

**Version**  
**April 2013**

**Program**

# **RSTAB 8**

**Structural Analysis of  
General Frameworks**

## **Program Description**

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of DLUBAL ENGINEERING SOFTWARE.

© **Ing.-Software Dlubal**  
**Am Zellweg 2 D-93464 Tiefenbach**

Tel.: +49 9673 9203-0  
Fax: +49 9673 9203-51  
E-mail: [info@dlubal.com](mailto:info@dlubal.com)  
Web: [www.dlubal.com](http://www.dlubal.com)



# Contents

Contents		Page	Contents		Page
<b>1.</b>	<b>Introduction</b>	<b>6</b>	4.10	Member Nonlinearities	93
1.1	New in RSTAB 8	6	4.11	Sets of Members	96
1.2	Program Capacities	7	<b>5.</b>	<b>Load Cases and Combinations</b>	<b>98</b>
1.3	Company Profile	7	5.1	Load Cases	98
1.4	The RSTAB Team	8	5.2	Actions	103
1.5	Using the Manual	9	5.3	Combination Expressions	106
<b>2.</b>	<b>Installation</b>	<b>10</b>	5.4	Action Combinations	116
2.1	System Requirements	10	5.5	Load Combinations	120
2.2	Installation Process	10	5.5.1	User-defined Combinations	121
2.2.1	Installation from DVD	11	5.5.2	Generated Combinations	126
2.2.2	Network Installation	12	5.6	Result Combinations	128
2.2.3	Installing Updates and Other Modules	12	5.6.1	User-defined Combinations	128
2.2.4	Parallel Installation of RSTAB Versions	12	5.6.2	Generated Combinations	134
<b>3.</b>	<b>Graphical User Interface</b>	<b>13</b>	5.7	Combination Scheme	136
3.1	Overview	13	5.8	Super Combinations	137
3.2	Terminology	14	<b>6.</b>	<b>Loads</b>	<b>142</b>
3.3	Special Terms in RSTAB	17	6.1	Nodal Loads	146
3.4	RSTAB User Interface	18	6.2	Member Loads	148
3.4.1	Menu bar	18	6.3	Imposed Nodal Deformations	157
3.4.2	Toolbars	18	6.4	Imperfections	158
3.4.3	Project Navigator	21	6.5	Generated Loads	162
3.4.4	Tables	24	<b>7.</b>	<b>Calculation</b>	<b>163</b>
3.4.5	Status Bar	25	7.1	Checking the Input Data	163
3.4.6	Control Panel	27	7.1.1	Plausibility Check	163
3.4.7	Default Buttons	31	7.1.2	Structure Check	164
3.4.8	Keyboard Functions	32	7.1.3	Regenerate Model	167
3.4.9	Mouse Functions	33	7.1.4	Delete Not Used Loads	167
3.4.10	Configuration Manager	34	7.2	Calculation Parameters	168
<b>4.</b>	<b>Model Data</b>	<b>36</b>	7.2.1	Load Cases and Load Combinations	169
4.1	Nodes	41	7.2.1.1	Dialog Tab <i>Calculation Parameters</i>	169
4.2	Materials	46	7.2.1.2	Dialog Tab <i>Modify Stiffness</i>	173
4.3	Cross-sections	53	7.2.1.3	Dialog Tab <i>Extra Options</i>	174
4.4	Member End Releases	64	7.2.2	Result Combinations	175
4.5	Member Eccentricities	70	7.2.3	Global Calculation Parameters	176
4.6	Member Divisions	72	7.3	Start Calculation	180
4.7	Members	73	<b>8.</b>	<b>Results</b>	<b>184</b>
4.8	Nodal Supports	85	8.0	Results Balance	184
4.9	Member Elastic Foundations	91	8.1	Members - Internal Forces	185

# Contents

Contents		Page	Contents		Page
8.2	Sets of Members - Internal Forces	189	10.1.6	Insert Graphics and Texts	238
8.3	Cross-sections - Internal Forces	190	10.1.7	Printout Report Template	240
8.4	Nodes - Support Forces	191	10.1.8	Adjust Layout	241
8.5	Members - Contact Forces	195	10.1.9	Create Cover	242
8.6	Nodes - Deformations	197	10.1.10	Print the Printout Report	244
8.7	Members - Local Deformations	198	10.1.11	Export Printout Report	244
8.8	Members - Global Deformations	200	10.1.12	Language Settings	246
8.9	Members - Member Coefficients for Buckling	201	10.2	Direct Graphic Printout	248
8.10	Member Slendernesses	202	10.2.1	General	249
<b>9.</b>	<b>Results Evaluation</b>	<b>203</b>	10.2.2	Options	252
9.1	Available Results	203	10.2.3	Color Spectrum	254
9.2	Results Selection	204	10.2.4	Mass Print	255
9.3	Results Display	205	<b>11.</b>	<b>Tools</b>	<b>257</b>
9.4	Info about Member	207	11.1	General Functions	257
9.5	Result Diagrams	208	11.1.1	Language Settings	257
9.5.1	Result diagram	208	11.1.2	Display Properties	258
9.5.2	Smoothing Results	210	11.1.3	Units and Decimal Places	261
9.6	Multiple Windows View	211	11.1.4	Comments	262
9.7	Filter Results	212	11.1.5	Measure Functions	264
9.7.1	Views	212	11.1.6	Search Functions	265
9.7.1.1	Views Navigator	212	11.1.7	Viewpoint and View Angle	266
9.7.1.2	Visibility Buttons and Menu	216	11.1.8	Determination of Centroid	267
9.7.2	Clipping Plane	218	11.1.9	Rendering	268
9.7.3	Filter Functions	220	11.1.10	Lighting	270
9.8	Animation of Deformations	221	11.2	Selection	271
<b>10.</b>	<b>Printout</b>	<b>223</b>	11.2.1	Selecting Objects Graphically	271
10.1	Printout Report	223	11.2.2	Selecting Objects by Criteria	274
10.1.1	Create or Open Printout Report	223	11.3	Work Window	275
10.1.2	Working in the Printout Report	225	11.3.1	Work Planes	275
10.1.3	Define Contents of Printout Report	227	11.3.2	Grid	278
10.1.3.1	Selecting Model Data	228	11.3.3	Object Snap	279
10.1.3.2	Selecting Load Case and Combination Data	229	11.3.4	Coordinate Systems	283
10.1.3.3	Selecting Load Data	230	11.3.5	Dimensions	286
10.1.3.4	Selecting Results Data	231	11.3.6	Comments	288
10.1.3.5	Selecting Data of Add-on Modules	232	11.3.7	Guidelines	290
10.1.4	Adjust Printout Report Header	233	11.3.8	Line Grid	294
10.1.5	Insert RSTAB Graphics	236	11.3.9	Visual Objects	296
			11.3.10	Background Layers	297
			11.3.11	Margins and Stretch Factors	300



# Contents

Contents		Page	Contents		Page
11.4	Edit Functions	300	11.8.3	Other Loads	362
11.4.1	Move and Copy	301	11.8.3.1	Member Loads from Free Line Load	362
11.4.2	Rotate	304	11.8.3.2	Member Loads from Coating	362
11.4.3	Mirror	305	11.8.3.3	Loads from Movements	363
11.4.4	Project	306	11.8.4	Snow Loads	364
11.4.5	Scale	307	11.8.4.1	Flat/Monopitch Roof	364
11.4.6	Shear	309	11.8.4.2	Duopitch Roof	365
11.4.7	Divide Members	310	11.8.5	Wind Loads	366
11.4.8	Connect Members	312	11.8.5.1	Vertical Walls	366
11.4.9	Merge Members	313	11.8.5.2	Flat Roof	368
11.4.10	Extend Members	313	11.8.5.3	Monopitch Roof	369
11.4.11	Join Members	314	11.8.5.4	Duopitch/Troughed Roof	370
11.4.12	Insert a Node	315	11.8.5.5	Vertical Walls with Roof	372
11.4.13	Insert a Member	316	<b>12.</b>	<b>File Management</b>	<b>373</b>
11.4.14	Assign Member Properties Graphically	317	12.1	Project Manager	373
11.4.15	Round Corners	318	12.1.1	Project Management	375
11.4.16	Change Numbering	318	12.1.2	Model Management	379
11.5	Table Functions	321	12.1.3	Data Backup	381
11.5.1	Editing Functions	321	12.1.4	Settings	383
11.5.2	Selection Functions	323	12.1.4.1	View	383
11.5.3	View Functions	325	12.1.4.2	Recycle Bin	384
11.5.4	Table Settings	327	12.1.4.3	Directories	385
11.5.5	Filter Functions	328	12.2	Creating a New Model	386
11.5.6	Import and Export of Tables	329	12.2.1	General	387
11.6	Parameterized Input	332	12.2.2	Options	391
11.6.1	Concept	332	12.2.3	History	392
11.6.2	Parameter List	332	12.3	Network Management	393
11.6.3	Formula Editor	335	12.4	Block Manager	394
11.6.4	Formulae in Tables and Dialog Boxes	338	12.4.1	Create a Block	395
11.7	Model Generators	339	12.4.2	Import a Block	396
11.7.1	Copies and Extrusions	339	12.4.3	Delete a Block	398
11.7.1.1	Set Parallel Member	339	12.5	Interfaces	399
11.7.1.2	Extrude Member into Grid	340	12.5.1	Direct Data Exchange	399
11.7.2	Model Generators	341	12.5.2	File Formats for Data Exchange	400
11.8	Load Generators	354	12.5.3	RF-LINK Import *.step, *.iges, *.sat	406
11.8.1	General Features	354	<b>A</b>	<b>Index</b>	<b>407</b>
11.8.2	Member Loads from Area Loads	357			
11.8.2.1	Member Loads From Area Load via Plane	357			
11.8.2.2	Member Loads From Area Load via Cells	361			

# 1. Introduction

## 1.1 New in RSTAB 8

With RSTAB, Dlubal's structural analysis program for spatial frameworks (German: *Räumliche STABwerke*), you've got a powerful tool to cope with a variety of different tasks in civil engineering. The program represents the basis for Dlubal's analysis software composed of various design modules: RSTAB determines internal forces, deformations and support reactions of general framework structures. Then, the results can be used in add-on modules for specific designs and other analyses.

The program version RSTAB 8 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 RSTAB 8 listed below:

- Graphical user interface in French, Italian, Polish, Portuguese, Russian, Spanish
- 32-bit and 64-bit program versions
- Usage of template models
- Member eccentricities from cross-section dimensions
- Input option for hybrid timber cross-sections
- Filter in cross-section library with favorites
- Inserting a member to existing member
- Adjustable stiffnesses for cross-sections and members
- Shifting/copying in user-defined coordinate system
- Import of files from Bentley ISM, SEMA, cadwork and Scia Engineer
- Import of 3D objects
- Graphical assignment of member properties
- Color symbols in tables for cross-sections and member types
- Input of inclination and precamber in absolute values
- Member loads for pipe content full/partly filled
- Load creation from multilayer structures such as ceiling and floor structures
- Selection with ellipse, annulus or intersection line
- Work planes defined by three points or surface axes
- Grid lines in work plane
- Color management for cross-section and member types as well as visibilities
- Automatic creation of load and result combinations according to standard specification
- Output of member coefficients and member slendernesses
- 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 RSTAB 8.

Your team from DLUBAL ENGINEERING SOFTWARE

## 1.2 Program Capacities

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

### Model data

99,999 objects of each category (nodes, members, 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 (non-linear calculation)	9,999
Result combinations	9,999
Super combinations	9,999

Table 1.1: Program limits of RSTAB

## 1.3 Company Profile

Since its beginnings in 1987, DLUBAL ENGINEERING SOFTWARE has been involved in the development of user-friendly and powerful 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 7000 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 The RSTAB Team

The following people were involved in the development of RSTAB 8:

### Program coordination

Dipl.-Ing. Georg Dlupal  
 Ing. Pavel Bartoš  
 Ing. Pavol Červeňák  
 Dipl.-Ing. (FH) Matthias Entenmann

Dipl.-Ing. (FH) Younes El Frem  
 M.Eng. Dipl.-Ing. (BA) Andreas Niemeier  
 M.Eng. Dipl.-Ing. (FH) Walter Rustler

### Programming

Ing. Radek Brettschneider  
 Ing. Michal Búzik  
 Dipl.-Ing. Georg Dlupal  
 Jan Fenár  
 Ing. Jan Gregor  
 Ing. Jiří Kubiček  
 MSc. Olga Melnikova  
 Ing. Jan Miléř  
 Ing. Daniel Molnár  
 Ing. Pavel Němeček  
 Ing. Petr Novák  
 Ing. Jan Otradovec  
 Mgr. Petr Oulehle  
 Mgr. Jiří Patrák

Mgr. Andor Patho  
 Mgr. Petr Pitka  
 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

Dr.-Ing. Jaroslav Lain  
 Ing. Martin Budáč

Dipl.-Ing. Georg Dlupal

### Program design, dialog figures, icons

Dipl.-Ing. Georg Dlupal  
 MgA. Robert Kolouch

Zdeněk Ballák  
 Ing. Jan Miléř

### Blocks

Ing. Tommy Brtek  
 Ing. Dmitry Bystrov

Ing. Evžen Haluzík

### Program supervision

Ing. Alexandra Bayrak  
 Ing. Tommy Brtek  
 Ing. Ondřej Čížek  
 Ing. Tomáš Ferencz  
 Ing. Vladimír Gajdoš  
 Ing. Jakub Harazín  
 Ing. Martin Hlavačká  
 Ing. Iva Horčíčková  
 Karel Kolář  
 Ing. František Knobloch

Ing. Ctirad Martinec  
 Ing. 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

### Localization, manual

Msc. Eliška Bartůňková  
 Ing. Fabio Borriello  
 Ing. Dmitry Bystrov  
 Eng.º Rafael Duarte  
 Ing. Jana Duníková  
 Ing. Lara Freyer  
 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  
 Ing. Petr Míchal  
 MA SKT Anton Mitleider  
 Dipl.-Ü. Gundel Pietzcker  
 Mgr. Petra Pokorná  
 Ing. Zoja Rendlová  
 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  
 Dipl.-Ing. (FH) Steffen Clauß  
 Dipl.-Ing. (FH) Matthias Entenmann  
 Dipl.-Ing. Frank Faulstich  
 Dipl.-Ing. (FH) René Flori  
 Dipl.-Ing. (FH) Stefan Frenzel  
 Dipl.-Ing. (FH) Walter Fröhlich  
 Dipl.-Ing. Wieland Götzler  
 Dipl.-Ing. (FH) Andreas Hörold  
 Dipl.-Ing. (FH) Paul Kieloch

Dipl.-Ing. (FH) Bastian Kuhn  
 Dipl.-Ing. (FH) Ulrich Lex  
 M.Sc. Dipl.-Ing. (FH) Frank Lobisch  
 Dipl.-Ing. (BA) Sandy Matula  
 Dipl.-Ing. (FH) Alexander Meierhofer  
 M.Eng. Dipl.-Ing. (BA) Andreas Niemeier  
 M.Eng. Dipl.-Ing. (FH) Walter Rustler  
 M.Sc. Dipl.-Ing. (FH) Frank Sonntag  
 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 RSTAB: 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](http://www.dlubal.com/downloading-manuals.aspx). In this way, you can get quickly familiar with the most important features of RSTAB. The example can also be performed within the restrictions of the demo version.



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 don't find what you are looking for, please check our website [www.dlubal.com](http://www.dlubal.com) where you can go through our *FAQ* pages by selecting particular criteria.

## 2. Installation

### 2.1 System Requirements

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

- Operating system Windows XP/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.



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

No product recommendations are made, with the exception of the operating system, as RSTAB basically runs on all systems that fulfill the system requirements mentioned above. If RSTAB 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, which can slow down the computer significantly. Therefore, upgrading the main memory will usually speed up the calculation more than a faster processor.



As the RSTAB analysis core supports several processor kernels, you can completely exploit the potential of 64-bit operating systems. 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, it is possible to calculate also big models.

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

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

### 2.2 Installation Process

The **RSTAB** program family is delivered on DVD. In addition to the main program RSTAB, the DVD contains all additional modules that belong to the RSTAB program family, for example **STEEL EC3**, **TIMBER Pro**, **RSBUCK** etc.

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

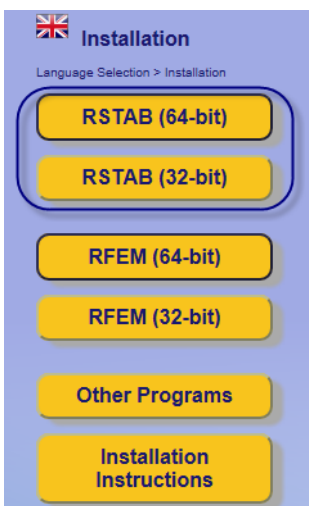


Please make sure that you are logged on as administrator or to have administrator rights for installing the programs. When working with RSTAB later, user rights will be sufficient. Please find detailed instructions shown in the [User Rights video](#) available on our website .

## 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 doesn't start, the *autorun* function may be inactive. In this case, start the file *setup.exe* on 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).
- Select the language in the start dialog box.



Select installation



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*.

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 RSTAB manual in PDF format. To look at the manual, you need the Acrobat Reader that you can install from the DVD.

### RSTAB 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 RSTAB as a full version or a trial version 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](http://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 required. However, if the content will be changed, it cannot be used anymore for authorization.

It is also possible to run the RSTAB full version with a *softlock* 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 *autostart.exe* from the client. The following steps do not differ from the DVD installation.

#### Network licenses

First, 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 another 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 RSTAB.

Old program files are removed and replaced by new ones while installing the update. Of course, your project data is preserved!



If you use printout report headers that you have defined yourself, save them before installing the update. The headers are normally stored in the file **DlubalProtocolConfig.cfg** that you find in the general master data folder *C:\ProgramData\Dlubal\Stammdat*. 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 an update. They are stored in the file **RSTABProtocolConfig.cfg** in the folder *C:\ProgramData\Dlubal\RSTAB 8.01\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\ProMan* (see Figure 12.21, page 385). If you want to uninstall RSTAB before installing the update, you should save this file, too.

### 2.2.4 Parallel Installation of RSTAB Versions

The Dlubal applications RSTAB 6, RSTAB 7 and RSTAB 8 can be run parallel on the computer since the program files are stored in different directories. The default folders of these program generations are the following for a 64-bit operating system:

- RSTAB 6: C:\Program Files (x86)\Dlubal\RSTAB6
- RSTAB 7: C:\Program Files (x86)\Dlubal\RSTAB7
- RSTAB 8: C:\Program Files\Dlubal\RSTAB 8.01

All models created with the previous version RSTAB 7 can be opened and edited in RSTAB 8.

Models from RSTAB 7 won't be overwritten when saving them in RSTAB 8 as both programs use different file endings: RSTAB 7 saves model data in the format **\*.rs7**, RSTAB 8 in **\*.rs8**.

Model files of RSTAB 8 are downward compatible with certain restrictions. When you open an RSTAB 8 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 RSTAB, your screen should look like shown in Figure 3.1. The graphical user interface corresponds to usual Windows standards.

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

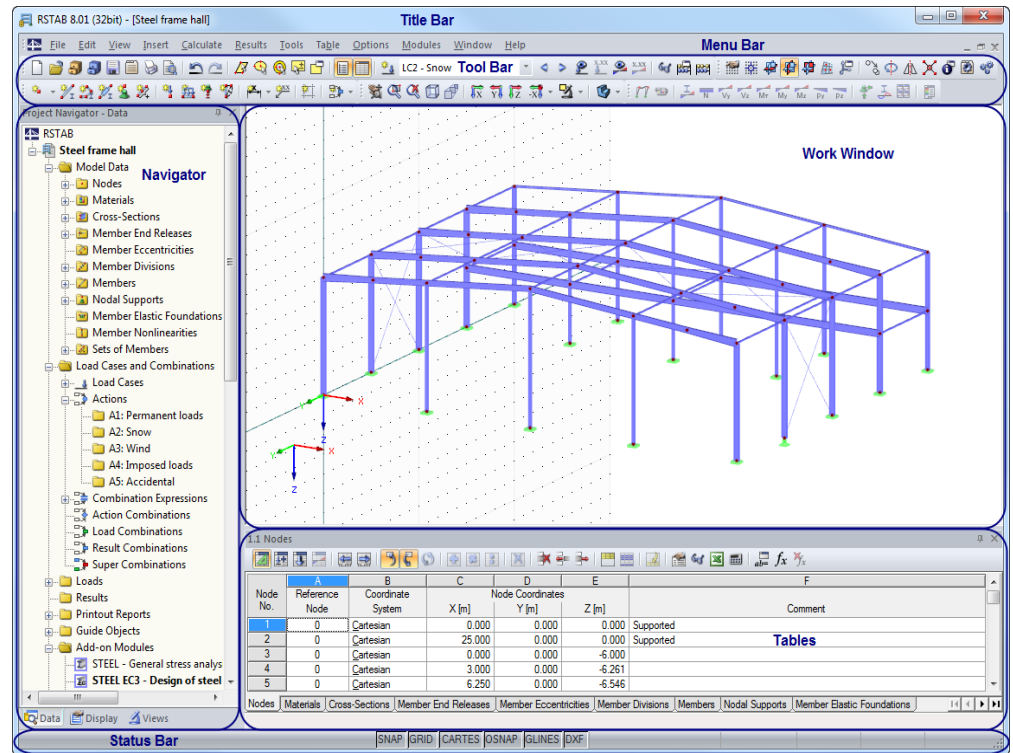


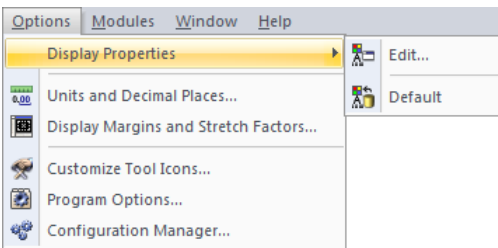
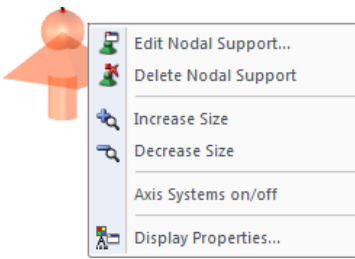

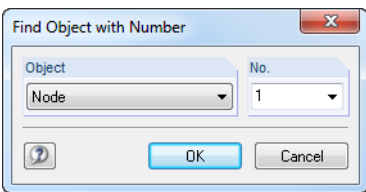
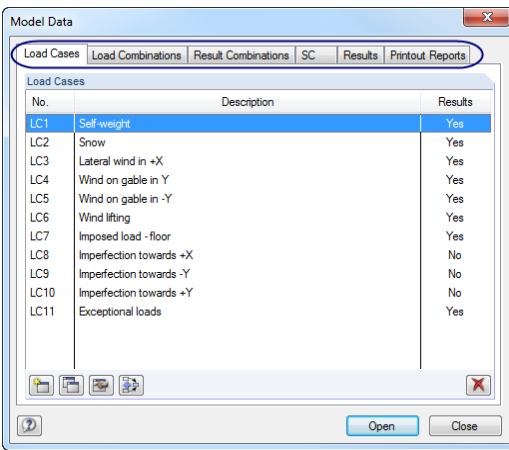
Figure 3.1: RSTAB graphical user interface

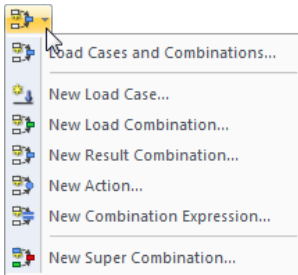
### 3.2 Terminology

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

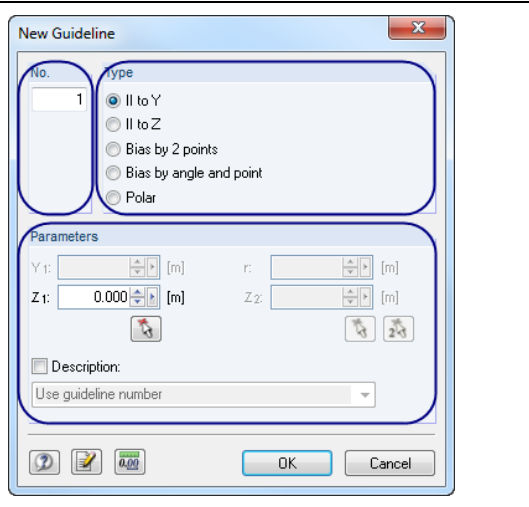
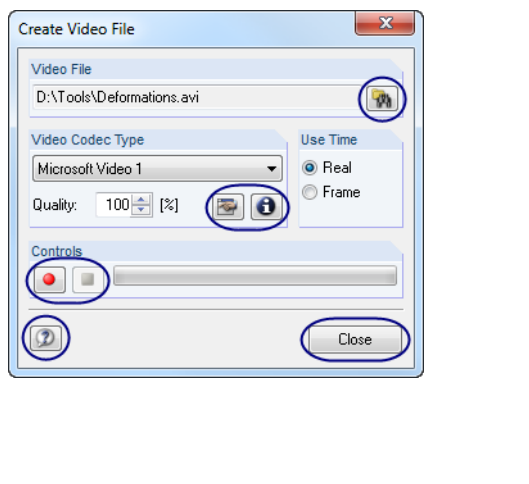
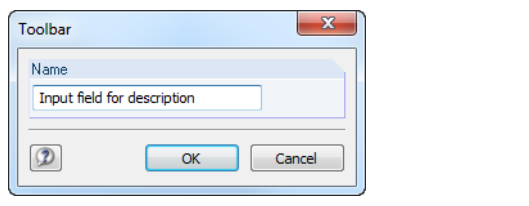
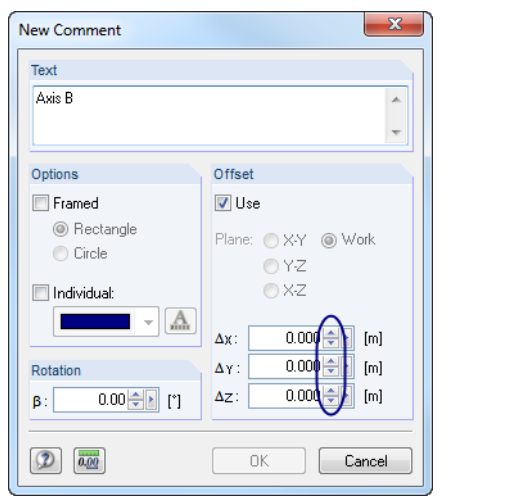
Different terms are used to describe the elements of the user interface. This manual uses English expressions often referring to the *Microsoft Manual of Style for Technical Publications*. Some terms are summarized if their differentiation is not essential for the operation of RSTAB.

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 <b>right</b> 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

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

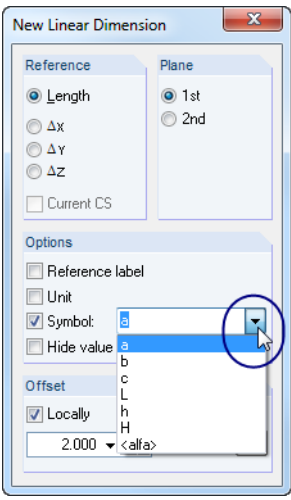
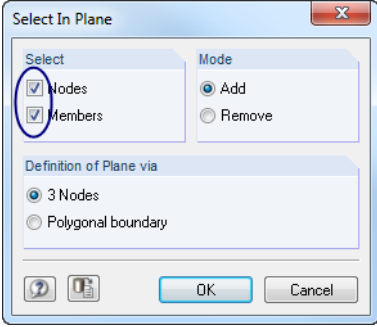
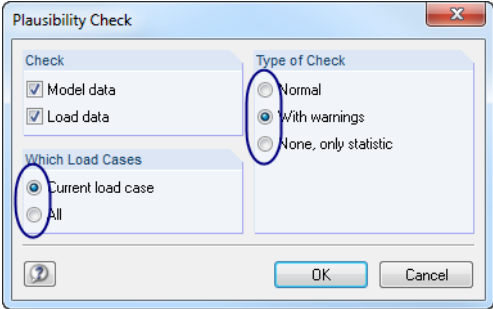
List		List box, combo box	Options for input fields  Sometimes user-defined specifications can be added.
Check box		Check box	Yes/No decision by ticking or clearing the check box
Selection field		Option button	Only one of the options can be selected.

Table 3.1: Terms of user interface

### 3.3 Special Terms in RSTAB

This chapter explains some important terms specific to RSTAB.

Term	Explanation
Nodes	In the 3D model, a node is defined by its coordinates (X/Y/Z). The geometry of the model is defined by nodes.
Member	The straight connection between two nodes represents a member. 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. <b>Continuous members</b> join members continuously like in a continuous beam. A <b>group of members</b> , consisting of connected members, can join more than two members on a single node.
Nodal support	The degrees of freedom are limited for the node.
Nodal load	Force or moment applied to a node.
Member load	A member is stressed by a linear or single load. The load distribution can be either uniform, linearly variable or parabolic.  In addition to forces and moments, temperature actions and prestresses are possible.
Load case LC	The loads of an action are managed in a load case, for example 'self-weight' or 'wind'.  The loads should be defined as characteristic loads (which means 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 <b>loads</b> 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 <b>results</b> 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 <i>Or</i> combination.  However, the additive principle of superposition does not apply for results calculated according to second-order analysis.
Super combination SC	A super combination superimposes the <b>results</b> of load cases, load combinations or result combinations from different RSTAB models. Super combinations can be used to analyze different stages of construction.  The add-on module SUPER-RC is required.

Table 3.2: RSTAB-specific terms

## 3.4 RSTAB User Interface

This chapter describes the individual control elements of RSTAB (see Figure 3.1, page 13). The program follows the general standards for Windows applications.

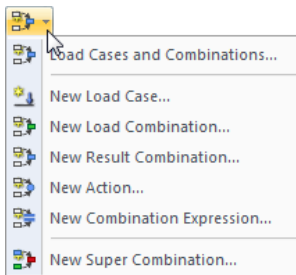
### 3.4.1 Menu bar

Below the title bar you see the menu bar. All functions of RSTAB 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 Window standards. Use shortcuts to start the functions directly with keyboard keys (for example [Ctrl] + [S] saves data).



List button of toolbar

### 3.4.2 Toolbars

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.

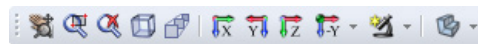


Figure 3.2: Docked position of View toolbar

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 (see Figure 3.4). Customizing toolbars follows Windows standards.

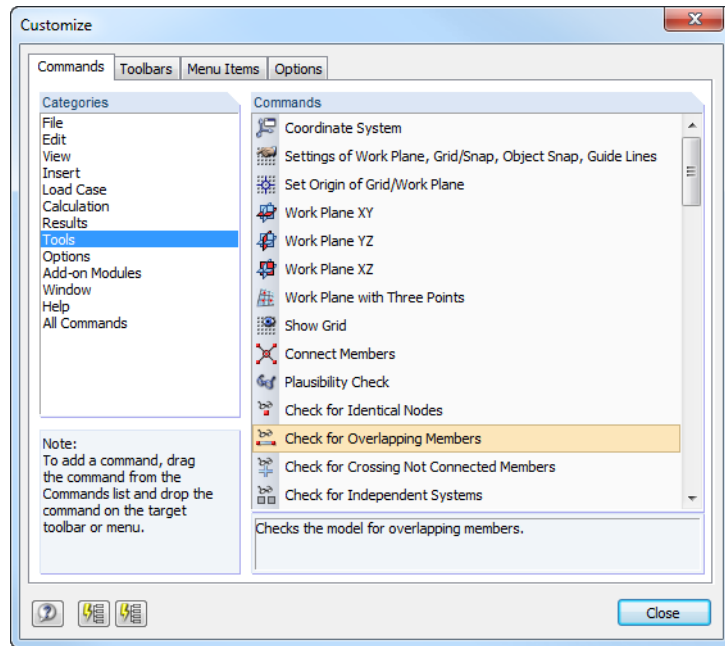


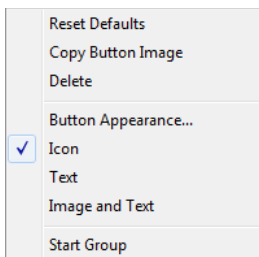
Figure 3.4: *Customize* dialog box, tab *Commands*

All commands of RSTAB 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 place such additional buttons in a new toolbar (see Figure 3.6) because the remaining toolbars may be reset to the default entries when an update is done.

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:



Context menu of a button or menu item

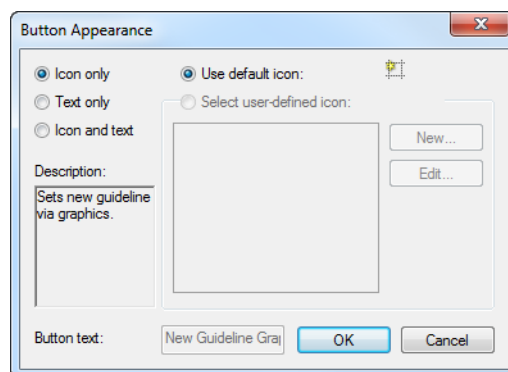


Figure 3.5: Dialog box *Button Appearance*

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.

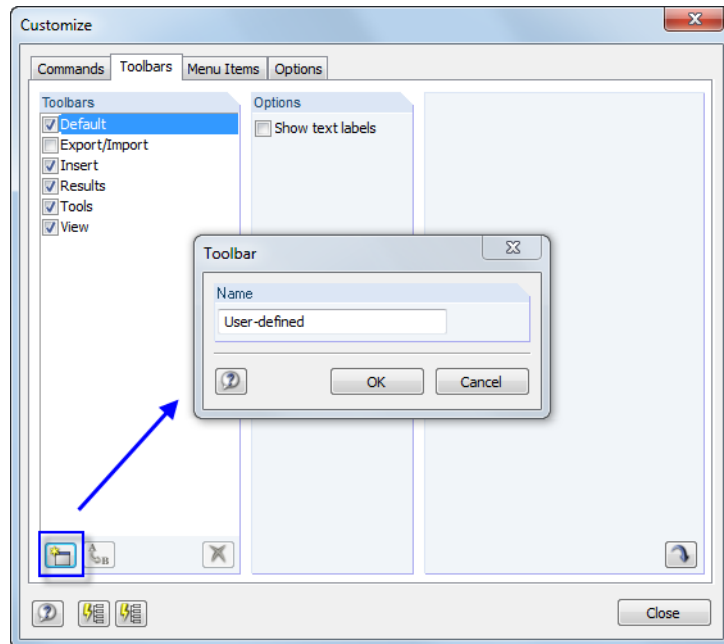


Figure 3.6: Creating a new toolbar

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 RSTAB 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 RSTAB user interface. The following *Designs* can be selected:

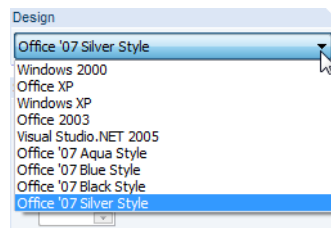


Figure 3.7: Available designs for the user interface

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.

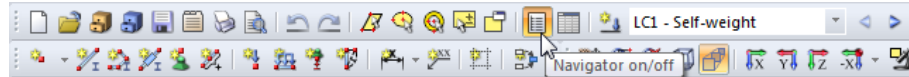


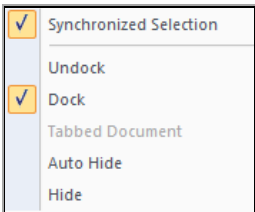
Figure 3.8: Navigator button in the *Default* toolbar

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.

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.



Context menu of navigator

When the menu item *Synchronized Selection* is ticked, RSTAB shows an object that is selected in the navigator also highlighted with colors 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 22).

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

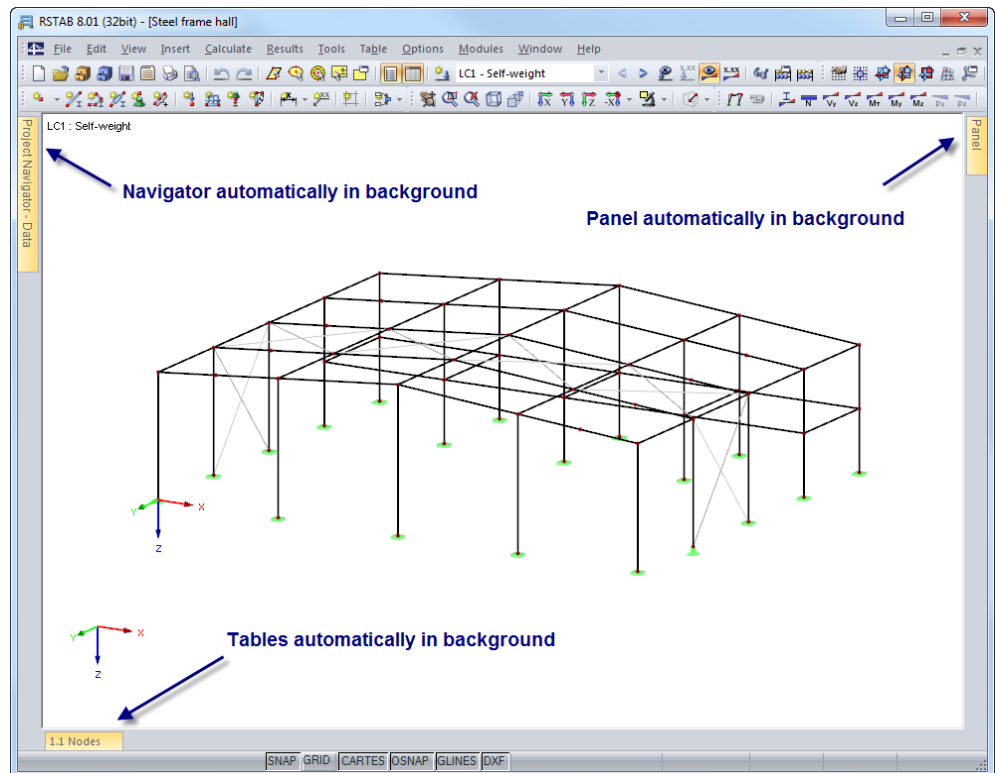


Figure 3.9: Navigator, tables and panel in auto hide mode

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.

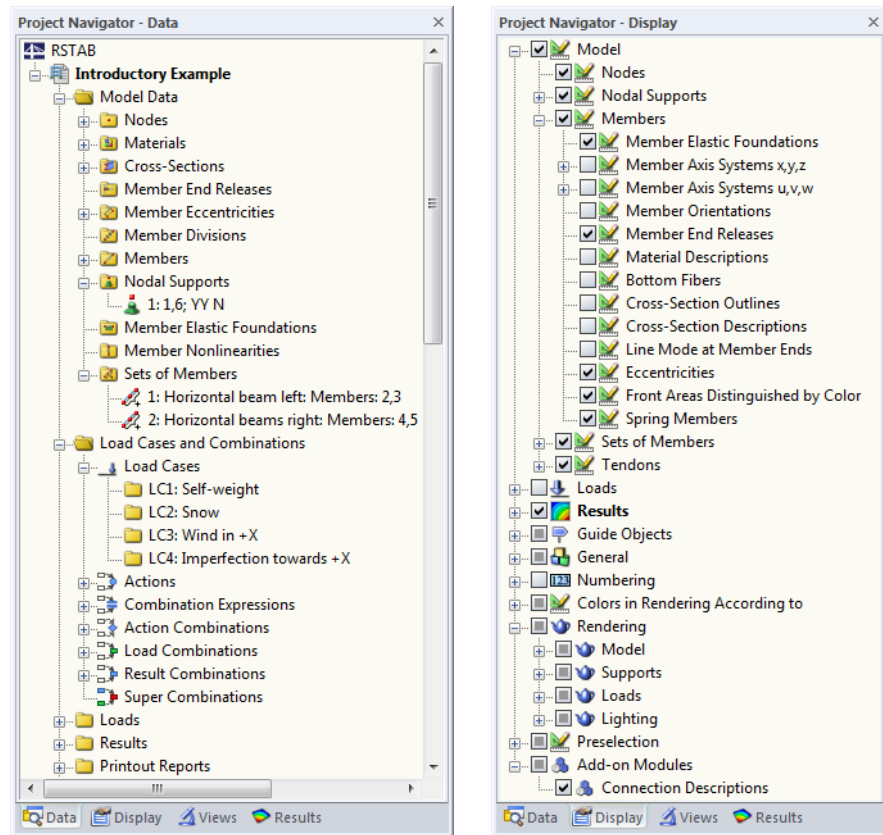
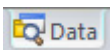


Figure 3.10: Tabs for *Data* and *Display* in Project Navigator

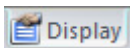
#### **Data navigator**



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.

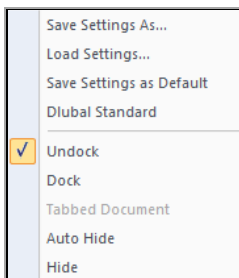
Incorrectly defined objects are displayed in red, unused objects are displayed in blue letters.

#### **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.



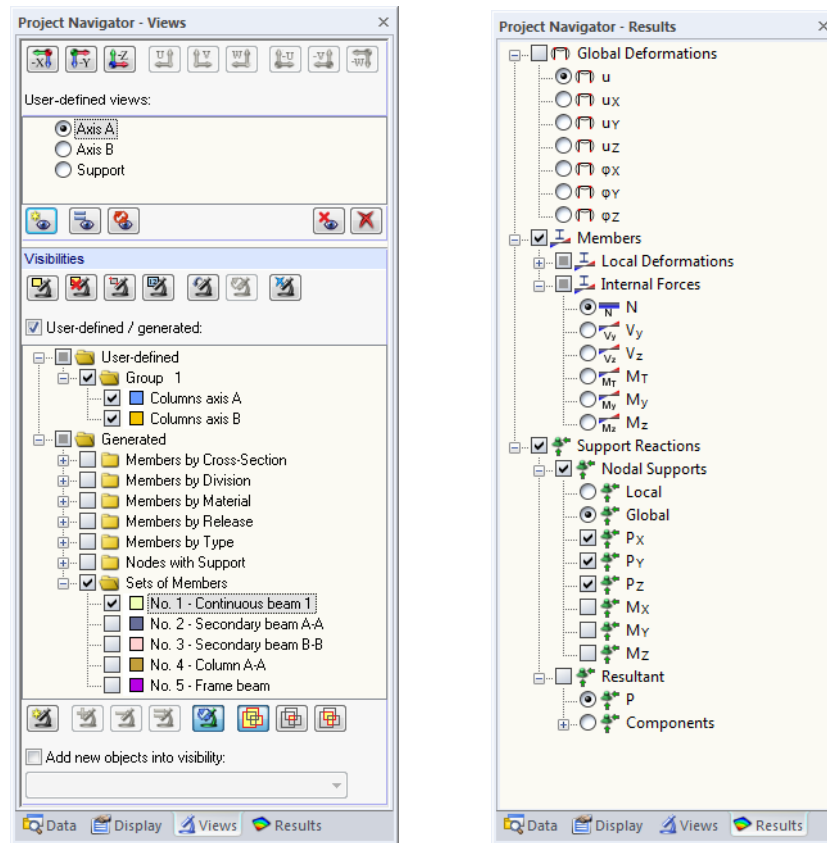


Figure 3.11: Tabs for Views and Results in Project Navigator

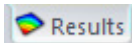
#### Views navigator



This navigator manages user-defined views as well as user-defined and automatically created visibilities of objects (used to be "partial views" and "groups" in RSTAB 7). 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 9.7.1 on page 212.

#### Results navigator



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

#### 3.4.4 Tables



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

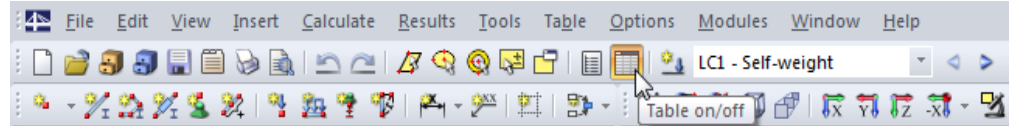
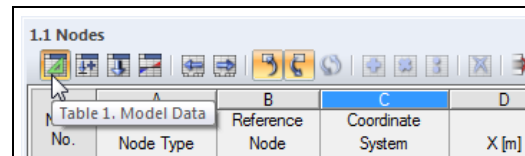


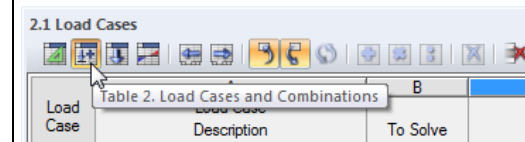
Figure 3.12: Button *Table on/off* in the *Default* toolbar

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.



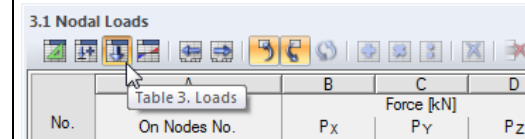
Tables for model data

**Table menu → Go To → Model Data**



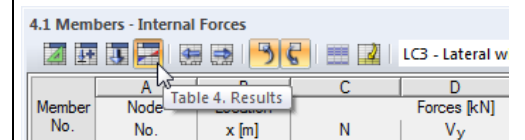
Tables for load cases and combinations

**Table menu → Go To → Load Cases and Combinations**



Tables for loads

**Table menu → Go To → Loads**



Tables for results

**Table menu → Go To → 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 321).

By working through the tables successively RSTAB ensures that all data is captured. The tables represent the internal organization of RSTAB data. 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 21). 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 "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 327).

#### 3.4.5 Status Bar

At the bottom of the RSTAB work window you see the status bar.

On the **View** menu, click **Status Bar** to switch the bar on and off.

The status bar consists of three areas.

##### Left area

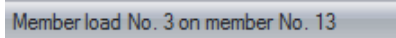


Figure 3.13: Left area of status bar

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 RSTAB 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

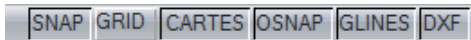
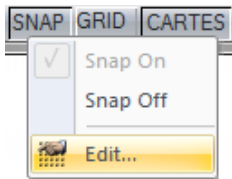


Figure 3.14: Center area of status bar

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 278).



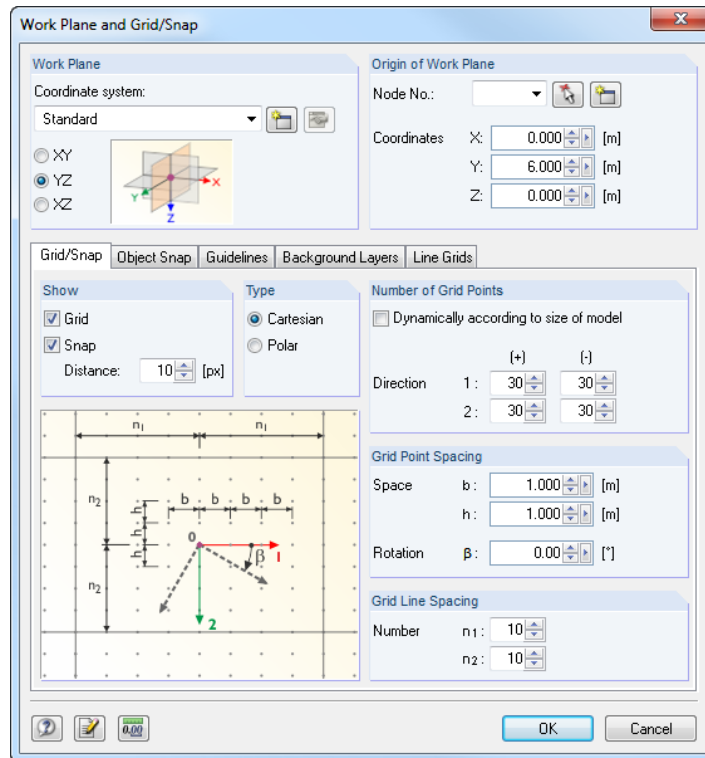


Figure 3.15: Dialog box *Work Plane and Grid/Snap*

#### 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 279).

#### GLINES

The button controls the display of guidelines (see chapter 11.3.7, page 290).

#### DXF

This button switches the display of background layers on and off (see chapter 11.3.10, page 297).

#### Right area

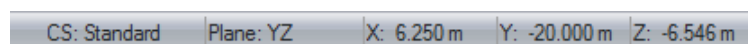
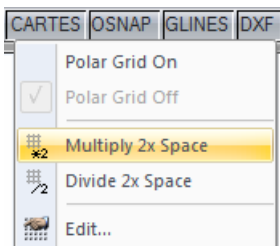
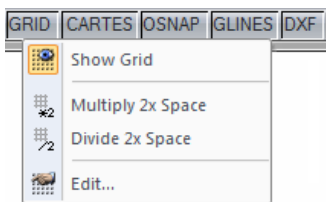


Figure 3.16: Right area of status bar

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 21). 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* and *Filter*.

#### Color scale

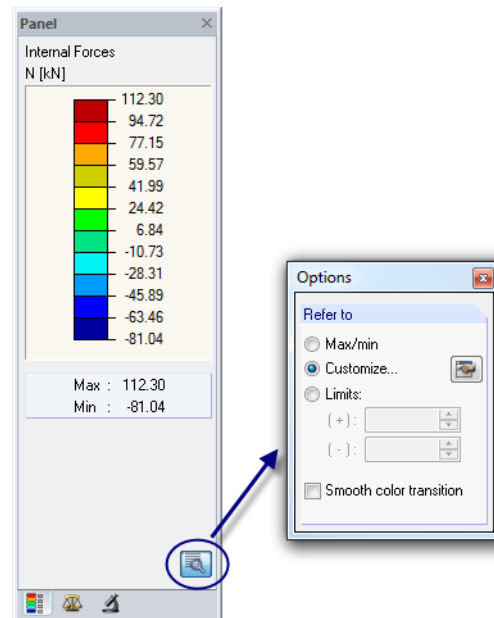


Figure 3.17: Control panel, tab *Color scale* with active *Options* dialog box

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.

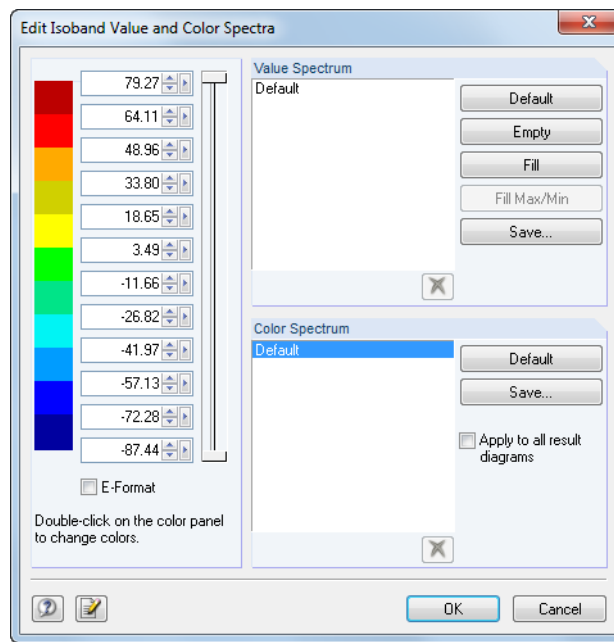


Figure 3.18: Dialog box *Edit Isoband Value and Color Spectra*

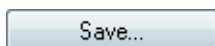
Use the vertical sliders to the right of the values to reduce the number of color ranges at both ends of the color spectrum.

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
Standard	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.



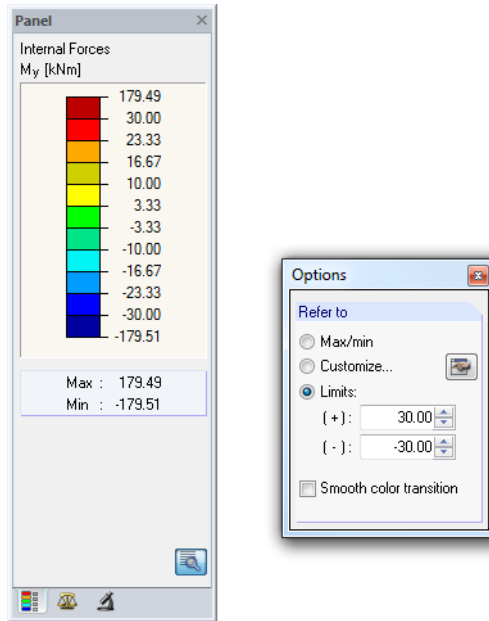


Figure 3.19: Dialog box *Options*, option *Limits +/-*

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  $\pm 30$  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

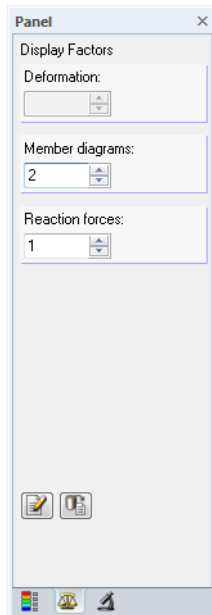


Figure 3.20: Control panel, tab *Factors*

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 *Deformations*, *Member diagrams* (internal forces) and *Reaction forces*.

#### Filter



Figure 3.21: Control panel, tab *Filter*

With the *Color scale* tab you can filter result values in general. Use the *Filter* tab to control the results display with regard to particular members.



You have to enter the numbers of the relevant members 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 (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: For example, when you restrict the results display in the panel to two members, table 4.1 *Members - Internal Forces* will also 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	Description	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 262
	Units and Decimal Places	Settings for units and decimal places → chapter 11.1.3, page 261
	Standard	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 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 whole 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]	<i>Redo</i> 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]	<i>Undo</i> 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 11.52, page 287).

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, the tables and the navigator.

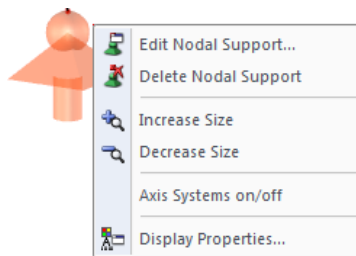


Figure 3.22: Context menu of nodal support in the graphic



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.

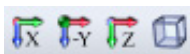


Press the wheel button to move the model directly within the workspace, which means without previously activating the toolbar button [Move, Zoom]. When you press the [Ctrl] key additionally, you can rotate the model. Rotating the model is also possible by using the wheel button and holding down the right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



To rotate the view about 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 RSTAB.



Furthermore, RSTAB 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.

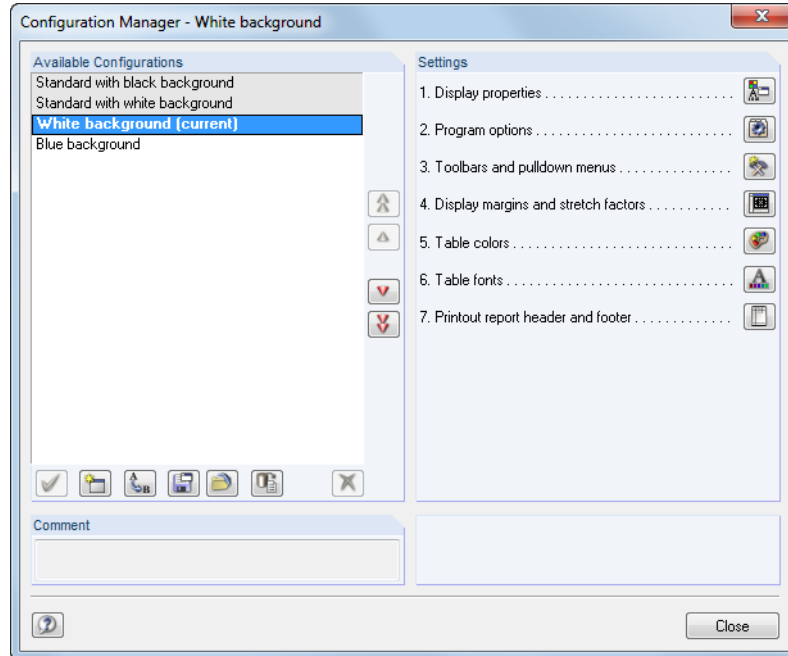


Figure 3.23: Dialog box *Configuration Manager*

#### 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 are reserved for the following functions:








Button	Function
	Sets the entry selected above as new <i>current</i> configuration
	Creates a new configuration from current settings (see 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 <i>Standard</i> and <i>current</i> )

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.

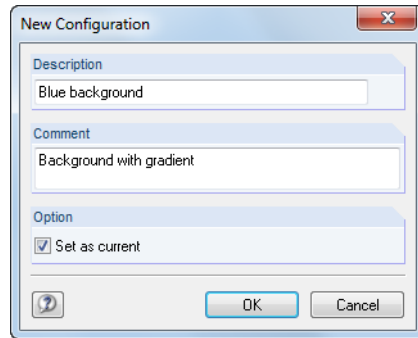


Figure 3.24: Dialog box *New Configuration*

### 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.








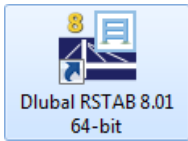
Button	Description	Function
	Display properties	Opens the dialog box <i>Display Properties</i> → chapter 11.1.2, page 258
	Program options	Opens the multi-tab dialog box <i>Program Options</i> → chapter 7.2.3, page 183 → chapter 9.8, page 222 → chapter 11.1.1, page 257 → chapter 11.1.4, page 263
	Toolbars and menus	Opens the dialog box <i>Customize</i> → chapter 3.4.2, page 19
	Margins and stretch factors	Opens the dialog box <i>Display Margins and Stretch Factors</i> → chapter 11.3.11, page 300
	Table colors	Opens the dialog box <i>Colors</i> for the table colors → chapter 11.5.4, page 327
	Table fonts	Opens the dialog box <i>Font</i> for the table fonts → chapter 11.5.4, page 327
	Header and footer of printout report	Opens the dialog box <i>Printout Report Header</i> → chapter 10.1.4, page 233

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

# 4. Model Data



## Starting RSTAB

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 386).

RSTAB 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 RSTAB 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](http://www.dlubal.com/downloading-manuals.aspx)

## Open the input dialog box

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

### Menu Insert

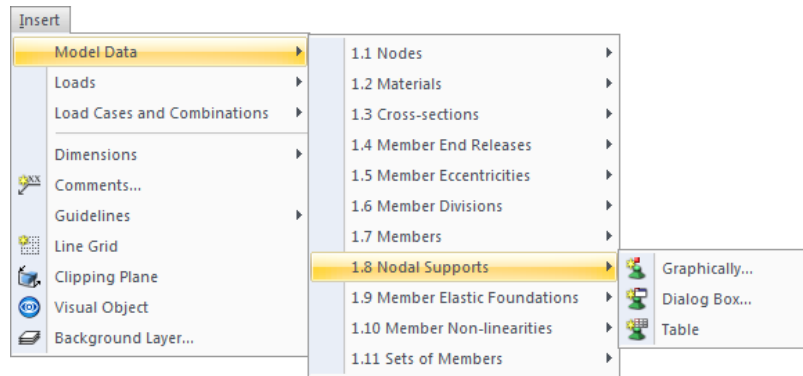


Figure 4.1: Menu Insert → Model Data

### Toolbar Insert



Figure 4.2: Toolbar Insert



Context menu in *Data* navigator

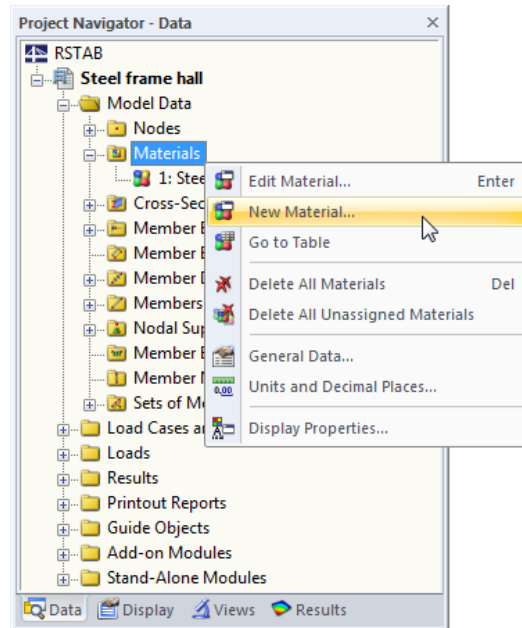


Figure 4.3: Context menu of model data objects in the *Data* navigator

Context menu or double-click in table

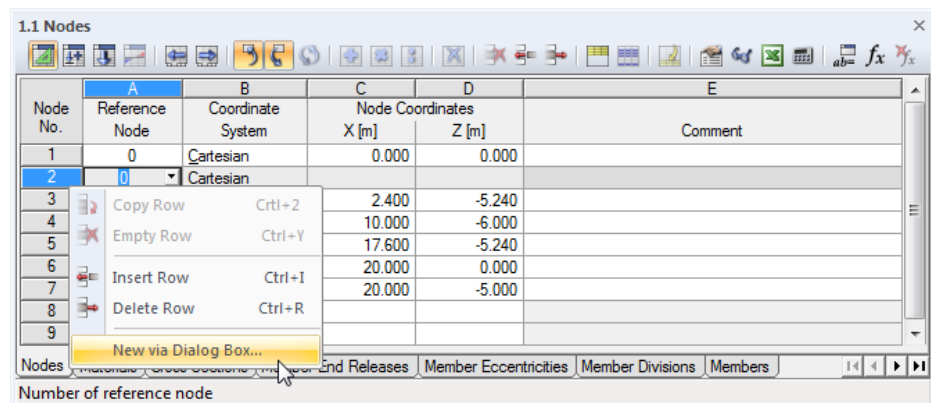


Figure 4.4: Context menu in model data tables

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

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

#### Menu *Edit*

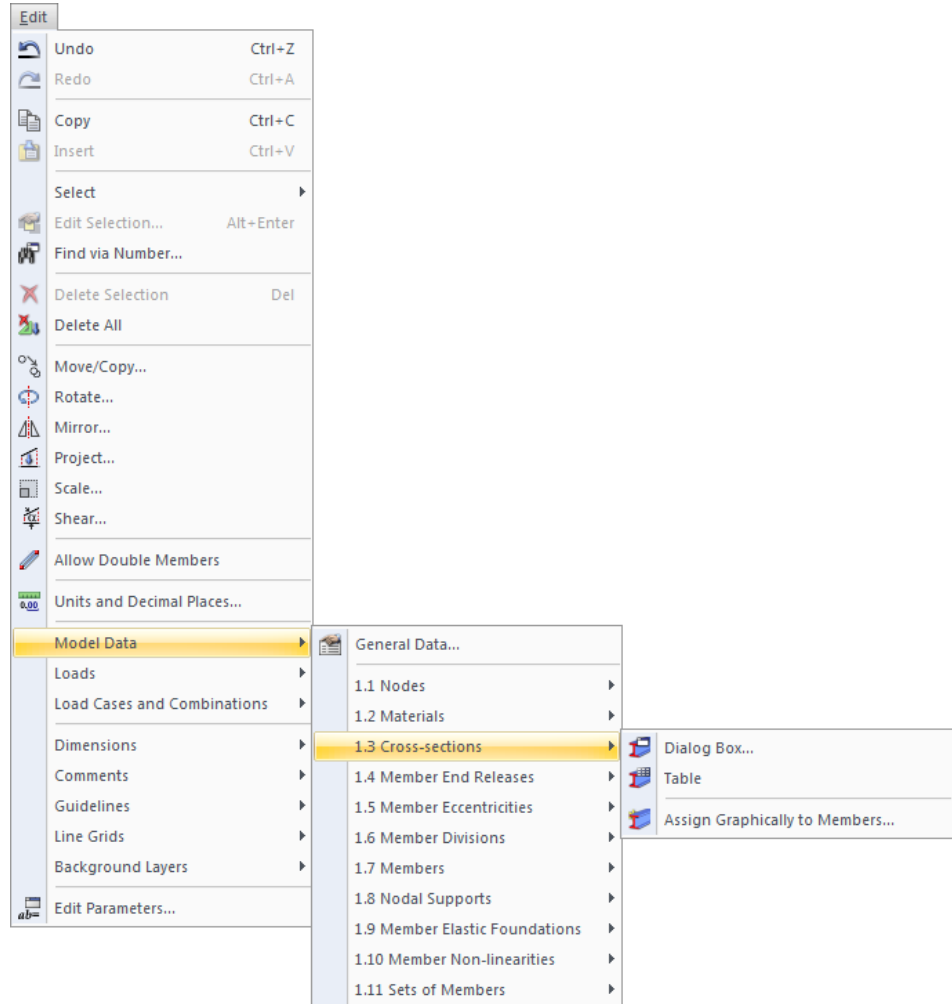


Figure 4.5: Menu *Edit* → *Model Data*

### Context menu or double-click in graphic

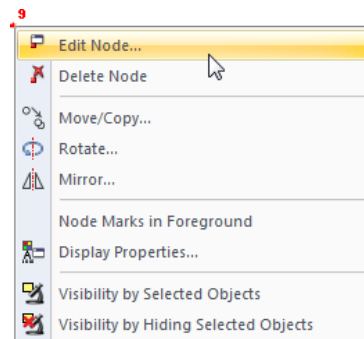


Figure 4.6: Context menu of a node in the work window

Context menu or double-click in *Data* navigator

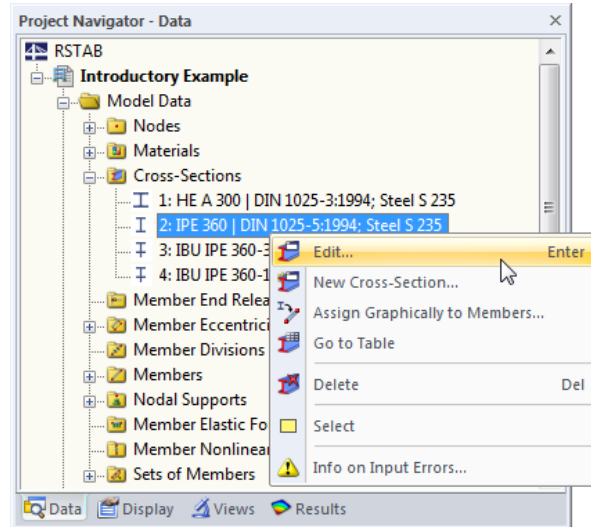


Figure 4.7: Context menu of model data objects in the *Data* navigator

Context menu or double-click in table

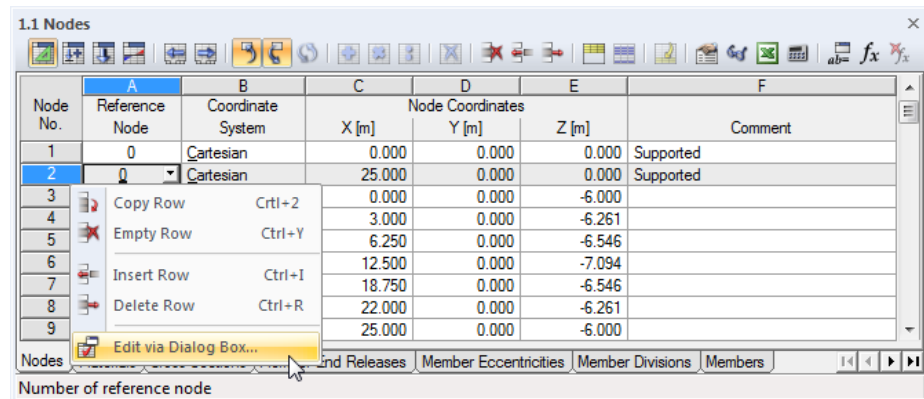


Figure 4.8: Context menu in model data tables

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.

Node No.	Node	Coordinate System	Node Coordinates		Comment
			X [m]	Z [m]	
1	0	Cartesian	0.000	0.000	
2	0	Cartesian	0.000	-5.000	Supported
3	0	Cartesian	2.400	-5.240	
4	0	Cartesian	10.000	-6.000	
5	0	Cartesian	17.600	-5.240	
6	0	Cartesian	20.000	0.000	
7	0	Cartesian	20.000	-5.000	
8					
9					

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 321).



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 262). Moreover, comments are part of ScreenTips for graphical objects.

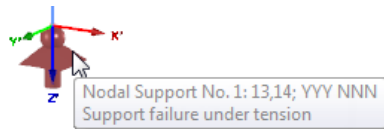


Figure 4.10: ScreenTip of a nodal support

## 4.1 Nodes

### General description

The geometry of the model is defined by nodes. They are essential for creating members. 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.

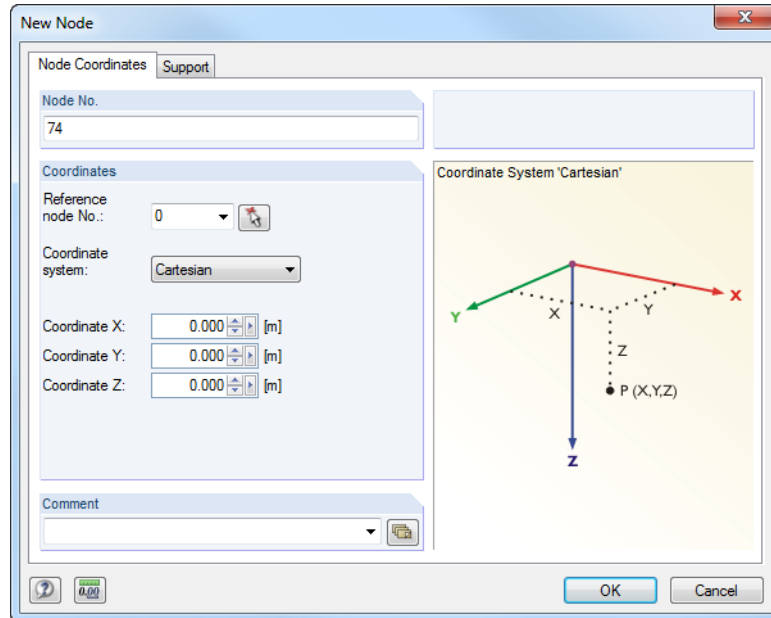


Figure 4.11: Dialog box *New Node*

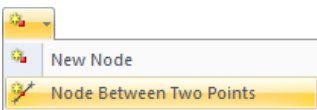
Node No.	Reference Node	Coordinate System	Node Coordinates			Comment
			X [m]	Y [m]	Z [m]	
1	0	Cartesian	0.000	0.000	0.000	Supported
2	0	Cartesian	25.000	0.000	0.000	Supported
3	0	Cartesian	0.000	0.000	-6.000	
4	0	Cartesian	3.000	0.000	-6.261	
5	0	Cartesian	6.250	0.000	-6.546	
6	0	Cartesian	12.500	0.000	-7.094	
7	0	Cartesian	18.750	0.000	-6.546	
8	0	Cartesian	22.000	0.000	-6.261	

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 11.4.16, page 318).

Furthermore, RSTAB provides a special function to create a node on the connection line of two nodes already existing (see chapter 11.4.12, page 315).



### 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 RSTAB recognizes the origin automatically.

Any node can be defined 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.

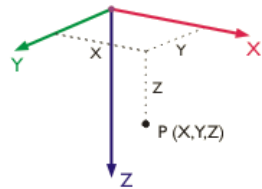


Figure 4.13: Cartesian coordinate system

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  $\theta$  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.

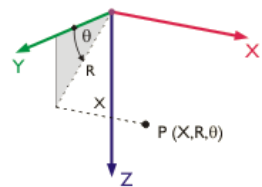


Figure 4.14: X-cylindrical coordinate system

#### Y-cylindrical

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

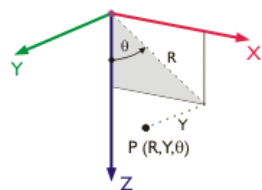


Figure 4.15: Y-cylindrical coordinate system

**Z-cylindrical**

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

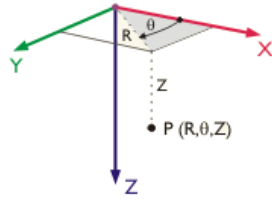


Figure 4.16: Z-cylindrical coordinate system

**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  $\theta$  and  $\Phi$ .

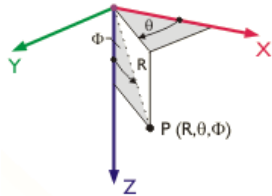


Figure 4.17: Polar coordinate system



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 by clicking the toolbar button shown on the left. Usually, nodes snap on grid points which are aligned with the active user-defined or global coordinate system (CS).

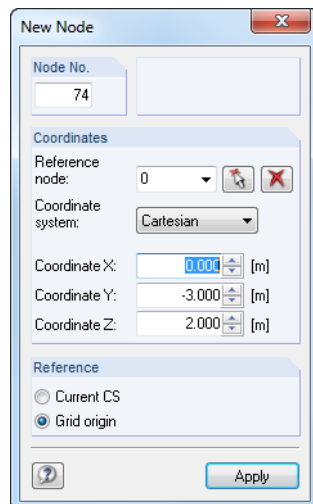


Figure 4.18: Floating dialog box *New Node*

For more information about user-defined coordinate systems, see chapter 11.3.4 page 283.

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.

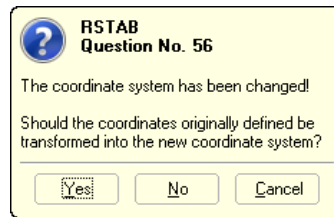


Figure 4.19: RSTAB query

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 has been restricted to a planar 2D system or a continuous beam in the *General Data* dialog box, you cannot access all three input fields or table 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 are placed in one plane: Select the relevant nodes and double click one of them to open the dialog box *Edit Node*. Only those input fields are filled with coordinates data 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 329). Furthermore, you can determine node coordinates with the Formula Editor of RSTAB (see chapter 11.6, page 332). In addition, you can take advantage of several model generators facilitating the input (see chapter 11.7.2, page 341).





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

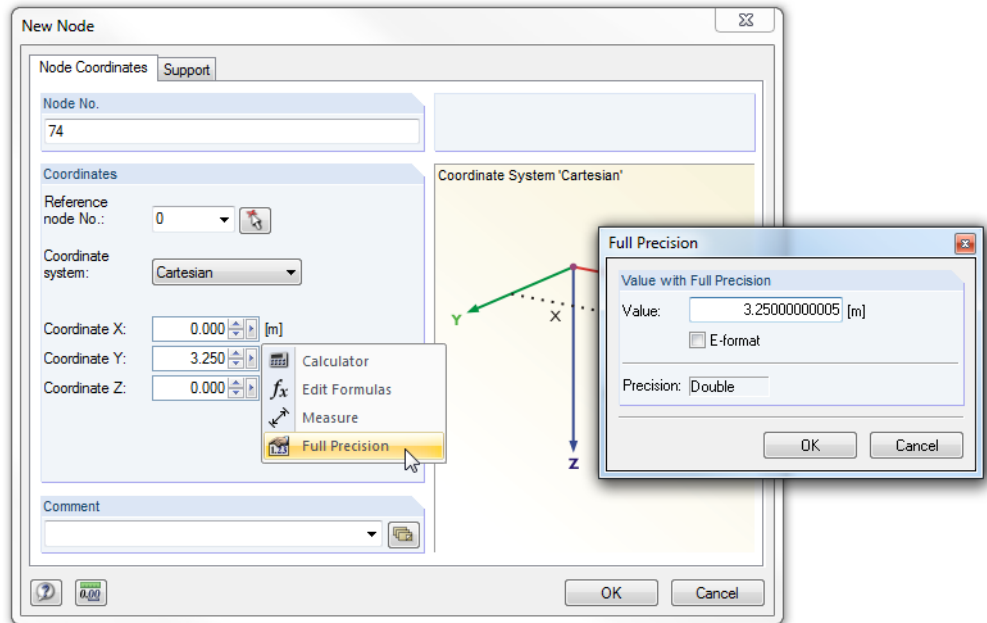


Figure 4.20: Context menu in the dialog box *New Node* and dialog box *Full Precision*

### Comment



You can enter user-defined notes. Use the button [Apply Comment] to import saved comments (see chapter 11.1.4, page 262).

## 4.2 Materials

### General description

Materials are required to define cross-sections. The material properties affect the stiffnesses of members.

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 268).

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

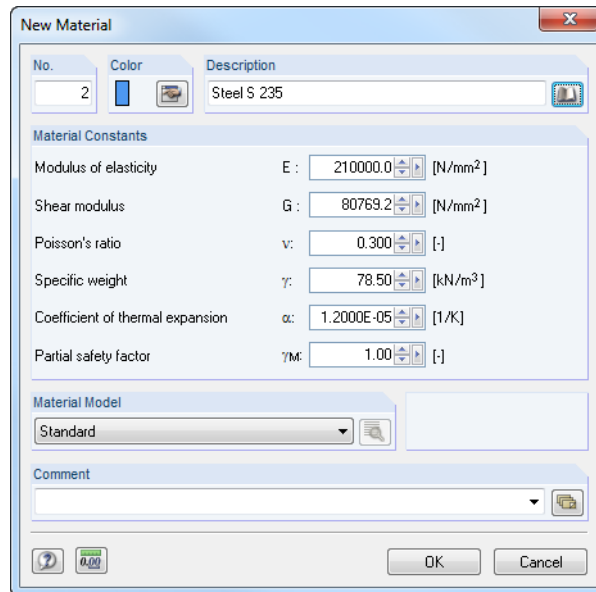


Figure 4.21: Dialog box *New Material*

Material No.	Material Description	Modulus of Elasticity E [kN/cm <sup>2</sup> ]	Shear Modulus G [kN/cm <sup>2</sup> ]	Poisson's Ratio ν [-]	Specific Weight γ [kN/m <sup>3</sup> ]	Coeff. of Th. Exp. α [1/K]	Partial Factor γ <sub>M</sub> [-]
1	Glulam Timber GL28h   DIN 1052:2008-1...	1260.00	78.00	7.077	5.00	5.00E-06	1.30
2	Poplar and Coniferous Timber C24   ONO	1100.00	69.00	6.971	4.20	5.00E-06	1.30
3	Glulam Timber GL24h   DIN 1052:2008-12	1160.00	72.00	7.056	5.00	5.00E-06	1.30
4	Steel S 235   DIN 18800:1990-11	21000.00	8100.00	0.296	78.50	1.20E-05	1.10
5							
6							

Figure 4.22: Table 1.2 *Materials*

### Material description

Any name can be chosen for the *Description* of the material. When the entered name corresponds to an entry of the library, RSTAB 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  $\nu$ . 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 $\nu$

The following relation exists between elastic and shear modulus and the Poisson's ratio  $\nu$ .

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

Equation 4.1



When you define the properties of an isotropic material manually, RSTAB 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 $\gamma$

The specific weight  $\gamma$  describes the weight of the material per volume unit.

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

### Coefficient of thermal expansion $\alpha$

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 $\gamma_M$

The value describes the safety factor for the material resistance. Therefore the index  $M$  is used. Use the factor  $\gamma_M$  to reduce stiffness for calculations according to second-order and large deformation analysis (see chapter 7.2.1, page 172).

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

### Material model

Two material models are available for selection in the list. The parameters of the material model *Isotropic Thermal-Elastic* can be defined in another dialog box that you access with a click on the [Details] button.



#### Standard

The linear-elastic stiffness properties of the isotropic material do not depend on directions. The following conditions are valid:

- $E > 0$
- $G > 0$
- $-1 < \nu$

#### 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).

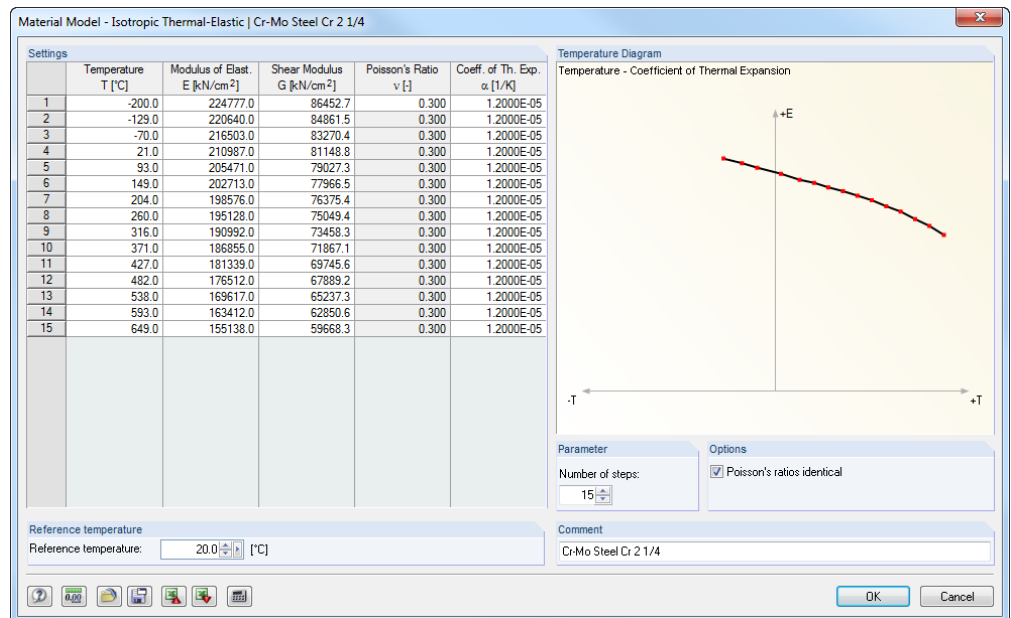


Figure 4.23: Dialog box *Material Model - Isotropic Thermal-Elastic*

The *Reference temperature* defines the stiffnesses for those members that do not bear any temperature loads. For example, when a reference temperature of 300 °C is set, RSTAB applies to all members the reduced modulus of elasticity of this point of the temperature curve.

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.



Click the [Save] button in the dialog box to save the stress-strain diagram so that you can use it for other models. Use the [Load] button to import user-defined diagrams (see Figure 4.24).

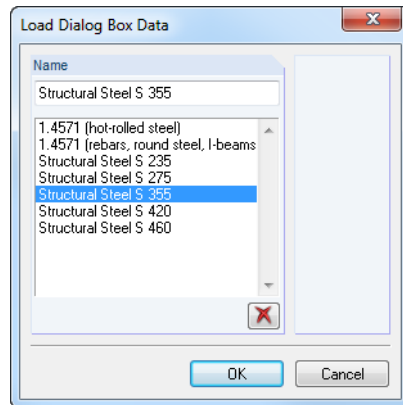


Figure 4.24: Dialog box *Load Dialog Box Data*

### 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 4.21, page 46) in the dialog box *New Material*. You can open the database also in table 1.2 *Materials* (cf. Figure 4.22, page 46): Place the cursor into table column A and click the button [...] shown on the left, or use the function key [F7] on the keyboard.

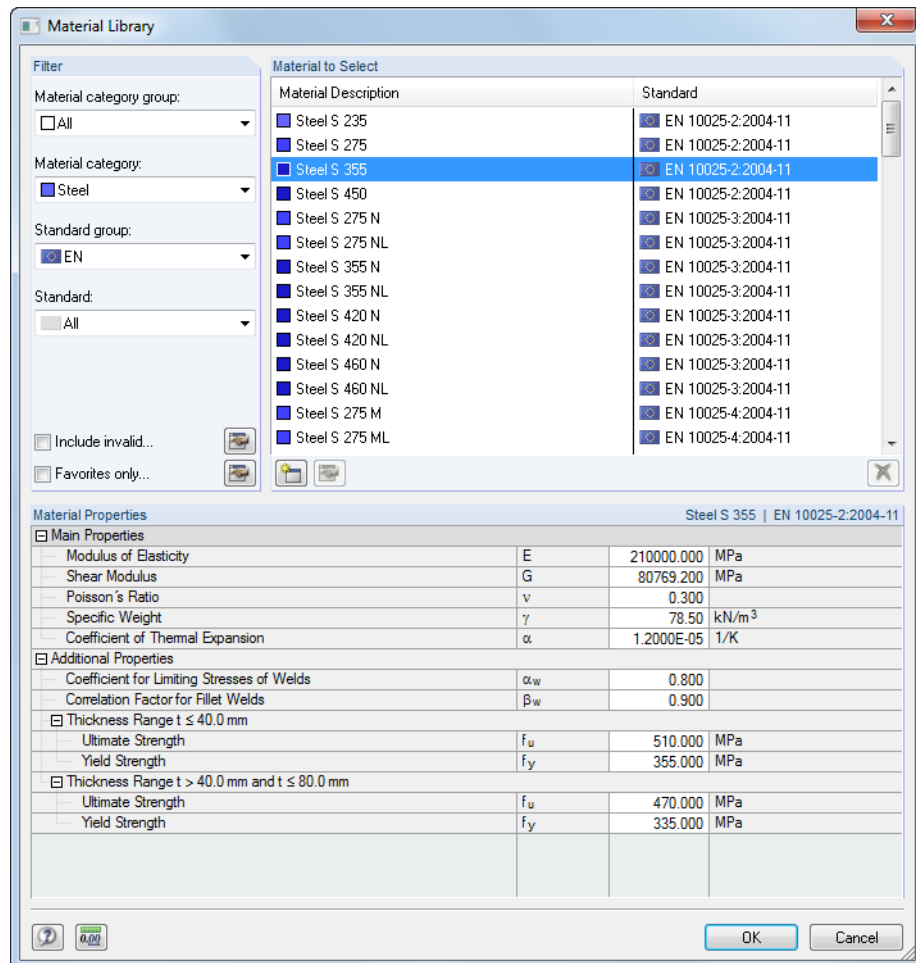


Figure 4.25: Dialog box *Material Library*

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 import it to the previous dialog box or 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.

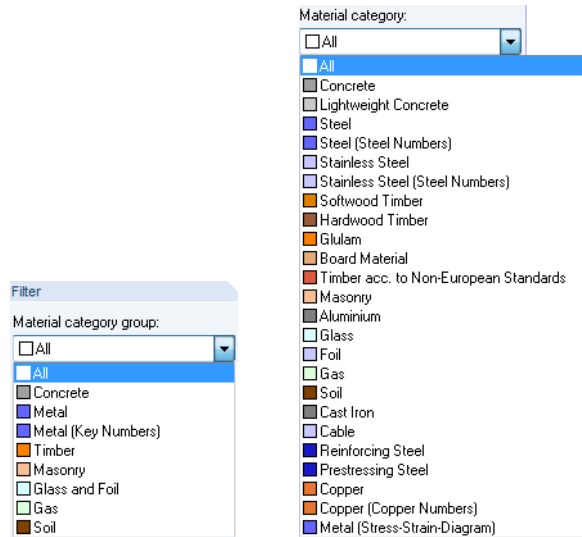


Figure 4.26: Filter for *Material category group* and *Material category*

**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 4.28) to open the dialog box for defining preferred materials.

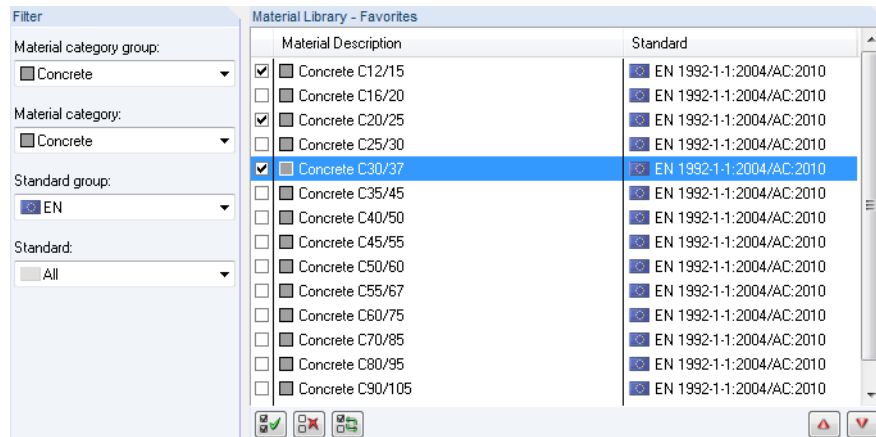


Figure 4.27: Dialog box *Material Library - Favorites* (dialog section)



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*.

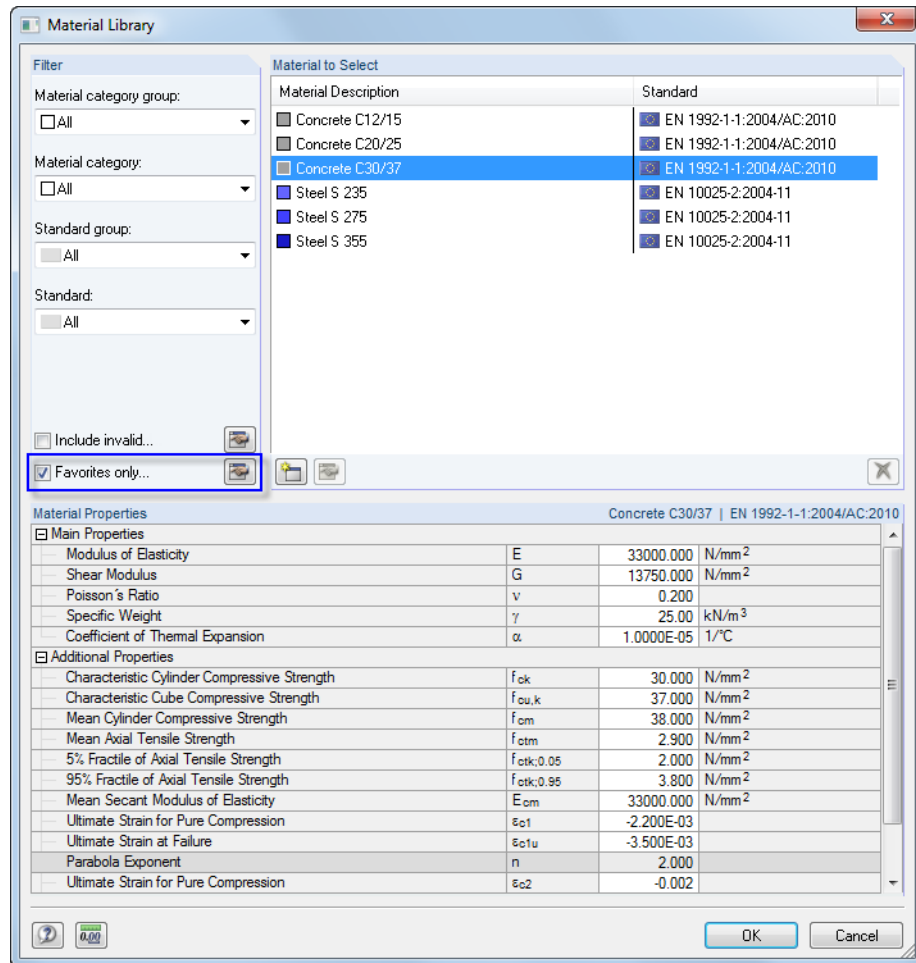


Figure 4.28: Dialog box *Material Library* with option *Favorites only*

With the check box for *Include invalid* in the *Filter* dialog section you can decide if materials of 'old' standards also appear in the library.

**Add materials to 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 4.28). The dialog box *New Material* opens (see Figure 4.29). 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.

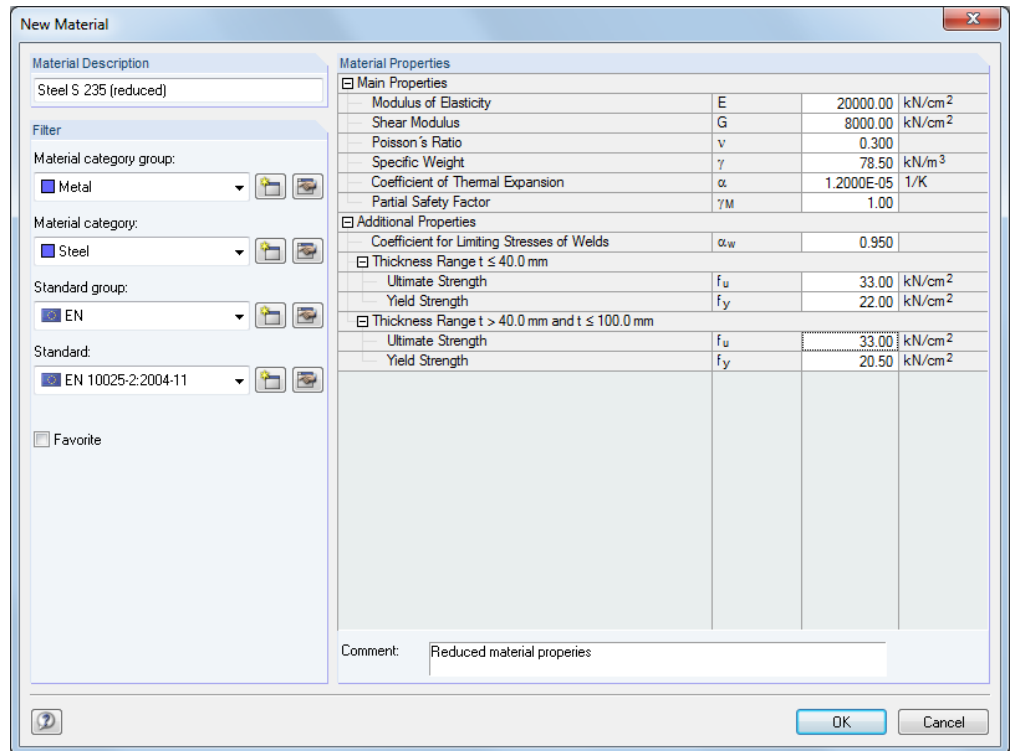


Figure 4.29: Dialog box *New Material*

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.

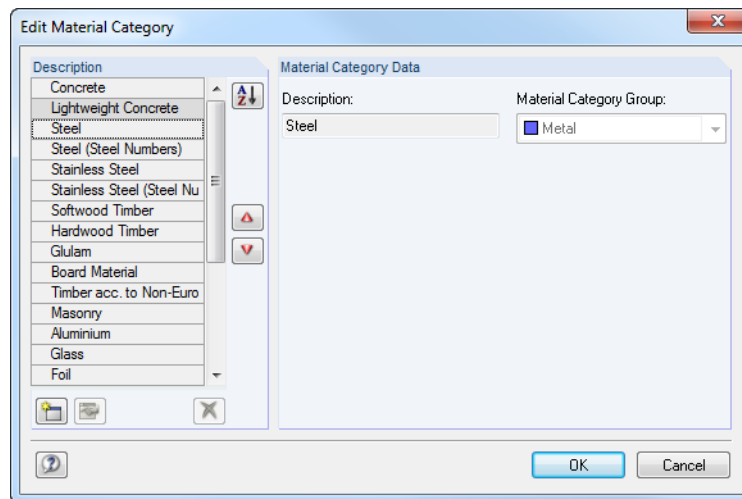


Figure 4.30: Dialog box *Edit Material Category*

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. You can find it in the master data folder of RSTAB 8 C:\ProgramData\Dlubal\RSTAB 8.01\General Data.



### 4.3 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 268).

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. RSTAB determines the variable stiffnesses along the member automatically.

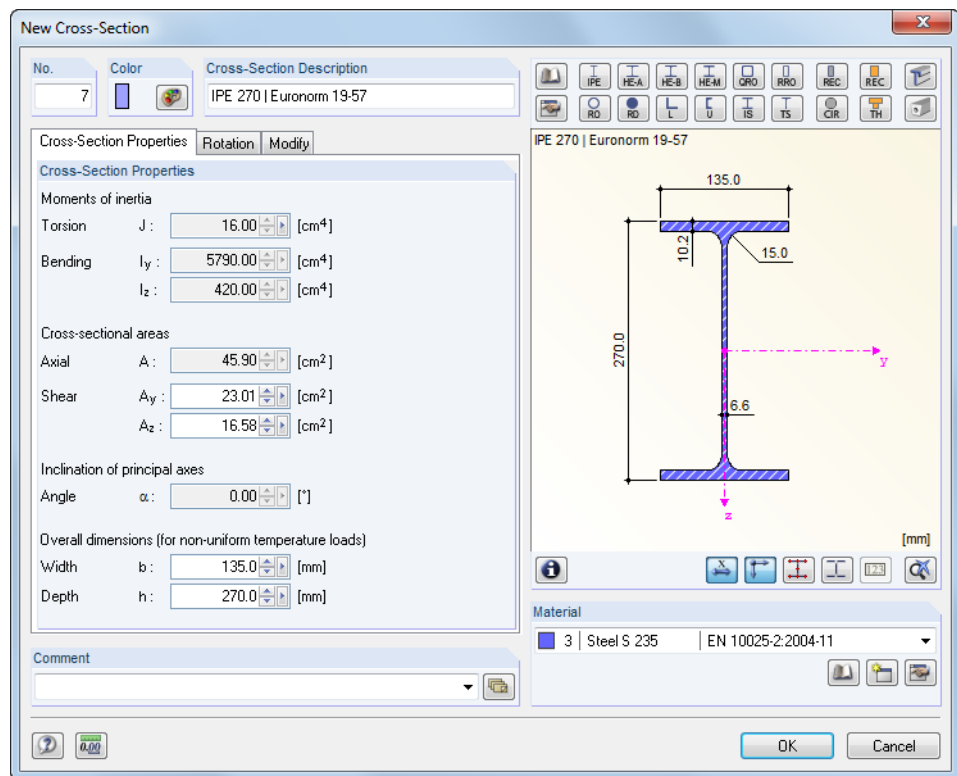


Figure 4.31: Dialog box *New Cross-Section*, tab *Cross-Section Properties*

Section No.	Cross-Section Description	Material No.	Moments of Inertia [cm <sup>4</sup> ]			Cross-Sectional Areas [cm <sup>2</sup> ]			Principal Axes		Rotation α [°]
			Torsion J	Bending I <sub>y</sub>	Bending I <sub>z</sub>	Axial A	Shear A <sub>y</sub>	Shear A <sub>z</sub>	α [°]		
1	I W 16x45   AISC 9	2	46.20	24391.16	1365.24	85.81	43.31	34.14	0.00	0.00	
2	I W 16x40   AISC 9	2	32.88	21560.79	1202.91	76.13	38.50	30.21	0.00	0.00	
3	Circle 300	4	795215603	397607801	397607801	70685.83	59903.25	59903.25	0.00	0.00	
4	T-Rectangle 20/35	3	60029.02	71458.33	23333.33	700.00	583.33	583.33	0.00	0.00	
5	SHS 3x3x0.125   AISC 13	2	118.21	74.09	74.09	8.39	4.19	4.19	0.00	0.00	
6	CHS 5.563x0.258   AISC 9	2	1259.22	631.10	631.10	27.74	13.77	13.77	0.00	0.00	
7	I W 8x21   AISC 9	2	11.65	3134.22	406.66	39.74	22.72	12.10	0.00	0.00	

Figure 4.32: Table 1.3 *Cross-Sections*

You do not need to enter the cross-section properties manually. RSTAB 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, RSTAB will import the cross-section parameters. In this case, it is not possible to change the values for the *Moments of Inertia* and the 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. RSTAB 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 RSTAB 8 is the member type *Rigid Member* (see page 75), so the definition of a *Dummy Rigid* is no longer necessary.

### Material no.

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 these buttons to access the material library, to create a new material or to edit materials.

For more detailed information about materials, see chapter 4.2, page 46.

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*.

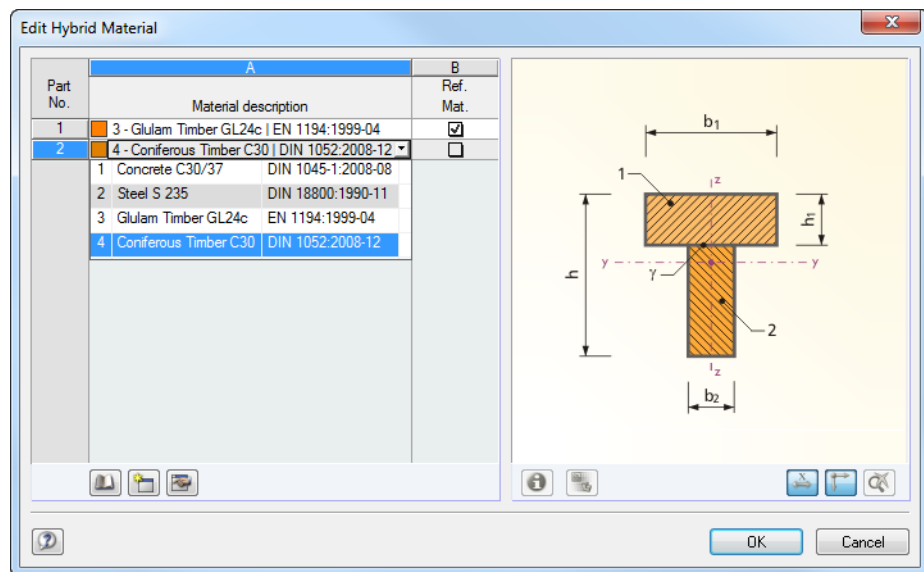
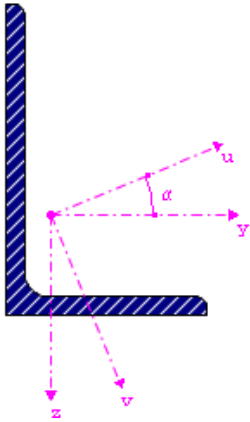


Figure 4.33: Dialog box *Edit Hybrid Material*

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.

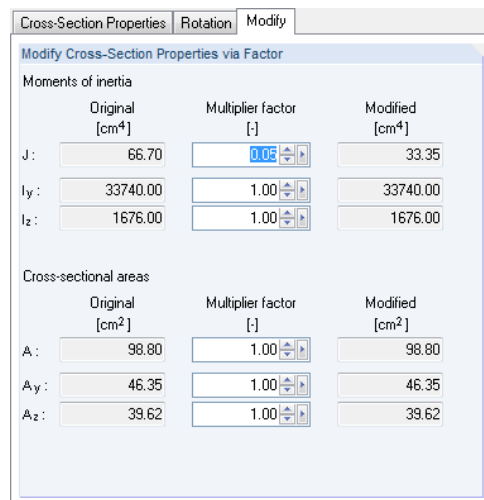


Figure 4.34: Dialog box *New Cross-Section*, tab *Modify*

With the entry in Figure 4.34, RSTAB considers the torsional constant only with 5 %.

### 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<sub>y</sub>* and *A<sub>z</sub>*.

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

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

Equation 4.2

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

Equation 4.3

- where
- A Total area of cross-section
- I<sub>z/y</sub> Moments of inertia of cross-section
- Q<sub>z/y(x)</sub> Statical moments of cross-section at location x
- t(x) Width of cross-section at location x

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. Parameters can also be controlled in the dialog tab *Global Calculation Parameters* of the dialog box *Calculation Parameters* (see Figure 7.14, page 176). 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 $\alpha$

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  $\alpha$  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 in the left margin above). This angle is defined clockwise as a positive angle. When symmetrical cross-sections are set, angle  $\alpha$  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 4.4



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

### Cross-section rotation $\alpha'$

The angle of rotation  $\alpha'$  describes the angle about which the sections of all members using that cross-section are rotated. This angle represents a global cross-sectional angle of rotation. In addition, each member can be rotated separately about a member rotation angle  $\beta$ .

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.

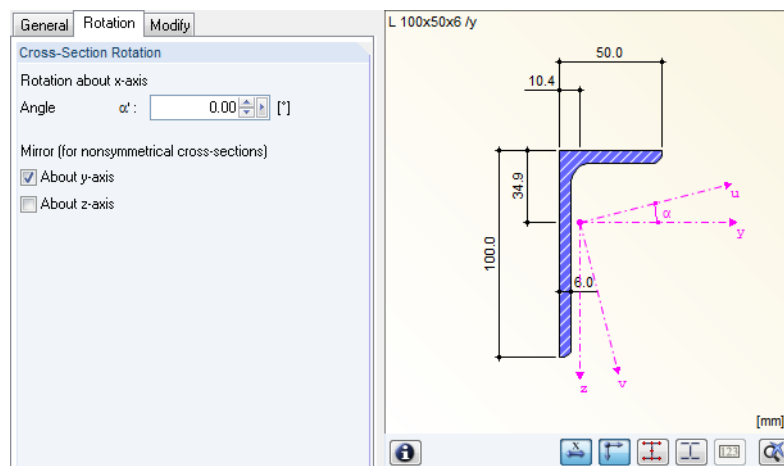


Figure 4.35: Dialog box *New Cross-Section*, tab *Rotation*

When you import a cross-section from the cross-section library or SHAPE-THIN, you do not need to take care of the angle  $\alpha'$ . RSTAB imports this angle in the same way as the 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

Width  $b$  and Depth  $h$  of the cross-section are relevant 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 1.3 *Cross-Sections*, you have direct access to frequently used cross-section tables:

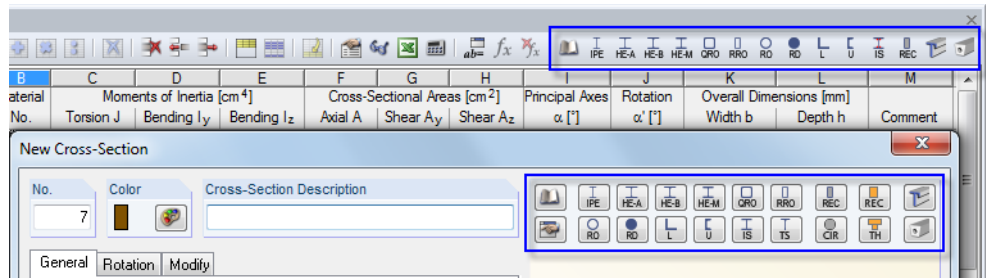


Figure 4.36: Buttons of frequently used cross-sections in table (above) and dialog box (below)



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.

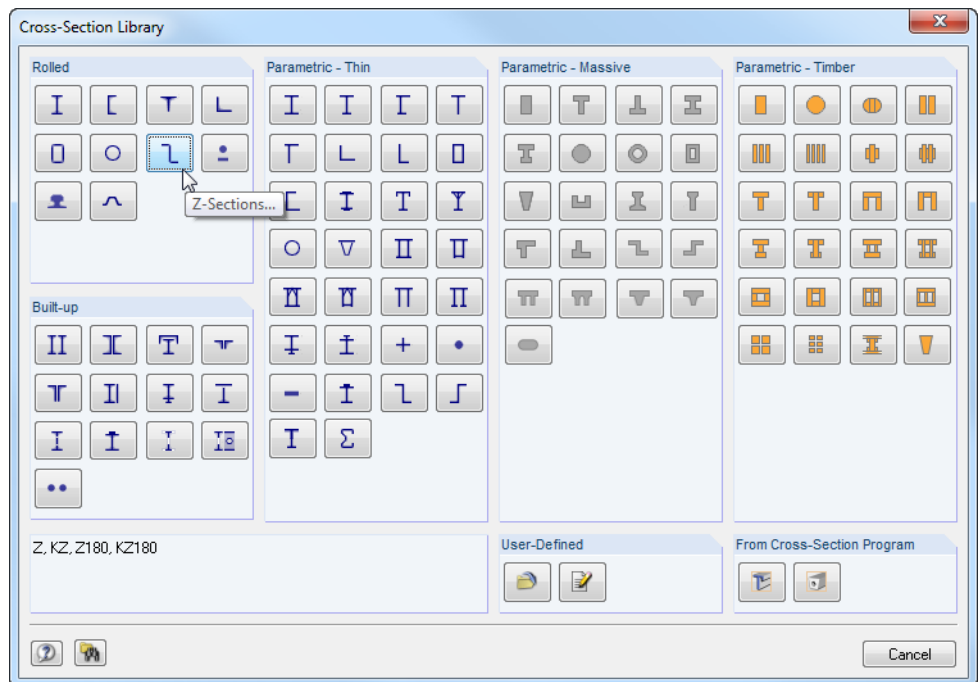


Figure 4.37: Cross-Section Library

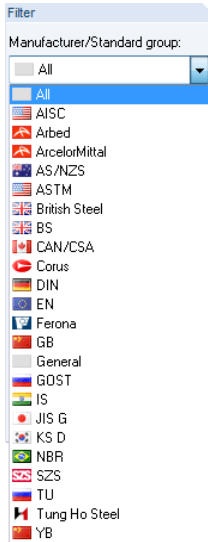
The cross-section library is divided into several sections which are described on the following pages.

### 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* (see figure below).





Filter for Manufacturer/Standard group

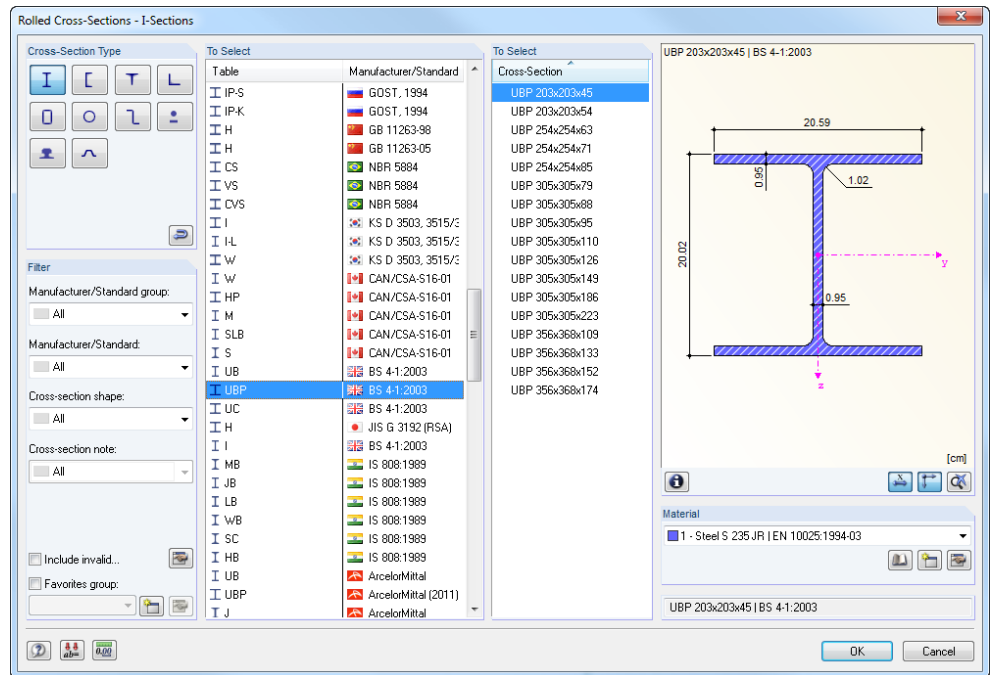


Figure 4.38: Selecting a rolled cross-section

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 *Inclusive 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] in the right corner of the *Filter* dialog section.

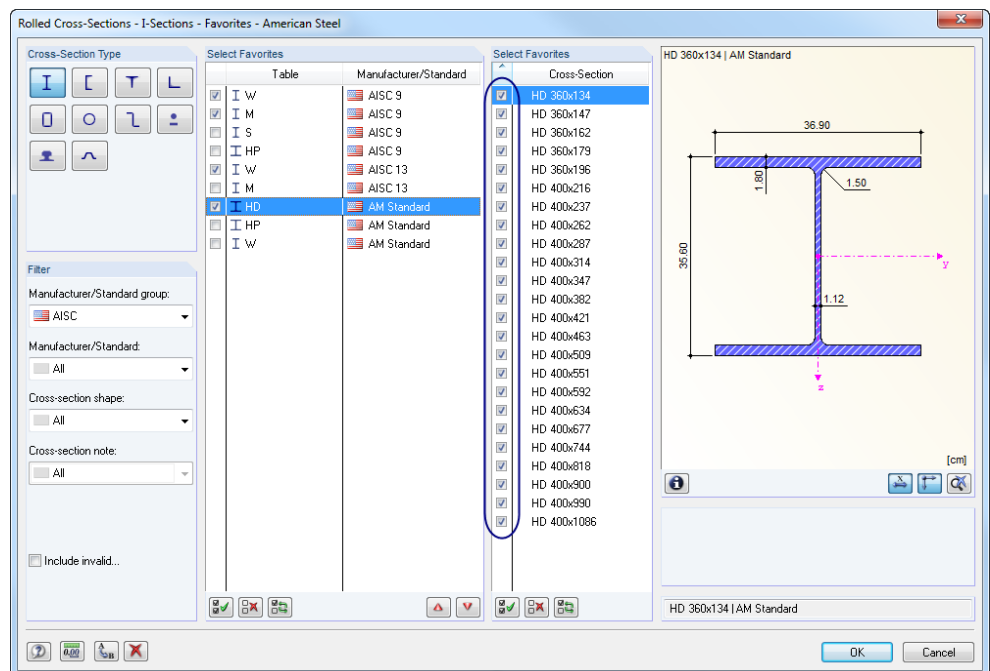
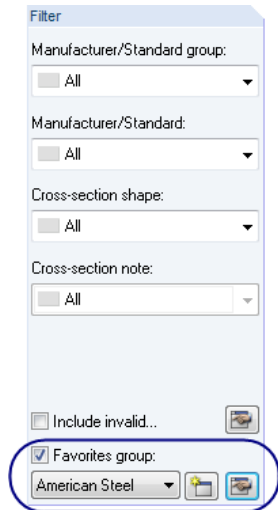


Figure 4.39: Dialog box *Rolled Cross-Sections - I-Sections - Favorites*

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*.

**Built-up cross-sections**

Rolled cross-sections can be combined by specifying parameters.

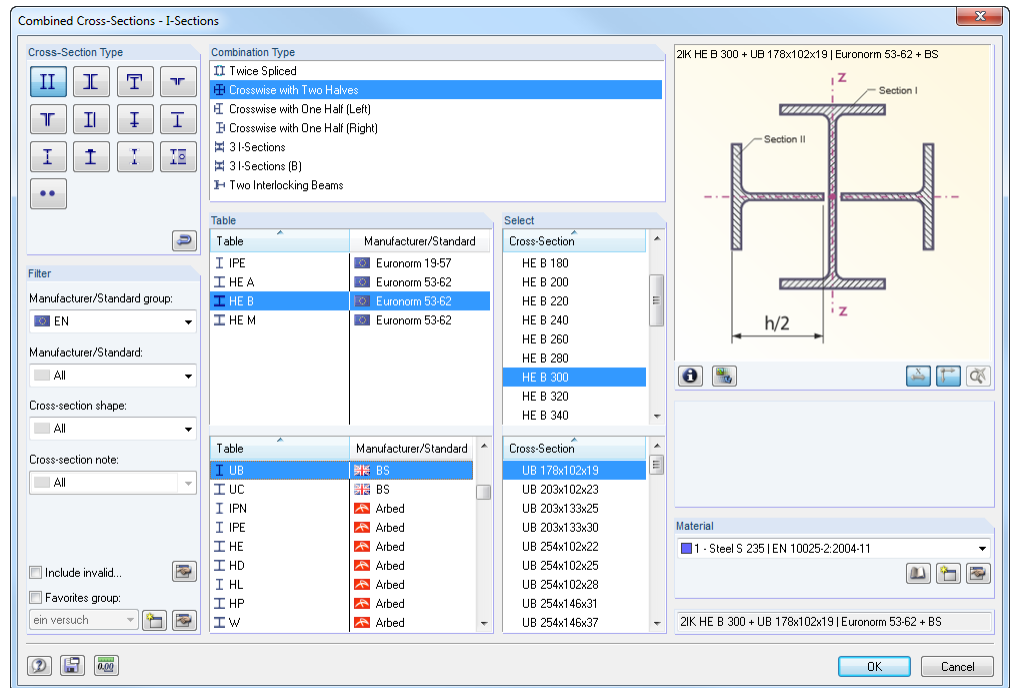
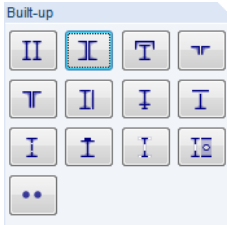


Figure 4.40: Dialog box *Combined Cross-Sections - I-Sections*



Use the [Save] button to save a combined cross-section. RSTAB stores it with its accurate description (for example *2IK HE B 300 + UB 178x102x19* 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 4.42), if possible.

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



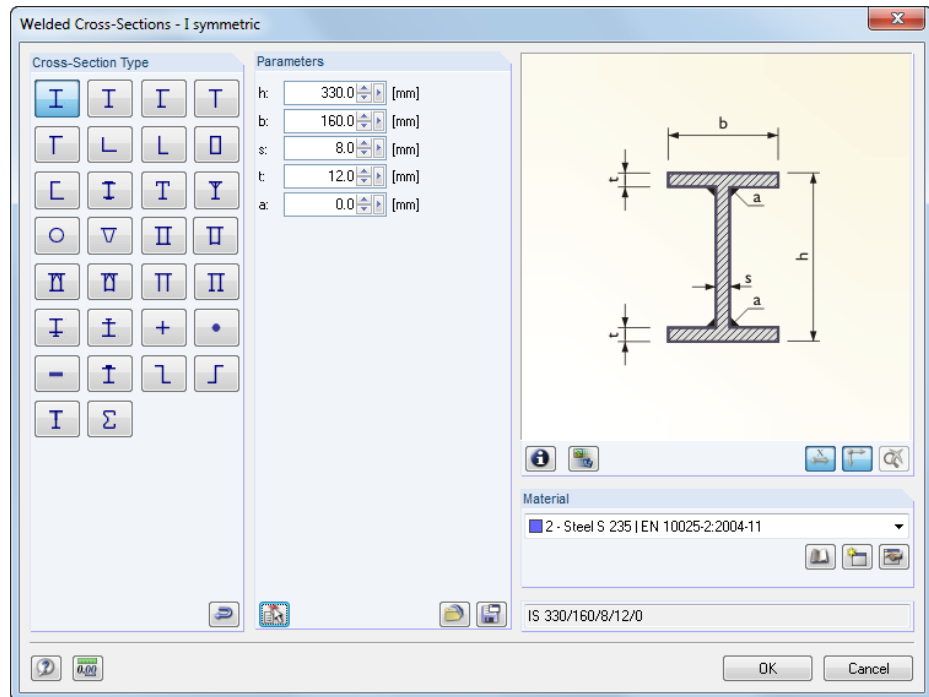


Figure 4.41: Input dialog box of a parameterized, thin-walled cross-section

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 these input fields you can freely define the parameters of 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.

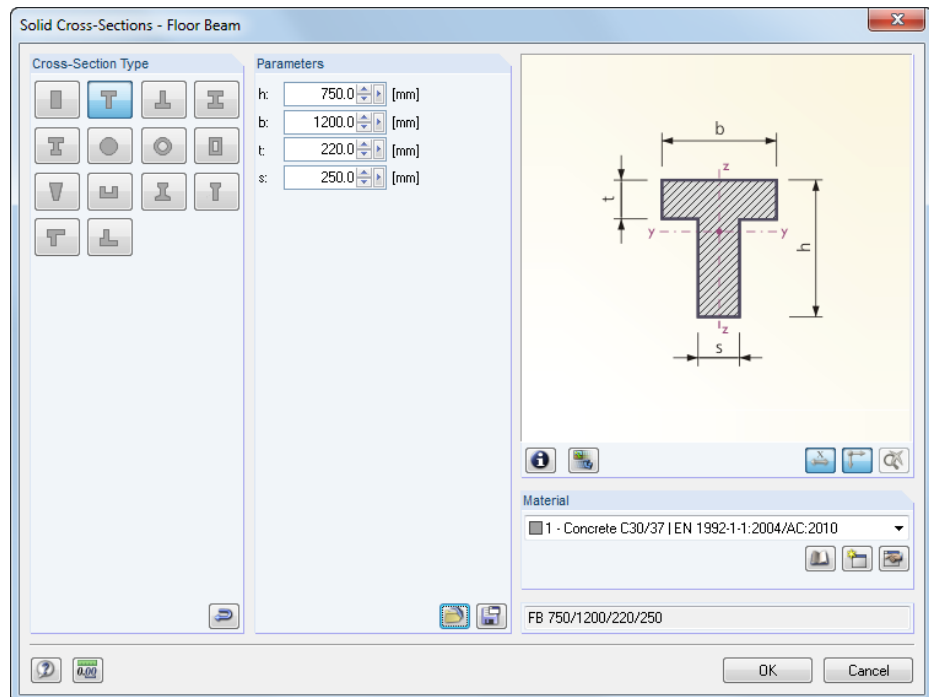


Figure 4.42: Input dialog box of a massive cross-section





**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.

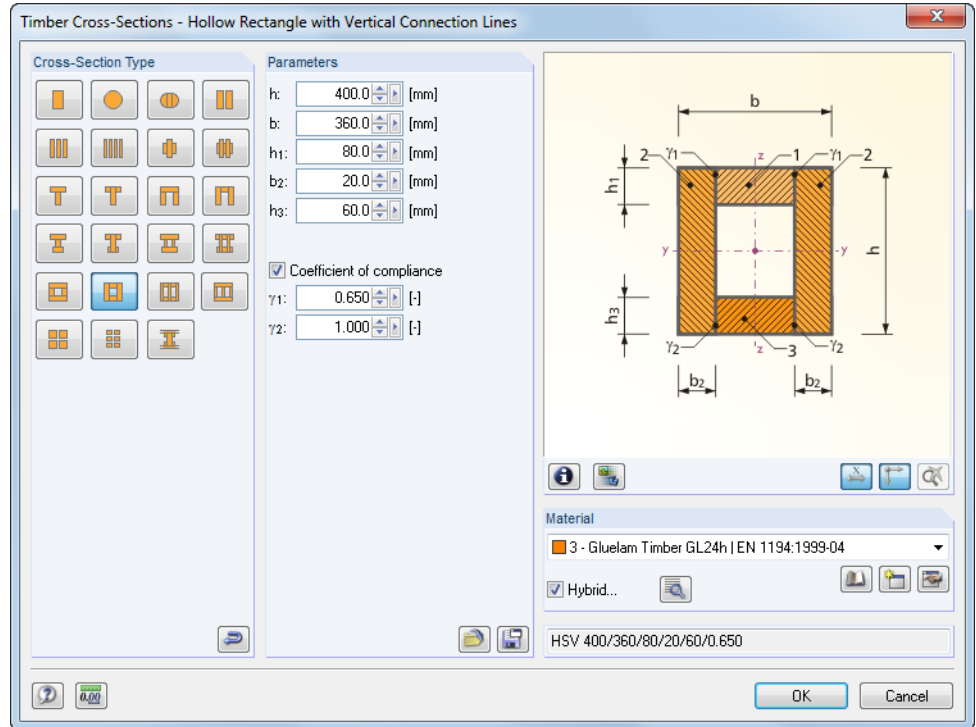
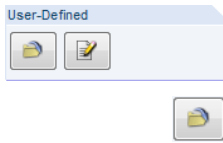


Figure 4.43: Input dialog box of a timber cross-section

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  $\gamma$ .



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 4.33, page 54).



**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.

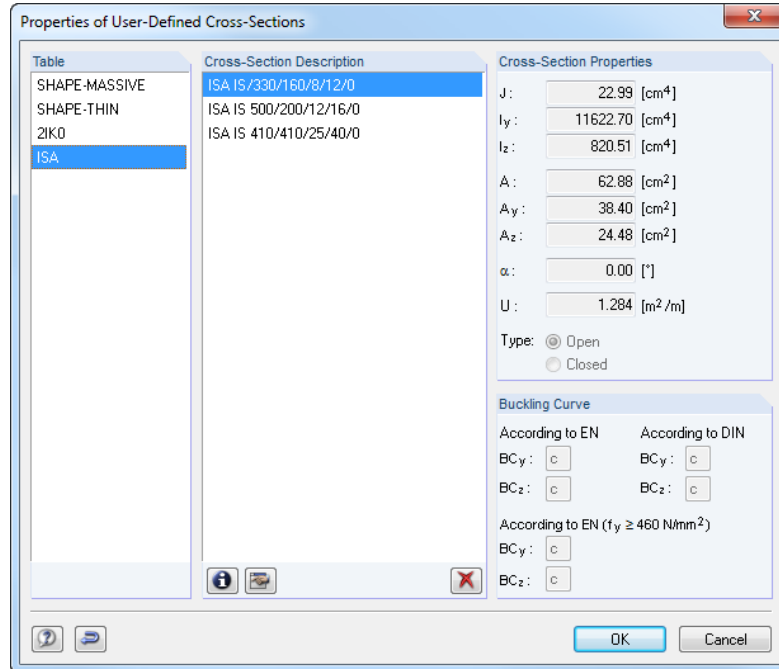


Figure 4.44: Dialog box *Properties of User-Defined Cross-Sections*



**Create a user-defined cross-section**

Click the [Create] button shown on the left to create user-defined cross-sections.

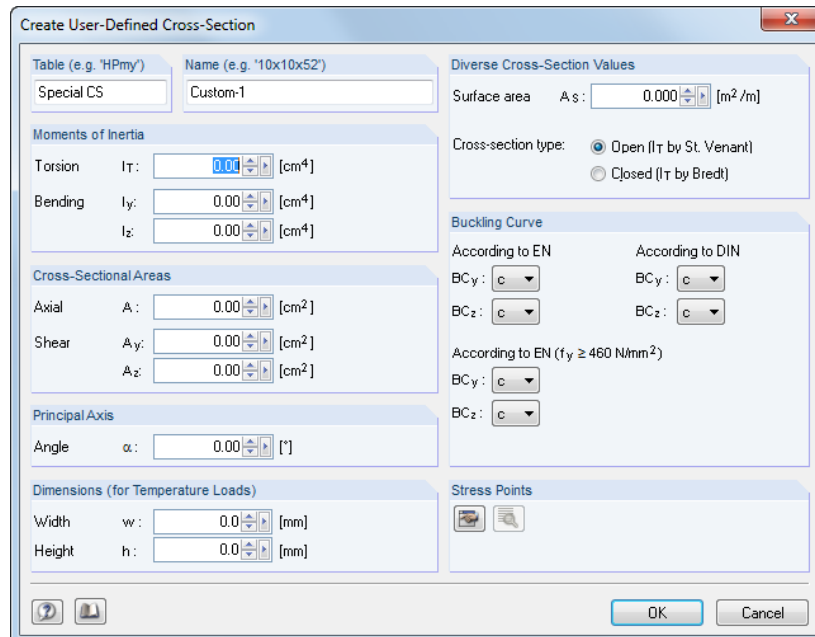
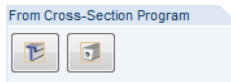


Figure 4.45: Dialog box *Create User-Defined Cross-Section*

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 SHAPE-THIN or SHAPE-MASSIVE 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 RSTAB cross-section table.

Example: Import of double symmetrical I-sections

The cross-sections are managed in the **IS** table (cf. Figure 4.41). 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	<b>Description</b>	<b>h</b>	<b>b</b>	<b>s</b>	<b>t</b>	<b>a</b>
2	Cross-section 1	400.00	200.00	10.00	10.00	0.00
3						
4						
5						

Figure 4.46: 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.

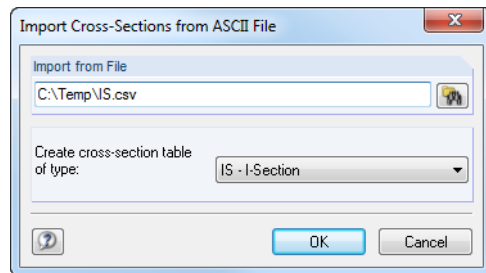


Figure 4.47: Dialog box *Import Cross-Sections from ASCII File*

Finally, you find the imported cross-sections available in the category *User-Defined* cross-sections (see Figure 4.44).

When importing cross-sections, RSTAB calculates the cross-section values and stress points so that stress designs can be performed as well.

## 4.4 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.

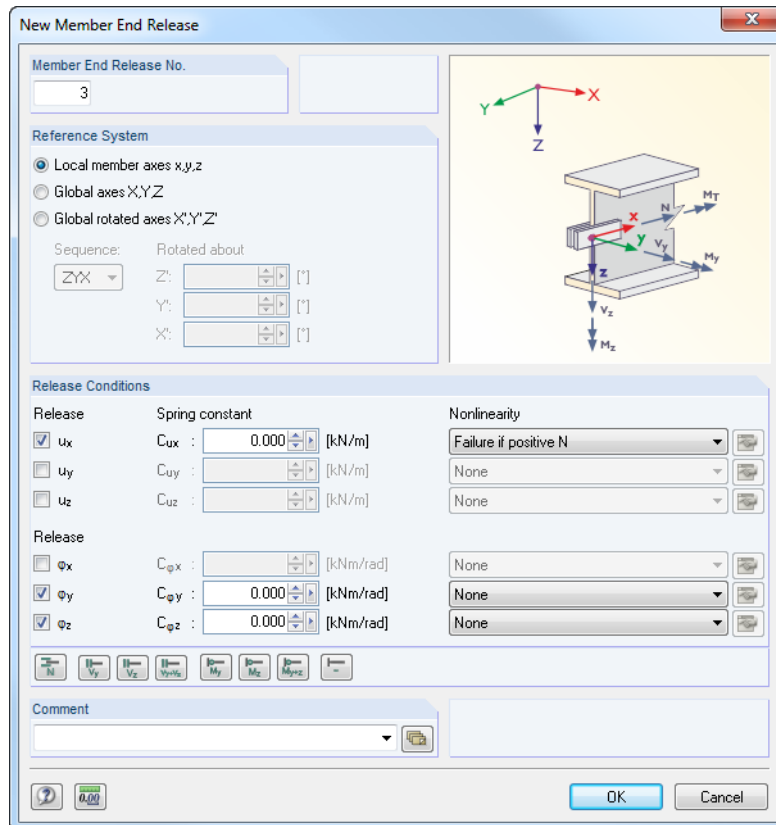
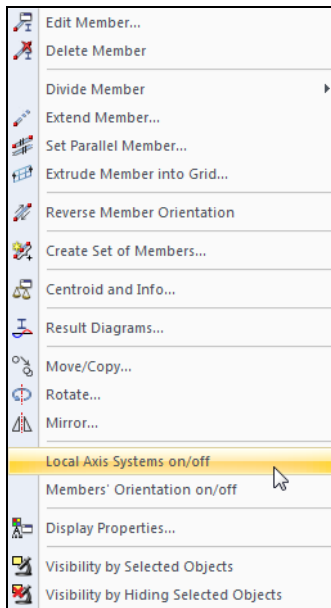


Figure 4.48: Dialog box *New Member End Release*

1.4 Member End Releases								
Release No.	A Reference System	B Axial/Shear Release or Spring [kN/m]			E Moment Release or Spring [kNm/rad]			H Comment
		N	V <sub>y</sub>	V <sub>z</sub>	M <sub>T</sub>	M <sub>y</sub>	M <sub>z</sub>	
1	Local x,y,z	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
2	Local x,y,z	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
3	Global X,Y,Z	<input type="checkbox"/>	<input type="checkbox"/>		Yes	Scissors	Scissors	
4					No			
5					Spring constant			
6					Failure if negative V <sub>z</sub>			
7					Failure if positive V <sub>z</sub>			
8					Partial activity...			
9					Diagram...			

Figure 4.49: Table 1.4 *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  $X,Y,Z$
- Global rotated coordinate system  $X',Y',Z'$

Use the *Display* navigator or the member context menu shown on the left to display the local member axes (see Figure 4.72, page 80).

For detailed information about the orientation of local member axes in the global coordinate system  $X,Y,Z$ , see chapter 4.7 on page 80.

Normally, releases are related to the local axis system  $x,y,z$ . Scissors releases (see release 3 in Figure 4.51), however, must be related to the global coordinate system. Spring constants and non-linearities must be defined in relation to the local member axis system.

### Axial/shear release or spring

To define an axial or shear force release, tick the check box of the respective internal force in the dialog box or table. The check mark means that the corresponding internal force is blocked 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 or bending moments like releases for forces. Again, the check mark means that the corresponding internal force is not 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 non-linear 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.

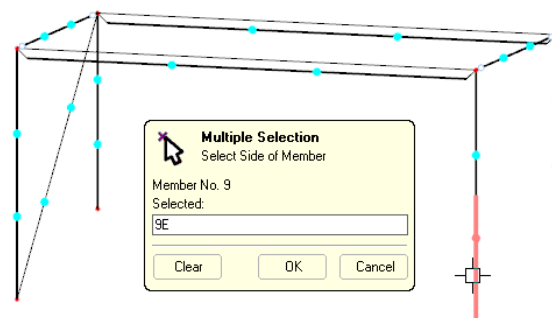


Figure 4.50: Assigning member end releases graphically

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 represent 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.

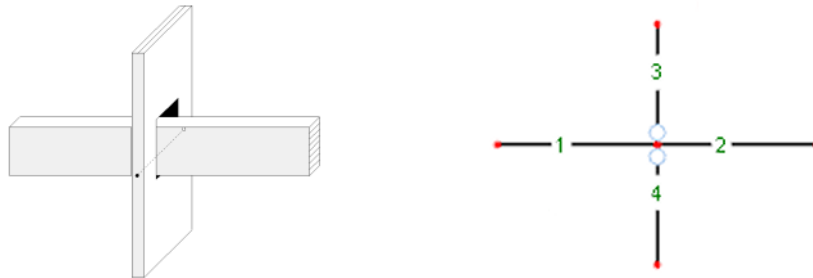


Figure 4.51: Beam crossing

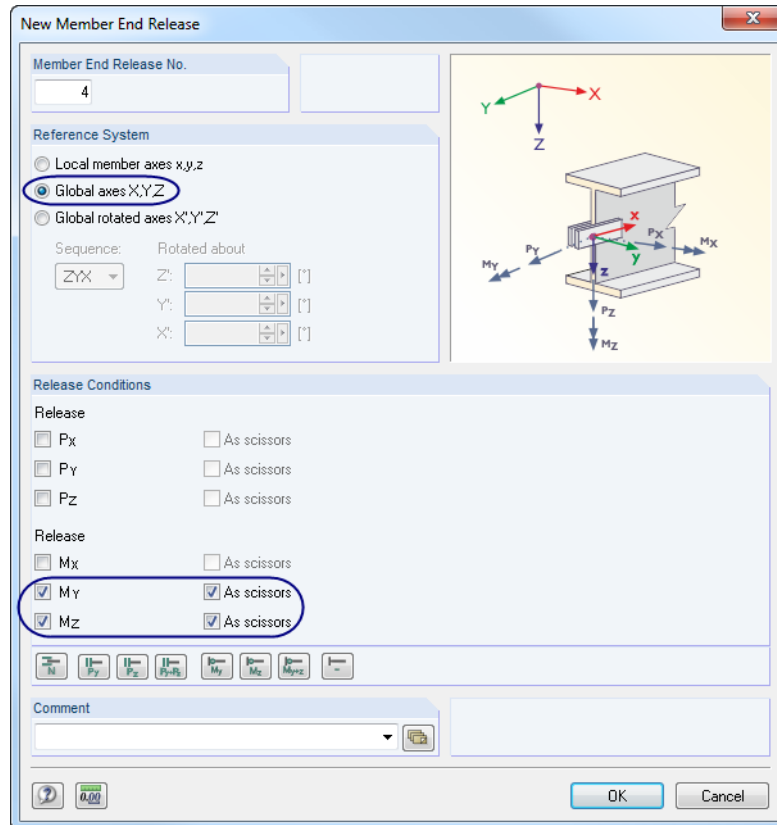


Figure 4.52: Dialog box *New Member End Release*

In this case, the release must be assigned either to members 1 and 2 or to members 3 and 4. The other crossing member pair will be modeled as bending-resistant without release.

### Non-linearities

Non-linear properties can be assigned to member end releases. In this way, you can control the transfer of internal forces in detail. The list of non-linearities offers the following options:

- Failure if internal force negative
- Failure if internal force positive
- Partial activity
- Diagram

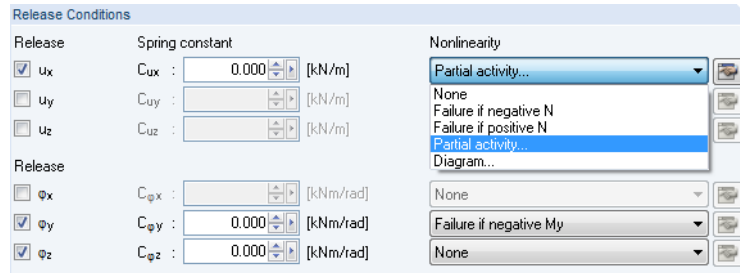


Figure 4.53: List of nonlinear properties

In the table, release types with non-linear properties are marked in blue.

#### Failure if internal force negative or positive

Use the two options to control the release activity for each internal force depending on the direction. For example, setting an axial force release with the nonlinearity *Failure if positive N* makes the release effective only for negative axial forces. Thus, only tensile forces (positive) are transferred at the member end, but no compressive forces (negative).

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 4.49, page 64).

#### Partial activity

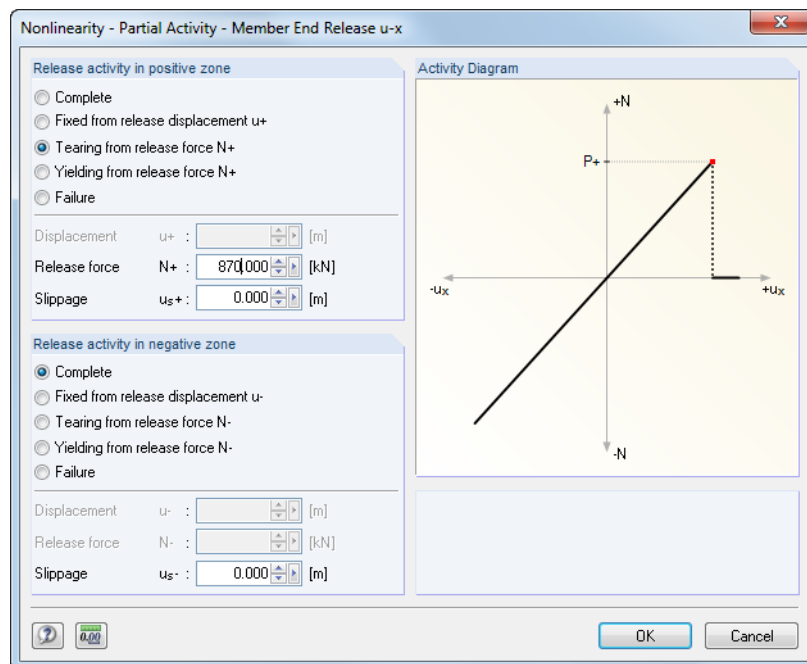


Figure 4.54: Dialog box *Nonlinearity - Partial Activity*

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**

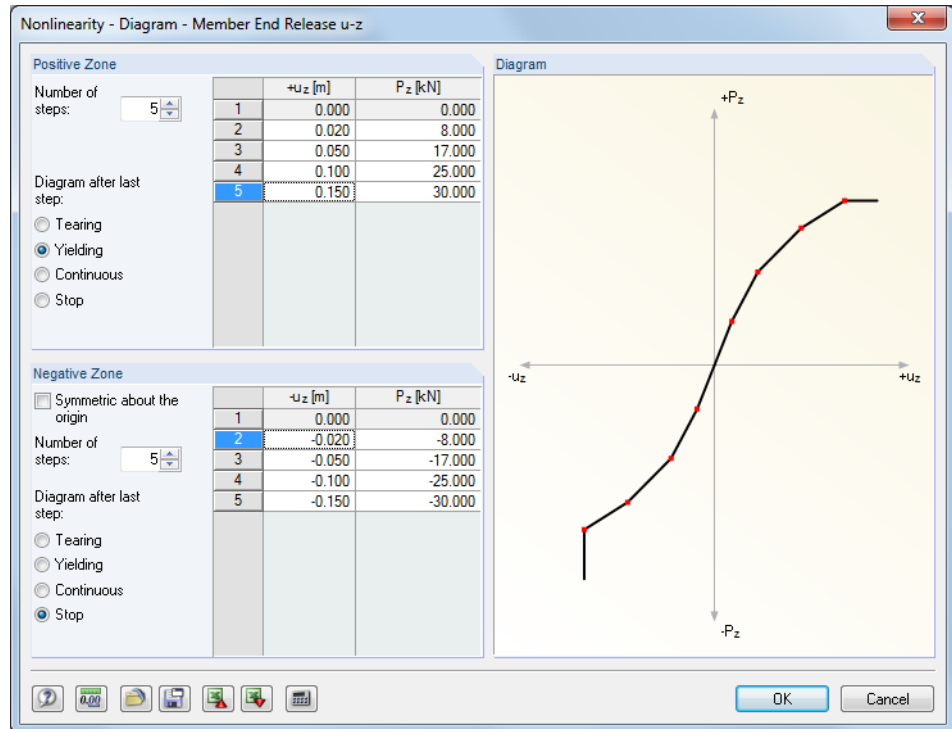


Figure 4.55: Dialog box *Nonlinearity - Diagram*

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 several options for the *Diagram after last step*: *Tearing* for 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

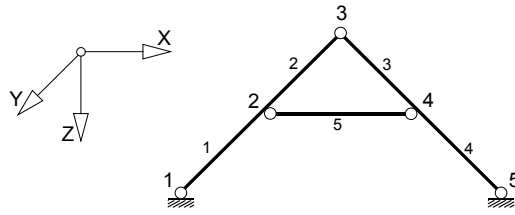


Figure 4.56: Rafter roof

A planar system is used. The release must be defined as follows:

1.4 Member End Releases

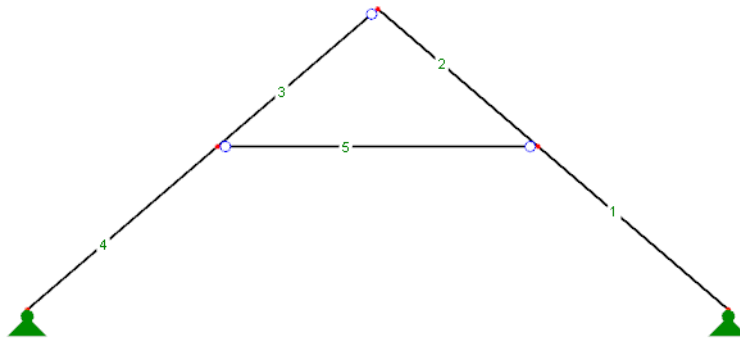
Release No.	A Reference System	B Release or Spring [kN/m]	C Release or Spring [kNm/rad]	D M <sub>y</sub>	E Comment
1	Local x,y,z	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
2					
3					

Nodes | Materials | Cross-Sections | Member End Releases | Member Eccentricities | Member Divisions | Members | Nodal Supports

Release Condition ('Y'es / 'N'o / Spring Constant / Nonlinear / F7 to select). Assign the release type to the member in Table 1.7.

Figure 4.57: Table 1.4 Member End Releases

Now, the release type can be assigned to the members.



1.7 Members

Member No.	A Member Type	B Node No. Start	C Node No. End	D Member Rotation Type	E Member Rotation β [°]	F Cross-Section No. Start	G Cross-Section No. End	H Release No. Start	I Release No. End	J Eccentr. No.	K Division No.	L Taper Shape	M Length L [m]	N Weight W [kg]
1	Beam	1	2	Angle	0.00	1	1	0	0	0	0		5.000	72.0
2	Beam	2	3	Angle	0.00	1	1	0	0	0	0		4.220	60.8
3	Beam	3	4	Angle	0.00	1	1	1	0	0	0		4.220	60.8
4	Beam	4	5	Angle	0.00	1	1	0	0	0	0		5.000	72.0
5	Beam	2	4	Angle	0.00	1	1	1	1	0	0		6.407	92.3

Nodes | Materials | Cross-Sections | Member End Releases | Member Eccentricities | Member Divisions | Members | Nodal Supports

Number of node

Figure 4.58: Graphic and table 1.7 Members

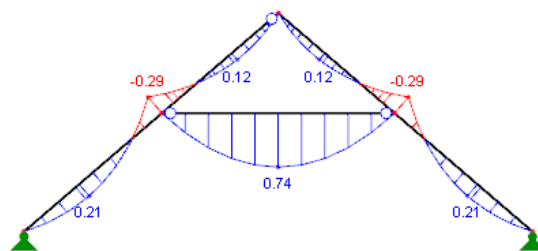


Figure 4.59: Moment diagram in load case Self-weight

## 4.5 Member Eccentricities

### General description

The length of a member in RSTAB corresponds to the distance between two nodes defining the member. However, in some modeling situations (connections of cross-sections or down-stand 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 of the respective member.

To check the entered eccentricities, use the photo-realistic imaging of the 3D rendering.

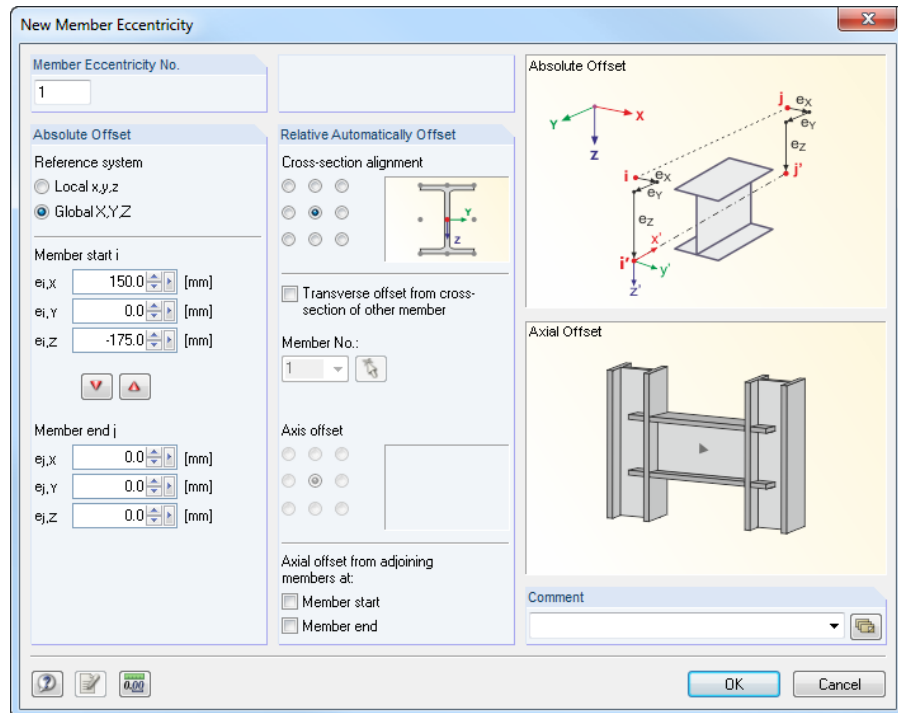
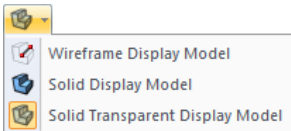


Figure 4.60: Dialog box *New Member Eccentricity*

1.5 Member Eccentricities												
Eccen. No.	Reference System	Member Start - Eccentricity [mm]	Member End - Eccentricity [mm]			Cross-Section Alignment		Transverse offset from cross-section of a		Object Type	Object No.	y-Axis
		ei,x	ei,y	ei,z	ej,x	ej,y	ej,z	y-Axis	z-Axis			
1	Global	0.0	0.0	0.0	0.0	0.0	0.0	Middle	Bottom (+z)	None	12	Middle
2	Global	150.0	0.0	-175.0	0.0	0.0	0.0	Middle	Middle	None	0	Middle
3												
4												
5												

Figure 4.61: Table 1.5 *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 4.72, page 80).

### Eccentricity for member start/member end

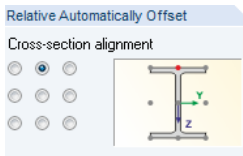
In the dialog section *Absolute Offset*, respectively in table columns B to G, you define the eccentricities for the *Member start i* and the *Member end j*. The distances are related to the selected axis system indicated by the upper- and lower-case indexes which are also shown in the dialog graphic.



In the dialog box, you can use the buttons [▼] and [▲] to transfer the values from one side to the other.

### Cross-section alignment

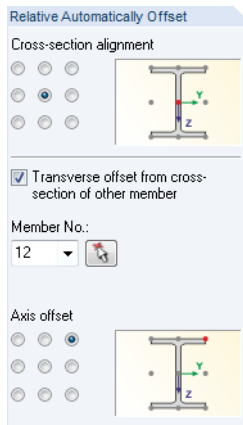
In the dialog section *Relative Automatically Offset*, use the nine selection options to define the cross-section point relevant for the determination of the eccentricity. In the table, specify the position of the point in columns H and I. The point defines the distance by which the cross-section is 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 member

With a *Transverse offset* you can arrange a member parallel to another member in a particular distance. Select the number of the relevant member 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* as a result of the cross-sectional geometry that you define by the nine check boxes. In the table, define the axis offset in columns L and M.



For example, when the points are defined on the top cross-section border and in the cross-section center as shown in the figures on the left, a horizontal beam cross-section is connected with its upper chord to the head of a column 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. RSTAB takes into account modified 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 4.50, page 65). To assign an eccentric connection to both member ends, click the member in its center area.

## 4.6 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 (RSTAB internally uses a more refined partition). Therefore, in most cases, member divisions are not required.

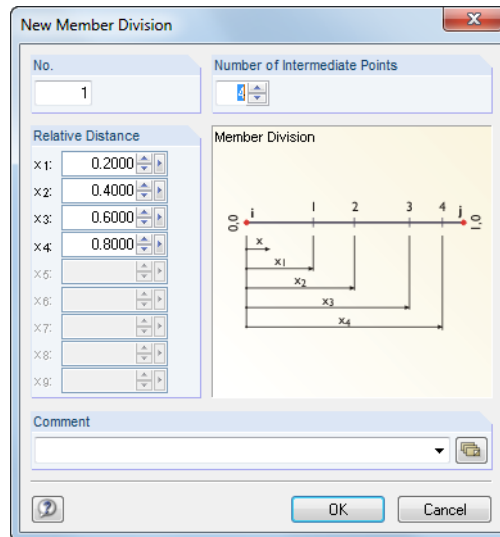


Figure 4.62: Dialog box *New Member Division*

Division No.	Number of Points	x1	x2	x3	x4	x5	x6	x7	x8	x9	Comment
1	3	0.2500	0.5000	0.7500							
2	9	0.1000	0.2000	0.3000	0.4000	0.5000	0.6000	0.7000	0.8000	0.9000	
3											
4											
5											
6											

Figure 4.63: Table 1.6 *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:  $x_1 < x_2 < x_3 \dots$



Moreover, any x-location on the member can specifically evaluated graphically (see chapter 9.5, page 208). Thus, in most cases entering member divisions manually with troublesome determination of relative distances is unnecessary.

## 4.7 Members

### General description



The geometry of the model is defined by members. Every member is defined by a start and an end node. By assigning a cross-section (by which also a material is defined), the member receives a stiffness.

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* or *Continuous*. The option *Inserted Member* is described in chapter 11.4.13 on page 316.

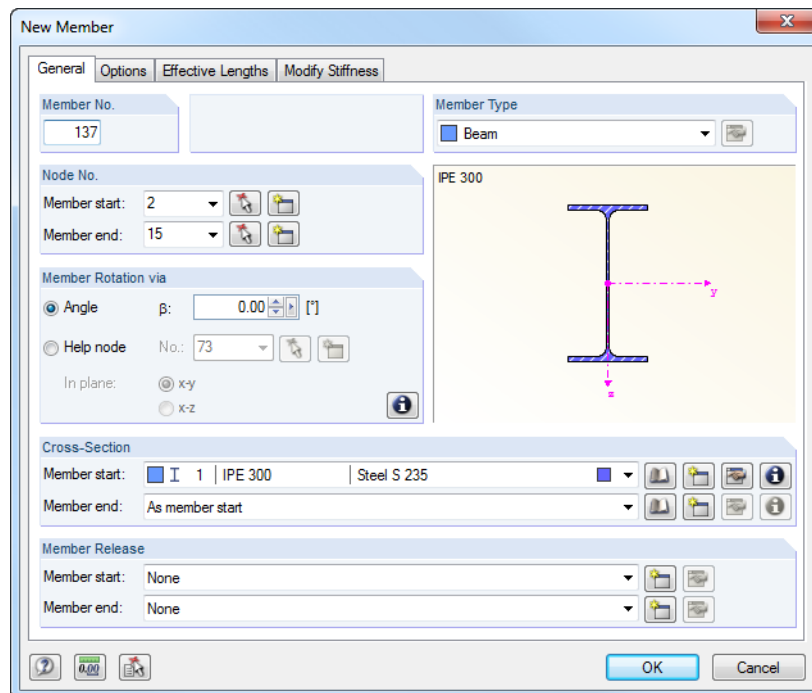


Figure 4.64: Dialog box *New Member*, tab *General*

1.7 Members															
Member No.	Member Type	Node No. Start	Node No. End	Member Rotation Type	Member Rotation $\beta$ [°]	Cross-Section No. Start	Cross-Section No. End	Release No. Start	Release No. End	Eccentr. No.	Division No.	Taper Shape	Length L [m]	Weight W [kg]	Comment
19	Beam	17	15	Angle	0.00	I 7	I 7	0	1	0	0		4,008	125,0	XZ
20	Truss (only N)	7	12	Angle	0.00	C 5	C 5			0			5,000	32,9	Y
21	Beam	19	14	Angle	90.00	I 7	I 7	0	0	0	0		4,008	125,0	XZ
22	Truss (only N)	9	14	Angle	0.00	C 5	C 5			0			5,000	32,9	Y
23	Truss (only N)	10	15	Angle	0.00	C 5	C 5			0			5,000	32,9	Y
24	Truss (only N)	16	20	Angle	0.00	C 5	C 5			0			5,000	32,9	Y
25	Truss (only N)	18	21	Angle	0.00	C 5	C 5			0			5,000	32,9	Y
26	Beam	20	10	Angle	-45.00	I 2	I 2	0	0	0	0		4,008	239,5	XZ
27	Truss (only N)	20	17	Angle	0.00	C 5	C 5			0			5,000	32,9	Y
28	Beam	21	9	Angle	0.00	I 2	I 2	0	0	0	0		4,008	239,5	XZ

Figure 4.65: Table 1.7 *Members*

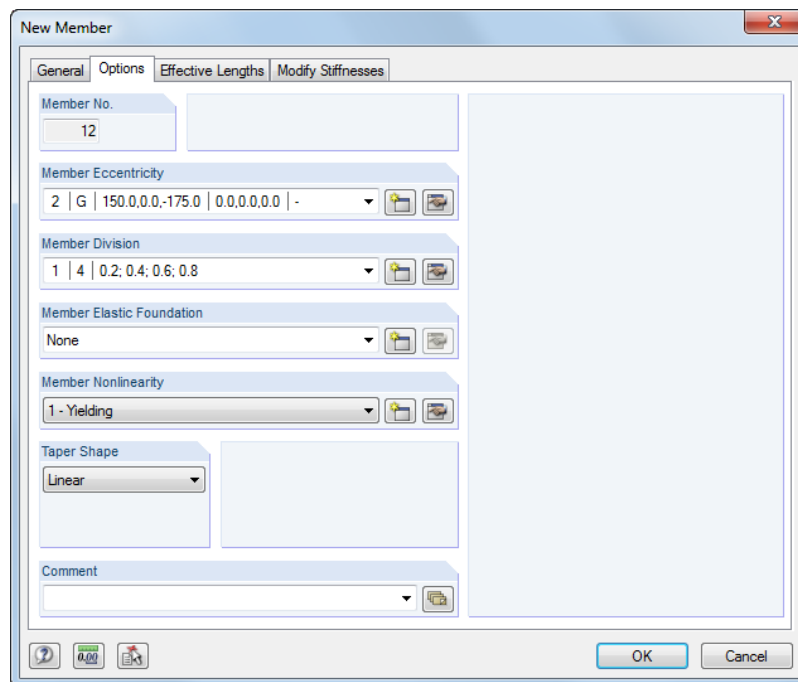
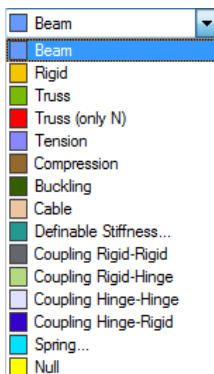


Figure 4.66: Dialog box *New Member*, tab *Options*

### 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 268).



Member type	Short description
Beam	Bending-resistant member that can transmit all internal forces.
Rigid	Coupling member with rigid stiffness
Truss	Beam with moment releases at both ends
Truss (only N)	Member with stiffness $E \cdot A$ 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.
Definable stiffness	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 and hinged connection
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 and bending-resistant connection
Null	Member that will be ignored in the calculation
Spring (in preparation)	Member with spring stiffness, definable activity zones and damping coefficients

Table 4.1: 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 77). 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. RSTAB shows internal forces also for rigid members.

The following stiffnesses are assumed (applies also to couplings and *Dummy Rigid*):

- 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)  $G_{Ay} = G_{Az} = 10^{16} \cdot l^3$

Due to this type of member it is no longer necessary to define a *Dummy Rigid* (see page 54) which is assigned as cross-section.

### 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. RSTAB 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 member / Compression member

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, RSTAB 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. You can control the *Reactivation of Failing Members* in the dialog tab **Global Calculation Parameters** of the **Calculation Parameters** dialog box that you open by selecting **Calculation Parameters** on the **Calculate** menu. Details can be found in chapter 7.2, page 176.

**Buckling member**

A buckling member absorbs unlimited quantities of tensile forces. Compressive forces, however, can be absorbed only until the critical Euler load is reached.

$$N_{cr} = \frac{\pi^2 \cdot E \cdot I}{l_{cr}^2} \quad \text{where } l_{cr} = l$$

Equation 4.5

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 7.2.1, page 170). It is required to define the complete cable as cable chain consisting of several cable members.

To create catenaries quickly, point to **Generate Model** on the **Tools** menu and select **Arc** (chapter 11.7.2, page 351). 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 29) to "1" so that tightening effects are represented realistically.

**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.

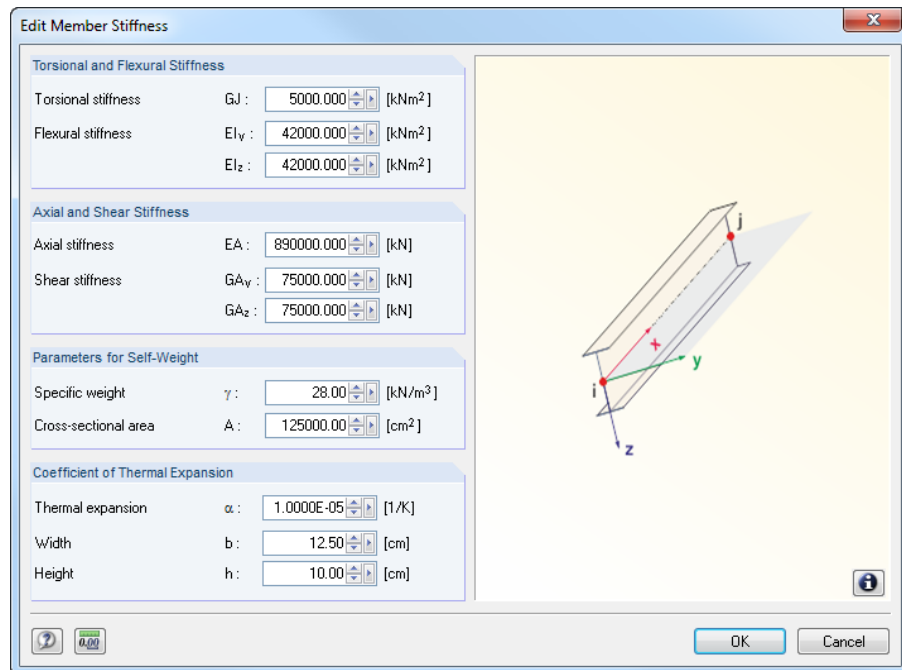
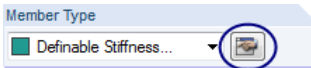


Figure 4.67: Dialog box *Edit Member Stiffness*

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.

RSTAB calculates stiffnesses of couplings depending on the model in order to exclude numerical problems.



With the alternative *Rigid* member (see page 75) you can define coupling members considering also springs and nonlinearities of releases.

To control the display of coupling results, use the *Display* navigator.

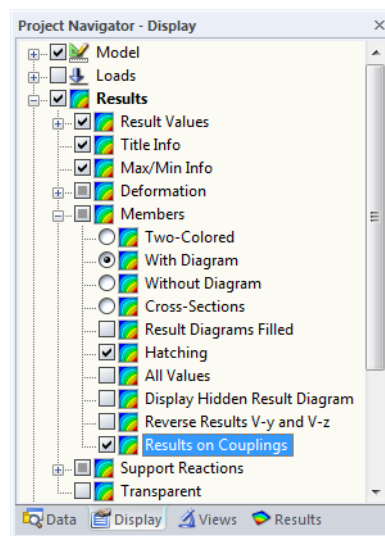


Figure 4.68: Activating the display for results of coupling members in the *Display* navigator

### Null (dummy member)

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, loading will be kept as well.

**Spring (in preparation)**

If *Spring* non-linearities are set, you can open a new dialog box by using the [Edit] dialog button or the [...] button in the table.

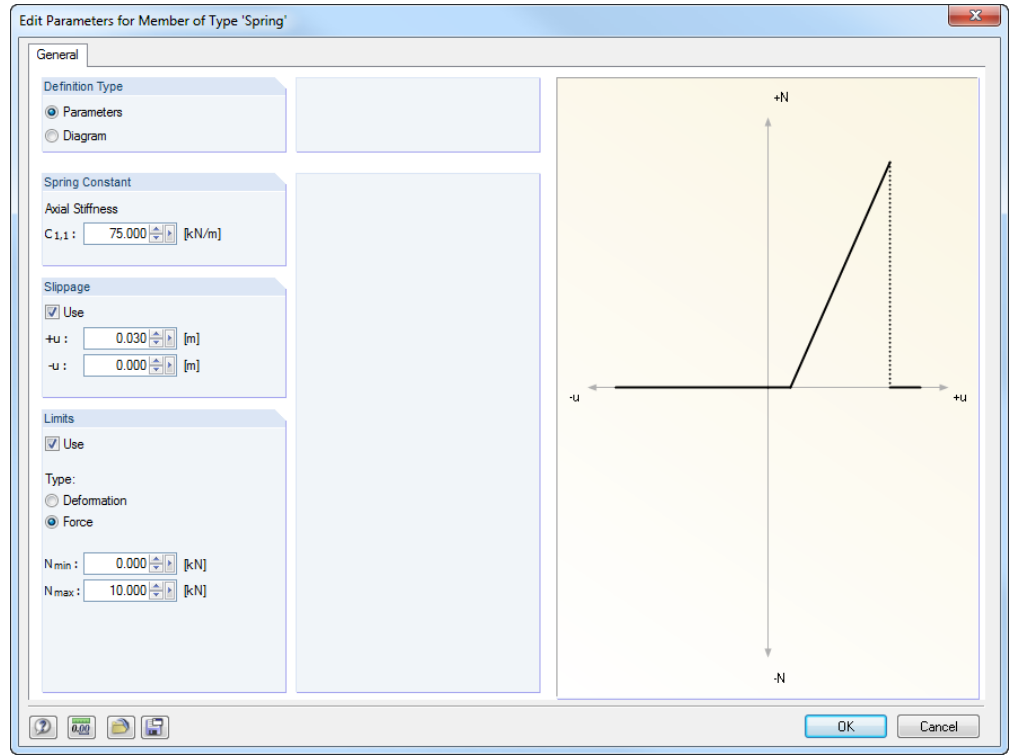


Figure 4.69: Dialog box *Edit Parameters for Member of Type Spring*

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 4.6

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. The settings are largely identical with the options available for nonlinear member releases (see chapter 4.4, page 68).

### Node no. start / end

Every member is geometrically defined by a start and an end node. With this definition the member orientation is defined, which affects also the member coordinate system. The nodes can be entered manually, graphically selected or redefined (see chapter 4.1, page 41).

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

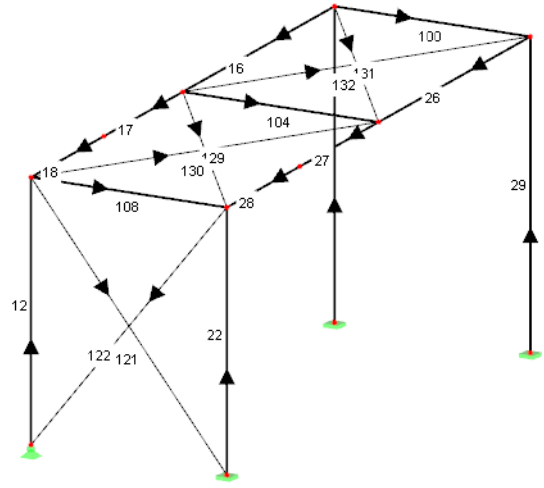
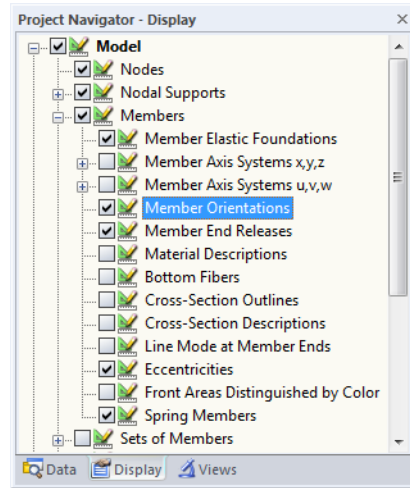


Figure 4.70: Activating *Member Orientations* in the *Display* navigator



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

### 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 member (positive direction). Member axes  $y$  and  $z$ , respectively  $u$  and  $v$  for unsymmetrical cross-sections, represent the principal axes of the member.

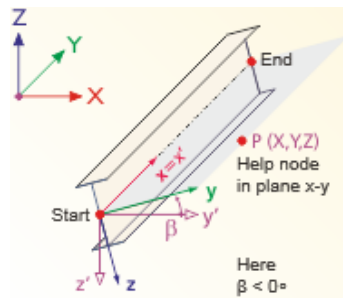
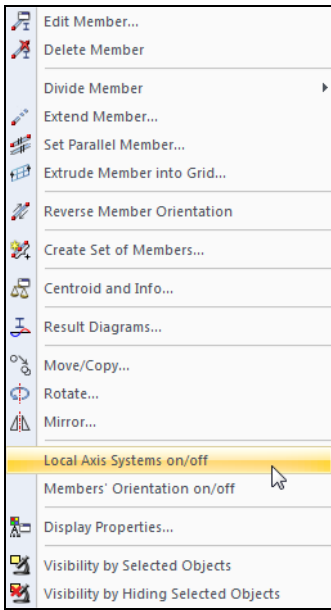


Figure 4.71: Member rotation and local member axes  $x,y,z$  (any spatial position)

The position of the local axes  $y$  and  $z$  is set automatically. Axis  $y$  runs perpendicular to the longitudinal axis  $x$  and, as far as possible, 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.



Member context menu

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*.

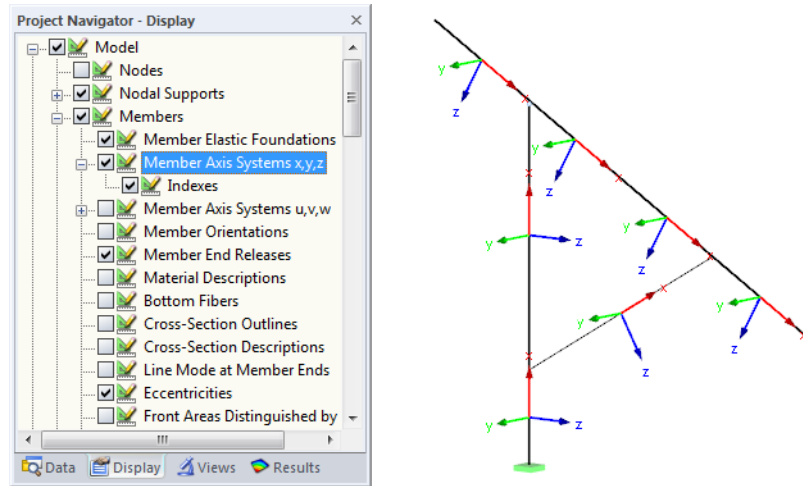


Figure 4.72: Selecting the local member axis systems in the *Display* navigator

Table column **O** 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.

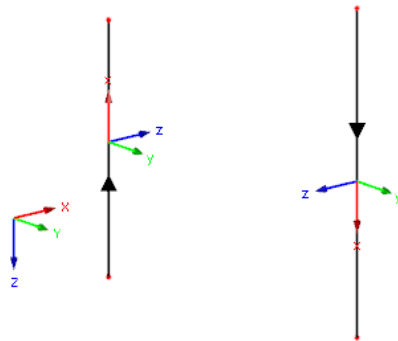


Figure 4.73: Vertical member position for members with different member orientations ( $\beta = 0^\circ$ )

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. RSTAB 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 167).

Member rotations can be applied in two ways:

**Member rotation via angle  $\beta$**

You define an *Angle*  $\beta$  about which the member is rotated. If the rotation angle  $\beta$  is positive, axes y and z are rotated clockwise around the longitudinal member axis x.

Please note that the member rotation angle  $\beta$  and the cross-section rotation angle  $\alpha'$  (see chapter 4.3, page 56) are summed up.

In 2D models, only member rotation angles of  $0^\circ$  and  $180^\circ$  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 determines the plane x-y or 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.

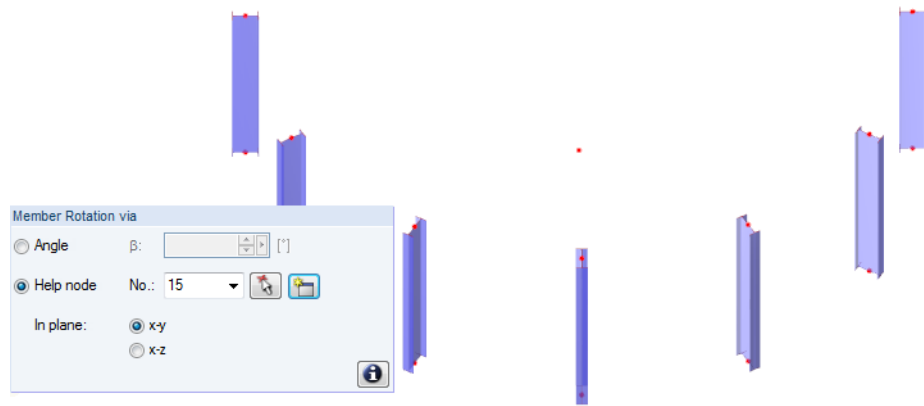


Figure 4.74: Member rotation via help node

Changes of the local member axis system may affect the signs of internal forces. The following figure illustrates the general sign rule.

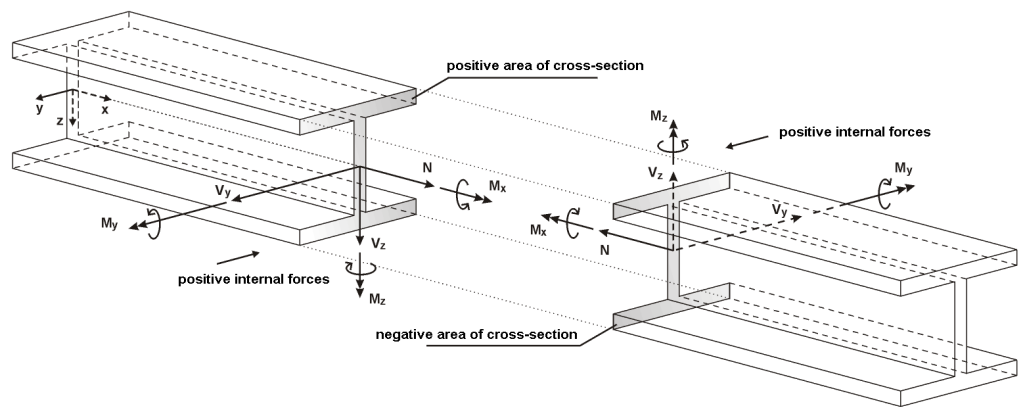


Figure 4.75: Positive definition of internal forces



The bending moment  $M_y$  is positive if tensile stresses occur on the positive member side (in direction of the z-axis).  $M_z$  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.



Tapered member

### Cross-section no. start / end

The two input fields or table columns are used to define the cross-sections for the member start and end. The numbers of the cross-sections refer to the entries in table 1.3 *Cross-Sections* (see chapter 4.3, page 53). Assignment is made easier by colors related to different cross-sections.

When you enter different numbers for the start and end cross-sections, a taper is created. RSTAB 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 82).

### Member release no. start / 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 in table 1.4 *Member End Releases* (see chapter 4.4, page 64).

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 4.66), you can assign an eccentric connection to the member. The numbers of the eccentricities refer to table 1.5 *Member Eccentricities* (see chapter 4.5, page 70). 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 4.6, page 72). 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 1.6 *Member Divisions*.

A member division has neither influence on the determination of extreme values nor on the graphical results diagram (RSTAB internally uses a more refined partition). As member divisions are not required in most cases, the default setting is 'None' or '0'.

### 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 *Linear* and *Quadratic* taper layout. 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 to the end cross-section, 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. RSTAB applies  $g = 10 \text{ m/s}^2$  for gravitational acceleration.

## Position

Table column **O** informs you about the global axis running parallel to the member, or it 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. RSTAB 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 167).

## Member elastic foundation

With this input field of the *Options* tab (see Figure 4.66) you can assign an elastic foundation to the member. The numbers of the elastic foundations are managed in table 1.9 *Member Elastic Foundations* (see chapter 4.9, page 91).

## Member nonlinearity

This input field in the *Options* dialog tab makes it possible to provide the member with nonlinear properties (see Figure 4.66, page 74). The numbers of the non-linearities refer to the entries in table 1.10 *Member Nonlinearities* (see chapter 4.10, page 93).

## Effective lengths

The dialog tab *Effective Lengths* manages the *Effective Length Factors*  $k_{cr,y}$  and  $k_{cr,z}$ .

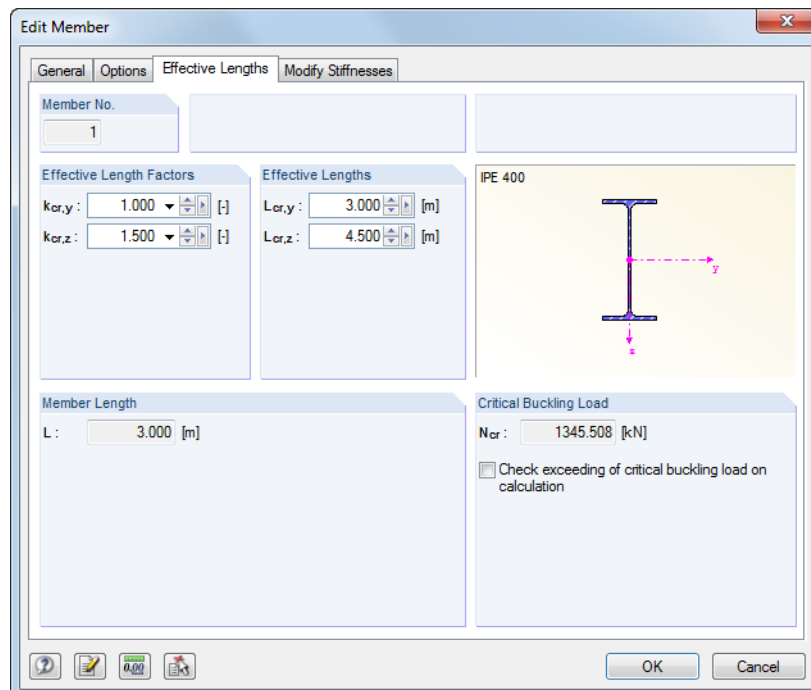


Figure 4.76: Dialog box *Edit Member*, tab *Effective Lengths*

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 STEEL EC3 where stability analyses are performed, but they play a secondary role for RSTAB 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 7.14, page 176) offers a global setting option for this kind of control.

### Modify stiffness

The dialog tab *Modify Stiffness* allows you to have an influence on the member stiffnesses.

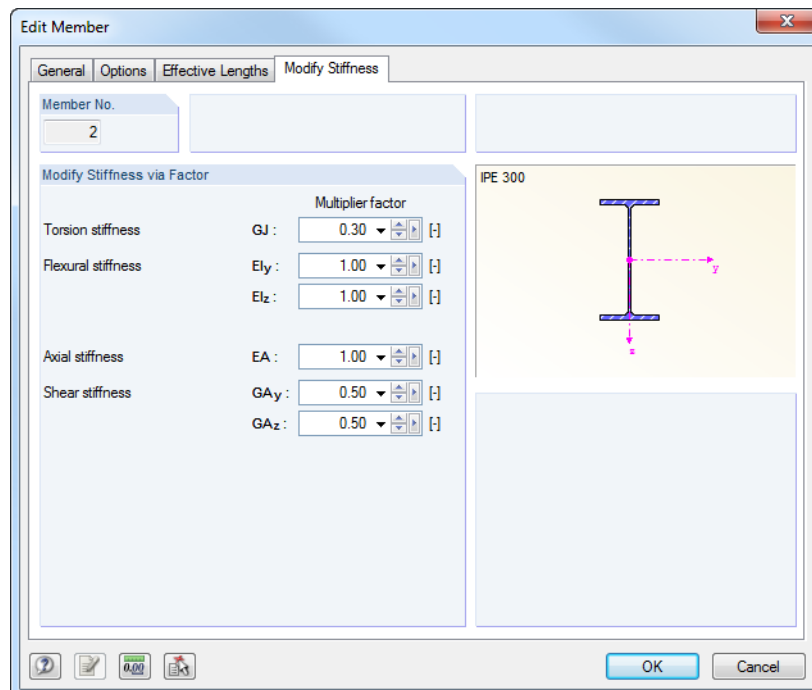


Figure 4.77: Dialog box *Edit Member*, tab *Modify Stiffness*

RSTAB presets a *Multiplier factor* of 1.00 for the torsional, flexural, axial and shear stiffness of the member: All stiffness components are taken into account by the calculation as they stand. If required, it is possible to define a reduction or increase factor in the input fields so that you can adapt the member stiffnesses.



In case changes have also been made for the cross-section stiffnesses (see chapter 4.3, page 55), they are additionally taken into account by the calculation.

### 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, RSTAB will delete the old member automatically.



To prevent RSTAB from deleting already defined members, select *Allow Double Members* on the *Edit* menu. RSTAB will consider the stiffnesses of both members in the calculation.



## 4.8 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 member. The boundary conditions of a member must be considered as well to exclude a double release on the supported node.

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).

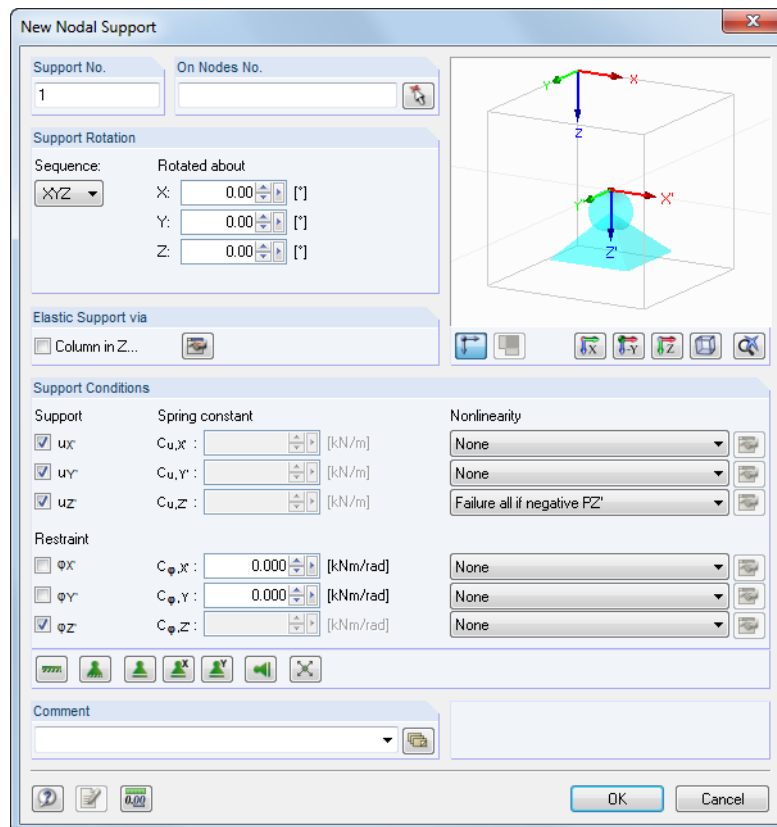


Figure 4.78: Dialog box *New Nodal Support*

Support No.	A	B	Support Rotation [°]			Support or Spring [kN/m]			Rotational Restraint or Spring [kNm/rad]			L
			On Nodes No.	Sequence	about X	about Y	about Z	ux	uy	uz	φx	
1	12.22	XYZ	0.00	0.00	0.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
2	46	XYZ	0.00	0.00	0.00	<input type="checkbox"/>	<input type="checkbox"/>	11320.000	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
3	4	XYZ	0.00	0.00	0.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
4												
5												
6												
7												
8												
9												

Figure 4.79: Table 1.8 *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:

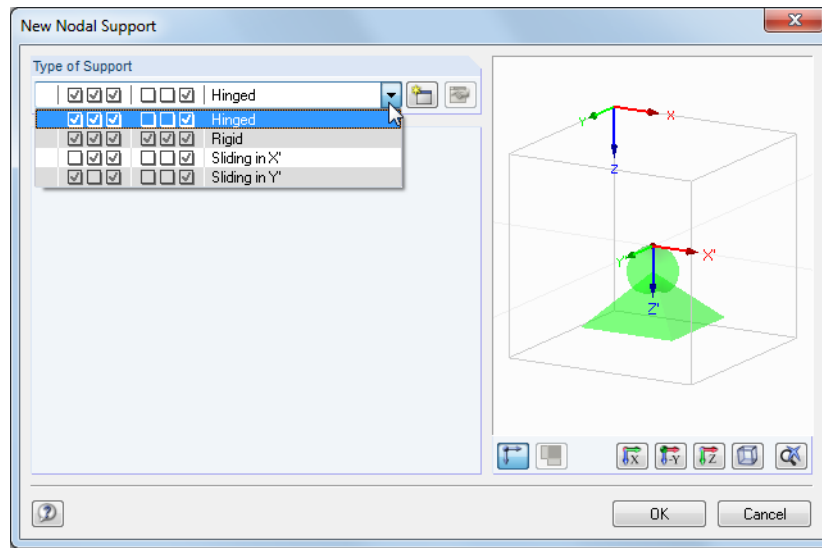


Figure 4.80: Dialog box *New Nodal Support*

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 4.78 appears.

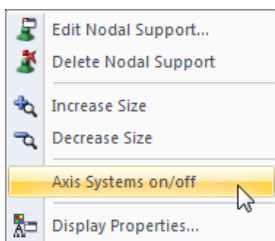
### On nodes

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

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 use the dialog buttons [▶] to define the support rotation also graphically (see Figure 4.81).



Context menu of nodal support

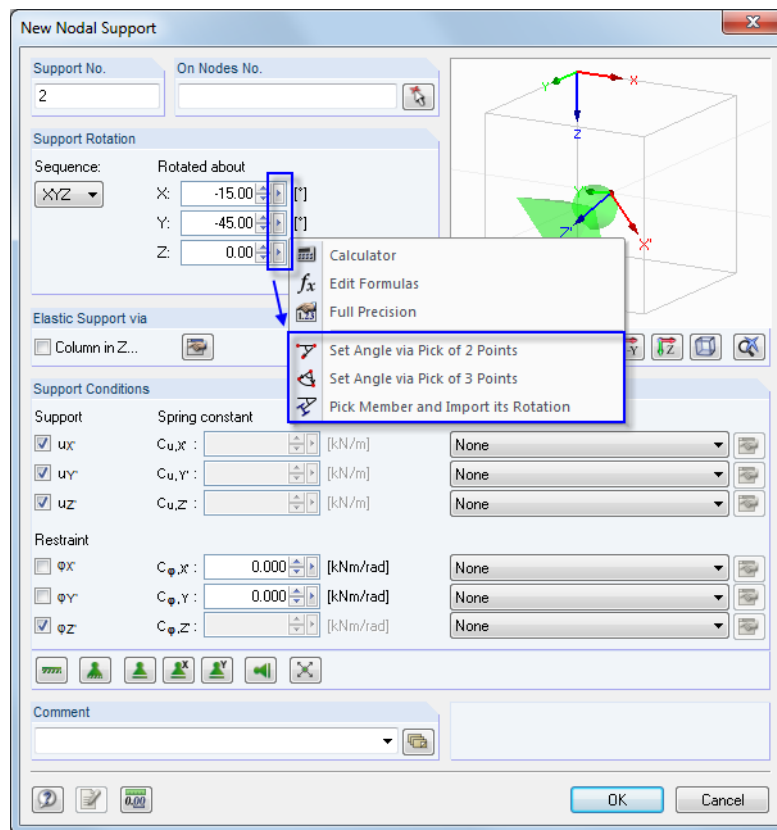


Figure 4.81: Dialog box *New Nodal Support* with options for support rotation

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.

### 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, RSTAB 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 4.78, page 85) provides buttons for different support types, making the definition of degrees of freedom easier.



Figure 4.82: Buttons in the dialog box *New Nodal Support*

The buttons are reserved for the following 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 4.2: Buttons in *Nodal Support* dialog box

### Non-linearities

To control in detail the transfer of internal forces, it is possible to provide nodal supports with nonlinear properties. The list of non-linearities 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 non-linear properties can be accessed in the dialog box and table by using the list (see Figure 4.78 and Figure 4.79). 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 non-linear properties are indicated by a blue check box.

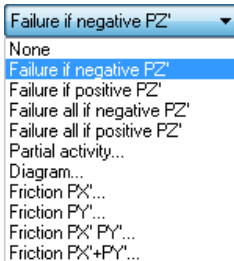
#### Failure if support force/moment is negative or positive

Both options control easily for each support component whether only positive or negative forces/moments are transferred on the supported node. If this condition is not fulfilled, the relevant component of the support fails. The remaining retentions and restraints are still effective.

*Positive or negative* refers to the forces or moments that are introduced to 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 negative or positive

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

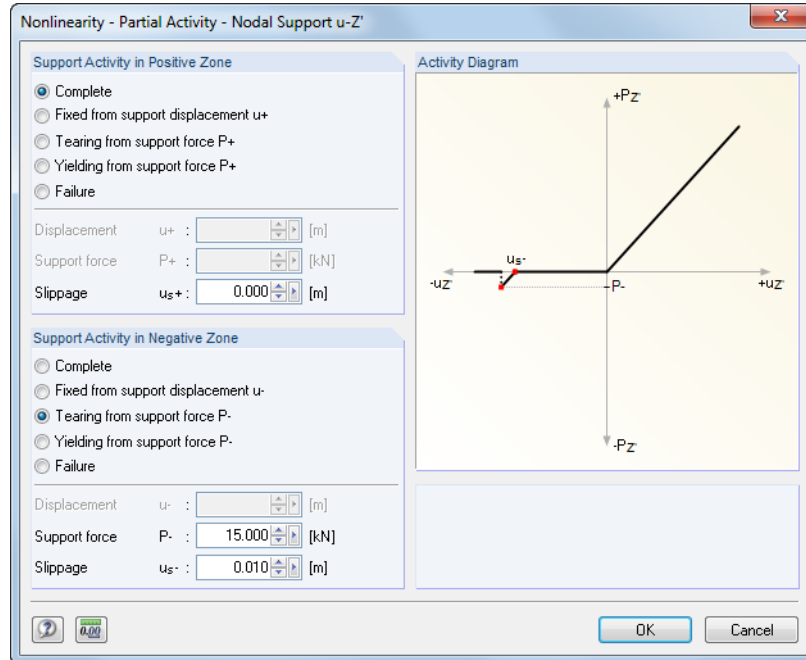


Figure 4.83: Dialog box *Nonlinearity - Partial Activity*

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

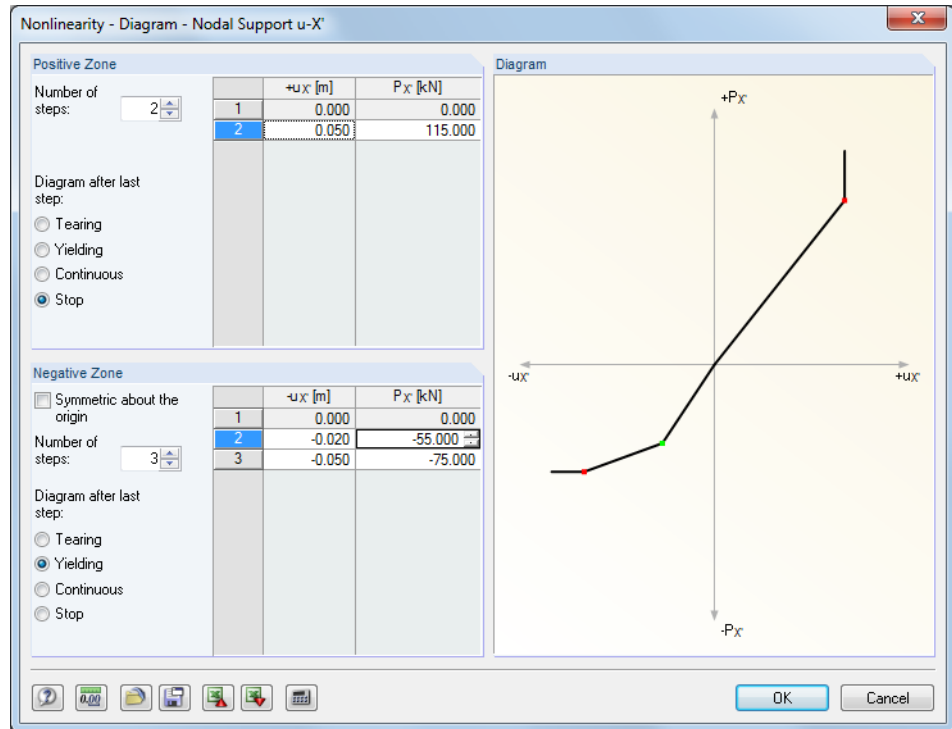


Figure 4.84: Dialog box *Nonlinearity - Diagram*

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*  $\mu$ .

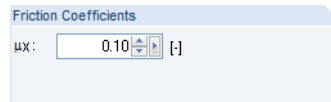
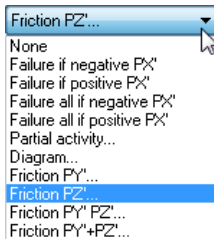


Figure 4.85: Dialog box *Friction in uX'* (dialog section)

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

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

Equation 4.7



## 4.9 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.

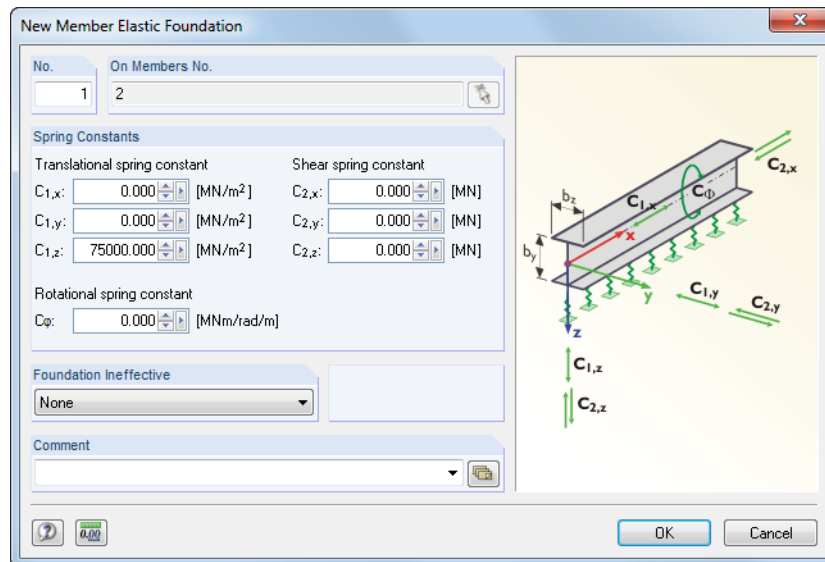


Figure 4.86: Dialog box *New Member Elastic Foundation*

Found. No.	A On Members No.	B C <sub>1,x</sub> [kN/m <sup>2</sup> ]	C C <sub>1,y</sub> [kN/m <sup>2</sup> ]	D C <sub>1,z</sub> [kN/m <sup>2</sup> ]	E C <sub>2,x</sub> [kN]	F C <sub>2,y</sub> [kN]	G C <sub>2,z</sub> [kN]	H C <sub>φ</sub> [kNm/rad/m]	I Foundation Ineffectiveness	J Comment
1	5	0.000	0.000	75000.000	0.000	0.000	0.000	0.000		
2	1,6	0.000	0.000	50000.000	0.000	0.000	0.000	0.000	If contact stress is	
3									None	
4									If contact stress is negative	
5									If contact stress is positive	

Figure 4.87: Table 1.9 *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.

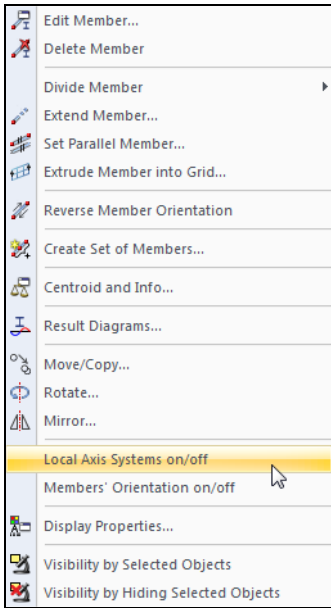
### Spring constants

#### 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 4.3 serve as reference values. Please note that input in RSTAB refers to the foundation module 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 4.3: Stiffness moduli of selected soil types in  $[N/mm^2]$

The values of Table 4.3 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 4.72, page 80).

**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  $\nu$  to be assumed between 0.125 and 0.5 for sand and gravelly 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 not 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 4.72, page 80). 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 7.2, page 176).



## 4.10 Member Nonlinearities

### General description

Member non-linearities are used to represent non-linear relations between force (or moment) and strain in members.

Some non-linear properties can be set already when defining 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 needing a verifiable force for it.

In principle, member non-linearities 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 54).

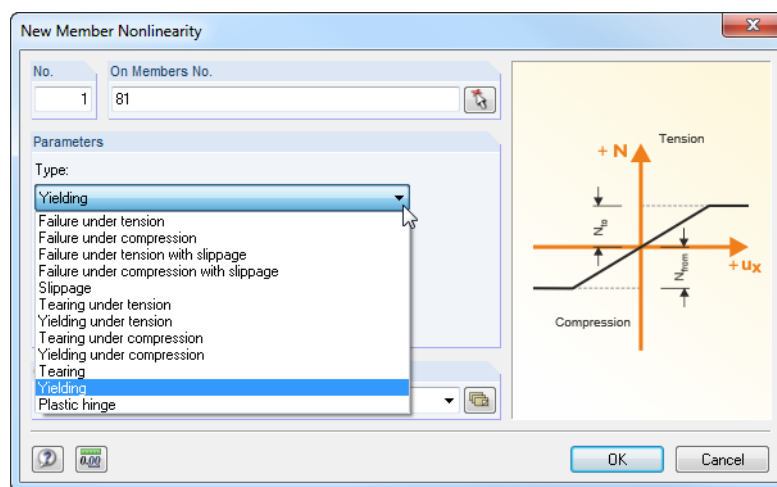
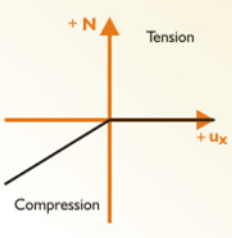
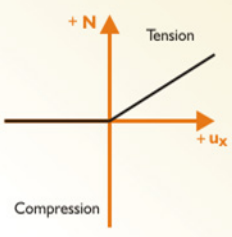
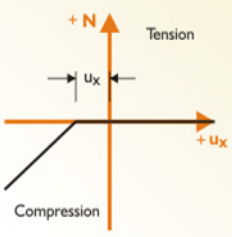
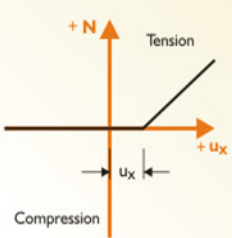
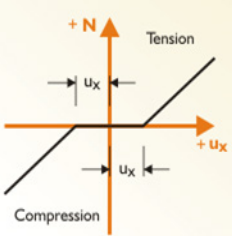
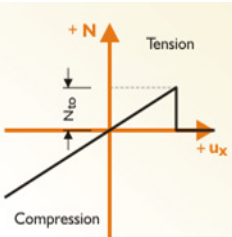


Figure 4.88: Dialog box *New Member Nonlinearity*

1.10 Member Nonlinearities									
Nonlin. No.	On Members No.	Type of Nonlinearity	Nonlinear Parameters						
			$N_{pl}$ [kN]	$V_{y,pl}$ [kN]	$V_{z,pl}$ [kN]	$M_{T,pl}$ [kNm]	$M_{y,pl}$ [kNm]	$M_{z,pl}$ [kNm]	Comment
1	10	Yielding	300.00	300.00					
2	5	Failure under tension							
3	2,3	Failure under tension with slippage							
4	4	Plastic hinge	9999.00	9999.00	9999.00	9999.00	150.00	9999.00	
5									
6									
7									

Figure 4.89: Table 1.10 *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 $u_x$ .
Tearing under tension		The member absorbs compressive forces without limitation but fails if tensile forces exceed $N_{to}$ .



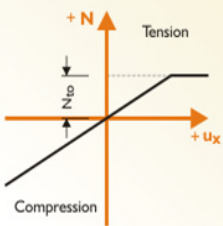
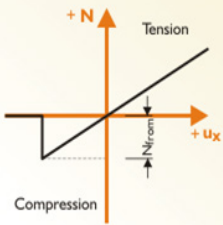
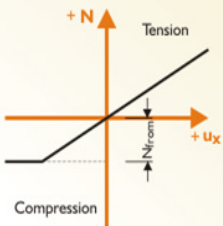
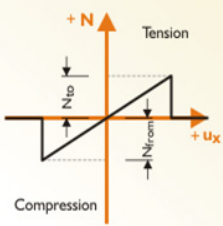
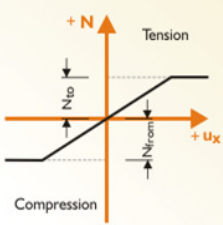
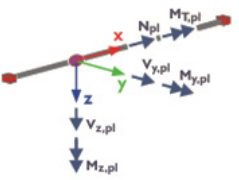
<p>Yielding under tension</p>		<p>The member absorbs compressive forces without limitation, but only a maximum tensile force of <math>N_{to}</math>.</p> <p>If the strain increases, the tensile force remains constant in the member.</p>
<p>Tearing under compression</p>		<p>The member absorbs tensile forces without limitation but fails if compressive forces exceed <math>N_{from}</math>.</p>
<p>Yielding under compression</p>		<p>The member absorbs tensile forces without limitation, but only a maximum compressive force of <math>N_{from}</math>.</p> <p>If the strain increases, the compressive force remains constant in the member.</p>
<p>Tearing</p>		<p>The member fails when reaching the compressive force <math>N_{from}</math> or the tensile force <math>N_{to}</math>.</p>
<p>Yielding</p>		<p>The member starts to yield if the compressive force <math>N_{from}</math> or the tensile force <math>N_{to}</math> are reached: If the strain increases, the force remains constant.</p>
<p>Plastic hinge</p>		<p>If a plastic design force is reached on a location of the member, a plastic hinge is formed there for the internal force.</p> <p>The internal forces must be entered as absolute values. For components of internal forces not resulting in plastifications, you have to enter high values.</p>

Table 4.4: Member nonlinearities

## 4.11 Sets of Members

### General description



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).

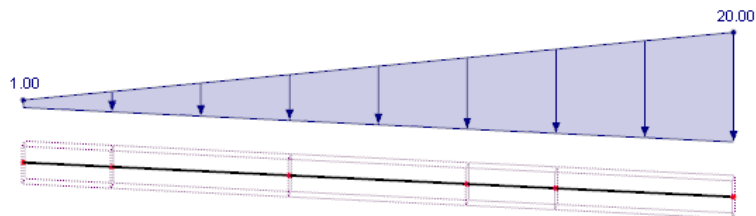


Figure 4.90: Continuous members with trapezoidal load

The figure above shows a linearly variable load acting on the complete length of a set of members.

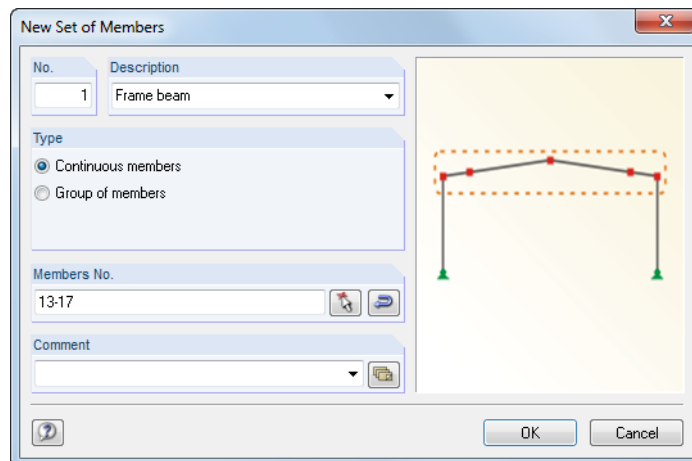


Figure 4.91: Dialog box *New Set of Members*

Set of M. No.	Set of Members Description	Type	Members No.	Length [m]	Weight [kg]	Comment
1	Continuous beam 1	Continuous	51,52	6.000	253.4	
2	Secondary beam A-A	Continuous	13-15	12.548	566.2	
3	Secondary beam B-B	Continuous	41-43	12.548	566.2	
4	Column A-A	Group	33,34,59,60,110,116	22.546	509.0	
5	Frame beam	Continuous	35,36	7.094	174.9	Frame for lateral-torsional buckling analysis
6						
7						

Figure 4.92: Table 1.11 *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 and groups of members.

**Continuous members** are created by connected members that are not branching out. They could be drawn with a pencil without interrupting the continuous line.

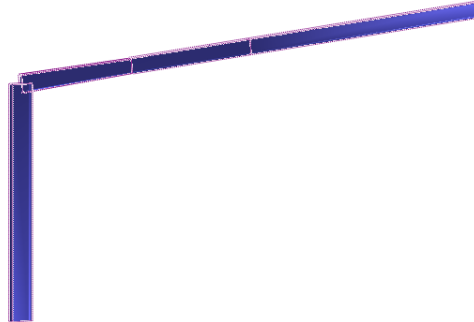


Figure 4.93: Continuous members

A **group of members** consists of connected members that may branch out.

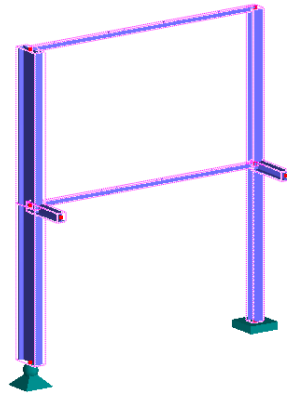


Figure 4.94: Group of members

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 the following: 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 represents 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.



# 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 388).



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, click **Load Cases and Combinations** and select **New Load Case**.
- Use the toolbar button [New Load Case] shown on the left.

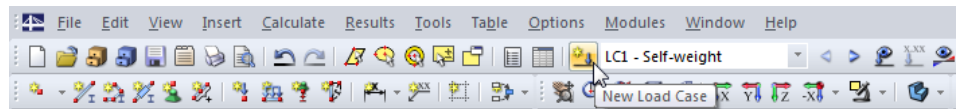


Figure 5.1: Button *New Load Case* in the toolbar

- Use the context menu of the navigator entry *Load Cases*.

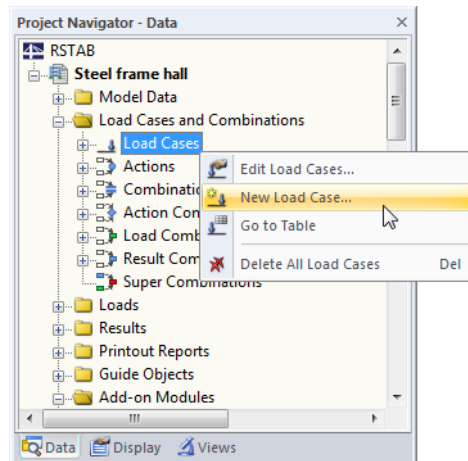


Figure 5.2: Context menu of *Load Cases* in the *Data* navigator

The dialog box *Edit Load Cases and Combinations* appears. A new load case is preset in the dialog tab *Load Cases*.

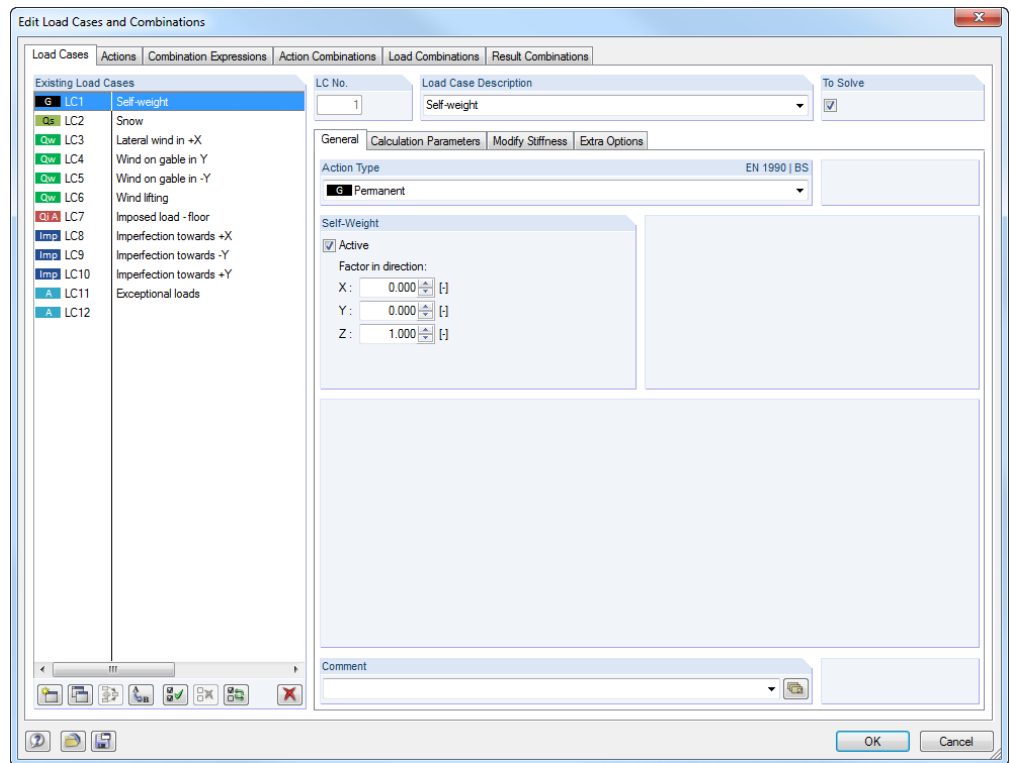


Figure 5.3: Dialog box *Edit Load Cases and Combinations*, tab *Load Cases*

- It is also possible to enter a new load case in an empty row of table 2.1 *Load Cases*.

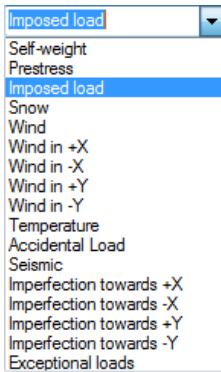
Load Case	Load Case Description	To Solve	Action Category	Self-Weight - Active	Factor in Direction X	Factor in Direction Y	Factor in Direction Z	Comment
LC1	Self-weight	<input checked="" type="checkbox"/>	G Permanent	<input checked="" type="checkbox"/>	0.000	0.000	1.000	
LC2	Snow	<input checked="" type="checkbox"/>	Qs Snow (H ≤ 1000 m a.s.l.)	<input type="checkbox"/>				
LC3	Lateral wind in +X	<input checked="" type="checkbox"/>	Qw Wind	<input type="checkbox"/>				
LC4								
LC5	Imposed load - floor	<input checked="" type="checkbox"/>	Qi A Imposed - category A: domestic, residar	<input type="checkbox"/>				
LC6	Imperfection towards +X	<input checked="" type="checkbox"/>	Imp Imperfection	<input type="checkbox"/>				
LC7	Imperfection towards -Y	<input checked="" type="checkbox"/>	Imp Imperfection	<input type="checkbox"/>				

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, RSTAB 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 [Renummer] dialog button (see Table 5.1 and chapter 11.4.16, page 320).



### Load case description

You can enter any name manually. You can also choose a name from the list to describe the load case shortly.

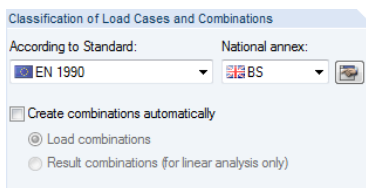
### To solve

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 category

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.

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 388).



Standard settings in the dialog box *Model - General Data*

Action Type		EN 1990   BS
G	Permanent	1
P	Prestress	2
Q1 A	Imposed - category A: domestic, residential areas	3.A
Q1 B	Imposed - category B: office areas	3.B
Q1 C	Imposed - category C: congregation areas	3.C
Q1 D	Imposed - category D: shopping areas	3.D
Q1 E	Imposed - category E: storage areas	3.E
Q1 F	Imposed - category F: traffic area - vehicle weight ≤ 30 kN	3.F
Q1 G	Imposed - category G: traffic area - vehicle weight ≤ 160 kN	3.G
Q1 H	Imposed - category H: roofs	3.H
Qs	Snow (H > 1000 m a.s.l.)	4.A
Qs	Snow (H ≤ 1000 m a.s.l.)	4.B
Qw	Wind	5
Qt	Temperature (non fire)	6
A	Accidental	7
AE	Earthquake	8
Imp	Imperfection	

Figure 5.5: Action categories according to EN 1990, British annex

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 7.2.1 on page 169.



## 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 **Load Cases and Combinations**, and then select **Load Case - General Data** (current load case).
- On the **Edit** menu, point to **Load Cases and Combinations**, 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.

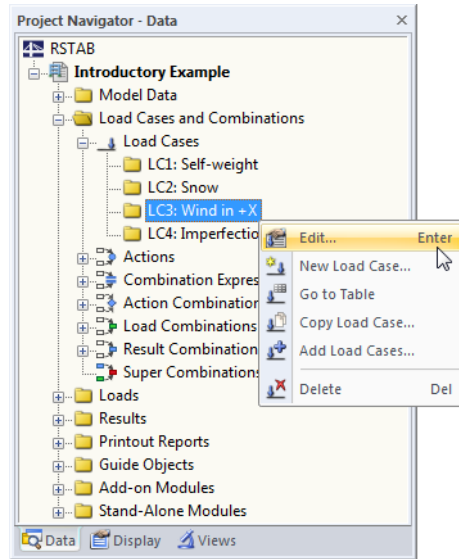


Figure 5.6: Context menu of a load case



- Use the button [Edit load cases] in the toolbar of the loads tables (current load case).

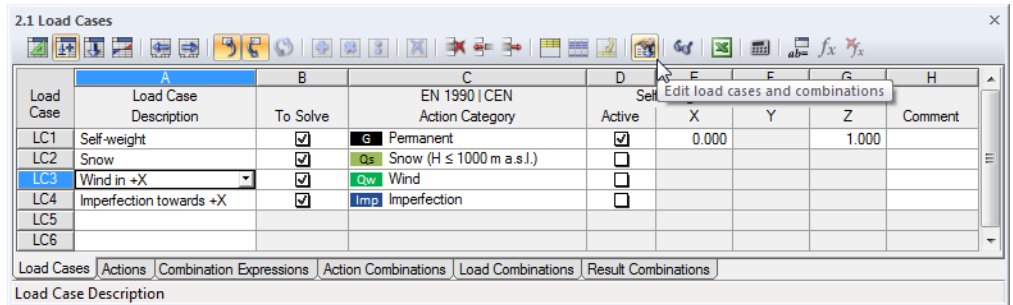
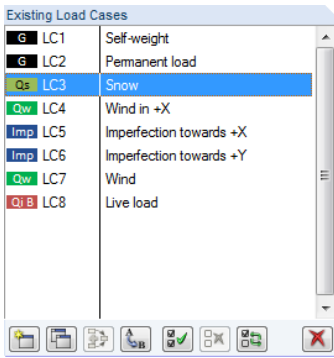


Figure 5.7: Button [Edit load cases and combinations] in the toolbar of the loads tables



### Buttons

In the dialog box *Edit Load Cases and Combinations*, several buttons are available below the load case list (see Figure 5.3, page 99). They are reserved for 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
	Cancel 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 RSTAB 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

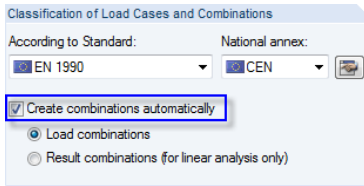
### General description

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 386).

The load cases defined in table 2.1 (see previous chapter 5.1) represent the base data for the automatic superpositioning. RSTAB 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 superpositioning in RSTAB. The action type defined for the load cases (see chapter 5.1, page 100) controls the assignment to action categories conforming to standards.



Check box in dialog box *Model - General Data*

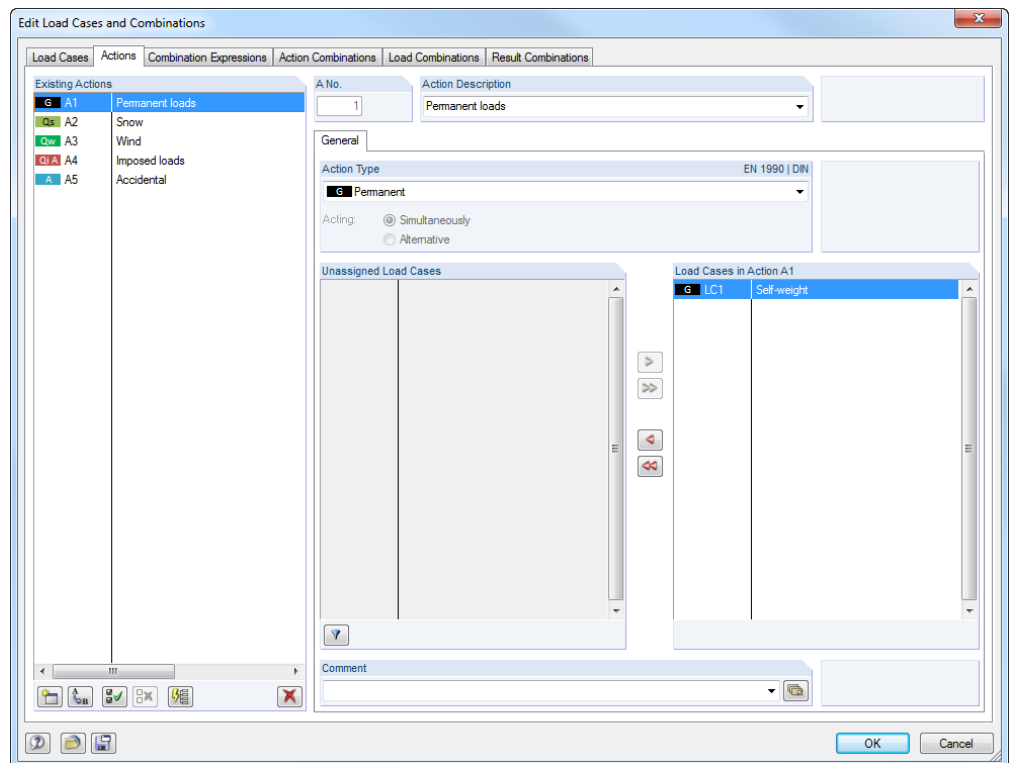
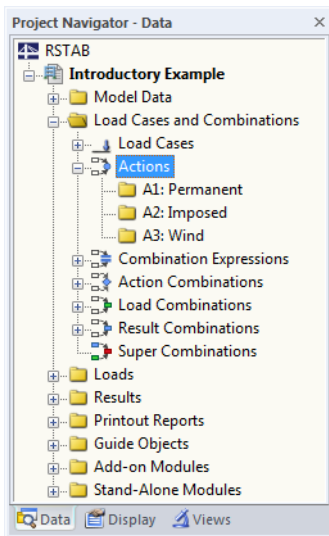


Figure 5.8: Dialog box *Edit Load Cases and Combinations*, tab *Actions*

Action	A		B		C	D				H
	Action Description	EN 1990   DIN Action Category	EN 1990   DIN Action Category	EN 1990   DIN Action Category		Load Cases in Action				
A1	Permanent loads	G Permanent			Acting	LC.1	LC.2	LC.3	LC.4	Comment
A2	Snow	Qs Snow (H ≤ 1000 m a.s.l.)				LC1				
A3	Wind	Qw Wind			Alternative	LC3	LC4	LC5	LC6	
A4	Imposed loads	Q1A Imposed - category A: domestic, residential areas			Alternative	LC7				
A5	Accidental	A Accidental			Simultaneously	LC11				
A6										

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 [Reorder] 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 100).

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 always or never 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 [Left Arrow] 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 not assigned to any action are not taken into account when generating combinations.



EN 1990   CEN Action Category	
G	Permanent
G	Permanent
Q1B	Imposed - category B: office
Qs	Snow (H ≤ 1000 m a.s.l.)



Load Cases in Action		
LC.1	LC.2	LC.3
LC1		
LC4		
LC8	G LC2	Permanent load



### 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* are reserved for the following functions:







	Creates a new action
	Renumbers the selected actions
	Selects all actions
	Cancel 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)

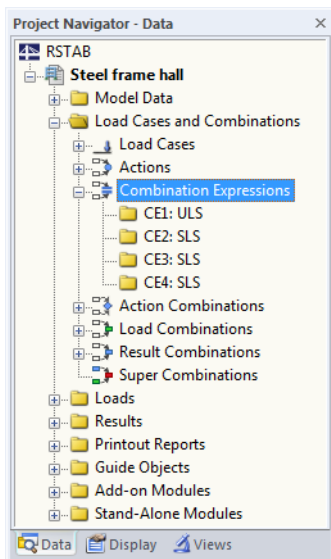
$$\sum_{j \geq 1} G_{k,j} + P + A_d + (\psi_{1,1} \text{ or } \psi_{2,1}) \cdot Q_{k,1} + \sum_{i > 1} \psi_{2,i} \cdot Q_{k,i}$$

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

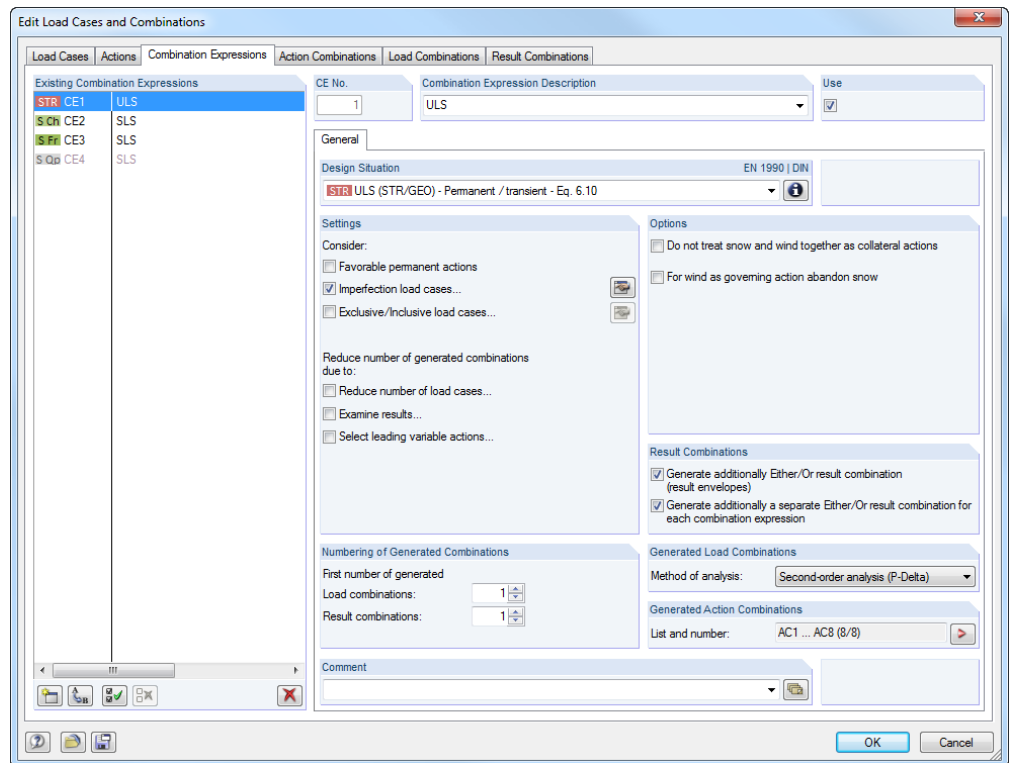
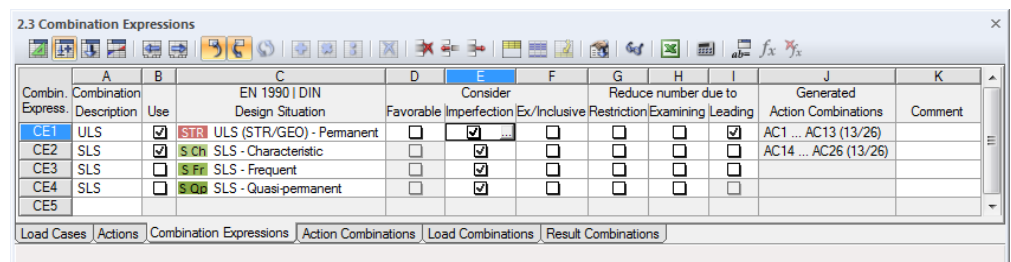


Figure 5.10: Dialog box *Edit Load Cases and Combinations*, tab *Combination Expressions*



Comb. Express.	Combination Description	Use	EN 1990   DIN Design Situation	Consider Favorable	Imperfection	Ex/Inclusive	Restriction	Examining	Leading	Generated Action Combinations	Comment
CE1	ULS	<input checked="" type="checkbox"/>	STR ULS (STR/GEO) - Permanent	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/>	AC1 ... AC13 (13/26)	
CE2	SLS	<input checked="" type="checkbox"/>	S Ch SLS - Characteristic	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	AC14 ... AC26 (13/26)	
CE3	SLS	<input type="checkbox"/>	S Ff SLS - Frequent	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		
CE4	SLS	<input type="checkbox"/>	S Qd SLS - Quasi-permanent	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>		
CE5											

Figure 5.11: Table 2.3 *Combination Expressions*

### Combination expression

When you access the dialog box or table, RSTAB 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:

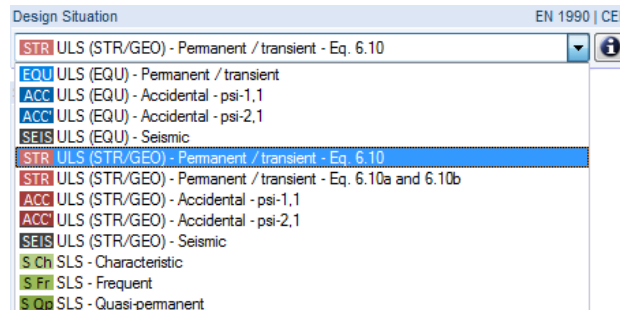
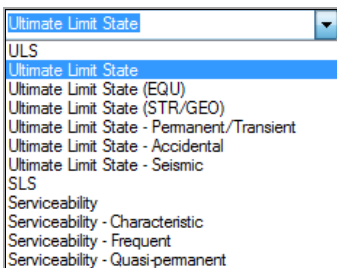


Table 5.12: Design situations according to EN 1990



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).



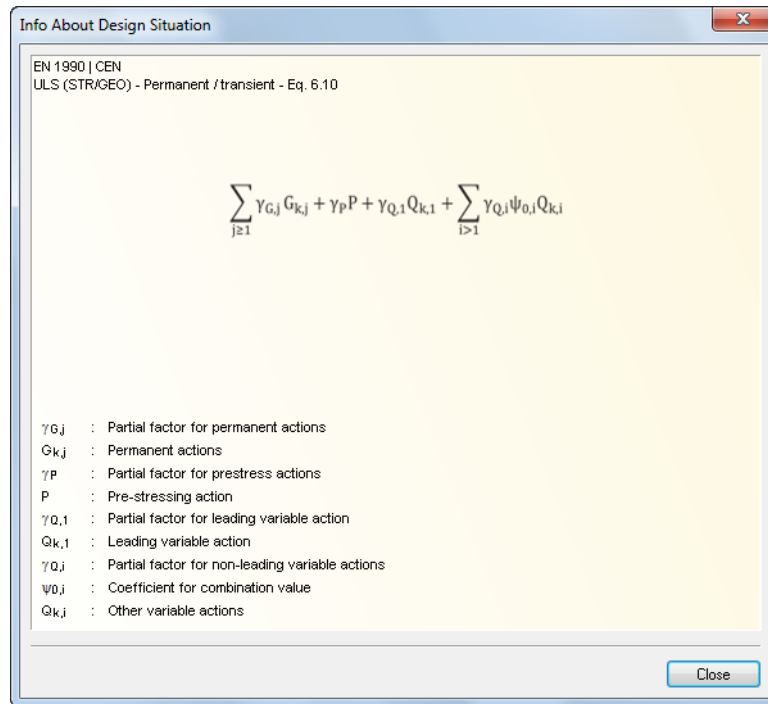
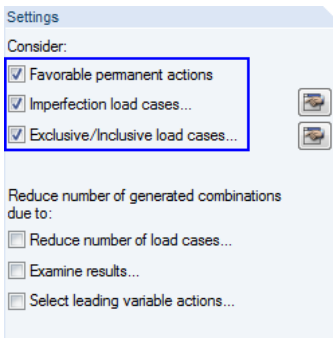


Figure 5.13: Dialog box *Info About Design Situation*



## Favorable permanent actions

Due to this option, RSTAB 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

RSTAB 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.

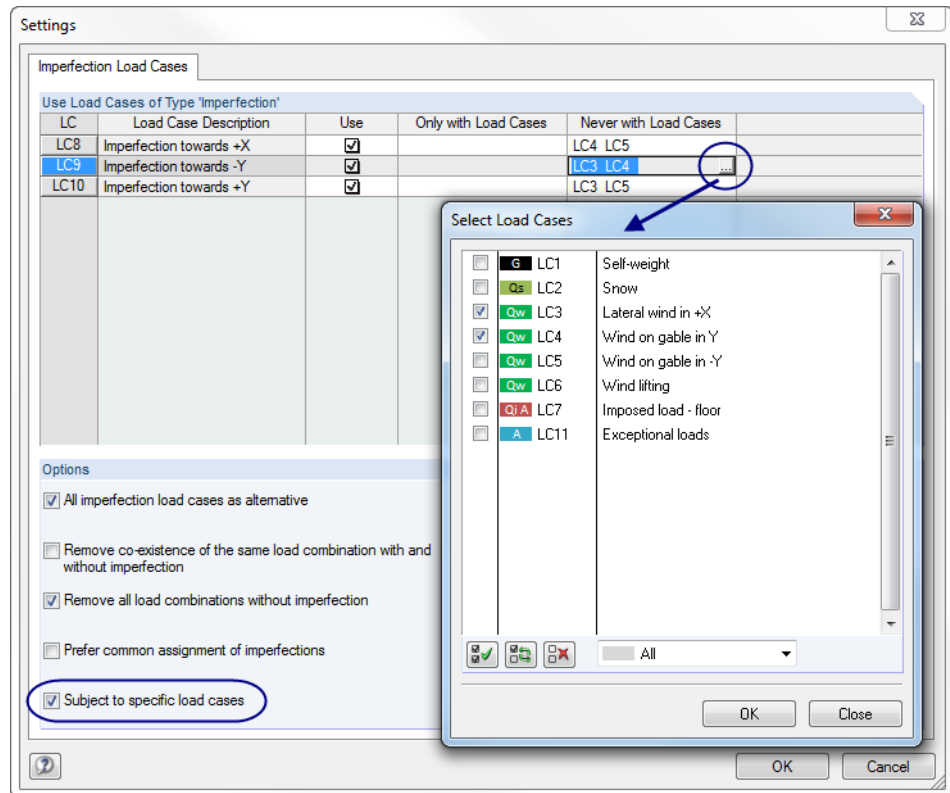


Figure 5.14: Dialog box Settings with dialog box Select Load Cases for selecting load cases

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 100). 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*, RSTAB 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.

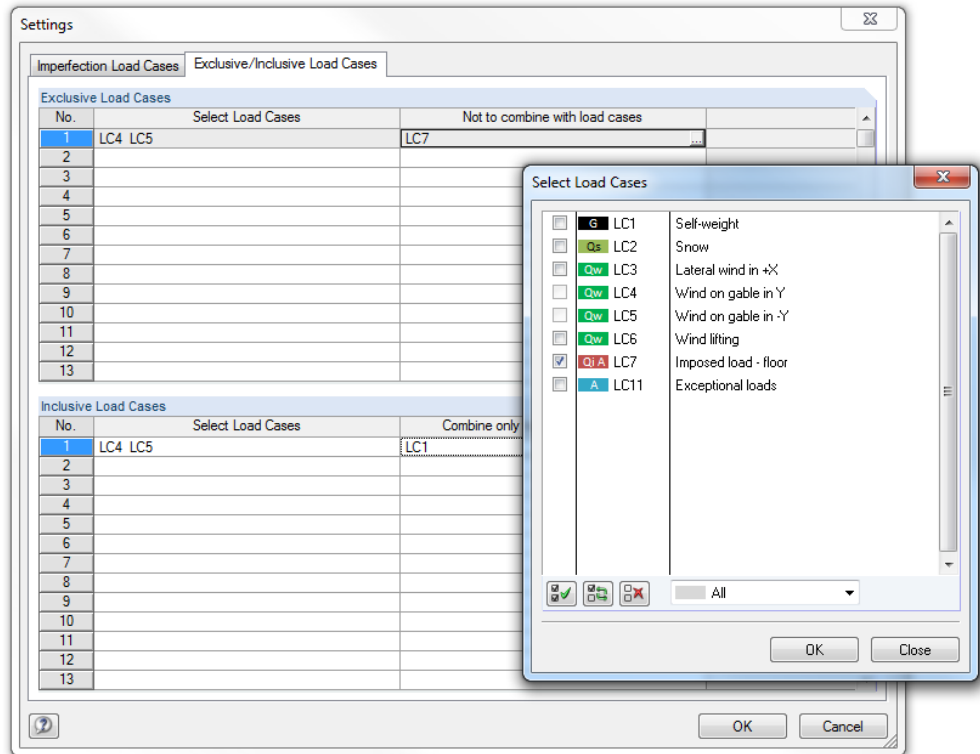


Figure 5.15: Dialog box Settings, tab Exclusive/Inclusive Load Cases with dialog box Select Load Cases



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 every 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 generated combinations. RSTAB 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.

#### 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 107). RSTAB 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 positive effect 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.

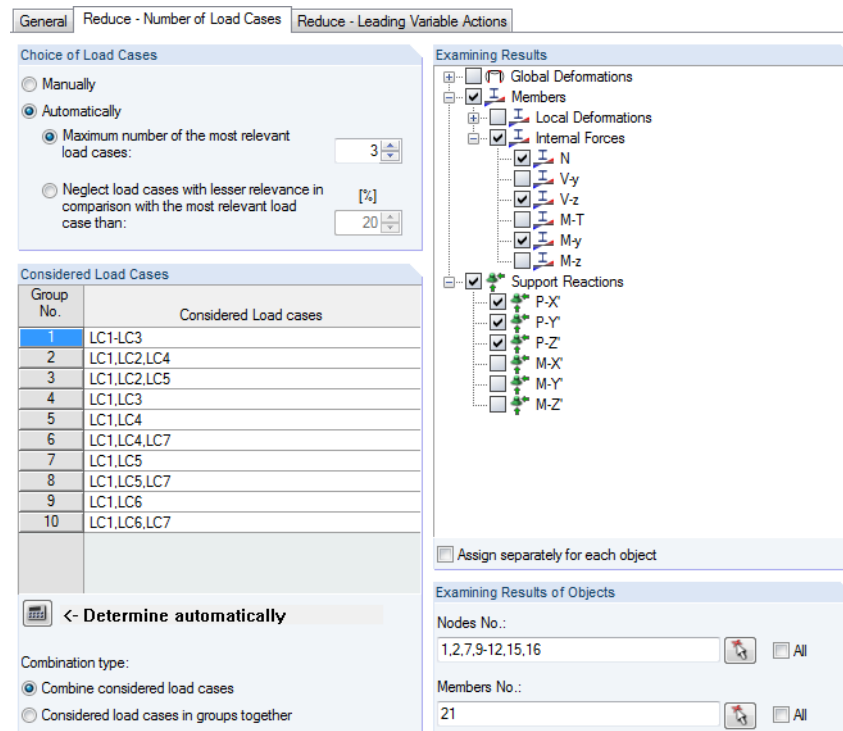
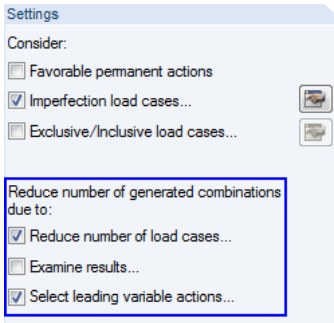


Figure 5.16: Dialog tab *Reduce - Number of Load Cases* for combination expressions

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, member internal forces, support reactions) and *Objects* (nodes, members) you want to consider for the evaluation of the load cases. The nodes and members 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 nodes and members 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**

RSTAB 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.

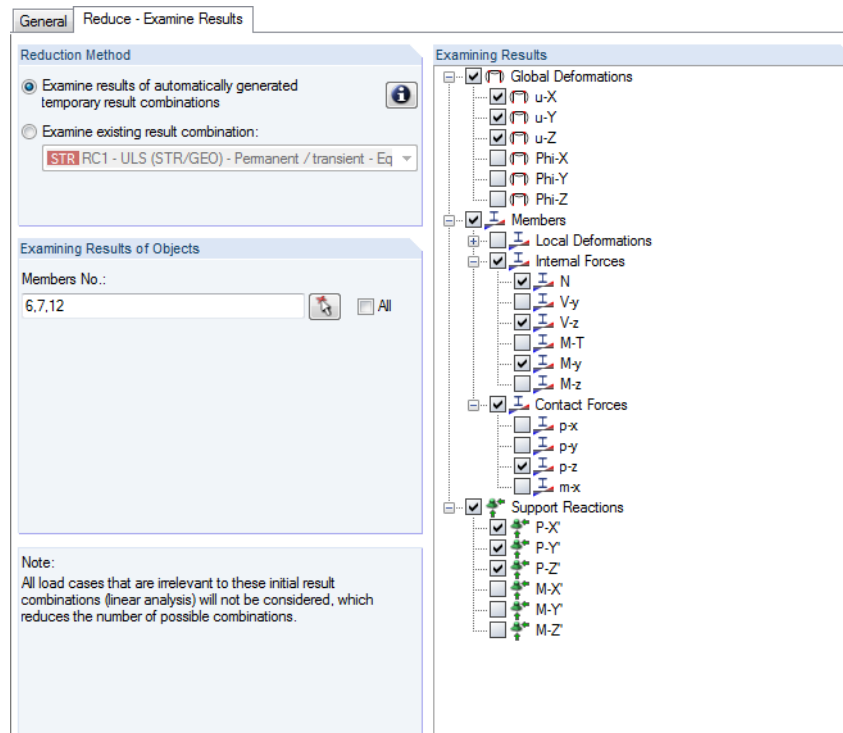
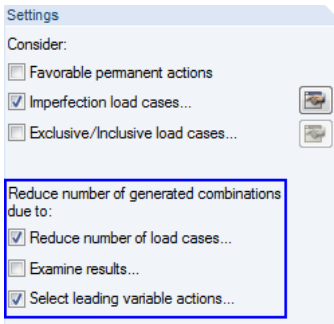


Figure 5.17: Tab *Reduce - Examine Results* for combination rules

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 the x-locations RSTAB 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 or support reactions you want to take into account for the determination of extreme values.



The dialog section *Examining Results of Objects* provides an option to restrict the extreme value analysis to results of selected members. Use the [^] function to select the members 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.

General   Reduce - Examine Results   Reduce - Leading Variable Actions					
Select Leading Variable Actions					
Action	Action Description	EN 1990   DIN Action Category	Load Cases in Action	Leading Actions	
A2	Snow	Qs Snow (H ≤ 1000 m a)	LC2	<input checked="" type="checkbox"/>	
A3	Wind	Qw Wind	LC3 ... LC6	<input type="checkbox"/>	
A4	Imposed loads	QjA Imposed - category /	LC7	<input checked="" type="checkbox"/>	

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 107) affects the *First number of generated Load combinations* or *Result combinations* that are created in RSTAB.

**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 7.2.1.1, page 169). RSTAB presets the non-linear 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, RSTAB 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 dialog tab with the dialog button [▶], RSTAB 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.

## 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.

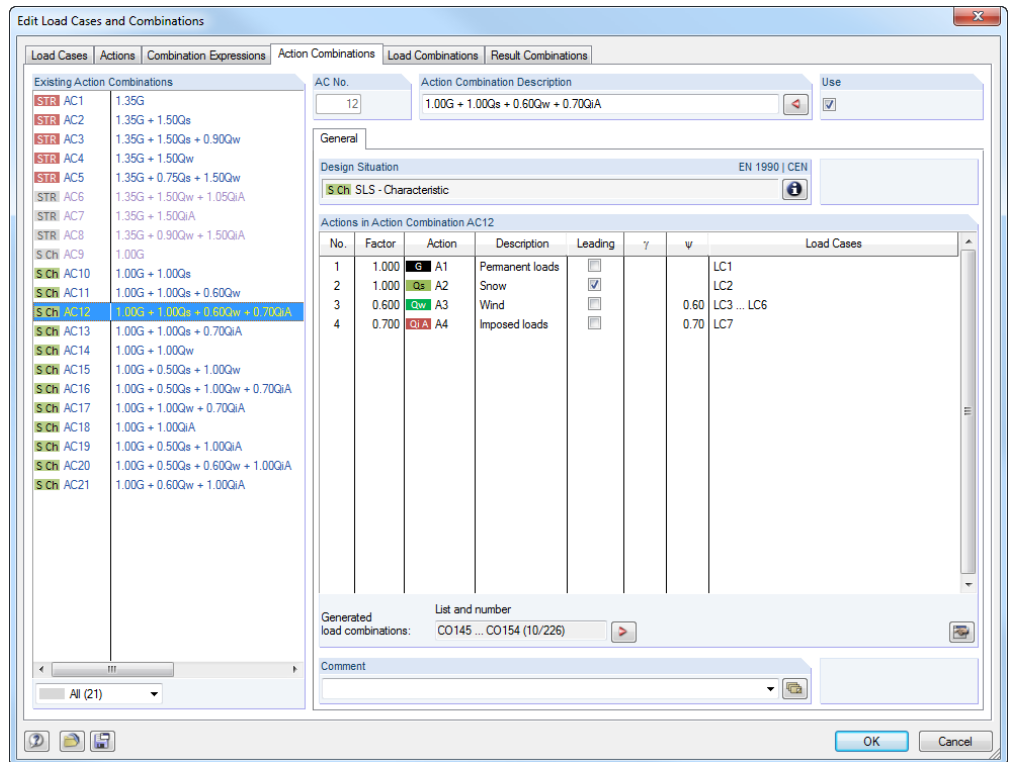
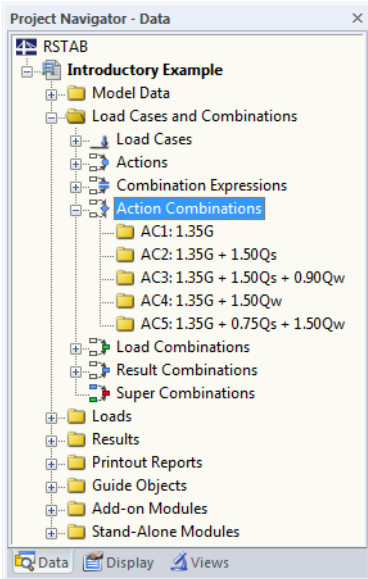


Figure 5.19: Dialog box *Edit Load Cases and Combinations*, tab *Action Combinations*

Action Combin.	A	B	C	D	E	F	G	H	I	J	K	L
	Action Combination Description	Use	EN 1990   CEN Design Situation	Action.1 Factor	Action.2 No.	Action.2 Factor	Action.2 No.	Action.3 Factor	Action.3 No.	Action.4 Factor	Action.4 No.	Generated Load Combinations
AC1	1.35G	<input checked="" type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1							CO1 ... CO8, CO13 ... CO16, CO9 ... CO12, CO17 ... CO20
AC2	1.35G + 1.50Qs	<input checked="" type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	1.50	Qs A2					CO21 ... CO22, CO35 ... CO3
AC3	1.35G + 1.50Qs + 0.90Qw	<input checked="" type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	1.50	Qs A2	0.90	Qw A3			CO23 ... CO24, CO37 ... CO3
AC4	1.35G + 1.50Qw	<input checked="" type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	1.50	Qw A3					CO25 ... CO26, CO39 ... CO4
AC5	1.35G + 0.75Qs + 1.50Qw	<input checked="" type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	0.75	Qs A2	1.50	Qw A3			
AC6	1.35G + 1.50Qw + 1.05QjA	<input type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	1.50	Qw A3	1.05	Qj A4			
AC7	1.35G + 1.50QjA	<input type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	1.50	Qj A4					
AC8	1.35G + 0.90Qw + 1.50QjA	<input type="checkbox"/>	STR ULS (STR/GEO)	1.35	G A1	0.90	Qw A3	1.50	Qj A4			
AC9	1.00G	<input checked="" type="checkbox"/>	S Ch SLS - Characterist	1.00	G A1							
AC10	1.00G + 1.00Qs	<input checked="" type="checkbox"/>	S Ch SLS - Characterist	1.00	G A1	1.00	Qs A2					CO121 ... CO124 (4/226)
AC11	1.00G + 1.00Qs + 0.60Qw	<input checked="" type="checkbox"/>	S Ch SLS - Characterist	1.00	G A1	1.00	Qs A2	0.60	Qw A3			CO125 ... CO134 (10/226)
AC12	1.00G + 1.00Qs + 0.60Qw	<input checked="" type="checkbox"/>	S Ch SLS - Characterist	1.00	G A1	1.00	Qs A2	0.60	Qw A3	0.70	Qj A4	CO135 ... CO144 (10/226)
AC13	1.00G + 1.00Qs + 0.70QjA	<input checked="" type="checkbox"/>	S Ch SLS - Characterist	1.00	G A1	1.00	Qs A2	0.70	Qj A4			CO145 ... CO148 (4/226)

Figure 5.20: 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.

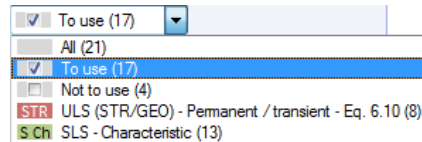


Figure 5.21: Filter option in the dialog box *Edit Load Cases and Combinations*

### Description of action combination

RSTAB assigns automatically brief descriptions based on the safety factors and symbols of actions, expressing combination rules. You can change the descriptions, if necessary.



Click the dialog button [◀] to jump to the previous dialog tab where RSTAB 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 RSTAB 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 109).

### 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 106).

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  $\gamma$ , the combination factors  $\psi$ , the reduction factors  $\xi$  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 388. The tab *Combination Coefficients* manages the factors  $\psi$  and  $\xi$ .

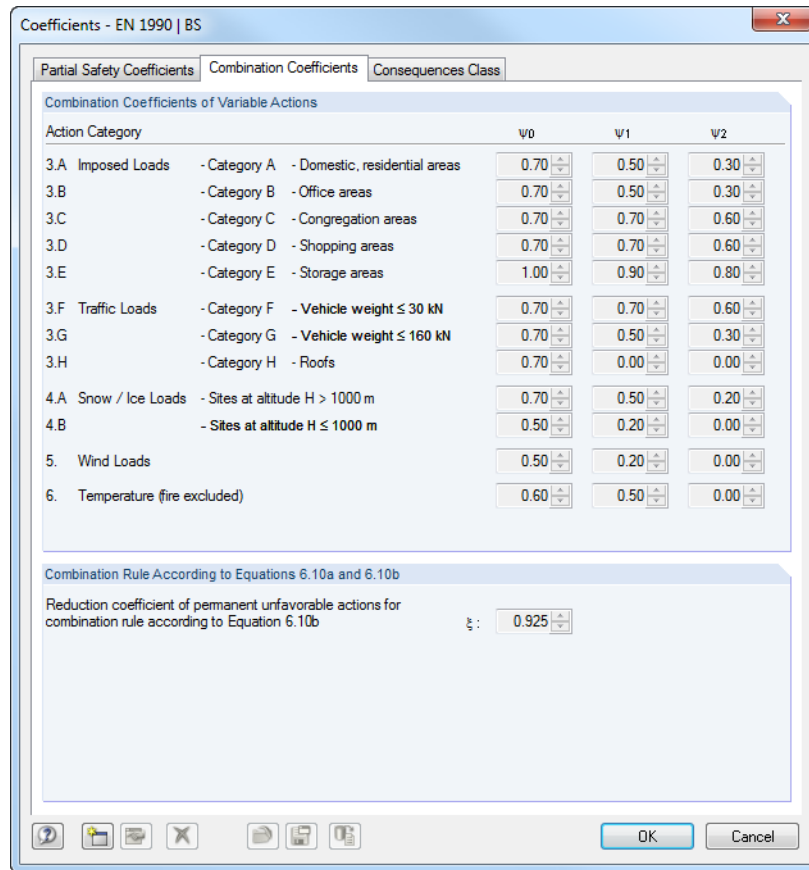


Figure 5.22: Dialog box *Coefficients*, tab *Combination Coefficients*

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.

Generated Load Combinations
CO1 ... CO3 (3/47)
CO4 ... CO6 (3/47)
CO7 ... CO10 (4/47)
CO11 ... CO14 (4/47)
CO15 ... CO17 (3/47)
CO18 ... CO21 (4/47)
CO22 ... CO25 (4/47)
CO26 ... CO29 (4/47)
CO30 ... CO33 (4/47)
CO34 ... CO36 (3/47)
CO37 ... CO39 (3/47)
CO40 ... CO43 (4/47)
CO44 ... CO47 (4/47)

**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  $\gamma = 1.35$  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  $\gamma = 1.50$  in all four load combinations. This action is a leading action.

Figure 5.23: 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 RSTAB 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

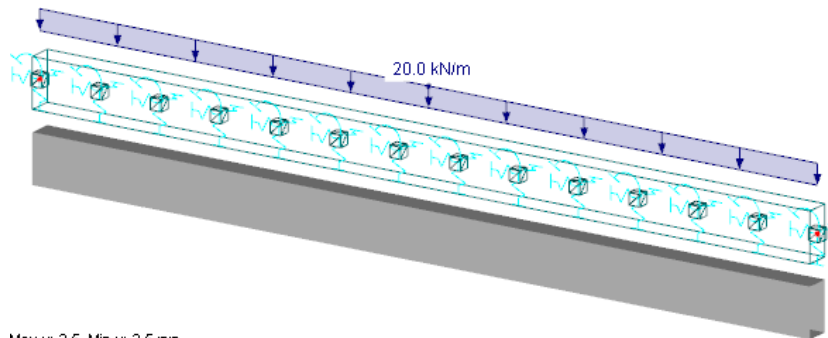
Load cases can be superimposed in a load combination (CO) and in a result combination (RC).

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 128) all included load cases are calculated first. Then, results will be superimposed, taking into account the partial safety factors.

Load cases can be combined manually (see chapter 5.5.1) or superimposed automatically by RSTAB (see chapter 5.5.2), depending on your settings in the dialog box *Model - General Data* (see Figure 12.23, page 386). 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.

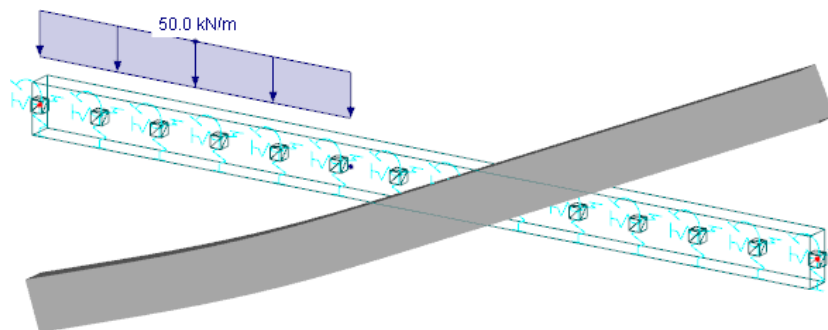
A beam with an elastic foundation is stressed by two different load cases: In load case 1, the member load acts on the entire beam. In load case 2, the load acts only on a part of the member. The self-weight is not taken into account. The member elastic foundation is ineffective in case of tension. Therefore, no lifting forces are absorbed.



Max u: 2.5, Min u: 2.5 mm

Figure 5.24: Load and deformation in LC 1

The foundation in load case 1 is effective for the entire length of the member.

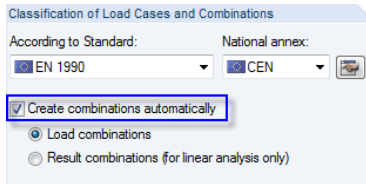


Max u: 6.4, Min u: 0.1 mm

Figure 5.25: Load and deformation in LC 2

In load case 2, the elastic foundation is effective only for the left part of the member. The right part of the member is lifted.

Difference between  
load and result combination

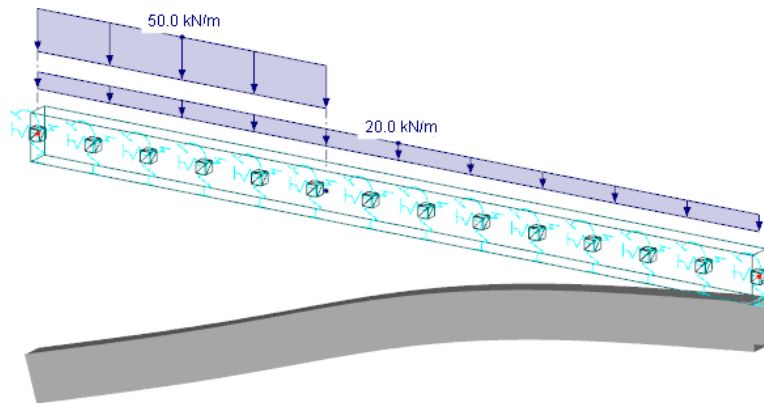


Check box in dialog box  
*Model - General Data*



When combining both load cases in a result combination, RSTAB 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. In a result combination, the lift of the right part resulting from load case 2 would be visible.

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.



Max u: 8.5, Min u: 1.7 mm

Figure 5.26: 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 select **Load Combination**
- use the toolbar button [New Load Combination] shown on the left

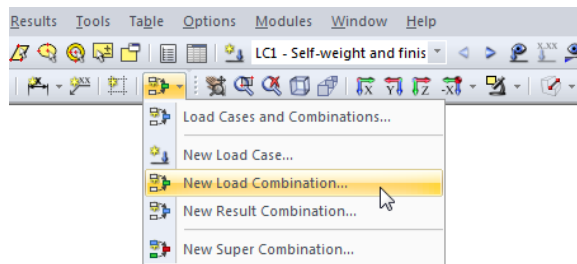


Figure 5.27: Button *New Load Combination* in the toolbar

- context menu of navigator entry *Load Combinations*

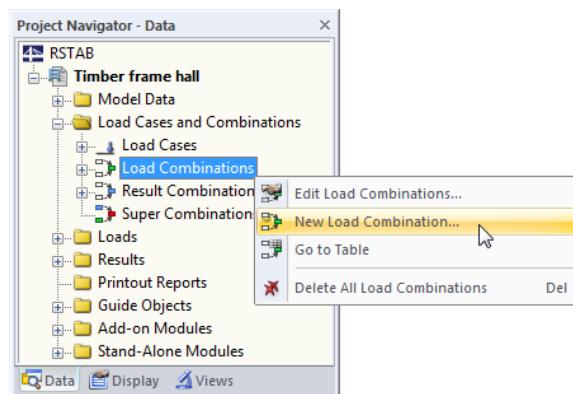


Figure 5.28: Context menu for *Load Combinations* in the *Data* navigator

The dialog box *Edit Load Cases and Combinations* appears. A new load combination is preset in the dialog tab *Load Combinations*.

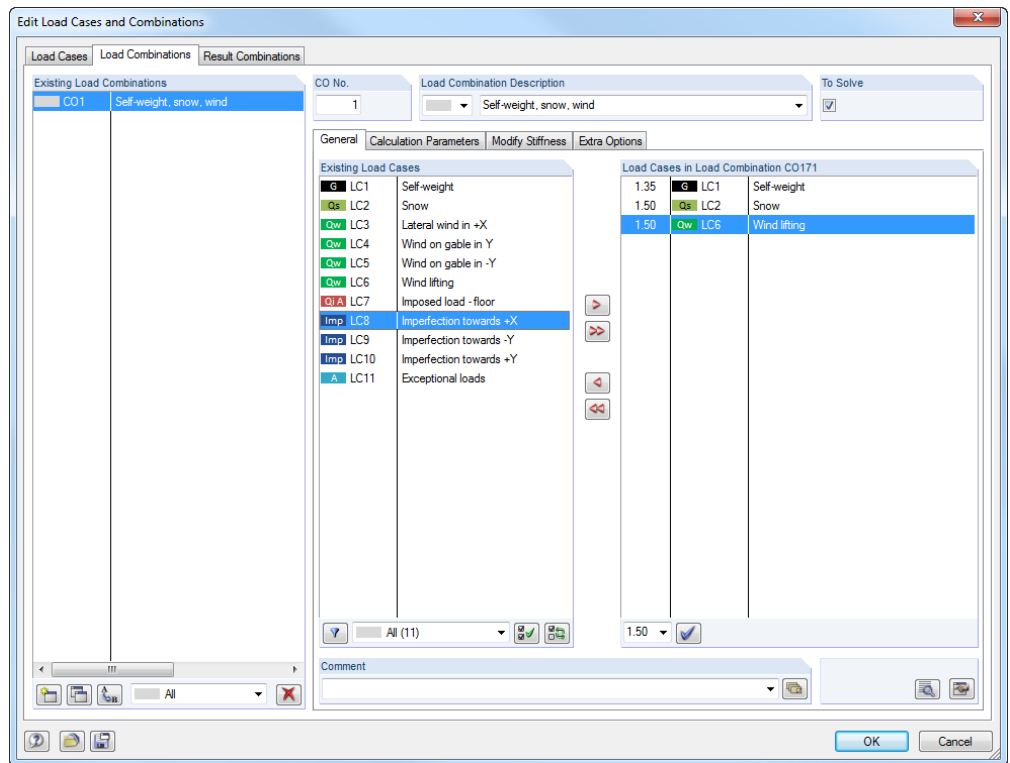


Figure 5.29: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

The following description refers to the tab *General*. The dialog tab *Calculation Parameters* is described in chapter 7.2.1 on page 169.

- It is also possible to enter a new load combination in an empty row of table 2.5 *Load Combinations*.

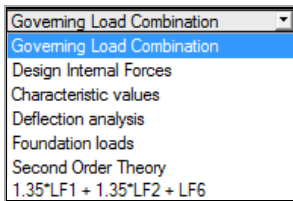
Load Combin.	DS	Description	To Solve	Factor	LC.1 No.	LC.2 Factor No.	LC.3 Factor No.	LC.4 Factor No.	
CO168	STR	1.35*LC1 + 1.5*LC7 + LC8	<input type="checkbox"/>	1.35	G LC1	1.50	Qi A LC7	1.00	Imp LC8
CO169	STR	1.35*LC1 + 1.5*LC7 + LC9	<input type="checkbox"/>	1.35	G LC1	1.50	Qi A LC7	1.00	Imp LC9
CO170	STR	1.35*LC1 + 1.5*LC7 + LC10	<input type="checkbox"/>	1.35	G LC1	1.50	Qi A LC7	1.00	Imp LC10
CO171		Selfweight, snow, wind	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qs LC2	0.90	Qw LC6
CO172		Selfweight, snow	<input checked="" type="checkbox"/>	1.35	G LC1	1.50	Qs LC2		
CO173									
CO174									

Figure 5.30: 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 [Renummer] dialog button (see Table 5.3 and chapter 11.4.16, page 320).





### 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 all other structures.

### 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  $\gamma$ , the combination factors  $\psi$ , the reduction factors  $\xi$  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, use the dialog button [Factors] or the table button [...]. 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 388. The tab *Combination Coefficients* manages the factors  $\psi$  and  $\xi$  (see Figure 5.22, page 118). 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 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.

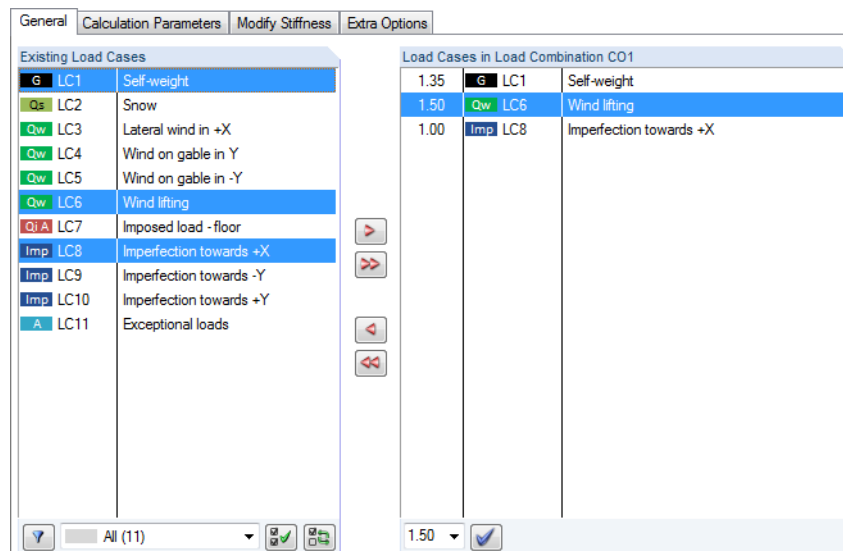
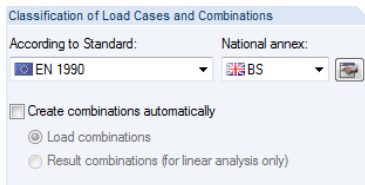


Figure 5.31: Multiple selection of load cases and load combination created according to EN 1990

The factors are created in accordance with the standard set in the dialog box *Model - General Data* (see chapter 12.2.1, page 388).



Standard settings in the dialog box *Model - General Data*



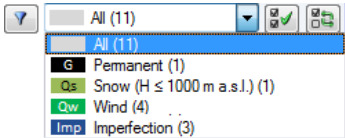
The preset factors can be checked in the *Coefficients* dialog box that you open with the button [Factors]. Furthermore, you can adjusted them for user-defined standards (see Figure 5.22, page 118 and Figure 12.27, page 388).



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 the 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 5.3 on page 125.

To define load combinations manually, use the [Edit] button in the bottom right corner of the loading dialog box.

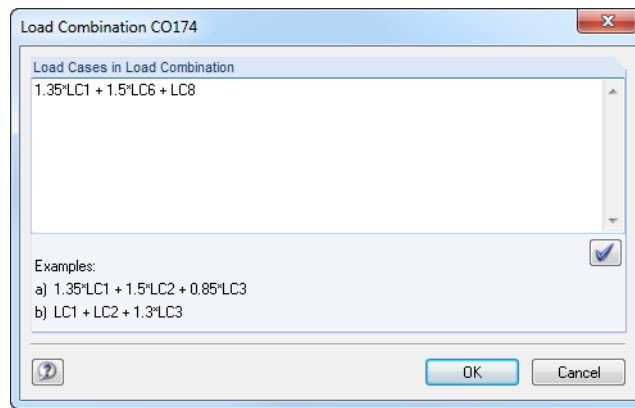


Figure 5.32: Dialog box *Load Combination* for definition via editing field

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 the 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 7.2.1 on page 169.



### 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

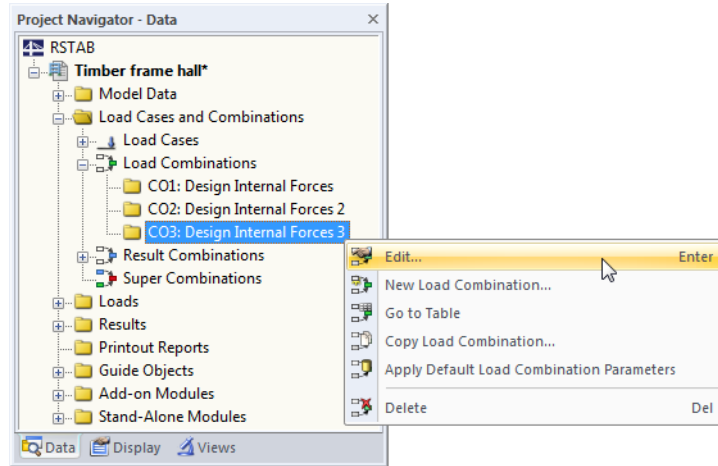


Figure 5.33: Context menu of a load combination

In the dialog box *Edit Load Cases and Combinations* (see Figure 5.29, page 122), 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*. They are reserved for 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 5.3: Buttons in the dialog tab *Load Combinations*

### 5.5.2 Generated Combinations

When switching to the dialog tab *Load Combinations* or to table 2.5, RSTAB creates the combinations automatically. As the load cases are not superimposed manually, the *General* tab looks differently (see Figure 5.29, page 122 for user-defined combinations).

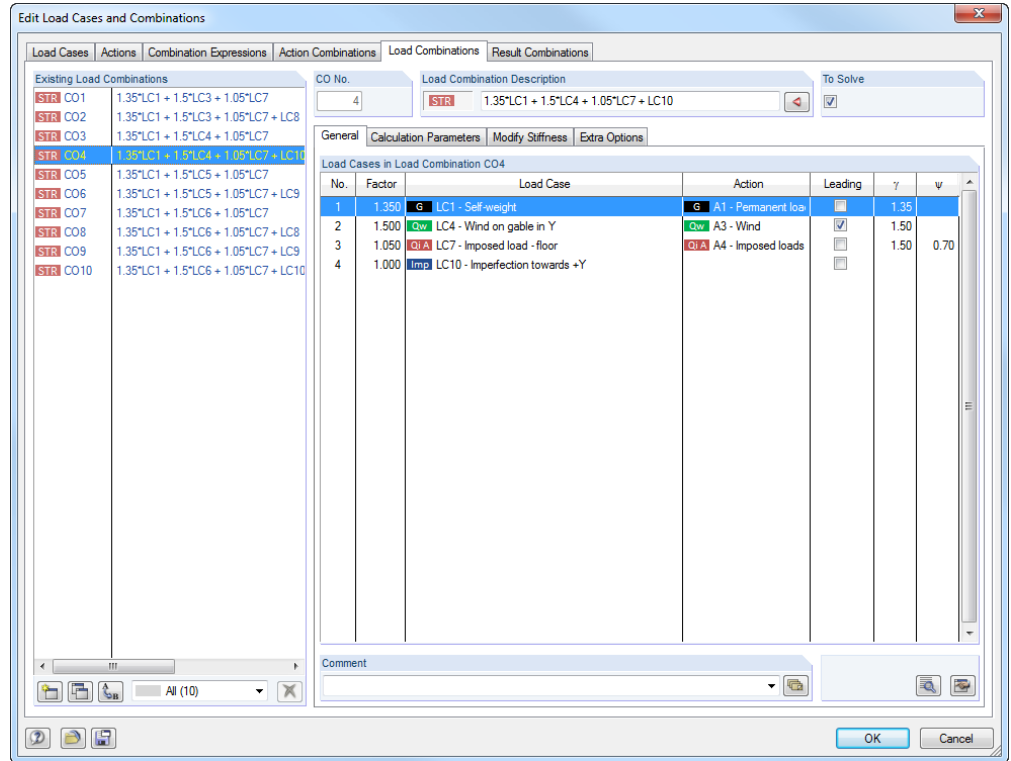
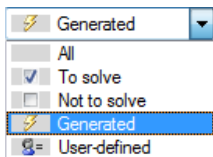


Figure 5.34: Dialog box *Edit Load Cases and Combinations*, tab *Load Combinations*

#### 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

RSTAB 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 116) 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 388 and Figure 5.22, page 118).

### 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.

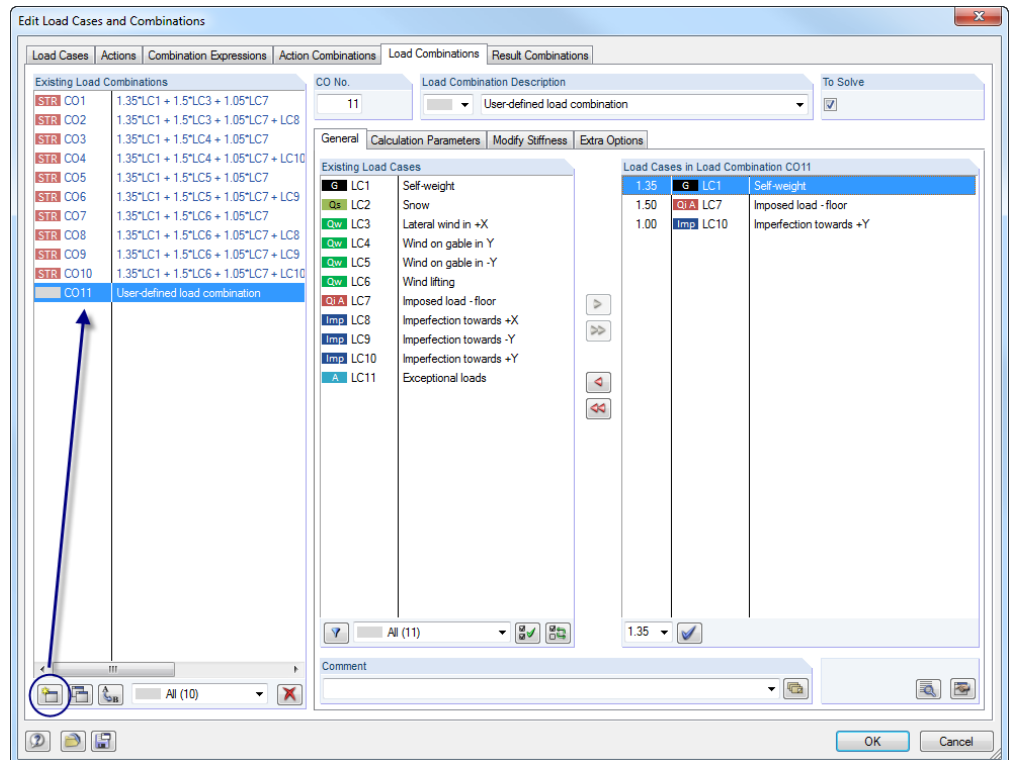


Figure 5.35: Adding a user-defined load combination

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**).

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 120) 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 RSTAB (see chapter 5.7), depending on your settings in the dialog box *Model - General Data* (see Figure 12.23, page 386). 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 non-linear 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 120).

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 102) 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

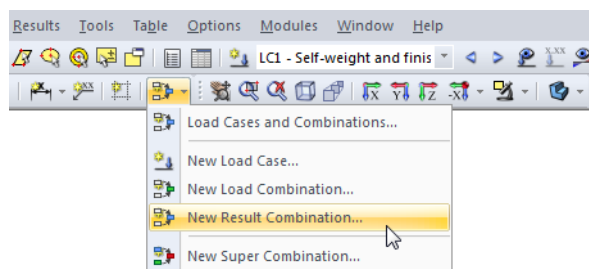
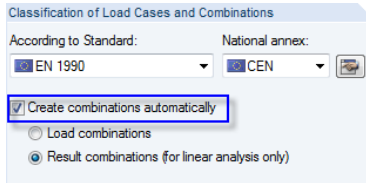


Figure 5.36: Button *New Result Combination* in the toolbar

Difference between result and load combination



Check box in dialog box *Model - General Data*



- use the context menu of the navigator entry *Result Combinations*

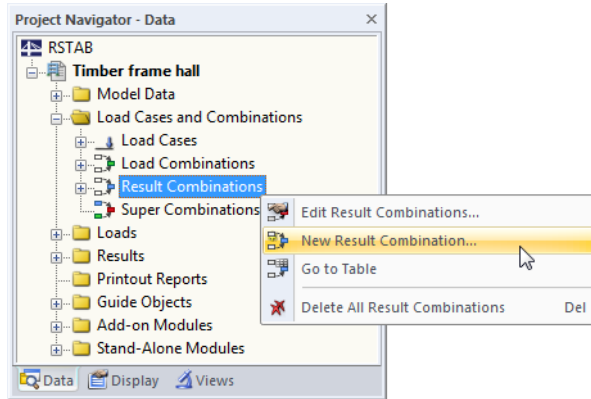


Figure 5.37: Context menu of *Result Combinations* in Data navigator

The dialog box *Edit Load Cases and Combinations* appears. A new result combination is preset in the dialog tab *Result Combinations*.

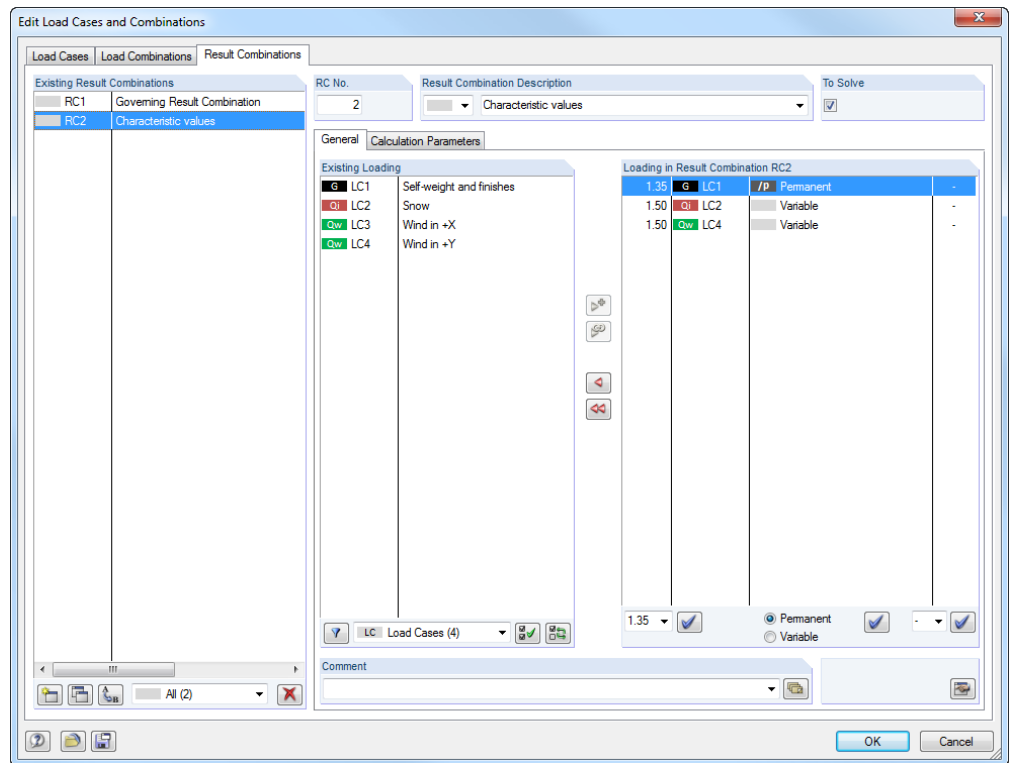


Figure 5.38: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

The following description refers to the tab *General*. The dialog tab *Calculation Parameters* with settings for the square addition is described in chapter 7.2.2 on page 175.

- It is also possible to enter a new result combination in an empty row of table 2.6 *Result Combinations*.

Result Combin.	DS	Result Combination Description	To Solve	Factor	Loading.1			Loading.2			Loading.3		
					No.	Crit.	Gr.	Factor	No.	Crit.	Gr.	Factor	No.
RC1		Governing Result Combination	<input checked="" type="checkbox"/>	1.00	CO1	1	1.00	CO2	1	1.00	CO3	1	
RC2		Characteristic values	<input checked="" type="checkbox"/>	1.35	LC1	/P	1.50	LC2	-	1.50	LC4	-	
RC3													
RC4													
RC5													

Figure 5.39: 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 5.4 and chapter 11.4.16, page 320).

### 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 all other structures.

### 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

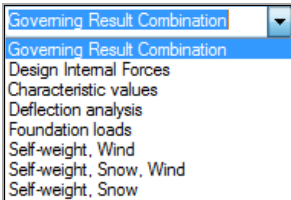
The columns inform you about the load cases, load and result combinations including related factors.

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  $\gamma$ , the combination factors  $\psi$ , the reduction factors  $\xi$  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 [Factors] or the table button [...]. 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 388. The tab *Combination Coefficients* manages the factors  $\psi$  and  $\xi$  (see Figure 5.22, page 118). 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 [►∞] to transfer the selected entries in the list *Loading in Result Combination* to the right.



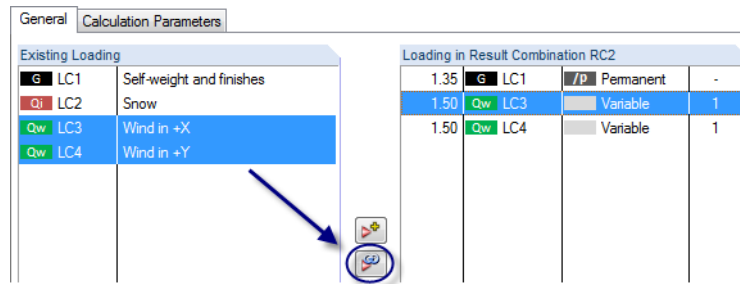


Figure 5.40: Multiple selection for alternative analysis of two load cases

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 388) 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**

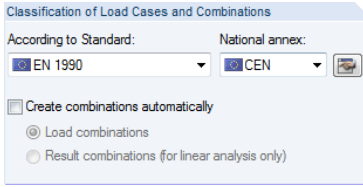
- For the alternative analysis using the "or" criterion, respectively the abbreviation "o", RSTAB treats the results of particular loadings as mutually exclusive. RSTAB will consider only values of the loading making the maximum unfavorable contribution. Use the dialog button [▶<sup>o</sup>] to transfer selected loadings to the definition list of the result combination.

- Loadings acting alternatively are marked with the same number in the table column *Group*.

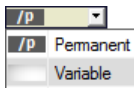
- The criterion "orto" (*or to*) combines a list of alternative loadings from the first to the last object. Objects lying in between are not listed.

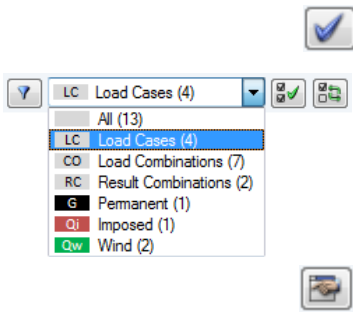
- All loadings listed in the alternative superposition must be marked consistently as 'Permanent' or 'Variable'. Thus, it is not allowed to enter for example "LC1/p or LC2.

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).



Standard settings in the dialog box *Model - General Data*





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 5.4 on page 133.

You can define result combinations manually in a separate dialog box. To open it, use the [Set] button in the bottom right corner of the dialog box *Edit Load Cases and Combinations*.

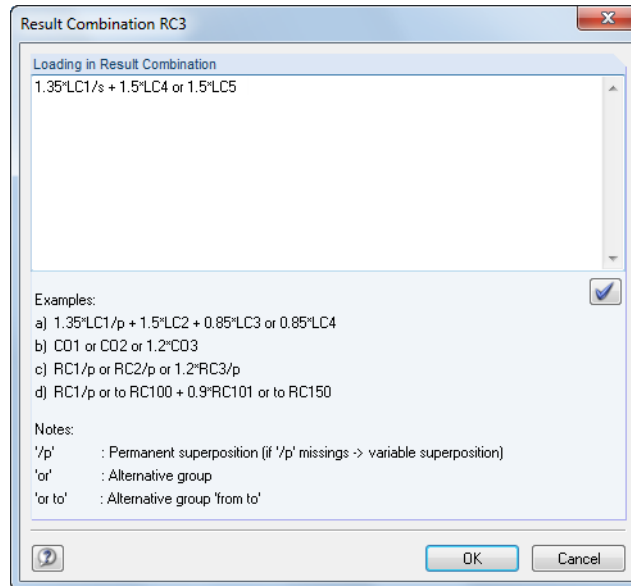


Figure 5.41: Dialog box *Result Combination* for definition via string box

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 the 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 or to 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 or RC2/p or RC3/p or RC4/p** (corresponds to **RC1/p or to 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 7.2.1 on page 169.

### 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

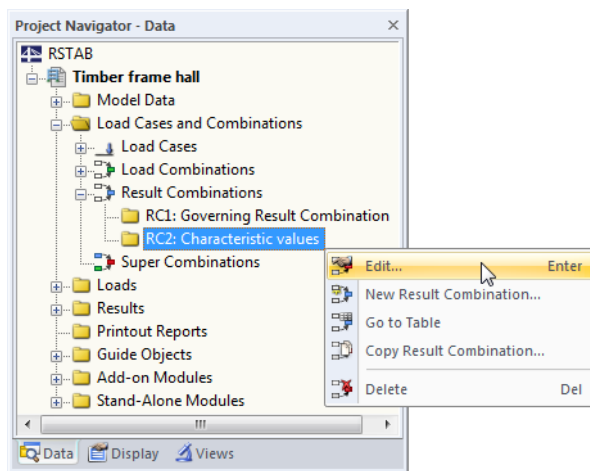


Figure 5.42: Context menu of a result combination

In the dialog box *Edit Load Cases and Combinations* (see Figure 5.38, page 129), 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*. They are reserved for 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 5.4: Buttons in the tab *Result Combinations*

### 5.6.2 Generated Combinations

When switching to the dialog tab *Result Combinations* or to table 2.6, RSTAB creates the combinations automatically. As the load cases are not superimposed manually, the *General* tab looks differently (cf. Figure 5.38, page 129 for user-defined combinations).

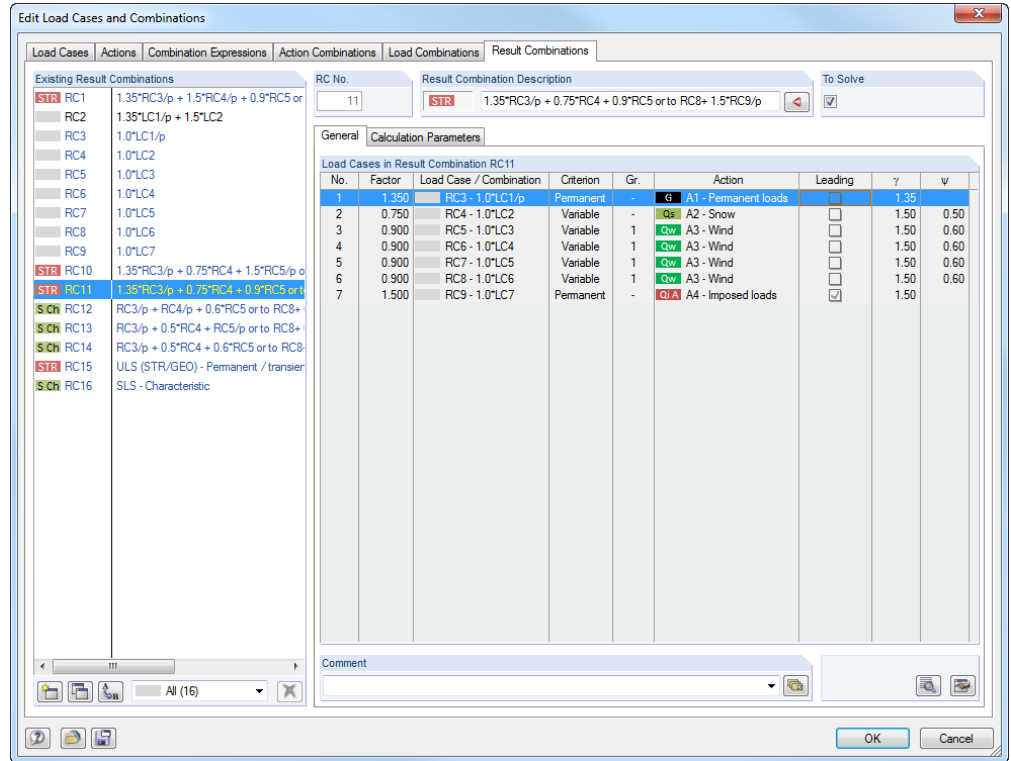
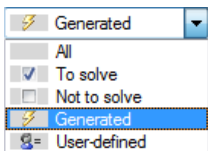


Figure 5.43: Dialog box *Edit Load Cases and Combinations*, tab *Result Combinations*

#### 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

RSTAB 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 116) 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 [Factors]. The dialog box *Coefficients* is subdivided into several tabs (see Figure 12.27, page 388 and Figure 5.22, page 118).

### 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.

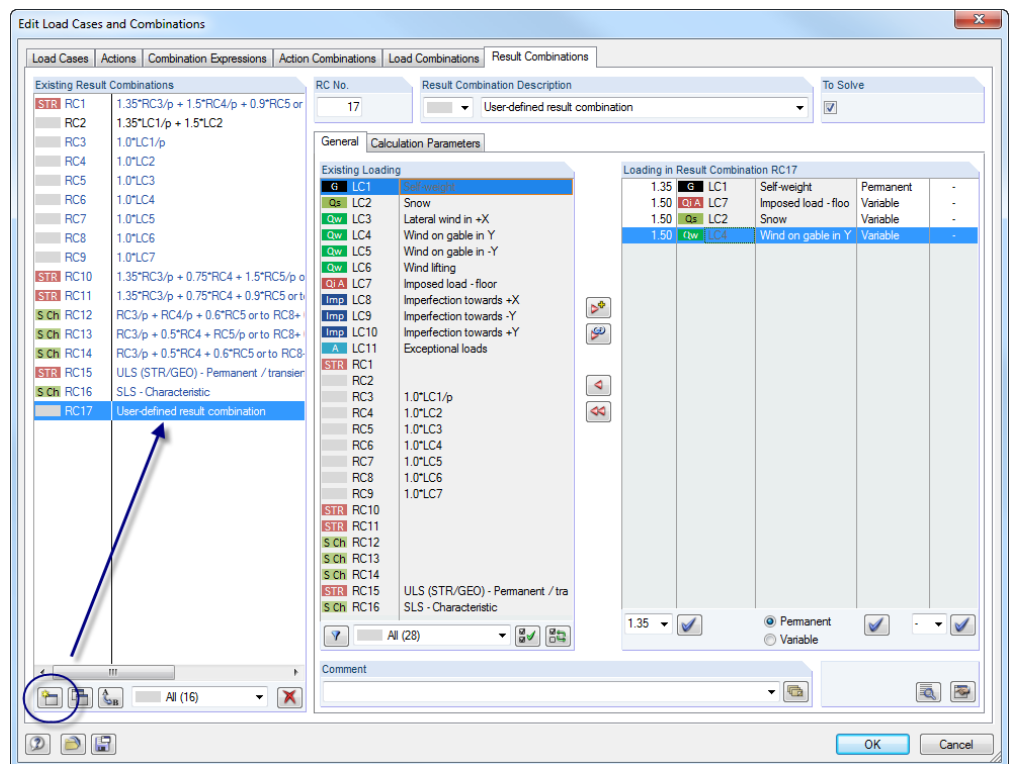


Figure 5.44: Adding a user-defined result combination

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.

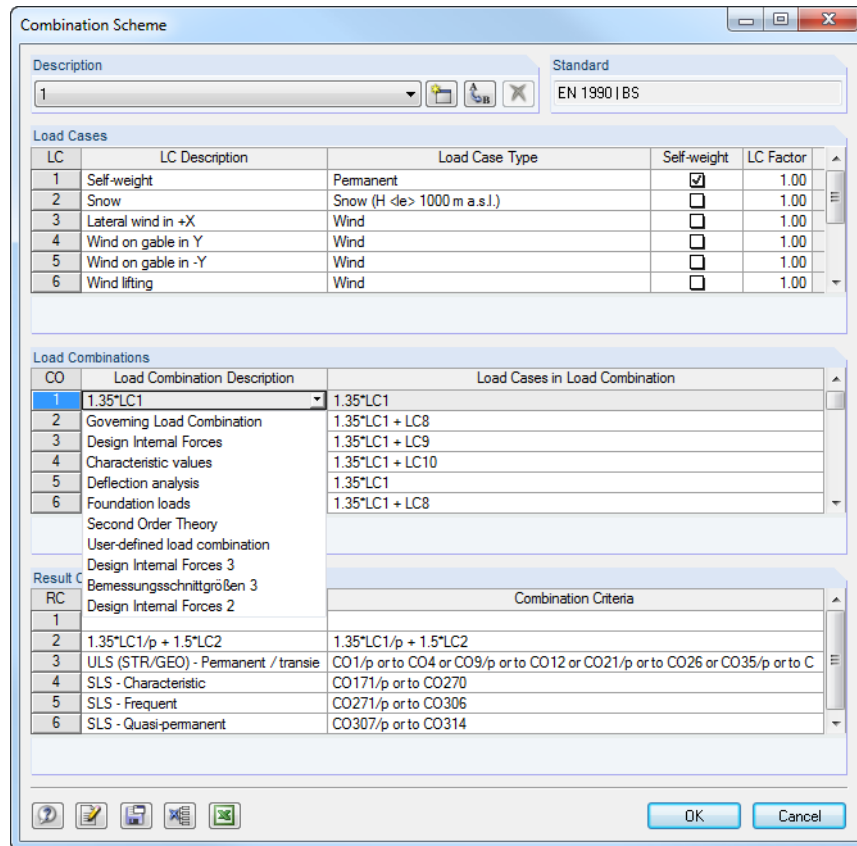


Figure 5.45: Dialog box *Combination Scheme*



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 RSTAB 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].

## 5.8 Super Combinations

### General description



Super combinations can only be used when the add-on module **SUPER-RC** has been licensed.

A super combination (**SC**) is similar to a result combination (see chapter 5.6.1, page 128). In a super combination, however, it is possible to superimpose also load cases and combinations that come from different models. In this way, it is possible to determine construction phases with variable system and loading conditions as they occur for example in bridge construction.

In the modeling process, you have to create an initial model first. Then, you can adjust the model to the progress of construction and save it each time as a copy. Make sure to have a coherent numbering of nodes and members so that internal forces are determined correctly when superimposing loads in a super combination later. It might be helpful to use dummy or divided members already applied in the initial model.



Only the results of models stored in the same project folder can be superimposed. If results of some models are not yet available, they are determined automatically before the superpositioning starts. It is also possible to superimpose the results of a super combination in another super combination.

The internal forces of super combinations can be used in a lot of RSTAB add-on modules for subsequent designs.

### Create a new super combination

There are several possibilities to open the loading dialog box for creating a super combination:

- point to **Load Cases and Combinations** on the **Insert** menu, and then select **Super Combination**.
- Use the toolbar button [New Super Combination].

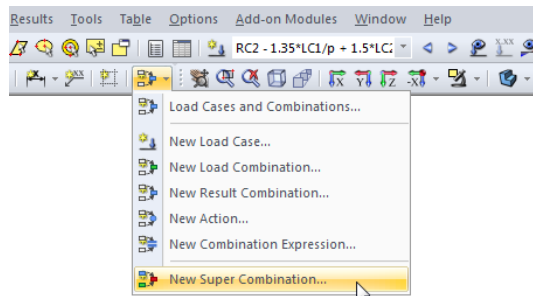


Figure 5.46: Button *New Super Combination* in the toolbar

- Use the context menu of the navigator entry *Super Combinations*.

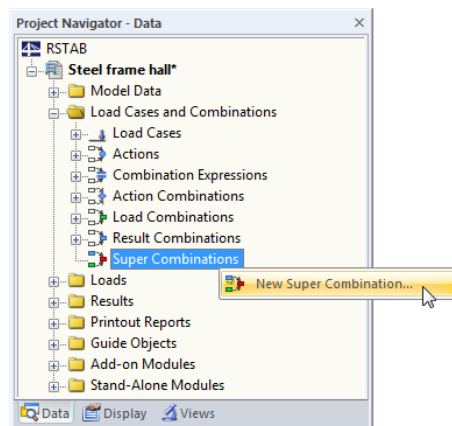


Figure 5.47: Context menu *Super Combinations* in the *Data* navigator

The dialog box *New Super Combination* appears.

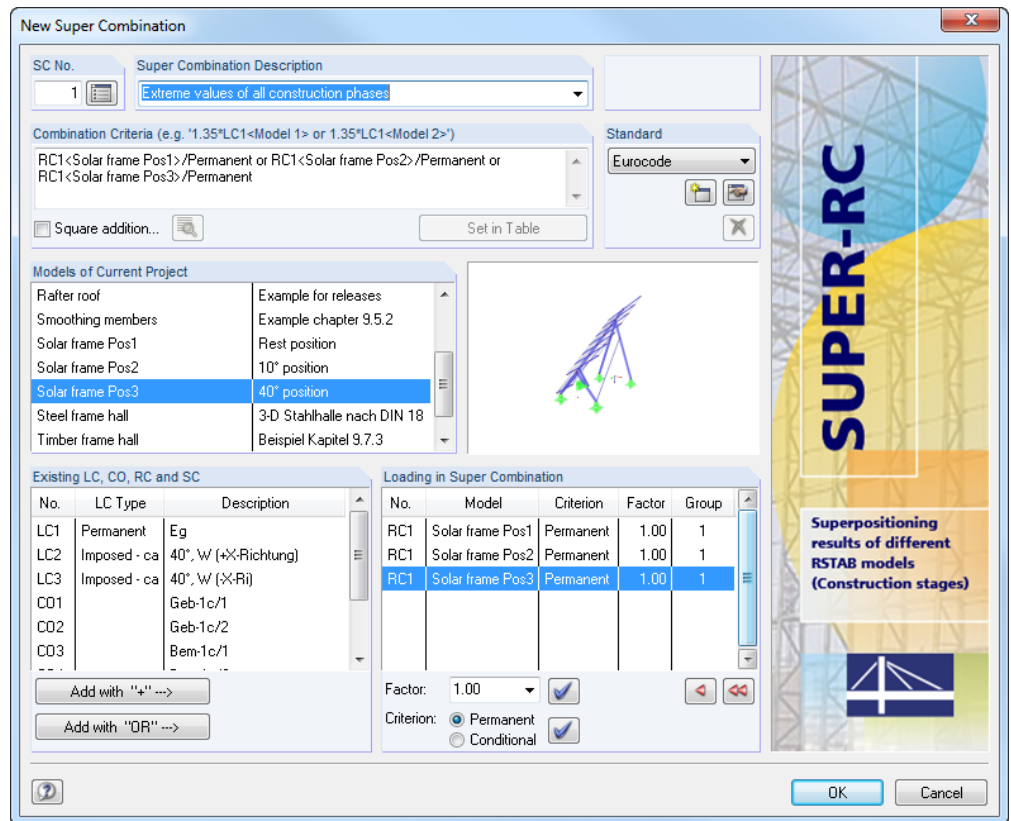


Figure 5.48: Dialog box *New Super Combination*

### SC No.

The number of the new super combination is preset but can be modified in the input field *SC No.*. Use the button [Index of All Available Super Combinations] to check which super combinations are already available.

### Super combination description

You can enter any name manually. You can also choose a name from the list to describe the super combination shortly. As manually entered descriptions are stored in the list, they are also available for other models.

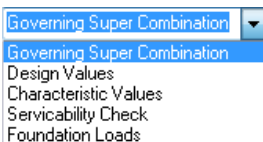
### Combination criteria

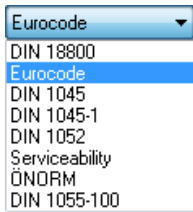
In this input field, you can add load cases and combinations from different models with any factor. You can also combine them with the "or" criterion. Each model name must be set in square brackets.

The way how you can create a super combination is described for the dialog section *Loading in Super Combination* explained below. The combination criteria is automatically entered when the load cases and combinations are transferred.

The *Square Addition* is deactivated by default. Thus, internal forces are superimposed by additive superposition. This default setting is appropriate for most application cases. However, a square addition of internal forces is relevant for dynamic analyses, for example for the combination of load cases occurring due to centrifugal forces (see chapter 7.2.2 on page 175). The [Settings] button can be used to adjust the treatment of signs for the square addition.

If you define the combination criteria manually, you can use the button [Set in Table] to transfer the entries to the dialog section *Loading in Super Combination*.





### Standard

The list contains a selection of rules and standards describing the principles for the ultimate limit state, serviceability and resistance of structural systems. The selected standard determines the rules by which the super combination is created. The standard also decides which partial safety factors  $\gamma$ , combination coefficients  $\psi$ , reduction factors  $\xi$  etc. are applied (see chapter 5.4, page 116).

The factors can be adjusted in the dialog box *Edit Standard* that you can access with the [Edit] button shown on the left.

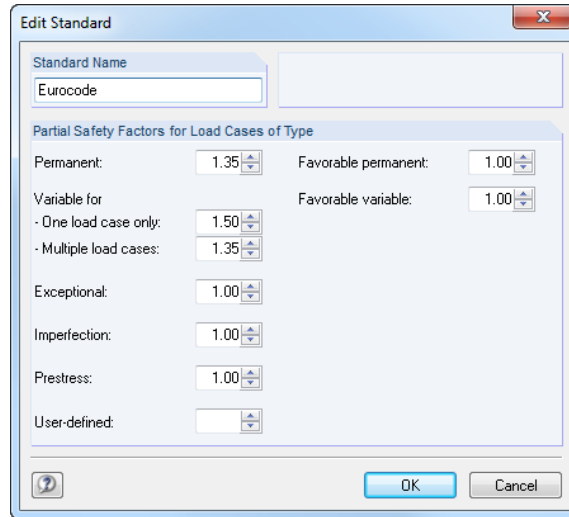


Figure 5.49: Dialog box *Edit Standard*

To create a user-defined standard, use the [New] button.



### Models of current project

This dialog section lists all models existing in the current project. To the right you see a thumbnail picture of the selected model, which makes the selection easier.

### Existing LC, CO, RC and SC

All load cases and combinations available in the model selected above are shown. Use both [Add] buttons to transfer the entries to the list *Loading in Super Combination* to the right (see the following).



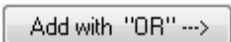
### Loading in super combination

The table contains all load cases and combinations contained in the super combination. SUPER-RC shows for each entry the model where the load case comes from as well as the superposition criteria and factors that are applied.

### Combining loadings

- First, in the dialog section *Models of Current Project*, select the model whose results are considered for the superposition. Then, you can select the relevant load cases and combinations in the dialog section *Existing LC, CO, RC and SC*. A multiple selection is possible with holding down the [Ctrl] key (see Figure 5.40, page 131).
- Use the dialog buttons [Add with "+"] or [Add with "OR"] to transfer the selected entries to the list *Loading in Super Combination* to the right. SUPER-RC applies the factors according to the *Standard* set in the dialog section above. Load cases of the action category *Permanent* loads (see Figure 5.5, page 100) are automatically taken into account with the *Criterion Permanent*.

If you click the [Add with "+"] button, results are combined additively with the criterion "+". If you click the [Add with "OR"] button, results are treated as mutually exclusive (see also





examples in chapter 5.6.1 *User-defined Combinations*, page 132). All load cases considered in an "Or"-superposition must be defined uniformly as *Permanent* or *Conditional*.

It is possible to adjust the factors or criteria for transferred load cases individually: Select the load case in the list *Loading in Super Combination*, and then enter an appropriate *Factor* into the input field below. You can also use the list to select a factor. The appropriate *Criterion* can also be defined below the dialog section *Loading in Super Combination*. Finally, click the [Set] button to apply the new specifications to the loads.

To remove a load case or combination from a super combination, select the relevant entry in the dialog section *Loading in Super Combination*. Use the [◀] button or double-click the entry to return it to the dialog section *Existing LC, CO, RC and SC*.

- When you have entered the load cases and combinations of one model into the super combination, select the next model in the dialog section *Models of Current Project*. Now, you can select all relevant load cases and combinations of this model to transfer them to the super combination as described above.

### Examples:

- **CO4<Model A>/Permanent or CO4<Model B>/Permanent**

The load combinations 4 of two different models are compared as permanently acting.

- **1.35\*LC1<A>/p + 1.50\*LC2<A> + 1.35\*LC1<B>/p + RC6<C> or RC67<D>**

In the superposition, load cases 1 of models A and B are taken into account as permanently acting with the factor 1.35. Load case 2 of model A, however, is considered with the factor 1.50 only if it has an unfavorable effect. Finally, only the higher contribution of result combinations 6 of models C and D is taken into account and considered respectively with a factor of 1.00.

### Edit a super combination

There are several possibilities to change super combinations subsequently:

- point to **Load Cases and Combinations** on the **Edit** menu, and then click **Super Combinations**.
- On the *Data* navigator, right-click a super combination to open its context menu, or double-click the super combination itself.

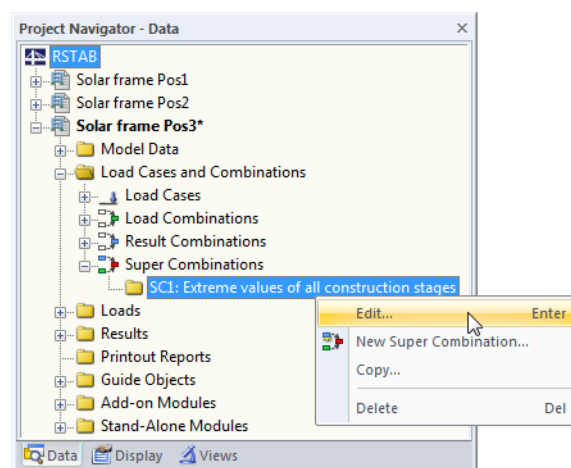


Figure 5.50: Context menu of a super combination

Then, you can adjust the definition criteria in the dialog box *Edit Super Combination*.



### Buttons

The buttons in the dialog box *New Super Combination* are reserved for the following functions:





	Opens a list with already available super combinations
	Assigns the <i>Factor</i> or <i>Criterion</i> to marked loading
	Removes marked entry from the list <i>Loading in Super Combination</i>
	Empties complete entries list of <i>Loading in Super Combination</i>

Figure 5.5: Buttons in the dialog box *New Super Combination*

## 6. Loads

RSTAB 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*

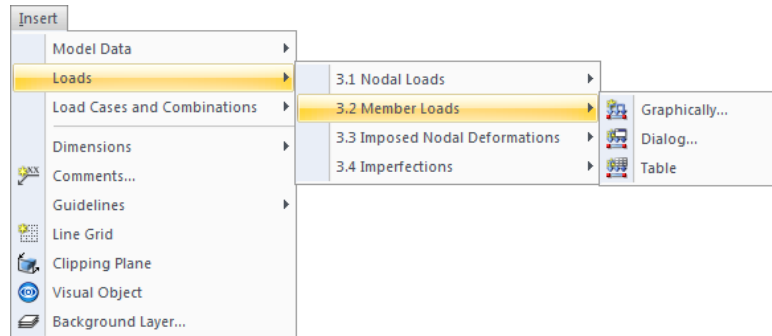


Figure 6.1: Menu *Insert* → Loads

#### Toolbar *Insert*



Figure 6.2: Toolbar *Insert*

#### Context menu in *Data navigator*

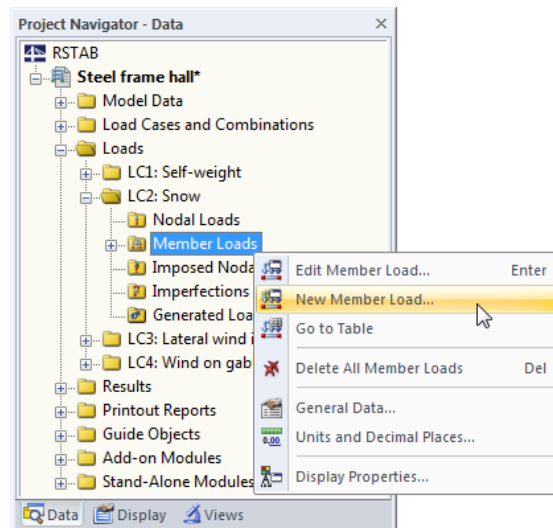


Figure 6.3: Context menu of load objects in the *Data navigator*



Context menu or double-click in table

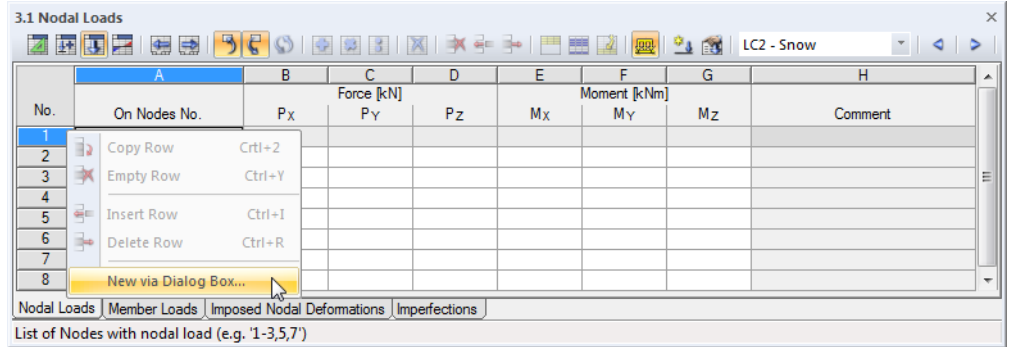


Figure 6.4: Context menu in load tables

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

RSTAB provides different possibilities to open a dialog box for editing a load object.

Menu Edit

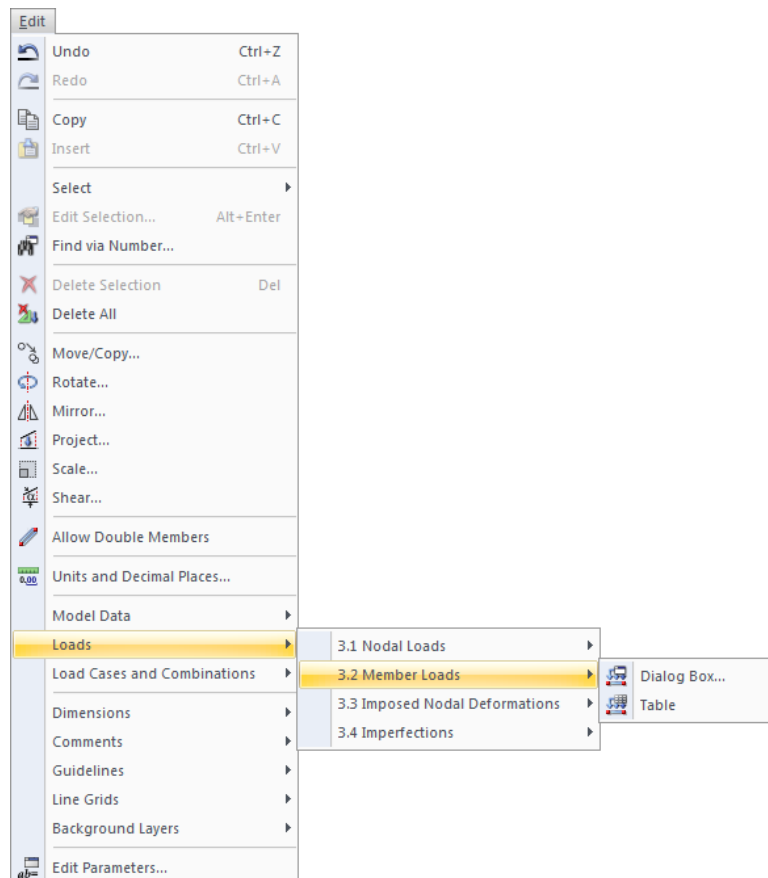


Figure 6.5: Menu Edit → Loads

When selecting the *Dialog Box* option, it is necessary to define a load object before the dialog box appears.

**Context menu or double-click in graphic**

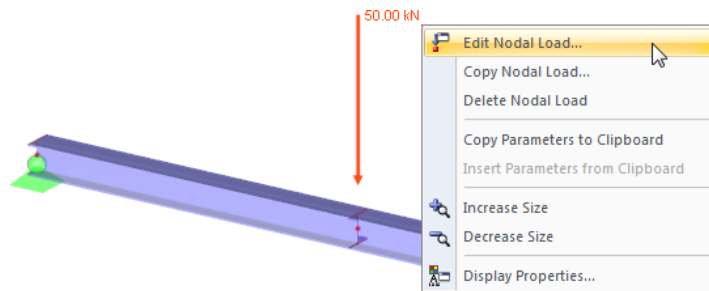


Figure 6.6: Context menu of a load in work window

**Context menu or double-click in Data navigator**

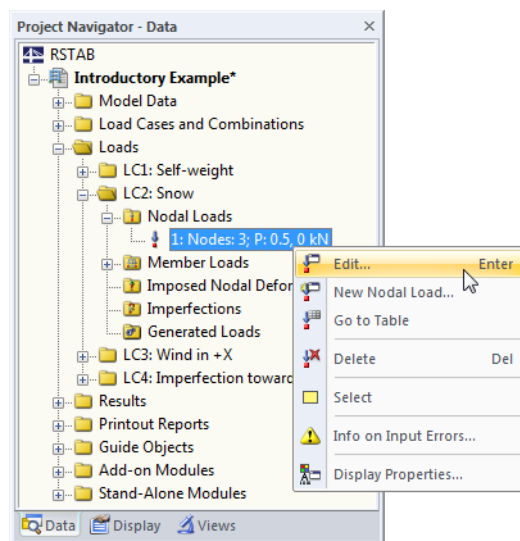


Figure 6.7: Context menu of load objects in the Data navigator

**Context menu or double-click in table**

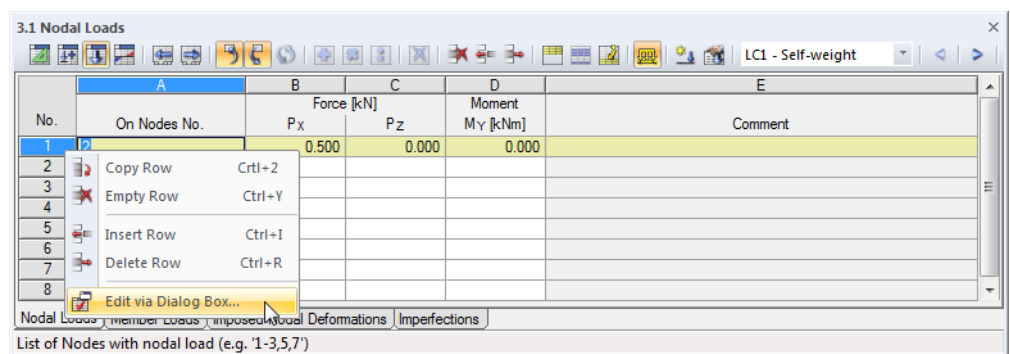


Figure 6.8: Context menu in load tables

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.

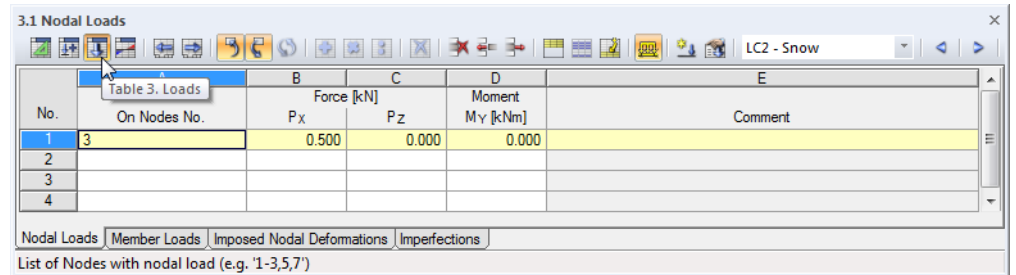


Figure 6.9: Button [Table 3. Loads]

Input in the form of spreadsheet data entered in tables can be quickly edited and imported (see chapter 11.5, page 321).

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 262).



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 41).

To apply a nodal load, a node must have been previously defined.

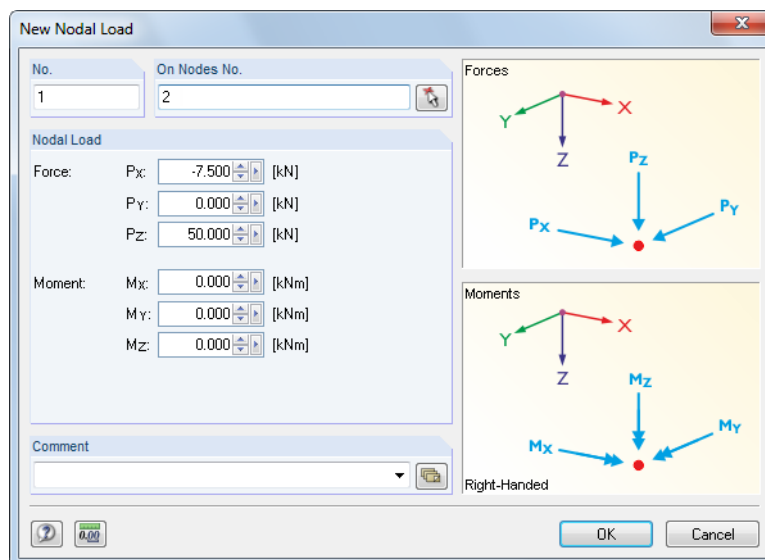


Figure 6.10: Dialog box *New Nodal Load*

No.	On Nodes No.	Force [kN]			Moment [kNm]			Comment
		P <sub>X</sub>	P <sub>Y</sub>	P <sub>Z</sub>	M <sub>X</sub>	M <sub>Y</sub>	M <sub>Z</sub>	
1	2	-7.50	0.00	50.00	0.00	0.00	0.00	
2	1,11	0.00	0.00	12.35	0.00	0.00	0.00	
3								
4								
5								

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 no.



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<sub>X</sub> / P<sub>Y</sub> / P<sub>Z</sub>

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 RSTAB graphic.

In addition to vectors, moments can be represented as arcs. To control the display properties (see chapter 11.1.2, page 258),

point to **Display Properties** on the **Options** menu, and select **Edit**.

The dialog box *Display Properties* opens where you set *Category* → *Loads* → *Nodal Loads* → *Nodal Moments*. Then, the display option *Arc* is available for selection in the tab to the right.

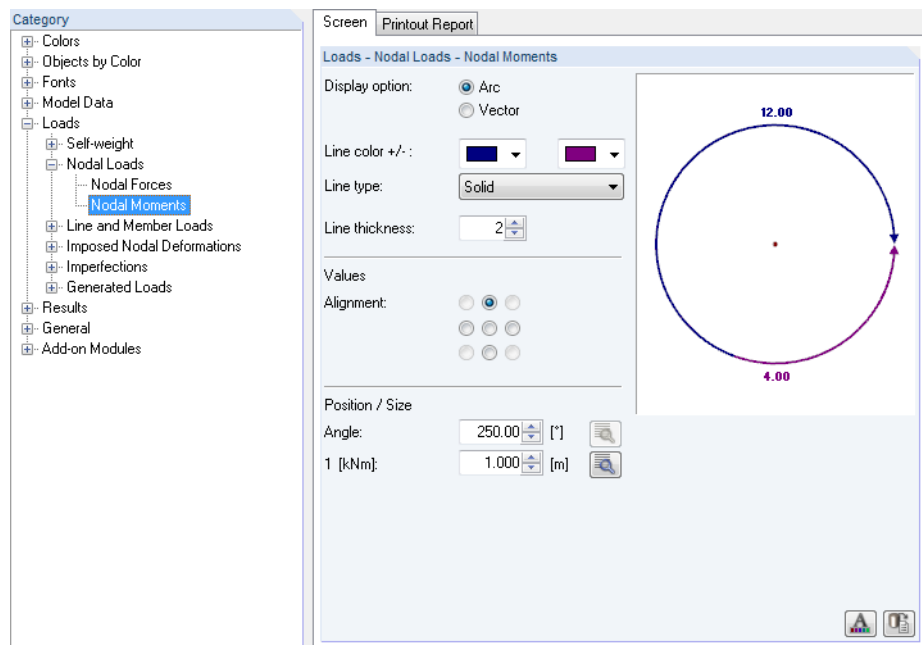


Figure 6.12: Dialog box *Display Properties* (dialog section): *Nodal Moments* with display option *Arc*



It is also possible to import nodal loads from Excel spreadsheets (see chapter 12.5.2, page 401).

## 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.

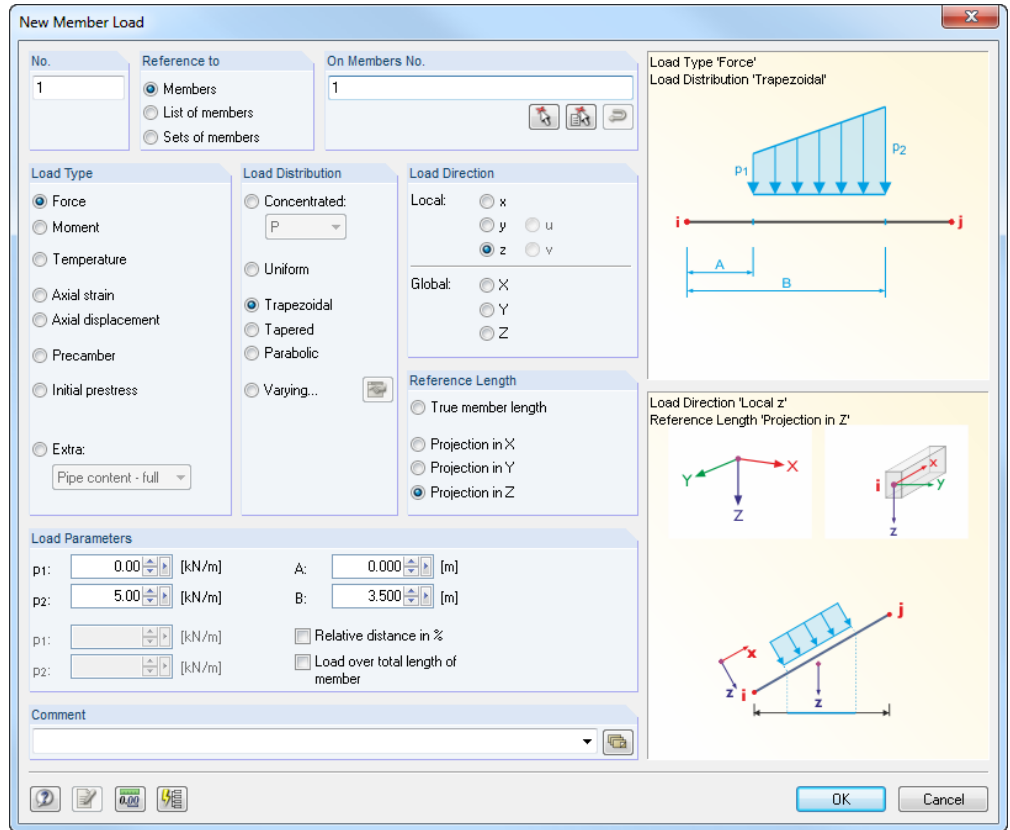


Figure 6.13: Dialog box *New Member Load*

No.	A	B	C	D	E	F	G	H	I	J	K
	Reference to	On Sets of Members No.	Load Type	Load Distribution	Load Direction	Reference Length	p [kN/m]		Member Load Parameters		
							p1	p2			A
1	Members	3-8,13,14,61-64	Force	Uniform	Z	True Length	0.75				
2	Members	121-124,129-136	Initial Prestress	Uniform	x	True Length	2.00				
3	Sets of Members	2	Force	Uniform	z	True Length	2.00				
4	List of Members	15-17	Force	Concentrated	z	True Length	5.50				0.000
5	Members	15-18,23-28,41-46	Moment	Uniform	Z	True Length	1.50				
6			Temperature								
7			Axial Strain								
8			Axial Displacement								
			Precamber								
			Initial Prestress								
			Pipe content - full								
			Pipe content - partial								

Figure 6.14: 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 objects 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 6.15.

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 4.11, page 96). 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.

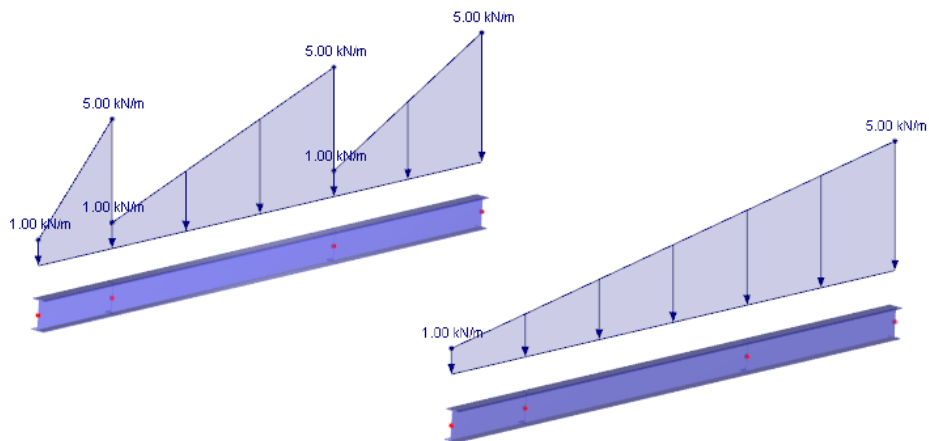


Figure 6.15: Trapezoidal load with reference to members (left) and to a list of members (right)

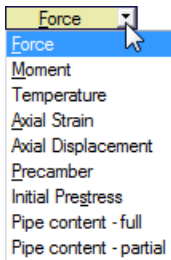
**On members no.**

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 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 $\epsilon$ 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 description in Figure 6.16).
Axial displacement	Imposed tensile or compressive strain $\Delta l$ 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.
Pipe content - full	Uniform load due to complete filling of pipe Specify weight density $\gamma$ of pipe content.
Pipe content - partial	Uniform load due to partial filling of pipe In addition to weight density $\gamma$ of pipe content, specify filling height $d$ .

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.



The parameters for member loads due to shrinkage can be defined in a separate dialog box (see the following figure).

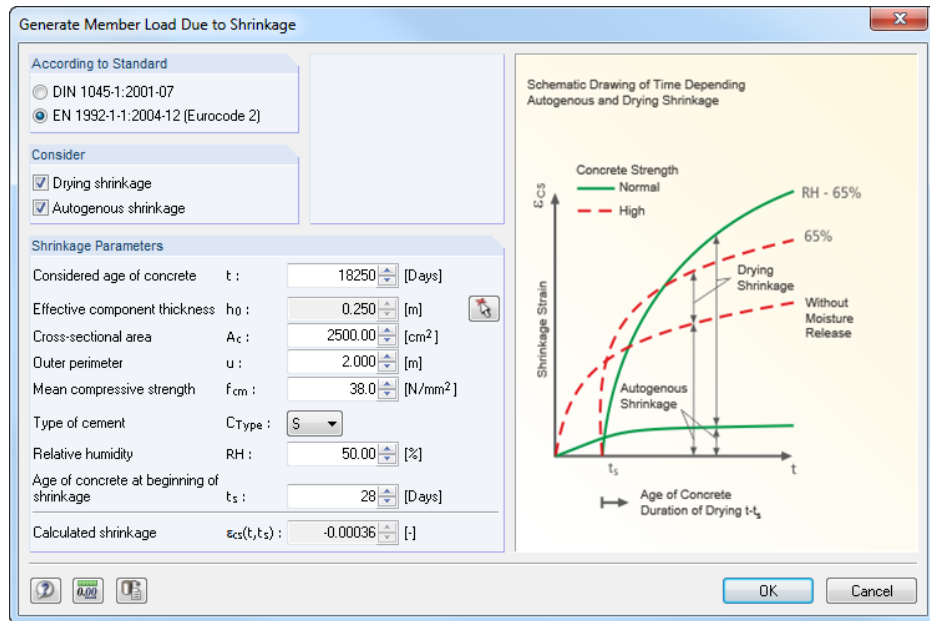


Figure 6.16: Dialog box *Generate Member Load Due to Shrinkage*

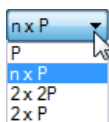
Shrinkage as a time-dependent change in volume without external load action or effects of temperature can be classified in drying shrinkage, autogenous shrinkage, plastic shrinkage and carbonation shrinkage.

Based on essential influence values of the shrinkage process (relative humidity  $RH$ , effective component thickness  $h$ , concrete strength  $f_{cm}$ , type of cement  $C_{Type}$ , age of concrete at beginning of shrinkage  $t_s$ ), RSTAB determines the shrinkage  $\epsilon_{cs}(t, t_s)$  at the moment of the considered concrete age  $t$ .

Click [OK] to transfer the value as axial strain  $\epsilon$  to the dialog box *New Member Load*.

### 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

Load distribution	Diagram	Description
Concentrated P	Load Type 'Force' Load Distribution 'Concentrated' 	Concentrated load, concentrated moment  In the dialog section <i>Load Parameters</i> , 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	Load Type 'Force' Load Distribution '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 <i>Load Parameters</i> , define the size of the concentrated load, the distance between first load and member start, and the spacing of loads among each other.

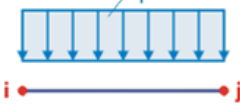
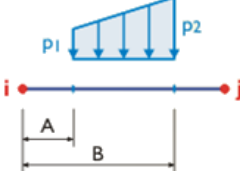
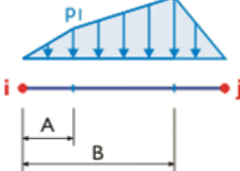
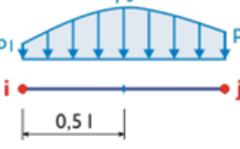
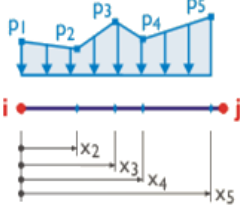
<p>Uniform</p>	<p>Load Type 'Force' Load Distribution 'Uniform'</p> 	<p>Uniformly distributed load, uniformly distributed moment</p> <p>In the dialog section <i>Load Parameters</i>, specify the size of the uniform member load or moment.</p>
<p>Trapezoidal</p>	<p>Load Type 'Force' Load Distribution 'Trapezoidal'</p> 	<p>Trapezoidal load, trapezoidal moment</p> <p>In the dialog section <i>Load Parameters</i>, 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.</p> <p>When the check box <i>Relative distance in %</i> is ticked, you can specify the distances relatively to the member length.</p>
<p>Tapered</p>	<p>Load Type 'Force' Load Distribution 'Tapered'</p> 	<p>Triangular-trapezoidal load, triangular-trapezoidal moment</p> <p>In the dialog section <i>Load Parameters</i>, specify the load values and distances for a linearly variable load distribution as shown in the dialog graphic.</p>
<p>Parabolic</p>	<p>Load Type 'Force' Load Distribution 'Parabolic'</p> 	<p>Parabolic load, parabolic moment</p> <p>The load acts as a parabolic load on the entire member. The load values at the member start and member end as well as in the member center must be entered in the dialog section <i>Load Parameters</i>.</p>
<p>Varying</p>	<p>Load Type 'Force' Load Distribution 'Varying'</p> 	<p>Polygonally distributed load</p> <p>Click the button [Edit Varying Load] shown on the left to open the dialog box shown in Figure 6.17 where you can enter or import the parameters of the load distribution.</p>

Table 6.2: Load distributions

If you want to represent a variable load, you can freely define the  $x$ -locations on the member with the corresponding load ordinates  $p$  in another dialog box (see the following figure). Make sure that the  $x$ -locations are defined in an ascending order. Use the interactive graphic to check your input immediately.

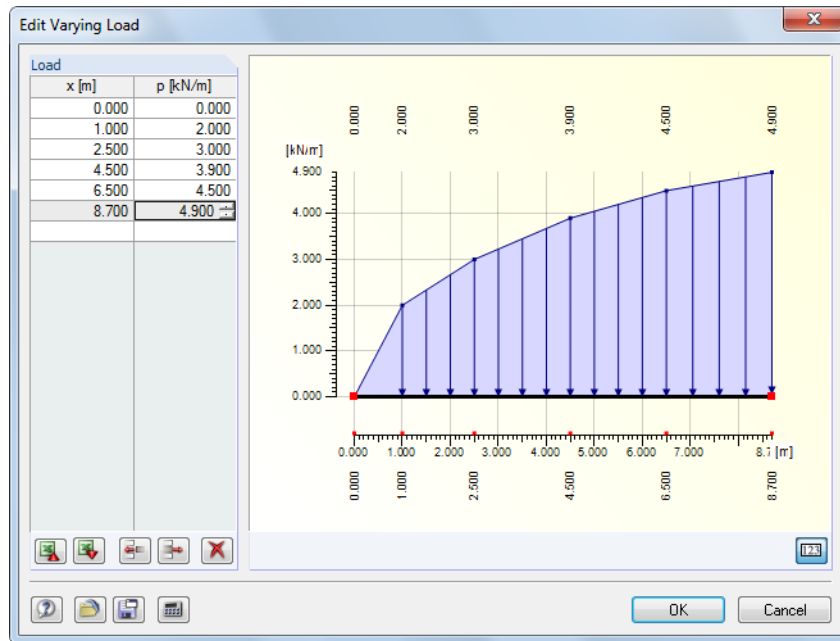


Figure 6.17: Dialog box *Edit Varying Load*

The buttons in this dialog box are reserved for 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

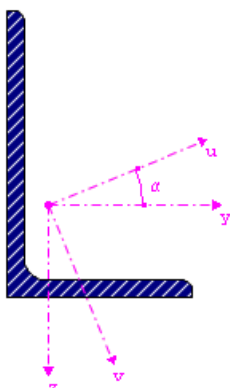
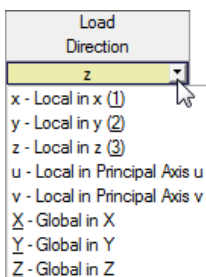
The load can be effective in direction of the global axes X, Y, Z or the local member axes x, y, z or u, v (see chapter 4.3, page 56). For the calculation, it makes no difference whether a load is defined as local or as equivalently global. RSTAB treats all loads conservatively, i.e. they are applied independently of the member deformations that may occur in geometrically nonlinear calculations. Therefore, the direction of local loads is always related to the undeformed model.

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 4.7, section *Member rotation* on page 79. 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 or 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 reference 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.



RSTAB 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, an additional torsional moment determined from for example load · distance to the shear center must be applied if the load acts in the centroid.

### 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).

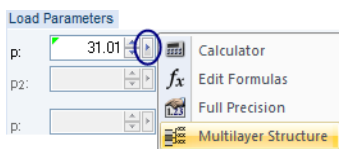
The dialog graphic in the upper right corner helps you when entering parameters.

#### Load of multi-layer 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 corresponding function in the dialog box *New Member Load* (Figure 6.13) where you click the button  displayed to the right of the load value's input field. In the context menu, select *Multilayer Structure*.

The *Multilayer Library* opens where you can enter user-defined material layers.



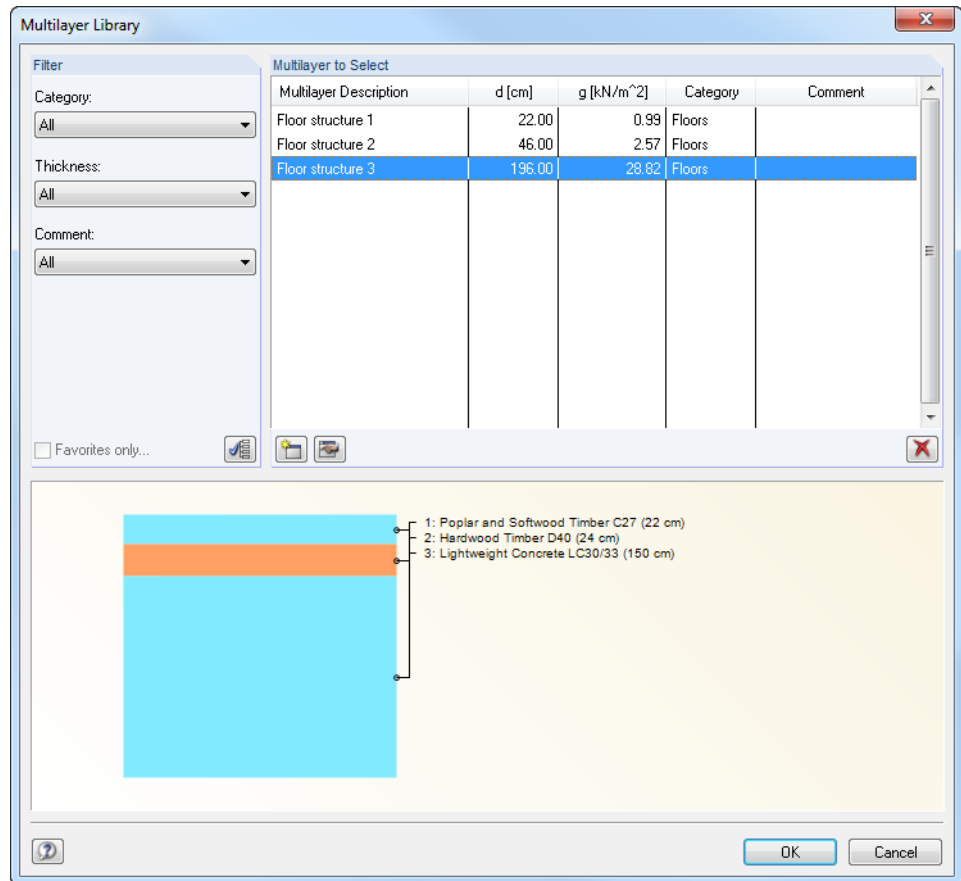


Figure 6.18: Dialog box *Multilayer Library*



The concept of the multilayer database is similar to the material library (see chapter 4.2, page 49). Use the library buttons [New] and [Edit] to create or modify multi-layer structures.

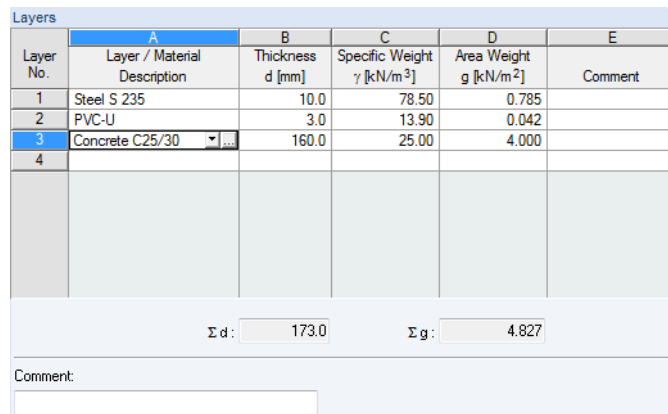


Figure 6.19: Dialog box *New Multilayer*, dialog section *Layers*



The *Layers* can be composed individually. Moreover, you can use the button [...] to access the material library (see chapter 4.2, page 49).

RSTAB determines the area weight (table column D) from *Thickness* and *Specific Weight*. An arrow shown in the dialog graphic indicates the current layer.

After confirming the dialog box another dialog box appears: *Convert Area Load into Line Load* where you have to specify the *Influence width* of the load (see the following figure).

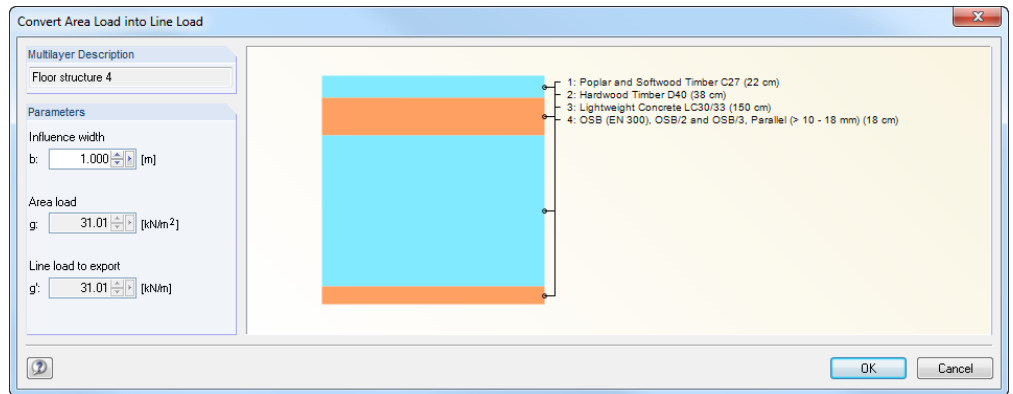


Figure 6.20: Dialog box *Convert Area Load into Line Load*

Confirm all dialog boxes with [OK] to import the member weight to the initial dialog box. A green triangle appears in the input field (see graphic shown in the margin on page 154), indicating the parameterized input value. Click the triangle to access again the input parameters for modifications.

### 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 *A* and *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:

In our example, member loads are defined for a planar framework. You can see that the members do not need to be divided by intermediate nodes to apply concentrated loads.

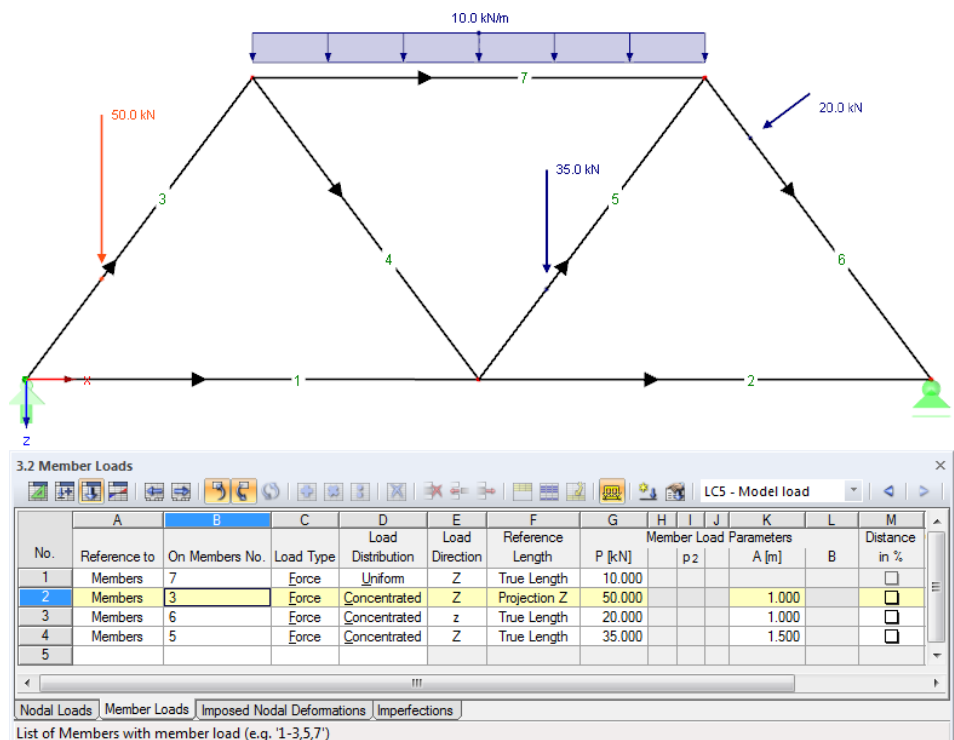


Figure 6.21: Framework with uniform load on upper chord and concentrated loads on diagonals



## 6.3 Imposed Nodal Deformations

### General description

An imposed nodal deformation is the displacement 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.

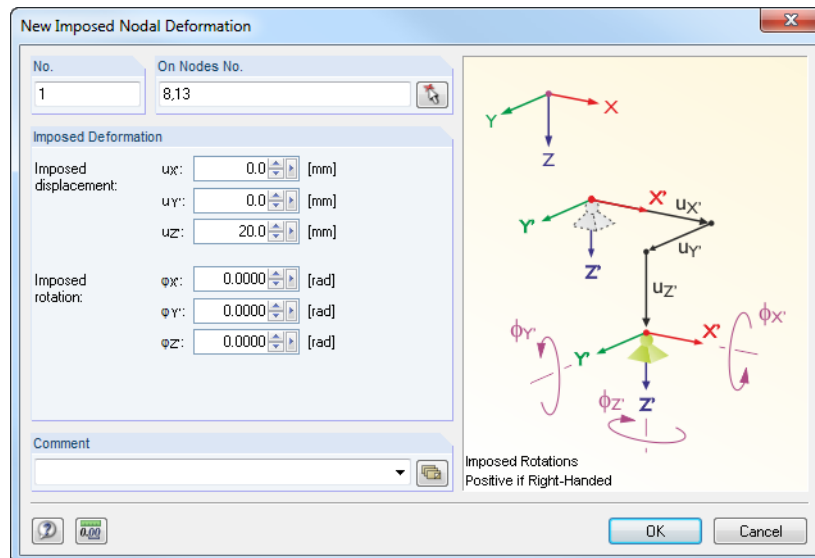


Figure 6.22: Dialog box *New Imposed Nodal Deformation*

3.3 Imposed Nodal Deformations								
No.	On Nodes No.	Displacement			Rotation			Comment
		u <sub>x</sub> [mm]	u <sub>y</sub> [mm]	u <sub>z</sub> [mm]	φ <sub>x</sub> [rad]	φ <sub>y</sub> [rad]	φ <sub>z</sub> [rad]	
1	13,14	0.0	0.0	20.0	0.0000	0.0000	0.0000	
2								
3								
4								

List of supported nodes with imposed deformation (e.g. '1-3,5,7').

Figure 6.23: 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 no.

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 $\phi_X / \phi_Y / \phi_Z$

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.

## 6.4 Imperfections

### General description

There are two ways how imperfections can be represented in RSTAB:

- **equivalent loads** for members and sets of members
- preformed **equivalent model** from the add-on module RSIMP

This chapter describes imperfections in the form of equivalent loads. For more information about generating equivalent models with RSIMP, see the RSIMP manual.

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.

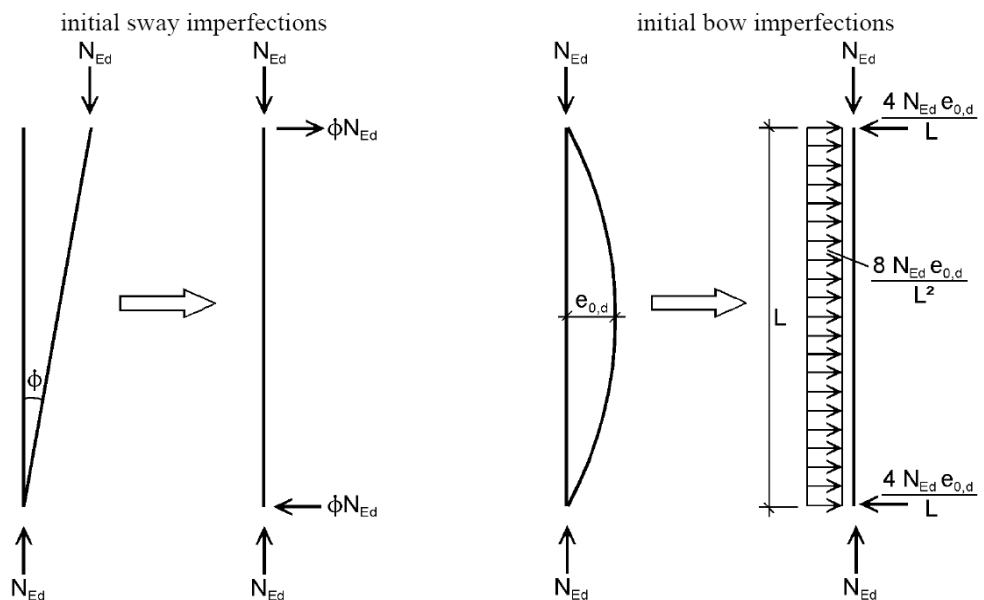


Figure 6.24: Equivalent loads according to EN 1993-1-1

Equivalent loads are also taken into account by RSTAB 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 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 99). 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.

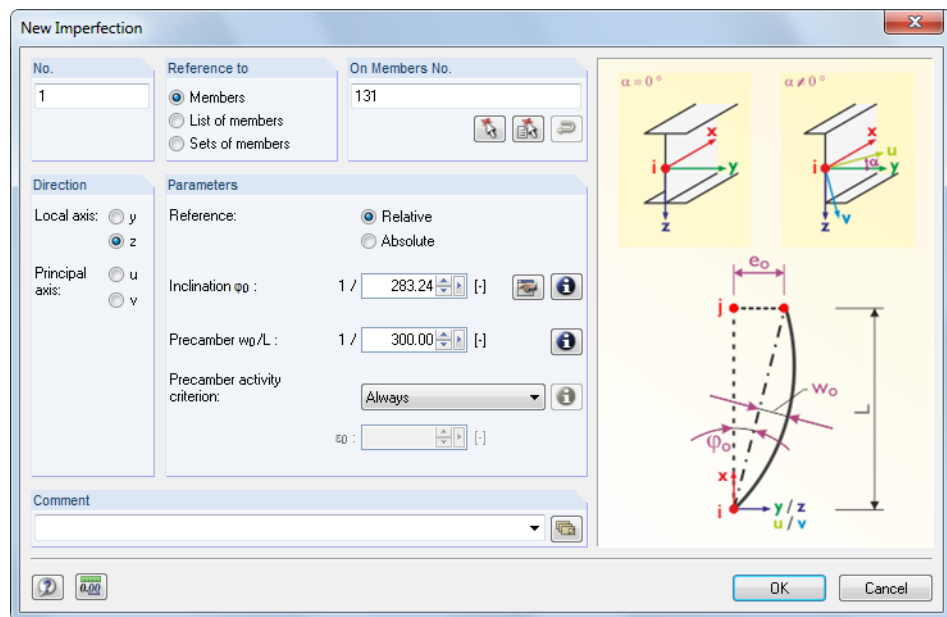


Figure 6.25: Dialog box *New Imperfection*

No.	Reference to	On Members No.	Direction	Reference	Inclination $1/\varphi_0$ [°]	Precamber $l/w_0$ [°]	Activity Criterion	Apply $w_0$ from $\varphi_0$ [°]
1	Members	1	z	Relative	200.00	300.00	EN 1993-1-1 (5.8)	
2	Members	81	y	Relative	250.00	300.00	EN 1993-1-1 (5.8)	
3	Members	82	y	Relative	250.00	300.00	EN 1993-1-1 (5.8)	
4	Members	83	y	Relative	250.00	300.00	EN 1993-1-1 (5.8)	
5	List of Members	2,11	z	Relative	200.00	300.00	EN 1993-1-1 (5.8)	
6	List of Members	3,12	z	Relative	200.00	300.00	EN 1993-1-1 (5.8)	
7	Members	14	z	Relative	283.24	300.00	EN 1993-1-1 (5.8)	

Figure 6.26: Table 3.4 *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 6.27.

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 4.11, page 96). 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.

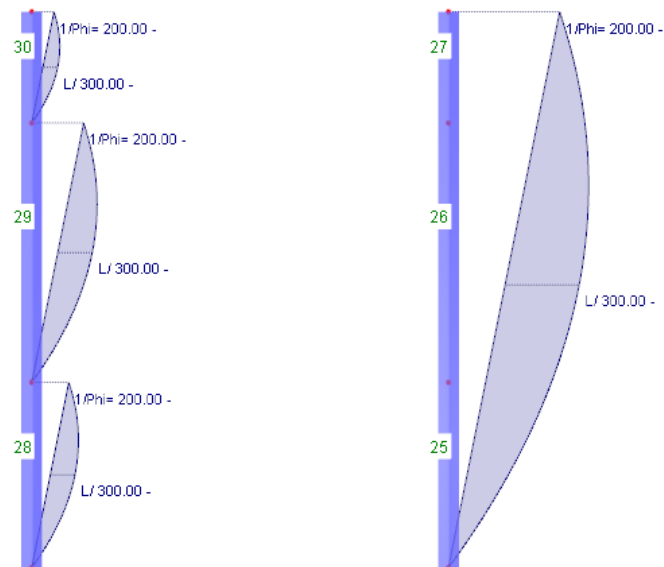


Figure 6.27: Imperfection with reference to members (left) and to a list of members (right)

### On members no.

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 4.3, page 56). It is not possible to define a globally acting inclination or precamber.

The orientation of member axes is described in chapter 4.7, section *Member rotation* on page 79. 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  $\varphi_0$  and  $w_0$  in relation to the member length, *Absolute* allows for specifying geometric dimensions directly.

## Inclination $1/\varphi_0$

$\varphi_0$  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  $\varphi_0$ , 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.

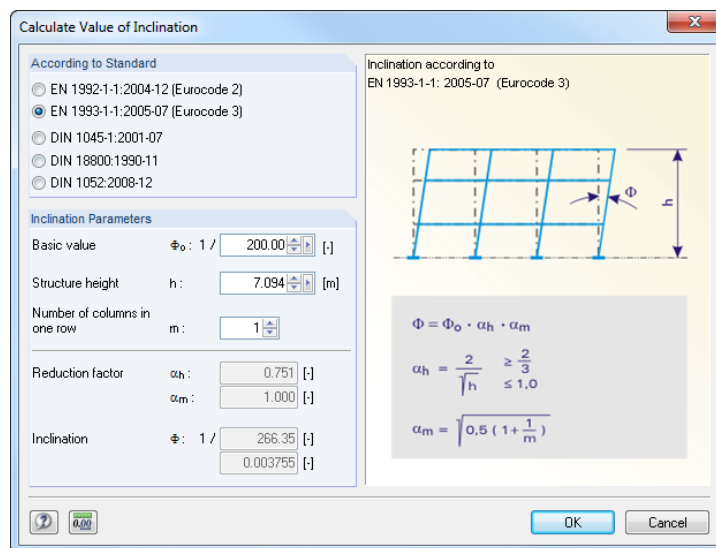


Figure 6.28: Dialog box *Calculate Value of Inclination*

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.

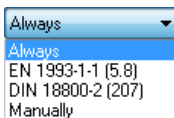
## Precamber $w_0/L$

The precamber  $w_0$  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  $\varepsilon$  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.

**Apply  $w_0$  from  $\epsilon_0$**

A precamber is considered in addition to inclination if the member coefficient  $\epsilon$  is higher than the value defined in this input field. DIN 18800 part 2 el. (207) specifies  $\epsilon > 1.6$  for most cases.

## 6.5 Generated Loads

RSTAB offers several generators that you can use to create loads easily (see chapter 11.8 on page 354). Generated member loads are reflected in table 3.5 and in the *Data* navigator.

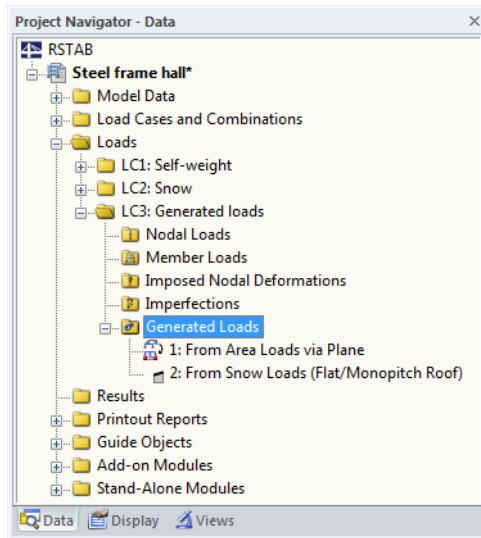


Figure 6.29: Data navigator for *Generated Loads*

No.	Generated Load Type	Comment
1	From Area Loads via Plane	
2	From Snow Loads (Flat/Monopitch Roof)	

Figure 6.30: Table 3.5 *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 11.176, page 364) 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. RSTAB 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.

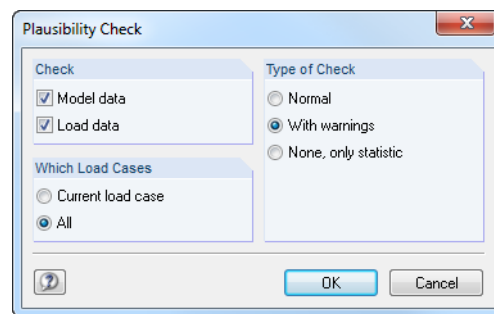


Figure 7.1: Dialog box *Plausibility Check*

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.

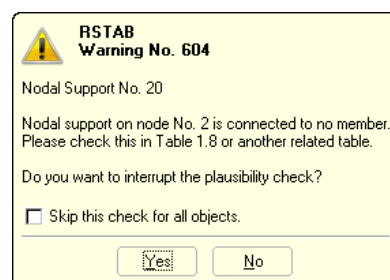


Figure 7.2: Plausibility check with warning

- **None, only statistic**

Only a summary of input data is reported (dimensions, total mass, number of nodes, members, supports, member loads etc.).

When the plausibility check was successful, the check result appears showing you a summary of input data.

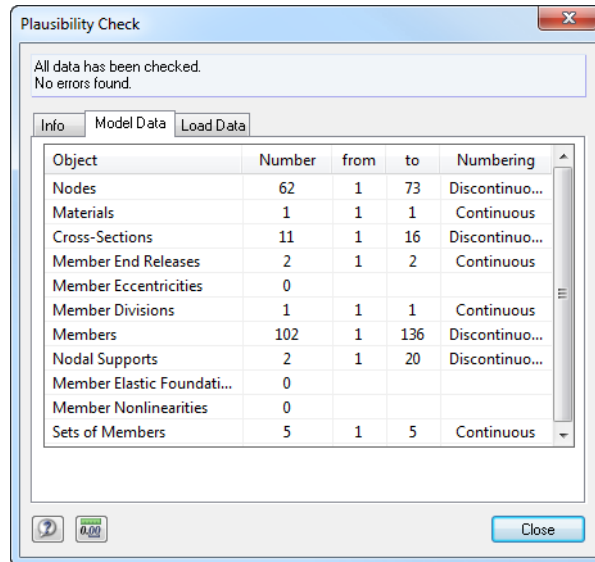


Figure 7.3: Result of plausibility check, tab *Model Data*

### 7.1.2 Structure Check

In addition to the general plausibility check, you can use the structure check to search specifically for discrepancies produced during the modeling. To open the corresponding dialog box,

point to **Structure Check** on the **Tools** menu

and select one of several check options.

#### Identical nodes



RSTAB filters all nodes with identical coordinates. They are combined in groups shown in a dialog box.

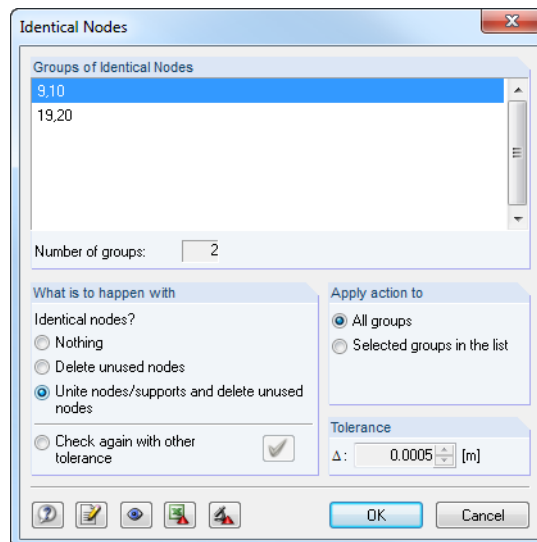


Figure 7.4: Results of structure check for identical nodes



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 where lines are often short because of nodes lying closely together. If such nodes are filtered with an appropriate tolerance and then unified, it is possible to avoid numerical problems due to short members.

### 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.

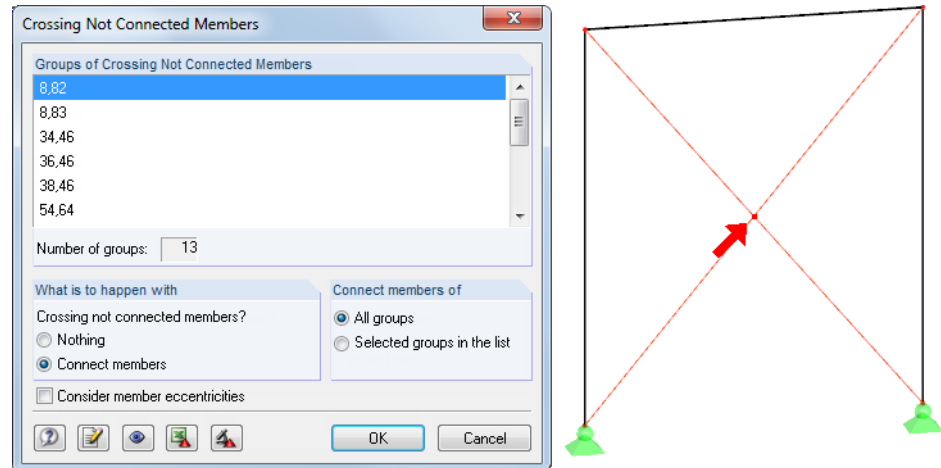


Figure 7.5: Results of structure check for crossing members

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.



### Independent systems

There is no connection between particular members of a model consisting of independent substructures. RSTAB is able to calculate such subsystems provided that they are considered to be individually stable. Often, however, substructures are generated unintentionally during the modeling process or when data is imported from CAD programs.

With this option you can check if the model is contiguous.

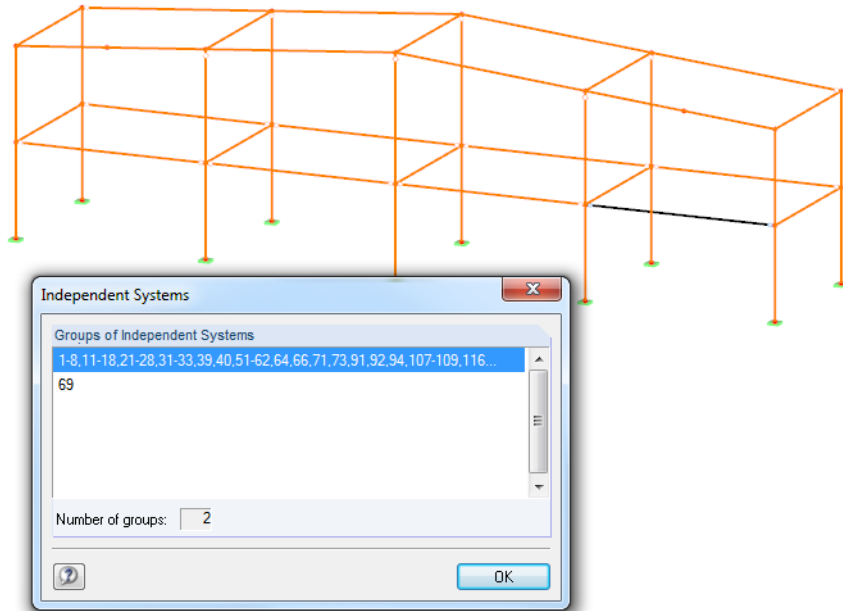


Figure 7.6: Results of structure check for independent systems

After the check you can see the groups of the independent systems listed in a dialog box. The current group is indicated by its selection color shown in the work window so that input errors can be detected quickly.

When you want to clean the model, the structure check for *Identical Nodes* (described above) is helpful, too.

### Buttons

The buttons in the dialog boxes of the structure check are reserved for the following functions:





	Applies changes of dialog section <i>What is to happen with</i>
	Jumps to RSTAB 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 structure check

### 7.1.3 Regenerate Model



RSTAB 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.

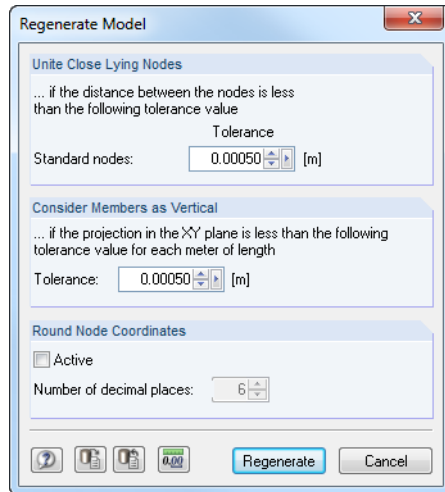


Figure 7.7: Dialog box *Regenerate Model*

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 Members as Vertical*, you can control the position of the local member axes. The orientation of axes for members in vertical position differs essentially from members in general (inclined) position (see chapter 4.7, page 80). 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.

Finally, you have the option to *Round Node Coordinates*. When the check box is ticked, you can define the relevant number of decimal places.

### 7.1.4 Delete Not Used Loads

Loads can only be defined on objects existing in the model. However, during the modeling process it may happen that members or nodes with assigned loads are removed from the system. Normally, RSTAB 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.



Menu *Tools* → *Delete Loads*

## 7.2 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*.

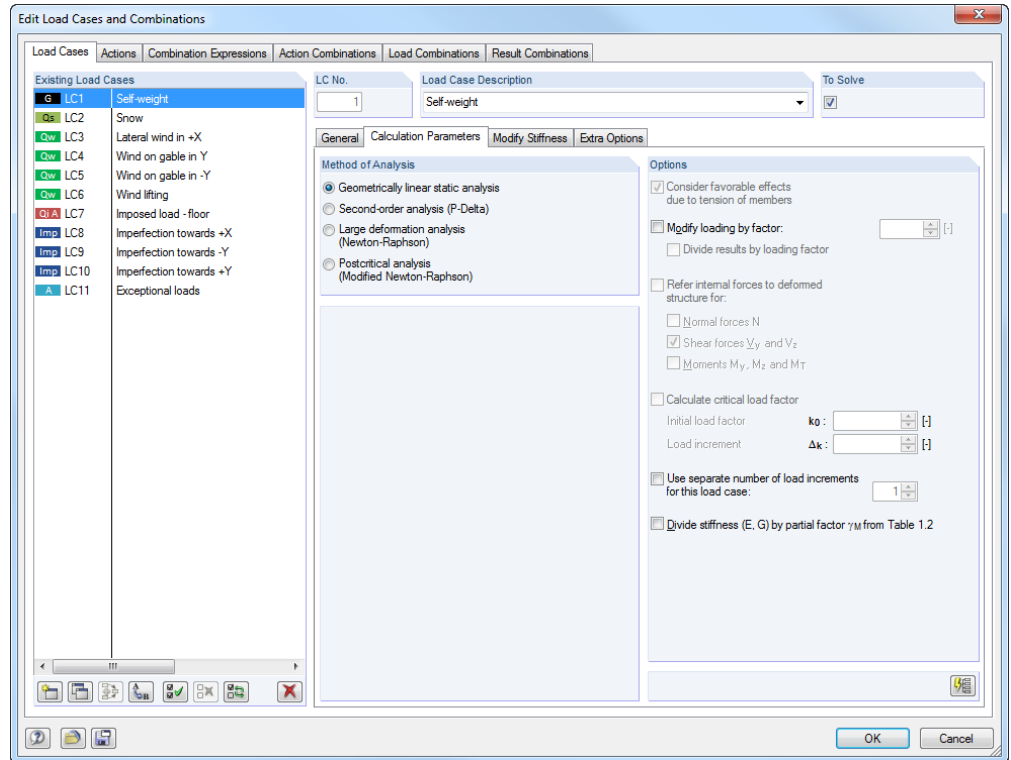


Figure 7.8: Dialog box *Edit Load Cases and Combinations*, tabs *Load Cases* and *Calculation Parameters*

In addition to an offered 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 7.9: 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 7.14, page 176), you can check and, if necessary, adjust specifications that are universally valid.

## 7.2.1 Load Cases and Load Combinations

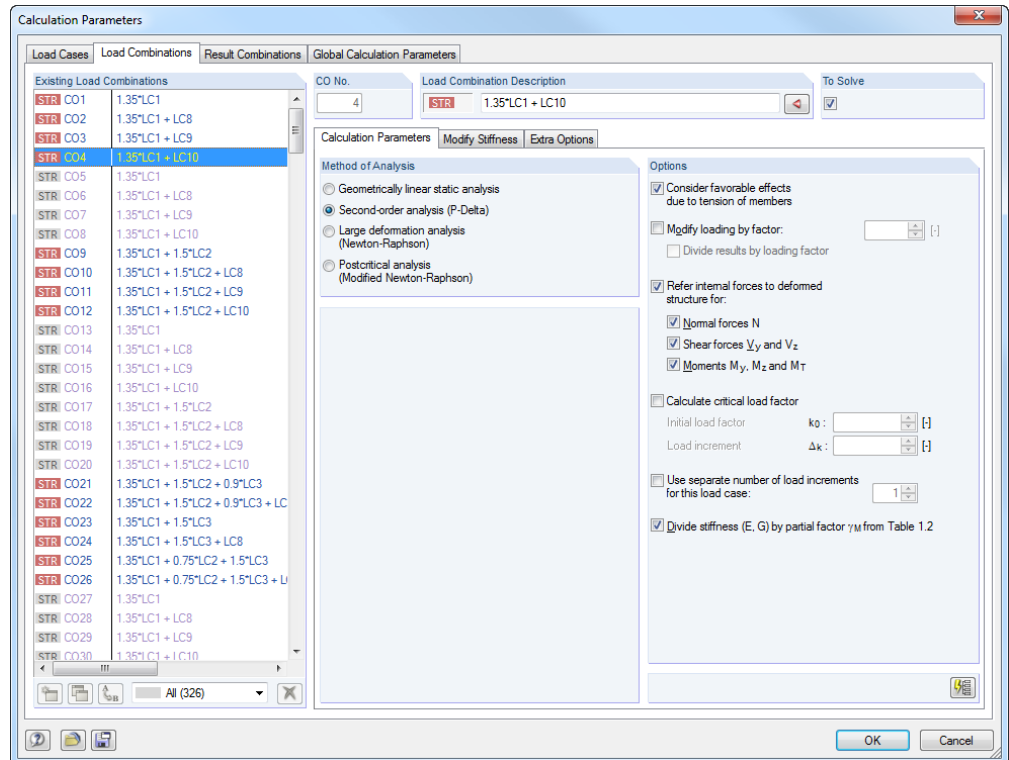


Figure 7.10: Dialog box *Calculation Parameters*, tab *Load Combinations*

The dialog section *Existing Load Cases* respectively *Existing Load Combinations* lists all created or generated 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 tabs *Calculation Parameters* and, if applicable, *Modify Stiffness* (see page 173) and *Extra Options* (see page 174).

### 7.2.1.1 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.

RSTAB 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 large deformation analysis, the remaining members according to the selected method of calculation.

#### 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 will be applied only if the axial forces are considerably higher than the

shear forces. The additional bending moment  $\Delta M$  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  $\gamma$ -fold actions.

The approaches according to second-order analysis are based on trigonometric functions. RSTAB applies the analytical solution of the differential equation for the displacement of the member by taking into account the axial force. The interaction between bending and torsion is not taken into account. If the influence of the second-order analysis for torsional buckling is relevant, you can use the add-on module FE-LTB.

RSTAB checks the member coefficient  $\varepsilon$ :

$$\varepsilon = L \cdot \sqrt{\frac{|N|}{E \cdot I}}$$

Equation 7.2

To avoid numerical problems, the RSTAB calculation uses series approaches for small member coefficients.

The axial force difference in the iterations serves as a break-off criterion. The stiffness-modifying axial force that is decisive for the second-order analysis is assumed to be constant along the entire member. The calculation stops as soon as a particular value of the axial force difference falls below the limit. It is possible to control this threshold in the dialog section *Precision and Tolerance* of the dialog tab *Global Calculation Parameters*.

For non-linear 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  $\varepsilon$ .

In the course of the calculation according to second-order analysis, the shear forces  $V_y$  and  $V_z$  are transformed in relation to the deformed member axis systems.

### Large deformation analysis

The large deformation analysis ("third order theory", "cable theory") takes into account longitudinal and transversal forces during the analysis of internal forces. If the calculation according to large deformation analysis is selected, all types of members will be calculated according to this analysis approach.

The procedure according to NEWTON-RAPHSON is used. The non-linear equation system is solved numerically by means of iterative approximations with tangents. You can influence the performance of convergence by the number of load increments to be set in the dialog tab *Global Calculation Parameters*.



After each iteration step the complete deformation of the model is corrected, and the stiffness matrix and the right side of the equation system is created for the deformed system. The right side contains the external loads and the internal forces of the deformed members (thus the complete equilibrium vector). The internal forces are transformed by the deformed member axis systems. If a globally defined load acts on a member, the load keeps its direction if the member axis is being deformed. A locally defined member load is treated "conservatively" as well: It is acting with constant size and constant load direction like on the undeformed system - regardless of the deformation.

### 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).

### Options

#### 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, RSTAB 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.

#### Modify loading by factor

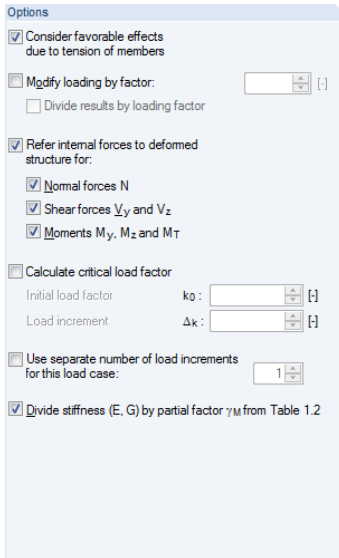
After ticking the check box, you can enter a factor into the input field by which all loads contained in the load case or combination will be multiplied. The factor is also reflected in the load vectors and values of the graphic. Generally, also negative factors are permitted.

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 not be edited with any factors. Instead, partial safety factors and combination coefficients must be applied for the superposition in the load and result combinations.

#### Refer internal forces to deformed structure

The option enables output for non-linear 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*.



### Calculate critical load factor

When calculating according to second-order or large deformation analysis, you can determine the critical load factor of a load case or combination iteratively. Based on the *Initial load factor*, the loading is increased continuously according to the *Load increment* until the model becomes unstable.

Make sure that the initial load factor is not too high and the load increment is not too wide so that the first eigenmode is not skipped. Furthermore, make sure to have set a sufficient number of possible iterations (see *Global Calculation Parameters*).



Determining the critical load factor has an unfavorable effect on the computing time because of the high number of load steps. Therefore, it is recommended to use this option only for special stability analyses.

### Use separate number of load increments for this load case

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 7.2.3, page 176).

### Divide stiffness (E, G) by partial safety factor $\gamma_M$

When the check box is ticked, RSTAB divides stiffnesses on the basis of the moduli of elasticity and shear by the partial safety factors  $\gamma_M$  for material. The factor  $\gamma_M$  must be defined for each material separately (see chapter 4.2, page 46).



### 7.2.1.2 Dialog Tab *Modify Stiffness*

The dialog tab is displayed only when the check box for advanced options is ticked in the *Options* tab of the dialog box *General Data* (see Figure 12.31, page 391).

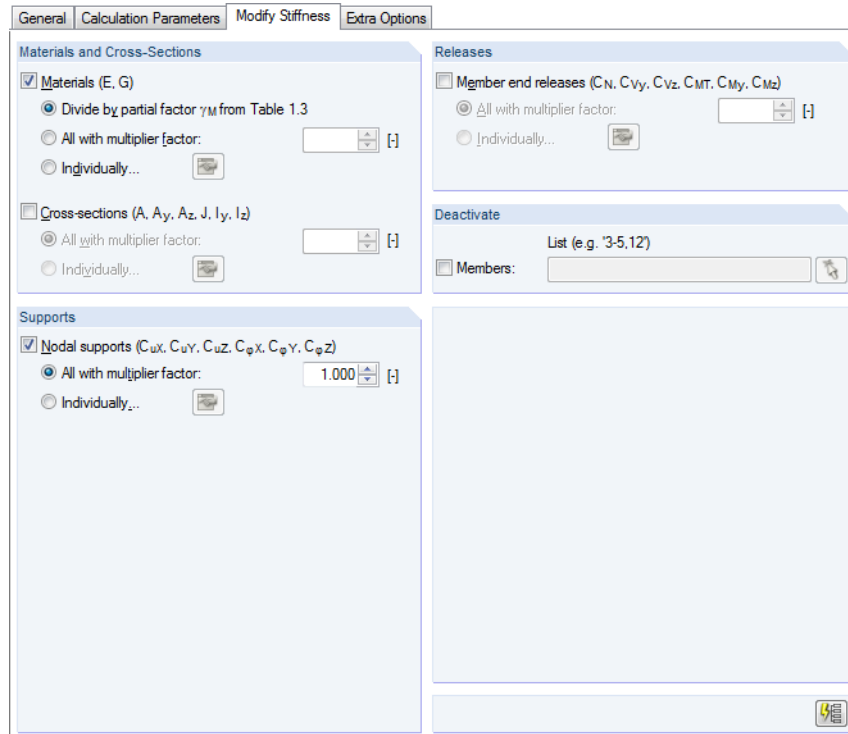


Figure 7.11: Dialog tab *Modify Stiffness*



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.

In general, the standard requires that material stiffnesses are divided by partial safety factors  $\gamma_M$ . Factors are stored with the material properties (see chapter 4.2, page 46). If a reduction by the provided factors is not wanted, individual specifications are possible:

- *All with multiplier factor*  
Specify a factor by which the stiffness of all materials (moduli of elasticity and shear) are globally multiplied.
- *Individually*  
Use the [Modify] button to open a new dialog box where you can assign a specific stiffness factor to each material.



Both selection fields are available for *Cross-Sections*, *Supports* and *Releases*. In this way, you can influence the calculation specifically.

#### Deactivate

You can define the *Members* which are not affected by the defined stiffness modifications, that means which are considered with the factor 1.0 in the calculation. You can select the members also graphically with the [^] function.



### 7.2.1.3 Dialog Tab *Extra Options*

The dialog tab is displayed only when the check box for advanced options is ticked in the *Options* tab of the dialog box *General Data* (see Figure 12.31, page 391).

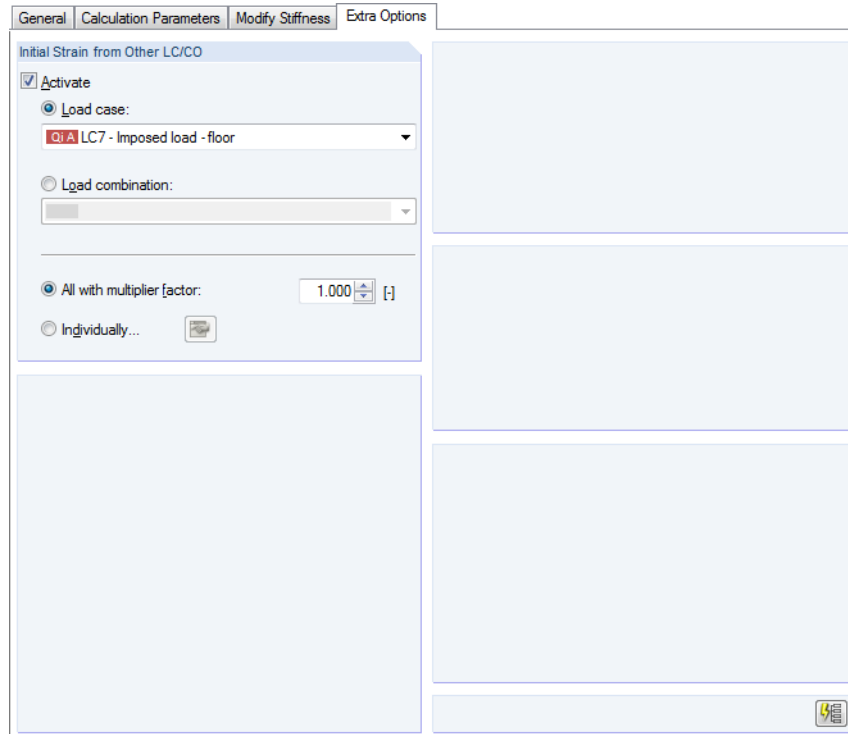


Figure 7.12: Dialog tab *Extra Options*

#### Initial strain from other load case/combination

Select a load case or load combination whose deformations you want to consider as initial deformation in the calculation. The 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:

- *All with multiplier factor*  
Deformations of the members are globally multiplied with 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.



## 7.2.2 Result Combinations

For basic information about superpositioning load cases in result combinations, see chapter 5.6 *Result Combinations* on page 128.

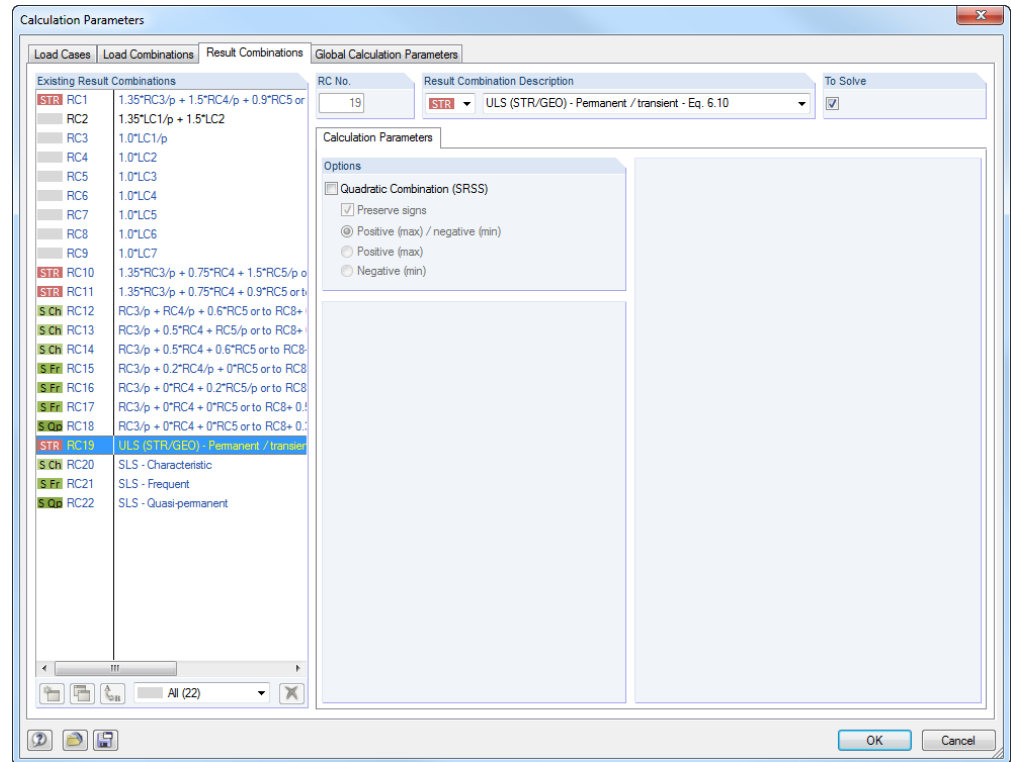


Figure 7.13: Dialog box *Calculation Parameters*, tab *Result Combinations*

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:

$$B = A_1 + A_2 + \dots + A_n$$

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:

$$B = \sqrt{A_1^2 + A_2^2 + \dots + A_n^2}$$

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.

### 7.2.3 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.

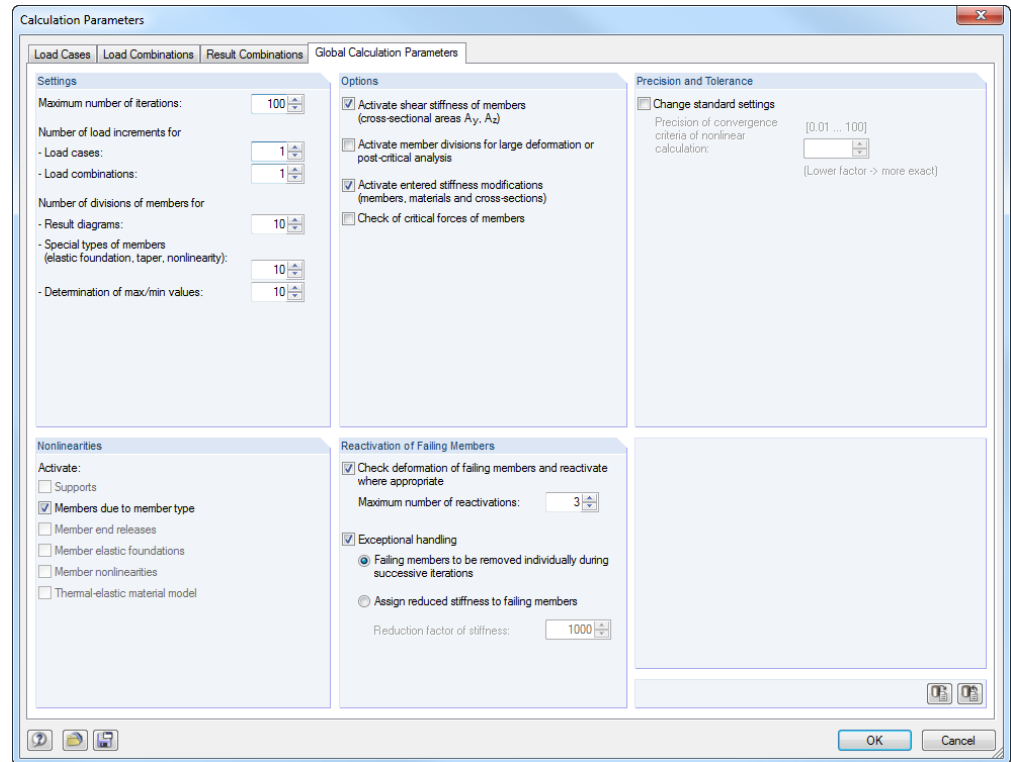


Figure 7.14: Dialog box *Calculation Parameters*, tab *Global Calculation Parameters*

## 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, RSTAB displays a corresponding message. The results can be displayed anyway.

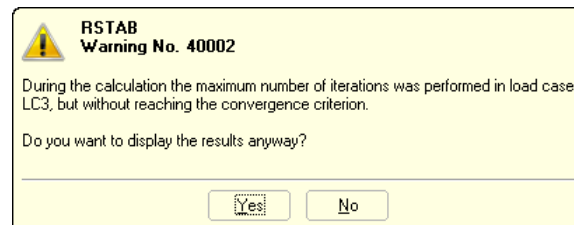


Figure 7.15: Message displayed in case of converging problems

### Number of load increments

Specifications of the input field take effect only for calculations according to second-order or large deformation analysis. 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 7.2.1.1, page 172). Then, the global specifications will be ignored.

#### Number of divisions of members for result diagrams

This input field has an influence on the graphical result diagram of members. If a division of 10 is set, RSTAB divides the length of the longest member in the system by 10. Based on such a system-related division length, RSTAB determines for each member the graphical result distributions on the division points.

#### Number of divisions of members for special types of members (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 foundation members (contact stresses), tapered members (interpolation of cross-section values) and members with plastic properties (yielding zones).

#### 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: 10) represents the basis for the extreme values shown in the results tables and graphic.

### Options

Options

Activate shear stiffness of members (cross-sectional areas  $A_y$ ,  $A_z$ )

Activate member divisions for large deformation or post-critical analysis

Activate entered stiffness modifications (members, materials and cross-sections)

Check of critical forces of members

#### 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. As the shear deformation is almost irrelevant for rolled and welded cross-sections, the check box is clear by default. 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

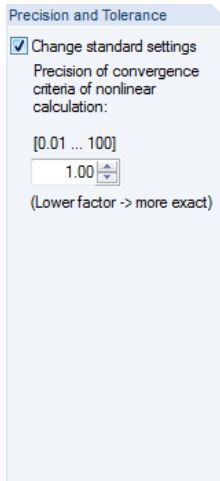
Beams can be divided by intermediate nodes for the calculation according to large deformation analysis to calculate such members with a higher accuracy. The number of divisions is taken from the input field for cable and foundation members.

#### Activate entered stiffness modifications (members, materials and cross-sections)

With the check box you can control if adjustments of the stiffnesses for members (see chapter 4.7, page 84) and cross-sections (see chapter 4.3, page 55) are considered in the calculation. The member and cross-section dialog boxes preset factors each with 1.00. Thus, the check mark in the check box usually involves no reduction or increase of the stiffnesses.

#### 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.



## Precision and tolerance

It is only rarely necessary to adjust the preset convergence parameters. Tick the check box *Change standard settings* to enable the input field below.

If non-linear effects are involved, or if data is calculated according to second-order or large deformation analysis, you can influence the calculation by means of the *Precision 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 factor entered in the input field you can define a certain sensitivity in order to prevent this oscillation effect.

The accuracy also effects the convergence criterion for deformation changes in calculations according to large deformation analysis where geometric non-linearities are considered.

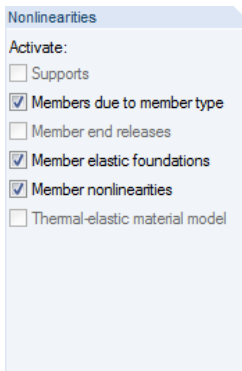
The default value is 1.0. The minimum factor is 0.01, the maximum value is 100.0. The higher the factor is, the less sensitive the break-off limit will be.

## Nonlinearities

When nonlinearly acting elements are used in the model, you can deactivate the effect of the following elements for the calculation:

- Failing supports (→ chapter 4.8, page 88)
- Failing members due to member type (→ chapter 4.7, page 75)
- Member end releases (→ chapter 4.4, page 64)
- Member elastic foundations (→ chapter 4.9, page 92)
- Member nonlinearities (→ chapter 4.10, page 93)
- Material nonlinearities (→ chapter 4.2, page 47)

It is recommended to disable the nonlinear effects only for test purposes, for example 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.



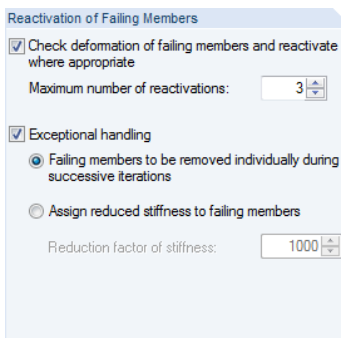
## Reactivation of failing members

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 structure 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 structure is unstable without the ties.

### Check deformation of failing members and reactivate where appropriate

When the check box is ticked, RSTAB analyzes the nodal displacements in each iteration. If member ends of a failed tie move away from each other, the member is 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 RSTAB 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, RSTAB 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, RSTAB 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 RSTAB displays on members small internal forces which can actually not be absorbed by the member due to its definition.

### 7.3 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 163).

#### Calculate all



To start the corresponding function,  
 select **Calculate All** on the **Calculate** menu,  
 or use the toolbar button shown on the left.



Figure 7.16: 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 structural systems with failing supports for tension, instabilities may occur during the batch calculation of the single load cases.
- If many load combinations and module design cases are available, RSTAB 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.

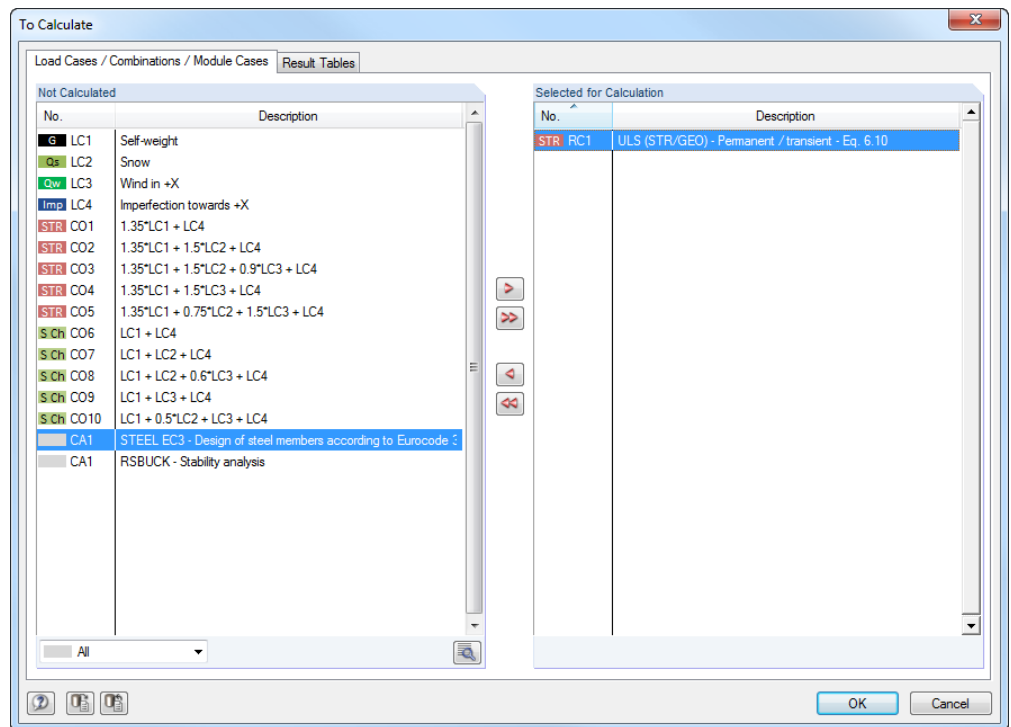


Figure 7.17: Dialog box To Calculate



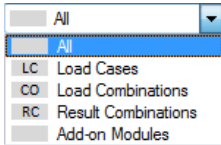


In the dialog section *Not Calculated* on the left, RSTAB lists all load cases, load combinations and result combinations as well as analysis cases of add-on modules for which no results exist. Use the button [▶] to transfer the selected entries to the list *Selected for Calculation*. You can also double-click 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 7.2, page 176) 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.

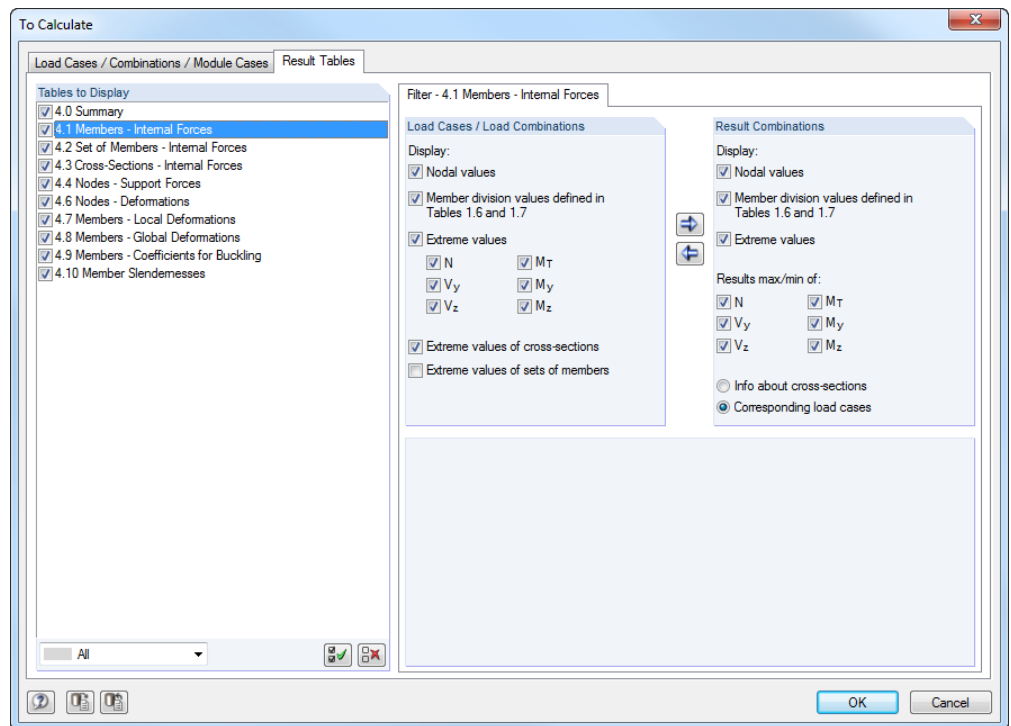


Figure 7.18: Dialog box *To Calculate*, tab *Result Tables*

More filter options are available for some results tables. They are described in chapter 8 *Results* together with the respective output tables (see for example Figure 8.4, page 186).

### 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 7.19: 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.

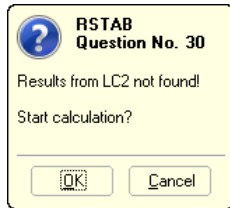


Figure 7.20: Query before calculation

### Calculate selected results

The toolbar menu *Calculate* offers you additional options for selecting results to be calculated:

- RSTAB results only
- Modules results only
- All results of all open models
- RSTAB 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 *Calculation* window. When nonlinear calculations are performed, you can observe the graphs of the maximum axial forces in a diagram in addition to the calculation steps through which RSTAB is running.

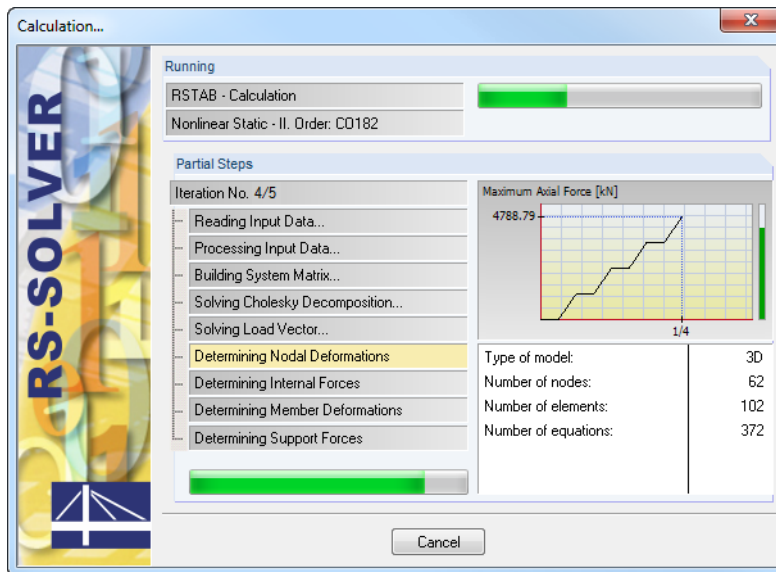


Figure 7.21: Calculation process

The green vertical bar on the right in the window visualizes 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.



It is necessary for the calculation to make sure 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 check if the control of the RAM disk space is activated.

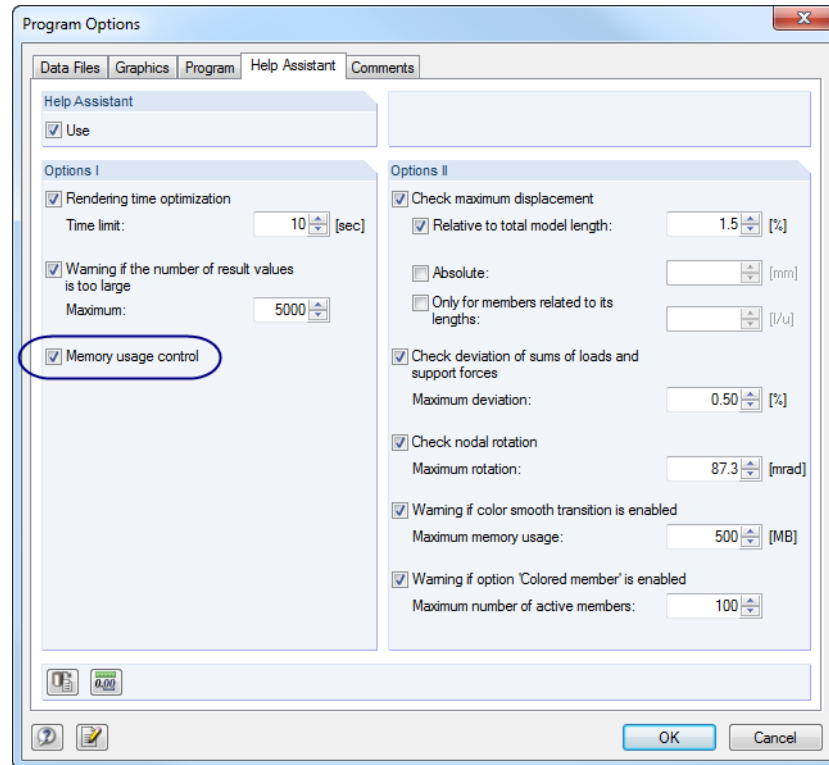


Figure 7.22: Dialog box *Program Options*, tab *Help Assistant*

## 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.



When data has been calculated, the additional tab *Results* appears in the navigator (see chapter 3.4.3, page 22) for controlling the graphical results display. The results are listed numerically in separate tables (see chapter 3.4.4, page 24).

### Colored relation scales in tables

The result columns of tables are partly highlighted in red or blue (see Figure 8.10, page 191). 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

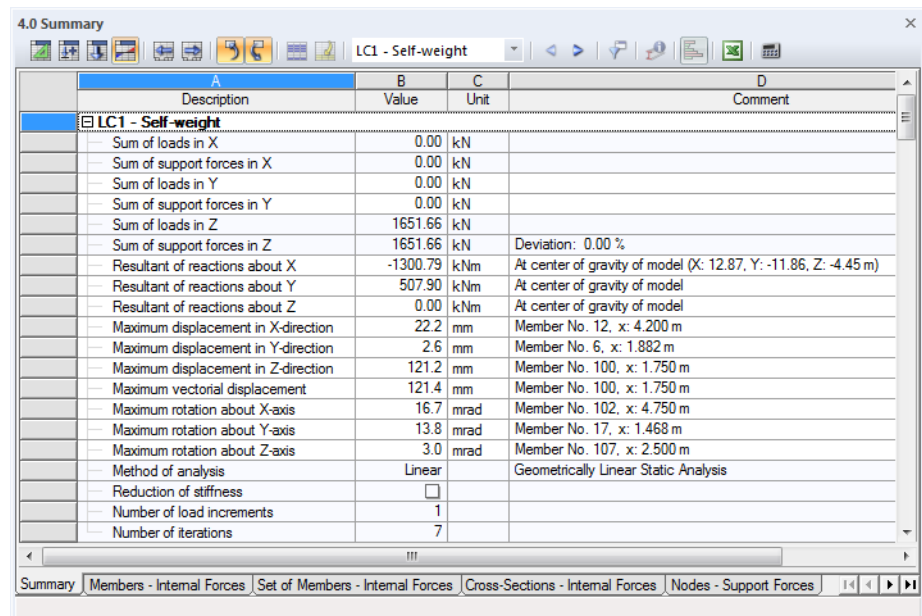


The displayed tables depend on the selections set in the dialog tab *Result Tables* of the dialog box *To Calculate* (see chapter 7.3, page 181).

## 8.0 Results Balance

### Table

Table 4.0 *Summary* represents a summary of the calculation process, sorted by load cases and combinations.



Description	Value	Unit	Comment
<b>LC1 - Self-weight</b>			
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	1651.66	kN	
Sum of support forces in Z	1651.66	kN	Deviation: 0.00 %
Resultant of reactions about X	-1300.79	kNm	At center of gravity of model (X: 12.87, Y: -11.86, Z: -4.45 m)
Resultant of reactions about Y	507.90	kNm	At center of gravity of model
Resultant of reactions about Z	0.00	kNm	At center of gravity of model
Maximum displacement in X-direction	22.2	mm	Member No. 12, x: 4.200 m
Maximum displacement in Y-direction	2.6	mm	Member No. 6, x: 1.882 m
Maximum displacement in Z-direction	121.2	mm	Member No. 100, x: 1.750 m
Maximum vectorial displacement	121.4	mm	Member No. 100, x: 1.750 m
Maximum rotation about X-axis	16.7	mrad	Member No. 102, x: 4.750 m
Maximum rotation about Y-axis	13.8	mrad	Member No. 17, x: 1.468 m
Maximum rotation about Z-axis	3.0	mrad	Member No. 107, x: 2.500 m
Method of analysis	Linear		Geometrically Linear Static Analysis
Reduction of stiffness	<input type="checkbox"/>		
Number of load increments	1		
Number of iterations	7		

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 7.14, page 176: dialog box *Calculation Parameters*, tab *Global Calculation Parameters*).

## 8.1 Members - Internal Forces

To control the graphical display of member internal forces, tick the check box for *Members* in the *Results* navigator. Table 4.1 shows the internal forces and moments in numerical form.

If the structure is a 2D model, RSTAB displays only the table columns of internal forces that are relevant for a planar structural system.

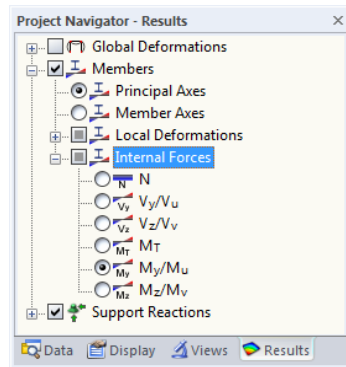


Figure 8.2: Results navigator: Members → Internal Forces

Member No.	Node No.	Location x [m]	N	V <sub>y</sub>	V <sub>z</sub>	M <sub>x</sub>	M <sub>y</sub>	M <sub>z</sub>	Corresponding Load Cases	
1	3	6.000	max N	-6.92	0.04	-6.26	0.01	-22.88	-0.19	CO 153
			min N	-57.97	-0.01	-56.25	0.02	-210.31	-0.76	CO 50
			max M <sub>y</sub>	-6.93	0.05	-6.14	0.01	-22.68	-0.22	CO 152
		0.000	min M <sub>y</sub>	-57.78	-1.57	-56.29	0.37	-210.46	-0.26	CO 49
			Max N	-6.92	0.04	-6.26	0.01	-22.88	-0.19	CO 153
			Min N	-126.25	0.52	-57.02	0.00	133.54	1.20	CO 50
2	2	0.000	Max M <sub>y</sub>	-126.05	2.72	-57.06	0.00	133.61	3.97	CO 49
			Min M <sub>y</sub>	-57.78	-1.57	-56.29	0.37	-210.46	-0.26	CO 49
			max N	-75.97	0.05	6.31	0.00	-14.96	0.06	CO 153
		1.500	min N	-127.16	0.49	59.92	0.00	-145.27	0.93	CO 50
			max M <sub>y</sub>	-75.97	0.05	6.31	0.00	-14.96	0.06	CO 153
			min M <sub>y</sub>	-126.97	0.02	64.06	-0.01	-157.31	-0.12	CO 22
3.000	2	1.500	max N	-58.71	0.05	6.36	0.00	-5.46	-0.01	CO 153
			min N	-109.39	0.42	60.86	-0.04	-54.71	0.22	CO 50
			max M <sub>y</sub>	-58.71	0.05	6.36	0.00	-5.46	-0.01	CO 153
		4.500	min M <sub>y</sub>	-109.12	-0.03	64.24	0.00	-61.11	-0.17	CO 22
			max N	-41.45	0.04	6.35	0.00	4.07	-0.08	CO 153
			min N	-92.08	0.26	60.78	0.02	36.50	-0.29	CO 50
4.500	2	max M <sub>y</sub>	-91.73	0.61	60.83	0.02	36.51	-1.60	CO 49	
		min M <sub>y</sub>	-41.50	0.06	6.53	0.00	3.93	-0.06	CO 152	
		max N	-24.20	0.04	6.32	0.00	13.57	-0.14	CO 153	
			min N	-75.25	0.06	60.09	0.00	127.27	-0.50	CO 50

Figure 8.3: Table 4.1 Members - Internal Forces

LC2 - Snow

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 4.6, page 72)
- 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.

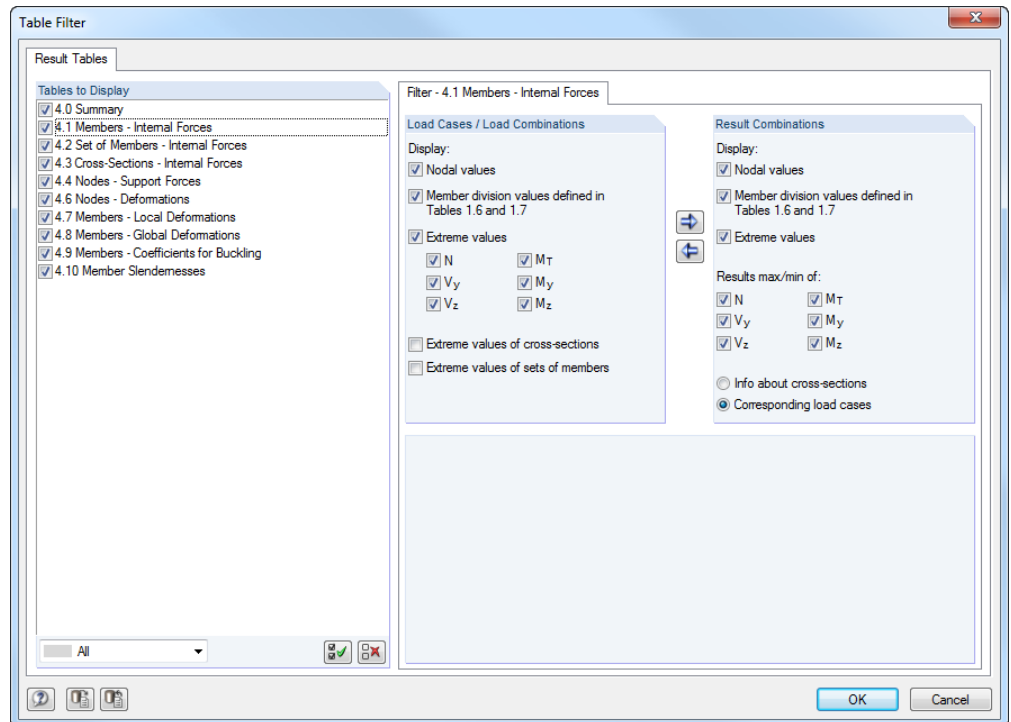


Figure 8.4: Dialog box *Table Filter*

The check boxes in the dialog box *Table Filter* control the type and amount of numerical output (see chapter 11.5.5, page 328).

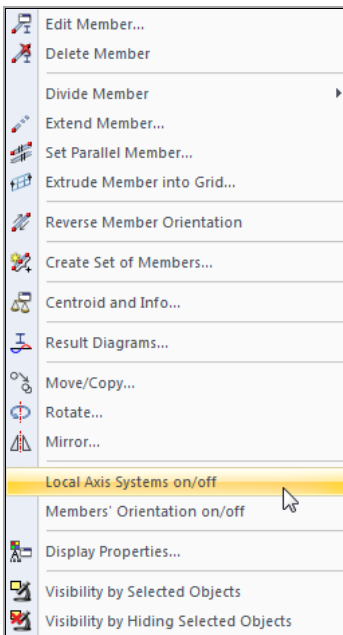
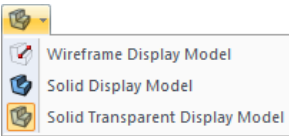
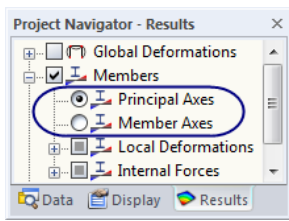
The diagram graphic for internal forces is based on the result values available in the member divisions that have been defined in the dialog tab *Global Calculation Parameters* of the dialog box *Calculation Parameters* (see chapter 7.2.3 page 177).

### 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 55)
$V_z / V_v$	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 8.1: Internal forces of members



Member context menu

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 4.7, page 79). When asymmetric cross-sections are used, you can select if internal forces refer to the principal axes  $u$  and  $v$  (see graphic on page 55) 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 non-linear 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 7.2.1, page 171).

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).

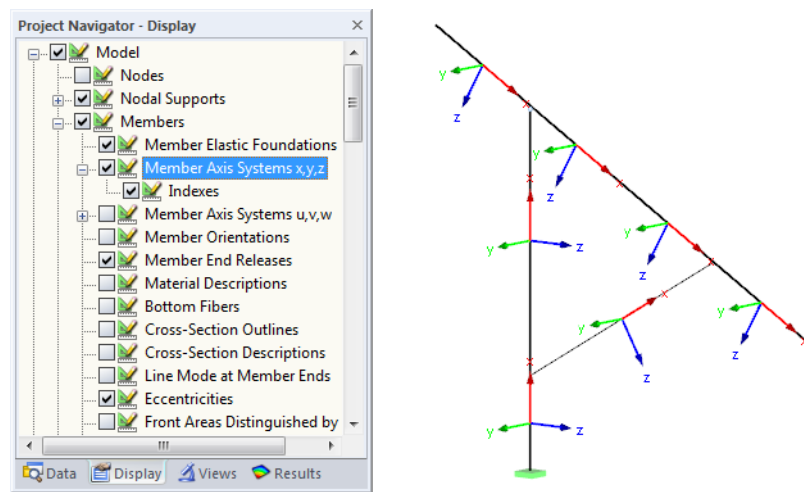


Figure 8.5: Selecting the local member axis systems in the *Display* navigator

The display of member axes can also be activated in the member context menu shown on the left.

The local member axis system affects the signs of internal forces.

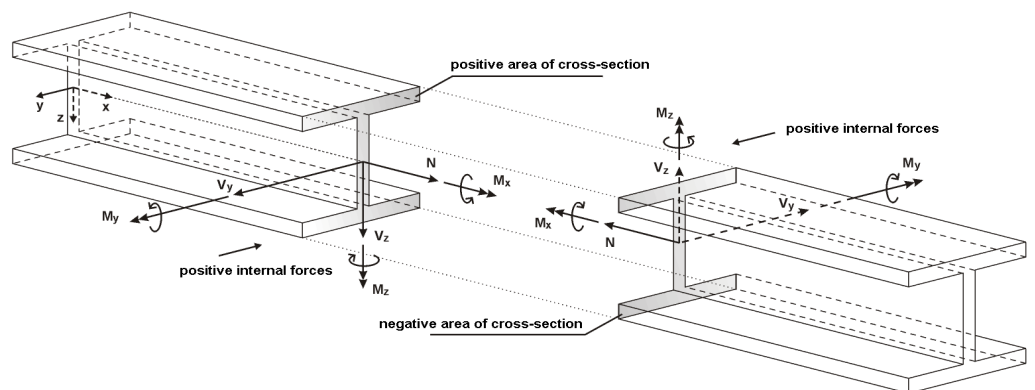


Figure 8.6: Positive definition of internal forces



The bending moment  $M_y$  is positive if tensile stresses occur on the positive member side (in direction of the  $z$ -axis).  $M_z$  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 8.4, page 186), RSTAB 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 328).

### 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 8.3). 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 131).

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 8.4, page 186). 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.



## 8.2 Sets of Members - Internal Forces

Table 4.2 shows the internal forces sorted by sets of members (see chapter 4.11, page 96).

Member No.	Node No.	Location x [m]	C	D	Forces [kN]			Moments [kNm]			Corresponding Load Cases
					N	V <sub>y</sub>	V <sub>z</sub>	M <sub>T</sub>	M <sub>y</sub>	M <sub>z</sub>	
<b>Set of Members No. 2: Secondary beam A-A</b>											
13	13	0.000	max N	<b>-31.71</b>	-0.03	26.23	-0.01	<b>-112.75</b>	-0.08	LC 1	
			min N	<b>-93.51</b>	-0.07	75.11	-0.05	<b>-332.56</b>	-0.14	LC 1,2	
			max V <sub>z</sub>	<b>-93.51</b>	-0.07	75.11	-0.05	<b>-332.56</b>	-0.14	LC 1,2	
			min V <sub>z</sub>	-31.71	-0.03	26.23	-0.01	-112.75	-0.08	LC 1	
			max M <sub>y</sub>	-31.71	-0.03	26.23	-0.01	<b>-112.75</b>	-0.08	LC 1	
			min M <sub>y</sub>	<b>-93.51</b>	-0.07	75.11	-0.05	<b>-332.56</b>	-0.14	LC 1,2	
15	16	6.274	max N	<b>-27.36</b>	0.08	-1.37	0.00	60.21	-0.36	LC 1	
			min N	<b>-80.86</b>	0.27	-4.81	-0.02	179.69	-1.27	LC 1,2	
			max V <sub>z</sub>	-27.36	0.08	-1.37	0.00	60.21	-0.36	LC 1	
			min V <sub>z</sub>	-80.86	0.27	-4.81	-0.02	179.69	-1.27	LC 1,2	
			max M <sub>y</sub>	-80.86	0.27	-4.81	-0.02	<b>179.69</b>	-1.27	LC 1,2	
			min M <sub>y</sub>	-27.36	0.08	-1.37	0.00	<b>60.21</b>	-0.36	LC 1	
15	6.274		MAX N	<b>-27.36</b>	0.08	-1.37	0.00	60.21	-0.36	LC 1	
13	0.000		MIN N	<b>-93.51</b>	-0.07	75.11	-0.05	<b>-332.56</b>	-0.14	LC 1,2	
13	0.000		MAX V <sub>z</sub>	<b>-93.51</b>	-0.07	75.11	-0.05	<b>-332.56</b>	-0.14	LC 1,2	
15	6.274		MIN V <sub>z</sub>	-80.86	0.27	-4.81	-0.02	179.69	-1.27	LC 1,2	
15	5.647		MAX M <sub>y</sub>	-81.31	0.27	0.32	-0.02	<b>181.10</b>	-1.10	LC 1,2	
13	0.000		MIN M <sub>y</sub>	-93.51	-0.07	75.11	-0.05	<b>-332.56</b>	-0.14	LC 1,2	
<b>Set of Members No. 3: Secondary beam B-B</b>											
41	41	0.000	max N	<b>-58.45</b>	0.01	28.95	0.03	-113.73	0.01	LC 1	
			min N	<b>-179.16</b>	0.01	88.54	0.09	<b>-348.74</b>	-0.08	LC 1,2	

Figure 8.7: Table 4.2 Set of Members - Internal Forces

The table's structure is similar to the one of table 4.1 *Members - Internal Forces* described in chapter 8.1. 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 328). 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.

### 8.3 Cross-sections - Internal Forces

Table 4.3 shows the internal forces sorted by cross-sections.

Member No.	Node No.	Location x [m]	Forces [kN]			Moments [kNm]		
			N	V <sub>y</sub>	V <sub>z</sub>	M <sub>T</sub>	M <sub>y</sub>	M <sub>z</sub>
<b>Cross-Section No. 15: HE A 200</b>								
51	51	0.000	-121.06	0.33	0.06	0.00	-0.08	0.76
	56	3.000	-86.54	0.35	0.06	0.00	0.11	-0.30
52	56	0.000	-37.96	-0.25	-0.04	-0.01	0.04	-0.51
	61	3.000	-3.44	-0.26	-0.04	-0.01	-0.08	0.25
59	55	0.000	-121.09	0.28	-0.06	0.00	0.05	0.53
	60	3.000	-86.57	0.29	-0.06	0.00	-0.15	-0.35
60	60	0.000	-37.97	-0.30	0.05	0.01	-0.07	-0.60
	65	3.000	-3.45	-0.30	0.05	0.01	0.07	0.30
52	MAX N	3.000	<b>-3.44</b>	-0.26	-0.04	-0.01	-0.08	0.25
59	MIN N	0.000	<b>-121.09</b>	0.28	-0.06	0.00	0.05	0.53
51	MAX V <sub>z</sub>	1.200	-107.25	0.36	<b>0.06</b>	0.00	0.00	0.35
59	MIN V <sub>z</sub>	0.750	-112.46	0.29	<b>-0.06</b>	0.00	0.00	0.32
51	MAX M <sub>y</sub>	3.000	-86.54	0.35	0.06	0.00	<b>0.11</b>	-0.30
59	MIN M <sub>y</sub>	3.000	-86.57	0.29	-0.06	0.00	<b>-0.15</b>	-0.35
<b>Cross-Section No. 16: Rectangle 200/200</b>								
57	54	0.000	-200.10	0.02	-1.26	0.02	4.46	0.08
	59	3.000	-150.93	0.02	-1.30	0.02	0.59	0.04
57	MAX N	3.000	<b>-150.93</b>	0.02	-1.30	0.02	0.59	0.04
57	MIN N	0.000	<b>-200.10</b>	0.02	-1.26	0.02	4.46	0.08
57	MAX V <sub>z</sub>	0.000	-200.10	0.02	<b>-1.26</b>	0.02	4.46	0.08
57	MIN V <sub>z</sub>	2.850	-153.39	0.02	<b>-1.30</b>	0.02	0.79	0.04
57	MAX M <sub>y</sub>	0.000	-200.10	0.02	-1.26	0.02	<b>4.46</b>	0.08
57	MIN M <sub>y</sub>	3.000	-150.93	0.02	-1.30	0.02	<b>0.59</b>	0.04
<b>Cross-Section No. 2 - 3 : IPE 300</b>								
8	8	0.000	-22.12	-0.09	-14.24	0.05	-26.33	0.15
	9	3.011	-22.62	-0.14	-19.37	0.01	-76.70	0.74
18	18	0.000	-23.39	-0.16	-23.51	0.02	-38.13	0.14
	19	3.011	-24.20	-0.17	-31.62	-0.03	-120.91	0.72
28	28	0.000	-41.27	0.09	-24.61	-0.01	-49.73	-0.05
	29	3.011	-42.15	0.11	-32.53	0.08	-135.51	-0.52
46	46	0.000	-59.22	0.12	-21.14	-0.02	-42.40	-0.07

Figure 8.8: Table 4.3 Cross-Sections - Internal Forces

The table's structure is similar to the one of table 4.1 *Members – Internal Forces* described in chapter 8.1. 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 328).

## 8.4 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 4.4 shows the support forces and moments in numerical form.

If the structure is a 2D model, RSTAB displays only the table columns of support forces and moments that are relevant for a planar structural system.

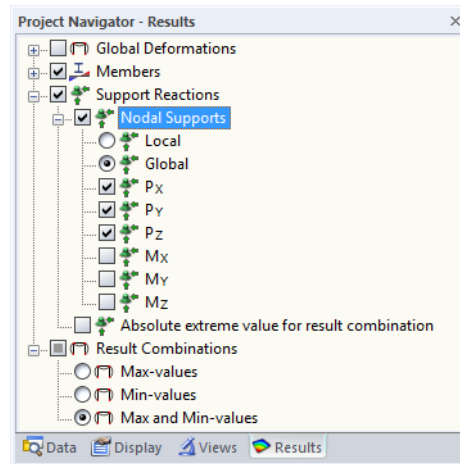
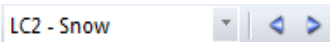


Figure 8.9: Results navigator: Support Reactions → Nodal Supports

Node No.	Support Forces [kN]			Support Moments [kNm]			
	P <sub>X</sub>	P <sub>Y</sub>	P <sub>Z</sub>	M <sub>X</sub>	M <sub>Y</sub>	M <sub>Z</sub>	
55	-0.05	-0.06	4.61	-0.23	0.04	0.00	
71	-0.03	0.05	0.00	0.30	0.22	0.00	
72*	0.01	0.05	0.00	0.37	-0.08	0.00	φ <sub>Y</sub> = -45.00 °
73*	0.06	0.02	0.00	0.14	-0.41	0.00	φ <sub>Y</sub> = -45.00 °
Σ Forces	0.00	0.00	27.30				
Σ Loads	0.00	0.00	71.21				

Figure 8.10: Table 4.4 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<sub>X</sub> / P<sub>Y</sub> / P<sub>Z</sub>

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 8.10. 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 introduced 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<sub>Z</sub>, and a wind load against the global axis X has a negative support force P<sub>X</sub>. 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 also display in the work window the signs of the forces that are transferred into the support. Select *Results* in the *Display* navigator and tick the corresponding check box.

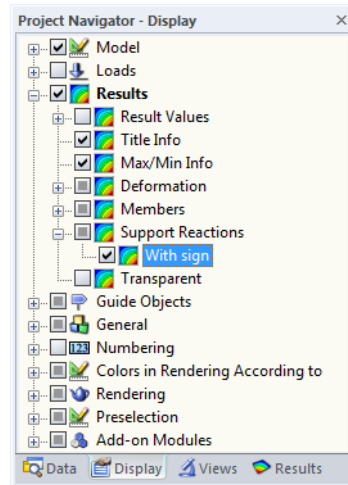


Figure 8.11: *Display* navigator, *Results* → *Support Reactions* → *With sign*

The signs in the graphic refer to the global axis system XYZ or the rotated local axis system X'Y'Z' which is rotated. It is advisable to activate these signs only for visualizing the transferred forces because otherwise they lead to misinterpretations.

### 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 8.11). 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 display type,

point to **Display Properties** on the **Options** menu, and select **Edit**.

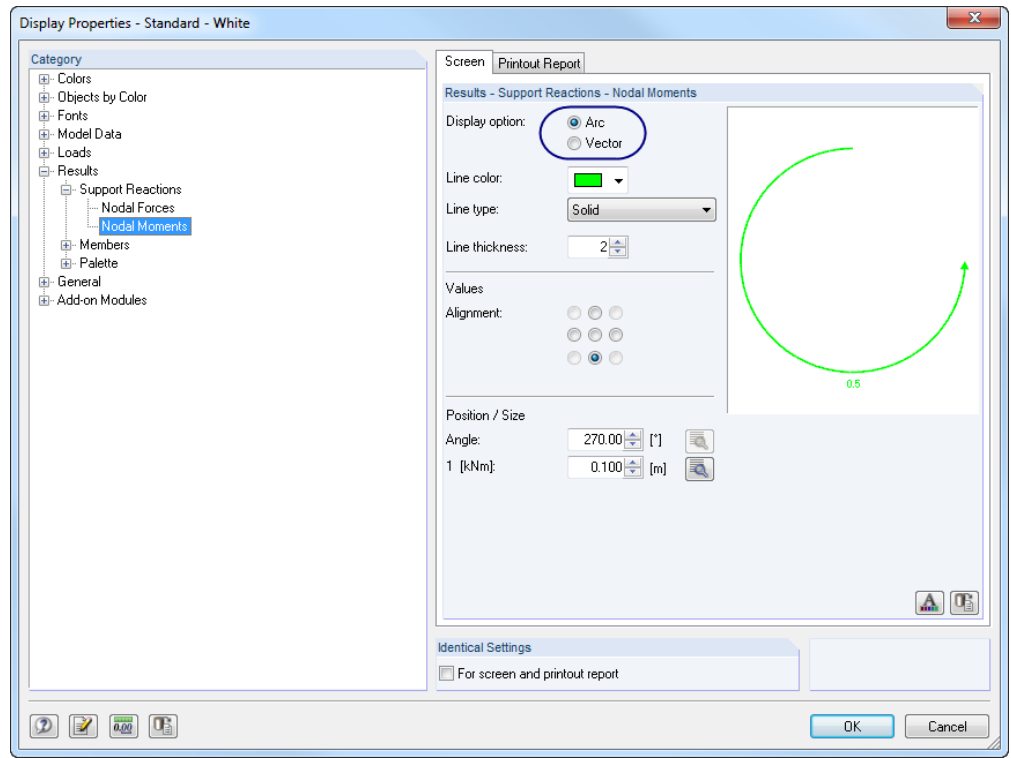


Figure 8.12: Dialog box *Display Properties*: nodal moments with display option *Arc*

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 8.10, page 191). The corresponding nodes are marked by an asterisk (\*).

### Check sums

For load cases and load combinations RSTAB displays check sums of loads and support reactions at the end of the table. Differences will occur between  $\Sigma Forces$  and  $\Sigma Loads$  if the model has also members with elastic foundations. Therefore, the  $\Sigma Forces$  of table 4.5 must also be considered for the total summary.



### 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.

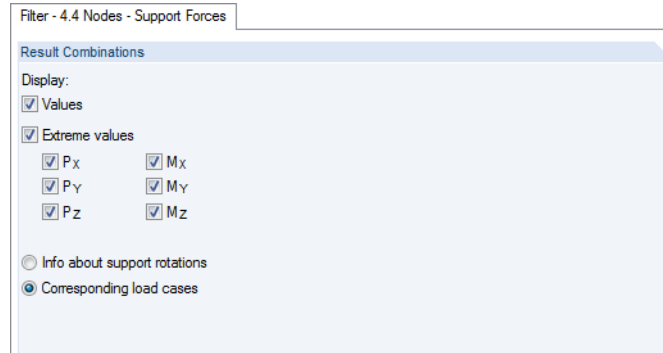


Figure 8.13: Dialog box *Table Filter* (dialog section)

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 184). Use the *Results* navigator to visualize the resultant forces also on the model.

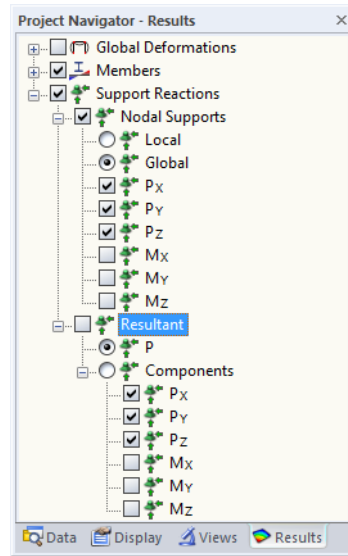


Figure 8.14: *Results* navigator: *Support Reactions* → *Resultant*

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.

## 8.5 Members - Contact Forces

When members with elastic foundations exist in the model (see chapter 4.9, page 91), the contact forces and moments are shown numerically in table 4.5. To control the graphical display of results, tick the check box for *Members* in the *Results* navigator.

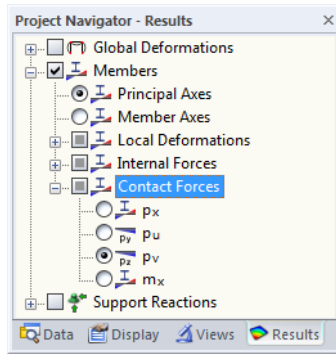


Figure 8.15: Results navigator: Members → Contact Forces

Member No.	Node No.	Location x [m]	Contact Forces [kN/m]			Moments m <sub>x</sub> [kNm/m]	Cross-Section
			p <sub>x</sub>	p <sub>y</sub>	p <sub>z</sub>		
69	39	0.000	0.000	0.000	9.337	0.000	16 - Rectangle 200/450
	40	6.250	0.000	0.000	1.018	0.000	
	Max p <sub>y</sub>	0.000	0.000	0.000	9.337	0.000	
	Min p <sub>y</sub>	0.000	0.000	0.000	9.337	0.000	
	Max p <sub>z</sub>	3.125	0.000	0.000	13.491	0.000	
Min p <sub>z</sub>	6.250	0.000	0.000	1.018	0.000		
74	59	0.000	0.000	0.000	2.835	0.000	16 - Rectangle 200/450
	60	6.250	0.000	0.000	1.008	0.000	
	Max p <sub>y</sub>	0.000	0.000	0.000	2.835	0.000	
	Min p <sub>y</sub>	0.000	0.000	0.000	2.835	0.000	
	Max p <sub>z</sub>	3.125	0.000	0.000	13.932	0.000	
Min p <sub>z</sub>	6.250	0.000	0.000	1.008	0.000		
Σ Forces			0.000	0.000		132.477	
Σ Loads			0.00	0.00		625.00	

Figure 8.16: Table 4.5 Members - Contact Forces

### Node no.

In the first two table rows, the numbers of the start and end node are displayed for each foundation member. The remaining rows inform you about the types of extreme values available for contact forces and moments.

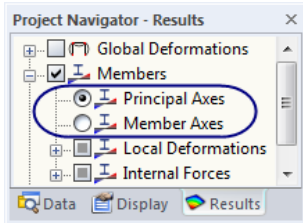


To adjust the default settings for the output of extreme values, select **View** on the **Table** menu and click **Result Filter**, or use the button in the table toolbar shown on the left.

### 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 4.6, page 72)
- 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 55) 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 8.5). The signs comply with the usual definitions explained in chapter 8.1 on page 187 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 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 RSTAB displays check sums of loads and support reactions at the end of the table. Differences will occur between  $\Sigma Forces$  and  $\Sigma Loads$  if the model has also nodal supports. Therefore, the  $\Sigma Forces$  of table 4.4 must also be considered for the total summary.



The contact forces are determined with the member divisions which apply to the *Result diagrams* and *max/min values*. Find the divisions shown in the dialog tab *Global Calculation Parameters* of the *Calculation Parameters* dialog box (see Figure 7.14, page 176). The entry for *Special types of members* has only an effect if a foundation member is used with an asymmetric cross-section.



## 8.6 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 4.6 shows the deformations of nodes in numerical form.

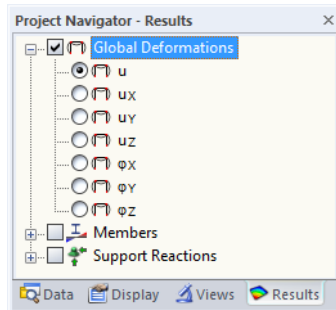


Figure 8.17: Results navigator: Global Deformations

Node No.	Displacements [mm]				Rotations [mrad]		
	u	ux	uy	uz	φx	φy	φz
37	0.8	0.0	0.7	0.4	0.3	0.9	0.0
38	1.5	1.1	0.9	0.4	0.4	-0.5	-0.2
39	2.3	2.2	0.2	0.4	0.1	-2.1	-0.6
40	3.4	3.3	-0.7	0.5	-0.4	-9.5	-0.6
41	24.0	-24.0	1.5	0.8	0.4	-8.8	0.5
42	66.3	-18.9	2.8	63.5	11.8	-33.0	-0.8
43	192.7	-8.2	2.6	192.5	32.4	-40.7	-3.2
44	333.7	3.2	3.9	333.7	49.4	0.5	-0.8
45	188.8	14.9	1.2	188.2	26.8	40.8	1.7
46	65.5	25.6	1.8	60.3	9.2	32.2	0.9

Figure 8.18: Table 4.6 Nodes - Deformations

The displacements and rotations are listed by nodes.

### Displacements / rotations

The deformations have the following meanings:

u	Total displacement
ux	Displacement in direction of the global axis X
uy	Displacement in direction of the global axis Y
uz	Displacement in direction of the global axis Z
φx	Rotation about the global axis X
φy	Rotation about the global axis Y
φz	Rotation about the global axis Z

Table 8.2: Nodal deformations

## 8.7 Members - Local 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 55) or to the standard input axes *y* and *z*. Table 4.7 shows the members' local deformations in numerical form.

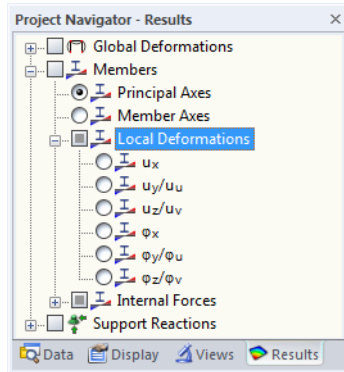
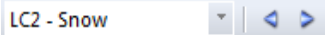


Figure 8.19: Results navigator: Members → Local Deformations

Member No.	Node No.	Location x [m]	u	Displacements [mm]			Rotations [mrad]			Cross-Section
				u <sub>x</sub>	u <sub>y</sub> / u <sub>u</sub>	u <sub>z</sub> / u <sub>v</sub>	φ <sub>x</sub>	φ <sub>y</sub> / φ <sub>u</sub>	φ <sub>z</sub> / φ <sub>v</sub>	
107	9	0.000	49.4	1.8	49.3	0.5	-4.0	0.0	-38.3	12-QRO 80
	19	5.000	74.4	1.8	74.4	0.7	-31.5	0.2	3.3	
	Max u <sub>x</sub>	5.000	74.4	1.8	74.4	0.7	-31.5	0.2	3.3	
	Min u <sub>x</sub>	0.000	49.4	1.8	49.3	0.5	-4.0	0.0	-38.3	
	Max u <sub>y</sub>	5.000	74.4	1.8	74.4	0.7	-31.5	0.2	3.3	
	Min u <sub>y</sub>	1.250	25.7	1.8	25.6	1.0	-10.8	-0.6	-2.5	
	Max u <sub>z</sub>	2.750	40.3	1.8	40.2	1.6	-19.1	0.0	18.0	
	Min u <sub>z</sub>	0.000	49.4	1.8	49.3	0.5	-4.0	0.0	-38.3	
	Max φ <sub>x</sub>	0.000	49.4	1.8	49.3	0.5	-4.0	0.0	-38.3	
	Min φ <sub>x</sub>	5.000	74.4	1.8	74.4	0.7	-31.5	0.2	3.3	
	Max φ <sub>y</sub>	4.250	67.4	1.8	67.4	1.0	-27.3	0.6	14.2	
	Min φ <sub>y</sub>	1.250	25.7	1.8	25.6	1.0	-10.8	-0.6	-2.5	
	Max φ <sub>z</sub>	3.250	49.7	1.8	49.7	1.5	-21.8	0.3	19.4	
	Min φ <sub>z</sub>	0.000	49.4	1.8	49.3	0.5	-4.0	0.0	-38.3	
108	19	0.000	74.4	1.8	74.4	0.7	-31.5	0.2	3.3	12-QRO 80
	29	5.000	70.2	1.7	70.2	0.8	-19.6	-0.4	-2.1	

Figure 8.20: Table 4.7 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 4.6, page 72)
- Extreme values (*Max/Min*) of displacements and rotations



To adjust the default setting of the displayed x-locations, select **View** on the **Table** menu and click **Result Filter**, or use the button in the table toolbar shown on the left.

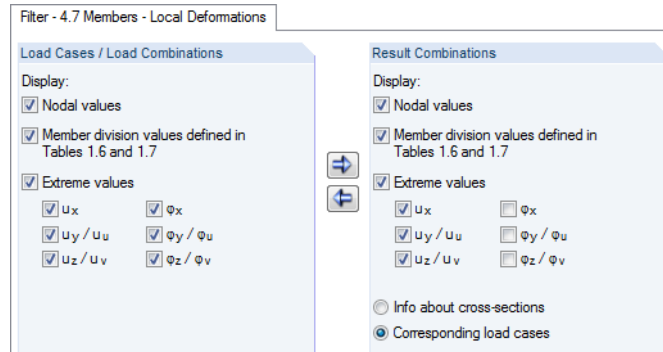


Figure 8.21: Dialog box *Table Filter* (dialog section)

The check boxes in the dialog box *Table Filter* control the type and amount of numerical output.

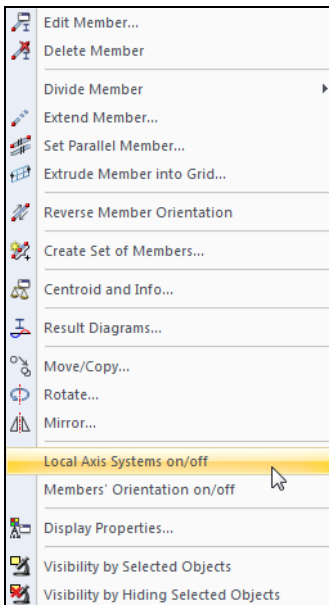
### Displacements / rotations

The member deformations have the following meanings:

$ u $	Absolute total displacement (not for result combinations)
$u_x$	Displacement of member in direction of its longitudinal axis
$u_y / u_u$	Displacement of member in direction of local axis y or u (see page 55)
$u_z / u_v$	Displacement of member in direction of local axis z or v
$\varphi_x$	Rotation of member about its longitudinal axis
$\varphi_y / \varphi_u$	Rotation of member about local axis y or u
$\varphi_z / \varphi_v$	Rotation of member about local axis z or v

Table 8.3: Member deformations

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 8.5, page 187). You can also use the member context menu shown on the left.



Member context menu

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 206).

Moreover, member deformations can be visualized as animation of the deformation process (see chapter 9.8, page 221).



## 8.8 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 4.8 shows the global deformations of members in numerical form.

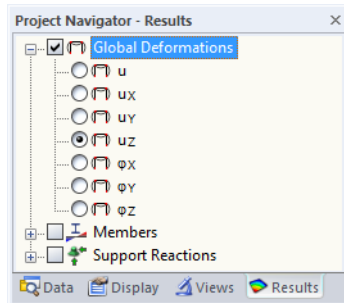


Figure 8.22: Results navigator: Global Deformations

Member No.	Node No.	Location x [m]	u	Displacements [mm]			Rotations [mrad]			Cross-Section
				u <sub>x</sub>	u <sub>y</sub>	u <sub>z</sub>	φ <sub>x</sub>	φ <sub>y</sub>	φ <sub>z</sub>	
43	43	0.000	192.8	-8.2	3.0	192.6	32.5	-40.7	-3.1	2 - IPE 300 ;
	44	6.274	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	
		5.960	333.7	3.2	3.7	333.6	48.6	-1.8	-1.2	
	Max u <sub>x</sub>	0.000	192.8	-8.2	3.0	192.6	32.5	-40.7	-3.1	
	Min u <sub>x</sub>	0.000	192.8	-8.2	3.0	192.6	32.5	-40.7	-3.1	
	Max u <sub>y</sub>	6.274	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	
	Min u <sub>y</sub>	3.585	307.5	1.3	0.4	307.4	42.0	-20.4	-3.7	
	Max u <sub>z</sub>	6.274	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	
	Min u <sub>z</sub>	0.000	192.8	-8.2	3.0	192.6	32.5	-40.7	-3.1	
	Max φ <sub>x</sub>	6.274	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	
	Min φ <sub>x</sub>	0.000	192.8	-8.2	3.0	192.6	32.5	-40.7	-3.1	
	Max φ <sub>y</sub>	6.274	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	
	Min φ <sub>y</sub>	0.000	192.8	-8.2	3.0	192.6	32.5	-40.7	-3.1	
	Max φ <sub>z</sub>	6.274	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	
	Min φ <sub>z</sub>	2.196	272.0	-1.6	1.2	272.0	38.2	-30.4	-4.3	
44	44	0.000	333.9	3.2	4.8	333.8	49.4	0.5	-0.8	2 - IPE 300 ;
	45	6.274	188.9	14.9	1.5	188.3	27.0	40.8	1.6	

Figure 8.23: Table 4.8 Members - Global Deformations

The table columns *Node No.* and *Location x* correspond to the columns of the previous results table 4.7 *Members - Local Deformations*.

### Displacements / rotations

The member deformations have the following meanings:

u	Absolute total displacement (not for result combinations)
u <sub>x</sub>	Displacement of member in direction of global axis X
u <sub>y</sub>	Displacement of member in direction of global axis Y
u <sub>z</sub>	Displacement of member in direction of global axis Z
φ <sub>x</sub>	Rotation of member about global axis X
φ <sub>y</sub>	Rotation of member about global axis Y
φ <sub>z</sub>	Rotation of member about global axis Z

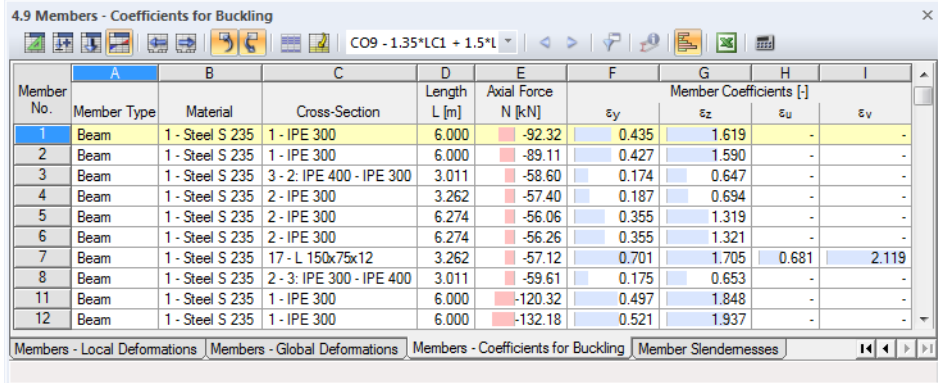
Table 8.4: Global member deformations

## 8.9 Members - Member Coefficients for Buckling

When you calculate member models subjected to pressure according to second-order analysis, the member coefficient  $\varepsilon$  is important (see chapter 7.2.1, page 170). Each member has its own 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 4.9 shows the member coefficients which are governing for buckling. There is no graphical output option.



Member No.	Member Type	Material	Cross-Section	Length L [m]	Axial Force N [kN]	Member Coefficients [-]			
						$\varepsilon_y$	$\varepsilon_z$	$\varepsilon_u$	$\varepsilon_v$
1	Beam	1 - Steel S 235	1 - IPE 300	6.000	-92.32	0.435	1.619	-	-
2	Beam	1 - Steel S 235	1 - IPE 300	6.000	-89.11	0.427	1.590	-	-
3	Beam	1 - Steel S 235	3 - 2: IPE 400 - IPE 300	3.011	-58.60	0.174	0.647	-	-
4	Beam	1 - Steel S 235	2 - IPE 300	3.262	-57.40	0.187	0.694	-	-
5	Beam	1 - Steel S 235	2 - IPE 300	6.274	-56.06	0.355	1.319	-	-
6	Beam	1 - Steel S 235	2 - IPE 300	6.274	-56.26	0.355	1.321	-	-
7	Beam	1 - Steel S 235	17 - L 150x75x12	3.262	-57.12	0.701	1.705	0.681	2.119
8	Beam	1 - Steel S 235	2 - 3: IPE 300 - IPE 400	3.011	-59.61	0.175	0.653	-	-
11	Beam	1 - Steel S 235	1 - IPE 300	6.000	-120.32	0.497	1.848	-	-
12	Beam	1 - Steel S 235	1 - IPE 300	6.000	-132.18	0.521	1.937	-	-

Figure 8.24: Table 4.9 Members - Coefficients for Buckling

The listed member coefficients are sorted by member numbers.

### Member type

The member types are indicated for information (see chapter 4.7, page 74). RSTAB 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 $\varepsilon_y / \varepsilon_z$

The member coefficient depends on the member length L, the compressive force N and the stiffness  $E \cdot I$ .

$$\varepsilon = L \cdot \sqrt{\frac{N}{E \cdot I}}$$

Equation 8.1: Member coefficient  $\varepsilon$

Table columns F and G show the member coefficients referring to the local member axis system y and z. When asymmetric cross-sections are used, columns H and I appear where also the member coefficients are shown in relation to the principal axes u and v.

## 8.10 Member Slendernesses

Table 4.10 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.

Member No.	Cross-Section	Length L [m]	Effective Length Factors [-]		Slenderness [-]			
			$k_{cr,y}$	$k_{cr,z}$	$\lambda_y$	$\lambda_z$	$\lambda_u$	$\lambda_v$
1	1 - IPE 300	6.000	1.000	1.000	48.13	179.07	-	-
2	1 - IPE 300	6.000	1.000	1.000	48.13	179.07	-	-
3	3 - 2: IPE 400 - IPE 300	3.011	1.000	1.000	24.16	89.87	-	-
4	2 - IPE 300	3.262	1.000	1.000	26.17	97.37	-	-
5	2 - IPE 300	6.274	1.000	1.000	50.33	187.25	-	-
6	2 - IPE 300	6.274	1.000	1.000	50.33	187.25	-	-
7	17 - L 150x75x12	3.262	1.000	1.000	66.24	206.02	66.24	206.02
8	2 - 3: IPE 300 - IPE 400	3.011	1.000	1.000	24.16	89.87	-	-
11	1 - IPE 300	6.000	1.000	1.000	48.13	179.07	-	-
12	1 - IPE 300	6.000	1.000	1.000	48.13	179.07	-	-

Figure 8.25: Table 4.10 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 4.7, page 83), the EULER buckling mode 2 is assumed: The buckling length is consequently equal to the member length. More accurate analyses can be performed with the add-on module RSBUCK or in various design modules such as STEEL EC3.

### Slendernesses $\lambda_y$ / $\lambda_z$

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  $\lambda$

Table columns E and F show the slendernesses referring to the local member axis system y and z. When asymmetric cross-sections are used, columns G and H appear where also the slenderness ratios are shown in relation to the principal axes u and v.

# 9. Results Evaluation

## 9.1 Available Results

To open the corresponding dialog box, select **Available Results** on the **Results** menu.

A dialog box with an overview about all calculated load cases and combinations appears.

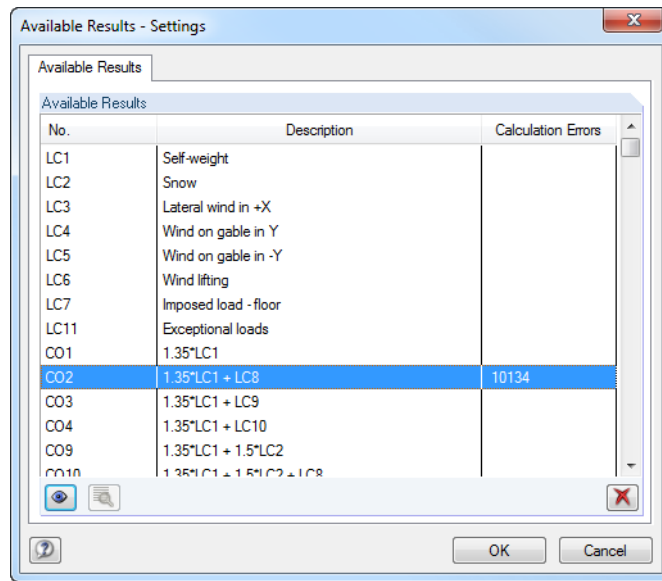


Figure 9.1: Dialog box *Available Results - Settings*

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 327).



## 9.2 Results Selection



Use the *Results* navigator to control the display for deformations, internal forces, contact forces and/or support reactions.

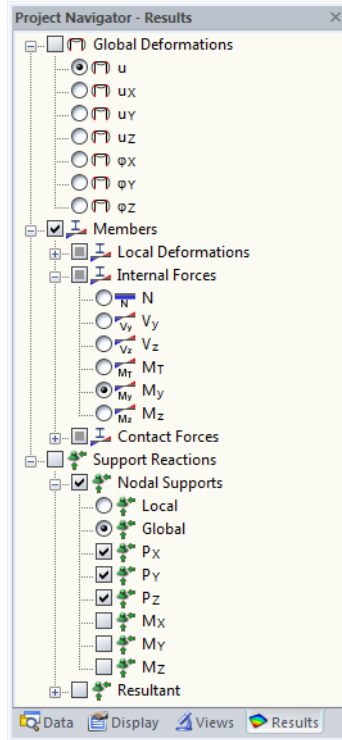


Figure 9.2: *Results* navigator

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.

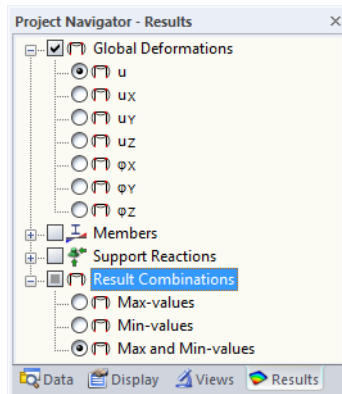


Figure 9.4: *Results* navigator for a result combination



You can select between three options affecting the graphical results display of deformations, internal and support forces: The *Max-values* and *Min-values* can be displayed separately. To display both envelopes from all extreme values at the same time, select *Max and Min-values*.

### 9.3 Results Display

The way results are represented is set in the *Display* navigator.

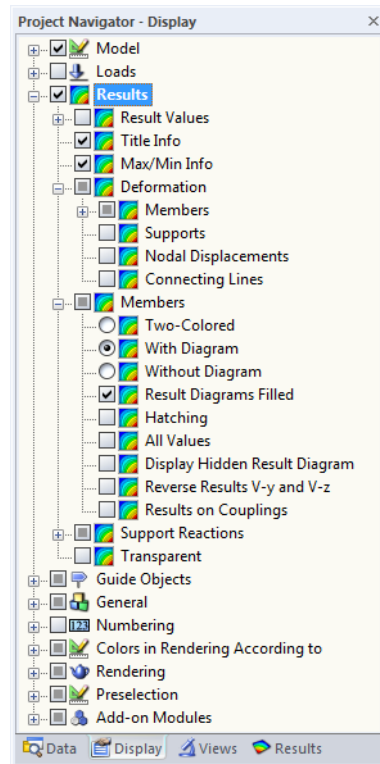


Figure 9.5: Display navigator: Results



In the *Results* navigator, you specify which results are displayed. The *Display* navigator defines the way how results are represented.

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 7.14, page 176). If a division of 10 is set, RSTAB divides the length of the longest member in the system by 10. With the system-related division length RSTAB determines for each member the graphical result distributions on the division points.



If the member internal forces are represented with colors using the display options *With Diagram* and *Without Diagram*, 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 27.

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).

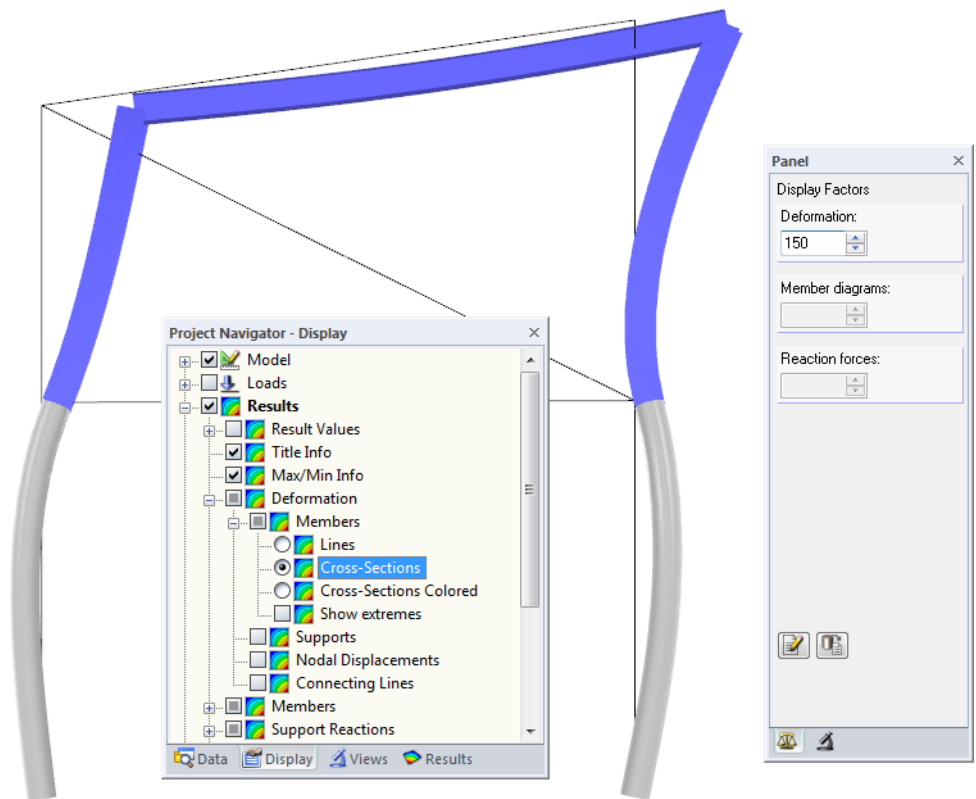


Figure 9.6: Scaled imaging of member deformations in 3D rendering



You can control the scaling of deformations and internal forces by settings in the control panel tab *Display Factors* (middle or left). The *Filter* tab (right) is used for a specific selection of members whose results you want to display (see Figure 9.24, page 221). Both panel tabs are described in chapter 3.4.6, page 29.

## 9.4 Info about Member



RSTAB offers a special output function for member results. To open the corresponding dialog box,

select **Info About Member** on the **Tools** menu

or use the info button in the toolbar.

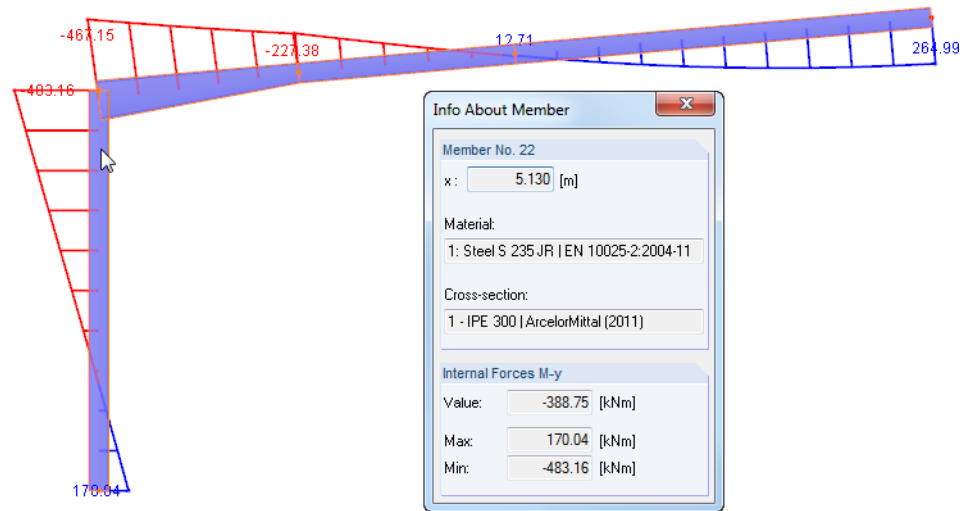


Figure 9.7: Dialog box *Info About Member*

An *Info* window appears. When you move the pointer across a member, the window informs you in addition to material and member cross-section about the deformation values or internal forces available on the current pointer position (location *x* of the member).

## 9.5 Result Diagrams

### 9.5.1 Result diagram

The result diagram makes it possible to see in detail the result distribution of members and sets of members.

First, select the members or sets of members (multiple selection by holding down the [Ctrl] key) in the work window. Then, to access the corresponding function,

select **Result Diagrams for Selected Members** on the **Results** menu,

use the toolbar button shown on the left or the context menu of the corresponding member or set of members.

A new window opens showing the result diagrams of the selected objects.

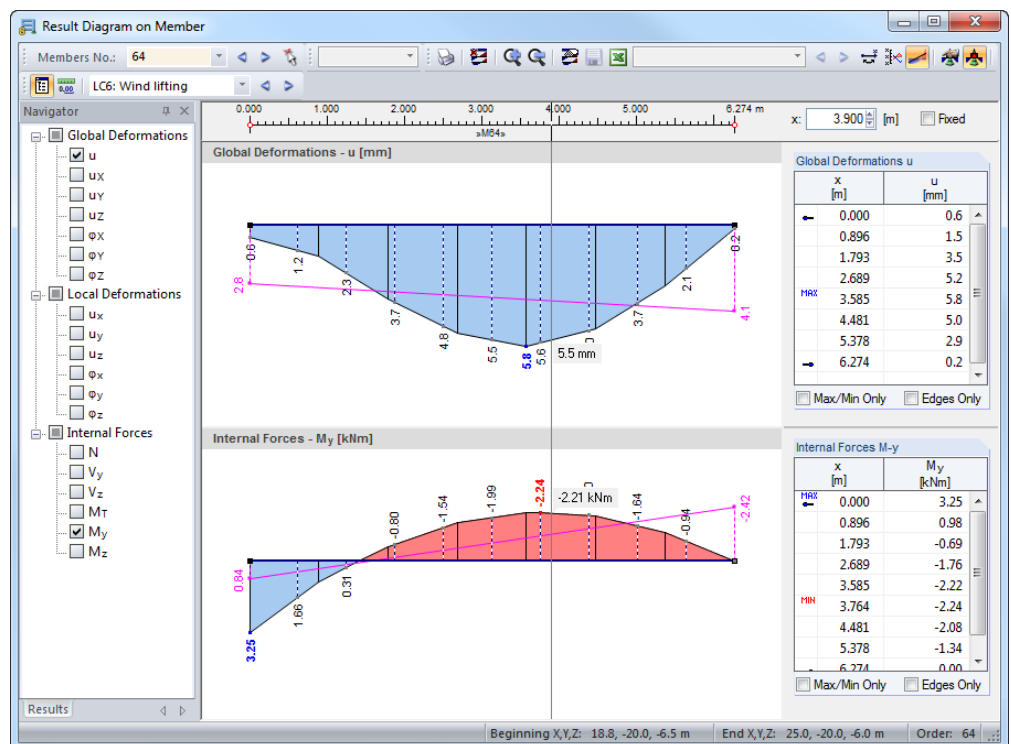


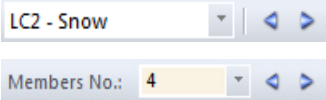
Figure 9.8: Dialog box *Result Diagram on Member*

The *Results* navigator on the left manages the deformations, internal forces and contact forces appearing in the result diagram. Use the list in the toolbar to choose a particular load case, load combination or result combination.

The numbers of the selected members or sets of members are listed in the upper left corner of the window. It is also possible to enter member numbers manually into the input field *Members No.* In this way, you can extend, reduce or completely reorganize the selection.

When you move the mouse along the member displayed in the result diagram, you can see the "moving" result values for the current x-location. The location x is related to the 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 member divisions according to specifications set in the dialog tab *Global Calculation Parameters* of the *Calculation Parameters* dialog box (see Figure 7.14, page 176).



The buttons in the toolbar *User operations*, in particular the smoothing options for internal forces (see chapter 9.5.2), help you to evaluate results for civil engineering purposes.



Figure 9.9: 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 9.10
	Saves smoothed result diagrams
	Opens dialog box <i>Export table</i> (see Figure 11.122, page 330)
	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 9.11, page 210)
	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.

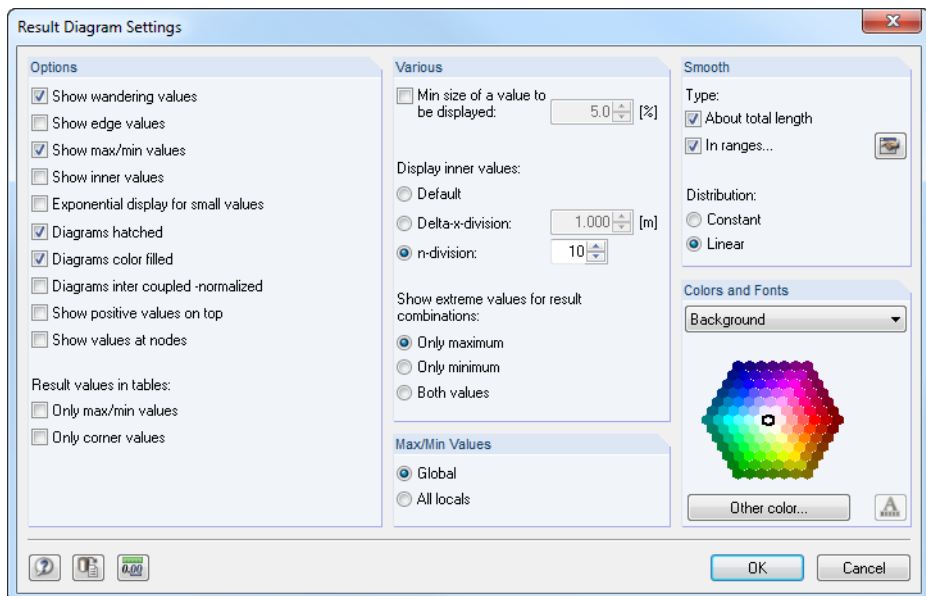


Figure 9.10: Dialog box *Result Diagram Settings*

### 9.5.2 Smoothing Results



In the *Result Diagram* dialog box, 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:

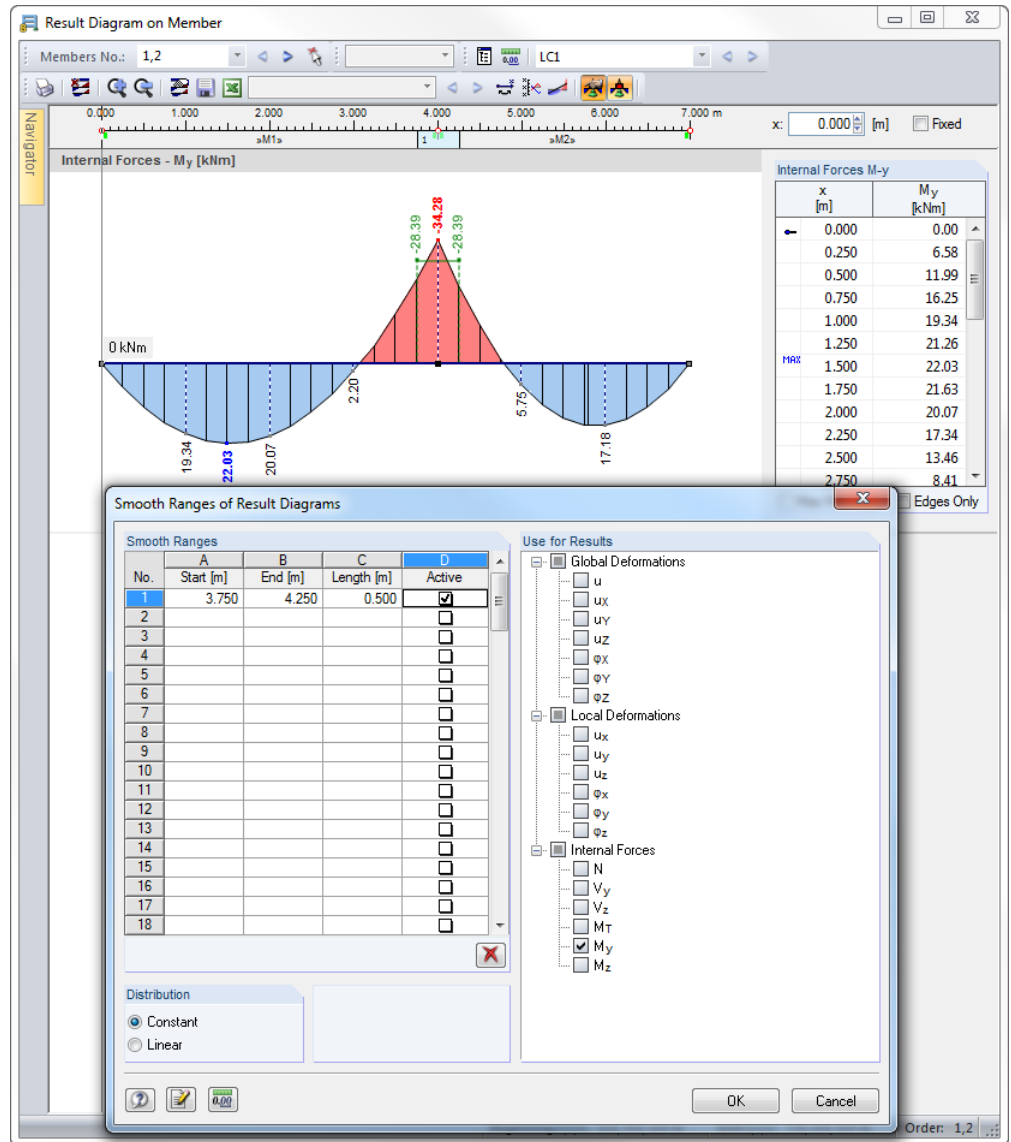


Figure 9.11: Dialog box *Smooth Ranges of Result Diagrams*

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.

## 9.6 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.

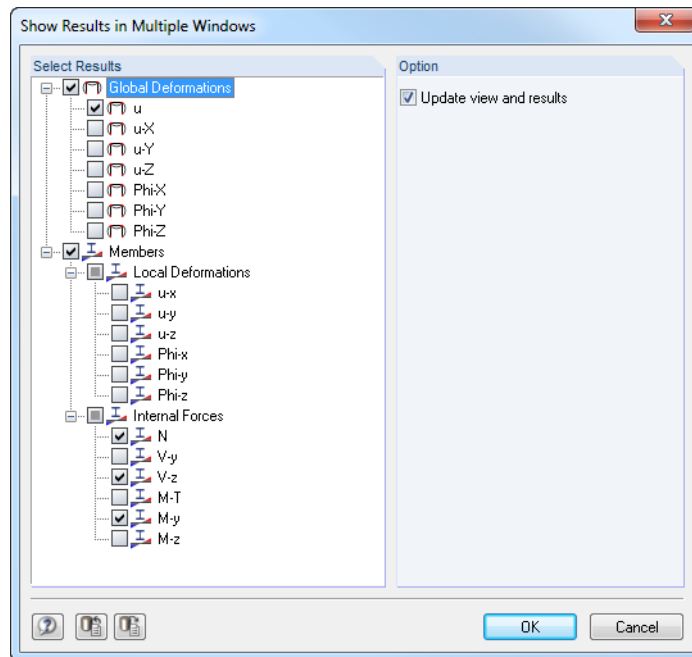


Figure 9.12: Dialog box *Show Results in Multiple Windows*

The multiple windows view can also be used for the printout (see chapter 10.2.1, page 250).

## 9.7 Filter Results

Various filter functions are available which are especially useful in case of complex structural systems for evaluating and documenting results.

### 9.7.1 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 for example to activate for the display only the members of a particular plane or members with a particular cross-section. 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 9.7.1.1) and by using **list buttons** or **menu functions** (chapter 9.7.1.2).

#### 9.7.1.1 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.

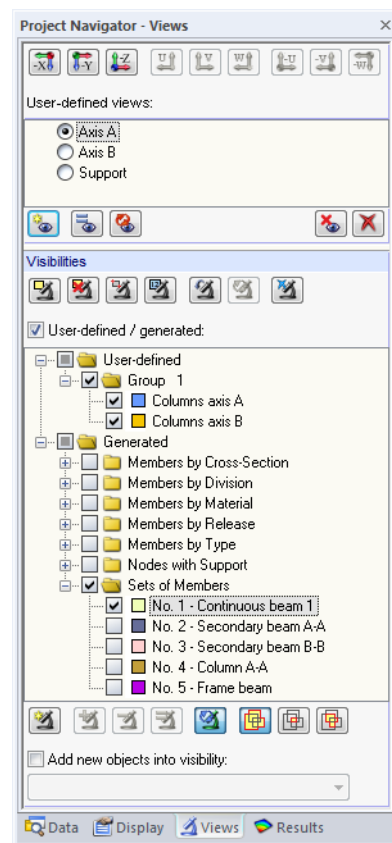


Figure 9.13: Tab Views of Project Navigator



### 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: RSTAB 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 set the following standard angles of view quickly:







	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 275)
	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 are reserved for the following functions:






	Creates a new <i>User-defined view</i> from current view (Figure 9.14)
	Redefines the active <i>User-defined view</i> by current view
	Restores the active <i>User-defined view</i> after modifications
	Deletes the entry selected in the list <i>User-defined views</i>
	Deletes all <i>User-defined views</i>

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.

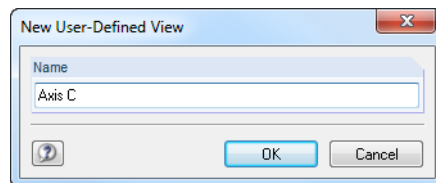


Figure 9.14: Dialog box *New User-Defined View*

### Visibilities

With the so-called "visibilities" it is possible to display partial views of the model or groups of objects such as members lying in one plane or columns of a particular story.

#### Visibility buttons

The buttons above the *Visibilities* list (see Figure 9.13, page 212) allow you to select the objects for representation by particular criteria. They are reserved for 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 216)
	Defines a new visibility by means of object numbers (see page 216)
	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 271) 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 9.18, page 217).

The buttons below the *Visibilities* list are reserved for the following functions:









	The dialog box <i>New User-Defined Visibility</i> appears (see Figure 9.18, page 217).
	Adds objects selected in work window to group marked in list above (see page 217)
	Removes objects selected in work window from group marked in list above (see page 217)
	Reassigns selected objects to group marked above
	Reverses current display (new visibility: hidden objects)
	Shows all objects activated in <i>Visibilities</i> list
	Shows only objects available in each active <i>Visibilities</i> entry
	Shows objects available in each active <i>Group</i>

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, members, supports 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 assigned automatically 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 269). 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**

RSTAB generates automatically visibilities for nodes and members according to particular criteria.

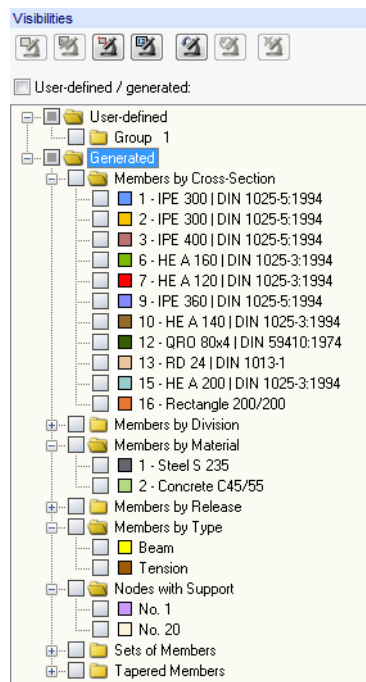


Figure 9.15: Generated visibilities in *Views* navigator

Those generated visibility types help you to have 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 RSTAB.



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.



### 9.7.1.2 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.

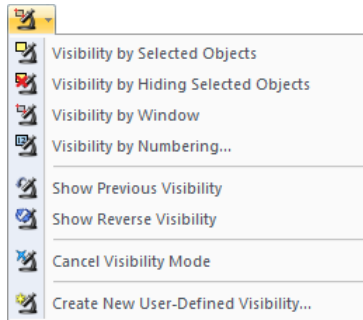


Figure 9.16: List buttons for *Visibility*

#### 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 partial view additionally includes objects that are cut by the window.

#### Visibility by numbering



Enter the numbers of *Nodes* or *Members* relevant for the partial view in a dialog box.

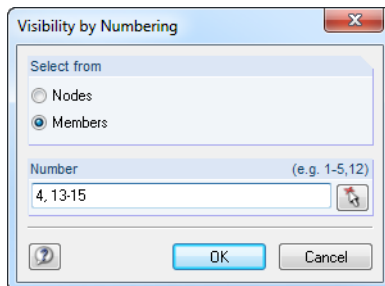


Figure 9.17: Dialog box *Visibility by Numbering*

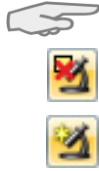
#### Cancel visibility mode



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 271 and chapter 11.2.2, page 274). 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.

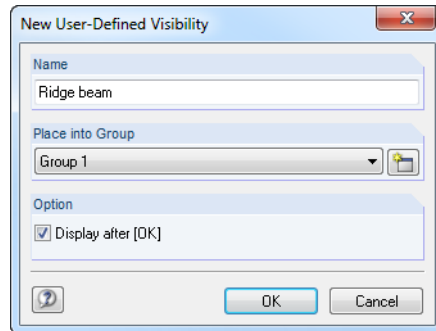


Figure 9.18: Dialog box *New User-Defined Visibility*

Define the *Name* and *Group*. If you want to use more visibility groups, click the [New] button to create another group.

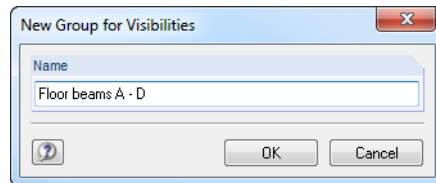


Figure 9.19: Dialog box *New Group for Visibilities*

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 9.13, page 212).

### 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. RSTAB 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 9.25, page 222).

The display of background objects can be turned on and off in the *Display* navigator.

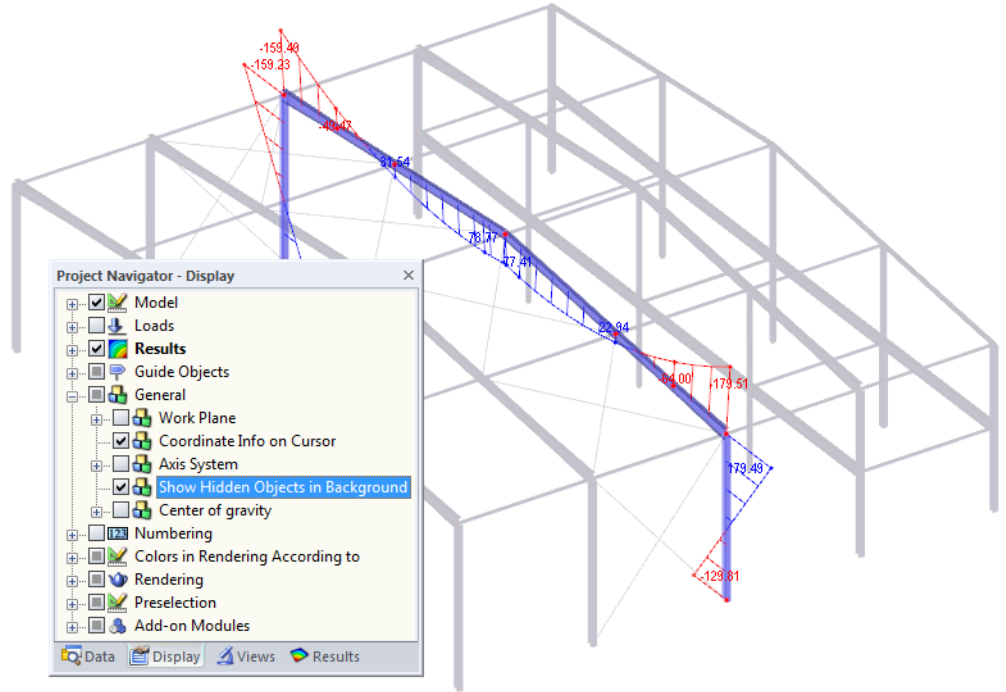


Figure 9.20: *Display* navigator: Option *General* → *Show Hidden Objects in Background*

### 9.7.2 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 to represent for example the results of a closed or curved model more clearly.

RSTAB 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.

To access the corresponding function, select **Clipping Plane** on the **Insert** menu.

The following dialog box appears:

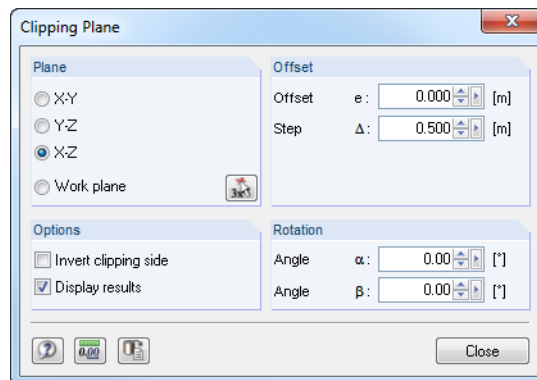



Figure 9.21: Dialog box *Clipping Plane*



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  $\alpha$  (about the last named axis of the plane) and angle  $\beta$  (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 are no options for storage and printing. Quit the function with the [Close] button.

The following example shows a clipping plane cutting through a container construction.

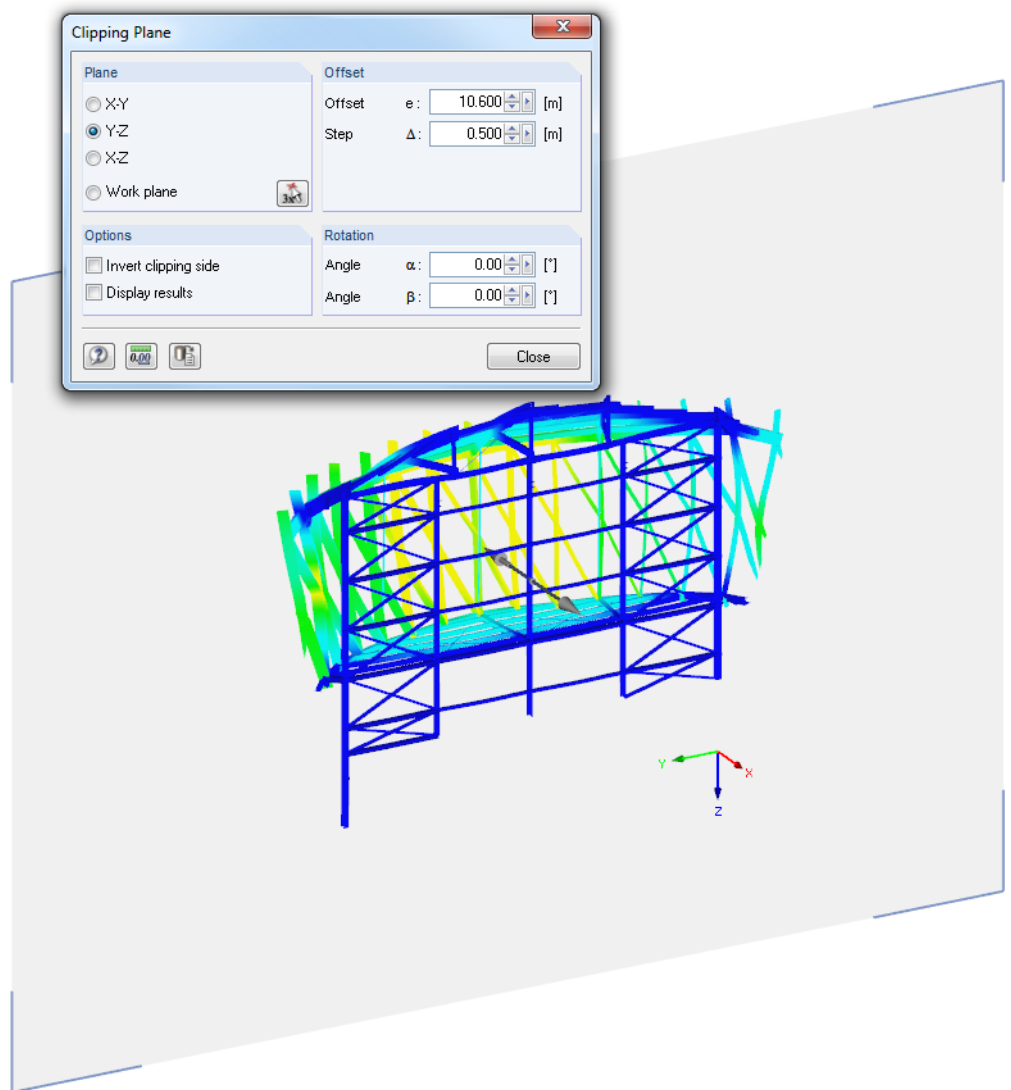


Figure 9.22: Clipping plane cutting through a container

### 9.7.3 Filter Functions

The grouping options described in chapter 9.7.1 *Views* refer to the objects of the model. Additionally, you can use internal forces and deformations as filter criteria.

#### Filtering results

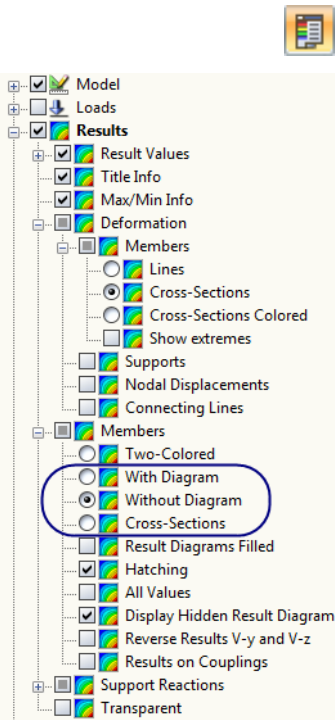
Results are filtered by means of the control panel. If the panel is not displayed, select **Control Panel (Color Scale, Factors, Filter)** on the **View** menu or use the toolbar button shown on the left.

The control panel is described in chapter 3.4.6 on page 27.

The filter settings for results must be defined in the panel tab *Color Spectrum*. 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).

You can set specific displays in the panel, for example compressive forces displayed only if they exceed a particular value, or bending moments within the range of  $\pm 30$  kNm using a fine gradation (see Figure 3.19, page 29).

In the following example of a hall, only compression forces between -160 kN and -320 kN are displayed on the model.



Settings in *Display* navigator for multi-color member results

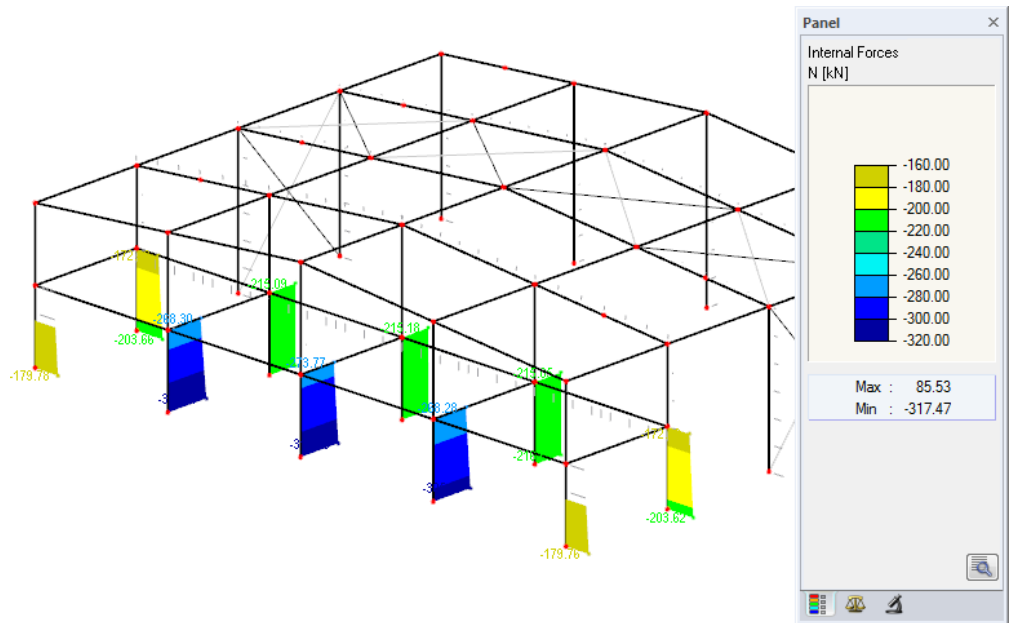


Figure 9.23: Filtering axial forces with adjusted color spectrum

The color spectrum has been modified in such a way that a color range covers exactly 20 kN. RSTAB does not show any results for members whose internal forces are beyond the defined range of values.

To represent the hidden diagrams of internal forces with dotted lines, go to the *Display* navigator and select *Results* → *Members* → *Display Hidden Result Diagram*.



### Filtering objects



In the *Filter* tab of the control panel, you can enter the numbers of selected members to show their result diagrams in a filtered display. The function is described in chapter 3.4.6 on page 30.

In contrast to the visibility function, the model is displayed completely in the graphic.

The following figure shows the bending moments available in the horizontal beams of a timber hall. The columns are shown in the model but displayed without internal forces.

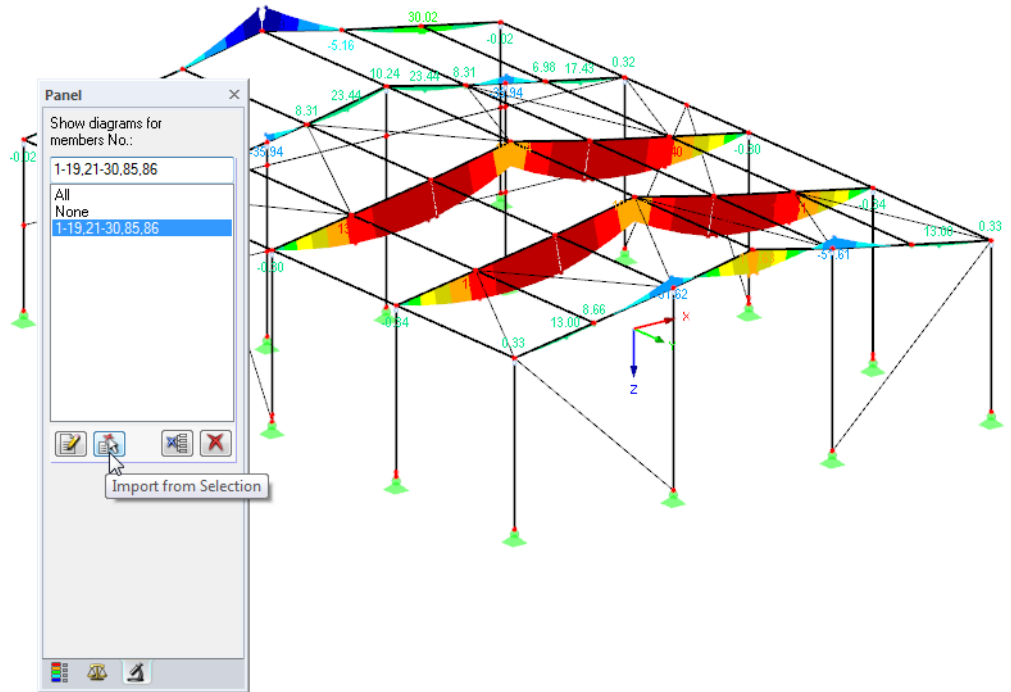


Figure 9.24: Filtering members: moments of horizontal beams



The filter settings of the panel also affect the objects in the results tables: When you restrict the results display in the panel for example to two members, table 4.1 *Members - Internal Forces* will also list only the results of those two members.

## 9.8 Animation of Deformations



Normally, deformations of objects are displayed in their final state.

But it is also possible to show the deformation process as a motion. To open the corresponding dialog box,



select **Animation** on the **Results** menu

or use the toolbar button shown on the left. 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*.

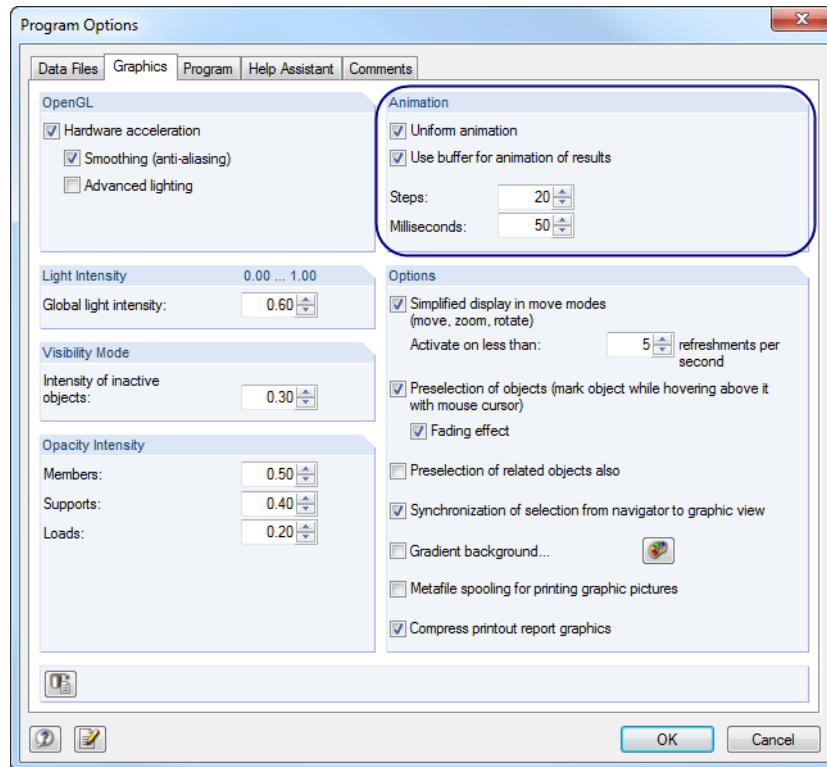


Figure 9.25: Dialog box *Program Options*, tab *Graphics*



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.

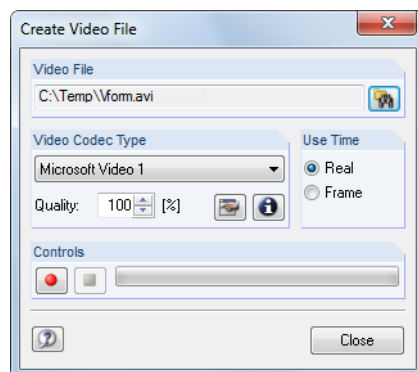


Figure 9.26: Dialog box *Create Video File*



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 RSTAB 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 member and cross-section results. In this way, you can reduce time of waiting.

It is also possible to create different printout reports in one RSTAB 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.

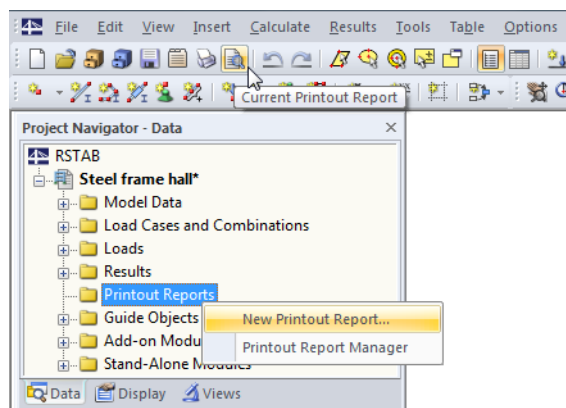


Figure 10.1: Button and context menu of *Printout Report*

The following dialog box appears:

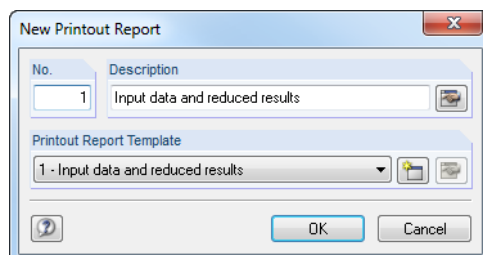


Figure 10.2: Dialog box *New Printout Report*

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 240).

The buttons in the dialog box are reserved for the following functions:



	Creates a new report template
	Edits selection of report (→ chapter 10.1.3, page 227).

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.

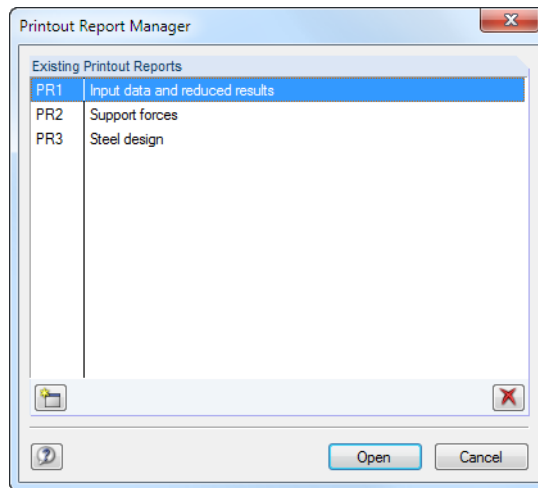


Figure 10.3: Dialog box *Printout Report Manager*

You can select the relevant report from the list.

The buttons in the dialog box are reserved for 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

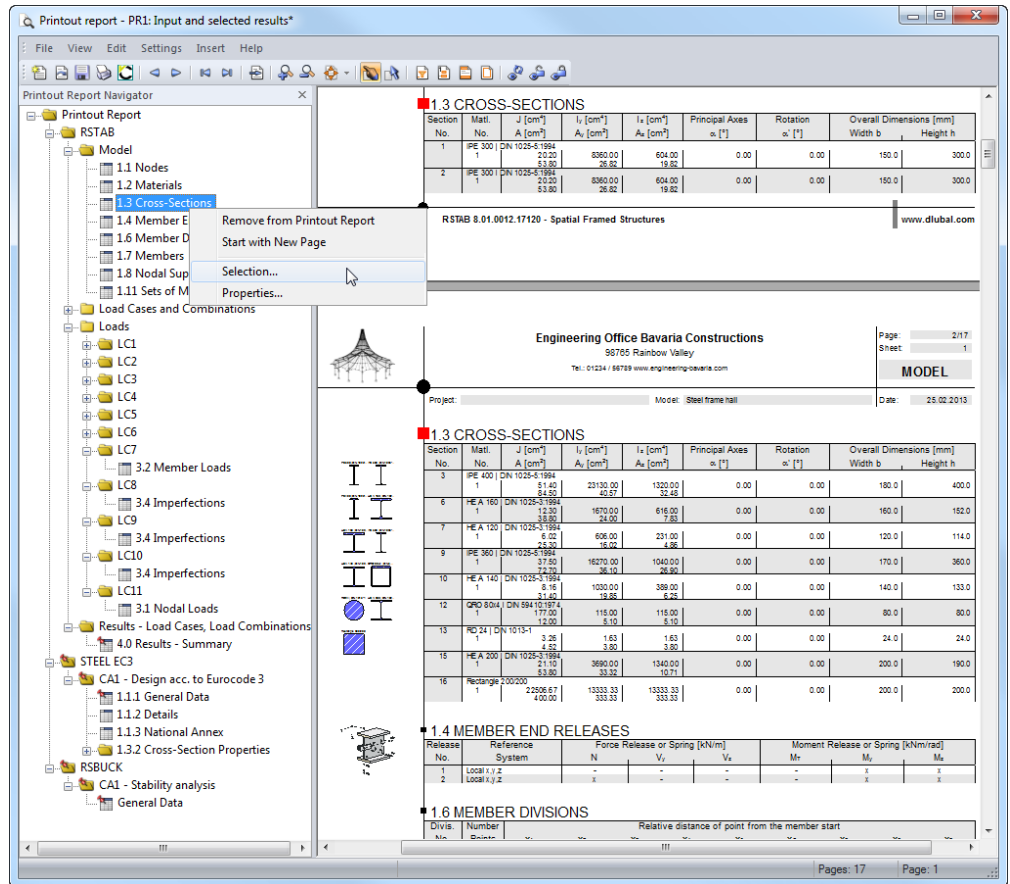


Figure 10.4: Printout report with context menu

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 more options for adjusting the printout report. As common for Windows applications, multiple selections are possible by using the [Ctrl] and [⇧] keys.

#### 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 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.



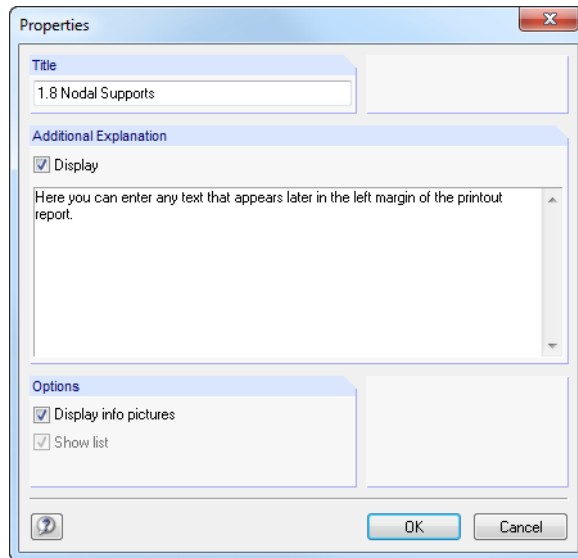


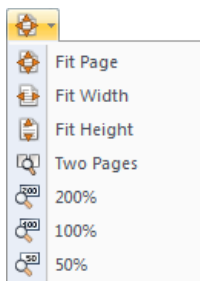
Figure 10.5: Dialog box *Properties*

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 <i>Zoom</i> 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 open the corresponding dialog box,



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.

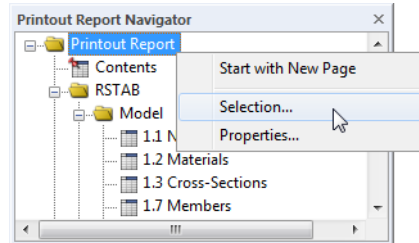


Figure 10.6: Open the global selection via the *Printout Report* context menu

The following dialog box appears:

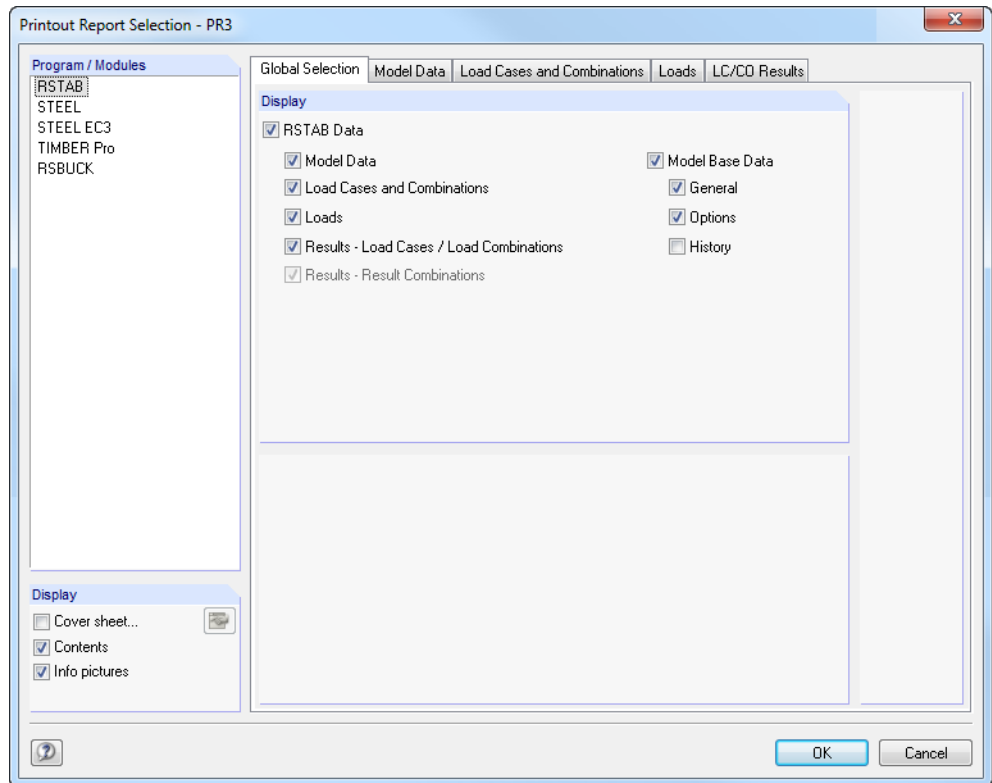


Figure 10.7: Dialog box *Printout Report Selection*, tab *Global Selection*

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 a check box, the corresponding detail tab disappears.

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

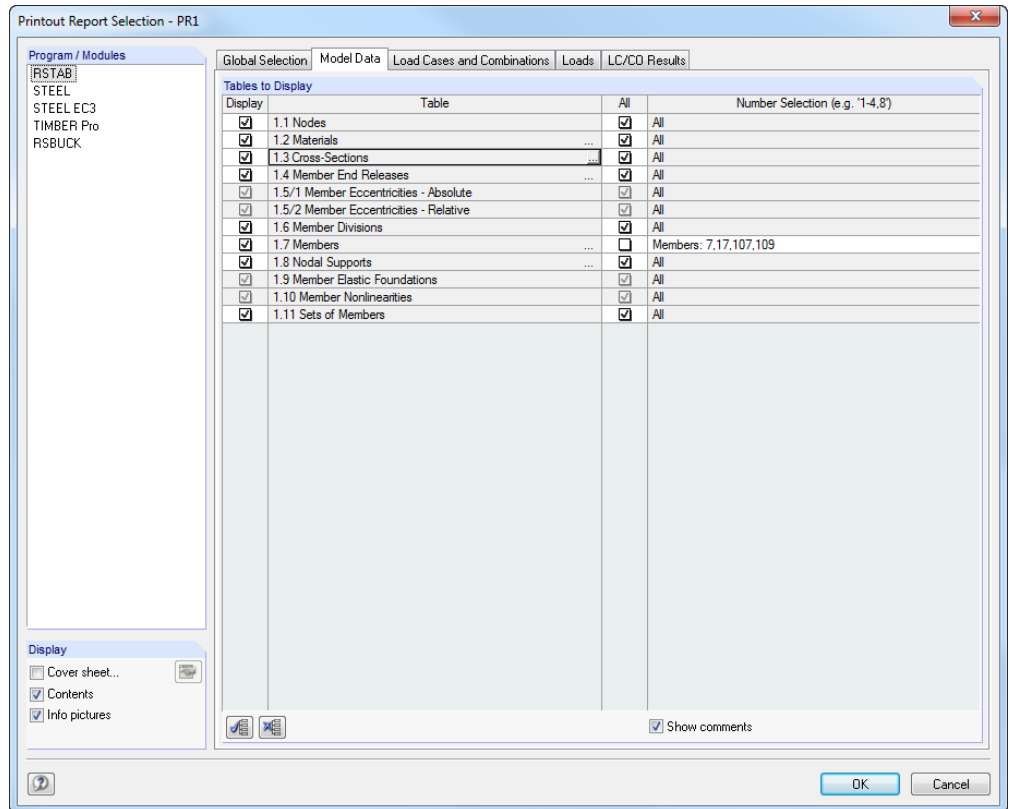


Figure 10.8: Dialog box *Printout Report Selection*, tab *Model Data*

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 *1.3 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.

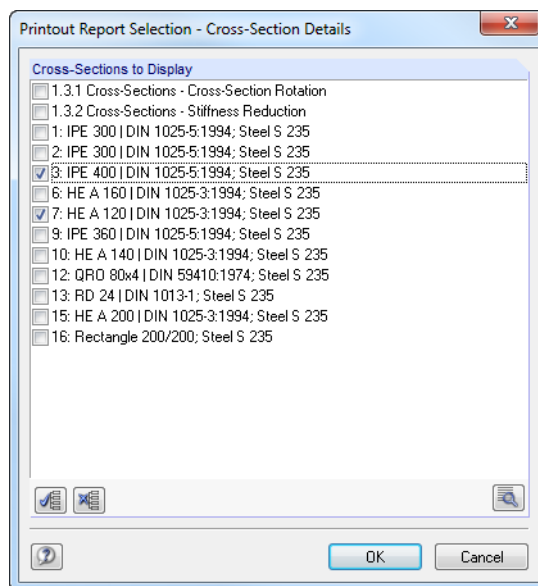


Figure 10.9: Dialog box *Printout Report Selection - Cross-Section Details*



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, members and sets of members graphically in the work window. For the remaining objects a list with table rows appears.

### 10.1.3.2 Selecting Load Case and Combination Data

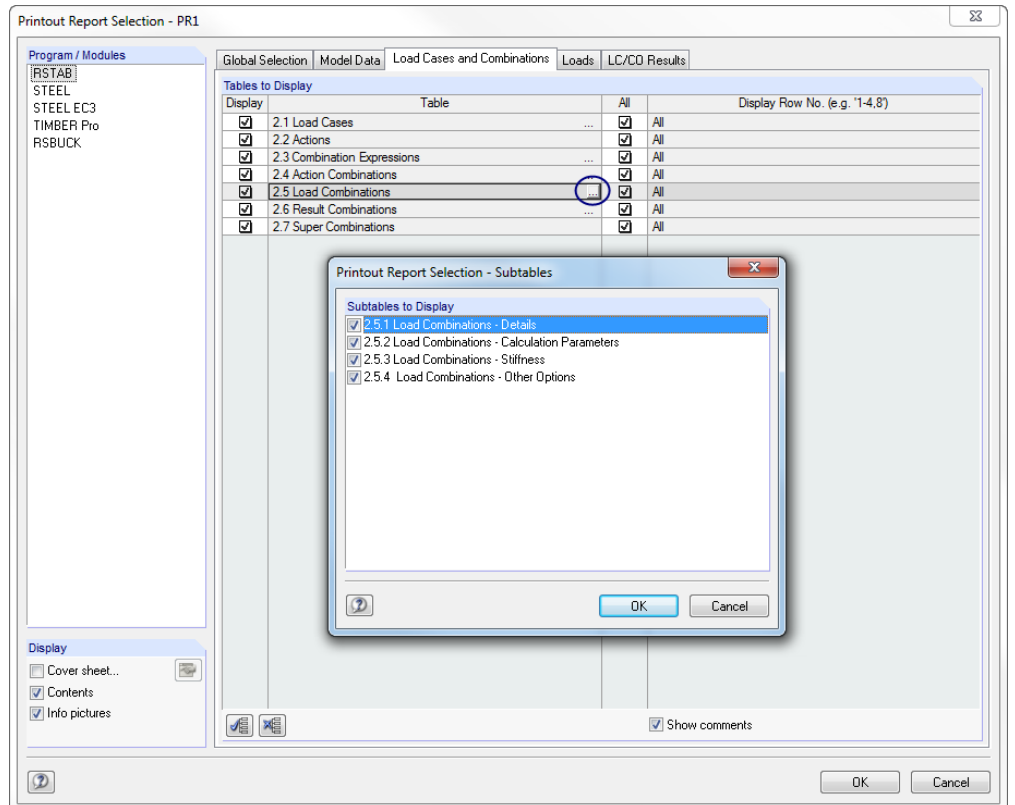


Figure 10.10: Dialog box *Printout Report Selection*, tab *Load Cases and Combinations*



With this dialog tab you decide which information about load cases and combinations appear in the printout report. Data is selected as described in the previous chapter. Use the [...] buttons to access the individual subtables.

### 10.1.3.3 Selecting Load Data

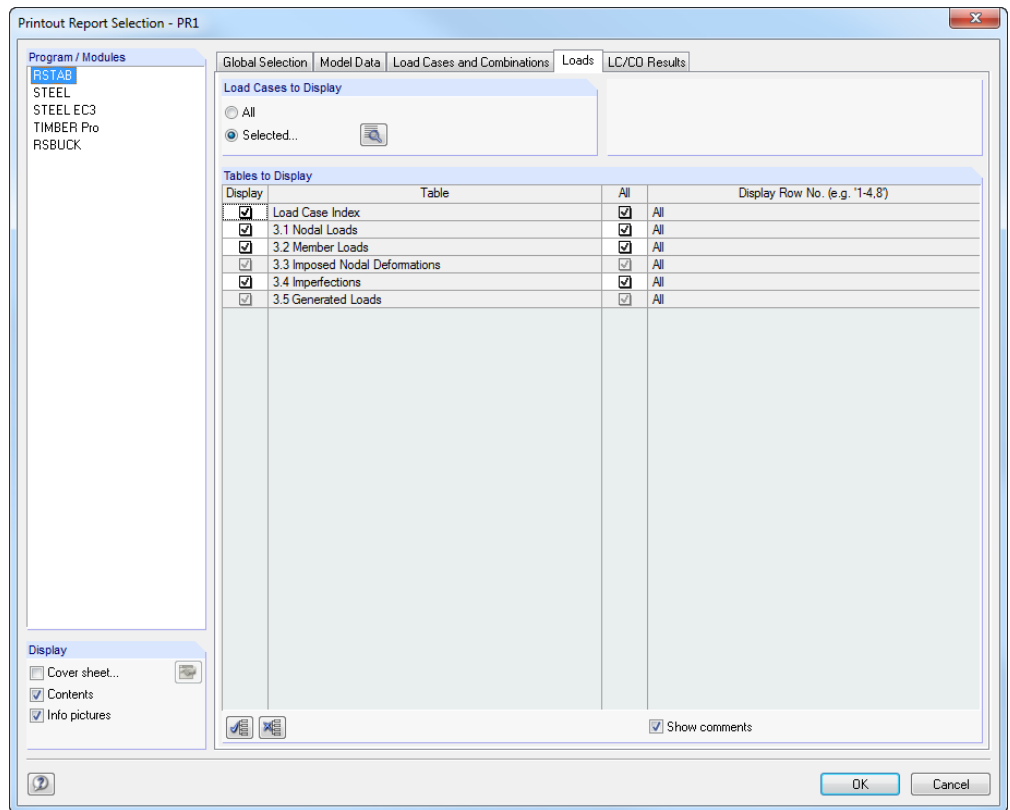


Figure 10.11: Dialog box *Printout Report Selection*, tab *Loads*

Tables are selected as described in 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.

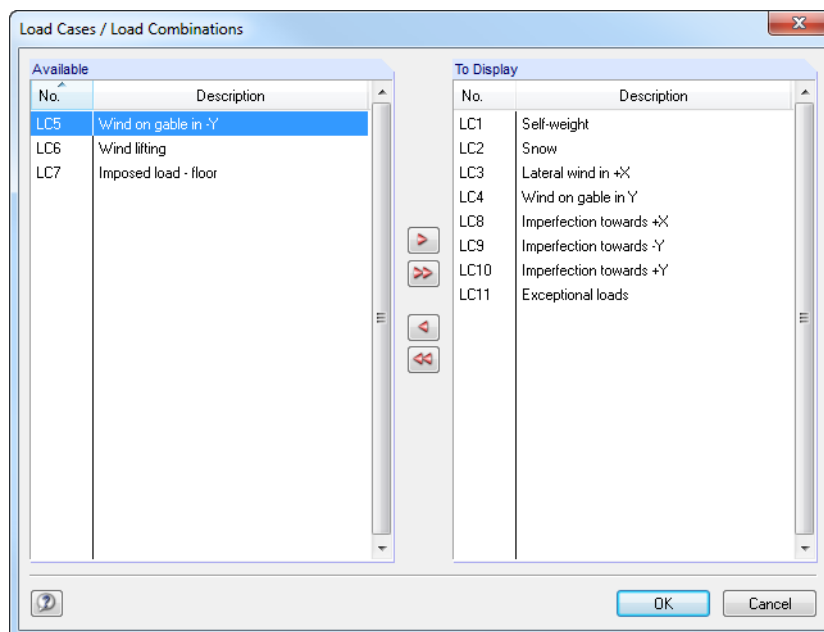


Figure 10.12: Selection of load cases

### 10.1.3.4 Selecting Results Data

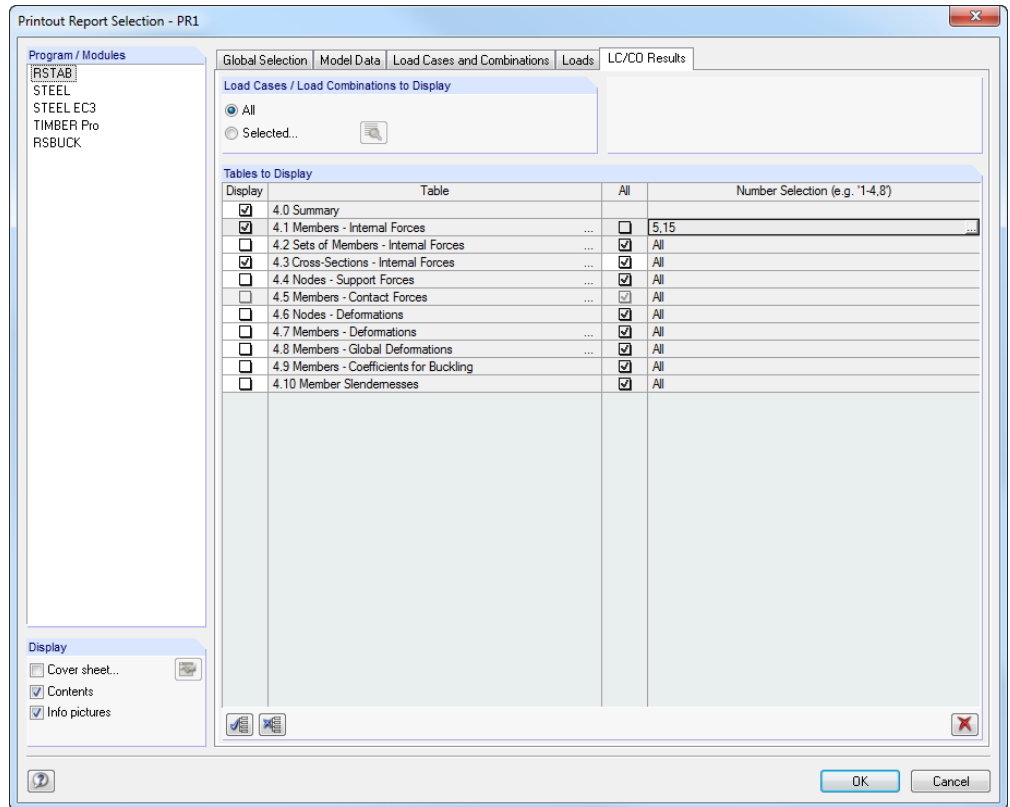


Figure 10.13: Dialog box *Printout Report Selection*, tab *LC/CO Results*

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 as well as the results of super combinations.



Results data can be prepared like load data (see previous chapter 10.1.3.3): 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.

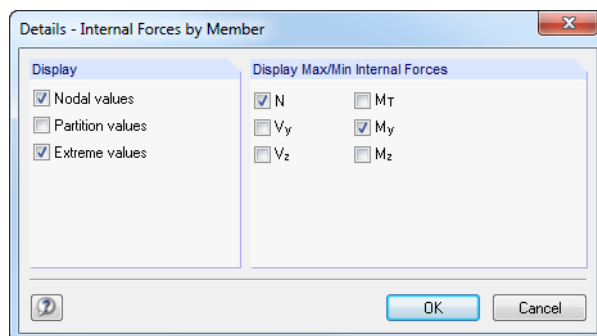


Figure 10.14: Dialog box *Details - Internal Forces by Member*

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 4.6, page 72)
- Extreme values (*Max/Min*) of results (see chapter 8.1, page 188)

The selection is connected with the *Table Filter* settings (see Figure 11.118, page 328).



You can considerably reduce the printout extent by restricting output data to results that are indispensable for your documentation.

### 10.1.3.5 Selecting Data of Add-on Modules

The data of all add-on modules is also managed for printing in the RSTAB printout report. You can summarize data of modules together with the RSTAB data in a single report, or you organize it in separate reports. For complex structural systems with many design cases, it is recommended to split data into several printout reports which makes data arrangement clearer.

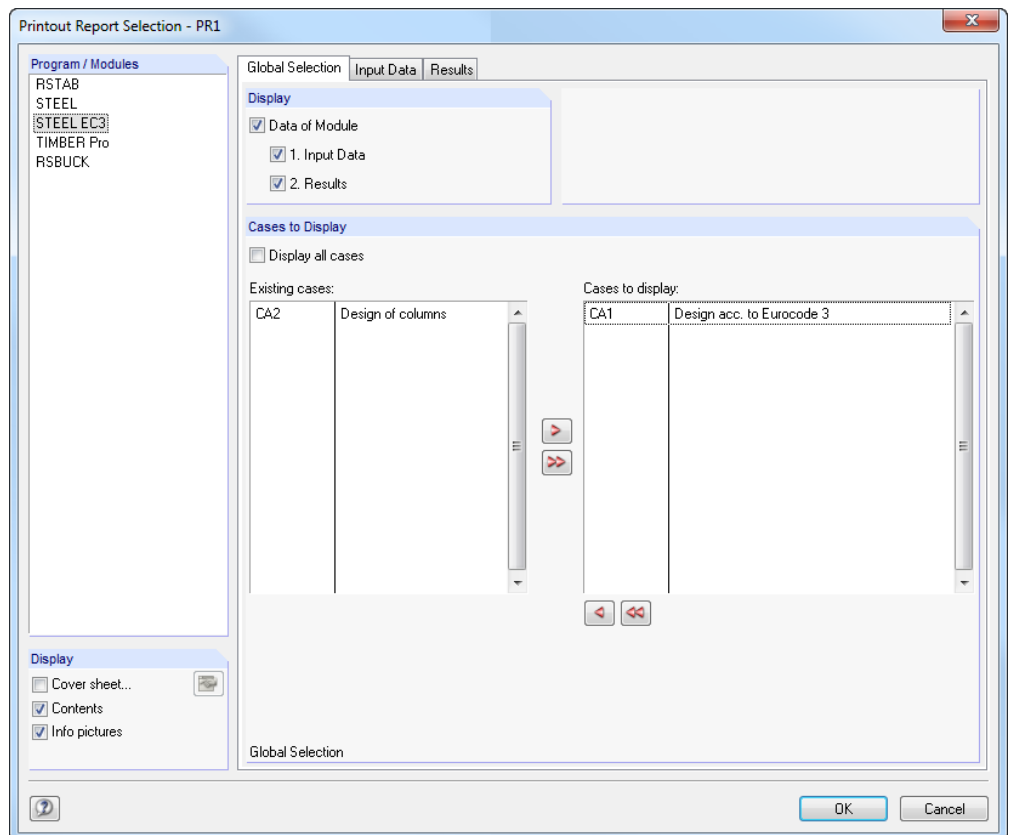


Figure 10.15: Dialog box *Printout Report Selection*, tab *Global Selection* of add-on module **STEEL EC3**

In addition to RSTAB, the list in the dialog section *Program / Modules* contains all add-on 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 10.1.3.4 *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.

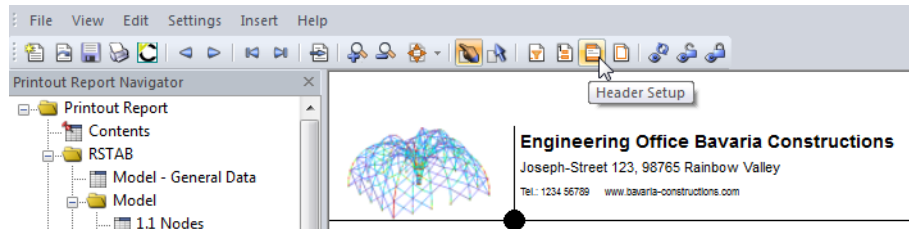


Figure 10.16: Button *Header Setup*

The following dialog box appears:

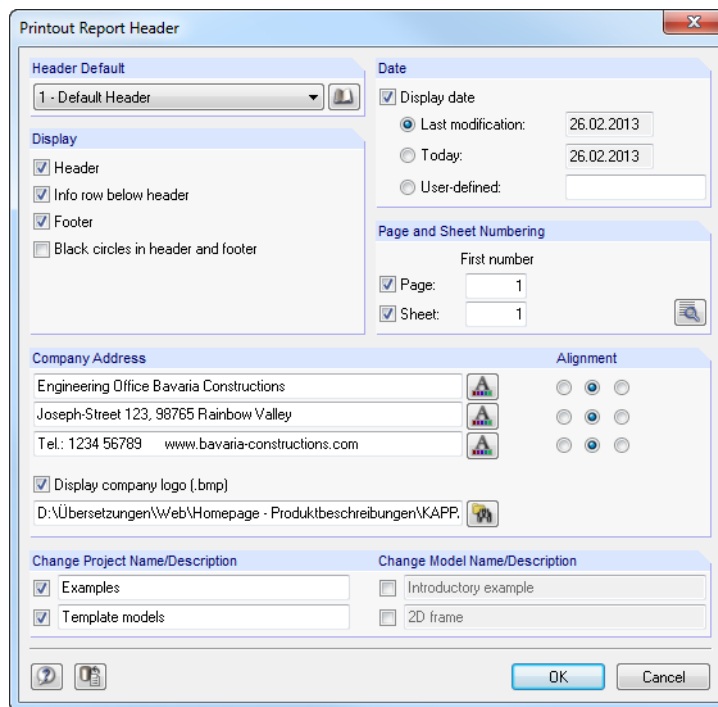


Figure 10.17: Dialog box *Printout Report Header*

### 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.

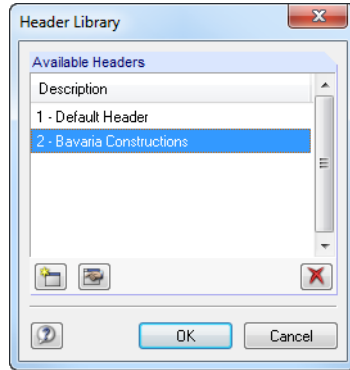


Figure 10.18: Dialog box *Header Library*

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 <i>Header</i> dialog box (see Figure 10.17).
	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 **DlubalProtocolConfig.cfg** that you find in the general master data folder *C:\ProgramData\Dlubal\Stammdat*. 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 378). The model description is taken from the base data of the model (see chapter 12.2, page 386). The default settings can be adjusted for the printout in the dialog sections *ChangeProjectName/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

RSTAB 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.



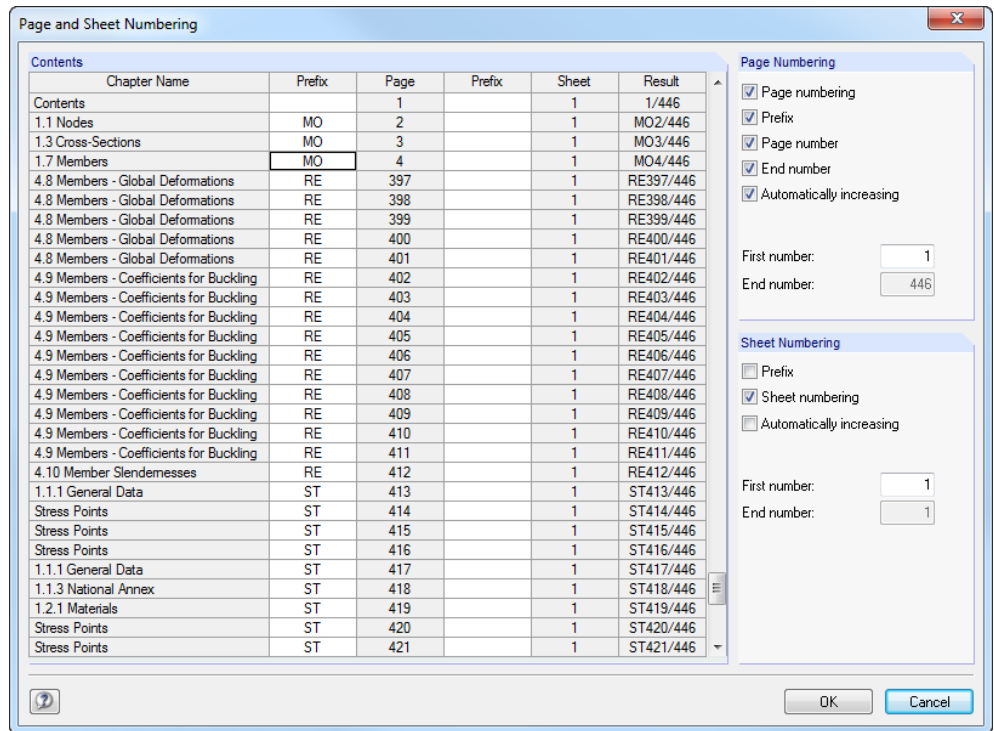


Figure 10.19: Dialog box *Page and Sheet Numbering*

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: MO2/446".

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.

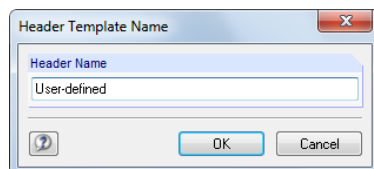


Figure 10.20: Dialog box *Header Template Name*

### Project name/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 RSTAB Graphics



Every picture displayed in the work window can be integrated into the printout report. Furthermore, it is possible to include result diagrams of members and sets of members 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.

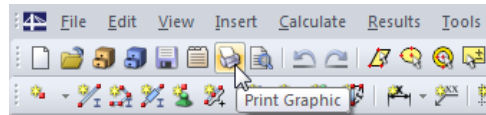


Figure 10.21: Button *Print Graphic* in the toolbar of the work window

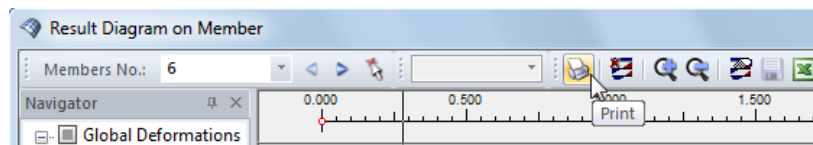


Figure 10.22: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog box appears:

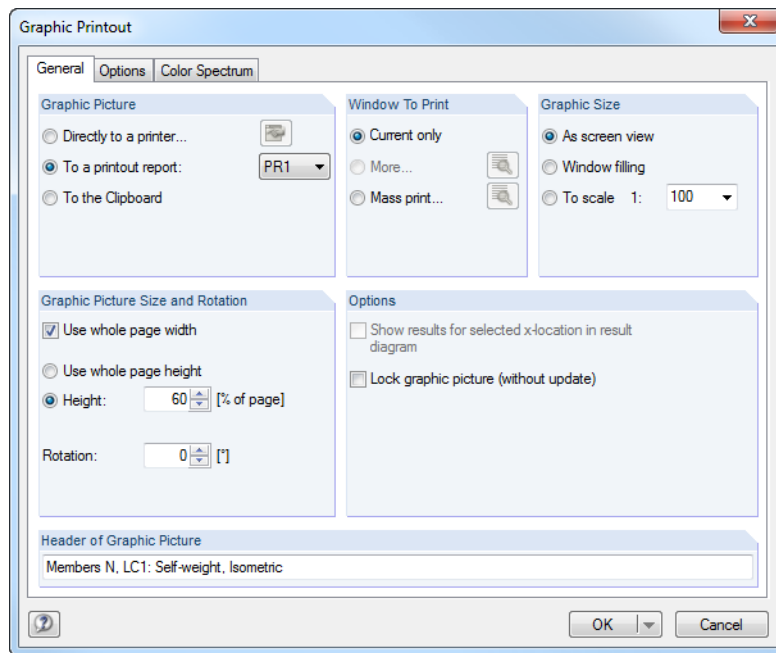


Figure 10.23: Dialog box *Graphic Printout*, tab *General*

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

Standard of RSTAB 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 225). 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 225). They are reserved for 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

### OK

The [OK] button of the dialog box is a list button.

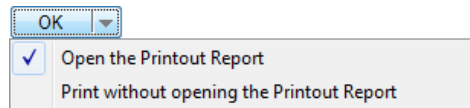


Figure 10.24: List button [OK]

With the default setting *Open the Printout Report* RSTAB opens the printout report and shows you the integrated graphic after clicking [OK]. This may be annoying, for example when you want to take several graphics one after the other to the printout report. The option *Print without opening Printout Report* allows you to print pictures without waiting time when creating the printout report.



The remaining functions and tabs of the dialog box *Graphic Printout* are explained in chapter 10.2 on page 248.

## 10.1.6 Insert Graphics and Texts

External graphics and texts can be integrated as well into the printout report of RSTAB.

### Graphics

To insert a picture that is not an RSTAB 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.

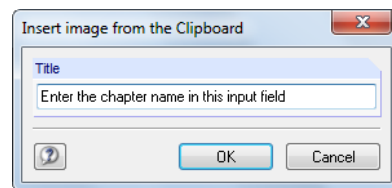


Figure 10.25: Dialog box *Insert image from the Clipboard*

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.

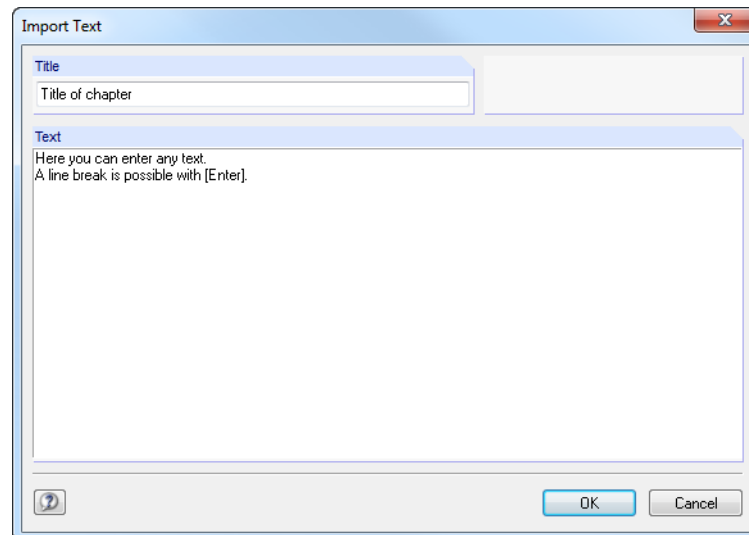


Figure 10.26: Dialog box *Import Text*

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 226) 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 226) you can modify the text subsequently by right-click. The dialog box *Import Text* appears for user-defined adjustments.

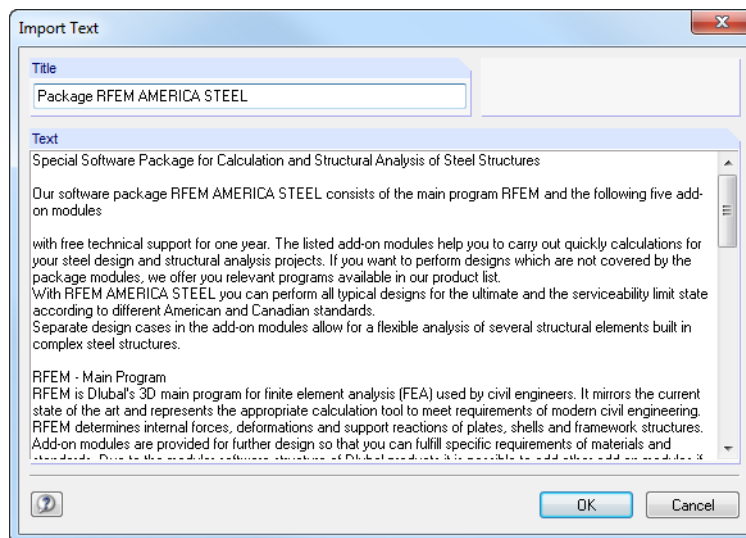


Figure 10.27: Dialog box *Import Text*

### 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 227 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.

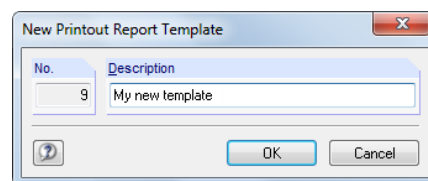


Figure 10.28: Dialog box *New Printout Report Template*

#### 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 10.28).

#### 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,

point to **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*.

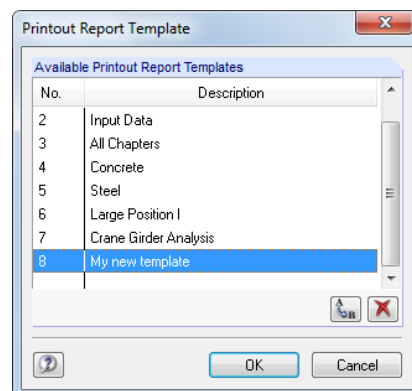


Figure 10.29: Dialog box *Printout Report Template*

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.

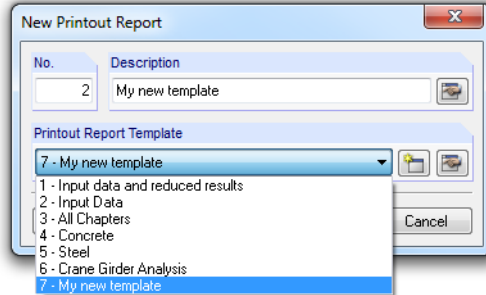


Figure 10.30: Dialog box *New Printout Report* with list of templates

### 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 10.29 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 **RSTABProtocolConfig.cfg** that can be found in the master data folder for RSTAB 8 C:\ProgramData\Dlubal\RSTAB 8.01\General Data. The 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, margin settings and table design.



To open the dialog box where you can edit the page layout, select **Cover** on the **Settings** menu in the printout report or use the toolbar button in the printout report shown on the left.

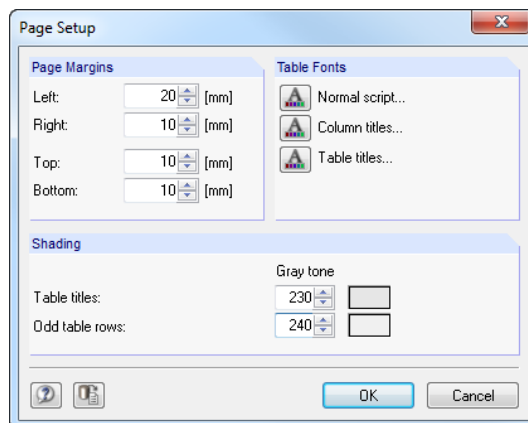


Figure 10.31: Dialog box *Page Setup*



The default fonts for table contents and table headers are relatively small. However, you should be careful with changing the default settings **Arial 5, 6 or 8**: Larger fonts do not always fit in the columns.



The layout settings also apply to the printout reports of the RSTAB 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.

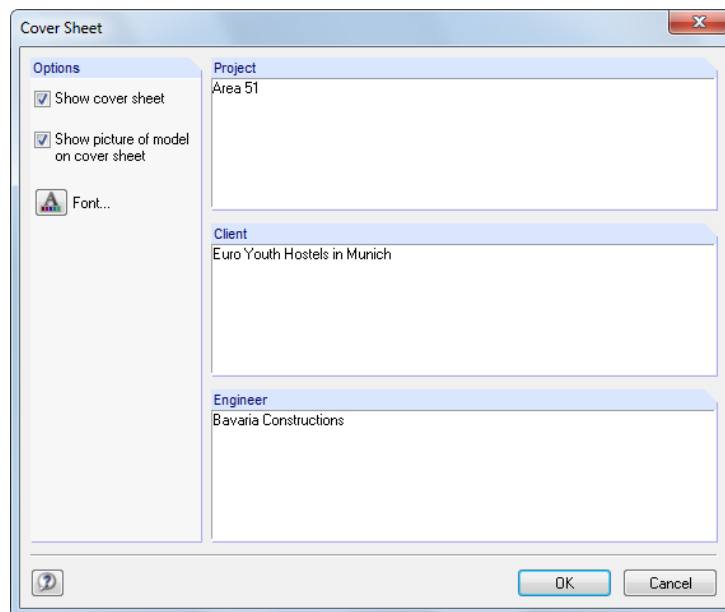


Figure 10.32: Dialog box *Cover Sheet*

When the input is complete, click [OK] to create the cover in the report.

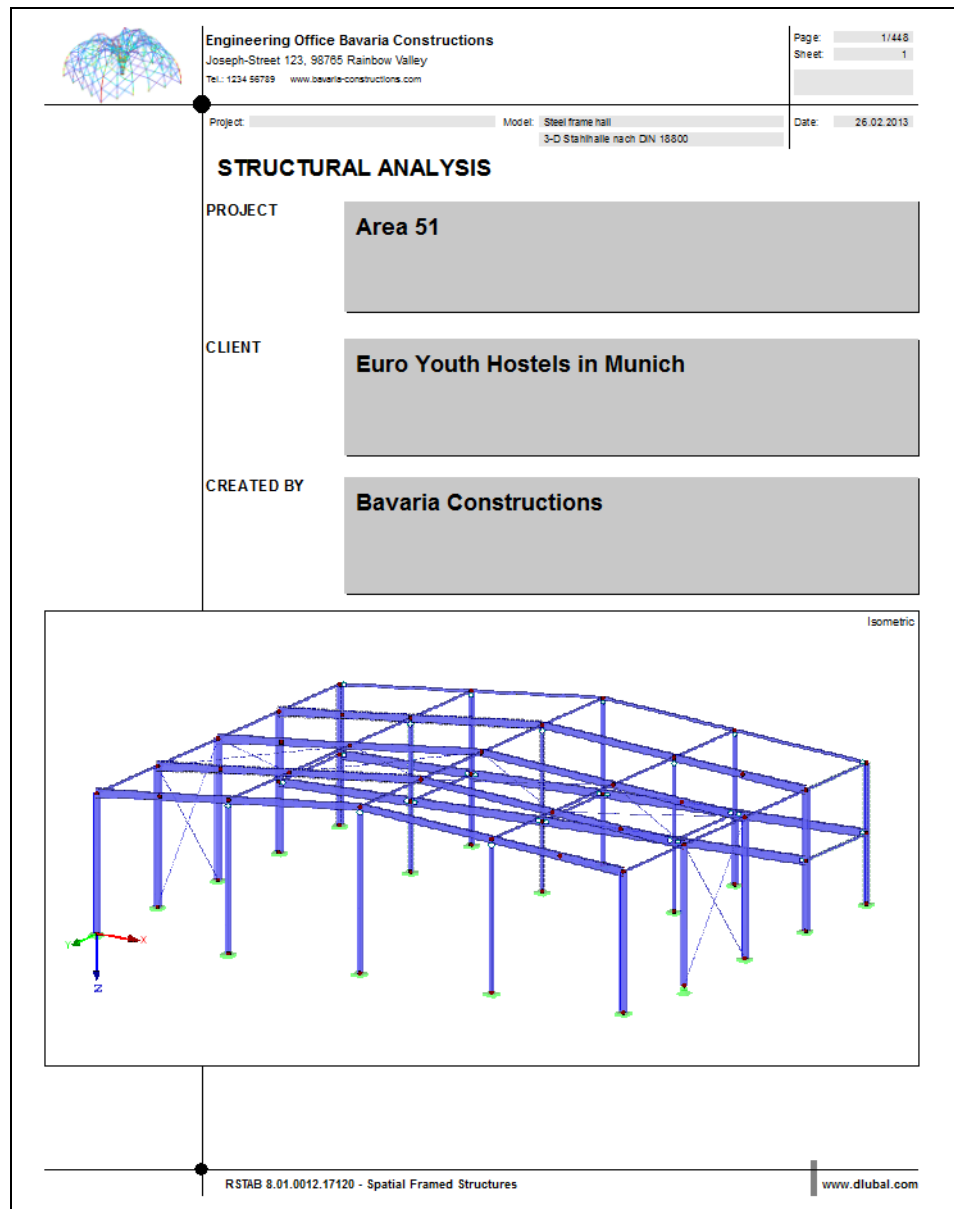


Figure 10.33: Cover sheet in the printout report



The contents of the cover sheet can be modified once again by a double-click in the selection mode (see Table 10.3, page 226). 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  
 or use the button in the printout report toolbar shown on the left.

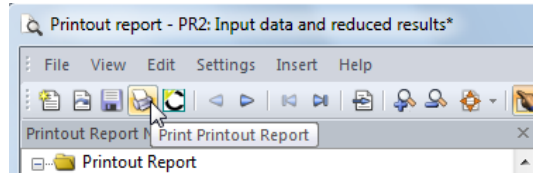


Figure 10.34: Button *Print Printout Report*

The dialog box for the printer set by default in Windows opens. Select the printer and determine the pages that you want to print.

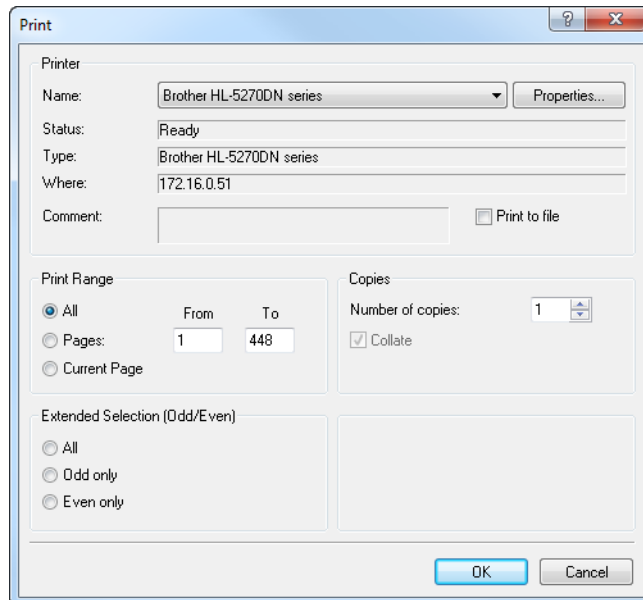


Figure 10.35: Dialog box *Print*

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 RSTAB.

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*.

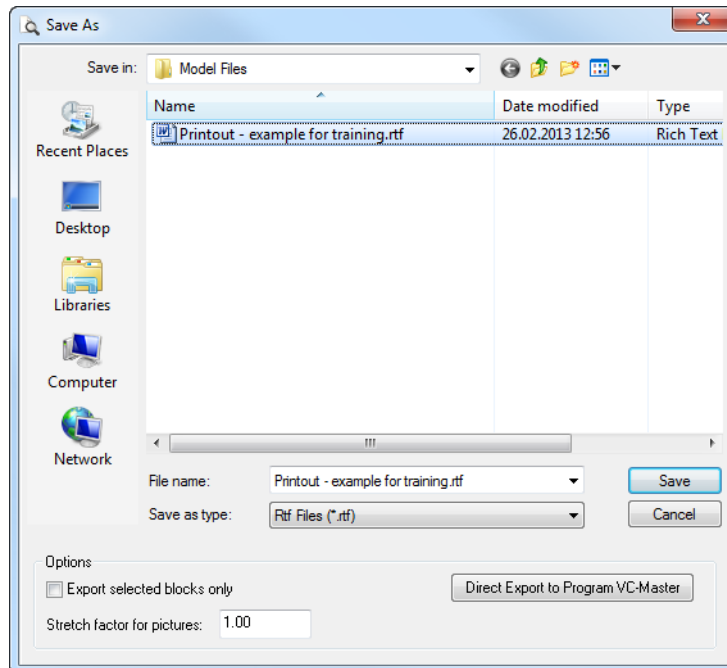
#### RTF export

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 10.36: 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 RSTAB 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 10.36) 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 VC-Master] in the printout report toolbar shown on the left.



The dialog box shown in Figure 10.36 appears where you have to tick the check box for *Direct Export to Program VC-Master*.

It is not necessary to enter a file name, but *VC-Master* should run in the background. To start the import module of *VC-Master*, click [OK].

### 10.1.12 Language Settings

The language in the printout report can be set independently of the language used in the graphical user interface of RSTAB. Thus, you can create for example a German or Italian printout report though 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.

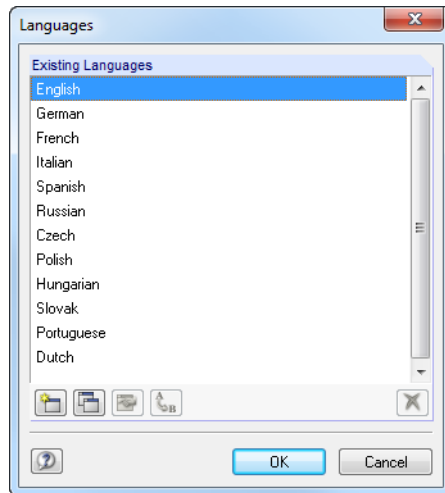


Figure 10.37: Dialog box *Languages*

#### 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 10.37), 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.

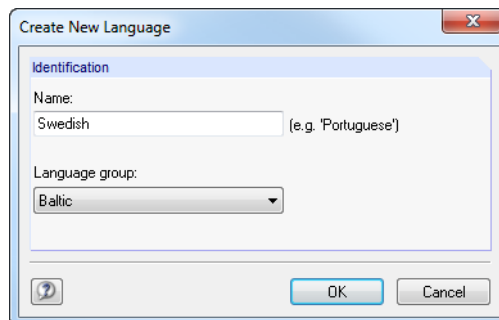


Figure 10.38: Dialog box *Create New Language*

Click [OK] to confirm the dialog box. The new language is now available in the list *Existing Languages*.

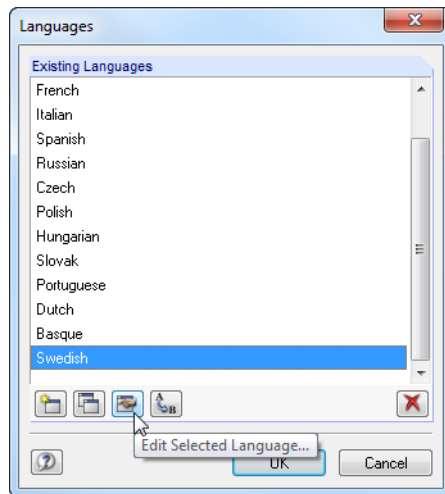


Figure 10.39: Dialog box *Languages*, button *Edit Selected Language*

Use the [Edit] button to enter the strings of the new language.

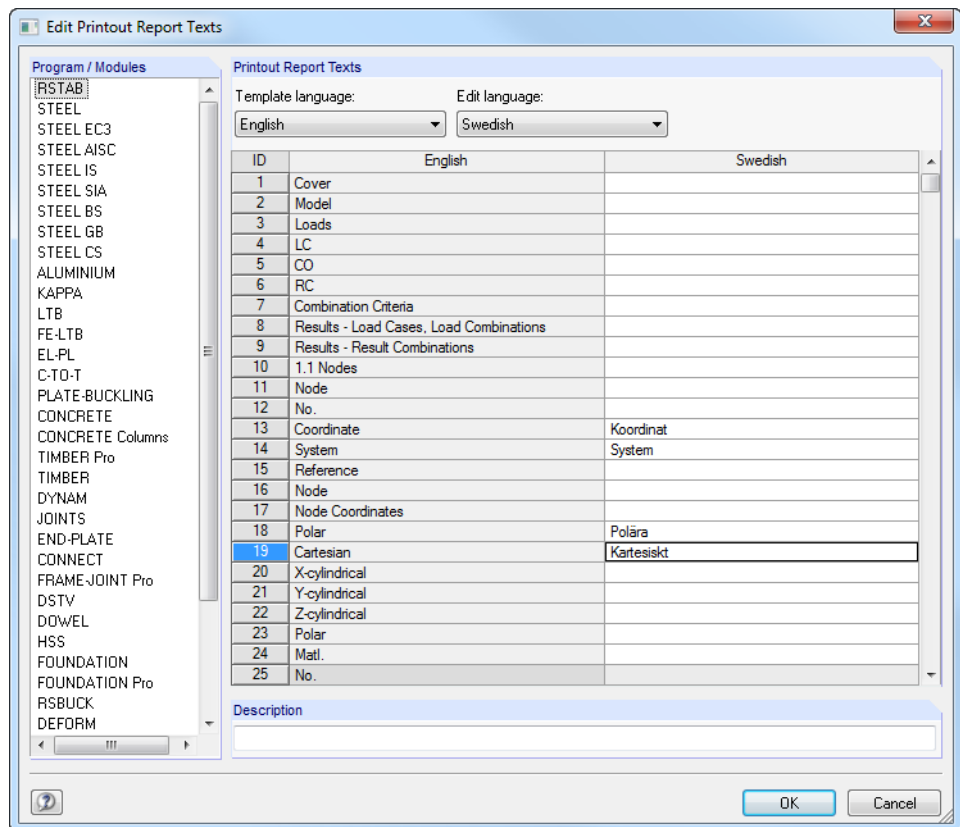


Figure 10.40: Dialog box *Edit Printout Report Texts*

Only user-defined languages can be edited.



Copy a language

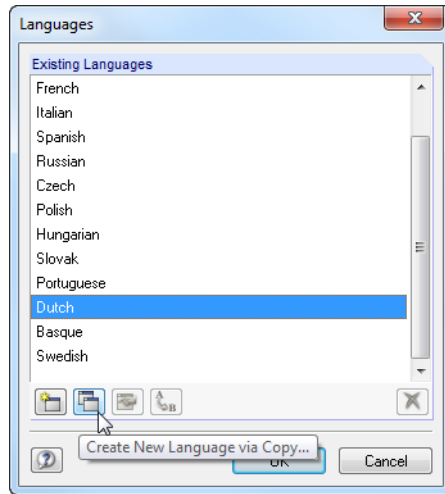


Figure 10.41: Dialog box *Languages*, button *Create New Language via Copy*

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 10.40, column *Swedish*) 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

Each graphic of the work window can be printed immediately without embedding in the print-out report (see chapter 10.1.5, page 236). Result diagrams of members and sets of members 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 respective button in the toolbar.

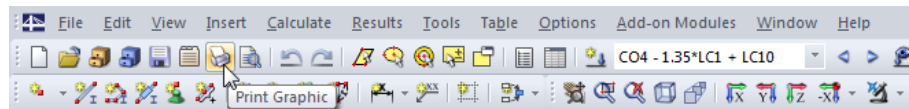


Figure 10.42: Button *Print Graphic* in the toolbar of the main window

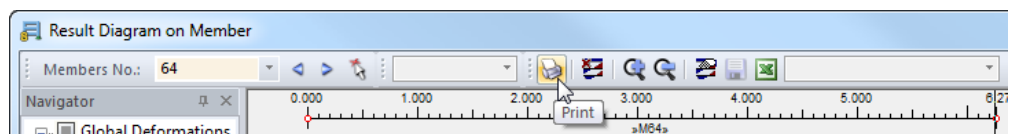


Figure 10.43: Button *Print* in the toolbar of the *Result Diagram* window

A dialog box with several tabs appears which are described in the following chapters.

## 10.2.1 General

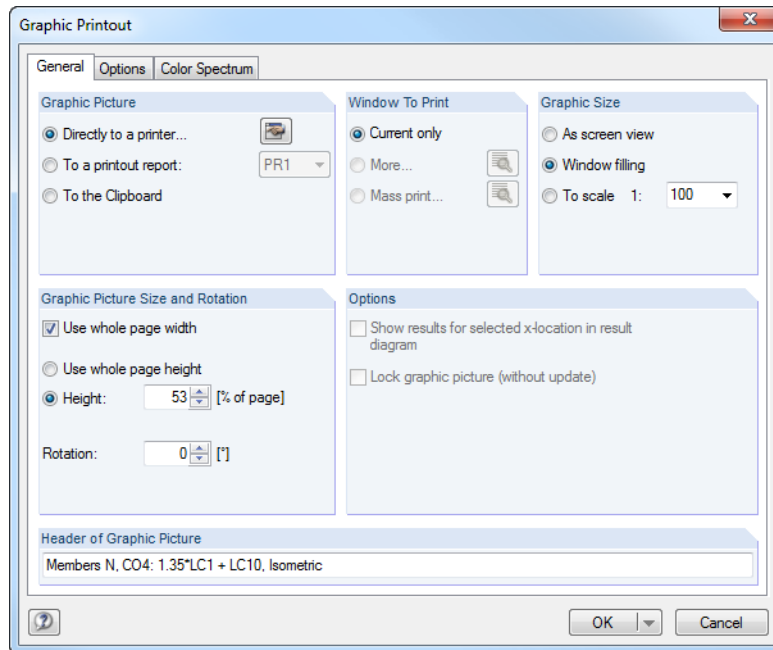


Figure 10.44: Dialog box *Graphic Printout*, tab *General*

### 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 236)
- 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 233).

### 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 10.45).

Please note when printing several graphic windows (see chapter 9.6, page 211) that you can only print graphics of one and the same model. A cross-model printout is not possible.

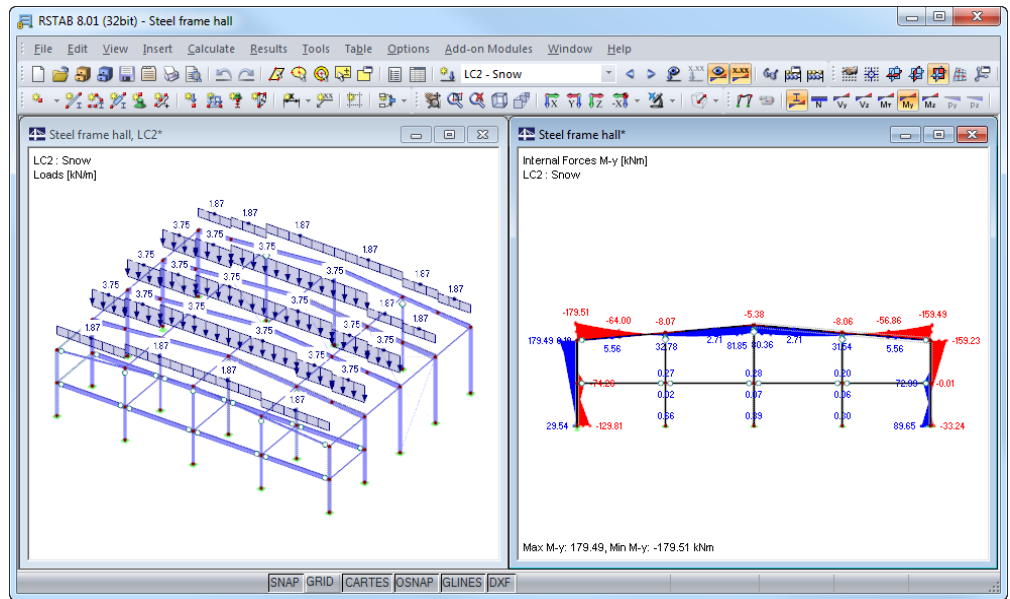


Figure 10.45: Display with two windows of same model



Select *More* to enable the button [Edit Window Arrangement] that opens a dialog box with control options for the print arrangement of graphics.

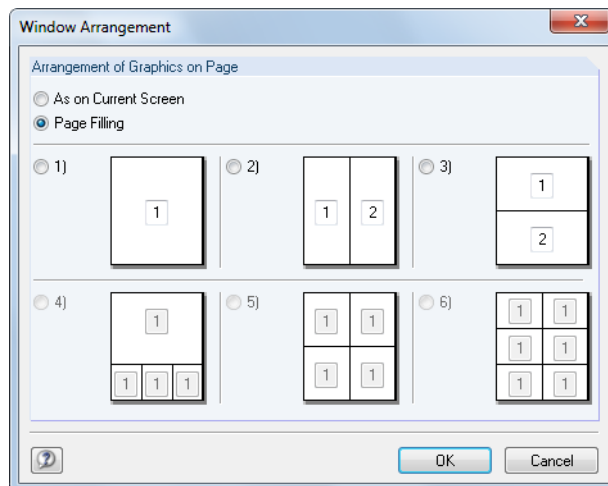


Figure 10.46: Dialog box *Window Arrangement*

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. When the option is selected, three additional tabs become available where you can define the parameters (see chapter 10.2.4, page 255).

### Graphic size

The dialog section in the upper right corner of the dialog box *Graphic Printout* (Figure 10.44) 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.

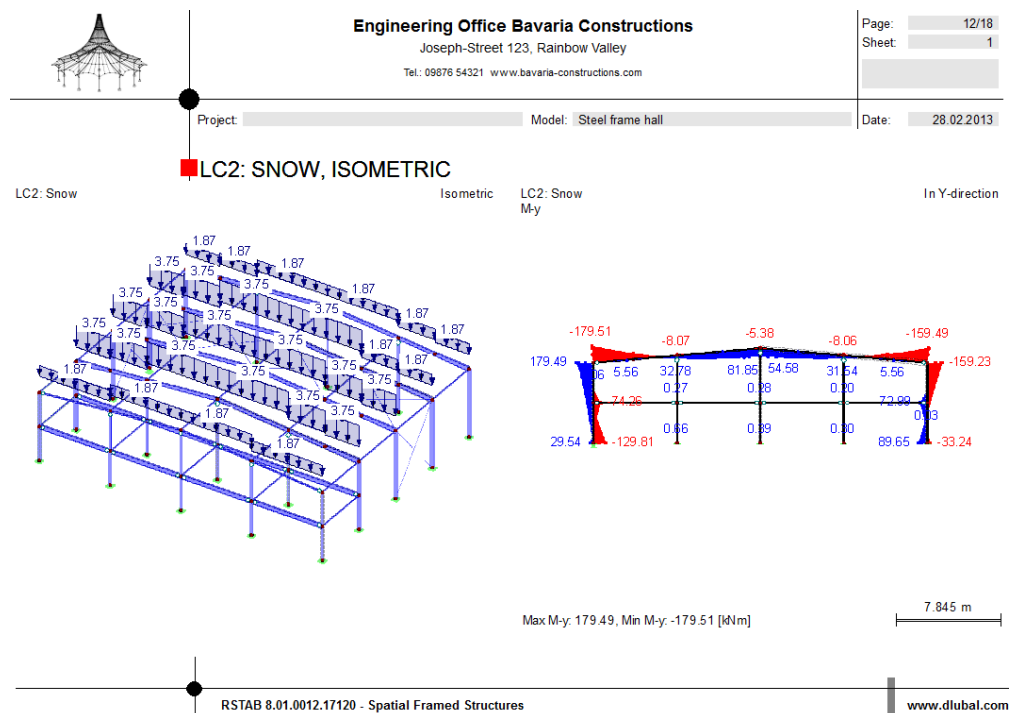


Figure 10.47: Graphic printout in landscape format: result of options *All windows* and *Use whole page width*

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 9.8, page 208).

### 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

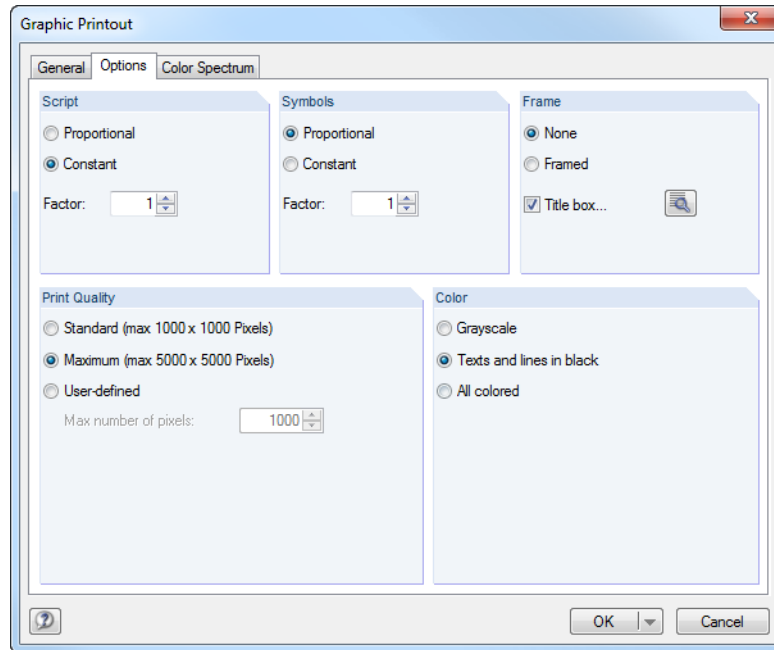


Figure 10.48: Dialog box *Graphic Printout*, tab *Options*

### Script / Symbols

In most cases, it is not necessary to change the default settings in both dialog sections. However, for printing with plotters using large formats, it may be required to adjust the factors.

The size of font and graphic symbols (nodes, supports, members 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*.

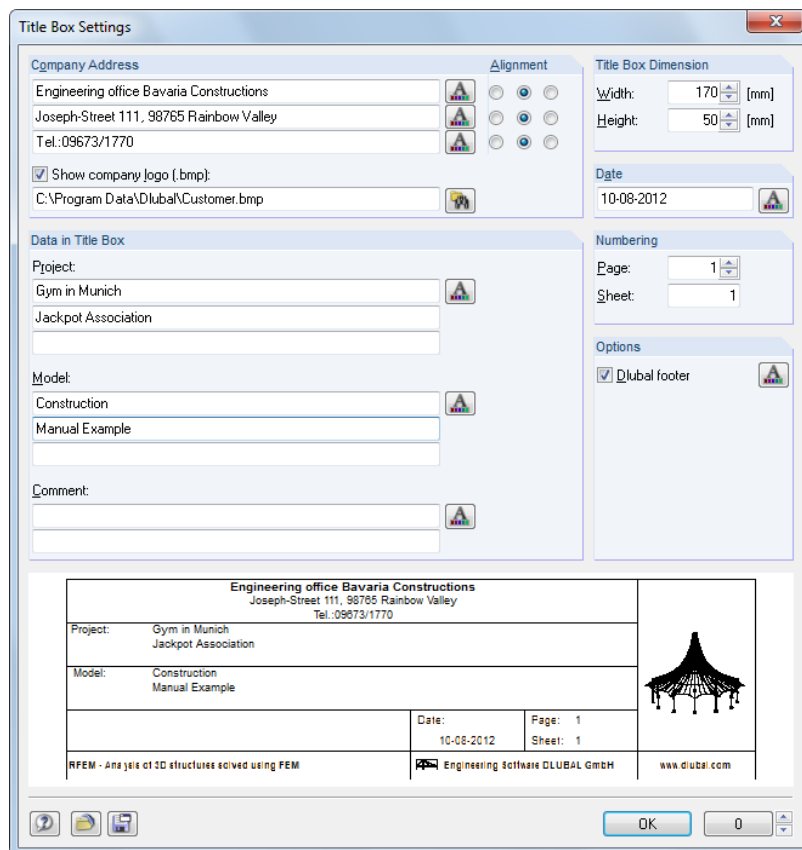
### Frame

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 10.49: 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* (Figure 10.48). 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 print-out.



The conversion from colored result diagrams to gray scales is always done by the printer driver. Corresponding setting options do not exist in RSTAB.

## 10.2.3 Color Spectrum

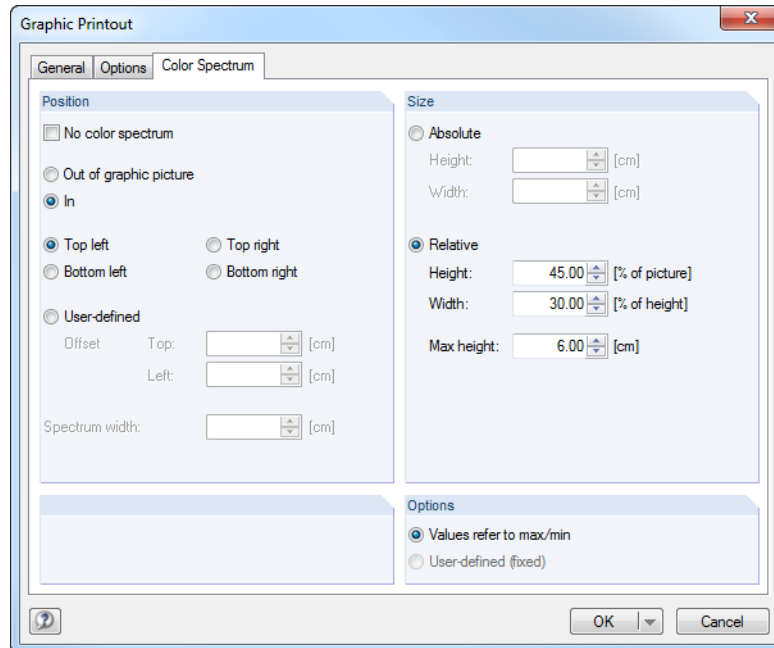


Figure 10.50: Dialog box *Graphic Printout*, tab *Color Spectrum*

The tab is only available when results are shown in a multicolor display (see chapter 9.3, page 205).

### 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.

### Options

The color-value assignment in the work window can be user-defined (see chapter 3.4.6, page 28).

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 button [Mass Print Settings] is enabled when the option *Mass print* has been selected in the **General** tab. Use mass print settings to decide which default graphics of the model, loads and results are integrated automatically into the printout report.

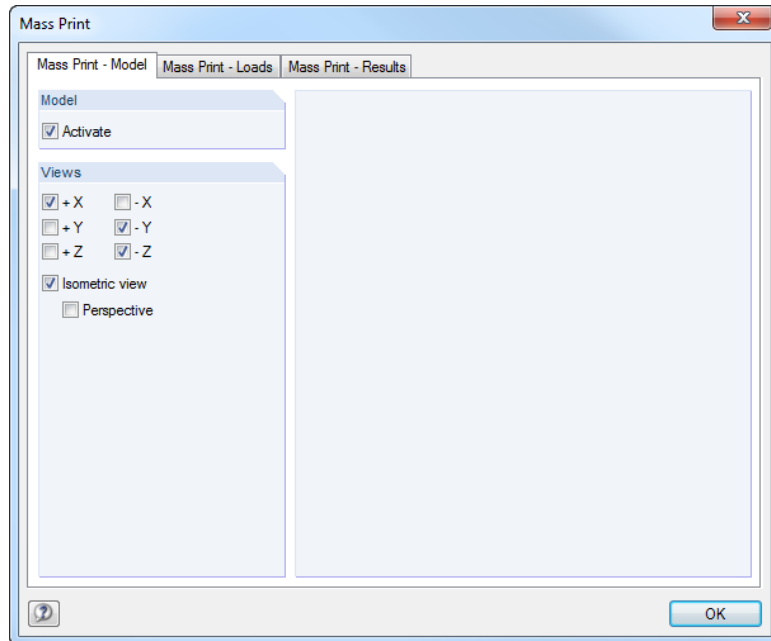


Figure 10.51: Dialog box *Mass Print*, tab *Mass Print - Model*

Seven different default *Views* are available for selection. Furthermore, you can activate a 3D *Perspective* for the model display.

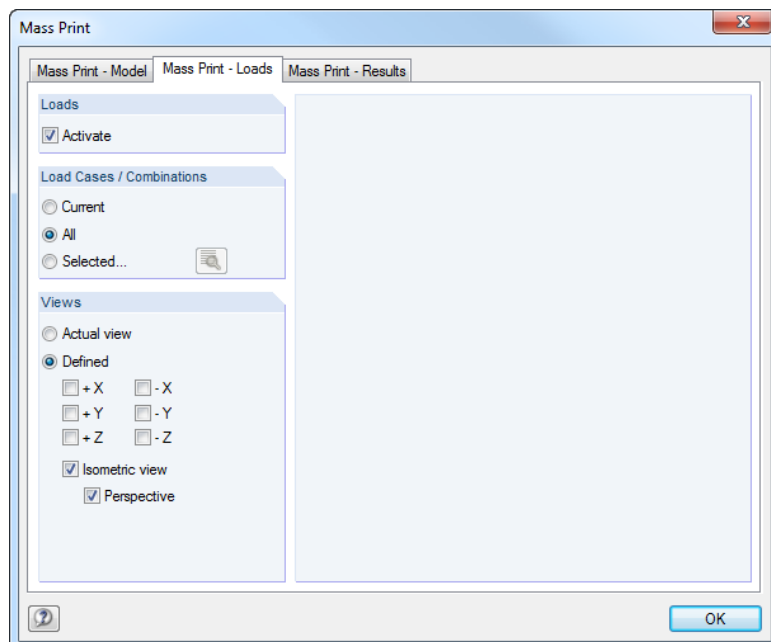


Figure 10.52: Dialog box *Mass Print*, tab *Mass Print - Loads*



In the *Loads* dialog section, 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 10.54, page 256).

Finally, in the dialog section *Views*, decide which angles of view are used for the default graphics.

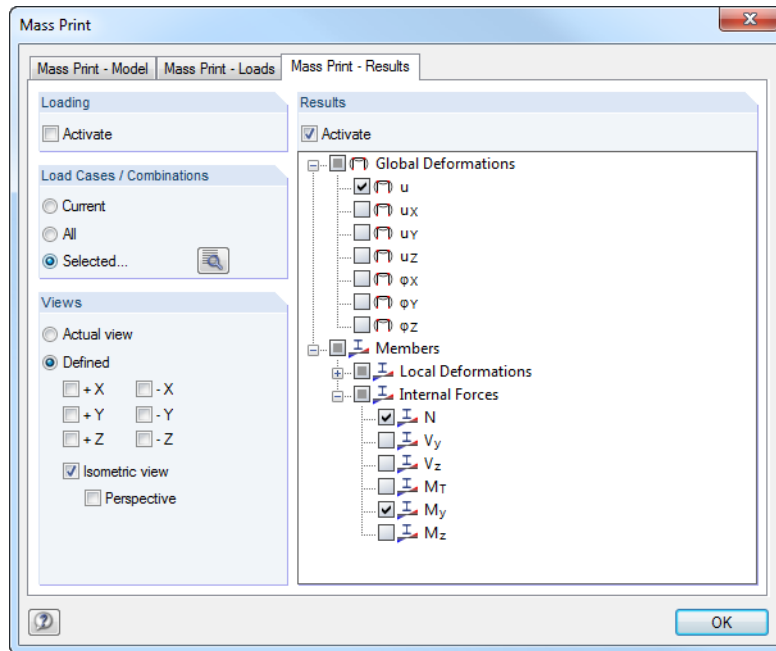


Figure 10.53: Dialog box *Mass Print*, tab *Mass Print - Results*

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 *Loading* 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.

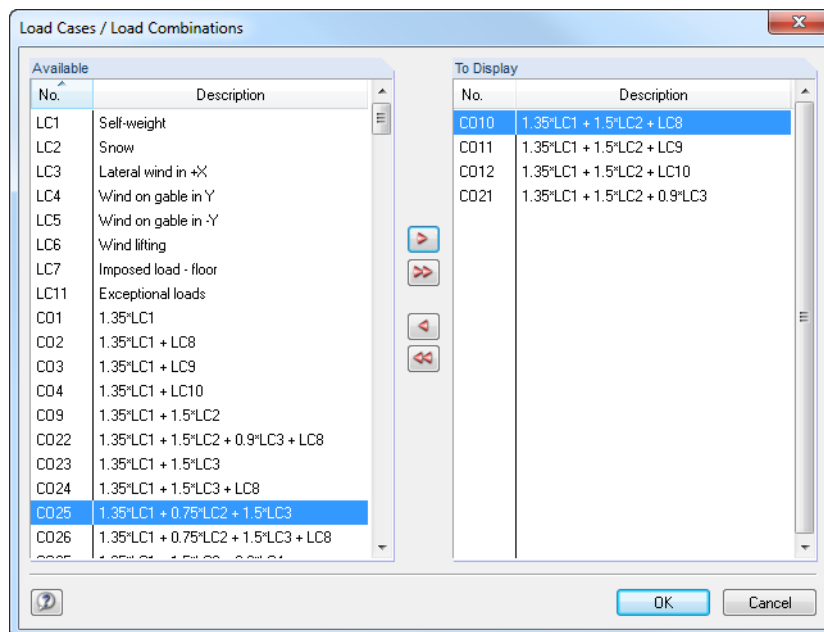


Figure 10.54: Dialog box *Load Cases / Load Combinations*

The graphics' angle of view is defined in the dialog section *Views*.

# 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 RSTAB.

### 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 RSTAB,



select **Program Options** on the **Options** menu

or use the toolbar button shown on the left.

In the dialog tab *Program*, you can select another *Program Language* in the list.

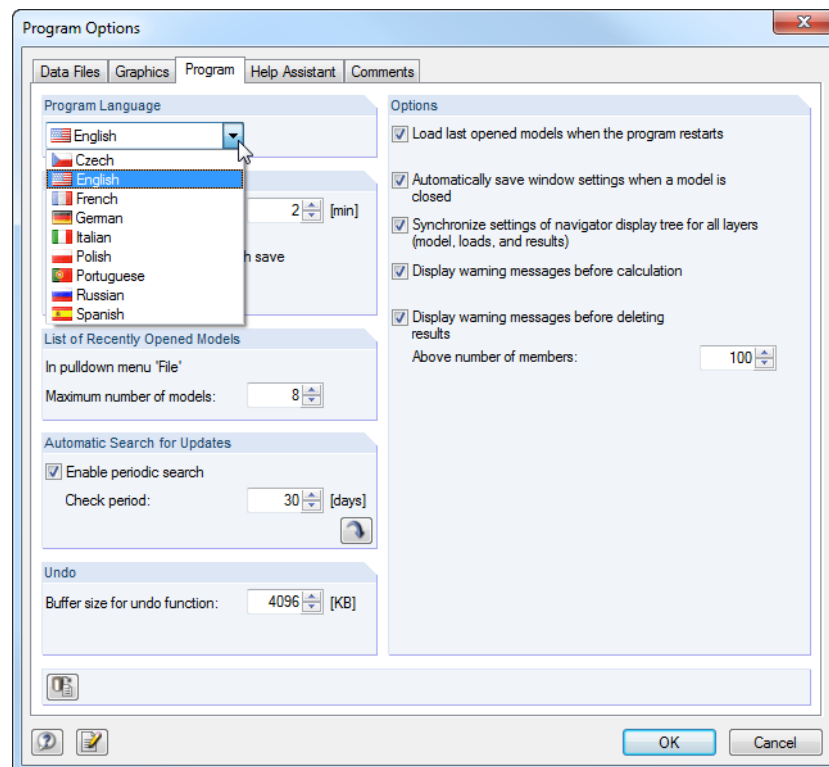


Figure 11.1: Changing the *Program Language* in the dialog box *Program Options*

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 22).

### 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 34).

It is also possible to access the display properties of each graphical object (model, load or result symbol) directly: 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).

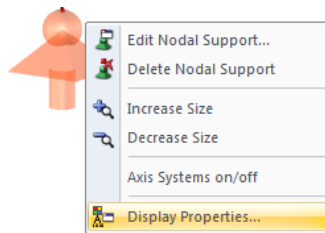


Figure 11.2: Context menu of nodal support

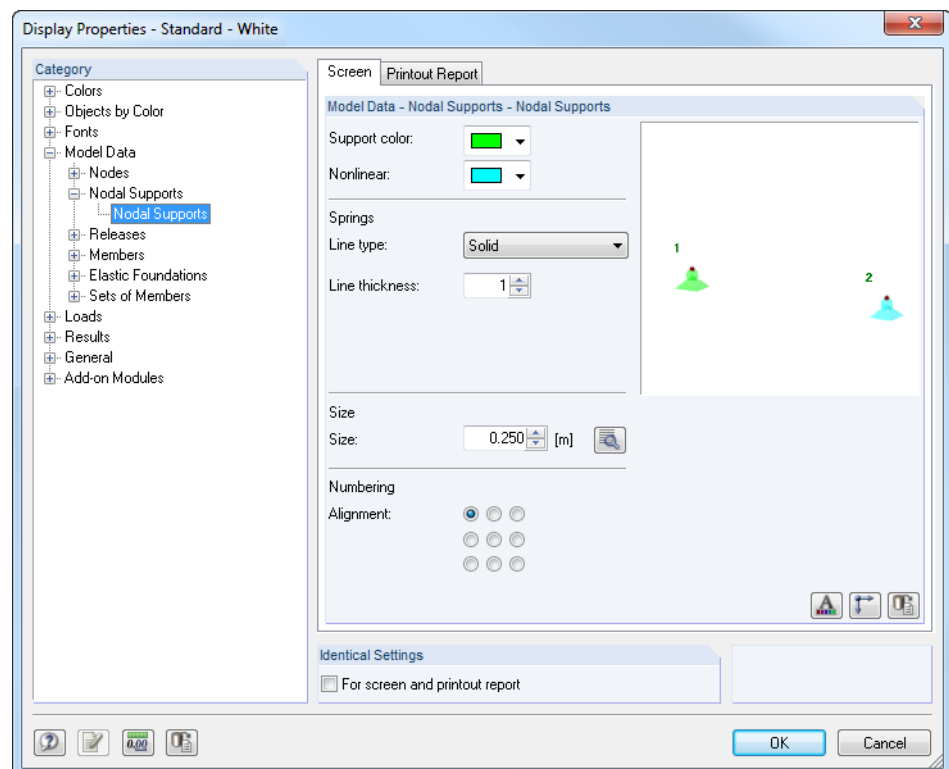


Figure 11.3: Dialog box *Display Properties* (for category *Nodal Supports*)



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 following settings that are defined subsequently 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.

RSTAB offers additional [Details] buttons for some parameters.

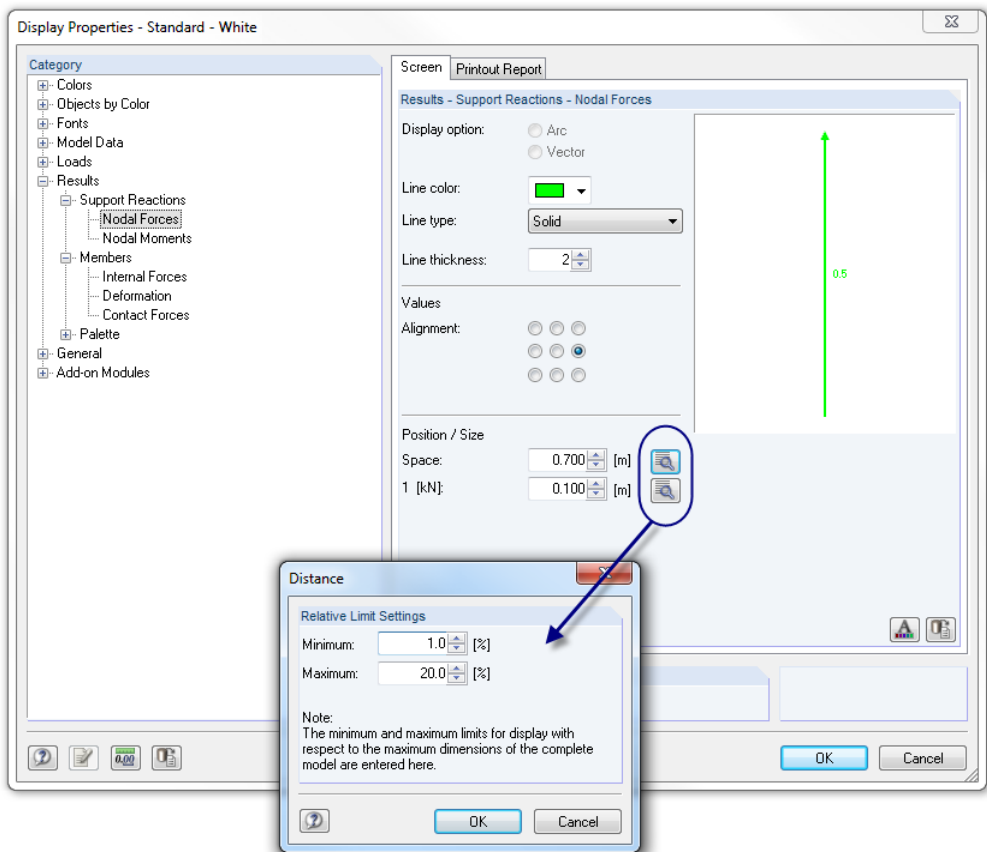


Figure 11.4: Dialog box *Distance* for *Nodal Forces*

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 are reserved for the following functions:

	Opens the <i>Font</i> 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 <i>Relative Positions</i> (Figure 11.5) for arranging descriptions
	Restores default settings

Table 11.1: Buttons in dialog box *Display Properties*



For objects that are relevant for members it is possible to arrange the description or symbol by user-defined settings. A dialog box opens where you can define the position of the information by means of a relative distance to the member start.

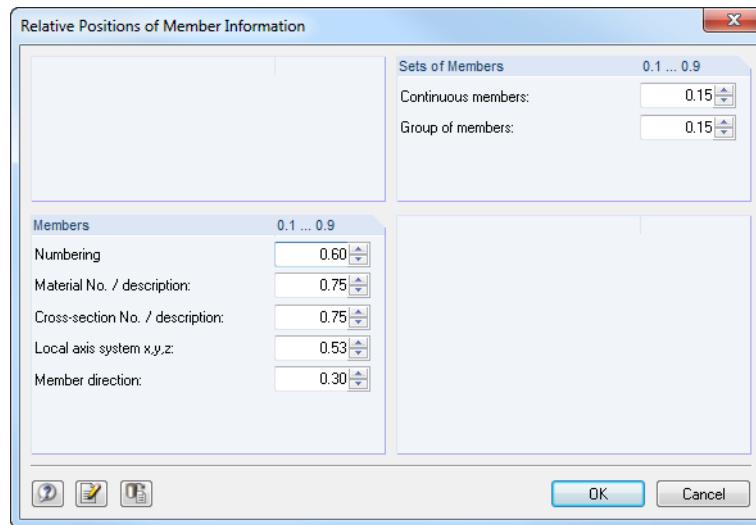


Figure 11.5: Dialog box *Relative Positions of Member Information*

### Save display configuration

The dialog box *Display Properties* allows you to adjust the display configuration to the given requirements. So it is possible to create for example different settings for the screen with colored background and for the monochrome printer 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 34 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 34).
- Create a [New] configuration.
- Enter a description in the dialog box *New Configuration*, and then confirm the input with [OK].



### 11.1.3 Units and Decimal Places

The units and decimal places for RSTAB 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 that 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.

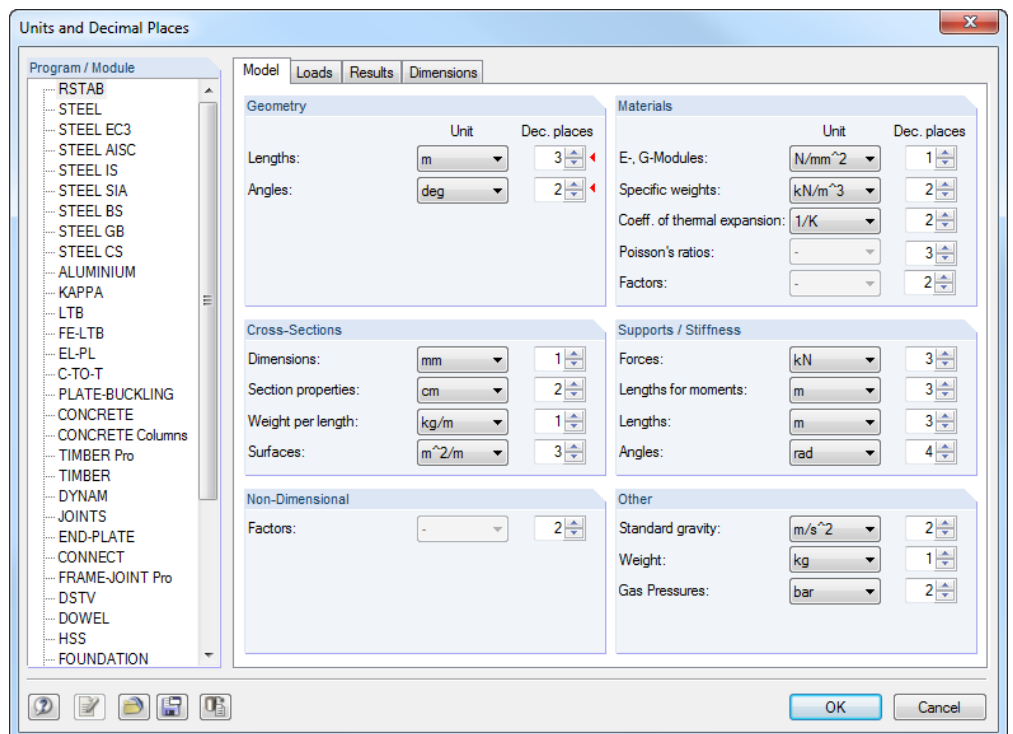


Figure 11.6: Dialog box *Units and Decimal Places*

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 RSTAB 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 the figure above.

#### Saving and import of units as user profile

The settings in the dialog box *Units and Decimal Places* can be saved and reused in other models. Thus, creating specific unit profiles for models consisting of steel and reinforced concrete, for example, is possible.

The button shown on the left opens a dialog box where you specify the *Name* of the new units user profile.



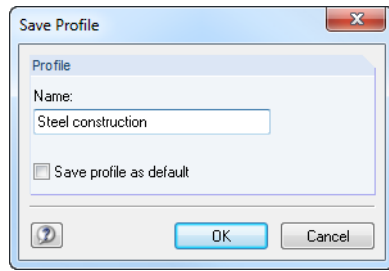


Figure 11.7: Dialog box *Save Profile*

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.

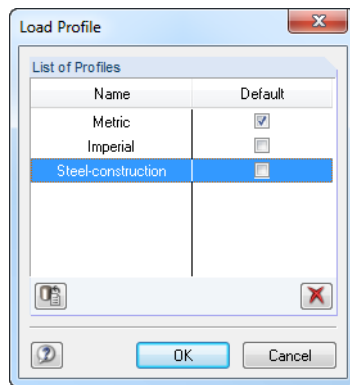


Figure 11.8: Dialog box *Load Profile*

### 11.1.4 Comments

This chapter describes the comment fields available in dialog boxes and tables (see for example Figure 4.12, page 41). The comments that you can insert graphically are described in chapter 11.3.6 on page 288.

#### 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.

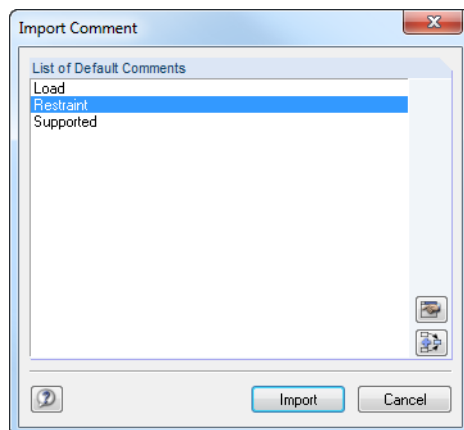


Figure 11.9: Dialog box *Import Comment*

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.

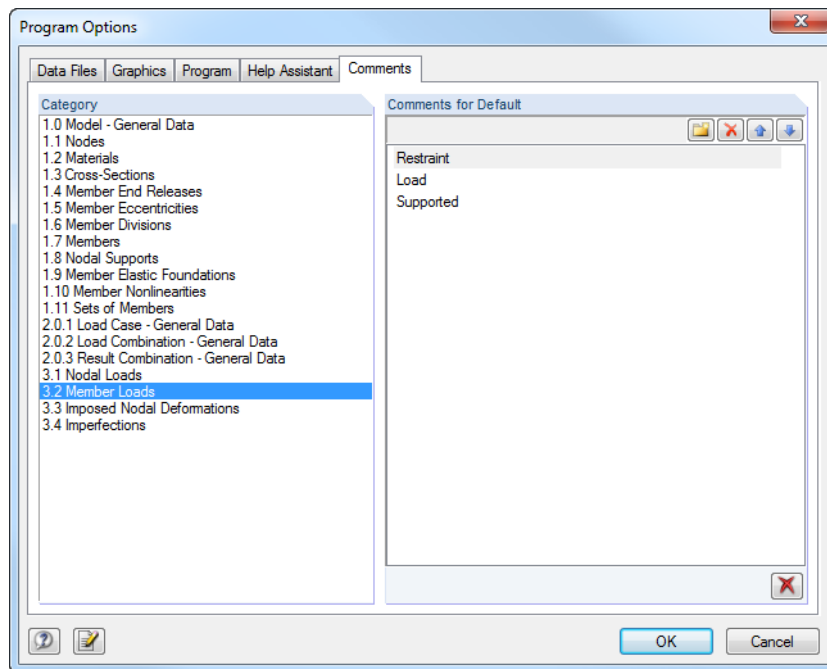


Figure 11.10: Dialog box *Program Options*, tab *Comments*

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 are reserved for the following functions:

Button	Description
	Creates a new comment within the marked <i>Category</i> . 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 274), you can filter data by user-defined comments.

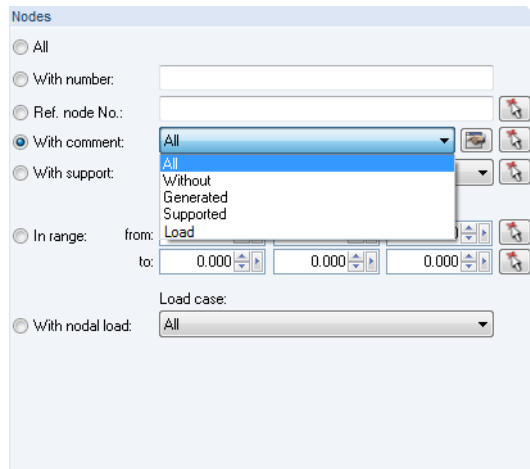


Figure 11.11: Dialog box *Special Selection* (dialog section) for nodes filtered by *comment*

### 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

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.

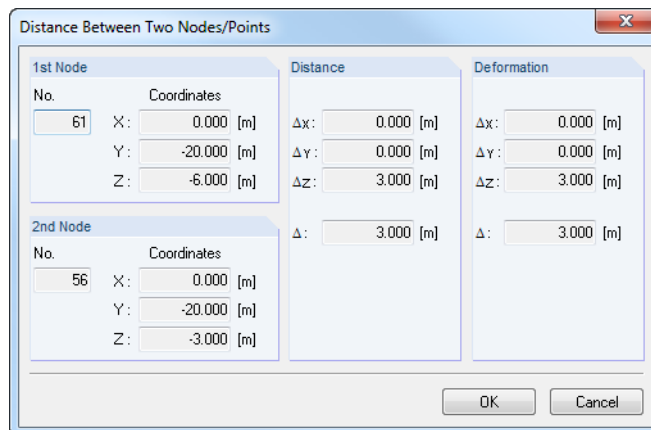


Figure 11.12: Dialog box *Distance Between Two Nodes/Points*

### 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 327).

#### Searching by object number



In RSTAB you can also search for objects specifically, which is especially recommended for rather large and complex structures. To access the search function, select **Find via Number** on the **Edit** menu.

The following dialog box appears:

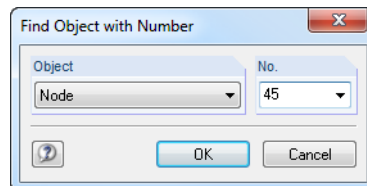
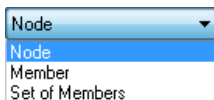


Figure 11.13: Dialog box *Find Object with Number*



In the dialog section *Object*, use the list to define the object category for searching: node, member or set of members. Then, enter the *No.* of the object directly into the input 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



RSTAB 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. Additional 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 9.7.1.1, page 213).

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.

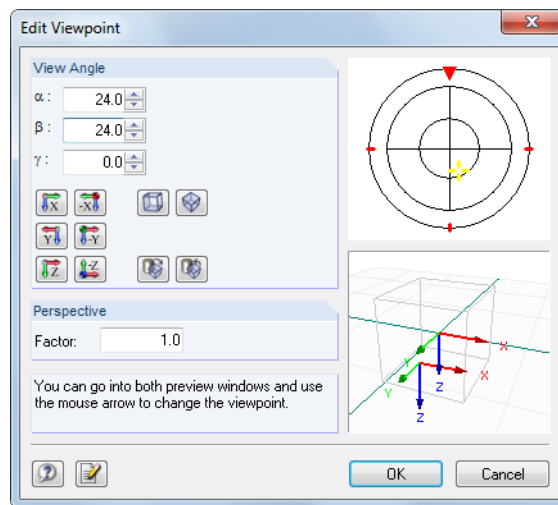


Figure 11.14: Dialog box *Edit Viewpoint*

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 option *Center of gravity* is ticked in the *Display* navigator under the navigator item *General*. Color and size can be adjusted in the dialog box *Display Properties*: Click *Colors* → *Other* → *Center of Gravity* (see chapter 11.1.2, page 258).

Moreover, it is possible to determine the centroid of particular objects: Select the relevant members, for example by multiple selection or by opening a selection window (see chapter 11.2, page 271). Activate the context menu shown on the left by right-clicking one of the selected members. Then, click the menu item *Centroid and Info* to open a dialog box with information about the selected objects.

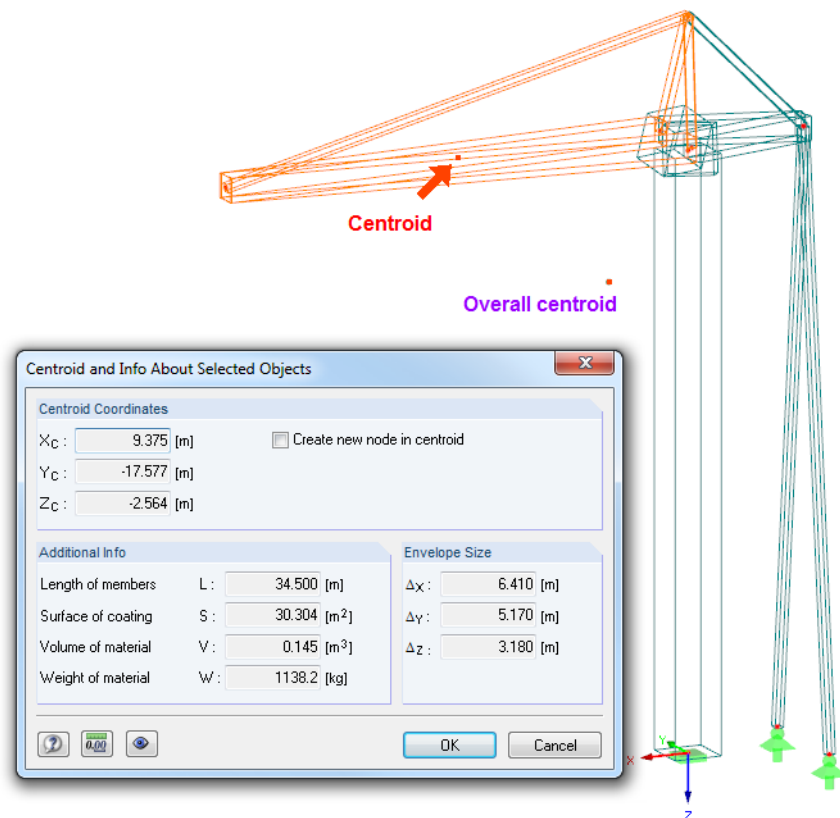
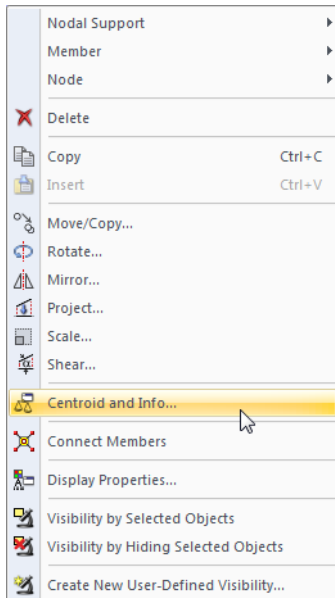
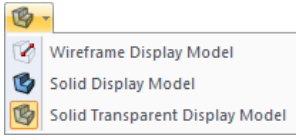


Figure 11.15: Dialog box *Centroid and Info about Selected Objects*

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 an arrow. Optionally, you can *Create a new node in the centroid*.

In addition to the global dimensions of the selected objects (*Envelope Size*), the following *Additional Info* is displayed:

- Length of members
- Surface area of visible surfaces of all members
- 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 can be specified in the *Display* navigator under the navigator item **Rendering**.

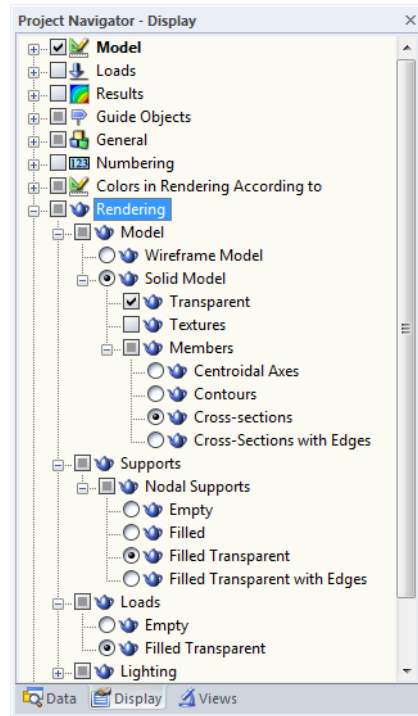


Figure 11.16: *Display* navigator with options for *Rendering* of model and load objects

RSTAB provides control options for the *Solid Model* representation of members as well as the display of supports and loads.

#### Textures

When *Textures* are activated, RSTAB shows the surface textures in the rendered model. To access detail settings for the textures,

point to **Display Properties** on the **Options** menu, and then 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*.



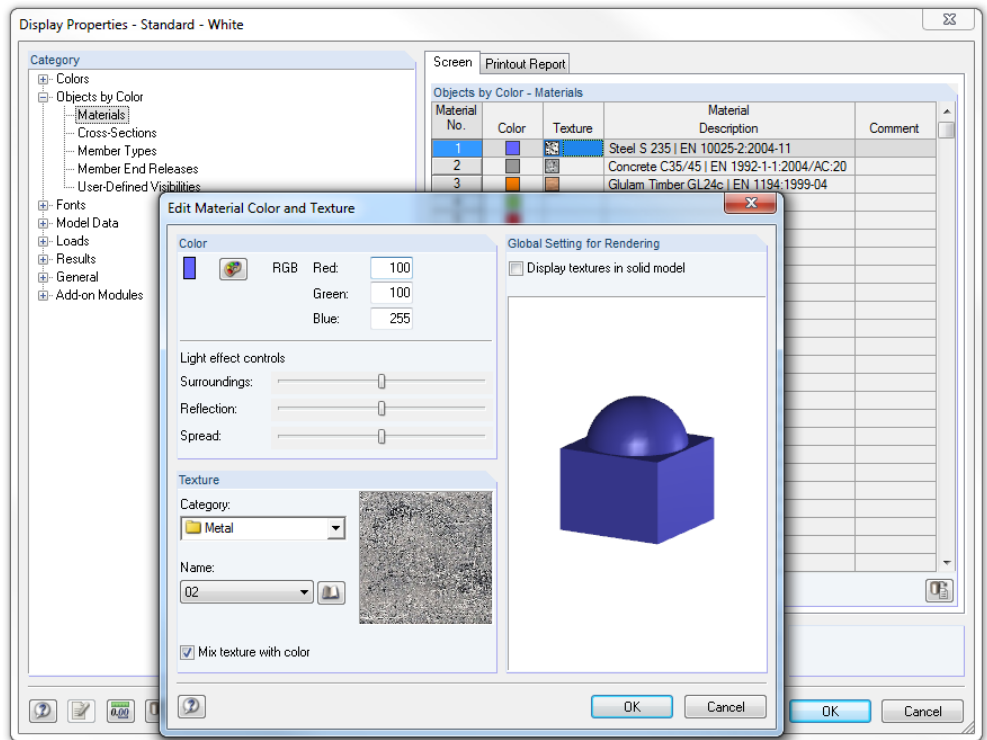


Figure 11.17: Dialog box *Edit Material Color and Texture*



Use the dialog box to adjust *Color* and *Texture* of the selected material. RSTAB provides a color palette and a comprehensive library with textures (see dialog buttons).

### Color control

The *Display* navigator item **Colors in Rendering According to** (see Figure 11.16) contains several selection fields. An activated field controls the assignment of colors for the objects in the rendering. By default, RSTAB uses the material colors defined for the individual construction materials (see chapter 4.2, page 46). With the remaining options it is possible to check graphically also cross-sections, member types, etc. based on the assigned colors.

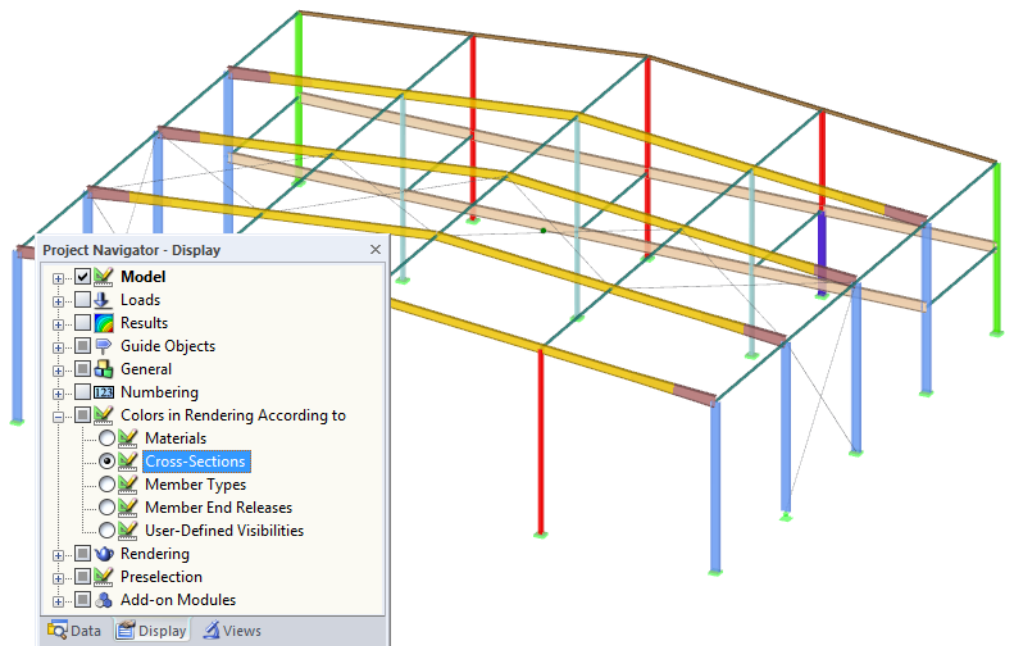


Figure 11.18: Option *Colors in Rendering According to Cross-Sections* for checking cross-section types

### 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.

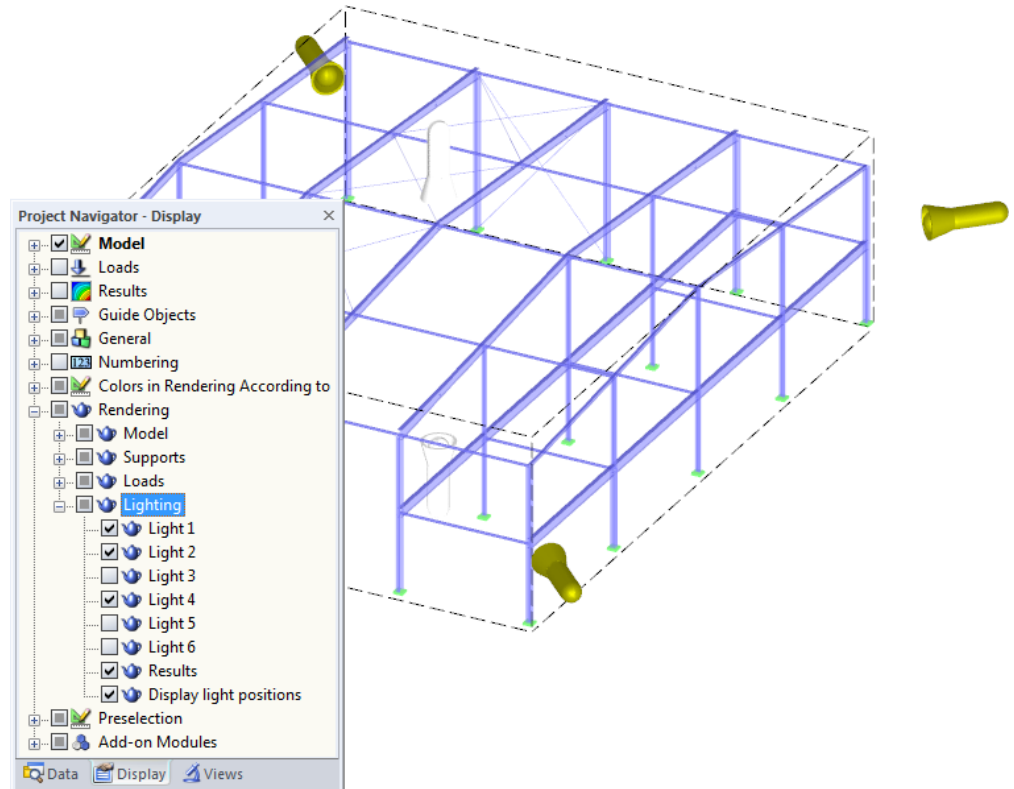


Figure 11.19: Display of light positions using the *Display* navigator

It is also possible to use the light effects for *Results*.

## 11.2 Selection

With the selection functions, you can define objects for subsequent editing: nodes, members, sets of members, supports, 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 use the tables, too: 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 327).

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.

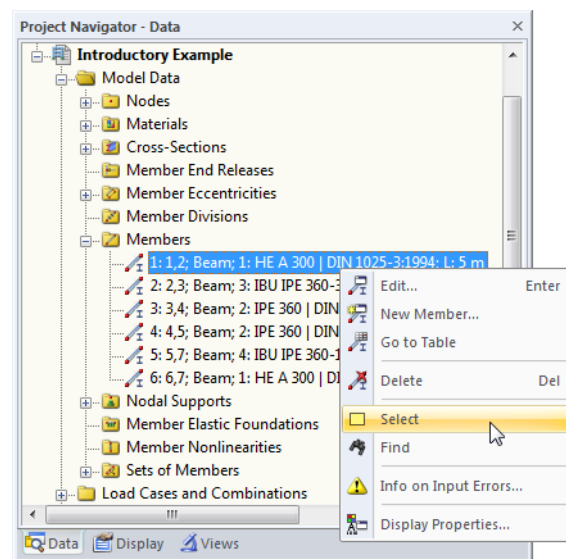


Figure 11.20: Context menu in *Data* navigator

### 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.

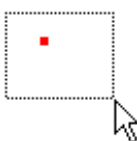


If you want to select more than one object by clicking, hold down the [Ctrl] key additionally. Another possibility is to switch to the setting *Add to Selection* by using the toolbar button shown on the left. You can also point to *Select* on the *Edit* menu where objects can be clicked separately to 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.





### 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.

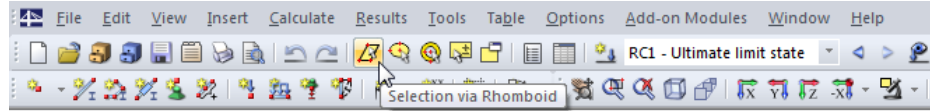


Figure 11.21: Button *Selection via Rhomboid*

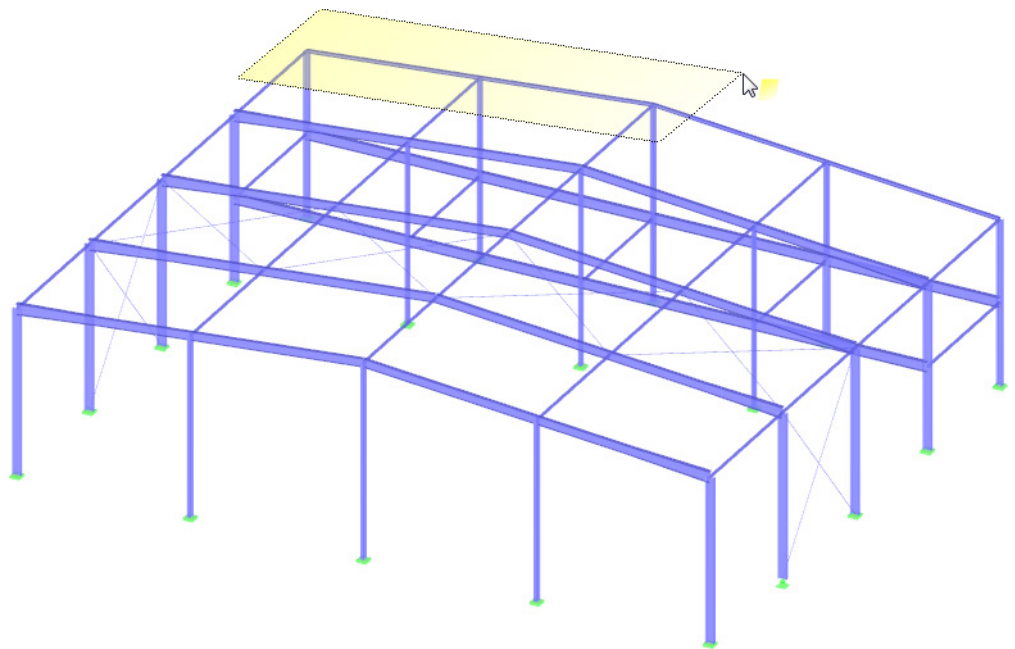


Figure 11.22: Selecting with rhomboid



### Selecting with ellipse/circular ring

The possibilities to select objects by an ellipse or circular ring represent alternatives to the rhomboid selection and can be used for example for circular models. 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.

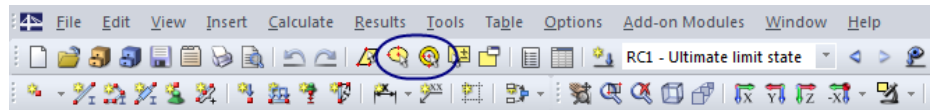


Figure 11.23: Buttons *Selection via Ellipse* or *Circular Ring*

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.

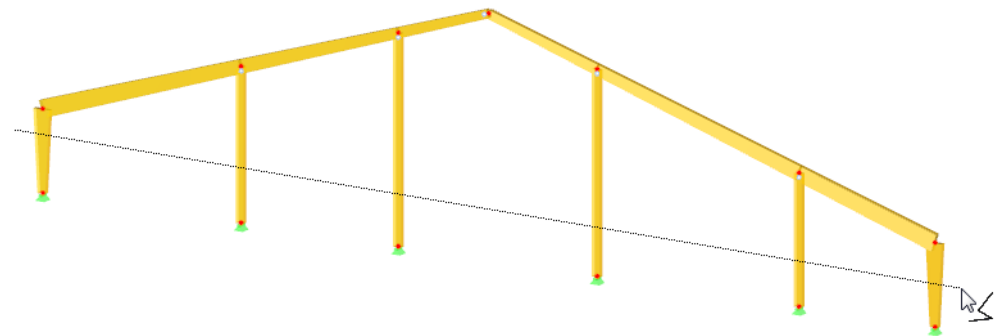


Figure 11.24: Selecting all columns with a section line



### 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.

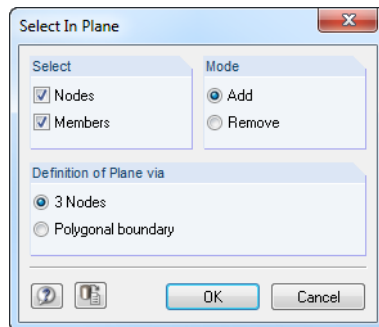


Figure 11.25: Dialog box *Select In Plane*

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 which are not used for the definition of members,

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 a member by clicking, the nodes belonging to the member 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 the supports of members quickly into the selection and to save them as related objects in a user-defined visibility (see chapter 9.7.1.2, page 217).

### 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.

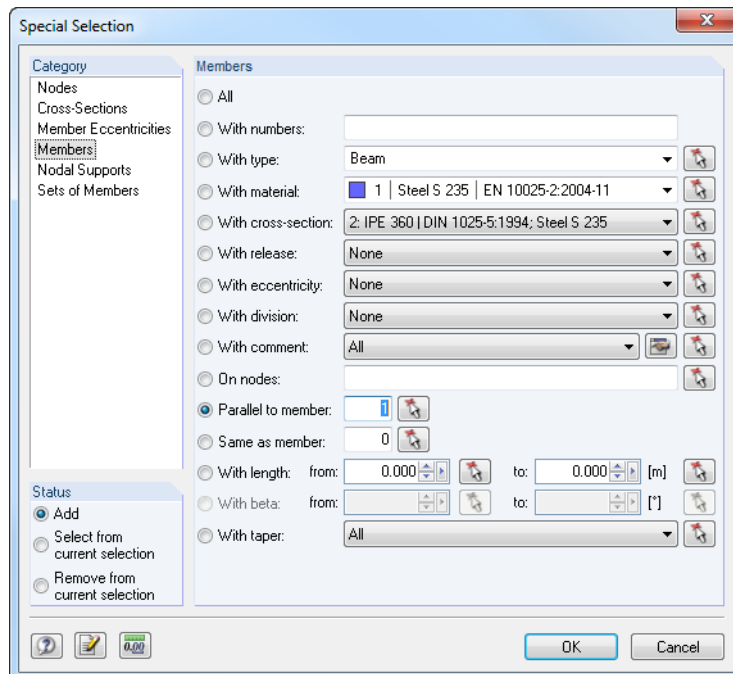


Figure 11.26: Dialog box *Special Selection*

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 column members that are modeled *Parallel to member 1* are selected. You can also use the [ $\wedge$ ] button in the dialog box to define the template member graphically.

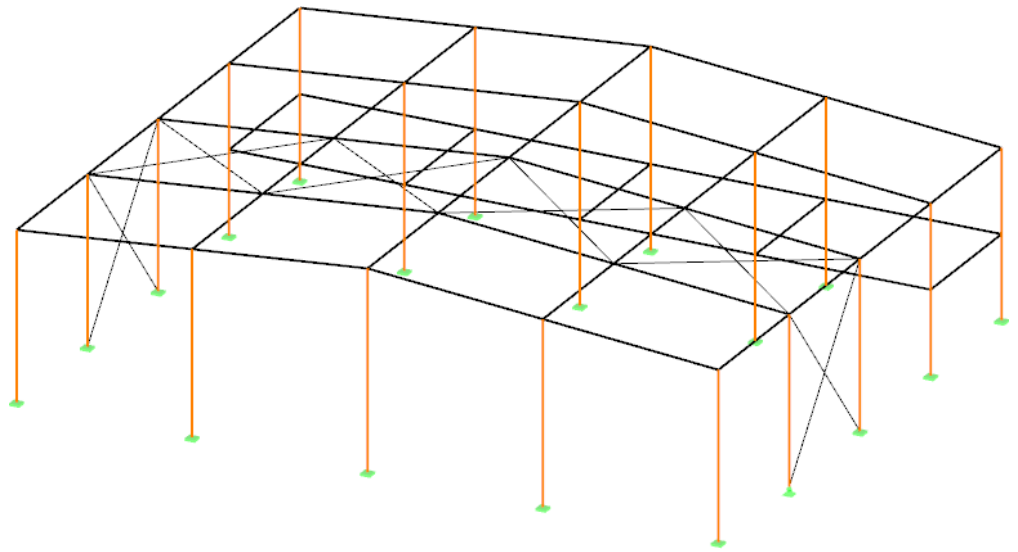


Figure 11.27: Selection of parallel members

## 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".

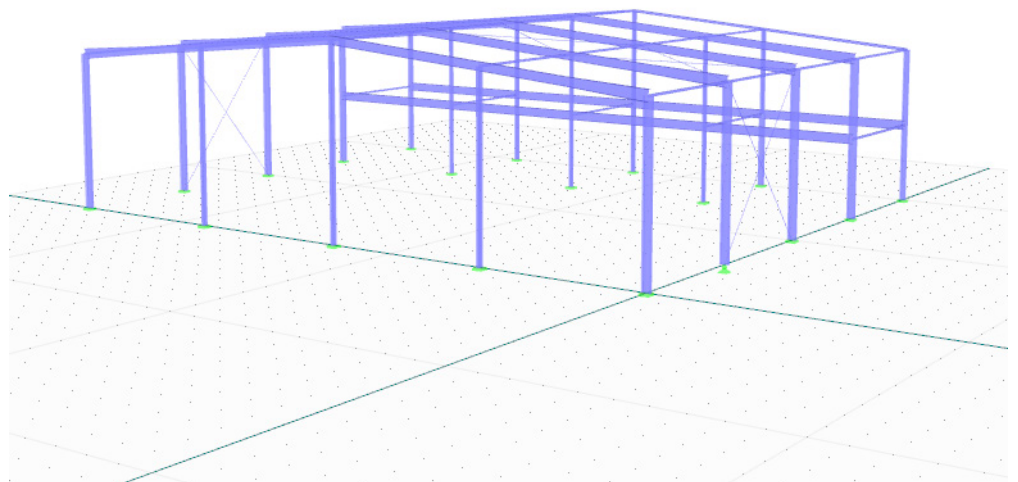


Figure 11.28: Work plane represented in the graphic

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 member 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.

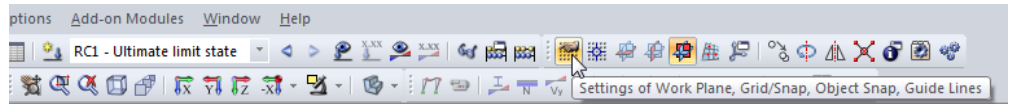


Figure 11.29: Button *Settings of Work Plane*

The dialog box shown in Figure 11.34 on page 278 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 <i>Work Plane</i> dialog box	Selection in toolbar
XY		
YZ		
XZ		

Table 11.3: Selection of work plane

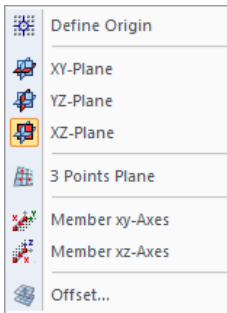
To find more options for defining work planes, point to **Select Work Plane** on the **Tools** menu or use the corresponding toolbar buttons.

### 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 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 4.7, page 80). 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. Axis *U* shows in direction of member axis *x* (see the following figure).





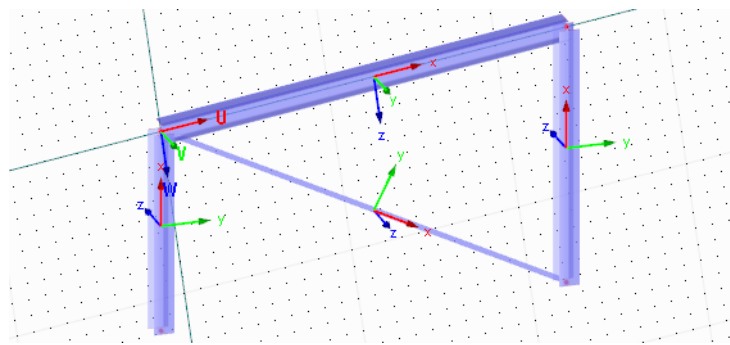


Figure 11.30: Work plane in roof inclination of member axis xz

### 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*.

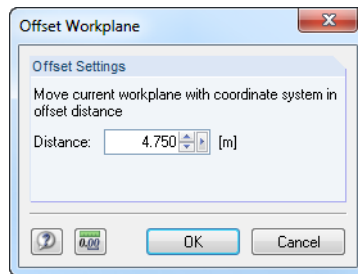


Figure 11.31: Dialog box *Offset Workplane*



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 [ $\leftarrow$ ] 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.

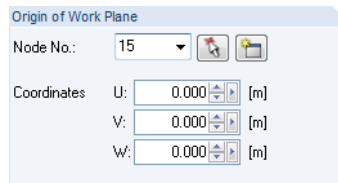


Figure 11.32: Dialog box *Work Plane*, dialog section *Origin of Work Plane*



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 276).

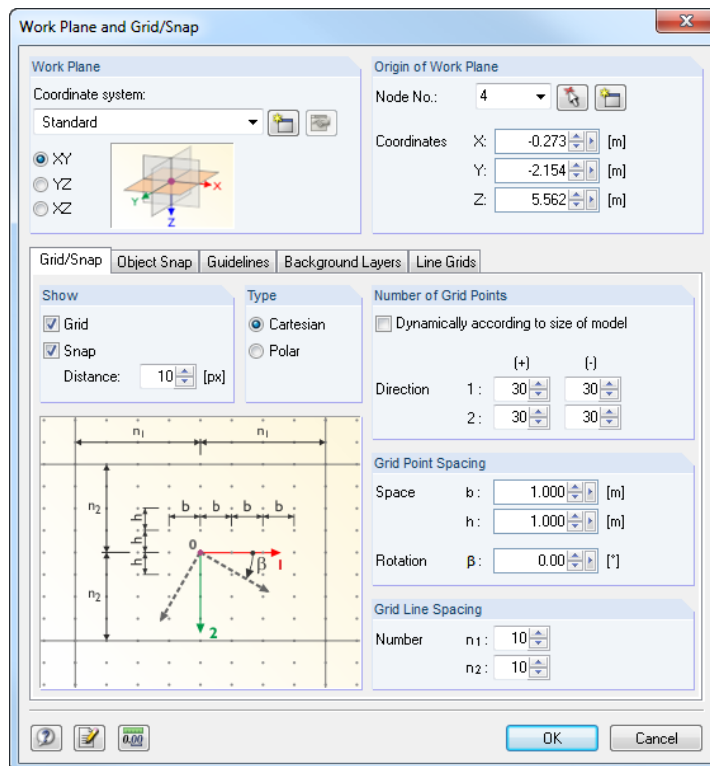
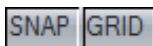


Figure 11.34: Dialog box *Work Plane and Grid/Snap*

The setting options relevant for the grid are available in the dialog tab *Grid/Snap*.

### Show

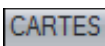
To display the grid in the work window, tick the *Grid* check box. The snap function can be turned on and off independently of the grid by means of the *Snap* check box. Thus, the snap function on the grid points can be effective while the grid is invisible.



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  $\alpha$  controls the spacing of grid points on the circles.

Optionally, the Cartesian and the polar grid can be rotated about the rotation angle  $\beta$ .

If needed, the number of pixels controlling the snap *Distance* (see dialog section *Show*) can be adjusted.

### 11.3.3 Object Snap

The object snap facilitates the CAD-like modeling when defining members. In addition to nodes, several snap points along the members 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 276).

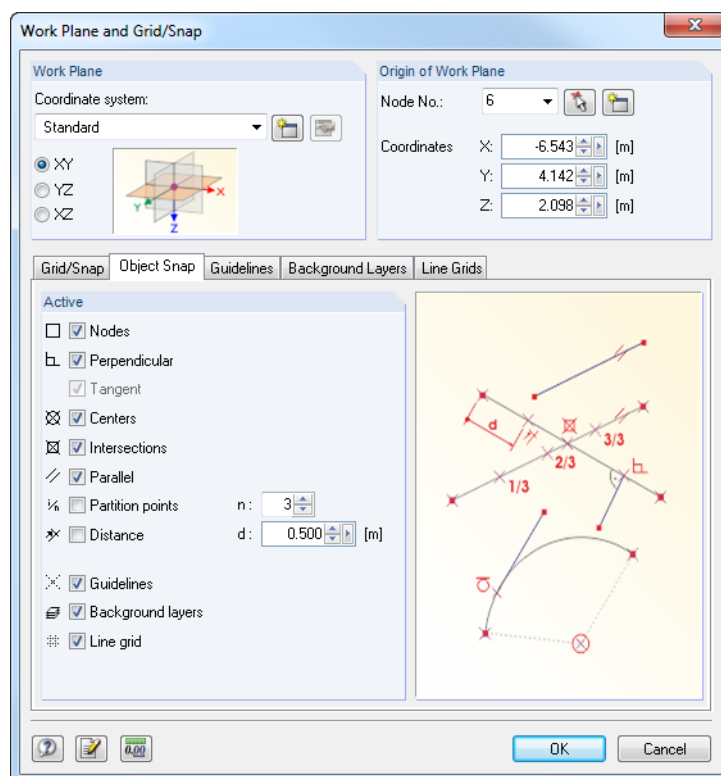


Figure 11.35: Dialog box *Work Plane and Grid/Snap*

The dialog tab *Object Snap* manages the different snap functions.

OSNAP

To make the functions of the object snap effective, make sure that the button [OSNAP] is activated in the status bar.

### Nodes

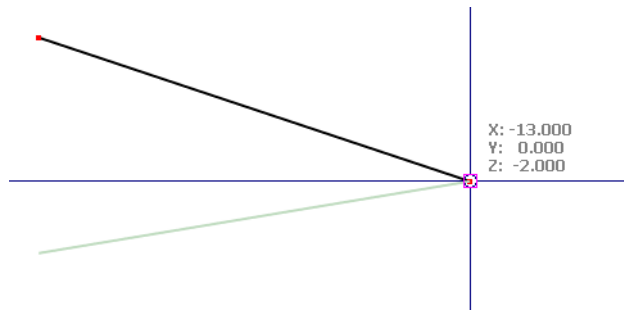


Figure 11.36: Snapping a node



When defining new members, the existing nodes will be captured. Snap points are symbolized by squares.

### Perpendicular

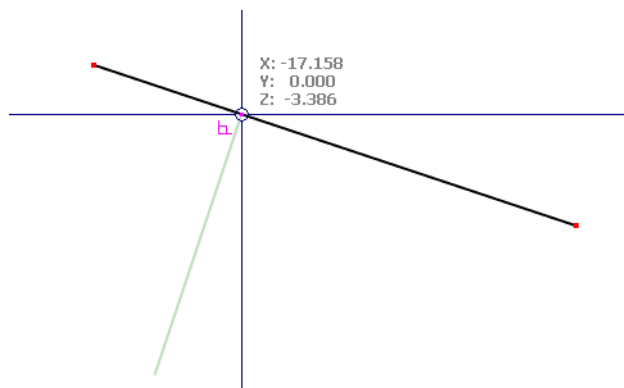


Figure 11.37: Connecting a member perpendicularly



When you define a new member, the pointer will be snapped if you move the mouse pointer near the perpendicular point of an existing member. The snap point is symbolized by a perpendicular symbol.

### Centers

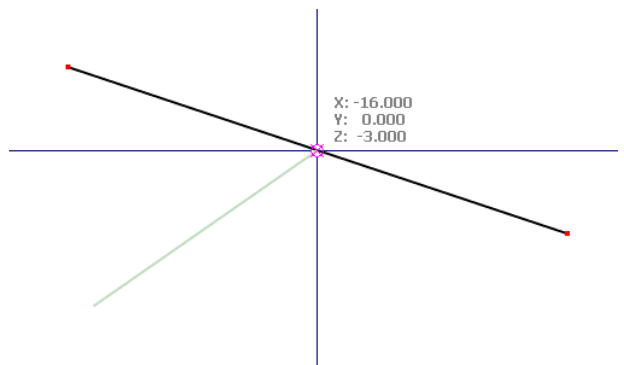


Figure 11.38: Connecting a member in center



If you move the pointer near the center (middle) of a member, it will be snapped. The center symbol appears on the snap point.

### Intersections

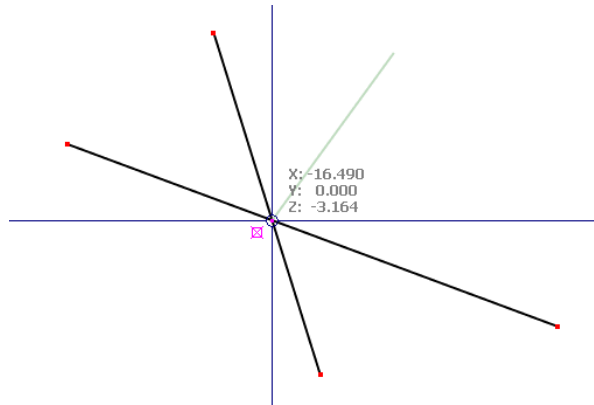


Figure 11.39: Snapping members at the intersection point



The pointer will be captured at the intersection point of two crossing members that have no node in common. The snap point is symbolized by the intersection symbol shown on the left.

### Parallel

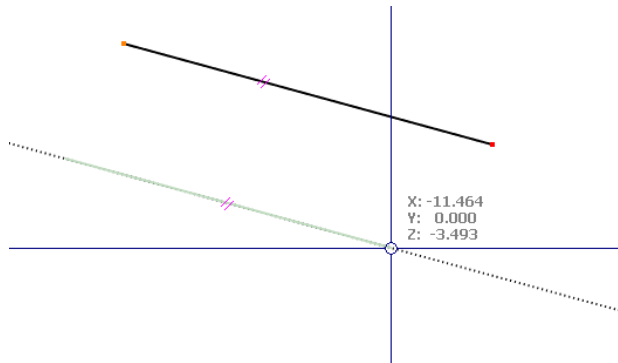


Figure 11.40: Snapping a parallel member



Use this function to set parallel members: Define the start node of the new member, and then move the pointer over a template member. Now, if you move the pointer near a possible end node of the new member running parallel to the template, the parallel symbol shown on the left appears on both members.

### Partition points

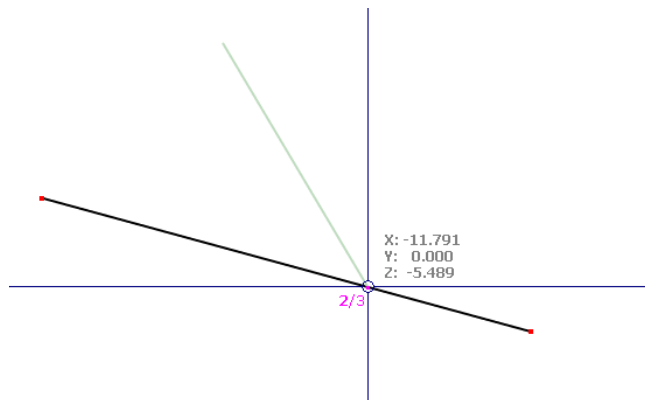


Figure 11.41: Snapping a member at the partition point (example: 2/3)



Use this option to enter a number of  $n$  member divisions. Then, when you move the pointer along a member, it will be snapped on the partition points. The partition is displayed as fraction on the pointer.

### Distance

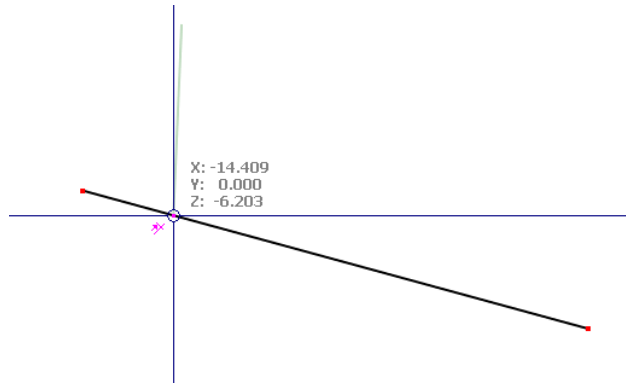


Figure 11.42: Connecting a member in a defined distance



Use this option to enter a distance  $d$  for the member division. Then, when you move the pointer along a member, it will be snapped at the defined distance from the member start as well as member end. The distance symbol appears on the pointer.

### Guidelines

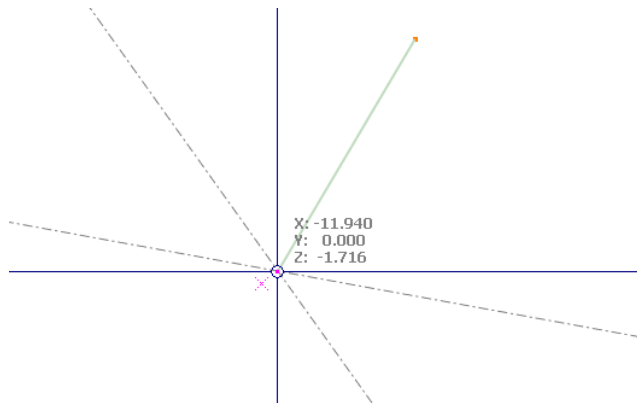


Figure 11.43: Snapping guidelines at intersection point



When you move the pointer near the intersection point of two guide lines (see chapter 11.3.7, page 290), it will be snapped. The intersection symbol appears on the snap point.

### Background layers

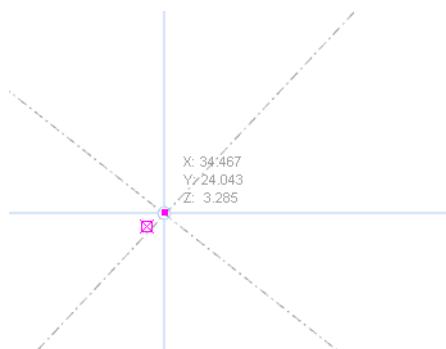


Figure 11.44: Snapping background layers on intersection point



Use this function to set nodes on intersection points of background layers (see chapter 11.3.7, page 290). The intersection symbol appears on the snap point.

### Line grid

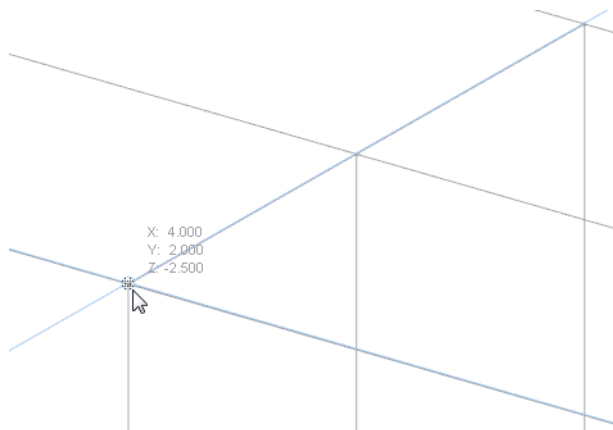


Figure 11.45: Snapping points of line grid

Use this function to place objects into the intersection points of a line grid (see chapter 11.3.8, page 294).

### 11.3.4 Coordinate Systems

User-defined coordinate systems make entering inclined parts of a model easier. Normally, these systems have nothing to do with the axis systems of members, unless they are defined graphically by the axes of particular members (see chapter 11.3.1, page 277).



To open the dialog box *Coordinate System*,  
 select **Coordinate System** on the **Tools** menu  
 or use the toolbar button shown on the left.



Figure 11.46: Button *Coordinate System*



You can also use the dialog box *Work Plane and Grid/Snap* (see Figure 3.15, page 26) where you find the [New] button for creating a user-defined system of coordinates.

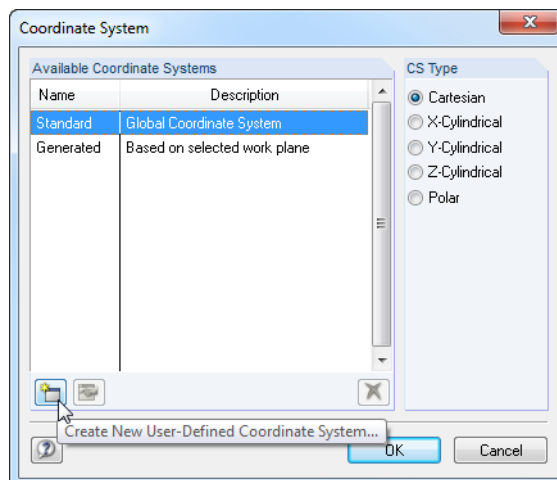


Figure 11.47: Dialog box *Coordinate System*

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 11.47 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 26).

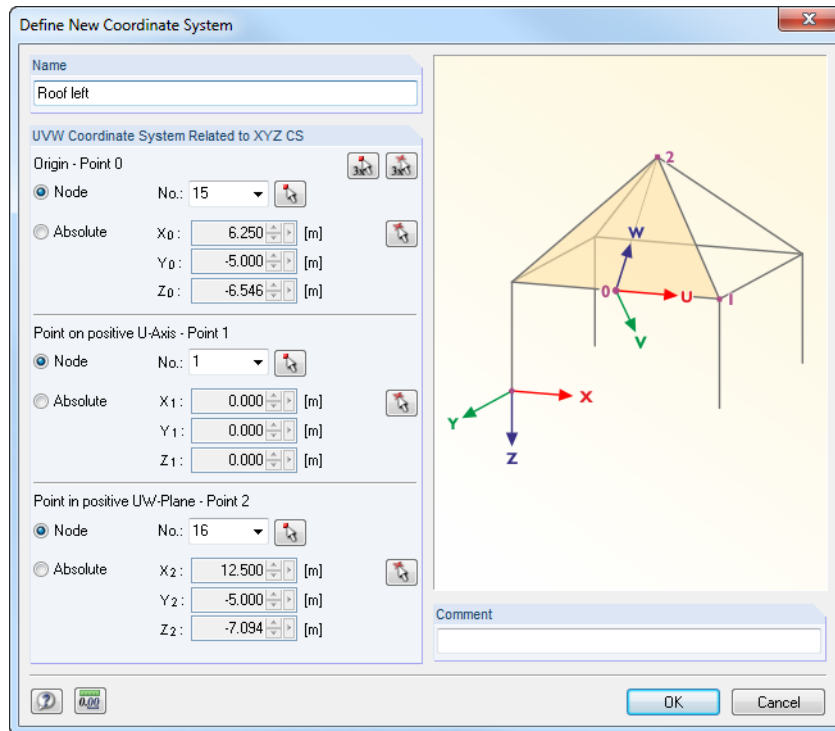


Figure 11.48: Dialog box *Define New Coordinate System*

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 276), RSTAB 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*.

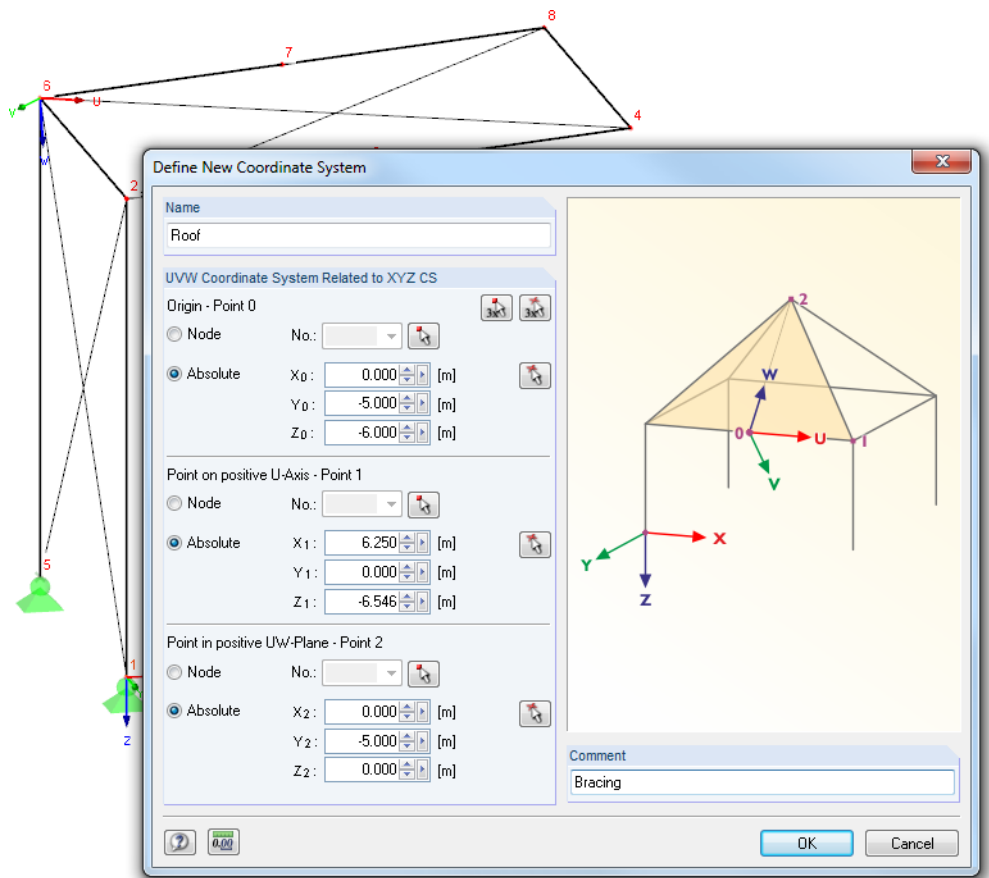


Figure 11.49: User-defined coordinate system **UVW** in a frame joint

Now, the grid refers to the work planes UV, VW and UW where you can define new objects (see chapter 11.3.1, page 275).

### 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.

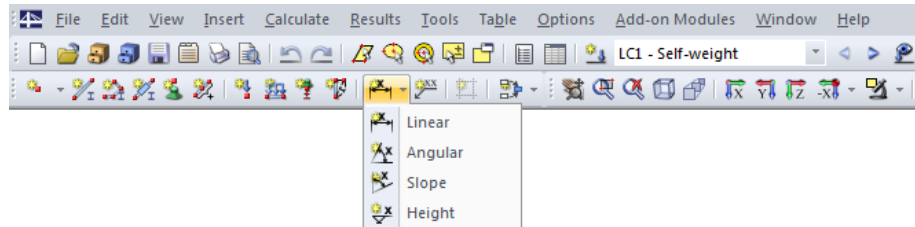


Figure 11.50: *New Dimension* buttons

The following dimension options can be selected:

Dimension	Dimensioned Objects
Linear	Lengths between two or several nodes
Angular	Angles between three nodes or two members
Slope	Inclination angle between a member and a global 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.

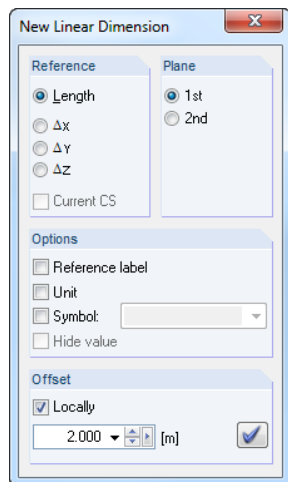


Figure 11.51: Dialog box *New Linear Dimension*

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 user-defined system of coordinates UVW. 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).

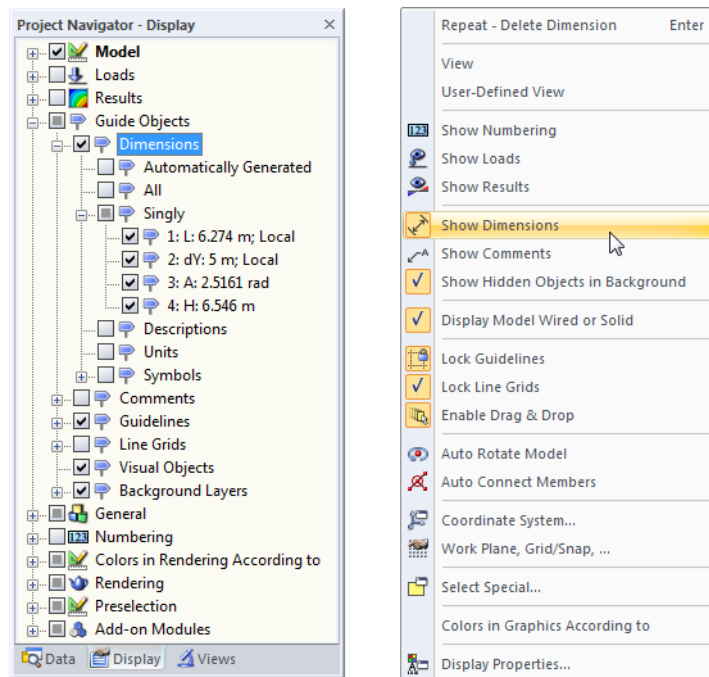


Figure 11.52: *Display* navigator (*Guide Objects* → *Dimensions*) and general context menu



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 members, 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 262)
- Comments in work window

This chapter describes how comments are set graphically.

You can place comments in reference to nodes and centers of members. They can be placed also anywhere in the current work plane or 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.

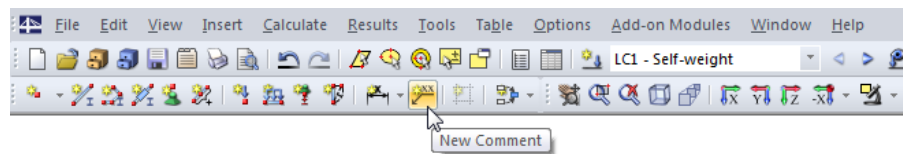


Figure 11.53: Button *New Comment*

The dialog box *New Comment* opens.

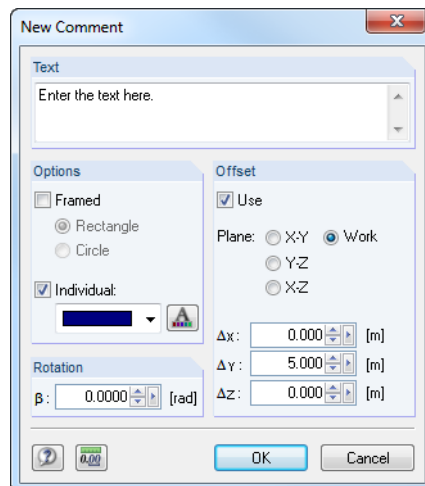


Figure 11.54: Dialog box *New Comment*



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. RSTAB 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).

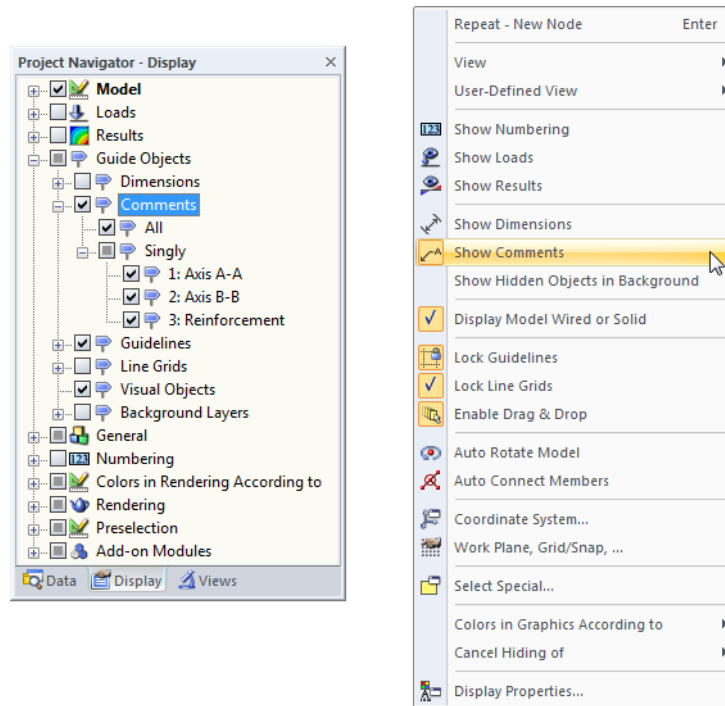


Figure 11.55: Display navigator (Guide Objects → Comments) and general context menu



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 282).

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.

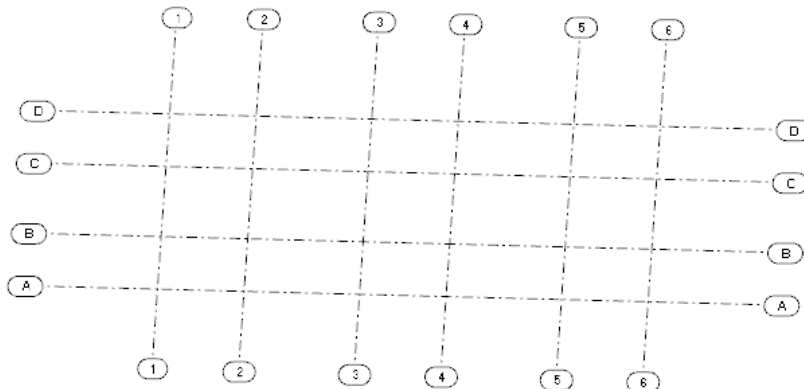


Figure 11.56: Grid of guidelines

#### 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.

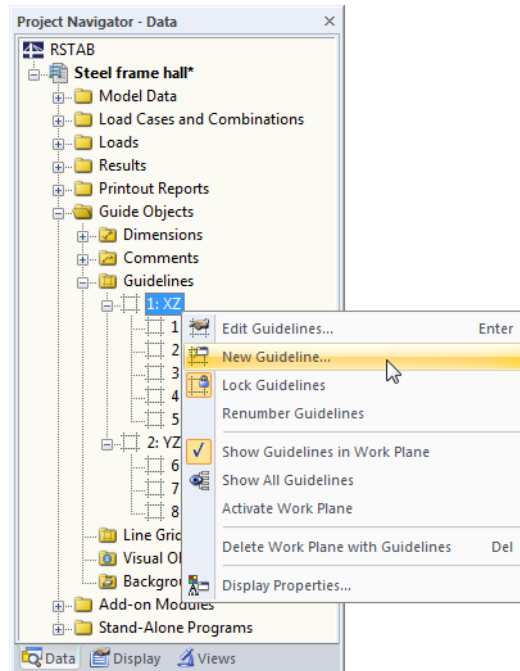


Figure 11.57: Context menu of Guidelines in Data navigator

The following dialog box appears:

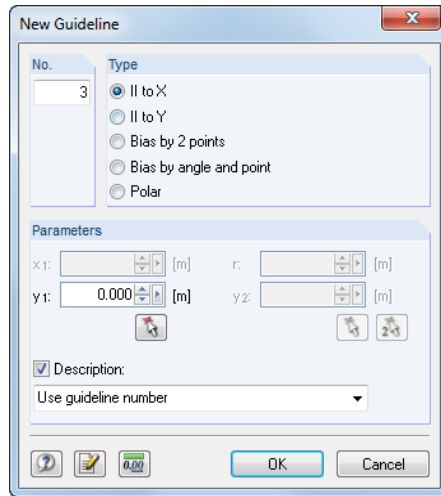


Figure 11.58: Dialog box *New Guideline*

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.

Type	Explanation
Il 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 <i>Parameters</i> .
Bias by 2 points	In the dialog section <i>Parameters</i> , 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 <i>Parameters</i> dialog section. The guideline will be created in the current work plane.
Polar	In the <i>Parameters</i> 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 [ $\wedge$ ] 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).

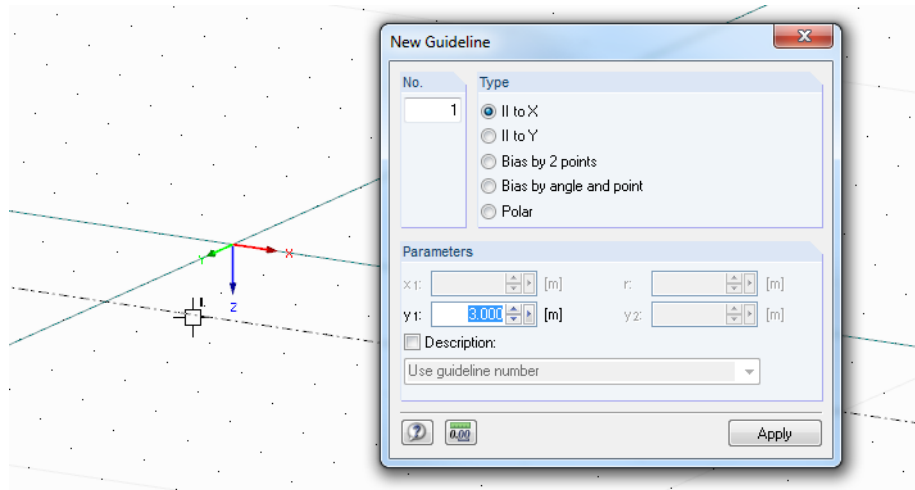
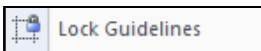


Figure 11.59: Creating a guideline graphically

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 (see figure below).



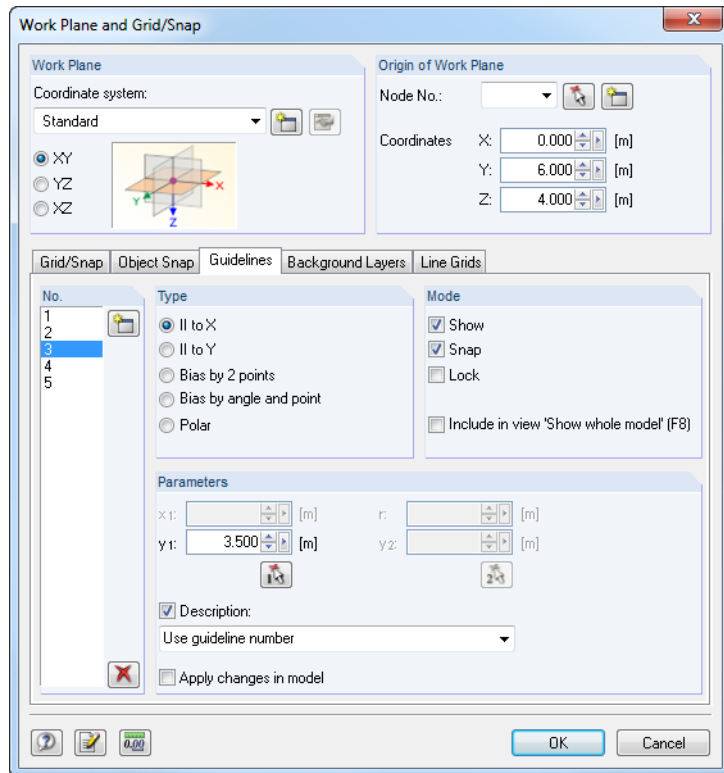


Figure 11.60: Dialog box *Work Plane and Grid/Snap*, dialog tab *Guidelines*

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.

### Lock guidelines

When guidelines are locked, they cannot be selected, edited or moved. In this way, they do not affect the graphical input of objects. Nevertheless, the snap function on the intersection points remains active.

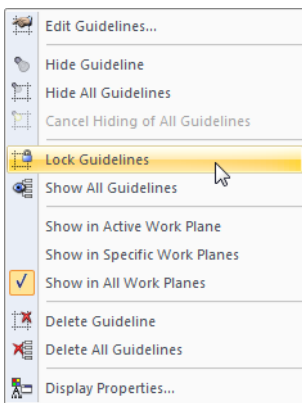
To lock or unlock all guidelines,

- right-click a guideline and select *Lock Guidelines* in the context menu,
- point to **Guidelines** on the **Edit** menu, and then select **Lock** or
- right-click *Guidelines* in the navigator and select *Lock Guidelines* in 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 301.



Context menu of guidelines

### Show guidelines

The *Display* navigator controls the graphical representation of guidelines in detail.

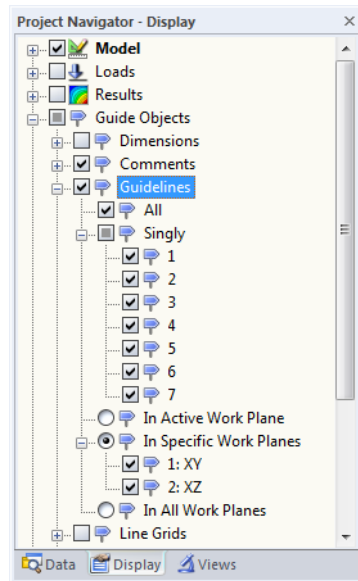


Figure 11.61: Guideline settings in the *Display* navigator

### 11.3.8 Line Grid

User-defined line grids help you to model structures consisting of girder grillages or grids. The intersection points of the grid represent definition points for nodes 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.

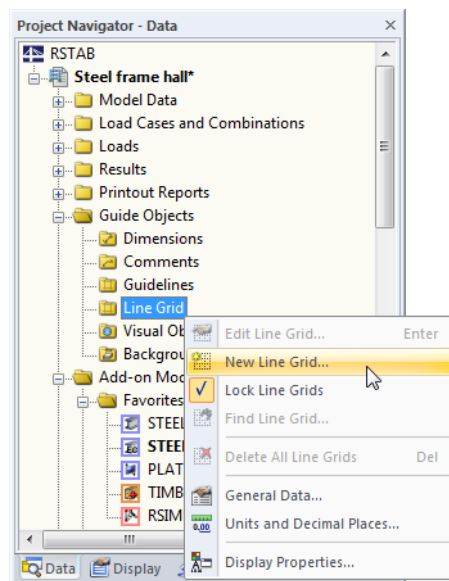


Figure 11.62: Context menu of *Line Grid* in *Data* navigator

The dialog box *Line Grid* appears where you can define the new grid.

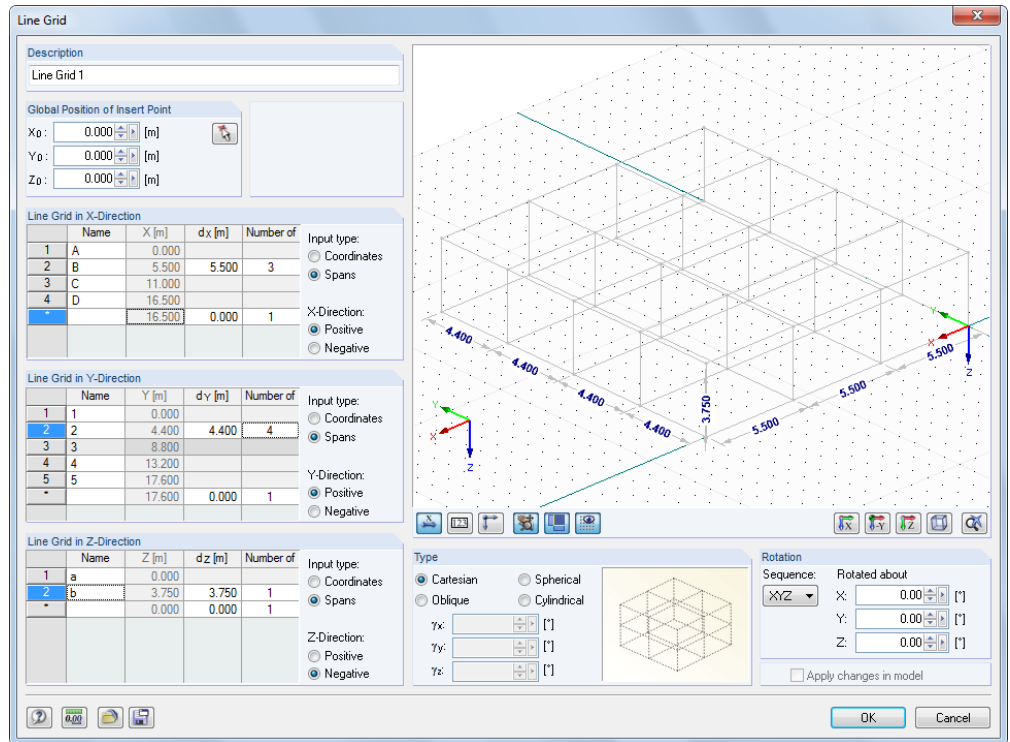


Figure 11.63: Dialog box *Line Grid*



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
- Oblique (grid that can be rotated for each axis about any rotation angle  $\gamma$ )
- Cylindrical

The small graphic to the right is interactive with the type specification.

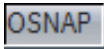
In the dialog sections *Line Grid in X-/Y-/Z-Direction*, enter the distances  $d$  and the *Number of spans* for each direction. The *Name* is preset but can be adjusted. It is also possible to enter the *Coordinates* of the distances or to adjust them subsequently.

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 RSTAB functions used to 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 33).



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 279).

### 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 RSTAB 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.

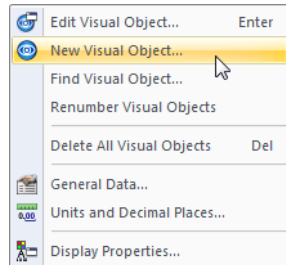


Figure 11.64: Context menu in the *Data* navigator, *Guide Objects* → *Visual Objects*

The dialog box *New Visual Object* opens where you have to specify the *Description* and *File Name*.

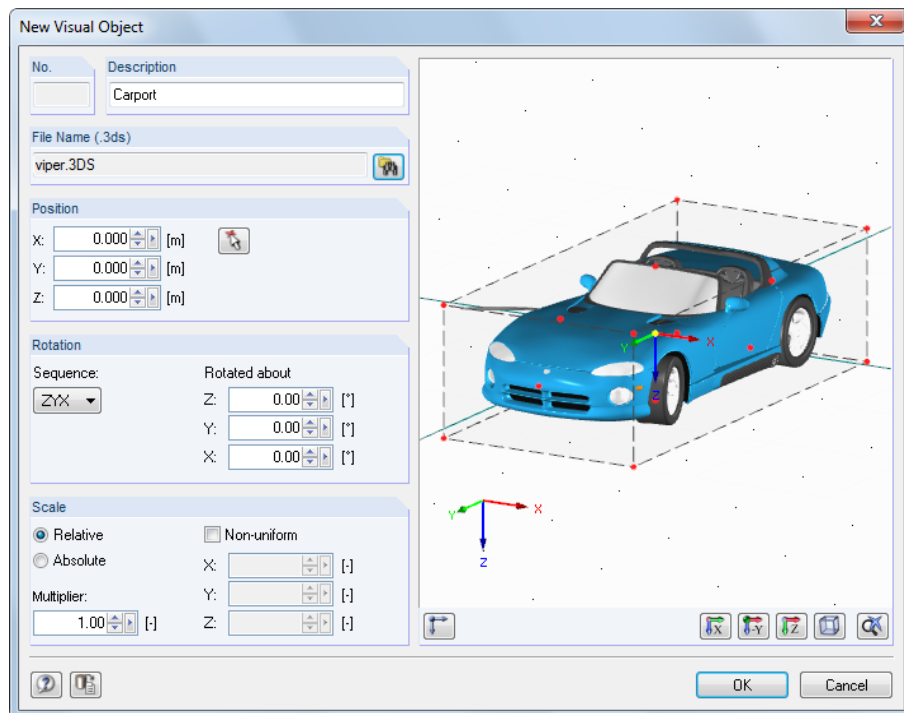


Figure 11.65: Dialog box *New Visual Object*



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 403) 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.

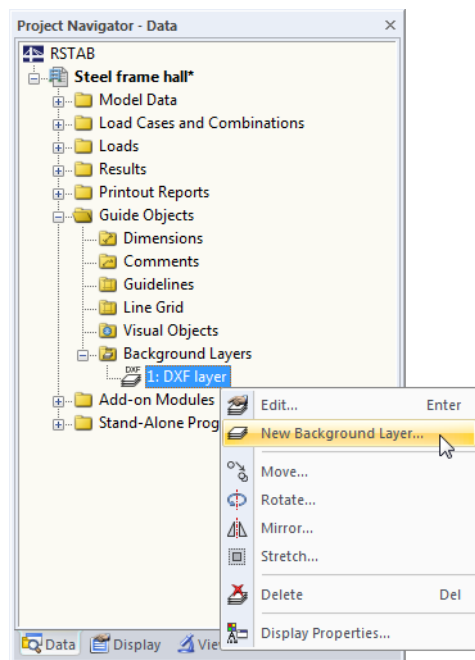


Figure 11.66: Context menu of *Background Layers* in the *Data* navigator

The Windows dialog box *Open* appears (see figure below). Enter the directory and the name of the DXF file.

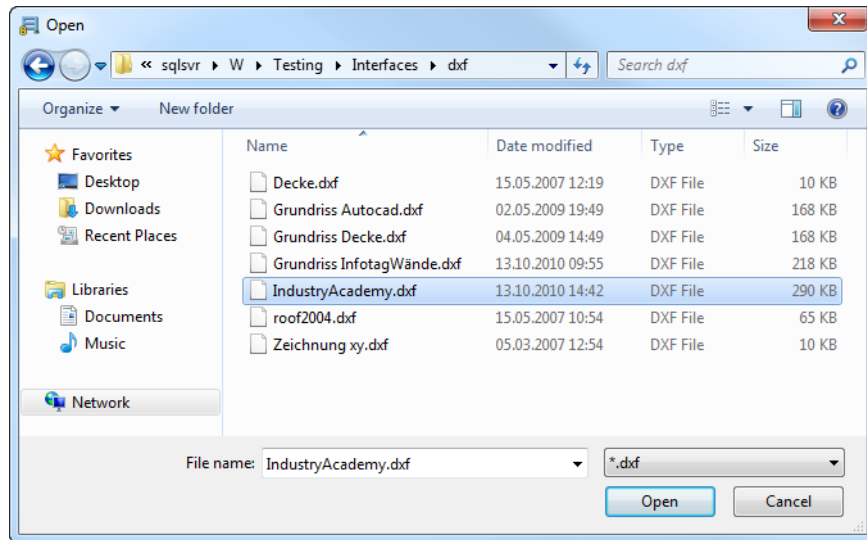
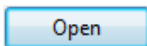


Figure 11.67: Dialog box *Open*



Click the [Open] button to access the dialog box *Background Layer*.

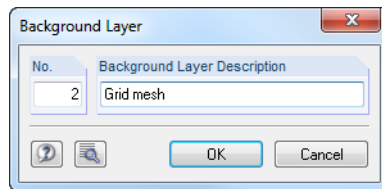


Figure 11.68: Dialog box *Background Layer*

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 403.

After clicking [OK] RSTAB imports the layer which appears gray in the background of the work window. In the gray line model, you can define nodes and members now.

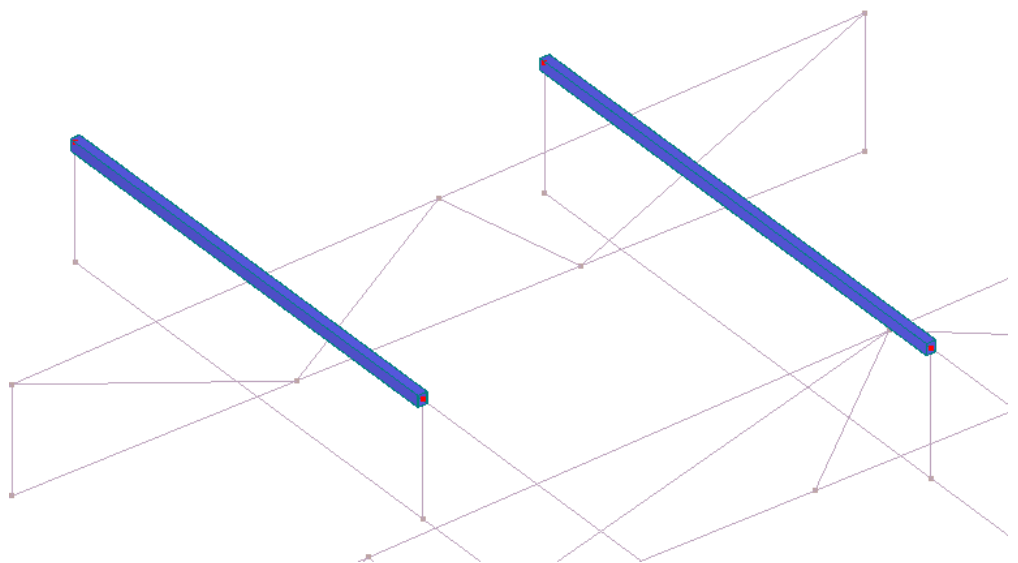


Figure 11.69: Defining members with background layer



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 can not only activate the snap but edit, delete or hide and display layers as well as create new layers.

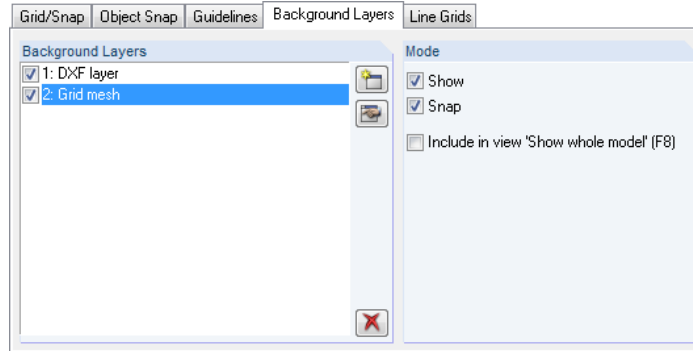


Figure 11.70: Dialog box *Work Plane and Grid/Snap*, tab *Background Layers* (dialog section)

### 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 11.66, page 297). You can also use the dialog tab *Background Layers* available in the dialog box for work plane settings (see Figure 11.70): 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 301.

### Display of background layers

The *Display* navigator controls the representation of background layers in detail.

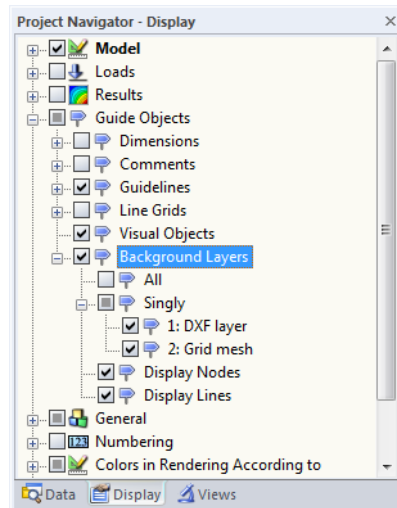


Figure 11.71: Background layer settings in the *Display* navigator

### 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.

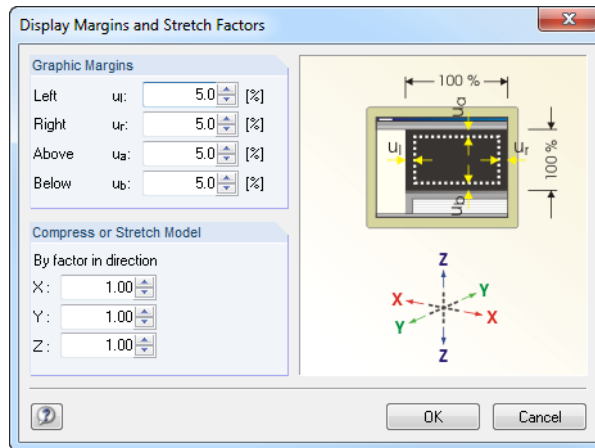
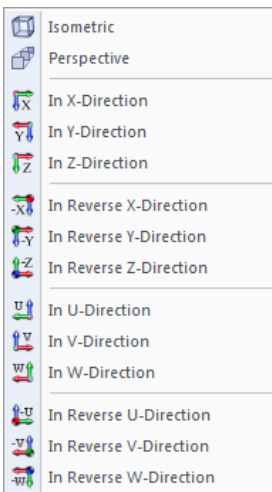


Figure 11.72: Dialog box *Display Margins and Stretch Factors*

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 produce a distortion of the model, use the *Scale* function available on the *Edit* menu (see chapter 11.4.5, page 307).



Buttons of menu item *Select View*

## 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 in the following help you to model new objects.

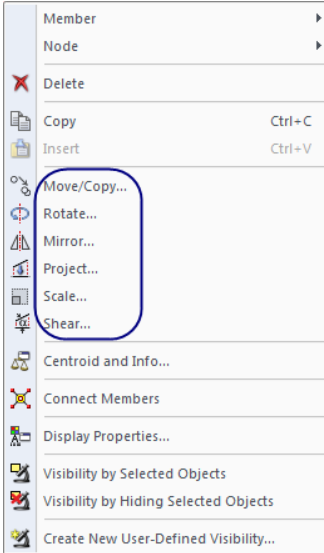
In addition, the present chapter describes how members can be divided, comments placed or numberings changed.



### 11.4.1 Move and Copy

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.



Context menu of selected objects



Figure 11.73: Button *Move and/or Copy*

The following dialog box appears:

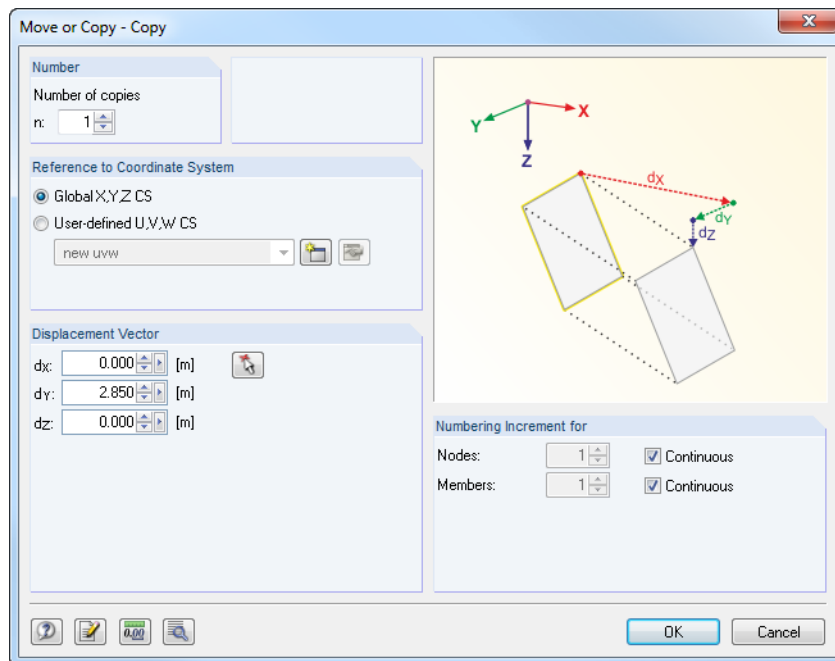


Figure 11.74: Dialog box *Move or Copy - Copy*

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 283). 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 and members 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

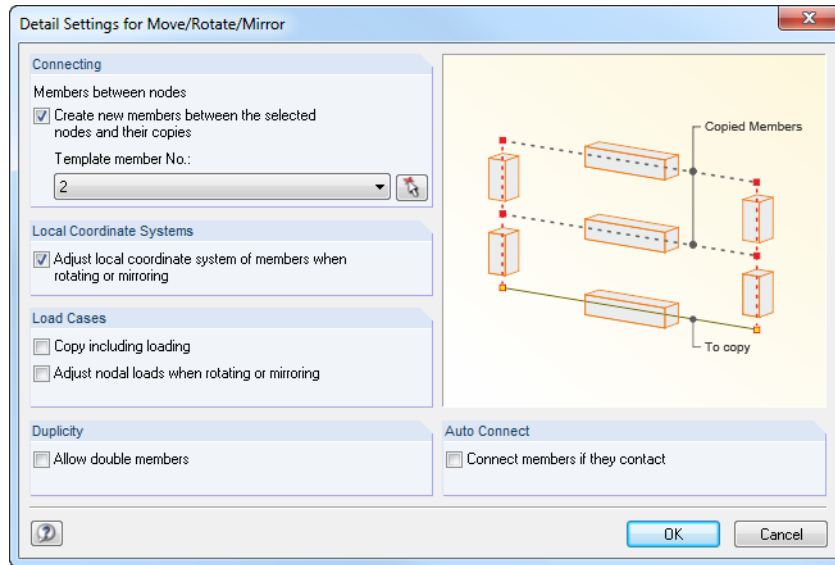


Figure 11.75: Dialog box *Detail Settings for Move/Rotate/Mirror*

### Connecting

You can create *new members* between the selected nodes and their copies.

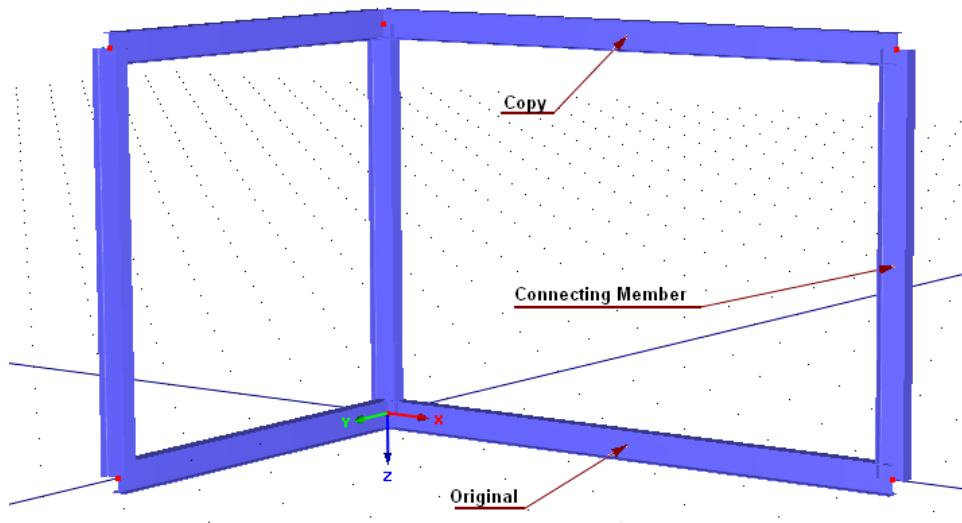


Figure 11.76: Copy with connecting members



When a *Template member* is selected in the list or graphic by using the [↖] function, its properties are used for the connecting members.

### Local coordinate system

You can adjust the local member coordinate system 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 4.7, page 80).

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 members, use the check box *Adjust nodal loads when rotating or mirroring*. When it is ticked, RSTAB 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.

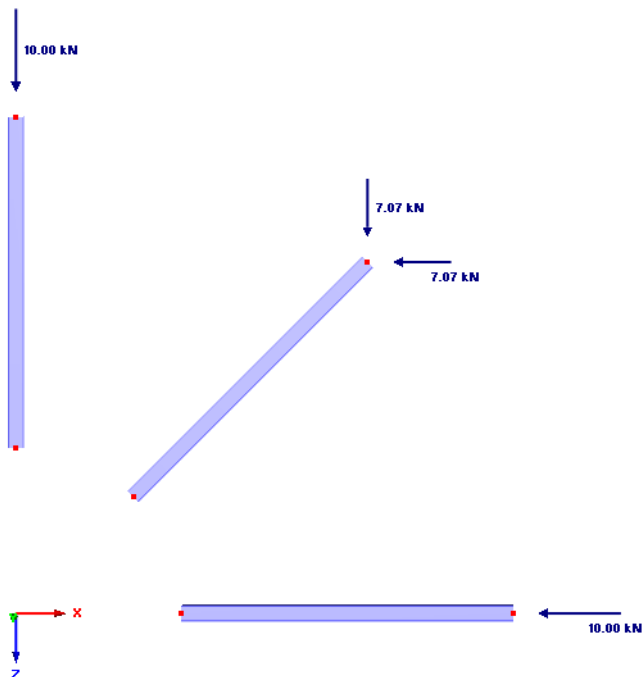


Figure 11.77: Adjusted nodal loads when rotating two times about 45°

### 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.

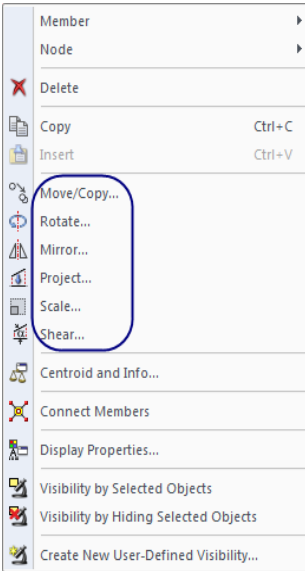
### Auto connect

Use the check box to decide if the copies of members will be connected automatically to the already existing 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 an axis, select **Rotate** 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

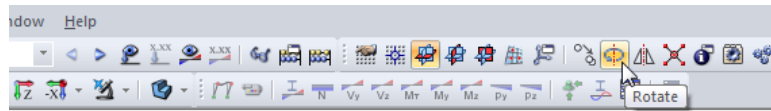


Figure 11.78: Button *Rotate*

The following dialog box appears:

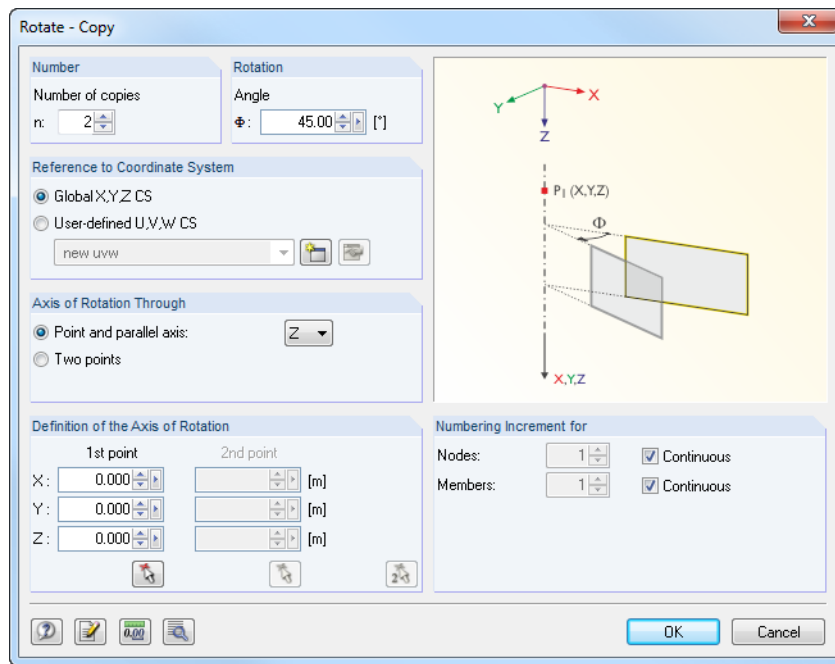


Figure 11.79: Dialog box *Rotate - Copy*

When the *Number* of copies is set 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 XYZ. 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*.

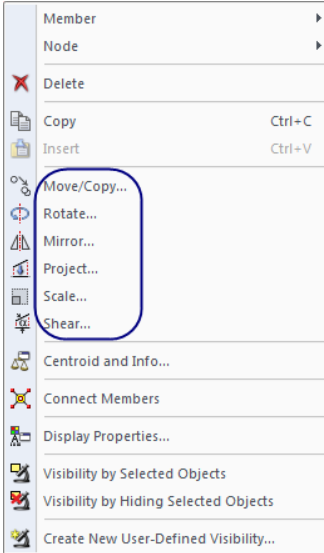
Use the [Edit] button shown on the left to open another dialog box with useful options that are described in chapter 11.4.1 on page 302.



### 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

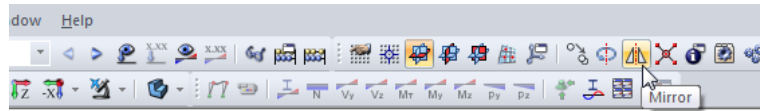


Figure 11.80: Button *Mirror*

The following dialog box appears:

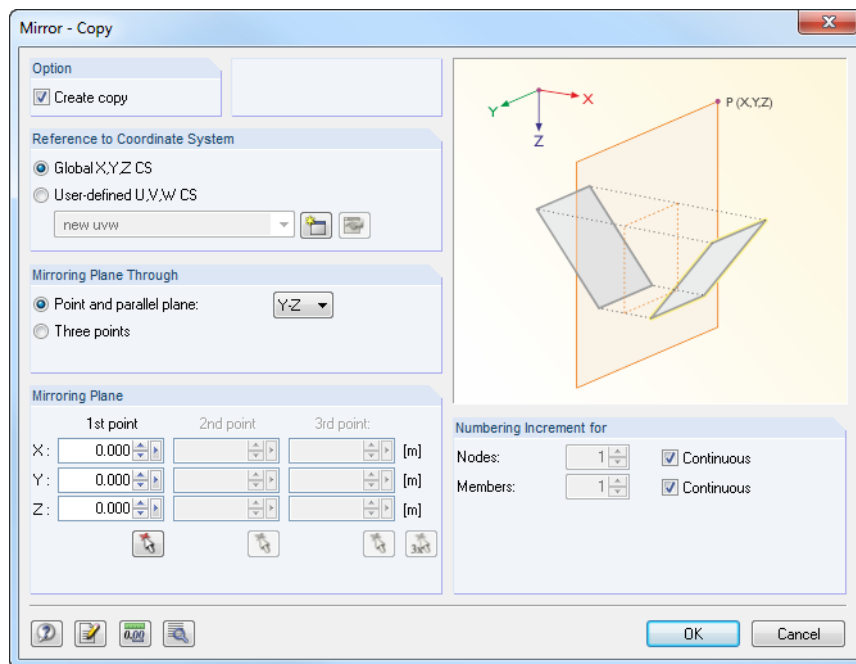


Figure 11.81: Dialog box *Mirror - Copy*

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 302.



### 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.

**Example:**

A member is projected in direction X on the plane YZ.

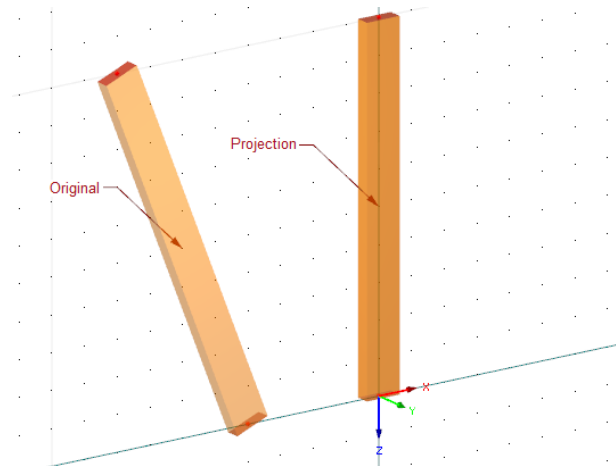
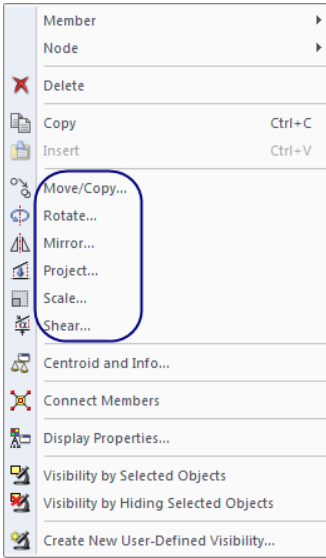


Figure 11.82: Original member and projected copy on plane YZ



Context menu of selected objects

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.

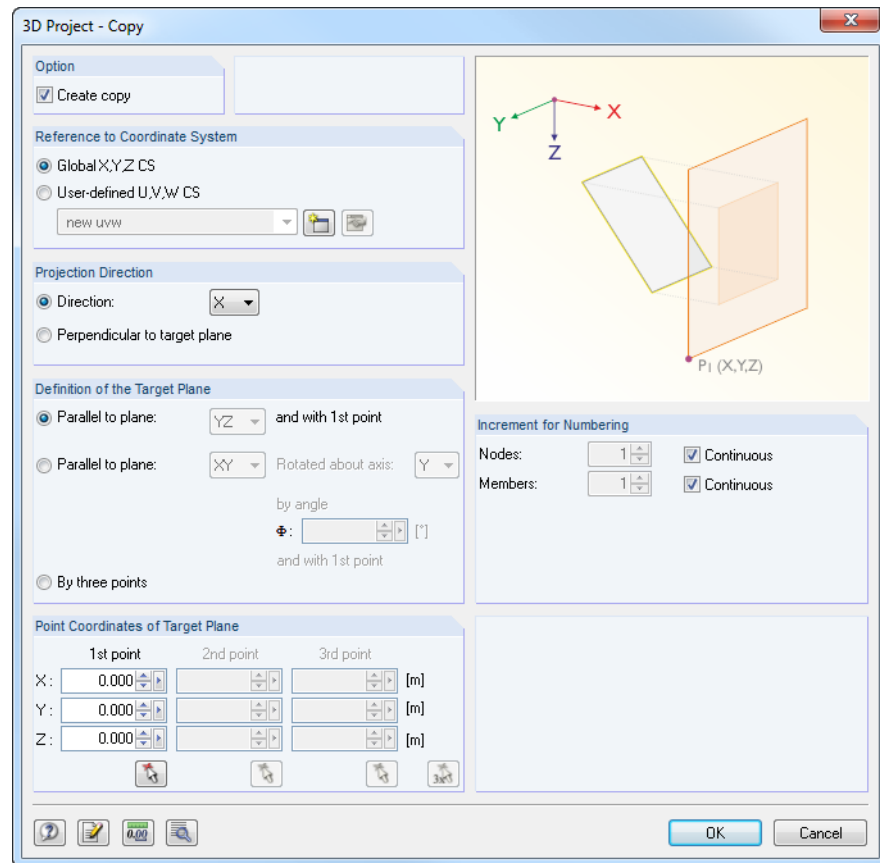


Figure 11.83: Dialog box 3D Project - Copy

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 XYZ. 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 *Numbering for Increment*.



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 302.

### 11.4.5 Scale

Use this function to scale selected objects in relation to a point.

#### Example:

A member is equally scaled starting from the origin in all three directions by the factor 2.

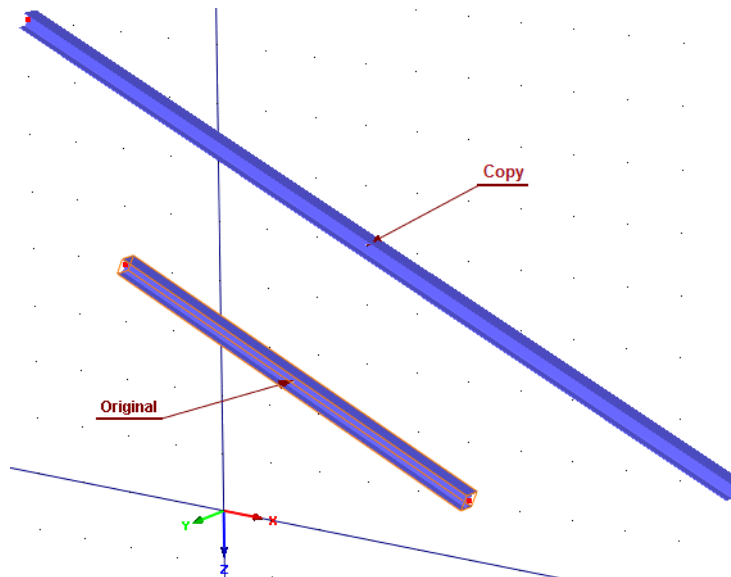


Figure 11.84: Original member and scaled copy



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 11.82).

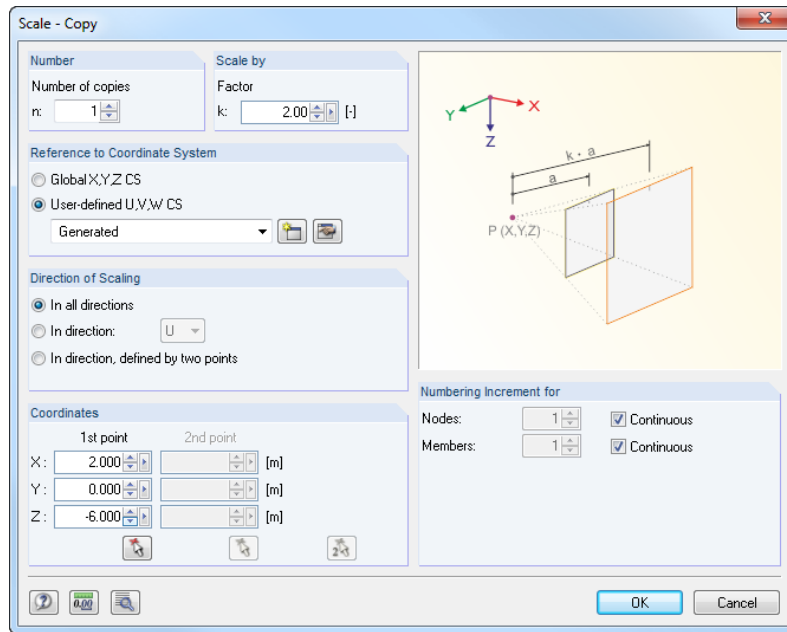


Figure 11.85: Dialog box *Scale - Copy*

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 <i>Coordinates</i> .
	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 <i>Coordinates</i> dialog section.
	In direction that is defined by two points	In the dialog section <i>Coordinates</i> , specify a vector by entering two points. Objects will be scaled into the vector direction.

Table 11.7: Dialog section *Direction of Scaling*

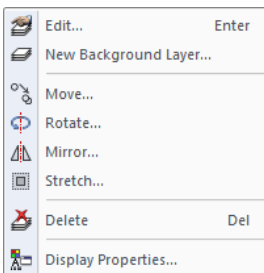
If a copy is created, you can influence the numbering of new objects in the dialog section *Numbering Increment for*.



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 302.

It is also possible to scale background layers. To open the corresponding dialog box, 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 scaling factor in the dialog box *Stretch Background Layer*.



Context menu of background layers

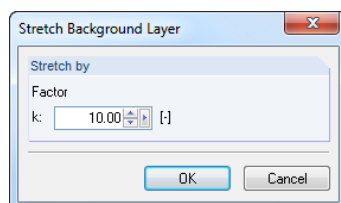


Figure 11.86: Dialog box *Stretch Background Layer*



### 11.4.6 Shear

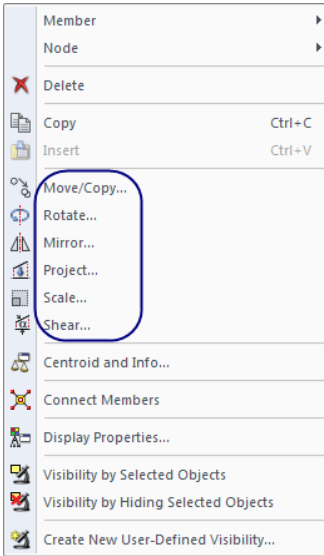
The function rotates objects about an axis and adjusts only the coordinates of one single direction. You can use the shear function for example to shift horizontal members into the inclination plane of a roof. The member lengths will be adjusted, the horizontal components of the coordinates remain unchanged.

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

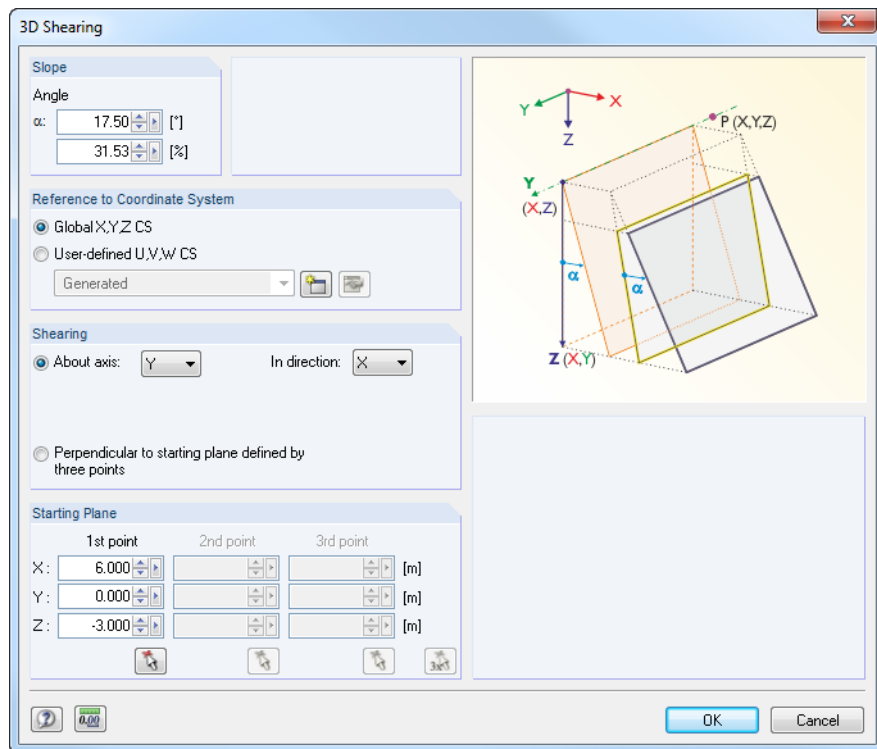


Figure 11.87: Dialog box 3D Shearing

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 Members

Members can be divided quickly: Right-click the member and select *Divide Member* in the context menu.

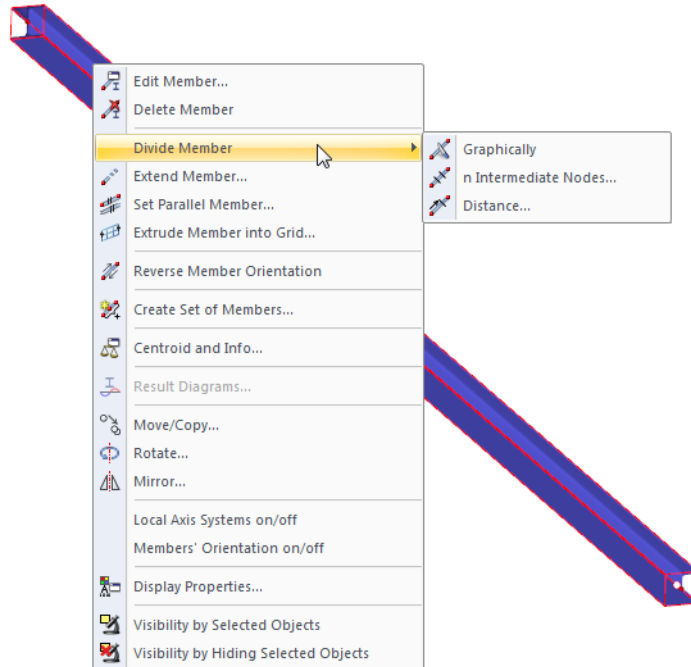


Figure 11.88: Context menu *Divide Member*

The menu item offers three division options.

#### Graphically

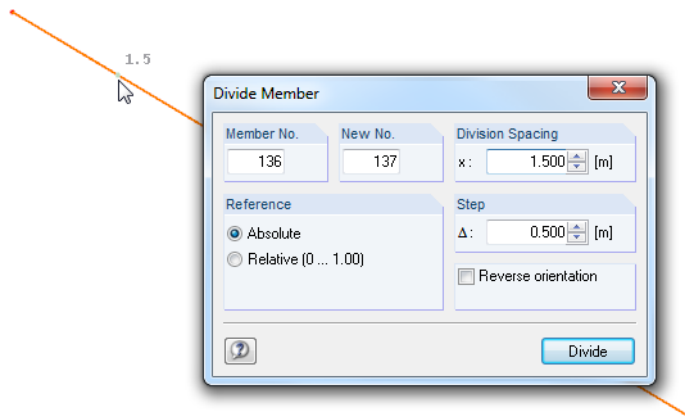


Figure 11.89: Dialog box *Divide Member*

The dialog box *Divide Member* opens. When you move the pointer along the member, 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, define the member that you want to divide in the input field *Member No.*. Then, enter the number of the new member in the input field *New No.*. If you want to relate the division spacing to the member end, you can change the member orientation with the check box *Reverse orientation*.

### n Intermediate Nodes

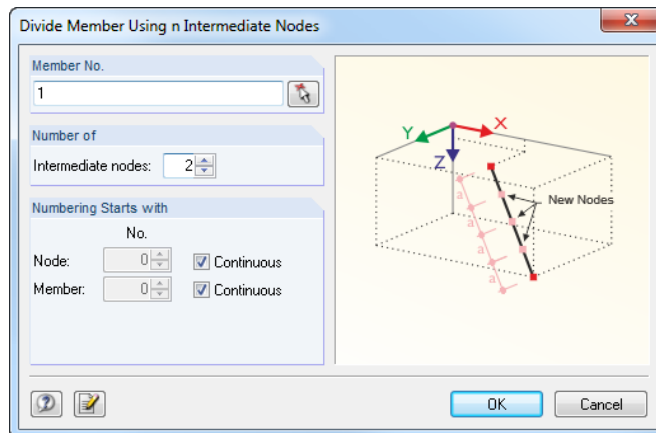


Figure 11.90. Dialog box *Divide Member Using n Intermediate Nodes*

Use this function to divide the member equally into several member parts. In the dialog section *Number of*, you can define the number of *Intermediate nodes* needed for the line division. In the dialog section *Numbering Starts with*, you can influence the numbering of new nodes and members.

### Distance

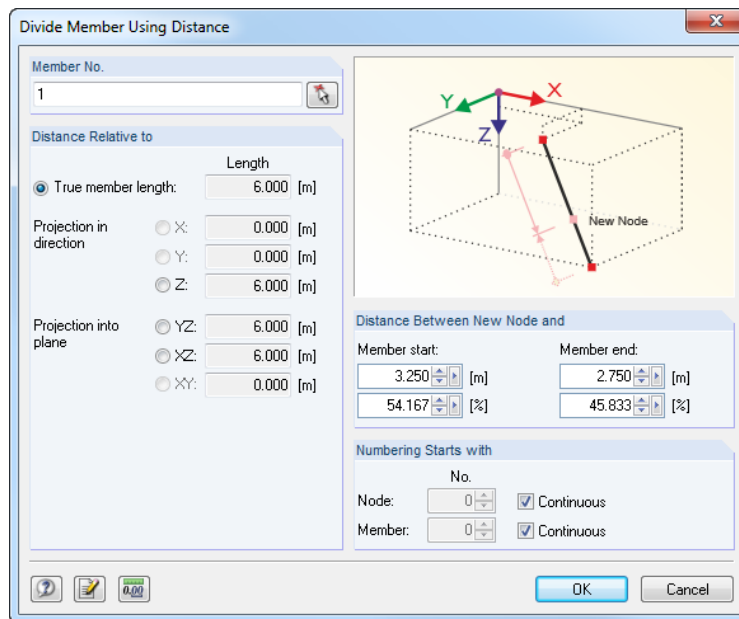


Figure 11.91: Dialog box *Divide Member Using Distance*

Use this function to generate a division node on a particular location of the member. Settings in the dialog section *Distance Relative to* control the reference of the division distance. The distance can be related to the real member length (normal case) or to a projection.

The *Distance Between New Node and* start or end node of the member 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 member orientation. The orientations and axis systems of members can be switched on and off in the context menu or the *Display* navigator (see Figure 4.70, page 79 and Figure 4.72, page 80).

The dialog section *Numbering Starts with* controls the numbering of new objects.

### 11.4.8 Connect Members

Use this function to connect members that cross each other but do not have a common node.

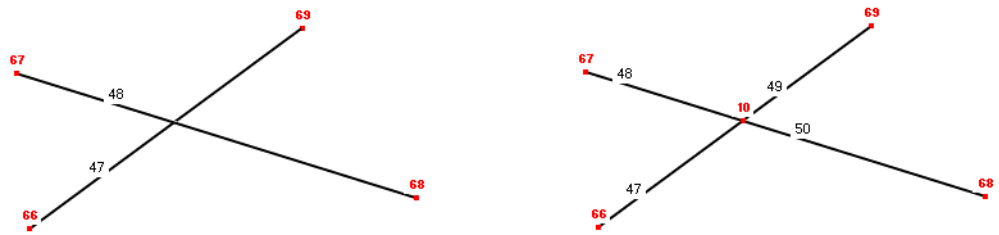


Figure 11.92: Original on the left (intersecting, unconnected members) and result on the right (connected members)



To access the corresponding function, select **Connect Members** on the **Tools** menu or use the toolbar button shown on the left.

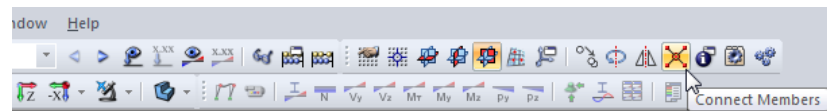


Figure 11.93: Button *Connect Members*

Go into the work window and draw a window across the zone where you want to connect the members. It is not necessary to catch the objects completely.

The *Auto Connect* function is preset for the graphical definition of new members, as shown in the figure below. But connection nodes will only be created when members are connected to other members, that means when members end on members. So, when you define crossing diagonals, no intersection node will be generated.



In the *New Member* dialog box appearing when members are defined graphically, you can use the [Details] button to determine if members are connected automatically when they are generated.

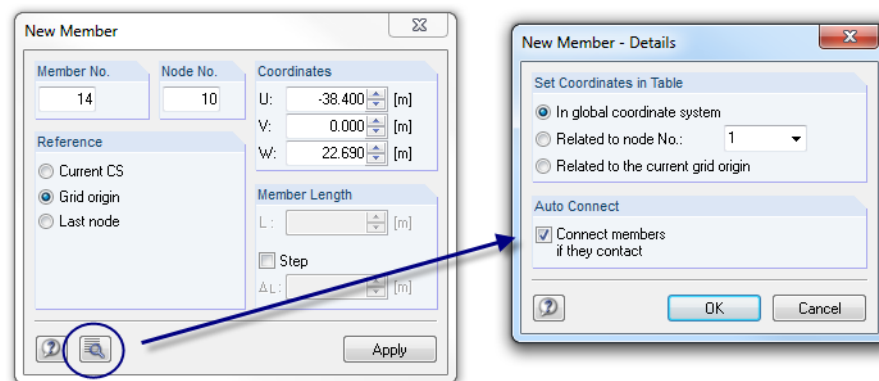


Figure 11.94: Dialog box *New Member - Details*

### 11.4.9 Merge Members

Members connected to one another can be unified to be one single member. This function is only available in the context menu of division nodes. Right-click the division node to open its context menu.

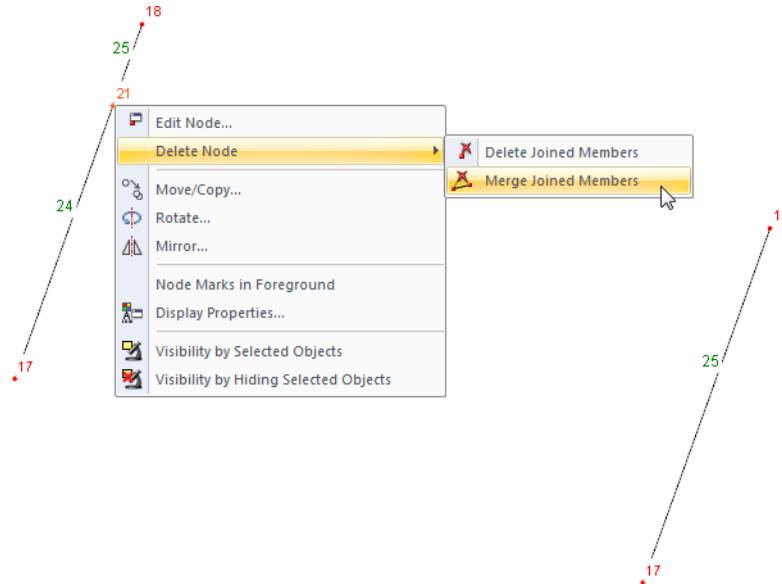


Figure 11.95: Context menu items *Delete Node* → *Merge Joined Members* with result (right)

The context menu offers extended options for the function *Delete Node* whereas the [Del] key simply deletes the selected node and consequently the joined members. But these special options are only provided for nodes to which exactly two members are connected.

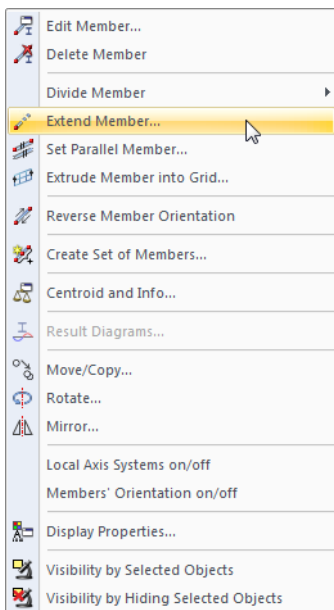
In case the members do not lie on a straight line, RSTAB will create a new member between the edge nodes when merging.

### 11.4.10 Extend Members

Use this function to adjust the length of a member or to extend the member until it reaches another one.

To access this editing function, use the member context menu shown on the left.

The dialog box *Extend Member* opens.



Member context menu

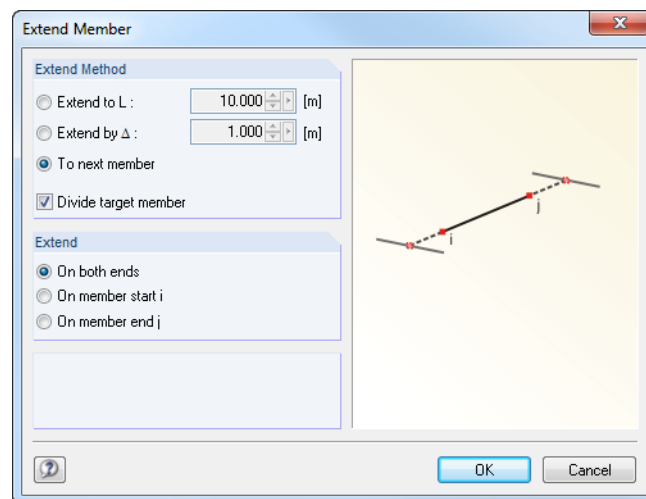


Figure 11.96: Dialog box *Extend Member*

The dialog section *Extend Method* offers three options:

- *Extend to L* changes the total length of the member to a dimension that you specify in the input field.
- *Extend by  $\Delta$*  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* to extend the member to the nearest member that will produce an intersection with the extended straight line of the member. When the check box for *Divide target member* is ticked, the members 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 member center, or extend the member on both sides by the value  $\Delta$  or until the next two members are reached. Alternatively, use the options *On member start i* and *On member end j* to adjust the length of the member on only one end.

Use the Display navigator or the context menu of the member to display the member orientations (see Figure 4.70, page 79).

### 11.4.11 Join Members

Contrary to connecting members (see chapter 11.4.8, page 312), 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:

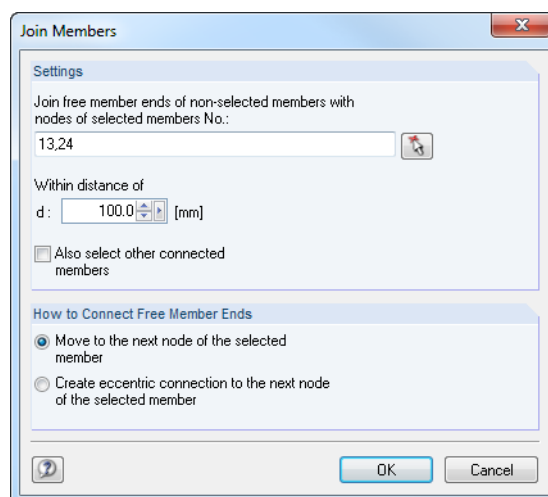


Figure 11.97: Dialog box *Join Members*



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 [ $\wedge$ ] function. The input field below specifies the *distance*, that means the circumference where RSTAB looks for free member ends. If the check box for *Also select other connected members* is ticked, RSTAB 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 RSTAB 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 member and divide it by an intermediate node (see chapter 11.4.7, page 310).

To access the corresponding function,

point to **Model Data** on the **Insert** menu, select **Nodes** and then 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:

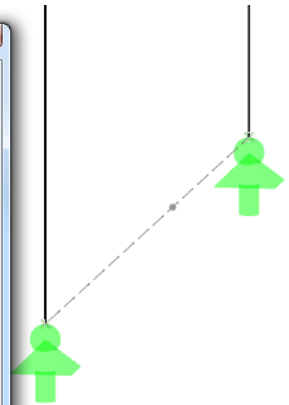
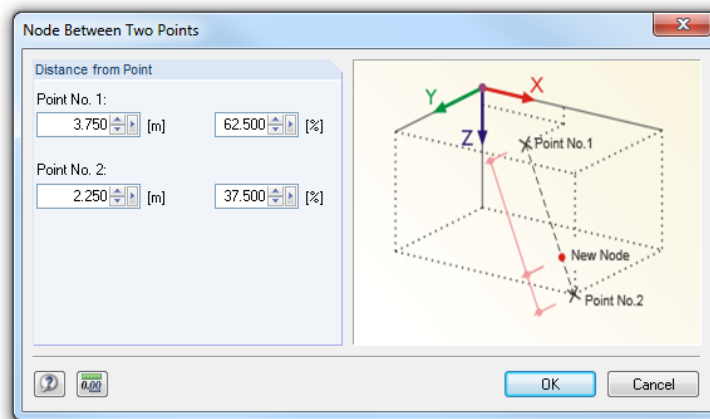
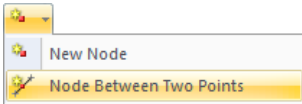
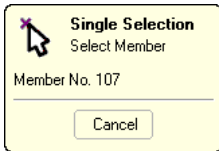


Figure 11.98: Dialog box *Node Between Two Points*

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:

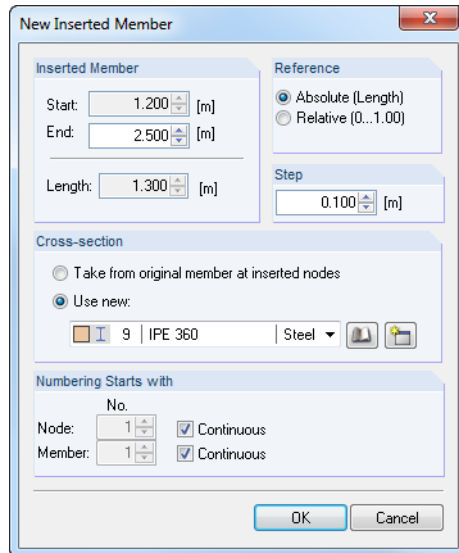


Figure 11.99: Dialog box *New Inserted Member*

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:

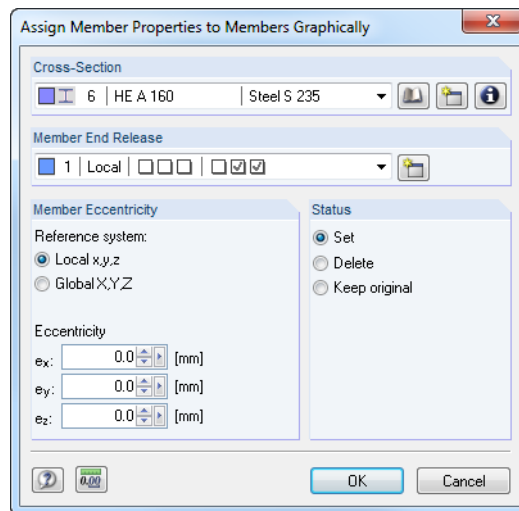


Figure 11.100: Dialog box *Assign Member Properties to Members Graphically*



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 4.4, page 64).

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 4.5, page 70).

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 4.50, page 65). 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 in the model may result in moment peaks. To open the dialog box for modeling corners close to reality using fillet members,

select **Create Round or Angled Corner** on the **Tools** menu.

It is not necessary to select both members previously. The following dialog box appears:

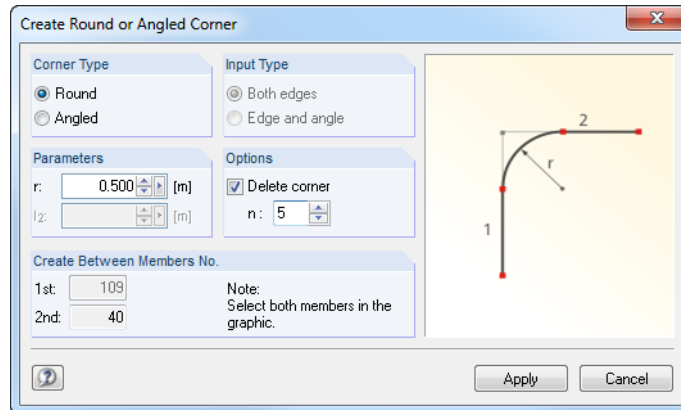


Figure 11.101: Dialog box *Create Round or Angled Corner*

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*.

When rounding corners, use the dialog section *Options* where you can define the number of members  $n$  required to represent the rounding as a polygonal chain (minimum 3). If the check box for *Delete corner* is ticked, RSTAB deletes the corner nodes together with the extensions of the original members overlapping in the corner zone.

Select both corner members by mouse-click in the work window without closing the dialog box. You can see the member numbers shown in the dialog section *Create Between Members No.*

### 11.4.16 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

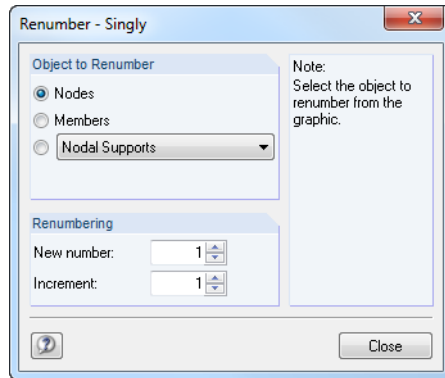


Figure 11.102: Dialog box *Renumber - Singly*

In the dialog section *Object to Renumber*, you decide whether nodes, 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 RSTAB can allocate only free numbers that are not yet assigned.

### Automatically

First, select the nodes and members (see chapter 11.2.1, page 271) whose numbering you want to adjust. Then, open the following dialog box.

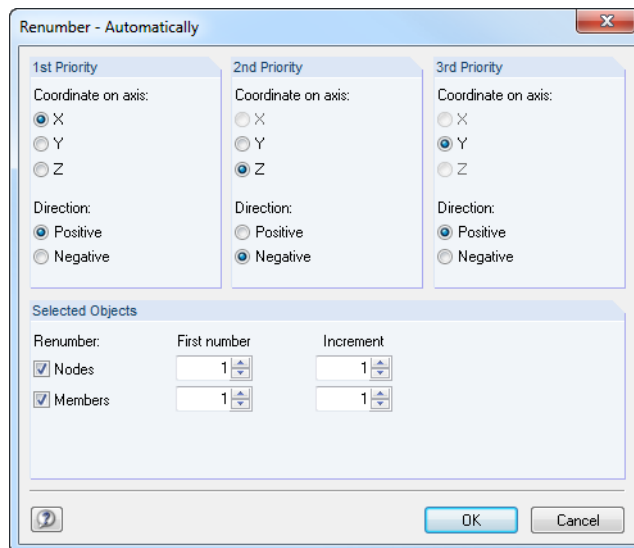


Figure 11.103: Dialog box *Renumber - Automatically* for nodes and members

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 members) with the smallest 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 *Selected Objects* controls which nodes 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, RSTAB 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.

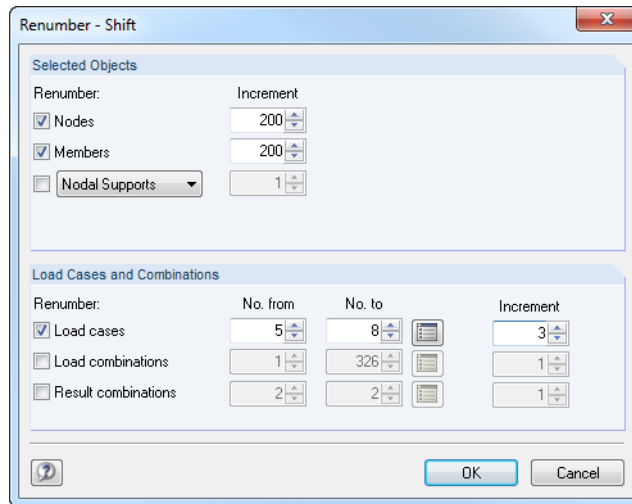


Figure 11.104: Dialog box *Renumber - Shift*



In the dialog section *Selected Objects*, define the objects that you want to renumber: In addition to nodes and members you can select other objects of the model in a list. 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 24). 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.

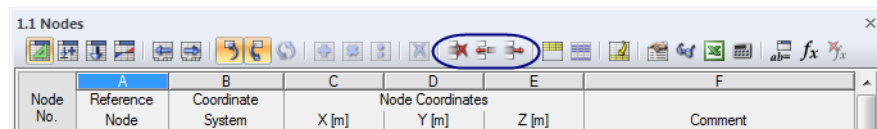


Figure 11.105: Buttons for several editing functions in the table toolbar

Alternatively, use the context menu in the table to access the functions.

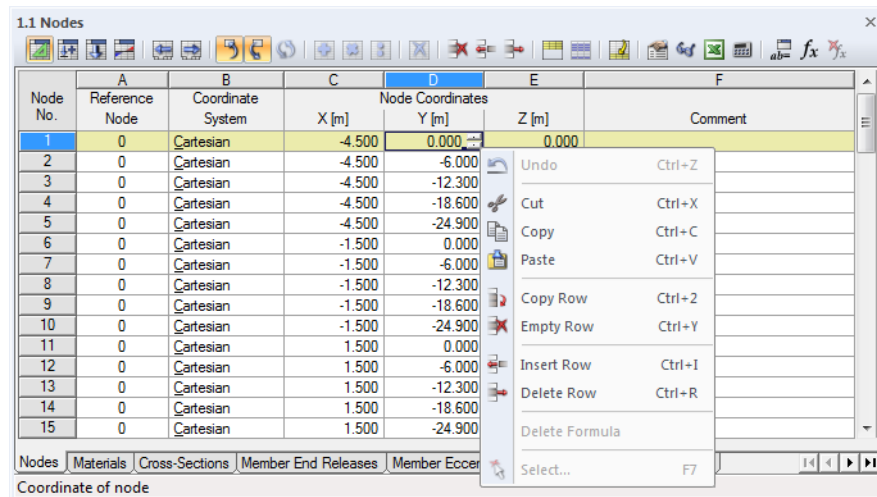


Figure 11.106: Editing functions in the table context menu

### 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	X [m]	Y [m]	Z [m]
Cartesian	12.500	0.000	-7.094
Cartesian	18.750	0.000	-6.546
Cartesian	22.000	0.000	-6.261
Cartesian	25.000	0.000	-6.000
Cartesian	0.000	-5.000	0.000
Cartesian	25.000	-5.000	0.000
Cartesian	0.000	-5.000	-6.000
Cartesian	3.000	-5.000	-6.261

Figure 11.107: 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.

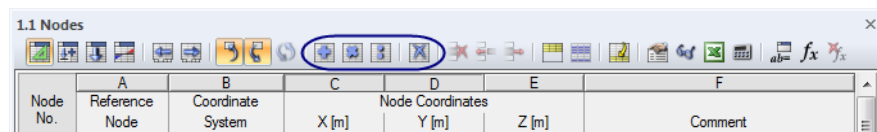


Figure 11.108: Buttons for several selection functions in the table toolbar

Alternatively, use the context menu in the table to access the functions.

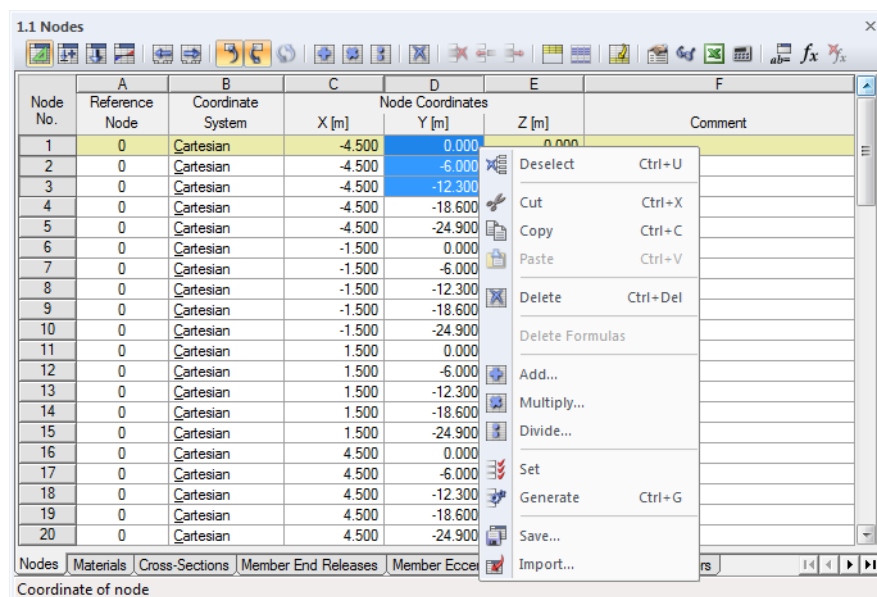


Figure 11.109: Selection functions in the table context menu

### 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 if 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 value of topmost selected cell to all cells in selection
Generate [Ctrl+G]	Used for cells with numerical values. Generates cells between first and last selected cell by interpolation of both basic 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).

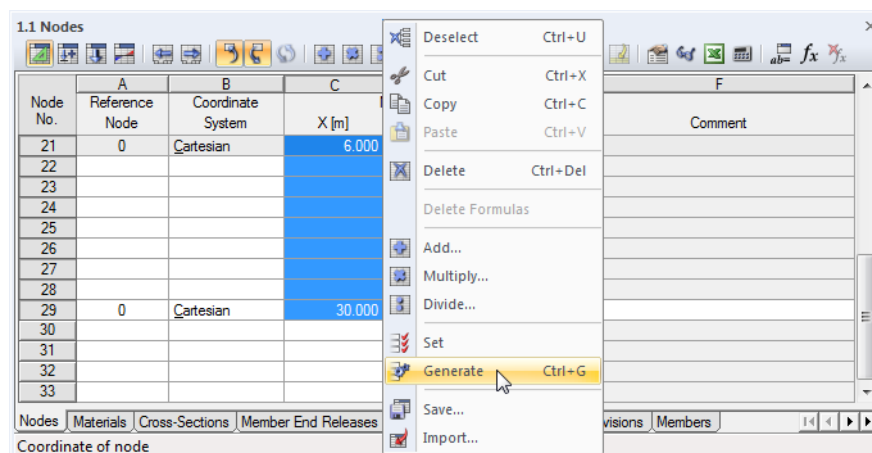


Figure 11.110: Context menu of selection



After the *Generate* function has been applied, the intermediate cells are filled with interpolated values.

Node No.	A	B	Node Coordinates			Comment
	Reference Node	Coordinate System	X [m]	Y [m]	Z [m]	
21	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	0	Cartesian	30.000	0.000	0.000	

Figure 11.111: 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 as well as

**Optimize Load Data** on the **Table** menu.



Some of the view functions can be accessed in the toolbar of the table.

Figure 11.112: Buttons for several view functions in the table toolbar

#### Functions

Function	Effect
	Hides all empty table rows
	Shows only selected rows
	Shows only objects selected in graphic
	In addition to loads, the associated model objects (members, nodes, sets of members) are selected in the graphic. Only available in loads tables 3.
	Summarizes objects with same loads in one single table row in loads tables
	Lists loads for each object individually

Result filter 	Table output can be restricted to particular result types (see chapter 11.5.5, page 328).
Info about cross-section 	Shows characteristic values of current cross-section
Show result diagrams 	Displays results of selected member graphically in new window (see chapter 9.5, page 208)
Colored relation scales 	Switches display of red and blue bars in 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.

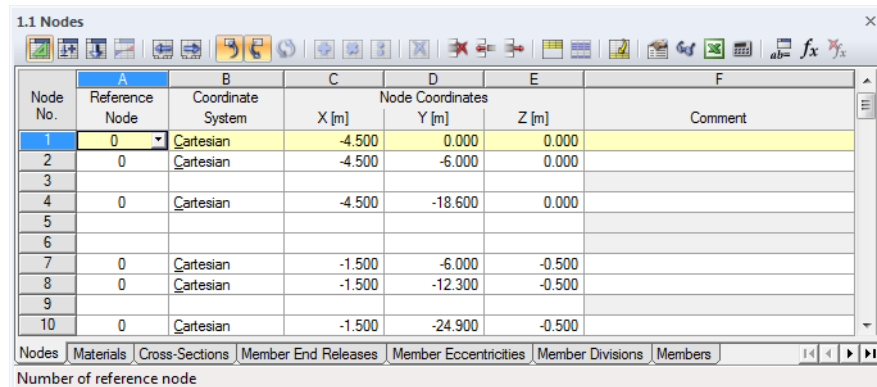


Figure 11.113: Table with empty rows



Use the button *Only Filled Rows* to hide all empty rows.

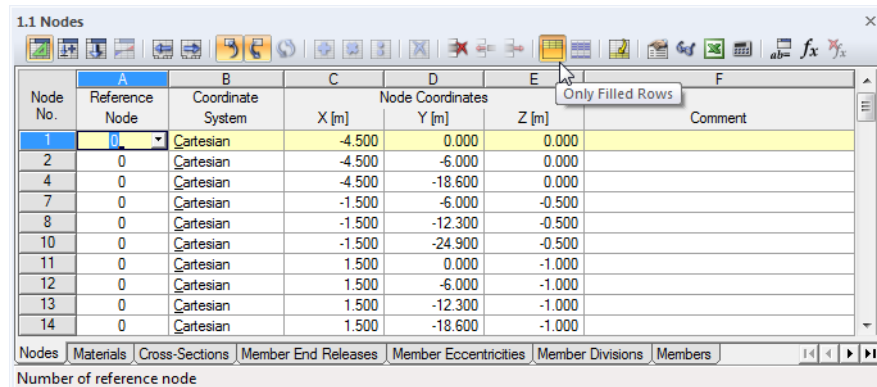


Figure 11.114: Table without empty rows

### 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 access the setting options, select **Settings** on the **Table** menu.

To activate and deactivate the synchronization of the selection, you can also use the table toolbar buttons.

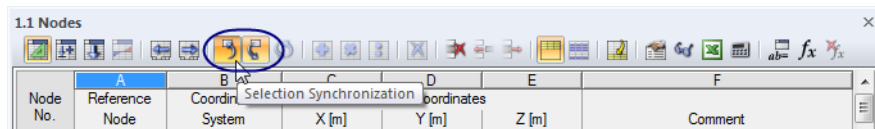


Figure 11.115: Buttons Selection Synchronization

#### Functions



Function	Effect
Colors	Opens the dialog box <i>Colors</i> (see Figure 11.116). Colors of individual table objects can be adjusted separately.
Fonts	Opens the dialog box <i>Font</i> (see Figure 11.116). 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

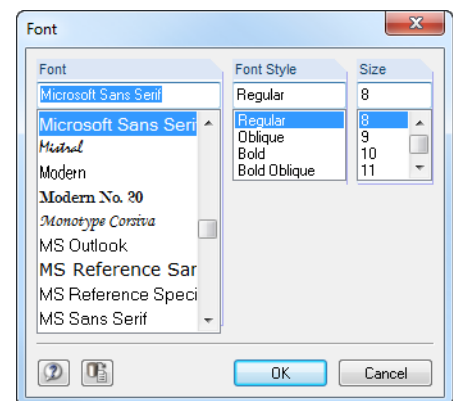
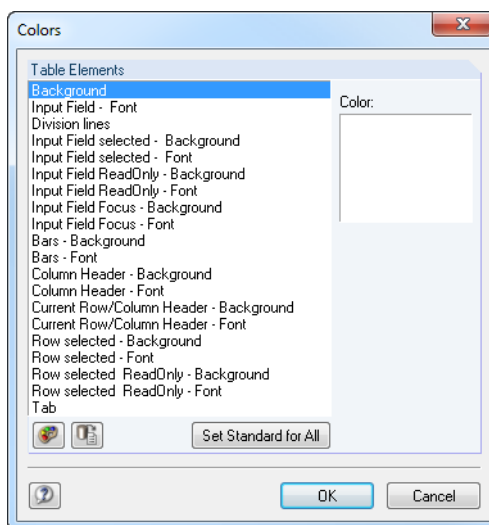


Figure 11.116: Dialog boxes *Colors* and *Font*

## 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 support forces of result combinations (see chapter 8.4, page 194).

### Access to filter functions

To access the filter functions,

point to **View** on the **Table** menu and select **Result Filter**

or use the button in the table toolbar shown on the left.

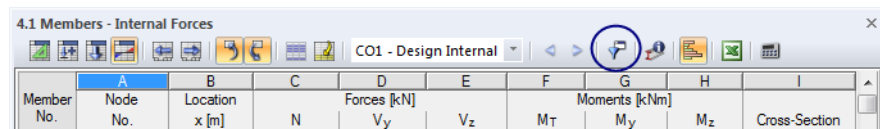


Figure 11.117: Button *Result Filter*

The following dialog box appears:

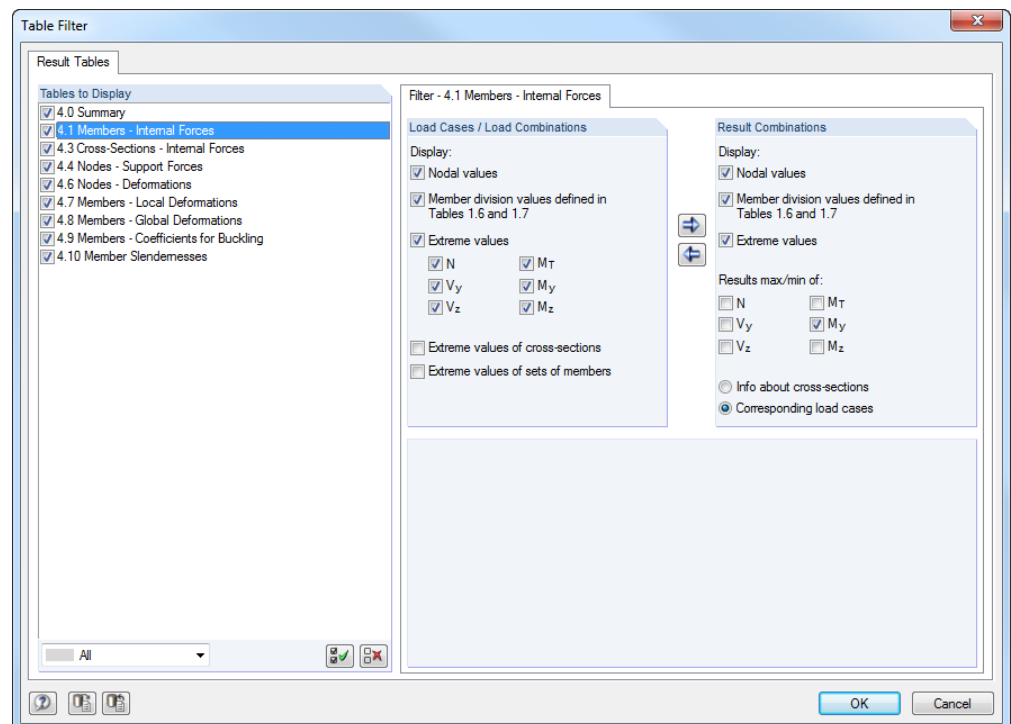


Figure 11.118: Dialog box *Table Filter*

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 / 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 4.6) 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 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 11.118 result in the following results table 4.1 Members - Internal Forces.

Member No.	A Node No.	B Location x [m]	C	D Forces [kN]			G M <sub>T</sub>	H Moments [kNm]		I M <sub>z</sub>	J Corresponding Load
				N	V <sub>y</sub>	V <sub>z</sub>		M <sub>y</sub>	M <sub>z</sub>		
11	16	0.000	max M <sub>y</sub>	-15.09	0.02	21.24	0.00	<b>-20.12</b>	0.02	CO1	
			min M <sub>y</sub>	-21.08	1.45	31.62	-0.02	<b>-30.03</b>	-0.11	CO4	
		2.233	max M <sub>y</sub>	-15.09	0.02	21.24	0.00	<b>-20.12</b>	0.02	CO1	
			min M <sub>y</sub>	-21.08	1.45	31.62	-0.02	<b>-30.03</b>	-0.11	CO4	
			max M <sub>y</sub>	-15.77	1.27	10.70	-0.01	<b>18.63</b>	-4.64	CO15	
			min M <sub>y</sub>	-19.33	0.01	6.86	-0.01	<b>9.99</b>	-0.02	CO2	
	3.350	max M <sub>y</sub>	-15.78	0.00	0.06	-0.02	<b>24.64</b>	-5.39	CO14		
		min M <sub>y</sub>	-19.34	0.00	-0.27	-0.01	<b>13.67</b>	-0.02	CO2		
		max M <sub>y</sub>	-15.78	0.00	0.06	-0.02	<b>24.64</b>	-5.39	CO14		
		min M <sub>y</sub>	-19.34	0.00	-0.27	-0.01	<b>13.67</b>	-0.02	CO2		
	4.467	6.700	max M <sub>y</sub>	-15.77	-1.27	-10.60	-0.02	<b>18.75</b>	-4.63	CO14	
			min M <sub>y</sub>	-19.33	-0.01	-7.41	-0.01	<b>9.38</b>	-0.01	CO2	
		3.350	max M <sub>y</sub>	-12.38	-2.30	-21.83	0.02	<b>-20.58</b>	-0.27	CO8	
			min M <sub>y</sub>	-22.71	-0.04	-31.68	0.00	<b>-31.17</b>	0.04	CO3	
6.700	max M <sub>y</sub>	-12.38	-2.30	-21.83	0.02	<b>-20.58</b>	-0.27	CO8			
	min M <sub>y</sub>	-22.71	-0.04	-31.68	0.00	<b>-31.17</b>	0.04	CO3			

Figure 11.119: Results filtered by nodal values, division points and extreme values M<sub>y</sub>

Table column H shows the maximum and minimum bending moments M<sub>y</sub> 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<sub>y</sub> and Min M<sub>y</sub> 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 RSTAB input table. The programs involved must be opened. It is also possible to export the current RSTAB table all or part to Excel or Open Office.org Calc.

**Access to import and export function**

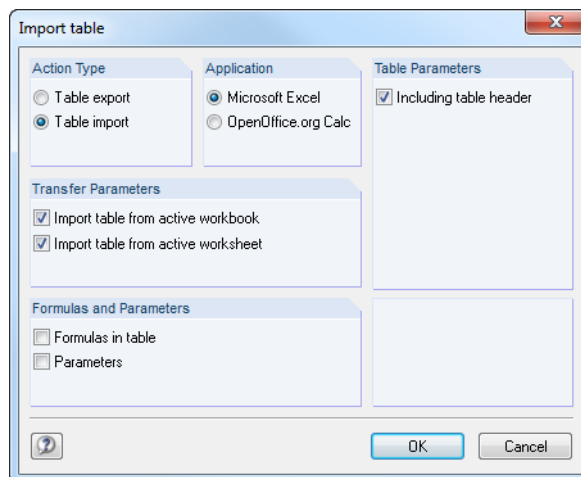
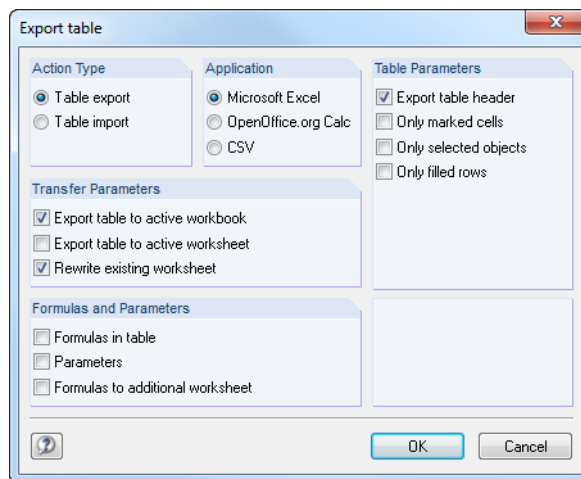
To apply the import or export function, click the button [Export/Import Table] in the table toolbar.



Node No.	A	B	D Node Coordinates		F Comment
	Reference Node	Coordinate System	X [m]	Y [m]	

Figure 11.120: Button Export/Import Table in the table toolbar

Use this button to open the Export table and Import table dialog box (see figures below).

Figure 11.121: Dialog box *Import table*Figure 11.122: Dialog box *Export table*

## Import table

The workbook from MS Excel or OpenOffice must have been opened before the import starts. If headers exist 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 RSTAB 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 RSTAB 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.



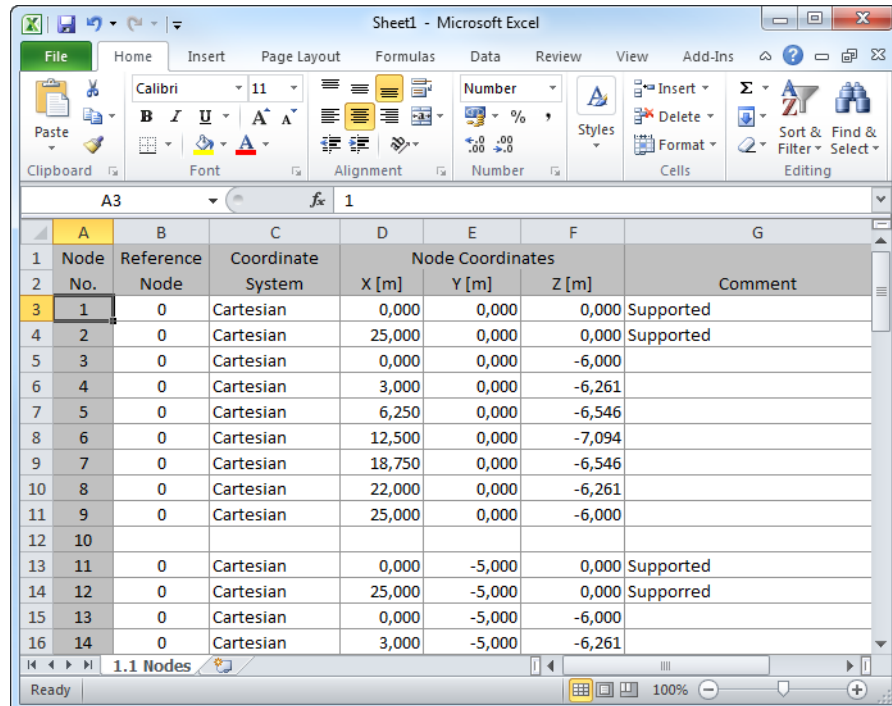
If you want to import only particular parts of the worksheet, the copy function is recommended: Select the relevant area in the Excel table and copy it to the clipboard with [Ctrl]+[C]. Then, place the pointer into the corresponding cell of the RSTAB table and insert the contents of the clipboard with [Ctrl]+[V].

## Export table

To export RSTAB 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 4.3, page 63).

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
1	Node	Reference	Coordinate	Node Coordinates			
2	No.	Node	System	X [m]	Y [m]	Z [m]	Comment
3	1	0	Cartesian	0,000	0,000	0,000	Supported
4	2	0	Cartesian	25,000	0,000	0,000	Supported
5	3	0	Cartesian	0,000	0,000	-6,000	
6	4	0	Cartesian	3,000	0,000	-6,261	
7	5	0	Cartesian	6,250	0,000	-6,546	
8	6	0	Cartesian	12,500	0,000	-7,094	
9	7	0	Cartesian	18,750	0,000	-6,546	
10	8	0	Cartesian	22,000	0,000	-6,261	
11	9	0	Cartesian	25,000	0,000	-6,000	
12	10						
13	11	0	Cartesian	0,000	-5,000	0,000	Supported
14	12	0	Cartesian	25,000	-5,000	0,000	Supported
15	13	0	Cartesian	0,000	-5,000	-6,000	
16	14	0	Cartesian	3,000	-5,000	-6,261	

Figure 11.123: 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 323).

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 327).

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, RSTAB 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, RSTAB will search in the workbook for a table with the same name as in RSTAB 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 RSTAB will be exported.

To start the export of the current RSTAB 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 401). 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 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

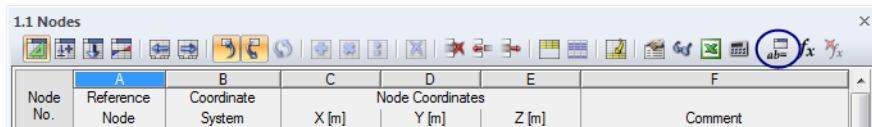


Figure 11.124: Button *Edit Parameters* in the table toolbar

- in the Formula Editor.

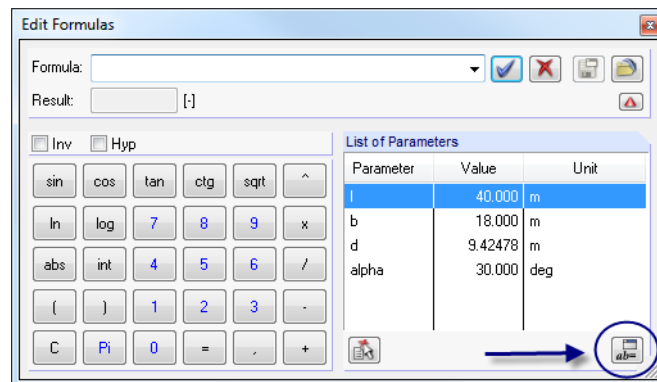


Figure 11.125: Button *Edit Parameters* in the Formula Editor

#### Description

The dialog box *Edit Parameters* appears.



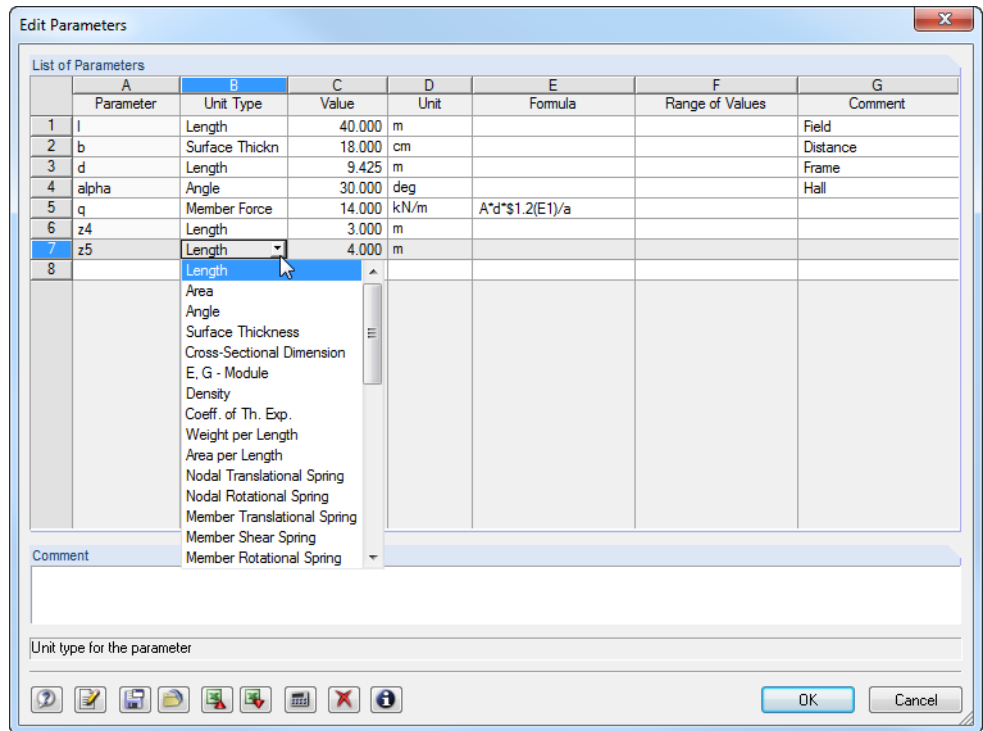


Figure 11.126: Dialog box *Edit Parameters*

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*.

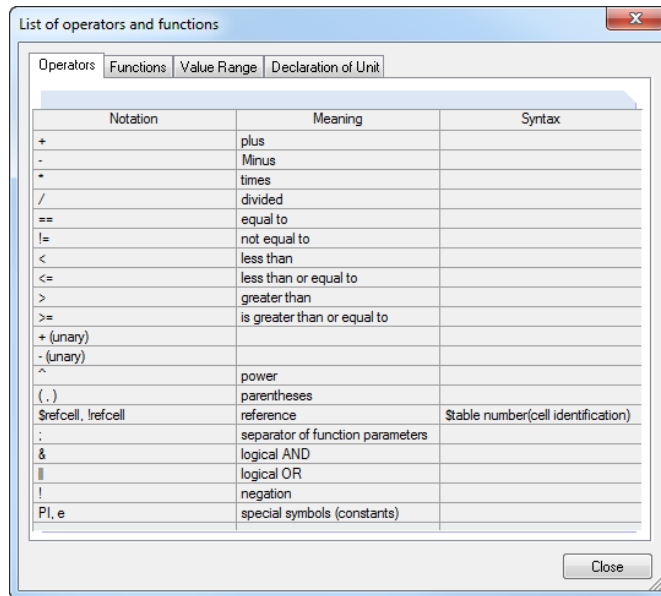


Figure 11.127: Dialog box *List of operators and functions*

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 321).

When several cells are marked as a selection, the following context menu appears.

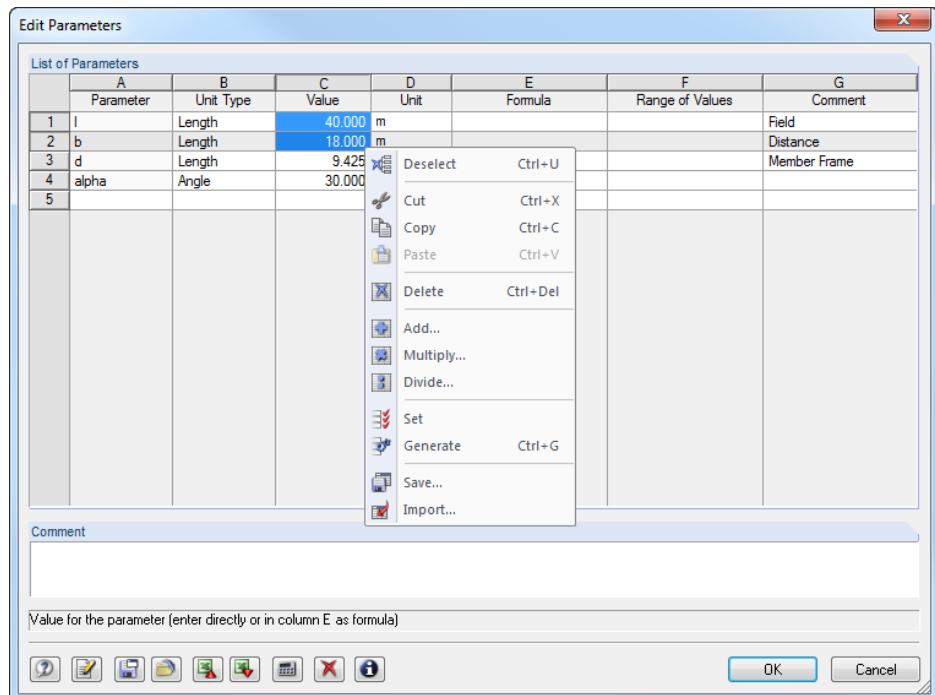


Figure 11.128: Context menu of a selection in parameter list

Find a description of the menu functions in chapters 11.5.1 and 11.5.2, page 321.

### Buttons

In addition to the default buttons, the following functions are available in the parameter list.

Button	Description
	Saves parameter list in a file
	Loads a saved parameter list
	Exports parameter list to MS Excel
	Imports data from opened Excel table
	Opens calculator and imports its result
	Deletes entire contents of parameters list
	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

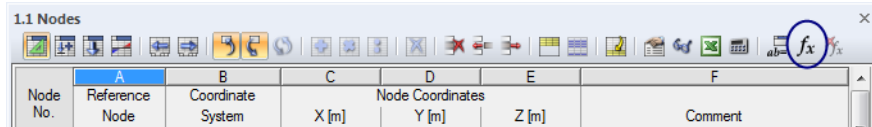


Figure 11.129: Button *Edit Formulas* in the table toolbar

- click the yellow or red corner of the table cell (a red corner indicates an error in the formula) or

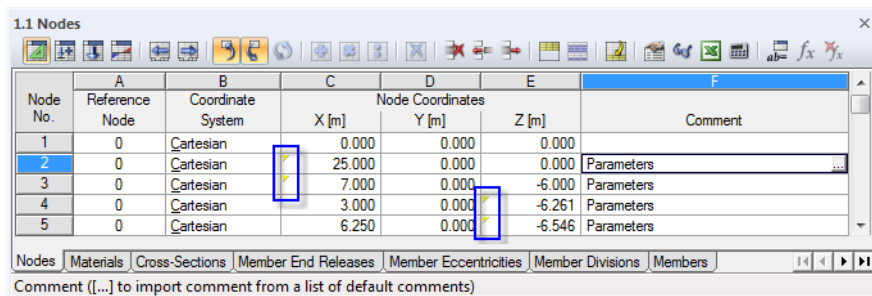


Figure 11.130: Marked cell corners in table 1.1 *Nodes*

- use the function buttons next to the input fields in dialog boxes (see Figure 11.135).

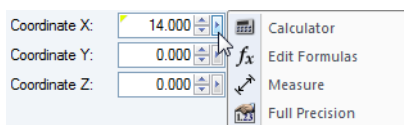


Figure 11.131: Function buttons with context menu in dialog box *Edit Node*

It is also possible to import formulas saved in Excel and to export formulas from RSTAB to Excel. For more information about the data exchange with Excel, see chapter 12.5.2 on page 401.

### Description

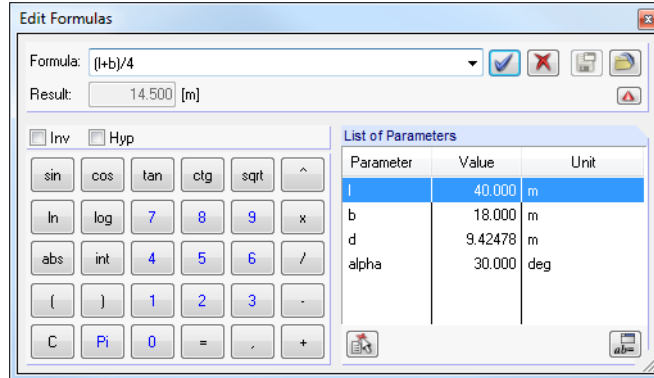


Figure 11.132: Dialog box *Edit Formulas*

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.

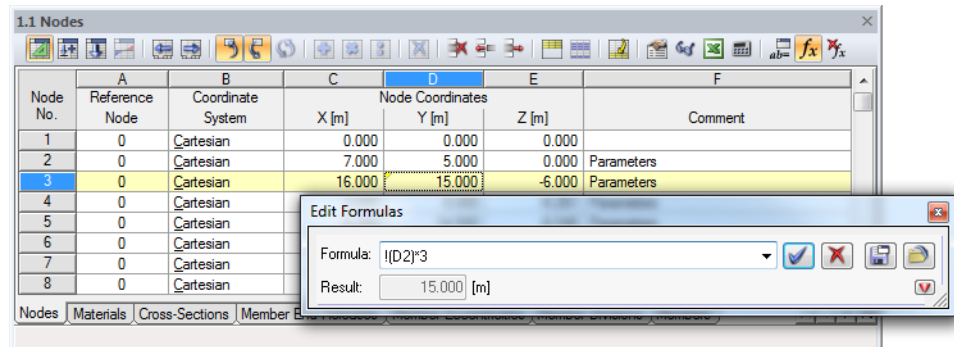


Figure 11.133: Formula Editor with a reference

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 enter formulas also directly into table cells (for example =2.5\*P). 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
	Sine
	Cosine
	Tangent
	Cotangent
	Square root
	Power
	Natural logarithm
	Logarithm to the base 10
	Absolute value
	Integer, for example $int(5.638) = 5$
	Clear formula line
<input type="checkbox"/> Inv	Inverse, for example $inv\ sqrt(5)$ means $5^2$
<input type="checkbox"/> 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 332) to open the parameter list where you can modify or complete the parameters.

### Buttons

The buttons available in the Formula Editor are reserved for 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



0.30

When cells are marked by a yellow or red flag (triangle) in the upper left corner, a formula has been linked (see Figure 11.130, page 335). 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.

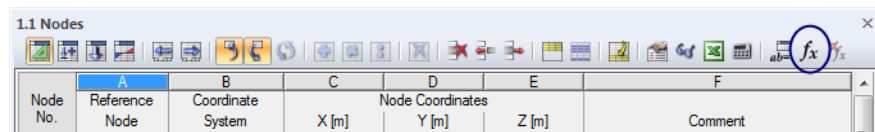


Figure 11.134: Button *Edit Formulas* in the table toolbar



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.

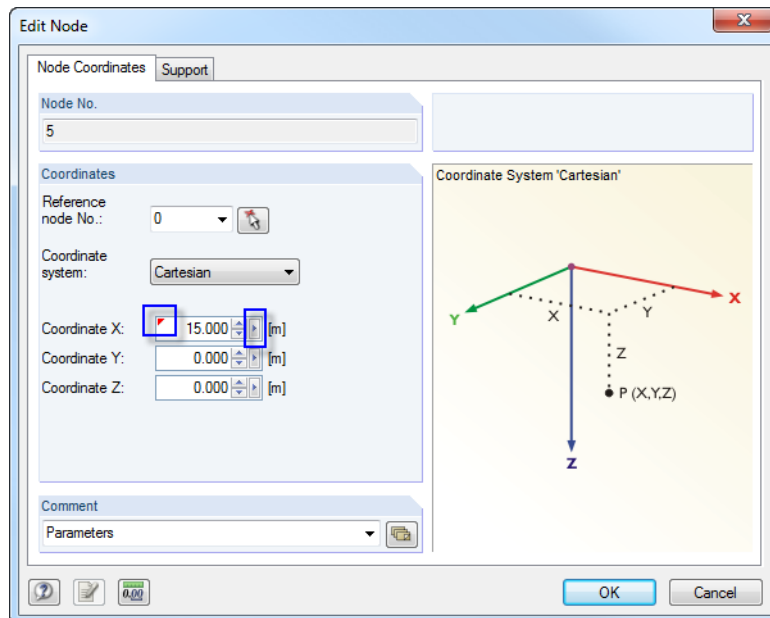


Figure 11.135: Dialog box with linked formula and function button

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 11.131 on page 335 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, RSTAB provides special dialog boxes for generating member models.

### 11.7.1 Copies and Extrusions

#### 11.7.1.1 Set Parallel Member

It is easy to copy selected 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 members, you can enter specific settings in a dialog box. To access the corresponding function,

select **Set Parallel Member** on the **Tools** menu

or use the context menu of the member (see Figure 4.72, page 80).

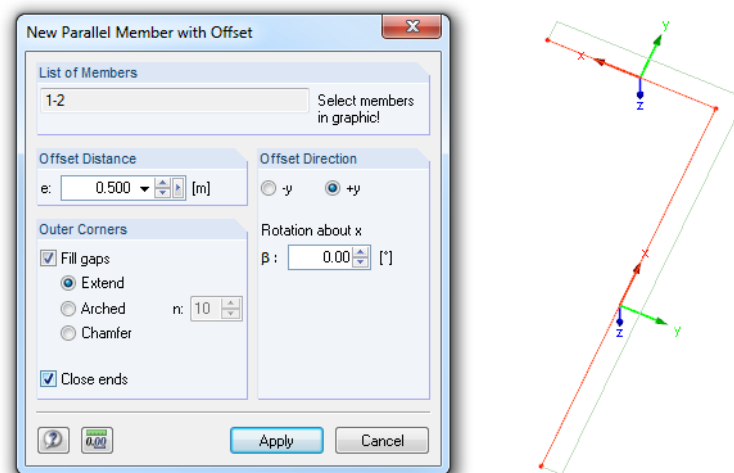


Figure 11.136: Dialog box *New Parallel Member with Offset*

The selected member appears in the *List of Members*. If necessary, other 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 *Offset Distance*, you specify the copy's distance to the original.

When several members are copied by parallel offset, you have several possibilities offered in the dialog section *Outer Corners* to adjust the copied members. The figure above shows the copied members (represented without axes) extended to the common point of intersection. Moreover, both ends are connected with the original members by the ticked check box *Close ends*.

The settings in the dialog section *Offset Direction* define the side on which the members 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 member axes. Due to the input field *Rotation about x*, it is possible to copy the objects out of the plane.

### 11.7.1.2 Extrude Member into Grid

By extruding members you can quickly create 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 344).

To access the corresponding function,

select **Extrude Member into Grid** on the **Tools** menu

or use the context menu of the member (see Figure 4.72, page 80).

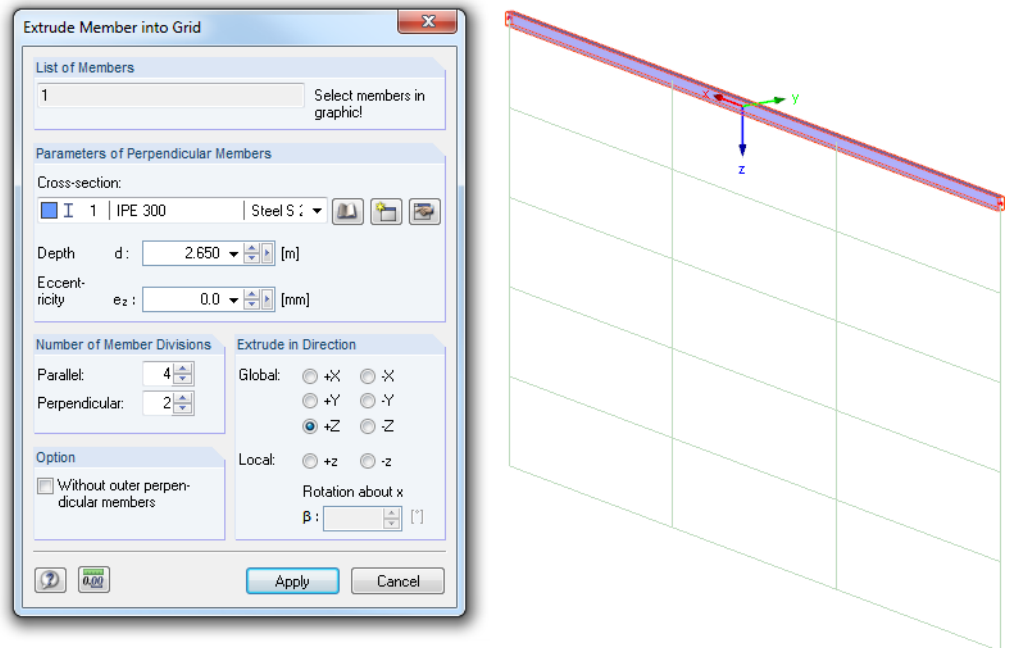


Figure 11.137: Dialog box *Extrude Member into Grid*

The selected member appears in the *List of Members*. If necessary, other 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 4.5, page 70).

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.



### 11.7.2 Model Generators

To access the dialog boxes for creating model objects, select **Generate Model** on the **Tools** menu.

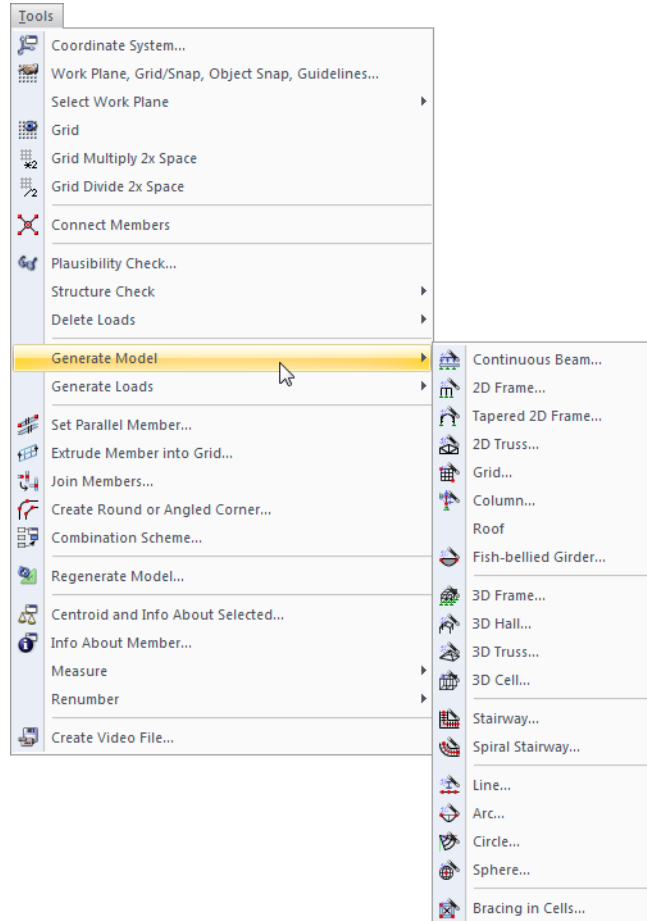


Figure 11.138: Menu *Tools* → *Generate Model*

In the following, the functions are described in detail. However, you won't find a detailed description of the single generator dialog boxes. 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.

### Continuous beam

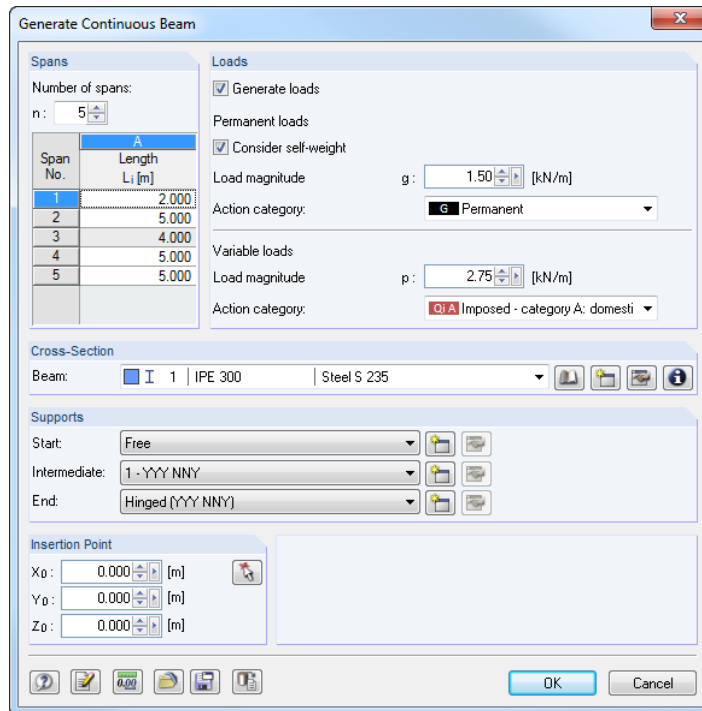


Figure 11.139: Dialog box *Generate Continuous Beam*

RSTAB creates a continuous beam with uniform cross-section, supports and irregular spans. Optionally, load cases and result combinations are created, too.

### 2D frame

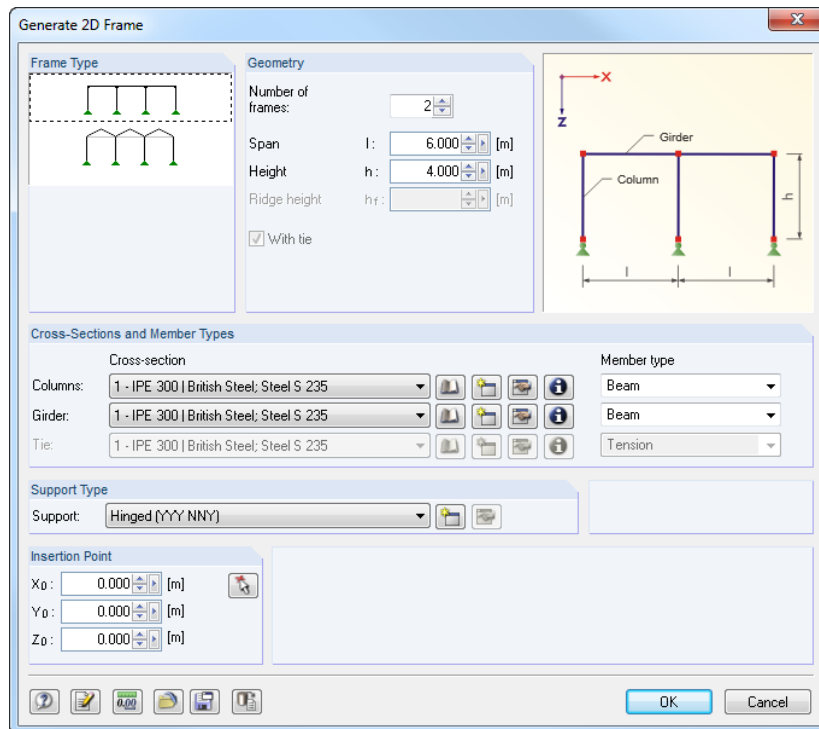


Figure 11.140: Dialog box *Generate 2D Frame*

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

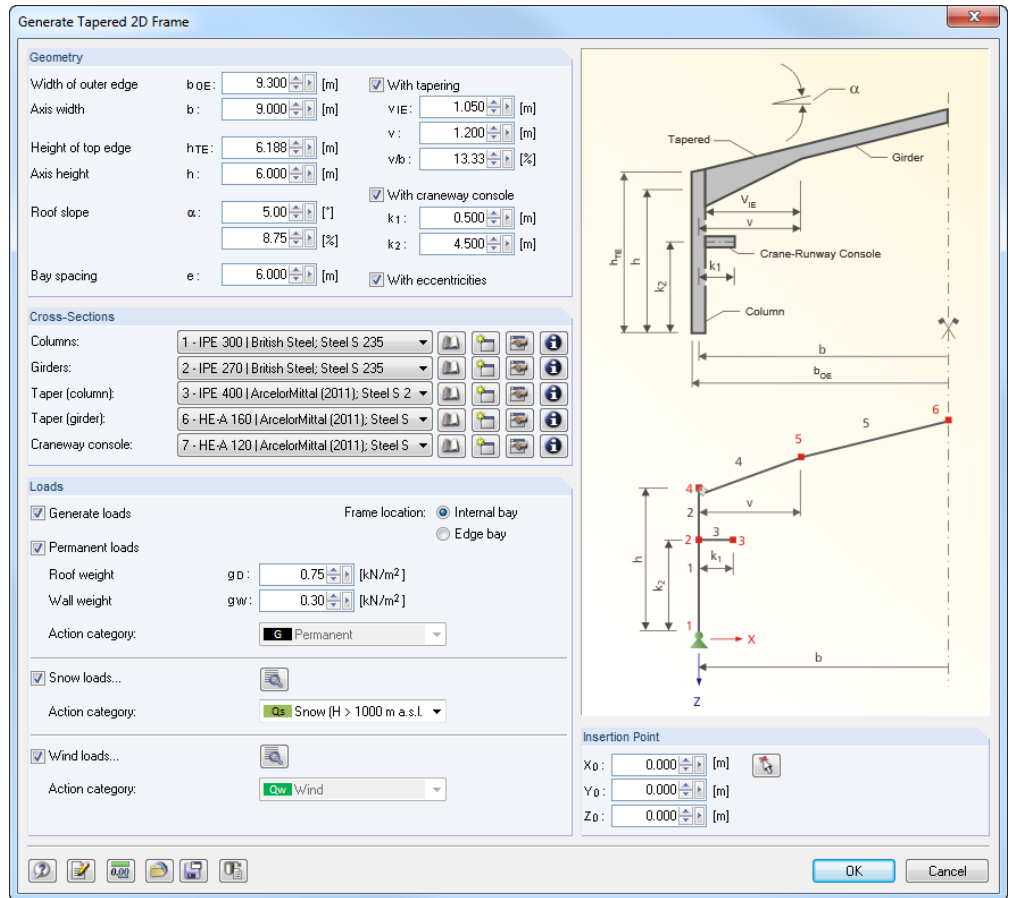


Figure 11.141: Dialog box *Generate Tapered 2D Frame*



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.

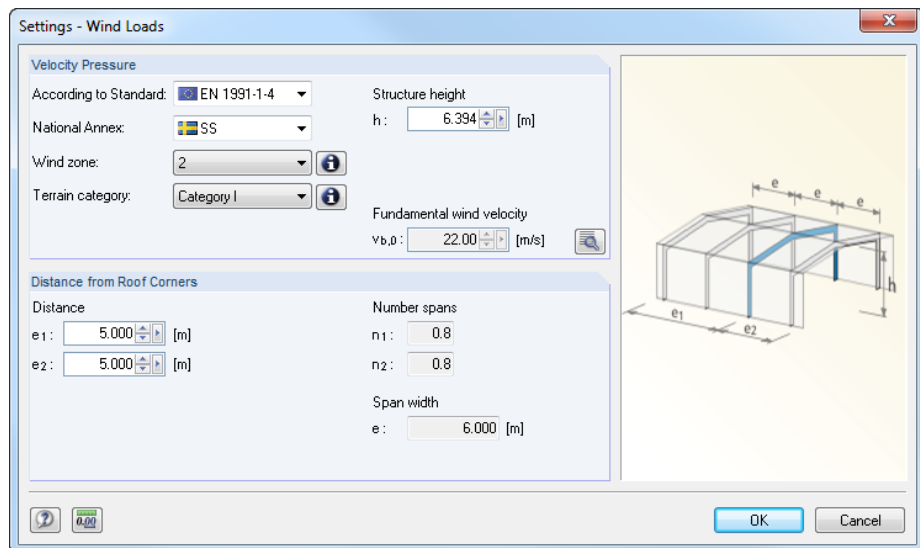


Figure 11.142: Dialog box *Settings - Wind Loads*

### 2D truss

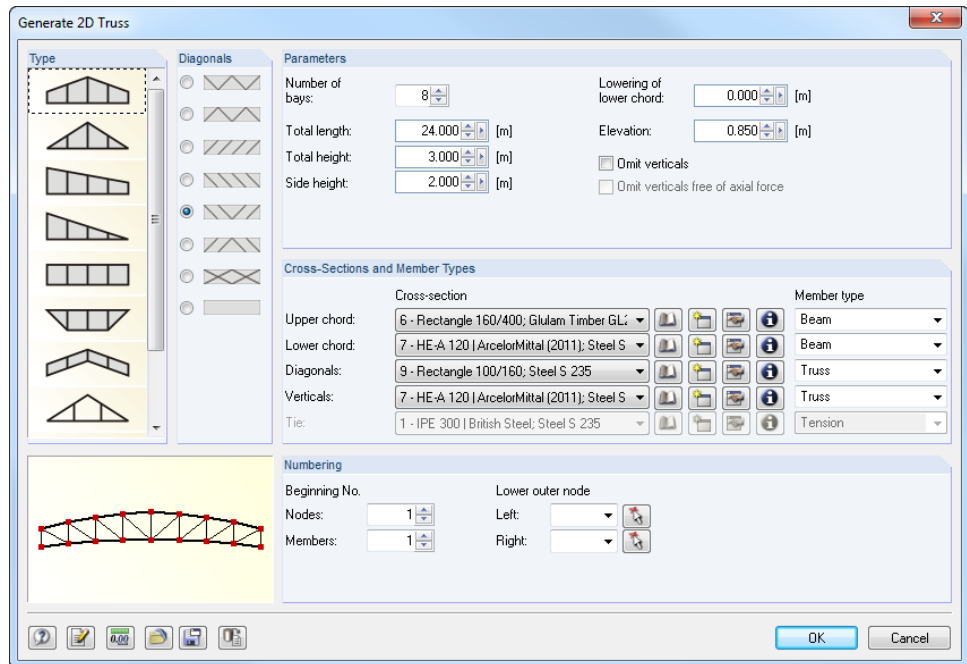


Figure 11.143: Dialog box *Generate 2D Truss*

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

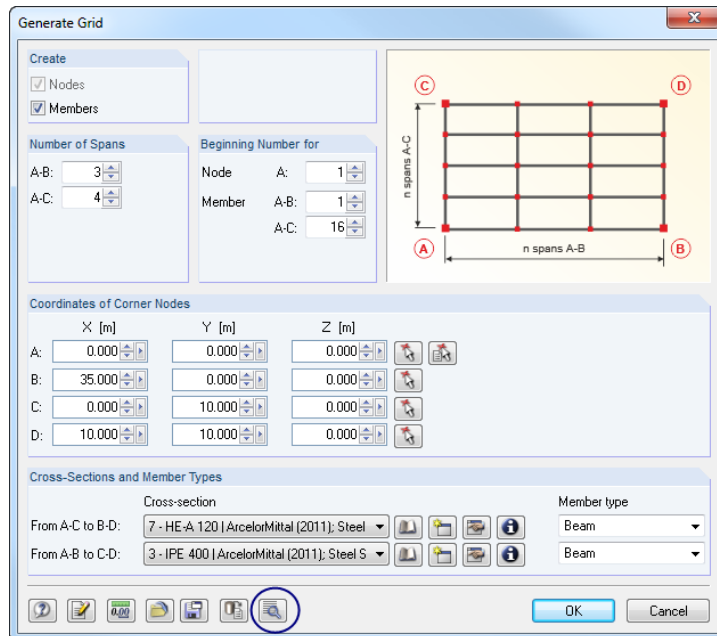


Figure 11.144: Dialog box *Generate Grid*

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 386).

To generate irregular grids, use the button [Edit Advanced Settings] shown on the left.



### Column

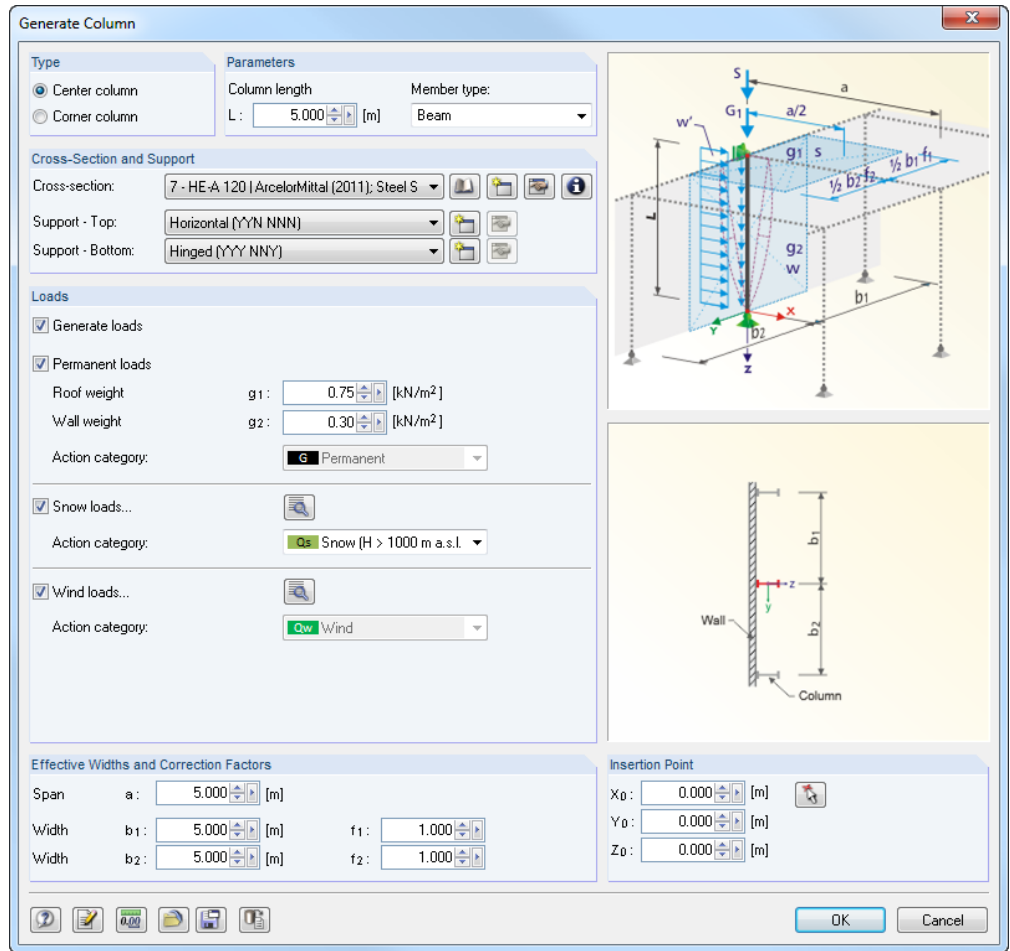


Figure 11.145: Dialog box *Generate Column*

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  $f_1$  and  $f_2$  are used to scale the geometric widths  $b_1$  and  $b_2$  for the structural model, or to consider special standard 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 (see figures below).

The [Settings] buttons available in the roof dialog boxes help you to determine wind and snow loads (see Figure 11.142, page 343).

Collar roof

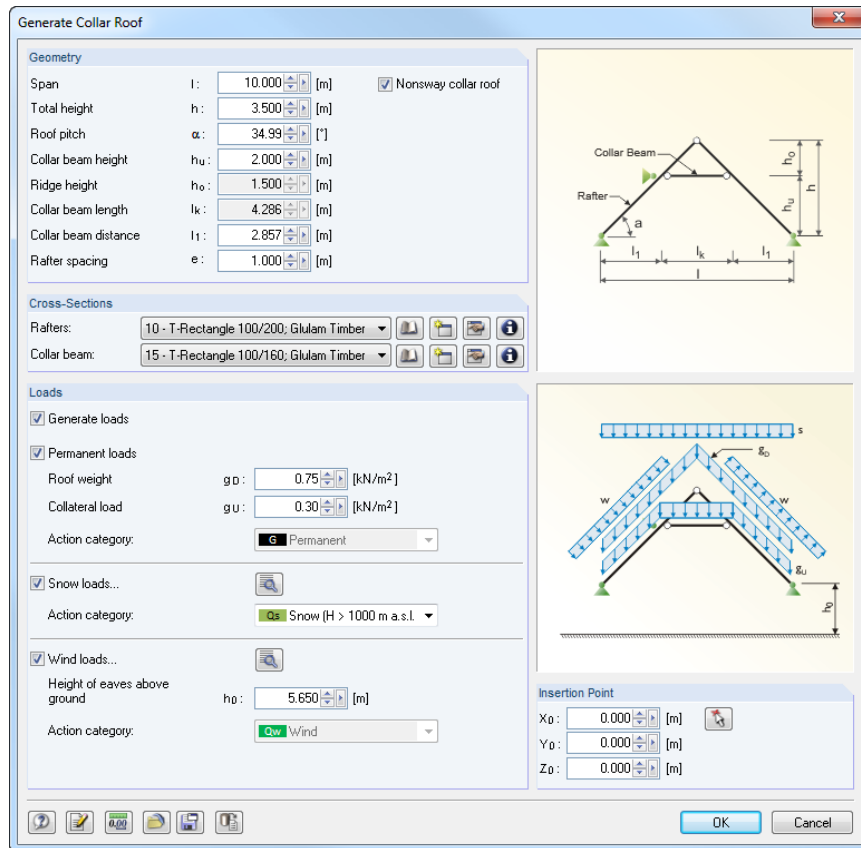


Figure 11.146: Dialog box *Generate Collar Roof*

Rafter roof

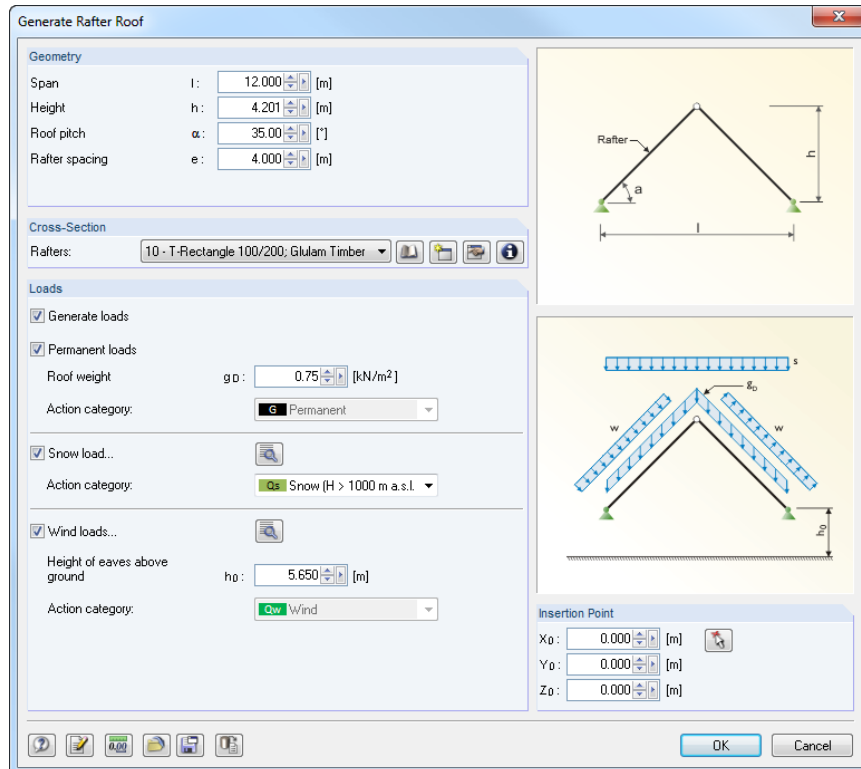


Figure 11.147: Dialog box *Generate Rafter Roof*

Purlin roof

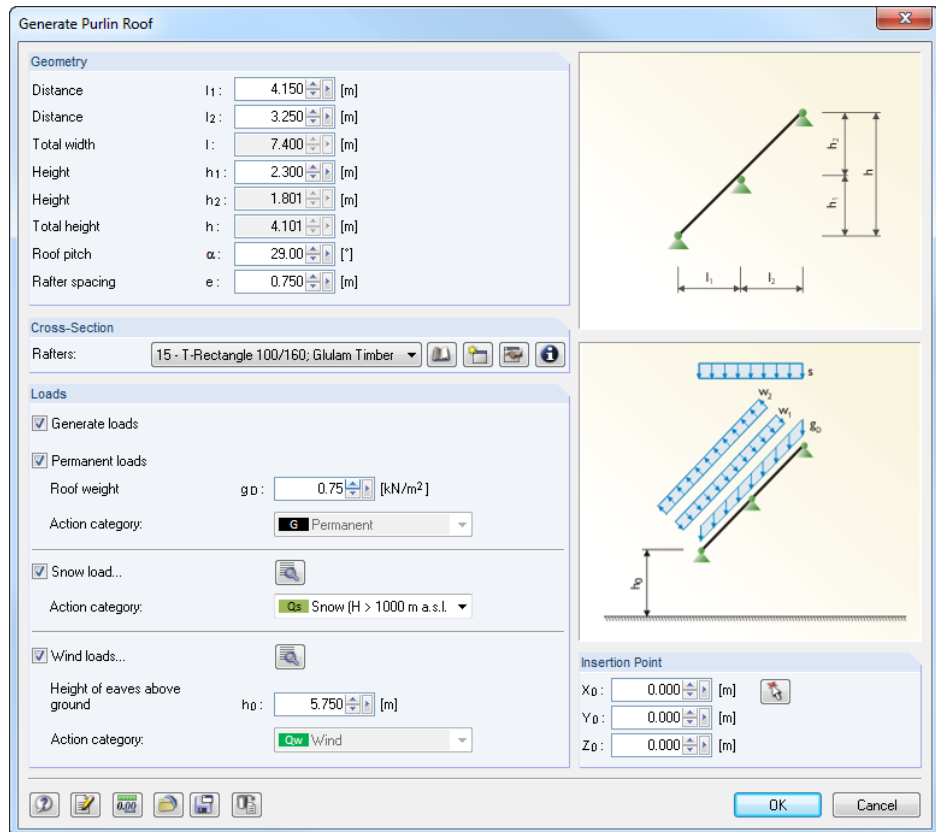


Figure 11.148: Dialog box *Generate Purlin Roof*

Fish-bellied girder

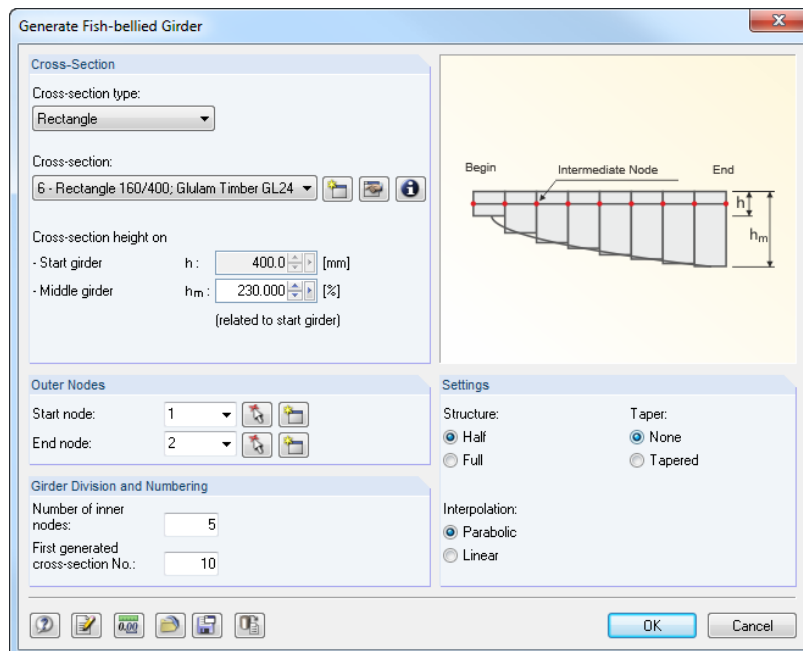


Figure 11.149: Dialog box *Generate Fish-bellied Girder*

For the generation of fish-bellied girders commonly used in timber construction the rectangular and ITS cross-section types (symmetric I-beams) are available for selection in the *Cross-section type* list.

### 3D frame

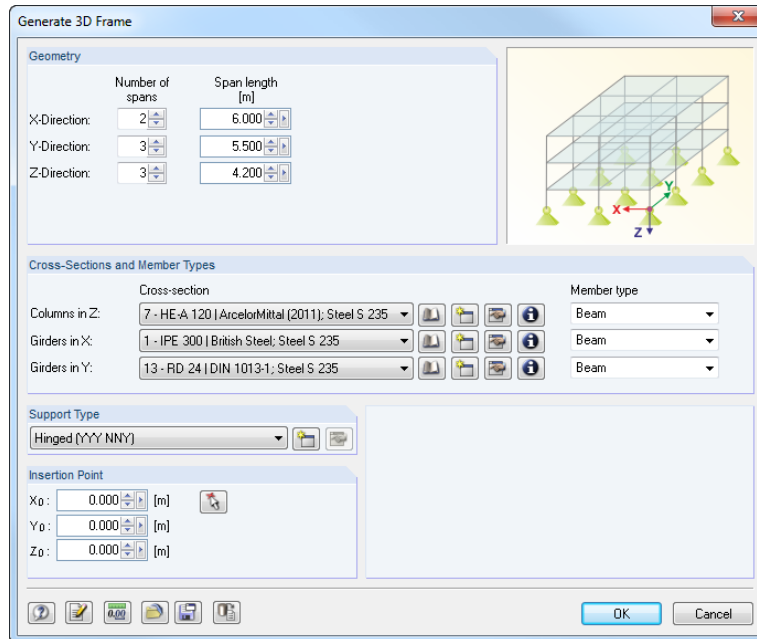


Figure 11.150: Dialog box *Generate 3D Frame*

Use this generator to create regular frame models. The columns of the frame receive equal support conditions.

### 3D hall

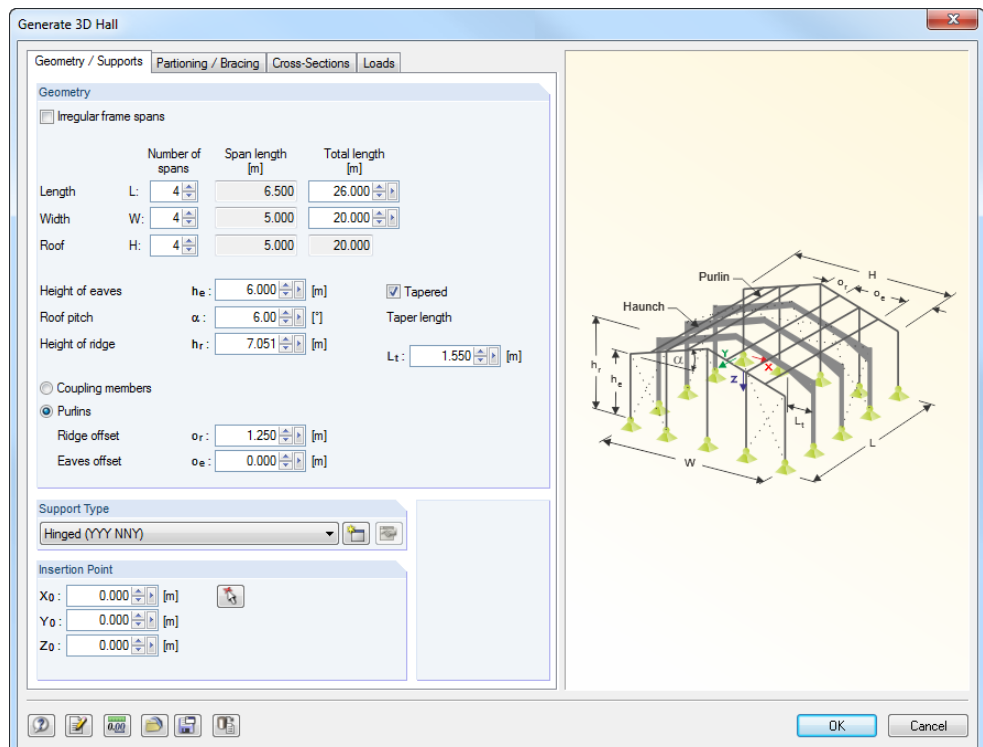


Figure 11.151: Dialog box *Generate 3D Hall*

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

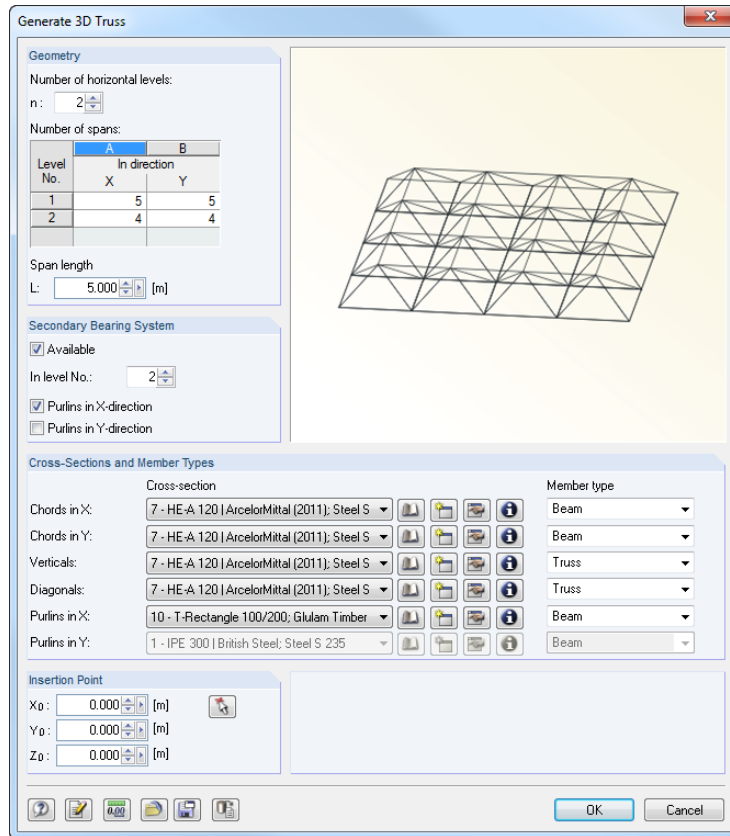


Figure 11.152: Dialog box *Generate 3D Truss*

Use this generator to create a spatial load-bearing structure according to the *Bernauer* system ([www.raumtragwerke.de](http://www.raumtragwerke.de)).

### 3D cell

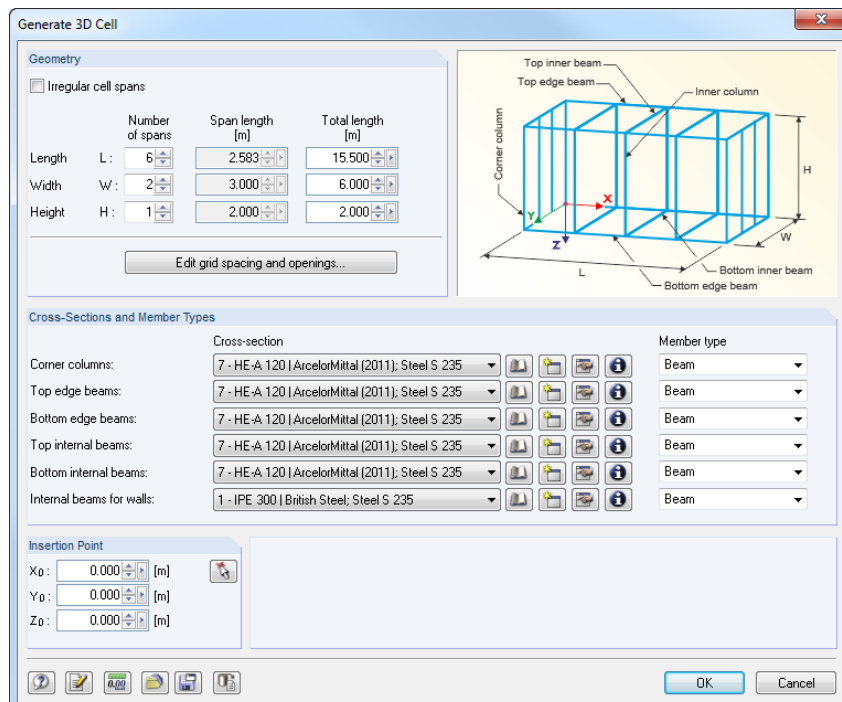


Figure 11.153: Dialog box *Generate 3D Cell*

Edit grid spacing and openings...

The generator creates a spatial cell with several fields. Use the button [Edit grid spacing and openings] to open another dialog box where you can define openings as well as the grid arrangement for irregular field spacings.

### Stairway

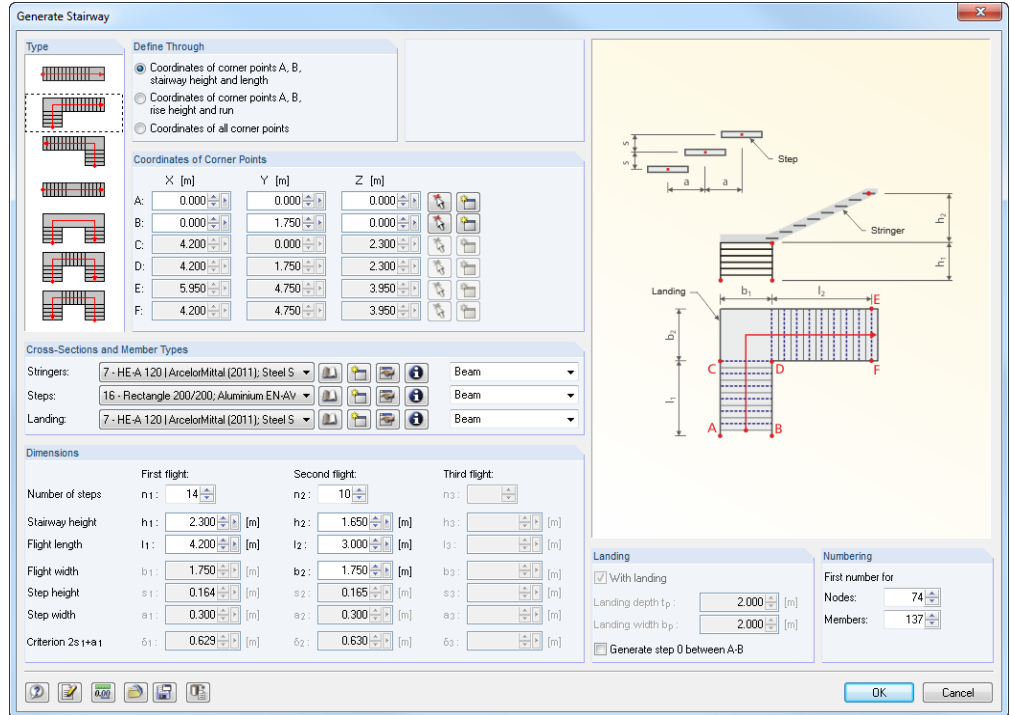


Figure 11.154: Dialog box *Generate Stairway*

In the list, select the *Type* controlling the remaining parameters.

### Spiral stairway

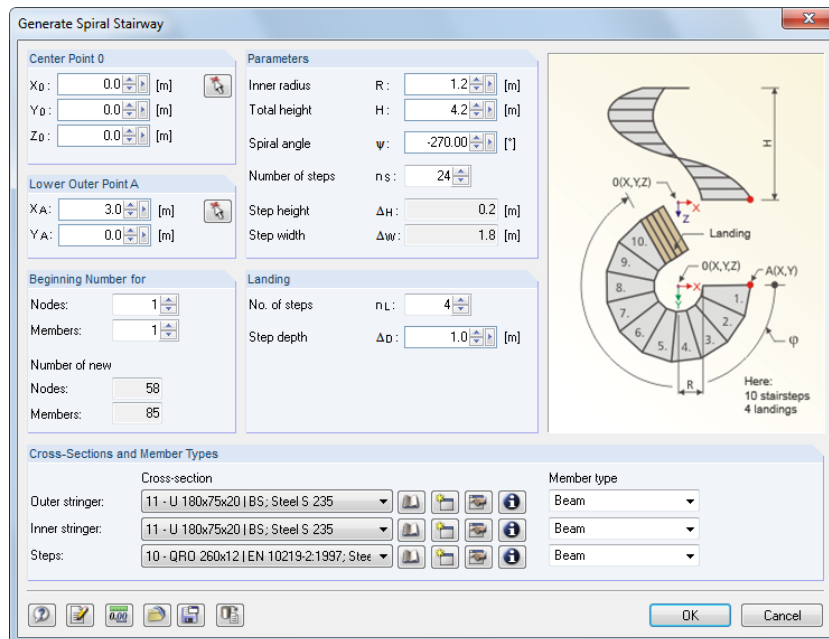


Figure 11.155: Dialog box *Generate Spiral Stairway*

### Straight line

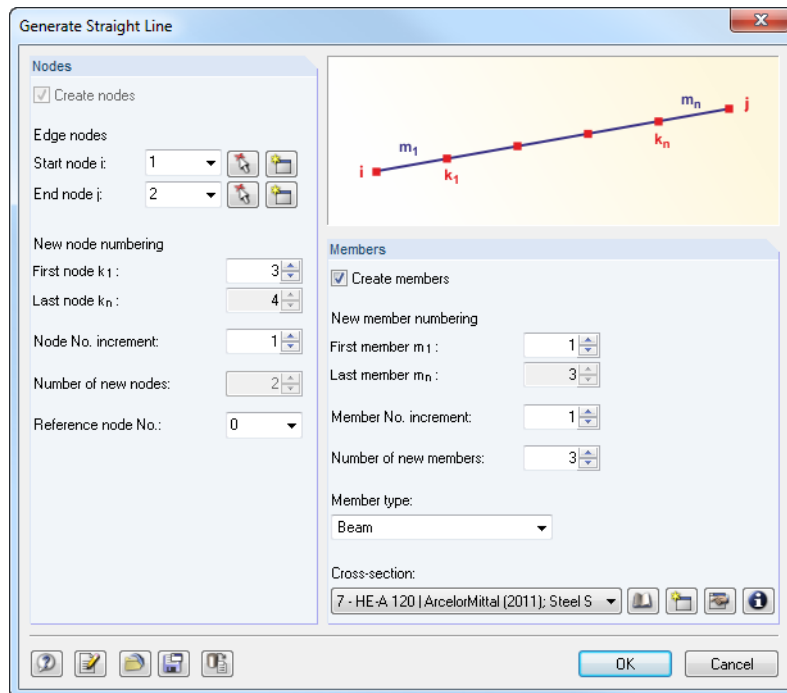


Figure 11.156: Dialog box *Generate Straight Line*

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

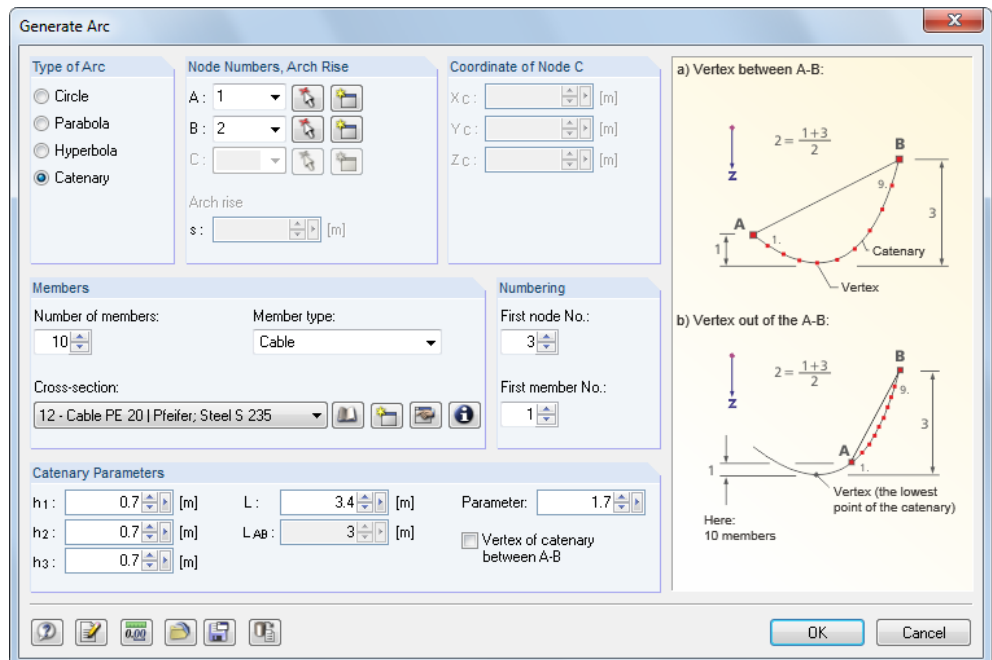


Figure 11.157: Dialog box *Generate Arc*

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:

$$y(x) = a \cdot \cosh\left(\frac{x - v_x}{a}\right) + v_y \quad \text{where } v_x \text{ or } v_y: \quad \text{displacements in } x \text{ or } y$$

Equation 11.1

The larger the number of members is, the more precisely the arc will be modeled as a polygonal chain.

### Circle

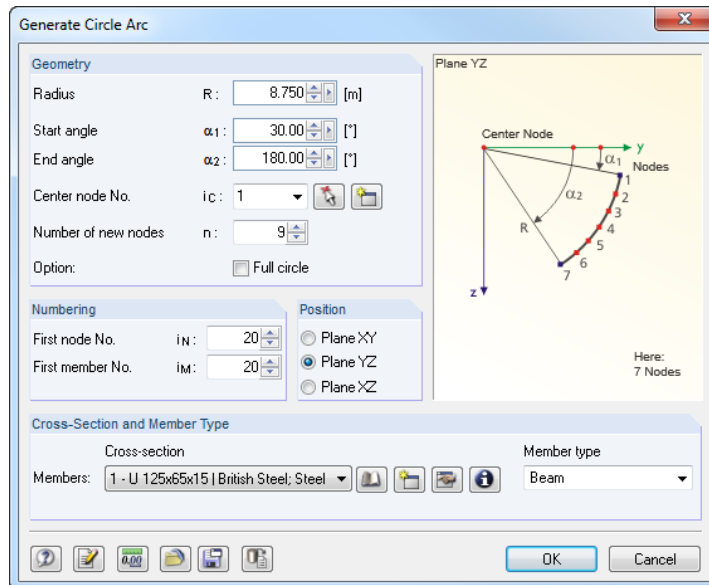


Figure 11.158: Dialog box *Generate Circle Arc*

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

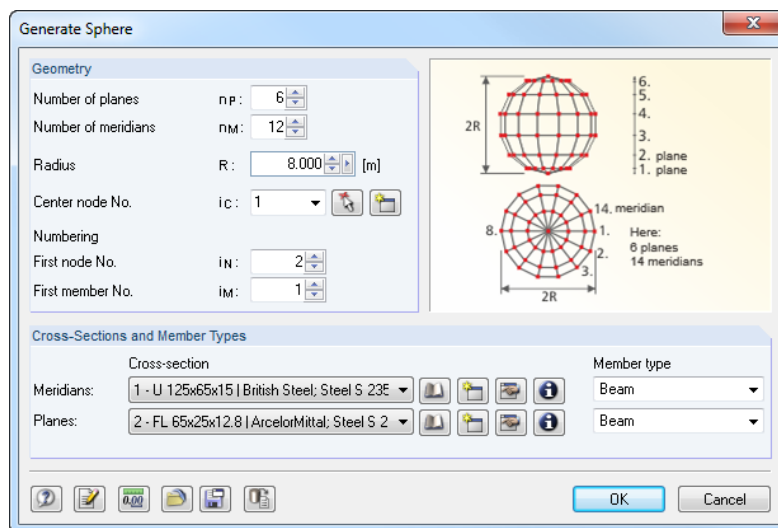


Figure 11.159: Dialog box *Generate Sphere*

The larger the *Number of planes* and *meridians* is, the rounder the shape of the sphere will be. Polygonal chains approximate the spherical form with each member representing a segment.

## Bracing in cells

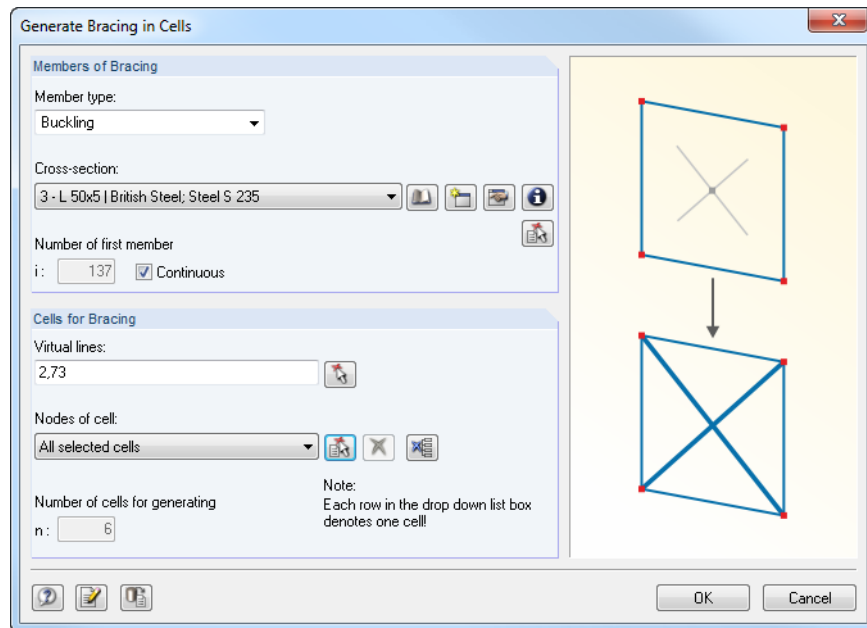


Figure 11.160: Dialog box *Generate Bracing in Cells*



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.

## 11.8 Load Generators

The second group of generators helps you to apply member loads: On the one hand, it is possible to convert area loads (for example snow, wind) acting on the structural system into member 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 area 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 11.167, page 357) 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.

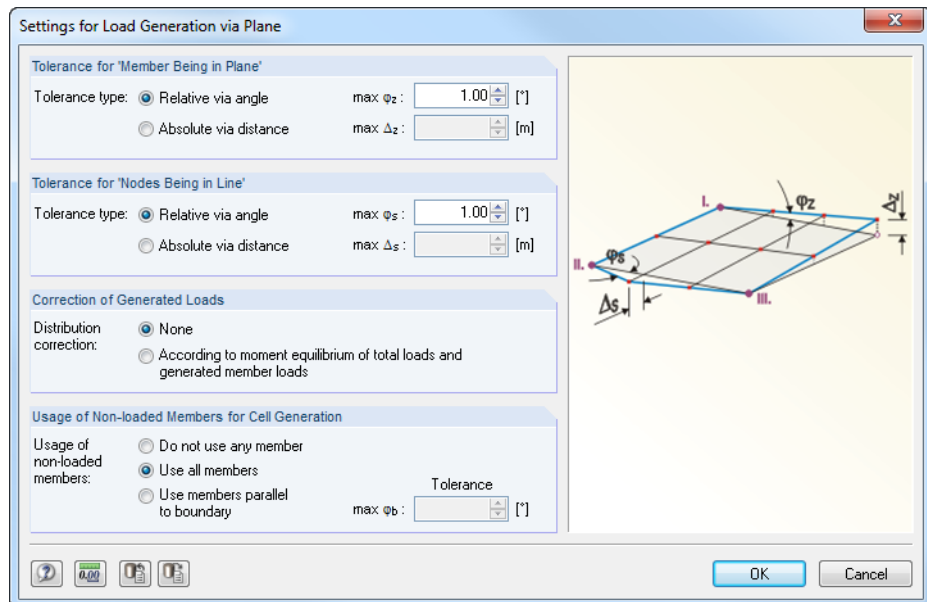


Figure 11.161: Dialog box *Settings for Load Generation via Plane*

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, RSTAB 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 that appear after load generation and before the final conversion into member loads will be performed (see Figure 11.171, page 360). In case of minor differences, it is recommended to correct the distribution according to the *moment equilibrium*.

The following applies:

$$\int_{L_{\text{cell}}} (q_{\text{member}} + q_{\text{correct}}) dL = \int_{S_{\text{cell}}} q dS \quad \text{Equilibrium of forces}$$

$$\int_{L_{\text{cell}}} (q_{\text{member}} + q_{\text{correct}}) r dL = \int_{S_{\text{cell}}} q r dS \quad \text{Moment equilibrium}$$

where  $r = (x, y)$  Distance to centroid of cell

When correcting the generated loads according to 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, RSTAB finds out which members are enclosing cells. Next, the cells are subtracted from the total area. When excluding a member from the loading (option *Remove Influence from members*, see below), RSTAB 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.

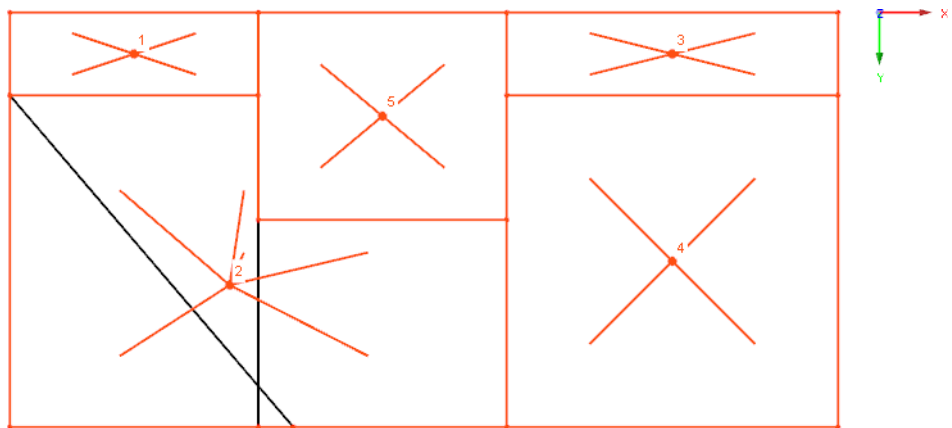


Figure 11.162: Platform construction with cells for load generation

- 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.

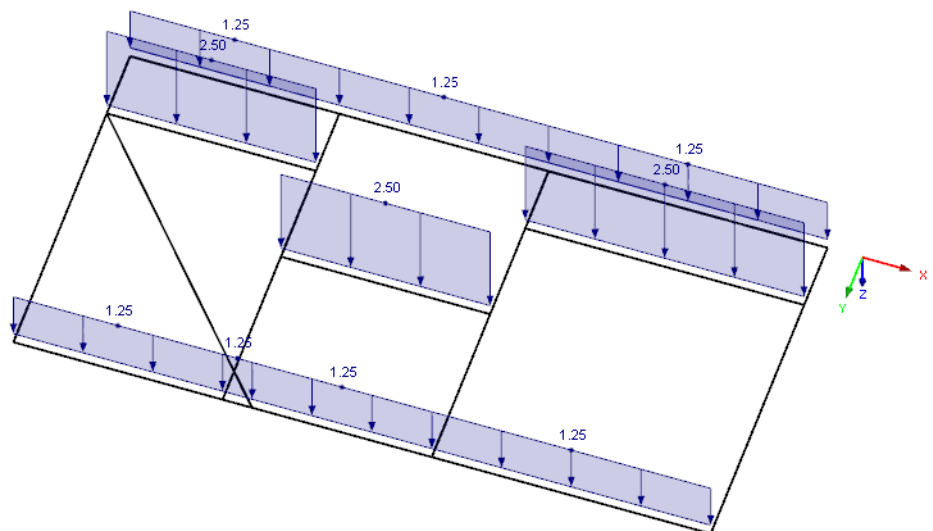


Figure 11.163: Result for *Do not use any members*

- *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.

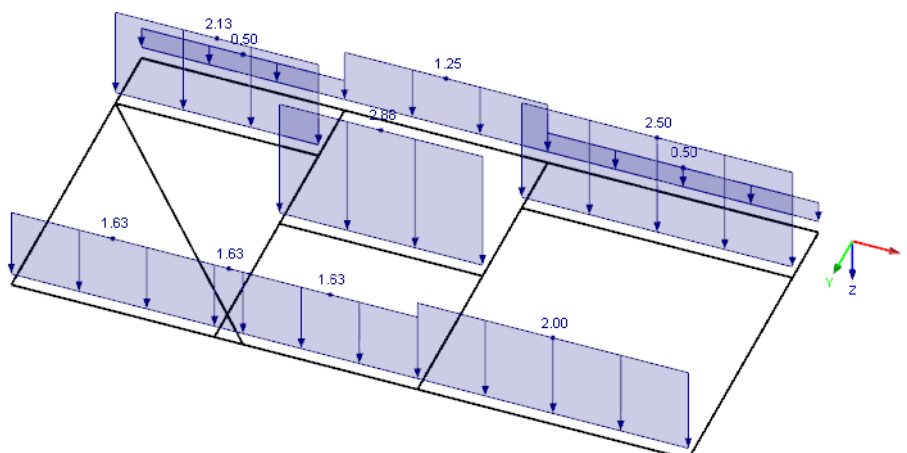


Figure 11.164: Result for *Use all members*

- *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  $\varphi_b$  is restricted to  $40.55^\circ$  in the *Settings* dialog box (see Figure 11.161, page 354), the load will be generated as expected.

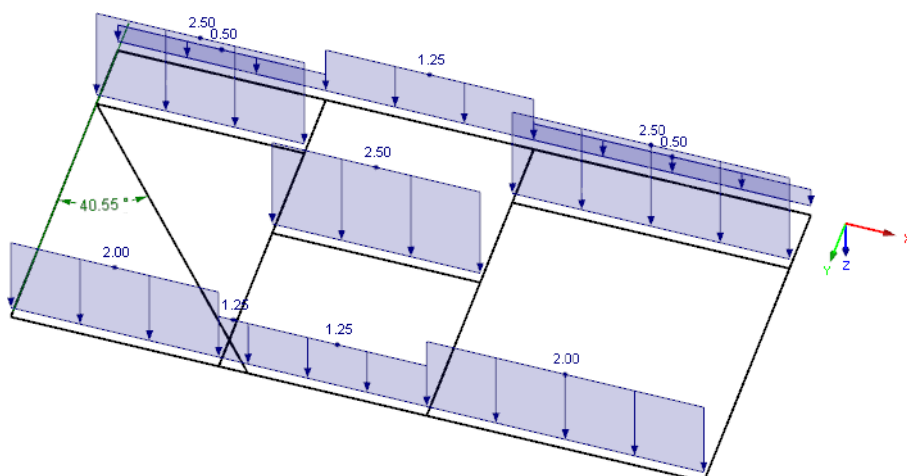


Figure 11.165: Result for *Use members parallel to boundary*

### Modify generated loads subsequently

After confirming a generator dialog box you find the generated loads transferred to load table 3.5. The additional entry *Generated Loads* appears in the *Data* navigator (see Figure 6.29, page 162). 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).



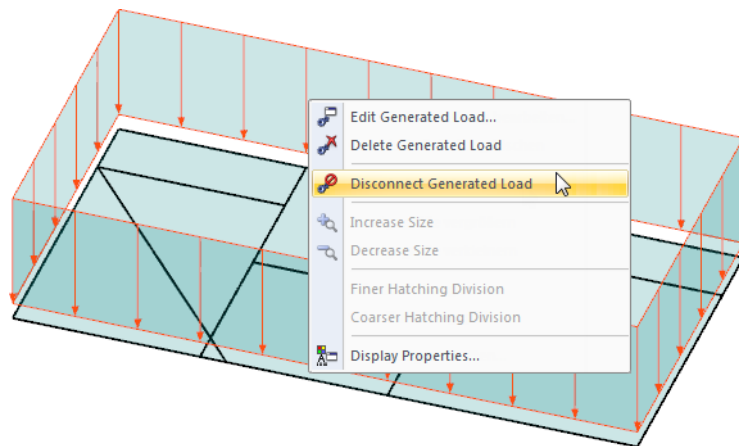


Figure 11.166: Context menu of generated load

You can also use the context menu of the generated load in the *Data* navigator.

## 11.8.2 Member Loads from Area Loads



### 11.8.2.1 Member Loads From Area Load via Plane

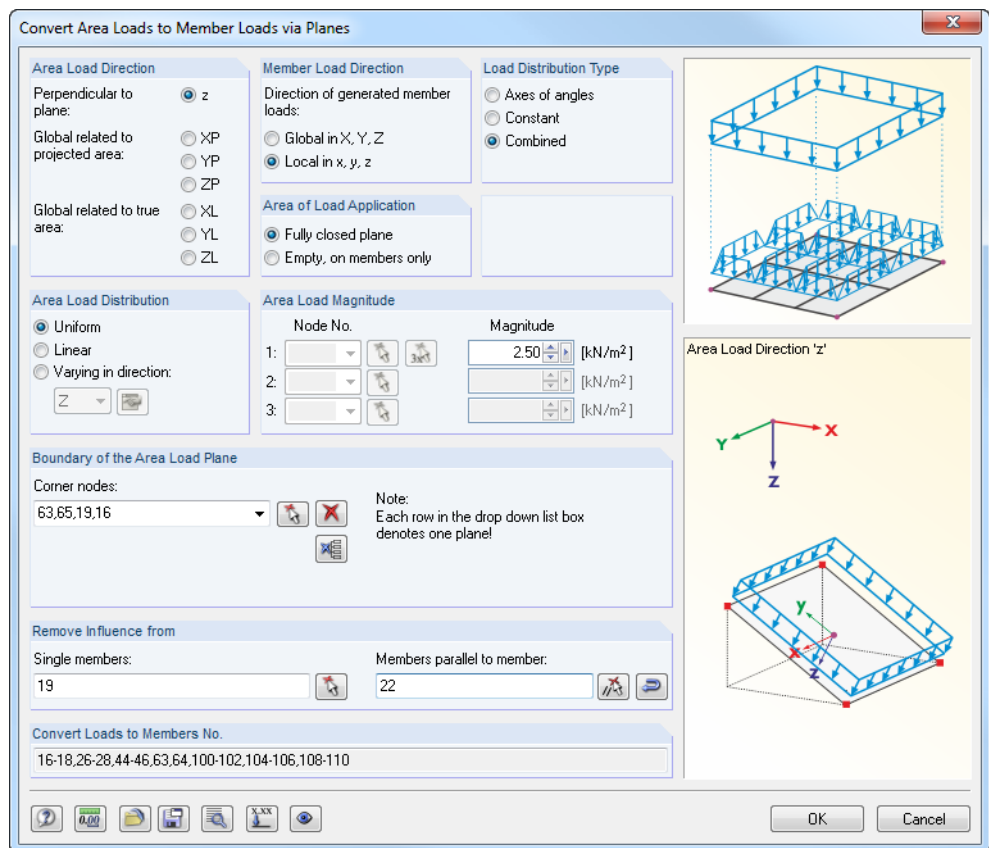


Figure 11.167: Dialog box *Convert Area Loads to Member Loads via Planes*

#### Area load direction

Decide whether the load acts perpendicular to the plane or globally related to the real or projected area. The dialog graphic in the right corner illustrates the selected load direction.

### Member load direction

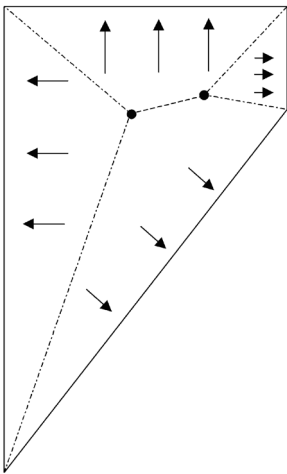
The generated member loads can be set to be global or local loads (see chapter 6.2, page 153). The difference is especially significant for non-linear calculations.

### 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 RSTAB model. In this case, RSTAB 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, RSTAB 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, RSTAB will also determine the centroid of the plane. If the intersection points of the bisecting lines lie in front of the centroid, triangular application areas will be generated. If they lie behind the centroid, a line that is parallel to the member will be drawn through the centroid, forming an application area with both angle bisectors.

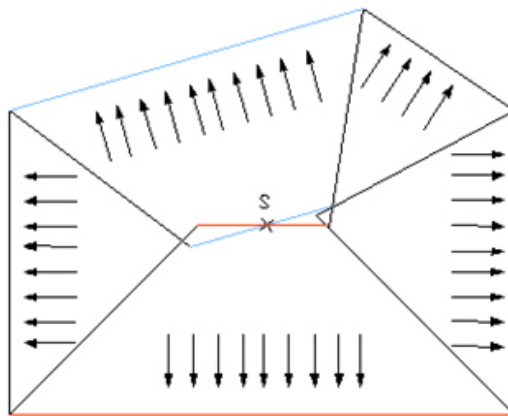


Figure 11.168: Load distribution type *Constant*

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, RSTAB switches automatically to the constant load distribution. Therefore, the combined method is set by default; RSTAB 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.



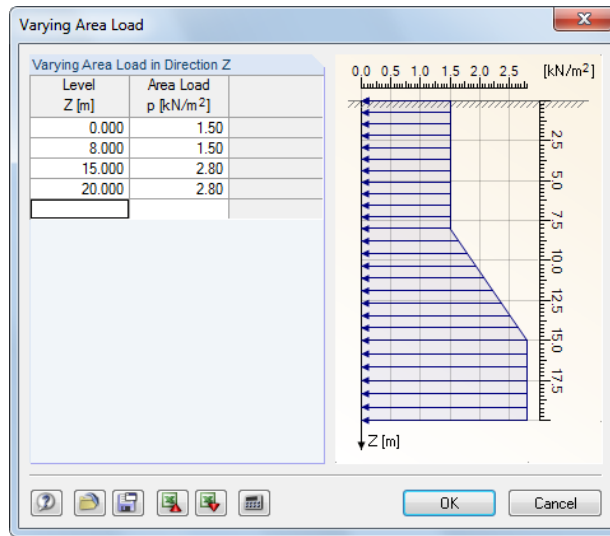


Figure 11.169: Dialog box *Varying Area Load*

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 11.161, page 354). 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 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 11.161, page 354). 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.



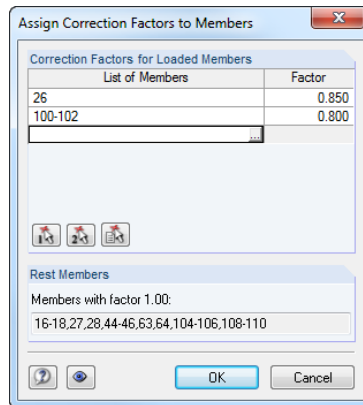


Figure 11.170: Dialog box *Assign Correction Factors to Members*



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.

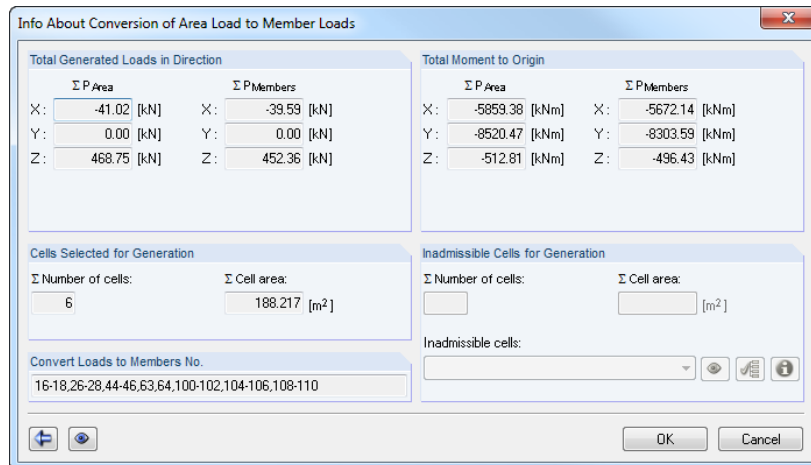


Figure 11.171: Dialog box *Info About Conversion of Area Load to Member Loads*



If inadmissible cells are listed, RSTAB 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 11.161, page 354) that you can access by using the [Settings] button.

The buttons at the bottom left in the info window are reserved for the following functions:

Button	Description
	The dialog box <i>Convert Area Loads to Member Loads</i> opens again and you can adjust the generation parameters.
	RSTAB shows you the work window where you can change the view (view mode). To return to the <i>Info</i> 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

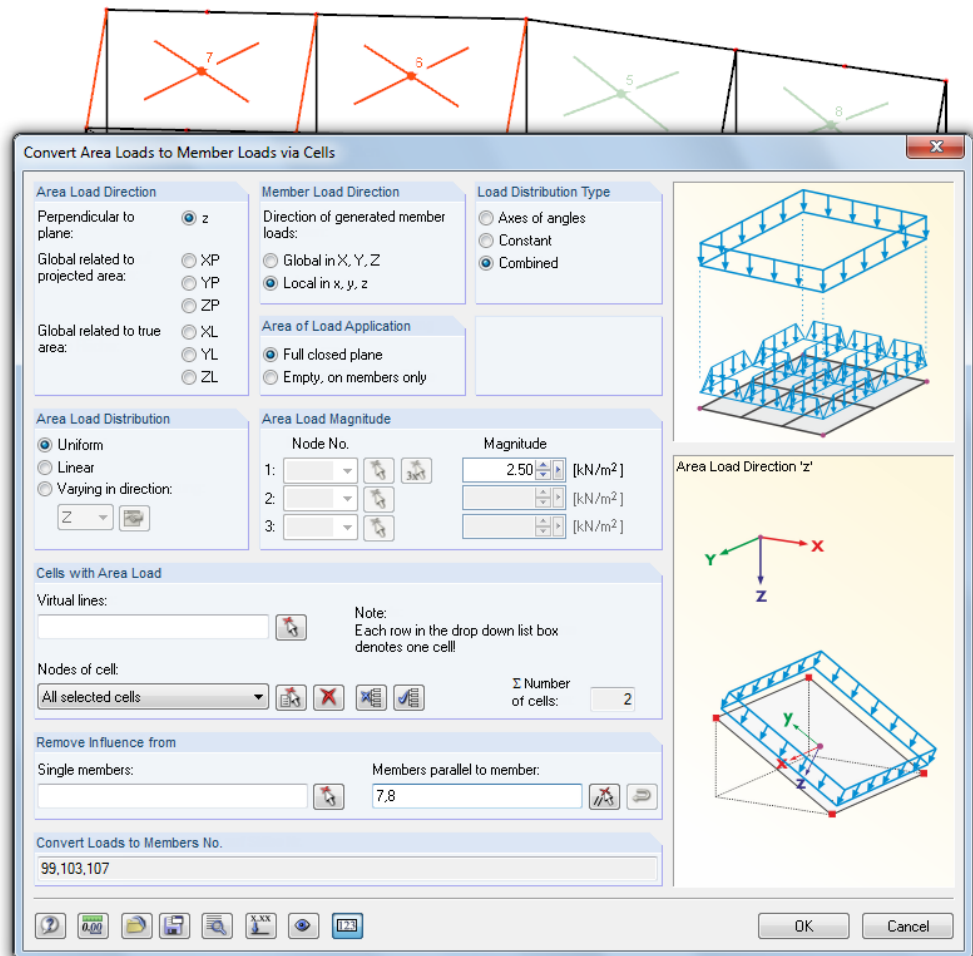


Figure 11.172: Dialog box *Convert Area Loads to Member Loads via Cells*

This dialog box is similar to the dialog box *Convert Area Loads to Member Loads via Planes* described on page 357. RSTAB 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: RSTAB 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 11.161, page 354). Then, you can adjust the tolerance for the integration of nodes in the load plane or correct the generated loads.

### 11.8.3 Other Loads

#### 11.8.3.1 Member Loads from Free Line Load

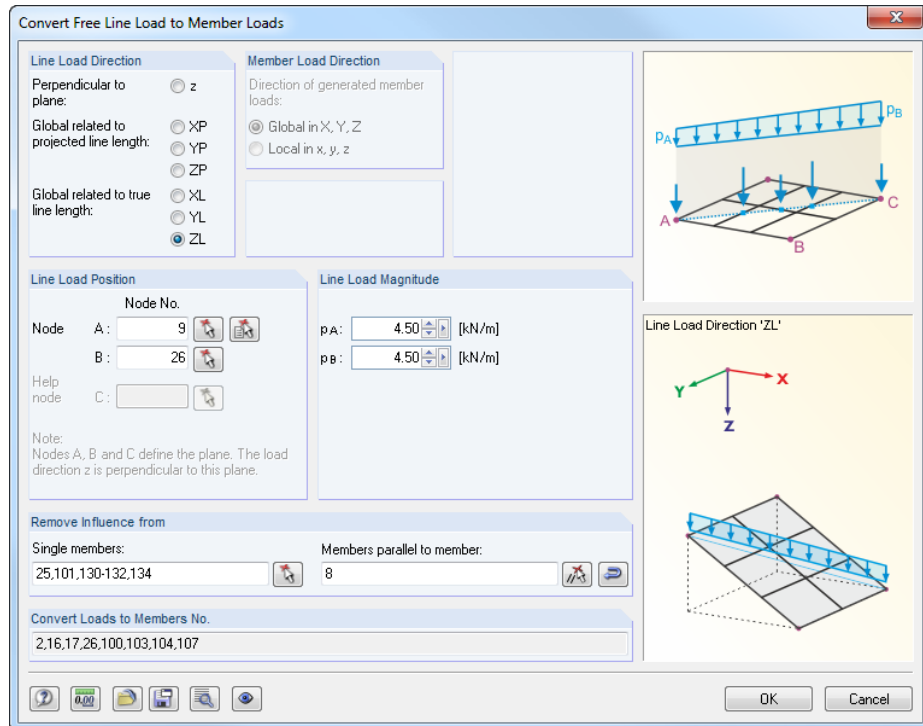


Figure 11.173: Dialog box *Convert Free Line Load to Member Loads*

Use this dialog box to define free line loads for example for girder grillages in order to prorate the loads to the 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 357.

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 11.161, page 354).

#### 11.8.3.2 Member Loads from Coating

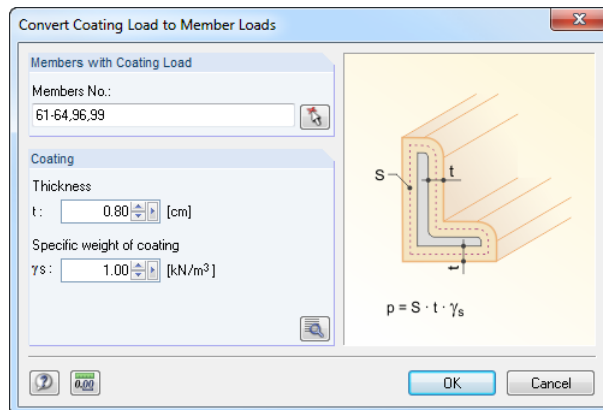


Figure 11.174: Dialog box *Convert Coating Load to Member Loads*



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 11.174). Thus, loads will be determined correctly even for small cross-sections with many edges.



### 11.8.3.3 Loads from Movements

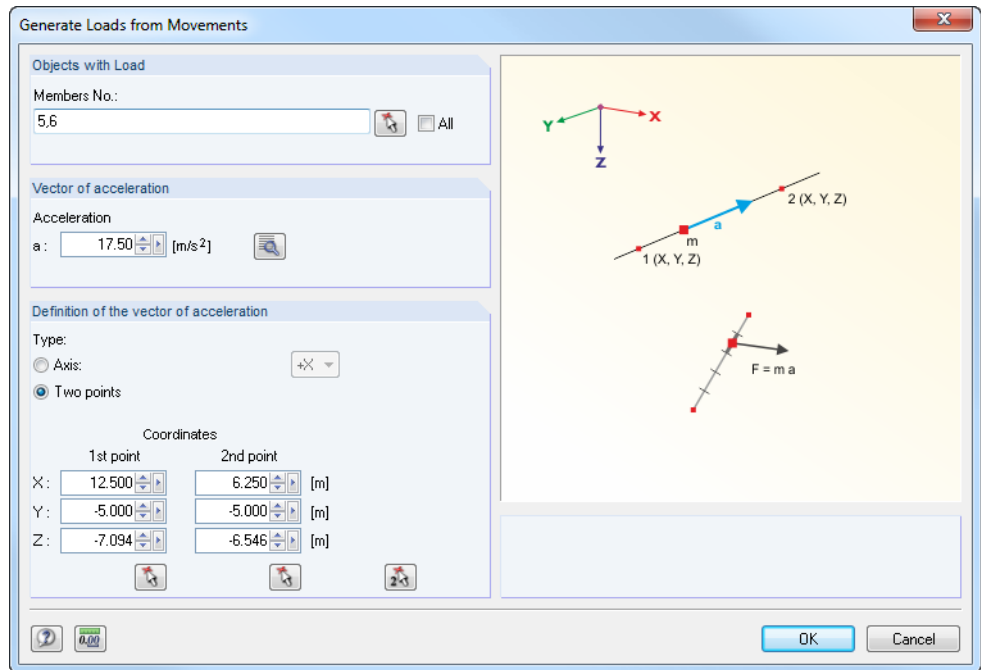


Figure 11.175: Dialog box *Generate Loads from Movements*

The generator creates loads as a result of an acceleration acting on particular members. The mass is determined from the self-weight.



In the dialog section *Objects with Load*, enter the numbers of the relevant members. You can select them also graphically with the [↖] function.



Specify the acceleration in the dialog section *Vector of acceleration*. You can also use the button shown on the left to [Open] a separate dialog box where the acceleration can be determined from the velocities 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

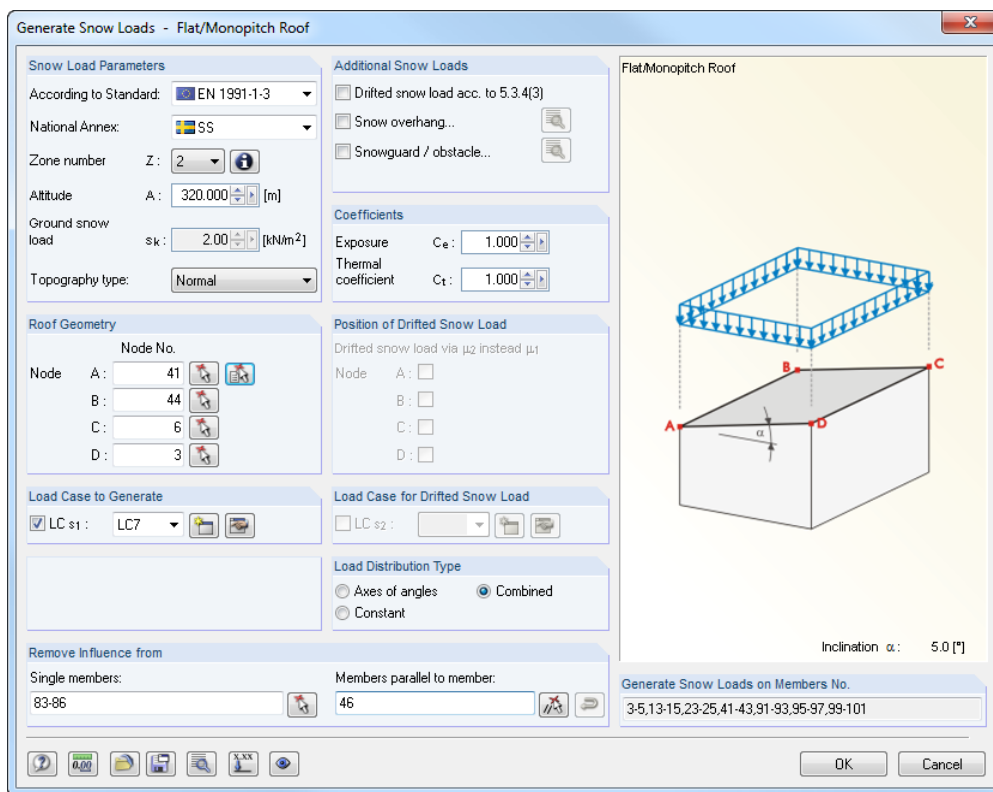


Figure 11.176: Dialog box *Generate Snow Loads - Flat/Monopitch Roof*

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, RSTAB 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 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.

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 358.



Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.161, page 354).



Use the button [Assign load correction factors] to scale the loads for particular members. Specifications will be entered in a separate dialog box (see Figure 11.170, page 360).



After confirming the generator dialog box with [OK], RSTAB 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 RSTAB, 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

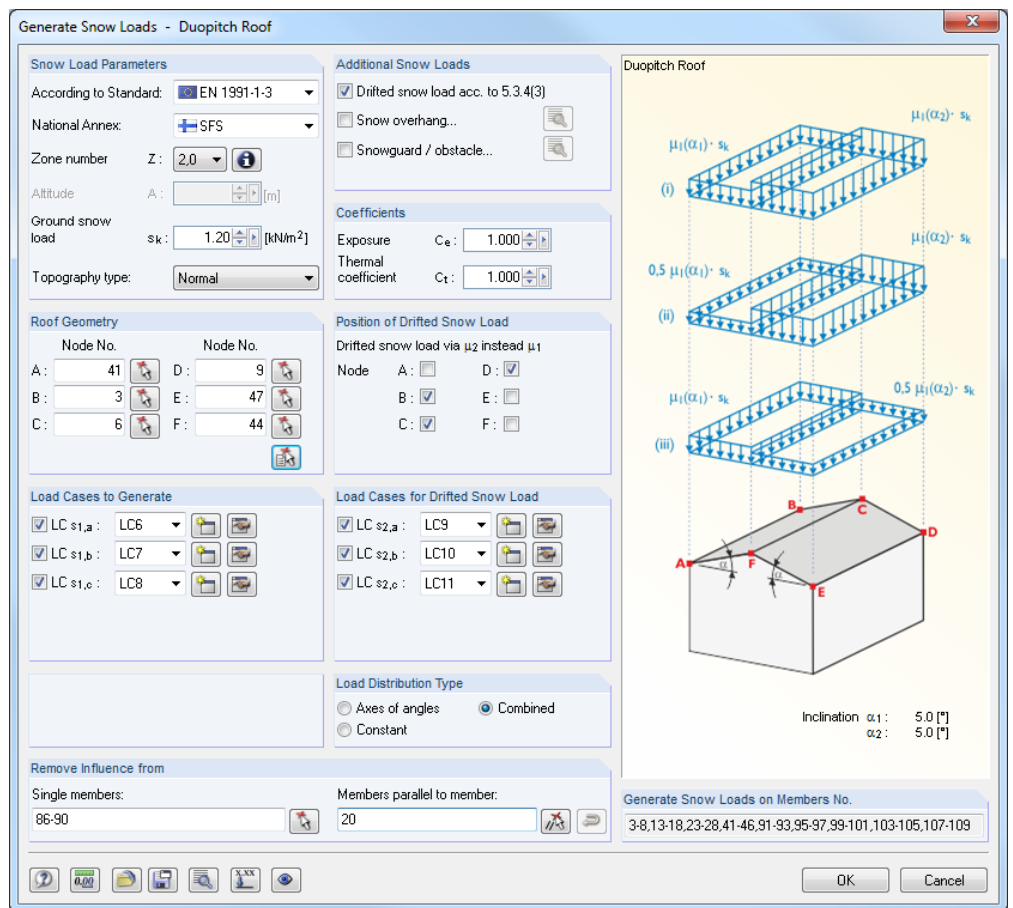


Figure 11.177: Dialog box *Generate Snow Loads - Duopitch Roof*

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

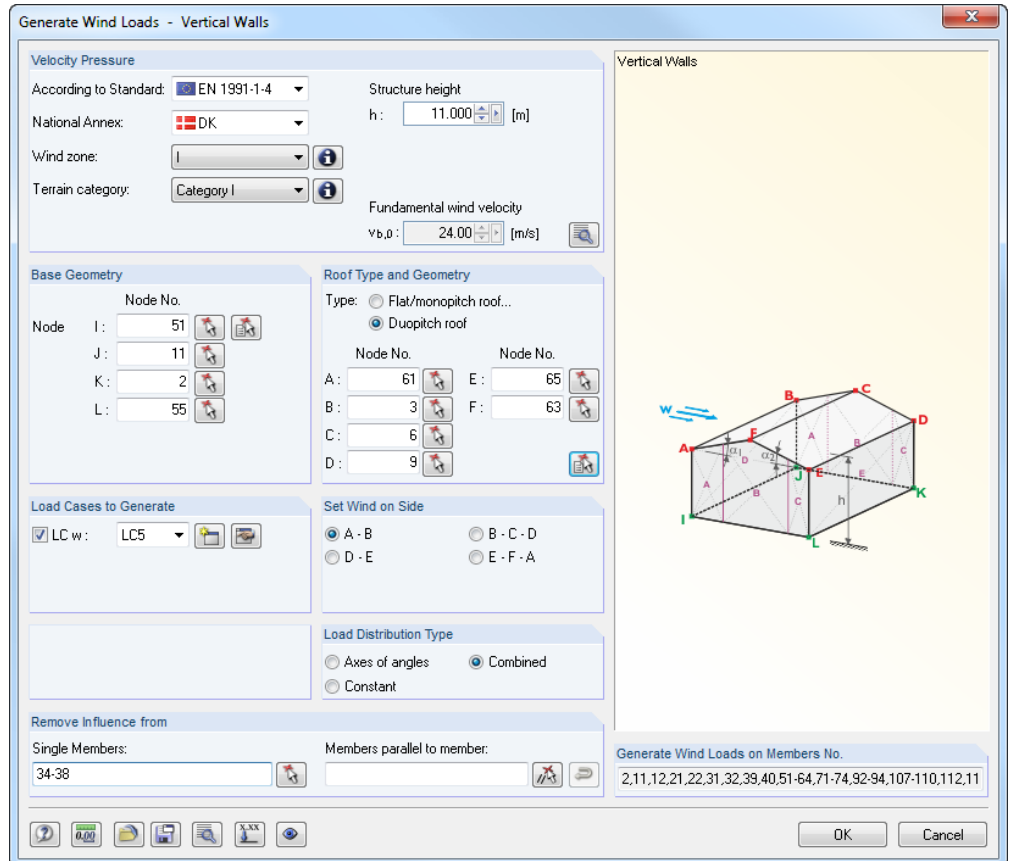


Figure 11.178: Dialog box *Generate Wind Loads - Vertical Walls* (roof geometry: *Duopitch roof*)

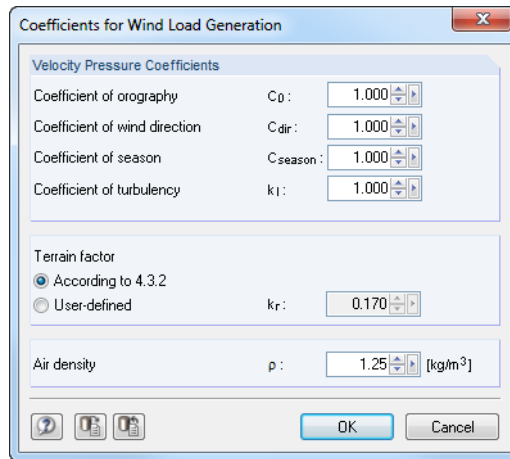
First, define the standard and, where applicable, the national annex in the dialog section *Velocity Pressure*. The setting controls the input fields enabled for access.



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, RSTAB 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 11.179: 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 Cases 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.

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 358.

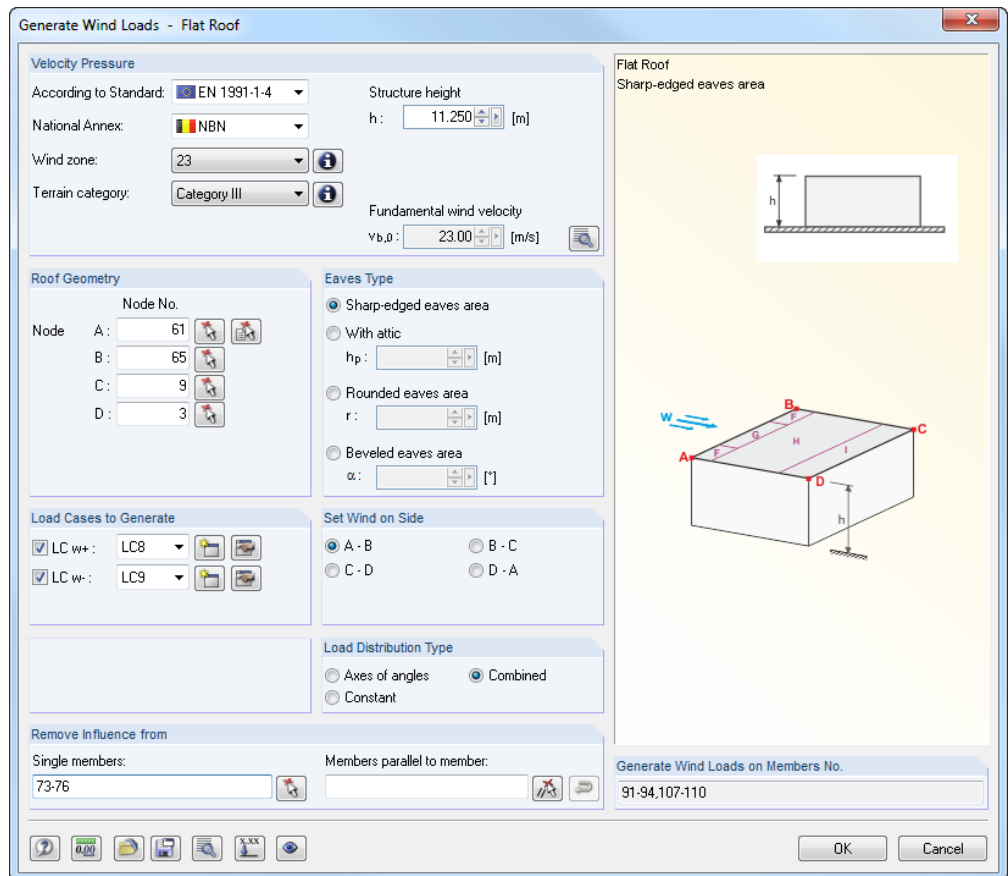


Click the [Settings] button shown on the left to open the dialog box *Settings for Load Generation* (see Figure 11.161, page 354).



After confirming the generator dialog box with [OK], RSTAB 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 RSTAB, 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 11.180: Dialog box *Generate Wind Loads - Flat Roof*

RSTAB considers a roof to be a flat roof if the roof inclination is  $\alpha < 5^\circ$ .

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], RSTAB shows you the results of the load generation for all load cases in an overview (see Figure 11.183, page 371). 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

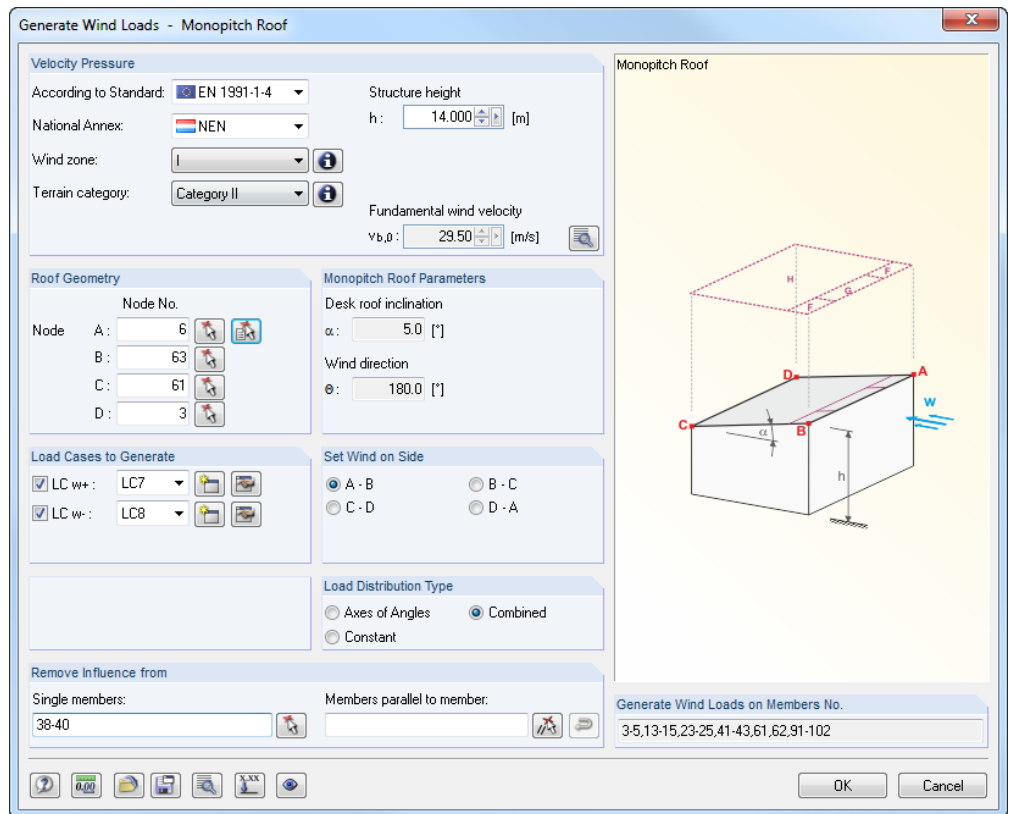


Figure 11.181: Dialog box *Generate Wind Loads - Monopitch Roof*

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.

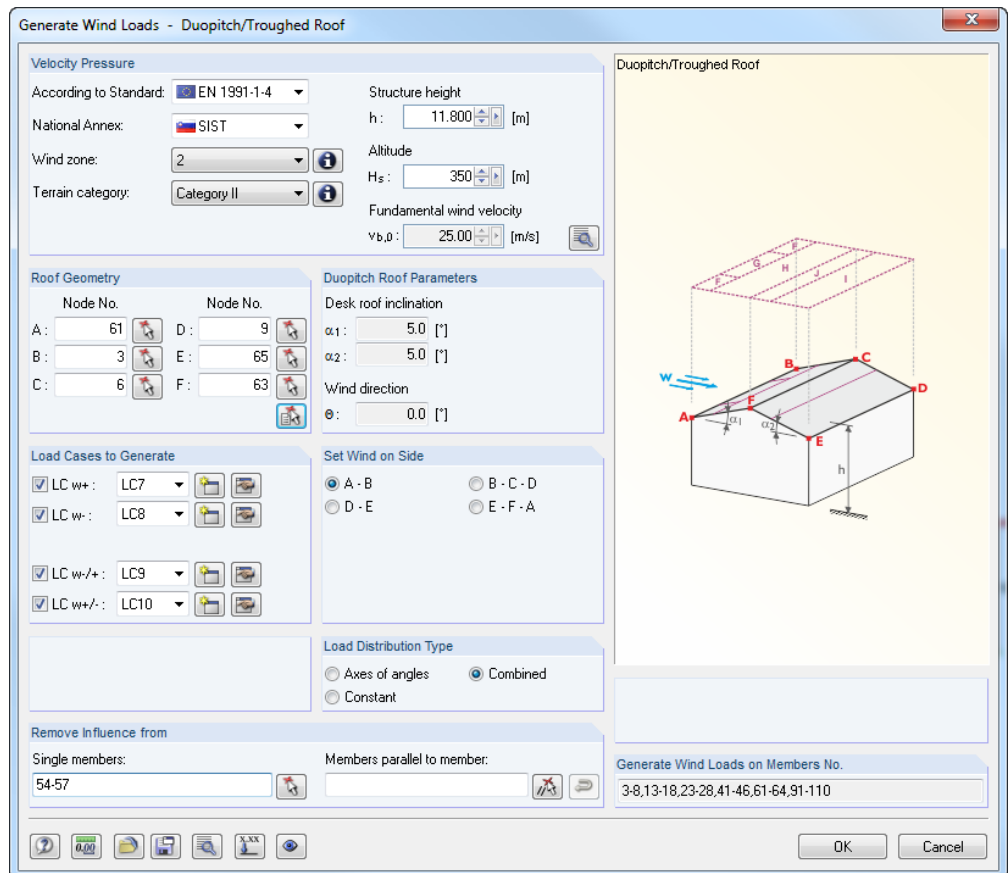


As described for example in EN 1991-1-4, table 7.3a, several load cases must be taken into account for a monopitch roof. In the dialog section *Load Cases to Generate*, specify the load case numbers for the load generation. The compression loads are created in the load case *LC w+*. The suction loads 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. Specifications will be entered in a separate dialog box (see Figure 11.170, page 360).

## 11.8.5.4 Duopitch/Troughed Roof

Figure 11.182: 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 366. 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], RSTAB 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.

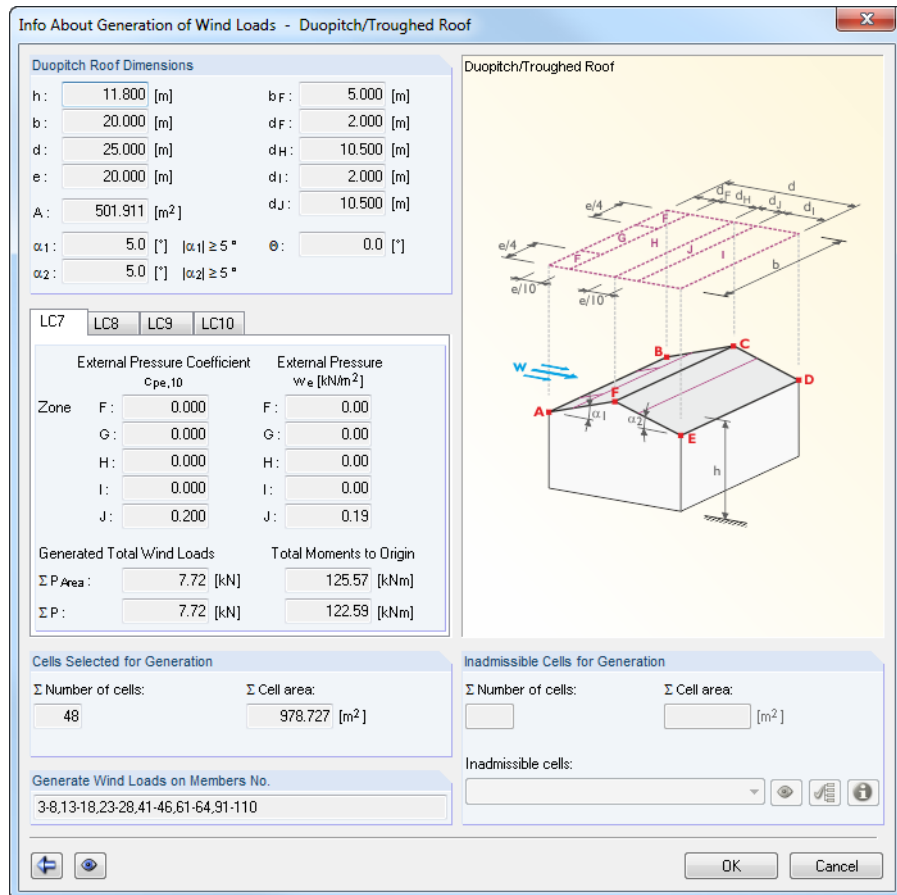
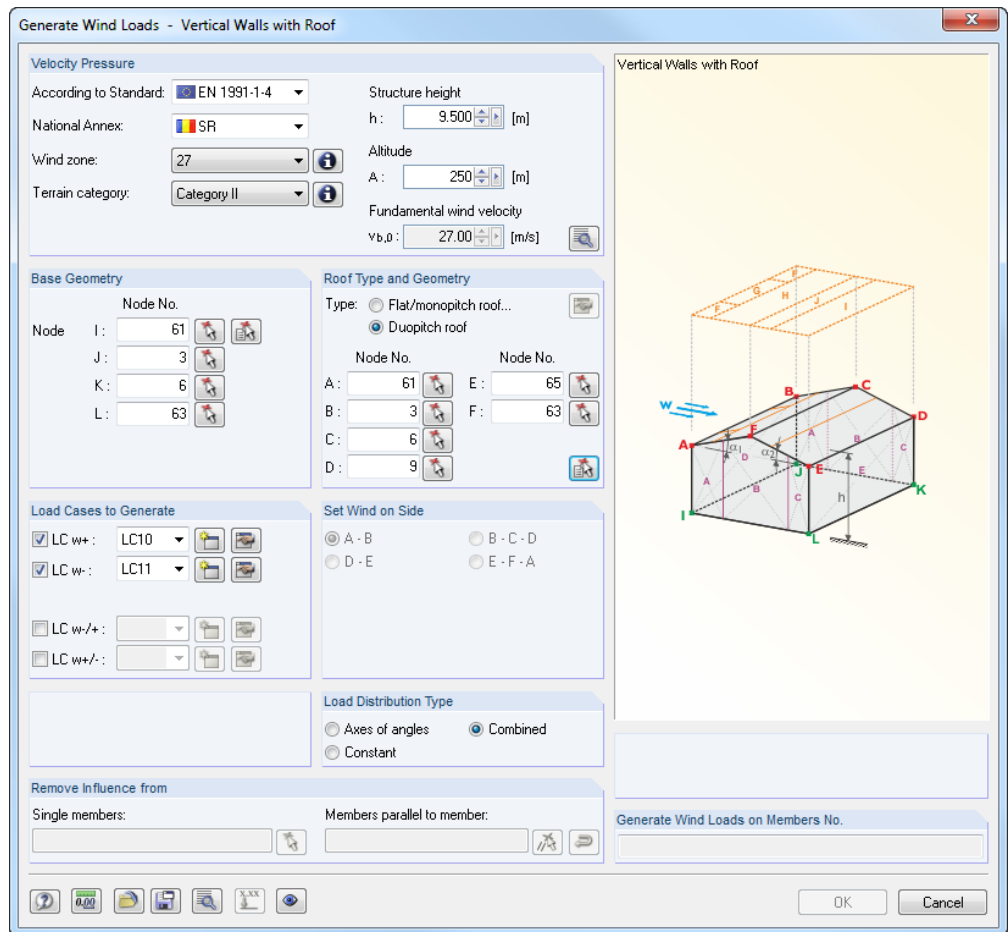


Figure 11.183: Dialog box *Info About Generation of Wind Loads - Duopitch/Troughed Roof*



Before the loads are transferred to RSTAB, 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 11.184: 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 366.



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. Specifications will be entered in a separate dialog box (see Figure 11.170, page 360).

After confirming the generator dialog box with [OK], RSTAB shows you the results of the load generation for all load cases in an overview (see Figure 11.183, page 371). 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 RSTAB models within the network (see chapter 12.3, page 393).

The Project Manager can be left open as a stand-alone application when working in RSTAB.

To open the Project Manager, select **Project Manager** on the **File** menu, or use the toolbar button shown on the left.

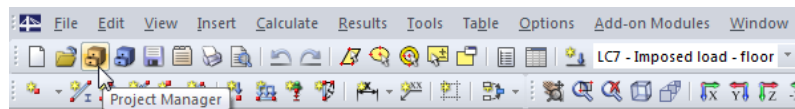


Figure 12.1: Button *Project Manager* in the toolbar



It is also possible to access the Project Manager in the model's *General Data* dialog box.

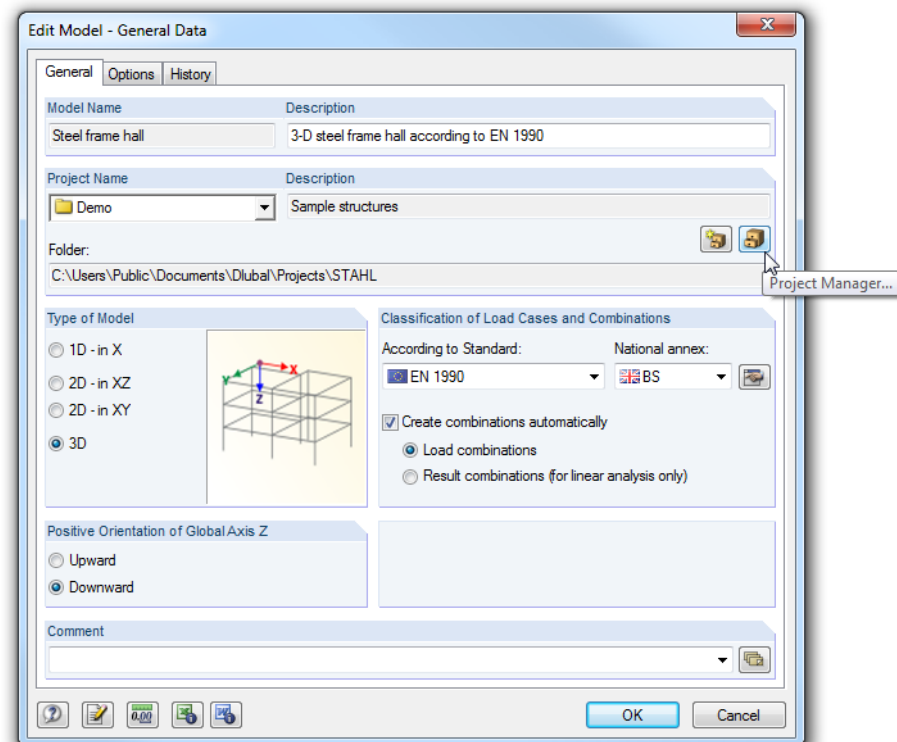


Figure 12.2: Button *Project Manager* in the dialog box *General Data*

When you open the Project Manager, the following multi-part window appears. It has its own menu and toolbar.

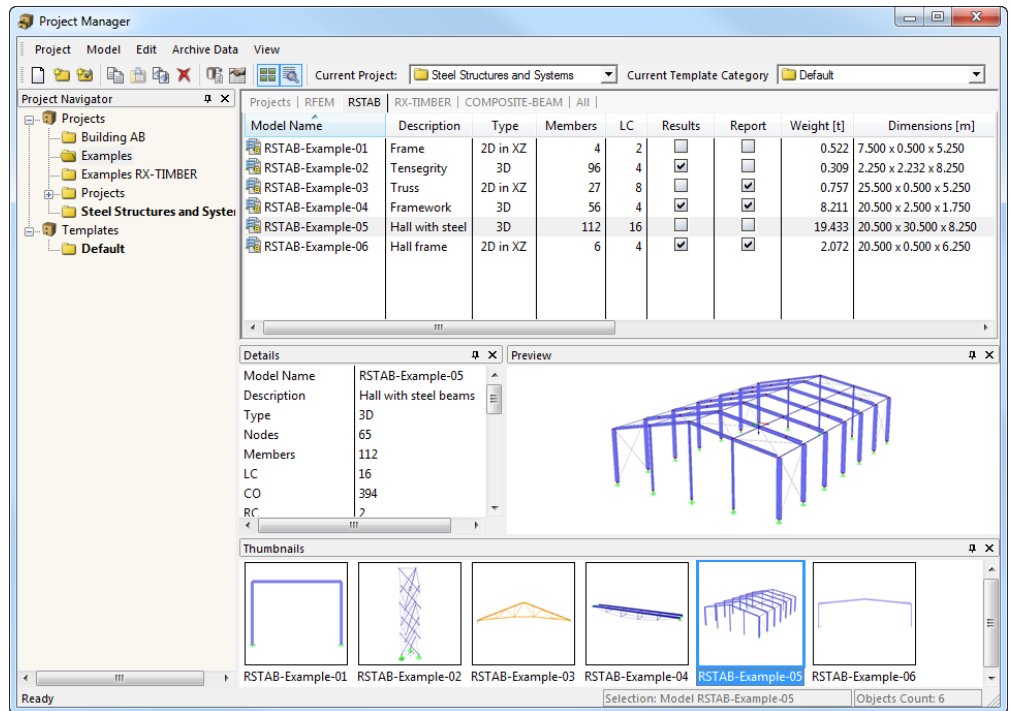


Figure 12.3: Project Manager

### 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 *RSTAB* tab lists all RSTAB 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 383).

### Details

This part of the window shows all information available for the model that is selected in the window section above.

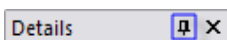
### Preview

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.

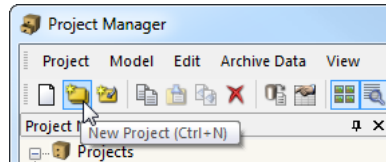


Figure 12.4: Button *New Project*

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.

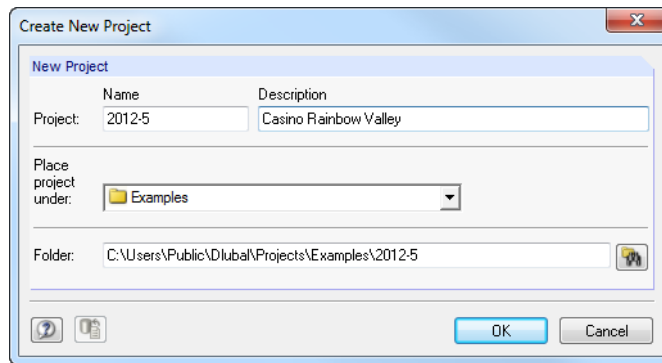


Figure 12.5: Dialog box *Create New Project*

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 RSTAB 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.

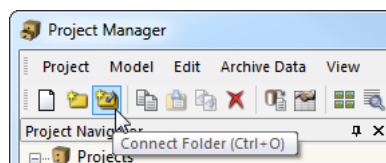
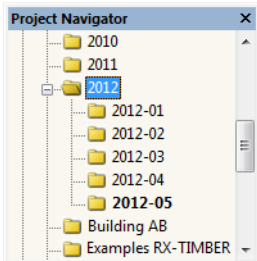


Figure 12.6: Button *Connect Folder*

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 **ProMan** (see chapter 12.1.4.3, page 385).





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.



By connecting folders it is possible to integrate projects of the program versions RSTAB 5 and RSTAB 6 into 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) or
- use the project's context menu in the navigator.

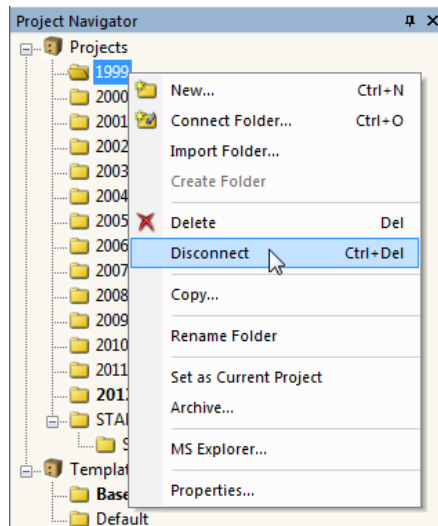


Figure 12.7: Context menu of a project



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 toolbar shown on the left
- use the **Delete** entry in the project's context menu available in the navigator (see figure above).

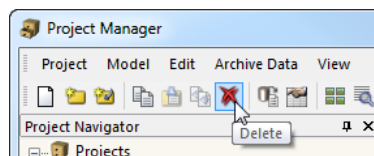


Figure 12.8: Button *Delete*

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 384.

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 to simply disconnect 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).

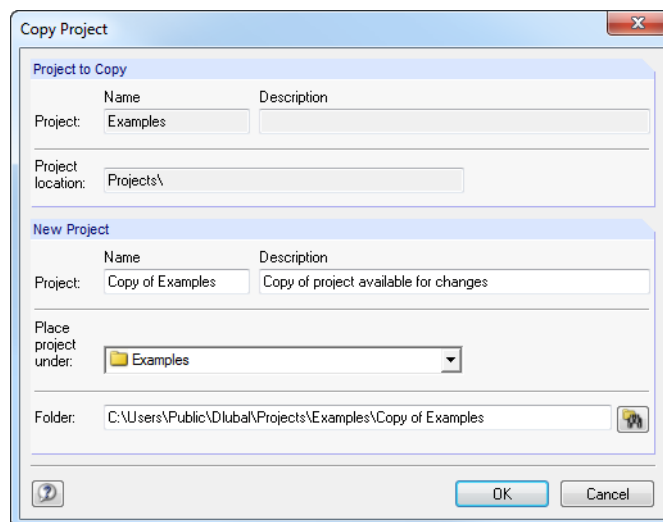


Figure 12.9: Dialog box *Copy Project*

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 chapter Figure 12.6, page 375).

### 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.

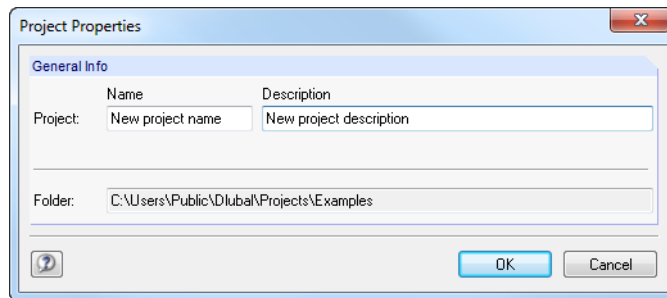


Figure 12.10: Dialog box *Project Properties*

### 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 393). 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.

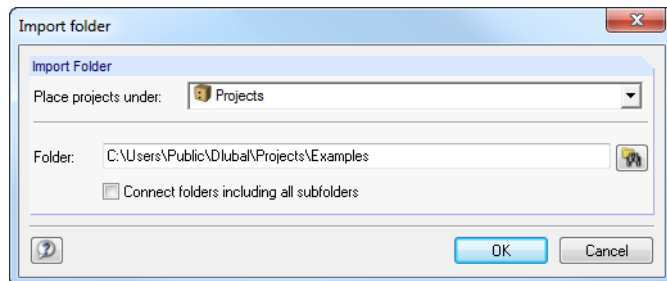


Figure 12.11: Dialog box *Import folder*



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.

## 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.

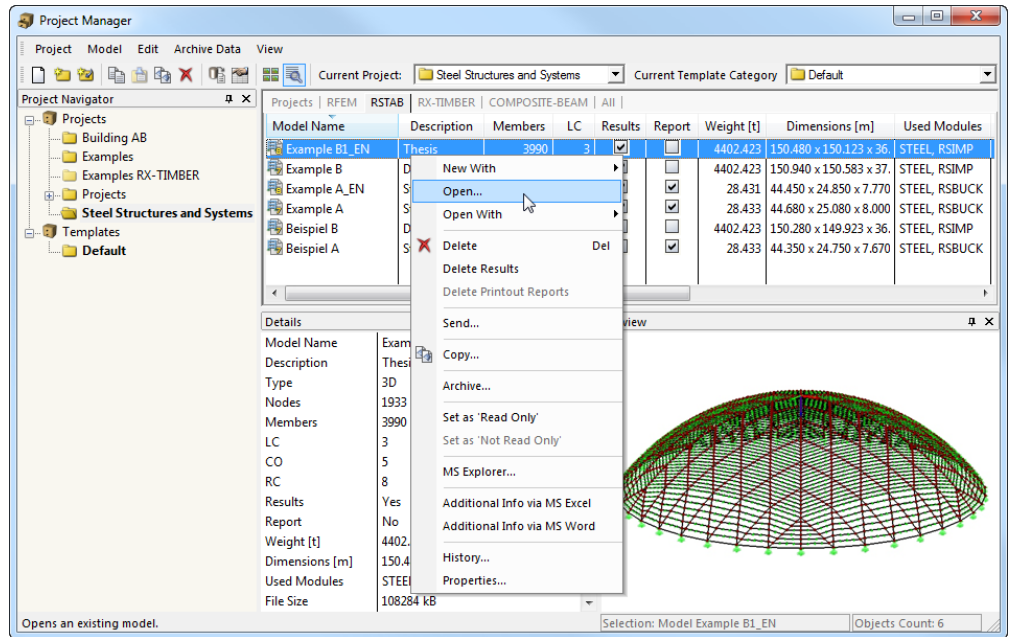
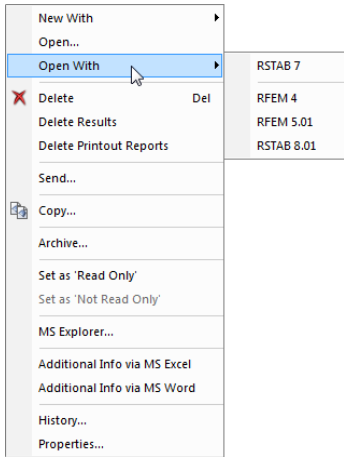


Figure 12.12: Context menu *Model*

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 RFEM directly in RSTAB.

### 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.

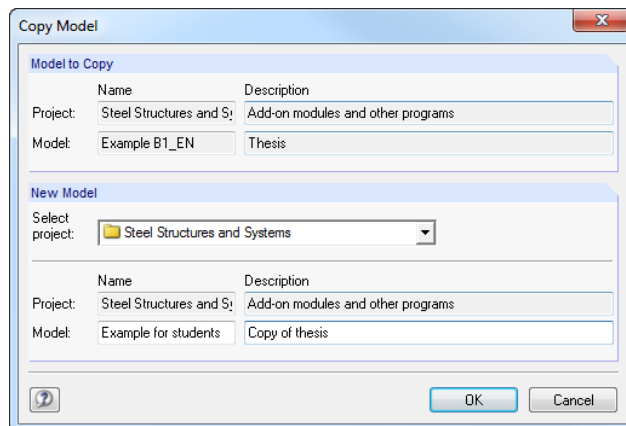


Figure 12.13: Dialog box *Copy Model*

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)
- use the **Properties** entry in the model's context menu (see Figure 12.12).

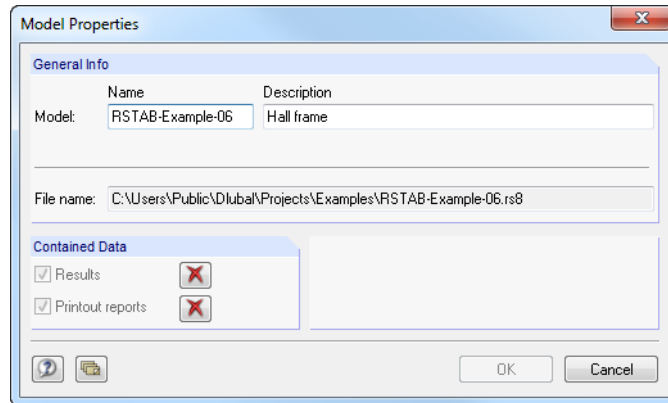


Figure 12.14: Dialog box *Model Properties*

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 384.

### 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)
- use the **History** entry in the model's context menu (see Figure 12.12).





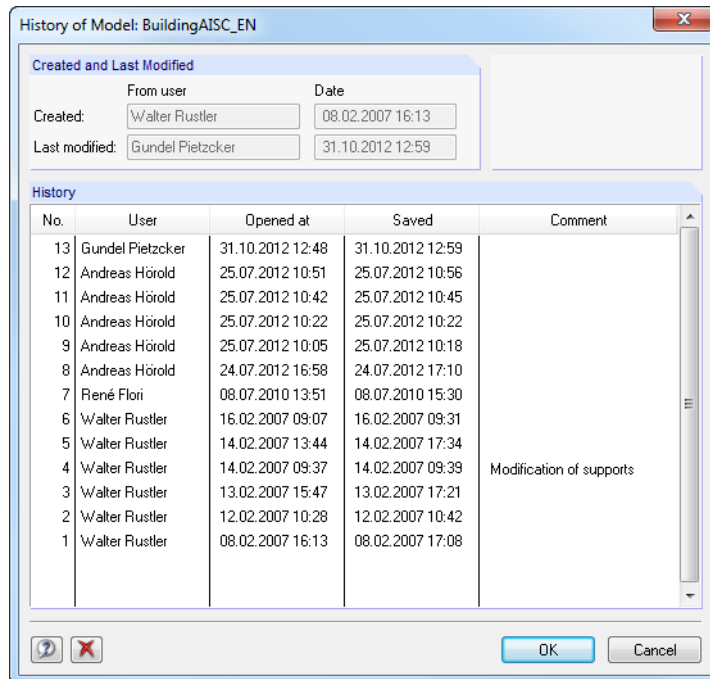


Figure 12.15: Info window *History of Model*

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 12.2.3, page 392).

### 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)
- use the context menu of the project (see Figure 12.7) or model (see Figure 12.12).

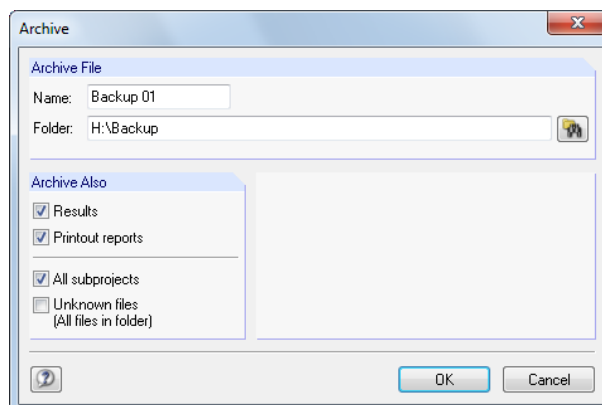


Figure 12.16: Dialog box *Archive*

The backup file can be generated with or without results and printout reports. Further options allow for the integration of subprojects and files that do not belong to any of the Dlubal applications.

When the *Name* and *Folder* of the archive file are defined, you can create a 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], the contents are displayed.

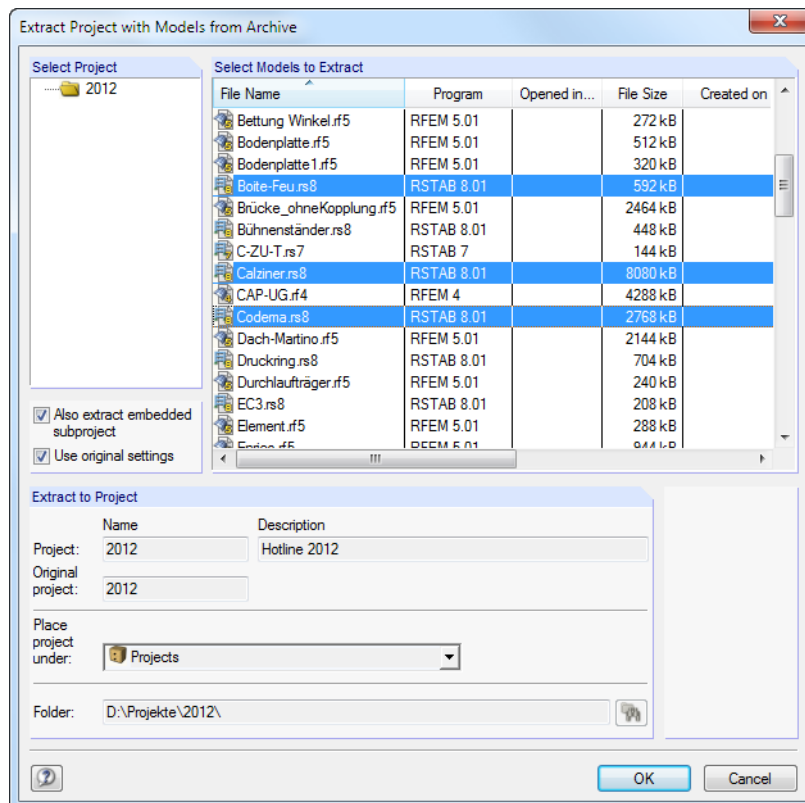


Figure 12.17: Dialog box *Extract Project with Models from Archive*



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

#### Sort 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.

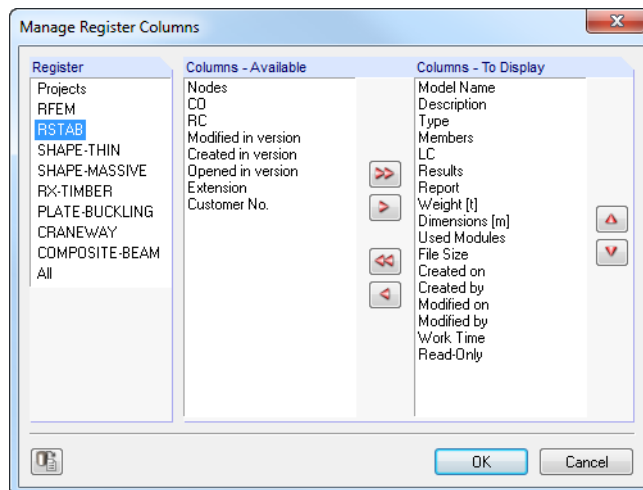


Figure 12.18: Dialog box *Manage Register Columns*



First, define the *Register* whose columns you want to adjust (for example RSTAB). 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.

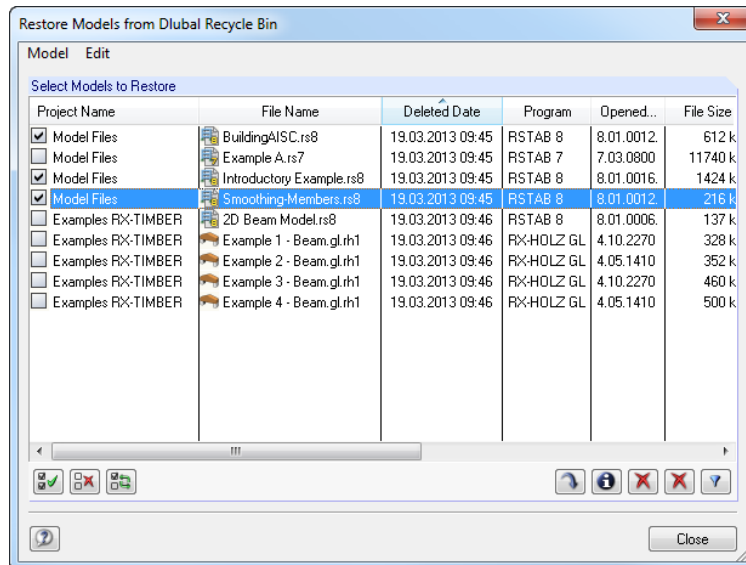


Figure 12.19: Dialog box *Restore Models from Dlubal Recycle Bin*



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.

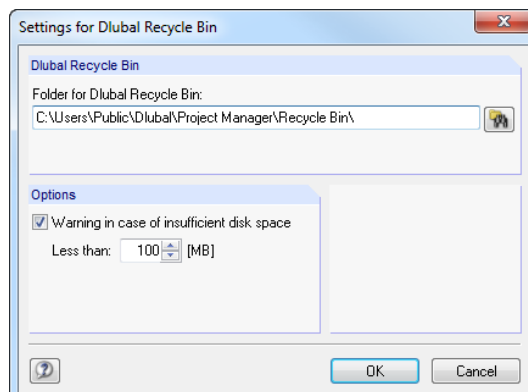


Figure 12.20: Dialog box *Settings for Dlubal Recycle Bin*

### 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.

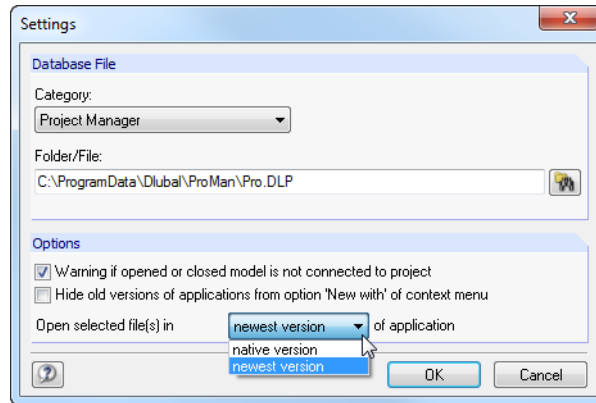


Figure 12.21: Dialog box *Settings*



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\ProMan* (Windows 7) or *C:\Documents and Settings\All Users\Application Data\Dlubal\ProMan* (Windows XP). 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 393).

The dialog section *Options* offers you general settings for handling RSTAB 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 RSTAB **File** menu
- 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 **RSTAB 8**.

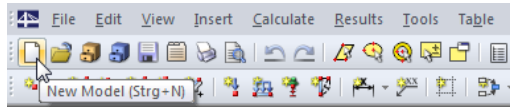


Figure 12.22: Button *New Model*

The dialog box *New Model - General Data* opens offering three tabs.

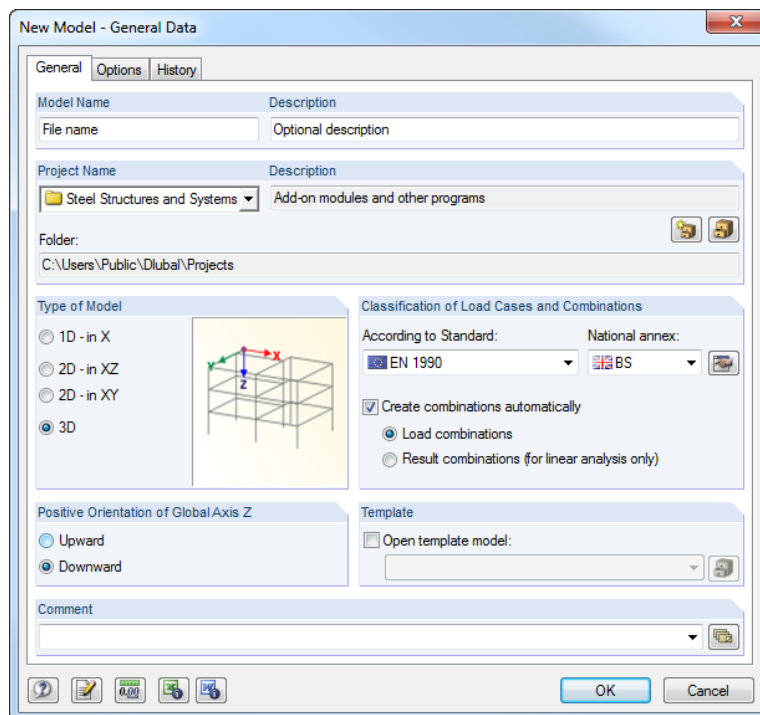


Figure 12.23: Dialog box *New Model - General Data*, tab *General*

When you want to edit the model's general data later,

- point to **Model Data** on the **Edit** menu, and then select **General Data**
- use the context menu of the model in the *Data* navigator.

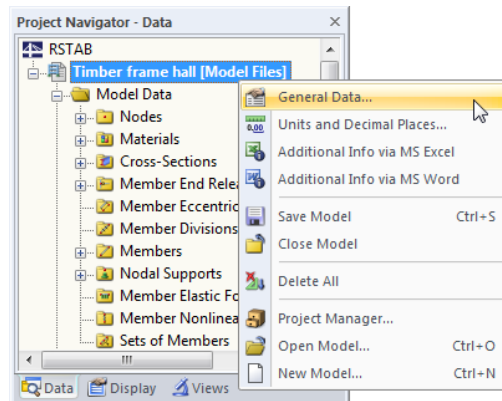


Figure 12.24: Context menu of model

## 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 373) 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

Among 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.

Type *1D - in X* represents a continuous beam. The use of type *2D - in XZ* is recommended for planar frame structures where moments are taken into account only about the strong member axes. Option *2D - in XY* is appropriate for girder grillages with loads acting perpendicular to the area plane.

It is possible to change the selected type of model subsequently. Please note that such a modification may result in data loss, for example when a 3D model is reduced to a grillage.

### Classification of load cases and combinations

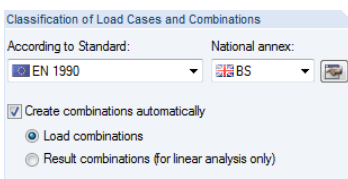
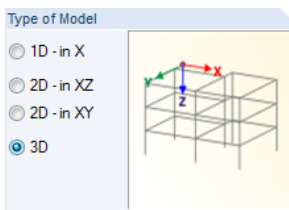
Loading is to be applied by load cases. Load cases may be for example self-weight, snow or live load.

The individual standards define rules for how to combine load cases. It is therefore important to assign load cases to particular action categories (see chapter 5.1, page 100). Thus, when creating load and result combinations, RSTAB 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 RSTAB are defined. This specification is especially significant for the automatic creation of combinations by RSTAB (see chapter 5.2, page 103 to chapter 5.4, page 116).

When *None* is set, no combination will be created automatically, which corresponds to the default setting in RSTAB 7. In this case, load cases must be superimposed manually (see chapter 5.5.1, page 121 and chapter 5.6.1, page 128).



When changing the standard subsequently it is required to reclassify the load cases and to adjust the combination. A corresponding warning appears.

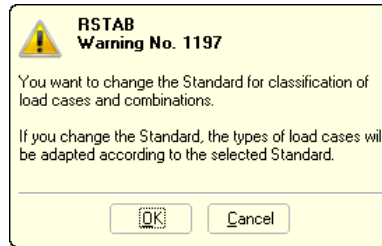


Figure 12.26: Warning when changing the standard



### 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*  $\gamma$  for the design situations "static equilibrium" and "ultimate limit state".

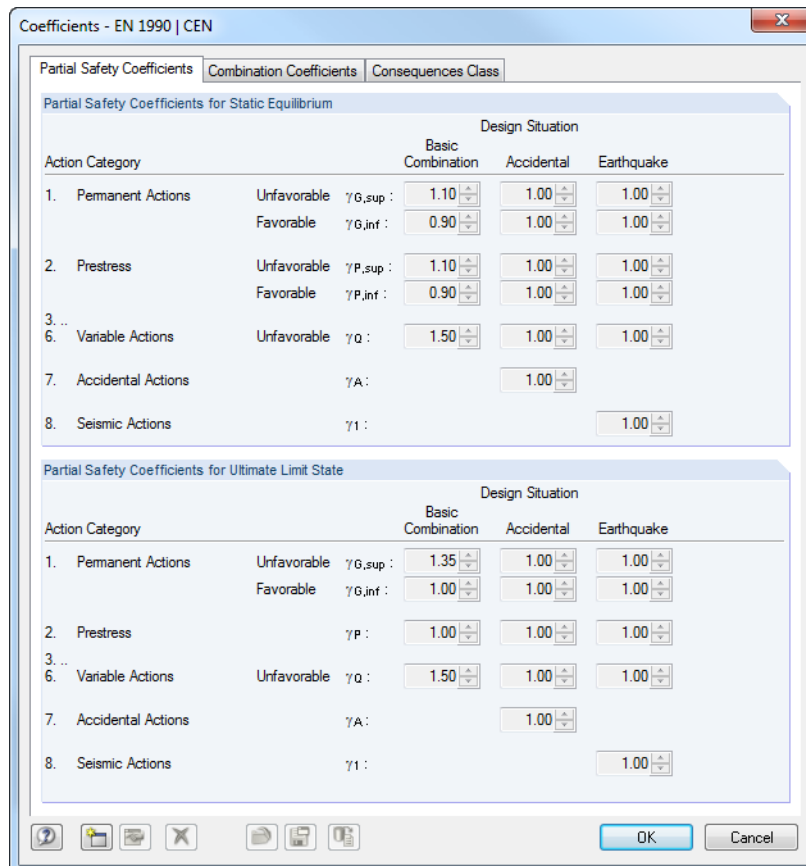


Figure 12.27: Dialog box *Coefficients*, tab *Partial Safety Coefficients*



The dialog tab *Combination Coefficients* (see Figure 5.22, page 118) controls the factors  $\psi$  and  $\xi$ . In the tab *Factors of Construction*, 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 RSTAB 7 (see chapter 5.5.1, page 121 and chapter 5.6.1, page 128). 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 103 to chapter 5.4, page 116.



The automatic superposition is based on the concept of the add-on module RSCOMBI 2006. Find additional information about combinatorics in the module's manual that is available for download at [www.dlubal.com](http://www.dlubal.com).

During the automatic superpositioning RSTAB creates either *Load combinations* or *Result combinations*. The difference between both combination possibilities is described in chapters 5.5 on page 120 and 5.6 on page 128.

### 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. Where necessary, the self-weight factor must be changed to  $-1.0$ .



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. If the axis direction is modified, the following query appears:

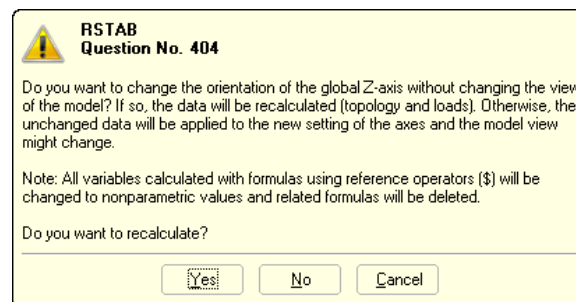
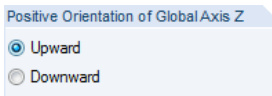
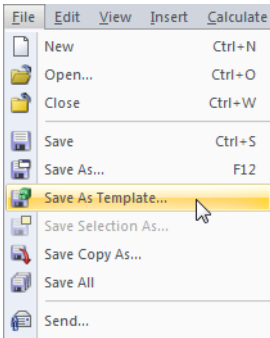


Figure 12.28: Query when changing the Z direction



### 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.

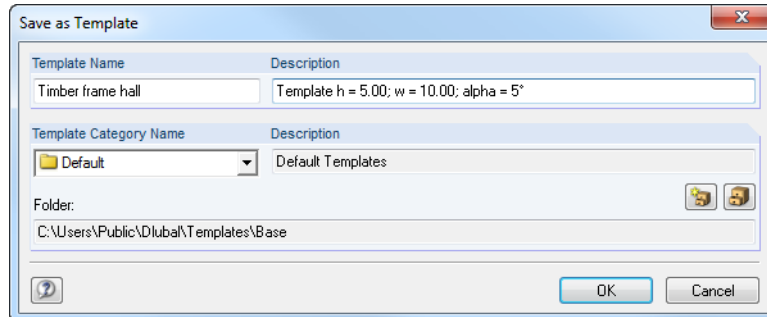


Figure 12.29: Dialog box *Save as Template*

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 374).

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.

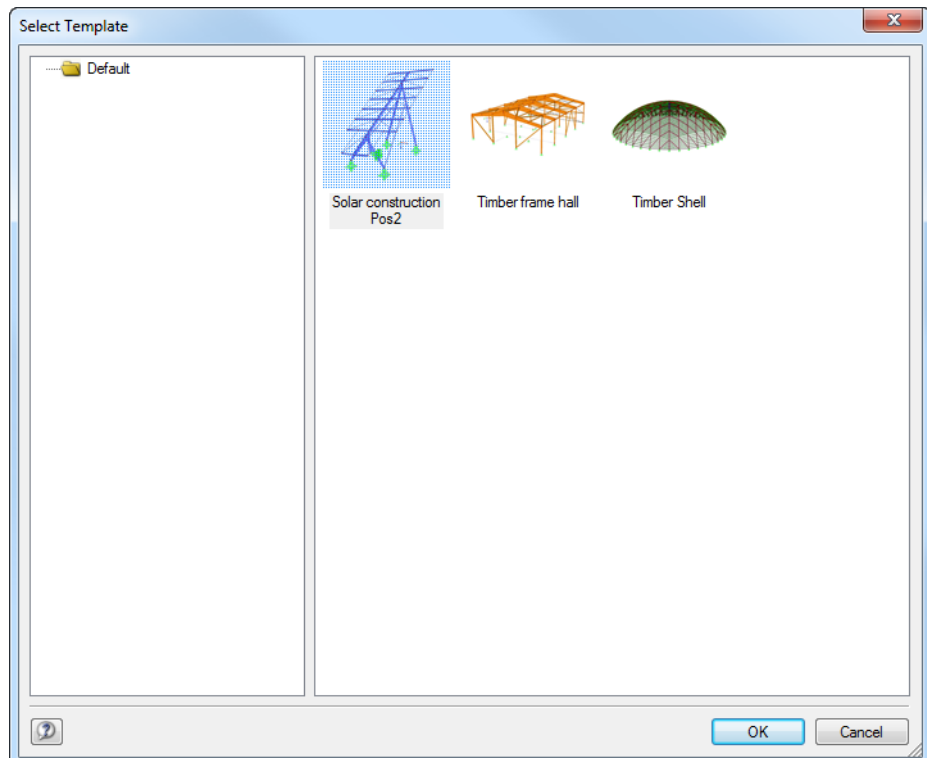
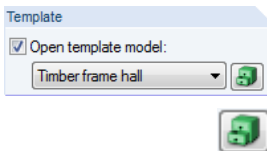


Figure 12.30: Dialog box *Select Template*

### 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 are reserved for the following functions:





Button	Description	Explanation
	Comment	→ chapter 11.1.4, page 262
	Units and Decimal Places	→ chapter 11.1.3, page 261
	MS Excel	Option for additional explanations in the form of XLS file saved in RSTAB file
	MS Word	Option for additional explanations in the form of DOC file saved in RSTAB file

Table 12.2: Dialog box *General Data*, buttons

### 12.2.2 Options

With the second tab of the dialog box *New Model - General Data* you decide if additional functions are displayed in the load case and combination dialog boxes.

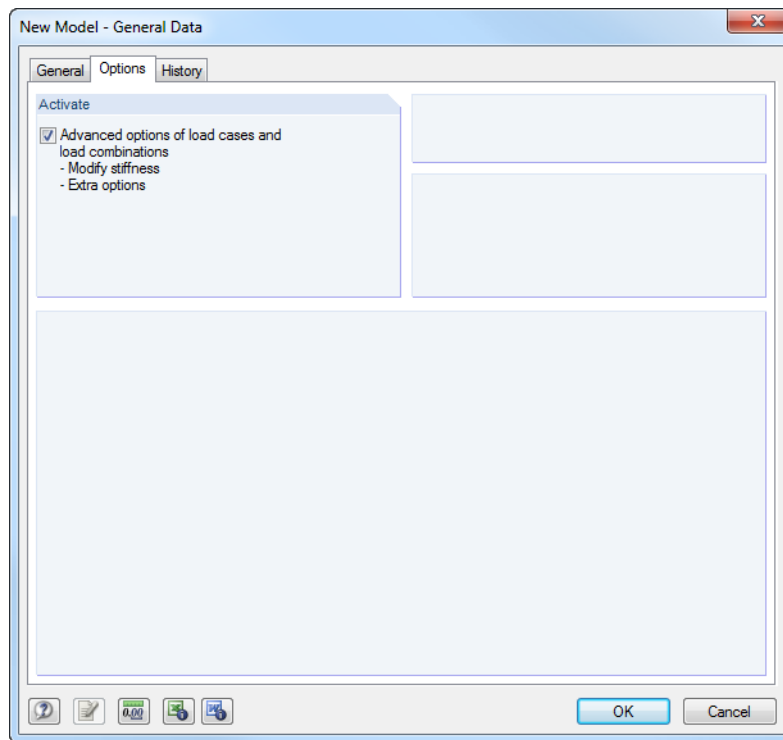


Figure 12.31: Dialog box *New Model - General Data*, tab *Options*

If the check box for *Advanced options of load cases and load combinations* is ticked, additional dialog tabs can be accessed in the dialog box *Edit Load Cases and Combinations*:

- *Modify stiffness* (see chapter 7.2.1.2, page 173)
- *Extra options* (see chapter 7.2.1.3, page 174)

In the additional tabs, you can adjust the stiffnesses of materials, cross-sections, supports and releases. Moreover, you can take account of the initial deformations from another load case.

### 12.2.3 History

The third dialog tab keeps record of processing in the form of a *History*.

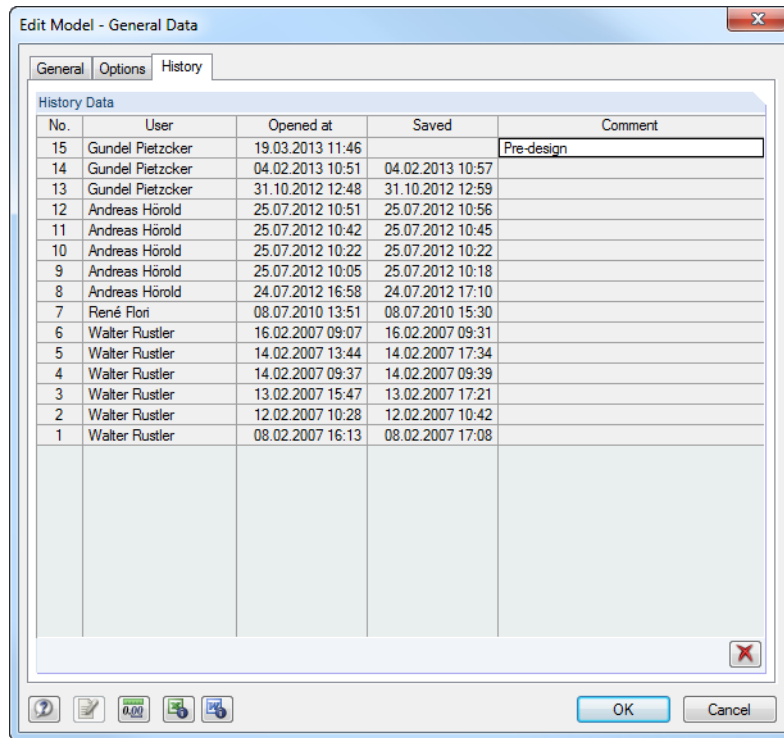


Figure 12.32: Dialog box *Edit Model - General Data*, tab *History*

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 381).



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 375. 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.

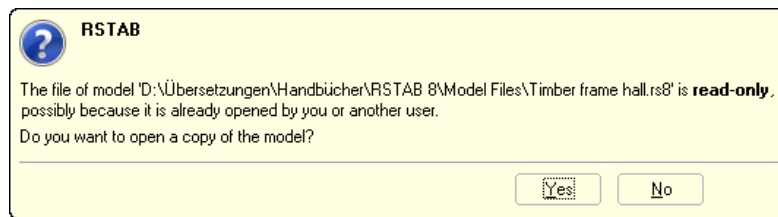


Figure 12.33: Query when opening a write-protected model



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:\Documents and Settings\All Users\Application Data\Dlubal\ProMan* (Windows XP) or *C:\ProgramData\Dlubal\ProMan* (Windows 7).

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 378).

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 385).

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 RSTAB, or use the toolbar button shown on the left.

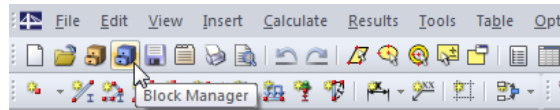


Figure 12.34: Button *Block Manager* in the toolbar

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.

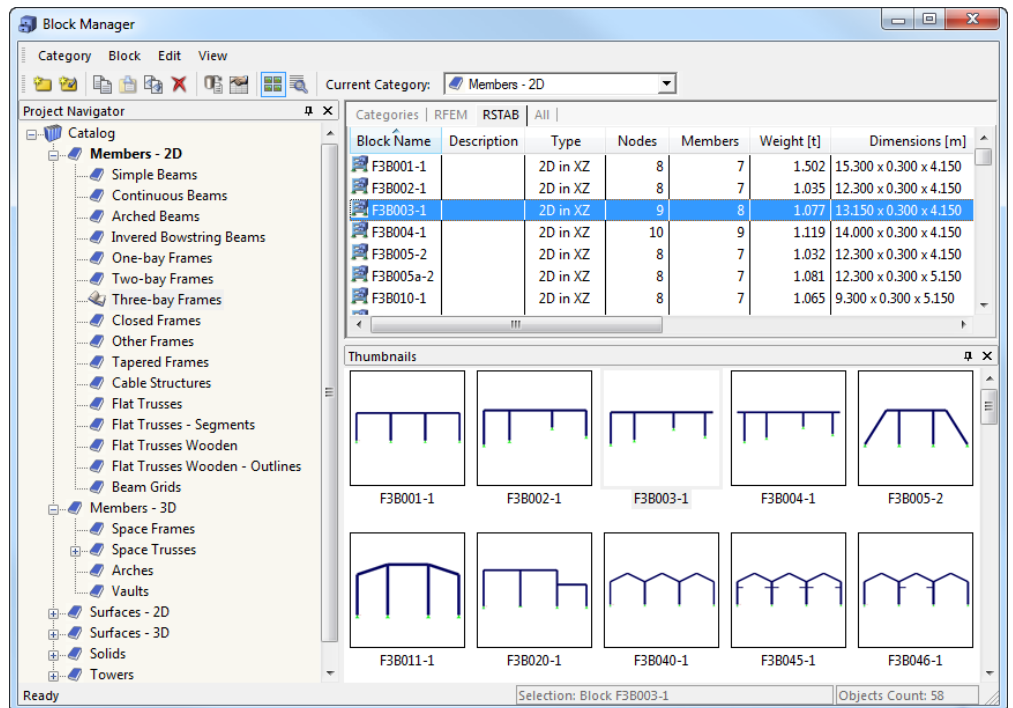


Figure 12.35: Block Manager

### Block 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. The following categories can be used for RSTAB:

- Members - 2D
- Members - 3D
- Towers

### 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 383).

### Details

The window section shows you detailed information about the selected block.

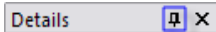
### Preview

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 RSTAB 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 RSTAB.

The following dialog box appears.

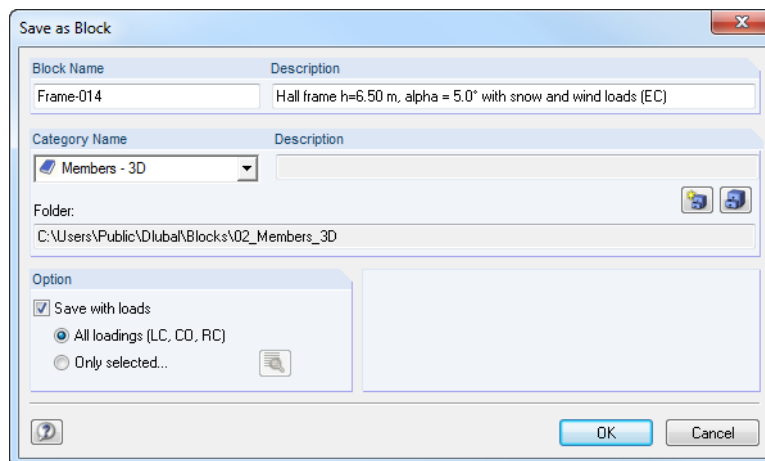


Figure 12.36: Dialog box *Save as Block*

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.

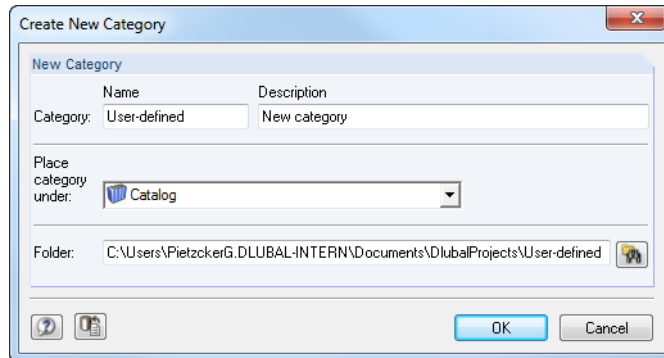


Figure 12.37: Dialog box *Create New Category*

The creation of a block is similar to the creation of a new project in the Project Manager (see chapter 12.1.1, page 375).

## 12.4.2 Import a Block



To import a block into the current RSTAB model, open the Block Manager (see Figure 12.34, page 394). First, select the category in the catalog. Then, you can select the relevant block with a mouse click in the *RSTAB* tab.

To start the import,

- select **Insert Block** on the **Block** menu
- use the context menu of the block.

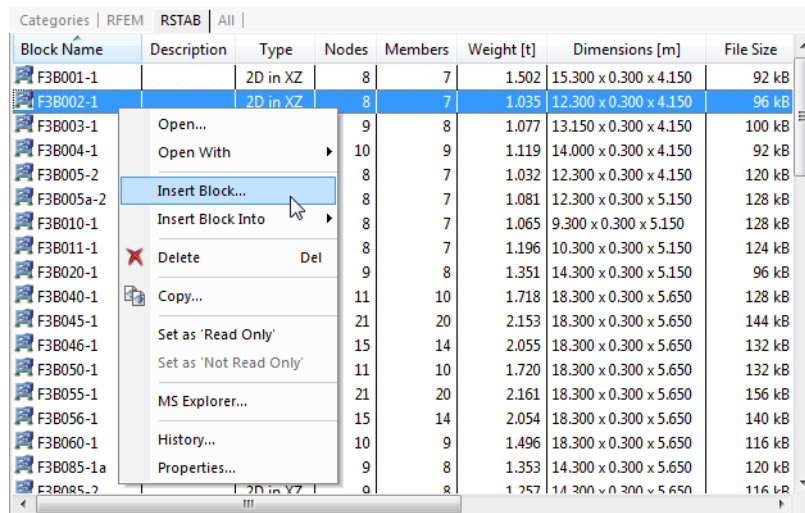


Figure 12.38: Context menu of block

You can also double-click the block in the table. The following dialog box opens.



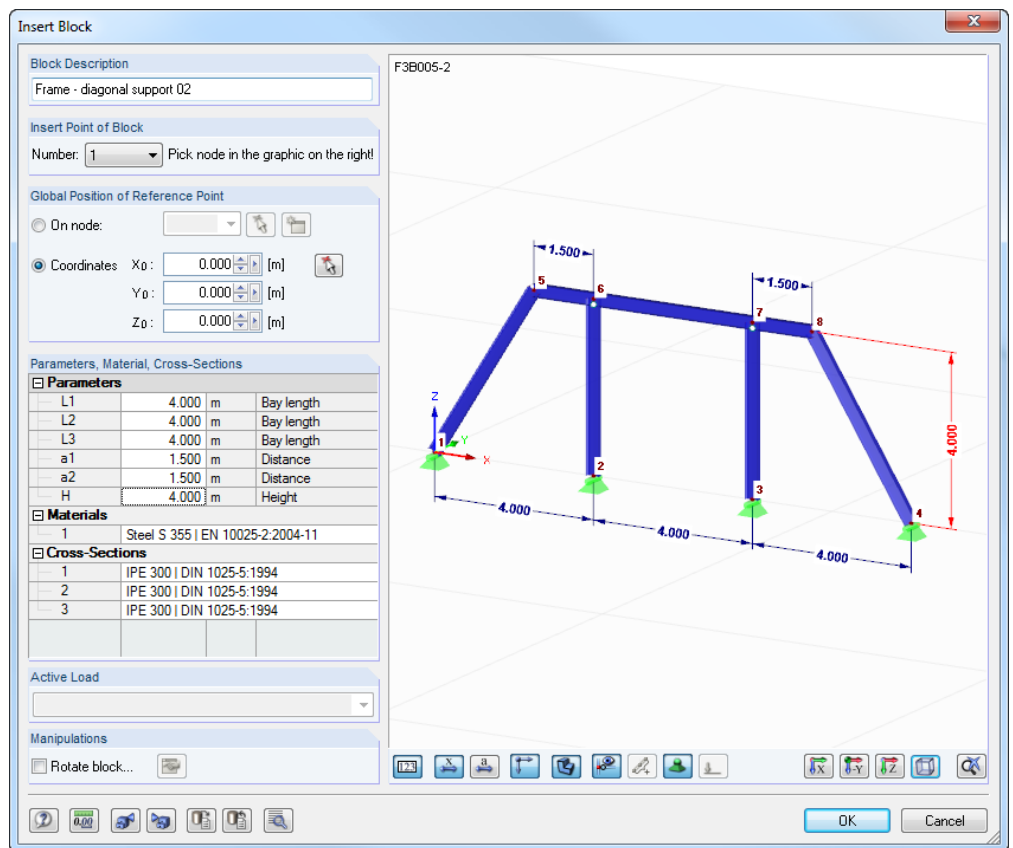


Figure 12.39: Dialog box *Insert Block*



Specify the *Insert Point of Block* (the "snap point") and the *Global Position of Reference Point* in the dialog box. The points can be selected also graphically in the block model or RSTAB 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.

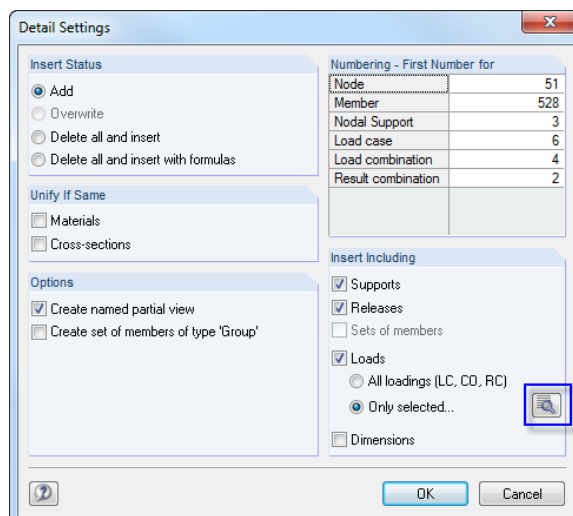


Figure 12.40: Dialog box *Detail Settings*

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.

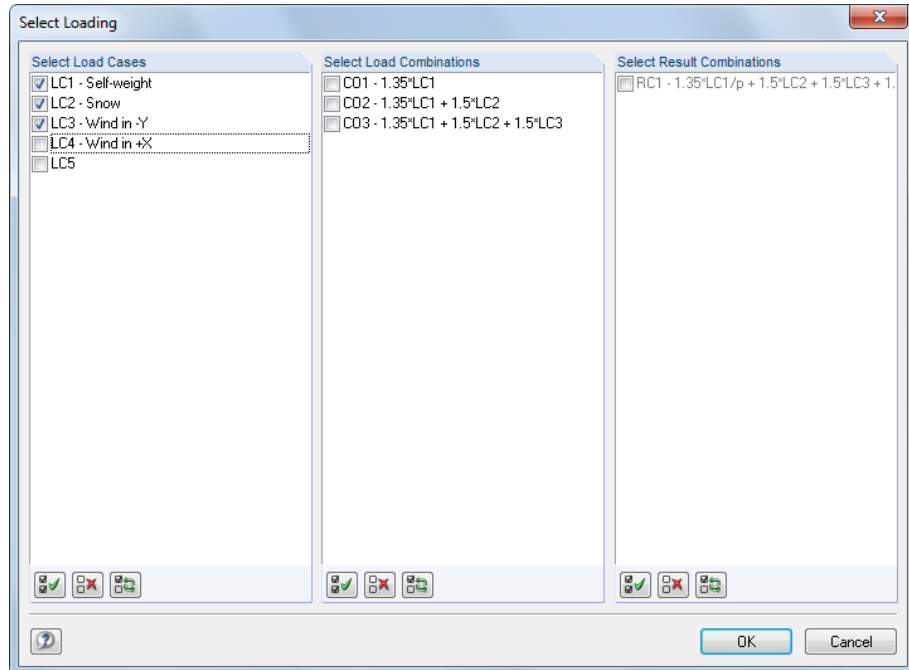


Figure 12.41: Dialog box *Select Loading*

### 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).

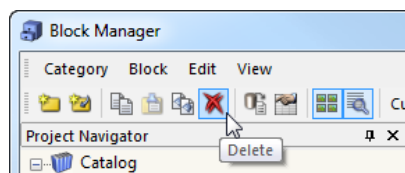


Figure 12.42: Button *Delete*

After confirming the security query, the block will be put into the Dlubal recycle bin.

## 12.5 Interfaces

RSTAB 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 244.

In addition, RSTAB can be run externally using a programmable interface based on COM technology (for example Visual Basic). With **RS-COM** which can be acquired as RSTAB add-on module you can use customized input macros and follow-up programs.

### 12.5.1 Direct Data Exchange

RSTAB provides an interface to software programs developed by the DLUBAL company. Input data of all previous **RSTAB** versions can be imported without problems. Furthermore, you can directly open files created with the finite element program **RFEM** (surfaces and solid elements will be ignored during the import). In the same way, you can open files created with RSTAB 8 in RFEM 5 where you can add surfaces and solids.

RSTAB has 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 RSTAB 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 RSTAB or use the toolbar buttons shown on the left.

The dialog box shown in Figure 12.43 or Figure 12.44 on page 400 opens where you can select the relevant CAD program in the dialog section *Direct Imports* or *Direct Exports*.

The buttons in the RSTAB toolbar *Export/Import* are reserved for 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](http://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 RSTAB. Vice versa, RSTAB 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.

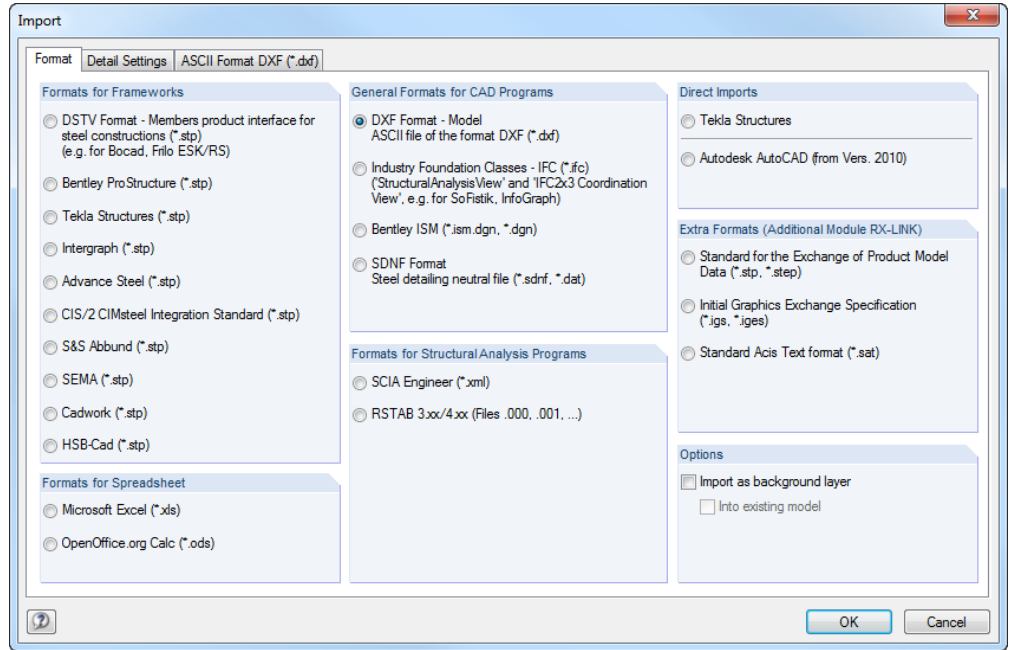


Figure 12.43: Dialog box *Import*

When the option *Import as background layer* is ticked, RSTAB will show you only a line model that can be used to set nodes, members etc. (see chapter 11.3.10, page 297).



To start the export of a RSTAB file, select **Export** on the **File** menu.

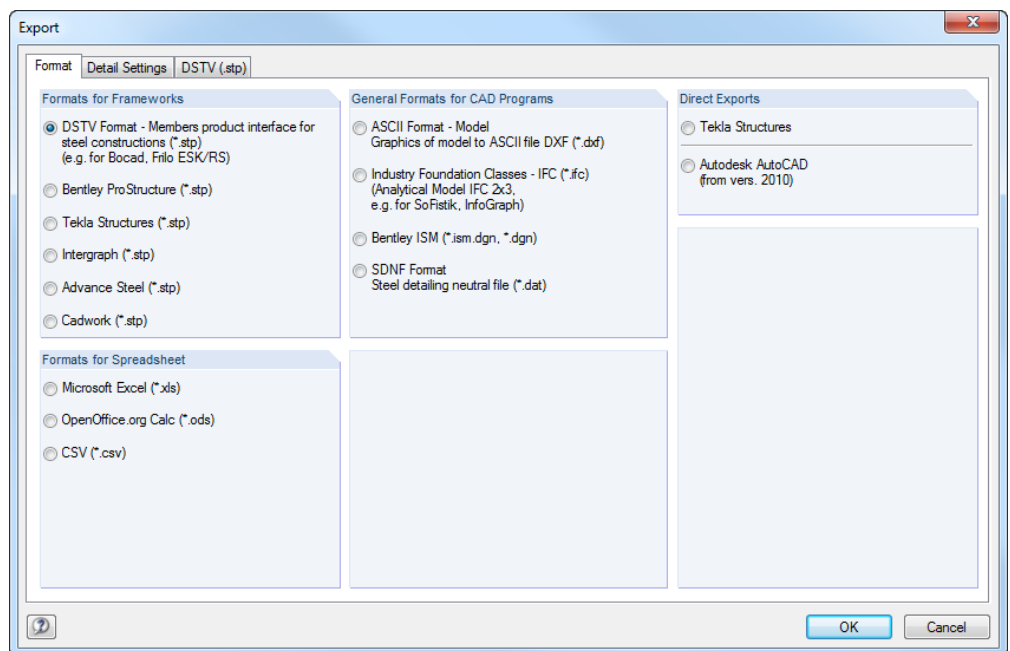


Figure 12.44: Dialog box *Export*

### 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*, *SEMA* 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. RSTAB, however, supports only the structural format with specific "entities" that can be found in a description (German only, PDF download of [Standardbeschreibung Produktschnittstelle Stahlbau](#)).

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)*.

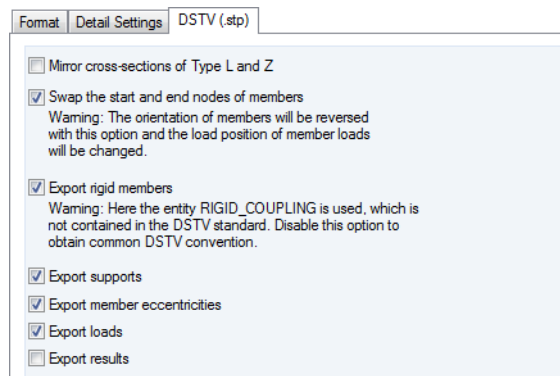


Figure 12.45: Dialog box *Export*, tab *DSTV (.stp)*

### File formats for spreadsheet

#### MS Excel format \*.xls

RSTAB is able to import and create tables as \*.xls files. Exchanging data with MS Excel is described in chapter 11.5.6 on page 329. However, the described exchange option is only available for the active RSTAB 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 RSTAB import dialog box (see Figure 12.43) to open the following dialog box.

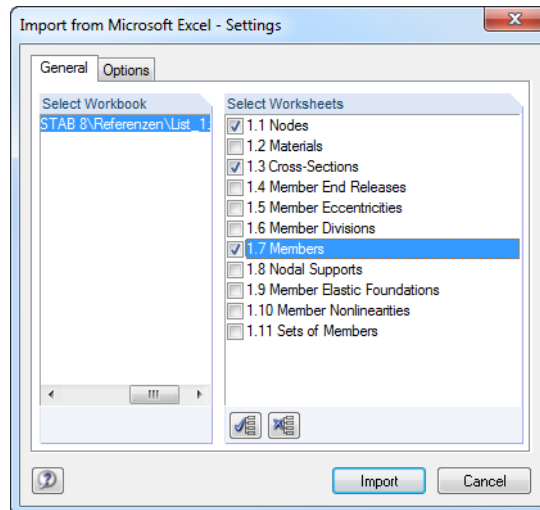


Figure 12.46: Dialog box *Import from Microsoft Excel - Settings*

Select the *Workbook* and *Worksheets* that you want to import. Descriptions, sequence and structure of worksheets must match exactly the data of RSTAB so that imported data can be written correctly into the RSTAB tables. If you are not quite sure, try to create an XLS file from the current RSTAB 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.

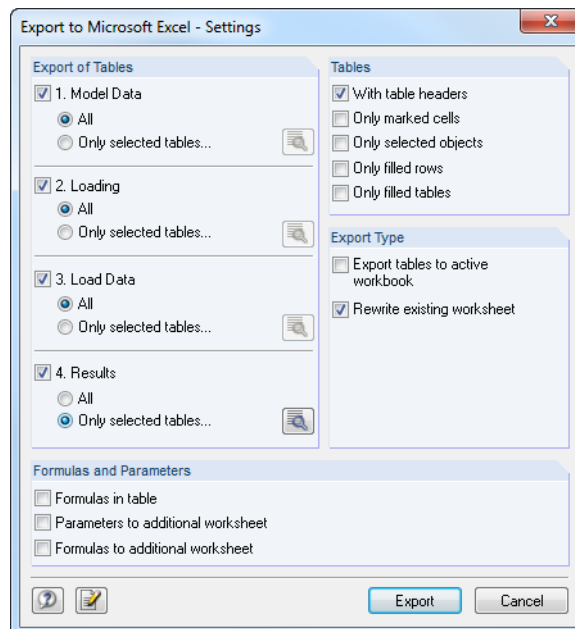


Figure 12.47: Dialog box *Export to Microsoft Excel - Settings*



In the dialog section *Export of Tables*, select the tables that you want to export. When you activate the option *Only selected tables*, RSTAB enables the respective [Select] button shown on the left. Click the button to open another dialog box for specific settings.

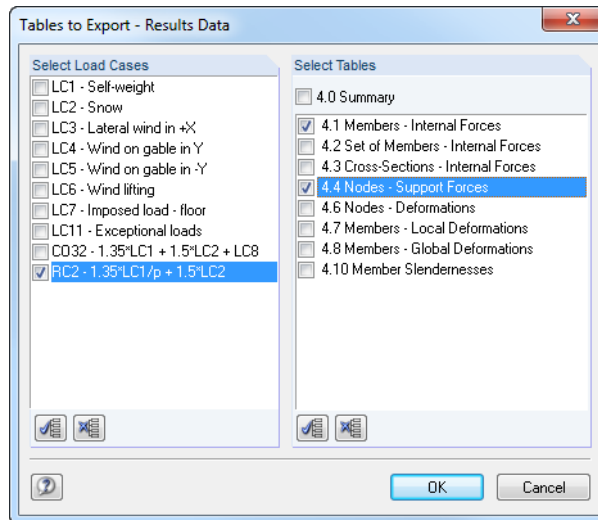


Figure 12.48: Dialog box *Tables to Export - Results Data*

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 RSTAB to MS Excel during the data exchange.

### OpenOffice format \*.ods

This interface is only available when *OpenOffice.org Calc* and *RSTAB 8 32-bit* are installed.

The import and export options are similar to the data exchange between RSTAB and MS 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. RSTAB 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.

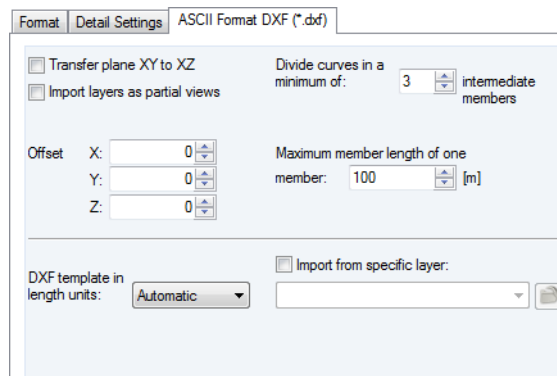


Figure 12.49: Dialog box *Import*, tab *ASCII Format DXF (\*.dxf)*

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.

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 RSTAB, however, it is normally 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 RSTAB.

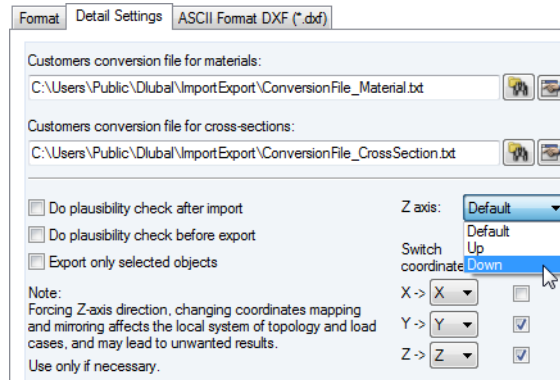


Figure 12.50: Dialog box *Import*, tab *Detail Settings*

For the DXF export it is also recommended to check the orientation of the Z axis.

### IFC format \*.ifc

The *Industry Foundation Classes* (IFC) are a global standard for 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 model 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.org](http://www.buildingsmart.org).

When you export an RSTAB 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*. RSTAB is able to import model data as well as to export RSTAB 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 structural analysis programs

### Scia format \*.xml

It is possible to import model data from the structural analysis program *SCIA* by NEMETSCHKE to RSTAB, provided that data is available in the \*.xml format.

### Dlubal RSTAB format \*.000

Use this interface to import DOS files created with RSTAB 3.xx/4.xx. Set the directory of the INP folder including input data in the *Open* dialog box.



IAI logo



### General Dlubal formats \*.xml, \*.st8

To save RSTAB 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*.

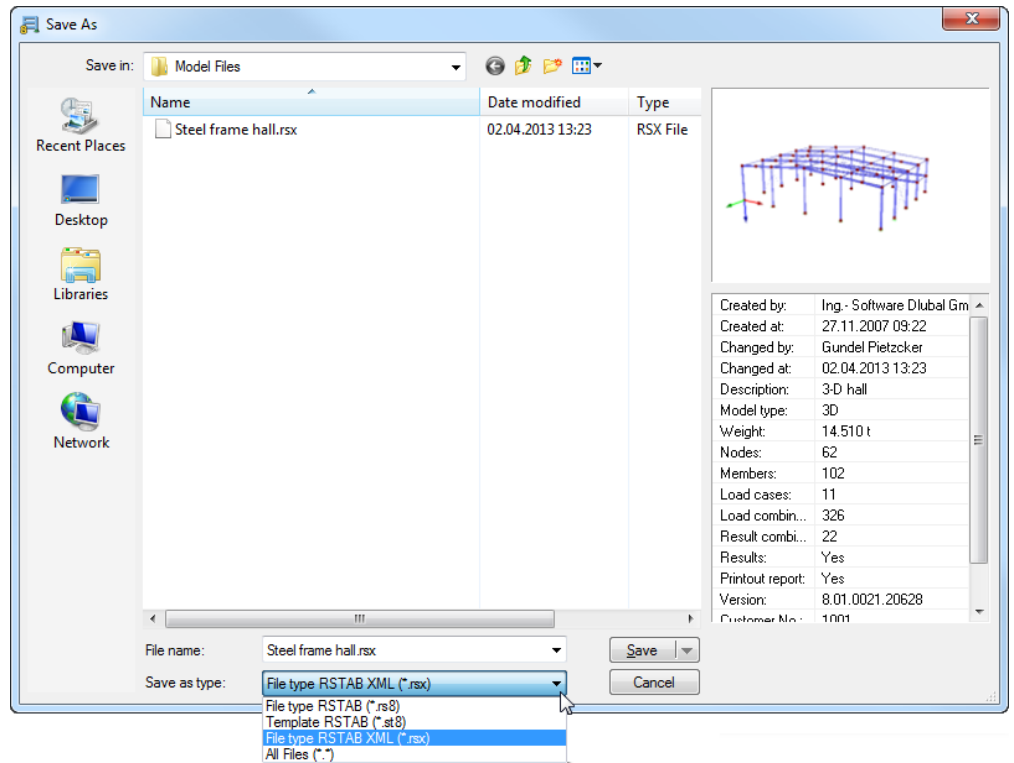


Figure 12.51: Dialog box *Save As*

With the ST8 format you save the model as a template which can be imported later when creating a new file (see Figure 12.23, page 386).

When you save the model with the file type RSX, 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 RSTAB) 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.



To import model files available in one of the formats mentioned above, select **Import** on the **File** menu.

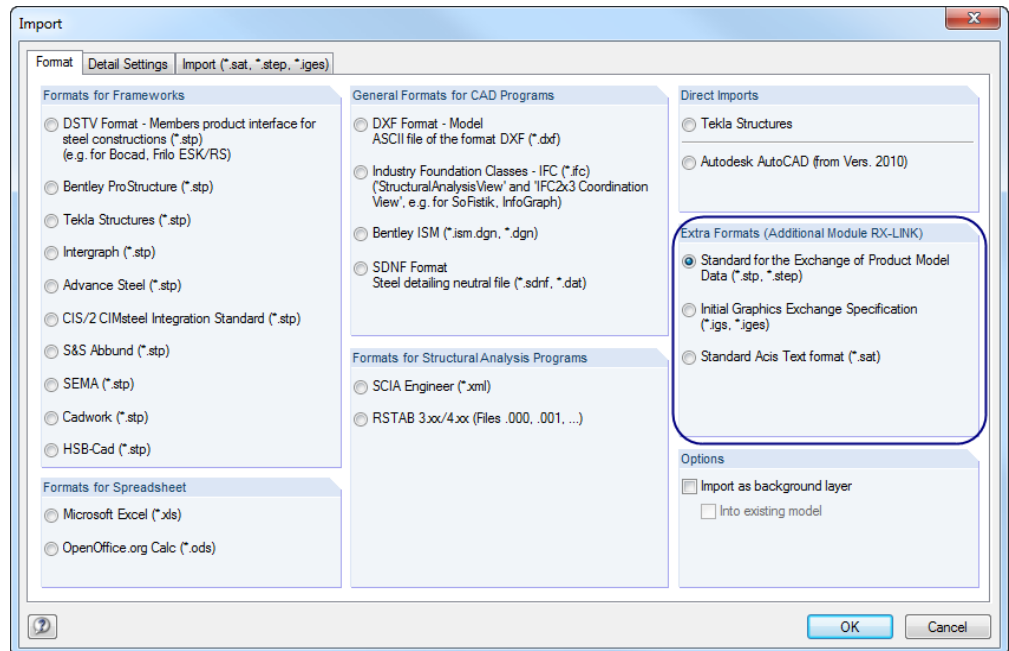


Figure 12.52: Dialog box *Import*

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.

Export options of RSTAB files in the STEP, IGES or SAT format are currently not available.

# A Index

<b>3</b>	
3 points plane.....	276
3D truss.....	349
<b>A</b>	
Acceleration .....	363
ACIS format .....	406
Action.....	103
Action category .....	100, 104
Action combination .....	115, 116, 117
Activity criterion .....	161
Add.....	324
Add load case .....	102
Add materials to library.....	51
Additional explanation .....	226
Additional moment.....	170
Additional snow loads.....	364
Additive combination.....	131
Add-on modules.....	181, 182
Alternative combination.....	131, 139
Alternatively acting .....	104
Angle .....	264, 286
Angle of principal axes .....	56
Animation .....	199, 221
Arc .....	351
Arc display .....	147, 193
Archive.....	381
Area.....	267
Area load.....	357, 361
Area of load application.....	358
ASCII file.....	63, 239
Axial displacement .....	150
Axial force .....	186
Axial strain .....	150
<b>B</b>	
Background layer .....	282, 297, 299, 308, 400
Beam.....	74, 75
Bending moment .....	186
Bentley.....	404
Block .....	395, 396
Block Manager.....	394
Bracing.....	353
Buckling.....	201
Buckling length coefficient .....	202
Buckling member .....	74, 76
Built-up cross-section.....	59
Button.....	15
<b>C</b>	
Cable .....	74, 76
Cable member .....	169
Calculation error .....	203
Calculation parameters.....	168, 176
Calculator .....	337
CARTES .....	26, 278
Cartesian coordinate system .....	42, 278
Category .....	396
Catenary.....	351
Cell .....	350, 353, 361
Center .....	280
Centroid .....	267
Chain dimensioning.....	287
Characteristic action .....	98
Check box.....	16
Check sum.....	184, 193
Circle .....	351, 352
Circle arc .....	352
Circular ring .....	272
Classification.....	387
Clipboard .....	249
Clipping plane .....	218
Coating load .....	362
Coefficient of thermal expansion .....	47
Coefficients .....	117, 123, 388
Collar roof.....	346
Color scale.....	27
Color spectrum .....	220, 254
Colored relation scales.....	184, 326
Colors in rendering.....	215, 269
Column.....	345
Column settlement .....	157
COM interface .....	399
Combination coefficient.....	123
Combination criteria.....	138
Combination expression .....	106
Combination rule.....	108

Combination scheme .....	136	Cut.....	322
Combination value .....	117	Cylindrical coordinate system .....	42
Combinations .....	389	<b>D</b>	
Comment .....	262, 288, 392	Data navigator .....	22
Comment field .....	262	Date .....	234
Company address .....	233, 235	Decimal places .....	261
Company logo.....	235	Default buttons .....	31
Compression.....	74	Default printer .....	223, 244
Compression member.....	75	Definition of axis .....	389
Concentrated load.....	151	Deformation - rendering .....	206
Configuration .....	34	Deformation process.....	222
Configuration Manager.....	34	Deformations .....	221
Connect folder .....	375	Deformed structure .....	171
Connect members .....	303, 312	Delete a project.....	376
Construction phases .....	137	Delete loads.....	167
Contact forces .....	196	Delete model.....	380
Contact moments .....	196	Design.....	20
Contact stresses.....	177	Design situation .....	108
Context menu .....	14, 33, 225, 258, 323	Diagram for nodal support.....	90
Continuous beam .....	342	Diagram for release.....	68
Continuous members.....	97	Dialog input.....	36, 142
Control panel.....	27, 220, 254	Differences in stiffness .....	56
Convergence .....	178, 179, 182	Dimension line.....	286
Coordinate system.....	42, 283, 285, 302	Dimensioning.....	286, 287
Copy.....	301, 322	Disconnect a folder .....	376
Copy load case .....	102	Disconnect load .....	356
Copy model.....	379	Displacement vector.....	301
Correction of load distribution .....	354	Displacements .....	197, 199, 200
Corresponding load cases.....	188	Display.....	213
Coupling.....	74, 77	Display navigator .....	22, 205, 220, 287
Cover sheet .....	242	Display properties .....	258, 259
Create model .....	386	Distance .....	264, 282
Create project.....	375	Divide.....	324
Criterion.....	131	Divide member.....	310
Critical buckling load .....	84	Division node .....	311
Critical forces .....	177	Division points.....	72
Critical load factor.....	172	Division spacing .....	310
Crossing members.....	165, 303, 312	Dlubal recycle bin .....	384
Cross-section .....	53	Double members .....	84
Cross-section description.....	54	Drag-and-drop.....	33, 225
Cross-section library.....	57	Drifted snow .....	364
Cross-section rotation .....	56	Dummy member.....	74
Cross-sectional angle of rotation.....	56	Dummy Rigid .....	54
Cross-sectional area .....	55	Duopitch roof.....	365, 370, 372
Current project.....	374	Duplicity.....	303

DXF file.....	297, 299, 403	Generator .....	339, 354
<b>E</b>		Girder grillage.....	344, 387
Eaves.....	368	Global load.....	153, 170
Edit stiffnesses.....	173	Grab mode .....	226
Effective length.....	83	Graphic margins.....	300
Effective length factor .....	83	Graphic printout.....	248
Either-or superposition .....	131, 139	Graphic size.....	250
Ellipse .....	272	Graphical input.....	36, 142
E-modulus.....	46	Graphical user interface.....	13
Envelope .....	132, 205	Gray scales.....	253
Equivalent loads .....	158	Grid.....	26, 278, 344
Excel.....	329, 331, 335, 336, 391, 401, 402	Grid point .....	278
Exceptional handling.....	179	Grid type .....	278
Export.....	331, 399, 400	Group.....	23, 131, 214
Extend member .....	313	Group of members .....	97
Extract from archive .....	382	Group of views.....	217
Extreme values.....	177, 186, 189, 198, 205, 328	Guideline .....	282, 290, 292
Extrude .....	340	<b>H</b>	
<b>F</b>		Hall.....	348
Failing members.....	179	Height.....	286
Failure of foundation .....	92	Help Assistant .....	183
Failure of release .....	67	Help node.....	81
Failure of support.....	88	Hidden objects .....	218
Favorites in cross-section library.....	58	History .....	380, 392
Favorites in material library .....	50	Hybrid material.....	54
Filter.....	30, 186, 194, 199, 206, 212, 220, 221, 328	Hyperbola.....	351
Find.....	265, 271	<b>I</b>	
Fish-bellied girder.....	347	Ice load .....	363
Flat roof .....	364, 368	Ideal cross-section values .....	55
Fonts.....	242	Identical nodes .....	164
Force.....	146, 150	IFC format.....	404
Formula .....	333, 336, 338	IGES format .....	406
Formula Editor .....	330, 331, 335, 337, 338, 403	Imperfection.....	109, 158
Foundation beam .....	92	Import.....	330, 399, 400
Foundation member.....	177, 195	Import cross-section table.....	63
Frame .....	342, 343, 348	Import project folder.....	378
Free line load .....	362	Imposed displacement .....	157
Friction.....	90	Imposed nodal deformation.....	157
Friction coefficient.....	90	Imposed rotation .....	158
<b>G</b>		Inclination .....	161
General data.....	386	Inclination angle .....	286
General member position .....	80, 83	Independent systems.....	166
Generate.....	324	Ineffectivity of foundation.....	92
Generated loads .....	356	Info pictures.....	226
Generated visibilities .....	215	Initial load factor .....	172

Initial prestress .....	150	Load distribution type .....	358
Initial strain .....	174	Load generator .....	354
Input field .....	15	Load increments .....	172, 176
Insert graphic.....	238	Load of multi-layer structures .....	154
Insert member.....	316	Load parameter.....	154
Insert nodes .....	315	Load type.....	150
Insert point.....	397	Loading factor.....	171
Insert text.....	238	Loads.....	325
Instability .....	176, 178	Local load .....	153, 170
Installation.....	10	Lock graphic .....	236
Interfaces .....	399	Locked guideline .....	292, 293
Intergraph.....	404	Logo .....	235
Intermediate node.....	311	<b>M</b>	
Internal forces - multicolor.....	220	Mass print .....	250, 255
Internal forces - rendering .....	205, 220	Material .....	46
Intersection point.....	281	Material color .....	268
Iterations .....	176	Material description.....	46
<b>J</b>		Material library.....	49
Join members .....	314	Material model.....	48
<b>K</b>		Member.....	73
Keyboard functions .....	32	Member axes.....	79, 153, 199, 302
<b>L</b>		Member coefficient.....	162, 170, 201
Language settings .....	246	Member contact forces.....	195
Large deformation analysis .....	170, 176	Member deformations .....	198, 200
Layer .....	297, 403	Member division .....	72, 177, 205
Layout .....	241	Member eccentricity.....	70
Leading action .....	114, 117, 126	Member elastic foundation.....	91
Light.....	270	Member end release.....	64
Light position .....	270	Member internal forces .....	185, 205
Lighting .....	270	Member length.....	82
Limit values .....	29	Member load.....	148
Limit values of spring.....	78	Member nonlinearity.....	93
Line .....	351	Member orientation.....	79, 310, 311, 314
Line grid .....	283, 294	Member position .....	80, 83, 187, 196
Linear static analysis.....	158	Member rotation.....	79, 80
List .....	16	Member slenderness .....	202
List button .....	15	Member type.....	74
List of members .....	149, 159	Merge members.....	313
Load application.....	358	Method of analysis .....	114, 169
Load bearing capacity .....	106	MicroStation .....	404
Load case .....	98, 303	Mirror .....	303, 305
Load combination.....	120, 121, 122, 125, 126, 389	Mirroring plane.....	305
Load correction factors .....	359, 365, 369, 372	Model description .....	234, 235, 387
Load direction .....	153	Model generator .....	341
Load distribution.....	151	Model type .....	387

Moment ..... 147, 150  
 Moment equilibrium .....354  
 Moment of inertia ..... 55  
 Moment release for member ..... 65  
 Monopitch roof .....364, 369, 372  
 Mouse functions ..... 33  
 Move .....301  
 Movement .....363  
 Multi-color internal forces .....205  
 Multiple windows view ..... 211, 249  
 Multiply .....324

**N**

National annex .....388  
 Navigator .....21  
 Network ..... 12  
 Network projects .....393  
 New page .....225  
 Newton-Raphson .....170  
 Nodal deformations .....197  
 Nodal load ..... 146, 303  
 Nodal support ..... 85  
 Nodal support forces .....191  
 Node coordinates ..... 44  
 Node number ..... 41  
 Nodes .....41, 273  
 Nonlinearities for releases ..... 67  
 Nonlinearity .....178  
 Non-linearity of supports ..... 88  
 Notes .....288  
 Null ..... 77  
 Number of load cases .....112  
 Number of reactivations .....178  
 Numbering .....234, 318, 319

**O**

Object information .....207  
 Object snap .....279, 290, 296, 299  
 Offset .....71, 219, 277, 287, 288  
 Open model ..... 379, 385  
 OpenOffice ..... 329, 331, 403  
 Options .....391  
 Organization of load data .....145  
 Origin .....275, 277, 284  
 OSNAP .....26, 279  
 Overall dimensions ..... 57  
 Overlapping members .....165

**P**

Page preview .....226  
 Panel ..... 27  
 Parabola .....351  
 Parabolic load ..... 152  
 Parallel installation ..... 12  
 Parallel member .....281, 339  
 Parameter list ..... 332, 335, 337  
 Parameterized input ..... 332  
 Parametric cross-section ..... 59  
 Partial activity ..... 67, 89  
 Partial safety factor .....98, 117, 123  
 Partial safety factor for material .....47, 172, 173  
 Partial view .....23, 215  
 Partition points .....281  
 Paste .....322  
 PDF file .....245  
 Permanent loading ..... 131  
 Perpendicular .....280  
 Pipe content .....150  
 Plane .....357  
 Plastic hinge ..... 95  
 Plausibility check ..... 163  
 Plotters .....252  
 Poisson's ratio ..... 47  
 Polar coordinate system .....43, 278  
 Polygonal chain .....352  
 Post-critical analysis .....171, 177  
 Precamber .....150, 161  
 Prefix .....235  
 Preselection .....271  
 Prestressing force ..... 150  
 Principal axes ..... 187  
 Print file .....244  
 Print graphic .....236, 248  
 Print quality .....253  
 Printing .....244  
 Printing color .....253  
 Printout report ..... 223, 229, 246  
 Printout report header ..... 233, 235, 249  
 Printout Report Manager .....224  
 Printout report template .....240, 241  
 Program capacities ..... 7  
 Program language .....257  
 Program options ..... 183

Project .....	306	Rotation axis .....	304, 309
Project description .....	234, 235, 378	Rotations .....	56, 197, 199, 200
Project Manager .....	12, 373	Round corner .....	318
Project Navigator .....	21	Row .....	322
Projection .....	154	RSCOMBI .....	389
Purlin roof .....	347	RSTAB 7 .....	12
<b>R</b>		RSX format .....	405
Rafter roof .....	346	RTF file .....	239, 244
Reactivate members .....	178	<b>S</b>	
Reactivation .....	178	Save cross-section .....	60
Recycle bin .....	377, 380, 384	Scale .....	259, 307
Reduce combinations .....	112	Scaling .....	206
Reduction factor .....	117, 123, 179	Scaling factor .....	29, 206
Reference length .....	154	Scia .....	404
Reference material .....	55	Scissors release .....	66
Reference node .....	42	ScreenTip .....	18, 40
Reference system .....	65	SDNF format .....	404
Reference temperature .....	48	Search .....	265, 322
Regenerate model .....	167	Second-order analysis .....	120, 169, 172, 176
Related objects .....	274	Section .....	15
Rename model .....	380	Section line .....	273
Rendering .....	199, 205, 268	Selecting - additive .....	271
Renumber .....	318	Selecting - alternative .....	271
Renumber load case .....	320	Selection .....	271, 323, 324, 334
Replace .....	322	Selection field .....	16
Report header template .....	234	Selection function .....	323
Report template .....	224, 240	Selection in printout report .....	227, 231, 232
Restraint .....	87	Selection mode .....	226
Result combination ....	114, 128, 130, 133, 134, 175, 188, 204, 389	Self-weight .....	100
Result diagrams .....	208, 251	Serviceability .....	107, 108
Result values .....	204, 208	Set language .....	257
Resultant force .....	194	Set of members .....	17, 96, 149, 160, 189
Results .....	184, 204	SHAPE-MASSIVE cross-section .....	63
Results balance .....	184	SHAPE-THIN cross-section .....	63
Results display .....	205	Shear .....	309
Results evaluation .....	203	Shear area .....	55
Results navigator .....	23, 204	Shear center .....	154
RF-LINK .....	406	Shear deformation .....	56
Rhomboid .....	272	Shear force .....	186
Rigid coupling .....	74, 77	Shear modulus .....	47
Rigid member .....	74, 75	Shear springs of foundation .....	92
Rolled cross-section .....	57	Shear stiffness .....	177
Rotate .....	303, 304, 309	Sheet numbering .....	234
Rotated nodal supports .....	193	Shrinkage .....	150
		Sign .....	175



Sign rule .....	81, 187, 199	Synchronization of selection .....	327
Signs of internal forces .....	81	Synchronized selection .....	21
Signs of support forces .....	191, 192	System requirements .....	10
Singularity .....	318	<b>T</b>	
Slenderness .....	202	Tab .....	14
Slippage .....	78, 94	Table input .....	321, 323
Smooth color transition .....	29	Table settings .....	325, 327
Smoothing .....	209, 210	Tables .....	24, 40, 142, 145, 181, 325
Snap .....	25, 278	Taper .....	53, 82, 177, 343
Snap distance .....	279	Taper shape .....	82
Snow guard .....	364	Target plane .....	307
Snow load .....	364, 365	Tearing .....	94, 95
Soil constants .....	91	Temperature .....	57, 150
Solid cross-section .....	60	Template .....	241, 390, 405
Special selection .....	264, 274	Tensile forces .....	171
Specific weight .....	47	Tension .....	74
Sphere .....	352	Tension member .....	75
Spiral stairway .....	350	Text file .....	239
Spring .....	65, 78, 91	Texture .....	268
Spring constant .....	78	Thermal material model .....	48
Square addition .....	138, 175	Three-dimensional cell .....	349
Stairway .....	350	Thumbnail .....	374, 383, 395
Standard .....	139, 387	Timber cross-section .....	61
Start calculation .....	180	Title .....	226
Start program .....	36	Title box .....	252
Statistic .....	164	Toolbar .....	18
Status bar .....	25	Torsion .....	154
STEP format .....	406	Torsional constant .....	55
Stiffness .....	76, 84, 172, 177	Torsional moment .....	186
Stiffness modulus $E_s$ .....	91	Total moment to origin .....	360
Stress-strain diagram .....	48	Transparency .....	218
Stretch factors .....	300	Trapezoidal load .....	152
Strip foundation .....	92	Triangular load .....	152
Structure check .....	164	True member length .....	154
Sub-project .....	375, 376	Truss .....	74, 344
Super combination .....	137, 138, 140, 231	Truss (only N) .....	74, 75
SUPER-RC .....	17, 137	Type of guideline .....	291
Support .....	85, 87	<b>U</b>	
Support forces .....	191	Ultimate limit state .....	108
Support moments .....	192	Uniformly distributed load .....	152
Support reactions .....	191, 192	Unite nodes .....	167
Support rotation .....	86, 191, 193	Units .....	261
Support type .....	86, 88	Updates .....	12
Surface area .....	267	User profile .....	261
Swap file .....	182	User-defined cross-section .....	62

User-defined view .....213  
User-defined visibilities .....214

**V**

Value spectrum .....28  
Variable load .....152  
Variable loading .....131  
VCmaster .....245  
Vertical position .....80, 167  
Video .....222  
View .....33, 212, 213, 266  
View angle .....266  
View mode .....360  
Viewpoint .....266  
Views navigator .....23  
Virtual line .....353, 361  
Visibilities .....212, 214, 216, 217  
Visual object .....296

**W**

Wall .....366, 372  
Weight .....83  
Wheel button .....33  
Wind load .....358, 366, 368, 369, 370, 372  
Window .....216  
Window selection .....271  
Word .....391  
Work plane .....219, 275

**X**

XML file .....405

**Y**

Yielding .....95

**Z**

Z-axis .....404  
Zero point .....284