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

All rights, including those of translations, are reserved.

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

© 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	Page		Contents	Page
1.	Introduction	6	6.3	RF-COMBI	59
1.1	About RX-TIMBER	6	6.4	Calculation	60
1.2	Company Profile	7	7.	Results	61
1.3	RX-TIMBER Team	8	7.1	Result Combinations	61
1.4	Using the Manual	8	7.2	Design - All	63
2.	Installation	9	7.3	Design by X-Location	64
2.1	System Requirements	9	7.4	Support Forces	65
2.2	Installation Process	9	7.5	Deformations	66
2.2.1	Installation from DVD	10	7.6	Result Diagrams	67
2.2.2	Network Installation	11	8.	Printout	68
2.2.3	Installing Updates and Other Programs	11	8.1	Printout Report	68
2.2.4	Parallel Installation of Dlubal Programs	11	8.2	Selection of Printing Data	71
3.	Principles for RX-TIMBER	12	8.3	Printout Report Header	73
3.1	Restrictions	12	8.4	Result Diagrams	75
3.2	Start RX-TIMBER	12	8.5	Graphics and Texts	77
4.	File Management	13	8.6	Printout Report Template	79
4.1	Project Manager	13	8.7	Presentation	80
4.1.1	Project Management	14	8.7.1	Layout	80
4.1.2	Model Management	19	8.7.2	Cover	81
4.1.3	Data Backup	21	8.7.3	Language	82
4.1.4	Settings	23	8.8	Printout	85
4.1.4.1	View	23	8.8.1	Direct Printing	85
4.1.4.2	Recycle Bin	24	8.8.2	Export	85
4.1.4.3	Directories	25	9.	General Functions	87
4.2	Creating a New Model	26	9.1	Units and Decimal Places	87
4.3	Network Management	27	9.2	Language Settings	88
5.	Input	28	9.3	Display Properties	89
5.1	Beam Type and Material	29	9.4	Export of Results	90
5.2	Geometry	36	10.	Glued-Laminated Beam	92
5.3	Loading	41	10.1	Example for Pitched Cambered Beam	92
5.4	Control Parameters	51	10.1.1	System and Loads	92
6.	Calculation	54	10.1.2	Input Data	93
6.1	Calculation Details	54	10.1.2.1	Beam Type and Material	93
6.2	Standard and National Annex	56	10.1.2.2	Geometry	95
6.2.1	General Parameters	56	10.1.2.3	Loads	96
6.2.2	Other Parameters	57	10.1.2.4	Control Parameters	97
6.2.3	Standards Used	58	10.1.3	Calculation	98



Contents

	Contents	Page		Contents	Page
10.1.3.1	Combinations with RF-COMBI	98	13.1	System and Loads	133
10.1.3.2	Start Calculation	98	13.2	Input of Model Data	134
10.1.4	Results	99	13.2.1	General Data	134
10.1.5	Documentation	102	13.2.2	Geometry	135
10.2	Example for Fish Beam	104	13.2.3	Loading	137
10.2.1	Geometry	104	13.2.4	Control Parameters	139
10.2.2	Lateral Buckling Design	105	13.2.5	RF-COMBI	140
11.	Continuous Beam	107	13.3	Results	141
11.1	System and Loads	107	13.3.1	Result Combinations	141
11.2	Input of Model Data	108	13.3.2	Designs	142
11.2.1	General Data	108	13.3.3	Other Results Windows	147
11.2.2	Geometry	109	14.	Purlin	148
11.2.3	Cross-section	109	14.1	System and Loads	148
11.2.4	Loads	110	14.2	Input of Model Data	149
11.2.5	Control Parameters	111	14.2.1	General Data	149
11.2.6	Effective Lengths	113	14.2.2	Geometry	150
11.3	RF-COMBI	113	14.2.3	Cross-section and Coupling	151
11.4	Results	117	14.2.4	Loads	152
11.4.1	Result combinations	117	14.2.5	Control Parameters	154
11.4.2	Design by Beam	118	14.2.6	Effective Lengths	155
11.4.3	Ultimate Limit State	120	14.2.7	Details	155
11.4.4	Serviceability Limit State	122	14.2.8	RF-COMBI	156
11.4.5	Support Forces	123	14.3	Results	157
11.5	Documentation	123	14.3.1	Result Combinations	157
12.	Column	124	14.3.2	Ultimate Limit State Designs	157
12.1	System and Loads	124	14.3.3	Other Results Windows	161
12.2	Input Data	125	15.	Brace	162
12.2.1	General Data	125	15.1	System and Loads	162
12.2.2	Loads	126	15.2	Input of Model Data	164
12.2.3	RF-COMBI	127	15.2.1	General Data	164
12.2.4	Control Parameters	128	15.2.2	Geometry	167
12.3	Results	129	15.2.3	Materials	168
12.3.1	Result Combinations	129	15.2.4	Cross-sections	169
12.3.2	Designs	130	15.2.5	Connections	170
12.3.3	Serviceability	132	15.2.6	Components	173
12.3.4	Other Results Windows	132	15.2.7	Loads	175
13.	Frame	133	15.2.8	Control Parameters	180

Dlubal

Contents

	Contents	Page		Contents	Page
15.2.9	Effective Lengths	181	15.3.4	Design by Cross-Section	184
15.2.10	RF-COMBI	181	15.3.5	Design by x-Location	185
15.3	Results	182	15.3.6	Support Forces	189
15.3.1	Load/Result Combinations	182	15.3.7	Printout Report	189
15.3.2	Design - All	183	A:	Literature	190
15.3.3	Design by Components	184	В:	Index	191



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.

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.

CEN	European Union
🛌 CSN	Czech Republic
🧰 DIN	Germany
DK 🗧	Denmark
💳 NEN	Netherlands
🚺 NF	France
🚍 ONORN	d Austria
📻 PN	Poland
📥 SFS	Finland
🌅 SS	Sweden
🚺 UNI	Italy

National annexes for EC 5



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

Programming

Ing. Martin Deyl Dipl.-Ing. Georg Dlubal Ing. Tomáš Ferencz Ing. Jiří Hanzálek Dipl.-Ing. (FH) Younes El Frem

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.° 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.

2

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.

E Language	
Ctond Alone	Deutsch
Stand-Alone	English
Applications Diubai	Česky
RX-HOLZ RX-TIMBER	Español
KRANBAHN CRANEWAY	Français
VERBUND-TR COMPOSITE-BEAM	Italiano
FE-BEUL PLATE-BUCKLING	Polski
	Português
	Русский
©2012 Ingenieur-Software Dlubal GmbH • www.dlubal.de • www.dlubal.com	

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*.

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 *Au*thor.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 <u>every</u> 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.



Program selection





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 standalone 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).

5

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-TIMBER2.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 doubleclicking 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.

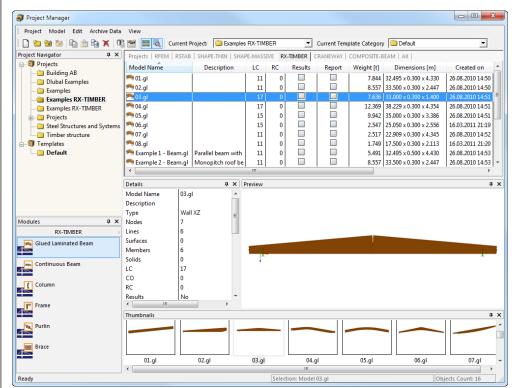


Figure 4.1: Project Manager

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.

I Project	Manage	r			
Project	Model	Edit	Arch	ive D	ata
	2	Þ 🖻		×	0
Project I N	ew Proje	ct (Ctrl+	N)	ц,	x
	Example	s	_		*
	Example	es RX-T	IMBE	R	

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.



Details



124

Ψ ×



4 File Management

Create Ne	w Project	×
New Proj	ect	
	Name	Description
Project:	Miller	New building for company hall
Place project under:	Projects	<u> </u>
Folder:	C:\Users\Pietz	cckerG.DLUBAL-INTERN\Documents\DlubalProjects\Miller
20	1	OK Cancel

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.

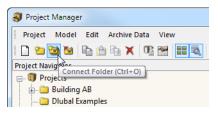


Figure 4.4: Button Connect Folder

•

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.

Project Navigator	ά×
🖃 🗊 Projects	
🖮 🍅 Building AB	
🚞 2012-01	Ξ
🚞 2012-02	_
🗀 2012-03	
🚞 2012-04	
🛅 Dlubal Examples	
🛅 Examples	
Examples RX-TIMBER	-
< III	- F

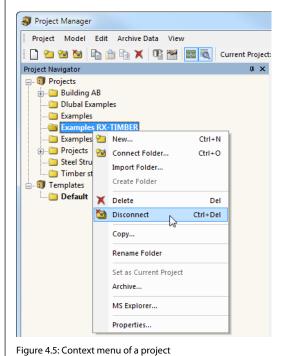




Disconnect a folder

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.





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).

Project Manager	
Project Model Edit	Archive Data View
i 🗋 🖄 🐿 🐚 💼 💼	Di 💓 🖬 💐
Project Navigator	Delete
🖃 🗊 Projects	Delete
🕀 👘 📴 Building AB	
🛅 Dlubal Examples	

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).

Project to	Сору	
	Name	Description
Project:	Examples RX-TIMBER	
Project location:	Projects\	
New Proje	ect	
	Name	Description
Project:	Copy of Examples RX-	RX-TIMBER - Reference examples description
Place project under:	Projects	T
Folder:	D:\Transfer\Timber Pro	iects - Buildings 🎆

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.

Project Pro	perties	×
General In	ıfo	
	Name	Description
Project:	Examples RX-TIMBER	RX-TIMBER - Reference examples description
Folder:	C:\Users\Public\Dlubal	\Projects\Timber structure\
٦		OK Cancel

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 <u>projects</u> 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.

Import fol	der	×				
Import Fo	Import Folder					
Place pro	ijects under: 🧊 Projects	. .				
Folder:	D:\Transfer\Timber Projects - Buildings					
	Connect folders including all subfolders					
D		OK Cancel				

Figure 4.9: Dialog box Import folder

124

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.

Project Manager												X
Project Model Edit Archive Data	View											
📄 🖄 🐿 🐚 💼 🔂 🗙 👊	M 📰 🔍 🖉	urrer	nt Project:	Exa	ample	s RX-TIMBE	R	Current	Template Category 📋	Default	•	
Project Navigator 🛛 📮 🗙	Projects RFE	M F	RSTAB SH	APE-THI	N S	HAPE-MASS	SIVE RX-TIN	MBER CRANI	EWAY COMPOSITE-BEA	M All		
E- 🗊 Projects	Model Name		L	C RC	2	Results	Report	Weight [t]	Dimensions [m]	Created on	Created by	*
Building AB	🦰 01.gl			11	0			7.844	32.495 x 0.300 x 4.330	26.08.2010 14:50	Ing Software	D
Copy of Examples RX-TIMBER Dlubal Examples	🖱 02.gl			11	0			8.557	33.500 x 0.300 x 2.447	26.08.2010 14:50	Ing Software	
Examples	🦰 03.gl		New With	n		•		7.636	33.000 x 0.300 x 1.400	26.08.2010 14:51	Ing Software	D -
Examples RX-TIMBER	🦰 04.gl		Open		N			12.369	38.229 x 0.300 x 4.354	26.08.2010 14:51	Ing Software	D
Projects	🦰 05.gl		Open Wi	th	3	•		9.942	35.000 x 0.300 x 3.386	26.08.2010 14:51	Ing Software	D
🗀 Steel Structures and Systems	🦰 06.gl							2.547	25.050 x 0.300 x 2.556	16.03.2011 21:19		
Timber structure	🦰 07.gl	×	Delete			Del		2.517	22.909 x 0.300 x 4.345	26.08.2010 14:52	Ing Software	D +
🗄 🧊 Templates	1		Delete Re	sults								F
Default	Details		Delete Pr	intout R	eport	s	Preview					аx
	Model Name		Send									-
	Description		Jena				1					
	Type		Сору									
	Nodes		Archive									
Modules 4 ×	Lines						ļ					
RX-TIMBER =	Surfaces		Set as 'Re	ad Only	r -							
Glued Laminated Beam	Members		Set as 'No	ot Read (Only		1.00					
	Solids		MS Explo								1	
Continuous Beam	LC		M3 Explo	ICI								
	со		Addition	al Info vi	ia MS	Excel						
	RC		Addition	al Info vi	ia MS	Word						
Column	Results		History									
	Thumbnails						-					ąх
Frame		L	Propertie	s						1	_	<i>+</i> ^
												=
Purlin												
	-		,			7					-	
Brace												
I												-
Opens an existing model.					_			Selection:	Model 02.gl	Obj	ects Count: 16	

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.

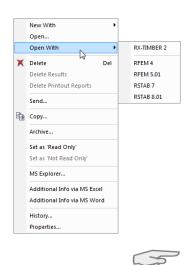
Shift / copy a model

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.



4 File Management

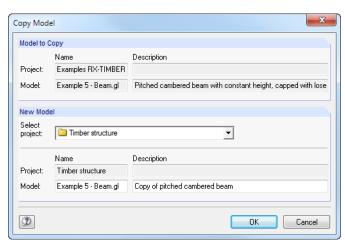


Figure 4.11: Dialog box Copy Model

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).

lodel Pro		X
General In Model:	Name New name	Description Pitched cambered beam with variable height
File name		\Dlubal\Projects\Timber structure\Example 6 - Beam.gl.rh2
Result	s 🔀	
2	à	OK Cancel

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.



X

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).

	From user		Date			
Created:	Ing Softwar	e Dlubal Gmb⊦	16.	03.2011 21:23		
Last modified	: Gundel Pietz	cker	08.	08.2013 16:56		
History						
No.	User	Opened a	t	Saved	Comment	
3 Gund	lel Pietzcker	08.08.201316	:55	08.08.2013 16:56		
2 Gund	lel Pietzcker	08.08.2013 16	:51	08.08.2013 16:53		
1 Ing	Software Dluba	16.03.2011 21	:23	16.03.2011 21:23		

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).

rchive		×
Archive	File	
Name:	Backup RX-TIMBER	
Folder:	D:\Transfer\Backup	A
Archive	Also	
🔽 Resi	ults	
🔽 Print	out reports	
🔽 All su	ubprojects	
	nown files les in folder)	
2		OK Cancel

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.

Select Pro	oject	Select Models to Extra	ct			
🔁 1	limber structure	File Name	Program	Opened i	File Size	Created
		🥱 01.clm.rh2	RX-TIMBER 2		384 kB	
		🦰 01.dlt.rh2	RX-TIMBER 2	1	340 kB	
		🦰 01.frm.rh2	RX-TIMBER 2		416 kB	
		🦰 01.gl.rh2	RX-TIMBER 2	1	328 kB	L
		🦰 01.pft.rh2	RX-TIMBER 2		324 kB	
		🙈 01.vrb.rh2	RX-TIMBER 2		420 kB	
		🦰 02.clm.rh2	RX-TIMBER 2		272 kB	
		🦰 02.dlt.rh2	RX-TIMBER 2		336 kB	
		🦰 02.frm.rh2	RX-TIMBER 2		852 kB	
		🦰 02.gl.rh2	RX-TIMBER 2		368 kB	
		🦰 02.pft.rh2	RX-TIMBER 2		460 kB	
		🦰 02. vrb. rh2	RX-TIMBER 2		460 kB	
		🦰 03.dlt.rh2	RX-TIMBER 2		900 kB	
🔽 Also e	xtract embedded	💏 03.frm.rh2	RX-TIMBER 2		320 kB	
subpre		💏 03.gl.rh2	RX-TIMBER 2		480 kB	
📃 Use o	riginal settings	← ∩2 ~# -#-2		1 1	AEC LD I	÷.
Extract to	Project					
	Name	Description				
Project:	Timber structure	•				
Original project:	Timber structure	•				
Place project						
under:	🛅 Dlubal Exan	nples	-			
Folder:	C:\Users\Public	:\Dlubal\Projects\2012\T	imber structure		M	

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
2	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.

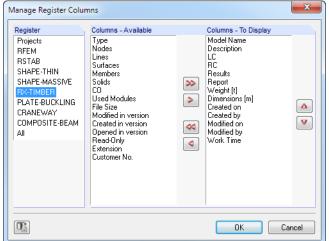
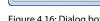


Figure 4.16: Dialog box Manage Register Columns

>>	
>	4

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.





```
To arra
```

4 File Management



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.

Select Models to Resi	ore				
Project Name	File Name	Deleted Date	Program	Opened in	*
Timber structure	🗪 Example 2 - Brace.vrb.rh1	08.08.2013 17:04	RF-HOVERB, RF	4.09.0270	
Timber structure	🦰 Example 1 - Brace.vrb.rh1	08.08.2013 17:04	RF-HOVERB, RF	4.10.2270	Ε
Timber structure	🦰 02.vrb.rh2	08.08.2013 17:04	RF-HOVERB, RF	4.09.0270	
Timber structure	🦰 01.vrb.rh2	08.08.2013 17:04	RF-HOVERB, RF	5.01.0075.3	
Timber structure	🐴 2D Beam Model.rs8	08.08.2013 17:04	RSTAB 8	8.01.0006.1	
Timber structure	🦐 Example 2 - Column.clm.rh1	08.08.2013 17:04	RX-HOLZ Columi	4.06.0690	
Timber structure	🦰 Example 1 - Column.clm.rh1	08.08.2013 17:04	RX-HOLZ Columi	4.10.2270	
Timber structure	🦰 02.clm.rh2	08.08.2013 17:04	RX-HOLZ Columi	5.01.0075.3	
Timber structure	🦰 01.clm.rh2	08.08.2013 17:04	RX-HOLZ Columi	4.05.1410	
Timber structure	🦐 Example 7 - Continuous Beam.dlt.rh1	08.08.2013 17:04	RX-HOLZ Contin	4.06.0690	
Timber structure	🦐 Example 6 - Continuous Beam.dlt.rh1	08.08.2013 17:04	RX-HOLZ Contin	4.05.1410	
Timber structure	🚗 Example 5 - Continuous Beam.dlt.rh1	08.08.2013 17:04	RX-HOLZ Contin	4.05.1410	
Timber structure	🙈 Example 4 - Continuous Beam.dlt.rh1	08.08.2013 17:04	RX-HOLZ Contin	4.05.1410	
Timber structure	🙈 Example 3 - Continuous Beam.dlt.rh1	08.08.2013 17:04	RX-HOLZ Contin	4.05.1410	
Timber structure	🗪 Example 2 - Continuous Beam.dlt.rh1	08.08.2013 17:04	RX-HOLZ Contin	4.05.1410	-
•		1		4	
X) ()	X X 1	7

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.

4 File Management



Settings for Dlubal Recycle Bin	X
Dlubal Recycle Bin	
Folder for Dlubal Recycle Bin:	
C:\Users\Public\Dlubal\Project Manager\Recy	cle Bin\ 🙀
Options ♥ Warning in case of insufficient disk space Less than: 100 ★ [MB]	
	OK Cancel

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.

Settings	x
Database File	
Category: Project Manager	
Folder/File:	
C:\Users\Public\Dlubal\Project Manager\Pro.DLP	<u>h</u>
Options	
Warning if opened or closed model is not connected to project Wilde old versions of applications from option 'New with' of context menu	
Open selected file(s) in newest version of application native version newest version OK	

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.

Project	Manager	·				
Project	Model	Edit	Archive	Data	View	1
Project Nav	idator Model Example	5		Œ		

Figure 4.20: Button New Model

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

Model Name	Description
Principal Beam	Main beam with maximum loading
Project Name	Description
Examples RX-TIM	
Folder:	
C:\Users\Public\Dlubal	\Projects\Timber structure\Principal Beam.gl.rh2

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.

RX-Timber
The file of model 'C:\Users\Public\Dlubal\Projects\Timber structure\02.gl.rh2' is read-only , possibly because it is already opened by you or another user.
Do you want to open a copy of the model?
Yes

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



5. Input



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

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

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.

Input Data	1.1 Beam Type and Material			
 Beam Type and Material Geometry 	Beam Type	According to	Standard / National Annex	
Loads Control Parameters Result Result Combinations Design - All Design by Location X Support Forces Deformations	 1. Parallel beam 2. Monopitch roof beam 3. Double tapered beam 4. Curved beam 5. Pitched cambered beam with constant height 6. Pitched cambered beam with variable height 7. Fish beam - Parabolic 8. Fish beam - Linear with roundings in central area Asymmetric layout Cantilevers: Taper 		i-1:2004:11 IN	RX-TIMBER Glued laminated beam
	Capped with loose ridge wedge	Stiffening Ele	ments for Transversal Tension	
	Notch on supports	🔽 Apply	Constructive if required	Single-span beam fo
	Material		 Full absorption of tension stresses perpendicular to the grain 	roof structures (solid web girder) built wit glued laminated tim
	Glulam Timber GL24c Comment	Bonded set	with wood-screw thread Did wood or wood-based strips	
0 5 5	Calculation Details Nat. Annex		BF-COMBI	OK Cancel

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



Cancel

OK

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 (\rightarrow chapter 6.1)
Nat. Annex	Opens the standard parameters for the national annex (\rightarrow chapter 6.2)
Report	Displays the print preview (\rightarrow 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 can			With lose ridge wedge	Stiffening ele- ments for trans- versal tension
Bea	Asyn out	Horizontal	Parallel	Taper	Offset	With lo: wedge	Stiff mer vers
1			х	х	x		
2		х		х	x		
3	х	х		х	x		х
4	х	х	х	х	x		х
5	х	х	х	х	x	x	х
6	х	х	х	х	х	x	х
7			х	х	х		
8			х	х	х		

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.

According to Standard / National Annex	
📧 EN 1995-1-1:2004-11 🔽 🔳 DIN	- 💽
🔳 DIN 1052:2008-12 Germany	
💽 EN 1995-1-1:2004-11 European Union	

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

15

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

	tandards Used								
Partial Factor Accord	ding to 2.4.1		Limit	/alues of Defor	rmations Ac	cording to Ta	ble 7.2		
PT: Permanent and	d transient design sit	uation	- Cha	racteristic (rare)	design situ	ation			
 Solid timber 	γM :	1.300 🚖			Fixed o	n Both Sides	0	verhanging	
 Glued laminated tin 	nber ym:	1.250 🜩	SC :	Winst	≤17	300 🚖	≤l _k /	150 🚖	
AC : Accidental des	ign situation		- Qua	si-permanent de	esign situati	on			
	- γM:	1.000 🚔	S1 :	Wfin - Wc	≤17	250 🔶	$\leq l_{k}/$	125 🔶	Eq. (7.2)
			S2:	W fin	≤17	150 🜩	$\leq l_k \ell$	75 🔶	
Partial Factor Accord	-								
For timber in fire	γM,fi∶	1.000 🜩							
Modification Factor k	-mod According to	Table 3.1							
		ervice Class							
LDC	1	2	3						
Permanent:	0.600	0.600 🚖	0.500 🚖						
Long-term:	0.700 🚔	0.700 🚖	0.550 🚖						
- Medium-term:	0.800 🚔	0.800 🚖	0.650 🚖						
- Short-term:	0.900 🚖	0.900 🚖	0.700 🚖						
- Instantaneous:	1.100 🚖	1.100 🚖	0.900 🚖						
Data for Fire Protect	ion According to EN	1995-1-2, 2.3, Table	3.1 and 4.2.	2					
	Softwood	Glulam	Hard	wood					
Charring rate	βn: 0.80 🚔	0.70 🚖	0.	55 🚔 (mm/mi	in]				
Increased charring	dg: 7.00 🚔	7.00 🚖	7.	00 🚖 [mm]					
Factor	k fi : 1.250 🚔	1.150 🚔	1.2	50 🚖					

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.

Glulam Timber GL24c	
Glulam Timber GL24c	
Glulam Timber GL28c	
Glulam Timber GL32c	
Glulam Timber GL36c	
Glulam Timber GL24h	
Glulam Timber GL28h	
Glulam Timber GL32h	
Glulam Timber GL36h	

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.

Filter	Material to Select			
Material category group:	Material Description	Standard		
Timber	- Glulam Timber GL24h	SS-EN	1995-1-1:2009	
	Glulam Timber GL28h	🔚 SS-EN	1995-1-1:2009	
Material category:	Glulam Timber GL 32h		1995-1-1:2009	
📒 Glulam	Glulam Timber GL36h		1995-1-1:2009	
	Glulam Timber GL24c		1995-1-1:2009	
Standard group:	Glulam Timber GL28c		1995-1-1:2009	
🔚 SS-EN	Glulam Timber GL32c		1995-1-1:2009	
			1995-1-1:2009	
Standard:	Glulam Timber GL36c			
22 SS-EN 1995-1-1:2009	Glulam Timber GL24h (Perpendicular to Grain)		1995-1-1:2009	
	Glulam Timber GL28h (Perpendicular to Grain)		1995-1-1:2009	
	Glulam Timber GL32h (Perpendicular to Grain)	SS-EN	1995-1-1:2009	
	🔲 Glulam Timber GL36h (Perpendicular to Grain)	🔚 SS-EN	1995-1-1:2009	
	Glulam Timber GL24c (Perpendicular to Grain)	🔚 SS-EN	1995-1-1:2009	
🗌 Include invalid 🛛	🗊 📒 Glulam Timber GL28c (Perpendicular to Grain)	🔚 SS-EN	1995-1-1:2009	
		1		
🔄 Favorites only 🔣				7
Material Properties		Giulam Timber	GL24c SS-EN	1995-1-1:20
Modulus of Elasticity		E	11600.00	N/mm ²
Shear Modulus		G	590.00	
- Specific Weight		γ		kN/m ³
Coefficient of Thermal Expan	sion	α	5.0000E-06	
Partial Safety Factor		7M	1.25	
Additional Properties				
Characteristic Strength for Be	ending	fm,k	24.00	N/mm ²
- Characteristic Strength for Be	ending for Middle Lamellas	fm,k,internal		N/mm ²
Characteristic Strength for Te	ension	ft,0,k		N/mm ²
 Characteristic Strength for Te 	ension Perpendicular	ft,90,k		N/mm ²
 Characteristic Strength for Co 		fc,0,k		N/mm ²
 Characteristic Strength for Co 		fc,90,k		N/mm ²
Characteristic Strength for Sh	near/Torsion	f _{v,k}		N/mm ²
Modulus of Elasticity Parallel		E0,mean	11600.00	
 Modulus of Elasticity Perpend 	dicular	E90,mean		N/mm ²
- Shear Modulus		Gmean		N/mm ²
 Density 		Pk	350.0	kg/m ³

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.



2

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

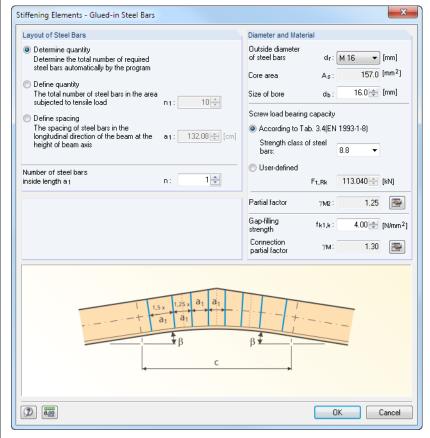


Figure 5.7: Dialog box Stiffening Elements - Glued-in Steel Bars

Program RX-TIMBER © 2013 Dlubal Software GmbH

5 Input



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 a₁. 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.

Bonded solid wood or wood-based strips

Layout of Wood Material Strips		Dimensions and Material
 Determine quantity Determine the total number of required strips automatically by the program Define quantity The total number of strips in the area subjected to tensile load Define spacing The spacing of strips in the longitudinal direction of the beam at the height of beam axis 	n1: 10 - 10 - 10 - 10 - 10 - 10 - 10 - 10	Length of strips along with beam axis Ir: 10.00 ♣ [cm] Thickness of wood material strips Ir: 2.00 ♣ [cm] Strength Image: Solid wood Plywood Image: Solid wood Poplar and Softwood Timber Image: Solid wood Image: Strength Tensile strength ft,k: 8.00 ♣ [N/mn²] Safety factor 7M: 1.30 Image: Solid wood
	.5x 1.25x a1 a1 a1 a1 a1	Joint strength f _{k3,k} : 1.50 ⊕ [N/tmm ²]
	4	

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 I_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_{tk}* and the *Safety factor* γ_M of the selected timber strength class are indicated for information.

The *Joint strength* $f_{k_{3,k}}$ 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

iffening Elements - Bonded A		Material
Length of agglomerated wood plates along beam axis	Ir: 583.09 [cm]	Strength class: F25/10 parallel
Thickness of agglomerated wood plates	tr: 2.00 [cm]	Tensile strength ft,k: 18.00 + [N/mm ²]
		Partial factor 7M: 1.20
		Joint strength f _{k3,k} : 1.50 [] [N/mm ²]
	. – Ir	

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

In the dialog section *Dimensions*, the *Length* I_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

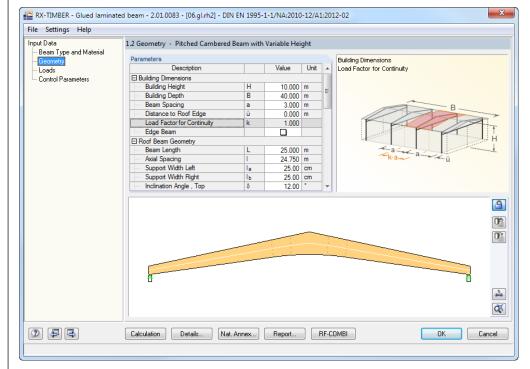


Figure 5.10: Window 1.2 Geometry

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 \ddot{u} allows you to consider an additional load application area for edge trusses. As load application area RX-TIMBER assumes $2 \cdot a/2$ by default, for edge trusses it assumes $a/2 + \ddot{u}$.



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 I* changes automatically. When you enter a value for the *Support Width Left Ia* or *Right Ib*, the axial spacing will be adjusted accordingly.

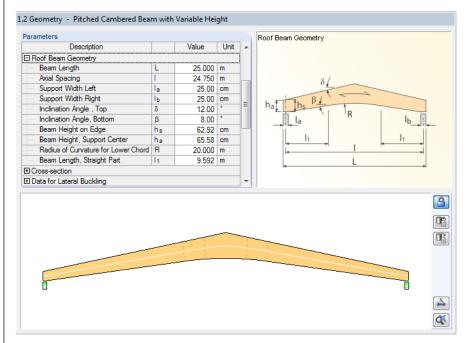


Figure 5.11: Parameters of the Roof Beam Geometry

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.

RX-TIMBER Warning No. 1015									
Non-compatible parameter input for geometry! Error: beta > delta									
Do you want to set the parameters used before the check was disabled? If not, please modify the input data.									

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).

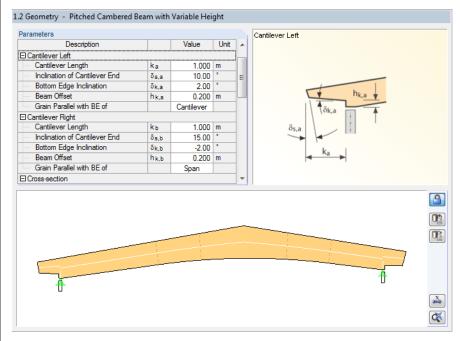
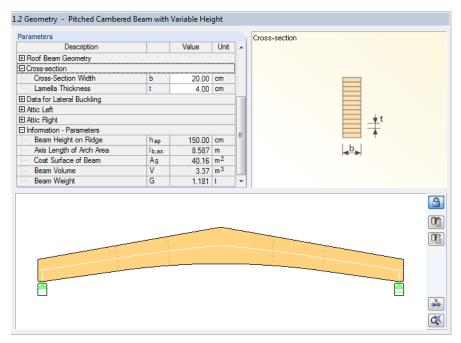


Figure 5.13: Parameters of Cantilever

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.







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.

Parameters				Data for Lateral Buckling	
Description		Value	Unit		
Data for Lateral Buckling					
 Beam Endangered by Lateral Bucklir 		2		-	
 Lateral Supports Available 		V			
 Spacing of Lateral Supports 	С	2.000	m		
 Bracing Distance 	е	60.00	cm		
Lateral supports resist to fire					
⊞ Attic Left					
⊞ Attic Right					C C
Information - Parameters				=	
 Beam Height on Ridge 	hap	150.00	cm		
 Axis Length of Arch Area 	I _{b,ax}	8.587	m		Ú,
 Coat Surface of Beam 	As	40.16	m ²	1-	
Beam Volume	V	3.37	m ³	1.	

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.

Parameters					Attic Left
Description		Value	Unit	-	x .
				1	
🗆 Attic Left					
- Existing		<			h _{p,a}
Height	h _{p,a}	0.500	m		
🗆 Attic Right					
Existing					h
Height	h _{p,b}	0.500	m		
Information - Parameters					*********
Beam Height on Ridge	hap	150.00	cm	Ξ	
 Axis Length of Arch Area 	I _{b,ax}	8.587	m		
 Coat Surface of Beam 	As	40.16			
Beam Volume	V	3.37	m ³		
Beam Weight	G	1,181	t	-	,

Figure 5.16: Parameters for Attic Left and Attic Right



The attic parameters affect the snow load generation:

Load Ca	ases										×	
Load	Case						Self-weig	ght				
LC No	o.: LC41 ▼ •	•	Consi factor	der with		÷						
Descr	iption: Snow (Both Side	es Full)										
Load	Type: Impos	ed Load Ca										
🔘 Pe	ermanent	-	C									
🔘 Va	ariable						Comment					
	cidental										^	
🔘 Se	Seismic											
Load	Load Load Case LC41											
	А	B	С	D	E	F	G	Н		J		
		Load			Load Para				Across Tot			
No.	Load Type	Direction	Load Reference		p2[kN/m]	A [m]	B [m]	in %	Length	Comme	int	
1	Line Load	ZP	Left Span Side	1.560 1.560								
2	Line Load	ZP	Right Span Side									
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		
6								-				
7												
8												
2	z								ОК	Ca	ncel	

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.

arameters					Information - Parameters
Description		Value	Unit	-	
Information - Parameters				1	lb,ax
 Beam Height on Ridge 	h _{max}	192.81	cm		lif
 Support Difference 	Δh	-0.587	m		
 Axis Length of Arch Area 	I _{b,ax}	13.568	m		h _{max} ,
 Beam Height on Edge Left 	h _{s,a}	29.19	cm		Elf,a
 Beam Height on Edge Right 	h _{s,b}	35.56	cm		Ha top
 Length of Free Ridge Wedge 	lıf	5.341	m		H
 Angle of Secondary Ridge, Left Side 	٤lf,a	9.78	•		δ_h
 Angle of Secondary Ridge, Right Sid 	۵lf,b	9.82	•	1	
 Coat Surface of Beam 	As	56.15	m ²		
 Beam Volume (without Ridge Wedge 	VT	4.15	m ³	Ξ	
 Ridge Wedge Volume 	VF	0.25	m ³		
- Total Volume	Vsum	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.

nput Data	1.3 Lo	1.3 Loading - Pitched Cambered Beam with Variable Height											
Beam Type and Material Geometry	Perm	anent Action			Snow Load								
Loads		A	B	C	D	-	Altitude A: 200 (m)						
Control Parameters	No.	Roof Layer	Thickness		_								
		Material	d [cm]	[kN/m ²]	Comment	= =	O Determine automatically						
		Trapezoidal Sheet M		0.150		-11	Snow load zone SZ: 2	6					
	2	Vapour Barrier Rock Wool	30.00	0.020		_							
	4	Purlins	30.00	0.300		- 11	Topography type TT: Normal 👻						
	5	Gypsum Boards incl		0.150			North German Plain						
		Gypsain boards inci					Snow load of overhang						
	Rool	f structure	gk,2'∶	0.640 🔶	[kN/m ²]	RA							
			gk,2 :	1.920 ≑	[kN/m]	BA	Snow load on snow guard	Ŀ					
	Bear	m self-weight	9k.1 :	0.701 🔶		BA	○ Define snow load s _K : 0.850 ← [kN/m ²] B.					
		rage)	9K,1 -	0.701	[KN/m]	RA	manually	В.					
				0.004			sk: 2.550 ⊕ [kN/m]	D.					
			gk:	2.621 ≑	[kN/m]	RA	Wind Load						
	Impo	sed Load					Determine automatically						
	Impo	sed load	pk::	0.000 🔶	[kN/m ²]	BA	Wind zone WZ: 1						
				0.000	ILNUm1	BA	Terrain category TC: Category III						
			pk:	0.000	[KN/m]	DA		-					
	Impo	sed load category					Fundamental basic wind velocity vb.p: 22.5 (m/s)						
		ording to EN 1991-1-1:		H 👻									
							Body shells with permeable walls						
	Serv	ice Class	Show P	Generated Lo	ads		○ Define peak velocity q(z)': 0.506 ↓ [kN/m ²	1 B					
	SEC	L: 2 🔻 🖍		resp.			pressure manually	, B.					
	020		Define.	Additional Lo	ads	B	q(z): 1.519 (kN/m)	В					

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.





~

1

Category	Mate	rial							
Timber and Timber-based Mater Concrete and Mortar	No.	A	B Specific Weight	С					
Metals	NO.	Material	[kN/m ²]	Commen					
Floor Coverings	1	Roof Panels	2.500	Commen					
Barier Materials, Insulating Mate Slabs, Meshes and Sheets	2	Flat Tiles, Flat Clay Roofing Tile incl.	0.550						
Roof Cladding Material	3	Concrete Roofing Tiles incl. Lathing	0.600						
Roof Substructure	4	Eat-tiled Concrete Bricks	0.600						
Moisture-proof roofing	5	Plain-tailed Cinderblock Double Boo	0.000						
Various Volume Loads Various Surface Loads	6	Double-skin Partition Roofing with 2	0.250						
Valious Surface Edaus	7	Etemit	0.200						
	8	Mission Tiling	0.900						
	9	Double Slate Roof Cladding	0.600						
	10	Single Slate Roof Cladding	0.500						
	11	Trapezoidal Sheet Metal	0.150						
	12	Glass	0.400						
	13	Plexiglass	0.080						
	14	Canvas	0.030						
	15	Shingles incl. Lathing	0.250						
	16	Straw Roofing incl. Lathing	0.700						
	17	Purlins	0.150						
	18	Rafters	0.100						
	19								
	•								

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: $p_k = p_k' \cdot (\frac{a}{2} + \ddot{u}) \cdot k$
- 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.

mpose	ed Load	Catego	ſy		Example				
_	Α	B	С	D	Freely walkable areas, e.g. museum				
No.	Cate	gory	Ratio	qk [kN/m²]	areas, exhibition grounds, etc. and entrance areas in public buildings				
1	Α	A1	Ceiling structures	1.500	and hotels, yard cellar floors not				
2		A2	Stairways	3.000	open to traffic				
3		A3	Balconies	3.000					
4	В	B1	Offices	2.500					
5	С	C1	Rooms, meeting halls and areas, which can be used	3.000	1				
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	н	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*.



0

Ē.

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.

0

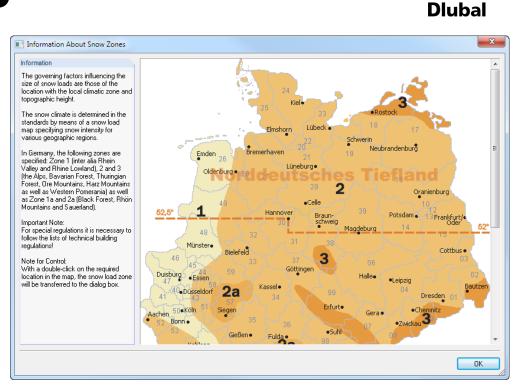


Figure 5.22: Map showing snow load zones in Germany

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

Snow Load of Overhang	X
Load Image	Snow load of overhang in kN/m $S_{e} = \frac{k \cdot s^{2}}{\gamma}$ ^S Snow load for roof in kN/m ² k Coefficient considering the irregular shape of snow (k=0.4) γ Specific weight of snow (γ = 3 kN/m ²)
	ОК

Figure 5.23: Dialog box Snow Load of Overhang

5 Input



Q)

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.

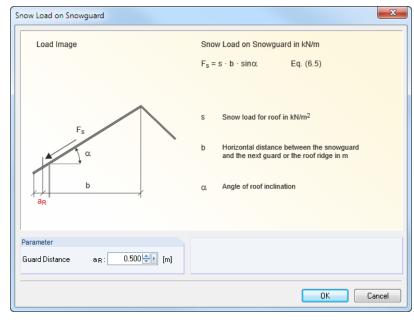


Figure 5.24: Dialog box Snow Load on Snowguard

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

5

0

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*.

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).

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).

4 Diubal

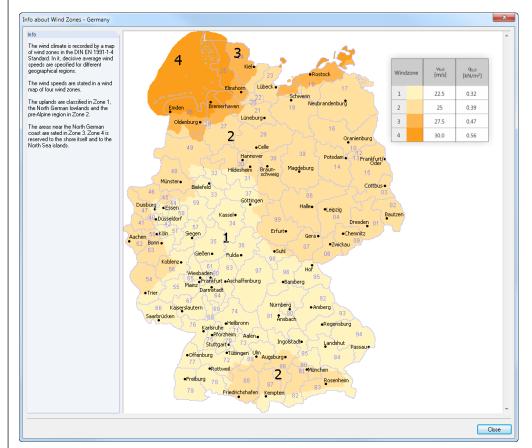


Figure 5.25: Wind zone map for Germany

The wind zone sets the value of the *Fundamental basic wind velocity* $v_{b,o}$ 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:

Coefficients for Wind Load Gen	neration X
Velocity Pressure Coefficients	
Coefficient of orography	Cp:
Coefficient of wind direction	Cdir: 1.000
Coefficient of season	Cseason : 1.000
Coefficient of turbulency	kı:
Terrain factor According to 4.3.2	
User-defined	kr:
Air density	ρ: 1.25 [kg/m ³]
	OK Cancel

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*).

5 Input



Q

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

Wall Parameters	
Automatically, cpi = according to EN 19	+0.2 or -0.3 91-1-4, 7.2.9(6), Note 2
User-defined wall pa according to EN 19	arameters 91-1-4, 7.2.9(6), Figure 7.13
	μв 1.00 🚔 [-]
μι [-]	μ _B 0.90 (-
μι []	
	♥ _y I
	μf 1.00 🜩 [·]
	OK

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.

Load C	ases									×			
Load	Load Case Self-weight												
LC N	o.: LC51 💌	• • •		Short	description:	w(q,I,AA)		Consider v	vith				
Desc	Description: Wind Transversely to Ridge (left)(AA)												
Load	Load Type: Imposed Load Category: Load Duration Class:												
	Permanent												
⊚ V	Variable Comment Comment												
- A	Accidental												
🔘 S	Seismic												
Load	Load Load Case LC51												
	A	B	С	D	E	F	G	Н		J			
N-		Load			Load Para				Across Tot				
No.	Load Type	Direction	Load Reference		p2[kN/m]	A [m]	B [m]	in %	Length	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													
8													
<u> </u>													
2	<u></u>									DK Cancel			

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.

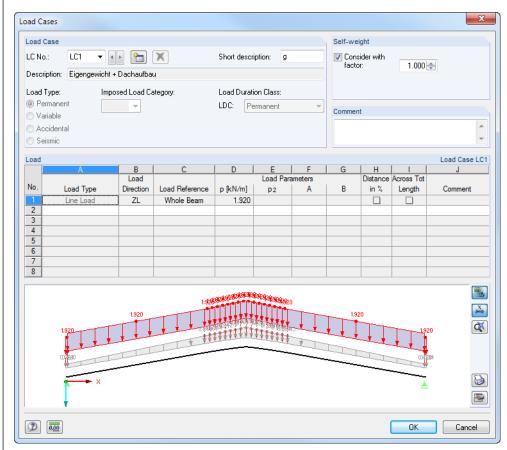


Figure 5.29: Dialog box Load Cases

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 [4] 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.

Show Generated Loads... resp. Define Additional Loads...

5 Input

Load Duration Class:

LDC: Short-term Long-term Medium-term Short-term Instantaneous



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.

 \sim

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.

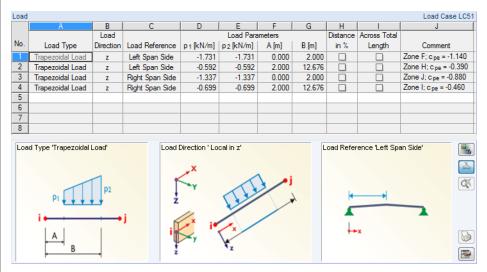


Figure 5.30: Dialog box Load Cases: explanatory graphics for load parameters

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.

5 Input



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.

Line Load Concentrated Load Line Load Trapezoidal Load Temperature Change Temperature Differential Concentrated Moment Line Moment Trapezoidal Moment

 ZL

 x - Local in x

 z - Local in z

 XL - Global in X on true length

 ZL - Global in Z on true length

 XP - Global in X on grojected length

 ZP - Global in Z on projected length

Whole Beam Whole Beam Cantilever Left Inner Span Left Span Side Right Span Side Cantilever Right











5.4 Control Parameters

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

put Data Beam Type and Material	1.4 Control Parameters - Pitched Cambered Beam with Varial	ble Height						
Geometry	Design of	Support Modelling						
Loads Control Parameters	 Static equilibrium (EQU) Ultimate limit state (STR) 	Left Support: Horizontal fixed Column Horizontal free	13					
	Serviceability limit state Precamber wc: 0.0 (mm) Imm	Right Support: Horizontal fixed Column Horizontal fixee As left column Supports on center line	P 1					
	✓ Fire protection	Optimization						
	Fire Resistance Class: O R 30	Perform optimization Maximum ratio: 1.00	ב כ					
	🔘 R 90	Description Symbol Parameter Un	nit					
	○ B 45 ÷ [min]	Optimize Beam Height						
		Beam Height, Support Center ha						
		- Minimum possible h _{a,min} 40.00 cm						
	Charring:	- Maximum possible ha,max 200.00 cm						
		Optimize Shape Optimize Cross-Section Width						
	V	Calculation Parameters						
	To Display	Generate supplementary combinations from favorable permanent actions						
	Support forces Deformations	Distribute permanent load field-by-field						
	Support compression	divisions for						

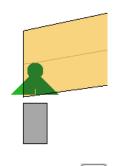
Figure 5.31: Window 1.4 Control Parameters

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.

 Rectangular Circular Cross-section from 	x: 35.00 = [cm] y: 35.00 = [cm] hibrary:		
Parameters			
Column height Material of column:	h: 5.000 [w] [m]		
4 Concrete C3	10/37 EN 1992-1-1:2004 👻 🔊 🖭		
Support conditions at column head:	© Hinged ⊙ Partial rigid		
Support conditions at	: 🗇 Hinged	Support Springs Due to C	olumn
column base:		Cu,X 495.206	[kN/m]
	💿 Rigid 🛛 🕰 🛨	Cu,Z 808500.000	[kN/m]
	17 7-11 7.	С _{о, Y} 0.000	[kNm/rad]

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.



1





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.

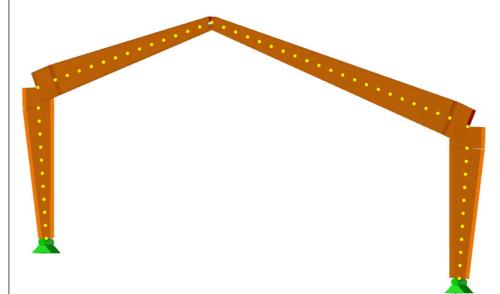


Figure 5.33: Principle of member divisions for result diagrams

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

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

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.

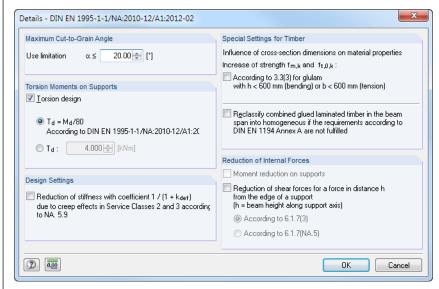


Figure 6.1: Dialog box Details according to DIN EN 1995-1-1 for RX-TIMBER Glued-Laminated Beam

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.

6 Calculation



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.

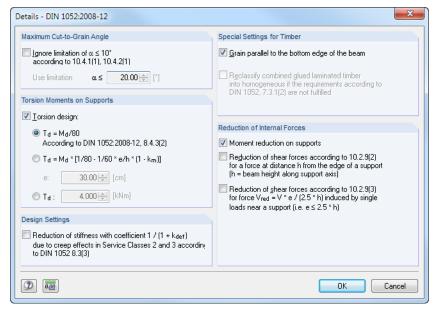


Figure 6.2: Dialog box Details according to DIN 1052 for RX-TIMBER Continuous Beam

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

Nat. 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

		5-1-1/NA:2010-12/	A1:2012-02						<u> </u>
General Other !									
Partial Factor Acco	rding to 2.4.1		Limit	Values of Defe	ormations A	ccording to Ta	able 7.2		
PT: Permanent ar	nd transient design si	tuation	- Ch	aracteristic (rare	e) design siti	uation			
 Solid timber 	γM :	1.300 🚖			Fixed	on Both Sides	0	verhanging	
 Glued laminated t 	imber γм:	1.300 🜲	sc	: Winst	≤17	300 🚖	≤l _k /	150 🚖	
AC : Accidental de	sign situation		- Qu	asi-permanent (design situal	ion			
	γм:	1.000 🚔	S1	W fin - W c	≤17	250 🔶	≤l _k /	125 🔶	Eq. (7.2)
Destini Frankra Array			S2	: Wifin	≤17	150 🔶	$\leq l_{k}/$	75 🚖	
Partial Factor Acco For timber in fire	rding to EN 1995-1-2								
For timber in fire	γM,fi∶	1.000 🚖							
Modification Factor	k-mod According to	Table 3.1							
		Service Class							
LDC	1	2	3						
Permanent:	0.600 🚔	0.600 🚖	0.500 🚖						
 Long-term: 	0.700 🚔	0.700 🚖	0.550 🚖						
- Medium-term:	0.800 🚖	0.800 🚖	0.650 🔶						
- Short-term:	0.900 🚖	0.900 🚖	0.700 🔶						
- Instantaneous:	1.100 🚖	1.100 🜲	0.900 🚖						
Data for Fire Protec	tion According to EN	l 1995-1-2, 2.3, Table	3.1 and 4.2	2.2					
	Softwood	Glulam	Har	dwood					
Charring rate	βn: 0.80 🚔	0.70 🚖	0).55 🚔 (mm/r	nin]				
Increased charring	do: 7.00 🚔	7.00 🚖	7	7.00 🚔 [mm]					
Factor	kri: 1.250 🚔	1.150 🚔	1.	250 🚖					
A 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1							_		
2 0.00								ОК	Cancel

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

National Annex Settings - DIN EN 1	1995-1-1/NA:2010-12/A1:2	012-02	X
General Other Standards Used			
Partial Factor for Stiffenings for Tra	insversal Tension	Shear Factor	
Glued-in steel bars	ум <u>2</u> : 1.250 🚔	According to 6.1.7	
Steel bars with wood-screw	ум <u>2</u> : 1.250 🚔	🔘 User-defined	
Plywood	ум: 1.200 🚔	Solid timber	ker:
Connections	γм: 1.300 🖨	Glued laminated timber	kor:
Volume Factor		Factor of Compression Perpendicul	ar to Grain
According to 6.4.3		According to 6.1.5	0
O User-defined		🔘 User-defined	
Solid timber	kvol : 1.000 🚔	Softwood timber	k _{c,90} : 1.500
Glued laminated timber	kvol:	Glued laminated timber	kc,90: 1.750 🚔
Factor of Stress Distribution		Factor of Stress Distribution Accord	ding to DIN EN 1995-1-1/NA:2010-12
According to 6.4.3		According to NCI NA.6.8.5	
User-defined		🔘 User-defined	
Double tapered and curved beams	k dis : 1.400 🚔	Double tapered and pitched cambered beams	kdis, DIN EN NA: 1.300 🚔
Pitched cambered beams	kdis: 1.700 🐳	Curved beams	kdis, DIN EN NA: 1.150
			OK Cancel

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.

Parameter Description I : Support length I _{i,j} , I _{i,r} : Increase of contact length (I _{i,j} + 1 + I _{i,r}) Ief : Effective length (I _{i,j} + 1 + I _{i,r}) F : Local force I1 : Distance F from support a : Cantilever length A _{ef} = lef * b Immin 30 mm, I, 1/2)	Info Automatic calculation of $k_{0,00}$ and A_{ef} (with distance conditions acc. to 6.1.5) is valid only for a model with a local force. For other cases (line load), the parameters $k_{0,00}$ and A_{ef} will be increased to the maximum.	
Distance Conditions l_{ef} : Effective length $(l_{i,i} + l + l_{i,r})$ F : Local force $l_1 >= 2h$: $k_{0,90}$ Depends on national annex and type of timber $l_1 < 2h$: $k_{0,90} = 1$ $l_1 < 2h$: $k_{0,90} = 1$ $l_{i,j} = min(30 mm, l, a)$ $l_{i,r} = min(30 mm, l, 1/2)$	I : Support length	
	Ief : Effective length (l _{i,l} + l + l _{i,r}) F : Local force I1 : Distance F from support a : Cantilever length	l 1 >= 2h : k _{0.90} Depends on national annex and type of timber l 1 < 2h : k _{0.90} = 1 l _i ,j = min(30 mm, l, a)

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

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.

ate Limit State Desig	 1.1 General Data 		Supplementary Examination
al Data s Categories 1 Categories 2			Reduce possible result combinations by examining RFEM results From automatically defined
	Combination Rules Acc	to Standard / Anney	Combinations
	EN 1990 + EN 199		From result combinations:
	Generating for Design		Settings for Combinations Acc. to EN 1990
	Static equilibrium:	Basic combination E Accidental E. Accidental Snow E Seismic E	Ultimate limit state basic combination Apply combination rule: © Equation 6.10 © Equations 6.10a
	☑ Ultimate limit state:	Basic combination U Accidental U Accidental - Snow U Seismic U	Accidental compliance according to Equations 6.11a and 6.11b Apply combination @ \u03c8 \u03c8 \u03c8 1.1 coefficient: \u03c8 \u03c8 \u03c8 \u03c8 1.1
	Serviceability limit state:	Characteristic S Frequent S Quasi-permanent S	Setting for Timber Structures
	Generate suppleme		Deformation coefficient kdef : 0.80
	Do not treat togethe as collateral actions	r snow and wind	Numbering Starting number for generated - Result combination:

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.

Calculation

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.

General Data Actions Apply Stuation LDC LC1 LC21 LC41 LC42 LC51 LC54																		
-General Data -Action Categories 1 -Action Categories 2 -Action Categories 2 	1 - Ultimate Limit State Desig 💌	2.2 Resu	lt Comb	inations														
RC Apply Stuation LDC LC1 LC21 LC41 LC42 LC43 LC51 LC52 LC53 LC54 LC55 LC55 LC56	out Data		A		С	D	E	F	G	H		J	K	L	M	N	0	<u> </u>
Action Categories 1 Action Categories 2 Action Categories 2 RC1 Q UB Permanent 1.35 - <td>General Data</td> <td></td>	General Data																	
RC2 U U B Short+em 1.35 1.50 <td< td=""><td>Actions</td><td></td><td></td><td></td><td></td><td></td><td>LC21</td><td>LC41</td><td>LC42</td><td>LC43</td><td>LC51</td><td>LC52</td><td>LC53</td><td>LC54</td><td>LC55</td><td>LC56</td><td>LC57</td><td>LC</td></td<>	Actions						LC21	LC41	LC42	LC43	LC51	LC52	LC53	LC54	LC55	LC56	LC57	LC
Result RC3 U U Short4em 1.35 1.50 0.75 - </td <td>Action Categories 1</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>-</td> <td>-</td> <td>-</td> <td>-</td> <td>1.0</td> <td>1.1</td> <td>•</td> <td>-</td> <td>-</td> <td></td> <td></td>	Action Categories 1							-	-	-	-	1.0	1.1	•	-	-		
Result Combinations by Actions Red U B Short term 1.35 1.50 0 0.75 -	Action Categories 2								-	-	-	1.0	1.1	•	-	-		
RC5 U UB Short+em 1.35 1.50 - 0.75 -	sults											1.0		•	-	-		
RC6 U UB Short+em/l 1.35 1.50 0.75 - - 0.90 - 0.90 - - - - 0.90 - - <t< td=""><td>Result Combinations by Actions</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td>-</td><td></td><td></td><td>•</td><td>-</td><td>-</td><td>-</td><td></td></t<>	Result Combinations by Actions										-			•	-	-	-	
RC7 ☑ UB Short+em/l 1.35 1.50 0.75 - - 0.90 - 0 - - - - 0 0 0 - - - 0 0 0 - - 0 0 0 0 0 0 0 0 0 0 0 0 0	 Result Combinations 			S					-			1.0	- ÷ -	•	-		-	
RC3 Ø UB Short+em/l 1.35 1.50 0.75 - - - 0.90 - 0.90 - - - 0.90 - - - 0.90 - - - 0.90 - - - 0.90 - - - 0.90 - - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90 - - 0.90	Result Combinations - Reducec									-	0.90		-	•	-	-	-	
RC9 Image: Construction of the construct									-	-	-	0.90		-	-	-	-	
RC10 ☑ UB Short⊀erm/l 1.35 1.50 0.75 - - - - 0.90 - - RC11 ☑ UB Short⊀erm/l 1.35 1.50 0.75 - - - - 0.90 - - 1.10									-	-	-	-	0.90		-	-	-	
RC11 ☑ UB Short+emr/l 1.35 1.50 0.75 - - - - 0.90 0.90 0.90 Load Cases in Generated Result Combination RC5 Utimate Limit State - Basic Con LC Load Case Description Action Category Leading KFi 7 y Cesi KFi.7 LC1 Bgengewicht + Dachaufbau Action Category Leading KFi 7 y Cesi KFi.7 LC2 Imposed Load Action 2.100 1.50 - 1											-			0.90			-	
Image: set of the set									-	-	-			•	0.90	_	-	
Load Cases in Generated Result Combination RC5 Utimate Limit State - Basic Control LC Load Case Description Action Category Leading KFI 7 v Cesi KFI 7 LC1 Bgengewicht + Dachaufbau AC1 1.A 1.00 1.35 - - 1 LC21 Imposed Load AC2 3.H V 1.00 1.50 - 1		RC11		UB	Short-term/I	1.35	1.50	0.75	-	-	-	1.0	1.1	•	-	0.90		
LC Load Case Description Action Category Leading KFI γ ψ Cesi KFI. γ LC1 Egengewicht + Dachaufbau AC1 1.A □ 1.00 1.35 - - 1 LC21 Imposed Load AC2 3.H ☑ 1.00 1.50 - - 1																		_
LC1 Egengewicht + Dachaufbau AC1 1.A I.00 1.35 - - 1 LC21 Imposed Load AC2 3.H II 1.00 1.50 - - 1			ases in C	Generated Re	sult Combinatio	n RC5							Ultir	mate Lir	nit State	e - Basi	c Combi	nation
LC21 Imposed Load AC2 3.H V 1.00 1.50 1		LC		Load C	ase Description	n I		Actio	n Ca	ategory	Leadir	ng K	FI	γ	ψ (C _{esi} Ki	FL·γ·Ψ	. Ces
		LC1	Eigen	gewicht + Dao	chaufbau			AC1		1.A		1	.00 1	.35	-	-	1.35	
LC43 Snow (Right Side Full) AC3 4.B 1.00 1.50 0.50 - 0		LC21	Impos	ed Load				AC2	2	3.H	1	1	.00 1	.50	-		1.50	
		LC43	Snow	(Right Side Fi	ull)			AC3	}	4.B		1	.00 1	.50 0).50	-	0.75	

Dlubal

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.

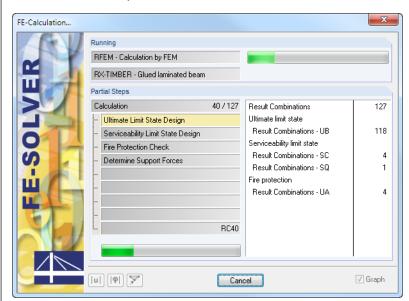


Figure 6.8: Window FE-Calculation

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

Calculation



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.



OK

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.

Beam Type and Material Geometry Loads Control Parameters	RC	A	В	С	1 .		
- Loads Control Parameters	PC				D	E	F
- Control Parameters				Design		Factor	Max.
	nc.	RC Description	Load Cases	Situation	LDC	k mod	Design
		Ultimate Limit State Desi	ign				
Results	RC1	g	1.35*LC1	UB	Permane	0.600	0.92
— Result Combinations	RC2	g+p	1.35*LC1 + 1.50*LC21	UB	Short-ter	0.900	0.6
- Design - All	RC3	g+p+s	1.35*LC1 + 1.50*LC21 + 0.75*LC41	UB	Short-ter	0.900	0.80
- Design by Location X	RC4	g + p + s(l)	1.35*LC1 + 1.50*LC21 + 0.75*LC42	UB	Short-ter	0.900	0.76
- Support Forces	RC5	g + p + s(r)	1.35*LC1 + 1.50*LC21 + 0.75*LC43	UB	Short-ter	0.900	0.76
- Deformations	RC6	g + p + s + w(g,I,AA)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC51	UB	Short-ter	1.000	0.72
1	RC7	g + p + s + w(g,I,BB)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC52	UB	Short-ter	1.000	0.7
1	RC8	g + p + s + w(g,I,AB)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC53	UB	Short-ter	1.000	0.7
le la	RC9	g + p + s + w(g,I,BA)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC54	UB	Short-ter	1.000	0.7
le la	RC10	g + p + s + w(g.r.AA)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC55	UB	Short-ter	1.000	0.7
	RC11	a + p + s + w(a.l.BB)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC56	UB	Short-ter	1.000	0.7
l l l l l l l l l l l l l l l l l l l	RC12	a + p + s + w(a,r,AB)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC57	UB	Short-ter	1.000	0.7
	RC13	a + p + s + w(a,I,BA)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC58	UB	Short-ter	1.000	0.7
-	RC14	g + p + s + w(p,A)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC59	UB	Short-ter	1.000	0.72
-	RC15	g + p + s + w(p,B)	1.35*LC1 + 1.50*LC21 + 0.75*LC41 + 0.90*LC60	UB	Short-ter	1.000	0.7
-	RC16	q + p + s(l) + w(q,l,AA)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC51	UB	Short-ter	1.000	0.6
-	RC17	g + p + s(l) + w(g,l,BB)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC52	UB	Short-ter	1.000	0.6
-	RC18	g + p + s(l) + w(g, l, AB)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC53	UB	Short-ter	1.000	0.6
-	RC19	g + p + s(l) + w(q,l,BA)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC54	UB	Short-ter	1.000	0.6
	RC20	g + p + s(l) + w(q,r,AA)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC55	UB	Short-ter	1.000	0.6
	RC21	q + p + s(l) + w(q,l,BB)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC56	UB	Short-ter	1.000	0.6
	RC22	q + p + s(l) + w(q,r,AB)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC57	UB	Short-ter	1.000	0.6
-	RC23	q + p + s(l) + w(q,l,BA)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC58	UB	Short-ter	1.000	0.6
-	RC24	g + p + s(l) + w(p,A)	1.35*LC1 + 1.50*LC21 + 0.75*LC42 + 0.90*LC59	UB	Short-ter	1.000	0.6
L		13.5.00.000	Max: 0.99 ≤1			2 3	

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

Ę.



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.

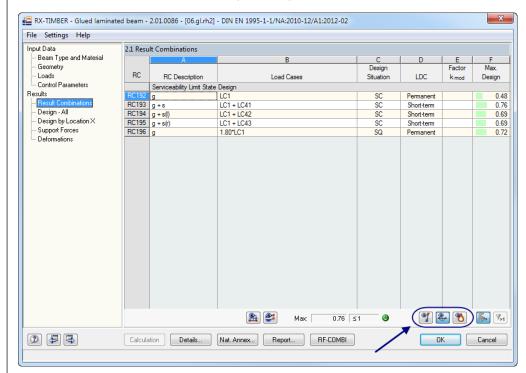


Figure 7.2: Window 2.1 Result Combinations with filter set only for serviceability limit state design

The buttons have the following functions:

Button	Function
Y	Hides and displays the ultimate limit state design
2	Hides and displays the serviceability limit state design
8	Hides and displays the fire protection design

Table 7.2: Buttons for designs

7_{>1}

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.

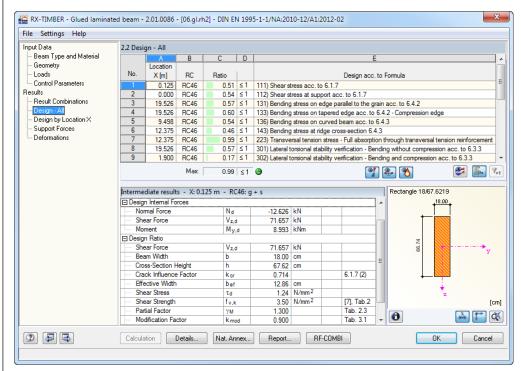


Figure 7.3: Window 2.2 Design - All

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

Max:

0.95 ≤1 🥹

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).

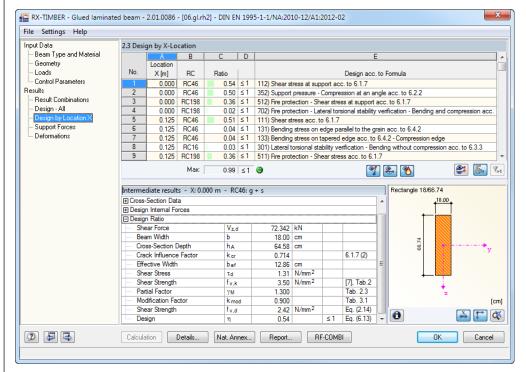


Figure 7.4: Window 2.3 Design by X-Location

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

5



ି 😫 🦄

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.

nput Data	2.4 Supp	ort Forces							
 Beam Type and Material 		A	B	C	D	E	F	G	
Geometry	LC	LC/	Left Sup	oport	Right S	upport	Equiv. Load	Max.Moment	
Loads	RC	RC Description	Ax [kN]	Az [kN]	Bx [kN]	Bz [kN]	q [kN/m]	My [kNm]	
Control Parameters	RC187	g + s(r) + w(q.I,BB)	0.000	53.941	0.000	57.561	0.204	353.148	
esults		g + s(r) + w(q.r,AB)	0.000	53.941	0.000	57.561	0.204	353.148	
— Result Combinations	RC189	g + s(r) + w(q,I,BA)	0.000	53.941	0.000	57.561	0.204	353,148	
Design - All	RC190	q + s(r) + w(p,A)	0.000	53.941	0.000	57,561	0.204	353,148	
 Design by Location X 	RC191	g + s(r) + w(p,B)	0.000	53.941	0.000	57.561	0.204	353.148	
- Support Forces	Max		0.000	73.849	0.000	73.849	0.268	464.822	_
Deformations	Min		0.000	44.892	0.000	44.892	0.165	285.647	
		Result Combinations for Services	ability Limit State (Characteristic V	(alues)				
	RC192	g	0.000	33.253	0.000	33.253	0.122	211.590	
	RC193		0.000	52.558	0.000	52.558	0.191	331.040	
	RC194	g + s()	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	q	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 Res	istance (Design V	alues)					
	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()	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	
	Tilt. mom pinned s	ent for upport Td: 5.810 [kNm	Axial force in] compr. edge			iquiv.load or1 beam q:	0.268	[kN/m]	2

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.

Loads RC Description uz ux X [m] max uz [mm] uz ux \psi_A Load Cases Results Load Cases 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Load Cases Support Forces 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 Design +All Design by Location X Snow (Left Side Full) 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 Support Forces Deformations Result Combinations - </th <th>A B C D E F G H I cometry pads ontrol Parameters termistions C LC/RC Supp. Left [mm] Max. Span Deflect. Supp. Night [mm] Rotations [mads on trol Parameters ts Load Cases Load Cases U ux V/r.A (v).1 (v).2 (u).2 (u).2</th> <th>ile Settings Help</th> <th></th>	A B C D E F G H I cometry pads ontrol Parameters termistions C LC/RC Supp. Left [mm] Max. Span Deflect. Supp. Night [mm] Rotations [mads on trol Parameters ts Load Cases Load Cases U ux V/r.A (v).1 (v).2 (u).2	ile Settings Help										
Geometry Loads LC LC/RC Supp. Left [mm] U2 Max. Span Deflect. Supp. Right [mm] U2 Notation U2 Control Parameters Results Load Cases V/A V/A V/A Design. All Design by Location X Suppot Forces Load Cases 0.0 0.0 12.375 24.7 0.0 20.6 -7.7.7 Load Cases 1.041 Snow (Roth Sides Full) 0.0 0.0 12.375 22.7 0.0 10.5 4.0 Load Cases Snow (Right Side Full) 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 Load Cases Snow (Right Side Full) 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 Load Cases RC5 \$ 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 Load Cases Sonw (Right Side Full) 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 RC5 \$ 0.0 0.0 13.462 17.1 0.0	LC LC/RC Supp. Left [mm] Max. Span Deflect. Supp. Right [mm] Rations [mad opt. A pads Description uz		2.5 Defo	rmations								
Loads Control Parameters Results Control Parameters Result Combinations X [m] max u Z [mm] U ux X [m] max u Z [mm] U ux V yA Pesults Load Cases 0.0 0.0 12.375 44.7 0.0 20.6 -7.7. Pesults Sow (Both Sides Full) 0.0 0.0 12.375 22.7 0.0 10.5 4.0 Design by Location X Support Forces Sow (Right Side Full) 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 LC43 Snow (Right Side Full) 0.0 0.0 11.288 17.1 0.0 7.9 -2.8 Support Forces Petromstions RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s(f) 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s(f) 0.0 0.0 12.375 22.7 0.0 10.5 -4.0	BC Description uz ux X (m) Description uz vx qm, m qm											
Control Parameters Load Cases Results Load Cases Result Combinations 0.0 0.0 12.375 44.7 0.0 20.6 7.7. Result Combinations Design by Location X Sign by Location X Sign by Location X 1.01 Eigengewicht + Dachaufbau 0.0 0.0 12.375 44.7 0.0 7.9 -3.2 Support Fromations Sign by Location X Sign by Location	Load Cases Load Cases Is Load Cases User Strategy 0.0 0.0 12.375 24.7 0.0 20.6 97.8 97.8 Is Load Cases Is Is 10.0 0.0 0.0 12.375 24.7 0.0 20.6 -7.7 Is Is can cases Is Snow (Both Sides Full) 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 LC41 Snow (Both Sides Full) 0.0 0.0 11.288 17.1 0.0 7.9 -2.8 LC42 Snow (Right Side Full) 0.0 0.0 11.288 17.1 0.0 7.9 -2.8 Proces RC5 s 0.0 0.0 11.288 17.1 0.0 7.9 -2.8 RC6 s() 0.0 0.0 11.288 17.1 0.0 7.9 -2.8 RC7 s() 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1		LC									•
Results Load Cases Pesign 24 Combinations 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Design -All Design by Location X Snow (Reft Side Full) 0.0 0.0 12.375 22.7 0.0 10.5 4.4.0 Design by Location X Support Forces Support Forces 0.0 0.0 12.375 22.7 0.0 10.5 4.4.0 RC5 s 0.0 0.0 12.482 17.1 0.0 7.9 -2.8 RC6 s(1) 0.0 0.0 12.462 7.7 0.0 10.5 4.4.0 RC6 s(1) 0.0 0.0 11.482 17.1 0.0 7.9 -2.8 RC6 s(1) 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s(1) 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC7 g s(2) s(1) 0.0 12.375 <td>Lood Cases Lood Cases Lood Cases CIC Egergwicht + Dachaufbau 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 desit Combinations esuit Combinations cign - All CIC Egergwicht + Dachaufbau 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 LC43 Snow (Left Side Full) 0.0 0.0 7.9 -2.8 Result Combinations RC5 s 0.0 10.5 </td> <td></td> <td>RC</td> <td></td> <td>uΖ</td> <td>uχ</td> <td>X [m]</td> <td>max u z [mm]</td> <td>uΖ</td> <td>uχ</td> <td>φy.A</td> <td>Фу.В</td>	Lood Cases Lood Cases Lood Cases CIC Egergwicht + Dachaufbau 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 desit Combinations esuit Combinations cign - All CIC Egergwicht + Dachaufbau 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 LC43 Snow (Left Side Full) 0.0 0.0 7.9 -2.8 Result Combinations RC5 s 0.0 10.5		RC		uΖ	uχ	X [m]	max u z [mm]	uΖ	uχ	φy.A	Фу.В
Result Combinations Oot Control Degramment in Destinational Oot Oot Case Control Control Degramment in Destinational Oot Oot Case Control Degramment in Destinational Oot Oot Case Case Control Destinational Oot Oot Case Case <thcase< th=""> Case <thcase< th=""> Case Case<td>Result Combinations Control Contro Control Control</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></thcase<></thcase<>	Result Combinations Control Contro Control Control											
Design - All Design - All Dot of the second procese (and procese bill) Dot of the second procese (and procese bi	esign - All Low Show (bour source of the full) 0.0 0.0 12.57 22.7 0.0 10.5 4.0 esign by Location X upport Forces LC42 Snow (Right Side Full) 0.0 0.0 11.257 22.7 0.0 10.5 4.0 esign by Location X upport Forces IC43 Snow (Right Side Full) 0.0 0.0 11.288 17.1 0.0 7.9 -2.8 Result Combinations			Eigengewicht + Dachaufbau	0.0	0.0		44.7	0.0			
Design by Location X Support Forces Design by Location X Support Forces Design by Location X (Ref side Full) 0.0 0.0 0.1 0.1 0.0 7.3 0.2 Result Combinations 0 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 Result Combinations 0 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC5 s 0.0 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s(r) 0.0 0.0 11.287 22.7 0.0 10.5 -4.0 RC6 s(t) 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s(r) 0.0 0.0 11.288 0.0 16.5 6.2 RC8 g 0.0 0.0 12.375 58.4 0.0 24.3 9.4 RC11 g+s(r) 0.0 0.0 12.375 80.5 0.0 37.1 1	esign by Location X upport Forces eformations estimation			Snow (Both Sides Full)	0.0	0.0		22.7	0.0			
Support Forces Result Combinations 0.0 0.0 12.375 22.7 0.0 10.5 4.0 RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s() 0.0 0.0 12.375 35.8 0.0 7.9 -2.2 RC7 s(r) 0.0 0.0 12.375 35.8 0.0 7.9 -2.2 RC8 g 0.0 0.0 12.375 35.8 0.0 26.9 -10.1 RC10 g + s() 0.0 0.0 12.375 58.4 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 12.375 80.5 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.375 80.5 0.0 24.3 -9.4 RC13 g 0.0 0.0 </td <td>Loss Diversion <thdiversion< th=""> <thdiversion< th=""> <thdivers< td=""><td></td><td>LC42</td><td>Snow (Left Side Full)</td><td>0.0</td><td>0.0</td><td>11.288</td><td>17.1</td><td>0.0</td><td>7.9</td><td>-3.2</td><td></td></thdivers<></thdiversion<></thdiversion<></td>	Loss Diversion Diversion <thdiversion< th=""> <thdiversion< th=""> <thdivers< td=""><td></td><td>LC42</td><td>Snow (Left Side Full)</td><td>0.0</td><td>0.0</td><td>11.288</td><td>17.1</td><td>0.0</td><td>7.9</td><td>-3.2</td><td></td></thdivers<></thdiversion<></thdiversion<>		LC42	Snow (Left Side Full)	0.0	0.0	11.288	17.1	0.0	7.9	-3.2	
Result Combinations RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC5 s() 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s() 0.0 0.0 13.462 17.1 0.0 7.9 -3.2 RC8 g 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC9 g.s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 RC10 g.s() 0.0 0.0 12.2375 58.4 0.0 24.3 -9.4 RC11 g.s() 0.0 0.0 12.2375 58.0 0.0 27.1 -13.4 RC12 g 0.0 0.0 12.2375 80.5 0.0 27.1 -13.4 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 RC13 g	Result Combinations RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s() 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s() 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s(r) 0.0 0.0 13.462 17.1 0.0 7.9 -3.2 RC8 g 0.0 0.0 12.375 58.8 0.0 16.5 -6.2 RC9 g + s() 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1 RC11 g + s(r) 0.0 0.0 12.375 88.4 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1 <td></td> <td>LC43</td> <td>Snow (Right Side Full)</td> <td>0.0</td> <td>0.0</td> <td>13.462</td> <td>17.1</td> <td>0.0</td> <td>7.9</td> <td>-2.8</td> <td></td>		LC43	Snow (Right Side Full)	0.0	0.0	13.462	17.1	0.0	7.9	-2.8	
RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 -4.0 RC6 s() 0.0 0.0 0.12.375 22.7 0.0 10.5 -4.0 RC7 s() 0.0 0.0 0.12.375 22.7 0.0 10.5 -4.0 RC7 s() 0.0 0.0 0.12.375 35.8 0.0 7.9 -3.2 RC7 s() 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC3 g + s() 0.0 0.0 12.375 56.4 0.0 26.9 -10.1 RC11 g + s() 0.0 0.0 12.175 80.4 0.0 24.3 -8.9 RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 Max/Min Deformations Max 0.0 0.0	RC5 s 0.0 0.0 12.375 22.7 0.0 10.5 4.0 RC6 s() 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s() 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s() 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC8 g 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC10 g + s() 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1 RC11 g + s(r) 0.0 0.0 12.375 52.8 0.0 24.3 -8.9 RC12 g 0.0 0.0 12.375 60.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1											
RC6 s() 0.0 0.0 11.288 17.1 0.0 7.9 -3.2 RC7 s() 0.0 0.0 0.0 13.462 17.1 0.0 7.9 -3.2 RC8 g 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 RC8 g 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC9 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 RC10 g + s() 0.0 0.0 12.375 58.4 0.0 24.3 -34. RC11 g + s() 0.0 0.0 12.375 80.5 0.0 27.1 -13.3 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 Max/Min Deformations 0.0 0.0 12.375 80.5 0.0 37.1 -2.6	RC6 s() 0.0 0.0 11.288 17.1 0.0 7.9 3.2 RC7 s(r) 0.0 0.0 0.0 13.462 17.1 0.0 7.9 3.2 RC8 g 0.0 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 RC8 g 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC9 g + s(r) 0.0 0.0 12.375 58.4 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 12.375 80.5 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -1.3.9 1 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations U U U U U U U U U U U <thu< td=""><td> Deformations</td><td></td><td>Result Combinations</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></thu<>	Deformations		Result Combinations								
RC7 stp 0.0 0.0 13.462 17.1 0.0 7.9 -2.6 RC8 g 0.0 0.0 12.375 33.8 0.0 16.5 -5.2 RC9 g + s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 RC10 g + stp 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC11 g + stp 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.375 80.5 0.0 24.3 -9.4 RC13 g 0.0 0.0 12.375 80.5 0.0 24.3 -9.4 RC13 g 0.0 0.0 12.375 80.5 0.0 27.1 -13.9 Max/Min Deformations	RC7 s(r) 0.0 0.0 13.462 17.1 0.0 7.9 -2.8 RC8 g 0.0 0.0 10.1 12.375 35.8 0.0 16.5 6.2 RC9 g + s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1 RC10 g + s(r) 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 12.739 52.8 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations U U 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1		RC5	s	0.0	0.0	12.375	22.7	0.0	10.5	-4.0	
RC8 gr 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC9 g+s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 RC10 g+s() 0.0 0.0 12.375 58.4 0.0 24.3 -9.4 RC11 g+s() 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 Max/Min Deformations	RC8 g 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC9 g + s 0.0 0.0 12.375 35.8 0.0 16.5 -6.2 RC10 g + s() 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1 RC11 g + s() 0.0 0.0 12.375 58.4 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 12.375 52.8 0.0 24.3 -9.9 RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations Max/Min Deformations U <thu< th=""> <</thu<>		RC6	s()	0.0	0.0	11.288	17.1	0.0	7.9	-3.2	
RC9 g + s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 RC10 g + s(r) 0.0 0.0 12.011 52.8 0.0 24.3 -34. RC11 g + s(r) 0.0 0.0 12.175 58.4 0.0 24.3 -34. RC12 g 0.0 0.0 12.375 80.5 0.0 24.3 -34. RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations U U U U U U U U U U U U U U U	RC9 g + s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1 RC10 g + s() 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.375 80.5 0.0 24.3 -8.9 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations		RC7	s(r)	0.0	0.0	13.462	17.1	0.0	7.9	-2.8	
RC9 g + s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 RC10 g + s(r) 0.0 0.0 10.01 12.375 58.4 0.0 26.9 -10.1 RC11 g + s(r) 0.0 0.0 12.011 52.8 0.0 24.3 -3.4 RC12 g 0.0 0.0 12.375 80.5 0.0 23.4 -3.4 RC13 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 Max/Min Deformations U Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.6	FC9 g + s 0.0 0.0 12.375 58.4 0.0 26.9 -10.1 1 RC10 g + s() 0.0 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 0.0 12.079 52.8 0.0 24.3 -9.4 RC12 g 0.0 0.0 0.0 12.375 80.5 0.0 24.3 -9.4 RC13 g 0.0 0.0 0.0 12.375 80.5 0.0 23.71 -1.3.9 1 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations		RC8	q	0.0	0.0	12.375	35.8	0.0	16.5	-6.2	
RC10 g + s() 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC11 g + s(r) 0.0 0.0 12.739 52.8 0.0 24.3 -9.4 RC12 g 0.0 0.0 12.735 80.5 0.0 24.3 -9.4 RC13 g 0.0 0.0 12.375 80.5 0.0 23.1 -13.8 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations 0.0 0.0 12.375 80.5 0.0 37.1 -2.6	RC10 g + s() 0.0 0.0 12.011 52.8 0.0 24.3 -9.4 RC11 g + s() 0.0 0.0 12.739 52.8 0.0 24.3 -8.9 RC12 g 0.0 0.0 12.739 52.8 0.0 24.3 -8.9 RC12 g 0.0 0.0 12.757 80.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations		RC9		0.0	0.0	12.375	58.4	0.0	26.9	-10.1	1
RC11 g + s(r) 0.0 0.0 12.739 52.8 0.0 24.3 -8.5 RC12 g 0.0 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations 0.0 0.0 12.375 80.5 0.0 37.1 -2.6	RC11 g + s(r) 0.0 0.0 12.739 52.8 0.0 24.3 -8.9 RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations Max/Min Deformations 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1		RC10		0.0	0.0	12.011	52.8	0.0	24.3	-9.4	
RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.5 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations	RC12 g 0.0 0.0 12.375 80.5 0.0 37.1 -13.9 1 RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations V		RC11		0.0	0.0	12,739	52.8	0.0	24.3	-8.9	
RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations 0.0 0.0 12.375 80.5 0.0 37.1 -2.6	RC13 g 0.0 0.0 12.375 44.7 0.0 20.6 -7.7 Max/Min Deformations Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1		RC12		0.0	0.0	12.375	80.5	0.0	37.1	-13.9	1
Max/Min Deformations Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.6	Max/Min Deformations 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1		RC13	-	0.0	0.0	12 375	44 7	0.0	20.6	-77	
Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.8	Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1											
Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.8	Max 0.0 0.0 12.375 80.5 0.0 37.1 -2.8 1			Max/Min Deformations								
			Max		0.0	0.0	12.375	80.5	0.0	37.1	-2.8	1
Min 00 00 11288 171 00 79 -139												
					0.0	0.0		17.1	0.0	7.5	10.0	
												-
												1
Oscillation Check acc.			to 9.3 (1) W G,inst + ¥2 * W Q,inst :	44.7	> 6.0	[mm]					_
Oscillation Check acc. to 9.3 (1) w g.inst + ψ2 * w g.inst : 44.7 >6.0 [mm]												
		2 5 3	Calcula	tion Details Standard	Report.		F-COMBI	r	1	OK		ancel

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.

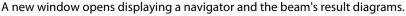
7 Results

2



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.



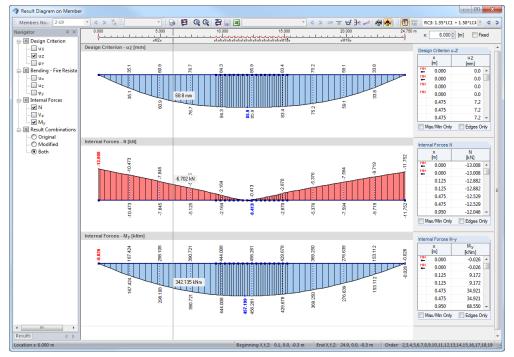


Figure 7.7: Dialog box Result Diagram on Member

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.

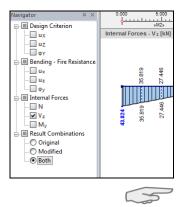
A dotted line in the result diagram of the RX-TIMBER design indicates the design ratio of 1.0 (see Figure 8.12, page 75). In this way, you can evaluate the significance of the results quickly.

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



b

Report...



8. Printout

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

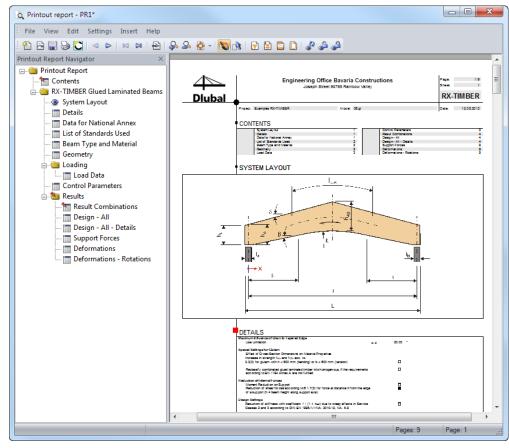


Figure 8.1: Printout report

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

select Rename on the File menu of the printout report.

Rename Printout Repo	ort X
Old Description	
All data	
New Description	
Reduced data	
2	OK Cancel

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 $[\hat{u}]$ keys.

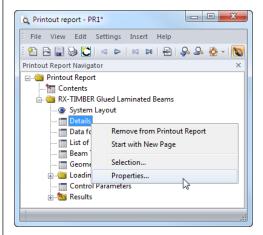


Figure 8.3: Context menu of report navigator

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.

roperties	×
Title	
Geometry	
Additional Explanation	
🔽 Display	
Here you can enter any text. The text will appear in the left report margin. Line breaks can be inserted with [Enter].	~
Options	
Options Image: Construction of the second	

Figure 8.4: Dialog box Properties



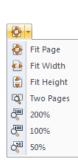
Navigation in the printout report

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
R	Go to first page in the page preview
×	Go to last page
-	Specify the number of a particular page in a dialog box.
S	Zoom in
9	Zoom out
	List button for <i>Zoom</i> to adjust display size
	Grab mode: Use the mouse for navigation within the report.
3	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:

Printout Report Selection - PR1	-X
Program / Modules FRCTIMBER Display Cover sheet Ø Contents Ø Info pictures	Global Selection Input Data Results Display Data of Module I I. Input Data I 2. Results
	OK Cancel

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.

8 Printout

Printout Report Selection - PR1	×
Printout Report Selection - PR1 Program / Modules RX-TIMBER	Global Selection Input Data Results Display System Layout Graphical Representation Graphical Rep
	 ✓ 1.2 Geometry ✓ 1.3 Loads ✓ Load Data Generated Loads ✓ Additional Loads ✓ 1.4 Control Parameters
Display Cover sheet Contents Info pictures	Global Selection
۷	OK Cancel

Dlubal

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

3

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.

Program / Modules	Global Selection Input Data Results	1	
RX-TIMBER	Display		
	2.1 Result Combinations	Filter:	
	 2.1 Hesuk Combinations 2.2 Design - All - Summary 	Filter:	
	2.2 Design - All - Details	Serviceability Limit State	
	2.3 Design by Location x - Summary	📝 Fire Resistance	
	2.3 Design by Location x - Details		
	2.4 Support Forces		
	🥅 Min/max only		
	2.5 Deformations		
	🥅 Min/max only		
Display			
Cover sheet			
Contents			
Info pictures	Global Selection		
D			OK Cancel

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.

File View Edit Settings Insert Help	
🖀 🖬 🍃 💟 🗠 🕨 🛛 🗲	a 🎝 🕹 🎸 - [📉 🔥 🗹 🖬 🖬 🔄 🔄 🖉 🖓 🖓 🦂
Printout Report Navigator >	Header Setup Engineering Office Bavaria Constructions
e following dialog box appea	ars:
Header Default	Date
2 - Bavaria Constructions	Display date Last modification: 12.08.2013
	Teday 10.00.0010
☑ Header☑ Info row below header	Today: 12.08.2013 User-defined: 12.08.2013
 ✓ Info row below header ✓ Footer 	 O User-defined: Page and Sheet Numbering First number ✓ Page: 1
 Info row below header Footer Black circles in header and footer 	 ○ User-defined: Page and Sheet Numbering First number ✓ Page: 1 ✓ Sheet: 1
Info row below header Footer Black circles in header and footer	 ○ User-defined: Page and Sheet Numbering First number ✓ Page: 1 ✓ Sheet: 1 ✓ Alignment
Info row below header Footer Black circles in header and footer Company Address Engineering Office Bavaria Constructions	© User-defined: Page and Sheet Numbering First number ☑ Page: 1 ☑ Sheet: 1 Alignment ⓐ ⓐ ⓐ ⓐ ⓐ ⓐ ⓐ ⓐ ⓑ ⓑ ⓑ ⓑ ⓑ ⓑ ⓑ ⓑ ⓑ ⓑ

Figure 8.9: Dialog box Printout Report Header

Change Project Name/Description

Examples RX-TIMBER

(2)

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

ΟK

Cancel

Change Model Name/Description

06.gl

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.

8 Printout



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.

ge and Sheet Numbering						
ontents						Page Numbering
Chapter Name	Prefix	Page	Prefix	Sheet	Result	Page numbering
Contents		1		1	1/9	
Data for National Annex		2		1	2/9	Prefix
Geometry		3		1	3/9	Page number
lesult Combinations		4		1	4/9	End number
lesign - All - Details		5		1	5/9	
esign - All - Details		6		1	6/9	Automatically increasing
esign - All - Details		7		1	7/9	
esign - All - Details		8		1	8/9	
upport Forces		9		1	9/9	First number: 1
						End number: 9
						Sheet Numbering
						Prefix
						Sheet numbering
						Automatically increasing
						First number: 1
						End number:
2						OK Cancel

Figure 8.10: Dialog box Page and Sheet Numbering



0

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.

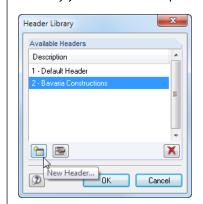


Figure 8.11: Dialog box Header Library

2



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.

Result Diagrams 8.4

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.

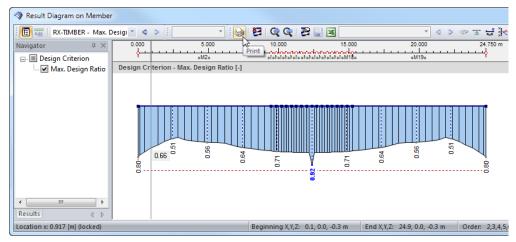


Figure 8.12: Button Print in the toolbar of the window Result Diagram on Member

The following dialog box appears:

Graphic Printout				
General Options				
Graphic Picture	Window To Print	Graphic Size		
Directly to a printer	Current only	As screen view		
To a printout report: PR1	O More	Window filling		
 To the Clipboard 	O Mass print	○ To scale 1: 100 ▼		
Graphic Picture Size and Rotation	Options			
☑ Use whole page width	Show results for selected x diagram	docation in result		
── Use whole page height	Lock graphic picture (with	out update)		
● Height: 68 ⇒ [% of page]				
	Show printout report on [O	ĸj		
Rotation: 0 😴 [*]				
Header of Graphic Picture Result diagrams in Member - M2,M3,M4,M5,M6,M7,M8,M9,M10,M11,M12,M13,M14,M15,M16,M17,M18,M19				
Tresult diagrams in member - mz,mo,m4,mo,mo,m/,mo,m5,m10,m11,m12,m13,m14,m13,m10,m17,m10,m15				
		OK Cancel		

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:

0	Refreshes all graphics
<mark>-}</mark>	Unlocks all graphics which can then be updated dynamically
3	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.

Insert image fro	m the Clipboard
Title	
Enter the chapt	ter name.
D	OK Cancel

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.

mport Text	×
Title Title of chapter	
Text	
Here you can enter any text. Line breaks can be inserted with [Enter].	
Dver all page width	OK Cancel

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 doubleclick. 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.

Title	
RFEM_customer-project WOOD CARVING	
Text	
Wood Carving Workshop An extraordinary project was born when the architectural office bergmeister-wolf designed a new workshop f wood carving at in Val Gardena, South Tyrol. As the triangulated facade could be used only patially for load application, the building represented a great challenge for the responsible timber construction engineer. But the Italian Dlubal customer Schrentewein & Partner mastered the task by primarily using the building's inside parts for transferring loads.	or
Construction The facade of the building brings to mind carvings in a block of wood, which can be appreciated from both outside the structure and also from the inside. The rooms for exhibition and sales are in the first floor. The workshop for the sculptors and painters (painting wooden figures) is located on the upper floor. In addition, a large apartment was set up in the attic. Another difficulty for the structural engineers was the fact that the first floor includes only a few walls, and on few walls in the building stand upon each other. Schrentewein & Partner found the following solution: They designed almost every wall in the upper floor as a wall-like deep beam consisting of cross laminated timber. I longest wall has a length of 13.7 m. The walls in the upper floor carry the loading from the floor and ceiling of floor and transfer the loading to the walls and columns made of steel round pipes on the first floor. The ceilings like the load-bearing interior walls on store to the struct. The ceilings like the load-bearing interior walls on the first floor. The ceilings like the load-bearing interior walls on the transfer the structure the structure and the structure and the structure and the structure walls and columns the test. The ceilings like the load-bearing interior walls on the structure the structure and the structu	lya he that
Dver all page width	ancel

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.

New Printo	ut Report Template
No.	Description New template
Ø	OK Cancel

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.

No. Description 1 All Chapters 2 New template	ole Printout Report Templates
2 New template	Description
	All Chapters
	New template
(La)	
(La 1997)	
🕹 🛛 🔪	
(b)	
AB 🖉	
	SB 7

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.

A B	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.

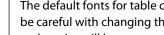
To open the dialog box where you can edit the page layout,

select Page on the Settings menu of the printout report or

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

Page Setup	×
Page Margins	Table Fonts
Left: 16 🚔 [mm) Normal script
Right: 10 🚔 (mm	1] 🔝 Column titles
Top: 10 🚔 (mm	1] Table titles
Bottom: 10 🚔 [mm	
	Display settings
Shading	
	Gray tone
Table titles:	230 🛬
Odd table rows:	230 🛬
2	OK Cancel

Figure 8.19: Dialog box Page Setup



A

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 Printout



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.

Cover Sheet	
Options	Project
Show cover sheet	Wood carving workshop
Show picture of model on cover sheet	
Font	
	Client
	Miller Artists
	Engineer
	Engineering Office Bavaria Constructions
D	OK Cancel

Figure 8.20: Dialog box Cover Sheet

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

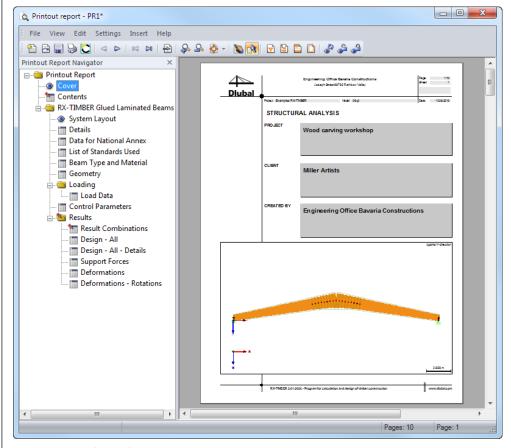


Figure 8.21: Cover sheet in the printout report



8

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.

Languages	×
Existing Languages	
English	
German	
French	
Italian	
Spanish	
Russian	
Czech	=
Polish	
Hungarian	
Slovak	
Portuguese	
Dutch	-
2	OK Cancel

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.

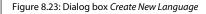
In the lower part of the dialog box (Figure 8.22), you see some buttons used to manage the languages.



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.

•
OK Cancel





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

Languages	×
Existing Languages	
German	*
French	
Italian	
Spanish	
Russian	
Czech	
Polish	=
Hungarian	
Slovak	
Portuguese	
Dutch	
Danish	
Edit Selected Language	ncel

Figure 8.24: Dialog box Languages, button Edit Selected Language

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

RFEM RX-TIMBER	Templa	e language: I	Edit language:			
RX-TIMBER	English	• • • • • • • • • • • • • • • • • • •	Danish	•		
	ID	Englis	h		Danish	
	1	Gas Concrete				
	2	Gypsum Mortar without San	ıd			
	3	Wood Chip Concrete				
	4	Lime Mortar, Lime-gypsum I	Mortar, Anhydrite Morta	Kalkmørtel		
	5	Cement-lime Mortar and Lim	e-trass Mortar			
	6	Loam Mortar				
	7	Light Concrete				
	8	Normal Concrete		Beton		
	9	to B10				
	10	to B15				
	11	Reinforced Concrete		Jembeton		
	12	to B10				
	13	Reinforced Light Concrete				
	14	Cement Mortar and Cement	Trass Mortar			
	15	Concrete Roofing Tiles incl	. Lathing			
	16	Flat-tiled Concrete Bricks				
	17	Plain-tailed Cinderblock Do	uble Roofing			
	18	Roof Panels				
	19	Double-skin Partition Roofin	ng with 22 mm Framew			
	20	Etemit				
	21	Flat Tiles, Flat Clay Roofing	Tile incl. Lathing			
	22	Glass		Glas		
	23	Mission Tiling				
	24	Purlins				
	25	Plexiglass				
	Descrip	ion				
	Descrip	1011				

Figure 8.25: Dialog box Edit Printout Report Texts

Only user-defined languages can be edited.

8 Printout



Copy a language

Languages	X
Existing Languages	
English	*
German	
French	
Italian	
Spanish	
Russian	=
Czech	-
Polish	
Hungarian	
Slovak	
Portuguese	
Dutch	
Danish	*
	×
Create New Language via Copy	Cancel

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.



1



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.

Q Printout report - PR1*	
File View Edit Settings Insert	Help
i 🖹 🖻 🔒 😡 💟 ⊲ ⊳ 🛤 (× 🖹 🐥 🕰
Printout Report Print Printout Report	
E- Printout Report	
Contents	
📄 🛅 RX-TIMBER Glued Laminate	d Beams
System Layout	

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.

rint Printer			? ×
Name:	Brother HL-5270DN series		▼ Properties
Status:	Ready		
Туре:	Brother HL-5270DN series		
Where:	172.16.0.51		
Comment:			Print to file
Print Range		Copies	
All	From To	Number of copies:	1 🚔
Pages:	1 9	Collate	
Current Page	•		
Extended Select	ion (Odd/Even)		
🔘 All			
🔘 Odd only			
🔘 Even only			
			OK Cancel

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.

Save As					×
Save in:	RFEM		•	3 👂 📂 🛄 -	
(And	Name	*		Date modified	Туре
Recent Places) 2010 2011 2012 2012 2013			14.05.2012 08:28 14.05.2012 08:29 07.08.2012 11:48 04.07.2013 07:46	File folder File folder File folder File folder
Libraries					
Computer					
Network	File name: Save as type:	III Glulam Position.rtf Rtf Files (*.rtf)			Save Cancel
Options	ted blocks only		Dire	ct Export to Program V	'Cmaster
Stretch factor fo	or pictures: 1.00				

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

ument.

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 doc-



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.

Units and Decimal Places						×
Program / Module	Model Loads Results					
····· RX-TIMBER	Deformations			Stresses		
		Unit	Dec. places		Unit	Dec. places
	Displacements:	mm 👻	1 🚖	Stresses:	N/mm^2 ▼	2 ≑
	Rotations:	mrad 👻	1 🚔	Design ratios:	% -	2 🚔 🖣
				- Factors:	% -	3 🌲 🖣
					<u>'</u> •	
	Support and Internal Force	es		Parts List		
	Forces:	kN 👻	3 🚔	Lengths:	m 🔻	3 🌩
	Lengths for moments:		3 🌲	Total lengths:		3 🌲
	Lengths:	m 🔻	3 🜩	Surface areas:	m^2 ▼	2 🌩
	Conguna.		J	Volumes:		
						2 🖨
				Weight:	t 🔻	3 🌩
2 🛛 🖻 🖬 🖪					ОК	Cancel

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.

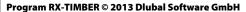
The units and decimal places that are relevant for the current table are marked with a red triangle as shown in the figure above.

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.





9 General Functions

4	
Dlubal	

Profile			
Name:			
Timber construct	tion		
Save profile a	s default		
(?)		ок	Cance

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.

Load Profile	×
List of Profiles	
Name	Default
Metric	V
Imperial	
Timber-construction	
	×
	Cancel

Figure 9.3: Dialog box Load Profile

9.2 Language Settings

The language that has already been selected for installation is preset. Materials and crosssection 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.

lit Archive Data	View					
Copy to Clipboard	d Ctrl+	C nt	Proje	ct: 📜 Example	es RX-TI	MBER
Paste from Clipbo	ard Ctrl+	V _{RS}	TAB	SHAPE-THIN	SHAPE-N	IASSIVE
Restore from Dlub	bal Recycle Bin			Description	LC	RC
Empty Dlubal Recy	ycle Bin				11	0
Settings for Dluba	al Recycle Bin				11	0
			_			0
Language (RX-TIM	IBER)	•	G	erman		0
Program Options		•	 Er 	nglish		0
	1.0	-	Cz	ech		3
ples			Ita	alian		0
X-TIMBER			Sp	anish	11	0
(-TIMBER			Fr	ench		
			Po	olish		
ith supports			Ru	ussian		
			Po	ortuguese		
	Copy to Clipboarn Paste from Clipbo Restore from Dlub Empty Dlubal Rec Settings for Dlub Language (RX-TIM Program Options X-TIMBER C-TIMBER	Copy to Clipboard Ctrl+ Paste from Clipboard Ctrl+ Restore from Dlubal Recycle Bin Empty Dlubal Recycle Bin Settings for Dlubal Recycle Bin Language (RX-TIMBER) Program Options ples T-TIMBER C-TIMBER th supports liles	Copy to Clipboard Ctrl+C nt Paste from Clipboard Ctrl+V Restore from Dlubal Recycle Bin Empty Dlubal Recycle Bin Settings for Dlubal Recycle Bin Language (RX-TIMBER) Program Options ples X-TIMBER (-TIMBER ith supports	Copy to Clipboard Ctrl+C Paste from Clipboard Ctrl+V Restore from Dlubal Recycle Bin Empty Dlubal Recycle Bin Settings for Dlubal Recycle Bin Language (RX-TIMBER) ↓ Program Options ples 07.gl 1008.gl X-TIMBER ith supports iles	Copy to Clipboard Ctrl+C Paste from Clipboard Ctrl+V Restore from Dlubal Recycle Bin Empty Dlubal Recycle Bin Eanguage (RX-TIMBER) Program Options Program Options Program Options Program Spanish C-TIMBER C-TIMBER ith supports iles	Copy to Clipboard Ctrl+C nt Project: Examples RX-TI Paste from Clipboard Ctrl+V RSTAB SHAPE-THIN SHAPE-A Restore from Dlubal Recycle Bin Description LC Empty Dlubal Recycle Bin 0 11 Settings for Dlubal Recycle Bin 11 11 Language (RX-TIMBER) 0 11 Program Options 0 German 15 Czech 11 11 11 Shapes 07.gl 11 11 K-TIMBER 08.gl Spanish 11 French Polish Russian 11 Its supports Russian Russian 11

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

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



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.

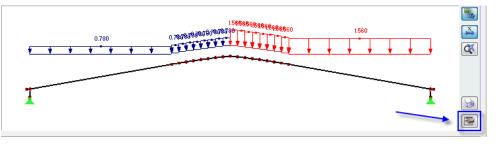


Figure 9.5: Button Display Properties in the dialog box Load Cases

The dialog box Display Properties appears.

Display Properties - Standard with black background	_					×
Category		Screen Printout P	Report			
Objects by Color	*	Loade Line and L	lember Loads - Uniform Forces			
· Fonts						
ia - Model Data ia - Loads		Line color +/- :				
i ⊡- Loads i i - Self-weight		Line type:	Solid 👻			
Sei-weight Nodal Loads		Line type.	Solid			
Line and Member Loads		Line thickness:	1 🚔			
- Concentrated Forces						
		Values			20.000	
Trapezoidal Forces		Alignment:		+ +	+ $+$ $+$	+ + +
Varying Forces		Algrinolic	000	-		
- Concentrated Moments				1 1	† † †	† † †
- Uniform Moments	=		\odot \odot \odot		20.000	
Trapezoidal Moments					20.000	
Uniform Axial Strains						
Trapezoidal Axial Strains		Position / Hatching	a / Size			
- Uniform Precambers		Space:	0.200 ≑ [m] 🔍			
- Trapezoidal Precambers		space.				
- Uniform Prestresses		Hatching:	1.000≑ [m] 🔍			
Trapezoidal Prestresses Uniform End Prestresses		1 [kN/m]:	0.500 ≑ [m]			
		r (Krivin).				
- Piping Content - full						
- Piping Content - partial						
- Uniform Temperature						
Trapezoidal Temperature						
- Concentrated Displacements						
- Uniform Displacements						
- Trapezoidal Displacements						
- Concentrated Rotations						A 06
Uniform Rotations						
Trapezoidal Rotations		Identical Settings				
Rotary Motion	-	For screen and p	printout report			
🗄 🗄 Curface Loade	Ŧ		······			
					OK	Cancel

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.

Show Generated Loads... resp. Define Additional Loads...



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 <u>subsequently</u> 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:

A	Opens the Font dialog box for changing type, size and color of font
*	Goes to display parameters of axes for current object
123	Opens the dialog box <i>Relative Positions</i> 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

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.

9 General Functions



Table Parameters	Application
📝 With table header	Microsoft Excel
Only marked rows	OpenOffice.org Calc
	CSV file format
Transfer Parameters	
Export table to active workbook	
Export table to active worksheet	
Rewrite existing worksheet	
Selected Tables	
Active table	Export tables with details
All tables	
🔲 Input tables	
📝 Result tables	

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.

X	- 1	- (* - +				Sheet1 - Microsoft Excel	х
Fi	le	Home	Insert	Page Lay	out	Formulas Data Review View Add-Ins 🛆 🕜 🗆 🗟	ε Σ
Î	<u>ل</u> ام ا	Calibri	- <u>u</u> -	0	= : : :		ĥ
Pas	te 🛷		🕭 - 🖉	<u>A</u> -	< >	E ≫r 500 500 E Cell Styles ▼ E Format ▼ 2 * Filter * Sele	
Clipb	oard	ā.	Font	Gi -	Alig	nment 🖬 Number 🖬 Styles Cells Editing	
	D	6	-	f,	· 0,	38	
	А	В	С	D	E	F	
1		Location					
2	No.	X [m]	RC	Ratio		Design acc. to Formula	
3	1	0,000	RC6	0,40	≤1	112) Shear stress at support acc. to 6.1.7	
4	2	0,000	RC6	0,22	≤1	352) Support pressure - Compression at an angle acc. to 6.2.2	
5	з	0,000	RC18	0,24	≤1	512) Fire protection - Shear stress at support acc. to 6.1.7	
6	4	0,154	RC6	0,38	≤1	111) Shear stress acc. to 6.1.7	
7	5	0,154	RC6	0,02	≤1	131) Bending stress on edge parallel to the grain acc. to 6.4.2	
8	6	0,154	RC6	0,02	≤1	133) Bending stress on tapered edge acc. to 6.4.2 - Compression edge	
9	7	0,154	RC6	0,02		301) Lateral torsional stability verification - Bending without compression acc. to 6.3.3	_
10	8	0,154	RC18	0,24		511) Fire protection - Shear stress acc. to 6.1.7	_
11	9	0,154	RC18	0,01		531) Fire protection - Bending stress on edge parallel to the grain acc. to 6.4.2	_
12	10	0,154	RC18	0,01		533) Fire protection - Bending stress on cut edge acc. to 6.4.2 - Compression edge	_
13	11	0,154	RC18	0,05		701) Fire protection - Lateral torsional stability verification - Bending without compression acc. to 6.3.3	_
14 15	12	0,175	RC6	0,37		111) Shear stress acc. to 6.1.7	_
16	13	0,175	RC6 RC6	0,02		131) Bending stress on edge parallel to the grain acc. to 6.4.2 133) Bending stress on tapered edge acc. to 6.4.2 - Compression edge	_
17	14	0,175	RC6	0,02		301) Lateral torsional stability verification - Bending without compression acc. to 6.3.3	-
18	16	0,175	RC18	0,02		511) Fire protection - Shear stress acc. to 6.1.7	-
14 4	► H		sian - All				► I
Rea	dv			A			(+)

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

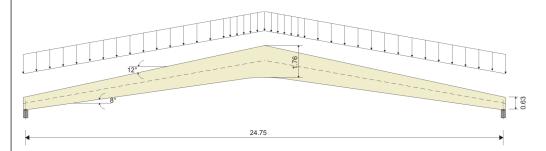


Figure 10.1: System and loads

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 $h_s = 629.2 \text{ mm}$ on the support edge and $h_{ap} = 1761 \text{ mm}$ 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/m². With a spacing of 3 m between the trusses we get a distributed load of 1.92 kN/m plus beam self-weight.



10 Glued-Laminated Beam



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.



10.1.2.1 Beam Type and Material

nput Data	1.1 Beam Type and Material		
-Beam Type and Material	BeamType	According to Standard / National Annex	
— Geometry — Loads — Control Parameters	 Deam type 1. Parallel beam 2. Monopitch roof beam 3. Double tapered beam 4. Curved beam 5. Pitched cambered beam with constant height 6. Pitched cambered beam with variable height 7. Fish beam - Parabolic 8. Fish beam - Linear with roundings in central area Asymmetric layout Cantilevers: Taper * 	Pitched Cambered Beam with Variable Height	
	Capped with loose ridge wedge	Stiffening Elements for Transversal Tension	
	Notch on supports		an beam
	Material	perpendicular to the grain web gird	ctures (so er) built v ninated ti
	Glulam Timber GL24c	Bonded steel bars Steel bars with wood-screw thread Bonded solid wood or wood-based strips Bonded wood-based plates	

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**.

10 Glued-Laminated Beam

Dlubal	

ayout of Steel Bars		Dimensions		Screw Load Bearing Capacity
Determine quantity		Nominal diameter of stee	el bars	According to Tab. 3.4(EN 1993-1-8)
Determine the total number of required steel bars automatically by the program		Predefined	d: 16.0 🔻 [mm]	Strength class of steel
) Define quantity		O User-defined	d : [mm]	bars: 8.8 ▼
The total number of steel bars in the area subjected to tensile load	n1:	Core area		Ft,Rk: 81.430 (kN)
) Define spacing The spacing of steel bars in the		Predefined		Partial factor y _{M2} : 1.25
longitudinal direction of the beam at the height of beam axis	a1:	O User-defined		Pull-off Strength
umber of steel bars			As: 113.1 ⊕ [mm ²]	Acc. to DIN EN/NA:2010-12, 6.8.5 (NA.3)
side length a 1	n: 1荣	Defined by diameter		C User-defined
		Not defined	Ad:	fk1,k
		Size of bore	d _b : 12.0 ≑ [mm]	Connection partial factor 7M: 1.30
	l 1.5x a1	1.25 x a ₁ a ₁ a ₁ β	β	

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

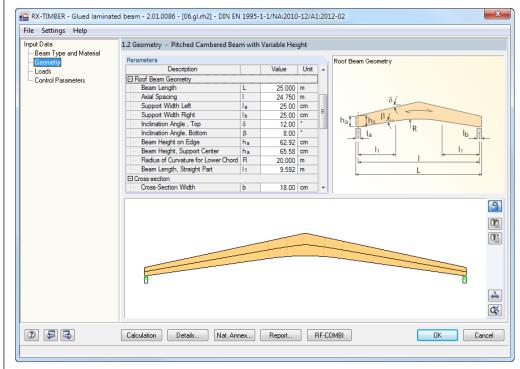


Figure 10.4: Window 1.2 Geometry - Pitched Cambered Beam with Variable Height

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/m ²
Vapour barrier	with	0.02 kN/m ²
• Rock wool (d = 30 cm)	with	0.30 kN/m ²
Purlins	with	0.15 kN/m ²
Gypsum boards incl. strapping	with	0.02 kN/m ²

Beam Type and Material	Perm	nanent Action					Snow Load			
Loads	No.	A Roof Laver	B Thickness	C Load g k	D	-	Altitude A: 200 🚔 [m]			
Control Parameters		Material	d [cm]	[kN/m ²]	Comment	t =	Determine automatically			
	1	Trapezoidal Sheet M		0.150			·			
	2	Vapour Barrier		0.020			Snow load zone SZ: 1 🔻			
	3	Rock Wool	30.00	0.300			Topography type TT: Normal 🗸			
	4	Purlins		0.150						
	5	Gypsum Boards incl		0.020		-	🔲 North German Plain			
	Roo	f structure	gk,2'∶	0.640 🔶	[kN/m ²]	RA	Snow load of overhang			
			gk,2∶	1.920 🌲	[kN/m]	BA	Snow load on snow guard			
		m self-weight rage)	gk,1:	0.701 🚖	[kN/m]	RA	○ Define snow load s _{k'} : 0.650 ⇒ [kN/m ² manually]		
			gk:	2.621 🔶	[kN/m]	BA	sk: 1.950 🔶 [kN/m]			
			-				Wind Load			
	Impo	sed Load				O Determine automatically				
	Impo	osed load	pk::	K: 0.000 🚔 [[kN/m²] B/	BA	Wind zone WZ: 1 💌			
			pk:	0.000 🚖	[kN/m]	BA	Terrain category TC: Category I	Ŧ		
							Fundamental basic wind			
		osed load category			1		velocity v _{b,0} : 22.5 🖨 [m/s]			
	acco	ording to EN 1991-1-1:	991-1-1: H 💌			لعت	Body shells with permeable walls			
		rice Class	Show G	ienerated Lo resp.	ads		Define peak velocity q(z)': 0.000 [[kN/m ² pressure manually c) 2.000 [kN/m ² pressure manually pres]		
	SEC	iL: 2 🔻 🖍	Define	Additional Lo		q(z): 0.000 🚔 [kN/m]				

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 selfweight of the beam. As the beam has a linearly variable height, a trapezoidal load for the selfweight 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.

```
Show Generated Loads...
resp.
Define Additional Loads...
```



0

....



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.

Input Data Beam Type and Material	1.4 Control Parameters - Pitched Cambered Beam with Variable Height									
Eeam Type and Material Geometry Loads Control Parameters	Design of	Support Modelling								
	 Static equilibrium (EQU) Ultimate limit state (STR) 	Left Support: Horizontal fixed Column O Horizontal free								
	✓ Serviceability limit state Do not consider cantilevers Precamber w₀: 0.0 (⇒) [mm]	Right Support. Horizontal fixed Column Image: Column Arrow of the								
	✓ Fire protection	Supports on center line								
	Fire Resistance Class: R 30	Optimization								
		Perform optimization Maximum ratio: 1.00								
	◎ R 90 ◎ R 45 ÷ [min]	Description Symbol Parameter Ur Optimize Beam Height								
	Charring:	Optimize Cross-Section Width Optimize Spacing of Lateral Supports								
	V									
	V	Calculation Parameters								
	To Display	Generate supplementary combinations from favorable permanent actions								
	✓ Support forces ✓ Deformations	Distribute permanent load field-by-field								
	Support compression	divisions for								

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.

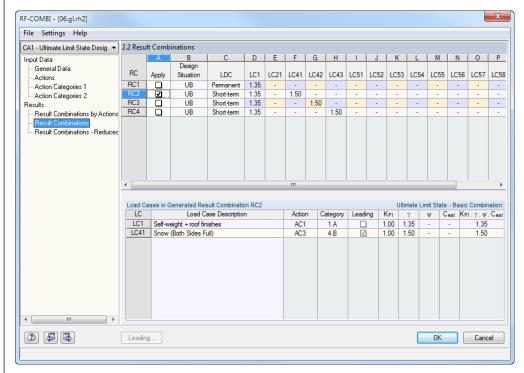


Figure 10.7: RF-COMBI window 2.3 Result Combinations - Reduced

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*.

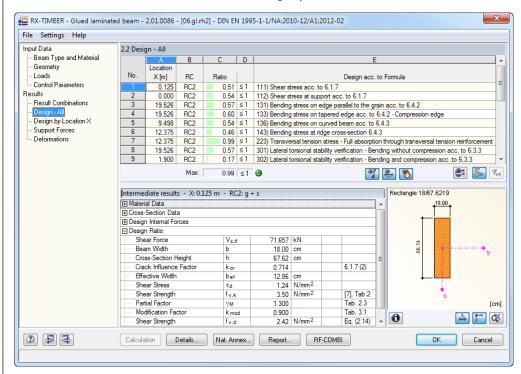


Figure 10.8: Window 2.2 Design - All for ultimate limit state design (shear stress at support)

୍

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

$$\tau_{d} = 1.5 \cdot \frac{V_{d}}{b_{ef} \cdot h_{A}} = 1.5 \cdot \frac{72.34 \cdot 10^{3}}{128.6 \cdot 645.8} = 1.31 \frac{N}{mm^{2}}$$

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

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



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)

 $x = \frac{1 \cdot h_a}{2 \cdot h_1} = \frac{25.00 \cdot 0.656}{2 \cdot 1.565} = 5.24 \text{ m}$

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

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

Dlubal

Input Data	2.2 Design - All												
 Beam Type and Material 		A	B	С	D				[
- Geometry	Nie	Location		_					_	_			
- Loads Control Parameters	No.	X [m]		Ratio					Design acc	to Formul	a		
Results	1	0.125	RC2		51 ≤1	111) Shear s							
- Result Combinations	2	0.000	RC2		54 ≤ 1	112) Shear s							
Design - All	3	19.526	RC2		57 ≤1				arallel to the g				
- Design by Location X	· · ·	4 19.526 RC2 0.60 ≤ 1 133) Bending stress on						mpression edge					
- Support Forces	5	9.498	RC2		54 ≤1				beam acc. to				
Deformations	6	12.375	RC2		46 ≤ 1				ss-section 6.				
Deronnations		7 12.375 RC2 0.99 ≤ 1 223) Transversal tension stress - Full absorption through 8 19.526 RC2 0.57 ≤ 1 301) Lateral torsional stability verification - Bending witho											
	8	19.526	RC2		57 ≤1								
	9	1.900	RC2	0.1	17 ≤ 1	302) Lateral	torsional sta	ability ve	erification - Be	nding and	compression acc	: to 6.3.3	
			Max:	0.9	99 ≤ 1	۲			9		8		
	Interme	diate result	s - X: 19.526	im -	- RC2: g + s Rectangle 18/103.595							5	
	(— Cu	t-to-Grain Angle		α	α	4.00	۰	•			~	18.00	
	Moment			N	1d l	306.552	kNm						
	Be	Beam Width Beam Depth				18.00	cm						
	Be					103.59	cm						
	 Longitudinal Stress 				m.α.d		N/mm ²		Eq. (6.37)				
			rength Perpen		c,90,d v.d		N/mm ²		Eq. (2.14)		- 103.60		
		Shear Strength Auxiliary factor Bending Strength					N/mm ²		Eq. (2.14)		1		
						0.951			Eq. (6.40)				
							N/mm ²		[7], Tab.2				
		- Partial Factor			М	1.300			Tab. 2.3				
		dification Fac			mod m.d	0.900			Tab. 3.1				
		 Bending Strength 					N/mm ²		Eq. (2.14)			[cr	
		nding Strengt	h	fr	m.α.d		N/mm ²		Eq. (6.38)	. 0	1	🚔 🚰 à	
	Design			η		0.60		≤1	Eq. (6.17)		J		

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

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

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

Details...

For beam types of this example the restraint of fork supports causes a torsion of the crosssection. 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*.

intout Report Selection -	PR1		
Program / Modules	Global Selection Input Data Results		
RX-TIMBER	Display		
	2.1 Result Combinations	Filter:	
	2.2 Design - All - Summary	✓ Ultimate Limit State	
	☑ 2.2 Design - All - Details	Serviceability Limit State	
	2.3 Design by Location x - Summary		
	2.3 Design by Location x - Details		
	2.4 Support Forces		
	Min/max only		
	✓ 2.5 Deformations		
	🥅 Min/max only		
isplay			
Contents			
Info pictures			
	Global Selection		
a			
D			K Cance

Figure 10.10: Dialog box Printout Report Selection

Report...



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

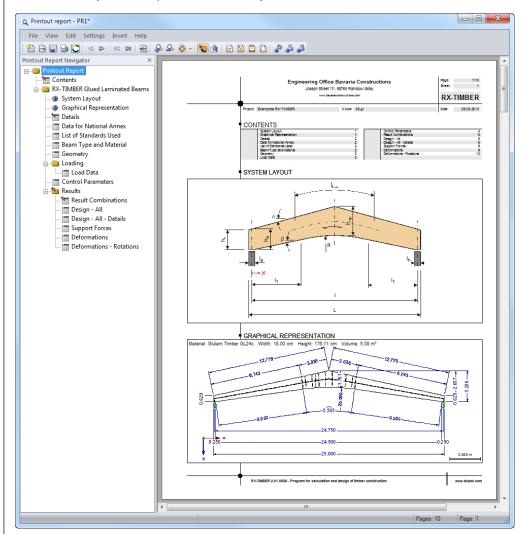


Figure 10.11: Print preview in printout report

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*).

material: glued laminated timber GL24c width: 20.00 cm height: 120.00 cm volume: 3.50 m³

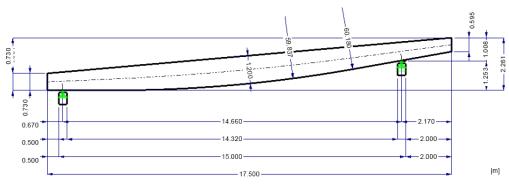


Figure 10.12: Geometry of fish beam

10.2.1 Geometry

The *Beam Type 8* presented in this example has a linear rounding in the center area with a straight line I₁ 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 I₁ 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.

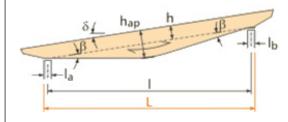


Figure 10.13: Definition parameters



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

Calculation

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.

	2.2 Desig	gn - All											
 Beam Type and Material 		A	B	С	D				E				
- Geometry		Location											
- Loads	No.	X [m]		atio					Design acc	. to	Formula		
Control Parameters	2	0.000	RC2		3 ≤ 1	112) Shear s							
Results 	3	7.347	RC2		5 ≤1	122) Tension							
	4	7.347	RC8		6 ≤1	123) Compre							
Design - All	5	6.809	RC2		5 ≤1								
 Design by Location X Support Forces 	6	14.660	RC2		1 ≤1								
Support Forces	7	6.702	RC2		5 ≤1								
	8	6.809	RC2		5 ≤1		eral torsional stability verification - Bending without compression acc. to 6.3.3						
	9	0.000	RC2		6 ≤1					ndicular to the grain of wood acc. to 6.1.5			
	10	14.660	RC2	0.4	3 ≤ 1	352) Support	pressure -	Compres	ssion at an a	ngle	acc. to 6.2.2		
			Max:	0.43	3 ≤1	۲			2			😫 🖪 🛛	
	-		s - X: 6.809 n								Rectangle 20/119.41	7	
		ment		M	d	277.494				*	20.0	ŧ.	
		am Width		b		20.00					• • •		
	S	am Depth		h		119.42							
		tion Modulu:	•	W		47534.70							
		ngitudinal Stre	ess s-Section Heial		n,d	5.838					19.42		
	EQU			-),65	120.00		_			116	У	
	Eau												
		uivalent Mem		lef		2.000			[7] Tab 2	=			
	Mo	dulus of Elas	ticity		f 0,05	9400.000			[7], Tab.2	н			
	More Mul	dulus of Elas Itiplication Fa	ticity actor	E	0,05	9400.000 1.40		<07	NCI Zu 6.3	ш			
	Mot Mul	dulus of Elas Itiplication Fa ative Slende	ticity actor mess Ratio	Eα	0,05 el,m	9400.000 1.40 0.407	MPa	≤ 0.7	NCI Zu 6.3 Eq. (6.30)	Ш			
	Mul Rel	dulus of Elas Itiplication Fa lative Slende ical Bending	ticity actor mess Ratio Stress	Ε(λη	0,05 el,m n,crit	9400.000 1.40 0.407 144.589	MPa	≤ 0.7	NCI Zu 6.3 Eq. (6.30) Eq. (6.32)			i i i	
	Mul Mul Rel Criti	dulus of Elas Itiplication Fa lative Slende ical Bending eral Buckling	ticity actor mess Ratio Stress Coefficient	E(λr σr ko	0,05 el,m n,orit	9400.000 1.40 0.407 144.589 1.000	MPa MPa	≤ 0.7	NCI Zu 6.3 Eq. (6.30) Eq. (6.32) Eq. (6.34)				
	Mot Mul Rel Criti Late Ber	dulus of Elas Itiplication Fa lative Slende ical Bending	ticity actor mess Ratio Stress Coefficient	Ε(λη	o,05 el,m n,orit prit	9400.000 1.40 0.407 144.589	MPa MPa	≤ 0.7	NCI Zu 6.3 Eq. (6.30) Eq. (6.32)	4 M		ء) المجار الح	

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 $0.65 \cdot l_{ef.}$

For the manual calculations we use the following values:

10 Glued-Laminated Beam

Dlubal

Relative slenderness according to eq. (6.30)

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

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

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

Stress on X-location = 6.81 m

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

Design ratio

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

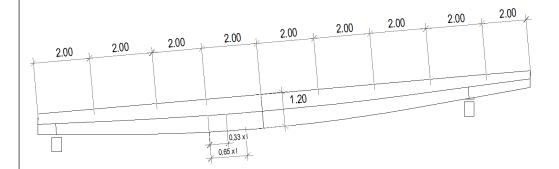


Figure 10.15: Equivalent height in manual calculations

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.





1. 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

LC1 - self-weight and structure

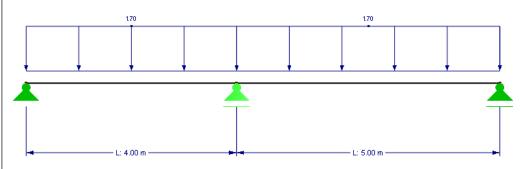


Figure 11.1: System and loads

Model

Cross-section:	$b/d = 70/240 \text{ mm} \text{ NKL } 1 \rightarrow k_{def} = 0.6$
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: $I_1 = 4.00m$ $I_2 = 5.00m$

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 = $1.35 \cdot LC1 = 2.3 \text{ kN/m}$

RC 2 = $1.35 \cdot LC1 + 1.5 \cdot LC 2 = 5.3 \text{ kN/m}$

Serviceability

• Characteristic situations

Restriction of deflection:

 $w_{inst} = w_{g,inst} + w_{q,1,inst} + \sum \psi_{0,i} \cdot w_{q,i,inst} \le I/300$

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

• Quasi-permanent design situation

$$\begin{split} w_{net,fin} &= w_{g,inst} \cdot \left(1 + k_{def}\right) + \sum_{i \ge 1} \psi_{2,i} \cdot w_{q,i,inst} \cdot \left(1 + k_{def}\right) - w_c \le I/300 \\ GQ &= (1 + k_{def}) \cdot g + \psi_{2,i} \cdot p \cdot \left(1 + k_{def}\right) = (1 + 0.6) \cdot 1.7 \text{ kN/m} + 0.3 \cdot 2 \text{ kN/m} \cdot 1.6 = 3.68 \text{ kN/m} \end{split}$$

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.

e Settings Help	
out Data General Data	1.1 General Data
- Geometry	Material According to Standard / National Annex
- Cross-Section - Loads	Poplar and Softwood Timber C24
Control Parameters	Beam Rotation About its Own Axis Slope β: 0.00 ☆ [1]
- Effective Lengths	Slope β: 0.00 ⊕ ['] →× → ↓ ↓ ↓
	Beam Rotation About Is Own Axis Slope B: 0.00 ☆ [1]
	Continuous
	a company sector
	Simple beam Continuous beam
	Gerber beam
	Comment
	Calculation Details Nat. Annex Report RF-COMBI OK Cancel



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

The *Beam Rotation* about its own axis is set to $\beta = 0^{\circ}$, 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 userdefined spring constants to the supports and releases.

General Data <mark>Geometry</mark> Cross-Section Loads	Number o											
Loads Control Parameters Effective Lengths	Number of spans n: 2 Identical span length Identical span length <th></th> <th colspan="2">(m)</th>										(m)	
		A	В	С	D	E	F	G	H		J	
	Support	Location	Span Length	Support Width			isplaceme			Rotation		
	No.	X [m]	l [m]	b [cm]	Type of Support	uχ	uγ	uΖ	φx	ΦY	φ2	
	1	0.100	4.000	20.00	Hinged	1	1	1	1			
	2	4.100	5.000	20.00	Hinged free		1	1	1			
	3	9.100		20.00	Hinged free		1	1	1			
	4											
	5											
	6											
		<mark>► X</mark>]	9.200		5.000				(

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 I:	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.

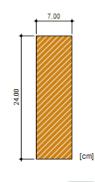
For the *Type of Support* we select a *Hinged* support for *Support No. 1* and two *Hinged free* supports for *No. 2* and *3* in direction X. By using the *User-defined* list option it would be possible to assign any individual degree of freedom as well as translational and rotational spring stiffnesses 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 $\mathbf{b} = \mathbf{7}$ cm and $\mathbf{h} = \mathbf{24}$ 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.



0

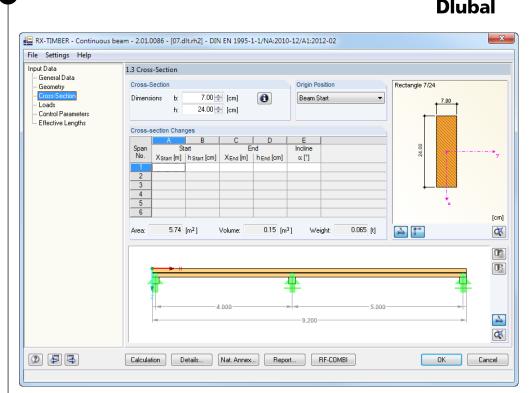


Figure 11.4: Window 1.3 Cross-Section

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).

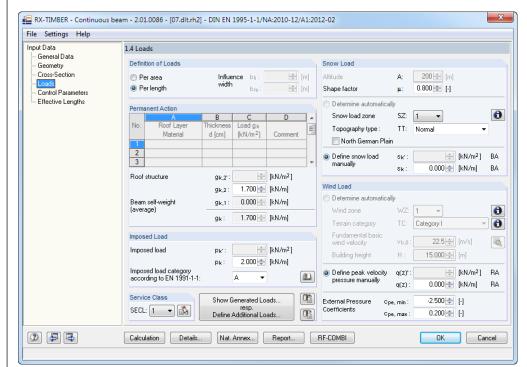


Figure 11.5: Window 1.4 Loads

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.



Show Generated Loads... resp. Define Additional Loads... 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.

Load (Cases									X
Load	Case						Self-weig	ght		
LC N	lo.: LC1 🔻 📢	•	×	Short descrip	ption: g		Consi factor	ider with)	
Desc	cription: Self-Weight + R	oof Finishes					Tactor			×
Load	d Type: Impos	ed Load Ca	ategory:	Load Duratio	on Class:					
<u> </u>	Permanent	-		LDC: Per	rmanent	-	Comment			
ΟV	ariable						Comment	1		
	ccidental eismic									* *
Load	I									Load Case LC1
	A	В	С	D	E	F	G	Н		J
Ne		Load			Load Para		_		Across Tot	
No.	Load Type	Direction	Load Reference	P [kN]	P2	A [m]	В	in %	Length	Comment
1	Concentrated Load	ZL	Span No. 1	0.170	4 700	0.000	4.000			Roof Structure
2	Trapezoidal Load Trapezoidal Load	ZL ZL	Span No. 1	1.700	1.700 1.700	0.000	4.000			Roof Structure Roof Structure
4	Concentrated Load	ZL	Span No. 2 Span No. 2	0.170	1.700	5.000	0.000			Roof Structure
5	Concentrated Load	2L	3part No. 2	0.170		5.000				Hoor Structure
6										
7										
8										
	, 0.170	1.700				1.700			• •	.170
									ОК	Cancel

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.

11 Continuous Beam



neral Data	1.5 Control Parameters	
ometry	Design of	Special Settings for Timber
ss-Section	🔲 Static equilibrium (EQU)	Effect of Cross-Section Dimensions on Material Properties
ids htrol Parameters	Ultimate limit state (STR)	Increase fm,y,k and ft,0,k
ective Lengths	Serviceability limit state	 According to 3.2(3) for massive timber with h < 150 mm (bending) or b < 150 mm (tension)
	Precamber we: 0.0 + [mm]	According to 3.3(3) for glulam with h < 600 mm (bending) or b < 600 mm (tension)
	Fire protection	Increase of Strength fm,z,k
	Fire Resistance Class:	According to 3.3(NA. 6), Edgewise bending (Mz) for glulam timber consisting of a minimum of four side-by-side lamellas
	🔿 R 90	
	○ R 45 (min)	Lamination thickness t : 40.0 - [mm]
		Number of lamellas - Upper ni,u: 2
	Charring:	- Inner ni,o: 2
	✓	Calculation Parameters
		Moment redistribution acc. to DIN EN 1990 1.5.6.4 Redistribution: 10.000 ∲
	To Display	Generate supplementary combinations
	Support forces	from favorable permanent actions Distribute permanent load span-by-span
	✓ Deformations	
	V Support compression	Image: Number of member of member of member of member of members. 10 ÷ Image: divisions for the members. 10 ÷

Figure 11.7: Window 1.5 Control Parameters

Nat. Annex...

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.

General Other S	Standards Used								
Partial Factor Accor	rding to 2.4.1		Limit Va	alues of Defor	mations A	ccording to Ta	ble 7.2		
PT: Permanent ar	ıd transient design sit	uation	- Chara	acteristic (rare)	design siti	uation			
- Solid timber	γм:	1.300 🚔			Fixed	on Both Sides	0	verhanging	
- Glued laminated ti	mber γм:	1.300 🚖	SC:	Winst	≤17	300 🔶	$\leq I_{\mathbf{k}} / $	150 🚖	
AC : Accidental de	sign situation		- Quas	i-permanent de	esign situa	tion			
	ум:	1.000 🚖		W fin - W c	≤17	300 🔶	$\leq l_{\rm k}/$	125 🚖	Eq. (7.2)
			S2:	W fin	≤17	300 🔶	$\leq l_k /$	75 🚖	
	rding to EN 1995-1-2								
For timber in fire	γM,fi:	1.000 ≑							
Modification Factor	k-mod According to	Table 3.1							
	-	ervice Class							
LDC	1	2	3						
- Permanent:	0.600 🚖	0.600 🚖	0.500 🚖						
 Long-term: 	0.700 🚖	0.700 🚖	0.550 🚖						
- Medium-term:	0.800 🚖	0.800 🚖	0.650 🚖						
- Short-term:	0.900 🚖	0.900 🚖	0.700 🔶						
 Instantaneous: 	1.100 🚖	1.100 🚖	0.900 🚖						
Data for Fire Protec	tion According to EN	1995-1-2, 2.3, Table	3.1 and 4.2.2	,					
	Softwood	Glulam	Hardw						
Charring rate	βn: 0.80 🚔	0.70 🚖	0.5	5 🚔 (mm/mi	n]				
Increased charring	dg: 7.00 🚔	7.00 🚖	7.0	10 🚖 [mm]					
Factor	k fi: 1.250 🚔	1.150 🔶	1.25	50 🚖					
2 0.00								οκ	Cancel

Figure 11.8: Dialog box National Annex Settings

The deformation can be defined independently of span and cantilever.

4 Diubal

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.

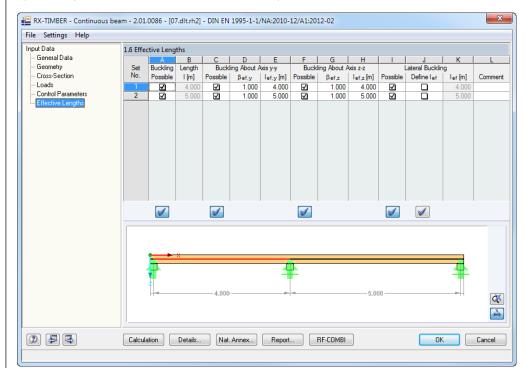


Figure 11.9 Window 1.6 Effective Lengths

11.3 RF-COMBI

[RF-COMBI] button.

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

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.

ability Limit State [-].1 General Data		
Data stegories 1 stegories 2		Supplementary Examination Produce possible result combinations by examining RFEM results From automatically defined combinations
Combination Rules Acc	to Standard / Annex	From result combinations:
💷 EN 1990 + EN 199	15 👻 💻 DIN 👻	
Generating for Design	Situations	Settings for Combinations Acc. to EN 1990
Static equilibrium:	Basic combination EB Accidental EA Accidental - Snow EN Seismic ES	Ultimate limit state basic combination Apply combination rule:
Ultimate limit state:	Basic combination UB Accidental UA Accidental - Snow UN Seismic US	Accidental combination according to Equations 6.11a and 6.11b Apply combination
Serviceability limit state:	Characteristic SC Frequent SF Quasi-permanent SQ	Setting for Timber Structures
Generate supplement from lavorable perm Do not treat togethe as collateral actions For wind as governi	anent actions	Deformation coefficient kdef: 0.60 v 🚵

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*.

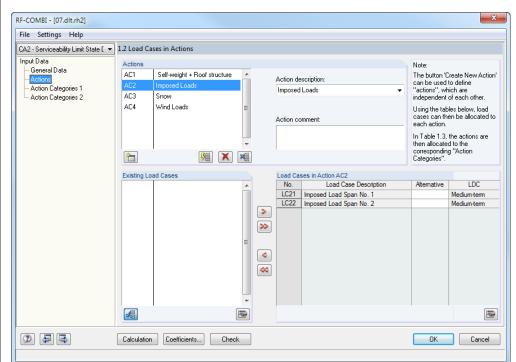


Figure 11.11: RF-COMBI window 1.2 Load Cases in Actions

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.

Coefficients...

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

efficier	nts - EN 1990 + 19	95 DIN					X
Partial 3	Safety Coefficients	Combination Coefficients	Factors	of Construction			
Partial	Safety Coefficients	for Static Equilibrium					
					Design Situation		
Action	Category			Basic Combination	Accidental	Earthquake	
1.A F	ermanent Actions	Unfavorable	γG,sup∶	1.10 *	1.00	1.00	
		Favorable	γG,inf∶	0.90	0.95	0.95	
1.B P	ermanent Actions	Unfavorable	γG,sup∶	1.05	1.00	1.00	
۷	With Small Oscillation	s Favorable	γG,inf∶	0.95	0.95	0.95 ÷	
2. F	restress	Unfavorable	γP,sup∶	1.10	1.00	1.00	
		Favorable	γP,inf∶	0.90 ÷	1.00 🐥	1.00	
3 8. V	/ariable Actions	Unfavorable	γο:	1.50	1.00	1.00	
9. A	Accidental Actions		γA:		1.00		
10. S	Seismic Actions		γ1 :			1.00	
Partial	Safety Coefficients	for Ultimate Limit State					
	Category			Basic Combination	Accidental	Earthquake	
1. F	^p ermanent Actions	Unfavorable	γG,sup∶	1.35	1.00	1.00	
		Favorable	γ/G,inf∶	1.00	1.00	1.00	
	Prestress		γP:	1.00	1.00	1.00	
3 8. V	/ariable Actions	Unfavorable	γο:	1.50	1.00	1.00	
9. A	Accidental Actions		γA:		1.00 📩		
10. S	Seismic Actions		γ 1 :			1.00	
2						ОК	Cancel

Figure 11.12: Dialog box Coefficients, tab Partial Safety Coefficients

Calculation

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.

Dlubal

esult Combinations A B Apply Stuation 1 SC 2 SC 3 SC 3 SC 5 SC 5 SC 6 SQ 7 SQ 8 SQ	C LDC Permanent Medium-ter Medium-ter Medium-ter Medium-ter Medium-ter	D LC1 1.00 1.00 1.00 1.60 1.60 1.60 1.60	E LC21 1.00 1.00 - - 0.48	F LC22 - 1.00 1.00	G LC41 - - -	H LC51 - -	I					
Apply Design Situation 1 1 1 SC 2 1 SC SC 3 1 SC SC 4 1 SC SC 5 1 SQ SQ 6 1 SQ SQ 7 1 SQ SQ	LDC Permanent Medium-ter Medium-ter Permanent Medium-ter Medium-ter	LC1 1.00 1.00 1.00 1.00 1.60 1.60 1.60	LC21 - 1.00 1.00 -	LC22 - 1.00 1.00	LC41 - -	LC51 - -	- -					
Apply Situation 11 12 SC 2 12 SC 3 12 SC 3 12 SC 4 12 SC 5 12 SQ 6 12 SQ 7 12 SQ	Permanent Medium-ter Medium-ter Permanent Medium-ter Medium-ter	1.00 1.00 1.00 1.00 1.60 1.60 1.60	- 1.00 1.00 - -	- - 1.00 1.00	•	•	- -					
Apply Station 1 I SC 2 I SC 3 I SC 4 I SC 5 I SQ 6 I SQ 7 I SQ	Permanent Medium-ter Medium-ter Permanent Medium-ter Medium-ter	1.00 1.00 1.00 1.00 1.60 1.60 1.60	- 1.00 1.00 - -	- - 1.00 1.00	•	•	- -					
2	Medium-ter Medium-ter Medium-ter Permanent Medium-ter Medium-ter	1.00 1.00 1.00 1.60 1.60 1.60	1.00 1.00 -	- 1.00 1.00	•	•	-					
3	Medium-ter Medium-ter Permanent Medium-ter Medium-ter	1.00 1.00 1.60 1.60 1.60	1.00 - -	1.00 1.00	-							
4 ☑ SC 5 ☑ SQ 6 ☑ SQ 7 ☑ SQ	Medium-ter Permanent Medium-ter Medium-ter	1.00 1.60 1.60 1.60	•	1.00								
5 🗹 SQ 6 🗹 SQ 7 🗹 SQ	Permanent Medium-ter Medium-ter	1.60 1.60 1.60	-		-		-					
5 🗹 SQ 6 🗹 SQ 7 🗹 SQ	Medium-ter Medium-ter	1.60 1.60		-		-	-					
6 √ SQ 7 √ SQ	Medium-ter	1.60	0.48			-	-					
7 🔽 SQ				-	•	•	-					
	Medium-ter	1.00	0.48	0.48	-	-	-					
		1.60	-	0.48	-	-	-					
d Cases in Generated R	esult Combination	on RC7			1			Service	eability	Limit Sta	ate - Qu	uasi-permanent
LC Load	Case Descriptio	n		Actio	n Ca	ategory	Leading	KFI	γ	Ψ	Cesi H	KFI.γ.ψ.Cesi
C1 Self-Weight + Roo	f Finishes			AC1		1.A		-		1.60		1.60
C21 Imposed Load Spa	an No. 1			AC2	2	3.A		-		0.48	-	0.48
C22 Imposed Load Spa	an No. 2			AC2	2	3.A		-		0.48		0.48

ubal

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.

File Settings Help							
Input Data	2.1 Resu	It Combinations					
General Data		A	В	C	D	E	F
Geometry				Design		Factor	Max.
- Cross-Section	RC	RC Description	Load Cases	Situation	LDC	kmod	Design
Loads		Ultimate Limit State De	sign				
 Control Parameters 	RC1	g	1.35*LC1	UB	Permane	0.600	1.0
Effective Lengths	RC2	g + p(F1)	1.35*LC1 + 1.50*LC21	UB	Medium-t	0.800	1.0
Results	RC3	g + p(F1) + p(F2)	1.35*LC1 + 1.50*LC21 + 1.50*LC22	UB	Medium-t	0.800	1.7
 Result Combinations 	RC4	g + p(F2)	1.35*LC1 + 1.50*LC22	UB	Medium-t	0.800	1.4
 Design by Beam 							
- Design by Span		Serviceability Limit Stat	e Design				
— Design by Location X	RC5	g	LC1	SC	Permane		0.4
Support Forces	RC6	g + p(F1)	LC1 + LC21	SC	Medium-t		0.5
- Deformations	RC7	g + p(F1) + p(F2)	LC1 + LC21 + LC22	SC	Medium-t		1.0
	RC8	g + p(F2)	LC1 + LC22	SC	Medium-t		1.2
	RC9	g	1.60*LC1	SQ	Permane		0.7
	RC10	g + p(F1)	1.60*LC1 + 0.48*LC21	SQ	Medium-t		0.6
	RC11	g + p(F1) + p(F2)	1.60*LC1 + 0.48*LC21 + 0.48*LC22	SQ	Medium-t		1.0
	RC12	g + p(F2)	1.60*LC1 + 0.48*LC22	SQ	Medium-t		1.1
			Max: 1.74 > 1		I		The second
	Calcula	ation Details	Nat. Annex Report RF-COMBI		OK		Cancel

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.

<u>&</u>

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).

Calculation

4
Dlubal

		А		В		С		D		E
				Sho	rt	Load Case				
LC	L	oad Case D	escr.	Descrip	otion	Туре	EN 1	991-1-1		LDC
LC1	Self-Weight + Roof			g		Permanent				Permanent
LC21	Imposed Load Span			p(F1)		Variable				Medium-term
LC22	Imposed Load Span	No. 2		p(F2)		Variable				Medium-term
LC41	Snow			s		Variable				Short-term
LC51	Wind Shape Factor			w(cpe,m		Variable				erm / Instantane
LC52	Wind Shape Factor	cpe,max		w(cpe,m	ax)	Variable			Short-te	erm / Instantane
No.	Load Type	Load Direction	Load Reference	P [kN]	Load Pa P2	arameters A [m]	в		ross Tot .ength	Comment
					p2		В		ength.	
1	Concentrated Load	ZL	Span No. 1	0.170	1 70	0.000	4.000		<u> </u>	Roof Structure Roof Structure
2	Trapezoidal Load Trapezoidal Load	ZL	Span No. 1 Span No. 2	1.700	1.70		4.000		<u> </u>	Roof Structure
-	Concentrated Load	ZL	Span No. 2 Span No. 2	0.170	1.70	5.000	0.000		<u>+</u>	Roof Structure
5	Concentrated Load	26	Sparrito. 2	0.170		5.000			<u> </u>	noor stractare
6										
	• 0.170	1.700				1.700	•	•	∎ 0.	170
	Z X		4						4	

Figure 11.15: Dialog box Load Cases

11.4.2 Design by Beam

This window shows all governing designs for the entire beam.

iput Data	2.2 Desi	gn by Bean	n								
General Data		A	В	С	D				E		
Geometry		Location									
Cross-Section	No.	X [m]	RC	Ratio				Design ad	c.to Fom	nula	
Loads	1	4.600	RC3	0.	97 ≤ 1	111) Cross-se	ection resistance	ce - Shear due to	shear for	ce Vz acc. to 6.1.	.7
Control Parameters	2	4.100	RC3		03 > 1					t Vz acc. to 6.1.7	1
Effective Lengths	3	4.100	RC3		40 > 1			ce - Uniaxial bend			
esults	4	4.100	RC3	0.	72 ≤ 1					the grain of wood	
Result Combinations	5	4.100	RC3		74 > 1					ompression force	
- Design by Beam	6	6.600	RC8		20 > 1					c. to 7.2 - Inner sp	
Design by Span	7	7.100	RC12		10 > 1						er span, z-direction
Design by Location X	8	7.100	RC12	1.	10 > 1	403) Service	ability - Design	Situation Quasi-p	emanent	acc. to 7.2 - Inne	er span, z-direction
Support Forces Deformations											
Deroimations			Max	1.	74 > 1	Ī		3		°)	2
			s - X: 4.6	500 m -	RC3: g	+ p(F1) + p(F2	2)		Rec	tangle 7/24	
	⊞ Mater								^	+ 7.00	·+
		 Section Dat 	•								
		n Internal Fo	rces								
	🖂 Desig										
		ear Force			z,d	13.370				54.00	
		ss-Section V		b		7.00				54	У
		ss-Section H		h		24.00	cm	0.4 7 40	Ξ		
	0.0	ck Influence	e Factor		cr	0.500		6.1.7 (2)			
		ective Area			lef	84.00			-	+	
		ear Stress			d		kN/cm ²	(0) T 1		÷	
		ear Strength			v,k		kN/cm ²	[8], Tab.1	-	2	
		tial Factor dification Fa			М	1.300		Tab. 2.3			[0
			ctor		mod	0.800	Labl (and 2	Tab. 3.1	- 0		👗 🔛 (
		ear Strength		1	v.d	0.246	kN/cm ²	Eq. (2.14)	-		

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.

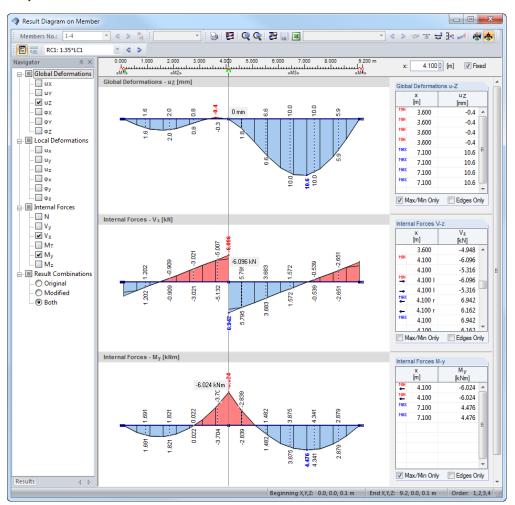


Figure 11.17: Dialog box Result Diagram on Member

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.



10

9

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].

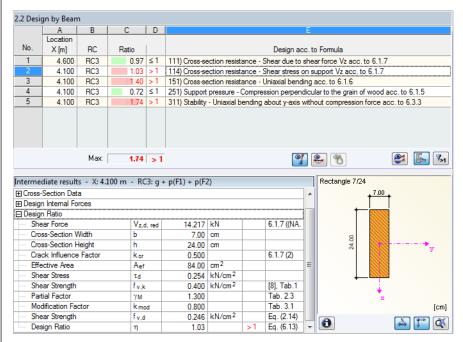


Figure 11.18: Window 2.2 Design by Beam for ultimate limit state only

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

Loading due to shear force on support according to 6.1.7

 $\frac{\tau_{d}}{f_{v,d}} = \frac{2.54}{2.46} = 1.03 > 1$

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
$$au_d = 1$$

$$\tau_{d} = 1.5 \cdot \frac{V_{d}}{b \cdot k_{cr} \cdot h} = 1.5 \cdot \frac{14.2 \text{ kN}}{3.5 \text{ cm} \cdot 24 \text{ cm}} = 0.254 \text{ kN/cm}^{2} = 2.54 \text{ N/mm}^{2}$$

Shear stren

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

Design ratio

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.

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

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

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

$$i_{-} = 0.289 \cdot b = 0.289 \cdot 7 \text{ cm} = 2.02 \text{ cm}$$

Slenderness

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

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

Lateral buckling factor

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

Design ratio

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

k,

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*.

2.



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.

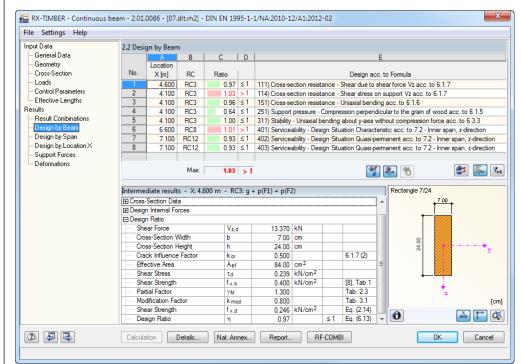


Figure 11.19: Window 2.2 Design by Beam

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.

Global Deformations - uz [mm]

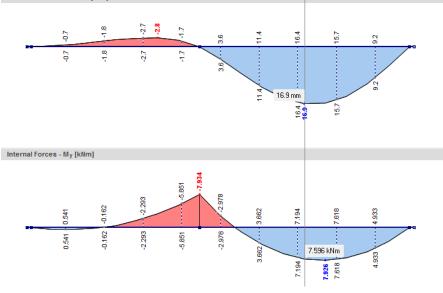


Figure 11.20: Result diagrams in RX-TIMBER for RC8



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 ($E = 1.300 \text{ kN/cm}^2$) 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.

Input Data	2.5 Sup	port Forces								
- General Data		A	В	С	D	E	F	G	н	1
Geometry	LC	LC/RC	Support	Location	Suppo	rt Reactions	[kN]	Suppor	rt Moments	kNm]
Cross-Section	RC	Description	No.	X[m]	Px	Py	Pz	Mx	MY	Mz
Loads		Load Cases (Characteristic Val	ues)							
- Control Parameters	LC1	Self-Weight + Roof Finishes	1	0.100	0.000	0.000	2.454	0.000	0.000	0.00
Effective Lengths			2	4.100	0.000	0.000	9.658	0.000	0.000	0.00
Results			3	9.100	0.000	0.000	3.528	0.000	0.000	0.00
 Result Combinations 	LC21	Imposed Load Span No. 1	1	0.100	0.000	0.000	3.756	0.000	0.000	0.00
- Design by Beam			2	4.100	0.000	0.000	4.800	0.000	0.000	0.00
Design by Span			3	9.100	0.000	0.000	-0.356	0.000	0.000	0.00
— Design by Location X	LC22	Imposed Load Span No. 2	1	0.100	0.000	0.000	-0.868	0.000	0.000	0.00
 Support Forces 			2	4.100	0.000	0.000	6.562	0.000	0.000	0.00
Deformations			3	9.100	0.000	0.000	4.506	0.000	0.000	0.00
	Max		1	0.100	0.000	0.000	3.756	0.000	0.000	0.00
	Min				0.000	0.000	-0.868	0.000	0.000	0.00
	Max		2	4.100	0.000	0.000	9.658	0.000	0.000	0.00
	Min				0.000	0.000	4.800	0.000	0.000	0.00
	Max		3	9.100	0.000	0.000	4.506	0.000	0.000	0.00
	Min				0.000	0.000	-0.356	0.000	0.000	0.00
		Result Combinations for Ultimat	e Limit State	e (Design Valu	ies) (STR)					
	RC1	g	1	0.100	0.000	0.000	3.314	0.000	0.000	0.00
			2	4.100	0.000	0.000	13.038	0.000	0.000	0.00
			3	9.100	0.000	0.000	4.762	0.000	0.000	0.00
	RC2	g + p(F1)	1	0.100	0.000	0.000	8.947	0.000	0.000	0.00
			2	4.100	0.000	0.000	20.238	0.000	0.000	0.00
			3	9.100	0.000	0.000	4.229	0.000	0.000	0.00
2 4 4	Calcula	ation Details Nat	Annex	Report	BF-CC				ок	Cance

Figure 11.21: Window 2.5 Support Forces

11.5 Documentation

Report...

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 print-out report is described in detail in chapter 8.

Modules

ľ Column

ſ Frame

Purlin

Brace

RX-TIMBER Glued Laminated Beam

Continuous Beam

ąх



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**.

System and Loads

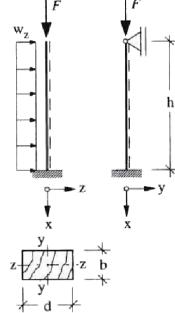


Figure 12.1: System and loads

Model

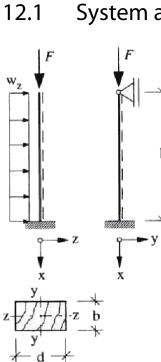
Cross-section:	d = 21 cm
Material:	CT C24
Height:	h = 3.20 m
SECL:	1
LDC:	permanent

Loading

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

Design values for load bearing capacity

$N = 1.35 \cdot F = 1.35 \cdot 45 \text{ KN} = 60.75 \text{ kN}$	$(k_{mod} = 0.6)$
$q = 1.5 \cdot w = 1.5 \cdot 1.5 \text{ KN/m} = 2.25 \text{ kN/m}$	$(k_{mod} = 0.9)$





12.2 Input Data

12.2.1 General Data

ut Data	1.1 System		
System Loads Control Parameters	Column Type 1. Hinged column @ 2. Bracket Elastic Foundation / Support Ineffectivity Head column Ø Ecoting column height H: 3.200 (m) Effective length for buckling Lef.z: Effective length for LT buckling Lef: 6.400 (m)	According to Standard / National Annex	RX-TIMBER Column
	Material Poplar and Softwood Timber C24 Comment Commen	Cross-section <u>Bectangular</u> b: 20.00 ⊕ [cm] h: 20.00 ⊕ [cm] <u>Becund</u> d: 21.00 ⊕ [cm]	

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.

Support Cond	altions		
Support	Spring Constant	Support Ineffectiv	ity
🔽 ux	[kN/m]	None	
🔽 uv	[kN/m]	None	-
🔽 uz	[kN/m]	None	Ŧ
Restraint	Spring Constant		
V 🔍	[kNm/ra	d]	
🗸 oy	[kNm/ra	d]	
V ØZ	[kNm/ra	d]	

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**.

RX-TIMBER - Column - 2.0	01.0086 - [02.clm.rh2] - DIN EN 1995-1-1/NA:2010-12/A1:2012-02	×
File Settings Help		
Input Data System	1.2 Loads	
Loads	Permanent Action Load Eccentricity	
Control Parameters	Vertical load on column head G _k : 45.000 ⊕ [kN] Eccentricity of vertical loads e _y : 0.00 ⊕ [cm]	
	Column self-weight g _{k,1} : 0.000 🔶 [kN/m]	
	Longit. load from wall weight gk.2: 0.000 [kN/m]	
	Imposed Load	
	Vertical load on column head Pk: 0.000 (kN) Pk	
	Imposed load category according to EN 1991-1-1: H 9 9k, g 0,2	
	Snow Load A B C No. LC Description Sk [kN] 1	
	Wind Load	
	No. LC Description W _k [kN] w _{k,Y} [kN/m] w _{k,Y} [kN/m] # 1 Wind 0.000 1.500 0.000 # Service Class Show Generated Loads 2 3	
	Calculation Nat. Annex Report RF-COMBI OK Ca	ancel

Figure 12.4: Window 1.2 Loads

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

Load Cases									×
Load Case						Self-we	ight		
LC No.: LC1 1 Description: LC1 0	n De	x escription ht + Vertical Load	Short descri LDC Permanent		Comment	Cons facto	ider with r:)	
Load Type: Permanen Variable	Imposed I Snow Wind	.oad	Short-term Short-term Short-term / In	stantaneous	5	Commen	t		
Accidental									* *
Load	В		(_		_	н		Load Case LC1
A	B Load	С	D	E Load Par	F ameters	G		Across Tot	J
No. Load Type	Direction	Load Reference	P [kN]	p2	A [m]	в	in %	Length	Comment
Concentrated Load	ZL	Whole Column	45.000		3.200				
2									
3 4									
5									
6									
7									
8									
			ţ	45 000					1
			+						
								OK	Cancel

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.

RF-COMBI

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

RF-COMBI - [02.clm.rh2]			X
File Settings Help			
CA1 - Ultimate Limit State Desig 💌	1.2 Load Cases in Actions		
Input Data General Data Actions - Action Categories 1 - Action Categories 2	Actions AC1 Self-weight + Vertical Load AC2 Imposed Loads AC3 Snow AC4 Wind Loads Existing Load Cases	Action description: Imposed Loads Action comment: Load Cases in Action AC2 No. Load Case Description LC21 Imposed Load	Note: The button 'Create New Action' can be used to define independent of each other. Using the tables below, load cases can them be allocated to each action. In Table 1.3, the actions are then allocated to the corresponding 'Action Categories''. Atemative LDC Short+em
	Calculation Coefficients Check		OK Cancel
	Laicuration Loerricients Lheck]	

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

X

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.

ut Data	1.3 Control Parameters						
– System – Loads – Control Parameters	Design of Static equilibrium (EQU) V Ukimate limit state (STR) Serviceability limit state	Special Settings for Timber Effect of Cross-Section Dimensions on Material Properties Increase fm,y,k and ft,g,k According to 3.2(3) for massive timber with h < 150 mm (bending) or b < 150 mm (tension) According to 3.3(3) for gular with h < 600 mm (bending) or b < 600 mm (tension)					
	 □ Fire protection Fire Resistance Class: □ R 60 □ R 90 □ R 45 □ [min] 						
		Design Settings Reduction of stiffness with coefficient 1 / (1 + kder) due to creep effects in Service Classes 2 and 3 according to NA. 5.9					
	To Display Support forces Deformations	Calculation Parameters Number of member - Result diagrams: 10 -					

Figure 12.7: Window 1.3 Control Parameters

Nat. Annex...

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.

RX-TIMBER - Column - 2.01	.0086 - [02	.clm.rh2] - DIN EN 19	95-1-1/NA:2010-12/A1:2012-02				×
File Settings Help							
Input Data	2.1 Resu	It Combinations					
System		A	B	С	D	E	F
Loads Control Parameters	RC			Design		Factor	Max.
Results	nu	RC Description	Load Cases	Situation	LDC	k mod	Design
	RC1	Ultimate Limit State De	sign 1.35*LC1	UB	Permanent	0.600	0.85
- Design - All	RC2	g g + w1	1.35*LC1 + 1.50*LC51	UB	Short-term / Instantaneous	1.000	1.20
 Design by Location x 	1102	g+wi	1.35 ECT + 1.50 EC51	UB	Short-term / Instantaneous	1.000	1.20
- Support Forces		Serviceability Limit Stat	e Design				
Deformations	RC3	g	LC1	SC	Permanent		
	RC4	g+w1	LC1 + LC51	SC	Short-term / Instantaneous		0.88
	RC5	g	1.60*LC1	SQ	Permanent		
						2 3	
			😤 😂 Max 🗆	1.20 >1			
	Calcula	ation	Nat. Annex Report	RF-COMBI	0		Cancel

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].

Calculation



12.3.2 Designs

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

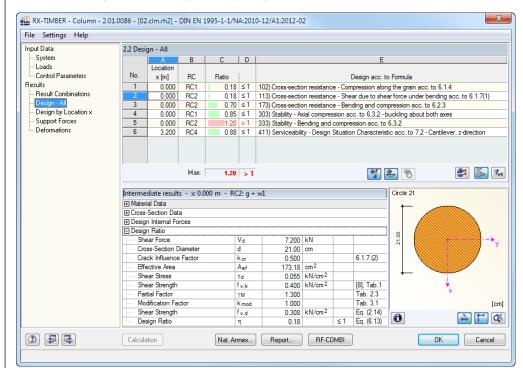


Figure 12.9: Window 2.2 Design - All

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

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

Compressive strength

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

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

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

Compressive strength

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

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

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

Relative slenderness ratio

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

k-factor

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

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.

out Data	2.2 Desi	an - All									
- System	Ziz Desi	A	В	С	D				E		
- Loads		Location	D	<u> </u>					C		
- Control Parameters	No.	x [m]	RC	Ratio	1				Design acc. t	to Formula	
esults	1	3.200	RC4	0.88	3 ≤1	411) Serviceabil	ity - Desig		-	stic acc. to 7.2 - Cantilever, z-dire	ction
- Result Combinations - Design - All - Design by Location x - Support Forces - Deformations											
		P	Max:		8 ≤1 (-			? (H
	Interme	diate result	s - x: 3.2	00 m - R	C4: g + 1	w1				Circle 21	
		al Data Section Dat	_								
	⊡ Defor		9								
		ection x			Wx	-0.4	mm				
	Dire	ection y			wy	0.0	mm			- 12	.
		ection z			Wz	18.7	mm				
	🖃 Desig			/							
		ormation at			Winst,z		mm				
		erence Leng it Value Crite			1/	3.200	m			- <u>i</u>	
		t Value Onte it Value of D				150.00				- 2	
		ign Ratio	eronnauor		Winst,lim η	0.88		≤1	Tab. 7.2	-	
	Dec	ign natio			4	0.00		- 21	100.7.2	-	_
									-	- 🖯 🔺	1

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

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

Serviceability limit state design

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

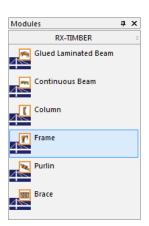
Design ratio

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

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.

Dlubal

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

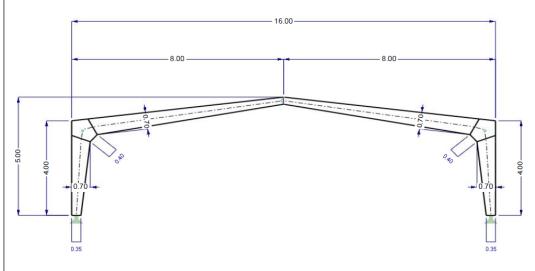


Figure 13.1: System and loads

Model

Cross-section width:	b = 30 cm
Cross-section depth in footing:	h₁ = 35 cm
Cross-section depth at vertex:	$h_f = 30 \text{ cm}$
Cross-section depth at frame joint:	$h_1 = 70 \text{ cm}$
Length of inserted wedge:	$I_{zw} = 40 \text{ cm}$

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.

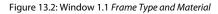


13.2 Input of Model Data

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.

RX-TIMBER - Frame - 2.01.0	086 - [01.frm.rh2] - DIN EN 1995-1-1/NA:2010	0-12/A1:2012-02	×
File Settings Help			
Input Data <mark>Frame Type and Material</mark> Geometry	1.1 Frame Type and Material Material	According to Standard / National Annex	
- Loads Control Parameters	Glulam Timber GL32c 🔹 🔹	■EN 1995-1-1:2004-11 ▼ ■DIN ▼ 🗃	
	Frame Type Column inside sloping Column outside sloping Symmetrical Asymmetrical HaiFrame Frame Joint Type Fringer joint Wh intermediate piece	Description	RX-TIMBER Frame
	User-defined frame joint	Direction of Lamellas O Parallel to inner edge O Parallel to outer edge	Timber Frame
	Comment	* *	
	Calculation Details Nat. Annex	. Report RF-COMBI	OK Cancel



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 userdefined spring constants to the supports and releases.

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

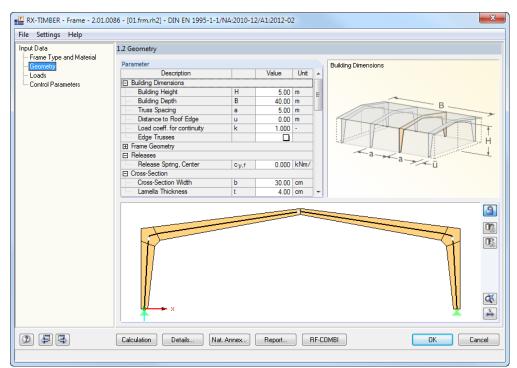


Figure 13.3: Window 1.2 Geometry

The Frame Geometry is defined by the following parameters:

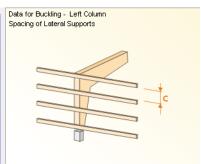
Parameter					F	rame Geometry
Description		Value	Unit			
Frame Geometry						
Frame Width	La	8.00	m			La
 Height on Edge 	Hs	4.00	m			ta br
Height in the Middle	Hf	5.00	m	Ξ		
 Cross-Section Height in Footing 	ha	35.00	cm	1		
 Cross-Section Height at Ridge 	hf	30.00	cm			h1
 Cross-Section Height at Frame Joir 	h1	70.00	cm	1		± + Hi kan
Length of Inserted Wedge	l _{zw}	40.00	cm	1		
				1		
Cross-Section				1		
Data for Buckling				1		
- User-defined				1		
- Left Column				-		

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	1
Data for Buckling				1
User-defined				1
- Left Column				1
 Beam Endangered by Lateral Buckling 		V		1
 Lateral Support Available 				1-
 — Spacing of Lateral Supports 	с	2.00	m	
 Bracing Spacing 	е	0.60	m	1
Coefficient for effective length	βs		-	1
🖃 Left Beam				1
 Beam Endangered by Lateral Buckling 		V		1
Lateral Support Available				1
 Spacing of Lateral Supports 	С	2.00	m	1
Bracing Spacing	е	0.60	m	1.



Dlubal

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					Information Parameters
Description		Value	Unit		
Left Attic					18
⊞ Right Attic					80 03
Information Parameters]	R R Tar
Inclination of Top Edge of Beam	ō	7.13	•	1	1
Angle at Frame Joint	E0	48.56	•		1
 Connection Angle in Frame Joint 	8	69.28	•		i lus
 Slope Angle in Frame Joint 	α	20.72	•		
 Slope Angle of Column 	α _s	6.41	•		
 Slope Angle of Beam 	٥Lr	3.22	•	Ξ	
 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/m ²
Purlins	with	0.15 kN/m ²
Vapour barrier	with	0.02 kN/m ²
• Rock wool (d = 20 cm)	with	0.20 kN/m ²
• Oriented strand boards (OSB) (d = 25 mm)	with	0.15 kN/m ²

File Settings Help								
Input Data	1.3 Loa	ads						
 Frame Type and Material 	Perm	anent Action					Snow Load	
Geometry Loads		A	В	C	D			
- Control Parameters	No.	Roof Layer	Thickness	Load g _k			Altitude A: 200 🜩 [m]	
Control 1 didilicitors		Material	d [cm]	[kN/m ²]	Comment	- =	Determine automatically	
	1	Trapezoidal Sheet M		0.15				_
	2	Purlins		0.15			Snow load zone SZ: 1 🔻	
	3	Vapour Barrier		0.02			Topography type : TT: Normal	-
	4	Rock Wool	20.00	0.20		-	North German Plain	
	5	OSB	2.50	0.15				
	Roof	structure	gk,2'∶	0.67 🚖	[kN/m²]	RA	Snow load on snow guard	
			gk,2:	3.35 🔶	[kN/m]	RA	Define snow load s _{k'} : 0.65 (kN/m ²) manually	B
	Bean (aver	n self-weight age)	gk,1∶	0.71 🚔	[kN/m]	RA	sk: 3.25 → [kN/m]	BA
			gk:	4.06 🌲	[kN/m]	RA	Wind Load	
	Impos	ed Load					Determine automatically	_
							Wind zone WZ: 1 🔻	
	Impo	sed load	рк: рк:		[kN/m ²] [kN/m]	BA BA	Terrain category TC: Category II	-
		sed load category	•		paring		Fundamental basic wind velocity vb.0 : 22.5 [m/s]	
	acco	rding to EN 1991-1-1:		H 👻			Body shells with permeable walls	
	Servi SECL	ce Class : 2 ▼ 💦		Generated Loa resp. Additional Loa			● Define peak velocity pressure manually q(z)*: 0.56 ☆ [kN/m²] q(z): 2.81 ☆ [kN/m]	
	Calcu	lation Details	. Nat.	Annex	Report		RF-COMBI OK C	ance

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 selfweight 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.



0

....

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.



Show Generated Loads... resp. Define Additional Loads... 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.

LC No: LC55 Verifician LC1 g Self-Veight + Roof Finishes Permarent Lc21 g Self-Veight + Roof Finishes Permarent Lc21 p Imposed Load Short-term Permanent LC22 p Imposed Load Short-term Value LC23 eg L	Load Case									Self	-weight			
Description: LC1 g Self-Weight + Roof Finishes Permanent Load Type: LC21 p Imposed Load Short-term Permanen LC22 p Imposed Load Short-term @ Variable LC42 sl() Snow (Both Sides Full) Short-term Accidenta LC42 sl() Snow (Right Side Full) Short-term / Instantaneous LC51 w(q,LA8) Wind Transversely to Ridge (left)(BA) Short-term / Instantaneous LC53 LC54 w(q,LA8) Wind Transversely to Ridge (left)(BA) Short-term / Instantaneous LC54 No. Lc55 w(q,LBA) Wind Transversely to Ridge (left)(BA) Short-term / Instantaneous H J 1 Concer LC56 w(q,LBA) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous Zone G; cpe = -1.11 2 Trape: LC58 w(q,LBA) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous Zone G; cpe = -0.56 3 Trape: LC56 w(q,LBA) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous Zone G; cpe = -0.56 4 <t< td=""><td>LC No.:</td><td></td><td></td><td><u>}</u></td><td></td><td></td><td>Short d</td><td>•</td><td></td><td></td><td></td><td>with</td><td></td><td></td></t<>	LC No.:			<u>}</u>			Short d	•				with		
LC1 g Self-Weight + Roof Finishes Permanent Load Type: LC21 p Imposed Load Short-term Permanen LC41 s Snow (Both Sides Full) Short-term Q Variable LC43 s(i) Snow (Right Side Full) Short-term LC43 s(ii) Snow (Right Side Full) Short-term / Instantaneous LC52 LC54 w(q,LA8) Wind Transversely to Ridge (left)(A4) Short-term / Instantaneous LC54 LC55 w(q,LB8) Wind Transversely to Ridge (left)(B4) Short-term / Instantaneous LC56 No. LC56 w(q,LB4) Wind Transversely to Ridge (left)(B4) Short-term / Instantaneous ance Across Tot 1 Concer LC56 w(q,LB4) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Zone G; c.p.e = 1.11 2 Trapes LC58 w(q,LB4) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Zone G; c.p.e = -0.56 2 Trapes LC59 w(q,LB4) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Zone G; c.p.e = -0.56 Zone H; c.p.e = 0.56 <td< td=""><td>Description:</td><td></td><td>Shd_vterm</td><td></td><td>Descrip</td><td>tion</td><td></td><td></td><td>LDC</td><td>Comm</td><td>ent^{pr:}</td><td></td><td></td><td></td></td<>	Description:		Shd _v term		Descrip	tion			LDC	Comm	ent ^{pr:}			
Permaner LC22 p Imposed Load Short-term Imposed Load Short-term Short-term Short-term Accidenta LC42 s(1) Short-term Short-term Seismic LC51 w(q,LA8) Wind Transversely to Ridge (left)(A4) Short-term / Instantaneous Lc54 w(q,LB8) Wind Transversely to Ridge (left)(B4) Short-term / Instantaneous Lc54 No. Lc54 w(q,LB8) Wind Transversely to Ridge (left)(B4) Short-term / Instantaneous Imposed Case LC No. Lc55 w(q,LB8) Wind Transversely to Ridge (left)(B4) Short-term / Instantaneous Imposed Case LC 1 Concert LC55 w(q,LBA) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Imposed Case LC 1 Concert LC55 w(q,LBA) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Imposed Case LC 2 Trape: LC58 w(q,LBA) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous Imposed Case LC 2 Trape: LC58 w(q,LBA) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous Imposed			g	Self-W	eight + Roof F	inishes		Permanent						
Image: Construction of the second	Lodd (Jpo.		· .											
Workshole LC42 s(i) Snow (Left Side Full) Short-term Accidenta LC42 s(i) Snow (Right Side Full) Short-term Seismic LC51 w(q,LA8) Wind Transversely to Ridge (left)(A8) Short-term / Instantaneous Load LC52 w(q,LA8) Wind Transversely to Ridge (left)(A8) Short-term / Instantaneous No. Los LC54 w(q,LA8) Wind Transversely to Ridge (left)(A8) Short-term / Instantaneous LC54 w(q,LA8) Wind Transversely to Ridge (left)(A8) Short-term / Instantaneous Load Case LC LC56 w(q,LA8) Wind Transversely to Ridge (left)(B4) Short-term / Instantaneous Load Case LC 1 Concer LC57 w(q,LA8) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Zone G; c.pe = -1.11 2 Trape: LC58 w(q,LA8) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Zone G; c.pe = -0.54 3 Trape: LC59 w(q,LA8) Wind Transversely to Ridge (light)(B4) Short-term / Instantaneous Zone G; c.pe = -0.56 5 Trape: LC59 w(q,LA8) Wind Tra	Permanen		P											
LC42 slip Snow (Leff Side Full) Short-term Seismic LC53 w(q,LAA) Wind Transversely to Ridge (left)(AA) Short-term / Instantaneous Load LC52 w(q,LAB) Wind Transversely to Ridge (left)(AB) Short-term / Instantaneous Load LC53 w(q,LAB) Wind Transversely to Ridge (left)(AB) Short-term / Instantaneous No. LC54 w(q,LBA) Wind Transversely to Ridge (left)(AB) Short-term / Instantaneous 1 Conceer LC55 w(q,LBA) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous 1 Conceer LC56 w(q,LBA) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous 2 Trape: LC59 w(q,LBA) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous Image: Conceer Zone G; cpe = -1.11 2 Trape: LC59 w(q,LBA) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous Image: Concer Zone G; cpe = -0.56 3 Trape: LC59 w(q,LBA) Wind Parallel to Ridge (R) Short-term / Instantaneous Image: Concer Zone I; cpe = -0.56 5	Variable		-	Snow (Both Sides Fu	all)					nt			
Seismic LC51 w(q,IAA) Wind Transversely to Ridge [left](AA) Short+erm / Instantaneous LC53 w(q,IAB) Wind Transversely to Ridge [left](BA) Short+erm / Instantaneous Load Case LC LC54 w(q,IAB) Wind Transversely to Ridge [left](BA) Short+erm / Instantaneous Load Case LC No. LC55 w(q,IAB) Wind Transversely to Ridge [ight](AA) Short+erm / Instantaneous Ance Across Tot 1 Concer LC56 w(q,IAB) Wind Transversely to Ridge [ight](BB) Short+erm / Instantaneous % Length Comment 2 Trape: LC58 w(q,IAB) Wind Transversely to Ridge [ight](BB) Short+erm / Instantaneous % Length Comment 2 Trape: LC58 w(q,IAB) Wind Transversely to Ridge [ight](BA) Short+erm / Instantaneous % Zone G; c.pe = -1.11 3 Trape: LC58 w(q,IAB) Wind Transversely to Ridge [ight](BA) Short+erm / Instantaneous Zone G; c.pe = -0.54 4 Trape: LC58 W(D,B) Wind Parallel to Ridge [ight](BA) Short+erm / Instantaneous Zone I; c.pe = -0.56 Zone I; c.pe = -0.56 Zone I; c.pe = -0.	~	LC42	s(I)	Snow (Left Side Full)			Short-term						
Lost u(qLBA) Wind Transversely to Ridge [left][AB) Short-term / Instantaneous LC52 w(qLBA) Wind Transversely to Ridge [left][AB) Short-term / Instantaneous No. LC55 w(qLBA) Wind Transversely to Ridge [left][AB) Short-term / Instantaneous I LC54 w(qLBA) Wind Transversely to Ridge [left][BA) Short-term / Instantaneous I LC56 w(qLBA) Wind Transversely to Ridge [light](AA) Short-term / Instantaneous I Concert LC57 w(qLBA) Wind Transversely to Ridge [light](BB) Short-term / Instantaneous I Concert LC57 w(qLBA) Wind Transversely to Ridge [light](BB) Short-term / Instantaneous Zone G; cpe = -1.11 2 Trape: LC58 w(qLBA) Wind Transversely to Ridge [light](BA) Short-term / Instantaneous Zone G; cpe = -0.54 3 Trape: LC59 w(nA) Wind Parallel to Ridge [B Short-term / Instantaneous Zone G; cpe = -0.56 5 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 Zone I; cpe = -0.56 6 Trapezoidal Load z Lef		LC43	s(r)	Snow (Right Side Fu	ll)		Short-term						
Load LC53 w(qLAB) Wind Transversely to Ridge (left)(AB) Short-term / Instantaneous Load Case LC No. LC54 w(qLAB) Wind Transversely to Ridge (left)(BA) Short-term / Instantaneous ance Across Tot 1 LC55 w(qLBB) Wind Transversely to Ridge (light)(BA) Short-term / Instantaneous ance Across Tot 1 Concer LC56 w(qLBB) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous ance Across Tot 1 Concer LC57 w(qLBA) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous ance Across Tot 1 Concer LC58 w(qLBA) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous ance Across Tot 1 Trapez (LC50 w(qLBA) Wind Transversely to Ridge (light)(BB) Short-term / Instantaneous ance Across Tot 1 Trapez (LC50 w(p,B) Wind Paralel to Ridge (B) Short-term / Instantaneous ance Across Tot 5 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 ance Ic pea = -0.56 7 Trapezoidal Load z Left B	💿 Seismic	LC51	w(q,I,AA)	Wind 1	ransversely to	o Ridge	(left)(AA)	Short-term	/Instantane	eous				1
LC33 W(qLAB) Wind Transversely to Ridge (eff(AB)) Short-term / Instantaneous No. LC54 W(qLAB) Wind Transversely to Ridge (night) Short-term / Instantaneous 1 Concert LC55 W(qLAB) Wind Transversely to Ridge (night) Short-term / Instantaneous 1 Concert LC55 W(qLAB) Wind Transversely to Ridge (night) Short-term / Instantaneous 2 Trape: LC58 w(qLAB) Wind Transversely to Ridge (night) Short-term / Instantaneous 2one G; cpe = -1.11 3 Trape: LC59 w(pA) Wind Parallet to Ridge (R) Short-term / Instantaneous 2one G; cpe = -0.54 4 Trape: LC59 w(pA) Wind Parallet to Ridge (R) Short-term / Instantaneous 2one I; cpe = -0.56 5 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 2one I; cpe = -0.56 7 Trapezoidal Load z Left Beam -1.57 -1.57 0.00 0.52 2one I; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 2one I; cpe = -0.56		LC52	w(q,I,BB)	Wind 1	ransversely to	o Ridge	(left)(BB)	Short-term	/Instantane	eous				
No. LC54 Wind Transversely to Ridge (right)(A) Short term / Instantaneous ance Across Tot I LC55 w(q.I.A8) Wind Transversely to Ridge (right)(AB) Short term / Instantaneous ance Across Tot I Concert LC56 w(q.I.AB) Wind Transversely to Ridge (right)(AB) Short term / Instantaneous ance Across Tot I Concert LC56 w(q.I.AB) Wind Transversely to Ridge (right)(AB) Short term / Instantaneous Image: Concert (right)(AB) Image: Concert (right)(AB) Short term / Instantaneous Image: Concert (right)(AB)	Load	LC53	w(q,I,AB)	Wind 1	ransversely to	Ridge	(left)(AB)	Short-term	/ Instantane	eous				LC
No. Los LCSS w(q.IAR) Wind Transversely to Ridge (right)(BA) Short-term / Instantaneous 1 Length Comment 1 Concer LCS6 w(q.IAR) Wind Transversely to Ridge (right)(BA) Short-term / Instantaneous 1 Zone G; cpe = -1.11 2 Trape: LCS8 w(q.IAR) Wind Transversely to Ridge (right)(BA) Short-term / Instantaneous 2 Zone G; cpe = -1.11 3 Trape: LCS9 w(q.IAR) Wind Parallel to Ridge (R) Short-term / Instantaneous 2 Zone G; cpe = -0.54 5 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 2 Zone I; cpe = -0.56 7 Trapezoidal Load z Left Beam -1.57 -1.57 0.00 0.52 2 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 2 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 Zone I; cpe = -0.56 1		LC54	w(q,I,BA)	Wind 1	ransversely to	Ridge	(left)(BA)	Short-term	/ Instantane	eous		- T.	J	4
Los LCS6 w(qLBB) Wind Transversely to Ridge (right)(BB) Short-term / Instantaneous 1 Concer LCS7 w(qLBB) Wind Transversely to Ridge (right)(BB) Short-term / Instantaneous	No	LC55	w(q,r,AA)	Wind 1	ransversely to	o Ridge	(right)(AA)	Short-term	/ Instantane				C	
Content LCS7 w(qr.AB) Wind Transversely to Ridge (right)(AB) Short-term / Instantaneous Zone G; cpe = -1.11 3 Trapez LCS8 w(qr.AB) Wind Transversely to Ridge (right)(BA) Short-term / Instantaneous Zone G; cpe = -1.11 4 Trapez LCS9 w(pA) Wind Parallel to Ridge (B) Short-term / Instantaneous Zone G; cpe = -0.54 5 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 Zone I; cpe = -0.56 7 Trapezoidal Load z Left Beam -1.57 -1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 Zone I; cpe = -0.56 1 68 1.568 1.928 1.508 Zone I; cpe = -0.56	LUC	LC56	w(q,I,BB)	Wind 1	ransversely to	o Ridge	(right)(BB)	Short-term	/Instantane	eous	1 %			_
3 Trape: LC59 Wind Transverse or Indge (Ight)(DA) Short-term / Instantaneous 4 Trape: LC59 Wind Parallel to Ridge (A) Short-term / Instantaneous Zone G; c pe = -0.54 5 Trape: LC59 Wind Parallel to Ridge (B) Short-term / Instantaneous Zone H; c pe = -0.54 5 Trape: Instantaneous Zone H; c pe = -0.54 Zone H; c pe = -0.56 6 Trape: codal Load z Left Beam -1.57 -1.57 0.52 6.68 Zone I; c pe = -0.56 7 Trape: codal Load z Left Beam -1.57 -1.57 0.00 0.52 Zone I; c pe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 Zone I; c pe = -0.56 1 68 1.568 1.928 1.508 0.622 Zone I; c pe = -0.56	Concer	LC57	w(q,r,AB)	Wind 1	ransversely to	Ridge	(right)(AB)	Short-term	/ Instantane	eous	<u> </u>			
4 Trape: LCS0 w(p,A) Wind Paralel to Nidge (A) Short-em / Instantaneous Image: LCS0 w(p,B) Zone H; cpe = -0.54 5 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 Zone I; cpe = -0.56 7 Trapezoidal Load z Left Beam -1.57 -1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -1.57 -1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -1.57 1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -1.57 1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -1.57 1.57 0.00 0.52 Zone I; cpe = -0.56 1 68 1.568 1.928 1.508 1.694 0.622 1.694		LC58	w(q,I,BA)	Wind 1	ransversely to	Ridge	(right)(BA)	Short-term	/ Instantane	eous	<u> </u>			
5 Trape LC80 with a rate to hidge (b) Dirichem / instanteous Zone J; cpe = 0.69 6 Trapezoidal Load z Left Beam -1.57 -1.57 0.52 6.68 Zone I; cpe = 0.56 7 Trapezoidal Load z Left Beam -1.57 -1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 Zone I; cpe = 0.56 8 Concentrated Load z Left Beam -0.66 0.00 Zone I; cpe = 0.56 1 68 1.568 1.928 1.508 0.622 Zone I; cpe = 0.56		LC59	w(p,A)	Wind F	Parallel to Ridg	je (A)		Short-term	/ Instantane	eous				
5 Trapezoldal Load 2 Left Beam -1.53 -1.53 0.56 7.56 1 20nel; cpe = -0.56 6 Trapezoldal Load z Left Beam -1.57 -1.57 0.52 6.68 1 Zonel; cpe = -0.56 7 Trapezoldal Load z Left Beam -1.57 -1.57 0.00 0.52 1 Zonel; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 1 Zonel; cpe = -0.56 1 1.57 1.57 1.50 0.00 0.52 2 Zonel; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 2 Zonel; cpe = -0.56 1 68 1.568 1.928 1.508 1.694 0.622 1		LC60	w(p,B)	Wind F	arallel to Rido	je (B)		Short-term	/ Instantane	eous				
7 Trapezoidal Load z Left Beam -1.57 -1.57 0.00 0.52 Zone I; cpe = -0.56 8 Concentrated Load z Left Beam -0.66 0.00 Zone I; cpe = -0.56 7 Trapezoidal Load z Left Beam -0.66 0.00 Zone I; cpe = -0.56 7 1.694 0.622	5 Trapez	oldal L	oad											
8 Concentrated Load z Left Beam -0.66 0.00 Image: Concentrated Load Zone I; cpe = -0.56 1 1.568 1.928 1.508 1.694 0.622 Image: Concentrated Load Image: C	- mapor			2										
1.568 1.568 1.508 1.508				2				-1.57		0.52				
1.568 1.928 1.508 1.694 0.622	8 Concen	trated	Load :	<u>د</u>	Left Beam		-0.66		0.00				Zone I; c pe = -0.56	
1.315 / 1.315 /		1		1	.568		1.928		1.508		1.1.296		,4	a ^X

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*.

out Data Frame Type and Material	1.4 Control Parameters	
- Geometry	Design of	Special Settings for Glulam
Loads Control Parameters	Static equilibrium (EQU) Ultimate limit state (STR) Serviceability limit state	Effect of Cross-Section Dimensions on Material Properties Increase fm,k and ft,p,k
	Fire protection	Support Modeling
	Fire Resistance Class: © F 30-8 © F 60-8 @ F 90-8	Left support: Horizontal fixed Horizontal fixed Right support: Horizontal fixed Column
	F 45 ☆ (min) Charring:	Horizontal free
	v	Calculation Parameters
	To Display ▼ Support forces	Generate supplementary combinations from favorable permanent actions Distribute permanent load span-by-span
	Support forces Support compression	Usuble permanent load span-by-span Image: Span by the s

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

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.

Iltimate Limit State Desig	 1.1 General Data 		
lata neral Data			Supplementary Examination
tions tion Categories 1			Reduce possible result combinations by examining RFEM results
tion Categories 2			From automatically defined combinations
	Combination Rules Acc.	to Standard / Annex	From result combinations:
	EN 1990 + EN 199	5 👻 🔳 DIN	
	Generating for Design S	Situations	Settings for Combinations Acc. to EN 1990
	Static equilibrium:	Basic combination E Accidental E. Accidental - Snow E Seismic E	Apply combination rule: Equation 6.10 Equations 6.10
	✓ Ultimate limit state:	Basic combination U Accidental U Accidental - Snow U Seismic U	Accidental combination according to Equations 6.11a and 6.11b Apply combination I The The According to the A
	Serviceability limit state:	Characteristic S Frequent S Quasi-permanent S	Setting for Timber Structures
	 Generate supplement from favorable permutification Do not treat togethe as collateral actions For wind as governing 	anent actions	Deformation coefficient kdef : 0.80 v kd Numbering Starting number for generated · Result combination: 1 kd

Figure 13.10: Add-on module RF-COMBI

Calculation

We create combinations by clicking the [Calculation] button.

RF-COMBI - [01.frm.rh2] File Settings Help																x
CA1 · Ultimate Limit State Desig 👻	2.2 Resu	lt Con	bination													
	2.2 11050	A	В	С	D	E	F	G	H		J	K	L	M	N	
Input Data General Data		~	Design	<u> </u>	0			u			0	IX.		141	IN	_
- General Data - Actions	RC	Apply	Situation	LDC	LC1	LC21	LC22	LC41	LC42	LC43	LC51	LC52	LC53	LC54	LC55	U E
- Action Categories 1	RC1		UB	Permanent	1.35				-							
Action Categories 2	RC2		UB	Short-term	1.35			1.50	-							
Results	RC3	$\overline{\square}$	UB	Short-term	1.35	-	-	-	1.50		-	-		-	-	
- Result Combinations by Actions	RC4	$\overline{\square}$	UB	Short-term	1.35	•	-		-	1.50	-	-	•			
-Result Combinations	RC5	Ø	UB	Short-term/Instantaneous	1.35	-	-	1.50	-		0.90	-		-	-	
Result Combinations - Reduced	RC6		UB	Short-term/Instantaneous	1.35	•	-	1.50	-	-		0.90	•	-	•	
Tresar company and the Treduced	RC7	Ø	UB	Short-term/Instantaneous	1.35	-	-	1.50	•	-	-	-	0.90	-	-	
	RC8	Ø	UB	Short-term/Instantaneous	1.35	-	-	1.50	•	-	-	-	•	0.90	-	
	RC9	√)	UB	Short-term/Instantaneous	1.35	-	-	1.50	-	-	-	-	•	-	0.90	
	RC10	V	UB	Short-term/Instantaneous	1.35	-	-	1.50	-	-	-	-	-	-	-	0
	RC11	V	UB	Short-term/Instantaneous	1.35	-	-	1.50	-	-	-	-	-	-	-	
	•															
	Load C	ases in		d Result Combination RC5 ad Case Description		Action		tegory	Leadir	na K					Combin	
	LC1	C-K		Roof Finishes		AC1		1.A		ig K 1.				esi NH	1.35	Ces
	LC41		weight +			AC1 AC3		4.B		1					1.35	
	LC51			sely to Ridge (left)(AA)		AC3		4.B						-	0.90	
	LCOT	VVID	a Transver	sely to Hidge (lett)(AA)		AC4	_	э.		1.	00 1.	50 U.	60	-	0.90	
	Leadir												ОК		Canc	
													wix.		Carlo	

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.

out Data	2.1 Resu	ult Combinations					
 Frame Type and Material 		A	В	С	D	E	F
Geometry	_			Design		Factor	Max.
Loads	RC	RC Description	Load Cases	Situation	LDC	k mod	Design
 Control Parameters 		Ultimate Limit State De	sign				
esults	RC1	g	1.35*LC1	UB	Permane	0.600	0.56
 Result Combinations 	RC2	g+s	1.35*LC1 + 1.50*LC41	UB	Short-ter	0.900	0.65
- Design - All	RC3	g + s(l)	1.35*LC1 + 1.50*LC42	UB	Short-ter	0.900	0.58
 Design by Member 	RC4	g + s(r)	1.35*LC1 + 1.50*LC43	UB	Short-ter	0.900	0.58
 Design by Location X 	RC5	g + s + w(q,I,AA)	1.35*LC1 + 1.50*LC41 + 0.90*LC51	UB	Short-ter	1.000	0.52
- Support Forces	RC6	g + s + w(q,I,BB)	1.35*LC1 + 1.50*LC41 + 0.90*LC52	UB	Short-ter	1.000	0.61
	RC7	g + s + w(q,I,AB)	1.35*LC1 + 1.50*LC41 + 0.90*LC53	UB	Short-ter	1.000	0.56
	RC8	g + s + w(q,I,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,I,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,I,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,I,AA)	1.35*LC1 + 1.50*LC42 + 0.90*LC51	UB	Short-ter	1.000	0.45
	RC16	g + s(l) + w(q,I,BB)	1.35*LC1 + 1.50*LC42 + 0.90*LC52	UB	Short-ter	1.000	0.55
	RC17	g + s(l) + w(q,I,AB)	1.35*LC1 + 1.50*LC42 + 0.90*LC53	UB	Short-ter	1.000	0.49
	RC18	g + s(l) + w(q,I,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,I,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,I,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
			隆 🥙 Max: 0.65	≦1 ම	Y		

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.

2

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).

Calculation



13.3.2 Designs

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

nput Data	2.2 Desid	gn - All											
 Frame Type and Material 		A	В	С	D					Е			
- Geometry	Member												
Loads	No.	X [m]	RC	Ratio					Design ad	cc. to	Formula		
Control Parameters	1	0.00	RC2	0.01	1 ≤1				Compression				
Results	1	0.00	RC2	0.44	≤1	111) Cross-se	ection resist	tance -	Shear due to	shea	r force Vz ac	c. to 6.1.7	
- Result Combinations	1	3.12	RC2		≤1							nd compressio	
— Design - All	4	9.89	RC58) ≤1							ension edge) a	
 Design by Member 	1	3.12	RC2		} ≤1							ompression ed	lge) and com
- Design by Location X	1	0.00	RC2		} ≤1	303) Stability							
Support Forces	1	2.50	RC2		j ≤1	323) Stability							
	4	4.16	RC2		} ≤1	341) Stability							
	1	3.12	RC2	0.65	j ≤1	361) Stability	- Design at	t finger f	frame joint ac	c.to	DIN EN 1995	5-1-1/NA:2010	-12, NA.11.3
			Max	0.65	5 ≤1	9			۲		2		
		diate result		00 m - RC	:2: g +	s					Rectangle 3		
		Section Dat	-							-	+	30.00	-+
		n Internal Fo	rces										
	Design												
	0.10	ar Force		Vz	,d	53.88							
		ss-Section V		b		30.00					35.00		
		ss-Section H		h		35.00	cm		0.4 7 (0)		35		У
		ck Influence	Factor	ka		0.714	2		6.1.7 (2)	=			
		ective Area		Ae		750.00	Cm ² N/mm ²	-		=			
				τd			N/mm ²		171 T-1-0	-1	+_₽		
		ar Strength		fv,			IN/mm-	-	[7], Tab.2 Tab. 2.3	-		÷	
		tial Factor dification Fa		YM I		1.300			Tab. 2.3	-		z	
		anounon ru	CLOF	k m		0.900	N/mm ²	-	Eq. (2.14)	-			[0
		C1 11				2.42	IN/mm ^		Eq. (2.14)				X 🗭 🛛
	- She	ar Strength aign Ratio		n 197	u	0.44		≤1	Eq. (6.13)	-	0	4	X 🛟 🕻

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

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

2



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.

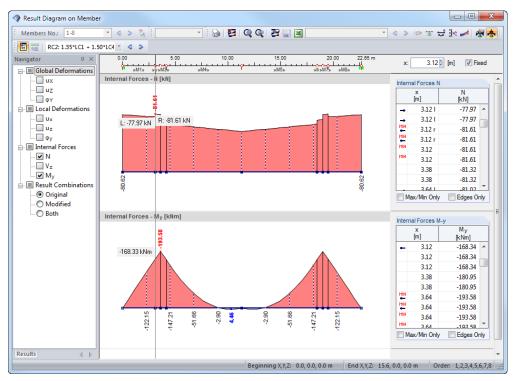


Figure 13.14: Distribution of internal forces N and M_y in RC2

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

	A	B	С	D					Е	
Member	Location									
No.	X [m]	RC	Ratio					Design ac	c.to	o Formula
1	0.00	RC2	0.	04 ≤1	102) Cross-se	ection resistar	nce - O	Compression	alon	ng the grain acc. to 6.1.4
1	0.00	RC2	0.	44 ≤ 1	111) Cross-se	ection resistar	nce - S	Shear due to	shea	ear force Vz acc. to 6.1.7
1	3.12	RC2	0.	31 ≤1	183) Cross-se	ection resistar	nce - l	Jniaxial bend	ling a	about y-axis and compression on edge pa
4	9.89	RC58	0.	10 ≤ 1	193) Cross-se	ection resistar	nce - l	Jniaxial bend	ling a	about y-axis (tension edge) and compress
1	3.12	RC2	0.	38 ≤ 1	203) Cross-se	ection resistar	nce - l	Jniaxial bend	ling a	about y-axis (compression edge) and com
1	0.00	RC2	0.	08 ≤1	303) Stability	- Axial compr	ression	acc. to 6.3.	2 - b	buckling about both axes
1	2.50	RC2	0.	35 ≤1	323) Stability	- Uniaxial ber	nding	and compres	sion	n acc. to 6.3.2
4	4.16	RC2	0.	13 ≤1	341) Stability	- Uniaxial be	ending	and compres	ssion	n acc. to 6.3.3
1	3.12	RC2	0.	65 ≤1	361) Stability	- Design at fi	inger fr	ame joint ac	c.to	DIN EN 1995-1-1/NA:2010-12, NA.11.3
nterme	diate result	s - X: 3.1	l2 m - F	C2: g	+ s					Rectangle 30/70
	diate result mal Force (0			RC2: g	+ s 77.97	kN				Rectangle 30/70
— Noi		Compressio		la j						
— Noi — Cro	rmal Force (C	Compressio Area	on) N A	la j	77.97 2100.00			Eq. (6.36)		
— Nor — Cro — Cor	mal Force (0 ss-Sectional	Compressio Area ress	on) N A	la V	77.97 2100.00 0.37	cm ²		Eq. (6.36) [7], Tab.2		
- Nor - Cro - Cor - Cor	mal Force (C ss-Sectional mpressive St	Compressio Area ress	on) N / c f	ld \ ic,0,d	77.97 2100.00 0.37	cm ² N/mm ²				
Nor Cro Cor Cor Par	mal Force (0 ss-Sectional mpressive St mpressive St	Compressic Area ress rength	on) N A c f 7	ld (ic,0,d c,0,k	77.97 2100.00 0.37 26.50	cm ² N/mm ²		[7], Tab.2		
Nor Cro Cor Cor Par Mo	mal Force (C ss-Sectional npressive St npressive St tial Factor	Compressic Area ress rength ctor	אר) אר קיין אריין אריין אריין אריין אריין אריין אריין אריין אריין אריין	ld (c,0,d c,0,k M	77.97 2100.00 0.37 26.50 1.300	cm ² N/mm ² N/mm ²		[7], Tab.2 Tab. 2.3		
Nor Cro Cor Cor Par Mo Cor	mal Force (C ss-Sectional mpressive St mpressive St tial Factor dification Fa	Compressic Area ress rength ctor rength	אר) א ק ק א א ק	ld ic,0,d c,0,k M imod	77.97 2100.00 0.37 26.50 1.300 0.900	cm ² N/mm ² N/mm ²	≤1	[7], Tab.2 Tab. 2.3 Tab. 3.1		
Not Cro Cor Par Mo Cor Des Cut	mal Force ((iss-Sectional mpressive St tial Factor dification Fa mpressive St sign - N-ratio :to-Grain An	Compressic Area ress rength ctor rength (Compres	אר) א ק ק א א ק	Id (c,0,d c,0,k M (mod c,0,d	77.97 2100.00 0.37 26.50 1.300 0.900 18.35	cm ² N/mm ² N/mm ²	≤1	[7], Tab.2 Tab. 2.3 Tab. 3.1 Eq. (2.14)		
Nor Cro Cor Par Mo Cor De: Cut Mo	mal Force ((ss-Sectional mpressive St tial Factor dification Fa mpressive St sign - N-ratio :to-Grain An ment	Compressic Area ress rength ctor rength (Compres	on) N A c f f y k f sion) c	Id (c,0,d c,0,k M (mod c,0,d	77.97 2100.00 0.37 26.50 1.300 0.900 18.35 0.00	cm ² N/mm ² N/mm ² N/mm ²	≤1	[7], Tab.2 Tab. 2.3 Tab. 3.1 Eq. (2.14)		
Nor Cro Cor Par Mo Cor De: Cut Mo Bez	mal Force ((ss-Sectional mpressive St tial Factor dification Fa mpressive St sign - N-ratio -to-Grain An ment am Width	Compressic Area ress rength ctor rength (Compres	Image: mail of the second se	اط (c,0,d c,0,k M (mod c,0,d (ر ال	77.97 2100.00 0.37 26.50 1.300 0.900 18.35 0.00 6.40 168.34 0.30	cm ² N/mm ² N/mm ² N/mm ² * kNm m	≤1	[7], Tab.2 Tab. 2.3 Tab. 3.1 Eq. (2.14)		
Nor Cro Cor Par Mo Cor Des Cut Mo Bes Bes	mal Force ((ss-Sectional mpressive St tial Factor dification Fa mpressive St sign - N-ratio -to-Grain An ment am Width am Depth	Compressic Area ress rength ctor rength (Compres	Implement Implement	اط (c,0,d c,0,k M (mod c,0,d (ر ال	77.97 2100.00 0.37 26.50 1.300 0.900 18.35 0.00 6.40 168.34 0.30 0.30	cm ² N/mm ² N/mm ² N/mm ² ° kNm m	≤1	[7], Tab.2 Tab.2.3 Tab.3.1 Eq. (2.14) Eq. (6.19)		
Nor Cro Cor Par Mo Cor Des Cut Mo Bes Bes Bes	mal Force ((ss-Sectional mpressive St tial Factor dification Fa mpressive St sign - N-ratio -to-Grain An ment am Width	Compression Area ress rength ctor rength (Compres gle	Image: Non (Image) Image:	اط (c,0,d c,0,k M (mod c,0,d (ر ال	77.97 2100.00 0.37 26.50 0.900 18.35 0.00 6.40 168.34 0.30 0.70 0.70 6.87	cm ² N/mm ² N/mm ² N/mm ² ° kNm m	≤1	[7], Tab.2 Tab. 2.3 Tab. 3.1 Eq. (2.14)		

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.

	A	В	С	D					Е	
Member	Location									
No.	X [m]	RC	Ratio					Design ac	c.to	to Formula
1	0.00	RC2	0.04	≤1	102) Cross-se	ection resist	ance -	Compression	alon	ng the grain acc. to 6.1.4
1	0.00	RC2	0.44	≤1	111) Cross-se	ection resist	ance -	Shear due to	she	ear force Vz acc. to 6.1.7
1	3.12	RC2	0.31	≤1	183) Cross-se	ection resist	ance -	Uniaxial bend	ling	about y-axis and compression on edge pa
4	9.89	RC58	0.10	≤1	193) Cross-se	ection resist	ance -	Uniaxial bend	ling	about y-axis (tension edge) and compressi
	3.12	RC2	0.38	≤1	203) Cross-se	ection resist	ance -	Uniaxial bend	ling	about y-axis (compression edge) and comp
1	0.00	RC2	0.08		303) Stability	- Axial com	pressio	n acc. to 6.3.	2 - b	buckling about both axes
1	2.50	RC2	0.35							n acc. to 6.3.2
4	4.16	RC2		≤1	341) Stability	– Uniaxial b	ending	and compres	ssior	n acc. to 6.3.3
1	3.12	RC2	0.65	≤1	361) Stability	- Design at	fingerf	rame joint ac	c.to	DIN EN 1995-1-1/NA:2010-12, NA.11.3
nterme	diate result	s - X:3.	12 m - RC	2: g +	s					Rectangle 30/70
	diate result ment	s - X: 3.1	L2 m - RC		s 168.34	kNm				Rectangle 30/70
— Moi		s - X: 3.1								-
Mor Bea	ment	s - X: 3.:	My		168.34 0.30 0.70	m m				-
Mor Bea Bea Ber	ment am Width am Depth nding Stress		My b h		168.34 0.30 0.70 6.87	m m N/mm ²		Eq. (6.37)		-
Mor Bea Bea Ber Ber	ment am Width am Depth nding Stress nding Streng	th	My b h	,d ,d .α.d	168.34 0.30 0.70 6.87 32.00	m m N/mm ² N/mm ²		[7], Tab.2		+ <u>+</u>
Mor Bea Bea Ber Ber Ber	ment am Width am Depth ading Stress ading Streng ading Streng	th	My b h σm fm, fm,	,d .α.d y,k y,d	168.34 0.30 0.70 6.87 32.00 22.15	m N/mm ² N/mm ² N/mm ²		[7], Tab.2 Eq. (2.14)		-
Mon Bea Bea Ber Ber Ber Cor	ment am Width am Depth nding Stress nding Streng nding Streng npressive St	th th rength	My b h σm fm, fo,s	,d .α.d y,k y,d 0,k	168.34 0.30 0.70 6.87 32.00 22.15 3.00	m N/mm ² N/mm ² N/mm ² N/mm ²		[7], Tab.2 Eq. (2.14) [7], Tab.2		+ <u>+</u>
Mor Bea Ber Ber Ber Cor Cor	ment am Width am Depth nding Stress nding Streng npressive St ear Strength	th th rength	My b h σm fm.; fo.s fv.i	,d .α.d y,k y,d 0,k	168.34 0.30 0.70 6.87 32.00 22.15 3.00 3.50	m M/mm ² N/mm ² N/mm ² N/mm ² N/mm ²		[7], Tab.2 Eq. (2.14) [7], Tab.2 [7], Tab.2		+ <u>+</u>
Moi Bez Ber Ber Ber Ber Cor She She	ment am Width am Depth nding Stress nding Streng nding Streng npressive St ear Strength ear Strength	th th rength	My b h σm fm, fo,s fv,i fv,c	,d α.d y,k y,d 0,k	168.34 0.30 0.70 6.87 32.00 22.15 3.00 3.50 2.42	m M/mm ² N/mm ² N/mm ² N/mm ² N/mm ² N/mm ²		[7], Tab.2 Eq. (2.14) [7], Tab.2 [7], Tab.2 Eq. (2.14)		+ <u>+</u>
Moi Bea Ber Ber Ber Cor She She Cor	ment am Width am Depth nding Stress nding Streng nding Streng npressive St ear Strength ear Strength npressive St	th th rength rength	My b fm, fm, fo,s fv,t fv,c fo,s	,d α.d y,k y,d 0,k	168.34 0.30 0.70 6.87 32.00 22.15 3.00 3.50 2.42 2.08	m M/mm ² N/mm ² N/mm ² N/mm ² N/mm ²		[7], Tab.2 Eq. (2.14) [7], Tab.2 [7], Tab.2		+ <u>+</u>
Moi Bea Ber Ber Cor She Cor Cor Cor	ment m Width am Depth nding Stress nding Streng npressive St ear Strength ar Strength npressive St to-Grain An	th th rength rength	My b h σm fm, fa,s fv,i fv,c fc,s a	,d .α.d y,k y,d 0,k : i 0,d	168.34 0.30 0.70 6.87 32.00 22.15 3.00 3.50 2.42 2.08 6.40	m M/mm ² N/mm ² N/mm ² N/mm ² N/mm ² N/mm ²		[7], Tab.2 Eq. (2.14) [7], Tab.2 [7], Tab.2 Eq. (2.14) Eq. (2.14)		
Mon Bea Bea Ber Ber Cor She Cor Cor Cut	ment m Width am Depth nding Stress nding Streng npressive St ear Strength ar Strength npressive St to-Grain An-	th th rength gle	My b h σm fm, fa,S fv,i fc,S fc,S c, fc,S c, sk km	,d .α.d y,k y,d 0,k : i 0,d	168.34 0.30 0.70 6.87 32.00 22.15 3.00 3.50 2.42 2.08 6.40 0.820	m m N/mm ² N/mm ² N/mm ² N/mm ² N/mm ² N/mm ² °		[7], Tab.2 Eq. (2.14) [7], Tab.2 [7], Tab.2 Eq. (2.14) Eq. (2.14) Eq. (2.14) Eq. (6.40)		+ <u>10.00</u>
Mor Bea Ber Ber Cor She Cor Cut Fac Ber	ment m Width am Depth nding Stress nding Streng npressive St ear Strength ar Strength npressive St to-Grain An	th th rength rength gle	My b h σm fm, fa,s fv,i fv,c fc,s a	,d .α.d y,k y,d 0,k : i 0,d	168.34 0.30 0.70 6.87 32.00 22.15 3.00 3.50 2.42 2.08 6.40 0.820	m M/mm ² N/mm ² N/mm ² N/mm ² N/mm ² N/mm ²	<u></u>	[7], Tab.2 Eq. (2.14) [7], Tab.2 [7], Tab.2 Eq. (2.14) Eq. (2.14)		

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

ession edge
$$k_{m,\alpha} = \frac{1}{\sqrt{1 + \left(\frac{f_{m,d}}{1.5 \cdot f_{v,d}} \cdot \tan \alpha\right)^2 + \left(\frac{f_{m,d}}{f_{c,90,d}} \cdot \tan^2 \alpha\right)^2}} = 0.82$$
$$\left(\frac{\sigma_{c,0,d}}{f_{c,0,d}}\right)^2 + \frac{\sigma_{m,d}}{k_{m,\alpha} \cdot f_{m,d}} = \left(\frac{0.39}{18.35}\right)^2 + \frac{6.87}{0.82 \cdot 22.15} = 0.38 < 100$$

Design ratio



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.

	A	В	C	D	E
Member	Location				
No.	X [m]	RC	Ratio		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 para
4	9.89	RC58	0.10	≤1	193) Cross-section resistance - Uniaxial bending about y-axis (tension edge) and compressio
1	3.12	RC2	0.38	≤1	203) Cross-section resistance - Uniaxial bending about y-axis (compression edge) and comp
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

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 $N_d = 78.64 \text{ kN}$
- Moment $M_{y,d} = 134.63 \text{ kNm}$

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
$$\beta_{5} = \sqrt{4 + \frac{\pi^{2} \cdot I_{5} \cdot s}{3 \cdot h \cdot I_{R}} + \frac{I_{5} \cdot N_{R} \cdot s^{2}}{I_{R} \cdot N_{5} \cdot h^{2}}} = \sqrt{4 + \frac{\pi^{2} \cdot 5.83E + 09 \cdot 7683}{3 \cdot 3641 \cdot 4.833E + 09} + \frac{5.83E + 09 \cdot 76209 \cdot 7683^{2}}{4.833E + 09 \cdot 85357 \cdot 3641^{2}}} = 4.143$$

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 $k_y = 0.5 \cdot (1 + \beta_c \cdot (\lambda_{rel,y} - 0.3) + \lambda_{rel,y}^2) =$ = $0.5 \cdot (1 + 0.1 \cdot (1.311 - 0.3) + 1.311^2) = 1.410$

$$k_{z} = 0.5 \cdot (1 + \beta_{c} \cdot (\lambda_{rel,z} - 0.3) + \lambda_{rel,z}^{2}) =$$

= 0.5 \cdot (1 + 0.1 \cdot (0.654 - 0.3) + 0.654^{2}) = 0.732

Dlubal

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

Normal Force (Compression)	Nd	78.64	kN		
Cross-Sectional Area	A	1890.00	cm ²		
Compressive Stress	σc,0,d	0.42	N/mm ²		Eq. (6.36
Equivalent Member Length	lef,y	14.99	m		
Equivalent Member Length	lef,z	3.64	m		
Radius of Inertia	iy	17.77	cm		
Radius of Inertia	iz	8.66	cm		
Slenderness Degree	λy	84.347			
Slenderness Degree	λz	42.051			
Relative Slendemess Ratio	λrel,y	1.312		> 0.3	Eq. (6.21
Relative Slendemess Ratio	λrel,z	0.654		> 0.3	Eq. (6.22
Straightness Factor	βc	0.100			Eq. (6.29
Auxiliary Buckling Coefficient	ky	1.411			Eq. (6.27
Auxiliary Buckling Coefficient	kz	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	fc,0,k	26.50	N/mm ²		[7], Tab.
Partial Factor	γM	1.300			Tab. 2.3
Modification Factor	kmod	0.900			Tab. 3.1
Compressive Strength	fc,0,d	18.35	N/mm ²		Eq. (2.14
Modulus of Elasticity	E0,05	11100.00	N/mm ²		[7], Tab.:
Reduction Factor	km	0.700			6.1.6
Moment	My,d	134.63	kNm		
Section Modulus	Wy	19845.00	cm ³		
Bending Stress	σm,y,d	0.70			
Bending Strength	fm,y,k	32.00	N/mm ²		[7], Tab.:
Bending Strength	fm,y,d	22.15	N/mm ²		Eq. (2.14
Design 1	η1	0.35		≤1	Eq. (6.23
Design 2	η2	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
$$\beta_{R}$$
 =

$$=\beta_{S}\cdot\sqrt{\frac{I_{R}\cdot N_{S}}{I_{S}\cdot N_{R}}}\cdot\frac{h}{s}=$$

$$=4.15 \cdot \sqrt{\frac{4.814 \text{E} + 09 \cdot 81610}{5.83 \text{E} + 09 \cdot 72860}} \cdot \frac{3641}{7683} = 1.89$$

$$\lambda_{\rm R} = \frac{l_{\rm ef,R}}{i_{\rm y}} = \frac{\beta_{\rm R} \cdot s}{i_{\rm y}} = \frac{1.89 \cdot 768 \text{cm}}{16.77 \text{cm}} = 86.6$$

Slenderness

Design ratio $\frac{f_{c,0}}{f_{c,\alpha}}$	$\frac{d}{d} \cdot \left(\frac{d}{k_c}\right)$	$\frac{5_{c,0,dS}}{5_{c,0,d}}$ +	$+\frac{\sigma_{m,ds}}{f_{m,d}}$	$\left(\frac{15.5}{12.4}\right) = \frac{15.5}{12.4}$	$\frac{59}{47} \cdot \left(\frac{0}{0.511}\right)$	$\frac{0.5}{1.15.59} + \frac{8.59}{18.83} = 0$
f _{c 0}	d (0	$\overline{D}_{c,0,dR}$	σ_{mdf}) 15.	59 (0	$\left(\frac{0.43}{9.15.59} + \frac{8.5}{18.83}\right) =$
<u> </u>	<u>~</u> ·	<i>c,o,an</i>	+ <u></u>	$\frac{1}{1} = \frac{1}{12}$	<u></u>	$\frac{1}{0} \frac{1}{10} \frac{1}{10} \frac{1}{10} =$
Γ _{c,α}	,d (Kc	R ⁺ ^T c,0,d	T _{m,d}) 12.4	47 (0.48)	9.12.29 18.83
			0.65	. 1		
	gover	ning:	0.65	< 1		
						1
Design Ratio	N	01.01	LAL	1	1	
Axial Force at Column	Ns	81.61				
Moment at Column Column Height	Ms h	168.34				
 Column Height Cross-Section Width 	n b	3.64				
 Cross-Section Vilatin Cross-Section Height, Colum 	-	30.00				
 Moment of Inertia for Column 		61.55				
 Buckling Length Factor for C 	-	583018.00	Cill 1		Tab. NA.23	
Column Slenderness	ps λs	4.150 85.047			Tab. INA.25	
Related Slendemess	λs λrel.cS	1.323			Eq. (6.21)	
- Factor		0.100			Eq. (6.21) Eq. (6.29)	
Buckling Coefficient	βc ks*	1.426			Eq. (6.23) Eq. (6.27)	
Buckling Factor	kes"	0.511			Eq. (6.27) Eq. (6.25)	
- Axial Force at Beam	Nr	72.86	L-N		Eq. (0.23)	
Moment at Beam	Mr	166.68				
- Column Length	S	7.68				
- Cross-Section Height, Beam	hr 0.65	0.58				
Moment of Inertia for Beam 5		481392.00				
 Buckling Length Factor for B 	-	1.891	Cill		Tab. NA.23	
Beam Slendemess	λR [*]	87.180			100.101.20	
Related Slenderness	λrel.cR*	1.356			Eq. (6.21)	
Buckling Factor	kR*	1.472			Eq. (6.27)	
Buckling Factor	k _c R*	0.489			Eq. (6.25)	
Angle	α	20.72				
Bending Strength	fm,k		N/mm ²			
Compressive Strength	fc,0,k		N/mm ²			
- Transversal Compressive Str			N/mm ²			
Shear Strength	f _{V,k}	3.50	N/mm ²			
Partial Factor	7M	1.300			Tab. 2.3	
Modification Factor	k mod	0.900			Tab. 3.1	
Bending Strength	fm,d	18.83	N/mm ²	11.3(NA6)	Eq. (2.14)	
Compressive Strength	fc,0,d		N/mm ²	11.3(NA6)	Eq. (2.14)	
 Transversal Compressive Str 	fc,90,d	2.08	N/mm ²		Eq. (2.14)	
 Shear Strength 	fv,d		N/mm ²		Eq. (2.14)	
 Transv. Compressive Strengt 	f _{c.α.d}		N/mm ²		Eq. (NA.152)	
Axial Stress	σc,0,dS		N/mm ²			
Bending Stress	σm,dS	8.59	N/mm ²			
Design Ratio 1	η1	0.65		≤1	Eq. (NA.147)	
 Axial Stress 	σc,0,dR		N/mm ²			
Bending Stress	σ _{m,dR}		N/mm ²			
 Design Ratio 2 	η2	0.64		≤1	Eq. (NA.147)	
Design Ratio		0.65		≤1	Eq. (NA.147)	•

Figure 13.19: Design of finger-jointed connection

Other Results Windows 13.3.3

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.

Dlubal



Purlin 4. 1

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.

Dlubal

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**.

System and Loads 14.1

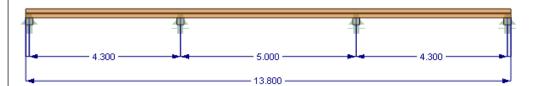


Figure 14.1: System

Model

b/d = 14/26 cm	
coniferous timber C24	
$I_1 = I_3 = 4.30m$	$I_2 = 5.00m$
6.0°	
inclined in roof plane	
nail connection d = 8 mm,	I = 280 mm as shown in Figure 14.2
	coniferous timber C24 $I_1 = I_3 = 4.30m$ 6.0° inclined in roof plane

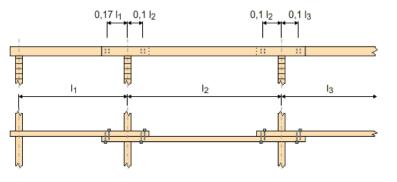


Figure 14.2: Purlin connection plan

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 ll	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.

File Settings Help	1.1 Beam Type and Material	
nput Data Beam Type and Material Geometry Cross-Section and Coupling Loads Control Parameters Effective Lengths	Material Poplar and Softwood Timber C24 Roof Type P Flat roof Monopitch roof G abled roof Beam Type P Continuous beam P Urin	
0 5 6	Comment	ncel

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.

file Settings Help	1.2 Geometry						
Beam Type and Material Geometry Cross-Section and Coupling Loads Control Parameters Effective Lengths	Number of spans n: 3 - V Total length of beam L: 13.800	 ☐ Identical span lengths ☆ [m] ○ Span length U: 	lengths Cantilever IRJE				
	Roof Parameters Supports Sup Building Height	port Spring Constants Releases F	lelease Spring Constants				
	Building Depth Distance of Purlin from Edge	B 40.000 m a 3.000 m	b				
	Purlin Distance Load Width Result Roof Inclination	bk 2.500 m	a 92	δ ¹ Η			
	Load Factor for Continuity Edge Purlin Purlin with Vertical Axis	k 1.000		B			
				0			
	Distance of Purlin from Edge a 3.000 m Purlin Distance b 2.500 m Load Width Result b_k 2.500 m Roof Inclination δ 6.00 ° Load Factor for Continuity k 1.000 Edge Purlin □						
	4.300			0			
0 5 5	Calculation Details Na		-сомві	OK Cancel			

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*.

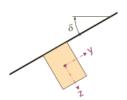
In the second tab Supports, we can define the span lengths and support widths.

Boof Parameters	Supports	Support Spring Constants	Beleases	Release Spring Constants

	А	B	C	D	E	F	G	H		J
Support	Location	Span Length	Support Width		D	isplaceme	nt		Rotation	
No.	X [m]	l [m]	b [cm]	Type of Support	uχ	uy	uΖ	φχ	ΦY	φz
1	0.100	4.300	20.00	Hinged	2	2	~	2		
2	4.400	5.000	20.00	Hinged free		1	1	2		
3	9.400	4.300	20.00	Hinged free		I	~	I		
4	13.700		20.00	Hinged free		2	~	2		
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**.

Input Data	1.3 Cross-Section and Coupling		
Beam Type and Material Geometry Cross Section and Coupling Loads Control Parameters Effective Lengths	Cross-Section Dimensions b: 14.00 + [cm] h: 26.00 + [cm] Other on end spans Dimensions b: 14.00 + [cm] h: 26.00 + [cm] Image: the spanse spans	Selected inner supports Supports: Coupling Length on Inner spans: 0.10 + x L [·] • Outer spans: 0.17 + x L [·]	Coupling Elements Nails n: 6 2 1 280 Type: Not prebored Prebored Ring and plate connectors and bolts Fastening system WT from SFS intec
0 7 3	Calculation Details Nat. Annex.		4.300

Figure 14.6: Window 1.3 Cross-Section and Coupling

The Couplings are provided for All inner supports.

6

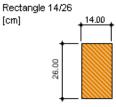
We don't modify the preset *Coupling Lengths*. We use the [Info] button shown on the left to check the connection plan of the purlin (see Figure 14.2, page 148).

We use Nails as Coupling Elements:

• Number *n*:

•

- Diameter D: 8 mm
 - Length *L*: **280 mm**
- Type: Prebored







14.2.4 Loads

This window manages the different types of loads.

Input Data Beam Type and Material	1.4 Loads									
	Perm	anent Action					Snow Load			
Geometry Geometry — Cross-Section and Coupling — Loads — Control Parameters — Effective Lengths		A	B	C	D		Altitude	A:	400 🔶 [m]	
	No.	Roof Layer	Thickness	3.4	_		Annuale	Α.	400 💌 [iii]	
- Control Parameters		Material	d [cm]	[kN/m ²]	Commer	t 😑	O Determine automatical	v		
- Effective Lengths	1	Trapezoidal Sheet		0.150		_	Snow load zone	SZ:		
	2	Fibrous Materials, B	2.00	0.040		_	Show load zone		2 🔻	
	3	Vapour Barrier	04.00	0.020		_	Topography type	TT:	Normal 🔹	•
	4	Steinwolle Sparren	24.00	0.240						
	5	Sparren		0.100						
	Roof	structure	gk,2"∶	0.550 🜲	[kN/m ²]	RA				
			gk,2 :	1.375 🜲	[kN/m]	BA	Define snow load	sk':	1.683 ÷ [kN/m ²]	
	Poor	n self-weight	gk.1 :		 	BA	manually	Sk :	4.184 🔶 [kN/m]	j
	(aver		<u>9к,</u> 1.	0.200 ≑	[kN/m]	RA		ък.	4.104 v [Kia/iii]	
	· · ·				1		Wind Load			
			gk:	1.575 ≑	[kN/m]	RA	Determine automatical			
	Impos	ed Load					Terrain category	TC:	Category II	•
	Impo	sed load	pk:	0.000 🔶	[kN/m ²]	BA	 Fundamental basic 	10.	Calegoly II	יע
			pk:	0.000		ΒА	wind velocity	νь,0:	22.5 🔶 [m/s]	
			PR.	0.000	Iron	500	Fundamental basic		THE PART IN A	
		sed load category rding to EN 1991-1-1:		H -]		velocity pressure	qь,0 :	0.316 (kN/m ²)	(
					_		Body shells with per	meable	e walls	
	Servi	ce Class	Show P	Generated Lo	ade		Define peak velocity	q(z)':	0.588 + [kN/m ²]	
	SECI	2 - 🚯		resp.			pressure manually			l
	020		Define	Additional Lo	ads	œ		q(z):	1.463 [KN/M]	

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 Determined 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.



0

....

Program RX-TIMBER © 2013 Dlubal Software GmbH

. Dlubal

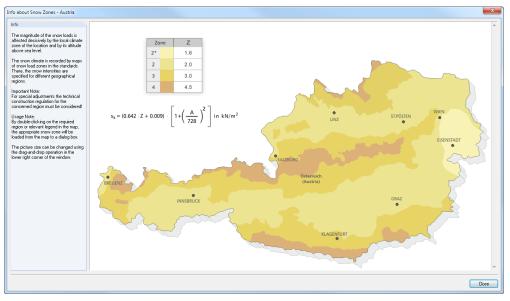


Figure 14.8: Map showing snow load zones for Austria

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.

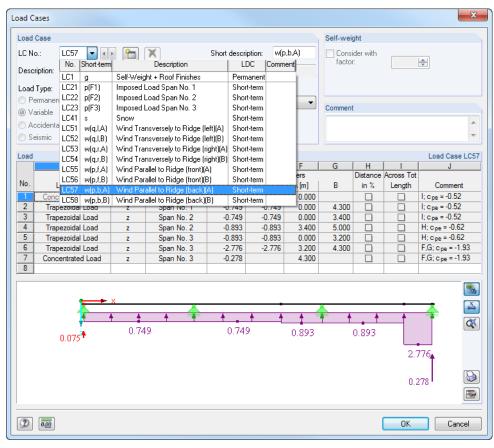


Figure 14.9: Dialog box Load Cases

Show Generated Loads... resp. Define Additional Loads...



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**.

ut Data	1.5 Control Parameters	
Beam Type and Material Geometry	Design of	Special Settings for Timber
Cross-Section and Coupling Loads Control Parameters Effective Lengths	Static equilibrium (EQU) Ultimate limit state (STR) Serviceability limit state	Effect of Cross-Section Dimensions on Material Properties Increase fm,k and ft,0,k
	Precamber wo: 0.0 (mm) Fire protection	According to 3.3(3) for glularn with h < 600 mm (bending) or b < 600 mm (tension)
	Fire Resistance Class: @ F 30-8	
	V	Calculation Parameters ✓ Moment redistribution acc. to 0NORM EN 1990 Redistribution: 10.000
	To Display	from favorable permanent actions
	Support forces Deformations Support compression	Distribute permanent load span-by-span Number of member - Result diagrams: 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.

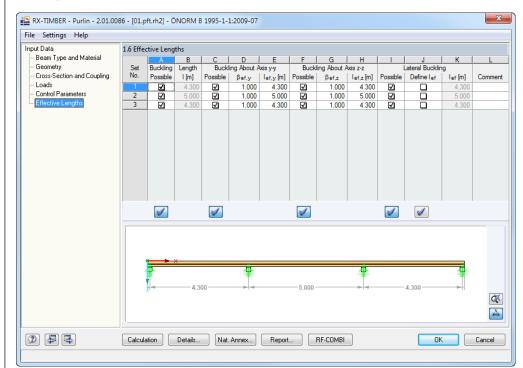


Figure 14.11 Window 1.6 Effective Lengths

14.2.7 Details

We click the [Details] button to open a dialog box offering specific design settings.

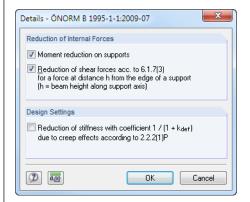


Figure 14.12: Dialog box Details

In this dialog box, we tick the options *Moment reduction on supports* and *Reduction of shear forces*.

Details...



14.2.8 RF-COMBI

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*.

A1 - Ultimate Limit State Desig 🦄	 1.2 Load Ca 	ases in Actions					
Input Data General Data Actions Action Categories	Actions AC1	Self-weight + Roof structure	•	فالمناسب الم	escription:	Note: The button 'l	Create New Actio
	AC2 AC3	Imposed Loads Snow		Wind Lo	•	"actions", w	can be used to define "actions", which are independent of each other.
	AC4	Wind Loads	=	Action co	omment:		bles below, load hen be allocated
		<u>7</u>	•			In Table 1.3, then allocate correspondin Categories".	
	Existing Lo	ad Cases		Load Ca	ses in Action AC4		
			<u> </u>	No.	Load Case Description	Alternative	LDC
		l f		LC51	Wind guer zum First (links)(A)	Wind	Short-term
				LC52	Wind guer zum First (links)(B)	Wind	Short-term
			>	LC53	Wind quer zum First (rechts)(A)	Wind	Short-term
				LCEA	Wind quer zum First (rechts)(B)	Wind	Short-term
			>	LC55	Wind parallel zum First (vome)(A)	Wind	Short-term
				LC56	Wind parallel zum First (vome)(B)	Wind	Short-term
				LC57	Wind parallel zum First (hinten)(A)	Wind	Short-term
			4	LC58	Wind parallel zum First (hinten)(B)	Wind	Short-term
					·		E

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

Calculation

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.

nput Data	2.1 Resu	ult Combinations					
 Beam Type and Material 		A	В	С	D	E	F
Geometry				Design		Factor	Max.
 Cross-Section and Coupling 	RC	RC Description	Load Cases	Situation	LDC	k mod	Design
Loads		Ultimate Limit State De	sign				
- Control Parameters	RC1	q	1.35*LC1	UB	Permane	0.600	0.46
Effective Lengths	RC2	g+s	1.35*LC1 + 1.50*LC41	UB	Short-ter	0.900	1.03
esults	RC3	g + s + w(q,I,A)	1.35*LC1 + 1.50*LC41 + 0.90*LC51	UB	Short-ter	0.900	0.94
Result Combinations	RC4	g + s + w(q,I,B)	1.35*LC1 + 1.50*LC41 + 0.90*LC52	UB	Short-ter	0.900	1.03
- Design by Beam	RC5	g + s + w(q,r,A)	1.35*LC1 + 1.50*LC41 + 0.90*LC53	UB	Short-ter	0.900	0.91
- Design by Span	RC6	g + s + w(q,r,B)	1.35*LC1 + 1.50*LC41 + 0.90*LC54	UB	Short-ter	0.900	0.91
 Design by Location X 	RC7	g + s + w(p,f,A)	1.35*LC1 + 1.50*LC41 + 0.90*LC55	UB	Short-ter	0.900	0.96
	RC8	g + s + w(p,f,B)	1.35*LC1 + 1.50*LC41 + 0.90*LC56	UB	Short-ter	0.900	0.96
	RC9	g + s + w(p,b,A)	1.35*LC1 + 1.50*LC41 + 0.90*LC57	UB	Short-ter	0.900	0.96
	RC10	g + s + w(p,b,B)	1.35*LC1 + 1.50*LC41 + 0.90*LC58	UB	Short-ter	0.900	0.96
	RC11	g + w(q,I,A)	1.35*LC1 + 1.50*LC51	UB	Short-ter	0.900	0.16
	RC12	g + w(g,I,B)	1.35*LC1 + 1.50*LC52	UB	Short-ter	0.900	0.31
	RC13	g + w(q,r,A)	1.35*LC1 + 1.50*LC53	UB	Short-ter	0.900	0.10
	RC14	g + w(q,r,B)	1.35*LC1 + 1.50*LC54	UB	Short-ter	0.900	0.10
	RC15	g + w(p.f.A)	1.35*LC1 + 1.50*LC55	UB	Short-ter	0.900	0.18
	RC16	g + w(p,f,B)	1.35*LC1 + 1.50*LC56	UB	Short-ter	0.900	0.18
	RC17	g + w(p,b,A)	1.35*LC1 + 1.50*LC57	UB	Short-ter	0.900	0.18
	RC18	g + w(p,b,B)	1.35*LC1 + 1.50*LC58	UB	Short-ter	0.900	0.18
	RC19	g + s + w(q,I,A)	1.35*LC1 + 0.75*LC41 + 1.50*LC51	UB	Short-ter	0.900	0.52
	RC20	g + s + w(q,I,B)	1.35*LC1 + 0.75*LC41 + 1.50*LC52	UB	Short-ter	0.900	0.67
	RC21	g + s + w(g,r,A)	1.35*LC1 + 0.75*LC41 + 1.50*LC53	UB	Short-ter	0.900	0.46
	RC22	g + s + w(q,r,B)	1.35*LC1 + 0.75*LC41 + 1.50*LC54	UB	Short-ter	0.900	0.46
	RC23	g + s + w(p,f,A)	1.35*LC1 + 0.75*LC41 + 1.50*LC55	UB	Short-ter	0.900	0.54
	RC24	a + s + w(p.f.B)	1.35*LC1 + 0.75*LC41 + 1.50*LC56	UB	Short-ter	0.900	0.54
			🔉 💕 Max 🛛 1.03 >1		°		

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.

Input Data	2.2 Desi	gn by Bean	n											
- Beam Type and Material		A	В	С	D					Е				
Geometry		Location												
 Cross-Section and Coupling 	No.	X [m]	RC	Ratio					Design ac					
Loads	1	4.300	RC4		2 ≤1				Shear due to					
- Control Parameters	2	4.300	RC2		5 ≤1				Shear due to					
Effective Lengths	3	4.400	RC4		4 ≤1				Shear stress of					
Results 	4	4.400	RC2		6 ≤1				Shear stress of					
	5	5.400	RC23		2 ≤1				Uniaxial bend			6		
Design by Beam	6	1.871	RC4		5 ≤1				Biaxial bendin					
 Design by Span Design by Location X 	7	4.400	RC4		3 > 1				sion combinat					
Design by Location X	8	1.871	RC4	0.3	9 ≤1	311) Stability	- Uniaxial t	bending	about y-axis	without	compres	ision force ac	c. to 6.3.3	
		-Section Dat n Internal Fo								^		14.00	+	
		-Section Dat	а									14.00	-+	
	Desig Desig		rces							-	+	_		
		n Hatio ear Force		M	z.d	18.285	LIN							
		ear Force ass-Section V	Vialtia	v: b	z,d	18.285		-						
		ss-Section V		h		26.00		26.00		N				
		iss-section r ick Influence		k		26.00	Cill		6.1.7 (2)		26			У
		ective Area	ractor	A		243.88	cm ²	-	0.1.7 (2)	=				
		ear Stress		τα			N/mm ²							
		ear Strength			.k		N/mm ²		[8], Tab.1		+		_	
		tial Factor		71	1	1.300		-	Tab. 2.3			*		
	Mo	dification Fa	ctor	k	mod	0.900			Tab. 3.1					ſa
	- She	ear Strength		F _v	,d	2.1	N/mm ²	1	Eq. (2.14)					
	De	sian Ratio		η		0.52		≤1	Eq. (6.13)	-			l ⇒	d
	She	ear Strength		fv		2.1		≤1	Eq. (2.14)	Ţ	9		x F	

Figure 14.15: Window 2.2 Design by Beam

2



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 $V_{\rm z}$ according to 6.1.7

The maximum shear force of 19.00 kN occurs on the inner supports in RC4.

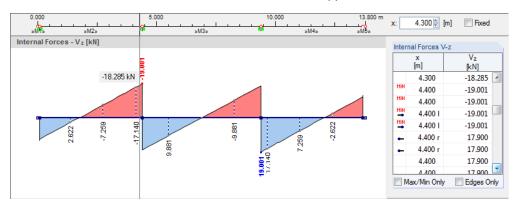


Figure 14.16: Distribution of shear force V_z in RC4

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

$$\frac{\tau_{\rm d}}{f_{\rm v,d}} = \frac{1.10}{2.15} = 0.51 < 1$$

2



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.

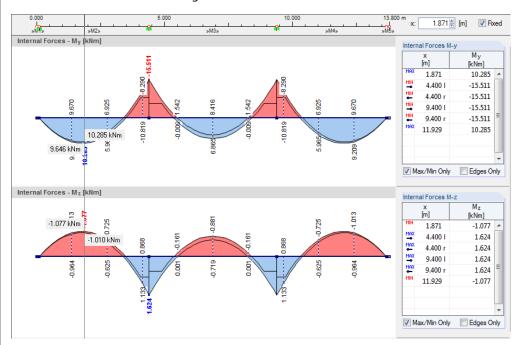


Figure 14.17: Distribution of moments My and Mz in RC4 with representation of original and modified values

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

	A	B	С	D					Е			
	Location											
No.	X [m]	RC	Ratio					Design ad	c.t	o Formula		
1	4.300	RC4	0.52	≤1	111) Cross-se	ection resist	ance -	Shear due to	she	ar force Vz a	acc. to 6.1.7	
2	4.300	RC2	0.05	≤1	112) Cross-se	ection resist	ance -	Shear due to	she	ar force Vy a	acc. to 6.1.7	
3	4.400	RC4	0.54	≤1	114) Cross-se	ection resist	ance -	Shear stress	on s	upport Vz ad	cc. to 6.1.7	
4	4.400	RC2	0.06	≤1	115) Cross-se	ection resist	ance -	Shear stress	on s	upport Vy a	cc. to 6.1.7	
5	5.400	RC23	0.02	≤1	151) Cross-se	151) Cross-section resistance - Uniaxial bending acc. to 6.1.6						
6	1.871	RC4	0.45		153) Cross-se							
7	4.400	RC4	1.03								nd Mz acc. to 8	
8	1.871	RC4	0.39	≤1	311) Stability	- Uniaxial b	ending	about y-axis	with	out compres	ssion force acc. t	to 6.3.3
erme	diate result	is - X:1.8	871 m - R0	04: g ·	+ s + w(q,I,B)					Rectangle	14/26	
	ediate result	:s - X:1.8				kNm				Rectangle	14/26	
Мо		rs - X:1.8	871 m - R0 My Mz,	d,	10.285	kNm kNm				Rectangle		
Mo Mo	ment		My	,d d	10.285	kNm cm ³				Rectangle		
Mo Mo See	ment ment	IS	My Mz	d d	10.285 1.077 1577.33 849.33	kNm cm ³ cm ³				Rectangle		
Mo Mo Ser Ser Ber	ment ment ction Modulu ction Modulu nding Stress	IS	My Mz Wy	,d d	10.285 1.077 1577.33 849.33 6.5	kNm cm ³ cm ³ N/mm ²				t		
Mo Mo Ser Ser Ber Ber	ment ment ction Modulu ction Modulu nding Stress nding Stress	IS	My Mz, Wy Wz σm σm	,d d y,d z,d	10.285 1.077 1577.33 849.33 6.5 1.3	kNm cm ³ cm ³ N/mm ² N/mm ²				Rectangle		
Mo See See Bee Bee Bee	ment ment ction Modulu ction Modulu nding Stress nding Stress nding Streng	ıs ıs th	My Mz, Wy Wz σm σm fm;	,d d y,d z,d y,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0	kNm cm ³ cm ³ N/mm ² N/mm ²		[8], Tab.1		t		
Mo See Bee Bee Bee Bee	ment ment ction Modulu ction Modulu nding Stress nding Stress nding Streng nding Streng	ıs ıs th	My Mz, Wy Wz σm σm	,d d y,d z,d y,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0 24.0	kNm cm ³ cm ³ N/mm ² N/mm ²		[8], Tab.1		t		
Mo See See Bee Bee Bee Bee Pa	ment ction Modulu ction Modulu nding Stress nding Stress nding Streng nding Streng rtial Factor	ıs ıs th th	My Mz, Wy Wz Gm Gm fm, fm, 7M	,d d y,d z,d y,k t,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0 24.0 24.0 1.300	kNm cm ³ cm ³ N/mm ² N/mm ²		[8], Tab.1 Tab. 2.3		t		,
Mo See See Bee Bee Bee Bee Par	ment ction Modulu ction Modulu nding Stress nding Streng nding Streng nding Streng ttial Factor idification Fa	is is th th ctor	My Mz, Wy Wz Gm Gm fm, fm, fm, YM km	,d d y,d z,d y,k z,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0 24.0 24.0 1.300 0.900	kNm cm ³ cm ³ N/mm ² N/mm ² N/mm ²		[8], Tab.1 Tab. 2.3 Tab. 3.1		t		
Mo See Bee Bee Bee Bee Par Mo Bee	ment ction Modulu ction Modulu nding Stress nding Streng nding Streng nding Streng ttial Factor vdification Fa nding Streng	is is th th ctor th	My Mz, Wy Wz Gm Gm fm; fm; fm; fm; fm; fm;	,d d y,d z,d y,k t,k v,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0 24.0 1.300 0.900 16.6	kNm cm ³ cm ³ N/mm ² N/mm ² N/mm ² N/mm ²		[8], Tab.1 Tab. 2.3 Tab. 3.1 Eq. (2.14)		t		
Mo See Bee Bee Bee Par Bee Bee Bee Bee Bee Bee Bee	ment ction Modulu ction Modulu nding Stress nding Streng nding Streng nding Streng ttial Factor idification Fa	is th th ctor th th	My Mz, Wy Wz Gm Gm fm, fm, fm, YM km	,d d y,d z,d y,k t,k v,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0 24.0 1.300 0.900 16.6	kNm cm ³ cm ³ N/mm ² N/mm ² N/mm ²		[8], Tab.1 Tab. 2.3 Tab. 3.1		t		y
Mo Mo See Bee Bee Bee Par Mo Bee	ment ction Modulu ction Modulu nding Stress nding Streng nding Streng nding Streng ttial Factor vdification Fa nding Streng	is is th th ctor th	My Mz, Wy Wz Gm Gm fm; fm; fm; fm; fm; fm;	,d d y,d z,d y,k t,k v,k	10.285 1.077 1577.33 849.33 6.5 1.3 24.0 24.0 1.300 0.900 16.6	kNm cm ³ cm ³ N/mm ² N/mm ² N/mm ² N/mm ²		[8], Tab.1 Tab. 2.3 Tab. 3.1 Eq. (2.14)		t		

Dlubal

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.

🛛 Left Side					
 Design Moment 	My,r	-5.905	kNm		
 Design Moment 	M _{z,r}	0.618	kNm		
 Coupling-to-Support Distance 	UI	0.731	m		
 Design Force in the Coupling 	K _{z,I}	8.078	kN		
 Design Force in the Coupling 	Ky,I	0.846	kN		
- 🕀 Coupling					
— Туре	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	kmod	0.900			Tab. 3.
- Partial Factor	γM	1.300			Tab. 2.
 Design Resistance 	F _{v.z.Rd}	19.665	kN		
Design Resistance	Fv.v.Rd	9.119	kN		
Design - Left Side		0.50		≤1	
Right Side					
Design Moment	MyJ	-8.290	kNm		
Design Moment	M _{z,I}	0.868	kNm		
Coupling-to-Support Distance	Ur	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	kmod	0.900			Tab. 3.
- Partial Factor	7M	1.300			Tab. 2.
 Design Resistance 	F _{v,z,Rd}	19.665	kN		
- Design Resistance	F _{v.v.Rd}	9.119			
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

$$K_{z,d} = \frac{M_{y,d}}{u} = \frac{8.26 \text{ kNm}}{0.5 \text{ m}} = 16.52 \text{ kN}$$
$$K_{y,d} = \frac{M_{z,d}}{u} = \frac{0.87 \text{ kNm}}{0.5 \text{ m}} = 1.73 \text{ kN}$$

$$F_{v,z,Rd} = n_{ef} \cdot F_{v,Rk} = 6 \cdot 4.734 \text{ kN} = 28.41 \text{ kN}$$

0.5 m

$$F_{v,y,Rd} = n_{ef} \cdot F_{ax,Rk} = 6 \cdot 2.195 \text{ kN} = 13.17 \text{ kN}$$

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)

$$F_{ax,Rk} = f_{head,k} \cdot d_h^2 = 8.58 \cdot 16^2 \cdot 10^{-3} = 2.195 \text{ kN}$$

Designed bearing capacity for loading perpendicular to nail axis

$$F_{v,z,Rd} = k_{mod} \cdot \frac{F_{v,z,Rk}}{\gamma_{M}} = 0.9 \cdot \frac{28.41 \text{ kN}}{1.3} = 19.67 \text{ kN}$$
$$F_{v,y,Rd} = k_{mod} \cdot \frac{F_{v,y,Rk}}{\gamma_{M}} = 0.9 \cdot \frac{13.17 \text{ kN}}{1.3} = 9.12 \text{ kN}$$

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

$$\lambda_{rel,m} = \sqrt{\frac{0.78 \cdot b^2}{h \cdot l_{ef}}} \cdot \mathsf{E}_{0.05} = \sqrt{\frac{0.78 \cdot 140^2}{260 \cdot 4300}} \cdot 7400 = 0.487$$

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

Dlubal

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

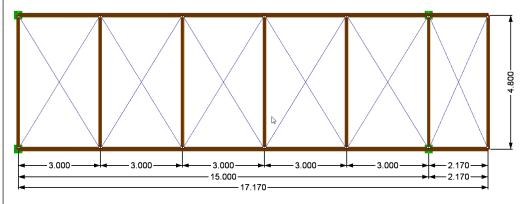


Figure 15.1: System

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: Depth:	6.0 m 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 finishe	s q _g = 0.08 kN/m	LDC = permanent
Load case 41: snow load	$q_s = 0.04 \text{ kN/m}$	LDC = short-term
Load case 56: wind parallel to ridge (A)	$q_{w(p,A)} = 0.04 \text{ kN/m}$	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.

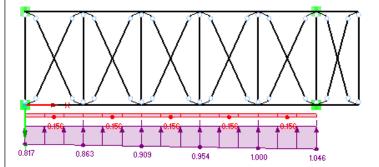


Figure 15.2: Wind load only in span

0

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.

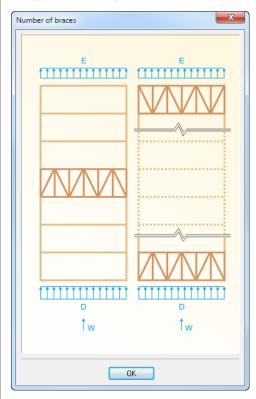


Figure 15.3: Number of braces

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.

Input Data General Data	1.1 General Data		
Geometry Geometry Materials Cross-Sections Connections Loads Loads Control Parameters Effective Lengths	Brace Type Roof shape: Monopitch Duopitch Curved	According to Standard / National Annex	
	× Fair	Asymmetric Carliever Central bay Capped with losse ridge wedge	iber
	¥+	structure	
	Comment		

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).



\angle	\checkmark
DI	ubal

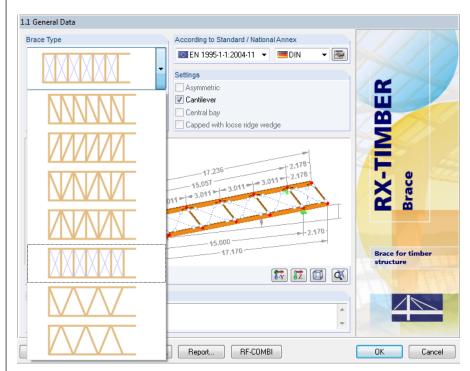
General Other S				
Partial Factor Accor	ding to 2.4.1		Limit Values of Deformations	
PT: Permanent an	d transient design sit	uation	- Characteristic (rare) design situation accord	ling to 7.2
 Solid timber 	γ M :	1.300 🚖	Fixed on Both Sides	
 Glued laminated tir 	mber γM:	1.300 ≑	SC: winst ≤1/ 1000 🖨	≤l _k / 500
AC: Accidental des	sign situation		- STR: Deformation with design values acco	rdina to 9.2.5
	ΥΜ :	1.000 🚖	UA: Winst,d ≤1/ 500) ≤l _k / 250 (
Partial Factor Accor	ding to EN 1995-1-2	, 2.3		
For timber in fire	γM,fi:	1.000 🚖		
Modification Factor I	k-mod According to	Table 3.1		
	S	ervice Class		
LDC	1	2	3	
- Permanent:	0.600 🚖	0.600 🚖	0.500 🜩	
- Long-term:	0.700 🚖	0.700 🚖	0.550 🜩	
- Medium-term:	0.800 🚖	0.800 🚖	0.650 🜩	
- Short-term:	0.900 🚖	0.900 🚖	0.700 ≑	
- Instantaneous:	1.100 🚔	1.100 🚔	0.900 🚔	
Data for Fire Protect	tion According to EN	1995-1-2, 2.3, Table	3.1 and 4.2.2	
	Softwood	Glulam	Hardwood	
Charring rate	βn: 0.80 🚔	0.70 🚖	0.55 🔿 [mm/min]	
Increased charring	dø: 7.00 🚔	7.00 🚖	7.00 🚖 [mm]	
Factor	kfi: 1.250 🚔	1.150 🚔	1.250 🜩	

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 **I/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.



Dlubal

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.

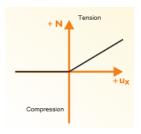


Figure 15.7: Failure under compression



15.2.2 Geometry

In the second program window, we enter the geometry of the bracing.

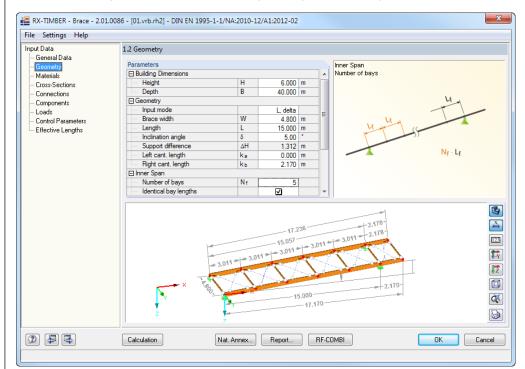


Figure 15.8: Window 1.2 Geometry

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:

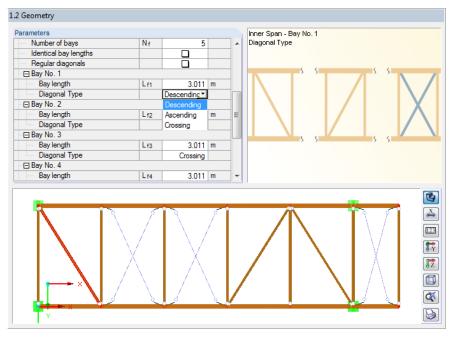


Figure 15.9: Window section *Parameters* with different types of diagonals



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.

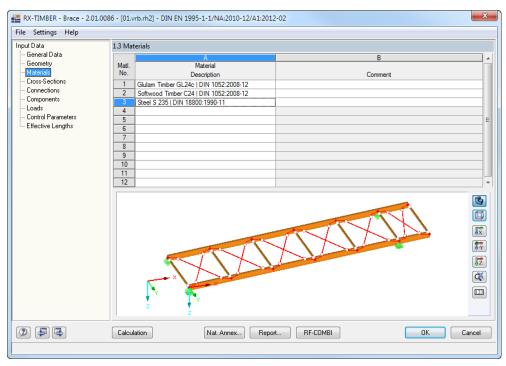


Figure 15.10: Window 1.3 Materials

.....

We access the material library by clicking the button [...] appearing at the end of the input row.

Filter	Material to Select			
Material category group:	Material Description	Standard		
Metal	▼ Steel S 235	🥅 DIN 18	800:1990-11	
- Model	Steel S 275		800:1990-11	
Material category:	Steel S 355		800:1990-11	
Steel	 Fine Grain Steel S 275 N 		800:1990-11	
_	Fine Grain Steel S 275 M		800:1990-11	
Standard group:	Fine Grain Steel S 275 M		800:1990-11	
🔳 DIN	• -			
	Fine Grain Steel S 355 M		800:1990-11	
Standard:	Fine Grain Steel S 460 N		800:1990-11	
👅 DIN 18800:1990-11	Fine Grain Steel S 460 M		800:1990-11	
	Improved Steel C 35+N		800:1990-11	
	Improved Steel C 45+N	🥅 DIN 18	800:1990-11	
	Steel St 37-2	🥅 DIN 18	800:1990-11	
	Steel USt 37-2	🔳 DIN 18	800:1990-11	
🔄 Include invalid	Steel RSt 37-2	🥅 DIN 18	800:1990-11	
Favorites only	2 h 2			7
laterial Properties		St	eel S 235 DIN 18	800:1990-
Main Properties				
 Modulus of Elasticity 		E	210000.0	
Shear Modulus		G	81000.0	N/mm ²
Poisson's Ratio		ν	0.296	1.814.2
 Specific Weight Coefficient of Thermal Expansion 		γ	/8.50 1.2000E-05	kN/m ³
Partial Safety Factor	ansion	α. 7M	1.2000E-05	1/0
Additional Properties		7 M	1.10	
Coefficient for Limiting Stres	sses of Welds	aw	0.950	
☐ Thickness Range t ≤ 4.00				
Yield Strength		fy		N/mm ²
Ultimate Strength		fu	360.0	N/mm ²
□ Thickness Range t > 4.00	cm and t ≤ 10.00 cm			
Yield Strength		fy		N/mm ²
Ultimate Strength		fu	360.0	N/mm ²
2 0.00		(ПК	Cancel

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.

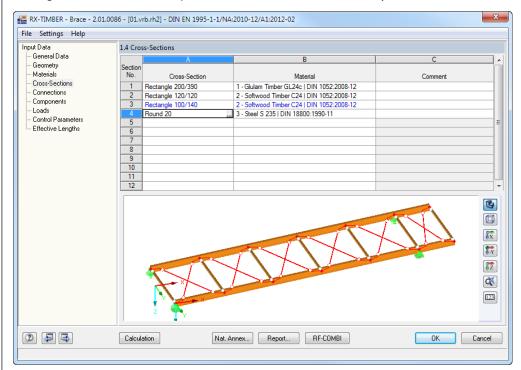


Figure 15.12: Window 1.4 Cross-Sections

Cross-Section Library			×
Rolled	Parametric - Thin-Walled	Parametric - Massive	Parametric - Timber
ILL	IIIIT		
	TLL		
1 •• • • •			ΤΤΠΠ
	Ο Σ Π Π	TLIJ	
Built-up	ΠΠΠ		
II I T T	Ţ <u>Ť</u> + •	T	
TIII	- İ l J		Standardized - Timber
IIII	ΤΣΓ		
••	L		
		User-Defined	From Cross-Section Program
2			Cancel

Figure 15.13: Cross-Section Library

Unacceptable or illogical cross-sections are highlighted in red in window 1.4.

- 1		A	В
-	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.

ut Data	1.5 Connections					
General Data Geometry	Parameters					X' X'
Materials	A	В		С		r the second sec
Cross-Sections	No. Components	Connection	1 C	omment		Y Y
Connections	1 Crossing diagona	al User-defined				· · / ·
Components	2 Vertical	To Flange				
Loads	3					
Control Parameters	4					
Effective Lengths	5				-	
Enective Lenguis	6				=	
	7					
	8					
	9					
	10					
	11					
	•				•	
	Connection Details				_	
	Eccentricity				^	
	In X' - direction	e-X'	0.000			
	In Y' - direction	e-Y'	0.000	m		
	⊟ Slip					
	Possible in plane		XY			
	Possible on		Both members		E	
	Connection Settings					
	 Number of shear pla 	anes n _{sp}	1			
	 Fastener type 		Nail 💌			
	 Number of fasteners 	s n	Nail	NS .		
	Fastener Settings		Screw			
	 Smooth shank diam 	eter d	5.0	mm	T	

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.

Connection Details			
Eccentricity			
 In X' - direction 	e-X'	0.500	m
In Y' - direction	e-Y'	0.250	m
Stiffness			
- Along x			
- Along y			
- About z		V	
Stiffness	Co	0.000	kNm/rad





Excursus:

A connection with through bolts can be modeled in different ways.

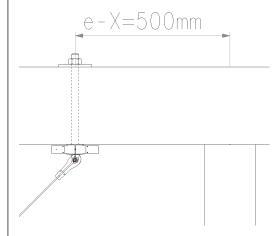


Figure 15.17: Details of through bolt

This connection can be represented by three variants:

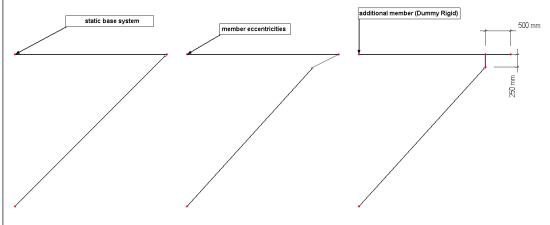


Figure 15.18: Modeling of eccentricities

- 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.

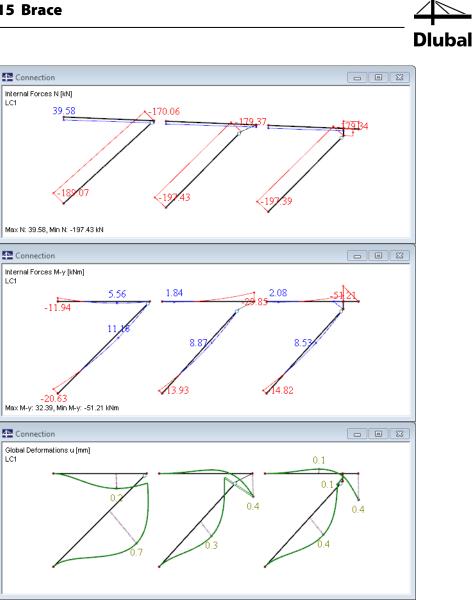


Figure 15.19: Internal forces and deformations for variants 1 to 3 (from the left to the right)

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.

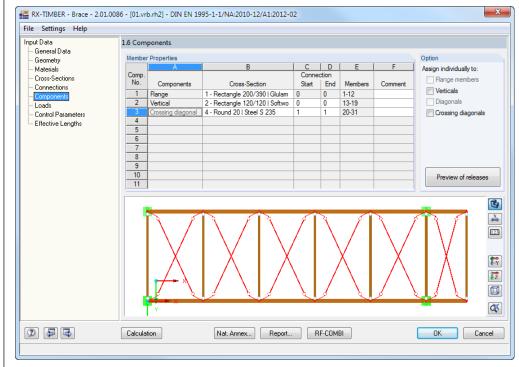


Figure 15.20: Window 1.6 Components

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*.

	A	В	C	D	E	F	Assign individually to:
Comp.			Conne	ection			Flange members
No.	Components	Cross-Section	Start	End	Members	Comment	
1	Flange	1 - Rectangle 200/390 Glulam Ti	0	0	1-12		Verticals
2	Vertical	2 - Rectangle 120/120 Softwor	0	0	13		Diagonals
3	Vertical	2 - Rectangle 120/120 Softwoo	0	0	14		Crossing diagonals
4	Vertical	2 - Rectangle 120/120 Softwoo	0	0	15		
5	Vertical	2 - Rectangle 120/120 Softwoo	0	0	16		
6	Vertical	2 - Rectangle 120/120 Softwoo	0	0	17		
7	Vertical	2 - Rectangle 120/120 Softwoo	0	0	18		
8	Vertical	2 - Rectangle 120/120 Softwoo	0	0	19		
9	Crossing diagonal	4 - Round 20 Steel S 235	1	1	20-31		
10							Preview of releases
11							

Figure 15.21: Assigning structural components individually

However, the individual modeling involves a considerable modeling effort.



Preview of releases

We use the button [Preview of releases] to look at the members with the defined eccentricities and stiffnesses in a table.

	Axial/Shea	Axial/Shear Release or Spring[kN/m] Moment Release or Spring[kNm/rad]					
	N	Vy	Vz	Мт	My	Mz	
E Member No. 13							
 Release at the member start 				V	2	V	
Release at the member end					~	V	
 Reduced release at the member start 				1	1	I	
Reduced release at the member end					1	V	
I Member No. 14							
Release at the member start				1	1	V	
Release at the member end					V	V	
Reduced release at the member start				1	1	V	
Reduced release at the member end					V	V	
I Member No. 15							
Release at the member start				1	V	V	
Release at the member end					1	V	
 Reduced release at the member start 				1	V	V	
Reduced release at the member end					1	V	
Member No. 16							
Release at the member start				1	1	V	
Release at the member end					V	V	
Reduced release at the member start				1	1	V	
Reduced release at the member end					V	V	
I Member No. 17							
Release at the member start				1	V	V	
Release at the member end					1	V	

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.

Input Data	1.7 Loads				
General Data Geometry	Number of Braces				Imposed Load Category
- Materials	More than one			•	EN 1991-1-1: H
- Connections	Equivalent Loads				Snow Load
Components <mark>Loads</mark>	Number of members for one brace	n:	4	•	🕅 North German Plain
- Control Parameters			Value		Wind Load
Effective Lengths	Description		[kN/m]	Use	Determine automatically
	Permanent Actions				<u> </u>
	LC1 - Self-Weight + Roof Finish	qg	0.080	√	Wind zone WZ: 1
	Imposed Load				Terrain category TC: Category I 🗸
	 LC21 - Imposed Load Inner Spa 		0.000		
	 LC22 - Imposed Load Cantileve 	q _p	0.000		Fundamental basic wind velocity vb.0: 22.5 [m/s]
	Snow Load				
	LC41 - Snow	qs	0.040		Body shells with permeable walls
	Wind Load				
	 LC51 - Wind Transversely to Ri 		0.000		🔿 Define manually
	 LC52 - Wind Transversely to Ri 		0.000		
	 LC53 - Wind Transversely to Ri 	Qw(q,r,A)	0.048		Peak velocity pressure q(z): 0.747 🚔 [kN/m ²] F
	 LC54 - Wind Transversely to Ri 		0.048		
	 LC55 - Wind Parallel to Ridge () 		0.039	√	
	 LC56 - Wind Parallel to Ridge (I 	Qw(p,B)	0.033	V	
	User-defined load				
	Number		0		
					Service Class Show Generated Loads
	Hide Zero Values	Hide Unu:	sed Values		SECL: 2 Resp. Define Additional Loads

Figure 15.23: Window 1.7 Loads

0

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.

Number of members for one brace
ОК

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.

		Value	
Description		[kN/m]	Us
Permanent Actions			
LC1 - Self-Weight + Roof Finish	99	0.080	5
Imposed Load			
 LC21 - Imposed Load Inner Spa 	qp	0.000	
 LC22 - Imposed Load Cantileve 	qp	0.000	C
Snow Load			
LC41 - Snow	qs	0.040	C
Wind Load			
 LC51 - Wind Transversely to Ri 	Qw(q,I,A)	0.000	C
 LC52 - Wind Transversely to Ri 	Qw(q,I,B)	0.000	C
 LC53 - Wind Transversely to Ri 	Qw(q,r,A)	0.048	
 LC54 - Wind Transversely to Ri 	Qw(q,r,B)	0.048	C
 LC55 - Wind Parallel to Ridge () 	Qw(p,A)	0.039	5
 LC56 - Wind Parallel to Ridge (I 	Qw(p,B)	0.033	
User-defined load			
Number		0	

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.



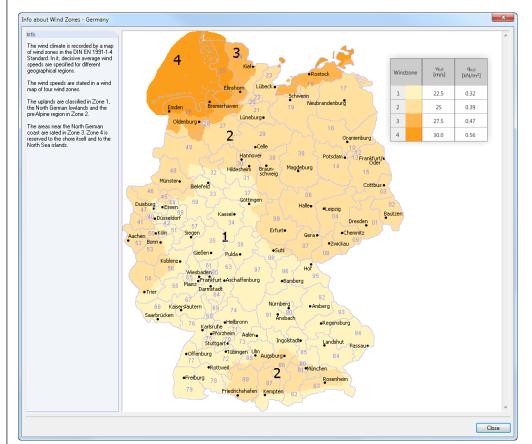


Figure 15.26: Wind zone map for Germany

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)
External pressure coefficient for area D $c_{\mbox{\tiny pe},10}$
External pressure coefficient for area D $c_{\mbox{\tiny pe},10}$
Wind pressure $c_{pe} \cdot q(z) = w_{e,D}$
M is a subtraction of $m(-)$

Wind suction $c_{pe} \cdot q(z) = w_{e,E}$

- = 0.5 kN/m²
 - 0.69 (interpolated for roof inclination of 5°)
- = 0.27 (interpolated for roof inclination of 5°)
- = 0.34 kN/m²
- = 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.

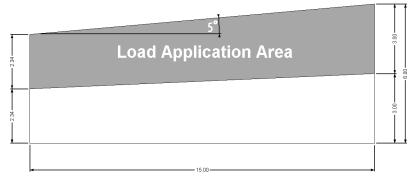


Figure 15.27: Scheme for load application area

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.

15 Brace



In our example, the settings result in the following load ordinates for the <u>wind pressure zone</u> (LC 155) with:

$$p_1 = w_{e,D} \cdot \frac{H}{2} \cdot \cos \alpha = 0.34 \text{ kN}/\text{m}^2 \cdot (3 \cdot \cos 5^\circ) = 1.02 \text{ kN}/\text{m}$$

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:

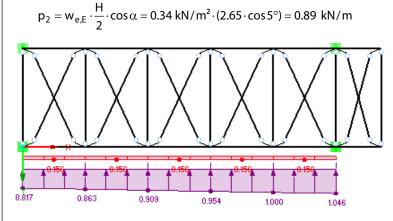


Figure 15.28: Load of wind pressure for brace 1

The load ordinates for the wind suction zone (LC 56) are the following:

$$p_1 = w_{e,D} \cdot \frac{H}{2} \cdot \cos \alpha = 0.14 \text{ kN/m}^2 \cdot (3 \cdot \cos 5^\circ) = 0.42 \text{ kN/m}$$

The load ordinate of the eaves with the eaves height of 5.33 m is the following:

$$p_2 = w_{e,E} \cdot \frac{H}{2} \cdot \cos \alpha = 0.14 \text{ kN/m}^2 \cdot (2.65 \cdot \cos 5^\circ) = 0.37 \text{ kN/m}$$

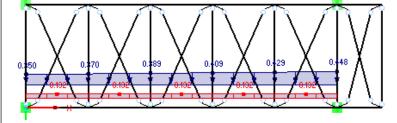


Figure 15.29: Load of wind suction for brace 2

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).

\angle	\swarrow
DI	ubal

File Settings Help									
Input Data	2.4 Supp	port Forces							
Beam Type and Material		A	B	С	D	E	F	G	
- Geometry	LC	LC/	Left Su	pport	Right Su	upport	Equiv. Load	Max.Moment	
Loads	RC	RC Description	A _X [kN]	Az [kN]	Bx [kN]	Bz [kN]	q [kN/m]	My [kNm]	
- Control Parameters		Load Cases (Characteristic Values	s)						
Results	LC1	Self-Weight + Roof Finishes	0.000	39.965	0.000	48.764	0.082	132.350	
— Result Combinations	LC41	Snow	0.000	20.422	0.000	25.078	0.041	66.522	
- Design - All	LC51	Wind Transversely to Ridge (left	-3.452	-21.925	0.000	-19.253	0.037	58.894	
 Design by Location X 	LC52	Wind Transversely to Ridge (left	0.252	-1.698	0.000	2.623	0.002	2.576	
Support Forces	LC53	Wind Transversely to Ridge (rig	-5.755	-21.596	0.000	-37.678	0.046	72.715	
	LC54	Wind Transversely to Ridge (rig	-5.755	-21.596	0.000	-37.678	0.046	72.715	
	LC55	Wind Parallel to Ridge (A)	-2.458	-15.087	0.000	-13.667	0.037	58.929	
	LC56	Wind Parallel to Ridge (B)	-2.394	-13.576	0.000	-13.786	0.032	50.706	
	Max		0.252	39.965	0.000	48.764	₩ Desele	ct Ctrl-	۰U
	Min		-5.755	-21.925	0.000	-37.678			
							of Cut	Ctrl	÷Χ
		Result Combinations for Ultimate L	imit State (Desig	gn Values) (STF)		Сору	Ctrl	. C
	RC1	g	0.000	53.953	0.000	65.831		and a	
	RC2	g + s	0.000	84.586	0.000	103.448	Paste	Ctrl	÷۷
	RC3	g + s + w(q,I,A)	-3.107	64.853	0.000	86.121			
	RC4	g + s + w(q,I,B)	0.227	83.058	0.000	105.809	Delete	Ctrl+E	el
	RC5	g + s + w(q,r,A)	-5.179	65.149	0.000	69.538	Add		
	RC6	g + s + w(q,r,B)	-5.179	65.149	0.000	69.538	Landate		
	RC7	g + s + w(p,A)	-2.212	71.007	0.000	91.148	🔜 Multip	ily	
	RC8	g + s + w(p,B)	-2.155	72.367	0.000	91.041	Divide		
	RC9	g + w(q,I,A)	-5.178	21.065	0.000	36.952			
	RC10	g + w(q,I,B)	0.378	51.406	0.000	69.766	📑 Set		
	RC11	g + w(q,r,A)	-8.632	21.559	0.000	9.314	😼 Genera	ate Ctrl+	
	Tilt. mom pinned s	nent for upport T _d : 3.481 [kNm]	Axial force i compr. edg			quiv.load or1 beam q			0
							🛛 📝 Import	t	

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.

oad Case	es									×
Load Cas	se						Self-wei	ght		
LC No.:	LC101 🔻 📢	•	×	Short descri	ption: g			ider with		-
Descripti	ion: Self-Weight + R	oof Finishes					facto	r:		
Load Typ	pe: Impo:	sed Load Ca	ategory:	Load Durati	on Class:					
-	Permanent 🚽			LDC: Permanent -			Comment			
Varial							Commen			*
Accio Seism										-
	nic									
Load	A	В	С	D	E	F	G	(н		Load Case LC10
	~	Load	C		Load Par				Across Tot	V
No.	Load Type	Direction	Load Reference	p [kN/m]	P 2	A	В	in %	Length	Comment
1	Line Load	YL	Inner Span	-0.320						
2 3										
4										
5										
6										
7										
8										
		0.390		.320 0.	390 • • • • • • • • • • • • • • • • • • •	0.390	0.390	2		
D	00								OK	Cancel

Figure 15.31: Dialog box Load Cases

Here, we can freely define the loads like in all other programs of the RX-TIMBER family.

Show Generated Loads... resp. Define Additional Loads...



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.

Data eneral Data	1.8 Control Parameters	
eneral Data eometry	Design of	Special Settings for Glulam
– Materials – Cross-Sections – Connections – Components – Loads – Edited Parameters – Effective Lengths	── Static equilibrium (EQU) ✓ Ultimate limit state (STR)	Special settings for glulam according to annotations in Table F.9
	Serviceability limit state	Increase of strength fm,k according to: ☐ Amotation b Flat ended bending strain My for lamellas of the dulam beam with h ≤ 600 mm
	Fire protection Fire Resistance Class: F 50-B F 90-B F 45 m (min)	quitam beam with h S bOU mm Annotation <u>c</u> Edgewise bending strain M ₂ for lamellas of homogenous glulam consisting of a minimum of four side-by- side lamellas
	Charring:	Calculation Parameters
		Generate supplementary combinations from favorable permanent actions
		Distribute permanent load span-by-span
	To Display	Number of member divisions for
	Support forces	result diagrams:

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.

5

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 service-ability 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.

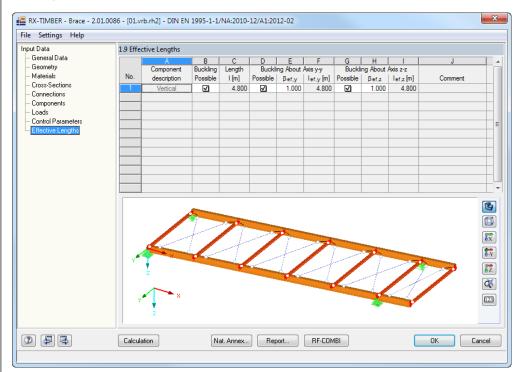


Figure 15.33 Window 1.9 Effective Lengths

15.2.10 RF-COMBI

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.

RF-COMBI - [01.vrb.rh2]			
File Settings Help			
CA1 - Ultimate Limit State Desig	👻 1.2 Load C	ases in Actions	
CA1 - Ultimate Limit State Design CA2 - Ultimate Limit State Design			
- Actions	AC1	Self-weight + Roof structure	-
- Action Categories 1	AC4	Wind Loads	
Action Categories 2			
			Ξ
			-

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

Calculation

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 cas- es. Therefore, always two result values per location are available: maximum and minimum.
	It is possible to determine the extreme internal forces and defor- mations from different load cases, load or result combinations by an <i>Or</i> 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!

put Data	2.1 Loa	d Combinations					
- General Data		A	В	C	D	E	F
Geometry	~~			Design		Factor	Max.
- Materials	CO	CO Description	Load Cases	Situation	LDC	kmod	Desigr
- Cross-Sections		Ultimate Limit State De					
Connections	CO1	g	1.35*LC1	UB	Permanent	0.600	0
Components	CO2	g + w(p,B)	1.35*LC1 + 1.50*LC56	UB	Short-term	0.900	. (
Loads	CO3	g	1.35*LC101	UB	Permanent	0.600	. (
- Control Parameters Effective Lengths	CO4	g + w(p,A)	1.35*LC101 + 1.50*LC155	UB	Short-term	0.900	(
– Design by Components – Design by Konst-Section – Design by Member – Design by x-Location – Support Forces			Max 0.82	51 0	(
			🎽 Max: 0.82	≤1 🥹			

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.

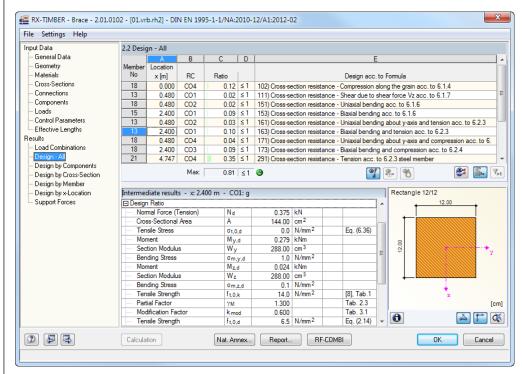


Figure 15.36: Window 2.2 Design - All

2

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.

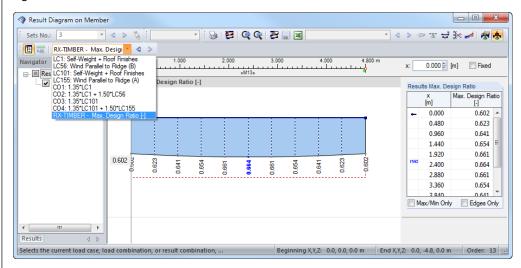


Figure 15.37: Result diagram on member 13



15.3.3 Design by Components

The results window shows the governing designs for the structural components *Front flange*, *Rear flange*, *Verticals* and *Diagonals*.

nput Data	2.3 Desig	gn by Com	ponents									
- General Data		A	B	С	D				E			
Geometry Materials	Member No		-									
- Cross-Sections	NO	x [m] Front flance	RC	Ratio				Desi	gn acc. to F	omula		
Connections	3	1.506	cO4		≤1 (91) STR - Defo	mations f	or design y	aluan aco tr	925. long	r ann w dim	otion
- Components	6	2.178	C04			92) STR - Defo						
- Loads		2.170	004	0.17		52,0111 2010		or dobigri vi	alabo abo. n	0.2.0 001	alover, y allo	000011
 Control Parameters 		Rear flange	, member	s: 7-12								
Effective Lengths	9	1.506	CO4	0.32	! ≤1 3	91) STR - Defo	rmations f	or design v	alues acc. to	9.2.5 - Inne	er span, y-dire	ection
lesults	12	2.178	CO4	0.17	/ ≤1 (92) STR - Defo	rmations f	or design v	alues acc. to	9.2.5 - Cari	tilever, y-dire	ction
- Load Combinations												
- Design - All Design by Components		Vertical No	.1									
- Design by Cross-Section			Max: [0.81	≤1 (•			?	- 8	(2 🖺 🛛
Design by Member Design by x-Location	Interme	diate result:	s - x:1.5	06 m - C	04: g + \	v(p,A)				Rectangle	20/39	
Support Forces		ial Data - Glu		er GL24c							20.00	+
		Section Data	3									
	Deform											
		ection x			Wx	-0.4						
										8		
	Dire	ection y			Wy	-9.6				8		
	Dire	ection z			Wy Wz	-9.6 46.3				39.00		* y
	Dire Dire Diresign	ection z n Ratio	ìnan		Wz	46.3	mm			39.00		
	Dire Dire Design Def	ection z			· ·	46.3	mm mm			38.00		я
	Dire Dire Design Def	ection z n Ratio formation in S	,th		Wz	46.3	mm mm			39.00		* y
	Dire Dire Desigr Def Ref	ection z n Ratio formation in S ference Leng	th rion		Wz Winst,y,c	46.3 -9.6 15.057	mm mm m			39.00	z	y
	Dire Dire Desigr Def Ref Limi	ection z n Ratio formation in S ference Leng it Value Crite	th rion		Wz Winst,y,c	46.3 -9.6 15.057 500.00	mm mm m	<u>≤1</u>		39.00	2	[c
	Dire Dire Desigr Def Ref Limi	ection z n Ratio formation in S ference Leng it Value Crite it Value of D	th rion		Wz Winst,y,c / Winst,lim	46.3 -9.6 15.057 500.00 30.1	mm mm m	≤1		Ļ	Z	
	Dire Dire Desigr Def Ref Limi	ection z n Ratio formation in S ference Leng it Value Crite it Value of D	th rion		Wz Winst,y,c / Winst,lim	46.3 -9.6 15.057 500.00 30.1	mm mm m	≤1 		33.00	z	с [с

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.

iput Data	2.4 Desig	gn by Cross	-Section	n							
General Data		A	В	C	D					E	
Geometry	Member										
Materials	No	x [m]	RC	Ratio	·				Design acc	c. to I	Formula
- Cross-Sections		Cross-section									
- Connections	3	1.506	CO4		.32 ≤1						to 9.2.5 - Inner span, y-direction
Components	12	2.178	CO4	0	.17 ≤1	392) STR - D	eformation	is for de	sign values a	icc.t	to 9.2.5 - Cantilever, y-direction
- Loads											
Control Parameters		Cross-section									
Effective Lengths	18	0.000	CO4		.12 ≤1						ng the grain acc. to 6.1.4
sults	13	0.480	C01		.02 ≤ 1						ear force Vz acc. to 6.1.7
Load Combinations	18	0.480	CO2		.02 ≤ 1						acc. to 6.1.6
Design - All	15	2.400	CO1	0	.09 ≤ 1	153) Cross-se	ection resis	tance -	Biaxial bendi	ng a	icc. to 6.1.6
Design by Components			Max:	0	.81 ≤ 1	۲					۹ 🕙 😫
 Design by Member 											
Design by x-Location	Interme	diate result	s - x: 0.0	000 m -	CO4: g	+ w(p,A)					Rectangle 12/12
		diate result ear Force	s - x: 0.0			4.4.4	kN	1			Rectangle 12/12
	- She		s - x: 0.0		CO4:g V _{y.d} V _{z.d}	+ w(p,A) -0.020 0.232				^	
	She	ear Force	s - x: 0.0		V _{y,d}	-0.020	kN				
	She	ear Force ear Force	s - x: 0.0		V _{y,d} V _{z,d} M _T	-0.020 0.232 0.000	kN kNm				
	She	ear Force ear Force sion	s - x: 0.0		V _{y,d} V _{z,d}	-0.020	kN kNm kNm				12.00
	She	ear Force ear Force sion ment ment	s - x: 0.0		V _{y,d} V _{z,d} M _T M _{y,d}	-0.020 0.232 0.000 0.000	kN kNm kNm				
	She She Tor Mor E Design	ear Force ear Force sion ment ment			V _{y,d} V _{z,d} M _T M _{y,d}	-0.020 0.232 0.000 0.000	kN kNm kNm kNm				12.00
	She She Tan Moi Desigi Nor	ear Force ear Force sion ment ment n Ratio	Compressio	on)	V _{y,d} V _{z,d} M _T M _{y,d} M _{z,d}	-0.020 0.232 0.000 0.000 0.000	kN kNm kNm kNm				12.00
	She She Ton Mor Design Nor Cro	ear Force ear Force sion ment ment n Ratio mal Force (C	Compressi	on)	V _{y,d} V _{z,d} M _T M _{y,d} M _{z,d}	-0.020 0.232 0.000 0.000 0.000 25.953 144.00	kN kNm kNm kNm		Eq. (6.36)		
	She She Tor Mor Desig Nor Cro	ear Force sar Force sion ment ment n Ratio mal Force (C ss-Sectional	Compressio Area ress	on)	Vy,d Vz,d MT My,d Mz,d Nd A	-0.020 0.232 0.000 0.000 0.000 25.953 144.00 1.8	kN kNm kNm kN cm ²		Eq. (6.36) [8], Tab.1		
	She She Tor Mor Design Nor Cro Cor	ear Force sion ment ment n Ratio mal Force (C ss-Sectional npressive Str	Compressio Area ress	on)	V _{y,d} V _{z,d} M _T M _{y,d} M _{z,d} N _d A σ _{c,0,d}	-0.020 0.232 0.000 0.000 0.000 25.953 144.00 1.8	kN kNm kNm kN cm ² N/mm ²				
	She She Tor Moi Desig Nor Cro Con Con	ear Force sion ment ment n Ratio mal Force (C ss-Sectional npressive Str npressive Str	Compressi Area ress rength	on)	V _{y,d} V _{z,d} MT M _{y,d} M _{z,d} Nd A σ _{c,0,d} f _{c,0,k}	0.232 0.000 0.000 0.000 0.000 25.953 144.00 1.8 21.0	kN kNm kNm kN cm ² N/mm ²		[8], Tab.1		
-Design by x-Location Support Forces	She She Tor Moi Design Nor Cro Cor Cor Par	ear Force sar Force sion ment ment n Ratio mal Force (C ss-Sectional npressive St npressive St tial Factor	Compression Area ress rength ctor	on)	V _{y,d} V _{z,d} MT My,d Mz,d Nd A Gc,0,d fc,0,k 7M	0.020 0.232 0.000 0.000 0.000 0.000 25.953 144.00 1.88 21.0 1.300 0.900	kN kNm kNm kN cm ² N/mm ²		[8], Tab.1 Tab. 2.3		
	She She Tor Moi Design Nor Cro Cor Cor Cor Cor Cor Cor Cor Cor C	ear Force sar Force sion ment matio mal Force (C ss-Sectional npressive Str npressive Str tial Factor dification Fac	Compression Area ress rength ctor	on)	V _{y,d} V _{z,d} MT M _{y,d} M _{z,d} N _d A G _{c,0,d} f _{c,0,k} 7M k mod	0.020 0.232 0.000 0.000 0.000 0.000 25.953 144.00 1.88 21.0 1.300 0.900	kN kNm kNm kNm cm ² N/mm ² N/mm ²		[8], Tab.1 Tab. 2.3 Tab. 3.1		

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.

nput Data	2.6 Decir	an by x-Loc	ation									
- General Data	2.0 Desig	A A	B	C	D				E			
Geometry	Member		D						C			
- Materials	No	x [m]	RC	Ratio				De	sion acc. to	Formula		
Cross-Sections	1	Member No	1									
- Connections		0.000	CO4	0.03	≤1	391) STR - Defo	mations	for design	values acc.	to 925 - Inner	span v-dire	ction
- Components		0.502	CO4			391) STR - Defo						
Loads		1.004	CO4			391) STR - Defo						
Control Parameters		1.506	CO4	0 19		391) STR - Defo						
Effective Lengths		2.008	CO4			391) STR - Defo						
esults		2.510	CO4	0.22		391) STR - Defo						
Load Combinations		3.011	CO4	0.23		391) STR - Defo						
Design - All				-								
 Design by Components 			L.			•					ſ	
- Design by Cross-Section			Max:	0.81	≤1	۲			9			😫 🖪
Design by Cross-Section Design by Member						-			? [2
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>		diate results	s - x: 2.0	08 m - C(-			?	Rectangle 2		2
- Design by Cross-Section Design by Member	⊞ Materia	al Data - Glu	s - x: 2.0 Iam Timbe	08 m - C(-] 🎦	Rectangle 2		2
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Tross-	al Data - Glu Section Data	s - x: 2.0 Iam Timbe	08 m - C(-				Rectangle 2	20/39	2
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Tross- Deform	al Data - Glu Section Data nations	s - x: 2.0 Iam Timbe	08 m - C(D4: g +	w(p,A)				Rectangle 2	20/39	
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Defom Dire	al Data - Glu Section Data nations ection x	s - x: 2.0 Iam Timbe	08 m - C0 erGL24c	D4: g +	w(p,A)				Rectangle 2	20/39	₽ []
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Deform Dire	al Data - Glu Section Data nations ection x ection y	s - x: 2.0 Iam Timbe	08 m - C0 erGL24c	D4: g +	w(p,A)	mm				20/39	
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Materia Cross- Defom Dire Dire Dire	al Data - Glu Section Data nations ection x ection y ection z	s - x: 2.0 Iam Timbe	08 m - C0 erGL24c	D4: g +	w(p,A)	mm			Rectangle 2	20/39	
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Defom Dire Dire Dire Dire	al Data - Glu Section Data nations ection x ection y ection z n Ratio	s - x: 2.0 Iam Timbo a	08 m - C(er GL24c	D4: g + Wx Wy Wz	w(p,A)	mm mm				20/39	¥ 🗐
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Deform Dire Deform Dire Deform Dire Deform Dire Deform Dire Deform Dire Deform Dire Deform Dire Dire Dire Dire Dire Dire Deform Dire Dire Deform Dire Deform Dire Deform Dire Deform Dire Deform Dire Deform Dire Dire Deform Dire Dire	al Data - Glu Section Data nations ection x ection y ection z n Ratio ormation in S	s - x: 2.0 Ilam Timbe a Opan	08 m - C(er GL24c	D4: g +	w(p,A) -0.3 -6.3 19.3 c -6.3	mm mm				20/39	₽
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross Defom Dire Dire Dire Dire Dire Dire Refu	al Data - Glu Section Data actions ection x ection y ection z n Ratio ormation in S erence Leng	s - x: 2,0 Iam Timbi a Span	08 m - Cl er GL24c	D4: g + Wx Wy Wz Winst,y,	w(p,A)	mm mm				20/39	♥ ● ●
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Deform Dire Dire	al Data - Glu Section Data actions ection x ection x ection z n Ratio ormation in S erence Leng t Value Criter	s - x: 2.0 Ilam Timbr a Span th	08 m - C(D4: g + Wx Wy Winst,y,d I I/	w(p,A) -0.3 -6.3 19.3 c -6.3 15.057 500.00	mm mm mm				20/39	₽ E (
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Deform Dire Dire	al Data - Glu Section Data nations ection x ection y ection z n Ratio ormation in S erence Leng t Value Criter t Value of De	s - x: 2.0 Ilam Timbr a Span th	08 m - C(D4: g + Wx Wy Winst.y. I I/ Winst.lim	w(p,A) -0.3 -6.3 19.3 c -6.3 15.057 500.00 n -30.1	mm mm				20/39	y
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Deform Dire Dire	al Data - Glu Section Data actions ection x ection x ection z n Ratio ormation in S erence Leng t Value Criter	s - x: 2.0 Ilam Timbr a Span th	08 m - C(D4: g + Wx Wy Winst,y,d I I/	w(p,A) -0.3 -6.3 19.3 c -6.3 15.057 500.00	mm mm mm	≤1			20/39	¥ 🗐
Design by Cross-Section Design by Member <mark>Design by x-Location</mark>	Materia Cross- Deform Dire Dire	al Data - Glu Section Data nations ection x ection y ection z n Ratio ormation in S erence Leng t Value Criter t Value of De	s - x: 2.0 Ilam Timbr a Span th	08 m - C(D4: g + Wx Wy Winst.y. I I/ Winst.lim	w(p,A) -0.3 -6.3 19.3 c -6.3 15.057 500.00 n -30.1	mm mm mm	 			20/39	y

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.



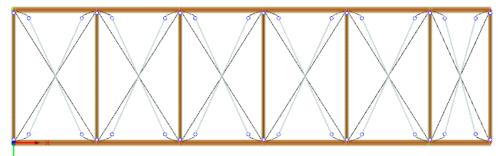
In our example, we get the following stiffness properties for load bearing capacity, serviceability and fire resistance of the respective components as shown in the table below. The stiffness properties of the fire resistance are to put on the same level with the ones for an accidental combination.

Component	Material	Load bearing capacity	Serviceability	Fire resistance
Chord	GL24c	$E = \frac{1160}{1.3} = 892 \text{ kN/cm}^2$	E _{mean} = 1160 kN/cm ²	E _{mean}
		$G = \frac{590}{1.3} = 454 \text{ kN/cm}^2$	G _{mean} = 590 kN/cm ²	G _{mean}
Verticals	C24	$E = \frac{1100}{1.3} = 846 \text{ kN/cm}^2$	E _{mean} = 1100 kN/cm ²	E _{mean}
		$G = \frac{690}{1.3} = 531 \text{kN/cm}^2$	$G_{mean} = 690 \text{ kN/cm}^2$	G _{mean}
Diagonals	S235	$E = \frac{21000}{1.1} = 19091 \text{ kN/ cm}^2$	E = 21000 kN/cm ²	-
		$G = \frac{8100}{1.1} = 7364 \text{ kN/cm}^2$	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 $1.35 \cdot LC101 + 1.5 \cdot LC155$ features the following loads, internal forces and deformations.

Loads

CO4 : 1.35*LC101 + 1.50*LC155 Loads [kN/m]



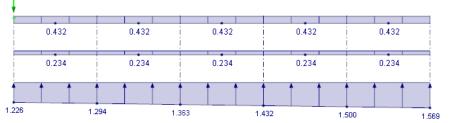


Figure 15.41: Loads



Deformation

Global Deformations u-Y [mm] CO4 : 1.35*LC101 + 1.50*LC155

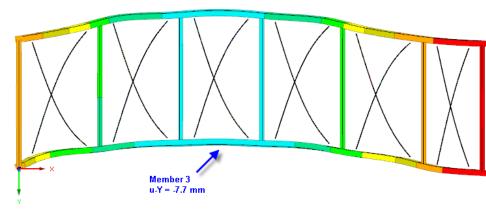


Figure 15.42: Deformations u_Y

Internal forces

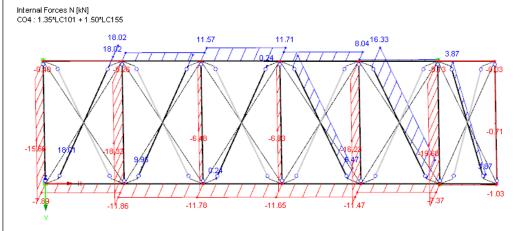


Figure 15.43: Axial forces N

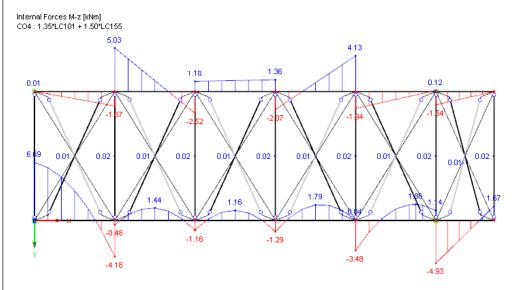


Figure 15.44: Moments Mz



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} = I/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 = $18.02 \text{ kN} / 3.14 \text{ cm}^2 = 5.74 \text{ kN/cm}^2$

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.

General Data Geometry Materials	LC	A LC/CO	В	C	0				
	LC	10/00			D	E	F	G	н
Materials		LC/CO	Support	Suppo	ort Reactions	[kN]	Support	t Moments	cNm]
	CO	Description	No.	Px	Py	Pz	Mx	MY	Mz
Cross-Sections		Load Cases (Characteristic Values)							
Connections	LC1	Self-Weight + Roof Finishes	1	0.000	0.000	3.653	-0.072	0.000	0.000
Components			2	0.000	0.000	4.974	-0.101	0.000	0.000
Loads			3	0.000	2.409	3.653	0.075	0.000	0.000
 Control Parameters 			4	0.000	2.409	4.974	0.099	0.000	0.000
 Effective Lengths 	LC56	Wind Parallel to Ridge (B)	1	0.000	0.000	0.000	0.002	0.000	0.000
esults			2	0.000	0.000	0.000	-0.002	0.000	0.000
 Result Combinations 			3	0.000	5.298	0.000	0.003	0.000	0.000
Design - All			4	0.000	5.666	0.000	-0.003	0.000	0.000
Design by Components	LC101	Self-Weight + Roof Finishes	1	0.000	0.000	3.653	-0.075	0.000	0.000
 Design by Cross-Section 			2	0.000	0.000	4.974	-0.099	0.000	0.000
 Design by Member 			3	0.000	-2.409	3.653	0.073	0.000	0.000
 Design by x-Location 			4	0.000	-2.409	4,974	0.101	0.000	0.000
 Support Forces 	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.



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 f
 ür 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] BLAB, H.J.; EHLBECK J.; KREUZINGER H.; STECK G.: Erläuterungen zu DIN 1052:2004-08; Informationsdienst Holz (März 2005)

A



B: Index

Additional load	48
Archive	21
ASCII file	78
Attic	
Axial spacing	
В	
Beam height	52
Beam type	29
Bearing capacity	61
Bending load	120, 143, 159
Bending stress	100
Brace	162
Bracing	
Browse through windows	28
Building dimensions	
C	
Calculation	
Calculation parameter	53
Cantilever 29,	36, 38, 46, 50
Center line	51
Column	51, 52, 124
Combination coefficients	43
Company address	74
Company header	73
Company logo	74
Compression strength	58
Connect folder	15
Connection partial factor	33
Context menu	69
Continuous beam	107
Control parameter	51
Copy model	19
Coupled purlin	148
Coupling	151, 161
Coupling element	151, 160
Cover	81
Create model	
Create project	14
Cross-section library	
CSV export	90
Current project	13

D Deformation53, 56, 66 Delete a model 20 Design ratio 62 Design situation61 Details.... Disconnect a folder 16 Е Earth quake 12 Effective lengths......113 F Filter Finger-jointed connection......146 G

General data26
Geometry
Glued-laminated beam92
Glued-laminated timber
Grab mode70
Grain55
н
Header template74
History21
Horizontal load12
I
Import project folder18
Imposed load42
Info parameters40
Inner beam42
Installation9
Intermediate results63
J
Joint strength
F
Lamella thickness
Lateral buckling
Lateral buckling analysis121
Lateral buckling analysis121 Lateral buckling design101, 105, 145, 161
Lateral buckling analysis121
Lateral buckling analysis121 Lateral buckling design101, 105, 145, 161 Lateral buckling moment
Lateral buckling analysis121 Lateral buckling design101, 105, 145, 161 Lateral buckling moment
Lateral buckling analysis
Lateral buckling analysis121 Lateral buckling design101, 105, 145, 161 Lateral buckling moment39 Lateral support39 Layout
Lateral buckling analysis
Lateral buckling analysis121Lateral buckling design101, 105, 145, 161Lateral buckling moment39Lateral support39Layout80LDC62Load49Load case48Load category42Load direction50Load reference50Load types50
Lateral buckling analysis121Lateral buckling design101, 105, 145, 161Lateral buckling moment39Lateral support39Layout80LDC62Load49Load case48Load category42Load direction50Load reference50Load types50Loading41
Lateral buckling analysis121Lateral buckling design101, 105, 145, 161Lateral buckling moment39Lateral support39Layout80LDC62Load49Load case48Load category42Load direction50Load reference50Load types50Loading41Loading due to shear force120, 130, 158
Lateral buckling analysis121Lateral buckling design101, 105, 145, 161Lateral buckling moment39Lateral support39Layout80LDC62Load49Load case48Load category42Load direction50Load reference50Load types50Loading41Loading due to shear force120, 130, 158Location x53
Lateral buckling analysis
Lateral buckling analysis121Lateral buckling design101, 105, 145, 161Lateral buckling moment39Lateral support39Layout80LDC62Load49Load case48Load category42Load direction50Load reference50Load types50Loading41Loading due to shear force120, 130, 158Location X63, 64Lock graphic76
Lateral buckling analysis121Lateral buckling design101, 105, 145, 161Lateral buckling moment39Lateral support39Layout80LDC62Load49Load case48Load category42Load direction50Load reference50Loading41Loading due to shear force120, 130, 158Location X63, 64Lock graphic76Logo74
Lateral buckling analysis 121 Lateral buckling design 101, 105, 145, 161 Lateral buckling moment 39 Lateral support 39 Layout 80 LDC 62 Load 49 Load case 48 Load category 42 Load direction 50 Load reference 50 Loading 41 Loading due to shear force 120, 130, 158 Location x 53 Location X 63, 64 Lock graphic 76 Logo 74



Model
Model description 26, 73
Modification factor
Monopitch roof beam29
Ν
National annex6, 30, 56
Navigator
Network11, 25, 27
Network projects
New page69
Notch
0
Open model 19, 25
OpenOffice90
Optimization52
Oscillation design
Overhang44
Ρ
Page numbering74
Page preview70
Parallel beam
Parallel installation
Partial safety coefficients115
Partial safety factor
PDF file
Permeable walls 46
Pitched cambered beam
Plausibility check
Plywood
Print file
Print graphic
Print text
Printing
Printout report
Printout report header73, 74
Printout report template79
PRO.DLP 15
Program filter
Project description 17, 73
Project Manager 11, 13
Pull-off strength161
Purlin
R
Ratio 63

Recycle bin 16, 20, 24
Redistribution of moments159
Reduction of internal forces55
Rename model20
Report template79
Restrictions12
Result combination 59, 61, 63, 67
Result diagram
Results windows60
RF-COMBI50, 59, 113
Roof edge
Roof load41
RTF file
RX-TIMBER 111
S
Screw load bearing capacity
Selection in printout report71
Selection mode70
Self-weight
Service class

Selection in printout report	71
Selection mode	70
Self-weight	
Service class	43
Service class category	
Serviceability	51, 61, 62, 132
Set language	
Shape	
Shape coefficient	45
Shear factor	57
Shear stress	
Short description	
Size of bore	
Snow guard	45
Snow load	12, 43
Snow load zone	43
Solid wood	
Span deflection	
Special settings	54
Standard	
Start program	12
Start RX-TIMBER	12
Steel bars	

Dlubal

Stiffening for transversal tension 32, 57
Stresses
Sub-project 15
Support 51, 54
Support compression121
Support conditions 52
Support forces
Support springs
Support width
т
Tapered members 53
Text file
Thumbnail image
Tilting moment
Timbger grade
Topography type 43
Torsion
Torsion design102
Trapezoidal load 50
U
Ultimate limit state
Units
Updates
User profile
V
VCmaster
Volume factor
w
Wheel button
Wind load12, 45, 49
Wind pressure
Wind suction
Wind zone
Wind zone map
Window
Wood material strips
Wood plates
Wood-based plates
Wood-based strips