Dlubal

Version
December 2013

Program

RFEM 5

Spatial Models Calculated
Acc. to Finite Element Method

Program Description

© Dlubal Software GmbH
Am Zellweg 2 D-93464 Tiefenbach

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

Contents

Contents

Contents

Contents

Contents

1. Introduction

1.1 New in RFEM 5

RFEM, the FEA program used to calculate plates, walls, shells, solids and frameworks, is a powerful tool to meet various challenges occurring in modern civil engineering. The program represents the basis for DLUBAL's analysis software composed of various design modules: RFEM determines internal forces, deformations and support reactions of general plate and shell models with our without member and solid elements.

The program version RFEM 5 offers you several useful features and options emphasizing user-friendliness and easy program handling when working on structural analysis projects. Once again, we would like to thank our customers for their valuable ideas and remarks.

Please find the most important innovations in RFEM 5 listed below:

- Graphical user interface in French, Italian, Polish, Portuguese, Russian, Spanish
- Direct setting of different types for openings
- Member eccentricities from cross-section dimensions
- Working diagrams and extended criteria for nonlinear nodal supports and releases
- Orthotropic properties for quadrangle and membrane surfaces as well as solids
- Input option for hybrid timber cross-sections
- Filter in cross-section library with favorites
- Solid modeling due to extrusion of surfaces in relation to plane or point, also with tapered sides
- Intersections of solids with Boolean operators
- Inserting a member to existing member
- Import of files from Bentley ISM, Ansys Apdl and Scia Engineer
- Import of 3D objects
- Graphical assignment of member properties
- Color symbols in tables for cross-sections, surfaces, solids, types of surfaces and members
- Selection with ellipse, annulus or intersection line
- Work planes defined by three points or line, member and surface axes
- Color management for types of surfaces, stiffnesses, members and solids
- Input of inclination and precamber in absolute values
- Specification of surface size and weight in input table
- Automatic creation of load and result combinations according to standard specification
- Gradual refinement of FE mesh in boundary areas of surfaces
- Large deformation analysis calculating according to NEWTON-RAPHSON, PICARD or as dynamic relaxation
- Output of load distribution, strains, member coefficients and member slendernesses
- Smooth ranges for evaluation of singularities
- User-defined settings for lighting
- Results evaluation by means of clipping plane
- *Views* navigator for user-defined and generated visibilities and angles of view
- Configuration Manager for display properties, toolbars, printout report headers etc.
- Mail merge of graphics
- PDF export of printout report

We hope you will enjoy working with RFEM 5.

Your team from DLUBAL SOFTWARE GMBH

1.2 Program Capacities

The following values represent the upper limits in the RFEM data structure. Please note that complex structures require powerful hardware.

Model data

99,999 objects of each category (nodes, lines, surfaces, cross-sections etc.)

Load data

99,999 objects of each type of load per load case

Load cases and combinations

Load cases (linear calculation)	9,999
Load combinations (nonlinear calculation)	9,999
Result combinations	9,999

Table 1.1: Program limits of RFEM

1.3 Company Profile

Since its beginnings in 1987, DLUBAL SOFTWARE GMBH has been involved in the development of both powerful and user-friendly programs for structural and dynamic analysis. In 1990, the company moved into its current location to Tiefenbach in Eastern Bavaria. A local branch exists since 2010 in Leipzig.

When looking at our programs you can feel the enthusiasm of everybody involved in the software development and you will notice the underlying philosophy of all our applications, which can be expressed in one word: user-friendliness. These two points combined with our expertise in engineering are forming the base for the ever-growing success of our products.

The software has been designed in such a way that even users with basic computer skills can handle the software successfully after a short while. With considerable pride, we now number more than 7,000 engineering offices as well as construction companies from a variety of fields and places of higher education among our satisfied customers all over the world. To remain true to our objectives, there are more than 150 internal and external employees working continuously on the development and improvement of DLUBAL applications. For general questions and problems, our customers can always rely on our qualified fax and email hotline.

The perfect balance between price and performance combined with excellent customer service provided by experienced civil engineers make DLUBAL programs an essential tool for anyone working in the areas of structural engineering, dynamics and design.

1.4 RFEM Team

The following people were involved in the development of RFEM 5:

Program coordination

Dipl.-Ing. Georg Dlubal
Ing. Pavel Bartoš
Ing. Pavol Červeňák
Dipl.-Ing. (FH) Younes El Frem
Dipl.-Ing. Frank Faulstich
M.Eng. Dipl.-Ing. (FH) Walter Rustler

Programming

RNDr. Miroslav Šejna, CSc
Ing. Radek Brettschneider
Jan Brnušák
Ing. Martin Budáč
Ing. Michal Búzik
Dipl.-Ing. Georg Dlubal
Jan Fenár
Ing. Jan Gregor
Ing. Jiři Kubíček
Dr.-Ing. Jaroslav Lain
Ing. Jan Miléř
Ing. Daniel Molnár
Ing. Petr Novák
Ing. Jan Otradovec
Mgr. Petr Oulehle
Mgr. Jiří Patrák
Mgr. Andor Patho
Mgr. Petr Pitka
Bc. Ondřej Planý
Ing. Jan Rybín, Ph.D.
Ing. Fatjon Sakiqi
Ing. Pavel Spilka
Ing. Roman Svoboda
RNDr. Stanislav Škovran
Dis. Jiří Šmerák
Ing. Jan Štalmach
Lukáš Tůma
RNDr. Miroslav Valeček
Ing. Vítězslav Zajíc
Michal Zelenka

Programming - analysis core

Doc. Ing. CSc. Ivan Němec
Ing. Jiří Buček
Ing. Jiří Doležal
Ing. Petr Horák
Ing. Jaromir Kabeláč
Ing. Ph.D. Radoslav Rusina
Ing. CSc. Ivan Ševčík
Ing. CSc. Zbyněk Vlk
Ing. Lukáš Weis
RNDr. Milan Zeiner

Program design, dialog figures, icons

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch
Zdeněk Ballák
Ing. Jan Miléř

Blocks

Ing. Tommy Brtek
Ing. Dmitry Bystrov
Ing. Evžen Haluzík

Program supervision

Ing. Alexandra Bayrak
Marian Bocek
Ing. Tommy Brtek
Ing. Ondřej Šašinka
Ing. Tomáš Ferencz
Ing. Vladimír Gajdoš
Ing. Jakub Harazín
Ing. Martin Hlavačká
Ing. Iva Horčičková
Karel Kolář
Ing. František Knobloch
Ing. Ctirad Martinec
Pavla Novotná
Ing. Vladimír Pátý
Ing. Evgeni Pirianov
Ing. Václav Rek
Ing. Jan Rybín, Ph.D.
Mgr. Ph.D. Vítězslav Stembera
Ing. Ondřej Šupčík
Ing. Martin Vasek
Marek Ženuch

Localization, manual

Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Ing. Jana Duníková
Ing. Lara Freyer
Alessandra Grosso
Bc. Chelsea Jennings
Jan Jeřábek
Ing. Ladislav Kábrt
Ing. Aleksandra Kociołek
Mgr. Michaela Kryšková
Dipl.-Ing. Tingting Ling
Ing. Roberto Lombino
Eng.º Nilton Lopes
Mgr. Ing. Hana Macková
Ing. Téc. Ind. José Martínez
MA SKT Anton Mitleider
Dipl.-Ü. Gundel Pietzcker
Mgr. Petra Pokorná
Ing. Zoja Rendlová
Dipl.-Ing. Jing Sun
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl
Ing. Marcin Wardyn

Technical support, quality management

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. Thomas Günthel
Dipl.-Ing. (FH) Sebastian Hawranke
Dipl.-Ing. Wieland Götzler
Dipl.-Ing. (FH) Andreas Hörold
Dipl.-Ing. (FH) Paul Kieloch
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Gerhard Rehm
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

1.5 Using the Manual

Many roads lead to Rome – this policy also applies to working with RFEM: graphics, tables and navigators are on an equal footing. The descriptions in this manual follow the sequence and structure of the tables provided for model, load and results data. The individual tables are described in detail column by column. Instead of presenting general Windows features, the manual often focuses on practical hints and tips.

If you are new to the program, you should work with the introductory example first, describing step by step how to enter data. Please find the PDF document available for download on our website www.dlubal.com/downloading-manuals.aspx. In this way, you can get quickly familiar with the most important features of RFEM. Advanced program users may try our detailed tutorial also available for download. Both examples can also be performed within the restrictions of the demo versions.

The text of the manual shows the described **buttons** in square brackets, for example [Apply]. At the same time, they are shown in the left margin. In addition, **expressions** used in dialog boxes, tables and menus are set in *italics* to clarify the explanations.

The index at the end of the manual helps you to find specific terms and subjects. However, if you do not find what you are looking for, please check our blogs at www.dlubal.de/blog/en where you can browse the articles to find a solution.

2. Installation

2.1 System Requirements

In order to use RFEM without any difficulties, the following system requirements are recommended:

- Operating system Windows Vista/7/8
- x86 CPU with 2 GHz
- 2 GB RAM
- DVD-ROM drive for installation (alternatively a network installation is possible)
- 10 GB hard disk capacity, including approximately 2 GB required for installation
- Graphic card with OpenGL acceleration and resolution of 1024 x 768 pixels. Onboard solutions and shared-memory-technologies are not recommended.

RFEM is not supported by Windows 95/98/Me/NT/2000/XP, Linux, Mac OS or server operating systems.

No product recommendations are made, with the exception of the operating system, as RFEM basically runs on all systems that fulfill the system requirements mentioned above. If RFEM is used for intensive calculations, the guiding principle 'more is better' applies.

When complex structural systems are calculated, huge amounts of data are produced. As soon as the main memory is not sufficient for taking the data, the hard drive will take over. This can slow down the computer significantly. Therefore, upgrading the main memory will usually speed up the calculation more than a faster processor.

As the RFEM analysis core supports several processor kernels, you can completely exploit the potential of the 64-bit operating system. For 32-bit systems the memory size used by the processor is limited to 2 gigabytes. Therefore, more memory can be used with the 64-bit technology. If you work with a computer having sufficient RAM memory using a 64-bit operating system, the fast and direct equation solving method can be applied even to bigger models.

To calculate complex structural systems, we recommend the following configuration:

- Quad-core processor
- Windows 7/8 64-bit
- 8 GB RAM

2.2 Installation Process

The program family **RFEM** is delivered on DVD. In addition to the main program RFEM, the DVD contains all additional modules that belong to the RFEM program family, for example **RF-CONCRETE**, **RF-STEEL**, **RF-STABILITY** etc.

Before you install RFEM, close all applications running in the background.

Please make sure that you are logged on as administrator or to have administrator rights for installing the programs. When working with RFEM, user rights will be sufficient. Please find detailed instructions shown in the User Rights video and the User Rights document available on our website.

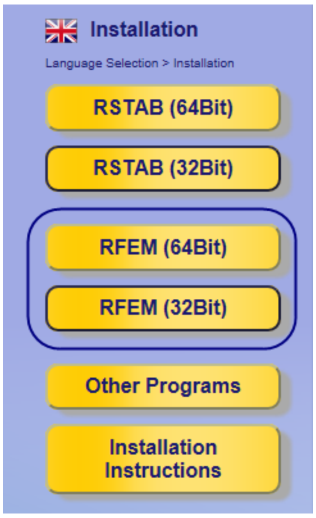
2.2.1 Installation from DVD

On the back side of the DVD case you can find instructions for installation.

- Insert the DVD into your DVD-ROM drive.
- The installation process starts automatically. If it does not start, the *autorun* function may be inactive. In this case, start the file *setup.exe* from the DVD either in the Explorer or by entering the command *'D':\setup.exe* in the input field of the Start menu (*'D'* refers to the drive letter of your DVD drive).
- In the start dialog box, select the language.

Figure 2.1: Select language

- In the next dialog box, define the program version (*64-bit* or *32-bit*).
- Follow the instructions of the *Installation Wizard*.



Select installation

Connect the dongle to a USB port of your computer only after the installation is complete. The dongle driver will be installed automatically.

The DVD also contains instructions for installation and the RFEM manual in PDF format. To look at the manual, you need the Acrobat Reader that you can install from the DVD.

RFEM as full or trial version

When you start the program for the first time after the installation has been completed successfully, you have to decide if you want to use RFEM as a full version or as a trial version running for 30 days.

To run the program as a full version, you need a dongle (hardlock) and an authorization file (*Author.ini*). The dongle is a plug to be plugged in a USB port of your computer. The authorization file contains coded information about your license(s). Usually, we send you the *Author.ini* file by e-mail. Also the Extranet to which you log in on our website www.dlubal.com provides access to your authorization file. Save the *Author.ini* file on your computer, a USB flash drive or in the network.

Each work station requires the authorization file. The file can be copied as many times as you want. However, if the content will be changed, it cannot be used anymore for authorization.

It is also possible to run the RFEM full version with a *Software Licence* without dongle.

2.2.2 Network Installation

Local licenses

The installation can be started from any drive of your computer or server. First, copy the contents of the DVD to the relevant folder. Then, start the file *setup.exe* from the client. The following steps do not differ from the DVD installation.

Network licenses

When you have network licenses, install the program on the work stations as described. Then, the licenses will be approved by the SRM network dongle. Find detailed information about installing the network dongle in the instructions available on our website.

2.2.3 Installing Updates and Other Modules

The DVD contains the complete program package including all add-on modules. When purchasing a new add-on module, you will not necessarily receive a new DVD but always a new authorization file *Author.ini*. To update the authorization without reinstallation, select *Load Authorization File* on the *Help* menu in RFEM.

Old program files are removed and replaced by new ones when updating the program within one series of a version (for example 5.**02**.xxxx). Of course, your project data is preserved! When updating the program to the next series (for example 5.**03**.xxxx), the new version will be installed parallel to the old version (see below).

If you use printout report headers that you have defined yourself, save them before installing the update. The headers are usually stored in the file **DlubalProtocolConfigNew.cfg** that you find in the general master data folder *C:\ProgramData\Dlubal\Global\General Data*. The file won't be overwritten during the update. Nevertheless, saving a backup file may be useful.

We also recommend to save your report templates before you install the update. They are stored in the file **RfemProtocolConfig.cfg** in the folder *C:\ProgramData\Dlubal\RFEM 5.xx\General Data*.

The projects linked in the Project Manager are managed in the ASCII file **PRO.DLP** which can normally be found in the folder *C:\ProgramData\Dlubal\Global\Project Manager* (see Figure 12.21, page 553). If you want to uninstall RFEM before installing the update, you should save this file, too.

2.2.4 Parallel Installation of RFEM Versions

RFEM 4 and the individual series of version RFEM 5 can be run parallel on the computer since the program files are stored in different directories. The default folders are the following for a 64-bit operating system:

- RFEM 4: C:\Programs (x86)\Dlubal\RFEM4
- RFEM 5.01: C:\Programs\Dlubal\RFEM 5.01
- RFEM 5.02: C:\Programs\Dlubal\RFEM 5.02
- RFEM 5.03: C:\Programs\Dlubal\RFEM 5.03 etc.

All models created with the previous version RFEM 4 can be opened and edited in RFEM 5.

Models from RFEM 4 won't be overwritten when saving them in RFEM 5 as both programs use different file endings: RFEM 4 saves model data in the format ***.rf4**, RFEM in ***.rf5**.

Model files of RFEM 5 are downward compatible with certain restrictions. When you open an RFEM 5 model file in a previous version, a message appears telling you for example that compatibility problems for members with unsymmetric cross-sections may occur.

3. Graphical User Interface

3.1 Overview

When you open one of the examples included in RFEM, the screen should look like the figure below (Figure 3.1). The graphical user interface corresponds to usual Windows standards.

The following figure shows the most important areas of the program interface.

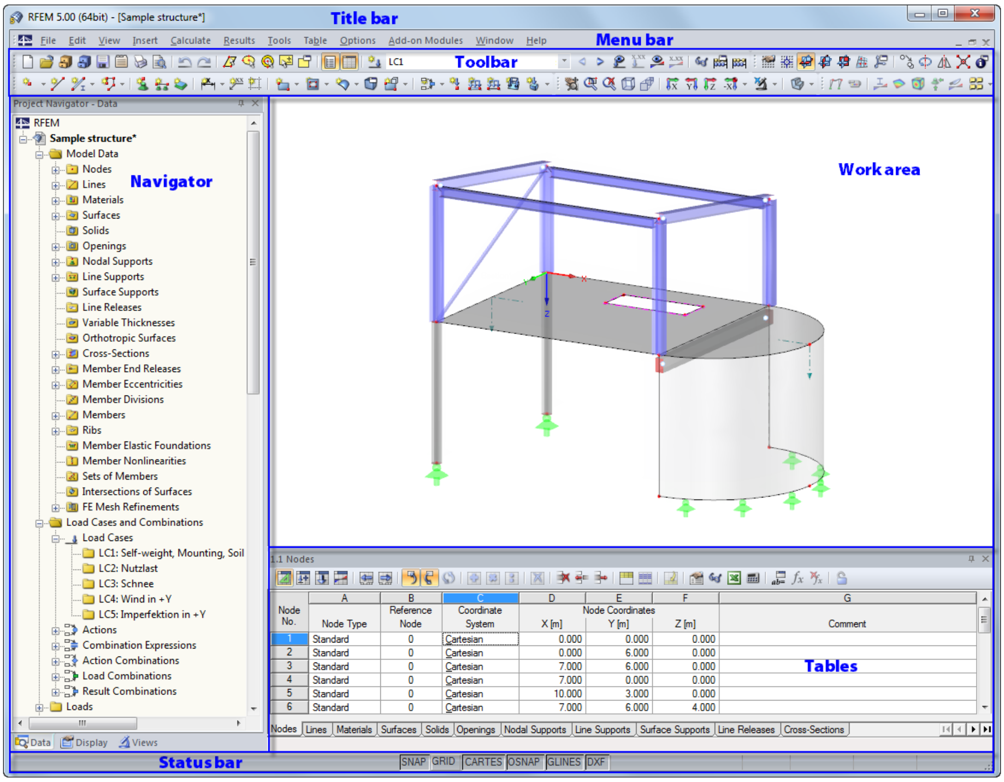


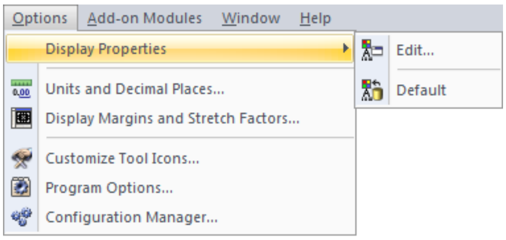
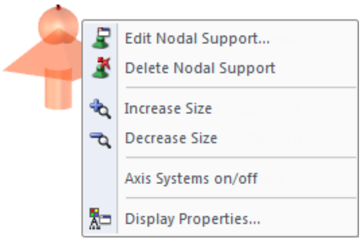
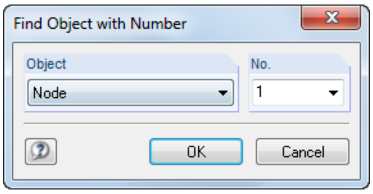
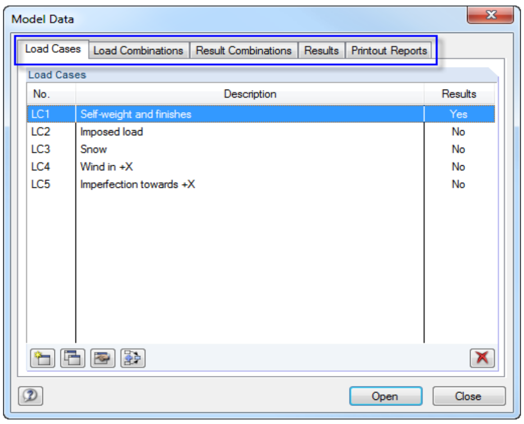
Figure 3.1: RFEM user interface

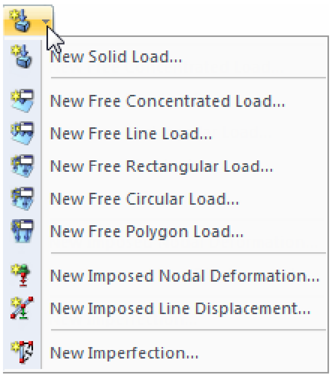
3.2 Terminology

This chapter explains important terms used in this manual relating to the user interface provided by Windows.

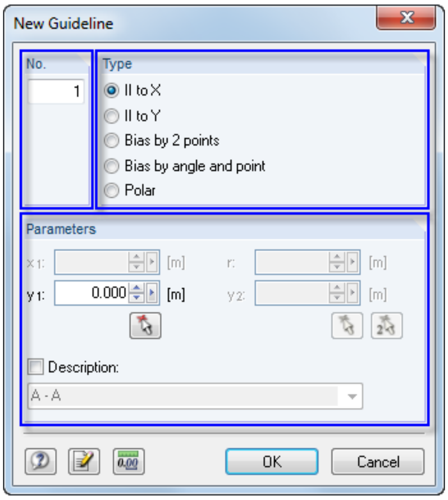
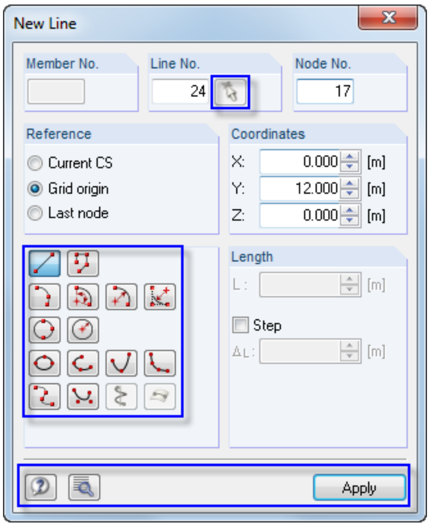
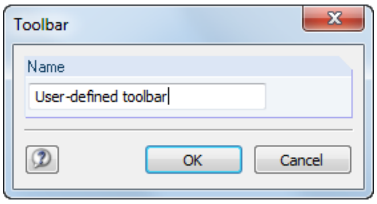
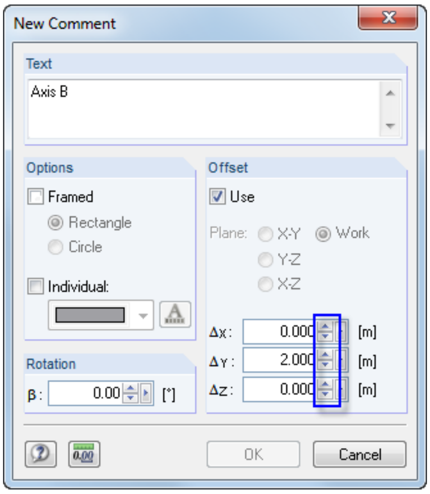
Different terms are used to describe the elements of the user interface. This manual uses English expressions often referring to the Microsoft Manual of Style for Technical Publications. Some terms are summarized if their differentiation is not essential for the operation of RFEM.

The following table describes frequently used terms.

Term	Figure	Synonym	Explanation
Menu		Pull down menu	Commands and functions below the title bar
Context menu		Pop-up menu	Open the context menu by clicking an object with the **right** mouse button. Contains useful commands and functions for the selected object.
Toolbar		Button bar	Collection of buttons below the menu bar
Dialog box			Window used for data input in the main window
Tabs		Register	Large dialog boxes are subdivided into several tabs. Click a tab to open the corresponding index card.



List button of toolbar

Section		Group, frame	Elements in a dialog box that belong together logically
Button		Icon	Click a button to start an action (for example to open a dialog box or to change data). The toolbar contains *list buttons*: Click [▼] to open a list with similar functions. The recently selected button is displayed at top.
Input field		Text box, input box	Field for entering text or numerical values
Spin box		Spin button	Two tiny buttons next to an input field Numerical values can be changed gradually.

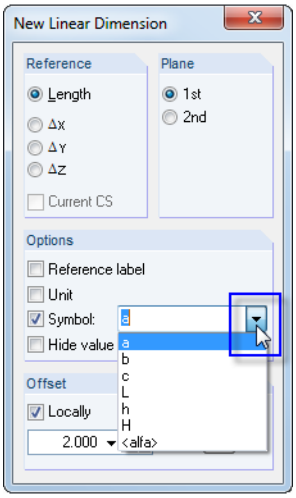
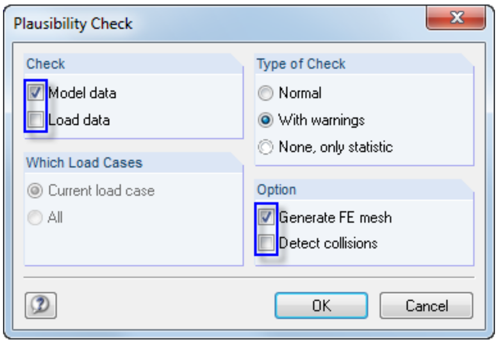
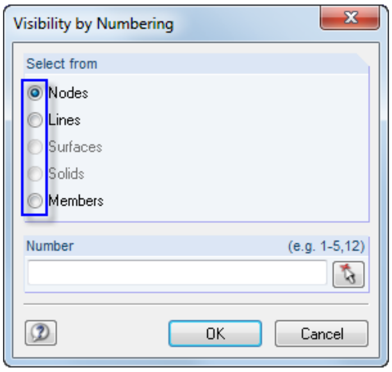
List		List box, combo box	Options for input fields Sometimes user-defined specifi-cations can be added.
Check box		Check box	Yes/No decision by ticking or clearing the check box
Selection field		Option but-ton	Only one of the options can be selected.

Table 3.1: Terms of user interface

3.3 Special Terms in RFEM

This chapter explains some important terms specific to RFEM.

Term	Explanation
Node	In the 3D model, a node is defined by its coordinates (X/Y/Z). Nodes are used to model the geometry of a structure.
Line	Nodes are connected by lines. Lines can be straight, bent or user-defined, for example arcs and splines.
Member	A member represents the property of a line. By properties for material and cross-section a certain stiffness is assigned to the member. A member is a 1D element.
Set of members	Members can be combined in a set of members. **Continuous members** join members continuously like in a continuous beam. A **group of members,** consisting of connected members, can join more than two members on a single node.
Surface	A surface is limited by boundary lines. By its material and thickness properties, a certain stiffness is assigned to the surface. Surfaces are 2D elements.
Solid	A solid is surrounded by boundary surfaces (usually type *Null*). The stiffness is defined by its material properties. Solids are 3D elements.
Nodal support	The degrees of freedom are limited for the node.
Line support	The degrees of freedom are limited for all nodes on the line.
Nodal load	Force or moment applied to a node.
Line load	A line is loaded by a concentrated load, a uniform or linearly variable load. The load acts as force or moment.
Member load	A member is loaded by a linear or single load. The load diagram can be either uniform or trapezoidal. In addition to forces and moments, temperature actions and prestresses are possible.
Surface load	A surface is loaded by a uniform or linearly variable load. In addition to forces, temperature loads and imposed deformations can act on the surface.
Solid load	A solid is loaded by effects of temperature or imposed deformations.
Load case *LC*	The loads from an action are managed in a load case, for example 'self-weight' or 'wind'. The loads should be defined as characteristic loads (thus without factors). Partial safety factors can be considered in load or result combinations. Usually, a load case is calculated according to linear static analysis, but a calculation according to second-order or large deformation analysis is also possible.

Load combination *CO*	A load combination is used to superimpose load cases, that means all **loads** of the load cases in question are summarized. Usually, a load combination is calculated according to second-order or large deformation analysis, but a calculation according to linear static analysis is also possible.
Result combination *RC*	A result combination sums up the **results** of the contained load cases. It is also possible to determine the extreme internal forces and deformations from different load cases, load or result combinations by an *Or* combination. However, the additive principle of superposition does not apply for results calculated according to second-order analysis.

Table 3.2: RFEM-specific terms

3.4 RFEM User Interface

This chapter describes the individual operating elements of RFEM (see Figure 3.1, page 15). The program follows the general standards for Windows applications.

3.4.1 Menu Bar

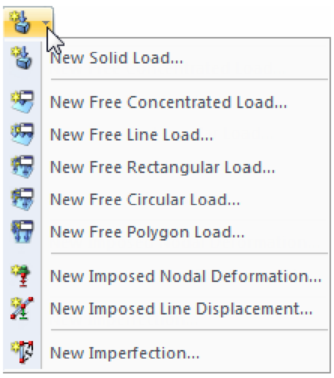
Below the title bar you see the menu bar. All functions of RFEM can be accessed in the menu bar. The functions are organized in logical blocks.

Open a menu by a single click on the left mouse button. You can also use the keyboard by holding down the [Alt] key in combination with the underlined letter of the menu title. Then, the menu opens and you can see its menu items. Select the entries by mouse-click or by pressing the underlined letter. You can also select an item by using the cursor keys [↑] and [↓] and finally pressing the [↵] key.

When a menu list is opened, you can switch between the menus or subentries by using the [→] and [←] keys.

For some menu items a keyboard shortcut is additionally shown. These key combinations follow the Windows standards. Use shortcuts to start the functions directly with keyboard keys (for example [Ctrl] + [S] saves data).

3.4.2 Toolbars



List button of toolbar

Below the menu bar you see the toolbars with various buttons. Use these buttons to access the most important functions directly by mouse-click. A short information of the button function appears when you point to a button using the mouse pointer (*ToolTip*, *ScreenTip*).

Some buttons provide subentries like a menu: These *list buttons* contain topic-related functions. Click [▼] next to the button symbol to access the functions. The recently selected button is preset at the top of the list.

To change the position of a toolbar, grab the bar in its front area with the left mouse-button. Then move it to the desired position.

Figure 3.2: Docked position of *View* toolbar

When you drag the toolbar into the workspace, it becomes a “floating” toolbar.

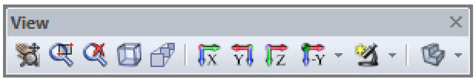


Figure 3.3: Floating position of *View* toolbar

To re-dock the floating toolbar, move it to the window frame with the mouse button. You can also double-click its headline.

On the **View** menu, click **Arrange Toolbars Customized** to open a dialog box for changing the content and look of toolbars. Customizing toolbars follows Windows standards.

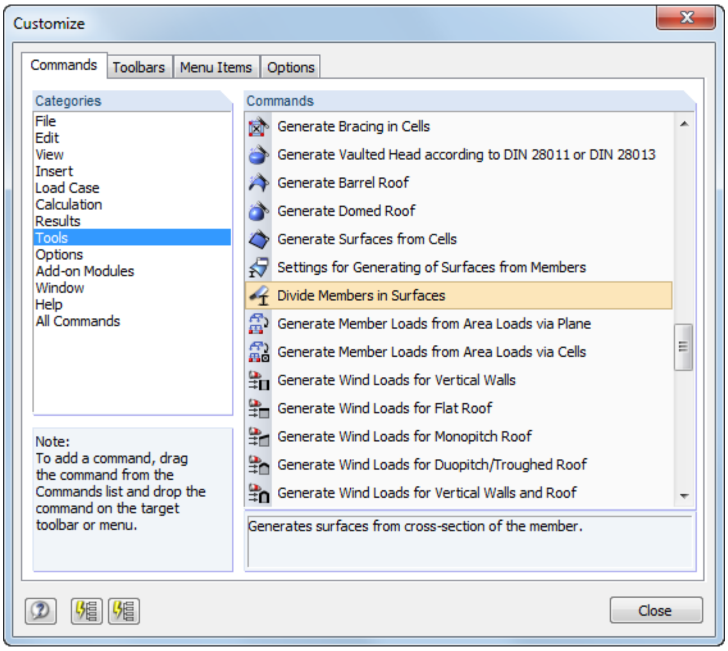
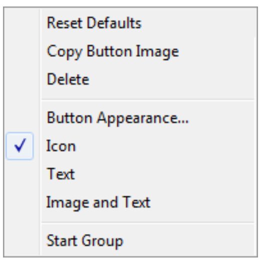


Figure 3.4: *Customize* dialog box, tab *Commands*

All commands of RFEM are sorted by *Categories*. Select an entry of the list to see the buttons of all associated *Commands* to the right. Click a button to get an explanation of the button function shown in the dialog section below. All buttons can be moved to any place in the toolbar by using the drag-and-drop function. It is recommended to integrate these additional buttons into a new toolbar (see Figure 3.6) as the remaining toolbars may be reset to the default entries when installing updates.



Context menu of a button or menu item

To remove a button from the toolbar, the *Customize* dialog box must be open. Then, you can drag and drop the button from the toolbar to the workspace. It is also possible to use its context menu shown on the left to *Delete* the button.

In addition to dragging commands into the toolbar, you can shift them into menus. In this way, you can create user-defined menus. Menu items can be deleted or adjusted by user specifications as described for toolbars.

The option *Button Appearance* available in the context menu opens the following dialog box:

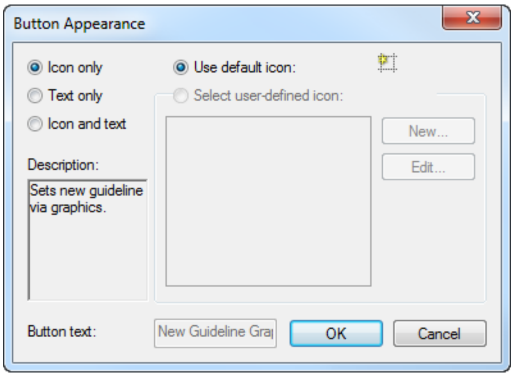


Figure 3.5: Dialog box *Button Appearance*

The dialog box helps you to change the *Text* of the button or menu item. Moreover, you can replace the default symbol by a *user-defined icon*.

All available toolbars are listed in the *Toolbars* tab of the *Customize* dialog box. You can switch off toolbars or create new ones by using the [New] button.

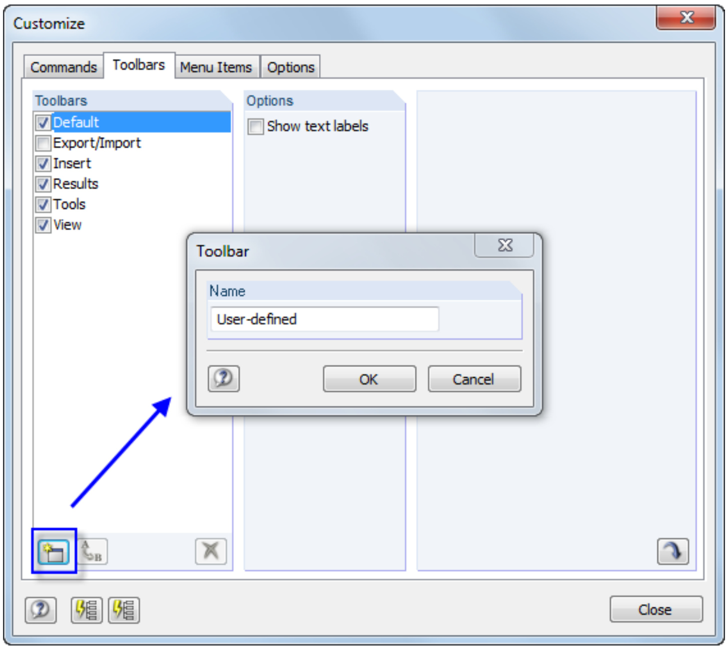


Figure 3.6: Creating a new toolbar

Enter the *Name* of the new toolbar in the *Toolbar* dialog box and click [OK]. The new bar will appear in a floating position on the display. You can shift the toolbar to an appropriate position and fill it with buttons by using the *Commands* tab (see above).

The button [Reset All User-defined Toolbars] resets the initial toolbar state. When the list contains a customized toolbar, the toolbar will be removed. The default toolbars of RFEM cannot be removed, but switched off only.

In the tab *Menu Items*, you can create user-defined pull-down menus. Proceed as described for creating new toolbars (see above).

Use the final dialog tab *Options* to change the appearance of the RFEM user interface. The following *Designs* can be selected:

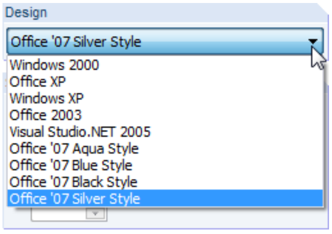


Figure 3.7: Available designs for the user interface

The new setting will be active immediately.

3.4.3 Project Navigator

To the left of the work window you see a navigator that looks like the Windows Explorer. To display or hide the *Project Navigator*, open the **View** menu and select **Navigator**, or use the corresponding toolbar button.

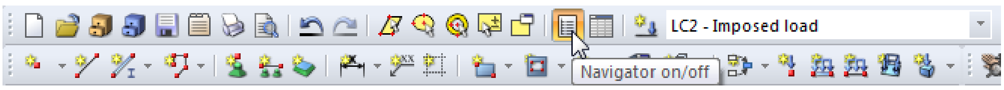
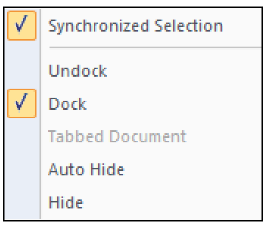


Figure 3.8: *Navigator* button in the *Default* toolbar

The navigator shows the model data of opened files in a tree structure. Click [+] to open a branch of the tree, click [-] to close it. You can also double-click the entry.

Similar to toolbars, you can use the mouse to "grab" the navigator in its title bar and move it to the workspace. To dock it again, double-click the title bar or move the navigator to the window frame. When moving the navigator, directional buttons shown on the left will additionally appear, facilitating the docking to one of the four sides of the work window. Drag the navigator to the arrow button of your choice and release the left mouse button as soon as the pointer is placed on the button.



Context menu of navigator

If you do not want the navigator to be docked to the window frame, clear the corresponding selection in the context menu of the navigator.

When the menu item *Synchronized Selection* is ticked, an object marked in the navigator will be highlighted with colors also in the model graphic.

The context menu option *Auto Hide* allows you to minimize a docked navigator: As soon as you click into the work window, the navigator slides to the edge and becomes a thin bar (see Figure 3.9). You can also use the pin button on the top right of the navigator to select this function (see Figure 3.10, page 24).

The navigator opens in full size when you move the pointer across the *Project Navigator* field highlighted in the docked navigator bar.

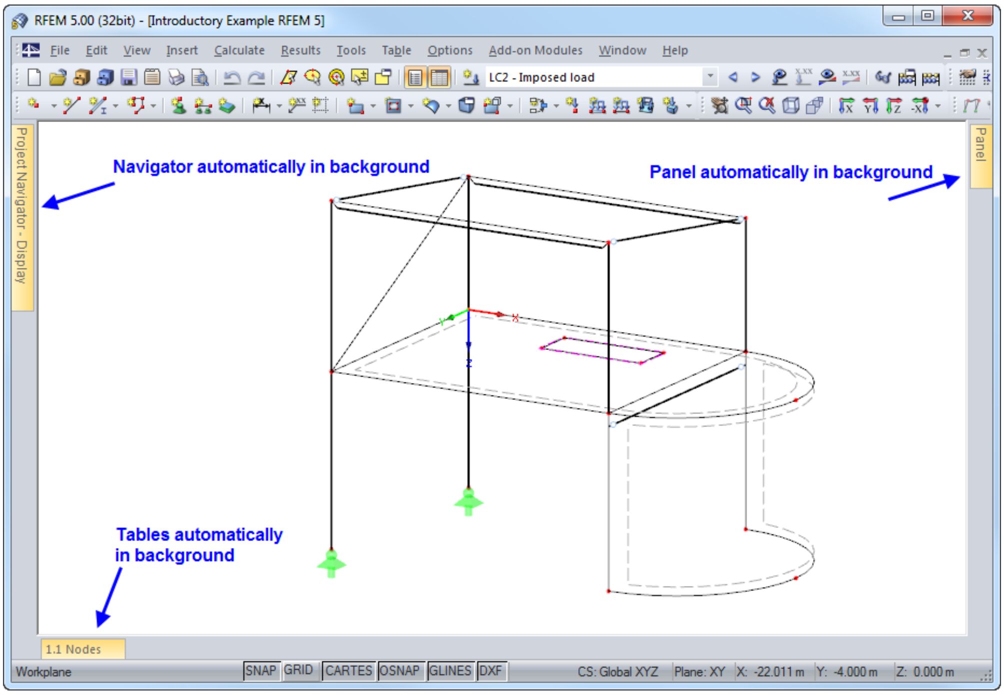


Figure 3.9: Navigator, tables and panel in auto hide mode

At the bottom edge of the navigator you see three tabs (four after calculations). Use the tabs to switch between *Data*, *Display*, *Views* and *Results* navigators.

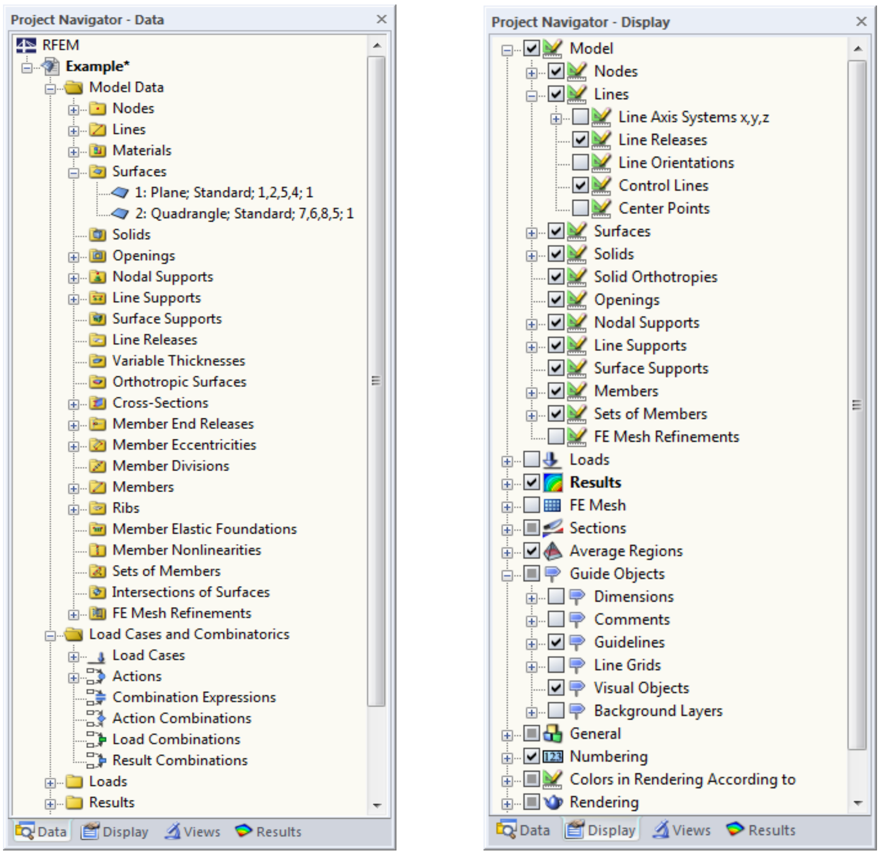


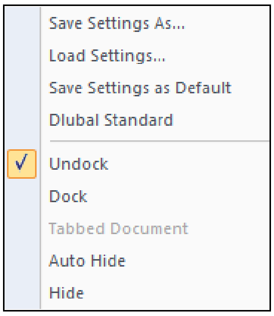
Figure 3.10: Tabs for *Data* and *Display* in Project Navigator

Data navigator

This navigator manages model and load data as well as calculated results. Double-click an entry (a "leaf" of the tree structure) to open a dialog box for changing the selected object. When you right-click an entry, a context menu appears with helpful functions used to create or modify the object.

Incorrectly defined objects are displayed in red, unused objects are displayed in blue letters.

Display navigator



This navigator controls the graphic display in the work window. When you clear the check box of an entry, the corresponding object will be hidden in the graphics.

Use the context menu of the navigator shown on the left to save or import user-defined settings. You can also apply saved settings as default for new models.

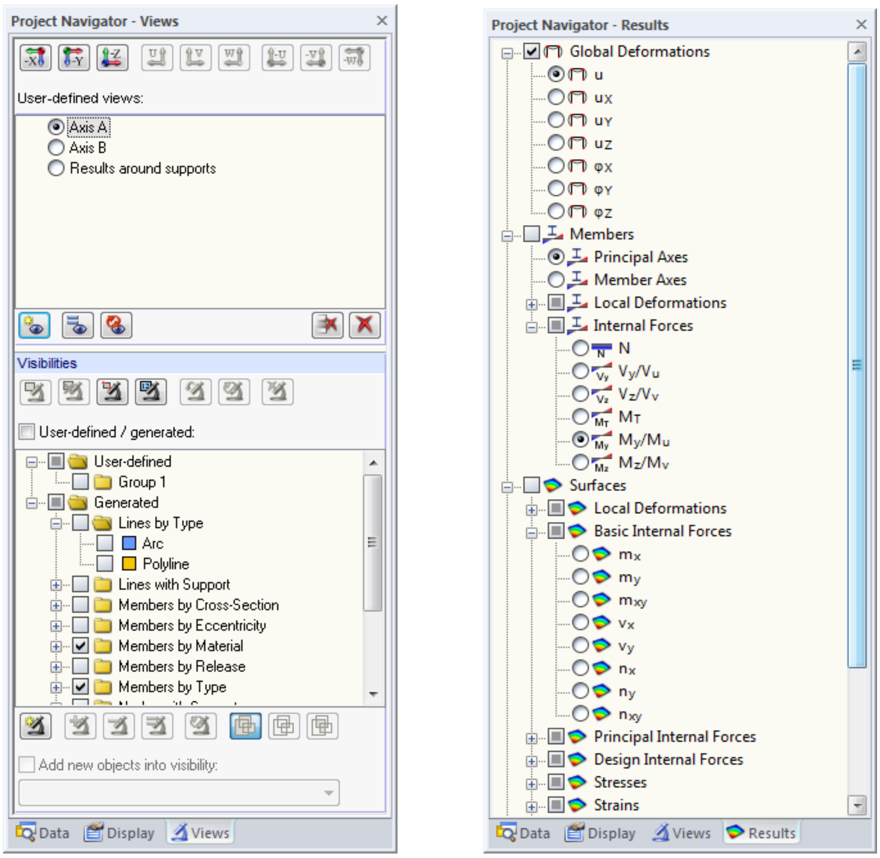


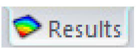
Figure 3.11: Tabs for *Views* and *Results* in Project Navigator

Views navigator

The navigator manages user-defined views as well as user-defined and automatically created visibilities of objects (used to be "partial views" and "groups" in RFEM 4). Buttons are available to create user-defined views, to set visibilities, to integrate objects into user-defined visibilities etc.

Working with views and visibilities is described in chapter 9.9.1 on page 369.

Results navigator



With the final navigator you control the results displayed in the graphic. Available entries depend on whether results of RFEM or an add-on module are displayed.

3.4.4 Tables

At the bottom edge of the RFEM window you see tables. To switch the tables on and off, click **Display** on the **Table** menu, or use the corresponding button.

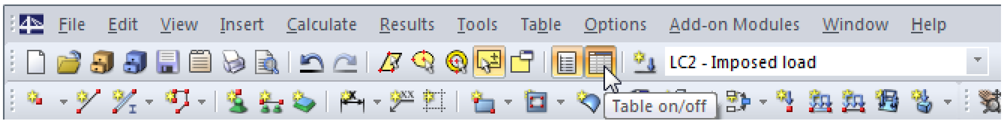


Figure 3.12: Button *Table on/off* in the *Default* toolbar

There are four groups of tables. To switch between them, use the first four buttons displayed in the toolbar of the table, or point to **Go To** on the **Table** menu.

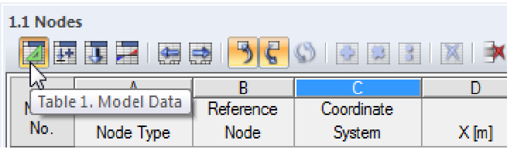
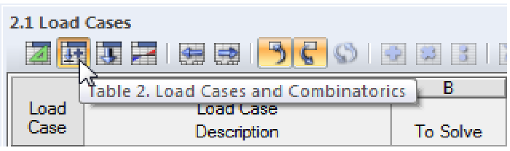
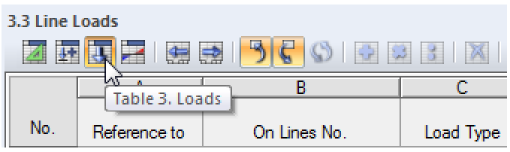
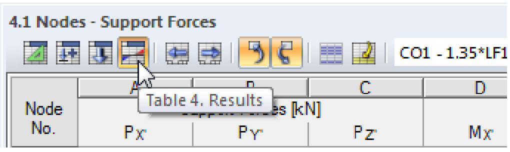
 **Table** menu → **Go To** → **Model Data**	Tables for model data
 **Table** menu → **Go To** → **Load Cases and Combinations**	Tables for load cases and combinations
 **Table** menu → **Go To** → **Loads**	Tables for loads
 **Table** menu → **Go To** → **Results**	Tables for results

Table 3.3: Buttons for control of table groups

The tables manage all model and load data numerically. Several powerful functions allow for an efficient data input (see chapter 11.5 on page 480).

By working through the tables successively RFEM ensures that all data is captured. The tables are representing the internal data organization of RFEM. Descriptions of input and output to be found in chapters 4, 5, 6 and 8 are based on the structure of these tables.

Similar to toolbars, you can use the mouse to "grab" tables in their title bar and move them into the workspace. To dock a table, double-click its title bar, or move the table to the window frame or one of the directional buttons shown on the left.

Docked tables can be minimized when the context menu option *Auto Hide* is set. As soon as you click into the work window, they slide to the edge (see Figure 3.9, page 24). You can also use the pin button on the top right of the table to select the minimize function. The tables open in full size again when you move the pointer over the docked bar.

When selecting a table row by mouse click, related objects are highlighted with colors in the graphic. Reciprocally, when an object is selected in the work window, the corresponding table row is displayed and highlighted, too. To control the settings for the so-called "synchronization of selection", point to **Settings** on the **Table** menu. You can also use the table toolbar buttons shown on the left (see chapter 11.5.4, page 486).

3.4.5 Status Bar

At the bottom of the RFEM work window you see the status bar. To activate or deactivate the bar, click **Status Bar** on the **View** menu.

The status bar consists of three areas.

Left area

Figure 3.13: Left area of status bar

The displayed text varies depending on the program function that is active. When the pointer moves across the work window, information appears about the object indicated by the pointer.

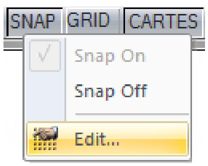
If you are an RFEM beginner, keep an eye on this section of the status bar: You may find useful hints and descriptions for toolbar buttons and dialog boxes.

Center area

Figure 3.14: Center area of status bar

Its functionality is similar to the one of a toolbar, controlling the display of the work window.

SNAP



The button enables or disables the snap function of the grid. Use the context menu to access the dialog box with specific settings for grid parameters (see chapter 11.3.2, page 437).

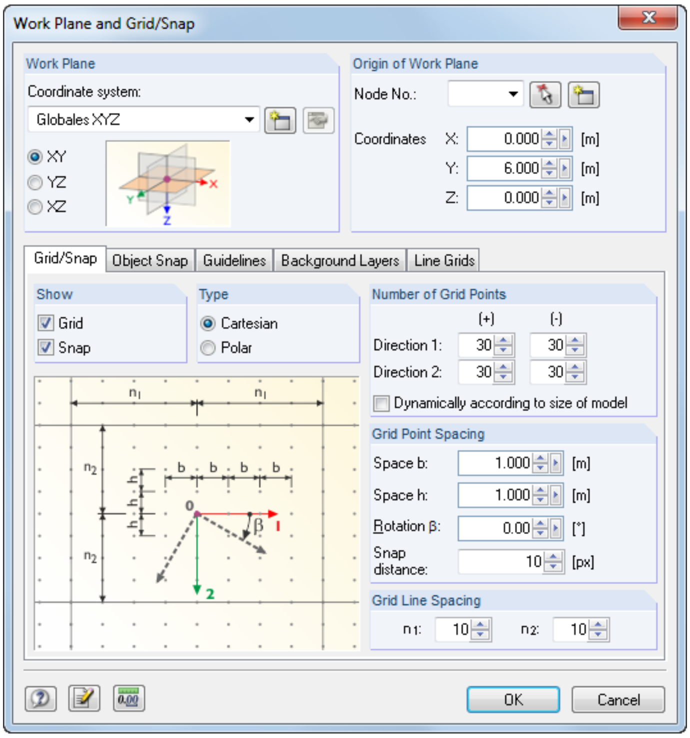
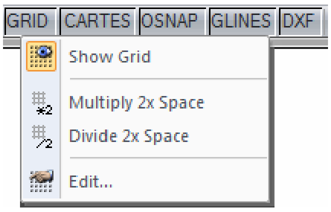
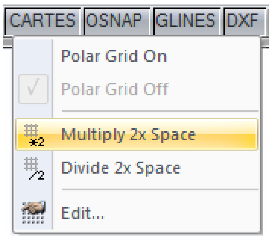


Figure 3.15: Dialog box *Work Plane and Grid/Snap*





GRID

Click the button to switch the grid on and off. Select *Edit* in the context menu to open the dialog box shown in Figure 3.15.

In addition, the context menu offers the possibility to maximize or minimize grid spacings gradually.

ORTHO / CARTES / POLAR

Use this button to select the orthogonal, Cartesian or polar grid. With the context menu you can open the dialog box shown in Figure 3.15. In addition, you can enlarge and reduce grid spacings gradually.

OSNAP

The button activates or deactivates the object snap (see chapter 11.3.3, page 438).

GLINES

The button controls the display of guidelines (see chapter 11.3.7, page 448).

DXF

Use this button to switch the display of background layers on and off (see chapter 11.3.10, page 455).

Right area

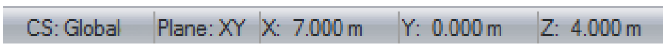


Figure 3.16: Right area of status bar

The right area of the status bar shows the following information about graphically entered data:

- Visibility mode (if active)
- Coordinate system *CS*
- Work plane
- Coordinates of current pointer position

3.4.6 Control Panel

As soon as internal forces or deformations are displayed graphically, the **panel** appears in the work window, offering different possibilities for display and control. To switch the panel on and off,

select **Control Panel (Color Scale, Factors, Filter)** on the **View** menu

or use the button shown on the left.

Similar to a toolbar, you can use the mouse to "grab" the panel in its title bar and shift it into the workspace. To dock the panel, double-click its title bar, or move it to the window frame or one of the directional buttons shown on the left.

The docked panel can be minimized when the context menu option *Auto Hide* is set. As soon as you click into the work window, it slides to the edge (see Figure 3.9, page 24). You can also use the pin button on the top right of the panel to select the minimize function. The panel opens in full size again when you move the pointer over the docked bar.

The control panel consists of the following tabs: *Color scale*, *Factors*, *Filter* and *Thicknesses*, if available.

Color spectrum

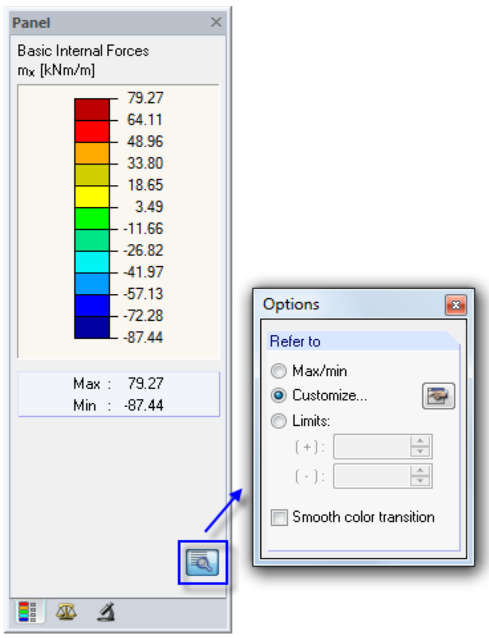


Figure 3.17: Control panel, tab *Color Spectrum* with active *Options* dialog box

When a multi-color results display is set, the first tab shows the color spectrum with assigned ranges of values. Eleven color zones are set by default, covering the range between extreme values in equally spaced intervals.


To adjust the color spectrum, double click one of the colors. You can also use the [Options] button available in the panel. The *Options* dialog box opens (Figure 3.17) where you can click the [Edit] button to access another dialog box for changing the ranges of colors and values.

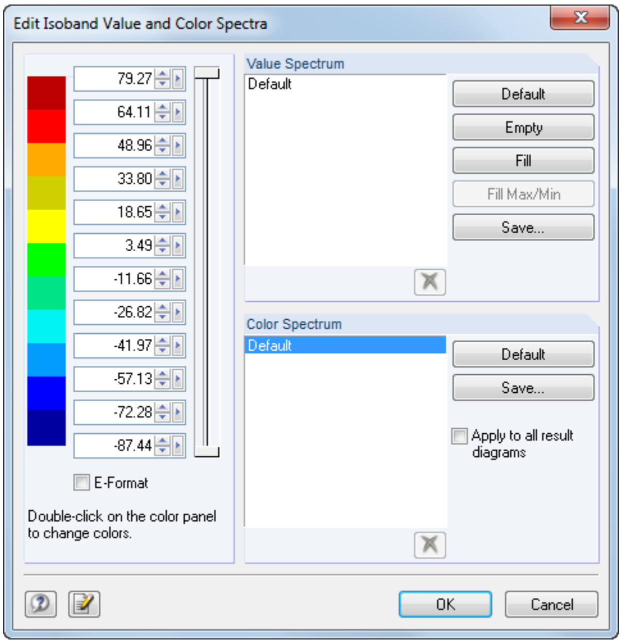


Figure 3.18: Dialog box *Edit Isoband Value and Color Spectra*

Use the vertical sliders to the right of the values to reduce the number of color ranges at both ends of the color spectrum.

You can change colors individually by double-clicking a color field.

Furthermore, you can adjust spectrum values manually. Please take care to follow a strictly ascending or descending order. Use the buttons in the dialog section *Value Spectrum* to assign values. The buttons are defined as follows:

Button	**Function**
Default	The eleven color zones will be set to default.
Empty	All values in the input fields will be deleted.
Fill	The values will be equidistantly intercalated between maximum and minimum depending on the number of color zones.
Fill Max/Min	For a reduced color spectrum, the interpolated values are calculated in relation to the absolute or manually entered extreme values.
Save	The value spectrum will be saved as a global sample.

Table 3.4: Buttons in the dialog section *Value Spectrum*

Tick the check box in front of *Apply to all result diagrams* to use the current color spectrum for the results display of all load cases, load and result combinations. The value spectrum remains unaffected because a global assignment for deformations, forces, moments and stresses would be difficult. Click [Save] to save the modified color spectrum as user-defined.

Use the [Options] button as shown in Figure 3.17 to select further options in the *Options* dialog box.

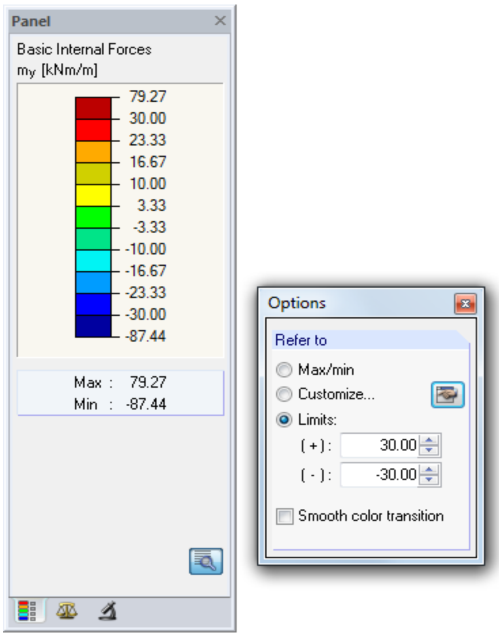


Figure 3.19: Dialog box *Options*, option *Limits* +/-

The reference to limit values allows you to evaluate results accurately within a defined zone. Exceedings of the upper and lower limits are represented by different colors. With the values set in Figure 3.19 you can see the moments m_y displayed in a fine gradation within the range of ± 30 kNm/m. Values beyond the defined zone appear in red or blue color.

Tick the check box for *Smooth color transition* in the *Options* dialog box to make the distinct color zones disappear. The smoothed color spectrum can be set independently, no matter which one of the three reference options is selected for the result values.

Factors

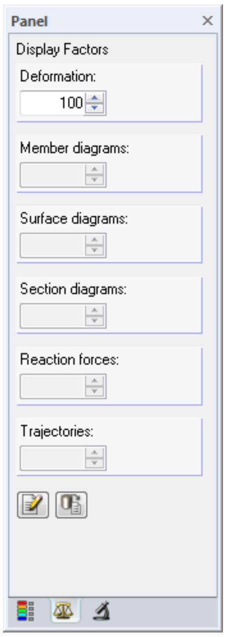


Figure 3.20: Control panel, tab *Factors*

Use the second panel tab to control the scaling factors for the graphic display. Depending on the currently set results graphic, you can access input fields for scaling *Deformation*, *Member diagrams*, *Surface diagrams*, *Section diagrams*, *Reaction forces* and *Trajectories*.

Filter

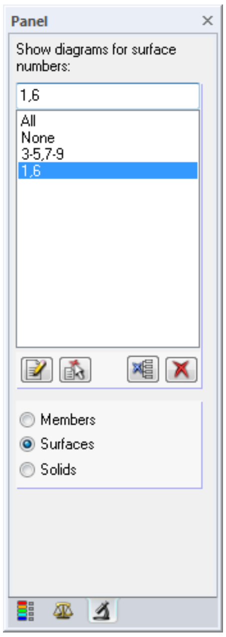


Figure 3.21: Control panel, tab *Filter*

With the *Color Spectrum* tab you can filter result values in general. Use the *Filter* tab to select different result displays for particular surfaces, members or solids.

The selection fields below the button row represent the three object categories for which you can display result diagrams. You have to enter the numbers of relevant members, surfaces or solids into the input field *Show diagrams for*. Then, with a click on the [Apply] button, you set the filter in the graphic.

It is also possible to take object numbers from the graphic: First, select the members, surfaces or solids (multiple selection via window or by holding down the [Ctrl] key). Then, click the button [Import from Selection].

The filter settings of the panel also affect the objects in the results tables: When you restrict the results display in the panel to for example two members, table 4.6 *Members - Internal Forces* will list only the results of those two members.

3.4.7 Default Buttons

Buttons are used in many dialog boxes. When you place the pointer on a button, its function will be displayed as short description after a moment.

The following overview describes frequently used default buttons.

Button	Name	Function
	New	Opens a dialog box to define an object
	Edit	Opens a dialog box to modify an object
	Delete	Deletes an object or entry
	Select	Graphical selection
	Apply	Import from the current selection
	Library	Opens a collection of stored data
	Help	Opens the help function
	Use	Applies changes without closing the dialog box
	Settings	Opens a dialog box for detailed settings
	Comments	Access to default text modules ➝ chapter 11.1.4, page 421
	Units and Decimal Places	Settings for units and decimal places ➝ chapter 11.1.3, page 420
	Default	Restores default dialog settings
	Set as Default	Saves the current settings as default
	Font	Option to set fonts and font sizes
	Colors	Option to set colors
	Info	Displays information about an object
	Transfer Selection	Transfers selected items from one list to another
	Transfer All	Transfers all items from one list to another
	Save	Stores user-defined entries
	Import	Imports stored entries
	Select	Selects certain or all objects
	Deselect	Deletes or cancels all entries

Table 3.5: Default buttons

3.4.8 Keyboard Functions

Often required functions in tables and graphical user interface can also be accessed with the keyboard.

[F1]	Help
[F2]	Next table
[F3]	Previous table
[F4]	Plausibility check for current table
[F5]	Plausibility check for all tables
[F7]	Selection function in tables
[F8]	Copies the table cell above, or shows entire model on screen
[F9]	Calculator
[F10]	Menu bar
[F12]	Saves the model under a new name
[Alt]	Menu bar
[Ctrl]+[2]	Copies a row of the table to the next row
[Ctrl]+[A]	*Redo* function
[Ctrl]+[C]	Copies to the clipboard
[Ctrl]+[E]	Exports data
[Ctrl]+[F]	Searches within the table
[Ctrl]+[G]	Generates entries in the table
[Ctrl]+[H]	Finds entries in the table and replaces them
[Ctrl]+[I]	Inserts a row in the table, or imports data
[Ctrl]+[L]	Jumps to a specific row number in the table
[Ctrl]+[N]	Creates a new model
[Ctrl]+[O]	Opens an existing model
[Ctrl]+[P]	Prints the graphic
[Ctrl]+[R]	Deletes a row in the table
[Ctrl]+[S]	Saves data
[Ctrl]+[U]	Clears selection in the table
[Ctrl]+[V]	Inserts from the clipboard
[Ctrl]+[X]	Cuts items in the table
[Ctrl]+[Y]	Deletes the content of a row in the table
[Ctrl]+[Z]	*Undo* function
[+] [-] *NumPad*	Zoom

Table 3.6: Keyboard functions

Provided that no dialog box is open, the [Enter] key calls up the last used function. Thus, reapplication of data is easier, for example placing model or load objects again in the work window.

3.4.9 Mouse Functions

The mouse functions follow general standards for Windows applications. To select an object for editing, simply click it with the **left** mouse button. Double-click the object when you want to open its dialog box for editing. You can apply these functions to objects of the work window as well as to entries in the *Data* navigator.

Model and load objects can be shifted in the work window by drag-and-drop. To copy objects, hold down the [Ctrl] key additionally. The drag-and-drop function can be switched on and off in the general context menu (see Figure 11.53, page 446).

When you click an object with the **right** mouse button, its context menu appears, providing object-related commands and functions.

Context menus are available in the graphic, navigator and tables.

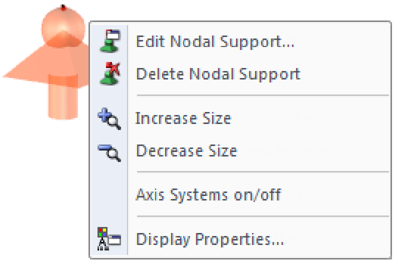
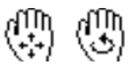


Figure 3.22: Context menu of nodal support in the graphic

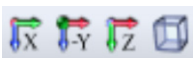
By scrolling the **wheel button** you can maximize or minimize the current model representation. The pointer position is always assumed as center of the zoom area.



Press the wheel button to move the model directly within the workspace, which means without previously activating the toolbar button [Move, Zoom]. When you press the [Ctrl] key additionally, you can rotate the model. Rotating the model is also possible by using the wheel button and holding down the right button of the mouse. The pointer symbols shown on the left are indicating the selected function.

To rotate the display around a particular node, select the node first. Now, hold down the [Alt] key and press the wheel button additionally to rotate the model about the selected node.

The options offered by a 3D mouse can also be used for working with the graphical user interface of RFEM.



Furthermore, RFEM offers a useful function to display selected objects quickly in maximized view: First, select the objects in the work window. Now, hold down the shift key [⇧] and click one of the buttons available in the *View* toolbar shown on the left. The work window will show you a maximized partial view of the object in the selected viewing direction.

3.4.10 Configuration Manager

The Configuration Manager provides access to all settings available for display properties, fonts, toolbars, print headers etc. To open the Configuration Manager, select **Configuration Manager** on the **Options** menu, or use the toolbar button shown on the left.

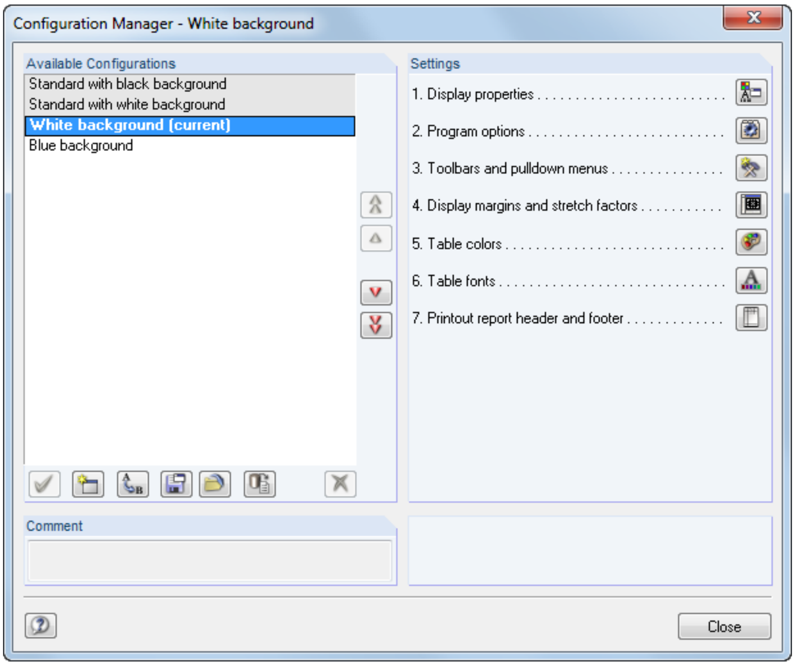


Figure 3.23: Dialog box *Configuration Manager*

Available configurations

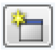
This dialog section lists all configurations that have been user-defined or created during installation. The setting currently used by the program is shown in bold and indicated as *current*.

The *Standard* configuration is preset, it cannot be deleted.

The buttons below the dialog section have the following functions:

Button	Function
	Sets the entry selected above as new *current* configuration
	Creates a new configuration from current settings (→ Figure 3.24)
	Renames the selected configuration
	Exports the selected configuration in a file
	Imports a configuration from a file
	Resets the default values
	Deletes the selected configuration (not possible for *Standard* and *current*)

Table 3.7: Buttons for *Available Configurations*



Use the [New] button to save the current settings as a new configuration. A dialog box opens where you have to enter a *Description*. An optional *Comment* makes it easier to select among various user-defined configurations later.

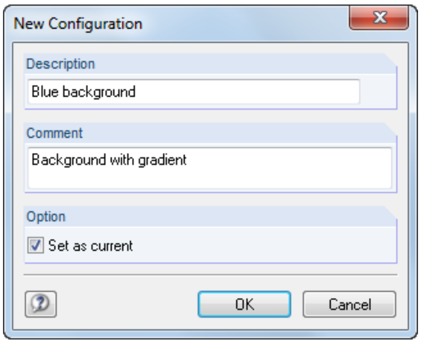


Figure 3.24: Dialog box *New Configuration*

Settings

The buttons available in the dialog section *Settings* provide access to different dialog boxes with configuration parameters. They are described in the following table.

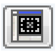
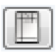
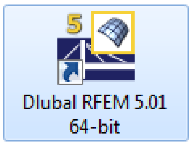
Button	Description	Function
	Display properties	Opens the dialog box *Display Properties* → chapter 11.1.2, page 417
	Program options	Opens the multi-tab dialog box *Program Options* → chapter 7.3.3, page 274 → chapter 9.10, page 379 → chapter 11.1.1, page 416 → chapter 11.1.4, page 422
	Toolbars and menus	Opens the dialog box *Customize* → chapter 3.4.2, page 21
	Margins and stretch factors	Opens the dialog box *Display Margins and Stretch Factors* → chapter 11.3.11, page 458
	Table colors	Opens the dialog box *Colors* for the table colors → chapter 11.5.4, page 486
	Table fonts	Opens the dialog box *Font* for the table fonts → chapter 11.5.4, page 486
	Header and footer of printout report	Opens the dialog box *Printout Report Header* → chapter 10.1.4, page 389

Table 3.8: Function of buttons in the dialog section *Settings*

4. Model Data



Starting RFEM

To start the program, use the Windows Start menu or the Dlubal icon on the desktop.

To enter data, a model must be created or opened (see chapter 12.2, page 554).

RFEM offers different options to enter data: You can define objects in a **dialog box**, a **table** and often directly in the **graphic**. All input is interactive, which means that graphical input is immediately reflected in the table and vice versa.

For first steps with RFEM we recommend to try the introductory example that may be helpful for beginners. You find the corresponding manual available for download on our website: www.dlubal.com/downloading-manuals.aspx

Open the input dialog box

You can access the input dialog boxes and the graphical input in different ways.

Menu *Insert*

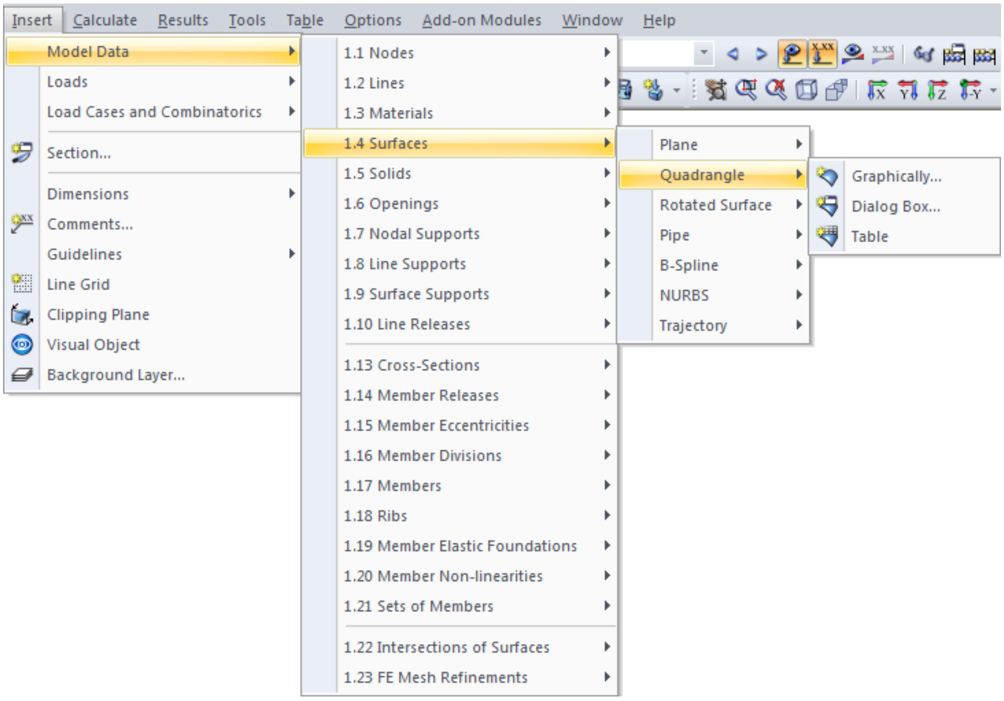


Figure 4.1: Menu *Insert* → *Model Data*

Toolbar *Insert*

Figure 4.2: Toolbar *Insert*

Context menu in *Data* navigator

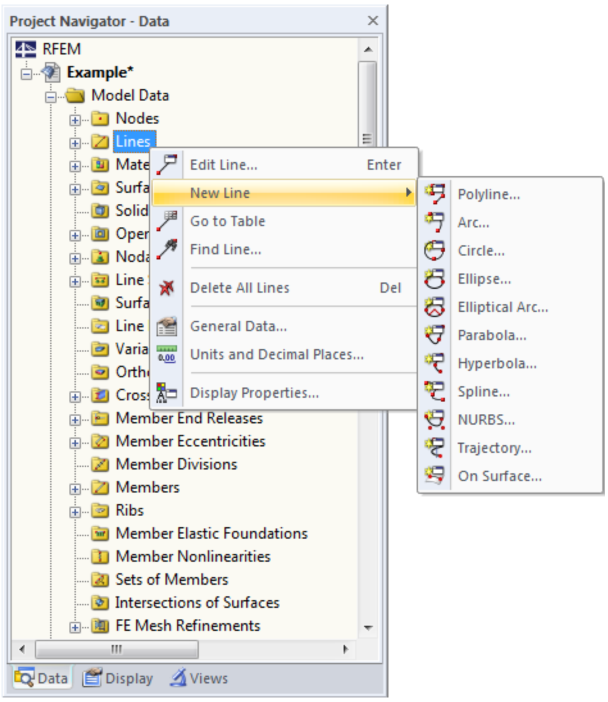


Figure 4.3: Context menu of model data objects in the *Data* navigator

Context menu or double-click in table

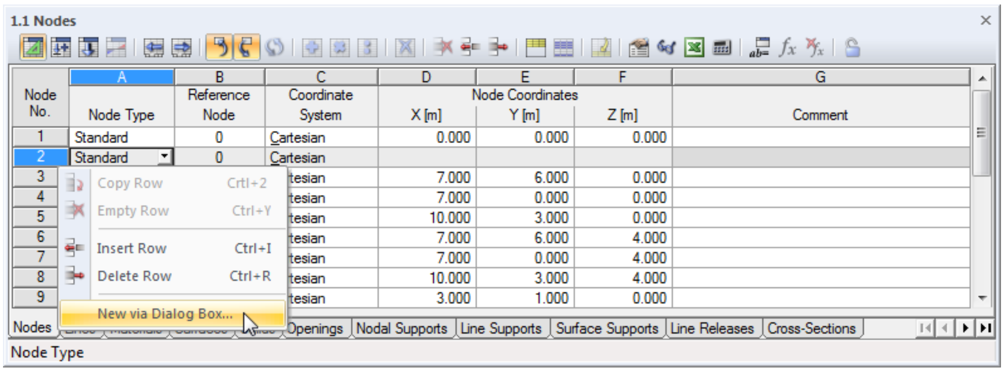


Figure 4.4: Context menu in model data tables

The input dialog box can be accessed by means of the context menu (or by double-click) of the row number.

Open the edit dialog box

RFEM provides different possibilities to open a dialog box for editing model objects.

Menu *Edit*

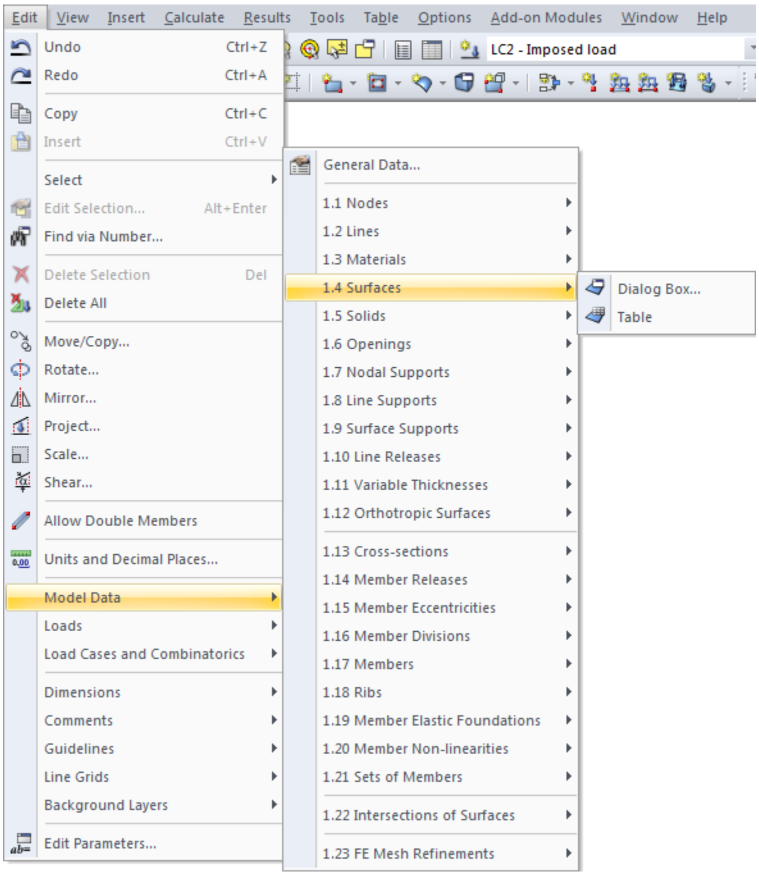


Figure 4.5: Menu *Edit* → *Model Data*

Context menu or double-click in graphic

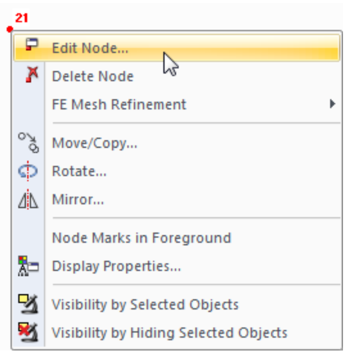


Figure 4.6: Context menu of a node in the work window

Context menu or double-click in *Data* navigator

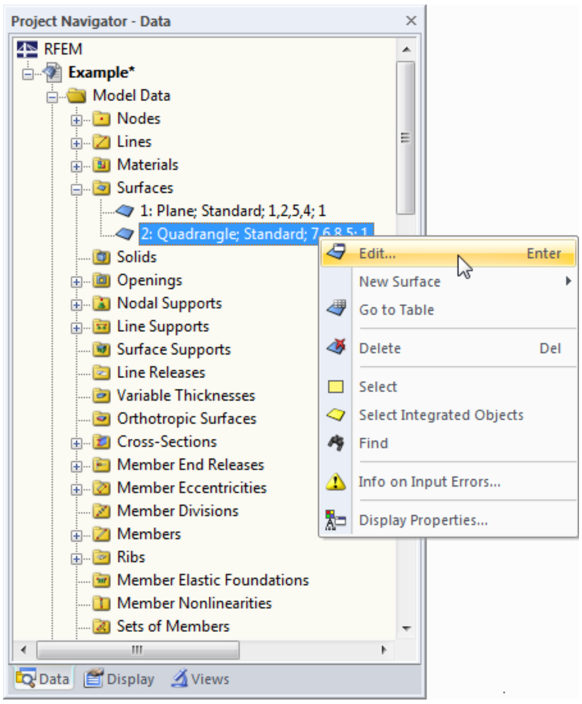


Figure 4.7: Context menu of model data objects in the *Data* navigator

Context menu or double-click in table

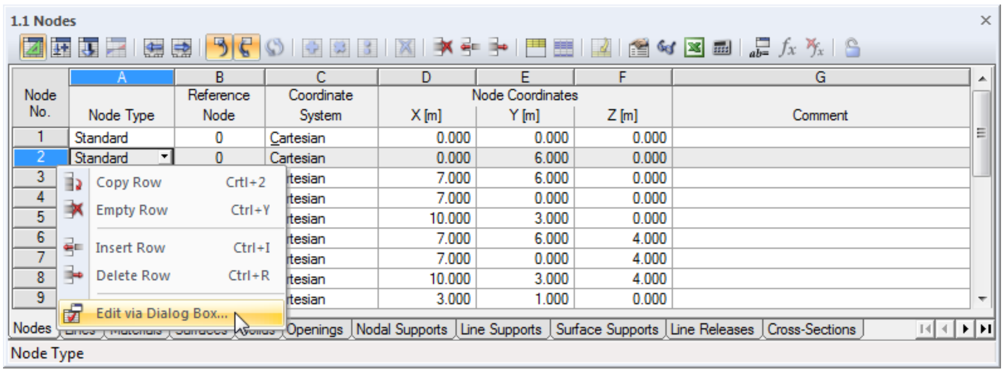
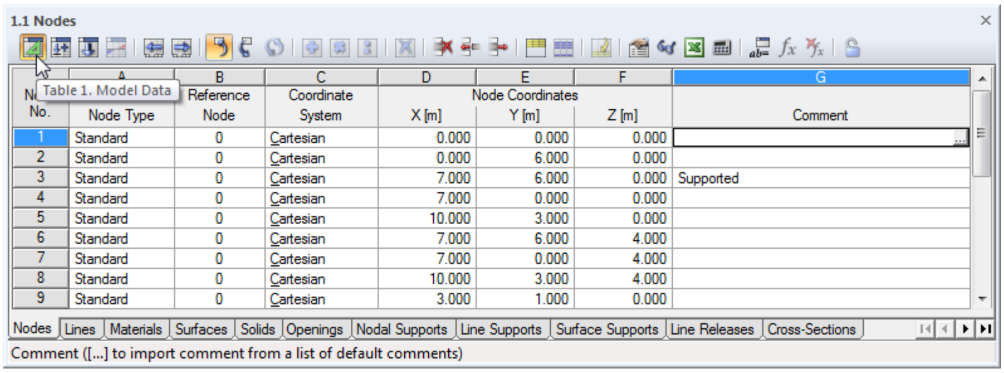


Figure 4.8: Context menu in model data tables

The edit dialog box can be accessed by means of the context menu (or by double-click) of the row number.

Table input

Input and modifications carried out in the graphical user interface are immediately shown in the tables, and vice versa. To open the model data tables, use the leftmost button in the table toolbar shown on the left.



	A	B	C	D	E	F	G
Node No.	Node Type	Reference Node	Coordinate System	X [m]	Node Coordinates Y [m]	Z [m]	Comment
1	Standard	0	Cartesian	0.000	0.000	0.000	
2	Standard	0	Cartesian	0.000	6.000	0.000	
3	Standard	0	Cartesian	7.000	6.000	0.000	Supported
4	Standard	0	Cartesian	7.000	0.000	0.000	
5	Standard	0	Cartesian	10.000	3.000	0.000	
6	Standard	0	Cartesian	7.000	6.000	4.000	
7	Standard	0	Cartesian	7.000	0.000	4.000	
8	Standard	0	Cartesian	10.000	3.000	4.000	
9	Standard	0	Cartesian	3.000	1.000	0.000	

Figure 4.9: Button [Table 1. Model Data]

Input in the form of spreadsheet data entered in tables can be quickly edited and imported (see chapter 11.5, page 480).

Unused objects are highlighted in blue in the tables and the *Data* navigator.

In each dialog box and table, it is possible to add a *Comment* specifying the object. You can also use predefined comments (see chapter 11.1.4, page 421). Moreover, comments are part of ScreenTips for graphical objects.

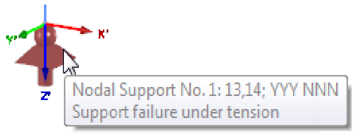


Figure 4.10: ScreenTip of a nodal support

4.1 Nodes

General description

The geometry of the model is defined by nodes. They are essential for creating lines, and thus members, surfaces and solids. Every node is specified by its coordinates (X,Y,Z). The coordinates usually refer to the origin of the global coordinate system, but it is also possible to define them in relation to another node.

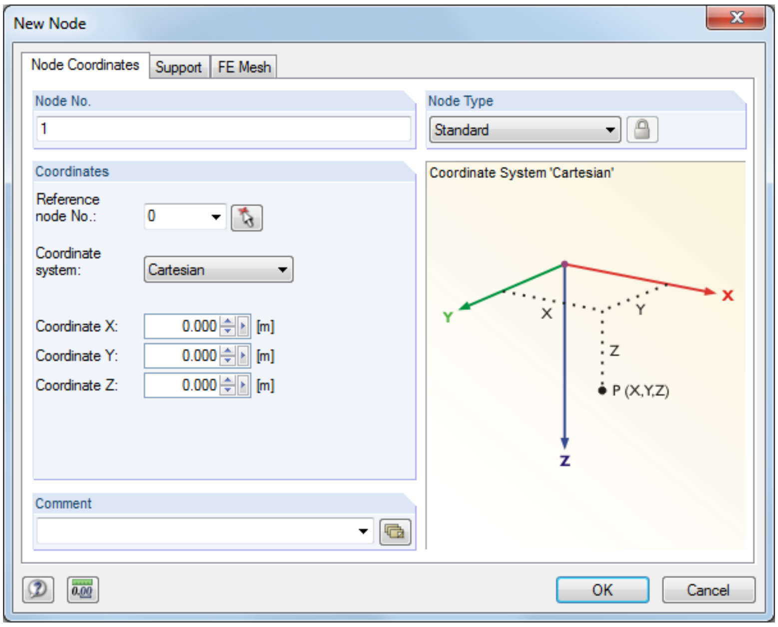
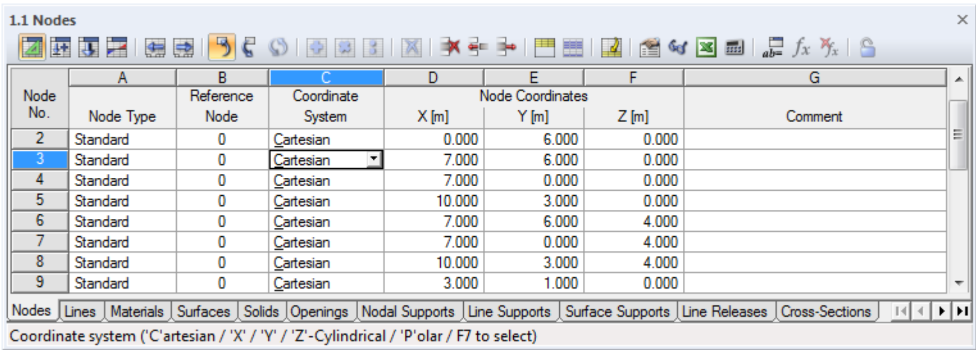


Figure 4.11: Dialog box *New Node*



1.1 Nodes

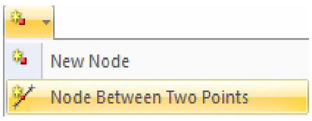
Node No.	A Node Type	B Reference Node	C Coordinate System	D Node Coordinates X [m]	E Y [m]	F Z [m]	G Comment
2	Standard	0	Cartesian	0.000	6.000	0.000	
3	Standard	0	Cartesian	7.000	6.000	0.000	
4	Standard	0	Cartesian	7.000	0.000	0.000	
5	Standard	0	Cartesian	10.000	3.000	0.000	
6	Standard	0	Cartesian	7.000	6.000	4.000	
7	Standard	0	Cartesian	7.000	0.000	4.000	
8	Standard	0	Cartesian	10.000	3.000	4.000	
9	Standard	0	Cartesian	3.000	1.000	0.000	

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Cross-Sections

Coordinate system ('C'artesian / 'X' / 'Y' / 'Z'-Cylindrical / 'P'olar / F7 to select)

Figure 4.12: Table 1.1 *Nodes*

The node number is assigned automatically in the dialog box *New Node* but can be modified in the input field. The order of the node numbering is not important and gaps are permitted.

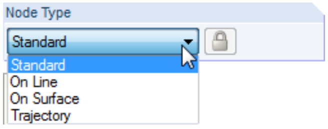


To adjust the order of node numbers subsequently, select **Renumber** on the **Tools** menu (see chapter 11.4.18, page 478).

Furthermore, RFEM provides a special function to create a node on the connection line of two nodes already existing (see chapter 11.4.12, page 473).

Node type

Default



This type of node is most frequently used. Standard nodes can be placed graphically in the work plane or anywhere in the workspace by coordinate specification. When you enter lines or rotated surfaces graphically, standard nodes will be created.

Standard nodes are displayed in red in the graphic.

On Line

Use this type of node to avoid that a line is divided into two lines. The complete line remains unchanged. The nodal parameter δ describes the relative distance to the start node of the line.

By creating nodes on lines it is possible to apply nodal loads anywhere on the line or to force an FE node.

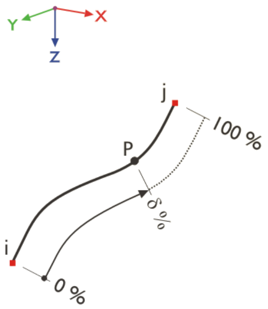


Figure 4.13: Node on line

Nodes on lines are displayed in light blue by default.

On Surface

It is difficult for quadrangle surfaces to determine the coordinates of nodes placed on the curved surface. With the node type *Node on Surface* you can place a node directly on a quadrangle surface displayed in the graphic. The nodal parameters δ_1 and δ_2 refer to the four corner nodes of the surface.

By creating nodes on surfaces it is possible to apply nodal loads anywhere on the curved surface or to force an FE node.

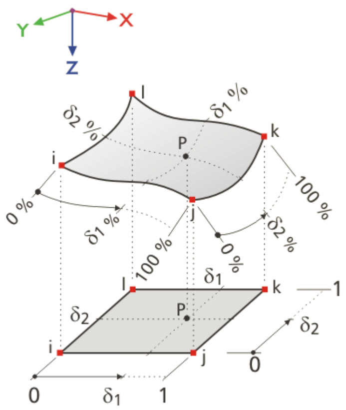


Figure 4.14: Node on surface

Coordinates in the table are stored in the Cartesian coordinate system. Nodes on surfaces are graphically displayed in light blue by default.

If you work with plane surfaces, use standard nodes.

Trajectory

This type of node is created when you define a spiral trajectory curve (see chapter 4.2, page 58). The nodal parameter δ describes the relative distance to the start node of the line.

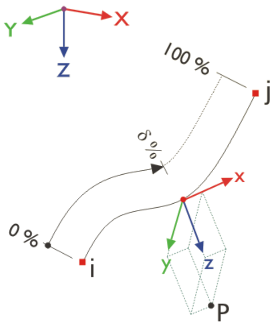


Figure 4.15: Trajectory

Trajectory nodes in the graphic are displayed in dark green by default.

Reference node

In general, coordinates of a node refer to the origin of the global coordinate system. You do not need to define the node (0/0/0) because RFEM recognizes the origin automatically.

Any node may serve as reference node. Even a node with a higher number is allowed to be used as reference node. Referring to another node may be useful to define for example a new node in a certain distance to an already known position. The table list with its option "*Previous* node" is especially useful in this case.

In the dialog box *New Node*, you can enter the reference node directly, select it from the list or define it graphically by using the [↖] button.

Coordinate system

The coordinates of a node always refer to a coordinate system that describes the position of the node in the workspace. Depending on the model geometry you can select between different coordinate systems. All coordinate systems are clockwise-oriented.

Cartesian

The global axes X, Y and Z describe a translational expansion (linear). All directions of coordinates are on an equal footing.

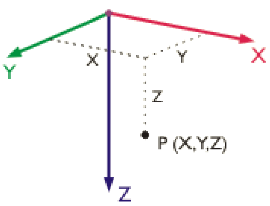


Figure 4.16: Cartesian coordinate system

In most cases, nodes can be defined in the Cartesian coordinate system.

X-cylindrical

The X-axis describes a translational expansion. The radius R defines the distance of the node to the X-axis. The angle θ defines the rotation of the coordinates about the X-axis.

The X-cylindrical coordinate system will be applied to represent for example tubular models whose central axis is the X-axis.

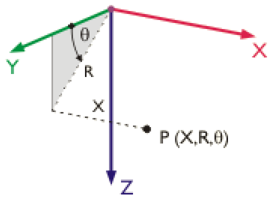


Figure 4.17: X-cylindrical coordinate system

Y-cylindrical

This coordinate system is similar to the X-cylindrical system, but now the longitudinal axis is represented by the Y-axis.

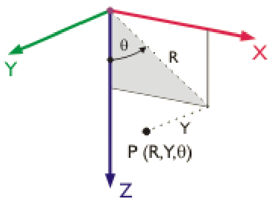


Figure 4.18: Y-cylindrical coordinate system

Z-cylindrical

The coordinate system is similar to the X-cylindrical system, but now the longitudinal axis is represented by the Z-axis.

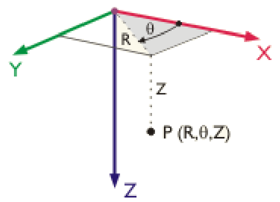


Figure 4.19: Z-cylindrical coordinate system

Polar

In the polar coordinate system, the node position is described by a radius defining the distance to the point of origin and the angles θ and Φ.

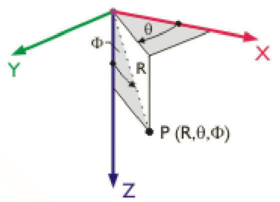


Figure 4.20: Polar coordinate system

If possible, organize the model input with regard to the global coordinate system in such a way that the X-, Y-, and Z-axes of the coordinate system are in line with the principal directions of the modeled framework. This allows for an easier definition of coordinates, conditions and loads.

To define nodes directly in the workspace, open the floating dialog box *New Node* for graphical input. Usually, nodes snap on grid points which are aligned with the active user-defined or global coordinate system (*CS*).

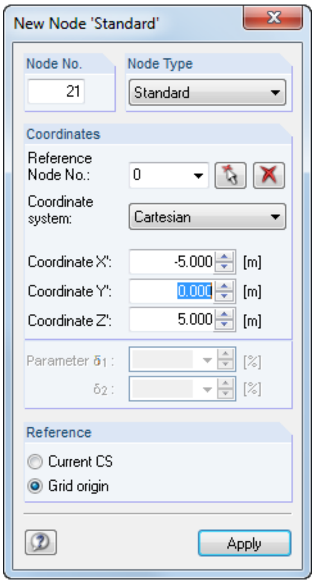


Figure 4.21: Floating dialog box *New Node*

For more information about user-defined coordinate systems, see chapter 11.3.4 page 442.

When the coordinate system is changed in the table, it is possible to convert node coordinates automatically to the new system. The following query will be displayed.

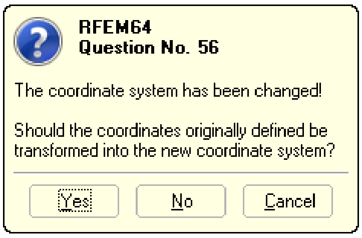


Figure 4.22: RFEM query

In the same way, you can convert node coordinates with the *Previous* reference node in relation to the origin.

Node coordinates

Node coordinates are defined in the coordinate system that you have previously set. When you model a 3D structure, the node is clearly defined by the coordinates X, Y and Z or the radius and angles. Coordinate parameters and table column titles are changing depending on the coordinate system.

When the model type is restricted to a 2D plate or wall, it is not possible to access all three input fields or columns.

To adjust *Lengths* and *Angles*, select **Units and Decimal Places** on the **Edit** menu, or use the corresponding button in the dialog box.

With the following procedure you can check if all nodes of a surface are placed in one plane: Select the relevant nodes and double click one of them to open the dialog box *Edit Node*. Coordinates data is filled only in those input fields whose values are conform for all selected nodes. If this is not the case, you can assign a uniform plane coordinate to the selected nodes now.

It is possible to import node coordinates from Excel spreadsheets (see chapter 11.5.6, page 488). Furthermore, you can determine node coordinates with the Formula Editor of RFEM (see chapter 11.6, page 491). In addition, you can take advantage of several model generators facilitating the input (see chapter 11.7.2, page 506).

To enter accurate, unrounded coordinates, select *Full Precision* in the dialog box *New Node*.

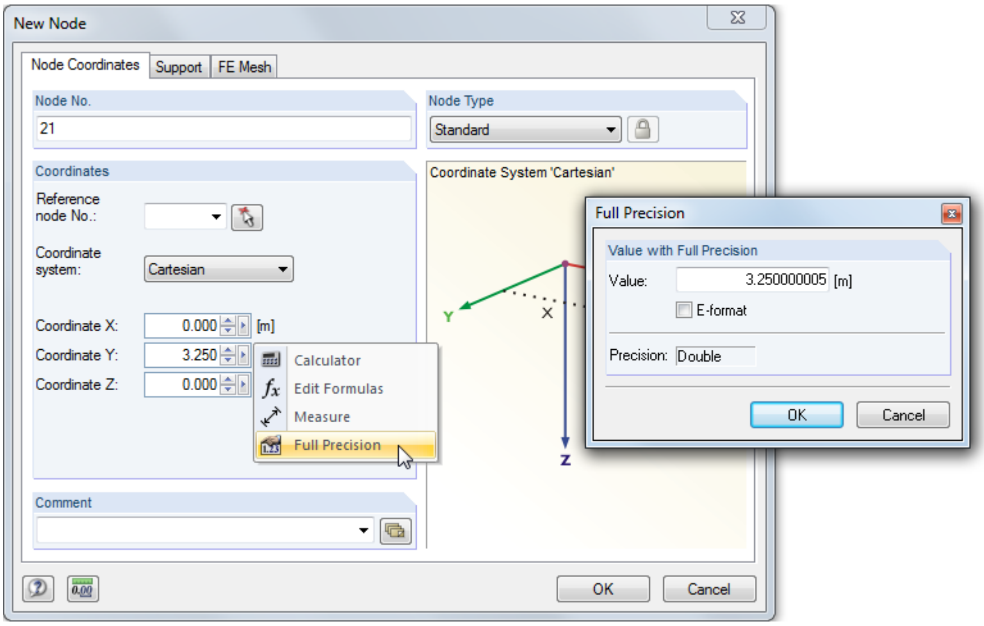


Figure 4.23: Context menu in the dialog box *New Node* and dialog box *Full Precision*

Comment

You can enter user-defined notes. Use the button [Apply Comment] to import saved comments (see chapter 11.1.4, page 421).

The comment *Generated* is displayed for nodes that are generated by RFEM when creating an intersection or a rotated surface. Click the button shown on the left, available in the dialog box and table, to "unlock" generated nodes, to make them accessible for modifications.

4.2 Lines

General description

The geometry of the model is defined by lines. They are essential for creating members, surfaces and solids. Every line is defined by a start and an end node. To define complex types of lines, intermediate nodes are additionally required.

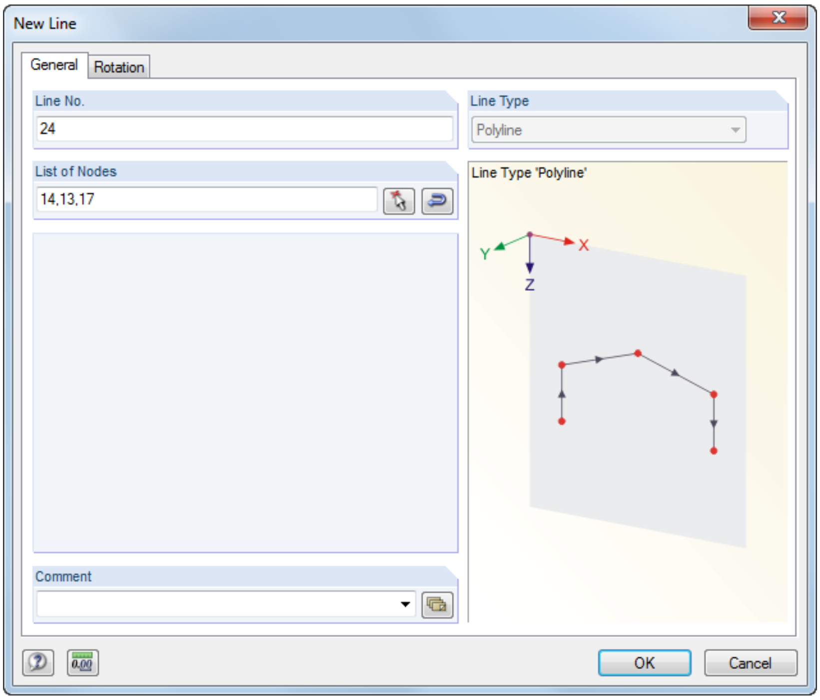
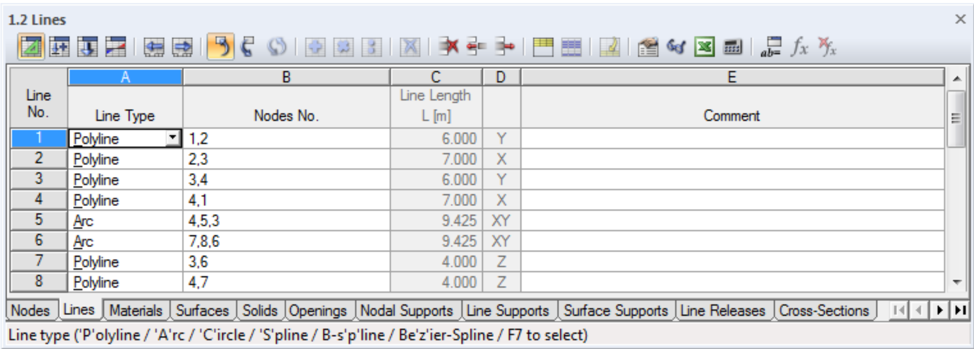


Figure 4.24: Dialog box *New Line*



1.2 Lines

Line No.	A Line Type	B Nodes No.	C Line Length L [m]	D	E Comment
1	Polyline	1,2	6.000	Y	
2	Polyline	2,3	7.000	X	
3	Polyline	3,4	6.000	Y	
4	Polyline	4,1	7.000	X	
5	Arc	4,5,3	9.425	XY	
6	Arc	7,8,6	9.425	XY	
7	Polyline	3,6	4.000	Z	
8	Polyline	4,7	4.000	Z	

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Cross-Sections

Line type ('P'olyline / 'A'rc / 'C'ircle / 'S'pline / B-s'p'line / Be'z'ier-Spline / F7 to select)

Figure 4.25: Table 1.2 *Lines*

The line number is assigned automatically in the dialog box *New Line* but can be changed in the input field. The order of the line numbering is not important.

To adjust the order of line numbers subsequently, select **Renumber** on the **Tools** menu (see chapter 11.4.18, page 478).

Line type

The following line types are available for selection on the menu as well as in the table list.

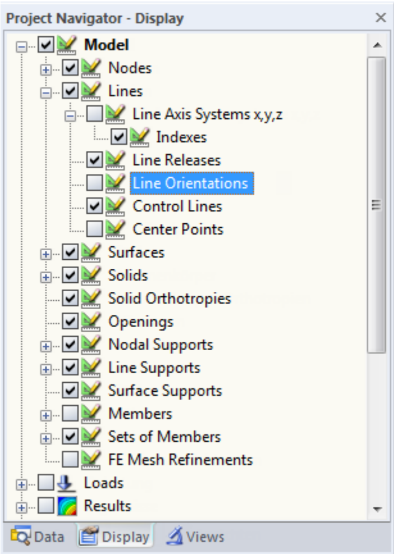
- Single line
- Polyline
- Arc
- Circle
- Ellipse
- Elliptical Arc
- Parabola
- Hyperbola
- Spline
- NURBS
- Trajectory
- On Surface

The different types of lines are described on the following pages.

Nodes

Every line is geometrically defined by a start and an end node. They define the orientation of a line that also affects the position of the line coordinate system. The nodes can be entered manually, graphically selected or redefined (see chapter 4.1, page 43). If a line requires control points or intermediate nodes, they are also included in the nodes list.

The display of line orientations can be activated in the *Display* navigator.



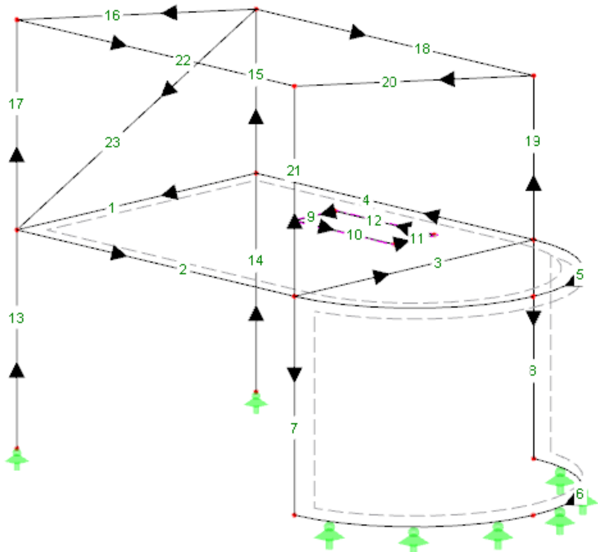


Figure 4.26: Activating *Line Orientations* in the *Display* navigator

The line orientation can be changed quickly in the graphic: Right-click the line and select *Reverse Line Orientation* in the context menu. The numbers of the start and end node will be interchanged.

The coordinate systems of lines can be activated in the *Display* navigator: Select *Model* and *Lines*, and tick *Line Axis Systems x, y, z* including *Indexes* (see Figure 4.96, page 101).

Line length

This table column shows the total length of the line.

Position

Table column **D** informs you about the global axis running parallel to the line or indicates the plane spanned by global axes where the line is lying. If there is no entry, the line is in an arbitrary spatial position.

Comment

You can enter user-defined notes. Use the button [Apply Comment] to import saved comments (see chapter 11.1.4, page 421). The comment *Generated* appears for lines created by RFEM (for example a pipe).

Line / Polyline

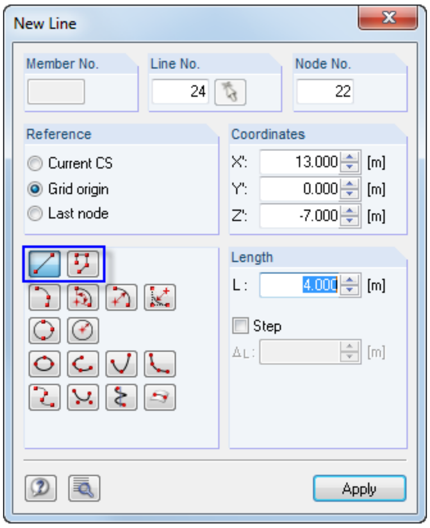


Figure 4.27: Graphical input dialog box *New Line*

Selecting the line entry in the pull-down menu opens the dialog box *New Line* shown in Figure 4.24 on page 49. The figure above shows the general dialog box for entering lines graphically. Use the list button in the toolbar to open it.

A "real" **line** is defined by only one start and only one end node. Such a line represents a direct connection between both nodes.

A **polyline** is a polygonal chain consisting of several straight line sections. Therefore, the intermediate nodes are listed in the dialog box (see Figure 4.24) in addition to the numbers of the start and end nodes. For reasons of simplified line management, "real" lines are handled as polylines.

When entering polylines graphically, already existing nodes, grid points or snap objects can be selected as definition nodes. It is also possible to set nodes freely into the work plane.

For example, if line loads or line supports are effective only for sections of a polyline, you can split the polyline into "real" lines subsequently: Right-click the polyline and select *Explode Polyline* in the context menu. You can also open the *Edit* menu where you point to *Model Data* and *Lines*, and then select *Explode Polyline*.

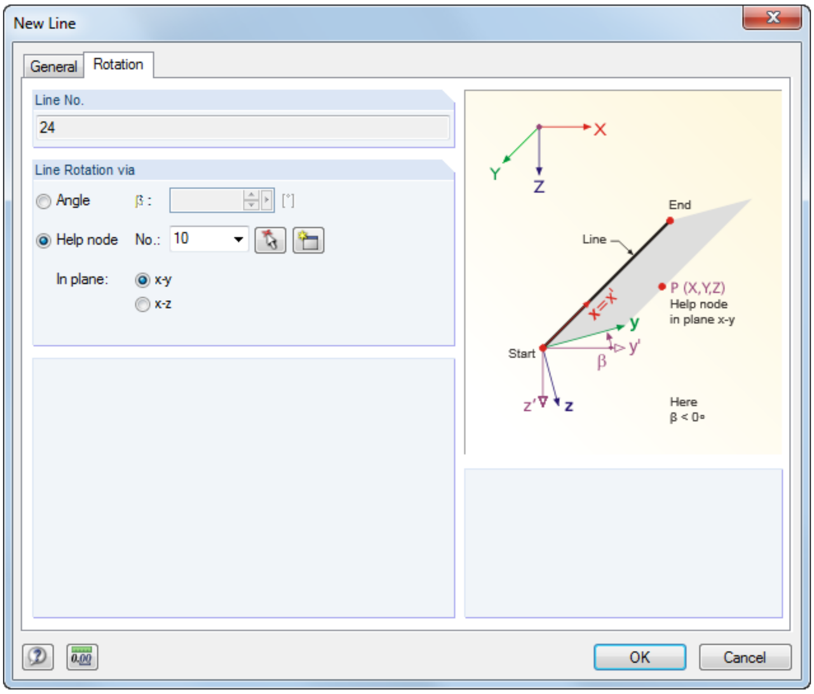


Figure 4.28: Dialog box *New Line*, tab *Rotation*

In the second tab of the dialog box you can define a *Rotation* of the line. Specify either an *Angle* β or a *Help node* to which the line axis y or z is aligned. The help node can be selected in the list or defined graphically. It is also possible to create a new node.

A rotation of the line may facilitate the input of line loads acting in local line direction. Line rotations do not affect surfaces or members because they have their own coordinate system.

The local line axis systems are shown in Figure 4.96 on page 101.

Arc

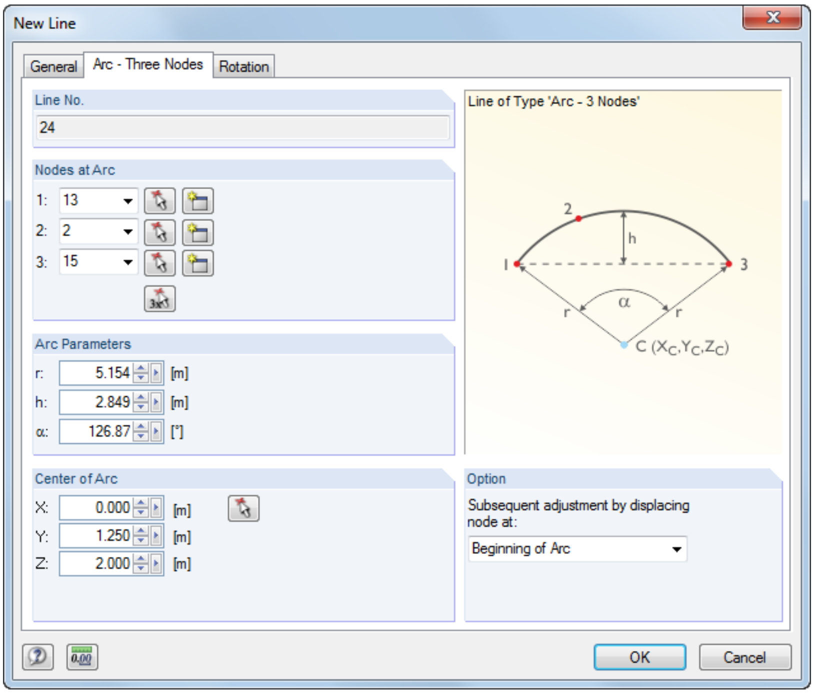
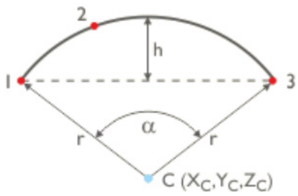


Figure 4.29: Dialog box *New Line*, tab *Arc*

An arc can be defined by the following parameters:

- three nodes
- center node, edge node and opening angle
- edge nodes and radius, opening angle or rise
- tangents and radius

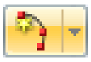


In the dialog section *Nodes at Arc*, you can define the start, intermediate and end node directly or select them graphically. You can also create new nodes. The node order is shown in the small dialog graphic.

Based on these three nodes, RFEM determines the *Arc Parameters* specified in the dialog section below. It is possible to change the radius *r*, the rise *h* and the opening angle α. The node coordinates will be adjusted accordingly.

The coordinates of the arc center resulting from the arc nodes or parameters are displayed in the dialog section *Center of Arc*. When you change data manually or select nodes graphically with the [↖] button, the coordinates of nodes will be adjusted as well.

In the list *Subsequent adjustment by displacing node at* you can define the node for which you want to change the coordinates.



When you define the arc graphically by using the toolbar button *Arc via Three Nodes*, you can select or create the nodes directly in the graphic.

If you select one of the other input options shown in Figure 4.30 and Figure 4.31 on the left, you have to select two nodes first. Then, when the second node is defined, another dialog box appears (shown on the right) where you define the arc parameters.

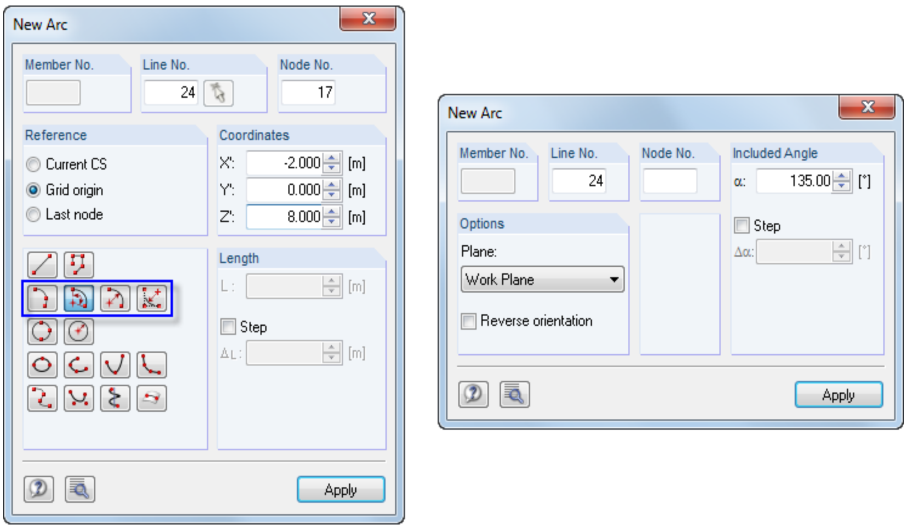
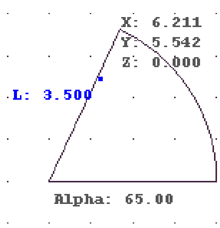


Figure 4.30: Dialog box *New Arc*, definition type *Arc via Center Node, Edge Node and Angle*



In the dialog section *Options* (in Figure 4.30 and Figure 4.31 on the right), you can select the plane of the arc from the list. Define the *Included Angle* α directly in the graphic or enter the angle manually and then click the [Apply] button.

To adjust an already defined arc, double-click its arc line. The dialog box *Edit Line* opens where you can modify entries in the dialog tab *Arc - Three Nodes* (see Figure 4.29, page 52).

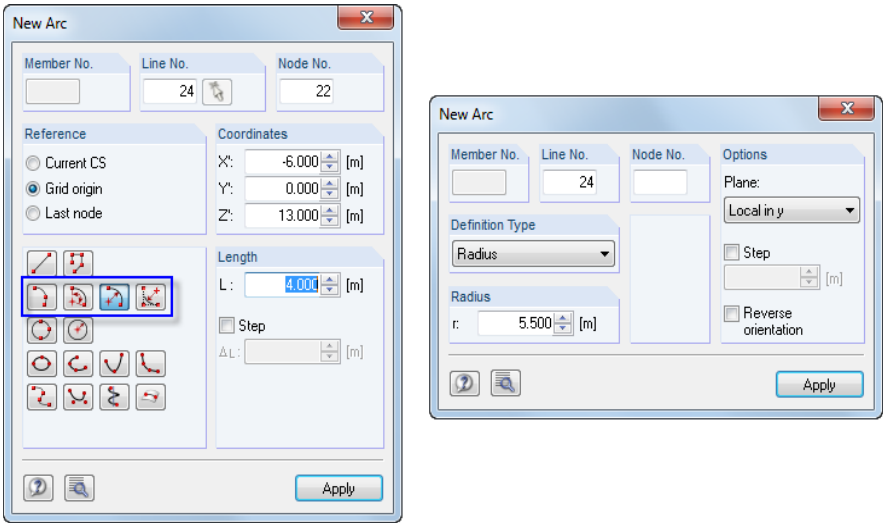
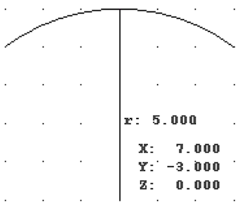


Figure 4.31: Dialog box *New Arc*, definition type *Arc via Edge Nodes and Radius, Angle or Rise*



In the dialog section *Definition Type* (right box), select the appropriate arc parameter from the list. Then, set the arc directly into the graphic, or enter it manually and click the [Apply] button.

In the input field *Step* you can enter the spacing by which the pointer snaps on when dragging the arc radius, angle or rise.

The orientation of the circular arc can be adjusted by ticking the check box *Reverse orientation*, determining whether the arc is placed to the "right" or "left" of both nodes.

Circle

A circle can be defined by the following parameters:

- three nodes
- center point and radius.

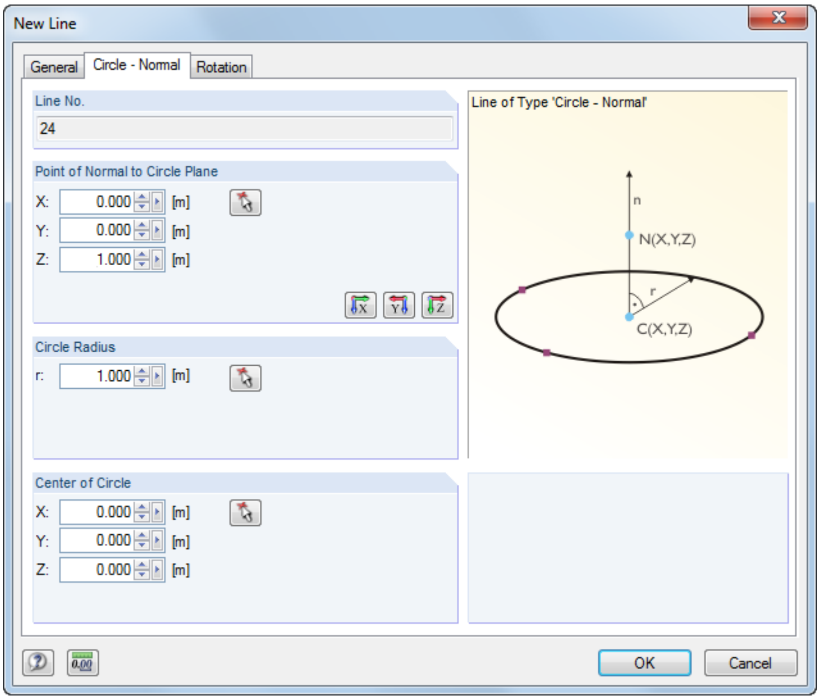


Figure 4.32: Dialog box *New Line*, tab *Circle - Normal*

You can enter the *Circle Radius* and the coordinates for the *Center of Circle* manually or graphically by using the [↖] button. The *Point of Normal to Circle Plane* determines the plane in which the circle is generated. Use the three buttons of the dialog section to select one of the global axes.

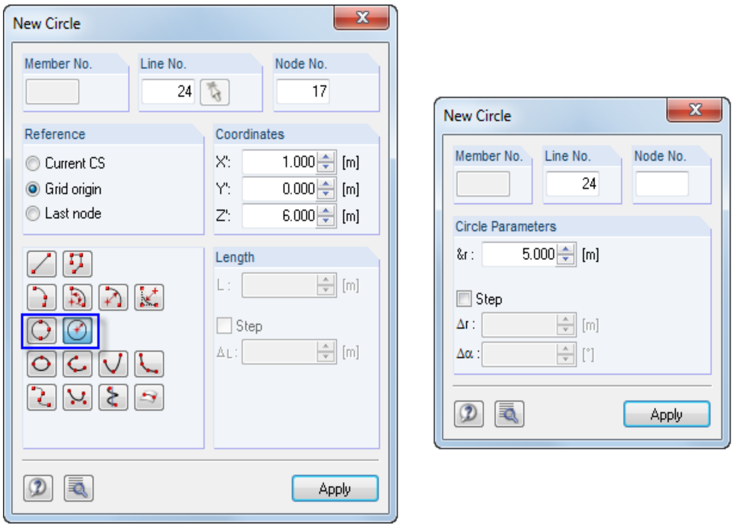
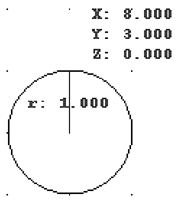


Figure 4.33: Dialog box *New Circle*, definition type *Center and Radius*



When you define the circle graphically by using one of the toolbar buttons, you can select or create the three nodes as well as the center and radius directly in the graphic.

Ellipse

To define an ellipse, three nodes are required.

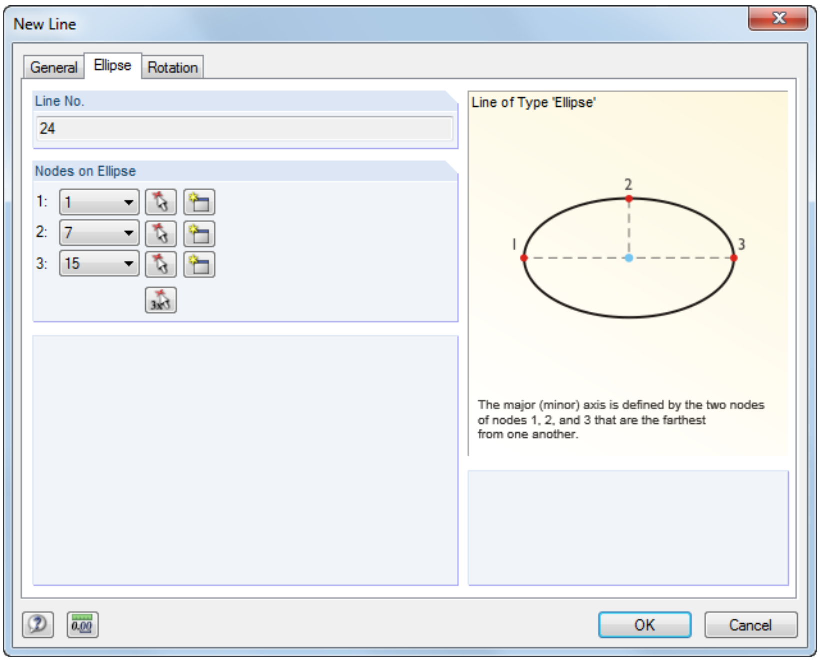
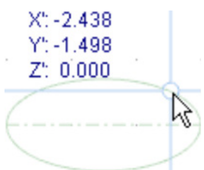


Figure 4.34: Dialog box *New Line*, tab *Ellipse*



The ellipse is generated by the *Nodes on Ellipse*: The largest distance between the three entered nodes is assumed to be the principal axis of the ellipse.

When you use the toolbar button to define the ellipse graphically, you can set it directly by selecting three nodes in the work plane.

Elliptical arc / parabola / hyperbola

The following curves of conic sections can be defined:

- Elliptical arc
- Parabola
- Hyperbola

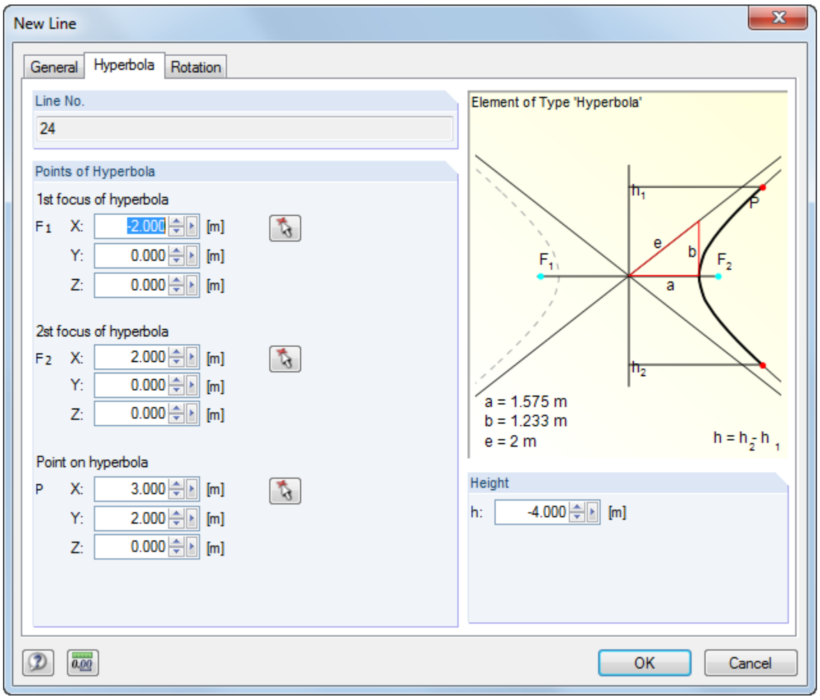
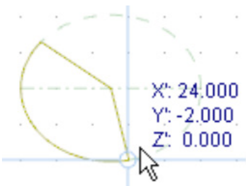


Figure 4.35: Dialog box *New Line*, tab *Hyperbola*



Enter the curve parameters (focus, angle, axis rotation etc.) manually in the corresponding tab of the dialog box *New Line*. You can also define them graphically.

When you enter line data graphically by using one of the toolbar buttons, you can define the curve parameters directly in the graphic.

Spline

Splines are used to model any kind of curves. Enter a spline line graphically by selecting determinant nodes of the curved line one after the other, or create nodes by mouse-click.

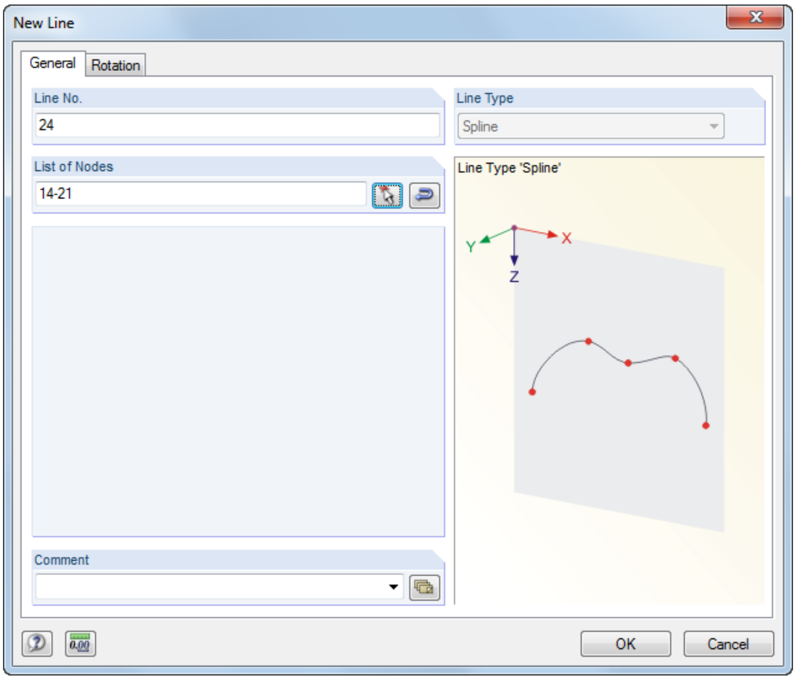


Figure 4.36: Dialog box *New Line* - line type *Spline*

NURBS

NURBS (*N*on-*U*niform *R*ational *B*-*S*plines) are required for modeling free form surfaces. NURBS are splines whose control points are not placed on the curve itself. Usually, these lines are entered graphically by selecting control points one after the other or by generating points by mouse-click.

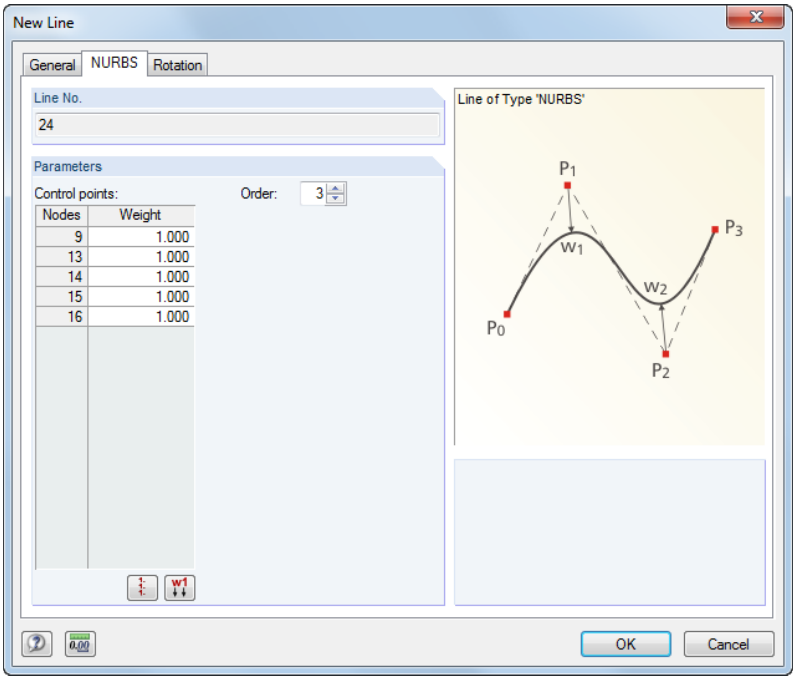


Figure 4.37: Dialog box *New Line* - line type *NURBS*

Trajectory curve

Use trajectory curves to create helical lines. Usually, they are entered graphically by using the toolbar button shown on the left. The following dialog box appears:

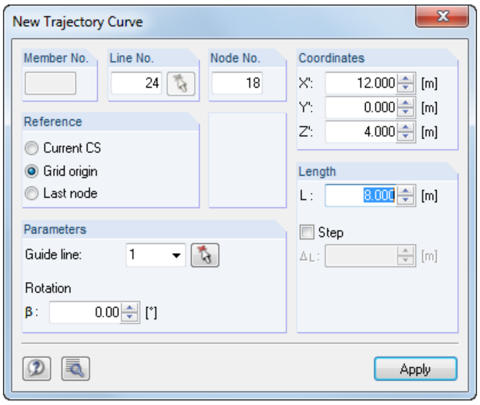
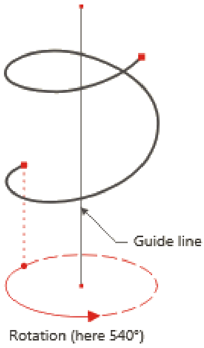


Figure 4.38: Dialog box *New Trajectory Curve*



First, define the *Coordinates* of the line start. Then, the dialog section *Parameters* is enabled where you can specify the total *Rotation* of the helix.

Define the *Coordinates* of the line end graphically, or enter them manually and click the [Apply] button. Alternatively, you can use the input field *Length*. Finally, the coordinates of the line end will be determined considering the specified rotation in the work plane.

To adjust an already defined trajectory curve, double-click its trajectory line. The dialog box *Edit Line* opens where you can modify entries in the dialog tab *Trajectory*.

Line on surface

In general, lines in plane surfaces are recognized automatically as integrated objects so that the line type *Polyline* is usually sufficient. However, to set a line on a curved surface, use the line type *Line On Surface*. You can also use this option to insert lines in plane surfaces which are not defined parallel to the global axes - without creating a new user-defined coordinate system.

The input dialog box is similar to the box of a polyline (see Figure 4.24, page 49).

Enter the start and end nodes of the line in the dialog box or select them graphically. To set nodes directly on a curved surface, use the floating dialog box *New Line of Type 'On Surface'*, but take care that the surface has been previously selected and the model display option *Solid* or *Solid Transparent* is active. RFEM will generate nodes of the type *On Surface*.

The surface where the line is integrated is defined in the second tab *On Surface* of the *New Line* dialog box. You can also check the parameters δ_1 and δ_2 of the start and end node (see Figure 4.14, page 44), but it is not possible to change them.

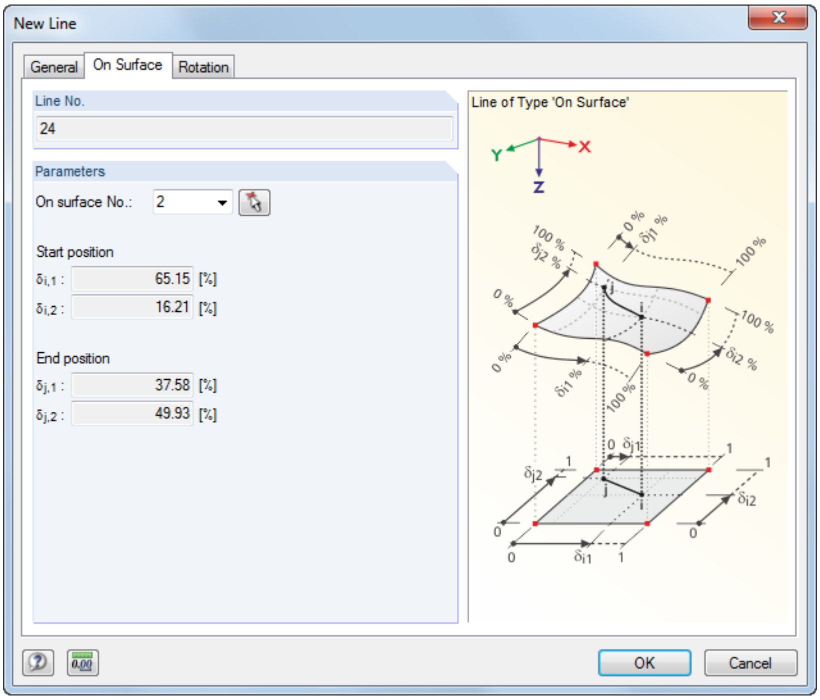
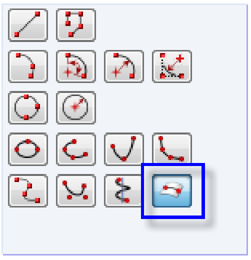


Figure 4.39: Dialog box *New Line*, tab *On Surface*



If you choose the graphical line input, using the toolbar button [Line on Surface], you can place the nodes directly on the curved surfaces. Please note that the display option *Wireframe Display Model* is not appropriate for the input.

4.3 Materials

General description

Materials are required to define surfaces, cross-sections and solids. The material properties affect the stiffnesses of model objects.

A *Color* is assigned to each material. Colors are used by default in the rendered model for the representation of objects (see chapter 11.1.9, page 427).

For new models RFEM presets the two materials that were last used.

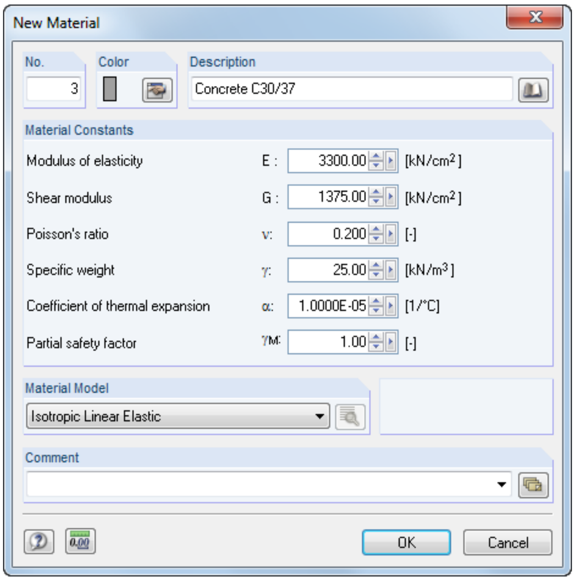
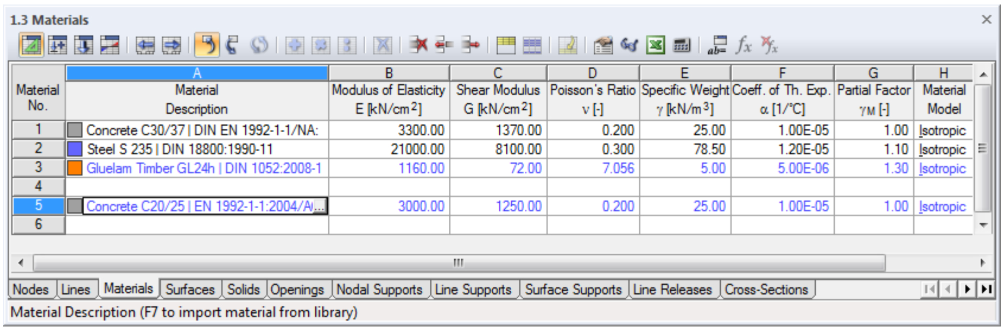


Figure 4.40: Dialog box *New Material*



Material No.	A Material Description	B Modulus of Elasticity E [kN/cm²]	C Shear Modulus G [kN/cm²]	D Poisson's Ratio ν [-]	E Specific Weight γ [kN/m³]	F Coeff. of Th. Exp. α [1/°C]	G Partial Factor γM [-]	H Material Model
1	Concrete C30/37 | DIN EN 1992-1-1/NA:	3300.00	1370.00	0.200	25.00	1.00E-05	1.00	Isotropic
2	Steel S 235 | DIN 18800:1990-11	21000.00	8100.00	0.300	78.50	1.20E-05	1.10	Isotropic
3	Gluelam Timber GL24h | DIN 1052:2008-1	1160.00	72.00	7.056	5.00	5.00E-06	1.30	Isotropic
4								
5	Concrete C20/25 | EN 1992-1-1:2004/A|...	3000.00	1250.00	0.200	25.00	1.00E-05	1.00	Isotropic
6								

Figure 4.41: Table 1.2 *Materials*

Material description

Any name can be chosen as *Description* of the material. When the entered name corresponds to an entry of the library, RFEM will import the material properties. The import of materials from the library is described later.

Modulus of elasticity E

The modulus of elasticity describes the ratio between normal stress and strain.

To adjust the settings for *Materials*, click **Units and Decimal Places** on the **Edit** menu, or use the corresponding button.

Shear modulus G

The shear modulus G is the second parameter used to describe the elastic behavior of a linear, isotropic and homogenous material.

The shear modulus of the materials listed in the library is calculated according to Equation 4.1 from the modulus of elasticity E and the Poisson's ratio ν. Thus, a symmetrical stiffness matrix is ensured for isotropic materials. The shear modulus values determined in this way may slightly deviate from the specifications in the Eurocodes.

Poisson's ratio ν

The following relation exists between elastic and shear modulus and the Poisson's ratio ν.

$$E = 2G(1+\nu)$$

Equation 4.1

When you define the properties of an isotropic material manually, RFEM will determine automatically the Poisson's ratio from the values of elastic and shear modulus (respectively shear modulus from modulus of elasticity and Poisson's ratio).

Generally, the Poisson's ratio of isotropic materials is between 0.0 and 0.5. Therefore, for a value higher than 0.5 (for example rubber) we assume that the material is not isotropic. Before the calculation starts, a query appears asking if you want to use an orthotropic material model.

Specific weight γ

The specific weight γ describes the weight of the material per volume unit.

The specification is especially important for the load type 'self-weight'. The automatic self-weight of the model is determined by the specific weight and the cross-sectional areas of the used members or surfaces and solids.

Coefficient of thermal expansion α

The coefficient describes the linear correlation between changes in temperature and axial strains (elongation due to heating, shortening due to cooling).

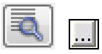
The value is important for the load types 'temperature change' and 'temperature differential'.

Partial safety factor γ_M

The value describes the safety factor for the material resistance. Therefore the index *M* is used. Use the factor γ_M to reduce the stiffness for calculations according to second-order and large deformation analysis (see chapter 7.3.1, page 267).

Do not confuse the factor γ_M with the safety factors for the determination of design internal forces. The partial safety factors γ on the action side take part in combining load cases for load and result combinations.

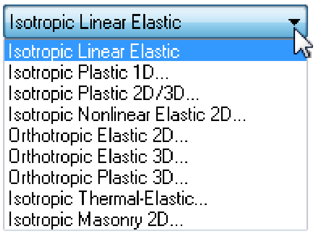
Material model



Nine material models are available for selection in the list. Use the [Details] button in the dialog box or table to access dialog boxes where you can define the parameters of the selected model.

If the add-on module **RF-MAT NL** is not licensed, you can use only the material models *Isotropic Linear Elastic* and *Orthotropic Elastic 2D/3D*.

Isotropic linear elastic



The linear-elastic stiffness properties of the material do not depend on directions. They can be described according to Equation 4.1. The following conditions apply:

- $E > 0$
- $G > 0$
- $-1 \leq \nu < 0.5$ (only for surfaces and solids, not bounded above for members)

The elasticity matrix (inverse of stiffness matrix) for surfaces is the following:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & & & \\ -\frac{\nu}{E} & \frac{1}{E} & & & \\ & & \frac{1}{G} & & \\ & & & \frac{1}{G} & \\ & & & & \frac{1}{G} \end{bmatrix} \cdot \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{Bmatrix}$$

Equation 4.2

Isotropic plastic 1D

If you have set the *3D* model type (see Figure 12.23, page 554), you can define the plastic properties of the isotropic material in a dialog box. RFEM will take into account these properties for member elements, for example for plastic calculations of a kinematic chain.

Only if a sufficient number of FE nodes is created on the member, the nonlinear material behavior will be determined correctly in the calculation. The following possibilities are available:

- Dialog box *Divide Member Using n Intermediate Nodes* (see Figure 11.91, page 469), method of division: *without dividing it*
- Dialog box *FE Mesh Settings* (see Figure 7.10, page 258), option *Use division for straight members* with a *Minimum number of member divisions* of 10

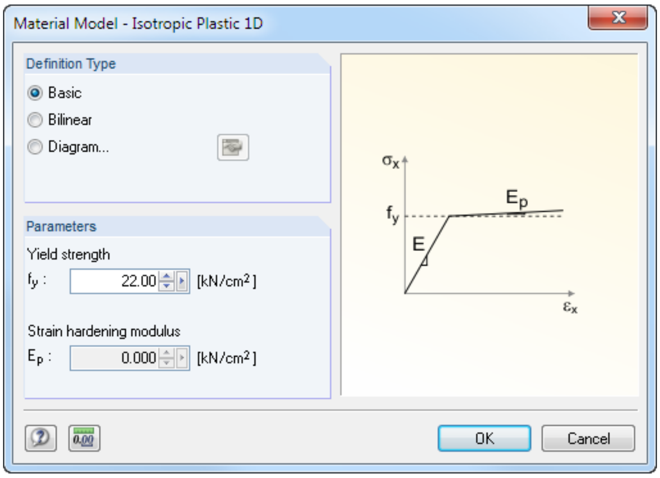


Figure 4.42: Dialog box *Material Model - Isotropic Plastic 1D*

Define the parameters of the ideally or bilinearly plastic material. You can also define a stress-strain *Diagram* to represent the material behavior close to reality.

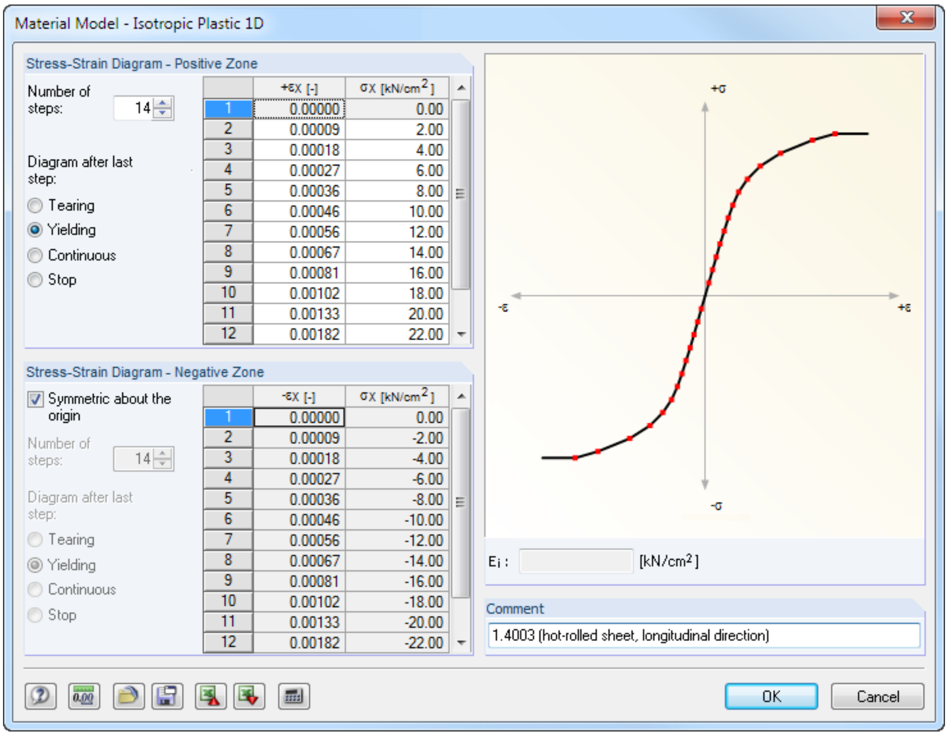


Figure 4.43: Dialog box *Material Model - Isotropic Plastic 1D*, stress-strain diagram

The material properties can be defined separately for the *Positive* and the *Negative Zone*. The *Number of steps* determines the number of definition points respectively available. Enter the strains ε and the corresponding normal stresses σ in the two lists.

You have several options for the *Diagram after last step*: *Tearing* for material failure when exceeding a certain stress, *Yielding* for restricting the transfer of a maximum stress, *Continuous* as in the last step or *Stop* for restricting to a maximum allowable deformation.

It is also possible to import parameters from an [Excel] worksheet.

Watch the dynamic graphic to the right to check the material properties. The dialog field E_i below the graphic shows the modulus of elasticity for the current definition point.

Use the [Save As] button in the dialog box to store the stress-strain diagram so that you can apply it to different models. To import user-defined diagrams, click the [Load] button (see figure below).

The check box *Activate shear stiffness of members (cross-sectional areas A_y, A_z)* in the dialog box *Calculation Parameters* (see Figure 7.22, page 271) is without effect for members with isotropic elastic-plastic material properties. This material model uses the beam theory according to EULER BERNOULLI where shear distortions are neglected.

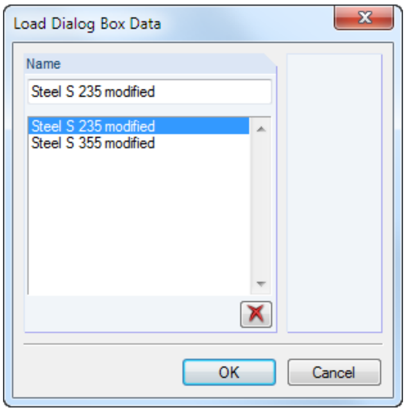


Figure 4.44: Dialog box *Load Dialog Box Data*

Isotropic plastic 2D/3D

This material model shows an isotropic material behavior in the elastic zone. The plastic zone is based on the yielding according to VON MISES (J2 plasticity) with a user-defined *Yield strength* of the equivalent stress for surfaces and solids.

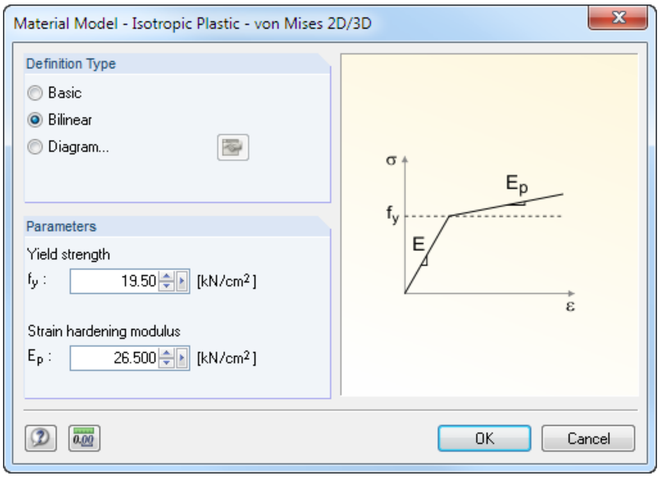


Figure 4.45: Dialog box *Material Model - Isotropic Plastic - von Mises 2D/3D*

Define the parameters of the ideally or bilinearly plastic material. You can also define a stress-strain *Diagram* to represent the material behavior close to reality (see Figure 4.43). The same relations apply for tension and compression.

The yielding is the following:

$$\sigma_v = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$$

for 2D elements

Equation 4.3

$$\sigma_v = \frac{1}{\sqrt{2}} \sqrt{\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + \left(\sigma_x - \sigma_z\right)^2 + 6\left(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2\right)}$$

for 3D elements

Equation 4.4

For plastic material properties calculations are carried out iteratively and with load increments (see chapter 7.3, page 274). If the stress is exceeded in a finite element, the modulus of elasticity will be reduced there and a new calculation run starts. The process is repeated until a convergence is reached. When the calculation is done, stiffness reductions can be checked also graphically (see chapter 9.3.2, page 349).

When evaluating results, it is recommended to use the smoothing option *Constant on Elements* (see Figure 9.31, page 363). The setting ensures that the defined stress limit is displayed as maximum in the results panel. Plastic effects can be considered only element by element in the calculation. For the remaining smoothing options, however, RFEM interpolates or extrapolates the results. This may lead to distortions that are more or less distinct depending on the mesh.

Isotropic nonlinear elastic 2D

This material model is similar to the model *Isotropic Plastic 2D/3D* described above but no energy is delivered to the model (conservative analysis). As the same stress-strain relations apply for loading and relief of load, no permanent plastic distortions are available after a relief.

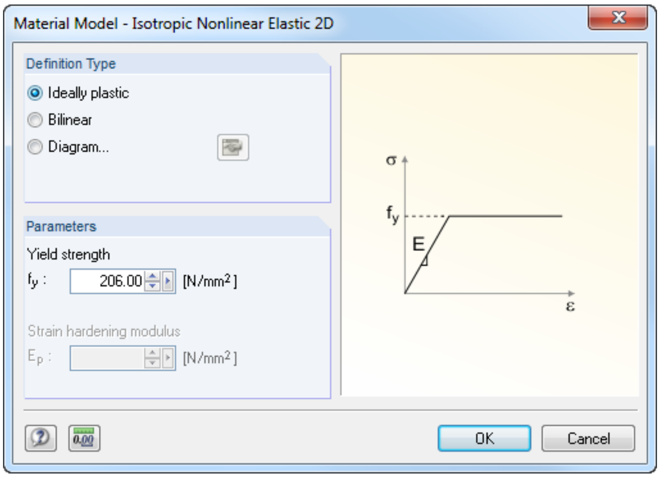


Figure 4.46: Dialog box *Material Model - Isotropic Nonlinear Elastic 2D*

The elasticity matrix is damped isotropically in order that the stress-strain relations of the equivalent stresses and distortions according to VON MISES are fulfilled. They are defined by the following conditions:

$$\sigma_v = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2}$$

$$\varepsilon_v = \frac{\sigma_v}{E}$$

Equation 4.5

In the *Diagram*, it is possible to define the stress-strain relations separately for the tension and the compression zone (see Figure 4.43).

Generally, many iterations are required for this material model until convergence is reached. Therefore, it is recommended to specify a minimum value of 300 as the *Maximum number of iterations* among the calculation parameters (see chapter 7.3.3, page 271).

Orthotropic elastic 2D

You can define stiffness properties that appear differently in both surface directions x and y. In this way, you can model for example ribbed floors or stress directions of reinforced ceilings. The surface axes x and y are perpendicular to each other in the surface plane (see Figure 4.73, page 83).

The RFEM 4 material models *Orthotropic* and *Orthotropic Extra* are converted into this model.

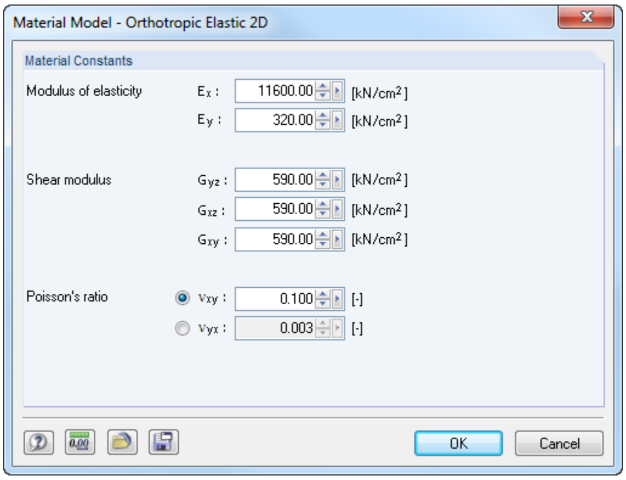


Figure 4.47: Dialog box *Material Model - Orthotropic Elastic 2D*

With this material model you can assign an orthotropic property globally to all surfaces consisting of a particular material. Alternatively, it is possible to define the parameters for each surface individually (see chapter 4.12 *Orthotropic Surfaces*, page 112).

An orthotropic elastic material is characterized by the moduli of elasticity E_x and E_y, the shear moduli G_{yz}, G_{xz} and G_{xy} as well as the Poisson's ratios ν_{xy} and ν_{yx}. The elasticity matrix (inverse of stiffness matrix) is defined as follows:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{xy}}{E_x} & & & \\ -\frac{\nu_{yx}}{E_y} & \frac{1}{E_y} & & & \\ & & \frac{1}{G_{xy}} & & \\ & & & \frac{1}{G_{yz}} & \\ & & & & \frac{1}{G_{xz}} \end{bmatrix} \cdot \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{Bmatrix}$$

Equation 4.6

The following correlation exists between principal Poisson's ratio ν_{xy} and secondary Poisson's ratio ν_{yx}:

$$\frac{\nu_{yx}}{E_y} = \frac{\nu_{xy}}{E_x}$$

Equation 4.7

The following conditions must be met for a positively definite stiffness matrix:

- $E_x > 0; \quad E_y > 0$
- $G_{yz} > 0; \quad G_{xz} > 0; \quad G_{xy} > 0$
- $\left|\nu_{xy}\right| < \sqrt{\frac{E_x}{E_y}}$

Orthotropic elastic 3D

In a three-dimensional material model, you can define elastic stiffnesses separately in all directions of the solid. In this way, you can represent for example strength properties of wood-based materials.

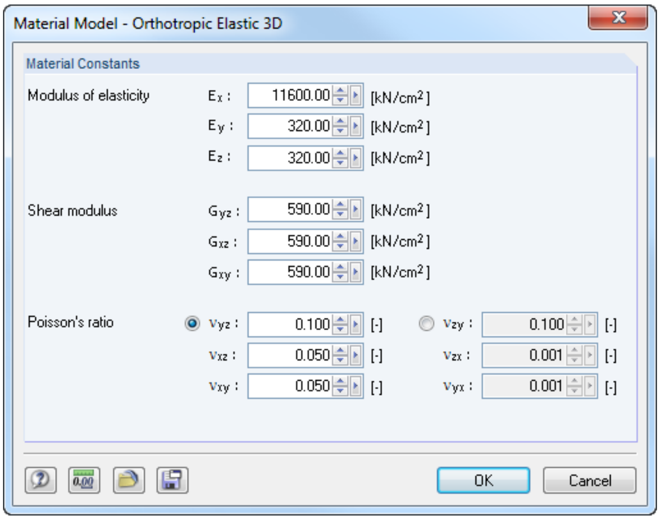


Figure 4.48: Dialog box *Material Model - Orthotropic Elastic 3D*

The elasticity matrix is defined as follows:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{yz} \\ \gamma_{xz} \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{xy}}{E_x} & -\frac{\nu_{xz}}{E_x} & & & \\ -\frac{\nu_{yx}}{E_y} & \frac{1}{E_y} & -\frac{\nu_{yz}}{E_y} & & & \\ -\frac{\nu_{zx}}{E_z} & -\frac{\nu_{zy}}{E_z} & \frac{1}{E_z} & & & \\ & & & \frac{1}{G_{yz}} & & \\ & & & & \frac{1}{G_{xz}} & \\ & & & & & \frac{1}{G_{xy}} \end{bmatrix} \cdot \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{yz} \\ \tau_{xz} \\ \tau_{xy} \end{Bmatrix}$$

Equation 4.8

The following correlations exist between the principal Poisson's ratios ν_{yz}, ν_{xz}, ν_{xy} and the secondary Poisson's ratios ν_{zy}, ν_{zx}, ν_{yx}:

$$\frac{\nu_{zy}}{E_z} = \frac{\nu_{yz}}{E_y} \;;\quad \frac{\nu_{zx}}{E_z} = \frac{\nu_{xz}}{E_x} \;;\quad \frac{\nu_{yx}}{E_y} = \frac{\nu_{xy}}{E_x}$$

Equation 4.9

The following conditions must be met for a positively definite stiffness matrix:

- $E_x > 0;\ \ E_y > 0;\ \ E_z > 0$
- $G_{yz} > 0;\ \ G_{xz} > 0;\ \ G_{xy} > 0$
- $|\nu_{yz}| < \sqrt{\frac{E_y}{E_z}} \;;\quad |\nu_{xz}| < \sqrt{\frac{E_x}{E_z}} \;;\quad |\nu_{xy}| < \sqrt{\frac{E_x}{E_y}}$
- $1 - \nu_{yz}^2 \frac{E_z}{E_y} - \nu_{xz}^2 \frac{E_z}{E_x} - \nu_{xy}^2 \frac{E_y}{E_x} - 2\frac{E_z}{E_x}\nu_{yz}\nu_{xz}\nu_{xy} > 0$

Orthotropic plastic 3D

The material model according to TSAI-WU unifies plastic with orthotropic properties. In this way, you can enter special modelings of materials with anisotropic characteristics like plastics or timber. When the material is yielding, stresses remain constant. A redistribution is carried out according to the stiffnesses available in the individual directions.

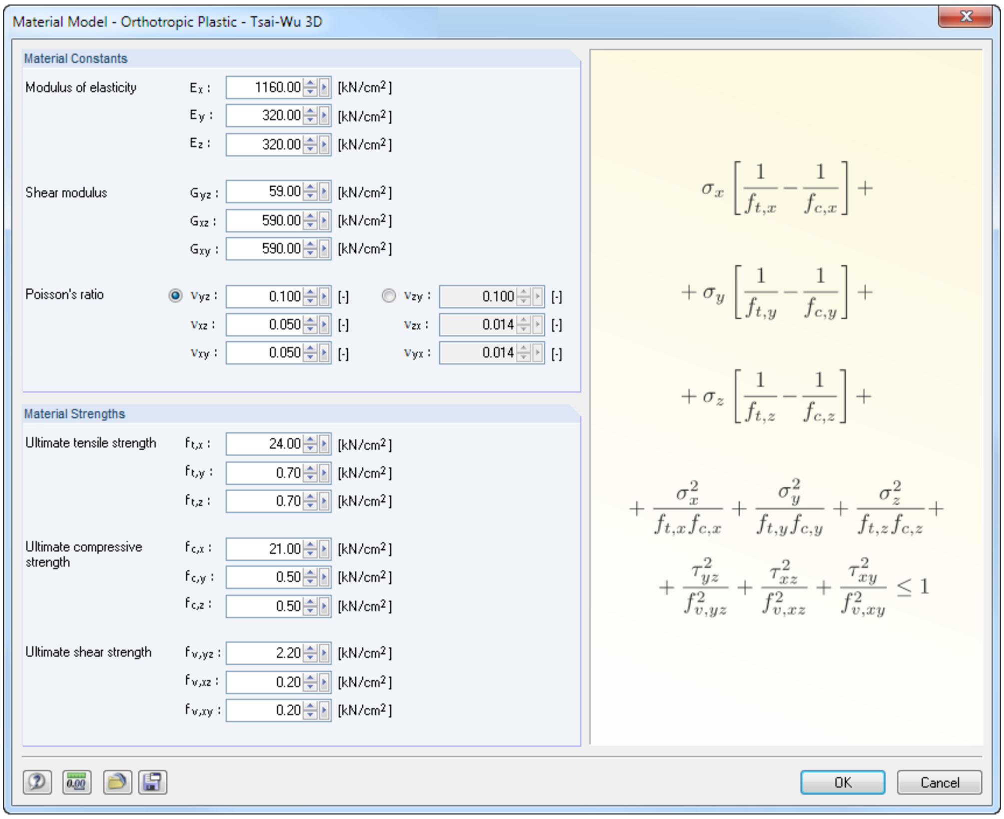


Figure 4.49: Dialog box *Material Model - Orthotropic Plastic - Tsai-Wu 3D*

The elastic zone corresponds to the material model *Orthotropic Elastic 3D* (see above). For the plastic zone the yielding according to TSAI-WU applies:

$$f_y(\sigma)=\sigma_x\left(\frac{1}{f_{t,x}}-\frac{1}{f_{c,x}}\right)+\sigma_y\left(\frac{1}{f_{t,y}}-\frac{1}{f_{c,y}}\right)+\sigma_z\left(\frac{1}{f_{t,z}}-\frac{1}{f_{c,z}}\right)+\frac{\sigma_x^2}{f_{t,x}f_{c,x}}+\frac{\sigma_y^2}{f_{t,y}f_{c,y}}+\frac{\sigma_z^2}{f_{t,z}f_{c,z}}+\frac{\tau_{yz}^2}{f_{v,yz}^2}+\frac{\tau_{xz}^2}{f_{v,xz}^2}+\frac{\tau_{xy}^2}{f_{v,xy}^2}$$

where $f_{t,x}$, $f_{t,y}$, $f_{t,z}$ Plastic ultimate tensile strength in direction x, y or z
$f_{c,x}$, $f_{c,y}$, $f_{c,z}$ Plastic ultimate compressive strength in direction x, y or z
$f_{v,yz}$, $f_{v,xz}$, $f_{v,xy}$ Plastic shear strength in direction yz, xz or xy

Equation 4.10

All strengths must be defined positively.

The stress criteria can be imagined as a surface in the shape of an ellipse within a six-dimensional space of stresses. If one of the three stress components is applied as constant value, the surface can be projected on a three-dimensional stress space (see figure below):

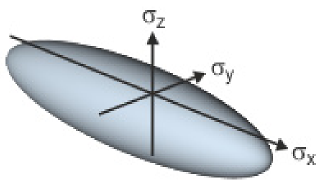


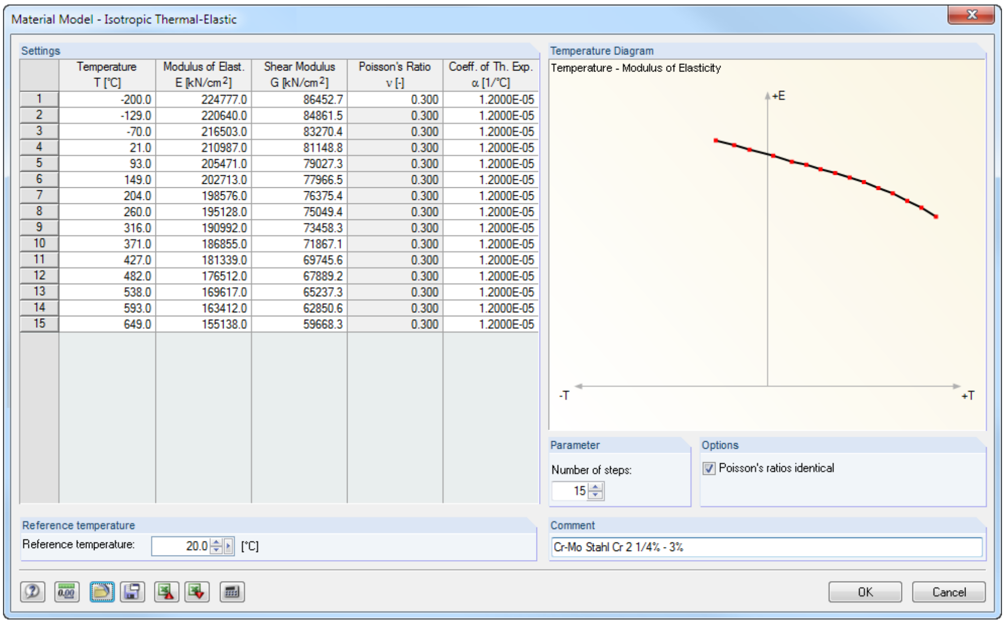
Figure 4.50: Projection of yielding surfaces for normal stresses according to TSAI-WU

When the value for $f_y(\sigma)$ according to Equation 4.10 is lower than 1, stresses lie in the elastic zone. The plastic zone is reached as soon as $f_y(\sigma) = 1$. Values higher than 1 are not allowed. The model behavior is ideal-plastic, which means no strengthening takes place.

Equation 4.10 is only valid for the local FE coordinate system. If it is not conform to the solid's coordinate system used for the stress output in RFEM, the values must be transformed accordingly.

Isotropic thermal-elastic

Temperature-dependent stress-strain properties of an elastic isotropic material can be defined in a diagram or imported from [Excel]. These properties will be considered for member and surface elements subjected to thermal load (changes or differences in temperature).



	Temperature T [°C]	Modulus of Elast. E [kN/cm²]	Shear Modulus G [kN/cm²]	Poisson's Ratio ν [-]	Coeff. of Th. Exp. α [1/°C]
1	-200.0	224777.0	86452.7	0.300	1.2000E-05
2	-129.0	220640.0	84861.5	0.300	1.2000E-05
3	-70.0	216503.0	83270.4	0.300	1.2000E-05
4	21.0	210987.0	81148.8	0.300	1.2000E-05
5	93.0	205471.0	79027.3	0.300	1.2000E-05
6	149.0	202713.0	77966.5	0.300	1.2000E-05
7	204.0	198576.0	76375.4	0.300	1.2000E-05
8	260.0	195128.0	75049.4	0.300	1.2000E-05
9	316.0	190992.0	73458.3	0.300	1.2000E-05
10	371.0	186855.0	71867.1	0.300	1.2000E-05
11	427.0	181339.0	69745.6	0.300	1.2000E-05
12	482.0	176512.0	67889.2	0.300	1.2000E-05
13	538.0	169617.0	65237.3	0.300	1.2000E-05
14	593.0	163412.0	62850.6	0.300	1.2000E-05
15	649.0	155138.0	59668.3	0.300	1.2000E-05

Figure 4.51: Dialog box *Material Model - Isotropic Thermal-Elastic*

The *Reference temperature* defines stiffnesses for the members or surfaces that have no temperature loads. For example, when a reference temperature of 300 °C is set, RFEM applies the reduced elastic modulus of this point of the temperature curve to all members and surfaces.

With the setting in the dialog section *Options* you decide if *identical Poisson's ratios* are applied to the complete temperature diagram. Remove the checkmark to access the table column *Poisson's Ratio* when you want to enter individual entries.

Use the [Load] button to import predefined temperature diagrams for different ferrous alloys (see Figure 4.44, page 64).

Click the [Save] button to save user-defined temperature diagrams so that you can use them for other models.

Isotropic masonry 2D

Use this material model to take into account masonry walls not able to bear tension forces but reacting with crack formation.

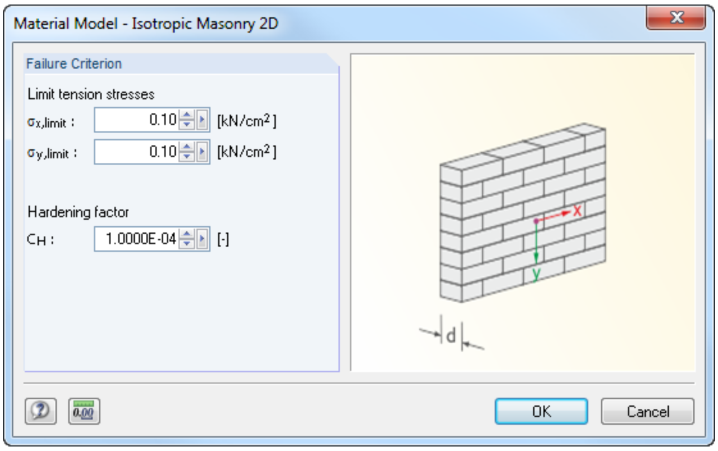


Figure 4.52: Dialog box *Material Model - Isotropic Masonry 2D*

The dialog box allows for the definition of the *Limit tension stresses* in direction of the surface axes x and y, which means parallel and perpendicular to the interstices of support. Then, when calculating data, RFEM finds out by several iterations which finite elements become stress-free due to the failure criterion.

When the limit tension stress is set to zero, RFEM applies a limit value of 10^{-11} N/mm² in calculations for reasons of stability. Thus, minor tensile stresses are not completely excluded.

If numerical problems occur during the calculation, you can try to reach convergence by increasing the *Hardening factor* C_H.

If the masonry material has already been defined in the library before you open the dialog box *Material Model*, the following limit values are preset:

Standard	$\sigma_{x,limit}$	$\sigma_{y,limit}$
DIN 1053-100	f_{x2} tensile strength parallel to interstice of support	0
EN 1996-1-1	f_{xk2} tensile strength parallel to interstice of support	f_{xk1} tensile strength perpendicular to interstice of support

Table 4.1: Limit tension stresses according to masonry standards

Material Library

The properties of many materials are stored in a comprehensive database that can be extended.

Open the library

To access the library, click the [Material Library] button (cf. Figure 4.40, page 60) in the dialog box *New Material*. You can open the database also in table 1.3 *Materials* (cf. Figure 4.41, page 60): Place the cursor into table column A and click the button [...] shown on the left, or use the function key [F7] on the keyboard.

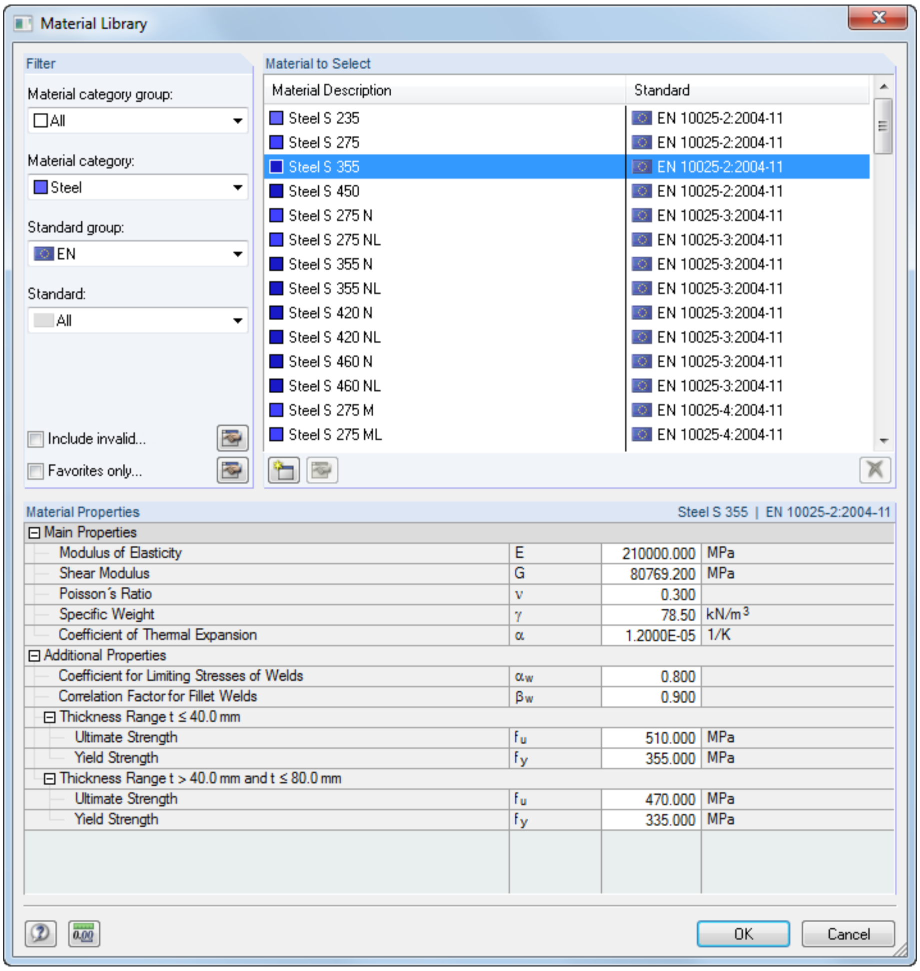


Figure 4.53: Dialog box *Material Library*

Select a material from the list *Material to Select* and check the corresponding parameters in the lower part of the dialog box. Click [OK] or [↵] to accept it for the previous dialog box or the table.

Library filter

As the material library is very large, you find various selection options available in the dialog section *Filter*. You can filter the material list according to *Material category group*, *Material category*, *Standard group* and *Standard*. In this way, you can reduce offered data.

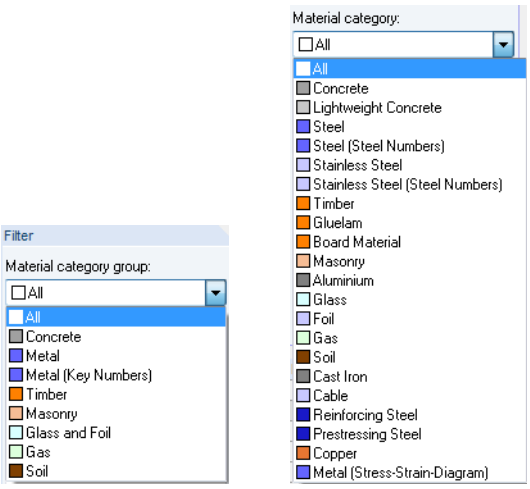


Figure 4.54: Filter for *Material category group* and *Material category*

Create favorites

Often, the use of a few materials is already sufficient for daily engineering work. You can mark these materials as your favorites. Use the button [Edit Favorites] (see Figure 4.56) to open the dialog box for defining preferred materials.

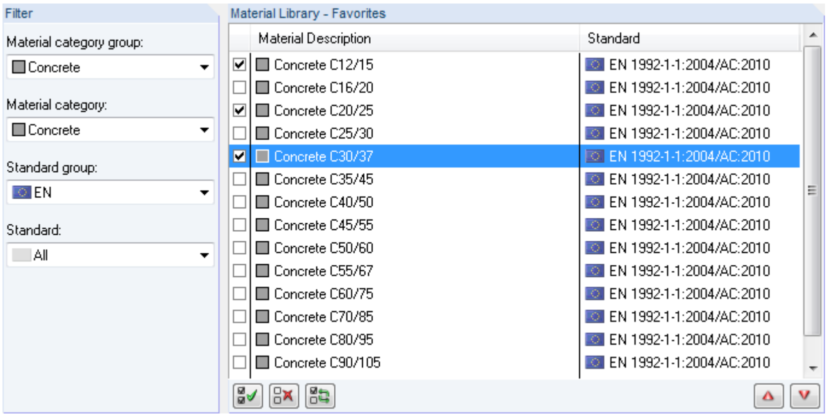


Figure 4.55: Dialog box *Material Library - Favorites* (dialog section)

The dialog box looks like the material library. You can use the filter options described above. In the dialog section *Material Library - Favorites*, you can select your preferred materials by ticking their check boxes. To change the sequence of materials, use the buttons [▲] and [▼].

After closing the dialog box, the material library presents a clear favorites overview as soon as you activate the option *Favorites only*.

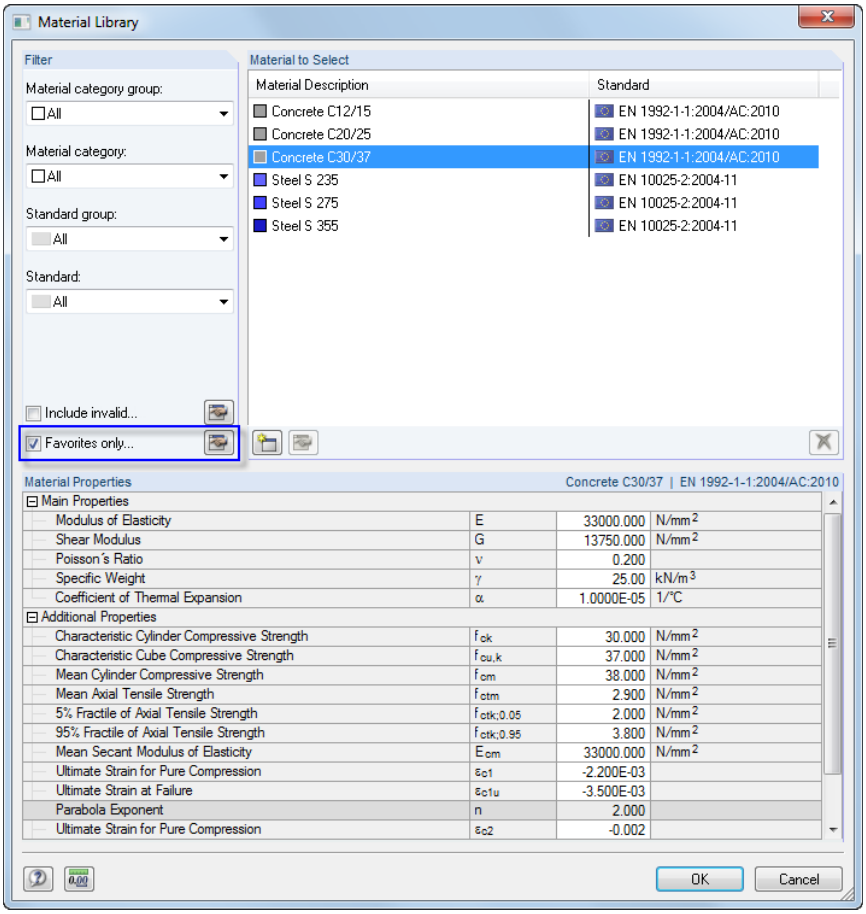


Figure 4.56: Dialog box *Material Library* with option *Favorites only*

With the option *Include invalid* in the *Filter* dialog section you can integrate also materials of 'old' standards into the library.

Complete library

The *Material Library* can be extended. When a new material is added, it can be used for all available models.

Click the [New] button in the library (to the right of the [Favorites] button, see Figure 4.56). The dialog box *New Material* opens. You can see that parameters of the entry selected in the list *Material to Select* are preset. So creating a new material is easier when you choose a material with similar properties before you access the dialog box.

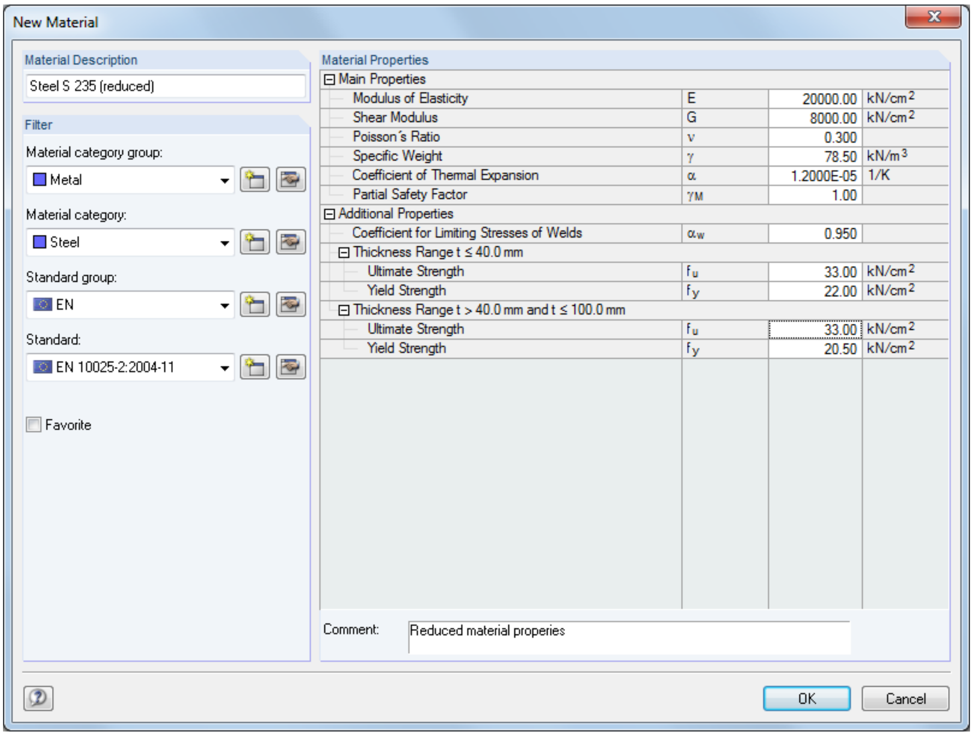


Figure 4.57: Dialog box *New Material*

Enter the *Material Description*, define the *Material Properties*, and assign the material to the appropriate categories for *Filter* functions.

Use the buttons shown on the left to create and edit categories.

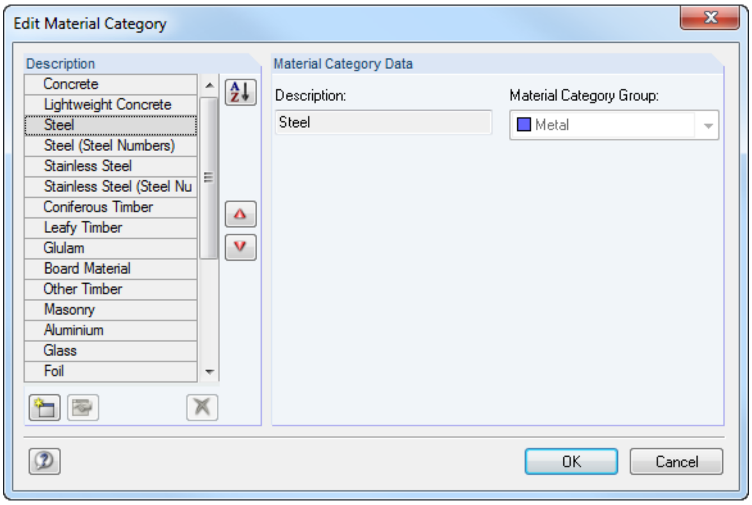


Figure 4.58: Dialog box *Edit Material Category*

To adjust the sequence of entries, use the buttons [▲] and [▼].

Saving user-defined materials

If you use customized materials, you should save the file **Materialien_User.dbd** before installing an update. The file can be found in the master data folder of RFEM 5 *C:\ProgramData\Dlubal\RFEM 5.xx\General Data.*

4.4 Surfaces

General description

In addition to structure geometry, surfaces describe the stiffness resulting from material and thickness properties. When generating the FE mesh, 2D elements are created on surfaces. For detailed information on used elements, see chapter 7.2.1 on page 256.

The stiffness type *Null* is to be used for geometry descriptions of solids.

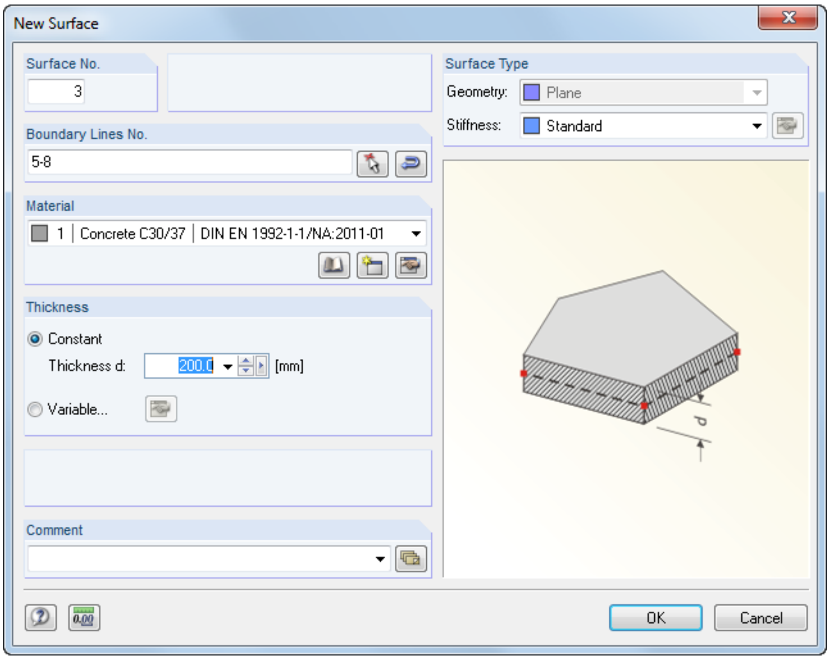
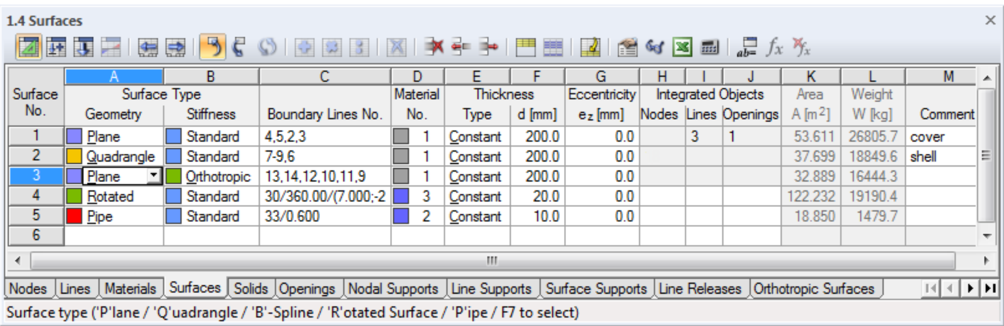


Figure 4.59: Dialog box *New Surface*



1.4 Surfaces

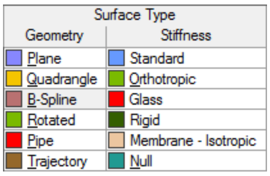
Surface No.	A Surface Type Geometry	B Stiffness	C Boundary Lines No.	D Material No.	E Thickness Type	F d [mm]	G Eccentricity e_z [mm]	H Integrated Objects Nodes	I Lines	J Openings	K Area A [m²]	L Weight W [kg]	M Comment
1	Plane	Standard	4,5,2,3	1	Constant	200.0	0.0		3	1	53.611	26805.7	cover
2	Quadrangle	Standard	7-9,6	1	Constant	200.0	0.0				37.699	18849.6	shell
3	Plane	Orthotropic	13,14,12,10,11,9	1	Constant	200.0	0.0				32.889	16444.3	
4	Rotated	Standard	30/360.00/(7.000;-2	3	Constant	20.0	0.0				122.232	19190.4	
5	Pipe	Standard	33/0.600	2	Constant	10.0	0.0				18.850	1479.7	
6													

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Orthotropic Surfaces

Surface type ('P'lane / 'Q'uadrangle / 'B'-Spline / 'R'otated Surface / 'P'ipe / F7 to select)

Figure 4.60: Table 1.4 *Surfaces*

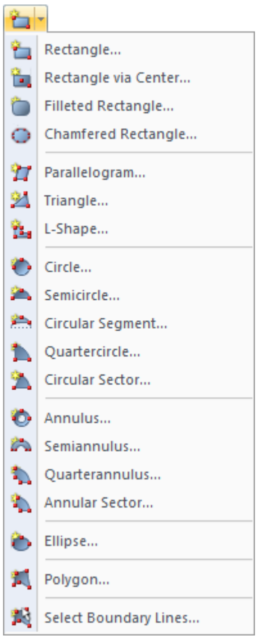
Different *Geometry* and *Stiffness* properties are available for structure modeling. It is possible to combine entries of both *Surface Type* lists or table columns – within type-specific limits and conditions.



Color symbols help you to assign various types for *Geometry* and *Stiffness*. You can use the colors in the model to represent individual surface types. Colors are controlled in the *Display* navigator with the option *Colors in Rendering According to* (see chapter 11.1.9, page 427).

Surface type - geometry

Plane surface



Plane surfaces can be defined graphically by drawing a rectangle, parallelogram, circle, annulus, polygon etc. Use the menu or the list button in the toolbar shown on the left to access different shapes of plane surfaces.

The following dialog box appears when you enter surface data by using one of the toolbar buttons.

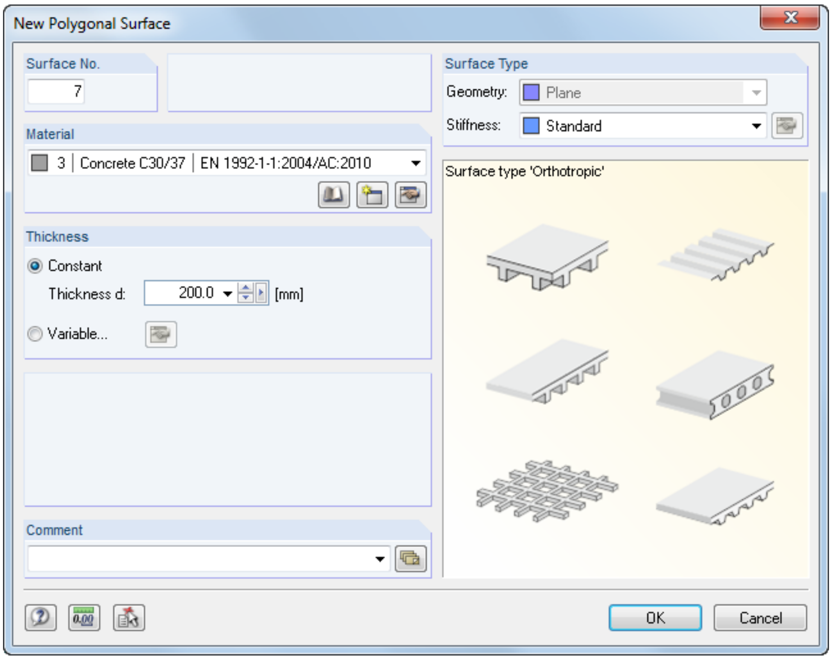


Figure 4.61: Dialog box *New Polygonal Surface*

First, enter the parameters for *Material*, *Thickness* and *Stiffness* in addition to the surface number. Click [OK], and then define the boundary lines of the surface in the work window by selecting relevant corner points.

With the menu option [Select Boundary Lines] you can select existing lines graphically. The lines must be arranged in a polygonal chain lying in one plane. The line types are described in chapter 4.2, page 52.

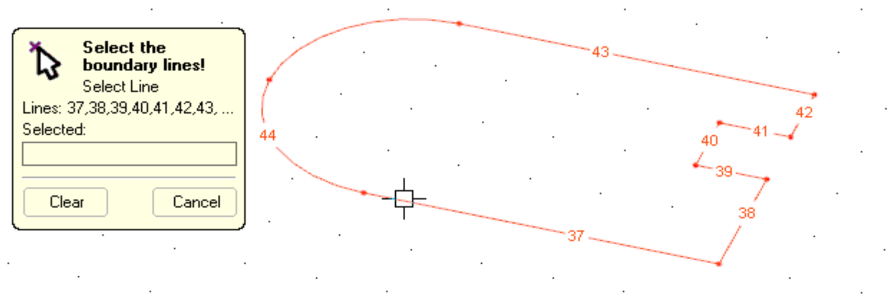


Figure 4.62: Selecting boundary lines in the graphic

RFEM recognizes the surfaces automatically as soon as a sufficient number of boundary lines has been defined.

Quadrangle surface

This type of surface represents a general quadrilateral surface. In addition to straight lines, you can use arcs, polylines and splines as boundary lines. Use this surface type to model shells as boundary lines do not have to be arranged in one plane.

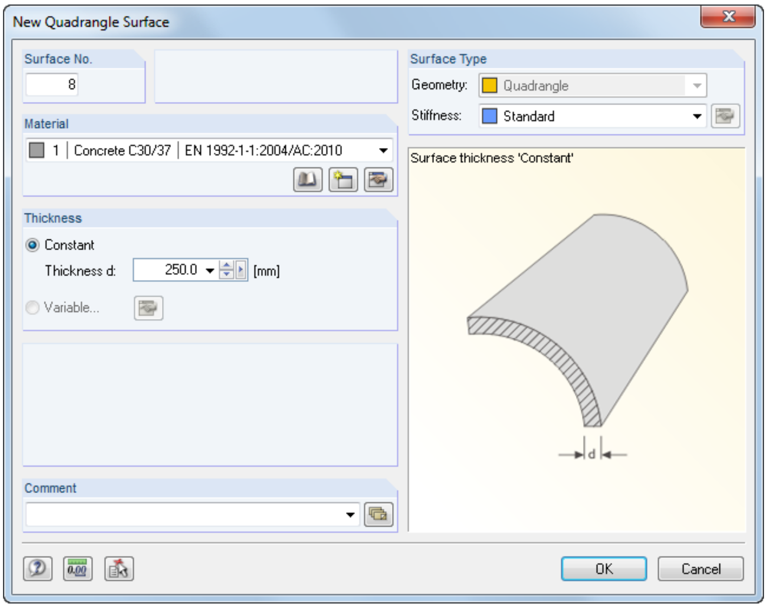


Figure 4.63: Dialog box *New Quadrangle Surface*

You can select the boundary lines graphically after clicking [OK].

Rotated surface

A rotated surface is created by rotating a line about a fixed axis. The surface results from the start and end position of the line as well as the line's rotated definition points.

The dialog box *New Rotated Surface* consists of two tabs. Define *Material*, *Thickness* and *Stiffness* of the surface in the *General* dialog tab (cf. Figure 4.64). A variable surface thickness is not allowed.

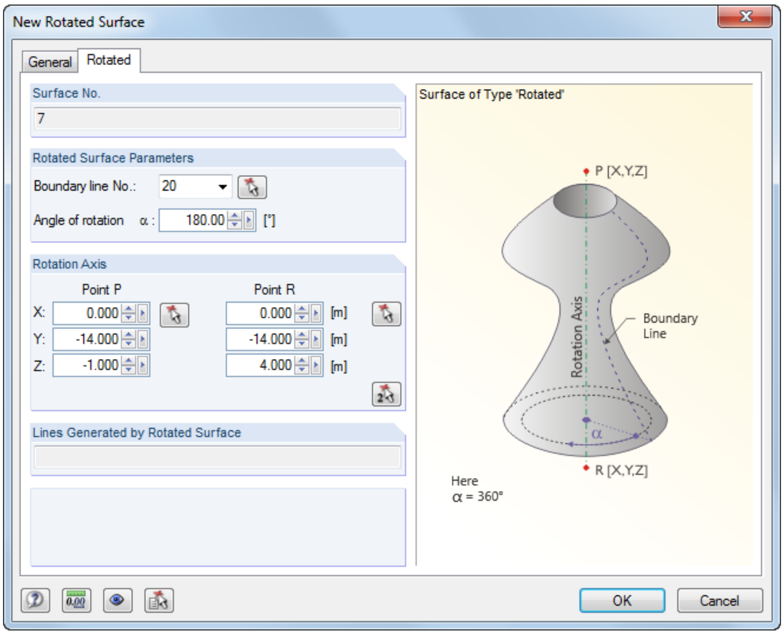


Figure 4.64: Dialog box *New Rotated Surface*, tab *Rotated*

In the *Rotated* tab, specify the *Angle of rotation* α. Both points of the *Rotation Axis* can be defined either by entering their coordinates or by using the [↖] function. Click [OK], and then define the boundary line for rotation in the work window.

Moreover, rotated surfaces can be created from generated lines.

Pipe

A pipe surface is created by rotating the center line of the pipe about the center axis in a distance of a specific radius.

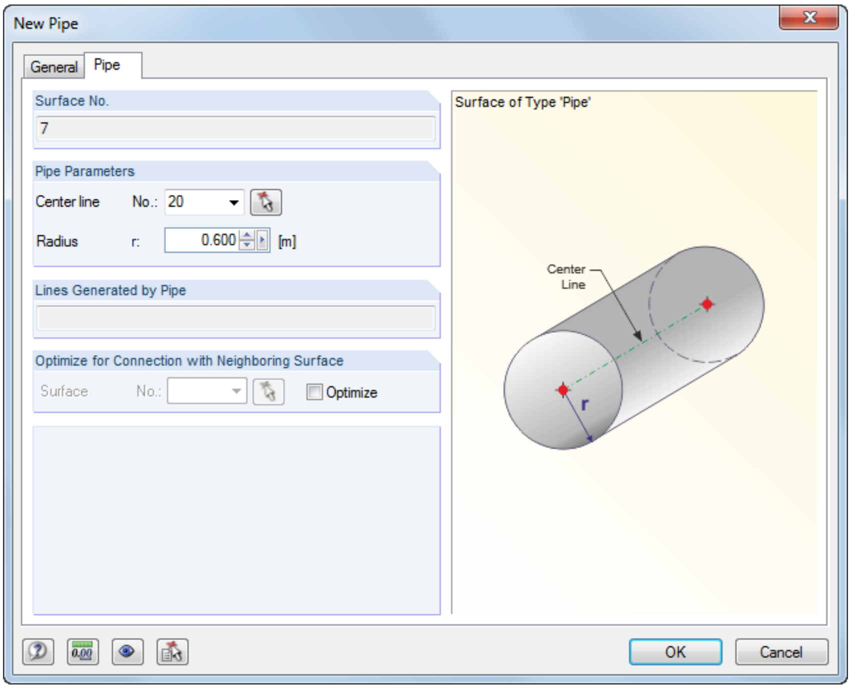


Figure 4.65: Dialog box *New Pipe*, tab *Pipe*

The dialog box *New Pipe* has two tabs. In the *General* tab, you enter the parameters for *Material*, *Thickness* and *Stiffness* of the surface. In the *Pipe* tab, you specify the *Center line* and *Radius r*. You can define the center line also graphically.

Use this type of surface to create two circles and a polyline that is parallel to the pipe axis.

B-Spline surface

A B-Spline surface is similar to a quadrangle surface (see Figure 4.63) but help nodes are created additionally on the surface. The surface shape can be influenced by adjusting the coordinates of help nodes subsequently.

The input dialog box has two tabs. In the *General* tab, you define the parameters for *Material*, *Thickness* and *Stiffness* of the surface. A variable surface thickness is not allowed.

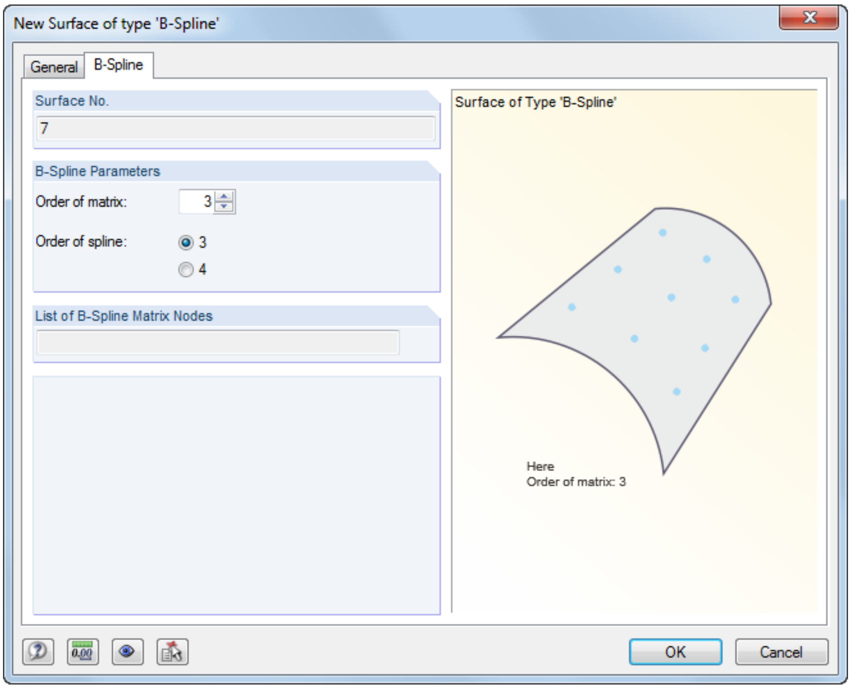


Figure 4.66: Dialog box *New Surface of type 'B-Spline'*, tab *B-Spline*

In the *B-Spline* tab, you enter the number of help nodes into the input field *Order of matrix*: For example, if you enter "3", you create a grid of 3 x 3 help nodes placed across the surface. The selection field *Order of spline* specifies whether a polynomial of the third or fourth degree is used for the calculation of the surface.

NURBS surface

NURBS surfaces are defined by four connected NURBS lines (see chapter 4.2, page 57). By using NURBS surfaces you can model almost any free form surface.

When entering boundary lines, make sure that opposite pairs of NURBS lines are "compatible" with each other. Only if the number of control points is equal, opposing NURBS lines are arranged in the same order.

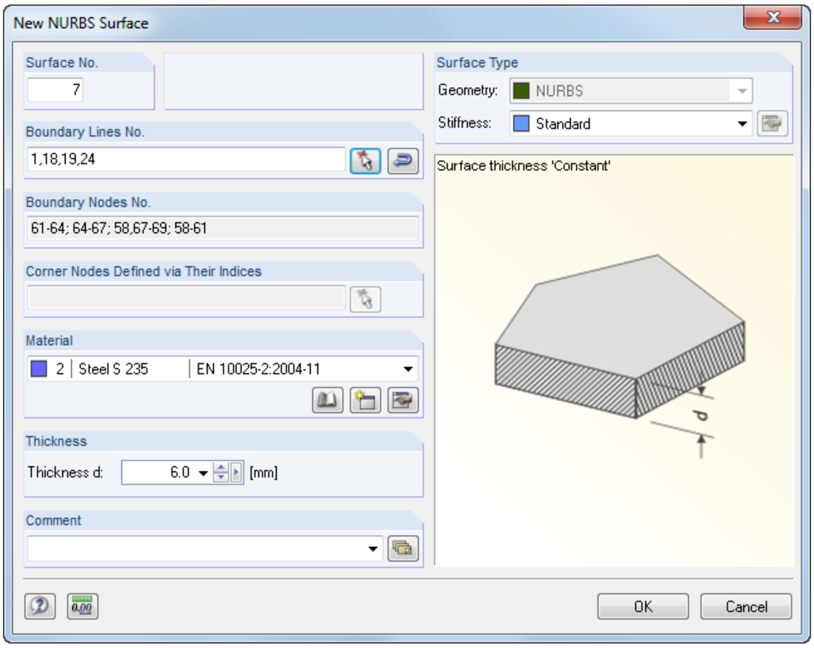


Figure 4.67: Dialog box *New NURBS Surface*

Trajectory surface

Use this type of surface to create a spatially curved surface from a certain start profile in relation to any trajectory.

The dialog box *New Trajectory Surface* has two tabs. In the *General* tab, you define the parameters for *Material*, *Thickness* and *Stiffness* of the surface.

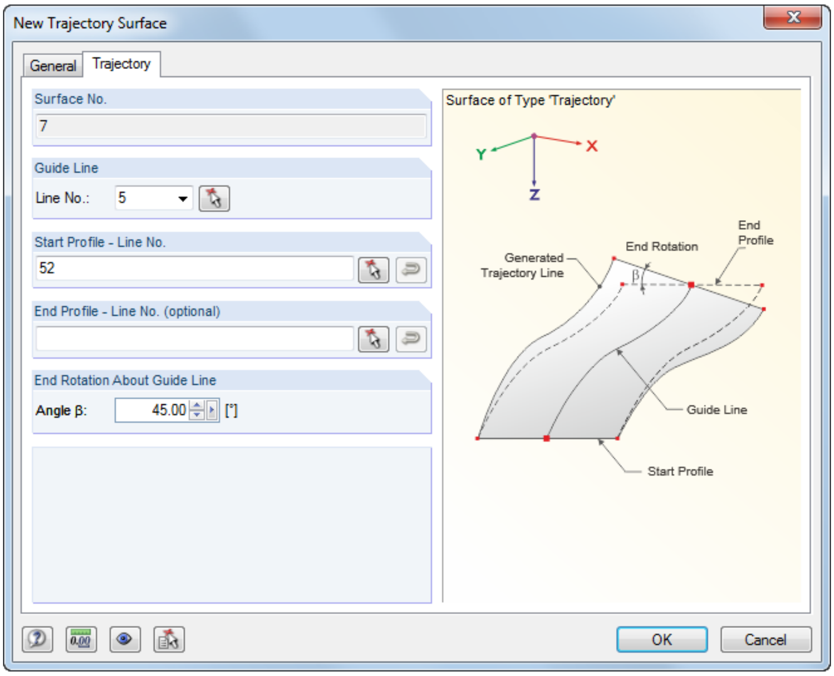


Figure 4.68: Dialog box *New Trajectory Surface*, tab *Trajectory*

In the *Trajectory* tab, enter the number of the *Guide Line* representing the reference line of the surface. You can select it also graphically. Then, determine the *Start Profile* in the graphic. If necessary, define a second line as *End Profile*. The *Angle* β describes the rotation of the parallel boundary line generated in relation to the trajectory.

Component

This entry appears in the table column and navigator only if an intersection of surfaces has been created (see chapter 4.22, page 160). The editing functions for components of intersection surfaces provided by RFEM are the same as for "normal" surfaces. So it is possible to modify properties of surface components quickly without creating an intersection again.

The original surface of a component is indicated in the *Component* tab of the *Edit Surface* dialog box.

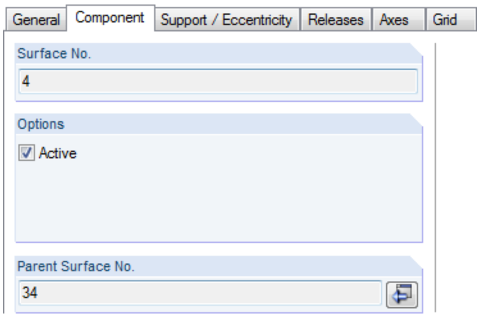
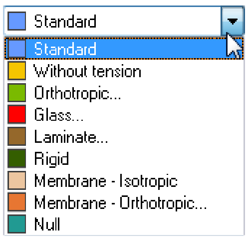


Figure 4.69: Dialog box *Edit Surface*, tab *Component*

Use the button [Go to Parent Surface] to access the edit dialog box of the original surface.

Surface type - stiffness{



The list available in the dialog box and table provides several stiffness models which you can select to model the structure more close to reality.

Standard

The surface transfers moments and axial forces. The approach describes the general behavior of a homogeneous and isotropic material. The stiffness properties of the surface do not depend on directions.

Without tension

Moments and membrane forces are transferred. For axial forces causing tension, however, a failure of affected surface elements occurs (example: hole bearing).

Orthotropic

Set this type of stiffness for surfaces with different stiffnesses in both surface directions (see chapter 4.12, page 112). Use the [Edit] button to define the parameters.

Alternatively, you can assign an orthotropic property to the material (see chapter 4.3, page 66). In this way, you can avoid defining properties for each single surface.

Glass

This type of stiffness is required for the add-on module RF-GLASS. Moments and membrane forces are transferred, but stresses are not determined in RFEM. The actual stress calculation is carried out later in the add-on module RF-GLASS.

Laminate

This stiffness type transfers moments and axial forces. The add-on module RF-LAMINATE is required to calculate the laminate model. Corresponding stresses are not included in the results output of RFEM, you need the module for stress calculation.

Rigid

Use this type of stiffness to generate very stiff surfaces creating a rigid connection between adjacent objects.

Membrane

The surface has a uniform stiffness in all directions. Only membrane forces are transferred.

Membrane orthotropic

Only membrane forces are transferred. Stiffnesses are different in both surface directions (chapter 4.12, page 112). Use the [Edit] button to define the parameters.

Null

Null surfaces are required for the definition of solids (see chapter 4.5, page 85).

Boundary lines No.

The boundary lines of a surface are listed in the corresponding input field or table column. They must form a polygonal chain.

When rotated surfaces were generated, generation parameters are displayed in the table column.

Material No.

You can choose an entry from the list of materials that have already been created. Material colors make the assignment easier.

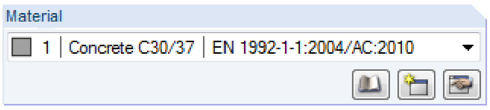


Table 4.70: Buttons in the dialog section *Material*

In the dialog box *New Surface*, you can see three buttons below the list. Use the buttons to access the material library or to create and edit materials.

For more detailed information on materials, see chapter 4.3 on page 60.

Thickness

Type

You can select between two types of surface thickness.

- **Constant**
  The surface has the same thickness everywhere.
- **Variable**
  The thickness of the surface is linearly variable (see chapter 4.11, page 111). Use the [Edit] button to define the parameters.

Thickness d

Specify the surface thickness *d* unless a variable thickness or a Null surface has been defined. The thickness is used to determine self-weight and stiffness for the stiffness types *Standard*, *Without tension*, *Glass* and *Membrane isotropic*. For the *Orthotropic* stiffness this value is only used for self-weight calculation (stiffnesses must be defined separately for orthotropic surfaces).

Surface thicknesses can be visualized with different colors in the model: In the *Display* navigator, select *Model* and *Surfaces*, and then tick the check box for *Color Scale of Thicknesses in Panel* (see figure below).

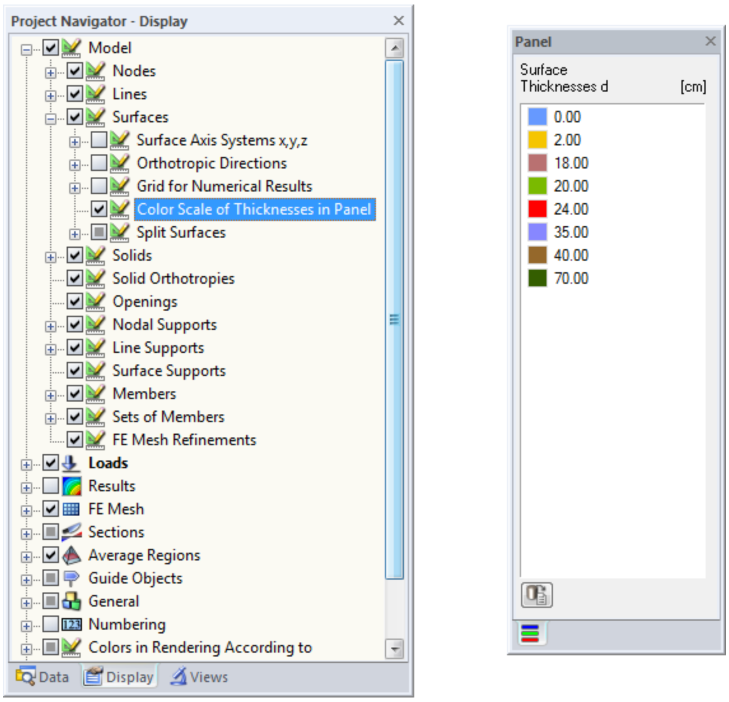


Figure 4.71: *Display* navigator: *Surfaces* → *Color Scale of Thicknesses in Panel*

Eccentricity

The plane in the surface center represents the reference surface for the thickness assumed to be in equal parts on both sides of the "centroidal plane". To check the center, set the *Display* navigator and select the options *Rendering*, *Model*, *Solid Model*, *Surface* and *Filled incl. thickness* (see Figure 4.110, page 112).

By specifying an *Eccentricity* e_z you can define an offset of height for the surface. In this way, you can create uniform top or bottom edges for adjoining surfaces that have different thicknesses.

The eccentricity in the form of additional moments has an influence on the surface's internal forces.

Integrated objects

In general, RFEM recognizes automatically all objects lying on a surface but are not used for surface definition. In the table columns or input fields of the dialog box, all numbers of nodes, lines and openings are displayed.

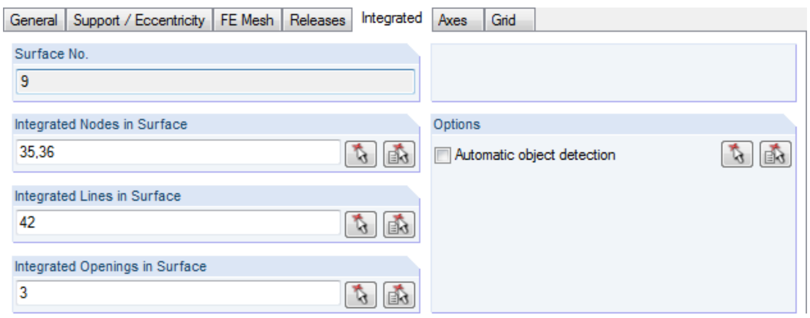


Figure 4.72: Dialog box *Edit Surface*, tab *Integrated*

If an object is not recognized, it is possible to integrate it manually: Double-click the surface to open the dialog box *Edit Surface*. Then, in the *Integrated* tab, deactivate the option *Automatic object detection*. The input fields of the dialog sections to the left will be enabled for access. Use the [↖] button to select the objects graphically.

Area

The area of each surface is shown in the table column so that you can check the surface data. Areas of openings are not taken into account, thus the value represents the net area.

Weight

The mass of each surface is indicated. It is determined from the area and the material's specific weight.

Comment

Enter a user-defined note or select an entry from the list.

Axis system

Each surface has a local coordinate system. The axis system of the surface is significant for various input parameters such as orthotropic and foundation properties or directions of surface loads. The base internal forces are also related to a surface axis system.

RFEM shows you the coordinate systems as soon as you move the pointer across a surface. You can also use the context menu of a surface to switch them on and off.

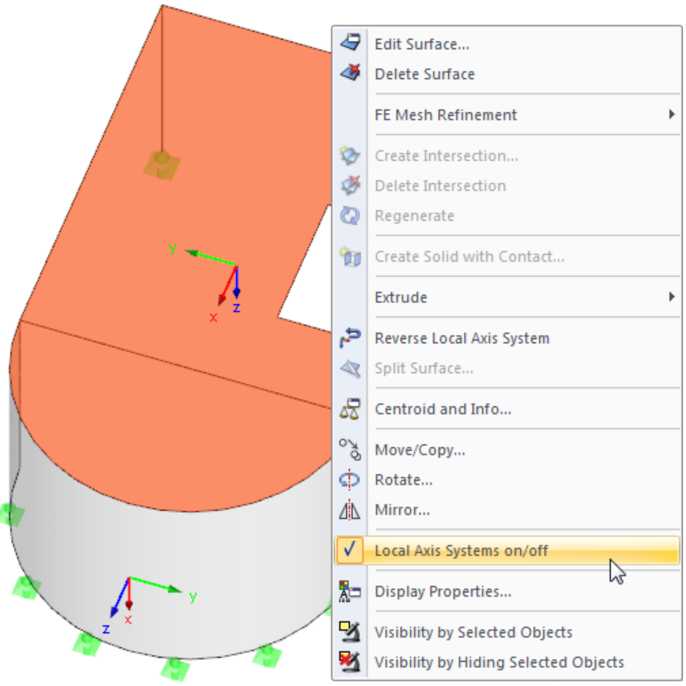


Figure 4.73: Surface context menu

If required, you can adjust the local surface axes:

- Context menu option *Reverse Local Axis System*

  The orientation of the local z-axis is reversed, the remaining axes are aligned according to the right-hand rule. As a result, foundations are put on the other side of the surface, or the "top" and "bottom" reinforcement layers for the reinforced concrete design change surface sides.
- Dialog box *Edit Surface*

  To open the dialog box *Edit Surface*, double-click the surface. In the *Axes* tab, you can adjust the local surface *Axes for Input* as well as *for Results* (see figure below).

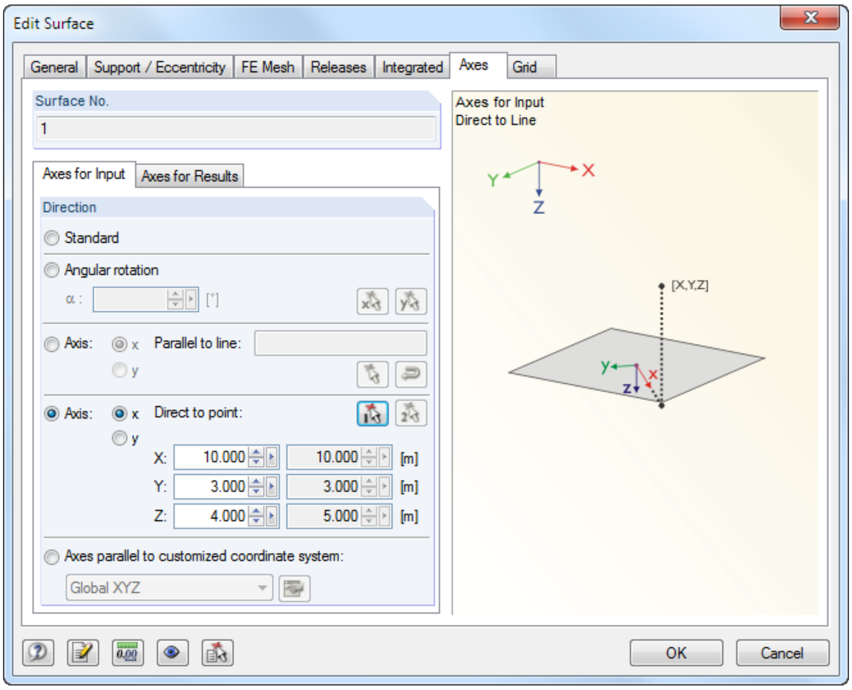


Figure 4.74: Dialog box *Edit Surface*, tab *Axes*

In the two sub-tabs, it is possible to direct the local surface axis *x* or *y* to a *line*, a *point* or a *customized coordinate system* (see chapter 11.3.4, page 443).

4.5 Solids

General description

In RFEM, 3D objects are described by solids. When generating the FE mesh, 3D elements are created. You can use solids to model also orthotropic properties or contact problems between surfaces. In addition, solids can have gas properties.

In general, boundary surfaces of solids are defined with the stiffness type *Null* (see chapter 4.4, page 81). However, if no other solid is connected to a model representing the contact between two surfaces, both contact surfaces have to be characterized with a stiffness.

In the graphic, solids can be created quickly from surfaces. Corresponding generation functions are described in chapters 11.7.1.3 and 11.7.1.4 on page 500.

FE mesh refinements for 3D elements can be specified as well.

Reinforced concrete designs are currently not implemented for solids.

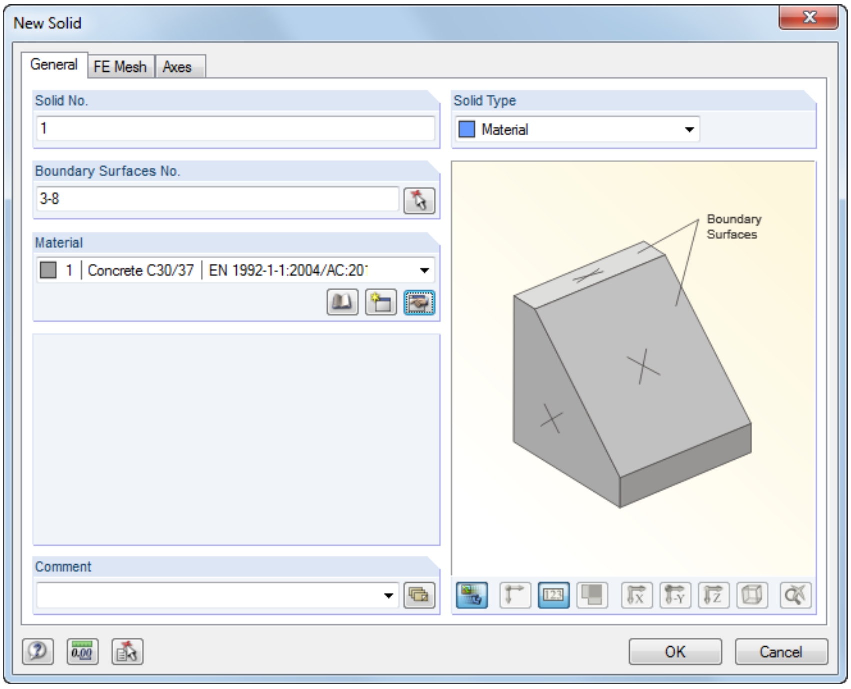
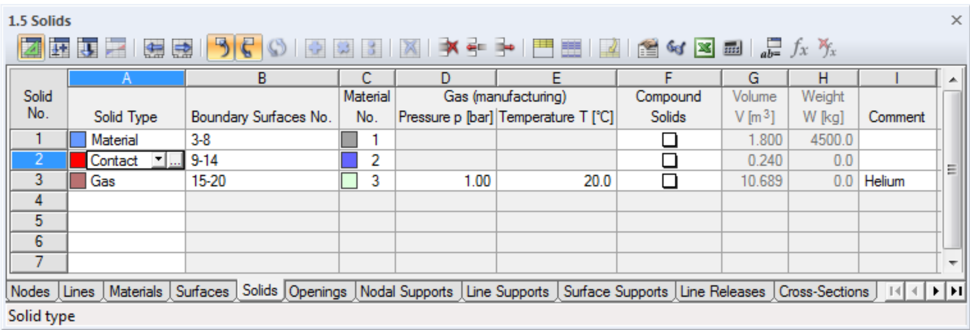


Figure 4.75: Dialog box *New Solid*

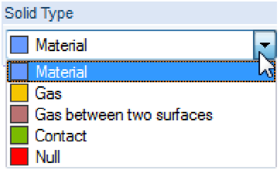

1.5 Solids

Solid No.	A Solid Type	B Boundary Surfaces No.	C Material No.	D Gas (manufacturing) Pressure p [bar]	E Temperature T [°C]	F Compound Solids	G Volume V [m³]	H Weight W [kg]	I Comment
1	Material	3-8	1			☐	1.800	4500.0	
2	Contact	9-14	2			☐	0.240	0.0	
3	Gas	15-20	3	1.00	20.0	☐	10.689	0.0	Helium
4									
5									
6									
7									

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Cross-Sections

Solid type

Figure 4.76: Table 1.5 *Solids*



Solid type

In the table as well as the list available in the dialog box, several stiffness types can be selected to model structures more close to reality. Each type has its own color that can be used in the model to differentiate solids. Colors are controlled in the *Display* navigator with the option *Colors in Rendering According to* (see chapter 11.1.9, page 427).

Material

The standard model is represented by a 3D object with the solid-specific properties of a homogeneous and isotropic material. Therefore, boundary surfaces should be defined by the stiffness type *Null*.

If the solid has orthotropic properties, stiffnesses are derived from the material characteristics, too. Define the elastic stiffnesses of the three-dimensional material model in the dialog box *Material Model - Orthotropic Elastic 3D* (see Figure 4.48, page 67).

Gas

Use this option to model solids with properties of an ideal gas. The gas parameters have to be defined in a separate tab of the dialog box (see Figure 4.79).

Gas between two surfaces

It is recommended to form a solid with properties of an ideal gas as an object that is relatively thin (for example gas layer in insulating glass). With this option RFEM creates exactly two finite elements between cover and base area of the solid so that calculations converge faster than for the *Gas* type. The parameters have to be defined in a separate tab of the dialog box (see Figure 4.79). However, for general situations (for example container, bouncing castle) use the type *Gas*.

Contact

This solid type is appropriate for modeling contact properties between two surfaces. The parameters have to be defined in a separate tab of the dialog box (see Figure 4.80).

Null

Neither a Null solid nor its loads will be considered for the calculation. Null solids are used to analyze for example changes in the model's structural behavior if a solid is not effective. You do not need to delete the solid, the loading will be kept as well.

Boundary surfaces

A solid is defined by surfaces completely enclosing a certain space. Enter the numbers of the surfaces into the input field, or select them in the graphic by using the [↖] function.

When you have defined all boundary surfaces in the dialog box *New Solid*, you can click the button [Show Figure or Rendering] below the graphic to see a preview of the solid.

Material

You can choose an entry from the list of materials that have already been created. Material colors make the assignment easier.

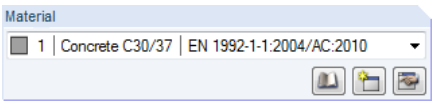
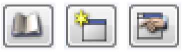


Table 4.77: Buttons in the dialog section *Material*



In the dialog box *New Solid*, you can see three buttons below the list. Use the buttons to access the material library or to create and edit materials.

For more detailed information on materials, see chapter 4.3, page 60.

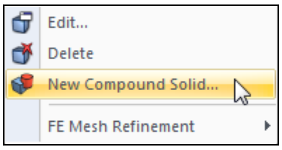
Compound solids

When an intersection of solids has been created, you can see this column displayed in the table.

In addition to surfaces, you can generate intersections for solids. RFEM determines the intersection lines of intersecting solids and creates 3D solid objects as a union, a section or as pure intersecting set. In this way, a new solid is generated from two original objects.

Determining the solid intersection is time-consuming and computationally intensive. Each time the model is changed, a recalculation of geometry is required.

Creating an intersection



You can create intersections of solids quickly in the graphic: Select two solids by drawing a selection window across the objects, or use the multiple selection by holding down the [Ctrl] key. Then, right-click one of the solids to open its context menu where you

point to **Solid** and select **New Compound Solid**.

The dialog box *New Solid* opens. With the settings in the dialog tab *Compound Solids* you specify how both solids are combined.

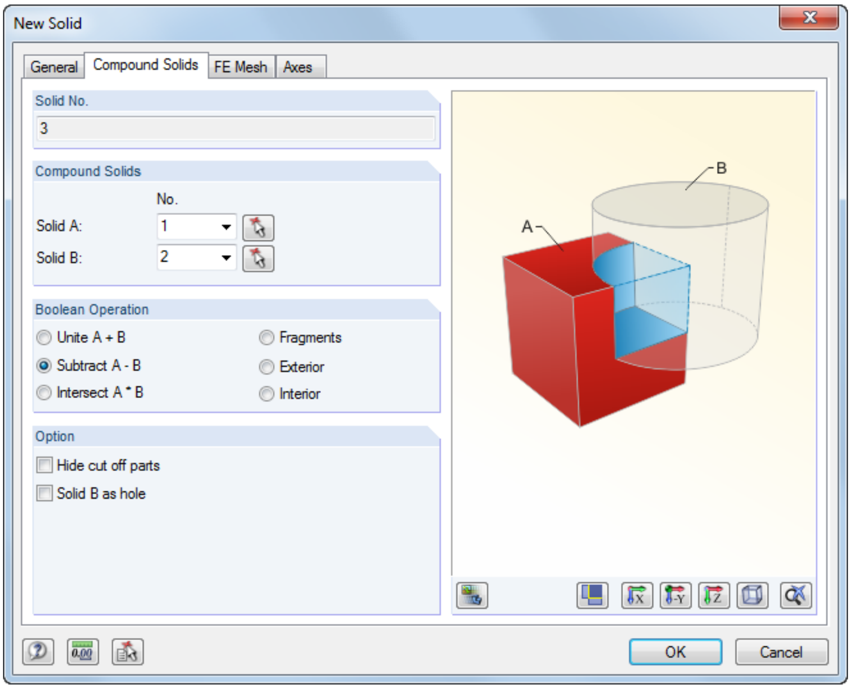


Figure 4.78: Dialog box *New Solid*, tab *Compound Solids*

The numbers of the two selected solids are entered into the input fields. Use the list or the [↖] function to change the entries.

Boolean operation

You have three possibilities to combine solids in a new object:

- *Unite*: Solids A and B are merged to a unit.
- *Subtract*: Solid B is cut out of solid A.
- *Intersect*: RFEM determines the area shared by solids A and B.

The dialog graphic to the right demonstrates the principle of combinations. Use the button [Show Figure or Rendering] to switch between scheme and model display.

In the dialog section *Option*, you decide how parts that were cut are displayed in the graphic of the work window.

Click [OK] to create the combined solid. As a result, intersections of surfaces are generated (see chapter 4.22, page 160) with active or inactive surface components (see chapter 4.4, page 80). At the same time, RFEM sets the original solids to the type *Null*.

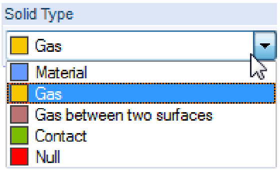
Volume V

The table column shows the volume of each solid.

Weight W

The mass of each solid is indicated. It is determined from the volume and the material's specific weight.

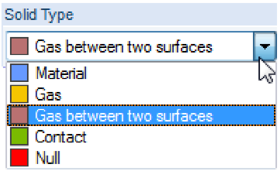
Gas



This dialog tab is available if you have selected the solid type *Gas* in the dialog tab *General*.

You have to define the *Gas Parameters* pressure p_p and temperature T_p (see Figure 4.79).

Gas between two surfaces



This dialog tab is available if you have selected the solid type *Gas between two surfaces* in the dialog tab *General*. With the tab settings you can model gas-specific contact solids representing the effect of pressure on two opposite surfaces (for example insulating glass).

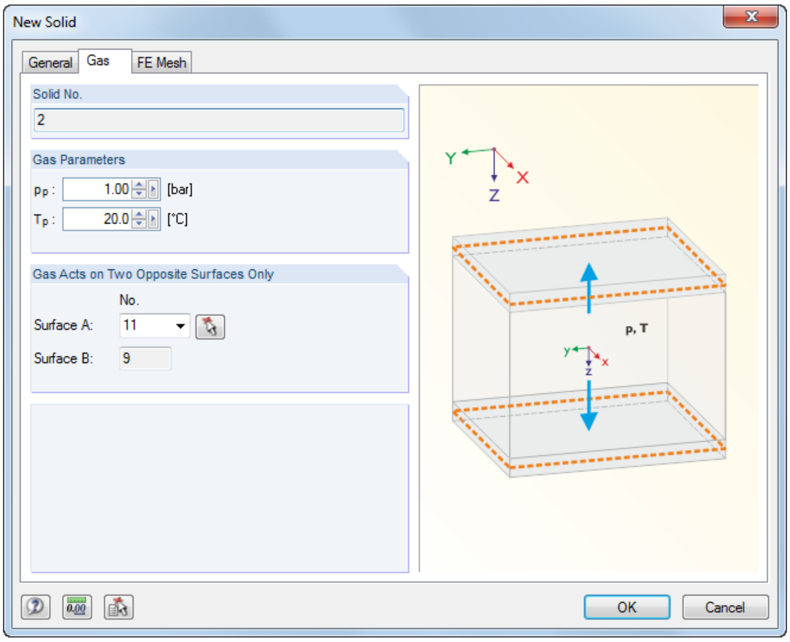
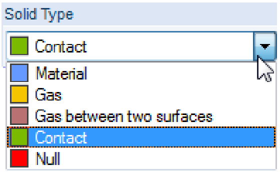


Figure 4.79: Dialog box *New Solid*, tab *Gas*

In addition to the *Gas Parameters* pressure p_p and temperature *Tp*, you have to specify two surfaces enclosing the gas layer. *Surface A* can be selected in the list or defined graphically by using the [↖] function. The parallel *Surface B* will be entered automatically.



Contact

This dialog tab is available if the solid type *Contact* has been previously selected in the dialog tab *General*.

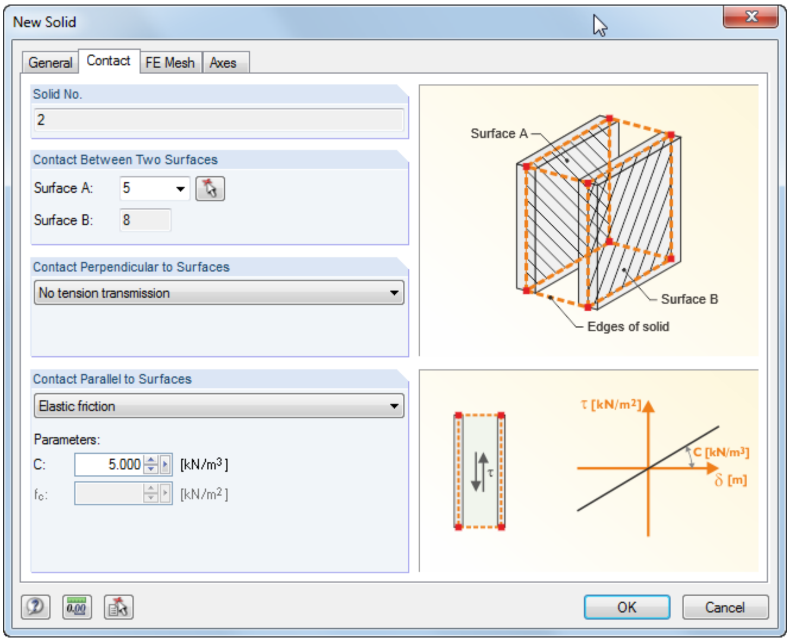


Figure 4.80: Dialog box *New Solid*, tab *Contact*

The following items must be observed when entering a contact solid:

- Both contact surfaces must be arranged parallel and created identically. It is recommended to create the second contact surface by copying.
- Each lateral connecting surface existing between the contact surfaces must created as a simple surface consisting of four boundary lines. Splitting a connecting surface, for example into two surface components at half of the height, is not allowed.
- When modeling curved contact surfaces, you have to split the contact solid into several simple parts.
- RFEM generates undivided 3D elements (parallel "columns") between the finite elements of the contact surfaces, creating a direct connection. Therefore, the FE division of the surface needs to be adjusted to the spacing of the contact surfaces.
- Polygonal solids are to be preferred to triangular solids.

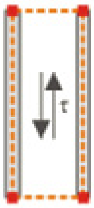
RFEM tries to find the contact surfaces automatically. In the dialog section *Contact Between Two Surfaces*, you can change *Surface A* by using the list. You can also use the [↖] function to select the surface graphically. Automatically, RFEM enters *Surface B* as a solid surface that is parallel to the first one.

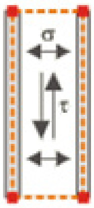
In the dialog section *Contact Perpendicular to Surfaces*, you can select between three options:

- Full force transmission
- No compression transmission
- No tension transmission

The failure criteria *No compression transmission* and *No tension transmission* are taken into account in the calculation by the deformations of solid FE mesh nodes.

The *Contact Parallel to Surfaces* can be defined independently, regardless of contact properties acting perpendicular to the two contact surfaces.





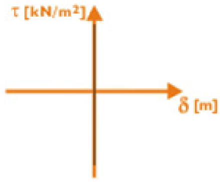
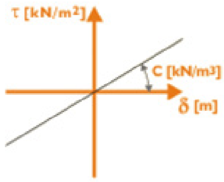
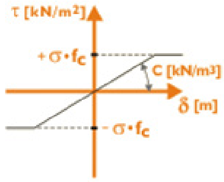
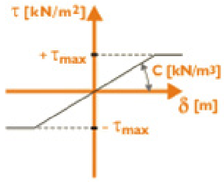
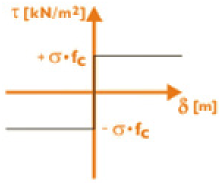
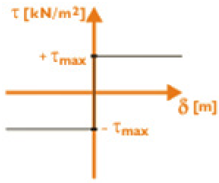
Contact	Diagram	Description
Full force transmission		All forces are transmitted.
Elastic friction		The "friction" represents an elastic behavior: The shear force increases proportionally to the deformation. There is no limit for deformation. The spring stiffness **C** must be entered as parameter (which means the force required to move a surface of 1 m^2 about 1 m).
Coulomb friction		This type of contact is similar to an elastic friction but has nonlinear impact. When a shearing stress of $\sigma \cdot f_c$ is reached, the shear stress remains constant even if the deformation increases. The stress σ is representing the normal stress in the relevant finite element. The spring stiffness **C** and the Coulomb friction factor $\mathbf{f_c}$ must be entered as parameters.
Coulomb friction with limit		In contrast to the elastic Coulomb friction, the maximum shear stress does not depend on the normal stress. Only one defined shear stress can be absorbed. The spring stiffness **C** and the shear stress $\boldsymbol{\tau_{max}}$ must be entered as parameters.
Rigid friction		This type of nonlinearity is similar to the elastic Coulomb friction. As the elastic zone is missing, the Coulomb friction is immediately effective. The Coulomb friction factor $\mathbf{f_c}$ must be entered as parameter.
Rigid friction with limit		This type of nonlinearity is similar to the elastic Coulomb friction with limit. As the elastic zone is missing, the limit due to the shear stress is immediately effective. The shear stress $\boldsymbol{\tau_{max}}$ must be entered as parameter.

Table 4.2: Contact properties parallel to contact surfaces

Axis system

Each solid has a local coordinate system. The axis system is significant for example for orthotropic properties. Stresses and distortions are related to the local axis system as well.

RFEM shows you the coordinate systems as soon as you move the pointer across a surface. You can also use the context menu of a solid to switch them on and off.

In the dialog box *Edit Solid*, you can adjust the solid coordinate system. Double-click the solid to open the dialog box. The orientation of the local axes is managed in the dialog tab *Axes*.

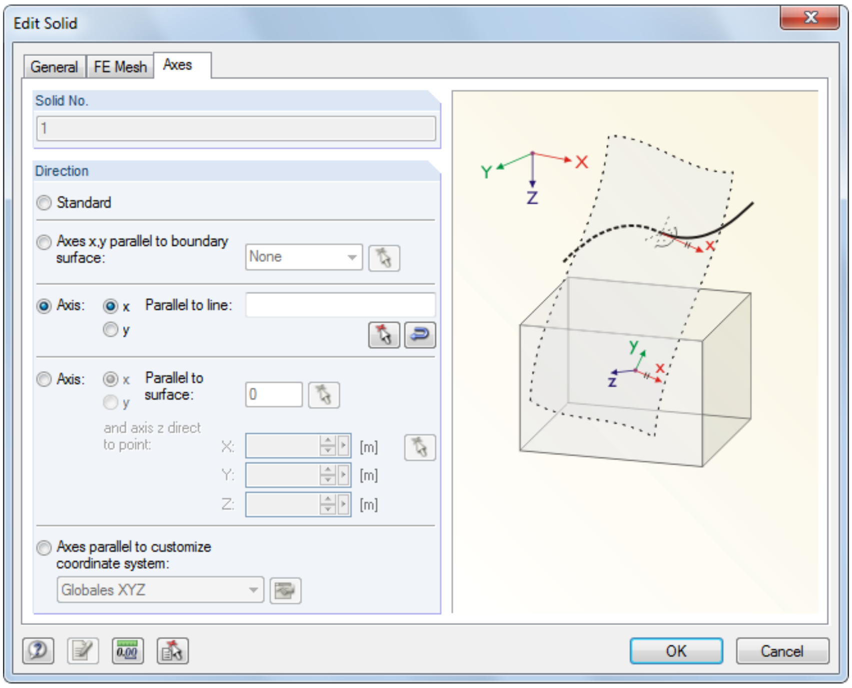


Figure 4.81: Dialog box *Edit Solid*, tab *Axes*

The solid's local axes *x* and *y* can be directed parallel to the axes of a *boundary surface*, a *line*, a *surface* or in direction of a *customize coordinate system* (see chapter 11.3.4, page 443).

4.6 Openings

General description

Openings are used to create cutouts into surfaces. Neither finite elements are generated nor surface loads are applied in areas where openings are placed.

Openings can be set graphically into surfaces. RFEM creates a polygonal chain for various types of openings and integrates it into the surface.

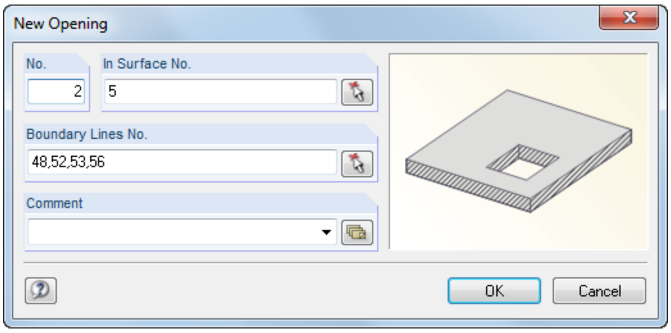
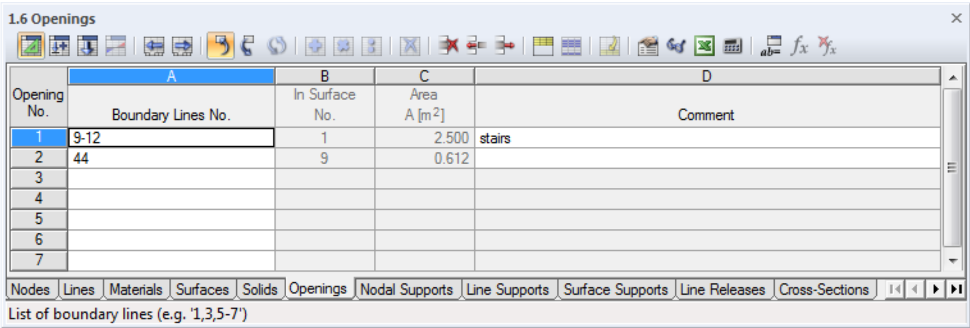


Figure 4.82: Dialog box *New Opening*


1.6 Openings

Opening No.	A Boundary Lines No.	B In Surface No.	C Area A [m2]	D Comment
1	9-12	1	2.500	stairs
2	44	9	0.612	
3				
4				
5				
6				
7				

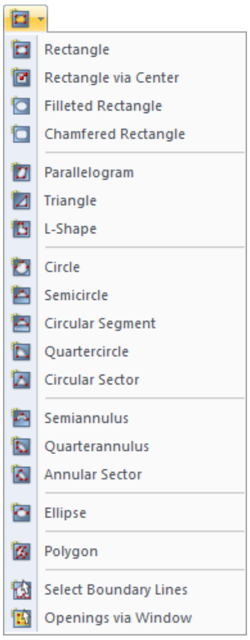
Figure 4.83: Table 1.6 *Openings*

Boundary lines

Enter the lines which enclose the opening. They must be defined as polygonal chain. Openings on the edge of a surface are allowed, too.

When you use the graphical selection, click the opening's boundary lines one after the other in the graphic. RFEM recognizes the opening automatically as soon as a sufficient number of boundary lines has been defined.

Use the button *Select Boundary Lines* in the openings menu list to select already defined lines graphically. They must form a polygonal chain.



Openings can be placed directly into a surface lying in the work plane by using one of the buttons shown on the left. The definition types for openings are based on the line types described in chapter 4.2, page 49 (for example circle, ellipse). The opening is created as soon as the contour lines have been determined. With this input option it is not necessary to create lines for the opening in advance.

In surface

For plane surfaces the automatic integration is active by default. For curved surfaces you have to integrate the opening manually. In the dialog box *Edit Surface*, select the tab *Integrated* and enter the number of the opening into the input field (see Figure 4.72, page 82).

Area

The table column shows the area of the opening.

4.7 Nodal Supports

General description

Supports are used to transfer loads applied on a structural system into the foundations. Without any supports all nodes would be free and could be displaced or rotated. If you want a node to act as a support, at least one of its degrees of freedom must be blocked or restricted by a spring. In addition, the node must be part of a surface or member. The boundary conditions of members must be considered as well in order to exclude double releases on the supported nodes.

Nodal supports are required in order to apply imposed deformations.

It is possible to provide nodal supports with nonlinear properties (failure criteria for tensile or compressive forces, working and stiffness diagrams).

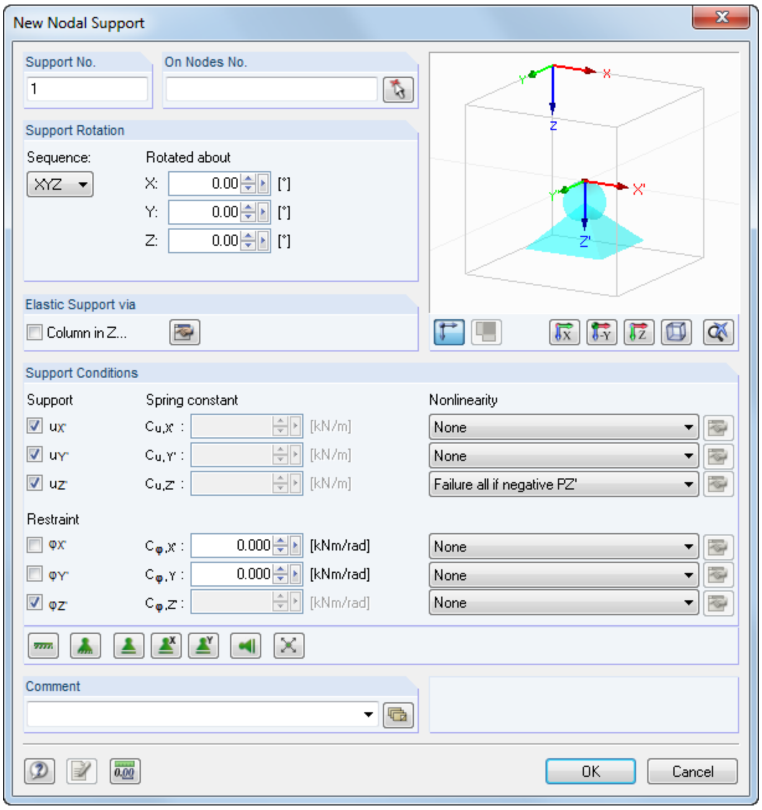


Figure 4.84: Dialog box *New Nodal Support*

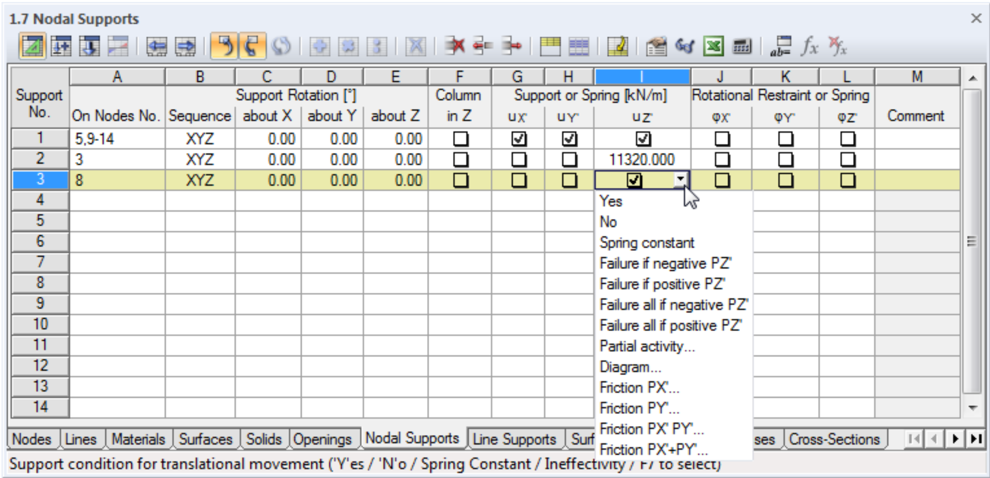


Figure 4.85: Table 1.7 *Nodal Supports*

To open the following dialog box, open the **Insert** menu, point to **Model Data** and **Nodal Supports**, and then select **Graphically**, or use the toolbar button shown on the left.

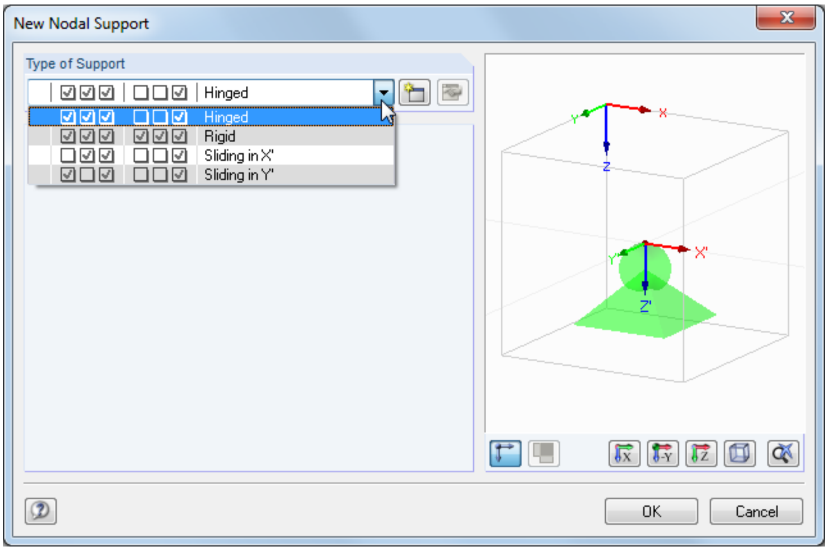


Figure 4.86: Dialog box *New Nodal Support*

The following support types are predefined and can be selected from the list:

- Hinged (YYY NNY)
- Rigid (YYY YYY)
- Sliding in X' (NYY NNY)
- Sliding in Y' (YNY NNY)

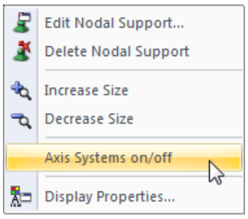
After clicking the [OK] button you can assign the selected support type to nodes in the graphic.

Use the [New] button to create another type of support. The dialog box shown in Figure 4.84 appears.

On nodes

Singular supports can be defined only on nodes. Enter the node number into the table column or the input field of the dialog box. You can select it also graphically.

Support rotation



Context menu of nodal support

Each nodal support has a local coordinate system which is oriented parallel to the global axes X, Y and Z by default. Use the context menu of the nodal support to activate the display for the support coordinate systems.

It is possible to rotate the support's local axis system. First, select the *Sequence* that determines the order of the local support axes X', Y' and Z'. Then, enter the angle of rotation for the global axes X, Y and Z into the input fields below *Rotated about*. You can also use the dialog buttons [▶] to define the support rotation graphically.

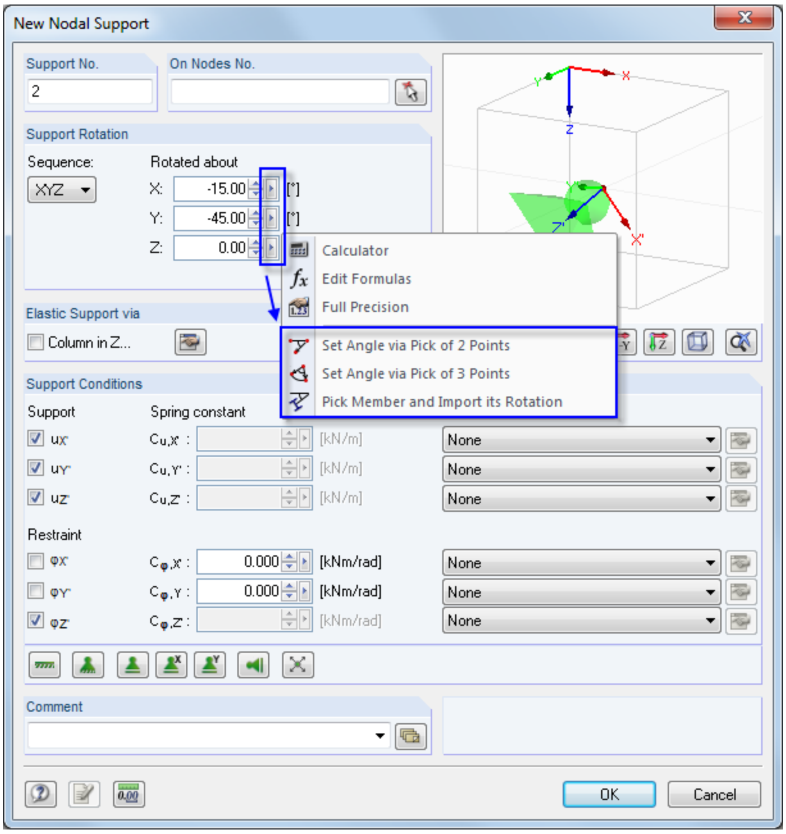


Figure 4.87: Dialog box *New Nodal Support* with options for support rotation

The entered support rotation is shown in the interactive dialog graphic.

When the calculation is complete, you can evaluate the support reactions of a rotated nodal support in relation to the global as well as the local axis system.

Column in Z

Often, real structural conditions are not sufficiently represented by a nodal support, for example when the support zone has great dimensions. Such support conditions can be represented in RFEM by special column macro elements considering material and geometry of the column. RFEM calculates the spring stiffnesses and adjusts the support conditions. Due to the realistic modeling you can avoid singularities that would be produced in a single FE node when a rigid support is defined.

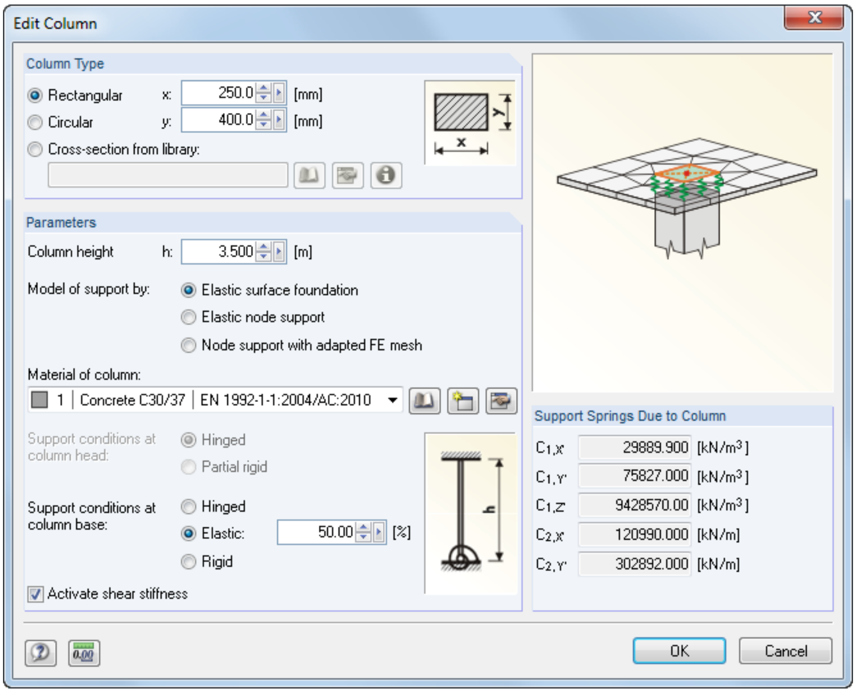


Figure 4.88: Dialog box *Edit Column*

Define the column cross-section in the dialog section *Column Type*. The input fields are changing according to the selected column type *Rectangular*, *Circular* or *Cross-section from library*. Thus, in addition to reinforced concrete columns, you can use steel sections as column cross-sections.

Furthermore, the *Column height h* affects the constants of translational and rotational springs. In the dialog section *Parameters*, you define how columns are modeled in the structure:

- The *Elastic surface foundation* represents an internal subsurface (cut out area) of the column area which is supported elastically. The foundation coefficients are calculated from the column geometry and material.
- The *Elastic node support* represents also an internal subsurface but is supported only at one node. The support is modeled by means of translational and rotational springs which are calculated from the column geometry and its material. Internally, the surface thickness is duplicated to account for higher bending rigidity within the column area.
- The *Node support with adapted FE mesh* corresponds to the elastic nodal support, but no springs are applied to the punctiform supports.

In the add-on module RF-CONCRETE Surfaces, cut-out surfaces cannot be designed for any of the three model options. Instead, the internal forces at the column's boundary lines are used.

When you select *Elastic surface foundation* or *Elastic node support*, you have to enter further data for the column. Select the *Material of column* from the list of already defined materials, or create a new column material (see chapter 4.3, page 60).

To determine the spring stiffnesses, specifications to the *Support conditions at column head* and *column base* are required. If you tick the check box to *Activate shear stiffness*, the shear stiffness will affect also the constants of the *Support Springs Due to Column*.

The nodal springs determined from the entered parameters are shown in the dialog section to the right.

Support or spring

To define a support, select the corresponding option in the dialog box or table. The check mark indicates that the corresponding degree of freedom is blocked and the node displacement in the corresponding direction is not possible.

If you don't want to define supports, clear the corresponding check box. Then, RFEM sets the constant of the translational spring to zero in the *Nodal Support* dialog box. It is always possible to modify the spring constant in order to represent an elastic support of the node. In the table, enter the constant directly into the table column.

The spring stiffnesses must be entered as design values.

Assigning nonlinear support properties is described below.

Restraint or spring

Restraints are defined in a similar way as supports. Again, the check mark indicates that the corresponding degree of freedom is blocked and the node displacement in the corresponding direction is not possible. The constants for rotational springs can be defined as soon as the check boxes are cleared. In the table, enter the spring constant directly into the corresponding table column.

The dialog box *New Nodal Support* (see Figure 4.84, page 93) provides buttons for different support types, making the definition of degrees of freedom easier.

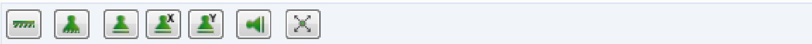


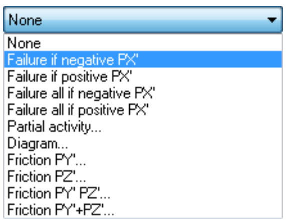
Figure 4.89: Buttons in the dialog box *New Nodal Support*

The buttons have the following functions used for support properties:

Button	Support type
	Rigid
	Hinged with restraint about Z'
	Sliding in X' and Y' with restraint about Z'
	Sliding in X' with restraint about Z'
	Sliding in Y' with restraint about Z'
	Sliding in Z' and Y' with restraint about Z'
	Free

Table 4.3: Buttons in *Nodal Support* dialog box

Nonlinearities



To control in detail the transfer of internal forces, it is possible to provide nodal supports with nonlinear properties. The list of nonlinearities includes the following options:

- Failure of component if support force or moment is negative or positive
- Complete failure of support if support force or moment is negative or positive
- Partial activity
- Diagram
- Friction depending on remaining support forces

The nonlinear properties can be accessed in the dialog box and table by using the list (see Figure 4.84 and Figure 4.85). In this way, you can define for each support's degree of freedom whether and which forces or moments are transferred at the supported node.

Nonlinear effective supports are displayed with a different color in the graphic. In the table, support elements having nonlinear properties are indicated by a blue check box.

Failure if support force/moment is positive or negative

Both options represent an easy control to decide whether the support can take only positive or negative forces/moments: If the internal force (force or moment) acts in the forbidden direction, the relevant component of the support will fail. The remaining retentions and restraints will still be effective.

The directions *positive* or *negative* refer to the forces or moments that are placed in the nodal support with regard to the respective axes (they do not refer to the reaction forces of the support). So signs are resulting from the direction of the global axes. If the global Z-axis is oriented downwards, the load case 'Self-weight' results in a positive support force P_Z.

Failure all if support force/moment is positive or negative

In contrast to the failure of a single component described above, the support fails completely as soon as the component is ineffective.

To access the following dialog boxes, use the buttons [Edit Nonlinearity] or [▼] to the right of the list available in the dialog box and table.

Partial activity

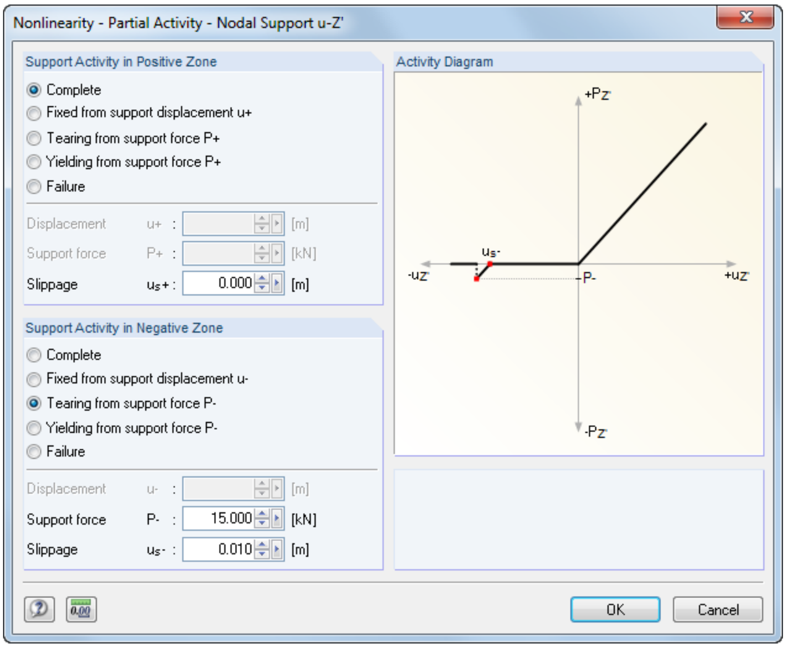


Figure 4.90: Dialog box *Nonlinearity - Partial Activity*

The support's effect can be defined separately for the *Positive* and *Negative Zone*. The sign rule is described in the previous paragraph. In addition to *Complete* activity or complete *Failure*, the support can be set to be effective only when it is displaced or rotated to a certain degree. In this case, a translational or rotational spring should be defined before.

Furthermore, *Tearing* (failure of support when exceeding a certain force or moment) as well as *Yielding* (effective only until force or moment is reached) can be set in combination with a *Slippage*.

Look at the dynamic dialog graphic called *Activity Diagram* to check the support properties.

Diagram

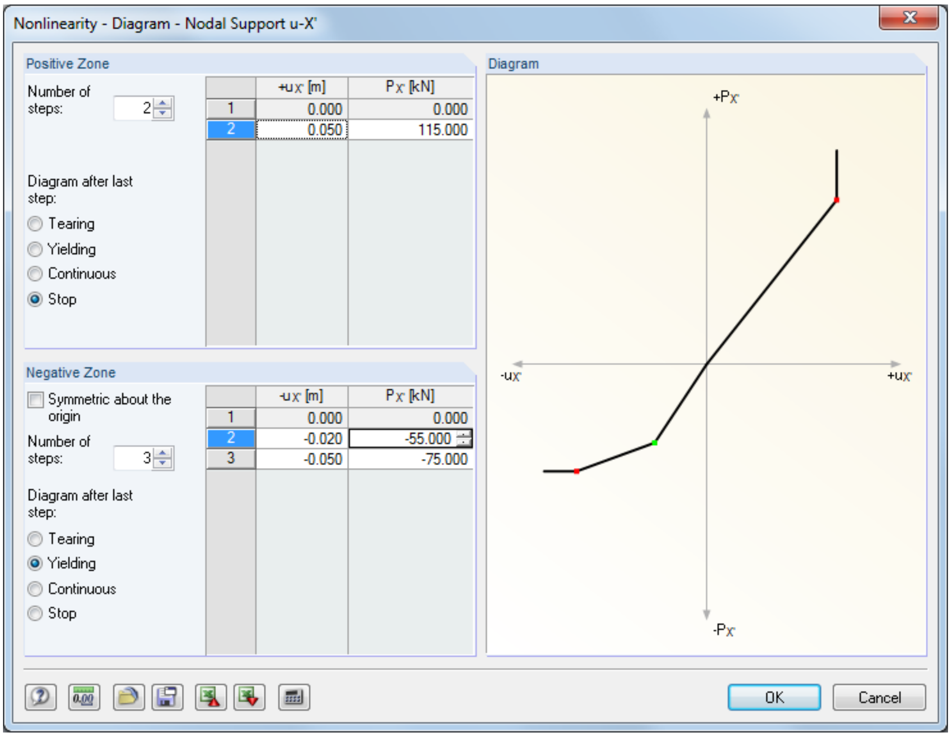
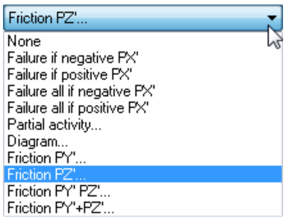


Figure 4.91: Dialog box *Nonlinearity - Diagram*

The support's effect can be defined separately for the *Positive* and *Negative Zone*. First, define the *Number of steps* (that means definition points) for the working diagram. Then, enter abscissa values of the displacements or rotations with corresponding support forces or moments into the list to the right.

You find different options for the *Diagram after last step*: *Tearing* for support failure when exceeding, *Yielding* for restricting the transfer to a maximum allowable support force or moment, *Continuous* as in the last step, or *Stop* for restricting to a maximum allowable displacement or rotation followed by a rigid or restrained support activity.



Friction depending on support force

Use the four friction options to set the transferred support forces in relation to the compressive forces acting in another direction. Depending on your selection, the friction depends on only one support force or on the total force of two support forces acting simultaneously.

Click the [Edit] dialog button to open a dialog box where you can define the *Friction Coefficient* μ.

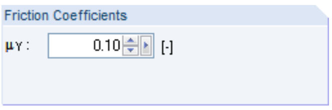


Figure 4.92: Dialog box *Friction in uX'* (dialog section)

The following relation exists between axial force and friction force of the support:

$$P_{\text{Support}} = \mu \cdot P_{\text{Axial force}}$$

Equation 4.11

4.8 Line Supports

General description

Line supports describe the boundary conditions of all FE nodes available along a line. Displacements and rotations on these internal nodes can be prevented or limited by translational and rotational springs.

You can assign nonlinear properties to displacements of line supports so that supports will be ineffective in case of tension or compression.

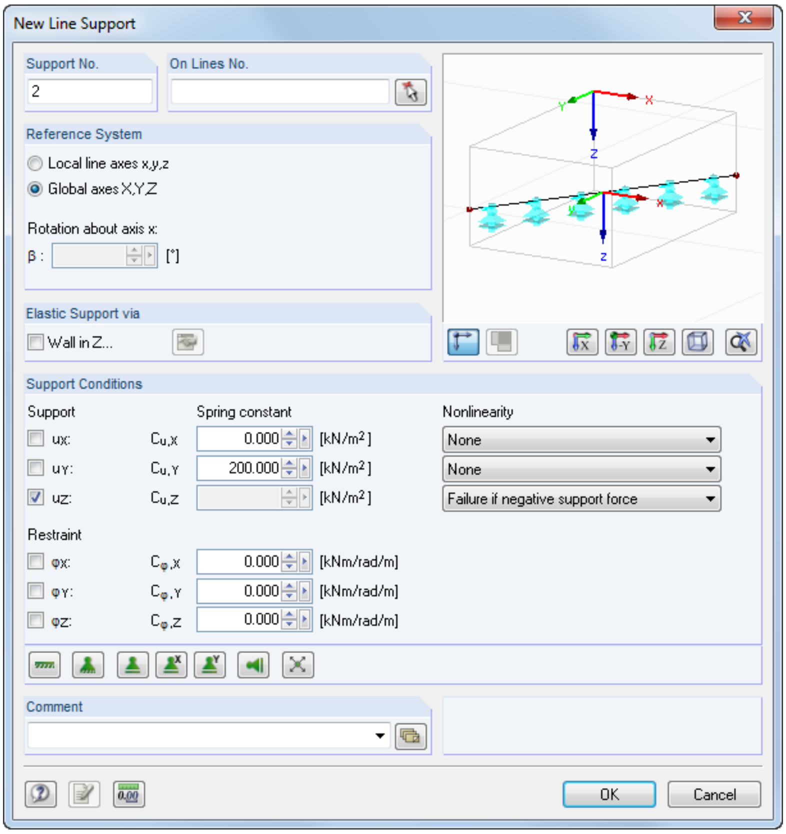


Figure 4.93: Dialog box *New Line Support*

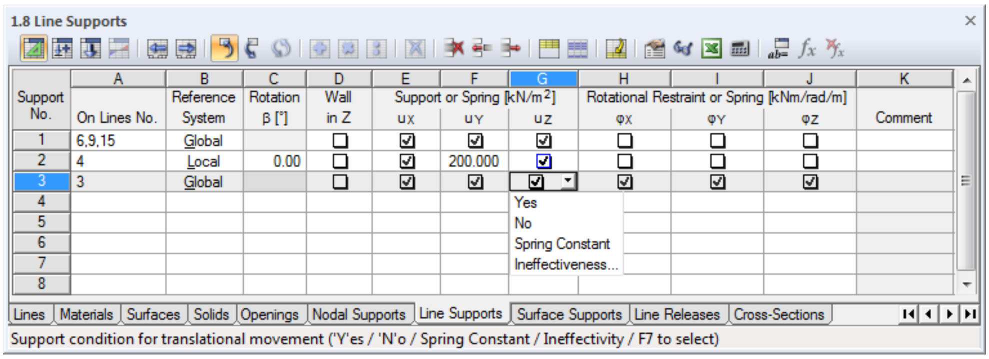


Figure 4.94: Table 1.8 *Line Supports*

To open the following dialog box, open the **Insert** menu, point to **Model Data** and **Line Supports**, and then select **Graphically**, or use the toolbar button shown on the left:

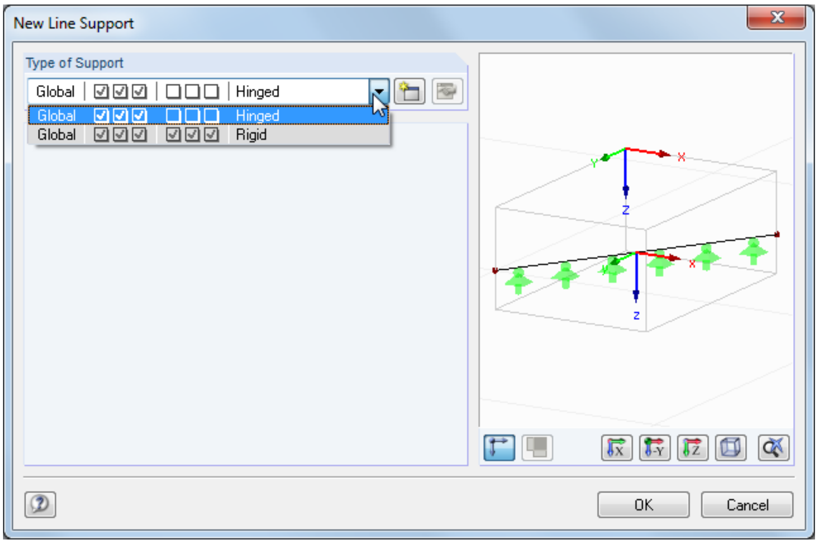


Figure 4.95: Dialog box *New Line Support*

The support types *Hinged (YYY NNN)* and *Rigid (YYY YYY)* are predefined and can be selected from the list. After clicking the [OK] button you can assign the selected support type to lines in the graphic.

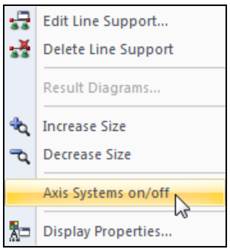
Use the [New] button to create another type of support. The dialog box shown in Figure 4.93 appears.

On lines

Line supports can be defined only on lines that belong to a surface or a solid. Enter the line number into the table column or input field of the dialog box. You can select it also graphically.

Reference system

The support parameters can be related to the *Local* line axes x, y and z or the *Global* axes X, Y and Z. Indexes in the dialog section *Support Conditions* as well as headlines of table columns E to J are changing depending on the selected setting.



Context menu of line support

The display of the local axis system of lines including numbering can be set in the *Display* navigator. You can also use the context menu of a line support.

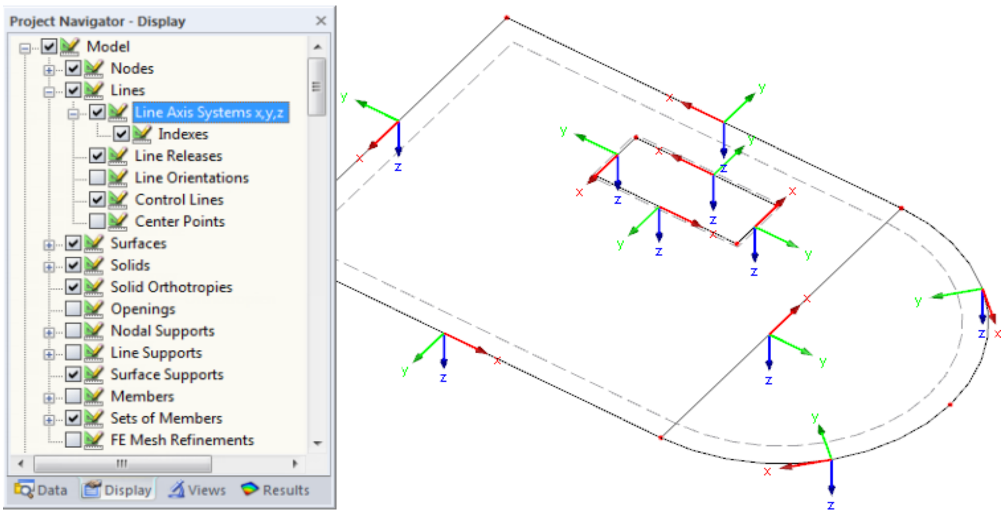


Figure 4.96: Activating the local line axis systems x,y,z in the *Display* navigator

Support rotation

It is possible to rotate the axis system of a local line support. The *Rotation* about a positive angle β rotates the support clockwise around the positive line axis x.

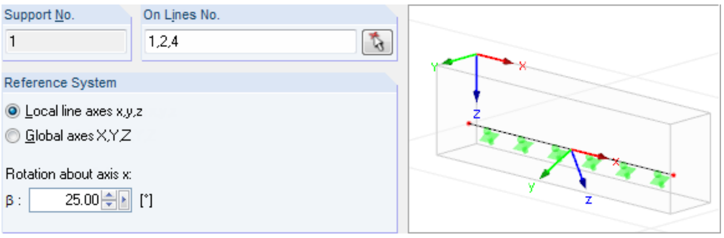


Figure 4.97: Support rotation

The entered support rotation is shown in the dynamic dialog graphic.

When the calculation is complete, it is possible to evaluate support reactions of a rotated line support in relation to the global and the local axis system.

Wall in Z

When a surface is supported by a wall, it is considered as an elastic support that depends on the stiffness of the wall. A fixed line support would not represent flexibility properly. For such a support type you can define a *Wall*: RFEM will calculate the constants of translational and rotational springs from the wall's material and geometry. This option is especially useful for 2D plates in order to avoid singularities which would occur for a rigidly supported line.

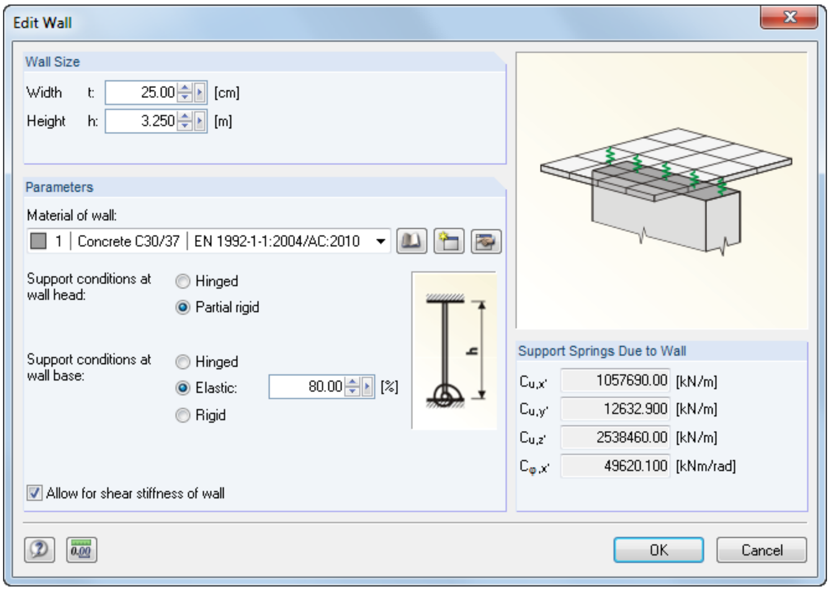


Figure 4.98: Dialog box *Edit Wall*

In the dialog section *Wall Size*, you define the geometry. In addition to the *Width t*, the *Height h* affects the constants of the translational and rotational springs.

In the dialog section *Parameters*, you select the *Material of wall* from the list of already defined materials. You can also create a new wall material (see chapter 4.3, page 60).

To determine the spring stiffnesses, specifications for *Support conditions at wall head* and *wall base* are required. If you tick the check box to *Allow for shear stiffness*, the shear stiffness will also affect the constants of the *Support Springs Due to Wall*.

The spring constants determined from the entered parameters are shown in the dialog section to the right.

The widths of the wall are displayed on the supported line in the graphic.

Support or spring

To define a support, select the corresponding option in the dialog box or table. The check mark indicates that the relevant degree of freedom is blocked and the line displacement in the respective direction is not possible.

If you don't want to define supports, clear the corresponding check box. RFEM sets the constant of the translational spring to zero in the *Line Support* dialog box. It is always possible to modify the spring constant in order to represent an elastic support of the line. In the table, enter the constant directly into the table column.

The spring stiffnesses are considered as design values.

Assigning a failure criterion is described below.

Restraint or spring

Restraints are defined in a similar way as supports. Again, the check mark indicates that the relevant degree of freedom is blocked and the line displacement in the respective direction is not possible. The constants for rotational springs can be defined as soon as the check boxes are cleared. In the table, enter the spring constant directly into the corresponding table column.

The dialog box *New Line Support* (see Figure 4.93, page 100) provides buttons for different support types, making the definition of degrees of freedom easier.

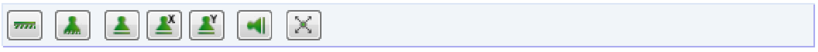


Figure 4.99: Buttons in the dialog box *New Line Support*

The buttons have the following functions used for support properties:

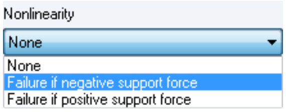
Button	**Support type**
	Rigid
	Hinged with restraint about Z'
	Sliding in X' and Y' with restraint about Z'
	Sliding in X' with restraint about Z'
	Sliding in Y' with restraint about Z'
	Sliding in Z' and Y' with restraint about Z'
	Free

Table 4.4: Buttons in *Line Support* dialog box

Nonlinearities

You can assign the following nonlinear features to the supports or translational springs of a line support:

- Failure if support force is positive
- Failure if support force is negative



The nonlinear properties can be accessed in the dialog box and table by using the list (see Figure 4.93 and Figure 4.94). Use these settings to define for each element of the support whether only positive or negative forces are transferred on the supported line.

Positive or *negative* refers to the forces that are introduced to the support in direction of the respective axes (they do not refer to the reaction forces of the line support). So, signs result from the direction of the local or global axes. For example, if the local z-axis of a line is directed downwards, the load case 'Self-weight' results in a positive support force $p_{Z'}$.

Nonlinear effective supports of lines are displayed with a different color in the graphic. In the table, you can recognize support elements with a failure criterion by a blue check box.

4.9 Surface Supports

Theoretical background

An elastic surface foundation represents an elastic support of all 2D elements of a surface.

In the WINKLER foundation model, the soil is assumed to be an ideal liquid upon which the slab is floating. This model is based on significant differences between the moduli of elasticity for concrete and (linearized) soil, which are typically represented by 1000:1 and more. Mathematically, the assumption by Winkler is the following:

$$p_z = C_z \cdot w_z$$

Equation 4.12

In each point, the contact pressure p_z is put into relation to the displacement w_z by means of the foundation constant C_z. However, the assumption implies that each point is displaced independently of all other nodes of the floor plan. Thus, the surrounding soil is irrelevant for the deformation of a surface (Figure 4.100a).

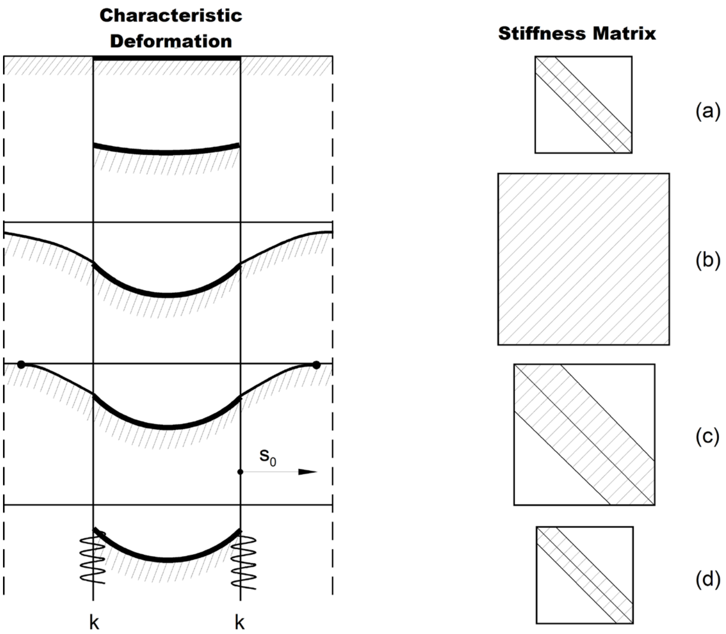


Figure 4.100: Correlation between deformation figure and bandwidth of the stiffness matrix

This rudimentary foundation model does no longer match modern requirements.

An improved type of modeling elastic foundations is based on the *stiffness modulus approach*. In this model, the soil is considered as an elastic half-space with all its nodes correlating mathematically and mechanically. This results in an “infinite“ subsidence basin with damping effects. The disadvantage of this foundation soil model is an enormous system matrix (Figure 4.100b).

The RFEM algorithm makes use of the effective soil model according to KOLAR / NEMEC, unifying the advantages of both models. This foundation soil model is based on the theory of PASTERNAK [8]:

- For the slab that is in contact with the soil, only the mechanical properties of the nonlinear elastic or plastic half-space in the contact gap are important. Therefore, the three-dimensional foundation effects are condensed in the contact gap, that means transferred to a 2D problem.
- The WINKLER model carries out the 2D condensation (see Equation 4.12), though it is defective in terms of energy. By including a second coefficient C_v for the shear capacity of soil, the coaction of soil beyond the slab's edge is established. A natural subsidence basin with limited dimensions is formed, as it can be verified in reality.
- A two parametric system (C_u, C_v) is created. $C_{u,z}$ corresponds approximately to the WINKLER foundation constant and can be applied in this way in practical calculations. In detail, the entire system consists of five parameters: $C_{u,x}$, $C_{u,y}$, $C_{u,z}$, $C_{v,x}$ and $C_{v,y}$.

Figure 4.100c shows this soil model in comparison. Numerically, the FE model is as stable as the WINKLER model. However, incorporating the soil elements in the subsidence basin results in a larger system matrix.

The KOLAR/NEMEC model has been enhanced as well. Experience has shown that soil elements can be eliminated from the system with the help of appropriate measures. The *effective soil model* implemented in RFEM is symbolically shown in Figure 4.100d. Thus, the disadvantage of the larger system matrix is eliminated. Find a detailed description of the effective foundation soil model in [4].

The surrounding soil ("wedge of soil") is eliminated from the surface model by converting its rigidity into an elastic boundary line and corner node support.

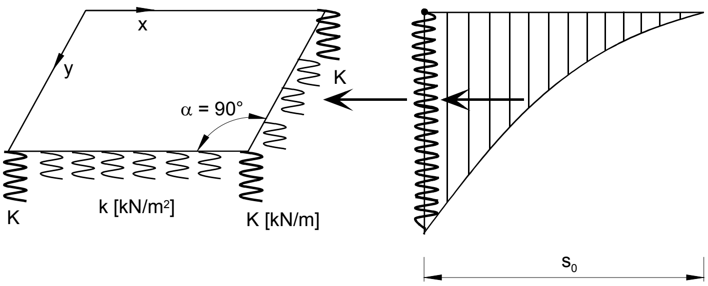


Figure 4.101: Converting the surrounding soil into line and corner nodal supports

In a first approximation, the spring constants *k* and *K* of the line and corner node support are calculated according to the following equations:

$$k = \sqrt{C_{u,z} \cdot C_{v,perpendicular}}$$

Equation 4.13: Spring constant of line support

$$K = \frac{C_{v,x} + C_{v,y}}{4}$$

Equation 4.14: Spring constant of corner nodal support

In Equation 4.13 you have to insert the parameter C_v that acts perpendicular to the border line.

Equation 4.14 is used for corners with an angle of α = 90° (see [17] for other angle sizes). Angles larger than 90° result in smaller values of *K*. However, if α = 0°, *K* = 0 as well.

The springs determined in this way must be arranged, in addition to the elastic foundation of the surface, as line and nodal supports in the model.

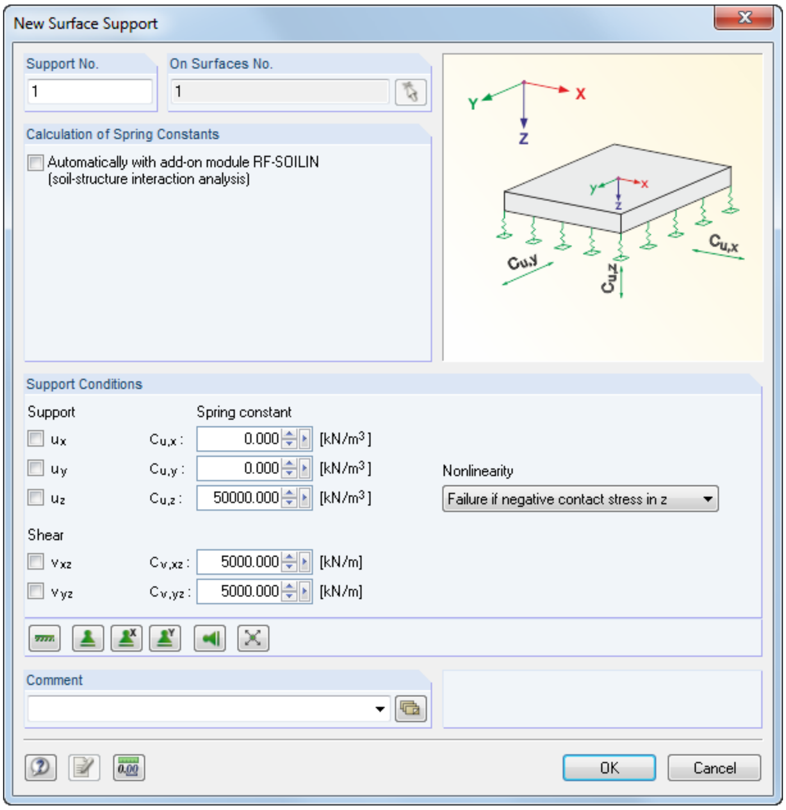


Figure 4.102: Dialog box *New Surface Support*

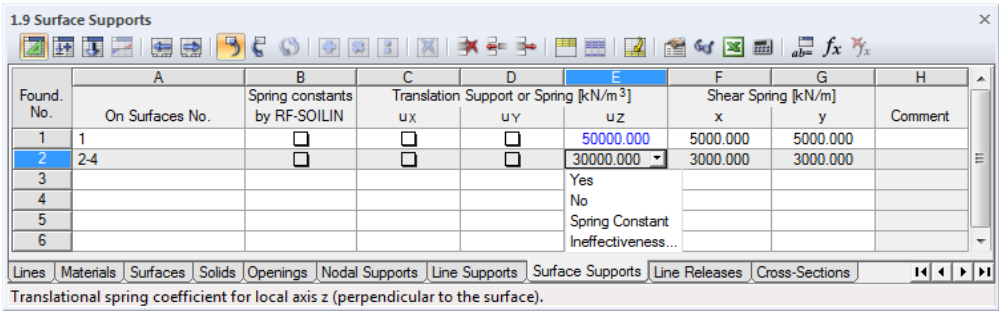


Figure 4.103: Table 1.9 *Surface Supports*

On surfaces

Enter the numbers of the surfaces to be supported into the table column or the input field of the dialog box. You can select them also graphically.

Spring constants with RF-SOLIN

Every soil has more or less distinctive nonlinear elastic or plastic features. To determine foundation coefficients easily, use Dlubal's add-on module RF-SOILIN. The program carries out calculations of settlements which are based on load actions and results of test borings. Then, it determines the spring coefficients in each finite element. In the add-on module, different layers of soil can be considered at several recording points.

If the option is selected and no results of RF-SOILIN are available, the foundation coefficients will be determined before the RFEM calculation is carried out.

Supports u_x /u_y /u_z or springs $C_{u,x}$ / $C_{u,y}$ / $C_{u,z}$

Directions of supports or springs refer to the surfaces' local axes *x*, *y* and *z*. Use the *Display* navigator or the context menu of a surface to show them in the graphic (see Figure 4.73, page 83).

The spring stiffnesses must be entered as design values.

If the support acts perpendicular to the surface, enter the support or spring constant into the input field $\mathbf{C_{u,z}}$. The parameter is practically equal to the WINKLER modulus of foundation C_z. It can be taken from a soil expertise.

The parameters $\mathbf{C_{u,x}}$ and $\mathbf{C_{u,y}}$ represent translational springs describing the foundation's resistance against displacement in the surface directions *x* or *y*. In case of a floor slab, they define the (load-independent) resistance in the horizontal directions.

In the graphic, springs are always placed in direction of the positive surface axis *z*. If spring symbols are on the "wrong" side of the surface, you can change the orientation of the local *z*-axis quickly: Right-click the surface to open the context menu and select *Reverse Local Axis System*. This option is only available for 3D models, not for planar models. When changing settings, please note that the failure criterion will change the action direction as well.

Rigid supports make it possible, for example for symmetrical solid models, to represent only one part of the model. In this way, you can increase the speed of calculation considerably.

Shear springs $C_{v,x}$ / $C_{v,y}$

These input fields are used to consider the shear capacity of soil in direction of the surface axes x or y. In most cases, the PASTERNAK constant C_v lies between $0.1 \cdot C_z$ (minor shear capacity) and $0.5 \cdot C_z$ (medium shear capacity). Generally, $C_{v,x} = C_{v,y}$ can be applied.

The central idea of the Effective Model is to link the parameters $C_{u,z}$ and C_v by means of the coefficient *s* according to the following equations.

$$C_{v,x} = C_{u,z} \cdot s_x^2$$

Equation 4.15: Shear spring constant $C_{v,x}$

$$C_{v,y} = C_{u,z} \cdot s_y^2$$

Equation 4.16: Shear spring constant $C_{v,y}$

The value *s* is an analogy of the elastic length for beams with elastic foundation. This empiric equation was derived from settlement measurements (see Figure 4.100c, page 104):

$$s_0 = 4.0 \text{ s to } 5.0 \text{ s} \qquad \text{Average:} \qquad s_0 = 4.5 \text{ s}$$

Equation 4.17: Subsidence basin s_0

The subsidence basin s_0 mentioned in Equation 4.17 is understood rather in an energetic than geometric sense of term. In practical construction, s_0 is defined as distance from the plate edge where settlements fall below 1 % of the foundation edge values. If a reference value for s_0 is known, *s* is calculated according to Equation 4.17. As a result, we get the value of C_v according to Equation 4.15 and Equation 4.16. If no measurings are available, but determining or estimating the value of C_v from the soil type is possible, you can derive the value of *s* as follows:

$$s_x = \sqrt{\frac{C_{v,x}}{C_{u,z}}} \qquad \text{or} \qquad s_y = \sqrt{\frac{C_{v,y}}{C_{u,z}}}$$

Equation 4.18

The determination of C_v is the main problem when applying the foundation model according to PASTERNAK. If C_v approaches zero, this model changes over to the energetic defective WINKLER model. If C_v approaches infinity, reaching the subsidence basin s_0 becomes endless as well.

Then, the energy of the soil's deformation is an infinite expression, settlement changes as well as settlements approach zero. Therefore, unrealistically high values of C_v lead to numerical problems in the FE algorithm.

For loose sand for example, C_v approaches zero. For compact types of rock, however, it can be assumed near 1.0 · $C_{u,z}$.

KOLAR [17] provides a summarizing table with the following orientation values. Please note that they do not substitute the values of a soil expertise.

Soil consistence	**$C_{u,z}$**	**Shear capacity C_v**		
		None	**Medium**	**High**
	[kN/m³]	[kN/m]	[kN/m]	[kN/m]
very soft	1,000	0	500	1,000
medium-dense	10,000	0	5,000	10,000
compact	100,000	0	50,000	100,000

Table 4.5: Guide values for $C_{u,z}$ and C_v

The dialog box *New Surface Support* (see Figure 4.102, page 106) provides buttons for different support types, making the definition of degrees of freedom easier.

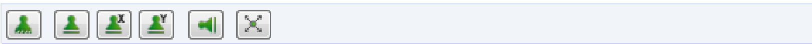


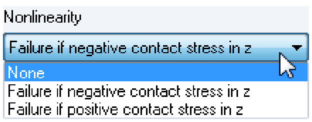
Figure 4.104: Buttons in the dialog box *New Surface Support*

The buttons have the following functions used for support properties:

Button	**Support type**
	Rigid
	Sliding in x and y
	Sliding in x
	Sliding in y
	Sliding in z
	Free

Table 4.6: Buttons in dialog box *Surface Support*

Nonlinearity



The support can be excluded for positive or negative contact stresses occurring in direction of the displacement u_Z: The foundation fails for example in case of lifting forces. Specify settings by means of the list available in the dialog box or table (see Figure 4.103, page 106).

Positive or *negative* refer to the stresses acting in direction (or opposite direction) of the local z-axis of the surface: Positive contact stresses are produced when a floor slab is stressed by self-weight, and the global axis Z as well as the local axis z are both orientated downwards. If the surface axis z was orientated upwards, the contact stress would be negative.

The display option of the local surface axes is shown in Figure 4.73, page 83.

Nonlinear effective surface supports are displayed with a different color in the graphic. In the table, characteristic values u_z of such supports are shown in blue.

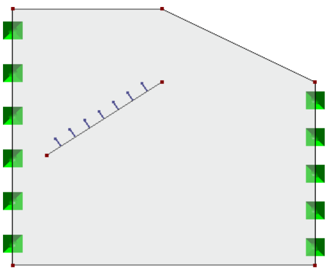
When a nonlinearity is existent, RFEM calculates the deformations and stresses by several iterations. The program finds out which finite elements become stress-free if the foundation is no longer active because of failure.

Please note that it may happen for result combinations with nonlinear acting supports that results are combined with locally-different support failure. In those cases, it is recommended to use load combinations (see example in Figure 5.28, page 194).

4.10 Line Releases

General description

Surfaces touching each other on one line are rigidly connected on it. With a line release you are able to exclude particular degrees of freedom from the transfer.



Line releases can be arranged on boundary lines of surfaces. In addition, they can be assigned to lines integrated in a surface as shown on the left.

A line release is an attribute of a surface, not of a line. Thus, it must be assigned to a surface. To assign the line release graphically,

select **Model Data** on the **Insert** menu, point to **Line Releases** and click **Assign to Lines Graphically**.

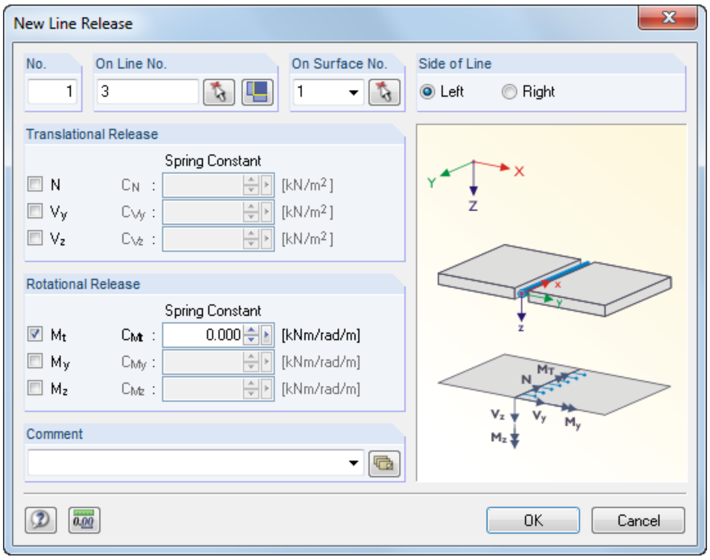
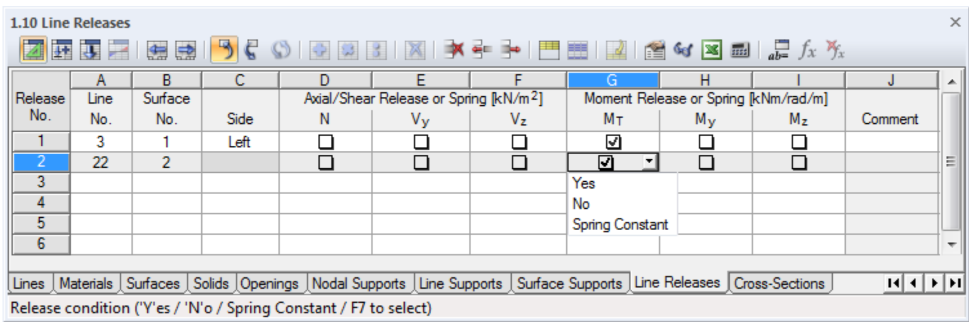


Figure 4.105: Dialog box *New Line Release*



1.10 Line Releases

Release No.	A Line No.	B Surface No.	C Side	D Axial/Shear Release or Spring [kN/m²] N	E Vy	F Vz	G Moment Release or Spring [kNm/rad/m] MT	H My	I Mz	J Comment
1	3	1	Left	☐	☐	☐	☑	☐	☐	
2	22	2		☐	☐	☐	☑	☐	☐	
3							Yes			
4							No			
5							Spring Constant			
6										

Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Cross-Sections

Release condition ('Y'es / 'N'o / Spring Constant / F7 to select)

Figure 4.106: Table 1.10 *Line Releases*

On line

Enter the number of the line on which you want to define the release. You can also use the list or select the line graphically. When you select the surface before you define the settings in the dialog box, you can import [All Boundary Lines] of the surface by clicking the blue button.

On surface

Assign the line release to a surface. As the release is a surface property, you can adjust it subsequently in the dialog box *Edit Surface*.

Side

The options in the dialog box, respectively the column in the table, are only accessible if the line is an integrated object of the surface. The arrangement of the release determines the way how the finite elements on the line sides are taken into account for stiffness.

To allocate the release *Left* or *Right* of the line, check the direction of the line (▲ in the figure below) and the direction of the local surface axis z. The following rule illustrates how to assign the side correctly: "Stand on the line with surface axis z pointing in direction of your feet. Then look in the direction of the line. Left and right are the directions of your arms".

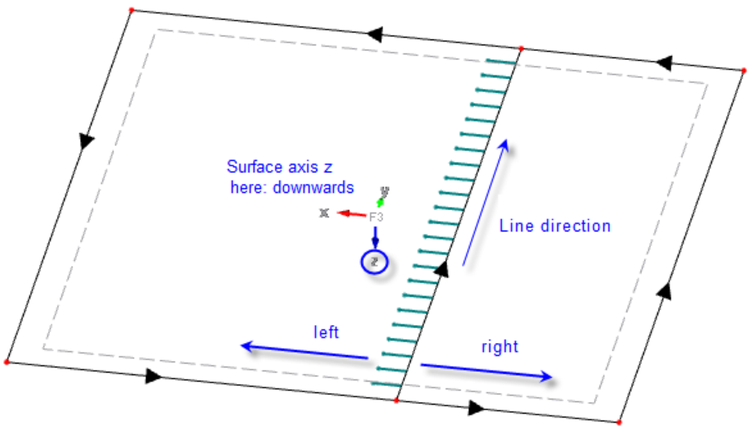


Figure 4.107: Definition of line sides

The side option is locked for the boundary line of a surface because the side of the line on which the release is effective is clearly defined by the assignment to the surface.

Axial/shear release or spring

The dialog input fields and table columns control the degrees of freedom for axial and shear forces. If a check box is ticked, it means that displacement in the relevant direction is possible, and thus the force won't be transferred. It is also possible to enter the constant of a translational spring.

The degrees of freedom are based on this definition of the axis system: the *x*-axis represents the direction the line, the *y*-axis the tangent of the surface plane; *z* points in the direction of the normal of the surface.

Moment release or spring

The degrees of freedom for moments refer to the local axis system of the hinge, too (*x*-axis in direction of the line, *y*-axis as tangent and *z*-axis as normal to the surface plane). The check mark means that the rotation is free so that the internal force is not transferred. It is also possible to enter the constant of a rotational spring.

The graphic in the dialog box *New Line Release* shows the direction of moments. For a "hinge joint" between two surfaces, the release type φ_x is to be applied, for example. Thus, a moment hinge about the longitudinal axis of the line is created.

4.11 Variable Thicknesses

General description

A variable thickness describes a linear decrease or increase of the surface thickness. Use variable thicknesses to model tapered surfaces. The variable thickness must be defined on three points in order to interpolate linearly between them.

A variable thickness is not entered directly but set as parameter when defining a surface. When you create a surface, define the *Thickness* as **Variable** (see chapter 4.4, page 81). Then, the [Edit] buttons shown on the left become active in the dialog box and table.

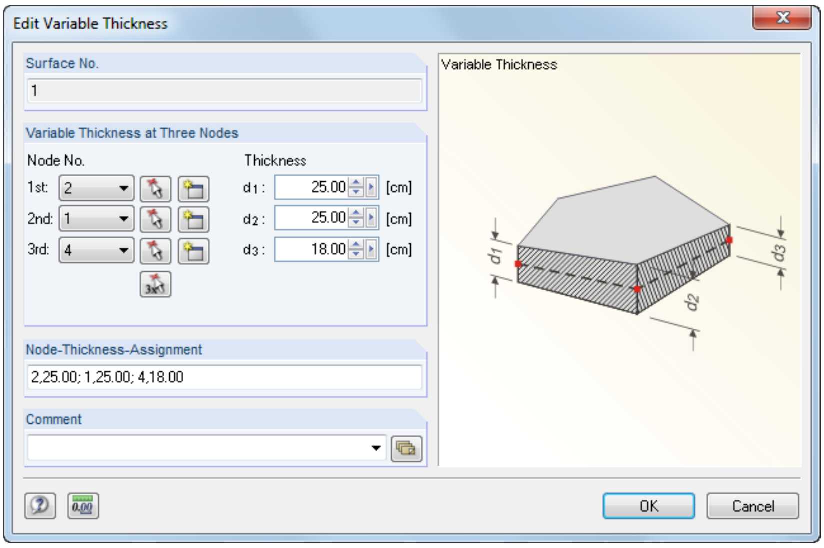
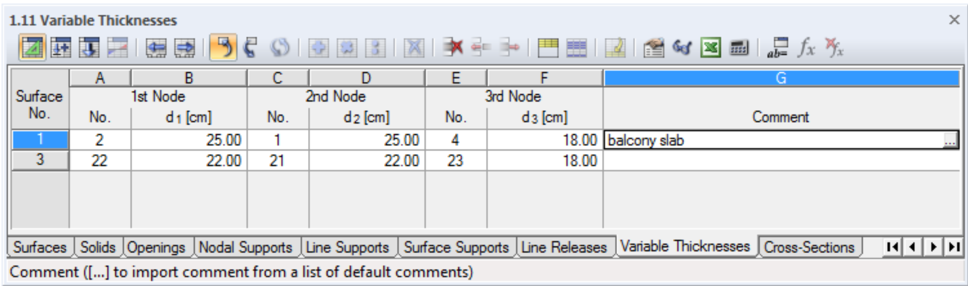


Figure 4.108: Dialog box *Edit Variable Thickness*


1.11 Variable Thicknesses

Surface No.	A 1st Node No.	B 1st Node d_1 [cm]	C 2nd Node No.	D 2nd Node d_2 [cm]	E 3rd Node No.	F 3rd Node d_3 [cm]	G Comment
1	2	25.00	1	25.00	4	18.00	balcony slab
3	22	22.00	21	22.00	23	18.00	

Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases | Variable Thicknesses | Cross-Sections

Comment ([...] to import comment from a list of default comments)

Figure 4.109: Table 1.11 *Variable Thicknesses*

Surface

Variable thicknesses can be used only for plane surfaces. They cannot be applied, for example, to curved surfaces.

Variable thickness at three nodes

To define a variable thickness, specify three nodes so that RFEM can interpolate linearly between them. You can select any nodes within the plane of the surface for the thickness definition. They do not need to belong to the surface, but it is necessary that FE nodes can be generated on theses definition points.

Select the three nodes from the list, or use the [↖] function to select them graphically. It is also possible to create [New] nodes. Then, assign the corresponding *Thickness d*.

The dialog section *Node-Thickness-Assignment* represents a short input overview. Node numbers and thicknesses are separated by comma, single node-thickness pairs by semicolon.

It is possible to display the distribution of surface thicknesses in the rendering mode to check data: Select the option *Filled incl. thickness* in the *Display* navigator.

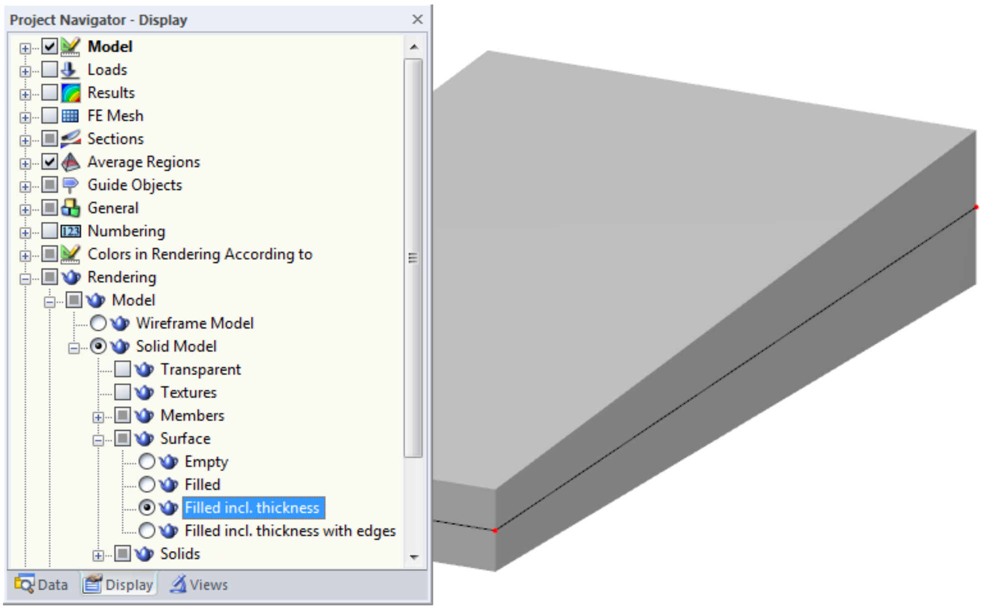


Figure 4.110: *Display* navigator: *Rendering* → *Model* → *Solid Model* → *Surface* → *Filled incl. thickness*

4.12 Orthotropic Surfaces

General description

Orthotropic surfaces have different stiffnesses in direction of the local surface axes x and y. Use orthotropic surface properties to model for example glued-laminated girders or ribbed floors. Orthotropic properties can be set for plane and quadrangle surfaces.

You can define orthotropic properties by material (material orthotropy with invariable geometry), by geometry (irregular shape of surface with isotropic material) or in a combination of both.

The following figure shows the general stiffness matrix of an orthotropic surface in RFEM.

$$\begin{Bmatrix} m_x \\ m_y \\ m_{xy} \\ v_x \\ v_y \\ n_x \\ n_y \\ n_{xy} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & 0 & 0 & D_{16} & D_{17} & D_{18} \\ & D_{22} & D_{23} & 0 & 0 & D_{26} & D_{27} & D_{28} \\ & & D_{33} & 0 & 0 & D_{36} & D_{37} & D_{38} \\ & & & D_{44} & D_{45} & 0 & 0 & 0 \\ & & & & D_{55} & 0 & 0 & 0 \\ & & \text{sym.} & & & D_{66} & D_{67} & D_{68} \\ & & & & & & D_{77} & D_{78} \\ & & & & & & & D_{88} \end{bmatrix} \begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

Elements for bending and torsional rigidity
Elements for shear
Membrane elements
Eccentricity elements

Figure 4.111: Matrix with stiffness coefficients

Orthotropic surfaces can be calculated according to linear static analysis, second-order analysis or large deformation analysis. In case of matrices with pure membrane coefficients, only a large deformation analysis is possible.

Please find detailed information about *Orthotropy* in an English document that you can request from DLUBAL SOFTWARE GMBH.

An orthotropy is not entered directly but set as parameter when defining a surface. When you create a new surface, define the *Stiffness* as **Orthotropic** or **Membrane orthotropic** (see chapter 4.4, page 80). Then, the [Edit] buttons shown on the left become active in the dialog box and table.

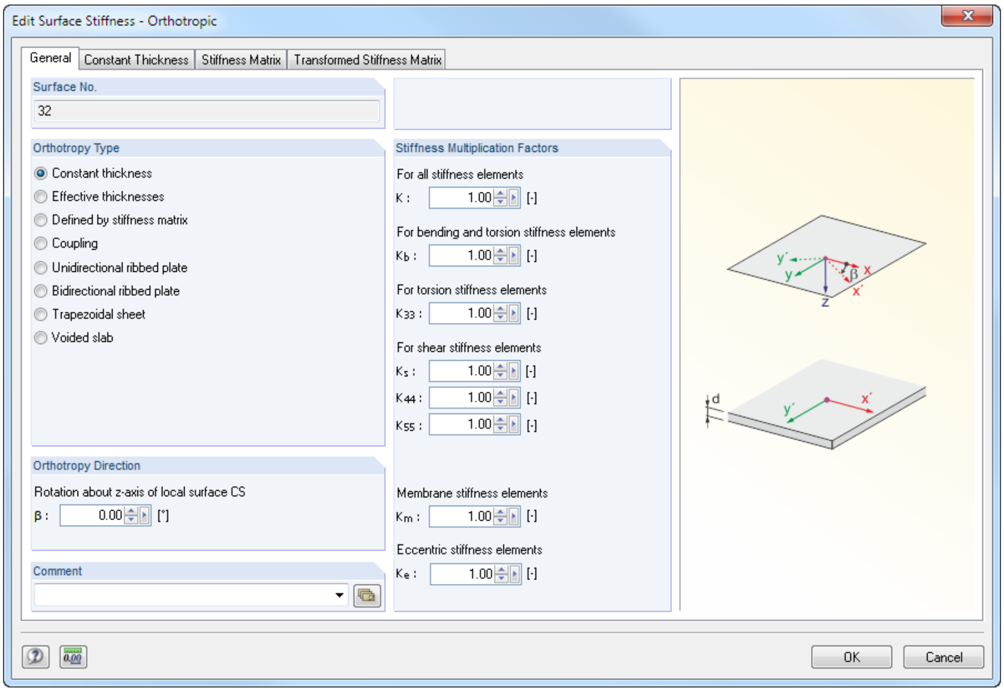
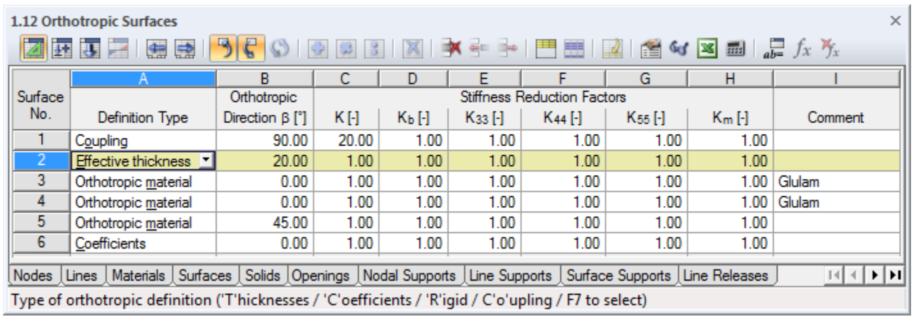


Figure 4.112: Dialog box *Edit Surface Stiffness - Orthotropic*



1.12 Orthotropic Surfaces

Surface No.	A Definition Type	B Orthotropic Direction β [°]	C K [-]	D Kb [-]	E K33 [-]	F K44 [-]	G K55 [-]	H Km [-]	I Comment
1	Coupling	90.00	20.00	1.00	1.00	1.00	1.00	1.00	
2	Effective thickness	20.00	1.00	1.00	1.00	1.00	1.00	1.00	
3	Orthotropic material	0.00	1.00	1.00	1.00	1.00	1.00	1.00	Glulam
4	Orthotropic material	0.00	1.00	1.00	1.00	1.00	1.00	1.00	Glulam
5	Orthotropic material	45.00	1.00	1.00	1.00	1.00	1.00	1.00	
6	Coefficients	0.00	1.00	1.00	1.00	1.00	1.00	1.00	

(Stiffness Reduction Factors: columns C–H)

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases

Type of orthotropic definition ('T'hicknesses / 'C'oefficients / 'R'igid / C'o'upling / F7 to select)

Figure 4.113: Table 1.12 *Orthotropic Surfaces*

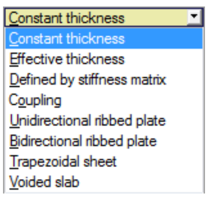
The dialog box is subdivided into several tabs which depend on the selected *Orthotropy Type*.

In the dialog section *Stiffness Multiplication Factors*, you can reduce stiffnesses either globally using the factor *K* or individually for bending, torsion, shear and membrane stiffness elements (see page 116).

In the *Stiffness Matrix* tab, the respective elements of the matrix are shown (see Figure 4.114).

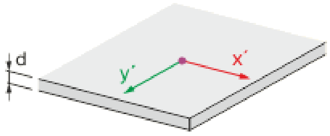
Stiffness matrix elements will be adjusted according to Equation 4.1 during the RFEM 4 import.

Definition



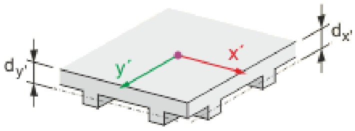
You can define orthotropic surfaces by material and geometry parameters or directly by coefficients of the local stiffness matrix. Depending on your specifications, tabs of the dialog box are changing.

The orthotropy types are described on the following pages. For each definition type you have to specify the *Thickness* that you want to apply for the self-weight determination.



Constant thickness

RFEM uses the orthotropic material properties that have been defined in the dialog box *Material Model - Orthotropic Elastic 2D* (see Figure 4.47, page 66). This type is appropriate only for homogenous surfaces of equal thickness whose material has distinctive orthotropic properties.



Effective thicknesses

In the dialog tab *Effective Thicknesses*, you can define different thicknesses in direction x' and y' to reproduce unequal stiffness conditions.

The self-weight is not determined from the thicknesses entered in this dialog box, but RFEM uses the surface thickness entered in the dialog box *Edit Surface* or in table 1.4 *Surfaces*.

RFEM shows you the moduli of elasticity and shear for the material that is used (see chapter 4.3, page 66) so that you can check corresponding data. Alternatively, it would be possible to control the orthotropic properties by means of material settings and to define the same thicknesses for the directions x' and y'.

Furthermore, RFEM does not calculate any stresses for orthotropic surfaces: The different stiffness coefficients would cause "blurred" results because they refer to an average value of the thickness. These stresses do not correspond to the orthotropy model.

Stiffness matrix

The coefficients of the local stiffness matrix can be defined manually.

With this option you can adjust also generated coefficients (for example a coupling or ribbed floor) by user-defined settings.

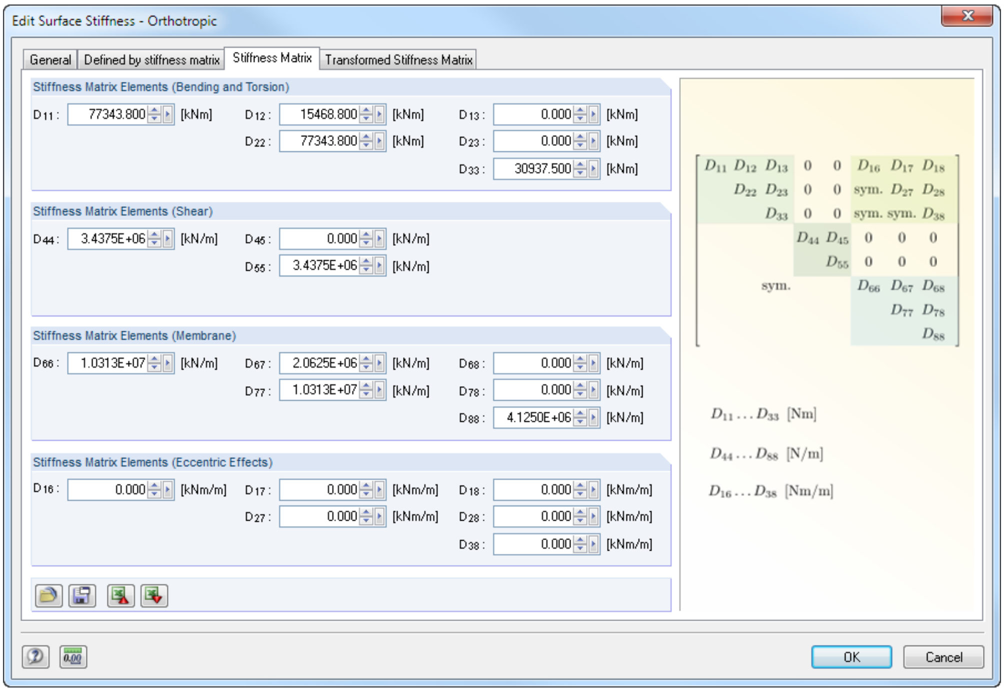


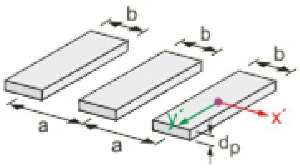
Figure 4.114: Dialog box *Edit Surface Stiffness - Orthotropic*, tab *Stiffness Matrix*

The [Info] button informs you about the relevance of coefficients in the stiffness matrix.

If the axes of the orthotropy are not consistent with the axes of the coordinate system of elements, you have to transform the matrices (see [16], page 305-313).

Furthermore, appropriate adjustments of coefficients are required if you find out, when checking data before performing calculations, that the stiffness matrix is not positively definite.

Coupling



Use this setting to model connections between surfaces or members which are represented by coupling elements consisting of isotropic material.

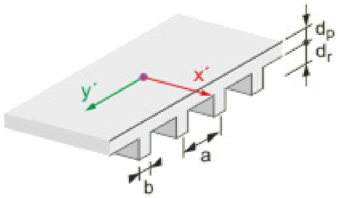
In the dialog tab *Coupling*, enter the parameters coupling thickness d_p, coupling spacing *a* and coupling width *b* according to the scheme. A realistic coupling model is given when the distance *a* is larger than the width *b* of the coupled elements.

The effective thickness d* is determined according to the following equation:

$$d^* = d_p \cdot \frac{b}{a}$$

Equation 4.19

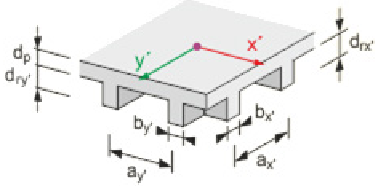
Unidirectional ribbed plate



The orthotropic properties of a ribbed floor are based on the principle of an uniaxially stressed T-beam ceiling. RFEM determines stiffnesses from the geometry parameters of the slab thickness d_p, rib height d_r, rib spacing *a* and rib width *b* which you have to specify according to the scheme shown in the dialog tab *Unidirectional Ribbed Plate*.

Please note that crack development (for example state II for concrete) is <u>not</u> taken into account when the stiffnesses are determined. Only isotropic materials are allowed.

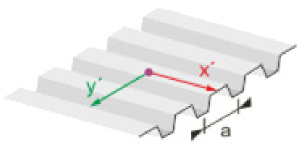
Bidirectional ribbed plate



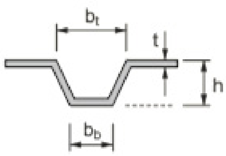
This type of ceiling is characterized by webs crossing perpendicularly in a uniform grid, subdividing the floor into coffers. The orthotropic properties can be described like for ribbed floors by geometry (see above). You need to specify the stiffness parameters for two directions.

In the dialog tab *Bidirectional Ribbed Plate*, you specify the parameters for slab thickness d_p, rib height d_r, rib spacing *a* and rib width *b* for the directions x' and y' according to the scheme.

Trapezoidal sheet



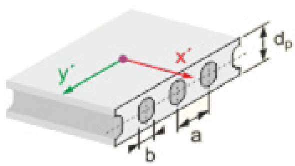
The possibility to reproduce trapezoidal sheets as surfaces with orthotropic properties facilitates modeling surfaces considerably. RFEM determines the stiffness coefficients from the geometry parameters of the cross-section. Only isotropic materials are allowed.



In the dialog tab *Trapezoidal Sheet*, you specify the parameters for the sheet thickness *t*, total profile height *h*, rib spacing *a*, top flange width b_t and bottom flange width b_b according to the scheme.

Only isotropic materials are allowed like for all other geometric orthotropies (effective thicknesses, ribbed floor, coffered ceiling, hollow concrete floor).

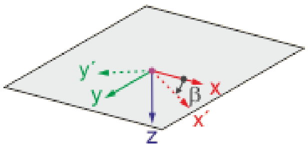
Voided slab



Hollow units built in a ceiling reduce the self-weight but produce orthotropic structural behavior. RFEM determines stiffnesses from the geometry parameters of the slab thickness d_p, void spacing *a* and void diameter *b* which you have to specify according to the scheme shown in the dialog tab *Voided Slab*.

Please find detailed information about stiffness components determined from geometric specifications in an English document that you can request from DLUBAL SOFTWARE GMBH.

Orthotropy direction β



The orthotropic direction refers to the surface's local axes *x* and *y*. The angle β describes the rotation of the x'-axis to the local x-axis of the surface. It is responsible for transforming the matrices available in the dialog tab *Transformed Stiffnes Matrix*.

Use the *Display* navigator or the context menu of the surface to display the coordinate systems of the surface in the graphic.

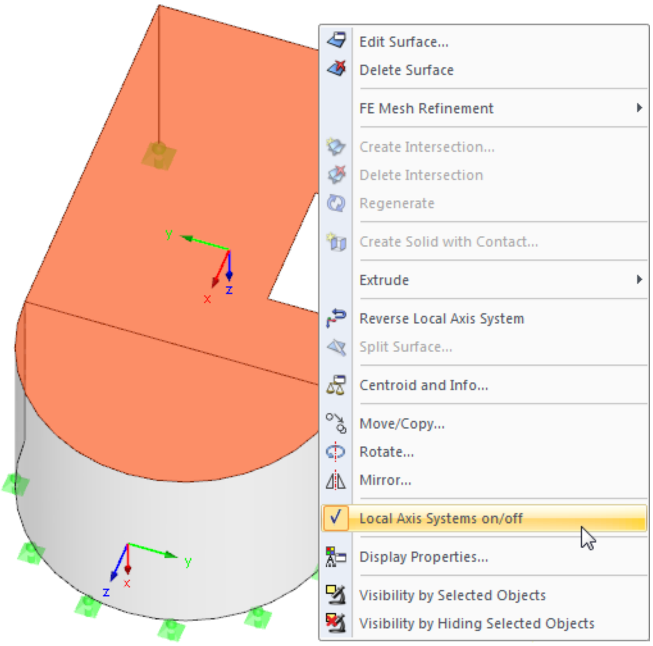


Figure 4.115: Surface context menu used to display the local surface axis systems **xyz**

The positive angle β is defined clockwise around the positive local z-axis of the surface.

Stiffness multiplication factors

You can reduce stiffnesses either globally using the factor *K*, or individually for bending, torsion, shear and membrane elements of the matrix (see Figure 4.111, page 112).

All stiffness elements

All coefficients of the stiffness matrix are globally multiplied by a factor.

Bending stiffness elements

Use the factor K_b to adjust the coefficients D_{11}, D_{12}, D_{22} and D_{33} of the stiffness matrix; they represent the bending components. It is allowed to enter factors between 0 (no flexural resistance) and 1 (full flexural resistance).

Torsion stiffness elements

With the value entered in the input field K_{33}, you control the factor for torsional rigidity D_{33} about the axes x' and y'. The input ranges from 0 (no twisting rigidity) to 1 (full twisting rigidity). For example for composite constructions with semi-rigid connections a small value is recommended.

Shear stiffness elements

Factors K_{44} and K_{55} affect the coefficients D_{44} and D_{55} of the matrix (components for shear).

Membrane stiffness elements

Use the factor K_m to adjust the coefficients D_{66}, D_{77}, D_{67} and D_{88} of the stiffness matrix; they represent the axial force components. It is allowed to enter factors between 0 (no membrane stiffness) and 1 (full membrane stiffness).

4.13 Cross-sections

General description

Before you can enter a member, a cross-section must be defined. The cross-section properties and material characteristics that are assigned determine the stiffness of the member.

Each cross-section has its own *Color* that can be used in the model to represent different profiles. Colors are controlled in the *Display* navigator with the option *Colors in Rendering According to* (see chapter 11.1.9, page 427).

You do not have to use each defined cross-section for input in the model. Thus, when modeling the structure, it is possible to make experiments without deleting cross-sections. Please note, however, that the cross-sections cannot be renumbered.

To represent a tapered beam, you have to define different start and end cross-sections for the member. RFEM determines the variable stiffnesses along the member automatically.

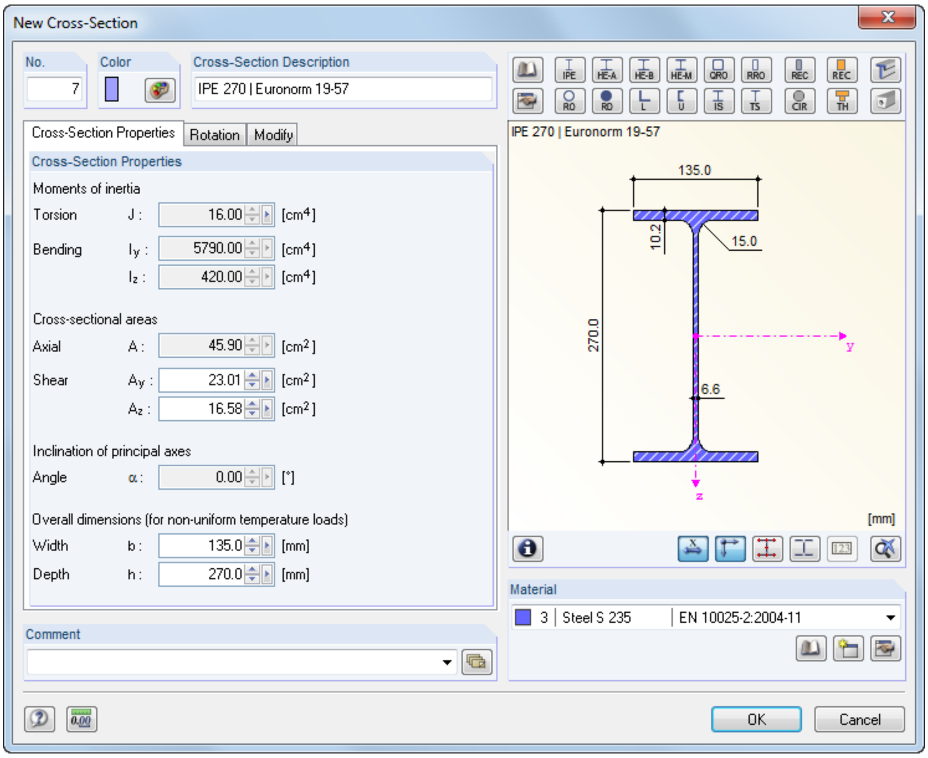
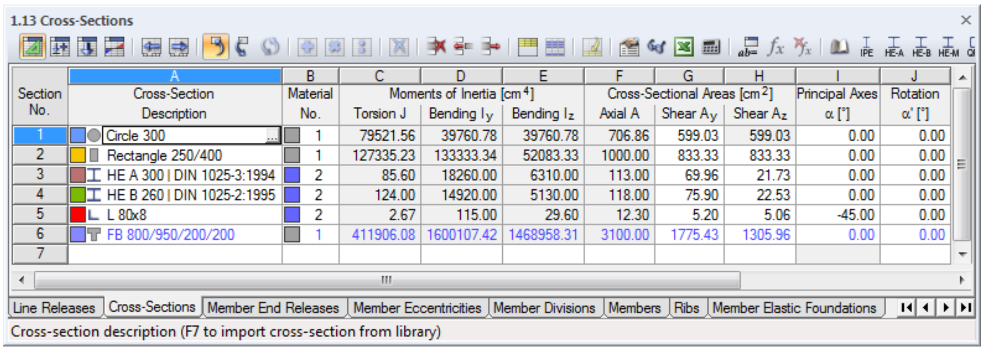


Figure 4.116: Dialog box *New Cross-Section*, tab *Cross-Section Properties*



1.13 Cross-Sections

Section No.	A Cross-Section Description	B Material No.	C Moments of Inertia [cm4] Torsion J	D Bending Iy	E Bending Iz	F Cross-Sectional Areas [cm2] Axial A	G Shear Ay	H Shear Az	I Principal Axes α [°]	J Rotation α' [°]
1	Circle 300	1	79521.56	39760.78	39760.78	706.86	599.03	599.03	0.00	0.00
2	Rectangle 250/400	1	127335.23	133333.34	52083.33	1000.00	833.33	833.33	0.00	0.00
3	HE A 300 | DIN 1025-3:1994	2	85.60	18260.00	6310.00	113.00	69.96	21.73	0.00	0.00
4	HE B 260 | DIN 1025-2:1995	2	124.00	14920.00	5130.00	118.00	75.90	22.53	0.00	0.00
5	L 80x8	2	2.67	115.00	29.60	12.30	5.20	5.06	-45.00	0.00
6	FB 800/950/200/200	1	411906.08	1600107.42	1468958.31	3100.00	1775.43	1305.96	0.00	0.00
7										

Line Releases | Cross-Sections | Member End Releases | Member Eccentricities | Member Divisions | Members | Ribs | Member Elastic Foundations

Cross-section description (F7 to import cross-section from library)

Figure 4.117: Table 1.13 *Cross-Sections*

You do not need to enter the cross-section properties manually. RFEM provides an extensive and extendable cross-section library as well as import options.

Cross-section description

The *Cross-Section Description* can be selected freely. When the entered cross-section name corresponds to an entry of the cross-section library, RFEM will import the cross-section parameters. In this case, it is not possible to change the values for the *Moments of Inertia* and area *Axial A*. For user-defined cross-section descriptions you can enter constants and cross-section areas manually.

The characteristic values of parameterized cross-sections are also imported automatically. For example, when you enter "Rectangle 80/140", the cross-section parameters of this cross-section will appear. The selection of cross-sections from the library is described later.

It is also possible to use a rigid dummy cross-section to model couplings. RFEM applies stiffnesses to this cross-section type like for a coupling member. Enter the name **Dummy Rigid** as description for the cross-section without defining the cross-section values in detail. In this way, you can use members with a high degree of stiffness, taking account of releases or other member properties. A new variant in RFEM 5 is the member type *Rigid Member* (see page 139), so the definition of a *Dummy Rigid* is no longer necessary.

Material

The cross-section's material can be selected from the list of already defined materials. The assignment is made easier by material colors that are used by default for the rendered graphical representation.

In the dialog box *New Cross-Section*, you can see three buttons below the material list. Use the buttons to access the material library, to create a new material or to edit materials.

For more detailed information about materials, see chapter 4.3, page 60.

The option *Hybrid* available in the dialog box for rectangular timber cross-sections can be accessed only for parameterized timber profiles. Use this option to assign specific material properties to cross-section elements if different material grades are provided (for example timber of low class for webs).

With a click on the [Edit] button you can open the dialog box *Edit Hybrid Material*.

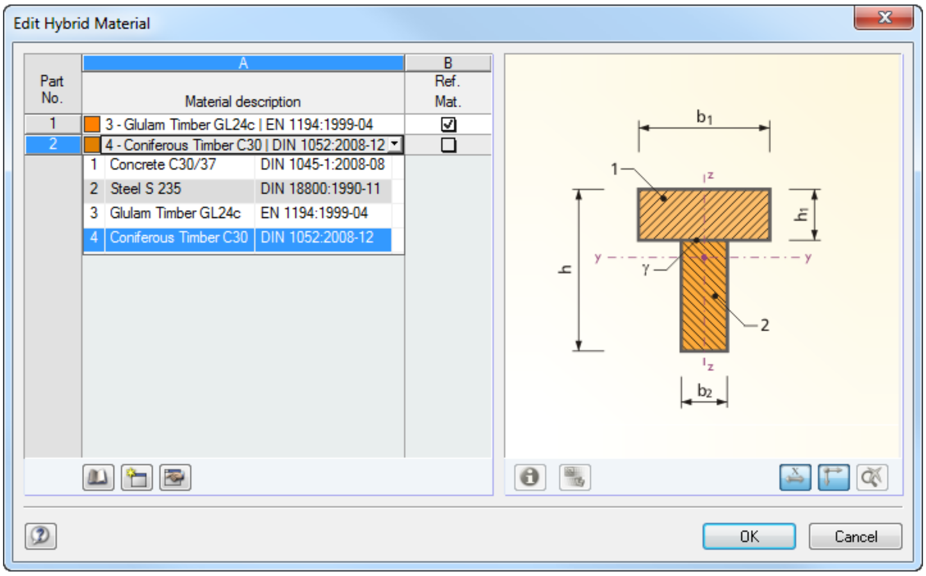
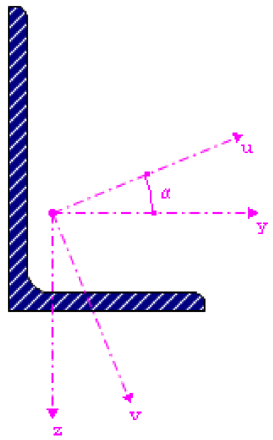


Figure 4.118: Dialog box *Edit Hybrid Material*

Assign materials to the single cross-section parts according to the graphical scheme. They can be selected from the list. One of the materials must be defined as *Reference Material* used to determine the ideal cross-section properties.



Moments of inertia

The moments of inertia are required for the cross-section stiffness: The torsional constant *J* describes the stiffness against rotation about the longitudinal axis. The second moments of area I_y and I_z describe the stiffnesses against bending about the local axes y and z. Axis y is considered to be the "strong" axis. The local cross-sectional axes are shown in the dialog graphic of the dialog box *New Cross-Section*.

The moments of inertia for unsymmetrical sections are displayed about the cross-section's principal axes *u* and *v*.

Moments of inertia as well as cross-sectional areas can be adjusted with the help of factors in the dialog tab *Modify*. In the table, you can access the tab with the [...] button that appears when you click into the table cell.

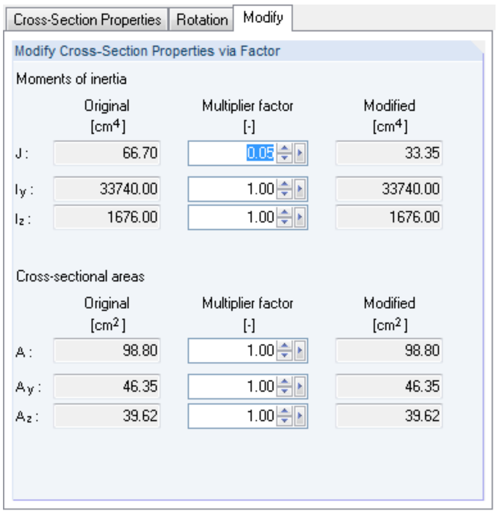


Figure 4.119: Dialog box *New Cross-Section*, tab *Modify*

With the specification set in Figure 4.119, RFEM will consider the torsional moment of inertia with 5 % only.

Cross-sectional areas

The cross-section parameters of the cross-sectional areas are subdivided into the total area *Axial A* and the shear areas *Shear* A_y and A_z.

Shear area A_y relates to the moment of inertia I_z, shear area A_z relates to I_y. Using a correction factor κ we see the following correlation existing between the shear areas A_y and A_z as well as the total area A.

$$A_y = \frac{A}{\kappa_y}; \qquad A_z = \frac{A}{\kappa_z}$$

Equation 4.20

$$\kappa_{y/z} = \frac{A}{I^2_{z/y}} \cdot \iint_A \frac{S^2_{z/y(x)}}{t^2_{(x)}} dA$$

Equation 4.21

where	A	Total area of cross-section
	$I_{z/y}$	Moments of inertia of cross-section
	$Q_{z/y(x)}$	Statical moments of cross-section at location x
	$t_{(x)}$	Width of cross-section at location x

The shear areas A_y and A_z affect the shear deformation which should be taken into account especially for short, massive members. When the shear areas are set to zero, the influence of shear will not be considered. Those parameters can also be controlled in the dialog tab *Global Calculation Parameters* of the dialog box *Calculation Parameters* (see Figure 7.22, page 271). If extremely low values are set for shear areas, numerical problems may occur because the shear areas are contained in the denominator of equations.

Select the values for cross-section areas realistically. Extreme differences in the cross-sectional areas of cross-sections involve significant differences in stiffness that may lead to numerical problems when solving the equation system.

Angle of principal axes α

The principal axes are described with *y* and *z* for symmetrical sections, and with *u* and *v* for unsymmetrical sections (see above). The rotation angle of principal axes α describes the position of the principal axes in relation to the standard system of coordinates for symmetrical sections. For unsymmetrical sections it is the angle between the **y**-axis and the **u**-axis (see graphic above shown in the left margin). This angle is defined clockwise as a positive angle. When symmetrical cross-sections are set, angle α is 0. The inclination of principal axes for sections from the library cannot be edited.

The angle of rotation for the principal axes is determined by the following equation:

$$\tan 2\alpha = \frac{2 \cdot I_{yz}}{I_z - I_y}$$

Equation 4.22

When you work with 2D models, only 0° and 180° are allowed to be set as cross-sectional angles of rotation.

Cross-section rotation α'

The angle of rotation α' describes the angle about which the sections of all members using this cross-section are rotated. Thus, the angle represents a global cross-sectional angle of rotation. In addition, each member can be rotated separately about a member rotation angle β.

Moreover, the dialog tab *Rotation* provides the option to *Mirror* nonsymmetrical cross-sections. Use this option for example to put an L-section into the correct position.

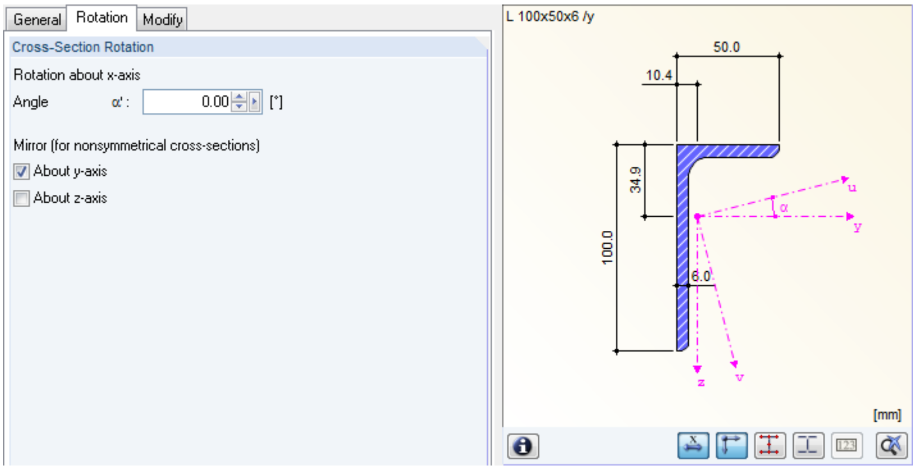


Figure 4.120: Dialog box *New Cross-Section*, tab *Rotation*

When you import a cross-section from the cross-section library or the add-on module SHAPE-THIN, you do not need to take care of the angle α'. RFEM imports this angle in the same way as other cross-sectional values. For user-defined sections, however, you have to determine the angle of principal axes yourself and to adjust it manually by means of the cross-section rotation.

Overall dimensions

The cross-section's *Width b* and *Depth h* are significant for temperature loads.

Cross-section Library

Numerous cross-sections are already available in the cross-section data base.

Open the library

In the dialog box *New Cross-Section* and in table 1.13 *Cross-Sections*, you have direct access to frequently used cross-section tables:

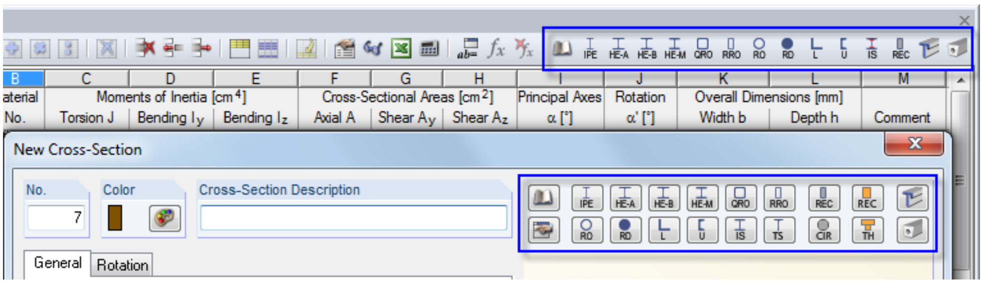


Figure 4.121: Buttons of frequently used cross-sections in table (above) and dialog box (below)

Use the button [Import Cross-Section from Library] to access the complete cross-section data-base. When you work in the table, place the cursor into table column A to enable the button [...] which you can use like the function key [F7] to open the cross-section library.

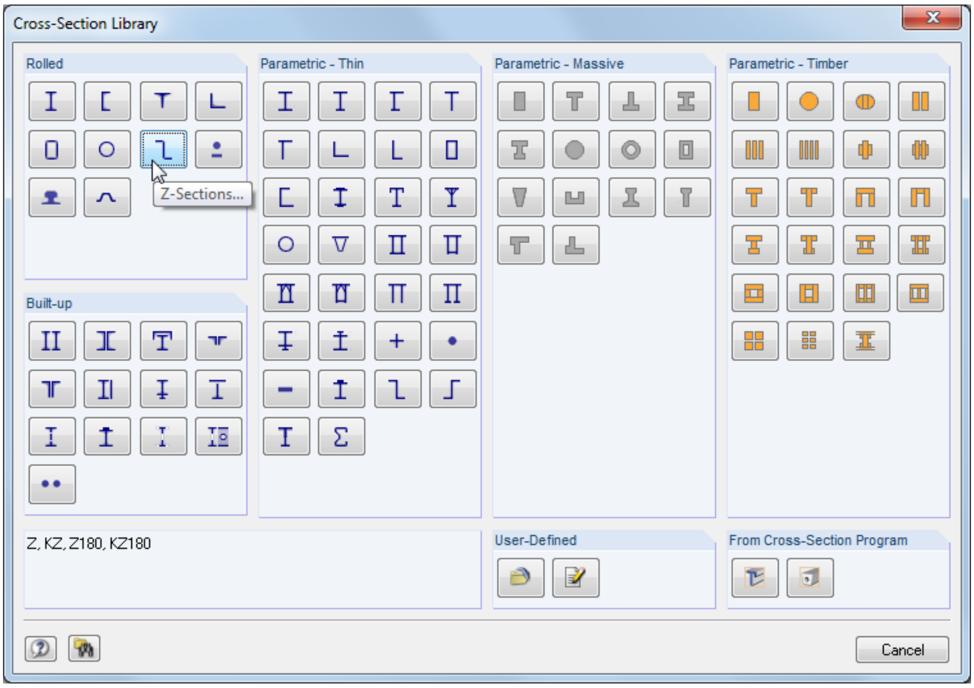


Figure 4.122: Cross-Section Library

The cross-section library is divided into several sections which are described on the following pages.

Rolled cross-sections

The table values of many rolled cross-sections are stored in a database.

First, click one of the ten buttons to define the *Cross-Section Type*. Another dialog box opens where you select the *Table*. Then, select an appropriate *Cross-Section*.

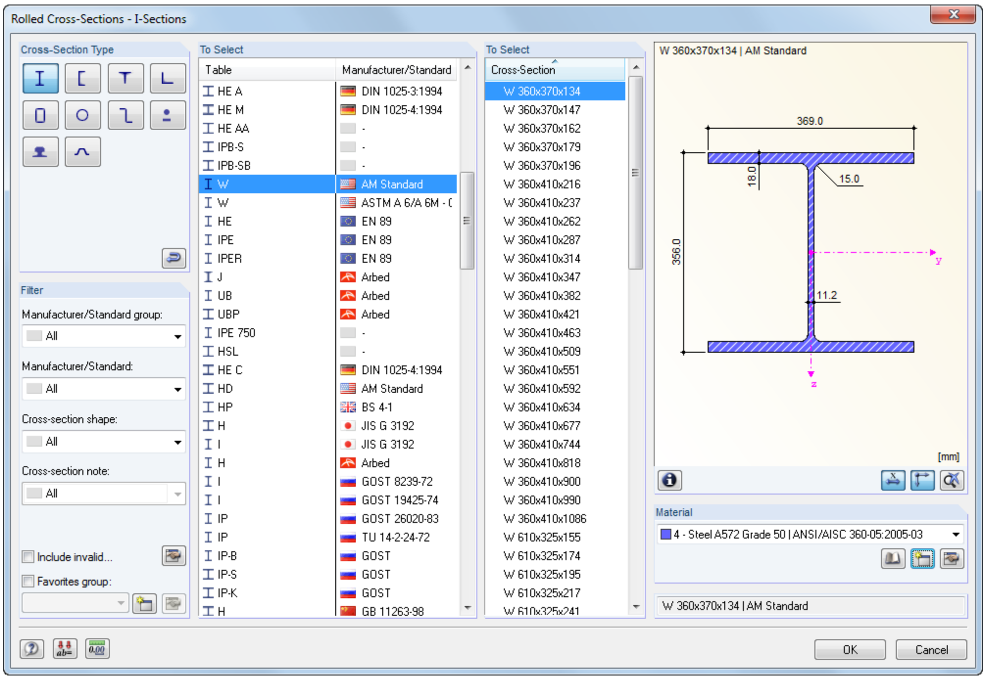
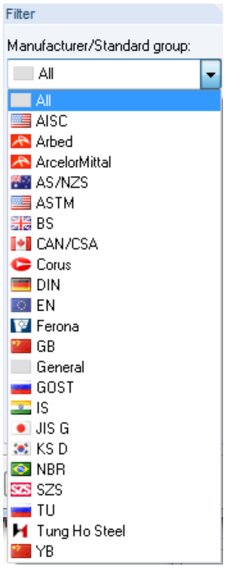


Figure 4.123: Selecting a rolled cross-section

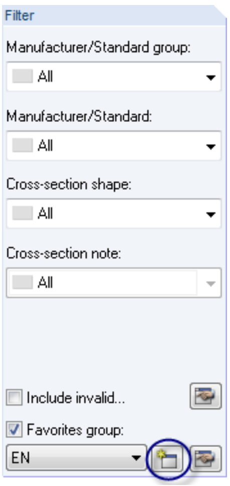


Filter for *Manufacturer/Standard group*

In the dialog section *Filter*, you can filter library entries by different criteria: *Manufacturer/Standard group*, *Manufacturer/Standard*, *Cross-section shape* and *Cross-section note*. In this way, it is easier to overview the offered tables and cross-sections. Displayed data can be sorted by clicking the headings of table columns.

If cross-sections of old standards are needed, tick the checkbox for *Include invalid* in the dialog section *Filter* to display also such sections.

Create favorites



Preferred cross-sections can be set as favorites. To access the dialog box for creating favorite cross-sections, use the button [Create New Favorites Group] at the bottom of the *Filter* dialog section. When the name for the new group has been defined, the following dialog box appears:

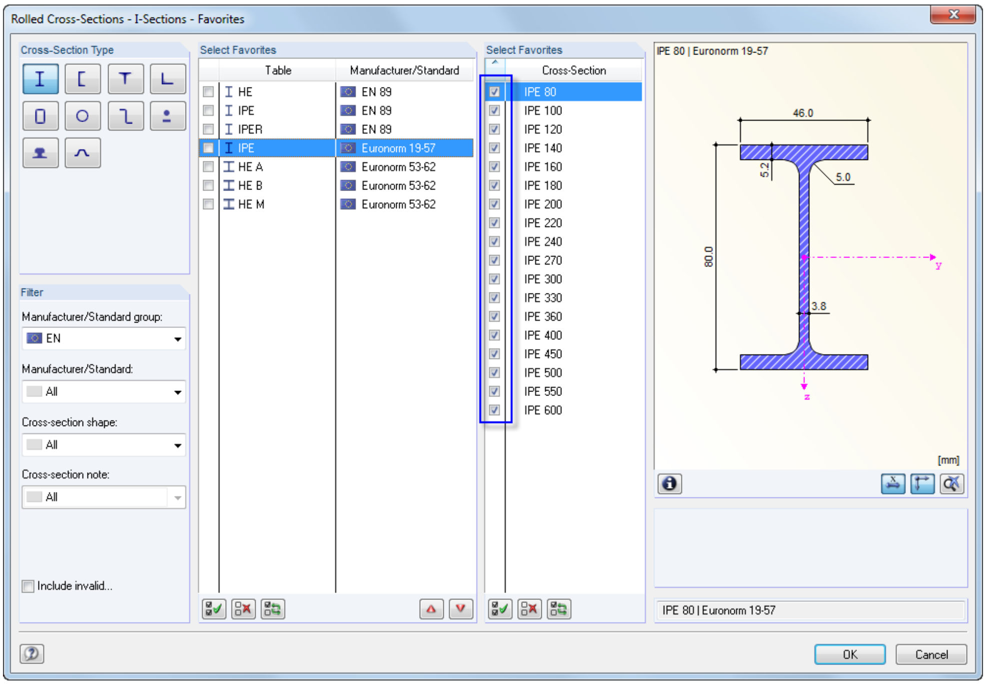
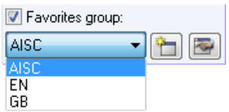


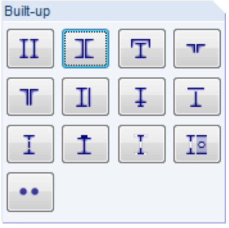
Figure 4.124: Dialog box *Rolled Cross-Sections - I-Sections - Favorites*



The dialog box looks like the cross-section library. You can use the filter options described above. In the dialog sections *Select Favorites*, you can choose preferred tables and cross-sections with a check mark.

After closing the dialog box, the cross-section library presents a clear overview of favorites as soon as you activate the option *Favorites group*.

In this way, it is possible to create different groups of favorites that are available for selection in the list at the bottom of the dialog section *Filter*.



Built-up cross-sections

Rolled cross-sections can be combined by specifying parameters.

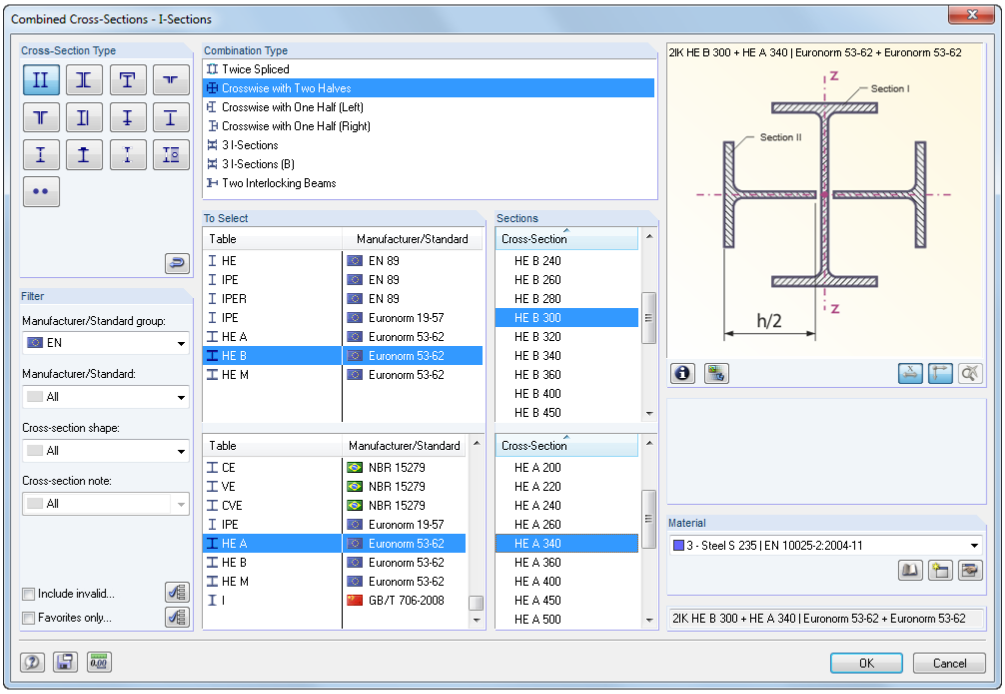


Figure 4.125: Dialog box *Combined Cross-Sections - I-Sections*

Use the [Save] button to save a combined cross-section. RFEM stores it with its accurate description (for example *2IK HE B 300 + HE A 340* in the figure above) in the category *User-Defined* from where you can reimport it later.

Parametric cross-sections - thin

With the offered input fields you can freely define parameters for a cross-section composed of sheets. The cross-section values will be calculated according to the theory for thin-walled cross-sections. The theory applies only to cross-sections whose element thickness is clearly smaller than the respective element length. If this condition is not fulfilled, define the cross-section in the *Massive* category (see Figure 4.127), if possible.

Parameter *a* represents the weld root, not the fillet radius (see figure below).

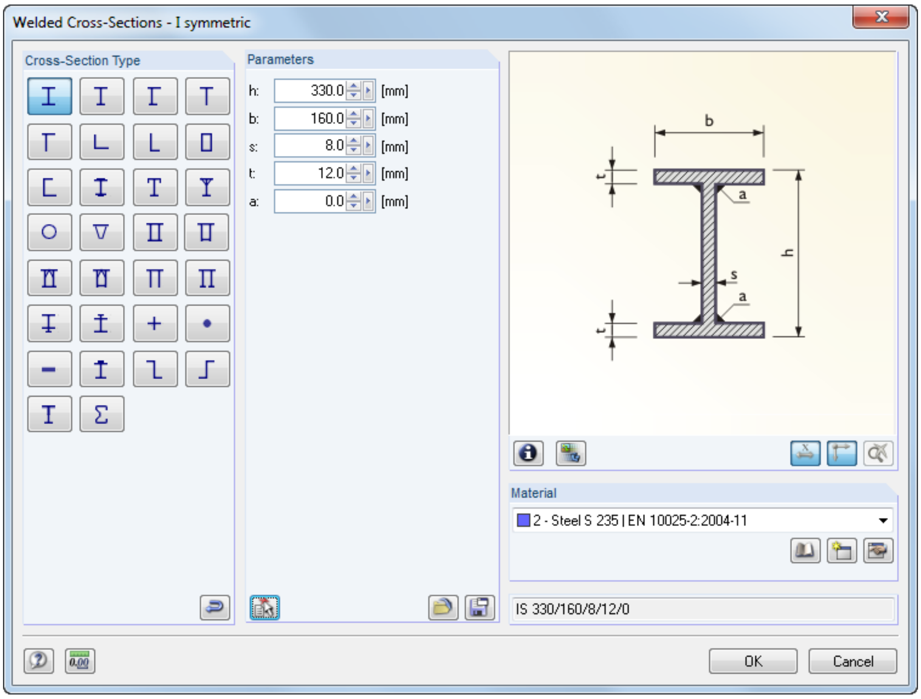


Figure 4.126: Input dialog box of a parameterized, thin-walled cross-section

Use the button shown on the left to import the parameters of a rolled cross-section. By using the selection function you can avoid entering lots of parameters.

Use the [Save] button to save a parametric cross-section with its exact name, for example *IS 330/160/8/12/0* in the figure above. Click the [Load] button shown on the left to import it.

Parametric cross-sections - massive

With the offered input fields you can freely define parameters for massive cross-sections (for example reinforced concrete sections). The cross-section values will be calculated according to the theory for massive cross-sections provided for elements with distinctive wall thicknesses.

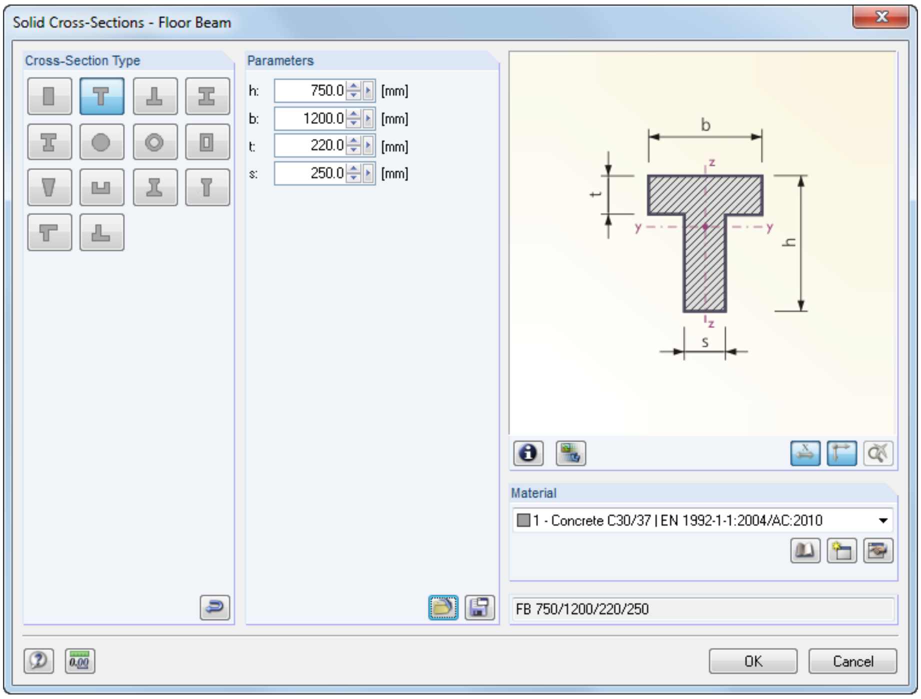
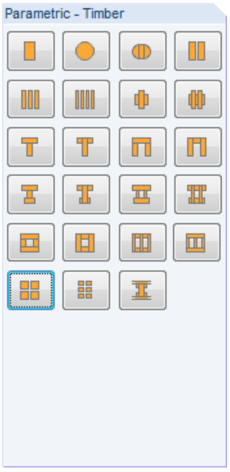


Figure 4.127: Input dialog box of a massive cross-section



Parametric cross-sections - timber

With the offered input fields you can freely define parameters for timber cross-sections. The cross-section values of both solid and combined cross-sections will be calculated according to the theory for massive cross-sections.

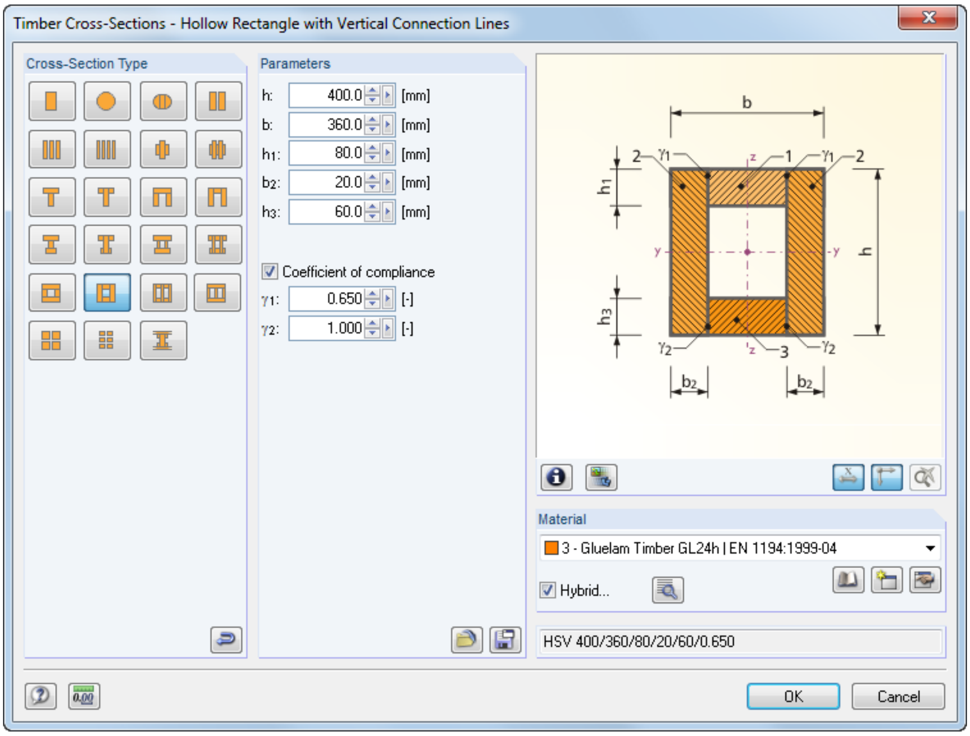


Figure 4.128: Input dialog box of a timber cross-section

Ticking the option *Coefficient of compliance* allows you to determine the effective stiffnesses for composite structural components from semi-rigidly connected cross-section elements, for example according to DIN 1052:2008-12, 8.6.2 (3). In this case, specify the reduction factors γ.

When you work with a material of the type *Hybrid*, use the [Edit] button to assign the properties of the cross-section parts (see Figure 4.118, page 118).

User-defined cross-sections

Import saved cross-section

Click the [Load] button shown on the left to open a dialog box where all cross-sections created with the help of the **Save** function are displayed.

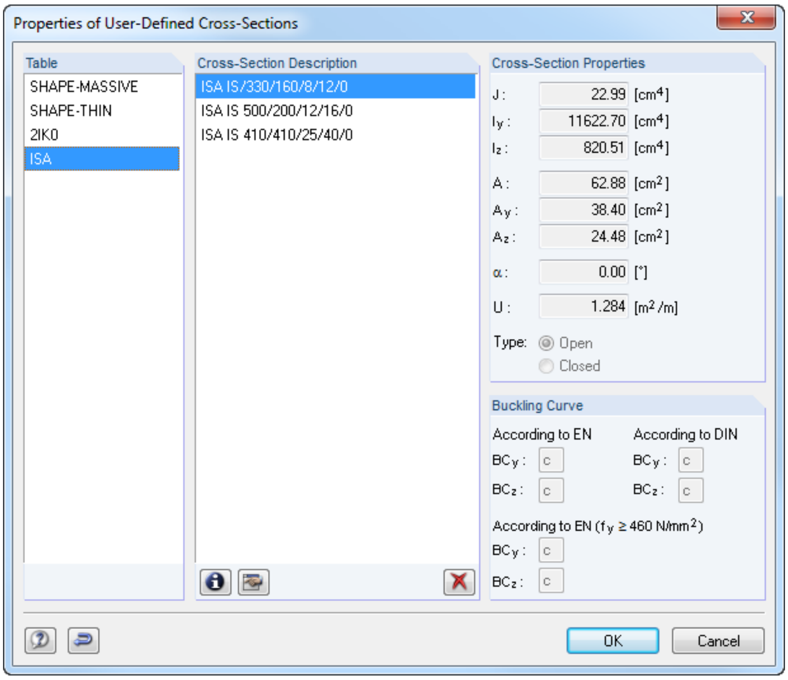


Figure 4.129: Dialog box *Properties of User-Defined Cross-Sections*

Create a user-defined cross-section

Click the [Create] button shown on the left to create user-defined cross-sections.

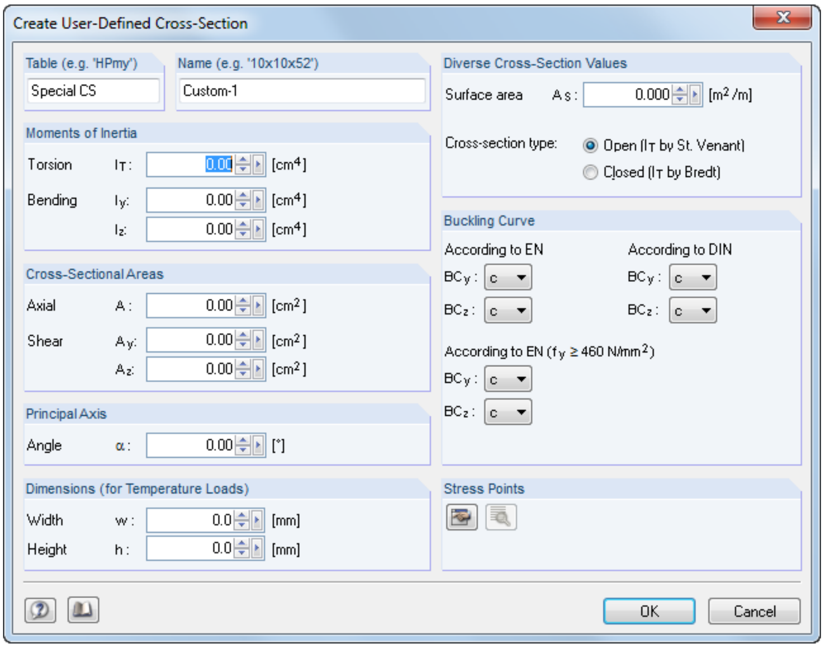
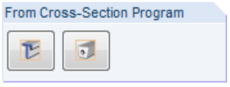


Figure 4.130: Dialog box *Create User-Defined Cross-Section*

Enter the *Table* to define the place where the cross-section is managed. Specify also the *Name* to describe the new cross-section. Then, enter the cross-section parameters and define the buckling curves.



Cross-sections from cross-section program

It is also possible to import cross-sections from the DLUBAL cross-section programs **SHAPE-THIN** and **SHAPE-MASSIVE**.

Please note that the cross-sections must be calculated and saved in the SHAPE modules before the cross-sectional values can be imported.

Import cross-section table from ASCII file

Use the button in the bottom left corner of the library to import a complete cross-section table from a file. The file must be a comma separated values file (CSV). Any Excel file can be saved in this format. Make sure that the syntax of the ASCII table corresponds to the definition parameters of the corresponding RFEM cross-section table.

Example: Import of double symmetrical I-sections

The cross-sections are managed in the **IS** table (cf. Figure 4.126). For IS cross-sections, the following parameters are required: h, b, s, t, a. The table in Excel is structured as shown below:

	A	B	C	D	E	F
1	**Description**	**h**	**b**	**s**	**t**	**a**
2	Cross-section 1	400.00	200.00	10.00	10.00	0.00
3						
4						
5						

Figure 4.131: Excel spreadsheet with cross-section parameters

In the import dialog box, specify the directory of the CSV file. Then, use the list to select the cross-section table where you want to manage the imported cross-sections.

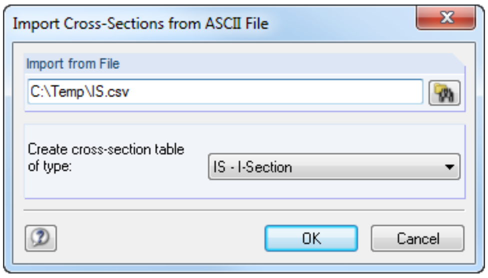


Figure 4.132: Dialog box *Import Cross-Sections from ASCII File*

Finally, you find the imported cross-sections available in the category *User-Defined* cross-sections (see Figure 4.129).

When importing cross-sections, RFEM calculates the cross-section values and stress points so that stress designs can be performed as well.

4.14 Member End Releases

General description

Member releases limit the internal forces transferred from one member to others. Releases are assigned only to member ends (nodes). They can never be assigned to other locations, for example to the middle of the member.

Some member types are already provided with releases. A truss, for example, does not transfer moments. A cable neither transfers moments nor shear forces. When entering data, keep in mind that the assignment of releases is blocked for members of such member types.

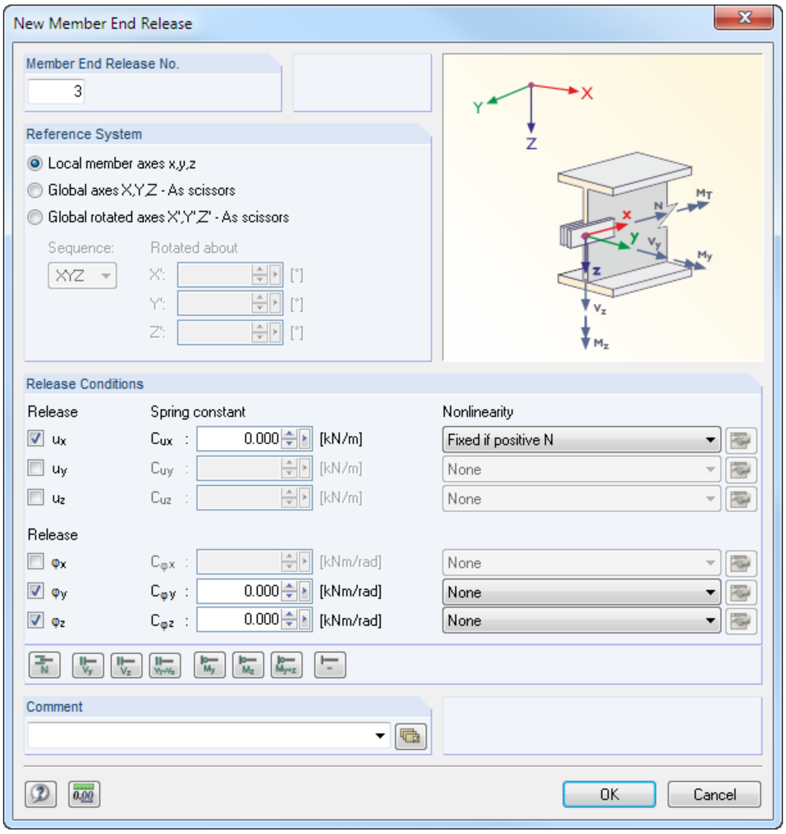


Figure 4.133: Dialog box *New Member End Release*

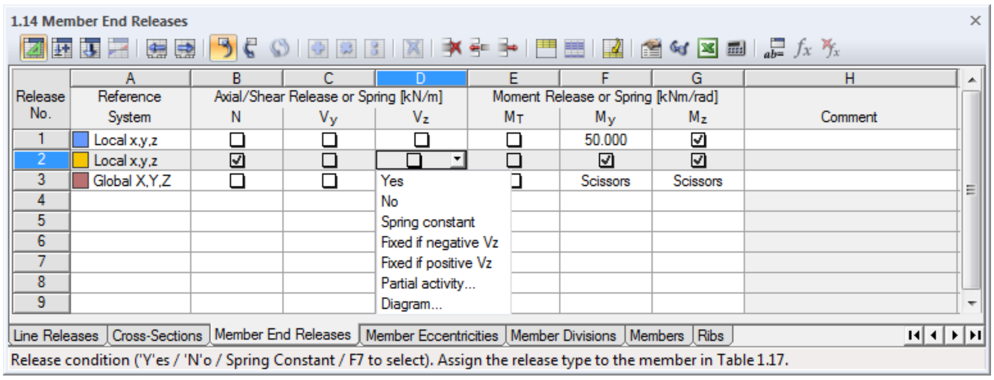
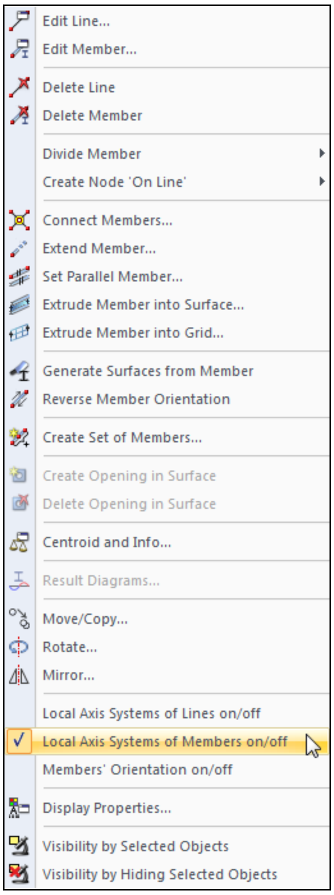


Figure 4.134: Table 1.14 *Member End Releases*



Member context menu

Reference system

A member release can be related to one of the following axis systems:

- Local member axis system x,y,z
- Global coordinate system X,Y,Z (only scissors release)
- Global rotated coordinate system X',Y',Z' (only scissors release)

Use the *Display* navigator or the member context menu shown on the left to display the local member axes (see Figure 4.158, page 145).

For detailed information about the orientation of local member axes in the global coordinate system X,Y,Z, see chapter 4.17 on page 145.

Normally, releases are related to the local axis system *x,y,z*. Scissors releases (see Figure 4.136) can only be related to the global coordinate system. Spring constants and nonlinearities must be defined in relation to the local member axis system.

Axial/shear release or spring

To define an axial or shear force release, set the respective displacement free by ticking the relevant check box in the dialog box or table. The check mark means that the axial respectively shear force cannot be transferred at the member end because a release has been set. Look at the *Member End Release* dialog box: A zero value is shown for the constant of the translational spring in the input field to the right of the check mark.

You can always change the spring constant to represent for example a semi-rigid connection. In the table, enter the constant directly into the table column. The stiffnesses of the springs are considered as design values.

Moment release or spring

Define releases for torsion and bending moments like releases for forces. Again, the check mark means that torsion is free and the internal force won't be transferred.

Elastic connections can be modeled by means of spring constants that you can enter directly. Pay attention not to use extreme stiffness values because otherwise numerical problems may occur during the calculation. Instead of very big or small constants, apply rigid connections (no check mark) or releases (check mark).

The option for defining nonlinear release properties is described at the end of this chapter.

Assign releases graphically

To assign releases in the work window graphically,

select **Model Data** on the **Insert** menu, point to **Member End Releases** and select **Assign to Members Graphically** or

open the **Edit** menu, point to **Model Data** and **Member End Releases**, and then select **Assign Graphically to Members**.

First, select a release type from the list or create a new one. After clicking [OK], members are divided graphically at one-third division points.

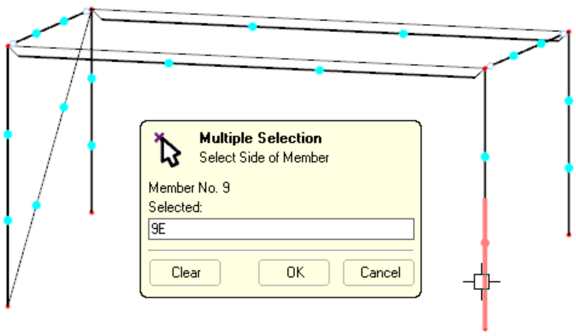


Figure 4.135: Assigning member end releases graphically

Now, you can click the member sides to which you want to apply the selected release. To assign the release to both member ends, click the member in its center area.

Scissors release

With scissors releases you can model crossing of beams. For example: You have four members connected in one node. Each of the two member pairs transfers moments in its 'continuous direction', but they do not transfer any moments to the other pair. Only axial and shear forces are transferred in the node.

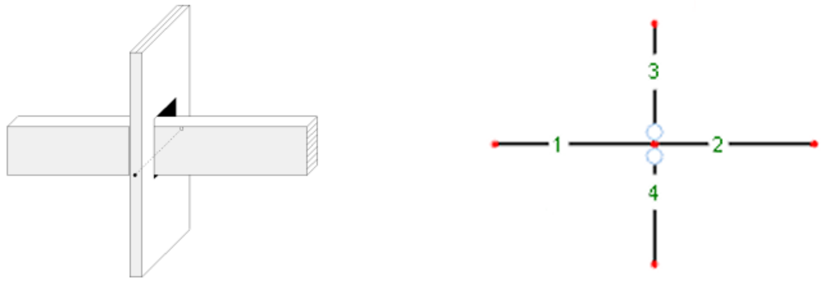


Figure 4.136: Beam crossing

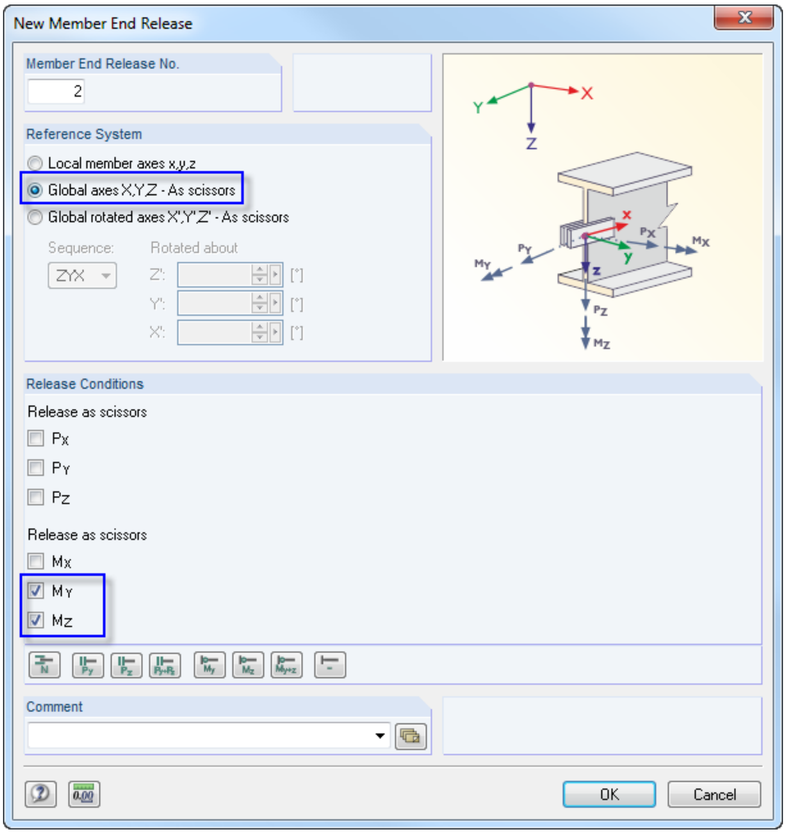


Figure 4.137: Dialog box *New Member End Release*

In this case, the release must be assigned either to members 1 and 2 <u>or</u> to members 3 and 4. The other crossing member pair will be modeled as bending-resistant without release.

Nonlinearities

Nonlinear properties can be assigned to member end releases. In this way, you can control the transfer of internal forces in detail. The list of nonlinearities offers the following options:

- Fixed if internal force negative
- Fixed if internal force positive
- Partial activity
- Diagram

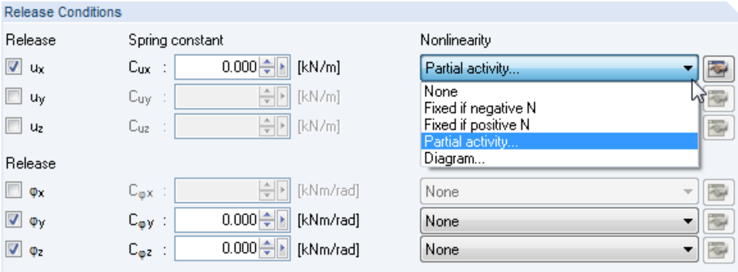


Figure 4.138: List of nonlinear properties

In the table, release types with nonlinear properties are marked in blue.

Fixed if internal force negative or positive

With these two options you can control the release activity depending on the direction for each internal force. For example: An axial force release defined with the nonlinearity *Fixed if positive N* has the effect that tensile forces (positive) can be transferred at the end of the member, but no compressive forces (negative). In case of negative axial forces the release wil be effective.

The internal forces are related to the local member axis system xyz.

The remaining entries of the *Nonlinearity* list offer detailed modeling options for release properties. To access the options, use the [Edit] dialog buttons to the right of the list or the button [▼] in the table (see Figure 4.134, page 128).

Partial activity

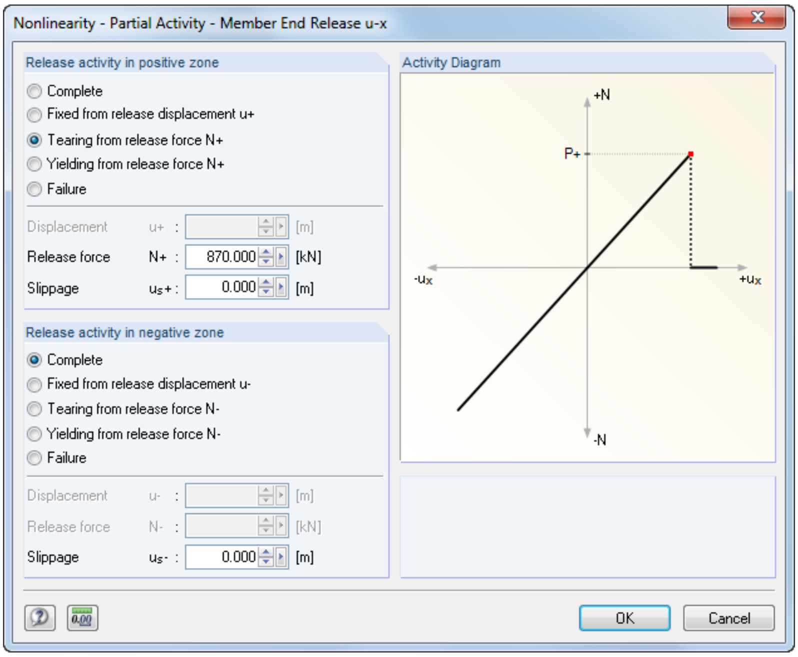


Figure 4.139: Dialog box *Nonlinearity - Partial Activity*

The activity of the release can be defined separately for the *positive* and *negative zone*. In addition to full effectiveness or failure, the release can loose its effect when a certain displacement or rotation is reached. Then, it begins to act as a fixed or rigid connection. Also *Tearing* (no internal force will be transferred anymore after exceeding a certain value) and *Yielding* (internal forces will be transferred only up to a certain value also in case of larger deformations) are possible in combination with a *Slippage*.

The limit values can be defined in the input fields below. In the dialog section *Activity Diagram*, the release properties are shown in a dynamic graphic.

Diagram

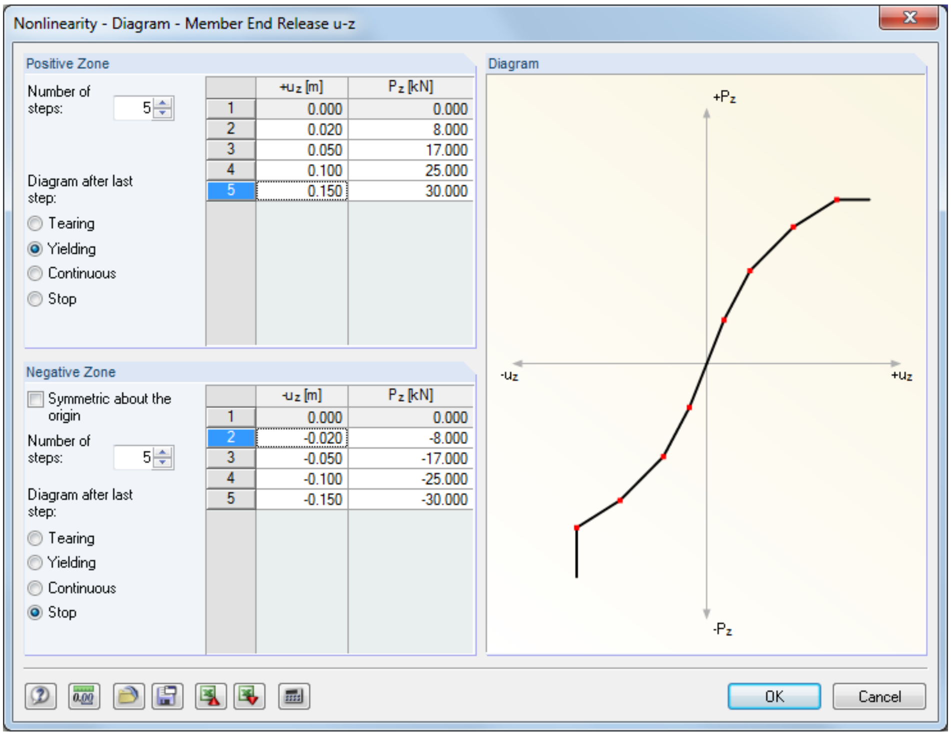


Figure 4.140: Dialog box *Nonlinearity - Diagram*

The activity of the release can be defined separately for the *Positive* and *Negative Zone*. First, enter the *Number of steps* (that means definition points) represented in the diagram. Then, you can enter the abscissa values of the internal forces with the assigned displacements or rotations into the list to the right.

You find different input possibilities for the *Diagram after last step*: *Tearing* for the failure of the release (no internal force will be transferred any longer), *Yielding* for restricting the transfer to a maximum allowable internal force, *Continuous* as in the last step or *Stop* for restricting to a maximum allowable displacement or rotation followed by a fixed or rigid release activity.

In the dialog section *Diagram*, the release properties are shown in a dynamic graphic.

Example: rafter roof

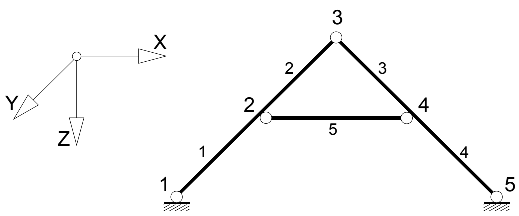
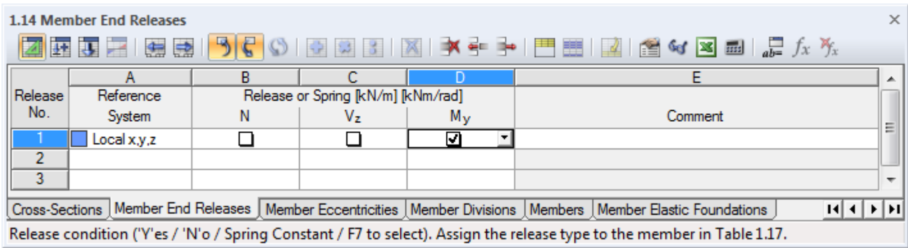


Figure 4.141: Rafter roof

A planar system is used. The release must be defined as follows:


1.14 Member End Releases

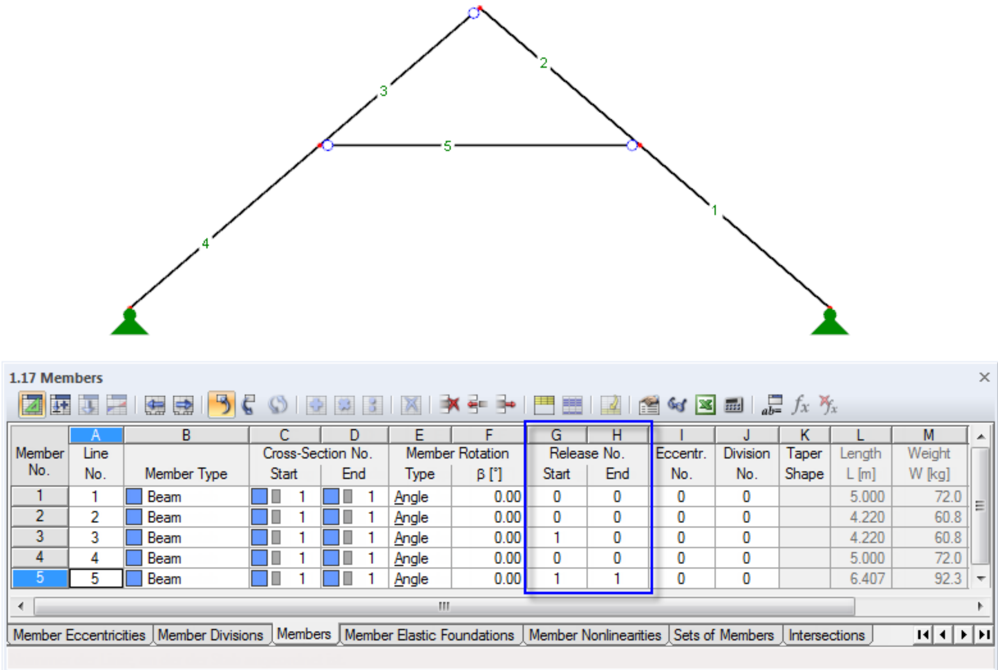
Release No.	A Reference System	B Release or Spring [kN/m] [kNm/rad] N	C V_z	D M_y	E Comment
1	Local x,y,z	☐	☐	☑	
2					
3					

Cross-Sections | Member End Releases | Member Eccentricities | Member Divisions | Members | Member Elastic Foundations

Release condition ('Y'es / 'N'o / Spring Constant / F7 to select). Assign the release type to the member in Table 1.17.

Figure 4.142: Table 1.14 *Member End Releases*

Now, the release type can be assigned to the members.


1.17 Members

Member No.	A Line No.	B Member Type	C Cross-Section No. Start	D Cross-Section No. End	E Member Rotation Type	F β [°]	G Release No. Start	H Release No. End	I Eccentr. No.	J Division No.	K Taper Shape	L Length L [m]	M Weight W [kg]
1	1	Beam	1	1	Angle	0.00	0	0	0	0		5.000	72.0
2	2	Beam	1	1	Angle	0.00	0	0	0	0		4.220	60.8
3	3	Beam	1	1	Angle	0.00	1	0	0	0		4.220	60.8
4	4	Beam	1	1	Angle	0.00	0	0	0	0		5.000	72.0
5	5	Beam	1	1	Angle	0.00	1	1	0	0		6.407	92.3

Member Eccentricities | Member Divisions | Members | Member Elastic Foundations | Member Nonlinearities | Sets of Members | Intersections

Figure 4.143: Graphic and table 1.17 *Members*

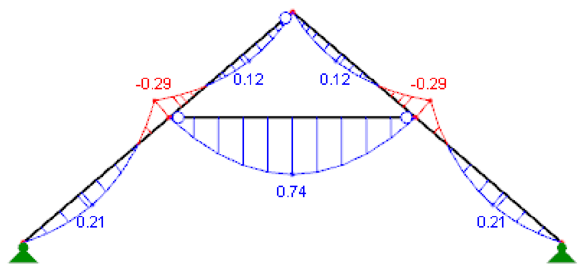
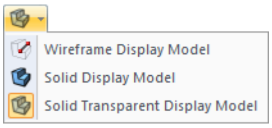


Figure 4.144: Moment diagram in load case 'Self-weight'

4.15 Member Eccentricities

General description

The length of a member corresponds to the distance between two nodes defined by the member line. However, in some modeling situations (connections of cross-sections or downstand beams), reality is represented only to a certain degree. With member eccentricities you can connect members eccentrically due to special member end sections. In this way, you can reduce for example design moments on horizontal beams for frames with big column cross-sections. Member eccentricities are taken into account by a transformation of the degrees of freedom in the local element stiffness matrix.



To check the entered eccentricities, use the photo-realistic imaging of the 3D rendering.

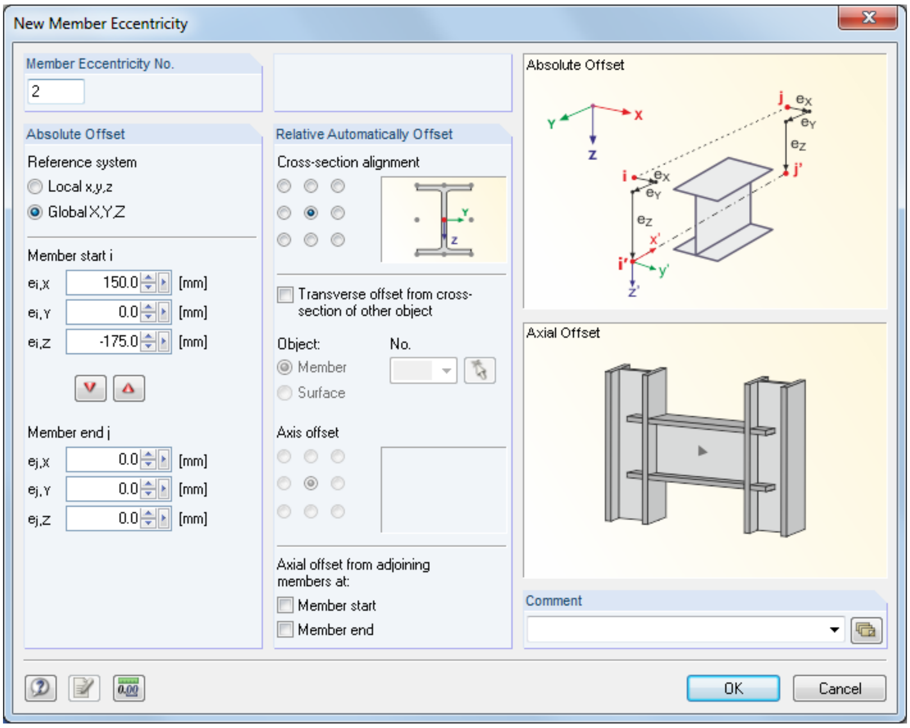
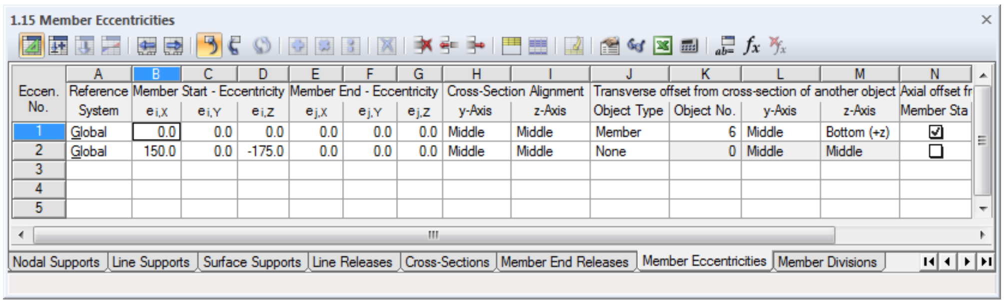


Figure 4.145: Dialog box *New Member Eccentricity*



1.15 Member Eccentricities

Eccen. No.	A Reference System	B Member Start - Eccentricity $e_{i,X}$	C $e_{i,Y}$	D $e_{i,Z}$	E Member End - Eccentricity $e_{j,X}$	F $e_{j,Y}$	G $e_{j,Z}$	H Cross-Section Alignment y-Axis	I z-Axis	J Transverse offset from cross-section of another object Object Type	K Object No.	L y-Axis	M z-Axis	N Axial offset fr Member Sta
1	Global	0.0	0.0	0.0	0.0	0.0	0.0	Middle	Middle	Member	6	Middle	Bottom (+z)	☑
2	Global	150.0	0.0	-175.0	0.0	0.0	0.0	Middle	Middle	None	0	Middle	Middle	☐
3														
4														
5														

Figure 4.146: Table 1.15 *Member Eccentricities*

Reference system

A member eccentricity can be related to one of the following axis systems:

- Local member axis system x,y,z
- Global coordinate system X,Y,Z

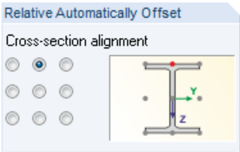
Use the *Display* navigator or the context menu of the member to display the local member axes x,y,z (see Figure 4.158, page 145).

Eccentricity for member start/member end

In the dialog section *Absolute Offset*, respectively in table columns B to G, you define the eccentricities for the *Member start i* and the *Member end j*. The distances refer to the selected axis system indicated by the upper- and lower-case indexes which are also shown in the dialog graphic.

In the dialog box, you can use the buttons [▼] and [▼] to transfer the values from one side to the other.

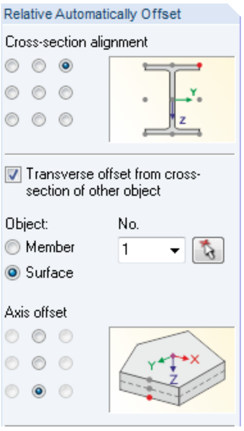
Cross-section alignment



In the dialog section *Relative Automatically Offset*, use the nine selection options to define the cross-section point relevant for the determination of the eccentricity. In the table, specify the position of the point in columns H and I. The point defines the distance by which the cross-section is shifted on the start or end node.

By defining the point in the middle of the top flange, as shown in the picture on the left, you can attach for example a horizontal beam with its top edge to a column by flush connection (without extension).

Transverse offset from cross-section of other object



With a *Transverse offset* you can arrange a member in a particular distance parallel to an object (member in surface, member in same direction). Select the number of the relevant object, a *Member* or a *Surface*, from the list. You can also use the [↖] function to select it in the work window. The eccentricity is determined from the *Cross-section alignment* defined above and the *Axis offset* (cross-sectional geometry or surface thickness) that you define by selecting one of the available nine or three check boxes. In the table, define the axis offset in columns L and M.

By defining the points on the edge of the top flange and on the bottom side of the surface, as shown in the picture on the left, you can arrange for example a steel cross-section on the edge under a plate by flush connection.

Axial offset from adjoining members at

The last option in the dialog section *Relative Automatically Offset* allows you to connect easily for example a member eccentrically to a flange of a column. The offset can be arranged separately for *Member start* and *Member end*. The eccentricity is determined automatically from the cross-section geometry of the adjacent members. In the table, assign the axial offset in columns N and O.

The dialog graphic *Axial Offset* is interactive with the input, illustrating the effectiveness of the selected check boxes.

You may prefer the input in the dialog section *Relative Automatically Offset* because you can directly adjust the eccentricities when cross-sections are changed. RFEM takes into account modified surface or cross-section dimensions automatically.

Assign eccentricities graphically

Furthermore, eccentricities can be assigned to members graphically in the work window. To open the corresponding dialog box,

> select **Model Data** on the **Insert** menu, point to **Member Eccentricities** and select **Assign to Members Graphically** or
>
> open the **Edit** menu, point to **Model Data** and **Member Eccentricities**, and then select **Assign Graphically to Members**.

First, define the reference system and the eccentricities.

After clicking [OK], members are divided graphically at one-third division points. Now, you can click the member sides to which you want to apply the eccentricity (see Figure 4.135, page 129). To assign an eccentric connection to both member ends, click the member in its center area.

4.16 Member Divisions

General description

Member divisions are used to define points on members for which internal forces and deformations are displayed later in the results tables and the numerical printout. A member division has neither influence on the determination of extreme values nor on the graphical results diagram (RFEM internally uses a more refined partition). Therefore, in most cases, member divisions are not required.

Do not confuse member divisions with FE divisions for members. FE nodes on "free" (not belonging to a surface) lines with member properties will be generated only if the lines have an FE mesh refinement (see chapter 4.23, page 164).

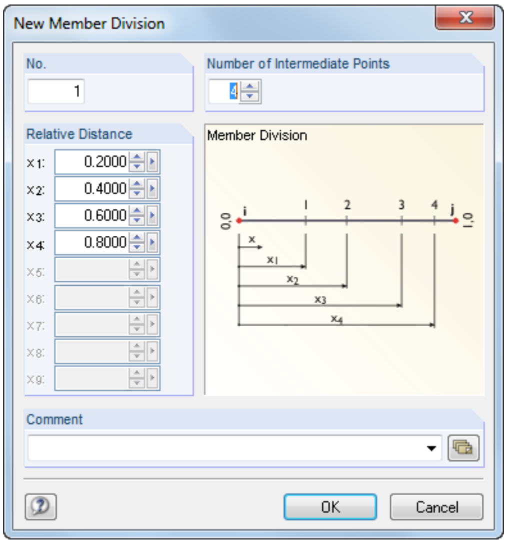
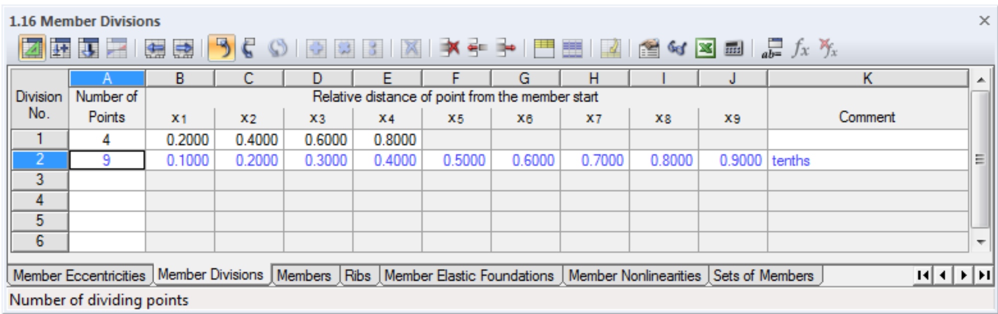


Figure 4.147: Dialog box *New Member Division*



1.16 Member Divisions

Division No.	A Number of Points	B x1	C x2	D x3	E x4	F x5	G x6	H x7	I x8	J x9	K Comment
1	4	0.2000	0.4000	0.6000	0.8000						
2	9	0.1000	0.2000	0.3000	0.4000	0.5000	0.6000	0.7000	0.8000	0.9000	tenths
3											
4											
5											
6											

Member Eccentricities | Member Divisions | Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members

Number of dividing points

Figure 4.148: Table 1.16 *Member Divisions*

Number of points

You can enter a maximum number of 99 division points in the dialog box. An entry divides the member into the desired number of equidistant points.

Relative distance of point from member start

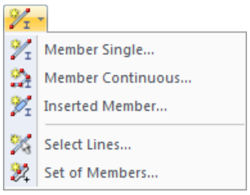
When you create a new division in the dialog box, the distances of three intermediate points are preset. They represent the relative distances in the interval of 0 (member start) to 1 (member end).

It is also possible to define irregular divisions for the specified points as you can enter the relative distances freely. Only make sure that you follow the correct order of intervals: $x_1 < x_2 < x_3$ ...

Moreover, any x-location on the member can specifically evaluated graphically (see chapter 9.5, page 356). Thus, in most cases entering member divisions manually with troublesome determination of relative distances is unnecessary.

4.17 Members

General description



List button *Member*

Members are attributes of lines. By assigning a cross-section (by which also a material is defined), the member receives a stiffness. When generating the FE mesh, 1D elements are created on members.

Members can be connected with each other only on nodes. When members cross each other without sharing a common node, no connection exists. No internal forces are transferred on such crossings.

Graphically, you can apply members as *Single*, *Continuous* or to already existing *Lines*. The option *Inserted Member* is described in chapter 11.4.13 on page 474.

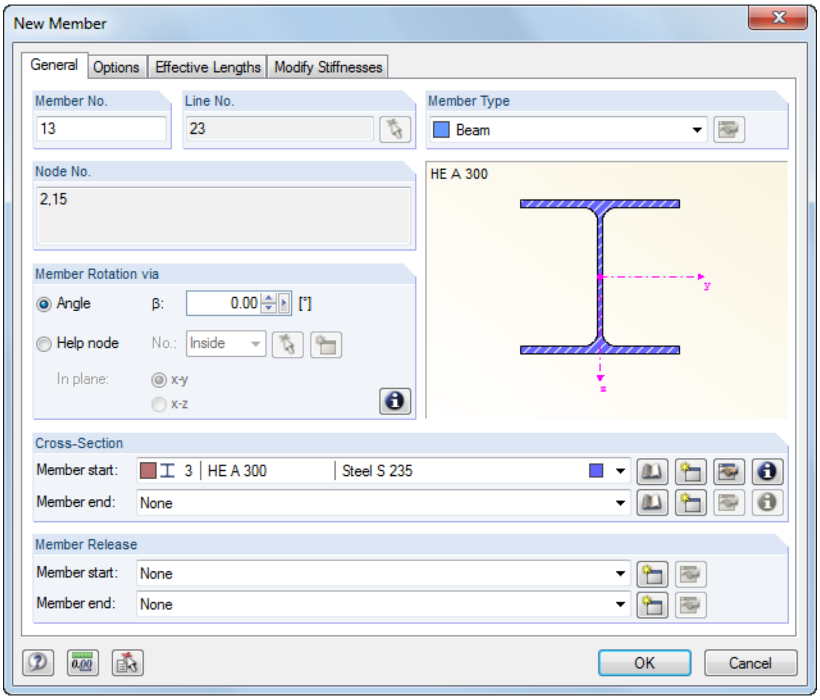
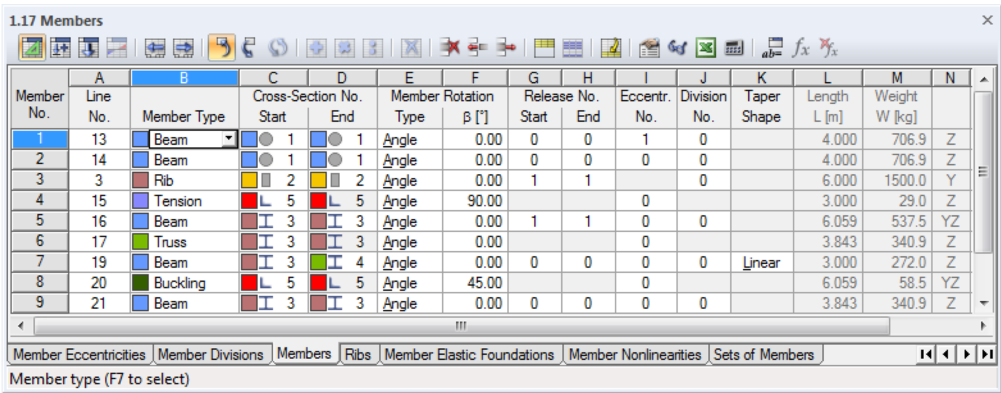


Figure 4.149: Dialog box *New Member*, tab *General*



1.17 Members

Member No.	A Line No.	B Member Type	C Cross-Section No. Start	D Cross-Section No. End	E Member Rotation Type	F Member Rotation β [°]	G Release No. Start	H Release No. End	I Eccentr. No.	J Division No.	K Taper Shape	L Length L [m]	M Weight W [kg]	N
1	13	Beam	1	1	Angle	0.00	0	0	1	0		4.000	706.9	Z
2	14	Beam	1	1	Angle	0.00	0	0	0	0		4.000	706.9	Z
3	3	Rib	2	2	Angle	0.00	1	1		0		6.000	1500.0	Y
4	15	Tension	5	5	Angle	90.00			0			3.000	29.0	Z
5	16	Beam	3	3	Angle	0.00	1	1	0	0		6.059	537.5	YZ
6	17	Truss	3	3	Angle	0.00			0			3.843	340.9	Z
7	19	Beam	3	4	Angle	0.00	0	0	0	0	Linear	3.000	272.0	Z
8	20	Buckling	5	5	Angle	45.00			0			6.059	58.5	YZ
9	21	Beam	3	3	Angle	0.00	0	0	0	0		3.843	340.9	Z

Member Eccentricities | Member Divisions | Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members

Member type (F7 to select)

Figure 4.150: Table 1.17 *Members*

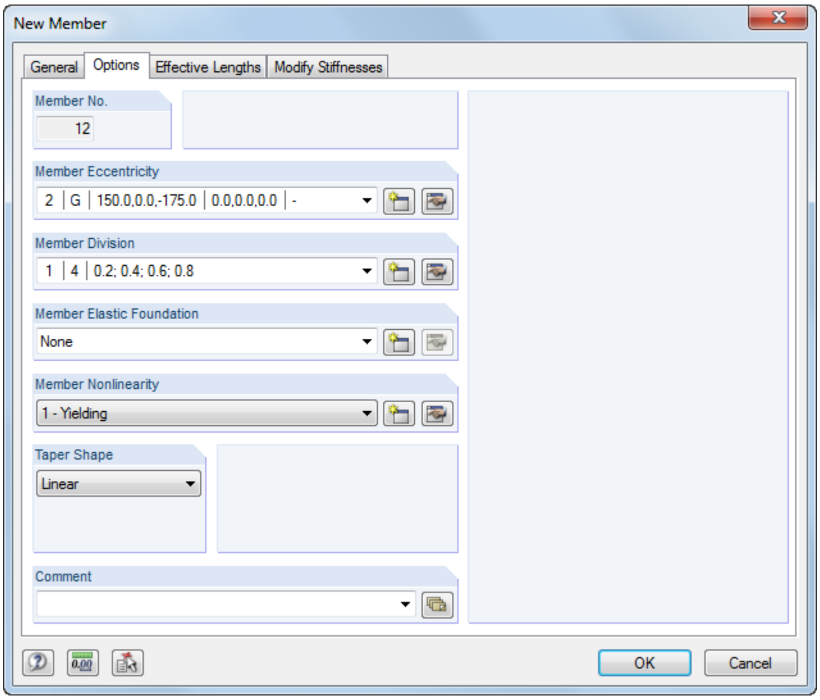


Figure 4.151: Dialog box *New Member*, tab *Options*

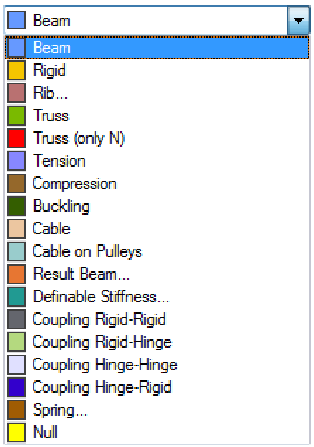
Line

Enter the number of the line with member properties into the input field of the dialog box, respectively the column in the table. In the dialog box *New Member*, you can select the line also graphically.

The start and end nodes of the line define the member orientation which also affects the position of the local coordinate system of the member (see "member rotation" in this chapter). The member orientation can be changed quickly in the graphic: Right-click the member and select *Reverse Member Orientation* in the context menu.

Member type

With the member type you define the way how internal forces are absorbed or which properties are assumed for the member.



Different options are available for selection in the *Member Type* list. Each member type has its own *Color* that can be used in the model to represent different kinds of members. Colors are controlled in the *Display* navigator with the option *Colors in Rendering According to* (see chapter 11.1.9, page 427).

Member type	Short description
Beam	Bending-resistant member that can transmit all internal forces.
Rigid member	Coupling member with rigid stiffness
Rib	Downstand beam considering the effective slab width
Truss	Beam with moment releases at both ends
Truss (only N)	Member with stiffness E · A only
Tension	Truss (only N) with failure in case of compression force
Compression	Truss (only N) with failure in case of tension force
Buckling	Truss (only N) with failure in case of compression force > N_{cr}

Cable	Member transferring tension forces only. Calculation is performed according to large deformation analysis.
Cable on pulleys	Member on polyline, can only be shifted in longitudinal direction, absorbing tensile forces only (pulley).
Result beam	Member for integration of surface, solid or member results
Stiffnesses	Member with user-defined stiffnesses
Coupling rigid-rigid	Rigid coupling with bending-resistant connections at both ends
Coupling rigid-hinge	Rigid coupling with bending-resistant connection at member start and hinged connection at member end
Coupling hinge-hinge	Rigid coupling with hinged connections at both ends (only axial and shear forces are transmitted, but no moments).
Coupling hinge-rigid	Rigid coupling with hinged connection at member start and bending-resistant connection at member end
Spring	Member with spring stiffness, definable activity zones and damping coefficients
Null (dummy member)	Member that will be ignored in the calculation.

Table 4.7: Member types

Beam

A beam does not have any releases defined on its member ends. When two beams are connected with each other and no release has been defined for the common node, the connection is bending-resistant. Beams can be stressed by all types of loads.

Rigid member

It couples the displacements of two nodes by a rigid connection. Thus, it corresponds to a coupling member in principle (see page 143). Use a rigid member to define members with high stiffness taking into account releases which may also have spring constants and nonlinearities. Hardly any numeric problems will occur as stiffnesses are adjusted to the system. RFEM shows internal forces also for rigid members.

The following stiffnesses are assumed (applies also to couplings and *Dummy Rigids*):

- Longitudinal and torsional stiffness $E \cdot A = G \cdot I_T = 10^{13} \cdot l$ (l = member length)
- Flexural resistance $E \cdot I = 10^{13} \cdot l^3$
- Shear stiffness (if activated) $GA_y = GA_z = 10^{16} \cdot l^3$

Due to this type of member it is no longer necessary to define a *Dummy Rigid* (see page 118) which is assigned as cross-section.

Rib

Ribs are described in chapter 4.18, page 150.

Truss (only N)

This type of a truss member absorbs axial forces in the form of tension and compression. A truss member has internal moment releases on its member ends. Therefore, an additional release definition is not allowed. RFEM shows you only node internal forces (which are transferred to the connecting members). The member itself shows a linear distribution of internal forces. An exception is the concentrated load on the member, which means that no moment diagram will be visible as a result of self-weight or a line load. The boundary moments are zero because of the release. A linear distribution is assumed along the member. The nodal forces, however, are calculated from the member loads, which guarantees a correct transmission.

The reason for special treatment is that a truss girder, as it is commonly understood, transfers only axial forces. Moments are not of interest. Therefore, they are deliberately neither shown in the output nor calculated as a part of the design. To get and see moments from the member loads, use the member type *Truss*.

Tension / Compression

A tension member can absorb only tension forces and a compression member only compression forces. The calculation of a framework structure with these types of members is carried out iteratively. In the first iteration, RFEM determines the internal forces of all members. If tension members have negative axial forces (compression), or if compression members have positive axial forces (tension), an additional iteration step is started in which the rigidity of these members won't be considered anymore - they have failed. This iteration process continues as long as tension or compression members are failing. Depending on modeling and loading, the system may become unstable due to failure of tension or compression members.

A failed tension or compression member can be considered again in the stiffness matrix if it is reactivated in a later iteration step due to redistributions in the system. On the **Calculate** menu, select **Calculation Parameters** to open the dialog box *Calculation Parameters* where you can specify the *Global Calculation Parameters*. In the dialog section *Reactivation of Failing Members* you can set the *Exceptional handling* of failing members. Details can be found in chapter page 7.3 on page 271.

Buckling

A buckling member absorbs unlimited quantities of tensile forces. Compressive forces, however, can be absorbed only until the critical Euler load is reached.

$$N_{cr} = \frac{\pi^2 \cdot E \cdot I}{l_{cr}^2} \qquad \text{where} \quad l_{cr} = l$$

Equation 4.23

With this type of member you can often avoid instabilities occurring in calculations according to second order or large deformation analysis due to buckling of truss members. If you replace trusses – close to reality – by buckling members, the critical load is increased in many cases.

Cable

Cable members absorb only tension forces. They are used to analyze cable chains with longitudinal and transversal forces by iterative calculations taking into account the cable theory (large deformation analysis - see chapter 7.3.1, page 264). It is required to define the complete cable as cable chain consisting of several cable members.

To create quickly a catenary, point to **Generate Model - Members** on the **Tools** menu and select **Arc** (chapter 11.7.2, page 516). The more accurately the starting shape of the catenary corresponds to the real cable chain, the more stable and faster you can perform the calculation.

It is recommended to prestress cable members in order to prevent compression forces resulting in failure. Furthermore, cables should be used only if deformations have a considerable part in changes of the internal forces, that means when large deformations occur. For simple straight riggings like transverse bracings (projecting roof), tension members are completely sufficient.

When evaluating deformations of cable members, set the scaling factor in the control panel (see Figure 3.20, page 31) to "1" so that tightening effects are represented realistically.

Cable on pulleys

The cable on pulleys absorbs only tensile forces and is calculated according to the cable theory (large deformation analysis). In contrast to a cable, it can only be applied to a polyline with at least three nodes. This member type is appropriate for pulley systems where axial forces are passed on by means of sheaves.

In comparison to a normal cable member, only a displacement within the internal nodes in the longitudinal direction u_x is possible. Therefore, the member must not be stressed by member loads acting in the local directions y or z.

The displacement in longitudinal direction is not allowed to be free at the ends of the cable.

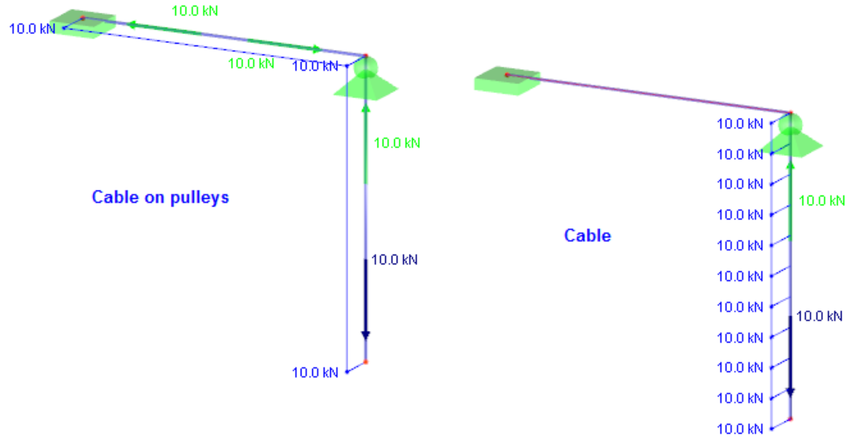


Figure 4.152: System with cable on pulley and cable member - axial forces and support reactions

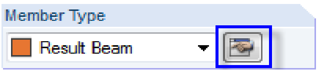
It is not important for the internal nodes of the polyline whether a nodal support is available or the member is connected to another construction. RFEM analyzes the total model of the cable member along the length of the polyline.

RFEM takes into account only displacements u_x and axial forces *N* for members of the member type *Cable on Pulleys*.

Result beam

Like a cut through the model, a result beam can be placed anywhere in the model as virtual member. Use it to display the internal forces of surfaces, members and solids in the form of integrated results. In this way, you can read in the display for example the resulting shear forces of a surface used for masonry design.

The result beam requires neither a support nor a connection to the model. Furthermore, it is not possible to apply loads to it.



The integration parameters must be set in a dialog box (see Figure 4.153) that you open by using the [Edit] button.

In the dialog section *Integrate Stresses and Forces*, define the result beam's zone of influence. The dialog graphic illustrates the parameters relevant for the individual options.

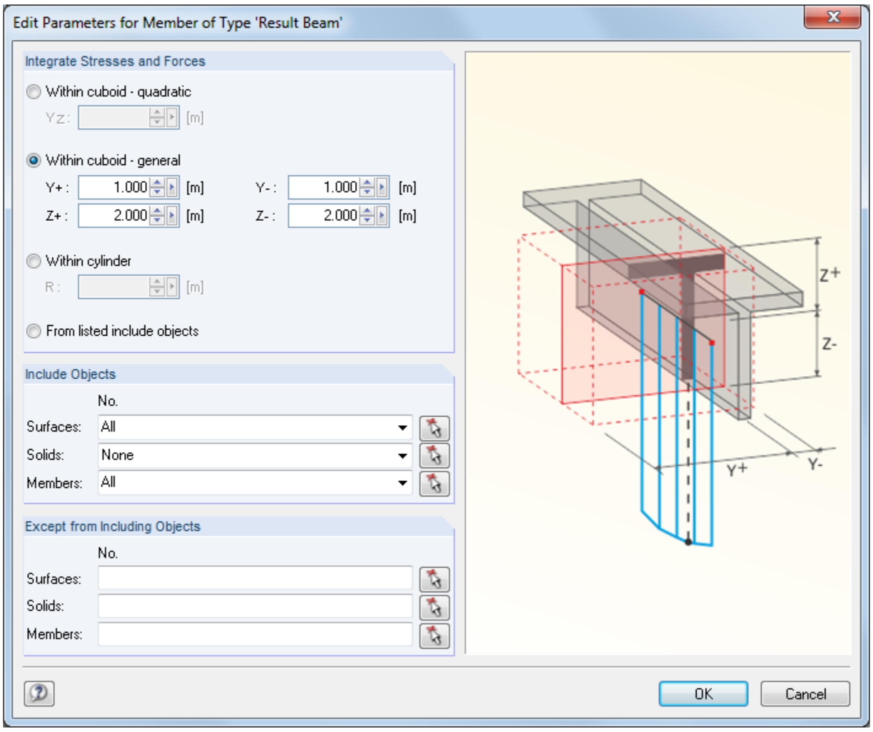
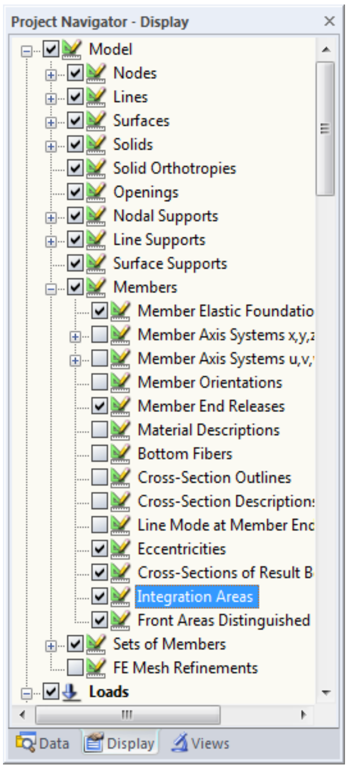


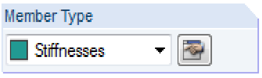
Figure 4.153: Dialog box *Edit Parameters for Member of Type 'Result Beam'*



The dialog section *Include Objects* allows for a specific selection of model elements whose results should be taken into account for the integration: surfaces, solids, members.

When the result beam is defined, you can activate and deactivate the display of integration areas in the *Display* navigator (see picture shown on the left).

Stiffnesses



The member stiffnesses can be directly specified in a dialog box that you open with the [Edit] button. Thus, the assignment of a cross-section is unnecessary.

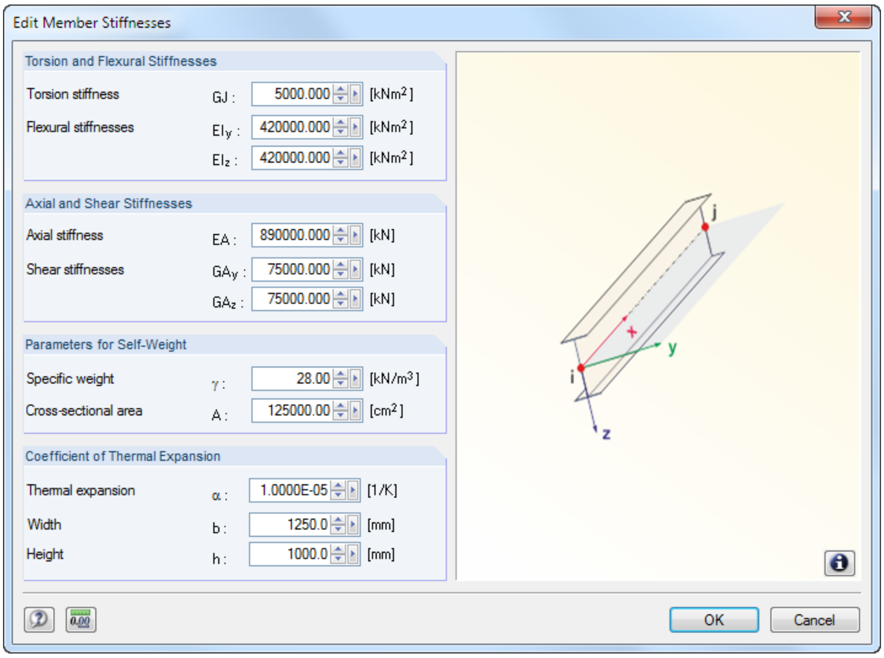


Figure 4.154: Dialog box *Edit Member Stiffnesses*

To look at the definition of the stiffness matrix, use the [Info] button.

Coupling

A coupling member is a virtual, very stiff member with definable rigid or hinged properties. It is possible to couple the degrees of freedom of the start and end nodes in four different ways. The axial and shear forces, respectively torsional and bending moments, are transferred directly from one node to the other. Couplings can be used to model special situations for the transfer of forces and moments.

RFEM calculates stiffnesses of couplings depending on the model in order to exclude numerical problems.

With the alternative *Rigid* member (see page 139) you can define coupling members considering also springs and nonlinearities of releases.

To control the display of coupling results, use the *Display* navigator.

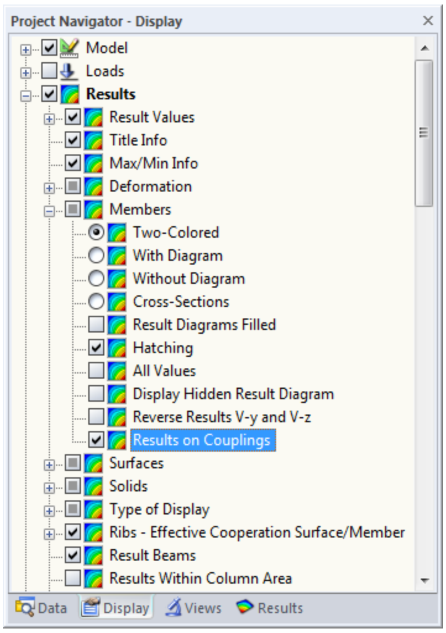


Figure 4.155: Activating the display for results of coupling members in the *Display* navigator

Spring

If *Spring* nonlinearities are set, you can open a new dialog box by using the [Edit] dialog button or the [...] button in the table.

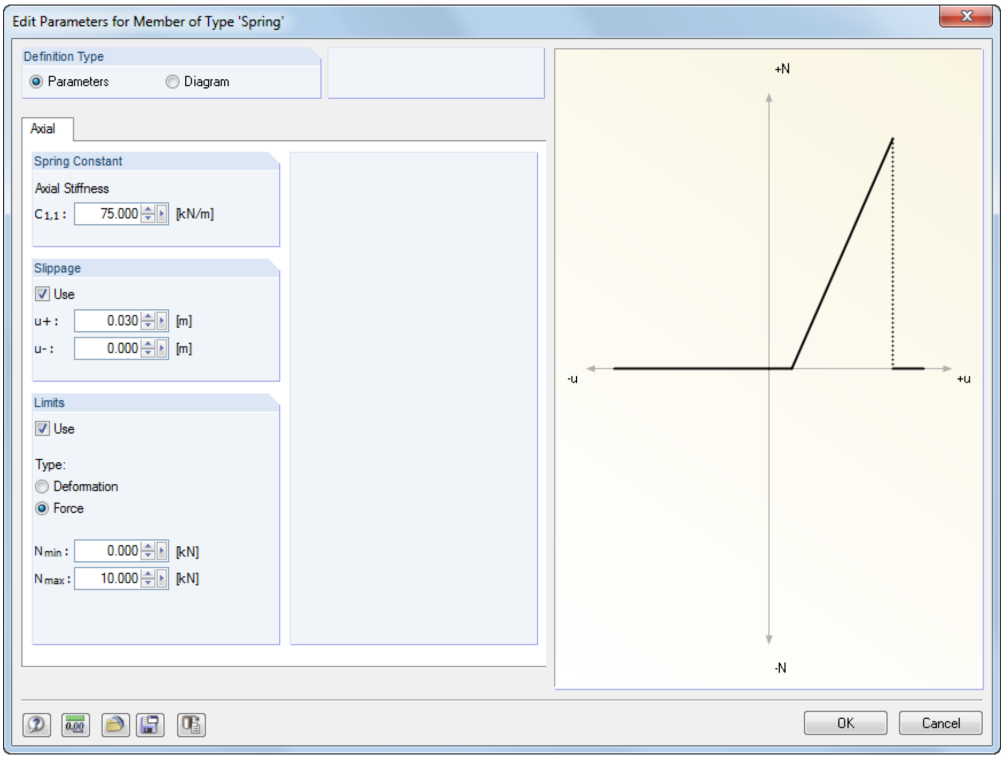


Figure 4.156: Dialog box *Edit Parameters for Member of Type 'Spring'*

Define the spring properties by *Parameters* or in a *Diagram*. The spring constant $C_{1,1}$ describes the stiffness of the member in its local x-direction according to the following relation:

$$k = \frac{E \cdot A}{l}$$

Equation 4.24

The *Slippage* specifies a zone of the deformation where the spring does not absorb any forces.

Furthermore, you have two options to define the spring *Limits*:

- *Deformation*: The values u_{min} and u_{max} define the geometric activity zone of the spring. The spring will act as a rigid member (stop) for deformations beyond the specified zone.
- *Force*: The values N_{min} and N_{max} define the activity zone for the forces that can be absorbed by the spring. If the axial force is beyond the defined limits, the spring fails.

When the *Diagram* option is set, you can define spring properties even more precisely. Those settings are largely identical with the options available for nonlinear member releases (see chapter 4.14, page 132).

Null

Neither a dummy nor its loads will be considered for the calculation. Use dummies to analyze, for example, changes in structural behavior if certain members are not effective. You do not need to delete these members, their loading will be kept as well.

Cross-section at member start and member end

The two input fields or table columns are used to define the cross-sections for the member start and end. The cross-section numbers refer to the entries in table 1.13 *Cross-Sections* (see chapter 4.13, page 117). Assignment is made easier by colors related to different cross-sections.

Tapered member

When you enter different numbers for the start and end cross-sections, a taper is created. RFEM interpolates the variable stiffnesses along the member according to polynomials of higher grade. Input of nonsense like a taper consisting of an IPE cross-section and a round steel will be identified by the plausibility check before the calculation starts.

The internal determination of tapered cross-section values is controlled by the *Taper Shape* set in the *Options* tab of the *New Member* dialog box, respectively the table column (see page 148).

Member rotation

The member-related coordinate system *x,y,z* is defined clockwise by right angles. The local axis **x** represents always the centroidal axis of the member, connecting the start node with the end node of the line (positive direction). Member axes **y** and **z**, respectively **u** and **v** for unsymmetrical cross-sections, represent the principal axes of the member.

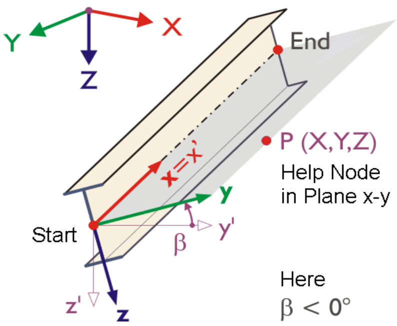
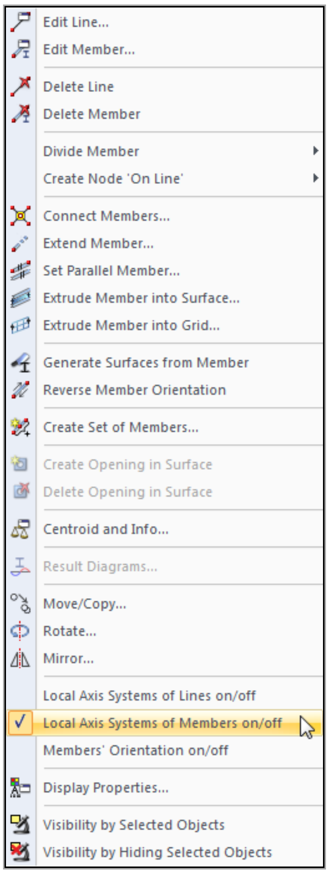


Figure 4.157: Member rotation and local member axes x,y,z (any spatial position)

The position of the local axes y and z is set automatically. Axis **y** runs perpendicular to the longitudinal axis x and parallel to the global plane XY. The position of axis **z** is determined by the right-hand rule. The z' component of the z-axis always shows downwards (which means in direction of the gravity) - irrespective of whether or not the global axis Z is oriented downward or upward.

To check the member position, use the 3D rendering. You can also use the *Display* navigator or the member context menu to display the *Member Axis Systems x,y,z*.



Member context menu

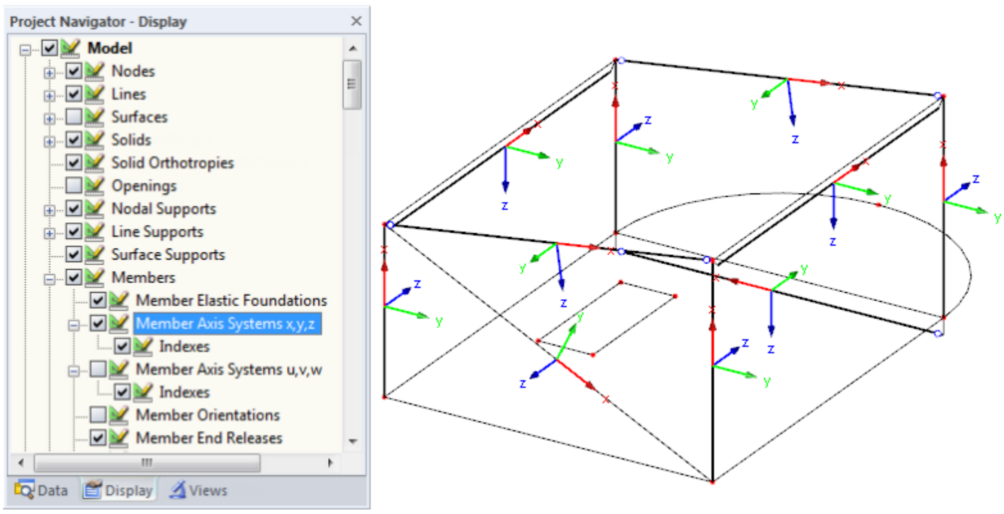


Figure 4.158: Selecting the local member axis systems in the *Display* navigator

Table column **N** informs you about the global axis running parallel to the member or indicates the plane spanned by the global axes where the member is lying. If there is no entry, the member is in an arbitrary spatial position.

If a member is aligned parallel to the global axis Z, which means in vertical position, the local axis **z**, of course, has no Z-component. In this case, the following rule applies: The local axis **y** will be aligned parallel to the global axis Y. Then, the position of the **z**-axis is determined by the right-hand rule.

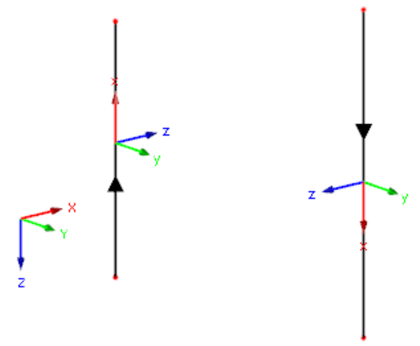


Figure 4.159: Vertical member position with different member orientations (β = 0°)

When a member located in a continuous set of column members is not exactly in vertical position (because of minor deviations of the nodal coordinates X or Y), the axes of the member can change their orientation: RFEM classifies the position of a member that is slightly inclined as "general". If you want to classify members in general position still as *vertical*, select **Regenerate Model** on the **Tools** menu (see chapter 7.1.3, page 255).

Member rotations can be applied in two ways:

Member rotation via angle β

You define an *Angle* β about which the member is rotated. If the rotation angle β is positive, axes y and z are rotated clockwise around the longitudinal member axis x.

Please note that the member rotation angle β and the cross-section rotation angle α' (see chapter 4.13, page 120) are summed up.

In 2D models, only member rotation angles of 0° and 180° are allowed.

Member rotation via help node

The member axis system is directed to a particular node. First, select the axis (y or z) that you want to be affected by the help node. Accordingly, the help node defines the plane x-y or the plane x-z of the member. Then, enter the help node. It is also possible to select it graphically or to create a new one. However, make sure that the node does not lie on the straight line that is defined by the x-axis of the member.

The following example shows columns that are aligned towards the center point.

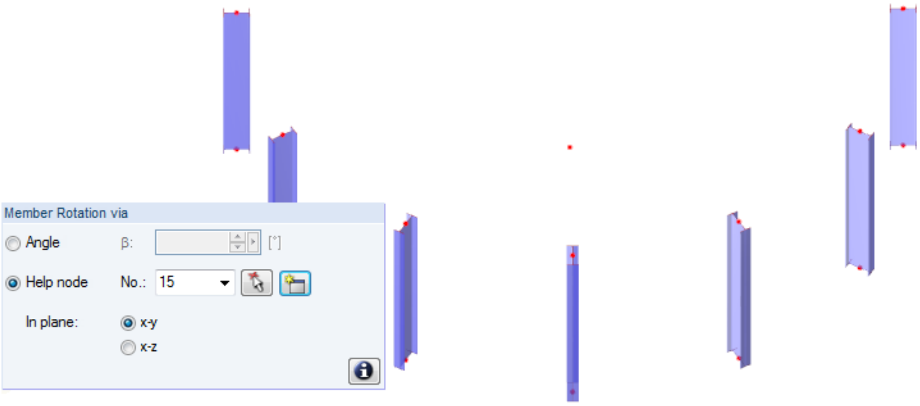


Figure 4.160: Member rotation via help node

Changes of the local member axis system may affect the signs of internal forces. The following figure illustrates the general sign rule.

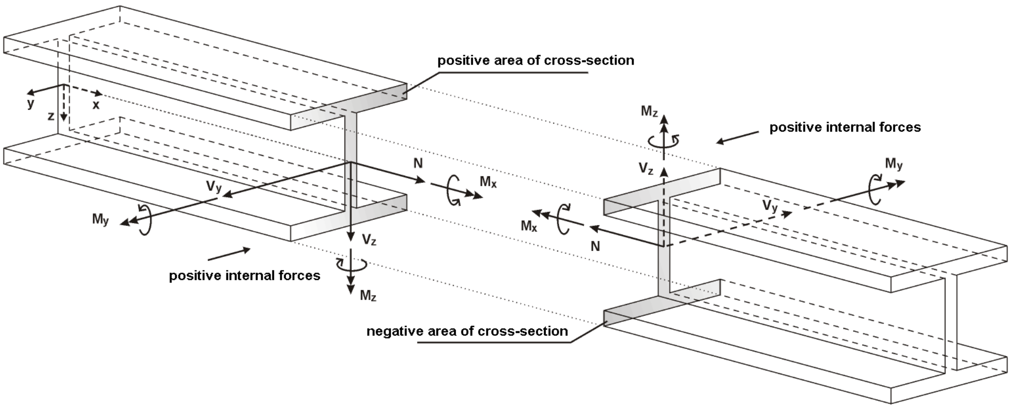


Figure 4.161: Positive definition of internal forces

The bending moment $\mathbf{M_y}$ is positive if tensile stresses occur on the positive member side (in direction of the z-axis). $\mathbf{M_z}$ is positive if compressive stresses occur on the positive member side (in direction of the y-axis). The sign definition for torsional moments, axial forces and shear forces conforms to the usual conventions. These internal forces are positive if they act in a positive direction.

Release at member start and member end

In these two table columns or input fields of the *New Member* dialog box, you can define releases controlling the transfer of internal forces on nodes. The release numbers refer to the entries available in table 1.14 *Member End Releases* (see chapter 4.14, page 128).

For some of the member types, entries are not possible because internal releases already exist.

Member eccentricity

In this table column or input field of the *Options* dialog tab (see Figure 4.151), you can assign an eccentric connection to the member. The numbers of the eccentricities refer to table 1.15 *Member Eccentricities* (see chapter 4.15, page 134). A connection type determines the eccentricities of both the member start and the member end.

Member division

Member divisions control the numerical output of internal forces and deformations along the member (see chapter 4.16, page 136). Use the settings in the table column or the input field of the *Options* dialog tab to assign divisions or to create new ones. The numbers of the divisions refer to the entries in table 1.16 *Member Divisions*.

A member division has neither influence on the determination of extreme values nor on the graphical results diagram (RFEM internally uses a more refined partition). As member divisions are not required in most cases, the default setting is 'None' or '0'.

Member elastic foundation

With this input field of the *Options* tab (see Figure 4.151) you can assign an elastic foundation to the member. The numbers of the elastic foundations are managed in table 1.19 *Member Elastic Foundations* (see chapter 4.19, page 153).

Member nonlinearity

This input field in the *Options* dialog tab (see Figure 4.151, page 138) makes it possible to provide the member with nonlinear properties. The numbers of the nonlinearities refer to the entries in table 1.20 *Member Nonlinearities* (see chapter 4.20, page 155).

Taper shape

If different cross-sections are defined for the member start and member end, this table column or input field in the *Options* tab offers you the choice between a *Linear* and a *Quadratic* taper application. In this way, it is possible to describe the taper geometry for the determination of the interpolated cross-section values.

In most cases, a linear taper course is existent: The height of the cross-section is changing evenly from the start of the cross-section to its end, the width remains more or less constant. However, if also the width of the cross-section is changing distinctly along the member (for example taper made of solid sections), it is recommended to use a square function for the interpolation of cross-section values.

Length

This table column indicates the absolute length of the member as distance between start and end node. Eccentricities are taken into account.

You can read the member length also in the work window: Place the mouse pointer on a member and wait a moment until the ScreenTip of the member appears.

Weight

The mass of a member is determined from the product of the cross-sectional area A and the specific weight of the material. RFEM applies g = 10 m/s² as gravitational acceleration.

Position

Table column **N** informs you about the global axis running parallel to the member or indicates the plane spanned by the global axes where the member is lying. If there is no entry, the member is in an arbitrary spatial position.

When a member located in a continuous set of column members is not exactly in vertical position (because of minor deviations of the nodal coordinates X or Y), the axes of the member can change their orientation: RFEM classifies the position of a member that is slightly inclined as "general". If you want to classify members in general position still as *vertical*, select **Regenerate Model** on the **Tools** menu (see chapter 7.1.3, page 255).

Effective lengths

The dialog tab *Effective Lengths* manages the *Effective Length Factors* $k_{cr,y}$ and $k_{cr,z}$.

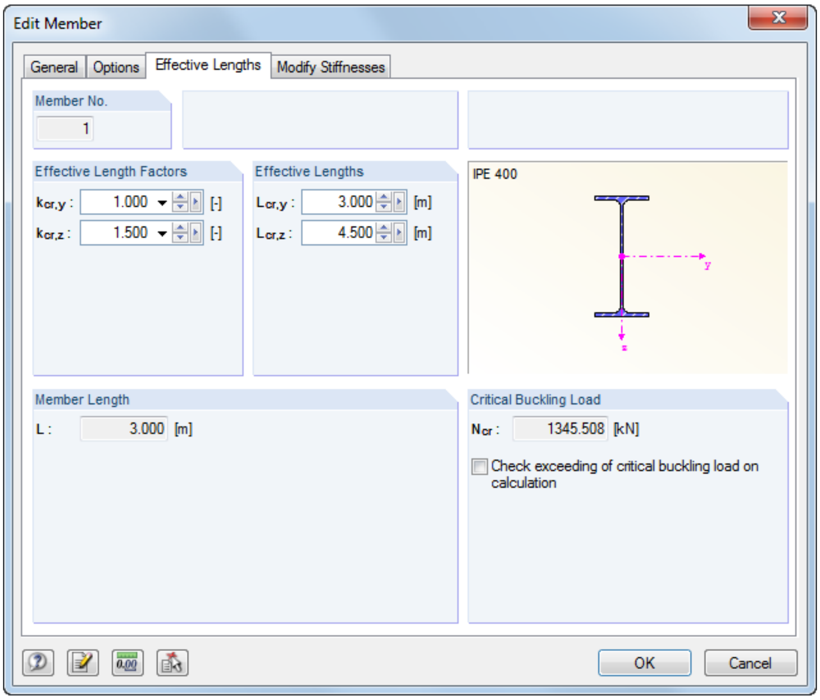


Figure 4.162: Dialog box *Edit Member*, tab *Effective Lengths*

The effective length factors can be adjusted separately for both member axes. The dialog fields to the right show the *Effective Lengths* resulting from the entered factors and the member length.

Effective length factors are significant for add-on modules like RF-STEEL EC3 where stability analyses are performed, but they play a secondary role for RFEM as for example buckling lengths for buckling members are determined internally from the boundary conditions, and then they are applied exactly.

In the dialog section *Critical Buckling Load*, you can decide if the flexural buckling load of the member will be checked during the calculation. The check box is ticked by default for truss, compression and buckling members. The dialog tab *Global Calculation Parameters* of the *Calculation Parameters* dialog box (see Figure 7.22, page 271) offers a global setting option for this kind of control.

Member as surface model

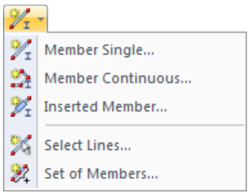
The context menu function *Generate Surfaces from Member* can be used to convert a member (1D elements) into adequate surface elements (2D elements) for detailed designs. The function is described in chapter 11.7.1.5 on page 504.

Double members

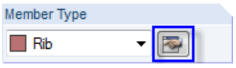
Generally, overlapping members in the model are not desired. So when you define a new member on the nodes of an already existing member, RFEM will delete the old member automatically.

To prevent RFEM from deleting already defined members, select *Allow Double Members* on the *Edit* menu. RFEM will consider the stiffnesses of both members in the calculation.

4.18 Ribs



List button *Member*



General description

Ribs are a special type of members. To create a rib, a member must already exist. Ribs can be used to represent T-beams in the FEA model by defining eccentricities and effective widths.

Ribs are primarily suited for models with reinforced concrete elements: You can use the rib internal forces and cross-sections for design in the add-on module **RF-CONCRETE Members**. However, when you want to model a steel plate with a welded "rib", use a surface with an eccentrically connected member.

You can define a rib directly with the navigator context menu of *Ribs* or by dialog input. When you create a new member and you select the *Member Type* **Rib** (see chapter 4.17, page 138), you can use the enabled [Edit] button to define the parameters. It is also possible to access the dialog box below by using the menu or the context menu in the navigator.

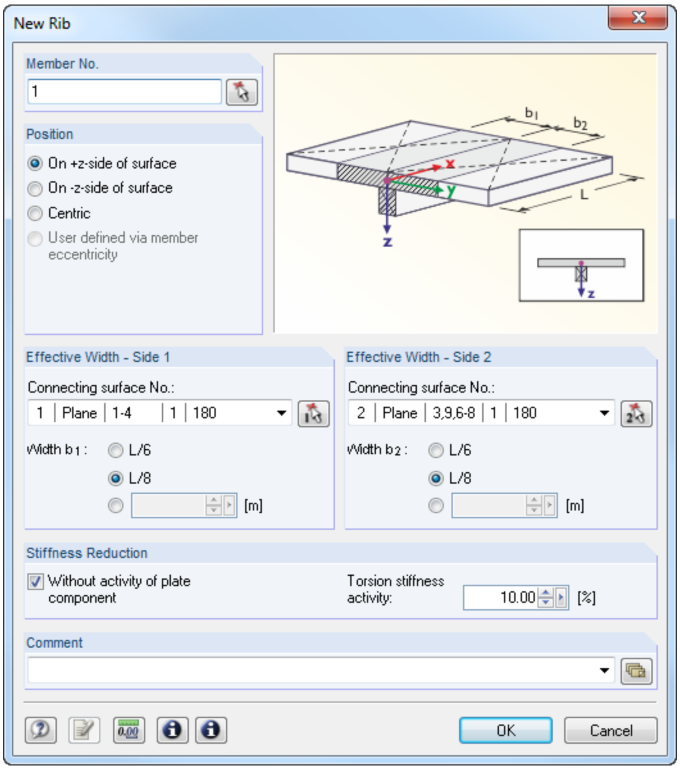
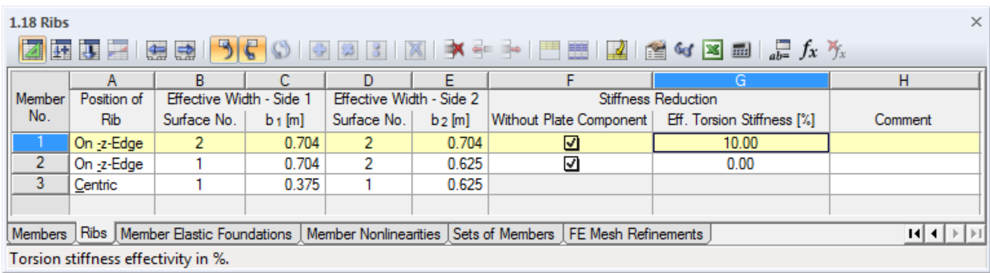


Figure 4.163: Dialog box *New Rib* (for model type *2D - XY*)


1.18 Ribs

Member No.	Position of Rib	Effective Width - Side 1 Surface No.	b1 [m]	Effective Width - Side 2 Surface No.	b2 [m]	Stiffness Reduction Without Plate Component	Eff. Torsion Stiffness [%]	Comment
1	On -z-Edge	2	0.704	2	0.704	☑	10.00	
2	On -z-Edge	1	0.704	2	0.625	☑	0.00	
3	Centric	1	0.375	1	0.625			

Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members | FE Mesh Refinements

Torsion stiffness effectivity in %.

Figure 4.164: Table 1.18 *Ribs*

Position of rib

Generally, a rib is a member that is eccentrically arranged. The eccentricity is determined automatically from half of the surface thickness and half of the member height (table 1.15 *Member Eccentricities* is not affected). You can also define it manually. Due to the eccentricity of the rib the rigidity of the model is increased.

The following arrangement options are available:

On +/–z-side of surface

The eccentricity as sum from half of the surface thickness and half of the web height is automatically applied in direction of the positive or negative surface axis **z**. To display and check the surface axes *x,y,z* in the graphic, use the *Display* navigator (see Figure 4.115, page 116).

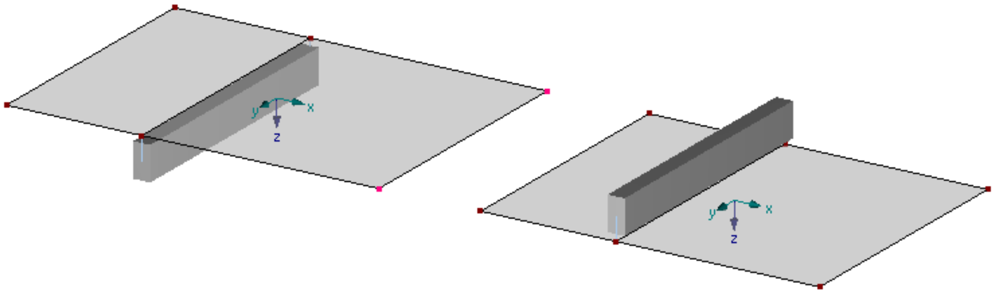


Figure 4.165: Ribs on positive z-side (left) and negative z-side (right) of the surfaces

Centric

The rib is modeled without eccentricity. The centroidal axis lies in the center of the surface.

User-defined via member eccentricity

You define the member eccentricity manually in the dialog box *New Member Eccentricity*, respectively in table 1.15 (see chapter 4.15, page 134), and then you assign it to the member.

You can check the rib position in the rendering mode without problems: In the *Display* navigator, select the two display options for solid models: *Members* → *Cross-sections* and *Surface* → *Filled incl. thickness*.

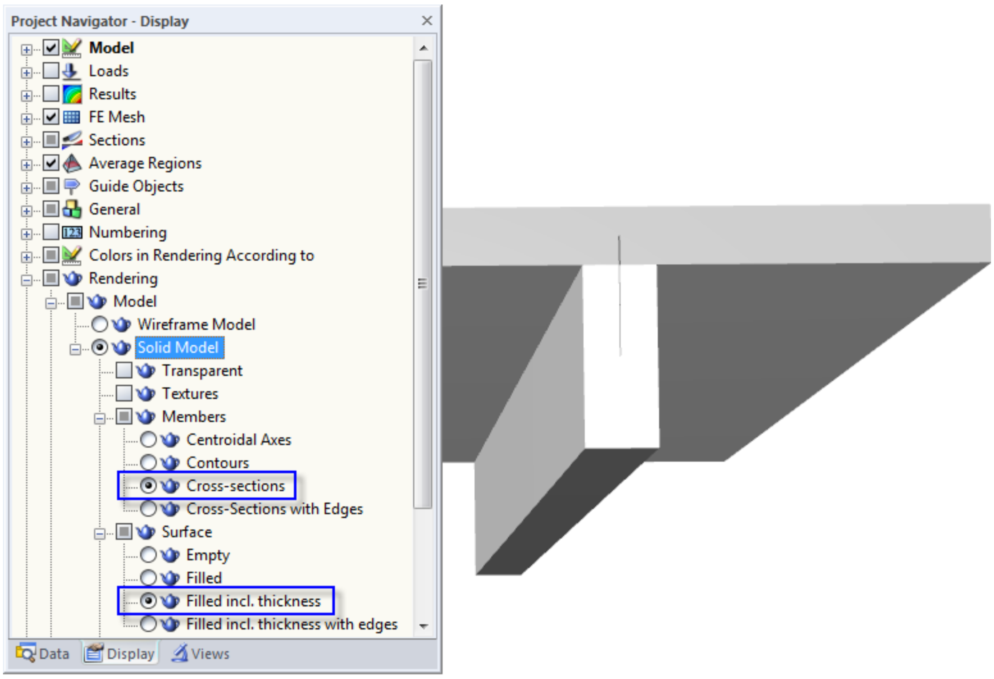


Figure 4.166: *Display* navigator: *Rendering* → *Model* → *Solid Model*

Effective width

When modeling 3D structures, the effective width has no influence on the stiffness because the increased stiffness is already taken into account by the eccentric member. The effective width affects only the internal forces. For 2D models (model type *2D - XY*), however, the stiffness is controlled by the settings applied to the *Stiffness Reduction* (see paragraph below).

If an eccentrically connected beam is used instead of a rib, then the RFEM 4 model assigns internal forces to both member and slab. But in reinforced concrete design the member and a certain part of the surface are considered as a single unit, namely the floor beam (T-beam). To determine the internal forces for the floor beam, the bending moment in the member must be increased by the product from the axial force in the slab and the eccentricity. To determine the axial force in the plate, you have to know the area where the axial forces are summed up. Therefore, you have to specify the effective widths as well as the surfaces.

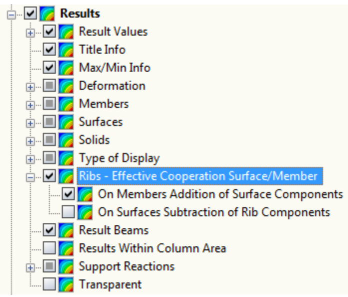
Connecting surface

The effective widths of the rib must be defined separately for the left and right side. Often, you can keep the *Autodetect* setting in the *Connecting surface* list available in the dialog box *New Rib*. Only if more than two surfaces adjoin each other along the line of the rib, you have to determine the connecting surfaces explicitly.

Effective width

The *Width* b_1 respectively b_2 can be directly entered into the input field or calculated automatically from the member length by selecting the options *L/6* and *L/8*. When confirming the dialog box, RFEM determines the effective widths and enters the values.

Please note: When the member length is modified subsequently, the effective widths will not be adjusted automatically!

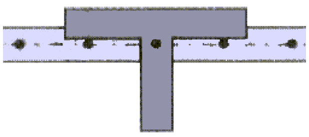


After the calculation, the effective components of the surfaces can be considered for the member results. In the *Display* navigator, click *Results* and select *Effective Cooperation Surface/Member*. The member result diagrams allow for a specific evaluation of the rib internal forces as well (see chapter 9.5, page 357).

Stiffness reduction

This dialog section respectively these table columns are only shown if the model type *2D - XY* has been set in the general data (see Figure 12.23, page 554). In comparison to spatially defined models where ribs can be taken into account in the FE analysis as eccentrically arranged members anyway, RFEM uses another analysis approach for floor beams.

Without activity of plate component



For the calculation RFEM applies a substitute cross-section whose stiffness is determined from the member cross-section and the effective plate component of the surfaces. Thus, the stiffness of the plate is determined twice for eccentrically arranged ribs because it is effective in the substitute cross-section as well as directly by the surface elements. If the check box *Without activity of plate component* is ticked, the stiffness component of the plate will not be considered in the substitute cross-section.

Torsion stiffness activity

This input field is used to reduce the torsional rigidity of the rib.

4.19 Member Elastic Foundations

General description

While nodal supports represent a support on both member ends, member elastic foundations provide an elastic support of the member along its entire length. Use elastic member foundations to model for example foundation beams considering soil properties. If the elastic foundation is not effective in case of tensile or compressive stresses, it is possible to take into account nonlinear effects in the calculation.

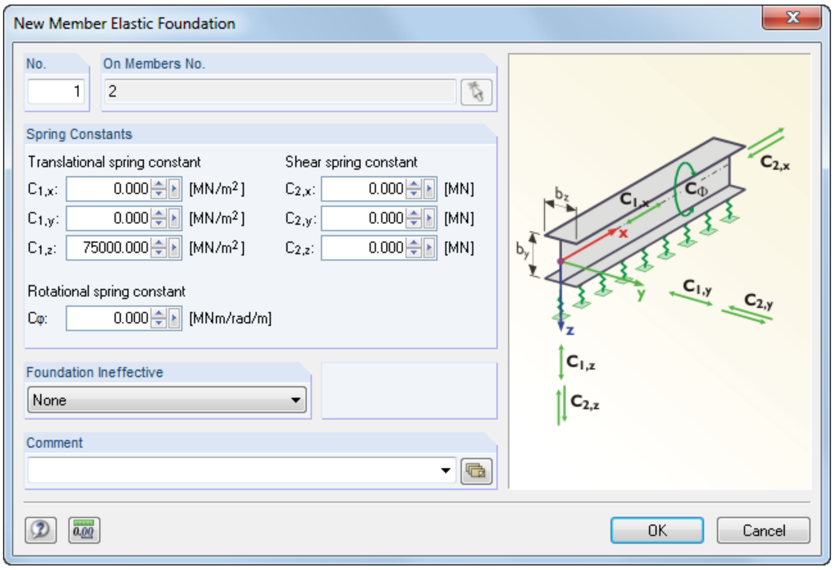
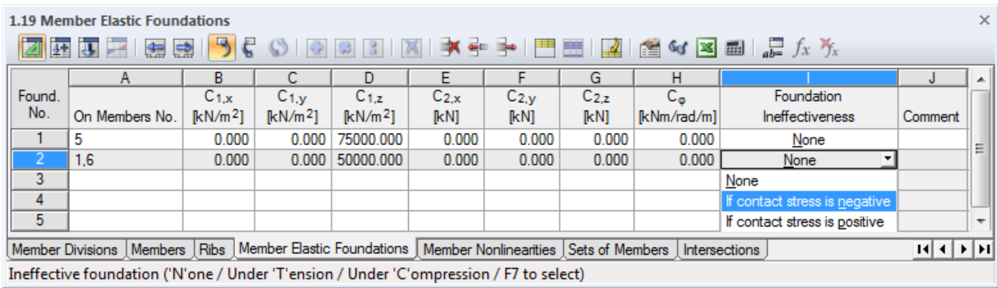


Figure 4.167: Dialog box *New Member Elastic Foundation*


1.19 Member Elastic Foundations

Found. No.	A On Members No.	B $C_{1,x}$ [kN/m2]	C $C_{1,y}$ [kN/m2]	D $C_{1,z}$ [kN/m2]	E $C_{2,x}$ [kN]	F $C_{2,y}$ [kN]	G $C_{2,z}$ [kN]	H C_{φ} [kNm/rad/m]	I Foundation Ineffectiveness	J Comment
1	5	0.000	0.000	75000.000	0.000	0.000	0.000	0.000	None	
2	1,6	0.000	0.000	50000.000	0.000	0.000	0.000	0.000	None	
3									None	
4									If contact stress is negative	
5									If contact stress is positive	

Member Divisions | Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members | Intersections

Ineffective foundation ('N'one / Under 'T'ension / Under 'C'ompression / F7 to select)

Figure 4.168: Table 1.19 *Member Elastic Foundations*

On members

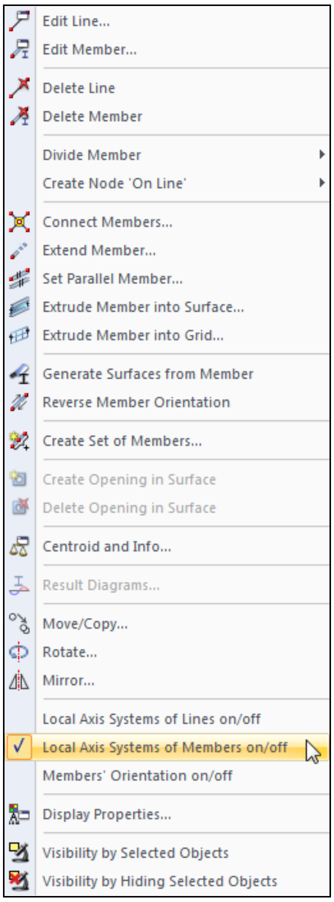
Member elastic foundations can only be defined for the member type *Beam*. Enter the number of the member into the table column or input field. You can also define it graphically.

Spring constants

Translational spring

You have to specify the parameters of translational springs in direction of the local member axes x, y and z.

The stiffness moduli E_S of Table 4.8 serve as reference values. Please note that input in RFEM refers to the modulus of subgrade reaction to be determined by taking into account the form factor.



Member context menu

Soil type	E_S (static loading)	E_S (dynamic loading)
Sand, compact	40 – 100	200 – 500
Gravel sand, compact	80 – 150	300 – 800
Clay, semi-solid to solid	8 – 30	120 – 250
Clay, stiff-plastic	5 – 20	70 – 150
Mixed soil, semi-solid to solid	20 – 100	200 – 600

Table 4.8: Stiffness moduli of selected soil types in [N/mm^2]

The values of Table 4.8 represent area-related characteristic values: They describe the area force in [N/mm^2] that is required to compress the soil by 1 mm. Thus, the unit would be interpreted in a solid-related way as [N/mm^3].

For foundation beams used for example to model strip foundations, you have to determine the spring coefficient taking into account the cross-section width. In this way, you get a translational spring in [N/mm^2] that is related to the member. The spring indicates the member force in [N/mm] that is required to compress the soil by 1 mm – therefore the unit [N/mm^2] for the input. The result must be entered as translational spring $C_{1,z}$: For strip foundations (members in horizontal position) the local z-axis is usually directed downwards.

The spring stiffnesses are considered as design values.

Use the *Display* navigator or the context menu of the member to display the local member axes (see Figure 4.158, page 145).

Shear spring

Shear springs are used to determine the shear capacity of the soil. The spring constants C_2 are determined from the product of $\nu \cdot C_{1,z}$, with the Poisson's ratio ν to be assumed between 0.125 and 0.5 for sand and gravely soil, and between 0.2 and 0.4 for clayey soil.

Rotational spring

Enter the constant of a rotational spring into the dialog input field or table column. The constant hinders the member rotating about its longitudinal axis.

Ineffective foundation

If the elastic foundation is not effective in case of tensile or compressive stresses, assign the nonlinear property *Failure* to the foundation type.

Please note that the failure criterion *Failure if negative* or *positive contact stress in z* only refers to the local member axis **z**. The nonlinearity does <u>not</u> apply to the translational springs in direction of the local axes x or y! Thus, a biaxially effective failure of foundation members is not possible.

An ineffectivity in case of a negative contact stress has the following meaning: The foundation is without effect if a member element moves in opposite direction of the local axis z.

When failure criteria is applied, it is recommended to check position and orientation of the local z-axes (see Figure 4.158, page 145). It might be necessary to rotate members.

The member division of members with elastic foundations can be adjusted in the dialog tab *Global Calculation Parameters* of the dialog box *Calculation Parameters* (see chapter 7.3, page 271).

4.20 Member Nonlinearities

General description

Member nonlinearities are used to represent nonlinear relations between force (or moment) and strain in members.

Some nonlinear properties can be defined already when specifying the member type. A tension member, for example, is a truss for which the strain is increasing proportional to the tension force, but whose strain may rise under compression without a verifiable force being required for it.

In principle, member nonlinearities can be assigned to any type of member. Of course, combinations have to make sense. A compression member with the design criterion "Failure under compression" would cause problems during the calculation. Therefore, member nonlinearities are not allowed for the member types tension, compression, buckling and cable member as well as for members with cross-sections of the type *Dummy Rigid* (see page 118).

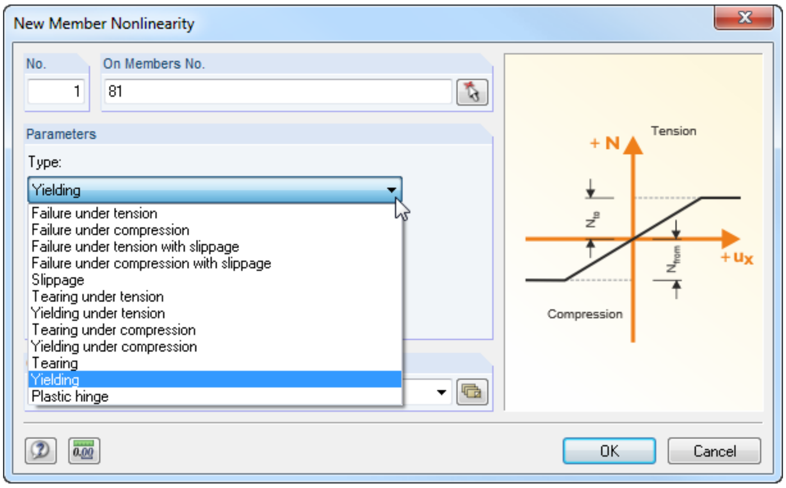
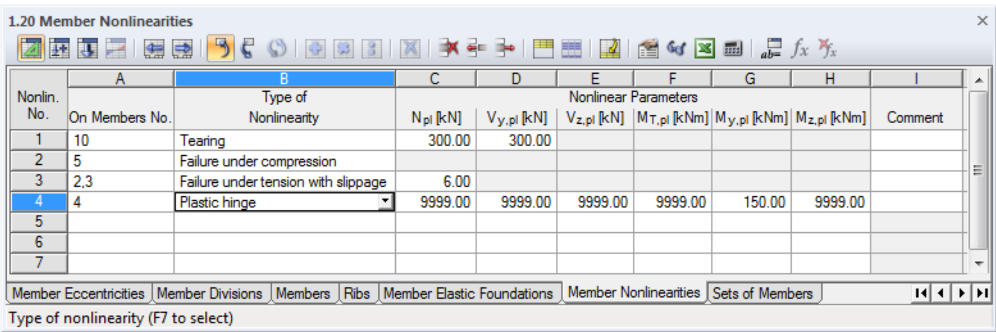


Figure 4.169: Dialog box *New Member Nonlinearity*

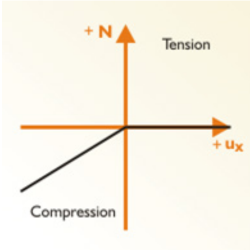
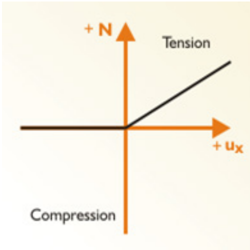
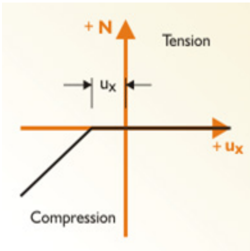
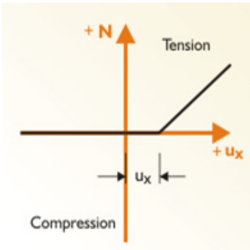
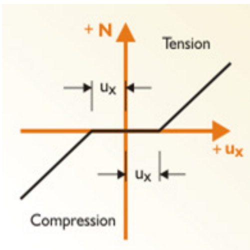
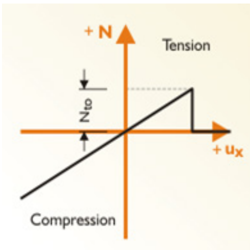

1.20 Member Nonlinearities

Nonlin. No.	A On Members No.	B Type of Nonlinearity	C N_{pl} [kN]	D $V_{y,pl}$ [kN]	E $V_{z,pl}$ [kN]	F $M_{T,pl}$ [kNm]	G $M_{y,pl}$ [kNm]	H $M_{z,pl}$ [kNm]	I Comment
1	10	Tearing	300.00	300.00					
2	5	Failure under compression							
3	2,3	Failure under tension with slippage	6.00						
4	4	Plastic hinge	9999.00	9999.00	9999.00	9999.00	150.00	9999.00	
5									
6									
7									

Member Eccentricities | Member Divisions | Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members

Type of nonlinearity (F7 to select)

Figure 4.170: Table 1.20 *Member Nonlinearities*

Nonlinearity	**Diagram**	**Description**
Failure under tension		The member cannot absorb tensile forces.
Failure under compression		The member cannot absorb compressive forces.
Failure under tension with slippage		The member cannot absorb tensile forces. Compressive forces are not absorbed until the slippage u_x is overcome.
Failure under compression with slippage		The member cannot absorb compressive forces. Tensile forces are not absorbed until the slippage u_x is overcome.
Slippage		The member absorbs axial forces only after having exceeded a strain or shortening by the value u_x. **Please note**: A line refinement on a member with *Slippage* causes an internal member division into small member parts. The slippage criterion will be applied to each of these partial members.
Tearing under tension		The member absorbs compressive forces without limitation but fails if tensile forces exceed N_{to}.

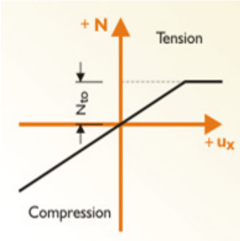
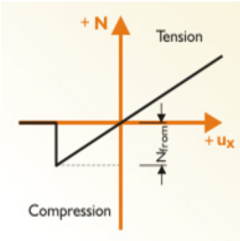
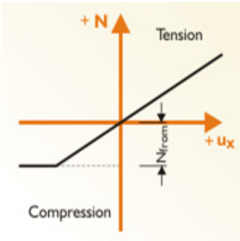
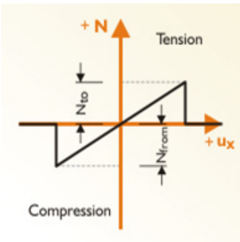
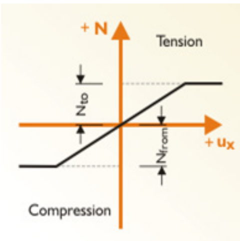
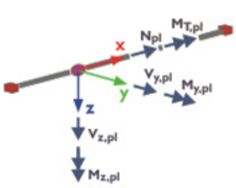
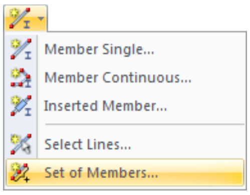
Yielding under tension		The member absorbs compressive forces without limitation, but only a maximum tensile force of N_{to}. If strain increases, tensile force remains constant in the member.
Tearing under compression		The member absorbs tensile forces without limitation but fails if compressive forces exceed N_{from}.
Yielding under compression		The member absorbs tensile forces without limitation, but only a maximum compressive force of N_{from}. If strain increases, compressive force remains constant in the member.
Tearing		The member fails when reaching the compressive force N_{from} or the tensile force N_{to}.
Yielding		The member starts to yield if the compressive force N_{from} or the tensile force N_{to} are reached: If strain increases, force remains constant.
Plastic hinge		If a plastic design force is reached on a location of the member, a plastic hinge is formed there for the internal force. The internal forces must be entered as absolute values. For components of internal forces not resulting in plastifications, you have to enter high values.

Table 4.9: Member nonlinearities

4.21 Sets of Members

General description



List button *Member*

Sets of members must be understood as combined members. Use a set of members to treat several members like a single member as it may be preferable for some locations in the structural system (for example for lateral-torsional buckling analysis, design of continuous beams, load application).

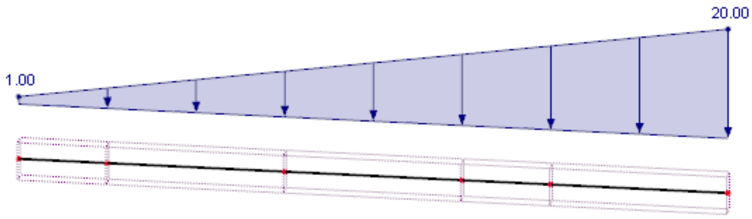


Figure 4.171: Continuous members with trapezoidal load

The figure above shows a linearly variable load acting on the complete length of a set of members.

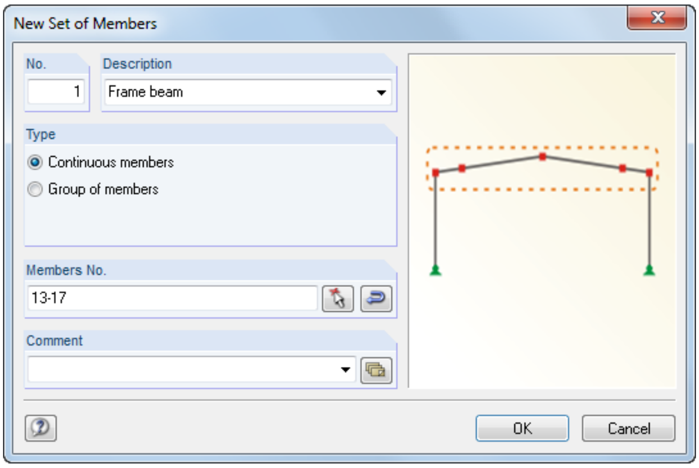
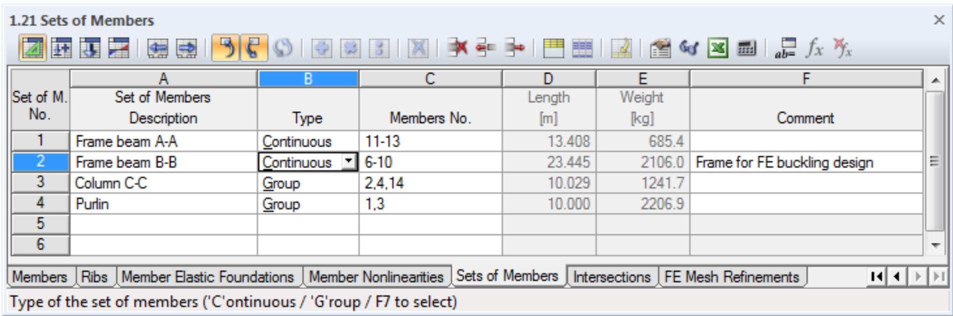


Figure 4.172: Dialog box *New Set of Members*


1.21 Sets of Members

Set of M. No.	A Set of Members Description	B Type	C Members No.	D Length [m]	E Weight [kg]	F Comment
1	Frame beam A-A	Continuous	11-13	13.408	685.4	
2	Frame beam B-B	Continuous	6-10	23.445	2106.0	Frame for FE buckling design
3	Column C-C	Group	2,4,14	10.029	1241.7	
4	Purlin	Group	1,3	10.000	2206.9	
5						
6						

Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members | Intersections | FE Mesh Refinements

Type of the set of members ('C'ontinuous / 'G'roup / F7 to select)

Figure 4.173: Table 1.21 *Sets of Members*

Description

You can enter any name for the set of members. You can also use the list to select a name. Manually entered descriptions are saved in the list and are instantly available for selection.

Type

There are two different types of member sets:

- **continuous members**
- **group of members**

Continuous members are created by connected members that are not branching out. It would be possible to draw them with a pencil without interrupting the continuous line.

Figure 4.174: Continuous members

A group of members consists of connected members that may branch out.

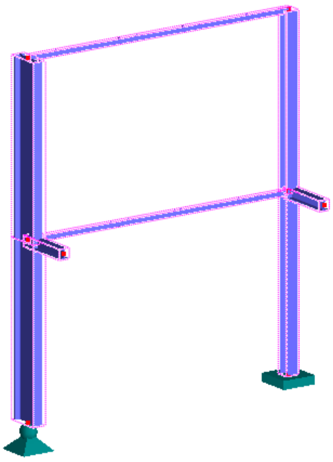


Figure 4.175: Group of members

In some add-on modules it is possible to design sets of members. Often, the design can be performed only for continuous members because parameters such as buckling lengths must be clearly defined.

Members

In the input field of the dialog box or the column in the table, enter the numbers of the members that form the set of members. You can also use the [↖] function to select them graphically in the work window. Use the button [Reverse Orientation of Members] to change the order of member numbers and thus the direction of the member set.

The quickest way to define a set of members is as follows: Select the relevant members in the work window by using the pointer drawing an enclosing window. You can also use the multiple selection by holding down the [Ctrl] key. Then, right-click one of the selected members. The context menu of the member opens where you point to **Member** and select **Create Set of Members** (for member groups) or select **Create Set of Members** (for continuous members). The dialog box *New Set of Members* opens and presets the numbers of the selected members.

Length

The total length of the set of members is determined from the sum of the individual member lengths.

Weight

The mass of the set of members is determined from the sum of the individual member masses.

4.22 Intersections

General description

If surfaces intersect and internal forces are transferred on the common line, you have to create an intersection. Otherwise, you would have two independent subsystems without any connection. The following example demonstrates the effect.

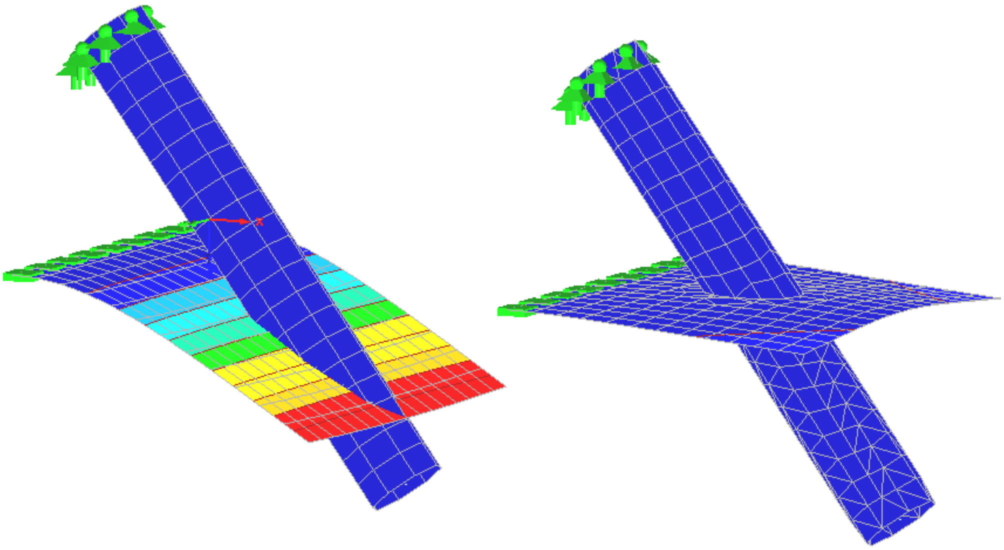


Figure 4.176: Deformations due to self-weight without intersection (left) and with intersection (right)

Each time the model is changed, RFEM must recalculate the intersections. Recalculating data is very time-consuming for the graphical representation when complex models are designed. The input is slowed down accordingly.

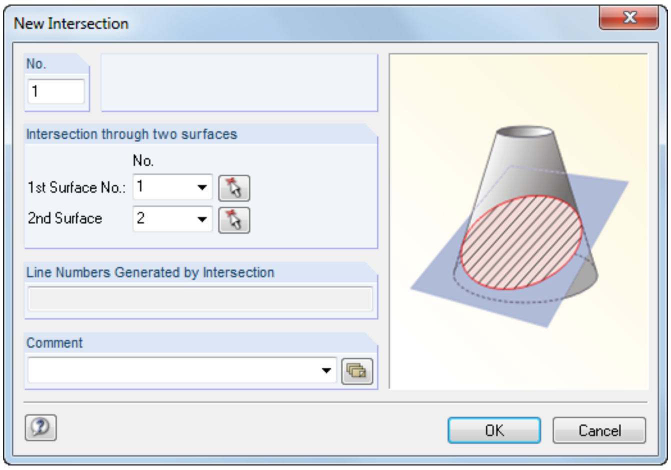
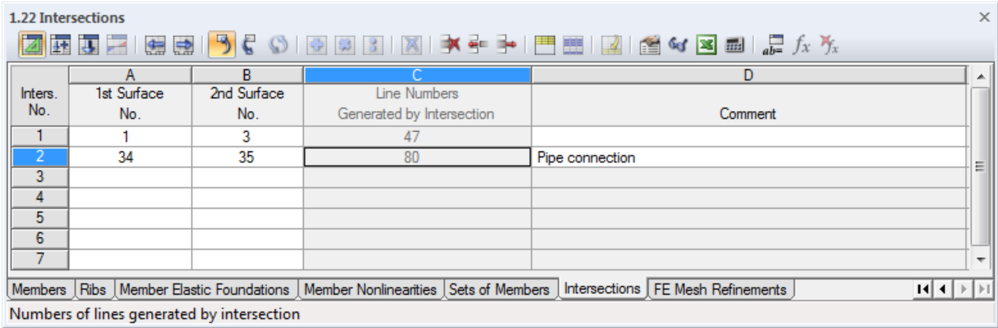


Figure 4.177: Dialog box *New Intersection*


1.22 Intersections

Inters. No.	A 1st Surface No.	B 2nd Surface No.	C Line Numbers Generated by Intersection	D Comment
1	1	3	47	
2	34	35	80	Pipe connection
3				
4				
5				
6				
7				

Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members | Intersections | FE Mesh Refinements

Numbers of lines generated by intersection

Figure 4.178: Table 1.22 *Intersections*

Intersection of two surfaces

In the input fields or table columns, enter the numbers of the two intersecting surfaces. In the dialog box *New Intersection*, you can select the surfaces from the list. You can also use the [↖] function to select them graphically.

Intersections (of even more than two surfaces, if needed) can be created quickly in the graphic: Select the surfaces by using the pointer drawing an enclosing window. You can also use the multiple selection by holding down the [Ctrl] key. Then, right-click one of the selected surfaces. The context menu opens where you point to **Surface** and select **Create Intersection**. Then, RFEM creates the intersection automatically.

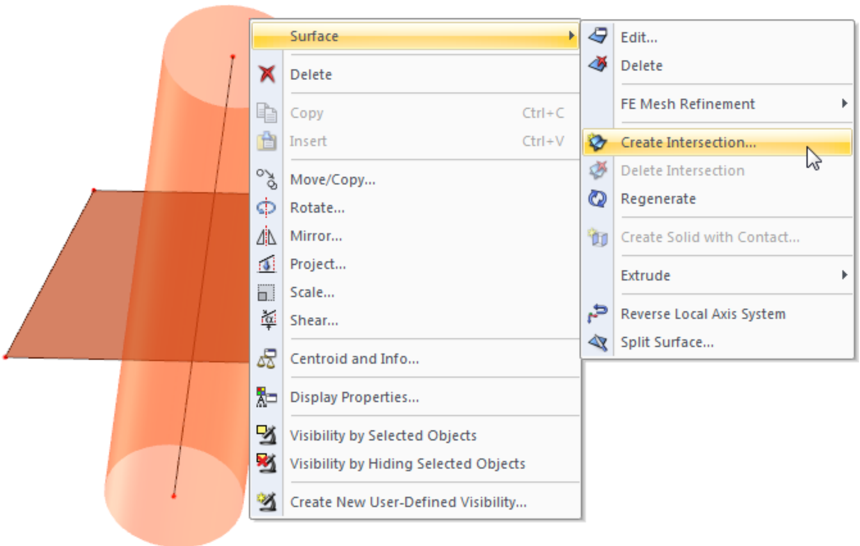


Figure 4.179: Context menu *Surface* → *Create Intersection*

Line numbers generated by intersection

When creating an intersection, RFEM generates a line that is shared by both surfaces ("intersection line"). The number of the new line is shown in the dialog field and table column.

Intersection lines are marked as line type *Intersection* in table 1.2 *Lines*. The comment identifies them as *Generated* lines. The dialog box *Edit Line* can also be used for intersection lines so that you can assign member or support properties.

Active surface components

An intersection line divides a surface into components that can be individually set active or inactive. Inactive surface components are not displayed in the work window. Neither finite elements are created nor loading is applied. Only active surface components are existent for the equation solver.

Components of intersections can be activated and deactivated as follows:

- Dialog box *Edit Surface*

Double-click the original surface in the *Data* navigator. When you double-click the surface in the work window, use the button [Go to] available in the dialog tab *Component* (see Figure 4.69, page 80) of the dialog box *Edit Surface* to open the edit dialog box of the original surface.

Open the dialog tab *Integrated / Components* where the dialog section *Active Surface Components* lists all surface components that have been created during the generation of the intersection.

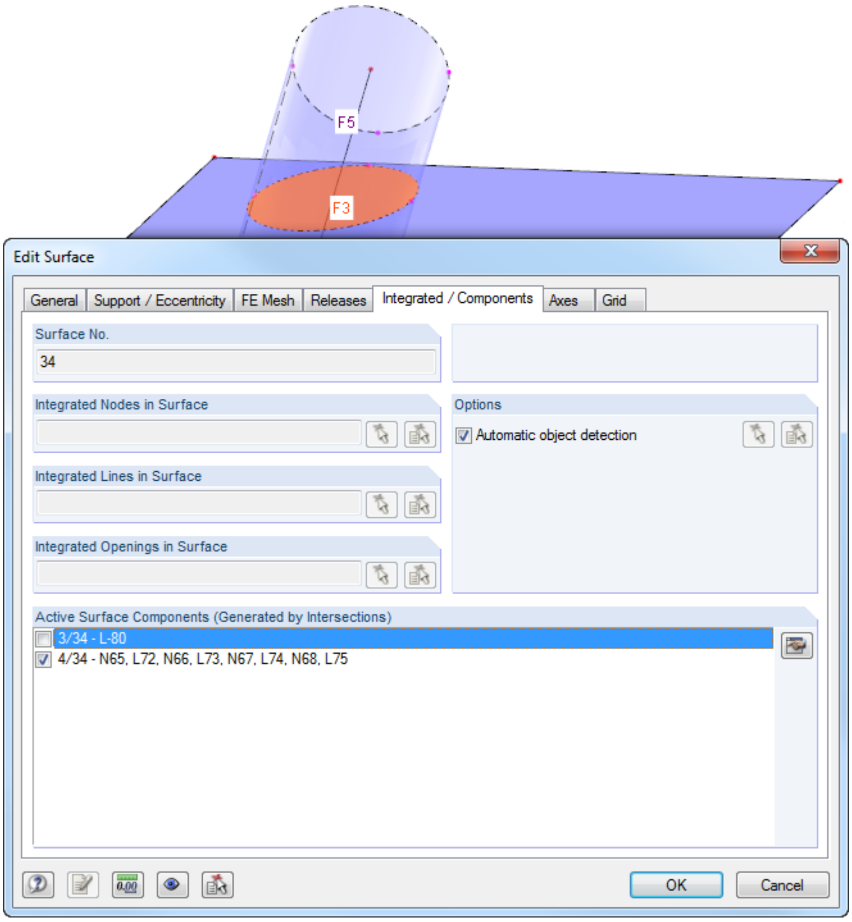


Figure 4.180: Dialog box *Edit Surface*, tab *Integrated / Components*

The surface component marked in the list is highlighted in color in the work window. To set a component inactive, clear the corresponding check box. Then, the inactive surface component is shown without filling color.

- Context menu of surface component in the *Data* navigator

  Directly right-click the navigator entry *Surfaces* or the component in the work window. Then, use the context menu to activate or deactivate the surface component.

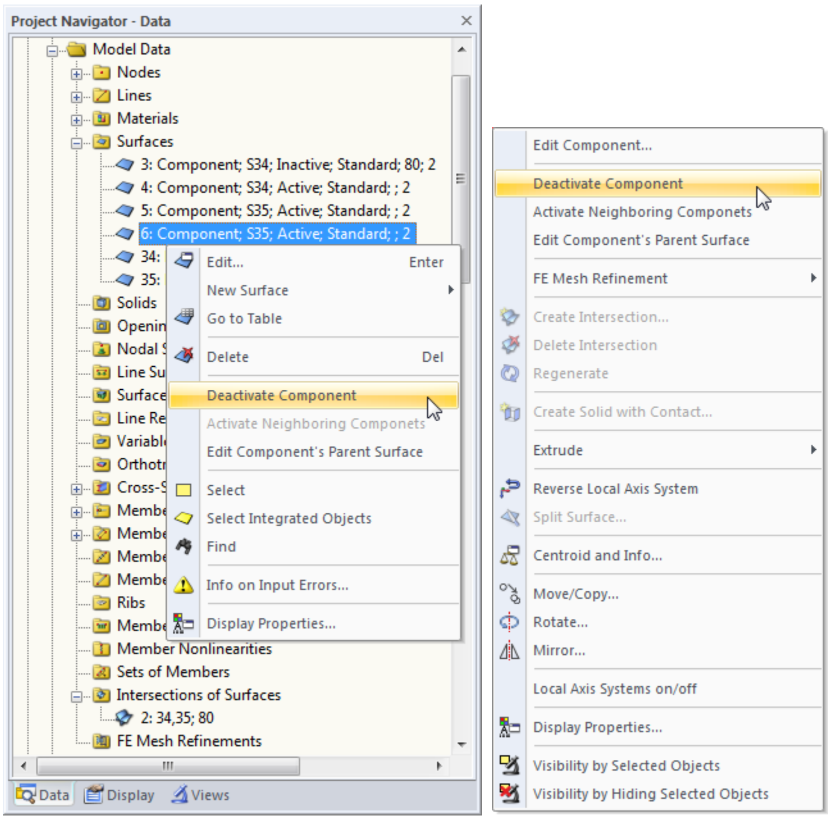


Figure 4.181: Context menu of a surface *Component* in the navigator and work window

The context menu provides further helpful functions for editing the component or original surface.

The geometric information of the original surface is kept internally by the program as it is needed for recalculating the intersection after modifications. Therefore, inactive surface components cannot be deleted but only be hidden.

4.23 FE Mesh Refinements

General description

If no FE mesh refinements have been defined, the FE mesh is generated with the preset target FE length. The global parameters of the FE mesh are described in chapter 7.2.2 on page 258.

The concept of the FE-mesh generator does not allow subsequent adjustments to the mesh. However, you can use FE mesh refinements to influence the mesh generation for specific areas. In this way, a user-defined discretization is done, as it may be required for example in nooks, for connections of members to surfaces or for a dynamic analysis of members.

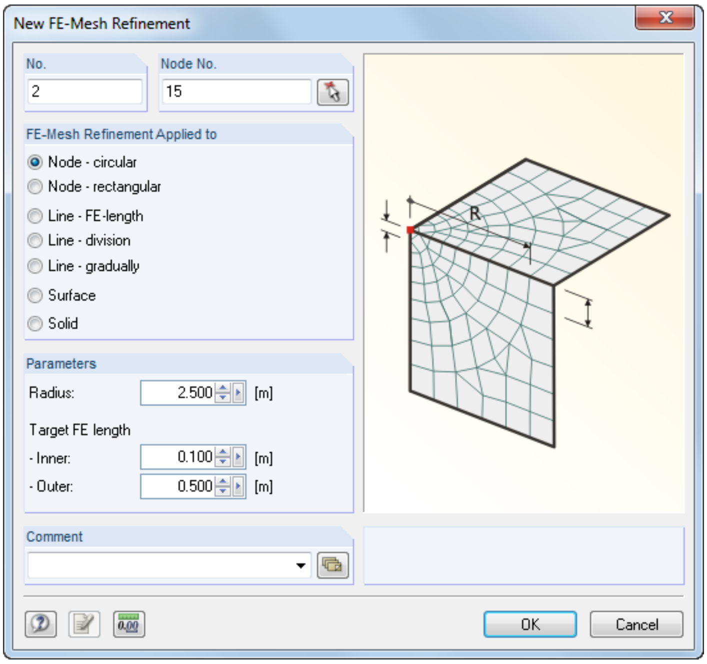
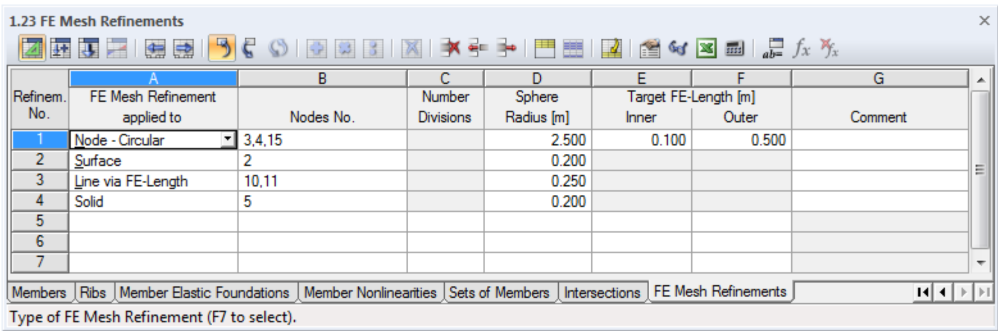


Figure 4.182: Dialog box *New FE-Mesh Refinement*



1.23 FE Mesh Refinements

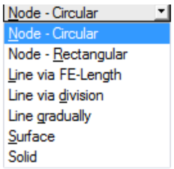
Refinem. No.	A FE Mesh Refinement applied to	B Nodes No.	C Number Divisions	D Sphere Radius [m]	E Target FE-Length [m] Inner	F Outer	G Comment
1	Node - Circular	3,4,15		2.500	0.100	0.500	
2	Surface	2		0.200			
3	Line via FE-Length	10,11		0.250			
4	Solid	5		0.200			
5							
6							
7							

Members | Ribs | Member Elastic Foundations | Member Nonlinearities | Sets of Members | Intersections | FE Mesh Refinements

Type of FE Mesh Refinement (F7 to select).

Figure 4.183: Table 1.23 *FE Mesh Refinements*

FE-mesh refinement applied to



With the settings in this dialog section or table column, you decide which objects are included by the FE mesh refinement and how the refinement is carried out. Different options are available for selection.

Node - circular

Define a radial refinement area around a node which is extended in all spatial directions.

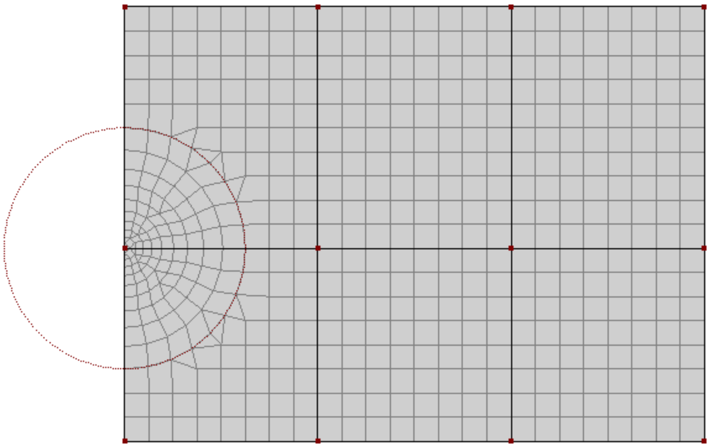


Figure 4.184: Circular refinement around a node

Node - rectangular

Instead of a circular refinement area, you can specify a rectangular zone for refinement.

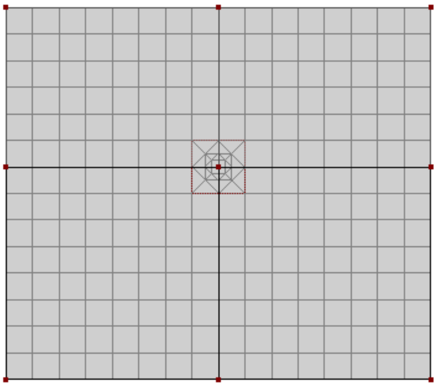


Figure 4.185: Rectangular refinement around a node

Refinement on line by FE length

Define regular spacings of FE nodes on a line.

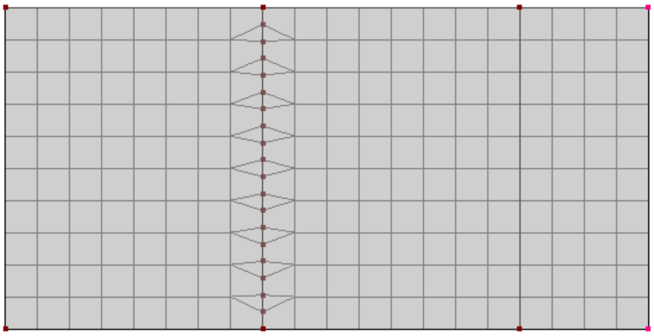


Figure 4.186: Refinement on line by FE length

Refinement on line by division

The FE mesh of a line can be refined in regular intervals. This type of refinement is especially useful for lines with member properties.

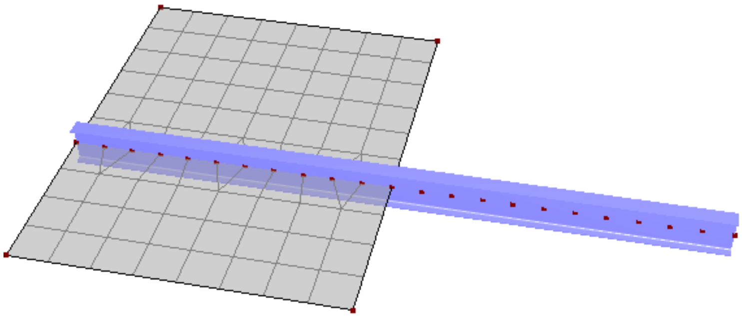


Figure 4.187: Refinement on line by division

Gradually refinement on line

The finite elements adjoining the line can be subdivided by a defined number of *n* rows. In this way, you can cover for example boundary areas of surfaces with a refinement. This refinement type is similar to the generation option *FE mesh refinement along lines* (see chapter 7.2.2, page 260) available for 2D plates.

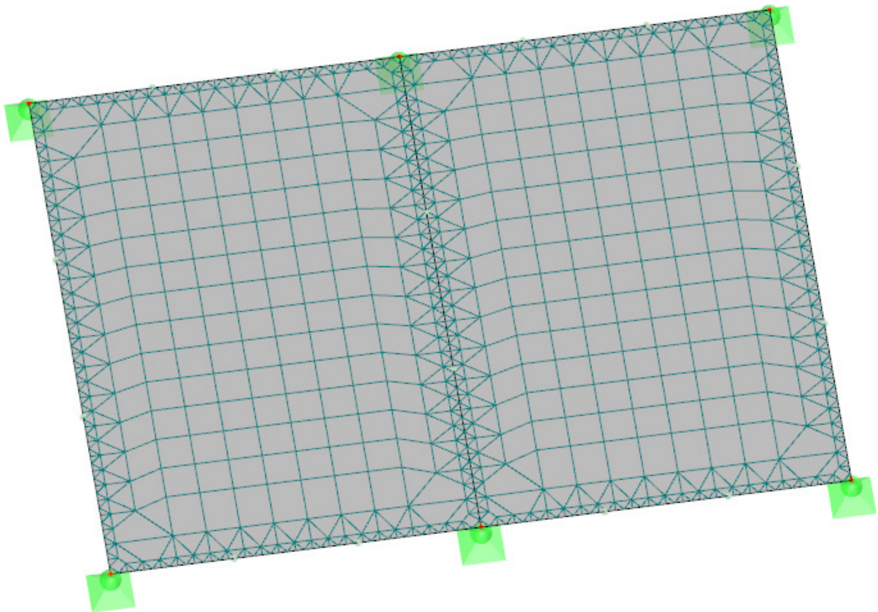


Figure 4.188: Gradual refinement on a line by two rows

Refinement on surface

Specify a lateral length of finite elements that is set as target length and mesh size for the entire surface.

This type of refinement can be used also for surfaces with low significance for the analysis: As "refinement" you enter a mesh size that is larger than the global target FE length.

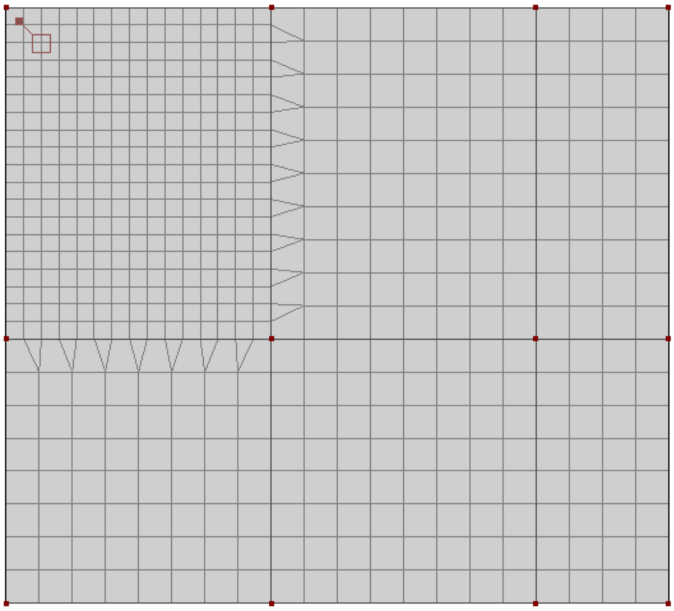


Figure 4.189: Refinement on a surface

Refinement on solid

FE mesh refinements can also be defined for solids to influence the generation of 3D elements.

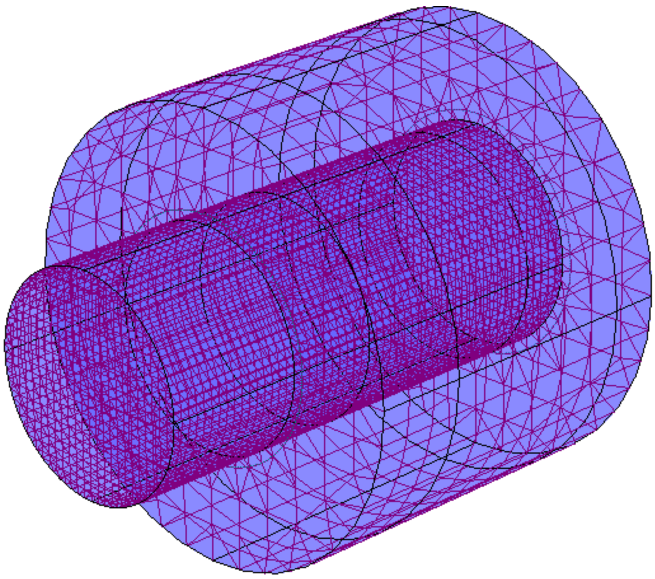


Figure 4.190: Refinement on a solid

Nodes / lines / surfaces / solids

In the input field of the dialog box, respectively the column of the table, enter the numbers of the objects to which you want to apply the refinement of the FE mesh. In the dialog box *New FE-Mesh Refinement*, you can use the [↖] function to select the objects graphically.

Parameters

Radius

When you select a circular refinement around a node, you have to specify the *Radius* of the refinement area. In addition, you have to define the *Target FE Length* in the center (*Inner*) and at the periphery (*Outer*) of the circle. If the FE length on the boundary of the refinement area corresponds to the global mesh size, RFEM refines the mesh gradually from outside to inside.

If there is a major difference between inner and outer FE length, specify a broader radius. In this way, you can avoid to generate acute-angled triangular elements within the refinement area.

Side length

When you select a rectangular refinement around a node, you have to define the area of refinement by its side length. In addition, you have to specify the *Target FE length* in the center (*Inner*).

Number of division nodes

When you select a line refinement by division, you have to define the number of division nodes. Then, the defined number of equally spaced FE nodes will be generated on the line.

Number of rows

When you select a gradual line refinement, you have to define the number of rows *n*. Then, RFEM divides the finite elements of the surface which are directly adjoining the line into the corresponding row number. Thus, a refinement in direction of the line is generated.

FE length for line / surface / solid

If the refinement has been selected on a line, surface or solid, the target FE length for the corresponding objects must be entered.

5. Load Cases and Combinations

Loads acting on the model are managed in different load cases. It is possible to superimpose these load cases, either manually or automatically, in load and result combinations (see chapter 12.2.1, page 556).

Before you can define loads (see chapter 6), you have to create a load case.

5.1 Load Cases

General description

The loads from a particular action are stored in a load case (**LC**). Load cases are for example: self- weight, snow or live load.

The loads in the load case should be defined as characteristic actions, that means **without factors**. The partial safety factors can be considered later when the load cases are superimposed in load or result combinations.

For each load case you can separately define which calculation method (linear static, second-order or large deformation analysis), approach and calculation parameters (load increment factor, stiffness reduction by partial safety factor of material) you want to use.

Create a new load case

There are several possibilities to open the loading dialog box for creating a new load case:

- On the **Insert** menu, point to **Loads** and select **New Load Case**.
- Use the toolbar button [New Load Case] shown on the left.

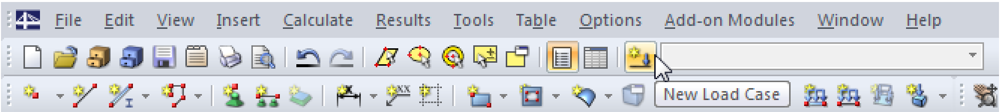


Figure 5.1: Button *New Load Case* in the toolbar

- Use the context menu of the navigator entry *Load Cases*.

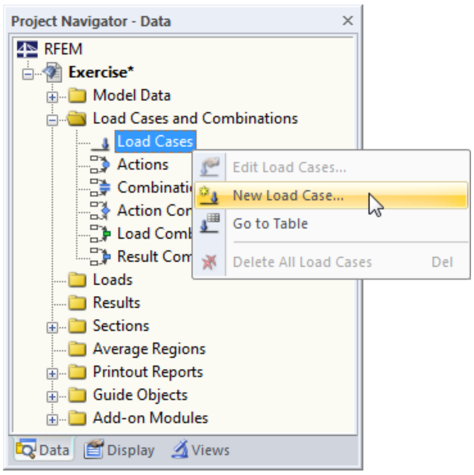


Figure 5.2: Context menu of *Load Cases* in the *Data* navigator

The dialog box *Edit Load Cases and Combinations* appears. A new load case is preset in the dialog tab *Load Cases*.

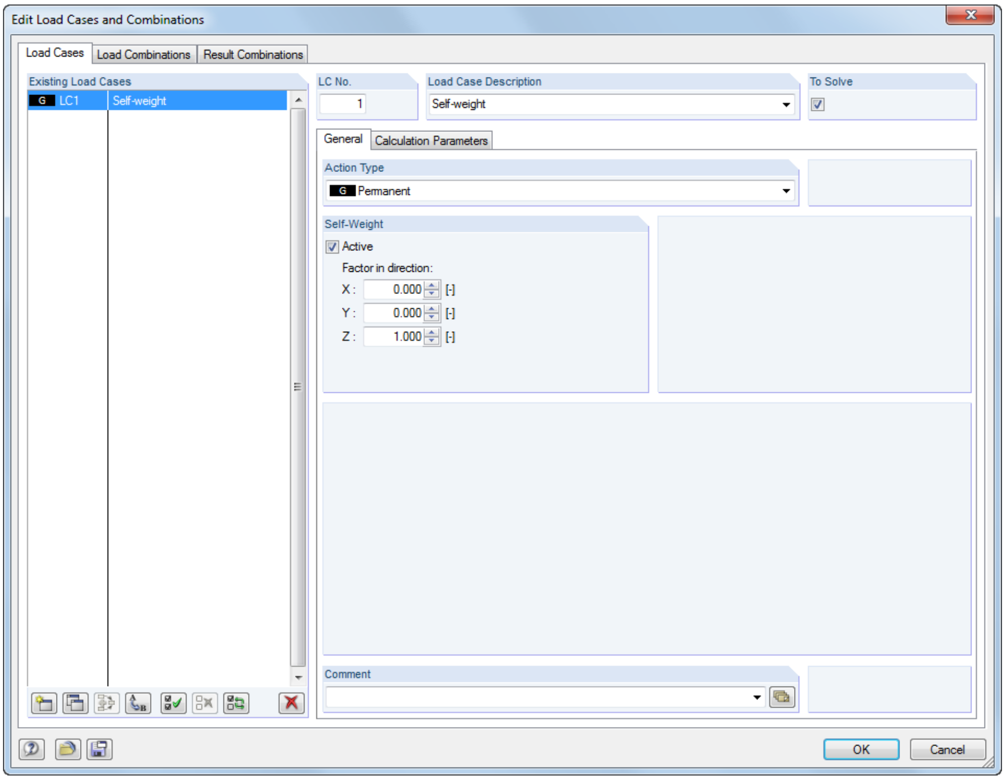
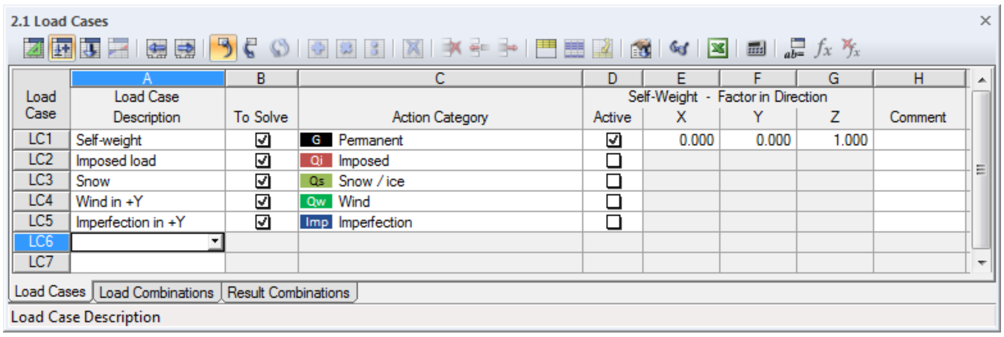


Figure 5.3: Dialog box *Edit Load Cases and Combinations*, tabs *Load Cases*

- It is also possible to enter a new load case in an empty row of table 2.1 *Load Cases*.



2.1 Load Cases

Load Case	A Load Case Description	B To Solve	C Action Category	D Active	E Self-Weight - Factor in Direction X	F Y	G Z	H Comment
LC1	Self-weight	☑	G Permanent	☑	0.000	0.000	1.000	
LC2	Imposed load	☑	Qi Imposed	☐				
LC3	Snow	☑	Qs Snow / ice	☐				
LC4	Wind in +Y	☑	Qw Wind	☐				
LC5	Imperfection in +Y	☑	Imp Imperfection	☐				
LC6								
LC7								

Load Cases | Load Combinations | Result Combinations

Load Case Description

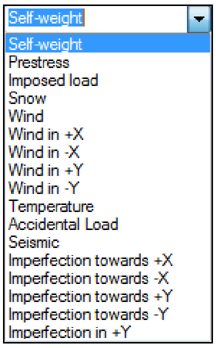
Figure 5.4: Table 2.1 *Load Cases*

Load case

The number of the new load case is preset but can be modified in the dialog input field *LC No*. If the entered number has already been assigned, RFEM displays a warning when closing the dialog box.

The creation of load cases should be well organized. Gaps in the numbering are allowed so that you can insert additional load cases later. The order of load cases can be changed subsequently by means of the [Renumber] dialog button (see Table 5.1 and chapter 11.4.18, page 479).

Load case description



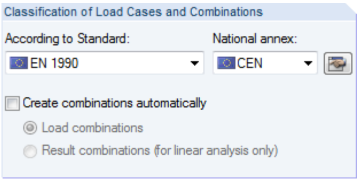
You can enter any name manually. You can also choose a name from the list to describe the load case shortly.

To solve

Use the check box to decide if the load case is considered as independent load case in the calculation. In this way, it is possible to exclude load cases from the calculation which do not occur in isolation (for example wind without considering self-weight) or whose results are not relevant for a preliminary design.

Action type

Standards mention different action categories controlling the superposition of load cases as well as the partial safety factors and combination coefficients. Each load case must be assigned to a category.



Standard settings in dialog box *Model - General Data*

The list of the dialog box and table provides several categories for selection. They depend on the standard that is set in the dialog box *Model - General Data* (see chapter 12.2.1, page 556).

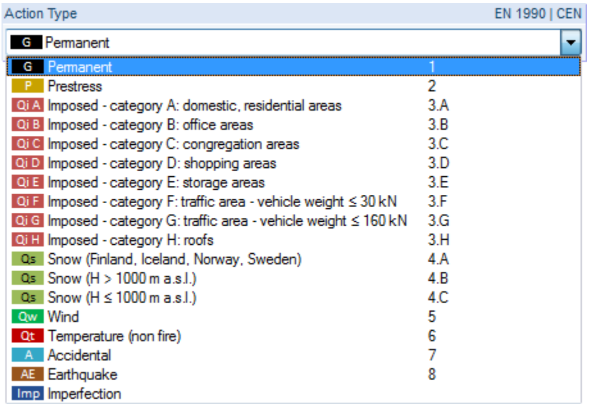


Figure 5.5: Action categories according to EN 1990

These categories are significant for combining load cases manually or automatically. The classification of the load case determines which factors are applied when creating load and result combinations.

Self-weight

When you want to take into account the construction's self-weight as load, tick the *Active* check box. The load's direction of action can be defined in one of the three input fields by means of the self-weight factor. The default setting is 1.00 in direction Z, respectively -1.00 if the global axis Z points upwards.

When the automatic self-weight is applied in several load cases, you have to consider this for the combination of load cases.

Comment

Enter a user-defined note or select an entry from the list to describe the load case in detail.

Calculation parameters

The tab *Calculation Parameters* in the loading dialog box offers different options for controlling the calculation. Find a detailed description of these parameters in chapter 7.3.1 on page 263.

Edit general data of a load case

There are several possibilities to change the general data of an existing load case:

- On the **Edit** menu, point to **Loads**, and then select **Load Case - General Data** (current load case).
- On the **Edit** menu, point to **Loads**, and then select **Load Cases** (selection from all load cases).
- In the *Data* navigator, right-click a load case to open its context menu, or double-click the load case itself.

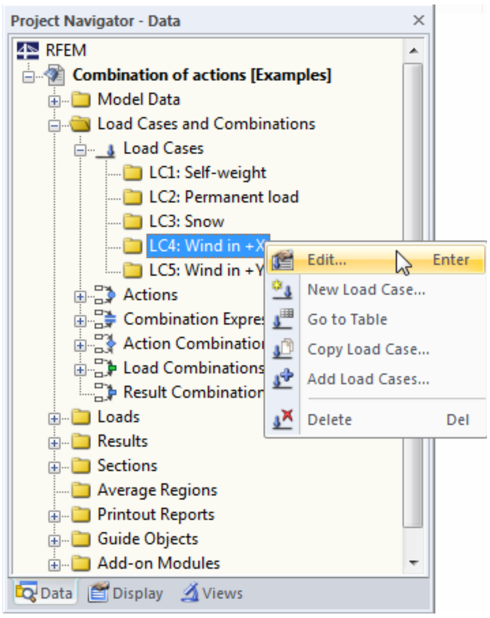


Figure 5.6: Context menu of a load case

- Use the button [Edit load cases] in the toolbar of the loads tables (current load case).

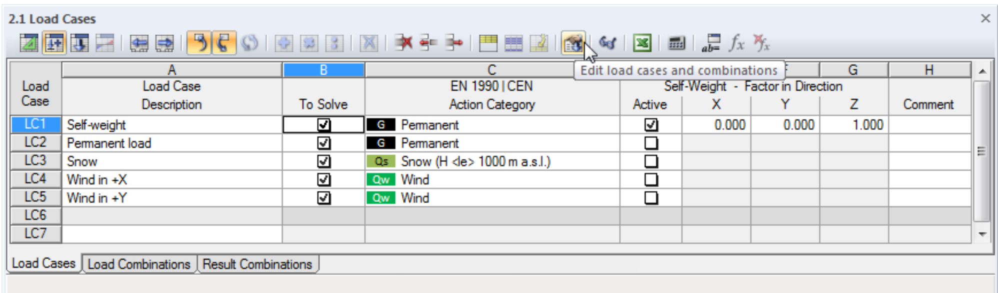
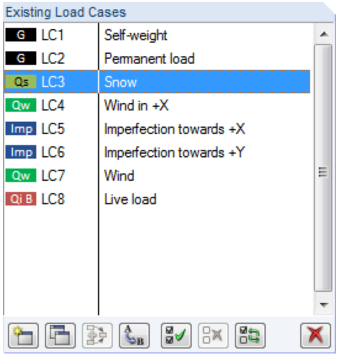


Figure 5.7: Button [Edit load cases and combinations] in the toolbar of the loads tables



Buttons

In the dialog box *Edit Load Cases and Combinations*, several buttons are available below the load case list (see Figure 5.3, page 170). The buttons have the following functions:

	Creates a new load case
	Creates a new load case as copy of the selected load case (see below)
	If several load cases are selected, all contained loads are copied to a new load case (see below).
	Assigns a new number to the selected load case. Specify the number in a separate dialog box. It is not allowed to enter a number that has already been assigned.
	Selects all load cases
	Cancels the selection in the list
	Inverts the selection of load cases
	Deletes the selected load case

Table 5.1: Buttons in the tab *Load Cases*

Copy and add load cases

You can use already existing load cases to create new load cases.

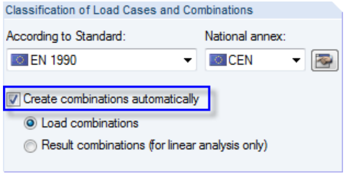
To **copy** a load case, select the relevant load case in the list *Existing Load Cases*. By clicking the [Copy] button you create a copy of the load case with the next available free number. Then, you can adjust the description of the new load case and the loads.

When **adding** load cases RFEM copies the loads of several load cases into a new load case. First, select the relevant load cases in the list *Existing Load Cases* (multiple selection by holding down the [Ctrl] key). Use the [Add] button to copy the loads into a new load case.

5.2 Actions

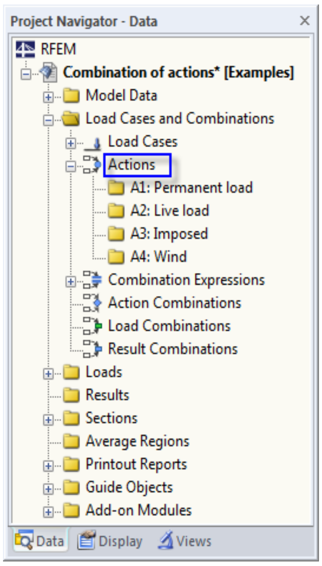
General description

When using the latest standards, for example EN 1990 or DIN 1055-100 (Germany), it is often time-consuming to take into account all load situations coming into question and to select the decisive situations for the designs. In the dialog box *Model - General Data*, you have the possibility to create combinations automatically (see Figure 12.23, page 554).



Check box in dialog box *Model - General Data*

The load cases defined in table 2.1 (see previous chapter 5.1) represent the base data for the automatic superpositioning. RFEM distinguishes between two load case categories: standard load cases and load cases of the type *Imperfection*. Moreover, for combining load cases it is important to know in which action category the standard load cases have been organized.



Standards provide rules for the combination of independent actions in various design situations. Actions are independent of each other if they arise out of different origins and if the correlation existing between them may be neglected with regard to the reliability of the structural system.

In accordance with this concept, *Actions* to which load cases are assigned must be defined for the automatic superposition in RFEM. The action type defined for the load cases (see chapter 5.1, page 171) controls the assignment to action categories conforming to standards.

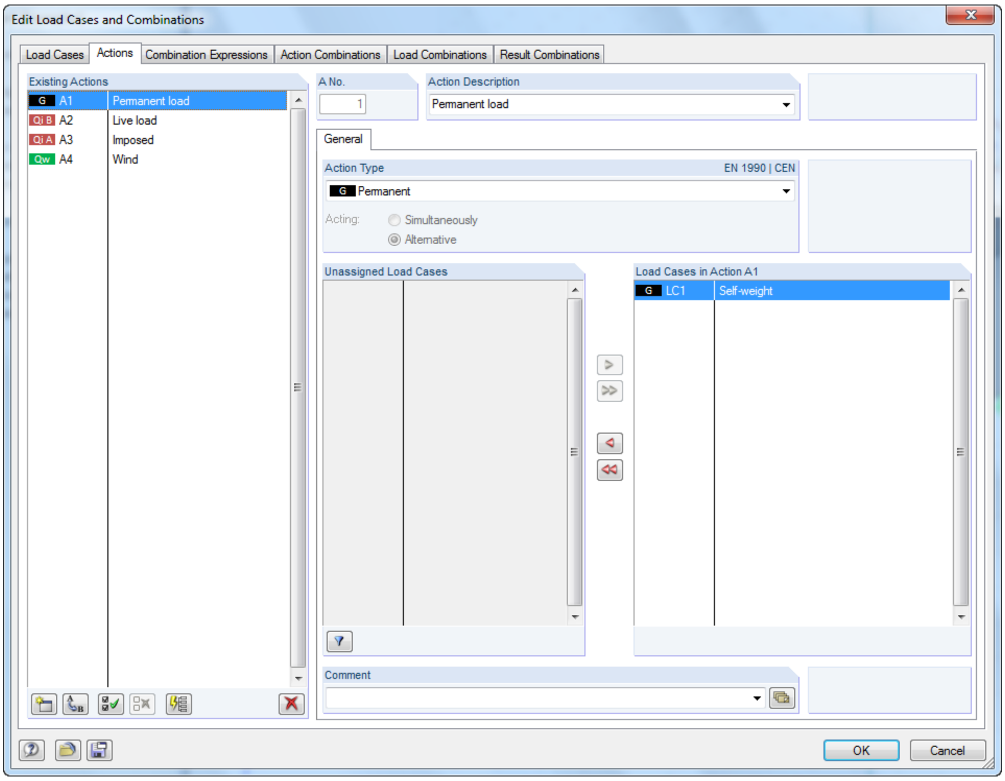


Figure 5.8: Dialog box *Edit Load Cases and Combinations*, tabs *Actions*

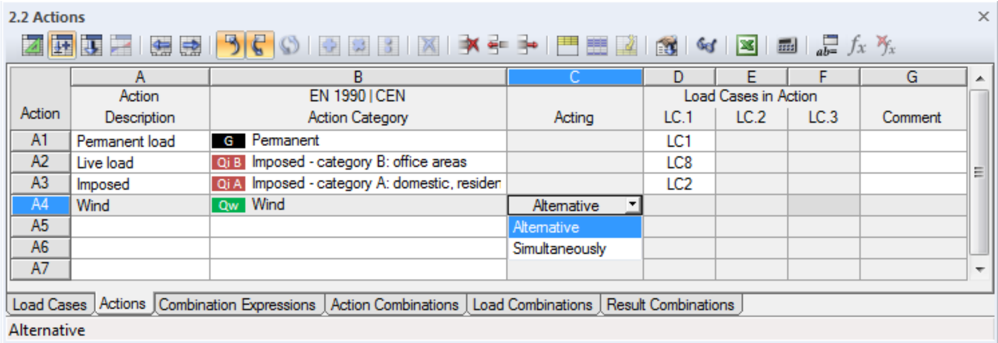


Figure 5.9: Table 2.2 *Actions*

Action

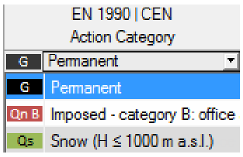
Actions are created already when defining load cases. They are consecutively numbered. The sequence is not important but can be modified, if necessary, by means of the [Renumber] button available in the dialog box.

In the table you can add actions manually for example to assign load cases by user-defined specifications when huge models are designed.

Action description

The description of the action is derived from the action type that has been selected for the load cases. The preset description can be changed, if necessary.

Action category



Standards mention different action categories controlling the partial safety factors and combination coefficients (see chapter 5.1, page 171).

The list of the dialog box and table provides only the categories which have been used for the definition of the single load cases. Therefore, in order to create a new category, a new action type must be assigned in the general data of a load case.

Acting

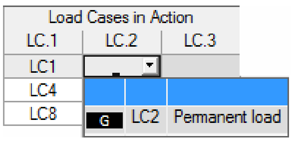
Two or more load cases can be defined as *Simultaneously* or *Alternative* acting. That means that these load cases occur either <u>always</u> or <u>never</u> together in a load or result combination.

For example, load cases with wind from different directions are "alternative" acting.

Load cases in action

Load cases are assigned according to the specifications of the LC action type, so assignment is largely automatic.

To remove a load case from an action, select the load case in the dialog section *Load Cases in Action*. Use the [◀] button or double-click the entry to transfer it to the dialog section *Unassigned Load Cases*. In the table, it is also possible to set a load case inactive: Select the empty entry in the list of the relevant table cell.



Manually removed load cases, taking into account the action type, are transferred to the list *Unassigned Load Cases*. This also means that only load cases of the same action type can be included in an action category. It is not possible for example to select load cases of the type "live loads" for actions of "snow", neither in the dialog box nor in the list of the table (see picture on the left). Therefore, type-different load cases are not visible in the list *Existing Actions*. Use the button [Show Unused] below the dialog section to display load cases of other categories. They are shown as locked and cannot be selected.

Load cases that not assigned to any action will not be taken into account when generating combinations.

Comment

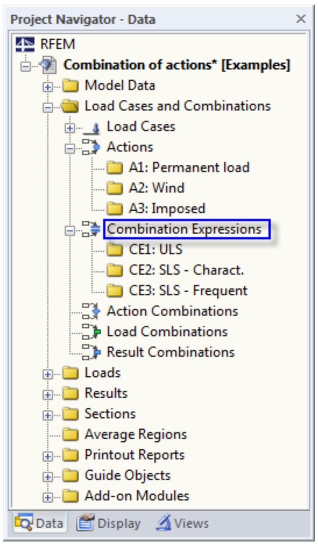
Enter a user-defined note or select an entry from the list.

The buttons in the *Actions* tab of the dialog box *Edit Load Cases and Combinations* have the following functions:

	Creates a new action
	Renumbers the selected actions
	Selects all actions
	Cancels the selection in the list
	Assigns unassigned load cases to actions automatically
	Deletes the selected actions

Table 5.2: Buttons in the tab *Actions*

5.3 Combination Expressions



General description

Standards describe how to combine actions. For example, EN 1990 requires the design of the ultimate and the serviceability limit states. Ultimate limit states for **load bearing capacity** have to be designed in four design situations for which particular combination rules must be applied:

1. Permanent situations involving common conditions of use of a structural system as well as temporary situations referring to time-limited stages of the structure (for example construction stage, repairs)

   As combination rule for permanent and temporary situations (basic combination) you have to apply either

   $$\sum_{j\geq 1} \gamma_{G,j} \cdot G_{k,j} + \gamma_P \cdot P_k + \gamma_{Q,1} \cdot Q_{k,1} + \sum_{i>1} \gamma_{Q,i} \cdot \psi_{0,i} \cdot Q_{k,i}$$

   Equation 5.1

   or the more unfavorable combination with Equation 5.2 and Equation 5.3 for the limit states STR and GEO.

   $$\sum_{j\geq 1} \gamma_{G,j} \cdot G_{k,j} + \gamma_P \cdot P_k + \gamma_{Q,1} \cdot \psi_{0,1} \cdot Q_{k,1} + \sum_{i>1} \gamma_{Q,i} \cdot \psi_{0,i} \cdot Q_{k,i}$$

   Equation 5.2

   $$\sum_{j\geq 1} \xi_j \cdot \gamma_{G,j} \cdot G_{k,j} + \gamma_P \cdot P_k + \gamma_{Q,1} \cdot Q_{k,1} + \sum_{i>1} \gamma_{Q,i} \cdot \psi_{0,i} \cdot Q_{k,i}$$

   Equation 5.3

2. Extraordinary situations referring to extraordinary actions of the structural system or its environment (for example fire, explosion, collision)

   $$\sum_{j\geq 1} G_{k,j} + P + A_d + (\psi_{1,1} \text{ oder } \psi_{2,1}) \cdot Q_{k,1} + \sum_{i>1} \psi_{2,i} \cdot Q_{k,i}$$

   Equation 5.4

3. Situations in case of earthquakes

   $$\sum_{j\geq 1} G_{k,j} + P_k + A_{Ed} + \sum_{i\geq 1} \psi_{2,i} \cdot Q_{k,i}$$

   Equation 5.5

According to EN 1990, you have to design **serviceability** limit states in three design situations for which the following combination rules must be applied.

1. Characteristic situations with irreversible (lasting) effects on the structural system

$$\sum_{j\geq 1} G_{k,j} + P_k + Q_{k,1} + \sum_{i>1} \psi_{0,i} \cdot Q_{k,i}$$

Equation 5.6

2. Frequent situations with reversible (non-lasting) effects on the structural system

$$\sum_{j\geq 1} G_{k,j} + P_k + \psi_{1,1} \cdot Q_{k,1} + \sum_{i>1} \psi_{2,i} \cdot Q_{k,i}$$

Equation 5.7

3. Quasi-permanent situations with long-term effects on the structural system

$$\sum_{j\geq 1} G_{k,j} + P_k + \sum_{i\geq 1} \psi_{2,i} \cdot Q_{k,i}$$

Equation 5.8

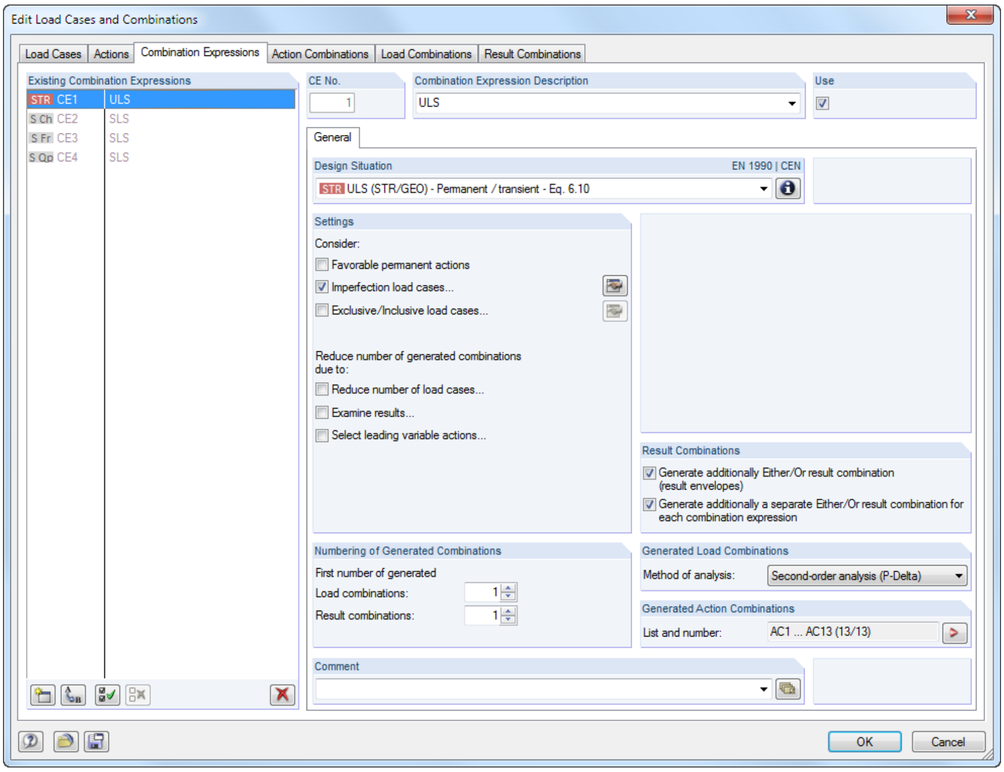
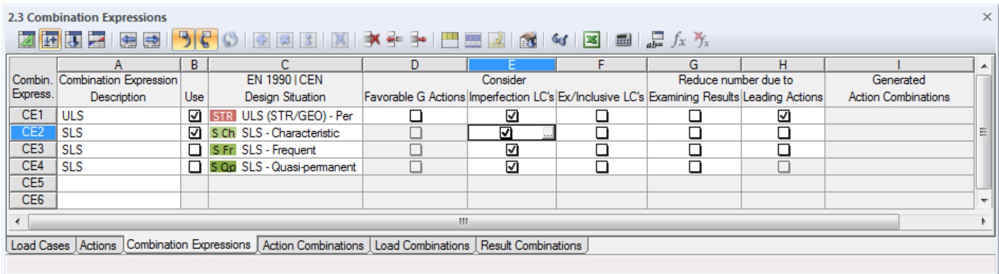


Figure 5.10: Dialog box *Edit Load Cases and Combinations*, tab *Combination Expressions*



2.3 Combination Expressions

Combin. Express.	A Combination Expression Description	B Use	C EN 1990 | CEN Design Situation	D Consider Favorable G Actions	E Consider Imperfection LC's	F Consider Ex/Inclusive LC's	G Reduce number due to Examining Results	H Reduce number due to Leading Actions	I Generated Action Combinations
CE1	ULS	☑	STR ULS (STR/GEO) - Per	☐	☑	☐	☐	☑	
CE2	SLS	☑	S Ch SLS - Characteristic	☐	☑	☐	☐	☐	
CE3	SLS	☐	S Fr SLS - Frequent	☐	☑	☐	☐	☐	
CE4	SLS	☐	S Qp SLS - Quasi-permanent	☐	☑	☐	☐	☐	
CE5									
CE6									

Load Cases | Actions | Combination Expressions | Action Combinations | Load Combinations | Result Combinations

Figure 5.11: Table 2.3 *Combination Expressions*

Combination expression

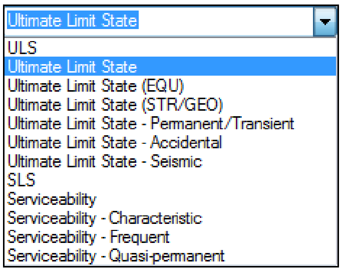
When you access the dialog box or table, RFEM presets the combination rules of the following design situations:

- *STR (ULS)* ultimate limit state for permanent or temporary situation
- *S Ch (SLS)* serviceability limit state for characteristic situation
- *S Fr (SLS)* serviceability limit state for frequent situation
- *S Qp (SLS)* serviceability limit state for quasi-permanent situation

You can create a new combination rule in another table row or in the dialog box by using the [New] button. The design situations described below are available for selection.

Combination rules marked in the dialog list can be deleted with the [Delete] button.

Description of combination expression



The brief description of combination rules can be changed subsequently. The list provides some suggestions for selection.

Use

Use the check box to decide if the selected combination rule is considered when creating result combinations. In this way, it is possible to reactive or exclude design situations from the generation.

Design situation

Standards describe the situations for which designs of structural systems must be performed. These design situations determine the conditions expected during the construction and use of the building.

The following design situations for EN 1990 are available for selection in the list:

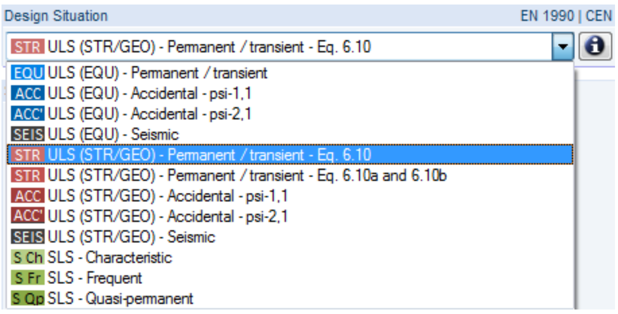


Table 5.12: Design situations according to EN 1990

For the standards DIN 1055-100, DIN EN 1990 and EN 1990 + DIN EN 1995, RFEM offers additionally the design situations *Accidental - Snow* where the factors for the North German Plain are taken into account.

Use the [Info] button to check the combination rule of the current design situation. A dialog box opens explaining the equation with relevant parameters (see figure below).

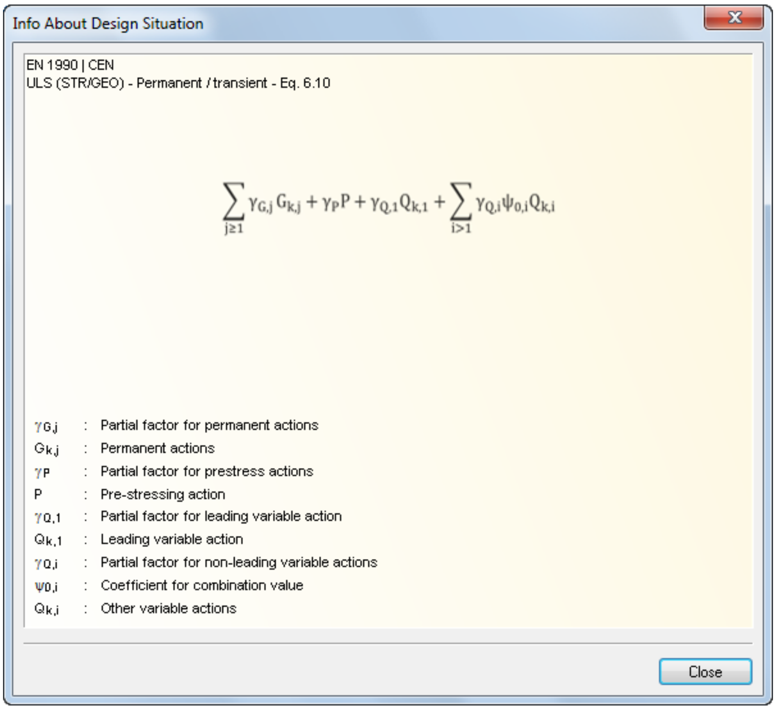
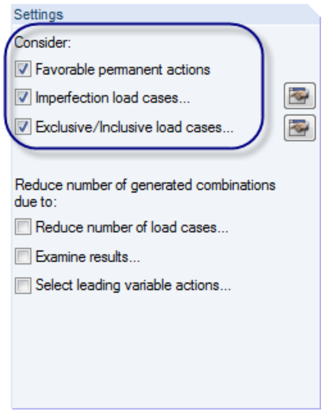


Figure 5.13: Dialog box *Info About Design Situation*



Favorable permanent actions

Due to this option, RFEM can distinguish between favorable and unfavorable acting permanent actions during the generation. They are considered with different partial safety factors in the superpositioning. Additional combinations are generated.

Settings of the check box affect only design situations for load bearing capacity. The distinction between favorable and unfavorable acting permanent actions is done automatically for the design situation "static equilibrium", whereas permanent actions for the design situation "serviceability" are not differentiated.

Imperfection load cases

RFEM distinguishes between two load case categories: standard load cases and load cases of the type *Imperfection*. Due to the special treatment of imperfections, it is possible to form any possible load combination once with imperfection and once without.

Imperfection load cases are taken into account only for generating load combinations. Moreover, settings of the check boxes are globally valid: Imperfections can be considered either consistently for all combination rules or not at all. It is not possible to apply imperfections separately for individual combination expressions.

When the check box is ticked, the [Settings] button or the button [...] is enabled. Use the buttons to access a dialog box with specific settings for imperfection load cases.

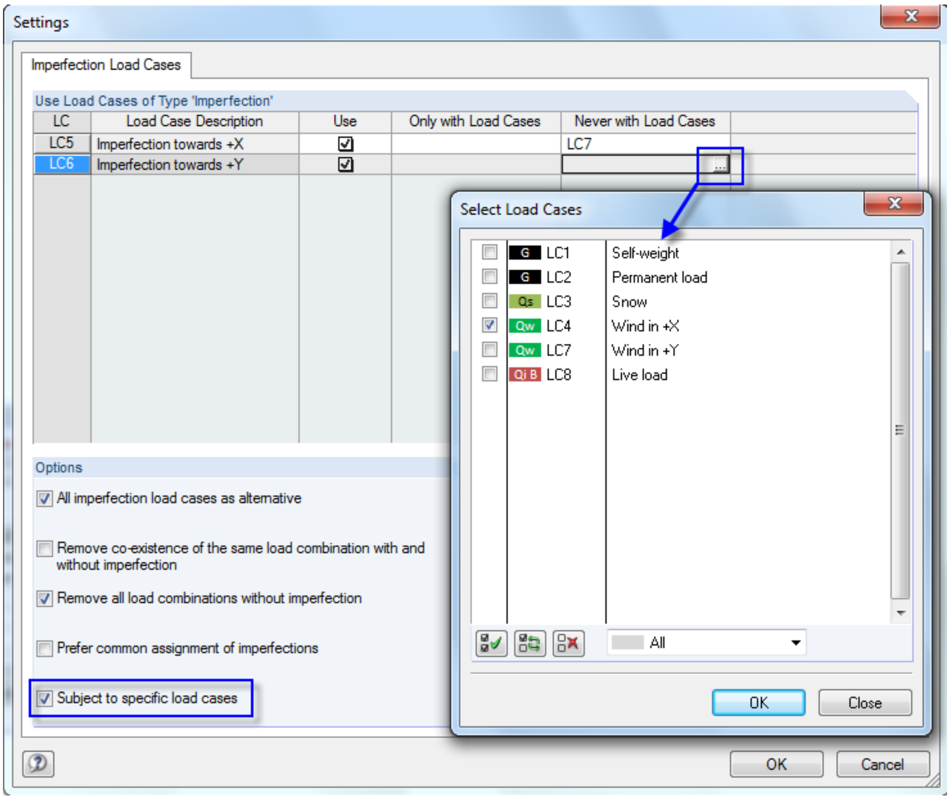


Figure 5.14: Dialog box *Settings* with dialog box *Select Load Cases* for selecting load cases

The dialog section *Use Load Cases of Type 'Imperfection'* lists all load cases that have been classified as action type "imperfection" (see chapter 5.1, page 171). Use the check boxes in the *Use* column to control the load cases in detail and to decide which one is included in the generation of load combinations.

The columns *Only with Load Cases* and *Never with Load Cases* are shown if the imperfection load cases are *Subject to specific load cases* (see description below).

With settings in the dialog section *Options* you determine how imperfection load cases are taken into account. When *All imperfection load cases* act *as alternative*, RFEM applies only one imperfection load case for each load combination.

If at least one imperfection load case is activated, any possible load combination will be created once with imperfection and once without. In case you want to create only load combinations with imperfection, tick the check box for *Remove co-existence of the same load combination with and without imperfection*.

With the option *Subject to specific load cases* you can further reduce the number of generated load combinations. If the option is ticked, the two additional columns *Only with Load Cases* and *Never with Load Cases* are shown in the dialog section above. Click into a cell to enable the [...] button that you can use to access the dialog box *Select Load Cases* where you can define a relation between the imperfection load case and one or more belonging respectively alternative load cases (see Figure 5.14).

Exclusive/inclusive load cases

To further reduce the number of created load combinations, it is possible to classify load cases to be mutually exclusive or occurring only together.

Ticking the check box enables the dialog button [Settings] or the table button [...] that you can use to open a dialog box with detailed settings for the application of load cases.

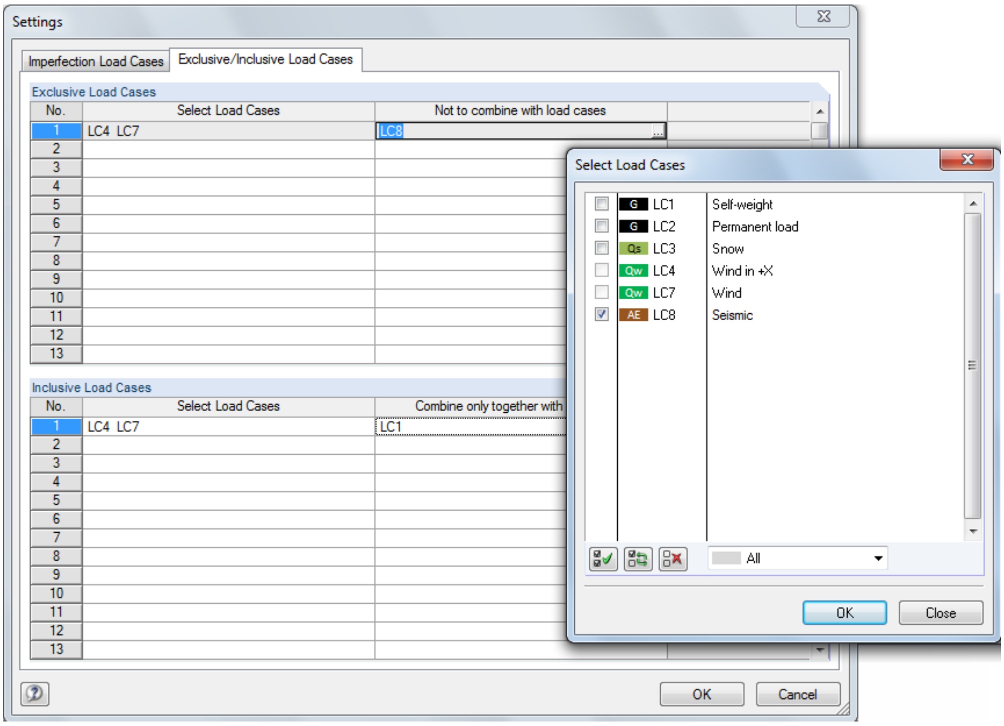
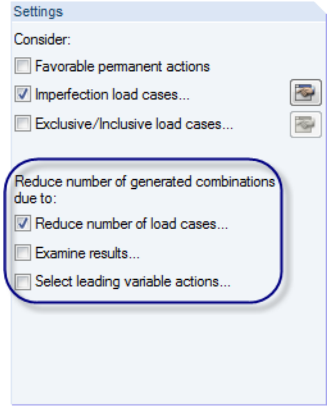


Figure 5.15: Dialog box *Settings*, tab *Exclusive/Inclusive Load Cases* with dialog box *Select Load Cases*

First, enter a load case into the column *Select Load Cases* in the dialog section *Exclusive Load Cases*. You can also use the [...] button to open the dialog box *Select Load Cases* where you can select the case. Then, in the column *Not to combine with load cases*, define which load case(s) you never want to take into account together with the case in the load combination. In this way, it is possible to avoid for example the combination of snow with human load cases.

In the dialog section *Inclusive Load Cases*, you can specify settings analogously for load cases that you want to appear always together in each load combination. However, these relations are only effective if the option *Reduce number of generated combinations due to Examine results* (see below) is not activated.

Specifications in the dialog section *Inclusive Load Cases* are taken into account only for the generation of load combinations, not of result combinations.



Reduce number of generated combinations due to

The complexity of the structural system as well as the number of actions and load cases have a significant influence on the number of generated combinations. RFEM offers three possibilities for reducing the number of constellations with great effect. The first two procedures are only available for the generation of load combinations but not for result combinations. They are described in an example to be found on page 186.

Reduce number of load cases

With this option you can generally limit the number of load cases occurring in the load combinations. Access to the check box is available in the *General* tab of the *Combination Expressions* (see Figure 5.10, page 178). RFEM finds out which load cases provide positive respectively negative internal forces and deformations. Then, all positively acting and all negatively acting load cases are combined. Thus, combinations will take into account only those load cases which are relevant for the maximum or minimum values.

The advantage of this method is the possibility to reduce the number of combinations considerably, which has a favorable impact on the speed of calculation as well as the evaluation. A disadvantage may be the fact that there is a certain factor of uncertainty for the reduction to find the extreme values in case of unfavorable load arrangements and specifications.

When you tick the check box, an additional dialog tab appears called *Reduce - Number of Load Cases* where you can specify in detail which load cases, internal forces and objects you want to be considered for the creation of governing combinations.

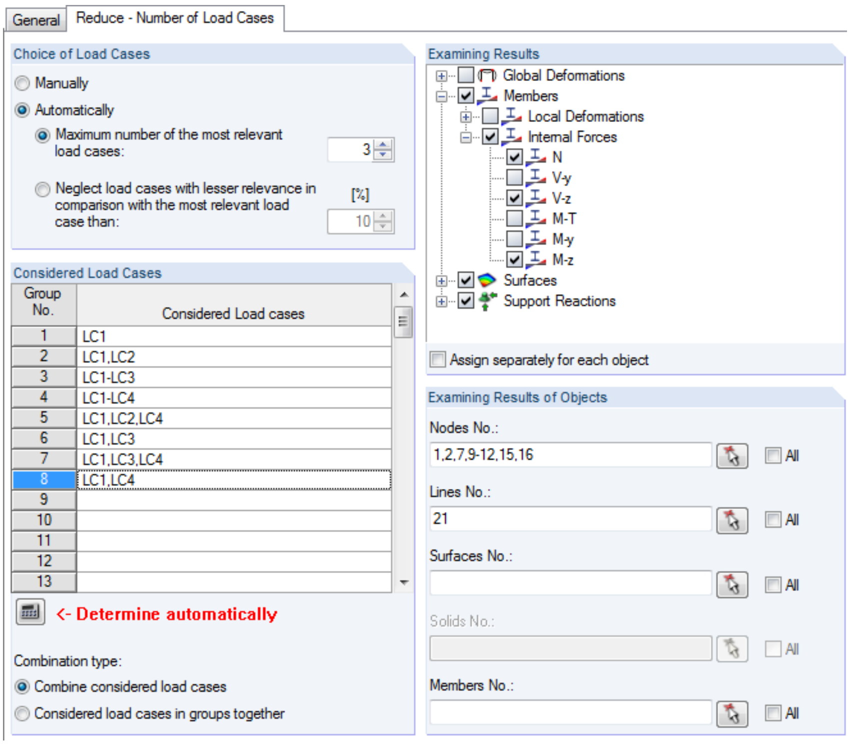


Figure 5.16: Dialog tab *Reduce - Number of Load Cases* for combination expressions

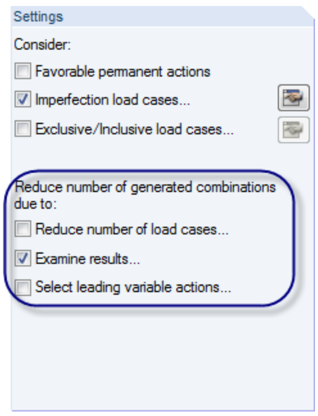
The load cases can be selected *Manually* or determined *Automatically* on the basis of relevance criteria. Clicking the button [Determine automatically] starts a calculation to determine the maximum and minimum internal forces, deformations and support reactions in the load cases.

When the automatic determination is selected, define which *Results* (deformations, internal forces of members and surfaces, support reactions) and *Objects* (nodes, surfaces, members etc.) you want to consider for the evaluation of the load cases. The relevant objects can be selected graphically with the [↖] function as soon as the check box *All* is clear. Above, you can use the check box *Assign separately for each object* to assign specific result types to objects for the analysis.

The number of load cases contained in a *Group* after calculating load case data depends on settings defined in the dialog section *Choice of Load Cases*:

- When the option **Maximum number of the most relevant load cases** is selected, a group provides either the specified maximum number of load cases or only positively respectively negatively acting load cases in a smaller number.
- It is possible to **Neglect load cases** which have only a very small share in the maximum and minimum values. The percentage refers to the internal forces, deformations and support forces of the load cases respectively providing the extreme values.

Imperfection load cases are not considered when the automatic creation of groups is set.



Examine results

RFEM creates only the governing load combinations (this option is not available for result combinations).

When ticking the check box, the new tab *Reduce - Examine Results* is added to the dialog box.

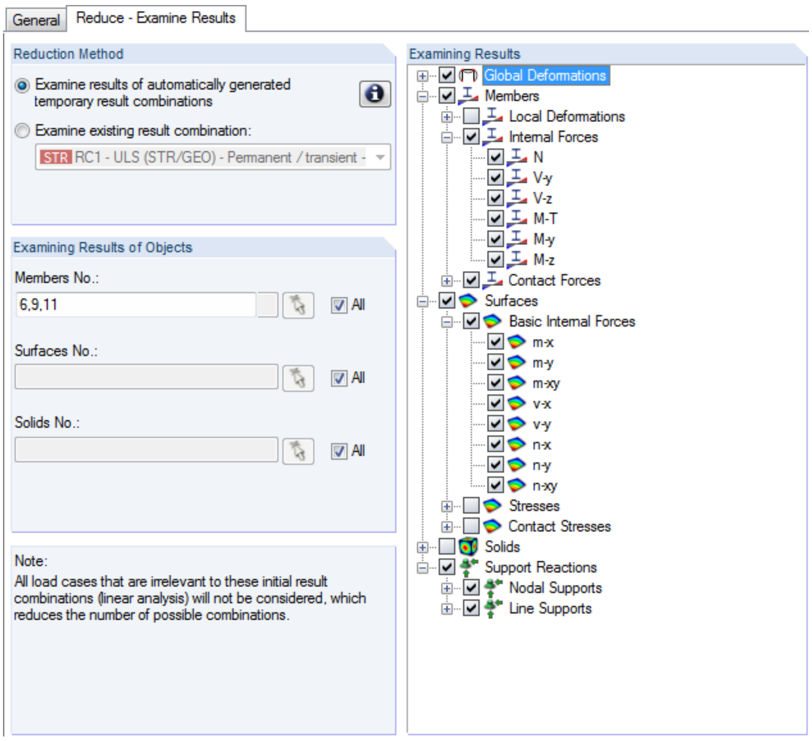


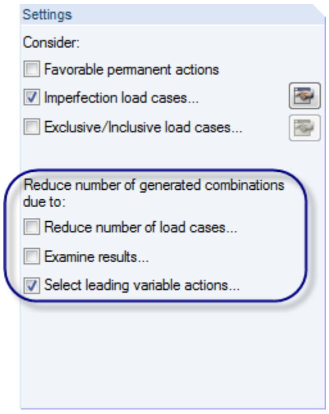
Figure 5.17: Tab *Reduce - Examine Results* for combination rules

With the first *Reduction Method* you can evaluate generated temporary result combinations automatically. Temporary result combinations include all load cases created in the model and consider all relations existing among them. By means of results available on each FE node, RFEM can analyze which of the simultaneously acting load cases produce a maximum or minimum on the corresponding locations. The reduction method is based on the assumption that only those combinations can be governing which contain exactly these simultaneously acting load cases.

Alternatively, it is possible to use the results of a user-defined result combination for the results reduction.

In the dialog section *Examining Results* to the right, you can define which deformations, internal forces, stresses or support reactions you want to take into account for the determination of extreme values.

The dialog section *Examining Results of Objects* provides options to restrict the extreme value analysis to results of selected members, surfaces and solids. You can use the [↖] function to select objects graphically.



Select leading variable actions

The third possibility to reduce the number of generated combinations is to classify only selected actions as leading actions. This option is available for the generation of load and result combinations.

When ticking the check box, the new tab *Reduce - Leading Variable Actions* is added to the dialog box.

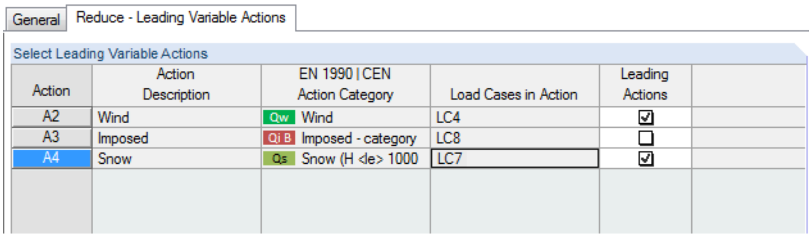


Figure 5.18: Tab *Reduce - Leading Variable Actions* for combination rules

The list of leading actions contains only variable actions.

When you remove the check mark of an entry in the column *Leading Actions*, the corresponding action will be superimposed only as accompanying variable action.

Numbering of generated combinations

Entering data in this section of the dialog box *Edit Load Cases and Combinations* (see Figure 5.10, page 178) affects the *First number of generated Load combinations* or *Result combinations* that are created in RFEM.

Result combinations

Optionally, you can *Generate additionally* an *Either/Or result combination (results envelopes)*. This result combination superimposes the extreme values of all load or result combinations according to the following scheme:

"CO1/permanent or CO2/permanent or CO3/permanent etc."

If several combination expressions are specified for the generation, it is possible to *Generate additionally a separate Either/Or result combination for each combination expression*.

Method of analysis

Use the list to decide which method of calculation you want to apply to analyze combinations (see chapter 7.3.1.1, page 263). RFEM presets the nonlinear calculation according to second-order analysis (P-Delta) for load combinations.

Generated action combinations

The dialog section, respectively table column, is filled during the generation starting automatically when closing the dialog tab or table. The dialog field shows you a short overview of the number of generated combinations.

With the data entered in the dialog box or table, RFEM creates so-called "action combinations" (AC). They are described in the following chapter. You can use the entries shown in the current dialog box to estimate the way how combination rules affect the number of combinations.

Generated Action Combinations
AC1 ... AC13 (13/47)
AC14 ... AC26 (13/47)
AC27 ... AC39 (13/47)
AC40 ... AC47 (8/47)

In the example shown on the left, a total of 47 action combinations is generated for the four specified design situations:

- ULS (STR/GEO): AC1 to AC13
- SLS - characteristic: AC14 to AC26
- SLS - frequent: AC27 to AC39
- SLS - quasi-permanent: AC40 to AC47

When jumping to the next tab with the dialog button [▶], RFEM determines the action combinations automatically. The first action combination created with the current combination expression is selected in the subsequent dialog tab.

Comment

Enter a user-defined note or select an entry from the list.

Example: Reducing generated combinations

The target of combining actions is finding the most unfavorable load arrangement for each location in the structural system. To reach it, you can

- either determine all combinations that are mathematically possible
- or try to find logical relationships before combining the actions to reduce the number of possible combinations.

For example, a symmetrical two-hinged frame has the following load cases:

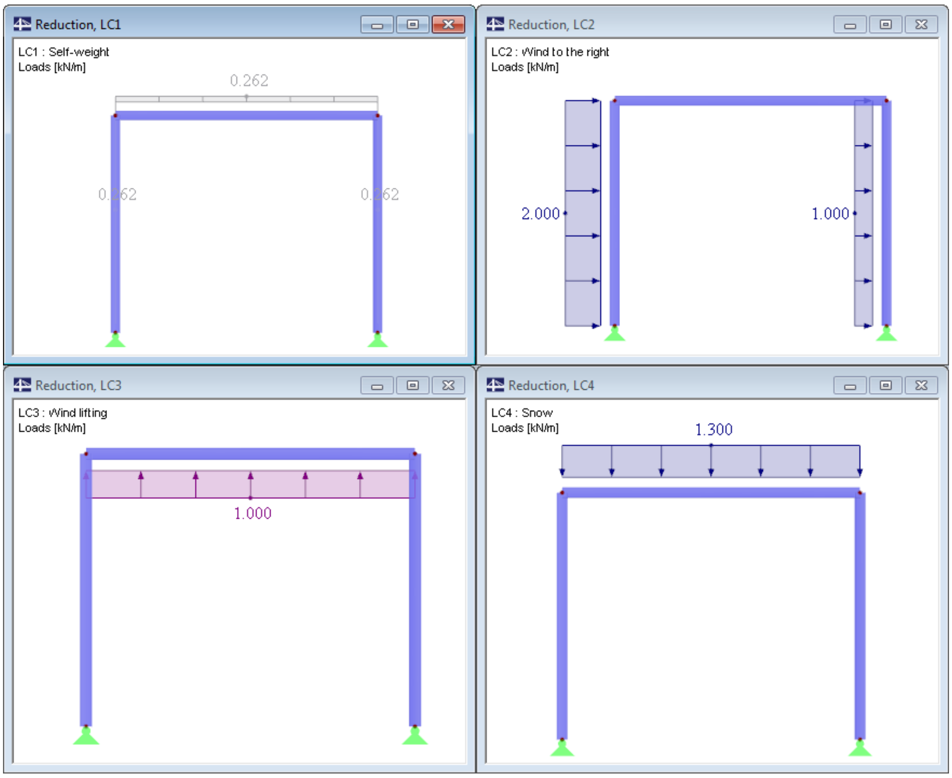
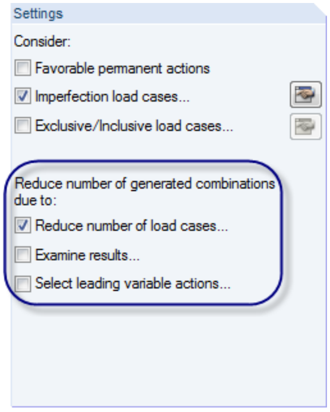


Figure 5.19: Two-hinged frame with four load cases



- ## Option *Reduce number of load cases*

The load cases result in the following axial forces available in the columns:

Load case	Description	Effect	Axial force left column	Axial force right column
1	self-weight	permanent	compression	compression
2	wind to the right	alternative	tension	compression
3	wind lifting	alternative	tension	tension
4	snow	alternative	compression	compression

Table 5.3: Tensile and compressive forces of columns

Because LC1 is always acting and the remaining load cases occur alternatively, eight combinations are theoretically possible:

CO1: LC1 + LC2 + LC3 + LC4
CO2: LC1
CO3: LC1 + LC2
CO4: LC1 + LC3
CO5: LC1 + LC4
CO6: LC1 + LC2 + LC3
CO7: LC1 + LC3 + LC4
CO8: LC1 + LC2 + LC4

These eight combinations can be reduced if we want to find for example only the arrangements with the extreme values of the columns' axial forces. It is possible to create a group of load cases for each column providing only tensile and compressive forces, considering the permanently acting load case 1.

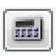
Group	Left column	Right column
tension	LC1, LC2, LC3	LC1, LC3
compression	LC1, LC4	LC1, LC2, LC4

Table 5.4: Groups of load cases

Thus, the result is no longer eight but only four combinations of load cases.

This reduction can be done in the dialog box *Edit Load Cases and Combinations* (see Figure 5.10, page 178) by

- ticking the check box *Reduce number of load cases*,
- ticking only the axial forces in the dialog section *Examining Results* in the tab *Reduce - Number of Load Cases* and
- entering only the numbers of the column members in the dialog section *Examining Results of Objects* (see figure on the following page).



After clicking the button [Determine automatically] RFEM performs a short calculation. Then, the table in the dialog section *Considered Load Cases* lists the four groups of load cases that are are also shown in Table 5.4.

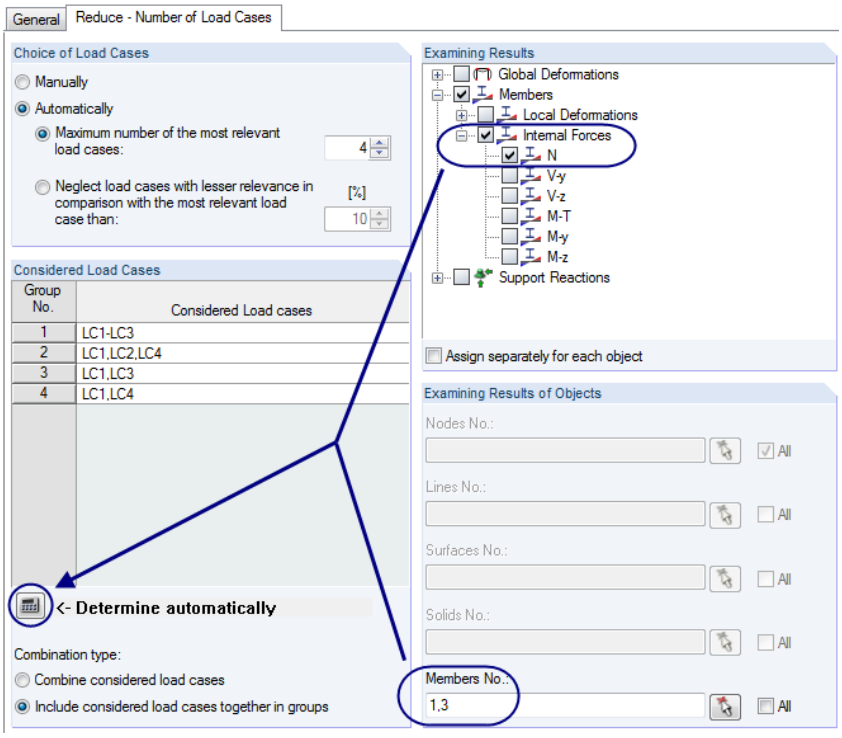
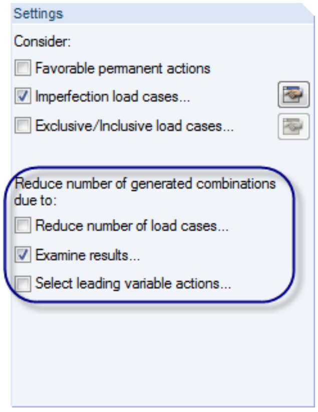


Figure 5.20: Automatic determination of load cases in groups



- ## Option *Examine results*

With this method a linear result combination is created from the load cases. For each point RFEM evaluates the extreme values and the involved load cases so that a max- and a min-combination of load cases is respectively available. Then, these extreme combinations are used for creating the load case combinations.

The load cases result in the following axial forces occurring in the columns:

Load case	**Description**	**Effect**	**Axial force left column**	**Axial force right column**
1	self-weight	permanent	compression (-10 kN)	compression (-10 kN)
2	wind to the right	alternative	tension (5 kN)	compression (-5 kN)
3	wind lifting	alternative	tension (3 kN)	tension (3 kN)
4	snow	alternative	compression (-12 kN)	compression (-12 kN)

Table 5.5: Tensile and compressive forces of columns

RFEM creates this temporary result combination: LC1/permanent + LC2 + LC3 + LC4

For the axial forces of the two columns we get the following extreme value results when superimposing:

Group	**Column left**	**Column right**
Maximum N	-2 kN (LC1, LC2, LC3)	-7 kN (LC1, LC3)
Minimum N	-22 kN (LC1, LC4)	-27 kN (LC1, LC2, LC4)

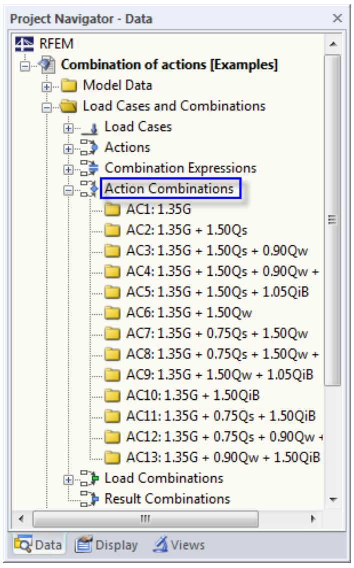
Table 5.6: Groups of load cases

Again, the result is no longer eight but only four combinations of load cases.

Setting specifications in the tab *Reduce - Examine Results* is analogous to Figure 5.20.

5.4 Action Combinations

General description



When you open the dialog tab or table 2.4, actions are superimposed automatically according to the combination rules and identified as so-called "action combinations". This overview is sorted by actions, and thus corresponds to the way how actions are described in standards. Now, you can define which action combinations will finally come into question for the generation of load or result combinations.

An action combination includes all possibilities how load cases can be combined in the action. Therefore, do not confuse it with a load or result combination representing only a single variant of these possibilities.

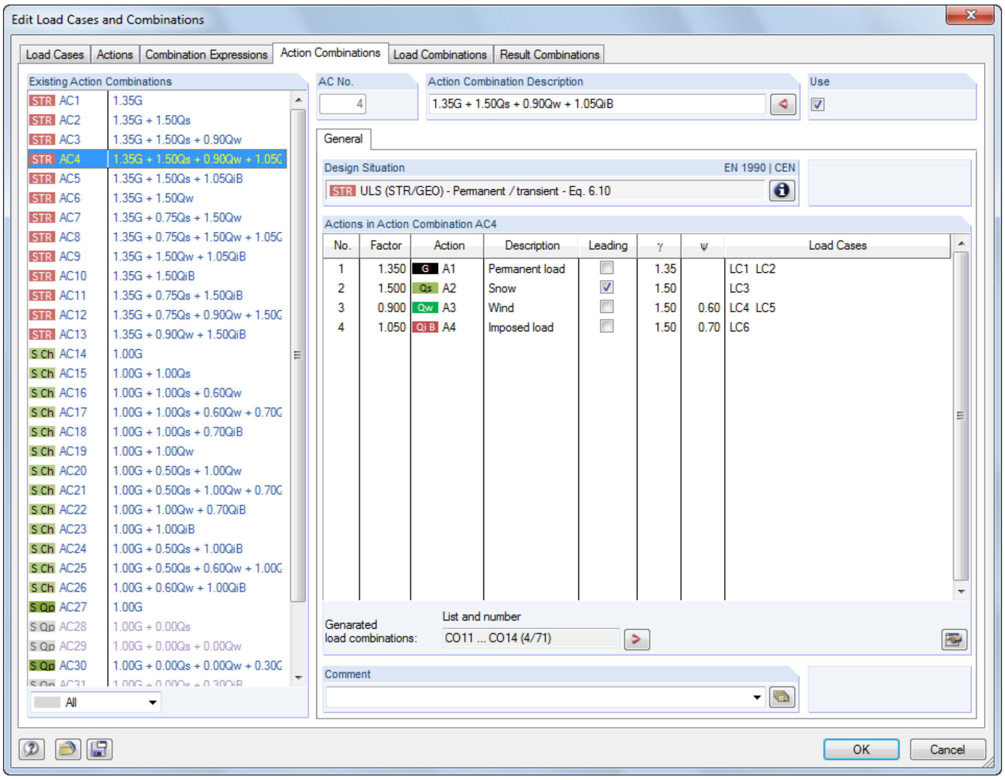
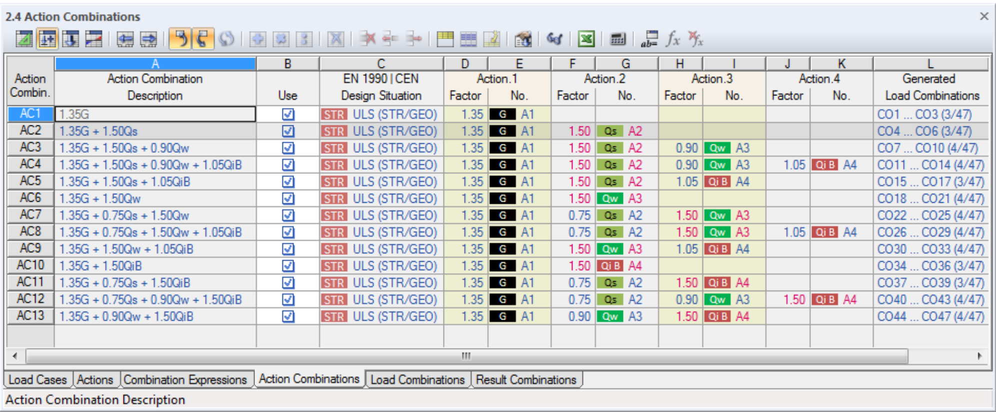


Figure 5.21: Dialog box *Edit Load Cases and Combinations*, tab *Action Combinations*



2.4 Action Combinations

Action Combin.	A: Action Combination Description	B: Use	C: EN 1990 | CEN Design Situation	D: Action.1 Factor	E: Action.1 No.	F: Action.2 Factor	G: Action.2 No.	H: Action.3 Factor	I: Action.3 No.	J: Action.4 Factor	K: Action.4 No.	L: Generated Load Combinations
AC1	1.35G	☑	STR ULS (STR/GEO)	1.35	G A1							CO1 ... CO3 (3/47)
AC2	1.35G + 1.50Qs	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qs A2					CO4 ... CO6 (3/47)
AC3	1.35G + 1.50Qs + 0.90Qw	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qs A2	0.90	Qw A3			CO7 ... CO10 (4/47)
AC4	1.35G + 1.50Qs + 0.90Qw + 1.05QiB	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qs A2	0.90	Qw A3	1.05	Qi B A4	CO11 ... CO14 (4/47)
AC5	1.35G + 1.50Qs + 1.05QiB	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qs A2	1.05	Qi B A4			CO15 ... CO17 (3/47)
AC6	1.35G + 1.50Qw	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qw A3					CO18 ... CO21 (4/47)
AC7	1.35G + 0.75Qs + 1.50Qw	☑	STR ULS (STR/GEO)	1.35	G A1	0.75	Qs A2	1.50	Qw A3			CO22 ... CO25 (4/47)
AC8	1.35G + 0.75Qs + 1.50Qw + 1.05QiB	☑	STR ULS (STR/GEO)	1.35	G A1	0.75	Qs A2	1.50	Qw A3	1.05	Qi B A4	CO26 ... CO29 (4/47)
AC9	1.35G + 1.50Qw + 1.05QiB	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qw A3	1.05	Qi B A4			CO30 ... CO33 (4/47)
AC10	1.35G + 1.50QiB	☑	STR ULS (STR/GEO)	1.35	G A1	1.50	Qi B A4					CO34 ... CO36 (3/47)
AC11	1.35G + 0.75Qs + 1.50QiB	☑	STR ULS (STR/GEO)	1.35	G A1	0.75	Qs A2	1.50	Qi B A4			CO37 ... CO39 (3/47)
AC12	1.35G + 0.75Qs + 0.90Qw + 1.50QiB	☑	STR ULS (STR/GEO)	1.35	G A1	0.75	Qs A2	0.90	Qw A3	1.50	Qi B A4	CO40 ... CO43 (4/47)
AC13	1.35G + 0.90Qw + 1.50QiB	☑	STR ULS (STR/GEO)	1.35	G A1	0.90	Qw A3	1.50	Qi B A4			CO44 ... CO47 (4/47)

Load Cases | Actions | Combination Expressions | Action Combinations | Load Combinations | Result Combinations

Action Combination Description

Figure 5.22: Table 2.4 *Action Combinations*

Action combination

The combinations generated from actions are consecutively numbered. An action combination includes all possibilities how load cases contained in the action can be considered. These possibilities depend on the action category and the combination expressions.

In the dialog box *Edit Load Cases and Combinations* below the list *Existing Action Combinations*, it is possible to filter generated combinations by design situation or relevance.

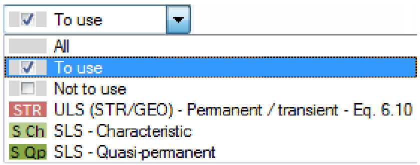


Figure 5.23: Filter option in the dialog box *Edit Load Cases and Combinations*

Description of action combination

RFEM assigns automatically brief descriptions based on the safety factors and symbols of actions, expressing combination rules. You can change these descriptions, if necessary.

Click the dialog button [◀] to jump to the previous dialog tab where RFEM shows you the combination expression by which the current action combination was created.

Use

Use the check box to decide if the selected action combination is considered for creating load or result combinations. In this way, it is possible to reactive or exclude action combinations from the generation.

If RFEM creates an action combination twice because of special constellations, one of them is automatically deactivated.

Design situation

The design situation of the current action combination is again indicated so that you can check data. Use the [Info] button to look at the combination rule of the design situation. A dialog box with explanations opens (see Figure 5.13, page 180).

Actions in action combinations

The columns inform you about actions including corresponding partial safety factors and combination coefficients.

If an action is assumed to be *Leading* in the combination, it is marked accordingly in the dialog box. In this case, it is inserted as action $Q_{k,1}$ in Equation 5.1 to Equation 5.7 (see page 177).

The values indicated in the table column *Factor* are based on coefficients depending on the selected standard. For EN 1990 they are the partial safety factors γ, the combination factors ψ, the reduction factors ξ and, if applicable, the reliability factors K_{FI} of each action resulting from design situation and action category.

Use the buttons [Settings] or [...] to check and, in case of a user-defined standard, adjust the partial safety factors and combination coefficients. The factors are organized in several tabs of the dialog box *Coefficients*. The first tab *Partial Safety Coefficients* is shown in Figure 12.27 on page 556. The tab *Combination Coefficients* manages the factors ψ and ξ.

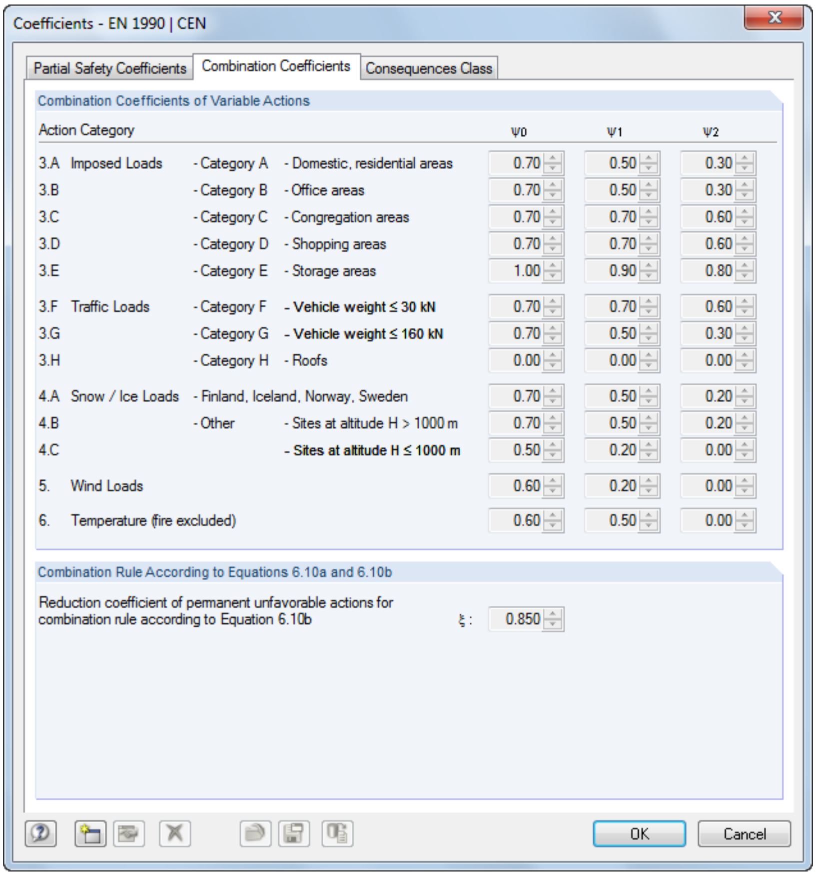


Figure 5.24: Dialog box *Coefficients*, tab *Combination Coefficients*

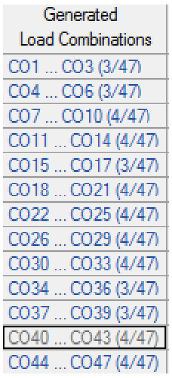
The dialog section *Actions in Action Combination* lists the *Load Cases* contained in the actions with all possibilities how they can be taken into account in the action. The possibilities depend on the action type and the defined action (simultaneous or alternative). It is precondition that all assigned load cases are always used together for the action types "permanent loads" and "prestress" unless the relation is defined as "alternative". In case of variable, extraordinary and seismic actions, assigned load cases can be superimposed in all relevant combinations.

Generated load or result combinations

The dialog section, respectively table column, is filled during the generation starting automatically when closing the dialog tab or table. The dialog field shows you a short overview about the number of generated load or result combinations.

Load and result combinations are described in the following chapters 5.5 and 5.6.

Example



In the example shown on the left, a total of 47 load combinations is generated for the design situation ULS. For the action combination **AC12** (penultimate row) the four load combinations CO40 to CO43 occur with the following background:

The first action A1 has been categorized as action category "permanent loads" and provided with the factor γ = 1.35 in the generated load combinations. The contained load cases 1 and 2 occur together in all load combinations.

As second action A2 we have the action category "snow" included in the load combination with the factor $\gamma * \psi = 1.50 * 0.50 = 0.75$.

The third action A3 doubles the number of the generated load combinations because the category "wind" is available with the two load cases 4 and 5 acting alternatively. This action is multiplied with the factor $\gamma * \psi = 1.50 * 0.60 = 0.90$ in the load combinations.

The fourth action A4 is classified as action type "imposed load category B" and provided with the factor γ = 1.50 in all four load combinations. This action is a leading action.

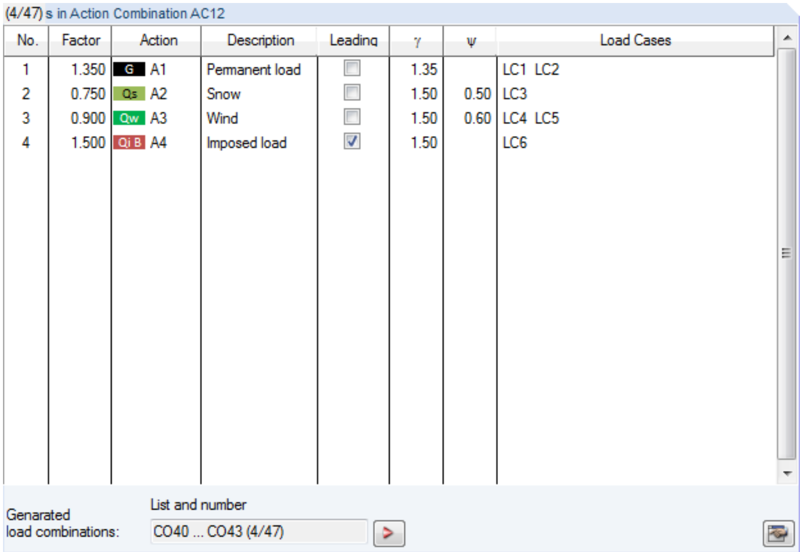


Figure 5.25: Actions in action combination AC12

In addition, we have to take into account the two imperfection load cases 7 and 8 which are coupled with the directions of both wind load cases. We want to create load combinations once with imperfections and once without.

With these specifications RFEM forms the following load combinations for AC12:

- CO40: 1.35*LC1 + 1.35*LC2 + 0.75*LC3 + 0.9*LC4 + 1.5*LC6
- CO41: 1.35*LC1 + 1.35*LC2 + 0.75*LC3 + 0.9*LC4 + 1.5*LC6 + LC7
- CO42: 1.35*LC1 + 1.35*LC2 + 0.75*LC3 + 0.9*LC5 + 1.5*LC6
- CO43: 1.35*LC1 + 1.35*LC2 + 0.75*LC3 + 0.9*LC5 + 1.5*LC6 + LC8

Click the dialog button [▶] to jump to dialog the tab *Load Combinations* where the first combination created from the current action combination is selected.

Comment

Enter a user-defined note or select an entry from the list.

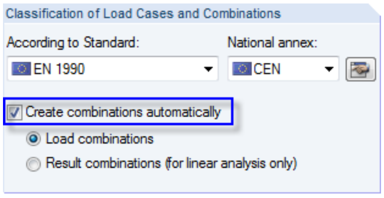
5.5 Load Combinations

General description

Load cases can be superimposed in a load combination (**CO**) and in a result combination (**RC**).

Difference between
load and result combination

Taking into account partial safety factors, a load combination combines the loads of the contained load cases in "one big load case" that will be calculated. In a result combination (see chapter 5.6, page 201) all included load cases are calculated first. Then, results will be superimposed, taking into account the partial safety factors.



Check box in dialog box
Model - General Data

Load cases can be combined manually (see chapter 5.5.1) or superimposed automatically by RFEM (see chapter 5.5.2), depending on settings in the dialog box *Model - General Data* (see Figure 12.23, page 554). Settings affect also the appearance of the dialog tab *Load Combinations* in the loading dialog box.

When you want to calculate combined load cases according to second-order or large deformation analysis, you generally have to create load combinations. The same applies to models with nonlinear elements. The following example is used to demonstrate the subject.

Two load cases act on a slab with elastic foundations. In load case 1, the surface load acts on the entire plate. In load case 2, it stresses only one part of the surface. The self-weight is not taken into account. The elastic foundation of the slab is ineffective in case of tension. Therefore, no lifting forces are absorbed.

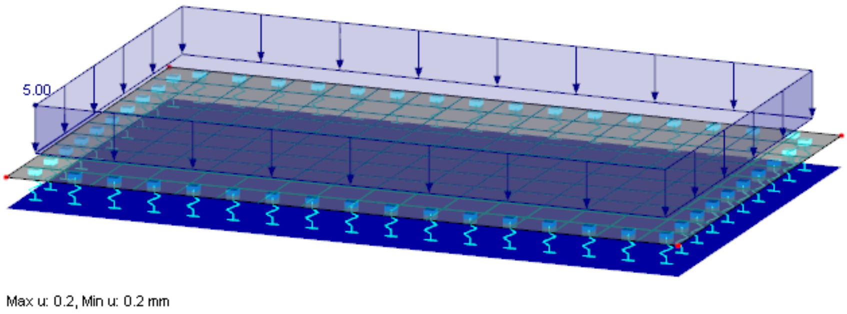


Figure 5.26: Load and deformation in LC 1

The foundation in load case 1 is effective for the entire surface.

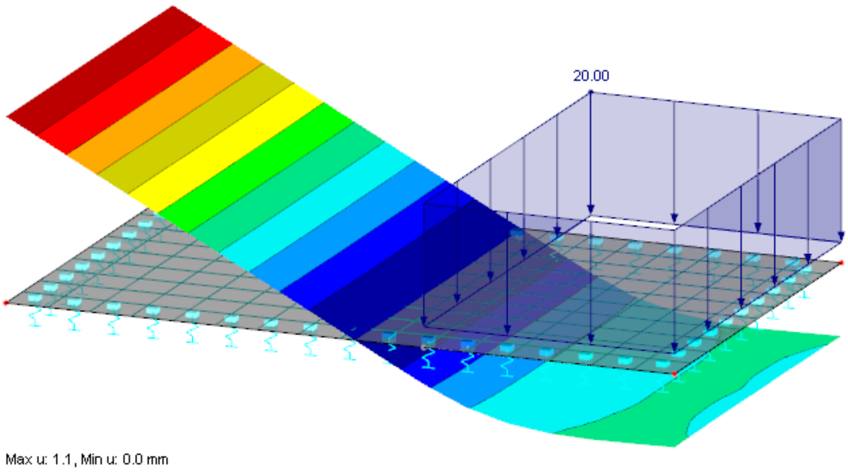


Figure 5.27: Load and deformation in LC 2

The foundation in load case 2 is effective only for the right part of the surface. The left part of the slab is lifting.

When combining both load cases in a result combination, RFEM will show you a warning because adding results would be unacceptable due to nonlinear effects: Deformations in both load cases are based on different structural systems. For a result combination you would see the lifting in the left zone shown in the second case.

Therefore, it is correct to superimpose the two load cases in a load combination. In the figure below, we see that the elastic foundation is effective for the added loads without failure.

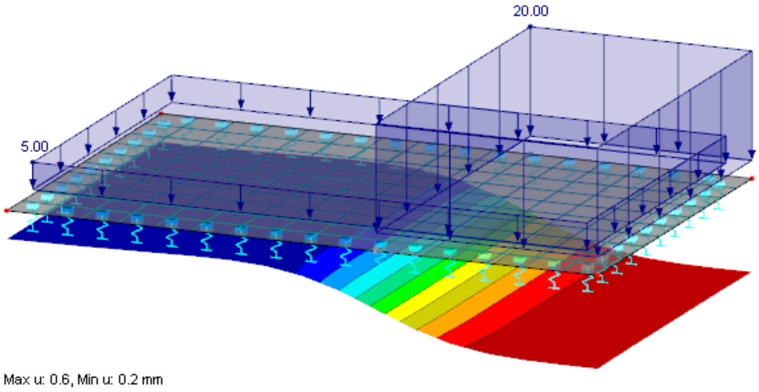


Figure 5.28: Load and deformation of the load combination

5.5.1 User-defined Combinations

Create a new load combination

There are several possibilities to open the dialog box *Edit Load Cases and Combinations* for creating a load combination:

- point to **Load Cases and Combinations** on the **Insert** menu, and then select **Load Combination**

- use the toolbar button [New Load Combination] shown on the left

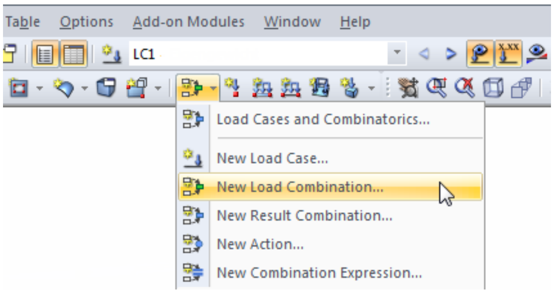


Figure 5.29: Button *New Load Combination* in the toolbar

- context menu of navigator entry *Load Combinations*

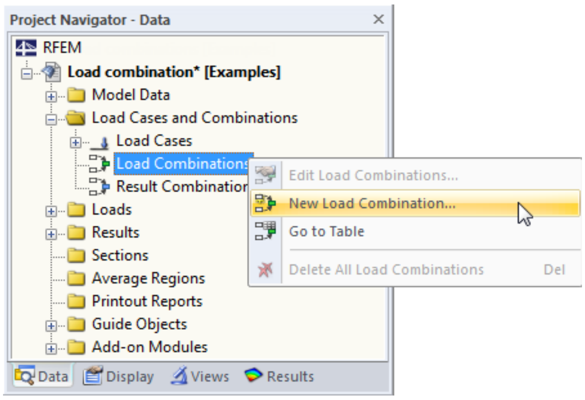


Figure 5.30: Context menu for *Load Combinations* in the *Data* navigator

The dialog box *Edit Load Cases and Combinations* appears. A new load combination is preset in the dialog tab *Load Combinations*.

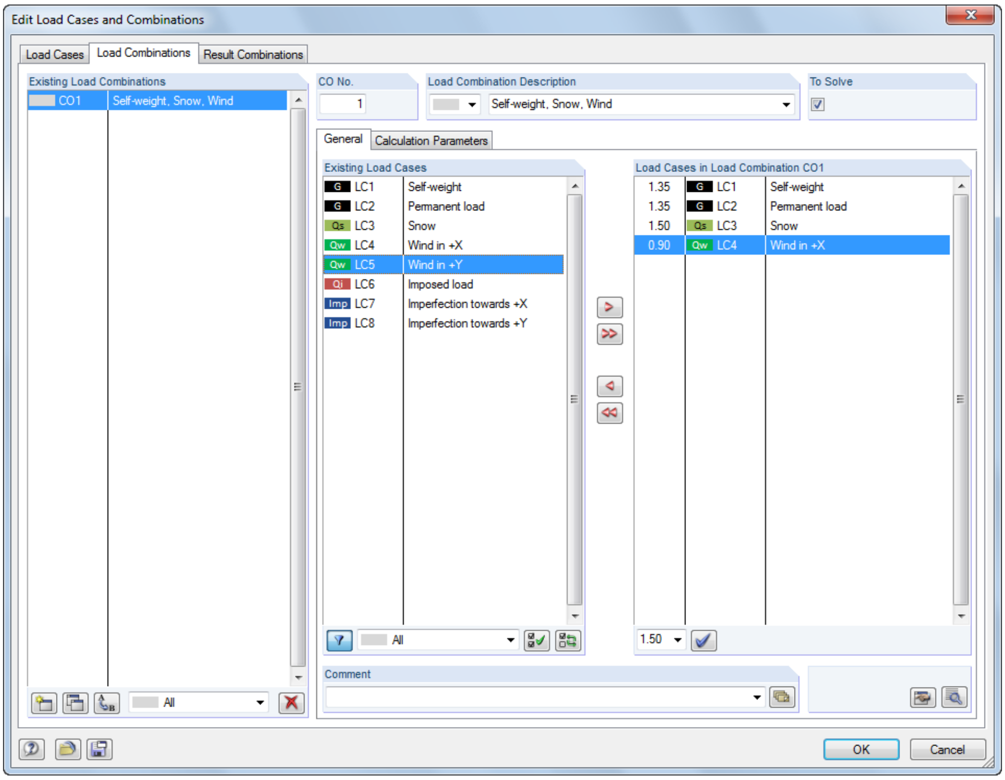
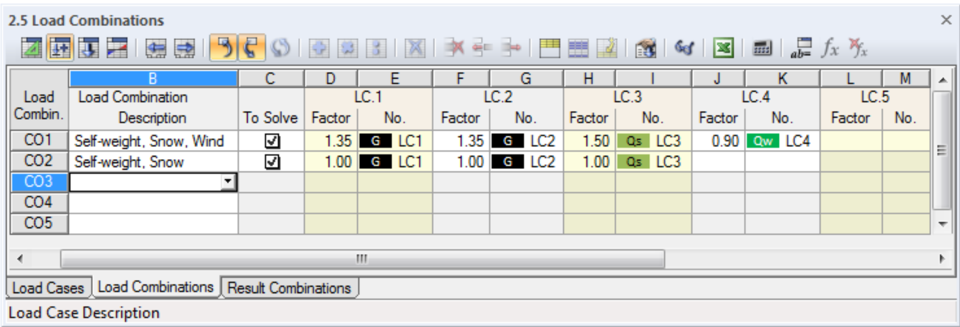


Figure 5.31: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

The following description refers to the tab *General*. The dialog tab *Calculation Parameters* is described in chapter 7.3.1 on page 263.

- It is also possible to enter a new load combination in an empty row of table 2.5 *Load Combinations*.

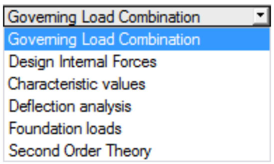


A	B	C	D	E	F	G	H	I	J	K	L	M
Load Combin.	Load Combination Description	To Solve	LC.1 Factor	No.	LC.2 Factor	No.	LC.3 Factor	No.	LC.4 Factor	No.	LC.5 Factor	No.
CO1	Self-weight, Snow, Wind	☑	1.35	G LC1	1.35	G LC2	1.50	Qs LC3	0.90	Qw LC4		
CO2	Self-weight, Snow	☑	1.00	G LC1	1.00	G LC2	1.00	Qs LC3				
CO3												
CO4												
CO5												

Figure 5.32: Table 2.5 *Load Combinations*

Load combination

The number of the new load combination is preset but can be modified in the dialog input field *CO No.*. The order of load combinations can be adjusted subsequently by means of the [Renumber] dialog button (see Table 5.7 and chapter 11.4.18, page 479).



Description of load combination

You can enter any name manually. You can also choose a name from the list to describe the load combination shortly. As manually entered descriptions are stored in the list, they are also available for other models.

To solve

Use the check box to decide if the load combination is considered in the calculation. In this way, it is possible to active or exclude load combinations from the calculation.

Load cases in load combination

The columns inform you about the load cases together with corresponding factors.

The values indicated in the table column *Factor* are based on coefficients depending on the selected standard. For EN 1990 they are the partial safety factors γ, the combination factors ψ, the reduction factors ξ and, if applicable, the reliability factors K_{Fl} of each action resulting from design situation and action category.

To check and adjust the partial safety factors and combination coefficients, use the dialog button [Combination Coefficients]. The dialog box *Coefficients* opens where you find various factors organized in several tabs. The first tab *Partial Safety Coefficients* for EN 1990 is shown in Figure 12.27 on page 556. The tab *Combination Coefficients* manages the factors ψ and ξ (see Figure 5.24, page 191). The reliability factor K_{Fl} can be defined in an input field of the dialog tab *Consequences Class*, but you can also enter a user-defined value.

Combining load cases

In the dialog box *Edit Load Cases and Combinations*, you can assimilate load cases in combinations as follows: Select the relevant load cases in the list *Existing Load Cases* by clicking. You can press the [Ctrl] key (as usual in Windows) to apply the multiple selection. Use the button [▶] to transfer selected load cases to the right in the list *Load Cases in Load Combination*, at the same time partial safety factors and combination coefficients are added automatically.

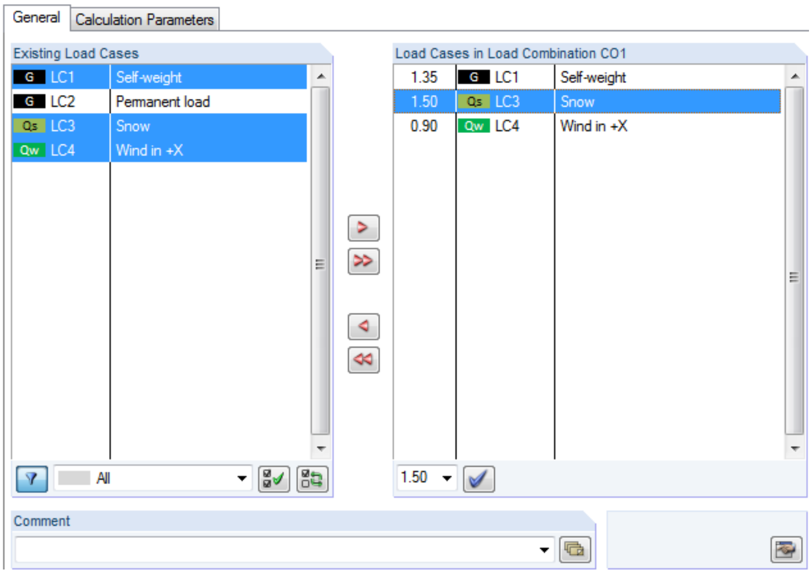
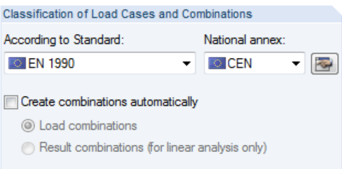


Figure 5.33: Multiple selection of load cases and load combination created according to EN 1990



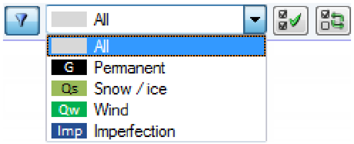
Standard settings in dialog box *Model - General Data*

The factors are created in accordance with the standard set in the dialog box *Model - General Data* (see chapter 12.2.1, page 556).

The preset factors can be checked in the *Coefficients* dialog box that you open with the button [Factors]. Furthermore, you can adjusted them for user-defined standards (see Figure 5.24, page 191 and Figure 12.27, page 556).

To modify the factor of a load case that has been transferred into a load combination, select the load case in the list *Load Cases in Load Combination*. Now, you can enter an appropriate factor into the input field below. You can also select the factor from the list. Finally, click the button [Set Factor] to apply the new factor to the load case.

To remove a load case from a load combination, select the load case in the dialog section *Load Cases in Load Combination*. Use the [◀] button or double-click the entry to return it to the dialog section *Existing Load Cases*.



Several filter options are available below the list *Existing Load Cases*. With the help of those options it is easier to assign load cases sorted by action categories or to select from load cases not yet assigned. The buttons are described in Table 5.7 on page 198.

To define load combinations manually, use the [Edit] button in the bottom right corner of the loading dialog box.

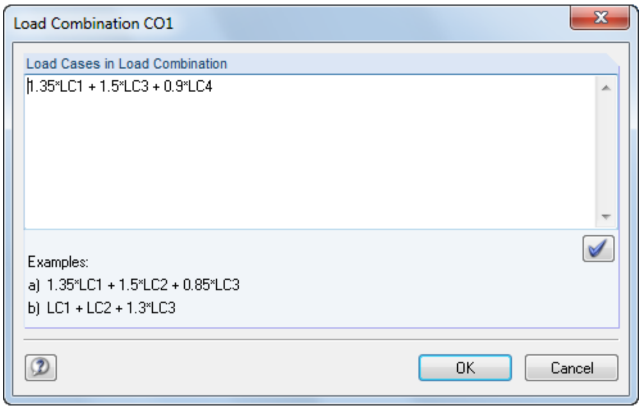


Figure 5.34: Dialog box *Load Combination* for definition via editing field

A dialog box opens offering the input field *Load Cases in Load Combination* where load cases can be added (or subtracted if necessary) by any factor. However, nesting of input is not permitted.

Example: LC1 + 0.5*LC3

To the simple load of load case 1 half the load of load case 3 is added.

Use the button [Set Input] to transfer the entry to the list *Load Cases in Load Combination* of the initial dialog box.

Comment

Enter a user-defined note or select an entry from the list to describe the load combination in detail.

Calculation parameters

The tab *Calculation Parameters* in the loading dialog box offers different options for controlling the calculation. Find a detailed description of these parameters in chapter 7.3.1 on page 263.

Edit a load combination

There are several possibilities to change load combinations subsequently:

- point to **Load Cases and Combinations** on the **Edit** menu, and then click **Load Combinations**
- use the context menu or double-click a load combination in the *Data* navigator

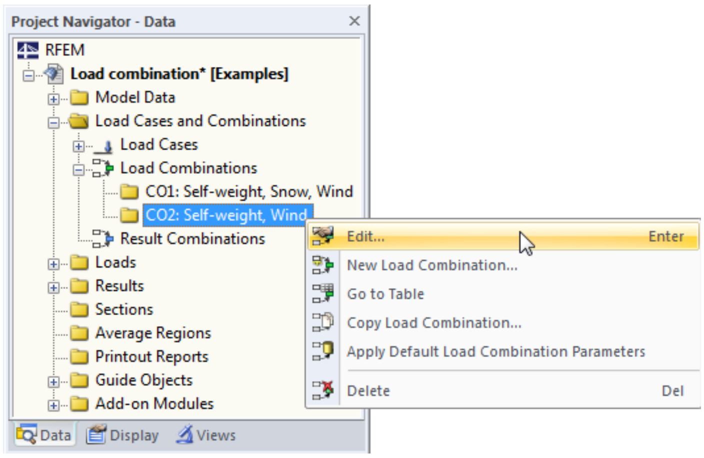


Figure 5.35: Context menu of a load combination

In the dialog box *Edit Load Cases and Combinations* (see Figure 5.31, page 195), select the CO by clicking. Then, you can edit the definition criteria.

Buttons

In the dialog box *Edit Load Cases and Combinations*, you see different buttons below the lists *Existing Load Combinations* and *Existing Load Cases*. The buttons have the following functions:

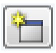
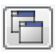
	Creates a new load combination
	Creates a new load combination as copy of the selected combination
	Assigns a new number to the selected load combination. Specify the number in a separate dialog box. It is not allowed to enter a CO number that has already been assigned.
	Deletes the selected load combination
	The list shows only the load cases that are not yet contained in the load combination.
	Selects all load cases in the list
	Inverts the selection of load cases

Table 5.7: Buttons in the tab *Load Combinations*

5.5.2 Generated Combinations

When switching to the dialog tab *Load Combinations* or to table 2.5, RFEM creates the combinations automatically. As the load cases are not superimposed manually, the *General* tab looks differently (see Figure 5.31, page 195 for user-defined combinations).

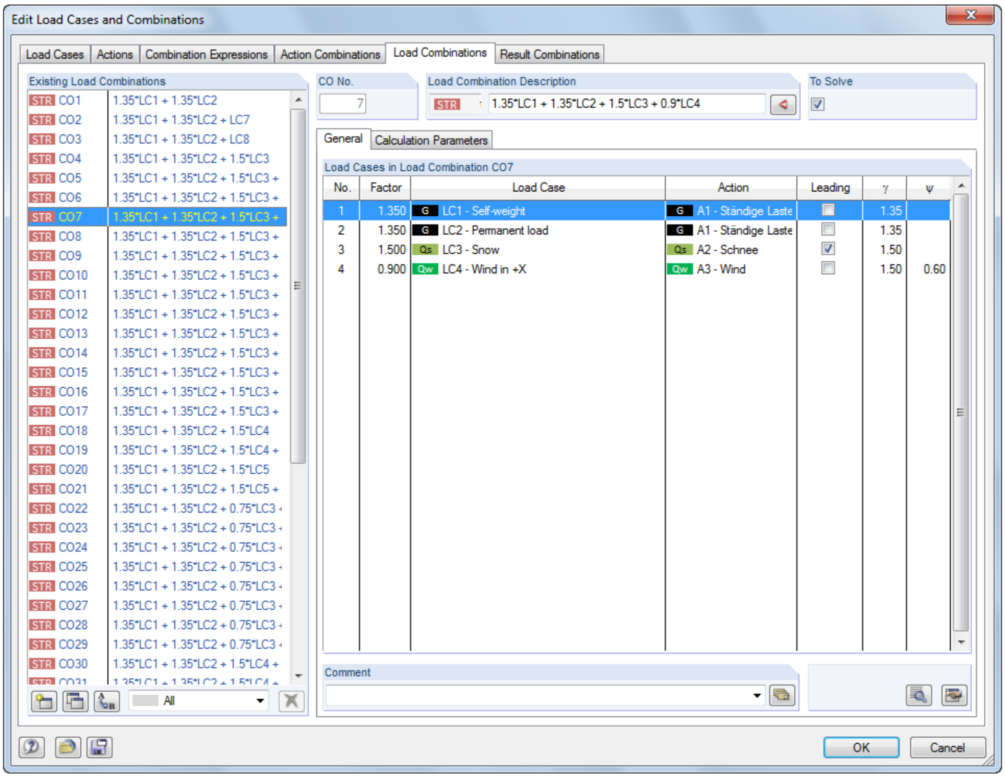
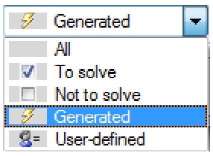


Figure 5.36: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

Load combination



Load combinations generated from action combinations are consecutively numbered.

You can filter the generated combinations by particular criteria, using the selection field in the bottom left corner below the dialog section *Existing Load Combinations*.

Description of load combination

RFEM assigns brief descriptions based on the safety factors and load case numbers, expressing combination rules. You can change these descriptions, if necessary.

Click the dialog button [◀] to jump back to the dialog tab *Action Combinations* (see chapter 5.4, page 189) where the action combination is selected by which the current load combination has been created.

To solve

The check box controls the result determination for the selected load combination(s).

Load cases in load combination

The columns inform you about the load cases including corresponding partial safety factors and combination coefficients. It is not possible to modify the factors of generated combinations.

If a load case is assumed to be *Leading* in the combination, it is marked accordingly in the dialog box.

To check and, if necessary, to adjust the partial safety factors and combination coefficients, use the dialog button [Info about Factors]. The dialog box *Coefficients* is subdivided into several tabs (see Figure 12.27, page 556 and Figure 5.24, page 191).

Add a load combination

The generated load combinations cannot be edited but only deleted or excluded from the calculation by using the check box *To Solve*.

With the [New] button in the left bottom corner below the dialog section *Existing Load Combinations* you can add a user-defined combination. To enable the manual definition, the dialog tab *General* changes its appearance.

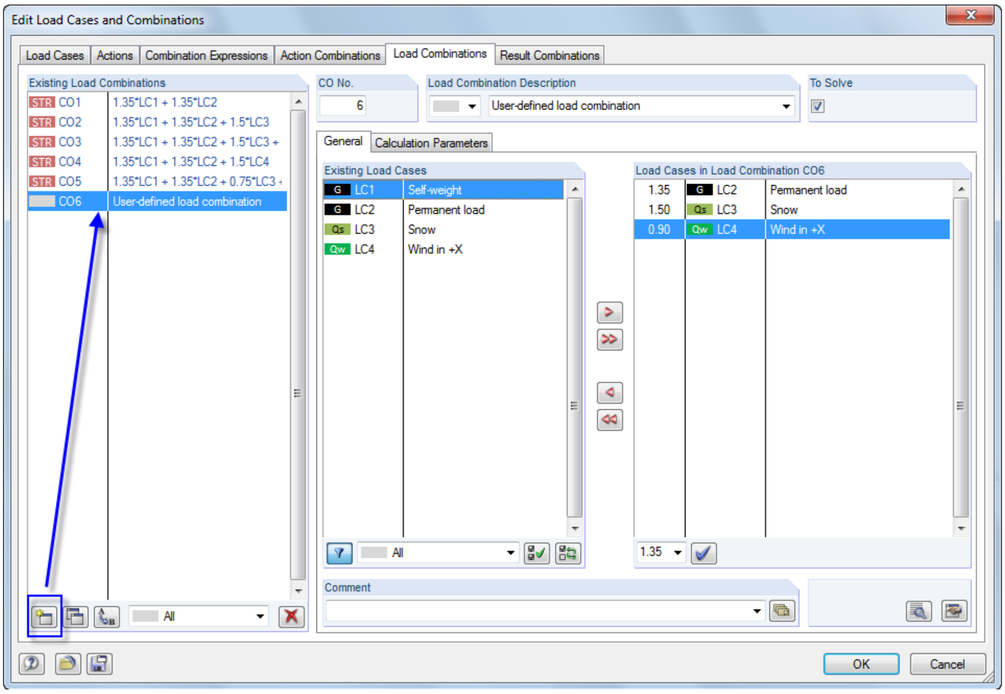


Figure 5.37: Adding a user-defined load combination

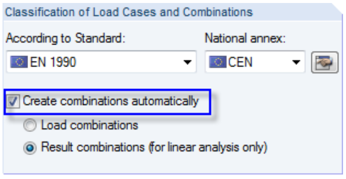
The previous chapter 5.5.1 describes in detail how load combinations can be created manually.

5.6 Result Combinations

General description

Load cases can be superimposed in a result combination (**RC**) and in a load combination (**CO**).

Difference between
result and load combination



Check box in dialog box
Model - General Data

In a result combination, included load cases are calculated first. Then, the results are superimposed, taking into account the partial safety factors. A load combination (see chapter 5.5, page 193) combines the loads of contained load cases in "one big load case" first, taking into account the partial safety factors. Then, the big case is calculated.

Load cases can be combined manually (see chapter 5.6.1) or superimposed automatically by RFEM (see chapter 5.7), depending on settings in the dialog box *Model - General Data* (see Figure 12.23, page 554). Settings affect also the appearance of the dialog tab *Result Combinations* in the dialog box *Edit Load Cases and Combinations*.

Result combinations are not appropriate for nonlinear calculations because they lead to falsified results: In most cases, failure of nonlinear elements (for example tension members, foundations) happens unequally in the individual load cases. Redistribution effects occur so that the internal forces would be combined from different models (see example in chapter 5.5 on page 193).

In a result combination, you can superimpose the results of load cases and load combinations as well as results of other result combinations.

Usually, the internal forces are summed up. In principle, subtractions are also possible. Please note, however, that in this case signs of the internal forces will be reversed: Tensile forces become compressive forces etc. Therefore, as an alternative, it is recommended to copy the load case (see chapter 5.1, page 173) and to set the load factor to –1.00 for the load case copy in the dialog tab *Calculation Parameters*. Then, the load case can be added in a result combination.

5.6.1 User-defined Combinations

Create a new result combination

There are several possibilities to open the loading dialog box for creating a result combination:

- point to **Load Cases and Combinations** on the **Insert** menu, and then select **Result Combination**
- click the button [New Result Combination] in the toolbar

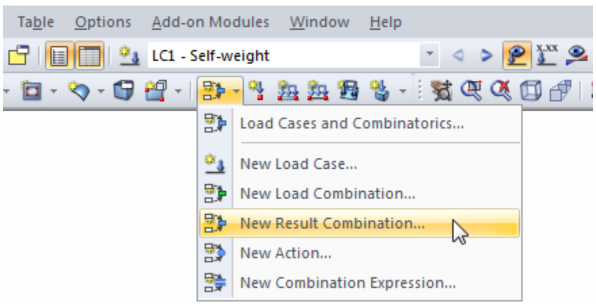


Figure 5.38: Button *New Result Combination* in the toolbar

- use the context menu of the navigator entry *Result Combinations*

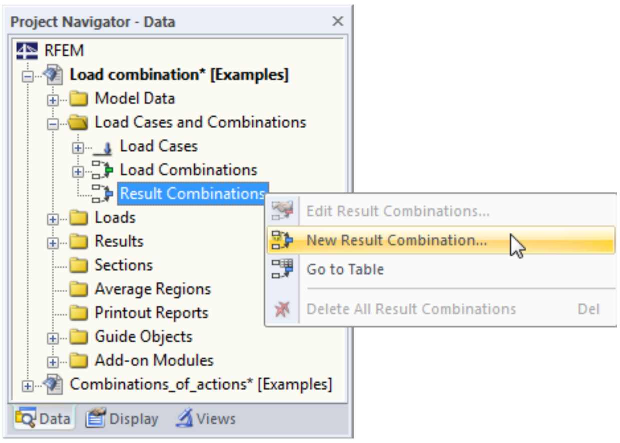


Figure 5.39: Context menu of *Result Combinations* in *Data* navigator

The dialog box *Edit Load Cases and Combinations* appears. A new result combination is preset in the dialog tab *Result Combinations*.

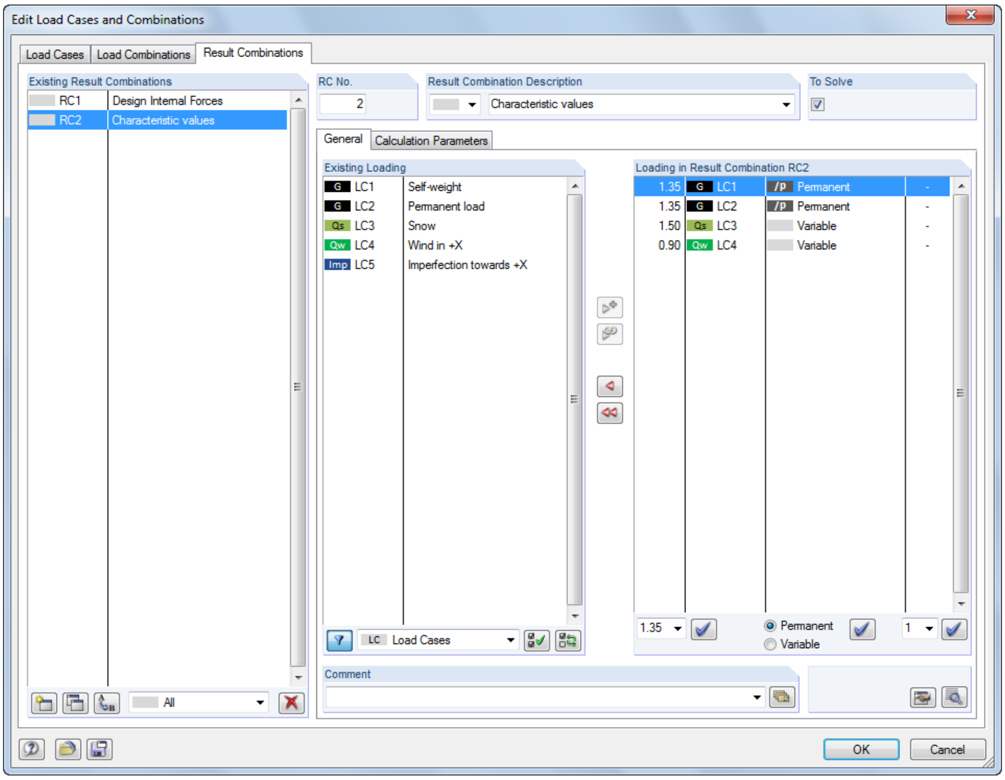
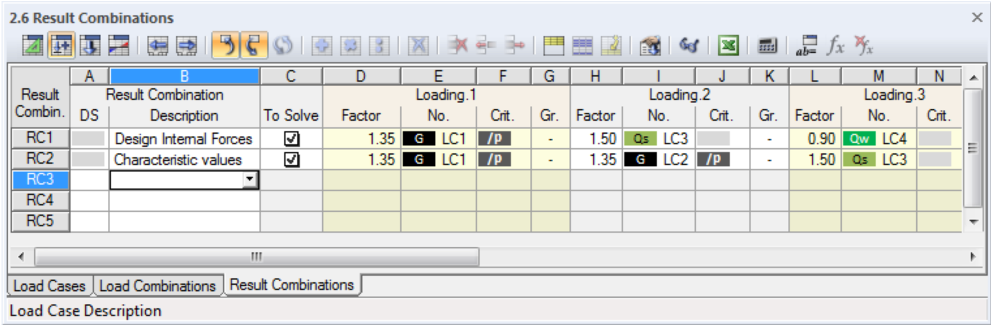


Figure 5.40: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

The following description refers to the tab *General*. The dialog tab *Calculation Parameters* is described in chapter 7.3.2 on page 270.

- It is also possible to enter a new result combination in an empty row of table 2.6 *Result Combinations*.



2.6 Result Combinations

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
Result Combin.	Result Combination			Loading.1				Loading.2				Loading.3		
	DS	Description	To Solve	Factor	No.	Crit.	Gr.	Factor	No.	Crit.	Gr.	Factor	No.	Crit.
RC1		Design Internal Forces	☑	1.35	G LC1	/p	-	1.50	Qs LC3		-	0.90	Qw LC4	
RC2		Characteristic values	☑	1.35	G LC1	/p	-	1.35	G LC2	/p	-	1.50	Qs LC3	
RC3														
RC4														
RC5														

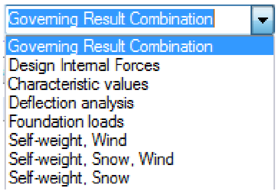
Load Cases | Load Combinations | Result Combinations

Load Case Description

Figure 5.41: Table 2.6 *Result Combinations*

Result combination

The number of the new result combination is preset but can be modified in the dialog input field *RC No.* The order of result combinations can be adjusted also subsequently by means of the [Renumber] dialog button (see Table 5.8 and chapter 11.4.18, page 479).



Description of result combination

You can enter any name manually. You can also choose a name from the list to describe the result combination shortly. As manually entered descriptions are stored in the list, they are also available for other models.

To solve

Use the check box to decide if the result combination is considered in the calculation. In this way, it is possible to reactive or exclude result combinations specifically from the calculation.

Loading in result combination

The columns inform you about the load cases, load and result combinations including related factors.

The values indicated in the table column *Factor* are based amongst others on coefficients that depend on the selected standard. For EN 1990 they are the partial safety factors γ, the combination factors ψ, the reduction factors ξ and, if applicable, the reliability factors K_{FI} of each action resulting from design situation and action category.

To check and adjust the partial safety factors and combination coefficients conforming to standards, use the dialog button [Combination Coefficients]. The dialog box *Coefficients* opens where you find various factors organized in several tabs. The tab *Partial Safety Coefficients* for EN 1990 is shown in Figure 12.27 on page 556. The tab *Combination Coefficients* manages the factors ψ and ξ (see Figure 5.24, page 191). The reliability factor K_{FI} can be defined in an input field of the dialog tab *Consequences Class*, but you can also enter a user-defined value.

Combining loadings

In the dialog box *Edit Load Cases and Combinations*, you can superimpose load cases, load and result combinations in a combination as follows: Select the relevant entries in the list *Existing Loading* by clicking. You can press the [Ctrl] key (as usual in Windows) to apply the multiple selection (see the figure below). Use the dialog buttons [▶$^{+}$] and [▶$^{\infty}$] to transfer the selected entries in the list *Loading in Result Combination* to the right.

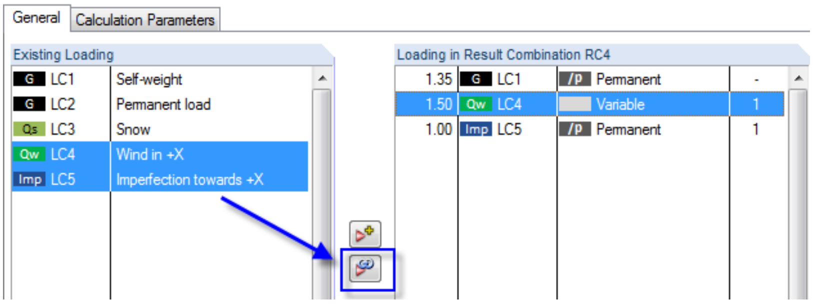
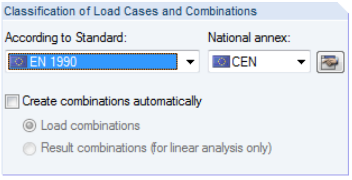


Figure 5.42: Multiple selection for alternative analysis of two load cases



Standard settings in dialog box *Model - General Data*

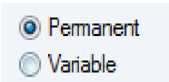
The load case factors are applied according to the standard set in the dialog box *Model - General Data*. If required, you can adjust the preset partial safety factors (see chapter 12.2.1, page 556) by using the dialog button [Factors].

To remove a loading from a result combination, select the relevant entry in the dialog section *Loading in Result Combination*. Use the [◀] button or double-click the entry to return it to the dialog section *Existing Loading*.

The load cases, load and result combinations contained in the result combination can be superimposed in accordance with their effect:

- **Loading criteria**

  Permanent effect



  If you want to apply the loading permanently or unconditionally, the criterion *Permanent* or */p* must be added to the loading.

  Variable effect

  A loading with the criterion *Variable* is considered in the superposition only if its internal forces make an unfavorable contribution to the result.

- **Criteria for superposition**

  Additive combination

  The results of loadings are combined additively with the criterion "+". Use the button [▶+] available in the dialog box to transfer the marked load cases, load and result combinations to the definition list of the result combination.

  Alternative combination

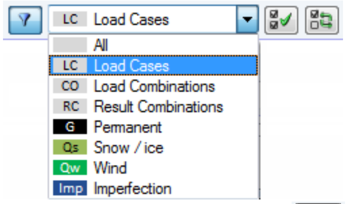
  For the alternative analysis using the "*or*" criterion, respectively the abbreviation "*o*", RFEM treats the results of particular loadings as mutually exclusive. RFEM will consider only values of the loading making the maximum unfavorable contribution. Use the dialog button [▶∞] to transfer selected loadings to the definition list of the result combination.

  Loadings acting alternatively are marked with the same number in the table column *Group*.

  The criterion "orto" (*or to*) combines a list of alternative loadings from the first to the last object. Objects lying in between are not listed.

  All loadings listed in the alternative combination group must be marked consistently as 'Permanent' or 'Variable'. Thus, it is not allowed to enter for example "LC1/p or LC2.

It is possible to adjust the factors of transferred loadings individually: Select the loading(s) in the list *Loading in Result Combination*, and then enter an appropriate factor into the input field. You can also use the list to select a factor. Finally, click the button [Set Factor] to apply the new factor to the loading(s).



Analogously, you can subsequently change the loading criteria (permanent or variable effect) or the group membership of an alternative loading. To assign the new criterion to the selected loading, use the dialog button [Set].

Several filter options are available below the list *Existing Loading*. With the help of the options it is easier to assign loadings sorted by load cases, load and action combinations as well as action categories. In addition, it is possible to restrict the listing to loadings not yet assigned. The buttons are described in Table 5.8 on page 206.

You can define result combinations manually in a separate dialog box. To open it, use the [Edit] button in the bottom right corner of the dialog box *Edit Load Cases and Combinations*.

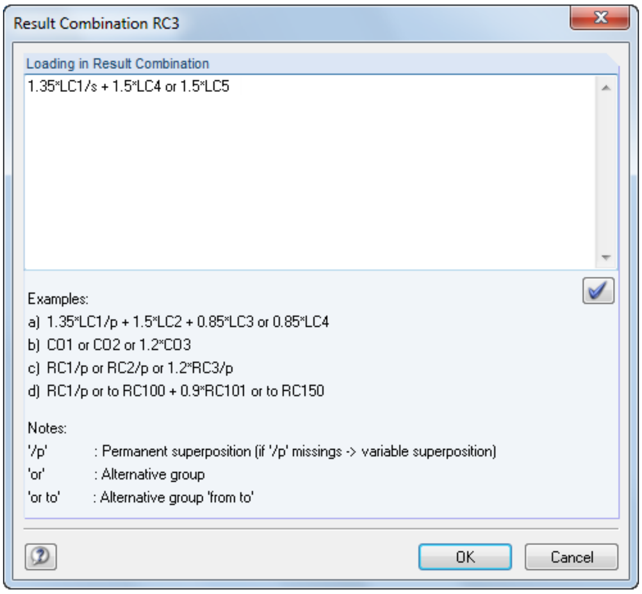


Figure 5.43: Dialog box *Result Combination* for definition via string box

A dialog box opens offering the input field *Loading in Result Combination* where load cases can be added by any factor or combined with the "or" criterion. However, nesting of input is not permitted.

Examples:

- **LC1/p + LC2/p + LC3**

  Load cases 1 and 2 are superimposed as permanent, load case 3 as variable.

- **LC1/p + CO2 + LC3 or LC4 or LC5 (corresponds to LC1/p + CO2 + LC3 orto LC5)**

  Load case 1 is considered as permanent in the superposition, load combination 2 as variable. The most unfavorable case of the load cases 3, 4 or 5 is also superimposed with the 'variable' criterion (that means only one of them is effective - if it increases the result values).

- **1.2*CO1/p + 0.2*RC1 or -0.2*RC1**

  The 1.2-fold of load combination 1 is superimposed as permanent with the most unfavorable contribution of the positive or negative 0.2-fold result combination 1.

- **RC1/p o RC2/p o RC3/p o RC4/p (corresponds to RC1/p orto RC4/p)**

  Result combinations 1 to 4 are compared among each other as permanent acting. The enveloping is determined as the most unfavorable result.

Use the [Set] button to transfer the entry to the list *Loading in Result Combination* of the initial dialog box.

Comment

Enter a user-defined note or select an entry from the list to describe the result combination in detail.

Calculation parameters

The tab *Calculation Parameters* in the loading dialog box offers different options for controlling the calculation. Find a detailed description of these parameters in chapter 7.3.1 on page 263.

Edit a result combination

There are several possibilities to change result combinations subsequently:

- point to **Load Cases and Combinations** on the **Edit** menu, and then click **Result Combinations**
- use the context menu or double-click a result combination in the *Data* navigator

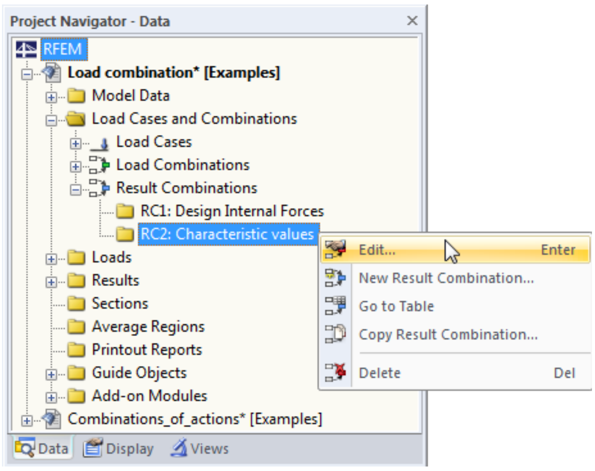


Figure 5.44: Context menu of a result combination

In the dialog box *Edit Load Cases and Combinations* (see Figure 5.40, page 202), select the RC by clicking. Then, you can edit the definition criteria.

Buttons

In the dialog box *Edit Load Cases and Combinations*, several buttons are available below the lists *Existing Result Combinations* and *Existing Loading*. The buttons have the following functions:

	Creates a new result combination
	Creates a new result combination as copy of the selected combination
	Assigns a new number to the selected result combination. Specify the number in a separate dialog box. It is not allowed to enter a RC number that has already been assigned.
	Deletes the selected result combination
	The list shows only load cases not contained in the result combination.
	Selects all load cases in the list
	Inverts the selection of load cases

Table 5.8: Buttons in the tab *Result Combinations*

5.6.2 Generated Combinations

When switching to the dialog tab *Result Combinations* or to table 2.6, RFEM creates the combinations automatically. As the load cases are not superimposed manually, the *General* tab looks differently (se Figure 5.40, page 202 for user-defined combinations).

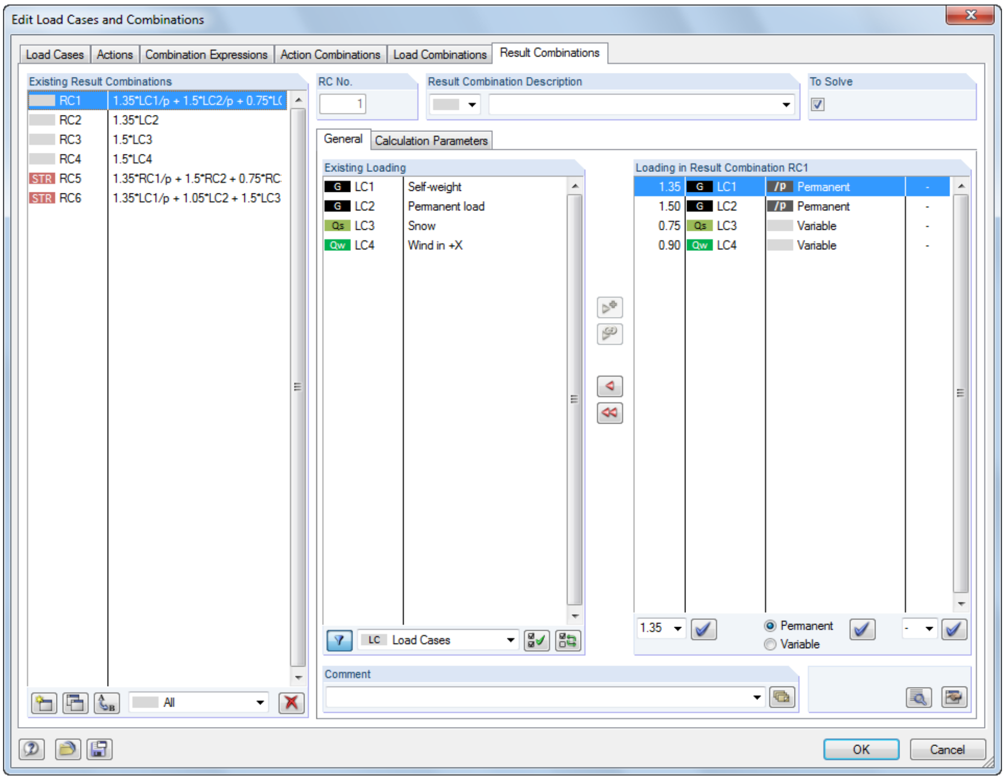
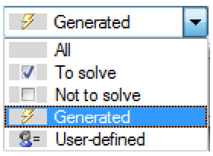


Figure 5.45: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

Result combination

The result combinations generated from action combinations are consecutively numbered.



You can filter the generated combinations by different criteria, using the selection field in the left dialog corner below the dialog section *Existing Result Combinations*.

Description of result combination

RFEM assigns brief descriptions based on the safety factors and load case numbers, expressing combination rules. You can change these descriptions, if necessary.

Click the dialog button [◀] to jump back to the dialog tab *Action Combinations* (see chapter 5.4, page 189) where the action combination is selected by which the current result combination has been created.

To solve

The check box controls the result determination for the result combination(s) selected to the left.

Load cases in result combination

The columns inform you about the load cases including corresponding partial safety factors and combination coefficients. It is not possible to modify the factors of generated combinations.

If a load case is assumed to be *Leading* in the combination, it is marked accordingly in the dialog box.

To check and, if necessary, to adjust the partial safety factors and combination coefficients, use the dialog button [Combination Coefficients]. The dialog box *Coefficients* is subdivided into several tabs (see Figure 12.27, page 556 and Figure 5.24, page 191).

Add a result combination

The generated result combinations cannot be edited but only deleted or excluded from the calculation by using the check box *To Solve*.

With the [New] button in the left bottom corner below the dialog section *Existing Result Combinations* you can add a user-defined combination. To enable the manual definition, the dialog tab *General* changes its appearance.

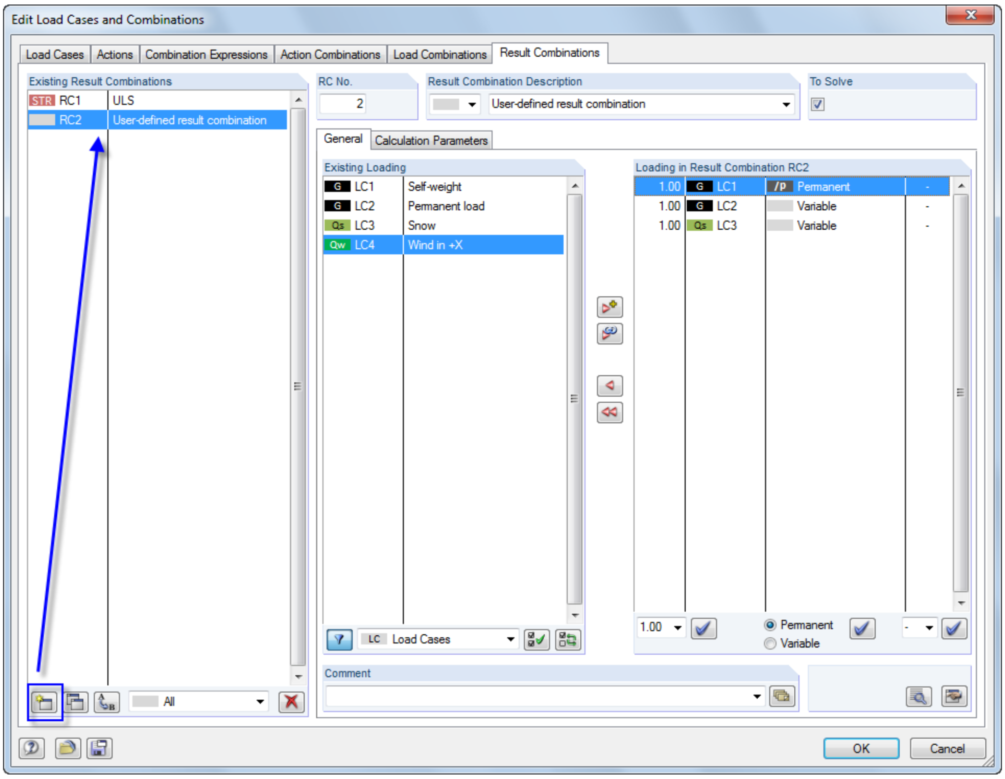


Figure 5.46: Adding a user-defined result combination

The previous chapter 5.6.1 describes in detail how result combinations can be created manually.

5.7 Combination Scheme

Load case constellations can be saved as combination schemes and reused for similar applications. To open the corresponding dialog box,

select **Combination Scheme** on the **Tools** menu.

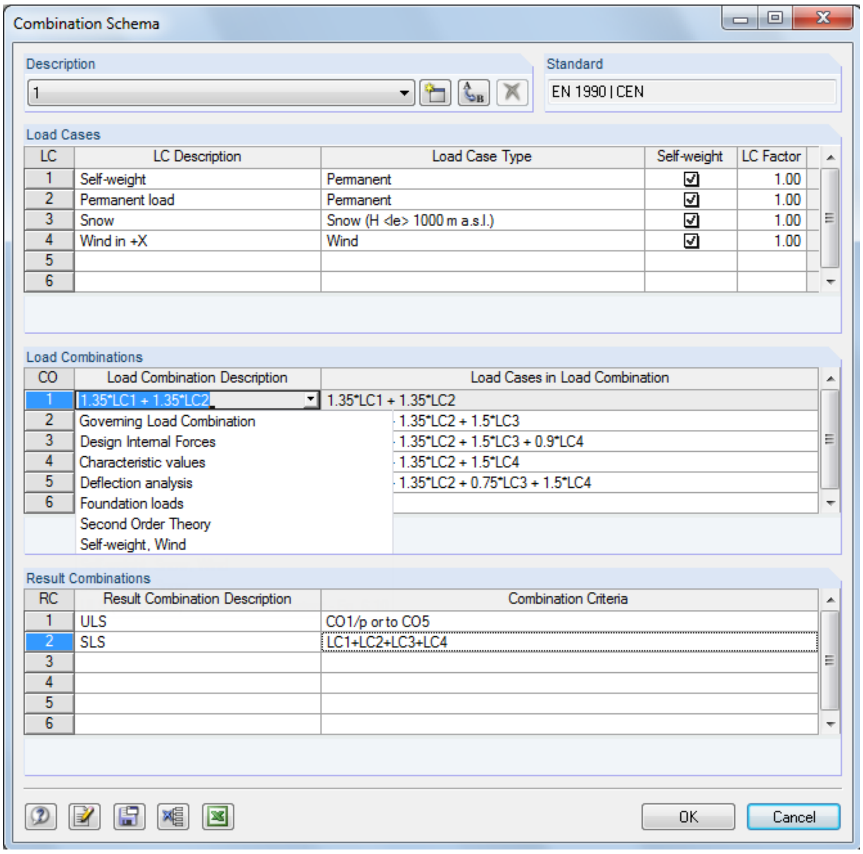


Figure 5.47: Dialog box *Combination Schema*

In the dialog section *Description*, you can select a combination scheme from the list. You can also use the [New] button to create a new scheme.

When load cases have already been defined in the model, they are entered in the dialog section *Load Cases*. Load cases can be added by confirming the last row of the list with the [Enter] or [Tab] key. In the dialog column *LC Description*, you can select predefined descriptions from a list.

The dialog sections *Load Combinations* and *Result Combinations* manage the conditions of superpositioning for load combinations (see chapter 5.5) and result combinations (see chapter 5.6).

To save the combination scheme, click the [Save] button shown on the left. Confirm the dialog box with the [OK] button so that RFEM can create the load cases, load and result combinations.

Do not forget to enter the loading: The combination scheme generates only a frame of load cases, loads and result combinations!

For models based on the same load scheme you can generate all load cases, load and result combinations without entering any more data. Open the scheme dialog box, select the combination scheme from the *Description* list and import it by clicking [OK].

6. Loads

RFEM offers different possibilities to enter loads: You can define loads in a **dialog box**, a **table** and often directly in the **graphic**.

Open the input dialog box

You can access the input dialog boxes and the graphical input in different ways.

Menu *Insert*

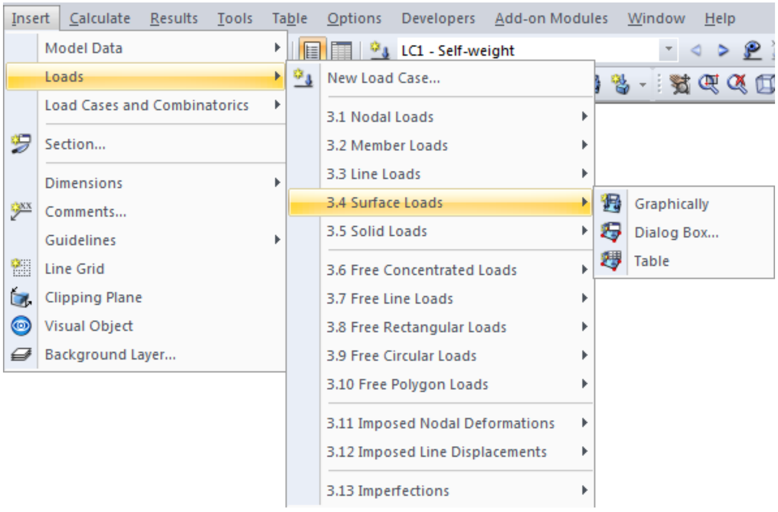


Figure 6.1: Menu *Insert* → *Loads*

Toolbar *Insert*

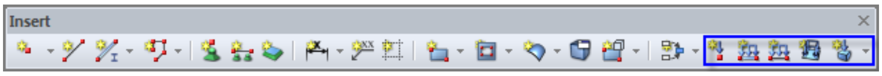


Figure 6.2: Toolbar *Insert*

Context menu in *Data* navigator

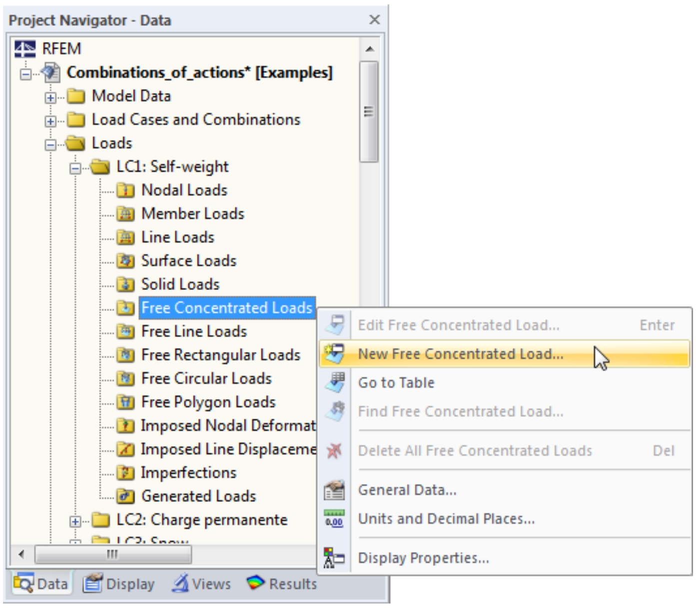
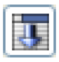


Figure 6.3: Context menu of load objects in the *Data* navigator



Context menu or double-click in table

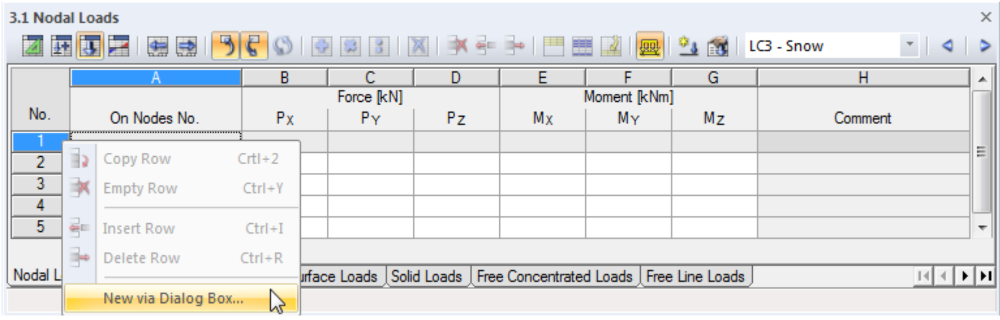


Figure 6.4: Context menu in load tables

The input dialog box can be accessed by means of the context menu (or by double-click) of the row number.

Open the edit dialog box

RFEM provides different possibilities to open a dialog box for editing a load object.

Menu *Edit*

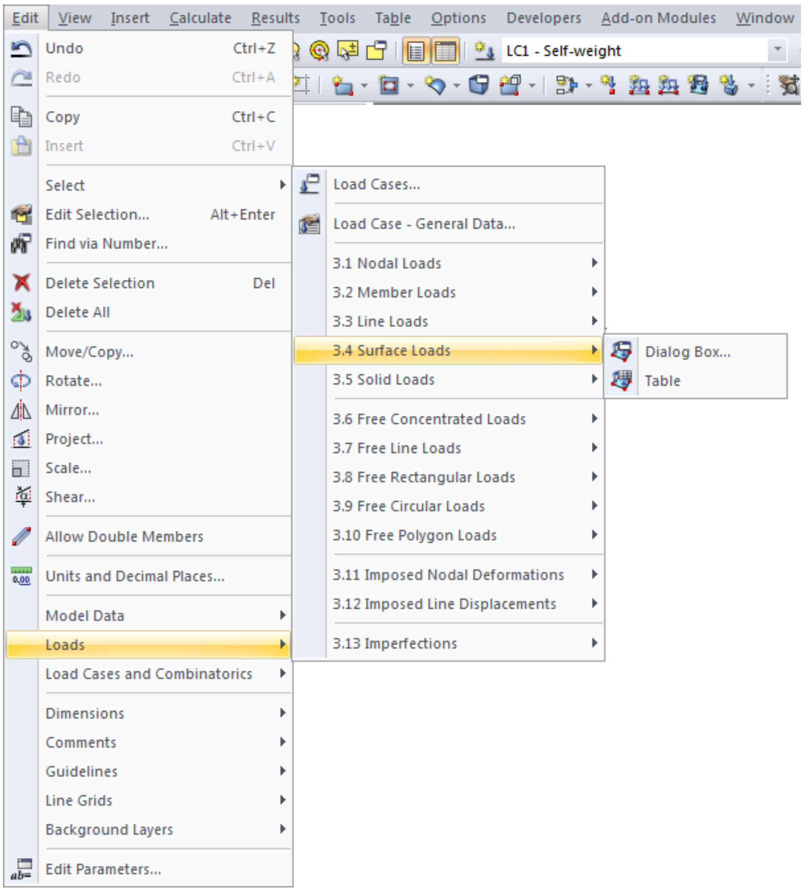


Figure 6.5: Menu *Edit* → *Loads*

The menu option *Dialog Box* can be accessed only when the load object has been selected before.

Context menu or double-click in graphic

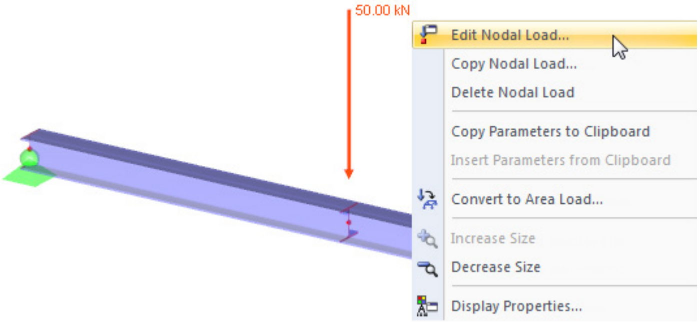


Figure 6.6: Context menu of a load in work window

Context menu or double-click in *Data* navigator

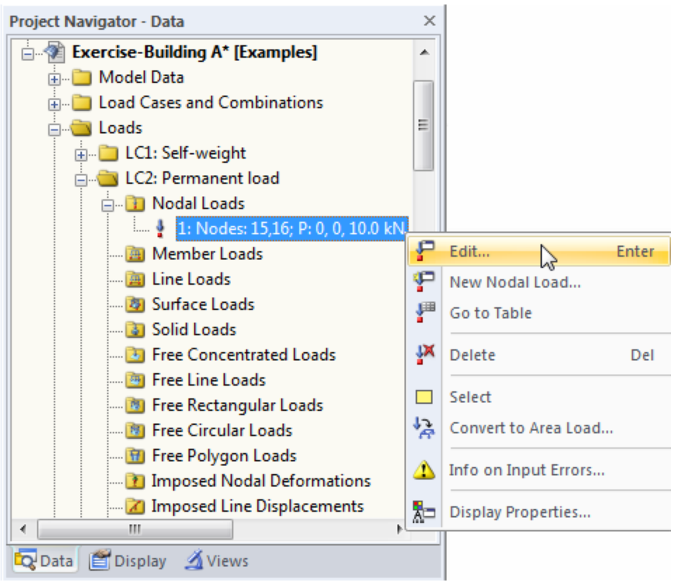


Figure 6.7: Context menu of load objects in the *Data* navigator

Context menu or double-click in table

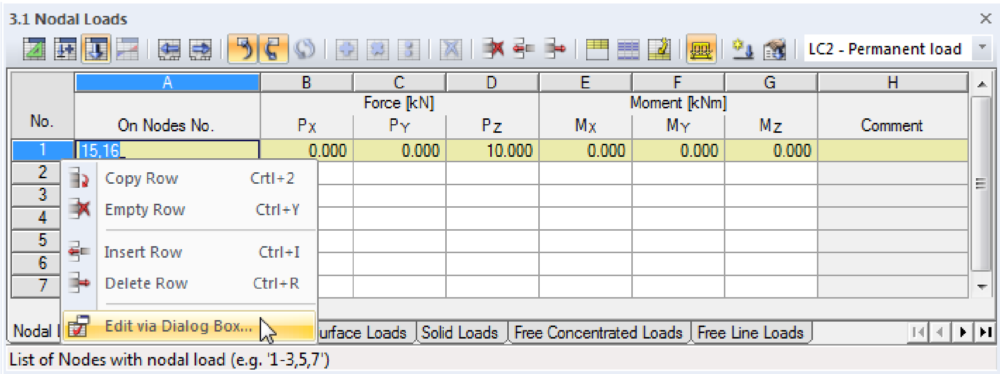
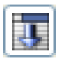


Figure 6.8: Context menu in load tables

The edit dialog box can be accessed by means of the context menu (or by double-click) of the row number.

Table input



Input and modifications carried out in the graphical user interface are immediately shown in the tables and vice versa. To access the load tables, use the third button from the left available in the table toolbar.

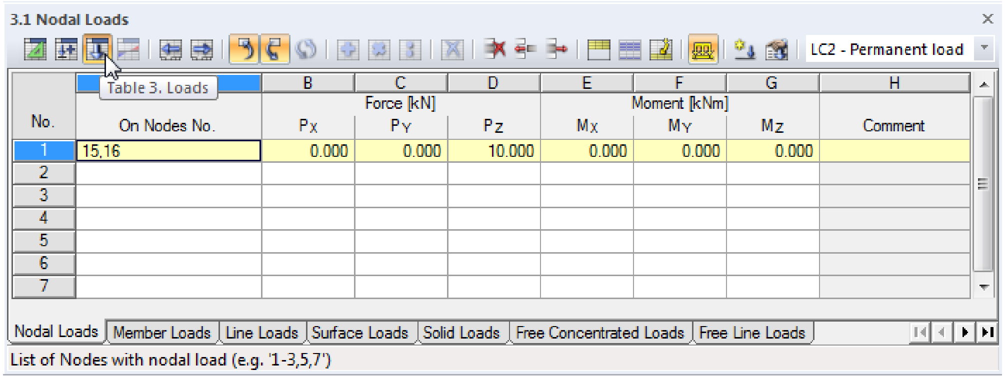


Figure 6.9: Button [Table 3. Loads]

Input in the form of spreadsheet data entered in tables can be quickly edited and imported (see chapter 11.5, page 480).

In each dialog box and table, it is possible to add a *Comment* specifying the load. You can also use predefined comments (see chapter 11.1.4, page 421).

To control whether loads are either listed row by row or summarized in the current table, respectively in all tables, select **Optimize Load Data** on the **Table** menu. You can also use the buttons in the table toolbar shown on the left to activate the settings. You find the buttons to the right of the load case list.

6.1 Nodal Loads

General description

Nodal loads are forces and moments that act on nodes (see chapter 4.1, page 43).

To apply a nodal load, a node must have been previously defined.

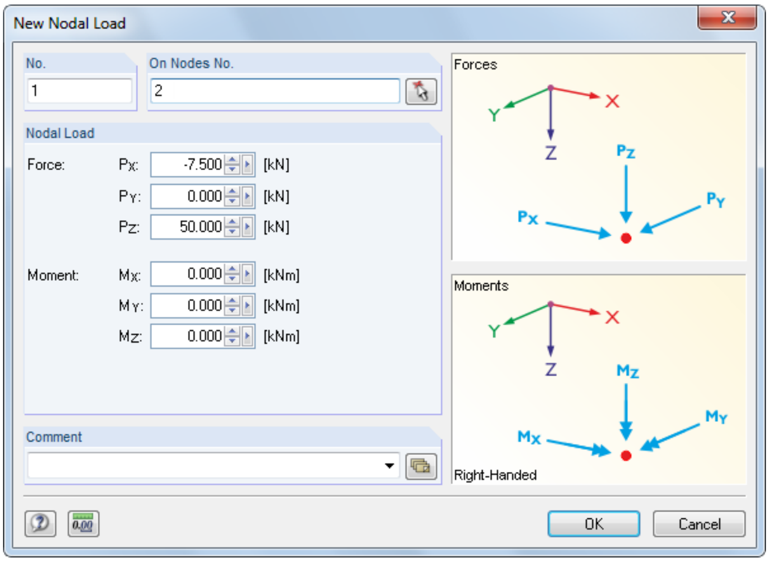
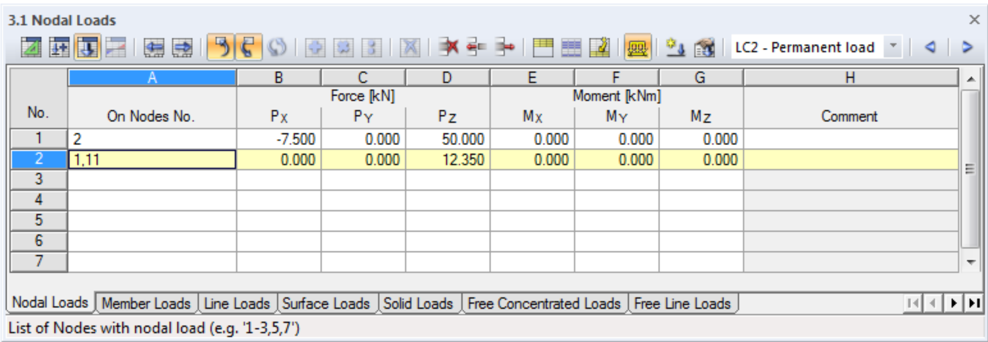


Figure 6.10: Dialog box *New Nodal Load*



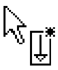
No.	A On Nodes No.	B Force [kN] Px	C PY	D Pz	E Moment [kNm] Mx	F MY	G Mz	H Comment
1	2	-7.500	0.000	50.000	0.000	0.000	0.000	
2	1,11	0.000	0.000	12.350	0.000	0.000	0.000	
3								
4								
5								
6								
7								

Figure 6.11: Table 3.1 *Nodal Loads*

The number of the nodal load is assigned automatically in the dialog box *New Nodal Load* but can be changed in the input field. The numbering order is not important.

On nodes

In this input field, define the numbers of the nodes on which the load is acting. In the dialog box *New Nodal Load*, you can select nodes also graphically by using the [↖] function.



When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter load data first. After clicking [OK] you can select the relevant nodes one after the other in the work window.

Force P_X / P_Y / P_Z

Nodal forces represent vectors referring to the global coordinate system. If a force does not act parallel to one of the global axes, its components X, Y and Z must be determined and entered in the corresponding input fields.

When the model type has been restricted to a planar system in the *General Data* dialog box, you cannot access all three input fields or table columns.

Moment M_X / M_Y / M_Z

Nodal moments refer to the global coordinate system X,Y,Z as well. Therefore, a moment acting in a sloping way must be split in its X, Y and Z-components which can then be entered in the respective input fields.

A positive moment acts clockwise about the corresponding positive global axis. Input is made clearer by the global axes of coordinates represented in the RFEM graphic.

In addition to vectors, moments can be represented as arcs. To control the display properties (see chapter 11.1.2, page 417),

point to **Display Properties** on the **Options** menu, and select **Edit**.

The dialog box *Display Properties* opens where you set the *Category Loads* → *Nodal Loads* → *Nodal Moments*. Then, the display option *Arc* is available for selection in the tab to the right.

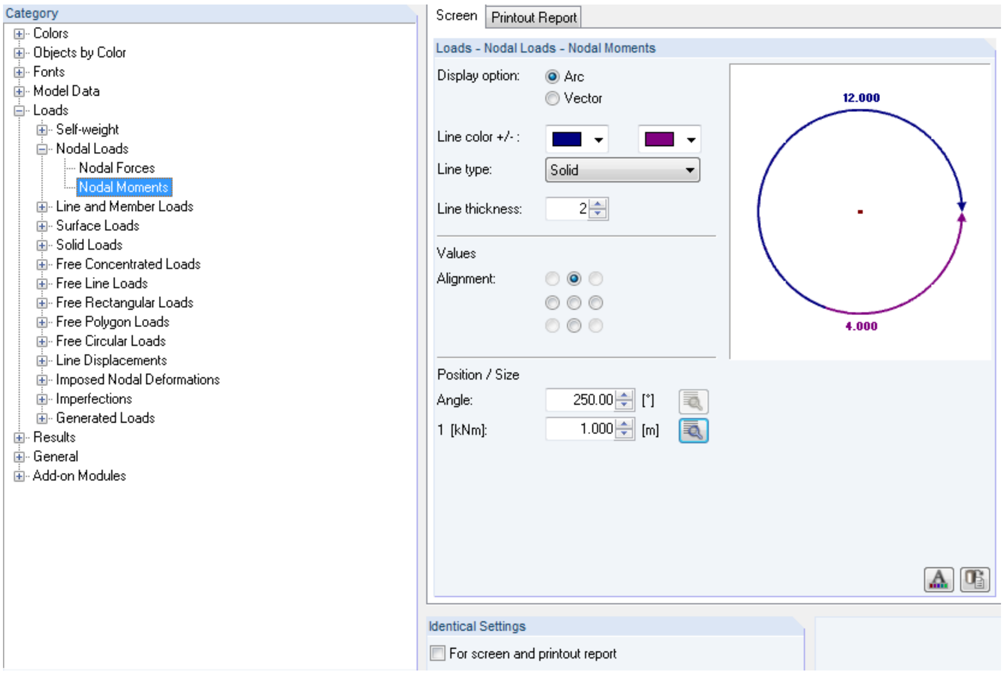
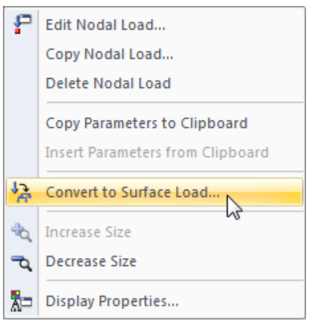


Figure 6.12: Dialog box *Display Properties* (dialog section): *Nodal Moments* with display option *Arc*

It is also possible to import nodal loads from Excel spreadsheets (see chapter 12.5.2, page 569).



Context menu of nodal load

Often, nodal loads result in singularities because the load is concentrated when introduced in a single FE node. To reduce this effect, select **Convert Nodal/Line Load to Surface Load** on the **Tools** menu. You can also use the context menu of a nodal load shown on the left to access the dialog box for converting nodal loads. Open the context menu by a right-click on the object.

A dialog box opens (see Figure 6.13) where you define the parameters for distributing the load. After clicking [OK] the corresponding free rectangular or circular load will be created.

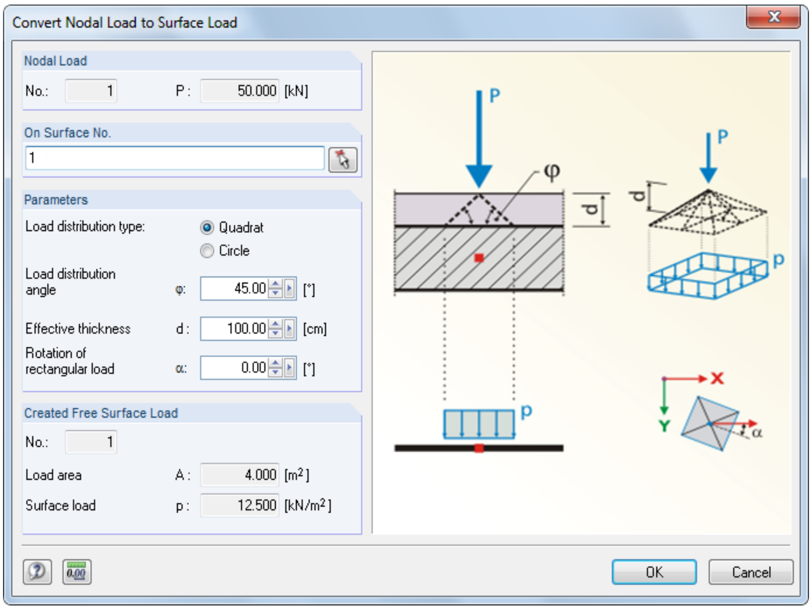


Figure 6.13: Dialog box *Convert Nodal Load to Surface Load*

6.2 Member Loads

General description

Member loads are forces, moments, temperature actions or imposed deformations that act on members.

To apply a member load, a member must have been previously defined.

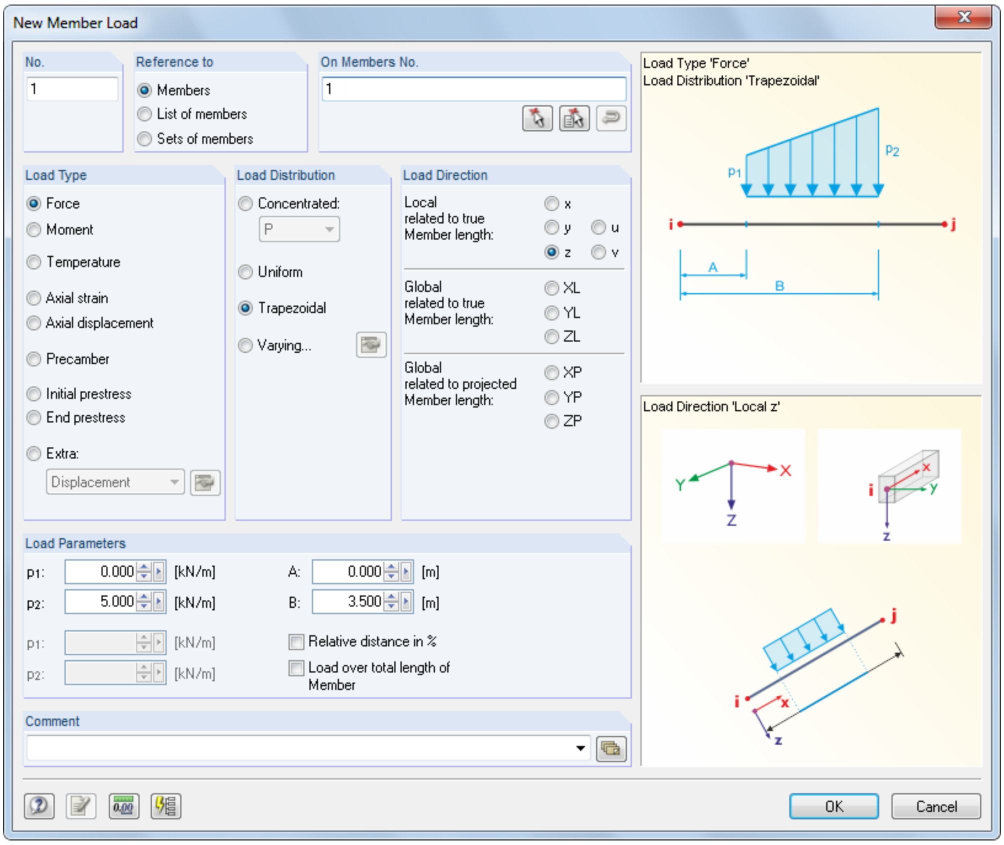


Figure 6.14: Dialog box *New Member Load*

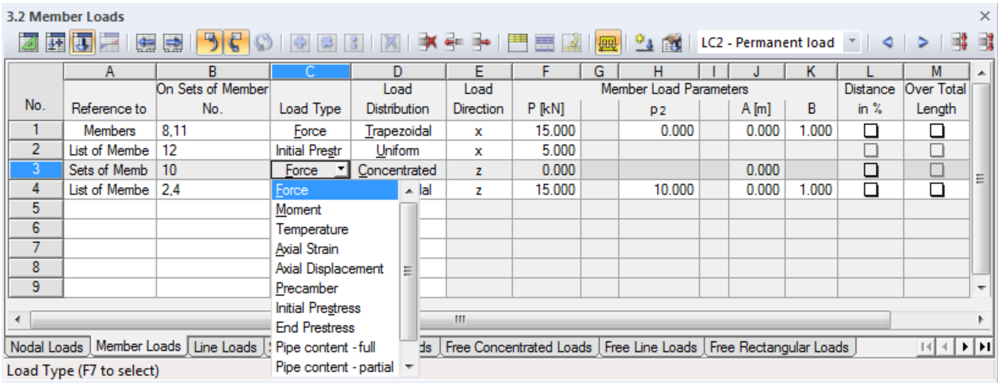


Figure 6.15: Table 3.2 *Member Loads*

The number of the member load is assigned automatically in the dialog box *New Member Load* but can be changed in the input field. The numbering order is not important.

Reference to

Define the model elements to which you want to apply the member load. The following options can be selected:

Members

The load acts on one single member or on each member of several members.

List of members

The load acts on the union of members that are defined in the list. Thus, when trapezoidal member loads are used, load parameters are not applied to each member individually but as total load to all members of the member list. The load effects of a trapezoidal member load on single members in contrast to a member list are shown in Figure 6.16.

Take advantage of a member list to apply loads over all members without defining continuous members. Moreover, it is possible to change the load reference to individual members quickly.

Sets of members

The load acts on a set of members or on each set of several sets of members. Similar to the member list described above, load parameters are applied to the union of members included in the member set.

Sets of members are subdivided into continuous members and groups of members (see chapter 4.21, page 158). Loads on sets of members can be applied to continuous members without problems. Member groups, however, need to be handled with care: The reference to a member group is usually problematic for trapezoidal loads.

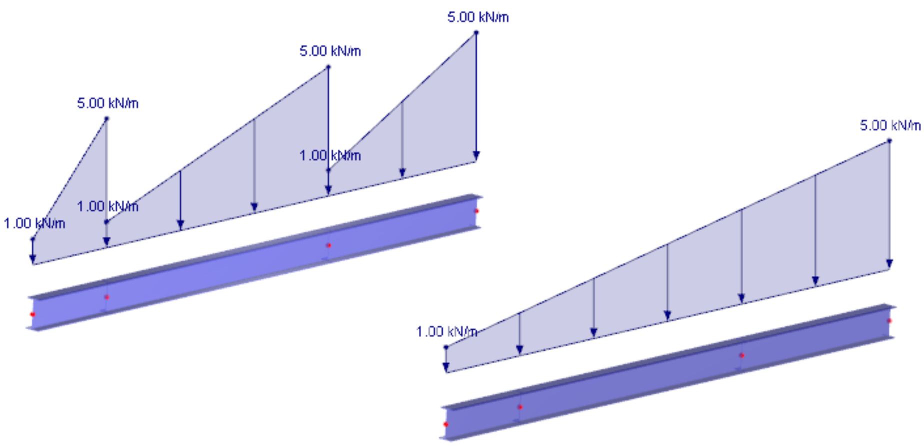
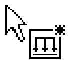


Figure 6.16: Trapezoidal load with reference to members (left) and to a list of members (right)

On members

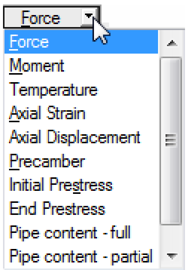
In the input field, enter the numbers of the members or sets of members on which the load is acting. In the dialog box, you can select nodes also graphically by using the [↖] function.



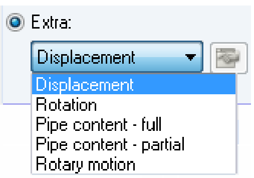
When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter load data first. After clicking [OK] you can select the relevant members or sets of members one after the other in the work window.

For trapezoidal or variable loads with load reference to a member list, you can adjust the member numbers by using the button [Reverse Orientation of Members] shown on the left.

Load type



In this dialog section you define the load type. Depending on your selection, certain parts of the dialog box, respectively columns of the table, are disabled. The following load types can be selected:



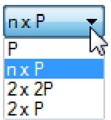
Load type	**Short description**
Force	Concentrated load, distributed load or trapezoidal load
Moment	Concentrated moment, distributed moment or trapezoidal moment
Temperature	Temperature load uniformly distributed over member cross-section, or temperature difference between top and bottom side of member Load applied as uniform or trapezoidal over member length or as trapezoidal over cross-section. A positive load value means that the member or the upper side is heating.
Axial strain	Imposed tensile or compressive strain ε of member A positive load value means that the member is extended. Thus, a prestress as member contraction must be entered negatively. Use the dialog button shown on the left to determine the strain due to shrinkage from the parameters for contraction and drying shrinkage (see Figure 6.24 with description on page 228).
Axial displacement	Imposed tensile or compressive strain Δl of member
Precamber	Imposed curvature of member
Initial prestress	Prestressing force acting on member before calculation A positive load value means that the member is extended.
End prestress	Axial force to be available on member after calculation (not possible for rigid members and cables) A positive load value means that the member is extended.
Displacement	Displacement about quantity Δ for determination of influence lines
Rotation	Rotation about angle φ for influence lines
Pipe content - full	Uniform load due to complete filling of pipe Specify weight density γ of pipe content.
Pipe content - partial	Uniform load due to partial filling of pipe In addition to weight density γ of pipe content, specify filling height *d*.
Rotary motion	Centrifugal force from mass and angular velocity ω on member The rotation axis can be defined in a separate dialog box that you open with the [Edit] button.

Table 6.1: Load types

The graphic in the right corner of the dialog box shows the selected load type including influence of signs set for forces and strains.

Load distribution

The dialog section *Load Distribution* offers different options to represent the effect of the load. The dialog graphic in the top right corner may help you to understand.



Multiple loads

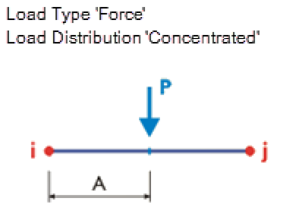
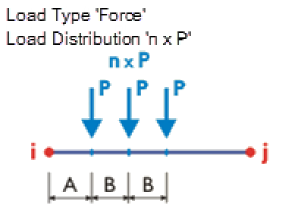
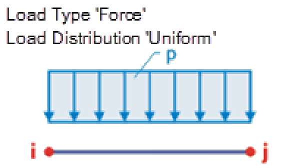
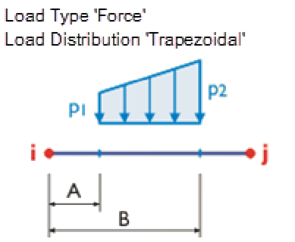
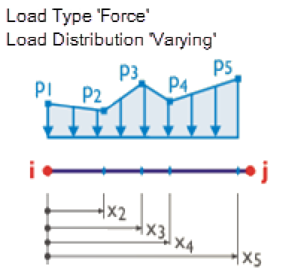
Distribution	Diagram	Description
Concentrated P		Concentrated load, concentrated moment In the dialog section *Load Parameters*, specify the size of the concentrated load or moment and the distance of the point of load application in relation to the member start.
Concentrated n x P		Multiple concentrated loads or moments The list offers several arrangement options for load pairs or multiple concentrated loads such as axle loads. The option shown on the left is appropriate for single forces that are equal in size and acting in a uniform spacing. In the dialog section *Load Parameters*, define the size of the concentrated load, the distance between first load and member start, and the spacing of loads among each other.
Uniform		Uniformly distributed load, uniformly distributed moment In the dialog section *Load Parameters*, specify the size of the uniform member load or moment.
Trapezoidal		Trapezoidal load, trapezoidal moment In the dialog section *Load Parameters*, define both load values and distances for a linearly variable load distribution as shown in the dialog graphic. A triangular load is created by setting one load value to zero. When the check box *Relative distance in %* is ticked, you can specify the distances relatively to the member length.
Varying		Polygonally distributed load Clicking the button [Edit Varying Load] shown on the left opens the dialog box shown in Figure 6.17 where you can enter or import the parameters of the load distribution.

Table 6.2: Load distributions

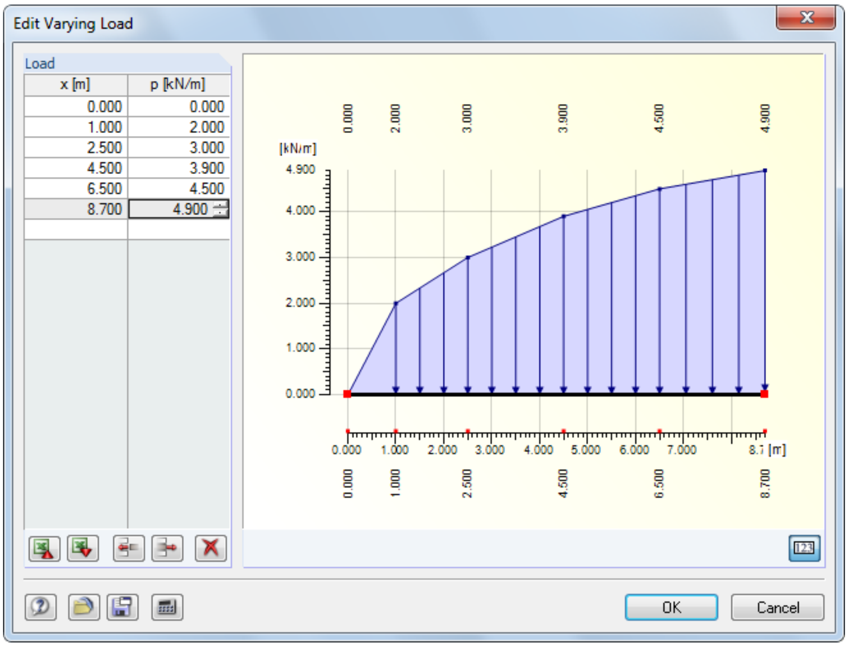


Figure 6.17: Dialog box *Edit Varying Load*

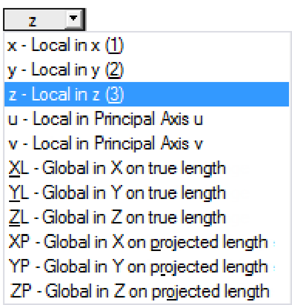
If you want to represent a variable load, you can freely define the *x*-locations on the member with the corresponding load ordinates *p*. You only have to make sure that the x-locations are defined in an ascending order. Use the interactive graphic to check your input immediately.

The buttons in this dialog box have the following functions:

Button	Function
	Table export to MS Excel
	Table import from MS Excel
	Inserts a blank line above pointer
	Deletes active row
	Deletes all entries

Table 6.3: Buttons of the dialog box *Edit Varying Load*

Load direction

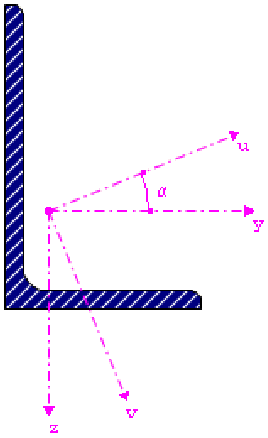


The load can be effective in direction of the global axes X, Y, Z or the local member axes x, y, z or u, v (see chapter 4.13, page 120). For the calculation according to linear static analysis it does not matter whether a load is defined as local or equivalent global. For geometrically non-linear calculations, however, differences between locally and globally defined loads are possible: If the load is defined with a global direction of action, it keeps this direction when finite elements start to twist. In case of a local direction of action, however, the load twists on the member according to the distortion of elements.

When the model type has been reduced to a planar system in the *General Data* dialog box, you cannot access all load directions.

Local

The orientation of member axes is described in chapter 4.17, paragraph *Member rotation* on page 145. The local axis *x* represents the longitudinal axis of the member. Axis *y* represents for symmetrical sections the so-called 'strong' axis, axis *z* accordingly the 'weak' axis of the member



cross-section. In case of unsymmetrical sections, loads can be related to the principal axes *u* and *v* as well as the standard input axes *y* and *z*.

Examples for loads defined as local are wind loads acting on roof structures, temperature loads or prestresses.

Global

The position of the local member axes is irrelevant for the load input if the load acts in direction of an axis of the global coordinate system XYZ.

Examples for loads defined as global are snow loads acting on roof constructions and wind loads on wall and gable columns.

The load impact can be related to different reference lengths:

- **related to true member length**
  The load is applied to the entire member length.
- **related to projected member length in X / Y / Z**
  The application length of the load is converted to the projection of the member in one of the directions of the global coordinate systems. Select this option to define for example a snow load on the projected ground-plan area of a roof.

RFEM applies member loads always in the shear center. An intended torsion originating from the cross-sectional geometry (centroid unequal shear center) is not considered. Therefore, when unsymmetrical cross-sections are used, a torsion moment determined from load x distance to the shear center must be applied additionally if loading is introduced for example in the centroid.

Member load parameters

In this dialog section, respectively table columns, the load values and, if applicable, additional parameters are managed. The input fields are labeled and accessible depending on the selection fields previously activated.

Load p_1 / p_2

Enter load values into the fields. Adjust the signs to the global or local orientations of axes. A positive load value for prestresses, temperature changes and axial strains means that the member is strained and consequently extended.

When a trapezoidal load is selected, specify two load values. The dialog graphic in the upper right corner shows the load parameters.

Distance A / B

In these two fields, enter the distances from the member start for concentrated and trapezoidal loads. It is also possible to define the distances also relative to the member length by ticking the check box *Relative distance in %* (see below).

The dialog graphic in the upper right corner helps you when entering parameters.

Relative distance in %

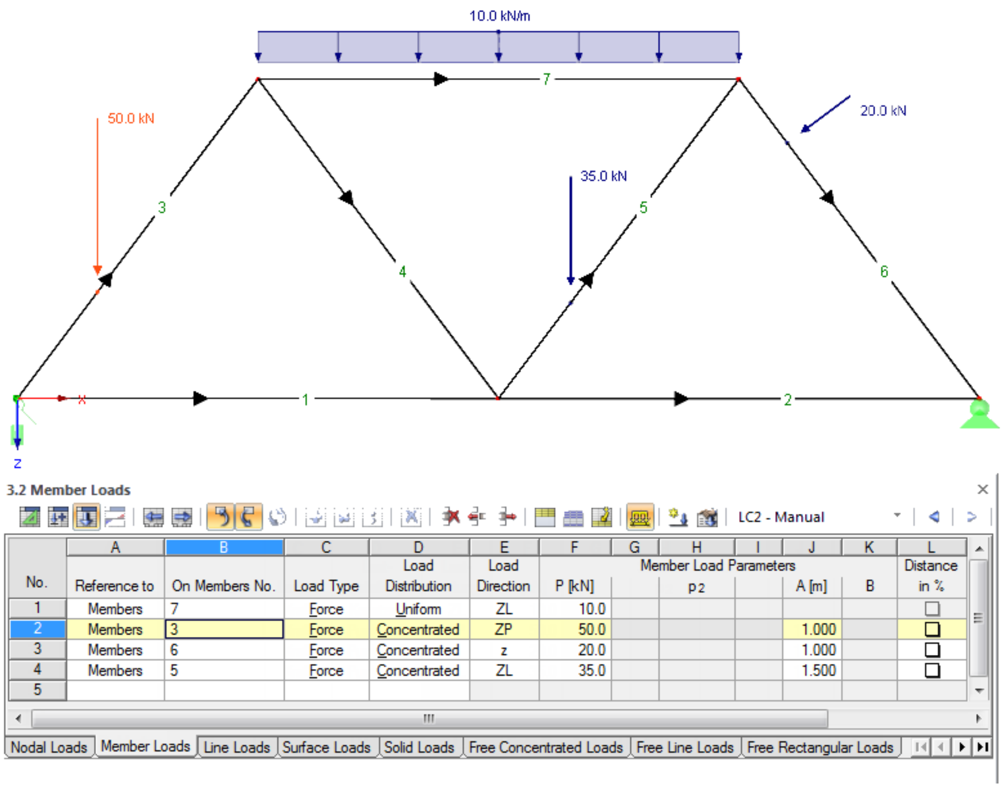
Tick this check box if you want to define the distances for concentrated and trapezoidal loads relative to the member length. Otherwise, the entries in the input fields for *Distance A / B* described above represent absolute ranges.

Load over total length of member

The check box can only be activated for trapezoidal loads. Select this option to arrange the application of the linearly variable load from the member start to the member end. The input fields *Load Parameters A / B* are no longer relevant and therefore disabled.

Example for member loads

The input of member loads is shown in the example below where member loads are applied to a planar frame structure. It is not necessary to divide members by intermediate nodes to apply concentrated loads.



No.	A Reference to	B On Members No.	C Load Type	D Load Distribution	E Load Direction	F P [kN]	G	H p2	I	J A [m]	K B	L Distance in %
1	Members	7	Force	Uniform	ZL	10.0						☐
2	Members	3	Force	Concentrated	ZP	50.0				1.000		☐
3	Members	6	Force	Concentrated	z	20.0				1.000		☐
4	Members	5	Force	Concentrated	ZL	35.0				1.500		☐
5												

Figure 6.18: Framework with uniform load on upper chord and concentrated loads on diagonals

6.3 Line Loads

General description

Line loads are forces and moments that act on lines (see chapter 4.2, page 49).

To apply a line load, a line must have been previously defined.

Line loads are similar to member loads. For line loads, however, it is not possible to allocate material properties (for example temperature loads or axial strains).

Line loads can act on members because a member is a property of a line. However, to apply a line load to a member, the line must belong to a surface. Consequently, line loads cannot be applied to members built in models consisting only of members.

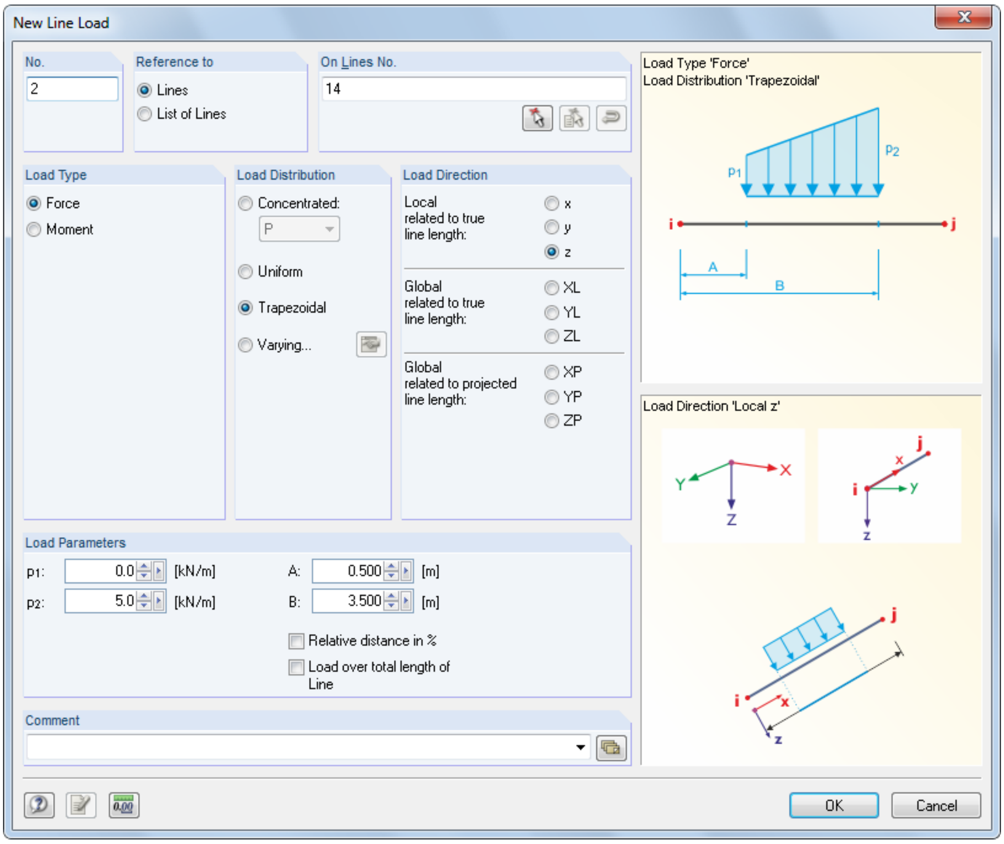


Figure 6.19: Dialog box *New Line Load*

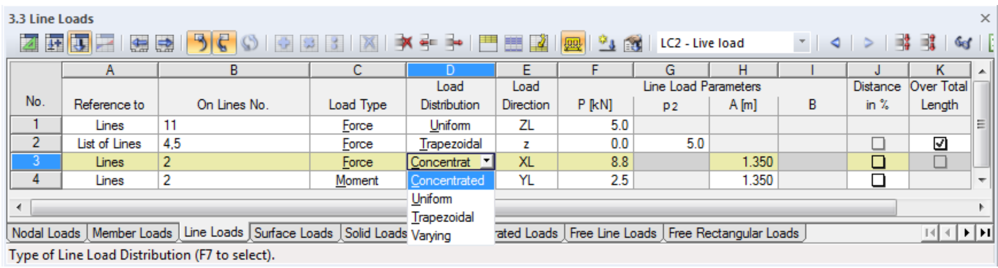
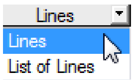


Figure 6.20: Table 3.3 *Line Loads*

The number of the line load is assigned automatically in the dialog box *New Line Load* but can be changed in the input field. The numbering order is not important.

Reference to



Define the objects to which you want to apply the line load. The following options can be selected:

Lines

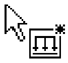
The load acts on one single line or on each line of several lines.

List of lines

The load acts on the union of lines that are defined in the list. Thus, when trapezoidal line loads are used, load parameters are not applied to each line individually but as total load to all lines of the line list (cf. Figure 6.16 on page 217).

On lines

In the input field, enter the numbers of the lines on which the load is acting. In the dialog box, you can select nodes also graphically by using the [↖] function.



When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter load data first. After clicking [OK] you can select the relevant lines one after the other in the work window.

Load type

In this dialog section or table column, you define the type of load. Depending on your selection, certain parts of the dialog box, respectively columns of the table, are disabled. The following load types can be selected:

Load type	**Short description**
Force	Concentrated, distributed, trapezoidal or variable load
Moment	Concentrated moment, distributed moment or trapezoidal moment

Table 6.4: Load types

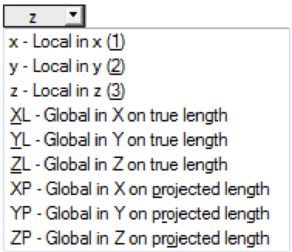
Load distribution

The dialog section *Load Distribution* offers different options to represent the effect of the load. The graphic in the dialog corner may help you to understand.

Load distribution	**Short description**
Concentrated	Concentrated load, concentrated moment
Uniform	Uniformly distributed load, uniformly distributed moment
Trapezoidal	Trapezoidal load, trapezoidal moment
Varying	Polygonally distributed load Click the button [Edit Varying Load] shown on the left to open the dialog box shown in Figure 6.17 on page 220 where you can enter the parameters of the load diagram.

Table 6.5: Load distributions

The load distributions of line loads largely correspond to the ones of member loads. The diagrams are described in detail in Table 6.2 on page 219.



Load direction

The load can be effective in direction of the global axes X, Y, Z or the local line axes x, y, z. For the calculation according to linear static analysis it does not matter whether a load is defined as local or equivalent global. For geometrically nonlinear calculations, however, differences between locally and globally defined loads are possible: If the load is defined with a global direction of action, it keeps this direction when finite elements start to twist. In case of a local direction of action, however, the load twists on the line according to the distortion of elements.

Local

The orientation of line axes is illustrated in Figure 4.96 on page 101. The local axis *x* represents the longitudinal axis of the line. Axis *z* is usually aligned parallel to the global axis Z.

Global

The position of the local line axes is irrelevant for the load input if the load acts in direction of an axis of the global coordinate system XYZ.

The load impact can be related to different reference lengths:

- **related to true line length**

  The load is applied to the entire line length.

- **related to projected line length**

  The application length of the load is converted to the projection of the line in one of the directions of the global coordinate systems. The projection lengths are shown in the dialog graphic to the right.

Line load parameters

In this dialog section, respectively table columns, the load values and, if applicable, additional parameters are managed. The input fields are labeled and accessible depending on the selection fields previously activated.

Load P / p / p_2 / M / m /m_2

Enter load values into the fields. Adjust the signs to the global or local orientations of axes. When a trapezoidal load is selected, specify two load values. The dialog graphic in the upper right corner shows the load parameters.

Distance A / B

In these two fields, enter the distances from the line start for concentrated and trapezoidal loads. You can define them also relatively to the line length by ticking the check box *Relative distance in %* (see below).

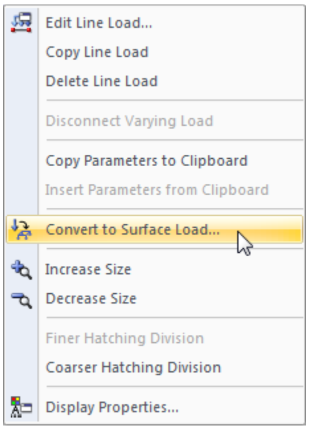
The dialog graphic in the upper right corner helps you when entering parameters.

Relative distance in %

Tick this check box to define the distances for concentrated and trapezoidal loads relative to the line length. Otherwise, the entries in the input fields for *Distance A / B* described above represent absolute ranges.

Load over total length of line

The check box can only be activated for trapezoidal loads. Select this option to arrange the application of the linearly variable load from the line start to the line end. The input fields *Load Parameters A / B* are no longer relevant and become unavailable.



Context menu of line load

Often, line loads result in singularities because the load is concentrated when introduced in a single line. To reduce this effect, select **Convert Nodal/Line Load to Surface Load** on the **Tools** menu. The option can be used for straight lines. You can also use the context menu of a line load shown on the left to access the dialog box for converting line loads. Open the context menu by a right-click on the object.

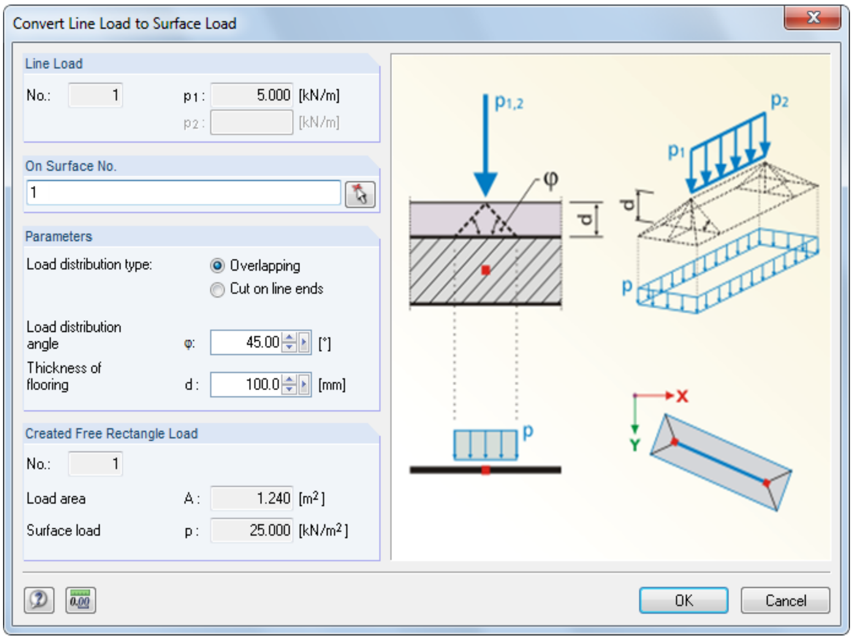


Figure 6.21: Dialog box *Convert Line Load to Surface Load*

A dialog box opens where you define the parameters for distributing the load. After clicking [OK], the corresponding free rectangular or polygonal load will be created.

6.4 Surface Loads

General description

Surface loads act on all 2D elements of a surface (see chapter 4.4, page 74).

To apply a surface load, a surface must have been previously defined.

If a surface is subdivided into surface components because of an intersection (see chapter 4.22, page 162), the surface load is not effective on components that are set inactive. Openings are omitted as well.

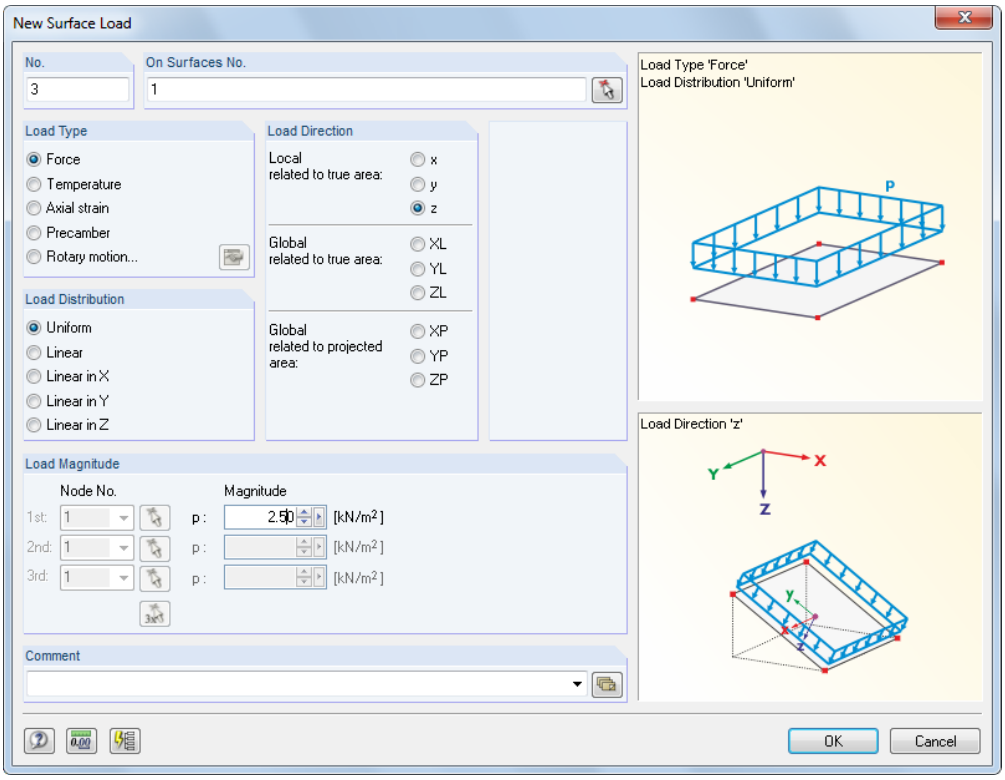


Figure 6.22: Dialog box *New Surface Load*

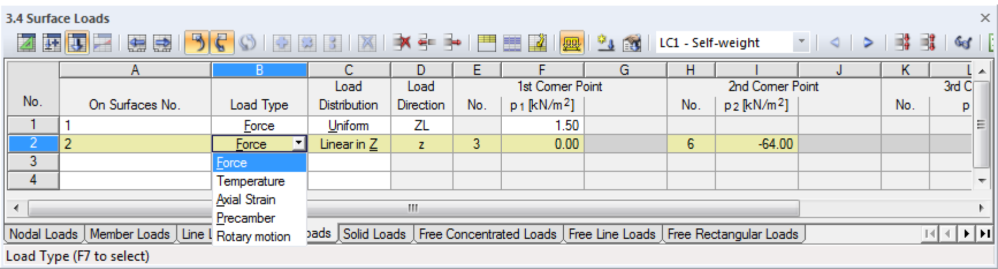
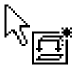


Figure 6.23: Table 3.4 *Surface Loads*

The number of the surface load is assigned automatically in the dialog box *New Surface Load* but can be changed in the input field. The numbering order is not important.

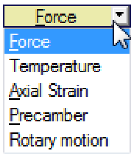
On surfaces

In this input field, define the numbers of the surfaces on which the load is acting. In the dialog box *New Surface Load*, you can select surfaces also graphically by using the [↖] function.



When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter load data first. After clicking [OK] you can select the relevant surfaces one after the other in the work window.

Load type



In this dialog section or table column, you define the type of load. Depending on your selection, certain parts of the dialog box respectively columns of the table are disabled. The following load types can be selected:

Load type	**Short description**
Force	Uniformly distributed or linearly variable force on surface
Temperature	Temperature load distributed as uniform or linearly variable over thickness of surface A positive load value means that the surface or its upper side is heating.
Axial strain	Imposed tensile or compressive strain ε of surface A positive load value means that the surface is extended. To determine the shrinkage strain, use the button shown on the left. The dialog box shown in Figure 6.24 opens where you can enter the parameters for shrinkage.
Precamber	Imposed curvature of surface
Rotary motion	Centrifugal force from mass and angular velocity ω on surface. The rotation axis can be defined in a separate dialog box that you open with the [Edit] button.

Table 6.6: Load types

The parameters for surface and member loads due to shrinkage can be defined in a separate dialog box.

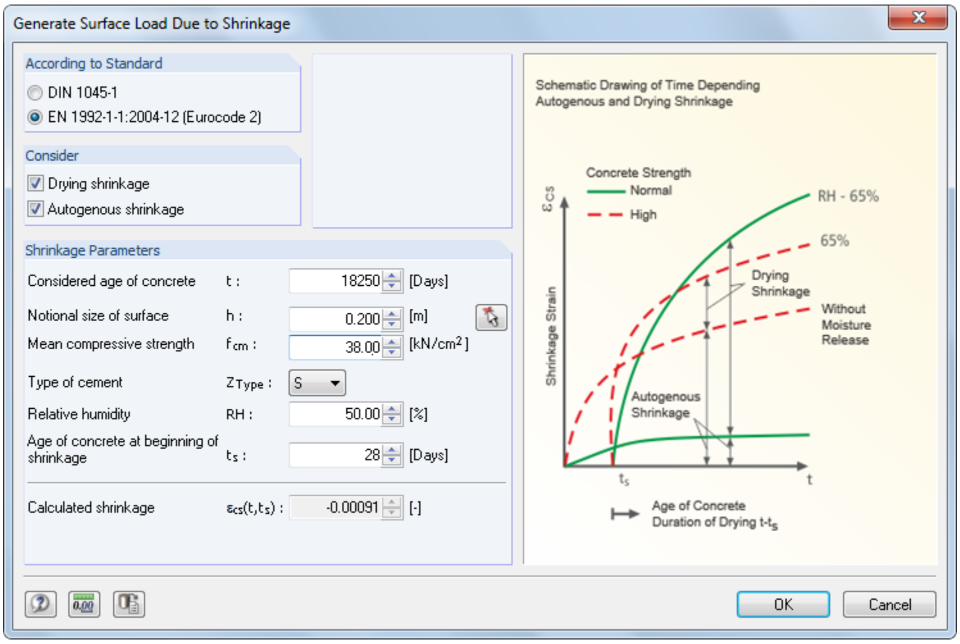


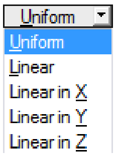
Figure 6.24: Dialog box *Generate Surface Load Due to Shrinkage*

Shrinkage as a time-dependent change in volume without external load action or effects of temperature can be classified in drying shrinkage, autogenous shrinkage, plastic shrinkage and carbonation shrinkage.

On the basis of the essential influencing variables in the shrinkage process (relative humidity *RH*, effective structural thickness *h*, concrete strength f_{cm}, type of cement Z_{Type}, age of concrete at beginning of shrinkage t_s) you determine the shrinkage $\varepsilon_{cs}(t,ts)$ at the moment of the considered concrete age *t*.

Click [OK] to transfer the value as axial strain ε to the dialog box *New Surface Load*.

Load distribution



The load can act on the surface as *Uniform* or *Linear* variable.

RFEM provides several options for linearly variable loads:

Linear

Define load values for three nodes. The nodes are used to define a plane.

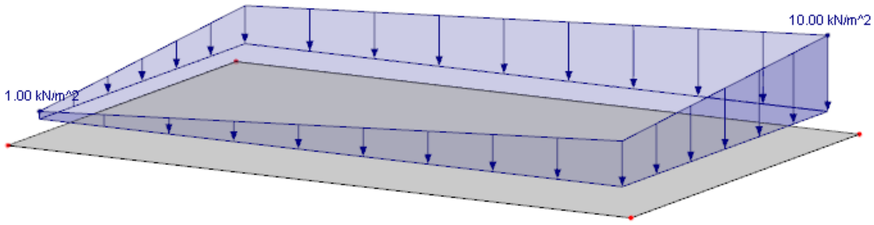


Figure 6.25: Linearly variable surface load

Linear in X / Y / Z

If the surface load is variable in direction of an axis of the global coordinate system, load values of only two nodes are required. They may lie outside of the stressed surface provided that FE nodes are generated there (nodes are not allowed to be free).

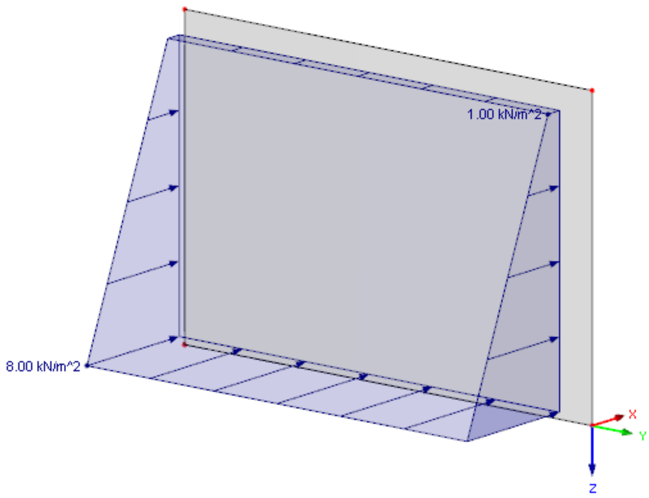
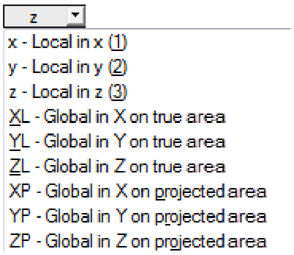


Figure 6.26: Surface load that is linearly variable in direction Z

Load direction



The load can be effective in direction of the local surface axes x, y, z or the global axes X, Y, Z.

Local related to true area

Loads acting perpendicular to the surface are usually defined as local in direction **z**. Examples of application are wind loads acting on roof surfaces or internal pressure on tank shells.

To display the surface axes, click **Model** in the *Display* navigator, select **Surfaces** and tick the check box for **Surface Axis Systems x,y,z**. You can also use the context menu of the surface (see Figure 4.115, page 116).

Global related to true area

The orientation of the local surface axes is irrelevant for the calculation according to linear static analysis if the load acts in direction of an axis of the global coordinate system XYZ. For nonlinear calculations, however, differences between locally and globally defined loads are possible: If the load is defined with a global direction of action, it keeps this direction when finite elements start to twist. In case of a local direction of action, however, the load twists according to the distortion of elements.

Global related to projected area

The load is converted to the projection of the surface in one of the directions of the global coordinate systems. Select this option to define for example a snow load on the projected ground-plan area of a roof.

The dialog graphic in the lower right corner shows the projected surfaces.

Surface load parameters

In this dialog section or table columns, the load values and, if applicable, the assigned nodes are managed. The input fields are labeled and accessible depending on the selection fields previously activated.

Load p / p_2 / p_3 / T / ΔT / ε / R / ω / α

Enter load values into the fields. Adjust the signs to the global or local orientations of axes.

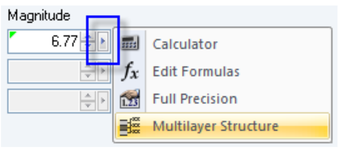
When a linearly variable load is selected, specify several load values. The dialog graphic in the upper right corner shows the load parameters.

Nodes

When a linearly variable load is selected, specify three nodes on which the load magnitudes can be determined. The nodes are used to define a plane. In the dialog box, you can select the nodes also graphically by using the [↖] function.

Load of multilayer structure

It is possible to create loads from area weights of materials acting as laminated layers. In this way, you can easily determine for example the structure of floorings or floor coverings.



Find the function in the dialog box *New Surface Load* (Figure 6.22) for access use the button [▶] to the right of the input field *Magnitude*). In the context menu, select *Multilayer Structure*.

The *Multilayer Library* opens where you can enter user-defined material layers.

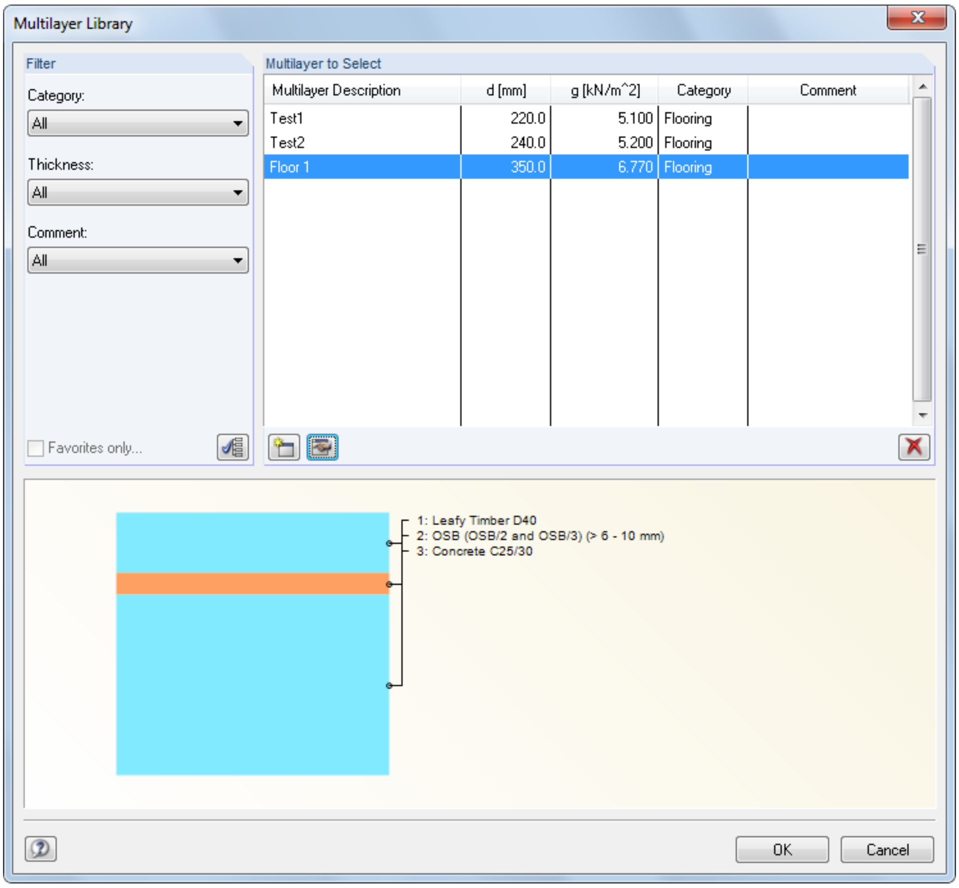


Figure 6.27: Dialog box *Multilayer Library*

The concept of the multilayer database is similar to the material library (see chapter 4.3, page 71). Use the library buttons [New] and [Edit] to create or modify multi-layer structures.

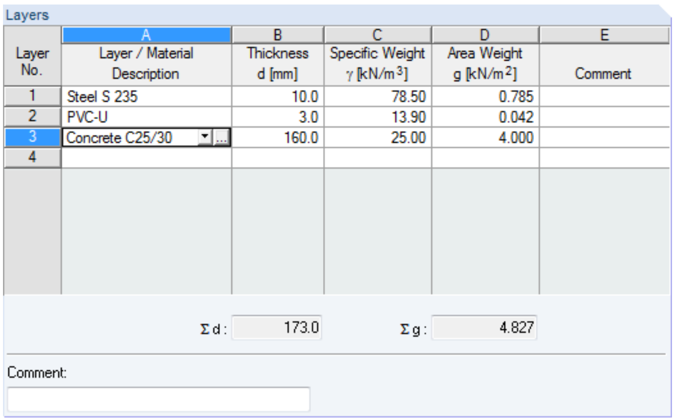


Figure 6.28: Dialog box *New Multilayer*, dialog section *Layers*

The *Layers* can be composed individually. Moreover, you can use the button [...] to access the material library (see chapter 4.3, page 71).

RFEM determines the area weight (table column D) from *Thickness* and *Specific Weight*. An arrow shown in the dialog graphic indicates the current layer.

Confirm all dialog boxes with [OK] to import the area weight into the initial dialog box. A green triangle appears in the input field (see graphic shown on the left on page 230), indicating the parameterized input value. Click the triangle to access again the input parameters for modifications.

6.5 Solid Loads

General description

Solid loads act on all 3D elements of a solid (see chapter 4.5, page 85).

To apply a solid load, a solid must have been previously defined.

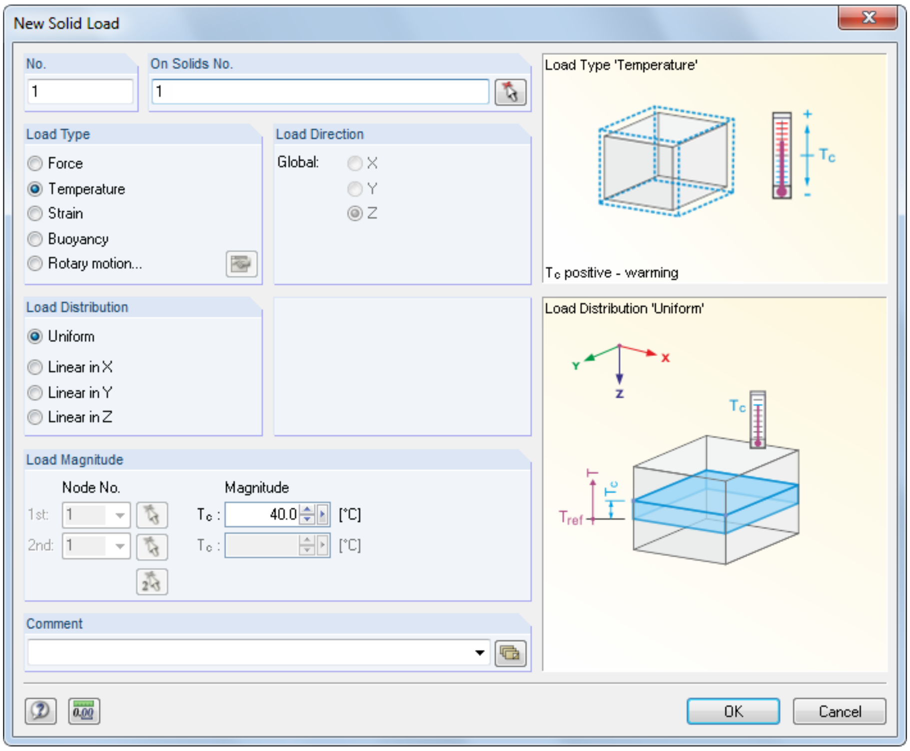
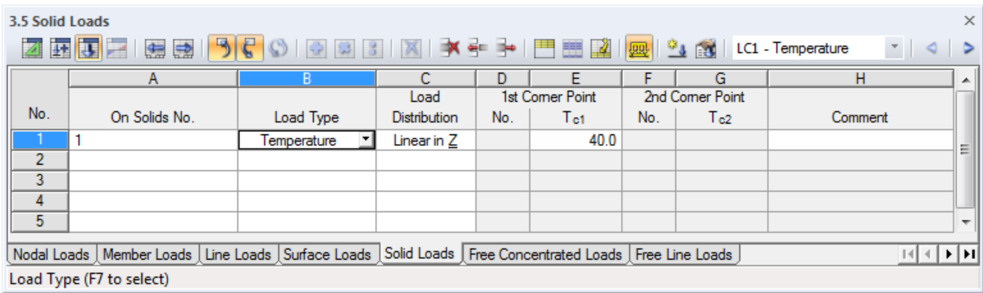


Figure 6.29: Dialog box *New Solid Load*



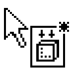
	A	B	C	D	E	F	G	H
No.	On Solids No.	Load Type	Load Distribution	1st Corner Point No.	1st Corner Point T_{c1}	2nd Corner Point No.	2nd Corner Point T_{c2}	Comment
1	1	Temperature	Linear in Z		40.0			
2								
3								
4								
5								

Figure 6.30: Table 3.5 *Solid Loads*

The number of the solid load is assigned automatically in the dialog box *New Solid Load* but can be changed in the input field.

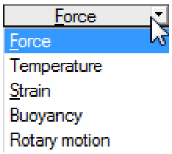
On solids

In the input field, enter the numbers of the solids on which the load is acting. In the dialog box *New Solid Load*, you can select solids also graphically by using the [↖] function.



When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter load data first. After clicking [OK] you can select the relevant solids one after the other in the work window.

Load type



In this dialog section or table column, you define the type of load. The following load types can be selected:

Load type	Short description
Force	Solid load acting uniformly in one of global directions
Temperature	Uniformly distributed or linearly variable temperature change in solid. A positive load value signifies a heating.
Strain	Imposed tensile or compressive strain of solid that is uniformly distributed or linearly variable. A positive load value means that the solid is extended.
Buoyancy	Weight of pushed away material whose density can be entered or selected in a [Library]. The *Environment Density* of *Air* refers to a standard atmosphere of 15°C on sea level.
Rotary motion	Centrifugal force from mass and angular velocity ω on solid. The rotation axis can be defined in a separate dialog box that you open with the [Edit] button.


Table 6.7: Load types

More forces can be applied on a solid in the form of surface or line loads.

Load distribution

The load can act on the solid as *Uniform* or *Linear* variable. It refers to one of the global axis X, Y or Z.

When linearly variable loads are selected, specify the load values of two nodes. Nodes are allowed to lie outside of the stressed solid provided that FE nodes are generated there.

Load magnitude

In this dialog section or table columns, the load values and, if applicable, the assigned nodes are managed. The input fields are labeled and accessible depending on the selection fields previously activated.

Nodes

When linearly variable loads are selected, specify two nodes on which RFEM can determine the magnitudes. The nodes are used to define a plane. In the dialog box, you can select nodes also graphically by using the [↖] function.

Magnitude

For a uniform load distribution, only one numerical value is required. For a linearly variable change in temperature or axial strain, specify two load values.

The graphics in the dialog box *New Solid Load* are useful when entering load parameters.

6.6 Free Concentrated Loads

General description

A free concentrated load acts as force or moment on any location of the surface. No FE nodes will be generated on the point of load application.

To apply a free concentrated load, a surface must have been previously defined.

Nodal support forces that have been imported from another model by using the function *Import Support Reactions as Load* (see Figure 8.14, page 289) are handled as free concentrated loads.

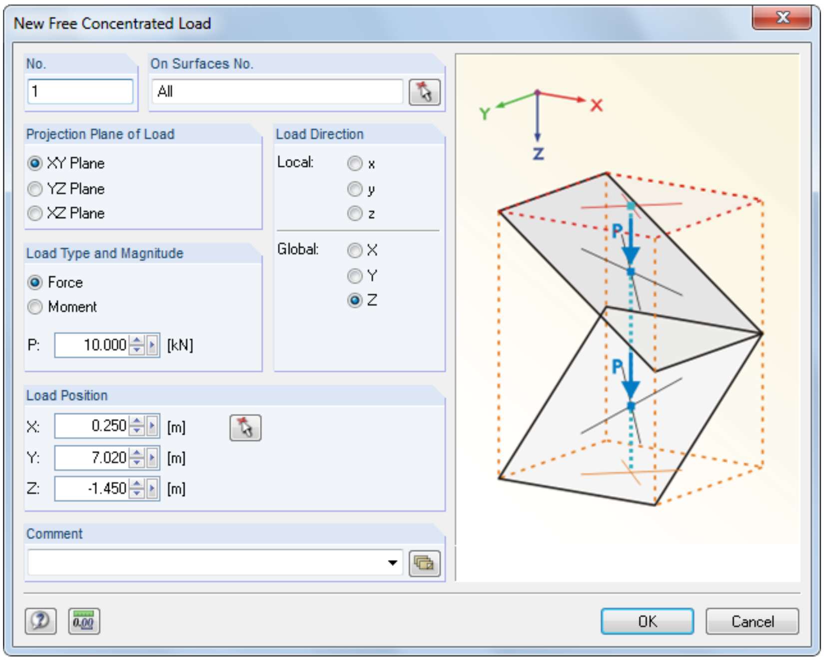
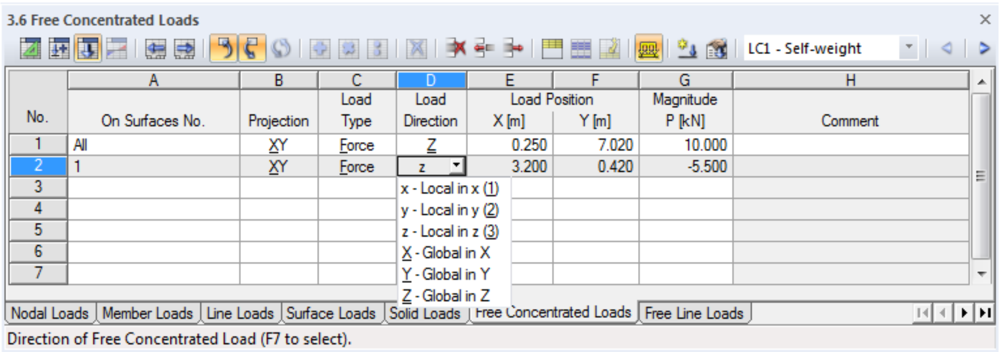


Figure 6.31: Dialog box *New Free Concentrated Load*



No.	A On Surfaces No.	B Projection	C Load Type	D Load Direction	E Load Position X [m]	F Load Position Y [m]	G Magnitude P [kN]	H Comment
1	All	XY	Force	Z	0.250	7.020	10.000	
2	1	XY	Force	z	3.200	0.420	-5.500	
3								
4								
5								
6								
7								

Figure 6.32: Table 3.6 *Free Concentrated Loads*

On surfaces

In the input field, enter the numbers of the surfaces on which the load is acting. In the dialog box *New Free Concentrated Load*, you can select surfaces also graphically by using the [↖] function.

Starting from the defined *Load Position*, an imaginary straight line is "set" perpendicular to the projection plane. When the line intersects any of the listed surfaces, the concentrated load is applied to the point of intersection. In this way, it is possible to allocate quickly loads of similar type to several surfaces.

Projection plane

The load can be projected on one of the global planes XY, YZ or XZ. As described above, an imaginary line is generated, starting from the load position and running perpendicular to the projection plane. The load is applied wherever the line intersects a surface.

The projection plane must not be perpendicular to a surface on which the load is acting: There is no clear intersection point with the surface.

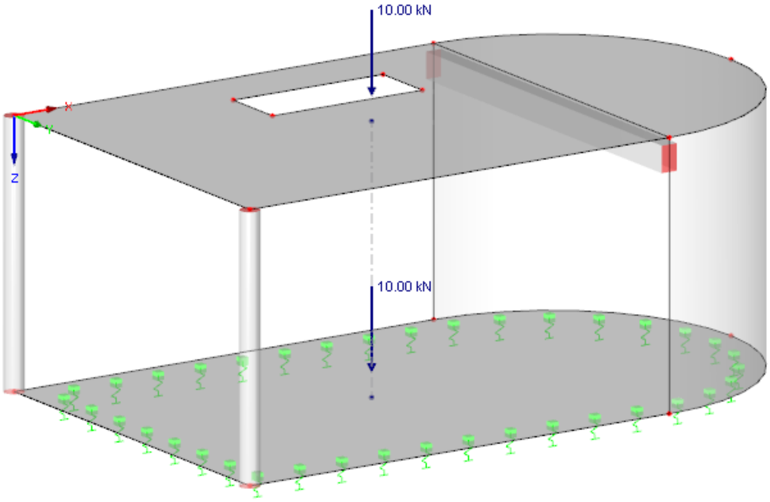


Figure 6.33: Free concentrated load acting on ceiling: load projection plane XY with impact on ceiling slab and floor slab

Load type

Specify whether a single force or concentrated moment is applied. In the dialog input field below, you can enter a numerical value of the load.

Load direction

The load can act in direction of the global axes X, Y, Z or the local surface axes x, y, z. To display the axes, use the context menu (see Figure 4.115, page 116) or the *Display* navigator where you select **Model** → **Surfaces** → **Surface Axis Systems x,y,z**.

Load position

Enter the coordinates of the load position into the input fields. In the dialog box, you can select the position of the load also graphically by using the [↖] function.

Magnitude

In the table column respectively input field, enter the numerical value of the concentrated force or moment.

6.7 Free Line Loads

General description

A free line load acts as a uniform or linearly variable force along a freely definable line of a surface. No FE nodes will be generated along this line.

To apply a free line load, a surface must have been previously defined.

Line support forces that have been imported from another model by using the function *Import Support Reactions as Load* (see Figure 8.14, page 289) are handled as free line loads.

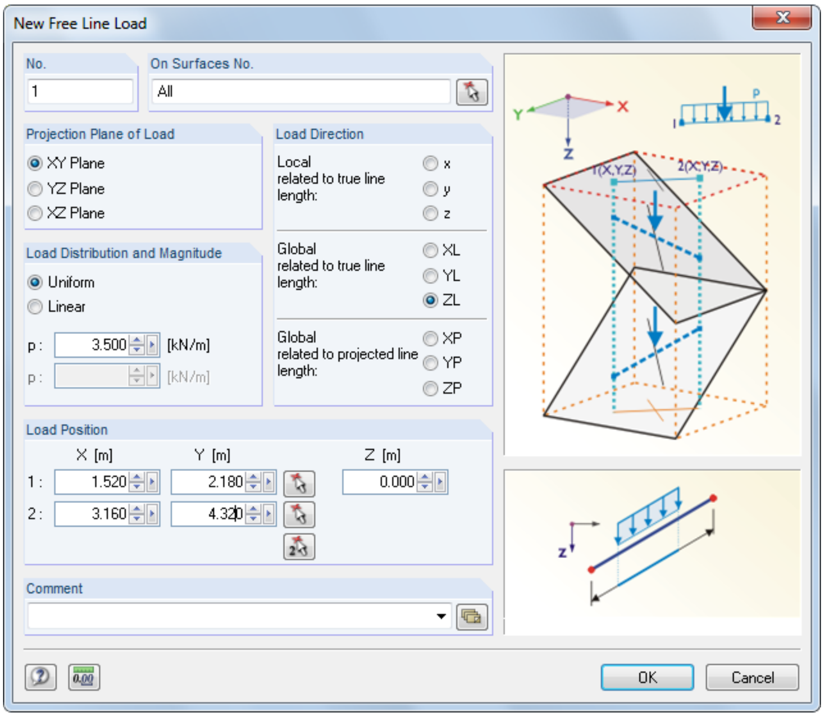
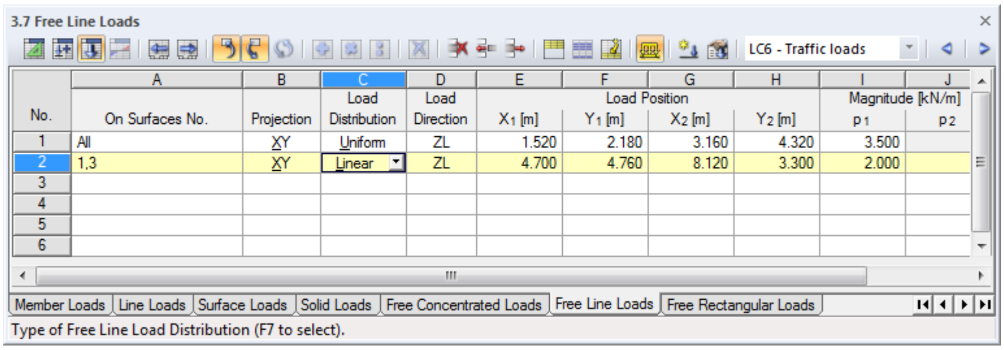


Figure 6.34: Dialog box *New Free Line Load*


3.7 Free Line Loads

LC6 - Traffic loads

No.	A On Surfaces No.	B Projection	C Load Distribution	D Load Direction	E Load Position X1 [m]	F Load Position Y1 [m]	G Load Position X2 [m]	H Load Position Y2 [m]	I Magnitude [kN/m] p1	J Magnitude [kN/m] p2
1	All	XY	Uniform	ZL	1.520	2.180	3.160	4.320	3.500	
2	1,3	XY	Linear	ZL	4.700	4.760	8.120	3.300	2.000	
3										
4										
5										
6										

Member Loads | Line Loads | Surface Loads | Solid Loads | Free Concentrated Loads | Free Line Loads | Free Rectangular Loads

Type of Free Line Load Distribution (F7 to select).

Figure 6.35: Table 3.7 *Free Line Loads*

On surfaces

This input field manages the numbers of surfaces on which the load is acting. In the dialog box *New Free Line Load*, you can select surfaces also graphically by using the [↖] function.

Starting from both points defined by the *Load Position*, two imaginary straight lines are "set" perpendicular to the projection plane. When the lines intersect any of the listed surfaces, the load is applied to the connecting line of both intersection points. In this way, it is possible to allocate quickly loads of similar type to several surfaces.

Projection plane

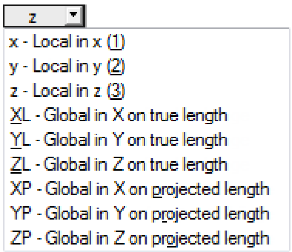
The load can be projected on one of the global planes XY, YZ or XZ. As described above, two imaginary lines are generated, starting from both load positions and running perpendicular to the projection plane. The start and end points of the free line load are assumed wherever the lines intersect a surface.

The projection plane must not be perpendicular to a surface on which the load is acting: There are no clear intersection points with the surface.

Load distribution

Specify whether a uniform or linearly variable force is applied. In the dialog input field below, you can enter one or two numerical values.

Load direction



The load can act in direction of the local surface axes x, y, z or the global axes X, Y, Z. Loads acting perpendicular to the surface are generally to be defined as local in direction **z**.

If a globally acting load is not running perpendicular to the line, the load impact can be related to different reference lengths:

- **related to true line length**

  The load is applied to the entire line length.

- **related to projected line length**

  The application length of the load is converted to the projection of the line in one of the directions of the global coordinate systems. The projection lengths are shown in the dialog graphic in the lower right corner.

Load position

Enter the coordinates of the load position into the input fields. In the dialog box, you can select the position of the load also graphically by using the [↖] function.

Magnitude

In the table column respectively input field, enter the numerical value of the line load.

6.8 Free Rectangular Loads

General description

A free rectangular load acts as a uniform or linearly variable surface load on a rectangular, freely definable zone of a surface.

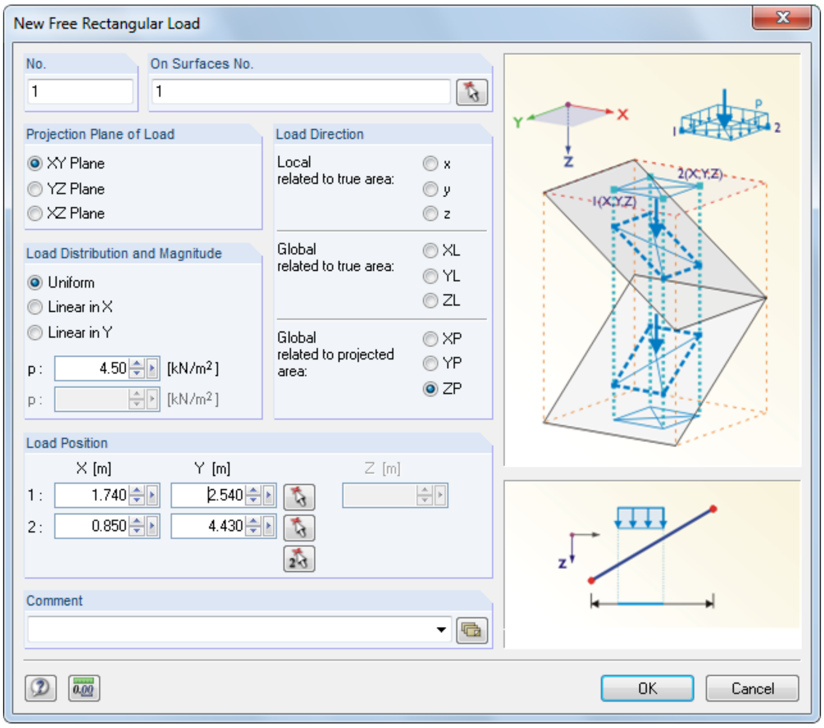


Figure 6.36: Dialog box *New Free Rectangular Load*

3.8 Free Rectangular Loads

LC6 - Traffic loads

No.	A On Surfaces No.	B Projection	C Load Distribution	D Load Direction	E Load Position X_1 [m]	F Y_1 [m]	G X_2 [m]	H Y_2 [m]	I Magnitude [kN/m²] p_1	J p_2
1	1	XY	Uniform	ZP	1.740	2.540	0.850	4.430	4.500	
2	1	XY	Linear in X	z	3.410	3.200	4.450	4.100	2.000	5.500
3										
4										
5										
6										

Line Loads | Surface Loads | Solid Loads | Free Concentrated Loads | Free Line Loads | Free Rectangular Loads | Free Circular Loads

Direction of Free Rectangle Load (F7 to select).

Figure 6.37: Table 3.8 *Free Rectangular Loads*

On surfaces

This input field manages the numbers of surfaces on which the load is acting. In the dialog box *New Free Rectangular Load*, you can select surfaces also graphically by using the [↖] function.

Starting from both points defined by the *Load Position*, two imaginary straight lines are "set" perpendicular to the projection plane. When the lines intersect any of the listed surfaces, the load is applied to the connecting diagonal of both intersection points. In this way, it is possible to allocate quickly loads of similar type to several surfaces.

Projection plane

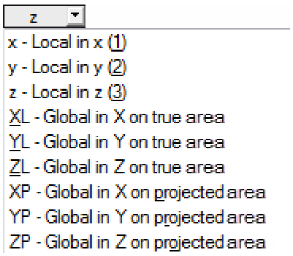
The load can be projected on one of the global planes XY, YZ or XZ. As described above, two imaginary lines are generated, starting from both load positions and running perpendicular to the projection plane. The start and end points of the rectangle's diagonal are assumed wherever the lines intersect a surface.

The projection plane must not be perpendicular to a surface on which the load is acting: There are no clear intersection points with the surface.

Load distribution

Specify whether a uniform or linearly variable load is applied. In the dialog input field below, you can enter one or two numerical values.

Load direction



The load can act in direction of the global axes X, Y, Z or the local surface axes x, y, z.

- **Local related to true area**

  Loads acting perpendicular to the surface are usually defined as local in direction **z**.

- **Global related to true area**

  The orientation of the local surface axes is irrelevant for the calculation according to linear static analysis if the load acts in direction of an axis of the global coordinate system XYZ. An example for load reference to the true area is self-weight.

- **Global related to projected area**

  The load is converted to the projection of the surface in one of the directions of the global coordinate systems. A case for application is for example snow load. The dialog graphic in the lower right corner shows the projected surfaces.

Load position

Enter the coordinates of the load position into the input fields. In the dialog box, you can select the position of the load also graphically by using the [↖] function.

Magnitude

In the table column respectively input field, enter the numerical value of the area load.

6.9 Free Circular Loads

General description

A free circular load acts as a uniform or linearly variable surface load on a circular, freely definable part of a surface.

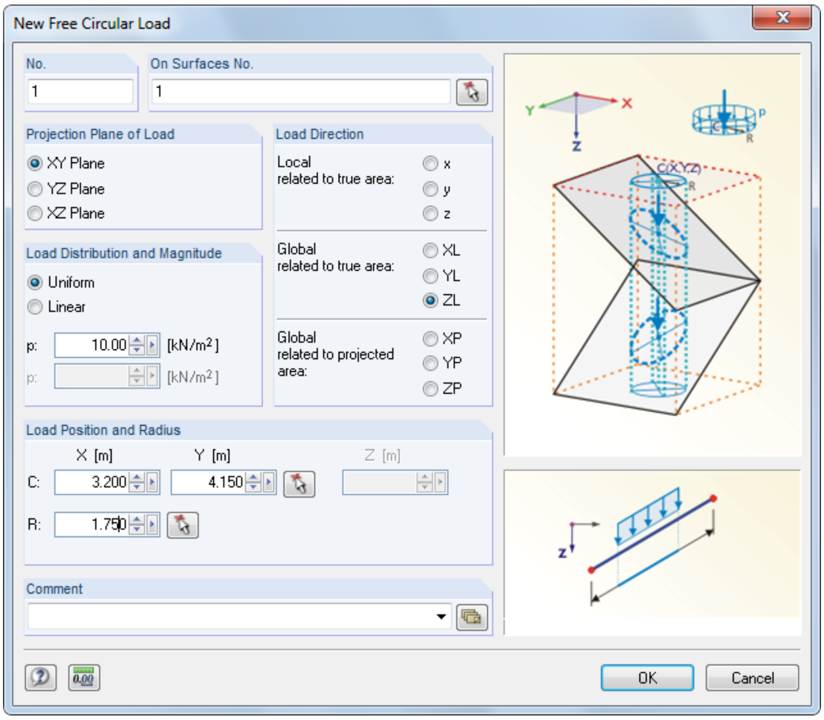


Figure 6.38: Dialog box *New Free Circular Load*

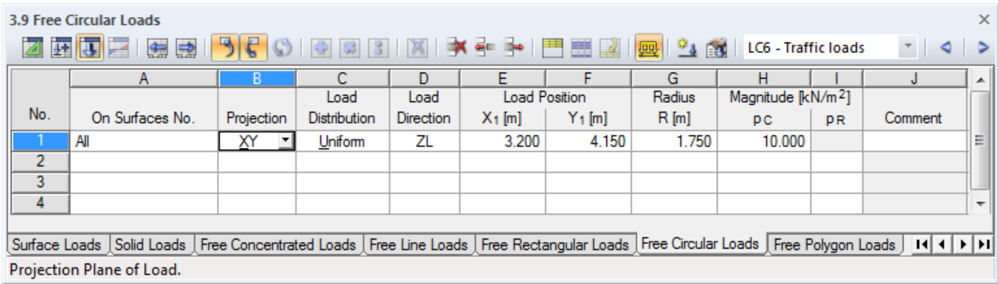


Figure 6.39: Table 3.9 *Free Circular Loads*

On surfaces

This input field manages the numbers of surfaces on which the load is acting. In the dialog box *New Free Circular Load*, you can select surfaces also graphically by using the [↖] function.

Starting from the defined *Load Position*, an imaginary straight line is "set" perpendicular to the projection plane. When the line intersects any of the listed surfaces, the circular load is applied to the point of intersection representing the circle center with the radius *R*. In this way, it is possible to allocate quickly loads of similar type to several surfaces.

Projection plane

The load can be projected on one of the global planes XY, YZ or XZ. As described above, an imaginary line is generated, starting from the load position and running perpendicular to the projection plane. The center of the circular load is assumed wherever the line intersects a surface.

The projection plane must not be perpendicular to a surface on which the load is acting: There are no clear intersection points with the surface.

Load distribution

Specify whether a uniform or linearly variable load is applied. In the dialog input field below, you can enter one or two numerical values.

Load direction

The load can act in direction of the global axes X, Y, Z or the local surface axes x, y, z. The load directions are described in the previous chapter 6.8 on page 239.

Load position

Into the input fields, enter the coordinates of midpoint *C* of the circle load. In the dialog box, you can select the circle center also graphically by using the [↖] function.

Radius

Enter the radius *R* of the circular area load into the input field or table column. In the dialog box, you can select the radius also graphically in the work window by using the [↖] function.

Magnitude

In the table column respectively input field, enter the numerical value of the area load.

6.10 Free Polygon Loads

General description

A free polygon load acts as a uniform or linearly variable surface load on a polygonal, freely definable zone of a surface.

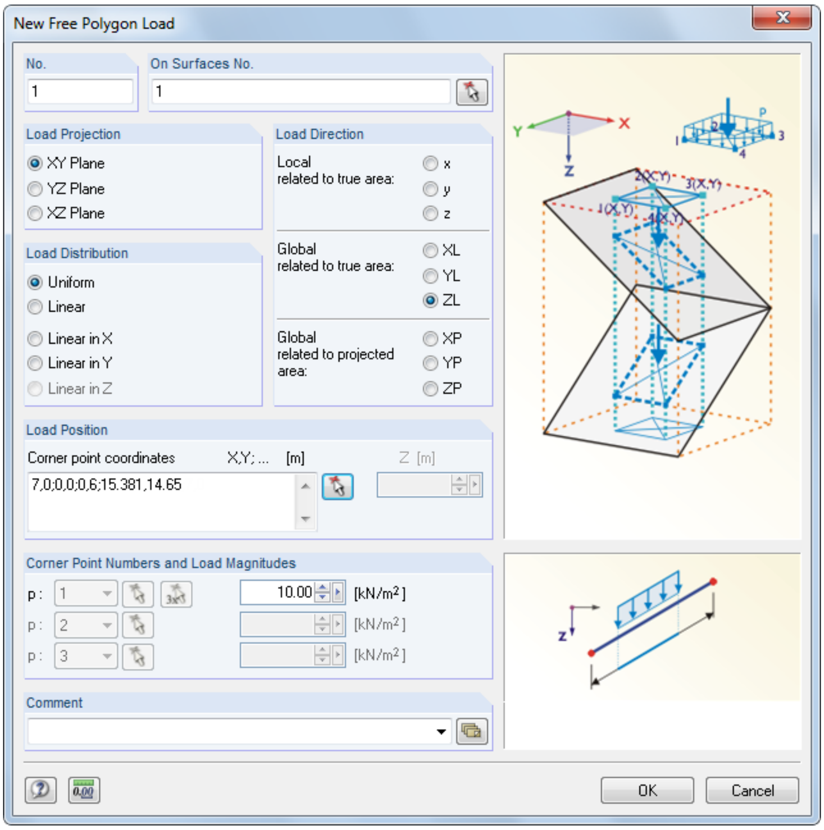
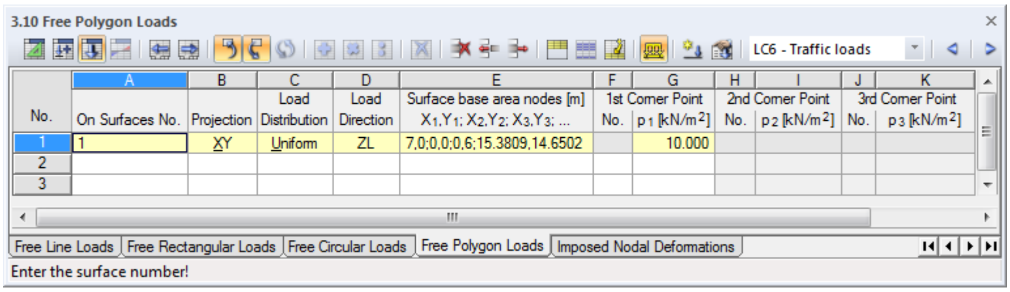


Figure 6.40: Dialog box *New Free Polygon Load*



	A	B	C	D	E	F	G	H	I	J	K
No.	On Surfaces No.	Projection	Load Distribution	Load Direction	Surface base area nodes [m] X1,Y1; X2,Y2; X3,Y3; ...	1st Corner Point No.	1st Corner Point p1 [kN/m2]	2nd Corner Point No.	2nd Corner Point p2 [kN/m2]	3rd Corner Point No.	3rd Corner Point p3 [kN/m2]
1	1	XY	Uniform	ZL	7,0;0,0;0,6;15.3809,14.6502		10.000				
2											
3											

Figure 6.41: Table 3.10 *Free Polygon Loads*

On surfaces

This input field manages the numbers of surfaces on which the load is acting. In the dialog box *New Free Polygon Load*, you can select surfaces also graphically by using the [↖] function.

Starting from the corner points defined by the *Load Position*, imaginary straight lines are "set" perpendicular to the projection plane. When they intersect any of the listed surfaces, the line connecting the points of intersection represents the boundary of the area load to be applied. In this way, it is possible to allocate quickly loads of similar type to several surfaces.

Load projection

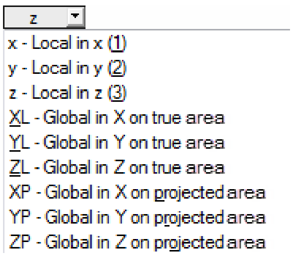
The load can be projected on one of the global planes XY, YZ or XZ. As described above, two imaginary lines are generated, starting from the load positions and running perpendicular to the projection plane. The corner points of the polygon load are assumed wherever the lines intersect a surface.

The projection plane must not be perpendicular to a surface on which the load is acting: There are no clear intersection points with the surface.

Load distribution

Specify whether a uniform or linearly variable area load is applied. In the dialog section *Corner Point Numbers and Load Magnitudes* below, you can enter one (*Uniform*), two (*Linear in X / Y / Z*) or three (*Linear*) numerical values.

Load direction



The load can act in direction of the global axes X, Y, Z or the local surface axes x, y, z. The load directions are described in chapter 6.8 on page 239.

Load position

Enter the *Corner point coordinates* into the input fields. It is recommended to define the polygonal chain graphically by using the dialog function [↖]. In the input field respectively table column, enter the point coordinates separated by comma. Coordinate pairs are separated from one another by semicolon.

Example: corner point coordinates X,Y [m] 2,3;1.6,4.7;5,45;6,25;3,2

The third coordinate component is defined automatically by the plane of load projection. In the example, it is the Z-coordinate that can be specified separately in an input field of the dialog box. When defining coordinates graphically, the component is irrelevant because the active work plane is decisive.

Corner point numbers

When defining linearly variable area loads, enter two (*Linear in X / Y / Z*) or three (*Linear*) corner points with the corresponding load values. You can define or graphically select only those corner points which are listed in the dialog section *Load Position* used to define the boundary lines. It is not possible to select RFEM nodes. Thus, the numbers of corner points are related to the sequence of the *Corner point coordinates*.

Magnitude

In the table column respectively input field, enter the numerical value of the area load. For a linearly variable distribution, two or three values must be entered.

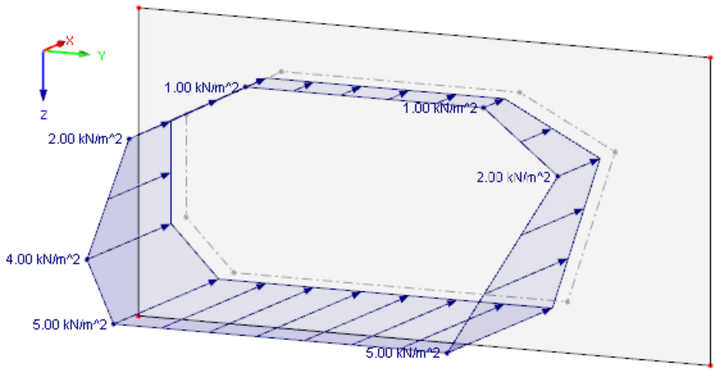


Figure 6.42: Polygon load that is linearly variable in direction **Z**: load projection plane **YZ** and local load direction in **z**

6.11 Imposed Nodal Deformations

General description

An imposed nodal deformation is a displacement or rotation of a supported node, for example due to a column settlement.

Imposed nodal deformations can only be applied to nodes that have a support in the direction of the deformation.

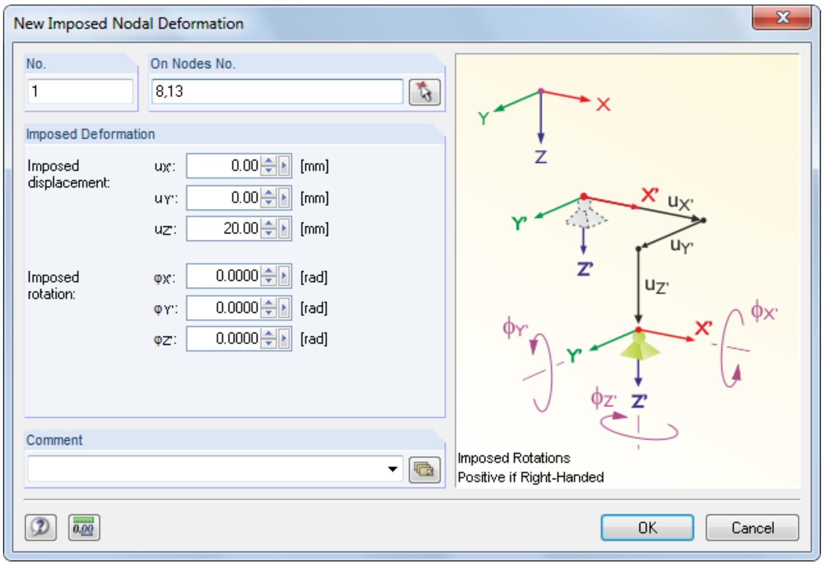
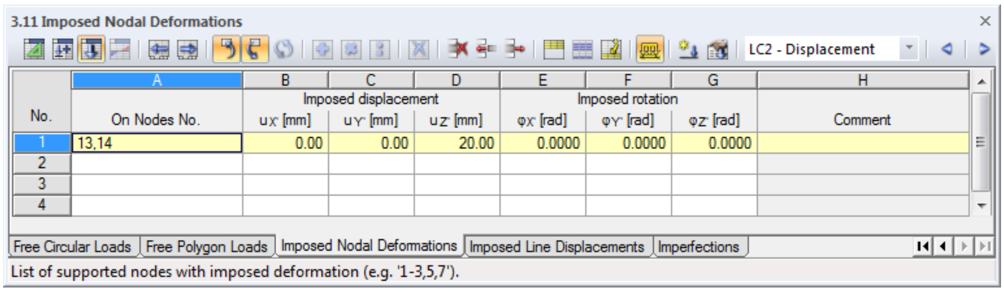


Figure 6.43: Dialog box *New Imposed Nodal Deformation*



	A	B	C	D	E	F	G	H
		Imposed displacement			Imposed rotation			
No.	On Nodes No.	$u_{X'}$ [mm]	$u_{Y'}$ [mm]	$u_{Z'}$ [mm]	$\varphi_{X'}$ [rad]	$\varphi_{Y'}$ [rad]	$\varphi_{Z'}$ [rad]	Comment
1	13,14	0.00	0.00	20.00	0.0000	0.0000	0.0000	
2								
3								
4								

Figure 6.44: Table 3.3 *Imposed Nodal Deformations*

The number of the load is assigned automatically in the dialog box *New Imposed Nodal Deformation* but can be changed in the input field.

On nodes

In this input field, define the numbers of nodes on which the imposed deformation is acting. In the dialog box, you can select nodes also graphically by using the [↖] function.

When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter the deformations first. After clicking [OK] you can select the relevant nodes one after the other in the work window.

Imposed displacement $u_{X'}$ / $u_{Y'}$ / $u_{Z'}$

Imposed displacements refer to the global coordinate system. If a displacement of a supported node does not act parallel to one of the global axes, its components X, Y and Z must be determined and entered in the corresponding input fields.

The graphic in the dialog box explains how displacements and signs are effective.

Imposed rotation $\varphi_{X'}$ / $\varphi_{Y'}$ / $\varphi_{Z'}$

Node rotations refer to the global coordinate system X,Y,Z as well. Therefore, a skew imposed rotation requires the division in X, Y and Z components.

A positive imposed rotation acts clockwise about the corresponding positive global axis.

6.12 Imposed Line Displacements

General description

An imposed line displacement is a displacement of a supported line, for example due to a foundation settlement.

Imposed line displacements can only be applied to lines that are supported in the direction of the displacement.

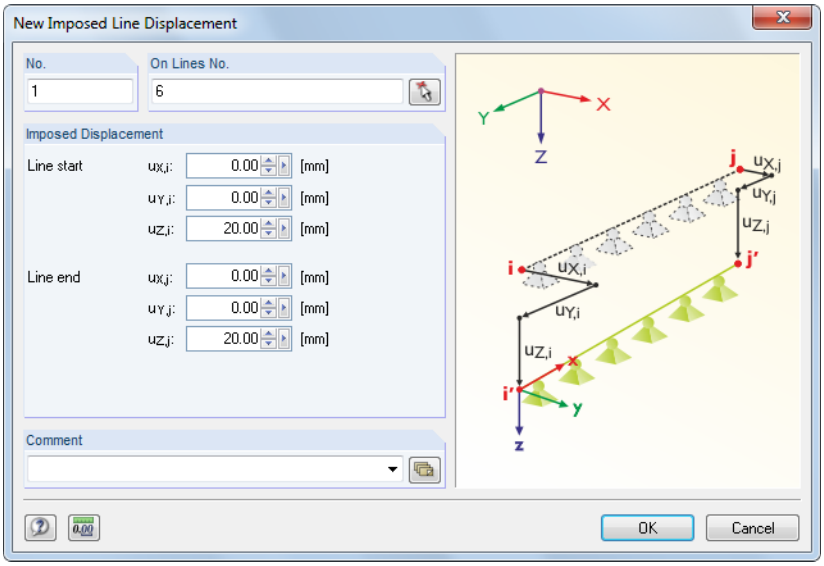
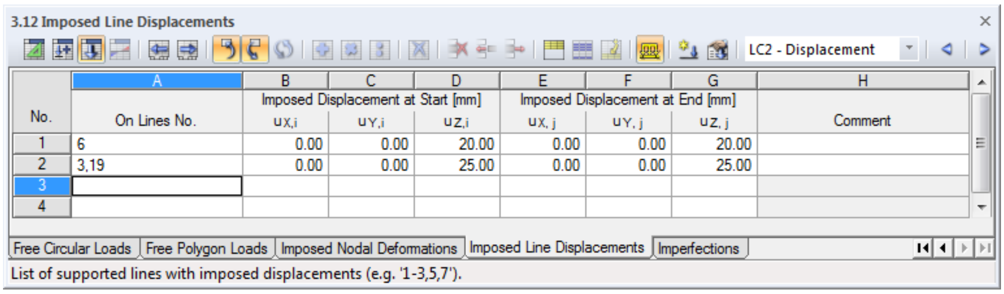


Figure 6.45: Dialog box *New Imposed Line Displacement*



3.12 Imposed Line Displacements

No.	A	B	C	D	E	F	G	H
		Imposed Displacement at Start [mm]			Imposed Displacement at End [mm]			
	On Lines No.	uX,i	uY,i	uZ,i	uX,j	uY,j	uZ,j	Comment
1	6	0.00	0.00	20.00	0.00	0.00	20.00	
2	3,19	0.00	0.00	25.00	0.00	0.00	25.00	
3								
4								

Free Circular Loads | Free Polygon Loads | Imposed Nodal Deformations | Imposed Line Displacements | Imperfections

List of supported lines with imposed displacements (e.g. '1-3,5,7').

Figure 6.46: Table 3.12 *Imposed Line Displacements*

The number of the load is assigned automatically in the dialog box *New Imposed Line Displacement* but can be changed in the input field.

On lines

In this input field, define the numbers of lines on which the imposed displacement is acting. In the dialog box, you can select nodes also graphically by using the [↖] function.

When the graphical input is selected, the input field is disabled and you have to enter the displacements first. After clicking [OK] you can select the relevant lines one after the other in the work window.

Imposed displacement u_X / u_Y / u_Z

Line start

Imposed displacements refer to the global coordinate system X,Y,Z. Three input fields are provided for the displacement on the start node of the supported line.

If a displacement of the supported line does not act parallel to one of the global axes, its X-, Y- and Z-components must be determined.

Line end

In the input fields, enter the displacement on the end node of the supported line.

To visualize the line orientation, use the *Display* navigator: Select **Model** → **Lines** → **Line Orientations** (see Figure 4.26, page 50).

6.13 Imperfections

General description

There are two ways how imperfections can be determined in RFEM:

- **Equivalent loads** are applied to members.
- A pre-deformed **equivalent model** is used.

This chapter describes imperfections in the form of equivalent loads. For detailed information on how to generate equivalent models using the add-on module **RF-IMP**, see chapter 7.3.1 on page 269.

To apply an imperfection, a member must have been previously defined.

Imperfections represent manufacturing deviations in model geometry and material properties. In EN 1993-1-1, clause 5.3, the application of imperfections is organized as precamber (deflection) and inclination (sway). Thus, imperfections are taken into account by equivalent loads.

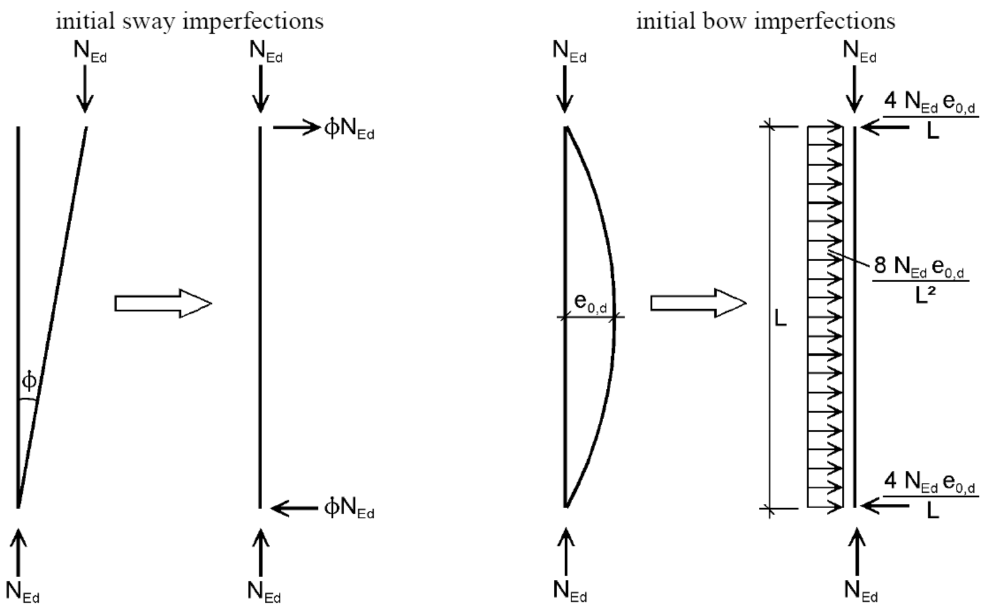


Figure 6.47: Equivalent loads according to EN 1993-1-1

Equivalent loads are also taken into account by RFEM when calculations are performed according to linear static analysis. Please note, however, that a pure imperfection load case will not produce any internal forces. The model must additionally have some “real” loads inducing axial forces in the imperfect member.

It is recommended to manage loads and imperfections in separate load cases. They can be combined with each other appropriately in load combinations. Load cases with pure imperfections must be categorized as action type **Imperfection** in the base data for load cases (see Figure 5.3, page 170). Otherwise, the plausibility check would display a message because of missing loads.

Generally, imperfections must be set affine with the lowest buckling eigenvalue in the most unfavorable direction.

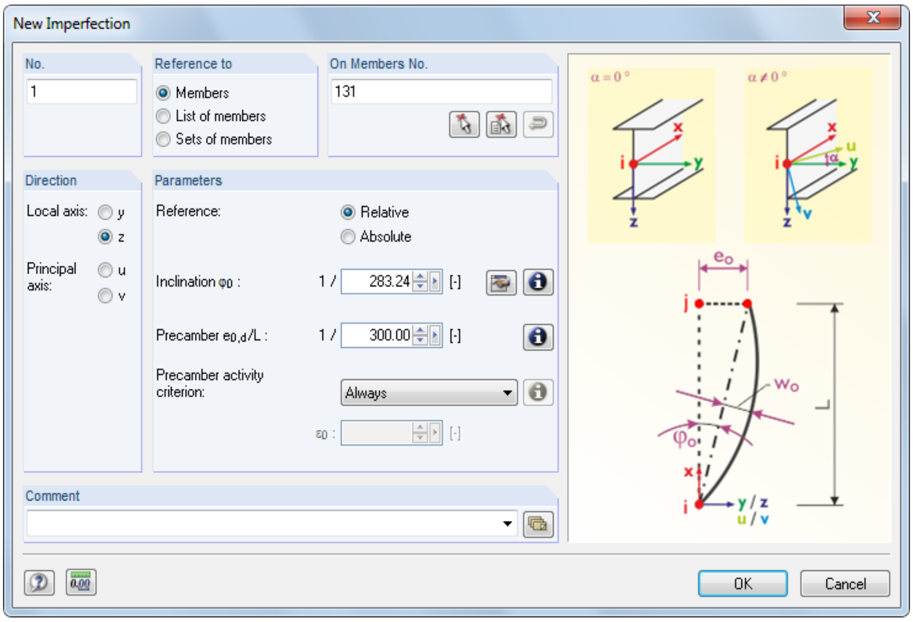


Figure 6.48: Dialog box *New Imperfection*

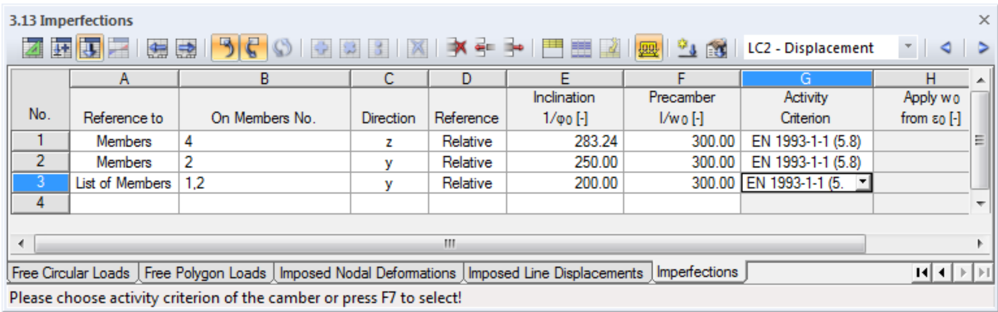


Figure 6.49: Table 3.13 *Imperfections*

The number of the imperfection is assigned automatically in the dialog box *New Imperfection* but can be changed in the input field. The numbering order is not important.

Reference to

Define the objects to which you want to apply the imperfection. The following options can be selected:

Members

The imperfection acts on one single member or on each member of several selected members.

List of members

The imperfection acts on the union of members that are defined in the list. Thus, pre-deformations and inclinations are not applied to each member individually but as total imperfection to all members of the member list. Load effects of an imperfection on single members in contrast to a member list are shown in Figure 6.50.

Take advantage of a list of members to apply imperfections over all members without defining continuous members.

Sets of members

The imperfection acts on a set of members or on each set of several sets of members. Similar to the member list described above, parameters are applied to the union of members included in the member set.

Sets of members are subdivided into continuous members and groups of members (see chapter 4.21, page 158). Imperfections for sets of members can only be applied to continuous members lying on one line. They are not adequate for member groups or continuous members which are buckled.

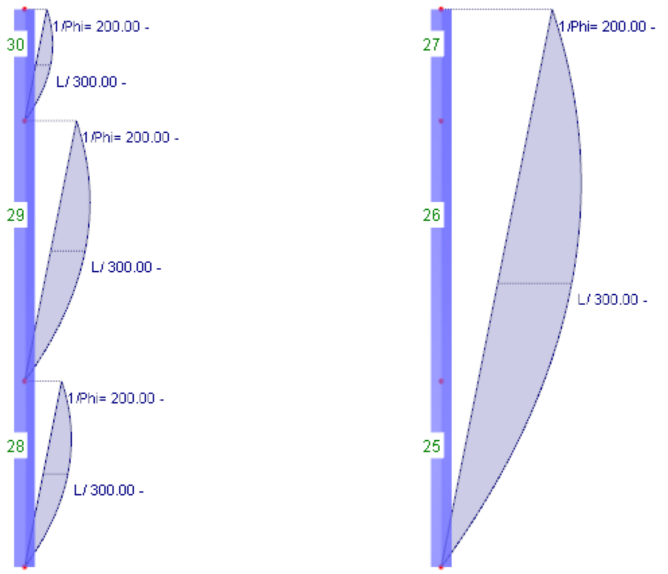


Figure 6.50: Imperfection with reference to members (left) and to a list of members (right)

On members

In the input field, enter the numbers of members or sets of members on which the imperfection is acting. In the dialog box, you can select nodes also graphically by using the [↖] function.

When you have selected the graphical input by clicking the toolbar button, the input field is disabled and you have to enter all imperfection data first. After clicking [OK] you can select the relevant members or sets of members one after the other in the work window.

For imperfections referring to a list of members it is possible to arrange member numbers appropriately by using the dialog button [Reverse Member Orientation], for example to reverse the inclination for the graphic display. However, the sequence is irrelevant for calculations because of the identical equivalent loads.

Direction

The imperfection can only be applied in direction of the local member axes *y* or *z*. When unsymmetrical cross-sections are used, the principal axes *u* and *v* are additionally available for selection (see chapter 4.13, page 120). It is not possible to define a globally acting inclination or precamber.

The orientation of member axes is described in chapter 4.17, paragraph *Member rotation* on page 145. For symmetrical sections, axis *y* represents the so-called 'strong' axis, axis *z* accordingly the 'weak' axis of the member cross-section.

When the model type for plates or walls was selected in the *General Data* dialog box, only the direction *z* can be selected.

Reference

The values for inclination and precamber can be defined in two ways:

Relative allows for entering the reciprocal values of φ_0 and w_0 in relation to the member length, *Absolute* allows for specifying geometric dimensions directly.

Inclination 1/φ_0

φ_0 indicates the degree of inclination as it is described for example in EN 1993-1-1, clause 5.3.2. Enter the reciprocal value of φ_0, respectively the absolute value, into the input field. An illustration of parameters can be displayed in the dialog box by using the [Info] button.

In addition, the dialog box offers you the button [Calculate inclination] to determine inclinations according to different standards in a separate dialog box.

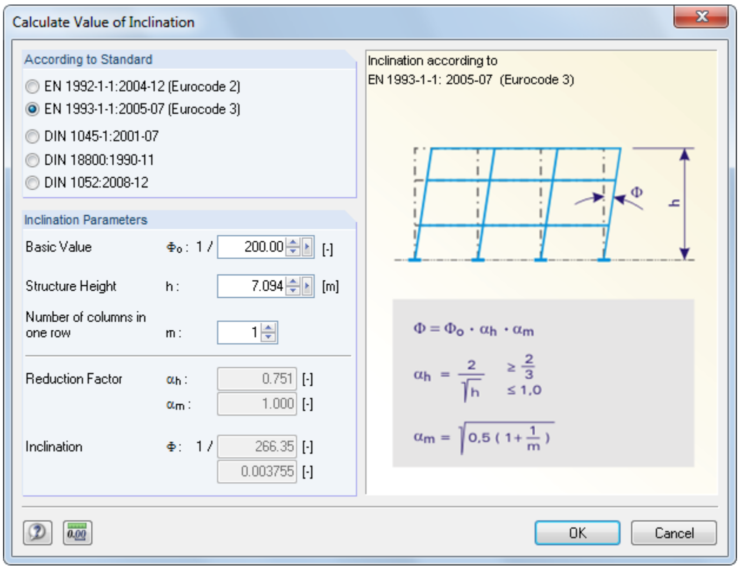


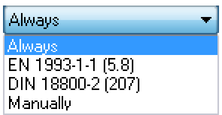
Figure 6.51: Dialog box *Calculate Value of Inclination*

Depending on the setting selected in the dialog section *According to Standard*, different input fields are available in the dialog section *Inclination Parameters*. Based on the values entered in the dialog input fields, reduction factors and inclinations are calculated conforming to standards. Click [OK] to transfer the values to the initial dialog box.

Precamber l/w_0

The precamber w_0 or $e_{0,d}$ defines the degree of deflection to be applied according to the standard (for example DIN 18800 part 2, el. (204) or EN 1993-1-1, clause 5.3.2). The precamber depends on the buckling stress curve of the cross-section and is related to the member length L or entered as absolute value.

Activity criterion



The following options are available for selection to define how precambers are handled in interaction with member inclinations:

- **Always**
  Precamber is taken into account in all cases.
- **EN 1993-1-1 (5.8)**
  The influence of the precamber $e_{0,d}$ is applied to members with a slenderness $\bar{\lambda}$ determined according to EN 1993-1-1:2005, clause 5.3.2 (6), eq. (5.8).
- **DIN 18800-2 (207)**
  w_0 is applied only if the member coefficient ε exceeds a certain value. This regulation refers to DIN 18800, part 2, el. (207).
- **Manually**
  The activity criterion can be user-defined.

To display the criteria in the dialog graphic, use the [Info] button.

To consider w_0 not before ε_0

A precamber is considered in addition to inclination if the member coefficient ε is higher than the value defined in this input field. DIN 18800-2 el. (207) specifies $\varepsilon > 1.6$ for most cases.

6.14 Generated Loads

RFEM offers several generators that you can use to create loads easily (see chapter 11.8, page 521 ff.). Generated member or surface loads are reflected in table 3.14 and in the *Data* navigator.

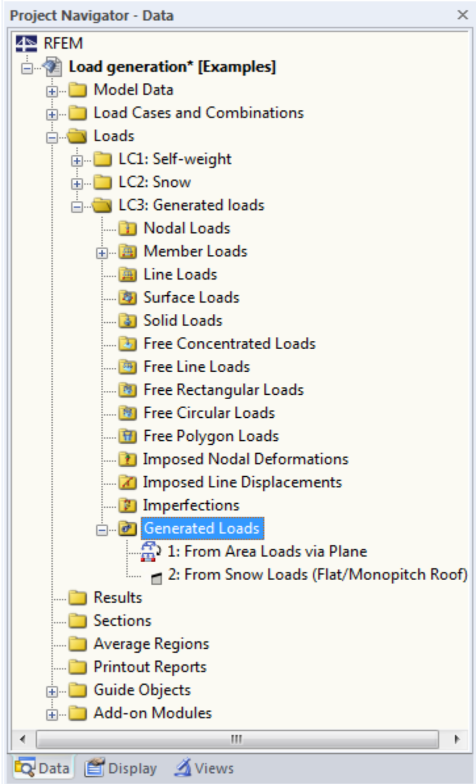


Figure 6.52: *Data* navigator for *Generated Loads*

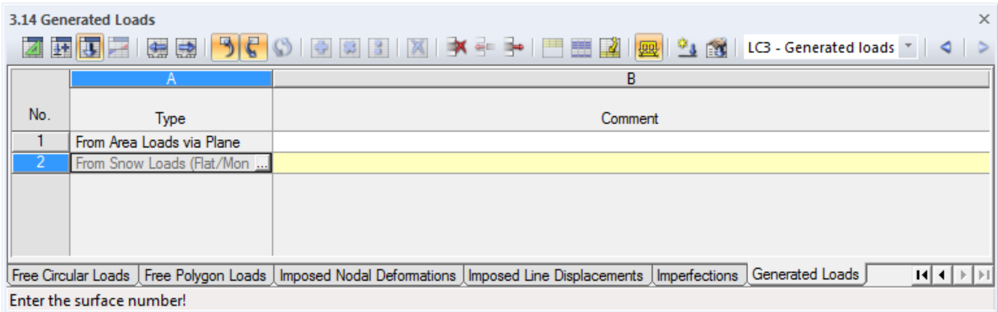


Figure 6.53: Table 3.14 *Generated Loads*

The original generator dialog boxes are stored as specific load objects which can also be accessed for modifications: Double-click a navigator item or use the table button [...] to open the initial dialog box again (see for example Figure 11.194, page 532) where you can adjust the parameters of load generation.

7. Calculation

7.1 Checking the Input Data

Before you start the calculation, it is recommended to check model and load data as well as the modeling. RFEM checks if data for each model and load object is completely available, if references of data sets are alright and if the modeling is correct.

Possible input errors can be corrected quickly as you can directly access the table row with the relevant problem (see Figure 7.2).

7.1.1 Plausibility Check

You can check model as well as load data for its coherent input. To open the dialog box for the plausibility check,

select **Plausibility Check** on the **Tools** menu

or use the toolbar button shown on the left.

A dialog box opens where you define the input data that you want to check.

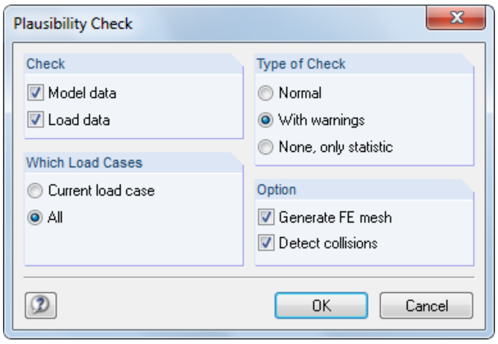


Figure 7.1: Dialog box *Plausibility Check*

In the dialog section *Type of Check*, you can choose between three options:

- **Normal**

  The standard option checks the completeness of input parameters and the correctness of data records.

- **With warnings**

  Select this option to carry out a detailed check of input data, finding also nodes with identical coordinates or releases with unlimited degrees of freedom.

  When a mismatch is detected, a message appears with detailed information about the problem. You can interrupt the check in order to eliminate the mistake.

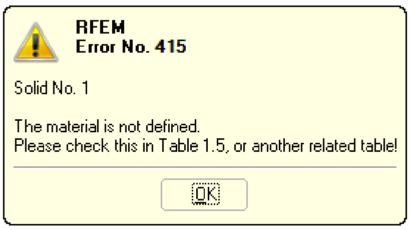


Figure 7.2: Plausibility check with warning

- **None, only statistic**

  Only a summary of input data is reported (model dimensions, total weight, number of nodes, lines, supports, surface and member loads etc.).

When the check box for *Generate FE mesh* is ticked below, the FE mesh can be generated during the plausibility check. For more detailed information, see chapter 7.2, page 256.

When the plausibility check was successful, the check result appears showing you a summary of input data.

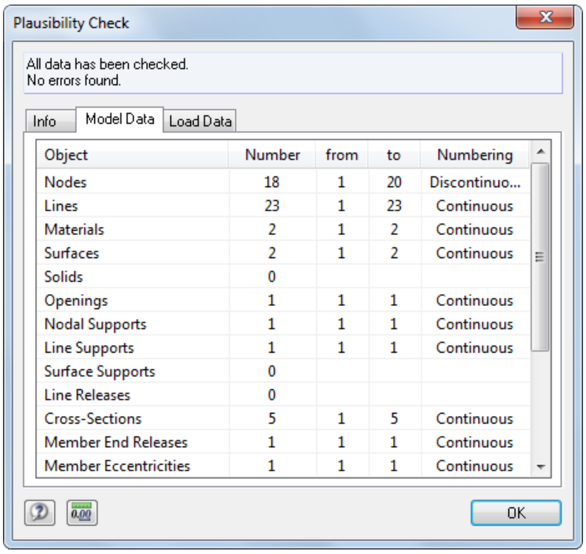


Figure 7.3: Result of plausibility check, tab *Model Data*

7.1.2 Model Check

In addition to the general plausibility check, you can use the model check to search specifically for discrepancies produced during the modeling. To open the corresponding dialog box,

point to **Model Check** on the **Tools** menu

and select one of several check options.

Identical nodes

RFEM filters all nodes with identical coordinates. They are combined in groups shown in a dialog box.

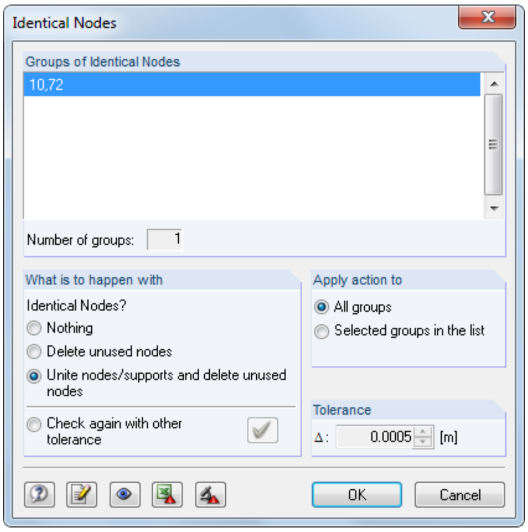


Figure 7.4: Results of model check for identical nodes

In the dialog section *What is to happen with Identical nodes?*, you can decide how double nodes are treated. In the dialog section *Apply action to*, you define whether your selection applies to all groups listed above or only to the selected row.

In the dialog section *Tolerance*, a kind of fine tuning is available to define the zone where coordinates are evaluated as identical. This function is especially useful for models imported from CAD programs. In this case, lines are often short because of nodes lying closely together. If such nodes are filtered with appropriate tolerance and then unified, it is possible to avoid numerical problems due to short members or lines.

Overlapping members

Use this option to filter all members overlapping partially or entirely in their lengths.

If overlapping members are detected, they are shown in a dialog box where they are sorted by groups. The current group is indicated by an arrow displayed in the work window. After clicking the [OK] button you can fix the problem.

Crossing not connected members

The check searches for members which are crossing but do not have a common node at the point of intersection.

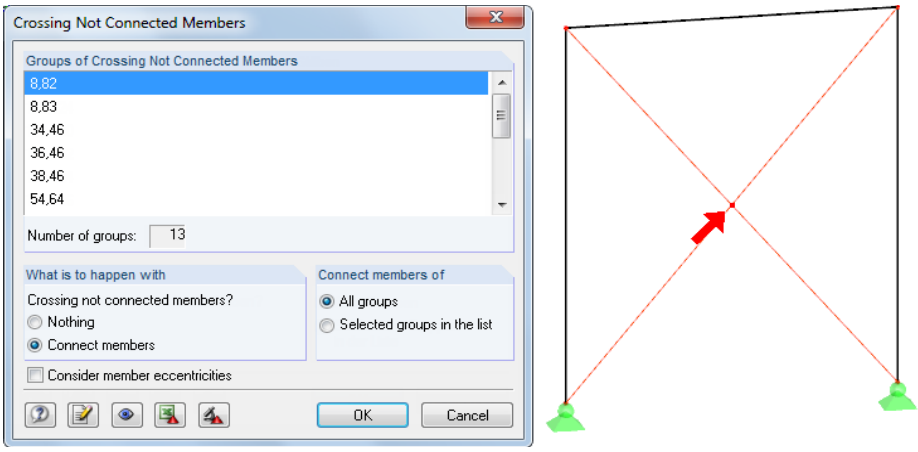


Figure 7.5: Results of model check for crossing members

The check results are shown in the dialog section *Groups of Crossing Not Connected Members*. The crossing members are listed in groups. The group that is currently selected is indicated by an arrow in the graphic.

In the dialog section *What is to happen with*, you decide what you want to do with the crossing members. The option *Connect members* is useful for actual possibilities of internal force transfer but not for example ordinary diagonal crossings with ties.

Overlapping lines

Use this option to filter all lines overlapping partially or entirely in their lengths.

If overlapping lines are detected, they are shown in a dialog box where they are sorted by groups. The current group is indicated by an arrow displayed in the work window. After clicking the [OK] button you can fix the problem.

Crossing not connected lines

Use this option to search for lines that are crossing without sharing a common node at the point of intersection. The results are shown in the dialog section *Groups of Crossing Not Connected Lines* (see Figure 7.5). The crossing lines are listed in groups. The group that is currently selected is indicated by an arrow in the graphic.

In the dialog section *What is to happen with*, you decide how to handle the crossing lines.

Overlapping surfaces

Use this option to filter all surfaces overlapping partially or entirely.

If overlapping surfaces are detected, they are shown in a dialog box where they are sorted by groups. The current group is indicated by its selection color shown in the work window. After clicking the [OK] button you can fix the problem.

Minimally curved surfaces

You can search for surfaces with minor plane deviation.

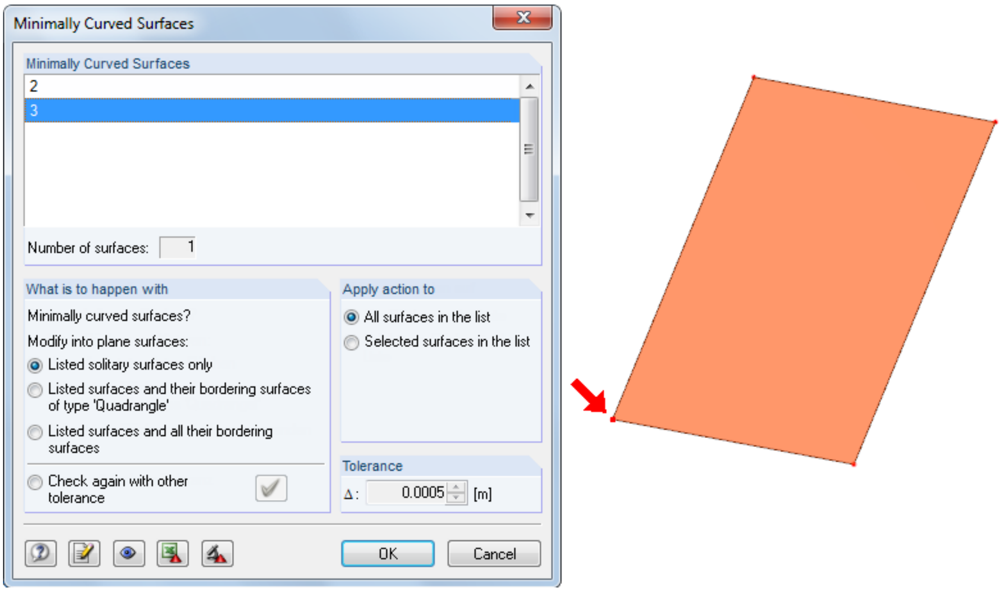


Figure 7.6: Results of model check for minimally curved surfaces

If surfaces with minor curvatures are detected, they are shown in a dialog box where they are sorted by groups. The current group with a node deviating from the plane is indicated by an arrow displayed in the work window.

The dialog section *What is to happen with* offers you specific control options how to treat such surfaces. In the dialog section *Apply action to*, you decide whether your settings apply to all groups listed above or only to the selected surface.

In the dialog section *Tolerance*, a kind of fine tuning is available for the plane definition. Furthermore, it is possible to *Check again with other tolerance* on curved surfaces in case of modifications.

Buttons

The buttons in the dialog boxes of the model check have the following functions:

	Applies changes of dialog section *What is to happen with*
	Jumps to RFEM work window to adjust the view
	Exports listed objects to Excel table
	Creates a new partial view for each object group

Table 7.1: Buttons in dialog boxes of model check

7.1.3 Regenerate Model

RFEM revises automatically small inconsistencies existing in the model produced during the modeling process or arising from data exchange with CAD programs. To access the corresponding function,

select **Regenerate Model** on the **Tools** menu.

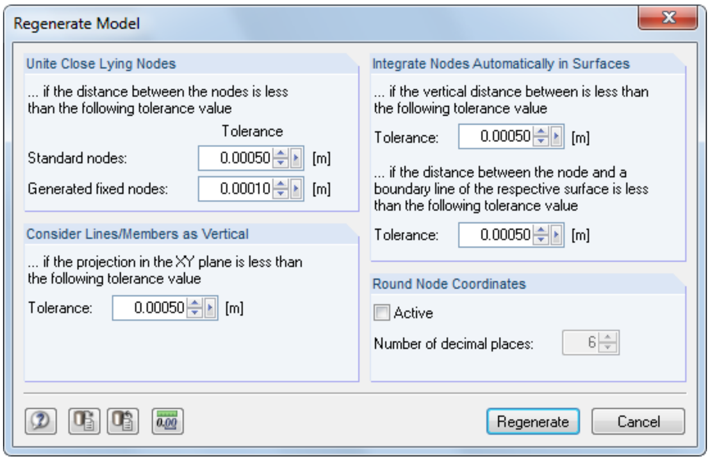


Figure 7.7: Dialog box *Regenerate Model*

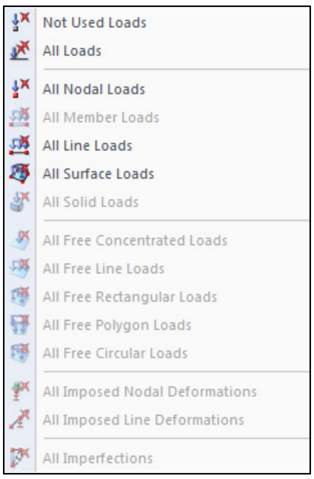
In the dialog section *Unite Close Lying Nodes*, define a threshold for the distances of nodes: When values fall below the *Tolerance*, nodes are considered to be identical and will be combined in a single node. As redundant nodes will be deleted, a renumbering of objects may be the result.

In the dialog section *Consider Lines/Members as Vertical*, you can control the position of local line and member axes. The orientation of axes for members in vertical position differs essentially from members in general (inclined) position (see chapter 4.17, page 146). To impose a vertical position for a general position, you can use the input field *Tolerance*. In this way, you prevent the member axes from "switching", which is also favorable for load input and output of internal forces.

Use the options in the dialog section *Integrate Nodes Automatically in Surfaces* for nodes with only a very small distance to a surface or boundary line to include them automatically into the list of integrated objects of the surface (see page 82). As a result, a manual integration is unnecessary. Please note that an internal check is carried out before the calculation starts: If the nodes' distance to the surface is too large, they will be considered as not belonging to the surface.

Finally, you can activate a function to *Round Node Coordinates* automatically. Define the relevant number of decimal places.

7.1.4 Delete Not Used Loads



Menu *Tools* → *Delete Loads*

Loads can only be defined on objects existing in the model. However, during the modeling process it may happen that members or surfaces with assigned loads are removed from the system. Normally, RFEM deletes their loads, too. If the plausibility check still finds loads on non-existing objects, it is possible to remove them. To find unused loads,

point to **Delete Loads** on the **Tools** menu, and then select **Not Used Loads**.

Use the menu shown on the left to select also other load objects for specific removal.

7.2 FE Mesh

With RFEM you can analyze member elements, plates, walls, shells and solids. Before you calculate data, the FE mesh must be generated to create the corresponding 1D-, 2D and 3D elements.

The FE analysis requires the division of the structural system into small subsystems represented by finite elements. For each element equilibrium conditions are set up. A linear system of equations is created that has many unknown quantities. The more refined the mesh size of the finite elements is, the more precise the results will be. On the other hand, computing time increases enormously due to the amount of data to be calculated, as additional equations must be solved for every further FE node.

The FE mesh is created automatically. However, there are some options that can be used to control the mesh generation.

Useful references concerning discretization and finite element method can be found in [15].

7.2.1 Basics of Finite Elements in RFEM

1D elements

For member elements, it is assumed that the cross-section remains planar when deforming. 1D member elements are used to represent beams, trusses, ribs, cables and rigid couplings. A 1D member element has in total 12 degrees of freedom: six at the start and six at the end of the element. They refer to displacements (u_x, u_y, u_z) and rotations (φ_x, φ_y, φ_z). When calculating structural data linearly, tension, compression and torsion are expressed as linear functions of the member axis x, independent of bending and shear. They are approximated by a 3rd order polynomial in x, including influence of shearing stresses resulting from shear forces V_y and V_z. The stiffness matrix $\mathbf{K_L}$(12, 12) describes the linear behavior of the 1D elements. The mutual interaction of axial force with bending in case of geometrically nonlinear problems is expressed in the stiffness matrix $\mathbf{K_{NL}}$(12, 12). Please find further information in [18] and [19].

For calculations according to large deformation analysis it is recommended to use an FE mesh refinement of lines (see chapter 4.23, page 166) so that results can be calculated accurately.

2D elements

Usually, quadrangle elements are used as 2D elements. The mesh generator adds triangular elements, where required.

The degrees of freedom in corner nodes of quadrangle and triangular elements are the same as for 1D elements: degrees of freedom of displacement (u_x, u_y, u_z) and rotation (φ_x, φ_y, φ_z). In this way, the compatibility of 1D and 2D elements in nodes is guaranteed. The parameters are defined in the planar, local coordinate system of elements and will be converted into the global coordinate system when creating the global stiffness matrix.

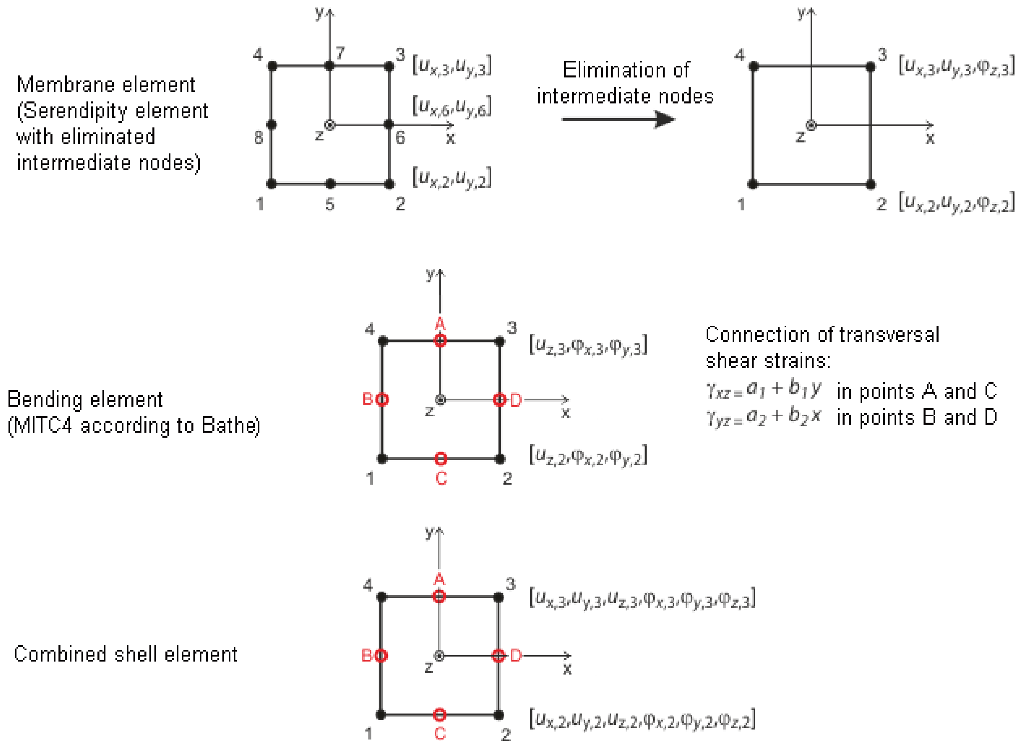


Figure 7.8: Shell elements of RFEM (quadrangle)

The planar shell elements are based on the MINDLIN/REISSNER theory. Figure 7.8 shows the approaches of elements in graphical representations. To ensure a direct coupling with member elements, a square approach in the shell plane (u_x, u_y) is chosen. By eliminating the intermediate nodes, a four-node element is created with an additional degree of freedom φ_x. This allows for wall elements to be coupled directly with beam elements. Based on a mixed interpolation of transversal deformations, cross-section rotations as well as transversal shear strains, the MITC4 elements (*M*ixed *I*nterpolation of *T*ensorial *C*omponents) as presented by BATHE and DVORKIN [24] are also applied.

At the moment, member elements are considered by directly solving the differential equation according to second-order analysis. Considering drilling effects is not possible with the Saint-Venant torsion.

The analysis of membranes is based on the principles of BERGAN [20], [21], [22]. The basic functions are subdivided, for example for triangular elements into three rigid-body deformations, three constant conditions of strain and three special linear gradients of stress and strain. Within an element, the deformation field is quadratic and the stress field is linear. The element stiffness matrix $\mathbf{K_L}$ is then transformed into nine collective parameters of the types u_x, u_y, φ_z. Components of this matrix are then added to the overall stiffness matrix (18, 18) , together with the components causing bending and shear effects. This matrix is the result of the LYNN/DHILLON concept. Then, so-called MINDLIN plates are applied, which means that plates with distinct shear distortion are analyzed according to TIMOSHENKO. Thus, RFEM is able to find the correct solution for both thick and thin plates (NAVIER plates).

In case of geometrically nonlinear problems, it is not possible to split the stress-strain condition into a planar state and into bending with shear. The mutual influences of these states are considered in the matrix $\mathbf{K_{NL}}$. RFEM uses a rather simple, but effective type of the K_{NL} matrix that is based on the approaches of ZIENKIEWICZ [23]. The square component ε_2 of the GREEN/LAGRANGE strain tensor $\varepsilon = \varepsilon_1 + \varepsilon_2$ is applied. A linear distribution of $u_z(x, y)$ of the planar stress condition and linear distributions of $u_x(x, y)$ and $u_y(x, y)$ of the interaction with bending are assumed. This assumption is possible because the main effect of the interaction depends on the first derivation of the differential equation, and because the influence of components of a higher order decreases quickly with the division into smaller elements. The correctness of this procedure has been proved in several numerical analyses.

To apply shell elements, thickness of elements must be considerably smaller than their dimension. If this is not the case, it is recommended to model objects as solids.

3D elements

The following 3D elements are implemented in RFEM: tetrahedron, pentahedron (prism, pyramid) and hexahedron. For detailed information about applied elements and matrices, please see [48]. Corresponding documentation can be requested from DLUBAL SOFTWARE GMBH.

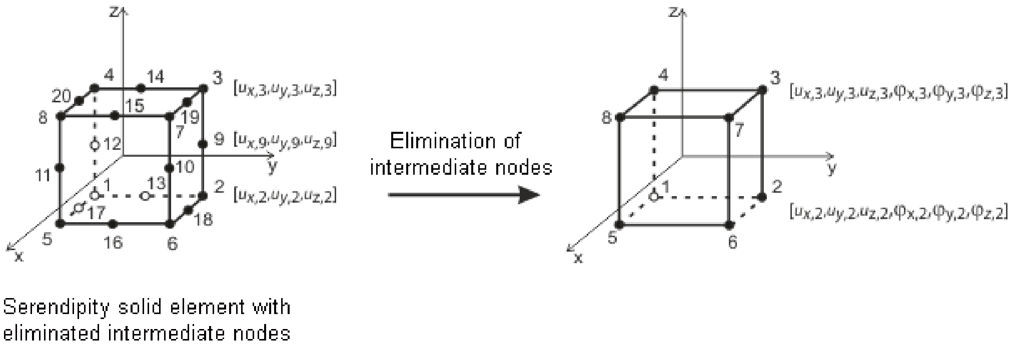


Figure 7.9: Solid element (hexahedron)

7.2.2 FE Mesh Settings

To open the dialog box for setting FE mesh parameters,

select **FE Mesh Settings** on the **Calculate** menu.

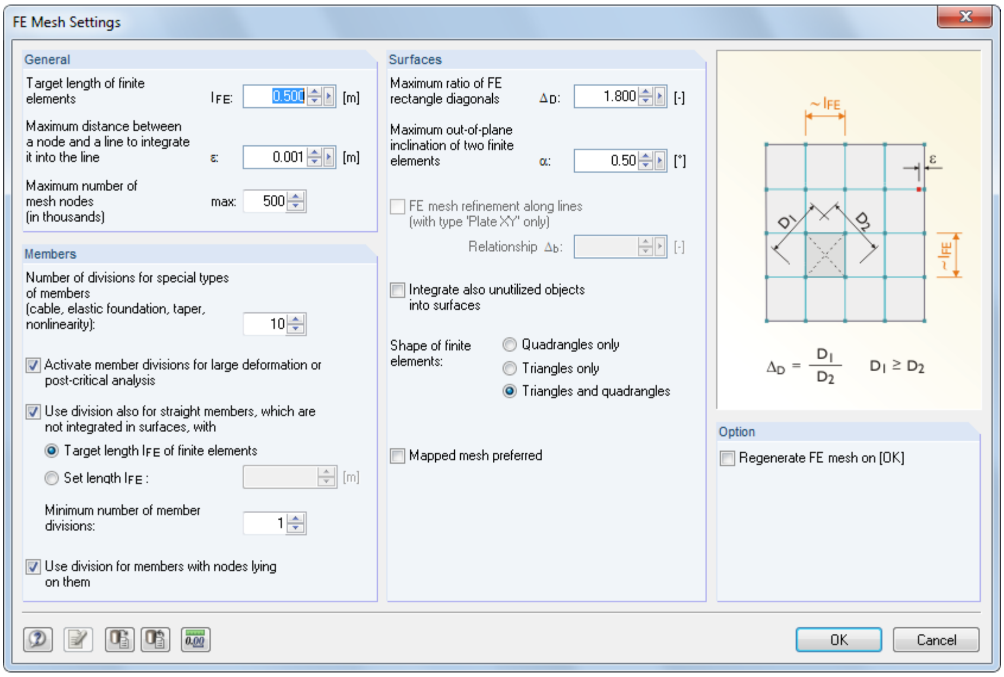


Figure 7.10: Dialog box *FE Mesh Settings*

General

The *Target length of finite elements* controls the global size of the mesh. The finer the mesh is, the more precise the results will generally be. However, the amount of data to be calculated as well as computing time are increased as well since additional equations must be solved for any further FE node. Furthermore, singularity effects occur more frequently in a fine-meshed FE mesh.

The discretization is important for the FE analysis to be performed. A mesh size that is too fine slows down the calculation without improving quality of results significantly. A target length that is too long cannot determine boundary conditions in a satisfying way. As a general rule, the following recommendation can be made for the appropriate lateral length of finite elements: Eight to ten finite elements should be generated between the boundary lines of a surface. If possible, avoid to define less than four elements.

In the second input field of this dialog section, you define the mesh node's allowable distance ε from a line. If the distance of a node is larger than the entered value, a new FE node will be created for it.

The *Maximum number of mesh nodes* is defined with an upper limit in the last input field of the dialog section *General* to restrict the number of generated nodes and thus to ensure the efficiency of program and computer.

Members

For cable, foundation and tapered members or members with plastic properties, you can specify the number of internal divisions, which leads to a real division of the member by intermediate nodes. However, if a member is arranged on a boundary line of a surface, or if the definition line has an FE mesh refinement, the specification has no effect.

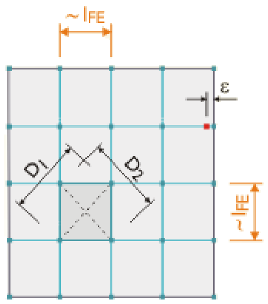
Tick the check box for *Activate member divisions for large deformation or post-critical analysis* to divide also beams by intermediate nodes for the calculation according to large deformation analysis so that these members are calculated with higher accuracy. The number of member divisions is determined by the input field above.

If you select *Use division also for straight members, which are not integrated in surfaces,* FE nodes will be generated on all free members and considered for calculations according to linear static and second-order analysis. The length of the finite elements is either determined by the global target length l_{FE} set in the dialog section *General* or entered manually.

With the ticked option *Use division for members with nodes lying on them*, RFEM generates FE nodes on those locations of the member where end nodes of other members are lying, without any connection existing between these members.

Surfaces

The most accurate results are determined for elements coming as close as possible to a shape of a square. For a square, the ratio of diagonals is $D_1/D_2=1$. In the input field *Maximum ratio of FE rectangle diagonals*, enter the limit value Δ_D for the diagonal ratio. If the value is set too high, there is a risk that elements are generated with very acute or reflex angles. This may result in numerical problems.



$$\Delta_D = \frac{D_1}{D_2} \qquad D_1 \geq D_2$$

Figure 7.11: Elements with diagonals D_1 and D_2

A curved surface is transformed into planar elements when creating the FE mesh. The value entered in the input field *Maximum out-of-plane inclination of two finite elements* defines the maximum allowable inclination angle α: The mesh will be refined automatically on the location where the value is exceeded.

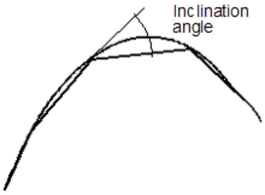


Figure 7.12: Inclination angle α between two finite elements

If the model is defined as plate of type *2D - in XY*, you can specify a *FE mesh refinement along lines* to create smaller finite elements on all lines, and thus to approximate better for example results along supported lines. The relation Δb refers to the global mesh size. It describes the edge distance of the refinement from the lines.

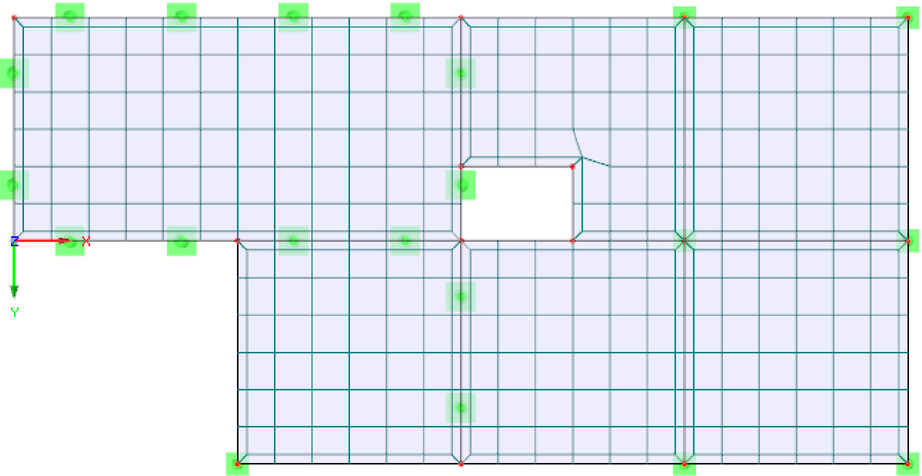
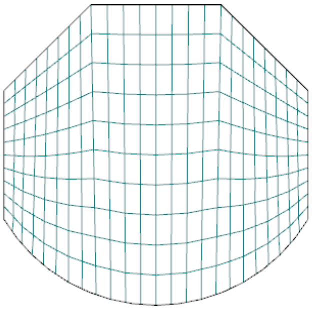


Figure 7.13: FE mesh refinement along the boundary lines of a 2D slab

Tick the check box *Integrate also unutilized objects into surfaces* to generate FE nodes also on objects that do not have any specific function for a surface (for example free nodes without support or load, constructional lines in surfaces). The function is disabled by default so that structurally irrelevant objects do not warp the FE mesh.

The *Shape of finite elements* can be determined by the following three options:

- Triangles and quadrangles: default setting
- Triangles only: option in case quadrangles cause strong mesh distortions
- Quadrangles only: option for higher accuracy of results



Mapped FE mesh

The ticked option *Mapped mesh preferred* tries to align the FE mesh with the boundary lines of surfaces. This type of FE mesh generation can be defined for each surface individually (dialog box *Edit Surface*, tab *FE Mesh*).

A mapped mesh is exclusively composed of quadrangles. In general, a mapped mesh provides "more accurate" results. Since also unknown quantities occur less frequently in the equation system, it is recommended for generating the mesh.

Option

Tick the check box for *Regenerate FE mesh on [OK]* if you want to generate a new FE mesh after confirming the dialog box.

7.2.3 FE Mesh Refinements

Take advantage of FE mesh refinements to influence the FE mesh generation. You can refine the mesh on relevant points, for example in connection zones. Refinements are also used to make an appropriate compromise between result accuracy and computing time.

Basically, there are four types of FE mesh refinements:

- Refinement around a node
- Refinement on a line
- Refinement on a surface
- Refinement on a solid

The definition of FE mesh refinements is described in chapter 4.23 on page 164.

7.2.4 FE Mesh Generation

To start the FE mesh generation,

select **Generate FE Mesh** on the **Calculate** menu.

Moreover, the FE mesh is generated automatically when you start the calculation of a load case. However, it is strongly recommended to check the generated mesh before starting the calculation, and to check whether a sufficient and "harmonic" discretization is available or if refinement areas are still needed.

Conversely, areas of minor relevance for the results evaluation can be covered by a more coarse-meshed FE mesh. For example, define a surface "mesh refinement" with a mesh size larger than the target length l_{FE}. In this way, it is possible to accelearte the calculation as well as the evaluation.

When the FE mesh generation has been successful,

select **FE Mesh Statistic** on the **Calculate** menu

to open a dialog box with information about the generated FE mesh.

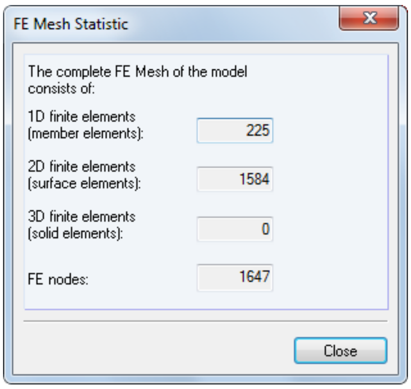


Figure 7.14: Dialog box *FE Mesh Statistic*

The statistics show types and numbers of the generated finite elements, which is helpful for selecting the appropriate equation solving method and to estimate the approximate calculation time (see chapter 7.3, page 274).

The FE mesh is deleted automatically when model data is modified. Besides, it is possible to delete the mesh specifically. To activate the corresponding function,

select **Delete FE Mesh** on the **Calculate** menu.

Please note that all results that may be available will be deleted, too.

7.3 Calculation Parameters

Dialog box *Edit Load Cases and Combinations*

When creating a load case or load combination, it is already possible to define calculation parameters. Settings can be specified in the respective dialog section tab *Calculation Parameters* of the dialog box *Edit Load Cases and Combinations*.

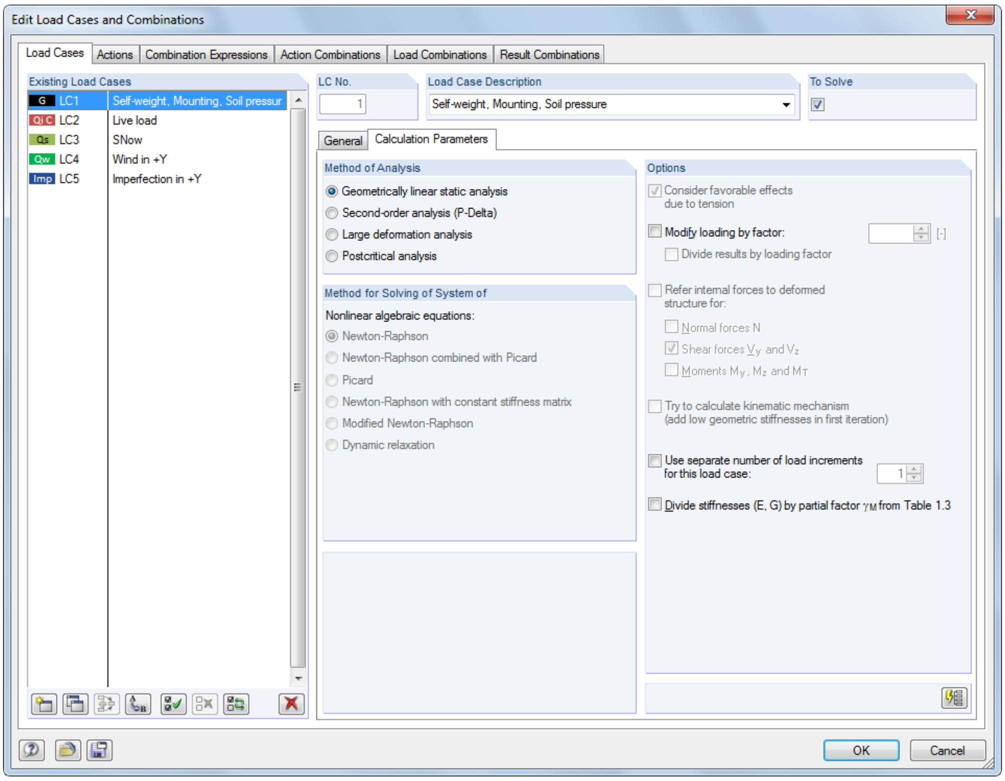


Figure 7.15: Dialog box *Edit Load Cases and Combinations*, tabs *Load Cases* and *Calculation Parameters*

In addition to offering an overview about all load cases and combinations, the dialog box *Edit Load Cases and Combinations* controls the calculation parameters for each load case, load combination and result combination.

Dialog box *Calculation Parameters*

Moreover, you can access the calculation parameters in a separate dialog box.

To open the dialog box *Calculation Parameters*,

select **Calculation Parameters** on the **Calculate** menu

or use the toolbar button shown on the left.

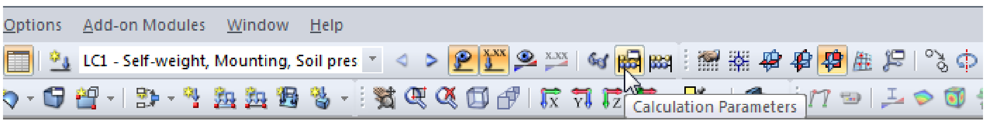


Figure 7.16: Button [Calculation Parameters]

The dialog box *Calculation Parameters* consists of four dialog tabs. The first three tabs manage the calculation parameters of each load case, respectively load and result combination. In the fourth tab *Global Calculation Parameters* (see Figure 7.22, page 271), you can check and, if necessary, adjust specifications that are universally valid.

7.3.1 Load Cases and Load Combinations

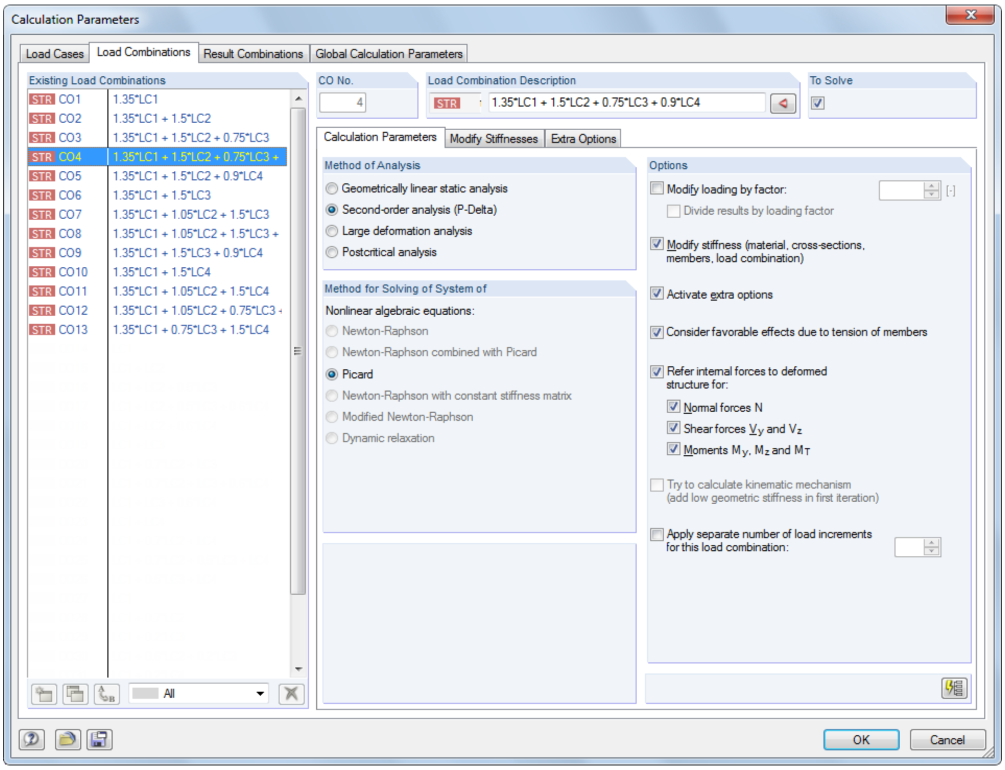


Figure 7.17: Dialog box *Calculation Parameters*, tab *Load Combinations*

The dialog section *Existing Load Cases*, respectively *Existing Load Combinations*, lists all available load cases and combinations. You can adjust the *Calculation Parameters* of the selected entry in the dialog section to the right.

The button [Apply settings] assigns the current specifications to all load cases and combinations.

The dialog tab *Load Combinations* is subdivided into the dialog section tabs *Calculation Parameters*, *Modify Stiffnesses* (see page 267) and, if applicable, *Extra Options* (see page 268).

7.3.1.1 Dialog Tab *Calculation Parameters*

Method of analysis

In this dialog section, you decide whether the load case/combination is calculated according to the *linear static*, *Second-order* or *Large deformation* analysis. Select the option *Postcritical analysis* to carry out a stability analysis according to large deformation analysis with regard to postcritical failure of the entire structure.

RFEM presets the linear calculation according to linear static analysis for load cases, and the nonlinear calculation according to second-order analysis for load combinations.

When the model includes cable members, a large deformation analysis is suggested in all cases. Cable members are always calculated according to the large deformation analysis, the remaining members according to the selected analysis method.

Second order analysis

The common "structural" second-order analysis is used to determine the equilibrium on the deformed system. Deformations are assumed to be small. If axial forces are available in the system, they will lead to an increase of bending moments. Thus, the calculation according to second-order analysis by TIMOSHENKO [10] has an effect only if axial forces are considerably higher than shear forces. The additional bending moment ΔM results from the axial force N and the elastic lever e_{el}.

$$\Delta M = N \cdot e_{el}$$

Equation 7.1

For structural systems subjected to pressure there is an overlinear relation between loading and internal forces. Normally, you have to calculate also with γ-fold actions.

The axial force difference in the iterations represents the break-off criterion. For member elements, the stiffness-modifying axial force decisive for the second-order analysis is assumed to be constant along the entire member. The calculation stops as soon as a certain value of the normal force difference falls below. It is possible to influence this break-off limit in the dialog section *Precision and Tolerance* of the dialog tab *Global Calculation Parameters*.

For nonlinear calculations according to second-order analysis, assumptions of the linear elastic analysis are the same with the following additions:

- No plastic deformations occur.
- The external forces stay true to the direction.
- For members with non-constant axial force (for example columns) the mean value of the axial force N is applied for determining the member coefficient ε.

Large deformation analysis

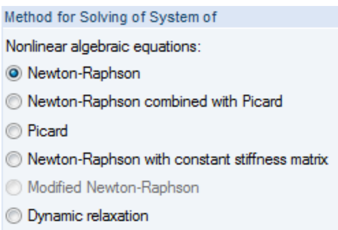
The large deformation analysis ("third order theory") takes into account longitudinal and transversal forces during the analysis of internal forces. If the calculation according to this large deformation analysis is selected, all surfaces and members will be calculated according to this calculation theory.

The stiffness matrix for the deformed system is created after each iteration step. Please note that there are significant differences between loads defined as local and global: For example, when a surface load defined as global in Z is acting on a floor, it keeps its direction if finite elements are warping. But when the load is effective in direction of the local surface axis z, it warps on each element according to the element's warping.

Post-critical analysis

A stability analysis with regard to postcritical failure is performed. The method represents a modified calculation according to large deformation analysis by NEWTON-RAPHSON where the influence of axial forces is considered for changes occurring in shear and bending stiffness. The tangential stiffness matrix is saved in each iteration step. In case of singularities (which means instability), the stiffness matrix of the previous iteration will be used for new geometric, incremental iterations until the tangential stiffness matrix of the current setting becomes regular (stable).

Method for solving of system of nonlinear equations



Six methods for solving the nonlinear, algebraic system of equations are available for selection:

Newton-Raphson

The approach according to NEWTON-RAPHSON is preset for the large deformation analysis. The nonlinear equation system is solved numerically by means of iterative approximations with tangents. The tangential stiffness matrix is determined as function of the current state of deformation; it is inverted in each cycle of iterations. In the majority of cases, a fast (quadratic) convergence is reached.

You can influence the performance of convergence by the number of load increments set in the dialog tab *Global Calculation Parameters*.

Newton-Raphson combined with Picard

First, the approach according to PICARD is applied (see below). After a few iterations, the program changes to the NEWTON-RAPHSON method. The basic idea of this approach is to use the relatively strong PICARD method for the first iteration steps to avoid instability messages. This initial approximation is followed by the fast method according to NEWTON-RAPHSON to find the ultimate state of equilibrium.

In the dialog section *Settings* of the dialog tab *Global Calculation Parameters*, you can define the percentage that is used for iterations according to PICARD when the combined method is applied (see Figure 7.22, page 271).

Picard

The method according to PICARD, also known as secant method, can be understood as finite difference approximation of the Newton method. The program takes into account the difference between the current iteration run and the original iteration run in the current load increment step.

Often, converging is slower than in the calculation method according to NEWTON-RAPHSON. But the method proves to be stronger with regard to nonlinear problems, making the calculation more stable.

Newton-Raphson with constant stiffness matrix

This version of the NEWTON-RAPHSON method can be selected for calculations according to large deformation analysis. The stiffness matrix is created only once in the first iteration step, and then it is used in all subsequent calculation loops.

Thus, the calculation runs faster but not that stable like calculations according to the normal or modified NEWTON-RAPHSON method.

Modified Newton-Raphson

This method is used to perform the post-critical analysis (see dialog section *Method of Analysis* above) where a range with instability must be overcome. If an instability is available and the stiffness matrix cannot be inverted, the program uses the stiffness matrix of the last stable iteration step. The program continues to calculate with this matrix until a stability range is reached again.

Dynamic relaxation

The final method is appropriate for calculations according to large deformation analysis and for solving problems with regard to post-critical analysis. An artificial time parameter is introduced. Taking into account inertia and damping, the failure can be handled as a dynamic problem. This approach uses the explicit time-integration method; the stiffness matrix won't be inverted.

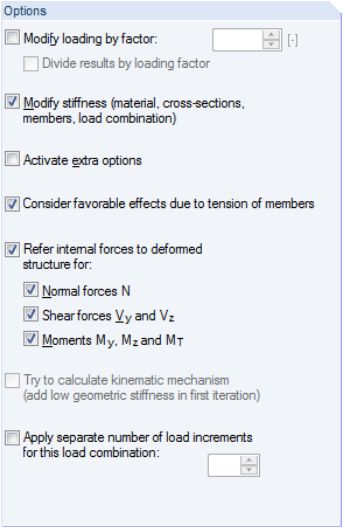
No part of the model is allowed to have a specific weight of zero when calculating with dynamic relaxation.

This method includes the RAYLEIGH damping that can be defined by means of the constants α and β according to the following equation with derivations by time:

$$M\ddot{u} + C\dot{u} + Ku = f$$

where	M	Mass matrix
	C	Damping matrix $C = \alpha M + \beta K$
	K	Stiffness matrix
	f	Vector of external forces
	u	Discretized displacement vector

Equation 7.2



Options

Modify loading by factor

After ticking the check box, you can enter a factor into the input field by which all loads contained in the load case or combination will be multiplied. The factor is also reflected in the load vectors and values of the graphic. Generally, also negative factors are permitted.

Older standards claim to multiply loads globally by a certain factor in order to increase effects according to second-order analysis for stability designs. On the other hand, design must be carried out with the characteristic loads. Both requirements can be fulfilled by entering a factor larger than 1.00 and ticking the check box *Divide results by loading factor*.

When analyzing structures according to current standards, loading should <u>not</u> be edited with any factors. Instead, partial safety factors and combination coefficients must be applied for the superposition in the load and result combinations.

Modify stiffness

If the check box is ticked, the stiffness factors of the materials (see chapter 4.3, page 61), cross-sections (see chapter 4.13, page 119) and members (see chapter 4.17, page 142) are considered in the calculation. Settings can be defined in detail in the dialog tab *Modify Stiffness* (see chapter 7.3.1.2, page 267).

Activate extra options

When ticking the check box the dialog tab *Extra Options* becomes available where you can activate the initial deformations or forces of a load case as well as the results of an add-on module for the calculation (see chapter 7.3.1.3, page 268).

Consider favorable effects due to tension

Tensile forces have a favorable effect on pre-deformed structural systems. Thus, the pre-deformation is reduced and the structure is stabilized.

There are different opinions on how to consider tensile forces acting in a favorable way. Standards contain regulations according to which relieving actions must be considered with a smaller partial safety factor than unfavorable effects.

Partial safety factors that are varying from one member to the other cannot be realized with an acceptable computing time. Therefore, RFEM offers you the option to set tensile forces generally to zero for calculations according to second-order analysis. With this approach you will be definitely on the safe side. If you want to use this option, clear the check box.

On the other hand, one can say that standards refer to actions and not to internal forces. Therefore, it is necessary to decide for the action as a whole whether it is favorable or unfavorable. Thus, if an unfavorable action has a favorable effect in certain zones of the model, it can definitely be considered. So, if you want to take account of the axial forces without any changes in the calculation according to this approach, the check box must be ticked (default setting).

The favorable effect of tensile forces should be considered in the majority of cases, for example for halls with bracings or structural systems affected by bending. But please keep in mind that relief due to tension force effects for beams with supporting cables may result in an unwanted reduction of deformations and internal forces.

Refer internal forces to deformed structure

The option enables output for nonlinear calculations showing axial and shear forces as well as bending and torsion moments of members in relation to the rotated coordinate systems of the deformed system. There are three check boxes available for the internal force types *Normal forces*, *Shear forces* and *Moments*.

Calculation of kinematic mechanism

You can try to make an unstable model available for calculation: Internally, small springs are applied stabilizing the model for the first iteration. When a stable initial state is reached, springs will be removed for the subsequent iterations.

Separate number of load increments

You can define an individual number of load increment steps for each load case and each load combination. Thus, the number specified in the dialog tab *Global Calculation Parameters* is no longer valid (see 7.3.3, page 272).

7.3.1.2 Dialog Tab *Modify Stiffness*

The dialog tab is displayed only when the check box for *Modify stiffness (material, cross-sections, members, load combination)* is ticked in the previous tab *Calculation Parameters*.

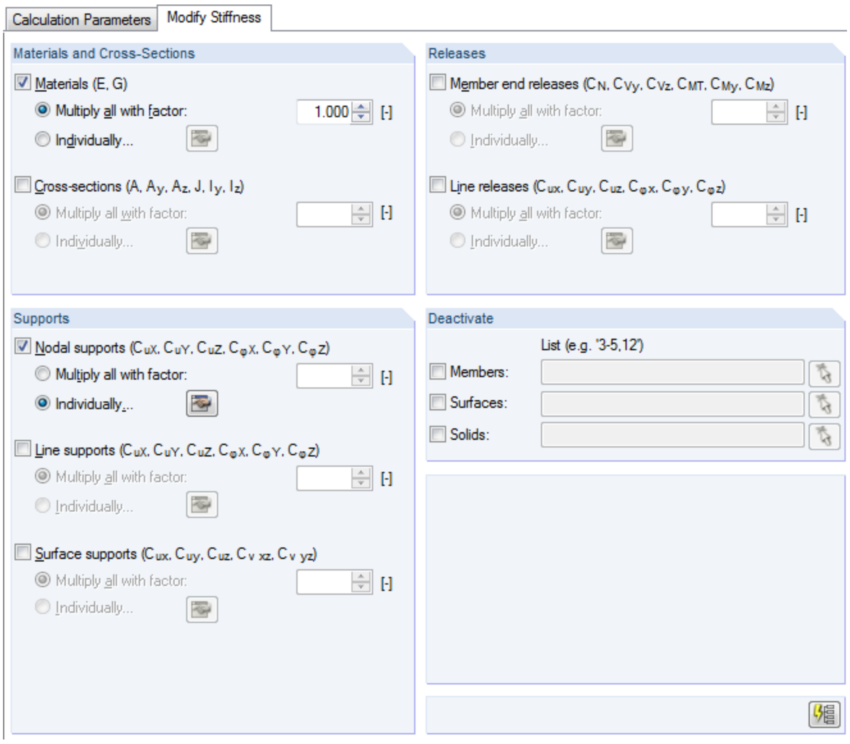


Figure 7.18: Dialog tab *Modify Stiffness*

Settings entered in this dialog tab affect only the load case or load combination that is selected in the list to the left. The button [Apply settings] transfers the current parameters to all load cases, respectively combinations.

Materials and cross-sections / supports / releases

With specifications defined in the three dialog sections you can decide how stiffnesses of the different model parameters are taken into account in the calculation.

- *Multiply all with factor*

  Specify a factor by which the stiffness of the materials, cross-sections, supports and releases is globally multiplied.

- *Individually*

  Use the [Modify] button to open a new dialog box where you can assign a specific stiffness factor to each object.

Deactivate

Use the three input fields to define which *Members*, *Surfaces* or *Solids* are not affected by the defined stiffness modifications, that means which are considered with the factor 1.0 in the calculation. You can select objects also graphically by using the [↖] function.

7.3.1.3 Dialog Tab *Extra Options*

The dialog tab is displayed only when the check box for *Activate extra options* is ticked in the dialog tab *Calculation Parameters* (see Figure 7.17, page 263).

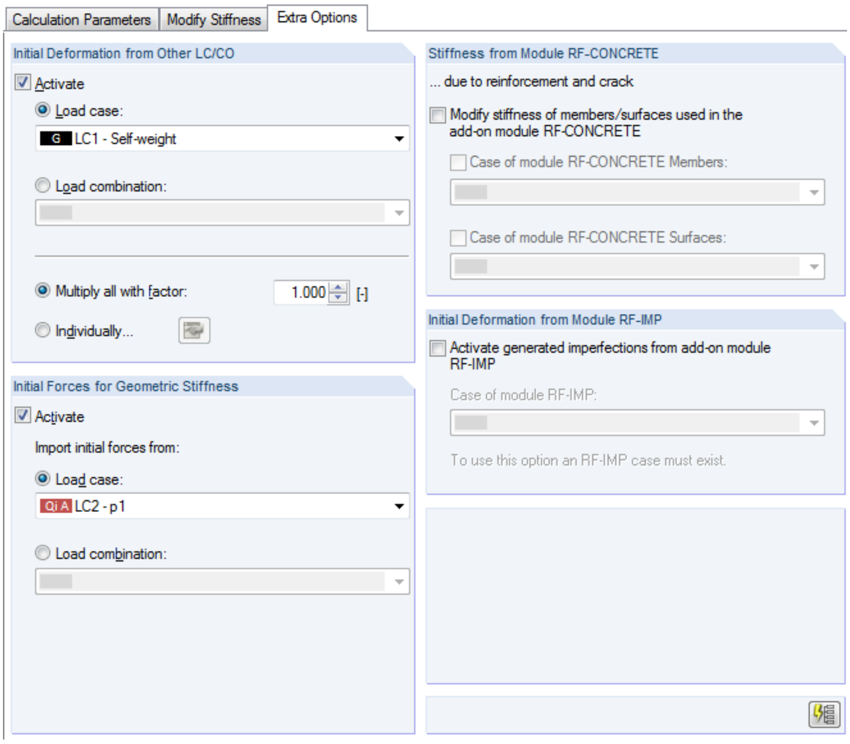


Figure 7.19: Dialog tab *Extra Options*

Initial deformation from other load case/combination

Select a load case or load combination whose deformations you want to consider as initial strain in the calculation. FE nodes are shifted accordingly before the calculation starts. If results are not yet available for the selected load case or combination, they will be calculated automatically.

Specify the factor by which you want to scale the deformations:

- *Multiply all with factor*
  Deformations of members, surfaces and solids are globally multiplied by the specified factor.
- *Individually*
  
  Use the [Edit] button to open a new dialog box where you can assign a specific scaling factor of deformation to each member, surface and solid.

Initial forces for geometric stiffness

You can select a load case or combination whose axial forces you want to use for an initial deformation. In this way, you can consider for example the stabilizing effect of another load case (than specified above in the dialog section *Initial Deformation*).

Access to the dialog section is only available for a calculation according to second-order analysis.

Stiffness from add-on module RF-CONCRETE

Stiffnesses of reinforced concrete elements, resulting from reinforcements and crack analysis according to nonlinear design method performed in the RF-CONCRETE modules, can be taken into account in the calculation. After ticking the check box, specify the analysis case of the add-on module **RF-CONCRETE Members** or **RF-CONCRETE Surfaces**.

Calculations with stiffnesses from RF-CONCRETE succeed only if design cases have been created and if designs are possible without non-designable situations.

Initial deformation from add-on module RF-IMP

Imperfections can be considered in the form of a pre-deformed equivalent model created in the add-on module **RF-IMP**. When the program is not licensed, equivalent imperfections for members (see chapter 6.13, page 246) or initial deformations of a load case (see above) can be applied manually.

The calculation on the pre-deformed equivalent model is only possible if this model has been previously created in the add-on module RF-IMP. The add-on module generates imperfections on the basis of eigenvalues from RF-STABILITY, RF-DYNAM or of deformations from an RFEM load case that are scaled to a maximum ordinate.

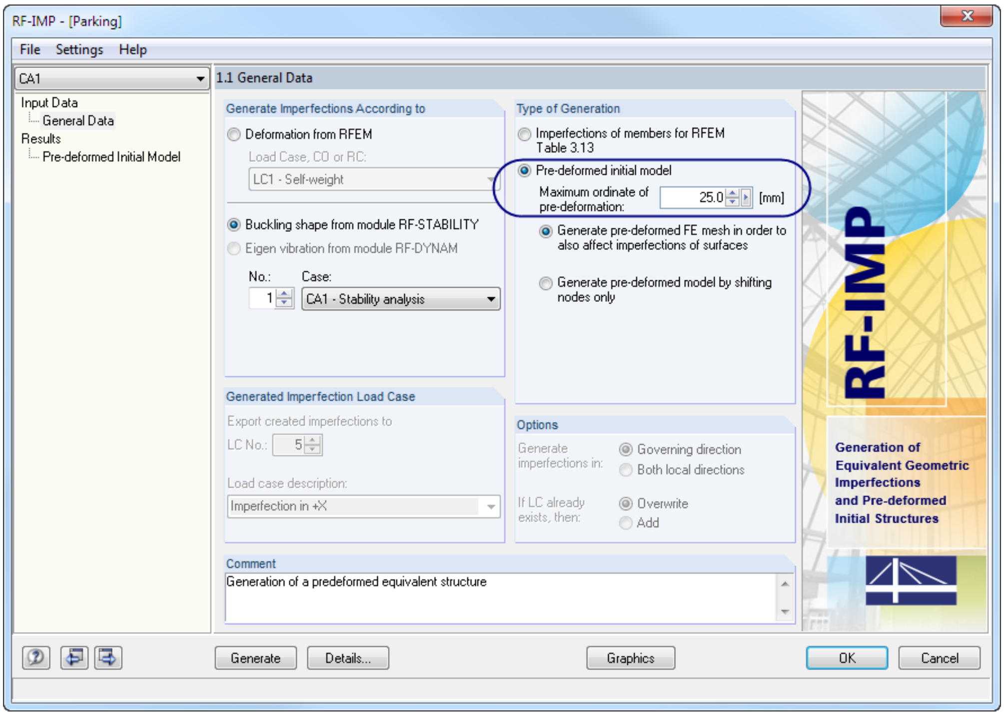


Figure 7.20: Add-on module RF-IMP with type of generation *Pre-deformed initial model*

The equivalent model is stored separately without changing the characteristic geometrical data. Coordinates of FE nodes are aligned with the equivalent model not until load combinations are calculated.

You can use a different equivalent model for each load combination. Select the relevant case in the list *Case of module RF-IMP*.

7.3.2 Result Combinations

For basic information about superpositioning load cases in result combinations, see chapter 5.6 *Result Combinations* on page 201.

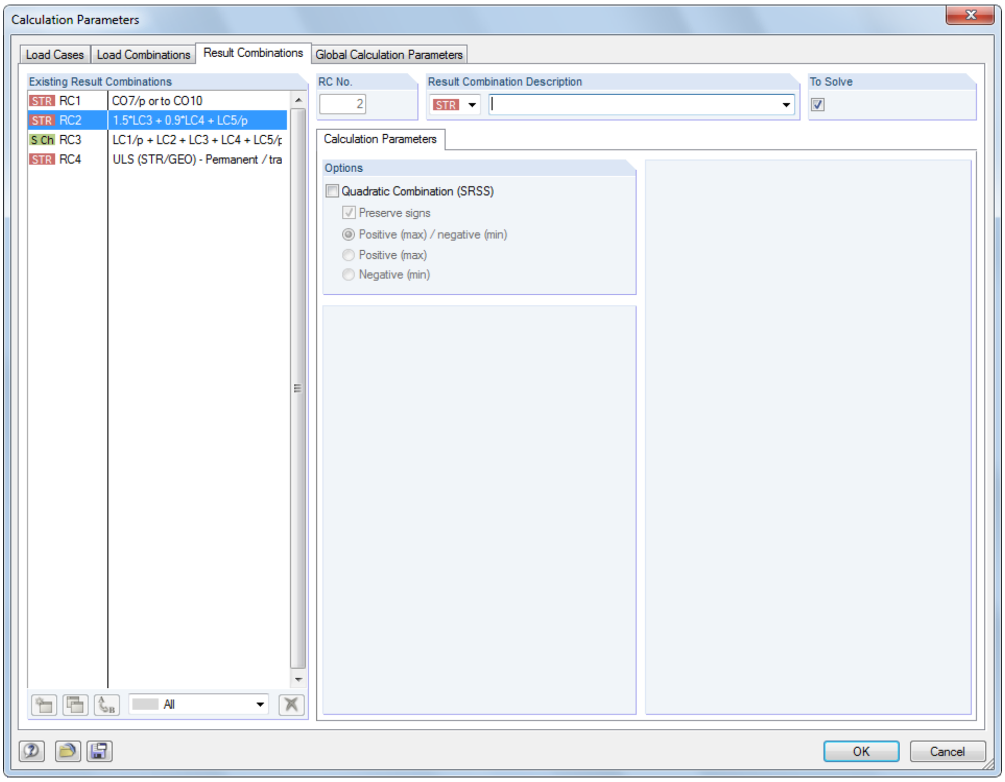


Figure 7.21: Dialog box *Calculation Parameters*, tab *Result Combinations*

In the dialog section *Existing Result Combinations*, you find a list of all created or generated result combinations. You can edit the *Calculation Parameters* of the selected entry in the dialog section to the right.

Options

The *Quadratic Combination* is deactivated by default. Thus, internal forces are superimposed by additive superposition:

$$B = A_1 + A_2 + \ldots + A_n$$

Equation 7.3

The default setting is appropriate for most application cases. A square addition of internal forces is relevant for dynamic analyses, for example when combining load cases due to centrifugal forces. In this case, the Pythagorean sum is created as follows:

$$B = \sqrt{A_1^2 + A_2^2 + \ldots + A_n^2}$$

Equation 7.4

When the square addition is activated, you can use the *Positive/Negative* options to decide which extreme values of the load cases will be considered in the super combination, and if you want to *Preserve signs*. In this way, the extreme values of the modal internal forces and deformations as well as the results belonging to the governing component can be determined conforming to signs.

7.3.3 Global Calculation Parameters

The dialog tab *Global Calculation Parameters* manages settings generally applied to all load cases and load combinations. To open the corresponding dialog box,

select **Calculation Parameters** on the **Calculate** menu

or use the toolbar button shown on the left.

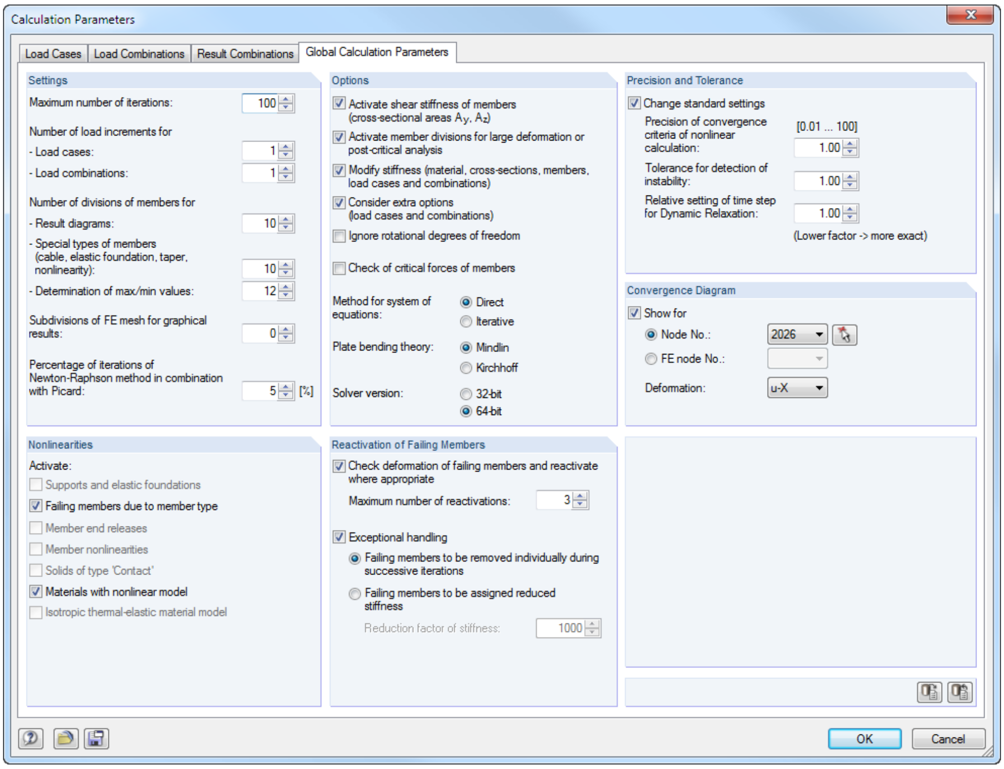


Figure 7.22: Dialog box *Calculation Parameters*, tab *Global Calculation Parameters*

Settings

Maximum number of iterations

When using second-order or large deformation analysis as well as objects that are nonlinearly effective, you have to calculate iteratively. The value of the input field defines the highest possible number of calculation runs. The specification has nothing to do with the iterative method set for the system of equations described for the dialog section *Options*.

When the calculation reaches the maximum number of iterations without achieving an equilibrium, RFEM displays a corresponding message. The results can be displayed anyway.

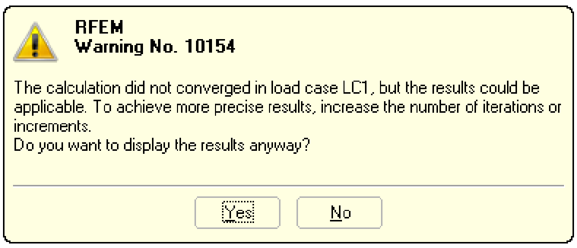


Figure 7.23: Message displayed for convergence problem

Number of load increments

Specifications of this input field take effect only for calculations according to second-order or large deformation analyses. Finding an equilibrium is often difficult when large deformations are considered. Instabilities can be avoided by applying the load in several steps.

For example, when two load increments are specified, half of the load will be applied in the first step. Iterations will be carried out until the equilibrium is found. Then, in the second step, the complete load will be applied to the already deformed system and iterations will be run again until the state of equilibrium is reached.

Please keep in mind that load increments have an unfavorable effect on the computing time. Therefore, the value *1* (no gradual load increment) is preset in the input field.

Moreover, you can define for each load case and combination how many load increments you want to apply (see chapter 7.3.1.1, page 267). Then, the global specifications will be ignored.

Number of divisions of members for result diagrams

The input field affects the graphical result diagram of members that have no other, distinct FE mesh division (for example due to a FE mesh refinement or a connected surface). If a division of *10* is set, RFEM divides the length of the longest member in the system by 10. With the system-related division length RFEM determines for each member the graphical result distributions on the division points.

In the dialog box *FE Mesh Settings*, there is another division option provided for straight members that are not integrated in surfaces (see Figure 7.10, page 258). With this option you can create FE nodes on all free members whose results are used for the graphical result diagrams.

Number of divisions of members for special types of members (cable, elastic foundation, taper, nonlinearity)

In contrast to the previous division option, a real division of the member is now defined by internal intermediate nodes. The specification affects cables, foundation members (contact stresses), tapered members (interpolation of cross-section values) and members with plastic properties (yielding zones) if they have not yet been divided otherwise by FE nodes: This division is irrelevant if a member is placed on a boundary line of a surface or if the definition line has a FE mesh refinement.

Number of divisions of members for determination of max/min values

The value specifies the internal division by which the maximum and minimum internal forces of members are determined. Thus, the division (default setting: *12*) represents the basis for the extreme values shown in the results tables and graphic. The division is also used for calculating member internal forces of load combinations.

Subdivisions of FE mesh for graphical results

The division controls the accuracy of graphical distributions within the finite elements. The following example compares the results with the divisions of *0* and *3*.

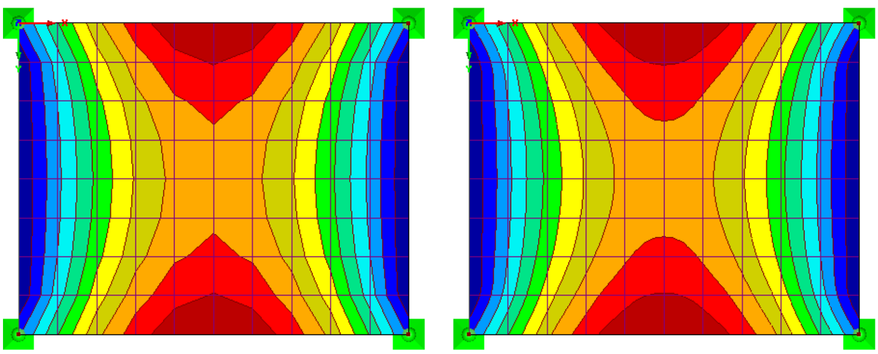


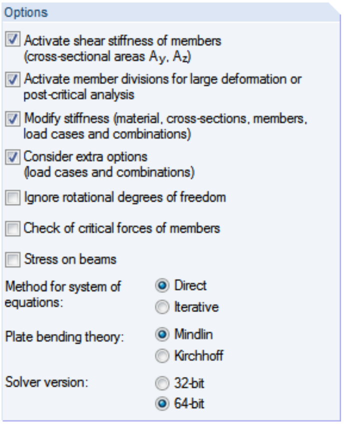
Figure 7.24: Graphical result diagram m-x with division *0* (left) and *3* (right)

Percentage of iterations of *Picard method in combination with Newton-Raphson*

The approach according to PICARD acts on the assumption of secant stiffenings, but the method of NEWTON-RAPHSON assumes tangent stiffenings (see chapter 7.3.1, page 264). When the calculation option *Newton-Raphson combined with Picard* is selected, secant stiffenings are used in the first iterations before tangent stiffenings are applied for the remaining iterations.

In the input field, you can define the percentage of the first iterations with secant stiffenings. Specify the value in relation to the total number of iterations.

Options



Activate shear stiffness of members (cross-sectional areas A_y, A_z)

Considering shear stiffnesses leads to an increase of deformations due to shear forces. The shear deformation is almost irrelevant for rolled and welded cross-sections. For solid and timber cross-sections, however, it is recommended to consider the shear stiffnesses for the deformation analysis.

Shear deformations have an effect only on the end nodes of members. Therefore, a single-span beam must be divided by intermediate nodes so that the increase becomes effective.

Activate member divisions for large deformation or post-critical analysis

Beams can be divided by intermediate nodes for the calculation according to large deformation analysis to calculate such members with a higher accuracy. The number of divisions is taken from the input field for cable and foundation members.

Modify stiffness (material, cross-sections, members, load cases and combinations)

Use this check box to define globally whether the adjustment factors of the stiffnesses for materials (see chapter 4.3, page 61), cross-sections (see chapter 4.13, page 119) and members (see chapter 4.17, page 142) are considered in the calculation of load cases and load combinations. Factors in the member and cross-section dialog boxes are preset each with 1.00. Thus, the check mark in the check box usually involves no reduction or increase of the stiffnesses.

Consider extra options

If *Extra Options* have been defined for the calculation parameters of load cases and combinations (see chapter 7.3.1.3, page 268), it is possible to activate or deactivate them by ticking and clearing this check box.

Ignore rotational degrees of freedom

Usually, six degrees of freedom must be considered in spatial structural systems. To save time, the option allows you to calculate models with only three degrees of freedom. The following simplifications are applicable: Only displacements are possible, but no rotations. Only axial forces are calculated as internal forces, but no bending or torsion moments and no shear forces – as if the system consisted only of truss girders, cables or membrane surfaces.

When this approach is used, the stiffness matrix is reduced to the half of the rows and tables, which affects computing time accordingly.

Check of critical forces of members

Often, exceeding the critical load already in the first iteration leads to an instability message. Use this check box to control if the critical load is checked for trusses, compression and buckling members. The defined effective lengths of members will be taken into account.

Method for system of equations

Both options control the method used for solving equation systems: *Direct* or *Iterative*. To prevent misunderstandings: When solving the equation system directly, an iterative calculation is performed as well, if nonlinearities are available or data is calculated according to second-order and large deformation analysis. *Direct* and *Iterative* refer to the data management during the calculation.

The choice of the solver method leading quickly to results depends on the complexity of the model as well as the size of the main memory (RAM) that is available:

- For small and medium size systems, the *Direct* solver method is more effective.
- For big and complex systems, the *Iterative* method is faster for results output.

When the matrices for the direct method cannot be stored any longer in the main memory, Windows starts to swap out parts of the data to the hard disk, which slows down the direct calculation. Hard disk activities increase and processor load is reduced, which is visible in the Windows Task Manager. By using the iterative ICG (*Incomplete Conjugate Gradient*) calculation method you can avoid this storage problem.

It is necessary to ensure that the swap file is large enough respectively the file size is assigned automatically by Windows. With a swap file that is too small program crashes may occur.

In the toolbar menu **Options**, select **Program Options**, or use the toolbar button shown on the left to open the dialog box *Program Options*. In the dialog tab *Help Assistant*, you can define the number of 2D and 3D elements for which, when it is exceeded, RFEM will show a warning of the calculation using the direct method.

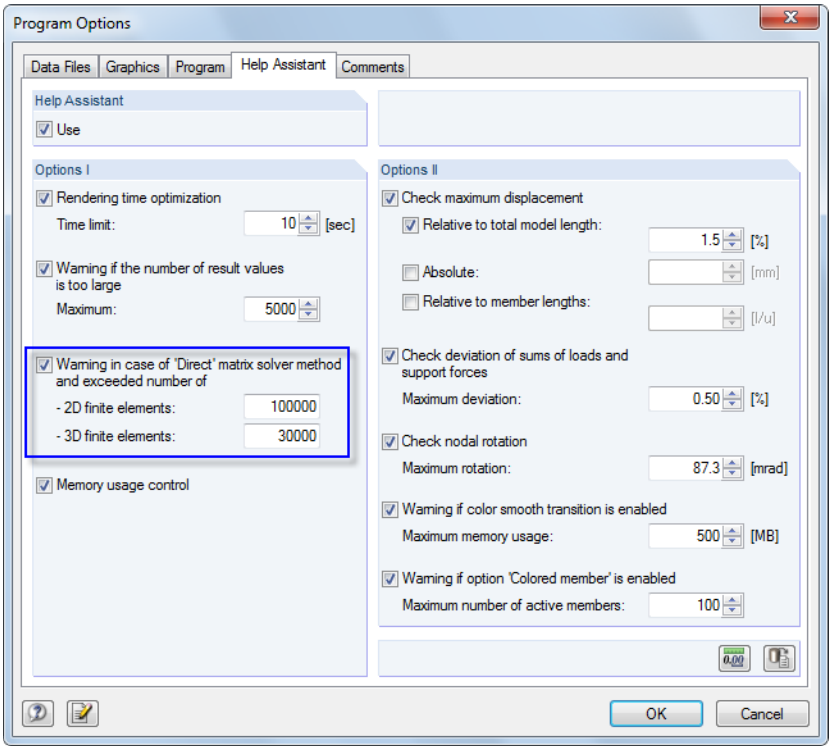


Figure 7.25: Dialog box *Program Options*, tab *Help Assistant*

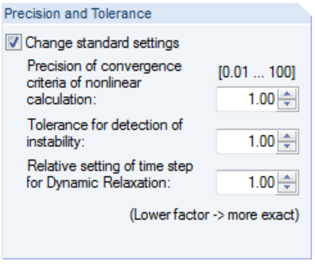
Plate bending theory

Surfaces can be calculated according to the bending theories of MINDLIN or KIRCHHOFF. The calculation according to MINDLIN includes shear force deformations, in the calculation according to KIRCHHOFF they are not considered. Therefore, the *Mindlin* calculation option is appropriate for relatively thick plates and shells used in solid construction. The *Kirchhoff* option is recommended for surfaces that are relatively thin like metal sheets in steel construction.

Solver version

The direct method for solving the system of equations (see above) is based on an analysis core using the extended RAM memory capacities of 64-bit operating systems. In this way, it is possible to calculate load cases and load combinations all at once even for big structural systems so that you can save time, provided that no object nonlinearities become effective. The RAM memory must be large enough in order to cover the stiffness matrix and all loading.

Precision and tolerance



Only rarely, it is required to adjust the preset convergence and tolerance parameters. Tick the check box *Change standard settings* to enable the input fields below.

Precision of convergence criteria of nonlinear calculation

If nonlinear effects are involved, or if the calculation is carried out according to second-order or large deformation analysis, you can influence the calculation by means of convergence criteria.

The change of axial forces of the two last iterations is compared member by member. As soon as the change reaches a specific fractional amount of the maximum axial force, the calculation stops. It is possible, however, that axial forces are swinging between two values during the iteration process instead of converging. With the value entered in this input field, the sensitivity can be defined in order to neglect these oscillation effects.

The accuracy also effects the convergence criterion for deformation changes in calculations according to large deformation analysis where geometric nonlinearities are considered.

The default value is 1.0. The minimum factor is 0.01, the maximum value is 100.0. The higher the factor is, the less sensitive the break-off limit will be.

Tolerance for detection of instability

There are different approaches by which the stability behavior of a model can be analyzed. However, not any of them is able to detect singular stiffness matrices with absolute reliability.

RFEM uses two procedures to determine the instability: On the one hand, elements on the principal diagonal of the stiffness matrix are compared with the same number in the iterations. On the other hand, each element of the principal diagonal is analyzed in relation to the adjacent number. The tolerance can be adjusted in the input field. The smaller the value is, the less 'sensitive' the analysis is performed.

Relative setting of time step for dynamic relaxation

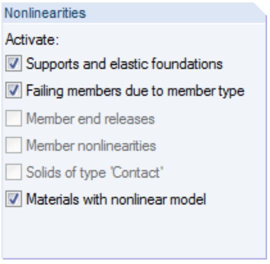
The time parameter controls the calculation by the method of dynamic relaxation (see 7.3.1.1, page 265). The smaller the value is, the smaller is the time step and the more accurate the results will be.

Convergence diagram

During the calculation, a diagram showing the developing of displacements and rotations is displayed (see Figure 7.31, page 279). The values displayed refer to the node with the maximum deformations.

In the dialog section *Convergence Diagram*, you have the possibility to define a particular node with deformation component whose results will be shown additionally in the dialog box *Convergence Diagram* (see Figure 8.2, page 281).

Nonlinearities

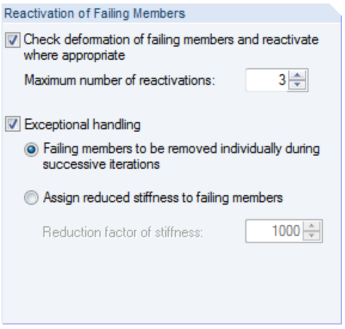


When nonlinearly acting elements are used in the model, you can deactivate the effect of the following elements for the calculation:

- Failing supports/elastic foundations (→ chap. 4.7, p. 97, chap. 4.8, p. 103, chap. 4.9, p. 108)
- Failing members (→ chapter 4.17, p. 140)
- Member end releases (→ chapter 4.14, p. 128)
- Member nonlinearities (→ chapter 4.20, p. 155)
- Contact solids (→ chapter 4.5, p. 89)
- Material nonlinearities (→ chapter 4.3, p. 61)

It is recommended to not suppress the nonlinear effects but for test purposes, for example for finding the cause of an instability. The options in this dialog section help you to find errors: Sometimes, inaccurately defined failure criteria is responsible for calculation break-offs.

Reactivation of failing members



Settings in this dialog section concern member elements that may fail (for example tension, compression or foundation members). Take advantage of the options to solve problems of instability caused by failing members: A model for example is stiffened by ties. Because of post shortenings due to vertical loads, the tension members receive small compressive forces in the first calculation step. They will be removed from the system. Then, in the second calculation run, the model is unstable without the ties.

Check deformation of failing members and reactivate where appropriate

When the check box is ticked, RFEM analyzes the nodal displacements in each iteration. If member ends of a failed tie move away from each other, the member will be reactivated.

In some cases, reactivating members may be problematic: A member is removed after the first iteration, but reactivated after the second one, removed again after the third iteration etc. The calculation would run this loop until reaching the maximum number of iterations without converging. This effect can be avoided by defining a *Maximum number of reactivations* specifying how often a member element is permitted to be reactivated before it will be definitely removed from the stiffness matrix.

Exceptional handling

After ticking the check box two methods for handling failing members are available for selection. They can be combined with the reactivation options described above.

- **Failing members to be removed individually during successive iterations**

After the first iteration RFEM does not remove for example all tension members with a compression force but only the tie with the greatest compressive force. Then, in the second iteration, only one member is missing in the stiffness matrix. In the next step, RFEM removes again the tie with the greatest compressive force. Often, a better convergence behavior can be achieved in this way for the system because of redistributing effects.

This calculation option requires more time because the program must run through a larger number of iterations. Furthermore, you have to make sure that a sufficient number of possible iterations is set in the *Settings* dialog section above.

- **Assign reduced stiffness to failing members**

Members which have failed are not removed from the stiffness matrix. Instead, RFEM assigns a very small stiffness to them. Specify it in the input field *Reduction factor of stiffness*: Factor 1000 means a reduction of stiffness to 1/1000.

Please keep in mind for this calculation option that RFEM displays on members small internal forces which can actually not be absorbed by the member due to its definition.

7.4 Start Calculation

You can select between several options for the calculation start. Before you start the calculation, it is recommended to carry out a short plausibility check of input data (see chapter 7.1.1, page 251).

Calculate all

To start the corresponding function,

select **Calculate All** on the **Calculate** ***menu***

or use the toolbar button shown on the left.

Figure 7.26: Button [Calculate All]

The command starts the calculation of all load cases, load combinations and result combinations as well as of all additional modules where input data is available.

Please use the function [Calculate All] with care:

- Many load cases cannot occur isolated. Wind loads, for example, act always together with the self-weight. For structures with failing supports for tension, instabilities may occur during the batch calculation of all single load cases.
- If many load combinations and module design cases are available, RFEM may need a lot of computing time.

Calculate selected load cases

To open the dialog box for selecting the load cases which are relevant for calculation,

select **To Calculate** on the **Calculate** menu.

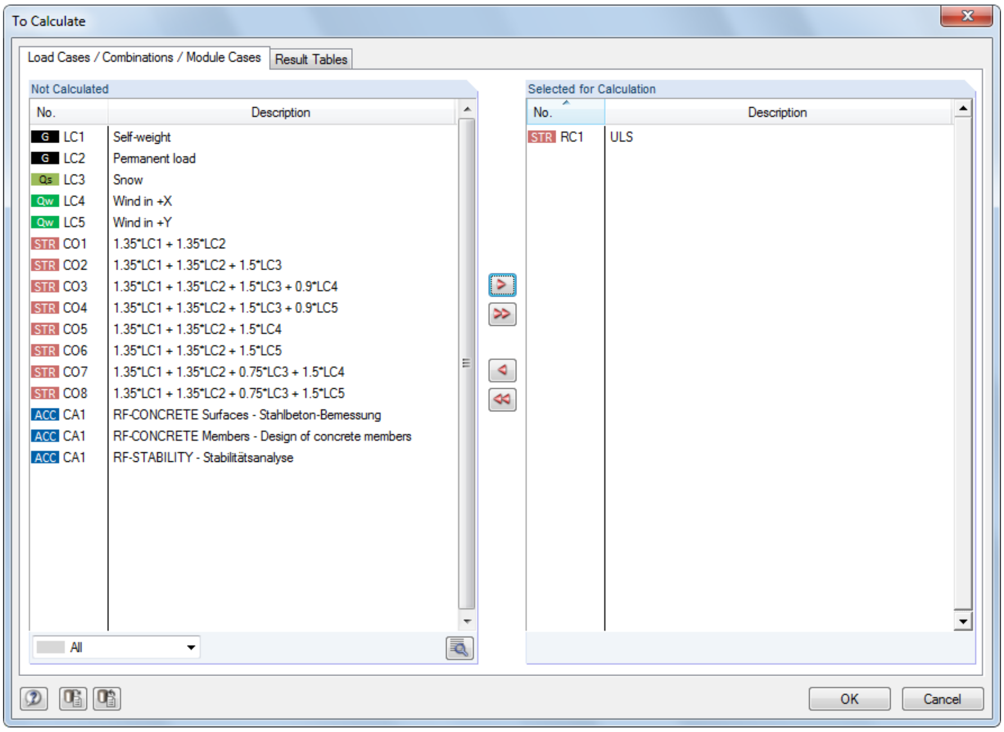
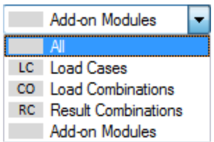


Figure 7.27: Dialog box *To Calculate*

In the dialog section *Not Calculated* on the left, RFEM lists all load cases, load combinations and result combinations as well as analysis cases of add-on modules for which no results exist. Use the button [▶] to transfer the selected entries to the list *Selected for Calculation*. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].

If result combinations or module cases are selected which require results from load cases, the relevant load cases will be calculated automatically.



Load items can be sorted by filter options available below the list according to the following criteria:

- Load cases
- Load combinations
- Result combinations
- Add-on modules

The button [Calculation Parameters] shown on the left opens the dialog box *Calculation Parameters* (see chapter 7.3, page 271) where settings can be checked and adjusted for the calculation.

The dialog tab *Result Tables* of the *To Calculate* dialog box controls the availability of tables shown after the calculation.

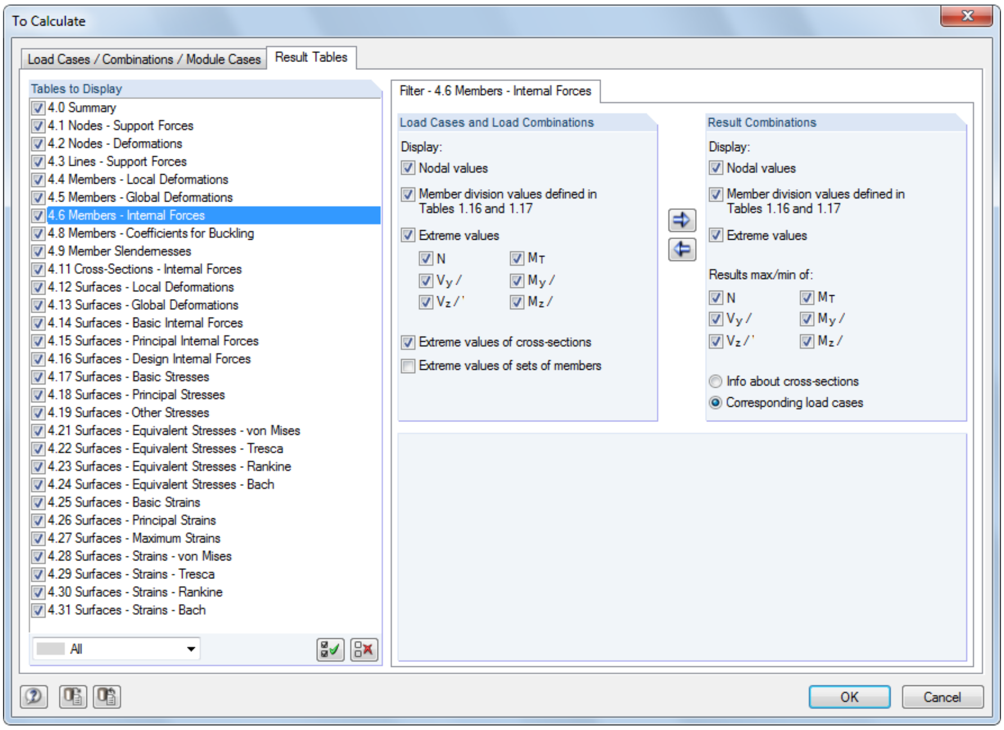


Figure 7.28: Dialog box *To Calculate*, tab *Result Tables*

More filter options are available for some results tables. They are presented in chapter 8 *Results* together with the respective output tables (see for example Figure 8.15, page 290).

Calculate current load case

It is possible to start the calculation of an individual load case directly: Select the load case, load or result combination in the toolbar list, and then click the button [Show Results].

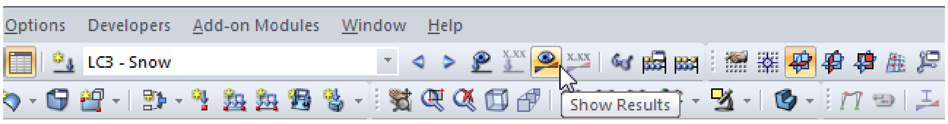


Figure 7.29: Calculating the load case directly by using the button [Show Results]

The calculation can be started after a message has been displayed that no results have been found.

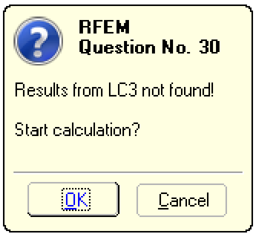


Figure 7.30: Query before calculation

Calculate selected results

The toolbar menu *Calculate* offers you additional options for selecting results to be calculated:

- RFEM results only
- Modules results only
- All results of all open models
- RFEM results only of all open models
- Modules results only of all open models

The calculation starts immediately after calling the corresponding function.

Calculation process

The calculation process is shown in the *FE-Calculation* window. In a diagram showing the convergence process you can observe the graphs of the maximum displacements in addition to the calculation steps RFEM is running through.

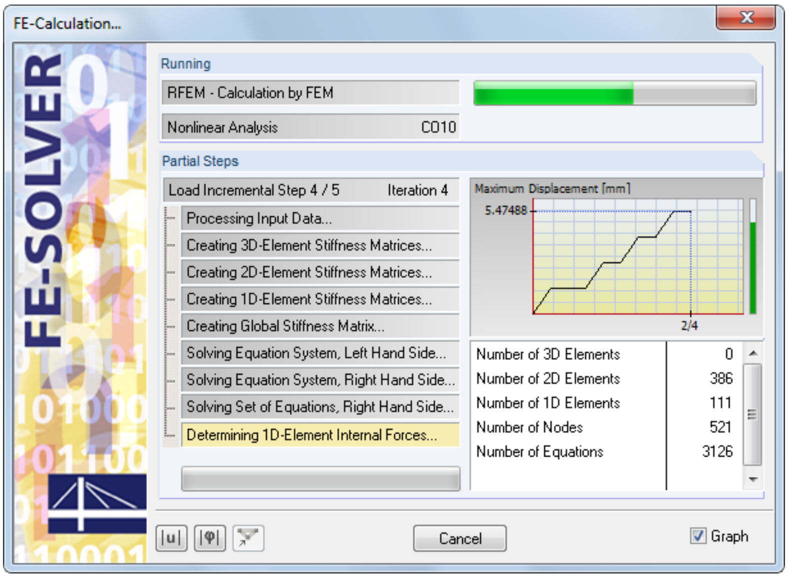


Figure 7.31: Calculation process

The green or red vertical bars on the right in the window visualize the convergence behavior during the calculation: Each load increment takes a part of the column, for example 4/5 in the figure above represents the fourth of five load increments. When the bar is green, deformations are in a tolerable zone. A red bar symbolizes oversized displacements or rotations (≥ 0.1 rad).

8. Results

The numbering of this manual chapter follows the numbering of results tables, which makes it easier to find the respective descriptions of the tabs.

Please note that any FE analysis represents an approximation. Results should be interpreted and checked for plausibility with engineering know-how.

When data has been calculated, the additional tab *Results* appears in the navigator (see chapter 3.4.3, page 24) for controlling the graphical results display. The results are listed numerically in separate tables (see chapter 3.4.4, page 26).

Colored relation scales in tables

The result columns of tables are partly highlighted in red or blue (see Figure 8.4, page 282. The colored bars represent result values graphically. They are scaled to the extreme values of the internal forces or deformations of all objects. Negative values are symbolized by red bars, positive ones by blue bars. Thus, the table allows also for a visual result evaluation.

To switch the colored bars on and off,

select **View on the Table menu**, and then click **Colored Relation Scales**

or use the button in the table toolbar shown on the left.

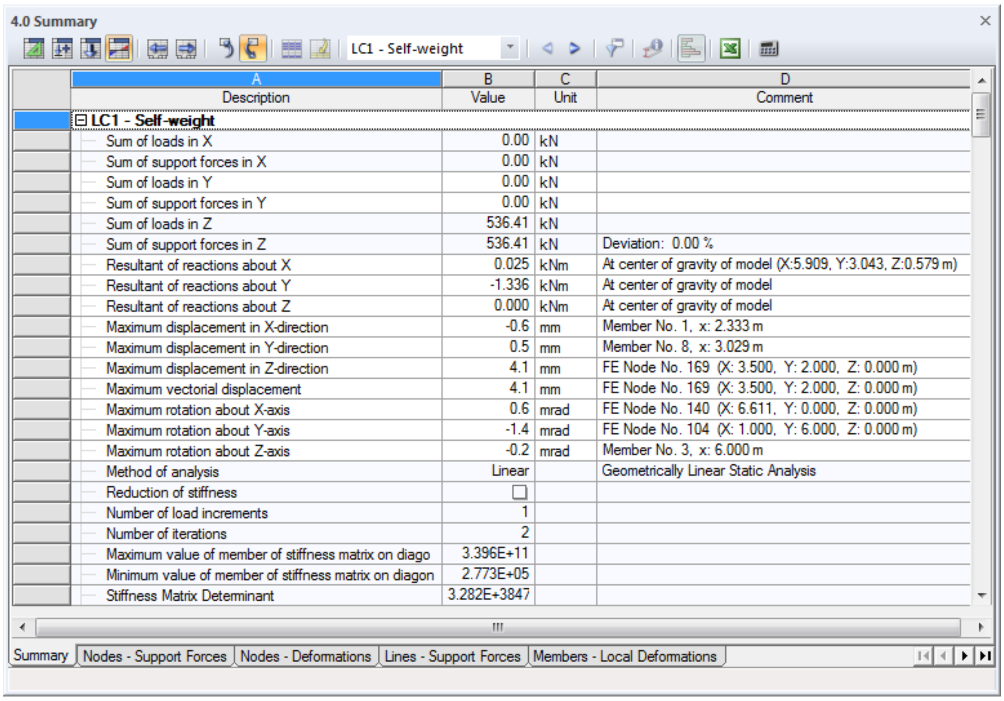
Table filter

The displayed tables depend on the selections set in the dialog tab *Result Tables* of the dialog box *To Calculate* (see chapter 7.4, page 278).

8.0 Results Summary

Table

Table 4.0 *Summary* represents a summary of the calculation process, sorted by load cases and combinations.



4.0 Summary

A Description	B Value	C Unit	D Comment
LC1 - Self-weight			
Sum of loads in X	0.00	kN	
Sum of support forces in X	0.00	kN	
Sum of loads in Y	0.00	kN	
Sum of support forces in Y	0.00	kN	
Sum of loads in Z	536.41	kN	
Sum of support forces in Z	536.41	kN	Deviation: 0.00 %
Resultant of reactions about X	0.025	kNm	At center of gravity of model (X:5.909, Y:3.043, Z:0.579 m)
Resultant of reactions about Y	-1.336	kNm	At center of gravity of model
Resultant of reactions about Z	0.000	kNm	At center of gravity of model
Maximum displacement in X-direction	-0.6	mm	Member No. 1, x: 2.333 m
Maximum displacement in Y-direction	0.5	mm	Member No. 8, x: 3.029 m
Maximum displacement in Z-direction	4.1	mm	FE Node No. 169 (X: 3.500, Y: 2.000, Z: 0.000 m)
Maximum vectorial displacement	4.1	mm	FE Node No. 169 (X: 3.500, Y: 2.000, Z: 0.000 m)
Maximum rotation about X-axis	0.6	mrad	FE Node No. 140 (X: 6.611, Y: 0.000, Z: 0.000 m)
Maximum rotation about Y-axis	-1.4	mrad	FE Node No. 104 (X: 1.000, Y: 6.000, Z: 0.000 m)
Maximum rotation about Z-axis	-0.2	mrad	Member No. 3, x: 6.000 m
Method of analysis	Linear		Geometrically Linear Static Analysis
Reduction of stiffness	☐		
Number of load increments	1		
Number of iterations	2		
Maximum value of member of stiffness matrix on diago	3.396E+11		
Minimum value of member of stiffness matrix on diagon	2.773E+05		
Stiffness Matrix Determinant	3.282E+3847		

Summary | Nodes - Support Forces | Nodes - Deformations | Lines - Support Forces | Members - Local Deformations

Figure 8.1: Table 4.0 *Summary*

This overview shows you the check sums of loadings and support forces. The deviations in each direction should be less than 1 %. If this is not the case, numerical problems have occurred because of considerable differences in stiffness. It may also be possible that the model has an insufficient stability, or the calculation has reached the maximum number of iterations without convergence. The overview informs you also about the resulting support reactions that are effective in an idealized way in the centroid of the model.

Moreover, the summary shows the maximum displacements and rotations related to the global axes X, Y and Z as well as the largest total displacement. Due to the check of deformations, reliability of results can be evaluated.

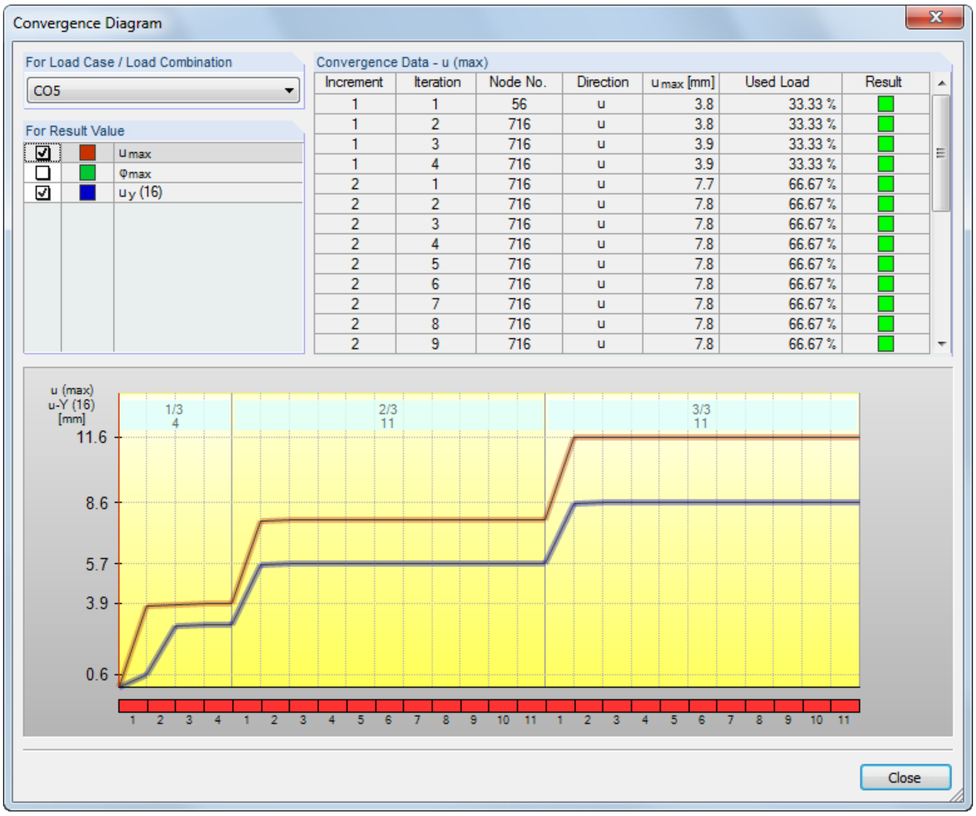
The summary that is listed by load cases is completed by the used calculation parameters. The *Number of iterations* required to obtain the results is of special interest here.

The table ends with a *Summary* of selected parameters of the analysis core as well as globally valid specifications of calculation (see Figure 7.22, page 271: dialog box *Calculation Parameters*, tab *Global Calculation Parameters*)

Convergence diagram

The deformation developing is represented graphically during the calculation (see Figure 7.31, page 279). You can use this iteration diagram also after the calculation for "record" evaluation. To open the corresponding dialog box,

select **Iteration Diagrams** on the **Results** menu.



Increment	Iteration	Node No.	Direction	u max [mm]	Used Load	Result
1	1	56	u	3.8	33.33 %	
1	2	716	u	3.8	33.33 %	
1	3	716	u	3.9	33.33 %	
1	4	716	u	3.9	33.33 %	
2	1	716	u	7.7	66.67 %	
2	2	716	u	7.8	66.67 %	
2	3	716	u	7.8	66.67 %	
2	4	716	u	7.8	66.67 %	
2	5	716	u	7.8	66.67 %	
2	6	716	u	7.8	66.67 %	
2	7	716	u	7.8	66.67 %	
2	8	716	u	7.8	66.67 %	
2	9	716	u	7.8	66.67 %	

Figure 8.2: Dialog box *Convergence Diagram*

The relevant load case can be selected in the list *For Load Case / Load Combination*.

In the dialog section *For Result Value*, the maximum displacements u_{max} and the maximum rotations φ_{max} are preset. When a particular node with corresponding deformation has been defined among the global calculation parameters (see Figure 7.22, page 271, dialog section *Convergence Diagram*), deformations related to this node are also contained in the list.

In the second half of the dialog box, you see the graphs of the deformations activated in the dialog section *For Result Value*.

With a click into one of the rows available in the dialog section *For Result Value*, you can update the result values of the respective deformation in the table *Convergence Data*. In this way, a specific evaluation of the iteration steps, governing nodes and deformation diagrams is possible.

Furthermore, the iteration diagram makes it possible to take remedial measures for "oscillating" (not converging) results. In addition, you can check the diagram of deformation in the iterations subsequently in case calculations are time-consuming.

Based on the convergence diagram, a load-deformation diagram can be created for load increments by copying the result values into the clipboard.

8.1 Nodes - Support Forces

With the entries under *Support Reactions* in the *Results* navigator you decide which components are displayed graphically in the work window. They can be related to the local axes of rotated supports or to the global axis system XYZ. Table 4.1 shows the support forces and moments in numerical form.

If the structure is a 2D model, RFEM displays only the table columns of support forces and moments that are relevant for a planar structural system.

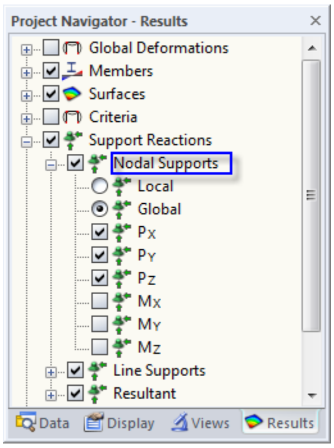
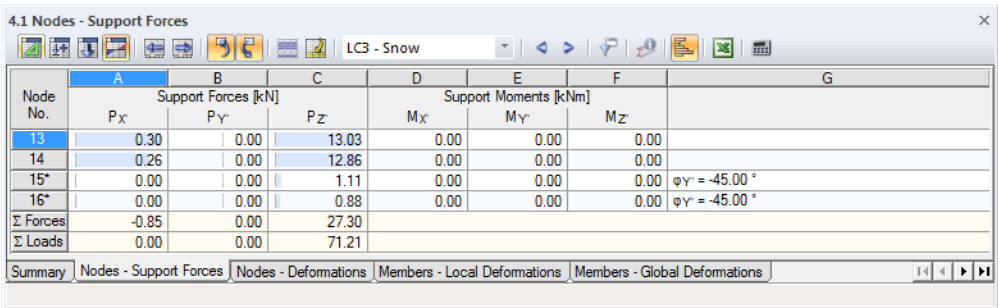
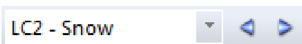


Figure 8.3: *Results* navigator: *Support Reactions* → *Nodal Supports*



Node No.	A Support Forces [kN] Px'	B Py'	C Pz'	D Support Moments [kNm] Mx'	E My'	F Mz'	G
13	0.30	0.00	13.03	0.00	0.00	0.00	
14	0.26	0.00	12.86	0.00	0.00	0.00	
15*	0.00	0.00	1.11	0.00	0.00	0.00	φY' = -45.00 °
16*	0.00	0.00	0.88	0.00	0.00	0.00	φY' = -45.00 °
Σ Forces	-0.85	0.00	27.30				
Σ Loads	0.00	0.00	71.21				

Figure 8.4: Table 4.1 *Nodes - Support Forces*



To display the support reactions of a particular load case, select the load case from the list in the main toolbar or the toolbar of the table.

Support forces P_X / P_Y / P_Z

The support forces are listed in three table columns where they are sorted by nodes. Usually, the forces refer to the axes X, Y and Z of the global coordinate system. To display the forces related to the local support axes X', Y' and Z' (rotated supports) in the graphic as well as in the table, go to the *Results* navigator and set **Support Reactions** → **Nodal Supports** → **Local**.

Nodes with support rotations are marked by an asterisk (*) as shown in Figure 8.4. Forces are put out in relation to the selected axis system. In the final table column, the support's rotation angle is indicated.

The table shows the forces which are passed into the support. Thus, with regard to signs, the table does not show the reaction forces on the part of the support. The signs result from the direction of the global axes. If the global axis Z is directed downwards, then the load case self-weight for example results in a positive support force P_Z, and a wind load against the global axis X has a negative support force P_X. Thus, the support forces shown in the table represent foundation loads.

In contrast, the green vectors displayed in the graphic of the work window show the reaction forces on the part of the supports. The components of the support reactions are visualized by the size and direction of the vectors.

You can display the signs of support reactions in the work window. Select *Results* in the *Display* navigator and tick the check box of the corresponding option.

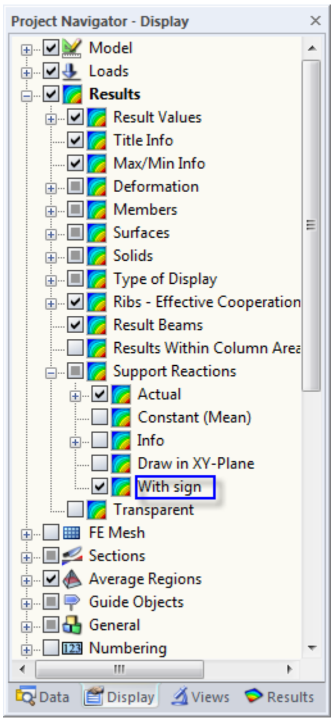


Figure 8.5: *Display* navigator, *Results* → *Support Reactions* → *With sign*

The signs in the graphic refer to the global axis system XYZ or the local axis system X'Y'Z' which is rotated. A positive support reaction acts in direction of the corresponding positive axis. For example, a wind load against the global axis X results in a positive support reaction P_X.

It is advisable to display these signs only for testing purposes. They may lead to misinterpretations because the vectors are already signed. The signs in the graphics are intended to be an extra feature of the vector display, indicating the directions for values in relation to the global axes.

Support moments M_X / M_Y / M_Z

The support moments are listed in three table columns where they are sorted by nodes. Usually, the moments refer to the axes X, Y and Z of the global coordinate system. Use the *Results* navigator to display the moments related to the local support axes X', Y' and Z' in the graphic as well as in the table.

The table shows the moments which are introduced into the support. With regard to signs, like for support forces, the table does not show the reactions on the part of the support. The signs result from the direction of the global axes. Thus, the support moments shown in the table represent foundation loads.

In the work window, however, reaction moments are shown on the part of the support.

Signs for support moments can be displayed in the graphic as well (see Figure 8.5). A positive support moment acts clockwise about the corresponding positive global axis. Similar to the vectors for support forces, vectors are already signed, and indications of value have to be considered independently: The signs indicate the directions of the moments in relation to the global axes.

In the graphic, support moments can be represented as vector or arc. To change the type of display,

point to **Display Properties** on the **Options** menu, and select **Edit**.

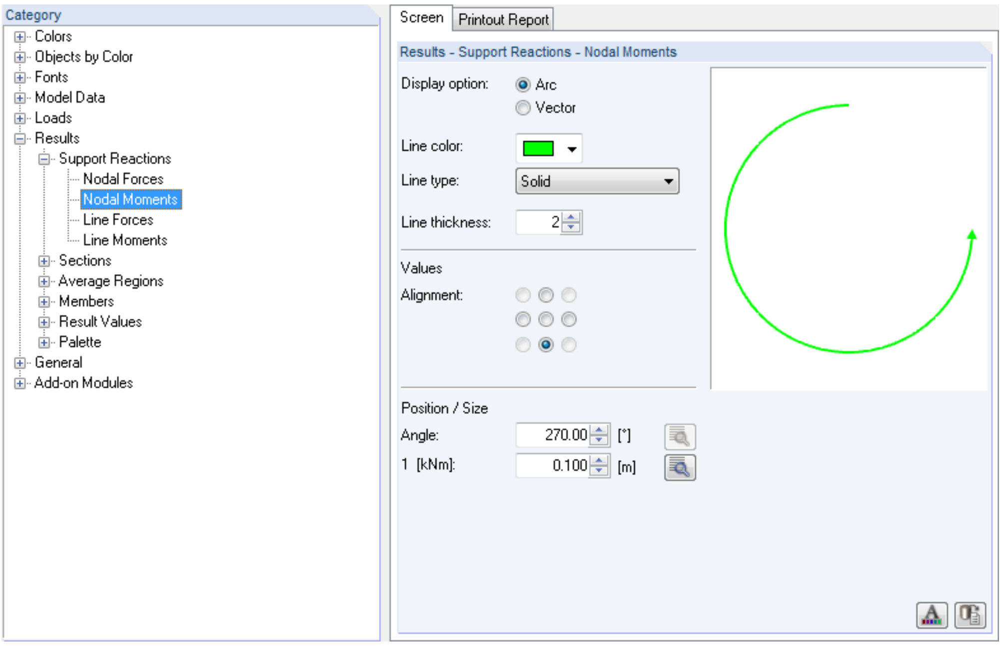


Figure 8.6: Dialog box *Display Properties* (dialog section): *Nodal Moments* with display option *Arc*

In the dialog section *Category* on the left, set *Results*, *Support Reactions* and *Nodal Moments*, and then select the *Display option Arc* to the right.

Rotated nodal supports

In the final table column, the rotation angles of rotated nodal supports are shown (see Figure 8.4, page 282). The corresponding nodes are marked by an asterisk (*).

Check sums

For load cases and load combinations RFEM displays check sums of loads and support reactions at the end of the table. Differences will occur between the sums of *Σ Forces* and *Σ Loads* if the model has additional line supports and members or surfaces with elastic foundations. Therefore, also the *Σ Forces* available in tables 4.3, 4.7 and 4.20 must be considered for the total summary.

Import support forces as load

Nodal support forces and moments of another RFEM model can be applied as loads in the model your are currently working on. In this way, loads can be transferred by floors to analyze 2D floor slabs. The function is described in chapter 8.3 on page 289.

Nodal support forces that have been imported are applied as free concentrated loads.

Filtering support forces of result combinations

For result combinations it is possible to adjust the default setting for the extreme values shown in the results table. To open the corresponding dialog box,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

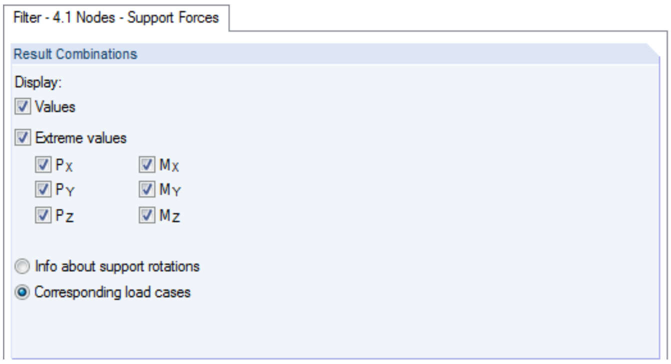


Figure 8.7: Dialog box *Table Filter* (dialog section)

The check boxes in the dialog box *Table Filter* control the type and amount of numerical output for support forces.

Resultant of support reactions

The resultants of support reactions for load cases and load combinations are shown in numerical form in table 4.0 *Summary* for each global direction (see Figure 8.1, page 280). Use the *Results* navigator to visualize the resultant forces also on the model.

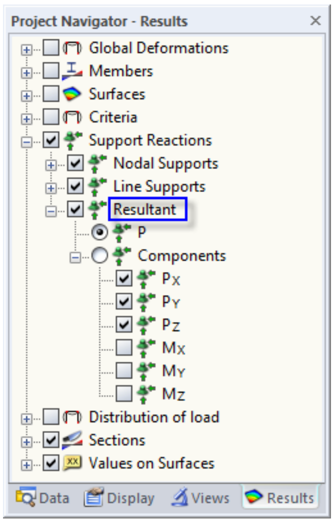


Figure 8.8: *Results* navigator: *Support Reactions* → *Resultant*

In addition to the total resultant *P*, it is possible to display the individual *Components* that are effective in an idealized way in the model's centroid. Thus, you can check the position and size of the resulting support forces at a glance.

8.2 Nodes - Deformations

To control the graphical display of nodal displacements and nodal rotations, tick the check box for *Global Deformations* in the *Results* navigator. Table 4.2 shows the deformations of nodes in numerical form.

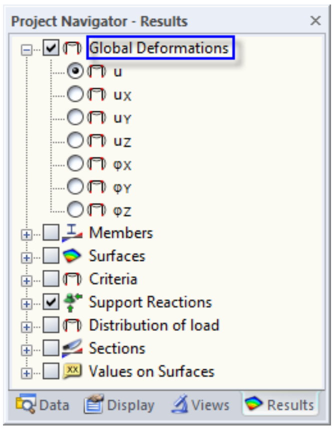
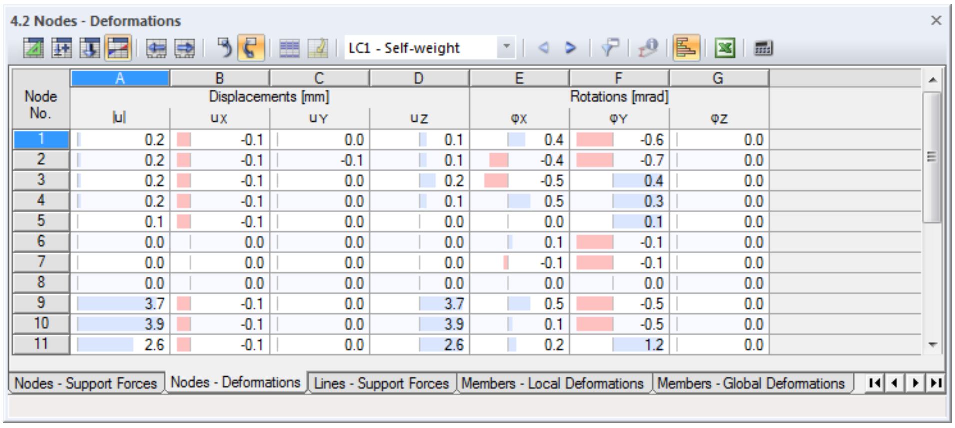


Figure 8.9: *Results* navigator: *Global Deformations*



	A	B	C	D	E	F	G
Node No.	Displacements [mm]				Rotations [mrad]		
	|u|	uX	uY	uZ	φX	φY	φZ
1	0.2	-0.1	0.0	0.1	0.4	-0.6	0.0
2	0.2	-0.1	-0.1	0.1	-0.4	-0.7	0.0
3	0.2	-0.1	0.0	0.2	-0.5	0.4	0.0
4	0.2	-0.1	0.0	0.1	0.5	0.3	0.0
5	0.1	-0.1	0.0	0.0	0.0	0.1	0.0
6	0.0	0.0	0.0	0.0	0.1	-0.1	0.0
7	0.0	0.0	0.0	0.0	-0.1	-0.1	0.0
8	0.0	0.0	0.0	0.0	0.0	0.0	0.0
9	3.7	-0.1	0.0	3.7	0.5	-0.5	0.0
10	3.9	-0.1	0.0	3.9	0.1	-0.5	0.0
11	2.6	-0.1	0.0	2.6	0.2	1.2	0.0

Figure 8.10: Table 4.2 *Nodes - Deformations*

The displacements and rotations are listed by nodes.

Displacements / Rotations

The deformations have the following meanings:

u	Total displacement
u_X	Displacement in direction of the global axis X
u_Y	Displacement in direction of the global axis Y
u_Z	Displacement in direction of the global axis Z
φ_X	Rotation about the global axis X
φ_Y	Rotation about the global axis Y
φ_Z	Rotation about the global axis Z

Table 8.1: Nodal deformations

8.3 Lines - Support Forces

With the entries under *Support Reactions* in the *Results* navigator you decide which components are displayed graphically in the work window. They can be related to the local axes of rotated supports or to the global axis system XYZ. Table 4.3 shows the support forces and moments in numerical form.

If the structure is a 2D model, RFEM displays only the table columns of support forces and moments that are relevant for a planar structural system.

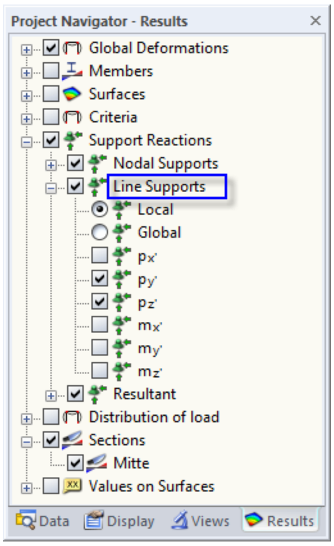
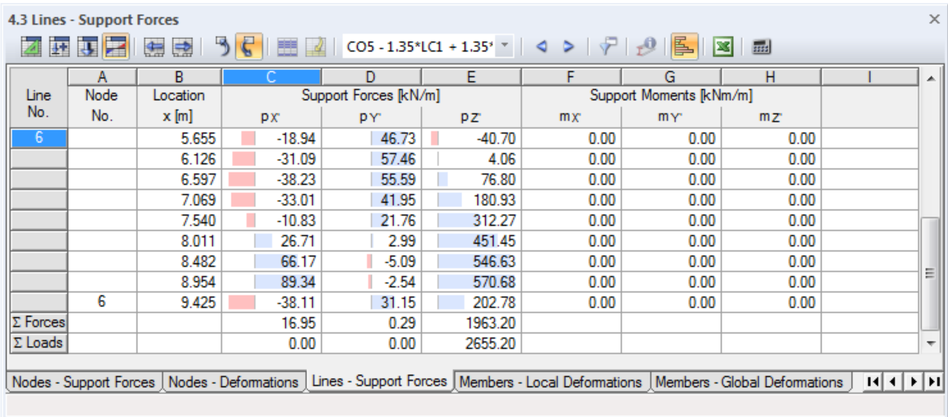


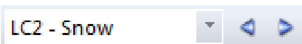
Figure 8.11: *Results* navigator: *Support Reactions* → *Line Supports*



4.3 Lines - Support Forces

Line No.	Node No.	Location x [m]	Support Forces [kN/m] px'	py'	pz'	Support Moments [kNm/m] mx'	my'	mz'
6		5.655	-18.94	46.73	-40.70	0.00	0.00	0.00
		6.126	-31.09	57.46	4.06	0.00	0.00	0.00
		6.597	-38.23	55.59	76.80	0.00	0.00	0.00
		7.069	-33.01	41.95	180.93	0.00	0.00	0.00
		7.540	-10.83	21.76	312.27	0.00	0.00	0.00
		8.011	26.71	2.99	451.45	0.00	0.00	0.00
		8.482	66.17	-5.09	546.63	0.00	0.00	0.00
		8.954	89.34	-2.54	570.68	0.00	0.00	0.00
	6	9.425	-38.11	31.15	202.78	0.00	0.00	0.00
Σ Forces			16.95	0.29	1963.20			
Σ Loads			0.00	0.00	2655.20			

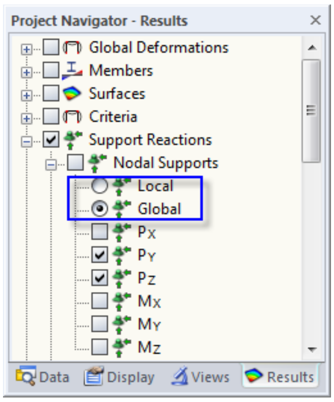
Figure 8.12: Table 4.3 *Lines - Support Forces*



To display the support reactions of a particular load case, select the load case from the list in the main toolbar or the toolbar of the table.

Location x

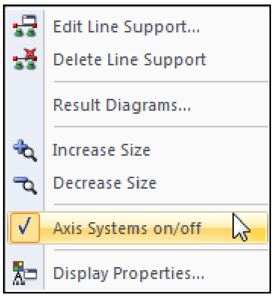
The support forces are listed by lines. The x-locations shown in the table column represent the spacings of FE nodes along the line. They are related to the start node of the line. The surface grid is not relevant for line support forces.



Support forces p_X / p_Y / p_Z

The support forces are listed in three table columns where they are sorted by lines. The forces can be related to the global axes X, Y and Z or the local axes X', Y' and Z' of the line supports. The axis reference in the table is controlled by settings in the *Results* navigator (see figure on the left).

The table shows the forces which are passed into the support. Thus, with regard to signs, the table does not show the reaction forces on the part of the support. If support forces refer to the global coordinate system, the signs result from the directions of the global axes. If the global axis Z is directed downwards, then the load case self-weight for example results in a positive support force p_Z, and a wind load against the global axis X has a negative support force p_X. Thus, the support forces shown in the table represent foundation loads.



Context menu of line support

If the local support forces $p_{X'}$, $p_{Y'}$ and $p_{Z'}$ are displayed, forces refer to the axes of the line supports X', Y' and Z'. Thus, the signs in the table for the introduced forces result from the directions of the local support axes. Use the *Display* navigator or the context menu of line supports to display these axes.

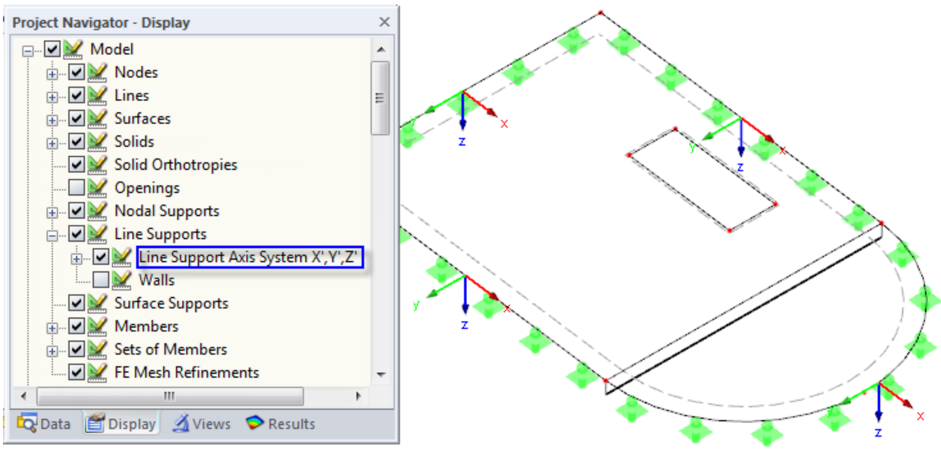


Figure 8.13: Activating the axis systems of the local line supports in the *Display* navigator

In contrast, the green vectors displayed in the graphic of the work window show the reaction forces on the part of the supports. The components of the support reactions are visualized by the size and direction of the vectors.

Support moments m_X / m_Y / m_Z

The support moments are listed in three table columns where they are sorted by lines. They are related to the global axis system XYZ or the local axis system of the line support X'Y'Z'. If you have set the local reference, the support moments are indicated with $m_{X'}$, $m_{Y'}$ and $m_{Z'}$.

The table shows the moments which are introduced into the support. Thus, with regard to signs, the table does not show the reactions on the part of the line support.

In the work window, however, the reaction moments are shown on the part of the support. In addition to the vector display, an arc display can be set: Point to *Display Properties* on the *Options* menu, and select *Edit* (see Figure 8.6, page 284).

Check sums

For load cases and load combinations RFEM displays check sums of loads and support reactions at the end of the table. They are always related to the global axis system. Differences will occur between the sums of *Σ Forces* and *Σ Loads* if the model has additional nodal supports and members or surfaces with elastic foundations. Therefore, also the *Σ Forces* available in these tables must be considered for the total summary.

Result diagrams

The result diagrams of line supports can be evaluated specifically in a new window: Right-click the (selected) line support(s), and then click the option *Result Diagrams* in the context menu (see figure in the left margin at Figure 8.13).

For more detailed information about the *Result Diagram* window, see chapter 9.5 on page 356.

In the work window, additional information is available for each line support:

- Σ Sum as resultant force
- Φ Mean value
- x Distance of the line's midpoint from line start
- e Eccentricity of resultant force related to the line's midpoint
- M Moment due to eccentricity of resultant force

To display this information, select *Results* in the *Display* navigator, double-click *Support Reactions*, and then tick the check box for *Info*.

Import support forces as load

The Z components of nodal and line support forces of another RFEM model can be applied as loads to the model your are currently working on. In this way, you can transfer for example loads by floors to analyze 2D floor slabs.

The support forces will be imported into the current load case. Therefore, it might be favorable if you first create a load case for the new loads.

To open the import dialog box,

select **Import Support Reactions as Load** on the **Tools** menu.

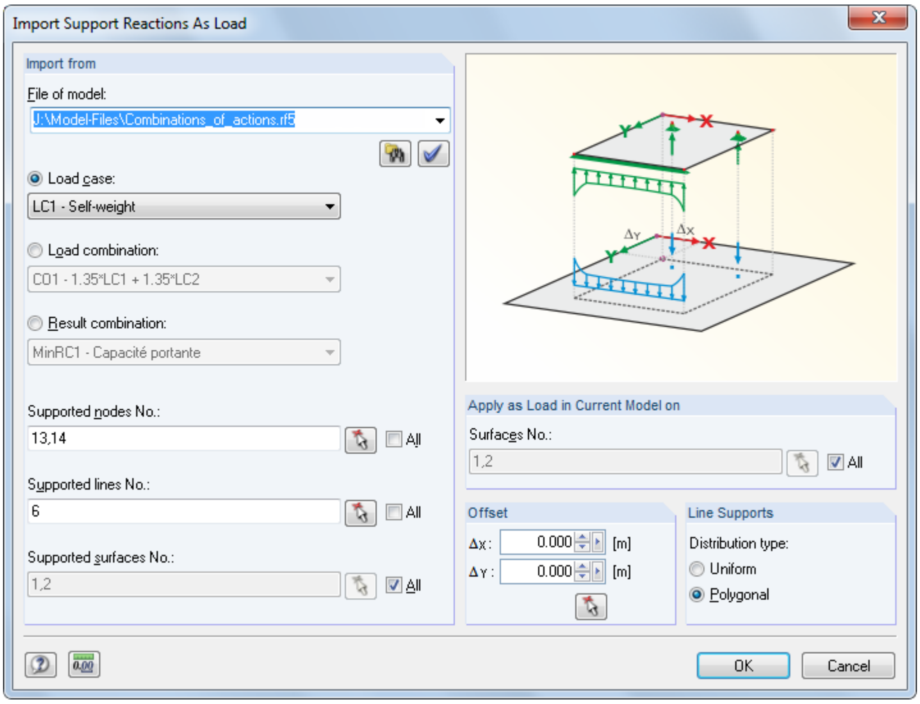


Figure 8.14: Dialog box *Import Support Reactions As Load*

First, specify the relevant model in the dialog section *Import from*. The button shown on the left helps you to select the right model. Then, RFEM imports the calculated load cases, load combinations and result combinations for which you have to make the next decision. When a result combination (*RC*) is set, you also have to define whether you want to import the maximum or minimum support forces.

If you don't want to import *All* support forces, you can specify the numbers of the relevant nodes, lines and surfaces. It is also possible to select them graphically in the original model by using the [↖] function.

In the dialog section *Apply as Load in Current Model on*, enter the numbers of the surfaces for which you want to create the loads. It is also possible to select them graphically.

If the original and target surfaces lie accurately one upon the other, no entry is required in the dialog section *Offset*. Otherwise, you can use the input fields to define the global offsets Δ*X* and Δ*Y* for the import. They refer to the global axes.

In the dialog section *Line Supports*, you can choose whether the support forces are created as free line loads with a uniform or a polygonal load distribution.

Filtering support forces of result combinations

For result combinations it is possible to adjust the default setting for the extreme values shown in the results table. To open the corresponding dialog box,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

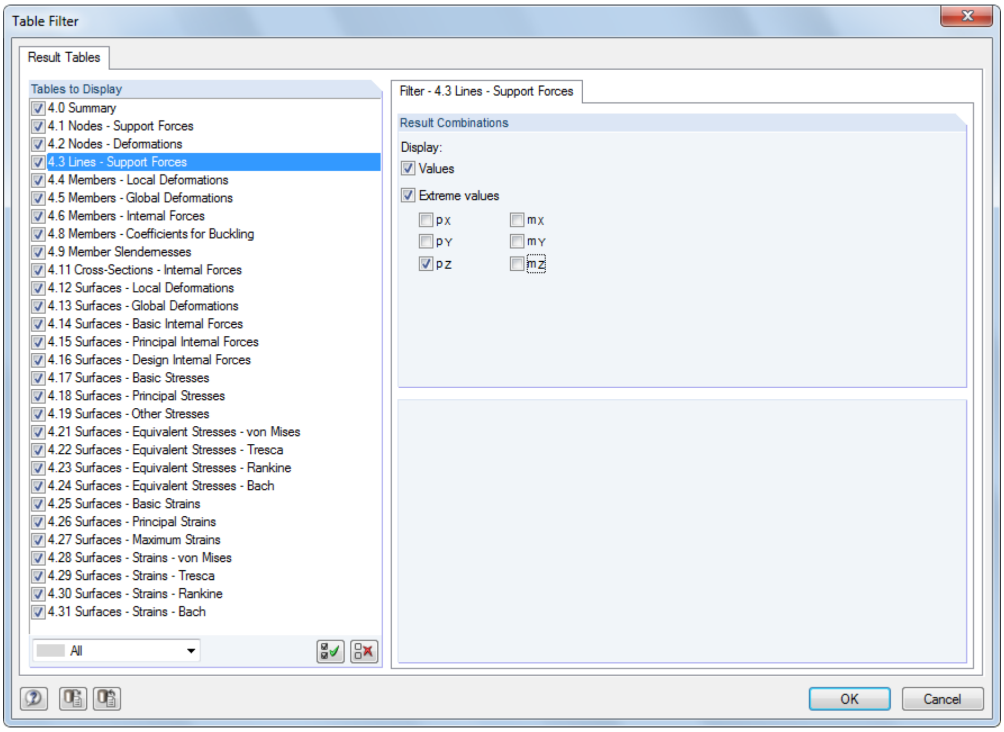


Figure 8.15: Dialog box *Table Filter*

The check boxes in the dialog box *Table Filter* control the type and amount of numerical output.

8.4 Members - Deformations

To control the graphical display of member displacements and member rotations, tick the check box for *Members* in the *Results* navigator. When asymmetric cross-sections are used, you can select if results refer to the principal axes *u* and *v* (see graphic on page 119) or to the standard input axes *y* and *z*. Table 4.4 shows the members' local deformations in numerical form.

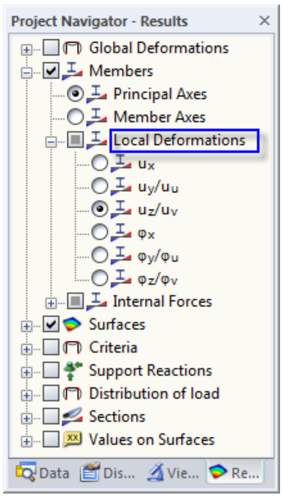
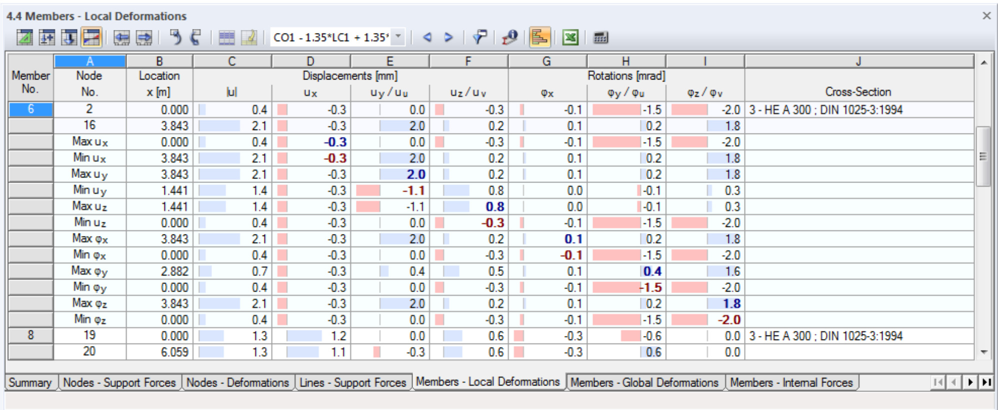
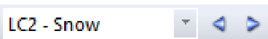


Figure 8.16: *Results* navigator: *Members* → *Local Deformations*



	A	B	C	D	E	F	G	H	I	J
Member	Node	Location	Displacements [mm]				Rotations [mrad]			
No.	No.	x [m]	|u|	u_x	u_y/u_u	u_z/u_v	φ_x	φ_y/φ_u	φ_z/φ_v	Cross-Section
6	2	0.000	0.4	-0.3	0.0	-0.3	-0.1	-1.5	-2.0	3 - HE A 300 ; DIN 1025-3:1994
	16	3.843	2.1	-0.3	2.0	0.2	0.1	0.2	1.8	
	Max u_x	0.000	0.4	-0.3	0.0	-0.3	-0.1	-1.5	-2.0	
	Min u_x	3.843	2.1	-0.3	2.0	0.2	0.1	0.2	1.8	
	Max u_y	3.843	2.1	-0.3	2.0	0.2	0.1	0.2	1.8	
	Min u_y	1.441	1.4	-0.3	-1.1	0.8	0.0	-0.1	0.3	
	Max u_z	1.441	1.4	-0.3	-1.1	0.8	0.0	-0.1	0.3	
	Min u_z	0.000	0.4	-0.3	0.0	-0.3	-0.1	-1.5	-2.0	
	Max φ_x	3.843	2.1	-0.3	2.0	0.2	0.1	0.2	1.8	
	Min φ_x	0.000	0.4	-0.3	0.0	-0.3	-0.1	-1.5	-2.0	
	Max φ_y	2.882	0.7	-0.3	0.4	0.5	0.1	0.4	1.6	
	Min φ_y	0.000	0.4	-0.3	0.0	-0.3	-0.1	-1.5	-2.0	
	Max φ_z	3.843	2.1	-0.3	2.0	0.2	0.1	0.2	1.8	
	Min φ_z	0.000	0.4	-0.3	0.0	-0.3	-0.1	-1.5	-2.0	
8	19	0.000	1.3	1.2	0.0	0.6	-0.3	-0.6	0.0	3 - HE A 300 ; DIN 1025-3:1994
	20	6.059	1.3	1.1	-0.3	0.6	-0.3	0.6	0.0	

Figure 8.17: Table 4.4 *Members - Local Deformations*



To display the deformations of a particular load case, select the load case from the list in the main toolbar or the toolbar of the table.

Node no.

The numbers of the start and end node are displayed for each member in the first two tables rows so that you can read the nodal values. In the subsequent rows, you see information about the deformation maximum or minimum shown in table columns D to I.

Location x

The table lists the deformations of each member on the following locations:

- Start and end node
- Division points according to defined member division (see chapter 4.16, page 136)
- Extreme values (*Max/Min*) of displacements and rotations

To adjust the default setting of the x-locations shown in the results table,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

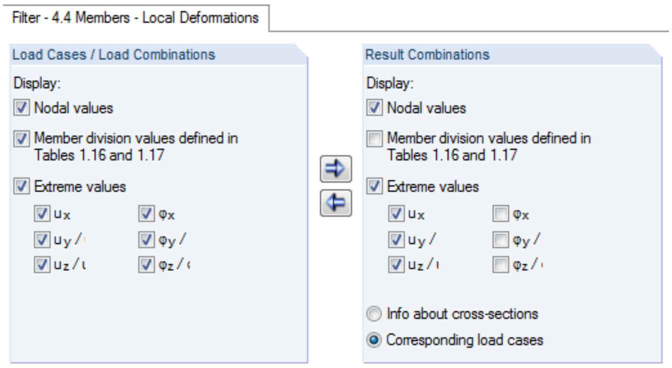


Figure 8.18: Dialog box *Table Filter* (dialog section)

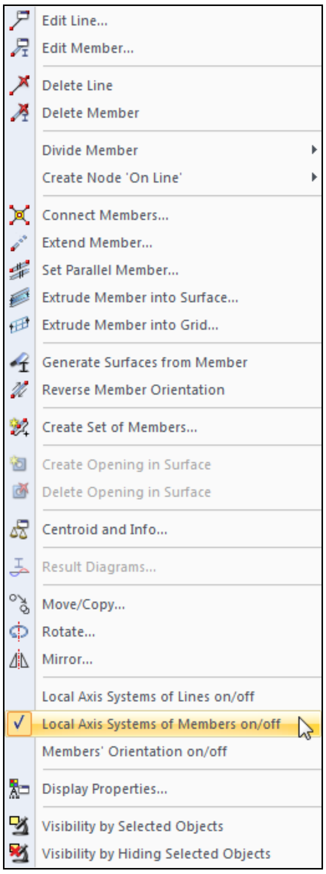
The check boxes in the dialog box *Table Filter* (see Figure 8.15, page 290) control the type and amount of numerical output.

Displacements / Rotations

The member deformations have the following meanings:

$\lvert u \rvert$	Absolute total displacement (not for result combinations)
u_x	Displacement of member in direction of its longitudinal axis
u_y / u_u	Displacement of member in direction of local axis y or u (see page 119)
u_z / u_v	Displacement of member in direction of local axis z or v
φ_x	Rotation of member about its longitudinal axis
φ_y / φ_u	Rotation of member about local axis y or u
φ_z / φ_v	Rotation of member about local axis z or v

Table 8.2: Member deformations



Member context menu

To check the position of the local member axes, select *Model* and *Members* in the *Display* navigator and activate *Member Axis Systems x,y,z* (see Figure 8.24, page 296). You can also use the member context menu shown on the left.

Furthermore, the local member axis system has an impact on the signs of deformations. A positive displacement follows the direction of the positive local axis, a positive rotation acts clockwise about the positive member axis.

Cross-section

The final table column informs you about the cross-sections used in members or about the corresponding load cases (for result combinations).

In the work window, deformations of members can be represented with a two- or multi-color display as well as in the rendering mode (see chapter 9.3, page 347).

Moreover, member deformations can be visualized as animation of the deformation process (see chapter 9.10, page 378).

8.5 Members - Global Deformations

To control the graphical display of member displacements and member rotations related to the global axes X, Y and Z, tick the check box for *Global Deformations* in the *Results* navigator. Table 4.5 shows the global deformations of members in numerical form.

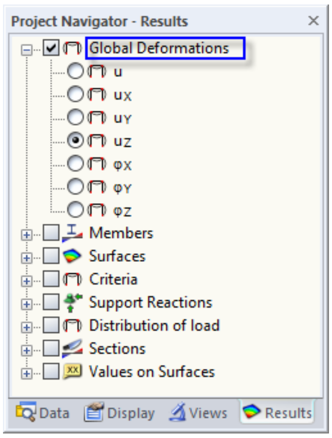
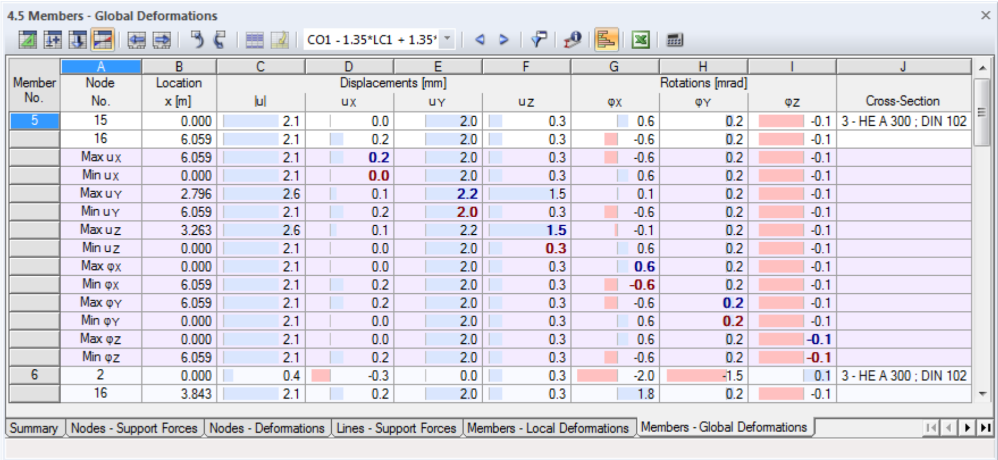


Figure 8.19: *Results* navigator: *Global Deformations*



4.5 Members - Global Deformations

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J
Member No.	Node No.	Location x [m]	Displacements [mm] |u|	uX	uY	uZ	Rotations [mrad] φX	φY	φZ	Cross-Section
5	15	0.000	2.1	0.0	2.0	0.3	0.6	0.2	-0.1	3 - HE A 300 ; DIN 102
	16	6.059	2.1	0.2	2.0	0.3	-0.6	0.2	-0.1	
	Max uX	6.059	2.1	0.2	2.0	0.3	-0.6	0.2	-0.1	
	Min uX	0.000	2.1	0.0	2.0	0.3	0.6	0.2	-0.1	
	Max uY	2.796	2.6	0.1	2.2	1.5	0.1	0.2	-0.1	
	Min uY	6.059	2.1	0.2	2.0	0.3	-0.6	0.2	-0.1	
	Max uZ	3.263	2.6	0.1	2.2	1.5	-0.1	0.2	-0.1	
	Min uZ	0.000	2.1	0.0	2.0	0.3	0.6	0.2	-0.1	
	Max φX	0.000	2.1	0.0	2.0	0.3	0.6	0.2	-0.1	
	Min φX	6.059	2.1	0.2	2.0	0.3	-0.6	0.2	-0.1	
	Max φY	6.059	2.1	0.2	2.0	0.3	-0.6	0.2	-0.1	
	Min φY	0.000	2.1	0.0	2.0	0.3	0.6	0.2	-0.1	
	Max φZ	0.000	2.1	0.0	2.0	0.3	0.6	0.2	-0.1	
	Min φZ	6.059	2.1	0.2	2.0	0.3	-0.6	0.2	-0.1	
6	2	0.000	0.4	-0.3	0.0	0.3	-2.0	-1.5	0.1	3 - HE A 300 ; DIN 102
	16	3.843	2.1	0.2	2.0	0.3	1.8	0.2	-0.1	

Summary | Nodes - Support Forces | Nodes - Deformations | Lines - Support Forces | Members - Local Deformations | Members - Global Deformations

Figure 8.20: Table 4.5 *Members - Global Deformations*

The table columns *Node No.* and *Location x* correspond to the columns of the previous results table 4.4 *Members - Local Deformations*.

Displacements / Rotations

The member deformations have the following meanings:

|u|	Absolute total displacement (not for result combinations)
u_X	Displacement of member in direction of global axis X
u_Y	Displacement of member in direction of global axis Y
u_Z	Displacement of member in direction of global axis Z
φ_X	Rotation of member about global axis X
φ_Y	Rotation of member about global axis Y
φ_Z	Rotation of member about global axis Z

Table 8.3: Global member deformations

8.6 Members - Internal Forces

To control the graphical display of member internal forces, tick the check box for *Members* in the *Results* navigator. Table 4.6 shows the internal forces and moments in numerical form.

If the structure is a 2D model, RFEM displays only the table columns of internal forces that are relevant for a planar structural system.

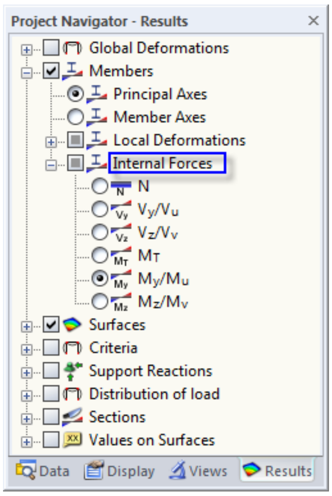
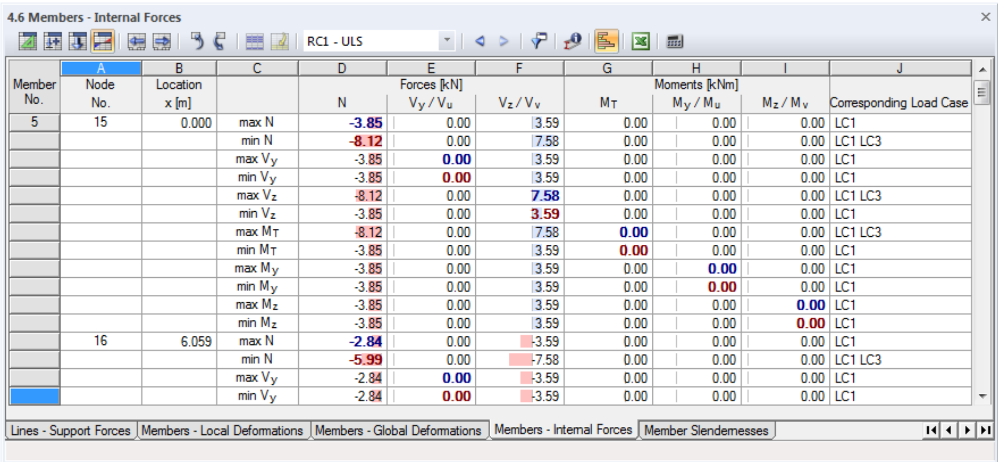


Figure 8.21: *Results* navigator: *Members* → *Internal Forces*



	A	B	C	D	E	F	G	H	I	J
Member No.	Node No.	Location x [m]		Forces [kN] N	V_y / V_u	V_z / V_v	Moments [kNm] M_T	M_y / M_u	M_z / M_v	Corresponding Load Case
5	15	0.000	max N	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			min N	-8.12	0.00	7.58	0.00	0.00	0.00	LC1 LC3
			max V_y	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			min V_y	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			max V_z	-8.12	0.00	7.58	0.00	0.00	0.00	LC1 LC3
			min V_z	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			max M_T	-8.12	0.00	7.58	0.00	0.00	0.00	LC1 LC3
			min M_T	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			max M_y	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			min M_y	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			max M_z	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
			min M_z	-3.85	0.00	3.59	0.00	0.00	0.00	LC1
	16	6.059	max N	-2.84	0.00	-3.59	0.00	0.00	0.00	LC1
			min N	-5.99	0.00	-7.58	0.00	0.00	0.00	LC1 LC3
			max V_y	-2.84	0.00	-3.59	0.00	0.00	0.00	LC1
			min V_y	-2.84	0.00	-3.59	0.00	0.00	0.00	LC1

Figure 8.22: Table 4.6 *Members – Internal Forces*

To display the internal forces of a particular load case, select the load case from the list in the main toolbar or the toolbar of the tables.

Location x

The table lists the internal forces of each member on the following locations:

- Start and end node
- Division points according to defined member division (see chapter 4.16, page 136)
- Extreme values (*Max/Min*) of internal forces

To adjust the default setting of the x-locations shown in the results table,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

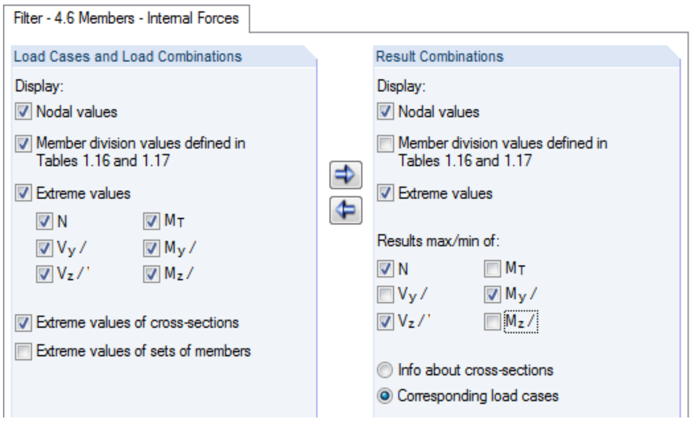


Figure 8.23: Dialog box *Table Filter* (dialog section)

The check boxes in the dialog box *Table Filter* control the type and amount of numerical output (see chapter 11.5.5, page 487).

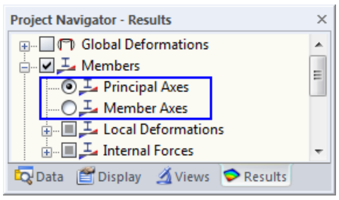
The diagram graphic for internal forces is based on the result values available in the FE mesh nodes or the member divisions that have been defined in the dialog tab *Global Calculation Parameters* of the dialog box *Calculation Parameters* (see chapter 7.3.3 page 272).

Forces / moments

The member internal forces have the following meanings:

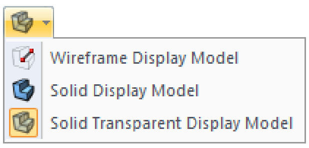
N	Axial force in member
V_y / V_u	Shear force in direction of local member axis y or u (see page 119)
V_z / V_v	Shear force in direction of local member axis z or v
M_T	Torsional moment
M_y / M_u	Bending moment about axis y or u
M_z / M_v	Bending moment about axis z or v

Table 8.4: Internal forces of members

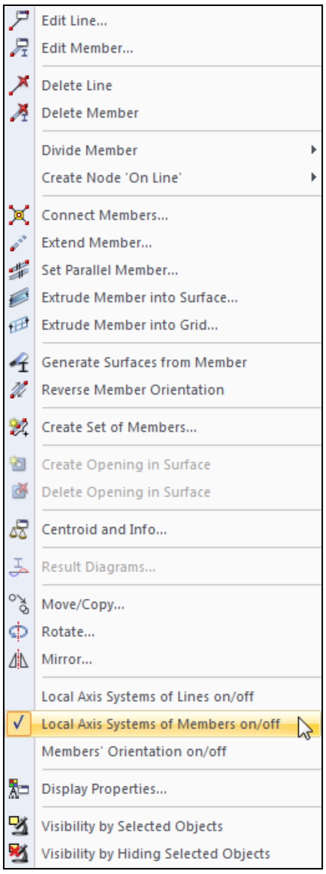


The local member axes y and z or u and v are the principal axes of the cross-section. Axis y or u represents the "strong" axis, the "weak" axis is represented by axis z or v (see chapter 4.17, page 145). When asymmetric cross-sections are used, you can select if internal forces refer to the principal axes *u* and *v* (see graphic on page 119) or to the standard input axes *y* and *z*. To set the results display, use the *Results* navigator as shown on the left. This display setting affects both the graphical results output and the output of results in the tables.

When a nonlinear analysis is performed, internal forces can also be related to the deformed member axis systems. The reference of the internal forces is set in the dialog section *Options* of the dialog box *Calculation Parameters* (see chapter 7.3.1, page 266).



To check the member position, use the 3D rendering. You can also use the *Display* navigator where you select *Model* and *Members*, and then tick the check box for *Member Axis Systems x,y,z* (see figure below).



Member context menu

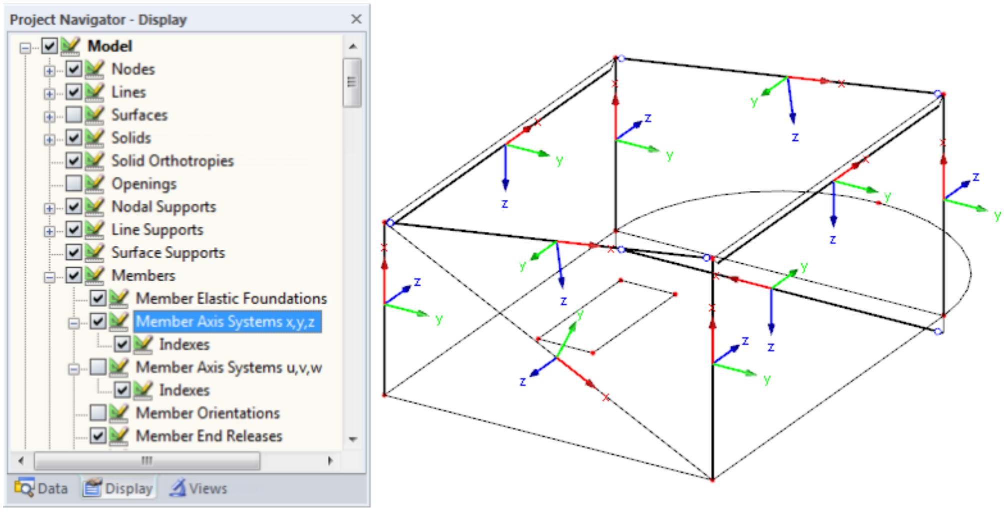


Figure 8.24: Selecting the local member axis systems in the *Display* navigator

The display of the member axes can also be activated in the member context menu which is shown on the left.

The local member axis system affects the signs of internal forces.

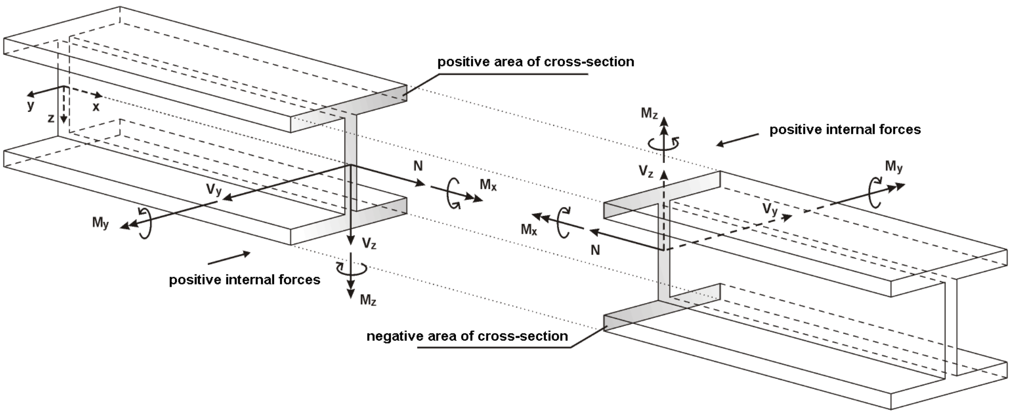
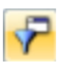


Figure 8.25: Positive definition of internal forces

The bending moment $\mathbf{M_y}$ is positive if tensile stresses occur on the positive member side (in direction of the z-axis). $\mathbf{M_z}$ is positive if compressive stresses occur on the positive member side (in direction of the y-axis). The sign definition for torsional moments, axial forces and shear forces conforms to the usual conventions. These internal forces are positive if they act in a positive direction.

Extreme values



If the table display of extreme values is activated (see Figure 8.23, page 295), RFEM shows you the maximum positive (*Max*) and the minimum negative (*Min*) internal forces for each member. In the results table, extreme values are highlighted in bold. The values in the remaining columns of the respective table row represent the internal forces related to the extreme value (see also chapter 11.5.5, page 487).

Cross-section / corresponding load cases

The final table column informs you about the cross-sections used in the members.

Result combinations

When you look at the results of result combinations, the column is entitled with *Corresponding Load Cases* (see Figure 8.22). The table shows the numbers of the load cases or combinations that have been used to determine the maximum or minimum internal forces of the respective table row. Load cases classified as *Permanent* appear always in this table column. *Variable* load cases are only displayed if their internal forces have an unfavorable effect on the result (see chapter 5.6, page 204).

At the same time, the table is extended by a new table column which is the third column C. At the end of the internal force list of a member you can read the maximum positive (***Max***) and the minimum negative (***Min***) values.

It is possible to reduce the amount of data in the result combination tables by using specific filter functions available in the dialog box *Table Filter* (see Figure 8.23, page 295). To open the dialog box,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

8.7 Members - Contact Forces

When members with elastic foundations exist in the model (see chapter 4.19, page 153), the contact forces and moments are shown numerically in table 4.6. To control the graphical display of results, tick the check box for *Members* in the *Results* navigator.

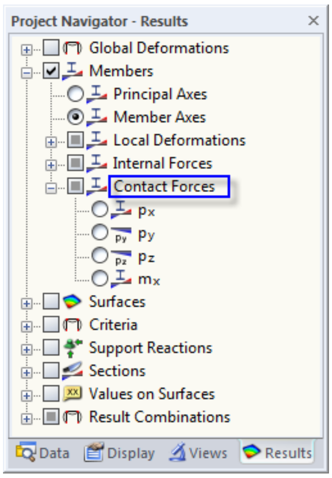
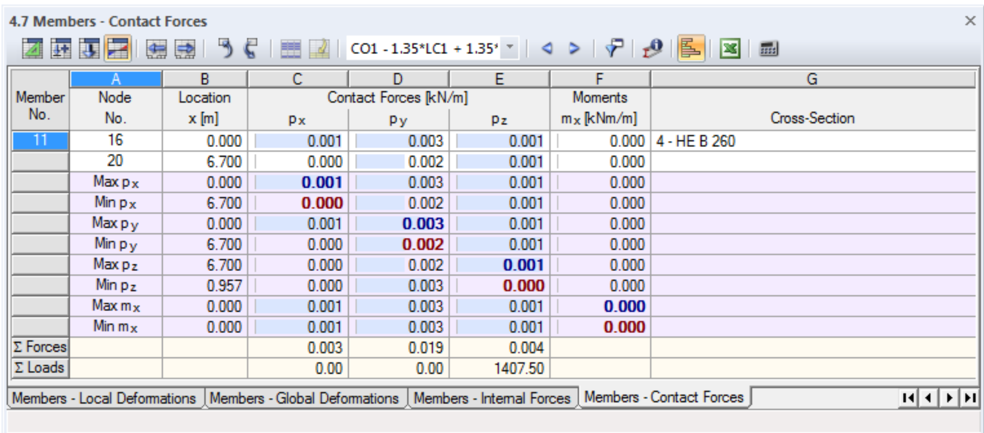


Figure 8.26: *Results* navigator: *Members* → *Contact Forces*


4.7 Members - Contact Forces

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G
Member No.	Node No.	Location x [m]	Contact Forces [kN/m] px	py	pz	Moments mx [kNm/m]	Cross-Section
11	16	0.000	0.001	0.003	0.001	0.000	4 - HE B 260
	20	6.700	0.000	0.002	0.001	0.000	
	Max px	0.000	0.001	0.003	0.001	0.000	
	Min px	6.700	0.000	0.002	0.001	0.000	
	Max py	0.000	0.001	0.003	0.001	0.000	
	Min py	6.700	0.000	0.002	0.001	0.000	
	Max pz	6.700	0.000	0.002	0.001	0.000	
	Min pz	0.957	0.000	0.003	0.000	0.000	
	Max mx	0.000	0.001	0.003	0.001	0.000	
	Min mx	0.000	0.001	0.003	0.001	0.000	
Σ Forces			0.003	0.019	0.004		
Σ Loads			0.00	0.00	1407.50		

Members - Local Deformations | Members - Global Deformations | Members - Internal Forces | Members - Contact Forces

Figure 8.27: Table 4.7 *Members - Contact Forces*

Node No.

In the first two table rows, the numbers of the start and end nodes are displayed for each foundation member. The remaining rows inform you about the types of extreme values available for contact forces and moments.

To adjust the default settings for the output of extreme values,

select **View** on the **Table** menu and click **Result Filter**

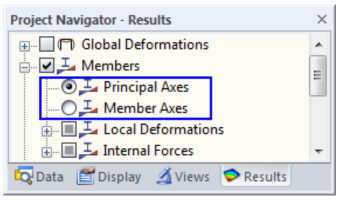
or use the button in the table toolbar shown on the left.

Location x

The table lists the contact internal forces of each member on the following locations:

- Start and end node
- Division points according to defined member division (see chapter 4.16, page 136)
- Extreme values (*Max/Min*) of contact forces and moments

Contact forces p_x / p_y / p_z



Contact forces that are effective in direction of the local member axes x, y and z are shown in relation to a standard length. When asymmetric cross-sections are used, you can select if contact forces refer to the principal axes *u* and *v* (see graphic on page 119) or to the standard input axes *y* and *z*. To set the results display, use the *Results* navigator.

To check the position of the local axes, select *Model* and *Members* in the *Display* navigator and activate *Member Axis Systems x,y,z* (see Figure 8.24). The signs comply with the usual definitions explained in chapter 8.6 on page 296 describing the internal forces of members.

When you want to determine soil contact pressures on the basis of the table values, you additionally have to divide the results by the respective cross-section widths.

Moments m_x

The contact moments about the longitudinal member axis x also refer to a standard length. The contact moments m_x are influenced by the rotational spring constant $C\varphi$.

Cross-section / corresponding load cases

The final table column informs you about the cross-sections used in members. When a result combination is set, you see the load cases and combinations that have been used to determine the maximum or minimum contact forces in the respective table row.

Check sums

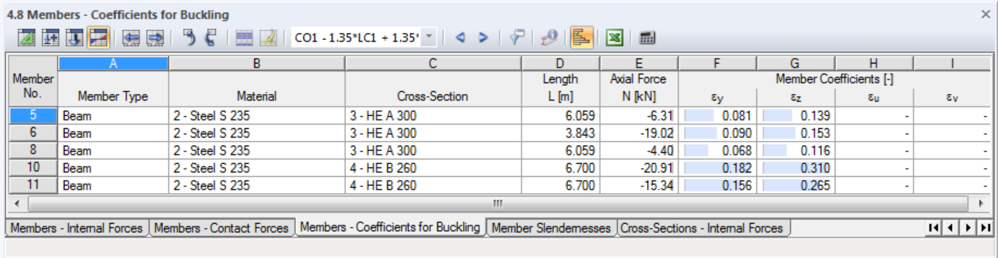
For load cases and load combinations RFEM displays check sums of loads and support reactions at the end of the table. Differences will occur between the sums of *Σ Forces* and *Σ Loads* if the model has additional nodal and line supports as well as surfaces with elastic foundations. Therefore, also the *Σ Forces* available in tables 4.1, 4.3 and 4.20 must be considered for the total summary.

8.8 Members - Member Coefficients for Buckling

When you calculate member models subjected to pressure according to second-order analysis, the member coefficient ε is important (see chapter 7.3.1, page 264). Each member has its own member coefficient that is determined from compressive force, member length and member stiffness.

Members with member coefficients higher than 1 have to be analyzed, where applicable, according to second-order analysis. Also standards of some countries such as the USA have rules where member coefficients must be limited.

Table 4.8 shows the member coefficients which are governing for buckling. There is no graphical output option.



4.8 Members - Coefficients for Buckling

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I
Member No.	Member Type	Material	Cross-Section	Length L [m]	Axial Force N [kN]	Member Coefficients [-] ε_y	ε_z	ε_u	ε_v
5	Beam	2 - Steel S 235	3 - HE A 300	6.059	-6.31	0.081	0.139	-	-
6	Beam	2 - Steel S 235	3 - HE A 300	3.843	-19.02	0.090	0.153	-	-
8	Beam	2 - Steel S 235	3 - HE A 300	6.059	-4.40	0.068	0.116	-	-
10	Beam	2 - Steel S 235	4 - HE B 260	6.700	-20.91	0.182	0.310	-	-
11	Beam	2 - Steel S 235	4 - HE B 260	6.700	-15.34	0.156	0.265	-	-

Members - Internal Forces | Members - Contact Forces | Members - Coefficients for Buckling | Member Slendernesses | Cross-Sections - Internal Forces

Figure 8.28: Table 4.8 *Members - Coefficients for Buckling*

The listed member coefficients are sorted by member numbers.

Member type

The member types are indicated for information (see chapter 4.17, page 139). RFEM determines member coefficients only for members that are able to absorb compressive forces.

Material

Characteristics of the material affect the member stiffness.

Cross-section

The cross-section's second moments of area are required to determine the member stiffnesses.

Length L

Table column D shows you the member lengths.

Axial force N

The column lists the axial forces used for determining the member coefficient. Here, the forces are the axial forces which are available in the member center (x = L/2).

Member coefficients are determined only for members that have compression forces in at least one portion of the member (truss girder) or along the entire member (compression member, buckling member etc.).

Member coefficients ε_y / ε_z

The member coefficient depends on the member length L, the compressive force N and the stiffness E · I.

$$\varepsilon = L \cdot \sqrt{\frac{|N|}{E \cdot I}}$$

Equation 8.1: Member coefficient ε

Table columns F and G show the member coefficients referring to the local member axis system y and z. When asymmetric cross-sections like angles are used, two more columns appear where slendernesses are shown also in relation to the principal axes u and v.

8.9 Member Slendernesses

Table 4.9 shows you the slenderness ratios of members. They are significant for the evaluation of the buckling behavior of members subjected to pressure. There is no graphical output option.

4.9 Member Slendernesses

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H
Member No.	Cross-Section	Length L [m]	Effective Length Factors [-] $k_{cr,y}$	$k_{cr,z}$	Slenderness [-] λ_y	λ_z	λ_u	λ_v
1	1 - Circle 300	4.000	1.000	1.000	53.33	53.33	-	-
2	1 - Circle 300	4.000	1.000	1.000	53.33	53.33	-	-
3	2 - Rectangle 250/	6.000	1.000	1.000	51.96	83.14	-	-
4	3 - HE A 300	3.000	1.000	1.000	23.60	40.15	-	-
5	3 - HE A 300	6.059	1.000	1.000	47.66	81.08	-	-
6	3 - HE A 300	3.843	1.000	1.000	30.23	51.43	-	-
7	3 - HE A 300	3.000	1.000	1.000	23.60	40.15	-	-
8	3 - HE A 300	6.059	1.000	1.000	47.66	81.08	-	-
9	3 - HE A 300	3.843	1.000	1.000	30.23	51.43	-	-
10	4 - HE B 260	6.700	1.000	1.000	59.58	101.61	-	-
11	4 - HE B 260	6.700	1.000	1.000	59.58	101.61	-	-
12	5 - L 80x8	6.708	1.000	1.000	219.39	432.43	219.39	432.43

Members - Internal Forces | Members - Contact Forces | Members - Coefficients for Buckling | Member Slendernesses

Figure 8.29: Table 4.9 *Member Slendernesses*

The listed member slendernesses are sorted by member numbers.

Cross-section

The cross-section's radii of gyration are required to determine the slendernesses.

Length L

The member lengths are indicated in table column B.

Effective length factors $k_{cr,y}$ / $k_{cr,z}$

The buckling length coefficients describe the ratio of buckling length and member length.

$$k_{cr} = \frac{L_{cr}}{L}$$

Equation 8.2: Buckling length coefficient k_{cr}

The buckling length L_{cr} refers to the buckling behavior perpendicular to the 'strong' member axis y, respectively the 'weak' member axis z. If no buckling lengths have been defined manually (see chapter 4.17, page 149), the EULER buckling mode 2 is assumed: In this case, the buckling length is equal to the member length. More accurate analyses can be performed with the add-on module RF-STABILITY or in Dlubal's design modules such as RF-STEEL EC3.

Slendernesses λ_y / λ_z

The slenderness ratio represents a pure geometric value. It is determined from the effective length factor k_{cr}, the member length L and the radius of gyration i.

$$\lambda = \frac{k_{cr} \cdot L}{i}$$

Equation 8.3: Slenderness λ

Table columns E and F show the slendernesses referring to the local member axis system y and z. When asymmetric cross-sections like angles are used, two more columns appear where slendernesses are shown also in relation to the principal axes u and v.

8.10 Sets of Members - Internal Forces

Table 4.10 shows the internal forces sorted by sets of members (see chapter 4.21, page 158).

4.10 Set of Members - Internal Forces

RC1 - ULS

	A	B	C	D	E	F	G	H	I	J
Member No.	Node No.	Location x [m]		Forces [kN] N	V_y / V_u	V_z / V_v	Moments [kNm] M_T	M_y / M_u	M_z / M_v	Corresponding Load Cases
	Set of Members No. 1: Purlin A-A									
51	56	3.000	min V_y	-7.07	**-2.23**	1.00	0.02	-4.47	3.33	LC3
			max V_z	-7.07	-2.23	**1.00**	0.02	-4.47	3.33	LC3
			min V_z	-86.58	-0.12	**0.06**	0.00	0.12	0.17	LC1
			max M_T	-7.07	-2.23	1.00	**0.02**	-4.47	3.33	LC3
			min M_T	-6.88	-0.06	0.07	**0.00**	0.13	0.00	LC2
			max M_y	-6.88	-0.06	0.07	0.00	**0.13**	0.00	LC2
			min M_y	-7.07	-2.23	1.00	0.02	**-4.47**	3.33	LC3
			max M_z	-7.07	-2.23	1.00	0.02	-4.47	**3.33**	LC3
			min M_z	-6.88	-0.06	0.07	0.00	0.13	**0.00**	LC2
52	56	0.000	max N	**-6.87**	-0.03	-0.05	-0.02	-0.01	-0.03	LC2
			min N	**-37.96**	-0.13	-0.04	-0.01	0.04	-0.11	LC1
			max V_y	-7.00	**0.17**	1.25	0.04	-4.40	3.19	LC3
			min V_y	-37.96	**-0.13**	-0.04	-0.01	0.04	-0.11	LC1
			max V_z	-7.00	0.17	**1.25**	0.04	-4.40	3.19	LC3
			min V_z	-6.87	-0.03	**-0.05**	-0.02	-0.01	-0.03	LC2
			max M_T	-7.00	0.17	1.25	**0.04**	-4.40	3.19	LC3
			min M_T	-6.87	-0.03	-0.05	**-0.02**	-0.01	-0.03	LC2
			max M_y	-37.96	-0.13	-0.04	-0.01	**0.04**	-0.11	LC1
			min M_y	-7.00	0.17	1.25	0.04	**-4.40**	3.19	LC3

Members - Internal Forces | Members - Coefficients for Buckling | Member Slendernesses | Set of Members - Internal Forces | Cross-Sections - Internal Forces

Figure 8.30: Table 4.10 *Set of Members - Internal Forces*

The table's structure is similar to the one of table 4.6 *Members – Internal Forces* described in chapter 8.6. Now, the results are sorted by continuous members or member groups. The descriptions of member sets remain fixed in the top row of the table so it is easier to overview results data when scrolling.

The table includes the member-by-member results of all members contained in the set of members. The results list of a set of members ends with the color-highlighted table rows: They show the total extremes ***MAX*** and ***MIN*** of each internal force type in the member set. The extreme values are highlighted in bold. The values in the remaining table columns of the respective table row represent the internal forces related to the extreme value.

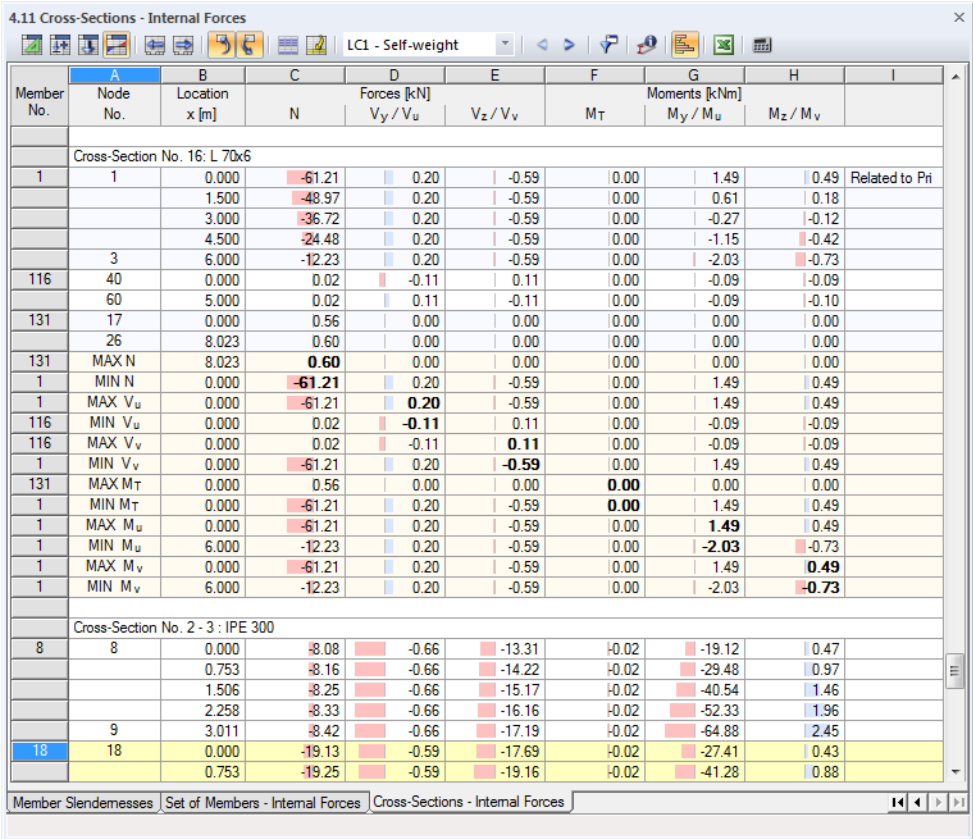
It is possible to reduce the amount of data in the table by using specific filter functions available in the dialog box *Table Filter* (see chapter 11.5.5, page 487). To open the dialog box,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

8.11 Cross-sections - Internal Forces

Table 4.11 shows the internal forces sorted by cross-sections.



4.11 Cross-Sections - Internal Forces

LC1 - Self-weight

Member No.	A Node No.	B Location x [m]	C Forces [kN] N	D V_y / V_u	E V_z / V_v	F Moments [kNm] M_T	G M_y / M_u	H M_z / M_v	I
	Cross-Section No. 16: L 70x6								
1	1	0.000	-61.21	0.20	-0.59	0.00	1.49	0.49	Related to Pri
		1.500	-48.97	0.20	-0.59	0.00	0.61	0.18	
		3.000	-36.72	0.20	-0.59	0.00	-0.27	-0.12	
		4.500	-24.48	0.20	-0.59	0.00	-1.15	-0.42	
	3	6.000	-12.23	0.20	-0.59	0.00	-2.03	-0.73	
116	40	0.000	0.02	-0.11	0.11	0.00	-0.09	-0.09	
	60	5.000	0.02	0.11	-0.11	0.00	-0.09	-0.10	
131	17	0.000	0.56	0.00	0.00	0.00	0.00	0.00	
	26	8.023	0.60	0.00	0.00	0.00	0.00	0.00	
131	MAX N	8.023	**0.60**	0.00	0.00	0.00	0.00	0.00	
1	MIN N	0.000	**-61.21**	0.20	-0.59	0.00	1.49	0.49	
1	MAX V_u	0.000	-61.21	**0.20**	-0.59	0.00	1.49	0.49	
116	MIN V_u	0.000	0.02	**-0.11**	0.11	0.00	-0.09	-0.09	
116	MAX V_v	0.000	0.02	-0.11	**0.11**	0.00	-0.09	-0.09	
1	MIN V_v	0.000	-61.21	0.20	**-0.59**	0.00	1.49	0.49	
131	MAX M_T	0.000	0.56	0.00	0.00	**0.00**	0.00	0.00	
1	MIN M_T	0.000	-61.21	0.20	-0.59	**0.00**	1.49	0.49	
1	MAX M_u	0.000	-61.21	0.20	-0.59	0.00	**1.49**	0.49	
1	MIN M_u	6.000	-12.23	0.20	-0.59	0.00	**-2.03**	-0.73	
1	MAX M_v	0.000	-61.21	0.20	-0.59	0.00	1.49	**0.49**	
1	MIN M_v	6.000	-12.23	0.20	-0.59	0.00	-2.03	**-0.73**	
	Cross-Section No. 2 - 3 : IPE 300								
8	8	0.000	-8.08	-0.66	-13.31	-0.02	-19.12	0.47	
		0.753	-8.16	-0.66	-14.22	-0.02	-29.48	0.97	
		1.506	-8.25	-0.66	-15.17	-0.02	-40.54	1.46	
		2.258	-8.33	-0.66	-16.16	-0.02	-52.33	1.96	
	9	3.011	-8.42	-0.66	-17.19	-0.02	-64.88	2.45	
18	18	0.000	-19.13	-0.59	-17.69	-0.02	-27.41	0.43	
		0.753	-19.25	-0.59	-19.16	-0.02	-41.28	0.88	

Member Slendernesses | Set of Members - Internal Forces | Cross-Sections - Internal Forces

Figure 8.31: Table 4.11 *Cross-Sections - Internal Forces*

The table's structure is similar to the one of table 4.6 *Members - Internal Forces* described in chapter 8.6. Now, the results are sorted by cross-sections. The descriptions of cross-sections remain fixed in the top row of the table so that it is easier to overview results data when scrolling.

The table includes the member-by-member results of all members that use the relevant cross-section. The results list for a cross-section ends with the color highlighted table rows: They show the total extremes ***MAX*** and ***MIN*** of each internal force type in the cross-section. The extreme values are highlighted in bold. The values in the remaining table columns of the respective table row represent the internal forces related to the extreme value.

It is possible to reduce the amount of data in the table by using specific filter functions available in the dialog box *Table Filter* (see chapter 11.5.5, page 487).

8.12 Surfaces - Local Deformations

To control the graphical display of local surface deformations, tick the check box for *Surfaces* in the *Results* navigator. Table 4.12 shows the surfaces' local deformations in numerical form.

For 2D structures RFEM shows only the relevant table columns of deformations.

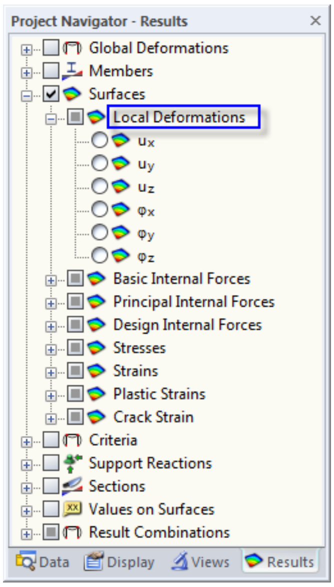
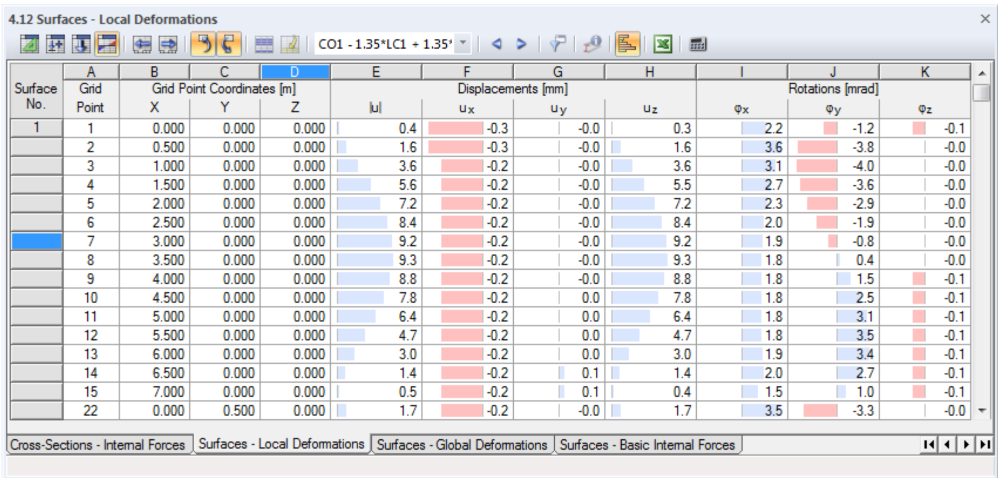


Figure 8.32: *Results* navigator: *Surfaces* → *Local Deformations*



	A	B	C	D	E	F	G	H	I	J	K
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Displacements [mm] |u|	u_x	u_y	u_z	Rotations [mrad] φ_x	φ_y	φ_z
1	1	0.000	0.000	0.000	0.4	-0.3	-0.0	0.3	2.2	-1.2	-0.1
	2	0.500	0.000	0.000	1.6	-0.3	-0.0	1.6	3.6	-3.8	-0.0
	3	1.000	0.000	0.000	3.6	-0.2	-0.0	3.6	3.1	-4.0	-0.0
	4	1.500	0.000	0.000	5.6	-0.2	-0.0	5.5	2.7	-3.6	-0.0
	5	2.000	0.000	0.000	7.2	-0.2	-0.0	7.2	2.3	-2.9	-0.0
	6	2.500	0.000	0.000	8.4	-0.2	-0.0	8.4	2.0	-1.9	-0.0
	7	3.000	0.000	0.000	9.2	-0.2	-0.0	9.2	1.9	-0.8	-0.0
	8	3.500	0.000	0.000	9.3	-0.2	-0.0	9.3	1.8	0.4	-0.0
	9	4.000	0.000	0.000	8.8	-0.2	-0.0	8.8	1.8	1.5	-0.1
	10	4.500	0.000	0.000	7.8	-0.2	0.0	7.8	1.8	2.5	-0.1
	11	5.000	0.000	0.000	6.4	-0.2	0.0	6.4	1.8	3.1	-0.1
	12	5.500	0.000	0.000	4.7	-0.2	0.0	4.7	1.8	3.5	-0.1
	13	6.000	0.000	0.000	3.0	-0.2	0.0	3.0	1.9	3.4	-0.1
	14	6.500	0.000	0.000	1.4	-0.2	0.1	1.4	2.0	2.7	-0.1
	15	7.000	0.000	0.000	0.5	-0.2	0.1	0.4	1.5	1.0	-0.1
	22	0.000	0.500	0.000	1.7	-0.2	-0.0	1.7	3.5	-3.3	-0.0

Figure 8.33: Table 4.12 *Surfaces - Local Deformations*

The table shows the displacements and rotations sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. They represent the characteristics of any surface. Number and arrangement of grid points can be adjusted in the *Grid* tab of the dialog box *Edit Surface*.

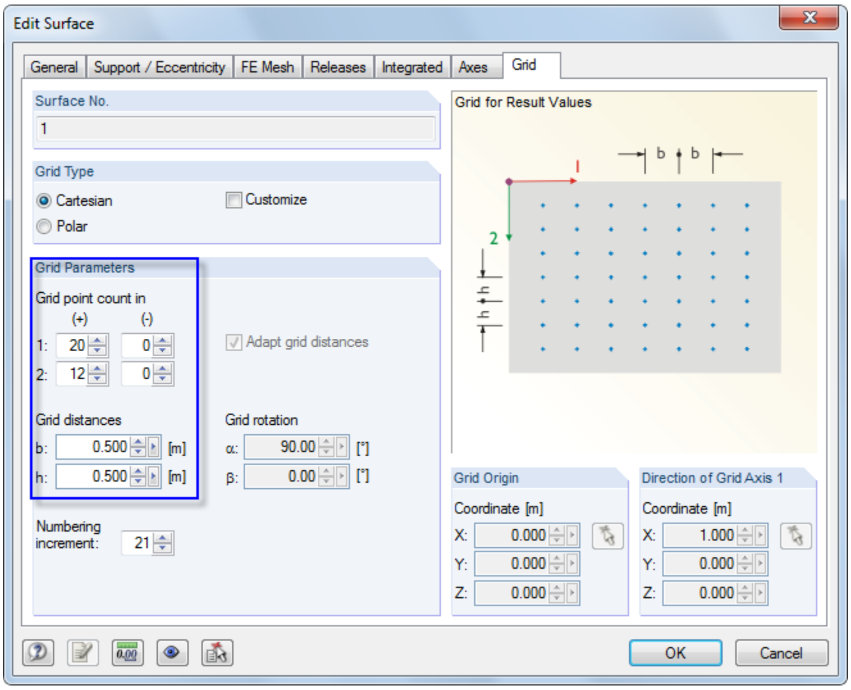


Figure 8.34: Dialog box *Edit Surface*, tab *Grid*

In the *Grid* dialog tab, you can define *Grid Type*, *Grid Parameters* and, if necessary, the *Grid Origin* as well as the *Direction of Grid Axis 1*. The default setting is a Cartesian surface grid with uniform *Grid distances* of 0.5 cm for the grid points in both directions.

The grid allows for a results output in equidistant, adjustable result points that does not depend on the FE mesh. For small surfaces the grid's standard mesh size of 0.5 m may produce only few grid points (or even only one result grid point in the grid origin). Then, *count* and *distances* of the grid points should be adjusted to the surface size in order to generate more grid points.

When the surface grid is modified, a new calculation of results is not necessary because the grid values are interpolated from the result values of the FE nodes.

The results output in the table is based on the surface results grid. In the work window, both the values of FE nodes and grid points can be displayed. To set the display, use the *Results* navigator:

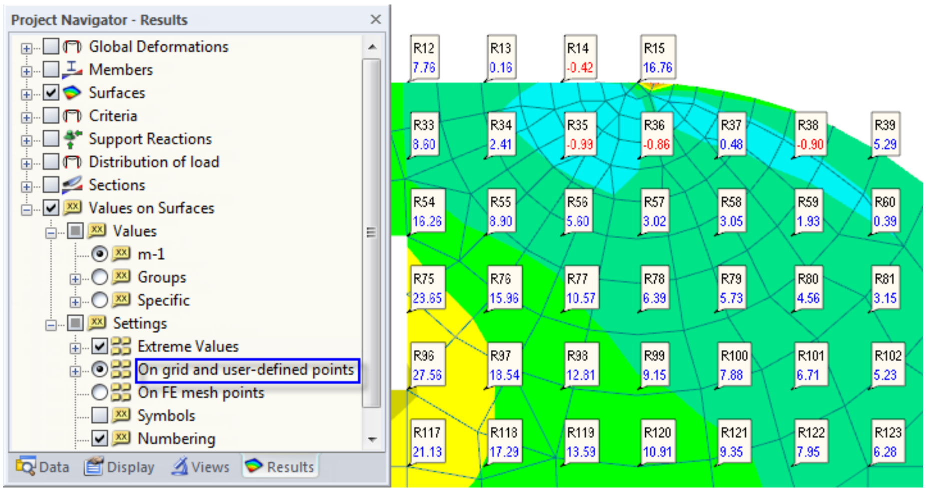


Figure 8.35: *Results* navigator: *Values on Surfaces* → *Settings* → *On **grid** points* or *On **FE mesh** points*

RFEM numbers the grid points automatically. To display the numbers of grid points in the results graphics, tick the check box for *Numbering* in the *Results* navigator as shown in the figure above.

Grid point coordinates

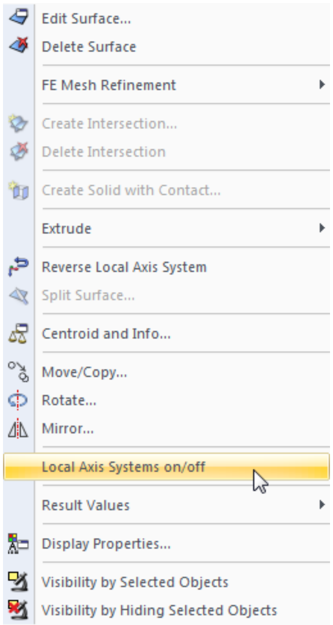
Table columns B to D show the coordinates of grid points in the global coordinate system XYZ. When you click into a table row, the corresponding grid point is indicated in the work window by an arrow.

Displacements / Rotations

The deformations have the following meanings:

$\lvert u \rvert$	Absolute total displacement (not for result combinations)
u_X	Displacement of surface in direction of local axis x
u_Y	Displacement of surface in direction of local axis y
u_Z	Displacement of surface in direction of local axis z
φ_X	Rotation of surface about local axis x
φ_Y	Rotation of surface about local axis y
φ_Z	Rotation of surface about local axis z

Table 8.5: Local surface deformations



Context menu of surface

To display the local surface axes, use the context menu or the *Display* navigator where you select **Model** → **Surfaces** → **Surface Axis Systems x,y,z**.

When you analyze curved surfaces, the surface axes refer to the axes of the finite elements (see Figure 8.40, page 308).

8.13 Surfaces - Global Deformations

To control the graphical display of surface displacements and rotations related to the global axes X, Y and Z, tick the check box for *Global Deformations* in the *Results* navigator. Table 4.13 shows the global deformations of surfaces in numerical form.

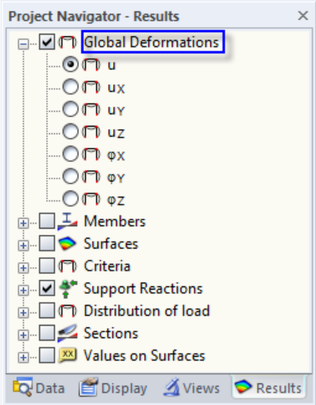
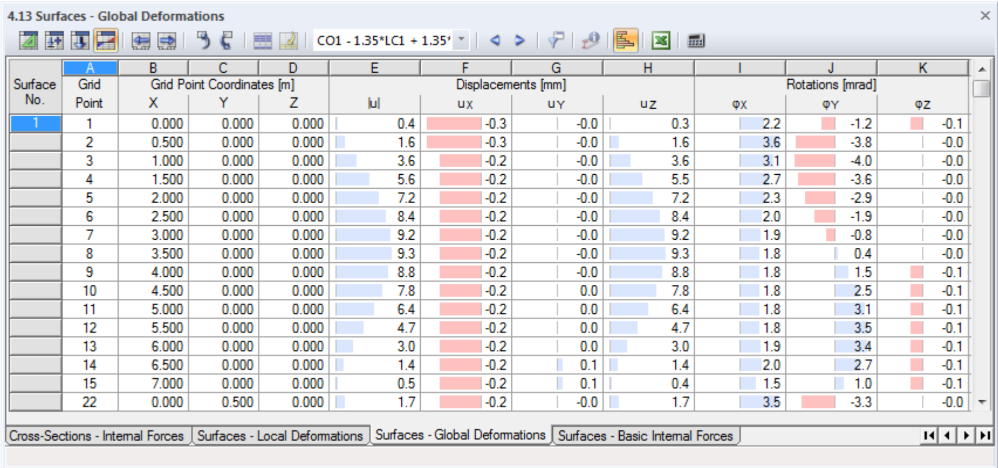


Figure 8.36: *Results* navigator: *Global Deformations*



4.13 Surfaces - Global Deformations

Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Displacements [mm] |u|	uX	uY	uZ	Rotations [mrad] φX	φY	φZ
1	1	0.000	0.000	0.000	0.4	-0.3	-0.0	0.3	2.2	-1.2	-0.1
	2	0.500	0.000	0.000	1.6	-0.3	-0.0	1.6	3.6	-3.8	-0.0
	3	1.000	0.000	0.000	3.6	-0.2	-0.0	3.6	3.1	-4.0	-0.0
	4	1.500	0.000	0.000	5.6	-0.2	-0.0	5.5	2.7	-3.6	-0.0
	5	2.000	0.000	0.000	7.2	-0.2	-0.0	7.2	2.3	-2.9	-0.0
	6	2.500	0.000	0.000	8.4	-0.2	-0.0	8.4	2.0	-1.9	-0.0
	7	3.000	0.000	0.000	9.2	-0.2	-0.0	9.2	1.9	-0.8	-0.0
	8	3.500	0.000	0.000	9.3	-0.2	-0.0	9.3	1.8	0.4	-0.0
	9	4.000	0.000	0.000	8.8	-0.2	-0.0	8.8	1.8	1.5	-0.1
	10	4.500	0.000	0.000	7.8	-0.2	0.0	7.8	1.8	2.5	-0.1
	11	5.000	0.000	0.000	6.4	-0.2	0.0	6.4	1.8	3.1	-0.1
	12	5.500	0.000	0.000	4.7	-0.2	0.0	4.7	1.8	3.5	-0.1
	13	6.000	0.000	0.000	3.0	-0.2	0.0	3.0	1.9	3.4	-0.1
	14	6.500	0.000	0.000	1.4	-0.2	0.1	1.4	2.0	2.7	-0.1
	15	7.000	0.000	0.000	0.5	-0.2	0.1	0.4	1.5	1.0	-0.1
	22	0.000	0.500	0.000	1.7	-0.2	-0.0	1.7	3.5	-3.3	-0.0

Cross-Sections - Internal Forces | Surfaces - Local Deformations | Surfaces - Global Deformations | Surfaces - Basic Internal Forces

Figure 8.37: Table 4.13 *Surfaces - Global Deformations*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.12 *Surfaces - Local Deformations*.

Displacements / Rotations

The surface deformations have the following meanings:

|u|	Absolute total displacement (not for result combinations)
u_X	Displacement of surface in direction of global axis X
u_Y	Displacement of surface in direction of global axis Y
u_Z	Displacement of surface in direction of global axis Z
φ_X	Rotation of surface about global axis X
φ_Y	Rotation of surface about global axis Y
φ_Z	Rotation of surface about global axis Z

Table 8.6: Global surface deformations

8.14 Surfaces - Basic Internal Forces

To control the graphical display of the basic internal forces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Basic Internal Forces*. Table 4.14 shows the basic internal forces of surfaces in numerical form.

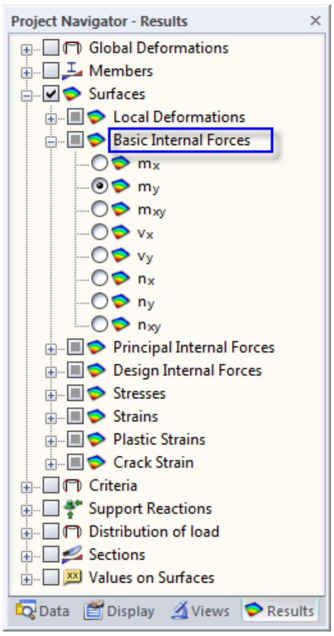
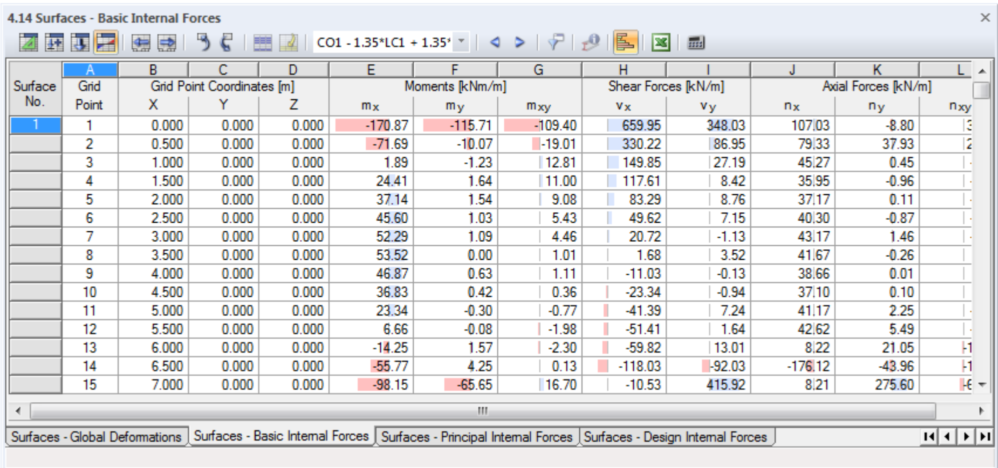


Figure 8.38: *Results* navigator: *Surfaces* → *Basic Internal Forces*



4.14 Surfaces - Basic Internal Forces

CO1 - 1.35*LC1 + 1.35*

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Moments [kNm/m] m_x	F m_y	G m_{xy}	H Shear Forces [kN/m] v_x	I v_y	J Axial Forces [kN/m] n_x	K n_y
1	1	0.000	0.000	0.000	-170.87	-115.71	-109.40	659.95	348.03	107.03	-8.80
	2	0.500	0.000	0.000	-71.69	-10.07	-19.01	330.22	86.95	79.33	37.93
	3	1.000	0.000	0.000	1.89	-1.23	12.81	149.85	27.19	45.27	0.45
	4	1.500	0.000	0.000	24.41	1.64	11.00	117.61	8.42	35.95	-0.96
	5	2.000	0.000	0.000	37.14	1.54	9.08	83.29	8.76	37.17	0.11
	6	2.500	0.000	0.000	45.60	1.03	5.43	49.62	7.15	40.30	-0.87
	7	3.000	0.000	0.000	52.29	1.09	4.46	20.72	-1.13	43.17	1.46
	8	3.500	0.000	0.000	53.52	0.00	1.01	1.68	3.52	41.67	-0.26
	9	4.000	0.000	0.000	46.87	0.63	1.11	-11.03	-0.13	38.66	0.01
	10	4.500	0.000	0.000	36.83	0.42	0.36	-23.34	-0.94	37.10	0.10
	11	5.000	0.000	0.000	23.34	-0.30	-0.77	-41.39	7.24	41.17	2.25
	12	5.500	0.000	0.000	6.66	-0.08	-1.98	-51.41	1.64	42.62	5.49
	13	6.000	0.000	0.000	-14.25	1.57	-2.30	-59.82	13.01	8.22	21.05
	14	6.500	0.000	0.000	-55.77	4.25	0.13	-118.03	-92.03	-176.12	-43.96
	15	7.000	0.000	0.000	-98.15	-65.65	16.70	-10.53	415.92	8.21	275.60

Surfaces - Global Deformations | Surfaces - Basic Internal Forces | Surfaces - Principal Internal Forces | Surfaces - Design Internal Forces

Figure 8.39: Table 4.14 *Surfaces - Basic Internal Forces*

The table shows the basic internal forces sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ. When you click into a table row, the corresponding grid point is indicated in the work window by an arrow.

Moments / shear forces / axial forces

In contrast to member internal forces, internal forces of a surface are symbolized by small letters. From the integral definition of the bending moments m_x and m_y arises the fact that moments are related to the directions of the surface axes where the corresponding normal stresses are created. To display the surface axes, use the surface context menu (see Figure 4.115, page 116).

When curved surfaces are analyzed, internal forces refer to the local axes of the individual finite elements. The axes can be displayed by ticking the corresponding check box in the *Display* navigator:

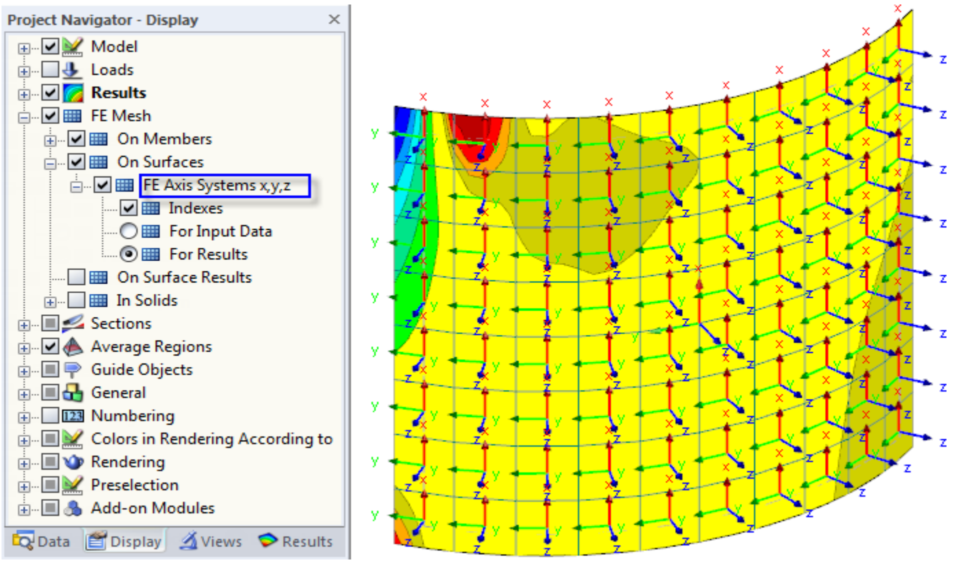


Figure 8.40: *Display* navigator: *FE Axis Systems x,y,z*

There is a basic difference in understanding internal forces of surfaces and members: A member moment M_y "rotates" <u>about</u> the local member axis y, whereas a surface moment m_y acts <u>in</u> direction of the local surface axis y, that means about the axis x of the surface.

The following figure explains the definition of basic internal forces of surfaces:

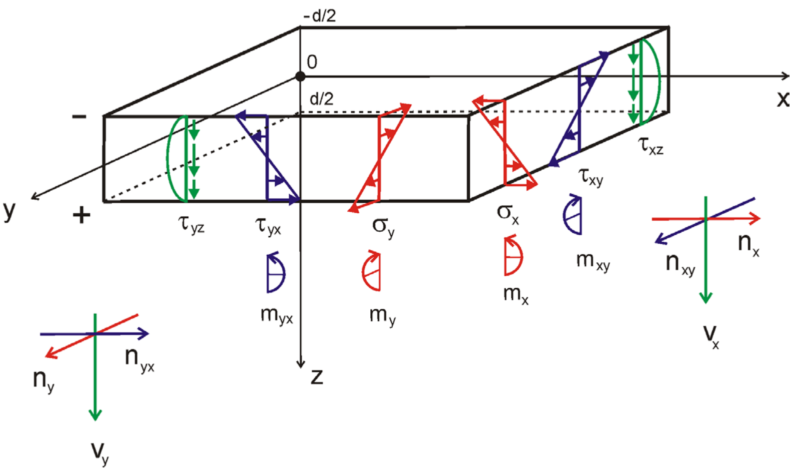


Figure 8.41: Surface internal forces and surface stresses

Moments as well as shear stresses acting perpendicular to the surface follow a parabolic graph across the surface thickness.

The signs help you to see on which side of the surface the internal force is available. However, signs depend also on the orientation of the global axis Z: If the global Z-axis is directed downwards (standard), positive internal forces generate tension stresses on the positive side of the surface (that means in direction of the positive surface axis z). They are visualized by blue bars in the table. Negative internal forces result in compression stresses on the positive side of the surface. They are represented by red bars in the table.

If the global axis Z is directed upwards, signs of the bending moments and shear forces are inverted.

When the Z-axis is directed downwards, basic internal forces are determined as follows:

m_x	Bending moment that creates stresses in direction of local axis x $m_x = \int_{-d/2}^{+d/2} \sigma_x z dz$
m_y	Bending moment that creates stresses in direction of local axis y $m_y = \int_{-d/2}^{+d/2} \sigma_y z dz$
m_{xy}	Torsional moment $m_{xy} = m_{yx} = \int_{-d/2}^{+d/2} \tau_{xy} z dz$
v_x	Shear force v_x $v_x = \int_{-d/2}^{+d/2} \tau_{xz} dz$
v_y	Shear force v_y $v_y = \int_{-d/2}^{+d/2} \tau_{yz} dz$
n_x	Axial force in direction of local axis x $n_x = \int_{-d/2}^{+d/2} \sigma_x dz$
n_y	Axial force in direction of local axis y $n_y = \int_{-d/2}^{+d/2} \sigma_y dz$
n_{xy}	Shear flow $n_{xy} = \int_{-d/2}^{+d/2} \tau_{xy} dz$

Table 8.7: Basic internal forces

8.15 Surfaces - Principal Internal Forces

To control the graphical display of the principal internal forces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Principal Internal Forces*. Table 4.15 shows the principal internal forces of surfaces in numerical form.

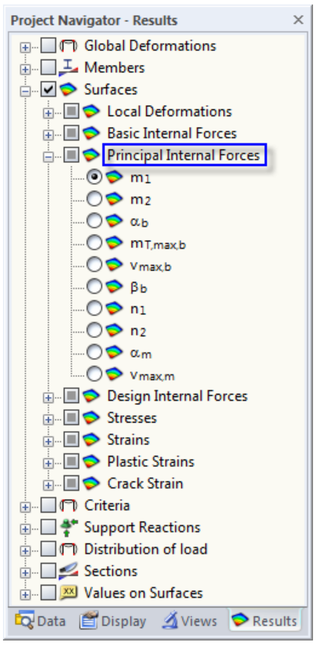


Figure 8.42: *Results* navigator: *Surfaces* → *Principal Internal Forces*

4.15 Surfaces - Principal Internal Forces

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Moments [kNm/m] m_1	F m_2	G α_b [°]	H $m_{T,max,b}$	I Shear Forces [kN/m] $v_{max,b}$	J β_b [°]	K Axial Force n_1	L n_2
1	1	0.000	0.000	0.000	-30.47	-256.11	-52.07	112.82	746.10	27.81	115.56	-17.34
	2	0.500	0.000	0.000	-4.67	-77.08	-74.16	36.21	341.47	14.75	95.02	22.24
	3	1.000	0.000	0.000	13.24	-12.57	41.53	12.90	152.30	10.28	45.31	0.42
	4	1.500	0.000	0.000	28.85	-2.81	22.00	15.83	117.91	4.09	35.95	-0.96
	5	2.000	0.000	0.000	39.32	-0.65	13.52	19.99	83.75	6.01	37.21	0.07
	6	2.500	0.000	0.000	46.25	0.38	6.85	22.94	50.13	8.20	40.44	-1.00
	7	3.000	0.000	0.000	52.68	0.70	4.94	25.99	20.75	-3.11	43.28	1.35
	8	3.500	0.000	0.000	53.54	-0.02	1.08	26.78	3.90	64.42	41.75	-0.34
	9	4.000	0.000	0.000	46.89	0.61	1.37	23.14	11.03	-179.34	38.66	0.00
	10	4.500	0.000	0.000	36.84	0.42	0.57	18.21	23.36	-177.71	37.10	0.10
	11	5.000	0.000	0.000	23.37	-0.33	-1.86	11.85	42.02	170.08	41.43	1.99
	12	5.500	0.000	0.000	7.20	-0.62	-15.23	3.91	51.44	178.17	42.96	5.14
	13	6.000	0.000	0.000	1.89	-14.58	-81.88	8.24	61.22	167.74	29.48	-0.21
	14	6.500	0.000	0.000	4.25	-55.77	89.87	30.01	149.67	-142.06	-42.88	-177.21
	15	7.000	0.000	0.000	-58.56	-105.24	71.85	23.34	462.94	87.75	393.93	-110.12

Surfaces - Global Deformations | Surfaces - Basic Internal Forces | Surfaces - Principal Internal Forces | Surfaces - Design Internal Forces

Figure 8.43: Table 4.15 *Surfaces - Principal Internal Forces*

The table shows the principal internal forces sorted by surfaces. The results are listed in reference to the grid points of each surface.

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.14 *Surfaces - Basic Internal Forces.*

Moments / shear forces / axial forces

The *Basic Internal Forces* described in the previous chapter refer to the more or less freely defined coordinate system xyz of a surface. In contrast, *Principal Internal Forces* represent the extreme values of the internal forces in a surface element. For this purpose, the basic internal forces are transformed in the directions of both principal axes. The principal axes 1 (maximum value) and 2 (minimum value) are arranged orthogonally.

The principal internal forces are determined from the basic internal forces:

m_1	Bending moment in direction of principal axis 1 $\frac{1}{2}\left(m_x + m_y + \sqrt{(m_x - m_y)^2 + 4 \cdot m_{xy}{}^2}\right)$
m_2	Bending moment in direction of principal axis 2 $\frac{1}{2}\left(m_x + m_y - \sqrt{(m_x - m_y)^2 + 4 \cdot m_{xy}{}^2}\right)$
α_b	Angle between local axis x (or y) and principal axis 1 (or 2) $\frac{1}{2}\left[\arctan\left(\frac{2 \cdot m_{xy}}{m_x - m_y}\right)\right]$
$m_{T,max,b}$	Maximum torsional moment $\frac{\sqrt{(m_x - m_y)^2 + 4 \cdot m_{xy}{}^2}}{2}$
$v_{max,b}$	Maximum resulting shear force from bending components $v_{max,b} = \sqrt{v_x{}^2 + v_y{}^2}$
β_b	Angle between principal shear force $v_{max,b}$ and local axis x $\beta = \arctan\frac{v_y}{v_x}$
n_1	Axial force in direction of principal axis 1 $\frac{1}{2}\left(n_x + n_y + \sqrt{(n_x - n_y)^2 + 4 \cdot n_{xy}{}^2}\right)$
n_2	Axial force in direction of principal axis 2 $\frac{1}{2}\left(n_x + n_y - \sqrt{(n_x - n_y)^2 + 4 \cdot n_{xy}{}^2}\right)$
α_m	Angle between axis x and principal axis 1 (for axial force n_1) $\frac{1}{2}\left[\arctan\left(\frac{2 \cdot n_{xy}}{n_x - n_y}\right)\right]$
$v_{max,m}$	Maximum shear force from membrane components $\frac{\sqrt{(n_x - n_y)^2 + 4 \cdot n_{xy}{}^2}}{2}$

Table 8.8: Principal internal forces

The directions of the principal axes α_b (for bending moments), β_b (for shear forces) and α_m (for axial forces) can be displayed as trajectories in the work window.

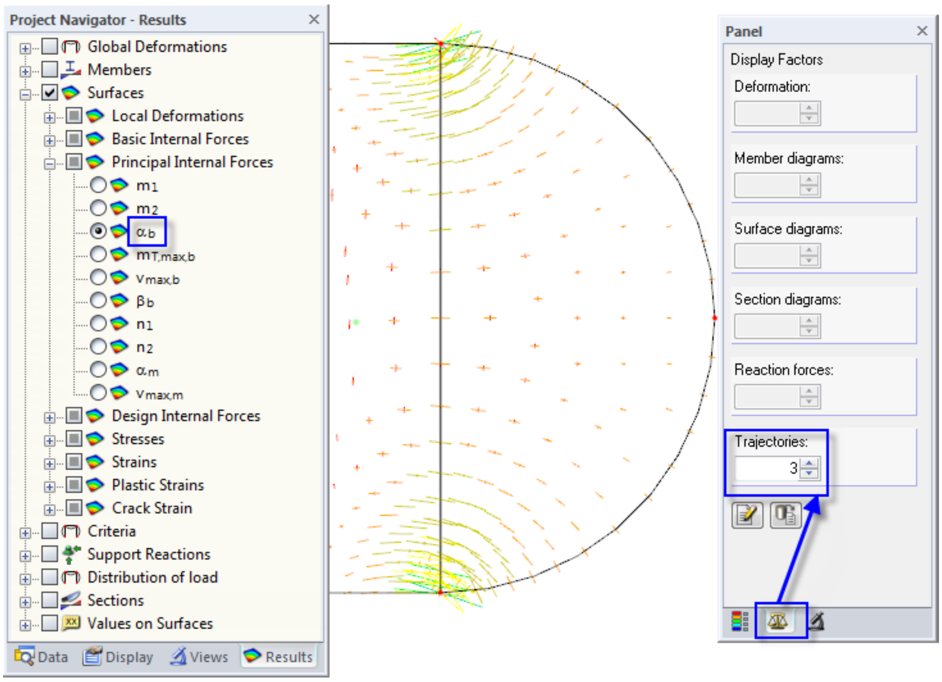


Figure 8.44: Trajectories of the principal axes

In the figure above, the display of angle α_b shows also the size of the respective principal moments because the trajectories are scaled to the values of the moments m_1 and m_2.

8.16 Surfaces - Design Internal Forces

To control the graphical display of the design internal forces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Design Internal Forces*. Table 4.16 shows the design internal forces of surfaces in numerical form.

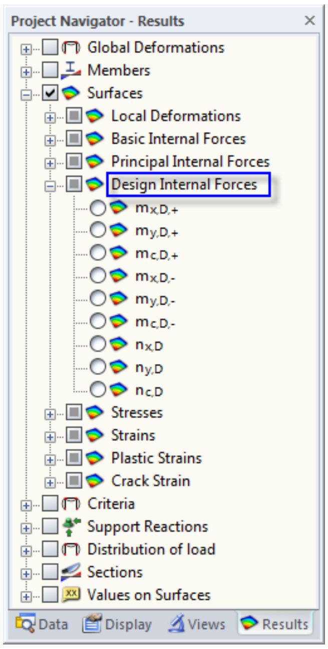


Figure 8.45: *Results* navigator: *Surfaces* → *Design Internal Forces*

4.16 Surfaces - Design Internal Forces

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J	K	L
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Moments [kNm/m] $m_{x,D+}$	$m_{y,D+}$	$m_{c,D+}$	$m_{x,D-}$	$m_{y,D-}$	$m_{c,D-}$	Axial Forces $n_{x,D}$	$n_{y,D}$
1	1	0.000	0.000	0.000	0.00	-45.67	-240.91	280.27	225.11	-218.80	139.60	2
	2	0.500	0.000	0.000	0.00	-5.03	-76.73	90.70	29.08	-38.02	109.26	6
	3	1.000	0.000	0.000	14.70	11.58	-25.61	10.91	14.03	-25.61	46.47	
	4	1.500	0.000	0.000	35.40	12.63	-21.99	0.00	3.32	-29.36	36.02	
	5	2.000	0.000	0.000	46.22	10.62	-18.17	0.00	0.69	-39.36	38.34	
	6	2.500	0.000	0.000	51.03	6.46	-10.86	0.00	-0.38	-46.24	42.66	
	7	3.000	0.000	0.000	56.75	5.55	-8.92	0.00	-0.71	-52.67	45.37	
	8	3.500	0.000	0.000	54.53	1.02	-2.03	0.00	0.02	-53.54	43.51	
	9	4.000	0.000	0.000	47.97	1.74	-2.21	0.00	-0.61	-46.89	39.13	
	10	4.500	0.000	0.000	37.19	0.78	-0.72	0.00	-0.42	-36.84	37.36	
	11	5.000	0.000	0.000	24.11	0.47	-1.54	0.00	0.33	-23.37	44.34	
	12	5.500	0.000	0.000	8.64	1.90	-3.96	0.00	0.67	-7.25	46.21	
	13	6.000	0.000	0.000	0.00	1.94	-14.62	16.55	0.74	-4.60	21.60	3
	14	6.500	0.000	0.000	0.00	4.25	-55.77	55.78	0.00	-4.26	0.00	-4
	15	7.000	0.000	0.000	0.00	-61.42	-102.39	114.85	82.36	-33.41	290.64	36

Surfaces - Basic Internal Forces | Surfaces - Principal Internal Forces | Surfaces - Design Internal Forces | Surfaces - Basic Stresses | Surfaces - Principal Stresses

Figure 8.46: Table 4.16 *Surfaces - Design Internal Forces*

The table shows the design internal forces sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ.

Moments / axial forces

The design moments and axial forces shown in this table are based on the approach described in DIN V ENV 1992-1-1, appendix 2, A 2.8 and A 2.9. In this way, RFEM provides users who do not have access to the design module RF-CONCRETE Surfaces with a kind of helping tool required for the manual reinforced concrete design. As the add-on module uses the method by BAUMANN, the design internal forces from RFEM are not applied in the module.

In this context, it is important to keep in mind that the design moments and axial forces of table 4.16 must not be combined. As explained in DIN V ENV 1992-1-1, annex 2.8, the moments refer exclusively to slab reinforcements. The axial forces are based on the design of wall elements described in annex 2.9.

The design internal forces have the following meanings:

$m_{x,D+}$	Design moment in direction of local axis x on positive side of surface (i.e. side in direction of positive surface axis z) $m_x + \|m_{xy}\|$ for $m_x \le m_y$ and $m_x \ge -\|m_{xy}\|$ for $m_x > m_y$ and $m_y \ge -\|m_{xy}\|$ 0 for $m_x \le m_y$ and $m_x < -\|m_{xy}\|$ $m_x + \frac{m_{xy}^2}{\|m_y\|}$ for $m_x > m_y$ and $m_y < -\|m_{xy}\|$
$m_{y,D+}$	Design moment in direction of local axis y on positive side of surface (i.e. side in direction of positive surface axis z) $m_y + \|m_{xy}\|$ for $m_x \le m_y$ and $m_x \ge -\|m_{xy}\|$ for $m_x > m_y$ and $m_y \ge -\|m_{xy}\|$ $m_y + \frac{m_{xy}^2}{\|m_x\|}$ for $m_x \le m_y$ and $m_x < -\|m_{xy}\|$ 0 for $m_x > m_y$ and $m_x < -\|m_{xy}\|$
$m_{c,D+}$	Design moment for concrete stress design on positive side of surface $-2 \cdot \|m_{xy}\|$ for $m_x \le m_y$ and $m_x \ge -\|m_{xy}\|$ for $m_x > m_y$ and $m_y \ge -\|m_{xy}\|$ $m_x - \frac{m_{xy}^2}{\|m_x\|}$ for $m_x \le m_y$ and $m_x < -\|m_{xy}\|$ $m_y - \frac{m_{xy}^2}{\|m_y\|}$ for $m_x > m_y$ and $m_x < -\|m_{xy}\|$
$m_{x,D-}$	Design moment in direction of axis x on negative side of surface $-m_x + \|m_{xy}\|$ for $m_x \le m_y$ and $m_y \le \|m_{xy}\|$ for $m_x > m_y$ and $m_x \le \|m_{xy}\|$ $-m_x + \frac{m_{xy}^2}{\|m_y\|}$ for $m_x \le m_y$ and $m_y > \|m_{xy}\|$ 0 for $m_x > m_y$ and $m_x > \|m_{xy}\|$

$m_{y,D-}$	Design moment in direction of axis y on negative side of surface $-m_y + \lvert m_{xy} \rvert$ for $mx \le my$ and $m_y \le \lvert m_{xy} \rvert$ for $m_x > m_y$ and $m_x \le \lvert m_{xy} \rvert$ 0 for $m_x \le m_y$ and $m_y > \lvert m_{xy} \rvert$ $-m_y + \frac{m_{xy}^2}{\lvert m_x \rvert}$ for $m_x > m_y$ and $m_x > \lvert m_{xy} \rvert$
$m_{c,D-}$	Design moment for concrete stress design on negative side of surface $-2 \cdot \lvert m_{xy} \rvert$ for $m_x \le m_y$ and $m_y \le \lvert m_{xy} \rvert$ for $m_x > m_y$ and $m_x \le \lvert m_{xy} \rvert$ $-m_y - \frac{m_{xy}^2}{\lvert m_y \rvert}$ for $m_x \le m_y$ and $m_y > \lvert m_{xy} \rvert$ $-m_x - \frac{m_{xy}^2}{\lvert m_x \rvert}$ for $m_x > m_y$ and $m_x > \lvert m_{xy} \rvert$
$n_{x,D}$	Design force in direction of local axis x $n_x + \lvert n_{xy} \rvert$ for $n_x \le n_y$ and $n_x \ge -\lvert n_{xy} \rvert$ for $n_x > n_y$ and $n_y \ge -\lvert n_{xy} \rvert$ 0 for $n_x \le n_y$ and $n_x < -\lvert n_{xy} \rvert$ $n_x + \frac{n_{xy}^2}{\lvert n_y \rvert}$ for $n_x > n_y$ and $n_y < -\lvert n_{xy} \rvert$
$n_{y,D}$	Design force in direction of local axis y $n_y + \lvert n_{xy} \rvert$ for $n_x \le n_y$ and $n_x \ge -\lvert n_{xy} \rvert$ for $n_x > n_y$ and $n_y \ge -\lvert n_{xy} \rvert$ $n_y + \frac{n_{xy}^2}{\lvert n_x \rvert}$ for $n_x \le n_y$ and $n_x < -\lvert n_{xy} \rvert$ 0 for $n_x > n_y$ and $n_y < -\lvert n_{xy} \rvert$
$n_{c,D}$	Design force for concrete stress design $-2 \cdot \lvert n_{xy} \rvert$ for $n_x \le n_y$ and $n_x \ge -\lvert n_{xy} \rvert$ for $n_x > n_y$ and $n_y \ge -\lvert n_{xy} \rvert$ $-\lvert n_x \rvert - \frac{n_{xy}^2}{\lvert n_x \rvert}$ for $n_x \le n_y$ and $n_x < -\lvert n_{xy} \rvert$ $-\lvert n_y \rvert - \frac{n_{xy}^2}{\lvert n_y \rvert}$ for $n_x > n_y$ and $n_y < -\lvert n_{xy} \rvert$

Table 8.9: Design internal forces

8.17 Surfaces - Basic Stresses

To control the graphical display of the basic stresses, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses*. Table 4.17 shows the basic stresses of surfaces in numerical form.

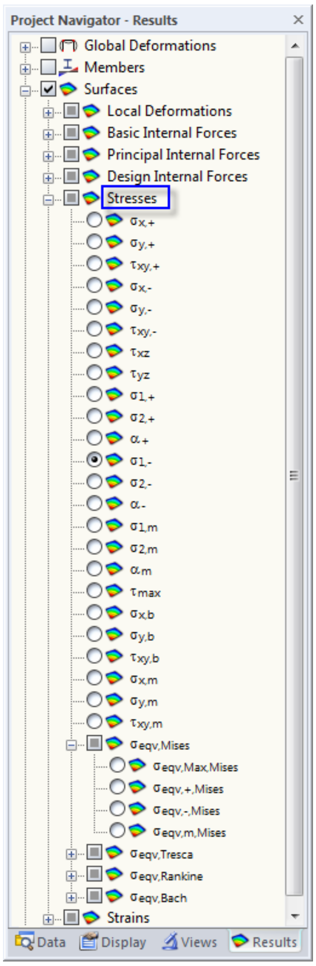
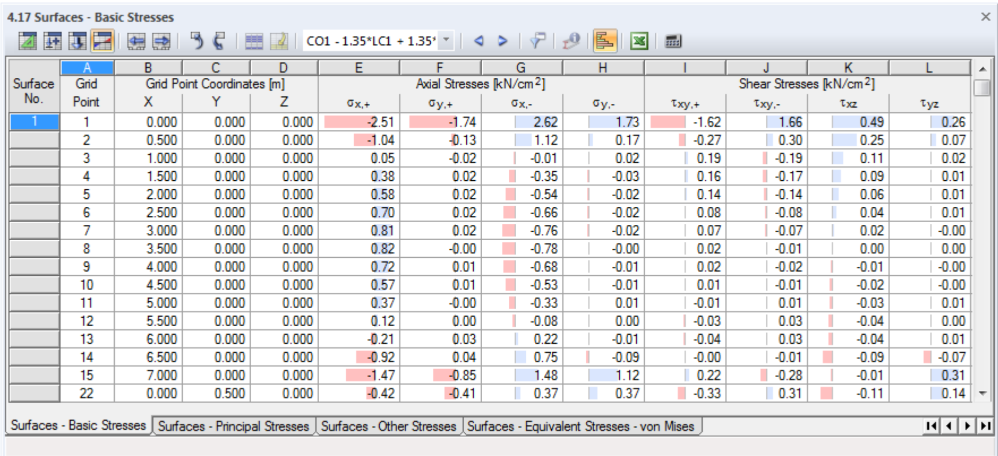


Figure 8.47: *Results* navigator: *Surfaces* → *Stresses*



4.17 Surfaces - Basic Stresses

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J	K	L
Surface No.	Grid Point	Grid Point Coordinates [m]			Axial Stresses [kN/cm²]				Shear Stresses [kN/cm²]			
		X	Y	Z	$\sigma_{x,+}$	$\sigma_{y,+}$	$\sigma_{x,-}$	$\sigma_{y,-}$	$\tau_{xy,+}$	$\tau_{xy,-}$	τ_{xz}	τ_{yz}
1	1	0.000	0.000	0.000	-2.51	-1.74	2.62	1.73	-1.62	1.66	0.49	0.26
	2	0.500	0.000	0.000	-1.04	-0.13	1.12	0.17	-0.27	0.30	0.25	0.07
	3	1.000	0.000	0.000	0.05	-0.02	-0.01	0.02	0.19	-0.19	0.11	0.02
	4	1.500	0.000	0.000	0.38	0.02	-0.35	-0.03	0.16	-0.17	0.09	0.01
	5	2.000	0.000	0.000	0.58	0.02	-0.54	-0.02	0.14	-0.14	0.06	0.01
	6	2.500	0.000	0.000	0.70	0.02	-0.66	-0.02	0.08	-0.08	0.04	0.01
	7	3.000	0.000	0.000	0.81	0.02	-0.76	-0.02	0.07	-0.07	0.02	-0.00
	8	3.500	0.000	0.000	0.82	-0.00	-0.78	-0.00	0.02	-0.01	0.00	0.00
	9	4.000	0.000	0.000	0.72	0.01	-0.68	-0.01	0.02	-0.02	-0.01	-0.00
	10	4.500	0.000	0.000	0.57	0.01	-0.53	-0.01	0.01	-0.01	-0.02	-0.00
	11	5.000	0.000	0.000	0.37	-0.00	-0.33	0.01	-0.01	0.01	-0.03	0.01
	12	5.500	0.000	0.000	0.12	0.00	-0.08	0.00	-0.03	0.03	-0.04	0.00
	13	6.000	0.000	0.000	-0.21	0.03	0.22	-0.01	-0.04	0.03	-0.04	0.01
	14	6.500	0.000	0.000	-0.92	0.04	0.75	-0.09	-0.00	-0.01	-0.09	-0.07
	15	7.000	0.000	0.000	-1.47	-0.85	1.48	1.12	0.22	-0.28	-0.01	0.31
	22	0.000	0.500	0.000	-0.42	-0.41	0.37	0.37	-0.33	0.31	-0.11	0.14

Surfaces - Basic Stresses | Surfaces - Principal Stresses | Surfaces - Other Stresses | Surfaces - Equivalent Stresses - von Mises

Figure 8.48: Table 4.17 *Surfaces - Basic Stresses*

The table shows the basic stresses sorted by surfaces. The results are listed in reference to the grid points of each surface.

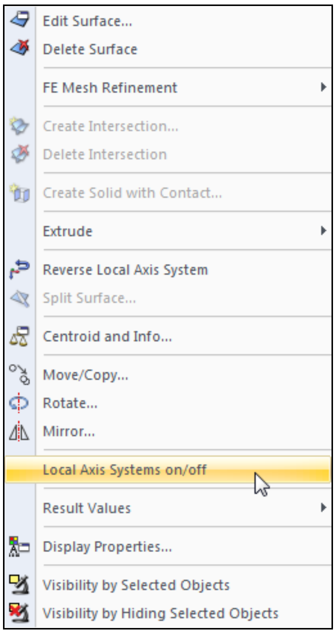
Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ.

Basic stresses



Context menu of surface

The stresses are related to the directions of the local surface axes. When you analyze curved surfaces, they refer to the local axes of the individual finite elements (see Figure 8.40, page 308).

The basic stresses are shown in Figure 8.41 on page 308 and have the following meanings:

$\sigma_{x,+}$	Stress in direction of local axis x on positive side of surface (i.e. side in direction of positive surface axis z) $\sigma_{x,+} = \frac{n_x}{d} + \frac{6 \cdot m_x}{d^2}$ with d: thickness of surface
$\sigma_{y,+}$	Stress in direction of local axis y on positive side of surface (i.e. side in direction of positive surface axis z) $\sigma_{y,+} = \frac{n_y}{d} + \frac{6 \cdot m_y}{d^2}$
$\sigma_{x,-}$	Stress in direction of axis x on negative side of surface $\sigma_{x,-} = \frac{n_x}{d} - \frac{6 \cdot m_x}{d^2}$
$\sigma_{y,-}$	Stress in direction of axis y on negative side of surface $\sigma_{y,-} = \frac{n_y}{d} - \frac{6 \cdot m_y}{d^2}$
$\tau_{xy,+}$	Torsional stress on positive side of surface $\tau_{xy,+} = \frac{n_{xy}}{d} + \frac{6 \cdot m_{xy}}{d^2}$
$\tau_{xy,-}$	Torsional stress on negative side of surface $\tau_{xy,-} = \frac{n_{xy}}{d} - \frac{6 \cdot m_{xy}}{d^2}$
τ_{xz}	Shear stress orthogonal to surface in direction of axis x $\frac{3 \cdot v_x}{2 \cdot d}$ with d: thickness of surface
τ_{yz}	Shear stress orthogonal to surface in direction of axis y $\frac{3 \cdot v_y}{2 \cdot d}$

Table 8.10: Basic stresses

8.18 Surfaces - Principal Stresses

To control the graphical display of principal stresses, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses* (see Figure 8.47, page 316). Table 4.18 shows the principal stresses of surfaces in numerical form.

4.18 Surfaces - Principal Stresses

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J	K	L	M	N
Surface No.	Grid Point	Grid Point Coordinates [m]			Principal Stresses [kN/cm²]									
		X	Y	Z	$\sigma_{1,+}$	$\sigma_{2,+}$	α_+ [°]	$\sigma_{1,-}$	$\sigma_{2,-}$	α_- [°]	$\sigma_{1,m}$	$\sigma_{2,m}$	α_m	τ_{max}
1	1	0.000	0.000	0.000	-0.46	-3.79	-51.66	3.89	0.46	37.52	0.06	-0.01	14.68	0.56
	2	0.500	0.000	0.000	-0.06	-1.11	-74.56	1.20	0.08	16.21	0.05	0.01	27.67	0.26
	3	1.000	0.000	0.000	0.21	-0.18	39.88	0.20	-0.19	-46.81	0.02	0.00	-1.53	0.11
	4	1.500	0.000	0.000	0.45	-0.04	21.24	0.04	-0.42	-67.19	0.02	-0.00	-0.41	0.09
	5	2.000	0.000	0.000	0.61	-0.01	13.08	0.01	-0.57	-76.02	0.02	0.00	-1.81	0.06
	6	2.500	0.000	0.000	0.71	0.01	6.56	-0.01	-0.67	-82.84	0.02	-0.00	-3.26	0.04
	7	3.000	0.000	0.000	0.81	0.01	4.74	-0.01	-0.77	-84.84	0.02	0.00	-3.01	0.02
	8	3.500	0.000	0.000	0.82	-0.00	1.12	0.00	-0.78	-88.95	0.02	-0.00	2.51	0.00
	9	4.000	0.000	0.000	0.72	0.01	1.35	-0.01	-0.68	-88.61	0.02	0.00	0.69	0.01
	10	4.500	0.000	0.000	0.57	0.01	0.56	-0.01	-0.53	-89.43	0.02	0.00	0.41	0.02
	11	5.000	0.000	0.000	0.37	-0.00	-2.01	0.01	-0.33	88.30	0.02	0.00	-4.63	0.03
	12	5.500	0.000	0.000	0.13	-0.01	-13.89	0.01	-0.09	72.95	0.02	0.00	-5.47	0.04
	13	6.000	0.000	0.000	0.04	-0.22	-80.65	0.22	-0.02	6.78	0.01	-0.00	-57.80	0.05
	14	6.500	0.000	0.000	0.04	-0.92	-89.76	0.75	-0.09	-0.55	-0.02	-0.09	-84.84	0.11
	15	7.000	0.000	0.000	-0.71	-1.60	-75.72	1.63	0.97	-30.41	0.20	-0.06	-65.51	0.35
	22	0.000	0.500	0.000	-0.08	-0.75	-45.23	0.68	0.06	45.09	-0.01	-0.03	-50.03	0.18

Surfaces - Design Internal Forces | Surfaces - Basic Stresses | Surfaces - Principal Stresses | Surfaces - Other Stresses | Surfaces - Equivalent Stresses - von Mises

Figure 8.49: Table 4.18 *Surfaces - Principal Stresses*

The table shows the principal stresses sorted by surfaces. The results are listed in reference to the grid points of each surface.

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.17 *Surfaces - Basic Stresses*.

Principal stresses

The basic stresses described in chapter 8.17 refer to the coordinate system xyz of the surface. The principal stresses, however, represent the extreme values of the stresses in a surface element. The principal axes 1 (maximum value) and 2 (minimum value) are arranged orthogonally.

It is possible to display the principal axis orientations α as trajectories in the work window (see Figure 8.44, page 312).

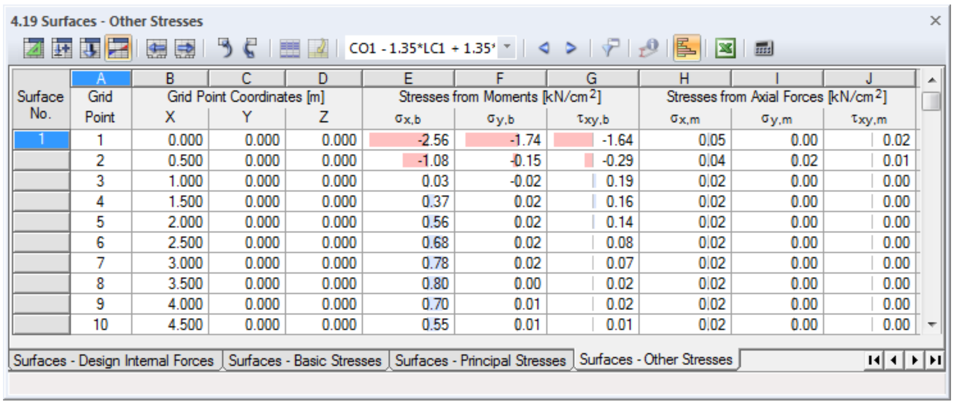
The principal stresses are determined from the basic stresses:

$\sigma_{1,+}$	Stress in direction of principal axis 1 on positive side of surface (i.e. side in direction of positive surface axis z) $$\sigma_{1,+} = \frac{1}{2}\left(\sigma_{x,+} + \sigma_{y,+} + \sqrt{(\sigma_{x,+} - \sigma_{y,+})^2 + 4 \cdot \tau_{xy,+}{}^2}\right)$$
$\sigma_{2,+}$	Stress in direction of principal axis 2 on positive side of surface (i.e. side in direction of positive surface axis z) $$\sigma_{2,+} = \frac{1}{2}\left(\sigma_{x,+} + \sigma_{y,+} - \sqrt{(\sigma_{x,+} - \sigma_{y,+})^2 + 4 \cdot \tau_{xy,+}{}^2}\right)$$
α_+	Angle between local axis x (or y) and principal axis 1 (or 2) for stresses on positive side of surface $$\alpha_+ = \frac{1}{2}\operatorname{atan2}\left(2 \cdot \tau_{xy,+}, \sigma_{x,+} - \sigma_{y,+}\right) \quad \in (-90°, 90°]$$
$\sigma_{1,-}$	Stress in direction of principal axis 1 on negative side of surface $$\sigma_{1,-} = \frac{1}{2}\left(\sigma_{x,-} + \sigma_{y,-} + \sqrt{(\sigma_{x,-} - \sigma_{y,-})^2 + 4 \cdot \tau_{xy,-}{}^2}\right)$$
$\sigma_{2,-}$	Stress in direction of principal axis 2 on negative side of surface $$\sigma_{2,-} = \frac{1}{2}\left(\sigma_{x,-} + \sigma_{y,-} - \sqrt{(\sigma_{x,-} - \sigma_{y,-})^2 + 4 \cdot \tau_{xy,-}{}^2}\right)$$
$\alpha-$	Angle between local axis x (or y) and principal axis 1 (or 2) for stresses on negative side of surface $$\alpha_- = \frac{1}{2}\operatorname{atan2}\left(2 \cdot \tau_{xy,-}, \sigma_{x,-} - \sigma_{y,-}\right) \quad \in (-90°, 90°]$$
$\sigma_{1,m}$	Membrane stress in direction of principal axis 1 $$\sigma_{1,m} = \frac{1}{2}\left(\sigma_{x,m} + \sigma_{y,m} + \sqrt{(\sigma_{x,m} - \sigma_{y,m})^2 + 4 \cdot \tau_{xy,m}{}^2}\right)$$
$\sigma_{2,m}$	Membrane stress in direction of principal axis 2 $$\sigma_{2,m} = \frac{1}{2}\left(\sigma_{x,m} + \sigma_{y,m} - \sqrt{(\sigma_{x,m} - \sigma_{y,m})^2 + 4 \cdot \tau_{xy,m}{}^2}\right)$$
α_m	Angle between local axis x and principal axis 1 for membrane stresses $$\alpha_m = \frac{1}{2}\operatorname{atan2}\left(2 \cdot \tau_{xy,m}, \sigma_{x,m} - \sigma_{y,m}\right) \quad \in (-90°, 90°]$$
τ_{max}	Maximum shear stress perpendicular to surface $$\tau_{max} = \sqrt{\tau_{xz}{}^2 + \tau_{yz}{}^2}$$

Table 8.11: Principal stresses

8.19 Surfaces - Other Stresses

To control the graphical display of stress components due to bending moments and membrane forces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses* (see Figure 8.47, page 316). Table 4.19 shows these stresses in numerical form.



4.19 Surfaces - Other Stresses

CO1 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J
Surface No.	Grid Point	Grid Point Coordinates [m]			Stresses from Moments [kN/cm²]			Stresses from Axial Forces [kN/cm²]		
		X	Y	Z	$\sigma_{x,b}$	$\sigma_{y,b}$	$\tau_{xy,b}$	$\sigma_{x,m}$	$\sigma_{y,m}$	$\tau_{xy,m}$
1	1	0.000	0.000	0.000	-2.56	-1.74	-1.64	0.05	0.00	0.02
	2	0.500	0.000	0.000	-1.08	-0.15	-0.29	0.04	0.02	0.01
	3	1.000	0.000	0.000	0.03	-0.02	0.19	0.02	0.00	0.00
	4	1.500	0.000	0.000	0.37	0.02	0.16	0.02	0.00	0.00
	5	2.000	0.000	0.000	0.56	0.02	0.14	0.02	0.00	0.00
	6	2.500	0.000	0.000	0.68	0.02	0.08	0.02	0.00	0.00
	7	3.000	0.000	0.000	0.78	0.02	0.07	0.02	0.00	0.00
	8	3.500	0.000	0.000	0.80	0.00	0.02	0.02	0.00	0.00
	9	4.000	0.000	0.000	0.70	0.01	0.02	0.02	0.00	0.00
	10	4.500	0.000	0.000	0.55	0.01	0.01	0.02	0.00	0.00

Surfaces - Design Internal Forces | Surfaces - Basic Stresses | Surfaces - Principal Stresses | Surfaces - Other Stresses

Figure 8.50: Table 4.19 *Surfaces - Other Stresses*

The table shows the other stresses sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ.

Stresses due to bending moments / axial forces

The stresses are related to the directions of the local surface axes. When you analyze curved surfaces, they refer to the axes of the finite elements (see Figure 8.40, page 308).

The stresses have the following meanings:

$\sigma_{x,b}$	Stress due to bending moment m_x $\sigma_{x,b} = \frac{6 \cdot m_x}{d^2}$ with d: thickness of surface
$\sigma_{y,b}$	Stress due to bending moment m_y $\sigma_{y,b} = \frac{6 \cdot m_y}{d^2}$
$\tau_{xy,b}$	Stress due to torsional moment m_{xy} $\tau_{xy,b} = \frac{6 \cdot m_{xy}}{d^2}$
$\sigma_{x,m}$	Membrane stress due to axial force n_x $\sigma_{x,m} = \frac{n_x}{d}$

$\sigma_{y,m}$	Membrane stress due to axial force n_y $\sigma_{y,m} = \frac{n_y}{d}$ with d: thickness of surface
$\tau_{xy,m}$	Membrane stress due to shear flow n_{xy} $\tau_{xy,m} = \frac{n_{xy}}{d}$

Table 8.12: Other stresses

8.20 Surfaces - Contact Stresses

When the model has surface supports (see chapter 4.9, page 104), table 4.20 shows the contact stresses ("soil contact pressures") of surfaces in numerical form. To control the graphical display of results, tick the check box for *Surfaces* in the *Results* navigator, and then select *Contact Stresses*.

For 2D-slabs only the table column σ_z is displayed.

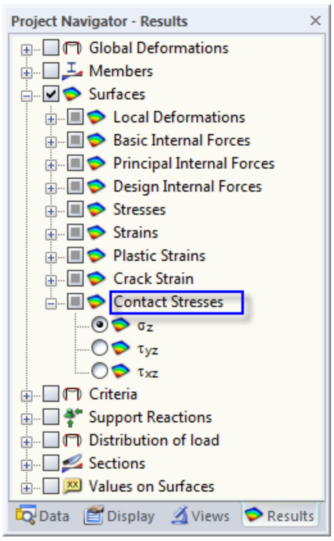
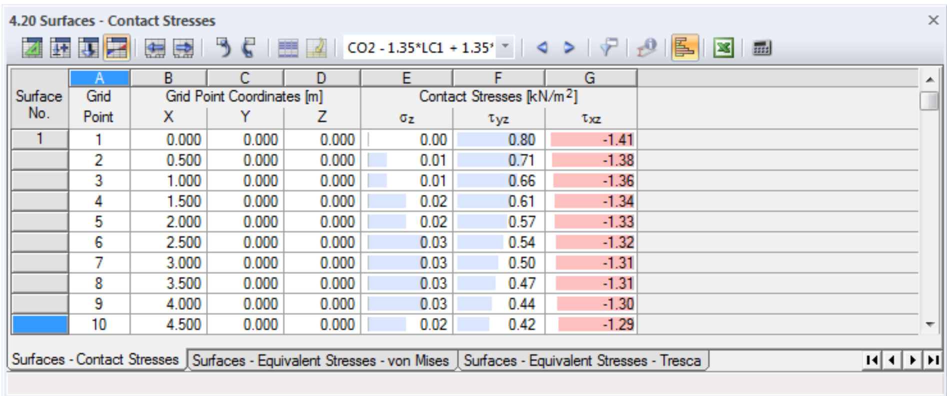


Figure 8.51: *Results* navigator: *Surfaces* → *Contact Stresses*



4.20 Surfaces - Contact Stresses

CO2 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Contact Stresses [kN/m²] σ_z	τ_{yz}	τ_{xz}
1	1	0.000	0.000	0.000	0.00	0.80	-1.41
	2	0.500	0.000	0.000	0.01	0.71	-1.38
	3	1.000	0.000	0.000	0.01	0.66	-1.36
	4	1.500	0.000	0.000	0.02	0.61	-1.34
	5	2.000	0.000	0.000	0.02	0.57	-1.33
	6	2.500	0.000	0.000	0.03	0.54	-1.32
	7	3.000	0.000	0.000	0.03	0.50	-1.31
	8	3.500	0.000	0.000	0.03	0.47	-1.31
	9	4.000	0.000	0.000	0.03	0.44	-1.30
	10	4.500	0.000	0.000	0.02	0.42	-1.29

Surfaces - Contact Stresses | Surfaces - Equivalent Stresses - von Mises | Surfaces - Equivalent Stresses - Tresca

Figure 8.52: Table 4.20 *Surfaces - Contact Stresses*

The table shows the contact stresses sorted by surfaces. The results are listed in reference to the grid points of each surface.

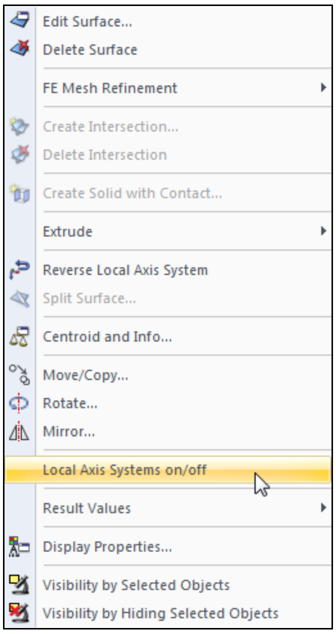
Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ. When you click into a table row, the corresponding grid point is indicated in the work window by an arrow, provided that the synchronization of selection is activated (see chapter 11.5.4, page 486).

Contact stresses



Context menu of surface

The stresses are related to the directions of the local surface axes. When you analyze curved surfaces, they refer to the axes of the finite elements (see Figure 8.40, page 308).

The contact stresses have the following meanings:

σ_z	Contact stress ("soil pressure") in direction of surface axis z $\sigma_z = \nu \cdot (\sigma_x + \sigma_y)$ with σ_x / σ_y : stresses in soil ν: Poisson's ratio of soil
τ_{yz}	Shear stress from surface support $\tau_{yz} = \frac{3 \cdot v_y}{2 \cdot d}$ with d: thickness of surface
τ_{xz}	Shear stress from surface support $\tau_{xz} = \frac{3 \cdot v_x}{2 \cdot d}$

Table 8.13: Contact stresses

Positive contact stresses are visualized by blue bars in the table. Accordingly, negative stresses are represented by red bars.

The table shows the stresses as forces per surface passed into the support. Thus, with regard to signs, the table does <u>not</u> show the reactions on the part of the support. If the local surface axis *z* is orientated downwards, a load for example acting in direction of the z-axis results in a positive stress σ_z. Thus, the signs result from the direction of the surface axis z (see Figure 4.73, page 83).

The orientation of the local surface axis z can be switched quickly for 3D models: Right-click the surface to open the surface context menu (see left figure above), and then select the option *Reverse Local Axis System*. Please note, however, that a defined ineffectivity will then change the direction of action as well.

8.21 Surfaces - Equivalent Stresses - von Mises

To control the graphical display of equivalent stresses of surfaces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses*. Table 4.21 shows the equivalent stresses determined according to VON MISES in numerical form.

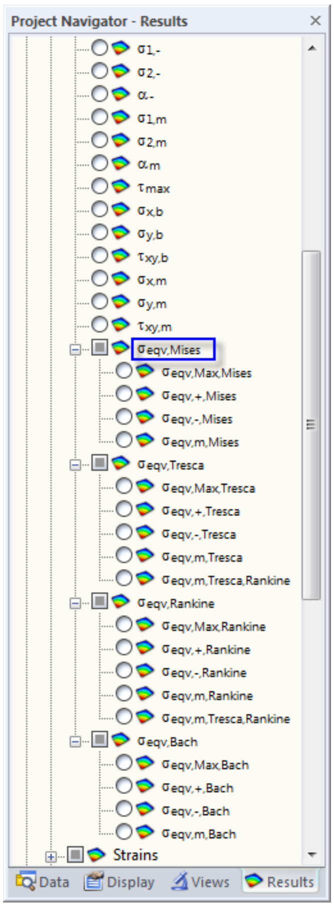
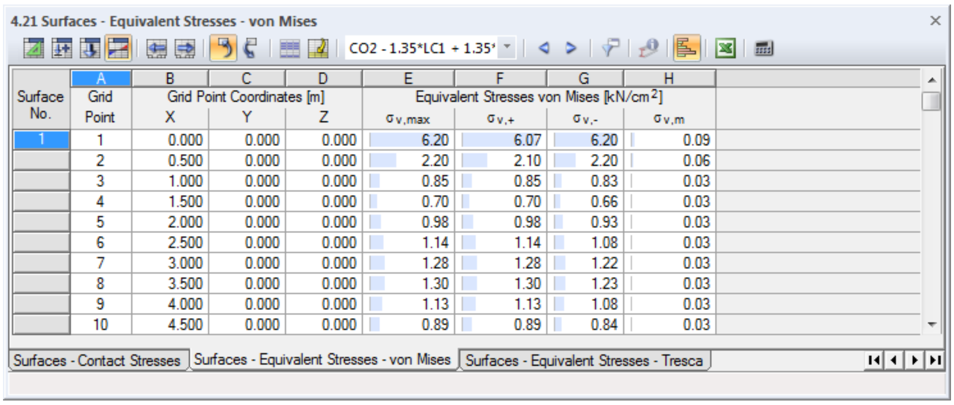


Figure 8.53: Equivalent stresses in *Results* navigator: *Surfaces* → *Stresses* → $\sigma_{eqv,Mises}$



4.21 Surfaces - Equivalent Stresses - von Mises

CO2 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Equivalent Stresses von Mises [kN/cm²] σv,max	σv,+	σv,-	σv,m
1	1	0.000	0.000	0.000	6.20	6.07	6.20	0.09
	2	0.500	0.000	0.000	2.20	2.10	2.20	0.06
	3	1.000	0.000	0.000	0.85	0.85	0.83	0.03
	4	1.500	0.000	0.000	0.70	0.70	0.66	0.03
	5	2.000	0.000	0.000	0.98	0.98	0.93	0.03
	6	2.500	0.000	0.000	1.14	1.14	1.08	0.03
	7	3.000	0.000	0.000	1.28	1.28	1.22	0.03
	8	3.500	0.000	0.000	1.30	1.30	1.23	0.03
	9	4.000	0.000	0.000	1.13	1.13	1.08	0.03
	10	4.500	0.000	0.000	0.89	0.89	0.84	0.03

Surfaces - Contact Stresses | Surfaces - Equivalent Stresses - von Mises | Surfaces - Equivalent Stresses - Tresca

Figure 8.54: Table 4.21 *Surfaces - Equivalent Stresses - von Mises*

The table shows the equivalent stresses sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ.

Equivalent stresses

In the *Results* navigator, four equivalent stress hypotheses for the plane stress condition are available for selection. The approach by VON MISES is also called "shape modification hypothesis". It is assumed that material fails as soon as the shape modifying energy exceeds a certain limit. This energy is the kind of energy that causes distortion or deformation of the object.

The approach represents the most well-known and frequently used equivalent stress hypothesis. It is appropriate for all materials that are not brittle. Therefore, it is widely used in steel building construction. However, the hypothesis is not adequate for hydrostatic stress conditions with equal principal stresses in all directions, as here the equivalent stress is zero.

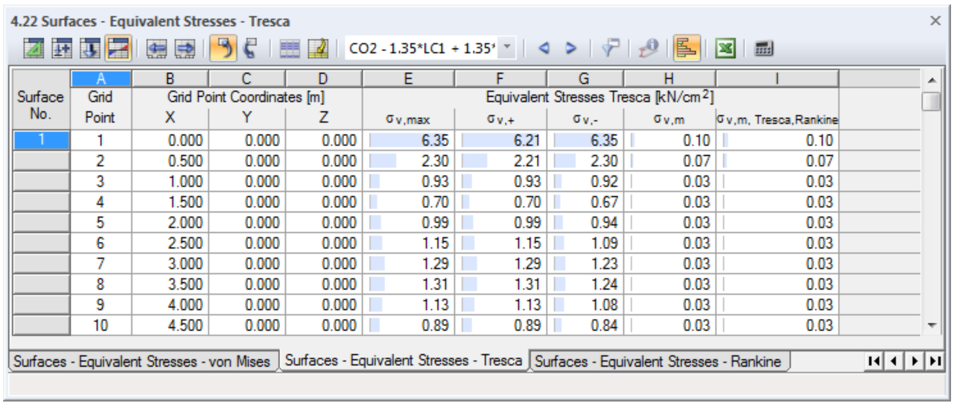
The equivalent stresses according to VON MISES for the plane stress conditions have the following meanings:

$\sigma_{eqv,+}$	Equivalent stress on positive side of surface (i.e. side in direction of positive surface axis z) $\sigma_{v,+} = \sqrt{\sigma_{x,+}^2 + \sigma_{y,+}^2 - \sigma_{x,+} \cdot \sigma_{y,+} + 3 \cdot \tau_{xy,+}^2}$
$\sigma_{eqv,-}$	Equivalent stress on negative side of surface $\sigma_{v,-} = \sqrt{\sigma_{x,-}^2 + \sigma_{y,-}^2 - \sigma_{x,-} \cdot \sigma_{y,-} + 3 \cdot \tau_{xy,-}^2}$
$\sigma_{eqv,m}$	Membrane equivalent stress $\sigma_{v,m} = \sqrt{\sigma_{x,m}^2 + \sigma_{y,m}^2 - \sigma_{x,m} \cdot \sigma_{y,m} + 3 \cdot \tau_{xy,m}^2}$

Table 8.14: Equivalent stresses according to VON MISES

8.22 Surfaces - Equivalent Stresses - Tresca

To control the graphical display of equivalent stresses of surfaces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses* (see Figure 8.53, page 323). Table 4.22 shows the surfaces' equivalent stresses determined according to TRESCA in numerical form.


4.22 Surfaces - Equivalent Stresses - Tresca

CO2 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Equivalent Stresses Tresca [kN/cm²] $\sigma_{v,max}$	$\sigma_{v,+}$	$\sigma_{v,-}$	$\sigma_{v,m}$	$\sigma_{v,m,Tresca,Rankine}$
1	1	0.000	0.000	0.000	6.35	6.21	6.35	0.10	0.10
	2	0.500	0.000	0.000	2.30	2.21	2.30	0.07	0.07
	3	1.000	0.000	0.000	0.93	0.93	0.92	0.03	0.03
	4	1.500	0.000	0.000	0.70	0.70	0.67	0.03	0.03
	5	2.000	0.000	0.000	0.99	0.99	0.94	0.03	0.03
	6	2.500	0.000	0.000	1.15	1.15	1.09	0.03	0.03
	7	3.000	0.000	0.000	1.29	1.29	1.23	0.03	0.03
	8	3.500	0.000	0.000	1.31	1.31	1.24	0.03	0.03
	9	4.000	0.000	0.000	1.13	1.13	1.08	0.03	0.03
	10	4.500	0.000	0.000	0.89	0.89	0.84	0.03	0.03

Surfaces - Equivalent Stresses - von Mises | Surfaces - Equivalent Stresses - Tresca | Surfaces - Equivalent Stresses - Rankine

Figure 8.55: Table 4.22 *Surfaces - Equivalent Stresses - Tresca*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.21 *Surfaces - Equivalent Stresses - von Mises*.

The approach by TRESCA is also known as "maximum shear stress theory". It is assumed that failure is caused by the maximum shear stress. As this hypothesis is especially applicable for brittle materials, it is frequently used in mechanical engineering.

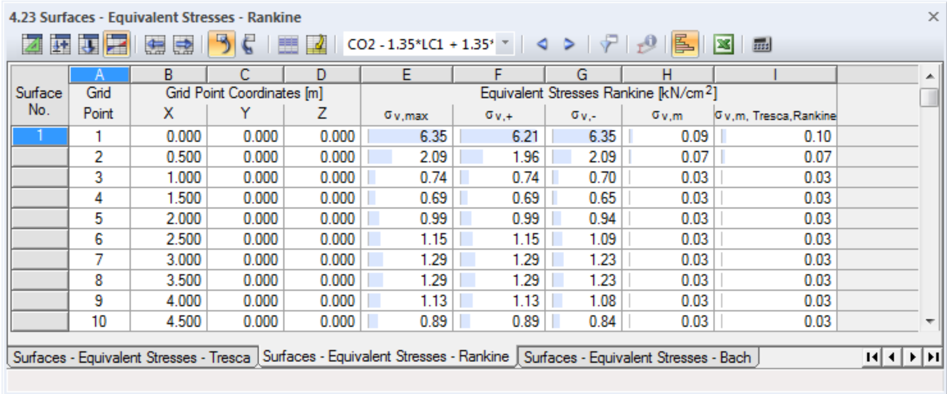
The equivalent stresses according to TRESCA are determined as follows:

$\sigma_{eqv,max}$	Maximum of equivalent stress on positive and negative side of surface
$\sigma_{eqv,+}$	Equivalent stress on positive side of surface $\sigma_{v,+} = \max\left(\left\|\sigma_{1,+} - \sigma_{2,+}\right\|;\left\|\sigma_{2,+}\right\|;\left\|\sigma_{1,+}\right\|\right)$ or $\sigma_{v,+} = \max\left(\sqrt{\left(\sigma_{x,+} - \sigma_{y,+}\right)^2 + 4 \cdot \tau_{xy,+}^{\;2}};\left\|\sigma_{2,+}\right\|;\left\|\sigma_{1,+}\right\|\right)$
$\sigma_{eqv,-}$	Equivalent stress on negative side of surface $\sigma_{v,-} = \max\left(\left\|\sigma_{1,-} - \sigma_{2,-}\right\|;\left\|\sigma_{2,-}\right\|;\left\|\sigma_{1,-}\right\|\right)$ or $\sigma_{v,-} = \max\left(\sqrt{\left(\sigma_{x,-} - \sigma_{y,-}\right)^2 + 4 \cdot \tau_{xy,-}^{\;2}};\left\|\sigma_{2,-}\right\|;\left\|\sigma_{1,-}\right\|\right)$
$\sigma_{eqv,m}$	Membrane equivalent stress $\sigma_{v,m} = \max\left(\left\|\sigma_{1,m} - \sigma_{2,m}\right\|;\left\|\sigma_{2,m}\right\|;\left\|\sigma_{1,m}\right\|\right)$ or $\sigma_{v,m} = \max\left(\sqrt{\left(\sigma_{x,m} - \sigma_{y,m}\right)^2 + 4 \cdot \tau_{xy,m}^{\;2}};\left\|\sigma_{2,m}\right\|;\left\|\sigma_{1,m}\right\|\right)$

Table 8.15: Equivalent stresses according to TRESCA

8.23 Surfaces - Equivalent Stresses - Rankine

To control the graphical display of equivalent stresses of surfaces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses* (see Figure 8.53, page 323). Table 4.23 shows the surfaces' equivalent stresses determined according to RANKINE in numerical form.



Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Equivalent Stresses Rankine [kN/cm²] $\sigma_{v,max}$	$\sigma_{v,+}$	$\sigma_{v,-}$	$\sigma_{v,m}$	$\sigma_{v,m,Tresca,Rankine}$
1	1	0.000	0.000	0.000	6.35	6.21	6.35	0.09	0.10
	2	0.500	0.000	0.000	2.09	1.96	2.09	0.07	0.07
	3	1.000	0.000	0.000	0.74	0.74	0.70	0.03	0.03
	4	1.500	0.000	0.000	0.69	0.69	0.65	0.03	0.03
	5	2.000	0.000	0.000	0.99	0.99	0.94	0.03	0.03
	6	2.500	0.000	0.000	1.15	1.15	1.09	0.03	0.03
	7	3.000	0.000	0.000	1.29	1.29	1.23	0.03	0.03
	8	3.500	0.000	0.000	1.29	1.29	1.23	0.03	0.03
	9	4.000	0.000	0.000	1.13	1.13	1.08	0.03	0.03
	10	4.500	0.000	0.000	0.89	0.89	0.84	0.03	0.03

Figure 8.56: Table 4.23 *Surfaces - Equivalent Stresses - Rankine*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the results table 4.21 *Surfaces - Equivalent Stresses - von Mises*.

The equivalent stress hypothesis by RANKINE is also known as "maximum principal stress criterion". It is assumed that failure is caused by the maximum principal stress.

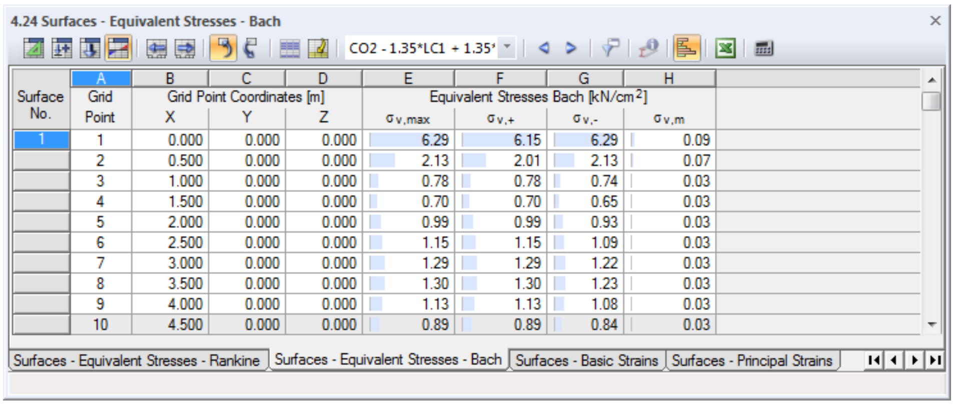
The equivalent stresses according to RANKINE are determined as follows:

$\sigma_{eqv,max}$	Maximum of equivalent stress on positive and negative side of surface
$\sigma_{eqv,+}$	Maximum absolute value of equivalent stress on positive side of surface $$\sigma_{v,+} = \frac{1}{2}\left\|\sigma_{x,+} + \sigma_{y,+}\right\| + \frac{1}{2}\sqrt{\left(\sigma_{x,+} - \sigma_{y,+}\right)^2 + 4 \cdot \tau_{xy,+}{}^2}$$
$\sigma_{eqv,-}$	Maximum absolute value of equivalent stress on negative side of surface $$\sigma_{v,-} = \frac{1}{2}\left\|\sigma_{x,-} + \sigma_{y,-}\right\| + \frac{1}{2}\sqrt{\left(\sigma_{x,-} - \sigma_{y,-}\right)^2 + 4 \cdot \tau_{xy,-}{}^2}$$
$\sigma_{eqv,m}$	Maximum absolute value of membrane equivalent stress $$\sigma_{v,m} = \frac{1}{2}\left\|\sigma_{x,m} + \sigma_{y,m}\right\| + \frac{1}{2}\sqrt{\left(\sigma_{x,m} - \sigma_{y,m}\right)^2 + 4 \cdot \tau_{xy,m}{}^2}$$

Table 8.16: Equivalent stresses according to RANKINE

8.24 Surfaces - Equivalent Stresses - Bach

To control the graphical display of equivalent stresses of surfaces, tick the check box for *Surfaces* in the *Results* navigator, and then select *Stresses* (see Figure 8.53, page 323). Table 4.24 shows the surfaces' equivalent stresses determined according to BACH in numerical form.


4.24 Surfaces - Equivalent Stresses - Bach

CO2 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H
Surface No.	Grid Point	Grid Point Coordinates [m]			Equivalent Stresses Bach [kN/cm²]			
		X	Y	Z	$\sigma_{v,max}$	$\sigma_{v,+}$	$\sigma_{v,-}$	$\sigma_{v,m}$
1	1	0.000	0.000	0.000	6.29	6.15	6.29	0.09
	2	0.500	0.000	0.000	2.13	2.01	2.13	0.07
	3	1.000	0.000	0.000	0.78	0.78	0.74	0.03
	4	1.500	0.000	0.000	0.70	0.70	0.65	0.03
	5	2.000	0.000	0.000	0.99	0.99	0.93	0.03
	6	2.500	0.000	0.000	1.15	1.15	1.09	0.03
	7	3.000	0.000	0.000	1.29	1.29	1.22	0.03
	8	3.500	0.000	0.000	1.30	1.30	1.23	0.03
	9	4.000	0.000	0.000	1.13	1.13	1.08	0.03
	10	4.500	0.000	0.000	0.89	0.89	0.84	0.03

Surfaces - Equivalent Stresses - Rankine | Surfaces - Equivalent Stresses - Bach | Surfaces - Basic Strains | Surfaces - Principal Strains

Figure 8.57: Table 4.24 *Surfaces - Equivalent Stresses - Bach*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the results table 4.21 *Surfaces - Equivalent Stresses - von Mises*.

The equivalent stress hypothesis by BACH is also called "principal strain criterion". It is assumed that the failure occurs in the direction of the greatest strain. The approach is similar to the stress determination according to RANKINE described in chapter 8.23. Here, the principal strain is used instead of the principal stress.

The equivalent stresses according to BACH are determined as follows:

$\sigma_{eqv,max}$	Maximum of equivalent stress on positive and negative side of surface
$\sigma_{eqv,+}$	Maximum absolute value of equivalent stress on positive side of surface $\sigma_{v,+} = \max\left[\frac{1-\nu}{2}\left\|\sigma_{x,+}+\sigma_{y,+}\right\| + \frac{1+\nu}{2}\sqrt{\left(\sigma_{x,+}-\sigma_{y,+}\right)^2 + 4\cdot\tau_{xy,+}{}^2},\ \nu\left\|\sigma_{x,+}+\sigma_{y,+}\right\|\right]$ with ν: Poisson's ratio (see chapter 4.3, page 61)
$\sigma_{eqv,-}$	Maximum absolute value of equivalent stress on negative side of surface $\sigma_{v,+} = \max\left[\frac{1-\nu}{2}\left\|\sigma_{x,-}+\sigma_{y,-}\right\| + \frac{1+\nu}{2}\sqrt{\left(\sigma_{x,-}-\sigma_{y,-}\right)^2 + 4\cdot\tau_{xy,-}{}^2},\ \nu\left\|\sigma_{x,-}+\sigma_{y,-}\right\|\right]$
$\sigma_{eqv,m}$	Maximum absolute value of membrane equivalent stress $\sigma_{v,+} = \max\left[\frac{1-\nu}{2}\left\|\sigma_{x,m}+\sigma_{y,m}\right\| + \frac{1+\nu}{2}\sqrt{\left(\sigma_{x,m}-\sigma_{y,m}\right)^2 + 4\cdot\tau_{xy,m}{}^2},\ \nu\left\|\sigma_{x,m}+\sigma_{y,m}\right\|\right]$

Table 8.17: Equivalent stresses according to BACH

8.25 Surfaces - Basic Strains

To control the graphical display of surface strains, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains*. Table 4.25 shows the basic strains of surfaces in numerical form.

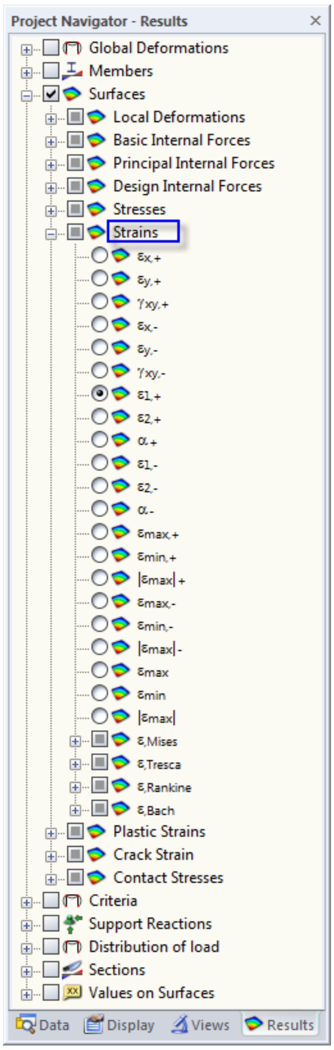
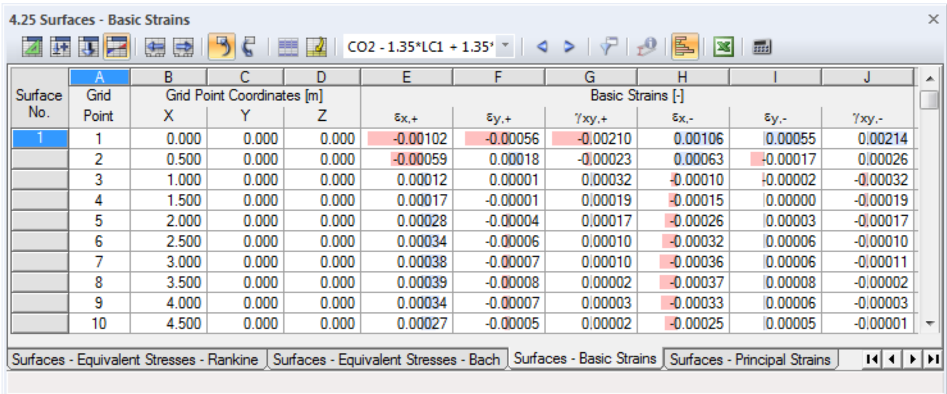


Figure 8.58: *Results* navigator: *Surfaces* → *Strains*



4.25 Surfaces - Basic Strains

CO2 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J
Surface No.	Grid Point	Grid Point Coordinates [m]			Basic Strains [-]					
		X	Y	Z	εx,+	εy,+	γxy,+	εx,-	εy,-	γxy,-
1	1	0.000	0.000	0.000	-0.00102	-0.00056	-0.00210	0.00106	0.00055	0.00214
	2	0.500	0.000	0.000	-0.00059	0.00018	-0.00023	0.00063	-0.00017	0.00026
	3	1.000	0.000	0.000	0.00012	0.00001	0.00032	-0.00010	-0.00002	-0.00032
	4	1.500	0.000	0.000	0.00017	-0.00001	0.00019	-0.00015	0.00000	-0.00019
	5	2.000	0.000	0.000	0.00028	-0.00004	0.00017	-0.00026	0.00003	-0.00017
	6	2.500	0.000	0.000	0.00034	-0.00006	0.00010	-0.00032	0.00006	-0.00010
	7	3.000	0.000	0.000	0.00038	-0.00007	0.00010	-0.00036	0.00006	-0.00011
	8	3.500	0.000	0.000	0.00039	-0.00008	0.00002	-0.00037	0.00008	-0.00002
	9	4.000	0.000	0.000	0.00034	-0.00007	0.00003	-0.00033	0.00006	-0.00003
	10	4.500	0.000	0.000	0.00027	-0.00005	0.00002	-0.00025	0.00005	-0.00001

Surfaces - Equivalent Stresses - Rankine | Surfaces - Equivalent Stresses - Bach | Surfaces - Basic Strains | Surfaces - Principal Strains

Figure 8.59: Table 4.25 *Surfaces - Basic Strains*

The table shows the strains sorted by surfaces. The results are listed in reference to the grid points of each surface.

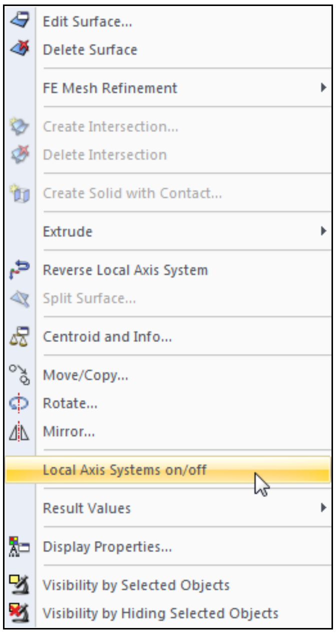
Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ.

Basic strains



Context menu of surface

The strains are related to the directions of the local surface axes. When you analyze curved surfaces, they refer to the axes of the finite elements (see Figure 8.40, page 308).

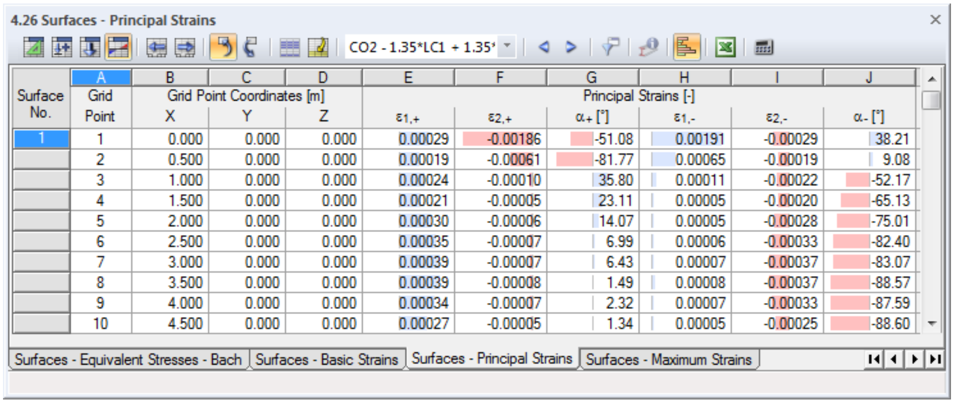
The basic strains have the following meanings:

$\varepsilon_{x,+}$	Strain in direction of local axis x on positive side of surface (i.e. side in direction of positive surface axis z) $\varepsilon_{x,+} = \frac{\partial u}{\partial x} + \frac{d}{2} \cdot \frac{\partial \varphi_y}{\partial x}$ with d: thickness of surface
ε_{y+}	Strain in direction of local axis y on positive side of surface (i.e. side in direction of positive surface axis z) $\varepsilon_{x,+} = \frac{\partial v}{\partial y} + \frac{d}{2} \cdot \left(- \frac{\partial \varphi_x}{\partial y} \right)$
$\gamma_{xy,+}$	Related rotation on positive side of surface $\gamma_{xy,+} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{d}{2} \cdot \left(\frac{\partial \varphi_y}{\partial y} - \frac{\partial \varphi_x}{\partial x} \right)$
$\varepsilon_{x,-}$	Strain in direction of axis x on negative side of surface $\varepsilon_{x,-} = \frac{\partial u}{\partial x} - \frac{d}{2} \cdot \frac{\partial \varphi_y}{\partial x}$
$\varepsilon_{y,-}$	Strain in direction of axis y on negative side of surface $\varepsilon_{x,-} = \frac{\partial v}{\partial y} - \frac{d}{2} \cdot \left(- \frac{\partial \varphi_x}{\partial y} \right)$
$\gamma_{xy,-}$	Related rotation on negative side of surface $\gamma_{xy,+} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} - \frac{d}{2} \cdot \left(\frac{\partial \varphi_y}{\partial y} - \frac{\partial \varphi_x}{\partial x} \right)$

Table 8.18: Basic strains

8.26 Surfaces - Principal Strains

To control the graphical display of strains, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains* (see Figure 8.58, page 328). Table 4.26 shows the principal strains of surfaces in numerical form.



	A	B	C	D	E	F	G	H	I	J
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Principal Strains [-] $\varepsilon_{1,+}$	$\varepsilon_{2,+}$	α_+ [°]	$\varepsilon_{1,-}$	$\varepsilon_{2,-}$	α_- [°]
1	1	0.000	0.000	0.000	0.00029	-0.00186	-51.08	0.00191	-0.00029	38.21
	2	0.500	0.000	0.000	0.00019	-0.00061	-81.77	0.00065	-0.00019	9.08
	3	1.000	0.000	0.000	0.00024	-0.00010	35.80	0.00011	-0.00022	-52.17
	4	1.500	0.000	0.000	0.00021	-0.00005	23.11	0.00005	-0.00020	-65.13
	5	2.000	0.000	0.000	0.00030	-0.00006	14.07	0.00005	-0.00028	-75.01
	6	2.500	0.000	0.000	0.00035	-0.00007	6.99	0.00006	-0.00033	-82.40
	7	3.000	0.000	0.000	0.00039	-0.00007	6.43	0.00007	-0.00037	-83.07
	8	3.500	0.000	0.000	0.00039	-0.00008	1.49	0.00008	-0.00037	-88.57
	9	4.000	0.000	0.000	0.00034	-0.00007	2.32	0.00007	-0.00033	-87.59
	10	4.500	0.000	0.000	0.00027	-0.00005	1.34	0.00005	-0.00025	-88.60

Figure 8.60: Table 4.26 *Surfaces - Principal Strains*

The table shows the principal strains sorted by surfaces. The results are listed in reference to the grid points of each surface.

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.25 *Surfaces - Basic Strains*.

Principal strains

The basic strains described in chapter 8.25 refer to the coordinate system xyz of the surface. The principal strains, however, represent the extreme values of the strains in a surface element. The principal axes 1 (maximum value) and 2 (minimum value) are arranged orthogonally.

It is possible to display the principal axis orientations α as trajectories in the work window (see Figure 8.44, page 312 for principal internal forces).

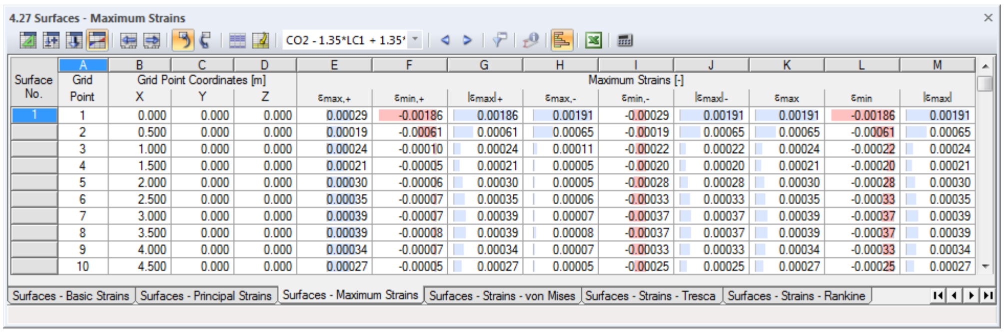
The principal strains have the following meanings:

$\varepsilon_{1,+}$	Strain in direction of principal axis 1 on positive side of surface (i.e. side in direction of positive surface axis z) $$\varepsilon_{1,+} = \frac{1}{2}\left(\varepsilon_{x,+} + \varepsilon_{y,+} + \sqrt{(\varepsilon_{x,+} - \varepsilon_{y,+})^2 + \gamma_{xy,+}{}^2}\right)$$
ε_{2+}	Strain in direction of principal axis 2 on positive side of surface (i.e. side in direction of positive surface axis z) $$\varepsilon_{2,+} = \frac{1}{2}\left(\varepsilon_{x,+} + \varepsilon_{y,+} - \sqrt{(\varepsilon_{x,+} - \varepsilon_{y,+})^2 + \gamma_{xy,+}{}^2}\right)$$
α_{+}	Angle between local axis x (or y) and principal axis 1 (or 2) for strains on positive side of surface $$\alpha_{+} = \frac{1}{2}\left(\arctan\left(\frac{\gamma_{xy,+}}{\varepsilon_{x,+} - \varepsilon_{y,+}}\right)\right)$$
$\varepsilon_{1,-}$	Strain in direction of principal axis 1 on negative side of surface $$\varepsilon_{1,-} = \frac{1}{2}\left(\varepsilon_{x,-} + \varepsilon_{y,-} + \sqrt{(\varepsilon_{x,-} - \varepsilon_{y,-})^2 + \gamma_{xy,-}{}^2}\right)$$
$\varepsilon_{2,-}$	Strain in direction of principal axis 2 on negative side of surface $$\varepsilon_{2,-} = \frac{1}{2}\left(\varepsilon_{x,-} + \varepsilon_{y,-} - \sqrt{(\varepsilon_{x,-} - \varepsilon_{y,-})^2 + \gamma_{xy,-}{}^2}\right)$$
α_{-}	Angle between local axis x (or y) and principal axis 1 (or 2) for stresses on negative side of surface $$\alpha_{-} = \frac{1}{2}\left(\arctan\left(\frac{\gamma_{xy,-}}{\varepsilon_{x,-} - \varepsilon_{y,-}}\right)\right)$$

Table 8.19: Principal strains

8.27 Surfaces - Maximum Strains

To control the graphical display for the extreme values of strains, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains* (see Figure 8.58, page 328). Table 4.27 shows these strains in numerical form.


4.27 Surfaces - Maximum Strains

CO2 - 1.35*LC1 + 1.35*

	A	B	C	D	E	F	G	H	I	J	K	L	M
Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Maximum Strains [-] $\varepsilon_{max,+}$	$\varepsilon_{min,+}$	$\lvert\varepsilon_{max}\rvert_+$	$\varepsilon_{max,-}$	$\varepsilon_{min,-}$	$\lvert\varepsilon_{max}\rvert_-$	ε_{max}	ε_{min}	$\lvert\varepsilon_{max}\rvert$
1	1	0.000	0.000	0.000	0.00029	-0.00186	0.00186	0.00191	-0.00029	0.00191	0.00191	-0.00186	0.00191
	2	0.500	0.000	0.000	0.00019	-0.00061	0.00061	0.00065	-0.00019	0.00065	0.00065	-0.00061	0.00065
	3	1.000	0.000	0.000	0.00024	-0.00010	0.00024	0.00011	-0.00022	0.00022	0.00024	-0.00022	0.00024
	4	1.500	0.000	0.000	0.00021	-0.00005	0.00021	0.00005	-0.00020	0.00020	0.00021	-0.00020	0.00021
	5	2.000	0.000	0.000	0.00030	-0.00006	0.00030	0.00005	-0.00028	0.00028	0.00030	-0.00028	0.00030
	6	2.500	0.000	0.000	0.00035	-0.00007	0.00035	0.00006	-0.00033	0.00033	0.00035	-0.00033	0.00035
	7	3.000	0.000	0.000	0.00039	-0.00007	0.00039	0.00007	-0.00037	0.00037	0.00039	-0.00037	0.00039
	8	3.500	0.000	0.000	0.00039	-0.00008	0.00039	0.00008	-0.00037	0.00037	0.00039	-0.00037	0.00039
	9	4.000	0.000	0.000	0.00034	-0.00007	0.00034	0.00007	-0.00033	0.00033	0.00034	-0.00033	0.00034
	10	4.500	0.000	0.000	0.00027	-0.00005	0.00027	0.00005	-0.00025	0.00025	0.00027	-0.00025	0.00027

Surfaces - Basic Strains | Surfaces - Principal Strains | Surfaces - Maximum Strains | Surfaces - Strains - von Mises | Surfaces - Strains - Tresca | Surfaces - Strains - Rankine

Figure 8.61: Table 4.27 *Surfaces - Maximum Strains*

The table shows the extreme values of strains sorted by surfaces. The results are listed in reference to the grid points of each surface.

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the results table 4.25 *Surfaces - Basic Strains*.

Maximum strains

These values represent the extreme values of the strains determined by the equations shown in Table 8.19.

$\varepsilon_{max,+}$	Maximum value of strain on positive side of surface (i.e. side in direction of positive surface axis z)
$\varepsilon_{min,+}$	Minimum value of strain on positive side of surface
$\lvert\varepsilon_{max}\rvert_+$	Maximum absolute value of both extreme values on positive side of surface
$\varepsilon_{max,-}$	Maximum value of strain on negative side of surface
$\varepsilon_{min,-}$	Minimum value of strain on negative side of surface
$\lvert\varepsilon_{max}\rvert_-$	Maximum absolute value of both extreme values on negative side of surface
ε_{max}	Maximum value of strain on positive or negative side of surface (columns E and H)
ε_{min}	Minimum value of strain on positive or negative side of surface (columns F and I)
$\lvert\varepsilon_{max}\rvert$	Maximum absolute value of strain on positive or negative side of surface (columns K and L)

Table 8.20: Maximum strains

8.28 Surfaces - Strains - von Mises

To control the graphical display of surface strains available with the equivalent stress hypothesis according to *von Mises*, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains*. Table 4.28 shows these strains in numerical form.

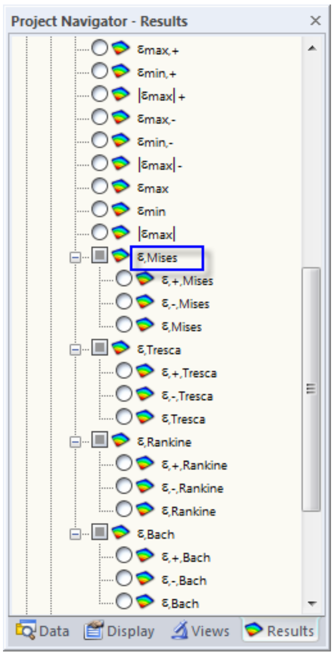
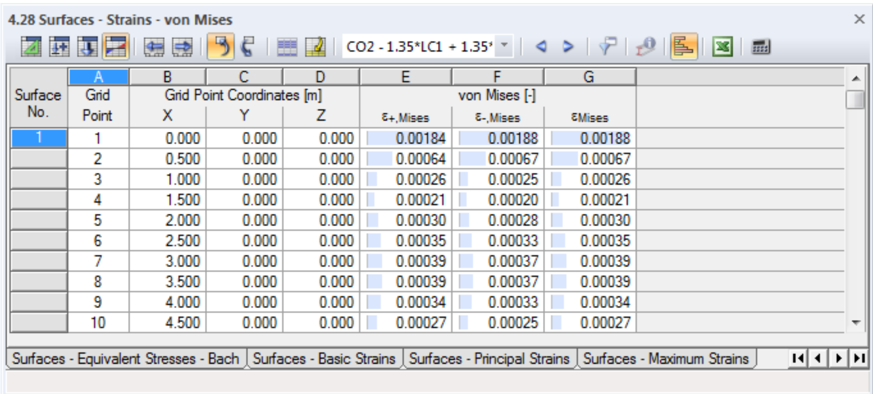


Figure 8.62: Equivalent strains in the *Results* navigator: *Surfaces* → *Strains*


4.28 Surfaces - Strains - von Mises

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E von Mises [-] $\varepsilon_{+,Mises}$	F $\varepsilon_{-,Mises}$	G ε_{Mises}
1	1	0.000	0.000	0.000	0.00184	0.00188	0.00188
	2	0.500	0.000	0.000	0.00064	0.00067	0.00067
	3	1.000	0.000	0.000	0.00026	0.00025	0.00026
	4	1.500	0.000	0.000	0.00021	0.00020	0.00021
	5	2.000	0.000	0.000	0.00030	0.00028	0.00030
	6	2.500	0.000	0.000	0.00035	0.00033	0.00035
	7	3.000	0.000	0.000	0.00039	0.00037	0.00039
	8	3.500	0.000	0.000	0.00039	0.00037	0.00039
	9	4.000	0.000	0.000	0.00034	0.00033	0.00034
	10	4.500	0.000	0.000	0.00027	0.00025	0.00027

Surfaces - Equivalent Stresses - Bach | Surfaces - Basic Strains | Surfaces - Principal Strains | Surfaces - Maximum Strains

Figure 8.63: Table 4.28 *Surfaces - Strains - von Mises*

The table shows the equivalent strains sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns B to D show the coordinates of grid points in the global coordinate system XYZ.

Strains according to von Mises

The approaches for the planar condition of strain described in chapters 8.21 to 8.24 are available for selection in the *Results* navigator. The approach by von Mises is also called "shape modification hypothesis". It is assumed that material fails as soon as the shape modifying energy exceeds a certain limit. This energy is the kind of energy that causes distortion or deformation of the object (see chapter 8.21, page 323).

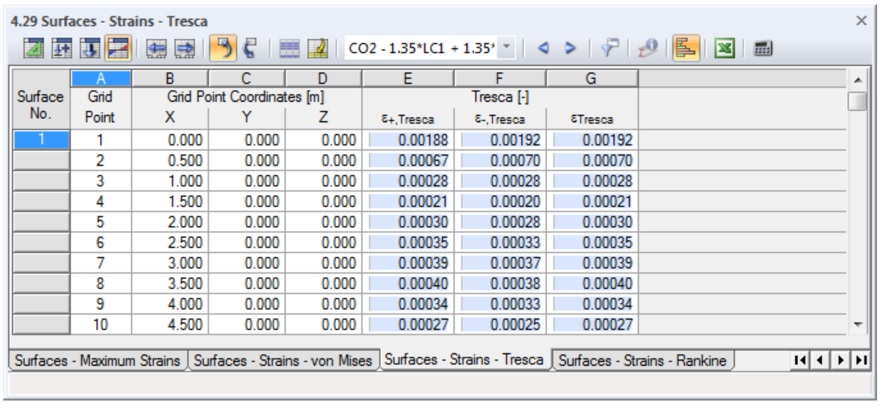
The strains according to von Mises for the plane condition of strain have the following meanings:

$\varepsilon_{+,Mises}$	Equivalent strain on positive side of surface (i.e. side in direction of positive surface axis z) $$\varepsilon_+ = \frac{\sqrt{\left(\varepsilon_{x,+} - \varepsilon_{y,+}\right)^2 + \left(\frac{\varepsilon_{x,+} + \nu \cdot \varepsilon_{y,+}}{1-\nu}\right)^2 + \left(\frac{\nu \cdot \varepsilon_{x,+} + \varepsilon_{y,+}}{1-\nu}\right)^2 + \frac{3}{2}\gamma_{xy,+}{}^2}}{\sqrt{2} \cdot (1+\nu)}$$
$\varepsilon_{-,Mises}$	Equivalent strain on negative side of surface $$\varepsilon_- = \frac{\sqrt{\left(\varepsilon_{x,-} - \varepsilon_{y,-}\right)^2 + \left(\frac{\varepsilon_{x,-} + \nu \cdot \varepsilon_{y,-}}{1-\nu}\right)^2 + \left(\frac{\nu \cdot \varepsilon_{x,-} + \varepsilon_{y,-}}{1-\nu}\right)^2 + \frac{3}{2}\gamma_{xy,-}{}^2}}{\sqrt{2} \cdot (1+\nu)}$$
ε_{Mises}	Maximum equivalent strain on positive or negative side of surface (columns E and F)

Table 8.21: Strains according to von Mises

8.29 Surfaces - Strains - Tresca

To control the graphical display of surface strains available with the equivalent stress hypothesis according to Tresca, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains* (see Figure 8.62, page 333). Table 4.29 shows these strains in numerical form.


4.29 Surfaces - Strains - Tresca

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Tresca [-] $\varepsilon_{+,Tresca}$	F $\varepsilon_{-,Tresca}$	G ε_{Tresca}
1	1	0.000	0.000	0.000	0.00188	0.00192	0.00192
	2	0.500	0.000	0.000	0.00067	0.00070	0.00070
	3	1.000	0.000	0.000	0.00028	0.00028	0.00028
	4	1.500	0.000	0.000	0.00021	0.00020	0.00021
	5	2.000	0.000	0.000	0.00030	0.00028	0.00030
	6	2.500	0.000	0.000	0.00035	0.00033	0.00035
	7	3.000	0.000	0.000	0.00039	0.00037	0.00039
	8	3.500	0.000	0.000	0.00040	0.00038	0.00040
	9	4.000	0.000	0.000	0.00034	0.00033	0.00034
	10	4.500	0.000	0.000	0.00027	0.00025	0.00027

Figure 8.64: Table 4.29 *Surfaces - Strains - Tresca*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.28 *Surfaces - Strains - von Mises*.

With the approach according to Tresca it is assumed that failure is caused by the maximum shear stress (see chapter 8.22, page 325).

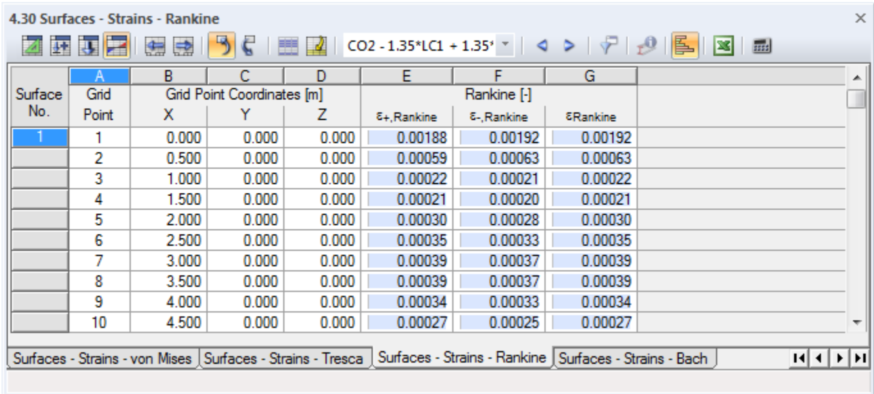
The strains according to TRESCA are determined as follows:

$\varepsilon_{+,Tresca}$	Equivalent strain on positive side of surface (i.e. side in direction of positive surface axis z) $$\varepsilon_{+} = \frac{\sqrt{(\varepsilon_{x,+} - \varepsilon_{y,+})^2 + \gamma_{xy,+}{}^2}}{1+\nu}$$ At the same time, the equivalent strain according to RANKINE is analyzed (see following chapter 8.30). If a larger strain is reached with this hypothesis, this value is shown in table column E.
$\varepsilon_{-,Tresca}$	Equivalent strain on negative side of surface $$\varepsilon_{-} = \frac{\sqrt{(\varepsilon_{x,-} - \varepsilon_{y,-})^2 + \gamma_{xy,-}{}^2}}{1+\nu}$$ If the hypothesis by RANKINE results in a larger equivalent strain, this value is shown in column F.
ε_{Tresca}	Maximum equivalent strain on positive or negative side of surface (columns E and F)

Table 8.22: Strains according to TRESCA

8.30 Surfaces - Strains - Rankine

To control the graphical display of surface strains available with the equivalent stress hypothesis according to RANKINE, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains* (see Figure 8.62, page 333). Table 4.30 shows these strains in numerical form.



Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Rankine [-] $\varepsilon_{+,Rankine}$	F $\varepsilon_{-,Rankine}$	G $\varepsilon_{Rankine}$
1	1	0.000	0.000	0.000	0.00188	0.00192	0.00192
	2	0.500	0.000	0.000	0.00059	0.00063	0.00063
	3	1.000	0.000	0.000	0.00022	0.00021	0.00022
	4	1.500	0.000	0.000	0.00021	0.00020	0.00021
	5	2.000	0.000	0.000	0.00030	0.00028	0.00030
	6	2.500	0.000	0.000	0.00035	0.00033	0.00035
	7	3.000	0.000	0.000	0.00039	0.00037	0.00039
	8	3.500	0.000	0.000	0.00039	0.00037	0.00039
	9	4.000	0.000	0.000	0.00034	0.00033	0.00034
	10	4.500	0.000	0.000	0.00027	0.00025	0.00027

Figure 8.65: Table 4.30 *Surfaces - Strains - Rankine*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the results table 4.28 *Surfaces - Strains - von Mises*.

With the approach according to RANKINE it is assumed that failure is caused by the maximum principal stress (see chapter 8.23, page 326).

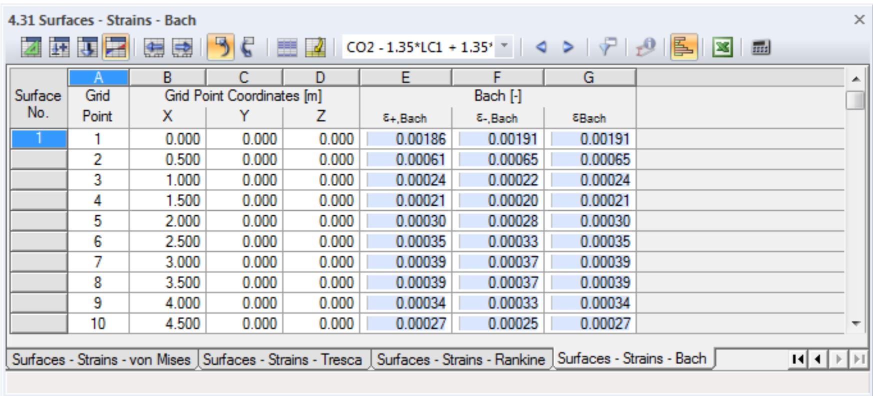
The strains according to RANKINE are determined as follows:

$\varepsilon_{+,\text{Rankine}}$	Equivalent strain on positive side of surface (i.e. side in direction of positive surface axis z) $$\varepsilon_{+} = \frac{1}{2}\left(\frac{\left|\varepsilon_{x,+} + \varepsilon_{y,+}\right|}{1-\nu} + \frac{\sqrt{\left(\varepsilon_{x,+} - \varepsilon_{y,+}\right)^2 + \gamma_{xy,+}^{\;2}}}{1+\nu}\right)$$
$\varepsilon_{-,\text{Rankine}}$	Equivalent strain on negative side of surface $$\varepsilon_{-} = \frac{1}{2}\left(\frac{\left|\varepsilon_{x,-} + \varepsilon_{y,-}\right|}{1-\nu} + \frac{\sqrt{\left(\varepsilon_{x,-} - \varepsilon_{y,-}\right)^2 + \gamma_{xy,-}^{\;2}}}{1+\nu}\right)$$
$\varepsilon_{\text{Rankine}}$	Maximum equivalent strain on positive or negative side of surface (columns E and F)

Table 8.23: Strains according to RANKINE

8.31 Surfaces - Strains - Bach

To control the graphical display of surface strains available with the equivalent stress hypothesis according to BACH, tick the check box for *Surfaces* in the *Results* navigator, and then select *Strains* (see Figure 8.62, page 333). Table 4.31 shows these strains in numerical form.



4.31 Surfaces - Strains - Bach

CO2 - 1.35*LC1 + 1.35*

Surface No.	A Grid Point	B Grid Point Coordinates [m] X	C Y	D Z	E Bach [-] $\varepsilon_{+,Bach}$	F $\varepsilon_{-,Bach}$	G ε_{Bach}
1	1	0.000	0.000	0.000	0.00186	0.00191	0.00191
	2	0.500	0.000	0.000	0.00061	0.00065	0.00065
	3	1.000	0.000	0.000	0.00024	0.00022	0.00024
	4	1.500	0.000	0.000	0.00021	0.00020	0.00021
	5	2.000	0.000	0.000	0.00030	0.00028	0.00030
	6	2.500	0.000	0.000	0.00035	0.00033	0.00035
	7	3.000	0.000	0.000	0.00039	0.00037	0.00039
	8	3.500	0.000	0.000	0.00039	0.00037	0.00039
	9	4.000	0.000	0.000	0.00034	0.00033	0.00034
	10	4.500	0.000	0.000	0.00027	0.00025	0.00027

Surfaces - Strains - von Mises | Surfaces - Strains - Tresca | Surfaces - Strains - Rankine | Surfaces - Strains - Bach

Figure 8.66: Table 4.31 *Surfaces - Strains - Bach*

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the results table 4.28 *Surfaces - Strains - von Mises*.

With the approach according to BACH it is assumed that failure occurs in direction of the maximum strain (see chapter 8.24, page 327).

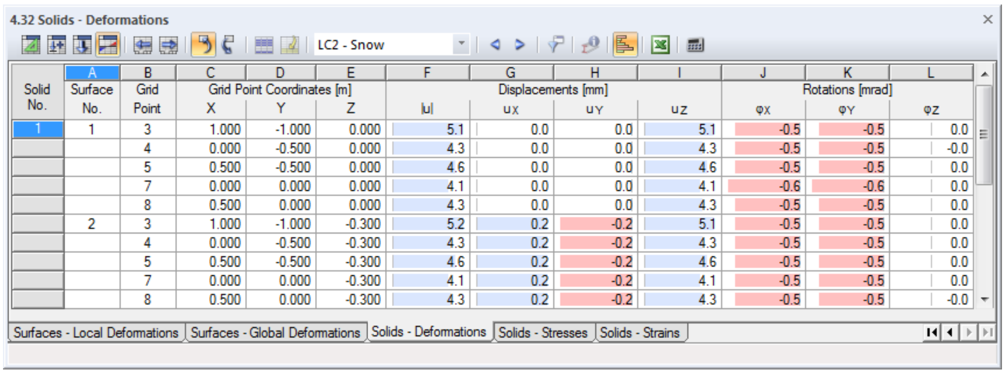
The strains according to BACH are determined as follows:

$\varepsilon_{+,\text{Bach}}$	Maximum absolute value of principal strain ε_{1+} or ε_{2+} on positive side of surface (see chapter 8.26, page 331).
$\varepsilon_{-,\text{Bach}}$	Maximum absolute value of principal strain ε_{1-} or ε_{2-} on negative side of surface (i.e. side in opposite direction of positive surface axis z)
$\varepsilon_{\text{Bach}}$	Maximum equivalent strain on positive or negative side of surface (columns E and F)

Table 8.24: Strains according to BACH

8.32 Solids - Deformations

To control the graphical display of the deformations for solids, tick the check box for *Global Deformations* in the *Results* navigator, (see Figure 8.36, page 306). Table 4.32 shows the deformations for the boundary surfaces of solids in numerical form.



	A	B	C	D	E	F	G	H	I	J	K	L
Solid No.	Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Displacements [mm] |u|	u_X	u_Y	u_Z	Rotations [mrad] φ_X	φ_Y	φ_Z
1	1	3	1.000	-1.000	0.000	5.1	0.0	0.0	5.1	-0.5	-0.5	0.0
		4	0.000	-0.500	0.000	4.3	0.0	0.0	4.3	-0.5	-0.5	-0.0
		5	0.500	-0.500	0.000	4.6	0.0	0.0	4.6	-0.5	-0.5	0.0
		7	0.000	0.000	0.000	4.1	0.0	0.0	4.1	-0.6	-0.6	0.0
		8	0.500	0.000	0.000	4.3	0.0	0.0	4.3	-0.5	-0.5	0.0
	2	3	1.000	-1.000	-0.300	5.2	0.2	-0.2	5.1	-0.5	-0.5	0.0
		4	0.000	-0.500	-0.300	4.3	0.2	-0.2	4.3	-0.5	-0.5	0.0
		5	0.500	-0.500	-0.300	4.6	0.2	-0.2	4.6	-0.5	-0.5	0.0
		7	0.000	0.000	-0.300	4.1	0.2	-0.2	4.1	-0.5	-0.5	0.0
		8	0.500	0.000	-0.300	4.3	0.2	-0.2	4.3	-0.5	-0.5	-0.0

Figure 8.67: Table 4.32 *Solids - Deformations*

The table shows displacements and rotations for the grid points of the individual boundary surfaces. Deformations inside the solid are not shown.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns C to E show the coordinates of grid points in the global coordinate system XYZ.

Displacements / Rotations

The deformations have the following meanings:

|u|	Absolute total displacement (not for result combinations)
u_X	Displacement of solid in direction of global axis X
u_Y	Displacement of solid in direction of global axis Y
u_Z	Displacement of solid in direction of global axis Z
φ_X	Rotation of solid about global axis X
φ_Y	Rotation of solid about global axis Y
φ_Z	Rotation of solid about global axis Z

Table 8.25: Solid deformations

8.33 Solids - Stresses

To control the graphical display of solid stresses, tick the check box for *Solids* in the *Results* navigator. Table 4.33 shows the stresses of solids in numerical form.

The results in the table refer to the grid points of the boundary surfaces. This means that the table does not list any stresses available inside the solid. However, stresses within the solid can be represented graphically on the interior FE mesh points: In the *Results* navigator, tick the check box for *Values on Surfaces*, and then select *Settings* and *On FE mesh points*. To display the values specifically, use a clipping plane (see chapter 9.9.2, page 375).

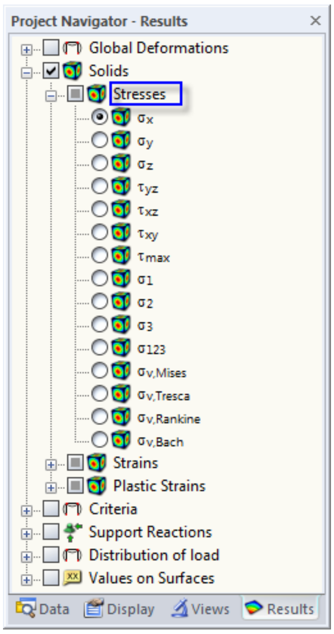
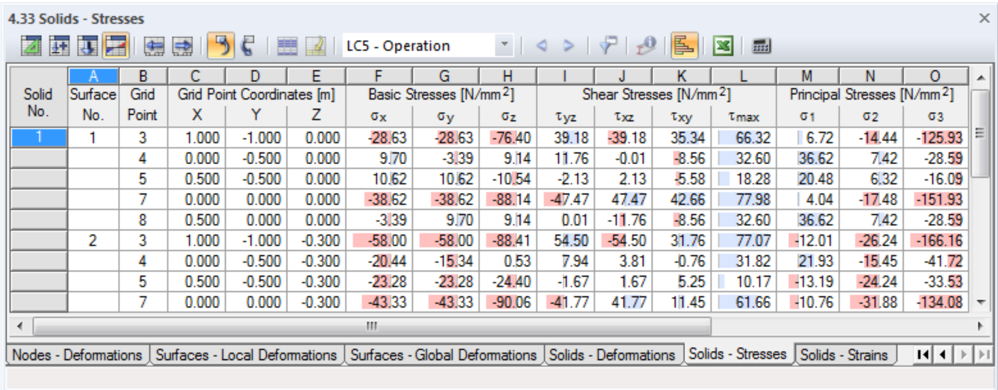


Figure 8.68: *Results* navigator: *Solids* → *Stresses*



4.33 Solids - Stresses

LC5 - Operation

Solid No.	A Surface No.	B Grid Point	C Grid Point Coordinates [m] X	D Y	E Z	F Basic Stresses [N/mm²] σ_x	G σ_y	H σ_z	I Shear Stresses [N/mm²] τ_{yz}	J τ_{xz}	K τ_{xy}	L τ_{max}	M Principal Stresses [N/mm²] σ_1	N σ_2	O σ_3
1	1	3	1.000	-1.000	0.000	-28.63	-28.63	-76.40	39.18	-39.18	35.34	66.32	6.72	-14.44	-125.93
		4	0.000	-0.500	0.000	9.70	-3.39	9.14	11.76	-0.01	-8.56	32.60	36.62	7.42	-28.59
		5	0.500	-0.500	0.000	10.62	10.62	-10.54	-2.13	2.13	-5.58	18.28	20.48	6.32	-16.09
		7	0.000	0.000	0.000	-38.62	-38.62	-88.14	-47.47	47.47	42.66	77.98	4.04	-17.48	-151.93
		8	0.500	0.000	0.000	-3.39	9.70	9.14	0.01	-11.76	-8.56	32.60	36.62	7.42	-28.59
	2	3	1.000	-1.000	-0.300	-58.00	-58.00	-88.41	54.50	-54.50	31.76	77.07	-12.01	-26.24	-166.16
		4	0.000	-0.500	-0.300	-20.44	-15.34	0.53	7.94	3.81	-0.76	31.82	21.93	-15.45	-41.72
		5	0.500	-0.500	-0.300	-23.28	-23.28	-24.40	-1.67	1.67	5.25	10.17	-13.19	-24.24	-33.53
		7	0.000	0.000	-0.300	-43.33	-43.33	-90.06	-41.77	41.77	11.45	61.66	-10.76	-31.88	-134.08

Nodes - Deformations | Surfaces - Local Deformations | Surfaces - Global Deformations | Solids - Deformations | Solids - Stresses | Solids - Strains

Figure 8.69: Table 4.33 *Solids - Stresses*

The table shows the solid stresses sorted by surfaces. The results are listed in reference to the grid points of each surface.

Grid point

The numbers of the grid points are listed by surface. For more information about grid points, see chapter 8.12 on page 304.

Grid point coordinates

Table columns C to E show the coordinates of grid points in the global coordinate system XYZ.

Basic stresses / shear stresses / principal stresses

Unlike surface stresses, solid stresses cannot be described by simple equations. The *Basic Stresses* σ_x, σ_y and σ_z as well as the *Shear Stresses* τ_{xy}, τ_{yz} and τ_{xz} are determined directly by the analysis core.

If a cube with the edge lengths d_x, d_y and d_z is cut from a 3D object with multiaxial loading, the stresses in each cubic surface can be split into normal and shear stresses. If neither spatial force nor stress differences on parallel surfaces are considered, the stress condition in the cube's local coordinate system can be described by nine stress components.

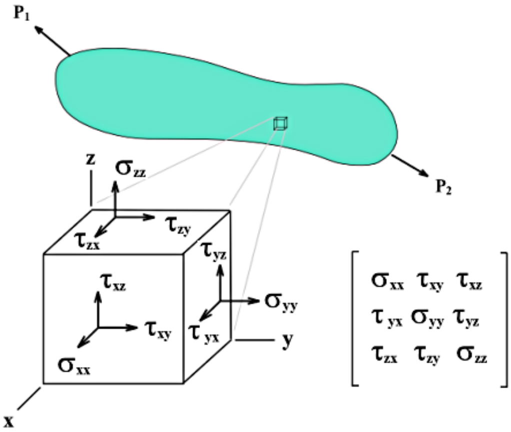


Figure 8.70: Solid element with stress components

The matrix of the stress tensor is the following:

$$S = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix}$$

Equation 8.4: Matrix of stress tensor

The *Principal Stresses* σ_1, σ_2 and σ_3 result from the eigenvalues of the tensor according to the following formula:

$$\det(S - \sigma E) = 0$$

with E: 3x3 unit matrix

Equation 8.5: Principal stresses

The maximum *Shear Stress* τ_{max} is determined according to Mohr's circle:

$$\tau_{max} = \frac{1}{2}(\sigma_1 - \sigma_3)$$

Equation 8.6: Maximum shear stress

The trajectories of the principal stresses can be represented graphically by selecting the navigator entry σ_{123}.

Equivalent stresses

The *Equivalent Stress* $\boldsymbol{\sigma_{eqv}}$ according to VON MISES can be expressed by the following homologous equations:

$$\sigma_{v,Mises} = \sqrt{\frac{1}{2} \cdot \left[(\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_2 - \sigma_3)^2 \right]}$$

Equation 8.7: Equivalent stress from principal stresses according to VON MISES

$$\sigma_{v,Mises} = \sqrt{\sigma_x{}^2 + \sigma_y{}^2 + \sigma_z{}^2 - \sigma_x \sigma_y - \sigma_x \sigma_z - \sigma_y \sigma_z + 3 \cdot \left(\tau_{xy}{}^2 + \tau_{xz}{}^2 + \tau_{yz}{}^2 \right)}$$

Equation 8.8: Equivalent stress from basic stresses according to VON MISES

For the determination of the *Equivalent Stress* $\boldsymbol{\sigma_{eqv}}$ according to TRESCA, RFEM analyzes the differences from the principal stresses to determine the maximum value out of them.

$$\sigma_{v,Tresca} = \max \left(|\sigma_1 - \sigma_2| , |\sigma_2 - \sigma_3| , |\sigma_3 - \sigma_1| \right)$$

Equation 8.9: Determination of equivalent stress according to TRESCA

The *Equivalent Stress* $\boldsymbol{\sigma_{eqv}}$ according to RANKINE is determined from the maximum absolute values of the principal stresses.

$$\sigma_{v,Rankine} = \max \left(|\sigma_1| , |\sigma_2| , |\sigma_3| \right)$$

Equation 8.10: Determination of equivalent stress according to RANKINE

For the determination of the *Equivalent Stress* $\boldsymbol{\sigma_{eqv}}$ according to BACH, RFEM analyzes the principal stress differences, taking the Poisson's ratio ν into account, to determine the maximum value out of them.

$$\sigma_{v,Bach} = \max \left[|\sigma_1 - \nu \cdot (\sigma_2 + \sigma_3)| , |\sigma_2 - \nu \cdot (\sigma_3 + \sigma_1)| , |\sigma_3 - \nu \cdot (\sigma_1 + \sigma_2)| \right]$$

Equation 8.11: Determination of equivalent stress according to BACH

8.34 Solids - Strains

To control the graphical display of solid strains, tick the check box for *Solids* in the *Results* navigator, and then select *Strains*. Table 4.34 shows these strains in numerical form.

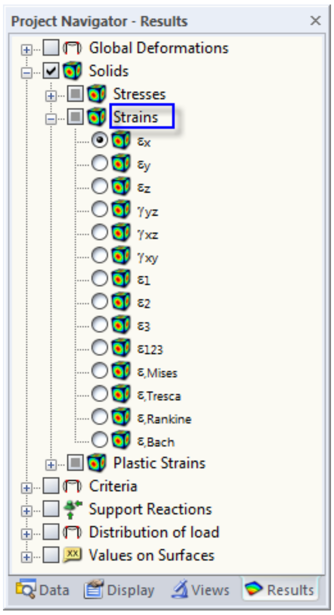
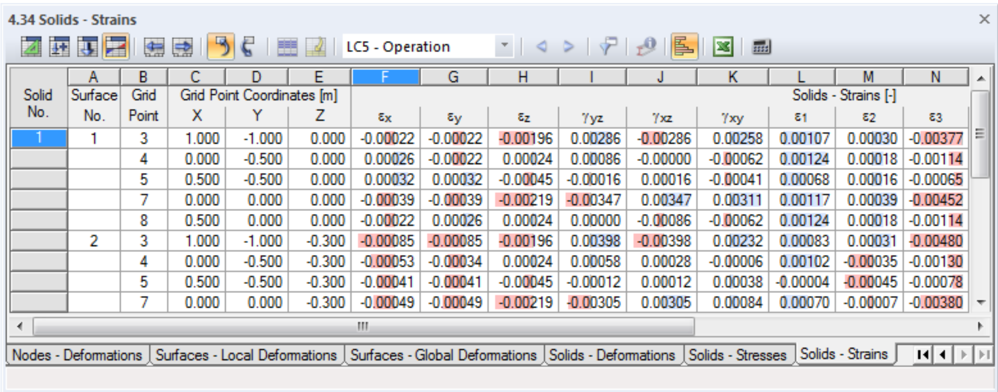


Figure 8.71: *Results* navigator: *Solids* → *Strains*



	A	B	C	D	E	F	G	H	I	J	K	L	M	N
Solid No.	Surface No.	Grid Point	Grid Point Coordinates [m] X	Y	Z	Solids - Strains [-] ε_x	ε_y	ε_z	γ_{yz}	γ_{xz}	γ_{xy}	ε_1	ε_2	ε_3
1	1	3	1.000	-1.000	0.000	-0.00022	-0.00022	-0.00196	0.00286	-0.00286	0.00258	0.00107	0.00030	-0.00377
		4	0.000	-0.500	0.000	0.00026	-0.00022	0.00024	0.00086	-0.00000	-0.00062	0.00124	0.00018	-0.00114
		5	0.500	-0.500	0.000	0.00032	0.00032	-0.00045	-0.00016	0.00016	-0.00041	0.00068	0.00016	-0.00065
		7	0.000	0.000	0.000	-0.00039	-0.00039	-0.00219	-0.00347	0.00347	0.00311	0.00117	0.00039	-0.00452
		8	0.500	0.000	0.000	-0.00022	0.00026	0.00024	0.00000	-0.00086	-0.00062	0.00124	0.00018	-0.00114
	2	3	1.000	-1.000	-0.300	-0.00085	-0.00085	-0.00196	0.00398	-0.00398	0.00232	0.00083	0.00031	-0.00480
		4	0.000	-0.500	-0.300	-0.00053	-0.00034	0.00024	0.00058	0.00028	-0.00006	0.00102	-0.00035	-0.00130
		5	0.500	-0.500	-0.300	-0.00041	-0.00041	-0.00045	-0.00012	0.00012	0.00038	-0.00004	-0.00045	-0.00078
		7	0.000	0.000	-0.300	-0.00049	-0.00049	-0.00219	-0.00305	0.00305	0.00084	0.00070	-0.00007	-0.00380

Figure 8.72: Table 4.34 *Solids - Strains*

The table shows the strains sorted by surfaces. The results are listed in reference to the grid points of each surface enclosing the solid.

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the previous results table 4.33 *Solids - Stresses*.

Solids - strains

The strains are directly determined by the analysis core on the basis of the eigenvalues of the strain matrix. When the model is analyzed according to linear static or second-order analysis, a linear calculation is performed. For a calculation according to large deformation analysis, strains are determined by logarithmic approach.

The equivalent strains are determined according to the four stress hypotheses as follows:

ε_{Mises}	$\varepsilon_{Mises} = \frac{1}{1+\nu}\sqrt{\varepsilon_x{}^2 + \varepsilon_y{}^2 + \varepsilon_z{}^2 - \varepsilon_x\varepsilon_y - \varepsilon_y\varepsilon_z - \varepsilon_z\varepsilon_x + \frac{3}{4}\left(\gamma_{xy}{}^2 + \gamma_{yz}{}^2 + \gamma_{xz}{}^2\right)}$
ε_{Tresca}	Maximum of eigenvalue differences according to matrix R (see Equation 8.12) $\varepsilon_{Tresca} = \max\left(\lvert R_1 - R_2\rvert, \lvert R_2 - R_3\rvert, \lvert R_3 - R_1\rvert\right)$
$\varepsilon_{Rankine}$	Maximum of eigenvalues according to matrix R $\varepsilon_{Rankine} = \max\left(\lvert R_1\rvert, \lvert R_2\rvert, \lvert R_3\rvert\right)$
ε_{Bach}	Maximum of eigenvalue differences by taking into account the Poisson's ratio ν according to matrix R $\varepsilon_{Bach} = \max\left[\lvert R_1 - \nu\cdot(R_2 + R_3)\rvert, \lvert R_2 - \nu\cdot(R_3 + R_1)\rvert, \lvert R_3 - \nu\cdot(R_1 + R_2)\rvert\right]$

Table 8.26: Equivalent strains

$$R = \frac{1}{1+\nu}\cdot\begin{bmatrix} \frac{(1-\nu)\cdot\varepsilon_x + \nu\cdot(\varepsilon_y + \varepsilon_z)}{1-2\nu} & \frac{\gamma_{xy}}{2} & \frac{\gamma_{xz}}{2} \\ \frac{\gamma_{xy}}{2} & \frac{(1-\nu)\cdot\varepsilon_y + \nu\cdot(\varepsilon_x + \varepsilon_z)}{1-2\nu} & \frac{\gamma_{yz}}{2} \\ \frac{\gamma_{xz}}{2} & \frac{\gamma_{yz}}{2} & \frac{(1-\nu)\cdot\varepsilon_z + \nu\cdot(\varepsilon_x + \varepsilon_y)}{1-2\nu} \end{bmatrix}$$

Equation 8.12: Matrix R

8.35 Solids - Gas Pressure

To control the graphical display of the gas pressure, tick the check box for *Solids* in the *Results* navigator, and then select *Stresses* and *Pressure P*. Table 4.35 shows the gas pressure of solids in numerical form.

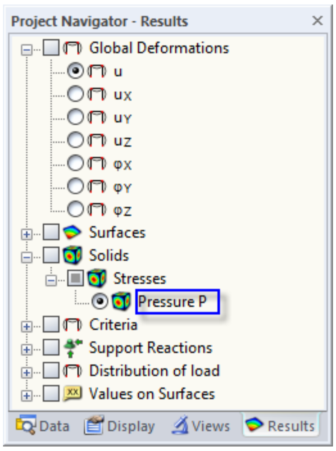
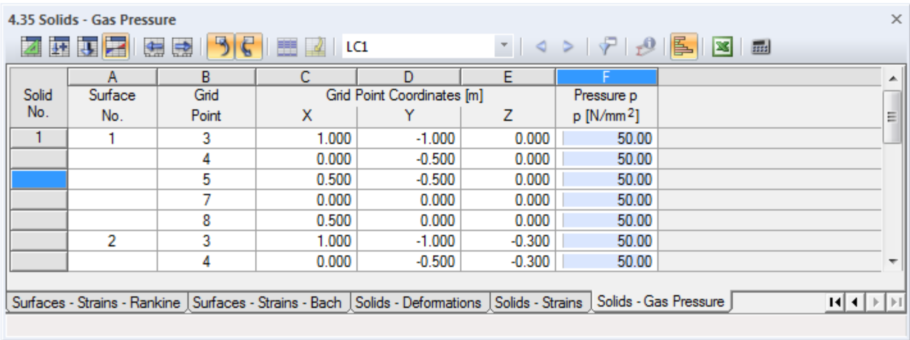


Figure 8.73: *Results* navigator: *Solids* → *Stresses* → *Pressure P*



4.35 Solids - Gas Pressure

Solid No.	A Surface No.	B Grid Point	C Grid Point Coordinates [m] X	D Y	E Z	F Pressure p p [N/mm²]
1	1	3	1.000	-1.000	0.000	50.00
		4	0.000	-0.500	0.000	50.00
		5	0.500	-0.500	0.000	50.00
		7	0.000	0.000	0.000	50.00
		8	0.500	0.000	0.000	50.00
	2	3	1.000	-1.000	-0.300	50.00
		4	0.000	-0.500	-0.300	50.00

Surfaces - Strains - Rankine | Surfaces - Strains - Bach | Solids - Deformations | Solids - Strains | Solids - Gas Pressure

Figure 8.74: Table 4.35 *Solids - Gas Pressure*

The table shows the distribution of pressure sorted by surfaces. The results are listed in reference to the grid points of each surface enclosing the solid.

The table columns *Grid Point* and *Grid Point Coordinates* correspond to the columns of the results table 4.33 *Solids - Stresses.*

Gas pressure p

The gas pressure is a specific type of stress for solids of the type "gas" (see chapter 4.5, page 88). It is determined with the functions of state for volume *V* and temperature *T* according to the following condition:

$$p \cdot \frac{V}{T} = \mathrm{const}$$

where *T* in [K] related to absolute zero point

Equation 8.13: State equation for gases

9. Results Evaluation

9.1 Available Results

To open the dialog box showing the available results,

select **Available Results** on the **Results** menu.

A dialog box with an overview about all calculated load cases and combinations appears.

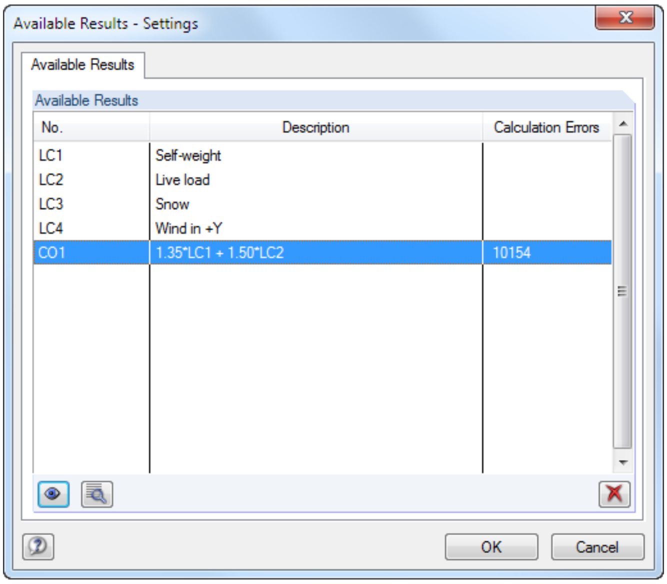
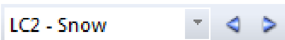


Figure 9.1: Dialog box *Available Results - Settings*

In the list you can see which load cases, load and result combinations were calculated. Any reasons for problems that may have occurred during the calculation process are indicated in the table column *Calculation Errors*. To view error details, select the relevant load case and click the [Details] button shown on the left.

To display a particular result in the graphic, select it in the dialog box and click the [Show] button. You can also double-click the entry. Results that are not required can be deleted by means of the button [X].



It is also possible to select load cases, load or result combinations in the load case list of the main toolbar or the toolbar of the results tables. Results graphic and table display are updated automatically if the synchronization of selection is active (see chapter 11.5.4, page 486).

9.2 Results Selection

Use the *Results* navigator to control the display for deformations, internal forces, stresses, strains and/or support reactions as well as sections and smooth ranges, where applicable.

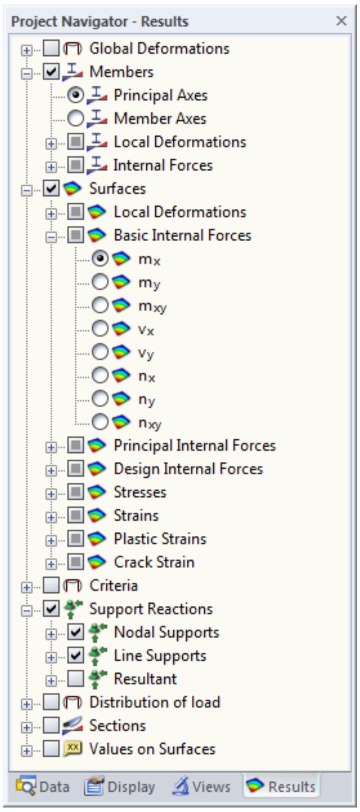


Figure 9.2: *Results* navigator

You can also select results by using the *Results* toolbar.

Figure 9.3: Buttons in the *Results* toolbar

To switch the display for the results graphic on and off, use the toolbar button [Show Results] shown on the left. To display the result values, use the toolbar button [Show Result Values] to the right.

For results of a result combination (RC) the additional entry *Result Combinations* is offered in the navigator.

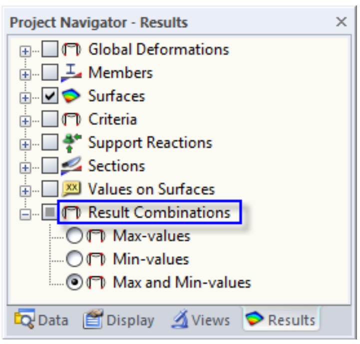


Figure 9.4: *Results* navigator for a result combination

You can choose among three options affecting the graphical results display of deformations, internal and support forces: The *Max-* und *Min-values* can be displayed separately. The option *Max and Min-values* shows both envelopes from all extreme values on the model.

9.3 Results Display

The way results are represented is set in the *Display* navigator.

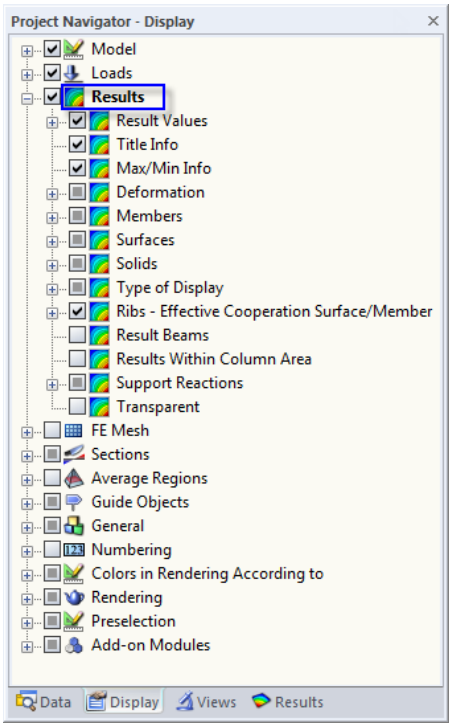


Figure 9.5: *Display* navigator: *Results*

In the *Results* navigator, you specify which results are displayed. The *Display* navigator defines the way how results are represented.

9.3.1 Member Results

The internal forces of members are displayed *Two-Colored* by default. Positive internal forces are represented by light blue lines, negative internal forces by red lines. Member deformations are shown as single-colored *Lines* by default.

The graphical result diagram is controlled by the number entered into the input field *Number of divisions of members for Result diagrams* in the dialog tab *Global Calculation Parameters* of the *Calculation Parameters* dialog box (see Figure 7.22, page 271). If a division of *10* is set, RFEM divides the length of the longest member in the system by 10. With the system-related division length RFEM determines for each member the graphical result distributions on the division points.

If the member internal forces are represented with colors using either the display option *With Diagram* or *Without Diagram*, the colors for the graphical results are assigned according to the spectrum shown in the control panel. Find notes for adjusting value and color spectra in chapter 3.4.6 on page 29.

The internal forces can also be displayed as *Cross-Sections:* A photorealistic representation of members appears showing color-coordinated diagrams of internal forces on the rendered members.

Analogously, you can display the deformation of *Cross-Sections* (3D rendering of deformation) or *Cross-Sections Colored* (rendering of deformation with color gradation).

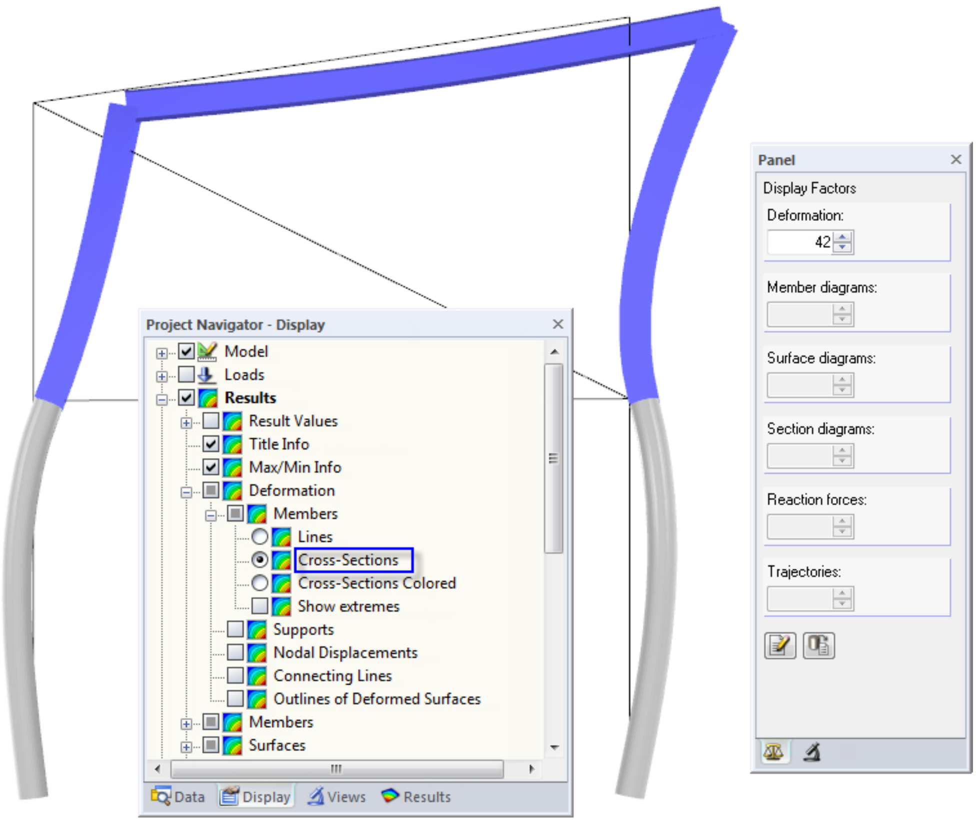


Figure 9.6: Scaled imaging of member deformations in 3D rendering

9.3.2 Surface and Solid Results

The results of surfaces and solids are displayed as *Isobands* by default. The color assignment is managed in the control panel (see chapter 3.4.6, page 29).

Furthermore, the *Display* navigator offers various display options for surface and solid results by selecting *Results* and *Type of Display*.

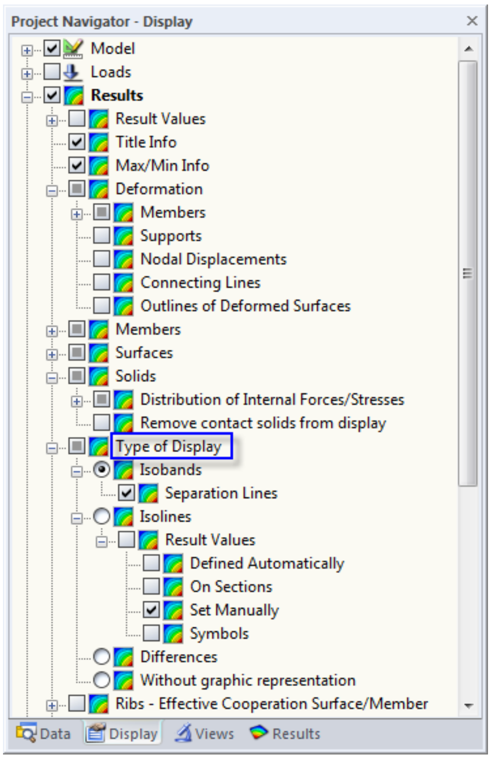


Figure 9.7: *Display* navigator: *Results* → *Type of Display*

The results of surfaces and solids can be displayed as *Isobands* or *Isolines*. Often, isolines prove to be useful for output by a monochrome printer.

The display option *Without graphic representation* allows for an output of pure result values: Isobands or isolines are hidden so that only the result values on the grid or FE mesh points are displayed. This setting is also appropriate for printing.

The display option *Differences* is only available for stresses. With this setting, you can see the stress changes in the finite elements, which makes it possible to draw conclusions concerning the quality of the FE mesh: If there are significant differences in adjacent FE elements, you should think about a FE mesh refinement on these locations.

Stresses in solids

You can select the option *Solid FE Nodes* to evaluate the stresses inside a solid provided that the FE mesh is fine enough. The color assignment of FE nodes follows the spectrum of the control panel. If you additionally activate *Values* in the FE mesh points in the *Results* navigator, you can directly see the solid stresses displayed on the model.

You can control the scaling of deformations and internal forces by settings in the control panel tab *Factors* (middle). The *Filter* tab (right) is used for a specific selection of members, surfaces or solids whose results you want to display (see Figure 9.51, page 378). Both panel tabs are described in chapter 3.4.6, page 31.

Criteria for nonlinear material properties

When a material model with nonlinear effects has been selected (see chapter 4.3, page 61), you can graphically check which areas are affected by reduced stiffnesses for example when reaching the yield strength. Please note that you can use this option only if the add-on module **RF-MAT NL** is licensed.

The results display option *Is nonlinear* shows the portion of GAUSS points that were nonlinearly analyzed at least once during the analysis. With the *Yield Criterion* you can for example find out which yielding zones are arising above the supports of a plate.

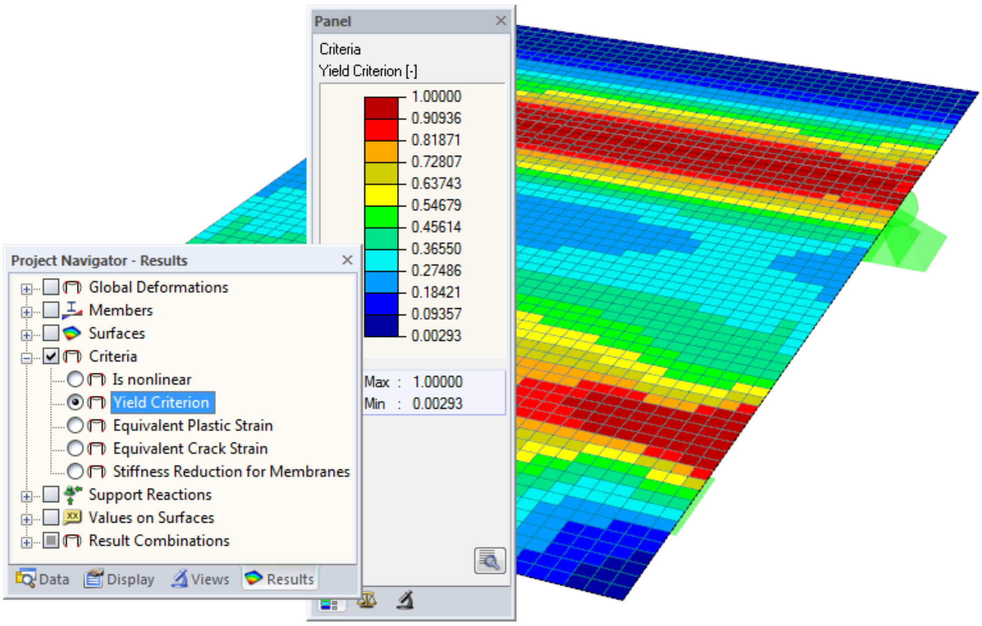


Figure 9.8: Yield criterion of a plate with yielding zones

Load distribution

By ticking the check boxes under the navigator entry *Distribution of load*, you can display the forces and moments received by *FE Nodes* or *FE Elements* from the entered loads. The forces of the finite elements can be represented in relation to the global axis system XYZ or the local surface axes xyz.

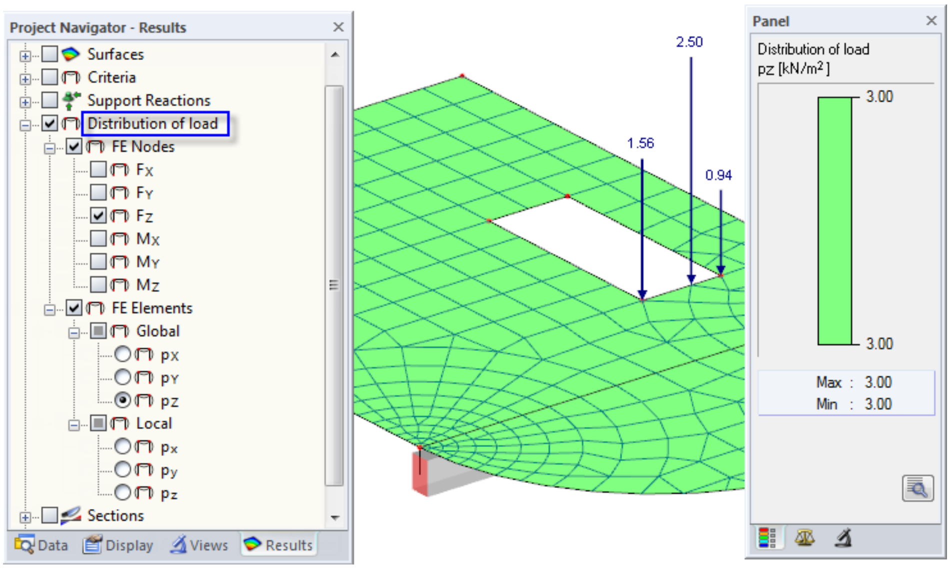


Figure 9.9: Load distribution with mesh loads for line load (F_Z) and surface load (p_Z)

Using mesh loads makes it possible to check the entered loads. For example for free concentrated loads you can check whether the load acts on all surfaces specified in the list *On Surfaces* (see Figure 6.31, page 234).

9.4 Value Display

The display of values is managed in the *Results* navigator (see chapter 3.4.3, page 25).

9.4.1 Result Values

The navigator category *Values* controls the result values displayed in the work window.

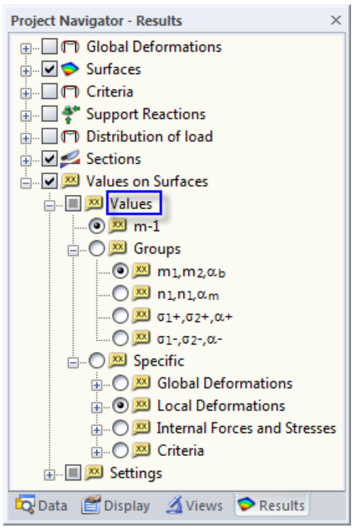


Figure 9.10: *Results* navigator: *Values on Surfaces* → *Values*

Values of results graphic

The first option (*m-1* in the figure above) is aligned with the result type that is displayed in the work window. When the selection field is active, RFEM displays the result values of the current deformation graphic, stress graphic or graphic of internal forces.

Values groups

With the option *Groups* it is possible to show two result values for each location for the surface results. Four groups are preset. The following figure shows the first group with the principal moments m_1 and m_2. The arranged grid values are rotated about the angle α_b.

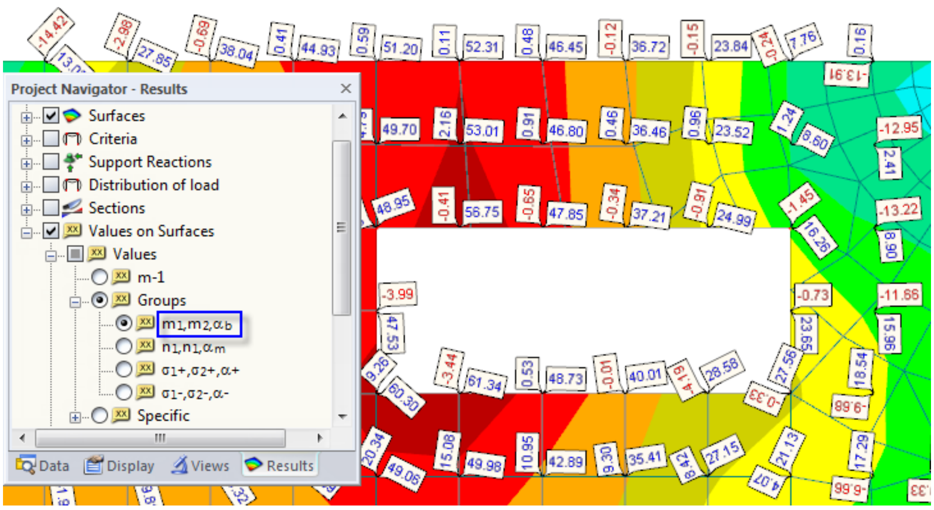
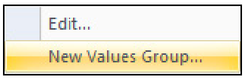


Figure 9.11: Value group of principal moments in the graphic



It is also possible to create user-defined groups of values: Right-click the navigator item *Groups* to open the context menu shown on the left. Select *New Values Group* to open the following dialog box.

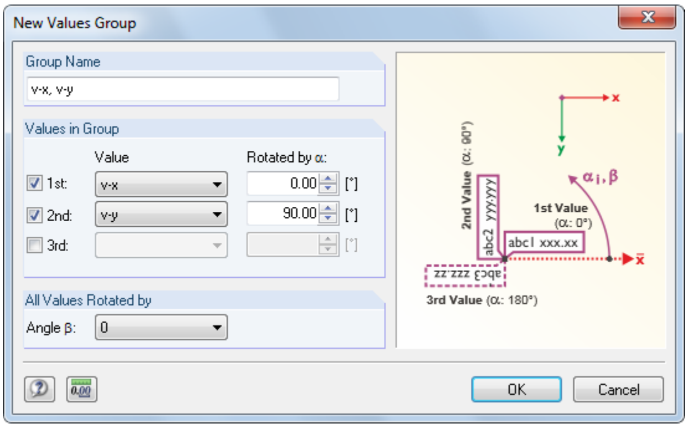
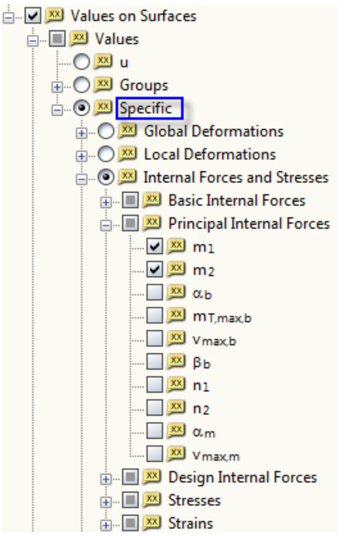


Figure 9.12: Dialog box *New Values Group*

First, define the *Group Name* that appears as item in the navigator later. In the dialog section *Values in Group*, you can select the result types from the lists *1st Value*, *2nd Value* and *3rd Value*. The rotation of the values is specified in the fields *Rotated by* α.



Specific result values

You can use the selection field *Specific* to determine the result values (deformations, internal forces, stresses, strains) you want to display, irrespective of the result type active in the work window. Thus, you can display data simultaneously, for example providing the deformations of a surface graphically and displaying the values of the principal internal forces m_1 and m_2 as shown on the left.

9.4.2 Settings

The display options available under the navigator entry *Settings* control the design locations of the result values and their representation.

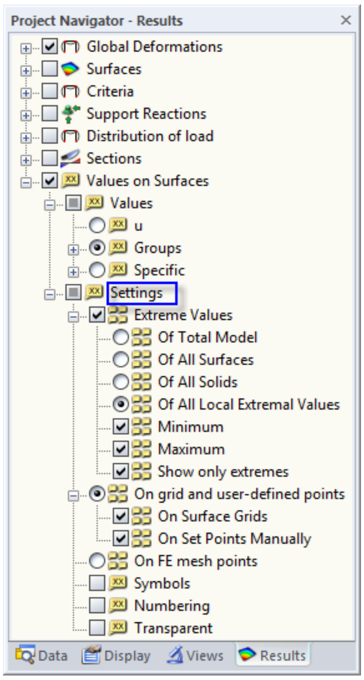


Figure 9.13: *Results* navigator: *Values on Surfaces* → *Settings*

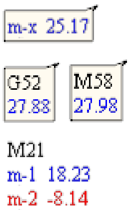
Extreme values

If the option *Extreme Values* is selected, only the respective minima or maxima, depending on the setting, are displayed.

Grid points / FE mesh points

In addition, result values can be displayed either *On grid and user-defined points* or *On FE mesh points*. Be careful with using the last mentioned option because importing all FE result values takes time for larger models.

Symbols / numbering / transparent



The final three check boxes listed under *Settings* control the type and extent of the labeling.

- The *Symbols* of the set result type (u, m_x, σ_z etc.) are shown, too.
- The *Numbering* of **g**rid points or FE **m**esh nodes (*G1*, *M1* etc.) can be displayed additionally.
- The values can be displayed in *Transparent* mode, i.e. without frame and background.

To adjust colors and fonts of the result values,

point to **Display Properties** on the **Options** menu, and then select **Edit**.

A dialog box with global settings for *Display Properties* opens. Select *Result Values* in the *Results* category, and then click *Result Values on Surfaces* to specify your settings.

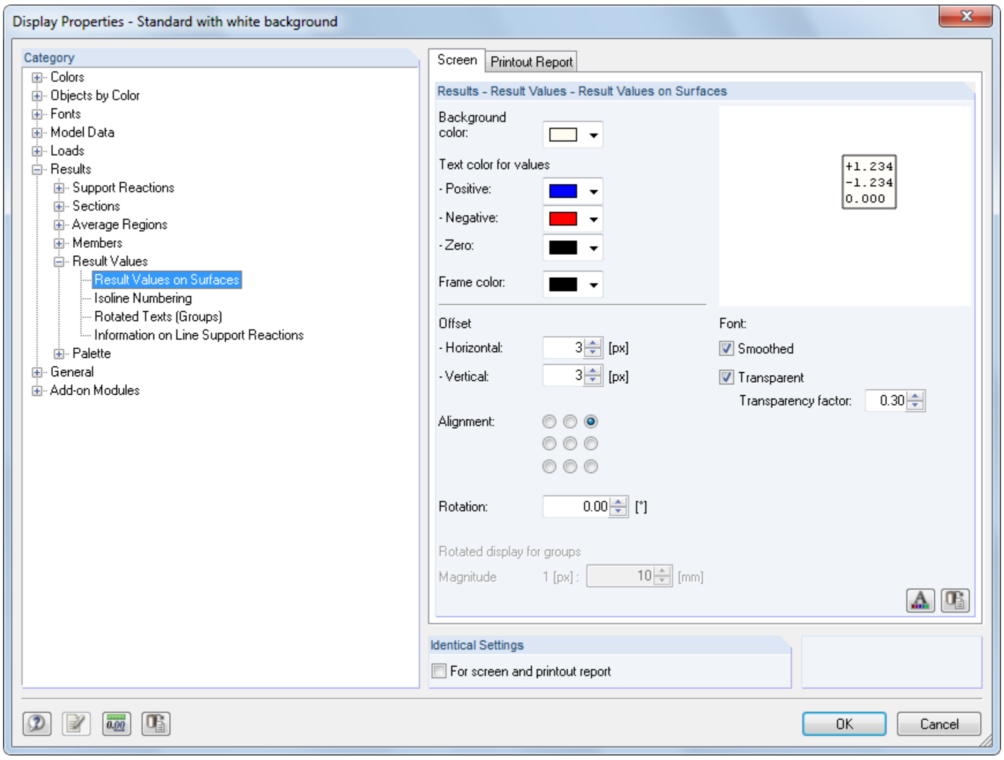


Figure 9.14: Dialog box *Display Properties*: category *Results* → *Result Values* → *Result Values on Surfaces*

9.4.3 User-defined Result Values

Grid values

Grid points represent an attribute of the surface. Therefore, number and arrangement of grid points can be adjusted in the *Grid* tab of the dialog box *Edit Surface*. The results output in tables is based on the result grid for surfaces. In the graphic, both the values of FE nodes as well as of grid points can be displayed.

For more information about grid points, see chapter 8.12 on page 304.

Graphic values

In the work window, you can set result values on any location of the model. If the results display is active, you can access the function in the following way:

Select **Set Result Values Manually** on the **Results** menu

or use the toolbar button shown on the left (see Figure 9.15).

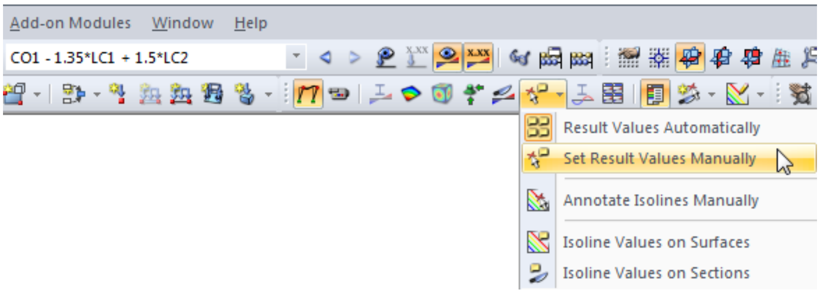


Figure 9.15: Function *Set Result Values Manually* in the *Results* toolbar

When you move the mouse across a surface, result values appear for the current position of the mouse pointer. Now, the result values can be placed by mouse click on the relevant locations.

A manually placed value can be deleted easily: Select the value by mouse click, and then press the [Del] key on your keyboard. For a multiple selection, keep the [Ctrl] key pressed or draw a window across the values that you want to select.

To access the context menu of result values, right-click one of the values. The menu contains specific display and filter functions for the graphical evaluation.

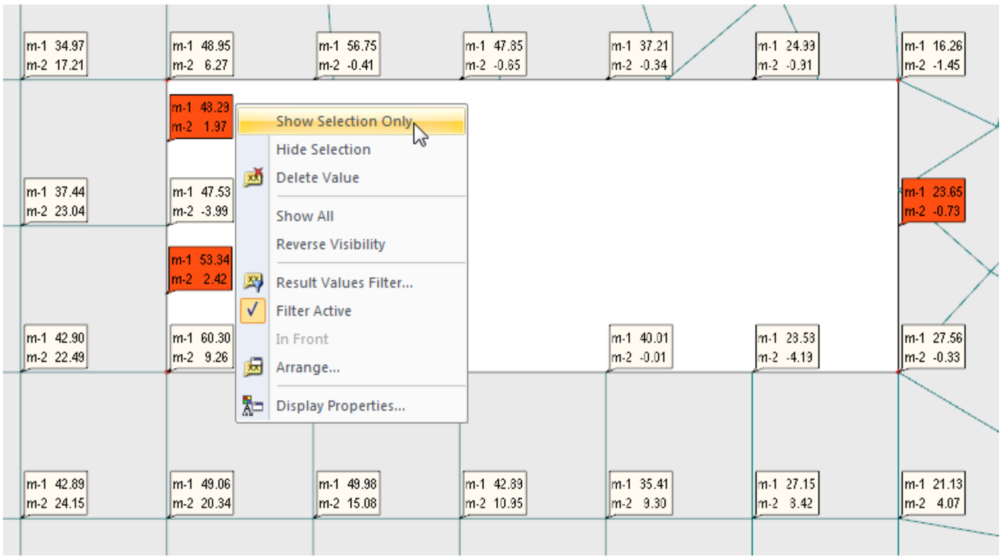
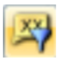


Figure 9.16: Context menu of result values



With the context menu function *Result Values Filter* (see figure above) you can define precise specifications for the result values to be displayed. To open the corresponding dialog box,

point to **Display Options** on the **Results** menu

where you select the corresponding entry. The following dialog box for entering filter criteria appears.

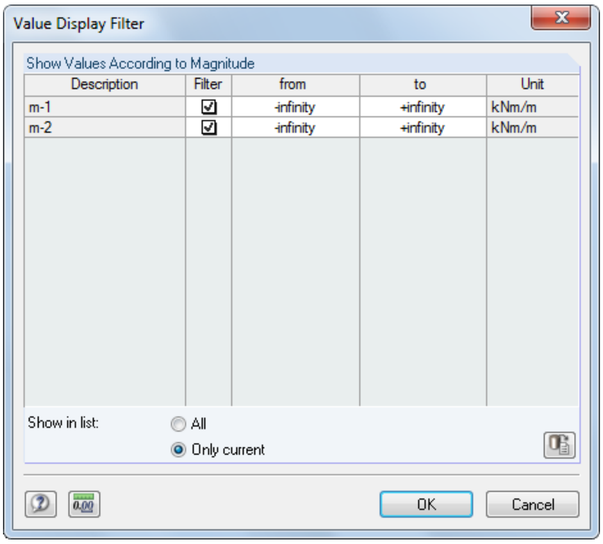


Figure 9.17: Dialog box *Value Display Filter*

In the list *Show Values According to Magnitude*, you can define limits for the result values in the table columns *from* and *to*. Values that are beyond these ranges will not be shown in the graphics later.

Search criteria for local extreme values

To control the output of graphical extreme values for surfaces,

point to **Display Options** on the **Results** menu, and then select **Search Criteria for Local Extreme Values**

or use the context menu of the *Results* navigator entry *Of All Local Extremal Values*.

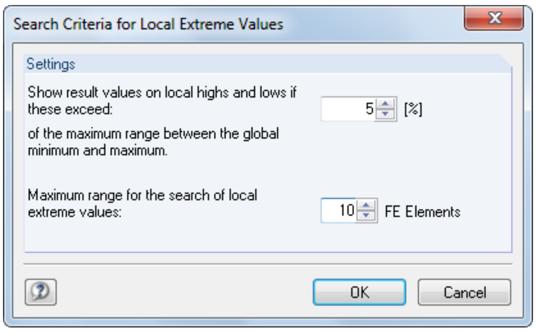


Figure 9.18: Dialog box *Search Criteria for Local Extreme Values*

In the first input field, you specify the percentage by which a result value is considered to be a local peak. Then, the difference from global maximum and global minimum of all active surfaces is multiplied by the specified value. The lower the threshold is, the more local extreme values will be displayed.

In the second input field, you can define how many finite elements generated around a point you want to apply for the analysis of extreme values. The higher the number is, the more local extreme values will be displayed.

9.4.4 Object Info

For member and surface results you can access a special output function. To open the corresponding dialog box,

select **Info About Object** on the **Tools** menu

or use the toolbar button shown on the left.

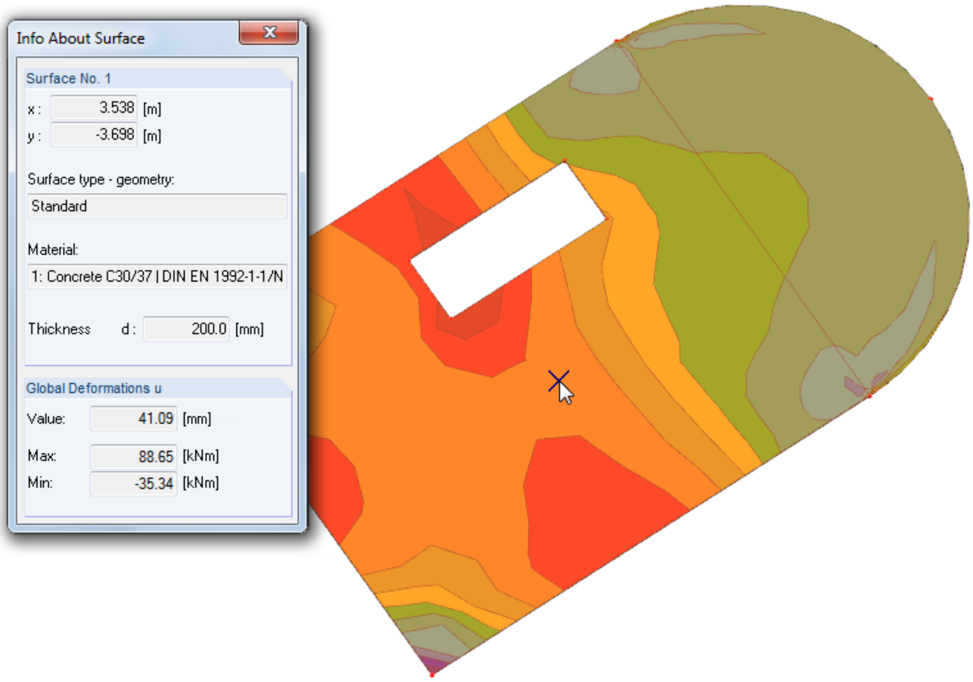
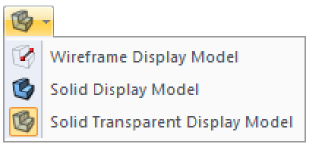


Figure 9.19: Dialog box *Info About Surface*

An *Info* window appears. When you move the pointer across a surface, a member or a solid, the window informs you about the object data (material, thickness, cross-section etc.) as well as deformation values, internal forces or stresses on the current pointer position.



For evaluating surface results we recommend to activate the representation type *Solid Transparent*.

9.5 Result Diagrams

The result diagram makes it possible to see the result distribution of objects in detail:

- Section
- Member
- Set of members
- Line
- Line support

First, select the object(s) in the work window (multiple selection by holding down the [Ctrl] key). Then, to access the corresponding function,

select **Result Diagrams for Selected Sections/Members/Sets of Members/Lines/Line Supports** on the **Results** menu

or use the context menu of the corresponding object. For members and sets of members, the toolbar button shown on the left is additionally available.

A new window opens showing the result diagrams of the selected object.

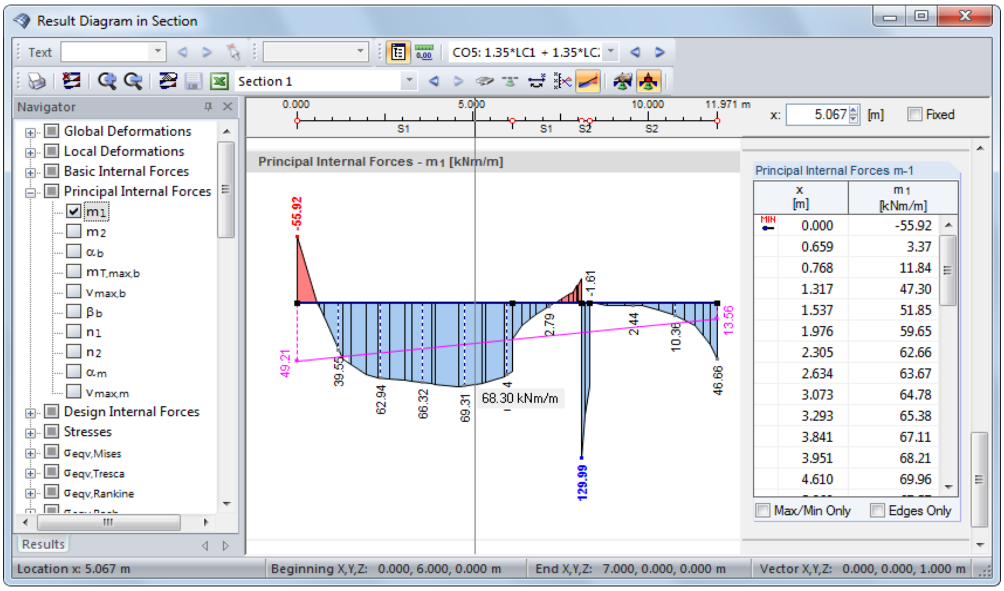
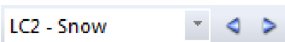
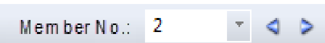


Figure 9.20: Dialog box *Result Diagram in Section*



The *Results* navigator on the left manages the deformations, internal forces, stresses, strains or support forces appearing in the result diagram. Use the list in the toolbar to choose a particular load case, load combination or result combination.



In the result diagrams for members, the numbers of the selected members are listed in the upper left corner of the window. It is also possible to enter member numbers manually into the input field *Member No.* In this way, you can extend, reduce or completely reorganize the selection.

When you move the mouse along the line or member selected in the result diagram, you can see the "moving" result values for the current x-location. The location *x* is related to the line or member start and is indicated in the upper right corner of the window. It is also possible to enter a specific location x manually into the input field. The check box *Fixed* pins the pointer to the indicated location.

In the right window section, the result values are listed numerically, representing results on edge nodes as well as on locations of the extreme values and division points. The latter correspond to the FE mesh nodes or member divisions according to specifications set in the tab *Global Calculation Parameters* of the *Calculation Parameters* dialog box (see Figure 7.22, page 271). In case of lines separating two surfaces from each other, the window shows the surface results of both sides.

The buttons in the toolbar *User operations*, in particular the smoothing options for support forces, lines or sections, help you to evaluate results for civil engineering purposes.

Figure 9.21: Floating toolbar *User operations*

The buttons have the following meanings:

Button	**Function**
	Prints result diagrams
	Removes all displayed result diagrams
	Maximizes result diagrams
	Minimizes result diagrams
	Access to control parameters shown in Figure 9.22
	Saves smoothed result diagrams
	Opens dialog box *Export table* (see Figure 11.126, page 489)
	Shows member results with or without rib components
	Switches on and off result diagram over column area
	Reverses direction x of member
	Switches on and off ordinates with maximum values
	Switches on and off display of average values
	Opens dialog box for defining smooth ranges (see Figure 9.36, page 366)
	Switches on and off display of smooth ranges

Table 9.1: Buttons of toolbar *User operations*

Use the button [Result Diagram Settings] to open a dialog box offering several options to adjust the *Result Diagram* window.

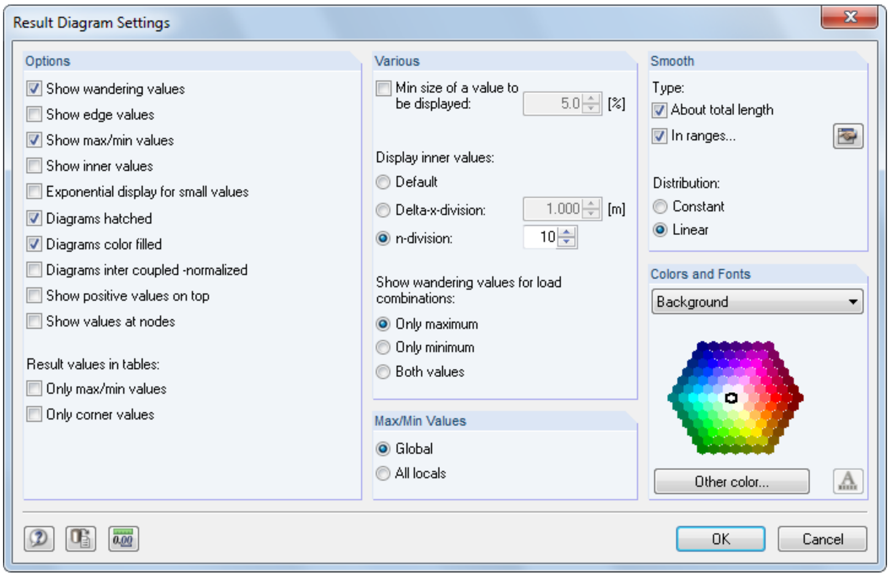


Figure 9.22: Dialog box *Result Diagram Settings*

9.6 Sections

You can create user-defined sections in RFEM by defining a plane slicing through the model. With the help of sections it is possible to evaluate in detail results that are available on the plane lines intersecting surfaces and solids. Sections are managed as independent objects in the tabs of the Project Navigator.

To generate a new section,

select **Section** on the **Insert** menu

or use the corresponding context menu in the *Data* navigator.

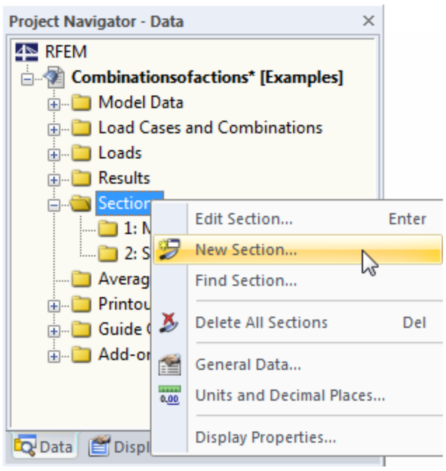


Figure 9.23: *Data* navigator: context menu of *Sections*

A dialog box opens where you can define the section parameters.

9.6.1 Section Through Surface

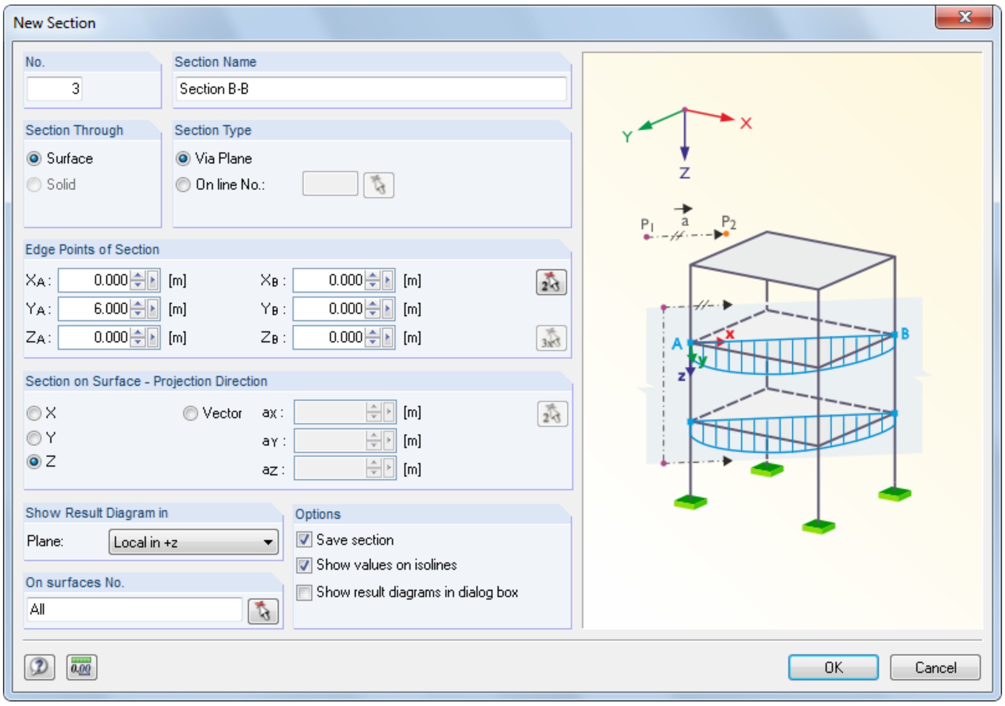


Figure 9.24: Dialog box *New Section* for surface

In addition to the *No.* of the section, you have to enter the *Section Name* that ensures a reliable assignment when evaluating results. All sections are stored under *Sections* in the *Data* navigator where they can be adjusted subsequently. Entering number and name is unnecessary if the check box *Save section* is cleared in the dialog section *Options*.

When you create a *Section Through* a *Surface*, you can see the surface-specific parameters displayed in the dialog sections *Section Type* and *Projection Direction* as well as the corresponding graphics to the right.

The *Section Type* can be created as *Plane* slicing the model. As an alternative, you can cut with the section along any *line* in the model. The line number can be entered manually or determined with the [↖] function in the work window.

The *Edge Points of Section* must be specified by the global coordinates XYZ of both points A and B. You can also select them graphically by using the [↖] function. To select free points (which means no nodes) in the work plane, adjust the current work plane, where necessary.

Starting from points A and B, two straight lines are "drawn" in the projection direction. If the lines intersect a surface contained in the list *On surfaces No.*, the result diagram will be displayed along the connecting line of both intersection points. In case several surfaces are cut by the projection plane, result diagrams will be displayed for each of these surfaces.

In addition to the global *Projection Direction*s in X, Y and Z, it is possible to define a vector. With the [↖] function you can select two points in the work window to define the vector.

The dialog section *Show Result Diagram in Plane* defines the surface plane in which the section will be represented. The setting affects only the work window (see Figure 9.27, page 361) but not the dialog box *Result Diagram* (Figure 9.25).

The dialog section *On surfaces No.* lists the numbers of surfaces on which the cutting lines are displayed. This option is useful if the section plane intersects with several surfaces. You can select relevant surfaces also graphically by using the [↖] function.

Use the three check boxes in the dialog section *Options* to decide whether the result diagrams are displayed as a dialog box (Figure 9.25) after clicking [OK] and if you want to *Save* the *section*. When you tick the check box for *Show values on isolines*, isolines will be labeled automatically in the work window.

When the dialog input is complete, click [OK]. Usually, the *Result Diagram* dialog box appear (see following figure).

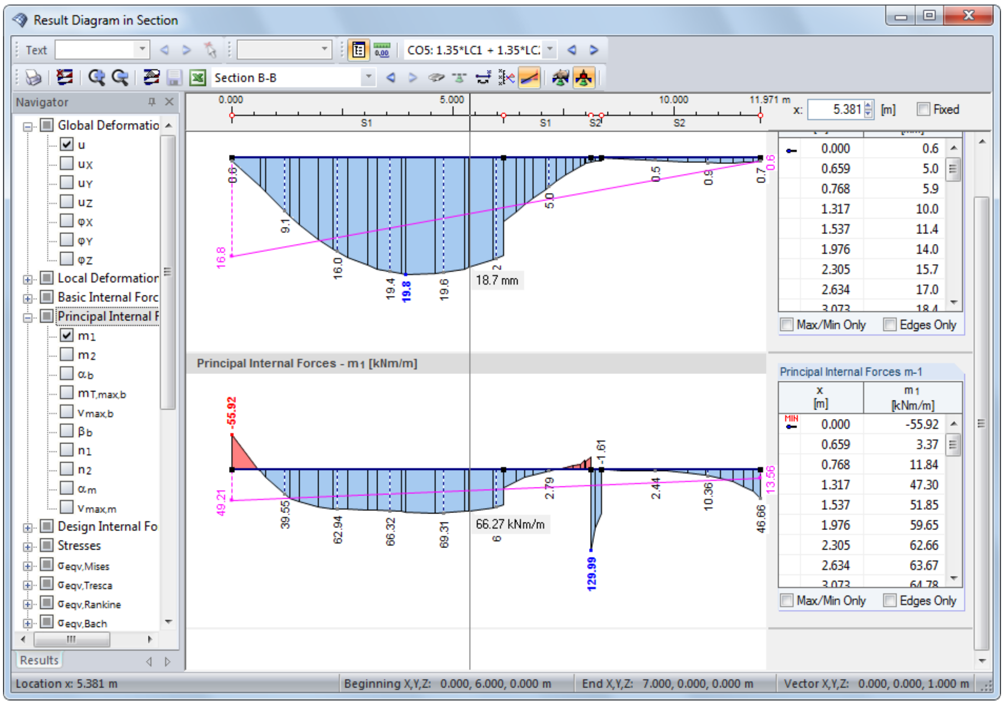


Figure 9.25: Dialog box *Result Diagram in Section*

When you move the mouse in the diagram along the section, you can see the "moving" result values for the current x-location. The location *x* is related to the section start A and indicated in the upper right corner of the window. It is also possible to enter a specific location x manually into the input field. The check box *Fixed* pins the pointer to the indicated location.

Use the list in the toolbar to switch between the individual sections.

The buttons of the dialog box *Result Diagram* are described in chapter 9.5 on page 357.

With the button shown on the left, you can switch on and off the sections in the work window. You can also use the *Results* navigator which additionally allows you to select specific sections among the sections that have been saved.

The *Display* navigator offers global setting options for representing sections.

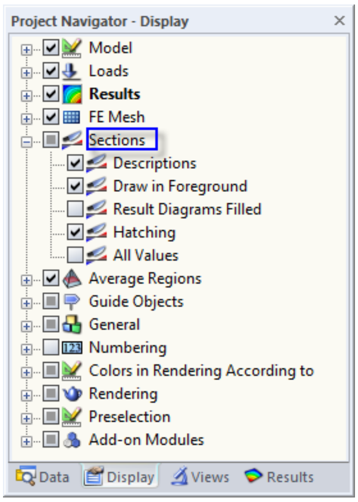


Figure 9.26: *Display* navigator for representation of sections

The following figure shows a section through a plane and a curved surface that are both cut by the section plane. For this graphic display, the *Sections* option *Result Diagrams Filled* has been ticked in the *Display* navigator.

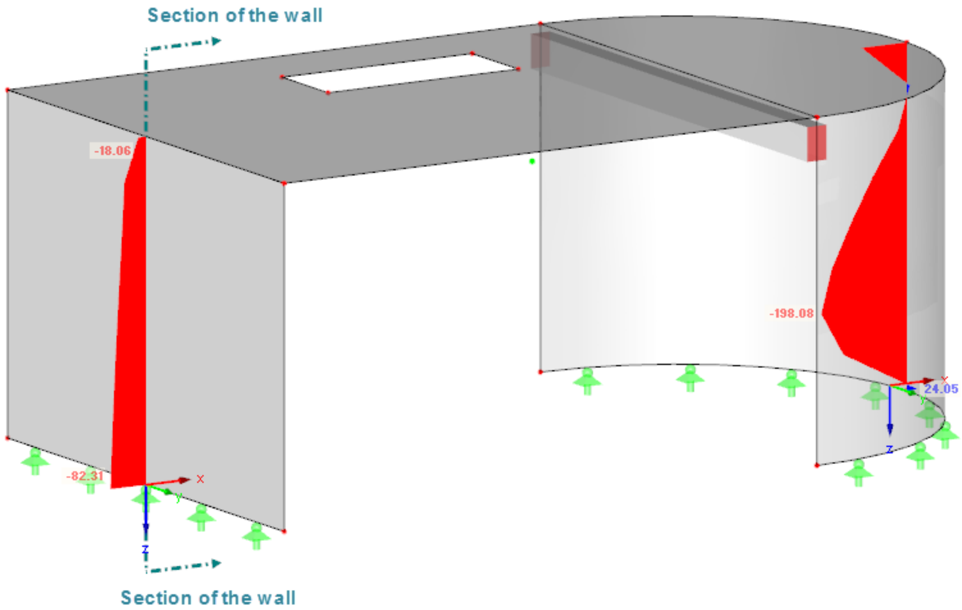


Figure 9.27: Section display of axial forces n-2 on RFEM model

9.6.2 Section Through Solid

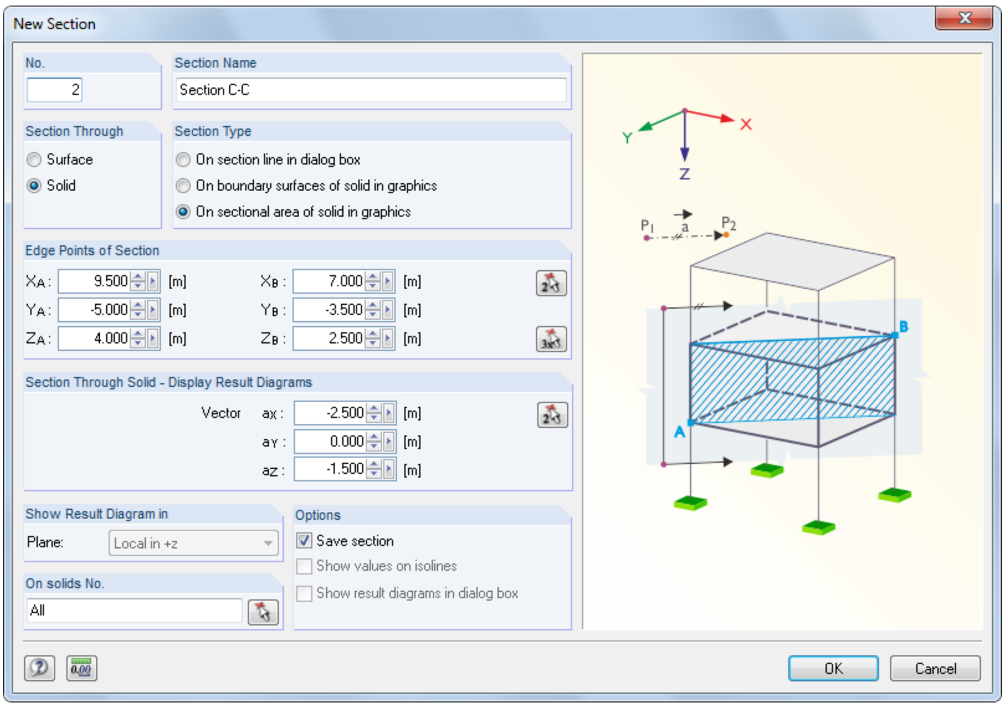


Figure 9.28: Dialog box *New Section* for solid

Similar to a section through surfaces, enter *No.* and *Section Name* if you want to *Save* the *section* (see dialog section *Options*).

When you create a *Section Through* a *Solid*, you can see the solid-specific parameters displayed in the dialog sections *Section Type* and *Display Result Diagrams* as well as the corresponding graphics to the right.

When the section cuts through a solid, you can generate a *section line* running through the object. Then, results are displayed like for surfaces in the dialog box *Result Diagram* (see Figure 9.25, page 360). Alternatively, results can be represented in the work window *On* the *boundary surfaces of solid* intersected by the plane. The option *On sectional area of solid in graphics* shows the results directly in the section plane.

The *Edge Points of Section* must be entered as described for surfaces, but you can select them also graphically.

Starting from points A and B, two straight lines are "drawn" in direction of the *Vector*. If the lines intersect a solid contained in the list *On solids No.*, results will be displayed along the connecting line of both intersection points, respectively as section plane between the straight lines. In case several solids are cut by the projection plane, result diagrams will be displayed for each of these solids.

The *Vector* defines the projection direction of the section. With the [↖] function you can select two points in the work window to define the vector.

9.7 Smoothing Results

The FE analysis determines the results for each FE mesh node. Usually, a continuous distribution of the internal force or deformation is preferable for the graphic. For this purpose, it is necessary to smooth the results, for example by interpolation.

The following smoothing options are available for surfaces and solids:

- Constant on elements
- Non continuous
- Continuous within surfaces/solids
- Continuous total

In addition, it is possible to define smooth ranges for result diagrams (see Figure 9.36, page 366).

9.7.1 Work Window

The *Display* navigator controls the results smoothing influencing the work window.

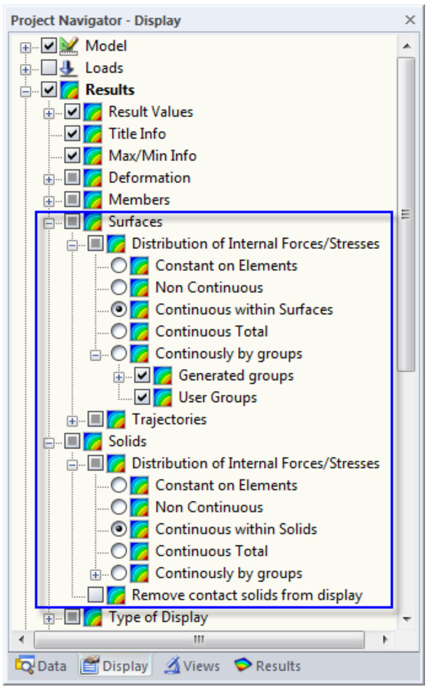


Figure 9.29: *Display* navigator: *Results* → *Surfaces* or *Solids* → *Distribution of Internal Forces/Stresses*

Example

An example shows you the effects of the different smoothing options.

A steel plate with the dimensions 3 m · 3 m and a thickness of 3 cm is supported on two lines with pinned supports facing each other. The plate is not modeled by one complete surface but by two surfaces with the same properties laying side by side. The local z-axes of both surfaces are orientated in opposite directions.

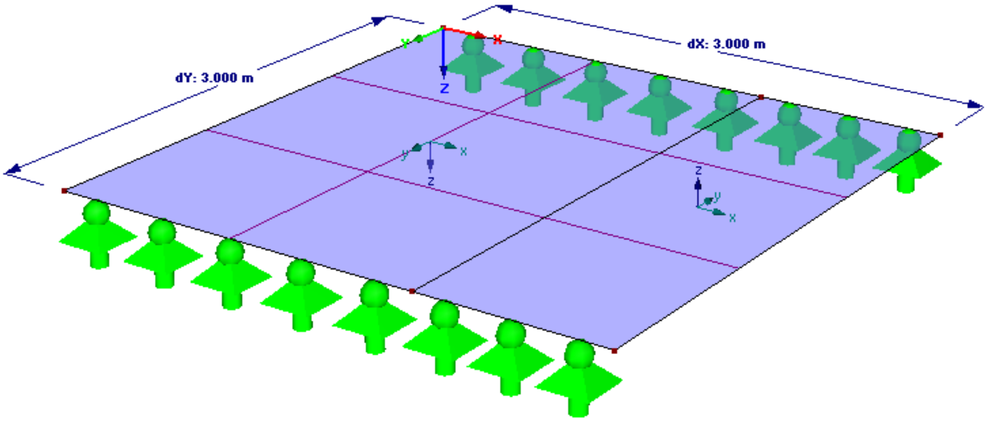


Figure 9.30: Steel plate – modeled with two surfaces

The size of the FE length has been generously set to 1 m. This element size cannot provide appropriate results. It is only used to demonstrate the result representations of the different smoothing modes.

The steel surface is stressed only by its self-weight.

Distribution of internal forces *Constant on Elements*

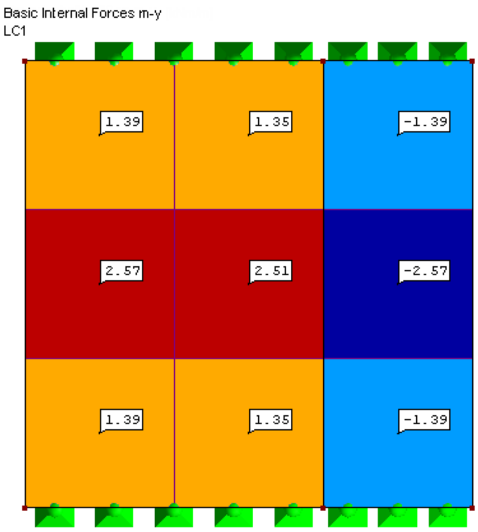


Figure 9.31: Distribution of internal forces *Constant on Elements* (*Display* nav.), values *On FE mesh points* (*Results* nav.)

The values of the FE nodes are averaged and the result is displayed in the centre of each element. The distribution in each finite element is constant. This type of results display is recommended for plastic material models (see chapter 4.3, page 61).

Distribution of internal forces *Non Continuous*

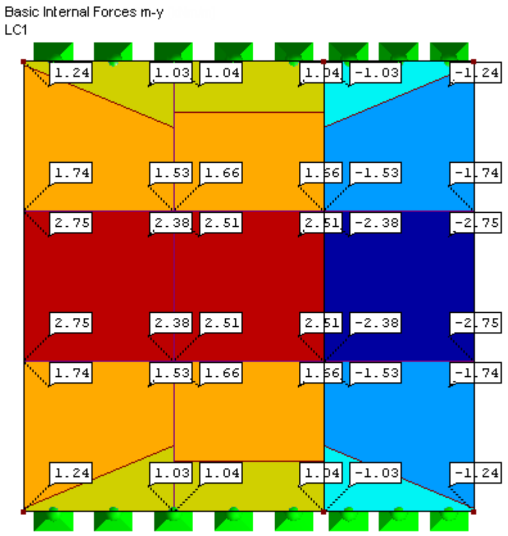


Figure 9.32: Distribution of internal forces *Non Continuous*, values *On FE mesh points*

The display shows the FE node values resulting from the displacements and rotations of each single element. Therefore, several values are displayed for each FE node. A dotted line on the node value indicates the element to which the value belongs.

For the graphic display, a plane is defined by the corner values of every element. As the results of adjacent elements are not taken into account, you see a discontinuous distribution.

Distribution of internal forces *Continuous within Surfaces/Solids*

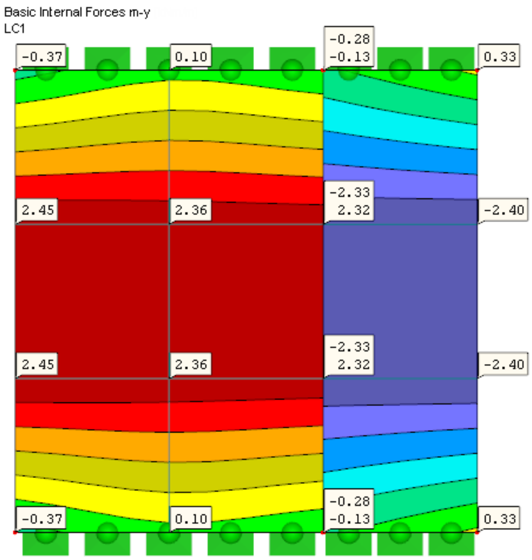


Figure 9.33: Distribution of internal forces *Continuous within Surfaces*, values *On FE mesh points*

The values on the FE nodes are averaged. Averaging stops on the surface boundary which may result in discontinuities between adjacent surfaces. However, this is absolutely correct in our example. At the boundary line, two FE node values are displayed.

This smoothing option is set by default because in most cases it provides the best results.

Distribution of internal forces *Continuous Total*

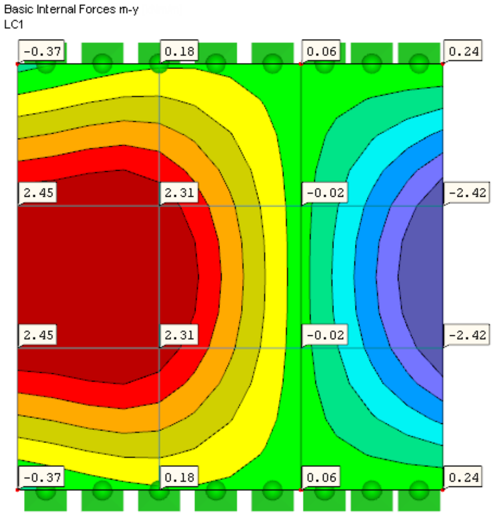


Figure 9.34: Distribution of internal forces *Continuous Total*, values *On FE mesh points*

The average values are determined by considering the FE values beyond the surface boundaries. This leads to a continuous distribution between adjoining surfaces, which is not correct for our example.

The following requirements must be met to apply this results display correctly:

- The orientation of the surface axis systems is congruent.
- Only two surfaces concur.
- The surfaces lie in one single plane.
- No line release has been defined on the boundary line.

If one of these conditions is not given, an incorrect distribution of results is displayed.

Distribution of internal forces *Continuously by groups*

In addition, it is possible to smooth results by groups. RFEM provides *Generated groups* with similar material properties.

Moreover, it is possible to create user-defined groups of surfaces or solids by using the navigator context menu (right-click on *User Groups*).

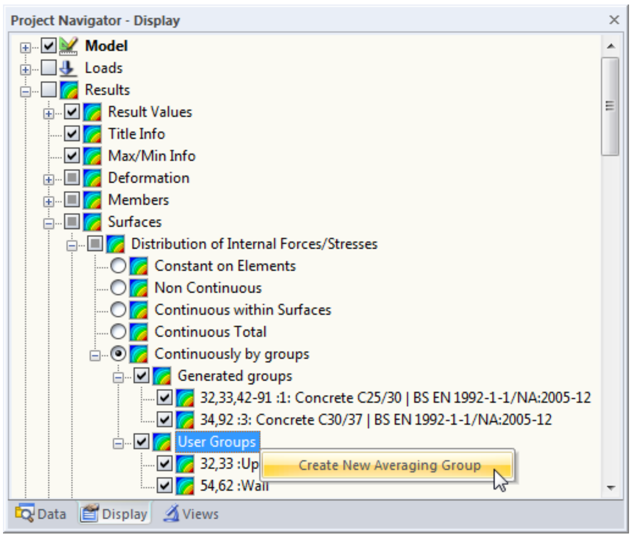


Figure 9.35: *Create New Averaging Group* via the navigator context menu

Then a dialog box opens where you can define the objects that you want to examine as a group.

9.7.2 Result Diagrams

In the *Result Diagram* dialog box (see chapter 9.5, page 356), you can create smooth ranges to prepare results for civil engineering purposes. To use this function, click the diagram toolbar button shown on the left. The following dialog box opens:

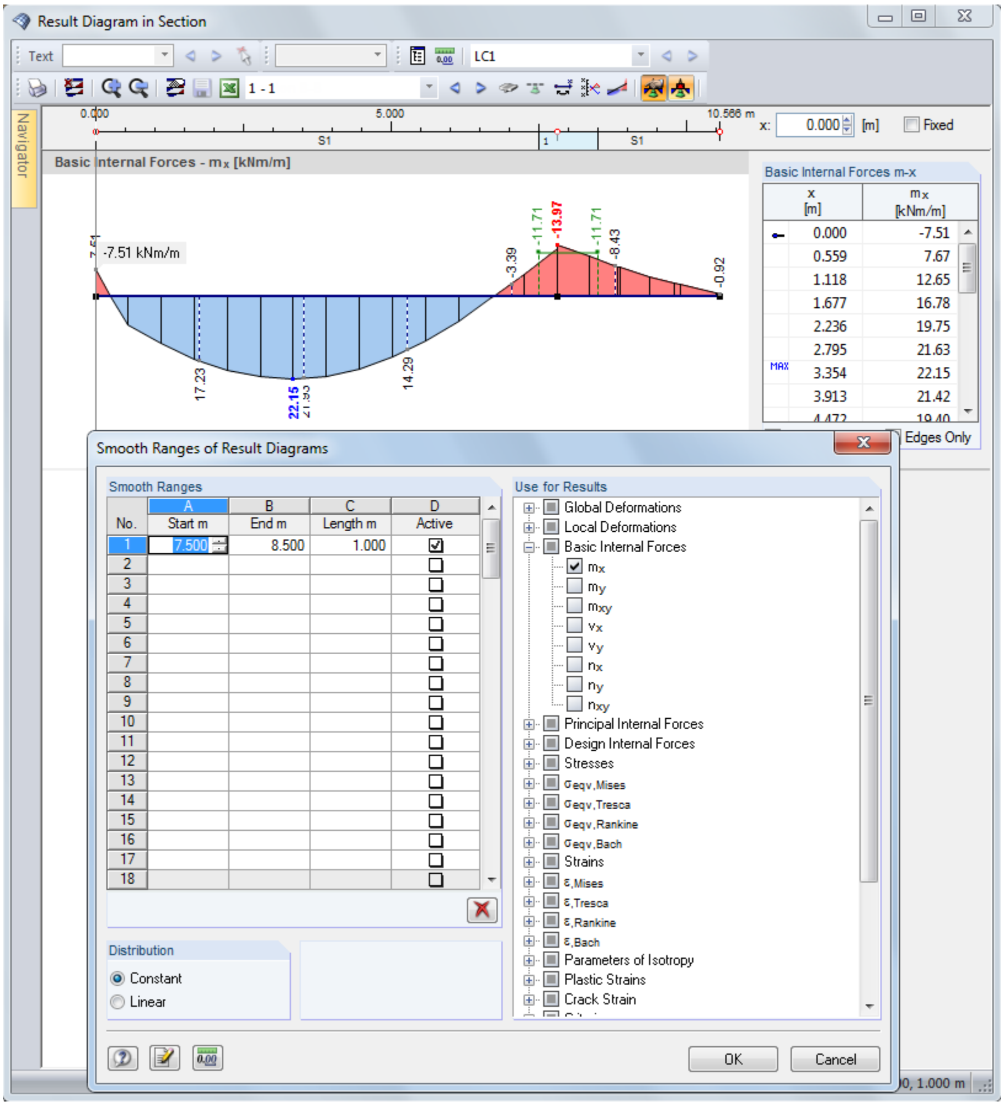
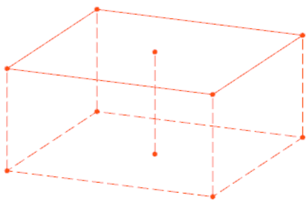


Figure 9.36: Dialog box *Smooth Ranges of Result Diagrams*

In the table columns on the left, define the *Smooth Ranges*. Please note that entries for *Start*, *End* and *Length* are interactive. Each range can be separately set to *Active*. In the dialog section *Use for Results* to the right, you decide for which deformations and internal forces you want to apply a smoothing.

The smoothing can be defined as *Constant* distribution (as shown in the figure above) or as *Linear* for all smooth ranges.

9.7.3 Average Region



It is possible to define a region in the model where the graphical results won't be displayed with the actual distributions but as average value. This average region allows for an evaluation of the averaged surface internal forces and stresses. Regions are managed as independent objects in the tabs of the Project Navigator.

To create an average region,

select **New Average Region** on the **Results** menu

or use the corresponding context menu in the *Data* navigator.

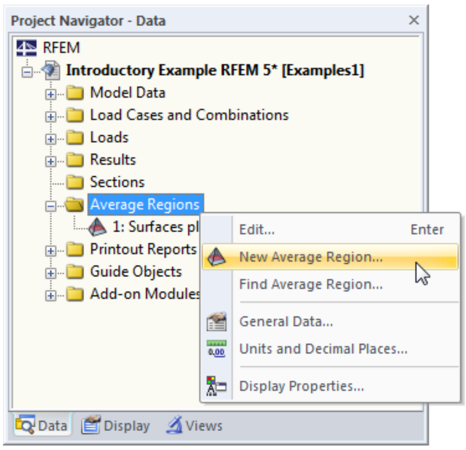


Figure 9.37: *Data* navigator: context menu of *Average Regions*

A dialog box opens where you can specify the parameters of the region.

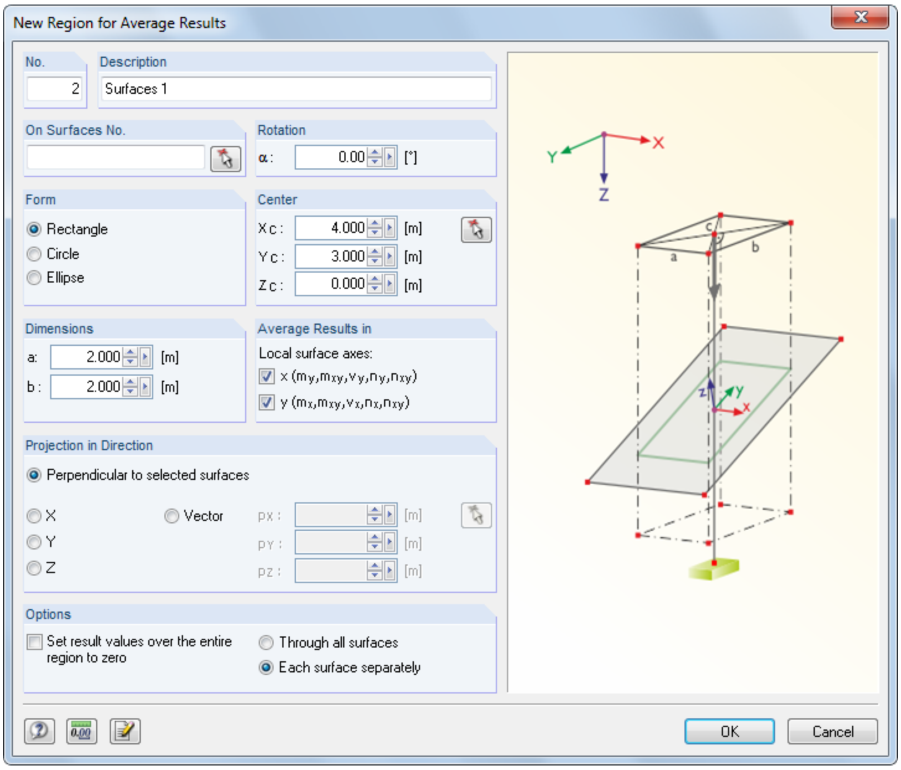


Figure 9.38: Dialog box *New Region for Average Results*

In addition to the *No.* you have to enter the *Description* of the region, making the selection easier when you evaluate results. The regions are stored under *Average Regions* in the *Data* navigator where they can be adjusted subsequently.

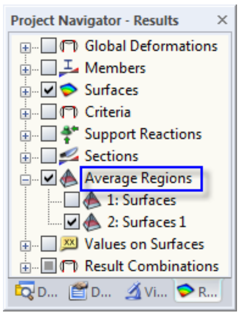
The dialog section *On Surfaces No.* lists the numbers of the surfaces for which you want to average results. This option is useful if the region's projection intersects with several surfaces. You can select relevant surfaces also graphically by using the [↖] function.

The *Form* of the region can be defined as rectangle, circle or ellipse. The respective parameters are shown in the dialog graphic to the right.

In the dialog section *Center*, specify the center point C of the region. The coordinates can be entered manually or selected graphically in the work window by using the [↖] function. The *Dimensions* describe the shape of the region by means of parameters relevant for your settings.

In the dialog section *Average Results in*, you can decide whether you want to average results in relation to both surface axes or if averaging refers only to one of the local axes.

The dialog section *Projection in Direction* describes the reference of the defined region to the surfaces. Generally, the projection is set perpendicular to the selected surfaces, but global projection directions in X, Y and Z are also possible as well as entering any projection vectors. With the [↖] function you can select two points in the work window to define the vector.



Control of regions in *Results* navigator

Finally, the dialog box provides an option to *Set result values over the entire region to zero*. In this way, it is possible to disable the result values in a selected zone of the model.

In the *Results* navigator, you can display and hide average regions individually or globally.

9.8 Multiple Windows View

On the screen, several windows showing different deformations or internal forces can be displayed together. To open the corresponding dialog box,

select **Arrange Results Window** on the **Results** menu

or use the toolbar button shown on the left.

A dialog box with a navigator tree opens where you can tick the result types that you want to be displayed in the single windows.

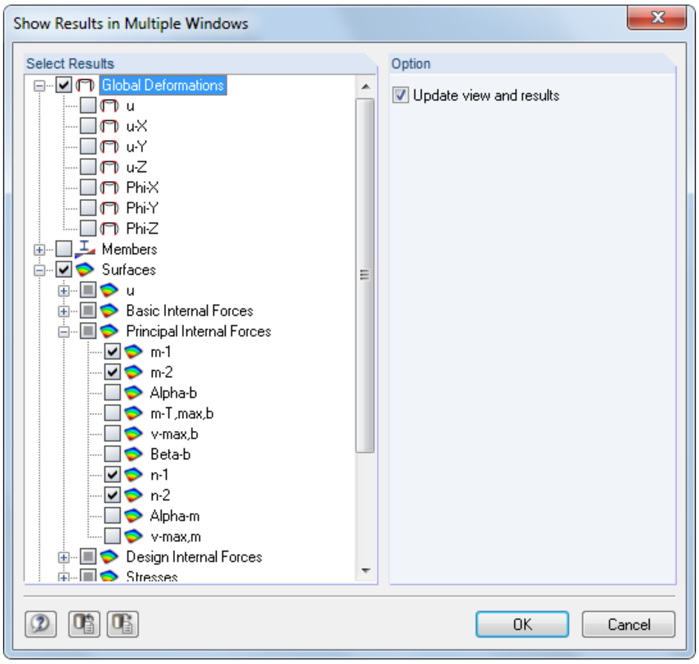


Figure 9.39: Dialog box *Show Results in Multiple Windows*

The multiple windows display can also be used for the printout (see chapter 10.2.1, page 406).

9.9 Filter Results

Various filter functions are available which are especially useful in case of complex models for evaluating and documenting results.

9.9.1 Views

User-defined views (angles of view, zoom settings etc.) make the results evaluation easier. By using "visibilities" you can subdivide the model into user-defined and generated partial views fulfilling certain criteria. Thus, it is possible to activate for example only the surfaces of a plane or members with a particular cross-section for the display. Of course, you can use these possibilities for both evaluation of results and input of model or load data.

You can access the different functions in an independent **navigator** (chapter 9.9.1.1) and by using **list buttons** or **menu functions** (chapter 9.9.1.2).

9.9.1.1 Views Navigator

The *Views* tab of the Project Navigator allows you to create user-defined views of the model which you can use for input and evaluation. The tab manages also the visibilities which can be user-defined or automatically created.

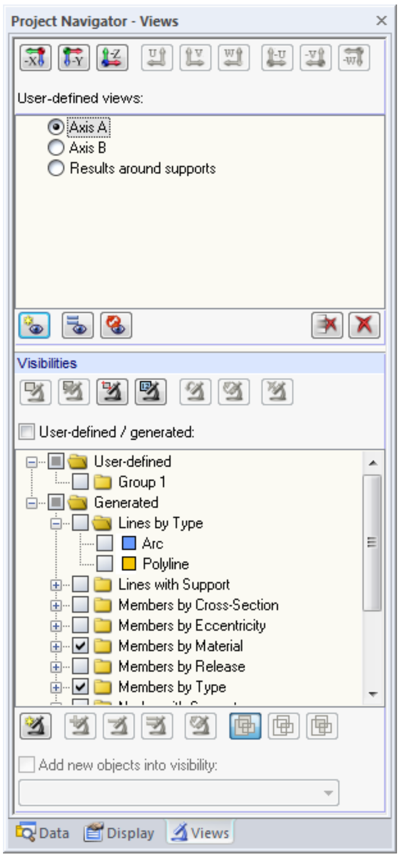


Figure 9.40: Tab *Views* of Navigator

User-defined views

In contrast to object-oriented *Visibilities* (see below), *User-defined views* allow you to save and import particular viewing angles, zoomed views as well as settings in the *Display* navigator.

The currently set view will be saved as display setting - no matter which filter specifications are effective in the *Visibilities* list: RFEM uses always the current visibilities settings for the object representation of a *User-defined view*. A user-defined view manages only the viewing angle, the zoom factor and the specifications set in the *Display* navigator.

Use the [View] buttons to quickly set the following standard angles of view:

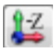
	View against axis X
	View against axis Y
	View against axis Z
	View in direction or against axis U of work plane (see chapter 11.3.1, page 434)
	View in direction or against axis V of work plane
	View in direction or against axis W of work plane

Table 9.2: [View] buttons

The buttons below the *Views* list have the following functions:

	Creates a new *User-defined view* from current view (see Figure 9.41)
	Redefines the active *User-defined view* by current view
	Restores the active *User-defined view* after modifications
	Deletes the entry selected in the list *User-defined views*
	Deletes all *User-defined views*

Table 9.3: Buttons in the navigator section *User-defined views*

Creating user-defined views

The currently set view can be saved by using the [New] button shown on the left. A dialog box appears where you have to enter the *Name* of the new display setting.

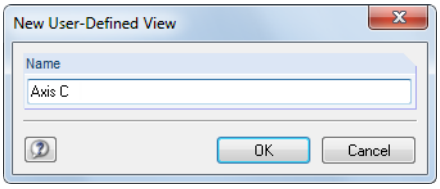


Figure 9.41: Dialog box *New User-Defined View*

Visibilities

With the so-called "visibilities" it is possible to display partial views of the model or groups of objects such as surfaces lying in one plane or columns of a particular story.

Visibility buttons

The buttons above the *Visibilities* list (see Figure 9.40, page 369) allow you to select the objects for representation by particular criteria. The buttons have the following functions:

	Displays objects selected in work window as partial view
	Hides objects selected in work window
	Creates a visibility by drawing a window (see page 373)
	Defines a new visibility by means of object numbers (see page 373)
	Restores previous visibility
	Reverses current display (new visibility: hidden objects)
	Quits visibility mode. All objects are displayed again.

Table 9.4: Buttons above *Visibilities* list

The *Visibilities* list contains user-defined and generated visibilities.

User-defined visibilities

With the graphical or numerical selection of objects (see chapter 11.2, page 430) you can create a visibility.

Use the button [Create New User-Defined Visibility] (below the *Visibilities* list) to save the current partial view. The dialog box *New User-Defined Visibility* opens where you define a name and the *Group* (see Figure 9.45, page 374).

The buttons below the *Visibilities* list have the following functions:

	The dialog box *New User-Defined Visibility* appears (see Figure 9.45, page 374).
	Adds objects selected in work window to group marked in list above (see page 374)
	Removes objects selected in work window from group marked in list above (see page 374)
	Reassigns selected objects to group marked above
	Reverses current display (new visibility: hidden objects)
	Shows all objects activated in *Visibilities* list
	Shows only objects available in each active *Visibilities* entry
	Shows objects available in each active *Group*

Table 9.5: Buttons below *Visibilities* list

With the check box *Add new objects into visibility* you can decide how you want to treat new nodes, lines, members etc. when you work in a user-defined visibility. If the option is ticked, you can define the relevant group in the list below.

A color symbol is automatically assigned to each user-defined visibility. The colors can be used as well in the *Display* navigator for graphical representation of objects (see chapter 11.1.9, page 428). In this way, you can detect the customized visibilities quickly in the model. To set the display for groups, use the *Views* navigator.

Generated visibilities

RFEM generates automatically visibilities for surfaces, lines, members etc. according to particular criteria.

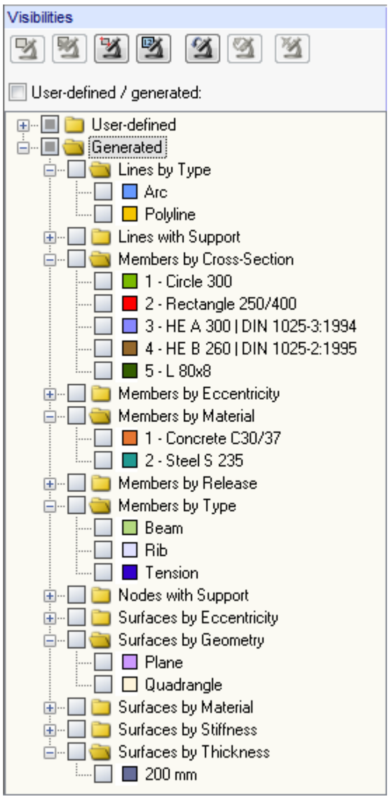
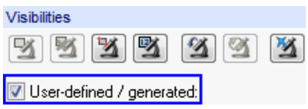


Figure 9.42: Generated visibilities in *Views* navigator

These generated visibility types help you to get a quick overview about the model as you can take the list to filter objects specifically. In this way, you can easily check both input and results in RFEM.

In addition to the multiple selection of generated views (default), the list allows for creating an intersecting set. Use the navigator buttons shown on the left to set the intersection. You find them below the list. The functions are described in Table 9.5 above.



With the check box *User-defined / generated* on the top of the list you can decide if the filter function is effective for the work window. All objects will be displayed again after removing the check mark.

9.9.1.2 Visibility Buttons and Menu

To access the different visibility functions,

point to **Visibility** on the **View** menu

or use the corresponding list button of the pull-down menu in the toolbar.

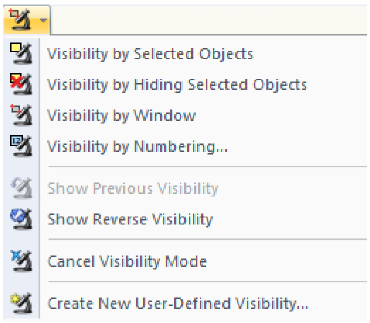


Figure 9.43: List buttons for *Visibility*

Visibility by window

Partial views can be created graphically by using the mouse and drawing a window.

When you open the window from the left to the right, the visibility includes only objects that are completely contained within the window. When opening the window from the right to the left, the visibility additionally includes objects that are cut by the window.

Visibility by numbering

Enter the numbers of *Nodes*, *Lines*, *Members*, *Surfaces* or *Solids* that are relevant for the visibility in a dialog box.

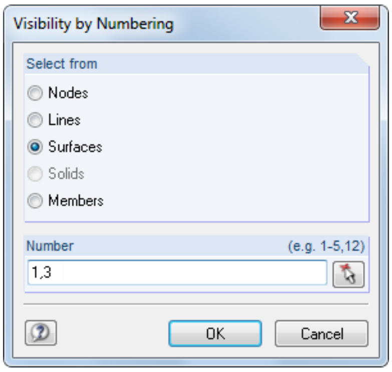


Figure 9.44: Dialog box *Visibility by Numbering*

Cancel visibility mode

Use this function to restore the view of all objects.

Create a user-defined visibility

Before you access the function, select the objects that you want to save as *Visibility* in the work window (see chapter 11.2.1, page 430 and chapter 11.2.2, page 433). The following selection function is useful: Point to **Select** on the **Edit** menu, and then select **Special**.

Only the objects that are selected in the work window will be integrated in a *Visibility*. Therefore, when you use the function [Visibility by Hiding Selected Objects], you have to select the displayed objects once again by drawing a window across them.

After a click on the [New] button shown on the left the following dialog box appears.

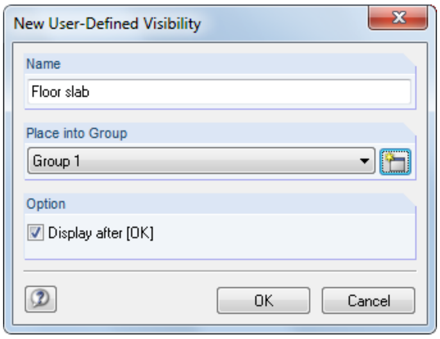


Figure 9.45: Dialog box *New User-Defined Visibility*

Define the *Name* and *Group*. If you want to use more visibility groups, click the [New] button to create another group.

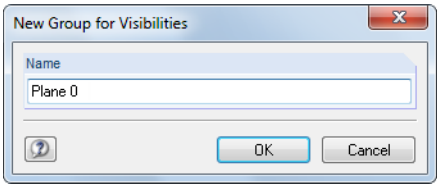


Figure 9.46: Dialog box *New Group for Visibilities*

Click the [OK] button to save the group of objects as new visibility.

The user-defined visibilities are managed in the *Views* navigator where they can be switched on and off individually (see Figure 9.40, page 369).

Change objects in visibilities

Objects can be integrated subsequently into existing visibilities: Quit the visibility mode by clicking the button shown on the left. You can also point to *Visibility* on the *View* menu where you select *Cancel Visibility Mode*. Now, select the objects that you want to add.

In the *Views* navigator, click the relevant entry in the *User-defined* list . RFEM enables the button [+] so that you can integrate the selected objects into the user-defined visibility.

In the same way, you can use the [-] button to remove selected objects from a user-defined visibility.

Click the button [=] to overwrite the objects available in the marked visibility of the *Views* navigator with the selection in the work window. Thus, existing visibilities can be redefined but the name is kept.

Transparency for hidden objects

When you use visibilities, it is possible to display hidden objects with minor intensity in the background. The degree of visibility is set individually in the *Graphics* tab of the dialog box *Program Options* (see Figure 9.52, page 379).

The display of background objects can be turned on and off in the *Display* navigator.

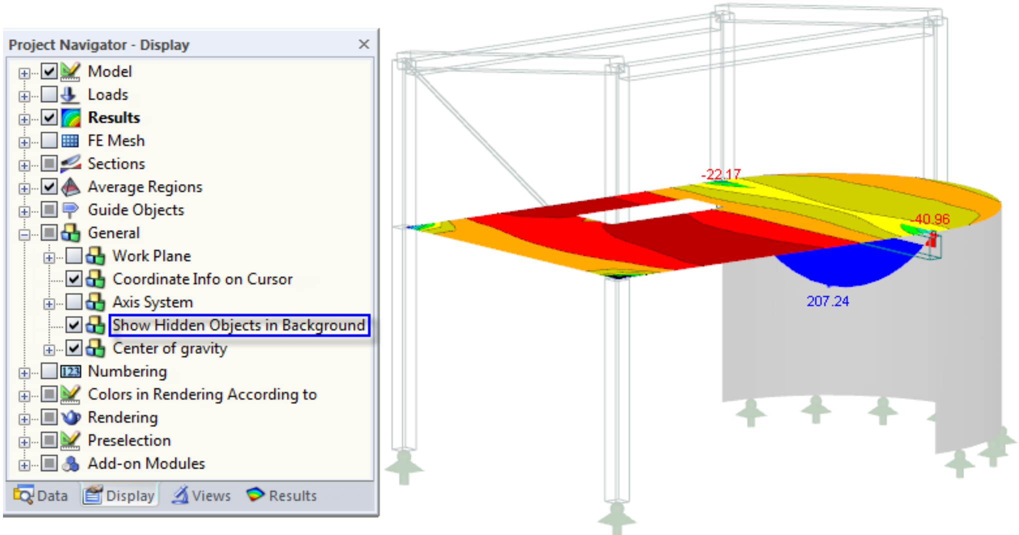
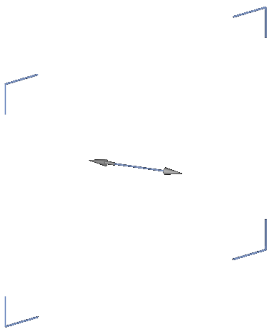


Figure 9.47: *Display* navigator: Option *General* → *Show Hidden Objects in Background*

9.9.2 Clipping Plane



You can define any section plane cutting through the model. The zone in front of (or behind) the plane will be hidden in the display. In this way, it is possible for example to look at the results in an intersection or in a solid.

RFEM places the clipping plane through the center of the geometric total dimensions. Thus, the plane is related to the model geometry. In the work window, the clipping plane is enclosed by a frame.

It is not possible to save a clipping plane.

To access the corresponding function,

select **Clipping Plane** on the **Insert** menu.

The following dialog box appears:

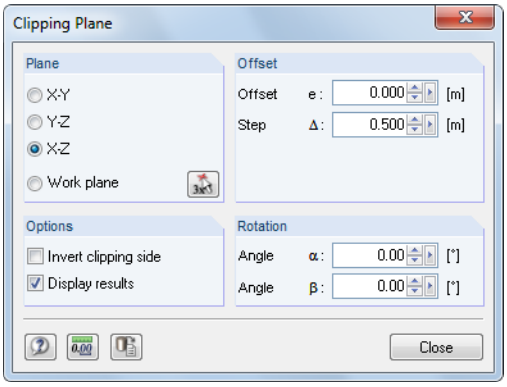


Figure 9.48: Dialog box *Clipping Plane*

You can arrange the *Plane* parallel to one of the planes spanned by the axes of the global coordinate system XYZ. In addition, you can place the plane into the current work plane. You can also select three points in the work window by clicking the [↖] button shown on the left.

The value entered into the input field *Offset* results in a parallel displacement of the plane in direction of the positive or negative axis that is perpendicular to the plane. Both directions are indicated by gray arrows in the work window. The offset can be entered directly or set with the spin box. The input field *Step* controls the interval of spacings by which the plane is shifted every time you click a button of the spin box.

In the dialog section *Options*, you have the possibility to change the active side of the clipping plane. Moreover, you can switch on and off the result diagrams available on the clipping borders.

Furthermore, it is possible to rotate the clipping plane by a *Rotation* about the angle α (about the last named axis of the plane) and angle β (about the first named axis). The graphic is interactive with the input.

When the dialog box *Clipping Plane* is open, you can use all edit and view functions in the work window, but there is no printout option. Quit the function with the [Close] button.

The following example shows a clipping plane running through a node of a pipe connection.

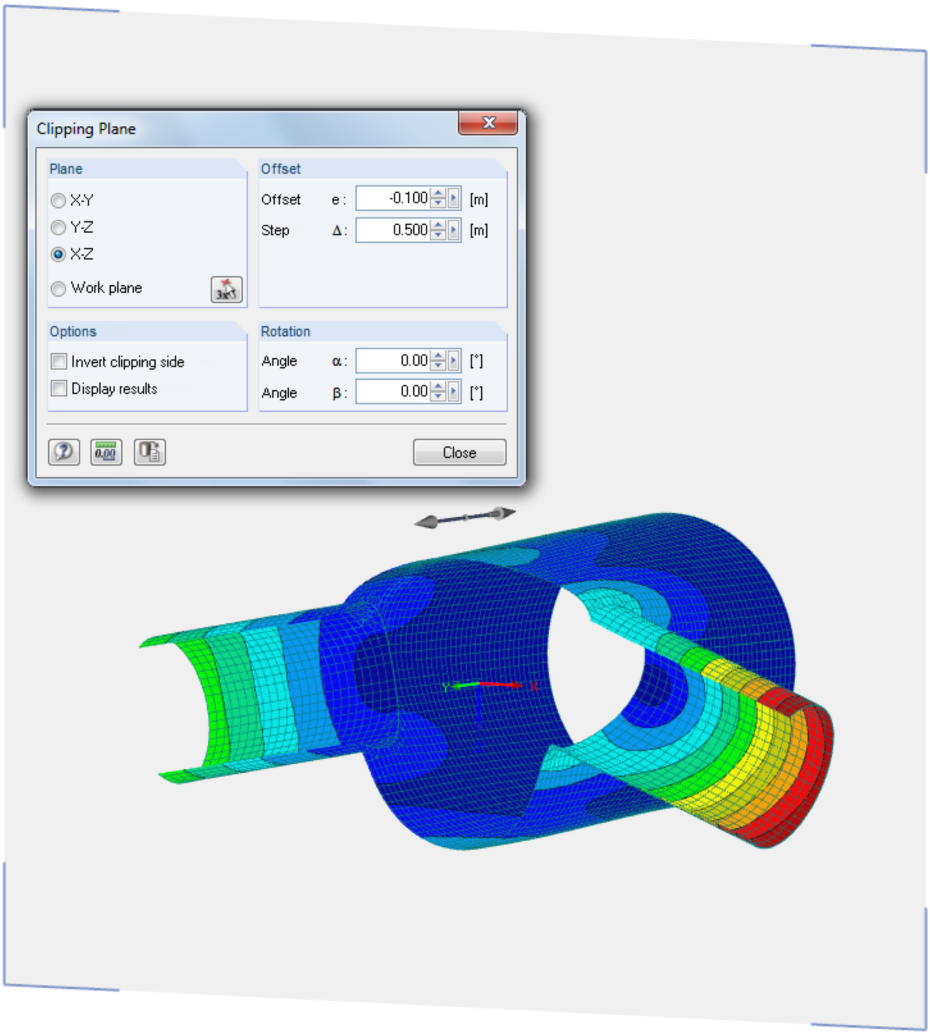


Figure 9.49: Clipping plane cutting through a pipe connection

9.9.3 Filter Functions

The grouping options described in chapter 9.9.1 *Views* refer to the objects of the model. Additionally, you can use internal forces, deformations and stresses as filter criteria.

Filtering results

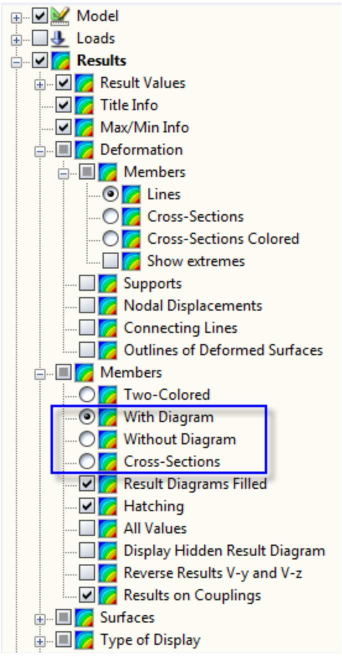
Results are filtered by means of the control panel. If the panel is not displayed,

select **Control Panel (Color Scale, Factors, Filter)** on the **View** menu

or use the toolbar button shown on the left.

The control panel is described in chapter 3.4.6 on page 29.

The filter settings for results are defined in the *Color Spectrum* tab that is shown for the isoband and isoline results of surfaces and solids (see Figure 3.17, page 29). As the tab is not available for the two-colored display of member internal forces, you have to switch to the *Display* navigator and set the display options *With/Without Diagram* or *Cross-Sections* (see figure shown on the left).



Settings in *Display* navigator for multi-color member results

In the panel it is possible to set specific displays, for example member moments displayed only if they exceed a particular value, or base internal forces of surfaces using a fine gradation shown within the range of ± 30 kNm (see Figure 3.19, page 31).

The following example represents a floor slab. RFEM displays contact stresses only between -120 kN/m² and -260 kN/m² on the model.

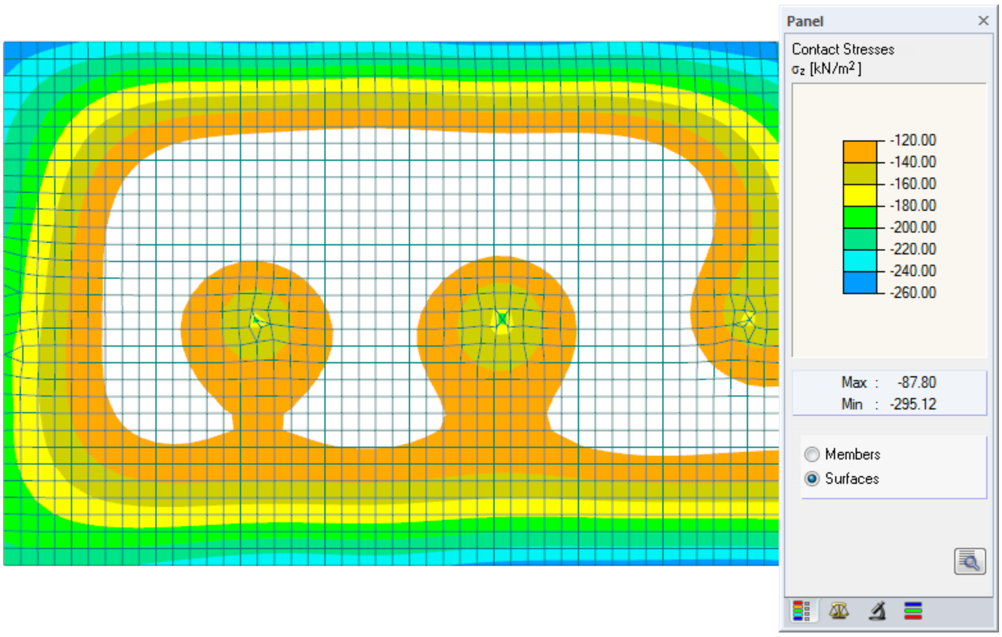


Figure 9.50: Filtering contact stresses with adjusted color spectrum

As the model provides also multi-colored member results, the selection field *Surfaces* is set active in the control panel. Furthermore, the color spectrum is modified in such a way that a color range covers exactly -20 kN/m². No results for surface elements with elastic foundations whose contact stresses are beyond the defined range of values are shown.

Filtering objects

In the *Filter* tab of the control panel, you can enter the numbers of selected members, surfaces or solids to display their result diagrams in a filtered display. The function is described in chapter 3.4.6 on page 32.

In contrast to the visibility function, the model is completely displayed in the graphic.

The following figure shows the bending moments available in the floor surfaces of a building. The walls are shown in the model but displayed without internal forces.

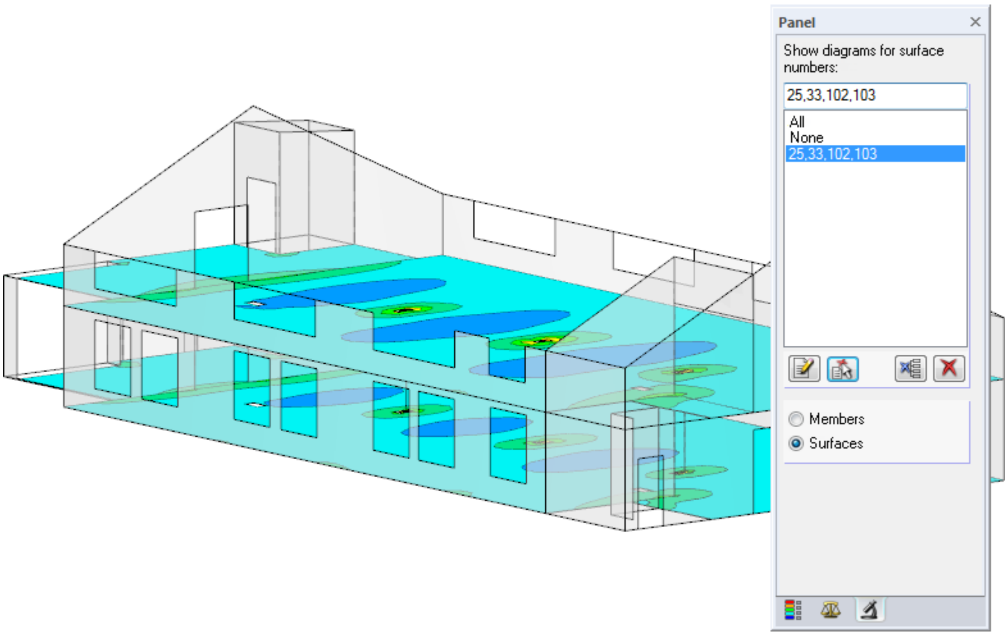


Figure 9.51: Filtering surfaces: bending moments of floors

The filter settings of the panel also affect the objects in the results tables: When you restrict the results display in the panel to for example two members, table 4.6 *Members - Internal Forces* will list only the results of those two members.

9.10 Animation of Deformations

Normally, deformations of objects are displayed in their final state.

But it is also possible to show the deformation process in motion. To open the corresponding dialog box,

select **Animation** on the **Results** menu

or click the corresponding button in the toolbar. To close the animated view, click the button again. You can also use the [Esc] key.

To define detailed settings for the animation process, use the *Program Options* dialog box.

Select **Program Options** on the **Options** menu,

and then open the dialog tab *Graphics*.

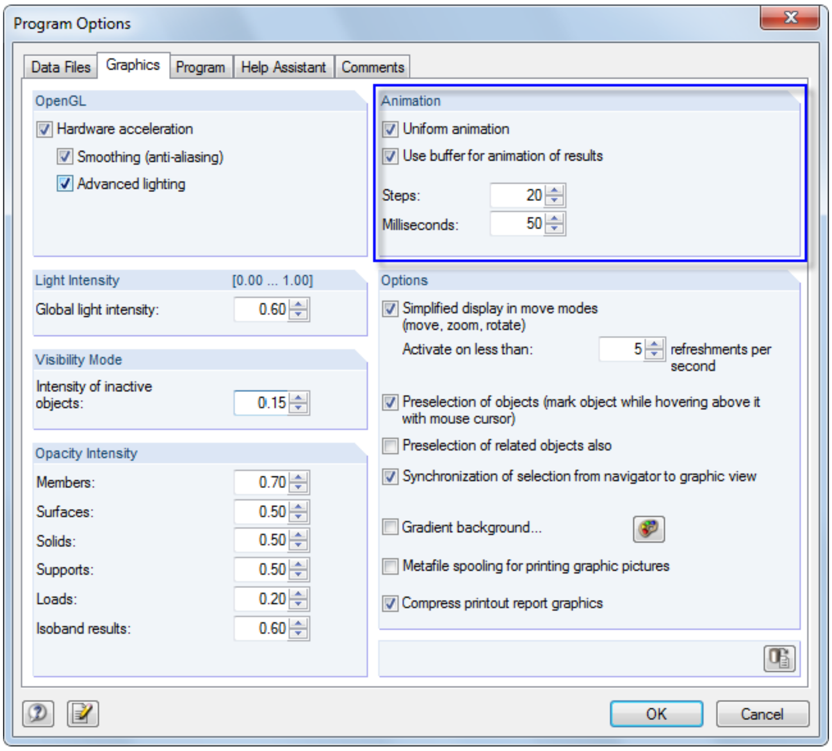


Figure 9.52: Dialog box *Program Options*, tab *Graphics*

The animation of deformations can be saved as video file. Set the animated graphic appropriately on the screen, and then select

Create Video File on the **Tools** menu.

You may see a message about OpenGL settings before the corresponding dialog box appears where you can define different settings for creating the video file.

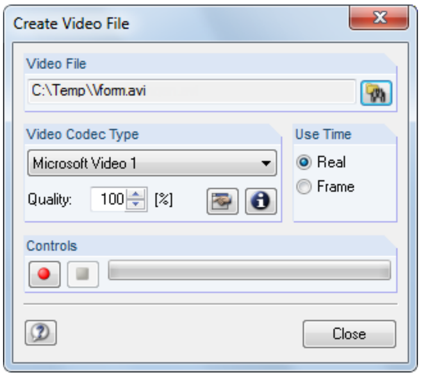


Figure 9.53: Dialog box *Create Video File*

Click the [Browse] button to define the name of the video file in a separate dialog box.

The red button [Record] starts the recording, and the blue button [Stop] stops it.

10. Printout

10.1 Printout Report

Normally, input and results data of RFEM are not sent directly to the printer. Instead, a so-called printout report is generated first to which you can add graphics, explanations, scans and other elements. In the printout report you define the data that will finally appear in the printout.

It is possible to create several printout reports for the model. When your structure is quite complex, it is recommended to split data into several small reports instead of creating a single report that is rather comprehensive. For example, you can create a report for input data, another one for support forces and one for surface results. In this way, you can reduce time of waiting.

It is also possible to create different printout reports in an RFEM model. Depending on the required data, the test engineer and the design engineer may receive different printout reports.

A printout report can only be called up if a default printer has been installed in Windows. The preview in the printout report uses the printer driver.

10.1.1 Create or Open Printout Report

To create a new printout report

select **Open Printout Report** on the **File** menu

or use the toolbar button shown on the left. You can also use the context menu of the corresponding entry in the *Data* navigator.

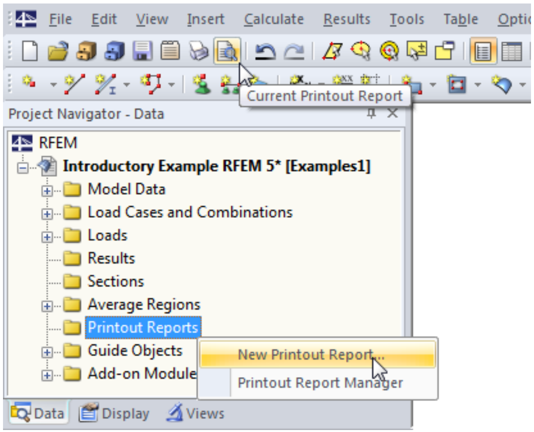


Figure 10.1: Button and context menu of *Printout Report*

The following dialog box appears:

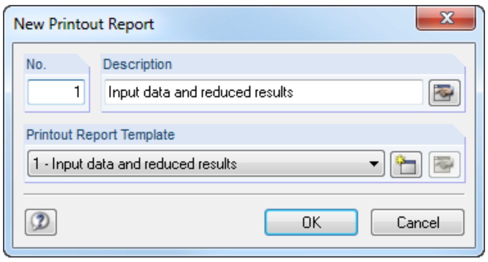


Figure 10.2: Dialog box *New Printout Report*

The *No.* of the report is preset but can be changed. In the input field *Description*, you can enter a name for the report making the selection in the lists easier later. This description does not show up in the printout.

Furthermore, you can select a particular report template from the list in the dialog section *Printout Report Template* (see chapter 10.1.7, page 396).

The buttons in the dialog box have the following functions:

	A new report template can be created.
	The selection of the report can be edited (→ chapter 10.1.3, page 384).

Table 10.1: Buttons in the dialog box *New Printout Report*

When a printout report is already available, and you select **Open Printout Report** on the **File** menu, the *Printout Report Manager* appears.

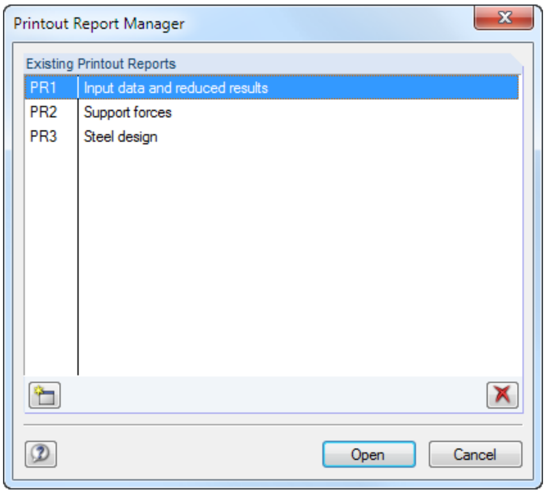


Figure 10.3: Dialog box *Printout Report Manager*

You can select the relevant report from the list.

The buttons in the dialog box have the following functions:

	Creates a new printout report
	Deletes selected printout report

Table 10.2: Buttons in the dialog box *Printout Report Manager*

10.1.2 Working in the Printout Report

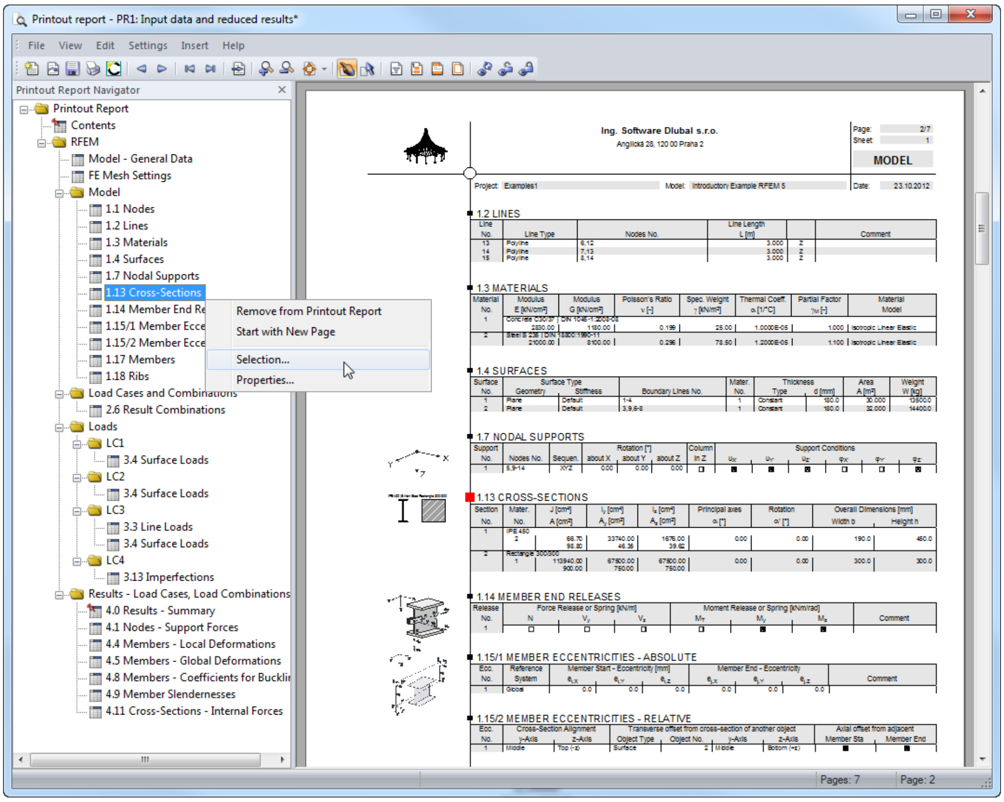


Figure 10.4: Printout report with context menu

When the printout report is open, you see the report navigator on the left. On the right, the page view with the preview of the printout is presented.

The individual chapters of the report can be shifted anywhere in the navigator by using the drag-and-drop function.

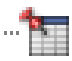
Context menu

The context menu offers additional options for adjusting the printout report. As common for Windows applications, multiple selections are possible by using the [Ctrl] or [Shift] key.

Remove from printout report

The selected chapter will be deleted. If you want to reinsert it, use the selection: Click *Selection* on the *Edit* menu to open a dialog box where you can choose data for display in the printout report.

Start with a new page



The selected chapter starts on a new page. It is marked by a red pin in the navigator (like chapter *Results - Summary* shown in the figure above).

Selection

You have access to the global selection that is described on the following pages. The selected chapter is preset.

Properties

Some general properties of the selected chapter can be modified.

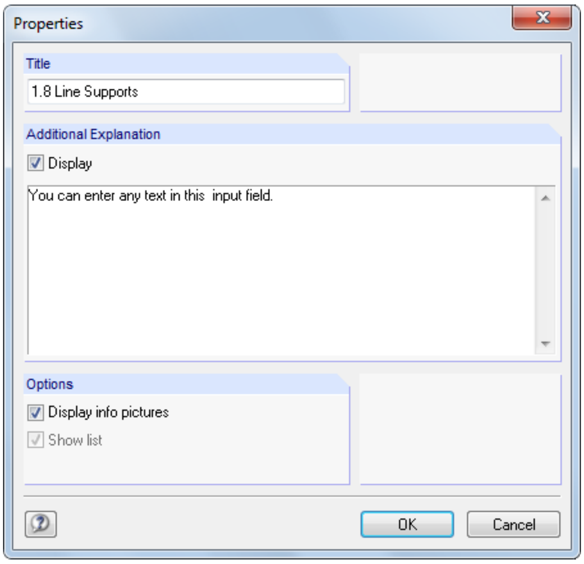


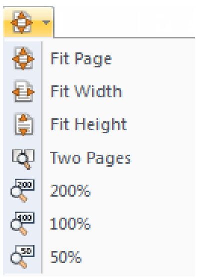
Figure 10.5: Dialog box *Properties*

It is possible to change the *Title* of the chapter and to enter an *Additional Explanation* which will appear in the left margin of the report. The additional text can be enabled and disabled for display like the *info pictures* of the chapter (for example drawings of cross-sections or loading).

Navigation in the printout report

To look at a particular section in the printout report, click the corresponding chapter entry in the navigator.

The **View** menu provides further functions for navigation. You can also use the buttons in the report toolbar to access the corresponding function.



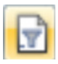
	Go to previous page in the page preview
	Go to next page
	Go to first page in the page preview
	Go to last page
	Specify the number of a particular page in a dialog box.
	Zoom in
	Zoom out
	List button for *Zoom* to adjust display size
	Grab mode: Use the mouse for navigation within the report.
	Select mode: Select and edit chapters by mouse click.

Table 10.3: Navigation buttons in the toolbar of the printout report

10.1.3 Define Contents of Printout Report

In the global selection, you can select the chapters that you want to appear in the printout report. To access the corresponding function,

select **Selection** on the **Edit** menu



or use the toolbar button shown on the left. You can also use the context menu of the *Printout Report* navigator item.

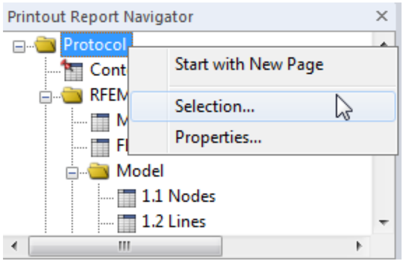


Figure 10.6: Open the global selection via the *Printout Report* context menu

The following dialog box appears:

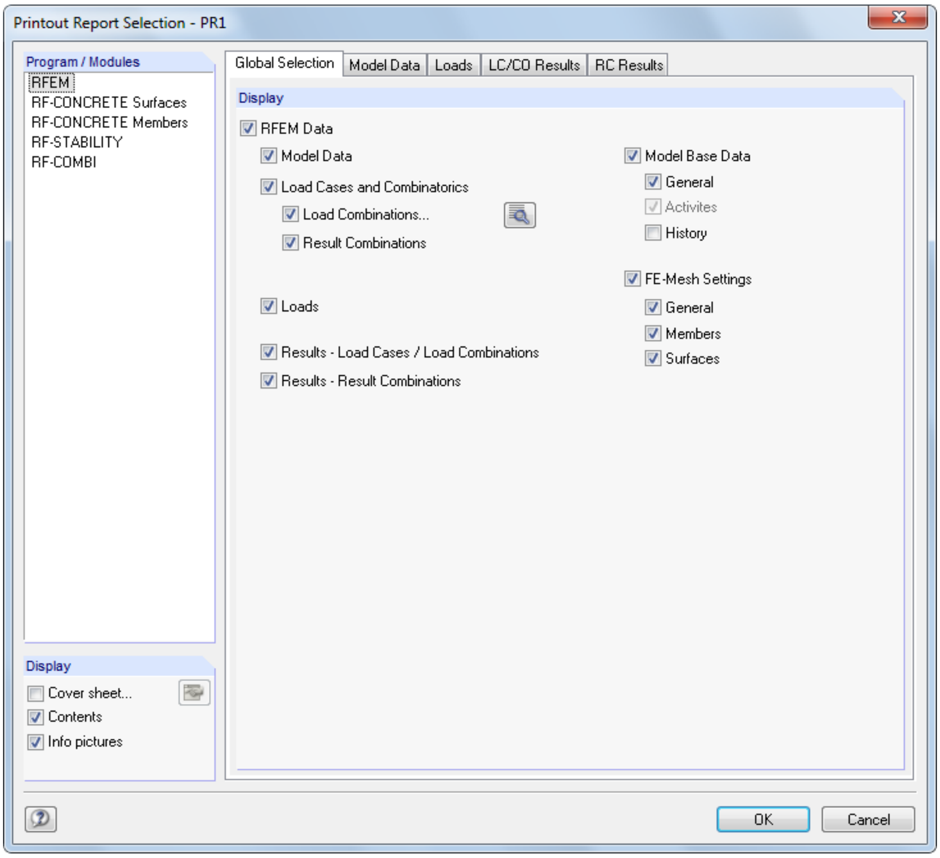


Figure 10.7: Dialog box *Printout Report Selection*, tab *Global Selection*

The list in the dialog section *Program / Modules* contains all add-on modules where input data is available. When a program is selected in the list, you can choose the chapters to be printed in the tabs to the right.

The *Global Selection* tab manages the main chapters of the report. If you clear the selection of a check box, the corresponding detail tab disappears as well.

Use the three check boxes in the dialog section *Display* (bottom left) to decide if a *Cover sheet*, a table of *Contents* and small *Info pictures* will be displayed in the report margin.

10.1.3.1 Selecting Model Data

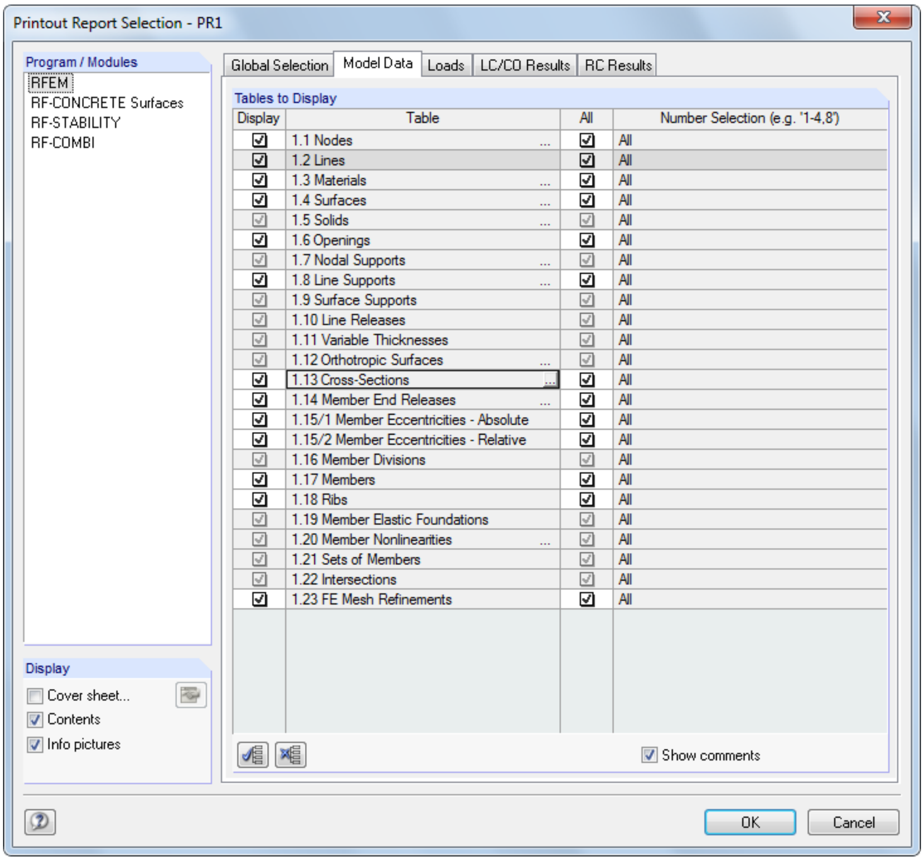


Figure 10.8: Dialog box *Printout Report Selection*, tab *Model Data*

With the check boxes in the *Display* column you decide which chapters appear in the printout report.

For some tables you find subchapters. When you click for example into the table field *1.13 Cross-Sections*, the button [...] is enabled and you can open another dialog box where it is possible to select sections for which also cross-section details will be shown. To define the types and amount of details, use the [Details] button shown on the left.

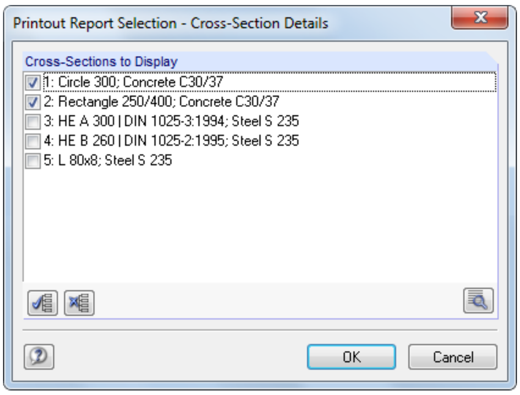


Figure 10.9: Dialog box *Printout Report Selection - Cross-Section Details*

The printout report is based on the input tables described in chapter 4. With the check boxes in the third column *All* you decide if all rows of the selected table will be included in the printout. When a check box is cleared, you can specify the numbers of selected objects (table rows) in the column *Number Selection*.

Again, it is recommended to use the button [...] becoming available at the end of the input field because in this way you can select nodes, lines, surfaces, members, sets of members, openings and solids graphically in the work window. For the remaining objects a list with table rows appears.

10.1.3.2 Selecting Load Data

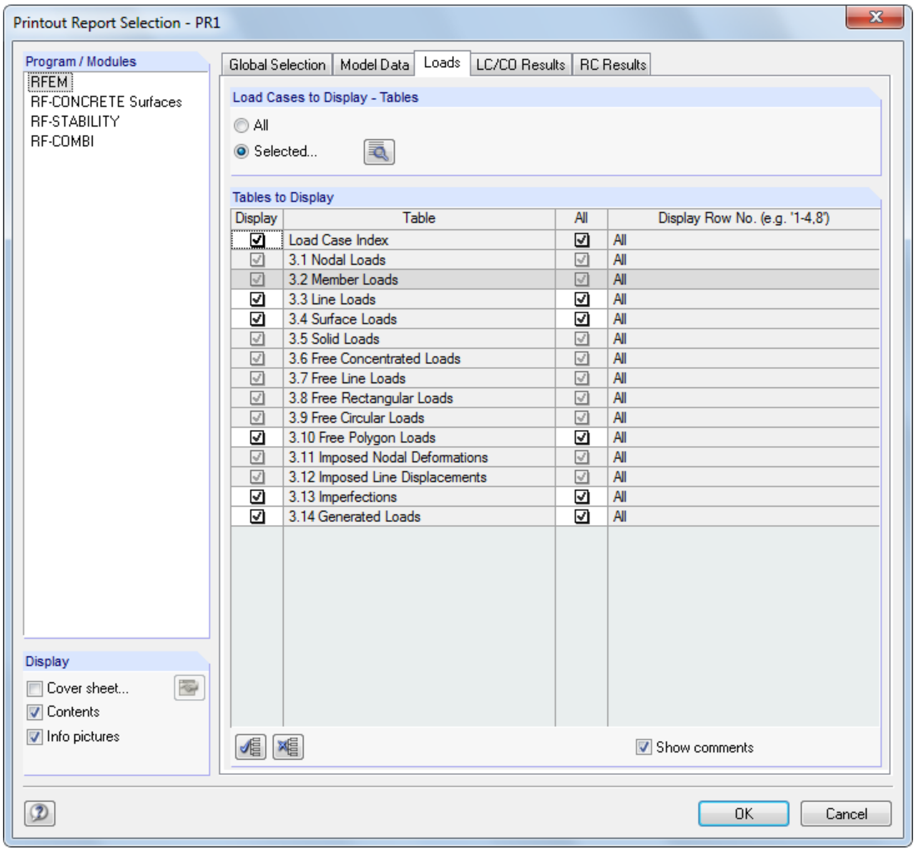


Figure 10.10: Dialog box *Printout Report Selection*, tab *Loads*

Tables are selected as described in the previous chapter 10.1.3.1.

Additional selection options are available in this tab: In the dialog section *Load Cases to Display*, you can decide whether the input data of *All* or only of particular load cases will appear in the printout. When the selection field *Selected* is activated, you can use the enabled [Details] button to open a new dialog box where you can select the load cases.

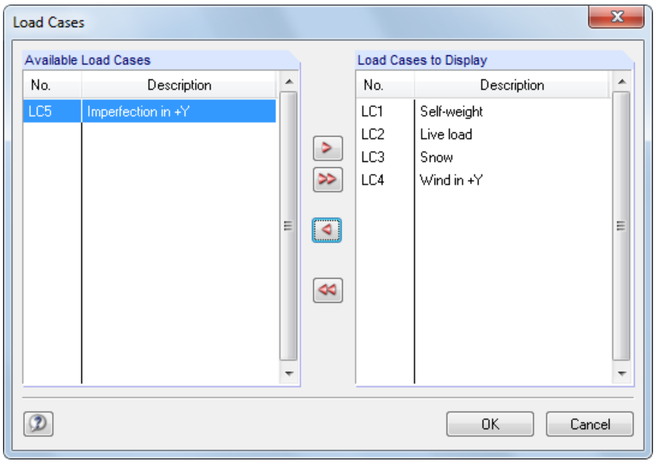


Figure 10.11: Selection of load cases

10.1.3.3 Selecting Results Data

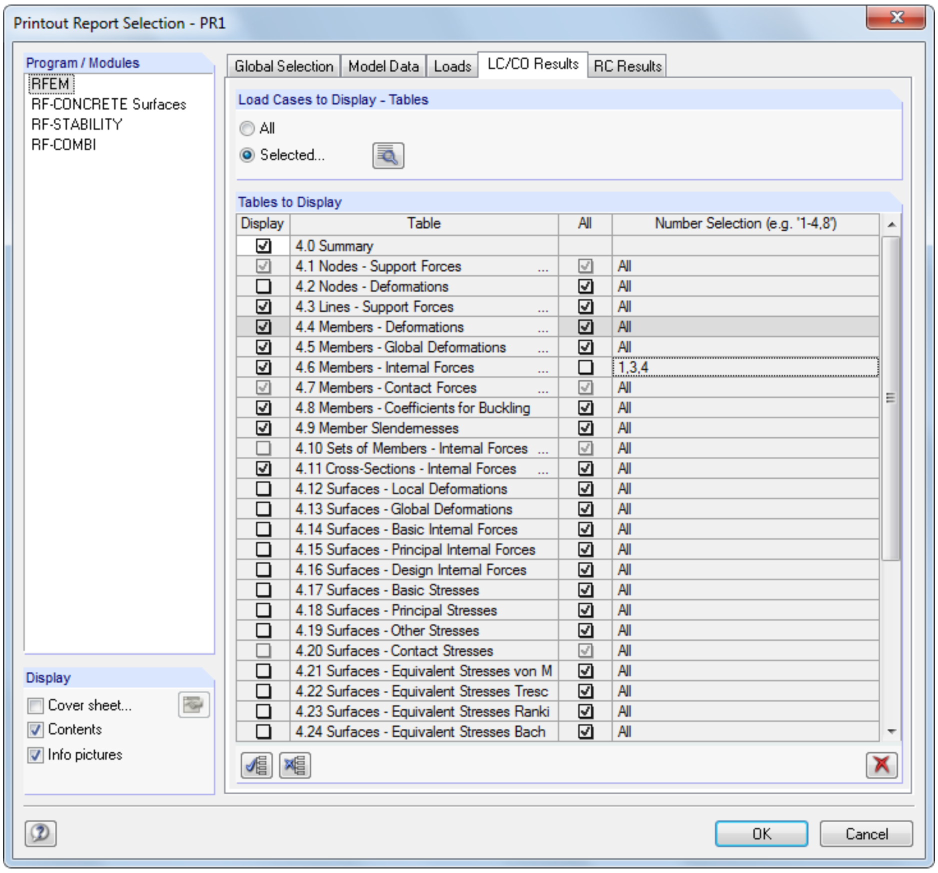


Figure 10.12: Dialog box *Printout Report Selection*, tab *LC/CO Results*

Selecting results data which is usually comprehensive is done in two dialog tabs: The tab *LC/CO Results* manages the selection for load cases and load combinations, the tab *RC Results* controls the printout for the results of result combinations.

Results data can be prepared like load data (see previous chapter 10.1.3.2): Use the selection field *Selected* to restrict the printout data to results of particular load cases or combinations. In the dialog section *Tables to Display*, you can select the tables and table rows as described in chapter 10.1.3.1. The column *Number Selection* allows you to specify particular objects or to select objects graphically by means of the button [...] that you can access at the end of the table row.

In the *Table* column, you see some table rows showing three dots at the end of the row. The dots indicate the button [...] that you can activate by clicking into the table row. Use this button to access more selection criteria, for example for member internal forces.

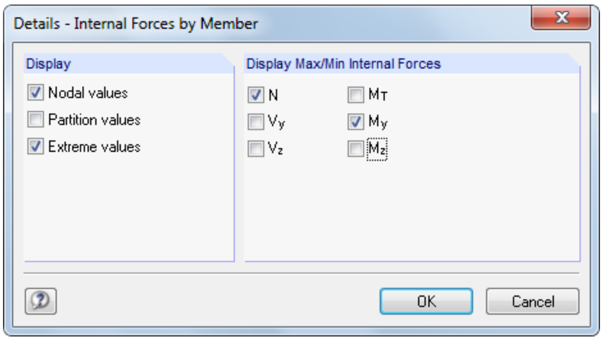


Figure 10.13: Dialog box *Details - Internal Forces by Member*

The printout report lists the results of each member on the following locations:

- Start and end node
- Division points according to defined member division (see chapter 4.16, page 136)
- Extreme values (*Max/Min*) of results (see chapter 8.6, page 296)

The selection is connected with the *Table Filter* settings (see Figure 11.122, page 487).

You can considerably reduce the printout extent by restricting output data to results that are indispensable for your documentation.

10.1.3.4 Selecting Data of Add-on Modules

All module data for printing is also managed in the RFEM printout report. You can summarize it together with the RFEM data in a single report, or you organize it in separate printout reports. For complex structural systems with a high number of design cases, it is recommended to split data into several printout reports which makes data arrangement clearer.

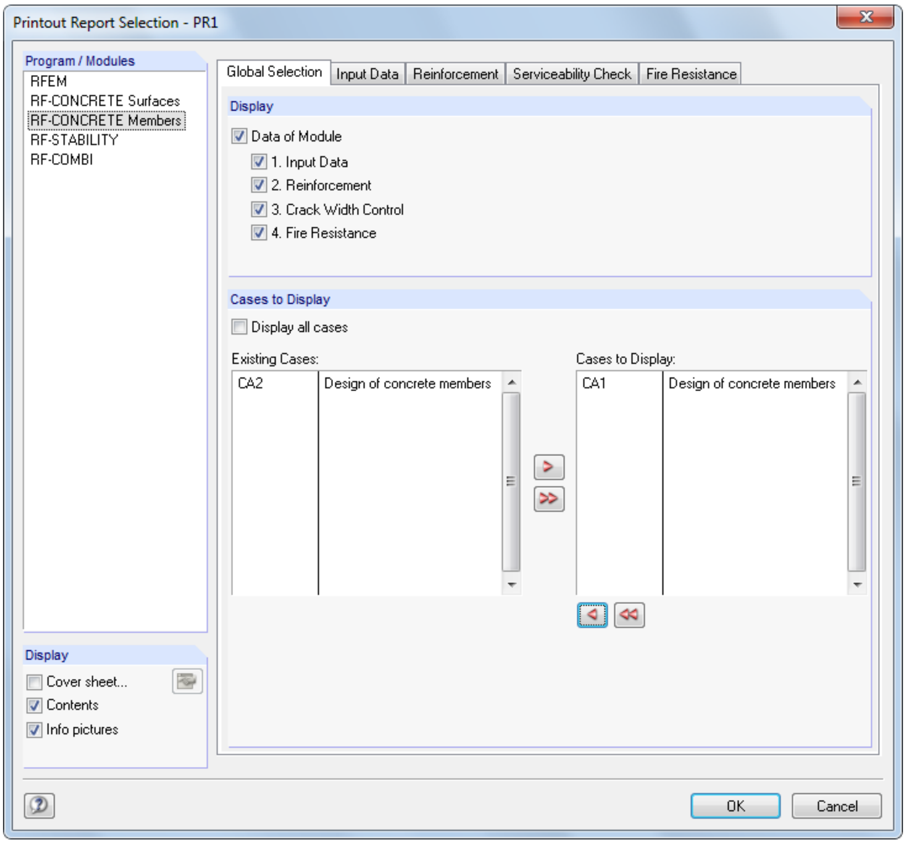


Figure 10.14: Dialog box *Printout Report Selection*, tab *Global Selection* of add-on module **RF-CONCRETE Members**

In addition to RFEM, the list in the dialog section *Program / Modules* contains all additional modules where entries have been made. When you select a module in the list, you can choose the chapters for printing in the tabs to the right.

The dialog tab *Global Selection* manages the main chapters of the add-on module data. When you clear a check box, the respective detail tab disappears as well.

In the dialog section *Cases to Display*, the option *Display all cases* is ticked by default. If you want to include only particular design cases in the printout report, clear the check box. Now, you can move the cases that you do not need from the list *Cases to Display* to the list *Existing cases*.

The selection in the detail tabs of input data and results is similar to the selection described in chapters 10.1.3.1 *Selecting Model Data* and 10.1.3.3 *Selecting Results Data* .

10.1.4 Adjust Printout Report Header

During the program installation a printout report header is created from the customer data. To change specifications,

select **Header** on the **Settings** menu in the printout report

or use the toolbar button in the printout report shown on the left.

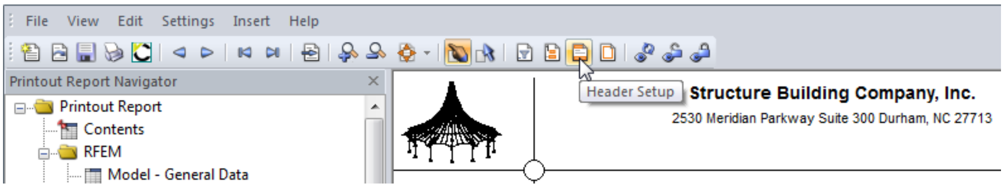


Figure 10.15: Button *Header Setup*

The following dialog box appears:

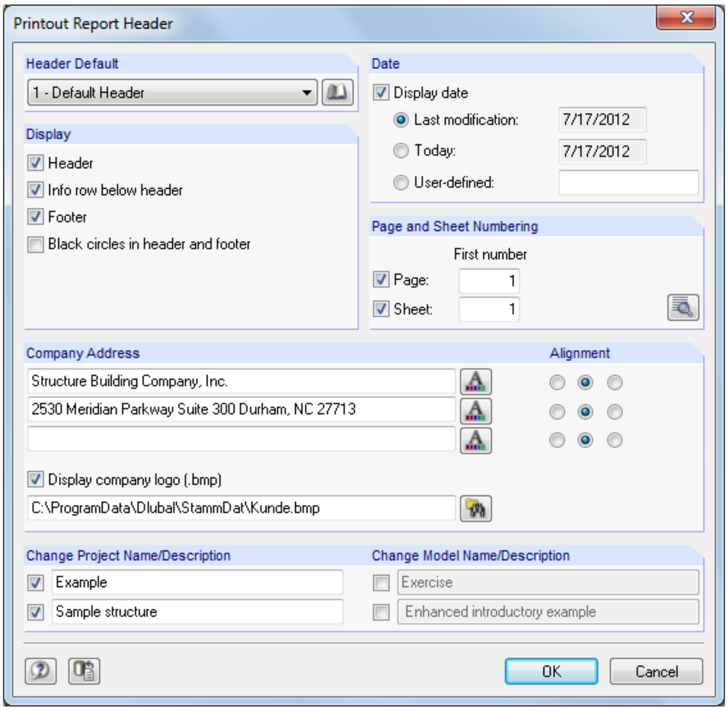


Figure 10.16: Dialog box *Printout Report Header*

Header default setting

In case several report headers are available, you can select the appropriate header in the list.

Furthermore, you can use the button [Header Library] to access different report headers. In addition, you can create, modify or delete headers in the library.

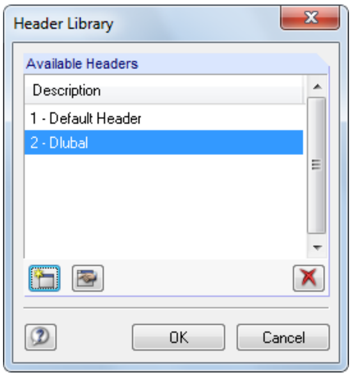


Figure 10.17: Dialog box *Header Library*

The buttons in the *Header Library* have the following meanings:

	A new header is created. Enter the specifications in another dialog box whose structure is similar to the one of the *Header* dialog box (see Figure 10.16).
	It is possible to edit the properties of the selected printout report header.
	The header that is selected in the list will be deleted.

Table 10.4: Buttons of the dialog box *Header Library*

The headers of the printout report are normally stored in the file **DlubalProtocolConfigNew.cfg** that you find in the general master data folder *C:\ProgramData\Dlubal\Global\General Data*. The file won't be overwritten during an update. Nevertheless, saving a backup file may be useful.

Display

Settings in this dialog section determine the header elements or the page layout shown.

The option *Info row below header* activates and deactivates the display of project and model data, with or without date (see below). The project description is taken from the project's general data filed in the Project Manager (see chapter 12.1.1, page 546). The model description is taken from the base data of the model (see chapter 12.2, page 554). It is possible to adjust the default specifications for the printout in the dialog sections *Change Project Name/Description* and *Change Model Name/Description* (see below).

The *Footer* can be switched on and off as well as the *Black circles* in the points of intersection of boundary line with header and footer line.

Date

RFEM provides automatic default settings and a *User-defined* specification option for displaying the date in the printout report header.

Page and sheet numbering

If *Page* and *Sheet* have the same initial numbers and the *Display* check boxes are ticked, there is no difference in numbering. But if you want to assign several pages to a sheet, it is possible to enter detailed specifications for the numbering by means of the [Settings] button shown on the left.

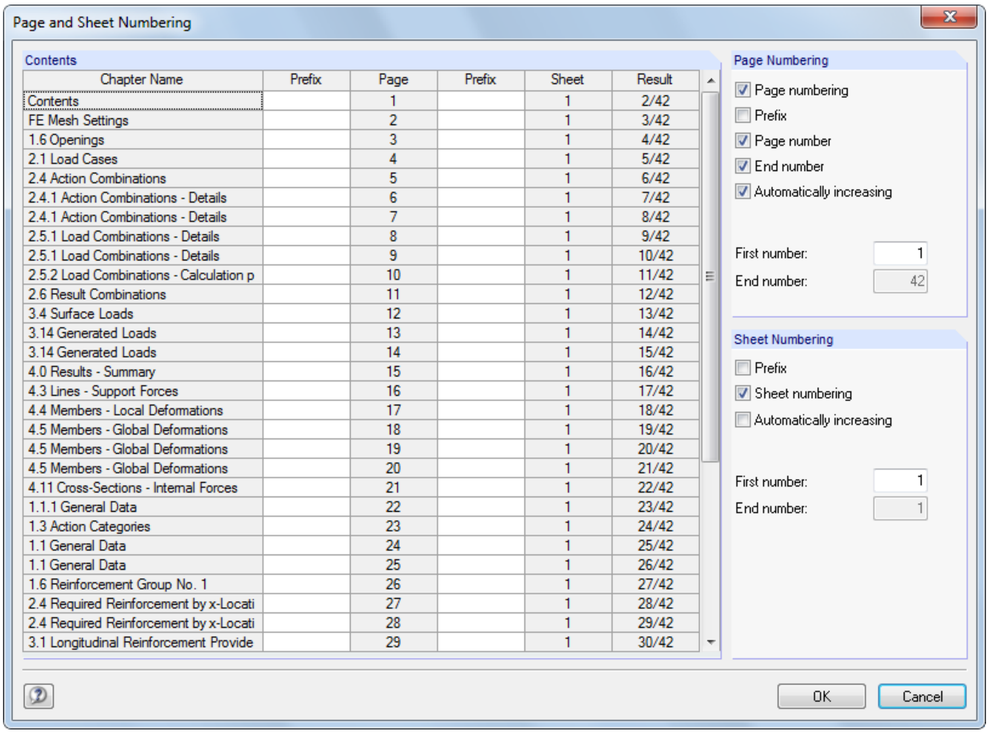


Figure 10.18: Dialog box *Page and Sheet Numbering*

Use this dialog box to decide if a *Prefix* is applied in front of the *Page numbering*. The prefix may be an abbreviation that is defined by chapter, indicating for example all model data in the numbering by a prefixed "MO". In addition, you can decide if the *End number* is included, for example "Page: MO3/25".

Use the two check boxes *Automatically increasing* in the dialog sections to the right to define a continuous numbering. Moreover, you can specify the *First number* for the page and sheet numbering. The table column *Result* shows the result of all specifications dynamically.

Company address

This dialog section of the dialog box *Printout Report Header* contains information from customer data that can be adjusted. A separate input field is available for each of the three rows of the report header. Use the [A] button shown on the left to change font and font size. The *Alignment* of rows can also be defined separately.

The left zone in the header is reserved for the company logo. The image must be available in a bitmap file format (MS Paint for example saves graphics as *.bmp).

To save the modified settings, click the button [Set Header as default] at the bottom of the dialog box. The dialog box *Header Template Name* opens where you have to enter a description. Then, the new report header will appear as *Header Default* on the top of the printout.

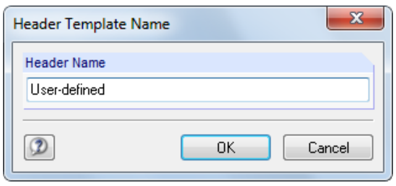


Figure 10.19: Dialog box *Header Template Name*

Change project/model name/description

In both dialog sections, the project and the model name including user-defined descriptions are preset. To modify the presettings, tick the check boxes in front of the relevant name. In this way, the input fields become accessible for new entries that appear in the printout later.

10.1.5 Insert RFEM Graphics

Every picture displayed in the work window can be integrated into the printout report. Furthermore, it is possible to include result diagrams of sections, members and line supports as well as cross-section details in the report by using the [Print] button in the respective dialog boxes.

To print the currently displayed graphic,

select **Print Graphic** on the **File** menu

or use the toolbar button shown on the left.

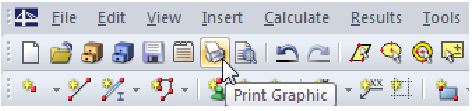


Figure 10.20: Button *Print Graphic* in the toolbar of the work window

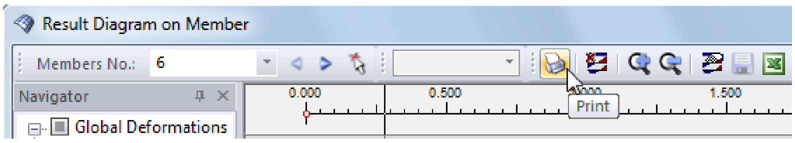


Figure 10.21: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog box appears:

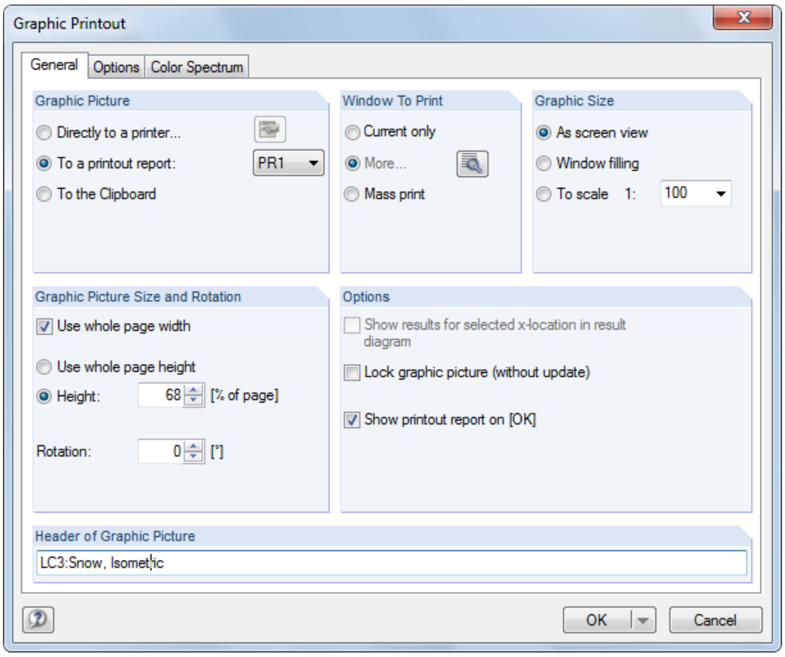


Figure 10.22: Dialog box *Graphic Printout*, tab *General*

Graphic picture

In the dialog section *Graphic Picture*, select the option *To a printout report*. If several printout reports are available, you can select the number of the target report in the list to the right.

Options

Lock graphic picture

Standard of RFEM is the generation of dynamic graphics: When model or results are modified, the graphics in the printout report will be updated automatically. If problems of performance occur in the report because of the graphics, you can stop the dynamic adjustment by ticking the check box for *Lock graphic picture (without update)* in the dialog section *Options*.

Of course, it is possible to unlock a graphic in the printout report: Right-click the graphic item in the report navigator to open its context menu (see Figure 10.4, page 382). Select *Properties* to access again the dialog box *Graphic Printout* for the picture. You can also mark the graphic in the report navigator and select *Chapter Properties* on the *Edit* menu.

The lock buttons in the toolbar of the printout report provide more functions to classify graphics as static or dynamic (see Figure 10.4, page 382). The buttons have the following functions:

	Refreshes all graphics
	Unlocks all graphics which can then be updated dynamically
	Locks all graphics which are then fixed in the printout report

Table 10.5: Graphic buttons in the printout report

Show printout report on [OK]

Normally, when closing the dialog box with [OK], the printout report opens so that you can check the print results. This may be annoying, for example when you want to take several graphics one after the other to the printout report. After clearing the check box, it is possible to print pictures without waiting time for the creation of the printout report.

The remaining functions and tabs of the dialog box *Graphic Printout* are explained in chapter 10.2 on page 404.

10.1.6 Insert Graphics and Texts

External graphics and texts can be integrated as well into the printout report of RFEM.

Graphics

To insert a picture that is not an RFEM graphic, you need to open the graphic file in an image editor first (for example MS Paint). Then, copy it to the clipboard with the keyboard keys [Ctrl]+[C].

To insert the graphic from the clipboard into the printout report,

select **Image from Clipboard** on the **Insert** menu.

You have to enter a chapter name for the new graphic before it is inserted.

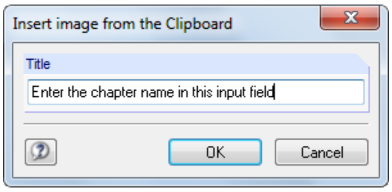


Figure 10.23: Dialog box *Insert image from the Clipboard*

The graphic will appear as a single chapter in the printout report.

Texts

Short user-defined notes can also be added to the printout report. To open the corresponding dialog box,

select **Text Block** on the **Insert** menu.

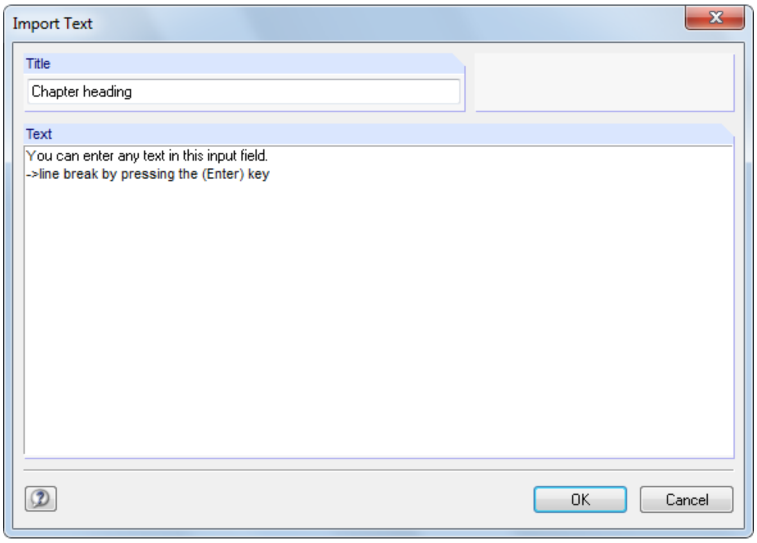


Figure 10.24: Dialog box *Import Text*

Enter the *Title* and the *Text* in the dialog box. After clicking [OK] the chapter will be inserted at the end of the printout report. Then, you can use the drag-and-drop function to move the chapter to an appropriate place in the printout report.

In the selection mode (see Table 10.3, page 383) you can modify the text subsequently by double-click. Alternatively, right-click the header in the report navigator, and then select *Properties* in the context menu.

Text and RTF files

It is possible to integrate text files available in ASCII format as well as formatted RTF files including embedded graphics into the printout report. Thus, you can save recurring texts in files to use them in the report.

Moreover, this function allows you to integrate analysis data from other design programs into the printout report, provided that the results are available in ASCII or RTF format.

To insert text and RTF files,

select **Text File** on the **Insert** menu.

First, the Windows dialog box *Open* appears where you can select the file. After clicking the [Open] button, the chapter will be added to the end of the printout report. Then, you can use the drag-and-drop function to move the chapter to an appropriate place in the printout report.

In the selection mode (see Table 10.3, page 383) you can modify the text subsequently by right-click. The dialog box *Import Text* appears for user-defined adjustments.

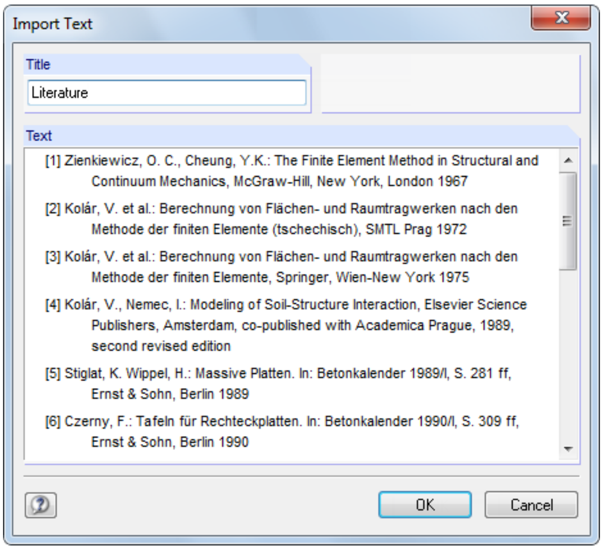


Figure 10.25: Dialog box *Import Text*

10.1.7 Printout Report Template

The selection described in chapter 10.1.3 is rather time-consuming. It is possible to save such a selection including graphics as template which you can use also for other models. Creating printout reports becomes more efficient on the basis of templates.

An existing printout report can be saved as template, too.

Create a new template

To define a new template,

point to **Printout Report Template** on the **Settings** menu of the printout report, and then select **New** or

point to **Printout Report Template** on the **Settings** menu of the printout report, and then select **New from Current Printout Report**.

New

First, the selection dialog box described in chapter 10.1.3 on page 384 opens.

Use the tabs to select the chapters that you want to print. When the selection is complete, click [OK] and enter a *Description* for the new report template.

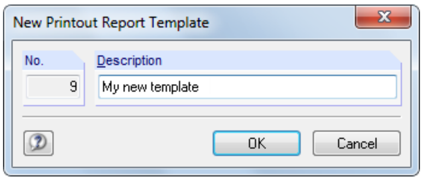


Figure 10.26: Dialog box *New Printout Report Template*

New from current printout report

The selection of the currently shown printout report is used for the new template. Enter the *Description* of the new report template in the dialog box (see Figure 10.26).

Apply a template

When a printout report is already open, you can apply the selected contents of a template to the current report. To open the corresponding dialog box,

select **Printout Report Template** on the **Settings** menu, and then click **Select**.

A dialog box opens where you can select the template from the list *Available Printout Report Templates*.

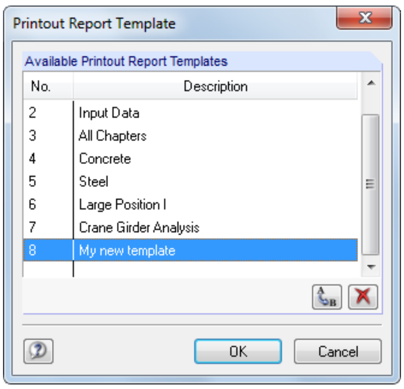


Figure 10.27: Dialog box *Printout Report Template*

Details to the buttons in this dialog box can be found in Table 10.6.

After confirming the dialog box and the subsequent security query, the current selection will be overwritten by the template.

Now, when you create a new printout report, you can select a template from the list *Printout Report Template* to apply specific settings to the new report.

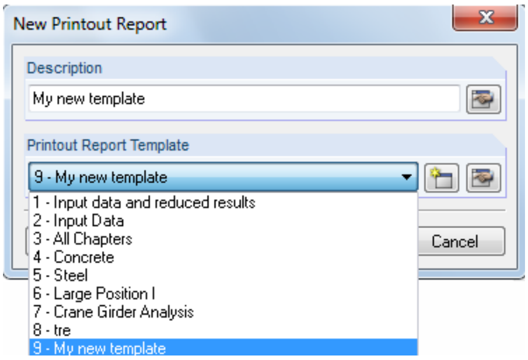


Figure 10.28: Dialog box *New Printout Report* with list of templates

Managing templates

All templates are managed in the dialog box *Printout Report Template*. To open the dialog box,

select **Printout Report Template** on the **Settings** menu, and then click **Select**.

The dialog box shown in Figure 10.27 appears. The functions of the buttons are enabled only for user-defined templates.

	The name of the selected template can be changed.
	The selected template will be deleted.

Table 10.6: Buttons in the dialog box *Printout Report Template*

The printout report templates are stored in the file **RfemProtocolConfig.cfg** that can be found in the master data folder for RFEM 5 *C:\ ProgramData\Dlubal\RFEM 5.xx\General Data*. This file won't be overwritten during an update. Nevertheless, saving a backup file may be useful.

10.1.8 Adjust Layout

The layout of a printout report can be adjusted concerning its fonts and font colors as well as its margin settings and table design.

To open the dialog box where you can edit the page layout,

select **Page** on the **Settings** menu in the printout report or

use the toolbar button in the printout report shown on the left.

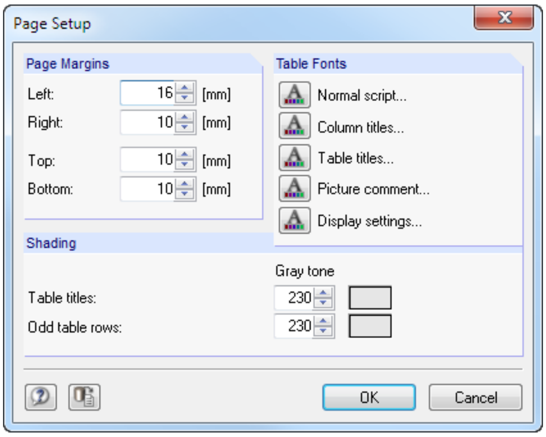


Figure 10.29: Dialog box *Page Setup*

The default fonts for table contents and table headers are relatively small. However, you should be careful with changing the **Arial** default settings: Larger fonts do not always fit in the columns.

The layout settings also apply to the printout reports of the RFEM add-on modules.

10.1.9 Create Cover

The printout report can be provided with a cover sheet. To open the dialog box where you can enter the cover data,

select **Cover** on the **Settings** menu in the printout report or

use the toolbar button in the printout report shown on the left.

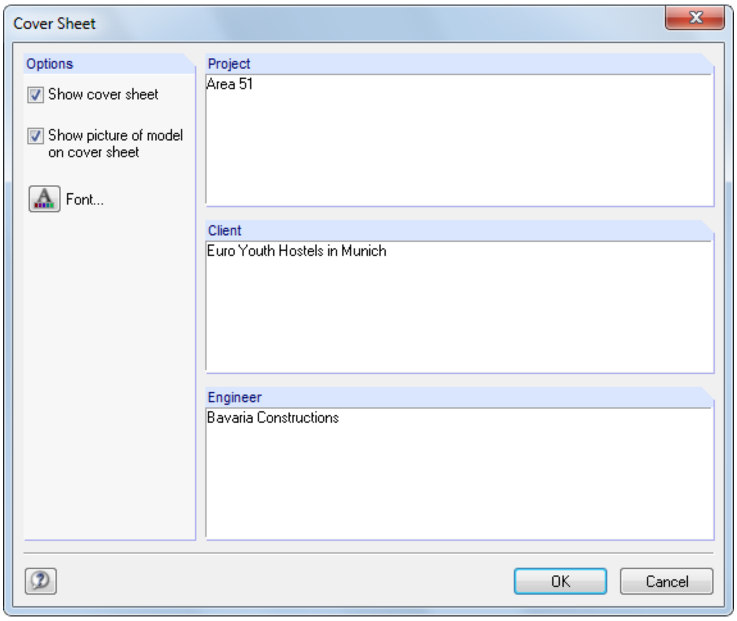
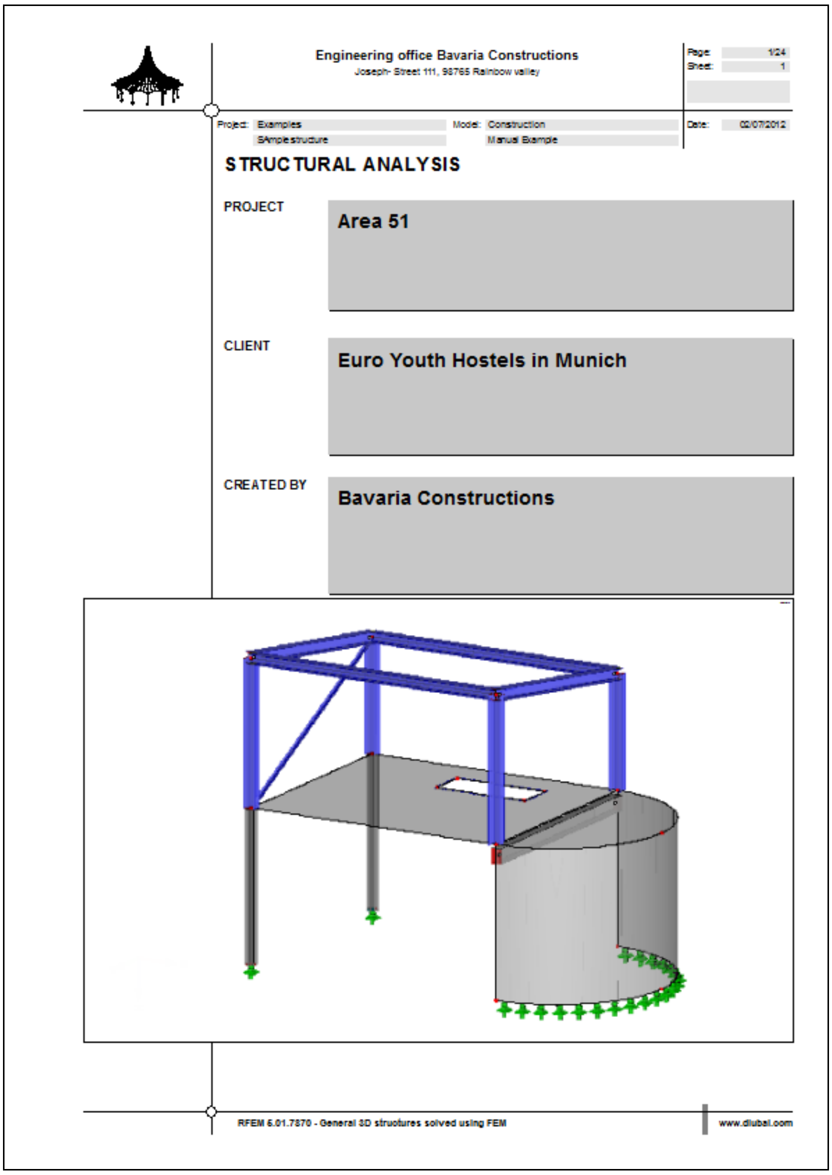


Figure 10.30: Dialog box *Cover Sheet*

When the input is complete, click [OK] to create the cover in the report.


Engineering office Bavaria Constructions
Joseph- Street 111, 98765 Rainbow valley

Page: 1/24
Sheet: 1

Project: Examples
SAmple structure

Model: Construction
Manual Example

Date: 02/07/2012

STRUCTURAL ANALYSIS

PROJECT
Area 51

CLIENT
Euro Youth Hostels in Munich

CREATED BY
Bavaria Constructions

RFEM 5.01.7870 - General 3D structures solved using FEM

www.dlubal.com

Figure 10.31: Cover sheet in the printout report

The contents of the cover sheet can be modified once again by a double-click in the selection mode (see Table 10.3, page 383). Alternatively, right-click the cover sheet in the report navigator and select *Properties* in the context menu.

10.1.10 Print the Printout Report

To start the printing process,

select **Print** on the **File** menu in the printout report

or use the button in the printout report toolbar shown on the left.

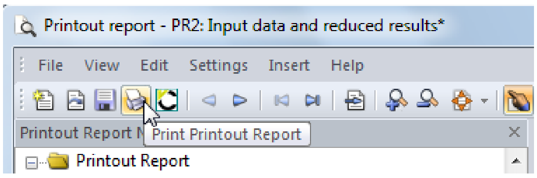


Figure 10.32: Button *Print Printout Report*

The dialog box for the printer set by default in Windows opens. Select the printer and determine the pages that you want to print.

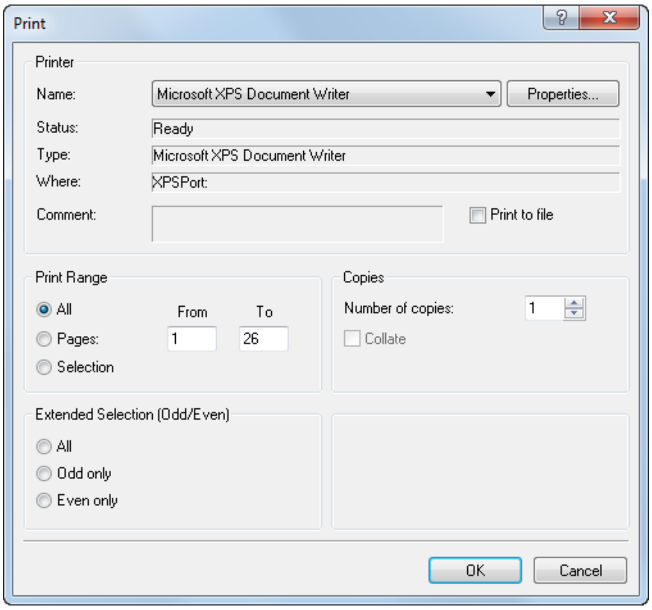


Figure 10.33: Dialog box *Print*

If you choose another printer than the default printer, the page break and therefore the page numbers printed on the paper might be different from the print preview in RFEM.

When you select the option *Print to file*, you can create a print file in PRN format that can be sent to the printer via the **copy** command.

10.1.11 Export Printout Report

The printout report can be exported in different file formats. It is also possible to export it directly to *VCmaster*.

RTF export

All common word processing programs support the RTF file format. To export the printout report including graphics as RTF document,

select **Export to RTF** on the **File** menu.

The Windows dialog box *Save As* opens:

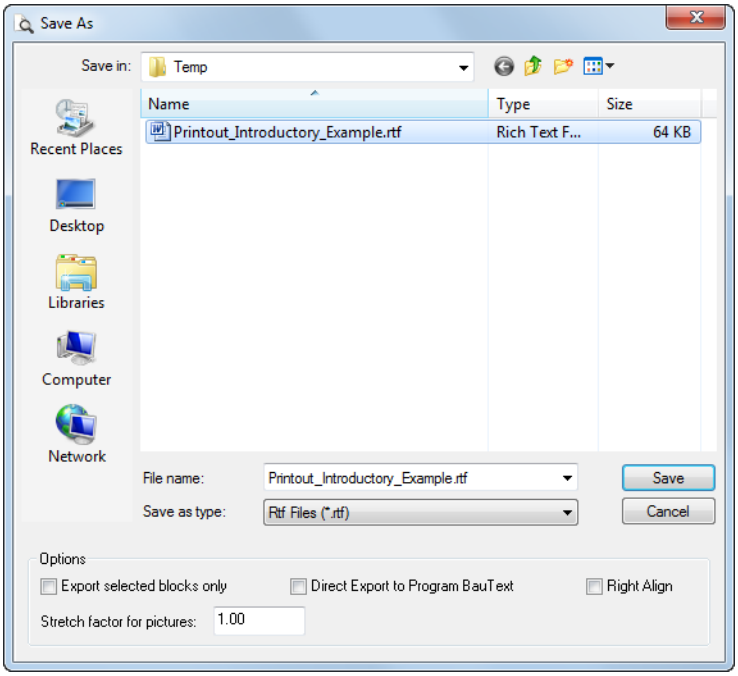


Figure 10.34: Dialog box *Save As*

Enter the storage location and the file name. If you tick the check box for *Export selected blocks only*, only the chapter(s) previously selected in the navigator will be exported instead of the entire report.

PDF export

The PDF print device integrated in RFEM makes it possible to put out report data in a PDF file. To open the corresponding dialog box,

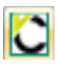
select **Export to PDF** on the **File** menu.

The Windows dialog box *Save As* opens (see Figure 10.34) where you enter the storage location and the file name. In the dialog section *Description* below, you can enter notes for the PDF file.

Moreover, the PDF file is created with bookmarks facilitating the navigation in the digital document.

VCmaster export

VCmaster from the VEIT CHRISTOPH company (formerly *BauText*) is a word processing program with specific extras for structural calculations.



To start the direct export to *VCmaster*,

select **Export to RTF** on the **File** menu

or use the button [Export to VCmaster] in the printout report toolbar shown on the left.

The dialog box shown in Figure 10.34 appears where you have to tick the check box for *Direct Export to Program VCmaster*.

It is not necessary to enter a file name, but *VCmaster* should run in the background. To start the import module of *VCmaster*, click [OK].

10.1.12 Language Settings

The language in the printout report can be set independently of the language that is used in the graphical user interface of RFEM. Thus, you can create for example a German or Italian printout report when you are working with the English program version.

Changing the language for printout

To change the language used in the printout report,

select **Language** on the **Settings** menu of the printout report.

A dialog box opens where you can select the report language from the list.

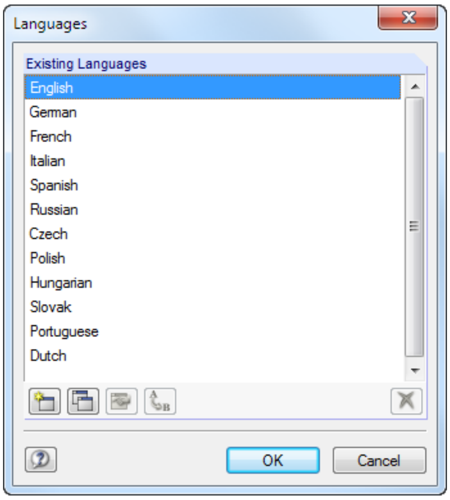


Figure 10.35: Dialog box *Languages*

Adding a language to the list

The expressions used in the printout report are stored in strings. Thus, adding new languages is rather easy.

First, open the dialog box *Languages* by

selecting **Language** on the **Settings** menu of the printout report.

In the lower part of the dialog box (Figure 10.35), you see some buttons used to manage the languages.

Create a new language

Click the button shown on the left to open the dialog box below. Specify the *Name* of the new language and select the *Language group* from the list so that the character set will be interpreted correctly.

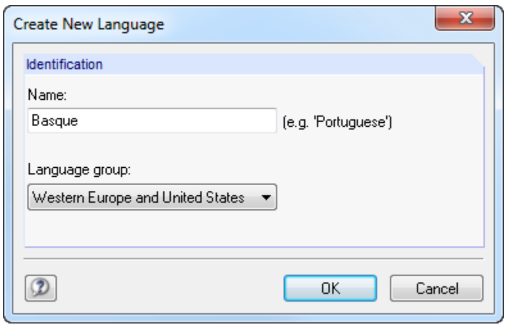


Figure 10.36: Dialog box *Create New Language*

Click [OK] to confirm the dialog box. The new language is now available in the list *Existing Languages*.

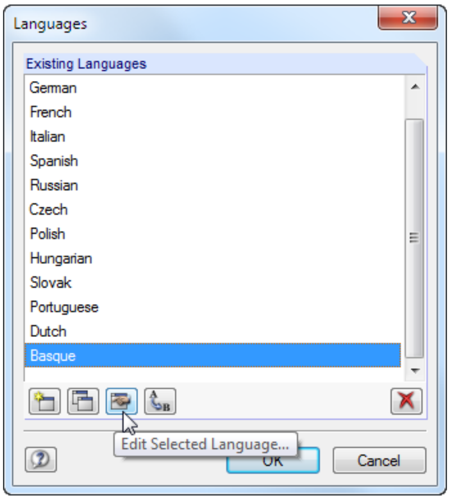


Figure 10.37: Dialog box *Languages*, button *Edit Selected Language*

Use the [Edit] button to enter the strings of the new language.

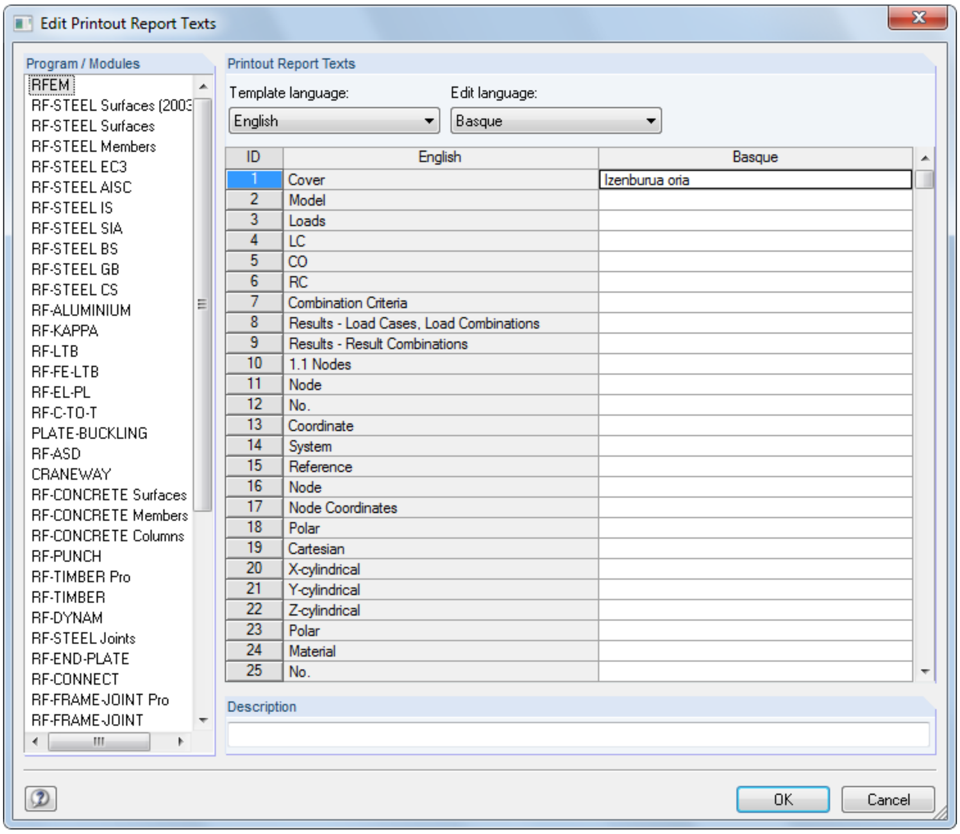


Figure 10.38: Dialog box *Edit Printout Report Texts*

Only user-defined languages can be edited.

Copy a language

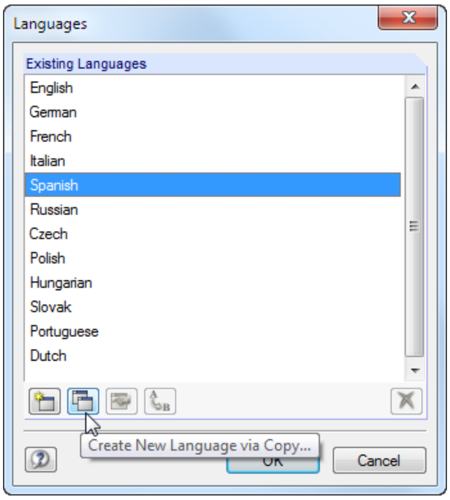


Figure 10.39: Dialog box *Languages*, button *Create New Language via Copy*

This function is similar to the creation of a new language. The difference is that you do not create an "empty" language column (see Figure 10.38, column *Basque*) since the terms of the selected language are already preset.

Rename or delete a language

Use the remaining buttons of the *Languages* dialog box to rename or delete a language. The two functions cannot be accessed for the preset default languages but only for user-defined languages.

10.2 Direct Graphic Printout

Every graphic of the work window can be printed immediately without embedding it in the printout report (see chapter 10.1.5, page 392). Result diagrams of sections, members, sets of members, lines and line supports as well as cross-section details can also be sent directly to the printer by using the [Print] button offered in the respective windows.

To print the currently displayed graphic directly,

select **Print Graphic** on the **File** menu

or use the toolbar button shown on the left.

Figure 10.40: Button *Print Graphic* in the toolbar of the main window

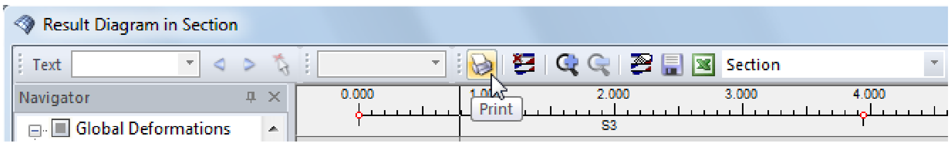


Figure 10.41: Button *Print* in the toolbar of the *Result Diagram* window

A dialog box with several tabs appears which are described in the following chapters.

10.2.1 General

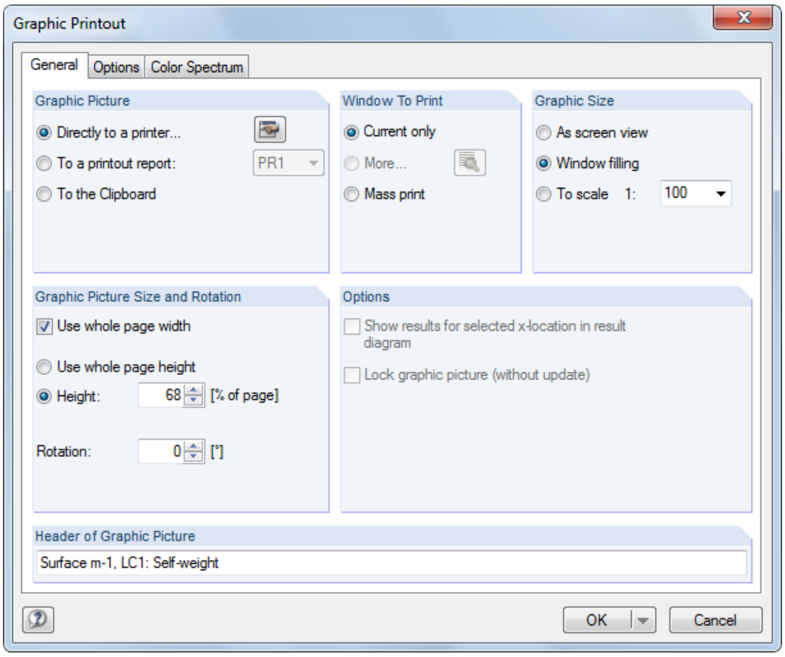


Figure 10.42: Dialog box *Graphic Printout*, tab *General*

Graphic picture

You have three options for graphical output: You can send the picture

- directly to a printer
- to a printout report (see chapter 10.1.5, page 392)
- to the clipboard.

The *Clipboard* makes the graphic available to other programs where it can usually be imported by selecting **Insert** on the **Edit** menu.

The option *Directly to a printer* results in the direct printout. It is possible to adjust the printout report header directly by using the button [Edit Printout Header] that opens the dialog box *Printout Report Header* (see chapter 10.1.4, page 389).

Window to print

The dialog section *Window To Print* is used for defining the printout settings of multiple windows views. Select *Current only* to print the graphic of the window that is currently active (for example the right window in Figure 10.43).

Please note when printing several graphic windows (see chapter 9.8, page 368) that you can only print graphics of one and the same model. A cross-model printout is not possible.

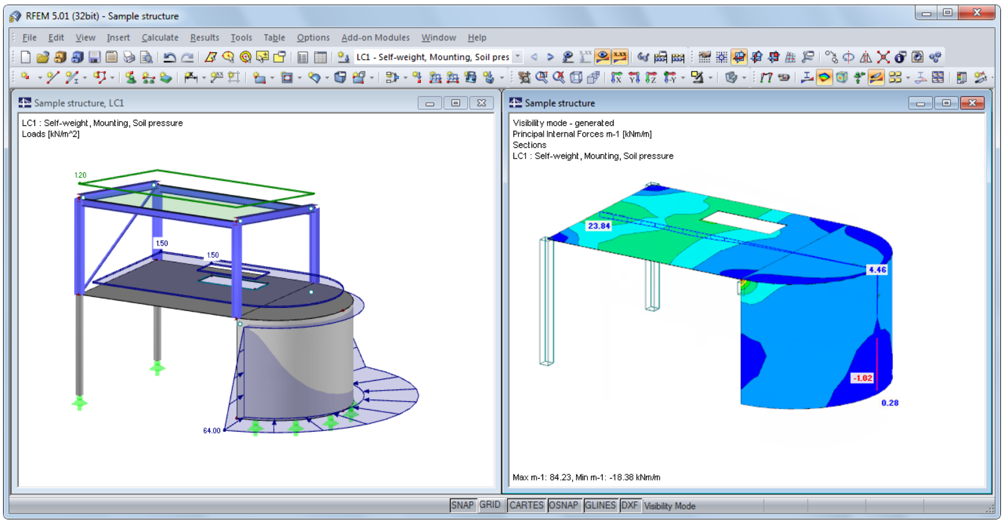


Figure 10.43: Display with two windows of same model

Select *More* to enable the button [Edit Window Arrangement] that opens a dialog box with control options for the print arrangement of graphics.

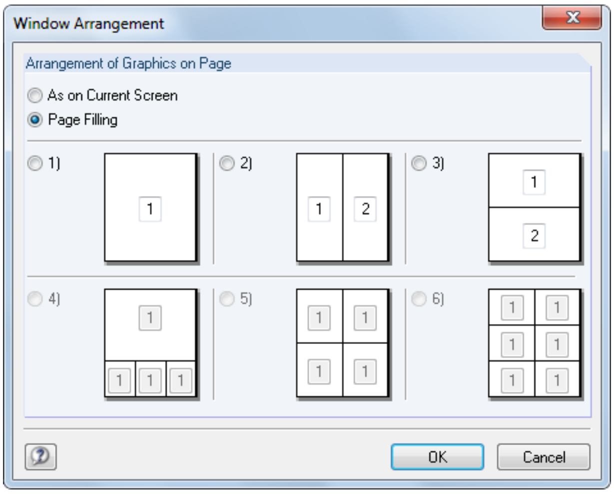


Figure 10.44: Dialog box *Window Arrangement*

Select *As on Current Screen* to arrange the windows on the printout sheet according to the proportions displayed on the screen. Then, the overall picture on the page will usually be wider than high – as it is presented on the screen. Select *Page Filling* to use the entire sheet size for the display of the windows.

With the *Mass print* option you can transfer default graphics simultaneously to the printout report. After selecting this option, a new dialog appears where you can define the parameters (see chapter 10.2.4, page 411).

Graphic size

The dialog section in the upper right corner of the dialog box *Graphic Printout* (Figure 10.42) manages the image scale of the graphic on the sheet.

If you want to use the same image size as displayed on the monitor, select *As screen view*. Take advantage of this option to print zoomed areas or special views.

The option *Window filling* prints the overall graphic on the sheet. The currently set angle of view is used to represent the whole model in the specified graphic picture size (see next dialog section).

With the option *To scale* the graphic will be printed with the scale that is selected in the list or entered manually into the input field. Again, the currently set angle of view is used. A perspective view is not suitable for the scale printout.

Graphic picture size and rotation

Settings in this dialog section define the size of the graphic on the sheet.

If the check box for *Use whole page width* is ticked, the left margin beyond the vertical separation line is additionally used for the graphic as shown in the figure below.

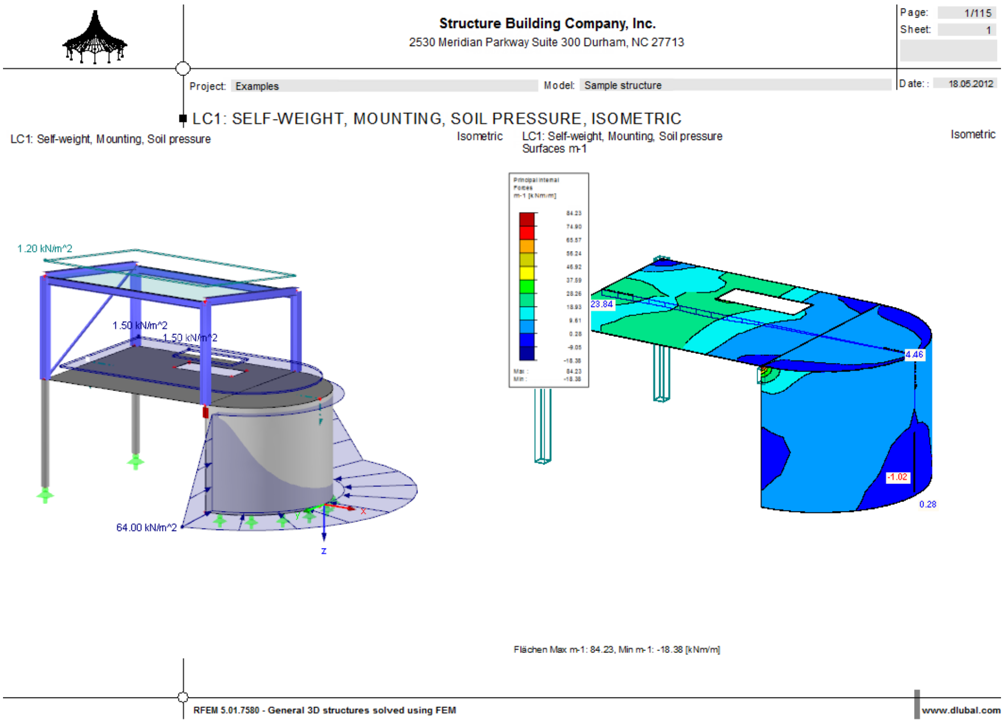


Figure 10.45: Graphic printout in landscape format: result of options *All* windows and *Use whole page width*

If you don't want to use the complete page size for the graphic, you can define the *Height* of the graphic area in percentage.

The rotation angle in the input field *Rotation* rotates the graphic for the printout.

Options

This dialog section is irrelevant for the direct printout of a work window graphic.

When printing result diagrams, you can use the check box for *Show results for selected x-location in result diagram* to decide if values appearing on the position of the vertical line will be printed (see Figure 9.20, page 356).

Header of graphic picture

When you open the dialog box *Graphic Printout*, a title is preset for the graphic. It can be modified in the input field.

10.2.2 Options

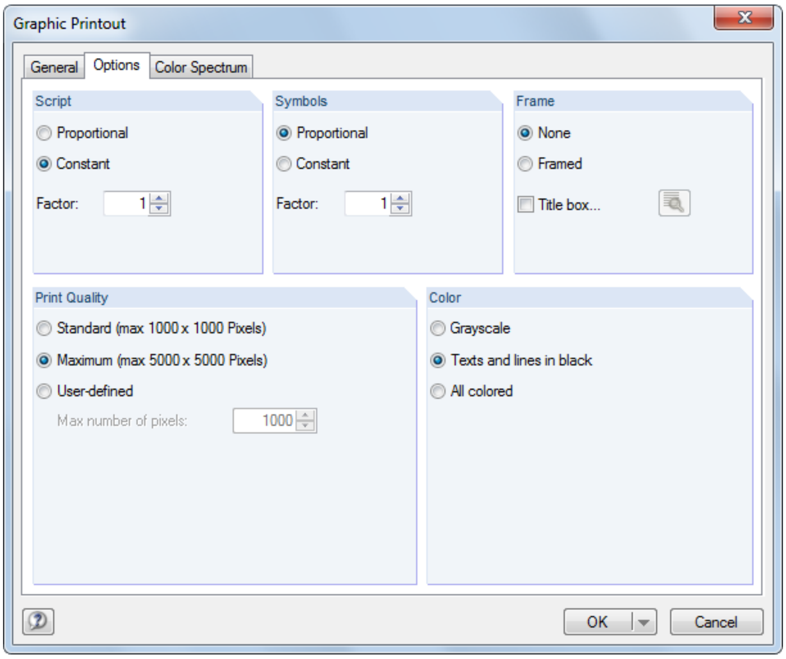


Figure 10.46: Dialog box *Graphic Printout*, tab *Options*

Script / Symbols

In most cases, it is not necessary to change the default settings in both dialog sections. For printing with plotters using large formats, however, you have to adjust the factors (see chapter 10.2.5, page 413).

The size of font and graphic symbols (nodes, supports, lines etc.) depends on the printer driver. If you are not satisfied with the printed results, scaling factors can be defined separately for *Script* and *Symbols*.

Frame

The graphic can be printed with or without frame around the graphic.

Furthermore, you have the option to add a title box to the printout. Click the button [Edit Title Box Settings] shown on the left to open the following dialog box where layout and contents of the title box can be defined. The lower part of the dialog box shows a preview.

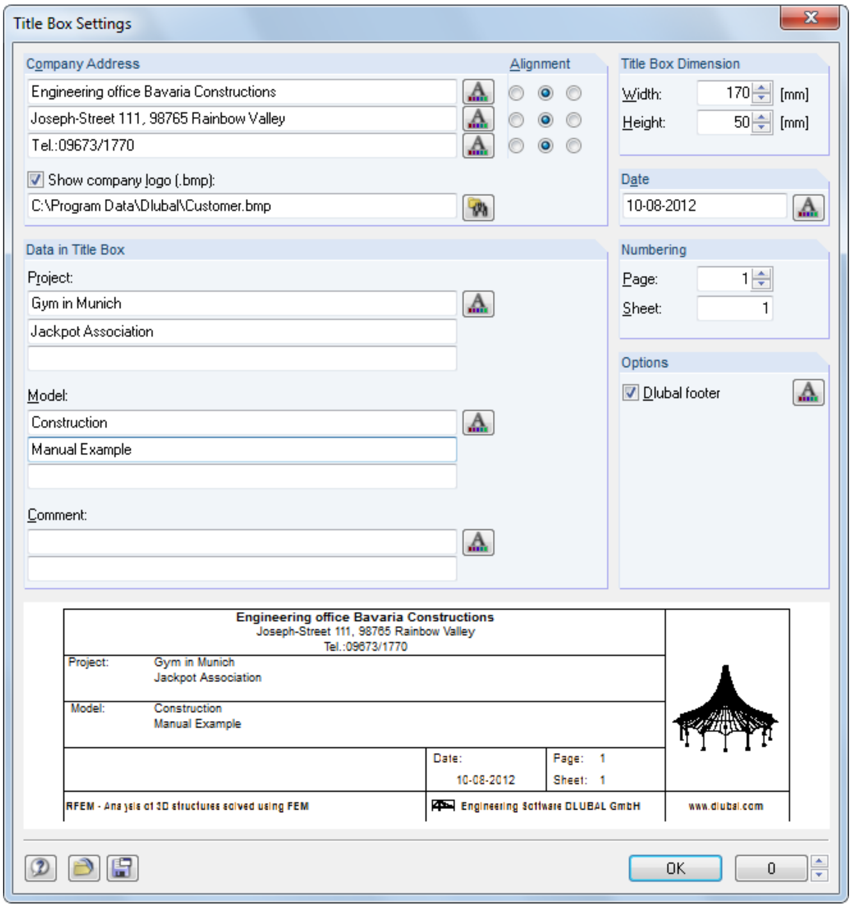


Figure 10.47: Dialog box *Title Box Settings*

Print quality

In most cases, it is not necessary to change the default settings in the dialog section *Print Quality* (see Figure 10.46). Select *Standard* to print the graphic as a bitmap file in a size of maximum 1000 x 1000 pixels. The *Maximum* size of up to 5000 x 5000 pixels together with a 32-bit color depth results in a data amount of about 100 MB. As this may cause problems for some printer drivers, be careful to select such a high resolution.

Color

When you direct the printing to a monochrome printer, you can print *Texts and lines in black* instead of gray scales to improve readability. Please note that some elements such as isobands and support symbols are not affected by the setting and therefore appear colored in the printout.

The conversion from colored result diagrams to gray scales is always managed by the printer driver. Corresponding setting options do not exist in RFEM.

10.2.3 Color Spectrum

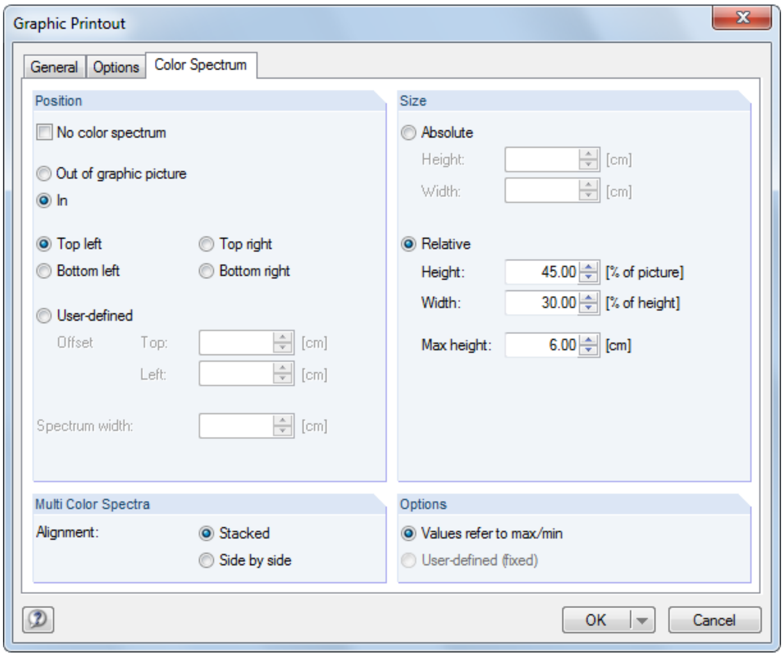


Figure 10.48: Dialog box *Graphic Printout*, tab *Color Spectrum*

The tab is only available when results are shown in a multicolor display (see chapter 9.3, page 346).

Position

The color spectrum of the control panel is usually printed into the printout. If you don't want to print it, tick the check box for *No color spectrum*.

When the panel lies *In* the graphic picture, the color spectrum overlaps a part of the picture. It is possible to specify the position of the panel: You can define it either for one of the four corners or as *User-defined* arrangement.

The option *Out of graphic picture* cuts off a strip of the graphic window and uses it only for the color spectrum. You can define the *Spectrum width* in the lower part of the dialog box.

Size

The size of the color spectrum can be defined either in absolute dimensions or relatively to the picture size.

Multi color spectra

If member and surface results are displayed together in the work window, it is possible to define the color spectrum which is relevant for the screen in the control panel (see Figure 9.50, page 377). On the sheet, however, two color spectra are displayed in this case. Their arrangement can be specified in this dialog section.

Options

The color-value assignment in the work window can be user-defined (see chapter 3.4.6, page 30).

You can determine whether the default color spectrum referring to the extreme values (*max/min*) or the user-defined color spectrum is used for the printout.

10.2.4 Mass Print

The dialog box *Mass Print* appears if you click the [Settings] button to the right of the **Mass print** option in the dialog tab *General* (see Figure 10.42, page 405). Three tabs are offered which you can use to decide which default graphics of the model, loads and results are integrated automatically into the printout report.

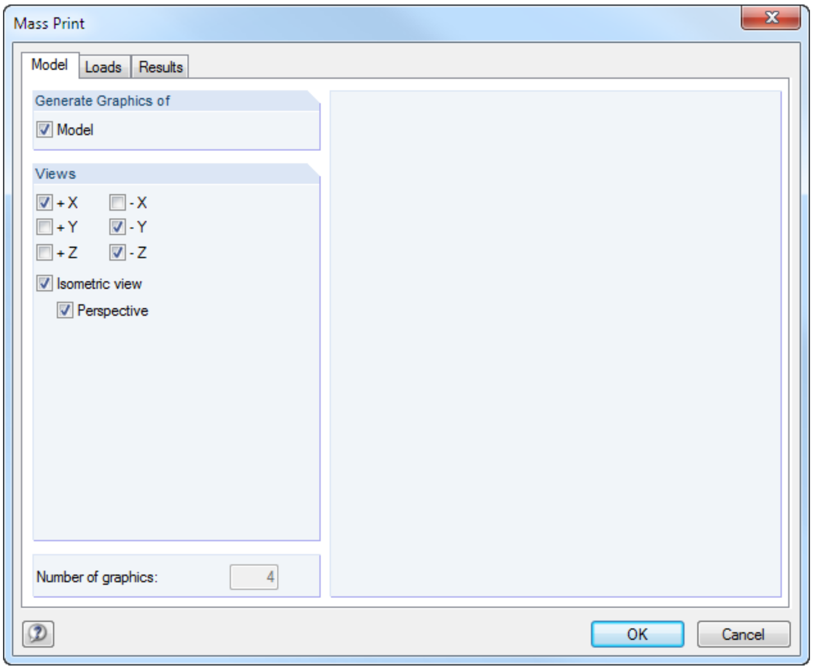


Figure 10.49: Dialog box *Mass Print*, tab *Model*

Seven standard views are available for selection. Additionally, you can activate the 3D *Perspective* for the model display.

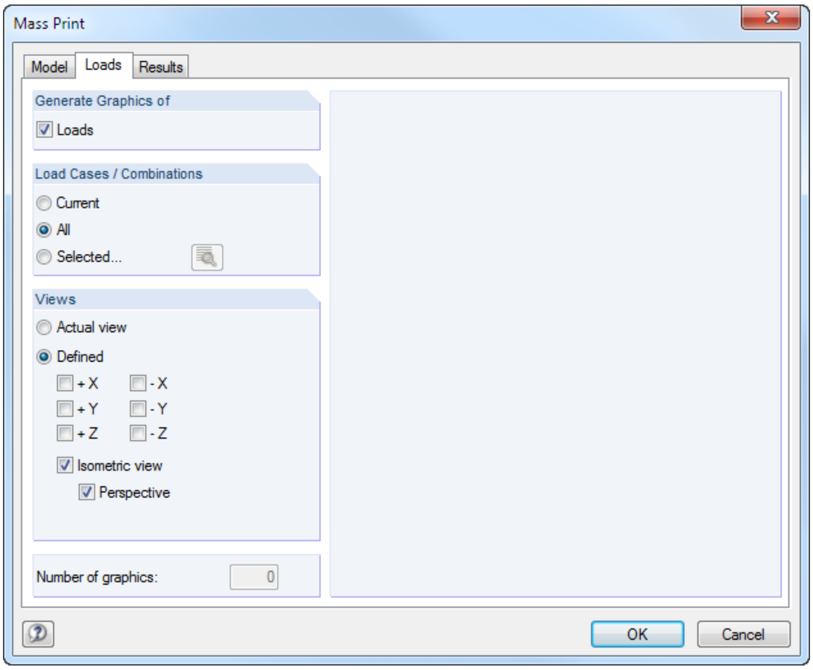


Figure 10.50: Dialog box *Mass Print*, tab *Loads*

In the dialog section *Generate Graphics of*, decide if automatic load graphics are created. Then, in the dialog section *Load Cases / Combinations*, specify the relevant load cases.

Use the [Select] button shown on the left to define *Selected* load cases in the dialog box *Load Cases / Load Combinations* (see Figure 10.52).

Finally, in the dialog section *Views*, decide which angles of view are used for the default graphics.

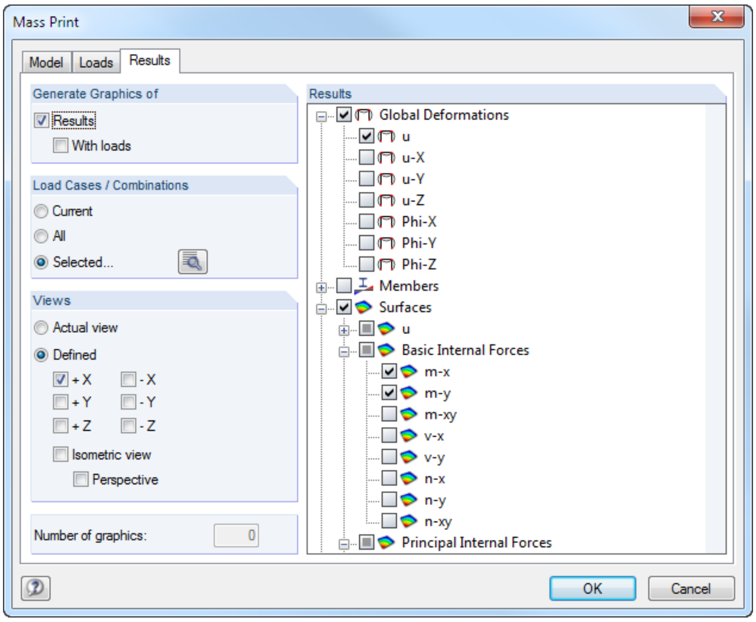


Figure 10.51: Dialog box *Mass Print*, tab *Results*

In the dialog section *Results*, you can select the relevant deformations and internal forces in the tree structure by ticking the check boxes.

With the settings in the dialog sections *Generate Graphics of* and *Load Cases / Combinations*, you decide if graphics are created with or without load representations and which load cases are relevant for the printing. Click the [Select] button shown on the left to define *Selected* load cases in a separate dialog box.

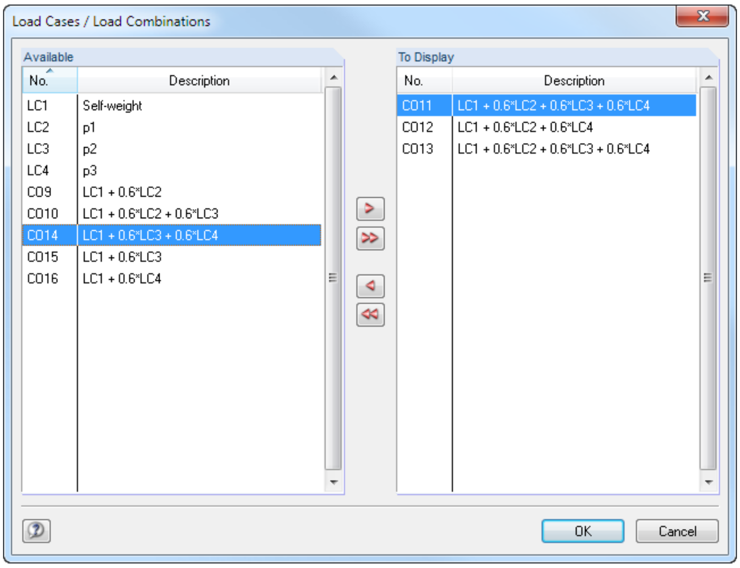


Figure 10.52: Dialog box *Load Cases / Load Combinations*

The graphics' angle of view is defined in the dialog section *Views*.

It is recommended to check the specified *Number of graphics*, in particular for the results: A small mistake during the selection may lead to a multitude of automatic graphics that slow down the creation of the report.

10.2.5 Notes for Plotting

Please note the following to achieve best results on the plotter.

Settings in operating system

Basically, RFEM uses the print system of Windows. Unlike some CAD programs, RFEM uses no special drivers to control the plotter. Therefore, the plotter must be installed as a normal printer under Windows.

Internal checks have shown that the drivers delivered by Windows are unstable and defective. Therefore, we recommend to use the original drivers or the current drivers made available on the website of the plotter producer.

When plotting huge amounts of data are processed. It is necessary to ensure that sufficient space is available on the system partition of your computer.

Do <u>not</u> set the plotter as default printer in the system. It is recommended to select the plotter not until printing is directly in process. Background: The printout report uses the standard printer driver for the print preview. Crashes occurred in the printout report with the tested plotter drivers.

Many plotter drivers offer the option to prepare the graphic in either the plotter or the computer. Generally, preparing graphics in the plotter is faster because it has a specialized processor. Besides, your work on the computer won't be affected. The problem however is that the plotter often provides only a small random access memory. If the memory is not sufficient anymore to record the picture, parts will get lost. When plotting RFEM graphics, you can see the loss in the form of missing descriptions or fillings, missing lines etc. In this case, the plotter will normally show you a corresponding message.

In case of doubt, prepare data in your computer. Please note that the default setting is set to the preparation in the plotter. In this case, adjust the printer properties accordingly.

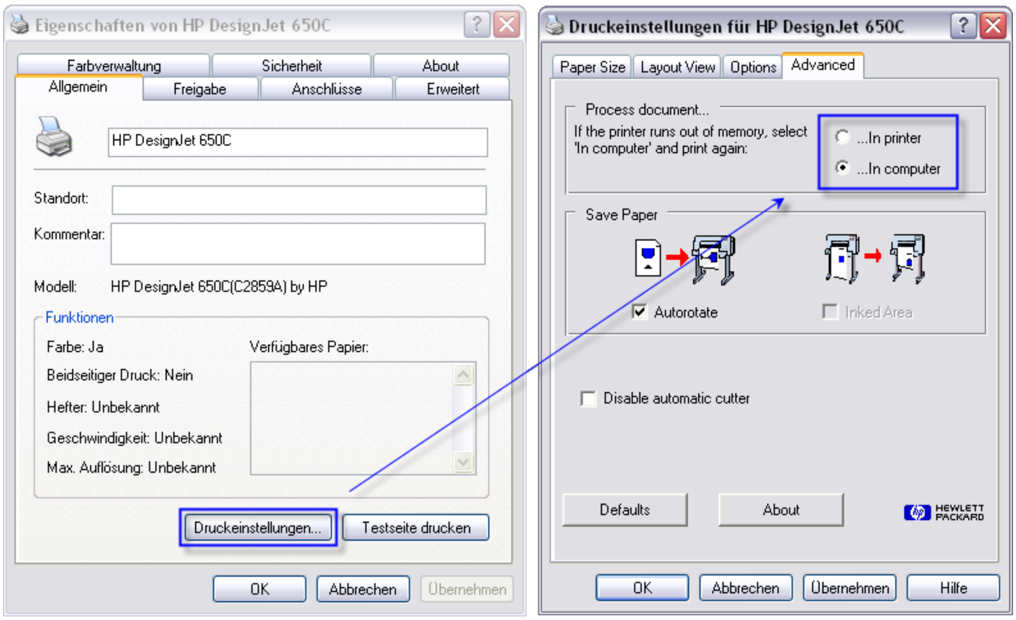


Figure 10.53: Dialog box *Printout Settings* for HP DesignJet under German Windows XP

Settings in RFEM

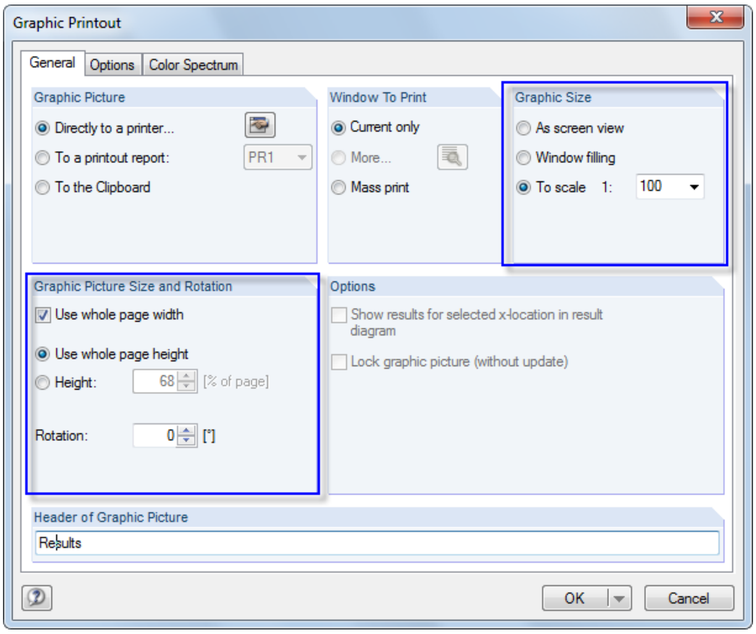


Figure 10.54: Dialog box *Graphic Printout*, tab *General*

It is recommended to select the graphic size **To scale** in the *General* tab of the dialog box *Graphic Printout* as the output on a A0 plan is almost always true to scale. Then, select the scale from the list or enter it directly into the input field.

Furthermore, it is recommended to use the complete sheet area for the plotter output: Tick the check box for **Use whole page width** in the dialog section *Graphic Picture Size and Rotation*.

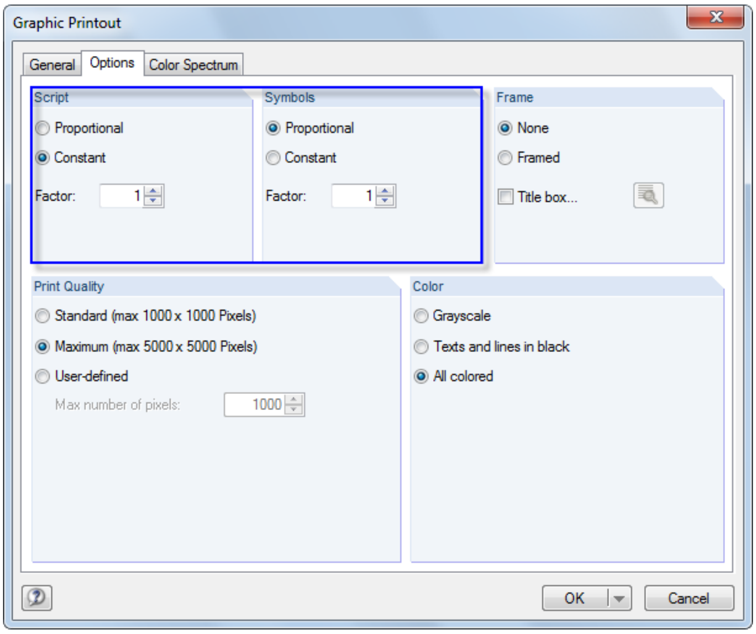


Figure 10.55: Dialog box *Graphic Printout*, tab *Options*

In the *Options* tab, you can define settings influencing the output quality. You will surely need some test plot printings to find the optimal settings. Unfortunately, it is not possible to give global recommendations because the settings' effect depends on the printer driver. The guide values shown in the figure above refer to the plotter HP DesignJet 650C with the driver version 4.62 running under Windows XP.

If crashes occur during the plotting process, we recommend to install the current printer driver as described above and to select it for the preparation of the graphic *In computer* (see dialog box for print settings). If crashes still occur when plotting, reduce the resolution gradually in the dialog section *Print Quality* in the *Options* tab of the dialog box *Graphic Printout*.

The dialog section *Script* in the *Options* tab of the dialog box *Graphic Printout* controls the scaling of the font size for numberings, dimensions and result values. Good results have been achieved for the factor 2 and the setting *Constant* for the A0 plot on the HP DesignJet 650C.

The dialog section *Symbols* does not only affect the size of support symbols, nodes or other elements but also line widths. If lines are too rough, you should reduce the factor. Good results have been achieved for the A0 plot on the HP DesignJet 650C when using a factor of 0.2 and the setting *Proportional*.

The factors set for symbols and script affect <u>all</u> fonts and symbols globally. To influence the appearance of particular objects specifically, use the settings in the dialog box *Display Properties* (see Figure 11.3, page 417). It is recommended to save adjustments for the plotter as new display configuration used for the printout report. For more information, see chapter 11.1.2 on page 417.

After clicking the [OK] button you see the *Print* dialog box of the operating system. Select the plotter from the list of printers. Click [Printer Properties] to open another dialog box where you can set page size and alignment.

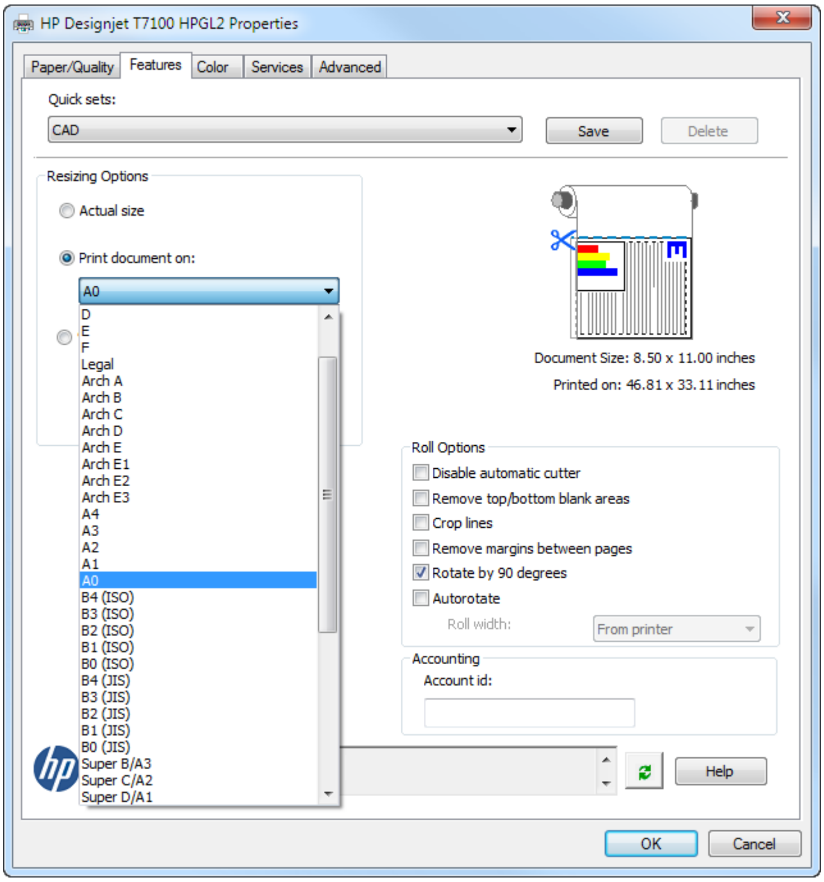


Figure 10.56: Plotter dialog box *Properties* for setting the page format

11. Tools

In the following, you find descriptions of functions for graphical and table input such as CAD tools for designing or generating model and load objects, edit options, operations in spreadsheets or parameterized input.

11.1 General Functions

This chapter describes program functions which are generally useful or provided in many dialog boxes of RFEM.

11.1.1 Language Settings

The language that has already been selected for installation is preset. Materials and cross-section tables in the libraries have also been set up by country-specific arrangements.

To change the graphical user interface of RFEM,

select **Program Options** on the **Options** menu

or use the corresponding button in the toolbar.

In the dialog tab *Program*, you can select another *Program Language* in the list.

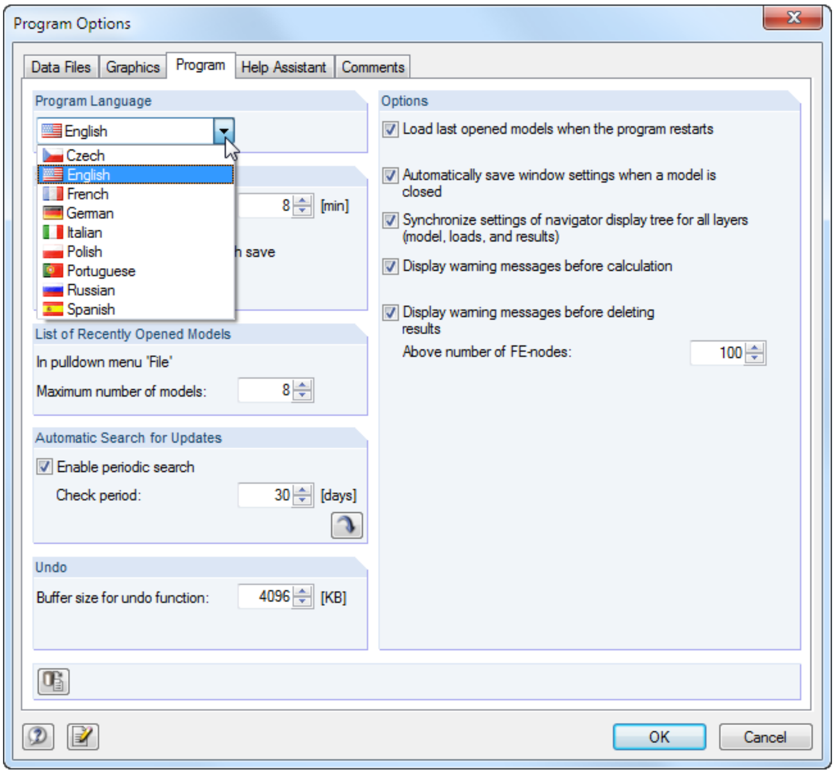


Figure 11.1: Changing the *Program Language* in the dialog box *Program Options*

The changed language settings will be effective after restarting the program.

When you change the language, please note the following:

- Some characters are only displayed correctly if the corresponding fonts are available in the operating system.
- The new language affects the arrangement of cross-section tables in the libraries.

11.1.2 Display Properties

The display properties determine the way how a graphical object is represented on the screen and in the printout. The *Display* navigator is the place to decide whether an object is represented or not (see chapter 3.4.3, page 24).

Adjust the display

To open the dialog box for adjusting the graphical display,

point to **Display Properties** on the **Options** menu, and then select **Edit**

or use the Configuration Manager (see chapter 3.4.10, page 36).

It is also possible to directly access the display properties of each graphical object (model, load or result symbol): Right-click the object to open its context menu and select the menu item *Display Properties*. Now, you can immediately adjust the object's display properties in the dialog box *Display Properties* (Figure 11.3).

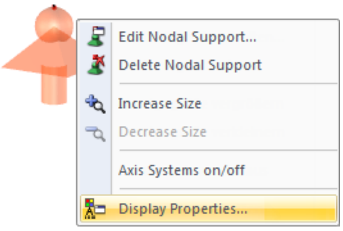


Figure 11.2: Context menu of nodal support

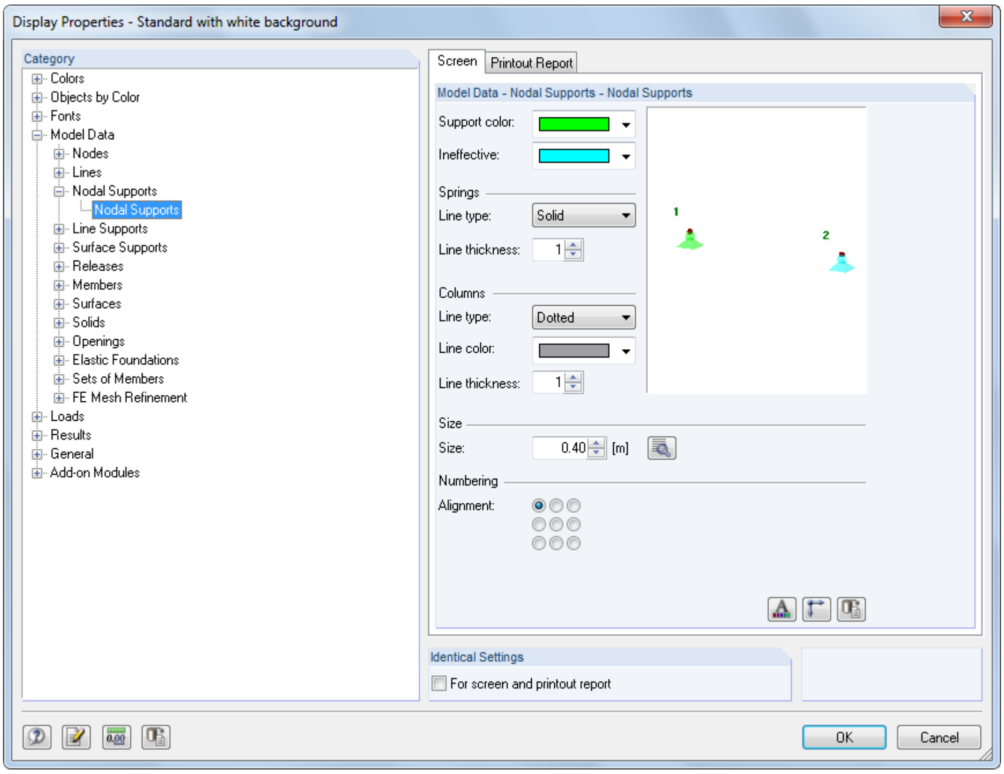


Figure 11.3: Dialog box *Display Properties* (for category *Nodal Supports*)

The settings for display on the *Screen* and in the *Printout Report* are managed in two dialog tabs. In this way, it is possible to define adjustments separately for the monitor graphic (for example size of support symbols with black background) and for the printout.

If you want to define *Identical Settings For screen and printout report*, use the check box below the tabs to synchronize the display properties for screen and printout report. If it is ticked, the settings that are defined <u>subsequently</u> are also enabled in the other dialog tab (*Screen* or *Printout Report*) of the current category. Settings that have already been defined cannot be transferred subsequently with this function.

The *Category* navigator shows the graphical objects listed in a directory tree. To change the display properties of an object, select the relevant entry. Then, adjust the object-specific display parameters in the dialog section to the right: color, line display, size in work window, type and arrangement of numbering, font, size of load vector etc.

RFEM offers additional [Details] buttons for some parameters.

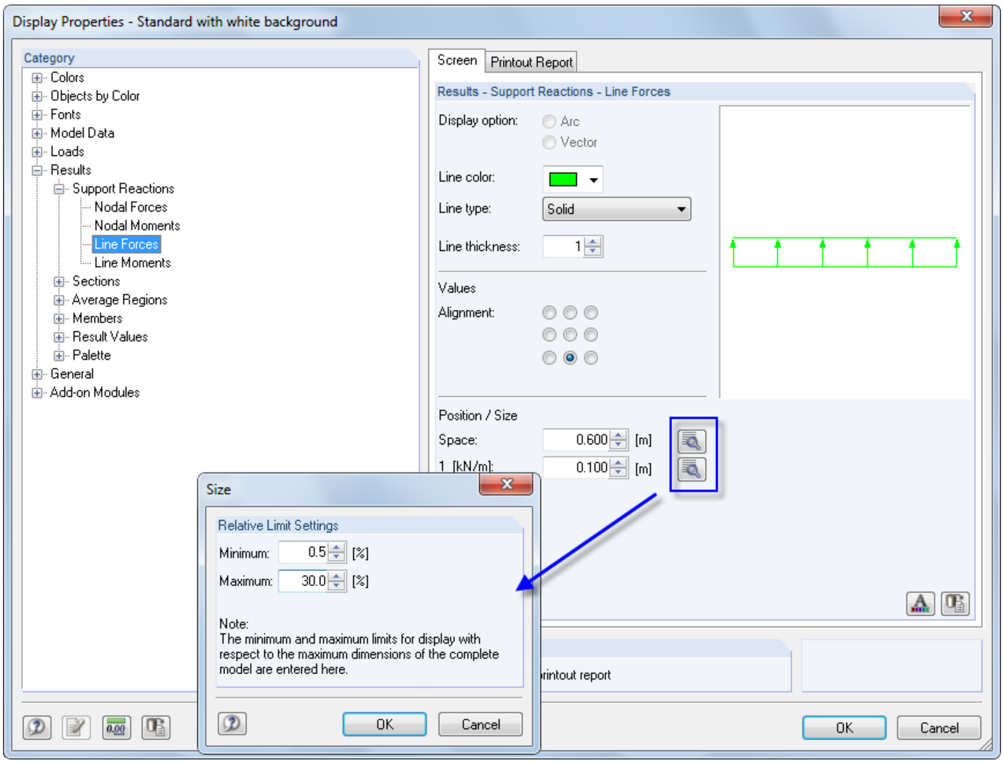


Figure 11.4: Dialog box *Size* for *Line Forces*

The buttons are used to open new dialog boxes where you can scale for example the distance or size of the object to the dimensions of the total structure.

The buttons below the parameters have the following functions:

	Opens the *Font* dialog box for changing type, size and color of font
	Goes to display parameters of axes of current object
	Returns to base data of object
	Opens the dialog box *Relative Positions* (Figure 11.5) for arranging descriptions
	Restores default settings

Table 11.1: Buttons in dialog box *Display Properties*

For objects that are relevant for lines and members it is possible to arrange the description or symbol by user-defined settings. A dialog box opens where you can define the information's position by means of a relative distance to the line or member start.

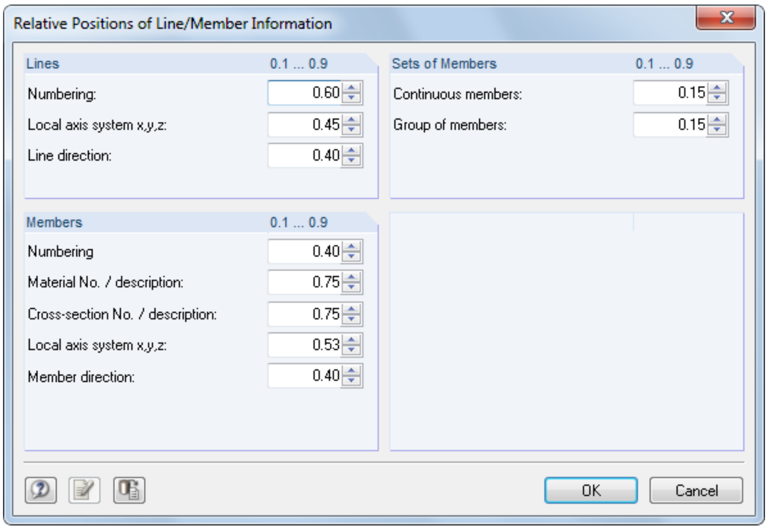


Figure 11.5: Dialog box *Relative Positions of Line/Member Information*

Save display configuration

The dialog box *Display Properties* allows you to adjust the display configuration to the given requirements. So it is possible, for example, to create different settings for the screen with colored background and for the plotter with specific settings.

However, modifications cannot be saved in the dialog box *Display Properties*. The *Configuration Manager* described in chapter 3.4.10 on page 36 is responsible for managing display configurations.

Therefore, proceed as follows when you want to create a new display profile based on your modifications:

- Confirm the modifications in the dialog box *Display Properties* with [OK].
- Open the *Configuration Manager* (see chapter 3.4.10, page 36).
- Create a [New] configuration.
- Enter a description in the dialog box *New Configuration*, and then confirm with [OK].

11.1.3 Units and Decimal Places

The units and decimal places for RFEM and all add-on modules are managed in one dialog box. The settings can be modified as required for modeling or evaluation. All numerical values will be converted or adjusted.

Changing units and decimal places

Many dialog boxes provide the button shown on the left which you can use to access the dialog box for changing units and decimal places (see Figure 11.4 for dialog box *Display Properties*).

To open the dialog box *Units and Decimal Places*, you can also

select **Units and Decimal Places** on the **Edit** menu.

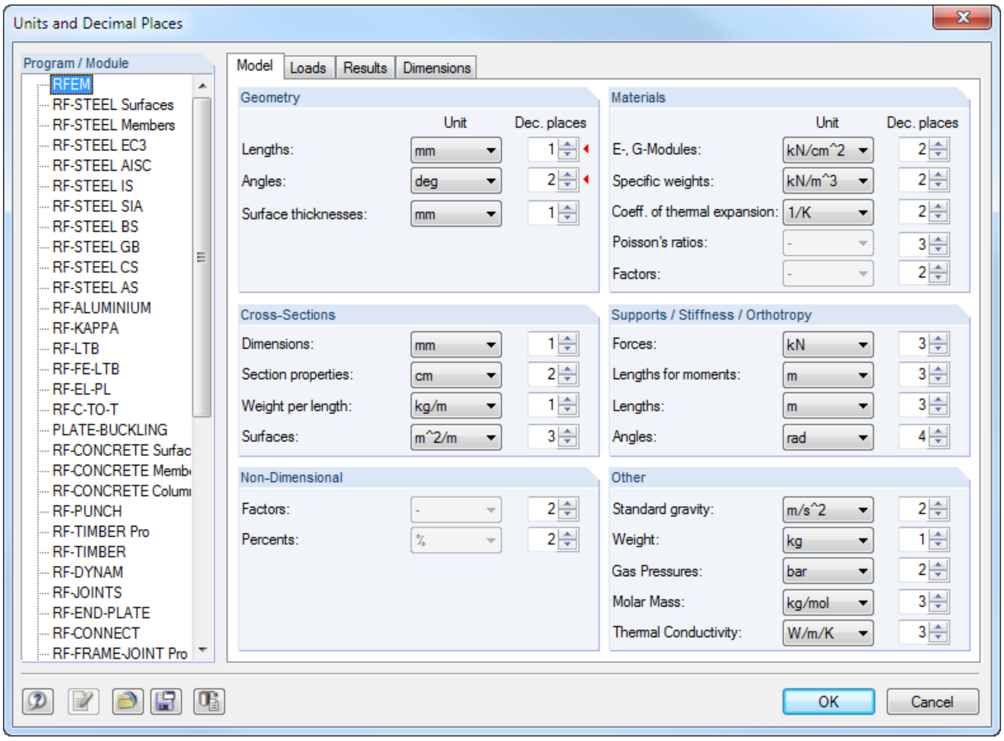


Figure 11.6: Dialog box *Units and Decimal Places*

First, select the module in the dialog section *Program / Module* for which you want to adjust the units or decimal places. Depending on the selection, the right side of the dialog box is changing.

Four dialog tabs are offered for RFEM so that you can specify settings separately for *Model*, *Loads* and *Results* data as well as *Dimensions*. For some add-on modules the right part of the dialog box is also subdivided into several tabs. The units and decimal places are summarized in groups.

When the dialog box has been opened from another dialog box (for example the *New Member* box), the relevant units and decimal places are marked with a red triangle on the right as shown in the figure above.

Save and import units as user profile

The settings in the dialog box *Units and Decimal Places* can be saved and used in other models. Thus, creating specific unit profiles for example for models consisting of steel and reinforced concrete is possible.

The button shown on the left opens a dialog box where you specify the *Name* of the new units user profile.

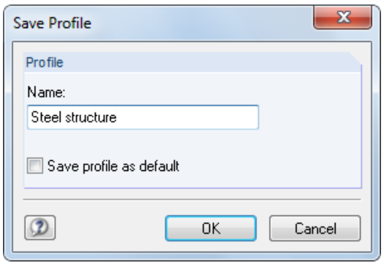


Figure 11.7: Dialog box *Save Profile*

To use this profile as default setting for new models, tick the check box *Save profile as default*.

A user profile can be imported with the button shown on the left. A dialog box opens where several profiles are available for selection. A metric and an imperial (Anglo-american) unit profile are preset as default settings.

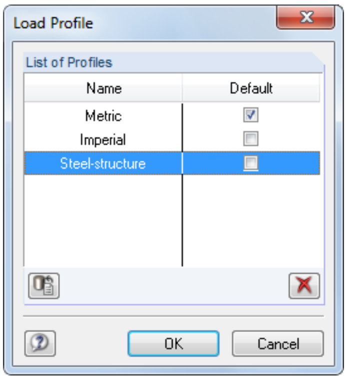


Figure 11.8: Dialog box *Load Profile*

11.1.4 Comments

This chapter describes the comment fields available in dialog boxes and tables (see for example Figure 4.12, page 43). The comments that you can insert graphically are described in chapter 11.3.6 on page 447.

Using comments

You can enter any kind of text into the comment fields. With the button [Import Comment] shown on the left you can take advantage of predefined text modules which are stored by cross-model management.

A dialog box appears showing a list of stored text modules.

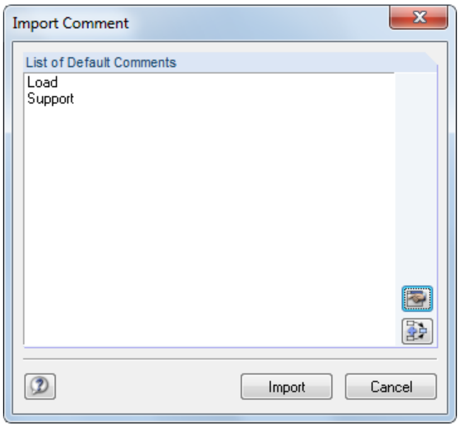


Figure 11.9: Dialog box *Import Comment*

Import

The *List of Default Comments* contains all comments that are suitable for the category. Click the [Import] button to insert the selected comment into the comment field of the dialog box. If the comment field already contains a text, it will be overwritten. Then, you may continue to edit the comment in the comment field.

Use the button shown on the left to add the selected comment to a comment field text that is already available.

Creating and managing comments

In the dialog box *Import Comment* (Figure 11.9), you can create new text modules by means of the button shown on the left. Alternatively, you can use the *Comments* tab in the *Program Options* dialog box where all comments are managed. To open the dialog box,

select **Program Options** on the **Options** menu

or use the toolbar button shown on the left.

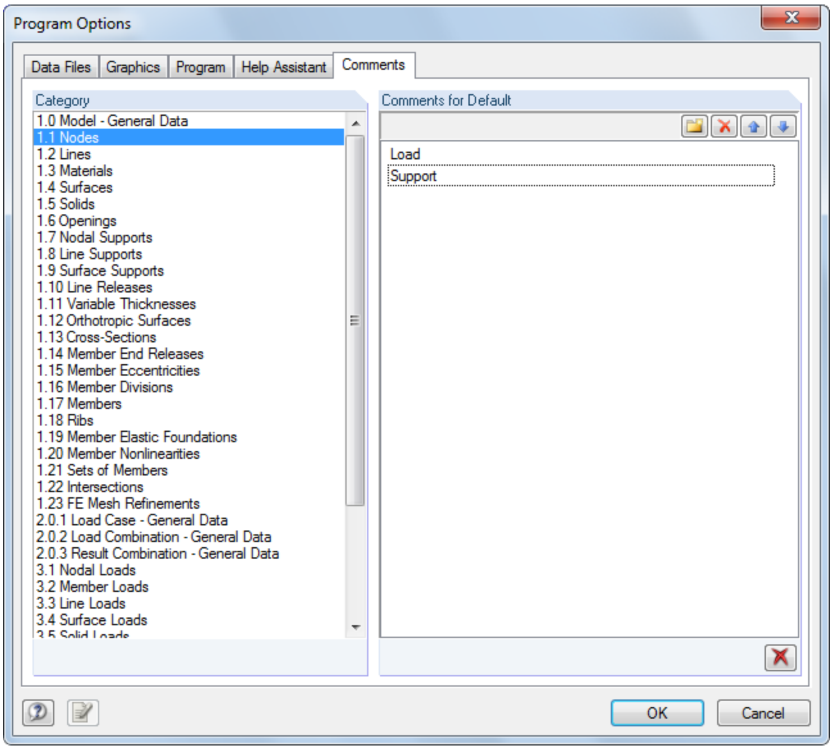


Figure 11.10: Dialog box *Program Options*, tab *Comments*

In the left dialog section *Category*, you determine the group (which means input table or input dialog box) to which you want to assign the comment text.

The right dialog section *Comments for Default* offers four buttons which have the following functions:

Button	Description
	Creates a new comment within the marked *Category*. Enter the text in the list.
	Deletes the comment that is selected in the list.
	Moves the selected comment upwards.
	Moves the selected comment downwards.

Table 11.2: Buttons in the dialog box *Program Options*, tab *Comments*

When the special selection is used (see chapter 11.2.2, page 433), you can filter data by user-defined comments.

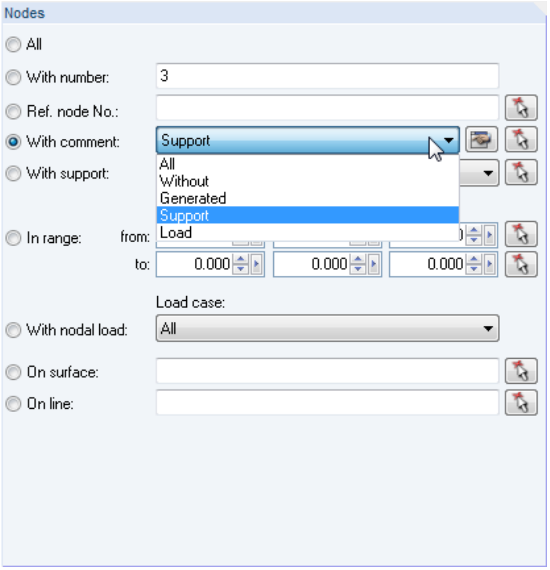


Figure 11.11: Dialog box *Special Selection* (dialog section) for *Nodes* filtered by *comment*

11.1.5 Measure Functions

In order to check entered data, distances and angles can be measured. To access the corresponding function,

point to **Measure** on the **Tools** menu.

The following measure functions are available for selection:

- Distance between 2 nodes
- Angle between 3 nodes
- Angle between 2 members
- Angle between 2 surfaces
- Angle between member and surface
- Angle between 2 lines
- Angle between member and line
- Angle between line and surface

Click the objects for measurement one after the other in the work window. Then, *Distance* and *Deformation* of the nodes are shown in a dialog box.

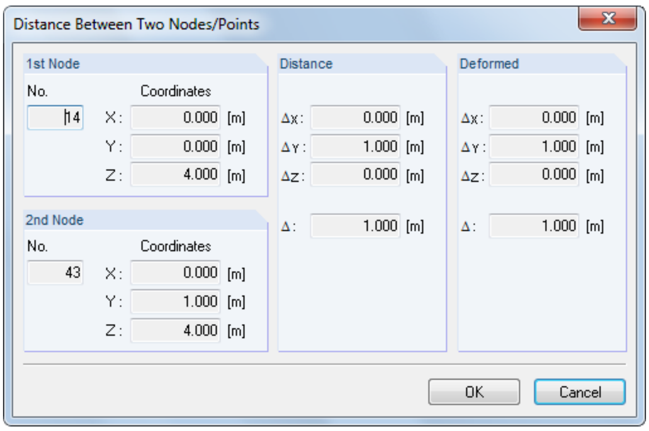


Figure 11.12: Dialog box *Distance Between Two Nodes/Points*

11.1.6 Search Functions

Selection with table

To find an object in the graphic, you can use the tables: Click into a table row and you see the relevant object highlighted with colors in the work window. Take advantage of this function for rather simple models to detect objects fast and easily in the graphic.

The graphical selection with the table works only if the synchronization of the selection is active (see chapter 11.5.4, page 486).

Searching by object number

In RFEM you can specifically search for objects, which is especially recommended for large and complex models. To access the search function,

select **Find via Number** on the **Edit** menu.

The following dialog box appears:

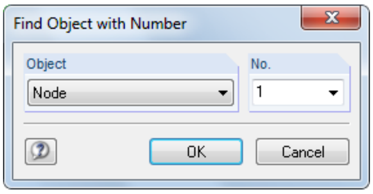
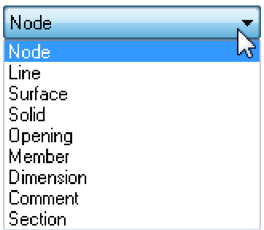


Figure 11.13: Dialog box *Find Object with Number*



In the dialog section *Object*, use the list to define the object category for searching: node, line, surface, solid, member or finite element. Then, enter the *No.* of the object directly into the in-put field to the right, or use the list to select a number.

Click [OK] to confirm the dialog box. Then, you see a big arrow indicating the object in the work window. The arrow will still be displayed when you adjust the area around the object appropriately by zooming or rotating the model. The arrow will disappear by a click into the workspace.

11.1.7 Viewpoint and View Angle

RFEM offers the standard views [in X/Y/Z] direction and [in Reverse X/Y/Z] direction as well as the [Isometric View] that can be selected by means of the buttons shown on the left. More buttons for user-defined coordinate systems and angles of view are available in the list button of the toolbar and in the *Views* navigator (see chapter 9.9.1.1, page 370).

If these views including rotating option (use toolbar button [Move] and hold down [Ctrl] key) do not result in the display view that you want to set, you can use the extended options of the dialog box *Edit Viewpoint*.

To open the dialog box,

select **Viewpoint** on the **View** menu.

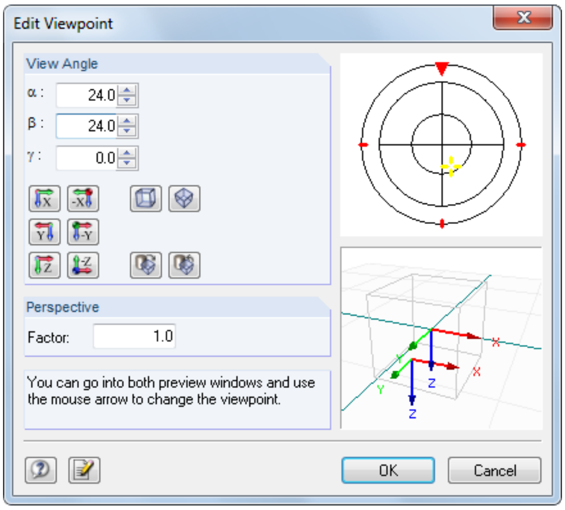
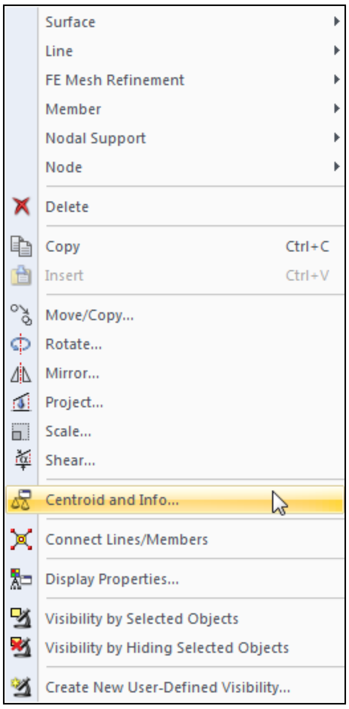


Figure 11.14: Dialog box *Edit Viewpoint*

Click into the preview windows on the right and move the mouse to set viewpoint and view angle. In addition, you can adjust the factor for the *Perspective*.

11.1.8 Determination of Centroid

The centroid of the overall model is displayed automatically when the FE mesh has been generated successfully and the corresponding option in the *Display* navigator under the navigator item *General* is ticked. Color and size can be adjusted in the dialog box *Display Properties*: Click *Colors* → *Other* → *Center of Gravity* (see chapter 11.1.2, page 417).



Moreover, it is possible to determine the centroid of particular objects: Select the relevant members, surfaces and solids for example by multiple selection or by opening a selection window (see chapter 11.2, page 430). Activate the context menu shown on the left by right-clicking one of the objects. Then, click the menu item *Centroid and Info* to open a dialog box with information about the selected objects.

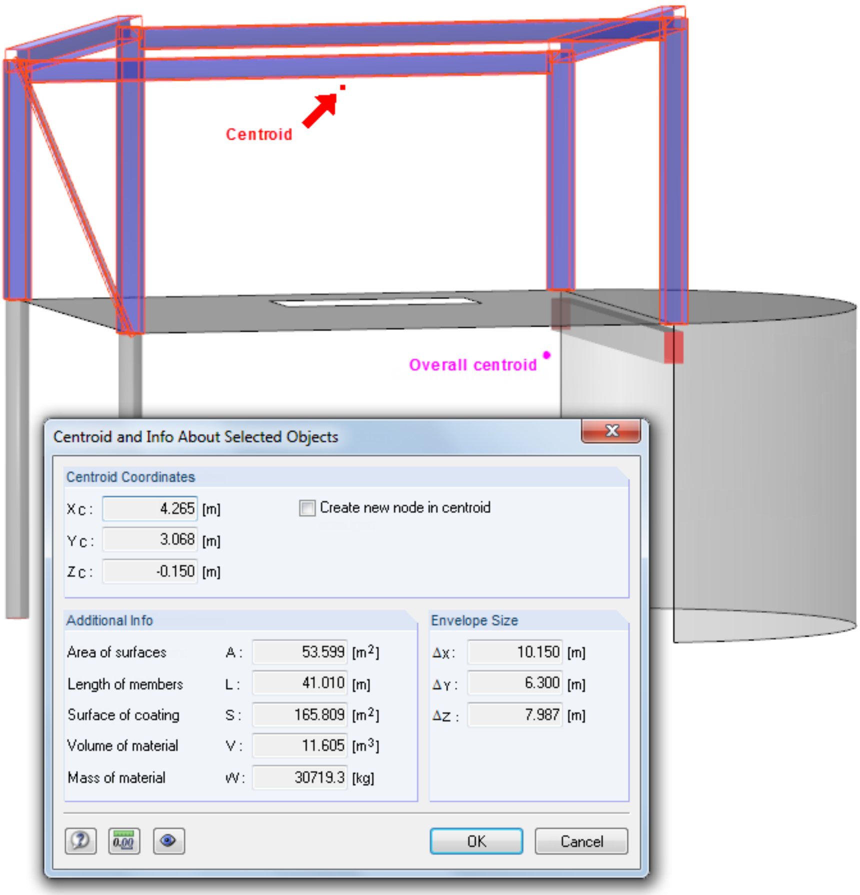


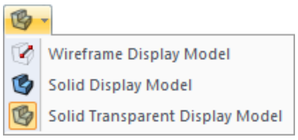
Figure 11.15: Dialog box *Centroid and Info About Selected Objects*

The dialog box shows you the *Centroid Coordinates* in relation to the origin of the global axis system XYZ. In the work window, the centroid is indicated by a big arrow. Optionally, you can *Create* a *new node in the centroid*.

In addition to the global *Envelope Size* of the selected objects, the following *Additional Info* is displayed:

- Area of all surfaces
- Length of all members
- Surface area of visible surfaces of all objects
- Net volume
- Total mass

11.1.9 Rendering



The model's representation in the work window can be set by user-defined control. Use the list button in the toolbar shown on the left to switch quickly between the display types *Wireframe*, *Solid* and *Solid Transparent Display Model*.

Detailed settings for the individual objects can be specified in the *Display* navigator under the navigator item **Rendering**.

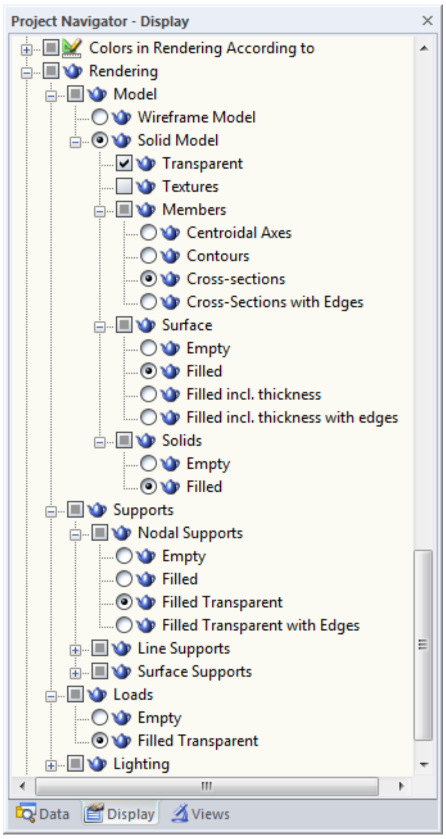


Figure 11.16: *Display* navigator with options for *Rendering* of model and load objects

The *Solid Model* representation can be adjusted individually for members, surfaces and solids. Similar control options are available for the display of supports and loads.

Textures

When *Textures* are activated, RFEM shows the surface textures in the rendered model. To access detail settings for the textures,

point to **Display Properties** on the **Options** menu, and select **Edit**.

The dialog box *Display Properties* opens where you select *Materials* in the category *Objects by Color*. Then, you see the materials listed with assigned colors and textures to the right. Double-click into a field of the table row to open the dialog box *Edit Material Color and Texture*.

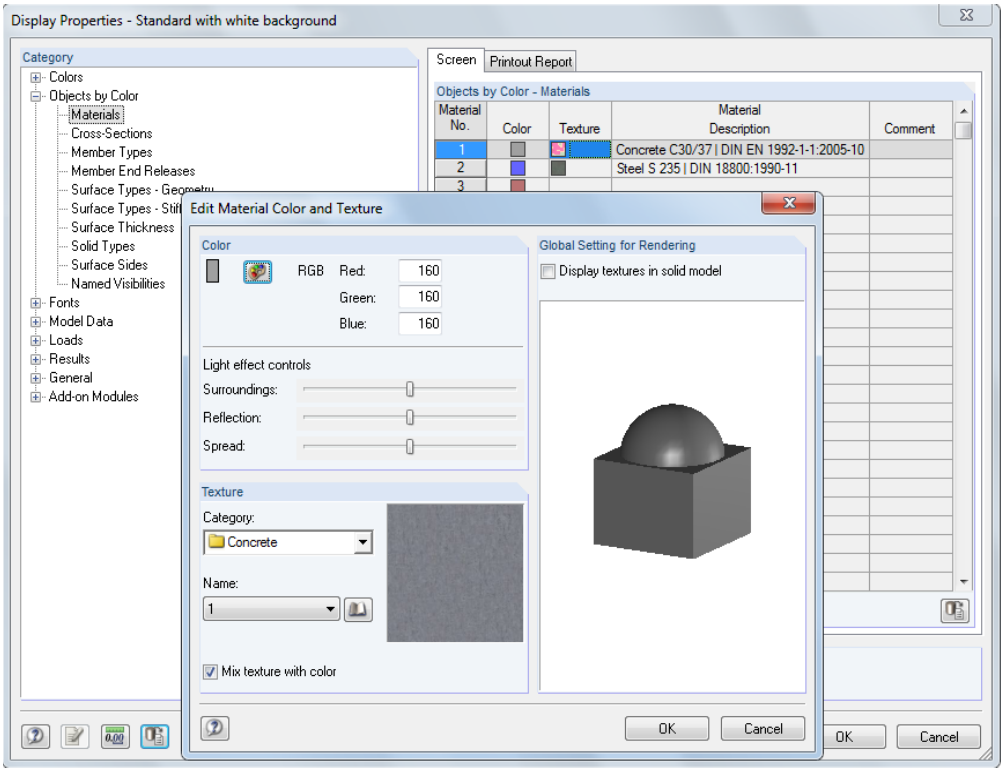


Figure 11.17: Dialog box *Edit Material Color and Texture*

Use the dialog box to adjust *Color* and *Texture* of the selected material. RFEM provides a color palette and a comprehensive library with textures (see dialog buttons).

Color control

The *Display* navigator item **Colors in Rendering According to** contains several selection fields. An activated field controls the assignment of colors for the objects in the rendering. By default, RFEM uses the material colors defined for the individual construction materials (see chapter 4.3, page 60). With the remaining options it is possible to graphically check also cross-sections and types of members, surfaces, solids etc. by means of the assigned colors.

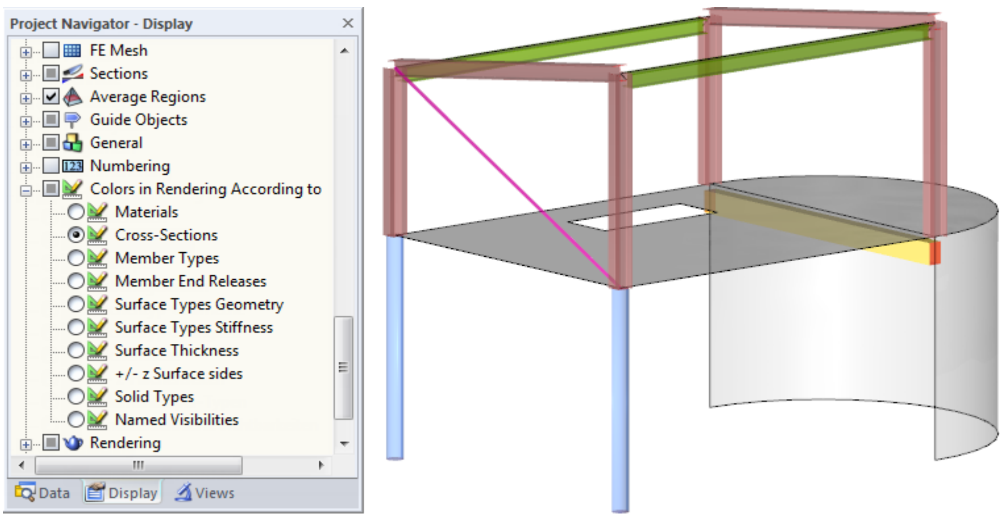


Figure 11.18: Option *Colors in Rendering According to Cross-Sections* for checking cross-section types

The option *+/- z Surface sides* is used for checking the position of surface sides which is important for defining foundations with failure or for the reinforcement layout. The top side of a surface is displayed in red, the bottom side is represented with blue color (standard).

11.1.10 Lighting

Lightness and light effects of the rendered model can be adjusted individually. To manage the lighting in the *Display* navigator,

select **Lighting** under **Rendering**.

Six different light sources are available for selection: Light 1 to 4 light the model from its side, light 5 and 6 from below and above. Each *Light* can be switched on and off individually.

Tick the check box for *Display light positions* to display the light sources in the work window. Active lights are represented in gold, inactive lights are shown in gray.

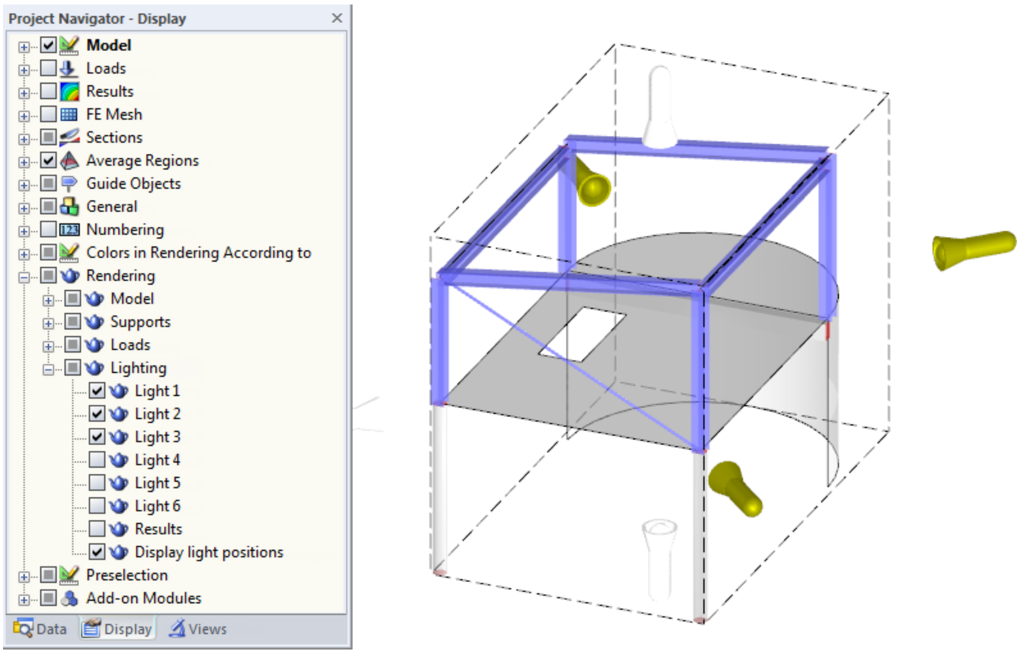


Figure 11.19: Display of light positions using the *Display* navigator

It is also possible to apply light effects to the *Results*. The check box is inactive by default because lighting often has an unfavorable effect on the visibility of surface and solid results.

11.2 Selection

With the selection functions you can define objects for subsequent editing. Objects are represented by nodes, lines, surfaces, solids, members, supports, FE mesh refinements etc. But it is possible to select also loads and guide objects (dimension lines, comments) graphically.

To select (or find) an object in the work window, you can also use the tables: Click into a table row and you see the corresponding object highlighted with colors in the graphic. However, this type of selection works only if the synchronization of the selection is set active (see chapter 11.5.4, page 486).

Using the *Data* navigator is another option to select objects: Right-click the relevant navigator entry, and then select the menu item *Select* in the context menu.

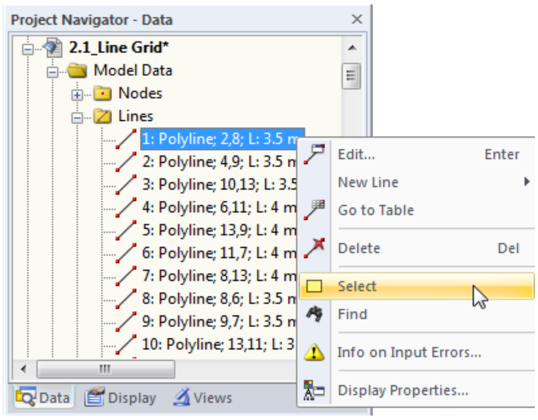


Figure 11.20: Context menu in *Data* navigator

11.2.1 Selecting Objects Graphically

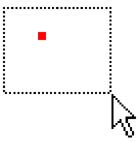
Selecting with mouse

Every object can be selected in the work window by a simple click of your mouse. Once selected, it is highlighted in the graphic with another color. Always, only the last clicked object remains selected provided that the default setting *New Selection* is not changed.

If you want to select more than one object by clicking, hold down the [Ctrl] key additionally. Another possibility is to switch to the setting *Add to Selection* by using the toolbar button shown on the left. You can also point to *Select* on the *Edit* menu where objects can be clicked separately to select them one after the other.

The so-called **preselection** allows you to locate relevant objects before clicking. If selecting objects proves to be difficult for complex structural systems, you can exclude non-required model objects from the graphical pre-selection in the *Display* navigator category *Preselection*.

Selecting with window



Use the window selection to mark a lot of objects in one single step: Hold the left mouse button down and draw a window across the relevant objects. If you open the window from the left to the right, all objects that are completely covered by the window are selected. If you open the window from the right to the left, you select also those objects that are only cut by the window.

Lines or nodes lying in a surface can be selected without displacing the surface unintentionally: Hold down the [Alt] key while you open the window across the objects inside the surface.

Selecting with rhomboid

In the isometric view, it is sometimes difficult to select an object with a rectangular window. Then, it is recommended to use the function *Selection via Rhomboid.*

Point to **Select** on the **Edit** menu, and then click **Rhomboid**

or use the toolbar button shown on the left.

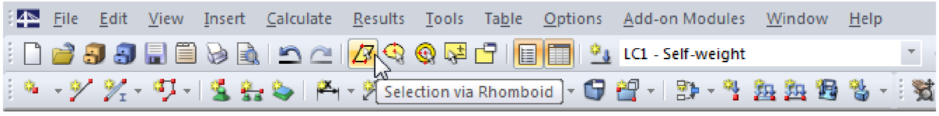


Figure 11.21: Button *Selection via Rhomboid*

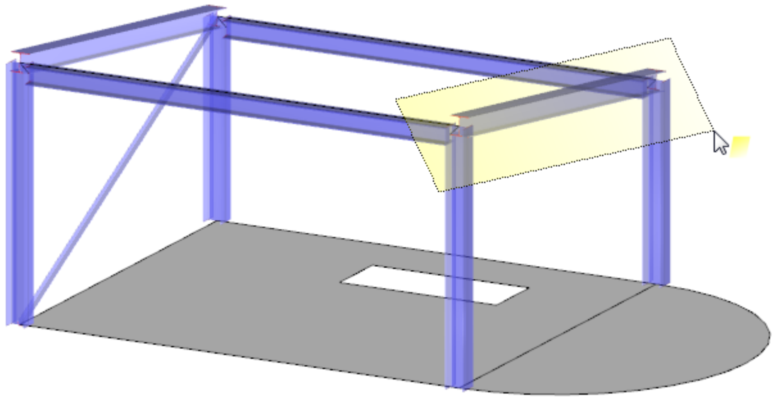
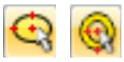


Figure 11.22: Selecting with rhomboid

Selecting with ellipse/circular ring



The possibilities to select objects by an ellipse or annulus that can be used for example for circular surfaces represent alternatives to the rhomboid selection. To access the corresponding functions,

point to **Select** on the **Edit** menu, and then select **Ellipse** or **Circular Ring**

or use the corresponding toolbar buttons.

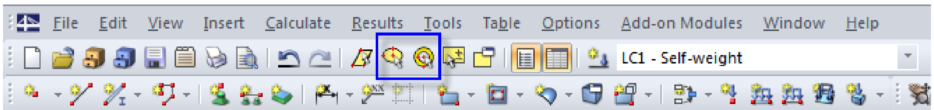


Figure 11.23: Buttons *Selection via Ellipse* or *Circular Ring*

The elliptical or annular selection zone can be set by mouse-click defining the center point and both radii.

Selecting with section line

You can select objects by means of a line running anywhere through the model. To access the function,

point to **Select** on the **Edit** menu, and then click **Section Line**.

The section line can be defined in the work window as a simple line or as polygon. Click relevant points one after the other by mouse click to define the line. The points are independent of the work plane: The selection includes all objects that are cut by the intersection line displayed in the current view.

After setting the endpoint of the section line, click it once again (alternative: double-click the last point). Make sure to place this point in an empty area of the work window.

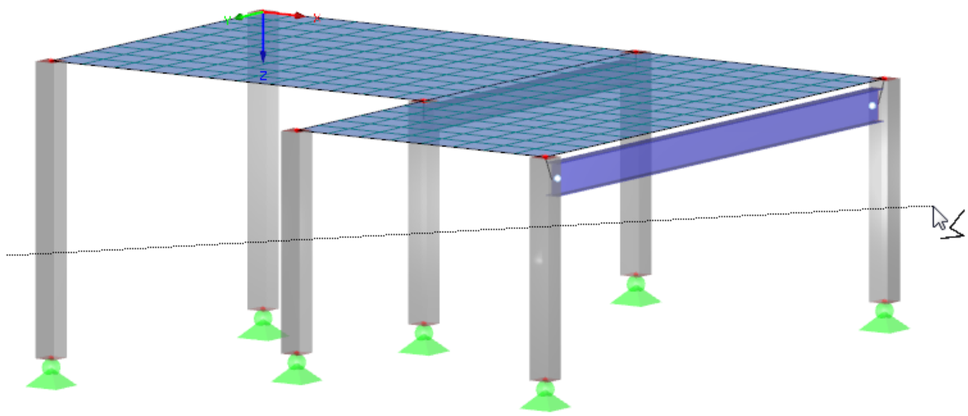


Figure 11.24: Selecting all columns with a section line

Selecting in plane

Objects lying in one plane (for example roof surfaces) can be selected easily by the selection function *In Plane*. To open the corresponding dialog box,

point to **Select** on the **Edit** menu, and then click **In Plane**.

A dialog box appears with detailed settings for selecting the objects and the plane.

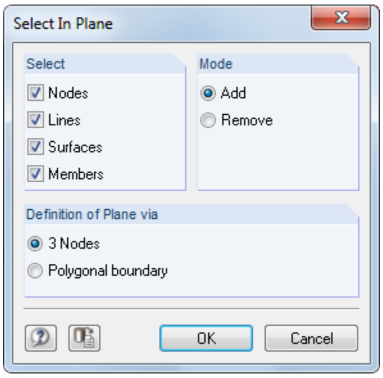


Figure 11.25: Dialog box *Select In Plane*

After clicking the [OK] button you can define the selection plane graphically: Click *3 Nodes*, or draw a *Polygonal* chain freely or with the help of nodes in the work plane.

Selecting free nodes

To select nodes that are not used for defining lines or surfaces,

point to **Select** on the **Edit** menu, and then click **Free Nodes**.

The easiest way to delete selected free nodes is to use the [Del] key.

Selecting related objects

When you select for example a surface by clicking, the nodes and lines belonging to the surface are not included in the selection. To select also the components of objects,

point to **Select** on the **Edit** menu, and then click **Related Objects**.

Use this function for example to integrate quickly the supports of members or surfaces into the selection and to save them as related objects in a user-defined visibility (see chapter 9.9.1.2, page 374).

11.2.2 Selecting Objects by Criteria

The function allows you to select objects by particular criteria. Moreover, specific objects can be added to or removed from an existing selection.

To open the dialog box used for the special selection,

point to **Select** on the **Edit** menu, and then click **Special**

or use the toolbar button shown on the left.

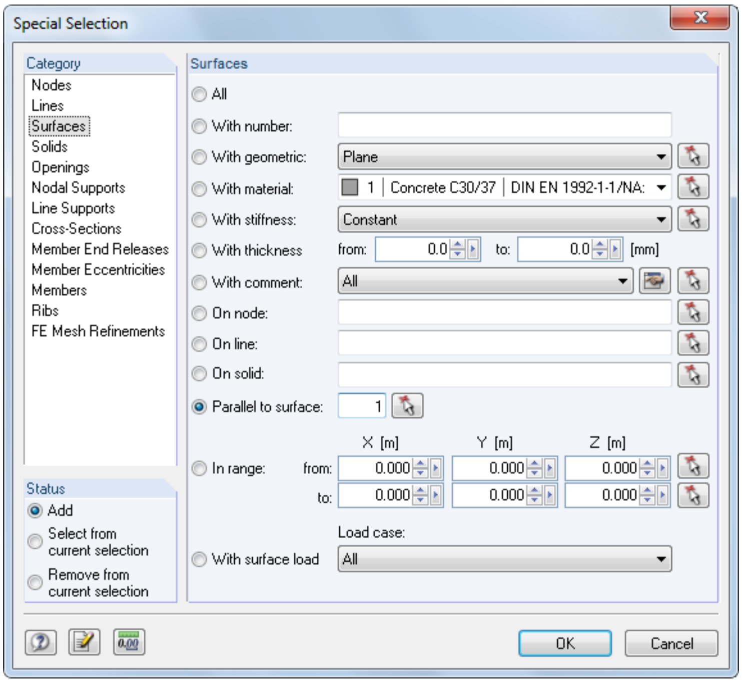


Figure 11.26: Dialog box *Special Selection*

The dialog section *Category* on the left lists the objects defined in the model. Settings in the right part of the dialog box depend on the selected object. Determine a selection criterion and specify detailed settings, if necessary.

Example

With the settings shown in Figure 11.26 all surfaces that are modeled *Parallel to surface 1* (floor slab) are selected. You can also use the [↖] button to define the template surface graphically.

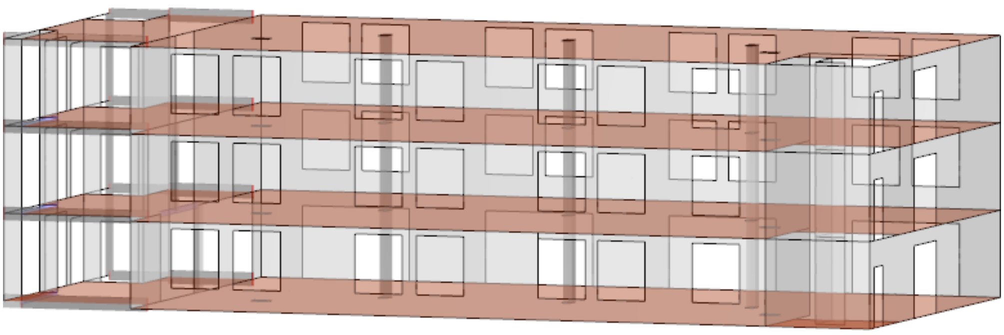


Figure 11.27: Selection of parallel surfaces

11.3 Work Window

Special CAD functions such as work planes, snap options, guidelines and user-defined coordinate systems help you to model graphical objects in the work window.

11.3.1 Work Planes

Although a model is defined spatially, it can be displayed only in two dimensions on the screen. Therefore, defining objects graphically is a problem because it must be organized in which plane objects are created when clicking into the graphic window. The work plane determines which coordinate is always "fixed".

The axes of coordinates of the currently set work plane are represented by two green, orthogonal lines. The lines' point of intersection is called "origin of the work plane".

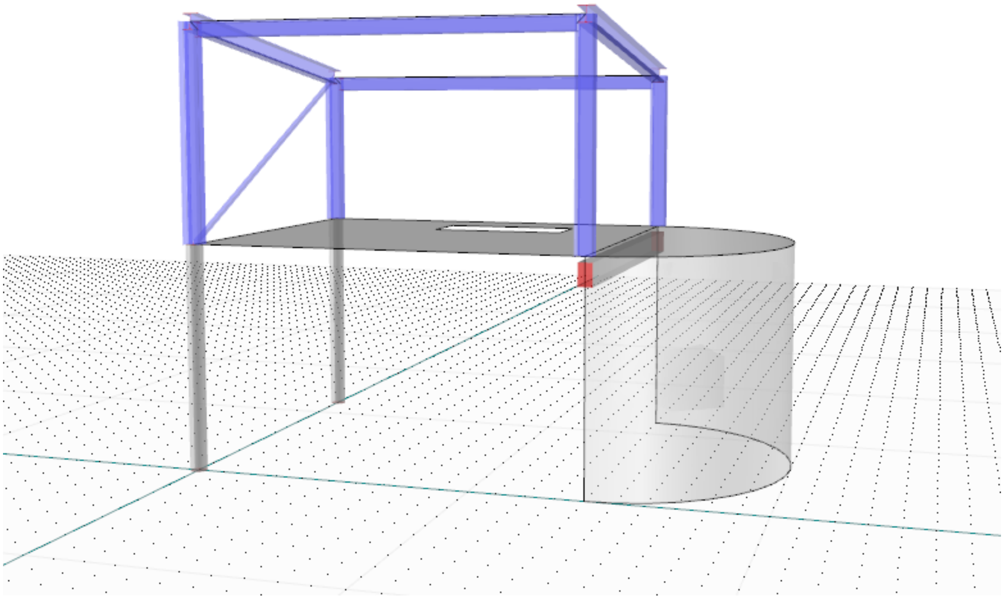


Figure 11.28: Work plane represented in the graphic

Normally, a work plane runs parallel to one of the global planes XY, YZ or XZ that are spanned by two axes of the global coordinate system. But it is also possible to specify a work plane directly as a plane with any inclination, or to define it by means of line, member and surface axes.

To open the dialog box *Work Plane and Grid/Snap* with the parameters of the work plane,

select **Work Plane, Grid/Snap, Object Snap, Guidelines** on the **Tools** menu

or use the toolbar button shown on the left.

Figure 11.29: Button *Settings of Work Plane*

The dialog box shown in Figure 11.34 on page 437 appears.

Parallel to global plane XY / YZ / XZ

The work plane can be aligned parallel with one of the following global planes.

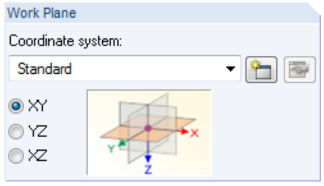
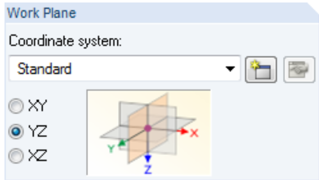
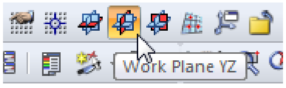
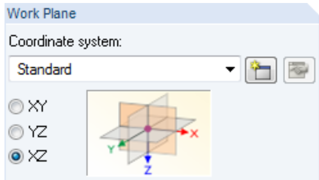
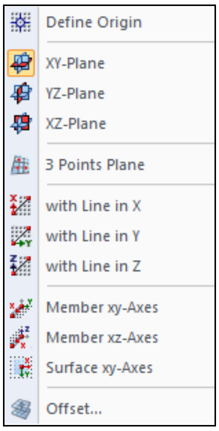
Plane	Selection in *Work Plane* dialog box	Selection in toolbar
XY		
YZ		
XZ		

Table 11.3: Selection of work plane



To find more options for defining work planes,

point to **Select Work Plane** on the **Tools** menu

or use the corresponding toolbar buttons.

3 points plane

In the work window, you can select three points defining a new work plane with the axis system *UVW*. The points must not be defined on a straight line.

Plane with line in X / Y / Z

The work plane is defined by one of the global axes and a line that you determine graphically in the work window. The zero point of the new work plane is placed into the start node of the line. The axis *U* is aligned parallel with the selected global axis. In this way, you can quickly shift the work plane for example into a roof area.

Plane with member axis xy / xz

The planes of the member axes xy ("weak axis") or xz ("strong axis") are used for defining the work plane (see chapter 4.17, page 146). The relevant member must be defined graphically in the work window. The zero point of the new work plane is placed into the start node of the member.

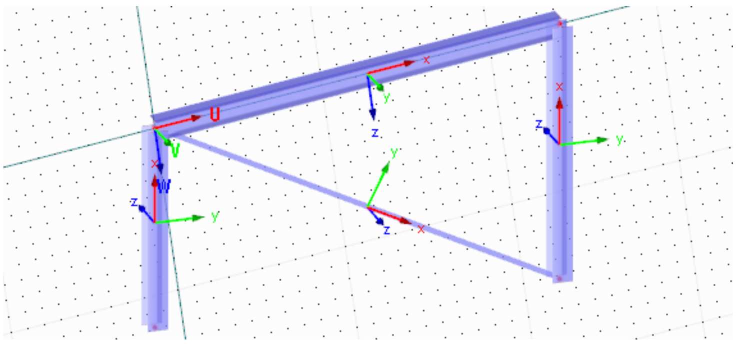


Figure 11.30: Work plane in roof inclination of member axis xz

Plane with surface axis xy

The work plane is defined by the xy-axes of a surface that you determine graphically in the work window (see chapter 4.4, page 84). The axes of the new work plane are called *UVW* (see Figure 11.30).

Offset of work plane

Use this function to shift the work plane perpendicular to the current plane. Specify the distance in the dialog box *Offset Workplane*.

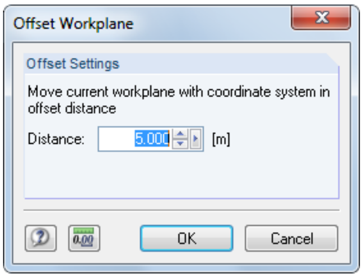


Figure 11.31: Dialog box *Offset Workplane*

The offset remains active until the function is canceled on the menu.

Zero point of work plane

The *Work Plane* dialog box (Figure 11.34) manages the zero point (origin) settings of the work plane. Use the [↖] function to select a node in the work window. Click the [New] button to define a new node. It is also possible to enter the coordinates of any point directly.

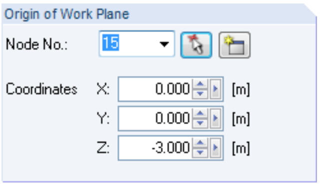


Figure 11.32: Dialog box *Work Plane*, dialog section *Origin of Work Plane*

The zero point of the work plane can also be defined graphically.

Point to **Select Work Plane** on the **Tools** menu, and then select **Define Origin** or use the toolbar button shown on the left.

Figure 11.33: Button *Set Origin of Grid/Work Plane*

11.3.2 Grid

Grid points are used to help you with the graphical input in the work plane. When nodes are defined graphically, the pointer snaps on the grid points.

The properties of grid points are managed in the dialog box *Work Plane and Grid/Snap*. To open the dialog box,

select **Work Plane, Grid/Snap, Object Snap, Guidelines** on the **Tools** menu

or use the toolbar button shown on the left (see Figure 11.29, page 434).

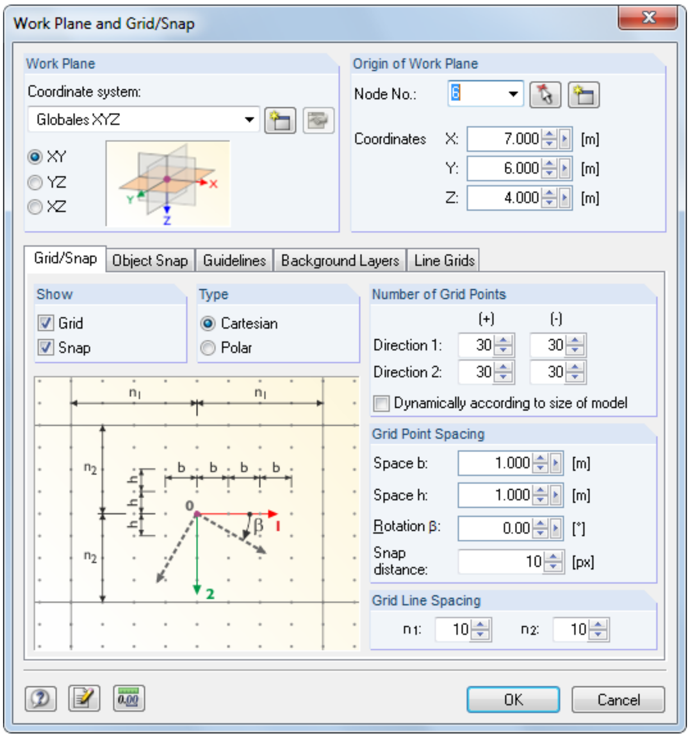


Figure 11.34: Dialog box *Work Plane and Grid/Snap*

The setting options relevant for the grid are available in the dialog tab *Grid/Snap*.

Show

To display the grid in the work window, tick the *Grid* check box. The snap function can be turned on and off independently of the grid by means of the *Snap* check box. Thus, the snap function on the grid points can be effective while the grid is invisible.

To switch both functions on and off quickly, use the buttons [SNAP] and [GRID] in the status bar.

Type

The grid points can be arranged in the Cartesian or the polar coordinate system. Depending on the selection, contents of the displayed dialog sections are changing.

Alternatively, you can select the coordinate system by means of the buttons [CARTES], [POLAR] or [ORTHO] in the status bar.

Number of grid points

When the Cartesian grid is set, you can define the number of grid points for both axis directions separately.

When the polar grid is set, you have to specify the number of concentric grid circles.

When the option *Dynamically according to size of model* is ticked, the grid will automatically be adjusted to the dimensions of the model. Thus, a sufficient number of grid points will always be available around the model. However, the required grid points will be recalculated after each input, which may slow down the speed for creating the graphic when you work on complex models.

Grid point spacing

When you use the Cartesian grid, you can define the spacing of grid points separately for the directions *1* and *2*.

For the polar grid you have to specify the radial spacing *R* for the grid circles. The angle α controls the spacing of grid points on the circles.

Optionally, the Cartesian and the polar grid can be rotated about the rotation angle β.

If needed, the number of pixels controlling the *Snap distance* can be adjusted.

11.3.3 Object Snap

The object snap facilitates the CAD-like modeling when defining lines. In addition to nodes, several snap points along the lines can be activated.

The settings for the object snap are defined as well in the *Work Plane* dialog box. To open the dialog box,

select **Work Plane, Grid/Snap, Object Snap, Guidelines** on the **Tools** menu

or use the toolbar button shown on the left (see Figure 11.29, page 434).

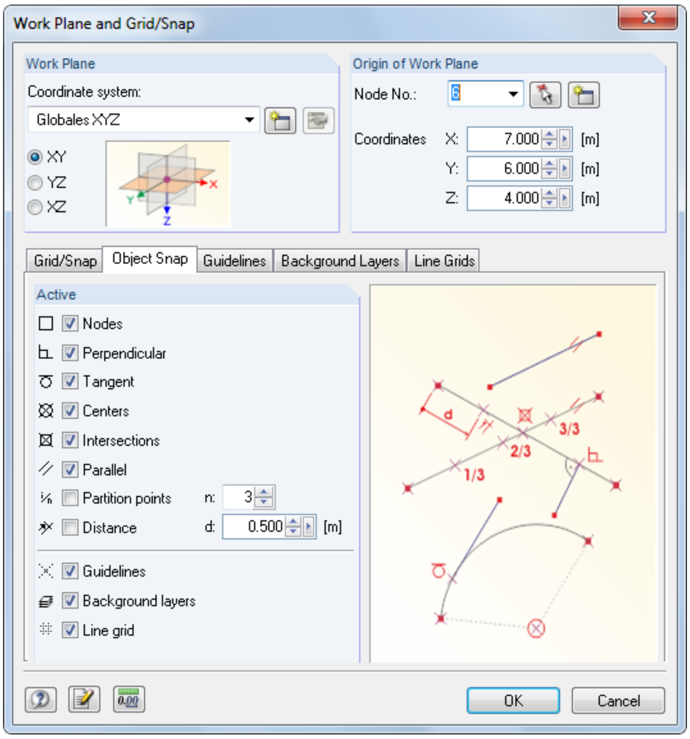


Figure 11.35: Dialog box *Work Plane and Grid/Snap*

The dialog tab *Object Snap* manages the different snap functions.

To make the functions of the object snap effective, make sure that the button [OSNAP] is activated in the status bar.

Nodes

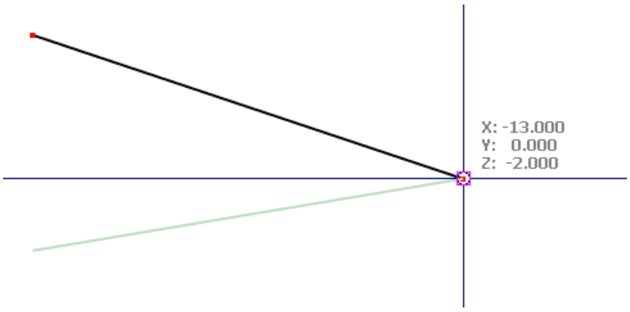


Figure 11.36: Snapping a node

When defining a new line existing nodes are captured. Snap points are symbolized by squares.

Perpendicular

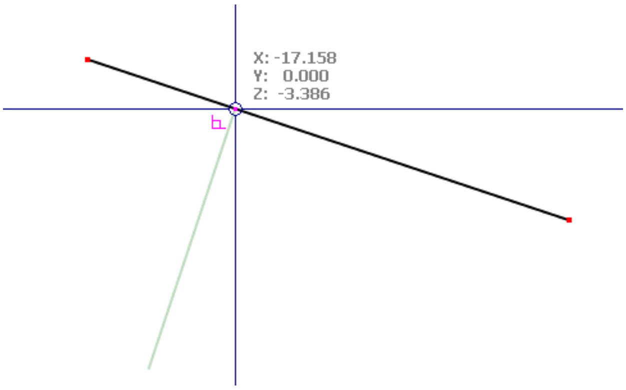


Figure 11.37: Connecting a line perpendicularly

When drawing a line the pointer is snapped when you move it near the perpendicular point. The snap point is symbolized by a perpendicular symbol.

Tangent

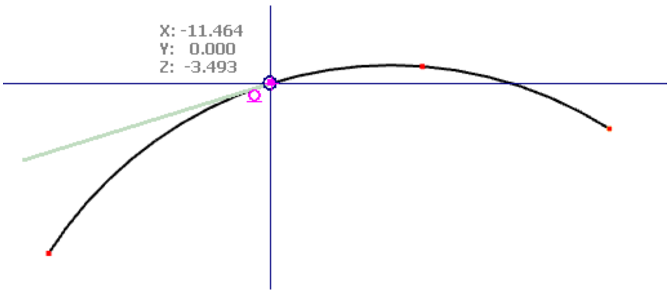


Figure 11.38: Connecting a tangent to an arc

A tangent is created on a circular arc. When drawing a line the pointer is snapped when you move it near the tangent point. The snap point is symbolized by a tangent symbol.

Centers

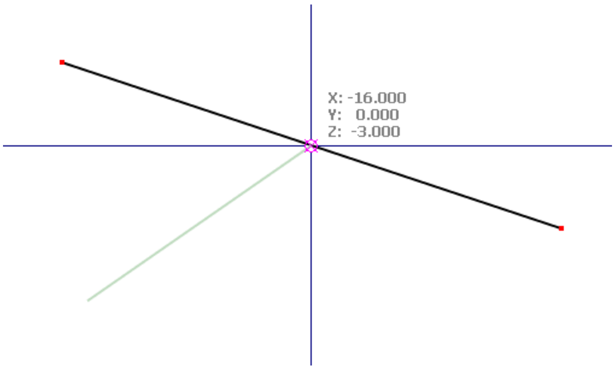


Figure 11.39: Connecting a line in center

When you move the pointer near the center (middle) of a line, it will be snapped. The center symbol appears on the snap point.

Intersections

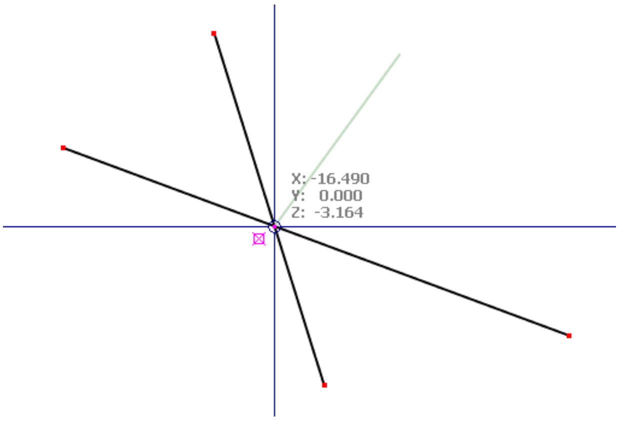


Figure 11.40: Snapping lines at intersection point

The pointer snaps on the intersection point of two crossing lines that have no common node. The snap point is symbolized by the intersection symbol shown on the left.

Parallel

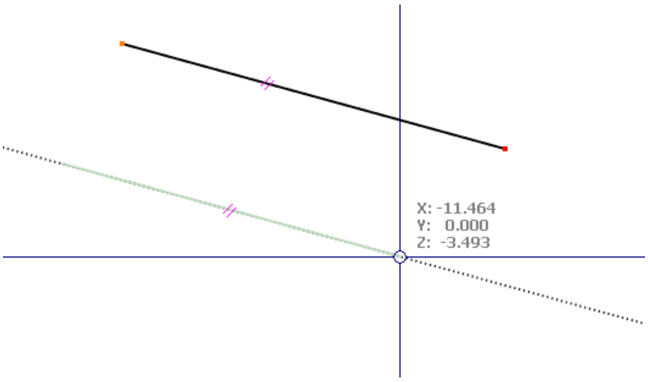


Figure 11.41: Snapping a parallel line

Use this function to set parallel lines: Define the start node of the new line, and then move the pointer over a template line. Now, if you move the pointer near a possible end node of the new line running parallel to the template, the parallel symbol shown on the left appears on both lines.

Partition points

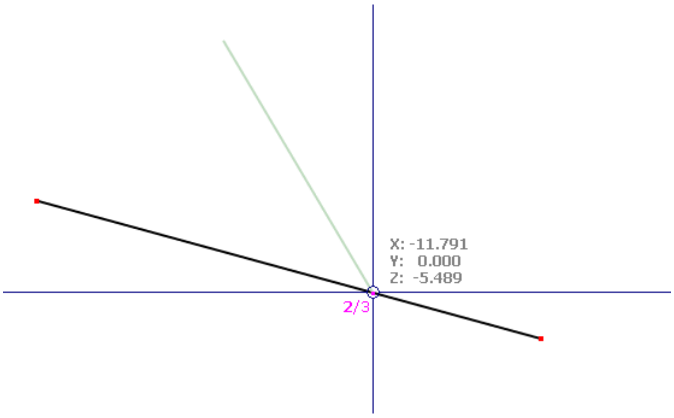


Figure 11.42: Snapping a line on partition point (example: 2/3-point)

In the dialog tab *Object Snap* of the *Work Plane* dialog box, you can enter a number of *n* line divisions. When you move the pointer along a line, it will be snapped on the partition points. The partition is displayed as fraction on the pointer.

Distance

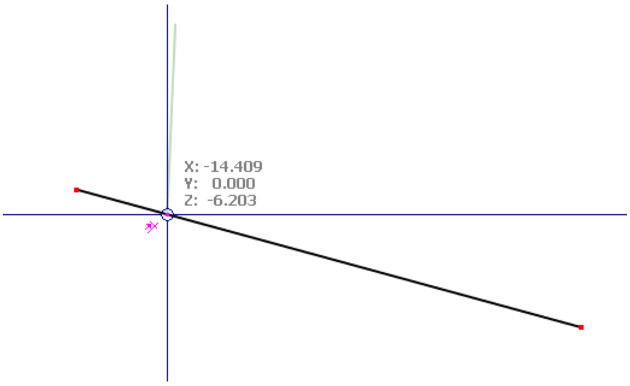


Figure 11.43: Connecting a line in a defined distance

In the dialog tab *Object Snap* of the *Work Plane* dialog box, you can enter a distance *d* for dividing a line. When you move the pointer across a line, it will be snapped at the defined distance from the line start and end. The distance symbol appears on the pointer.

Guidelines

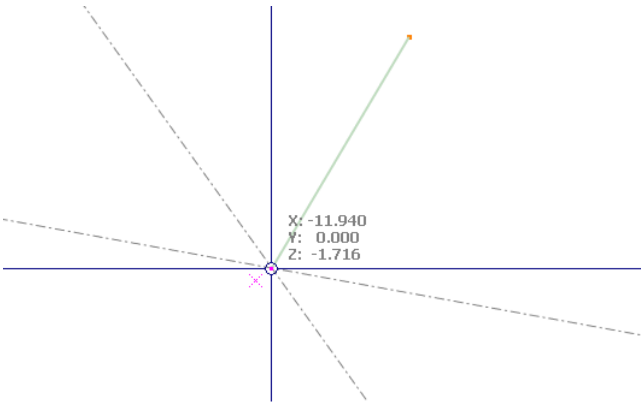


Figure 11.44: Snapping guidelines at intersection point

When you move the pointer near the intersection point of two guide lines (see chapter 11.3.7, page 448), it will be snapped. The intersection symbol appears on the snap point.

Background layers

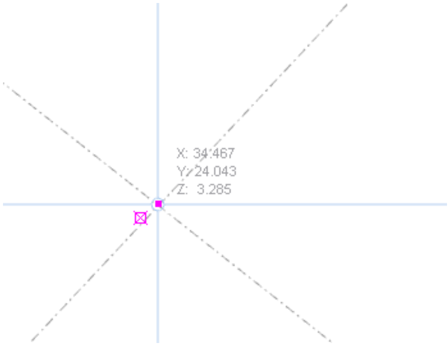


Figure 11.45: Snapping background layers on intersection point

Use this function to set nodes on intersection points of background layers (see chapter 11.3.7, page 448). The intersection symbol appears on the snap point.

Line grid

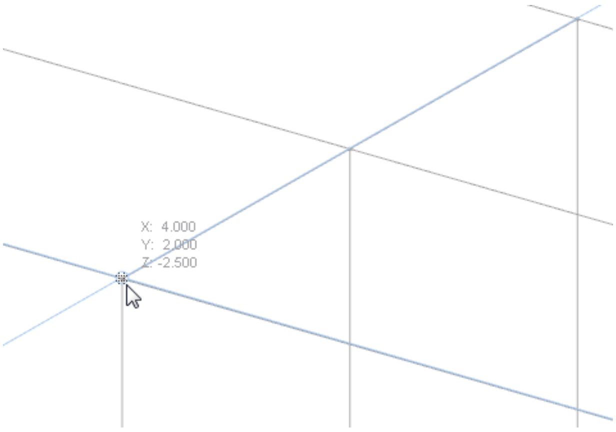


Figure 11.46: Snapping points of line grid

Use this function to place objects into the intersection points of a line grid (see chapter 11.3.8, page 452).

11.3.4 Coordinate Systems

User-defined coordinate systems make entering inclined parts of a model easier. They have nothing to do with the axis systems of lines, surfaces or members. As an alternative, you can define work planes graphically by means of points or axes of lines, members and surfaces (see chapter 11.3.1, page 435).

To open the dialog box *Coordinate System*,

select **Coordinate System** on the Tools **menu**

or use the toolbar button shown on the left.

Figure 11.47: Button *Coordinate System*

You can also use the dialog box *Work Plane and Grid/Snap* (see Figure 3.15, page 28) where you find the [New] button for creating a user-defined system of coordinates.

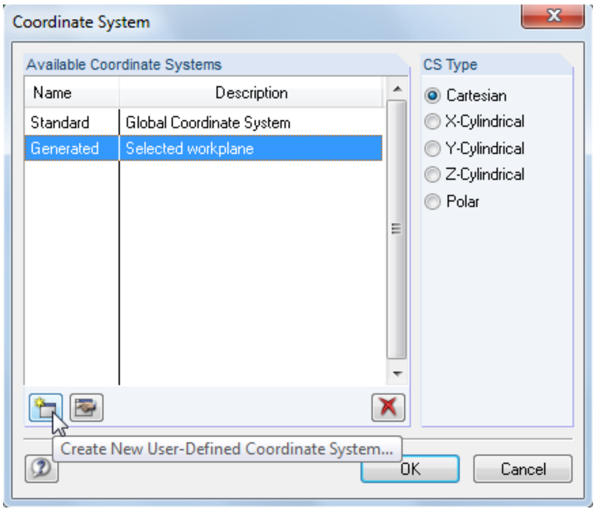


Figure 11.48: Dialog box *Coordinate System*

The *Standard* coordinate system that refers to the global axes X,Y,Z and the origin is preset.

Create a new coordinate system

Click the [New] button shown in Figure 11.48 to open the following dialog box. You find the same button in the dialog box *Work Plane and Grid/Snap* (see Figure 3.15, page 28).

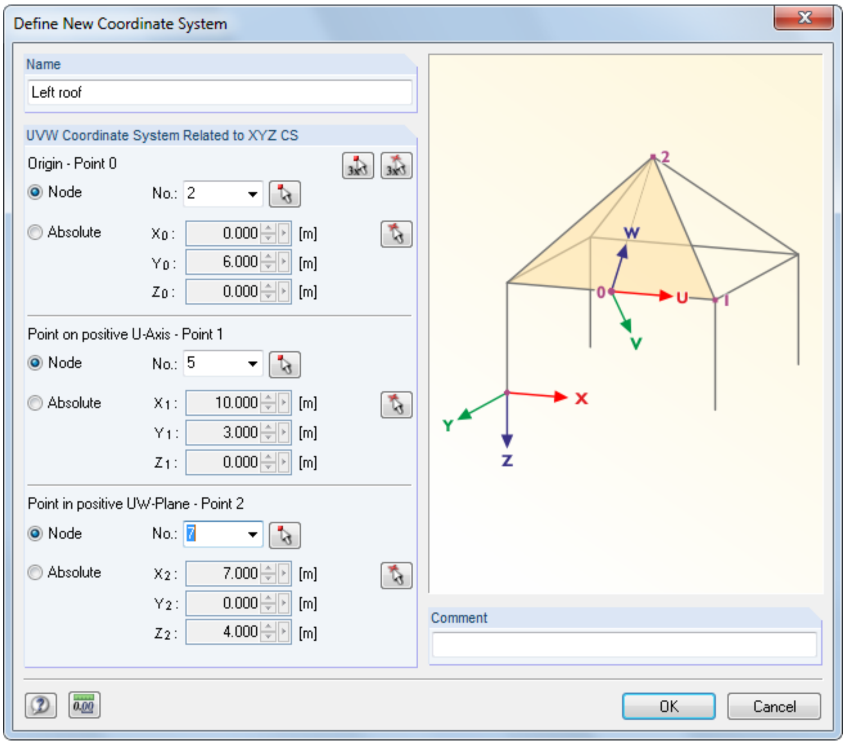


Figure 11.49: Dialog box *Define New Coordinate System*

Enter a *Name* for the new coordinate system. Then, define the axis system with the help of three parameters in the dialog section *UVW Coordinate System Related to XYZ CS*:

- Origin (zero point of new coordinate system)
- Point on positive U-axis (first axis)
- Point in positive UW-plane (rotation of plane about axis U)

Specify three points that you can enter directly or select graphically. The points must not be defined on a straight line.

You can use the buttons shown on the left to select the three points one after the other in the work window (please observe the sequence when defining points 0 to 2). With the left button you can select only *Nodes*, with the right button you can select any *Points*. The difference becomes especially significant when a node representing a definition point of the coordinate system is changed. Then, the coordinate system will be adjusted automatically. In case of any points, the system of coordinates is fixed.

If a user-defined work plane is defined with the help of three points (see chapter 11.3.1, page 435), RFEM creates automatically a new coordinate system with the name *Generated*.

Edit or delete a coordinate system

Only user-defined coordinate systems can be edited or deleted. Use the following two buttons available in the *Coordinate System* dialog box.

	Modifies selected coordinate system
	Deletes selected coordinate system

Table 11.4: Buttons in the dialog box *Coordinate System*

Example

In a frame joint, a new coordinate system is defined for the diagonal lying in the plane of the roof. The *Origin* is set in corner node **6**. End node **4** of the diagonal member is selected as *Point on positive U-Axis*. Base node **5** of the column is selected as *Point in positive UW-Plane*.

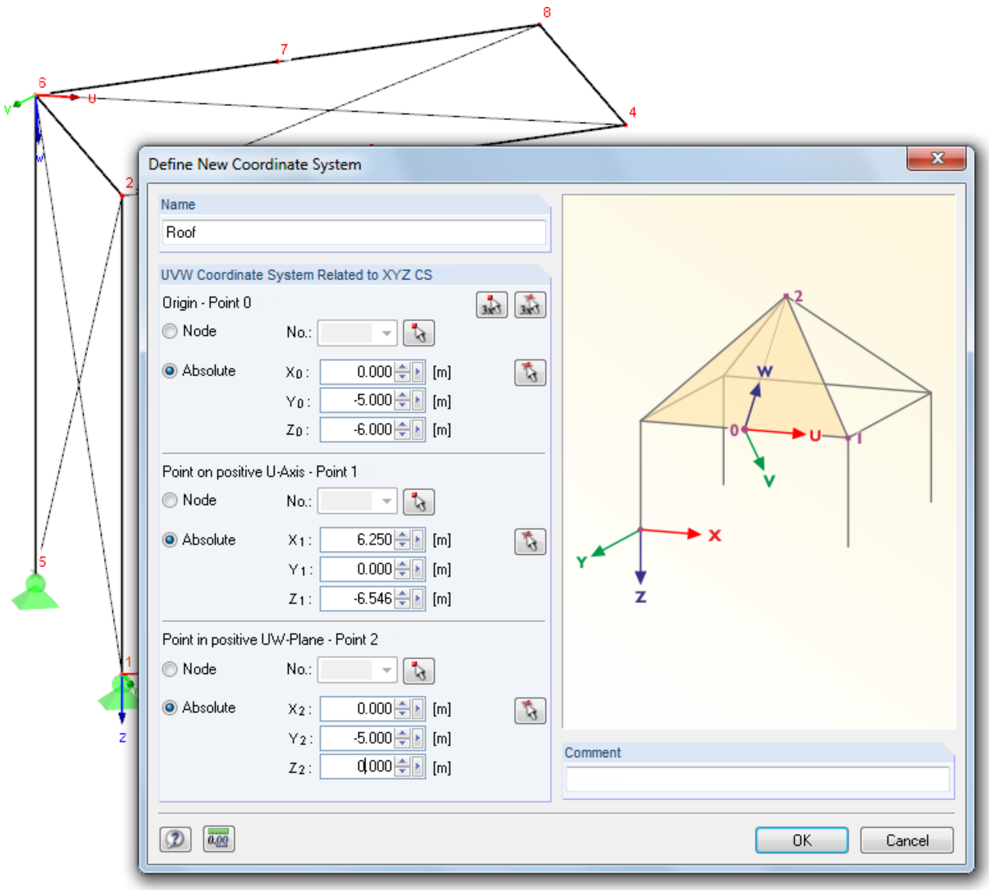


Figure 11.50: User-defined coordinate system **UVW** in a frame joint

Now, the grid refers to the work planes UV, VW and UW where you can define new objects (see chapter 11.3.1, page 434).

11.3.5 Dimensions

It is possible to add user-defined dimension lines to the model.

To apply dimensioning functions,

point to **Dimensions** on the **Insert** menu

or use the corresponding toolbar buttons.

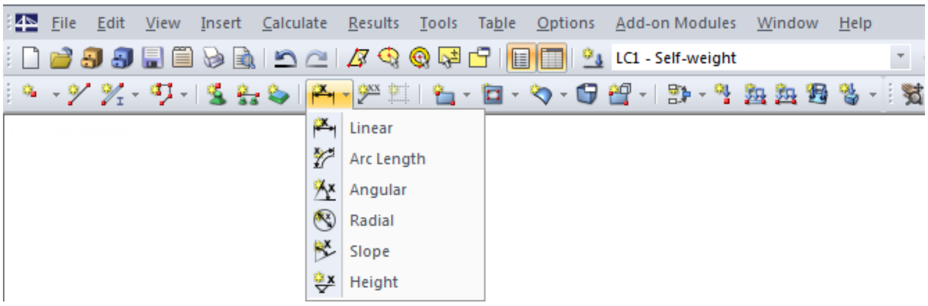


Figure 11.51: *New Dimension* buttons

The following dimension options can be selected:

Dimension	**Dimensioned Objects**
Linear	Lengths between two or several nodes
Arc length	Length between nodes of an arc
Angular	Angle between three nodes or two lines
Radial	Diameter or radius of circle and arc
Slope	Inclination angle between a line and a plane
Height	Height level of a node

Table 11.5: Dimensioning functions

The dialog box *New Dimension* opens. The appearance of the dialog box depends on your selection.

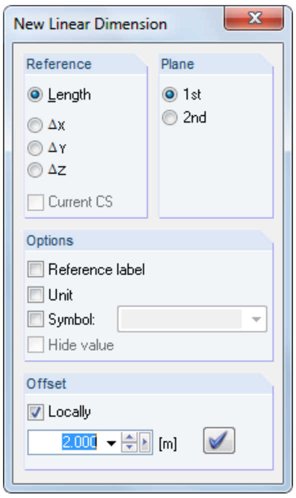


Figure 11.52: Dialog box *New Linear Dimension*

Use the pointer displayed with a selection symbol and click the objects representing the dimensioning's reference points one after the other. In the dialog section *Reference*, you can select the real length or the projection in one of the global axis directions.

In the dialog section to the right, you determine the *Plane* where the dimension line is applied. The setting refers to the axes of the global coordinate system XYZ, respectively the line axes. If you switch the plane and move the pointer in the graphic, you can see the effect of both selection fields.

Use the four check boxes in the dialog section *Options* to define the information appearing on the values. When you select *Symbol*, you can enter a dimensioning symbol. It is also possible to select it from the list. Tick *Hide value* to switch off the measured value so that only the symbol appears.

The *Offset* determines the distance of the dimension line from the first selected node. The distance can be defined also graphically by using the mouse pointer. To finally define the dimension line, click into the work window or use the button [Set Dimension] shown on the left.

To define a chain dimensioning with equal offset, click the individual nodes one after the other, and then specify the offset.

To set the display of dimension lines, use the *Display* navigator or the general context menu (right-click into an object-free area of the work window).

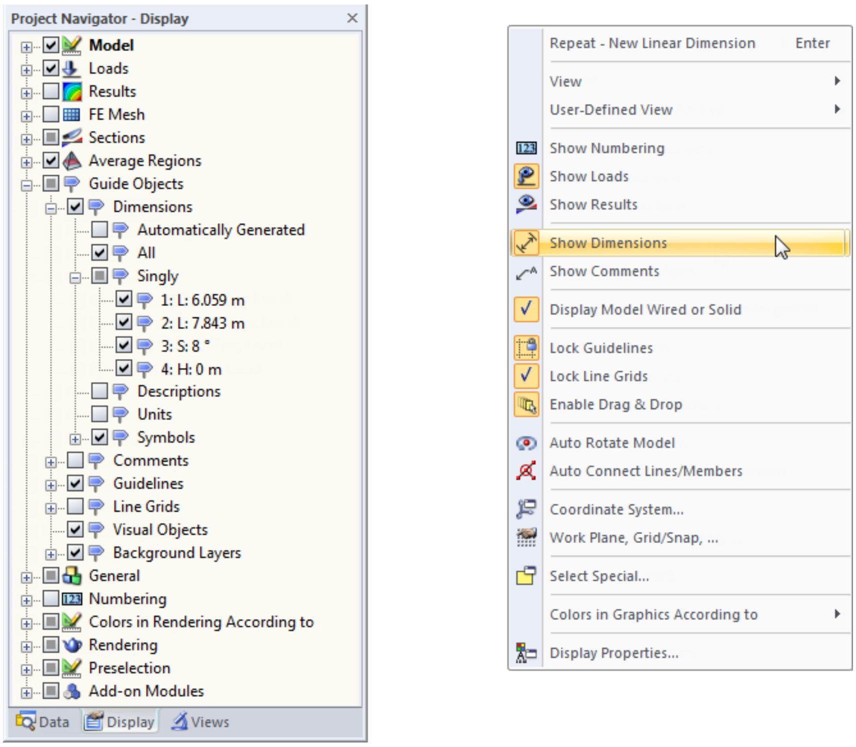


Figure 11.53: *Display* navigator (*Guide Objects* → *Dimensions*) and general context menu

When the model geometry is modified, dimensions will be adjusted automatically.

To open the dialog box *Edit Dimension*, double-click the relevant dimension. In this way, you can subsequently adjust the offset. However, if you want to relate the dimension line to other nodes or lines, delete the dimension first. Then you can redefine it.

11.3.6 Comments

There are two types of comments:

- Comments in dialog boxes and tables (see chapter 11.1.4, page 421)
- Comments in work window

This chapter describes how comments are set graphically.

You can place comments referring to nodes and centers of lines and members. They can also be placed anywhere in the current work plane or in a global plane.

To open the dialog box for applying comments,

select **Comments** on the **Insert** menu

or use the toolbar button shown on the left.

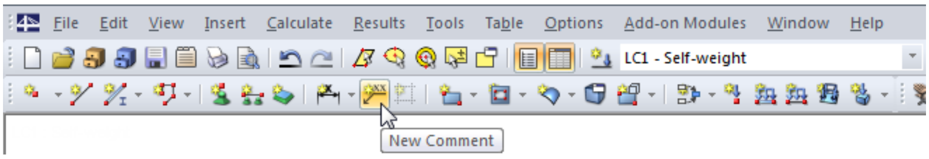


Figure 11.54: Button *New Comment*

The dialog box *New Comment* opens.

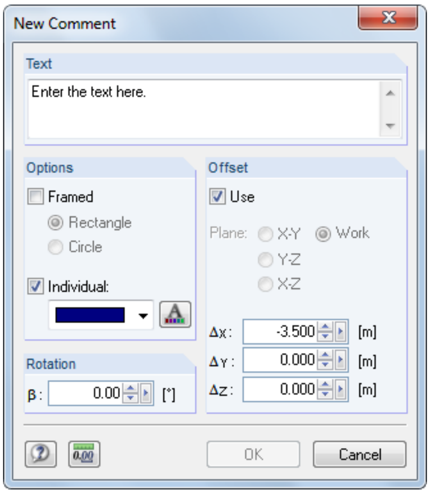


Figure 11.55: Dialog box *New Comment*

Enter the comment text into the dialog section *Text*. The appearance of the comment concerning colors and [Fonts] can be adjusted in the dialog section *Options*. Optionally, the comment is *Framed* by a rectangle or circle.

The *Rotation* of the comment allows you to user-define the comment text arrangement.

If the check box in the dialog section *Offset* is ticked, the comment will be arranged in a specified distance to the object. You can define the distance also graphically: First, click the object after entering the comment text. Then, use the pointer to locate the appropriate position where you enter the comment text with another mouse click. RFEM displays the current work plane so that you can place the comment correctly. If necessary, you can change the work plane before placing the comment.

To set the display of comments, use the *Display* navigator or the general context menu (right-click into an object-free area of the work window, see figure below).

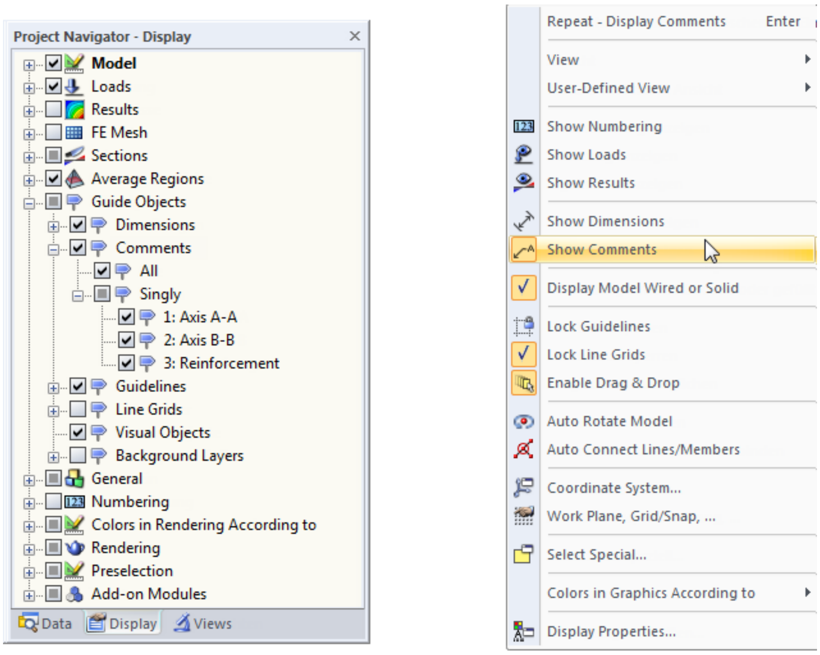


Figure 11.56: *Display* navigator (*Guide Objects* → *Comments*) and general context menu

When the model geometry is modified, comments will be adjusted automatically.

Comment texts including offset can be edited subsequently: Double-click the comment in the work window or its entry in the *Data* navigator.

You can shift comments by using the drag-and-drop function (for copying: hold down the [Ctrl] key). Please note the following: When you "grab" the arrow of the graphical comment at its head, you shift the entire comment. When you "grab" it on the text, the arrowhead continues to point to the object so that the position of the comment text can be adjusted in the work plane.

11.3.7 Guidelines

Guidelines represent a grid of axes and rows underneath the graphical workspace. The intersection points of guidelines are as well snap points for graphical input, provided that the snap function for *Guidelines-Intersections* is active in the object snap (see chapter 11.3.3, page 441).

Guidelines do not need to be parallel to the axes of the global coordinate system XYZ. Angles can be specified freely. You can even define a polar arrangement of guidelines. Also spacings among guidelines may be arbitrary.

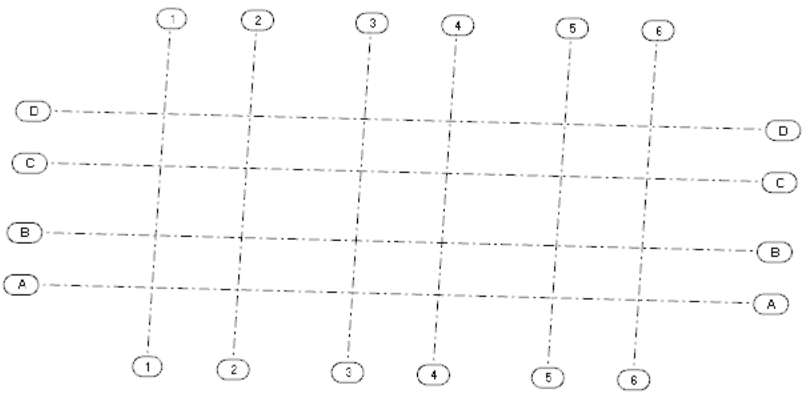


Figure 11-57: Grid of guidelines

Create guidelines

Dialog input

To open the dialog box for creating a new guideline,

point to **Guidelines** on the **Insert** menu, and then select **Dialog Box**

or use the context menu in the *Data* navigator.

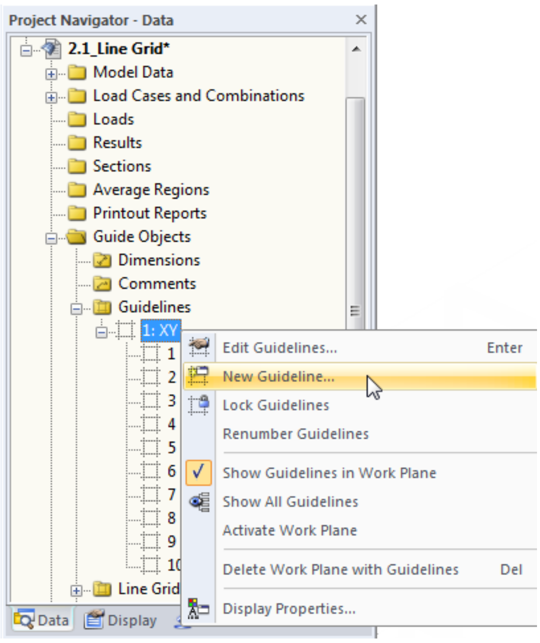


Figure 11.58: Context menu of *Guidelines* in *Data* navigator

The following dialog box appears:

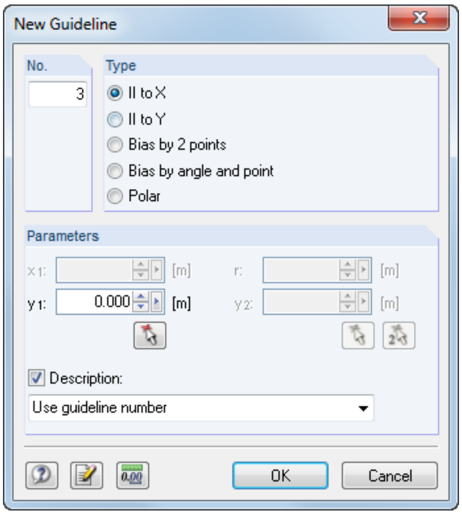


Figure 11.59: Dialog box *New Guideline*

The *No.* of the guideline is allocated by the program but can be adjusted, if necessary.

With the options in the dialog section *Type* you decide how the guideline will be created (see table below).

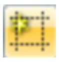
Type	Explanation
II to X / Y / Z (parallel to global axis X, Y or Z)	The guideline is created parallel to one of the global axes. Specify the distances x_1 / y_1 / z_1 of the respective global axes in the dialog section *Parameters*.
Bias by 2 points	In the dialog section *Parameters*, enter the coordinates of two points in the current work plane to define the guideline.
Bias by angle and point	The coordinates of a point and a rotation angle must be specified in the *Parameters* dialog section. The guideline will be created in the current work plane.
Polar	In the *Parameters* dialog section, the center point and the radius for the circular guideline must be specified.

Table 11.6: Types of guidelines

Enter the individual parameters into the input fields or determine them graphically in the work window by using the [↖] function.

When the check box *Description* is ticked, you can enter a description for the guideline into the input field. You can also select a description from the list.

Graphical input



To define a guideline graphically,

- point to **Guidelines** on the **Insert** menu, and then select **Graphically**
- use the button [New Guideline Graphically] shown on the left or
- grab an axis of the work plane and move it in a parallel direction (only possible if guidelines are not locked, see below).

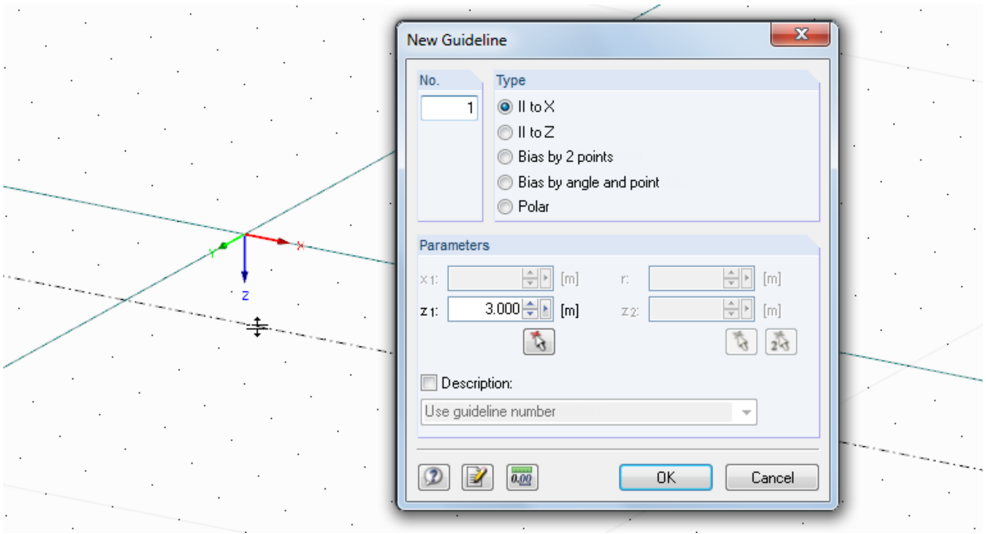


Figure 11.60: Creating a guideline graphically

The dialog box *New Guideline* is described above.

Edit and delete guidelines

To open the dialog box for editing guidelines, double-click a guideline in the graphic or its entry in the *Data* navigator.

If the guideline cannot be selected in the graphic, it is locked (see below). Guidelines can be unlocked quickly in the following way: Right-click in an empty space of the work window and deactivate the option *Lock Guidelines* in the context menu.

Another possibility to edit guidelines is to select *Work Plane, Grid/Snap, Object Snap, Guidelines* on the *Tools* menu, or to use the toolbar button shown on the left. A dialog box opens where you can use the *Guidelines* tab not only for activating the snap but for editing, deleting or hiding and displaying guidelines as well as creating new guidelines.

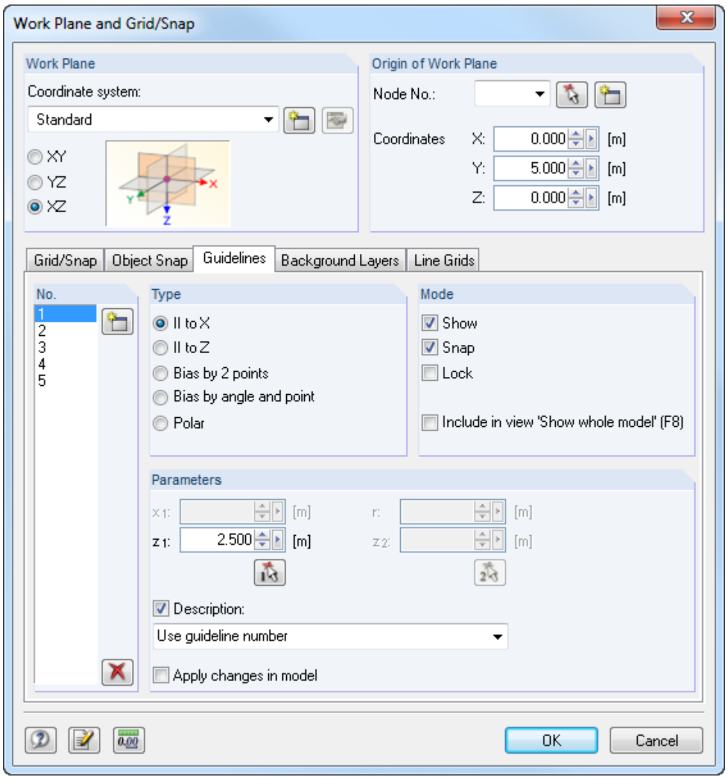
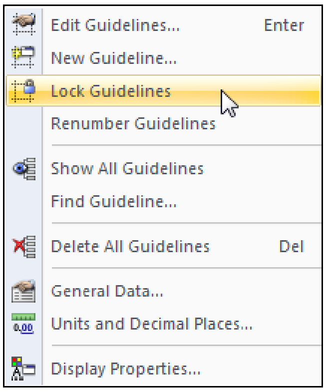


Figure 11.61: Dialog box *Work Plane and Grid/Snap*, tab *Guidelines*

Guidelines can be deleted in both the work window and the *Data* navigator: Right-click the guideline, and then select *Delete* or *Delete Guideline* in the context menu.

Locking guidelines



Context menu of guidelines

When guidelines are locked, they cannot be selected, edited, moved or deleted. In this way, they do not affect the graphical input of objects. Nevertheless, the snap function on the intersection points remains active.

To lock or unlock all guidelines,

- right-click a guideline and select *Lock Guidelines* on the context menu
- point to **Guidelines** on the **Edit** menu, and then select **Lock** or
- right-click *Guidelines* in the *Data* navigator and select *Lock Guidelines* on the context menu.

Copy and move guidelines

Guidelines are normal graphical objects for which you can use all common editing functions.

To move or copy a guideline, select the guideline first. Then, you can apply the function described in chapter 11.4.1 on page 459.

Show guidelines

The *Display* navigator controls the graphical representation of guidelines in detail (see the following figure).

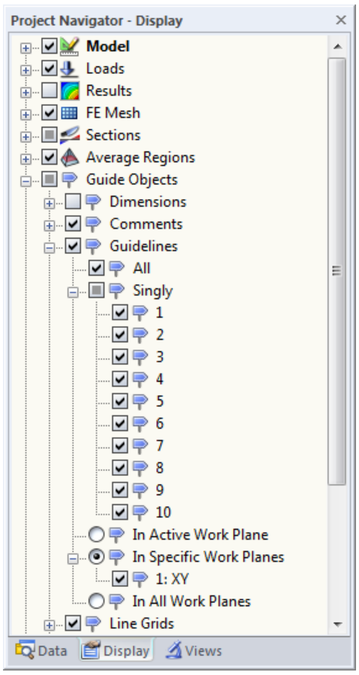


Figure 11.62: Guideline settings in the *Display* navigator

11.3.8 Line Grid

User-defined line grids help you to model structures consisting of surfaces, girder grillages or grids. The intersection points of the grid represent definition points for surfaces, lines and members.

It is possible to use several line grids in one model.

Create line grid

To open the dialog box for creating a new line grid,

select **Line Grid** on the **Insert** menu

or use the context menu in the *Data* navigator.

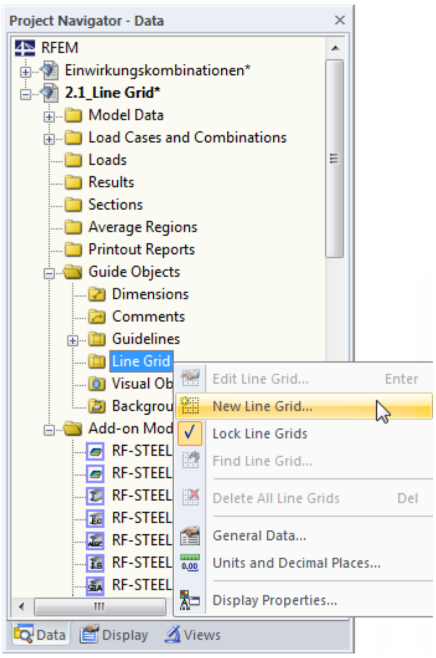


Figure 11.63: Context menu of *Line Grid* in *Data* navigator

The dialog box *Line Grid* appears where you can define the new grid.

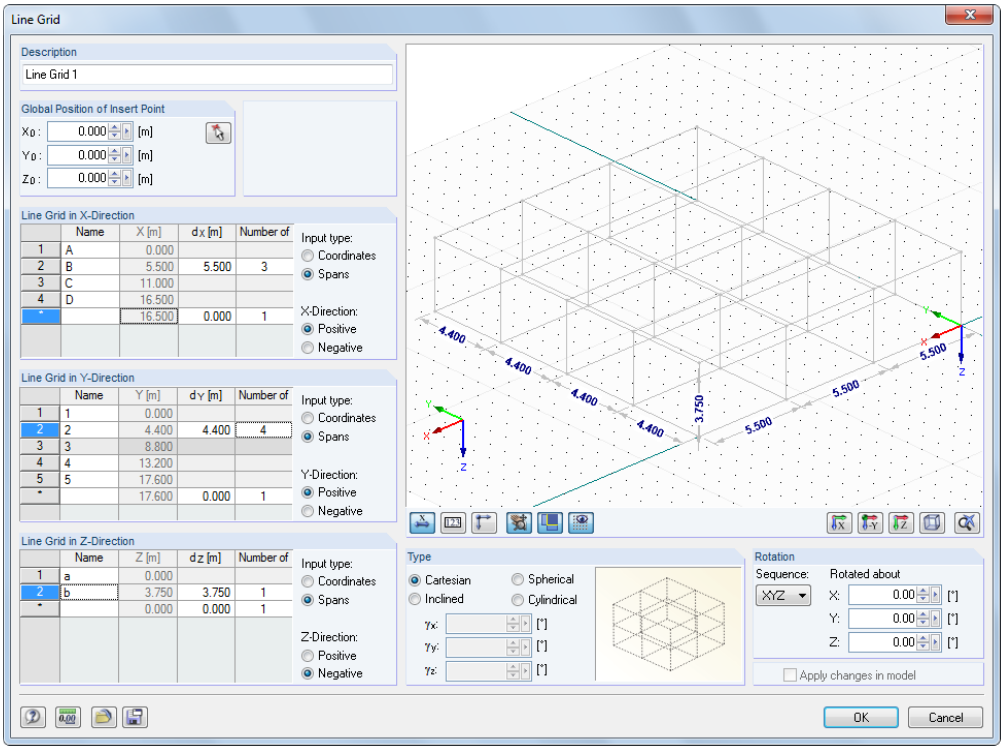


Figure 11.64: Dialog box *Line Grid*

The *Global Position of Insert Point* defines the origin of the line grid. The coordinates can be entered or selected in the work window by using the [↖] function.

The dialog section *Type* below the dialog graphic offers the following options to define the grid system before entering further data:

- Cartesian
- Spherical
- Inclined (grid that can be rotated for each axis about any rotation angle γ)
- Cylindrical

The small graphic to the right is interactive with the type specification.

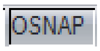
In the dialog sections *Line Grid in X-/Y-/Z-Direction*, enter the distances *d* and the *Number of* spans for each direction. The *Name* is preset but can be adjusted. It is also possible to enter the *Coordinates* of the distances or to adjust them subsequently.

The options *Positive* and *Negative* determine in which direction of the global axis the line grid will be created.

With the dialog section *Rotation* in the right bottom corner you have the possibility to rotate the line grid about an axis: First, select the *Sequence* determining the order of the local grid axes X', Y' and Z'. Then, enter the angle of rotation about the global axes X, Y and Z in the input fields for *Rotated about*. You can also use the field buttons [▶] to define the support rotation graphically.

A great part of the dialog box is covered by a graphic window where input is immediately represented graphically. The buttons below the window are familiar, you know them from RFEM. They control the display for dimensioning, numbering, axes and view. It is also possible to use the mouse control options for the big dialog graphic (see chapter 3.4.9, page 35).

Each line grid can be saved as template and reused later. Both buttons shown on the left are used to [Save] and [Load] grid data.



After closing the dialog box, you can set objects on the grid nodes. Make sure that the object snap is active (see chapter 11.3.3, page 438).

11.3.9 Visual Objects

Visual objects are 3D objects used for example in architectural design programs to represent model designs close to reality (for example people, cars, trees, textures etc.). You can also integrate 3D objects into the RFEM model to demonstrate the model's proportions.

Load visual object

To open the dialog box for importing a visual object,

select **Visual Object** on the **Insert** menu

or use the context menu in the *Data* navigator.

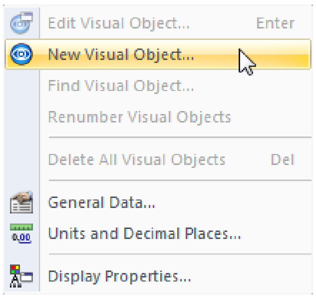


Figure 11.65: Context menu in the *Data* navigator, *Guide Objects* → *Visual Objects*

The dialog box *New Visual Object* opens where you have to specify the *Description* and *File Name*.

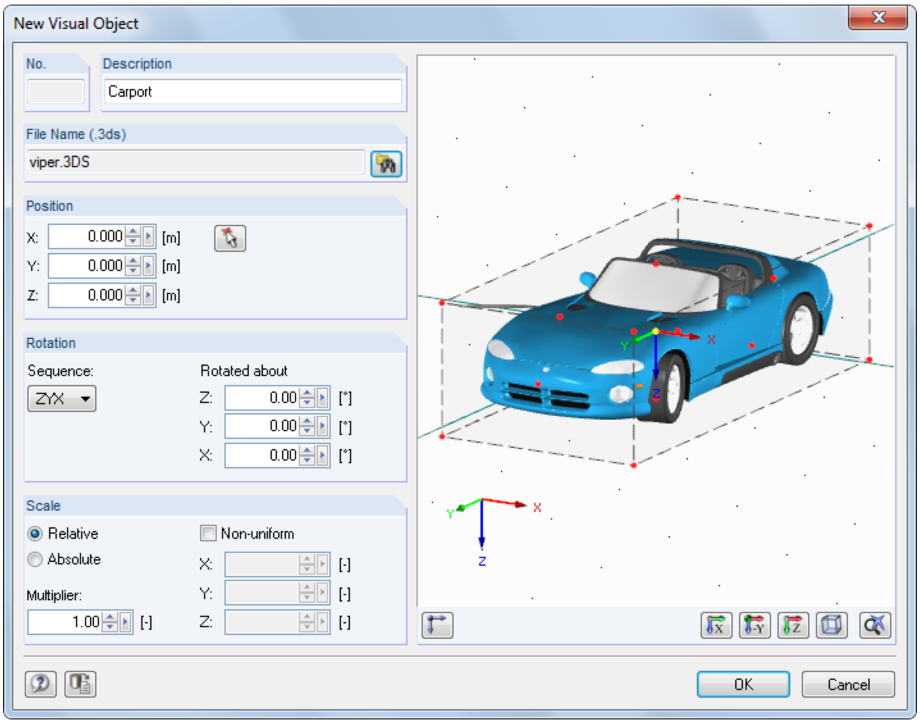


Figure 11.66: Dialog box *New Visual Object*

The visual object must be available in the format *.3ds*. Use the [Browse] button to select the file in the Windows dialog box *Open*.

Define the *Position* of the object in the model by entering coordinates. You can also use the [↖] function to define it graphically in the work window. The reference point of the 3D object is indicated by the selection color in the graphic to the right.

In addition, it is possible to define a *Rotation* of the object or to *Scale* the object.

Click [OK] to insert the object into the model.

The edit dialog box of a visual object can be accessed by double-clicking the object in the graphic or in the *Data* navigator.

11.3.10 Background Layers

A DXF file can be imported as background layer and used for the graphical input of objects. In contrast to the DXF import (see chapter 12.5.2, page 571) where the complete model is loaded being converted into nodes and lines, background layers represent some sort of transparent sheets for specific modeling.

It is possible to use several background layers in a model.

Create background layer

To open the dialog box for creating a new background layer,

select **Background Layer** on the **Insert** menu

or use the context menu in the *Data* navigator.

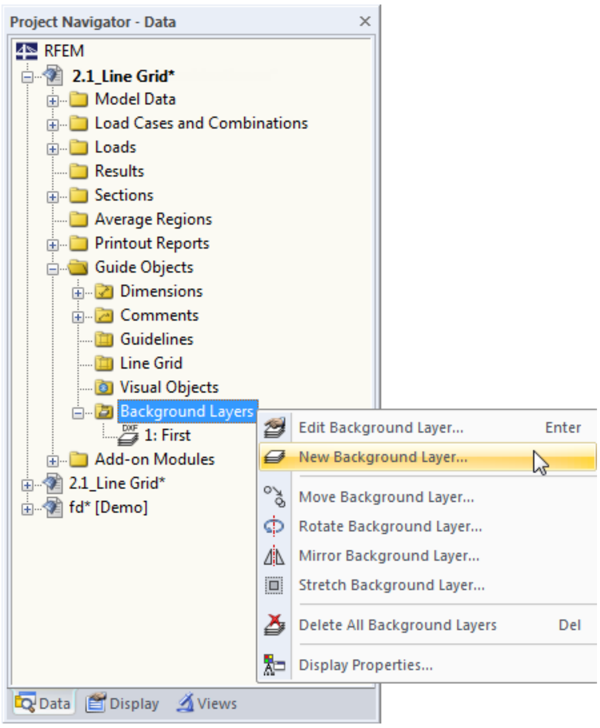


Figure 11.67: Context menu of *Background Layers* in the *Data* navigator

The Windows dialog box *Open* appears. Enter the directory and the name of the DXF file.

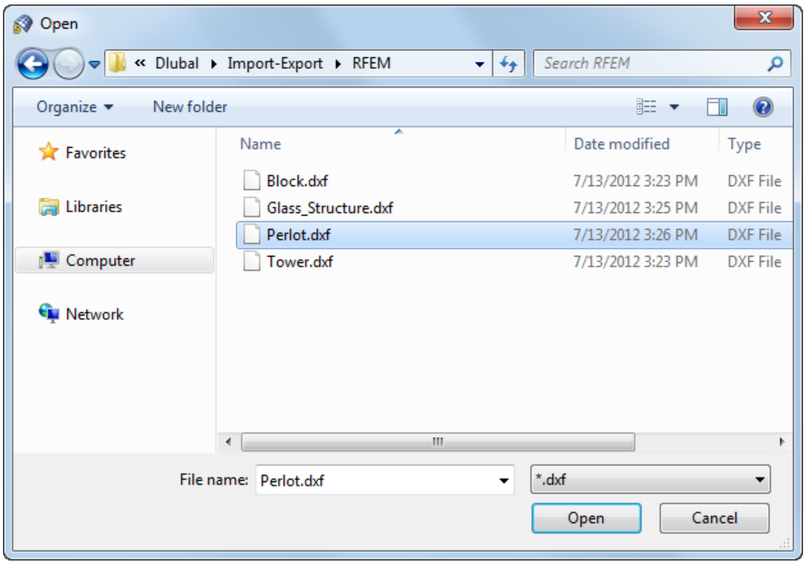
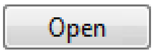


Figure 11.68: Dialog box *Open*



Click the [Open] button to access the dialog box *Background Layer*.

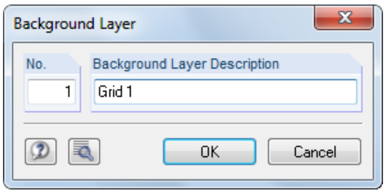


Figure 11.69: Dialog box *Background Layer*

The *No.* of the layer is allocated by the program. In the dialog section *Background Layer Description*, you can enter any name making the assignment easier later.

Use the [Edit] button shown on the left to access more settings for the DXF import. Details on the dialog box can be found in Figure 12.49 on page 571.

After clicking [OK] RFEM imports the layer which appears gray in the background of the work window. In the line model, you can now define nodes, lines and members.

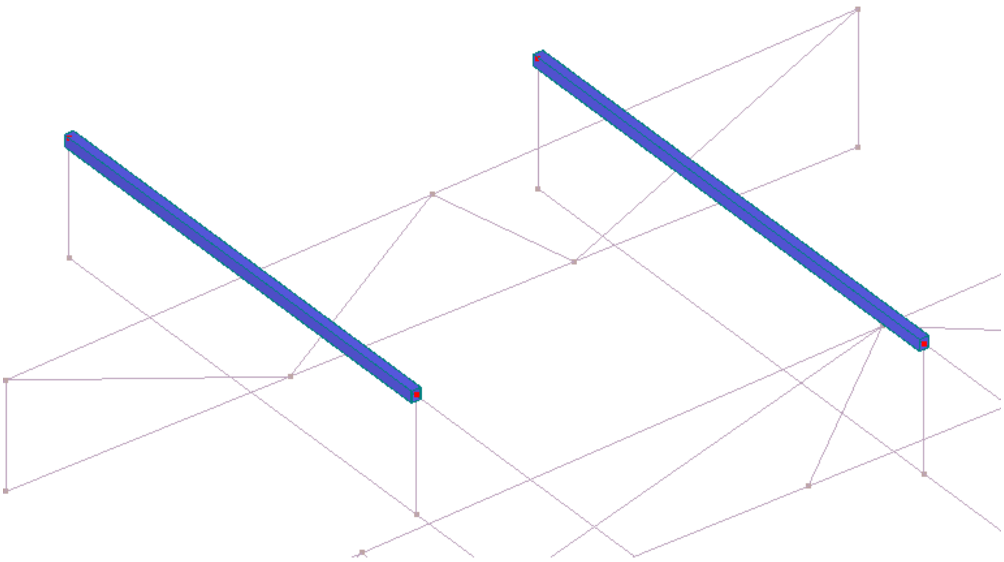


Figure 11.70: Defining members with background layer

Make sure that the object snap for background layers is activated so that you can arrange objects on the points available in the layer. To activate the object snap for DXF points, use the [DXF] button in the status bar. Alternatively, select *Work Plane, Grid/Snap, Object Snap, Guidelines* on the *Tools* menu, or use the toolbar button shown on the left.

The dialog box *Work Plane and Grid/Snap* opens. In the dialog tab *Background Layers*, you cannot only activate the snap but edit, delete or hide and display layers as well as create new ones.

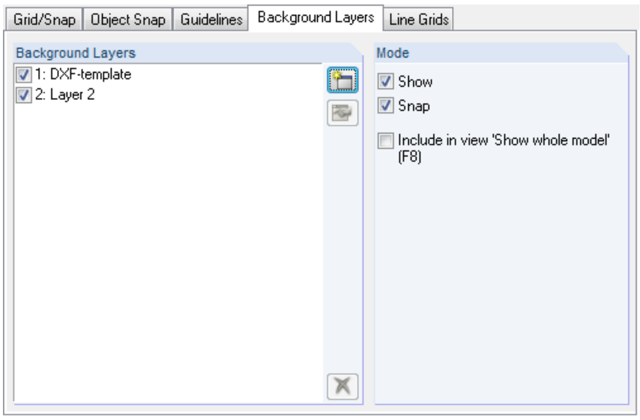


Figure 11.71: Dialog box *Work Plane and Grid/Snap*, tab *Background Layers* (dialog section)

Edit, delete or copy background layer

To open the edit dialog box, double-click the background layer or the relevant entry in the *Data* navigator (see Figure 11.67, page 455). You can also use the dialog tab *Background Layers* available in the dialog box for work plane settings (see Figure 11.71): After selecting the layer in the list, you can [Edit] it.

Deleting a background layer is also possible in the *Data* navigator.

To move, copy or mirror a background layer, select the layer first. Then, you can apply the function described in chapter 11.4.1 on page 459.

Display of background layers

The *Display* navigator controls the representation of background layers in detail.

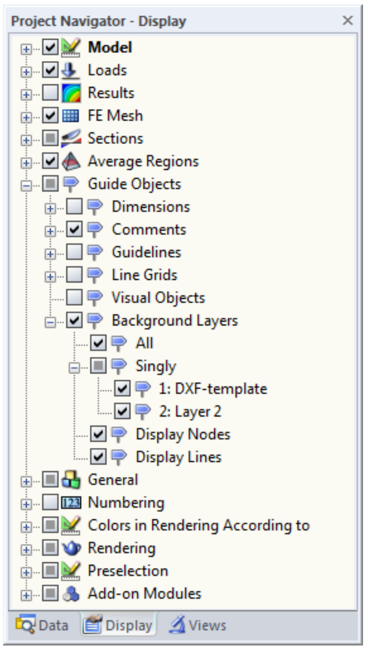
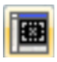


Figure 11.72: Background layer settings in the *Display* navigator

11.3.11 Margins and Stretch Factors



In most cases, it is not required to change the full screen arrangement or the scaling of the model in the work window. But if you have to adjust the global display parameters,

select **Display Margins and Stretch Factors** on the **Options** menu

to open a dialog box managing the default settings.

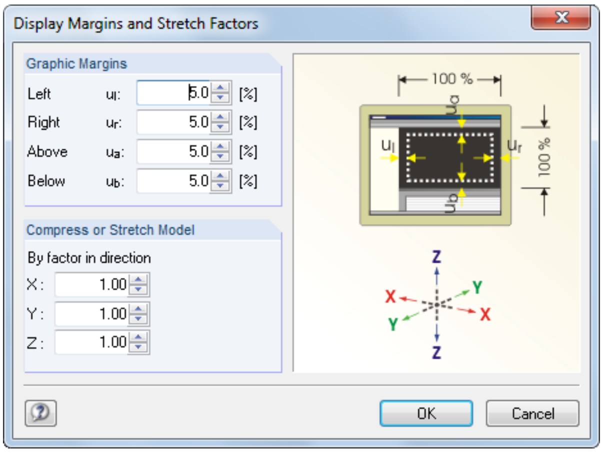


Figure 11.73: Dialog box *Display Margins and Stretch Factors*

Buttons of menu item *Select View*

Settings in the dialog section *Graphic Margins* determine the minimum distances that are kept for the representation of the model on the four sides of the work window's margins. The values are set in percentage and refer to the total height or width of the work window. They have an impact when using the buttons of the menu item *Select View* on the *View* menu (see figure on the left) or the function *Show Whole Model* [F8] for the window-filling graphical representation.

To display the model in a distorted view, you can define factors unequal to 1 for the global directions in the dialog section *Compress or Stretch Model*. However, customizing settings in this dialog section may be required only in exceptional cases. They affect only the display of the model but not the actual geometry. To scale the model, use the *Scale* function available on the *Edit* menu (see chapter 11.4.5, page 465).

11.4 Edit Functions

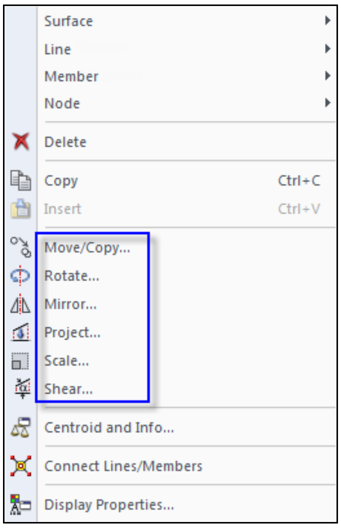
Use the graphical editing functions to modify objects previously selected in the graphic. The selected objects can be

- moved
- copied
- rotated
- mirrored
- projected
- scaled
- extruded
- sheared.

No selection is needed for the CAD functions described in chapter 11.3. The functions described on the following pages help you to model new objects.

This chapter also describes how to divide lines, place comments or change numberings.

11.4.1 Move and Copy



Context menu of selected objects

To move or copy selected objects,

select **Move/Copy** on the **Edit** menu

or use the context menu of the corresponding object. You can also use the toolbar button shown on the left.

Figure 11.74: Button *Move and/or Copy*

The following dialog box appears:

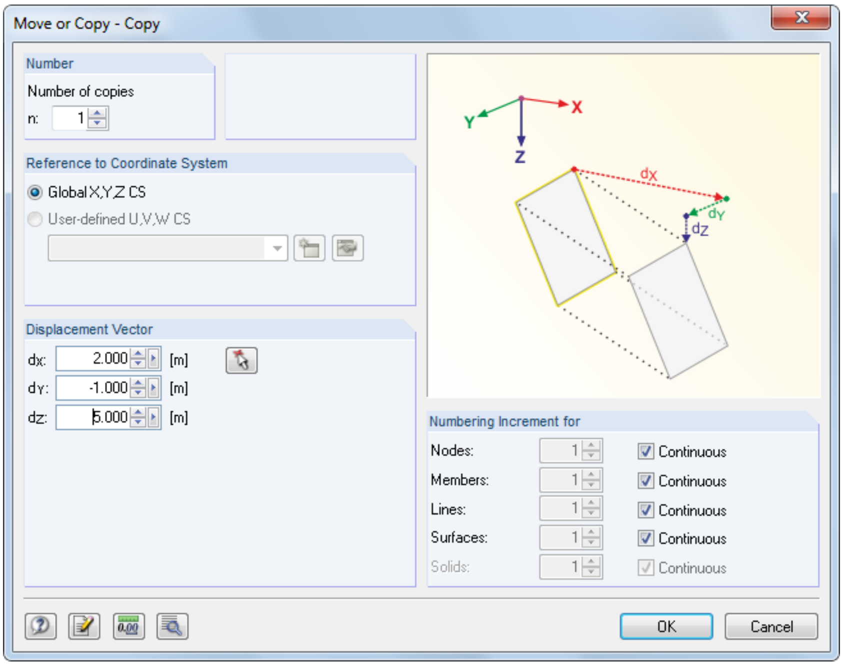


Figure 11.75: Dialog box *Move or Copy - Copy*

When the *Number of copies* is set to ***0***, the selected objects will be moved. Otherwise, the entered number of copies will be generated.

With the options in the dialog section *Reference to Coordinate System* you decide whether the objects are moved or copied in the global coordinate system XYZ or in a user-defined system of coordinates UVW (see 11.3.4, page 442). The user-defined coordinate system can be selected in the list or created with the [New] button.

The *Displacement Vector* is specified by the distances d_X, d_Y and d_Z, or d_u, d_v and d_w for a user-defined coordinate system. The vector can also be determined in the work window by using the [↖] function or by clicking two grid points or nodes.

If copies are created, you can influence the numbering of new nodes, members, lines, surfaces and solids in the dialog section *Numbering Increment for*.

Click the [Edit] button shown on the left to open another dialog box offering useful options for copying. The same dialog box is used also for other functions such as mirroring, rotating etc.

Detail settings

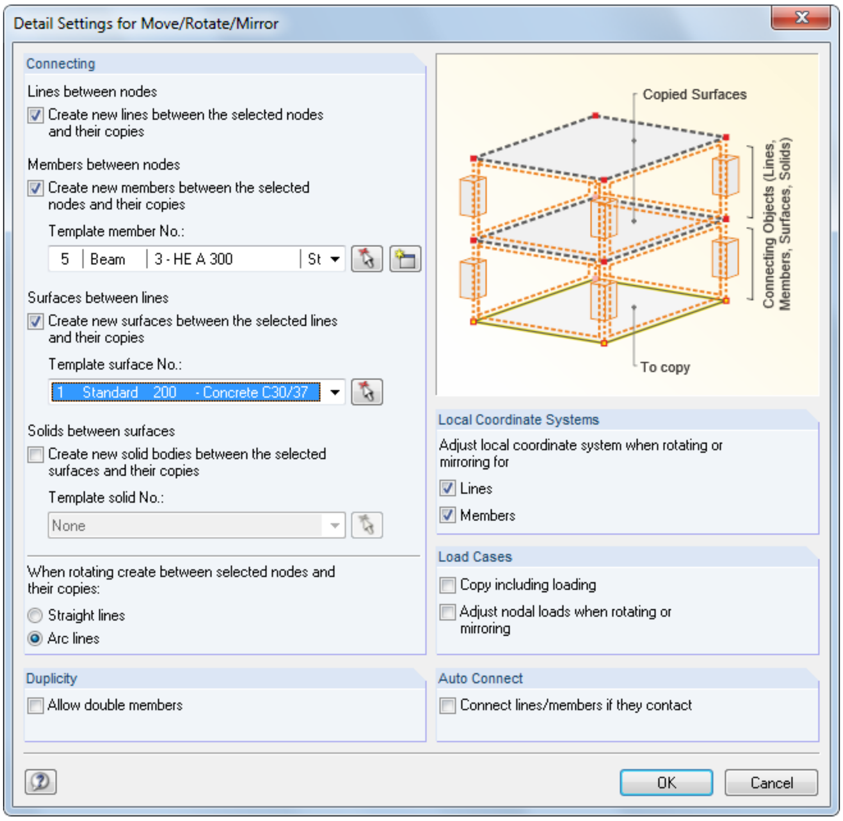


Figure 11.76: Dialog box *Detail Settings for Move/Rotate/Mirror*

Connecting

You can create new *Lines* and *Members* between the selected nodes and their copies. In addition, it is possible to generate *Surfaces* and *Solids* between the selected lines or surfaces and their copies.

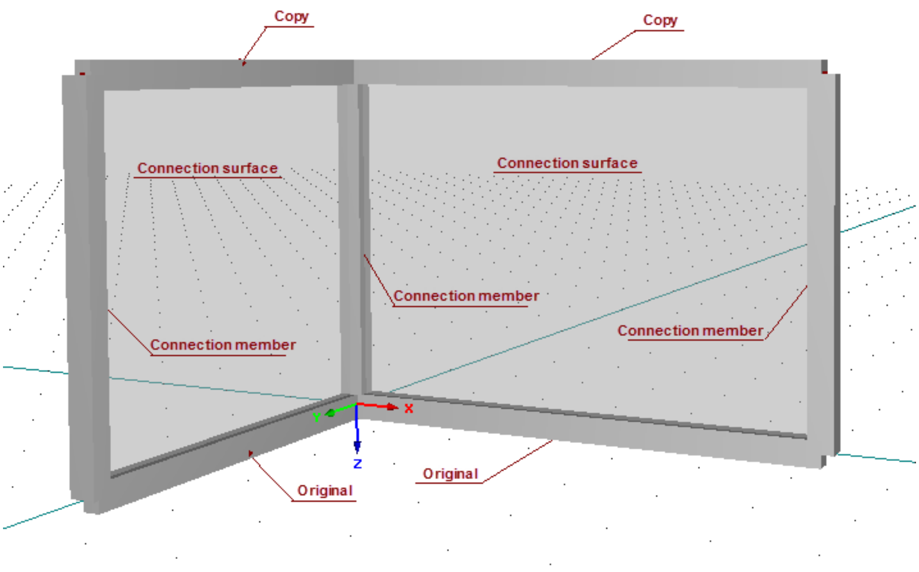


Figure 11.77: Copy with connecting members and connecting surfaces

When a *Template member*, *Template surface* or *Template solid* is selected in the list or in the graphic by using the [↖] function, its properties are used for the connecting objects.

Duplicity

Double members may be created when copying. Use the check box to decide if overlapping members are allowed or merged to be one member.

Local coordinate systems

You can adjust the local line and member coordinate systems to the new position when rotating and mirroring.

The automatic adjustment of local axes often becomes important when mirroring objects. The function proves to be useful as well for rotating a vertical member as its axis y is oriented parallel to the global Y-axis (see chapter 4.17, page 146).

Moreover, the function adjusts eccentric connections that are defined in direction of the global axes X, Y and Z.

Load cases

If the check box for *Copy including loading* is ticked, the loads acting on the selected objects will be transferred to the copies. Please note that the loads of all load cases will be copied, not only the loads of the currently selected load case.

Nodal loads can be defined only in direction of the global axes X,Y,Z. If you want to influence the direction of nodal loads when copying the surfaces or members, use the check box *Adjust nodal loads when rotating or mirroring*. When it is ticked, RFEM will convert the loads like local concentrated loads to the new position. In this case, make sure that the members are selected together with the nodal loads before rotating or mirroring. If the check box is clear, the global load direction will be kept.

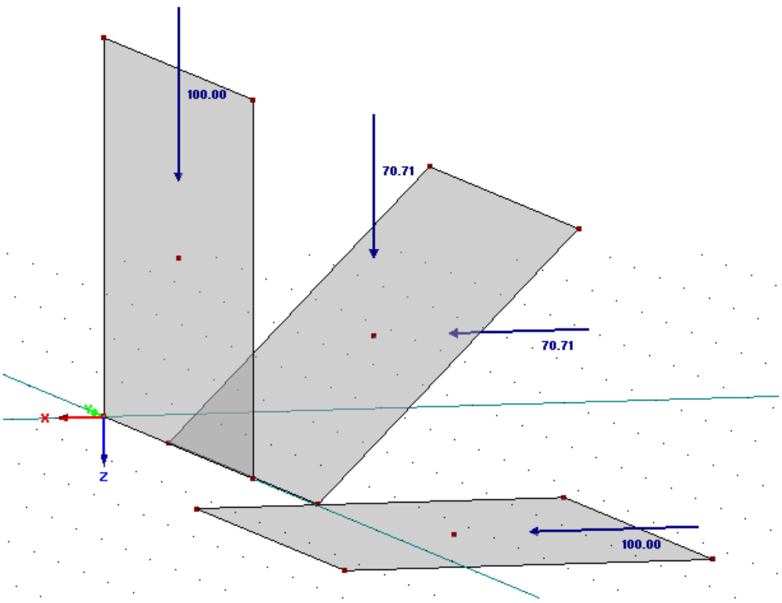


Figure 11.78: Adjusted nodal loads when rotating two times about 45°

Auto connect

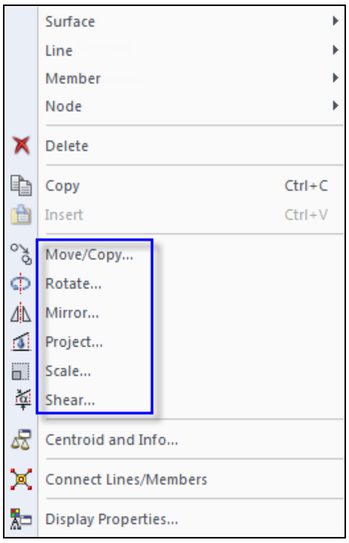
Use the check box to decide if the copies of the lines and members will be connected automatically to the already existing lines and members. When the box is ticked, a node will be created in the point of intersection.

11.4.2 Rotate

To rotate selected objects about a particular axis,

select **Rotate** on the **Edit** menu,

use the context menu of the corresponding object or the toolbar button shown on the left.



Context menu of selected objects

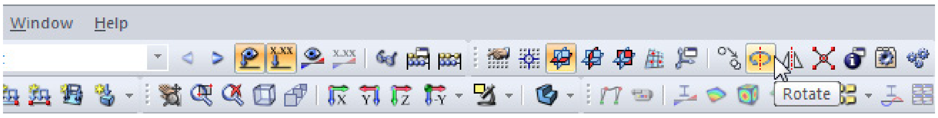


Figure 11.79: Button *Rotate*

The following dialog box appears:

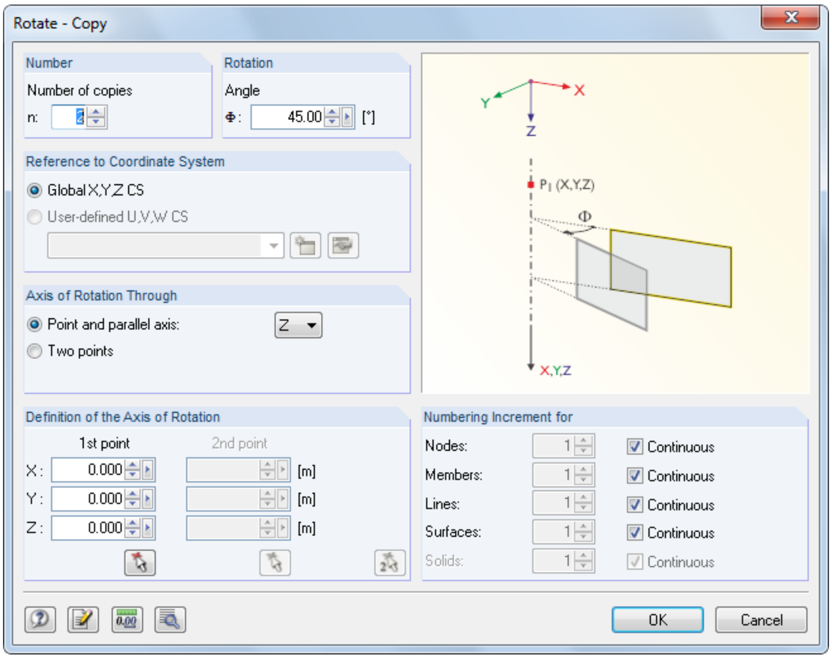


Figure 11.80: Dialog box *Rotate - Copy*

When you set the *Number of copies* to ***0***, the selected objects will be rotated. Otherwise, the entered number of copies will be generated.

Enter the rotation angle in the dialog section *Rotation*. The angle refers to a coordinate system that is clockwise-oriented.

The *Axis of Rotation* can be defined in two ways:

- The rotation axis runs parallel to an axis of the global axis system X,Y,Z. In this case, activate the first option and select the relevant axis from the list to the right. Then, in the dialog section *Definition of the Axis of Rotation*, specify a point through which the rotation axis is running.
- The rotation axis lies anywhere in the work plane. In this case, activate the second option. Then, in the dialog section *Definition of the Axis of Rotation*, specify two points defining the rotation axis.

If copies are created, you can influence the numbering of new objects in the dialog section *Numbering Increment for*.

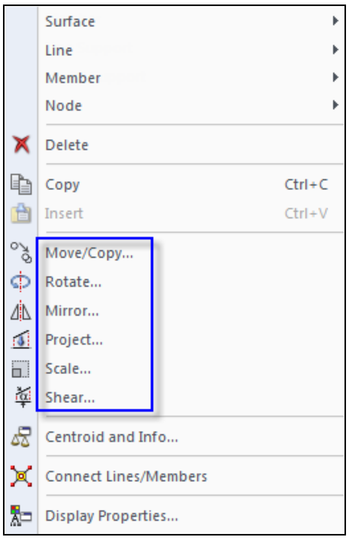
Use the [Edit] button shown on the left to open another dialog box with useful options that are described in chapter 11.4.1 on page 460. With entries in the dialog box for detail settings you can determine if the connecting lines created when copying are generated as straight lines or arcs.

11.4.3 Mirror

To mirror selected objects on a plane,

select **Mirror** on the **Edit** menu

or use the context menu of the corresponding object. You can also use the toolbar button shown on the left.



Context menu of selected objects

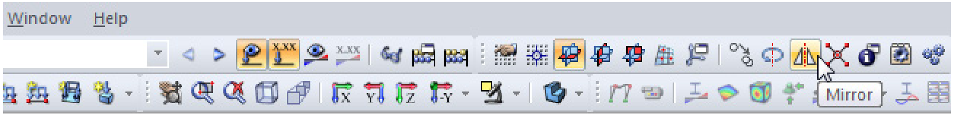


Figure 11.81: Button *Mirror*

The following dialog box appears:

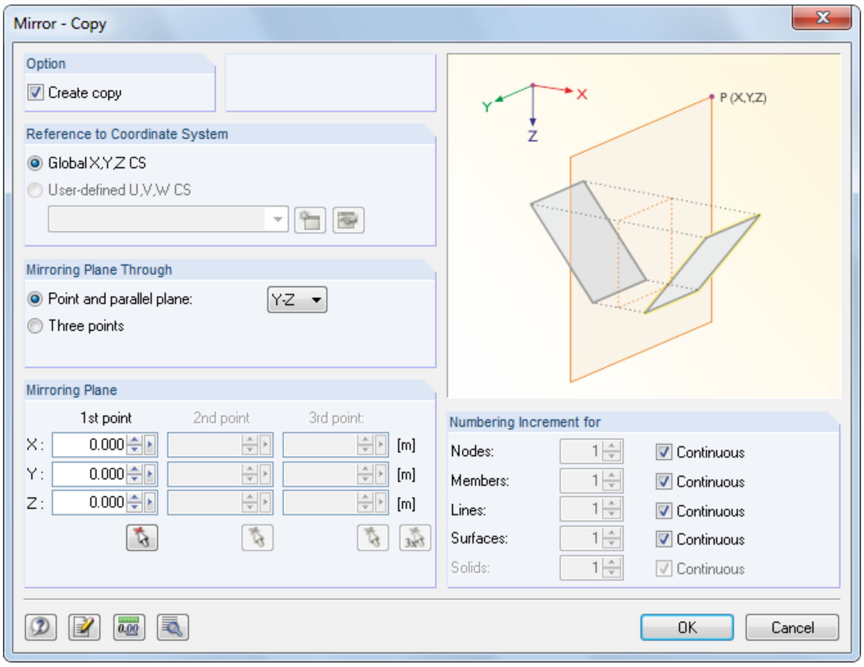


Figure 11.82: Dialog box *Mirror - Copy*

To maintain the original object, tick the check box for *Create copy*.

The *Mirroring Plane* can be defined in two ways:

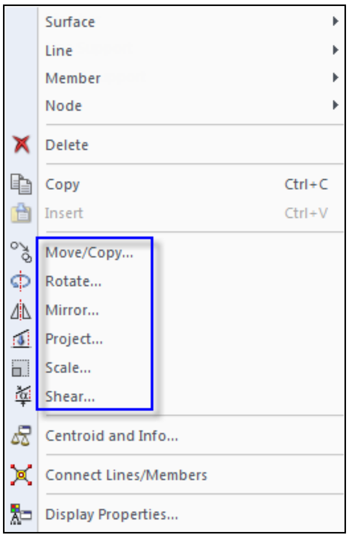
- The mirroring plane runs parallel to a plane that is spanned by the axes of the global axis system XYZ. In this case, activate the first option and select the relevant plane from the list to the right. Then, in the dialog section *Mirroring Plane*, enter a point lying in the plane set above.
- The mirroring plane lies anywhere in the work plane. In this case, activate the second option. Then, in the dialog section *Mirroring Plane*, enter three points that define the plane.

If a copy is created, you can influence the numbering of new objects in the dialog section *Numbering Increment for*.

Use the [Edit] button shown on the left to open another dialog box with useful options that are described in chapter 11.4.1 on page 460.

11.4.4 Project

Use this function to project selected objects on a plane. Thus, you can adjust for example the inclination angle of horizontal beams or rafter members.



Context menu of selected objects

Example

A member is projected in direction X on the plane YZ.

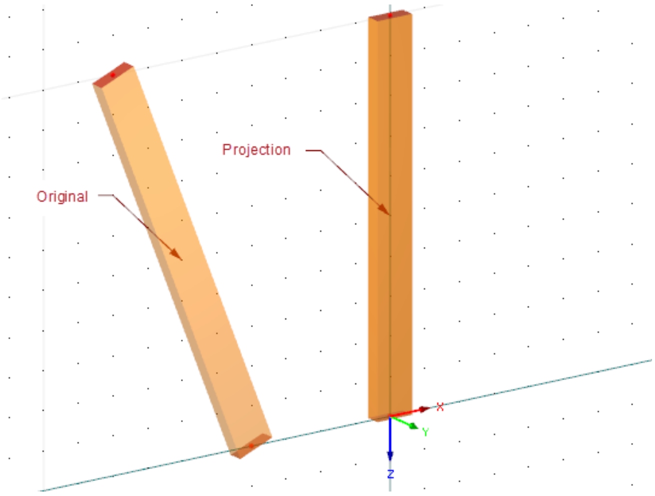


Figure 11.83: Original member and projected copy on plane YZ

To open the dialog box for entering the projection parameters,

select **Project** on the **Edit** menu

or use the context menu of the selected objects.

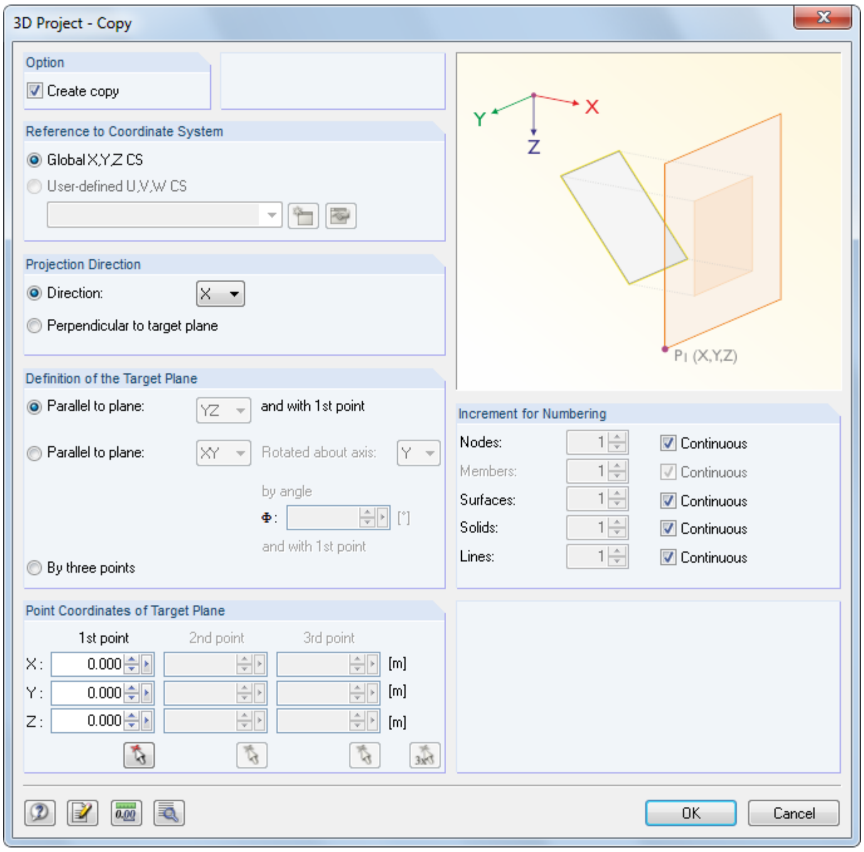


Figure 11.84: Dialog box *3D Project - Copy*

To maintain the original object, tick the check box for *Create copy*.

In the dialog section *Projection Direction*, you can decide whether the objects will be projected in the direction of a global axis (X, Y or Z) or perpendicular to a target plane.

The *Target Plane* can be defined in the following three ways:

- The target plane runs parallel to a plane that is spanned by the axes of the global axis system X,Y,Z. In this case, activate the first option and select the relevant plane from the list to the right. Then, in the dialog section *Point Coordinates of Target Plane*, enter a point that lies in the plane set above.
- The target plane runs parallel to a plane that is spanned by the axes of the global axis system XYZ but is additionally rotated about one of the axes. In this case, activate the second option. In the list to the right, select the relevant plane and specify the axis and angle of rotation. Then, in the dialog section *Point Coordinates of Target Plane*, enter a point that lies in the plane set above.
- The target plane lies anywhere in the work plane. In this case, activate the third option. Then, in the dialog section *Point Coordinates of Target Plane*, define the plane by entering three points.

If a copy is created, you can influence the numbering of new objects in the dialog section *Increment for Numbering*.

Use the [Edit] button shown on the left to open another dialog box with useful options that are described in chapter 11.4.1 on page 460.

11.4.5 Scale

Use this function to scale selected objects in relation to a point.

Example

A quadrangle surface is equally scaled starting from the origin in all three directions by the factor 2.

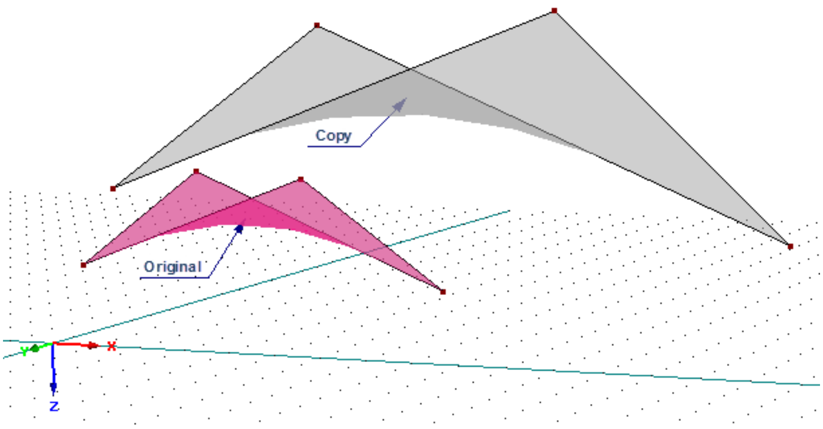
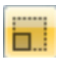


Figure 11.85: Original surface and scaled copy



To open the dialog box for entering the scale parameters,

select **Scale** on the **Edit** menu

or use the context menu of the selected objects (see figure in the margin to the left of Figure 11.83).

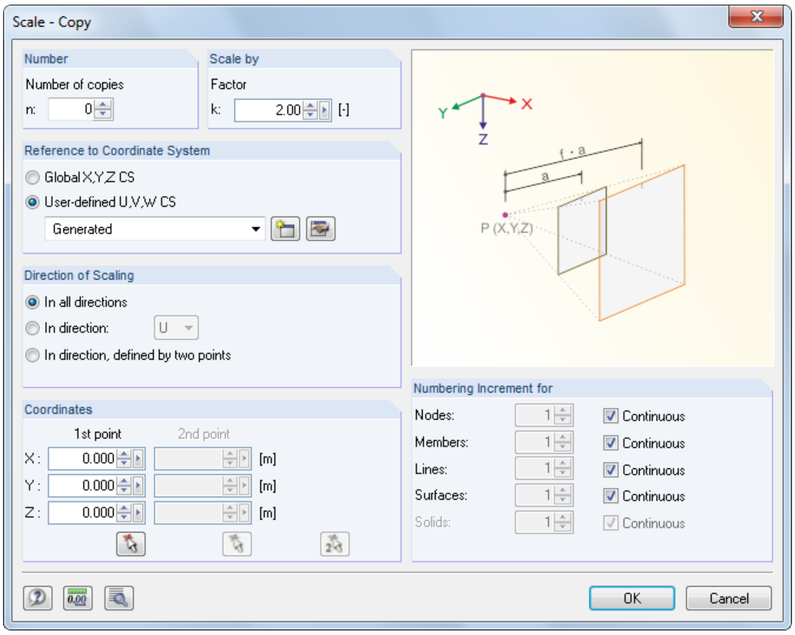


Figure 11.86: Dialog box *Scale - Copy*

When the *Number of copies* is set to ***0***, the selected objects will be scaled. Otherwise, the entered number of copies will be generated.

The dialog section *Scale by* manages the scaling factor *k* (see graphic in the dialog box).

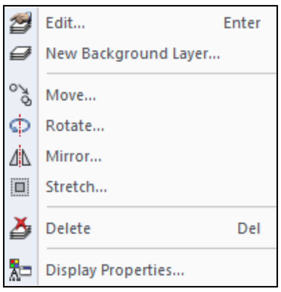
Three possibilities are available for selection to define the *Direction of Scaling*:

Equally in X,Y,Z	All object coordinates (X, Y and Z) will be scaled in relation to the starting point defined in the dialog section *Coordinates*.
In direction: X / Y / Z	You define one of the global axis. Only the object's coordinates of the selected global axis will be scaled in relation to the starting point defined in the *Coordinates* dialog section.
In direction, defined by two points	In the dialog section *Coordinates*, specify a vector by entering two points. Objects will be scaled into the vector direction.

Table 11.7: Dialog section *Direction of Scaling*

If a copy is created, you can influence the numbering of new objects in the dialog section *Numbering Increment for*.

Use the [Edit] button shown on the left to open another dialog box with useful options that are described in chapter 11.4.1 on page 460.



Context menu of background layers

It is also possible to scale background layers. To access the corresponding function,

point to **Background Layers** on the **Edit** menu, and then select **Stretch**

or use the context menu of background layers in the *Data* navigator.

In the dialog box *Select Background layer*, specify the relevant layer first. Then, you can define the stretch factor in the dialog box *Stretch Background Layer*.

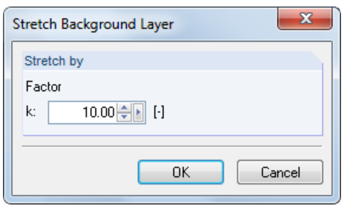


Figure 11.87: Dialog box *Stretch Background Layer*

11.4.6 Shear

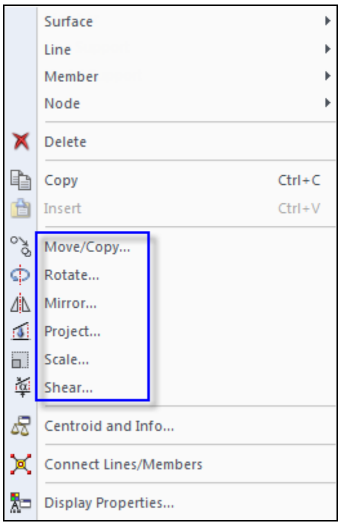
The function rotates objects about an axis and adjusts only the coordinates of one single direction. You can use the shear function for example to shift horizontal members into the inclination plane of a roof. The member lengths will be adjusted, the horizontal components of the coordinates remain unchanged.

Before you use the function, select the members together with the nodes that belong to them.

To open the dialog box for entering the shearing parameters,

select **Shear** on the **Edit** menu

or use the context menu of the selected objects.



Context menu of selected objects

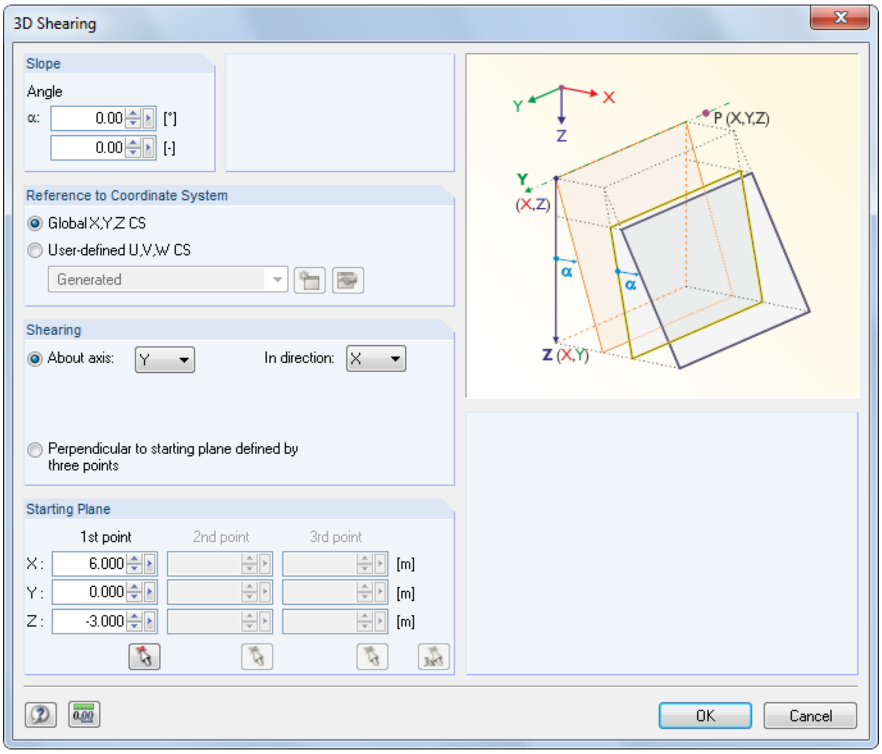


Figure 11.88: Dialog box *3D Shearing*

In the dialog section *Slope*, enter the rotation angle in [°] or [%].

The parameters for *Shearing* can be defined in two ways:

- The rotation axis runs parallel to a plane that is spanned by the axes of the global axis system XYZ. In this case, activate the option *About axis* and select the relevant axis of rotation from the list to the right. Then, in the list *In direction*, select the global axis that is relevant for adjusting the node coordinates. Finally, in the dialog section *Starting Plane*, enter the point of rotation.

- The rotation axis lies anywhere in the work plane. In this case, activate the second option. Then, in the dialog section *Starting Plane*, define both points of the rotation axis and another point for determining the plane. You can select the points also graphically by using the [↖] buttons.

11.4.7 Divide Lines and Members

Lines and members can be divided quickly: Right-click the object and select *Divide Line* or *Divide Member* in the context menu.

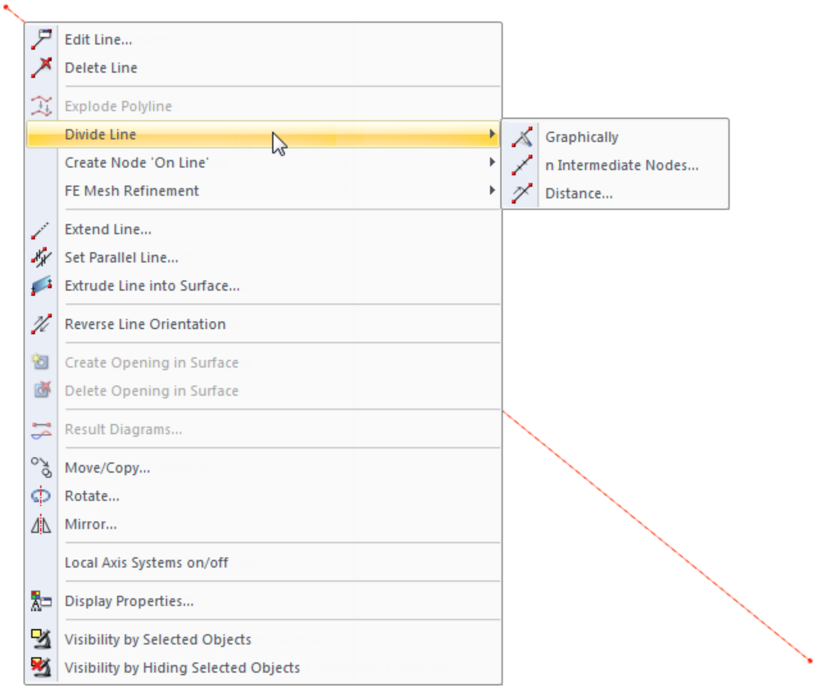


Figure 11.89: Context menu *Divide Line*

The menu item offers three division options.

Graphically

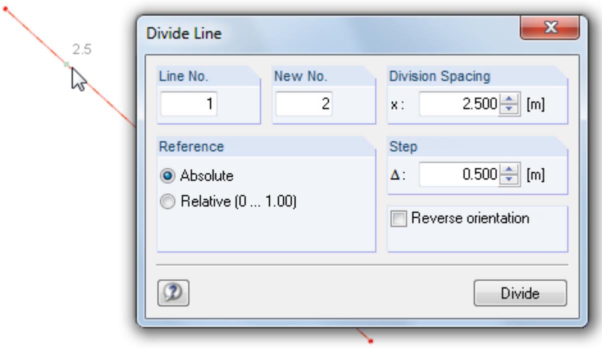


Figure 11.90: Dialog box *Divide Line*

The dialog box *Divide Line* opens. When you move the pointer along the line, it will be snapped at the distances specified in the dialog section *Step*. Click to define the division point. The *Reference* of the division spacings can be set in absolute distances or relatively to the total length.

It is also possible to enter the *Division Spacing* directly. Before entering the spacing, specify the line that you want to divide and the number of the new line in the input fields *Line No.* and *New No.* If you want to relate the division spacing to the line end, you can change the line orientation with the check box *Reverse orientation*.

n intermediate nodes

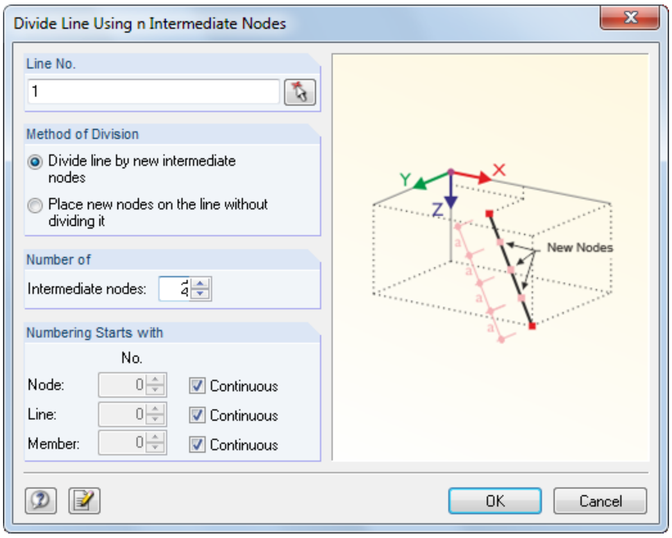


Figure 11.91: Dialog box *Divide Line Using n Intermediate Nodes*

Use this function to divide the line equally into several line parts. In the dialog section *Number of*, you can define the number of *Intermediate nodes* for the member division.

Decide if you want to divide the line into "real" lines by *new intermediate nodes*, or maintain the line while RFEM will create *nodes on the line* in equal spacings. Usually, the real division is preferred. However, if you want to change the course of a B-Spline line when dividing the line, the second option is the better choice.

In the dialog section *Numbering Starts with*, you can influence the numbering of new nodes, lines and members.

Distance

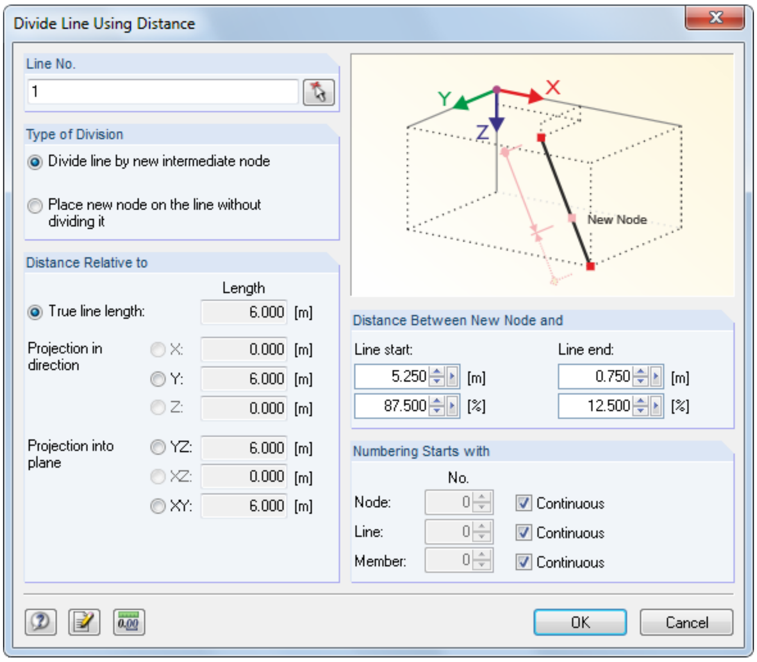


Figure 11.92: Dialog box *Divide Line Using Distance*

Use this function to generate a division node on a particular location of the line.

The line is either divided into "real" lines by a *new intermediate node* or maintained while RFEM creates a *new node on the line*.

Settings in the dialog section *Distance Relative to* control the reference of the division distance. The distance can be referred to the real line length (normal case) or to a projection.

The *Distance Between New Node and* start or end node of the line is to be specified as absolute value or relatively to the total length. The four input fields are interactive.

For entering the distance it is important to know the line or member orientation. The orientations and axis systems of lines and members can be switched on and off in the context menu or the *Display* navigator (see Figure 4.26, page 50 and Figure 4.158, page 145).

The dialog section *Numbering Starts with* controls the numbering of new objects.

11.4.8 Connect Lines and Members

Use this function to connect lines and members that cross each other but do not have a common node.

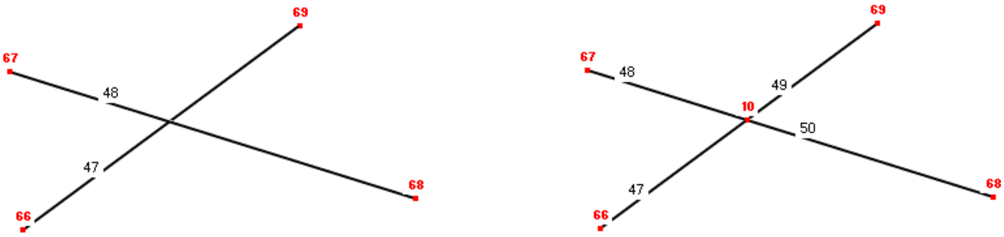


Figure 11.93: Original on the left (intersecting, unconnected lines) and result on the right (connected lines)

To access the corresponding function,

select **Connect Lines/Members** on the **Tools** menu

or use the toolbar button shown on the left.

Figure 11.94: Button *Connect Lines or Members*

Go into the work window and draw a window across the zone where you want to connect the lines or members. It is not necessary to catch the objects completely.

Furthermore, the function can be used to determine the intersection point of a line piercing a surface.

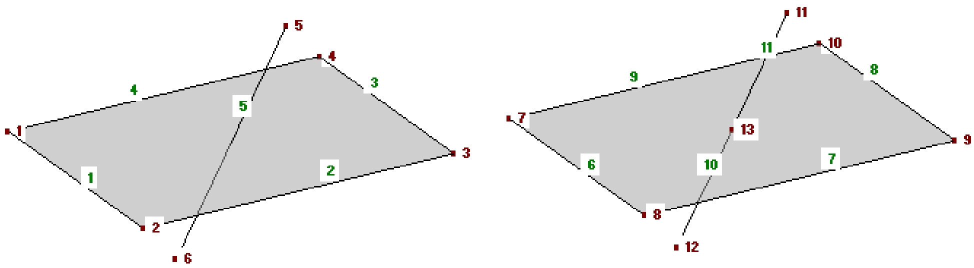


Figure 11.95: Creating an intersection point between line and surface: original (left) and copy with result (right)

The Auto Connect function is preset for setting new lines or members graphically, as shown in the figure below. But connection nodes will only be created when lines/members are connected to other lines/members, that means when they end on the corresponding object. So, when you define crossing diagonals, no intersection node will be generated.

In the *New Line* or *New Member* dialog box, you can use the [Details] button to determine if lines or members are connected automatically when they are generated.

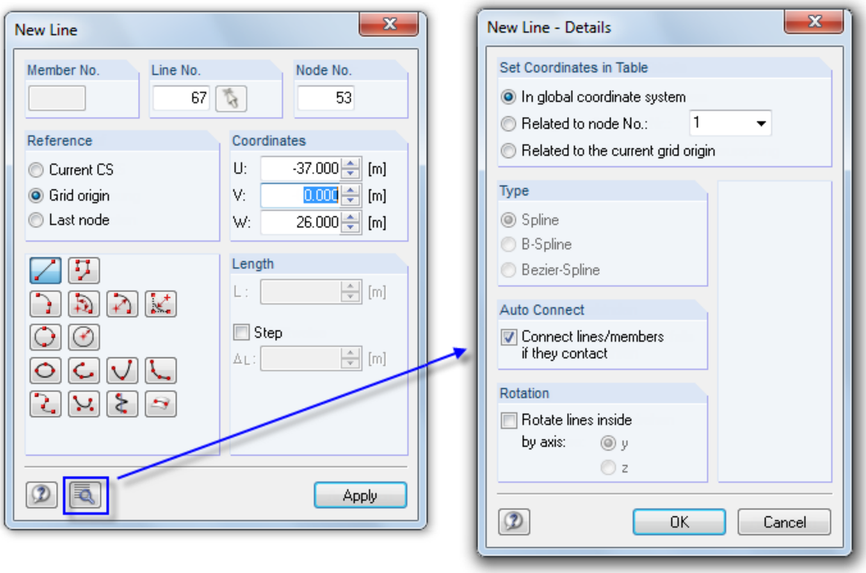


Figure 11.96: Dialog box *New Line - Details*

11.4.9 Merge Lines and Members

Lines or members that are connected with each other can be unified to be one single line or member. This function is only available in the context menu of division nodes. Right-click the division node to open its context menu.

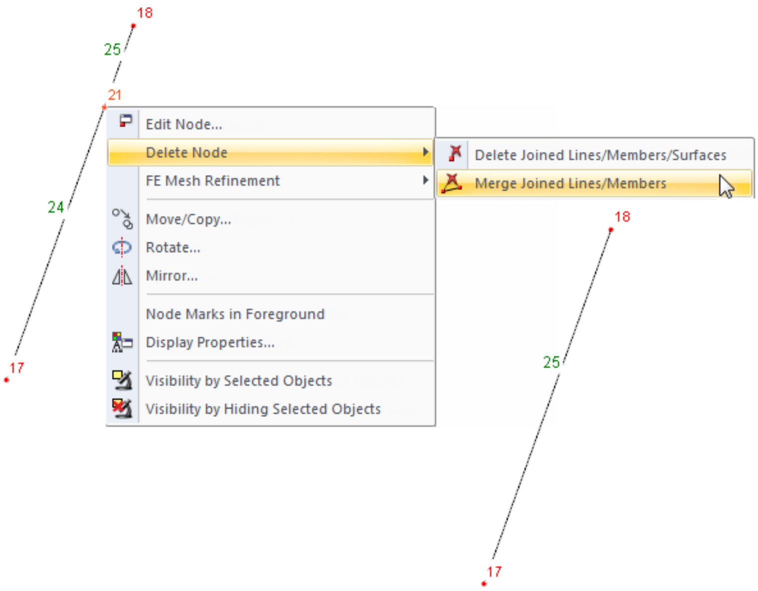
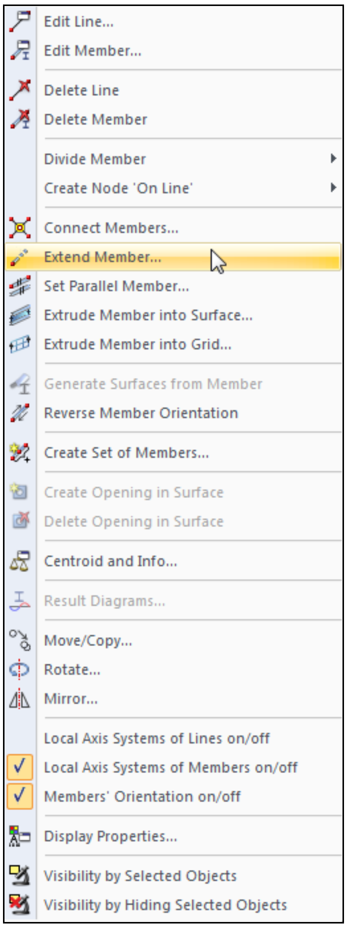


Figure 11.97: Context menu items *Delete Node* → *Merge Joined Lines/Members* with result (on the right)

The context menu offers extended options for the function *Delete Node* whereas the [Del] key simply deletes the selected node and consequently the joined lines, members and surfaces. But these special options are only provided for nodes to which exactly two lines or members are connected.

In case the lines or members do not lie on a straight line, RFEM will create a new line or member between the edge nodes when merging.

11.4.10 Extend Lines and Members



Member context menu

Use this function to adjust the length of a line or member in general, or to extend the line until it reaches another line.

To access the extend function, use the line context menu (see Figure 11.89, page 468) or the member context menu shown on the left.

The dialog box *Extend Line* or *Extend Member* opens.

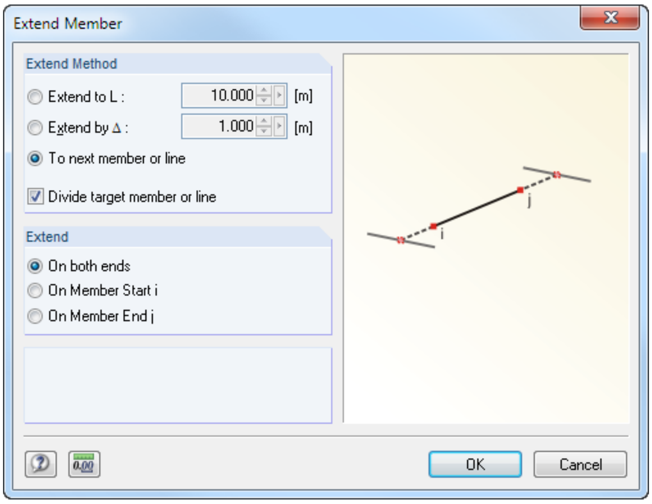


Figure 11.98: Dialog box *Extend Member*

The dialog section *Extend Method* offers three options:

- *Extend to L* changes the total length of the line or member to a dimension that you specify in the input field.
- *Extend by* Δ extends one member side or both member sides by a specified value, or shortens the side(s) if the value in the input field is negative.
- Select *To next member or line* to extend the object to the nearest line that will produce an intersection with the extended straight line of the line or member. When the check box for *Divide target member or line* is ticked, the objects will be connected automatically.

Specify the direction of extension in the dialog section below: The option *On both ends* results in an extension at both ends of the member. With this setting you can either refer the total length L to the line or member center, or extend the line on both sides by the value Δ or until the next two lines are reached. Alternatively, use the options *On Member Start i* or *On Member End j* to adjust the length of the member on only one end.

The display of line or member orientations can be set in the *Display* navigator (see Figure 4.26 on page 50).

11.4.11 Join Members

Contrary to connecting members (see chapter 11.4.8, page 470), the function does not require a common point of intersection. Thus, free members available in a certain distance to a member can be joined to the nodes of this member. However, if you want to connect the member by extending the member, use the function *Extend Member* (see chapter 11.4.10).

To access the corresponding function,

select **Join Members** on the **Tools** menu.

The following dialog box appears.

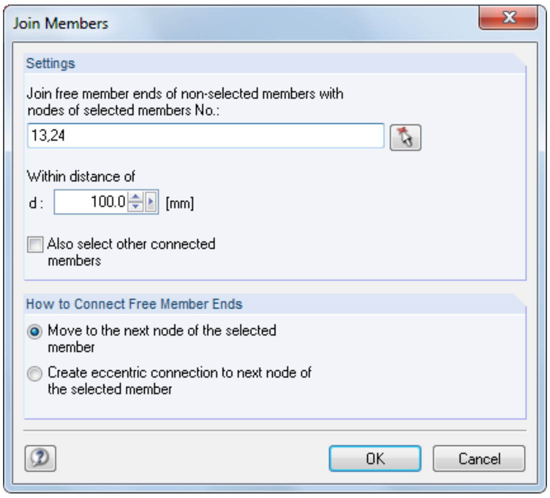


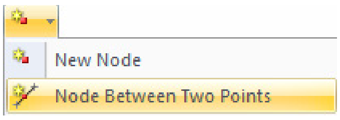
Figure 11.99: Dialog box *Join Members*

In the dialog section *Settings*, enter the number of the member to whose nodes you want to join the free members. You can select the member also graphically with the [↖] function. The input field below specifies the *distance*, that means the circumference where RFEM looks for free member ends. If the check box for *Also select other connected members* is ticked, RFEM will include also members that are connected with an already selected member into the member list of the input field above.

In the dialog section *How to Connect Free Member Ends*, you decide how RFEM will join the free member ends to the selected members: You can either move them to the nodes of the selected members or connect them by eccentric connections.

11.4.12 Insert a Node

Use this function to create a new node between any two nodes. In this way, you do not need to define a line and divide it by an intermediate node (see chapter 11.4.7, page 468).



To access the corresponding function,

point to **Model Data** on the **Insert** menu, select **Nodes** and click **Node Between Two Points**

or use the list button [New Node] in the toolbar.

Select the two points (nodes, grid points, any points) one after the other in the work window. Then, the following dialog box appears:

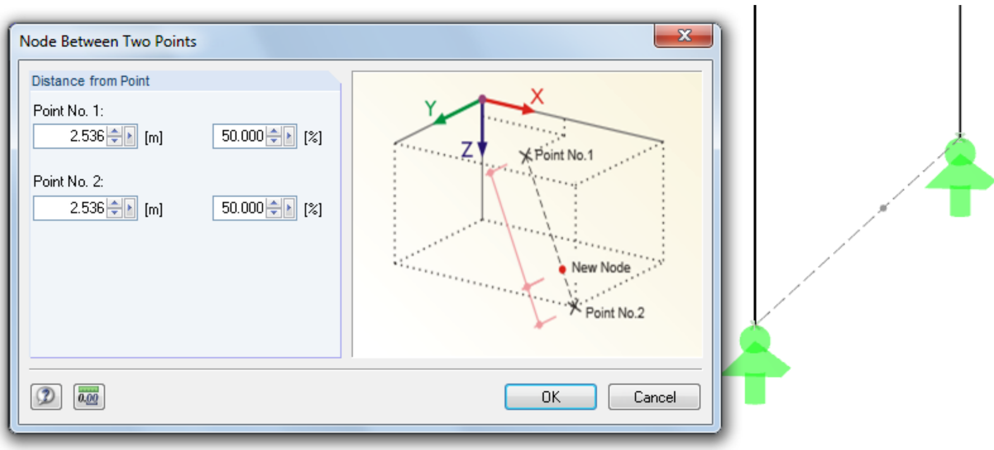


Figure 11.100: Dialog box *Node Between Two Points*

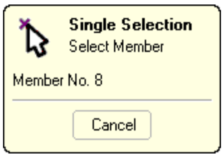
The *Distance from Point* can be defined in absolute or relative values. The work window shows you modifications immediately. To create the new node, click [OK].

11.4.13 Insert a Member

It is possible to define on an existing member a section that has different cross-section properties. The original member will be divided by two intermediate nodes.

To access the corresponding function,

> point to **Model Data** on the **Insert** menu, then select **Members** and **Graphically** and click **Inserted Member**.



After selecting the relevant member in the work window, the following dialog box appears:

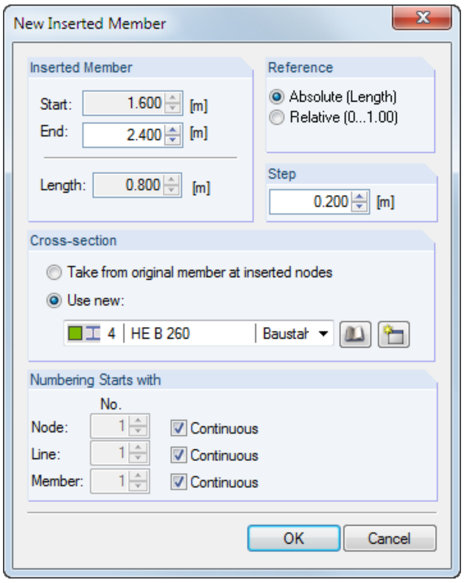


Figure 11.101: Dialog box *New Inserted Member*

Define both division points by mouse clicks in the work window. A cross on the pointer position indicates the current division point on the member. The distances shown when moving the pointer along the member are controlled by the input field *Step*.

The x-locations of the start and end node are displayed in the input fields of the dialog section *Inserted Member* where they can be modified, if necessary. The *Length* of the intermediate member appears below.

With the options in the dialog section *Reference* you decide whether the division spacings are related to the absolute lengths or to the relative distances from the member start.

The *Cross-section* can either be accepted or assigned as a new one selected from the list of already defined cross-sections. With the buttons shown on the left you can create a [New] cross-section or select a cross-section from the [Library].

The dialog section *Numbering Starts with* controls the numbering of new objects.

11.4.14 Assign Member Properties Graphically

Use this function to transfer the definition criteria of members for cross-section, release and eccentricity graphically to already created members.

To access the corresponding function,

select **Model Data** on the **Insert** menu, point to **Members** and select **Assign Member Properties to Members Graphically** or

open the **Edit** menu, point to **Model Data** and **Members**, and then select **Assign Member Properties to Members Graphically**.

The following dialog box appears:

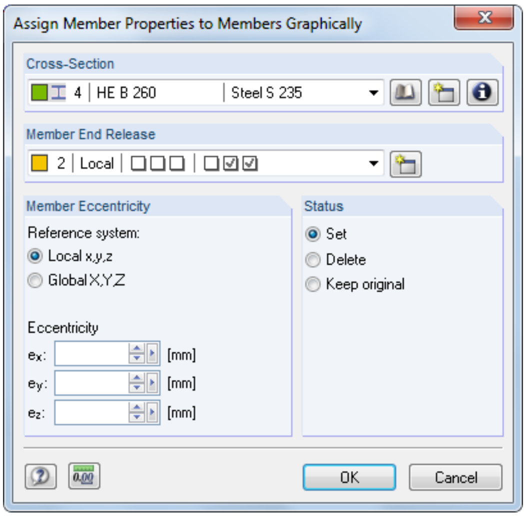


Figure 11.102: Dialog box *Assign Member Properties to Members Graphically*

Select the *Cross-Section* from the list or use the buttons shown on the left to select the cross-section from the [Library] or to create a [New] one. If necessary, you can define the *Member End Release* with a list, but it is also possible to create a [New] release type (see chapter 4.14, page 128).

You can relate the *Member Eccentricity* to the local member axis system xyz or the global coordinate system XYZ. If needed, define the eccentricity in the corresponding input fields (see chapter 4.15, page 134).

With the options in the dialog section *Status*, you decide if a member eccentricity is removed (*Delete*) or assigned as new (*Set*). Choose *Keep original* to change only the cross-section and the member end release but not any existing eccentricity.

After clicking [OK] you can see that the members are divided graphically at one-third division points (see Figure 4.135, page 129). Now, you can click the member sides to which you want to apply the selected properties (for example a release). To assign the release or the eccentricity to both member ends, click the member in its center.

11.4.15 Round Corners

Corners and edges in the model may result in singularity effects. To open the dialog box for modeling corners close to reality using fillet radii,

select **Create Round or Angled Corner** on the **Tools** menu.

It is not necessary to select both lines previously. The following dialog box appears:

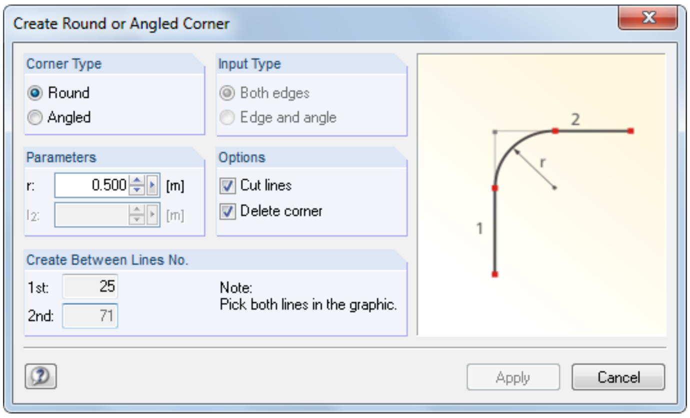


Figure 11.103: Dialog box *Create Round or Angled Corner*

In the dialog section *Corner Type*, you decide if the corner zone will be made *Round* or *Angled*. Depending on the selection, you have to enter the fillet radius *r* or a reduction by the lengths l_1 and l_2 in the dialog section *Parameters*.

Then, select both lines by mouse-click in the work window without closing the dialog box. The line numbers will be shown in the dialog section *Create Between Lines No*.

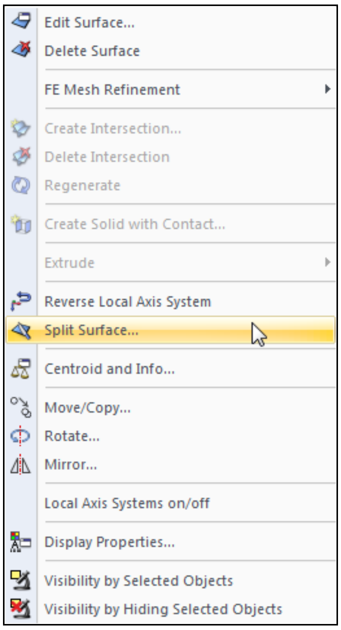
When the check box for *Cut lines* is ticked, RFEM deletes the extensions of the original lines overlapping in the corner zone after creating the arc or the new line. The option *Delete corner* removes also the node in the corner.

11.4.16 Split Surface

Surfaces can be split into surface components if one of the following conditions is fulfilled:

- The surface is defined by four lines and has no re-entrant corner. The lines are no intersecting lines, trajectory curves or similar objects.
- The surface is a rotated surface with a rotation angle of α < 360°.

To divide a surface, right-click it and select *Split Surface* in the context menu.



Surface context menu

A preview appears in the dialog box *Split Surface* (see Figure 11.104) suggesting a division that illustrates the preset parameters. Parameter settings and dialog graphic are interactive: When you change the *Number of divisions* for both boundary line pairs A + C and B + D, the graphic shows you the new sub-surfaces immediately.

It is possible to define a *Relative distance* for each division line. Irregular splitting patterns can be defined as well.

In the graphic window, you can use the common mouse functions such as zooming or rotating to change the view (see chapter 3.4.9, page 35).

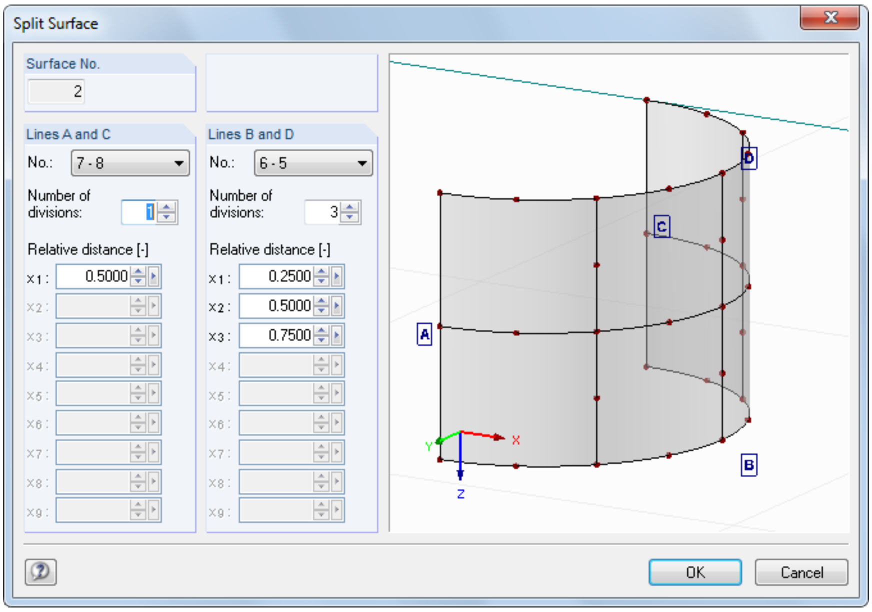


Figure 11.104: Dialog box *Split Surface*

11.4.17 Apply Tangent to Circles

You can easily create a tangent on an arc or a circle by using the object snap (see chapter 11.3.3, page 439). Another special function allows you to find tangents even for two circles or circular arcs. To open the corresponding dialog box,

select **Tangent to Two Circles/Arcs** on the **Tools** menu.

The following dialog box appears:

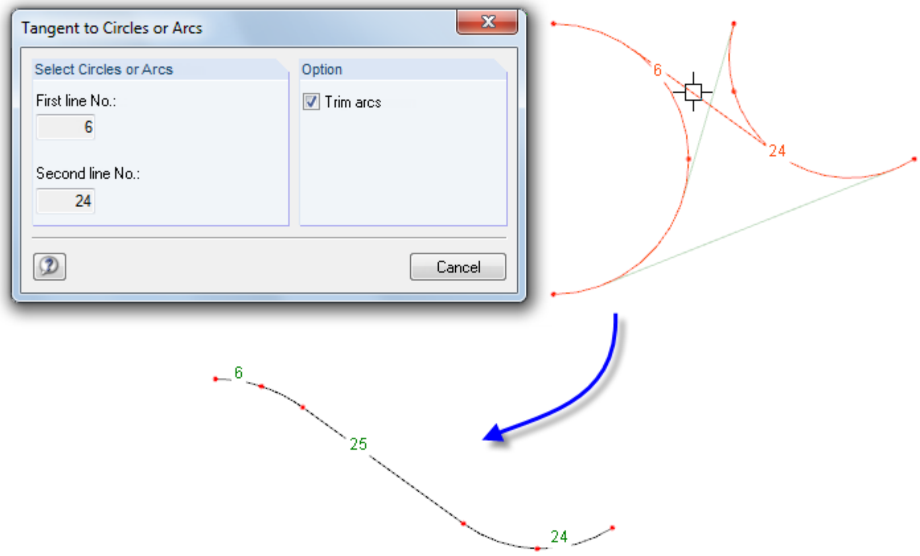


Figure 11.105: Dialog box *Tangent to Circles or Arcs* (above) with result (below)

First, click both circle or arc lines one after the other in the work window. RFEM draws the possible tangents as gray lines. Now, click the relevant line. RFEM divides the circular or arc line by nodes and creates the tangent as a new line.

By ticking the check box for *Trim arcs* you can remove overlapping line sections resulting from the division (see figure above).

11.4.18 Change Numbering

A regular, structured numbering proves to be useful for modeling as well as evaluations. However, graphical input and subsequent modifications may rearrange the numbering.

There are three options for adjusting the order of the numbering subsequently. To access the corresponding functions,

select **Renumber** on the **Tools** menu.

Loads are no problem when changing the numbering because the assigned loading will be transferred automatically to the new numbers of the objects.

Singly

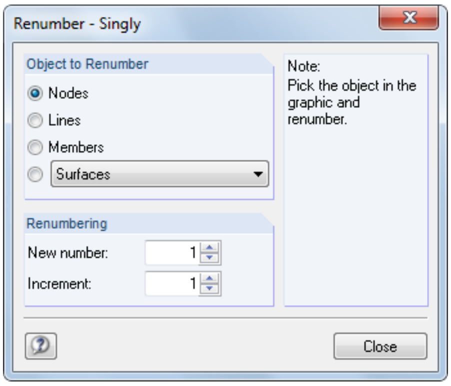


Figure 11.106: Dialog box *Renumber - Singly*

In the dialog section *Object to Renumber*, you decide whether nodes, lines, members or other model objects selected from the list will be renumbered. Specify the start number of the new numbering as well as the increment in the dialog section *Renumbering*.

After closing the dialog box with the [Close] button, you can select the relevant objects one after the other in the work window. Please note that RFEM can allocate only free numbers that are not yet assigned.

Automatically

First, select the nodes, lines and members (see chapter 11.2.1, page 430) whose numbering you want to adjust. Then, open the following dialog box.

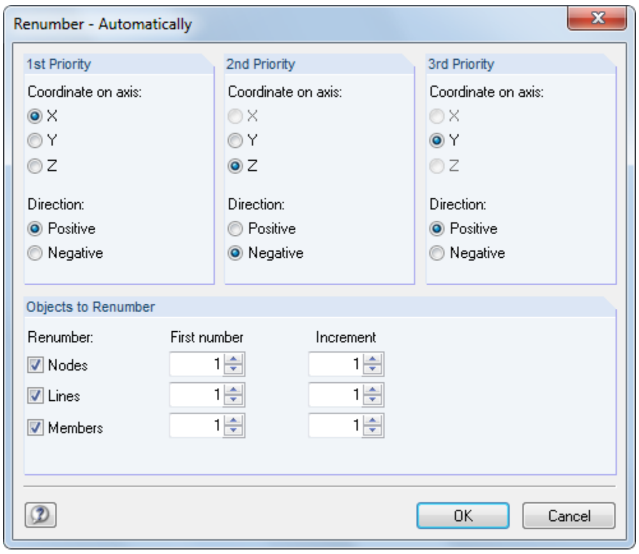


Figure 11.107: Dialog box *Renumber - Automatically* for nodes, lines and members

Specify the *Priority* of the global directions X, Y and Z for the new numbering. In addition, you have to decide if the ascending numbering will be applied in *Direction* of the respective positive or negative axis.

In the example above, the nodes (as well as lines and members) with the lowest X-coordinates receive new numbers first. The nodes are processed in the positive direction X. If two nodes have identical X-coordinates, the second priority decides which node will receive the lower number: This will be the node with the smaller Y-coordinate. The third priority will be decisive in case the Y-coordinates are identical as well.

The dialog section *Objects to Renumber* controls which nodes, lines and members will be renumbered and which start numbers and increments will be used for the renumbering. Already allocated numbers must not be assigned again. However, RFEM allows the use of numbers that have been allocated before changing the numbers but will become vacant during the renumbering.

Shift

First, select the objects whose numbering you want to adjust. Then, open the following dialog box by pointing to *Renumber* on the *Tools* menu.

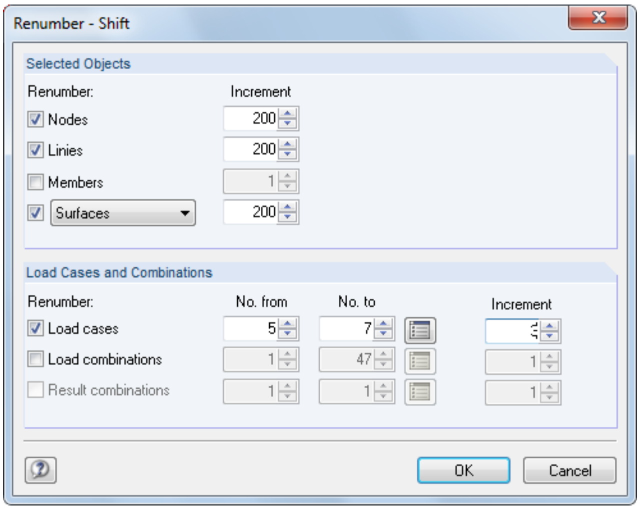
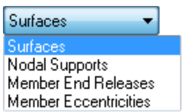


Figure 11.108: Dialog box *Renumber - Shift*



In the dialog section *Selected Objects*, define the objects that you want to renumber: Apart from nodes, lines and members, you can select other objects of the model in a list (see figure to the left). In the *Increment* column to the right, you can specify a value by which the numbers of the selected objects will be upgraded. Use negative increments to degrade the numbering. Make sure that no number will be smaller than 1.

In the dialog section *Load Cases and Combinations*, you can adjust the numbering of load cases, load and result combinations. Specify their numbers in the form of a list entered in the columns *No. from* and *No. to*. The *Increment* column to the right controls the value by which the numbers of the load objects are respectively upgraded.

After clicking [OK] the numbers will be shifted. Please note that only free, not yet assigned numbers can be allocated to the various model and load objects.

11.5 Table Functions

11.5.1 Editing Functions

The editing functions are tools making the data input in tables easier (see chapter 3.4.4, page 26). In contrast to the selection functions described in the following chapter 11.5.2, it is not necessary to select cells previously. The editing functions only affect the cell in which the pointer is placed.

To turn the tables on and off,

select **Display** on the **Table** menu

or use the toolbar button shown on the left.

Access to editing functions

To enable the editing functions for the table, place the pointer into a table cell. To access the editing functions,

point to **Edit** on the **Table** menu.

Some editing functions are available in the toolbar of the table.

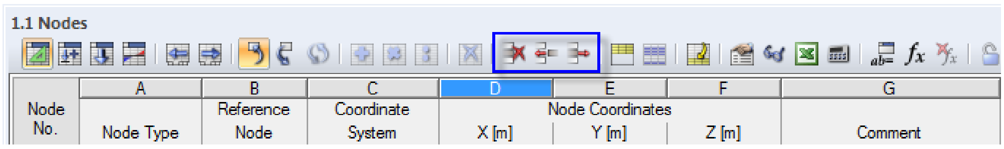


Figure 11.109: Buttons for several editing functions in the table toolbar

Alternatively, use the context menu in the table to access the functions.

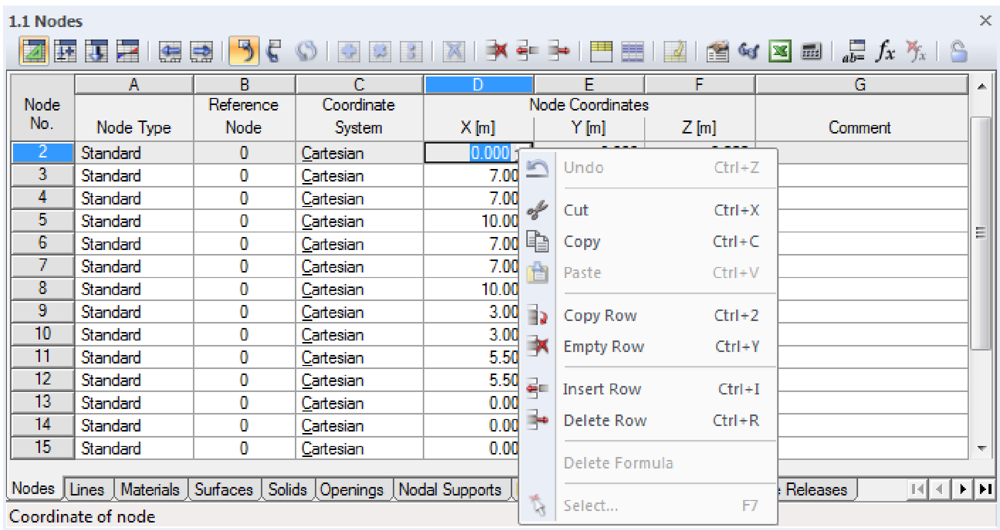


Figure 11.110: Editing functions in the table context menu

Functions and commands

Function	Effect
Cut [Ctrl+X]	Removes content of cell and saves it in clipboard
Copy [Ctrl+C]	Copies cell content to clipboard
Paste [Ctrl+V]	Inserts contents of clipboard into cell If the clipboard contents are bigger than the cell, the cells of the subsequent table columns and rows will be overwritten. A warning is displayed before.
Copy Row [Ctrl+2]	Overwrites subsequent row with contents of current row
Empty row [Ctrl+Y] 	Deletes contents of row without deleting the row itself
Insert row [Ctrl+I] 	Inserts a new, empty row. The subsequent rows will be moved downwards.
Delete row [Ctrl+R] 	Deletes the current row. The subsequent rows will be moved upwards.
Find [Ctrl+F]	Searches a number or string within the table
Replace [Ctrl+H]	Searches a number or string within the table and replaces it by another entry
Empty table	Deletes contents of current table completely without a warning
Empty all tables	Deletes contents of all tables
Select [F7]	Opens a list for selection in cell
Update graphics 	Transfers modifications entered in table to graphic
Edit in dialog box	Opens a dialog box where data of current row can be entered

Table 11.8: Editing functions

11.5.2 Selection Functions

The selection functions are tools making the data input in tables easier. In contrast to the editing functions described in chapter 11.5.1, you have to mark several, connected cells as a *Selection* first.

Coordinate System	Node Coordinates X [m]	Y [m]	Z [m]
Cartesian	0.000	6.000	0.000
Cartesian	7.000	6.000	0.000
Cartesian	7.000	0.000	0.000
Cartesian	10.000	3.000	0.000
Cartesian	7.000	6.000	4.000
Cartesian	7.000	0.000	4.000
Cartesian	10.000	3.000	4.000
Cartesian	3.000	1.000	0.000
Cartesian	3.000	2.000	0.000

Figure 11.111: Selection

It is not important whether the cells are empty or filled with content. A selection function modifies the contents of selected cells altogether.

Access to selection functions

First, mark a selection as a block of contiguous cells in the table: Move the mouse across several cells while holding the left mouse button down. A click into a table header (A, B, C ...) selects the whole table column. To select the entire table row, click the row number on the left.

To access the selection functions,

select **Selection** on the **Table** menu.

Some selection functions are available in the toolbar of the table.

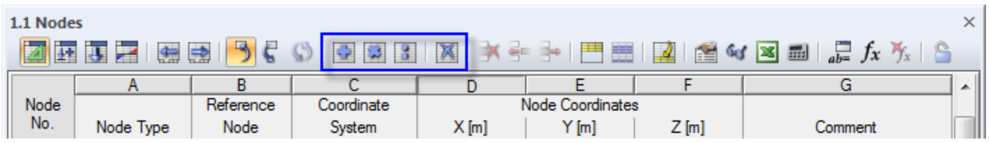


Figure 11.112: Buttons for several selection functions in the table toolbar

You can also access the functions by means of the context menu in the table.

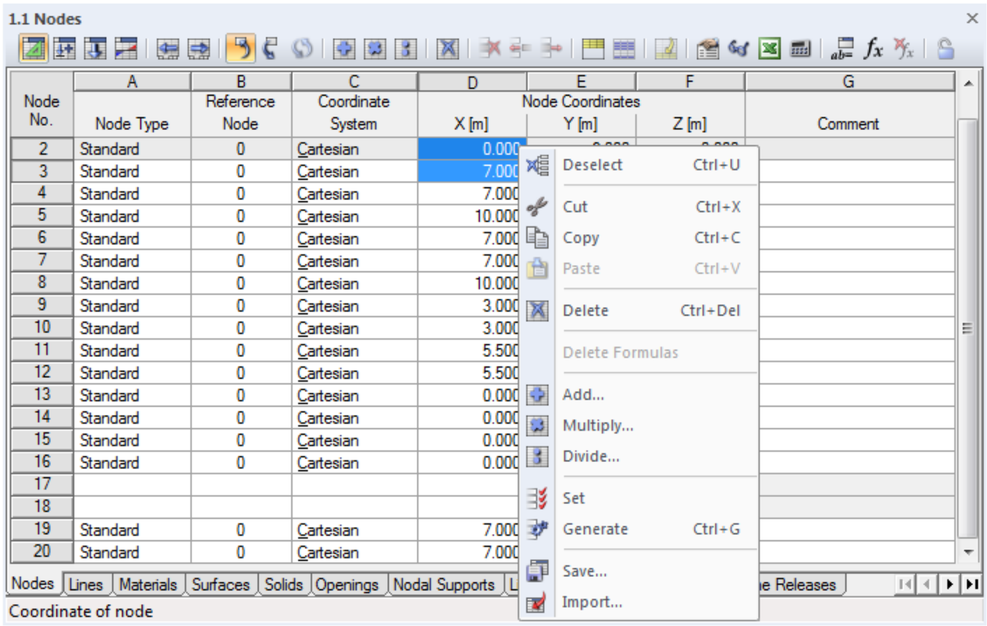


Figure 11.113: Selection functions in the table context menu

Functions and commands

Function	Effect
Deselect [Ctrl+D]	Cancels selection of row or column
Cut [Ctrl+X]	Removes content of selected cells and saves it in clipboard
Copy [Ctrl+C]	Copies content of selection to clipboard
Paste [Ctrl+V]	Inserts content of clipboard into table The command is only available when the clipboard contains appropriate data (for example from Excel).
Delete [Ctrl+Del]	Deletes all contents of selected cells
Add	Adds value to or subtracts value from cells with numerical values
Multiply	Multiplies cells with numerical values by a factor
Divide	Divides cells with numerical values by a divisor
Set	Assigns the value of the topmost selected cell to all cells of the entire selection.
Generate [Ctrl+G]	Used for cells with numerical values to generate entries between first and last selected cell by interpolation of both reference values (see example below)
Save	Saves selection as file
Import	Imports selection saved as file

Table 11.9: Selection functions

Example: Generating cell values

Use this function to fill empty cells quickly. The intermediate values are determined by a linear interpolation from the start value of the top cell (in example *6.000*) and the end value of the bottom cell (in example *30.000*).

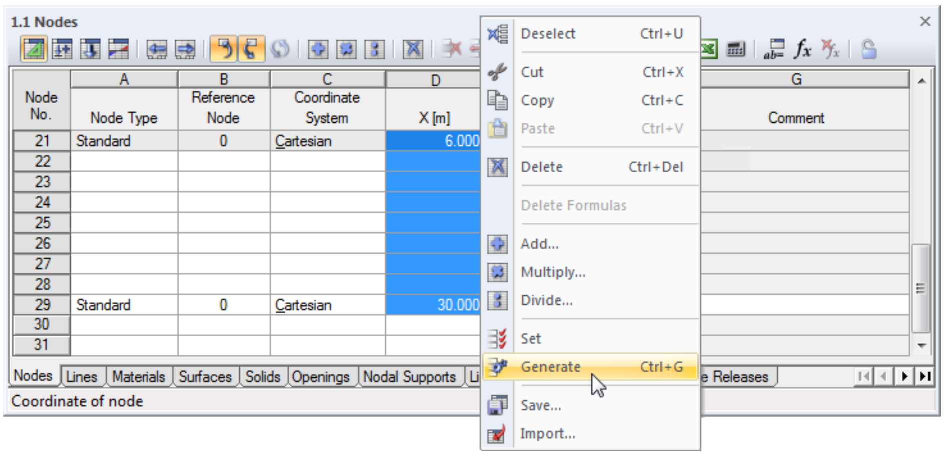
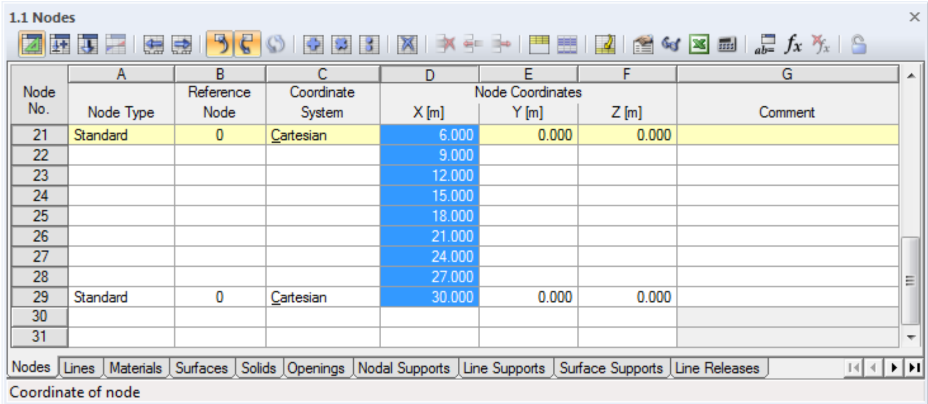


Figure 11.114: Context menu of selection

When you apply the *Generate* option, the intermediate cells are filled with interpolated values.



1.1 Nodes

Node No.	A Node Type	B Reference Node	C Coordinate System	D Node Coordinates X [m]	E Y [m]	F Z [m]	G Comment
21	Standard	0	Cartesian	6.000	0.000	0.000	
22				9.000			
23				12.000			
24				15.000			
25				18.000			
26				21.000			
27				24.000			
28				27.000			
29	Standard	0	Cartesian	30.000	0.000	0.000	
30							
31							

Figure 11.115: Result

11.5.3 View Functions

The table display can be adjusted by different view functions improving the data overview in the table.

Access to view functions

To access the view functions,

select **View** on the **Table** menu or

select **Optimize Load Data** on the **Table** menu.

Some of the view functions can be accessed in the toolbar of the table.

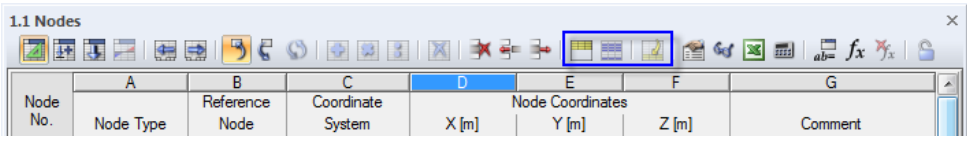


Figure 11.116: Buttons for several view functions in the table toolbar

Functions

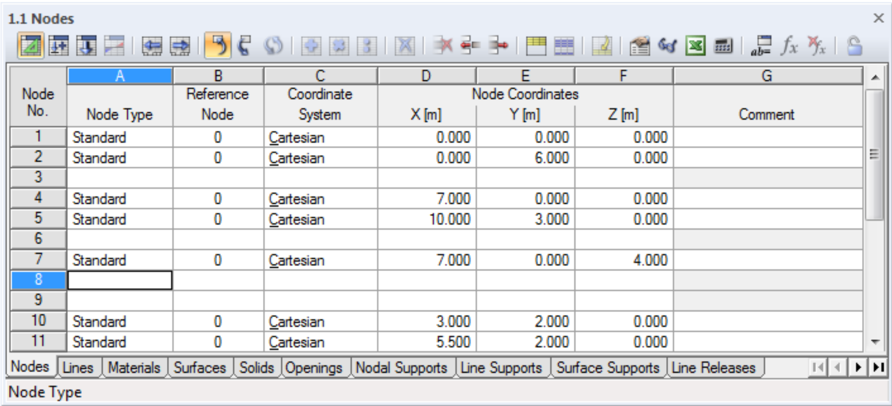
Function	Effect
Only filled rows 	Hides all empty table rows
Marked rows only 	Shows only selected rows
Selected objects only 	Shows only objects selected in graphic
Select related objects 	In addition to loads, associated model objects (nodes, surfaces, members etc.) are selected in the graphic. Only available in loads tables *3*.
Compress data 	Summarizes objects with same loads in one single table row in loads tables
Decompress data 	Lists loads for each object individually

Result filter	Table output can be restricted to particular result types (see chapter 11.5.5, page 487).
Info about cross-section	Shows characteristic values of current cross-section
Result diagrams	Displays results of selected member graphically in new window (see chapter 9.5, page 356)
Colored relation scales	Switches the display of the red and blue bars in the table on and off.
Title bar	Switches title bar on and off
Toolbar	Switches toolbar on and off
Column bar	Switches column headings (A, B, C, ...) on and off
Status bar	Switches status bar of table on and off
Highlight table row	The table row where the pointer is placed is highlighted with colors or won't be marked.

Table 11.10: View functions

Example: Only filled rows

A table contains empty rows disturbing the clear table overview.



1.1 Nodes

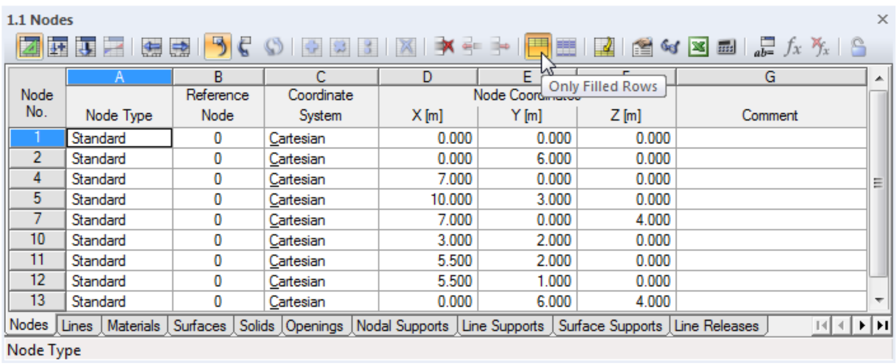
Node No.	A Node Type	B Reference Node	C Coordinate System	D Node Coordinates X [m]	E Y [m]	F Z [m]	G Comment
1	Standard	0	Cartesian	0.000	0.000	0.000	
2	Standard	0	Cartesian	0.000	6.000	0.000	
3							
4	Standard	0	Cartesian	7.000	0.000	0.000	
5	Standard	0	Cartesian	10.000	3.000	0.000	
6							
7	Standard	0	Cartesian	7.000	0.000	4.000	
8							
9							
10	Standard	0	Cartesian	3.000	2.000	0.000	
11	Standard	0	Cartesian	5.500	2.000	0.000	

Nodes | Lines | Materials | Surfaces | Solids | Openings | Nodal Supports | Line Supports | Surface Supports | Line Releases

Node Type

Figure 11.117: Table with empty rows

Use the button *Only Filled Rows* in the table toolbar to hide all empty table rows.



1.1 Nodes

Node No.	A Node Type	B Reference Node	C Coordinate System	D Node Coordinates X [m]	E Y [m]	F Z [m]	G Comment
1	Standard	0	Cartesian	0.000	0.000	0.000	
2	Standard	0	Cartesian	0.000	6.000	0.000	
4	Standard	0	Cartesian	7.000	0.000	0.000	
5	Standard	0	Cartesian	10.000	3.000	0.000	
7	Standard	0	Cartesian	7.000	0.000	4.000	
10	Standard	0	Cartesian	3.000	2.000	0.000	
11	Standard	0	Cartesian	5.500	2.000	0.000	
12	Standard	0	Cartesian	5.500	1.000	0.000	
13	Standard	0	Cartesian	0.000	6.000	4.000	

Figure 11.118: Table without empty rows

11.5.4 Table Settings

The font and color settings used in the tables can be adjusted individually. Moreover, it is possible to synchronize the selection in the graphic with the one in the table.

Access to table settings

To select a particular table setting,

select **Settings** on the **Table** menu.

To activate and deactivate the synchronization of the selection, you can also use the table toolbar buttons.

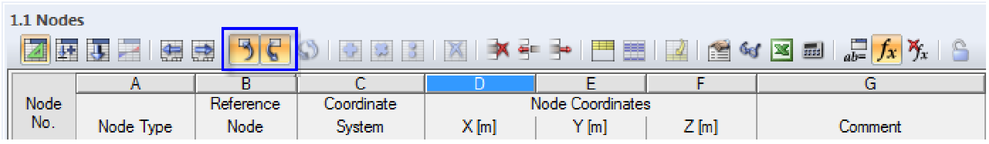


Figure 11.119: Buttons *Selection Synchronization*

Functions

Function	Effect
Colors	Opens the dialog box *Colors* (Figure 11.120). Colors of individual table objects can be adjusted separately.
Fonts	Opens the dialog box *Font* (Figure 11.120). Font, style and font size can be modified globally for all table objects.
Select current object in graphic 	Function is set active by default: Object of table row where pointer is placed is selected also in work window.
Show selected object in tables 	Function is set active by default: Objects selected in work window are highlighted with colors also in table.

Table 11.11: Table settings

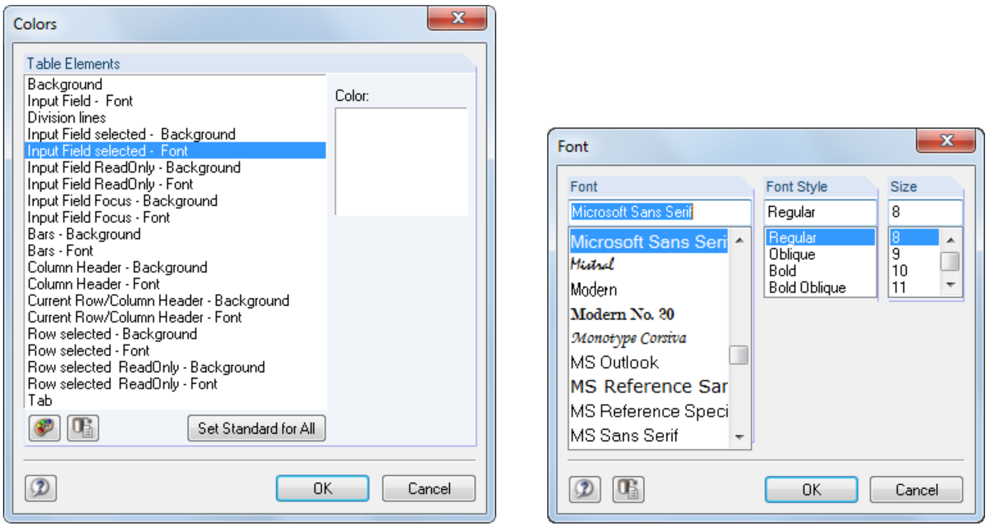


Figure 11.120: Dialog boxes *Colors* and *Font*

11.5.5 Filter Functions

Various filter functions allow you to evaluate specifically internal and contact forces as well as deformations in the member results tables. In addition, filter options are available for nodal and line support forces of result combinations (see chapter 8.1, page 285 and chapter 8.3, page 290).

Access to filter functions

To access the filter functions,

select **View** on the **Table** menu and click **Result Filter**

or use the button in the table toolbar shown on the left.

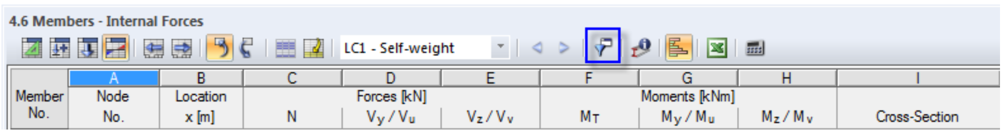


Figure 11.121: Button *Result Filter*

The following dialog box appears:

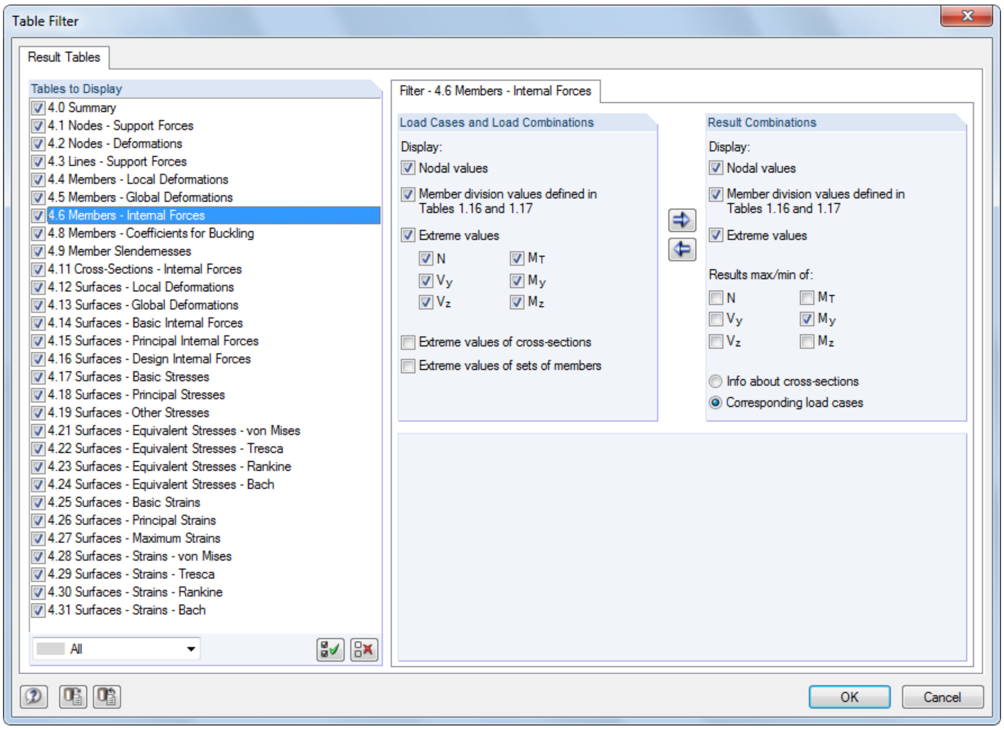


Figure 11.122: Dialog box *Table Filter*

Select the relevant results table in the dialog section *Tables to Display*. Then, use the dialog tab on the right to determine which values will be shown numerically.

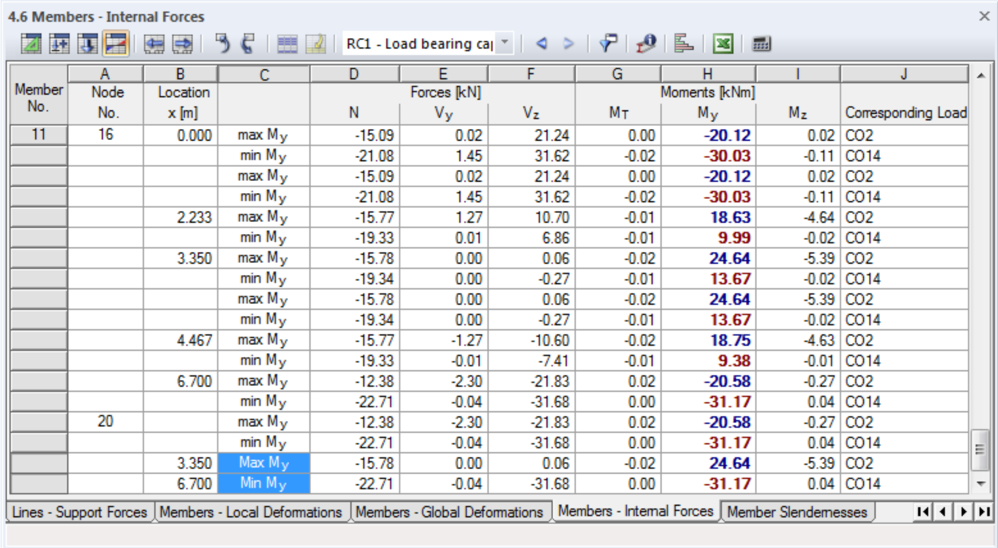
When the table for internal forces of members is set, you can define for *Load Cases and Load Combinations* and *Result Combinations* separately whether *Nodal values* (member start and member end), *Member division values* (intermediate points of user-defined member division, see chapter 4.16) and *Extreme values* of members are shown in the table. You have to tick at least one of the six check boxes for internal forces. The selected internal forces are shown on the locations of the result values that are activated by a check mark above.

Two result values appear on each location for result combinations – the minimum and the maximum internal forces with the corresponding internal forces.

Use the buttons shown on the left to transfer the filter criteria from one dialog section to the other.

Example

A member division with two intermediate points has been defined for member 11 that has a length of 6.70 m. The filter settings for result combinations shown in Figure 11.122 result in the following results table *4.6 Members - Internal Forces*.



4.6 Members - Internal Forces

RC1 - Load bearing ca|

Member No.	A Node No.	B Location x [m]	C	D Forces [kN] N	E V_y	F V_z	G Moments [kNm] M_T	H M_y	I M_z	J Corresponding Load
11	16	0.000	max M_y	-15.09	0.02	21.24	0.00	**-20.12**	0.02	CO2
			min M_y	-21.08	1.45	31.62	-0.02	**-30.03**	-0.11	CO14
			max M_y	-15.09	0.02	21.24	0.00	**-20.12**	0.02	CO2
			min M_y	-21.08	1.45	31.62	-0.02	**-30.03**	-0.11	CO14
		2.233	max M_y	-15.77	1.27	10.70	-0.01	**18.63**	-4.64	CO2
			min M_y	-19.33	0.01	6.86	-0.01	**9.99**	-0.02	CO14
		3.350	max M_y	-15.78	0.00	0.06	-0.02	**24.64**	-5.39	CO2
			min M_y	-19.34	0.00	-0.27	-0.01	**13.67**	-0.02	CO14
			max M_y	-15.78	0.00	0.06	-0.02	**24.64**	-5.39	CO2
			min M_y	-19.34	0.00	-0.27	-0.01	**13.67**	-0.02	CO14
		4.467	max M_y	-15.77	-1.27	-10.60	-0.02	**18.75**	-4.63	CO2
			min M_y	-19.33	-0.01	-7.41	-0.01	**9.38**	-0.01	CO14
		6.700	max M_y	-12.38	-2.30	-21.83	0.02	**-20.58**	-0.27	CO2
			min M_y	-22.71	-0.04	-31.68	0.00	**-31.17**	0.04	CO14
	20		max M_y	-12.38	-2.30	-21.83	0.02	**-20.58**	-0.27	CO2
			min M_y	-22.71	-0.04	-31.68	0.00	**-31.17**	0.04	CO14
		3.350	Max M_y	-15.78	0.00	0.06	-0.02	**24.64**	-5.39	CO2
		6.700	Min M_y	-22.71	-0.04	-31.68	0.00	**-31.17**	0.04	CO14

Lines - Support Forces | Members - Local Deformations | Members - Global Deformations | Members - Internal Forces | Member Slendernesses

Figure 11.123: Results filtered by nodal values, division points and extreme values M_y

Table column H shows the maximum and minimum bending moments $\mathbf{M_y}$ on the nodes and division points as well as locations of the absolute extreme values in bold. The latter appear with a capitalized initial letter as *Max* M_y and *Min* M_y at the end of the list (see marked cells in figure above). The values in the remaining columns represent the corresponding internal forces of the respective maximum and minimum values.

11.5.6 Import and Export of Tables

A table from MS Excel or Open Office.org Calc can be imported directly into the current RFEM input table. The programs involved must be opened. It is also possible to export the current RFEM table all or part to Excel or Open Office.org Calc.

Access to import and export function

To apply the import or export function, click the button [Export/Import Table] in the table toolbar.

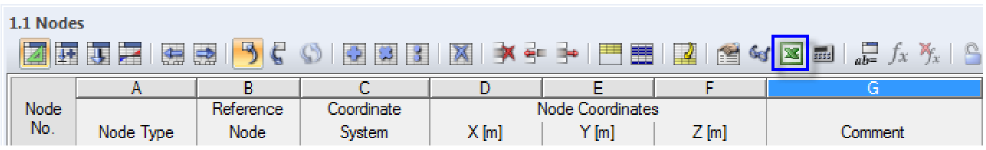


Figure 11.124: Button *Export/Import Table* in the table toolbar

Use this button to open the *Export table* and *Import table* dialog box.

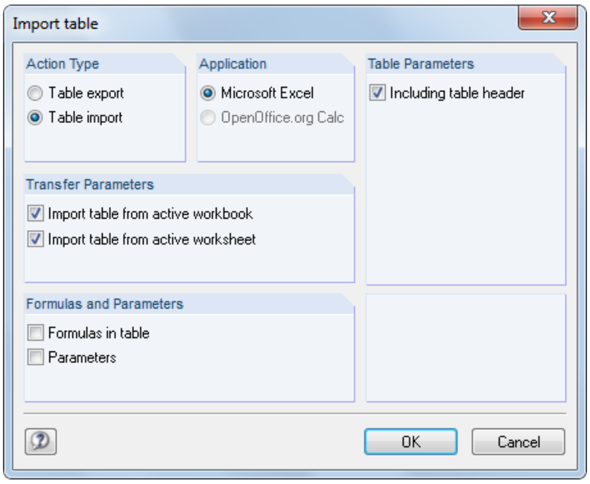


Figure 11.125: Dialog box *Import table*

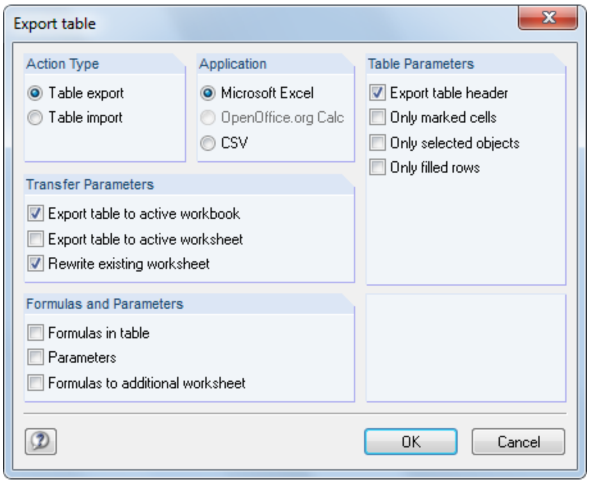


Figure 11.126: Dialog box *Export table*

Import table

The workbook from MS Excel or OpenOffice must have been opened before the import starts. If headers exists in the worksheets, tick the check box for *Including table header*. Then, the headers will be ignored during the import. Only the lists will be imported into the RFEM tables.

In the dialog section *Application*, you can select between the spreadsheets of Microsoft Excel and OpenOffice.org Calc.

The dialog section *Transfer Parameters* specifies whether the active workbook or only the active worksheet will be imported. When you import a complete workbook, the order and structure of worksheets must be completely consistent with the RFEM tables.

In the dialog section *Formulas and Parameters*, you can decide if formulas stored in Excel or OpenOffice will be imported as well when exchanging data.

Click [OK] to start the import.

If you want to import only particular parts of the worksheet, the copy function is recommended: Select the relevant area in the Excel table and copy it to the clipboard with [Ctrl]+[C]. Then, place the pointer into the corresponding cell of the RFEM table and insert the contents of the clipboard with [Ctrl]+[V].

Export table

To export RFEM tables, MS Excel or Open Office.org Calc do not need to run in the background.

In the dialog section *Application*, you can select between the spreadsheets Microsoft Excel and OpenOffice.org Calc. In addition, it is possible to create a file in the general spreadsheet *CSV* (see chapter 4.13, page 127).

In the dialog section *Table Parameters*, specify if the headers will be exported, too. When the check box for *Export table header* is ticked, the result in Excel looks like below:

	A	B	C	D	E	F	G	H
1	Node		Reference	Coordinate		Node Coordinates		
2	No.	Node Type	Node	System	X [m]	Y [m]	Z [m]	Comment
3	1	Standard	0	Cartesian	0.000	0.000	0.000	
4	2	Standard	0	Cartesian	0.000	6.000	0.000	
5	3	Standard	0	Cartesian	7.000	6.000	0.000	
6	4	Standard	0	Cartesian	7.000	0.000	0.000	
7	5	Standard	0	Cartesian	10.000	3.000	0.000	
8	6	Standard	0	Cartesian	7.000	6.000	4.000	
9	7	Standard	0	Cartesian	7.000	0.000	4.000	
10	8	Standard	0	Cartesian	10.000	3.000	4.000	
11	9	Standard	0	Cartesian	3.000	1.000	0.000	
12	10	Standard	0	Cartesian	3.000	2.000	0.000	
13	11	Standard	0	Cartesian	5.500	2.000	0.000	
14	12	Standard	0	Cartesian	5.500	1.000	0.000	
15	13	Standard	0	Cartesian	0.000	6.000	4.000	
16	14	Standard	0	Cartesian	0.000	0.000	4.000	
17	15	Standard	0	Cartesian	0.000	0.000	-3.000	
18	16	Standard	0	Cartesian	0.000	6.000	-3.843	

Figure 11.127: Excel table with exported headers

When you clear the check box, only the table contents will be transferred to Excel.

With the option *Only marked cells*, you can export selected table contents (see chapter 11.5.2, page 482).

Use the check box for *Only selected objects* to export data or results of selected row numbers. The selection is made easier by the synchronization of the selection between graphic and table (see chapter 11.5.4, page 486).

The option *Only filled rows* controls the way how empty rows are treated for the export.

In the dialog section *Transfer Parameters*, you can define the target tables where data will be written. When the first check box is clear, RFEM will create a new workbook. With the option *Export table to active worksheet* it is possible to use the current worksheet of the spreadsheet. If the check box for *Rewrite existing worksheet* is ticked, RFEM will search in the workbook for a table with the same name as in RFEM and will overwrite it then.

Using the check boxes in the dialog section *Formulas and Parameters*, you can decide if and how formulas saved in RFEM will be exported.

To start the export of the current RFEM table, click [OK].

To transfer several tables all at once to Excel or OpenOffice.org Calc, it is recommended to select **Export** on the **File** menu (see chapter 12.5.2, page 569). Then, you can select the relevant tables in a dialog box.

11.6 Parameterized Input

11.6.1 Concept

The parameterized input for model and load data makes use of variables (for example length, width, traffic load etc.) which are called "parameters" and stored in a **parameter list**.

The parameters can be used in formulas to determine a numerical value. The formulas are edited in the **Formula Editor**. If a parameter is modified in the parameters list, the results of all formulas using this parameter will be adjusted.

The parameterized input is useful for projects where many changes are to be expected. The stored formulas are easy to follow and bring more clarity to complex models. The parameter-controlled input is also quite suitable when editing recurring models that are similar in design: Simply open a template file and adjust the parameters.

11.6.2 Parameter List

The parameter list manages all parameters required for modeling.

Access to parameter list

To access the parameter list, click the [Edit Parameters] button:

- in the toolbar of an input table

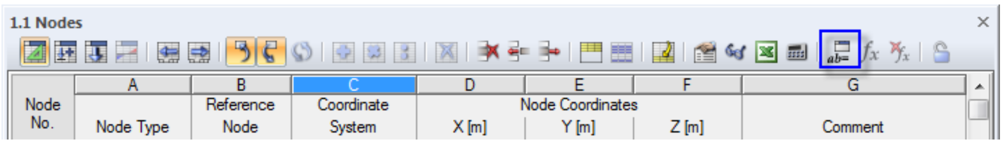


Figure 11.128: Button *Edit Parameters* in the table toolbar

- in the Formula Editor.

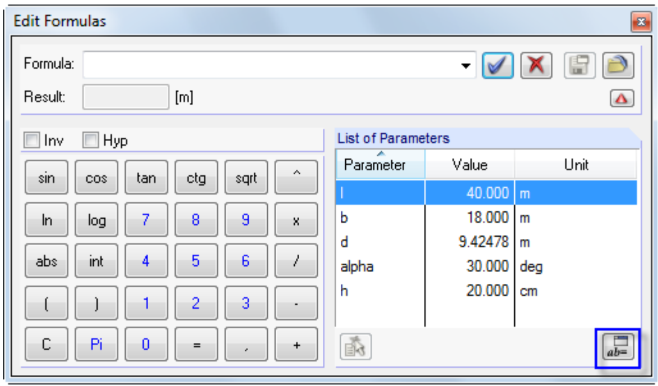


Figure 11.129: Button *Edit Parameters* in the Formula Editor

Description

The dialog box *Edit Parameters* appears.

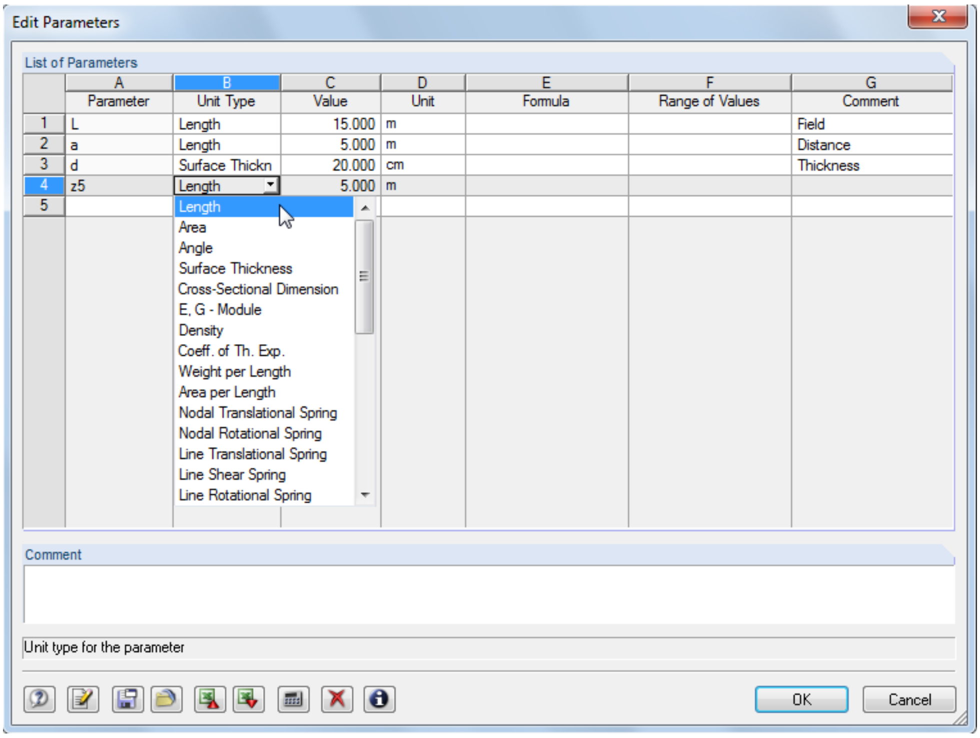


Figure 11.130: Dialog box *Edit Parameters*

Each table row manages a *Parameter*. In column ***A***, enter a name that consists of ASCII signs. The name must not contain any spaces. The description is used to refer to the parameter in the formulas. Each parameter name can be assigned only once.

In table column ***B***, define the *Unit Type* to determine if the parameter represents a length, load, density etc. The unit types are predefined. To access the selection list available in the column, use the context button [▼] or the keyboard key [F7].

In column ***C***, define the numerical *Value* of the parameter.

Specify the *Unit* in table column ***D***. To access the selection list of units available in the column, use the context button [▼] or the keyboard key [F7].

In column ***E***, you can enter a *Formula* to determine the value of the parameter for table column C. In addition to common mathematical operations, **IF-THEN** statements and **max/min** functions are available. With the **$**-reference you can refer to a particular table (for example **$1.1(A1)** uses the value of cell *A1* from table *1.1*).

Examples

if(A<B;10;B)	If parameter A is smaller than parameter B, the value 10 is applied. Otherwise, parameter B will be used.
max(A;B)	The larger value of both parameters A and B will be applied.
min(max(A;B);C)	The larger value of parameters A and B is determined which will then be compared to the value of parameter C. The smallest value will finally be applied.

Use the button [...] in table column E to access a *List of operators and functions*.

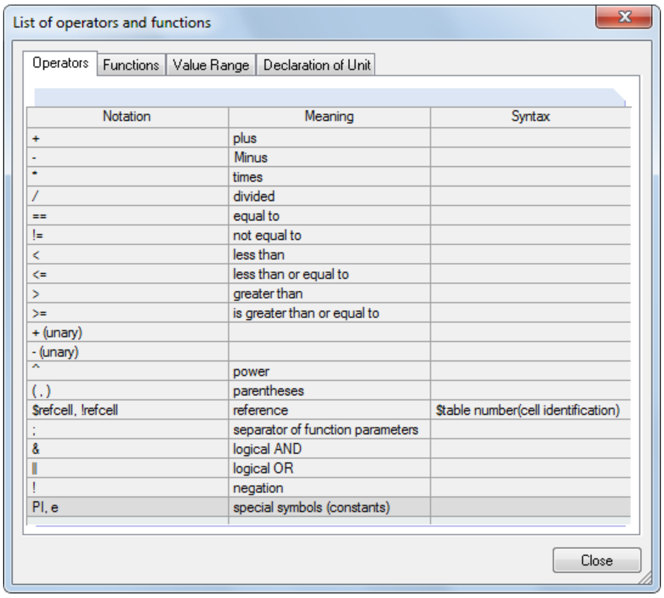


Figure 11.131: Dialog box *List of operators and functions*

In table column ***F***, you can define a *Range of Values* to control the values of column C.

Column ***G*** is reserved for entering any *Comment*.

Input functions

The parameters can be entered cell by cell.

Several tools for efficient input are available in the context menu that you open with a click of the right mouse button. The editing functions (empty row or insert row, replace etc.) are described in chapter 11.5.1 on page 480).

When several cells are marked as a selection, the following context menu appears.

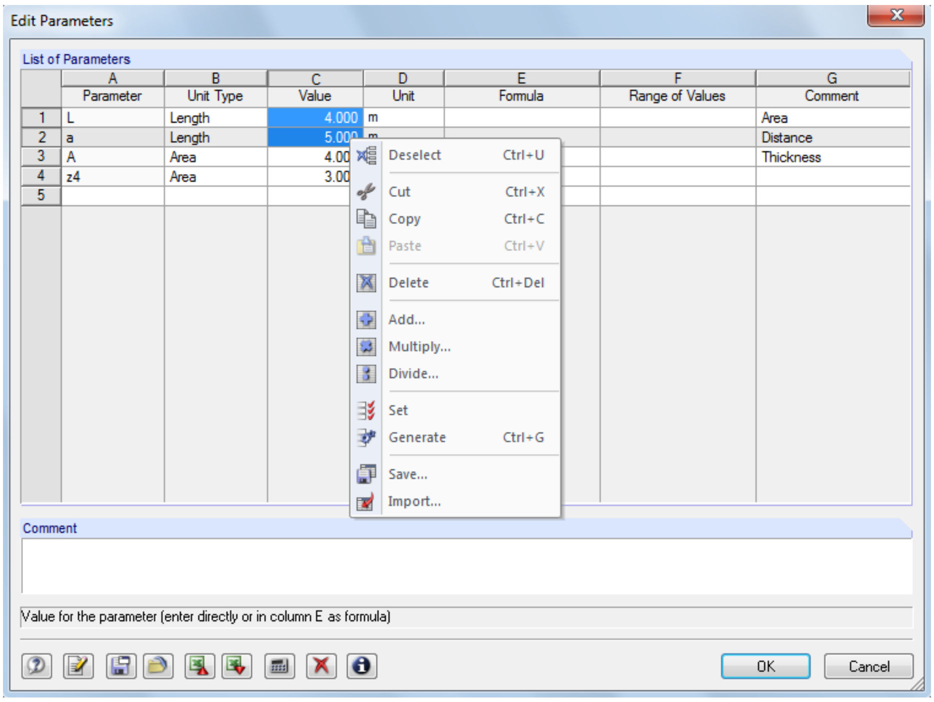


Figure 11.132: Context menu of a selection in parameter list

Find a description of the menu functions in chapters 11.5.1 and 11.5.2, page 480.

Buttons

In addition to the default buttons, the following functions are available in the parameter list.

Button	Description
	Saves the parameter list in a file.
	Loads a saved parameter list
	Export of parameter list to MS Excel
	Imports data from opened Excel table
	Opens calculator and imports its result
	Deletes the entire contents of the List of Parameters.
	Shows cross-section details of cross-sections used in model

Table 11.12: Dialog box *Edit Parameters*: Buttons

11.6.3 Formula Editor

The Formula Editor manages the equations of the parameterized input.

Access to Formula Editor

To open the Formula Editor,

- use the button in the table toolbar shown on the left

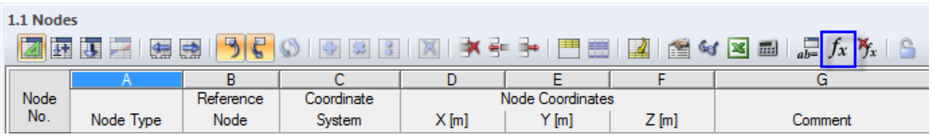


Figure 11.133: Button *Edit Formulas* in the table toolbar

- click the yellow or red corner of the table cell (a red corner indicates an error in the formula) or

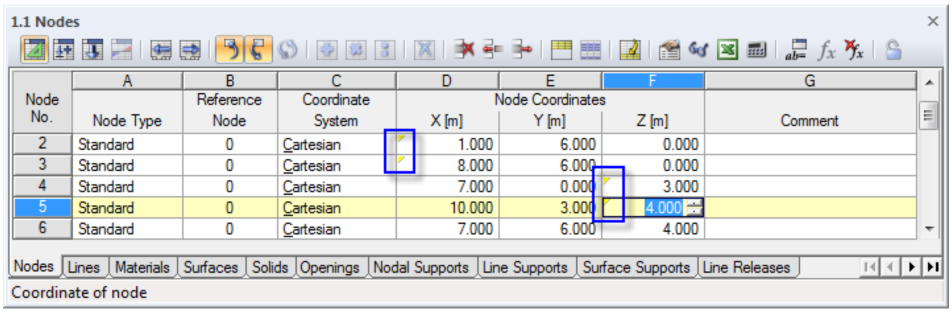


Figure 11.134: Marked cell corners in table 1.1 *Nodes*

- use the function buttons next to the input fields in dialog boxes (see Figure 11.139).

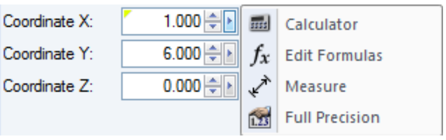


Figure 11.135: Function buttons with context menu in dialog box *Edit Node*

It is also possible to import formulas saved in Excel and to export formulas from RFEM to Excel. For more information about the data exchange with Excel, see chapter 12.5.2 on page 569.

Description

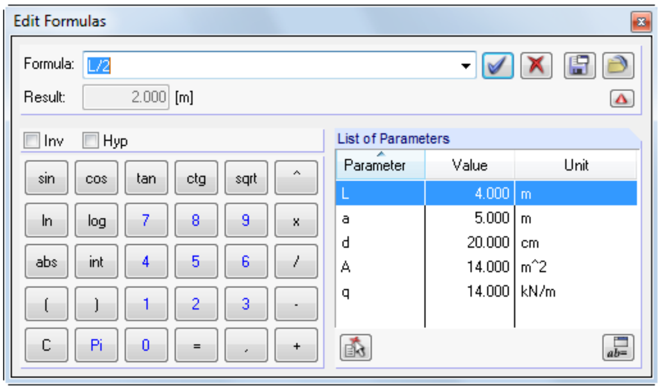


Figure 11.136: Dialog box *Edit Formulas*

In the input field *Formula*, any formula can be entered manually. When you use the calculator, its results will be transferred automatically.

The formula may consist of constant numerical values, parameters or functions. The result of the equation appears in the field below. Use the button [▼] at the end of the *Formula* line to select an entry from the list of already entered formulas.

Click the button [✓] to apply the formula to the table cell or input field of the dialog box. Delete the formula line with the [×] button. In case of misentries, formulas are displayed red in the *Formula* input field.

Contents of other cells can be used in formulas by means of references.

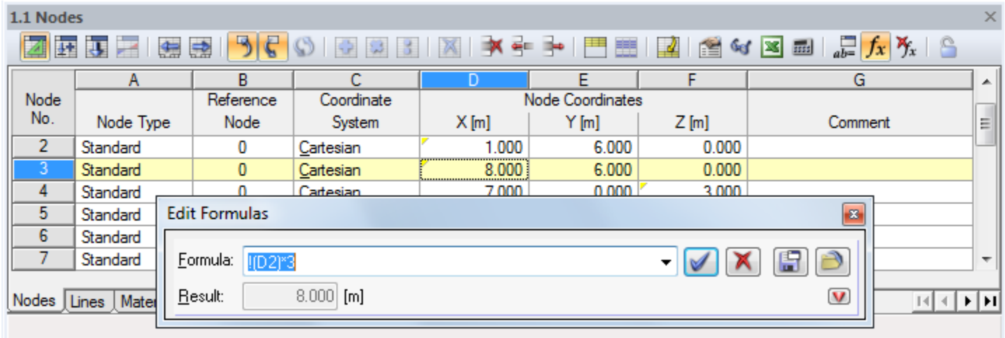


Figure 11.137: Formula Editor with a reference

A reference is introduced by an exclamation point (!). The reference cell is set in brackets. As shown in the figure above, the content of cell **D3** is three times the value of cell **D2**.

By means of a prefixed equal sign you can also enter formulas directly in the table cells (for example =*2.5*PI*). If values are used (for example =**22.1** + A*H), they are integrated in SI units with [m] or [N] into the formula.

The following functions are available in the calculator of the Formula Editor:

Function	Description
sin	Sine
cos	Cosine
tan	Tangent
ctg	Cotangent
sqrt	Square root
^	Power
ln	Natural logarithm
log	Logarithm to the base 10
abs	Absolute value
int	Integer, for example *int(5.638)* = 5
C	Clear formula line
Inv	Inverse, for example *inv sqrt(5)* means 5^2
Hyp	Hyperbolic function

Table 11.13: Calculator functions

The dialog section *List of Parameters* in the Formula Editor lists all parameters with the current values. To transfer a particular parameter to the *Formula* line, double-click the entry, or select the entry and use the button [Apply Selected Parameter] shown on the left.

Click the [Edit Parameters] button (see chapter 11.6.2, page 491) to open the parameter list where you can modify or complete the parameters.

Buttons

The buttons available in the Formula Editor have the following functions:

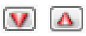
Button	Description
	Applies formula to table cell or dialog field
	Deletes formula input
	Saves contents of Formula Editor as a file
	Loads a saved file
	Displays or hides calculator and parameter list

Table 11.14: Dialog box *Edit Formulas*: Buttons

11.6.4 Formulae in Tables and Dialog Boxes

The equations stored in the Formula Editor can be used in both the cells of tables and the input fields of dialog boxes. As tables and dialog boxes are interactive, you can access the formulas in both input modes.

Formulas in tables

When cells are marked by a yellow or red flag (triangle) in the upper left corner, a formula has been linked (see Figure 11.134, page 494). Click the flag to open the Formula Editor.

To link a "normal" cell with a formula, place the pointer into the cell and open the Formula Editor by using the button shown on the left.

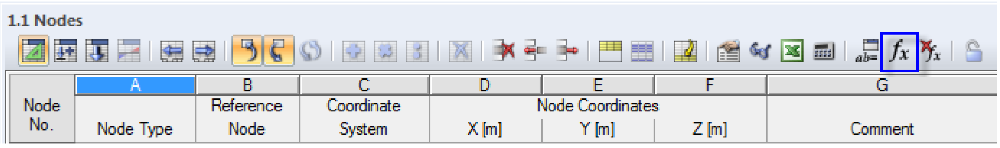


Figure 11.138: Button *Edit Formulas* in the table toolbar

A red flag means that there is an error in the definition of the formula. This flag corresponds to the red formula line in the Formula Editor. It is recommended to correct the formula.

Formulas in dialog boxes

The parameterized input has been developed primarily for the application in tables. However, it is also possible to use formulas in dialog boxes.

A function button to the right of the input fields in dialog boxes indicates that they can be linked with formulas.

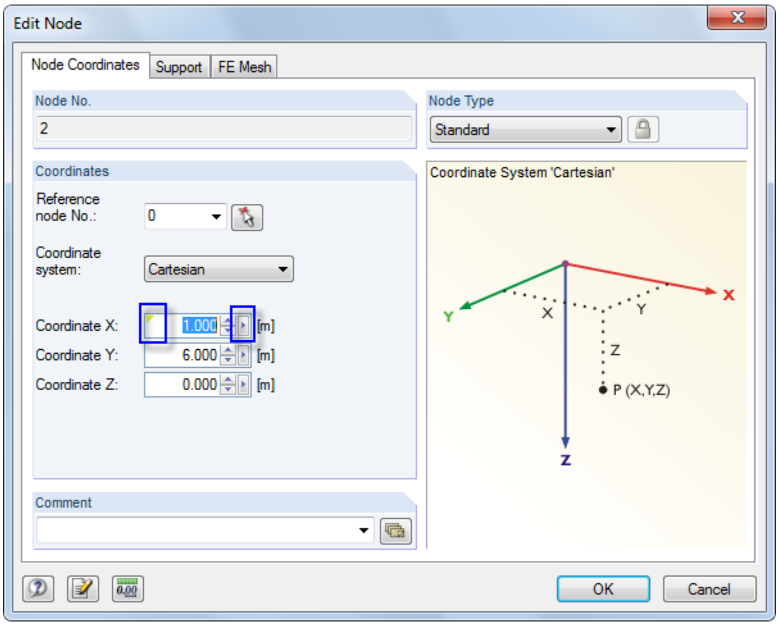


Figure 11.139: Dialog box with linked formula and function button

When the input field has already been linked with a formula, it is marked like a cell by a yellow flag (or red flag in case of incorrect input).

Click the function button to open the context menu shown in Figure 11.135 on page 494 where you can access the Formula Editor.

11.7 Model Generators

A variety of tools help you to create models or parts of structural systems. In addition to copy and extrude functions, RFEM provides special dialog boxes for generating member and surface models.

11.7.1 Copies and Extrusions

11.7.1.1 Parallel Offset of Lines and Members

It is easy to copy selected lines or members graphically: Move the objects to the desired place in the workspace by holding down the [Ctrl] key. The function follows the general standards for Windows applications.

If you want to create parallel lines or members, you can enter specific settings in a dialog box. To access the corresponding function,

select **Set Parallel Line** on the **Tools** menu or

select **Set Parallel Member** on the **Tools** menu

or use the context menu of the line or member (see Figure 11.149, page 505).

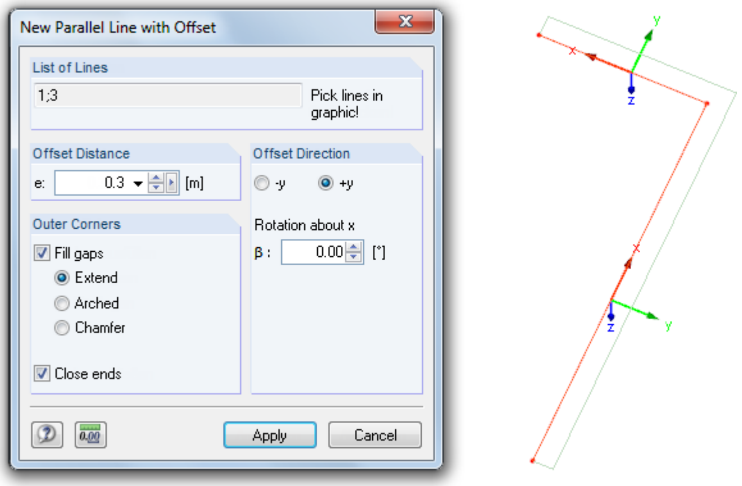


Figure 11.140: Dialog box *New Parallel Line with Offset*

The selected line appears in the *List of Lines*. If necessary, more lines can be added by clicking them in the work window. Please note that all lines of the list must lie in one plane.

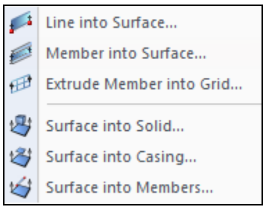
In the dialog section *Offset Distance*, you specify the copy's distance to the original.

When several lines are copied by parallel offset, you have several possibilities offered in the dialog section *Outer Corners* to adjust the copied lines or members. The figure above shows the copied lines (without axes) extended to the common point of intersection. Moreover, both ends are connected with the original lines by the ticked check box *Close ends*.

The settings in the dialog section *Offset Direction* define the side on which the lines will be copied. The directions *+y* and *-y* are directly displayed in the work window. They are especially used for this dialog box and do not depend on the currently set work plane. Thus, they do not necessarily reflect the line axes. The input field *Rotation about x* allows for copying objects out of the plane.

11.7.1.2 Extrude Lines and Members

By extruding lines and members you can quickly create surfaces, grids or grillages. But if you want to generate an irregular grid using extended specifications, it is recommended to use the dialog box *Generate Grid* (see chapter 11.7.2, page 509).



Menu *Tools* → *Extrude*

To access the extrude functions,

select **Extrude** on the **Tools** menu.

You can also use the context menu of the relevant line or member.

Extrude member/line into surface

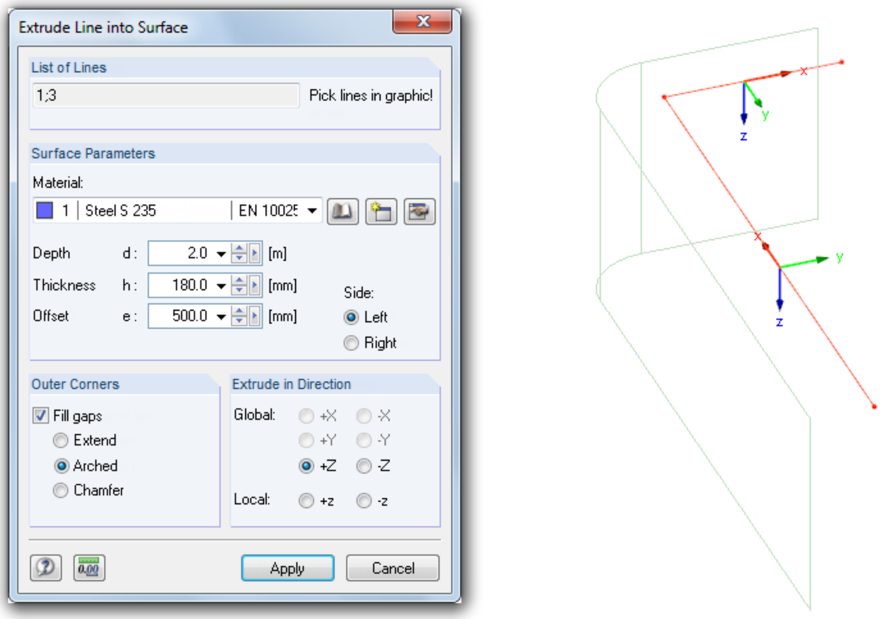


Figure 11.141: Dialog box *Extrude Line into Surface*

The selected line appears in the *List of Lines*. If necessary, more lines can be added by clicking them in the work window. Please note that all lines of the list must lie in one plane.

Then, enter the material, depth and thickness of the new surface as *Surface Parameters*. If an *Offset* is defined, the surface will be created in a lateral distance relating to the direction of the extrusion. In this case, specify the *Side*. The modified parameters are shown immediately in the graphic of the work window.

When several lines are extruded, you have different possibilities offered in the dialog section *Outer Corners* to adjust the copied lines. The figure above shows the lines (without axes) extruded with an offset and connected with an arc.

In the dialog section *Extrude in Direction*, define the global or local direction of the extrusion.

Extrude member into grid

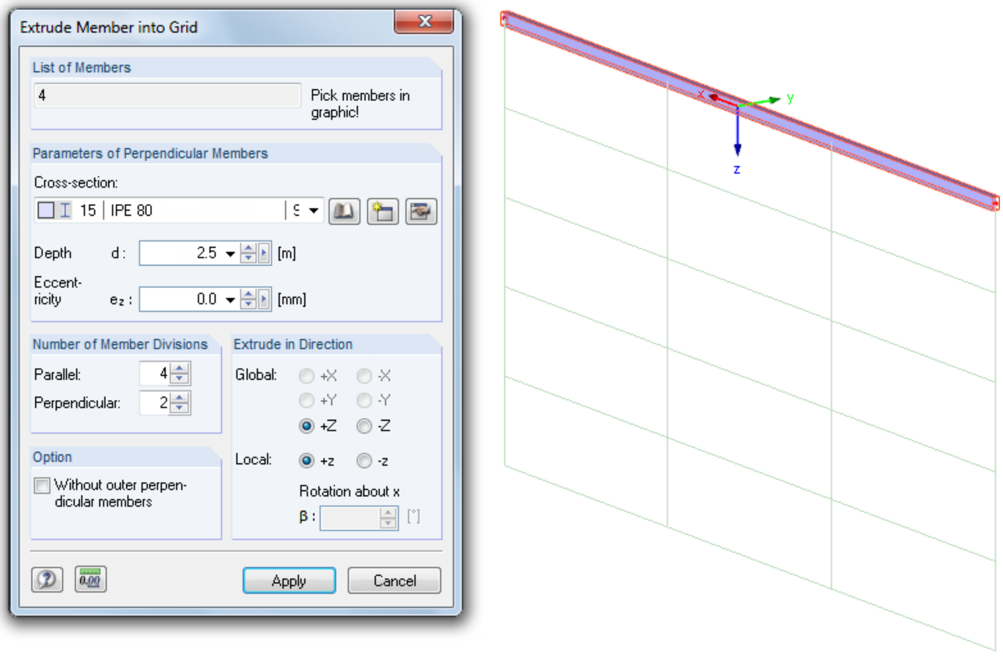


Figure 11.142: Dialog box *Extrude Member into Grid*

The selected member appears in the *List of Members*. If necessary, more members can be added by clicking them in the work window. All members of the list must lie in one plane.

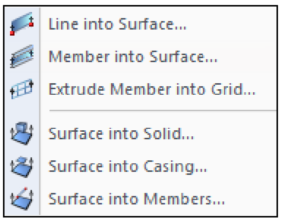
In the dialog section *Parameters of Perpendicular Members*, enter the cross-section of the vertical members and the depth as the value for the total height of the grid. Optionally, specify an eccentricity in order to connect the members by eccentric connection (see chapter 4.15, page 134).

Settings in the dialog section *Number of Member Divisions* control the division into a uniform grid consisting of parallel and vertical members. Furthermore, you have the *Option* to do without the generation of external vertical members.

In the dialog section *Extrude in Direction*, define the global or local direction where grid members will be created. The input field *Rotation about x* allows for copying objects out of the plane.

11.7.1.3 Extrude Surfaces

By extruding planar surfaces you can quickly create spatial geometric objects.



Menu *Tools* → *Extrude*

To access the corresponding functions,

select **Extrude** on the **Tools** menu.

You can also use the context menu of the relevant surface.

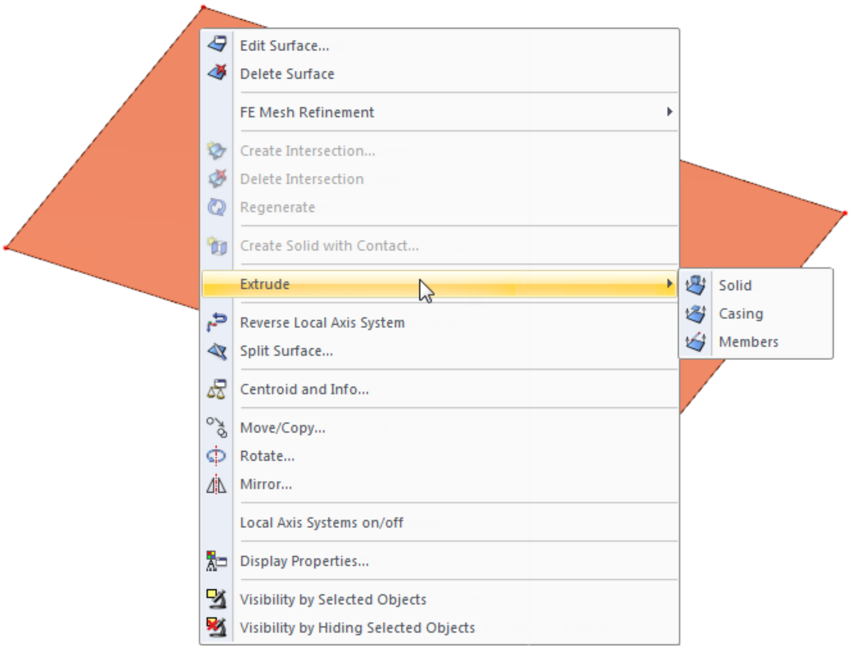


Figure 11.143: Surface context menu

To create extruded objects by shifting the surface parallel in the workspace, select one of the following three options in the context menu.

- **Solid**: A 3D-solid will be created (see chapter 4.5, page 85).
- **Casing**: Only surfaces encasing the spatial object will be generated.
- **Members**: Members will be created on the connection lines between the nodes and their copies. Optionally, the basic surface will be copied, too.

Depending on your choice, a new dialog box appears where you have to define the relevant parameters. The *Depth d* can be entered directly or determined graphically with the mouse.

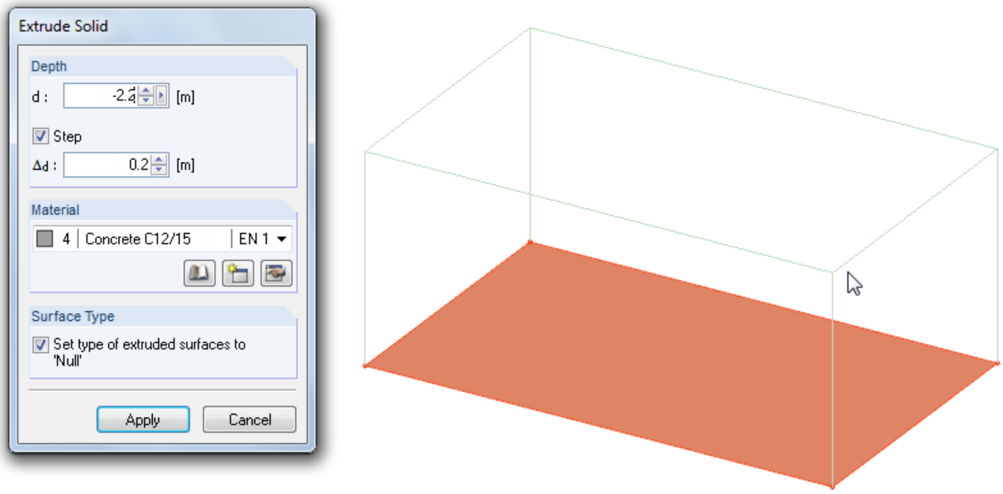


Figure 11.144: Dialog box *Extrude Solid*

11.7.1.4 Generate Solids

The previous chapter describes how existing surfaces can be used to create solids or casings. But if you want to generate a completely new solid, RFEM offers you special functions to create 3D objects: First, create the surfaces (rectangle with roundings, semi-circle etc.). In a second step, extrude the surfaces in relation to a point or a plane.

Extrude surface in relation to parallel plane

To access the corresponding function,

> point to **Model Data** on the **Insert** menu, select **Solids** and **Graphically**, and then click **Extrude Tapered**

or use the corresponding list button of the menu bar.

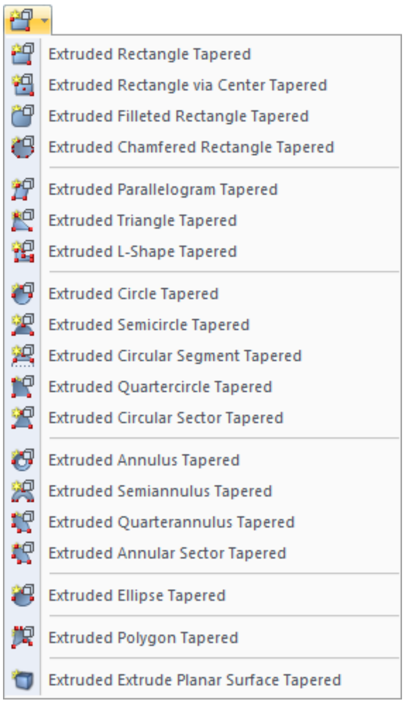


Figure 11.145: List button *Extrude Surface*

The menu contains a great number of planar surface shapes that can be defined graphically and then extruded parallel to the surface plane.

The functional principle is similar to the graphical input of surfaces (see chapter 4.4, page 75): First, define material and stiffness in a dialog box. Then, you can create the surfaces in the work window by clicking the definition points.

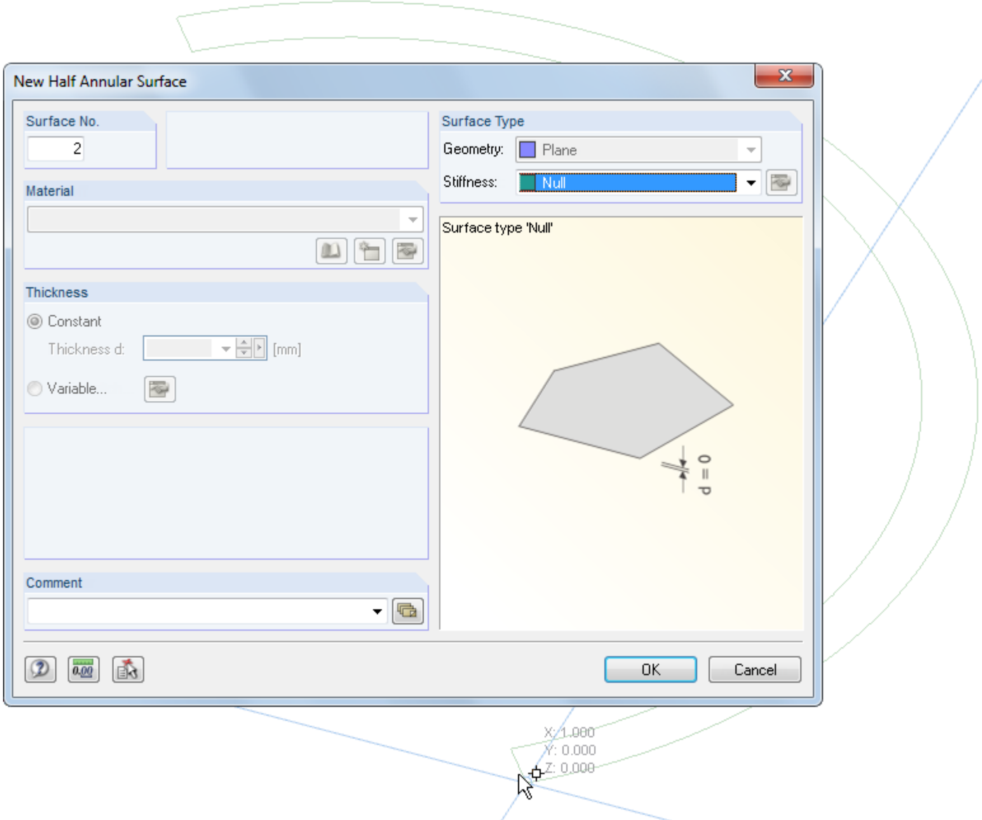


Figure 11.146: Dialog box *New Half Annular Surface* for graphical definition of surface

When the base area is set, define the parameters for creating the solid in the *Extrude* dialog box.

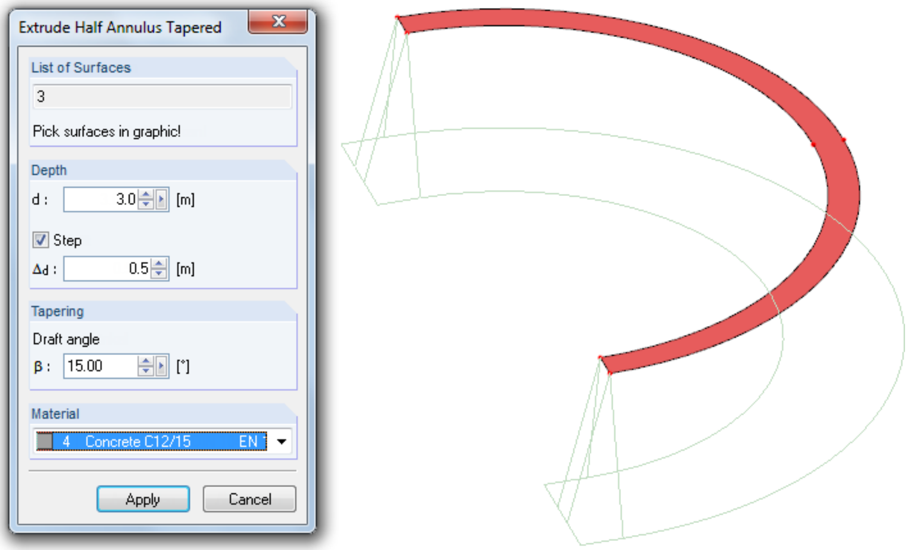
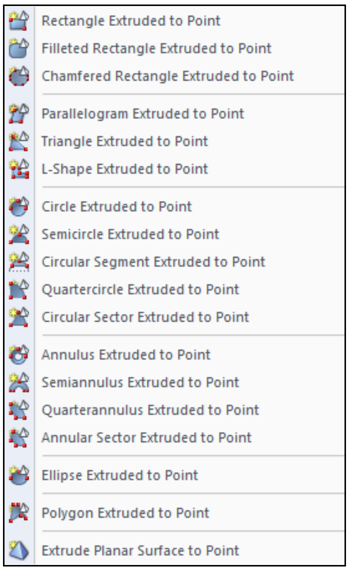


Figure 11.147: Dialog box *Extrude Half Annulus Tapered* with inclined lateral surfaces

The *Depth d* can be entered directly in the dialog box or determined graphically with the mouse. The projection direction is always right-angled to the plane of the base area.

By entering a value in the dialog section *Tapering* it is possible to create a parallel cover or base area with inclined sides. The angle β describes the inclination to the projection direction.

In addition, the *Material* of the new solid must be specified.



Extrude surface in relation to point

To access the corresponding function,

> point to **Model Data** on the **Insert** menu, select **Solids** and **Graphically**, and then click **Extrude to Point**.

The menu contains a great number of planar surface shapes that can be defined graphically and then extruded in relation to a point.

The functional principle is similar to extruding the object in reference to a parallel plane (see above): First, define the base area graphically. Then, you can enter the extrusion's projection point in the *Extrude* dialog box. You can define it also graphically.

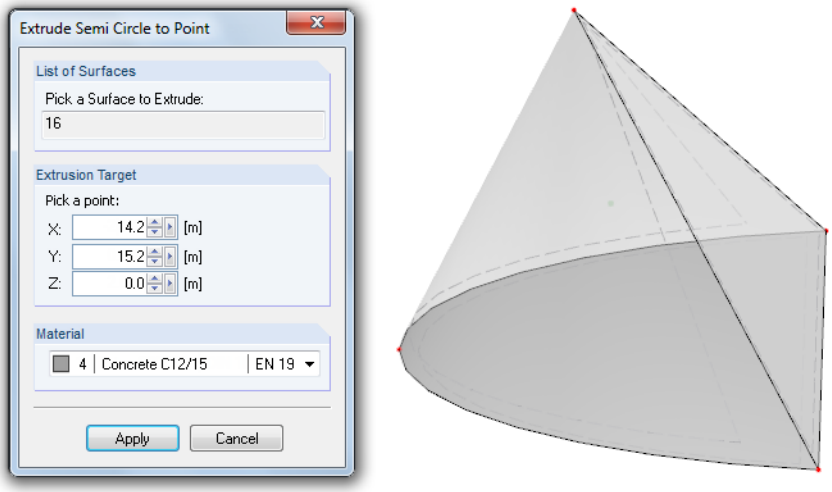


Figure 11.148: Dialog box *Extrude Semi Circle to Point* with result

11.7.1.5 Split Member into Surfaces

Sometimes, it is necessary to analyze particular zones of the framework model more in detail (for example evaluation on supports or frame joint as surface model). It would be possible but quite time-consuming to enter a cross-section using surface elements manually. The function *Generate Surfaces from Members* helps you to represent a 1D member element by means of 2D surface elements.

The function is only available if the model type has been defined as *3D* (see Figure 12.23, page 554).

To split a previously selected member,

> point to **Generate Surfaces from Members** on the **Tools** menu, and then select **Generate**.

This function is also available in the member context menu. Right-click the member to open its context menu.

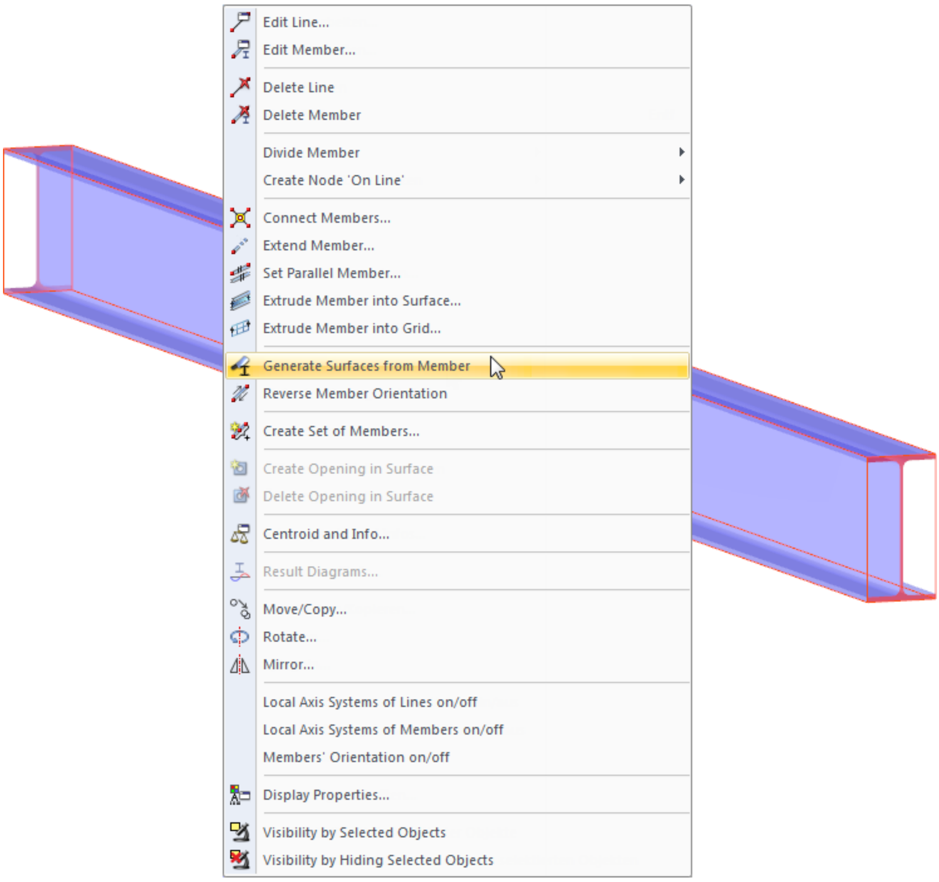


Figure 11.149: Context menu of member

The information about the member won't be lost: In addition to surfaces, a dummy member will be created in the centroidal axis. The dummy contains all member data but will not be considered in the calculation.

To access more options for the function *Generate Surfaces from Members*,

point to **Generate Surfaces from Members** on the **Tools** menu, and then select **Settings**.

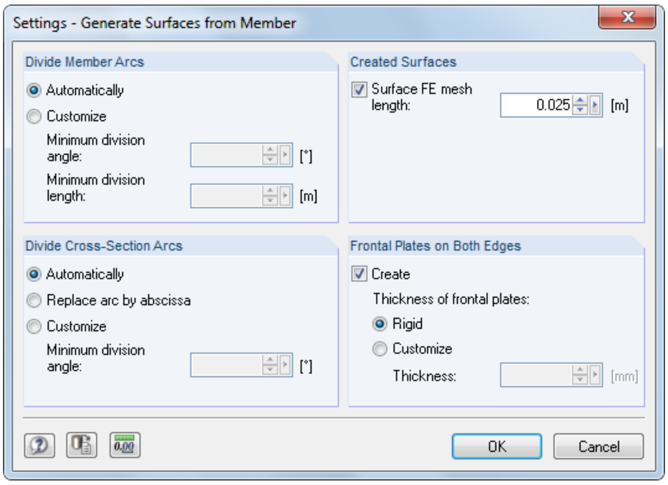


Figure 11.150: Dialog box *Settings - Generate Surfaces from Member*

Use the dialog section *Divide Member Arcs* to control how many times a member lying on a curved line is to be split. If a very rough polygonal chain is created with the default setting *Automatically*, you can customize the splitting and reduce the *division angle* or the *division length*.

The settings in the dialog section *Divide Cross-Section Arcs* affect the splitting of curved surfaces, for example members of the cross-section type "Pipe". Again, the division can be refined by a user-defined *division angle*.

In the dialog section *Created Surfaces*, you can define a FE mesh refinement for the generated surfaces (see chapter 4.23, page 166).

In the course of the conversion, *Frontal Plates* can be additionally created at the member ends. Characteristics of generated surfaces can be adjusted subsequently by editing the surfaces (see chapter 4.12, page 112).

11.7.2 Model Generators

To access the dialog boxes for creating model objects,

select **Generate Model - Members** on the **Tools** menu or

select **Generate Model - Surfaces** on the **Tools** menu.

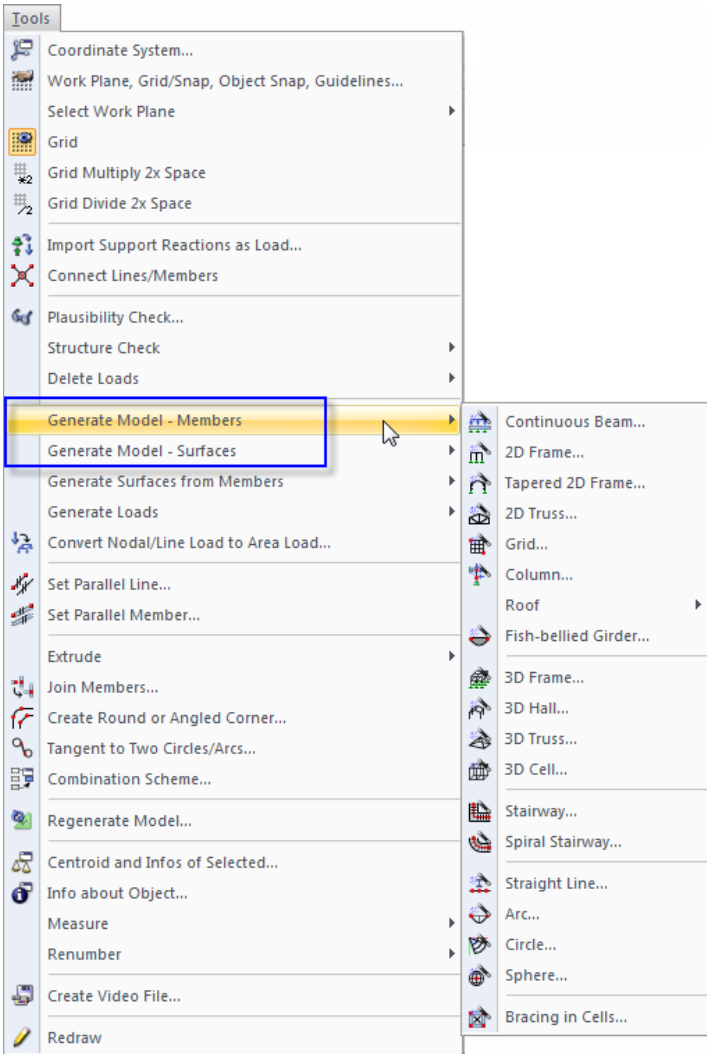


Figure 11.151: Menu *Tools* → *Generate Model - Members* or *Surfaces*

In the following, the single generators are presented. However, you won't find a detailed description of the dialog boxes because the dialog graphics illustrate the parameters adequately.

Each dialog input can be saved as a template and reused later. Both buttons shown on the left are used to save and load the generator data.

11.7.2.1 Members

Continuous beam

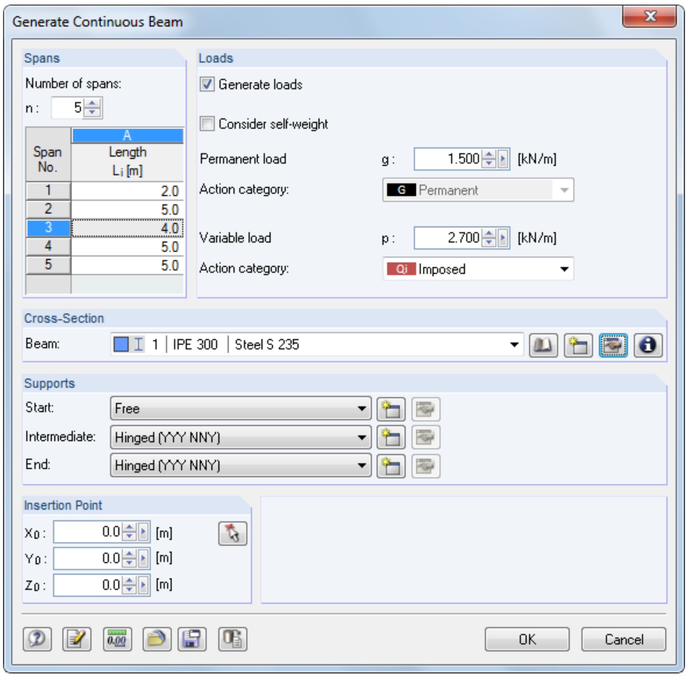


Figure 11.152: Dialog box *Generate Continuous Beam*

RFEM creates a continuous beam with uniform cross-section, supports and irregular spans. Optionally, load cases and result combinations are created, too.

2D frame

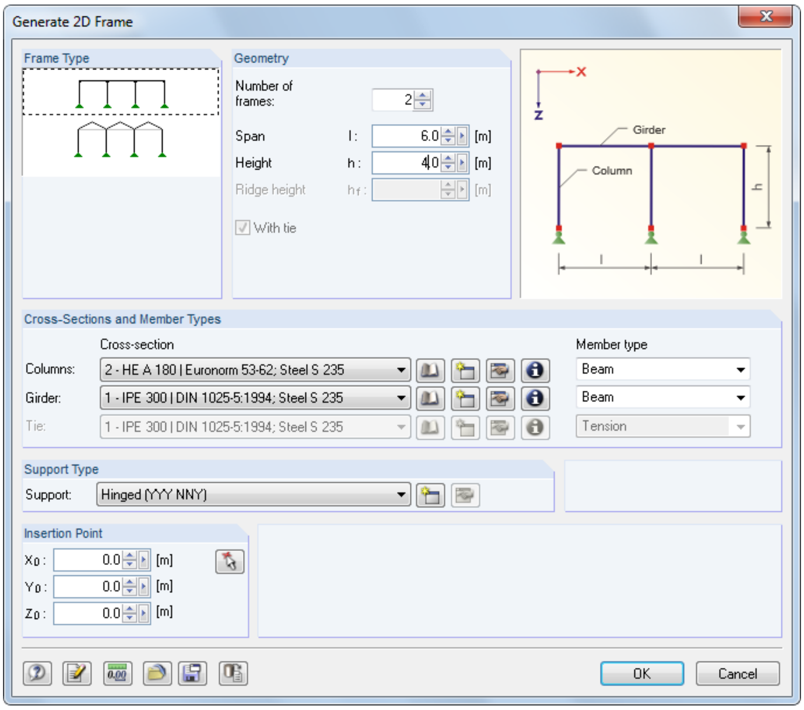


Figure 11.153: Dialog box *Generate 2D Frame*

Before you enter geometrical data and cross-section properties, select the *Frame Type*. The columns of the planar frame receive equal support conditions.

Tapered 2D frame

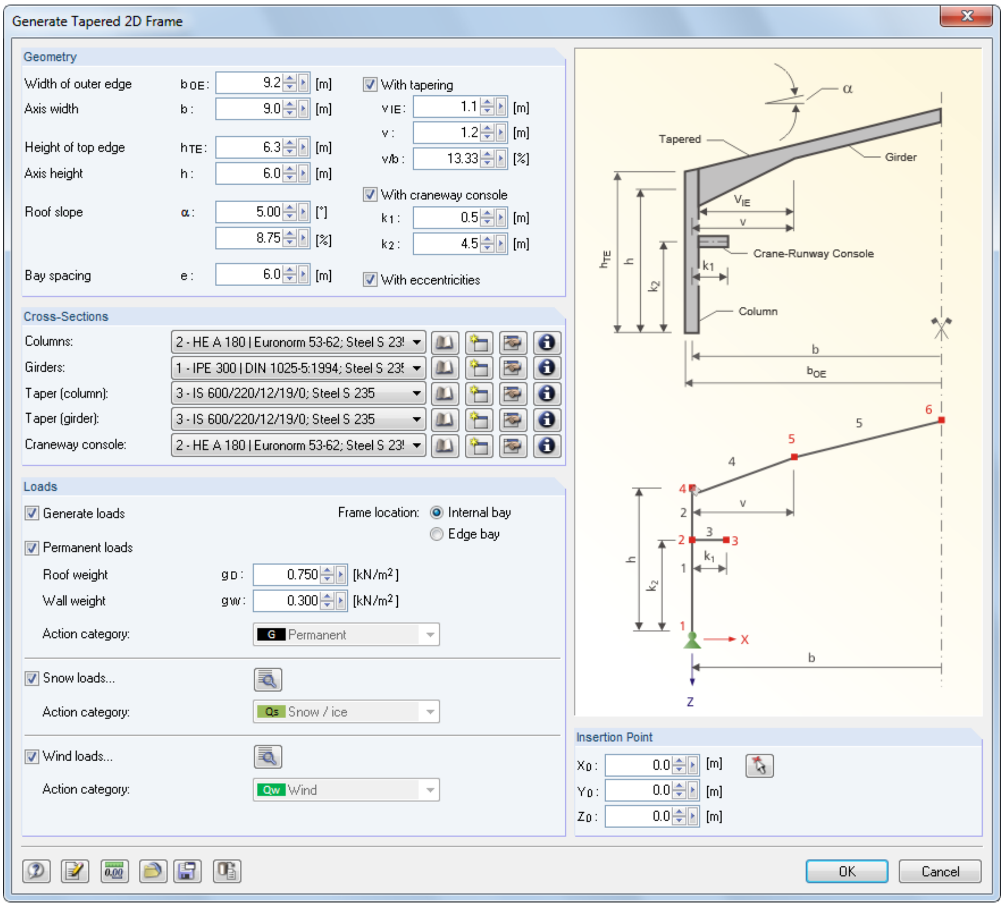


Figure 11.154: Dialog box *Generate Tapered 2D Frame*

The planar frame must be defined by its *Geometry* and *Cross-Sections*. You can create tapers, craneway consoles and eccentric connections. *Loads* can be generated additionally. The [Settings] buttons offer you access to the generator parameters. The *Frame location* is important for the load determination.

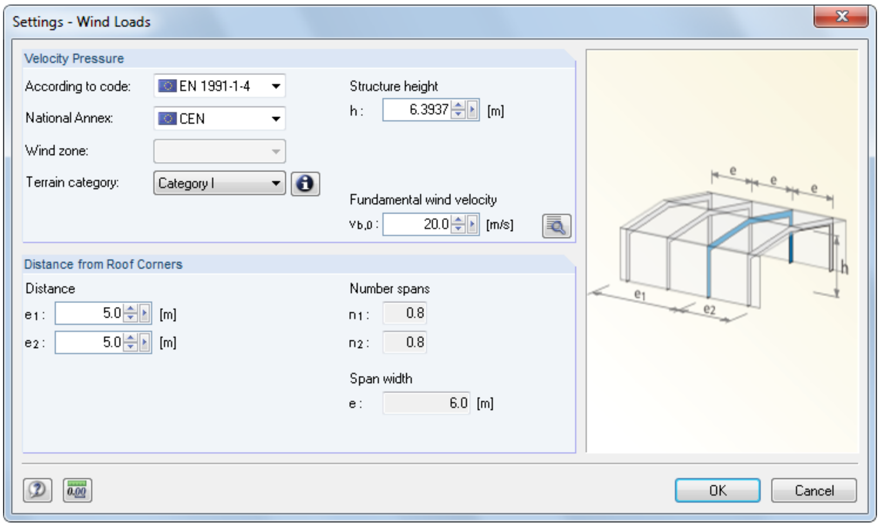


Figure 11.155: Dialog box *Settings - Wind Loads*

2D truss

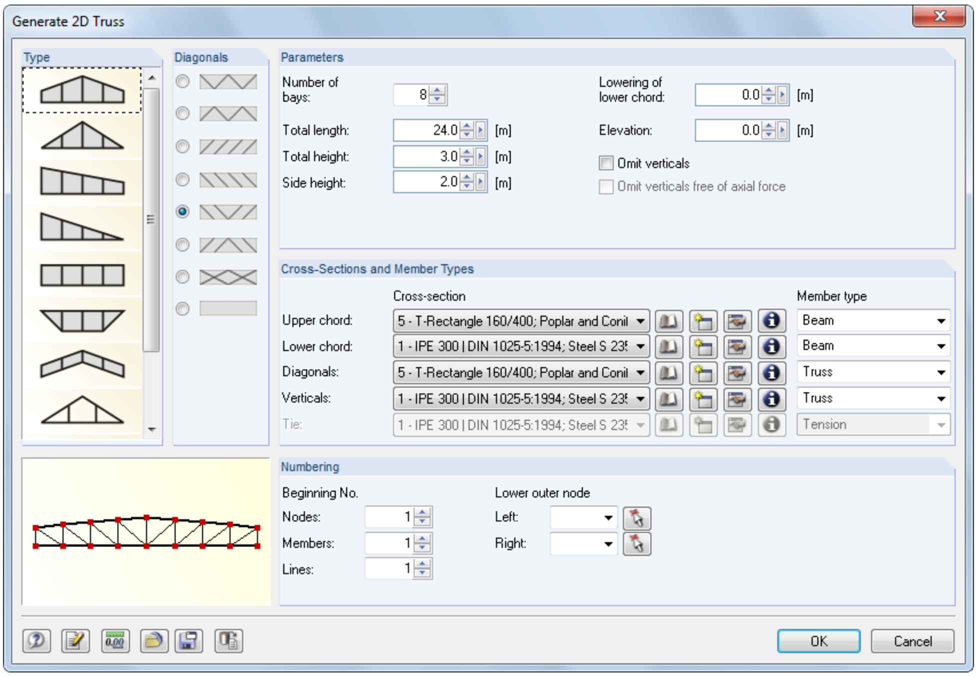


Figure 11.156: Dialog box *Generate 2D Truss*

First, define the *Type* of the truss and the arrangement of the *Diagonals*. Then, you can define the *Parameters*, *Cross-Sections and Member Types*.

Grid

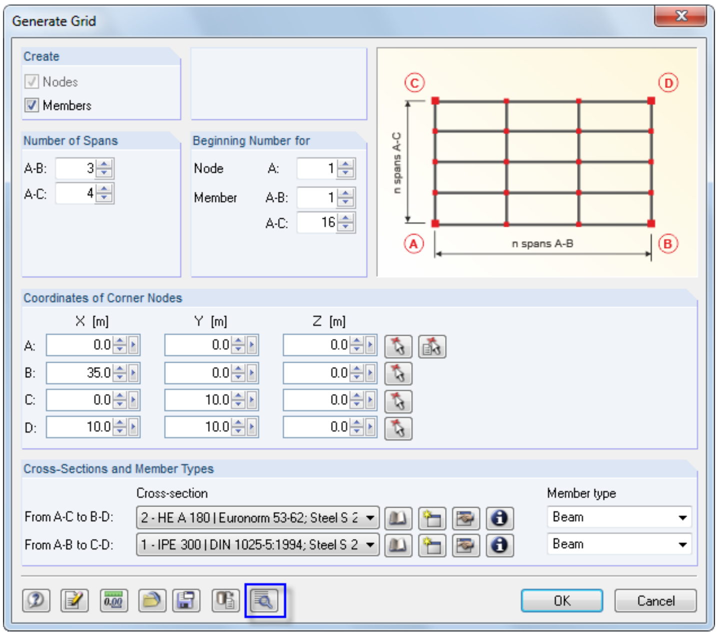


Figure 11.157: Dialog box *Generate Grid*

Use this generator to create models that have a uniform grid (for example gratings). They don't need to be designed with right angles as shown in the dialog graphic above. Any kind of spatial quadrangle model with four corner points is possible. To generate a "real" girder grillage, it is recommended to set the *Type of Model* to **2D - in XY** in the model's *General Data* dialog box (see chapter 12.2, page 554).

To generate irregular grids, use the button [Edit Advanced Settings] shown on the left.

Column

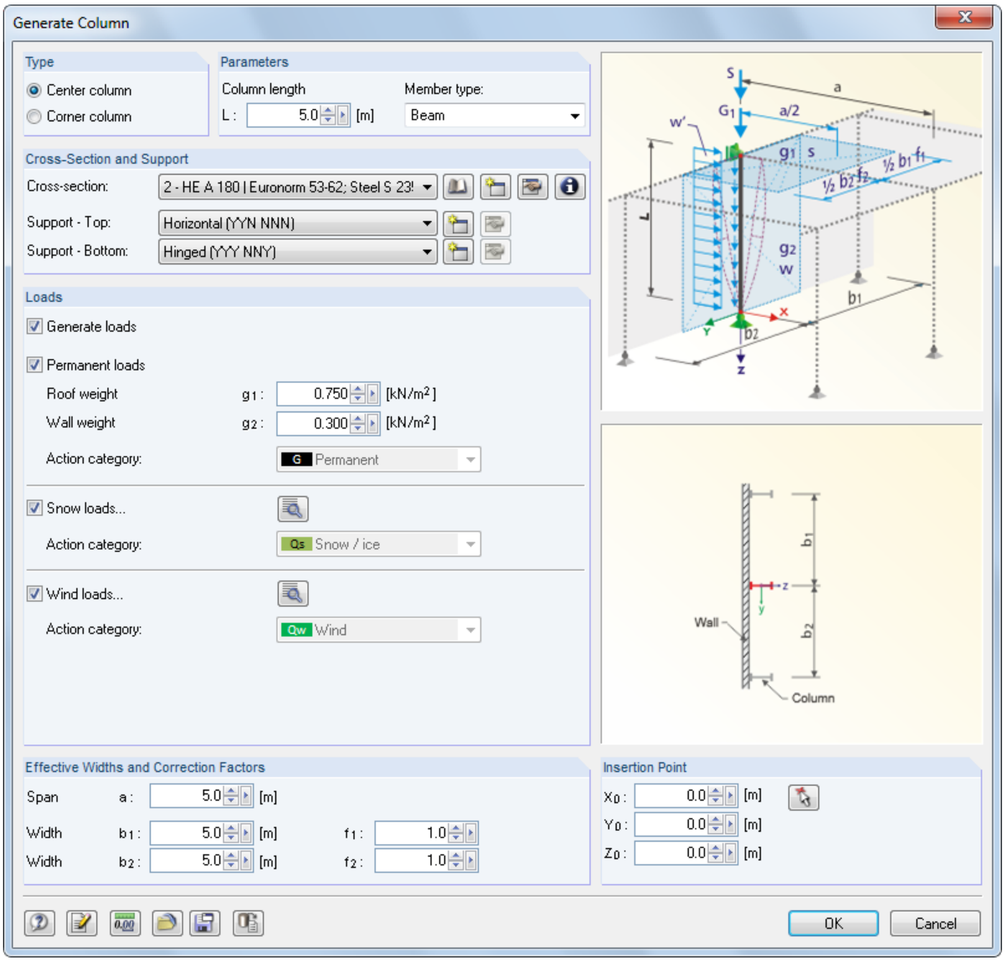


Figure 11.158: Dialog box *Generate Column*

In the dialog section *Type*, you decide whether a center or corner column will be generated. In case you want to generate *Loads*, you have to specify their *Effective Widths and Correction Factors*. For generating a gable column the ***Span a*** is required for the influence range in the longitudinal direction of the hall. The factors $\boldsymbol{f_1}$ and $\boldsymbol{f_2}$ are used to scale the geometric widths b_1 and b_2 for the static model or to fulfill special code requirements (for example load increment factors for individual designs).

Roof generators

The menu item *Roof* provides three roof generators which you can select to generate planar roof systems including loads. The [Settings] buttons available in the roof dialog boxes help you to determine wind and snow loads (see Figure 11.155, page 508).

Roof → Collar Roof

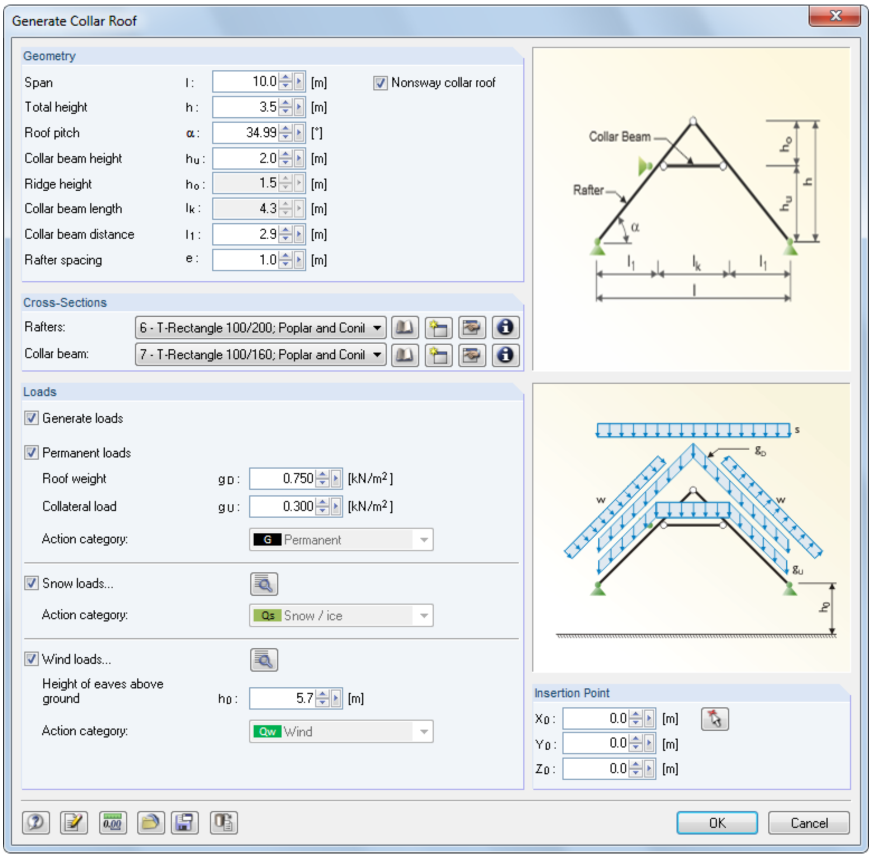


Figure 11.159: Dialog box *Generate Collar Roof*

Roof → Rafter Roof

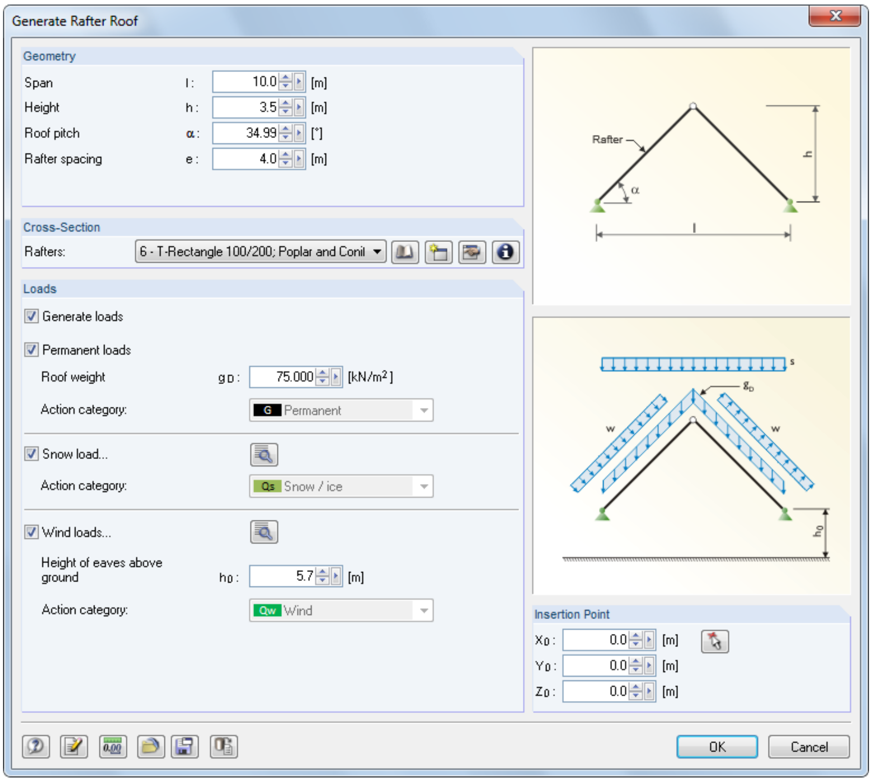


Figure 11.160: Dialog box *Generate Rafter Roof*

Roof → Purlin Roof

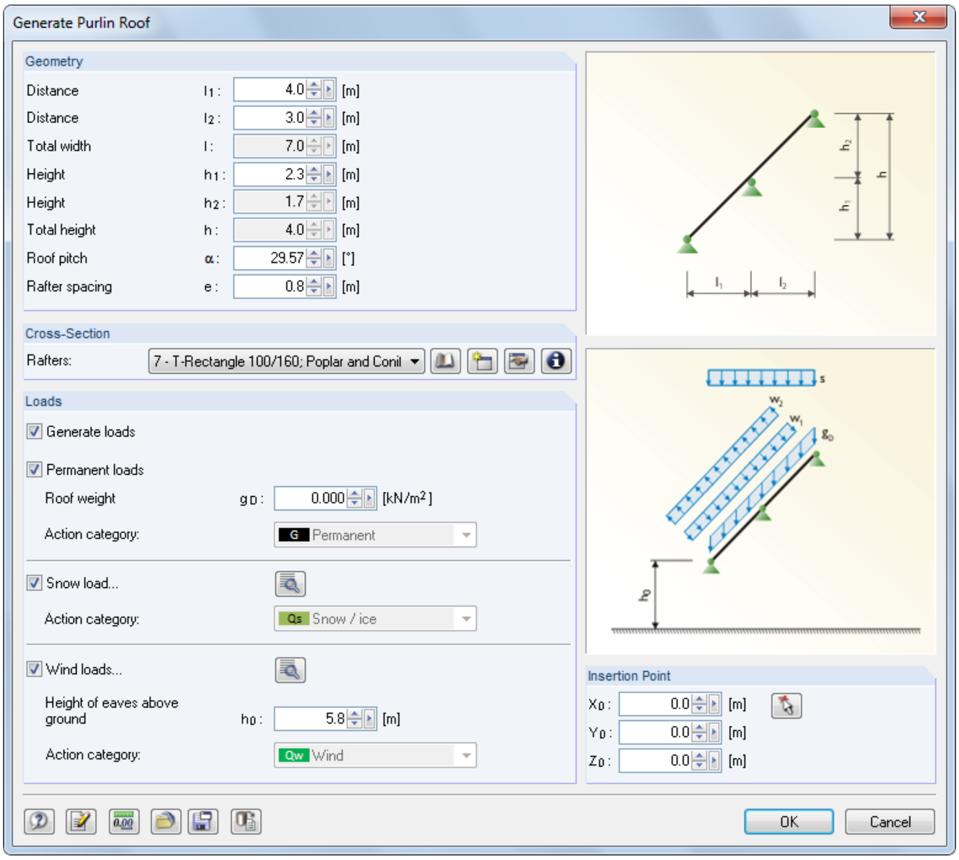


Figure 11.161: Dialog box *Generate Purlin Roof*

Fish-bellied girder

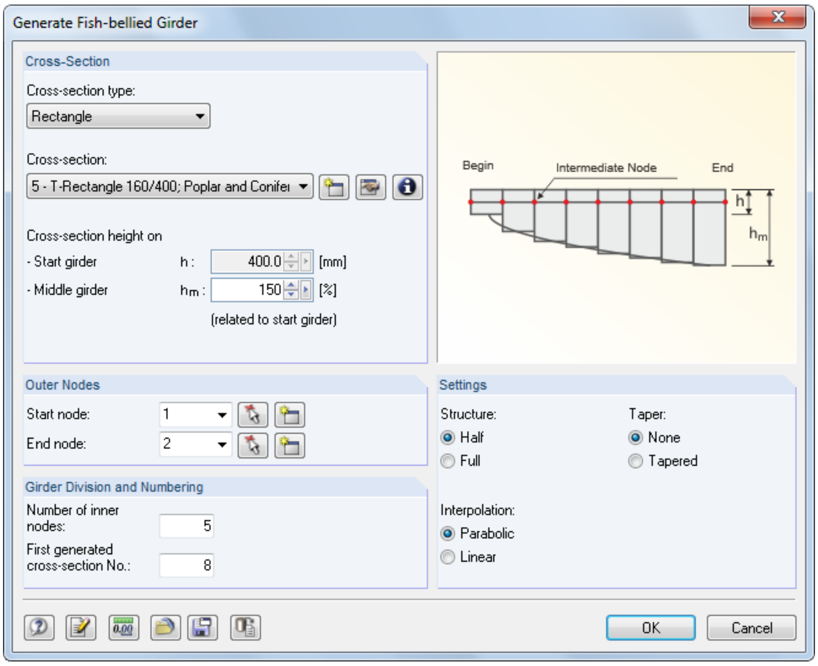


Figure 11.162: Dialog box *Generate Fish-bellied Girder*

For the generation of fish-bellied girders commonly used in timber construction, the rectangular and ITS section types (symmetric I-beams) can be selected in the *Cross-section type* list.

3D frame

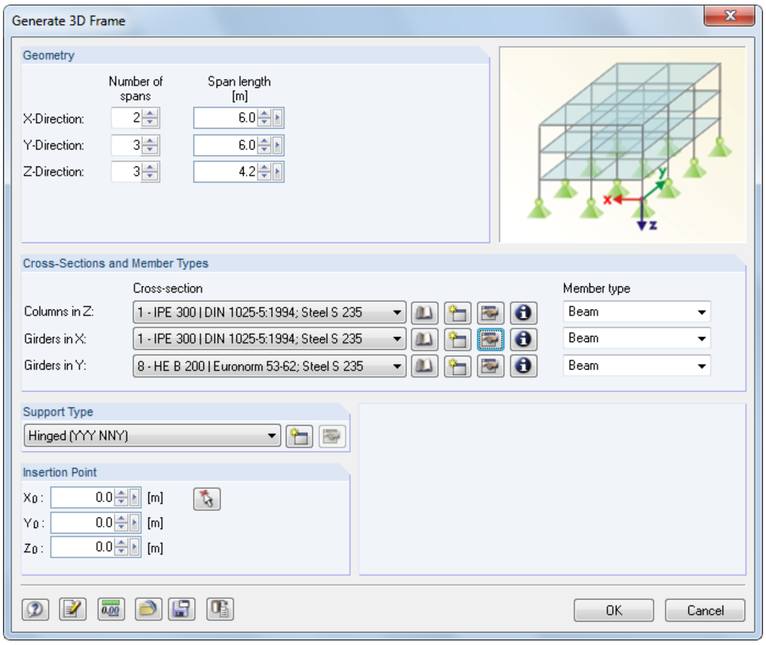


Figure 11.163: Dialog box *Generate 3D Frame*

Use this generator to create regular frame models. The columns of the frame receive equal support conditions.

3D hall

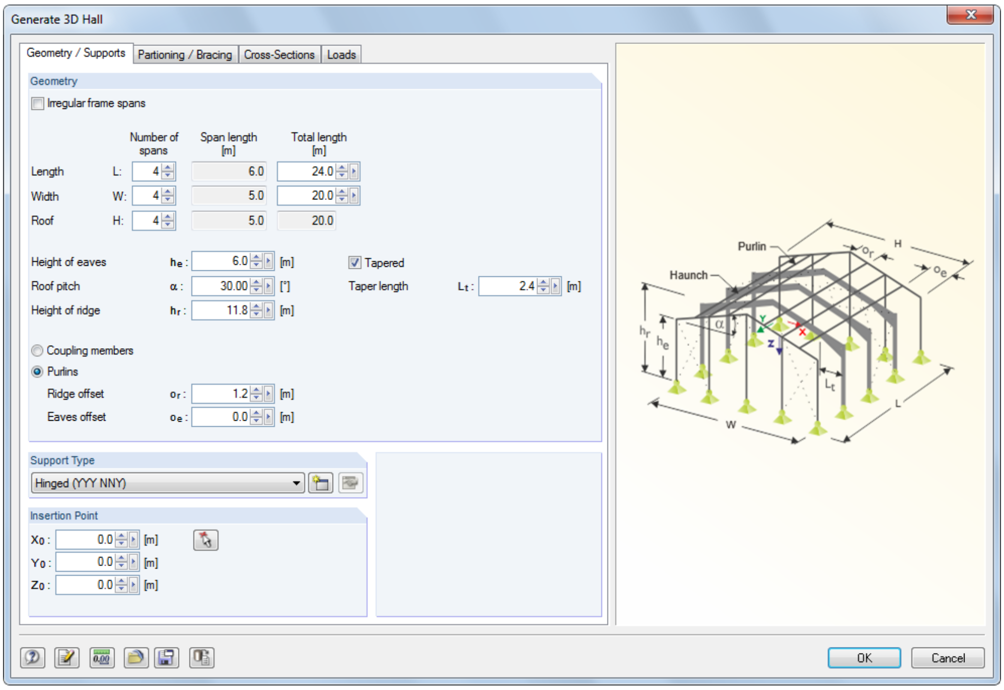


Figure 11.164: Dialog box *Generate 3D Hall*

This complex generator creates a complete hall including loads. Four dialog tabs are provided: *Geometry / Supports* manages the system geometry, *Partitioning / Bracing* controls irregular grid spacings and the arrangement of bracings. In the remaining two tabs, the *Cross-Sections* and *Loads* are defined.

3D truss

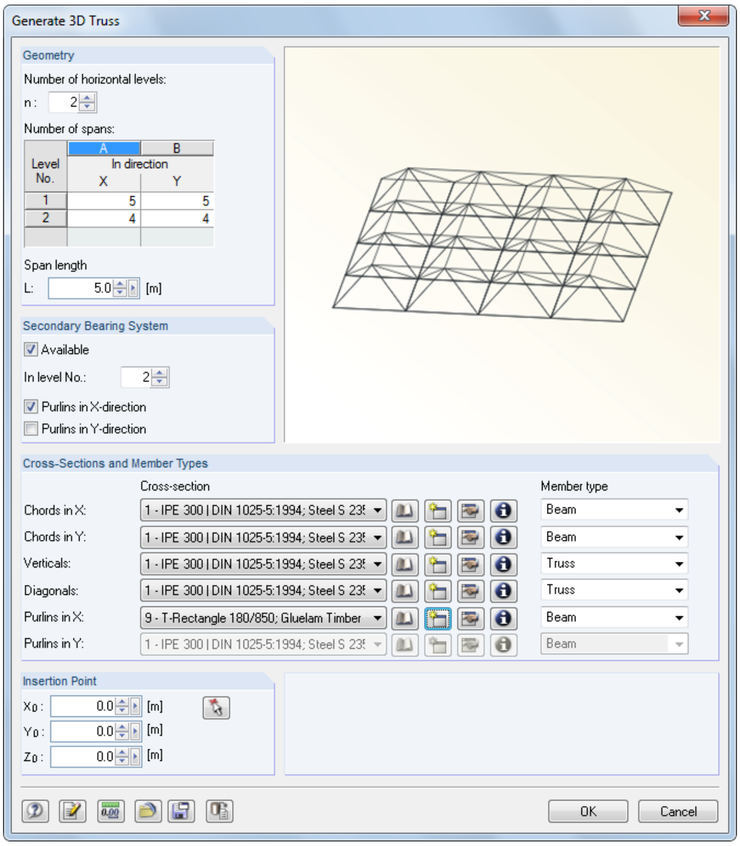


Figure 11.165: Dialog box *Generate 3D Truss*

Use this generator to create a spatial load-bearing structure according to the *Bernauer* system (www.raumtragwerke.de).

3D cell

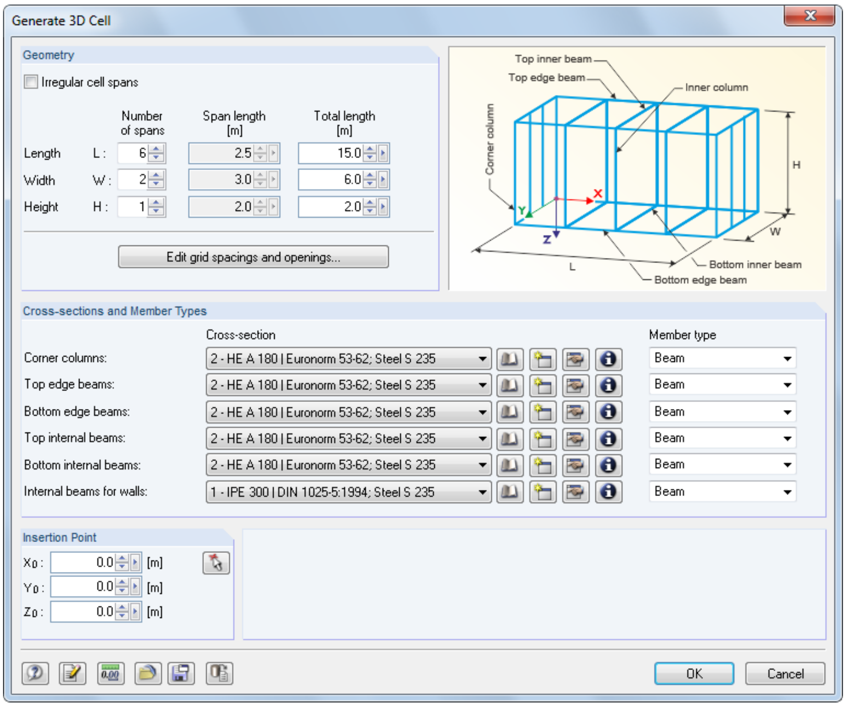


Figure 11.166: Dialog box *Generate 3D Cell*

Edit grid spacings and openings...

The generator creates a spatial cell with several fields. Use the button [Edit grid spacings and openings] to open another dialog box where you can define openings as well as the grid arrangement for irregular field spacings.

Stairway

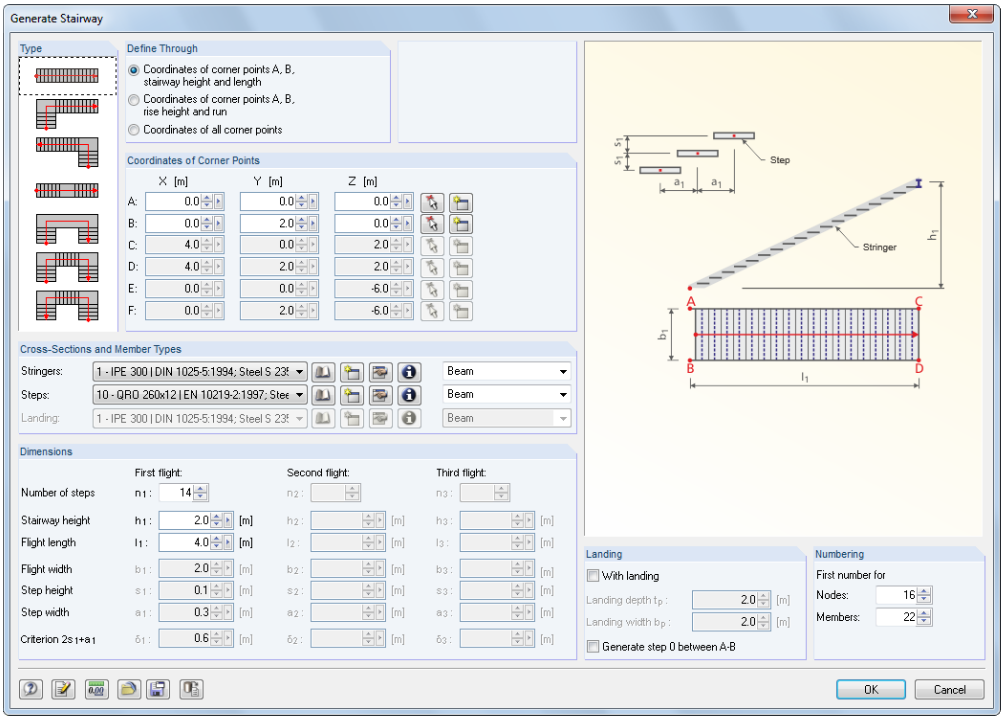


Figure 11.167: Dialog box *Generate Stairway*

In the list, select the *Type* controlling the remaining parameters.

Spiral stairway

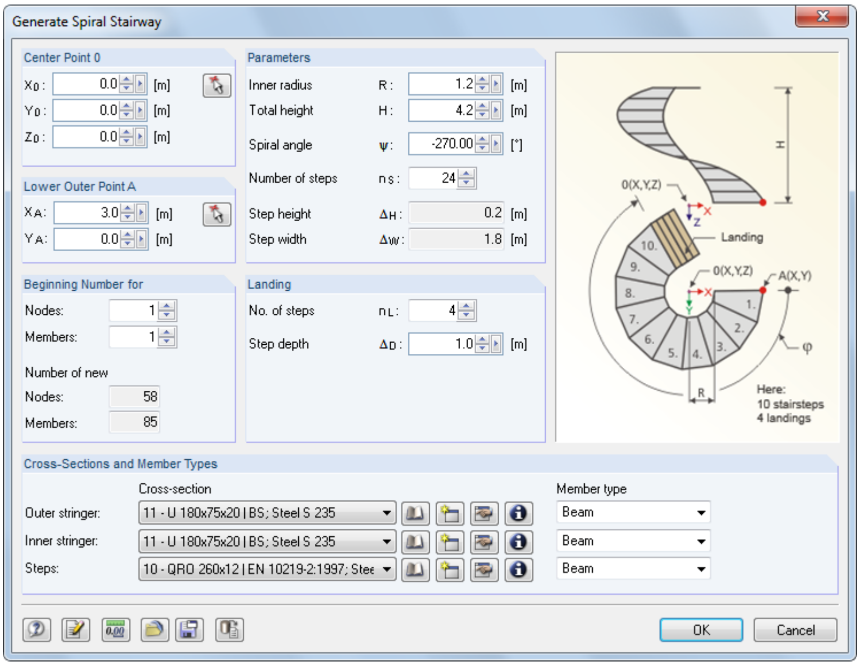


Figure 11.168: Dialog box *Generate Spiral Stairway*

Straight line

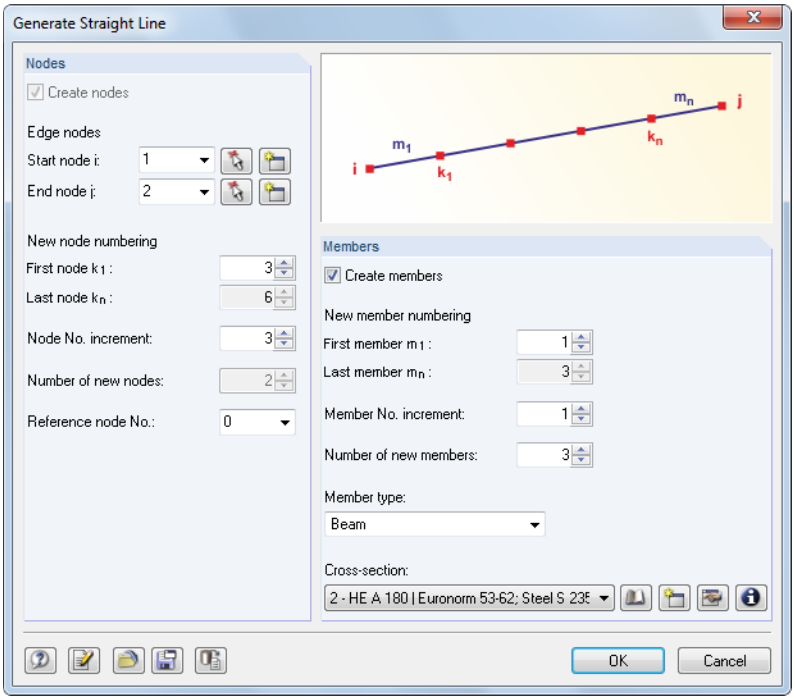


Figure 11.169: Dialog box *Generate Straight Line*

This function allows for generating straight lines based on new or already existing nodes. It is also possible to create only nodes placed on an imaginary straight line.

Arc

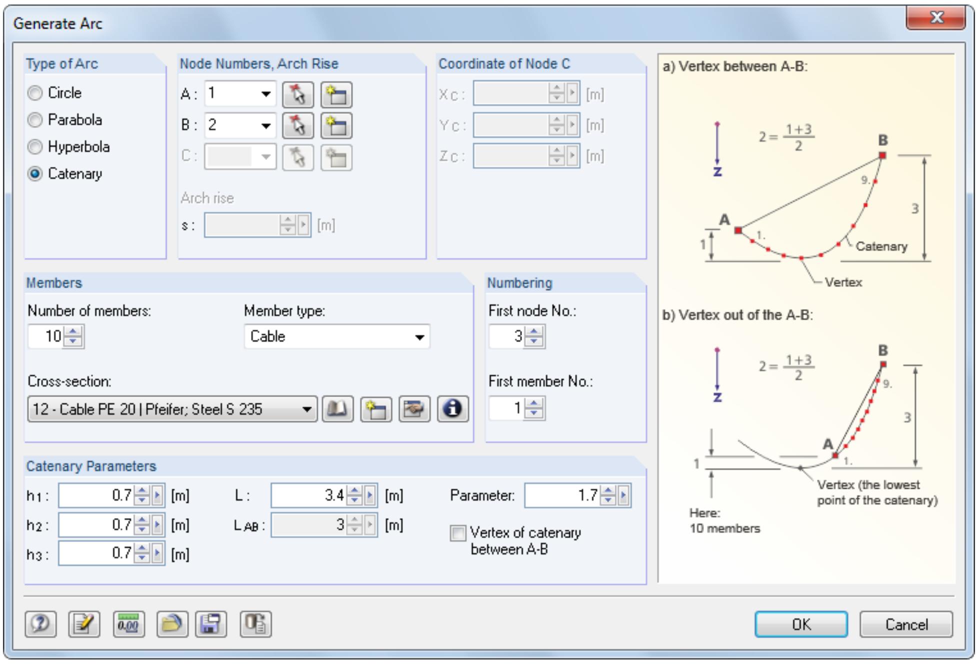


Figure 11.170: Dialog box *Generate Arc*

First, define the *Type of Arc*: circle, parabola, hyperbola or catenary. Points *A* and *B* represent both edge nodes of the arc, point *C* determines its arrangement. The *Arch rise* defines the sag.

The length of a catenary is defined by the parameter *L*. The heights h_1, h_2 and h_3 are interactive values. The *Parameter* describes the constant a in the following equation of the catenary curve:

$$y(x) = a \cdot \cosh\left(\frac{x - v_x}{a}\right) + v_y \qquad \text{where } v_x \text{ or } v_y: \quad \text{displacements in x or y}$$

Equation 11.1

The larger the *Number of members*, the more precisely the arc will be modeled as polygonal line.

Circle

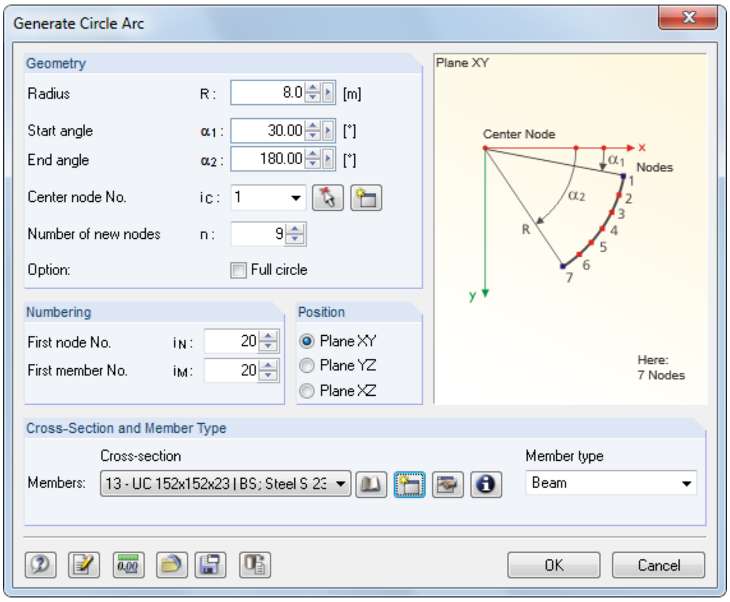


Figure 11.171: Dialog box *Generate Circle Arc*

The circle or circle arc is defined by *Radius* and angles. The object will be created around a center point that can be selected anywhere in one of the global planes.

Sphere

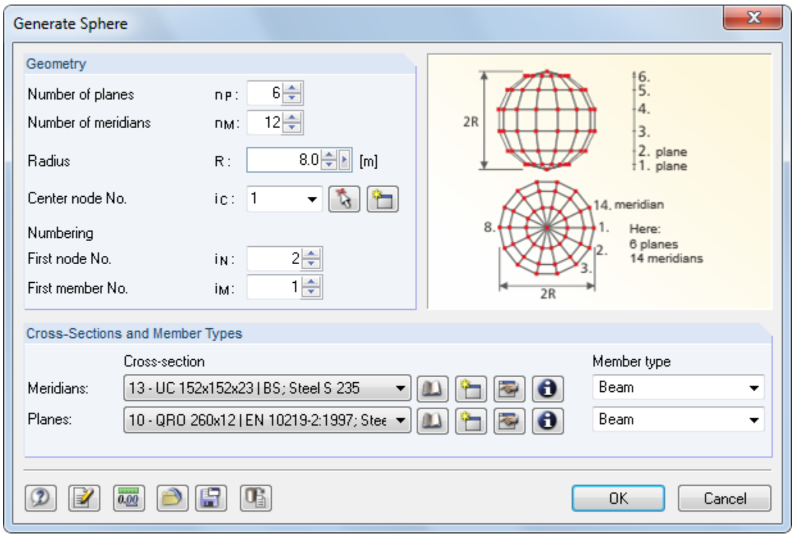


Figure 11.172: Dialog box *Generate Sphere*

The larger the *Number of planes* and *meridians*, the rounder the shape of the sphere will be. Polygonal lines approximate the spherical form, with each member representing a segment.

Bracing in cells

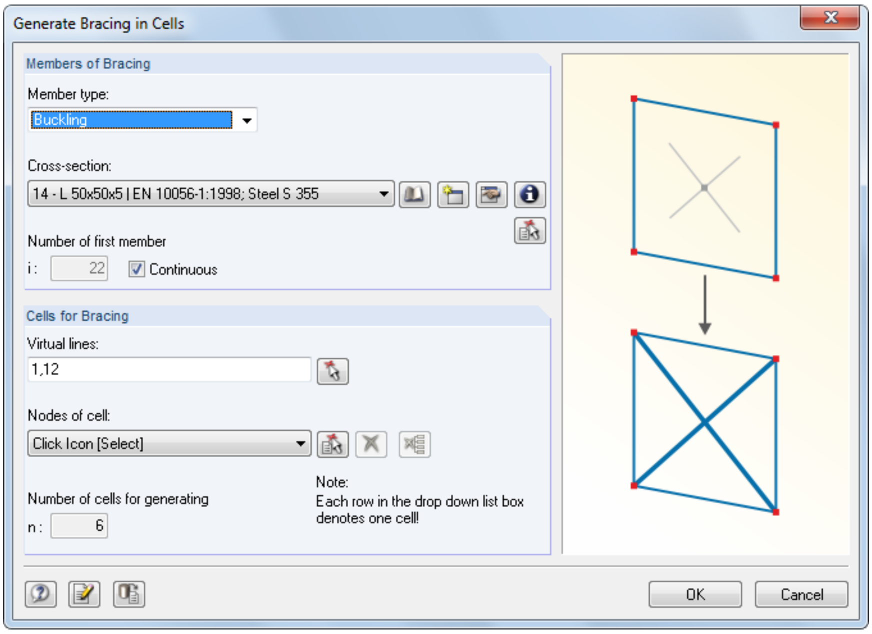


Figure 11.173: Dialog box *Generate Bracing in Cells*

Cells are defined by four corner nodes, enclosed by members on all sides and placed in one plane. In the generator dialog box, specify the *Members of Bracing* and the *Cells for Bracing*. You can also use the [↖] function to select them in the work window by clicking the cell crosses.

Furthermore, *Virtual lines* make it possible to close cells so that bracings can be created as well for example between wall supports.

11.7.2.2 Surfaces

Vaulted head according to DIN 28 011 or DIN 28 013

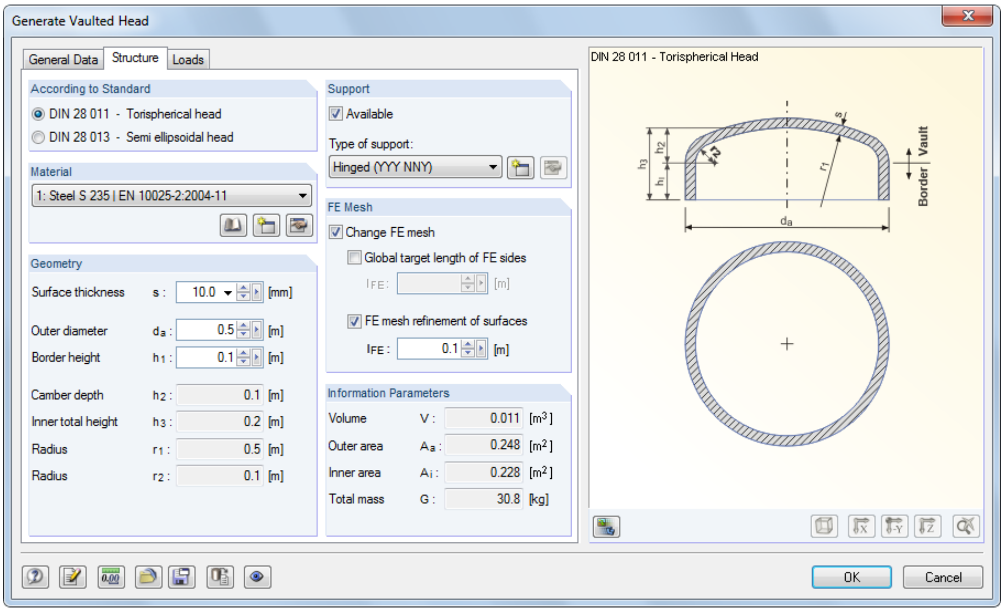


Figure 11.174: Dialog box *Generate Vaulted Head*, tab *Structure*

For creating a vaulted head, RFEM offers you the standard variants *Torispherical head* and *Semi ellipsoidal head*. Once the reference point for placing the head is set in the *General Data* tab, you can define material and generator parameters for surface thickness, outer diameter and border height in the dialog tab *Structure*. Moreover, it is possible to specify overpressure as surface load for the generation in the dialog tab *Loads*.

Barrel roof

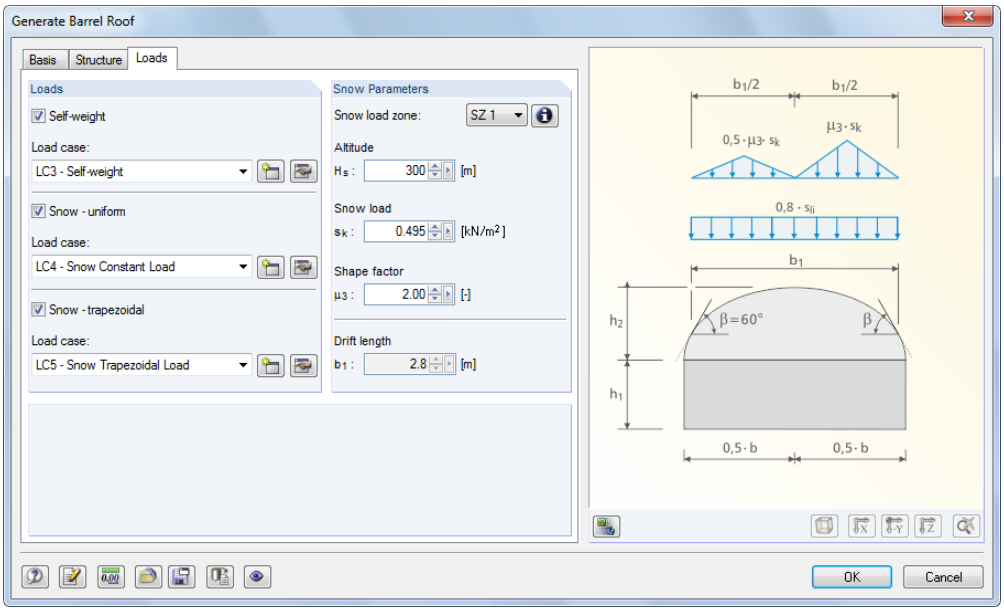


Figure 11.175: Dialog box *Generate Barrel Roof*, tab *Loads*

Define the parameters of the barrel roof in the dialog tabs *Basis* and *Structure*. In the *Loads* tab, enter the required data for the creation of snow load cases.

Domed roof

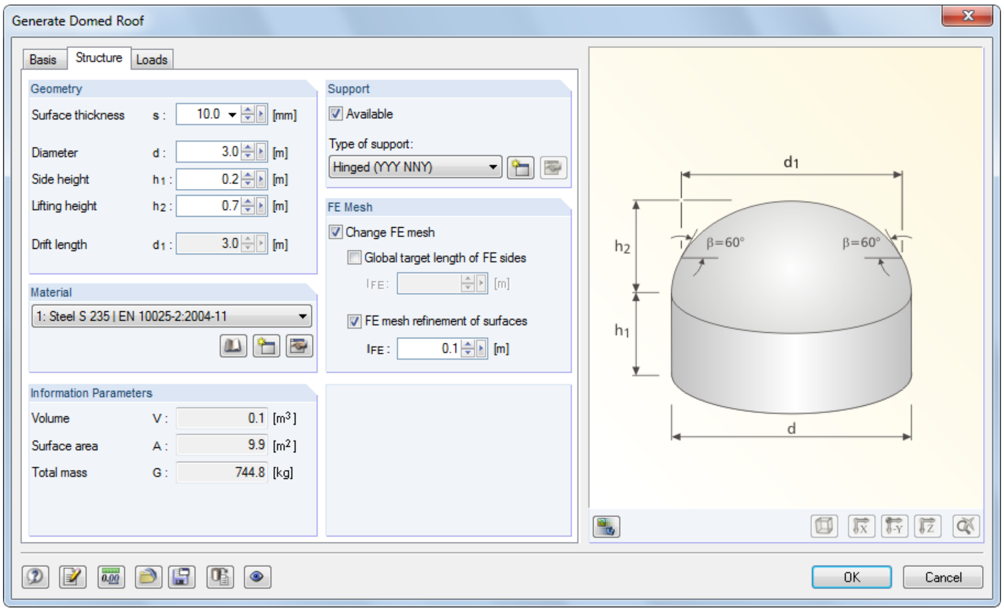


Figure 11.176: Dialog box *Generate Domed Roof*, tab *Structure*

Define the parameters of the domed roof in the dialog tabs *Basis* and *Structure*. In the *Loads* tab, enter the required data for the creation of snow load cases.

Surfaces from cells

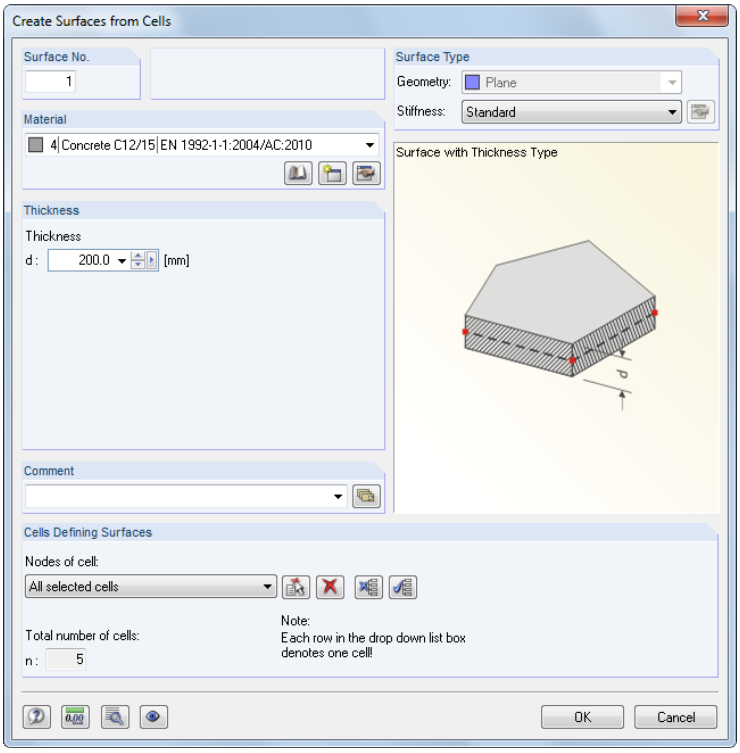


Figure 11.177: Dialog box *Create Surfaces from Cells*

Cells consists of at least three corner nodes. They are enclosed by lines or members on all sides and placed in one plane. To fill cells with surfaces, define the *Material* and *Thickness* of the surface first. Then, select the cells graphically by using the [↖] function clicking the cell crosses in the work window.

11.8 Load Generators

The second group of generators helps you to apply member and area loads: On the one hand, it is possible to convert area loads (for example snow, wind) acting on the structural system into member and surface loads. On the other hand, you can convert free line loads and coating loads due to frost into member loads.

To open the dialog boxes for generating member and surface loads,

select **Generate Loads** on the **Tools** menu.

11.8.1 General Features

Settings for load generation

Many generator dialog boxes offer you the [Settings] button (see Figure 11.184, page 524) that opens the dialog box *Settings for Load Generation* used to control the tolerance for the integration of nodes in the load plane and to correct the generated loads.

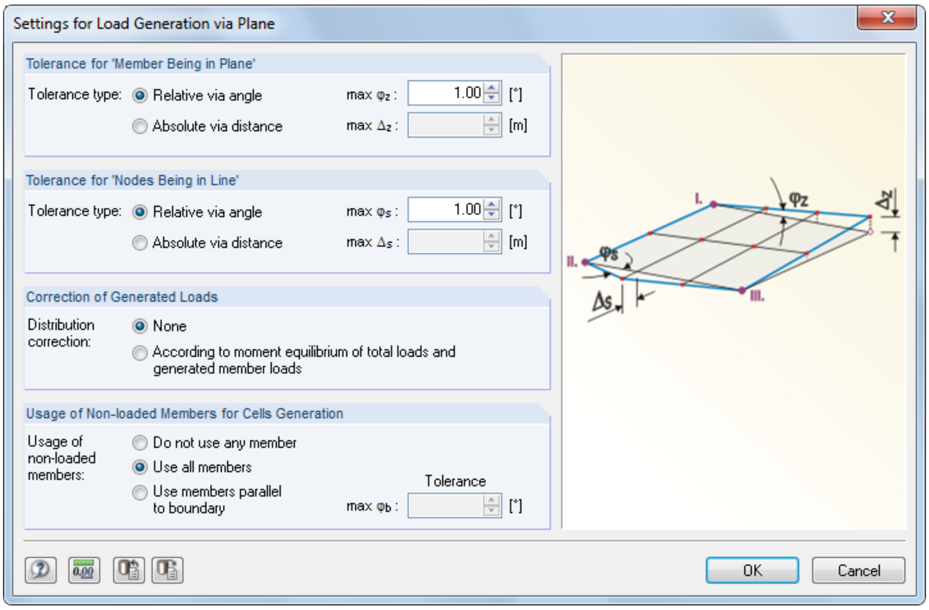


Figure 11.178: Dialog box *Settings for Load Generation via Plane*

Specifications in the settings dialog box are valid for all member load generators. The *Tolerance* determines the conditions under which members or nodes are considered to be belonging to a plane or line. Settings are possible by entering an angle or a distance. If nodes are lying within the defined thresholds, RFEM will recognize the cells and generate loads.

The dialog section *Correction of Generated Loads* allows for a comparison of the available area loads with the determined member loads. The check sums are displayed in the dialog boxes appearing after the load generation and before the final conversion into member loads will be performed (see Figure 11.188, page 527). In case of minor differences, you should make a correction of the distribution in accordance with the *moment equilibrium*. The following applies:

$$\int_{L_{cell}} (q_{member} + q_{correct})\,dL = \int_{S_{cell}} q\,dS \qquad \text{Equilibrium of forces}$$

$$\int_{L_{cell}} (q_{member} + q_{correct})\,r\,dL = \int_{S_{cell}} q\,r\,dS \qquad \text{Moment equilibrium}$$

where $r = (x, y)$ Distance to centroid of cell

When correcting generated loads by the *moment equilibrium*, the moment is formed from the area loads to the centroid and then compared with the moment from the member loads to the centroid. As a simplification, you may imagine the moment correction to be a recalculation of the support forces. This support force will then be applied as line load to the member. Take advantage of this correction option to create for example trapezoidal member loads from variable area loads.

Settings in the dialog section *Usage of Non-loaded Members for Cell Generation* primarily concern members that lie in an inclined position within the model. In the course of the load generation, the total area to be loaded will be determined first. Then, RFEM examines the members which enclose cells. Next, the cells are subtracted from the total area. When excluding a member from the loading (option *Remove Influence from* members, see below), RFEM will relocate its load to the remaining members of the plane or cell.

Now, the three options are explained by an example of a platform construction. We want to apply only traffic loads to members running in direction X. Like the Y-parallel members the inclined member is excluded from the load application, but depending on the setting it does affect the creation of member loads.

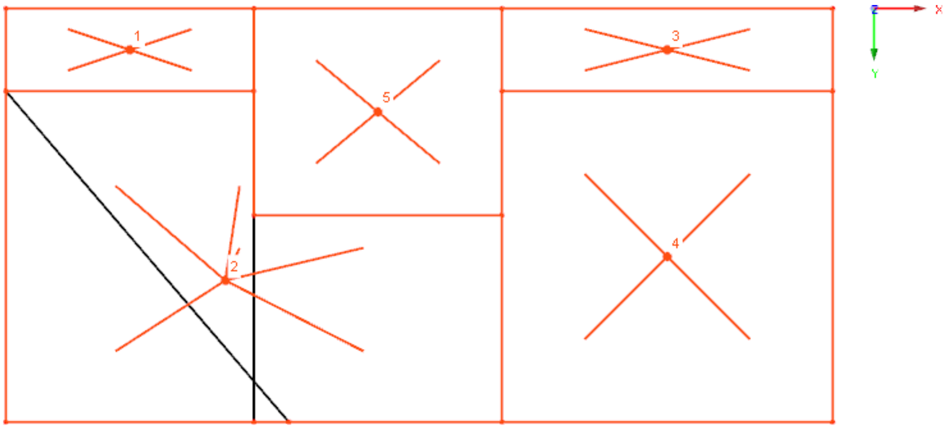


Figure 11.179: Platform construction with cells for load generation

- *Do not use any member*:
  The load is applied uniformly to the edge members and the intermediate members. With this setting all excluded members are ignored, which means applied internally for load distribution. After calculating the cell area, the load is distributed to the allowed members of the cell.

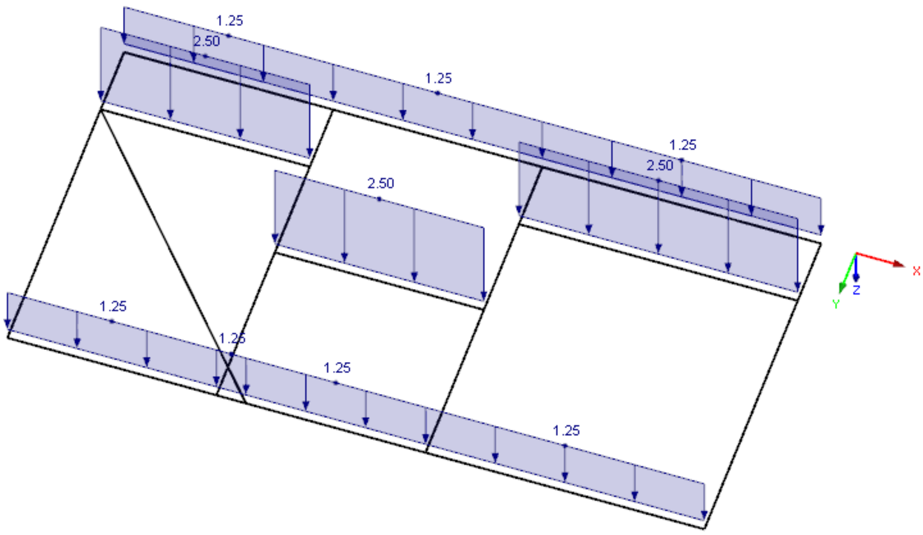


Figure 11.180: Result for *Do not use any member*

- *Use all members*:
  All unloaded members are excluded for the load generation. There is still a small problem in the load distribution because of the large, generated cell 2.

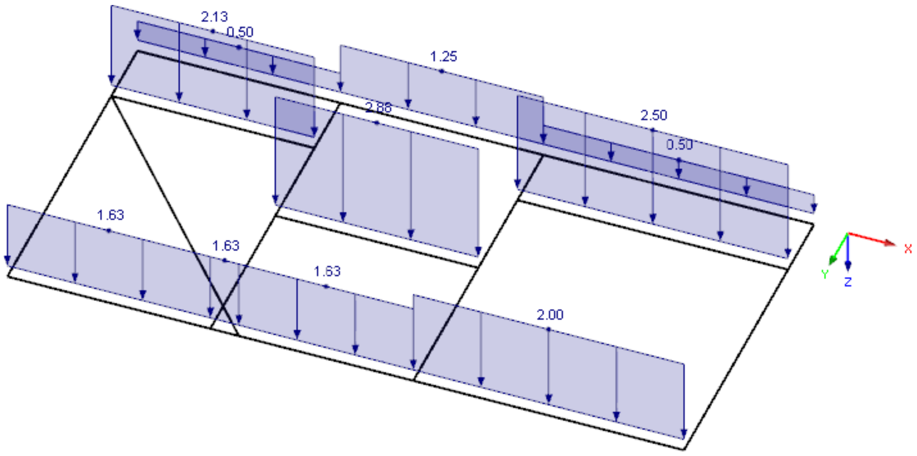


Figure 11.181: Result for *Use all members*

- *Use members parallel to boundary*:
  In this way, it is possible to exclude members lying in an inclined position. If the limit angle between members φ_b is restricted to 40.55° in the *Settings* dialog box (see Figure 11.178, page 521), the load will be generated as expected.

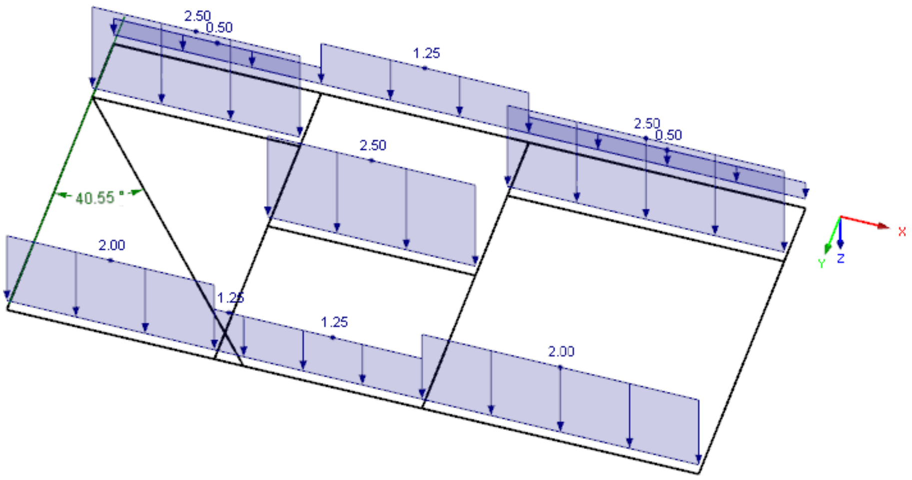


Figure 11.182: Result for *Use members parallel to boundary*

Modify generated loads subsequently

After confirming a generator dialog box you find the generated loads transferred to load table 3.14. The additional entry *Generated Loads* appears in the *Data* navigator (see Figure 6.52, page 250). The generator parameters won't be lost because the original dialog boxes remain accessible as input objects for changes. To open the initial dialog box again, double-click one of the entries in the navigator. You can also double-click a generated load in the work window. The original dialog box appears where you can adjust the parameters.

But if you want to treat the generated loads as isolated load objects, you have to release the loads from the overall concept and split them into their components. Access to this function is available in the load context menu that you open by right-clicking a generated load. Select *Disconnect Generated Load* in the context menu to create the individual loads (see figure below).

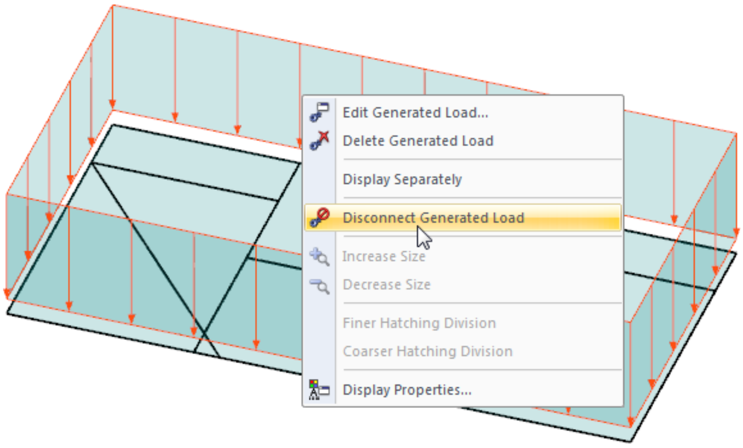


Figure 11.183: Context menu of generated load

You can also use the context menu of the generated load in the *Data* navigator.

11.8.2 Member/Linear Loads from Area Loads

11.8.2.1 Member Loads From Area Load via Plane

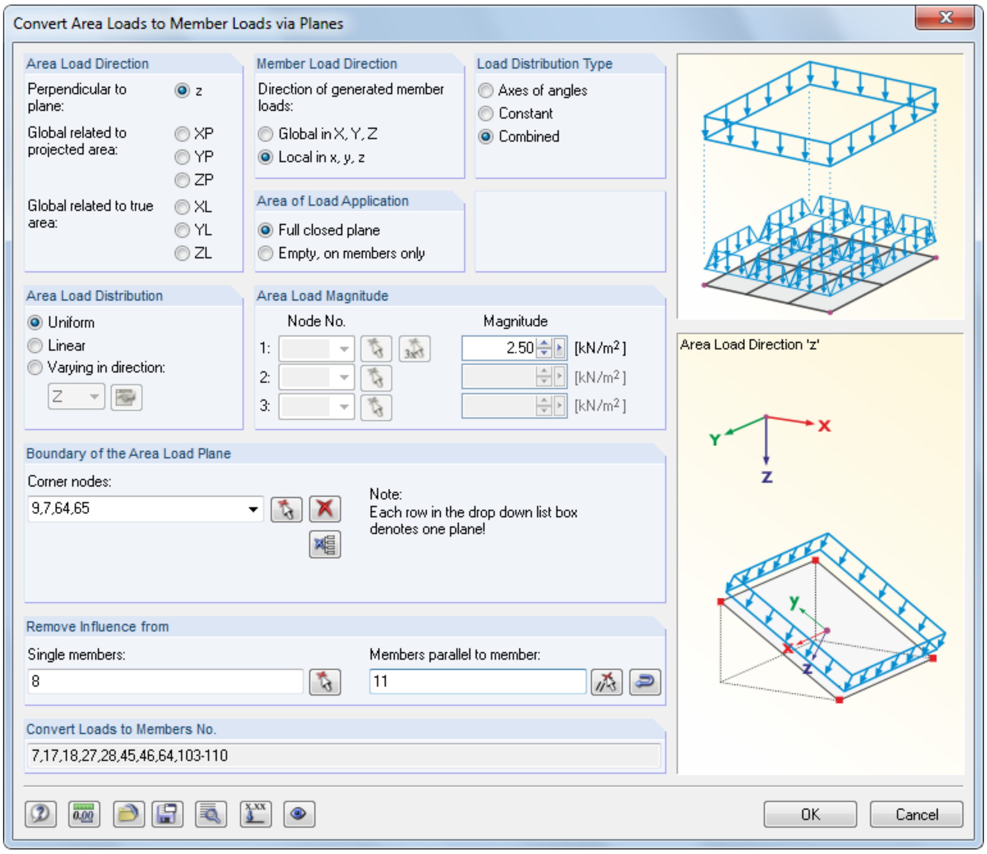


Figure 11.184: Dialog box *Convert Area Loads to Member Loads via Planes*

Area load direction

Decide whether the load acts perpendicular to the plane or globally related to the real or to the projected area. The dialog graphic in the right corner illustrates the selected load direction.

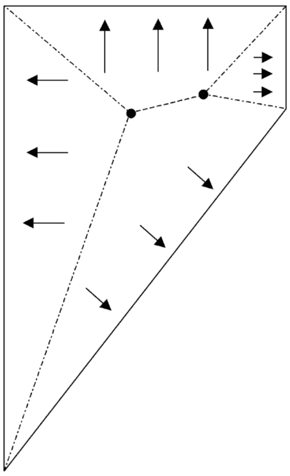
Member load direction

The generated member loads can be set to be global or local loads (see chapter 6.2, page 220). The difference is especially significant for non-linear calculations.

Area of load application

You have two selection options. Select *Full closed plane* when a surface exists in the load plane between the members (for example wall or roof surface) which is not represented in the RFEM model. In this case, RFEM converts the area load that acts on the entire plane to the members. But if the construction consists only of members (for example lattice tower), select the option *Empty, on members only*. Then, RFEM charges only the effective or projected area that is provided by the member cross-sections as "load application surface". The load will be applied in consideration of the member orientation.

Load distribution type



You decide how the area load components will be assigned to the members. Select *Axes of angles* for polygons that do not have a reflex angle. The intersection points of the bisecting lines will be connected in such a way that application areas are created as shown in the picture on the left. In this way, it is possible to distribute the area load clearly to the members without ambiguity.

The angle axes method is not applicable for planes with reflex angles or for polygons. In such cases, set the load distribution type to *Constant*. In addition to the angle bisectors, RFEM will also determine the centroid of the plane. If the intersection points of the bisecting lines lie <u>in front of</u> the centroid, triangular application areas will be generated. If they lie <u>behind</u> the centroid, a line that is parallel to the member will be drawn through the centroid, forming an application area with both angle bisectors.

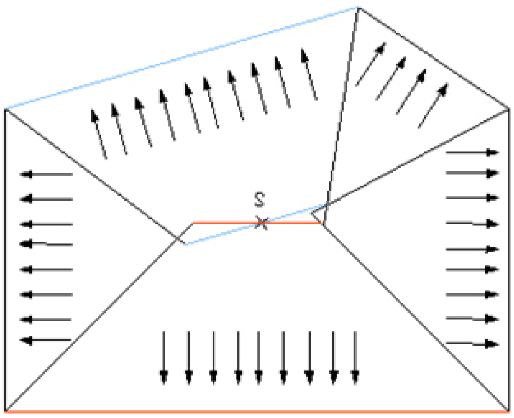


Figure 11.185: Load distribution type *Constant*

Using this method results in the fact that areas are either not taken into account or applied twice. The missing or remaining amount will be multiplied by a constant so that the sum of the area and member loads is equal.

The *Combined* option determines the application areas of triangles, quadrangles and polygons according to the angle axes method, where possible. If the method cannot be used, RFEM switches automatically to the constant load distribution. Therefore, the combined method is set by default; RFEM will select the appropriate method automatically.

Area load distribution

The load can act on the area as *Uniform* or *Linear* variable load. It is also possible to define area loads acting freely *Varying in direction* of a global axis (for example height-dependent wind load). Use the [Edit] button to open a dialog box where you can define the load parameters as a function of the height levels.

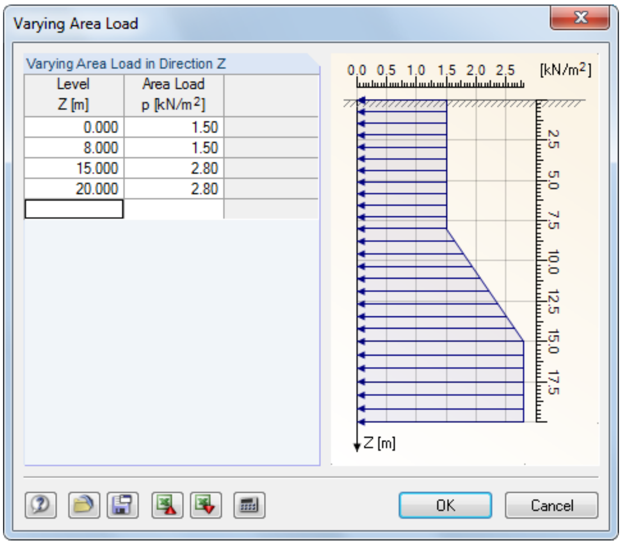


Figure 11.186: Dialog box *Varying Area Load*

In the left table column, enter the global ordinates of the *Level*. Assign the respective values of the *Area Load* to the right. The graphic illustrates the current state of input.

When freely variable loads are set, you have to select the correction of the distribution according to the moment equilibrium in the *Settings* dialog box (see Figure 11.178, page 521). Otherwise, constant member loads will be generated.

Area load magnitude

When the load acts uniformly on the area, enter the load value into the enabled input field. For linearly variable loads specify three node numbers with the respective loads. You can also use the [↖] function to select the nodes graphically in the work window.

Boundary of area load plane

The boundary is set by the corner nodes of the plane. Use the [↖] function and click the relevant nodes one after the other in the work window. The plane will be marked in the selection color. The completely entered plane will appear in cyan color. At least three nodes are required for defining a plane. The area does not need to be enclosed by lines or members on all sides.

It is possible to define different planes which appear then in the list *Corner nodes*.

If the dialog box is opened repeatedly, the last entered planes may be preset in the *Corner nodes* list. To avoid assigning double loads unintentionally to these planes, it is recommended to empty the list in this case with the button [Delete all area load planes].

Remove influence from

In the dialog section *Remove Influence from*, you can exclude members from the load application (for example purlins or bracings). The selection is carried out member by member or by entering a member template that is parallel to the load-free members. Again, it is recommended to use the [↖] function for graphical selection.

Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.178, page 521). Then, you can adjust the tolerance for the integration of nodes in the load plane or correct the generated loads.

Use the button [Assign load correction factors] to scale the loads for particular members. In this way, you can consider for example the effects of continuity of a roof sheathing on the edge rafters in order to generate reduced member loads there. The following dialog box opens.

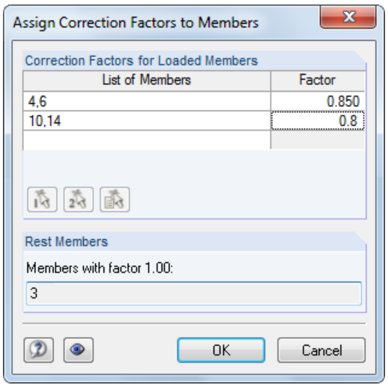


Figure 11.187: Dialog box *Assign Correction Factors to Members*

Use the [↖] buttons to select the members in the work window. Then, you can scale them with a *Factor*.

Click [OK] to start the generation of member loads. An overview appears with information about the cells and loads.

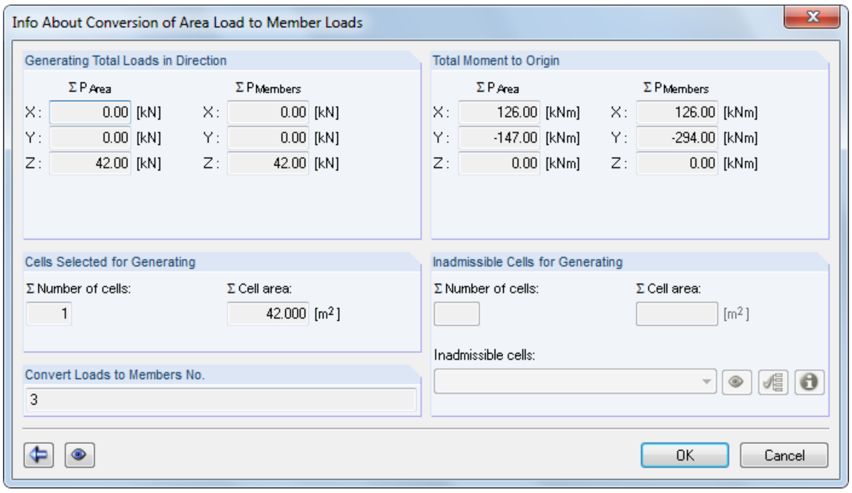


Figure 11.188: Dialog box *Info About Conversion of Area Load to Member Loads*

If inadmissible cells are listed, RFEM was not able to assign the loads without ambiguity. Use the eye button [Show Current Inadmissible Cell] to highlight the cell in the graphic. To show a list of reasons why the cells are invalid, click the [Info] button. Often, removed borders of the cell (that means edge members excluded from load application) or crossing members that are not connected are responsible for problems occurring when converting loads.

In the dialog section *Total Moment to Origin*, the determined member loads are compared with the applied area loads. In case of differences, you can use the [Back] button to access the initial dialog box where you can change parameters. Specifications are to be adjusted in the dialog box *Settings for Load Generation* (see Figure 11.178, page 521) that you can access by using the [Settings] button.

The buttons at the bottom left in the info window have the following functions:

Button	**Description**
	The dialog box *Convert Area Loads to Member Loads* opens again for you to adjust the generation parameters.
	RFEM shows you the work window where you can change the view (view mode). To return to the *Info* window, right-click into the work window or use the [Esc] key.

Table 11.15: Buttons in info window for converted member loads

11.8.2.2 Member Loads From Area Load via Cells

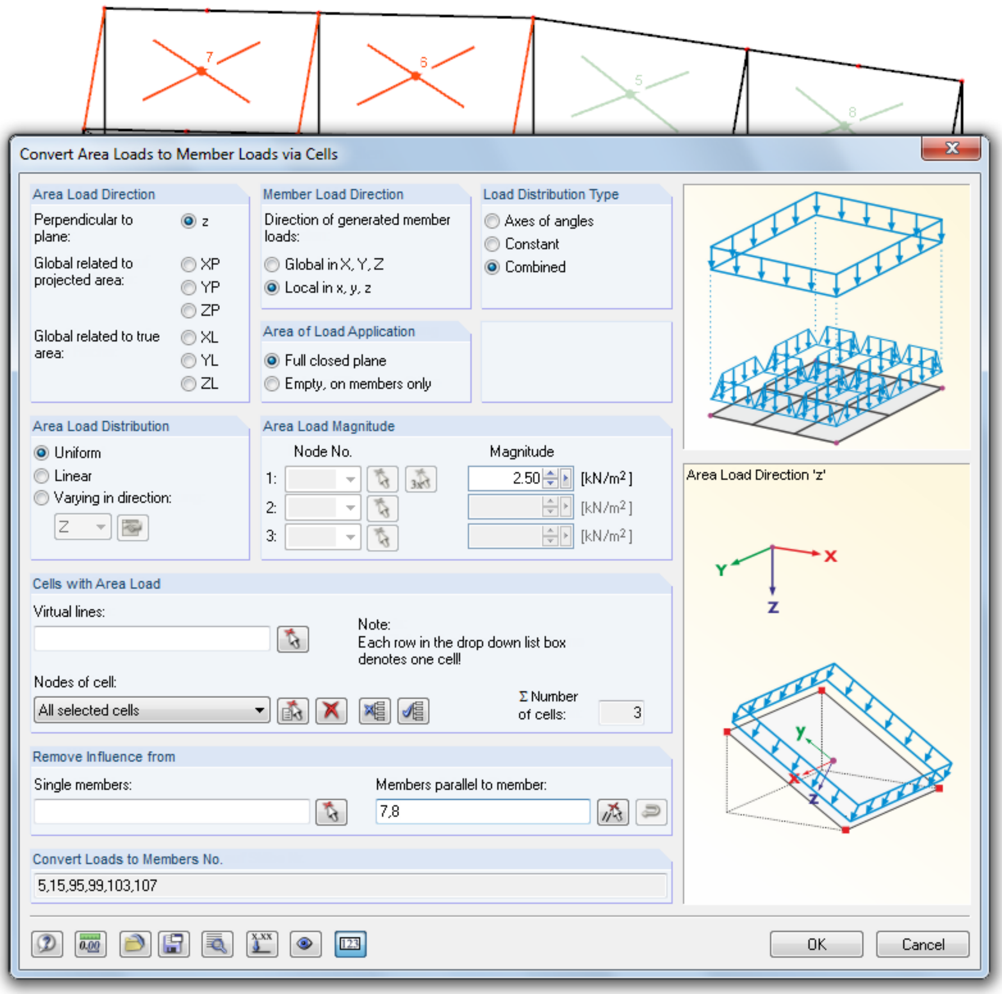


Figure 11.189: Dialog box *Convert Area Loads to Member Loads via Cells*

This dialog box is similar to the dialog box *Convert Area Loads to Member Loads via Planes* described on page 524. RFEM already checks the existence of cells in the model when opening the dialog box. Available cells are represented by cell crossings. Cells are zones defined by three or more corner nodes, enclosed by members on all sides and placed in one plane.

The load generator via cells cannot be used for wind loads, for example on a hall wall with columns: RFEM does not recognize any cells because members are missing between the footings. In such a case, you can create *Virtual lines* by clicking the start and end node using the [↖] function. In this way, cells are closed artificially and can be recognized by the generator.

The *Nodes of cell* can be selected with [↖] one after the other in the graphic. After the generation an overview with information about cells and loads appears.

Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.178, page 521). Then, you can adjust the tolerance for the integration of nodes in the load plane or correct the generated loads.

11.8.2.3 Line Loads from Area Loads on Openings

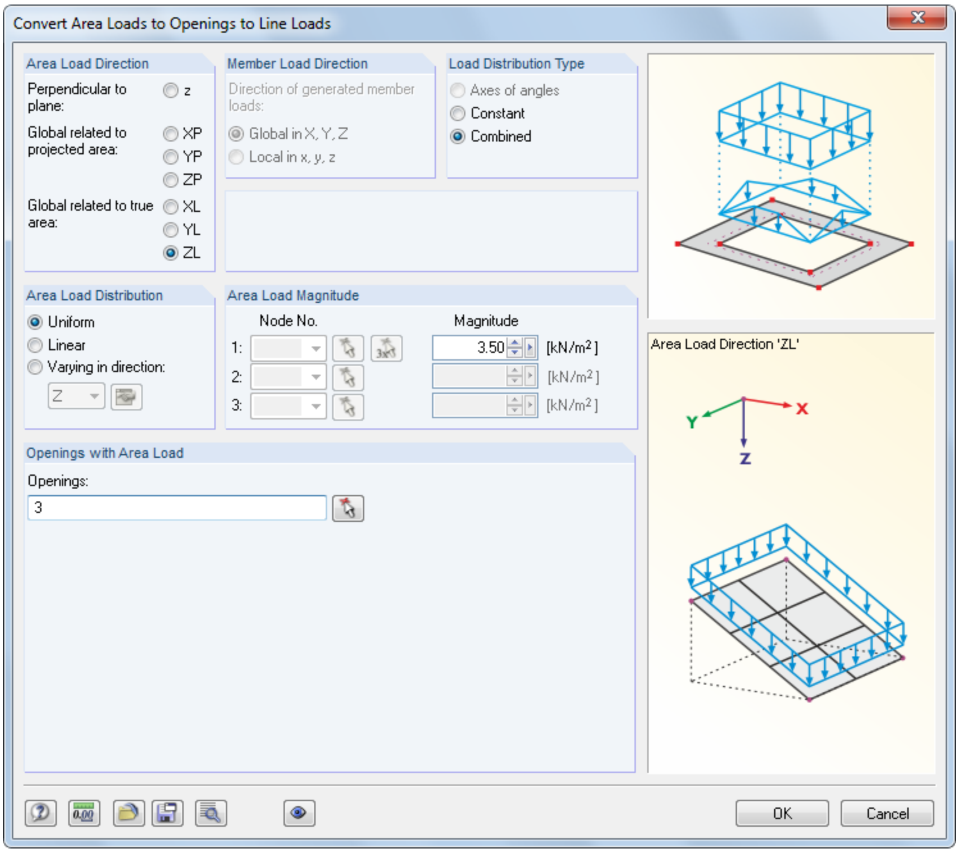


Figure 11.190: Dialog box *Convert Area Loads to Openings to Line Loads*

The dialog box resembles the dialog box *Convert Area Loads to Member Loads via Planes* described on page 524. In the upper dialog sections, you can define the different load parameters.

In the input field of the dialog section *Openings with Area Load*, enter the numbers of openings. You can also select them with the [↖] function in the work window.

Click [OK]. An overview with information about the generated loads appears. Then, click [OK] to create the line loads on the edges of the opening(s).

11.8.3 Other Loads

11.8.3.1 Member Loads from Free Line Load

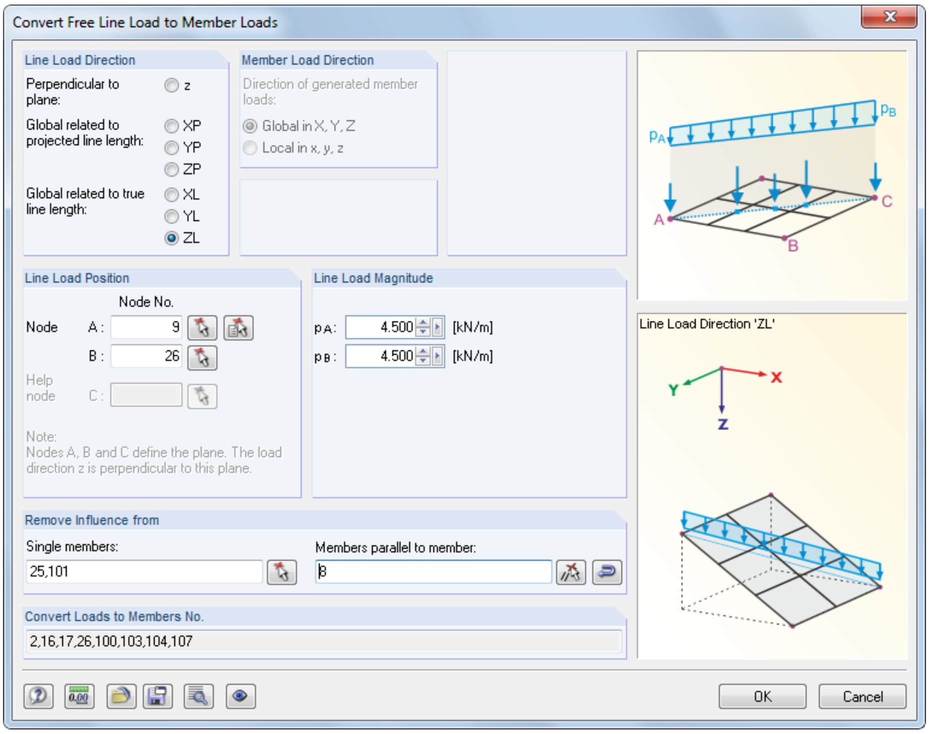


Figure 11.191: Dialog box *Convert Free Line Load to Member Loads*

Use the generator to define free line loads for pure member models like girder grillages to prorate the loads to members.

The correct load assignment requires specifications for *Line Load Direction* and *Member Load Direction*, where applicable. These dialog sections as well as the option for *Remove Influence from* is described for the function "Member Loads From Area Load via Plane" on page 524.

The *Line Load Magnitudes* can be defined constantly or linearly. The *Line Load Position* can be defined graphically with the [↖] function by clicking the start and end node. If the line load is directed perpendicular to the plane, enter the help node C additionally.

Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.178, page 521).

11.8.3.2 Member Loads from Coating

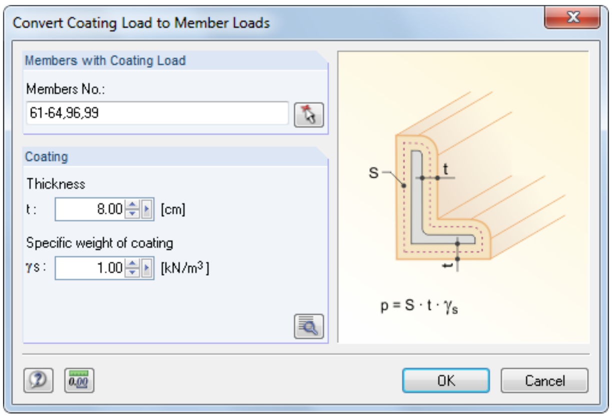


Figure 11.192: Dialog box *Convert Coating Load to Member Loads*

Members with Coating Load can be entered directly or determined graphically with [↖]. The *Coating* is to be defined by the thickness and the specific weight.

Use the [Info] button shown on the left to check the coating areas A_S of the selected member cross-sections to be applied for determining the ice load. The areas are related to the center lines of the ice load as shown in the dialog graphic (Figure 11.192). Thus, loads will be determined correctly even for small cross-sections with many edges.

11.8.3.3 Loads from Accelerated Movements

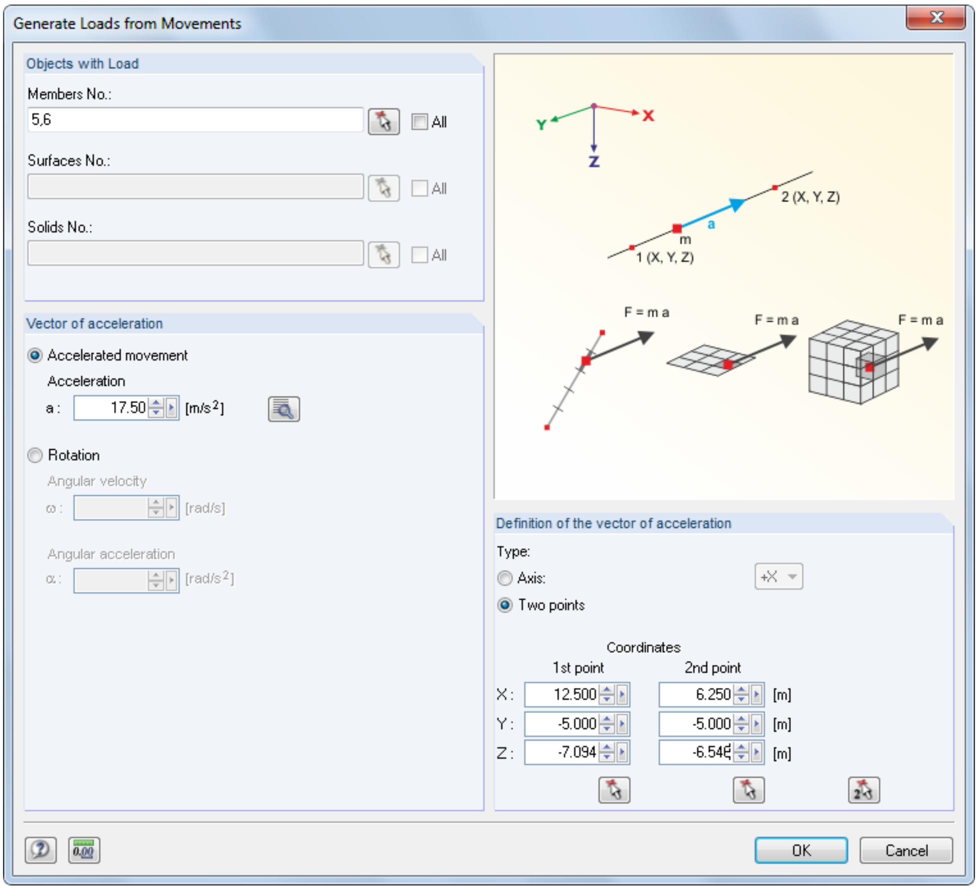


Figure 11.193: Dialog box *Generate Loads from Movements*

The generator creates loads as a result of an acceleration or rotation acting on particular objects of the model. The mass is determined from the self-weight.

In the dialog section *Objects with Load*, enter the numbers of relevant members, surfaces or solids. You can select them also graphically with the [↖] function.

Define the *Vector of acceleration* as acceleration or rotation (angular velocity ω, angular acceleration α). Use the [Open] button shown on the left to determine the acceleration from the velocities that are available on two points.

In the dialog section *Definition of the vector of acceleration*, you decide if the vector is related to a global axis or defined by two points. The vector can be defined graphically by using the [↖] buttons.

Click [OK] to create the loads for the currently set load case.

11.8.4 Snow Loads

11.8.4.1 Flat/Monopitch Roof

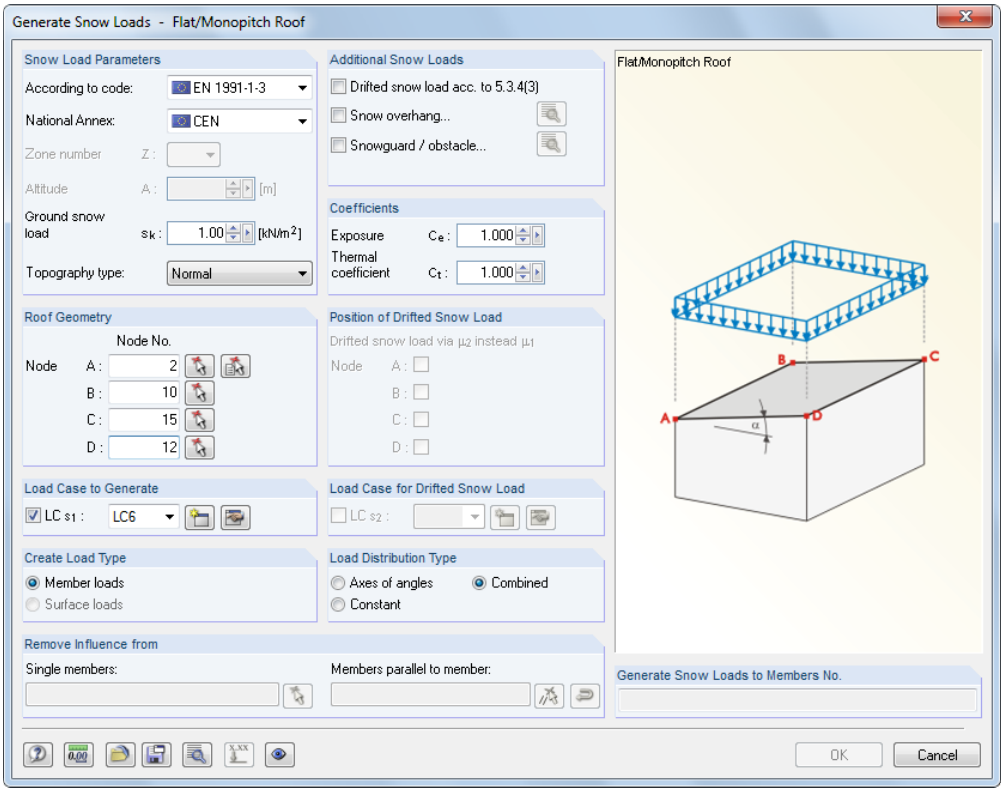


Figure 11.194: Dialog box *Generate Snow Loads - Flat/Monopitch Roof*

Flat and monopitch roofs are managed together in one dialog box. The shape coefficients for flat roofs or roofs with one-sided inclination are taken into account according to EN 1991-1-3 and DIN 1055-5.

First, define the standard and, if necessary, the national annex in the dialog section *Snow Load Parameters*. The setting controls the input fields enabled for access.

Use the [Info] button to open a map where the snow load zone *Z* can be selected graphically. Based on your specifications, RFEM determines the characteristic value of the snow load s_k on the ground, taking account of the altitude *A* (see level).

Use the three check boxes in the dialog section *Additional Snow Loads* to decide if other snow loads are considered:

- Snow accumulations because of snow drift
- Snow overhang on eaves
- Snow loads on snow guards

Use the [Edit] buttons to define the parameters for snow overhang and snow guard.

If required, you can adjust the *Exposure* coefficient C_e (EN 1991-1-3, table 5.1) as well as the *Thermal coefficient* C_t (EN 1991-1-3, clause 5.2 (8)) in the dialog section *Coefficients*.

Define the *Roof Geometry* by means of the roof corner nodes A to D in accordance with the dialog graphic. You can also use the [↖] function to determine them graphically in the work window. The plane will be marked in the selection color. At least three nodes are required for defining a plane. The area does not need to be enclosed by lines or members on all sides.

The *Position of Drifted Snow Load* can be defined by the edge nodes of the roof area.

In the dialog sections *Load Case to Generate* and *Load Case for Drifted Snow Load*, you specify the load case numbers for the load generation. Snow load cases can be created with the [New] button. If surfaces are available in the model, you can use the options in the dialog section *Create Load Type* to decide whether member or surface loads will be generated.

The dialog sections *Load Distribution Type* and *Remove Influence from* are described for the generator function "Member Loads From Area Load via Plane" on page 525.

Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.178, page 521).

Use the button [Assign load correction factors] to scale the loads for particular members. The specifications can be entered in a separate dialog box (see Figure 11.187, page 527).

After confirming the generator dialog box with [OK], RFEM shows you the results of the load generation for all load cases in an overview. Thus, the acting area loads can be compared with the converted loads. Before the loads are transferred to RFEM, you can click the [Back] button to access the initial dialog box where you can change the parameters of the loads.

11.8.4.2 Duopitch Roof

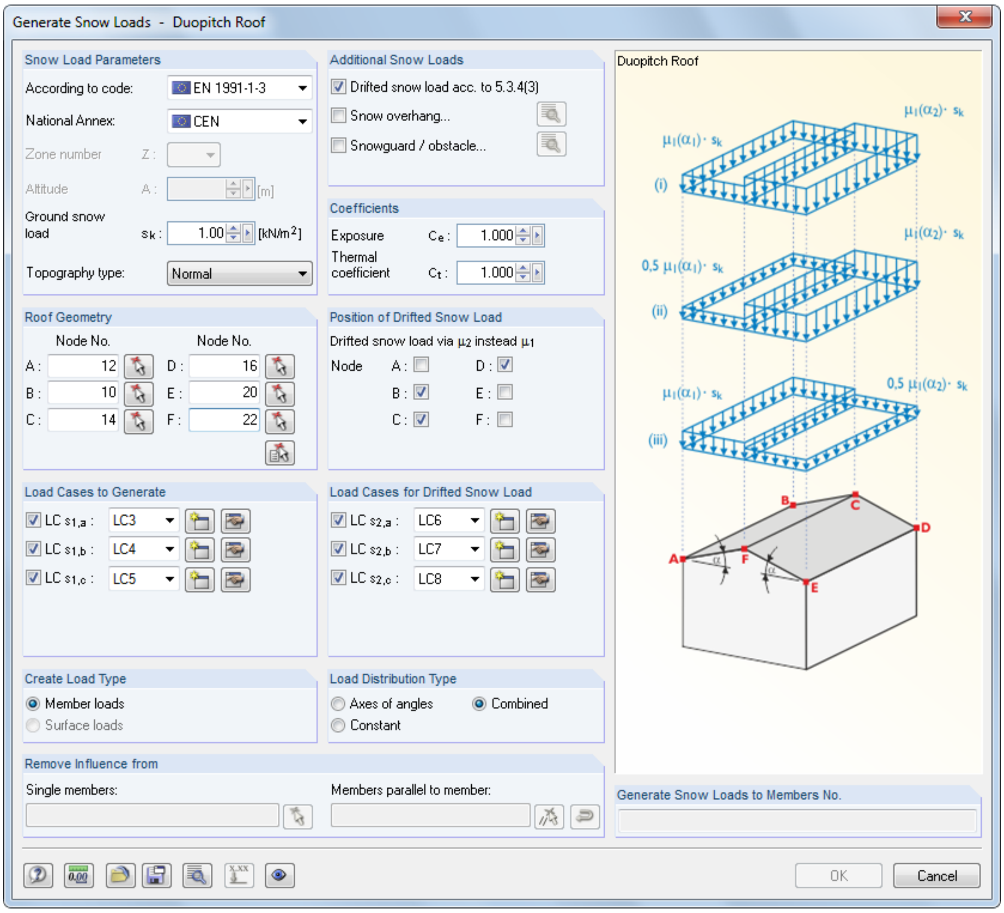


Figure 11.195: Dialog box *Generate Snow Loads - Duopitch Roof*

First, define the standard and, if necessary, the national annex in the dialog section *Snow Load Parameters*. The setting controls the input fields enabled for access.

Specify the parameters as described in the previous chapter. The *Roof Geometry* of a duopitch roof is defined by the roof's corner nodes A to F in accordance with the dialog graphic. You can also use the [↖] function to determine the nodes graphically in the work window.

In the dialog sections *Load Cases to Generate* and *Load Cases for Drifted Snow Load*, you specify the load case numbers for the load generation. Alternative load cases will be created when additional snow loads (for example DIN 1055-5, figure 4) or shape coefficients (for example EN 1991-1-3, figure 5.3) are taken into account. The relevant snow load cases can be created with the [New] button.

11.8.5 Wind Loads

11.8.5.1 Vertical Walls

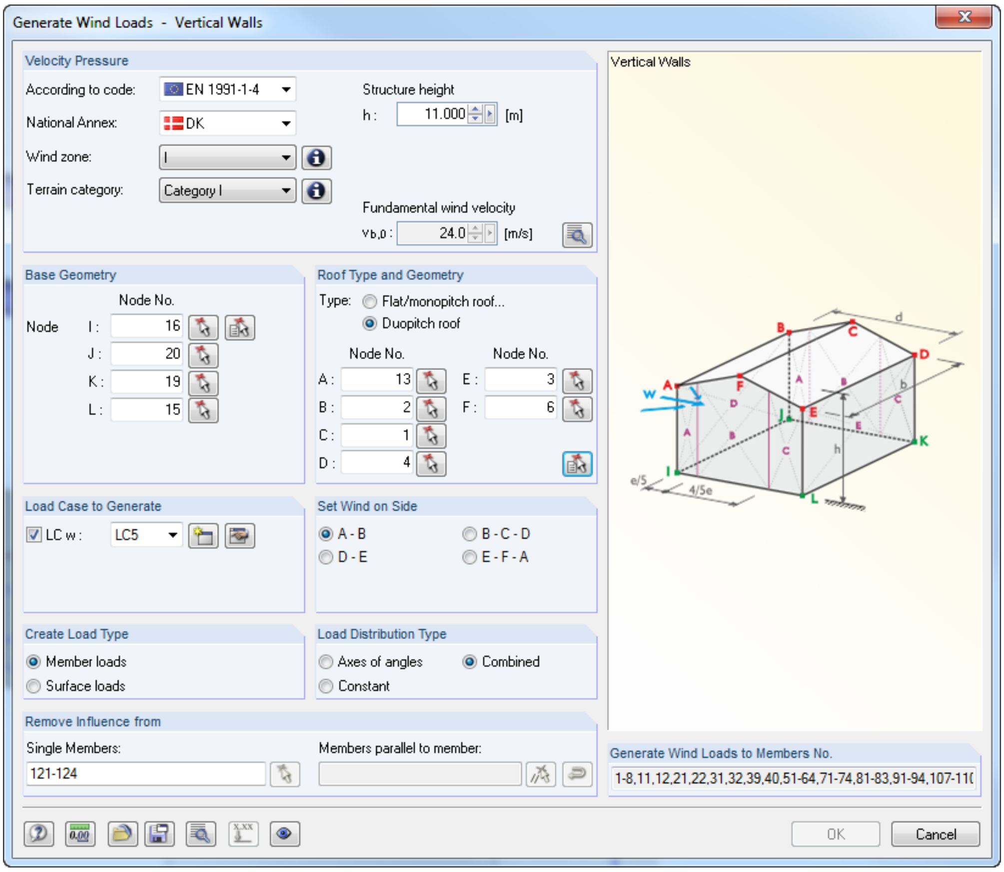


Figure 11.196: Dialog box *Generate Wind Loads - Vertical Walls* (roof geometry: *Duopitch roof*)

First, define the standard and, where applicable, the national annex in the dialog section *Velocity Pressure*. The setting controls the input fields enabled for access.

Wind zone and terrain category can be selected graphically in a map that you open with the [Info] button. The structure height *h* is not taken over automatically from the model but must be specified. Based on your settings, RFEM determines the basic value of the fundamental wind velocity $v_{b,0}$.

Click the [Edit] button shown on the left to access more coefficients used to determine wind loads.

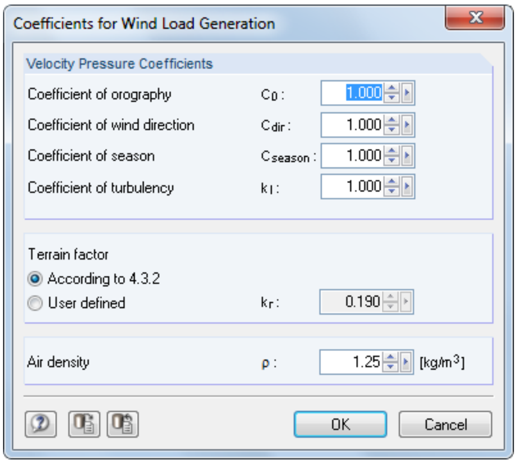


Figure 11.197: Dialog box *Coefficients for Wind Load Generation*

The walls are determined by the *Base Geometry* (nodes *I* to *L* for base area, bottom) and the *Roof Type and Geometry* (nodes *A* to *D* or *F* for roof planes, up). In case of roof overhangs, specify the upper wall nodes, not the roof nodes. As shown in the dialog graphic, wind loads can be generated for building objects closed on all sides with a quadrilateral base area. Please note when entering geometry that the start nodes *I* and *A* must lie upon each other. Moreover, the direction of clicking nodes must be consistent when determining the base and roof area. You can also use the [↖] buttons to define base and roof geometry graphically.

In the dialog section *Load Case to Generate*, enter the load case number for the load generation. With the [New] button you can create a wind load case.

The wind direction is defined in the dialog section *Set Wind on Side*. The wind acts perpendicular to the specified line.

If surfaces are available in the model, you can use the options in the dialog section *Create Load Type* to decide whether member or surface loads will be generated.

The dialog sections *Load Distribution Type* and *Remove Influence from* are described for the generator function "Member Loads From Area Load via Plane" on page 525.

Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.178, page 521).

After confirming the generator dialog box with [OK], RFEM shows you the results of the load generation in an overview. Thus, the acting area loads can be compared with the converted loads. Before the loads are transferred to RFEM, you can click the [Back] button to access the initial dialog box where you can change the parameters of the loads.

11.8.5.2 Flat Roof

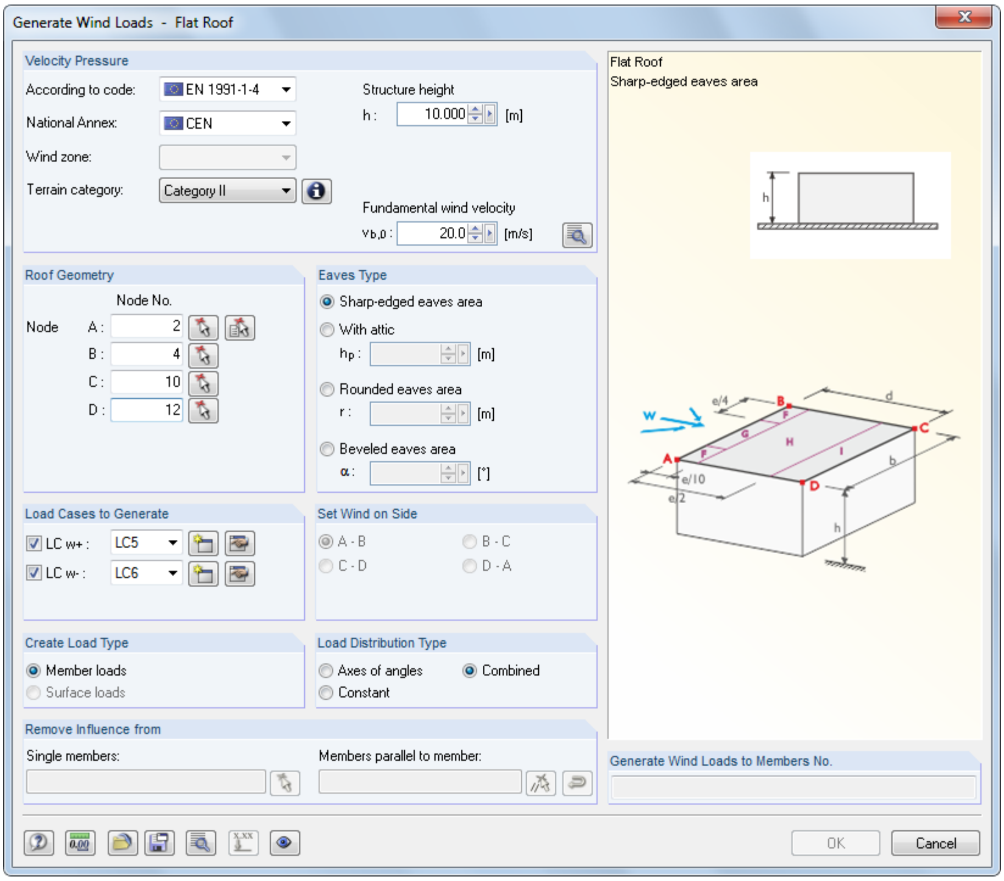


Figure 11.198: Dialog box *Generate Wind Loads - Flat Roof*

RFEM considers a roof to be a flat roof if the roof inclination is $\alpha < 5°$.

First, define the standard and, where applicable, the national annex in the dialog section *Velocity Pressure*. The setting controls the input fields enabled for access.

Specify the parameters as described in the previous chapter. The dialog section *Eaves Type* is linked to the interactive dialog graphics to the right illustrating the individual settings.

As described for example in EN 1991-1-4, table 7.2, several load cases must be taken into account for a flat roof. In the dialog section *Load Cases to Generate*, specify the load case numbers for the load generation. The compression loads are created in the load case *LC w+*. The suction loads are generated in *LC w-*. The relevant load cases can be created with the [New] button.

After confirming the generator dialog box with [OK], RFEM shows you the results of the load generation for all load cases in an overview (see Figure 11.201, page 539). The dialog tabs represent an important checking option because you can see for each load case the external pressure coefficient $c_{pe,10}$ and the external pressure w_e displayed by zones.

11.8.5.3 Monopitch Roof

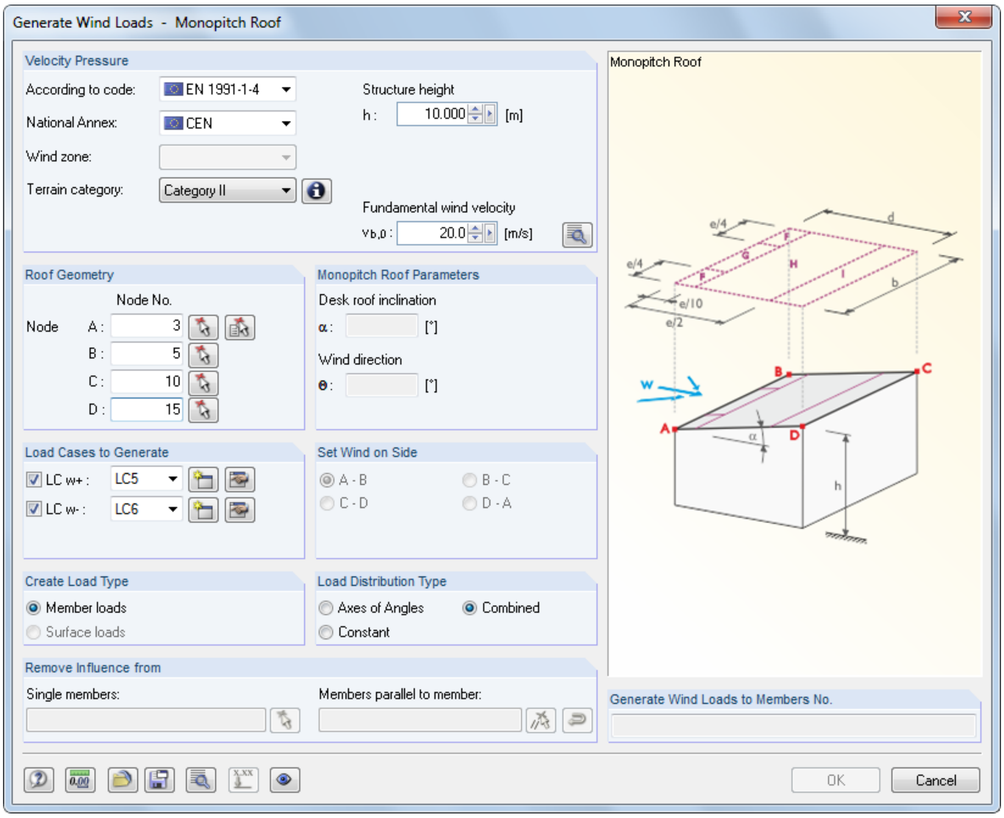


Figure 11.199: Dialog box *Generate Wind Loads - Monopitch Roof*

First, define the standard and, where applicable, the national annex in the dialog section *Velocity Pressure*. The setting controls the input fields enabled for access.

Specify the parameters as described in chapter 11.8.5.1. The *Monopitch Roof Parameters* are determined automatically from the roof geometry and the side where the wind blows.

As described for example in EN 1991-1-4, table 7.3a, several load cases must be taken into account for a monopitch roof. In the dialog section *Load Cases to Generate*, specify the load case numbers for the load generation. The compression loads are created in the load case *LC w+*. The suction loads are generated in *LC w-*. The relevant load cases can be created with the [New] button.

Use the button [Assign load correction factors] to scale the loads for particular members. In this way, you can consider for example the effects of continuity of a roof sheathing on the edge rafters in order to generate reduced member loads. The specifications can be entered in a separate dialog box (see Figure 11.187, page 527).

11.8.5.4 Duopitch/Troughed Roof

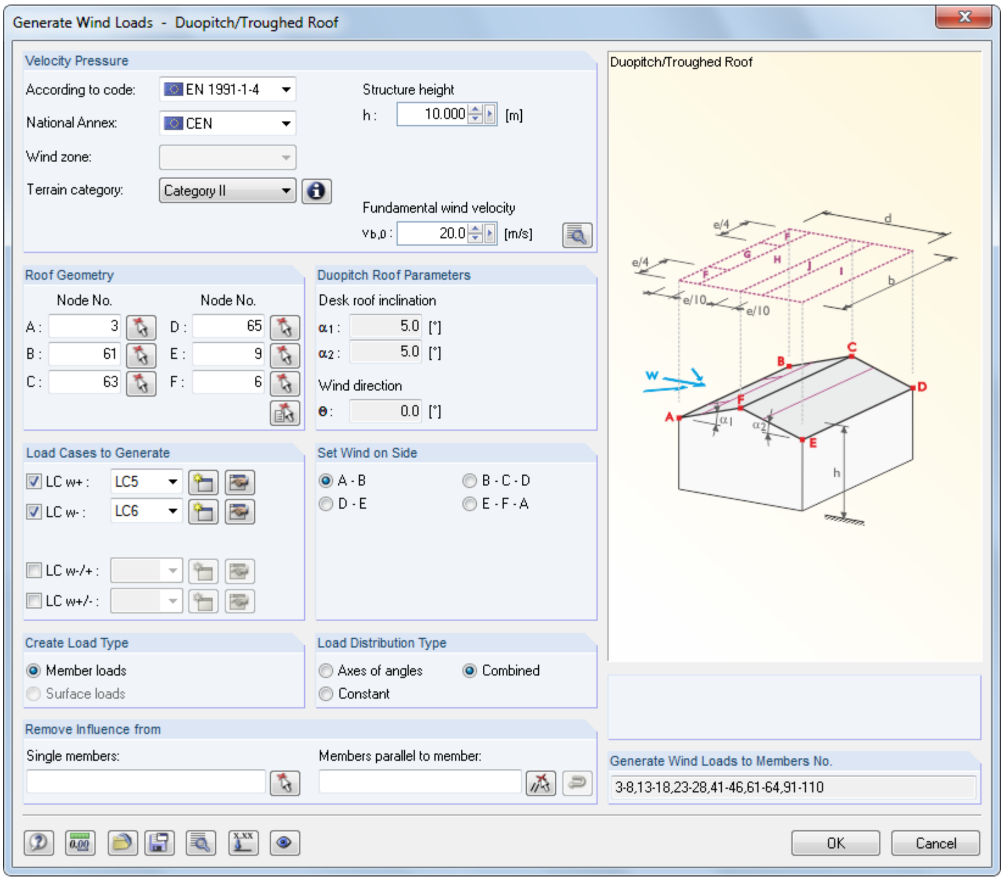
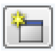


Figure 11.200: Dialog box *Generate Wind Loads - Duopitch/Troughed Roof*

First, define the standard and, where applicable, the national annex in the dialog section *Velocity Pressure*. The setting controls the input fields enabled for access.

Specify the parameters as described in chapter 11.8.5.1 on page 534. The *Duopitch Roof Parameters* are determined automatically from the roof geometry and the side where the wind blows.



As described for example in EN 1991-1-4, table 7.4a, several load cases must be taken into account for a duopitch roof. In the dialog section *Load Cases to Generate*, specify the load case numbers for the load generation. The compression loads are created in the load case *LC w+*. The suction loads are generated in *LC w-*. Combinations (compression on one side of the roof and suction on other side) are defined as *LC w-/+* and *LC w+/-*. The relevant load cases can be created with the [New] button.

After confirming the generator dialog box with [OK], RFEM shows you the results of the load generation for all load cases in an overview. The dialog tabs represent an important checking option because you can see for each load case the external pressure coefficient $c_{pe,10}$ and the external pressure w_e displayed by zones.

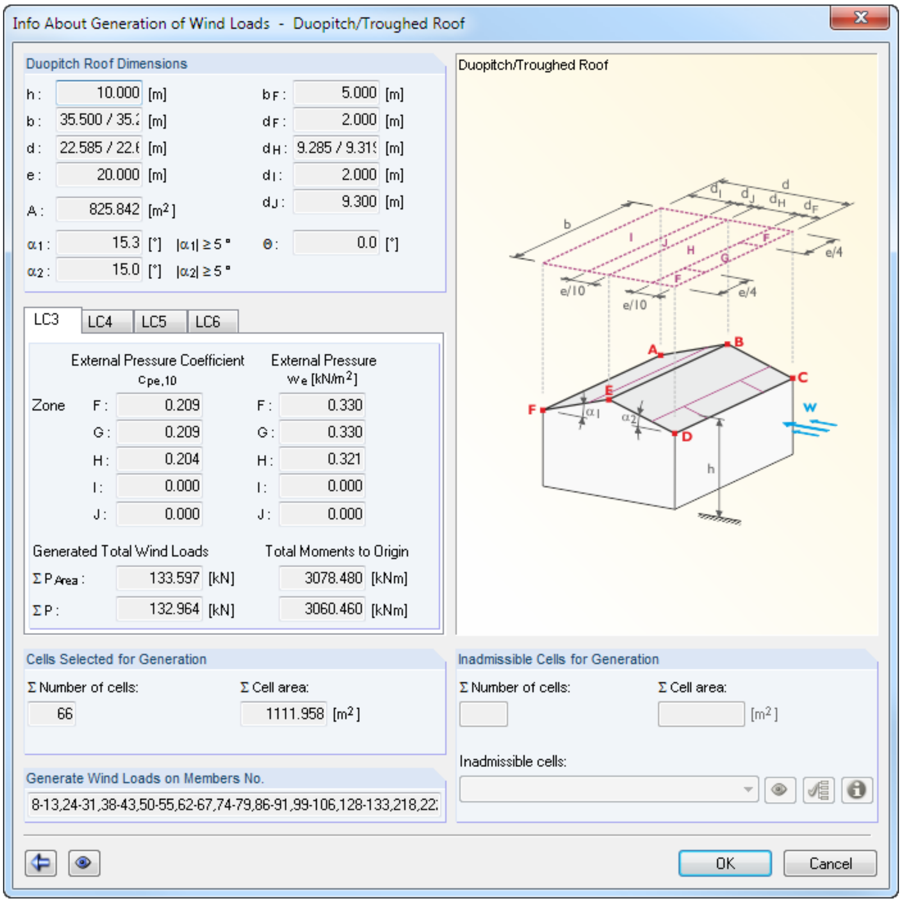


Figure 11.201: Dialog box *Info About Generation of Wind Loads - Duopitch/Troughed Roof*

Before the loads are transferred to RFEM, you can click the [Back] button to access the initial dialog box where you can change the parameters of the loads.

11.8.5.5 Vertical Walls with Roof

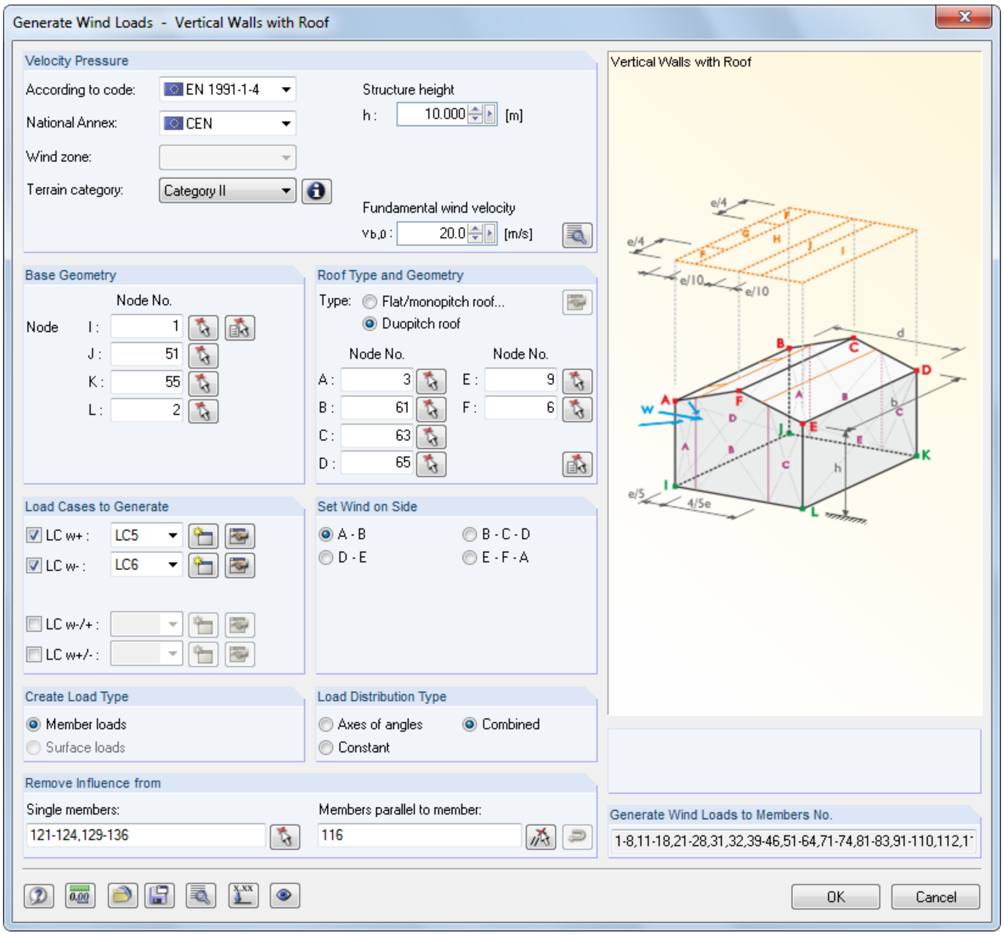
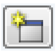


Figure 11.202: Dialog box *Generate Wind Loads - Vertical Walls with Roof* (roof geometry: *Duopitch roof*)

First, define the standard and, where applicable, the national annex in the dialog section *Velocity Pressure*. The setting controls the input fields enabled for access.

Specify the parameters as described in chapter 11.8.5.1 on page 534.



As described for example in EN 1991-1-4, table 7.4a, several load cases must be taken into account for a duopitch roof. In the dialog section *Load Cases to Generate*, specify the load case numbers for the load generation. The compression loads are created in the load case *LC w+*. The suction loads are generated in *LC w-*. Combinations (compression on one side of the roof and suction on other side) are defined as *LC w-/+* and *LC w+/-*. The relevant load cases can be created with the [New] button.

Use the button [Assign load correction factors] to scale the loads for particular members. The specifications can be entered in a separate dialog box (see Figure 11.187, page 527).

After confirming the generator dialog box with [OK], RFEM shows you the results of the load generation for all load cases in an overview (see Figure 11.201, page 539). The dialog tabs represent an important checking option because you can see for each load case the external pressure coefficient $c_{pe,10}$ and the external pressure w_e displayed by zones.

12. File Management

This chapter explains how data is organized in the Project Manager and how recurring model components are managed in blocks. In addition, the chapter describes the data import and export with the integrated interfaces for exchanging data with other programs.

12.1 Project Manager

In structural analysis, a project is often subdivided into several models. The *Project Manager* helps you to organize data of your Dlubal applications. You can also use it for managing RFEM models within the network (see chapter 12.3, page 561).

The Project Manager can be left open as a stand-alone application when working in RFEM.

To open the Project Manager, select **Project Manager** on the **File** menu, or use the toolbar button shown on the left.

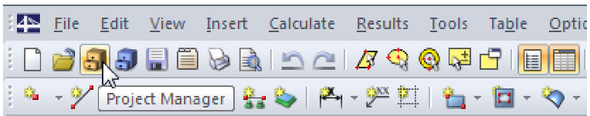


Figure 12.1: Button *Project Manager* in the toolbar

It is also possible to access the Project Manager in the model's *General Data* dialog box.

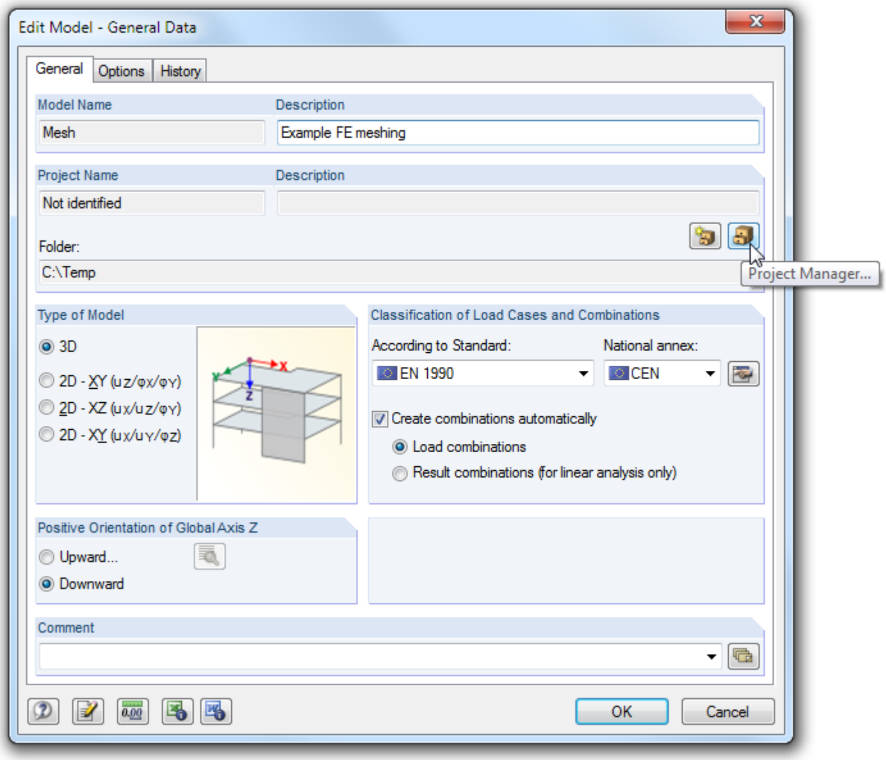


Figure 12.2: Button *Project Manager* in the dialog box *General Data*

When you open the Project Manager, the following multi-part window appears. It has its own menu and toolbar.

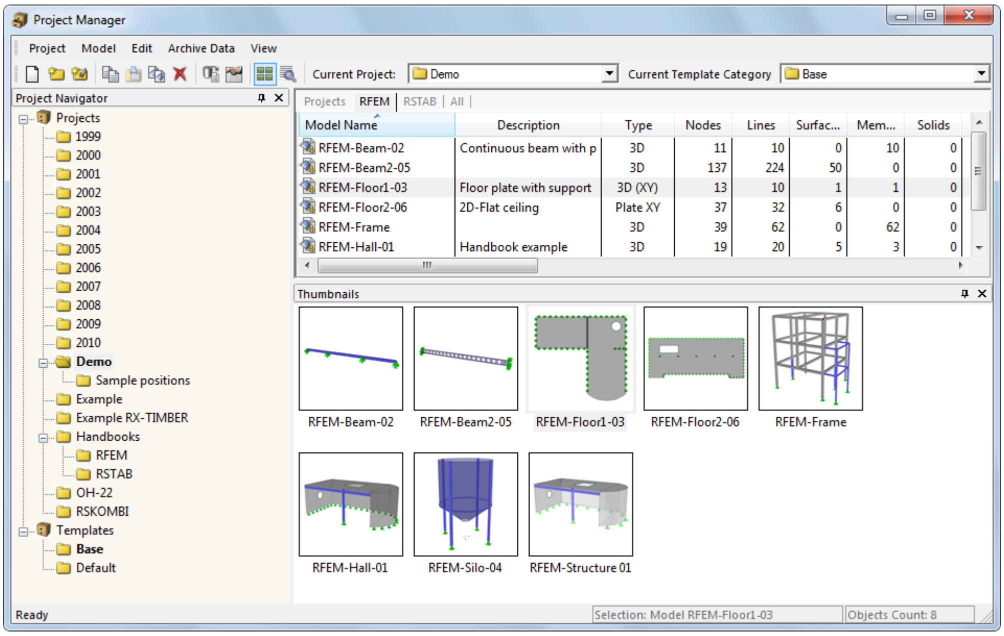


Figure 12.3: Project Manager

Project navigator

A navigator listing all projects in a tree structure is displayed on the left. The current project is set bold. To select another project, double-click the relevant entry or use the list *Current Project* in the toolbar. The table to the right of the navigator lists the models contained in the selected project.

Table of models

The models are arranged in several tabs, sorted by Dlubal applications. The *RFEM* tab lists all RFEM models contained in the selected project. The *Model Name* and *Description* as well as significant model and file information including the name of the user who created and edited the model are displayed respectively.

To adjust the column display, select **Manage Register Columns** on the **View** menu of the Project Manager, or use the toolbar button shown on the left (see page 551).

Details

This part of the window shows all information available for the model that is selected in the window section above.

Preview

The selected model is displayed in a preview. The size of the preview window can be adjusted by moving the upper edge of the window.

Thumbnails

The bottom area of the Project Manager offers you a graphical overview about the models contained in the selected project. The thumbnail images are interactive with the table above.

Use the pins to minimize particular window parts. They will be docked as tabs in the footer.

12.1.1 Project Management

Create a new project

To create a new project,

- select **New** on the **Project** menu of the Project Manager or
- click the button [New Project] in the toolbar shown on the left.

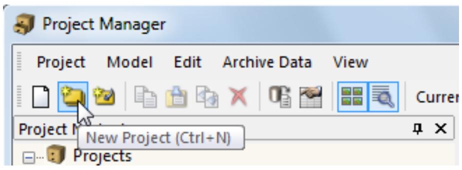


Figure 12.4: Button *New Project*

The dialog box *Create New Project* opens where you enter the *Name* of the new project. Then, select the *Folder* in which you want to save the models. Use the [Browse] button shown on the left to set the directory. You can also add a short project *Description*. It will be shown in the header of the printout report and has no further relevance.

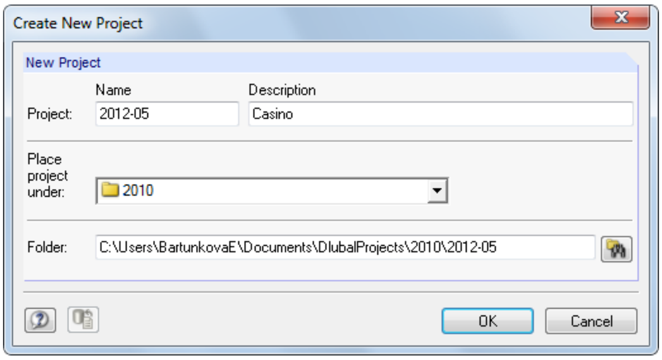
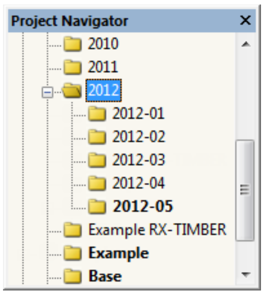


Figure 12.5: Dialog box *Create New Project*



It is also possible to create sub-projects in the Project Manager by selecting a project in the list *Place project under*. The new project will be displayed as sub-project in the navigator. If you do not want to use this setting, select the list entry *Projects* on the top of the list. Then, the project will appear as main entry in the navigator.

After clicking [OK], a new folder with the project name will be created on the local or network drive.

Connect existing folder

To integrate a folder containing already several RFEM models as a project,

- select **Connect Folder** on the **Project** menu of the Project Manager or
- use the button [Connect Folder] in the toolbar shown on the left.

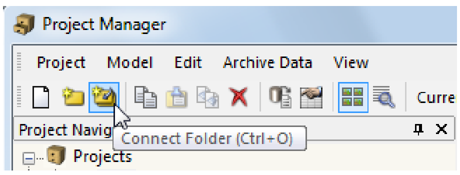


Figure 12.6: Button *Connect Folder*

It is irrelevant on which local or network drive the folder that you want to connect is located. It will be included into the file management and left at its location – similar to the creation of a shortcut on the desktop. The information is saved in the ASCII file **PRO.DLP** in the folder **Project Manager** (see chapter 12.1.4.3, page 553).

A dialog box opens that is similar to the dialog box shown in Figure 12.5. Enter the *Name* and *Description* of the project, and use the [Browse] button to set the directory for the relevant *Folder*. If a project is specified in the list *Place project under*, the connecting folder must be contained within the directory of this project. The folder will then be managed as a sub-project. But if you want the folder to appear as an independent project in the Project Manager, select *Projects* on the top of the list.

Tick the option *Connect folder including all subfolders* to connect all folders contained in the selected folder at once with the management of the Project Manager.

Disconnect a folder

To disconnect a folder integrated in the project management,

- select **Disconnect** on the **Project** menu of the Project Manager (project must have been previously selected),
- click the [Disconnect Project] button in the Manager toolbar shown on the left or
- use the project's context menu in the navigator.

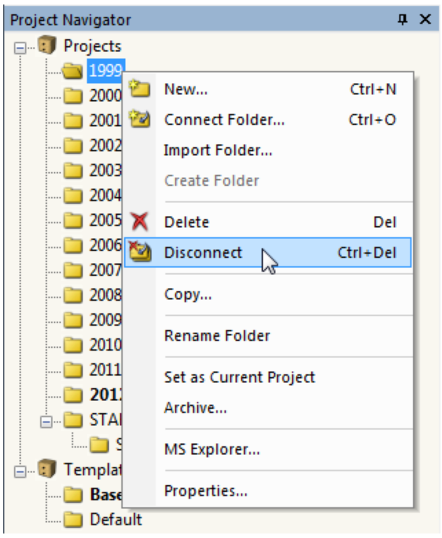


Figure 12.7: Context menu of a project

The project will be removed only from the internal management. The folder on the hard disk and its contents will be kept.

Delete a project

To delete a project,

- select **Delete** on the **Project** menu of the Project Manager (project must have been previously selected),
- click the [Delete] button in the Manager toolbar shown on the left or
- use the **Delete** entry in the project's context menu in the navigator (see figure above).

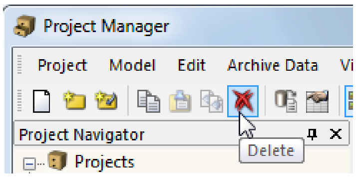


Figure 12.8: Button *Delete*

The folder including its contents will be completely deleted from the hard disk.

If the folder contains also files from other programs, only the files of Dlubal applications will be deleted. The folder itself will be preserved.

To undo the deletion of projects,

select **Restore from Dlubal Recycle Bin** on the **Edit** menu of the Project Manager.

The Dlubal recycle bin is described in chapter 12.1.4.2 on page 552.

In case files stored on a network drive are deleted, they are copied via network into the Dlubal recycle bin on the hard disk, which is different to the Windows standard where data is irrecoverable. In this way, you can restore files, deleted on network drives, from the relevant computer. If you don't want the files to be copied into the recycle bin, we recommend only disconnecting the project (see above). Then, you can delete the data from the network drive manually.

Copy a project

To copy a project,

- select **Copy** on the **Project** menu of the Project Manager (project must have been previously selected) or
- use the **Copy** entry in the project's context menu in the navigator (see Figure 12.7).

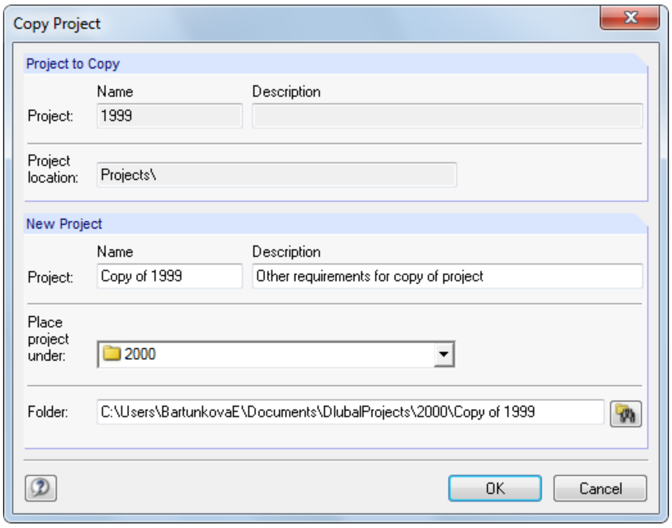


Figure 12.9: Dialog box *Copy Project*

Enter the *Name*, *Description* and the location of the new project in the Project Manager, and define the *Folder* that will be created by the copy function.

It is also possible to copy the project with the Windows-Explorer. Then, you can integrate the new folder as a connected folder into the management of the Project Manager (see Figure 12.6, page 543).

Rename a project / change description

To change the description of a project subsequently,

- select **Properties** on the **Project** menu of the Project Manager (project must have been previously selected) or
- use the **Properties** entry in the project's context menu in the navigator (see Figure 12.7).

The dialog box *Project Properties* opens where you can change the *Name* and *Description* of the project. The *Folder* of the project is also displayed.

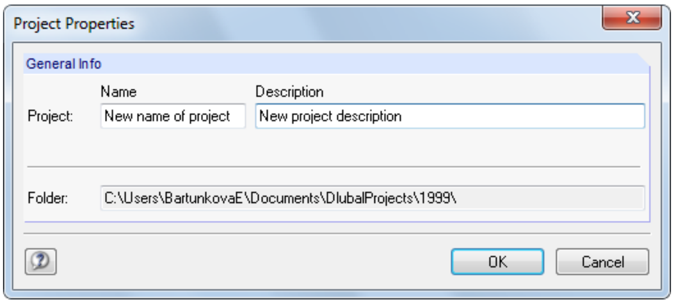


Figure 12.10: Dialog box *Project Properties*

Import a project folder

After changing the computer, you can restore the complete directory tree of the Project Manager without copying the file PRO.DLP (see chapter 12.3, page 561). All projects included in a folder will be entered in the project management (which means that this folder must contain projects, not models). In this way, the projects do not need to be connected individually.

To open the dialog box for importing a project folder,

select **Import Folder** on the **Project** menu of the Project Manager.

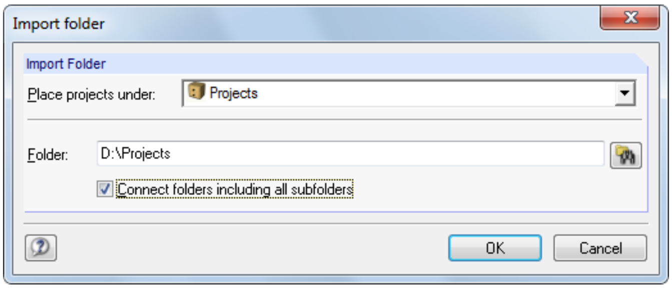


Figure 12.11: Dialog box *Import folder*

In the list *Place projects under*, define the way how you want to integrate the project folder into the management. If you want the folders to appear as independent projects in the Project Manager, select the list entry *Projects* on the top of the list. Use the [Browse] button shown on the left to set the directory for the *Folder* to be linked.

Tick the option *Connect folders including all subfolders* to integrate all subfolders of the folders into the management of the Project Manager.

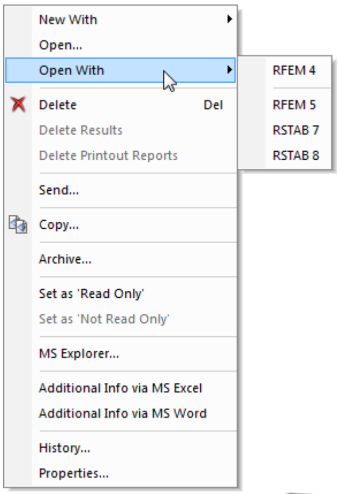
Dlubal

12.1.2 Model Management

Open a model

To open a model out of the Project Manager,

- double-click the model name or its thumbnail image,
- select **Open** on the **Model** menu of the Project Manager (model must have been previously selected)
- or use the context menu of the model.



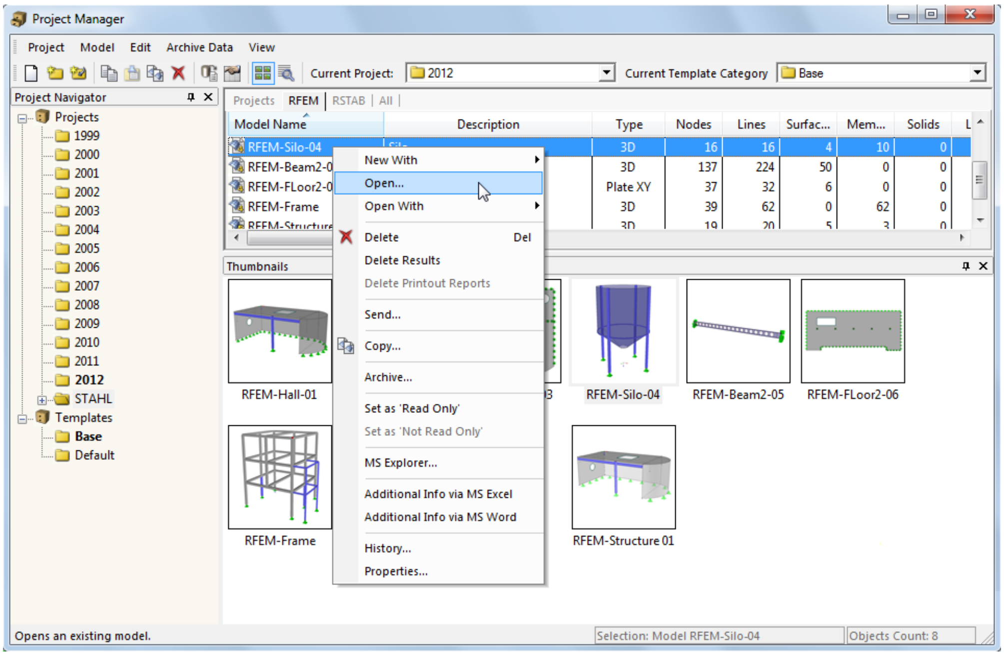


Figure 12.12: Context menu *Model*

Use the context menu option *Open With* shown on the left to select a particular Dlubal application with which you want to open the model.

It is possible to open files from RSTAB directly in RFEM.

Shift / copy a model

To copy a model to another project,

- select **Copy** on the **Model** menu (model must have been previously selected),
- use the **Copy** entry in the model's context menu (see figure above) or
- use the drag-and-drop function by holding down the [Ctrl] key.

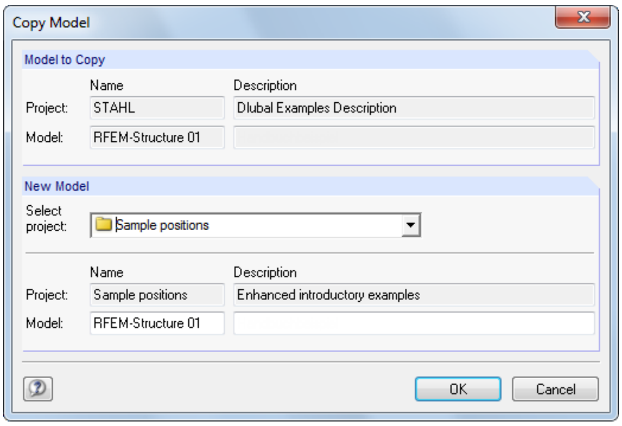


Figure 12.13: Dialog box *Copy Model*

In the dialog box *Copy Model*, specify the target project and enter the *Name* and *Description* for the copy of the model.

To shift a model, hold the left mouse button down when moving it into another folder.

Rename a model

To rename a model,

- select **Properties** on the **Model** menu of the Project Manager (model must have been previously selected) or
- use the **Properties** entry in the model's context menu in the navigator (see Figure 12.12).

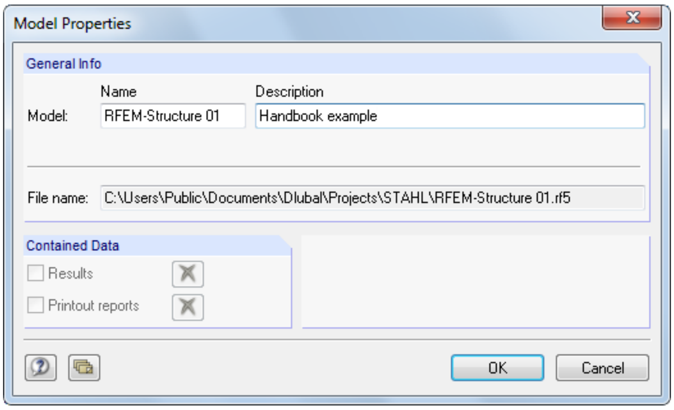


Figure 12.14: Dialog box *Model Properties*

In the dialog box *Model Properties*, you can change the *Name* and *Description* of the model. The *File name* and the model's directory are also displayed.

If the model contains also results and printout reports, you can remove such additional *Data* from the data record by using the [Delete] button.

Delete a model

To delete a model,

- select **Delete** on the **Model** menu of the Project Manager (model must have been previously selected)
- click the [Delete] button in the toolbar shown on the left
- use the **Delete** entry in the model's context menu (see Figure 12.12).

In the context menu, it is also possible to *Delete Results* and/or to *Delete Printout Reports* of the model specifically. In both cases, input data remains available.

To undo the deletion of models,

select **Restore from Dlubal Recycle Bin** on the **Edit** menu of the Project Manager.

The Dlubal recycle bin is described in chapter 12.1.4.2 on page 552.

Show the history

To check the history of a model,

- select **History** on the **Model** menu of the Project Manager (model must have been previously selected) or
- use the **History** entry in the model's context menu (see Figure 12.12).

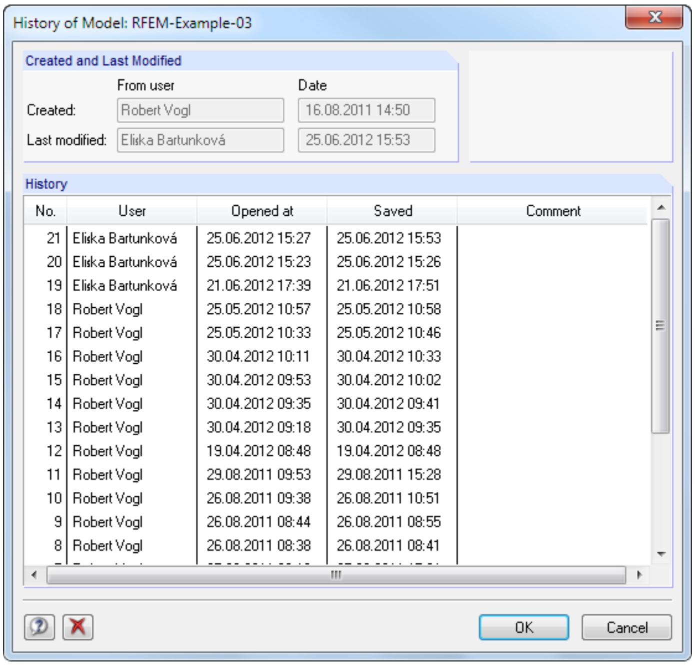


Figure 12.15: Info window *History of Model*

A dialog box appears showing you information about the users who created, opened or modified the model. The overview includes also the time when the individual actions were carried out.

Remarks listed in the *Comment* column are based on the model's general data. Corresponding entries in the dialog box *General Data* are managed in the dialog tab *History*. Take advantage of comments to describe the respective structural processing (see chapter 12.2.2, page 560).

12.1.3 Data Backup

Archive data

You can back up selected models or even an entire project folder in a compressed backup file. The original models remain available.

To start the archiving,

- select **Make Archive** on the **Archive Data** menu of the Project Manager (model or project must have been previously selected) or
- use the context menu of the project (see Figure 12.7) or model (see Figure 12.12).

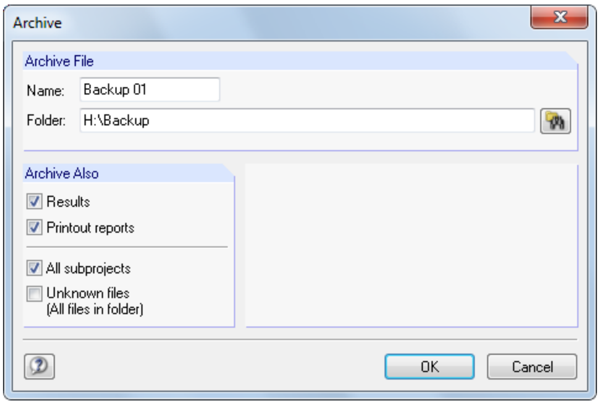


Figure 12.16: Dialog box *Archive*

The backup file can be generated with or without results and printout reports. Further options allow for the integration of subprojects and files that are not part of any Dlubal application.

When you have defined the *Name* and *Folder* of the archive file, create the ZIP file by clicking [OK].

Extract from archive

To extract data from the archive,

select **Extract Project/Models from Archive** on the **Archive Data** menu of the Project Manager.

The Windows dialog box *Open* appears where you can select the ZIP backup file. After clicking [OK], its contents are displayed.

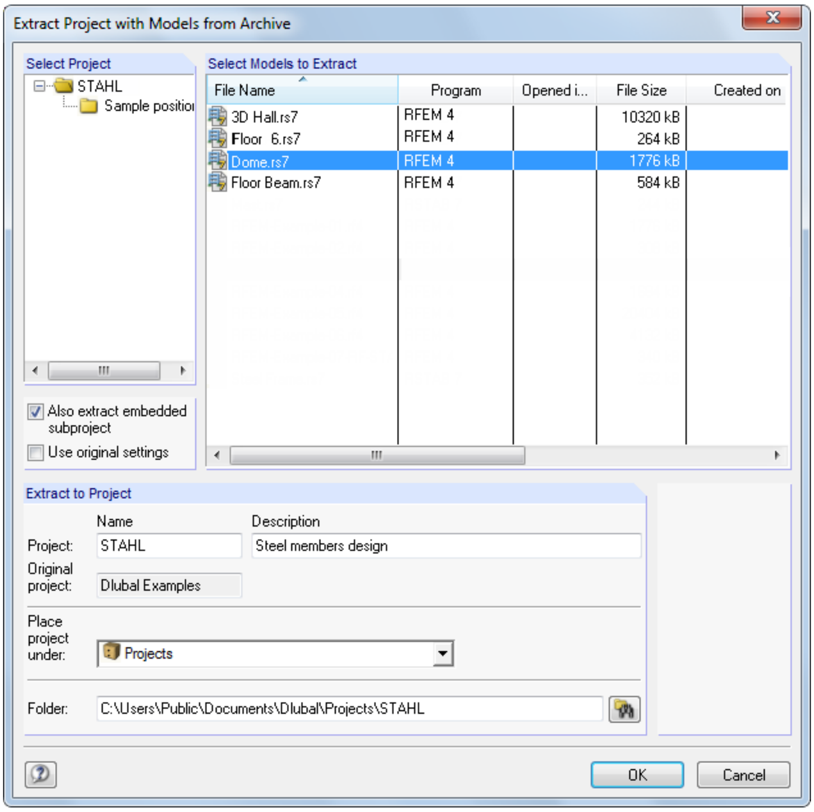


Figure 12.17: Dialog box *Extract Project with Models from Archive*

In the dialog section *Select Models to Extract*, select the models that you want to restore. They can be unpacked with either the original project settings or as new project. In the list *Place project under*, you can define the ranking in the management structure of the Project Manager. Alternatively, you can create a new directory by means of the [Browse] button.

12.1.4 Settings

12.1.4.1 View

Show thumbnails and details

The window area below the model table can be adjusted according to your preferences. You can choose two options for additional windows that can be activated independently of each other.

To set the display options,

select **Pictures Preview of All Models** on the **View** menu and

select **Details of Current Models** on the **View** menu of the Project Manager,

or use the respective toolbar buttons.

Button	Function
	Shows thumbnail images of all models in the project
	Shows model details and preview of model

Table 12.1: Buttons for setting the view

Sorting models

The arrangement of models in the table can be adjusted: As usual with Windows applications, you can sort the list in an ascending or descending order by clicking into the column titles. Alternatively, you can

select **Sort Models** on the **View** menu.

Adjust columns

To arrange the columns according to your needs,

- select **Manage Register Columns** on the **View** menu of the Project Manager
- or use the button [Manage Register Columns] in the Manager toolbar shown on the left.

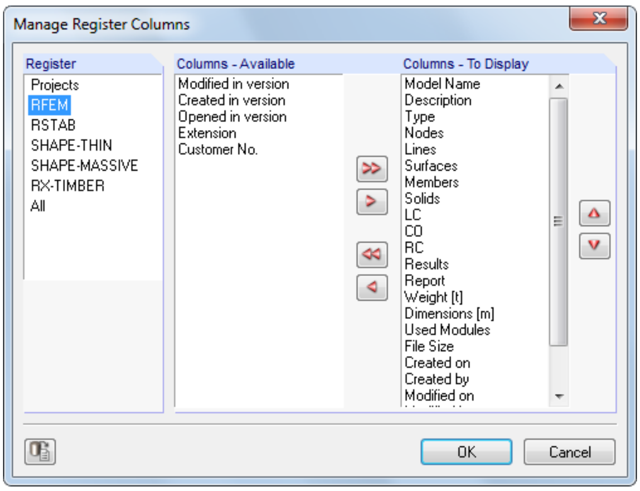
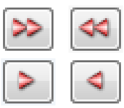


Figure 12.18: Dialog box *Manage Register Columns*



First, define the *Register* whose columns you want to adjust (for example RFEM). Now you can select relevant entries in the list *Columns - Available* to transfer them to the list *Columns - To Display*. Use the arrow buttons [▶] for the transfer. You can also double-click the items. Columns that you don't want to be displayed can be hidden with the [◀] buttons.

The order of columns in the models list can be changed by using the buttons [▲] and [▼] in the list *Columns - To Display*. Click them to shift a selected entry up and down.

To optimize the column widths in the models list, select **Arrange Automatically** on the **View** menu of the Project Manager. You can also use the toolbar button shown on the left.

12.1.4.2 Recycle Bin

To restore deleted projects and models,

select **Restore from Dlubal Recycle Bin** on the **Edit** menu of the Project Manager.

A dialog box appears where all deleted models are listed by projects.

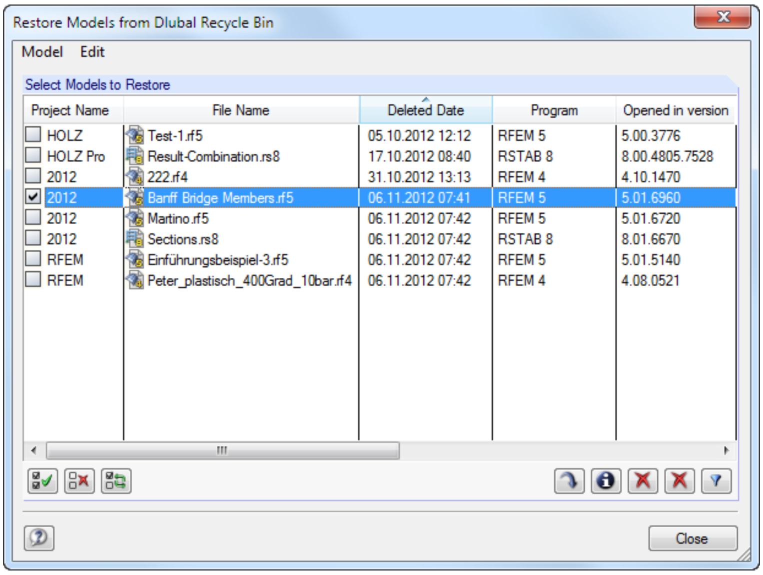


Figure 12.19: Dialog box *Restore Models from Dlubal Recycle Bin*

The models to be restored can be selected by mouse click. With the button [Select all] you can tick the entries all at once. Click the button [Restore Selected Models] to insert the deleted models into the original project folders.

To delete objects stored in the Dlubal recycle bin,

select **Empty Dlubal Recycle Bin** on the **Edit** menu of the Project Manager.

Before the hard delete is performed, a security query is displayed.

To adjust the settings for the Dlubal recycle bin,

select **Settings for Dlubal Recycle Bin** on the **Edit** menu of the Project Manager.

A dialog box appears where the settings for storage location and memory size are managed.

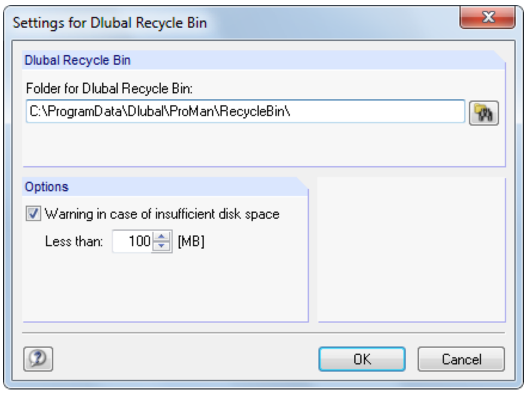


Figure 12.20: Dialog box *Settings for Dlubal Recycle Bin*

12.1.4.3 Directories

The directories of the Project Manager (and Block Manager) can be checked in the *Program Options*. To open the corresponding dialog box,

select **Program Options** on the **Edit** menu of the Project Manager.

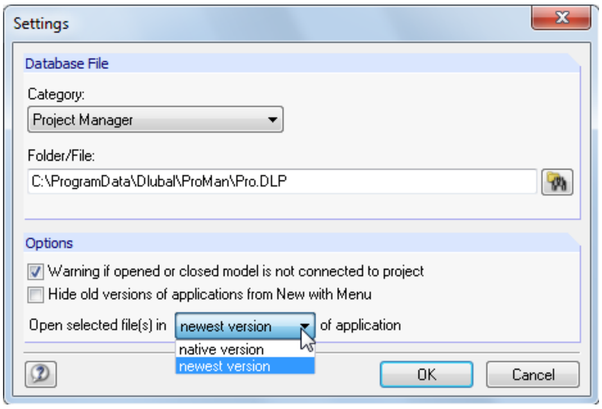


Figure 12.21: Dialog box *Settings*

The *Category* manages the settings separately for both the Project and the Block Manager. The folder and the file name are displayed in the input field below where they can be adjusted, if necessary. The projects are managed in the file **PRO.DLP** which can normally be found in folder *C:\ProgramData\Dlubal\Global\Project Manager*. The [Browse] button helps you to set another path.

As the Project Manager is network-compatible, it is possible to organize the data management for models contained in the Project Manager in a central place: Set the directory for the *PRO.DLP* file on the server (see chapter 12.3, page 561).

The dialog section *Options* offers you general settings for handling RFEM files: Usually, a message appears when opening a file out of the Explorer, an e-mail program etc. when the related folder is not integrated in the management of the Project Manager. The message can be deactivated. Moreover, you can decide which program version you want to use to create or open model files.

12.2 Creating a New Model

To create a model,

- select **New** on the **File** menu of RFEM,
- click the toolbar button [New Model] shown on the left,
- point to **New With** on the **Model** menu of the Project Manager, and then select **RFEM 5**.

Figure 12.22: Button *New Model*

The dialog box *New Model - General Data* opens offering two tabs.

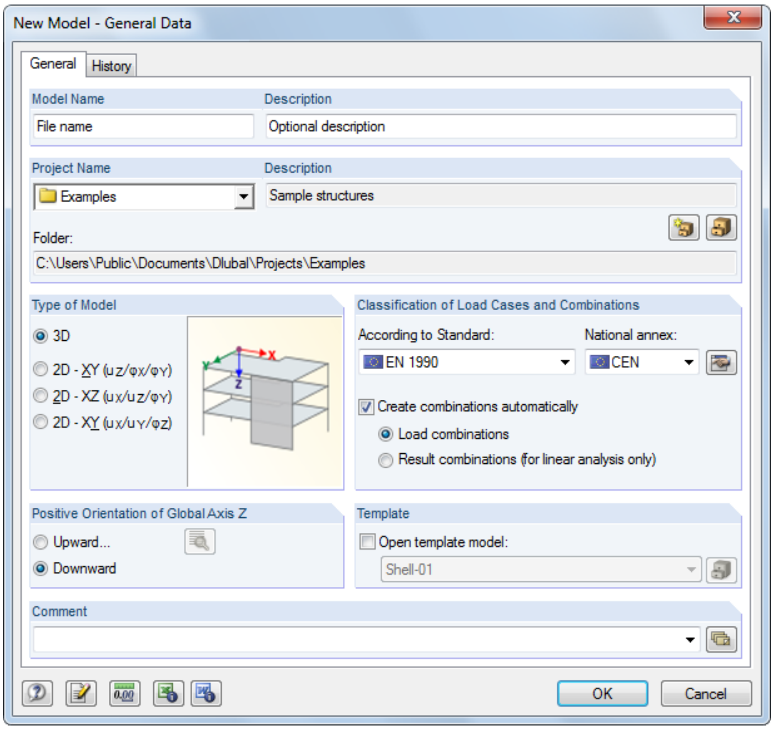


Figure 12.23: Dialog box *New Model - General Data*, tab *General*

When you want to edit the model's general data later,

- point to **Model Data** on the **Edit** menu, and then select **General Data** or
- use the context menu of the model in the *Data* navigator.

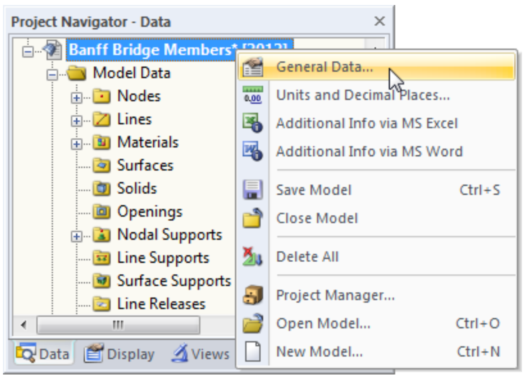


Figure 12.24: Context menu of model

12.2.1 General

The first dialog tab (see Figure 12.23) manages basic model parameters.

Model name / description

Enter a name into the input field for *Model Name*. At the same time, it is used as the model's file name. By entering a *Description* you can describe the model in detail. It appears in the printout report but has no further relevance.

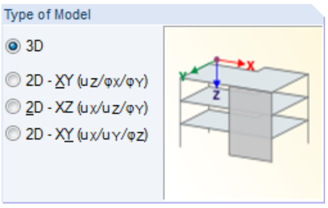
Figure 12.25: Model description in printout report

Project name / description

In the *Project Name* list, you can select the project folder where the model will be created. The current project is preset. If required, you can change the presetting in the Project Manager (see chapter 12.1, page 541) that you can access with the dialog button to the right.

The *Description* and *Folder* of the selected project are displayed for information.

Type of model

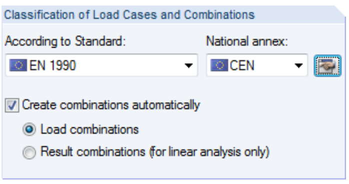


For the model's general data, you have to specify if your structure is a spatial or planar model. In case of 2D models, the effort for input is reduced due to the limited coordinates and degrees of freedom.

The type *2D - XY* is used for planar plate structures such as slabs that are stressed perpendicular to the surface plane. The model types *2D - XZ* and *2D - XY* can be used for walls and shells provided that loads are only acting in direction of the surface plane. The use of type *2D - XZ* is recommended for planar frame structures as this option takes into account moments only about the strong member axes.

It is possible to change the selected type of model subsequently. Please note that this modification may result in data loss, for example when a 3D model is reduced to a wall.

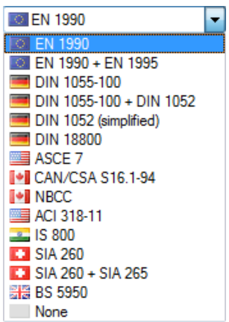
Classification of load cases and combinations



Loading is to be applied by load cases. Load cases may be for example self- weight, snow or live load.

The individual standards define rules for how to combine load cases. Therefore, it is important to assign load cases to particular action categories (see chapter 5.1, page 171). Thus, when creating load and result combinations, RFEM is able to provide the load cases automatically with correct partial safety factors and combination coefficients.

According to standard



The list *According to Standard* contains a variety of rules and standards describing the principles for ultimate limit state, serviceability and resistance of structural systems. With the selection of the standard the rules for creating load and result combinations in RFEM are defined. This specification is especially significant for the automatic creation of combinations by RFEM (see chapter 5.2, page 174 to chapter 5.4, page 189).

When *None* is set, no combination will be created automatically, which corresponds to the default setting in RFEM 4. In this case, load cases must be superimposed manually (see chapter 5.5.1, page 194 and chapter 5.6.1, page 201).

When changing the standard subsequently it is required to reclassify the load cases and to adjust the combination. A corresponding warning appears.

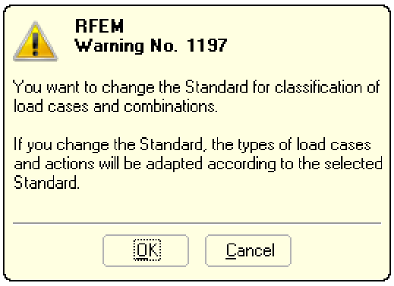


Figure 12.26: Warning when changing the standard

National annex

When the standard *EN 1990* is selected, an additional picklist appears: Though combination rules are defined in the Eurocode standard, countries are allowed to specify partial safety factors and combination coefficients themselves.

The list offers you a choice among national annexes of different countries. When the option *CEN* is set, the factors recommended by the European Commission are applied.

Use the dialog button [Edit] to check the partial safety factors and combination coefficients of the currently set standard. When a user-defined standard is set, you can even adjust them.

The factors are organized in several tabs in the dialog box *Coefficients*. The first tab manages the *Partial Safety Coefficients* γ for the design situations "static equilibrium" and "ultimate limit state".

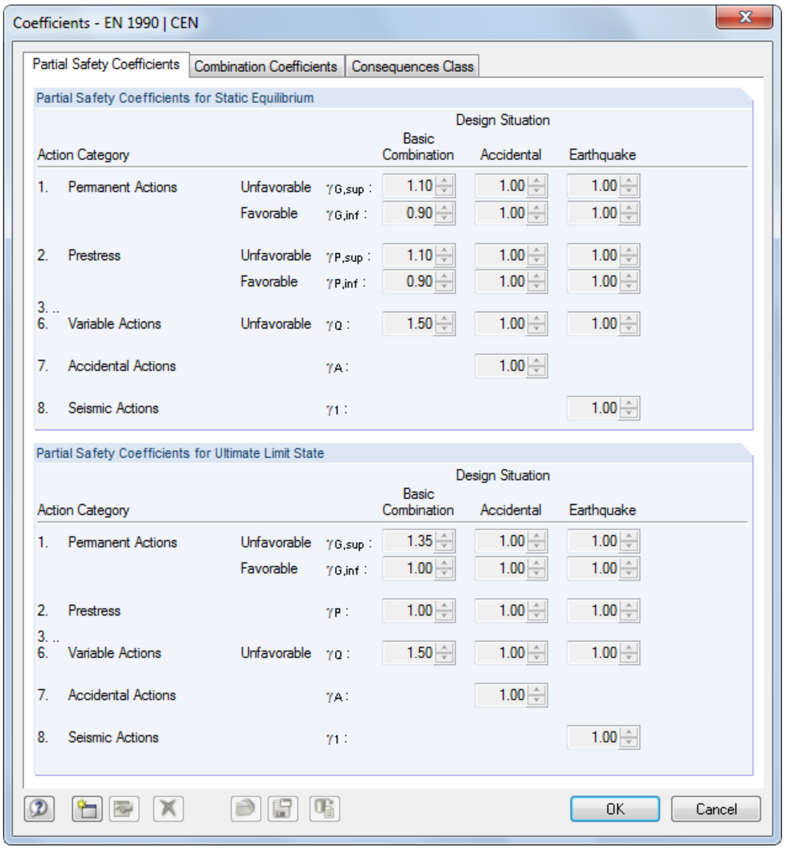


Figure 12.27: Dialog box *Coefficients*, tab *Partial Safety Coefficients*

The dialog tab *Combination Coefficients* (see Figure 5.24, page 191) controls the factors ψ and ξ. In the tab *Consequences Class* that is available for EN 1990 you can define the reliability factor K_{FI}.

Create combinations automatically

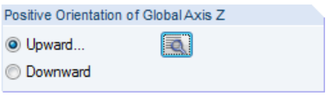
The check box is clear by default so that both options below are not enabled for access. Thus, the required load and result combinations must be created manually like in RFEM 4 (see chapter 5.5.1, page 194 and chapter 5.6.1, page 201). When combining load cases, the specified standard ensures that the partial safety and combination factors are assigned automatically.

As an alternative, you can *Create combinations automatically*. Then, additional dialog tabs are available in the dialog box *Edit Load Cases and Combinations* as well as separate entries in the *Data* navigator. In addition, tables 2.2. to 2.4 are enabled. Generating combinations is described in detail in chapter 5.2, page 174 to chapter 5.4, page 189.

The automatic superposition is based on the add-on module RF-COMBI for version RFEM 4. Find additional information about combinatorics in the module's manual that is still available for download at www.dlubal.com.

During the automatic superpositioning RFEM creates either *Load combinations* or *Result combinations*. The difference between both combination possibilities is described in chapters 5.5 on page 193 and 5.6 on page 201.

Positive orientation of global axis Z



This dialog section controls the orientation of the global axis Z. In CAD programs, the Z-axis is usually directed upwards. In programs used for structural analysis, it is normally directed downwards. The specification is irrelevant for the calculation.

If Z is defined *Upward* and the self-weight is specified with factor 1.0 in direction Z in the base data of the load case, the self-weight acts upwards. In this case, the self-weight factor needs to be changed to –1.0.

If the global Z-axis is directed upward, it is possible to define settings for surfaces and members by clicking the [Choose] button shown on the left. The dialog box *Orientation of Local z-Axis* appears.

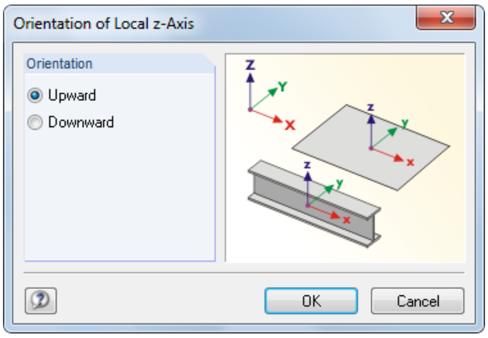


Figure 12.28: Dialog box *Orientation of Local z-Axis*

The local z-axis can be oriented *Upward* or *Downward* in order to assign for example elastic surface foundations or reinforcements of plates appropriately. Then, answer the query (see Figure 12.29) appearing when closing the *General Data* dialog box with *No*.

It is possible to change the orientation of the Z-axis subsequently. You also have the possibility to adjust the coordinates and global loads so that the view of the model will be kept. When the axis direction is modified, a query appears (see following figure).

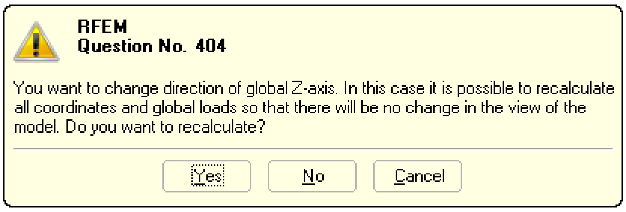
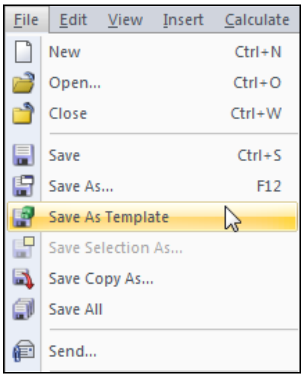


Figure 12.29: Query when changing the Z direction

Template



The model can be created according to a template that has been saved in another model. To access the save function,

select **Save As Template** on the **File** menu.

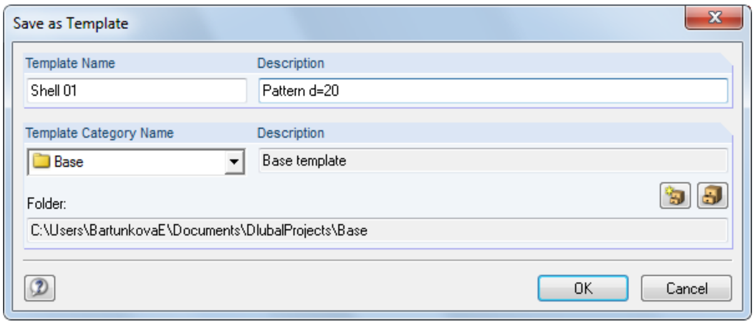
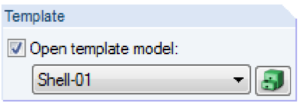


Figure 12.30: Dialog box *Save as Template*

In general, the templates are stored in the Dlubal folder for template models called *Base*. Access is also available in the navigator of the Project Manager by selecting *Default* under *Templates* (see Figure 12.3, page 542).



After ticking the check box in the dialog box *New Model - General Data*, you can select the relevant *template model* from the list.

Click the button shown on the left to open an overview with preview pictures helping you to choose among the templates (see following figure).

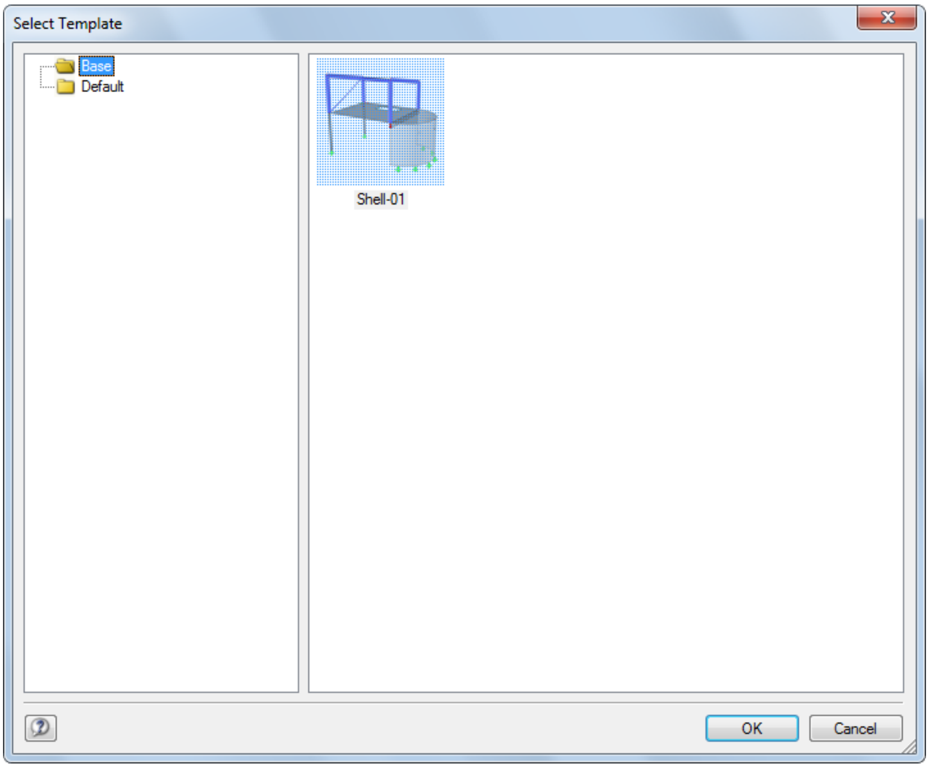


Figure 12.31: Dialog box *Select Template*

Comment

You can enter a text into the input field or select a text from the list to add a short description to the general data. The comment appears in the printout report, too.

The buttons in the *General Data* dialog box have the following functions:

Button	Name	Explanation
	Comment	→ chapter 11.1.4, page 421
	Units and decimal places	→ chapter 11.1.3, page 420
	MS Excel	Option for additional explanations in the XLS file format saved in RFEM file
	MS Word	Option for additional explanations in the DOC file format saved in RFEM file

Table 12.2: Dialog box *General Data*, buttons

12.2.2 History

The second dialog tab of the *General Data* dialog box keeps record of processing in the form of a *History*.

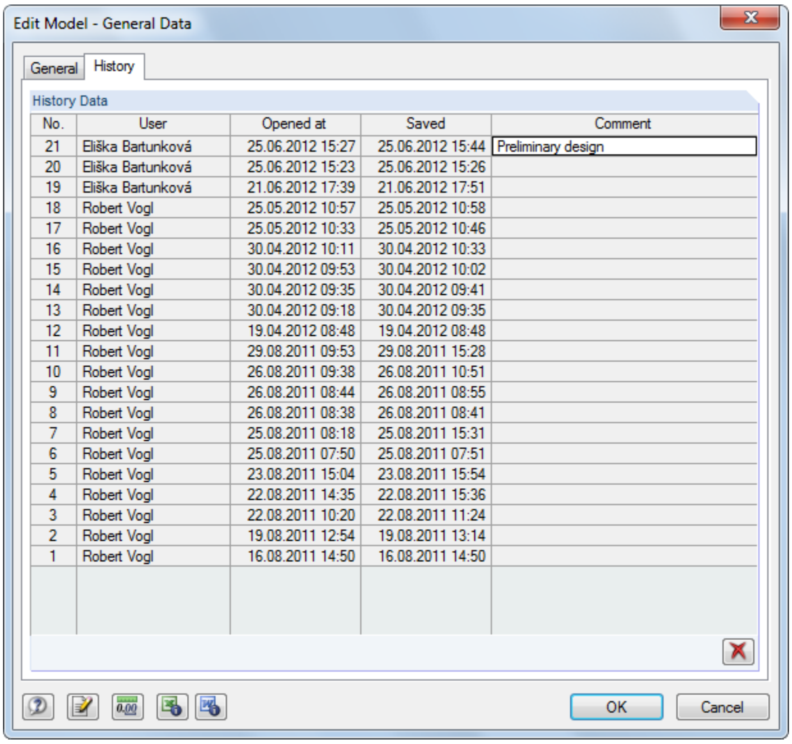


Figure 12.32: Dialog box *Edit Model - General Data*, tab *History*

The table shows information about the point of time when a *User* has *Opened* and *Saved* the model.

In the topmost table row, you can enter a *Comment* describing the current state of model processing. The remark will be effective for the history when saving the model next time. The comment appears not only in the *History* tab but is also available in the Project Manager (see Figure 12.15, page 549).

To delete the history, click the button [X]. In this way, it is possible to remove personal information from the file.

12.3 Network Management

When several users are working on the same projects, model management can be organized by the Project Manager, provided that the models are stored in a folder that is accessible on the network.

First, connect the network folder to the internal project management. Please find a description in chapter 12.1.1 on page 543. Now, you can directly access the models of this folder in the Project Manager, which means that you can open or copy the models, check their history or provide them with a write protection.

If another user is already working on the model that you want to open, a warning appears. In this case, you can open the model as a copy.

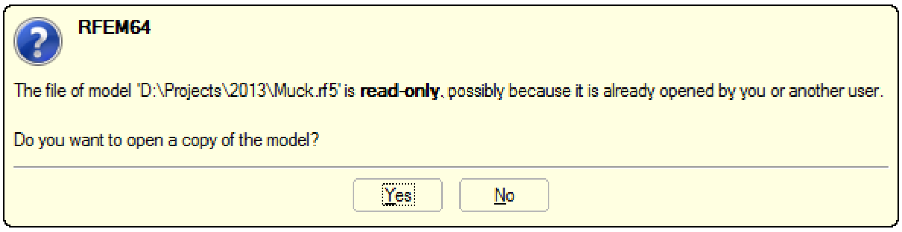


Figure 12.33: Query when opening a write-protected model

An automatic data synchronization of modifications is not possible.

Information about the projects registered in the Project Manager is stored in the file **PRO.DLP**. This is an ASCII file which is by default located under *C:\ProgramData\Dlubal\Global\Project Manager*.

By copying the PRO.DLP file to another computer you can avoid connecting folders project by project. In addition, the file can be edited by an editor. This facilitates the import of all relevant project folders into the internal file management of the Project Manager, especially after new installations. As an alternative, you can use the function *Import Folder* (see chapter 12.1.1, page 546).

Before copying the PRO.DLP file - like before uninstalling Dlubal applications - it is recommended to save the existing file.

The Project Manager is network-compatible. The file management can be organized in a central place so that all users are integrated in one common project management. To define the network settings,

select **Program Options** on the **Edit** menu of the Project Manager.

A dialog box opens where you can define the storage location for the file PRO.DLP (see Figure 12.21, page 553).

The Project Manager runs on every local computer, but each is using the central server file PRO.DLP. In this way, all users can carry out modifications to the project structure at the same time. For write access to the PRO.DLP file, the file is locked only for a short time and is unlocked immediately afterwards.

12.4 Block Manager

The Block Manager manages model blocks by cross-project management: Selected objects can be saved as blocks and reimported to other models. A multitude of typified elements is predefined in the Block Manager's *Catalog*.

To open the Block Manager, select **Block Manager** on the **File** menu in RFEM, or use the toolbar button shown on the left.

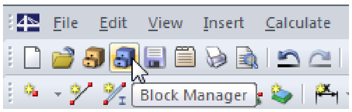


Figure 12.34: Button *Block Manager* in the toolbar

When you open the Block Manager, a multi-part window appears. Like the Project Manager (see chapter 12.1) it has its own menu and toolbar.

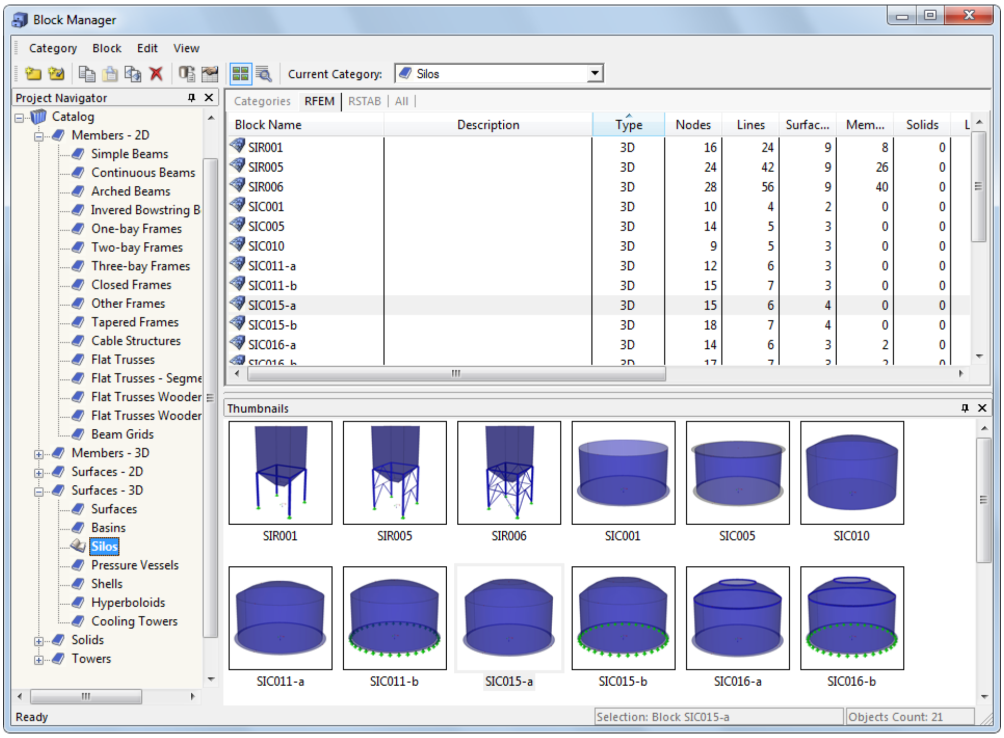


Figure 12.35: Block Manager

Project navigator

On the left, you see the navigator with the *Catalog* of all block categories. The current category is set bold. To select another category, double-click the relevant entry or use the list *Current Category* in the Manager toolbar. The table to the right of the navigator lists the objects filed in the selected category. Blocks for various member, surface and solid models are available for selection.

Table of blocks

The blocks of the selected category are listed one by one. The *Block Name* and *Description* as well as significant object and file information are shown respectively.

To adjust the displayed columns, select **Manage Register Columns** on the **View** menu of the Block Manager, or use the toolbar button shown on the left (see chapter 12.1.4.1, page 551).

Details

The window section shows you detailed information about the selected block.

Preview

The selected block is displayed in a preview. The size of the preview window can be adjusted by moving the upper edge of the window.

Thumbnails

The bottom area of the Block Manager offers you a graphical overview about the blocks contained in the selected category. The thumbnail images are interactive with the table above.

Use the pins to minimize particular window parts. They will be docked as tabs in the footer.

12.4.1 Create a Block

To create a block from particular objects, select the relevant objects in the current RFEM model in the work window. A multiple selection is possible by drawing a window with the mouse button. You can also click several elements by holding down the [Ctrl] key.

To create a new block,

select **Save as Block** on the **File** menu in RFEM.

The following dialog box opens.

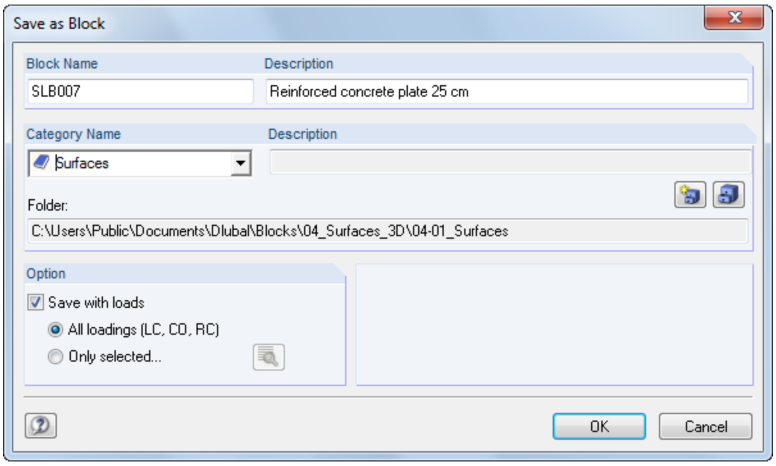


Figure 12.36: Dialog box *Save as Block*

Define the *Block Name* and *Category Name* under which you want to save the block. The category can be selected in the list. The *Description* is an optional entry used to describe the block shortly.

The directory of the block is indicated in the dialog field *Folder*.

In case loads are defined, they can be saved together with the block. In addition, you can use the settings in the dialog section *Option* to decide if all loads or only selected load cases are relevant.

To create a new block category, use the button [New Category] shown on the left.

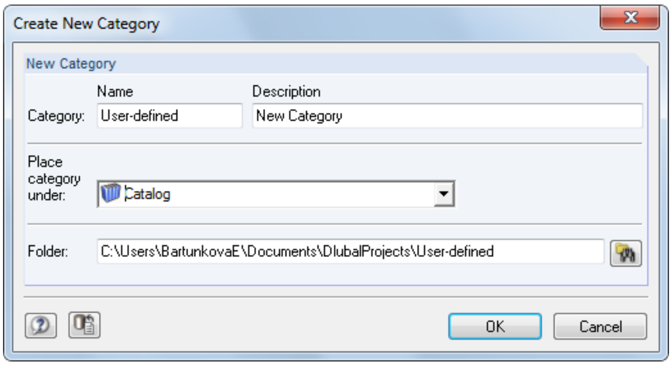


Figure 12.37: Dialog box *Create New Category*

The creation of a block is similar to the creation of a new project in the Project Manager (see chapter 12.1.1, page 543).

12.4.2 Import a Block

To import a block into the current RFEM model, open the Block Manager (see Figure 12.34, page 562). First, select the category in the catalog. Then, you can select the relevant block by mouse-click in the *RFEM* tab.

To start the import,

- select **Insert Block** on the **Block** menu or
- use the context menu of the block.

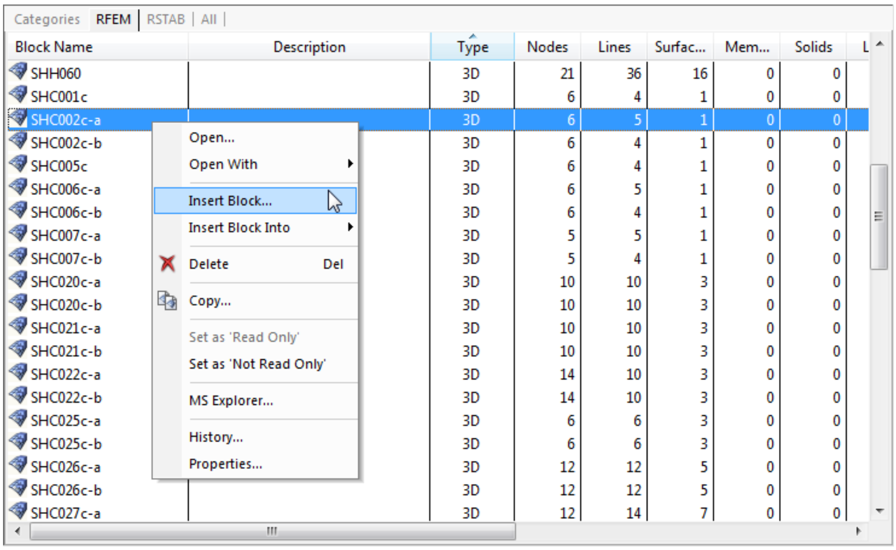


Figure 12.38: Context menu of block

You can also double-click the block in the table. The following dialog box opens.

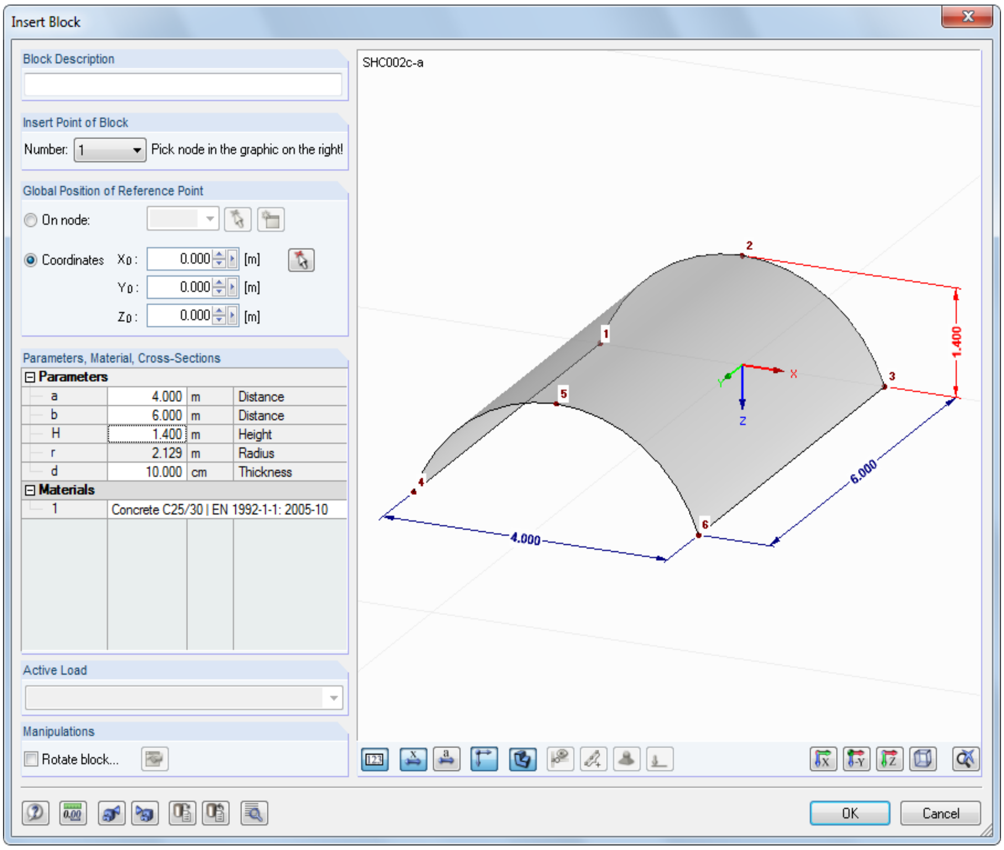


Figure 12.39: Dialog box *Insert Block*

Specify the *Insert Point of Block* (the "snap point") and the *Global Position of Reference Point* in the dialog box. The points can also be selected graphically in the block model or RFEM model.

Geometric *Parameters* can be modified as well as *Material* and *Cross-Sections*. A click into the relevant input field enables buttons that you can use to select items from a list or to open libraries.

For user-defined blocks it is even possible to import loads: The *Active Load* can be selected in the list.

Click the [Edit] button shown on the left to access specific import settings that can be defined in another dialog box.

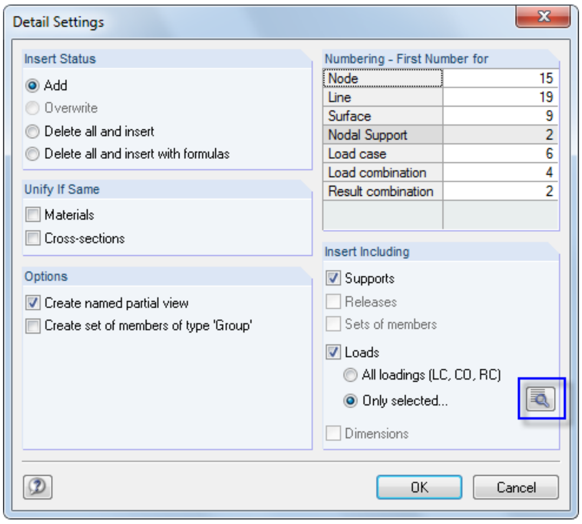


Figure 12.40: Dialog box *Detail Settings*

With the options available in the dialog box *Detail Settings* you determine how objects will be aligned with the existing model elements. Moreover, you can influence the *Numbering*.

Click the [Select] button to open a new dialog box where you can select the load cases, load and result combinations for the import.

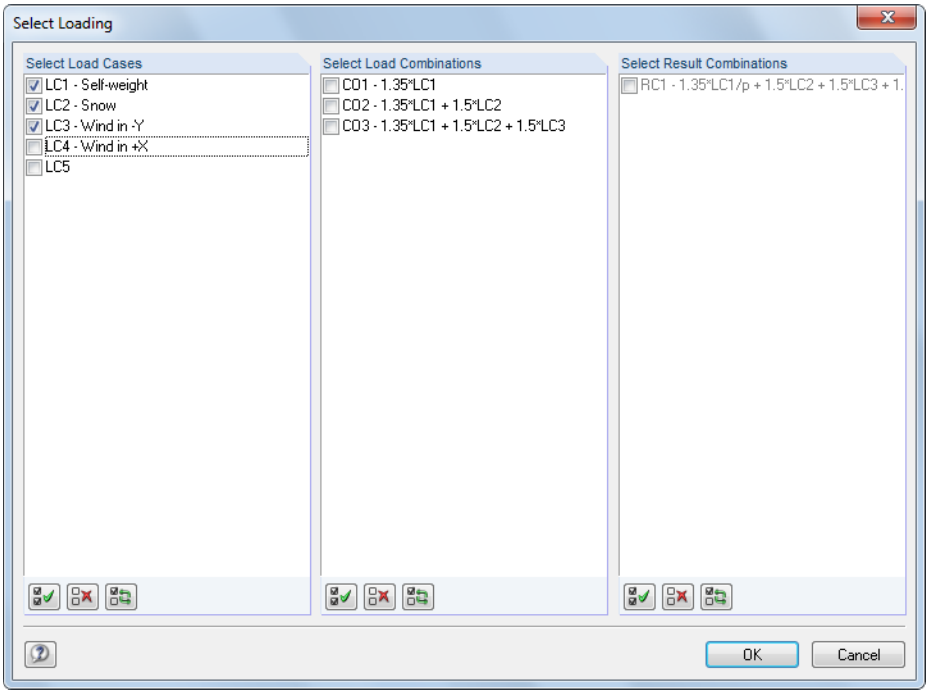


Figure 12.41: Dialog box *Select Loading*

12.4.3 Delete a Block

To delete a block,

- select **Delete** on the **Block** menu of the Block Manager (block must have been previously selected)
- click the [Delete] button in the toolbar shown on the left
- use the context menu of the block (see Figure 12.38).

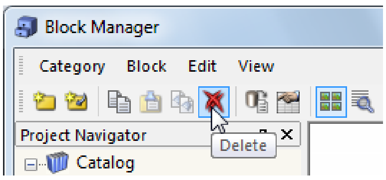


Figure 12.42: Button *Delete*

After confirming the security query, the block will be put into the Dlubal recycle bin.

12.5 Interfaces

RFEM offers you the possibility to exchange data with other programs. Thus, you can use for example CAD templates created in other applications. Furthermore, results of structural calculations from construction or design software can be made available.

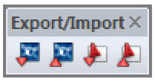
Exporting the printout report as **RTF** file and to **VCmaster** is described in chapter 10.1.11 on page 400.

RFEM can also be run externally using a programmable interface based on COM technology (for example Visual Basic). With **RF-COM** which can be acquired as RFEM add-on module you can use customized input macros and follow-up programs.

12.5.1 Direct Data Exchange

RFEM provides an interface to software programs developed by the DLUBAL company. Input data of all previous versions of **RFEM** can be imported without problems. Also files of the framework program **RSTAB** can be opened directly in RFEM so that surface or solid elements can be added. In the same way, you can open files created with RFEM 5 in RSTAB 8.

There is a direct connection to CAD programs from **Tekla Structures** and **Autodesk AutoCAD** (but not for LT versions). In this way, it is possible to take advantage of BIM (Building Information Modeling) with RFEM because data models can be exchanged directly for digital planning processes.



To start the direct data exchange,

select **Import** or **Export** on the **File** menu in RFEM

or use the toolbar buttons shown on the left.

The dialog box shown in Figure 12.43 or Figure 12.44 on page 568 opens where you can select the relevant CAD program in the dialog section *Direct Imports* or *Direct Exports*.

The buttons in the RFEM toolbar *Export/Import* have the following functions:

	Direct import from Tekla Structures
	Direct export to Tekla Structures
	Direct import from AutoCAD
	Direct export to AutoCAD

Table 12.3: Buttons of toolbar *Export/Import*

Find descriptions for the interfaces with Tekla Structures and Autodesk AutoCAD Revit at www.dlubal.com/manuals-for-category-interfaces.aspx.

- **RX-Tekla**
- **RX-Revit**

12.5.2 File Formats for Data Exchange

If CAD programs or programs for structural analysis can create files of the types *.**stp**, *.**dxf**, *.**fem**, *.**asf**, *.**dat**, *.**cfe** or *.**ifc**, corresponding data can be used as template for RFEM. Vice versa, RFEM is able to create files in formats appropriate for other programs.

To open the dialog box for importing a file,

select **Import** on the **File** menu.

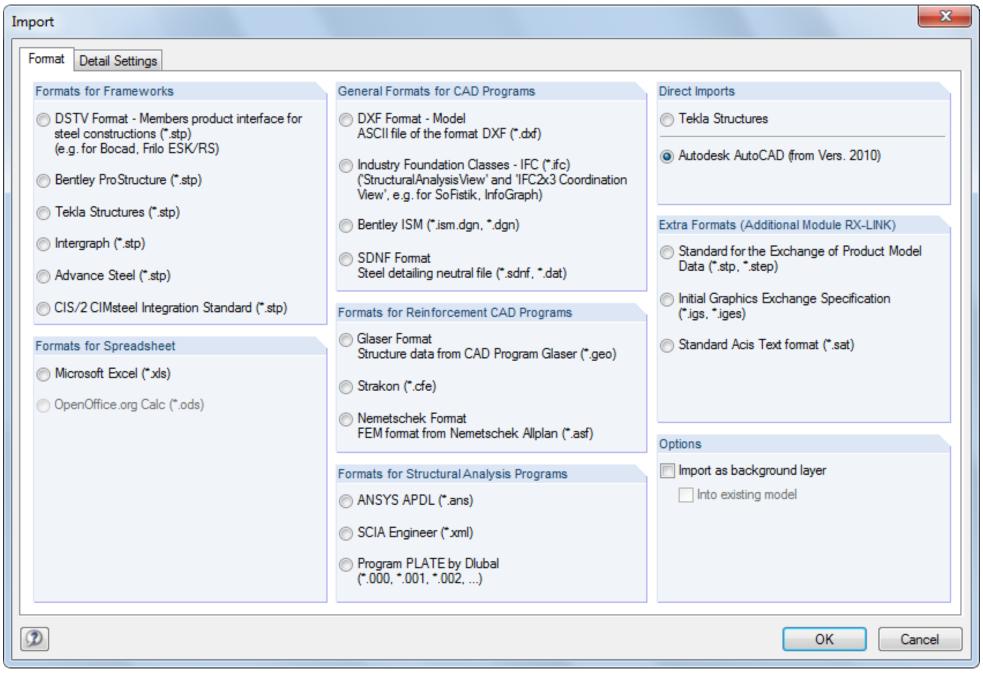


Figure 12.43: Dialog box *Import*

When the option *Import as background layer* is ticked, RFEM will show you only a line model in the work window that can be used to set nodes, lines etc. (see chapter 11.3.10, page 455).

To start the export of an RFEM file,

select **Export** on the **File** menu.

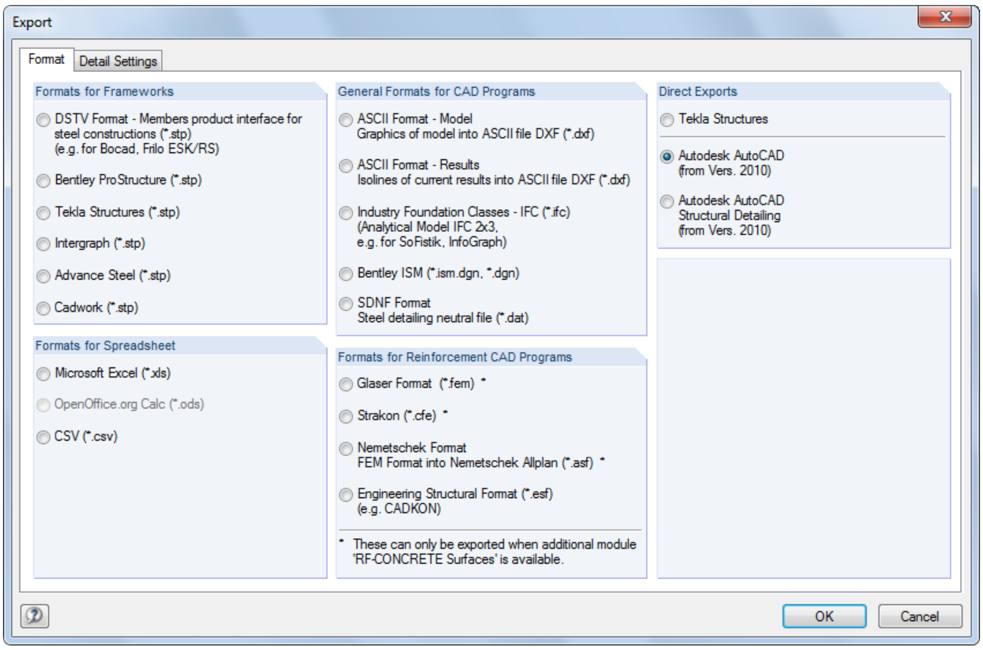


Figure 12.44: Dialog box *Export*

File formats for frameworks

DSTV format *.stp

When using the product interface from DSTV (German Steel Construction Association), transfer is not performed with framework files reduced to line models but files contain all model and load data that is necessary for efficient processing. DLUBAL, like many other software developers, work on the development of the product interface. So it is possible to exchange data with a variety of programs like *Bentley ProStructure*, *Tekla Structures*, *Intergraph Frameworks*, *Advance Steel*, *CIS/2 CIMSteel* or *cadwork*. You can select the programs also directly in the import and export dialog boxes.

The interface covers structural and CAD data in general. RFEM, however, supports only the structural format with specific “entities” that can be found in a description (PDF download at http://www.dlubal.com/download/pss_dstv-E.pdf).

The interface transfers information of nodes, members and cross-sections including member eccentricities and cross-section rotations. Furthermore, nodal supports, load cases, load and result combinations with nodal and member loads as well as imperfections are transferred. The results of the calculation can be stored in the exchange file as well.

More settings for data exchange can be defined in the dialog tab *DSTV (.stp)*.

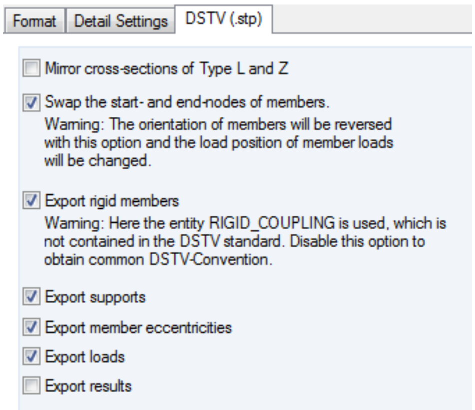


Figure 12.45: Dialog box *Export*, tab *DSTV (.stp)*

File formats for spreadsheet

MS Excel format *.xls

RFEM is able to import and create tables as *.xls files. The data exchange with MS Excel has already been described in chapter 11.5.6 on page 488. However, the described exchange option is only available for the active RFEM table. The function described in the following covers data of the model all at once. Thus, user-defined external generators for model and load data can be used.

To **import** an XLS file, open the file in MS Excel first. Then, you can use the option *Microsoft Excel* in the RFEM import dialog box (see Figure 12.43) to open the following dialog box.

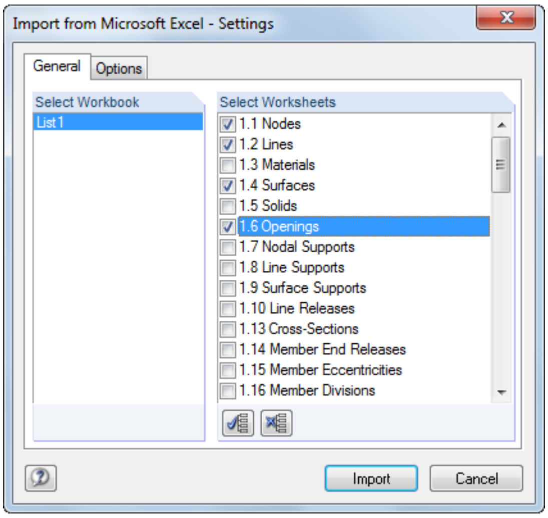


Figure 12.46: Dialog box *Import from Microsoft Excel - Settings*

Select the *Workbook* and *Worksheets* that you want to import. Descriptions, sequence and structure of worksheets must match exactly the data in RFEM so that imported data can be written correctly into the RFEM tables. If you are not quite sure, try to create a XLS file from the current RFEM file for test purposes.

In the *Options* tab, specify if worksheets will be imported with or without headers and how formulas will be represented in the worksheets.

When **exporting** a file, it is not necessary to open MS Excel. The spreadsheet program starts automatically.

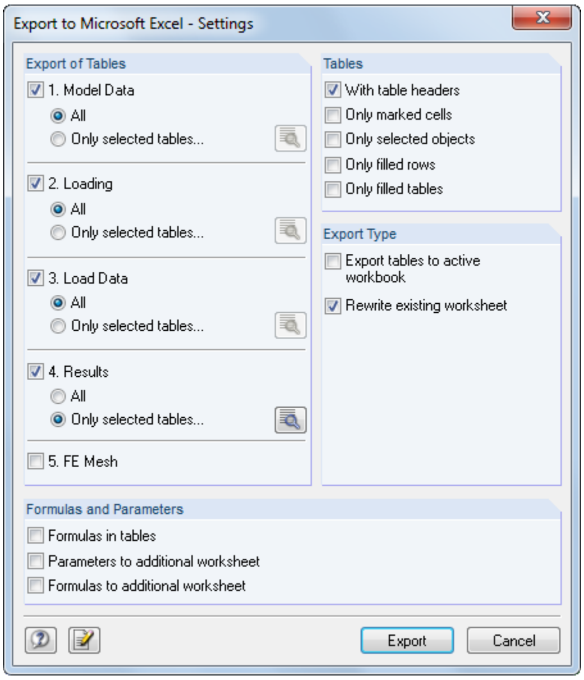


Figure 12.47: Dialog box *Export to Microsoft Excel - Settings*

In the dialog section *Export of Tables*, select the tables that you want to export. When you activate the option *Only selected tables*, RFEM enables the respective [Select] button shown on the left. Click the button to open another dialog box for specific settings.

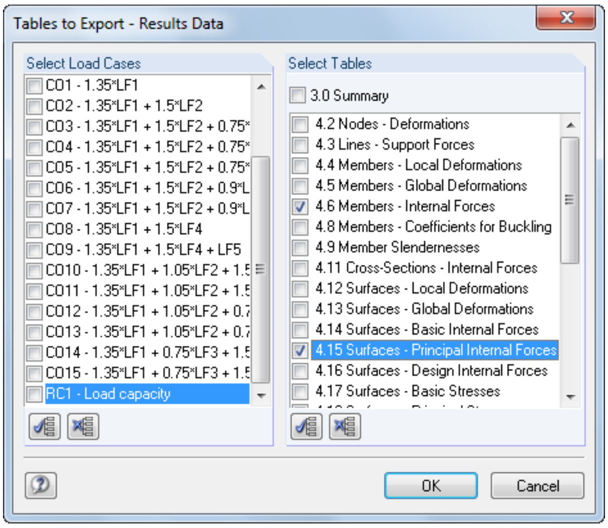


Figure 12.48: Dialog box *Tables to Export - Results Data*

In the dialog section *Formulas and Parameters* of the initial export dialog box (Figure 12.47), you can decide if stored formulas will also be transferred from RFEM to Excel during the data exchange.

OpenOffice format *.ods

This interface is only available when *OpenOffice.org Calc* and *RFEM 5 32-bit* are installed.

The import and export options are similar to the data exchange between RFEM and Excel described in detail above.

General file formats for CAD programs

ASCII format *.dxf

The DXF format transfers only general information concerning lines used in the model. RFEM is able to import a line model created for example in *AutoCAD* and to create a DXF file from the current model. A layer will be used for each cross-section. Nodal supports, loads etc. cannot be taken over.

More settings for data exchange can be defined in the dialog tab *ASCII Format DXF (*.dxf)*. It is recommended to check the parameters, especially before the import.

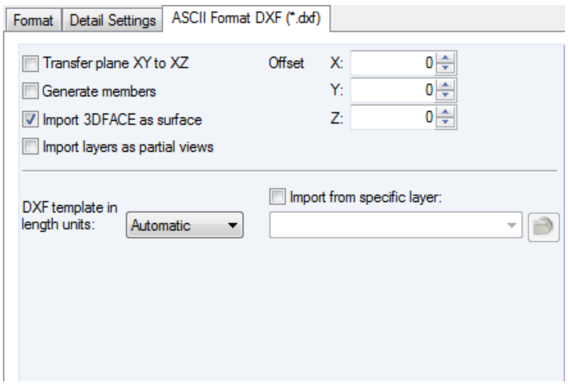


Figure 12.49: Dialog box *Import*, tab *ASCII Format DXF (*.dxf)*

It is also recommended to check the *length units* of the DXF template. Optionally, you can enter an *Offset* to place the DXF model in a certain distance to the origin. Select the option *Import 3D FACE as surface* to create 3D surfaces from the DXF template automatically as surfaces in RFEM.

If you want to *Import* a file *from a specific layer*, use the button [Select DXF File] shown on the left to select the DXF file. Then, the individual layers are available for selection in the list.

In most CAD programs, the Z-axis is directed upwards. In RFEM, however, it is usually directed downwards. Now, when you switch to the second dialog tab *Detail Settings* in the import dialog box, and set *Down* in the list for the *Z axis*, weight loads can be entered positively in RFEM.

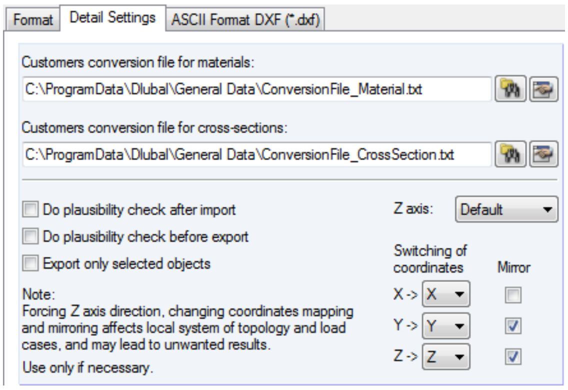


Figure 12.50: Dialog box *Import*, tab *Detail Settings*

The DXF export includes also values. However, only one value or one group of values is possible. If several values are specified, always the first value will be exported and a corresponding message will be displayed.

For the DXF export it is also recommended to check the orientation of the *Z axis*.

IFC format *.ifc

IAI logo

The *Industry Foundation Classes* (IFC) are a global standard of exchanging data for model-based approaches in the construction industry. They have been developed by the IAI (International Alliance for Interoperability). The IFC are structured in domains (architecture, design, structural analysis, electrical engineering etc.). DLUBAL software supports the domain for structural engineering which allows for the transfer of structural data like nodes, members, supports, load cases and loads. The IFC are still under development.

Please find a description of the interface at www.buildingsmart.de.

When you export an RFEM model as IFC model, an analytical model is created in the version IFC 2 x Edition 3.

Bentley format *.ism.dgn, *.dgn

The interface makes it possible to exchange data with the CAD product *MicroStation*. RFEM is able to import model data as well as to export RFEM files, using the possibilities of interoperability. Thus, a connection to all Bentley applications such as *ProSteel* is given on the basis of ISM (Integrated Structural Modeling).

SDNF format *.dat

The SDNF format (*Steel detailing neutral file*) is used to exchange geometrical data like nodes, cross-sections and members with INTERGRAPH.

File formats for CAD reinforcement programs

Glaser format *.geo, *.fem

RFEM provides an interface with the program *Glaser* by ISB CAD making the exchange of geometrical and reinforcement data possible.

If you want to export reinforcement results of the add-on module RF-CONCRETE Surfaces, make sure that surfaces are defined as plane and horizontal, that means created in the plane XY.

In the dialog tab *Results - Glaser (.fem)*, you can control the reinforcement results that are relevant for the export.

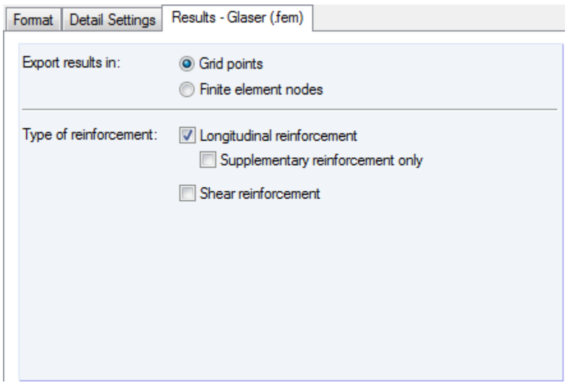


Figure 12.51: Dialog box *Export*, tab *Results - Glaser (.fem)*

The reinforcements exported to GLASER can be realized graphically in grid points or FE nodes. As they are available as values, it is possible to use them in reinforcement drawings.

Strakon format *.cfe

The Strakon format supports the exchange of geometrical data like surfaces with the CAD program system STRAKON produced by the software developer DICAD.

In the dialog tab *Results* of the *Export* dialog box, you can set the surfaces whose reinforcements you want to export (see Figure 12.52).

Nemetschek format *.asf

Data exchange is also possible with the program *Allplan* by NEMETSCHEK.

For the export of reinforcement results of the add-on module RF-CONCRETE Surfaces, please note that surfaces may be defined in any position but must be plane. During the export, RFEM creates a ASF file per plane surface. For example: When the RFEM model has 12 surfaces, 12 files will be created that can be merged to a 3D model in Allplan.

In the dialog tab *Results* of the *Export* dialog box, you can set the surfaces whose reinforcements you want to export.

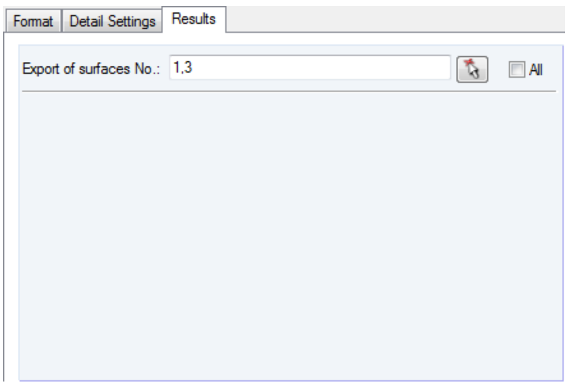


Figure 12.52: Dialog box *Export*, tab *Results*

ESF format *.esf

This interface has been developed especially for the program *CADKON* by AB STUDIO. It is possible to export plane surfaces with constant thicknesses including openings and material information. Furthermore, it is possible to export the reinforcements of RFEM surfaces in the format *.esf (*Engineering Structural Format*).

The import of *.esf files is not possible. You may use the DXF format instead.

File formats for structural analysis programs

Ansys format *.ans

Use the interface with the FE program ANSYS to import files available in the format *.ans. In this way, you can use data of this multifunctional program also for analyses performed with RFEM.

Scia format *.xml

It is also possible to import model data from the structural analysis program *Scia* by NEMETSCHEK to RFEM, provided that data is available in the *.xml format.

General Dlubal formats *.xml, *.ft5

To save RFEM files as XML files or templates,

select **Save As** on the **File** menu.

In the Windows dialog box *Save As*, use the list to set the relevant file type in the dialog field *Save as type*.

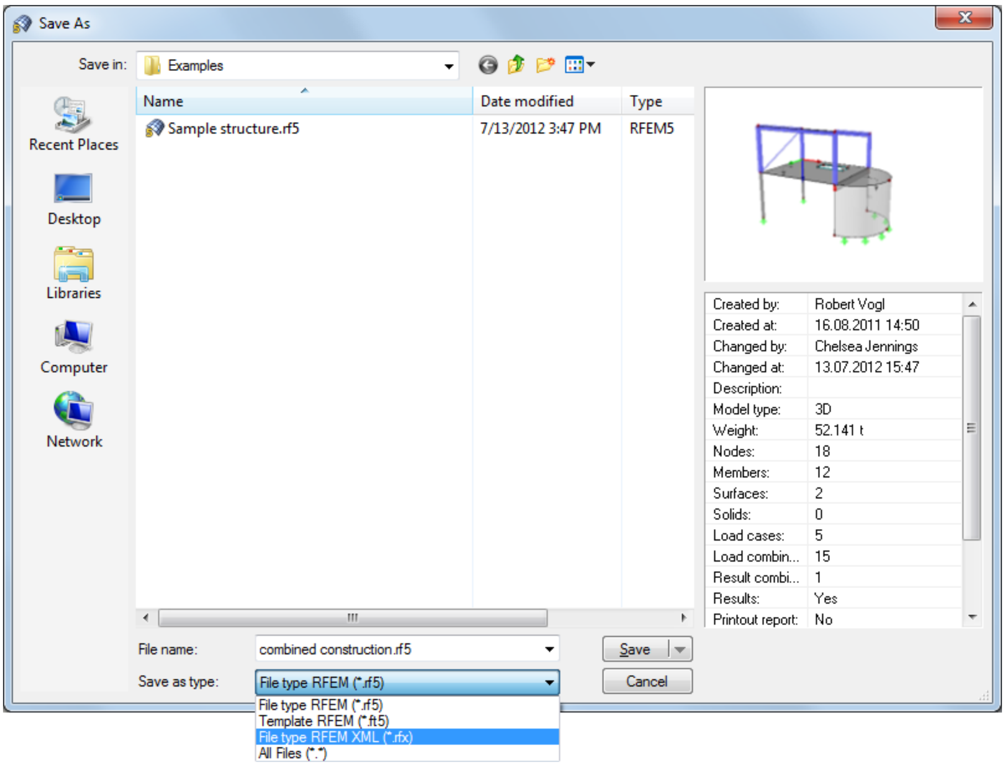


Figure 12.53: Dialog box *Save As*

With the FT5 format you save the model as a template which can be imported later when creating a new file (see Figure 12.23, page 554).

When you save the model with the file type RFX, tabular data will be converted into an XML format. The remaining data will be saved in binary format. Data is stored in a compressed file that can be opened like a ZIP archive file. In this way, it is possible to create files for CAD programs.

12.5.3 RF-LINK Import *.step, *.iges, *.sat

With the add-on module RF-LINK (not contained in RFEM) it is possible to import data in STEP, IGES or ACIS format. The file formats are mainly used in mechanical engineering, allowing for a transfer of model geometry in the form of boundary lines and surfaces.

To import model files available in one of the formats mentioned above,

select **Import** on the **File** menu.

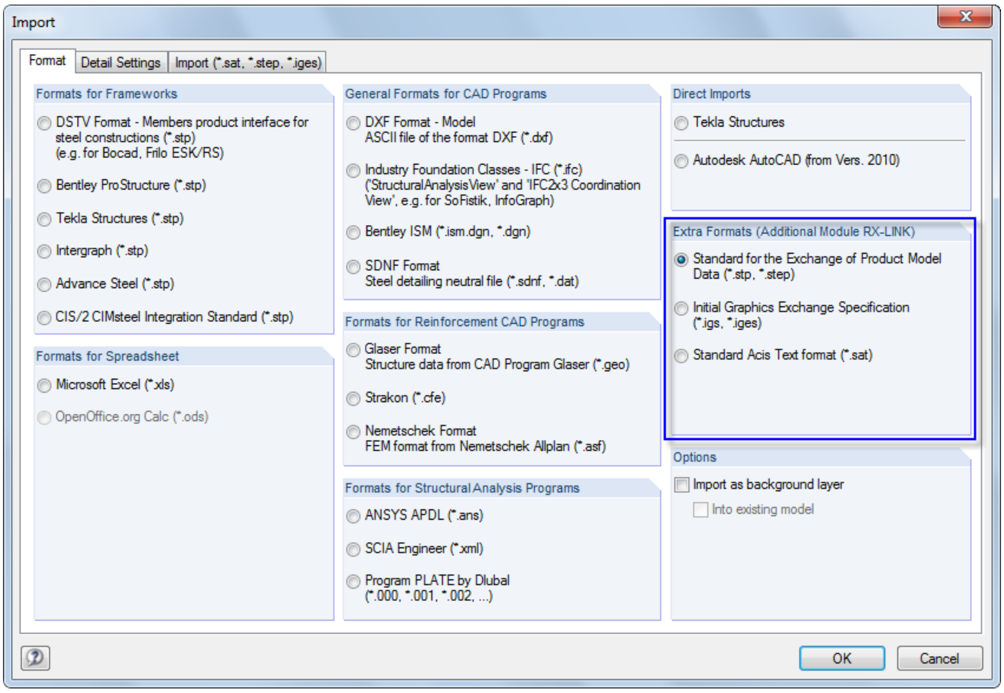


Figure 12.54: Dialog box *Import*

In the dialog section *Extra Formats* of the *Import* dialog box, you can define the relevant file format:

- *Standard for the Exchange of Product Model Data* (*.stp, *.step)
- *Initial Graphics Exchange Specification* (*.igs, *.iges)
- *Standard Acis Text format* (*.sat)

Access to the options is only available when RF-LINK has been installed. The installation requires a separate installation process.

In the dialog tab *Import (*.sat, *.step, *.iges)*, you can specify detailed settings for units and the treatment of lines and surfaces.

Export options of RFEM files in the STEP, IGES or SAT format are currently not available.

A Literature

[1] ZIENKIEWICZ, O. C., CHEUNG, Y.K.: The Finite Element Method in Structural and Continuum Mechanics, McGraw-Hill, New York, London 1967

[2] KOLÁR, V. et al.: Berechnung von Flächen- und Raumtragwerken nach den Methode der finiten Elemente (tschechisch), SMTL Prag 1972

[3] KOLÁR, V. et al.: Berechnung von Flächen- und Raumtragwerken nach den Methode der finiten Elemente, Springer, Wien-New York 1975

[4] KOLÁR, V., NEMEC, I.: Modeling of Soil-Structure Interaction, Elsevier Science Publishers, Amsterdam, co-published with Academica Prague, 1989, second revised edition

[5] STIGLAT, K. WIPPEL, H.: Massive Platten. In: Betonkalender 1989/I, S. 281 ff, Ernst & Sohn, Berlin 1989

[6] CZERNY, F.: Tafeln für Rechteckplatten. In: Betonkalender 1990/I, S. 309 ff, Ernst & Sohn, Berlin 1990

[7] WUNDERLICH, W. et al.: Modellierung und Berechnung von Deckenplatten mit Unterzügen. In: Bauingenieur 69, Heft 10, S. 381-389, Springer-Verlag 1994

[8] PASTERNAK, P.L.: Grundlagen einer neuen Methode der Berechnung von Fundamenten mittels zwei Bettungskoeffizienten, Gos. Isd. Stroj. i Arch., Moskau 1954 (Russian)

[9] KOLÁR, V. et al.: Kurs für Statiker von Gründungsbauwerken und Erdkörpern, S. 146 ff, Haus der Technik, Ostrau 1983 (tschechisch)

[10] TIMOSHENKO, S.P. und WOINOWSKI-KRIEGER, S.: Theory of Plates and Shells, 2. Auflage, McGraw-Hill, New York 1959

[11] GRASSER, E. und THIELEN, G.: Heft 240 DAfSt, Ernst & Sohn, Berlin-München-Düsseldorf, 1978, 2. überarbeitete Auflage

[12] GRASSER, E., KORDINA, K., QUAST, U.: Bemessung von Beton- und Stahlbetonbauteilen nach DIN 1045, DAfStb - Heft 220, Ernst & Sohn, Berlin 1979

[13] KOLÁR, V. - NEMEC, I.: Contact Stress and Settlement in the Structure-Soil Interface. Studie der tschechoslowakischen Akademie der Wissenschaften Nr. 16, Academia Prag 1991, 160 Seiten (englisch)

[14] PETERSEN, Chr.: Stahlbau, Vieweg & Sohn, Braunschweig-Wiesbaden 1988

[15] BARTH, C., RUSTLER, W.: Finite Elemente in der Baustatik-Praxis, Bauwerk, Berlin 2010

[16] NEMEC, I, KOLÁR, V. et al.: Finite Element Analysis of Structures - Principles and Praxis, Aachen 2010

[17] KOLÁR, V. et al.: Kurs für Statiker von Gründungsbauwerken und Erdkörpern, Haus der Technik, Ostrau, 1983 (tschechisch)

[18] KOLÁR, V. et al.: Bemessung von zwei- und dreidimensionalen Strukturen mit FEM, Springer-Verlag, New York/Wien, 1975, S. 425 ff. Kapitel 1 (1D-Element) und 6 (Variationsprinzip)

[19] KOLÁR, V., NEMEC, I.: Finite Element Analysis of Structures. United Nations Development Program, Economic Com. for Europe, Workshop on CAD Techniques, June 1984, Prague-Geneva, Vol. I, 248 pp.

[20] BERGAN, P. G. : Finite Elements Based on Energy Orthogonal Functions. Int. Journal for Numerical Methods in Engineering, 17 (1981), S. 154-155

[21] BERGAN, P.G. - NYGARD, M. K.: Finite Elements With Increased Freedom in Choosing Shape Functions. Int. Journal for Num. Meth. in Eng., 20 (1984), p. 643-664, (Free Formulation Concept)

[22] BERGAN, P.G. - FELIPPA, C. A.: A Triangular Membrane Element With Rotational Degrees of Freedom. Computer Methods in Applied Mechanics and Engineering, 50 (1985), pp. 25-69

[23] ZIENKIEWIC, O.C.: The Finite Element Method in Engineering Science, Mc Graw - Hill, London 3rd Ed., repr. 1979, 787 pp., Chapter 18 - 19 (Nonlinear Problems)

[24] DVORKIN, E.N. - BATHE, K.-J.: A continuum mechanics based four-node shell element for nonlinear analysis. In: Eng. Comput., 1984, vol. 1, pp. 77-88.

[25] BATHE, K.J.: Finite Element Procedures, New Jersey, 1996

[26] BAUMANN, Th.: Zur Frage der Netzbewehrung von Flächentragwerken. In: Der Bauingenieur 47 (1972), S. 36 ff, Springer-Verlag, Berlin 1972

[27] SCHLAICH, J., SCHÄFER, K.: Konstruieren im Stahlbetonbau. In: Betonkalender 1993, Teil II, S. 327 ff, Ernst & Sohn, Berlin 1993

[28] LEONHARDT, F.: Vorlesungen über Massivbau, Teil 6, Springer-Verlag, Berlin-Heidelberg-New York 1979

[29] DIN 1045 (07.88), Beton und Stahlbeton, Bemessung und Ausführung, Beuth, Berlin-Wien-Zürich 1988

[30] Deutscher Ausschuss für Stahlbeton: DIN 1045-1 Tragwerke aus Beton, Stahlbeton und Spannbeton - Teil 1: Bemessung und Konstruktion. Entwurf 12.1998.

[31] DIN 18800 (11.90) Teil 1, Stahlbauten, Bemessung und Konstruktion, Beuth, Berlin-Wien-Zürich 1992

[32] DIN 18800 (11.90) Teil 2, Stahlbauten, Stabilitätsfälle, Knicken von Stäben und Stabwerken, Beuth, Berlin-Wien-Zürich 1992

[33] Eurocode 2 Teil 1-1 (06.92), Planung von Stahlbeton- und Spannbetontragwerken, Beuth, Berlin-Wien-Zürich 1992

[34] Eurocode 3 Teil 1-1 (04.93), Bemessung und Konstruktion von Stahlbauten, Beuth, Berlin-Wien-Zürich 1993

[35] KLINGMÜLLER, O. LAWO, M., THIERAUF, G. (1983), Stabtragwerke, Matrizenmethoden der Statik und Dynamik, Teil 2: Dynamik, Fr. Vieweg & Sohn, Braunschweig

[36] KLOTTER, K. (1981), Technische Schwingungslehre, Bd. 1, Teil A: Lineare Schwingungen, Teil B: Nichtlineare Schwingungen, Bd. 2: Schwinger von mehreren Freiheitsgraden, Springer, Berlin

[37] KOLOUSEK, V. (1962), Dynamik der Baukonstruktionen, VEB-Verlag f. Bauwesen, Berlin

[38] KRÄMER, E. (1984), Maschinendynamik, Springer, Berlin

[39] LEHMANN, T. (1979), Elemente der Mechanik IV: Schwingungen, Variationsprinzipe, Fr. Vieweg & Sohn, Braunschweig

[40] LIPINSKI, J. (1972), Fundamente und Tragkonstruktionen für Maschinen, Bauverlag, Wiesbaden

[41] LORENZ, H. (1960), Grundbau-Dynamik, Springer, Berlin

[42] MÜLLER, F. P. (1978), Baudynamik, Betonkalender 1978, Ernst & Sohn, Berlin

[43] NATKE, H. G. (1989), Baudynamik, B. G. Teubner, Stuttgart

[44] NOWACKI, W. (1974), Baudynamik, Springerg, Berlin

[45] FLESCH, R. (1993), Baudynamik, praxisgerecht, Bauverlag, Wiesbaden-Berlin

[46] MESKOURIS, K. (1999), Baudynamik, Modelle Methoden Praxisbeispiele, Ernst & Sohn, Berlin

[47] BARES, R. A. (1989), Tabellen für die Berechnung von Platten und Wänden STNL, Prag

[48] ŠEVČÍK, I., 3D Finite Elements with Rotational Degrees of Freedom, FEM Consulting s.r.o., Brno

B Index

D

Dlubal