Dlubal

Version
October 2013

Program

RX-TIMBER 2

Glued-Laminated Beams, Continuous Beams, Columns, Frames, Coupled Purlins and Bracings Acc. to Eurocode 5 and DIN 1052

Program Description

© Dlubal Software GmbH
Am Zellweg 2 D-93464 Tiefenbach

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

Contents

Contents

Contents

1. Introduction

1.1 About RX-TIMBER

RX-TIMBER is a DLUBAL program family that meets specific demands in timber engineering. With the programs included in the RX-TIMBER package you can design glued-laminated beams, continuous beams, columns, frames, purlins and bracings. The governing result combinations are created automatically with the add-on module RF-COMBI that is integrated in RX-TIMBER.

All of the RX-TIMBER programs are able to perform the ultimate and the serviceability limit state design. The design can be carried out according to the following timber standards:

- **EN 1995-1-1:2004-11**
- **DIN 1052:2008-12**

In addition, it is possible to perform the fire protection design according to EN 1995-1-2 or DIN 4102, part 22.

CEN	European Union
CSN	Czech Republic
DIN	Germany
DK	Denmark
NEN	Netherlands
NF	France
ONORM	Austria
PN	Poland
SFS	Finland
SS	Sweden
UNI	Italy

National annexes for EC 5

Various national application documents are available for selection for EN 1995-1-1:2004. The list shown on the left presents the implemented national annexes (NAs) and is constantly being expanded.

RX-TIMBER Glued-Laminated Beam

The program is used for designing glued-laminated beams with long span lengths. The following beam types can be designed:

- Parallel beam
- Monopitch roof beam
- Curved beam
- Double tapered beam
- Pitched cambered beam with constant and variable height
- Fish beam - parabolic or linear with roundings

Depending on the selected beam type, different input settings are available, allowing you to calculate unsymmetric beams with and without cantilevers in various combinations. Moreover, you can take account of typical stiffenings for transversal tension such as bonded steel bars.

RX-TIMBER Continuous Beam

The program designs multi-span beams with up to 20 spans and cantilevers for biaxial bending. The loads are generated automatically according to EN 1990 or DIN 1055-100. The options for optimization offered in the standards are implemented in the program. The following beam types can be designed:

- Single-span beam
- Continuous beam
- Hinged girder system (Gerber)

For all beam types it is possible to define the support in x-, z- and y-direction. Optionally, you can assign moment and shear force releases.

RX-TIMBER Column

The program performs the design for columns with a rectangular or circular cross-section. To model the connection of a column to a roof truss close to reality, it is possible to define elastic supports for the column's head and footing.

RX-TIMBER Frame

The program is able to design symmetrical or asymmetrical three-hinged frames. The coupling element is represented by a knee connection with finger joints, optionally calculated with or without intermediate piece. When entering the frame geometry, sloping and direction of lamellas can be selected freely.

RX-TIMBER Purlin

The program designs coupled purlins in roofs. Optionally, you can set the design of couplings inactive to analyze nothing but a continuous beam (however, it is not possible to adjust the depth of the beam to the moment diagram by using tapers). Wind and snow loads are precisely determined by selecting the roof shape. Coupling elements are provided as nails, special timber connectors and bolts according to the WT system by SFS intec, or by a user-defined load bearing capacity.

RX-TIMBER Brace

This program is used to design the bracing of a roof structure. The brace can be defined by means of three roof constructions and adjusted at will. As steel diagonals are often used for a roof bracing, the program provides also steel cross-sections. The material nonlinearities of tension members as well as the reduction of stiffness are automatically considered in the ultimate limit state design.

Find information about news and developments of the timber programs and add-on modules on our homepage www.dlubal.com.

We hope you will enjoy working with the programs of the RX-TIMBER family.

Your team from DLUBAL SOFTWARE GMBH

1.2 Company Profile

Since its beginnings in 1987, DLUBAL SOFTWARE GMBH 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 which is 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.

DLUBAL software has been designed in such a way that even users with basic computer skills can handle the software successfully after a short while. With considerable pride, we now number more than 7,000 engineering offices as well as construction companies from a variety of fields and places of higher education all over the world among our satisfied customers. 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. In case of 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.

1.3 RX-TIMBER Team

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Bastian Kuhn
Ing. Jiří Hanzálek
Dipl.-Ing. (FH) Younes El Frem

Programming

Ing. Martin Deyl
Dipl.-Ing. Georg Dlubal
Ing. Tomáš Ferencz
Mgr. Jaroslav Krul
Mgr. Jiří Patrák

Program design, dialog figures and icons

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

Program supervision

Ing. Jiří Hanzálek
Dipl.-Ing. (FH) Bastian Kuhn
Ing. Vladimír Kabát
Ing. Zdeněk Kodera
Ing. Vladimír Pátý

Localization and manual

Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Ing. Jana Duníková
Ing. Lara Freyer
Alessandra Grosso
Ing. Ladislav Kábrt
Ing. Aleksandra Kociołek
Dipl.-Ing. (FH) Bastian Kuhn
Ing. Roberto Lombino
Eng.º Nilton Lopes
Ing. Téc. Ind. José Martínez
MA SKT Anton Mitleider
Dipl.-Ü. Gundel Pietzcker
Ing. Zoja Rendlová
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl
Ing. Marcin Wardyn

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
M. Eng. Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Gerhard Rehm
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

1.4 Using the Manual

Many roads lead to Rome – this policy also applies to working with RX-TIMBER. The descriptions in this manual follow the sequence and structure of the tables available in the RX-TIMBER programs. The first chapters describe general program functions such as file management, table arrangements and printout. Beginning with chapter 10, the manual presents the individual programs by describing examples.

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 where you can go through our FAQ pages by selecting particular criteria.

2. Installation

2.1 System Requirements

The following system requirements are recommended in order to use RX-TIMBER without any difficulties:

- 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 1 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.

RX-TIMBER 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 RX-TIMBER basically runs on all systems that fulfill the system requirements mentioned above. If RX-TIMBER is used for intensive calculations, the guiding principle 'the more, the better' applies.

2.2 Installation Process

The program family **RX-TIMBER** is delivered on the DVD called *Stand-Alone Applications*. The DVD contains the stand-alone programs RX-TIMBER, CRANEWAY, COMPOSITE-BEAM and PLATE-BUCKLING. The installation of RX-TIMBER includes all programs that are available in the RX-TIMBER product range.

Before you install RX-TIMBER, 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 program. When working with RX-TIMBER, user rights will be sufficient. For detailed instructions, see the user rights video on our website.

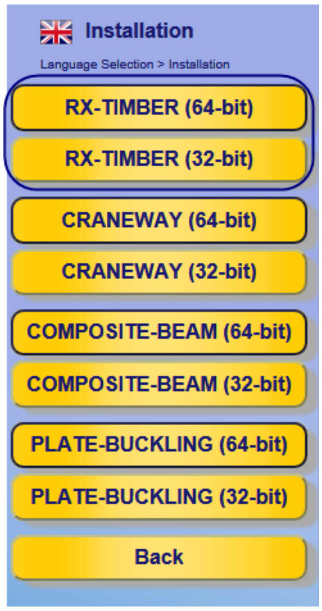
2.2.1 Installation from DVD

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

- Insert the DVD into your DVD-ROM drive.
- The installation process starts automatically. If it does not start, the *autorun* function is probably deactivated. 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).
- In the start dialog box, select the language.

Figure 2.1: Language settings

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



Program selection

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

Moreover, the DVD contains instructions for installation and the RX-TIMBER manual in PDF format. To open it, you need the Acrobat Reader that you can install from the DVD.

RX-TIMBER 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 RX-TIMBER as a full version or a trial version running for 30 days.

To use this program as full version, you need a dongle (hardlock) and an authorization file *Author.ini*. The dongle is a plug-in device that you have to connect to a USB port of the computer; the authorization file contains coded information for your license(s). Usually, we send you the *Author.ini* file via e-mail. You can also use the Extranet on www.dlubal.com to access the authorization file. Save this *Author.ini* file on your computer, an USB flash drive, or in the network.

The authorization file is needed for every work place. You can copy the file as often as you like. If, however, the content of this file is changed, the authorization becomes invalid.

It is also possible to run the full version of RX-TIMBER as 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 *setup.exe* from the client. The following steps do not differ from the DVD installation.

Network licenses

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

2.2.3 Installing Updates and Other Programs

The DVD contains the complete program package of RX-TIMBER together with other stand-alone programs. When purchasing a new add-on module, you will not necessarily receive a new DVD but always a new authorization file *Author.ini*. To update the authorization without reinstallation, select *Load Authorization File* on the *Help* menu in the respective RX-TIMBER program.

Old program files are removed and replaced by new ones when updating the program within one series of a version (for example 2.**02**.xxxx). Of course, your project data is preserved! When updating the program to the next series (for example 2.**03**.xxxx) the new version will be installed parallel to the old version (see below).

If you use printout report headers that you have defined yourself, save them before installing the update. The headers are usually stored in the file **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 the update. They are stored in the file **RfemProtocolConfig.cfg** in the folder *C:\ProgramData\Dlubal\RX-TIMBER 2.xx\General Data*.

The projects linked in the Project Manager are managed in the ASCII file **PRO.DLP** which can normally be found in the folder *C:\ProgramData\Dlubal\ProMan* (see Figure 4.19, page 25). If you want to uninstall RX-TIMBER before installing the update, you should save this file, too.

2.2.4 Parallel Installation of Dlubal Programs

The DLUBAL applications RX-TIMBER 1 and the individual series of version RX-TIMBER 2 can be run parallel on the computer since the program files are stored in different directories. The default folders are the following for a 64-bit operating system:

- RX-TIMBER 1: C:\Programs (x86)\Dlubal\RX-TIMBER1
- RX-TIMBER 2.01 C:\Programs\Dlubal\RX-TIMBER 2.01
- RX-TIMBER 2.02 C:\Programs\Dlubal\RX-TIMBER 2.02
- RX-TIMBER 2.03 C:\Programs\Dlubal\RX-TIMBER 2.03 etc.

All models created with RX-TIMBER 1 can be opened and edited in RX-TIMBER 2.

Models from RX-TIMBER 1 won't be overwritten when saving them in RX-TIMBER 2 because both programs use different file endings: RX-TIMBER 1 saves model data in the format ***.rh1**, RX-TIMBER 2 in ***.rh2**.

RX-TIMBER uses the analysis core of RFEM but is installed independently of RFEM. Thus, it runs stand-alone: An RFEM installation is not required.

A model created with RX-TIMBER can be opened in RFEM or RSTAB.

3. Principles for RX-TIMBER

3.1 Restrictions

Please note the following comments. In this way, you can avoid assuming automatisms that are not given in RX-TIMBER.

The concept of the software aims at avoiding unnecessary input. Therefore, defining snow and wind loads has been simplified as far as possible by integrated load generators. But as you may understand, considering all options of variation is not possible. The following items must be taken into account:

- When generating loads the program acts on the assumption of a hall whose sides are closed. Wind loads from roof-lifting internal wind can be taken into account for halls that are open on one side by the option *with Permeable Walls*.
- Blowings and slippings of snow loads must be defined manually.
- RX-TIMBER **Glued-Laminated Beam** does not generate any horizontal load from wind and earthquakes on the gable.
- The wind load in RX-TIMBER **Continuous Beam** is always determined for the maximum possible value of loading. For this purpose the program determines always the maximum and minimum stressed area of the respective geometry.

3.2 Start RX-TIMBER

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

File management and Project Manager are described in chapter 4.

The relevant RX-TIMBER program starts automatically after creating a new model or by double-clicking a model that is already available.

4. File Management

This chapter explains how data is organized in the Project Manager. In addition, the chapter describes the data import and export with the interfaces integrated in RX-TIMBER for exchanging data with other programs.

After starting the program on the Start menu in Windows or by clicking the RX-TIMBER icon, the Project Manager appears where you can create new models or select files that are already available.

The Project Manager provides access to the individual timber programs of the RX-TIMBER family.

4.1 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 models within the network (see chapter 4.3, page 27).

The Project Manager can be left open as a stand-alone application when working in RX-TIMBER.

When you open the Project Manager, the following multi-part window appears. It has its own menu and toolbar.

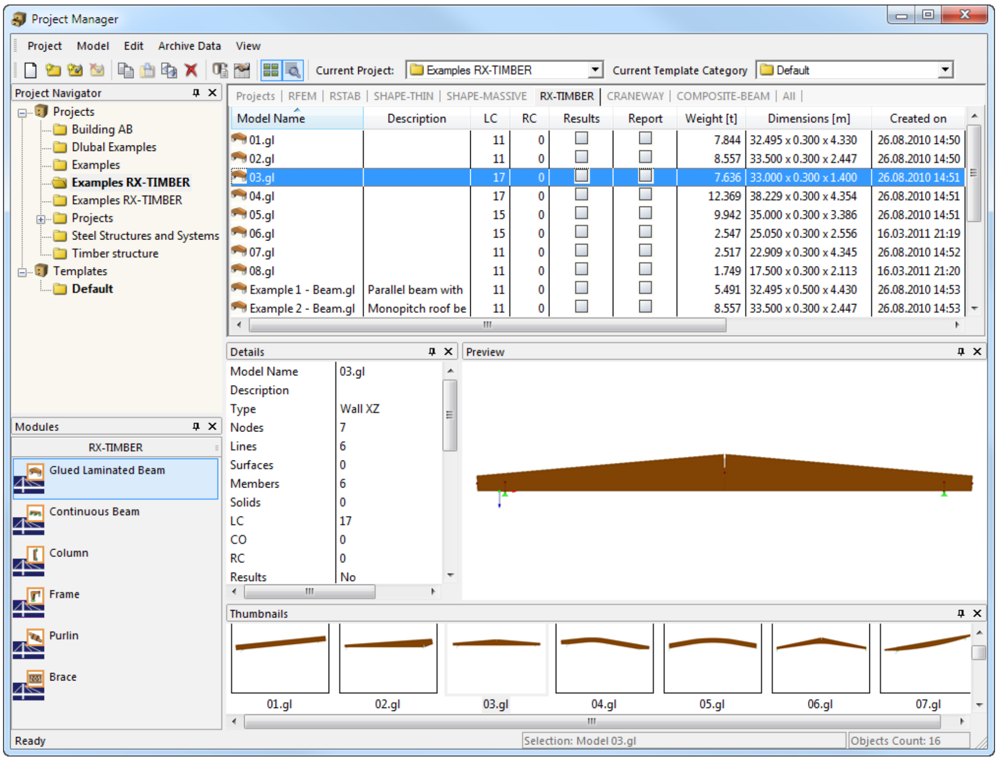


Figure 4.1: Project Manager

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.

RX-TIMBER programs

All programs of the RX-TIMBER family are displayed below the *Project Navigator*. They represent a filter possibility that can be used to show only the models that belong to a particular timber program.

For example, if you select *Glued Laminated Beam*, the Manager shows only the model files in the right part of the window that were created with the program *RX-TIMBER Glued Laminated Beam*. To create a new model in the program, click it with the right mouse button.

Table of models

The models are arranged in several tabs, sorted by Dlubal applications. The tab *RX-TIMBER* lists all RX-TIMBER models contained in the selected project. The *Model Name* and *Description* as well as significant model and file information including name of the user who created and edited the model are displayed.

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 23).

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.

4.1.1 Project Management

Create a new project

To create a new project,

- select **New** on the **Project** menu or
- click the button [New Project] in the toolbar shown on the left.

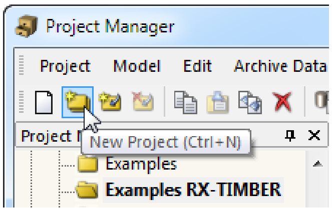


Figure 4.2: Button *New Project*

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.

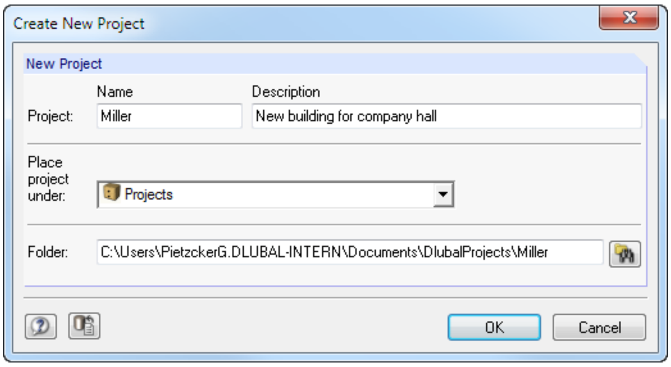
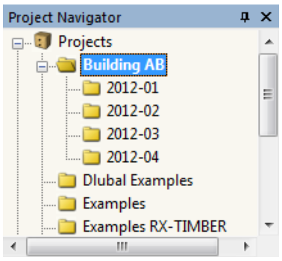


Figure 4.3: Dialog box *Create New Project*



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 an existing folder

To integrate a folder already containing several RX-TIMBER 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.

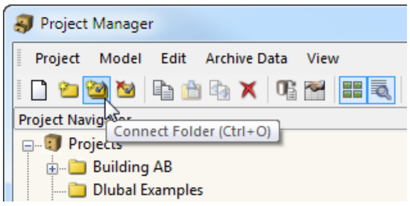


Figure 4.4: Button *Connect Folder*

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 **ProMan** folder (see chapter 4.1.4.3, page 25).

A dialog box opens that is similar to the dialog box shown in Figure 4.3. Enter the *Name* and *Description* of the project, and use the [Browse] button to set the directory for the relevant *Folder*. If a project is specified in the list *Place project under*, the connecting folder must be contained within the directory of this project. The folder will then be managed as a sub-project. But if you want the folder to appear as an independent project in the Project Manager, select *Projects* on the top of the list.

Tick the option *Connect folder including all subfolders* to connect all folders contained in the selected folder at once with the management of the Project Manager.

Disconnect a folder

To disconnect a folder integrated in the project management,

- select **Disconnect** on the **Project** menu of the Project Manager (project must have been previously selected) or
- use the project's context menu in the navigator.

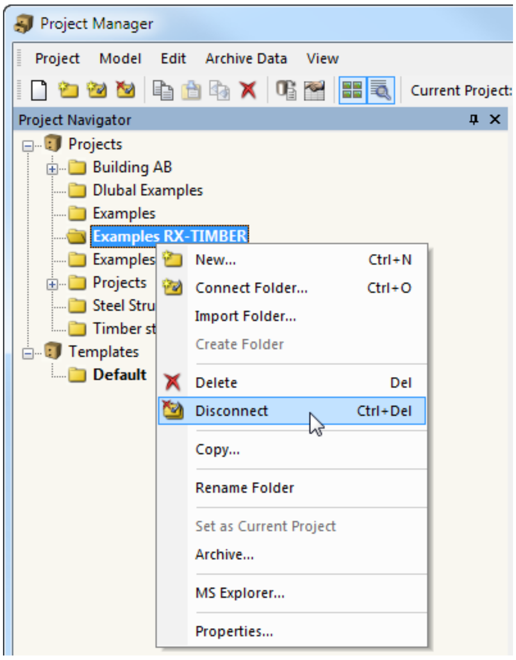


Figure 4.5: Context menu of a project

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 project's context menu in the navigator (see figure above).

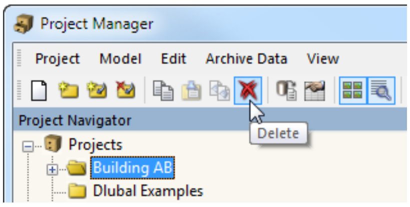


Figure 4.6: Button *Delete*

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 4.1.4.2 on page 24.

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 (select project previously) or
- use the **Copy** entry in the project's context menu in the navigator (see Figure 4.5).

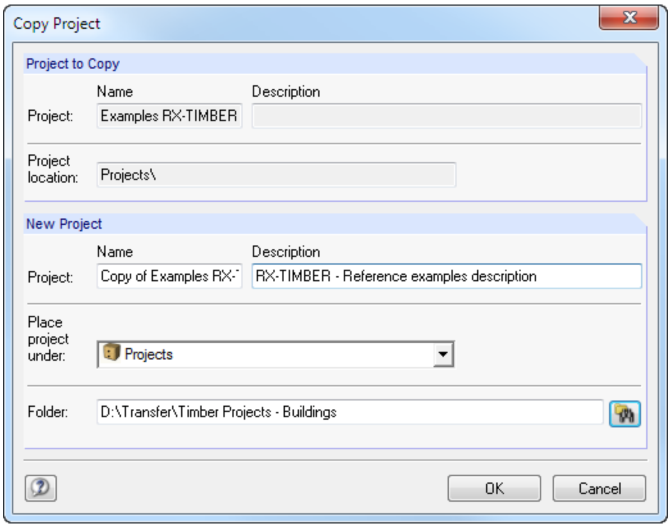


Figure 4.7: Dialog box *Copy Project*

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 4.4, page 15).

Rename a project / change description

To change the description of a project subsequently,

- select **Properties** on the **Project** menu of the Project Manager (select project previously) or
- use the **Properties** entry in the project's context menu in the navigator (see Figure 4.5).

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.

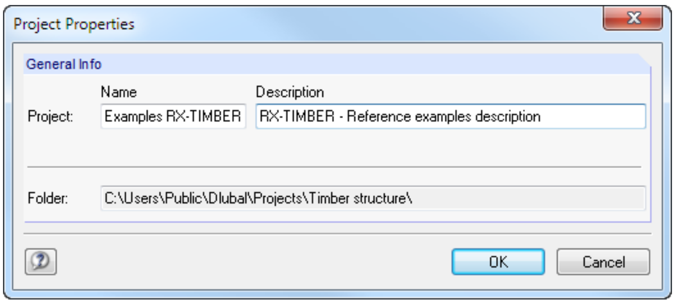


Figure 4.8: Dialog box *Project Properties*

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 4.3, page 27). 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.

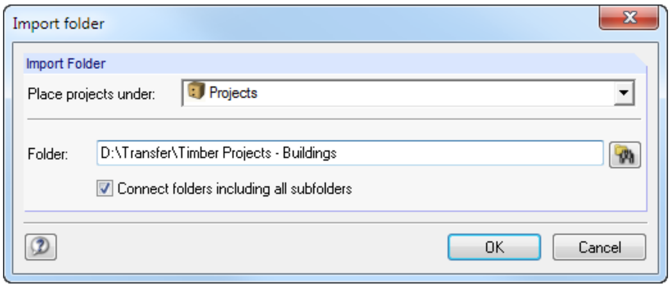


Figure 4.9: Dialog box *Import folder*

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.

4.1.2 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.

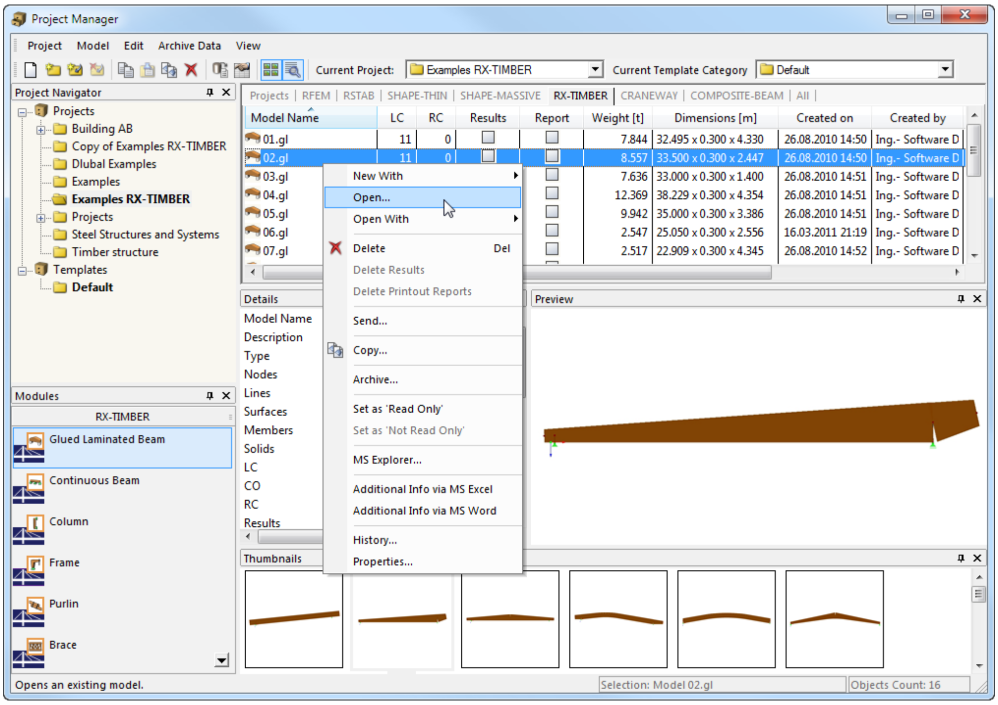
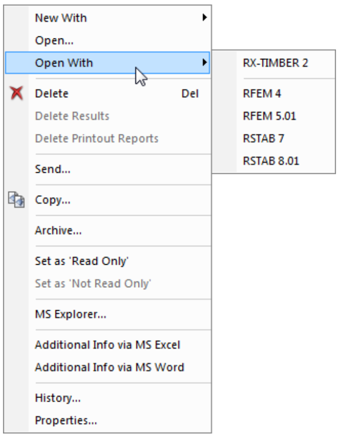


Figure 4.10: Context menu *Model*



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 also possible to open RX-TIMBER files directly in RFEM or 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.

In the dialog box *Copy Model* (see figure below), 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.

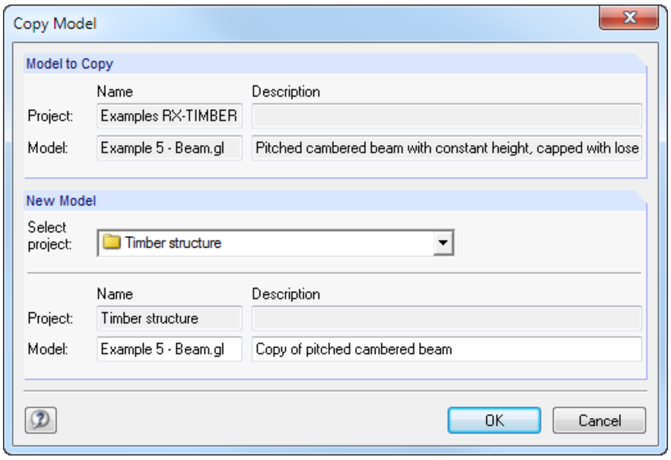


Figure 4.11: Dialog box *Copy Model*

Rename a model

To rename a model,

- select **Properties** on the **Model** menu of the Project Manager (model must have been previously selected) or
- use the **Properties** entry in the model's context menu in the navigator (see Figure 4.10).

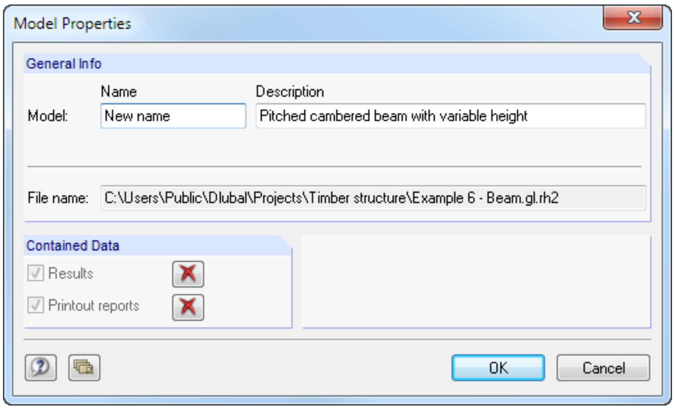


Figure 4.12: Dialog box *Model Properties*

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 (select model previously),
- click the [Delete] button in the toolbar shown on the left or
- use the context menu of the model (see Figure 4.10).

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 4.1.4.2 on page 24.

Show the history

To check the history of a model,

- select **History** on the **Model** menu of the Project Manager (select model previously) or
- use the **History** entry in the model's context menu (see Figure 4.10).

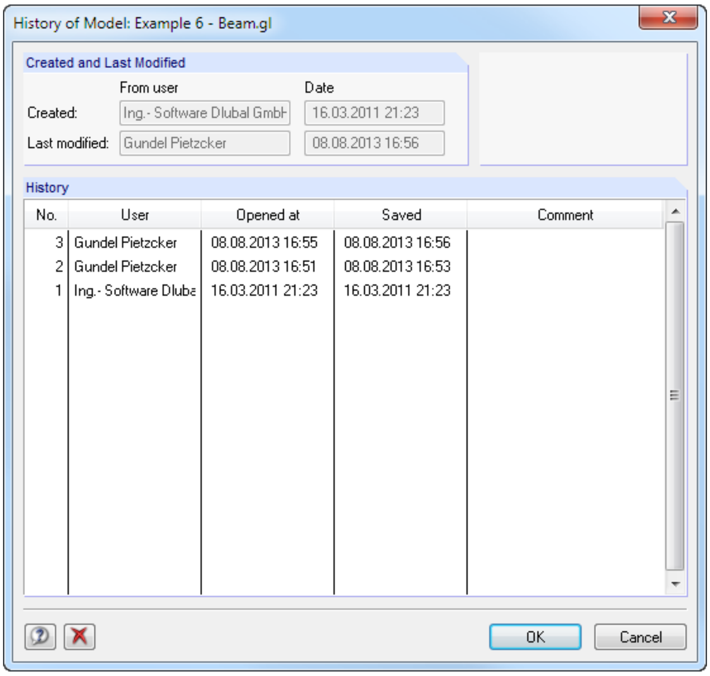


Figure 4.13: Info window *History of Model*

A dialog box appears showing information about the users who created, opened or modified the model. The overview includes also the time when the individual actions were carried out.

4.1.3 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 process,

- select **Make Archive** on the **Archive Data** menu of the Project Manager (model or project must have been previously selected) or
- use the context menu of the project or model (see Figure 4.10).

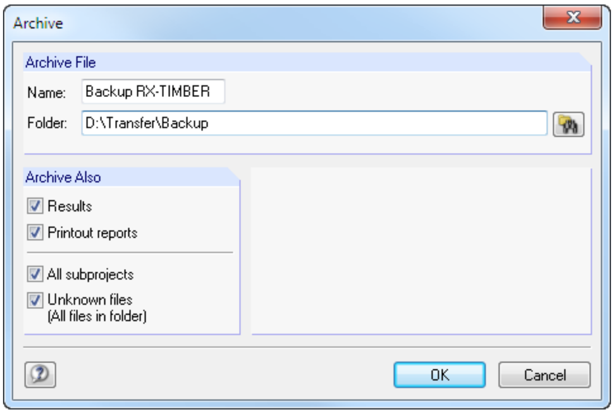


Figure 4.14: Dialog box *Archive*

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.

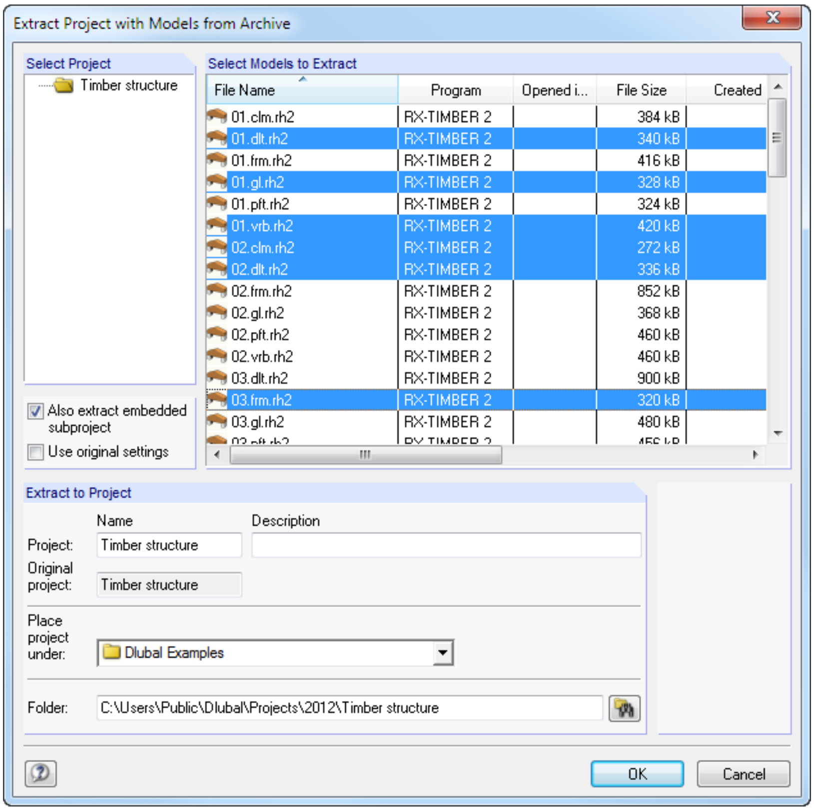


Figure 4.15: Dialog box *Extract Project with Models from Archive*

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.

4.1.4 Settings

4.1.4.1 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.

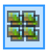
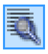
Button	Function
	Shows thumbnail images of all models in the project
	Shows model details and preview of model

Table 4.1: Buttons for setting the view

Sorting models

The arrangement of models in the table can be adjusted: As usual with Windows applications, you can sort the list in an ascending or descending order by clicking into the column titles. Alternatively, you can

select **Sort Models** on the **View** menu.

Adjust columns

To arrange the columns according to your needs,

- select **Manage Register Columns** on the **View** menu of the Project Manager
- or use the button [Manage Register Columns] in the Manager toolbar shown on the left.

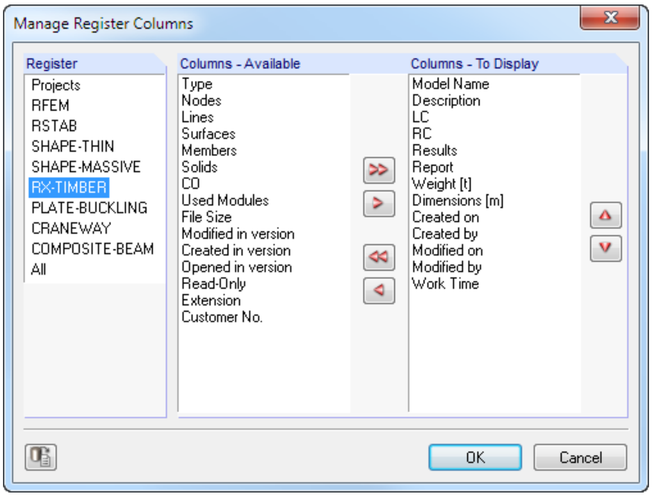
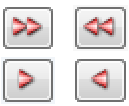


Figure 4.16: Dialog box *Manage Register Columns*



RX-TIMBER is preset in the *Register* dialog section. In the list *Columns - Available*, you can select the relevant entries 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.

4.1.4.2 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.

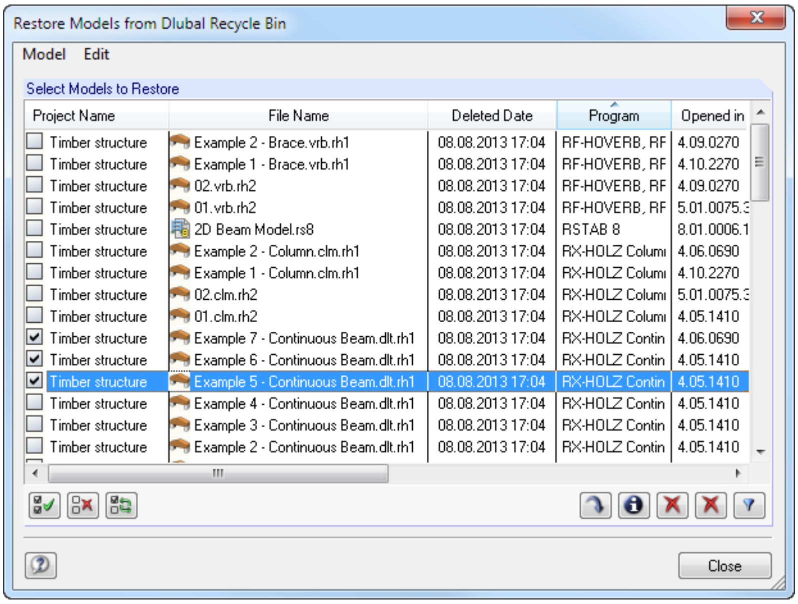


Figure 4.17: Dialog box *Restore Models from Dlubal Recycle Bin*

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 models 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.

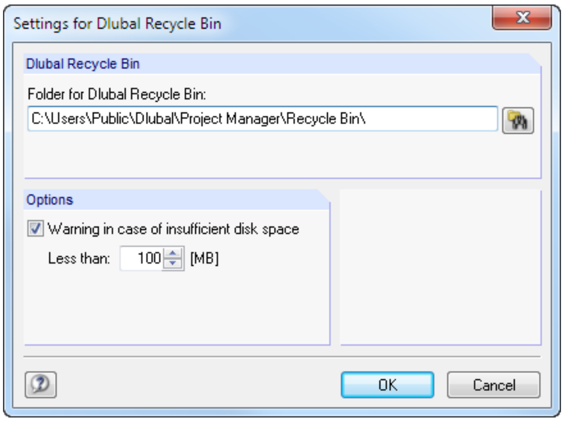


Figure 4.18: Dialog box *Settings for Dlubal Recycle Bin*

4.1.4.3 Directories

The directories of the Project Manager can be checked in the *Settings*. To open the corresponding dialog box,

select **Program Options** on the **Edit** menu of the Project Manager.

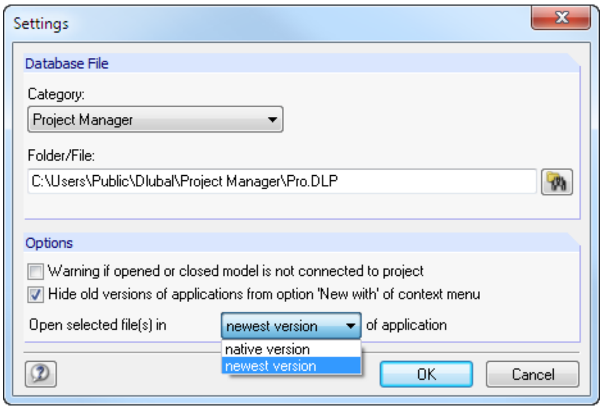


Figure 4.19: Dialog box *Settings*

Folder and file name of the category *Project Manager* are displayed in the input field below where they can be adjusted, if necessary. All projects are managed in the file **PRO.DLP** which can normally be found in the folders *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 the models contained in the Project Manager in a central place: Set the directory to the PRO.DLP file on the server (see chapter 4.3, page 27).

The dialog section *Options* offers more settings for handling RX-TIMBER files: Usually, a message appears when opening a file out of the Explorer, an e-mail program etc. if 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.

4.2 Creating a New Model

First, go to the *Modules* section in the Project Manager (see Figure 4.1, page 13) where you select the program that you want to use for calculating a new model.

To create a model,

- click the toolbar button [New Model] shown on the left or
- point to **New With** on the **Model** menu, and then select **RX-TIMBER 2**.

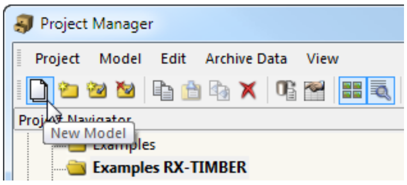


Figure 4.20: Button *New Model*

The dialog box *New Model for RX-TIMBER - General Data* opens.

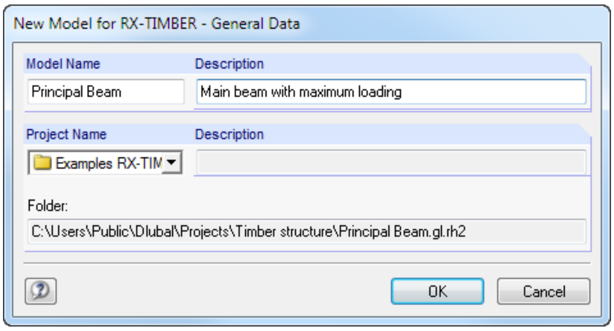


Figure 4.21: Dialog box *New Model for RX-TIMBER - General Data*

When you want to edit the model's general data later,

- select **Properties** on the **Model** menu of the Project Manager (model must have been previously selected) or
- use the *Properties* entry in the model's context menu in the navigator (see Figure 4.10, page 19).

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 will be shown in the header of the printout report but has no further relevance.

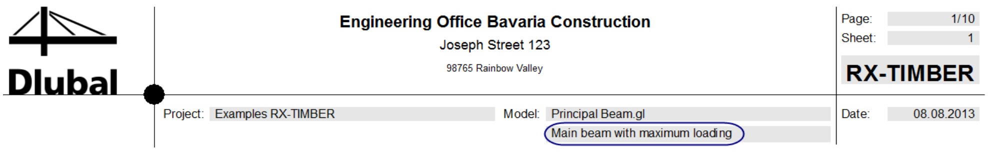


Figure 4.22: Model description in printout report

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 4.1.1, page 14).

In addition, the *Description* and *Folder* of the selected project are displayed for information.

4.3 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 4.1.1 on page 15. 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.

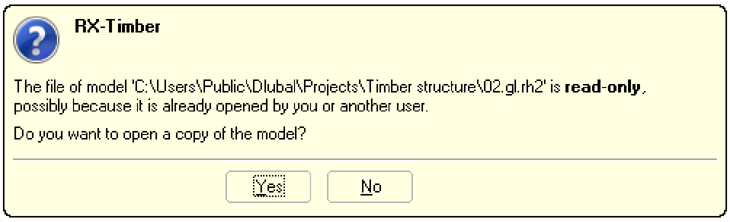


Figure 4.23: Query to open the model

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 normally 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 4.1.1, page 18).

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.

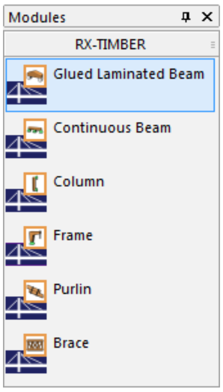
A dialog box opens where you can define the storage location for the file PRO.DLP (see Figure 4.19, page 25).

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.

5. Input

The input windows are presented using **RX-TIMBER Glued Laminated Beam** as an example program.

First, you have to create or open a model (see chapter 4.2, page 26) before you can start entering model and load data.



The program window of the selected RX-TIMBER program appears. On the left, a navigator managing the available windows is displayed. The term "window" refers to the central part of the window where you can specify data for the model and check the result details listed in the output after the calculation.

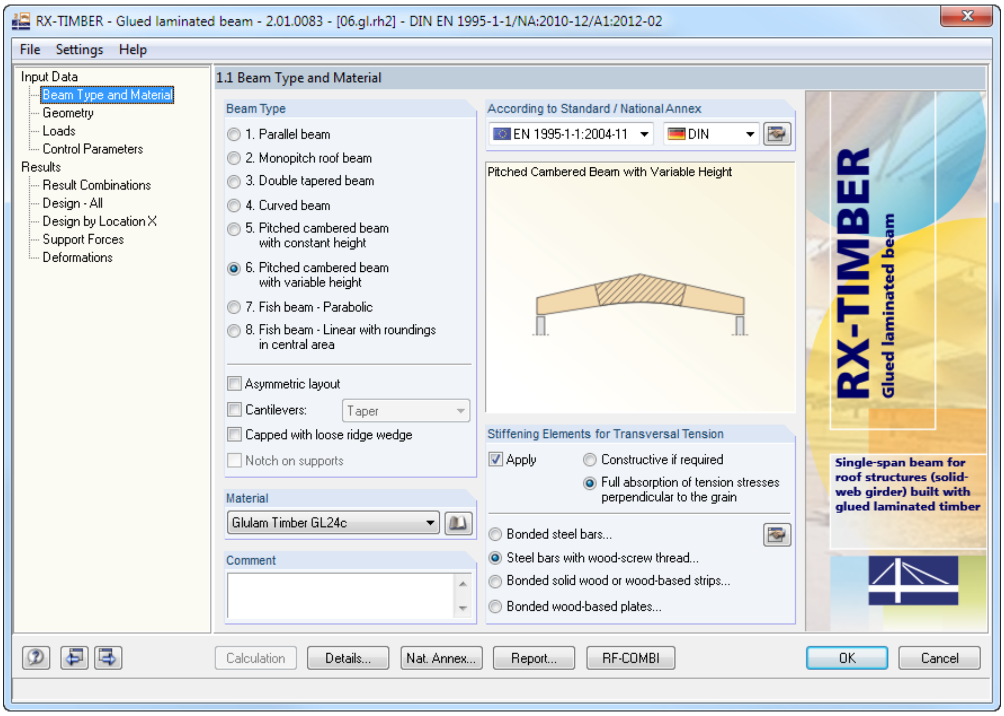


Figure 5.1: RX-TIMBER Glued laminated beam, window 1.1 *Beam Type and Material*

To go to a particular window, click its entry in the navigator. Use the buttons shown on the left to set the previous or next window. Alternatively, you can use the function keys [F2] (next) or [F3] (previous) to browse through the windows.

To save the input and exit the timber program, click [OK]. If you click [Cancel], you exit the program but without saving the data.

The buttons below the window have the following functions:

Button	**Function**
Calculation	Starts the calculation
Details...	Opens a dialog box with detailed settings (→ chapter 6.1)
Nat. Annex...	Opens the standard parameters for the national annex (→ chapter 6.2)
Report...	Displays the print preview (→ chapter 8)
RF-COMBI	Opens the add-on module RF-COMBI (→ chapter 6.3)

Table 5.1: Default buttons

5.1 Beam Type and Material

When you have started the program, window 1.1 *Beam Type and Material* is displayed (see Figure 5.1) where you enter the basic settings for the beam that you want to design. The beam is additionally shown in a graphic.

Beam type

The dialog section *Beam Type* offers you eight beam types for selection:

- Parallel beam
- Monopitch roof beam
- Double tapered beam
- Curved beam
- Pitched cambered beam with constant height
- Pitched cambered beam with variable height
- Fish beam - parabolic
- Fish beam - linear with roundings in central area

Depending on the beam type, up to four options are available below the list. These options allow you to specify the geometry in detail:

- Asymmetric layout
- With cantilevers: horizontal, parallel, taper or offset
- Capped with loose ridge wedge
- Notch on supports

The following table provides an overview about the possibilities how different beam types can be defined:

Beam type	Asymmetric lay-out	With cantilevers				With lose ridge wedge	Stiffening ele-ments for trans-versal tension
		Horizontal	Parallel	Taper	Offset		
1			x	x	x		
2		x		x	x		
3	x	x		x	x		x
4	x	x	x	x	x		x
5	x	x	x	x	x	x	x
6	x	x	x	x	x	x	x
7			x	x	x		
8			x	x	x		

Table 5.2: Overview about beam types

According to standard / national annex

The design standard is defined uniformly for all types of design. Two timber standards are available for selection.

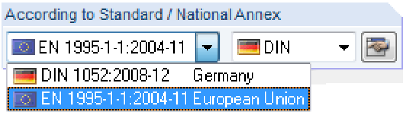


Figure 5.2: Selection of design standard

When EN 1995-1-1:2004 is set, you can select the *National Annex* from the list to the right.

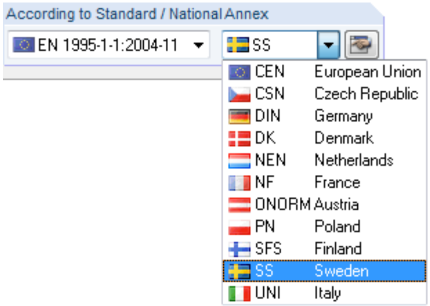


Figure 5.3: Selection of national annex

Use the [Edit] button to check the factors of the selected national annex. If necessary, you can adjust the coefficients.

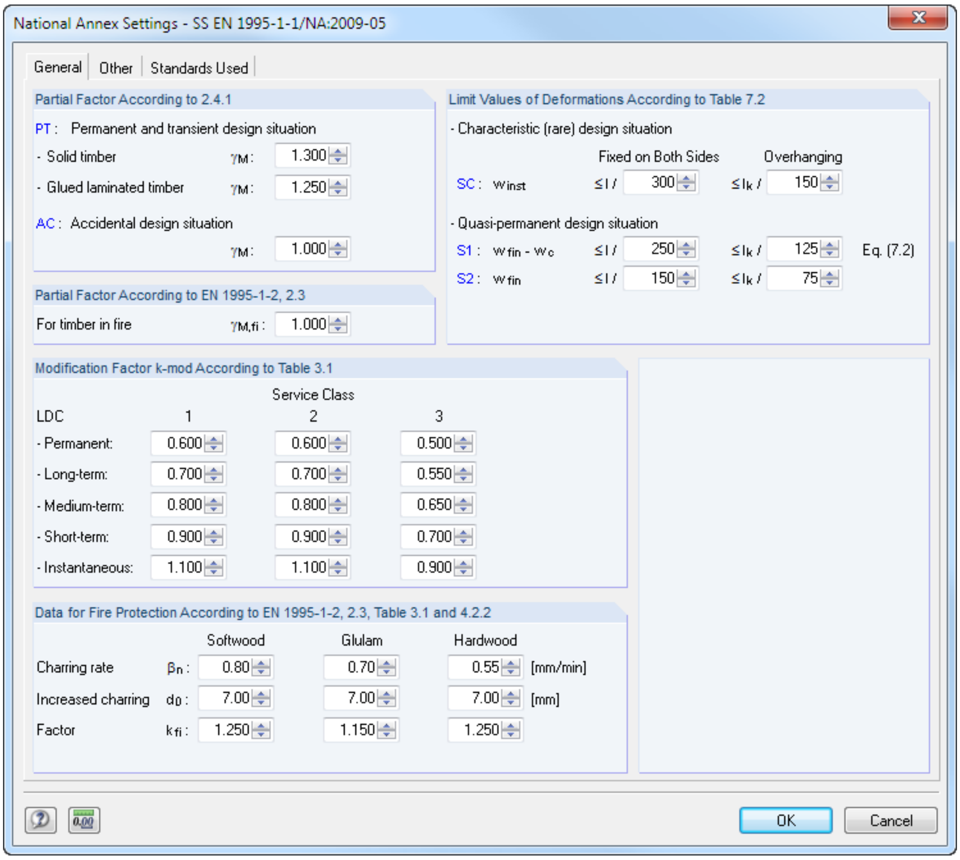


Figure 5.4: Dialog box *National Annex Settings*, tab *General*

The dialog box is described in detail in chapter 6.2 on page 56.

Material

In the window section *Material*, you can select the relevant timber grade in a list. The program provides standardized material grades for glued laminated timber according to EN 1995-1-1: 2004-11 and DIN 1052:2008-12.

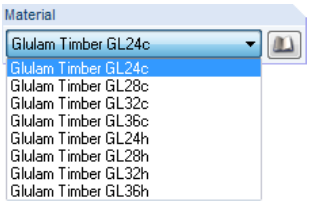


Figure 5.5: Selection of strength class for glued-laminated timber

Use the [Library] button to access the material library where you can check the material properties used for the calculation. The predefined timber grades cannot be edited. In this way, it is ensured that the material properties used by the program correspond to the ones of the displayed timber class.

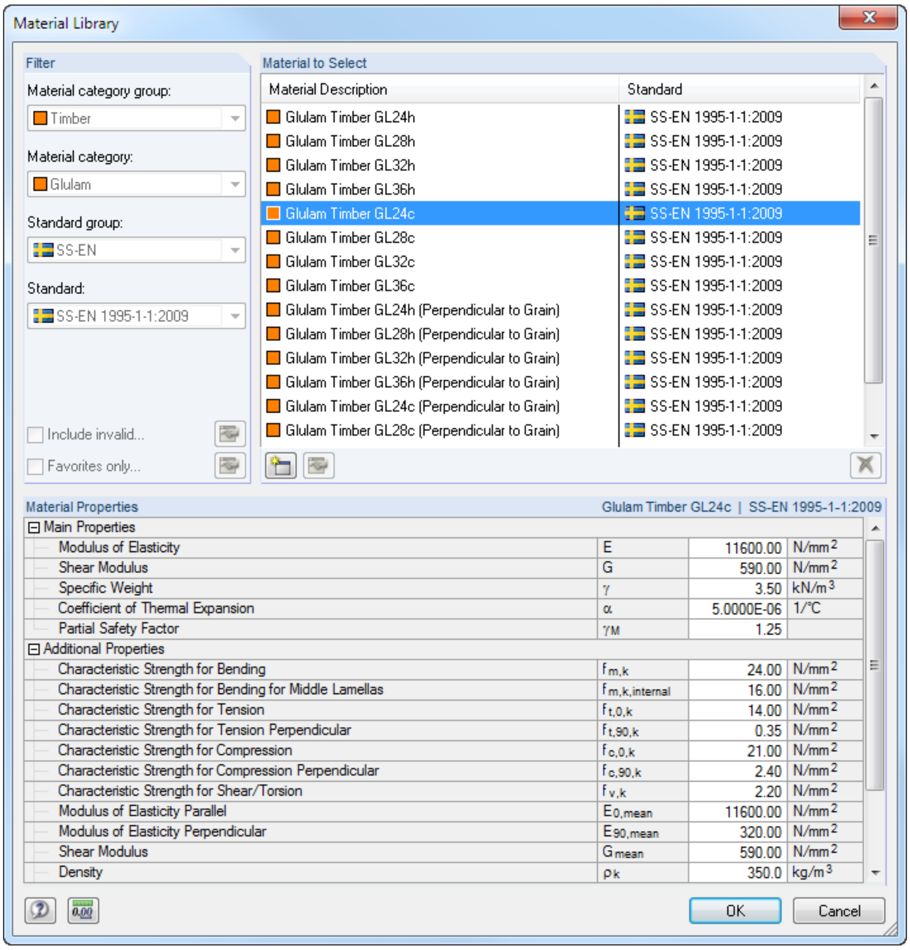


Figure 5.6: Material Library

To create a user-defined timber grade, click the [New] button in the library. The dialog box *New Material* opens. The parameters of the entry selected in the list *Material to Select* are preset. Creating a new material is easier when you choose a material with similar properties before you access the dialog box. Enter the *Material Description*, define the *Material Properties*, and assign the material to the *Filter* categories. The new timber grade is now available for all RX-TIMBER calculations.

You can also [Edit] or [Delete] user-defined materials in the library via the buttons.

Stiffening elements for transversal tension

In the window section *Stiffening Elements for Transversal Tension*, you decide if you want to take into account a stiffening for transversal tension. The setting is only relevant for the beam types 3 to 6. For the remaining beam types, the option is set inactive.

If the *Apply* check box is ticked, you can choose between

- constructive if required
- full absorption of tension stresses perpendicular to the grain.

If the first option is selected, the program checks if a structural stiffening for transversal tension according to the standard's specifications is required and must be taken into account, where applicable. If the second option is selected, the entire transversal tension stress is absorbed by stiffening elements, irrespective of whether a stiffening would be structurally needed.

The following stiffening elements can be defined for the transversal tension stiffening:

- Bonded steel bars
- Steel bars with wood-screw thread
- Bonded solid wood or wood-based strips
- Bonded wood-based plates

Use the [Setting] button in this dialog section to access the definition parameters for the selected element.

Bonded steel bars

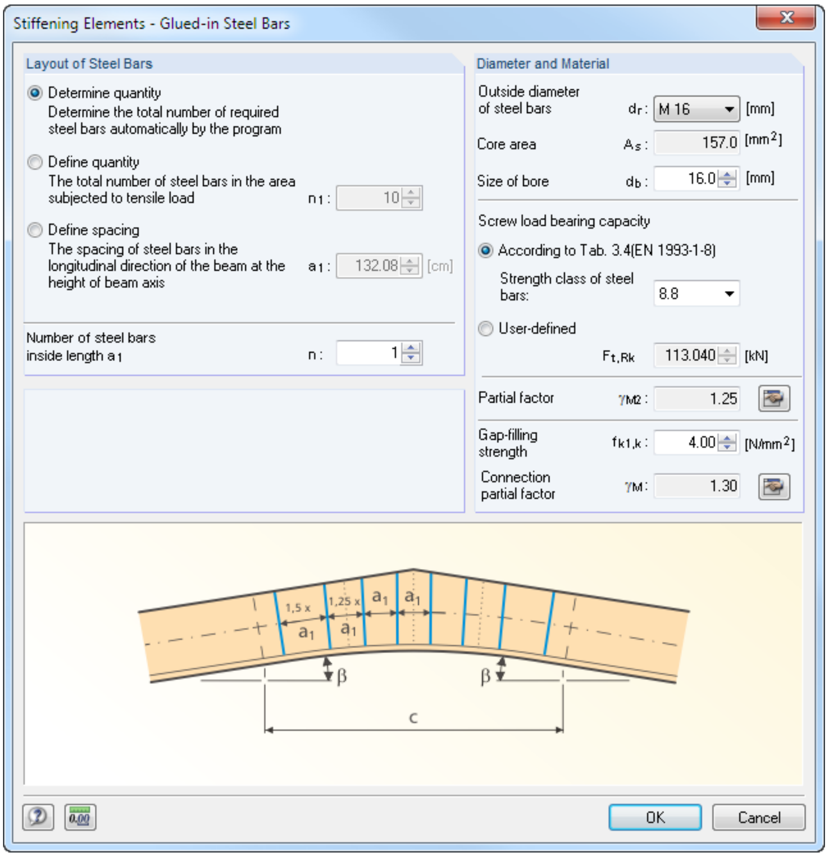


Figure 5.7: Dialog box *Stiffening Elements - Glued-in Steel Bars*

In the dialog section *Layout of Steel Bars*, you decide how the steel bars are arranged. The following three layouts are available for selection:

- *Determine quantity*
  RX-TIMBER automatically calculates the number of required steel bars and the spacings resulting from it.
- *Define quantity*
  Define the number n_1 of steel bars to be used. RX-TIMBER tries to perform the design with the specified setting. The members will be distributed automatically over the length *c* (see dialog graphic).
- *Define spacing*
  Define the spacing a_1 for the bars. RX-TIMBER determines the required number of bars and tries to incorporate them considering the specified spacing.

Irrespective of the selected layout, you can specify the number *n* of steel bars within the length a_1. The bars in the layout can be arranged side by side as well as offset to each other.

The dialog section *Diameter and Material* defines the properties of the steel bars. You can specify the *Outside diameter* d_r of the steel bars, the *Strength class* as well as the *Screw load bearing capacity* and the *Gap-filling strength*. More detailed settings are available: When specifying the outside diameter, an appropriate *Size of bore* d_b will be indicated automatically. If necessary, it can be adjusted in order to consider special methods of production for the calculation.

With the [Edit] button you can set the *Partial factor* γ_{M2} and the *Connection partial factor* γ_M: The dialog box *National Annex Settings* opens (see Figure 6.3, page 56) where you can adjusted the factors.

In case of a complete absorption of tension stresses perpendicular to the grain, the program designs the stiffenings according to EN 1995-1-1/NA:2010, equation (NA.95a) or (NA.95b). Equation (NA.95b) is tantamount to increasing the spacings of the stiffening for transversal tension by 50 %. Therefore, the factor 1.5 is used for a full absorption in the outside area.

In case of a structural stiffening, the program distributes the bars uniformly over the entire cross-section.

When number or spacing are user-defined, the beam graphic in window 1.2 *Geometry* also represents the corresponding steel bars (see Figure 5.10, page 36). When they are determined by the program, a visualization will be available only after the calculation.

Steel bars with wood-screw thread

The settings in the dialog box for defining steel bars with wood-screw threads are similar to the ones described above (see Figure 5.7). But now you specify the *Nominal diameter of steel bars* instead of the outside diameter.

Dlubal

Bonded solid wood or wood-based strips

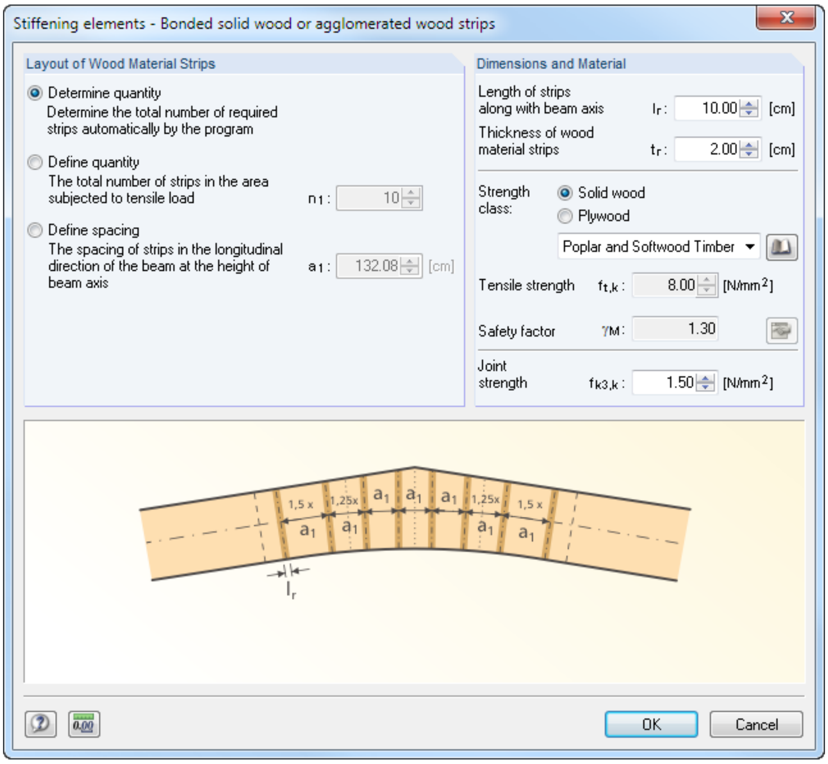


Figure 5.8: Dialog box *Stiffening elements - Bonded solid wood or agglomerated wood strips*

In the dialog section *Layout of Wood Material Strips*, you decide how the strips are arranged. Three layouts are available for selection:

- *Determine quantity*
  RX-TIMBER automatically calculates the number of required solid wood or wood-based strips and the spacings resulting from it.
- *Define quantity*
  Define the number n_1 of wood strips to be used. RX-TIMBER tries to perform the design with the specified setting. The assumed distribution of strips is represented in the graphic.
- *Define spacing*
  Define the spacing a_1 for the strips. RX-TIMBER determines the required number of strips and tries to incorporate them considering the specified spacing.

The dialog section *Dimensions and Material* defines the properties of the wood strips. First, specify the *Length* l_r in relation to the beam's longitudinal axis as well as the *Thickness* t_r of the strips. In addition, decide whether the strips are made of *Solid wood* or *Plywood*. If solid wood is selected, the program assumes that the grain of the wood strips runs perpendicular to the beam's longitudinal axis. The *Tensile strength* $f_{t,k}$ and the *Safety factor* γ_M of the selected timber strength class are indicated for information.

The *Joint strength* $f_{k3,k}$ defines the force transmission between wood strips and beam.

If plywood is selected, you can change the *Safety factor* γ_M by using the [Edit] button: The dialog box *National Annex Settings* opens (see Figure 6.3, page 56) where you can adjust the factor.

Bonded wood-based plates

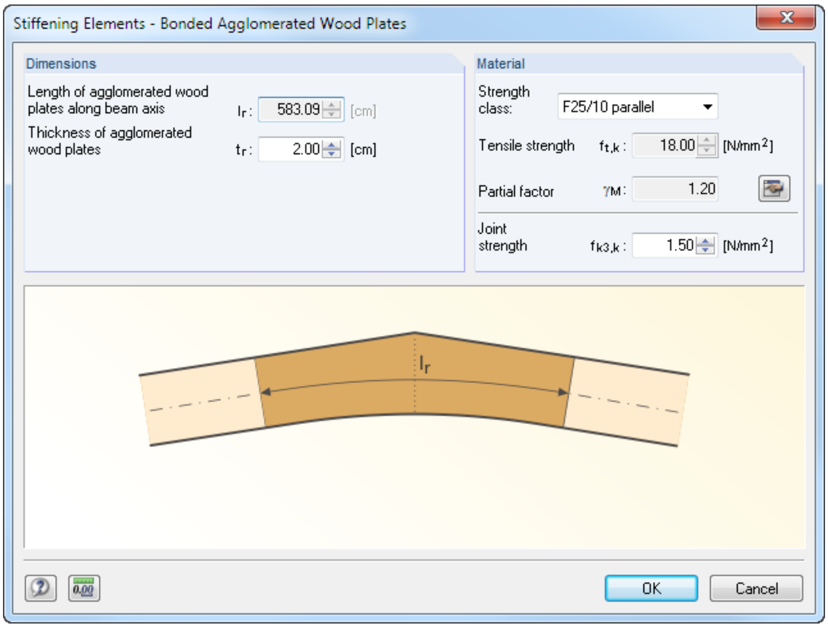


Figure 5.9: Dialog box *Stiffening Elements - Bonded Agglomerated Wood Plates*

In the dialog section *Dimensions*, the *Length* l_r of the bonded wood plates is specified: It represents the length of the curved beam axis resulting from the settings entered in window 1.2 *Geometry*. The *Thickness* t_r of the plates can be user-defined.

The *Strength class* list in the dialog section *Material* offers you, in addition to the timber grade, settings for a parallel or square arrangement of wood plates. The *Tensile strength* $f_{t,k}$ used for the calculation is indicated for information.

With the [Edit] button you can set the *Partial factor* γ_M: The dialog box *National Annex Settings* opens (see Figure 6.3, page 56) where you can adjusted the factor.

The *Joint strength* $f_{k3,k}$ defines the possible force transmission between wood plate and beam.

Comment

In the dialog section *Comment* of window 1.1 *Beam Type and Material*, you can enter a comment for example to describe the beam in detail. The comment appears in the printout report, too.

5.2 Geometry

In window 1.2 *Geometry*, you define the beam geometry based on the beam type that you have selected in window 1.1 *Beam Type and Material*. Depending on your choice and the selected additional options (asymmetric, with/without cantilever, lose ridge wedge), the following parameter groups are displayed:

- Building dimensions
- Roof beam geometry
- Cantilevers (left/right)
- Cross-section
- Data for lateral buckling
- Attic left
- Attic right
- Information parameters

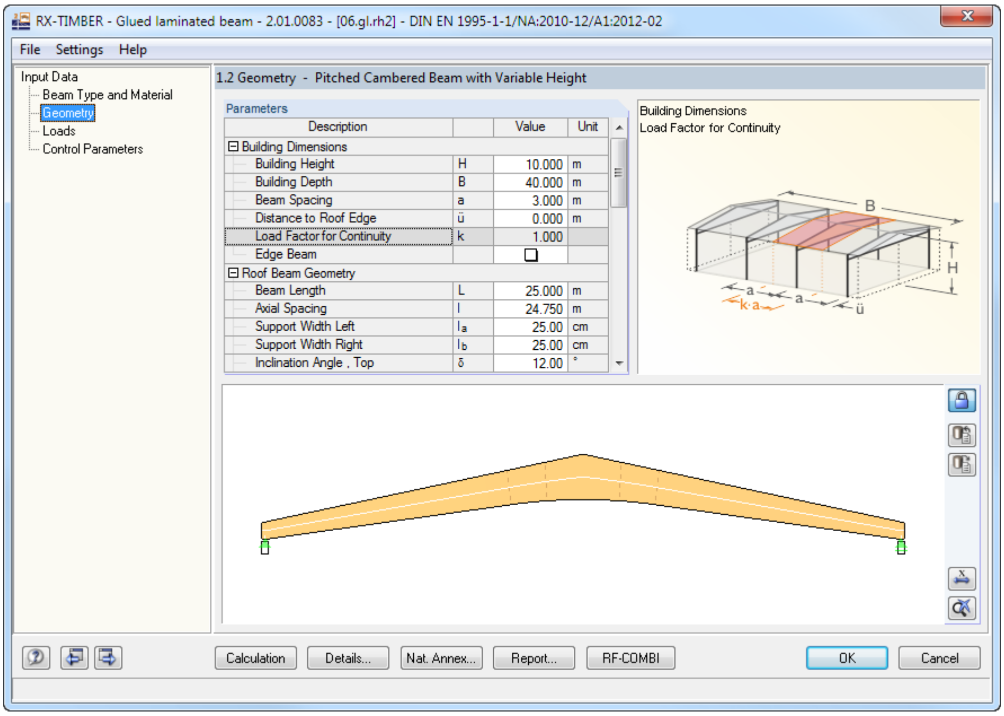


Figure 5.10: Window 1.2 *Geometry*

To have a clear data overview, you can open and close the groups by a simple click on the [-] or [+] sign indicated at the beginning of each group name.

Each parameter is interactive with the graphic to the right. The relevant value is highlighted in color in the outline of the building or beam.

Building dimensions

The *Building Dimensions* group (see Figure 5.10) is the same for all beam types. Its parameters are needed for the automatic load determination of wind and snow loads (*Building Height H* and *Building Depth B*) as well as for the definition of the application area (*Beam Spacing a* and *Distance to Roof Edge ü*).

The *Distance to Roof Edge ü* allows you to consider an additional load application area for edge trusses. As load application area RX-TIMBER assumes 2 · a/2 by default, for edge trusses it assumes a/2 + ü.

The *Load Factor for Continuity k* allows for increasing the resulting truss load in order to take into account the influence from effects of continuity. As the influence may vary depending on the roof area, you can determine the factor manually and enter it into the table.

When you tick the option *Edge Beam*, the program will assume the reduced load application area of an edge truss.

Roof beam geometry

This group defines the individual dimensions of the beam. The number of parameters varies according to the beam type. Each input value is displayed for better understanding in the graphic to the right. Some values depend on each other and thus are interactive: When you specify the *Beam Length L*, the *Axial Spacing l* changes automatically. When you enter a value for the *Support Width Left l_a* or *Right l_b*, the axial spacing will be adjusted accordingly.

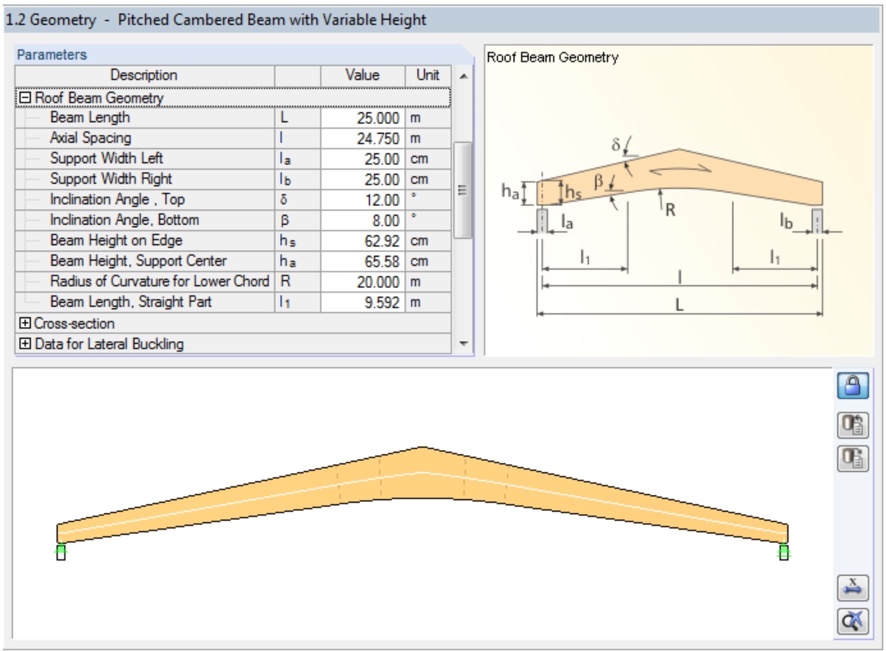


Figure 5.11: Parameters of the *Roof Beam Geometry*

The setting for adjusting parameters interactively can be deactivated or reactivated by using the button shown on the left. If the button [Plausibility Check] is disabled, you can enter any value. However, it is recommended to enable the check again when the input is complete to ensure a correct definition of the geometry. If the geometry is not defined within the specified limit values, a message appears.

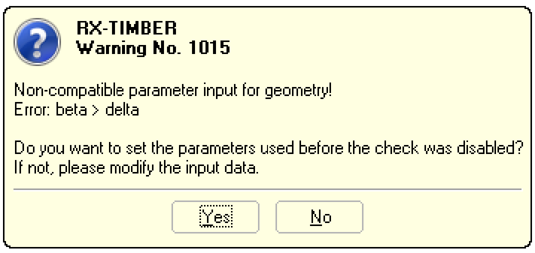


Figure 5.12: Plausibility check: warning in case of problems in geometry

The message informs you about the parameters that cause problems, offering two options for correction: [Yes] resets the default values resulting in a coherent geometry, [No] takes you back to the input window so that you can correct the parameters manually. The relevant parameters are highlighted in color.

Cantilever

This group appears only when the option *Cantilevers* has been selected in window 1.1. Depending on whether a symmetric or asymmetric beam has been specified, one group or two groups are displayed to define the overhang(s).

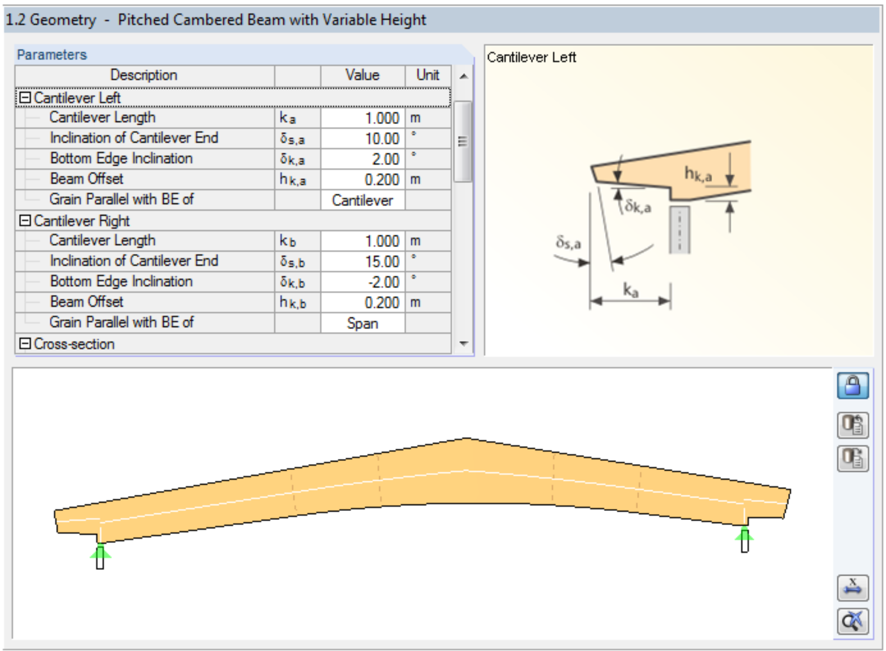


Figure 5.13: Parameters of *Cantilever*

A graphic is also displayed for cantilevers depending on the cantilever type, describing the meaning of the parameters.

Cross-section

This group defines the *Cross-Section Width b* as well as the *Lamella Thickness t*. The thickness *t* also affects the increment that can be defined later during the beam optimization.

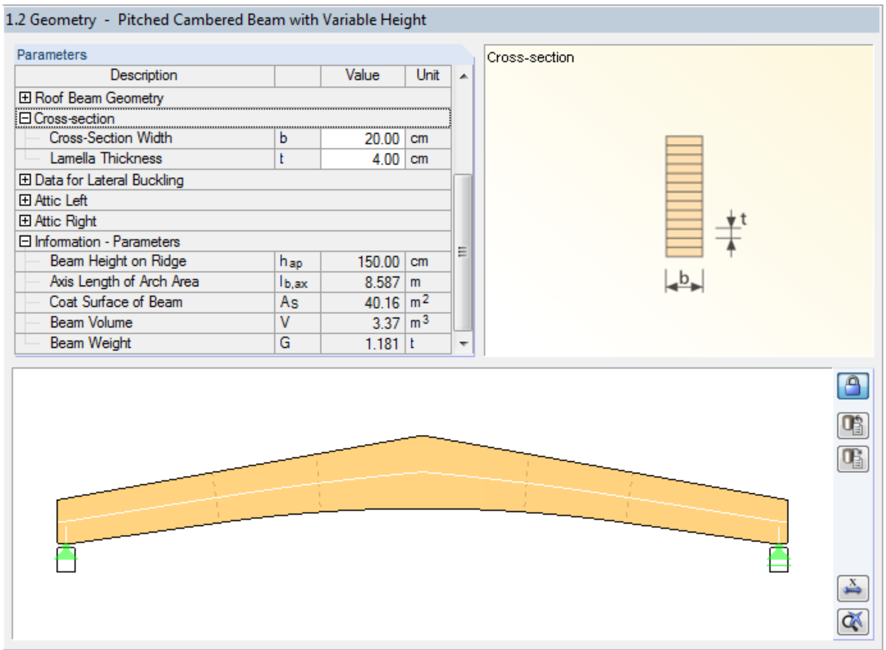


Figure 5,14 Parameters for *Cross-section*

Data for lateral buckling

With settings entered in this group you decide if an analysis for safety against lateral buckling is performed. If you select the option *Beam Endangered by Lateral Buckling*, the relevant parameters become accessible.

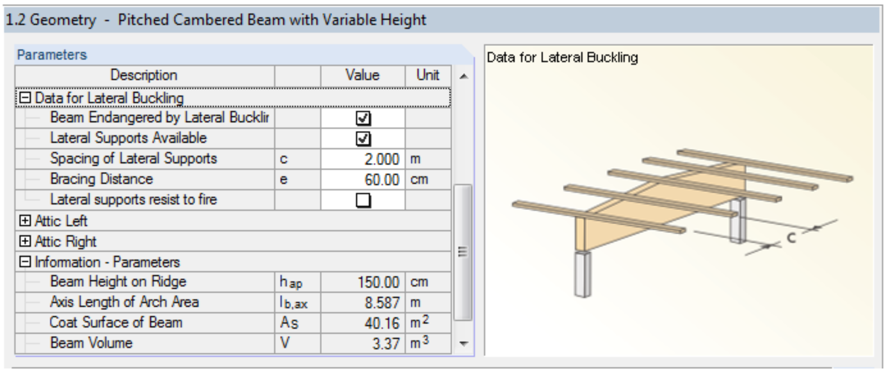


Figure 5.15: Parameters of *Data for Lateral Buckling*

By ticking the check box for *Lateral Supports Available*, you can take into account additional lateral supports, for example in the form of purlins. Define their spacings and position in the subsequent table rows.

With the value *Bracing Distance e* you can determine the lateral buckling moment T_d on the support for forked supports.

$$T_d = M_d \cdot \left[\frac{1}{80} - \frac{1}{60} \cdot \frac{e}{h} \cdot (1 - k_m) \right]$$

Equation 5.1: Determination of lateral buckling moment T_d according to DIN 1052: 2004-08, equation (14)

According to EN 1995-1-1, it should be possible to absorb the following moment on the support by the forked support or a bracing.

$$M_{tor,d} = M_d / 80$$

Equation 5.2: Moment $M_{tor,d}$ according to DIN EN 1995-1-1/NA:2010-12, equation (NA.129)

When designing the beam according to EN 1995-1-1, the program checks whether the slenderness for lateral buckling λ_{ef} is less than or equal to 225. If the condition is not fulfilled, the design won't be performed.

Attic left/right

With the settings for *Attic Left* or *Attic Right* you can decide if you want to consider an attic on the roof.

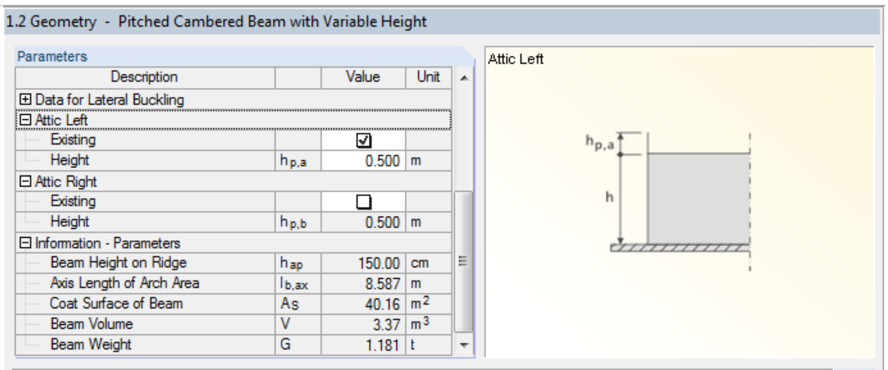
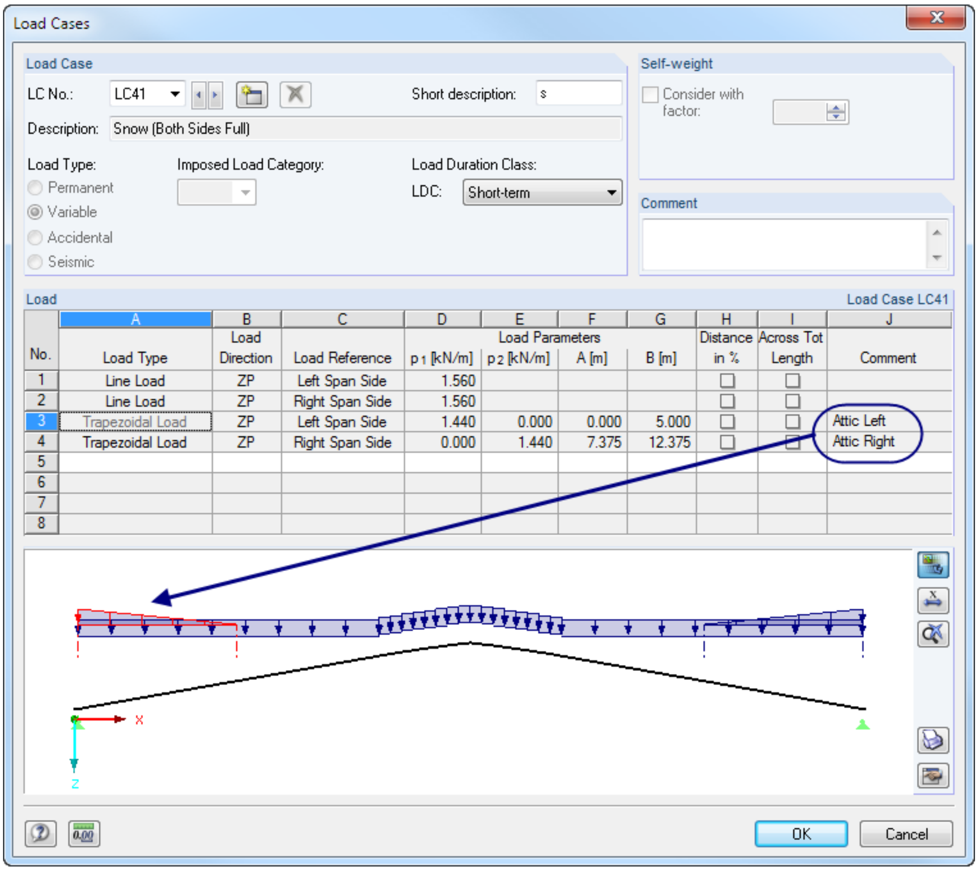


Figure 5.16: Parameters for *Attic Left* and *Attic Right*

The attic parameters affect the snow load generation:

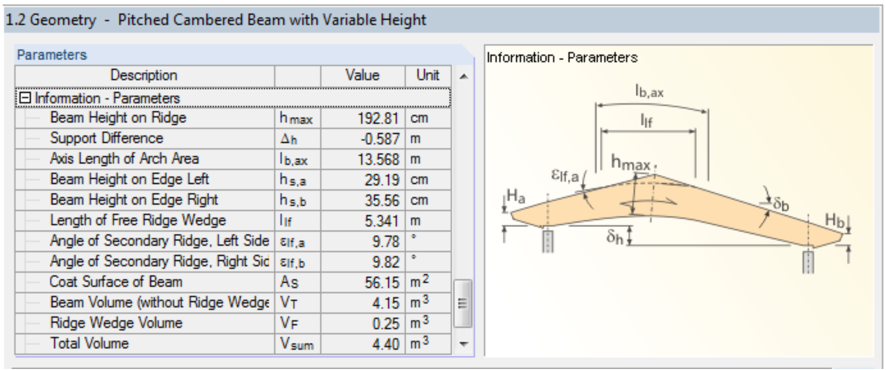


No.	A Load Type	B Load Direction	C Load Reference	D p1 [kN/m]	E p2 [kN/m]	F A [m]	G B [m]	H Distance in %	I Across Tot Length	J Comment
1	Line Load	ZP	Left Span Side	1.560				☐	☐	
2	Line Load	ZP	Right Span Side	1.560				☐	☐	
3	Trapezoidal Load	ZP	Left Span Side	1.440	0.000	0.000	5.000	☐	☐	Attic Left
4	Trapezoidal Load	ZP	Right Span Side	0.000	1.440	7.375	12.375	☐	☐	Attic Right
5										
6										
7										
8										

Figure 5.17: Influence of attic on the generation of snow loads

Information parameters

The final group is represented by the *Information - Parameters* including various geometrical data. The values cannot be modified, they are indicated only for information. The number of displayed values varies according to the beam type.



Description		Value	Unit
⊟ Information - Parameters			
Beam Height on Ridge	h_{max}	192.81	cm
Support Difference	Δh	-0.587	m
Axis Length of Arch Area	$l_{b,ax}$	13.568	m
Beam Height on Edge Left	$h_{s,a}$	29.19	cm
Beam Height on Edge Right	$h_{s,b}$	35.56	cm
Length of Free Ridge Wedge	l_{lf}	5.341	m
Angle of Secondary Ridge, Left Side	$\varepsilon_{lf,a}$	9.78	°
Angle of Secondary Ridge, Right Sic	$\varepsilon_{lf,b}$	9.82	°
Coat Surface of Beam	A_S	56.15	m^2
Beam Volume (without Ridge Wedge	V_T	4.15	m^3
Ridge Wedge Volume	V_F	0.25	m^3
Total Volume	V_{sum}	4.40	m^3

Figure 5.18: *Information - Parameters*

A graphic is displayed for each single value, describing the meaning of the corresponding parameter.

5.3 Loading

All loads are defined in this window. In the following, the individual window sections are described in detail.

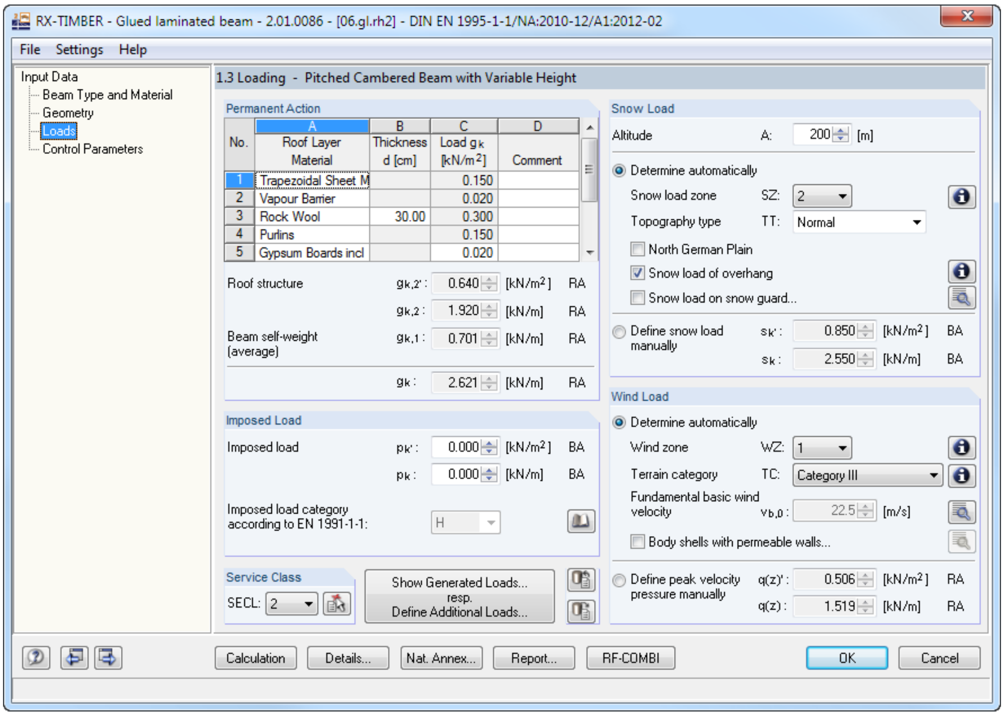


Figure 5.19: Window 1.3 *Loading*

Permanent action

The self-weight of the beam is automatically taken into account. You do not need to define it in particular. Please find a description on page 49 how to deactivate the default setting.

You can use the table to define additional, permanently acting roof loads such as the roofing. Click into a table cell of column A to activate the [...] button shown on the left (see Figure 5.19). Use this button to access the material library offering predefined roof loads (see Figure 5.20). They are sorted by different *Categories* and indicated in [kN/m²] or [kN/m³] referring to the area or volume. To select a material, click it and use the [OK] button to take it into the table. You can also double click the material to import it into window 1.3.

To access the table entries of the material library (Figure 5.20) for modification, use the [Edit] button shown on the left. In this way, you can adjust the description and the specific weight of the material currently selected.

The final row of the material table is empty so that you can extend the database: After entering a new material, a new empty row is added automatically. In this way, you can extend the library by specifying any materials.

After editing or extending the material library, close it by clicking the [OK] button to save the modifications. You can also use the [Apply] button shown on the left to save the adjustments.

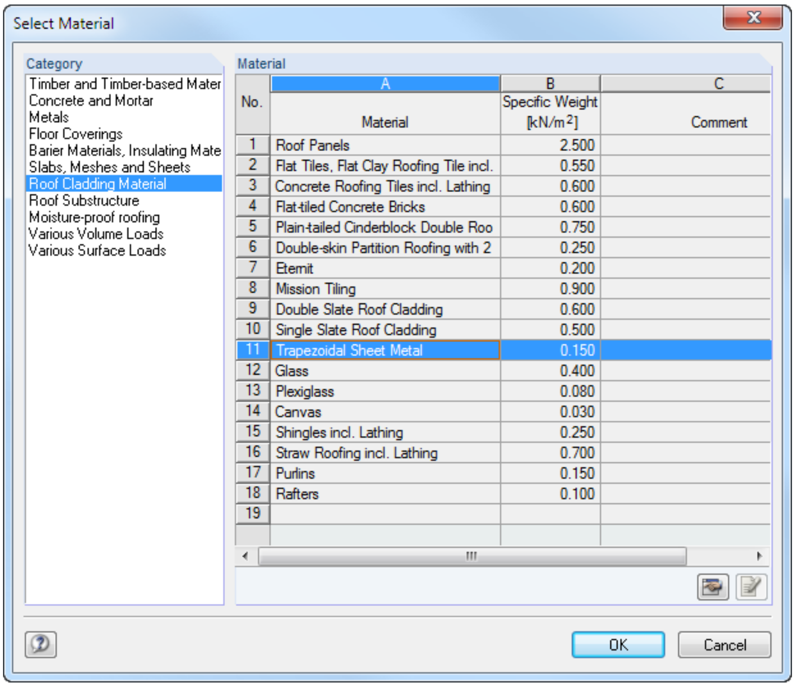


Figure 5.20: Material library for permanent actions

When you enter loads that are indicated as volume loads in the library, you have to specify additionally the *Thickness d* in table column B.

In addition, each load can be described by a comment entered in table column D.

Below the table the program sums up the area loads ($g_{k,2}'$), converts them into a distributed load ($g_{k,2}$) and displays them as permanent load (g_k) together with the self-weight ($g_{k,1}$). The beam's self-weight is shown as average value. However, during the calculation it is taken into account accurately as a variable distributed load.

Imposed load

The two input fields in this window section are connected with each other. For example, if you define the area load p_k' [kN/m²], the program converts it automatically into the corresponding distributed load p_k [kN/m] for a truss. The conversion between both values takes into account the specifications defined for the *Building Dimensions* in window 1.2:

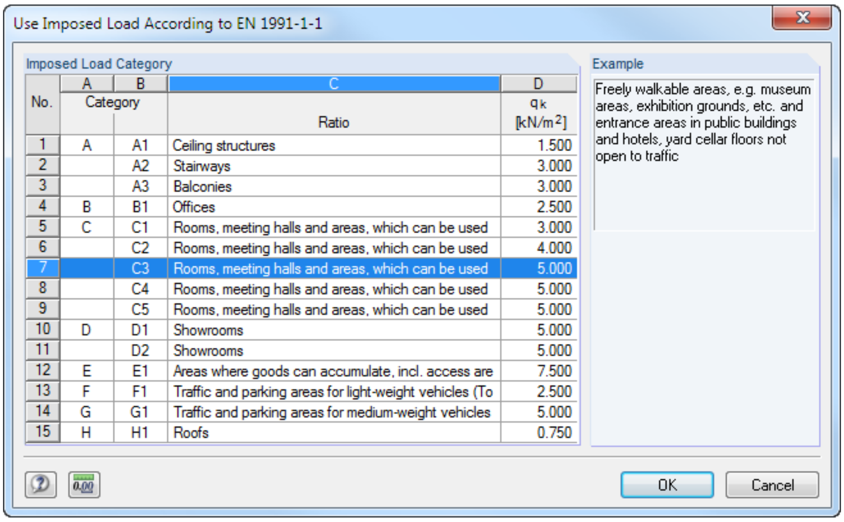
- Edge beam: $p_k = p_k' \cdot (\frac{a}{2} + ü) \cdot k$
- Inner beam: $p_k = p_k' \cdot a \cdot k$

Equation 5.1: Conversion of area loads into distributed loads

You can enter the distributed load p_k also directly. In this case, the input field for p_k' will display the corresponding value of the area load.

Use the button shown on the left to select the imposed load according to [3] or [7] from a table (see Figure 5.21).

When an imposed load is entered, the selection field for specifying the *Imposed load category according to EN 1991-1-1* (or *DIN 1052* and *DIN 1055-100*) is enabled. In most cases you can continue with the preset category *H* without modifying the setting. The category controls the ψ values according to [2] table A.2 or [9] table A.1.1 as well as the load duration class (LDC) according to [1] table 4 or [7] table NA.1.



No.	A Category	B	C Ratio	D q_k [kN/m²]
1	A	A1	Ceiling structures	1.500
2		A2	Stairways	3.000
3		A3	Balconies	3.000
4	B	B1	Offices	2.500
5	C	C1	Rooms, meeting halls and areas, which can be used	3.000
6		C2	Rooms, meeting halls and areas, which can be used	4.000
7		C3	Rooms, meeting halls and areas, which can be used	5.000
8		C4	Rooms, meeting halls and areas, which can be used	5.000
9		C5	Rooms, meeting halls and areas, which can be used	5.000
10	D	D1	Showrooms	5.000
11		D2	Showrooms	5.000
12	E	E1	Areas where goods can accumulate, incl. access are	7.500
13	F	F1	Traffic and parking areas for light-weight vehicles (To	2.500
14	G	G1	Traffic and parking areas for medium-weight vehicles	5.000
15	H	H1	Roofs	0.750

Figure 5.21: Dialog box *Use Imposed Load According to EN 1991-1-1*

The ψ values are taken from the add-on module RF-COMBI where they can be adjusted, if necessary (see Figure 11.12, page 115). For more information, see chapter 6.3 and the RF-COMBI manual that is available for download at www.dlubal.com.

Service class

The service class *SECL* is defined for the entire beam in the bottom left corner of window 1.3. Use the button shown on the left to access a selection dialog box where the individual service classes are shortly described.

Snow load

This window section manages the parameters for the automatic creation of snow load cases. The loads are generated according to the rules specified in EN 1991-1-3 (or DIN 1055-5).

First, enter the *Altitude* above sea level in m. Then, *Determine* the snow load *automatically* or *Define snow load manually*.

Determine automatically

The *Snow load zone SZ* can be selected directly from the list or determined by a double-click in the snow load map (Figure 5.22). The map with the snow load zones can be opened by clicking the [Info] button shown on the left.

If loads are generated according to EN 1991-1-3, the *Topography type TT* must be specified additionally. The list offers three options: *Windswept*, *Normal* and *Sheltered*.

Moreover, the program provides the options *North German Plain*, *Snow load of overhang* and *Snow load on snow guard* allowing together with the relevant additional settings for a specific load generation control. To display additional information, use the [Info] button.

In the snow load map of Germany, the *North German Plain* (Norddeutsches Tiefland - see figure below) is indicated. When you select the corresponding option, the program creates an additional, exceptional load case with 2.3-fold snow loads.

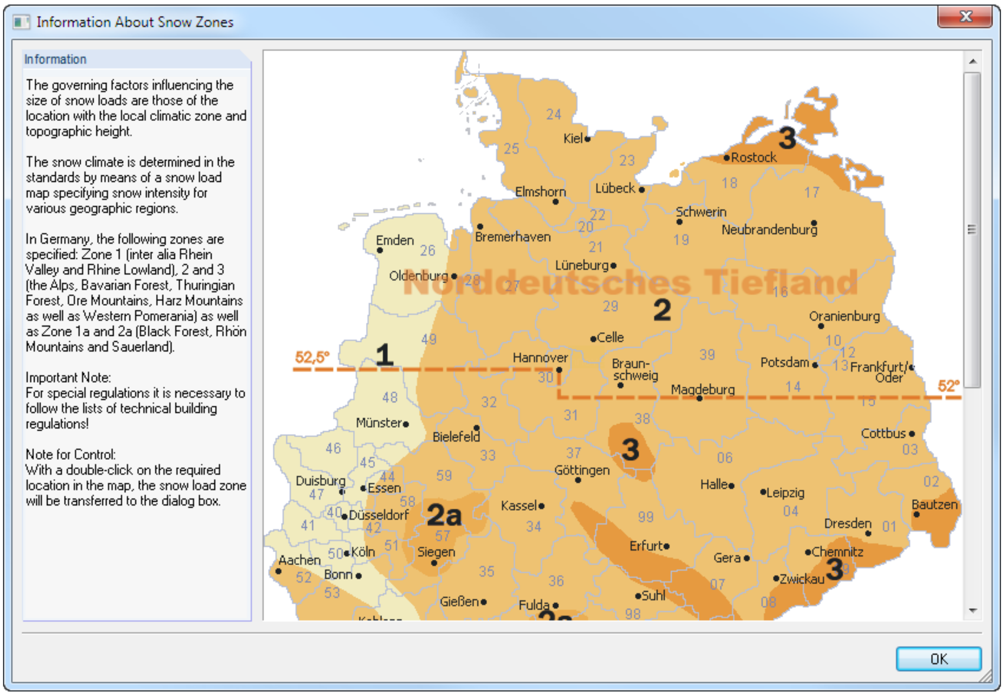


Figure 5.22: Map showing snow load zones in Germany

When you consider the *Snow load of overhang*, a window is available showing information about the equation used to determine the additional load.

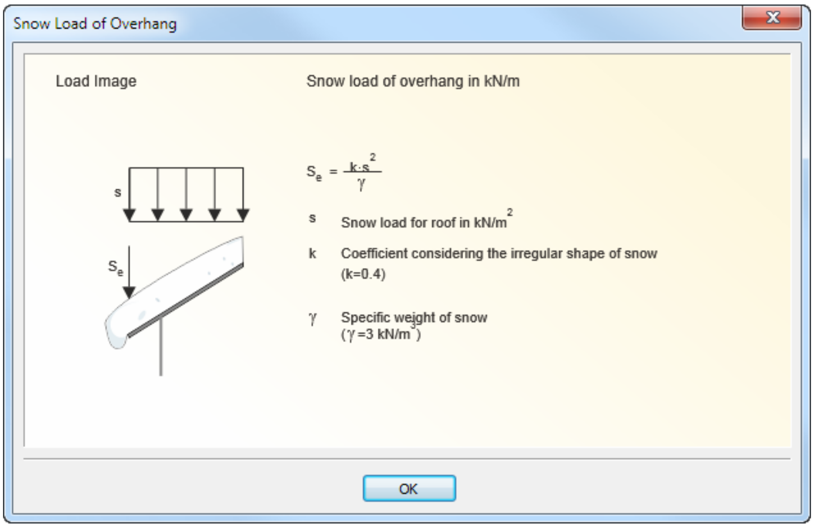


Figure 5.23: Dialog box *Snow Load of Overhang*

When you consider the *Snow load on snow guard*, you have to specify the guard's *Distance* a_R from the edge of the roof.

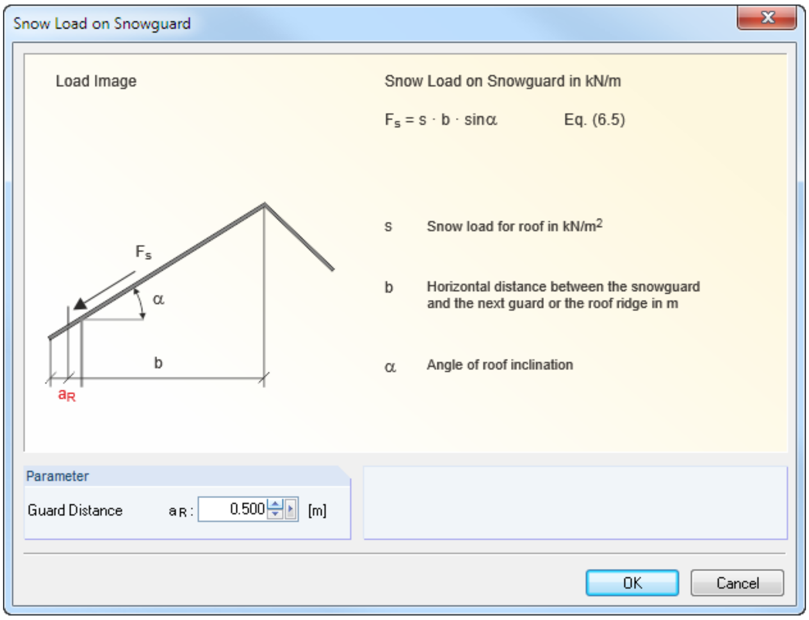


Figure 5.24: Dialog box *Snow Load on Snowguard*

Below, the snow load resulting from altitude and snow load zone is displayed for information as area and distributed load in the inactive input fields of the option *Define snow load manually*. The snow load value is represented by the characteristic snow load s_k. When generating the snow load using s_k, the program takes the shape coefficient according to [5] clause 4.2.5 or [10] clause 6.3, equation (6.5) additionally into account, which is why the value displayed here cannot directly be found later in the load representation.

Define snow load manually

By defining the load manually you can enter the snow load either as area load s_k' or distributed load s_k. The respective conversion between the loads is done analogously to the determination of the imposed load p_k (see Equation 5.1, page 42).

The relevant shape coefficient is also taken into account for the generated loads.

Wind load

This window section manages the parameters for the automatic creation of wind load cases. The loads are generated according to the rules specified in EN 1991-1-4 (or DIN 1055-4).

Wind loads always refer to a closed hall. Additional wind loads occurring for buildings that are open on one side or several sides must be entered specifically. Similar to the generation of snow loads, two options are available to define wind loads: You can choose between *Determine automatically* or *Define manually*.

Determine automatically

When you generate wind loads automatically, the *Wind Zone WZ* as well as the *Terrain Category TC* must be specified. The wind zone can be defined by double-clicking the relevant region in the wind zone map that you open with the [Info] button shown on the left (see figure below).

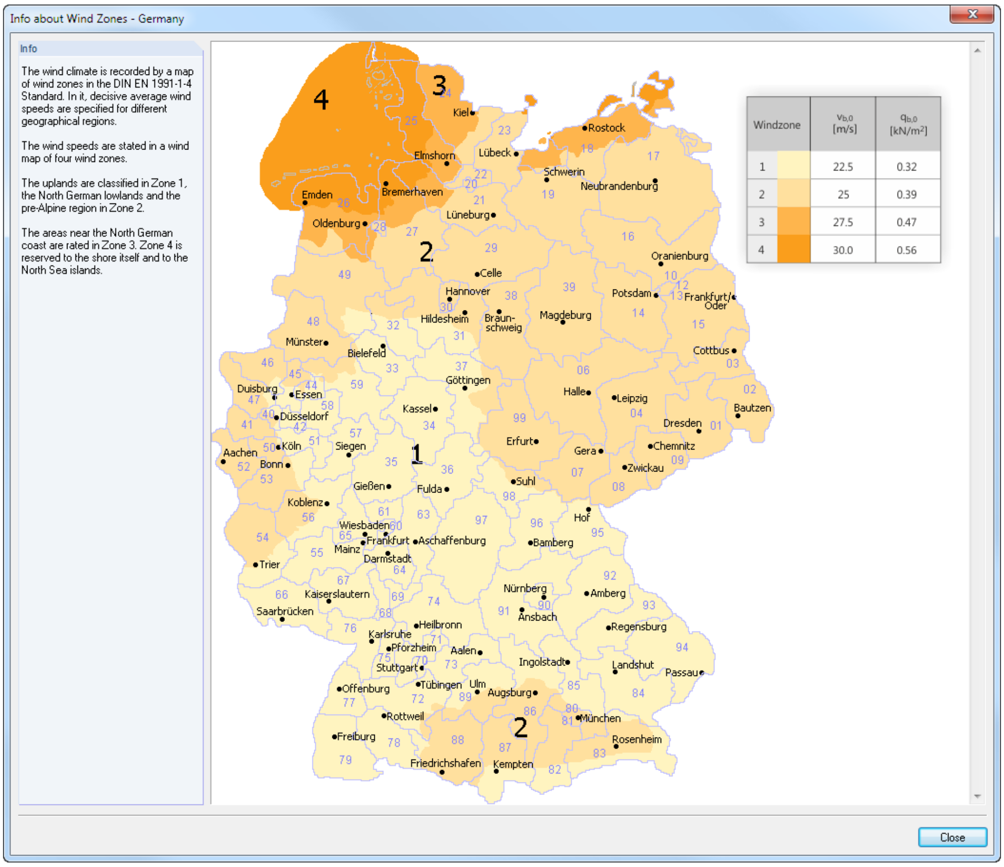


Figure 5.25: Wind zone map for Germany

The wind zone sets the value of the *Fundamental basic wind velocity* $v_{b,0}$ according to [11] clause 4.2. Use the [Edit] button shown on the left to adjust the values for the velocity pressure. The following dialog box appears:

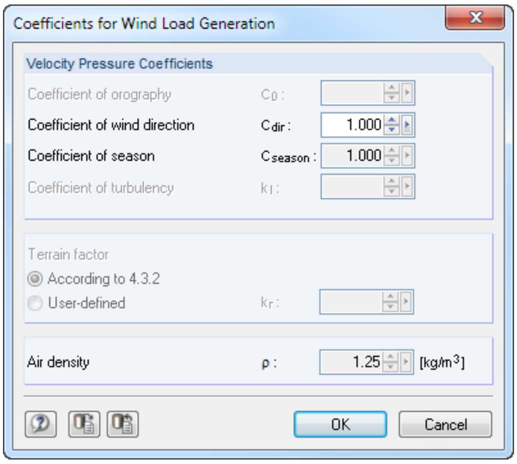


Figure 5.26: Dialog box *Coefficients for Wind Load Generation* according to EN 1991-1-4

By ticking the option *Body shells with permeable walls* it is possible to define a μ value according to [4] equation (19) or [11] equation (7.3) to take into account additional lifting forces due to permeable walls. If the beam has a cantilever, the program determines the load case "roof-lifting internal wind" as well in accordance with the connected wall area. However, the load generation is not able to consider all areas and special cases according to [4] or [11] so that you additionally have to define, if necessary, further actions from such loads (see the following description for *Show Generated Loads / Define Additional Loads*).

Click the [Settings] button shown on the left to open a dialog box where you can define the parameters of the walls in detail.

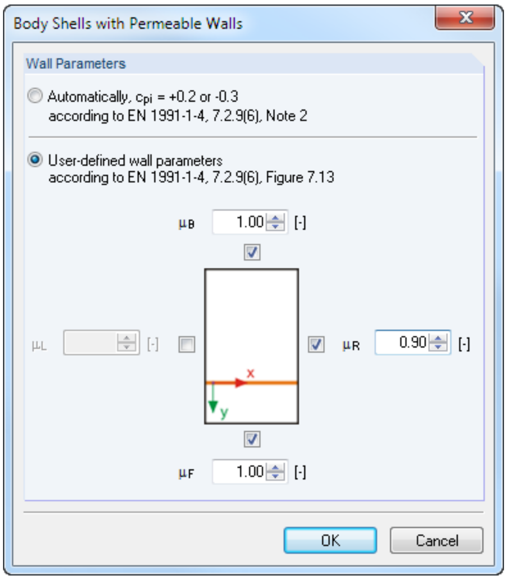


Figure 5.27: Dialog box *Body Shells with Permeable Walls* according to EN 1991-1-4

The dimensions of the building are decisive for the wind load generation. Based on the dimensions specified in window 1.2, the program determines the data fields in column F, G, H and I (see figure below) according to [4] or [11] for the respective beam type and generates the wind loads accordingly. For the zones where compressive as well as suction forces are applied, the program creates two load cases for each wind load case with suction or compressive forces.

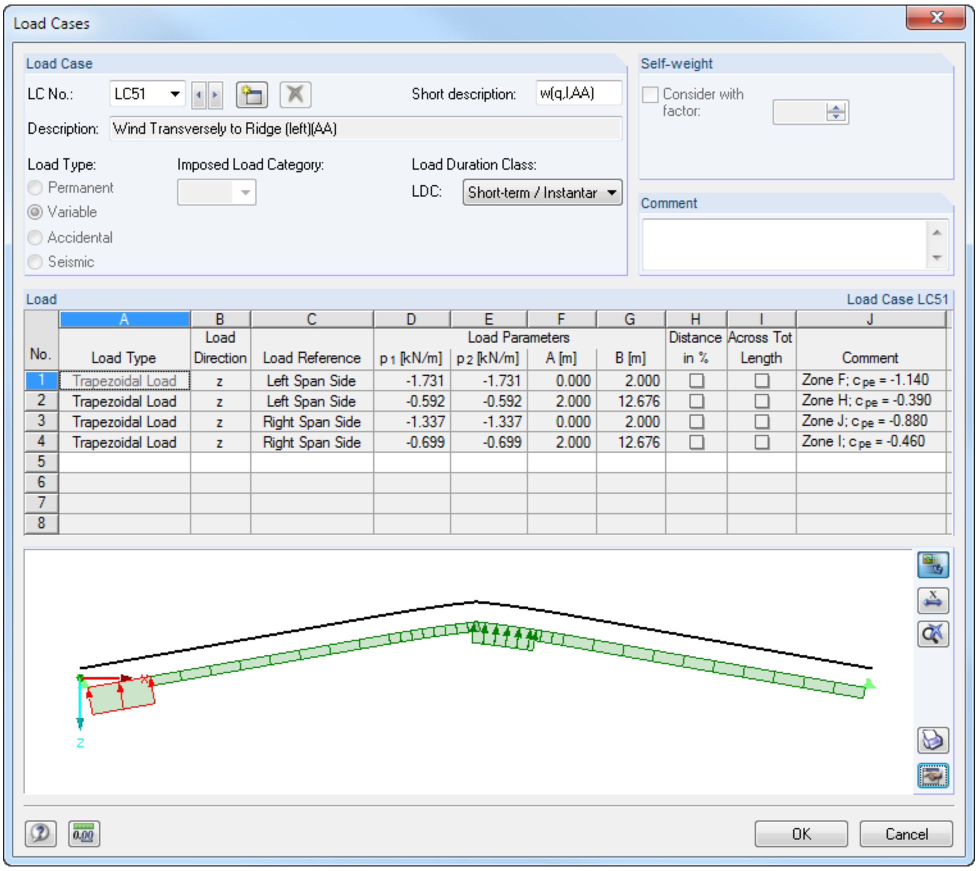


Figure 5.28: Wind suction in load case 51 (case AA)

In case of special regulations, like for the generation of snow loads, follow the lists of the technical construction regulations of your country.

Define peak velocity pressure manually

If you select the option *Define peak velocity pressure manually*, you can enter the wind load either as area load *q(z)'* or distributed load *q(z)*. The conversion between the loads is similar to the calculation of the imposed load p_k (see Equation 5.1, page 42).

Show Generated Loads / Define Additional Loads

The button [Show Generated Loads resp. Define Additional Loads] provides access to the loads generated with the specifications of window 1.3. The dialog box *Load Cases* opens.

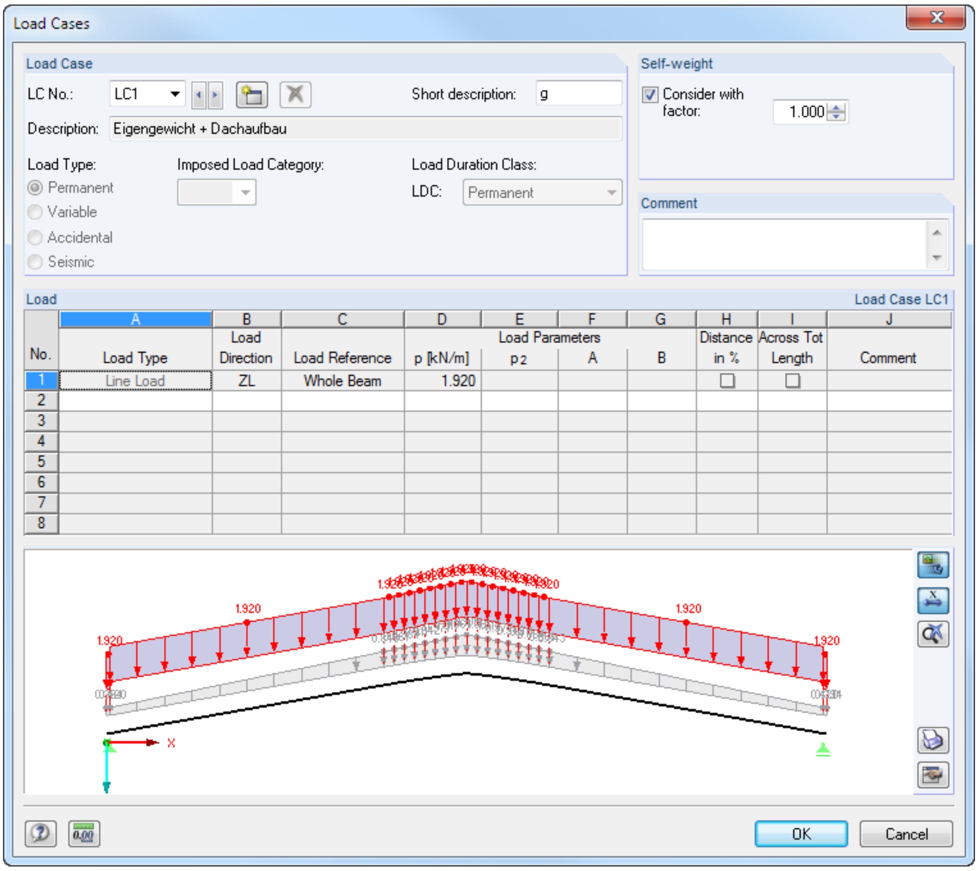
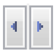


Figure 5.29: Dialog box *Load Cases*

Load case



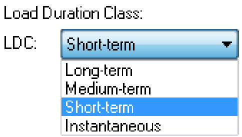
Use the list *LC No.* to select the load case that you want to be shown in the dialog section *Load*. With the arrow buttons [◄] and [►] you can set the next or pre

With the [New] button shown on the left you can create additional load cases. In this way, you can consider loads that are not generated automatically. The value of a user-defined load must be specified as *Load Parameter* in table column D. See page 50 to learn how to enter additional loads.

Additional load cases can be deleted by means of the [Delete] button shown on the left. Please note that generated load cases can be neither deleted nor modified.

The *Short description* makes it easier to keep track of the created result combinations (see chapter 7.1, page 61).

The *Description* helps you to specify the load. It appears also in the printout report where the load cases are listed.



For user-defined load cases you have to specify the *Load Type*. You can select the following options: *Permanent, Variable, Accidental* and *Seismic*. For variable loads it is possible to assign the *Imposed Load Category* like in window 1.3 as well as the *Load Duration Class LDC*.

Self-weight

By clearing the check box in the dialog section *Self-weight* you can deactivate the setting for taking into account the self-weight. It is also possible to scale the self-weight by a factor. Factor 1.0 is set by default.

Comment

Each load case can be described by user-defined notes.

Load

Table and graphic in the dialog section below display the loads of the load case set above.

In load case 1 "Self-Weight + Roof Finishes", the self-weight is represented graphically. The table, however, does not display it separately. It only lists the loads that have been defined as roof structure ($g_{k,2}$) in window 1.3.

The values of the snow and wind load are generated with the corresponding factors. Thus, the graphic displays the load that is applied effectively to the beam.

The wind load in the single load cases is generated perpendicular and parallel to the ridge direction. If the roof can be stressed by suction as well as compression according to the standard, the program generates two separate load cases for the same wind direction. They are distinguished by the indications *A* and *B*.

To display the respective explanatory graphics for illustrating the load parameters, use the button shown on the left. To reset the load graphic view, click the button again.

Load — Load Case LC51

No.	A Load Type	B Load Direction	C Load Reference	D Load Parameters p1 [kN/m]	E p2 [kN/m]	F A [m]	G B [m]	H Distance in %	I Across Total Length	J Comment
1	Trapezoidal Load	z	Left Span Side	-1.731	-1.731	0.000	2.000	☐	☐	Zone F; c_{pe} = -1.140
2	Trapezoidal Load	z	Left Span Side	-0.592	-0.592	2.000	12.676	☐	☐	Zone H; c_{pe} = -0.390
3	Trapezoidal Load	z	Right Span Side	-1.337	-1.337	0.000	2.000	☐	☐	Zone J; c_{pe} = -0.880
4	Trapezoidal Load	z	Right Span Side	-0.699	-0.699	2.000	12.676	☐	☐	Zone I; c_{pe} = -0.460
5										
6										
7										
8										

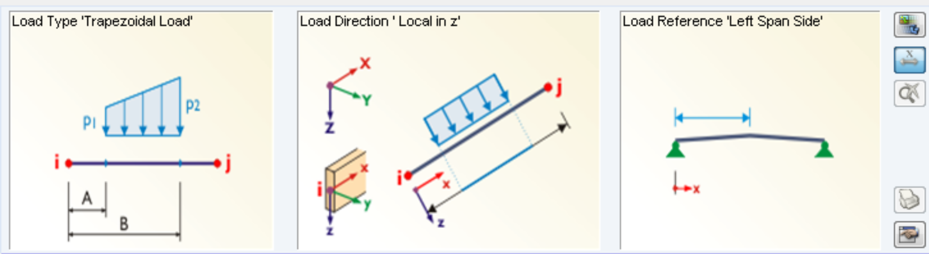
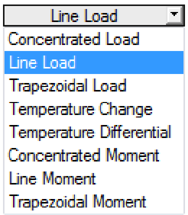


Figure 5.30: Dialog box *Load Cases*: explanatory graphics for load parameters

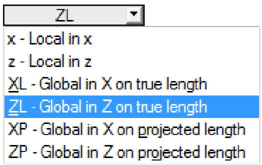
Table and graphic are interactive: When you select a load in the table, it is highlighted in the graphic. When you click a load in the graphic, the corresponding table row is marked.

Define additional loads

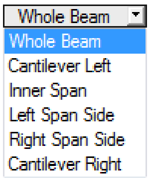
Additional loads can be entered directly into the next free table row. However, it is recommended to create a [New] load case to manage the cases separately. You have to define the descriptions, the load type and, if necessary, the service class category and the LDC.



In table column A of the dialog section *Load* (see Figure 5.30) you specify the *Load Type*. A list offers the load types shown on the left for selection. When the display with explanatory graphics is set, pictures appear in the bottom dialog section illustrating the individual **parameters** for load definition. Depending on the selected load type, other table columns become active or inactive in the table row.



When you define the *Load Direction*, three different settings are available: The loads can be defined in relation to the local member axes, in direction of the global axes on the effective length or in direction of the global axes on the projected length. The graphic in the bottom dialog section may help you to find the appropriate selection.



Further, you have to define the *Load Reference*, specifying the part of the beam where you want to apply the load. In order to simplify the input for structural beam parts, the following options are provided: whole beam, inner span, left/right span side, cantilever left/right. The *Cantilever* options are only available if an appropriate beam geometry has been set.

To arrange loads on any structural part of a beam, line loads must be defined as trapezoidal loads and line moments as trapezoidal moments. Only for these load types it is possible to define the start and end points of the load freely. It is recommended to always activate the graphic of loads so that you can check the input.

In table columns D to I, you can enter the loads according to the selected load type. For trapezoidal loads you can use the option *Across Total Length* to apply the load to the entire beam. Alternatively, you have to define the start and end points (*A* and *B*) of the load as well as the corresponding load values p_1 and p_2.

The button shown on the left allows for switching the load values on and off in the load graphic.

Particular graphic parts can be checked specifically by using the zoom function: Click into the graphic and then use the wheel button to minimize or maximize the graphic view.

The wheel button of the mouse can also be used for rotating the beam: Hold down the wheel button to move the structure anywhere in the graphic window. Now press the [Ctrl] key additionally to display the beam in the isometric view.

To reset the original graphic view, click the button [Show All Graphic] shown on the left.

When the input is complete, close the *Load Cases* dialog box by clicking the [OK] button to save the modifications. When you click [Cancel], you quit the dialog box but without saving the modifications. All modifications entered since the dialog box has been opened will be lost.

RF-COMBI

With the [RF-COMBI] button you can access the add-on module RF-COMBI that is used to combine load cases according to [2] or [9]. Usually, it is not required to modify data in RF-COMBI because the creation of combinations is performed automatically in the background also for user-defined loads. Therefore, you may use the button to understand the generation of combinations.

For more information on the features of RF-COMBI, see the manual available for download at www.dlubal.com.

5.4 Control Parameters

Program window 1.4 *Control Parameters* allows for numerous settings to control the calculation.

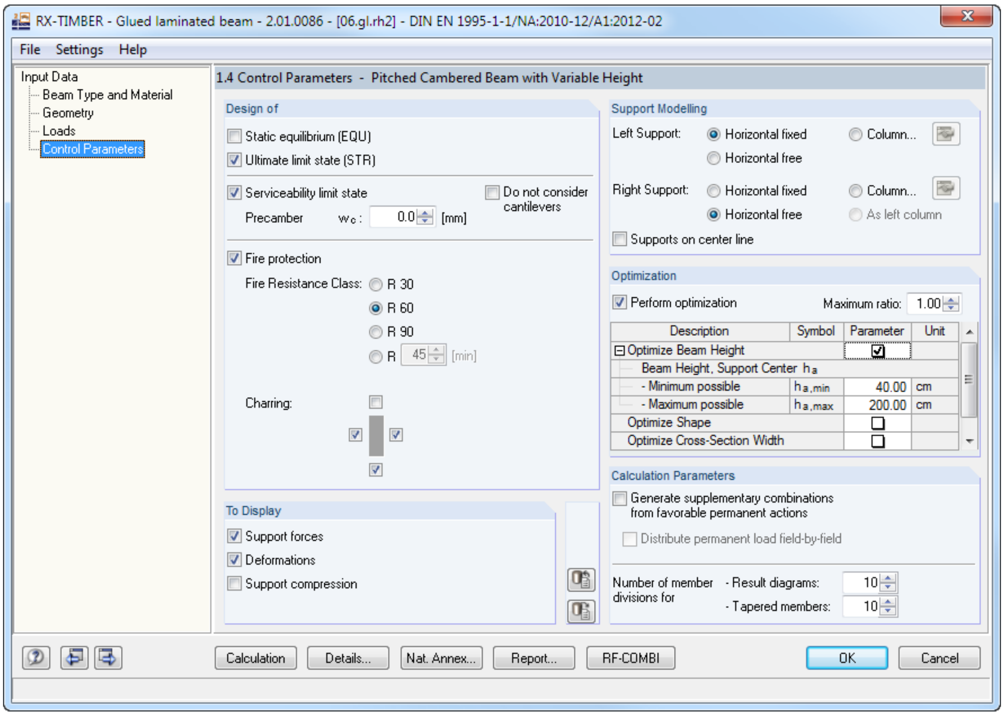


Figure 5.31: Window 1.4 *Control Parameters*

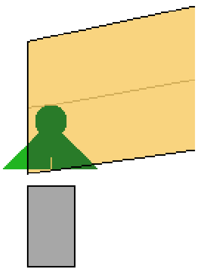
Design of

This window section sets the designs that are performed. The *Ultimate limit state* and the *Serviceability limit state* design are preset.

For the serviceability limit state design you can take into account an assumed *Precamber* of the beam by specifying a value for w_c or w_0. In addition, you have the option to *not consider cantilevers* for the analysis.

When you tick the option *Fire protection*, you can access the parameters for the fire protection design. To define the fire duration, select one of the fire resistance classes (R 30, R 60, R 90), or specify it by user-defined input. In the image below, you can define the beam sides for which you want to consider *Charring*.

Support modelling



Set the beam's support conditions. A successful calculation requires that the support is either defined as *Horizontal fixed* on at least one side or restrained by a bending-resistant column by selecting the *Column* option.

The supports are applied eccentrically by default. In this way, the wall-like structural behavior of beams which are often high and slender is taken into account. With the option *Supports on center line*, it is possible to shift the supports to the centroidal axis of the member (see picture in the margin) to reduce or eliminate the high boundary moments due to a compression force.

To specify the column parameters, use the [Edit] button shown on the left. The following dialog box opens.

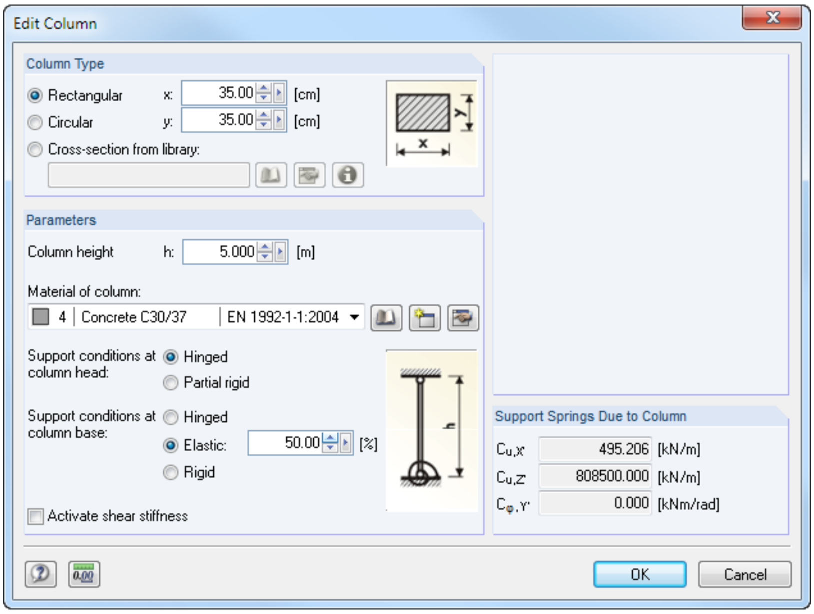


Figure 5.32: Dialog box *Edit Column* to determine support springs

In the dialog section *Column Type*, the options *Rectangular* and *Circular* are available for direct geometry input. The parameters are represented in the small graphic to the right. In case of special column cross-sections, use the option *Cross-section from library* that allows you access to a comprehensive cross-section database.

The settings for *Column height* and *Material* as well as *Support conditions* in the dialog section *Parameters* affect the support spring determination considerably. The *column base* setting offers three options whereas the setting for the *column head* allows only for a hinged connection or a partial restraint: The spring stiffness on the support is determined from the beam's point of view. As the column cannot provide a complete restraint, the program applies the rotational spring stiffness of the defined column for the option *Partial rigid*.

Based on the specified values, the program determines the spring stiffnesses in direction X and Z that will be used later as support conditions for the calculation. The dialog section *Support Springs Due to Column* shows the calculated spring values.

If you select the option *As left column* in window 1.4 (Figure 5.31), the program defines the column also for the right support. The values of the left column are preset in the dialog box *Edit Column* and can be adjusted, if necessary.

Optimization

With the settings in this window section you decide if and how the program *Performs* an *optimization* for the beam cross-section. The cross-section can be optimized by different criteria: Corresponding to the beam type you can adjust the *Beam Height*, the *Shape* or the *Cross-Section Width*.

The target of the optimization is preset with the *Maximum ratio* of 100 %. If necessary, you can specify another upper limit in this input field.

When optimizing the *Beam Height* you need to define the upper and lower limit for the height in the support center. When optimizing the *Shape* several boundary conditions can be defined (depth of beam in support center and on ridge, radius of curvature, inclination of lower chord etc.). The fewer parameters are set, the faster the beam will be optimized.

The *Cross-Section Width* can be optimized within user-defined limit values. The lamella thickness selected in window 1.2 is preset as *Increment* that you can adjust, if necessary. The procedure varies depending on the beam type.

The *Spacing of Lateral Supports* is important for the lateral buckling design. For beams prone to instability risks you can optimize the spacing within particular limits and a user-defined increment to fulfill the requirements of the design.

To display

By ticking the check boxes you decide if *Support forces*, *Deformations* and *Support compression* will appear in separate results windows. When designing the beam according to DIN 1052: 2008-12, an *Oscillation design* is possible.

The number of results windows appearing in the output depends on the design and output data that you select in this window section.

Calculation parameters

Settings in this window section affect the number of generated result combinations as well as the calculation accuracy.

With the option *Generate supplementary combinations from favorable permanent actions* the program is able to distinguish favorable and unfavorable permanent actions and to take them into account with different partial safety factors. Necessarily, the number of generated combinations will be increased in the output.

The *Number of member divisions for Result diagrams* determines how much x-locations are set on the longest member in the system. This division length will be valid also for the x-locations of all other members in the model. The x-locations represent the locations on the beam where designs are performed. The preset division value *10* for result diagrams has proven to be a good compromise between computational accuracy and speed.

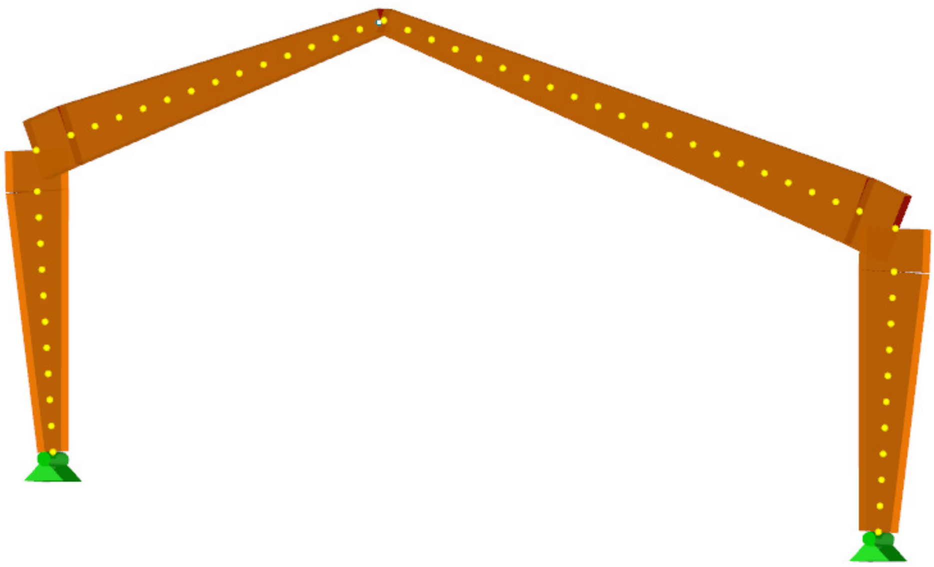


Figure 5.33: Principle of member divisions for result diagrams

To get a sufficient set of values for the calculation of *Tapered members* which are usually relatively short, it is possible to define the internal division for tapers specifically. The division value is also preset with *10*. This setting does not result in an increased number of output results but only in a reduction of the division length for the internal calculation of the internal forces.

6. Calculation

To start the calculation, use the [Calculation] button that is available in each input window. The designs are performed with the result combinations generated in RF-COMBI. Before you start the calculation, it is recommended to check the settings concerning calculation details and selected standard.

6.1 Calculation Details

To check important design parameters, use the [Details] button to open the dialog box below that can be accessed from any window in the program. The dialog box is aligned with the standard or the selected national annex. Figure 6.1 shows the *Details* dialog box for EN 1995-1-1 with the national annex for Germany.

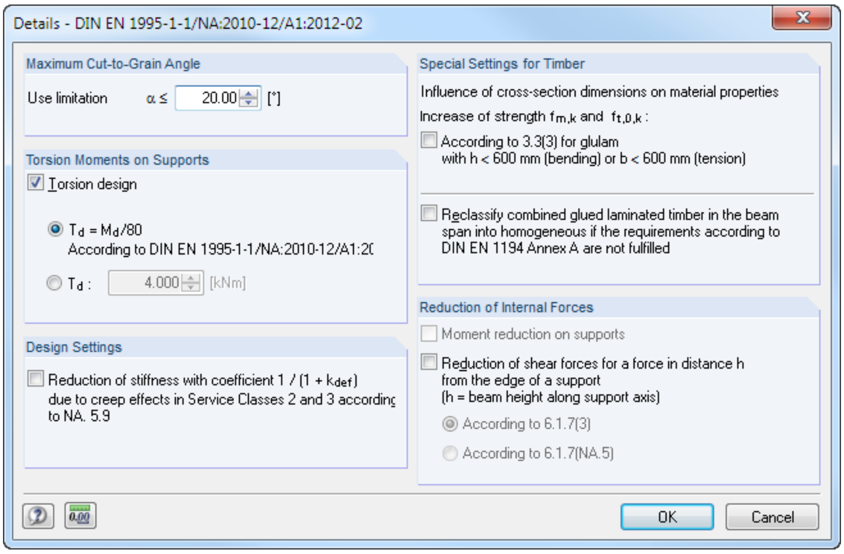


Figure 6.1: Dialog box *Details* according to DIN EN 1995-1-1 for RX-TIMBER Glued-Laminated Beam

Maximum Cut-to-Grain Angle

EN 1995-1-1 specifies no limitation for the cut-to-grain angle. The German standard DIN 1052, however, restricts the angle to 10° as the standard's equations are only valid for angles of up to 10°. In this dialog section, you can define the limit value yourself. The presetting in RX-TIMBER is 20°.

Torsion moments on supports

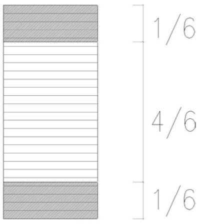
The torsion design on the supports according to [7], NAD for Germany, has been considerably eased: The design is to be performed only if the slenderness is larger than 225. RX-TIMBER checks this condition automatically.

For the design according to DIN 1052 it is additionally possible to apply the lateral loads of the trusses as stabilizing loads (see explanations for DIN 1052).

When the torsion moment T_d has been determined manually, it can be entered directly. The moment will be applied as cosine function descending from the supports.

Special settings for timber

When compact beams are used (height h < 600 mm), many design standards allow you to increase the strength values of the material. In case of glued laminated timber it is assumed that the material's weakening is not that strong due to the finger joint.



When the check box for *Reclassify combined glued laminated timber in the beam span into homogeneous* is ticked, the program checks for each design location if the conditions for combined glued laminated timber are geometrically fulfilled. If the higher strength of 1/6 in both boundary areas at top and bottom is not given, RX-TIMBER will automatically select the next lower strength grade and display a corresponding message. Then, you can either select a higher material grade in the critical areas or perform the design with reduced strengths.

Reduction of internal forces

Use the options in this dialog section to reduce moments and shear forces in the support area. The reduction of the supporting moment by a *Moment reduction* is only possible if your beam is a multi-span or a cantilevered beam.

As EN 1995-1-1 allows only for the reduction of concentrated loads in the support area, the option for reducing shear forces is not displayed for distributed loads. The request for a complete reduction of the concentrated loads cannot be answered by RX-TIMBER because a separate calculation of internal forces would be required.

The following figure shows the *Details* dialog box of the program RX-TIMBER Continuous Beam for the design according to DIN 1052.

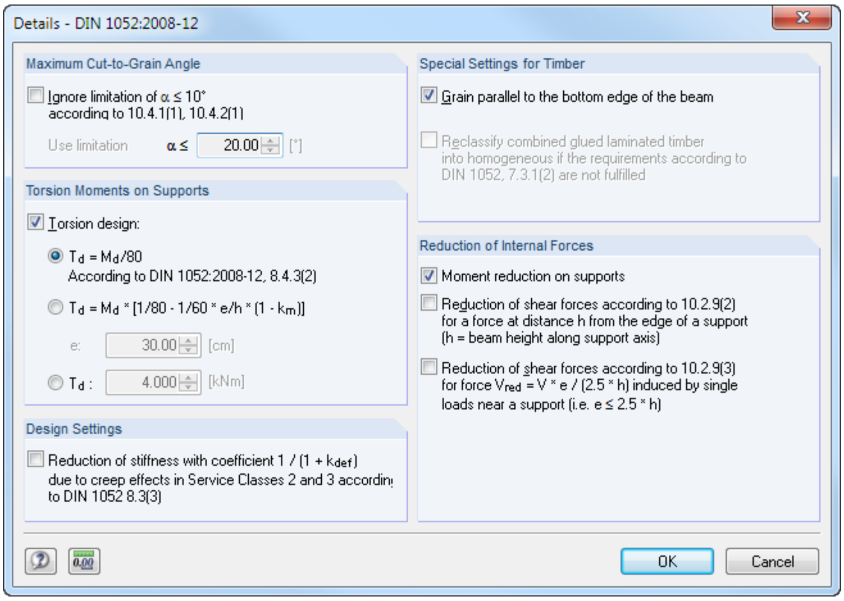


Figure 6.2: Dialog box *Details* according to DIN 1052 for RX-TIMBER Continuous Beam

In the dialog section *Special Settings for Timber* you can decide to define the *Grain parallel to the bottom edge of the beam*. The remaining setting options refer to specifications in the German standard DIN 1052. They correspond to the options represented in Figure 6.1.

6.2 Standard and National Annex

The button [National Annex] shown on the left provides access to significant design parameters such as the partial safety and modification factors as well as the limit values of the deformations. The dialog box is aligned with the standard or the selected national annex. The following figure shows the parameters for EN 1995-1-1 with the national annex for Germany.

The dialog box *National Annex Settings* consists of three tabs.

6.2.1 General Parameters

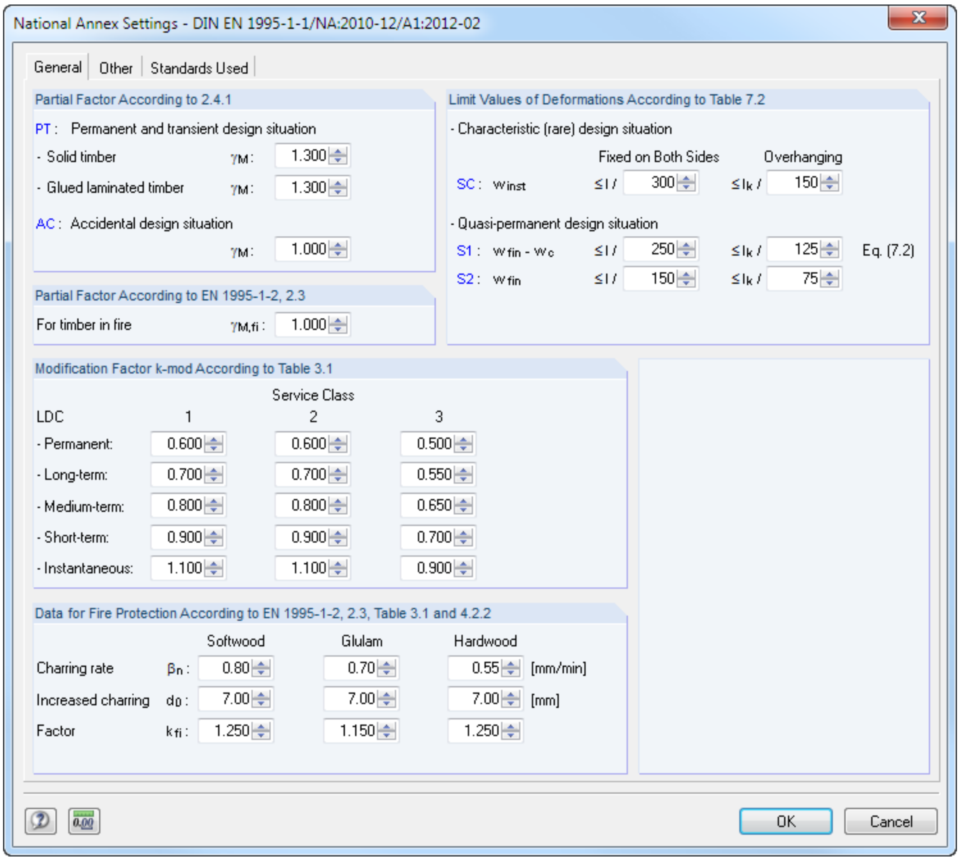


Figure 6.3: Dialog box *National Annex Settings*, tab *General*

Partial factor

In this dialog section you can check and, if necessary, adjust the partial safety factors of the material stiffnesses for the different design situations. The factors are preset according to the specifications of the standard selected in program window 1.1 *Beam Type and Material*.

Limit values of deformations

The deformation's limit values can be adjusted individually for the serviceability limit state design so that it is possible to perform the design according to specifications of different guidelines.

The allowable deflections can be defined separately for the individual design situations and boundary conditions (span, cantilever).

Modification factor k_{mod}

To take into account the moisture-dependent long-term effects of timber, it is possible to adjust the modification factors. They depend on the service and the load-duration class. The calculation values of the modification factors are specified in [1] table F.1 and [7] table 3.1.

Data for fire protection

The designs are carried out by the simplified method according to [6] clause 5.5.2.1 a) or [8] clause 2.3, 3.4.2 and 4.2.2. The preset values for β_n, d_0 and k_{fi} are valid for glued laminated timber. Thus, it is usually not necessary to modify them.

6.2.2 Other Parameters

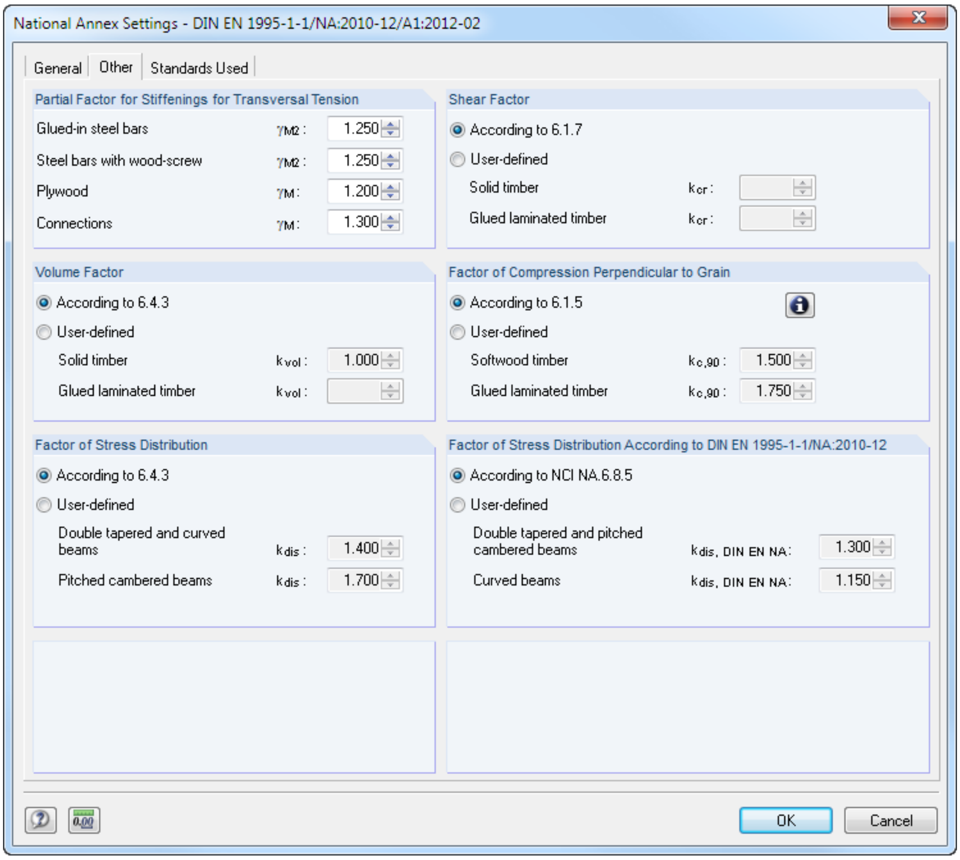


Figure 6.4: Dialog box *National Annex Settings*, tab *Other*

Partial factor for stiffenings for transversal tension

This dialog section controls the partial safety factors that must be taken into account for the different types of transversal tension stiffenings (see chapter 5.1, page 32).

The program presets the partial safety factors for material properties and load capacity recommended in the standard (for example EN 1995-1-1, table 2.3).

Shear factor

For shearing stresses along and perpendicular to the grain direction you must observe specific conditions according to EN 1995-1-1, clause 6.1.7.

The factor k_{cr} can be *User-defined* for solid and glued-laminated timber. Many countries still do not provide uniform material standards managing this stiffness.

Volume factor

When designing glued-laminated beams you can adjust the volume factor k_{vol} for solid and glued-laminated timber. You find the relevant standard specifications in EN 1995-1-1, clause 6.4.3 (6).

Factor of compression perpendicular to grain

For the design of the support compression the compression strength is adjusted perpendicular to the grain direction by means of the factor $k_{c,90}$ (see EN 1995-1-1, clause 6.1.5). With the default setting according to standard specifications RX-TIMBER increases the strength corresponding to the material type and the load situation. However, as it is usual in some countries to calculate generally with a strength increase of 75 %, the factor can be user-defined.

Click the [Info] button shown on the left to get more information on the factor determination.

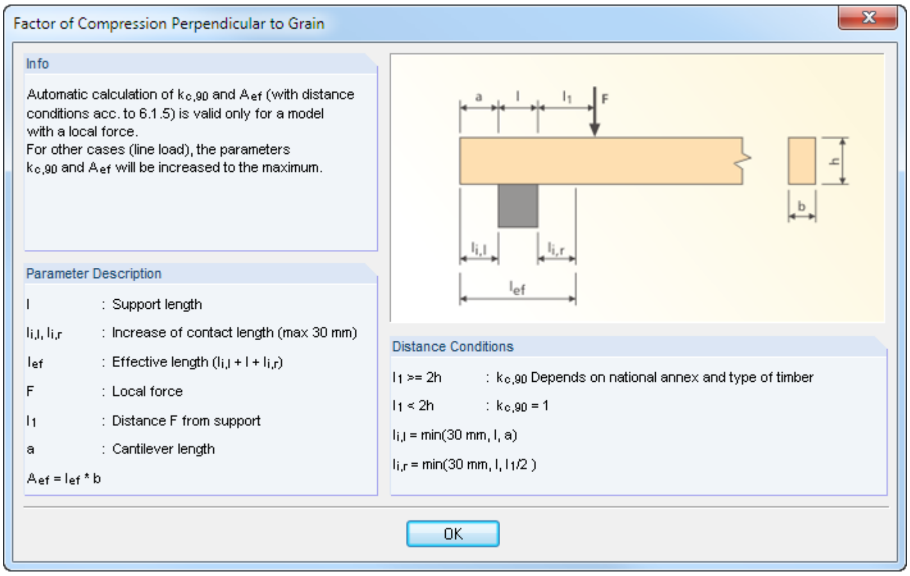


Figure 6.5: Dialog box *Factor of Compression Perpendicular to Grain*

Factor of stress distribution

For glued-laminated beams you can adjust the factor k_{dis} used to consider the stress distribution in the ridge zone. You find the relevant standard specifications in EN 1995-1-1, clause 6.4.3 (6) as well as in the national annexes.

6.2.3 Standards Used

The final tab lists all rules and standards that are relevant for the generation of loads and the design according to the selected standard.

6.3 RF-COMBI

The add-on module RF-COMBI is integrated in RX-TIMBER. It generates all result combinations automatically. Use the [RF-COMBI] button to start the add-on module.

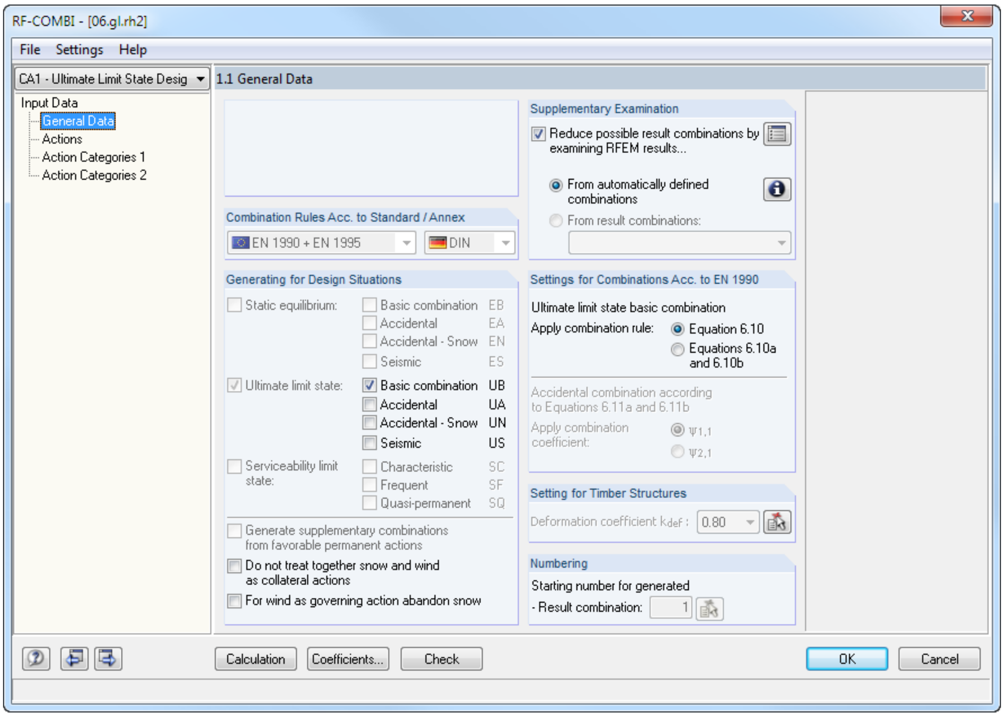


Figure 6.6: RF-COMBI window 1.1 *General Data*

To avoid generating an unnecessary high number of result combinations, it is recommended to *Reduce possible result combinations* in the window section *Supplementary Examination* of window 1.1 *General Data*. In this way, you make sure that the results of RF-COMBI include all governing result combinations but without generating more load cases than necessary.

Find more information about the add-on module RF-COMBI in the program manual available for download at www.dlubal.com.

After the [Calculation] you can check the results of the combinatorics: The generated result combinations are presented in three results windows sorted by different criteria.

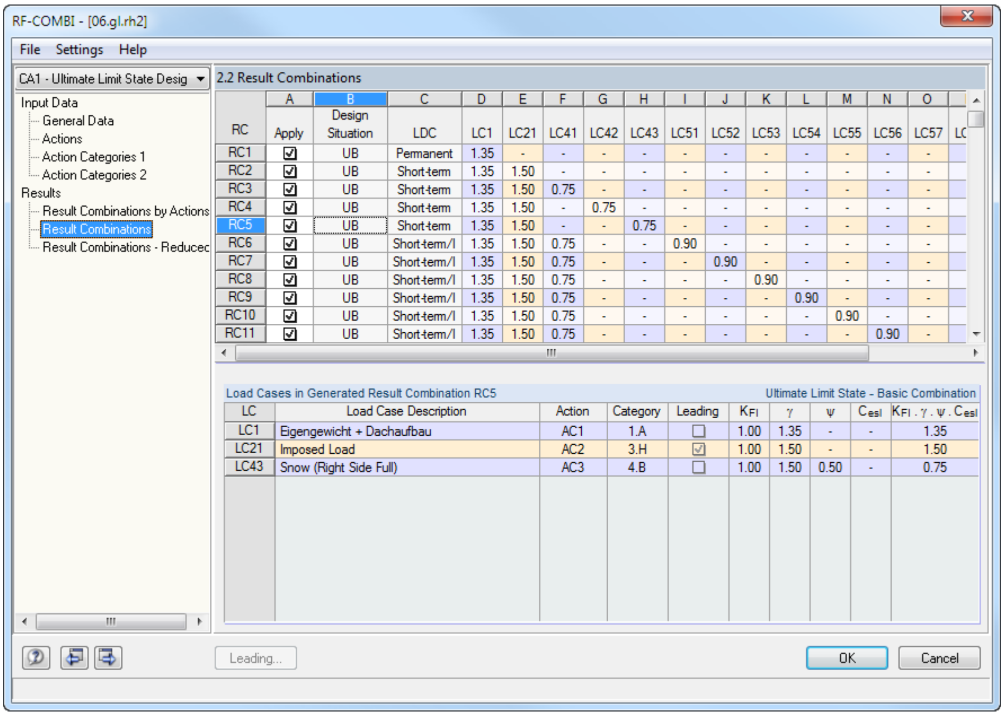


Figure 6.7: RF-COMBI window 2.2 *Result Combinations*

Click [OK] to return to the program RX-TIMBER.

6.4 Calculation

To start the calculation, click the [Calculation] button that is available in all input windows of RX-TIMBER. During the calculation the *FE-Solver* window is displayed, showing information about the calculation process.

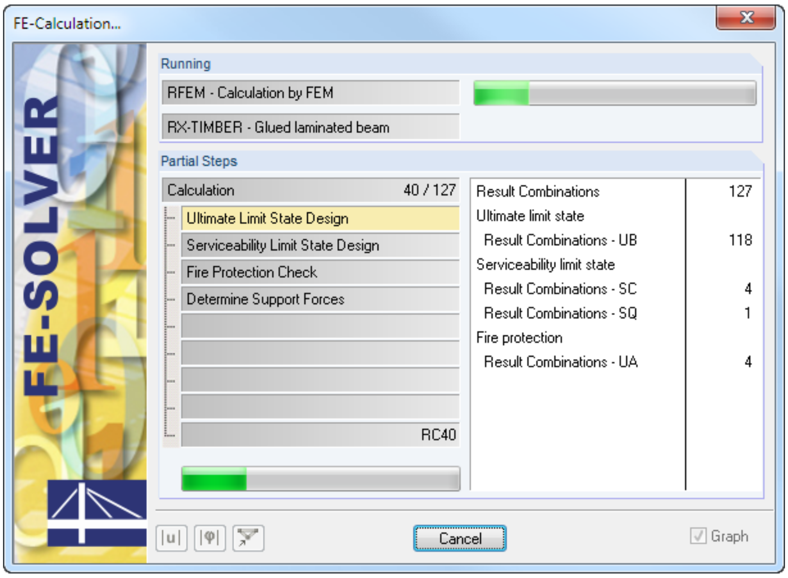
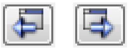


Figure 6.8: Window *FE-Calculation*

The results window 2.1 *Result Combinations* appears immediately after the calculation (see Figure 7.1).

7. Results

The designs are shown in the results windows 2.1 to 2.3 and sorted by different criteria. Windows 2.4 and 2.5 lists the support forces and deformations.



You can select every window by clicking the corresponding entry in the navigator. You can also use the buttons shown on the left to set the previous or next window. Alternatively, you can press the function keys [F2] and [F3].

To save the results and exit RX-TIMBER, click [OK].

Chapter 7 *Results* presents the results windows in sequence.

7.1 Result Combinations

This window shows the maximum design ratios for each single result combination. Table column *RC Description* lists the combinations with the short descriptions of the contained load cases (see page 48: Short description). Column B *Load Cases* displays the combination criterion with load case number and considered factors from RF-COMBI.

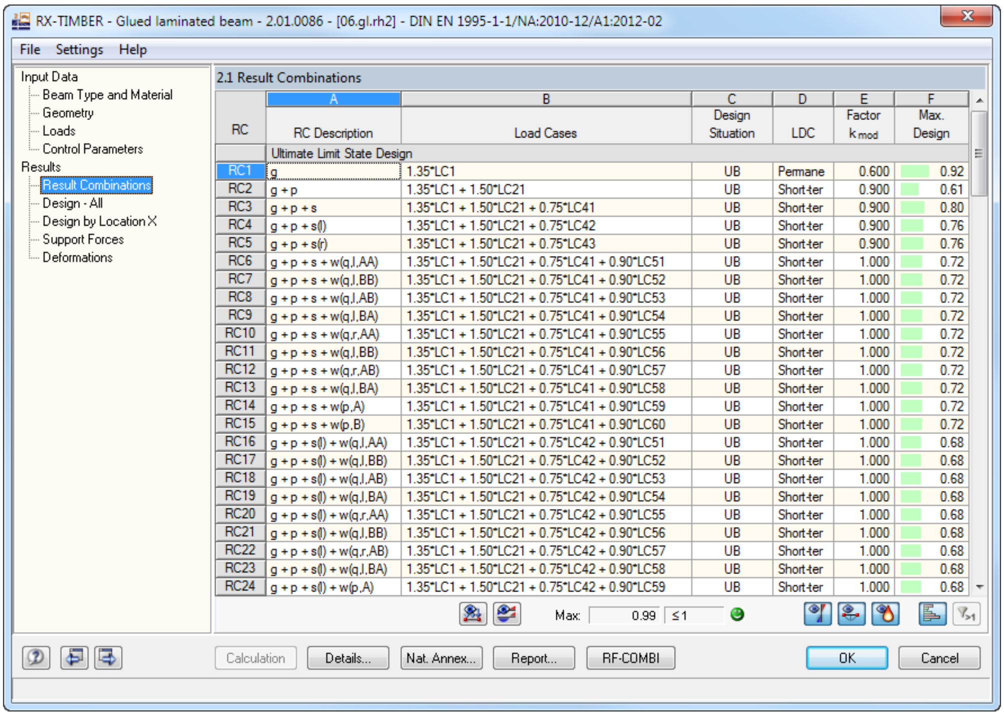


Figure 7.1: Window 2.1 *Result Combinations*

Table column C *Design Situation* informs you about the design situation to which the combination belongs. RX-TIMBER distinguishes the following situations:

UB	Fundamental combination for bearing capacity
UA	Accidental combination for bearing capacity
S1	Basic combination 1 for serviceability
S2	Basic combination 2 for serviceability
SQ	Quasi-permanent combination for serviceability

Table 7.1: Design situations in RX-TIMBER

Table columns D *LDC* and E *Factor* k_{mod} list the load-duration classes and the modification factors.

The final column F *Max. Design* indicates the maximum design ratio for each result combination. The ratios are represented by colored scales: For designs ≤ 1 a green scale is displayed, for designs > 1 a red scale appears. The color scales can be deactivated, if necessary, by means of the button shown on the left.

Filter functions

The results are arranged in blocks: The ultimate limit state design is displayed first, followed by the serviceability limit state design and the fire protection check. Below the table you find three buttons that can be used for filtering the design results in the list.

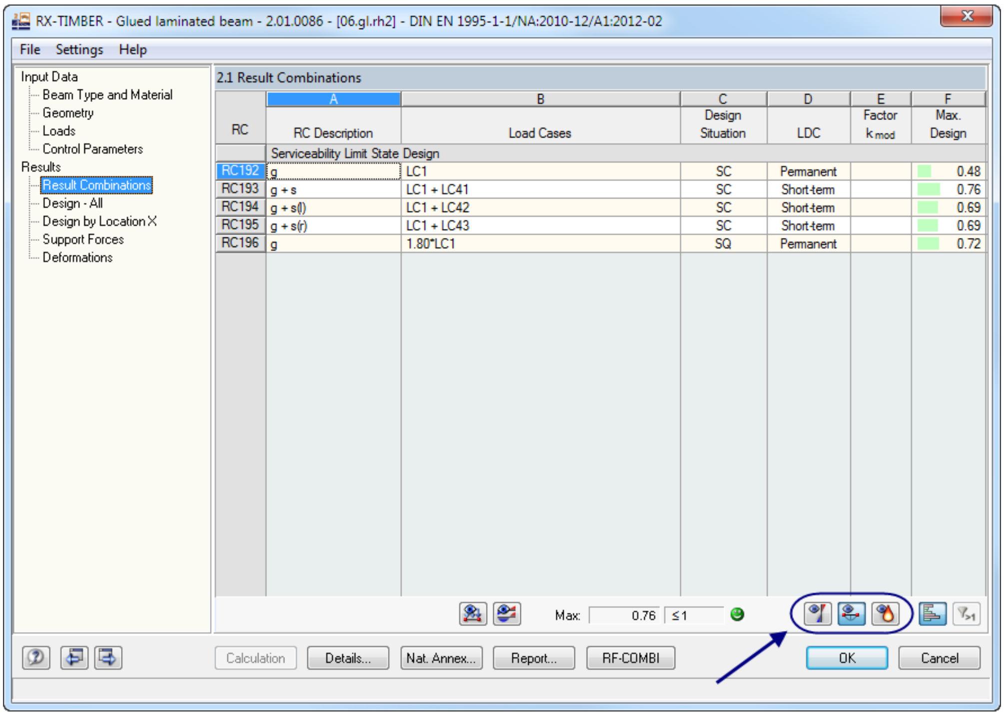


Figure 7.2: Window 2.1 *Result Combinations* with filter set only for serviceability limit state design

The buttons have the following functions:

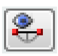
Button	**Function**
	Hides and displays the ultimate limit state design
	Hides and displays the serviceability limit state design
	Hides and displays the fire protection design

Table 7.2: Buttons for designs

If designs with a ratio of > 1 are available, an additional filter function is enabled: Click the button shown on the left to hide all designs ≤ 1. In this way, you can check all failed designs at a glance.

7.2 Design - All

The second results window presents an overview about all performed designs. The upper part of the window shows a table with the maximum design ratio for each type of design. This overview allows you to quickly evaluate the maximum design ratios for the individual designs (shear stress, bending stress, lateral buckling analysis, support pressure etc.).

The lower part of the window displays the *Intermediate results* for the load case set above. They provide detailed information about cross-section and design internal forces as well as design parameters including notes for tables and equations of the standard.

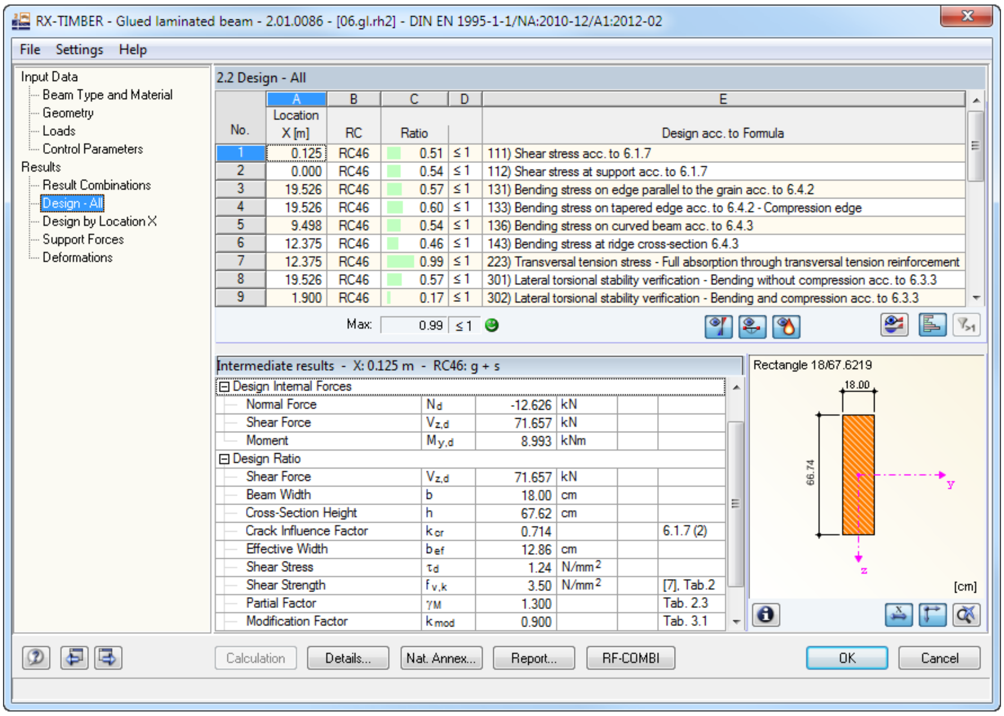


Figure 7.3: Window 2.2 *Design - All*

Location X

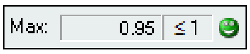
The first table column shows the respective X-location on the member for which the program has determined the maximum design criterion. The following member locations *X* are taken into account for the table output:

- Start and end node
- Division points according to member divisions defined in window 1.4 *Control Parameters* (see chapter 5.4, page 53)
- Extreme values of internal forces

RC

Table column B displays the numbers of the result combinations whose internal forces result in the respective maximum ratios.

Ratio



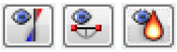
The program shows for each design type the design ratios according to the selected standard. The colored scales represent the corresponding ratio.

Design according to formula

The final table column lists the equations according to EN 1995-1-1 (or DIN 1052) that were used to carry out the design.

7.3 Design by X-Location

The structure of the results window is similar to the one of window 2.2 described in the previous chapter. The table lists in detail the results for each analyzed *Location X* on the beam.



The results are summarized in groups sorted by bearing capacity, serviceability and fire protection. Use the buttons shown on the left to filter the design types (see chapter 7.1, page 62).

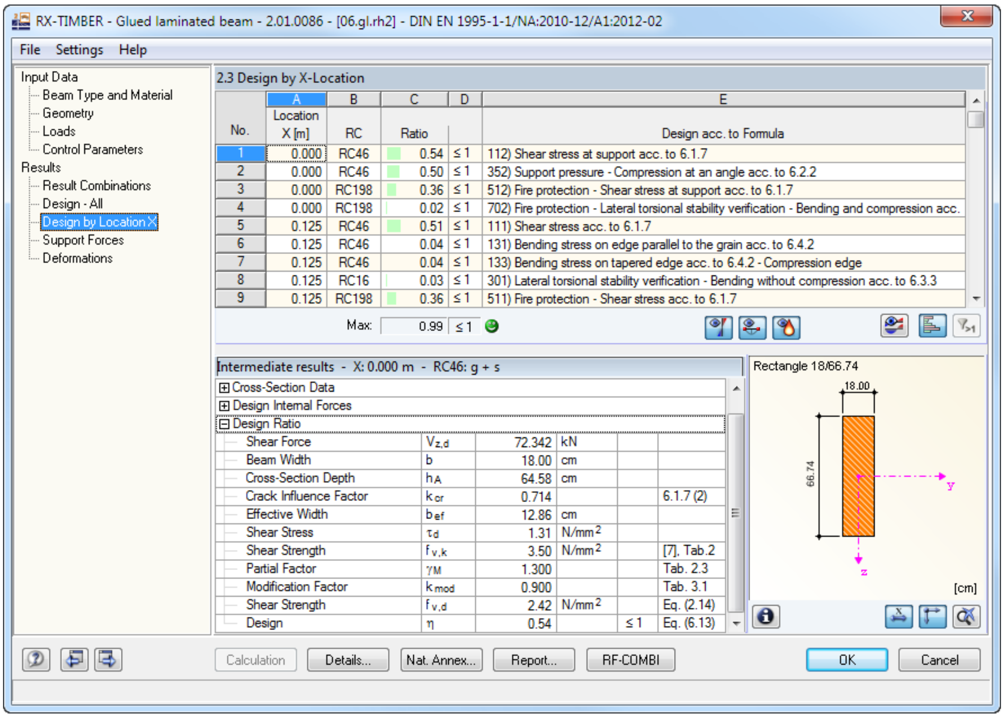


Figure 7.4: Window 2.3 *Design by X-Location*

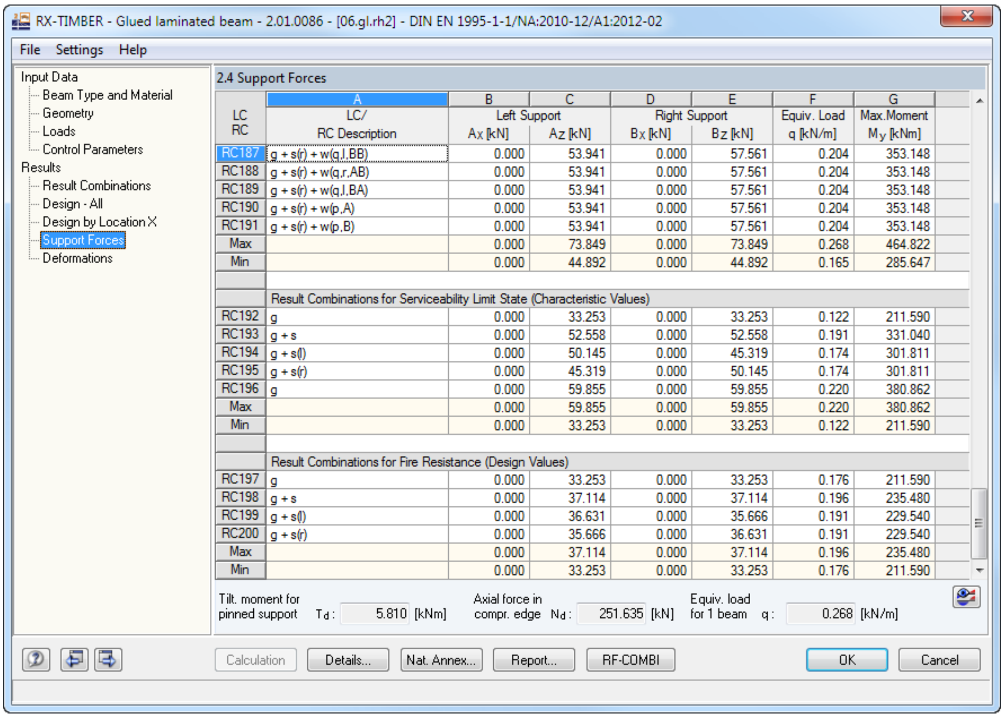
Due to the variety of loads it is not possible to clearly identify the decisive beam location immediately. Therefore, the beam is analyzed at different design locations *X*:

- Start and end node
- Division points according to member divisions defined in window 1.4 *Control Parameters* (see chapter 5.4, page 53)
- Extreme values of internal forces

The results windows generally display the *Locations X*. They are related to the global coordinate system and represent the length X projected to the floor plan. In contrast, the result diagrams (see chapter 7.6, page 67) refer to the local member axis x where the length x of the inclined beam is indicated.

7.4 Support Forces

The window lists the support forces of the individual load cases and result combinations. The extreme values of the result combinations are summarized in groups sorted by bearing capacity, serviceability and fire protection.



LC RC	A: LC/ RC Description	B: Left Support Ax [kN]	C: Left Support Az [kN]	D: Right Support Bx [kN]	E: Right Support Bz [kN]	F: Equiv. Load q [kN/m]	G: Max.Moment My [kNm]
RC187	g + s(r) + w(q,l,BB)	0.000	53.941	0.000	57.561	0.204	353.148
RC188	g + s(r) + w(q,r,AB)	0.000	53.941	0.000	57.561	0.204	353.148
RC189	g + s(r) + w(q,l,BA)	0.000	53.941	0.000	57.561	0.204	353.148
RC190	g + s(r) + w(p,A)	0.000	53.941	0.000	57.561	0.204	353.148
RC191	g + s(r) + w(p,B)	0.000	53.941	0.000	57.561	0.204	353.148
Max		0.000	73.849	0.000	73.849	0.268	464.822
Min		0.000	44.892	0.000	44.892	0.165	285.647
	Result Combinations for Serviceability Limit State (Characteristic Values)						
RC192	g	0.000	33.253	0.000	33.253	0.122	211.590
RC193	g + s	0.000	52.558	0.000	52.558	0.191	331.040
RC194	g + s(l)	0.000	50.145	0.000	45.319	0.174	301.811
RC195	g + s(r)	0.000	45.319	0.000	50.145	0.174	301.811
RC196	g	0.000	59.855	0.000	59.855	0.220	380.862
Max		0.000	59.855	0.000	59.855	0.220	380.862
Min		0.000	33.253	0.000	33.253	0.122	211.590
	Result Combinations for Fire Resistance (Design Values)						
RC197	g	0.000	33.253	0.000	33.253	0.176	211.590
RC198	g + s	0.000	37.114	0.000	37.114	0.196	235.480
RC199	g + s(l)	0.000	36.631	0.000	35.666	0.191	229.540
RC200	g + s(r)	0.000	35.666	0.000	36.631	0.191	229.540
Max		0.000	37.114	0.000	37.114	0.196	235.480
Min		0.000	33.253	0.000	33.253	0.176	211.590

Figure 7.5: Window 2.4 *Support Forces*

Support forces for load cases are displayed as characteristic loads. Thus, it is possible to use them for other calculations. Support forces for result combinations are displayed considering the design loads.

The following values are additionally displayed below the table:

- T_d *Tilting moment for pinned support* according to [1] eq. (14) or [7] (NA.129)
- N_d *Axial force in compression edge* according to [1] eq. (15) or [7] eq. (9.36)
- q *Equivalent load for 1 beam* according to [1] eq. (16)) or [7] eq. (9.37)

However, the indicated values are not taken into account for the design.

7.5 Deformations

The final results window lists the maximum deformations for the individual load cases and result combinations.

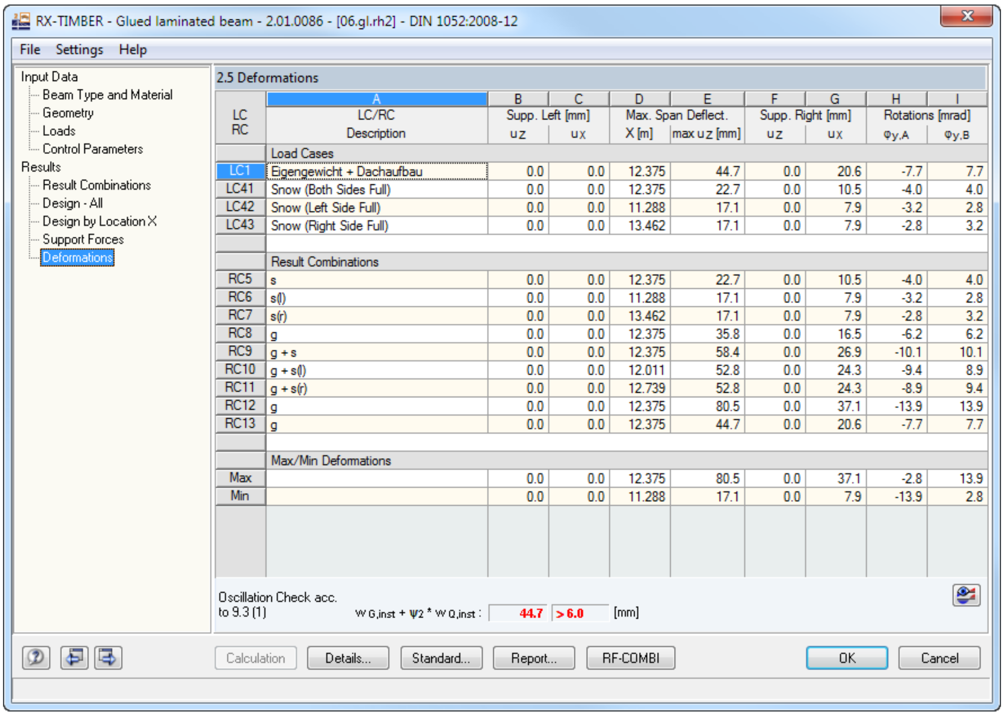


Figure 7.6: Window 2.5 *Deformations* with oscillation check according to DIN 1052

The window shows the displacements on each *Support* u_z and u_x as well as the *Maximum Span Deflection max* u_z at the governing X-locations.

If the model has cantilevers, the table is extended by the *Cantilever* column showing the displacements $u_{Z,c}$.

The right part of the table provides information about the *Rotations φ* on the supports and, if available, the cantilevers.

When the oscillation design according to [1] paragraph 9.3 (1) was selected in window 1.4 (see chapter 5.4, page 53), you can find the design result listed below the table in window 2.5.

7.6 Result Diagrams

The distribution of results on a beam can be evaluated graphically in the result diagram. To access the diagram, use the button shown on the left which is available in each results window.

A new window opens displaying a navigator and the beam's result diagrams.

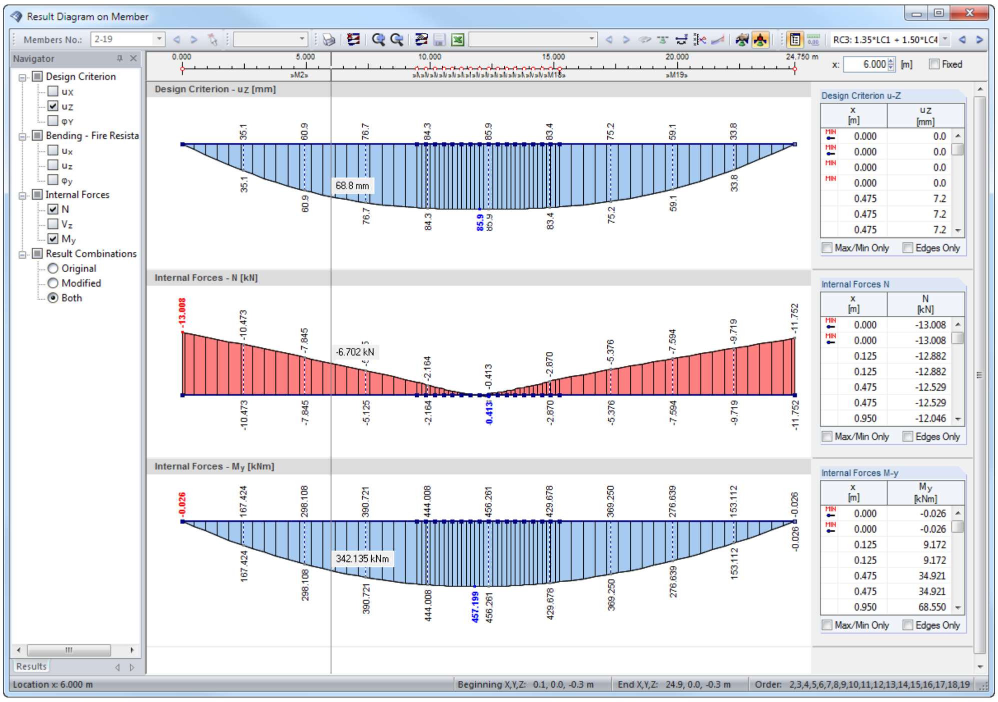
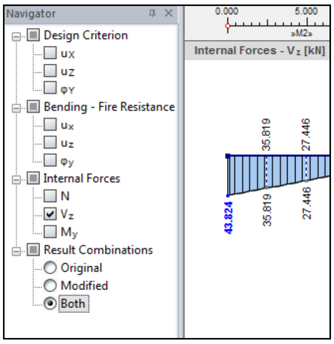


Figure 7.7: Dialog box *Result Diagram on Member*



Use the navigator on the left to select the deformations and internal forces that you want to display in the result diagram. Use the list in the toolbar to choose a particular load case, result combination or RX-TIMBER design case.

A dotted line in the result diagram of the RX-TIMBER design indicates the design ratio of 1.0 (see Figure 8.12, page 75). In this way, you can evaluate the significance of the results quickly.

With the navigator entry *Result Combinations* you have the possibility to display *Original* or *Modified* internal forces that are used for the design because of a moment or shear force reduction. It is also possible to display *Both* internal forces.

The results windows of RX-TIMBER show the global X-locations where the designs are performed, but the result diagrams refer to the local member axis x. However, the relation between global and local locations is ensured: When you have selected a (global) location X in a table and you open the result diagram, the position line jumps automatically to the corresponding (local) location x in the diagram.

The result diagrams can be either printed directly by using the print function or transferred into the printout report (see chapter 8.4, page 75).

For more detailed information about the dialog box *Result Diagram on Member*, see chapter 9.5 of the RFEM manual that is available for download at www.dlubal.com

8. Printout

The input and output data of RX-TIMBER is not automatically sent to a 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.

A printout report can only be called up if a default printer has been installed in Windows. The preview in the printout report uses that printer driver.

8.1 Printout Report

To open the printout report, use the button shown on the left.

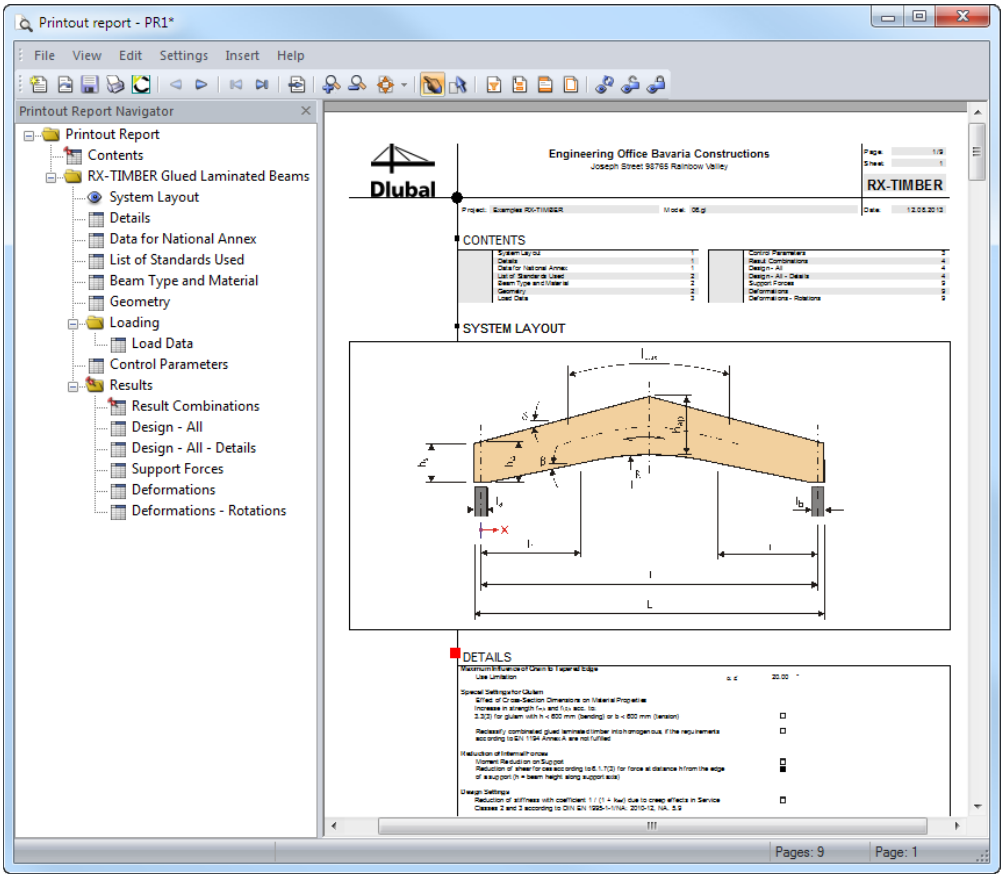


Figure 8.1: Printout report

If required, you can rename the printout report. To open the corresponding dialog box,

select **Rename** on the **File** menu of the printout report.

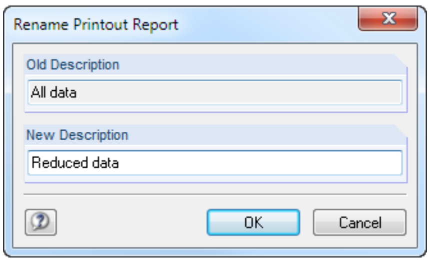


Figure 8.2: Dialog box *Rename Printout Report*

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.

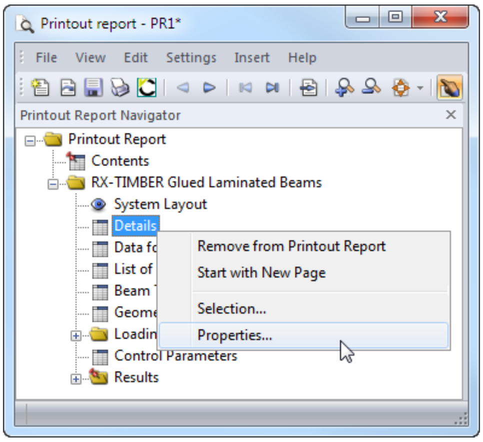
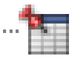


Figure 8.3: Context menu of report navigator

Remove from printout report

The selected chapter is 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 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.

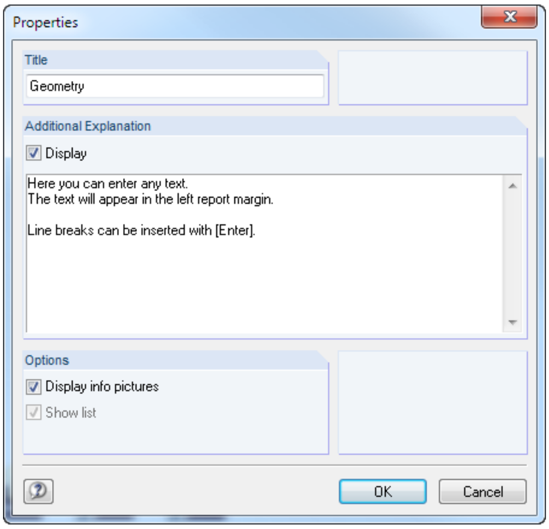
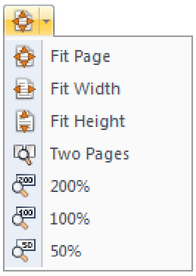


Figure 8.4: Dialog box *Properties*

Navigation in the printout report

To look at a particular paragraph in the printout report, click the corresponding chapter entry in the navigator.

The **Zoom** entry on the **View** menu as well as the **Edit** menu offer more functions for navigation. You can also use the buttons in the report toolbar to access the corresponding function.



	Go to previous page in the page preview
	Go to next page
	Go to first page in the page preview
	Go to last page
	Specify the number of a particular page in a dialog box.
	Zoom in
	Zoom out
	List button for *Zoom* to adjust display size
	Grab mode: Use the mouse for navigation within the report.
	Selection mode: Select and edit chapters by mouse click.

Table 8.1: Navigation buttons in the toolbar

8.2 Selection of Printing Data

In the global selection, you can select the chapters that you want to appear in the printout report. To access the corresponding function,

select **Selection** on the **Edit** menu of the printout report

or use the toolbar button shown on the left. You can also use the context menu of the navigator item *Contents*.

The following dialog box appears:

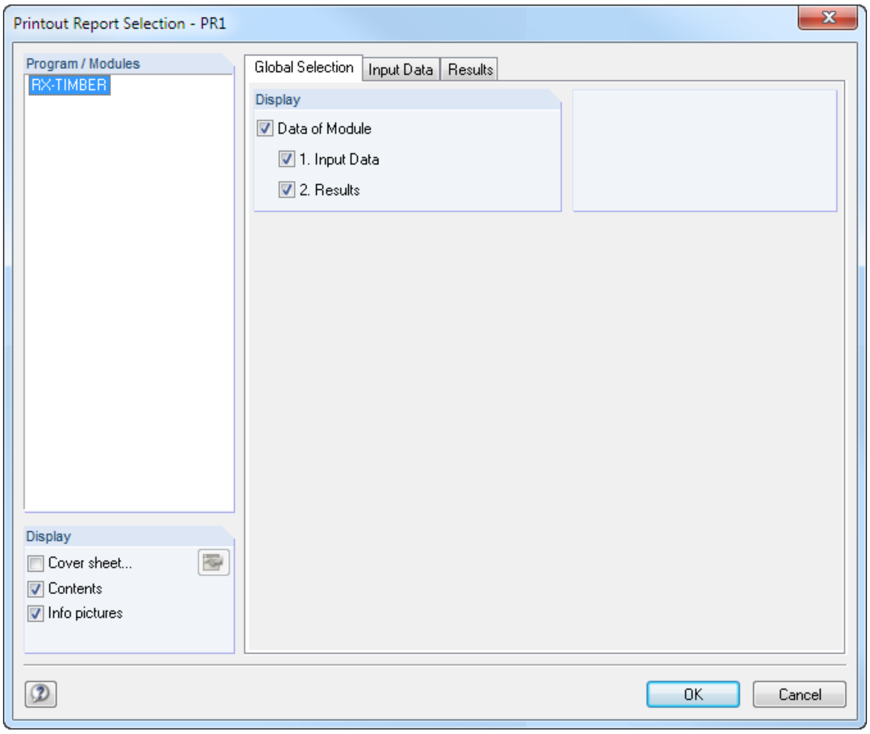


Figure 8.5: Dialog box *Printout Report Selection*, tab *Global Selection*

Global selection

The *Global Selection* tab manages the main chapters of the report. If you clear a check box, the corresponding detail tab disappears as well.

The *Display* section in the left corner of the dialog box provides three check boxes. Tick the check boxes to include a *Cover sheet*, the *Contents* or small *Info pictures* in the report margin.

Input data / results

With the *Input Data* and *Results* tabs you can set the printout's data amount. The designs performed for a beam may vary from a few pages to an amount of more than 150 pages. Therefore, a useful selection of output data is important. The design details are often very extensive, especially for the results sorted by X-location.

The data in the printout reflects the data arrangement in the program's in- and output.

Each selection is documented in the report navigator where it can also be removed.

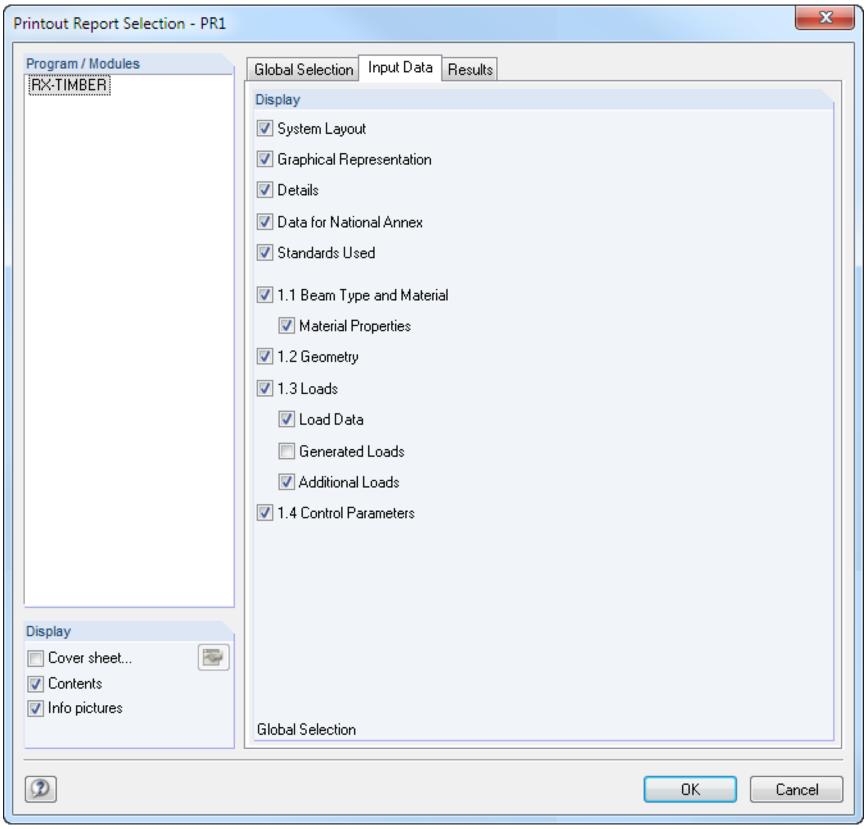


Figure 8.6: Dialog box *Printout Report Selection*, tab *Input Data*

Similar to the filter function of the results windows, the *Results* tab provides a *Filter* function for the ultimate limit state, the serviceability limit state and the fire protection design.

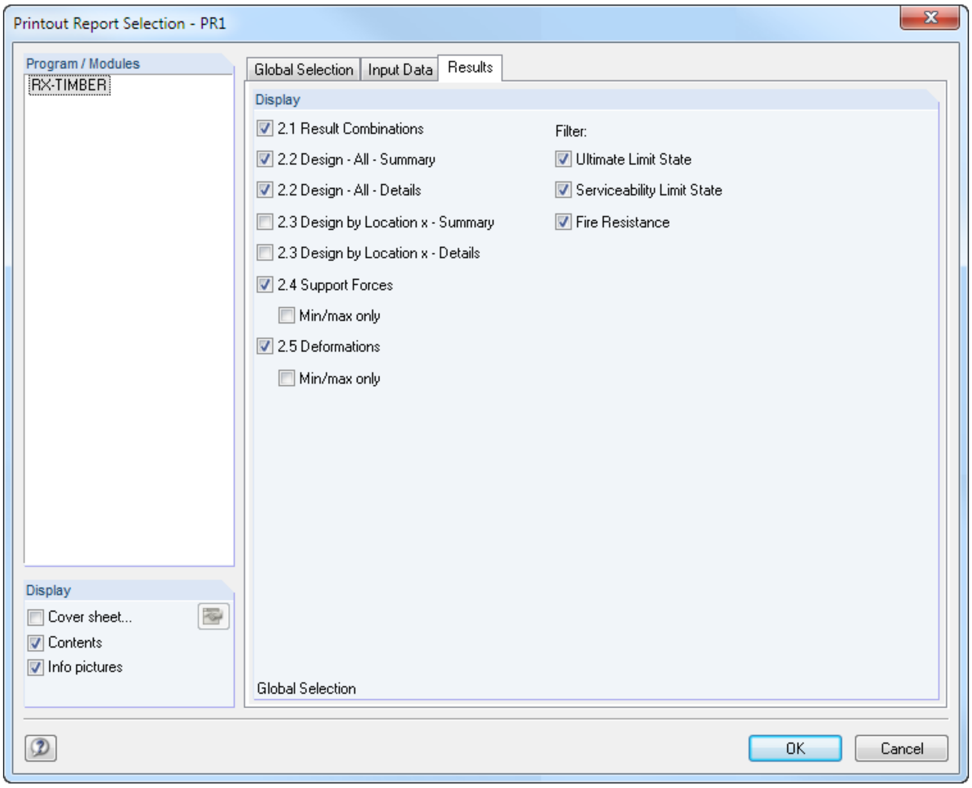


Figure 8.7: Dialog box *Printout Report Selection*, tab *Results*

8.3 Printout Report Header

A printout report header is created based on customer data during the program installation. To change this data,

select **Header** on the **Settings** menu of the printout report

or use the toolbar button shown on the left.

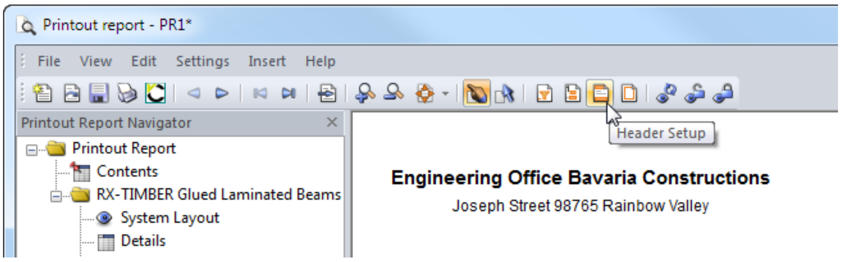


Figure 8.8: Button *Header Setup*

The following dialog box appears:

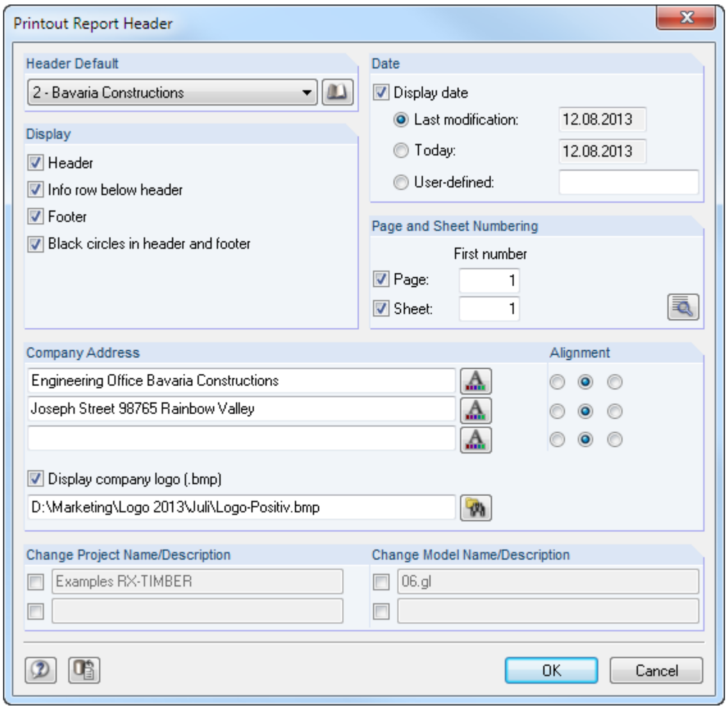


Figure 8.9: Dialog box *Printout Report Header*

In the dialog section *Display*, you select the elements that should be shown in general. In addition, it is possible to define *Date* settings.

The option *Info row below header* activates and deactivates the display of project and model data, with or without *Date*. The project description is taken from the project's general data filed in the Project Manager (see chapter 4.1.1, page 14). The model description is taken from the base data of the model (see chapter 4.2, page 26). It is possible to adjust the presettings in the dialog sections *Change Project Name/Description* and *Change Model Name/Description*.

The *Footer* can be switched on and off as well as the *Black circles* in the points of intersection of the boundary line with header and footer line.

The numbering can be adjusted in the dialog section *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.

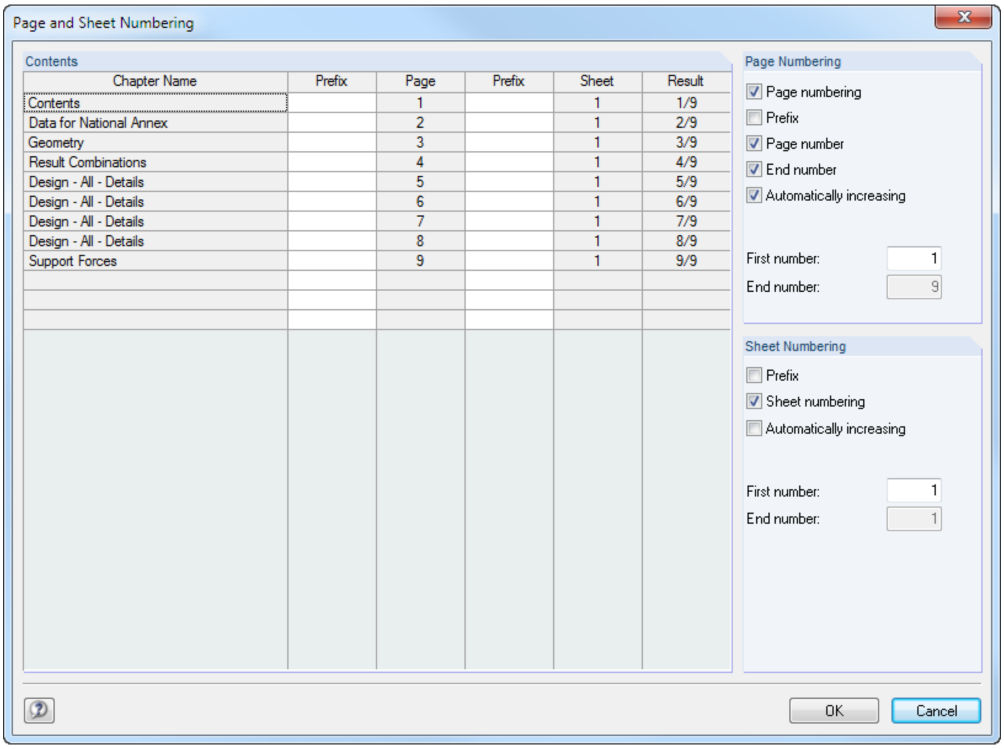


Figure 8.10: Dialog box *Page and Sheet Numbering*

The dialog section *Company Address* in the dialog box *Printout Report Header* (see Figure 8.9) 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 area of the header is reserved for the company logo. The image must be available as bitmap file format (MS Paint for example saves graphics in *.bmp format).

To save the modified settings, click the button [Set Header as default] in the bottom left corner 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.

Use the dialog button [Header Library] to access different printout report headers. In the *Header Library* you can create templates or modify and delete available headers.

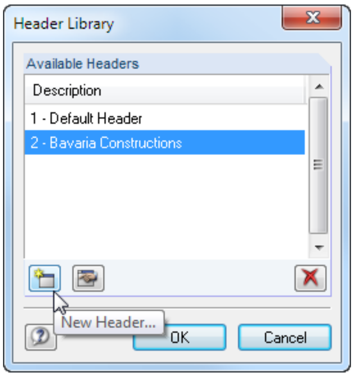


Figure 8.11: Dialog box *Header Library*

The headers are usually 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.

8.4 Result Diagrams

The printout report can be completed by graphics of the result diagrams (see chapter 7.6, page 67). First, set the corresponding result diagrams in RX-TIMBER by means of the button shown on the left.

Use the [Print] button to print the current graphic into the printout report.

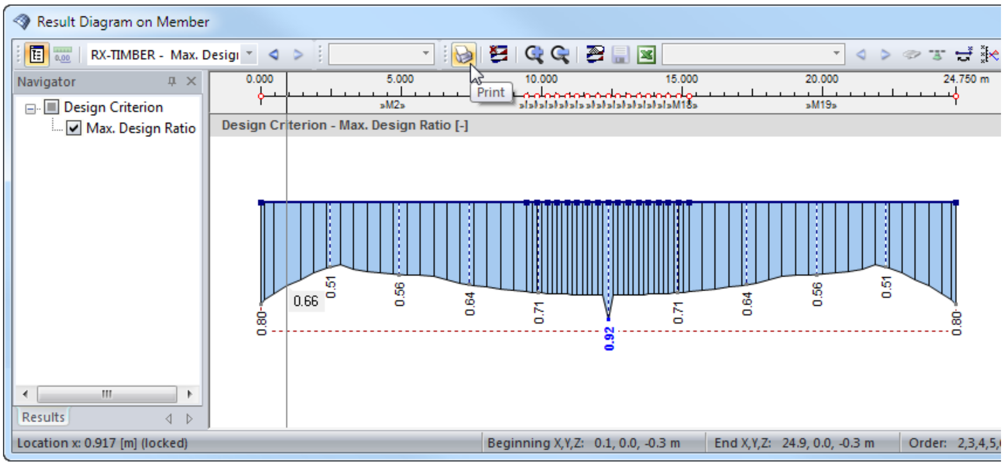


Figure 8.12: Button *Print* in the toolbar of the window *Result Diagram on Member*

The following dialog box appears:

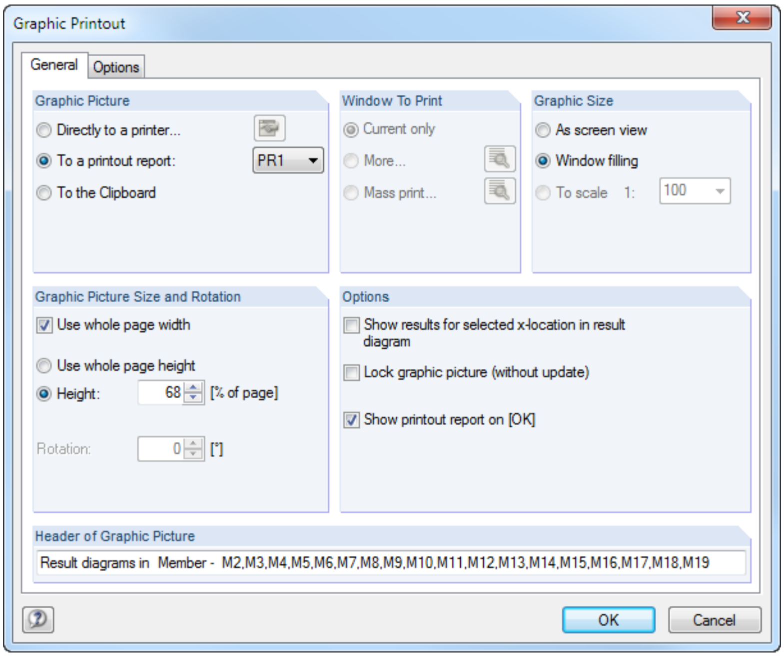


Figure 8.13: Dialog box *Graphic Printout*, tab *General*

Graphic picture

In the dialog section *Graphic Picture*, select the option *To a printout report*.

The dialog sections *Window To Print* and *Graphic Size* are irrelevant for the result diagrams of RX-TIMBER (they represent settings for RFEM graphics).

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.

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

When you move the mouse along the member set in the result diagram, you can see the "moving" result values for the current x-location. Use the check box *Show results for selected x-location in result diagram* to decide if values displayed on the position of the vertical line appear in the printout of the result diagram.

The program generates dynamic graphics by default: When the model is modified, the graphics in the printout report will be updated automatically. To disable the update function, use the check box *Lock graphic picture*.

You can unlock the graphic in the printout report: Right-click the graphic item in the report navigator to open its context menu (see Figure 8.3, page 69). 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 *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 8.1, page 68). The buttons have the following functions:

	Refreshes all graphics
	Unlocks all graphics which can then be updated dynamically
	Locks all graphics which are then fixed in the printout report

Table 8.2: Graphic buttons in the printout report

When you click the [OK] button in the *Graphic Printout* dialog box, the printout report usually opens. This may be annoying, for example when you want to print several graphics one after the other into the report. To prevent the report from being opened, clear the check box *Show printout report on [OK]* in the dialog section *Options*.

Header of graphic picture

When the dialog box *Graphic Printout* opens, a title with a list of member numbers is preset for the graphic. It can be adjusted in the input field.

8.5 Graphics and Texts

It is also possible to integrate external graphics and texts into the printout report.

Inserting graphics

To insert a picture that is not a graphic of the RX-TIMBER program, 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 of the printout report.

You have to enter a chapter name for the new graphic before it is inserted.

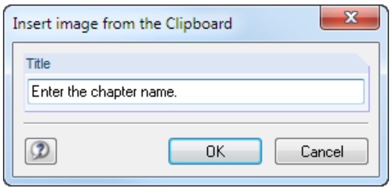


Figure 8.14: Dialog box *Insert image from the Clipboard*

The graphic will appear as a single chapter in the printout report.

Inserting 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 of the printout report.

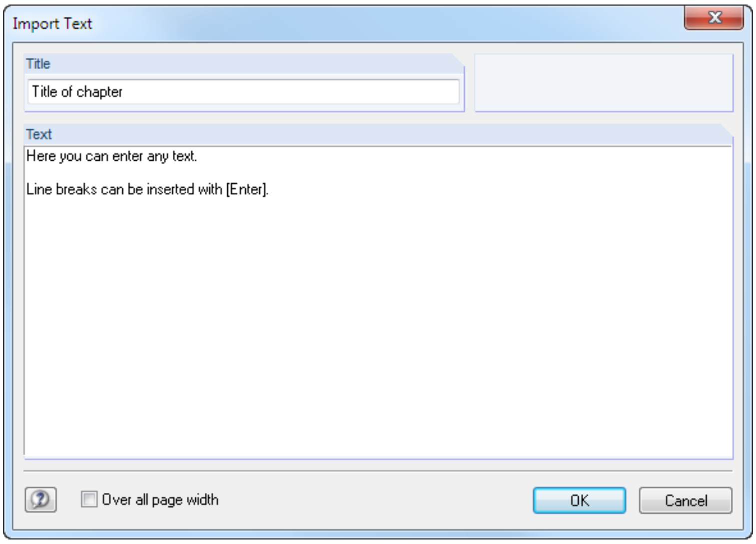


Figure 8.15: Dialog box *Import Text*

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 8.1, page 70) 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.

Inserting RTF and text 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 of the printout report.

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 8.1, page 70) you can modify the text (of .txt files) subsequently by right-click. The dialog box *Import Text* appears for user-defined adjustments.

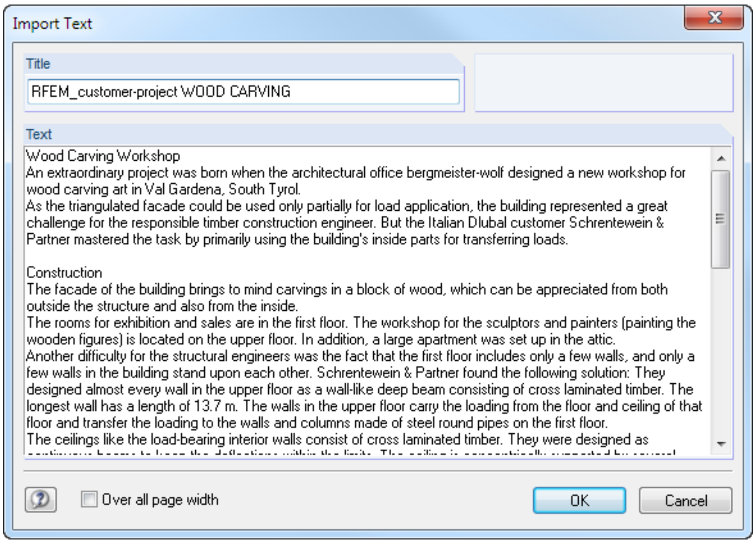


Figure 8.16: Dialog box *Import Text*

8.6 Printout Report Template

The selection described in chapter 8.2 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 8.2 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.

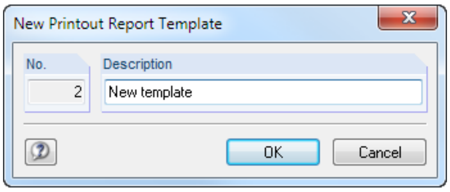


Figure 8.17: Dialog box *New Printout Report Template*

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 8.17).

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 of the printout report, and then click **Select**.

A dialog box opens where you can select the template from the list *Available Printout Report Templates*.

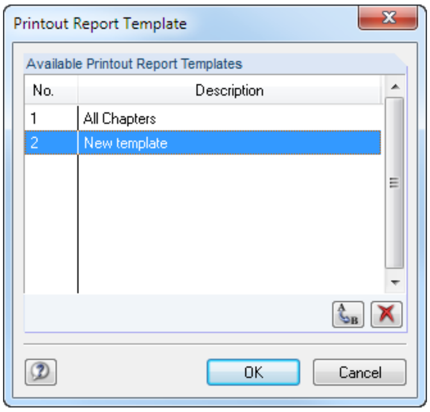


Figure 8.18: Dialog box *Printout Report Template*

Details to the buttons in this dialog box can be found in Table 8.3 below.

After confirming the dialog box and the subsequent security query, the current selection will be overwritten by the template.

Managing templates

All templates are managed in the dialog box *Printout Report Template*. To open the dialog box,

point to **Printout Report Template** on the **Settings** menu of the printout report, and then click **Select**.

The dialog box shown in Figure 8.18 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 8.3: Buttons in the dialog box *Printout Report Template*

The printout report templates are stored in the file **RfemProtocolConfig.cfg** that can be found in the master data folder for RX-TIMBER (*C:\ProgramData\Dlubal\RX-TIMBER 2\General Data*). The file won't be overwritten during an update. Nevertheless, saving a backup file may be useful.

8.7 Presentation

8.7.1 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 **Page** on the **Settings** menu of the printout report or

use the toolbar button in the printout report shown on the left.

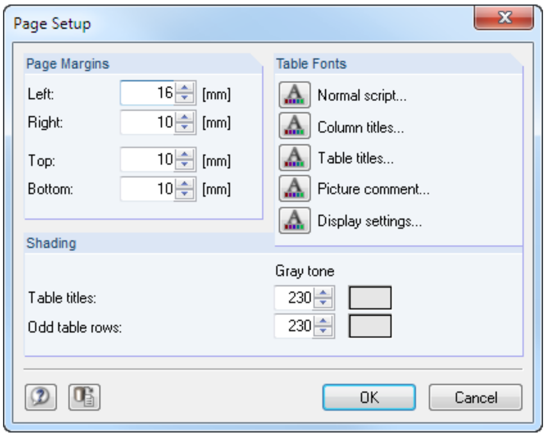


Figure 8.19: Dialog box *Page Setup*

The default fonts for table contents and table headers are relatively small. However, you should be careful with changing the **Arial** default settings. Larger fonts do not always fit in the columns and entries will be cut.

8.7.2 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 of the printout report or

use the toolbar button in the printout report shown on the left.

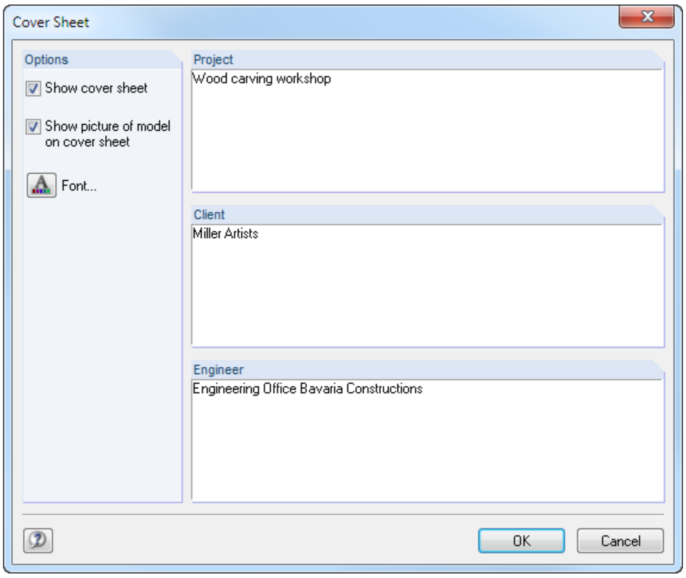


Figure 8.20: Dialog box *Cover Sheet*

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

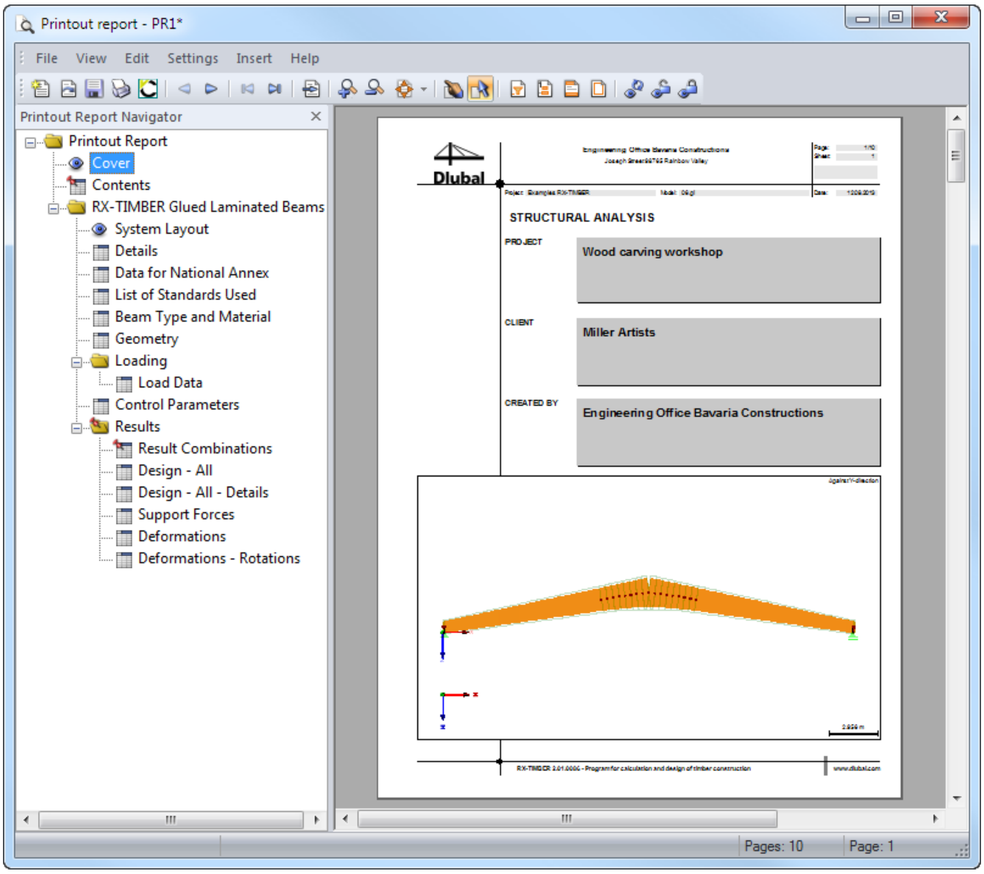


Figure 8.21: Cover sheet in the printout report

The contents of the cover sheet can be remodified in the selection mode (see Table 8.1, page 70) by a double-click. Alternatively, right-click the cover sheet in the report navigator and select *Properties* in the context menu.

8.7.3 Language

The language in the printout report can be set independently of the language applied for the RX-TIMBER user interface (see chapter 9.2, page 88). Therefore, you can create a German or Italian printout report though you are working with the English program version of RX-TIMBER.

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.

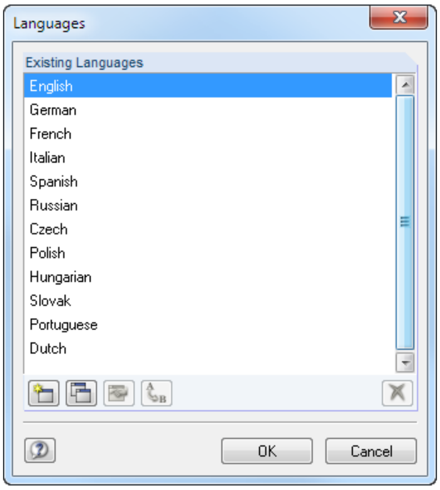


Figure 8.22: Dialog box *Languages*

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 8.22), you see some buttons used to manage the languages.

Create a new language

Click the button shown on the left to open a dialog box. Enter the *Name* of the new language and select the *Language group* from the list so that the character set is interpreted correctly.

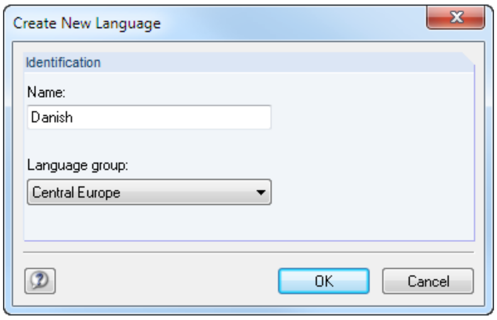


Figure 8.23: Dialog box *Create New Language*

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

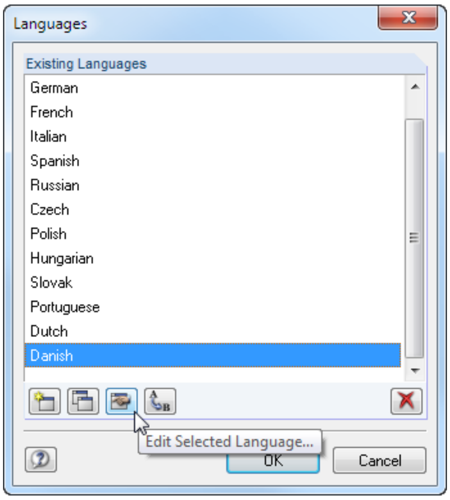


Figure 8.24: Dialog box *Languages*, button *Edit Selected Language*

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

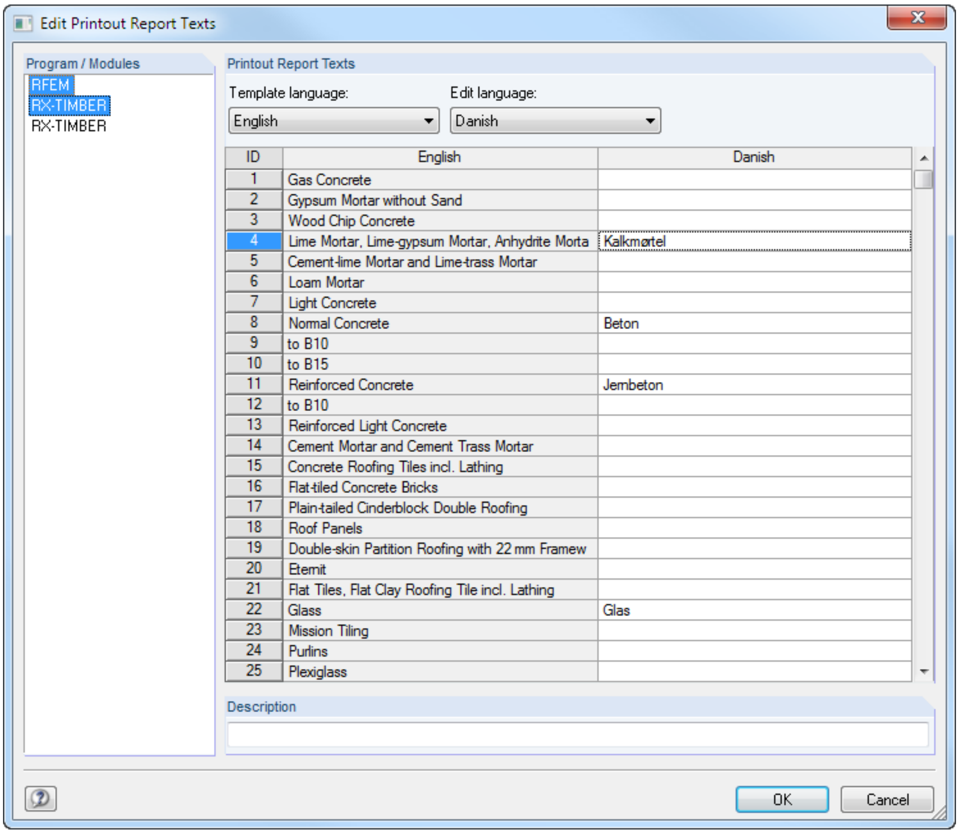


Figure 8.25: Dialog box *Edit Printout Report Texts*

Only user-defined languages can be edited.

Copy a language

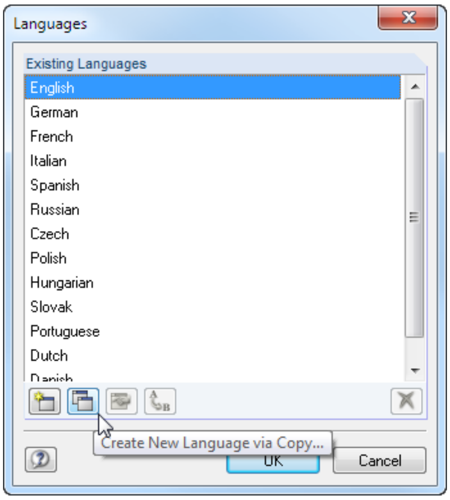


Figure 8.26: Dialog box *Languages*, button *Create New Language via Copy*

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 8.25, column *Danish*) 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.

8.8 Printout

8.8.1 Direct Printing

To start the printing process,

select **Print** on the **File** menu of the printout report

or use the toolbar button shown on the left.

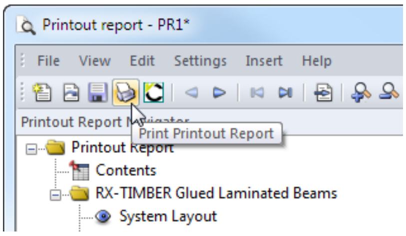


Figure 8.27: Button *Print Printout Report*

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

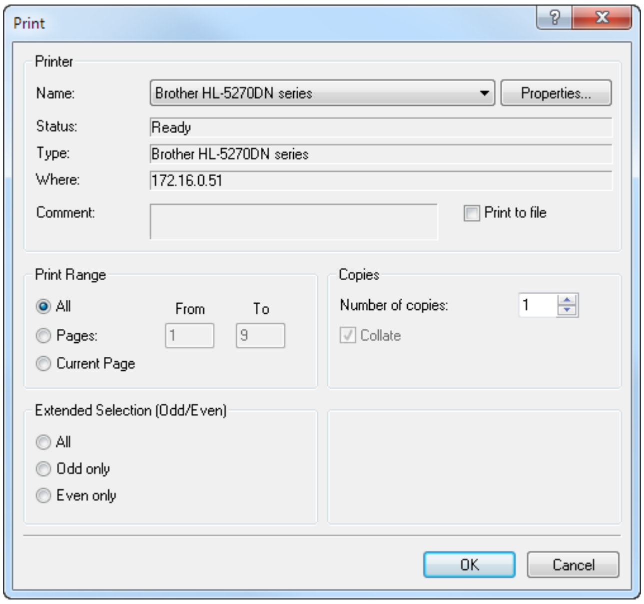


Figure 8.28: Dialog box *Print*

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 RX-TIMBER.

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.

8.8.2 Export

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 of the printout report.

The Windows dialog box *Save As* opens.

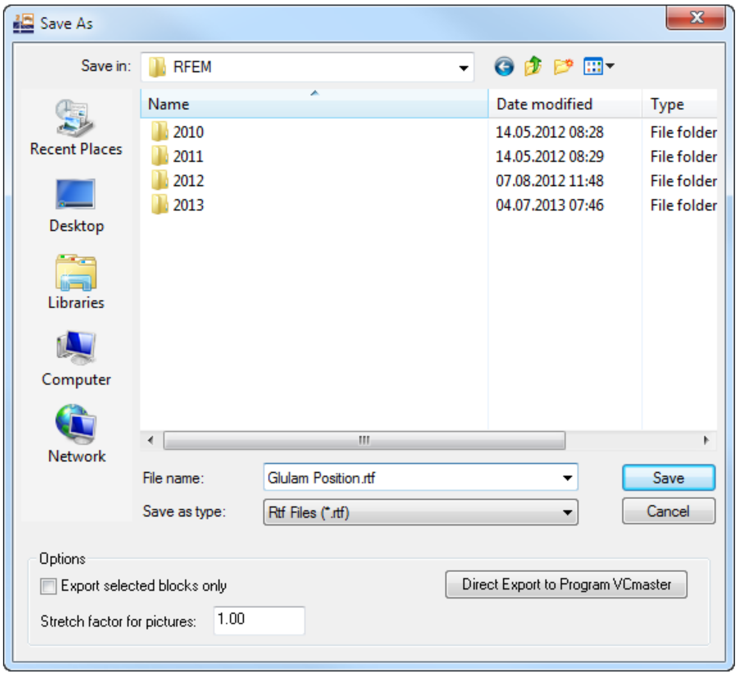


Figure 8.29: Dialog box *Save As*

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 integrated PDF printer 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 of the printout report.

The Windows dialog box *Save As* opens (see Figure 8.29Figure 8.29) 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 of the printout report

or use the button [Export to VCmaster] in the printout report toolbar shown on the left.

The dialog box shown in Figure 8.29 appears where you have to tick the check box for *Direct Export to Program VCmaster*.

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

9. General Functions

9.1 Units and Decimal Places

The units and decimal places can be modified however you want during the modeling or evaluation. All numerical values will be converted or adjusted.

Changing units and decimal places

Some dialog boxes provide the button shown on the left which you can use to access the dialog box for changing units and decimal places (see Figure 9.6 for dialog box *Display Properties*).

To open the dialog box *Units and Decimal Places*, you can also

select **Units and Decimal Places** on the **Settings** menu.

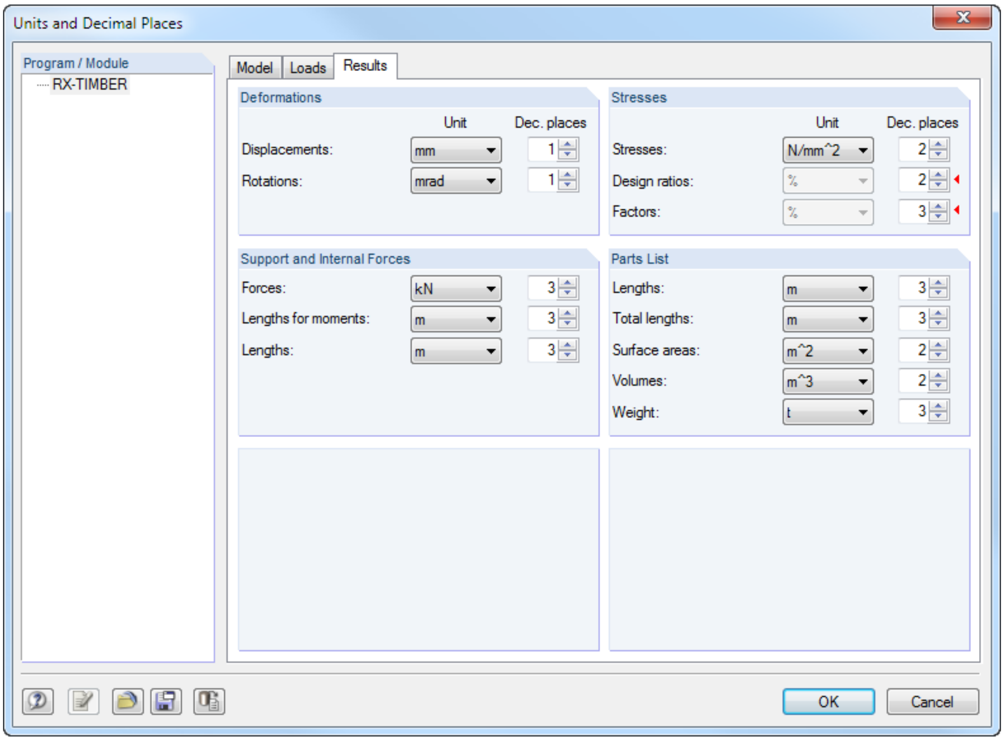


Figure 9.1: Dialog box *Units and Decimal Places*

Three dialog tabs are offered so that you can specify settings separately for *Model*, *Loads* and *Results* data. The units and decimal places are summarized in groups.

The units and decimal places that are relevant for the current table are marked with a red triangle as shown in 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, specific unit profiles for timber models are possible.

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

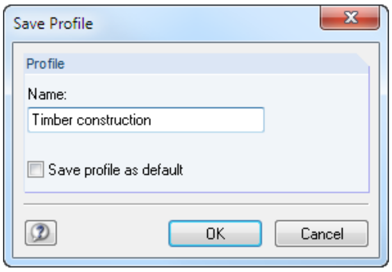


Figure 9.2: Dialog box *Save Profile*

To use this profile as default setting for new models, tick the check box for *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.

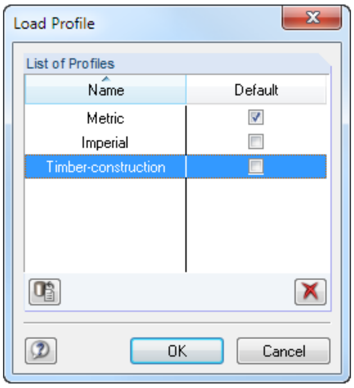


Figure 9.3: Dialog box *Load Profile*

9.2 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 (see chapter 4.1),

select **Language (RX-TIMBER)** on the **Edit** menu of the Project Manager.

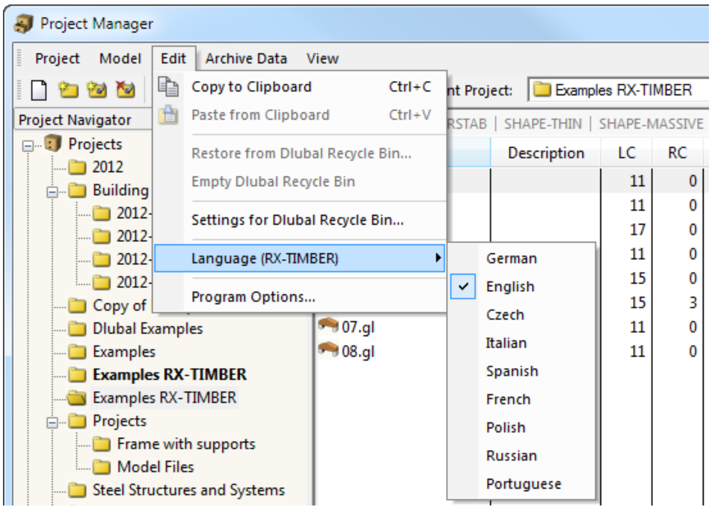


Figure 9.4: Project Manager menu *Edit* → *Language (RX-TIMBER)*

The changed language settings will be effective after restarting the program.

9.3 Display Properties

The display properties control how graphical objects are represented in the program windows and the printout.

Adjusting the display

To open the dialog box for adjusting the graphical display,

select **Display Properties** on the **Settings** menu of the RX-TIMBER program.

Alternatively, window 1.3 *Loading* offers a button to access the corresponding dialog box: First, click the button shown on the left to open the dialog box *Load Cases* (see Figure 5.29, page 48) where you find the button [Display Properties of Selected Load] next to the graphic.

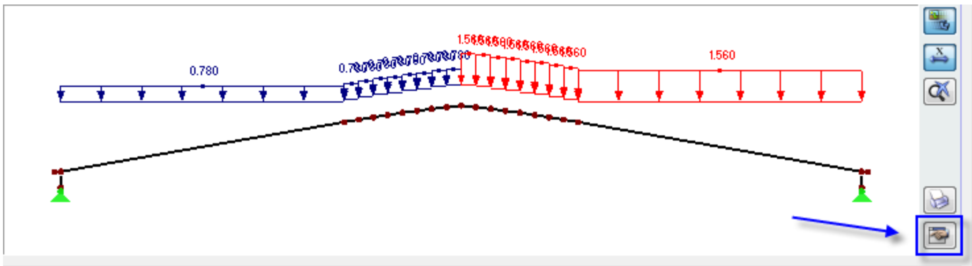


Figure 9.5: Button *Display Properties* in the dialog box *Load Cases*

The dialog box *Display Properties* appears.

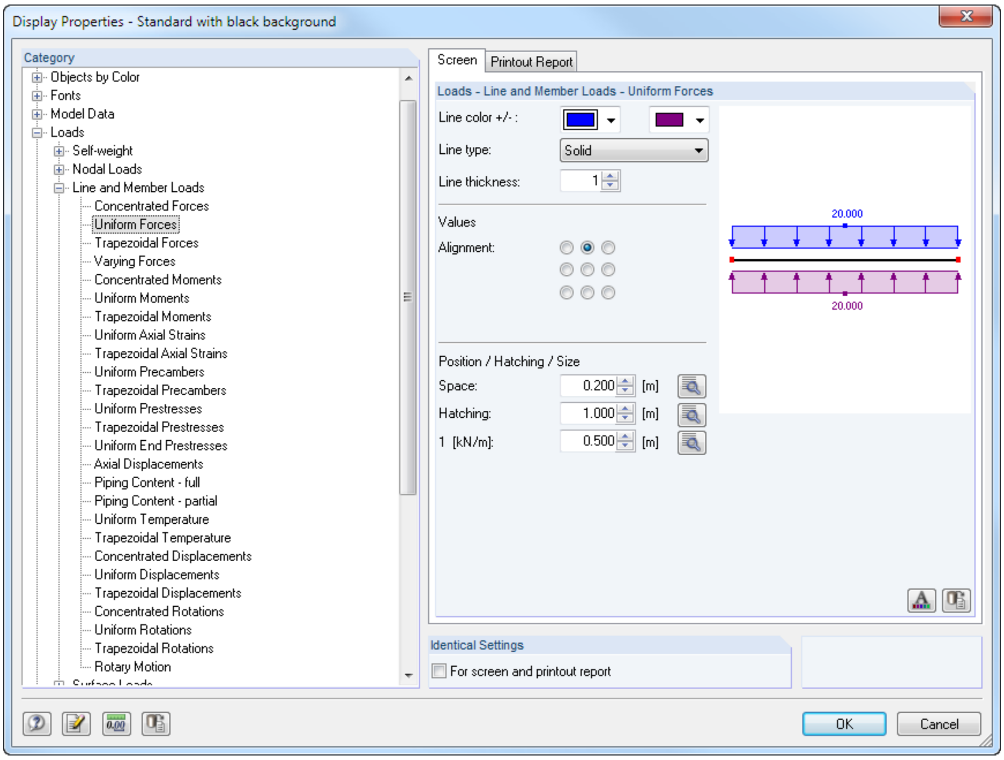


Figure 9.6: Dialog box *Display Properties*

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.

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, type and arrangement of numbering, font, size of load vector etc.

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 subsequently defined settings 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 buttons below the parameters have the following functions:

	Opens the *Font* dialog box for changing type, size and color of font
	Goes to display parameters of axes for current object
	Opens the dialog box *Relative Positions* for arranging descriptions
	Restores default settings

Table 9.1: Buttons in dialog box *Display Properties*

9.4 Export of Results

The results of the RX-TIMBER design can be used in other programs.

Clipboard

To copy cells selected in the results windows to the Clipboard, press [Ctrl]+[C]. To insert the cells, for example in a word processing program, press [Ctrl]+[V]. The headings of table columns are not exported.

Printout report

Data of RX-TIMBER programs can be printed in the printout report (see chapter 8.1, page 68) where it can be exported.

Select **Export to RTF** on the **File** menu of the printout report.

The function is described in chapter 8.8.2 on page 85.

Excel / OpenOffice

RX-TIMBER provides a function for the direct data export to MS Excel, OpenOffice.org Calc or the file format CSV. To open the corresponding dialog box,

select **Export to MS Excel** on the **File** menu of RX-TIMBER.

The following export dialog box appears.

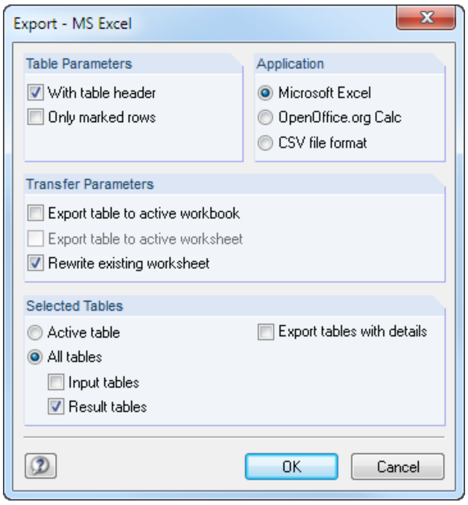
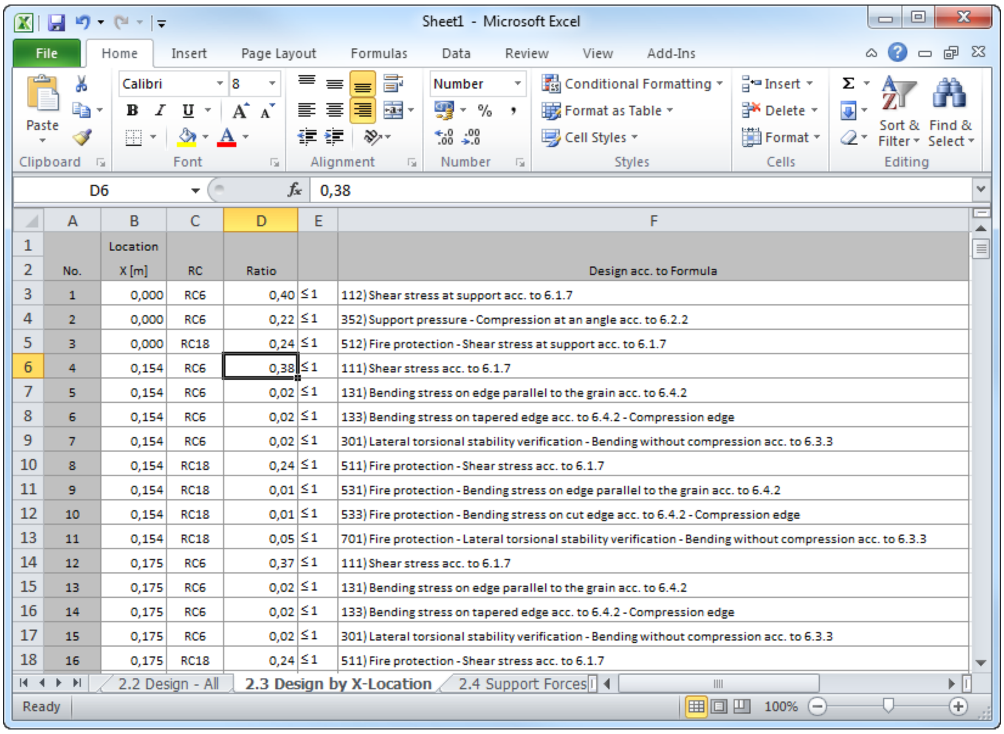


Figure 9.7: Dialog box *Export – MS Excel*

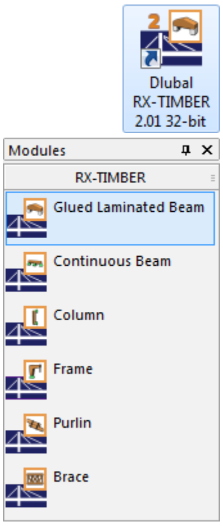
When your selection is complete, you can start the export by clicking [OK]. Excel or OpenOffice will be started automatically. It is not necessary to run the programs in the background.



No.	Location X [m]	RC	Ratio		Design acc. to Formula
1	0,000	RC6	0,40	≤ 1	112) Shear stress at support acc. to 6.1.7
2	0,000	RC6	0,22	≤ 1	352) Support pressure - Compression at an angle acc. to 6.2.2
3	0,000	RC18	0,24	≤ 1	512) Fire protection - Shear stress at support acc. to 6.1.7
4	0,154	RC6	0,38	≤ 1	111) Shear stress acc. to 6.1.7
5	0,154	RC6	0,02	≤ 1	131) Bending stress on edge parallel to the grain acc. to 6.4.2
6	0,154	RC6	0,02	≤ 1	133) Bending stress on tapered edge acc. to 6.4.2 - Compression edge
7	0,154	RC6	0,02	≤ 1	301) Lateral torsional stability verification - Bending without compression acc. to 6.3.3
8	0,154	RC18	0,24	≤ 1	511) Fire protection - Shear stress acc. to 6.1.7
9	0,154	RC18	0,01	≤ 1	531) Fire protection - Bending stress on edge parallel to the grain acc. to 6.4.2
10	0,154	RC18	0,01	≤ 1	533) Fire protection - Bending stress on cut edge acc. to 6.4.2 - Compression edge
11	0,154	RC18	0,05	≤ 1	701) Fire protection - Lateral torsional stability verification - Bending without compression acc. to 6.3.3
12	0,175	RC6	0,37	≤ 1	111) Shear stress acc. to 6.1.7
13	0,175	RC6	0,02	≤ 1	131) Bending stress on edge parallel to the grain acc. to 6.4.2
14	0,175	RC6	0,02	≤ 1	133) Bending stress on tapered edge acc. to 6.4.2 - Compression edge
15	0,175	RC6	0,02	≤ 1	301) Lateral torsional stability verification - Bending without compression acc. to 6.3.3
16	0,175	RC18	0,24	≤ 1	511) Fire protection - Shear stress acc. to 6.1.7

Figure 9.8: Result in MS Excel: window 2.3 *Design by X-Location*

10. Glued-Laminated Beam



The most important functions of **RX-TIMBER Glued-Laminated Beam** are presented in the following by describing two examples.

In the first example we enter and calculate a pitched cambered beam with inclined bottom chord and variable height (*Beam Type 6*). The second example in chapter 10.2 is the lateral buckling analysis of a fish beam (*Beam Type 8*).

Start the program by double-clicking the desktop icon **Dlubal RX-TIMBER 2.xx** (see chapter 3.2, page 12), and then select **Glued Laminated Beam** in the Project Manager.

The data of both examples can also be found in the models *06.gl* and *08.gl* of the project *Examples RX-TIMBER* that is created automatically during the program installation. However, if you are a program beginner, we recommend you to enter the example data manually.

10.1 Example for Pitched Cambered Beam

10.1.1 System and Loads

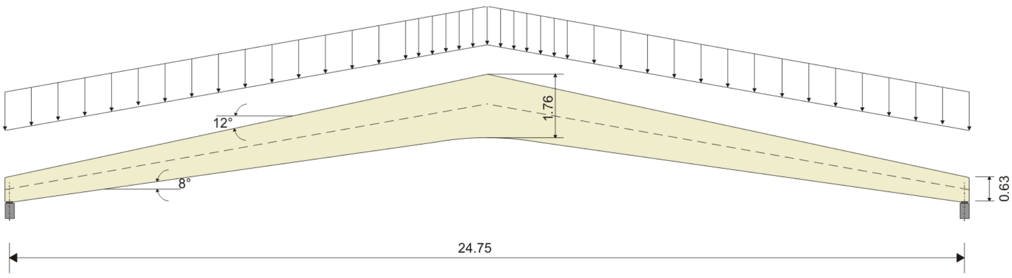


Figure 10.1: System and loads

Geometry and material

We use combined glued-laminated timber **GL24c** as material for the structure.

The linearly variable cross-section of the beam is defined with **h_s = 629.2 mm** on the support edge and **h_{ap} = 1761 mm** in the span center, resulting in an inclination of **12°** for the top beam side and **8°** for the bottom beam side.

The beam width is **180 mm**.

Load determination

Load case 1: self-weight

The load is given by the self-weight of the beam and the building's roof structure. We refer to chapter 5.3, page 41 when we determine the load and enter the roof structure.

In our example, we assume a roof structure of 0.64 kN/m^2. With a spacing of 3 m between the trusses we get a distributed load of 1.92 kN/m plus beam self-weight.

Load case 2: snow

The construction project lies in zone 1 with a height of 200 m above sea level. Together with the shape coefficient we get a snow load of:

$s = 0.8 \cdot 0.65 \text{ kN/m}^2 \cdot 3 \text{ m} = 1.56 \text{ kN/m}$

The snow load is applied as projected length with load direction Z.

The load case *wind* is not considered in the example.

10.1.2 Input Data

10.1.2.1 Beam Type and Material

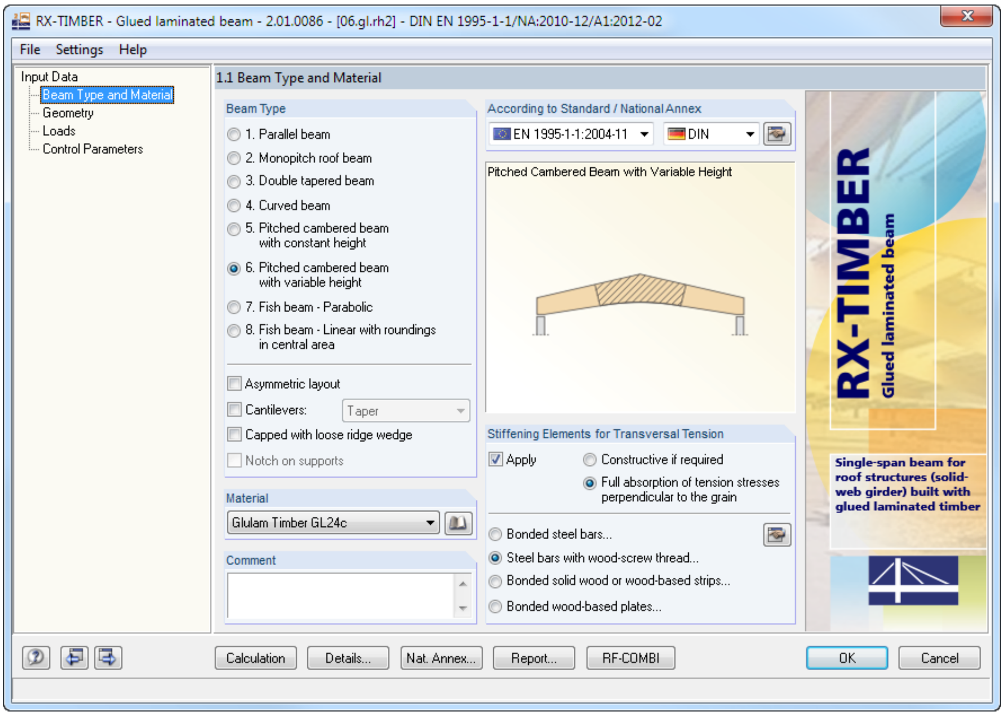


Figure 10.2: Window 1.1 *Beam Type and Material*

To enter a pitched cambered beam with variable height, we select **Beam Type 6** and the material **Glulam Timber GL24c** in window 1.1 *Beam Type and Material*.

The program presets a beam with *Cantilevers*. We deactivate the option for our example because we want to calculate the beam **without cantilevers**.

We design the beam according to **EN 1995-1-1:2004-11** and the national annex for Germany according to **DIN**.

Generally, there is a great danger of failure due to transversal tension for this type of beam. Therefore, we want the program to calculate the required **Stiffening Elements for Transversal Tension**.

Then, we select **Steel bars with wood-screw thread**. A new dialog box opens offering three options to define the *Layout of Steel Bars*. We select the first option **Determine quantity**.

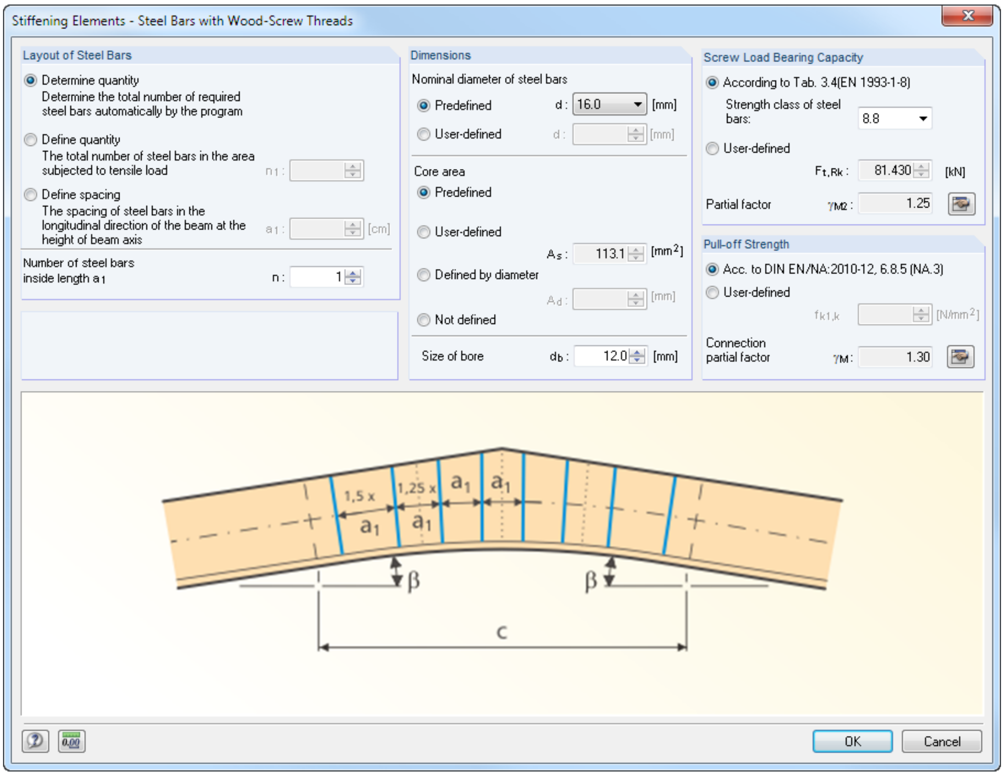


Figure 10.3: Dialog box *Stiffening Elements - Steel Bars with Wood-Screw Threads*

The *Nominal diameter of steel bars* is **16 mm**. The load bearing capacity of the screw $F_{t,Rk}$ is determined by the strength class of the steel bars **8.8** according to EN 1993-1-8, table 3.4.

10.1.2.2 Geometry

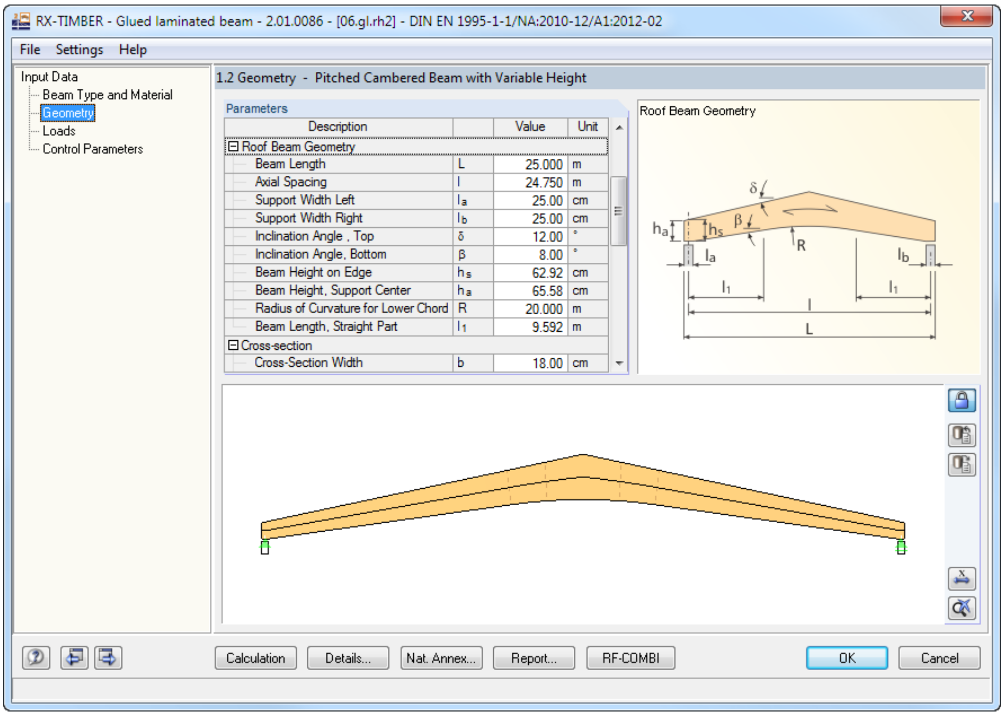


Figure 10.4: Window 1.2 *Geometry - Pitched Cambered Beam with Variable Height*

In the second program window, we enter the geometry of the beam and building. Based on the building dimensions, the wind and snow load generator determines in the program's background the size of the load area on the roof as well as the attached wall surfaces. The data input is interactive: As soon as the geometry is modified, the loading of the beam changes as well.

We select the following values for our example: *Building Height* **10.00 m**, *Building Depth* **40.00 m**, *Beam Spacing* **3.00 m**, entire *Beam Length* **25.00 m**, *Support Width* **25 cm**, *Beam Height on Edge* **62.92 cm**, *Inclination Angle, Top* **12°**, *Inclination Angle, Bottom* **8°**.

The *Radius of Curvature* must be limited in the service class 1-2 according to [1] equation (H.1) and [7] clause NA 11.4:

$$r = 2.5 \cdot 40^2 \text{ mm} + 117.5 \cdot 40 \text{ mm} = 8700 \text{ mm}$$

To reach an efficient ratio for transversal tension load and timber volume, we define the radius *R* with **20.00 m**.

The *Lamella Thickness* is **4.0 cm**. It is possible to enter any value in a range of 0.5 to 4.0 cm. The input field offers a selection list with the default values 3.3 and 4.0 cm. We set **18.0 cm** for the *Cross-section Width*.

Under the table entry *Data for Lateral Buckling* we specify the value for *Spacing of Lateral Supports* with **3.00 m**. The *Bracing Distance* from the beam axis is defined with **31.50 cm**.

The distance *e* as center distance of the stiffening from the horizontal retention of the bar on the support is represented in DIN 1052: 2004-08, figure E.2. The smaller the distance is defined, the larger is the moment that must be absorbed by the pinned support because the lever arm for the absorption of the tilting moment is reduced.

$$T_d = M_d \cdot \left[\frac{1}{80} - \frac{1}{60} \cdot \frac{e}{h} \cdot (1 - k_m) \right]$$

Equation 10.1: Determination of pinned support according to descriptions for DIN 1052: 2008-12, equation (14)

10.1.2.3 Loads

The first action that we enter in window 1.3 *Loading* is the *Permanent Action*, specifying the load "self-weight and roof structure". The following values are selected for the *Roof Layer*.

- Trapezoidal sheet metal with 0.15 kN/m²
- Vapour barrier with 0.02 kN/m²
- Rock wool (d = 30 cm) with 0.30 kN/m²
- Purlins with 0.15 kN/m²
- Gypsum boards incl. strapping with 0.02 kN/m²

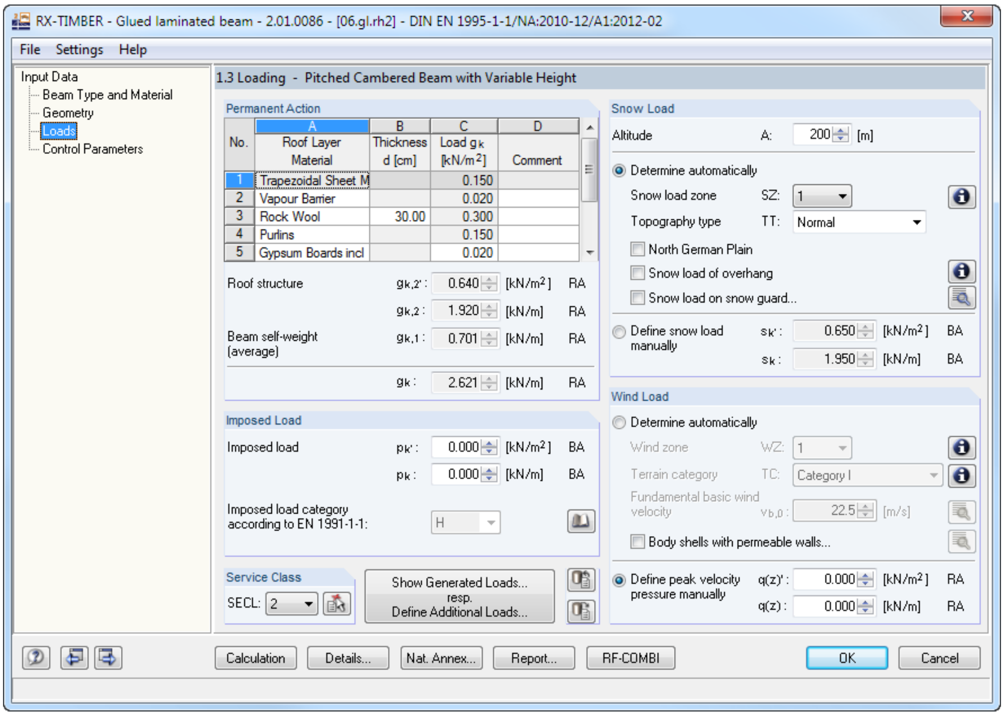


Figure 10.5: Window 1.3 *Loading*

We use the [...] button occurring in the *Roof Layer* column to select materials from the library. It is also possible to integrate a new loading into the material library.

The roof structure results in the permanent action g_k of 2.62 kN/m. This load includes the self-weight of the beam. As the beam has a linearly variable height, a trapezoidal load for the self-weight of the beam is automatically applied. To check the loading, use the button [Show Generated Loads resp. Define Additional Loads] shown on the left (see for example Figure 5.28, page 47). A dialog box opens with a table where you can enter also user-defined loads.

It would be possible to define an additional *Imposed load* by using the button shown on the left, selecting a load from the table and assigning the corresponding *Imposed load category* from the list for *EN 1991-1-1*. However, in our example we only define the *Service Class* **2** for the beam.

To determine the *Snow Load*, we enter **200 m** for the *Altitude* above sea level. We define the *Snow load zone* **SZ 1** by a double-click in the snow load map.

We do not want to analyze the wind load in our example. Therefore, we set the *peak velocity pressure* to **0.00** in both input fields.

10.1.2.4 Control Parameters

The final input window 1.4 *Control Parameters* manages the settings for the designs that you want to perform.

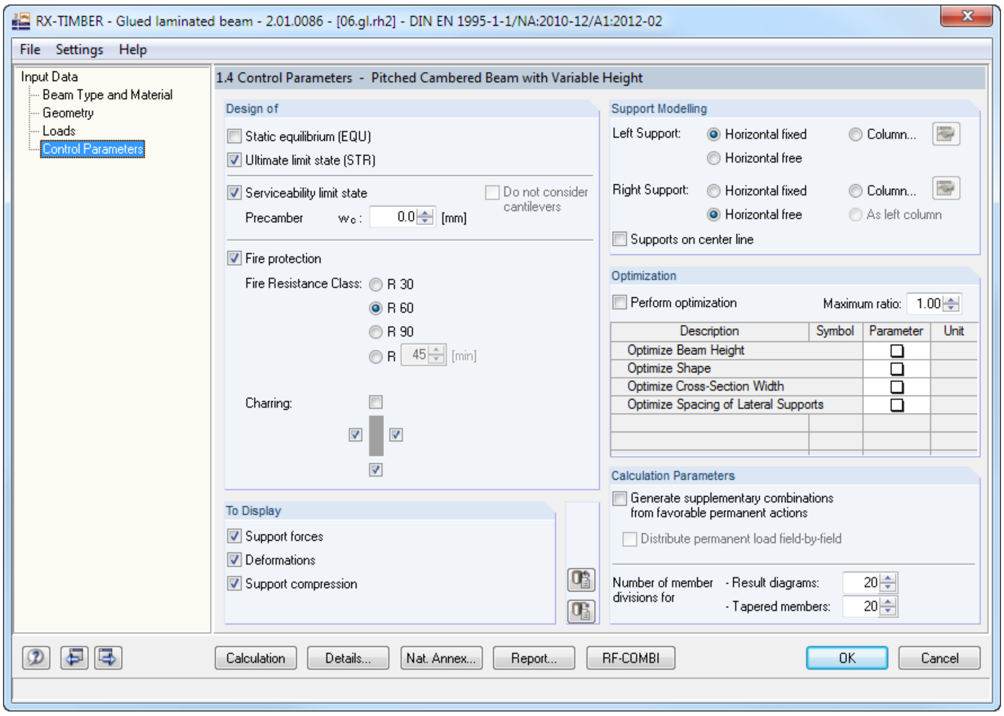


Figure 10.6: Window 1.4 *Control Parameters*

In our example, we analyze the **Ultimate limit state**, the **Serviceability limit state** and the **Fire protection**. The fire resistance class is set to **R 60**. As there is no *Charring* on the top beam side, we clear the relevant check box.

We check if the remaining settings match the ones shown in the figure above. Finally, the input of the geometry and loads is complete.

10.1.3 Calculation

10.1.3.1 Combinations with RF-COMBI

RF-COMBI

The add-on module RF-COMBI is directly integrated in RX-TIMBER. It generates all result combinations automatically. Use the [RF-COMBI] button to start the add-on module.

To avoid generating an unnecessary high number of result combinations, it is recommended to *Reduce possible result combinations* in the window section *Supplementary Examination* of the RF-COMBI window 1.1 *General Data*. In this way, you make sure that the results of RF-COMBI include all governing result combinations but without generating more load cases than necessary.

Find more information about RF-COMBI in the program manual available for download at www.dlubal.com.

Now, we start the generation of combinations in RF-COMBI with the [Calculation] button. Then, we only look at result combination 2. Therefore, we clear the check boxes for all the other load combinations in window 2.3 *Result Combinations - Reduced*. Only **RC 2** remains ticked. Finally, we click [OK] to quit the add-on module RF-COMBI.

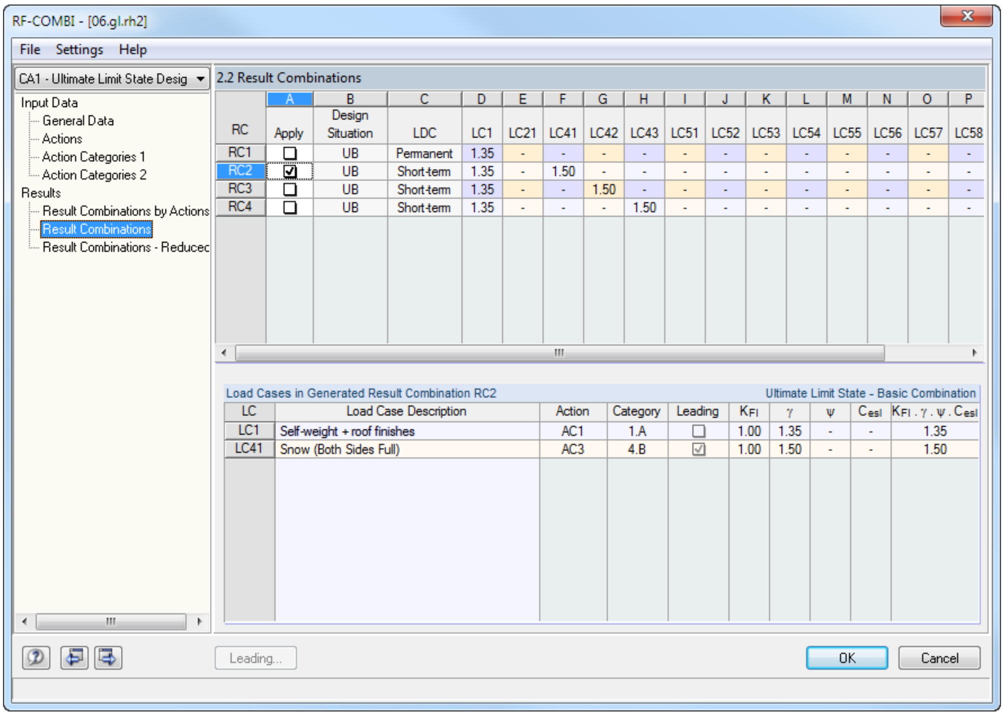


Figure 10.7: RF-COMBI window 2.3 *Result Combinations - Reduced*

10.1.3.2 Start Calculation

Calculation

When all entries have been set according to the specifications, click the [Calculation] button in the RX-TIMBER program to start the calculation.

10.1.4 Results

After the calculation the navigator shows more items with information about the *Results*.

As the location of the maximum bending stress in the beam is not necessarily the location of the maximum moment, RX-TIMBER divides the beam into small sections where it performs the designs. If an even more accurate calculation is needed, you can refine the member division in window 1.4 *Control Parameters*, dialog section *Calculation Parameters*. The designs for these division locations can bee seen in window 2.3 *Design by X-Location*.

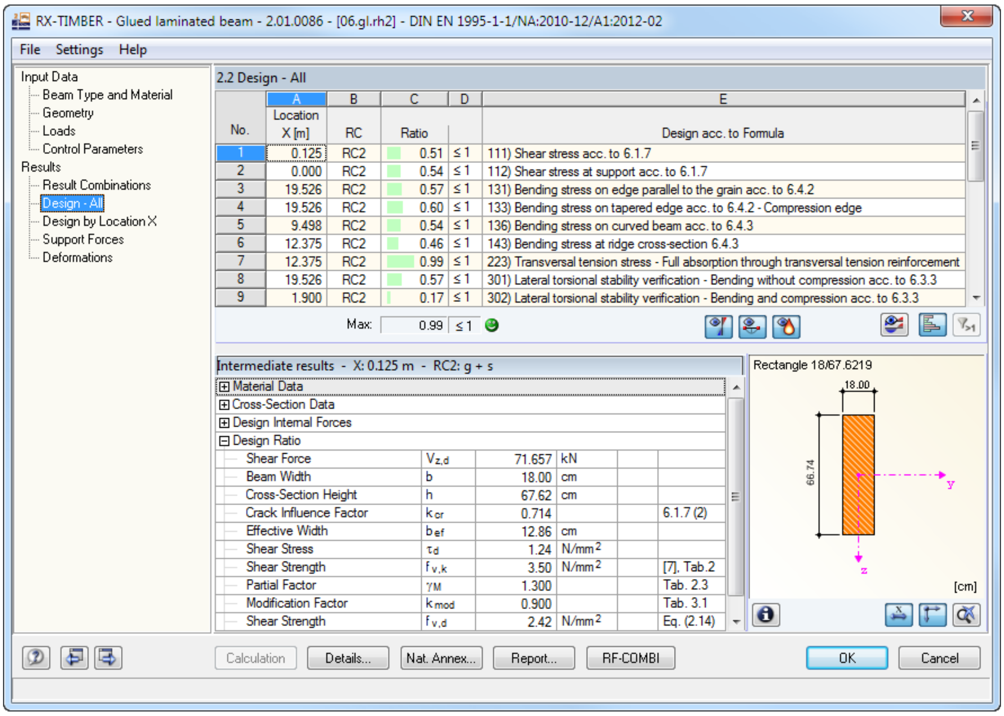


Figure 10.8: Window 2.2 *Design - All* for ultimate limit state design (shear stress at support)

We look at the [Ultimate Limit State Designs] for the result combination RC 2. Due to the notes listed in table column E referring to the equations of EN 1995-1-1: 2004-11, it is easy to understand the analysis. The window section *Intermediate results* below the table informs you about design details of the design currently selected above.

In the following hand calculations, we apply a linear load of 5.78 kN/m with an averaged beam self-weight. RX-TIMBER, however, takes exactly the linearly variable beam height for the self-weight proportion into account.

Design of shear stress

A reduction of the shear force is not necessary for this beam.

Shear stress at support

$$\tau_d = 1.5 \cdot \frac{V_d}{b_{ef} \cdot h_A} = 1.5 \cdot \frac{72.34 \cdot 10^3}{128.6 \cdot 645.8} = 1.31 \frac{N}{mm^2}$$

$$\text{where} \quad k_{cr} = \frac{2.5}{f_{v,k}}$$

$$b_{ef} = b \cdot k_{cr} = 180\ mm \cdot 0.714 = 128.6\ mm$$

Design value of strength

$$f_{v,d} = f_{v,k} \cdot \frac{k_{mod}}{\gamma_M} = 3.5 \cdot \frac{0.9}{1.3} = 2.42 \frac{N}{mm^2}$$

Design ratio

$$\frac{\tau_d}{\tau_{v,d}} = \frac{1.31}{2.42} = 0.54 < 1$$

This value is displayed also in RX-TIMBER for the shear design on the support as shown in Figure 10.8.

Design of bending stress (design 6.4.2)

As already mentioned, the design for the beam's bending stress is not necessarily performed on the location of the maximum bending moment. For our example, we calculate the location of the maximum loading and compare the results for this location with the values determined by RX-TIMBER.

Location of the maximum loading (global system of coordinates)

$$x = \frac{l \cdot h_a}{2 \cdot h_1} = \frac{25.00 \cdot 0.656}{2 \cdot 1.565} = 5.24 \text{ m}$$

Beam height on location x = 5.24 m

$$h'_x = h_a \cdot \left(2 - \frac{h_a}{h_{ap}}\right) = 0.656 \cdot \left(2 - \frac{0.656}{1.76}\right) = 1.06 \text{ m}$$

$$h_x = h_{x'} \cdot \cos\frac{\alpha + \beta}{2} = 1.06 \cdot \cos\frac{12 + 8}{2} = 1.05 \text{ m}$$

Section modulus

$$W_x = \frac{b \cdot h^2}{6} = \frac{18 \cdot 105^2}{6} = 33075 \text{ cm}^3$$

Design moment

$$M_{x,d} = \frac{q \cdot x \cdot (l - x)}{2} = \frac{5.9 \cdot 5.26 \cdot (25.00 - 5.26)}{2} = 306.3 \text{ kNm}$$

Bending stress

$$\sigma_{m,\alpha,d} = \frac{M_{x,d}}{W_x} = \frac{306.3 \cdot 10^3}{33075} = 9.3 \frac{N}{mm^2}$$

For the design in the compression zone on the upper beam edge we get a material resistance of:

$$f_{m,\alpha,d} = f_{m,d} \cdot k_{m,\alpha} = 24 \cdot \frac{0.9}{1.3} \cdot 0.951 = 15.81 \frac{N}{mm^2}$$

$k_{m,\alpha} = 0.951$ according to equation (6.40)

Design ratio

$$\frac{\sigma_{m,d}}{f_{m,\alpha,d}} = \frac{9.3}{15.8} = 0.59$$

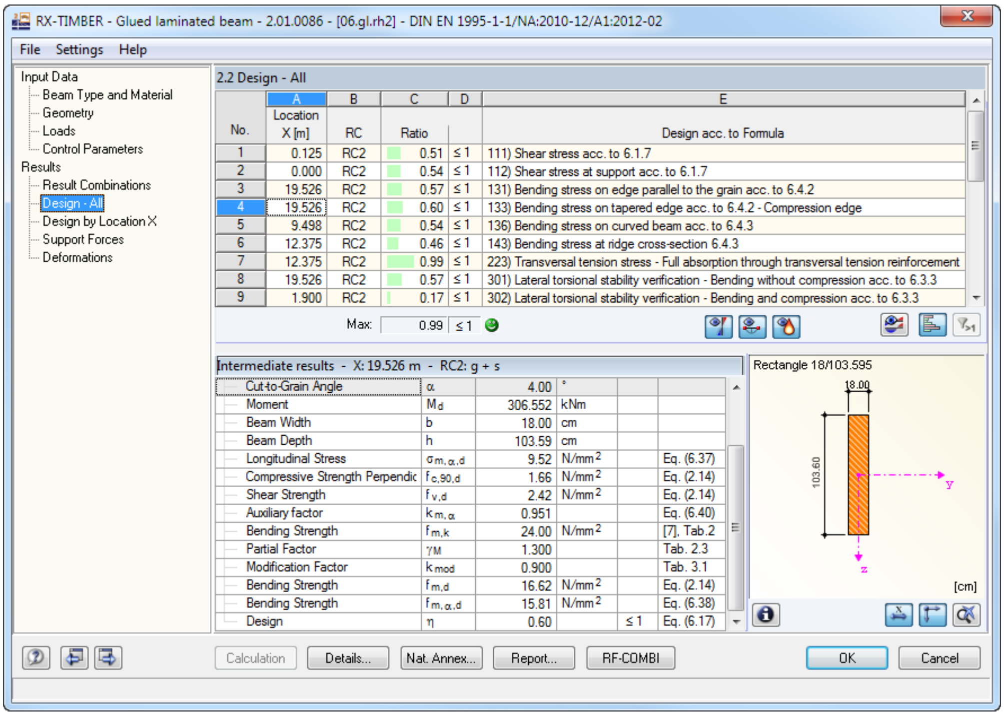


Figure 10.9: Window 2.2 *Design - All* for ultimate limit state design (bending stress on compression edge)

The program's design ratio of 0.60 on the x-location 19.526 deviates insignificantly (see Figure 10.9). RX-TIMBER does not necessarily perform the design as mentioned on the location with the maximum bending stress but analyzes the locations of division. Moreover, an averaged beam self-weight is applied in the hand calculations.

Lateral buckling design

To stiffen the beam, purlins suspended between the girders are provided, restricting the lateral buckling distance to 3.00 m.

Related lateral buckling slenderness on location x = 5.70 m

$$\lambda_{rel,m} = \sqrt{\frac{f_{m,k}}{\frac{0.78 \cdot b^2}{h \cdot l_{ef}} \cdot E_{0.05} \cdot 1.4}} = \sqrt{\frac{2.4}{\frac{18^2 \cdot 0.78}{113.60 \cdot 300} \cdot 940 \cdot 1{,}4}} = 0.496 \leq 0.7$$

For the determination of the lateral buckling slenderness the 5 % quantile of the stiffness properties is multiplied with the factor 1.4 in accordance with the national annex of Germany.

For the stability analysis of tapered cross-sections RX-TIMBER applies the cross-section properties that exist at a distance of the 0.65-fold member length from the end of the member with the smaller section. In this way, it is ensured that the design is performed correctly according to the equivalent member method.

According to equation (6.34), this results in:

$$k_{crit} = 1$$

Design value of strength

$$f_{m,d} = f_{m,k} \cdot \frac{k_{mod}}{\gamma_M} = 24 \cdot \frac{0.9}{1.3} = 16.62 \frac{N}{mm^2}$$

Design ratio

$$\frac{\sigma_{m,d}}{k_{crit} \cdot f_{m,d}} = \frac{9.52}{1 \cdot 16.62} = 0.57 < 1$$

According to equation (6.35) a superposition with compression force must additionally performed.

$$\left(\frac{\sigma_{m,d}}{k_{crit} \cdot f_{m,d}}\right)^2 + \frac{\sigma_{c,0,d}}{k_{c,z} \cdot f_{c,0,d}} = (0.57)^2 + \frac{0.003\text{kN/cm}^2}{0.86 \cdot 14.54} = 0.325 < 1$$

Torsion design

For beam types of this example the restraint of fork supports causes a torsion of the cross-section. Therefore, special torsion designs are required, but we don't perform them in our example. The specifications can be set in the *Details* dialog box that you open by clicking the [Details] button shown on the left.

The distance *e* of the stiffening bracing to be set in window 1.2 *Geometry* among the entries for lateral buckling is significant for the correct application of the procedure. RX-TIMBER increases the distance for a tapered cross-section automatically: The starting point for the program is the axis of the support. The distance *e* of the stiffenings is measured from the start of the member where it is related to the axis. Thus, the distance is also relevant for the design of the cross-section's torsion about its axis.

10.1.5 Documentation

We click the [Report] button to start the print preview of the calculated data.

To adjust the printout report,

select **Selection** on the **Edit** menu of the printout report.

The dialog box *Printout Report Selection* opens where the program RX-TIMBER is already preset in the navigator. In the *Results* tab, we select only the designs for the **Ultimate Limit State** by clearing the check boxes for *Serviceability Limit State* and *Fire Resistance*.

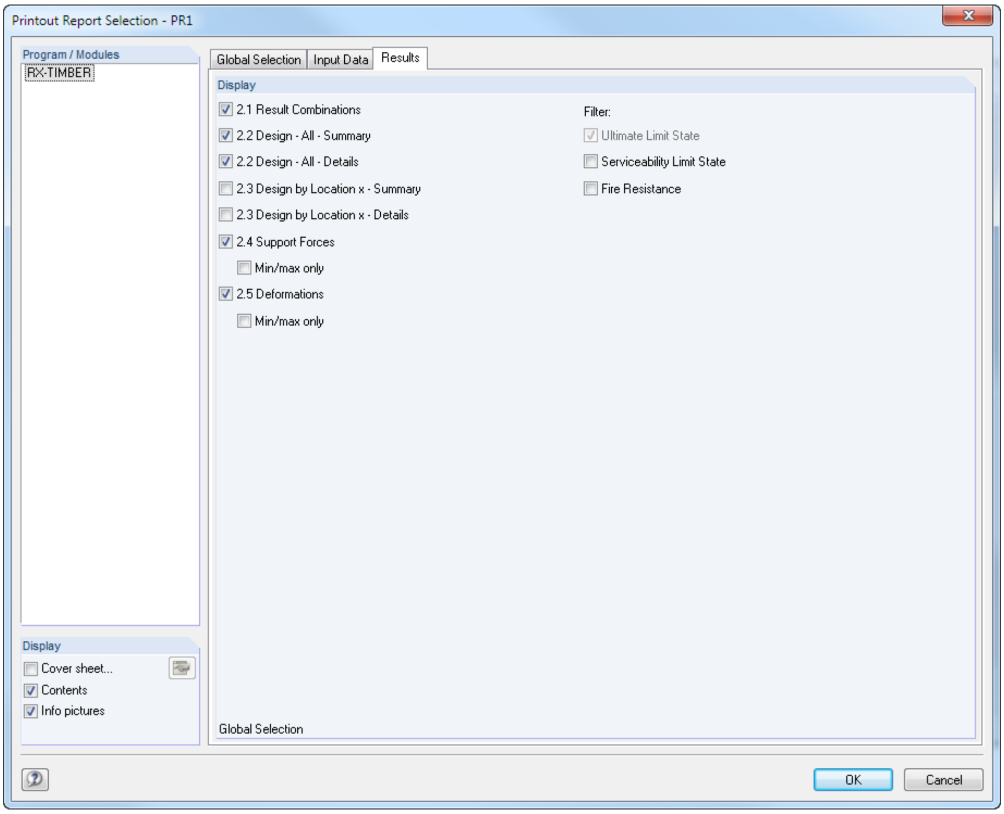


Figure 10.10: Dialog box *Printout Report Selection*

We don't modify the other preset values. Finally, we confirm our selection with the [OK] button.

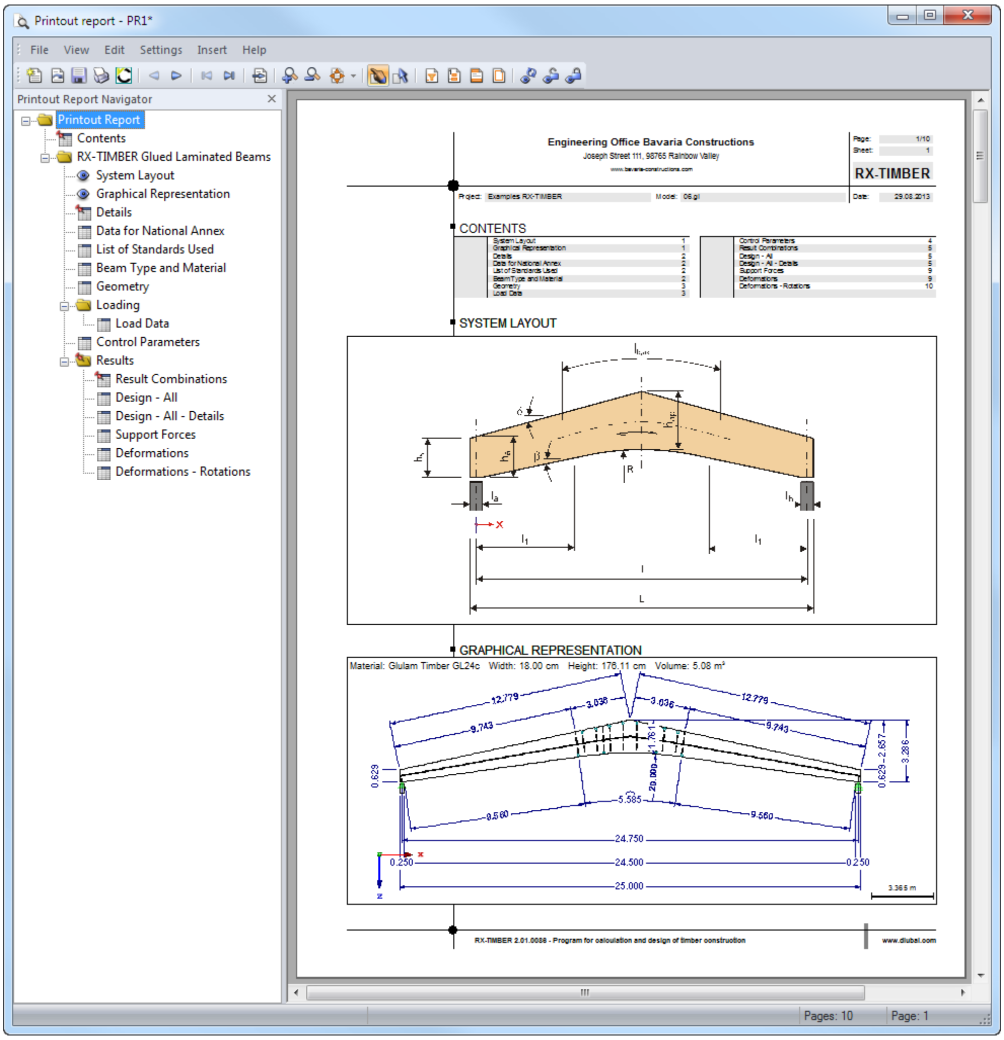


Figure 10.11: Print preview in printout report

Further information on the presentation of the printout report can be found in chapter 8.

10.2 Example for Fish Beam

In the previous example calculated for the pitched cambered beam with variable height we have seen that the design ratio of the program is slightly higher than the results of the manual calculations. This is due to the fact that the designs are calculated by subdividing the beam into a multitude of x-locations. The number of these locations and thus the accuracy of results can be set by the number of member divisions defined in window 1.4 *Control Parameters*.

The procedure as well as the advantages and disadvantages of this calculation method are now described for a fish beam (*Beam Type 8*).

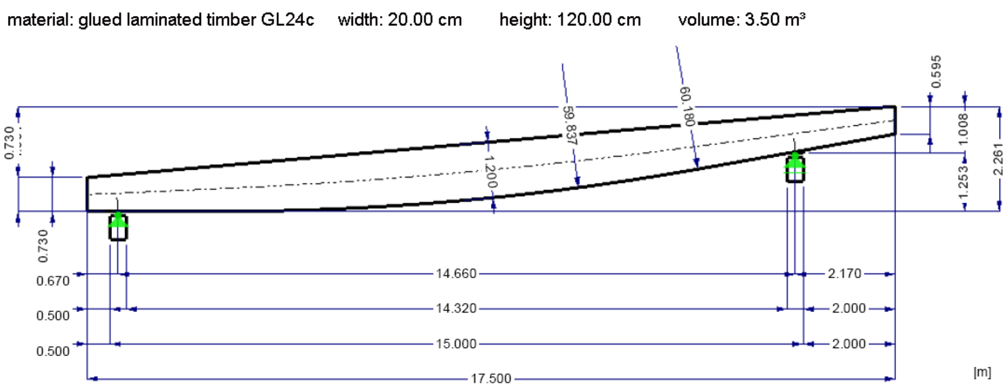


Figure 10.12: Geometry of fish beam

10.2.1 Geometry

The *Beam Type 8* presented in this example has a linear rounding in the center area with a straight line l_1 directed to both supports. The beam height on the support is generated on the inner edge of the support and must be identical on the left and the right support.

When generating the geometry the program reads the beam's geometrical conditions and checks if it is possible to integrate the minimum radius of the bottom edge according to [1] annex H as well as [7] clause NA11.4 into the inclination of the straight line l_1 depending on the lamella thickness: If the straight line's inclination becomes too steep or flat because of the height of the ridge and thus the radius cannot be integrated anymore into the beam, it is not possible to enter an unacceptable value.

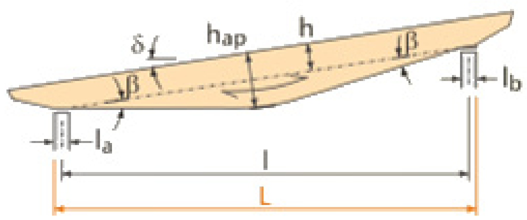


Figure 10.13: Definition parameters

10.2.2 Lateral Buckling Design

Generating loads is not described because it is already done in the previous example. To compare the results, we recommend to use the model *08.gl* that is automatically created during the program installation. For the lateral buckling analysis we only look at result combination 2 including loads from roof structure and snow.

RC2 = 1.35 · LC1 + 1.50 · LC41

In window 1.4 *Control Parameters* we select **20 divisions** for the truss resulting in a relatively accurate member division. Then, we click the [Calculation] button.

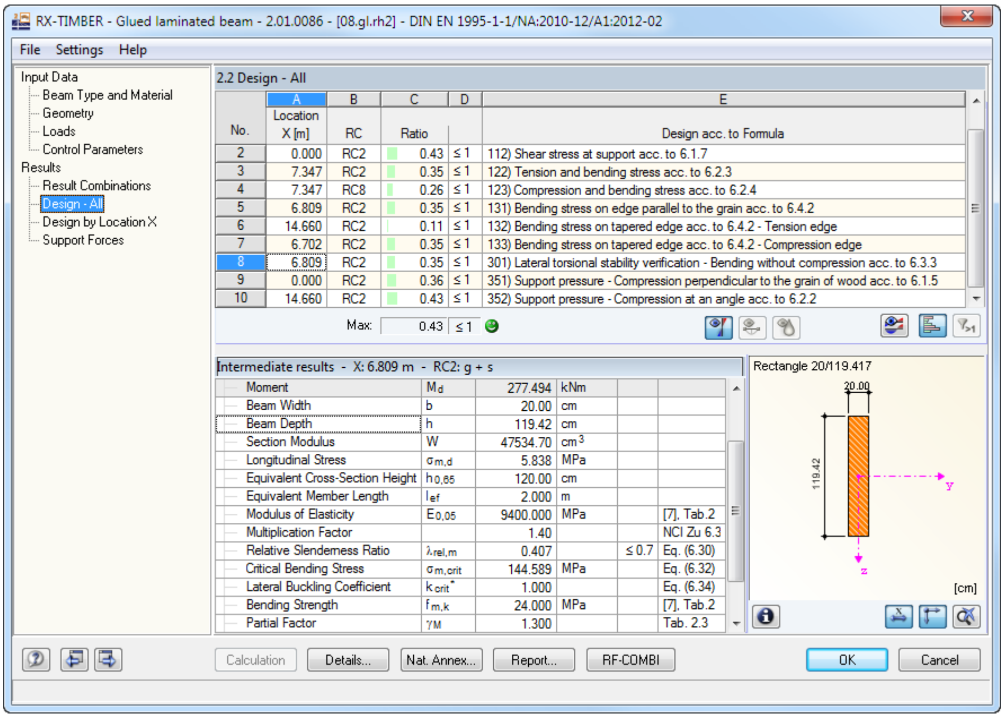


Figure 10.14: Window 2.2 *Design - All* for ultimate limit state design

For the X-location 6.809 m the program's output displays a design ratio of 35 % in the lateral torsional stability verification. The relative slenderness ratio is shown with $\lambda_{rel,m}$ = 0.375. Furthermore, the program calculates 120 cm for the equivalent cross-section height $h_{0.65}$. The equivalent height for the slenderness ratio is assumed by RX-TIMBER, as described in chapter 10.1.4, with the height that is available in a distance of 65 % of the entered equivalent member length from the member end with the minor cross-section (see DIN 1052:2008-12, 8.4.3 (3), 8.4.3 (4) and 8.4.4 (2)).

The equivalent member length for this beam was determined with 2.00. In hand calculations we would now determine the height on each point where the length for lateral buckling is reduced. With this height value we would determine the stress for the point. The lateral buckling factor, for example in the design (6.34) according to DIN EN 1995-1-1, is determined with an equivalent height in the distance of 0.65 · l_{ef}.

For the manual calculations we use the following values:

X-location	= 6.81 m
Height of cross-section	= 1.19 m
Equivalent height $h_{0.65}$	= 1.20 m
Flexural resistance $f_{m,k}$	= 24 N/mm^2 (GL24c)
Beam width b	= 180 mm

Relative slenderness according to eq. (6.30)

$$\lambda_{rel,m} = \sqrt{\frac{f_{m,k}}{\frac{0.78 \cdot b^2}{h_{0.65} \cdot l_{ef}} \cdot E_{0.05} \cdot 1.4}} = \sqrt{\frac{2.4}{\frac{20^2 \cdot 0.78}{120 \cdot 200} \cdot 940 \cdot 1.4}} = 0.375 \leq 0.7$$

Lateral buckling factor $k_{crit} = 1$

Design value of strength

$$f_{m,d} = f_{m,k} \cdot \frac{k_{mod}}{\gamma_M} = 24 \cdot \frac{0.9}{1.3} = 16.62 \frac{N}{mm^2}$$

Section modulus on X-location = 6.81 m

$$W_y = \frac{b \cdot h^2}{6} = \frac{20 \cdot 119.7^2}{6} = 47203\,cm^3$$

Stress on X-location = 6.81 m

$$\sigma_{m,d} = \frac{M_{y,d}}{W_y} = \frac{27749\,kNcm}{47203.3\,cm^3} = 0.588 \frac{kN}{cm^2} = 5.88 \frac{N}{mm^2}$$

Design ratio

$$\frac{\sigma_{m,d}}{k_m \cdot f_{m,d}} = \frac{5.88}{1 \cdot 16.62} = 0.35 < 1$$

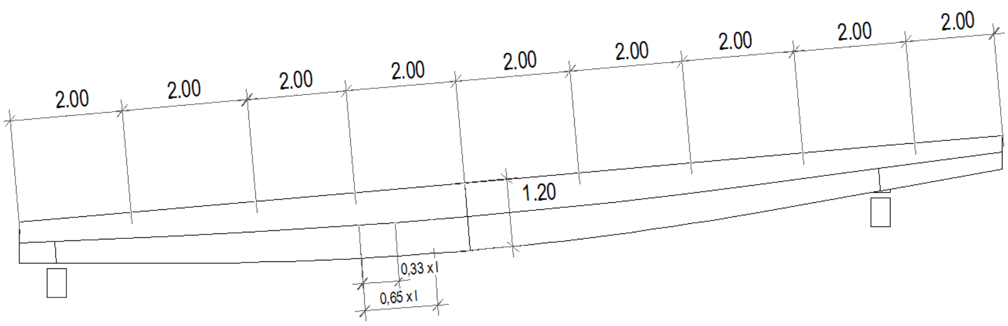
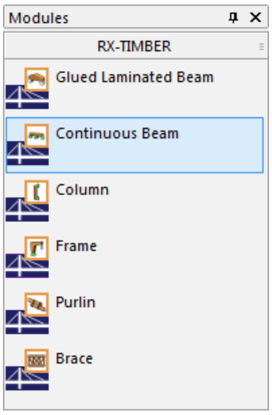


Figure 10.15: Equivalent height in manual calculations

We see in the figure above that the program, compared to the manual calculations, does not calculate with the 0.65-fold but the 0.33-fold effective member length.

Among other things, this is due to the fact that a program is not able to see if the division into X-locations respectively the resulting equivalent height leads to a useful result. Otherwise, it could happen in the ridge zone of the beam that the equivalent height is selected behind the ridge and thus calculated too low.

11. Continuous Beam



The following example describes the designs according to DIN EN 1995-1-1:2010 presenting a two-span beam with span lengths of 4.00 m and 5.00 m.

The model data is stored in the model *07.dlt* of the project *Examples RX-TIMBER*. We find it among the example files of the program **Continuous Beam**.

11.1 System and Loads

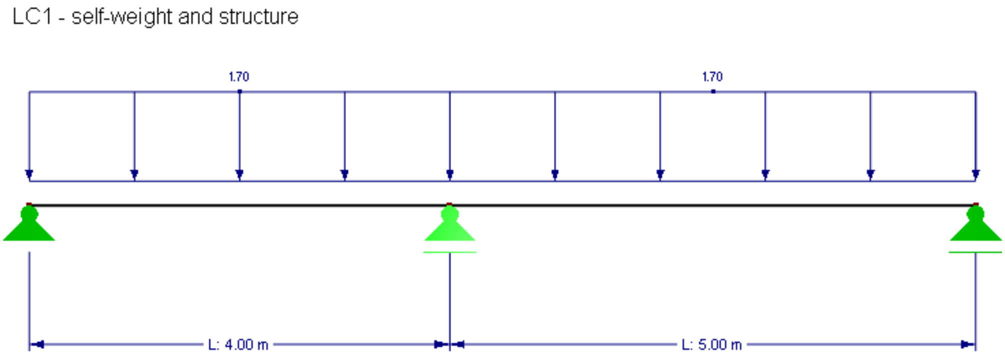


Figure 11.1: System and loads

Model

Cross-section: b/d = 70/240 mm NKL 1 → $k_{def} = 0.6$

Material: CT C24

The floor is built above living space. This results in the imposed load category **A** and the LDC **medium-term**.

Span lengths: $l_1 = 4.00\text{m}$ $l_2 = 5.00\text{m}$

Loads

Load case 1: self-weight and roof finishes g = 1.7 kN/m LDC = permanent

Load case 2: imposed load q = 2.0 kN/m LDC = medium-term

Result combinations

Load bearing capacity RC 1 = 1.35 · LC1 = 2.3 kN/m

RC 2 = 1.35 · LC1 + 1.5 · LC 2 = 5.3 kN/m

Serviceability

- **Characteristic situations**

  Restriction of deflection: $w_{inst} = w_{g,inst} + w_{q,1,inst} + \sum \psi_{0,i} \cdot w_{q,i,inst} \leq l/300$

  $G_C = g \cdot 1.0 + p \cdot 1.0 = 1.7\,\text{kN/m} + 2.0\,\text{kN/m} = 3.7\,\text{kN/m}$

- **Quasi-permanent design situation**

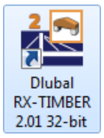
  $$w_{net,fin} = w_{g,inst} \cdot (1 + k_{def}) + \sum_{i \geq 1} \psi_{2,i} \cdot w_{q,i,inst} \cdot (1 + k_{def}) - w_c \leq l/300$$

  $$GQ = (1 + k_{def}) \cdot g + \psi_{2,i} \cdot p \cdot (1 + k_{def}) = (1 + 0.6) \cdot 1.7\,\text{kN/m} + 0.3 \cdot 2\,\text{kN/m} \cdot 1.6 = 3.68\,\text{kN/m}$$

Comparing the results from the manual calculations with the ones calculated by the program is easier when we don't take into account the beam's self-weight.

11.2 Input of Model Data

11.2.1 General Data



Start the program by double-clicking the desktop icon **Dlubal RX-TIMBER 2.xx** (see chapter 3.2, page 12), and then select **Continuous Beam** in the Project Manager.

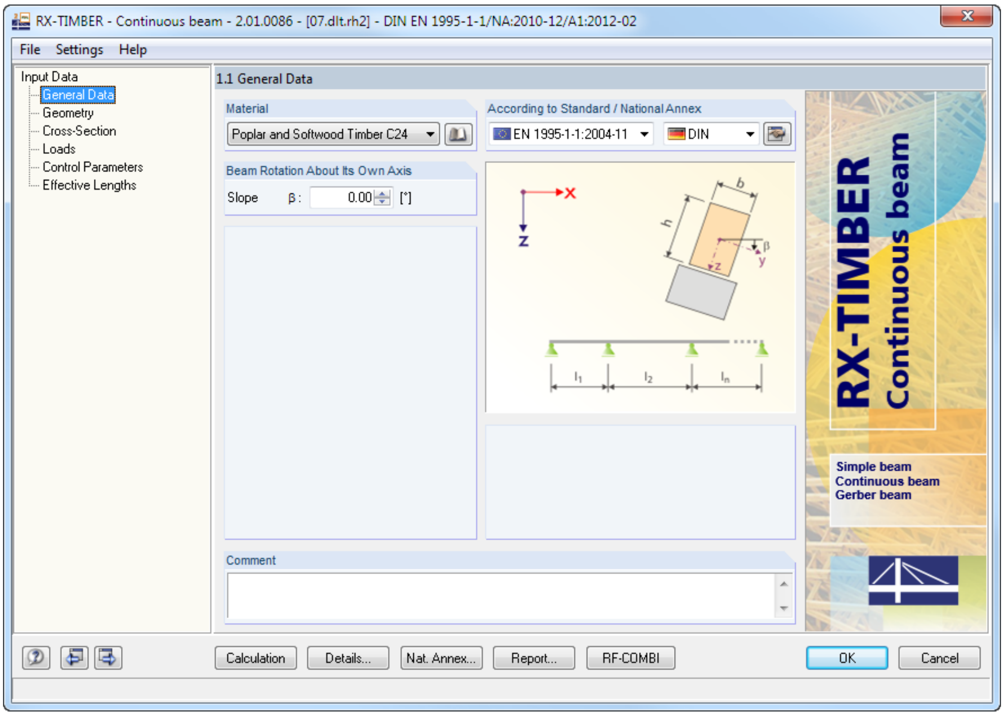


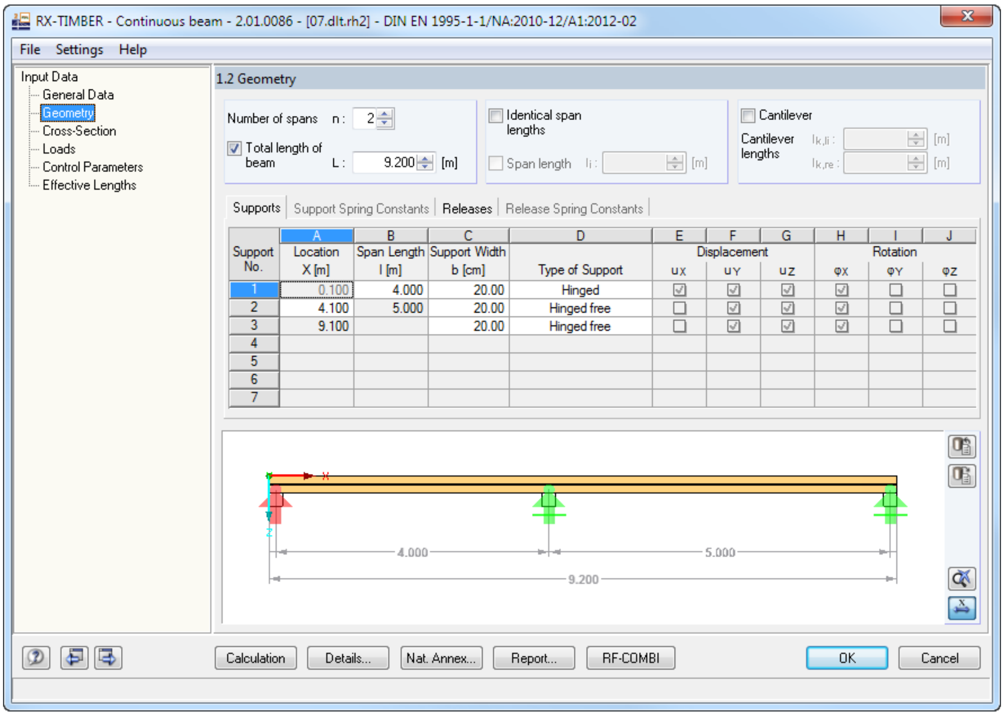
Figure 11.2: Window 1.1 *General Data*

We select **softwood timber** with the strength grade **C24** as *Material*.

The *Beam Rotation* about its own axis is set to β = **0°**, we don't modify it.

11.2.2 Geometry

In the second window, we enter the number of spans, the span lengths as well as the support and release conditions of the beam. Optionally, we can define cantilevers or assign user-defined spring constants to the supports and releases.



Support No.	A Location X [m]	B Span Length l [m]	C Support Width b [cm]	D Type of Support	E Displacement uX	F uY	G uZ	H Rotation φX	I φY	J φZ
1	0.100	4.000	20.00	Hinged	☑	☑	☑	☑	☐	☐
2	4.100	5.000	20.00	Hinged free	☐	☑	☑	☑	☐	☐
3	9.100		20.00	Hinged free	☐	☑	☑	☑	☐	☐
4										
5										
6										
7										

Figure 11.3: Window 1.2 *Geometry*

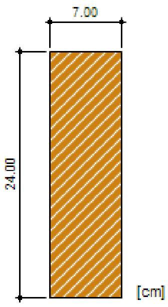
For our example we set:

- *Number of spans n*: **2**
- *Total length L*: **9.20 m**
- Two spans with *Span Length l*: **4.00 m** and **5.00 m**
- *Support Width b*: **20 cm**

The first *Location X* of **0.10 m** results from the support width.

For the *Type of Support* we select a *Hinged* support for *Support No. 1* and two *Hinged free* supports for *No. 2* and *3* in direction X. By using the *User-defined* list option it would be possible to assign any individual degree of freedom as well as translational and rotational spring stiffnesses to the support.

11.2.3 Cross-section



In the third window, we define cross-section dimensions and tapers, if applicable, that are adjusted to the moment diagrams. In the window section *Cross-section Changes*, we can describe the beam by specifying the length and height as well as the inclination of the change.

For the *Cross-Section* we enter the *Dimensions* **b = 7 cm** and **h = 24 cm**. The *Origin Position* is set for the **Beam Start**. We don't modify any of the remaining values (see Figure 11.4).

Below the table we see information about *Area*, *Volume* and *Weight* of the beam. We can display the beam's cross-section properties like the elastic section modulus etc. by clicking the [Info] button shown on the left.

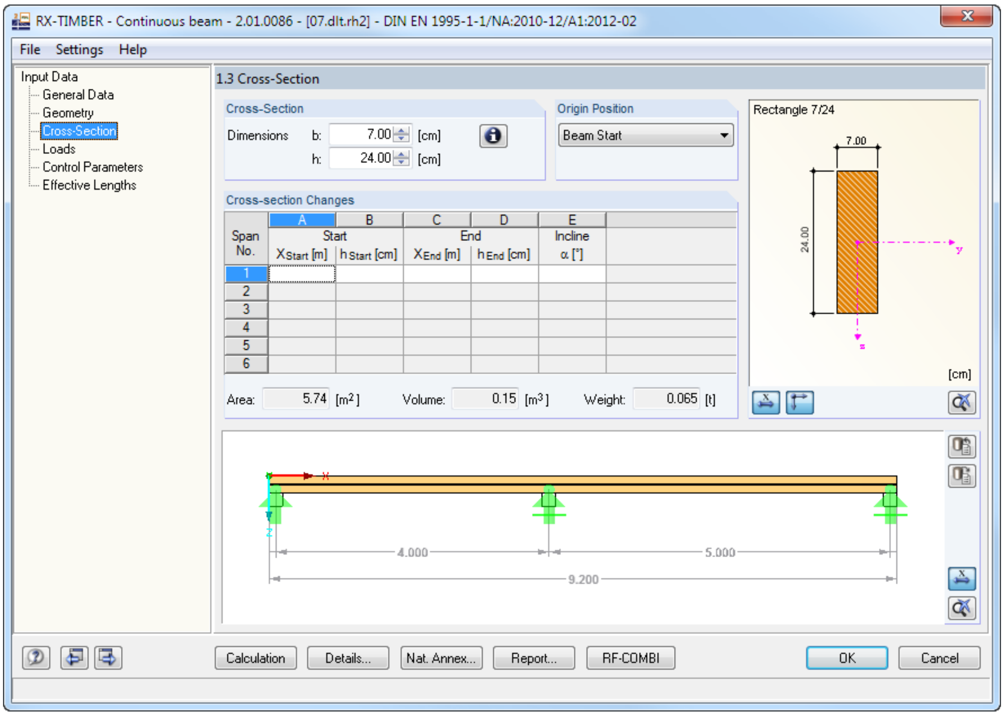


Figure 11.4: Window 1.3 *Cross-Section*

11.2.4 Loads

In the *Loads* table we enter the different load types. The program provides uniformly distributed loads, block loads, linearly variable loads, concentrated loads and moments. These loads can be entered separately for the directions x, y and z in the dialog box *Load Cases* (see Figure 11.6).

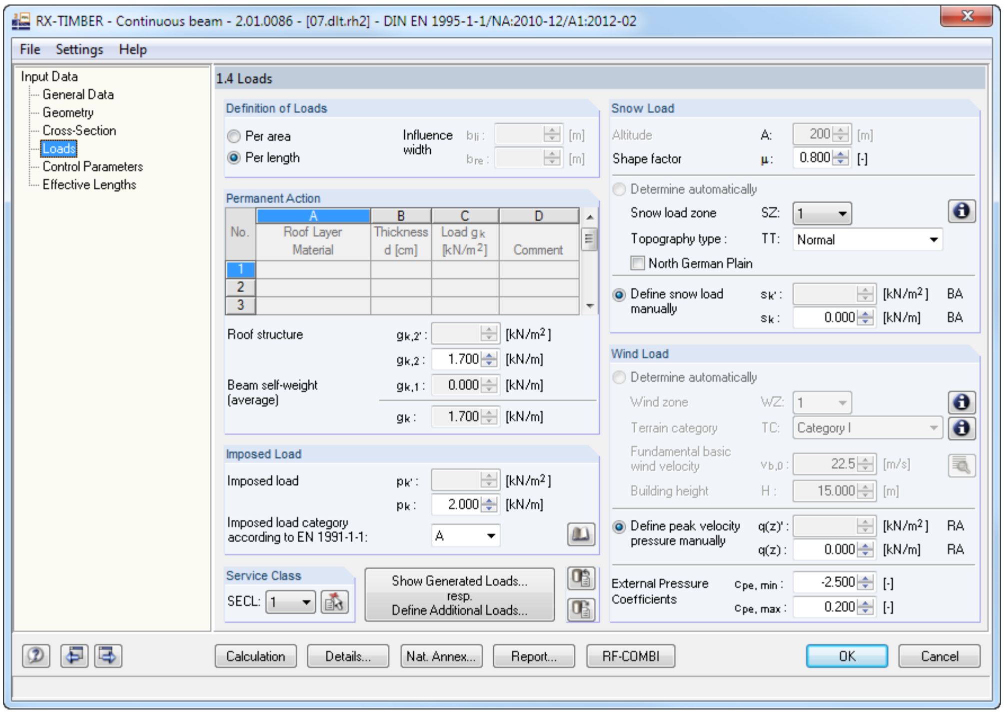


Figure 11.5: Window 1.4 *Loads*

In our example, we select a load per length. We enter the loads as shown in the figure above. We set the wind and snow loads manually to zero.

We use the button [Show Generated Loads] shown on the left to open the dialog box *Load Cases* where we can check the generated loads.

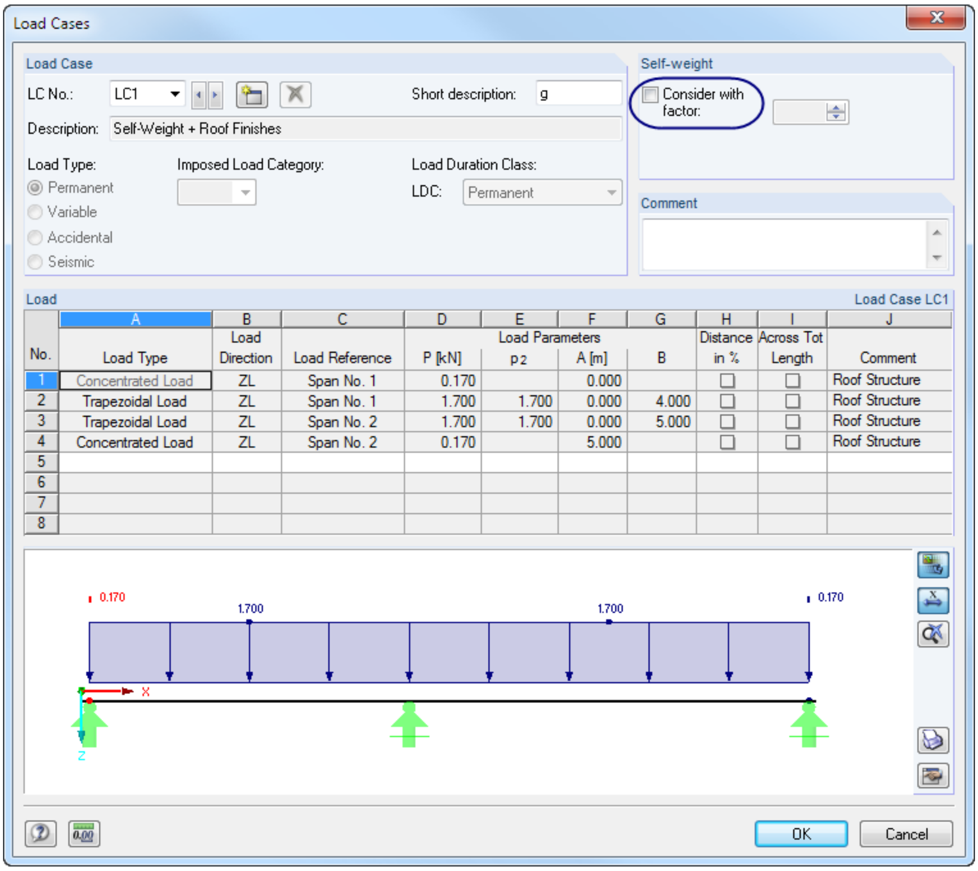


Figure 11.6: Dialog box *Load Cases*

The program generates a concentrated load resulting from the line load beyond the support axis, preventing a negative moment on the short overhang of the beam.

We deactivate the *Self-weight* for the calculation by clearing its check box.

11.2.5 Control Parameters

In window 1.5 *Control Parameters* (see Figure 11.7) we define the designs for the *Ultimate limit State*, the *Serviceability limit state* and the *Fire protection*. For the serviceability limit state design we can specify a *Precamber*. We do without the designs for the fire protection and deactivate the corresponding option.

In the window section *To Display*, we tick the check boxes of all options: *Support forces*, *Deformations* and *Support compression*.

We don't want to perform a *Moment redistribution* for non-used cross-section zones according to DIN EN 1990, clause 1.5.6.4.

The *Number of member divisions* determines the locations where RX-TIMBER performs the designs.

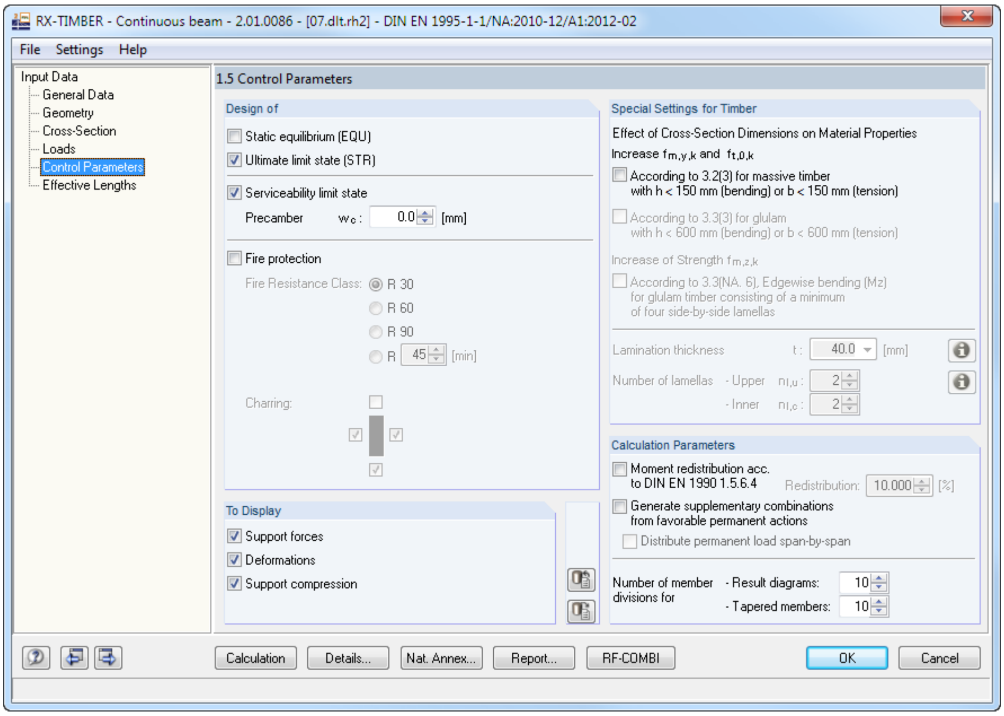


Figure 11.7: Window 1.5 *Control Parameters*

We use the [Nat. Annex] button shown on the left to open the dialog box *National Annex Settings* where we can check the partial safety factors and the limit values of the deformations.

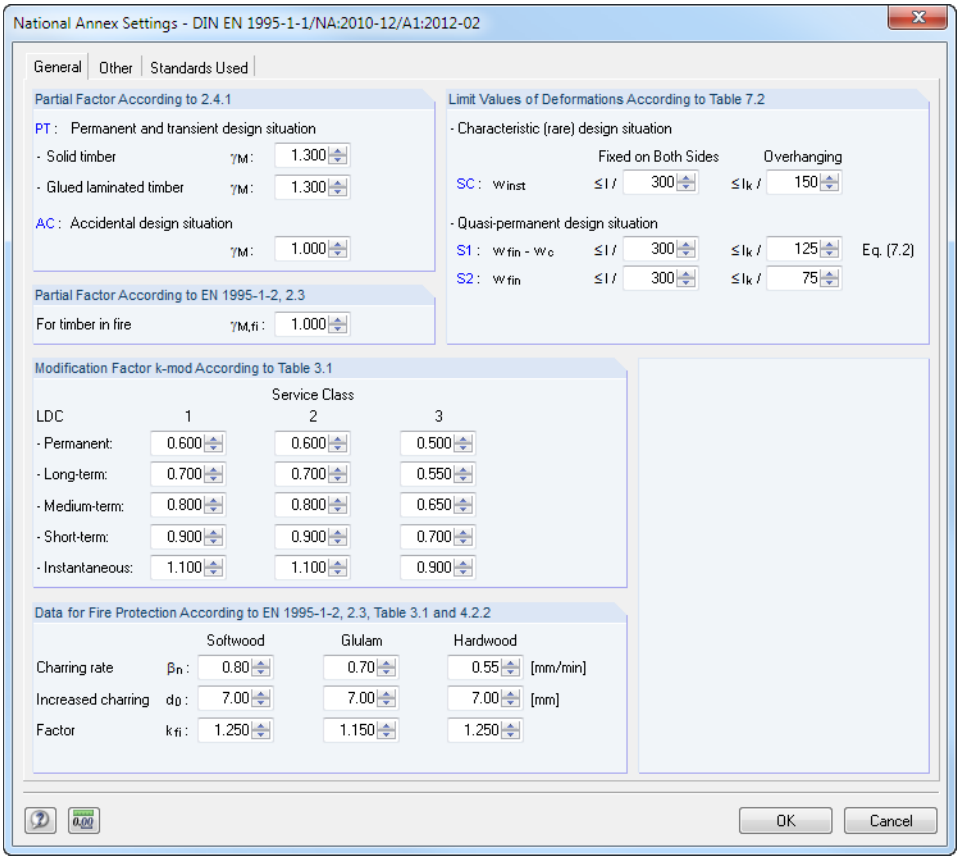


Figure 11.8: Dialog box *National Annex Settings*

The deformation can be defined independently of span and cantilever.

11.2.6 Effective Lengths

To be on the safe side, we set factor β to **1.0** in our example. The remaining values are preset by the program so that we don't need to enter any further data in this table.

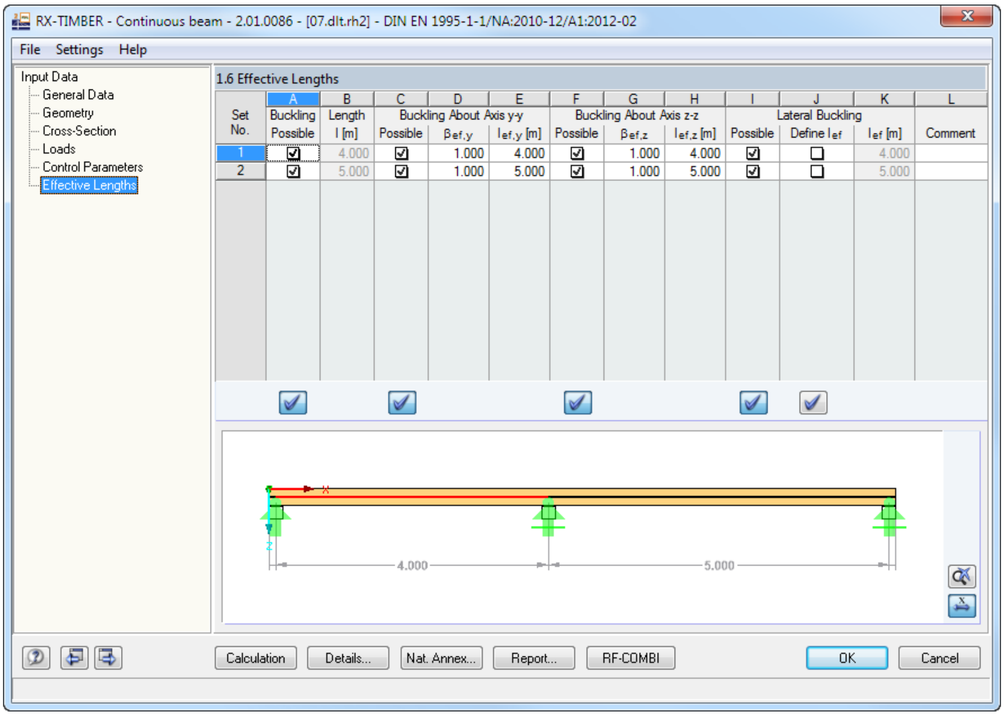


Figure 11.9 Window 1.6 *Effective Lengths*

11.3 RF-COMBI

The add-on module RF-COMBI used for the creation of combinations according to DIN EN 1990 and DIN EN 1995-1-1 is integrated in RX-TIMBER: The load cases are combined automatically so that we normally don't need to open the add-on module. However, we want to check the combinations that are generated in the background. We access the add-on module by using the [RF-COMBI] button.

We set *CA2* in the case list above the RF-COMBI navigator and look at the combinatorics of *Serviceability*.

The *Deformation coefficient* is defined automatically with $\mathbf{k_{def} = 0.6}$ in accordance with the respective service class. In our example, we have specified the service class 1.

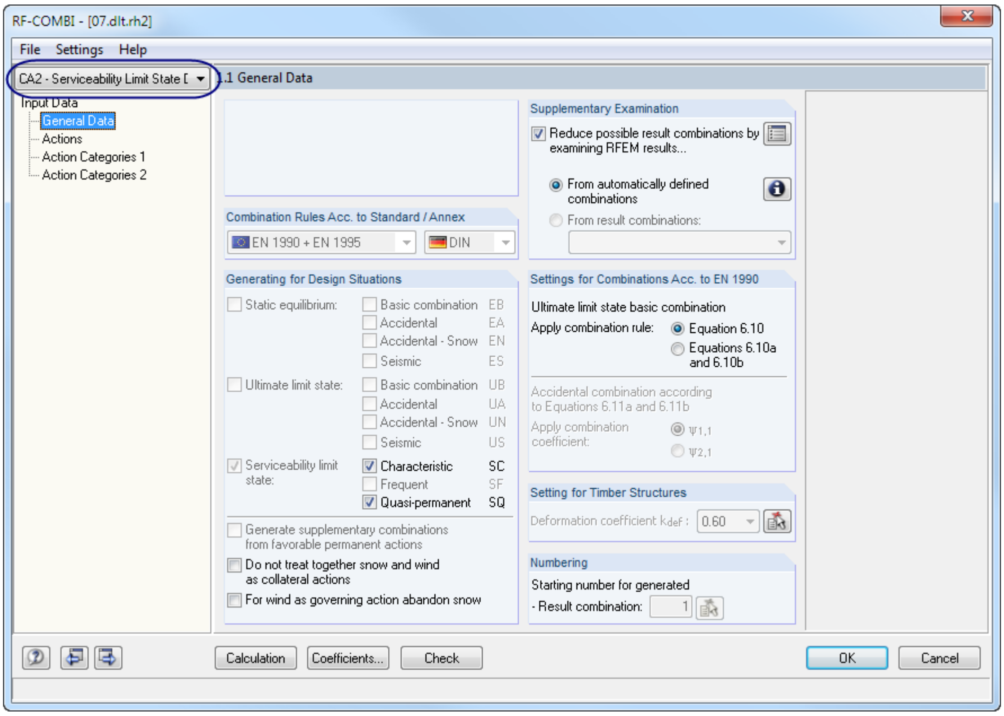


Figure 11.10: Add-on module *RF-COMBI*, case 2 *Serviceability Limit State Design*

In the next window 1.2 *Load Cases in Actions*, the loads are assigned to the relevant actions, and so the LDC is created.

For the imposed load the program already takes into account the load-duration class according to action *A* and the load duration *medium-term*.

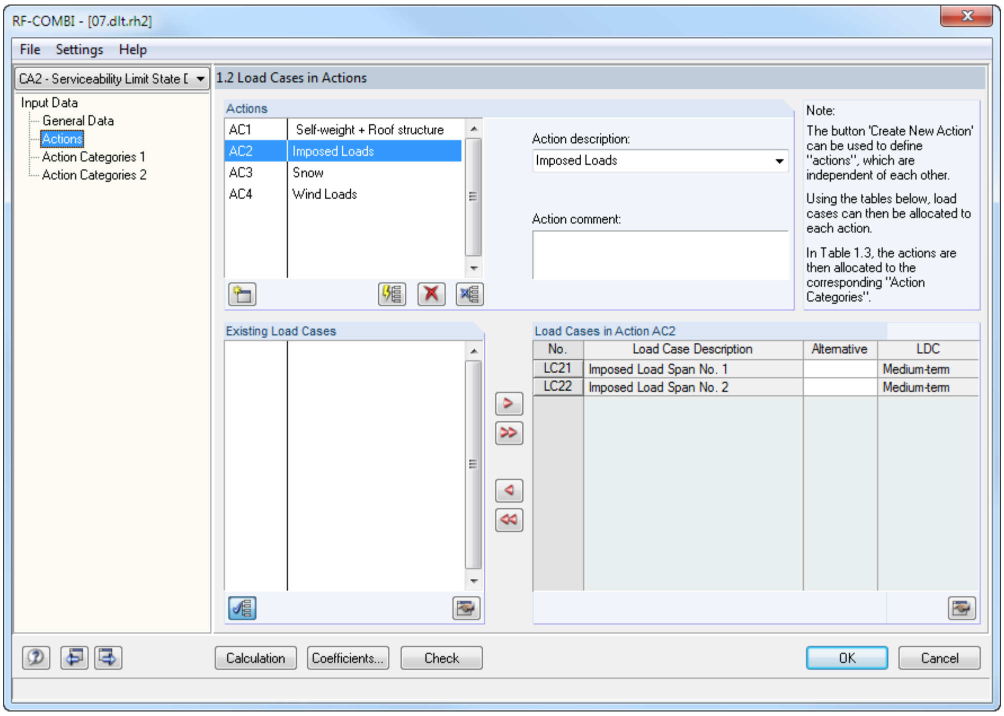


Figure 11.11: RF-COMBI window 1.2 *Load Cases in Actions*

For more information on the features of RF-COMBI, see the manual available for download at www.dlubal.com. For our simple example, the actions and respective load-duration classes have automatically been created correctly.

The used [Coefficients] can be checked in a separate dialog box.

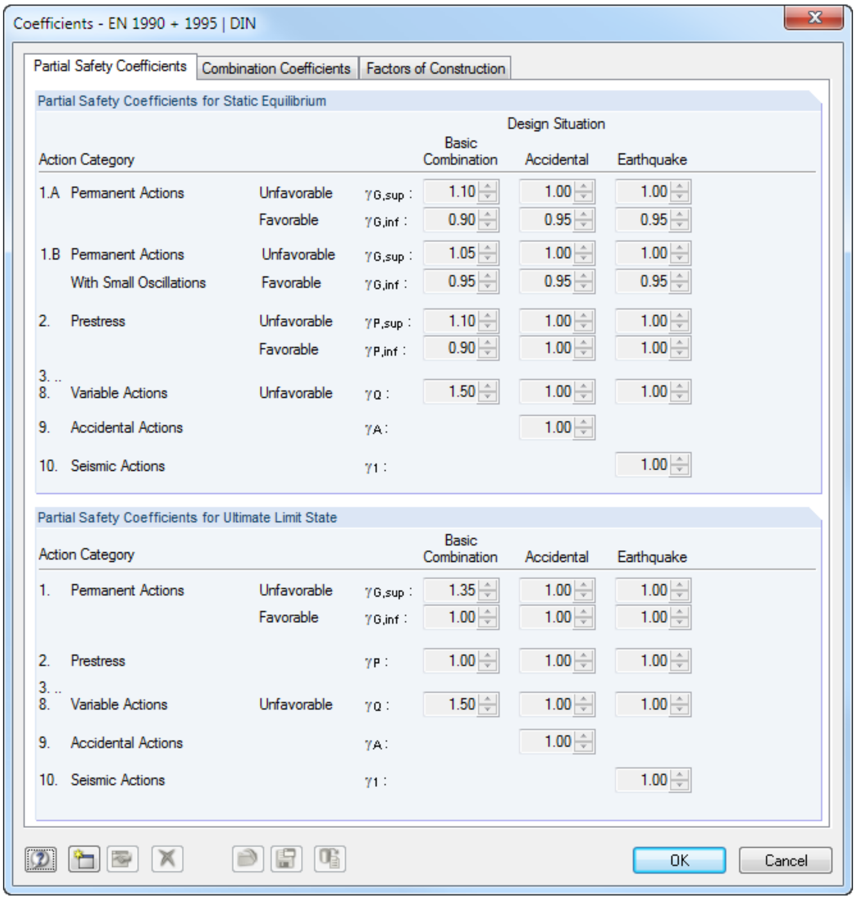


Figure 11.12: Dialog box *Coefficients*, tab *Partial Safety Coefficients*

After the [Calculation] we can check the results of the combinatorics by our hand calculations (see chapter 11.1, page 107). The calculation is performed correctly (see Figure 11.13).

The increased number of combinations in RF-COMBI is a result of the additionally required combination of the loads applied by span on the continuous beam.

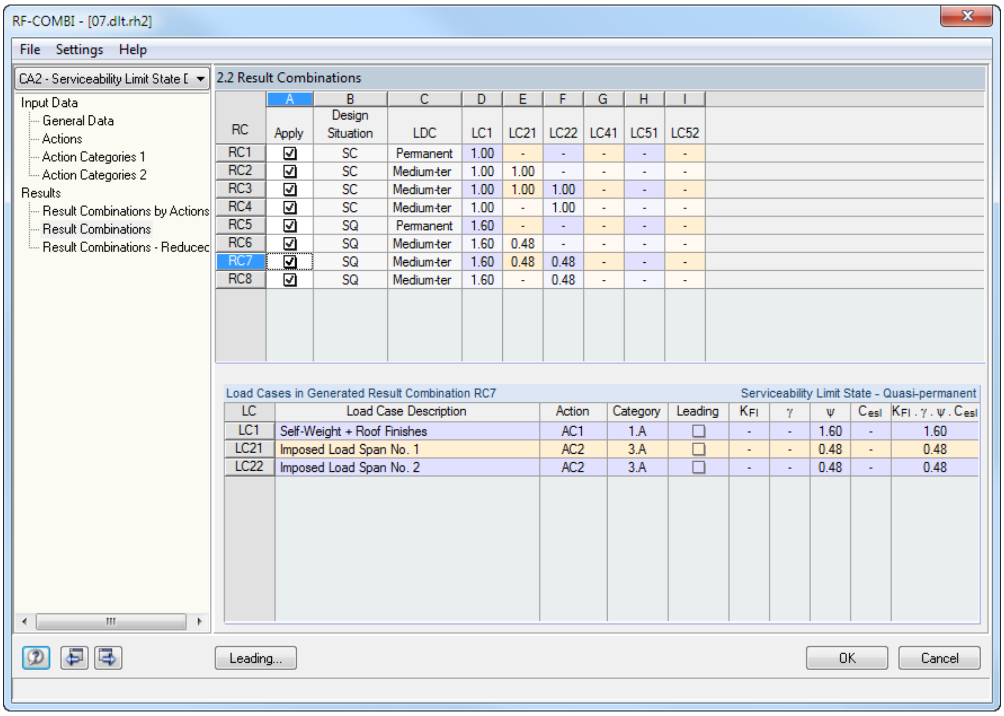


Figure 11.13: RF-COMBI window 2.2 *Result Combinations*

We have the possibility to select particular combinations for the calculation in the timber program *Continuous Beam*. However, this time we want to consider all combinations of window 2.2 for our example.

We click [OK] to return to *RX-TIMBER Continuous Beam*.

11.4 Results

11.4.1 Result combinations

We use the [Calculation] button to calculate the data. Results window 2.1 displays the designs of all combinations for the ultimate and the serviceability limit state including corresponding design ratio.

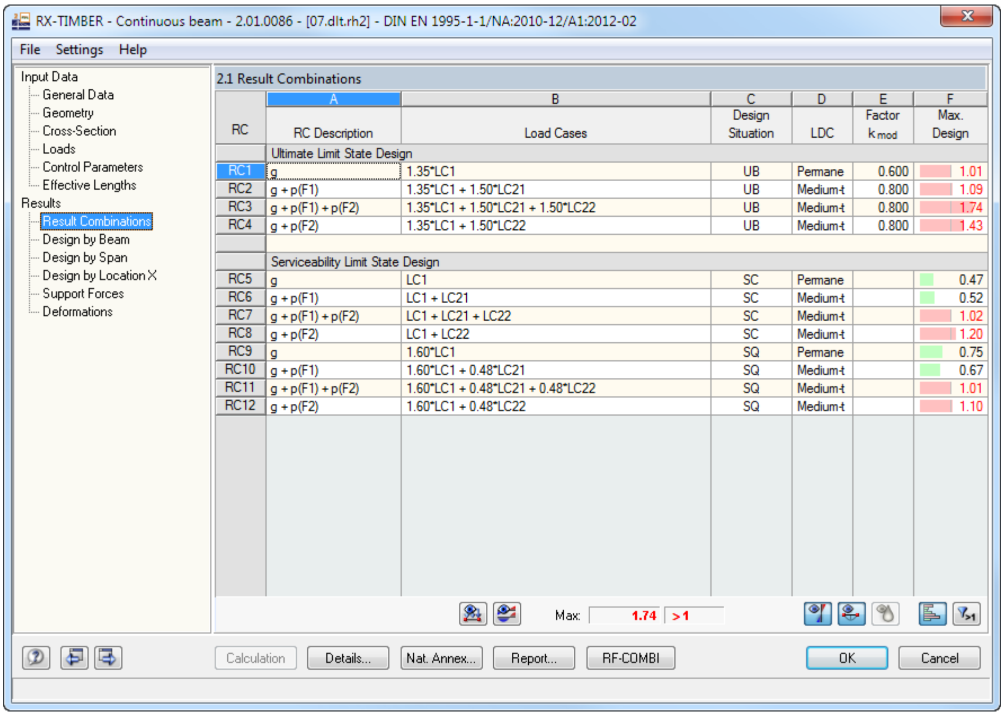


Figure 11.14: Window 2.1 *Result Combinations*

The factor k_{mod} used to consider the time-dependent settlement behavior due to load duration and specific climatic characteristics of timber is applied automatically to the required value according to the LDC and the SECL.

To check only the overstressed designs, we use the button [Shows only rows with ratios >1] which is available in each results window of RX-TIMBER. We click the button [Colored Relation Scale] to display and hide the colored representation of the design ratios.

We use the button [Load of Load Cases] to check the loads and the classes of the load actions that have been applied to the current result combination (that is the RC table row where the cursor is placed).

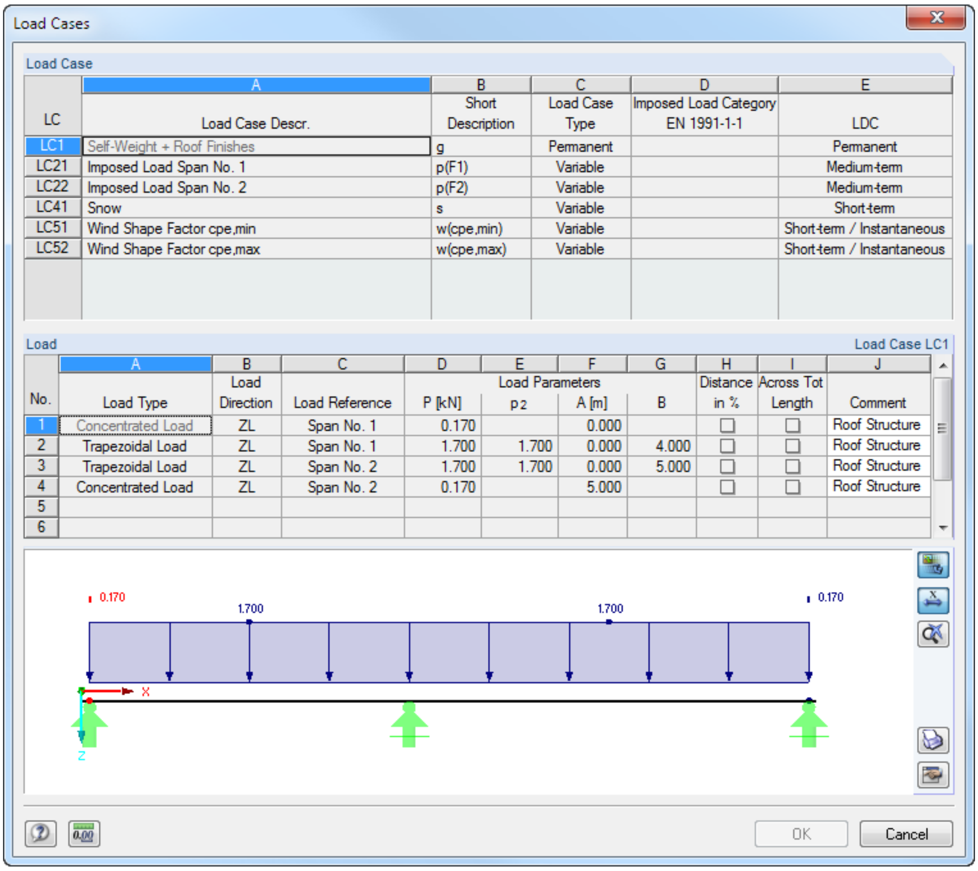


Figure 11.15: Dialog box *Load Cases*

11.4.2 Design by Beam

This window shows all governing designs for the entire beam.

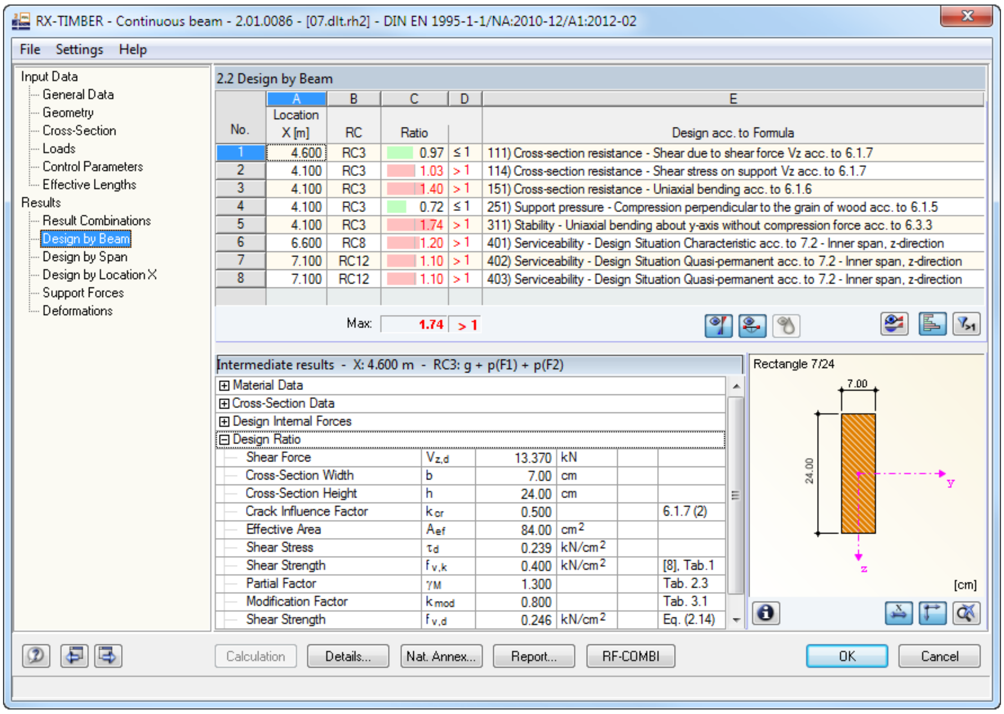


Figure 11.16: Window 2.2 *Design by Beam*

All factors and required design internal forces can be checked interactively: We click the relevant design in the table above to display all *Intermediate results* of the design in the window section below.

With the button [Result Diagrams] shown on the left we can check the distribution of internal forces and design ratios graphically. We find the button above the cross-section graphic. A new window opens where we can select the internal forces and deflections of the beam specifically for each result combination. The result combination can be set in the list above.

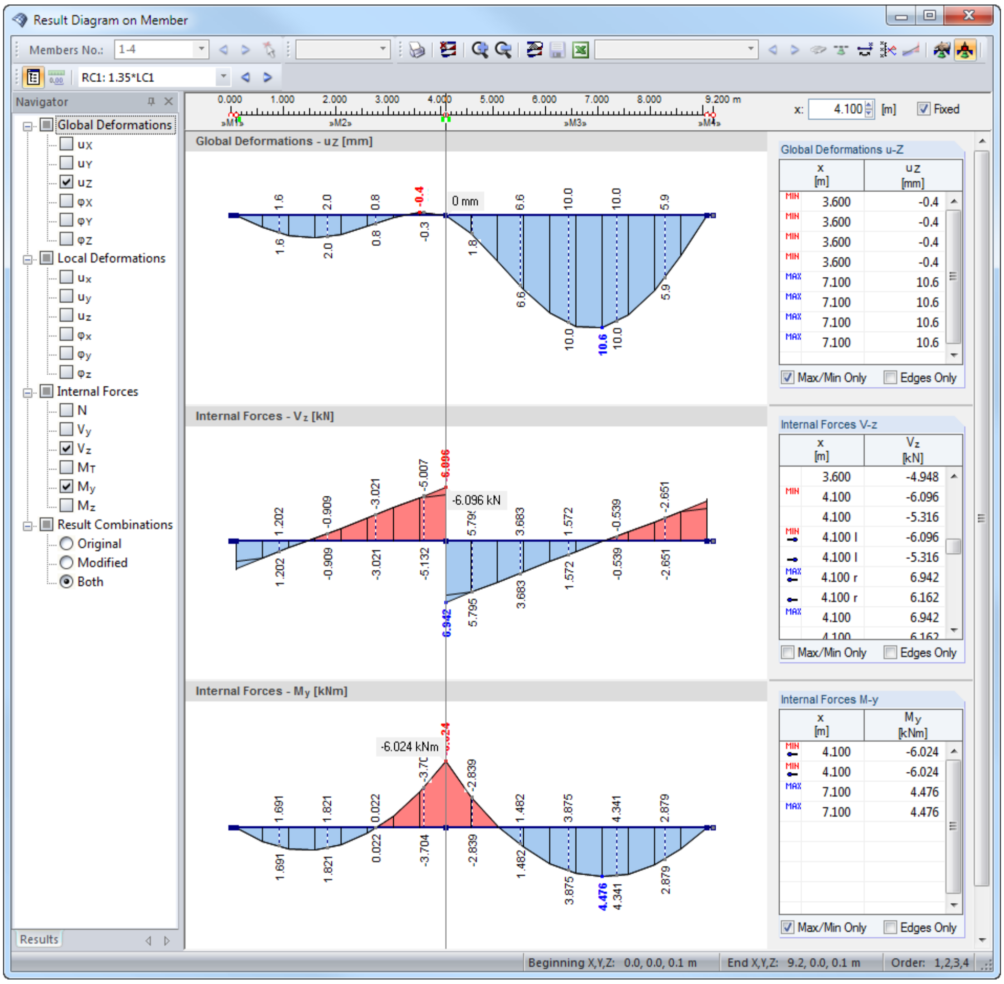


Figure 11.17: Dialog box *Result Diagram on Member*

Under the navigator entry *Result Combinations*, we can select the **Both** option to look at the reduction of forces according to the settings in the *Details* dialog box. This option is a useful function to evaluate for example the effect of the shear force reduction.

We use the [Print] button to transfer this graphic directly to the printout report.

11.4.3 Ultimate Limit State

In window 2.2 *Design by Beam*, it is also possible to display the designs only for the ultimate limit state: We disable the display for the serviceability limit state designs by clicking the button [Serviceability Limit State Designs].

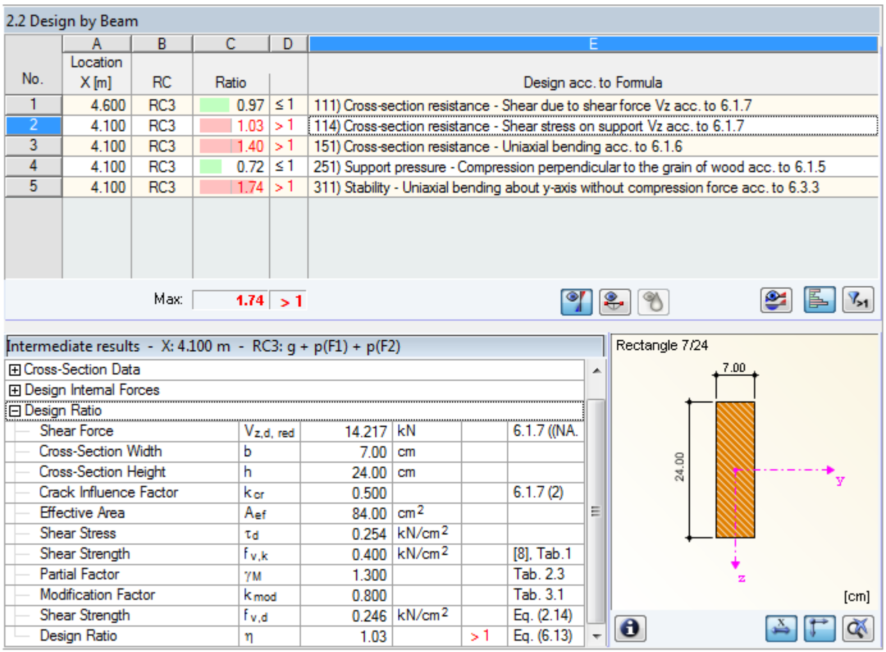


Figure 11.18: Window 2.2 *Design by Beam* for ultimate limit state only

The results of RX-TIMBER are checked by manual calculations.

Loading due to shear force on support according to 6.1.7

The maximum shear loading of 16.02 kN is determined above the central support. By means of the reduction according to 6.1.7 (NA.5) it is possible to reduce the loading to 14.2 kN and thus to the shear force occurring in a distance of 24 cm (= h) from the support.

Shear stress $$\tau_d = 1.5 \cdot \frac{V_d}{b \cdot k_{cr} \cdot h} = 1{,}5 \cdot \frac{14.2\ \text{kN}}{3.5\ \text{cm} \cdot 24\ \text{cm}} = 0.254\ \text{kN/cm}^2 = 2.54\ \text{N/mm}^2$$

Shear strength $$f_{v,d} = \frac{k_{mod} \cdot f_{v,k}}{\gamma_m} = \frac{0.8 \cdot 4\ \text{N/mm}^2}{1.3} = 2.46\ \text{N/mm}^2$$

Design ratio $$\frac{\tau_d}{f_{v,d}} = \frac{2.54}{2.46} = 1.03 > 1$$

Despite the reduction of the shear force, the design is not fulfilled. If glued-laminated timber was used, the design could be fulfilled.

Bending load according to 6.1.6

The maximum loading is available above the support center due to the supporting moment.

Bending stress $$\sigma_{m,y,d} = \frac{M_{d,max}}{W} = \frac{13.9\ \text{kNm} \cdot 10^{-3}}{672\ \text{cm}^3 \cdot 10^{-6}} = 20.68\ \text{N/mm}^2$$

Flexural resistance $$f_{m,d} = \frac{0.8 \cdot 24\ \text{N/mm}^2}{1.3} = 14.77\ \text{N/mm}^2$$

Design ratio $$\frac{\sigma_{m,y,d}}{f_{m,d}} = \frac{20.68}{14.77} = 1.40 > 1$$

Even if a moment reduction of 10 % was applied, which is not done in this case, the design ratio would be determined with 126 % for a design moment of 12.5 kNm. Only two options remain: selecting either a higher strength grade, for example C 35, or a larger cross-section.

Support compression according to 6.1.5

Like we expected the maximum support compression is reached at the central support with a loading of 30 kN. RX-TIMBER increases automatically the contact area by the allowed 3 cm along the timber grain. Thus, for the selected support with a width of 20 cm we get an effective area of 182 cm².

The maximum loading for this design is also determined above the support center due to the supporting moment.

Transversal compression stress $\sigma_{c,90,d} = \frac{F_{c,90,d}}{A_{ef}} = \frac{30\ \text{kN}}{182\ \text{cm}^2} = 0.165\ \text{kN/cm}^2 \approx 1.65\ \text{N/mm}^2$

Transversal compression strength $f_{c,90,d} = \frac{k_{mod} \cdot f_{c,90,d}}{\gamma_m} = \frac{0.8 \cdot 2.5\ \text{N/mm}^2}{1.3} = 1.54\ \text{N/mm}^2$

Design ratio $\frac{\sigma_{c,90,d}}{k_{c,90} \cdot f_{c,90,d}} = \frac{1.65}{1.5 \cdot 1.54} = 0.71 < 1$

Flexural members without compressive force according to 6.3.3 (lateral buckling analysis)

Additionally, we have to perform the stability analysis for this beam. The moment loading to be designed is identical with the beam's loading in the bending analysis within the ultimate limit state design.

Thus, bending stress and flexural resistance are identical with the design according to 10.2.6. Therefore, we don't describe it again.

Radius of gyration $i_z = 0.289 \cdot b = 0.289 \cdot 7\ \text{cm} = 2.02\ \text{cm}$

Slenderness $\lambda_{rel,m} = \sqrt{\frac{l_{ef}}{\pi \cdot i_z}} \cdot \sqrt{\frac{f_{m,k}}{\sqrt{E_{0.05} \cdot G_{05}}}} =$

$$= \sqrt{\frac{5.00\ \text{m}}{\pi \cdot 2.02\ \text{cm}}} \cdot \sqrt{\frac{24\ \text{N/mm}^2}{\sqrt{7333\ \text{N/mm}^2 \cdot 460\ \text{N/mm}}}} = 1.01$$

Lateral buckling factor $k_{crit} = 1.56 - 0.75 \cdot \lambda_{rel,m} = 1.56 - 0.75 \cdot 1.01 = 0.803$

Design ratio $\frac{\sigma_{m,d}}{k_m \cdot f_{m,d}} = \frac{20.68}{0.803 \cdot 14.77} = 1.74 > 1$

The overload to be expected for this high and slender beam could be reduced by reducing the buckling length. It would be possible to model a continuous lateral support in RX-TIMBER, which is represented in reality by a screwed-on OSB-boarding, by removing the buckling option in window 1.6 *Effective Lengths*.

11.4.4 Serviceability Limit State

In window 2.2 *Design by Beam*, we disable the display for the ULS designs by clicking the button [Ultimate Limit State Designs] so that we only see the serviceability limit state design results.

To eliminate the overloads from the ultimate limit state, we change the strength class of the material for this example from C24 to C35. Moreover, we calculate the beam as centrically fixed by defining the lengths for lateral buckling l_{ef} manually with **2.0 m** respectively **2.5 m** in window 1.6 *Effective Lengths*.

After the [Calculation] with the specified settings the results look much better.

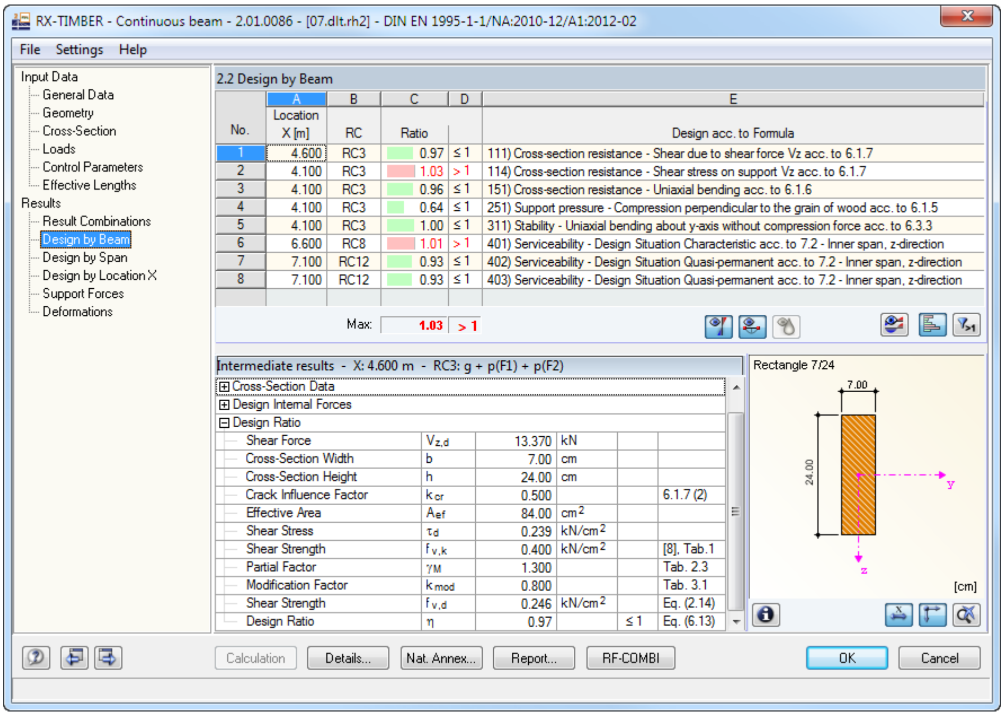


Figure 11.19: Window 2.2 *Design by Beam*

We perform the deflection analysis exemplarily for the quasi-permanent design situation. The loading from an imposed load becomes governing only in the right span.

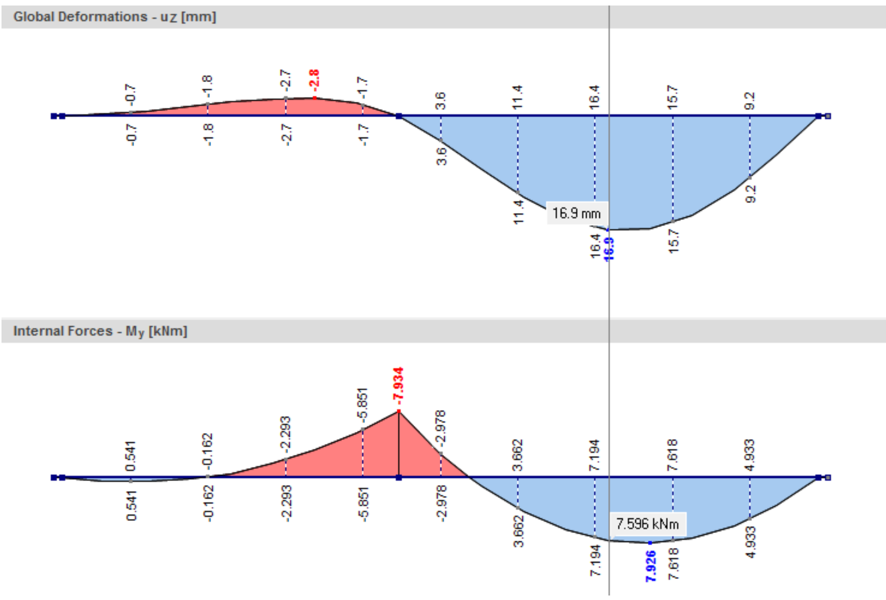


Figure 11.20: Result diagrams in RX-TIMBER for RC8

Loading according to SC (serviceability - characteristic) in right span:	3.7 kN/m
Loading according to SC in left span:	1.7 kN/m
Moment of span:	7.93 kNm
Supporting moment:	-7.93 kNm

Deflection

$$f = \frac{104 \cdot (7.93\ \text{kNm} + \frac{7.93\ \text{kNm}}{2}) \cdot 5.00^2\ \text{m}}{8064\ \text{cm}^4} + \frac{62.5 \cdot (-7.93\ \text{kNm}) \cdot 5.00^2\ \text{m}}{8064\ \text{cm}^4} = 2.299\ \text{cm}$$

To adjust the deflection design to the modified elastic modulus, we divide our result by 1.3 (E = 1.300 kN/cm^2) and receive a deflection of 1.77 cm.

For this design according to equation (42) RX-TIMBER shows us a deflection of **16.9 mm**.

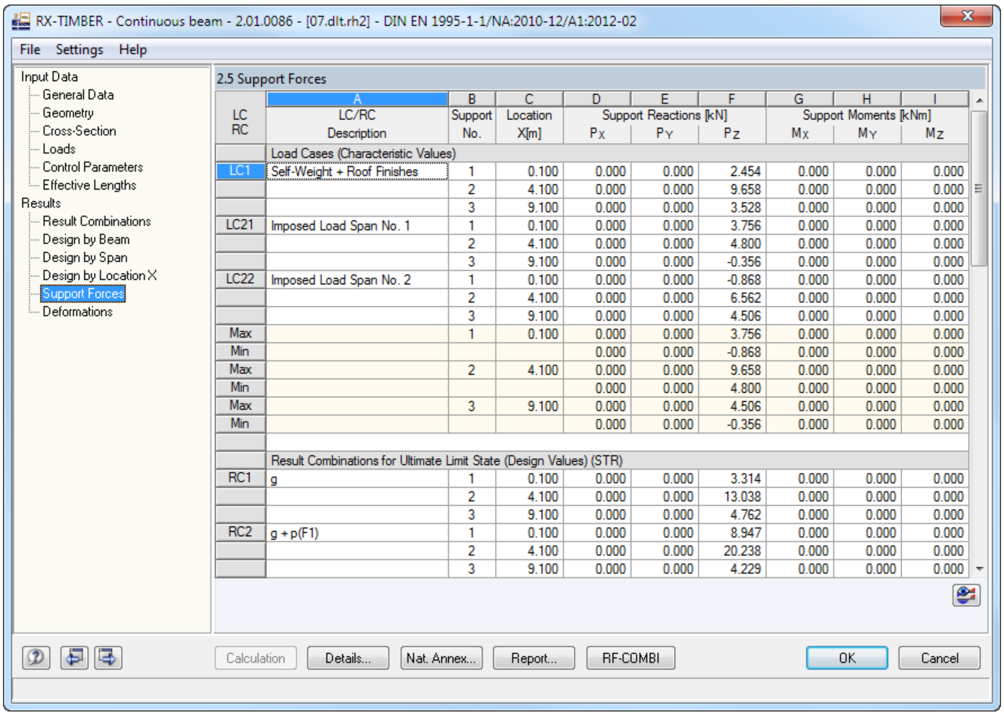
Thus, the deflection of 16.9 mm lies slightly above the limit value of

l/300 = 5000 mm/300 = 16.7 mm.

11.4.5 Support Forces

This window shows us the support forces of all load cases and result combinations. The support forces are listed separately for the design situations (ultimate limit state, serviceability limit state and, if available, fire protection).

In addition, the window presents the maximum and minimum values for each design situation.

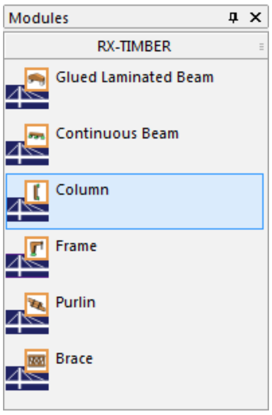


LC RC	A LC/RC Description	B Support No.	C Location X[m]	D Support Reactions [kN] Px	E Py	F Pz	G Support Moments [kNm] Mx	H My	I Mz
	Load Cases (Characteristic Values)								
LC1	Self-Weight + Roof Finishes	1	0.100	0.000	0.000	2.454	0.000	0.000	0.000
		2	4.100	0.000	0.000	9.658	0.000	0.000	0.000
		3	9.100	0.000	0.000	3.528	0.000	0.000	0.000
LC21	Imposed Load Span No. 1	1	0.100	0.000	0.000	3.756	0.000	0.000	0.000
		2	4.100	0.000	0.000	4.800	0.000	0.000	0.000
		3	9.100	0.000	0.000	-0.356	0.000	0.000	0.000
LC22	Imposed Load Span No. 2	1	0.100	0.000	0.000	-0.868	0.000	0.000	0.000
		2	4.100	0.000	0.000	6.562	0.000	0.000	0.000
		3	9.100	0.000	0.000	4.506	0.000	0.000	0.000
Max		1	0.100	0.000	0.000	3.756	0.000	0.000	0.000
Min				0.000	0.000	-0.868	0.000	0.000	0.000
Max		2	4.100	0.000	0.000	9.658	0.000	0.000	0.000
Min				0.000	0.000	4.800	0.000	0.000	0.000
Max		3	9.100	0.000	0.000	4.506	0.000	0.000	0.000
Min				0.000	0.000	-0.356	0.000	0.000	0.000
	Result Combinations for Ultimate Limit State (Design Values) (STR)								
RC1	g	1	0.100	0.000	0.000	3.314	0.000	0.000	0.000
		2	4.100	0.000	0.000	13.038	0.000	0.000	0.000
		3	9.100	0.000	0.000	4.762	0.000	0.000	0.000
RC2	g + p(F1)	1	0.100	0.000	0.000	8.947	0.000	0.000	0.000
		2	4.100	0.000	0.000	20.238	0.000	0.000	0.000
		3	9.100	0.000	0.000	4.229	0.000	0.000	0.000

Figure 11.21: Window 2.5 *Support Forces*

11.5 Documentation

The documentation in the printout report contains all performed designs and loadings. The elements of the report can be selected specifically and adjusted to individual needs. The printout report is described in detail in chapter 8.



12. Column

We perform the ultimate and the serviceability limit state design according to EN 1995-1-1 for a log wood column that is restrained and stressed by compression and bending.

The model data is stored in the model *02.clm* of the project *Examples RX-TIMBER*. We find it among the example files of the program **Column**.

12.1 System and Loads

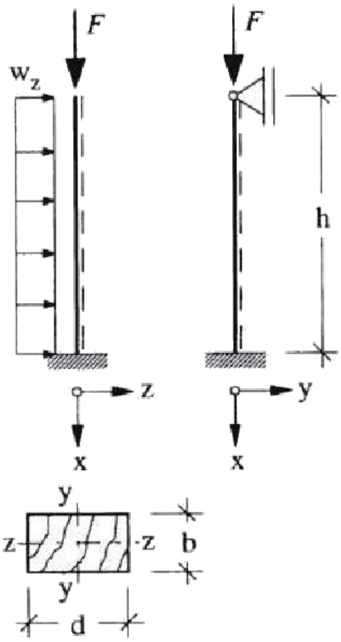


Figure 12.1: System and loads

Model

Cross-section:	d = 21 cm
Material:	CT C24
Height:	h = 3.20 m
SECL:	1
LDC:	permanent

Loading

Load case 1: self-weight	F = 45 kN
Load case 2: wind	w = 1.5 kN/m

Design values for load bearing capacity

$N = 1.35 \cdot F = 1.35 \cdot 45\ KN = 60.75\ kN$ $(k_{mod} = 0.6)$

$q = 1.5 \cdot w = 1.5 \cdot 1.5\ KN/m = 2.25\ kN/m$ $(k_{mod} = 0.9)$

12.2 Input Data

12.2.1 General Data

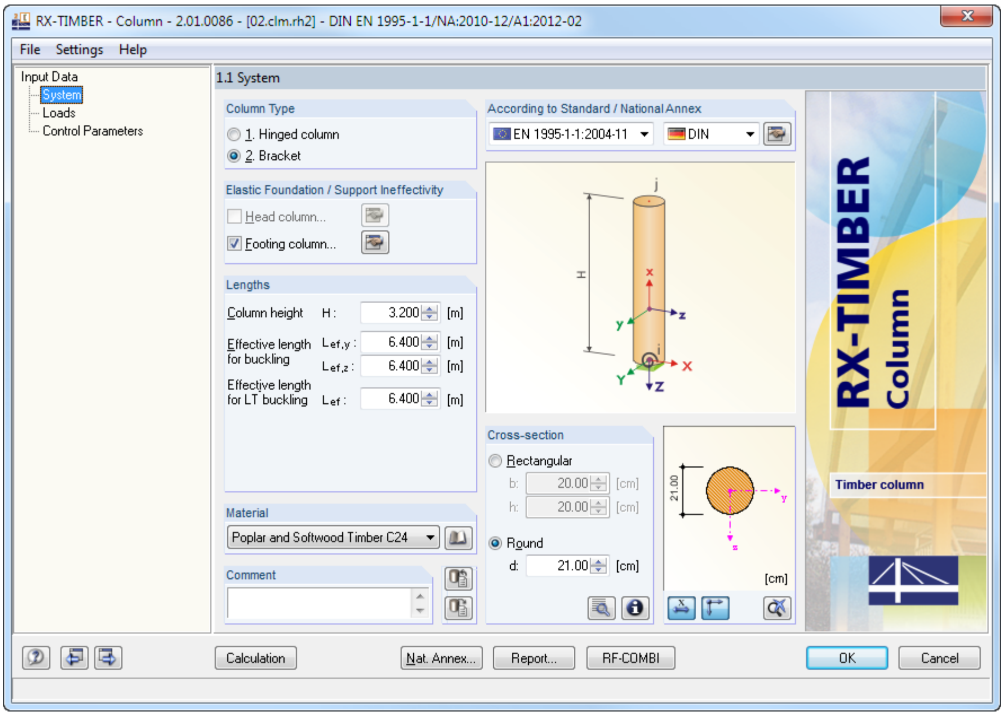


Figure 12.2: Window 1.1 *System*

In window 1.1 *System*, we define the geometric parameters of the column.

We design the *Column Type* **Bracket** *According to Standard* **EN 1995-1-1:2004-11** and the *National Annex* of Germany according to **DIN**.

The *Elastic Foundation* is defined only for the **Footing column**. We use the [Edit] button shown on the left to check if all degrees of freedom are restricted.

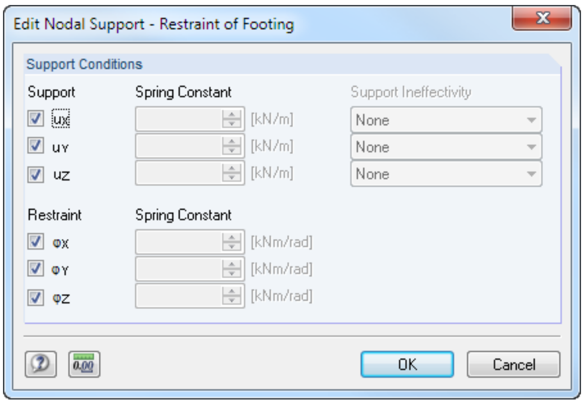


Figure 12.3: Dialog box *Edit Nodal Support - Restraint of Footing*

Then, we define the *Column height* with **3.20 m**. For our bracket model we set each *Effective length* to the doubled value of **6.40 m**.

As *Material* we select **Poplar and Softwood Timber** with the strength grade **C24**. We select the material directly by using the drop-down list. We can also use the [Library] button.

The *Cross-section* is **Round** having a diameter of **21 cm**.

12.2.2 Loads

Load case no. 1 for the self-weight is created automatically. We enter the self-weight as a *Permanent Action* representing a nodal force of **45 kN** on the column head.

The *Wind Load* is acting in the global direction X as distributed load $w_{k,X}$ with **1.50 kN/m**.

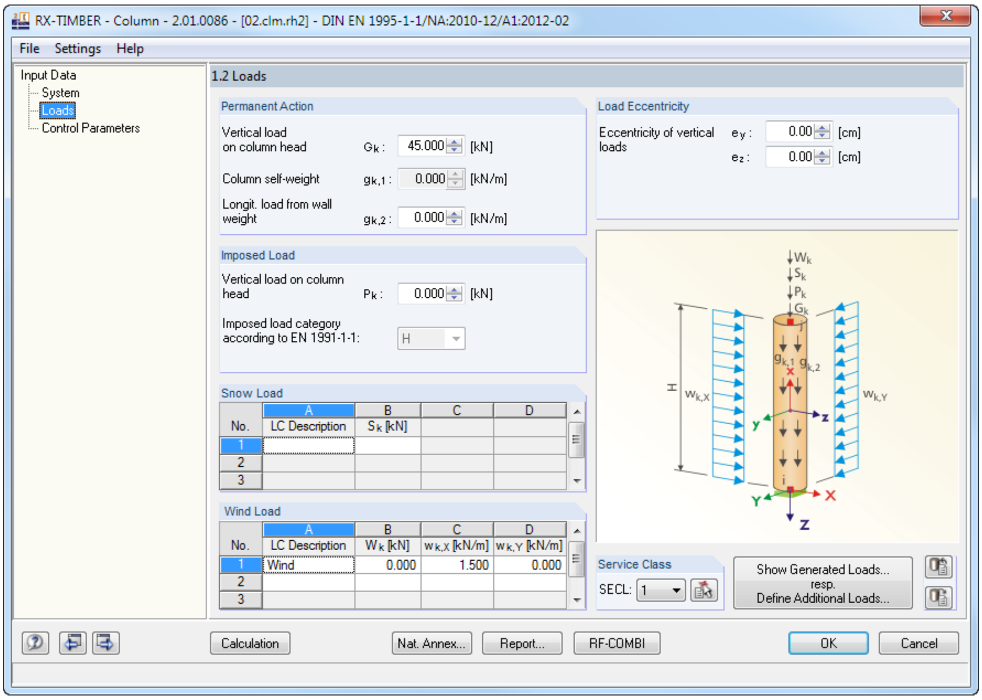


Figure 12.4: Window 1.2 *Loads*

We use the button [Show Generated Loads] to open the graphical display of the load cases.

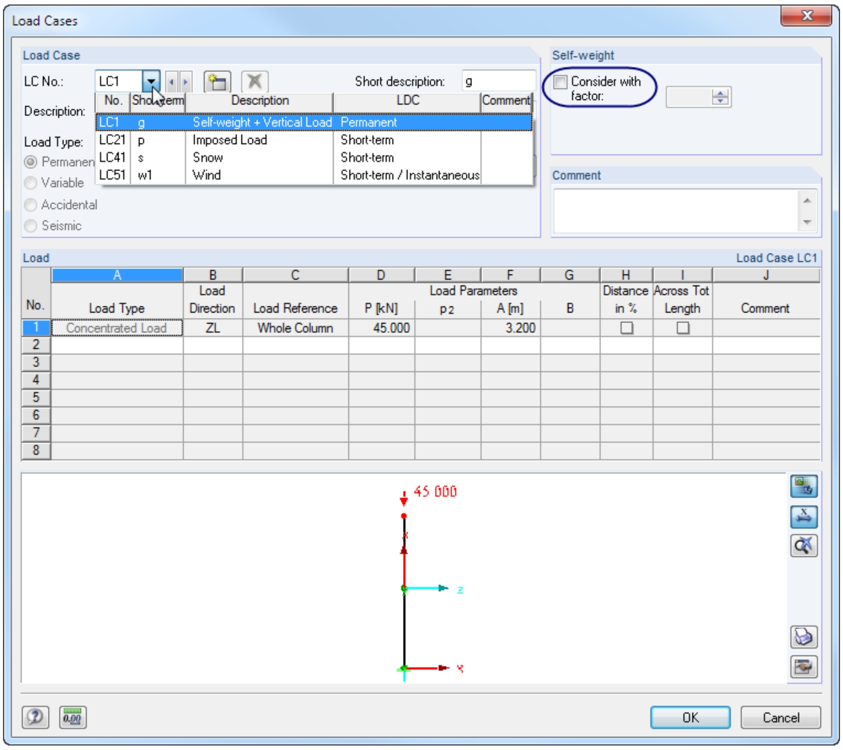


Figure 12.5: Dialog box *Load Cases*

We **clear the check box** used to consider the *Self-weight* for LC1 automatically.

12.2.3 RF-COMBI

Load cases LC21 *Imposed Load* and LC41 *Snow* are generated automatically (see Figure 12.5). However, they do not contain any loads because neither imposed nor snow loads were defined in window 1.2.

By means of the add-on module [RF-COMBI] it is possible to exclude these load cases from the generation of combinations.

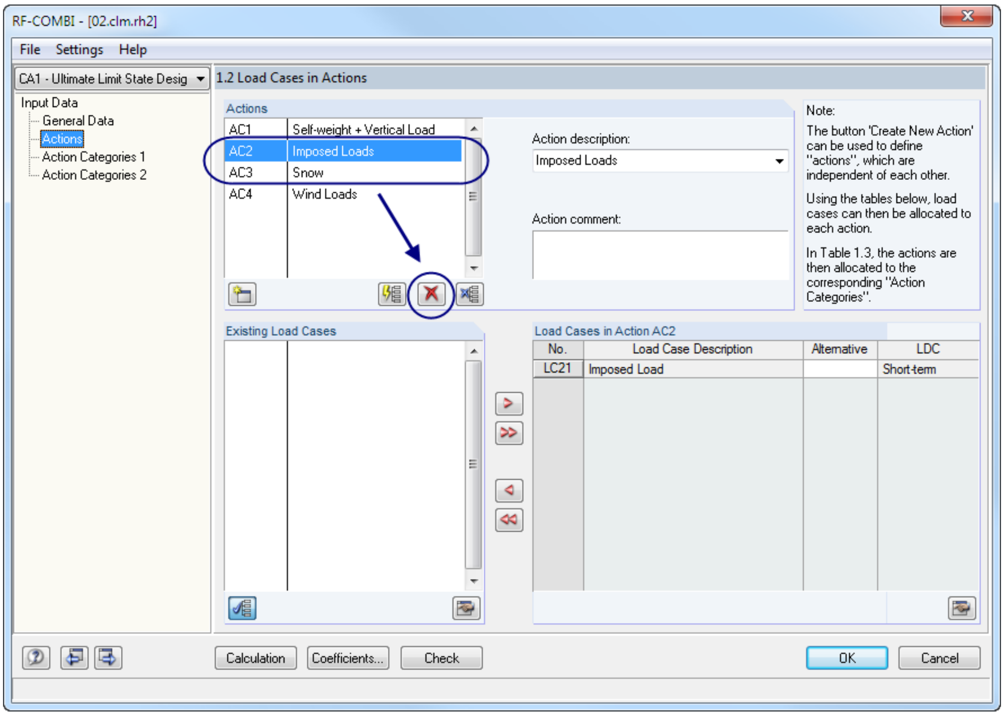


Figure 12.6: RF-COMBI window 1.2 *Load Cases in Actions*

We select the two actions **AC2** and **AC3** in the window section *Actions* and delete them by using the button shown on the left.

Then, in the list above the module navigator, we set the generation case *CA2 - Serviceability Limit State Design* and repeat the delete procedure.

We click the [OK] button to transfer the modifications to RX-TIMBER Column.

12.2.4 Control Parameters

We perform the **Ultimate limit state** and the **Serviceability limit state** design.

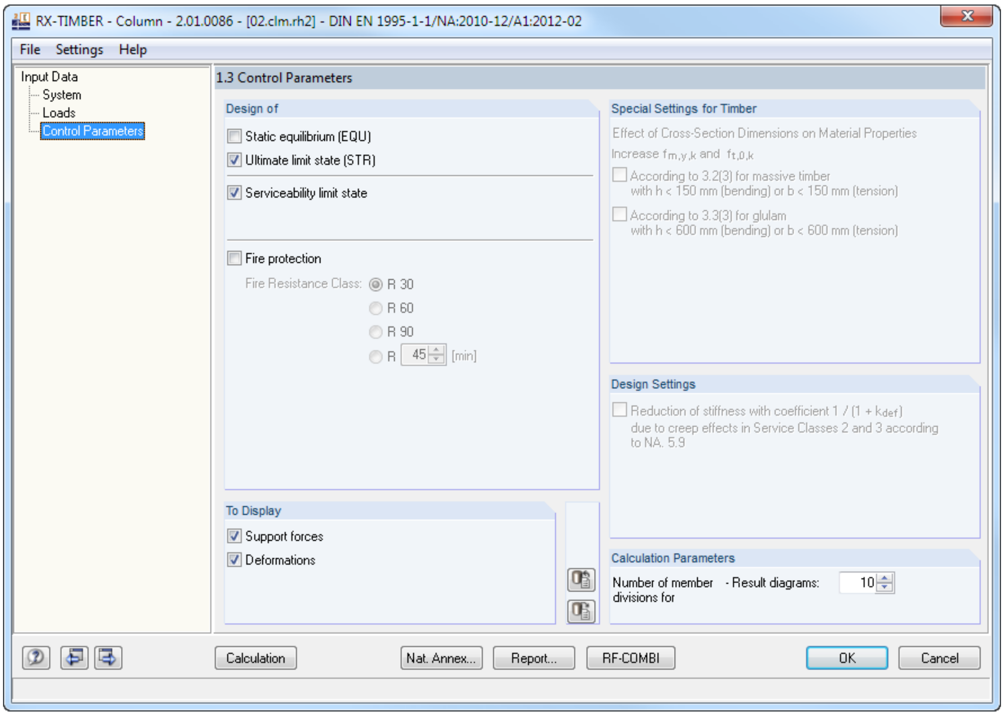


Figure 12.7: Window 1.3 *Control Parameters*

The [Nat. Annex] button shown on the left provides access to significant design parameters such as partial safety and modification factors as well as limit values for deformations (see Figure 6.3, page 56).

We don't need to modify the default settings of the dialog box *National Annex Settings* for our example.

12.3 Results

12.3.1 Result Combinations

After the [Calculation] the program shows the following designs for the generated result combinations.

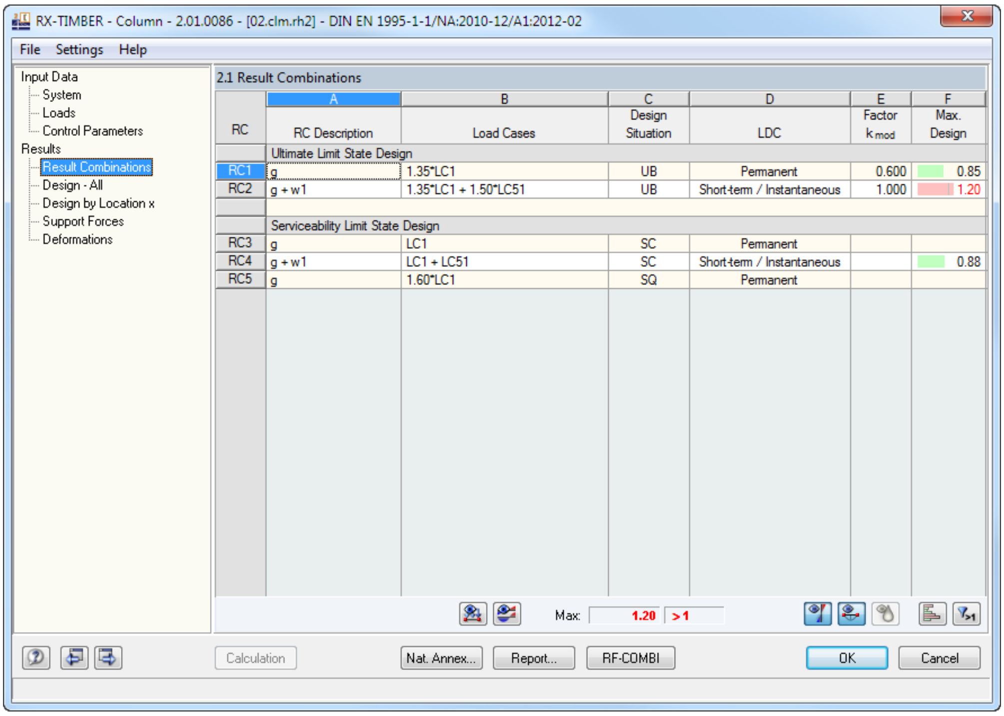


Figure 12.8: Window 2.1 *Result Combinations*

The window shows the design ratios for each result combination. The results are listed in two blocks sorted by the ultimate and the serviceability limit state design.

We use the buttons shown on the left to check the [Load of Load Cases] and the internal forces in the form of [Result Diagrams].

12.3.2 Designs

Window 2.2 *Design - All* lists the governing designs of the column.

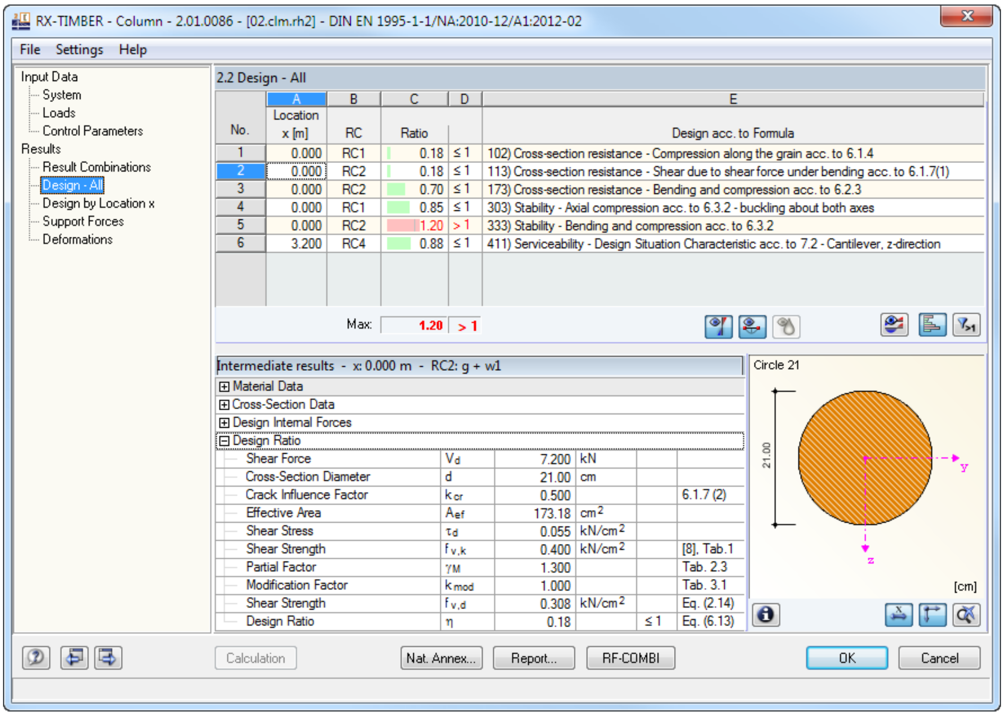


Figure 12.9: Window 2.2 *Design - All*

To explain the calculation of RX-TIMBER, we check some of the designs by hand calculations.

Loading due to shear force according to 6.1.7

The maximum shear loading of 7.2 kN is available at the column base in RC2.

Shear stress

The effective cross-section area of the cross-section is reduced by 50 % by a shear correction factor.

$$\tau_d = 1.33 \cdot \frac{V_d}{A \cdot k_{cr}} = 1.33 \cdot \frac{7.2\ \text{kN}}{346.4\ \text{cm}^2 \cdot 0.5} = 0.055\ \text{kN/cm}^2 = 0.55\ \text{N/mm}^2$$

Shear strength

$$f_{v,d} = \frac{k_{mod} \cdot f_{v,k}}{\gamma_m} = \frac{1.0 \cdot 4\ \text{N/mm}^2}{1.3} = 3.08\ \text{N/mm}^2$$

Design ratio

$$\frac{\tau_d}{f_{v,d}} = \frac{0.55}{3.08} = 0.18 < 1$$

Compression load according to 6.3.2 (stability)

Again, the maximum loading is available in the support area, but this time it is due to the modification factors in RC1 with the LDC 'permanent'.

Compressive stress

$$\sigma_{c,0,d} = \frac{N_d}{A} = \frac{60.75\ \text{kN}}{346.4\ \text{cm}^2} = 0.175\text{KN}/\text{cm}^2$$

Compressive strength

$$f_{c,0,d} = \frac{0.6 \cdot 2.1\,\text{kN}/\text{cm}^2}{1.3} = 0.969\ \text{kN}/\text{cm}^2$$

Design ratio

$$\frac{\sigma_{c,0,d}}{k_c \cdot f_{c,o,d}} = \frac{0.175}{0.21 \cdot 0.969} = 0.86 < 1$$

Bending and compression according to 6.3.2

Again, the governing design criterion is available in the support area for RC2.

Compressive stress

$$\sigma_{c,0,d} = \frac{N_d}{A} = \frac{60.75\ \text{kN}}{346.4\ \text{cm}^2} = 0.18\ \text{kN}/\text{cm}^2$$

Compressive strength

$$f_{c,0,d} = \frac{1.0 \cdot 2.1\,\text{kN}/\text{cm}^2}{1.3} = 1.62\ \text{kN}/\text{cm}^2$$

Governing radius of gyration

$$i_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{\frac{\pi}{4} * r^4}{r^2 * \pi}} = 5.25\text{cm}$$

Slenderness

$$\lambda = \frac{l_{eff}}{\sqrt{I}} \cdot \sqrt{A} = 121.9$$

Relative slenderness ratio

$$\lambda_{rel,c} = \frac{\lambda}{\pi}\sqrt{\frac{f_{c,0,k}}{E_{0.05}}} = 2.07$$

k-factor

$$k = 0.5 \cdot [1 + ß_c \cdot (\lambda_{rel,c} - 0.3) + \lambda^2{}_{rel,c}] = 2.81$$

Buckling coefficient

$$k_c = \frac{1}{k + \sqrt{k^2 - \lambda^2{}_{rel,c}}} = 0.21$$

Lateral buckling factor k_m according to 6.1.6: 1.0

Design according to equation (6.23)

$$\frac{\sigma_{c,0,d}}{k_c \cdot f_{c,0,d}} + \frac{\sigma_{m,y,d}}{k_m \cdot f_{m,y,d}} + k_m \cdot \frac{\sigma_{m,z,d}}{f_{m,z,d}} = \frac{0.18}{0.21 \cdot 1.62} + \frac{1.27}{1.0 \cdot 1.85} + 0.7 \cdot \frac{0}{16.62} = 1.22 > 1$$

12.3.3 Serviceability

The serviceability limit state design required to prevent damages to non-structural components is performed according to EN 1995-1-1, clause 7.2.

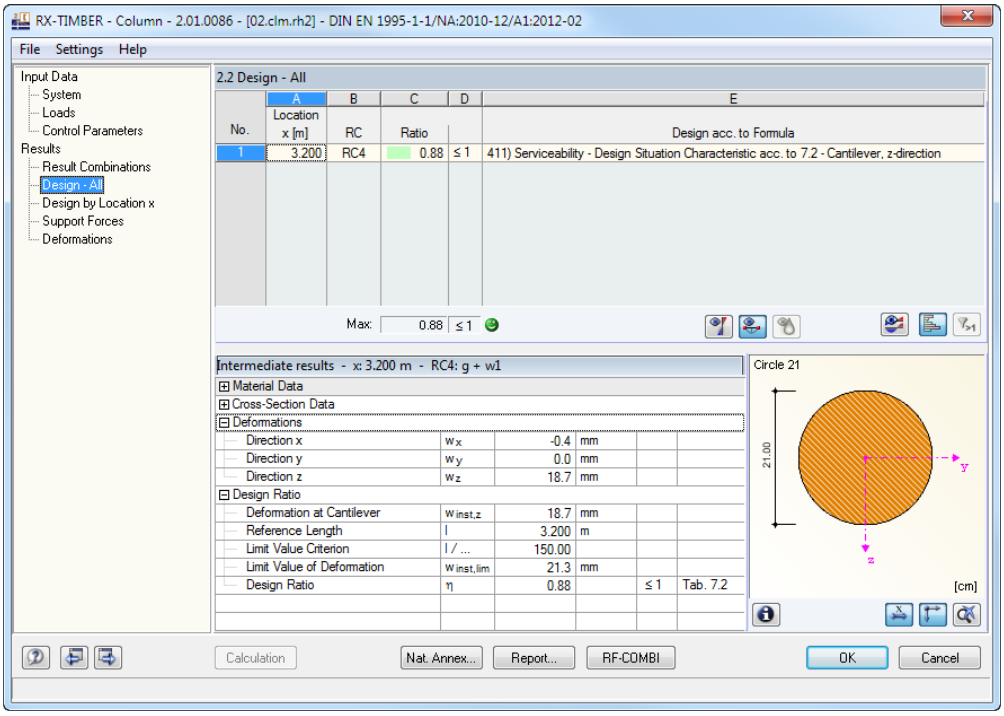


Figure 12.10: Window 2.2 *Design - All* (filter for serviceability limit state)

Due to the bracket's small allowable deformation of l/150, the characteristic design situation is governing.

Serviceability limit state design

$$w_{Q,inst} = \frac{w \cdot h^4}{8 \cdot E \cdot I_y} \leq \frac{l}{150}$$

$$w_{Q,inst} = \frac{1.5 \cdot 3.2^4}{8 \cdot 11000 \cdot 9546.56} \cdot \frac{10^{-1}}{10^{-8}} = 1.87\,\text{cm} < 2.13\,\text{cm} = \frac{320}{150}$$

Design ratio

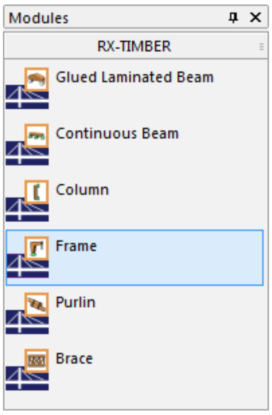
$$\frac{w_{Q,inst}}{w_{inst,grenz}} = \frac{1.87\text{ cm}}{2.13\text{ cm}} = 0.88 < 1$$

The result matches the designs of RX-TIMBER.

12.3.4 Other Results Windows

The remaining results windows as well as the printout report documentation are similar to the ones described in the examples presented in chapter 10 and 11.

13. Frame



This example presents the ultimate limit state design according to EN 1995-1-1 by describing a symmetrical frame. The column is inward-inclined, the frame joint is designed as finger-jointed connection with intermediate piece.

The model data is stored in the model *01.frm* of the project *Examples RX-TIMBER*. We find it among the example files of the program **Frame**.

13.1 System and Loads

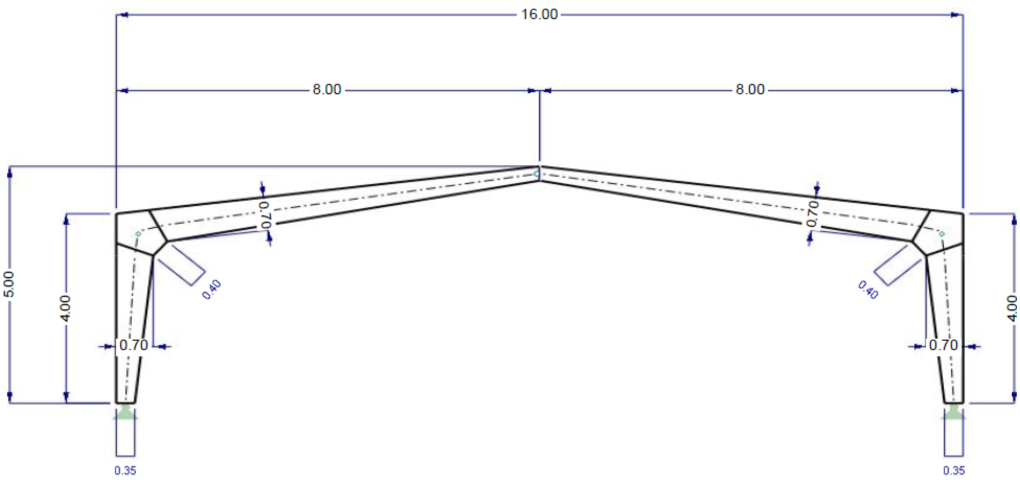


Figure 13.1: System and loads

Model

Cross-section width:	b = 30 cm
Cross-section depth in footing:	h_a = 35 cm
Cross-section depth at vertex:	h_f = 30 cm
Cross-section depth at frame joint:	h_1 = 70 cm
Length of inserted wedge:	l_{zw} = 40 cm

Loading

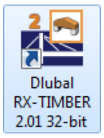
Action 1: self-weight/roof finishes	g = 4.32 kN/m	LDC = permanent
Action 2: snow	SZ 1, 200 m	LDC = short-term
Action 3: wind	WZ 1, TC II	LDC = short-term

The different snow and wind load cases are created automatically by the load generators integrated in the program.

The load cases for the ultimate limit state design are combined according to EN 1990 and EN 1995.

13.2 Input of Model Data

13.2.1 General Data



Start the program by double-clicking the desktop icon **Dlubal RX-TIMBER 2.xx** (see chapter 3.2, page 12), and then select **Frame** in the Project Manager.

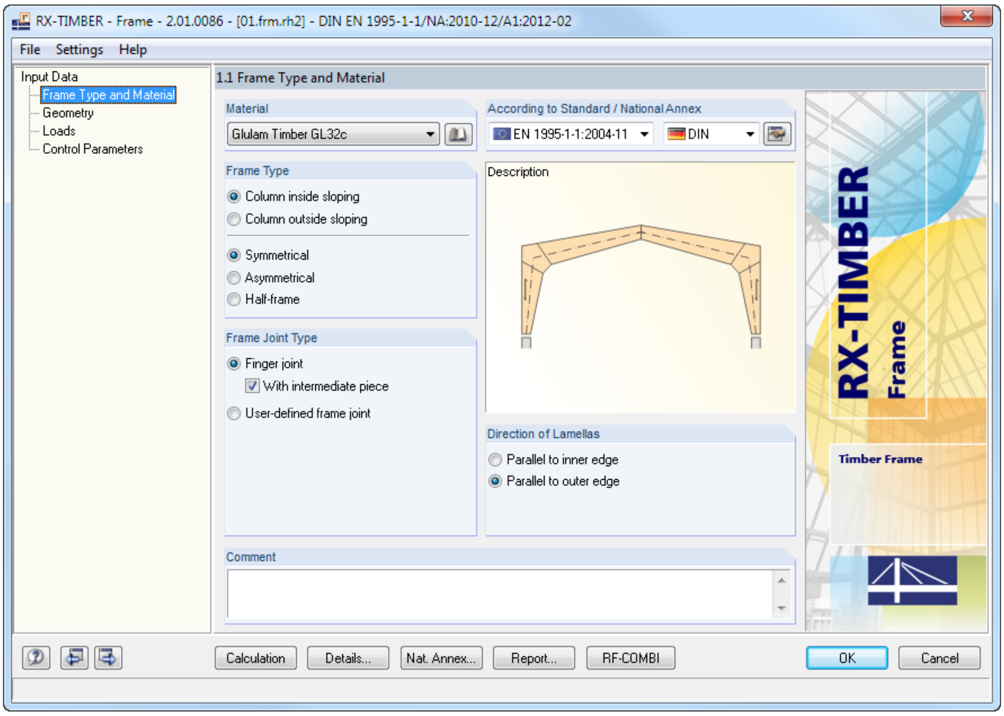


Figure 13.2: Window 1.1 *Frame Type and Material*

The material can be selected in the list or the extensive library. In addition, the [Library] offers us the possibility to specify a user-defined timber grade with specific strength properties.

We select **Gluelam Timber** with the strength grade **GL32c** as *Material*.

We design the frame *According to Standard* **EN 1995-1-1:2004-11** and the *National Annex* for Germany according to **DIN**.

We select a *Frame Type* with **Column inside sloping** (thus the outside is in vertical position) and **Symmetrical** arrangement.

The *Frame Joint Type* is a **Finger joint** defined **With intermediate piece**.

In the window section *Direction of Lamellas*, we specify the lamellas to run **Parallel to outer edge**.

13.2.2 Geometry

In the second window, we enter the number of spans, the span lengths as well as the support and release conditions of the beam. Optionally, we could define cantilevers or assign user-defined spring constants to the supports and releases.

We enter *Building Dimensions* and *Cross-Section* in our example as shown in the figure below.

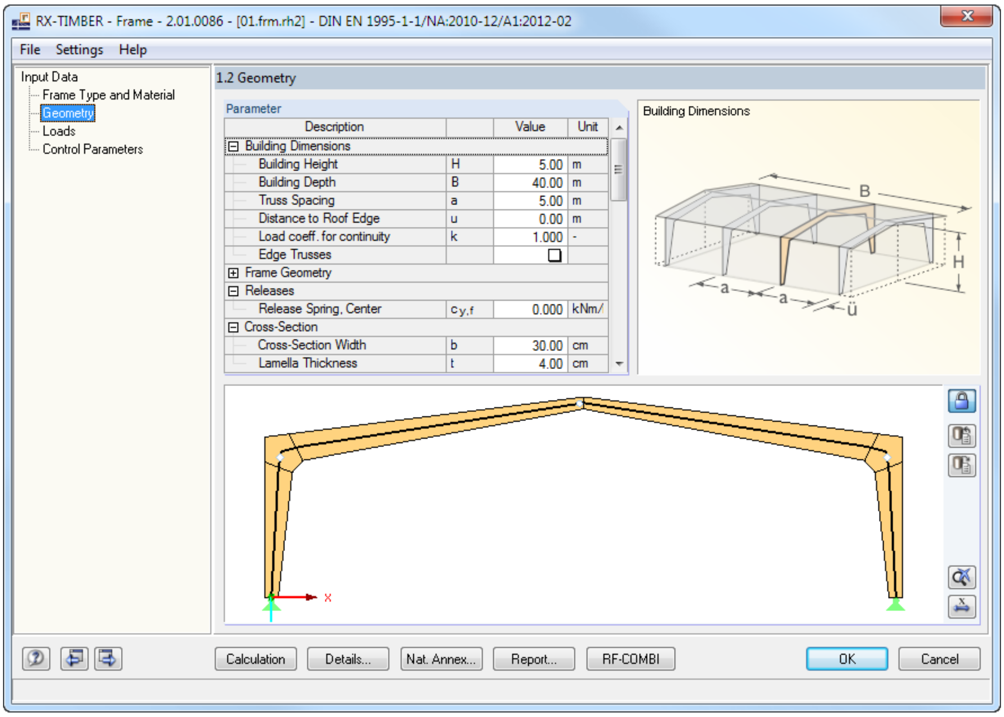


Figure 13.3: Window 1.2 *Geometry*

The *Frame Geometry* is defined by the following parameters:

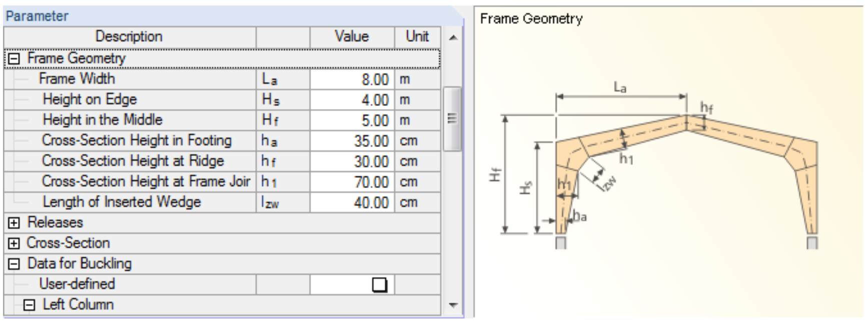


Figure 13.4: Parameters of *Frame Geometry*

Under the *Data for Buckling*, we specify that the *Left Column* as well as the *Left Beam* are **Endangered by Lateral Buckling** (see Figure 13.5).

If lateral supports are available, we can determine the spacing *c* for these intermediate supports. However, we don't do it in our example.

Parameter

Description		Value	Unit
⊟ Data for Buckling			
User-defined		☐	
⊟ Left Column			
Beam Endangered by Lateral Buckling		☑	
Lateral Support Available		☐	
Spacing of Lateral Supports	c	2.00	m
Bracing Spacing	e	0.60	m
Coefficient for effective length	β_s		-
⊟ Left Beam			
Beam Endangered by Lateral Buckling		☑	
Lateral Support Available		☐	
Spacing of Lateral Supports	c	2.00	m
Bracing Spacing	e	0.60	m

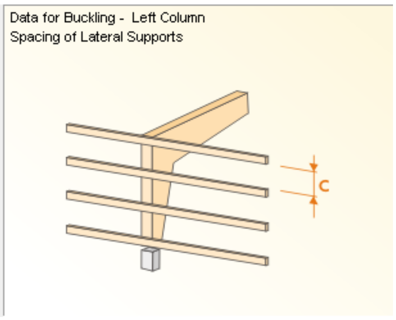


Figure 13.5: Parameters for *Buckling*

By ticking the *User-defined* option we activate two fields for entering the *Coefficient for effective length* directly. However, we do without this option and keep the presettings of the program.

The parameter list continues with specifications for an *Attic*, if applicable. We don't enter data.

Finally, we can check significant *Information Parameters* of the frame geometry.

Parameter

Description		Value	Unit
⊞ Data for Buckling			
⊞ Left Attic			
⊞ Right Attic			
⊟ Information Parameters			
Inclination of Top Edge of Beam	δ	7.13	°
Angle at Frame Joint	ε_0	48.56	°
Connection Angle in Frame Joint	ε	69.28	°
Slope Angle in Frame Joint	α	20.72	°
Slope Angle of Column	α_s	6.41	°
Slope Angle of Beam	α_r	3.22	°
Coat Surface of Frame	As	380365.13	cm^2
Frame Volume	V	3.63	m^3
Frame Weight	G	1.451	t

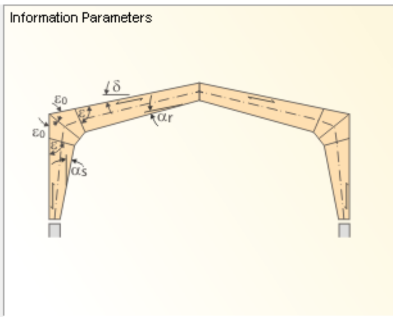


Figure 13.6: *Information Parameters*

13.2.3 Loading

The first action that we enter in window 1.3 *Loads* is the *Permanent Action* for the load "self-weight and roof structure". The following values are selected for the *Roof Layer*.

- Trapezoidal sheet metal with 0.15 kN/m²
- Purlins with 0.15 kN/m²
- Vapour barrier with 0.02 kN/m²
- Rock wool (d = 20 cm) with 0.20 kN/m²
- Oriented strand boards (OSB) (d = 25 mm) with 0.15 kN/m²

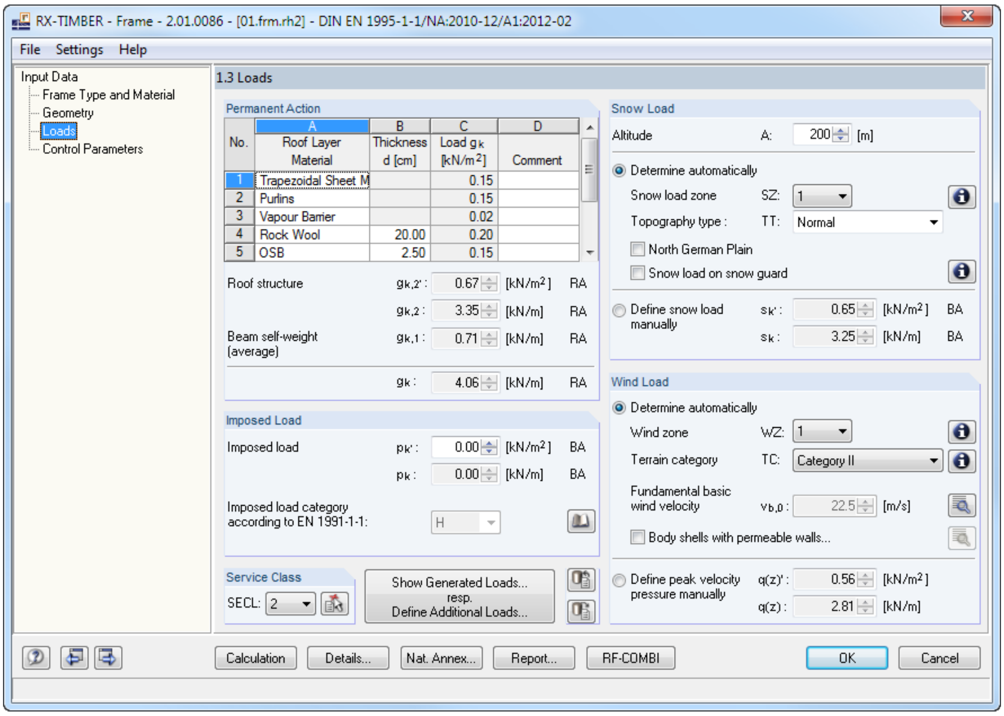


Figure 13.7: Window 1.3 *Loads*

We use the [...] button occurring in the *Roof Layer* column to select materials in a library.

The roof structure results in the permanent action g_k of 4.06 kN/m. This load includes the self-weight of the beam. As the beam has a linearly variable height, the self-weight of the beam is automatically applied as trapezoidal load.

It would be possible to define an additional *Imposed load* by using the button shown on the left, selecting a load from the table and assigning the corresponding *Imposed load category* from the list for *EN 1991-1-1*. However, in our example we only define the *Service Class* **2** for the beam.

To determine the *Snow Load*, we enter **200 m** for the *Altitude* above sea level. The *Snow load zone* **SZ 1** can be defined by a double-click in the snow load map.

For our example we **Determine automatically** the *Wind Load*. We only specify the *Wind zone* **WZ 1** and the *Terrain category* **TC II**. Informative dialog boxes facilitate the assignment.

We use the button [Show Generated Loads] shown on the left to open the dialog box *Load Cases* where we can check the generated loads.

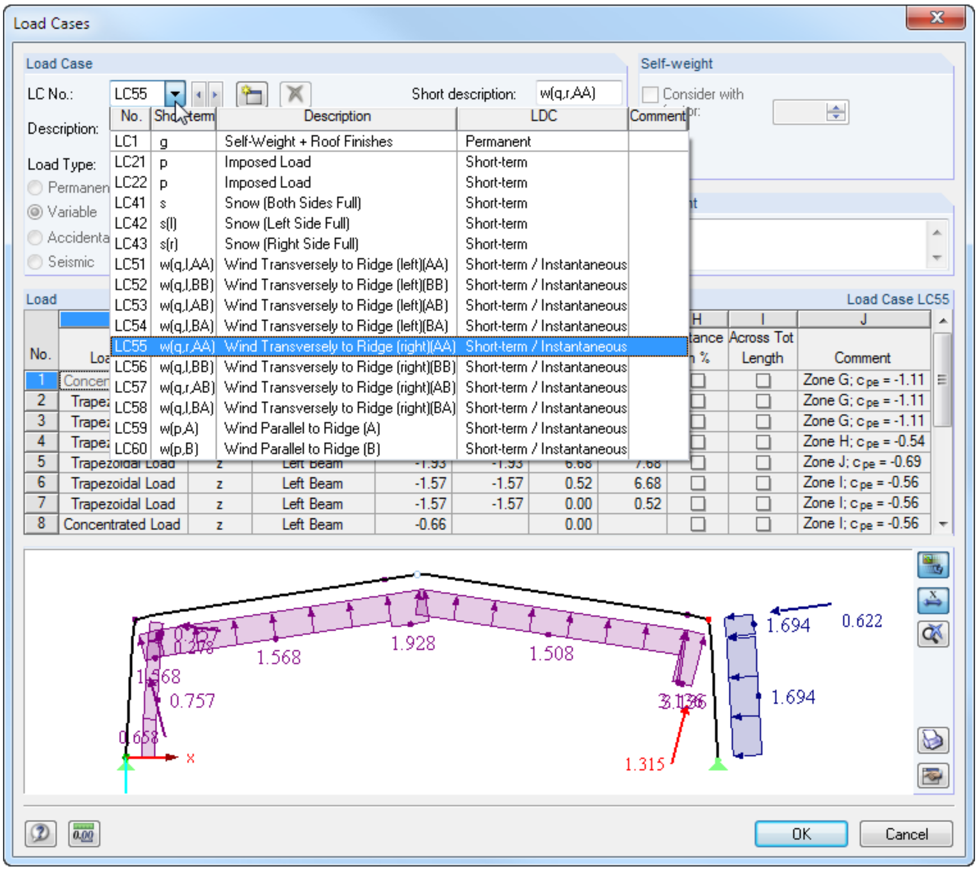


Figure 13.8: Dialog box *Load Cases* with list of generated snow and wind loads

The dimensions of the building are decisive for the wind load generation. Based on the dimensions specified in window 1.2 *Geometry*, the program determines the values in column F, G, H and I according to [4] or [11] and generates the wind loads accordingly.

For the zones where compressive as well as suction forces are applied, the program creates several load cases for each wind action with the corresponding suction or compressive forces.

13.2.4 Control Parameters

In window 1.4 *Control Parameters*, we set only the design of the **Ultimate limit state**. We clear the check boxes for *Static equilibrium*, *Serviceability limit state* and *Fire protection*.

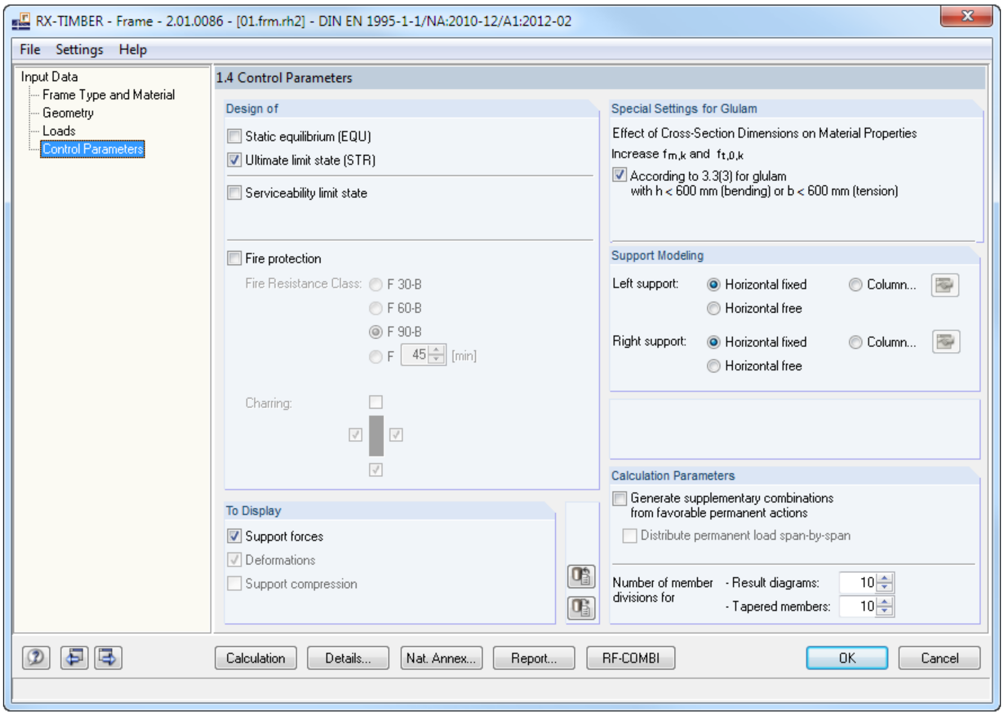


Figure 13.9: Window 1.4 *Control Parameters*

In the window section *To Display*, we tick the option **Support forces**.

We want the *Support Modelling* to be defined with **Horizontal fixed** supports on both sides.

We don't modify the presettings in the window sections *Special Settings for Glulam* and *Calculation Parameters*.

13.2.5 RF-COMBI

The add-on module RF-COMBI integrated in RX-TIMBER combines the load cases automatically. We don't need to open RF-COMBI. However, we want to check the combinations that are generated in the background. We access the add-on module by using the [RF-COMBI] button.

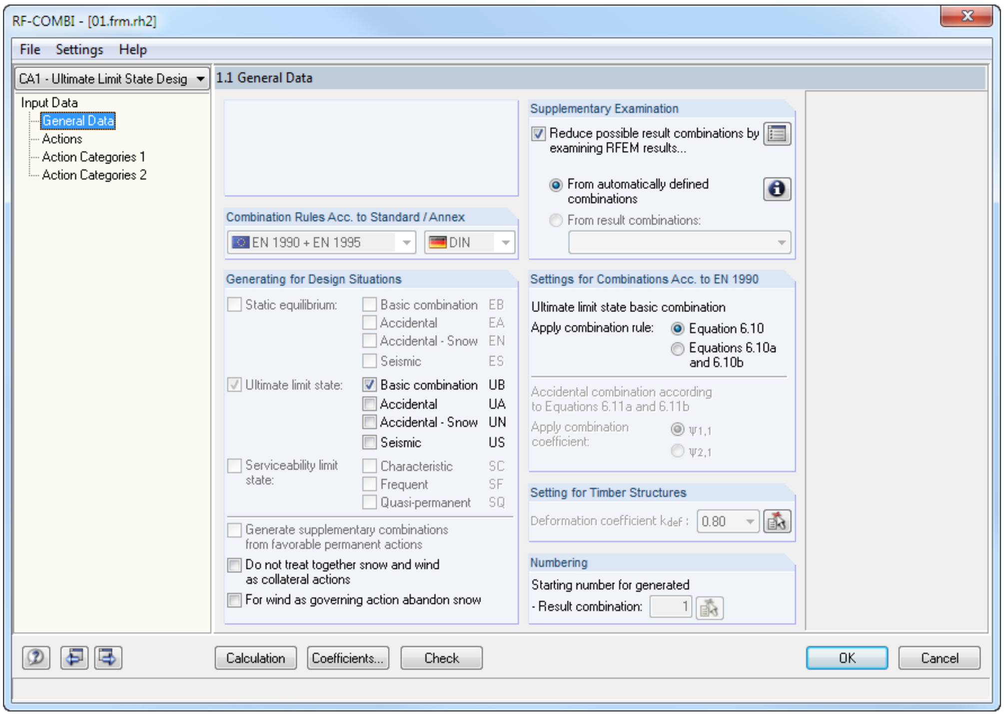


Figure 13.10: Add-on module *RF-COMBI*

We create combinations by clicking the [Calculation] button.

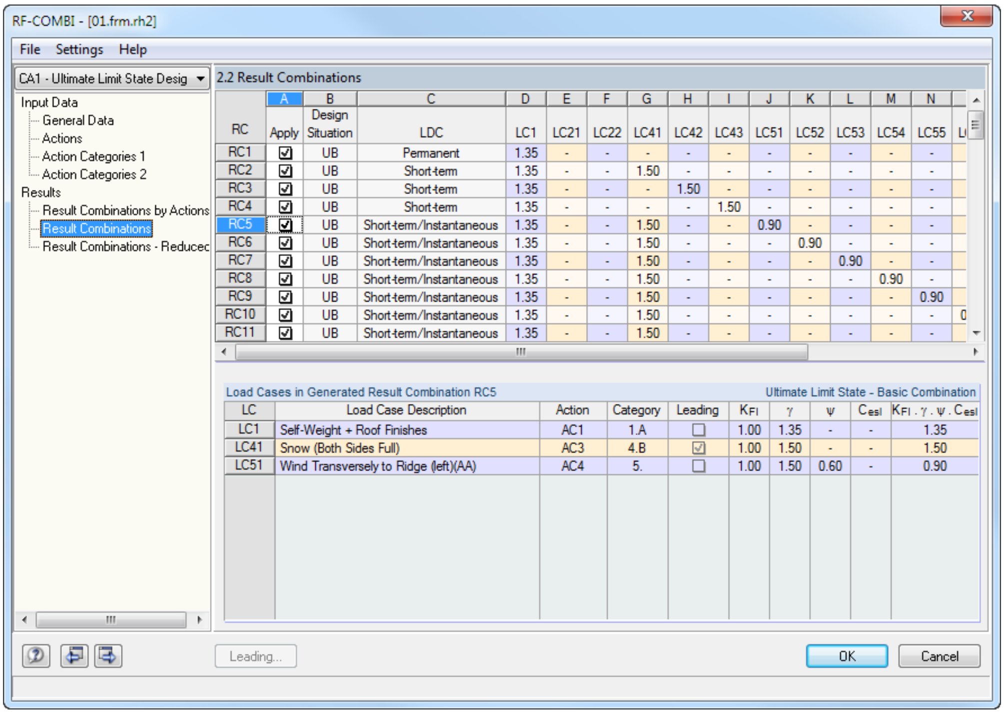


Figure 13.11: RF-COMBI window 2.2 *Result Combinations*

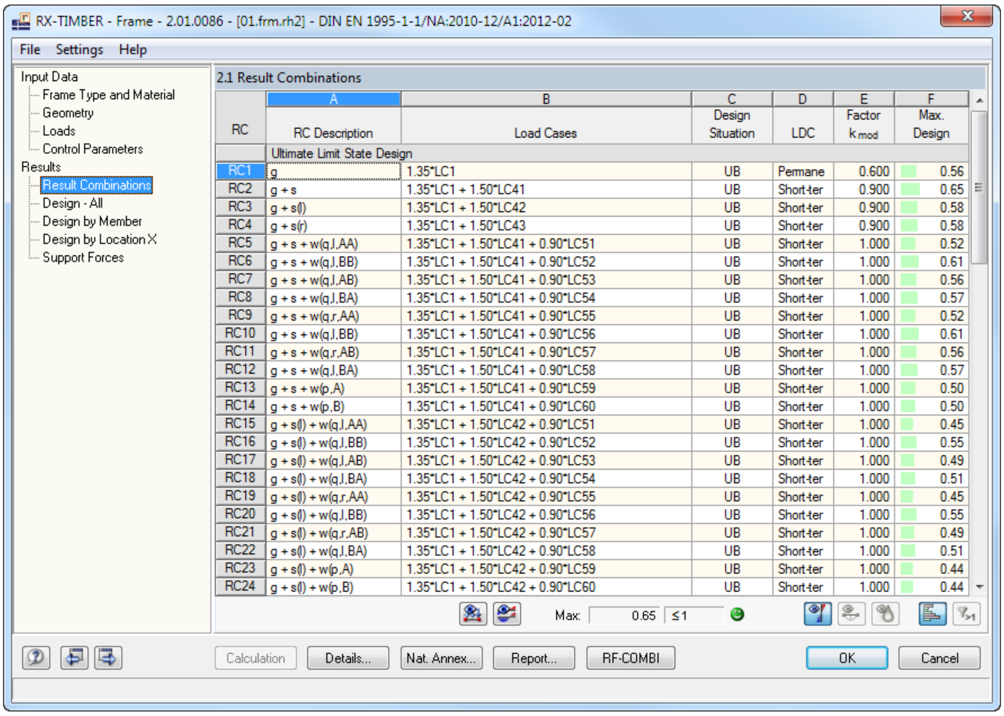
We click [OK] to return to *RX-TIMBER Frame*.

13.3 Results

13.3.1 Result Combinations

We start the calculation in RX-TIMBER by clicking the [Calculation] button shown on the left.

The designs of the combinations are shown with the respective design ratios in module window 2.1.



RC	A RC Description	B Load Cases	C Design Situation	D LDC	E Factor k_{mod}	F Max. Design
	Ultimate Limit State Design					
RC1	g	1.35*LC1	UB	Permane	0.600	0.56
RC2	g + s	1.35*LC1 + 1.50*LC41	UB	Short-ter	0.900	0.65
RC3	g + s(l)	1.35*LC1 + 1.50*LC42	UB	Short-ter	0.900	0.58
RC4	g + s(r)	1.35*LC1 + 1.50*LC43	UB	Short-ter	0.900	0.58
RC5	g + s + w(q,l,AA)	1.35*LC1 + 1.50*LC41 + 0.90*LC51	UB	Short-ter	1.000	0.52
RC6	g + s + w(q,l,BB)	1.35*LC1 + 1.50*LC41 + 0.90*LC52	UB	Short-ter	1.000	0.61
RC7	g + s + w(q,l,AB)	1.35*LC1 + 1.50*LC41 + 0.90*LC53	UB	Short-ter	1.000	0.56
RC8	g + s + w(q,l,BA)	1.35*LC1 + 1.50*LC41 + 0.90*LC54	UB	Short-ter	1.000	0.57
RC9	g + s + w(q,r,AA)	1.35*LC1 + 1.50*LC41 + 0.90*LC55	UB	Short-ter	1.000	0.52
RC10	g + s + w(q,l,BB)	1.35*LC1 + 1.50*LC41 + 0.90*LC56	UB	Short-ter	1.000	0.61
RC11	g + s + w(q,r,AB)	1.35*LC1 + 1.50*LC41 + 0.90*LC57	UB	Short-ter	1.000	0.56
RC12	g + s + w(q,l,BA)	1.35*LC1 + 1.50*LC41 + 0.90*LC58	UB	Short-ter	1.000	0.57
RC13	g + s + w(p,A)	1.35*LC1 + 1.50*LC41 + 0.90*LC59	UB	Short-ter	1.000	0.50
RC14	g + s + w(p,B)	1.35*LC1 + 1.50*LC41 + 0.90*LC60	UB	Short-ter	1.000	0.50
RC15	g + s(l) + w(q,l,AA)	1.35*LC1 + 1.50*LC42 + 0.90*LC51	UB	Short-ter	1.000	0.45
RC16	g + s(l) + w(q,l,BB)	1.35*LC1 + 1.50*LC42 + 0.90*LC52	UB	Short-ter	1.000	0.55
RC17	g + s(l) + w(q,l,AB)	1.35*LC1 + 1.50*LC42 + 0.90*LC53	UB	Short-ter	1.000	0.49
RC18	g + s(l) + w(q,l,BA)	1.35*LC1 + 1.50*LC42 + 0.90*LC54	UB	Short-ter	1.000	0.51
RC19	g + s(l) + w(q,r,AA)	1.35*LC1 + 1.50*LC42 + 0.90*LC55	UB	Short-ter	1.000	0.45
RC20	g + s(l) + w(q,l,BB)	1.35*LC1 + 1.50*LC42 + 0.90*LC56	UB	Short-ter	1.000	0.55
RC21	g + s(l) + w(q,r,AB)	1.35*LC1 + 1.50*LC42 + 0.90*LC57	UB	Short-ter	1.000	0.49
RC22	g + s(l) + w(q,l,BA)	1.35*LC1 + 1.50*LC42 + 0.90*LC58	UB	Short-ter	1.000	0.51
RC23	g + s(l) + w(p,A)	1.35*LC1 + 1.50*LC42 + 0.90*LC59	UB	Short-ter	1.000	0.44
RC24	g + s(l) + w(p,B)	1.35*LC1 + 1.50*LC42 + 0.90*LC60	UB	Short-ter	1.000	0.44

Figure 13.12: Window 2.1 *Result Combinations*

The factor k_{mod} used to consider the time-dependent settlement behavior due to load duration and specific climatic characteristics of timber is applied automatically to the required value according to the LDC and the SECL.

We use the button [Load of Load Cases] to check the loads and the classes of the load actions that refer to the current result combination (that is the RC table row where the cursor is placed).

13.3.2 Designs

The window lists all designs that are governing for the entire frame.

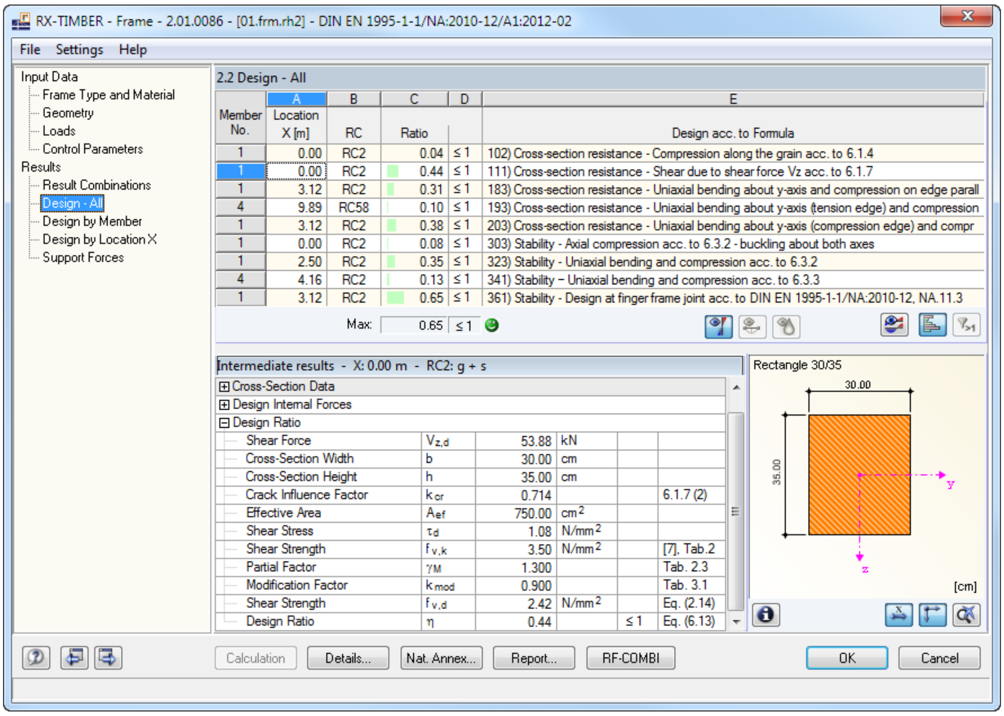


Figure 13.13: Window 2.2 *Design - All*

The factors and design internal forces can be checked interactively: We click a design in the table above to display all *Intermediate results* of the design in the window section below.

The results calculated by the program are checked by manual calculations.

Shear due to shear force V_z according to 6.1.7

The maximum shear force of 53.88 kN occurs at the left column base in RC2.

Shear stress

$$\tau_d = 1.5 \cdot \frac{V_d}{b_{ef} \cdot h} = 1.5 \cdot \frac{53.88 \text{ kN}}{21.4 \text{ cm} \cdot 35 \text{ cm}} = 0.108 \text{ kN/cm}^2 = 1.08 \text{ N/mm}^2$$

where $b_{ef} = k_{cr} \cdot b = 0.714 \cdot 30 \text{ cm} = 21.4 \text{ cm}$

$$k_{cr} = \frac{2.5}{f_{v,k}} = \frac{2.5}{3.5} = 0.714$$

Shear strength

$$f_{v,d} = \frac{k_{mod} \cdot f_{v,k}}{\gamma_m} = \frac{0.9 \cdot 3.5 \text{ N/mm}^2}{1.3} = 2.42 \text{ N/mm}^2$$

Design ratio

$$\frac{\tau_d}{f_{v,d}} = \frac{1.08}{2.42} = 0.44 < 1$$

Uniaxial bending and compression on edge parallel to grain

The cross-section resistance is analyzed according to clause 6.4.2 and 6.2.4.

The maximum loading due to bending and compression occurs in the left frame joint in RC2. The [Result Diagrams] represent the internal forces on the members stringed together if RC2 is set in the list above.

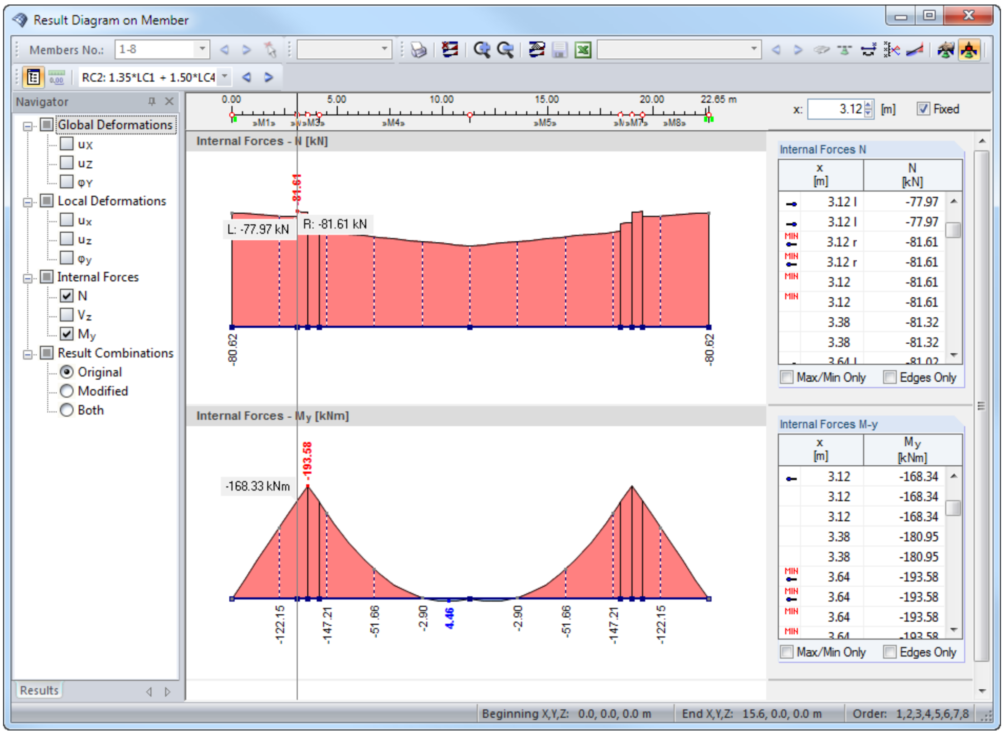
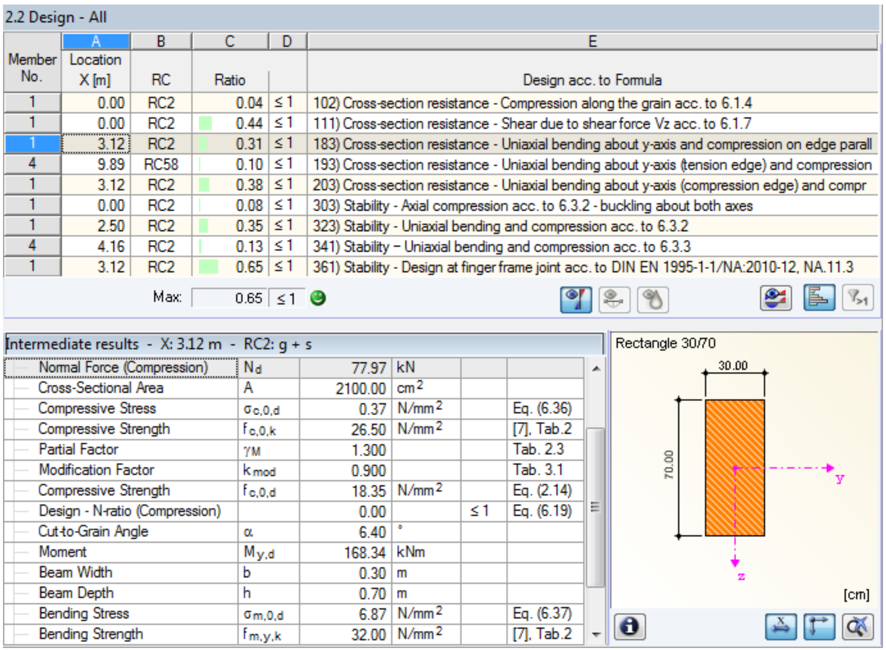


Figure 13.14: Distribution of internal forces N and M_y in RC2

RX-TIMBER designs the cross-section resistance on the location X = 3.12 as follows:



2.2 Design - All

Member No.	A Location X [m]	B RC	C Ratio	D	E Design acc. to Formula
1	0.00	RC2	0.04	≤ 1	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
1	0.00	RC2	0.44	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
1	3.12	RC2	0.31	≤ 1	183) Cross-section resistance - Uniaxial bending about y-axis and compression on edge parall
4	9.89	RC58	0.10	≤ 1	193) Cross-section resistance - Uniaxial bending about y-axis (tension edge) and compression
1	3.12	RC2	0.38	≤ 1	203) Cross-section resistance - Uniaxial bending about y-axis (compression edge) and compr
1	0.00	RC2	0.08	≤ 1	303) Stability - Axial compression acc. to 6.3.2 - buckling about both axes
1	2.50	RC2	0.35	≤ 1	323) Stability - Uniaxial bending and compression acc. to 6.3.2
4	4.16	RC2	0.13	≤ 1	341) Stability – Uniaxial bending and compression acc. to 6.3.3
1	3.12	RC2	0.65	≤ 1	361) Stability - Design at finger frame joint acc. to DIN EN 1995-1-1/NA:2010-12, NA.11.3
		Max:	0.65	≤ 1	

Intermediate results - X: 3.12 m - RC2: g + s

Normal Force (Compression)	N_d	77.97	kN		
Cross-Sectional Area	A	2100.00	cm²		
Compressive Stress	$\sigma_{c,0,d}$	0.37	N/mm²		Eq. (6.36)
Compressive Strength	$f_{c,0,k}$	26.50	N/mm²		[7], Tab.2
Partial Factor	γ_M	1.300			Tab. 2.3
Modification Factor	k_{mod}	0.900			Tab. 3.1
Compressive Strength	$f_{c,0,d}$	18.35	N/mm²		Eq. (2.14)
Design - N-ratio (Compression)		0.00		≤ 1	Eq. (6.19)
Cut-to-Grain Angle	α	6.40	°		
Moment	$M_{y,d}$	168.34	kNm		
Beam Width	b	0.30	m		
Beam Depth	h	0.70	m		
Bending Stress	$\sigma_{m,0,d}$	6.87	N/mm²		Eq. (6.37)
Bending Strength	$f_{m,y,k}$	32.00	N/mm²		[7], Tab.2

Figure 13.15: Design of cross-section resistance for bending and compression on edge parallel to grain

Compressive stress $$\sigma_{c,0,d} = \frac{N_d}{A} = \frac{77.97\ \text{kN} \cdot 10^3}{2100\ \text{cm}^2 \cdot 10^2} = 0.37\ \text{N/mm}^2$$

Design, axial force component $$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 = \left(\frac{0.39}{18.35}\right)^2 \approx 0.00$$

Bending stress $$\sigma_{m,y,d} = \frac{M_{y,d}}{W_y} = \frac{168.34\ \text{kNm} \cdot 10^6}{24500\ \text{cm}^3 \cdot 10^3} = 6.87\ \text{N/mm}^2$$

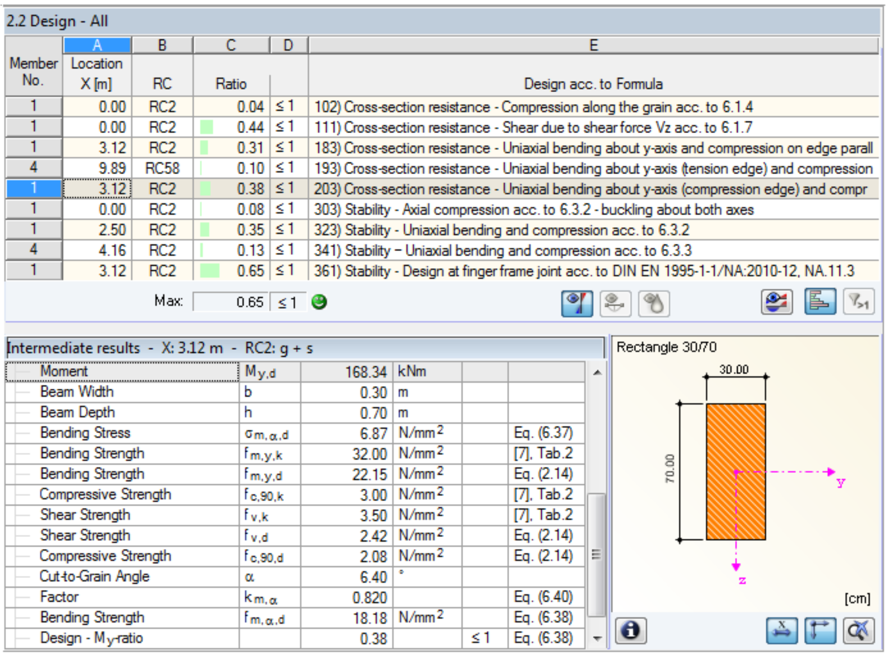
Flexural resistance $$f_{m,d} = k_{mod} \cdot \frac{f_{m,k}}{\gamma_M} = 0.9 \cdot \frac{32\ \text{N/mm}^2}{1.3} = 22.15\ \text{N/mm}^2$$

Design, moment $$\frac{\sigma_{m,y,d}}{f_{m,d}} = \frac{6.87}{22.15} = 0.31$$

Design ratio $$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,d}}{f_{m,d}} = 0.00 + 0.33 = 0.31 < 1$$

Uniaxial bending and compression on cut edge (compression edge)

The maximum loading occurs on the same location X, again in RC2.



2.2 Design - All

Member No.	A Location X [m]	B RC	C Ratio	D	E Design acc. to Formula
1	0.00	RC2	0.04	≤ 1	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
1	0.00	RC2	0.44	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
1	3.12	RC2	0.31	≤ 1	183) Cross-section resistance - Uniaxial bending about y-axis and compression on edge parall
4	9.89	RC58	0.10	≤ 1	193) Cross-section resistance - Uniaxial bending about y-axis (tension edge) and compression
1	3.12	RC2	0.38	≤ 1	203) Cross-section resistance - Uniaxial bending about y-axis (compression edge) and compr
1	0.00	RC2	0.08	≤ 1	303) Stability - Axial compression acc. to 6.3.2 - buckling about both axes
1	2.50	RC2	0.35	≤ 1	323) Stability - Uniaxial bending and compression acc. to 6.3.2
4	4.16	RC2	0.13	≤ 1	341) Stability – Uniaxial bending and compression acc. to 6.3.3
1	3.12	RC2	0.65	≤ 1	361) Stability - Design at finger frame joint acc. to DIN EN 1995-1-1/NA:2010-12, NA.11.3
		Max:	0.65	≤ 1	

Intermediate results - X: 3.12 m - RC2: g + s

Moment	$M_{y,d}$	168.34	kNm		
Beam Width	b	0.30	m		
Beam Depth	h	0.70	m		
Bending Stress	$\sigma_{m,\alpha,d}$	6.87	N/mm²		Eq. (6.37)
Bending Strength	$f_{m,y,k}$	32.00	N/mm²		[7], Tab.2
Bending Strength	$f_{m,y,d}$	22.15	N/mm²		Eq. (2.14)
Compressive Strength	$f_{c,90,k}$	3.00	N/mm²		[7], Tab.2
Shear Strength	$f_{v,k}$	3.50	N/mm²		[7], Tab.2
Shear Strength	$f_{v,d}$	2.42	N/mm²		Eq. (2.14)
Compressive Strength	$f_{c,90,d}$	2.08	N/mm²		Eq. (2.14)
Cut-to-Grain Angle	α	6.40	°		
Factor	$k_{m,\alpha}$	0.820			Eq. (6.40)
Bending Strength	$f_{m,\alpha,d}$	18.18	N/mm²		Eq. (6.38)
Design - M_y-ratio		0.38		≤ 1	Eq. (6.38)

Figure 13.16: Design of cross-section resistance for bending and compression on cut edge

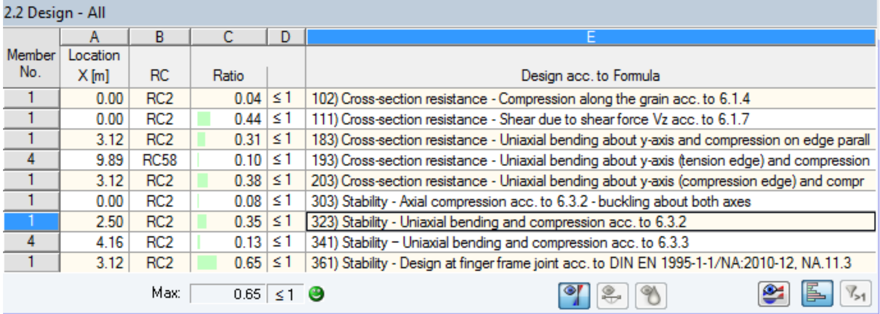
The same compressive and bending stresses are available like for the edge parallel to grain (see above). For the design on the cut edge the factor $k_{m,\alpha}$ is decisive.

Factor, compression edge $$k_{m,\alpha} = \frac{1}{\sqrt{1 + \left(\frac{f_{m,d}}{1.5 \cdot f_{v,d}} \cdot \tan\alpha\right)^2 + \left(\frac{f_{m,d}}{f_{c,90,d}} \cdot \tan^2\alpha\right)^2}} = 0.82$$

Design ratio $$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,d}}{k_{m,\alpha} \cdot f_{m,d}} = \left(\frac{0.39}{18.35}\right)^2 + \frac{6.87}{0.82 \cdot 22.15} = 0.38 < 1$$

Stability for uniaxial bending and compression force acc. to 6.3.2

For the stability analysis the lateral buckling design is given for uniaxial bending about the member axis y with compression force to be governing.



2.2 Design - All

Member No.	A Location X [m]	B RC	C Ratio	D	E Design acc. to Formula
1	0.00	RC2	0.04	≤ 1	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
1	0.00	RC2	0.44	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
1	3.12	RC2	0.31	≤ 1	183) Cross-section resistance - Uniaxial bending about y-axis and compression on edge parall
4	9.89	RC58	0.10	≤ 1	193) Cross-section resistance - Uniaxial bending about y-axis (tension edge) and compression
1	3.12	RC2	0.38	≤ 1	203) Cross-section resistance - Uniaxial bending about y-axis (compression edge) and compr
1	0.00	RC2	0.08	≤ 1	303) Stability - Axial compression acc. to 6.3.2 - buckling about both axes
1	2.50	RC2	0.35	≤ 1	323) Stability - Uniaxial bending and compression acc. to 6.3.2
4	4.16	RC2	0.13	≤ 1	341) Stability – Uniaxial bending and compression acc. to 6.3.3
1	3.12	RC2	0.65	≤ 1	361) Stability - Design at finger frame joint acc. to DIN EN 1995-1-1/NA:2010-12, NA.11.3
	Max:		0.65	≤ 1	

Figure 13.17: Stability designs

The governing location X = 2.496 m is determined on the left column. The following internal forces of RC2 are designed:

- Axial force $N_d = 78.64$ kN
- Moment $M_{y,d} = 134.63$ kNm

Compressive stress
$$\sigma_{c,0,d} = \frac{N_d}{A} = \frac{78.64\ \text{kN} \cdot 10^3}{1890\ \text{cm}^2 \cdot 10^2} = 0.42\ \text{N/mm}^2$$

Bending stress
$$\sigma_{m,y,d} = \frac{M_{y,d}}{W_y} = \frac{134.63\ \text{kNm} \cdot 10^6}{19845\ \text{cm}^3 \cdot 10^3} = 6.78\ \text{N/mm}^2$$

Buckling length coefficient
$$\beta_S = \sqrt{4 + \frac{\pi^2 \cdot I_S \cdot s}{3 \cdot h \cdot I_R} + \frac{I_S \cdot N_R \cdot s^2}{I_R \cdot N_S \cdot h^2}} =$$
$$= \sqrt{4 + \frac{\pi^2 \cdot 5.83E+09 \cdot 7683}{3 \cdot 3641 \cdot 4.833E+09} + \frac{5.83E+09 \cdot 76209 \cdot 7683^2}{4.833E+09 \cdot 85357 \cdot 3641^2}} = 4.143$$

Slenderness
$$\lambda_y = \frac{l_{ef,y}}{i_y} = \frac{\beta_S \cdot h}{i_y} = \frac{4.143 \cdot 3641}{177.7} = 84.283$$
$$\lambda_z = \frac{l_{ef,z}}{i_z} = \frac{3642}{86.6} = 42.051$$

Relative slenderness
$$\lambda_{rel,c,y} = \frac{\lambda_y}{\pi} \cdot \sqrt{\frac{f_{c,0,k}}{E_{0.05}}} = \frac{84.283}{\pi} \cdot \sqrt{\frac{26.5}{11100}} = 1.311$$
$$\lambda_{rel,c,z} = \frac{\lambda_z}{\pi} \cdot \sqrt{\frac{f_{c,0,k}}{E_{0.05}}} = \frac{42.051}{\pi} \cdot \sqrt{\frac{26.5}{11100}} = 0.654$$

Auxiliary buckling coefficient
$$k_y = 0.5 \cdot \left(1 + \beta_c \cdot (\lambda_{rel,y} - 0.3) + \lambda_{rel,y}^2\right) =$$
$$= 0.5 \cdot \left(1 + 0.1 \cdot (1.311 - 0.3) + 1.311^2\right) = 1.410$$
$$k_z = 0.5 \cdot \left(1 + \beta_c \cdot (\lambda_{rel,z} - 0.3) + \lambda_{rel,z}^2\right) =$$
$$= 0.5 \cdot \left(1 + 0.1 \cdot (0.654 - 0.3) + 0.654^2\right) = 0.732$$

Buckling coefficient

$$k_{c,y} = \frac{1}{k_y + \sqrt{k_y^2 - \lambda_{rel,y}^2}} = \frac{1}{1.410 + \sqrt{1.410^2 - 1.311^2}} = 0.519$$

$$k_{c,z} = \frac{1}{k_z + \sqrt{k_z^2 - \lambda_{rel,z}^2}} = \frac{1}{0.732 + \sqrt{0.732^2 - 0.654^2}} = 0.944$$

Design ratio

$$\frac{\sigma_{m,y,d}}{f_{m,y,d}} + \frac{\sigma_{c,0,d}}{k_{c,y} \cdot f_{c,0,d}} = \frac{6.78}{22.15} + \frac{0.42}{0.519 \cdot 18.35} = 0.35$$

$$\frac{\sigma_{m,y,d}}{f_{m,y,d}} + k_m \cdot \frac{\sigma_{c,0,d}}{k_{c,z} \cdot f_{c,0,d}} = \frac{6.78}{22.15} + 0.7 \cdot \frac{0.42}{0.944 \cdot 18.35} = 0.24$$

governing: 0.38 < 1

⊟ Design Ratio					
Normal Force (Compression)	N_d	78.64	kN		
Cross-Sectional Area	A	1890.00	cm^2		
Compressive Stress	$\sigma_{c,0,d}$	0.42	N/mm^2		Eq. (6.36)
Equivalent Member Length	$l_{ef,y}$	14.99	m		
Equivalent Member Length	$l_{ef,z}$	3.64	m		
Radius of Inertia	i_y	17.77	cm		
Radius of Inertia	i_z	8.66	cm		
Slenderness Degree	λ_y	84.347			
Slenderness Degree	λ_z	42.051			
Relative Slenderness Ratio	$\lambda_{rel,y}$	1.312		> 0.3	Eq. (6.21),
Relative Slenderness Ratio	$\lambda_{rel,z}$	0.654		> 0.3	Eq. (6.22),
Straightness Factor	β_c	0.100			Eq. (6.29)
Auxiliary Buckling Coefficient	k_y	1.411			Eq. (6.27)
Auxiliary Buckling Coefficient	k_z	0.732			Eq. (6.28)
Buckling Coefficient	$k_{c,y}$	0.518			Eq. (6.25)
Buckling Coefficient	$k_{c,z}$	0.944			Eq. (6.26)
Compressive Strength	$f_{c,0,k}$	26.50	N/mm^2		[7], Tab.2
Partial Factor	γ_M	1.300			Tab. 2.3
Modification Factor	k_{mod}	0.900			Tab. 3.1
Compressive Strength	$f_{c,0,d}$	18.35	N/mm^2		Eq. (2.14)
Modulus of Elasticity	$E_{0,05}$	11100.00	N/mm^2		[7], Tab.2
Reduction Factor	k_m	0.700			6.1.6
Moment	$M_{y,d}$	134.63	kNm		
Section Modulus	W_y	19845.00	cm^3		
Bending Stress	$\sigma_{m,y,d}$	6.78	N/mm^2		
Bending Strength	$f_{m,y,k}$	32.00	N/mm^2		[7], Tab.2
Bending Strength	$f_{m,y,d}$	22.15	N/mm^2		Eq. (2.14)
Design 1	η_1	0.35		≤ 1	Eq. (6.23)
Design 2	η_2	0.24		≤ 1	Eq. (6.24)
Design Ratio	η	0.35		≤ 1	

Figure 13.18: Stability analysis for uniaxial bending and compression according to 6.3.2

Finger-jointed connection acc. to DIN EN 1995-1-1/NA 2009, NA.11.3

For the design of the finger-jointed connection we first calculate the buckling lengths and stresses as presented above. Again, the internal forces of RC2 prove to be governing. In this case, they are available on the location X = 3.12.

Buckling length coefficient, horizontal beam $\beta_R = \beta_S \cdot \sqrt{\frac{I_R \cdot N_S}{I_S \cdot N_R}} \cdot \frac{h}{s} =$

$$= 4.15 \cdot \sqrt{\frac{4.814E+09 \cdot 81610}{5.83E+09 \cdot 72860}} \cdot \frac{3641}{7683} = 1.89$$

Slenderness

$$\lambda_R = \frac{l_{ef,R}}{i_y} = \frac{\beta_R \cdot s}{i_y} = \frac{1.89 \cdot 768cm}{16.77cm} = 86.6$$

Design ratio
$$\frac{f_{c,0,d}}{f_{c,\alpha,d}} \cdot \left(\frac{\sigma_{c,0,dS}}{k_{cS} \cdot f_{c,0,d}} + \frac{\sigma_{m,dS}}{f_{m,d}} \right) = \frac{15.59}{12.47} \cdot \left(\frac{0.5}{0.511 \cdot 15.59} + \frac{8.59}{18.83} \right) = 0.65$$

$$\frac{f_{c,0,d}}{f_{c,\alpha,d}} \cdot \left(\frac{\sigma_{c,0,dR}}{k_{cR} \cdot f_{c,0,d}} + \frac{\sigma_{m,dR}}{f_{m,d}} \right) = \frac{15.59}{12.47} \cdot \left(\frac{0.43}{0.489 \cdot 15.59} + \frac{8.5}{18.83} \right) = 0.64$$

governing: $0.65 < 1$

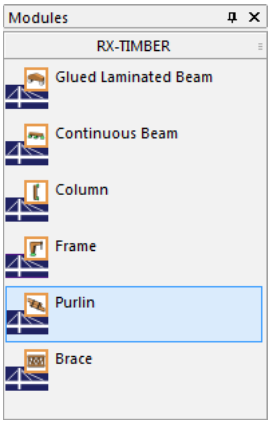
Design Ratio					
Axial Force at Column	N_s	81.61	kN		
Moment at Column	M_s	168.34	kNm		
Column Height	h	3.64	m		
Cross-Section Width	b	30.00	cm		
Cross-Section Height, Colum	$h_{s\,0,65}$	61.55	cm		
Moment of Inertia for Column	I_s^*	583018.00	cm^4		
Buckling Length Factor for C	β_S^*	4.150			Tab. NA.23
Column Slenderness	λ_S^*	85.047			
Related Slenderness	$\lambda_{rel,cS}^*$	1.323			Eq. (6.21)
Factor	β_c	0.100			Eq. (6.29)
Buckling Coefficient	k_S^*	1.426			Eq. (6.27)
Buckling Factor	k_{cS}^*	0.511			Eq. (6.25)
Axial Force at Beam	N_r	72.86	kN		
Moment at Beam	M_r	166.68	kNm		
Column Length	s	7.68	m		
Cross-Section Height, Beam	$h_{r\,0.65}$	0.58	m		
Moment of Inertia for Beam S	I_r^*	481392.00	cm^4		
Buckling Length Factor for B	β_R^*	1.891			Tab. NA.23
Beam Slenderness	λ_R^*	87.180			
Related Slenderness	$\lambda_{rel,cR}^*$	1.356			Eq. (6.21)
Buckling Factor	k_R^*	1.472			Eq. (6.27)
Buckling Factor	k_{cR}^*	0.489			Eq. (6.25)
Angle	α	20.72	°		
Bending Strength	$f_{m,k}$	32.00	N/mm^2		
Compressive Strength	$f_{c,0,k}$	26.50	N/mm^2		
Transversal Compressive Str	$f_{c,90,k}$	3.00	N/mm^2		
Shear Strength	$f_{v,k}$	3.50	N/mm^2		
Partial Factor	γ_M	1.300			Tab. 2.3
Modification Factor	k_{mod}	0.900			Tab. 3.1
Bending Strength	$f_{m,d}$	18.83	N/mm^2	11.3(NA6)	Eq. (2.14)
Compressive Strength	$f_{c,0,d}$	15.59	N/mm^2	11.3(NA6)	Eq. (2.14)
Transversal Compressive Str	$f_{c,90,d}$	2.08	N/mm^2		Eq. (2.14)
Shear Strength	$f_{v,d}$	2.42	N/mm^2		Eq. (2.14)
Transv. Compressive Strengt	$f_{c,\alpha,d}$	12.47	N/mm^2		Eq. (NA.152)
Axial Stress	$\sigma_{c,0,dS}$	0.49	N/mm^2		
Bending Stress	$\sigma_{m,dS}$	8.59	N/mm^2		
Design Ratio 1	η_1	0.65		≤ 1	Eq. (NA.147)
Axial Stress	$\sigma_{c,0,dR}$	0.43	N/mm^2		
Bending Stress	$\sigma_{m,dR}$	8.50	N/mm^2		
Design Ratio 2	η_2	0.64		≤ 1	Eq. (NA.147)
Design Ratio	η	0.65		≤ 1	Eq. (NA.147)

Figure 13.19: Design of finger-jointed connection

13.3.3 Other Results Windows

The remaining results windows as well as the printout report documentation are similar to the ones described in the examples presented in chapter 10 and 11.

14. Purlin



This example presents the ultimate limit state designs of a coupled purlin with irregular span spacings. The analysis is performed according to EN 1995-1-1:2004-11 with the national annex ÖNORM B 1995-1-1:2009-07.

The model data is stored in the model *01.pft* of the project *Examples RX-TIMBER*. We find it among the example files of the program **Purlin**.

14.1 System and Loads

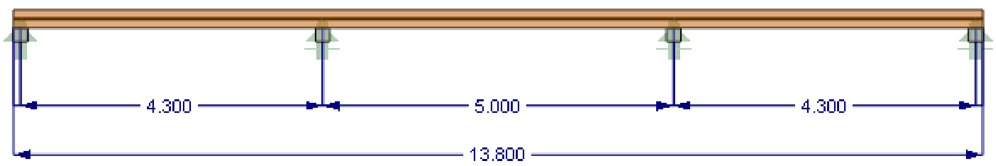


Figure 14.1: System

Model

Cross-section:	b/d = 14/26 cm	
Material:	coniferous timber C24	
Span lengths:	$l_1 = l_3 = 4.30m$	$l_2 = 5.00m$
Roof inclination:	6.0°	
Purlin axis:	inclined in roof plane	
Coupling:	nail connection d = 8 mm, l = 280 mm as shown in Figure 14.2	

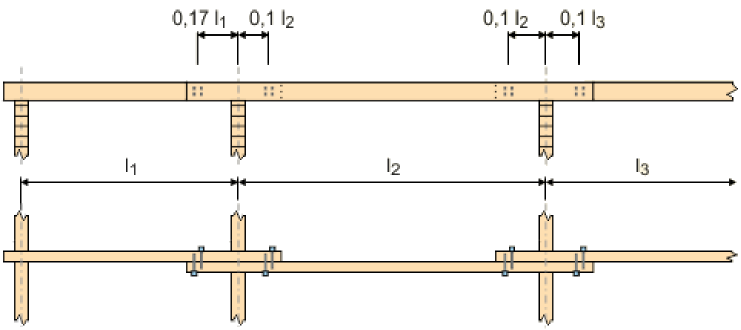


Figure 14.2: Purlin connection plan

Loading

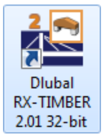
Load case 1: self-weight and roof finishes	g = 1.6 kN/m	LDC = permanent
Load case 2: snow load	SZ 2, 400 m	LDC = short-term
Load case 3: wind load	terrain category II	LDC = short-term

The loads for snow and wind are determined automatically by the load generators integrated in the program.

The actions for the ultimate limit state design need to be combined according to EN 1990 and EN 1995 with the national annex for Austria.

14.2 Input of Model Data

14.2.1 General Data



Start the program by double-clicking the desktop icon **Dlubal RX-TIMBER 2.xx** (see chapter 3.2, page 12), and then select **Purlin** in the Project Manager.

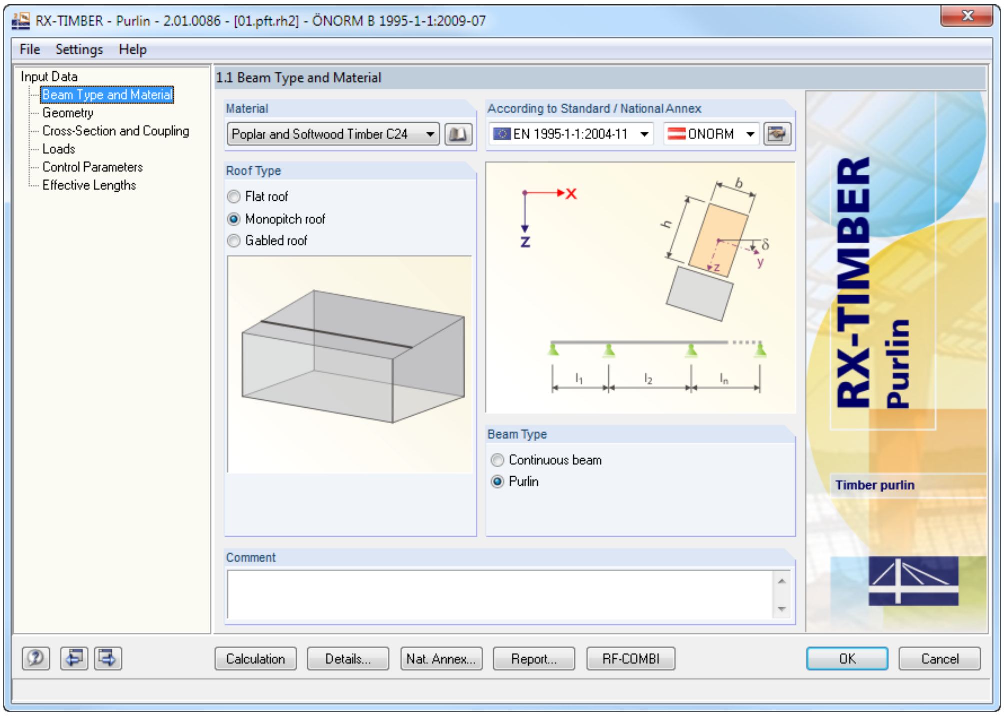


Figure 14.3: Window 1.1 *Beam Type and Material*

We select **softwood timber** with the strength grade **C24** as *Material*.

The *Roof Type* is a **Monopitch roof**. We design the beam *According to Standard* **EN 1995-1-1: 2004-11** and the *National Annex* for Austria according to **ÖNORM**.

The *Beam Type* is represented by a **Purlin**. Depending on our selection, we can define releases or couplings in the subsequent program windows.

14.2.2 Geometry

In the second window, we enter the number of spans, the span lengths as well as the roof parameters and support conditions of the purlin.

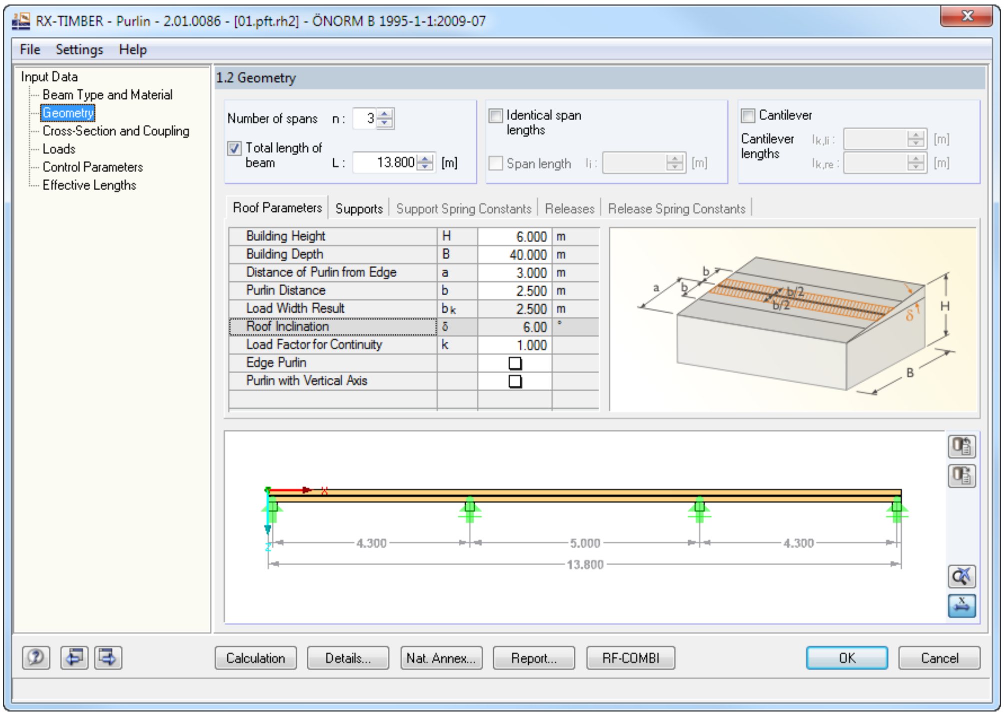
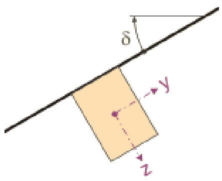


Figure 14.4: Window 1.2 *Geometry*, tab *Roof Parameters*

For our example we set:

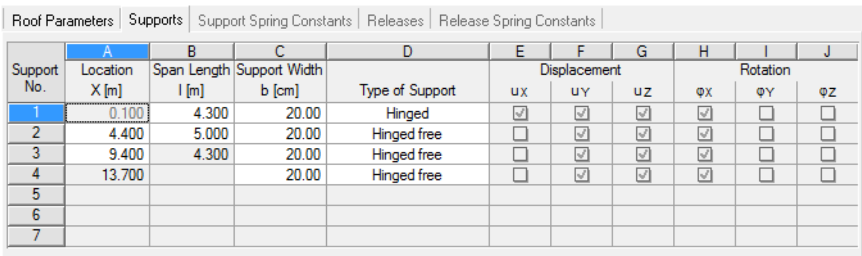
- *Number of spans n*: **3**
- *Total length L*: **13.80 m**

As the span lengths are different in size, we clear the check box for *Identical span lengths*.



We enter the geometric settings in the *Roof Parameters* tab as shown in the figure above. The purlin's axes are adjusted to the roof's inclination of **6.0°**. Therefore, we clear the check box for *Purlin with Vertical Axis*.

In the second tab *Supports*, we can define the span lengths and support widths.



Roof Parameters | Supports | Support Spring Constants | Releases | Release Spring Constants

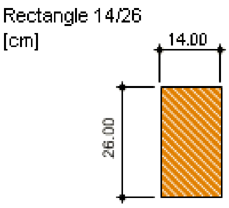
	A	B	C	D	E	F	G	H	I	J
Support No.	Location X [m]	Span Length l [m]	Support Width b [cm]	Type of Support	Displacement uX	uY	uZ	Rotation φX	φY	φZ
1	0.100	4.300	20.00	Hinged	☑	☑	☑	☑	☐	☐
2	4.400	5.000	20.00	Hinged free	☐	☑	☑	☑	☐	☐
3	9.400	4.300	20.00	Hinged free	☐	☑	☑	☑	☐	☐
4	13.700		20.00	Hinged free	☐	☑	☑	☑	☐	☐
5										
6										
7										

Figure 14.5: Window 1.2 *Geometry*, tab *Supports*

The *Location X* results from the span length and the support width that we enter as shown in the figure above. We define the purlin restrained on the first support in longitudinal direction, the remaining supports are hinged and can be shifted freely in X.

14.2.3 Cross-section and Coupling

In the next window, we define the cross-section dimensions, the couplings and the coupling elements.



For the *Cross-Section* we enter the *Dimensions* **b = 14 cm** and **h = 26 cm**.

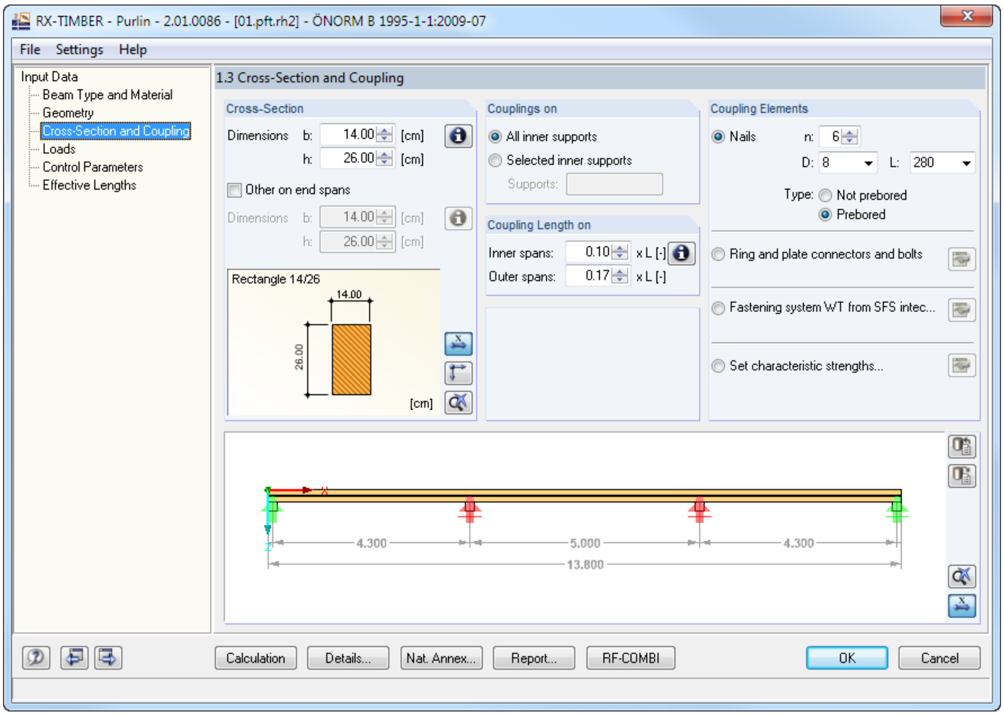


Figure 14.6: Window 1.3 *Cross-Section and Coupling*

The *Couplings* are provided for **All inner supports**.

We don't modify the preset *Coupling Length*s. We use the [Info] button shown on the left to check the connection plan of the purlin (see Figure 14.2, page 148).

We use **Nails** as *Coupling Elements:*

- Number *n*: **6**
- Diameter *D*: **8 mm**
- Length *L*: **280 mm**
- *Type*: **Prebored**

14.2.4 Loads

This window manages the different types of loads.

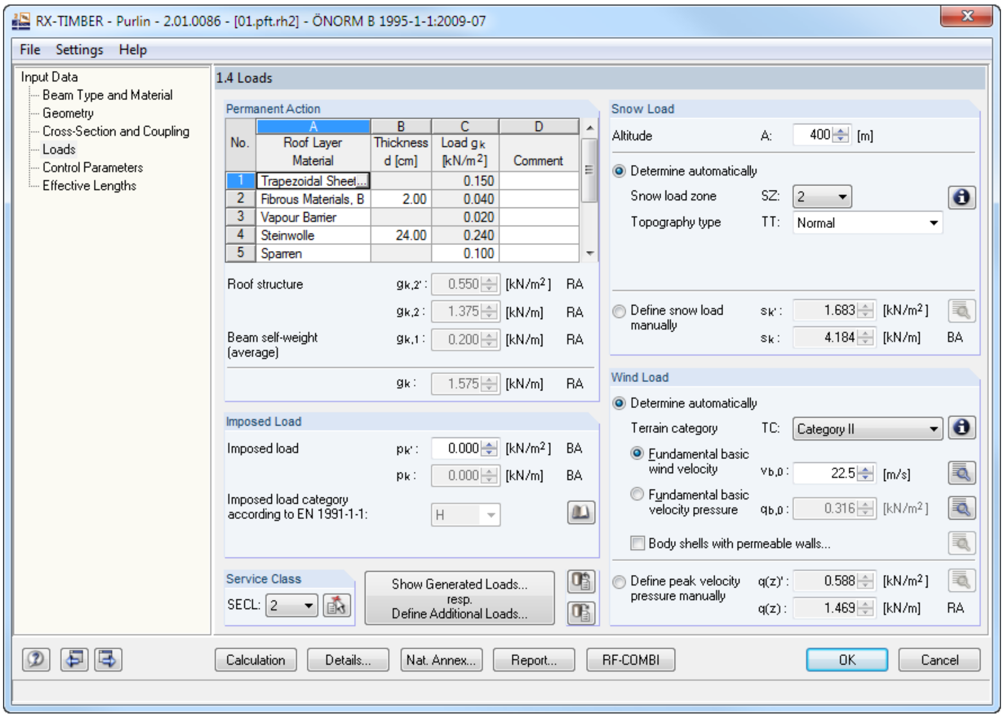


Figure 14.7: Window 1.4 *Loads*

We enter the *Permanent Action* as shown in the figure above. We use the [...] button to quickly select the materials in the library.

There is no *Imposed load* available in our example.

We specify the *Snow Load* with an altitude of **400 m** above sea level and *Determine* the load *automatically*.

- *Snow load zone SZ*: **2**
- *Topography type TT*: **Normal**

We click the [Info] button to open a map where the snow load zone can be selected also graphically (see Figure 14.8).

The *Wind Load* is *Determine*d *automatically* as well. We just specify the **Terrain category II**.

We click the [Info] button shown on the left to get detailed information on the single terrain categories.

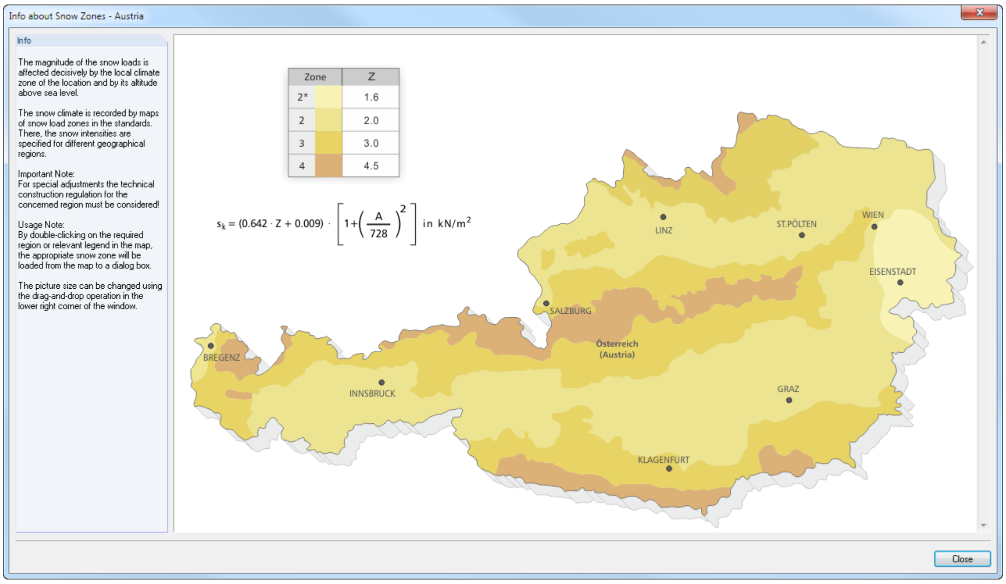


Figure 14.8: Map showing snow load zones for Austria

We use the button [Show Generated Loads] shown on the left to open the dialog box *Load Cases* where we can check the generated loads.

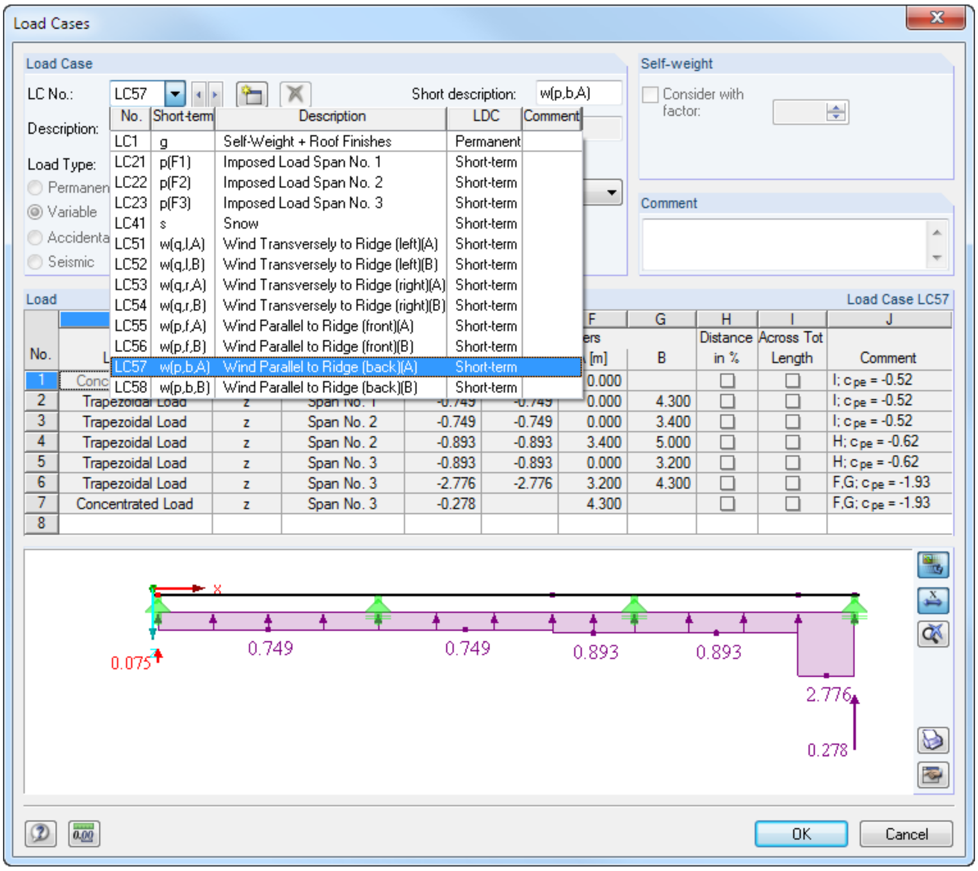


Figure 14.9: Dialog box *Load Cases*

14.2.5 Control Parameters

In window 1.5 *Control Parameters*, we set only the **Ultimate limit state** for the design (the serviceability limit state design is described in chapter 11.4.4 on page 122).

In the window section *Calculation Parameters*, we activate the **Moment redistribution** according to ÖNORM 1990. We don't modify the default setting of 10 %.

Finally, we define the number of *member divisions* with **20**.

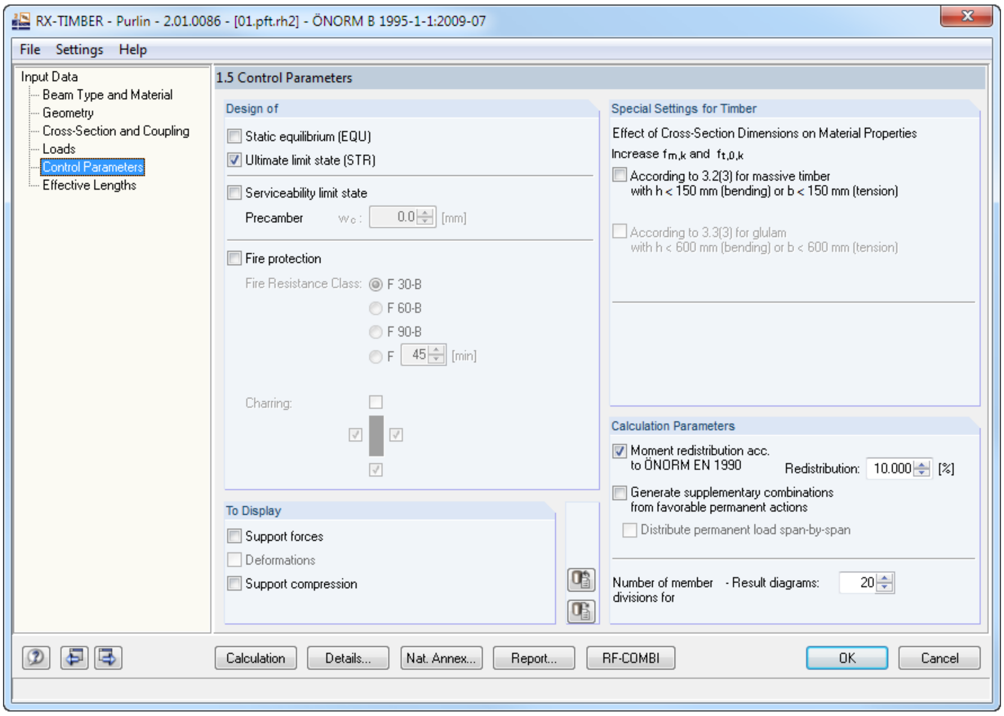


Figure 14.10: Window 1.5 *Control Parameters*

We use the button [Nat. Annex] to open the dialog box *National Annex Settings* where we can check the partial safety and modification factors preset for ÖNORM B 1995-1-1. It is not necessary to change them.

14.2.6 Effective Lengths

To be on the safe side, we select the factor β with **1.0** in our example. The values are preset by the program so that no data modification is required in this window.

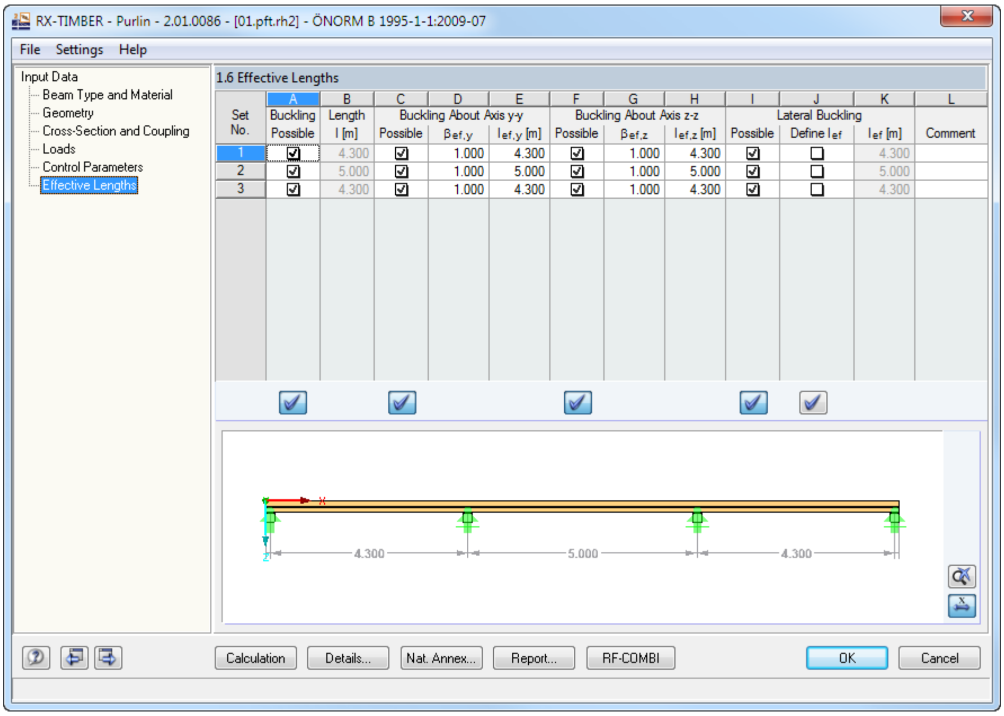


Figure 14.11 Window 1.6 *Effective Lengths*

14.2.7 Details

We click the [Details] button to open a dialog box offering specific design settings.

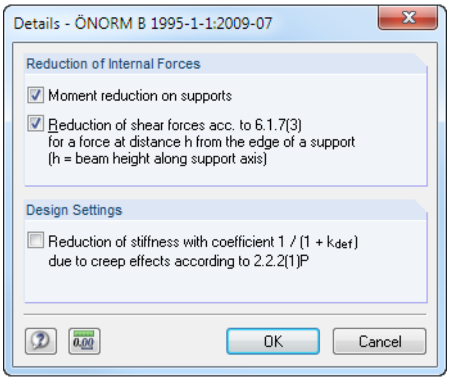


Figure 14.12: Dialog box *Details*

In this dialog box, we tick the options *Moment reduction on supports* and *Reduction of shear forces*.

14.2.8 RF-COMBI

We use the button [RF-COMBI] shown on the left to access the add-on module RF-COMBI integrated in RX-TIMBER. In RF-COMBI we can check the combinations generated in the program's background according to EN 1990 and EN 1995. The load cases are combined automatically so that we normally don't need to open the add-on module.

We have a look at window 1.2 *Load Cases in Actions*. For our simple example, the actions and respective load-duration classes are created correctly by the program.

In action AC4 *Wind Loads* the load cases 51 to 58 are summarized as acting alternatively. The load-duration class is preset with *Short-term*.

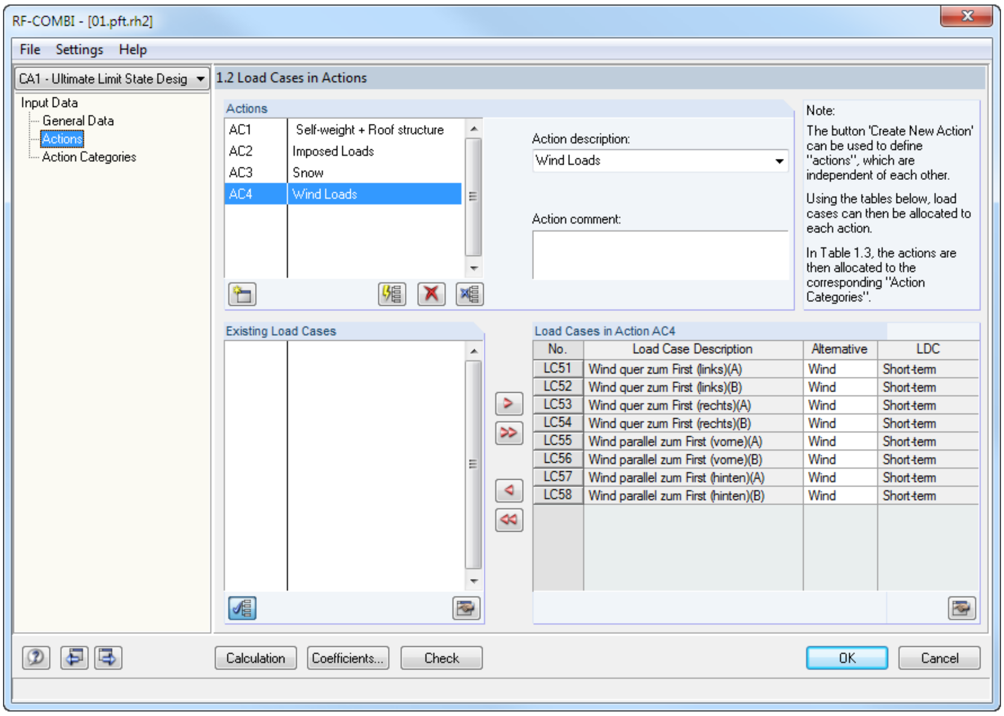


Figure 14.13: RF-COMBI window 1.2 *Load Cases in Actions*

For more detailed information about the features of RF-COMBI, see the manual available for download at www.dlubal.com. We click [OK] to return to *RX-TIMBER Purlin*.

14.3 Results

14.3.1 Result Combinations

We use the [Calculation] button to calculate the entered data. Window 2.1 shows the ultimate limit state designs for all combinations together with the respective design ratios.

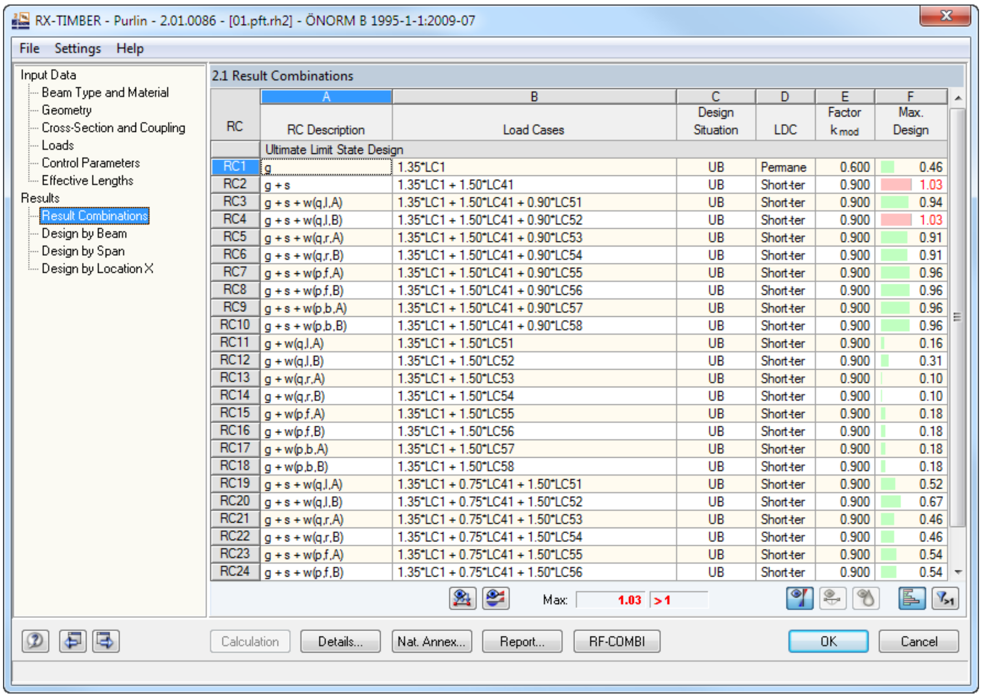


Figure 14.14: Window 2.1 *Result Combinations*

14.3.2 Ultimate Limit State Designs

Window 2.2 *Design by Beam* lists the governing designs of the purlin.

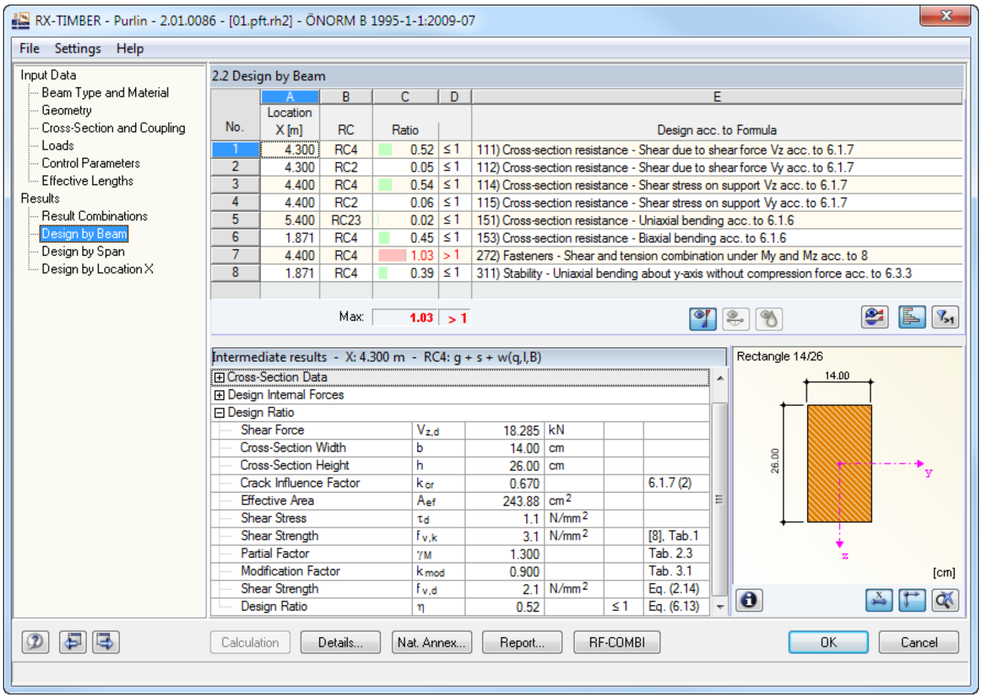


Figure 14.15: Window 2.2 *Design by Beam*

All factors and design internal forces can be checked interactively: We click a design in the table above to display all *Intermediate results* of the design in the window section below.

The results of RX-TIMBER are checked by manual calculations.

Shear due to shear force V_z according to 6.1.7

The maximum shear force of 19.00 kN occurs on the inner supports in RC4.

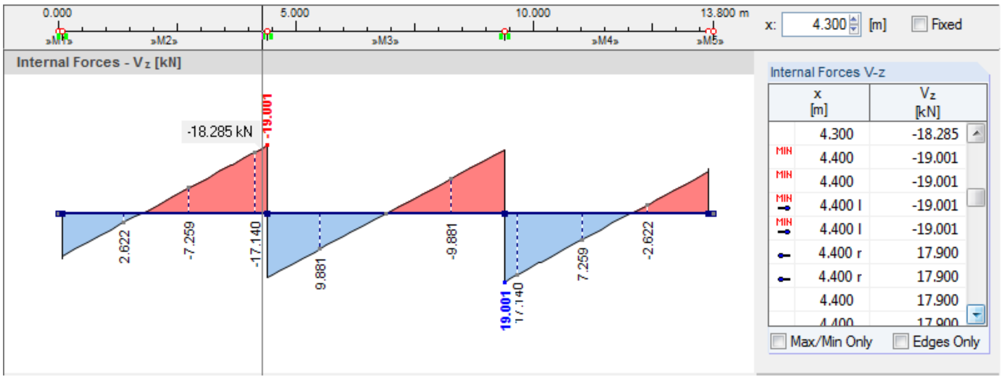


Figure 14.16: Distribution of shear force V_z in RC4

The support width of 20 cm results in a reduced design shear force of 18.22 on the support edge.

Shear stress
$$\tau_d = 1.5 \cdot \frac{V_d}{b_{ef} \cdot h} = 1.5 \cdot \frac{18.22\ \text{kN}}{9.6\ \text{cm} \cdot 26\ \text{cm}} = 0.11\ \text{kN/cm}^2 = 1.10\ \text{N/mm}^2$$

where $b_{ef} = k_{cr} \cdot b = 0.67 \cdot 14\ \text{cm} = 9.6\ \text{cm}$

Shear strength
$$f_{v,d} = \frac{k_{mod} \cdot f_{v,k}}{\gamma_m} = \frac{0.9 \cdot 3.10\ \text{N/mm}^2}{1.3} = 2.15\ \text{N/mm}^2$$

Design ratio
$$\frac{\tau_d}{f_{v,d}} = \frac{1.10}{2.15} = 0.51 < 1$$

Biaxial bending according to 6.1.6

The maximum loading occurs in the middle of the first span, again in RC4. The graphic with the result diagrams below shows the mathematical diagrams as well as the moment diagrams reduced or redistributed for the design.

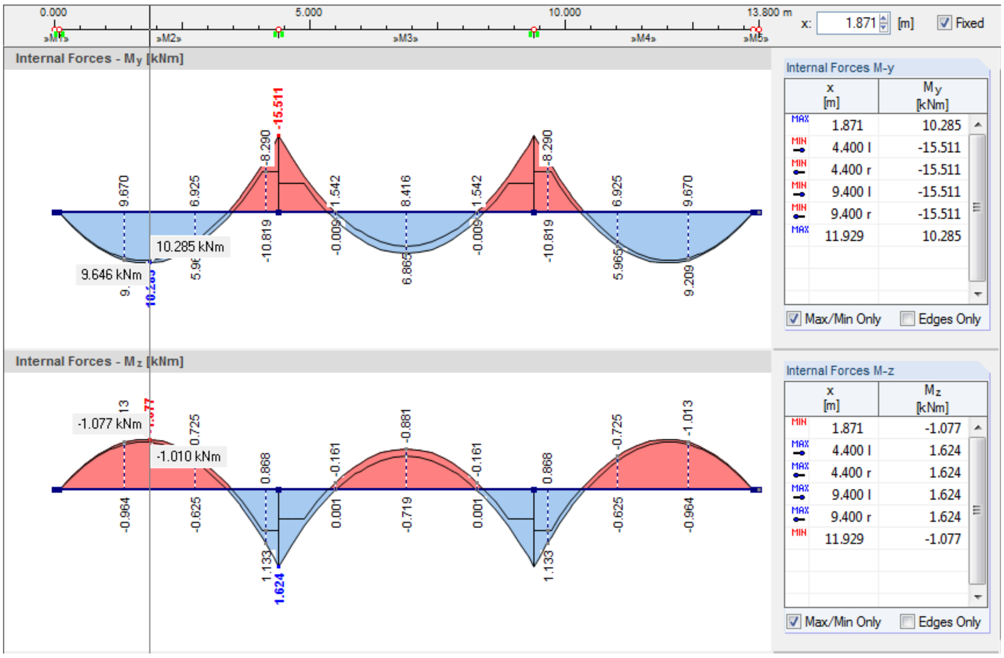


Figure 14.17: Distribution of moments M_y and M_z in RC4 with representation of original and modified values

Bending stresses
$$\sigma_{m,y,d} = \frac{M_{y,d}}{W_y} = \frac{10.25\ \text{kNm} \cdot 10^6}{1577\ \text{cm}^3 \cdot 10^3} = 6.50\ \text{N/mm}^2$$

$$\sigma_{m,z,d} = \frac{M_{z,d}}{W_z} = \frac{1.07\ \text{kNm} \cdot 10^6}{849.3\ \text{cm}^3 \cdot 10^3} = 1.26\ \text{N/mm}^2$$

Flexural resistance
$$f_{m,d} = k_{mod} \cdot \frac{f_{m,k}}{\gamma_M} = 0.9 \cdot \frac{24\ \text{N/mm}^2}{1.3} = 16.6\ \text{N/mm}^2$$

Design ratio
$$\frac{\sigma_{m,y,d}}{f_{m,d}} + k_m \cdot \frac{\sigma_{m,z,d}}{f_{m,d}} = \frac{6.50}{16.6} + 0.7 \cdot \frac{1.26}{16.6} = 0.44$$

$$k_m \cdot \frac{\sigma_{m,y,d}}{f_{m,d}} + \frac{\sigma_{m,z,d}}{f_{m,d}} = 0.7 \cdot \frac{6.50}{16.6} + \frac{1.26}{16.6} = 0.35$$

governing: $0.44 < 1$

2.2 Design by Beam

No.	A Location X [m]	B RC	C Ratio	D	E Design acc. to Formula
1	4.300	RC4	0.52	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
2	4.300	RC2	0.05	≤ 1	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7
3	4.400	RC4	0.54	≤ 1	114) Cross-section resistance - Shear stress on support Vz acc. to 6.1.7
4	4.400	RC2	0.06	≤ 1	115) Cross-section resistance - Shear stress on support Vy acc. to 6.1.7
5	5.400	RC23	0.02	≤ 1	151) Cross-section resistance - Uniaxial bending acc. to 6.1.6
6	1.871	RC4	0.45	≤ 1	153) Cross-section resistance - Biaxial bending acc. to 6.1.6
7	4.400	RC4	1.03	> 1	272) Fasteners - Shear and tension combination under My and Mz acc. to 8
8	1.871	RC4	0.39	≤ 1	311) Stability - Uniaxial bending about y-axis without compression force acc. to 6.3.3
		Max:	1.03	> 1	

Intermediate results - X: 1.871 m - RC4: g + s + w(q,l,B)

Moment	$M_{y,d}$	10.285	kNm		
Moment	$M_{z,d}$	1.077	kNm		
Section Modulus	W_y	1577.33	cm³		
Section Modulus	W_z	849.33	cm³		
Bending Stress	$\sigma_{m,y,d}$	6.5	N/mm²		
Bending Stress	$\sigma_{m,z,d}$	1.3	N/mm²		
Bending Strength	$f_{m,y,k}$	24.0	N/mm²		[8], Tab.1
Bending Strength	$f_{m,z,k}$	24.0	N/mm²		[8], Tab.1
Partial Factor	γ_M	1.300			Tab. 2.3
Modification Factor	k_{mod}	0.900			Tab. 3.1
Bending Strength	$f_{m,y,d}$	16.6	N/mm²		Eq. (2.14)
Bending Strength	$f_{m,z,d}$	16.6	N/mm²		Eq. (2.14)
Reduction Factor	k_m	0.700			6.1.6
Design 1	η_1	0.45		≤ 1	Eq. (6.11)

Rectangle 14/26

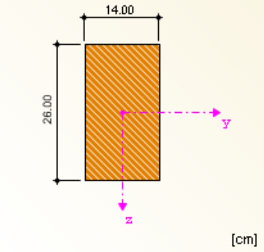


Figure 14.18: Designs for biaxial bending

Coupling elements for shear and tension due to biaxial bending acc. to 8

We have to design the coupling elements for both sides of the coupling. RX-TIMBER determines the maximum design criterion for the right side of the first intermediate support. Again, the internal forces of RC4 are governing.

Design Ratio					
Left Side					
Design Moment	$M_{y,r}$	-5.905	kNm		
Design Moment	$M_{z,r}$	0.618	kNm		
Coupling-to-Support Distance	u_l	0.731	m		
Design Force in the Coupling	$K_{z,l}$	8.078	kN		
Design Force in the Coupling	$K_{y,l}$	0.846	kN		
Coupling					
Type	Nails				
Diameter	D	8.0	mm		
Length	L	280.0	mm		
Opening type	Preborec				
Number	n	6			
Characteristic Resistance	$F_{v,z,Rk}$	28.405	kN		
Characteristic Resistance	$F_{v,y,Rk}$	13.171	kN		
Modification Factor	k_{mod}	0.900			Tab. 3.1
Partial Factor	γ_M	1.300			Tab. 2.3
Design Resistance	$F_{v,z,Rd}$	19.665	kN		
Design Resistance	$F_{v,y,Rd}$	9.119	kN		
Design - Left Side		0.50		≤ 1	
Right Side					
Design Moment	$M_{y,l}$	-8.290	kNm		
Design Moment	$M_{z,l}$	0.868	kNm		
Coupling-to-Support Distance	u_r	0.500	m		
Design Force in the Coupling	$K_{z,r}$	16.579	kN		
Design Force in the Coupling	$K_{y,r}$	1.736	kN		
Coupling					
Type	Nails				
Diameter	D	8.0	mm		
Length	L	280.0	mm		
Opening type	Preborec				
Number	n	6			
Characteristic Resistance	$F_{v,z,Rk}$	28.405	kN		
Characteristic Resistance	$F_{v,y,Rk}$	13.171	kN		
Modification Factor	k_{mod}	0.900			Tab. 3.1
Partial Factor	γ_M	1.300			Tab. 2.3
Design Resistance	$F_{v,z,Rd}$	19.665	kN		
Design Resistance	$F_{v,y,Rd}$	9.119	kN		
Design - Right Side		1.03		> 1	
Design Ratio	η	1.03		> 1	

Figure 14.19: Design of coupling elements

Right side

Design force in couplings $K_{z,d} = \frac{M_{y,d}}{u} = \frac{8.26\ kNm}{0.5\ m} = 16.52\ kN$

$K_{y,d} = \frac{M_{z,d}}{u} = \frac{0.87\ kNm}{0.5\ m} = 1.73\ kN$

Bearing capacity of coupling $F_{v,z,Rd} = n_{ef} \cdot F_{v,Rk} = 6 \cdot 4.734\ kN = 28.41\ kN$

$F_{v,y,Rd} = n_{ef} \cdot F_{ax,Rk} = 6 \cdot 2.195\ kN = 13.17\ kN$

where Load bearing capacity for nail per shear joint according to eq. (8.6) with governing condition (f) (percentage of cable effects are not applied in RX-TIMBER)

$$F_{v,Rk} = 1.15 \cdot \sqrt{2 \cdot M_{y,Rk} \cdot f_{h,1,k} \cdot d} =$$

$$= 1.15 \cdot \sqrt{2 \cdot 40115 \cdot 26.40 \cdot 8} \cdot 10^{-3} = 4.734\ kN$$

Pull-off strength for nail according to eq. (8.23) with governing condition (b)

$$F_{ax,Rk} = f_{head,k} \cdot d_h^2 = 8.58 \cdot 16^2 \cdot 10^{-3} = 2.195\ kN$$

Designed bearing capacity for loading perpendicular to nail axis

$$F_{v,z,Rd} = k_{mod} \cdot \frac{F_{v,z,Rk}}{\gamma_M} = 0.9 \cdot \frac{28.41\ kN}{1.3} = 19.67\ kN$$

$$F_{v,y,Rd} = k_{mod} \cdot \frac{F_{v,y,Rk}}{\gamma_M} = 0.9 \cdot \frac{13.17\ kN}{1.3} = 9.12\ kN$$

Design ratio $\frac{K_{z,d}}{F_{v,z,d}} + \frac{K_{y,d}}{F_{v,y,d}} = \frac{16.52}{19.67} + \frac{1.73}{9.12} = 1.03 > 1$

Stability analysis for uniaxial bending without compressive force acc. to 6.3.3

The lateral buckling design is performed for uniaxial bending about the member axis y.

The middle of the third span is determined as governing location. The bending stress on the location X = 11.929 is 6.5 N/mm² (see page 159).

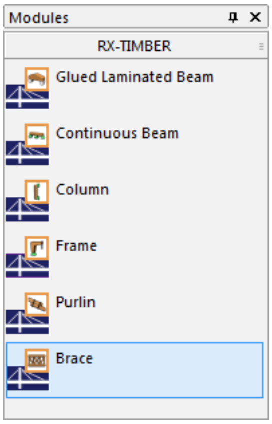
Slenderness $\lambda_{rel,m} = \sqrt{\frac{0.78 \cdot b^2}{h \cdot l_{ef}} \cdot E_{0.05}} = \sqrt{\frac{0.78 \cdot 140^2}{260 \cdot 4300} \cdot 7400} = 0.487$

Lateral buckling factor $k_{crit} = 1.0$

Design ratio $\frac{\sigma_{m,y,d}}{k_{crit} \cdot f_{m,y,d}} = \frac{6.5}{1.0 \cdot 16.62} = 0.39 > 1$

14.3.3 Other Results Windows

The remaining results windows as well as the printout report documentation are similar to the ones described in the examples presented in chapter 10 and 11.



15. Brace

The program RX-TIMBER Brace is also explained by an example: A stiffening brace is stabilizing the fish beam described in chapter 10.2

The model data is stored in the model *01.vrb* of the project *Examples RX-TIMBER*. We find it among the example files of the program **Brace**.

The short cantilever of the fish-bellied girder is structurally stiffened by an eaves board. The resulting load is directly transferred into the support.

15.1 System and Loads

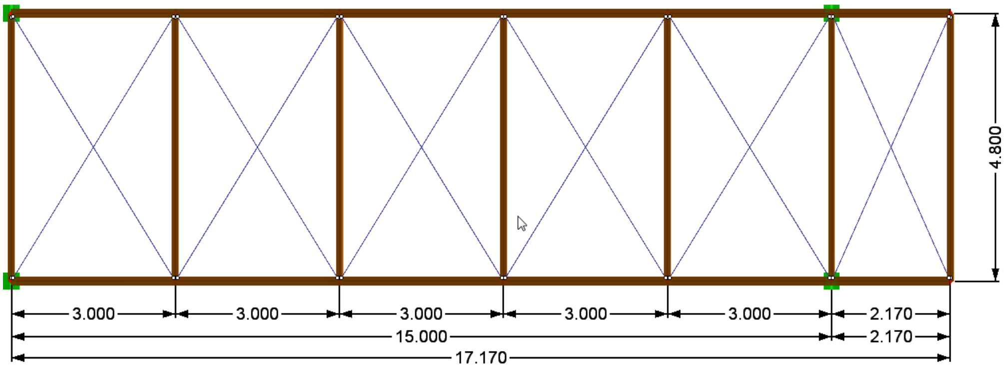


Figure 15.1: System

Model

	Cross-sections	Material
Upper/lower chord	20/39 cm	GL 24c
Verticals	12/12 cm	C 24
Diagonals	Round steel bars 20 mm	S 235

(Note: Half of the cross-section depth is used on the support for the chord.)

Roof shape: Monopitch roof with cantilevers

Building dimensions

Height: 6.0 m

Depth: 40.0 m

Geometry

Brace width: 4.8 m

Length: 15.0 m

Roof inclination: 5.0°

Cantilever length left: 0 m

Cantilever length right: 2.17 m

Inner span length: 3.0 m (in floor plan)

Number of bays: 5

Loading

The corresponding **equivalent load** is directly taken from the fish beam model *08.gl* that we find among the example files of the program *RX-TIMBER Glued Laminated Beam*.

Load case 1:	self-weight and roof finishes	q_g = 0.08 kN/m	LDC = permanent
Load case 41:	snow load	q_s = 0.04 kN/m	LDC = short-term
Load case 56:	wind parallel to ridge (A)	$q_{w(p,A)}$ = 0.04 kN/m	LDC = short-term
Load case 56:	wind parallel to ridge (B)	$q_{w(p,B)}$ = 0.03 kN/m	LDC = short-term

The wind loads are determined automatically by the load generator integrated in the program. To be on the safe side, they are applied only on the inner span of the bracing.

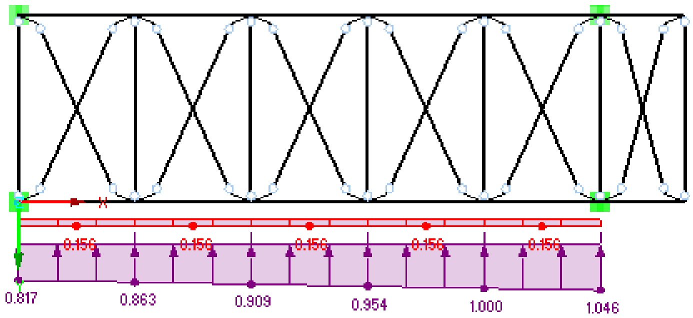


Figure 15.2: Wind load only in span

With the option *Number of Braces More than one* (in window 1.7 *Loads*) the wind loads "pressure on gable" (LC55) and "suction on gable" (LC56) are separately applied to the front respectively rear girder. The program automatically creates two new load cases, LC155 and LC156.

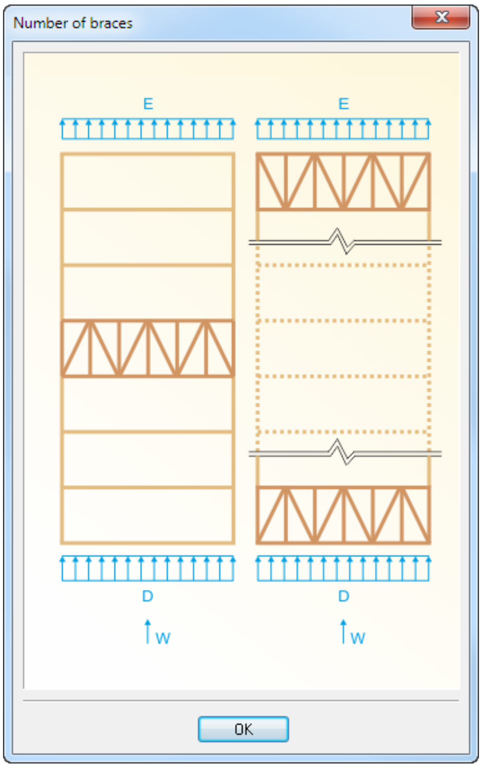
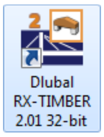


Figure 15.3: Number of braces

Equal load cases with reverse action directions are automatically created for other load cases, too. A changed load direction may have a positive effect especially with failing steel diagonals.

15.2 Input of Model Data

15.2.1 General Data



Start the program by double-clicking the desktop icon **Dlubal RX-TIMBER 2.xx** (see chapter 3.2, page 12), and then select **Brace** in the Project Manager.

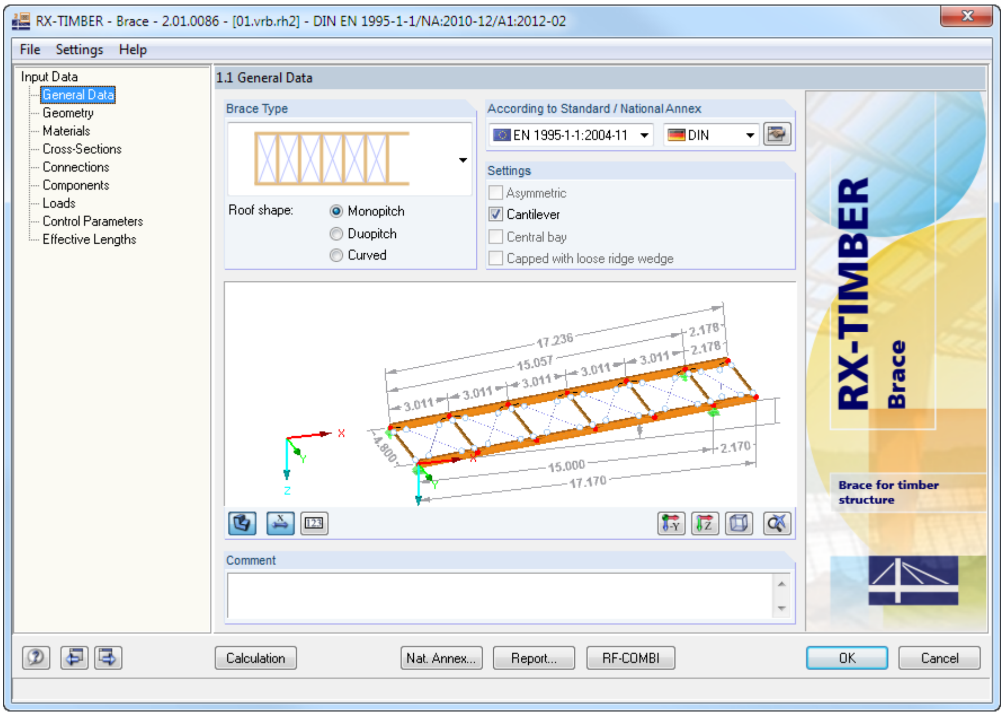


Figure 15.4: Window 1.1 *General Data*

For the *Roof shape* we select a **Monopitch** roof with **Cantilevers**.

We want to analyze the girder according to **DIN EN 1995-1-1**.

We click the [Edit] button shown on the left to open the dialog box *National Annex Settings* where we can check the standard's parameters (see figure below).

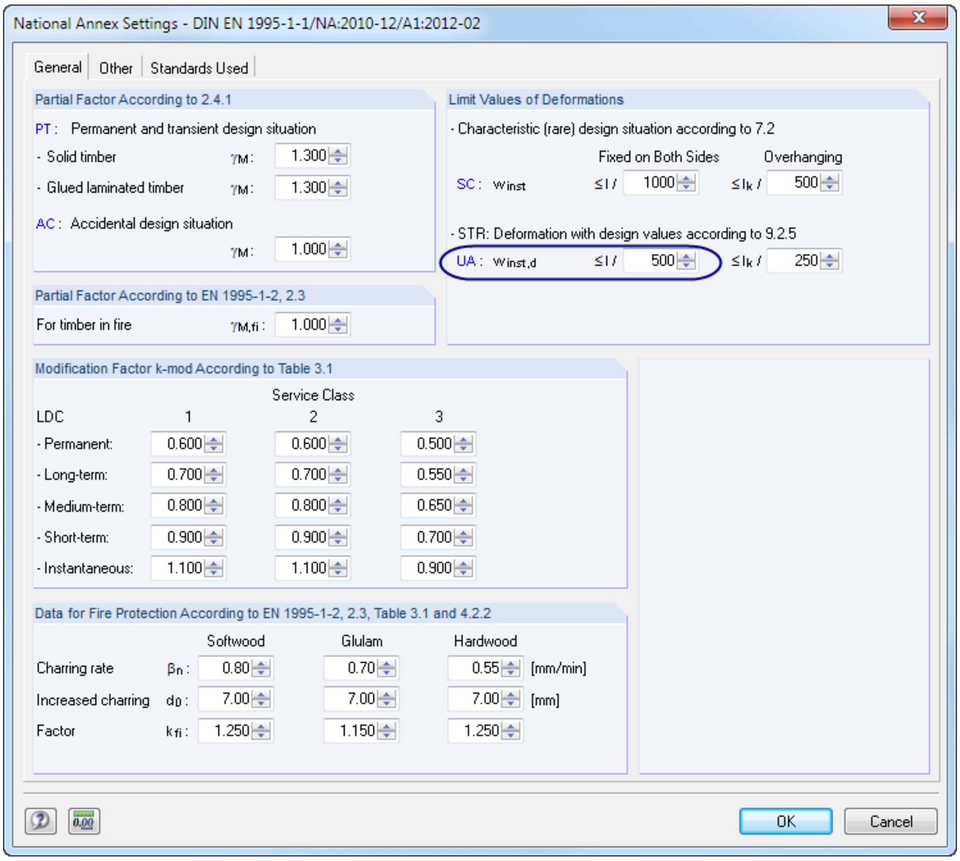


Figure 15.5: Dialog box *National Annex Settings - DIN EN 1995-1-1*

In accordance with clause 9.2.5 of DIN EN 1995-1-1, a deformation analysis for stiffening bracings must be performed also for the ultimate limit state design if we don't carry out any more accurate design. We restrict the deformations to **l/500** in the dialog section *Limit Values of Deformations*.

A limit of l/1000 is preset for the serviceability limit state design. In accordance with clause 7.2 of the standard, we could also calculate with l/200 or similar values. In this case, the serviceability limit state design would no longer be governing, of course, because then the mean values of the stiffnesses would be used for calculating the deformations. However, for the common setting of l/1000 as limit deformation there are still some cases imaginable where the serviceability limit state design becomes governing.

To avoid creating a multitude of combinations, we want to combine the load cases for the characteristic situation including all loads so that all defined load cases are always combined. So a division into the other design requirements of the standard is not necessary.

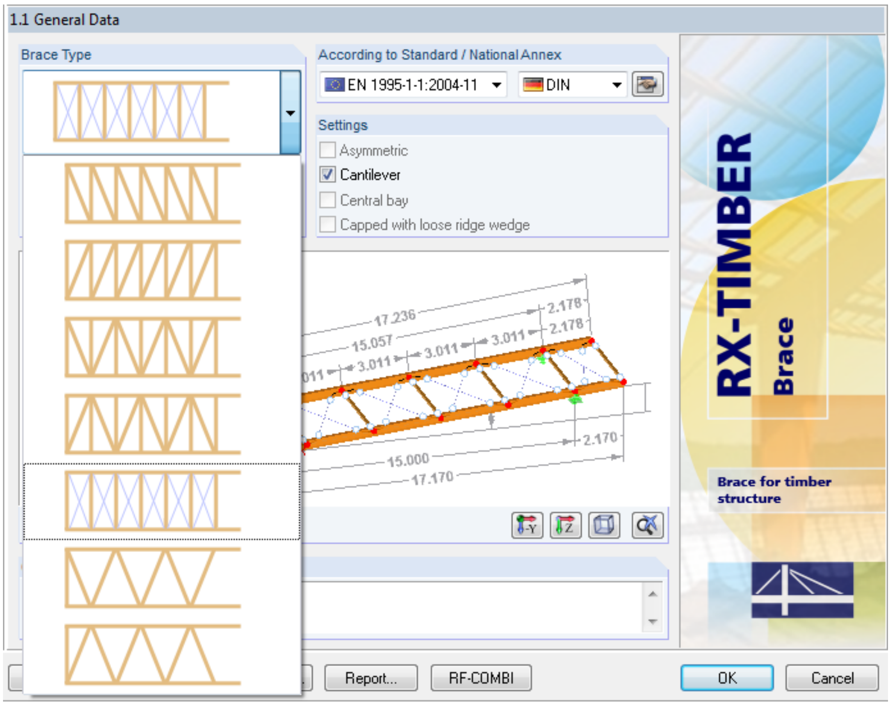


Figure 15.6: Window 1.1 *General Data*, list *Brace Type*

We define the *Brace Type* by using the list.

The variant with steel ties requires a nonlinear calculation. Therefore, load combinations are automatically created for this brace type. But the calculation is still performed according to linear static analysis.

The nonlinear calculation takes into account that a tension member is no longer effective in the system if a compressive force occurs in the member. Then, the model will be calculated without this tie in the following iteration steps.

Strictly speaking, the steel ties are truss girders with the member nonlinearity "failure under compression" as they can transfer also bending moments outside the plane.

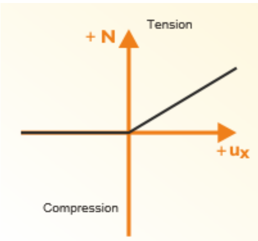


Figure 15.7: Failure under compression

15.2.2 Geometry

In the second program window, we enter the geometry of the bracing.

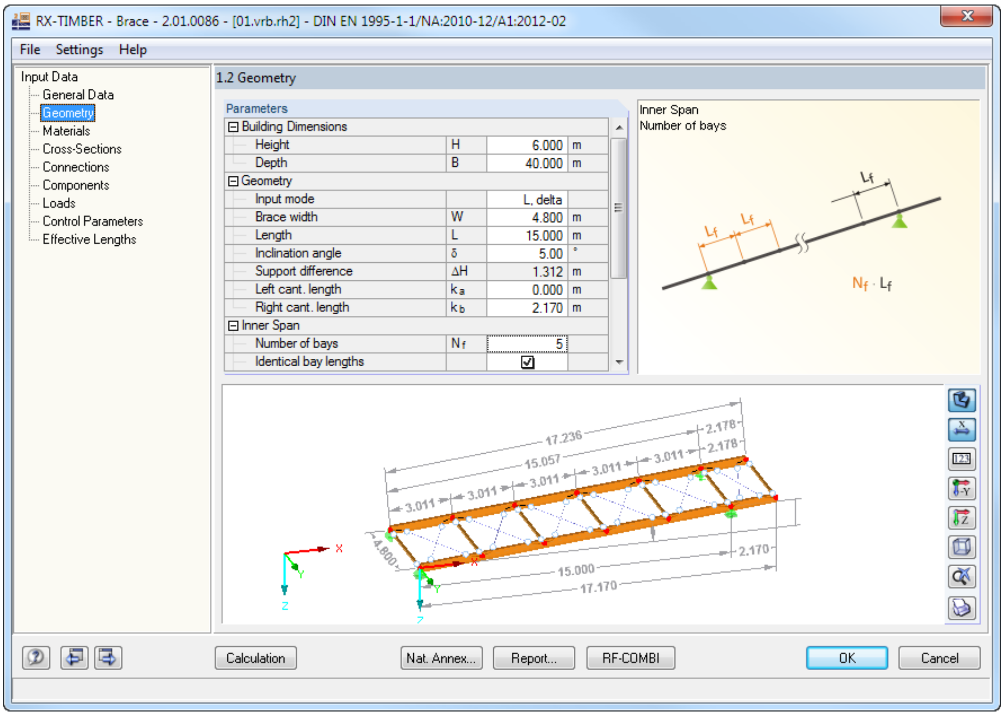


Figure 15.8: Window 1.2 *Geometry*

We set the geometry *Parameters* according to the system description on page 162.

To keep the effort for input as low as possible, we tick the check boxes for **Identical bay lengths** and **Regular diagonals**.

If the boxes are clear, it is possible to combine any descending, crossing or ascending diagonals in freely definable bays:

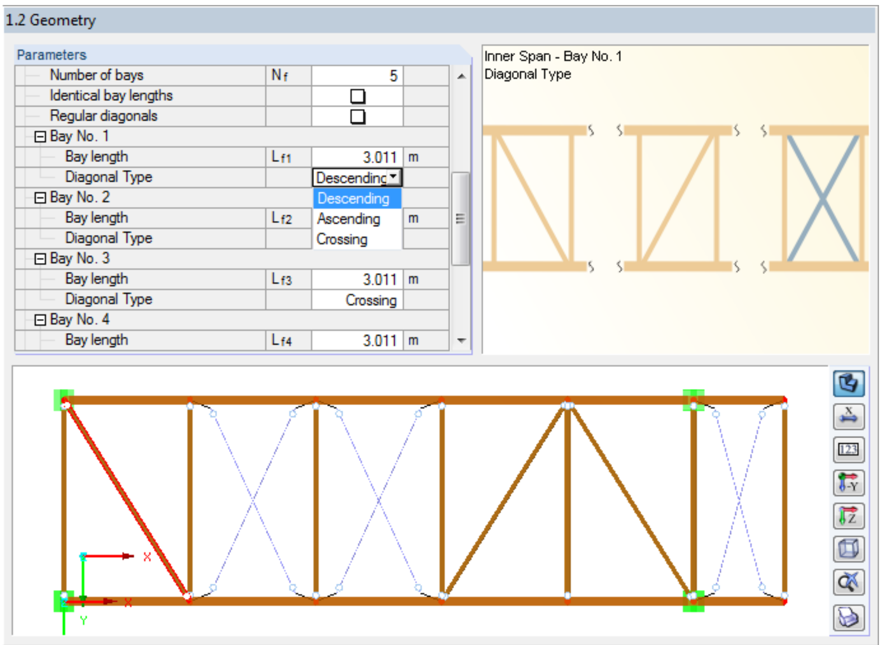


Figure 15.9: Window section *Parameters* with different types of diagonals

15.2.3 Materials

In window 1.3, we define the materials of the cross-sections. As timber and steel are often used together in bracings, *RX-TIMBER Brace* offers us the entire range of steel grades.

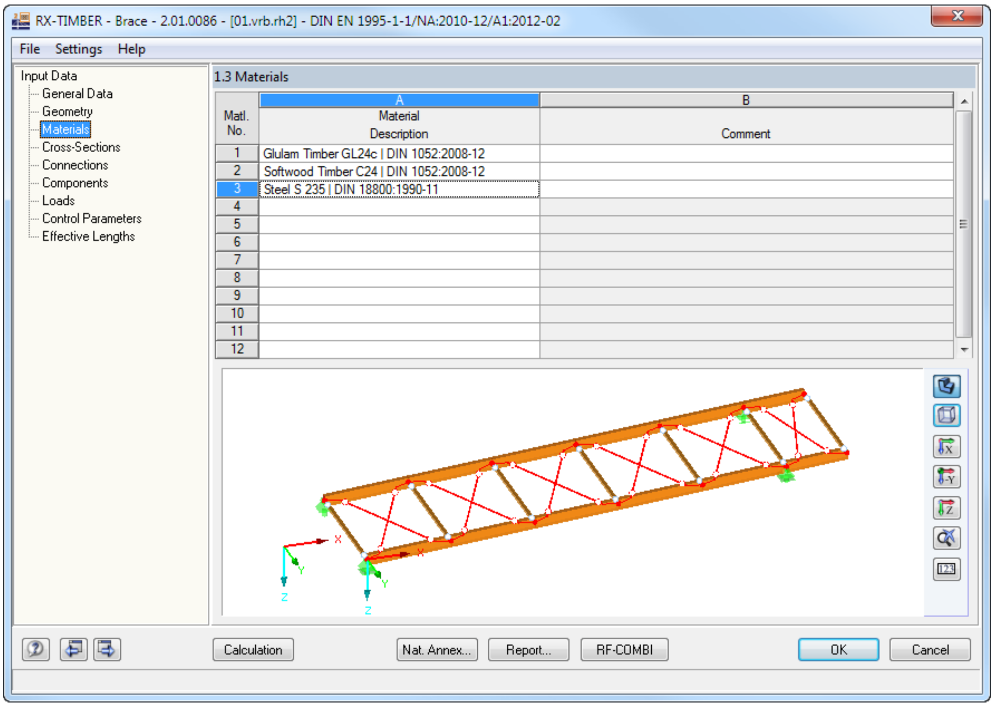


Figure 15.10: Window 1.3 *Materials*

We access the material library by clicking the button [...] appearing at the end of the input row.

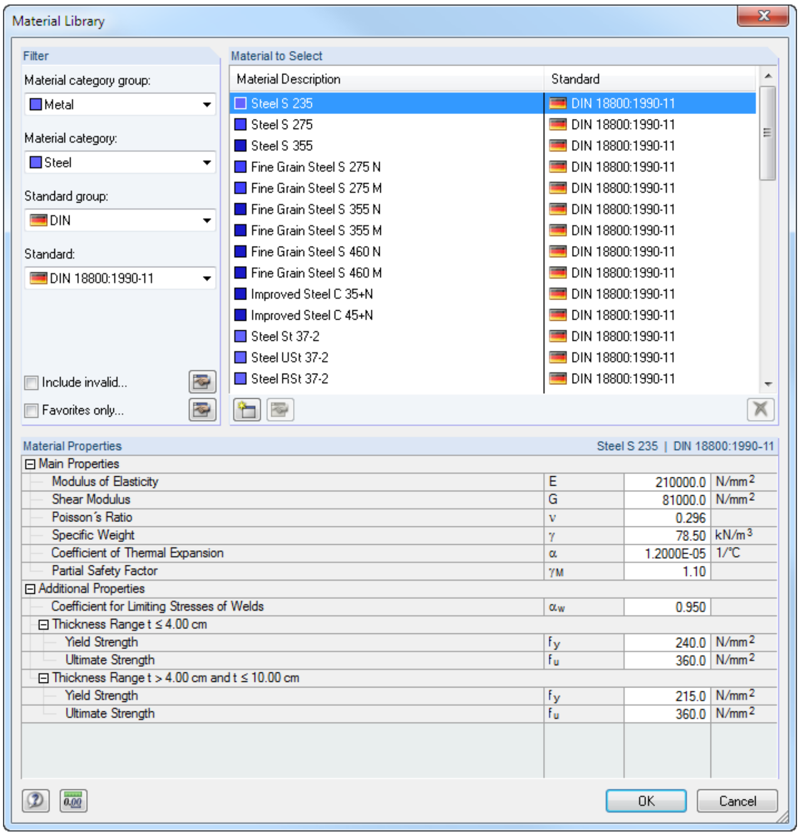


Figure 15.11: Material Library

15.2.4 Cross-sections

Cross-section types can be selected in a library by placing the pointer into the table row and clicking the [...] button. It is also possible to enter cross-sections directly in table column A.

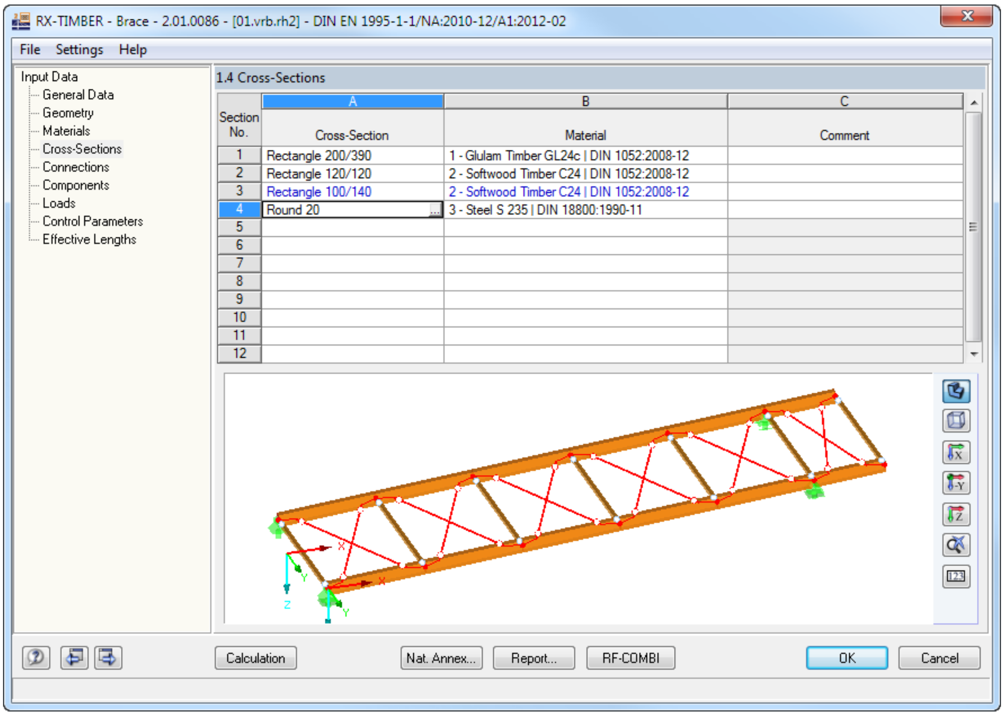


Figure 15.12: Window 1.4 *Cross-Sections*

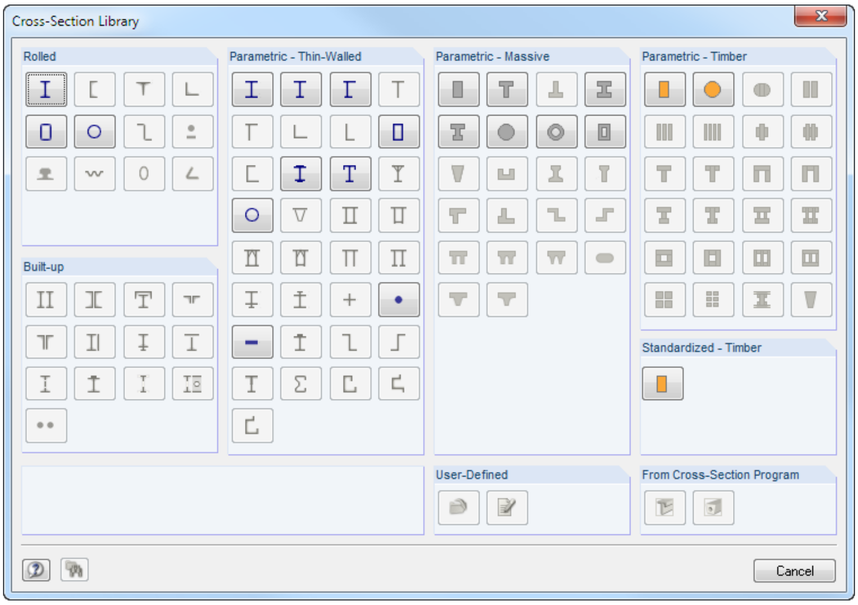


Figure 15.13: Cross-Section Library

Unacceptable or illogical cross-sections are highlighted in red in window 1.4.

	A	B
Section No.	Cross-Section	Material
1	Rectangle 200/390	1 - Glulam Timber GL24c | DIN 1052:2008-12
2	Rectangle 120/120	2 - Softwood Timber C24 | DIN 1052:2008-12
3	Rectangle 100/140	2 - Softwood Timber C24 | DIN 1052:2008-12
4	Round 20	3 - Steel S 235 | DIN 18800:1990-11
5	Rectangle 200/200	3 - Steel S 235 | DIN 18800:1990-11

Figure 15.14: Incorrect cross-sections highlighted in red

15.2.5 Connections

We can define the connections separately for verticals, diagonals and crossing diagonals.

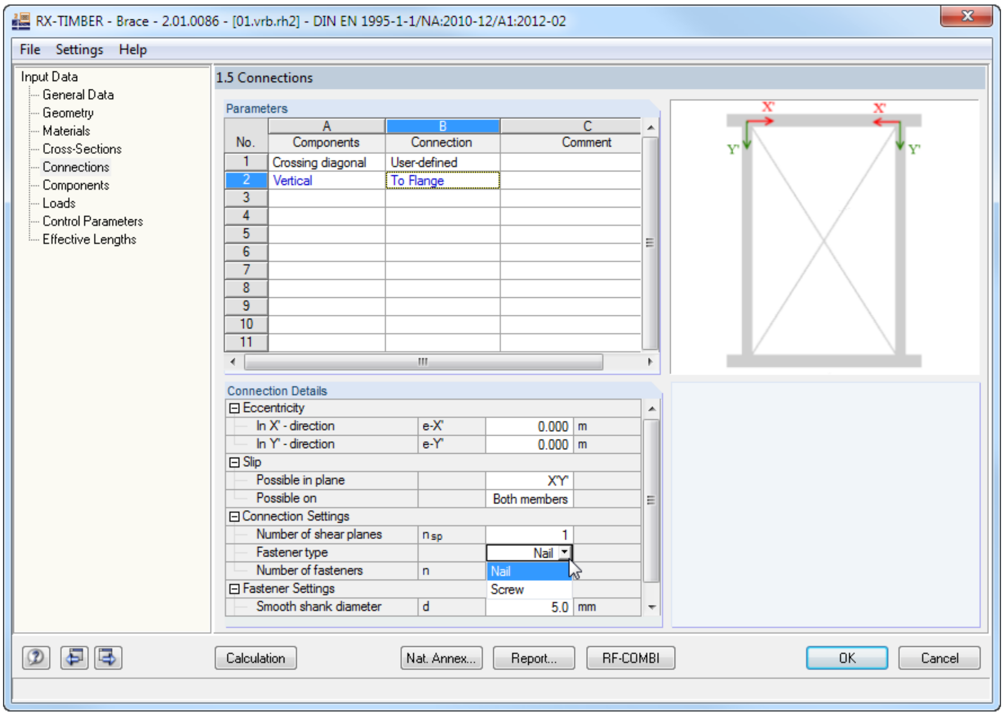


Figure 15.15: Window 1.5 *Connections*

In table column A *Components*, we specify the type of connection.

In table column B, we can decide if the stiffnesses are *User-defined* or determined by the program. If we want *RX-TIMBER Brace* to calculate the stiffnesses, a database with connecting elements is provided. In this case, the eccentricities and connection values must be set manually. The eccentricities always refer to the start node of the bay.

For the **Crossing diagonal** we enter an eccentricity of **0.500 m** in the global direction X', and another of **0.250 m** in the global direction Y'. We don't modify the translational or rotational spring stiffnesses.

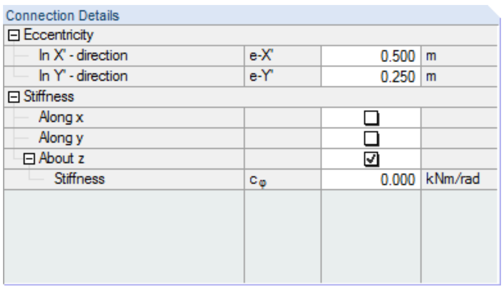


Figure 15.16: User-defined input of *Connection Details*

Excursus:

A connection with through bolts can be modeled in different ways.

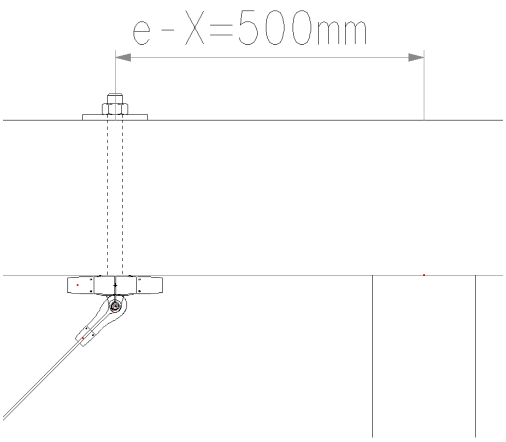


Figure 15.17: Details of through bolt

This connection can be represented by three variants:

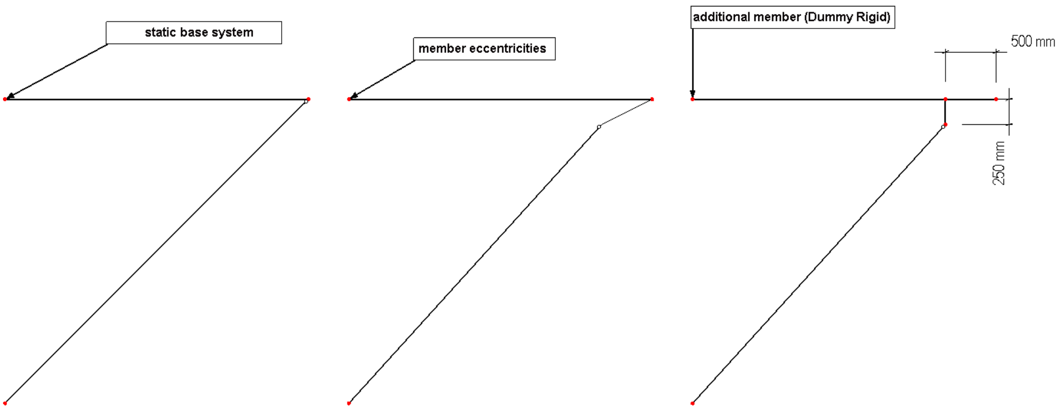


Figure 15.18: Modeling of eccentricities

1. The static base structure of *RX-TIMBER Brace* is used without considering eccentricities (simplest possibility).
2. A member eccentricity is defined for the tension member.
3. A new member with a very high stiffness is inserted for modeling the eccentricity.

Variant 3 provides very realistic results (see figure below) but involves considerable effort. It would be very difficult to do this kind of modeling in RX-TIMBER. Therefore, variant 2 is available in the program to be a feasible and quick modeling option.

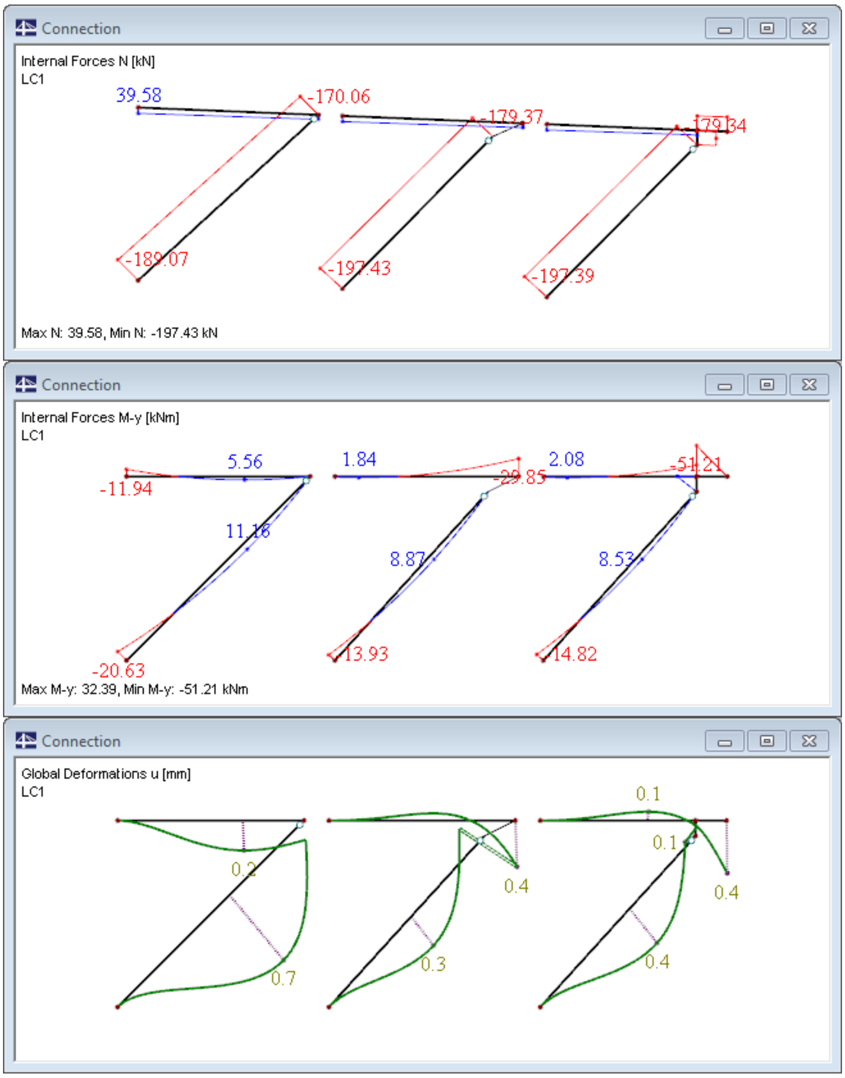


Figure 15.19: Internal forces and deformations for variants 1 to 3 (from the left to the right)

As you can see, variant 3 offers very realistic results. Variant 2 provides similar results, in particular for the deformations that are relevant for the brace. In variant 1, results are clearly different.

15.2.6 Components

Now, we assign the previously defined connection to the components.

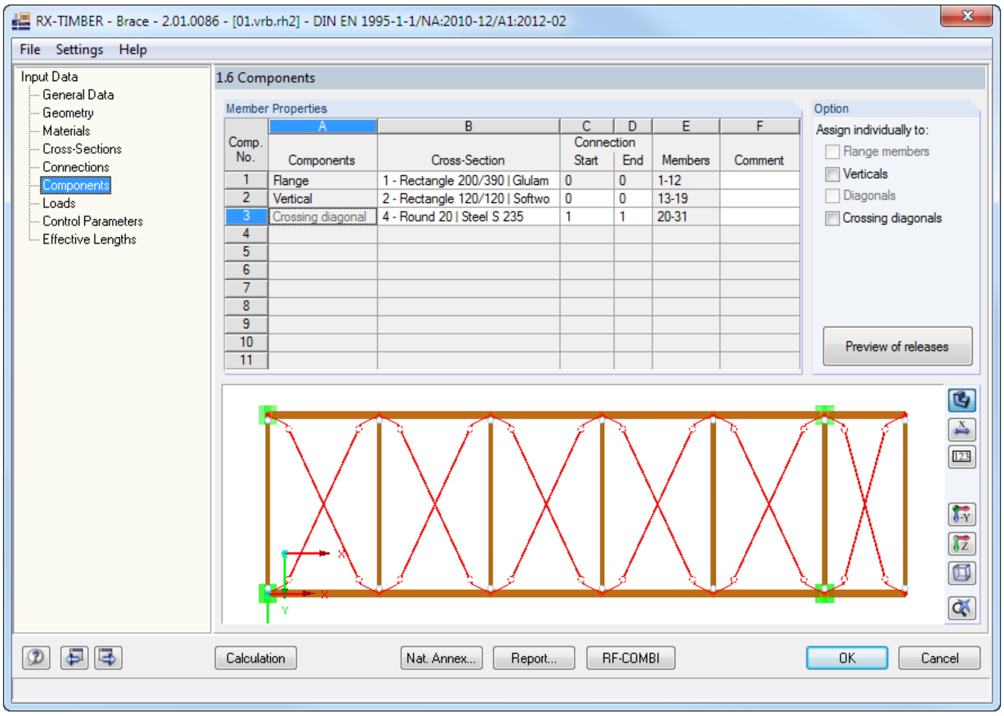


Figure 15.20: Window 1.6 *Components*

For the crossing diagonals we select connection **1** for the *Start* and *End* of the members.

If you want to define the eccentricity of a member individually, you can assign a distinct connection to each member in the window section *Option*. If the check boxes are ticked, it would also be possible to assign each cross-section as you want (see figure below). The cross-sections have to be defined previously in window 1.4 *Cross-Sections*.

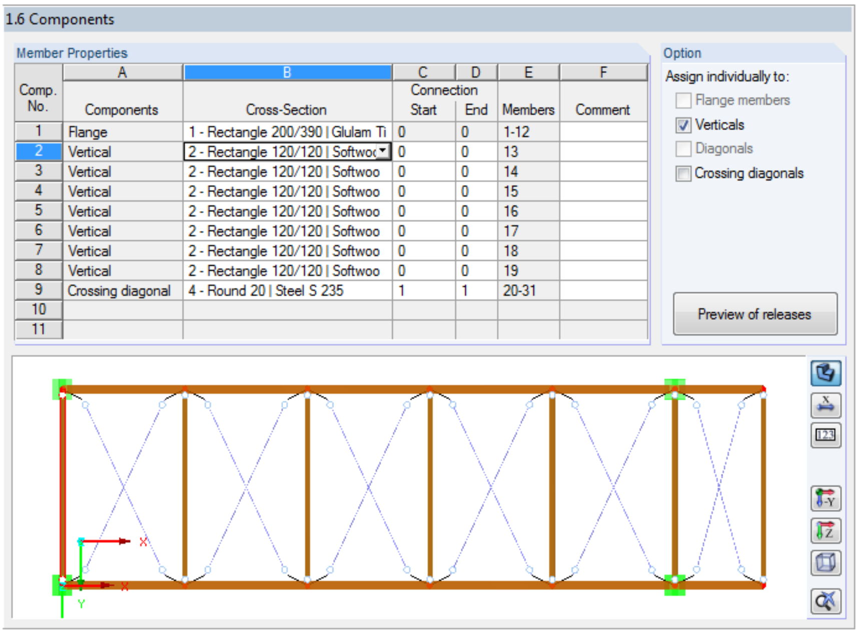
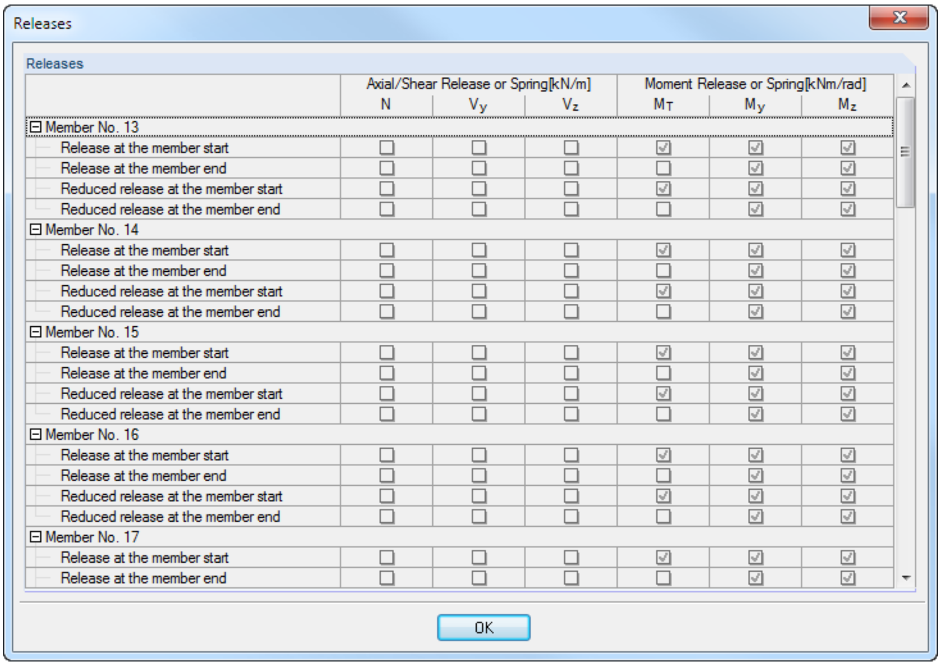


Figure 15.21: Assigning structural components individually

However, the individual modeling involves a considerable modeling effort.

We use the button [Preview of releases] to look at the members with the defined eccentricities and stiffnesses in a table.



Releases	Axial/Shear Release or Spring[kN/m] N	V_y	V_z	Moment Release or Spring[kNm/rad] M_T	M_y	M_z
⊟ Member No. 13						
Release at the member start	☐	☐	☐	☑	☑	☑
Release at the member end	☐	☐	☐	☐	☑	☑
Reduced release at the member start	☐	☐	☐	☑	☑	☑
Reduced release at the member end	☐	☐	☐	☐	☑	☑
⊟ Member No. 14						
Release at the member start	☐	☐	☐	☑	☑	☑
Release at the member end	☐	☐	☐	☐	☑	☑
Reduced release at the member start	☐	☐	☐	☑	☑	☑
Reduced release at the member end	☐	☐	☐	☐	☑	☑
⊟ Member No. 15						
Release at the member start	☐	☐	☐	☑	☑	☑
Release at the member end	☐	☐	☐	☐	☑	☑
Reduced release at the member start	☐	☐	☐	☑	☑	☑
Reduced release at the member end	☐	☐	☐	☐	☑	☑
⊟ Member No. 16						
Release at the member start	☐	☐	☐	☑	☑	☑
Release at the member end	☐	☐	☐	☐	☑	☑
Reduced release at the member start	☐	☐	☐	☑	☑	☑
Reduced release at the member end	☐	☐	☐	☐	☑	☑
⊟ Member No. 17						
Release at the member start	☐	☐	☐	☑	☑	☑
Release at the member end	☐	☐	☐	☐	☑	☑

Figure 15.22: Dialog box *Releases*

Buttons for window graphic

In window 1.6 *Components*, like in all other program windows offering a graphical representation of the brace, we see several buttons to the right of the graphic (see Figure 15.21).

In the rendering display, it is difficult to see whether the release has been assigned to the correct member end. Therefore, a model display reduced to the centerlines is possible in addition to the photo-realistic view. In the wireframe display model we see the members displayed as lines so that releases can be clearly recognized. We can switch between both display options with the [Display] button.

We use the button [Show Dimensions] to display the dimension lines on the bracing.

To turn on and off the numbering of nodes and members, we click the button [Show Numbering].

The girder's view can be changed by using the [View] buttons for the directions -Y and Z. Then, the graphic shows the brace displayed in the selected viewing direction.

The button shown on the left sets the isometric view.

We use the final button to reset the display of the entire model. This function is useful when a maximized partial view was set, which is possible by using the wheel button of the mouse.

15.2.7 Loads

In window 1.7 *Loads*, we have to specify several entries.

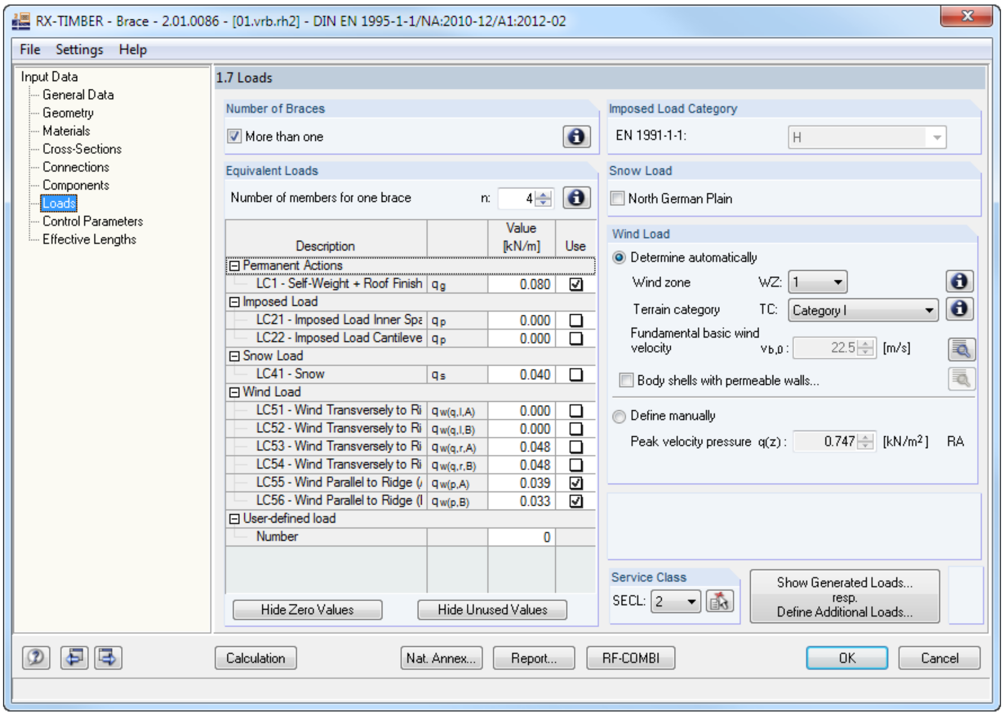


Figure 15.23: Window 1.7 *Loads*

In the window section *Number of Braces*, we define how many bracings are provided for the stiffening of the truss. For the normal case of a quadrilateral hall it would be reasonable to distribute the loads from wind flows among the braces. Therefore, we tick the check box **More than one**.

Due to the hall depth of 40 m defined in window 1.2, there are two braces each with four truss girders that must be stiffened by the bracing. Therefore, we enter the number **4** in the window section *Equivalent Loads*.

We click the [Info] button to open a dialog box illustrating the load application.

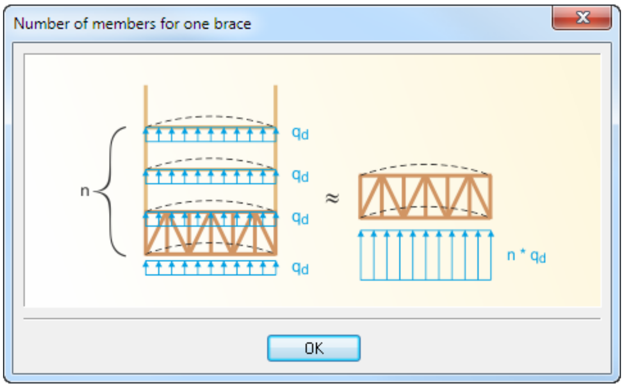


Figure 15.24: Dialog box *Number of members for one brace*

In the window section *Equivalent Loads*, we use the check boxes to decide if a load case is used for the automatic combination. The *Use* options are especially helpful for user-defined loads because this setting won't be overwritten by RF-COMBI.

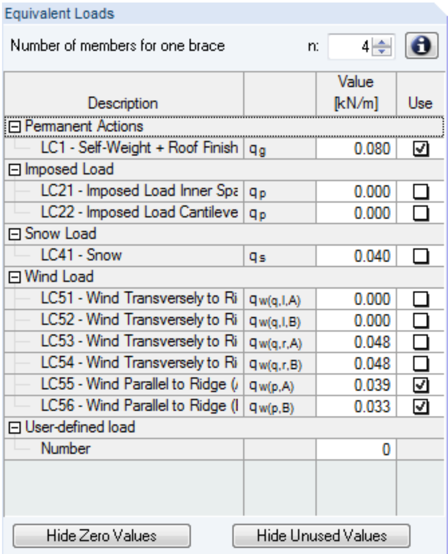


Figure 15.25: Window section *Equivalent Loads*

Here, it is recommended to *Use* load case 1 and the wind loads blowing on the gable. In most cases, they result in the governing deformations of the brace.

We can hide the used loads by clicking the buttons [Hide Zero Values] and [Hide Unused Values].

In window 1.7, we continue with setting the *Imposed Load Category* affecting the automatic superpositioning. When ticking the option *North German Plain* for the snow load below, an accidental combination is automatically created in RF-COMBI. Then, also the stiffnesses of the releases and of the materials with the characteristic stiffnesses are recalculated.

Like in other programs of the RX-TIMBER family the *Wind Load* is generated automatically, but it is also possible to define it manually.

We click the [Info] button to open the wind zone map of Germany. The relevant zone can be taken over by double click.

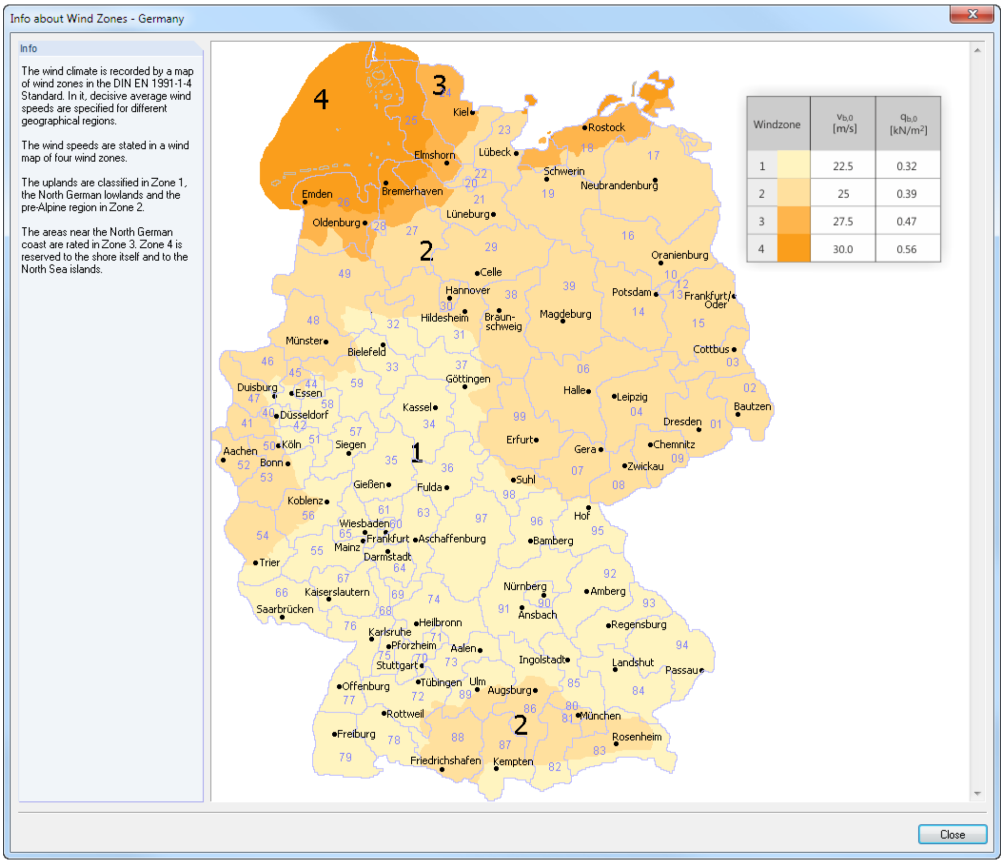


Figure 15.26: Wind zone map for Germany

The wind load is automatically interpolated for the selected roof inclination of 5°. Thus, in accordance with DIN EN 1991-1-4, we have the following values for the pressure and suction areas:

Gust velocity pressure for wind zone 1 q(z)	=	0.5 kN/m²
External pressure coefficient for area D $c_{pe,10}$	=	0.69 (interpolated for roof inclination of 5°)
External pressure coefficient for area D $c_{pe,10}$	=	0.27 (interpolated for roof inclination of 5°)
Wind pressure $c_{pe} \cdot q(z) = w_{e,D}$	=	0.34 kN/m²
Wind suction $c_{pe} \cdot q(z) = w_{e,E}$	=	0.14 kN/m²

Based on these factors and according to the building height defined in window 1.2 *Geometry*, the program calculates the wind load on the brace. To simplify the calculation, the program applies the half of the height of the eaves as well as the half of the height on the ridge to the load application area.

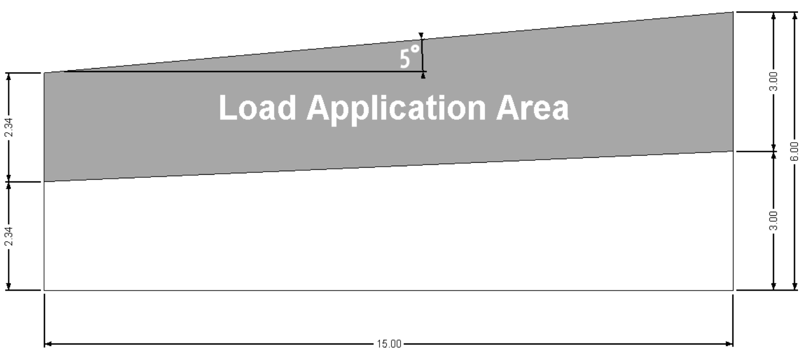


Figure 15.27: Scheme for load application area

Entering wind loads manually is also possible for more accurate load applications like of a concrete ring beam for which only half of the load application area would have to be considered.

In our example, the settings result in the following load ordinates for the wind pressure zone (LC 155) with:

$$p_1 = w_{e,D} \cdot \frac{H}{2} \cdot \cos\alpha = 0.34\ kN/m^2 \cdot (3 \cdot \cos 5°) = 1.02\ kN/m$$

The discrepancy of 0.03 kN/m results from the roundings in the manual calculations.

The load ordinate of the eaves for an eaves height of 5.33 m is:

$$p_2 = w_{e,E} \cdot \frac{H}{2} \cdot \cos\alpha = 0.34\ kN/m^2 \cdot (2.65 \cdot \cos 5°) = 0.89\ kN/m$$

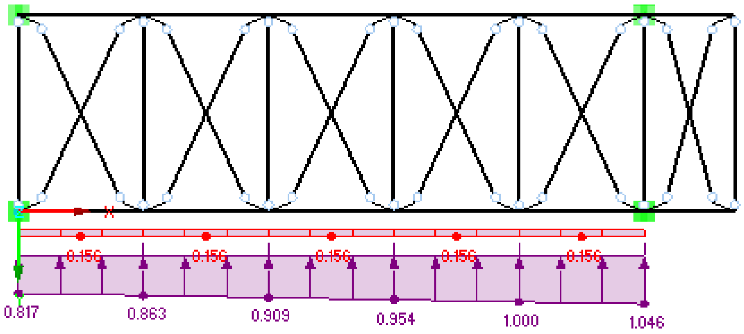


Figure 15.28: Load of wind pressure for brace 1

The load ordinates for the wind suction zone (LC 56) are the following:

$$p_1 = w_{e,D} \cdot \frac{H}{2} \cdot \cos\alpha = 0.14\ kN/m^2 \cdot (3 \cdot \cos 5°) = 0.42\ kN/m$$

The load ordinate of the eaves with the eaves height of 5.33 m is the following:

$$p_2 = w_{e,E} \cdot \frac{H}{2} \cdot \cos\alpha = 0.14\ kN/m^2 \cdot (2.65 \cdot \cos 5°) = 0.37\ kN/m$$

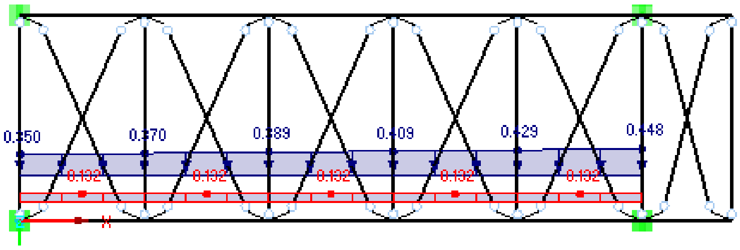


Figure 15.29: Load of wind suction for brace 2

The required equivalent load can be calculated according to eq. (9.37) of DIN EN 1995-1-1:

$$q_d = k_l \cdot \frac{n \cdot N_d}{30 \cdot l}$$

Alternatively, we can insert the values from window 2.4 *Support Forces* of the *RX-TIMBER Glued Laminated Beam* program into window 1.7 by using the clipboard (Ctrl + C and Ctrl + V).

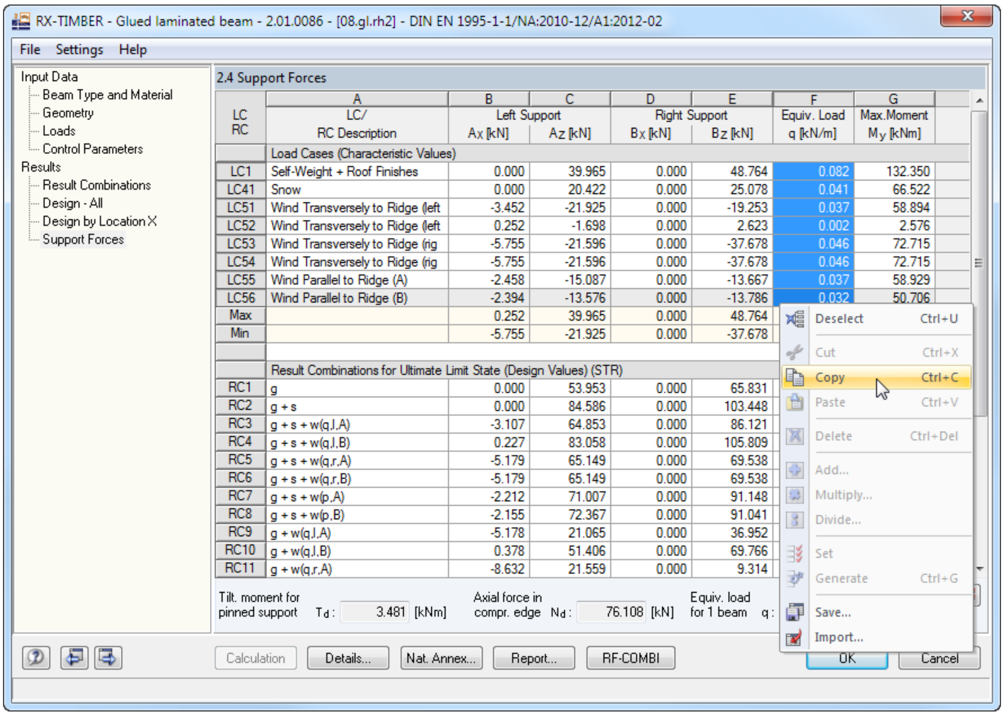


Figure 15.30: Window 2.4 Copying *Support Forces* in program RX-TIMBER Glued Laminated Beam

We use the button [Show Generated Loads] shown on the left to open the dialog box *Load Cases* where we can check the generated loads.

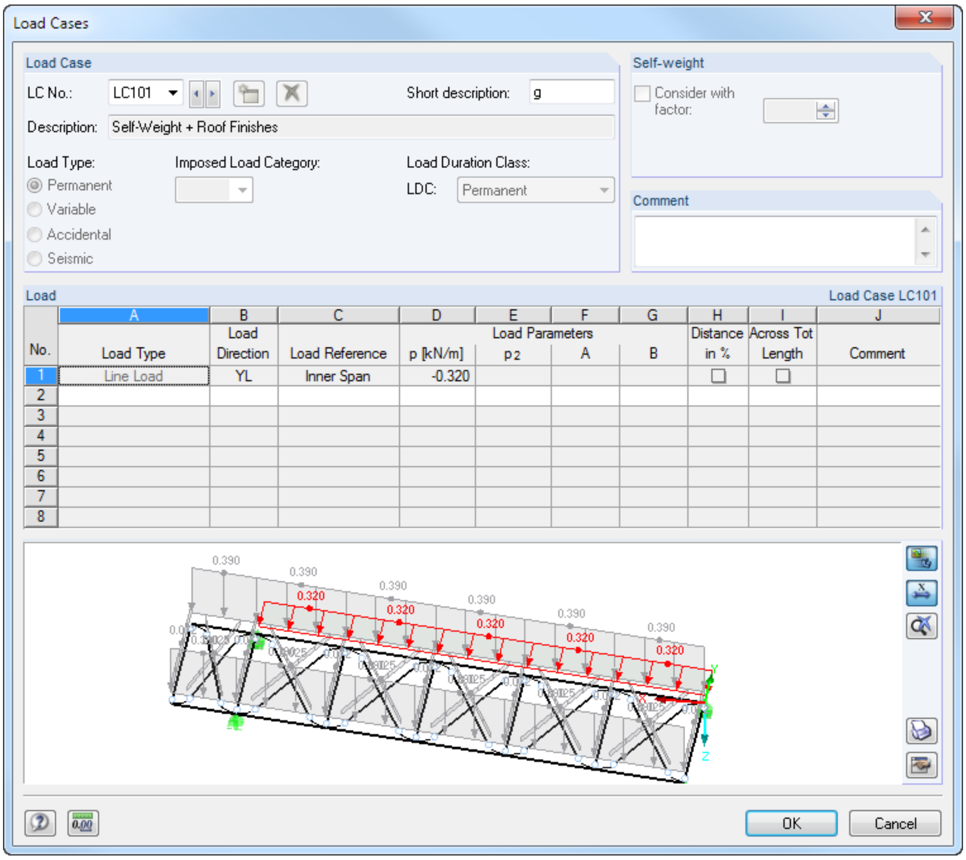


Figure 15.31: Dialog box *Load Cases*

Here, we can freely define the loads like in all other programs of the RX-TIMBER family.

15.2.8 Control Parameters

In window 1.8 *Control Parameters*, we set only the design of the **Ultimate limit state**. Usually, this design is sufficient because the program also analyzes the limitation of the deflection to u < l/500 according to clause 9.2.5 of DIN EN 1995.

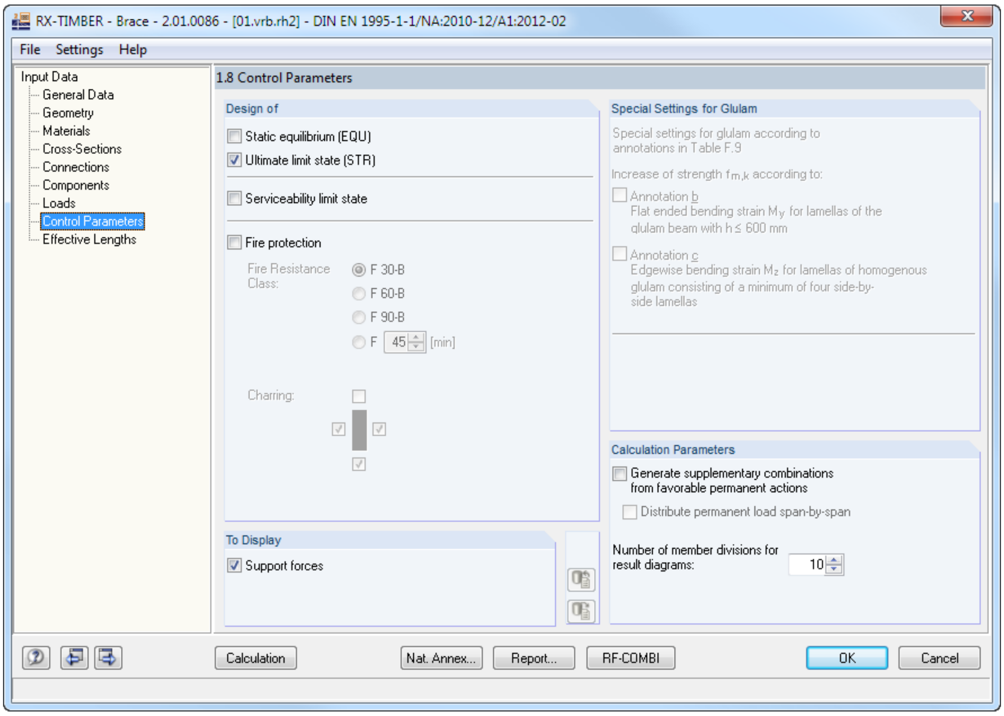


Figure 15.32: Window 1.8 *Control Parameters*

We click the button [Nat. Annex] to open the dialog box *National Annex Settings* (see Figure 15.5, page 165) where we can define the partial safety and modification factors as well as the limit deformations.

When determining the stiffnesses RX-TIMBER performs a double calculation of the internal forces and deformations:

When designing the ultimate limit state we have to calculate, in accordance with clause 2.2 of EN 1995-1-1 or clause 8.2 of DIN 1052, with the mean stiffness that is to be divided by the partial safety factor. In the serviceability limit state design, however, we have to apply the characteristic mean stiffnesses.

The combinatorics for the serviceability limit state design is globally solved in the program *RX-TIMBER Brace* with the rare characteristic combination according to EN 1991, respectively DIN 1055. The equation (22) according to DIN 1055-100 is:

$$E_{d,\,rare} = E\left\{ \sum_{j\,\geq 1} G_{k,\,1} \oplus P_k \oplus Q_{k,\,1} \oplus \sum_{i\,>\,1} \psi_{0,\,i} \cdot Q_{k,\,i} \right\}$$

Equation 15.1: Combination expression according to DIN 1055-100 for rare (characteristic) combination

Furthermore, note the following for the slip moduli K_{ser} of the release stiffnesses: They must be divided as well by the partial safety factor 1.3 in the ultimate limit state design. For the serviceability limit state design the program calculates with the characteristic displacement of K_{ser}.

RX-TIMBER calculates the data in a double calculation loop: In the first calculation run, the internal forces and deformations are determined in the ultimate limit state design on the design level. In the second calculation run, the program overwrites this stiffness of the materials and end releases and calculates the deformations with the characteristic stiffnesses.

15.2.9 Effective Lengths

To be on the safe side, we select the factor β with **1.0** in our example. The values are preset by the program so that no data modification is required in this window.

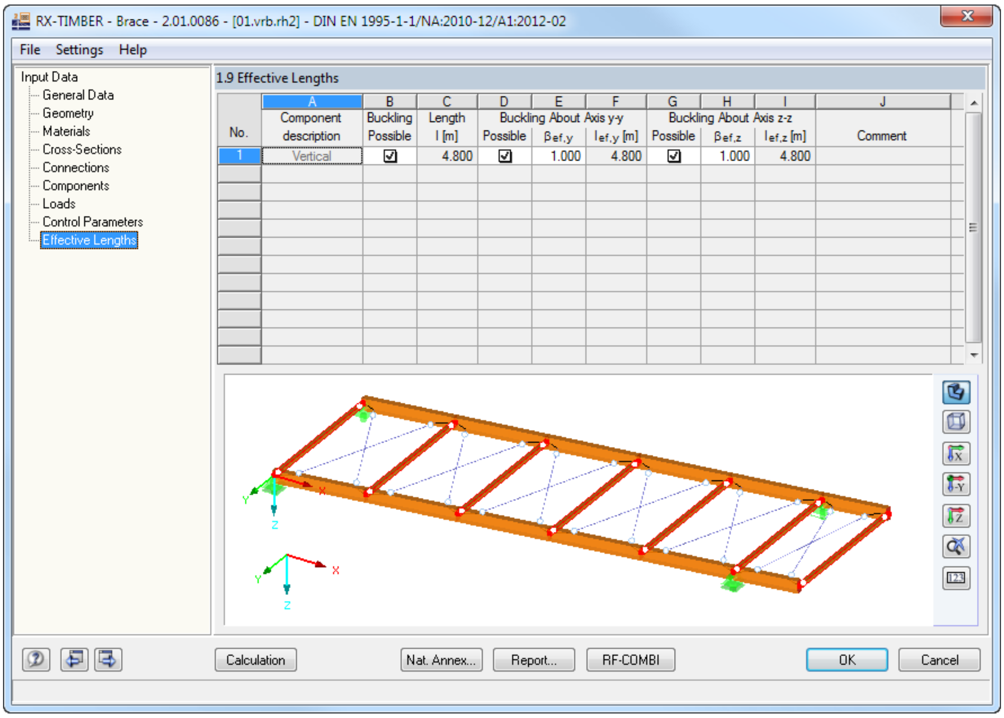
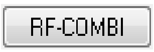


Figure 15.33 Window 1.9 *Effective Lengths*

15.2.10 RF-COMBI



We use the button [RF-COMBI] shown on the left to access the add-on module RF-COMBI integrated in RX-TIMBER. In RF-COMBI we can check the combinations generated in the program's background.

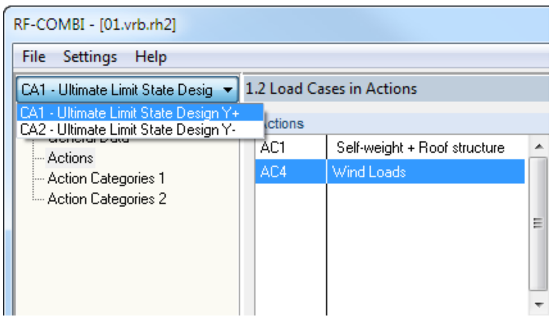


Figure 15.34: RF-COMBI window 1.2 *Load Cases in Actions*

In the list with the created cases (top left), two cases are available for the ultimate limit state design: As already mentioned in chapter 15.1, members may fail in particular load arrangements. To cover all eventualities, all loads are applied again in the reverse direction and combined automatically.

For more detailed information about the features of RF-COMBI, see the manual available for download at www.dlubal.com. We click [OK] to return to *RX-TIMBER Brace*.

15.3 Results

15.3.1 Load/Result Combinations

We use the [Calculation] button to calculate the entered data. Window 2.1 shows the ultimate limit state designs for all combinations together with the respective design ratios. As already mentioned, RX-TIMBER automatically creates load combinations in case of a material nonlinearity (which means failing tension members). Then, window 2.1 is renamed in *Load Combinations*.

Load combination *CO*	A load combination is used to superimpose load cases, that means all **loads** of the load cases in question are summarized.
Result combination *RC*	A result combination sums up the **results** of the contained load cases. Therefore, always two result values per location are available: maximum and minimum. It is possible to determine the extreme internal forces and deformations from different load cases, load or result combinations by an *Or* combination.

Table 15.1: Difference between load and result combination

When we perform a linear calculation, it does not matter which type of superposition is selected. But for a nonlinear calculation it is only the <u>load</u> combination that provides reasonable results!

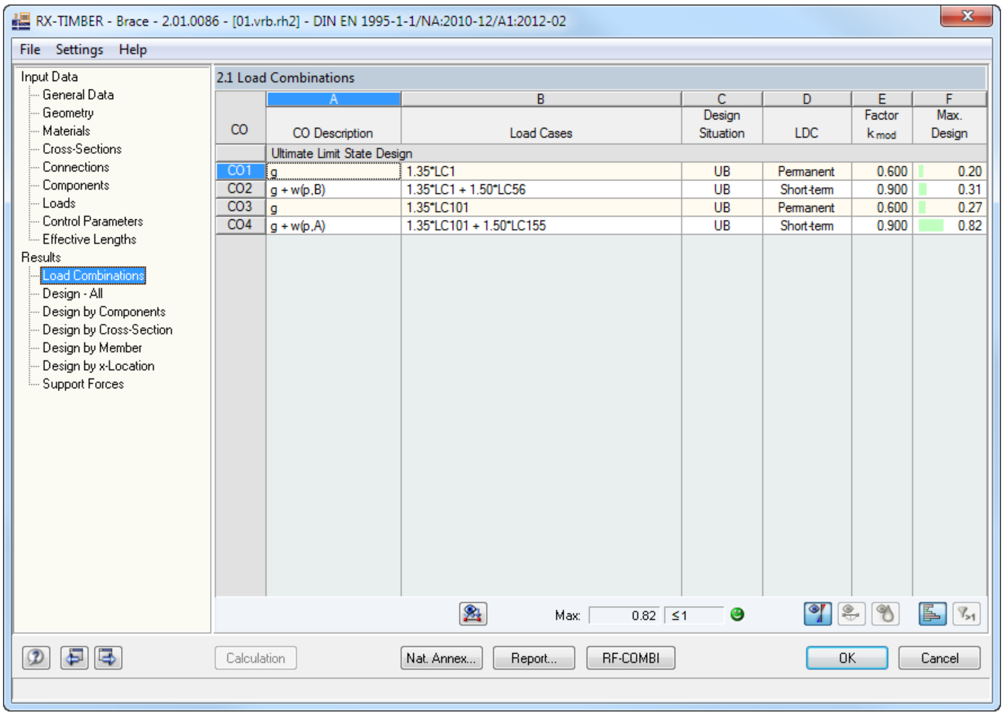


Figure 15.35: Window 2.1 *Load Combinations*

15.3.2 Design - All

Window 2.2 *Design - All* shows us an overview about the governing designs for each design situation. In this way, we can see overloaded areas immediately.

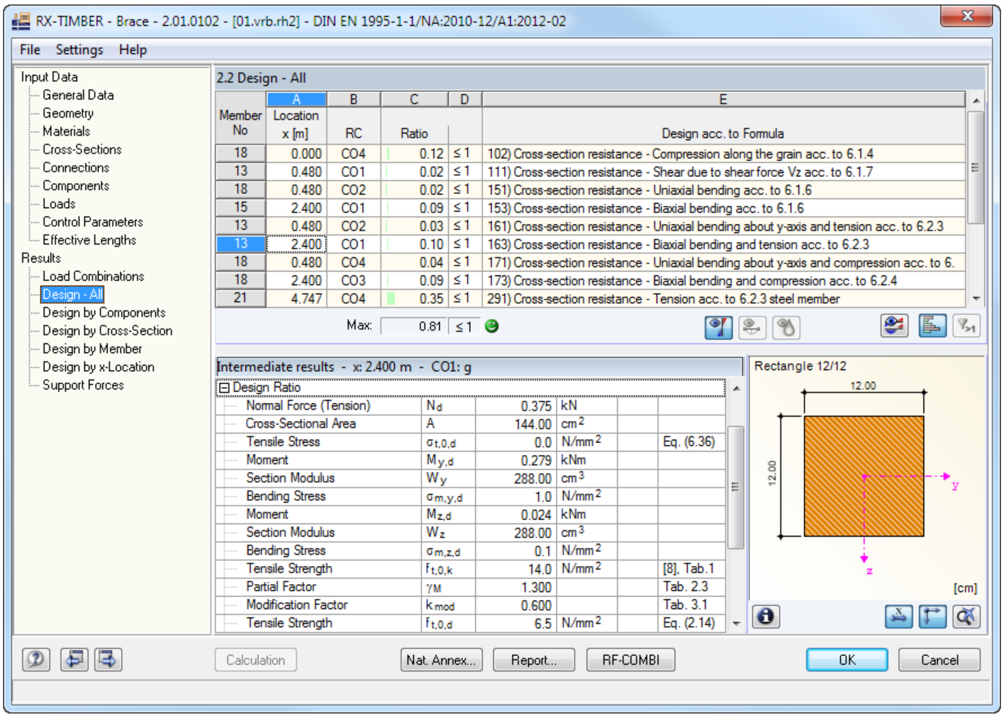


Figure 15.36: Window 2.2 *Design - All*

We have different possibilities for filters and result diagrams to evaluate the results (see chapter 7).

Please note the following for the *Result Diagrams on Member*: It is not possible to represent all results in the result diagram because *RX-TIMBER Brace* uses lots of members. Therefore, the result diagrams show only the results of the member that is selected in the current program window. If we place the pointer in the table row of member 13 as shown in the figure above and open the result diagram, the diagram window only shows us the internal forces and designs for member 13.

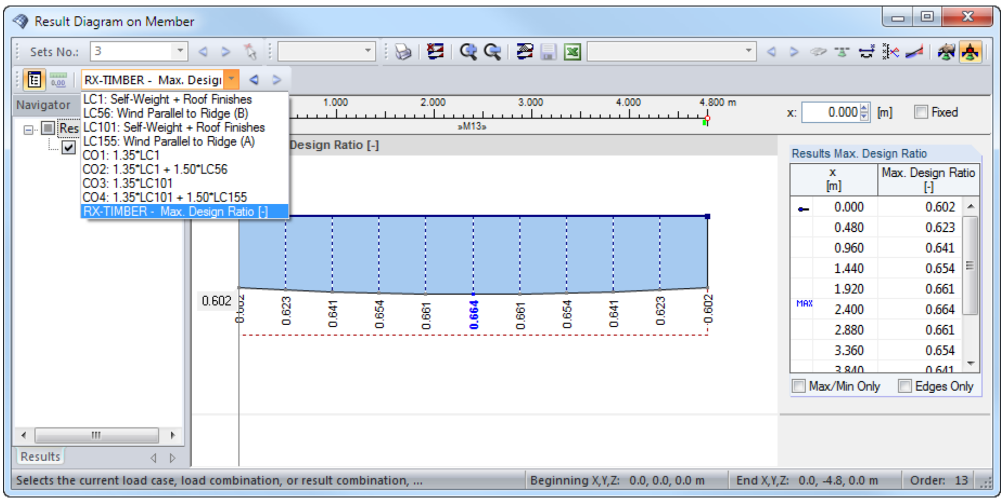


Figure 15.37: Result diagram on member 13

15.3.3 Design by Components

The results window shows the governing designs for the structural components *Front flange*, *Rear flange*, *Verticals* and *Diagonals*.

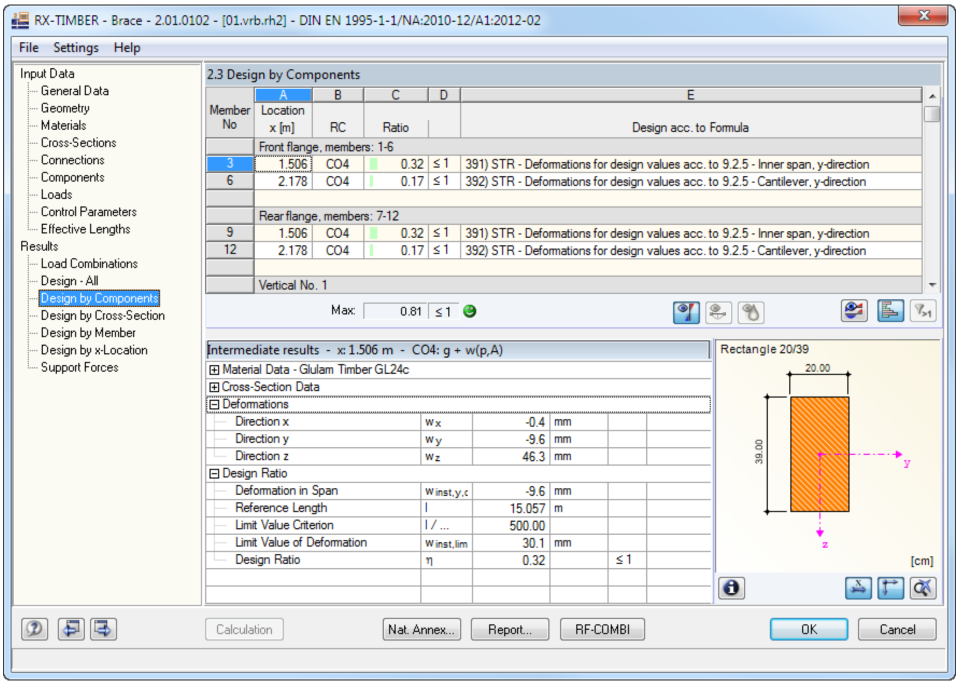


Figure 15.38: Window 2.3 *Design by Components*

15.3.4 Design by Cross-Section

Window 2.4 lists the designs sorted by cross-sections.

If no individual settings were specified in window 1.6 *Components*, this results window represents the clearest and most helpful table where the results in our example are listed separately for chord, verticals and diagonals.

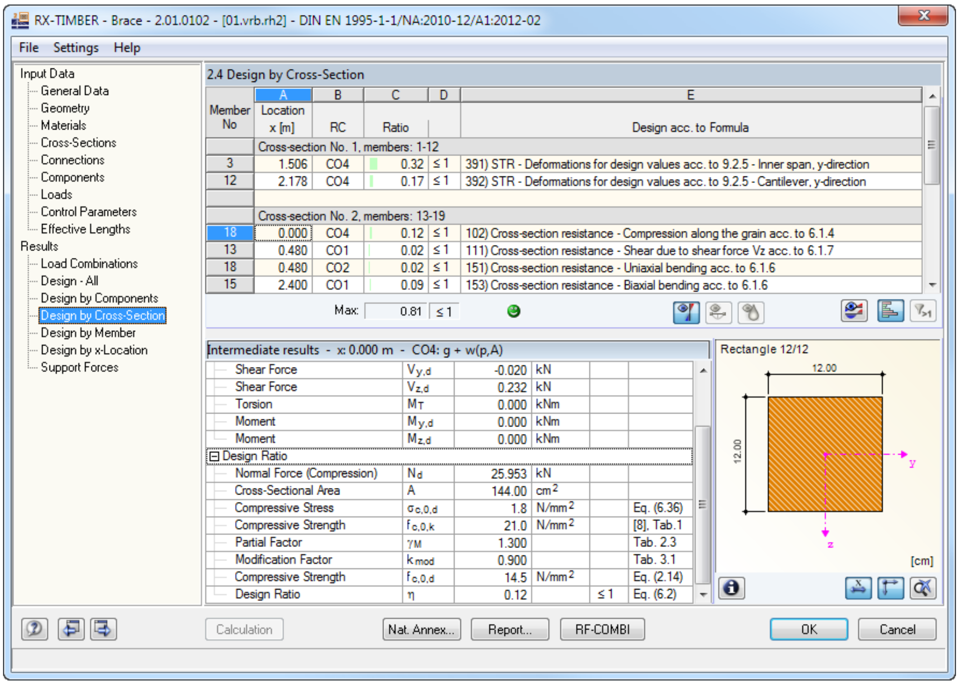


Figure 15.39: Window 2.4 *Design by Cross-Section*

15.3.5 Design by x-Location

The number of x-locations set in window 1.8 *Control Parameters* has an effect on the amount of data displayed in window 2.6.

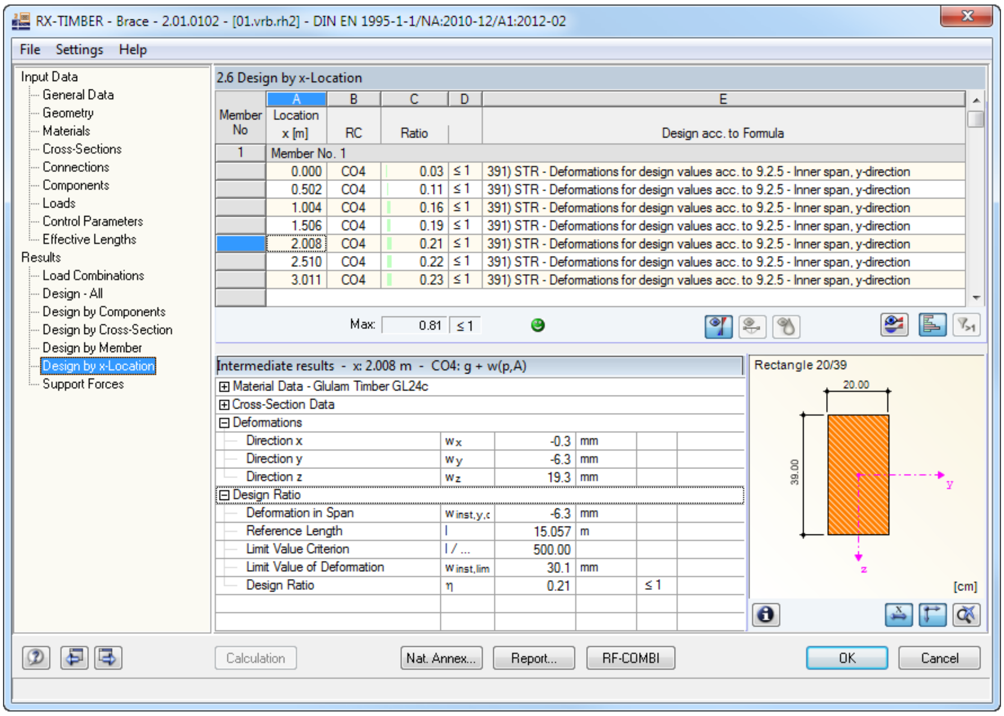


Figure 15.40: Window 2.6 *Design by x-Location*

The stress and stability analyses are already presented in previous examples. That's why we want to have a closer look at the most interesting bracing designs such as the analysis of deflection in the ultimate limit state design.

Analysis of deformation according to DIN 1052, paragraph 8.4.3(9)

If we don't perform a design that is more accurate, the calculated deflection from q_d and other external actions must not exceed l/500.

Here, the **stiffnesses** are applied on safety level by the program.

Stiffness properties	Properties for ultimate limit state	Mean values
Modulus of elasticity	$E = \frac{E_{mean}}{\gamma_M}$	E_{mean}
Shear modulus	$G = \frac{G_{mean}}{\gamma_M}$	G_{mean}
Slip modulus	$K_{u,mean} = \frac{K_{u,mean}}{\gamma_M}$	$K_{u,mean} = \frac{2}{3} \cdot K_{ser}$

The mean values of the stiffness can be found in annex F of DIN 1052, respectively EN 388 or EN 1194.

In our example, we get the following stiffness properties for load bearing capacity, serviceability and fire resistance of the respective components as shown in the table below.
The stiffness properties of the fire resistance are to put on the same level with the ones for an accidental combination.

Component	Material	Load bearing capacity	Serviceability	Fire resistance
Chord	GL24c	$E = \frac{1160}{1.3} = 892\ kN/cm^2$	$E_{mean} = 1160\ kN/cm^2$	E_{mean}
		$G = \frac{590}{1.3} = 454\ kN/cm^2$	$G_{mean} = 590\ kN/cm^2$	G_{mean}
Verticals	C24	$E = \frac{1100}{1.3} = 846\ kN/cm^2$	$E_{mean} = 1100\ kN/cm^2$	E_{mean}
		$G = \frac{690}{1.3} = 531\ kN/cm^2$	$G_{mean} = 690\ kN/cm^2$	G_{mean}
Diagonals	S235	$E = \frac{21000}{1.1} = 19091\ kN/cm^2$	$E = 21000\ kN/cm^2$	-
		$G = \frac{8100}{1.1} = 7364\ kN/cm^2$	$G = 8100\ kN/cm^2$	-
Connection	-	$K_{u,mean} = \frac{K_{u,mean}}{\gamma_M}$	K_{ser}	$K_{u,mean}$

The governing load combination CO4 with the actions 1.35 · LC101 + 1.5 · LC155 features the following loads, internal forces and deformations.

Loads

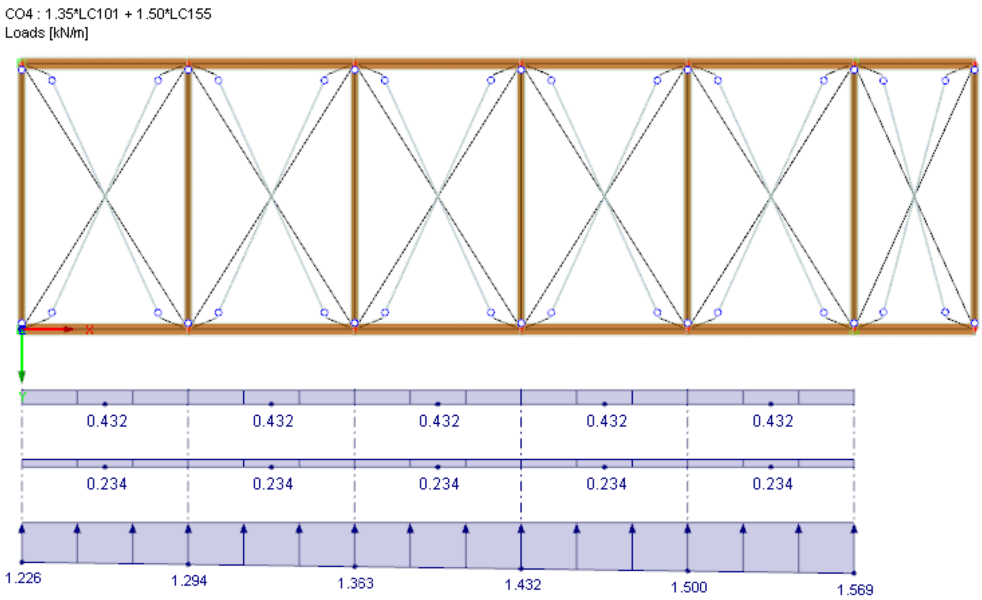


Figure 15.41: Loads

Deformation

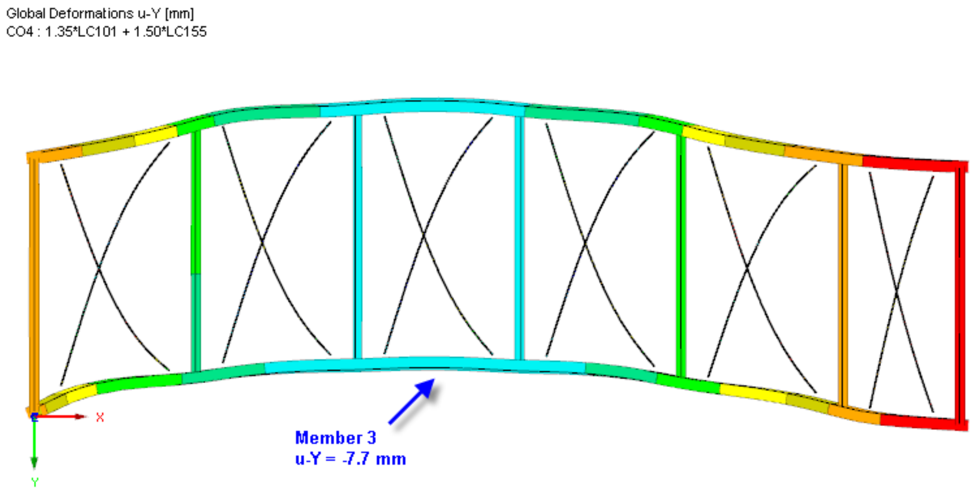


Figure 15.42: Deformations u_Y

Internal forces

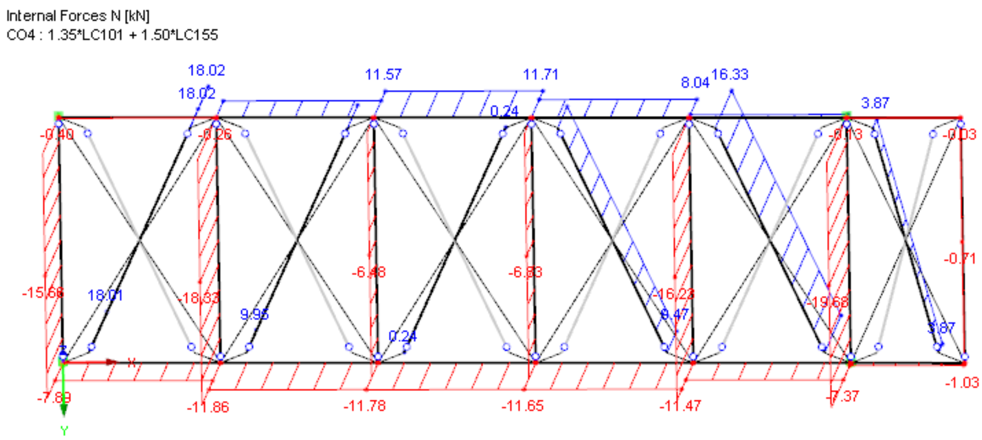


Figure 15.43: Axial forces N

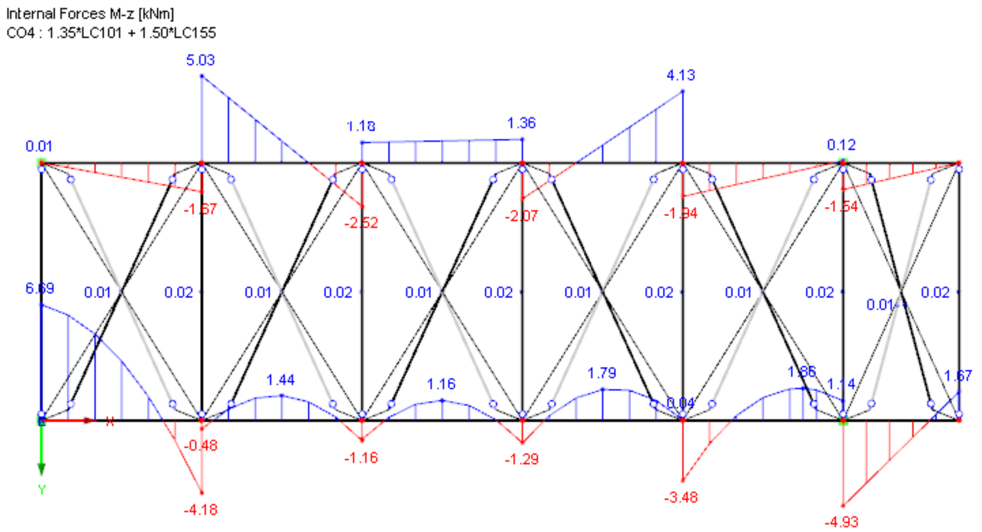


Figure 15.44: Moments M_Z

Deformation analysis

The deformation in the lower chord member 3 in direction Y is calculated by the program with 7.7 mm. The deformation of the tension diagonals is larger, of course, but irrelevant for the total deformation. Thus, we obtain the following design ratio:

Deformation = w_{inst} = -7.7 mm

Limit value = $w_{inst,limit}$ = l/500 = 15.06 m / 500 = 30.1 mm

Design ratio

w_{inst} / $w_{inst,limit}$ = 7.7 mm / 30.1 mm = 0.26 => 26 % design ratio

The girder is sufficiently stiff for the deformation analysis.

Design of tension members

As we can see in Figure 15.43, several diagonals are failing under the load, which means that they do not provide any internal forces.

The maximum tensile force occurs on the diagonal member 21 (left bay) with a force of 18.02 kN.

Tensile stress N / A = 18.02 kN / 3.14 cm² = 5.74 kN/cm²

Design ratio

$$\frac{\sigma_{N,t}}{f_{y,d}} = \frac{5.74\ \text{kN/cm}^2}{\frac{24\ \text{kN/cm}^2}{\gamma_M}} = \frac{5.74\ \text{kN/cm}^2}{\frac{24\ \text{kN/cm}^2}{1.1}} = 0.26 < 1$$

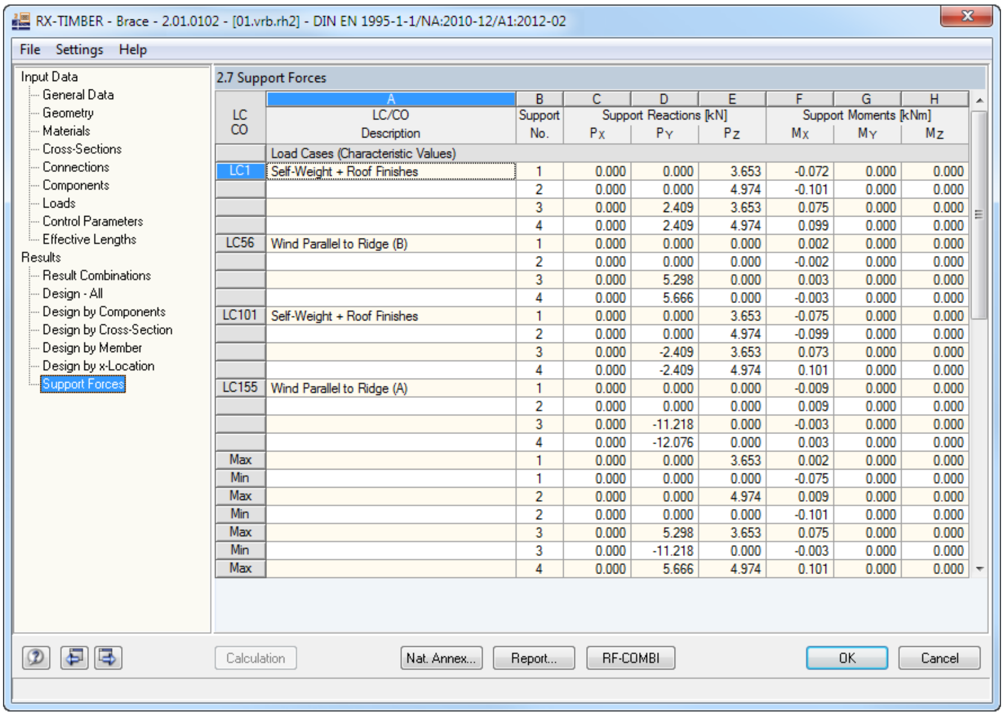
The member is used with a capacity of 26 %.

We do not analyze the chord because it has already been designed in the program *RX-TIMBER Glued Laminated Timber*. However, we need to consider the axial forces of the brace in a recalculation with *RX-TIMBER Glued Laminated Timber*.

15.3.6 Support Forces

Window 2.7 *Support Forces* is the final results window. The support forces are shown for all four supports of the brace.

In addition, a max/min evaluation of forces is displayed. In this way, the forces to be transferred can be quickly determined.



LC CO	A LC/CO Description	B Support No.	C Support Reactions [kN] Px	D Py	E Pz	F Support Moments [kNm] Mx	G My	H Mz
	Load Cases (Characteristic Values)							
LC1	Self-Weight + Roof Finishes	1	0.000	0.000	3.653	-0.072	0.000	0.000
		2	0.000	0.000	4.974	-0.101	0.000	0.000
		3	0.000	2.409	3.653	0.075	0.000	0.000
		4	0.000	2.409	4.974	0.099	0.000	0.000
LC56	Wind Parallel to Ridge (B)	1	0.000	0.000	0.000	0.002	0.000	0.000
		2	0.000	0.000	0.000	-0.002	0.000	0.000
		3	0.000	5.298	0.000	0.003	0.000	0.000
		4	0.000	5.666	0.000	-0.003	0.000	0.000
LC101	Self-Weight + Roof Finishes	1	0.000	0.000	3.653	-0.075	0.000	0.000
		2	0.000	0.000	4.974	-0.099	0.000	0.000
		3	0.000	-2.409	3.653	0.073	0.000	0.000
		4	0.000	-2.409	4.974	0.101	0.000	0.000
LC155	Wind Parallel to Ridge (A)	1	0.000	0.000	0.000	-0.009	0.000	0.000
		2	0.000	0.000	0.000	0.009	0.000	0.000
		3	0.000	-11.218	0.000	-0.003	0.000	0.000
		4	0.000	-12.076	0.000	0.003	0.000	0.000
Max		1	0.000	0.000	3.653	0.002	0.000	0.000
Min		1	0.000	0.000	0.000	-0.075	0.000	0.000
Max		2	0.000	0.000	4.974	0.009	0.000	0.000
Min		2	0.000	0.000	0.000	-0.101	0.000	0.000
Max		3	0.000	5.298	3.653	0.075	0.000	0.000
Min		3	0.000	-11.218	0.000	-0.003	0.000	0.000
Max		4	0.000	5.666	4.974	0.101	0.000	0.000

Figure 15.45: Window 2.7 *Support Forces*

15.3.7 Printout Report

The printout report documentation is similar to the one described in the examples presented in chapter 10 and 11.

Dlubal

A: Literature

[1] DIN 1052:2008-12 Entwurf, Berechnung und Bemessung von Holzbauwerken

[2] DIN 1055-100 Grundlagen der Tragwerksplanung, Sicherheitskonzept und Bemessungsregeln

[3] DIN 1055-3 Einwirkungen auf Tragwerke – Eigen- und Nutzlasten für Hochbauten

[4] DIN 1055-4 Einwirkungen auf Tragwerke – Windlasten

[5] DIN 1055-5 Einwirkungen auf Tragwerke – Schnee- und Eislasten

[6] DIN 4102-22 Brandverhalten von Baustoffen und Bauteilen – Teil 22: Anwendungsnorm zu DIN 4102-4 auf der Bemessungsbasis von Teilsicherheitsbeiwerten

[7] EN 1995-1-1: Bemessung und Konstruktion von Holzbauten
Teil 1-1: Allgemeines – Allgemeine Regeln und Regeln für den Hochbau
Deutsche Fassung EN 1995-1-1:2004 + AC:2006 + A1:2008

[8] EN 1995-1-2: Bemessung und Konstruktion von Holzbauten
Teil 1-2: Allgemeines – Tragwerksbemessung für den Brandfall
Deutsche Fassung EN 1995-1-2:2004 + AC:2009

[9] EN 1990 Grundlagen der Tragwerksplanung
Deutsche Fassung EN 1990:2002

[10] EN 1991-1-3: Einwirkungen auf Tragwerke
Teil 1-3: Allgemeine Einwirkungen – Schneelasten
Deutsche Fassung EN 1991-1-3:2003 + AC:2009

[11] EN 1991-1-4: Einwirkungen auf Tragwerke
Teil 1-4: Allgemeine Einwirkungen – Windlasten
Deutsche Fassung EN 1991-1-4:2005 + A1:2010 + AC:2010

[12] Schneider K.J.: Bautabellen für Ingenieure 16. Auflage, Werner Verlag (2007)

[13] Blaß, H.J.; Ehlbeck J.; Kreuzinger H.; Steck G.: Erläuterungen zu DIN 1052:2004-08; Informationsdienst Holz (März 2005)

B: Index