

Version September 2017

Add-on Module

RF-/TIMBER NBR

Design of Timber Members According to NBR 7190:1997

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 2017 Am Zellweg 2 93464 Tiefenbach Germany

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



Contents

Contents

Page

1.	Introduction	. 3
1.1	Add-on Module RF-/TIMBER NBR	
1.2	Using the Manual	
1.3	Open RF-/TIMBER NBR	
2.	Input Data	
2.1	General Data	
2.1.1	Ultimate Limit State	
2.1.2	Serviceability Limit State	
2.1.3	Standard	
2.2	Materials	
2.3	Cross-Sections	13
2.4	Load Duration	
2.5	Service Conditions - Members	
2.6	Service Conditions - Set of Members	
2.7	Effective Lengths - Members	
2.8	Effective Lengths - Sets of Members	
2.9	Serviceability Data	
3.	Calculation	
3.1	Detail Settings	
3.1.1	Resistance	
3.1.2	Stability	26
3.1.3	Serviceability	
3.1.4	Other	
3.2	Start Calculation	29
4.	Results	30
4.1	Design by Load Case	31
4.2	Design by Cross-Section	
4.3	Design by Set of Members	33
4.4	Design by Member	
4.5	Design by x-Location	
4.6	Governing Internal Forces by Member	35
4.7	Governing Internal Forces by Set of Members	
4.8	Member Slendernesses	36
4.9	Parts List by Member	38
4.10	Parts List by Set of Members	39
5.	Results Evaluation	40
5.1	Results in RFEM/RSTAB Model	41
5.2	Result Diagram	43
5.3	Filter for Results	44
6.	Printout	46
6.1	Printout Report	46
6.2	Graphic Printout	46
7.	General Functions	48
7.1	Design Cases	48
7.2	Cross-Section Optimization	50
7.3	Units and Decimal Places	52

4 Dlubal



7.4	Data Transfer	
7.4.1	Exporting Materials to RFEM/RSTAB	
7.4.2	Exporting Effective Lengths to RFEM/RSTAB	
7.4.3	Exporting Results	
8.	Example: Timber Beam	
8.1	System and Loads	
8.2	Calculation with RFEM/RSTAB	
8.3	Design with RF-/TIMBER NBR	
8.3.1	Ultimate Limit State Design	
8.3.2	Serviceability Limit State Design	
Α.	Literature	
В.	Index	

1 Introduction

1.1 Add-on Module RF-/TIMBER NBR

The Standard NBR 7190 - Design of wooden structures [1] provides general rules that must be followed for the design, construction and analysis of timber in Brazil. With the add-on modules RF-TIMBER NBR (for RFEM) and TIMBER NBR (for RSTAB), DLUBAL provides powerful tools for the design of timber beam models according to this standard.



In the following, the add-on modules of both main programs are described in one manual and are referred to as **RF-/TIMBER NBR**.

RF-/TIMBER NBR performs all cross-section resistance designs, optional stability analyses, and deformation analyses according to the standard [1]. For the stability analysis, additional bending moments from geometric imperfections (defined by a set of eccentricities) are considered. Alternatively, when imperfections are defined manually in RFEM or RSTAB and a second order analysis is performed, the design is possible without a stability analysis.

For timber models, the serviceability limit state can be important for the design. For this, it is possible to assign load cases, load and result combinations to different design situations. The limit deformations are preset by the standard and can be adjusted, if necessary. In addition, it is possible to specify reference lengths and precambers that are considered accordingly in the design.

The add-on module provides an automatic cross-section optimization with the possibility to export modified cross-sections to RFEM or RSTAB. Separate design cases allow for the flexibility to analyze individual structural components in complex structures.

Like other add-on modules, RF-/TIMBER NBR is completely integrated in RFEM or RSTAB. Thus, the design-relevant input data is preset when you open the module. Thus, the design-relevant input data is preset when you start the add-on module. After the design, you can use the graphical user interface of the main program to evaluate the results. As they are also included in the global printout report, the entire verification can be presented in a consistent and appealing form.

We hope you will enjoy working with RF-/TIMBER NBR.

Your DLUBAL team

1.2 Using the Manual

Topics like installation, graphical user interface, results evaluation, and printout are described in detail in the manuals of the main programs RFEM and RSTAB. The present manual focuses on typical features of the RF-/TIMBER NBR add-on module.

۲

The descriptions in this manual follow the sequence and structure of the module's input and results windows. In the text, the described **buttons** are given in square brackets, for example [View mode]. At the same time, they are pictured on the left. **Expressions** appearing in dialog boxes, windows, and menus are set in *italics* to clarify the explanations.

At the end of the manual, you can find the index. If you cannot find what you are looking for, go to the Knowledge Base where you can search for the solution of the problem. Or consult the FAQs on our website.



RFEM and RSTAB provide the following options to start the RF-/TIMBER NBR add-on module.

Menu

To start the program in the RFEM or RSTAB menu bar, click

Add-on Modules \rightarrow Design - Timber \rightarrow RF-/TIMBER NBR.

Add	l-on Modules Window	<u>H</u> elp		
400	Current Module	rüb	H - 4 > 🖗 🔭	ᆇ 💯 🐼 📾 📾 🦹 🎬 🧱 😰 🏘 🏚 🏦 🎾 🗞 💠 🛝
	Design - Steel	+ -	: 💥 🤻 🌂 🗊 🗗	🕅 📆 📆 - 🛂 - 🕜 - 17 🖘 🎿 🗇 🗊 🚏 ;
	Design - Concrete	- • [
	Design - Timber	1	RF-TIMBER	Design of timber members
	Design - Aluminium	• 2	RF-TIMBER Pro	Design of timber members
	Dynamic		RF-TIMBER AWC	Design of timber members according to AWC (LRFD or ASD)
	Connections		RF-TIMBER CSA	Design of timber members according to CSA
	Foundations	NB	RF-TIMBER NBR	Design of timber members according to NBR
	Stability	- F		
	Towers	- 1		
	Others	+		
	External Modules	÷		
	Stand-Alone Programs	÷		

Figure 1.1: Menu Add-on Modules \rightarrow Design - Timber \rightarrow RF-TIMBER NBR

Navigator

As an alternative, you can start the add-on module in the Data navigator by clicking

 $\textbf{Add-on Modules} \rightarrow \textbf{RF-/TIMBER NBR}.$

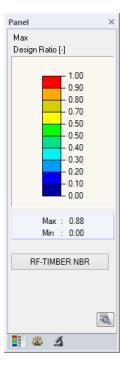
🛓 🛅 L	oads	
🛅 R	esults	
🗀 S	ections	
🛅 A	verage Regions	
🛅 P	rintout Reports	
🖶 🛅 G	uide Objects	
🗄 🚞 🗚	dd-on Modules	
ė	Favorites	
	RF-TIMBER NBR - Design of timber members according to NBR	
	RF-STEEL Surfaces - General stress analysis of steel surfaces	
1	RF-STEEL Members - General stress analysis of steel members	
	RF-STEEL EC3 - Design of steel members according to Eurocode 3	
	RF-STEEL AISC - Design of steel members according to AISC (LRFD or ASD))

Figure 1.2: Data navigator: Add-on Modules \rightarrow RF-TIMBER NBR

Panel

If results from RF-/TIMBER NBR are already available in the model, you can open the add-on module in the panel. Set the relevant RF-/TIMBER NBR design case in the load case list of the RFEM/RSTAB toolbar first.

Make sure that the solution is active so that the design results and the panel are shown. Then click the [RF-/TIMBER NBR] button in the panel to open the module.



2 Input Data

When you have started the add-on module, a new window opens. In this window, a *Navigator* is displayed on the left, managing the windows that can be currently selected. The drop-down list above the navigator contains the design cases (see Chapter 7.1, page 48).

The data relevant for the design is to be defined in several input windows. When you start RF-/TIM-BER NBR for the first time, these parameters are imported automatically:

- Members and sets of members
- Load cases, load and result combinations
- Materials

Cancel

OK

- Cross-sections
- Effective lengths
- Internal forces (in background, if calculated)

To select a window, click the corresponding entry in the navigator. To set the previous or next input window, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] window.

To save the results, click [OK]. Thus, you quit RF-/TIMBER NBR and return to RFEM or RSTAB. To exit the module without saving any changes, click [Cancel].

2.1 General Data

In the *1.1 General Data* Window, you select the members, sets of members and actions for the design. The two tabs manage the load cases, load and result combinations for the different types of design.

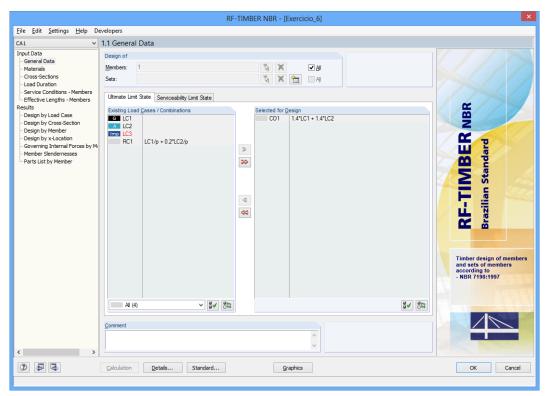


Figure 2.1: Window 1.1 General Data

2 Input Data

Design of

Design of			
Members:	174,176,177,186-196,198-206,351-362	X	🗖 All
Sets:	1-18	🖏 🗙 🎦	🔽 All

Figure 2.2: Design of members and sets of members

	X
í	*
l	ß

The design can be carried out for *Members* as well as for *Sets of Members*. If you want to design only selected objects, clear the *All* check box. Then you can access the text boxes to enter the numbers of the relevant members or sets of members. The [Delete] button clears the list of preset numbers. The [Select] button enables you to define the objects graphically in the work window of RFEM or RSTAB.

When you design a set of members, the program determines the extreme values of the analyses of all members contained in the set of members and takes into account the boundary conditions of connected members for the stability analysis. The results are shown in Windows 2.3 Designs by Set of Members, 3.2 Governing Internal Forces by Set of Members, and 4.2 Parts List by Set of Members.

Click [New] to create a new set of members. The dialog box that you already know from RFEM or RSTAB appears. There you can specify the parameters of the set of members.

Comment

<u>C</u> omment	
Design according to NBR 7190:1997	~
	\sim

Figure 2.3: User-defined comment

In this text box, you can enter user-defined notes describing, for example, the currently selected design case.

2.1.1 Ultimate Limit State

Ultimate Limit S	State Serviceability Limit S	tate				
Existing Load	Cases / Combinations		:	Selected for <u>D</u>	esign	
G LC1 A LC2				C01	1.4*LC1 + 1.4*LC2	
Imp LC3 CO2	LC1 + 0.2*LC2		>			
			>>			
			₽			
		<	4			
All (4)	~	8 ~				

Figure 2.4: Window 1.1 General Data, tab Ultimate Limit State

Existing Load Cases and Combinations

This column contains all load cases, load combinations, and result combinations that were created in RFEM or RSTAB.

To transfer selected items to the *Selected for Design* list on the right, click . Alternatively, you can double-click the items. To transfer the complete list to the right, click .

To select several items at once, click them while pressing the [Ctrl] key – as common for Windows applications.

A load case highlighted in red, like LC3 in Figure 2.4, cannot be designed: This happens when the load case is defined without any load data, or if the load cases contain only imperfections. When you transfer this load case, a warning will be shown.

Al (3)

Al (3)

C Load Cases (2)

Load Combinations (1)

R Result Combinations (1)

Load and Result Combinations (1)

Load and Result Combinations (1)

Load and Result Combinations - User-defined (1)

Load and Result Combinations - User-defined (1)

A Accidental (1)

At the end of the list, some filter options are available. They help you assign the items by load case, load combination, or action category. The buttons next to the box have the following functions:

	Selects all load cases in the list
85	Inverts the selection of load cases

Table 2.1: Buttons in the *Ultimate Limit State* tab

Selected for Design

The column on the right lists the load cases, load combinations, and result combinations selected for design. To remove an item from the list, click or double-click the item. To transfer the entire list to the left, click eq.



The design of an enveloping max/min result combination is faster than the design of all contained load cases and load combinations. In RF-/TIMBER NBR, however, only result combinations consisting of "permanent" load cases that are combined with the "OR" criterion can be designed.

2.1.2 Serviceability Limit State

Jltimate Limit	State Serviceability Lin	nit State				
Existing Load	Cases / Combinations			Selected for D	<u>)</u> esign	
G LC1 A LC2 CO1	Cases / Combinations		A V V	Selected for <u>E</u> RC1	<u>esign</u> LC1/p + 0.2°LC2/p	Permanent and live actions
All (3)		> 3√ 30				

Figure 2.5: Window 1.1 General Data, tab Serviceability Limit State

Existing Load Cases and Combinations

This section lists all load cases, load and result combinations that were created in RFEM or RSTAB.

Selected for Design



Load cases, load combinations, and result combinations can be added or removed as described in Chapter 2.1.1.

It is possible to assign different deformation limits for the deflection to the individual load cases, load and result combinations. These deformation limits are available for usual constructions (*Permanent and live actions*) and constructions with non-structural fragile materials (*Total deflection (including creep)*, Only variable actions) as well as for Vibration Design.

You can allocate the design situation via the drop-down list that you can open by clicking 🖾 at the end of the text box.

Selected for D			
S Ch CO16	G + QiA + 0.5Qs	Permanent and live actions	
S Ch CO21	G + 0.7QiA + Qs + 0.6Qw	Permanent and live actio 🗾	
		Permanent and live actions	9.2.1
		Total deflection (including creep)	9.2.2
		Only variable actions	9.2.2
		Vibration Design	9.3

Figure 2.6: Selecting design situtation

Standard...

The limit values of the deflections are controlled by the settings in the *Standard* dialog box (see Figure 3.3, page 27). To adjust those values, click the [Standard] button..

In the *1.9 Serviceability Data* Window, the reference lengths relevant for the deformation check are to defined (see Chapter 2.9, page 24).



For the serviceability limit state design, the effective modulus of elasticity, $E_{c_0,ef}$ should be used. However, all modification factors that are defined for the design of RF-/TIMBER NBR are not used in the calculation of RFEM/RSTAB. Therefore, it is necessary to define the final modification factor also in the calculation parameters of the relevant load case or load combination.



2.1.3 Standard

Standard...

To check and, if necessary, adjust the default parameters, click the [Standard] button in any input window. The *Standard* dialog box appears. It consists of three tabs.

Modification Factors

	Standard - NE	BR 7190:1997	×
Modification Factors Other Settings Used	Standards		
Type of Timber Solid Timber Glued Laminated Timber Plywood Recomposted Timber	Modification Factors Acc. to 6.4. Load Duration Class: - Permanent - Long-term - <u>M</u> edium-term - <u>S</u> hort-term - I <u>n</u> stantaneous	4 and Table 10 kmod, 1: 0.600 € kmod, 1: 0.700 € kmod, 1: 0.800 € kmod, 1: 0.900 € kmod, 1: 1.100 €	
	Modification Factors Acc. to 6.4. Moisture Class: - (1) - (2) - (3) - (4) Submerged timber:	4 and Table 11 kmod, 2 : 1.000 ★ kmod, 2 : 1.000 ★ kmod, 2 : 0.800 ★ kmod, 2 : 0.800 ★ kmod, 2 : 0.650 ★	
Ā	Modification Factors Acc. to 6.4. Classification of timber: - First Category - Second Category	4 kmod, 3 : 1.000 - kmod, 3 : 0.800 -	
2 · · · ·			OK Cancel

Figure 2.7: Dialog box Standard, tab Modification Factors

In the three sections of this tab, you can check or change the *Modification Factors*, $k_{mod,1}$, $k_{mod,2}$, and $k_{mod,3}$, if necessary.

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

Button	Function
3	Resets the default settings of the program
	Imports user-defined default settings
	Saves the current settings as default

Table 2.2: Buttons in the Standard dialog box

Other Settings

In the second tab of the *Standard* dialog box, you find other factors significant for the design: *Safety Factor of Resistance for Ultimate Limit State, Creep Coefficient, Factors for LTB Design* and *Interaction Factor.* These factors can also be modified, if necessary. Furthermore, the parameters of the Deformation Limits and Vibration Design are indicated.

Modification Fa Safety Factor Basic value fo

Basic value fo Creep Coeffici

			Standard	- NBR 7190):1997		
Modification Factors Oth	er Settings	Used Stand	dards				
Safety Factor of Resista	nce for Ultin	nate Limit Sta	te Acc. to 6.4.5			Factors for LTB Design Acc. t	0 7.5.6
Basic value for compres	sion parallel	to grain	Ywc :	1.400 🜩 [-]	1	γf: 1.400 🜩 [-]	
Basic value for tension p	arallel to gra	ain	γwt∶	1.800 🗘 [·]]	βε: 4.000 ÷ [·]	
Basic value for shear pa	rallel to grair	ı	Ywv I	1.800 ‡ [·]]		
Creep Coefficient Acc. to	Table 15					Interaction Factor Acc. to 7.3.	.4
			Moisture Class				
Load Duration Class		1	2	3	4	Rectangular cross-sections	
- Permanent	Ф:	0.800 🜩	0.800 🖨	2.000 ≑	2.000 ≑	km: 0.500 🜩 [·]	
- Long-term	Ф:	0.800 🜩	0.800 ≑	2.000 ≑	2.000 ≑		
- Medium-term	Ф:	0.300 🜩	0.300 ≑	1.000 🜩	1.000 ≑		
- Short-term	Ф:	0.100 🜲	0.100 🜩	0.500 🜩	0.500 🜩		
Deformation Limits Acc. t	0.9.2						

2

- Medium-term	Φ:	0.300 🜲	0.300 ≑	1.000 🜲	1.000 🜩		
- Short-term	Φ:	0.100 🜲	0.100 🖨	0.500 🜲	0.500 🜩		
Deformation Limits Ac	c. to 9.2						
Usual constructions a	acc. to 9.2.1	:					
			Fixed on bo	th sides	Overhanging		
Permanent and live a	ctions:		≤L/ 2	00 🗘	≤Lc/ 100	D ≑	
Constructions with no	n-structural	tragile materials a	cc. to 9.2.2:				
Total deflection (inclu	iding creep)		≤L/ 3	50 🚖	≤Lc/ 175	5 🜩	Absolute deformation
Only variable actions:			≤L/ 3	00 🗘	≤Lc/ 150)	15.000 ≑ [mm]
Vibration Design Acc.	to 9.3						
Deflection limit:			15.0	00(‡ [mm]			
Denection minit.			15.0	od 🔺 (uuu)			
2 000	r or						OK Cancel

Figure 2.8: Dialog box Standard, tab Others

Used Standards

The third tab of the Standard dialog box informs you about the Standards according to which the design is performed.

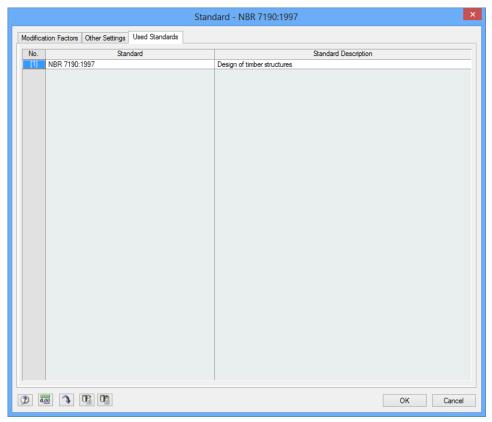


Figure 2.9: Dialog box Standard, tab Used Standards

2.2 Materials

The window is divided into two parts: The upper table lists all materials created in RFEM or RSTAB. The *Material Properties* section below shows the characteristics of the current material, i.e. the row currently selected in the upper table.

Material Description	Type of Timbe				
Description	Type of Timber				
			Comment		
Hardwood Timber C 40 NBR 7190:1997	Solid Timber				
Steel AR 350 ABNT NBR 8800:2008	Solid Timber				
Softwood Timber C 20 NBR 7190:1997	Solid Timber				
Softwood Timber C 30 NBR 7190:1997	Glued Laminated 1	Timber			
erties					
erties					
s of Elasticity	E	19500.000	MPa		
Aodulus	G	975.000	MPa		
: Weight	γ	9.50	kN/m ³		
ent of Thermal Expansion	α	5.0000E-06	1/K		
Safety Factor	γM	1.00			
Properties				Makasiath	In Allowed in
Shear Strength	fR,k			Material N	io. 1 used in
	E _{c0,m}			Cross-sec	ctions No.:
	Ec90,m			1	
				Mamham	No.:
	ft90,k				NO
	fc0,k			1	
	fc90,k				
eristic Strength for Shear/Torsion	fv0,k			Sets of m	embers No.:
ensity	ρbas,m	750.0	kg/m ³		
	Softwood Timber C 20 NBR 7190.1997 Softwood Timber C 30 NBR 7190.1997 Softwood Timber C 30 NBR 7190.1997 erties erties erties erties erties erties erties erties erties erties erties erties erties froperties Shear Strength of Elasticity for Compression ertistic Strength for Tension ertistic Strength for Compression ertistic Strength for Shear/Torsion	Softwood Timber C 201 NBR 7190:1997 Solid Timber Softwood Timber C 301 NBR 7190:1997 Glued Laminated ' Glued Laminated ' Glued Laminated ' erties arties of Basticity E lodulus G Weight γ ant of Themal Expansion α afety Factor γM Properties Shear Strength of Basticity for Compression Perpendicular Ee30,m eristic Strength for Tension fro.k eristic Strength for Compression Perpendicular fro.k	Softwood Timber C 20 NBR 7190.1997 Solid Timber Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Softwood Timber C 30 NBR 7190.1997 Glued Laminated Timber erties Glued Laminated Timber Softwood 70 NBR 7190.100 Weight T 7 or Timoin Perpendicular Fe30, M 1.00 erties Strength for Compression Perpendicular Fe30, K 1.000 erties Strength for Compression Perpendicular Fe30, K 1.000 erties Strength for Compression Perpendicular Fe30, K 0.00	Softwood Timber C 201 NBR 7190:1997 Solid Timber Softwood Timber C 301 NBR 7190:1997 Glued Laminated Timber Glued Laminated Timber Glued Laminated Timber erties File arties Softwood MPa obdulus G 975.000 MPa obdulus G 975.000 MPa obdulus G 975.000 MPa Weight γ 9.50 kN/m³ ant of Themal Expansion α 3 of Elasticity for Compression E eo.m 975.000 MPa Shear Strength f R.k 9 of Elasticity for Compression Perpendicular E sol.m 1 of Elasticity for Compression Perpendicular f tol.k 1 of Elasticity for Tension f tol.k 2 of Elasticity for Compression f tol.k	Softwood Timber C 20 NBR 7190:1997 Solid Timber Softwood Timber C 30 NBR 7190:1997 Glued Laminated Timber Softwood Timber C 30 NBR 7190:1997 Glued Laminated Timber Settwood Timber C 30 NBR 7190:1997 Glued Laminated Timber Settwood Timber C 30 NBR 7190:1997 Glued Laminated Timber Setters Setters Setters Setters Softwood Timber C 30 NBR 7190:1997 Glued Laminated Timber Setters Setters Setters Setters <

Figure 2.10: Window 1.2 Materials

Materials that will not be used in the design appear gray in color. Materials that are not allowed are highlighted in red. Modified materials are displayed in blue.

The material properties required to determine the internal forces (*Main Properties*) are described in Chapter 4.3 of the RFEM manual and Chapter 4.2 of the RFEM manual. The material properties required for design (*Additional Properties*) are stored in the global material library. These values are preset.

To adjust the units and decimal places of the material properties and stresses, select **Settings** \rightarrow **Units and Decimal Places** from the menu bar of the module (see Chapter 7.3, page 52).

Material Description

The materials defined in RFEM or RSTAB are preset, but you can always modify them: To select a material, click the description in column A. Then click I or press function key [F7] to open the list of materials.

Hardwood Timber C 40	NBR 7190:1997	<u> </u>
Softwood Timber C 20	NBR 7190:1997	
Softwood Timber C 25	NBR 7190:1997	
Softwood Timber C 30	NBR 7190:1997	
Hardwood Timber C 20	NBR 7190:1997	
Hardwood Timber C 30	NBR 7190:1997	
Hardwood Timber C 40	NBR 7190:1997	
Hardwood Timber C 60	NBR 7190:1997	

Figure 2.11: List of materials

According to the design concept of the NBR 7190:1997 [1], the list includes only materials of the NBR standard group.

When you have imported a material, the design relevant Material Properties are updated.

If you change the material description manually and the entry is stored in the material library, RF-/TIMBER NBR imports the material properties, too.

It is not possible to edit the material properties in the add-on module RF-/TIMBER NBR.

Type of Timber



Solid Timber is the default type of timber, but it is possible to change it to *Glued Laminated Timber*, *Plywood* or *Particle Board*: To select, click the type of timber in column B. Then click or press [F7] to open the list of types. The type of timber type is important for the modification factors that are applied in the design.

Material Library

Many materials are available in the library. To open the Material Library, select on the menu

Edit ightarrow Material Library

or click the corresponding button.

	Material Libra	ſy				
Filter	Material to Select					
Material category group:	Material Description	Standard				
Timber	Softwood Timber C 20	NBR 7190:1997				
	Softwood Timber C 25	S NBR 7				
Material category:	Softwood Timber C 30	S NBR 7				
	Hardwood Timber C 20	S NBR 7				
	Hardwood Timber C 20	NBR 7				
Standard group:						
S NBR	Hardwood Timber C 40	S NBR 7				
	Hardwood Timber C 60	💽 NBR 7	190:1997			
Standard:						
NBR 7190:1997	✓					
Indude invalid				×		
Material Properties		Softwood	d Timber C 20 N	NBR 7190:199		
☐ Main Properties						
Main Properties Modulus of Elasticity		E	3500.000	MPa		
Main Properties Modulus of Elasticity Shear Modulus		E G	3500.000 175.000	MPa MPa		
Main Properties Modulus of Elasticity Shear Modulus Specific Weight	ision	E	3500.000 175.000 5.00	MPa MPa kN/m ³		
Main Properties Modulus of Elasticity Shear Modulus	nsion	E G γ	3500.000 175.000	MPa MPa kN/m ³		
Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expan	nsion	E G γ	3500.000 175.000 5.00	MPa MPa kN/m ³ 1/K		
Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expan Additional Properties Rolling Shear Strength Modulus of Elasticity for Con	pression	Ε G γ α	3500.000 175.000 5.000E-06 1.000 3500.000	MPa MPa kN/m ³ 1/K MPa MPa		
 ☐ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expan ☐ Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con 	npression npression Perpendicular	Ε G γ α fR,k Ec0,m Ec90,m	3500.000 175.000 5.000E-06 1.000 3500.000 175.000	MPa MPa kN/m ³ 1/K MPa MPa MPa		
 ➡ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expansion Additional Properties Rolling Shear Strength Modulus of Elasticity for Cont Modulus of Elasticity for Cont Characteristic Strength for B 	npression npression Perpendicular ending	E G γ α fR,k Ec0,m Ec90,m ftM,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa		
 ➡ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expansion Additional Properties Rolling Shear Strength Modulus of Elasticity for Control Modulus of Elasticity for Control Characteristic Strength for B Characteristic Strength for Tormation 	npression npression Perpendicular ending ension	E G γ α fR,k Ec0,m Ec90,m ftM,k ft0,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa		
 ➡ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expan Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con Characteristic Strength for B Characteristic Strength for T Characteristic Strength for T 	ipression ipression Perpendicular ending ension ension Perpendicular	E G γ α fR,k E c0,m E c90,m f tM,k f t0,k f t90,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970 1.300	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa MPa		
Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expan Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con Characteristic Strength for T Characteristic Strength for T	pression pression Perpendicular ending ension ension Perpendicular ompression	E G γ α FR,k Ec0,m Ec90,m ftM,k ft0,k ft30,k ft30,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970 1.300 20.000	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa MPa MPa		
Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expai Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con Characteristic Strength for T Characteristic Strength for C Characteristic Strength for C	pression pression Perpendicular ending ension ension Perpendicular ompression ompression Perpendicular	E G γ α fR,k Ec0,m Ec90,m ftM,k ft0,k ft90,k fc90,k fc90,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970 1.300 20.000 5.000	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa MPa MPa MPa		
➡ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expai Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con Characteristic Strength for T Characteristic Strength for T Characteristic Strength for T Characteristic Strength for C Characteristic Strength for C Characteristic Strength for C	pression pression Perpendicular ending ension ension Perpendicular ompression ompression Perpendicular	E G γ α FR.k Ec0.m Ec90.m ftM.k ft0,k ft0,k ft0,k fo0,k fc0,k fc0,k fc0,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970 1.300 20.000 5.000 4.000	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa MPa MPa MPa MPa MPa		
➡ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expai Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con Characteristic Strength for B Characteristic Strength for T Characteristic Strength for C Characteristic Strength for C	pression pression Perpendicular ending ension ension Perpendicular ompression ompression Perpendicular	E G γ α fR,k Ec0,m Ec90,m ftM,k ft0,k ft90,k fc90,k fc90,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970 1.300 20.000 5.000 4.000	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa MPa MPa MPa		
➡ Main Properties Modulus of Elasticity Shear Modulus Specific Weight Coefficient of Thermal Expai Additional Properties Rolling Shear Strength Modulus of Elasticity for Con Modulus of Elasticity for Con Characteristic Strength for T Characteristic Strength for T Characteristic Strength for T Characteristic Strength for C Characteristic Strength for C Characteristic Strength for C	pression pression Perpendicular ending ension ension Perpendicular ompression ompression Perpendicular	E G γ α FR.k Ec0.m Ec90.m ftM.k ft0,k ft0,k ft0,k fo0,k fc0,k fc0,k fc0,k	3500.000 175.000 5.000E-06 1.000 3500.000 175.000 25.970 25.970 1.300 20.000 5.000 4.000	MPa MPa kN/m ³ 1/K MPa MPa MPa MPa MPa MPa MPa MPa MPa MPa		

Figure 2.12: Dialog box Material Library

In the *Filter* section, *NBR 7190:1997* is the default Standard. Select the material quality that you want to use for the design in the *Material to Select* list. You can check the corresponding properties in the dialog section below.

OK

Click [OK] or press [←] to transfer the selected material to Window 1.2 of RF-/TIMBER NBR.

Chapter 4.3 of the RFEM manual and Chapter 4.2 of the RSTAB manual describe in detail how materials can be filtered, added, or rearranged.

Material Properties

In the lower section of Window 1.2, the characteristic strengths for tension, $f_{t_0,k'}$ compression, $f_{c_0,k'}$ shear and torsion, $f_{v_0,k'}$ as well as other material properties are specified.

The design values of the material strengths are, as shown for example in [1] Chapter 7.2.6, to be determined with the modification factors k_{mod} and the safety factors of resistance γ_w .

$$f_{wd} = k_{\text{mod}} \cdot \frac{f_{wk}}{\gamma_w} \tag{2.1}$$

Standard...

Those factors can be modified in the Standard dialog box (see Figure 2.7, page 9), if necessary.

2.3 Cross-Sections

This window manages the cross-sections used for design. In addition, the module window allows you to specify optimization parameters.

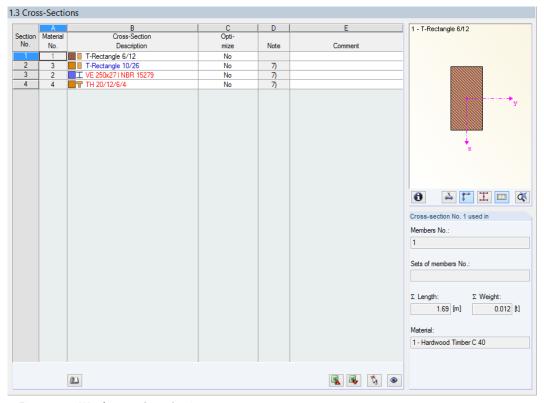
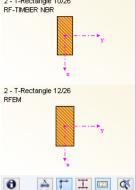


Figure 2.13: Window 1.3 Cross-Sections





Cross-Section Description

The cross-sections defined in RFEM or RSTAB are preset, together with their material numbers. The design is possible for the following types of cross-sections of the library:

- Parametric timber rectangular cross-sections
- Parametric timber circular cross-sections
- Standardized timber rectangular cross-sections

The new cross-section description can be entered in the text box. If the data base contains that entry, RF-/TIMBER NBR imports the cross-section parameters.

Modified cross-sections are highlighted in blue.

If the cross-section defined in RF-/TIMBER NBR is different from the one of RFEM or RSTAB, both sections are displayed in the graphic area. The design will then be performed with the internal forces of RFEM/RSTAB for the section defined in RF-/TIMBER NBR.

To modify a cross-section, click the entry in column B to select the text box. Use the **(L)** button or click **(L)** in the box, or press [F7] to open the dialog box of the current cross-section.

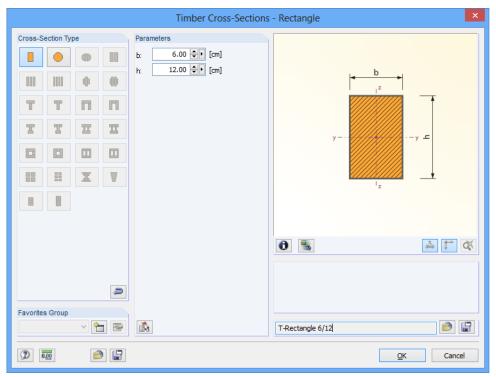


Figure 2.14: Parametric Timber Cross-Sections of the library

In the dialog box, you can select a different cross-section or a different cross-section table. To change the cross-section category, click and access the global cross-section library.

Chapter 4.13 of the RFEM manual and Chapter 4.3 of the RSTAB manual describe how cross-sections can be selected from the library.

Max. Design Ratio

This table column is shown only after the calculation. It is useful for the optimization: By means of the displayed design ratios and colored relation scales, you can see which cross-sections are little utilized and thus oversized, or overloaded and thus undersized.

Optimize

Details...

The optimization analyzes which cross-section comes as close as possible to a user-defined maximum utilization ratio. You can define this maximum ratio in the *Other* tab of the *Details* dialog box (see Figure 3.4, page 28).

If you want to optimize a cross-section, select the corresponding check box in column D or E. Recommendations on the optimization of cross-sections can be found in Chapter 7.2 on page 50.

Note

This column shows remarks as footers. They are described below the cross-section list.

Info About Cross-Section

0

In the Info About Cross-Section dialog box, you can check the section properties and stress points.

3	Info Abou	ut Cross-S	ection T-R	lectangle 6/12
Cross-Section Property	Symbol	Value	Unit	T-Rectangle 6/12
Width	b	6.00	cm	
Depth	d	12.00	cm	
Cross-sectional area	Α	72.00	cm ²	
Noment of inertia	Iγ	864.00	cm ⁴	6.00
1oment of inertia	Ιz	216.00	cm ⁴	+
Governing radius of gyration	ry	3.46	cm	
Soverning radius of gyration	rz	1.73	cm	
Veight	wt	6.8	kg/m	
urface	Asurf	0.360	m²/m	
orsional constant	J	593.24	cm ⁴	
lastic section modulus	Sy,max	144.00	cm ³	¥
lastic section modulus	S _{y,min}	-144.00	cm ³	
lastic section modulus	S _{z,max}	72.00	cm ³	
astic section modulus	Sz,min	-72.00	cm ³	
tatical moment of area	Q _{y,max}	108.00	cm ³	
tatical moment of area	Q _{z,max}	54.00	cm ³	
				Image: Stress points Image: Stress points Image: Stress points Image: Stress points Image: Stress points Image: Stress points Image: Stress points Image: Stress points
2				Close

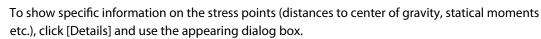
Figure 2.15: Dialog box Info About Cross-Section

The buttons below the graphic of the cross-section have the following functions:

Button	Function
Ŧ	Displays or hides the stress points
123	Displays or hides the numbers of stress points
	Shows the details of the stress points (see Figure 2.16)
×	Displays or hides the dimensions of the cross-section
*	Displays or hides the principal axes of the cross-section
X	Resets the full view of the cross-section graphic
	Prints the cross-section values and cross-section graphic

Table 2.3: Buttons of cross-section graphic

9



				Stress	Points of 1	-Rectangle	6/12	×
	A	B	C	D	E	F	G	T-Rectangle 6/12
StressP	Coordi	nates	Statical Mom	ents of Area	Thickness	Waŋ		-
No.	y [cm]	z [cm]	Qy [cm ³]	Q _z [cm ³]	t [cm]	W _{no} [cm ²]	Qw [cm 4]	
1	3.00	6.00	0.00	0.00	6.00	0.00	0.00	
2	-3.00	6.00	0.00	0.00	6.00	0.00	0.00	
3	-3.00	-6.00	0.00	0.00	6.00	0.00	0.00	
4	3.00	-6.00	0.00	0.00	6.00	0.00	0.00	3 4
5	0.00	0.00	108.00	0.00	6.00	0.00	0.00	
6	0.00	0.00	0.00	54.00	12.00	0.00	0.00	
								2 1 1 z
								ă 🕇 🛱
2								Close

Figure 2.16: Dialog box Stress Points

2.4 Load Duration

In this window, you define the load duration to consider factors reflecting the different load duration for all selected load cases, load and result combinations, as well as dynamic combinations.

	А	В	C	D	E	F	G	Н	Load Duration - Explanatory Notes
ad-		Load	Load Duration	Modification		Combination	Coefficient		
ng	Description	Туре	Class	Factor k mod,1	Criterion	¥1	Ψ2	Comment	Permanent: Design working life
C1		Permanent	Permanent	0.600	Permanent				Design working me
C2		Accidental	Long-term	0.700	Permanent				Long-term:
	1.4*LC1 + 1.4*LC2	-	Long-term	0.700		0.300	0.200		More than six months
02	LC1 + 0.2*LC2	-	Long-term	0.700		1.000	1.000		
									Medium-term:
									One week to six months
									Short-term:
									Less than one week
									Instantanenous:
									Very short

Figure 2.17: Window 1.4 Load Duration

Loading

All actions selected in the 1.1 General Data Window are listed here. For combinations, all contained load cases are listed as well.

Description

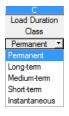
The descriptions of the load cases help you to classify them.

Load Type

This table column shows the action type of each load case as defined in RFEM or RSTAB. They are the basis for the default settings in the next column.

Load Duration Class

Loads and their superpositions must be assigned to the load duration classes (LDC) for the design. The classification of actions is specified e.g. in [1] Table 1.



For load cases and result combinations, the load duration can be changed via the list shown on the left: Click the cell in column C to select the box. The I button becomes available. For load combinations and "Or" result combinations, RF-/TIMBER NBR performs the classification automatically, taking into account the shortest load duration class of all contained load cases.

The load duration class is required to determine the modification factor, $k_{mod,1}$. This factor is also considered for the material stiffness.

Modification Factor $k_{mod,1}$

The impact of the load duration and type of timber on the strength properties is taken into account by means of the modification factor (see [1] Table 10).

Standard...

The factors can be checked in the Standard dialog box (see Figure 2.7, page 9).

Criterion



For load cases, the criterion *Permanent* or *Variable* can be changed via the list shown on the left. This option distributes the loading on permanent and variable components for the stability calculation according to [1] 7.5.5.

Column F is only displayed when the corresponding option in the *Stability* tab of the *Details* dialog box is selected (see Figure 3.2, page 26).

Combination Coefficient ψ_{1}/ψ_{2}

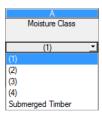
If compression members are defined by slendernesses $80 < \lambda < 140$, a buckling analysis can be performed according to [1] 7.5.5. Those settings can be activated in the *Stability* tab of the *Details* dialog box (see Figure 3.2, page 26). In this case, it is necessary to define the factors ψ_1 and ψ_2 for each combination in columns F and G.



In this window, moisture classes are allocated to members. Besides, the first or second category of timber is assigned. This makes it possible to modify strengths by other modification factors.

	А	B	C	DÍ	E	Moisture Class
ember	Moisture Class	Classification	Modificatio	n Factors		
No.		of Timber	k mod.2	k mod.3	Comment	Moisture Class (1):
1	(1)	Second Category	1.000	0.800		Relative humidity of environment:
2	(2)	First Category	1.000	1.000		U _{amb} ≤ 65%
3	(2)	First Category	1.000	1.000		Average moisture content of timber:
4	(3)	Second Category	0.800	0.800		U _{eq} = 12%
	(-)					Moisture Class (2):
						Relative humidity of environment:
						65% < U amb ≤ 75%
						Average moisture content of timber:
						U _{eq} = 15%
						Moisture Class (3):
						Relative humidity of environment:
						75% < U _{amb} ≤ 85%
						Average moisture content of timber:
						U _{eq} = 18%
						Moisture Class (4):
						Relative humidity of environment:
						U _{amb} > 85% for long periods
						Average moisture content of timber: U _{eq} ≥ 25%
						Classification of Timber
						First Category:
						The condition of first category timber can only be assumed if all the structural members are without defects. This has to the ensured by a visual method and by mechanical tests, which guarantees the homogenity of the applied timber.
						Second Category:
						All the types of timber which are not of firs
Set inpu	t for members No.:					category.
		*	✓ All		🕱 🖏	

Figure 2.18: Window 1.5 Service Conditions – Members



Classification

of Timber

Second Category First Category Second Category

Moisture Class

There are four moisture classes defined in [1] Table 7. The determination of the moisture class makes it possible to assign the modification factor $k_{mod,2}$. This factor is also dependent on the type of timber as specified in [1] Table 11.

Classification of Timber

The conditions of the first and second category of timber are specified in [1] 6.4.4. According to the classification, the modification factor $k_{mod,3}$ can be determined.

Modification Factors $k_{mod,2}/k_{mod,3}$

The modification factors $k_{mod,2}$ and $k_{mod,3}$ are automatically determined according to the choice in the previous columns A and B.

Standard...

These factors can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.7, page 9).

Below the table, you find the *Set input for members No.* check box. If it is selected, the settings entered <u>afterwards</u> will be applied to the selected or to *All* members. Members can be selected by entering their numbers. You can also select them graphically with the abutton. This option is useful when you want to assign the same conditions to several members. Please note that settings that have been already defined cannot be changed subsequently with this function.

2 Input Data

There are three more buttons available beneath the table. They have the following functions:

Button	Function
	Exports the table to MS Excel
N	Sets the row of member which can be selected in work window of RFEM/RSTAB
۲	Switches to work window of RFEM/RSTAB

Table 2.4: Buttons in Window 1.5 Service Conditions – Members

2.6 Service Conditions - Set of Members

This window appears if at least one set of members has been selected for the design in Window *1.1 General Data*.

	A	В	C	D	E	Moisture Class
Set	Moisture Class	Classification	Modificatio			Moisture Class (1):
No.		of Timber	k mod,2	k mod,3	Comment	Relative humidity of environment:
1	(1) 🗾	First Category	1.000	1.000		
		rinst Category	1.000	1.000		$\label{eq:constraint} \begin{array}{l} U_{amb} \leq 65\% \\ Average moisture content of timber: \\ U_{eq} = 12\% \\ \\ Moisture Class (2): \\ Relative humidity of environment: \\ 65\% < U_{amb} \leq 75\% \\ \\ Average moisture content of timber: \\ U_{eq} = 15\% \\ \\ \\ Moisture Class (3): \\ Relative humidity of environment: \\ 75\% < U_{amb} \leq 85\% \\ \\ Average moisture content of timber: \\ U_{eq} = 18\% \\ \\ \\ \\ Moisture Class (4): \\ \\ Relative humidity of environment: \\ U_{amb} \geq 85\% \mbox{ for long periods} \\ \\ Average moisture content of timber: \\ U_{eq} \geq 25\% \\ \end{array}$
						Classification of Timber
						First Category: The condition of first category timber can only be assumed if all the structural members are without defects. This has to be ensured by a visual method and by mechanical tests, which guarantees the homogenity of the applied timber.
						Second Category: All the types of timber which are not of first
Set ir	nput for members No.:	1	✓ Al		A 🛛	category.

1.6 Service Conditions - Sets of Members

Figure 2.19: Window 1.6 Service Conditions - Sets of Members

The concept of this window is similar to the one of the previous Window 1.5 Service Conditions - Members. Here you can assign the moisture and timber classes to sets of members.

2.7 Effective Lengths - Members



The appearance of the window depends on whether the stability analysis is carried out according to [1] 7.5 or second-order analysis. You select the method in the *Stability* tab of the *Details* dialog box (see Figure 3.2, page 26). The following description refers to the equivalent member method. Here, the parameters of buckling and lateral-torsional buckling have to be defined.

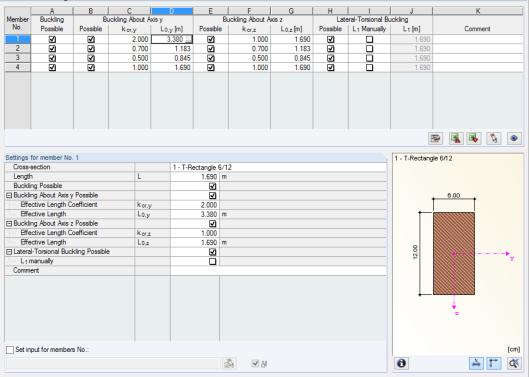


If the stability analysis is deactivated in the *Stability* tab of the *Details* dialog box, this window is not shown.

Window 1.7 consists of two parts: The upper table informs you about the factors concerning the lengths of buckling and lateral-torsional buckling of the members selected for the design. Furthermore, the equivalent member lengths are listed. The effective lengths defined in RFEM or RSTAB are preset. In the *Settings* section of this window, you can see further information on the member whose row is selected in the upper table.

Click 🚮 to select a member graphically and show its row.

You can make changes in the table as well as in the Settings tree.



1.7 Effective Lengths - Members

Figure 2.20: Window 1.7 Effective Lengths - Members for stability analysis according to [1] 7.5

The effective lengths can be entered manually in the table and in the *Settings* tree, or defined graphically in the work window after clicking . This button is enabled when you click in the box (see Figure 2.20).

The Settings tree manages the following parameters:

- Cross-Section
- Length of member
- Buckling Possible for member (cf. column A)
- Buckling about Axis y Possible (cf. columns B to D)
- Buckling about Axis z Possible (cf. columns E to G)
- Lateral-Torsional Buckling Possible (cf. columns H to J)

You can specify for the selected member whether a buckling or a lateral-torsional buckling design is to be performed. Furthermore, you can adjust the *Effective Length Coefficient* for the respective directions. When a coefficient is modified, the equivalent member length is adjusted automatically, and vice versa.

~

You can also define the buckling length of a member in a dialog box. To open it, click the button shown on the left. It is located on the right below the upper table.

Select Effective	e Length Factor
Buckling About y-Axis	Buckling About z-Axis
$\begin{array}{c c} k_{cr,y} = 2.0 \\ \hline \\ \text{Hinged - hinged} \\ k_{cr,y} = 1.0 \end{array}$	$k_{cr,z} = 2.0$ $\textcircled{linged - hinged}_{k_{cr,z} = 1.0}$
Rigid - hinged Z $k_{\sigma,\gamma} = 0.7$ Z	Rigid - hinged y kcr,z = 0.7 y
Rigid - rigid $k_{\sigma,\gamma} = 0.5$	O Rigid - rigid k _a ,z = 0.5
Ouser-defined ka,y =	User-defined krr,z =
Import from add-on module RF-STABILITY (Eigenvalue Analysis)	Import from add-on module RF-STABILITY (Eigenvalue Analysis) DE CTABILITY Comp
RF-STABILITY-Case:	RF-STABILITY-Cas <u>e</u> :
Bugkling mode No.:	Buckling mode No.:
Export effective length factor ker.y : 2.000 ‡ [-]	Export effective length factor kor,z : 1.000 🗘 [-]
	OK Cancel



For each direction, you can select one of the four EULER buckling modes or specify a *User-defined* buckling length coefficient. If an eigenvalue analysis was carried out in the add-on modules RF-STABILITY or RSBUCK, you can also select a *Buckling mode* in order to determine the coefficient.

Buckling Possible

A stability analysis for flexural buckling and lateral-torsional buckling requires that members can resist compressive forces. Therefore, members for which such resistance is not possible because of their member types (e.g. tension members, elastic foundations, rigid couplings) are generally excluded from the design. The corresponding rows appear dimmed and a note is displayed in the *Comment* column.

The *Buckling Possible* check boxes in table column A and in the *Settings* tree offer the possibility to control the stability analyses: They determine whether or not these analyses are carried out for a member.

Buckling About Axis y or Axis z

With the check boxes in the *Possible* table columns, you decide whether a member is susceptible to buckling about the y-axis and/or z-axis. Those axes represent the local member axes, where the y-axis is the major and the z-axis the minor member axis. The buckling length coefficients $k_{cr,y}$ and $k_{cr,z}$ for buckling about the major or the minor axis can be selected freely.

0

You can check the position of the member axes in the cross-section graphic in the *1.3 Cross-Sections* Window (see Figure 2.13, page 13). To access the work window of RFEM or RSTAB, click the [View mode] button. In the work window, you can display the local member axes via the member shortcut menu or the *Display* navigator (see Figure 2.22).

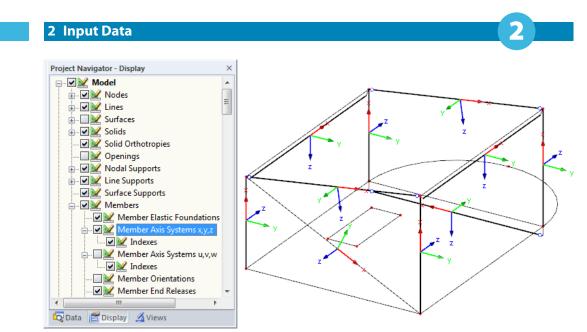


Figure 2.22: Activating the member axis systems in the Display navigator of RFEM

If buckling is possible about one or even both member axes, you can enter the buckling length coefficients as well as the buckling lengths in columns C and D as well as F and G. The same is possible in the *Settings* tree.

To specify the buckling lengths in the work window graphically, click \square . This button is available when you have selected a cell of the L₀ column (see Figure 2.20).

When you specify the buckling length coefficient k_{cr} , the program determines the effective length L_0 by multiplying the member length L by the buckling length coefficient. The text boxes of k_{cr} and L_0 are interactive.

Lateral-Torsional Buckling Possible

Table column H shows you for which members the program performs an analysis of lateral-torsional buckling.

L₁ Manually

The equivalent member length relevant for the lateral-torsional buckling is preset by the member length. Having selected the check box in column I, you can specify the length for lateral-torsional buckling L_1 in column J. You can also define it graphically after clicking \Box as the distance of the lateral supports. It can be useful to adjust a structural component if it consists of several members between the supports.

Below the *Settings* table, you find the *Set input for members No.* check box. If it is selected, the settings entered <u>afterwards</u> will be applied to the selected or to *All* members. Members can be selected by entering their numbers. You can also select them graphically with the button. This option is useful when you want to assign the same boundary conditions to several members. Please note that settings that have already been defined cannot be changed subsequently with this function.

Comment

In the last table column, you can enter you own comments to describe, for example, the effective lengths of specific members.



2.8 Effective Lengths - Sets of Members

This window appears only if at least one set of members has been selected for the design in Window 1.1 General Data and the stability check has been activated in the Stability tab of the Details dialog box (see Figure 3.2, page 26).

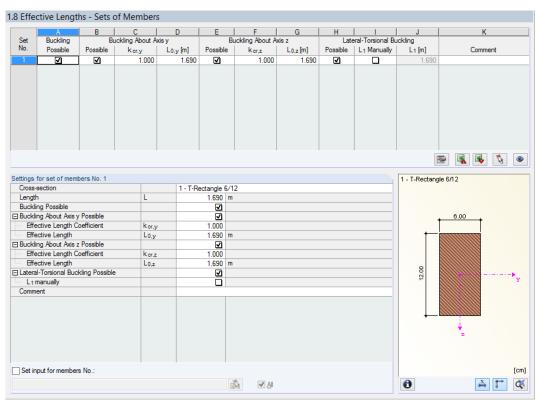


Figure 2.23: Window 1.8 Effective Lengths - Sets of Members

The concept of this window is similar to the one of the previous window 1.7 Effective Lengths - Members. Here you can enter the effective lengths for buckling as well as for lateral-torsional buckling of the set of members as described in Chapter 2.7. They determine the boundary conditions of the entire set of members that is treated as an equivalent member.

2.9 Serviceability Data

The last input window controls the settings for the serviceability limit state design of specific objects. It is available when one or more load cases or combinations have been selected in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 8).

	A	B	C	D	E	F	G	Н	
		Member		ce Length	Direc-	Preca	mber		
) .	Reference to	No.	Manually	L [m]	tion	wc,y[mm]	w _{c,z} [mm]	Beam Type	Comment
	Member	1		1.690	z		0.000	Beam	_
2	Member	2		1.690	z		2.000	Beam	
3	Member	3		1.690	у	0.000		Cantilever Start Free	
4	Member	4		1.690	У	0.000		Cantilever Start Free	
5	List of Members	1-4		6.760	y; z	0.000	0.000	Beam	
6	Set of Members	1		1.690	R	0.000	0.000	Beam	
7									
8									
9									
10									
11									
12									
13									
14									
15									
16									
17									
18									
19									
20									
21									
22									
23									
24									
25									
26									
27									
28									
29									
30									
31									
32									
33									

Figure 2.24: Window 1.9 Serviceability Data



Direction

y;z 💌

y;z R In column A, you define whether the deformation refers to single members, lists of members, or sets of members.

For a list or set of members, the orientation and rotation of all contained members must be identical. This will guarantee that the components of the deformation are taken into account correctly.

In column B, you can specify the numbers of the members or sets of members that are to be analyzed. The ... button enables you to select the objects graphically in the work window. In column D, the *Reference Length* of each object is shown. The geometrical lengths of the members, lists or sets of members are set by default. If necessary, you can adjust those values after having selected the *Manually* check box in column C.

Column E controls the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z or the resultant *R*.

Columns F and G enable you to consider a *Precamber* $w_{c,y}$ and $w_{c,z}$ for the design.

Beam Type	
Beam	
Beam	
Cantilever Start Free	
Cantilever End Free	
	_
Details	

The *Beam Type* is important for the correct reference to the limit deformations. In column G, you can specify a beam or a cantilever is to be analyzed. For the latter, you can define which end has no support.

The settings in the *Serviceability* tab of the *Details* dialog box control whether the deformations refer to the undeformed original model or to the shifted ends of members or sets of members (see Figure 3.3, page 27).

3 Calculation

3.1 Detail Settings

Before you start the calculation, it is recommended to check the design details. You can open the corresponding dialog box in all windows of the add-on module by clicking [Details].

The Details dialog box contains the following tabs:

- Resistance
- Stability
- Serviceability
- Other

Details...

3.1.1 Resistance

De	etails ×
Resistance Stability Serviceability Other	
Consideration of Connections	
✓ <u>R</u> eduction of limit tension stresses	
At nodes No.:	
1,2,6,8	
Connection length: 0.200 🜩 [m]	
Stress ratio inside connections: 60.00	
Stress ratio outside 100.00 🔪 [%]	
connections.	
	OK Cancel

Figure 3.1: Dialog box Details, tab Resistance

Consideration of Connections

The connection of members often weakens the cross-section in the zone of the joint. This effect can be accounted for by a *Reduction of limit tension stresses*. The numbers of the relevant nodes can be entered manually or selected graphically via the 🔊 button.

The Connection length defines the zone on the member where reduced stresses are to be applied. In the text box below, the maximum *Stress ratio inside connections* can be entered as percentage. If required, the *Stress ratio outside connections* can be modified as well.

3.1.2 Stability

Figure 3.2: Dialog box Details, tab Stability

Stability Analysis

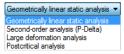
The *Perform stability analysis* check box controls whether a stability analysis is performed along with the cross-section design. If you clear the selection of the check box, Window 1.5 and 1.6 will not be shown.

The *Stability analysis* according to [1] 7.5 uses the internal forces determined by RFEM or RSTAB. For this method, make sure that the **Geometrically linear static analysis** has been set (the default for load combinations is second-order analysis). To perform the stability analysis according to [1] 7.5, the effective lengths of the members and sets of members subject to compression or compression and bending must be specified in Window 1.7 and 1.8.

If the bearing capacity of a structural system is significantly affected by its deformations, it is recommended to apply the *Stress/staility analysis according to second order theory*. This approach requires the definition of imperfections in RFEM/RSTAB and their consideration for the relevant load combinations. The flexural buckling analysis is carried out during the calculation of the load combinations in RFEM or RSTAB.



For the second order calculation, the lateral-torsional buckling design must be carried out nevertheless. Thus, the lengths for lateral-torsional buckling of members or sets of members must be specified in Window 1.7 or 1.8 manually. In this way, it is guaranteed that the lateral-torsional buckling analysis is performed with the appropriate factors (e.g. 1.0).



Method of analysis in RFEM/RSTAB

Buckling of Slender Compression Members

If the first check box is selected, a buckling analysis is performed for *members defined by slender*ness 80 < λ < 140 according to [1] 7.5.5. This option requires additional settings concerning the combination coefficients for loading which can be defined in Window 1.4 (see Chapter 2.4, page 16).

When the first check box is selected, it is possible to *Distribute loading on permanent and variable components*. Those settings are managed by Window 1.4 as well. If the first check box is not activated, this option is not available. Members with $\lambda > 80$ will then be verified as unsatisfactory.

3.1.3 Serviceability

Details	×
Resistance Stability Serviceability Other	
Servicability (Deflections)	
Deformation relative to:	
Shifted members ends / set of member ends	
◯ <u>U</u> ndeformed system	
	OK Cancel

Figure 3.3: Dialog box Details, tab Serviceability

The two options control whether the deformation is to be relative to the *Shifted member ends* (or ends of sets of members), i.e. the connection line between start and end nodes of the deformed system. Alternatively, the deformation can be referred to the *Undeformed system*. In most cases, the deformations can be checked relative to the displacements of the entire model.



You can find an example how to refer the deformations of members in the following article: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/001081.

Standard...

The limit deformations can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.7, page 9).

3.1.4 Other

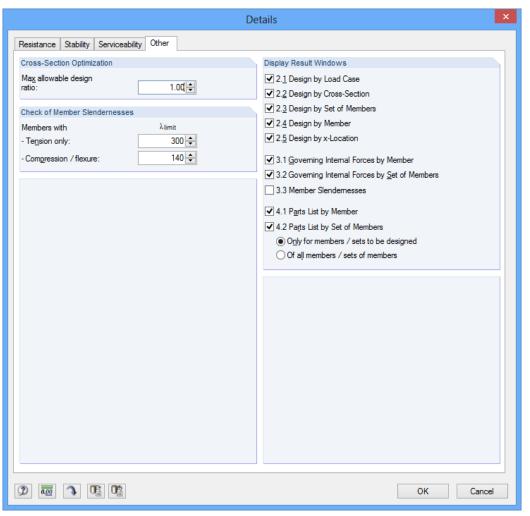


Figure 3.4: Dialog box Details, tab Other

Cross-Section Optimization

The optimization is targeted on the maximum design ratio of 100 %. If necessary, you can specify a different limit value in this text box.

Check of Member Slendernesses

The two text boxes in this section define the limit values λ_{limit} which control the member slendernesses. It is possible to enter specifications separately for members with pure tension forces and for members with bending and compression.

The limit values are compared to the real member slendernesses in Window 3.3. This window is available after the calculation (see Chapter 4.8, page 36) if the corresponding check box is selected in the *Display Result Windows* section (see below).

Display Result Windows

In this dialog section, you can select the results windows including parts lists that you want to display in the output. Those windows are described in Chapter 4.

Window 3.3 Member Slendernesses is deactivated by default.

3.2 Start Calculation



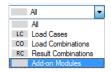
To start the [Calculation], click the corresponding button which is available in all input windows of RF-/TIMBER NBR.

The add-on module searches for the results of the load cases, load combinations, and result combinations to be designed. If those are not available yet, RF-/TIMBER NBR starts the calculation in RFEM or RSTAB to determine the relevant internal forces.

The calculation an also be started in the user interface of RFEM or RSTAB: The *To Calculate* dialog box (menu **Calculate** \rightarrow **To Calculate**) includes the design cases of all add-on modules, too.

		То	Calcul	ate		×
Load Cases /	Combinations / Module Cases Result Tables					
Not Calculate	ed			Selected for C	Calculation	
No.	Description	^		No.	Description	_
G LC1				CA1	RF-TIMBER NBR - Design acc. to NBR 7190:1997	
A LC2						
CO1	1.4*LC1 + 1.4*LC2					
RC1	LC1/p + 0.2*LC2/p					
			>			
			>>			
			4			
			~~			
		~				-
AI	×	Q		1	1	
		-				
D					OK	Cancel

Figure 3.5: Dialog box To Calculate of RFEM



۲

If the RF-/TIMBER NBR cases are missing in the *Not Calculated* section, select *All* or *Add-on Modules* in the drop-down list below.

To transfer the selected RF-/TIMBER NBR cases to the list on the right, use the button \ge . Click [OK] to start the calculation.

To calculate a design case directly, use the list in the toolbar. Select the RF-/TIMBER NBR case in the toolbar list, and then click [Show Results].



Figure 3.6: Direct calculation of a RF-TIMBER NBR design case in RFEM

Subsequently, you can observe the design in a separate dialog box.

4 Results

Window 2.1 Design by Load Case is displayed immediately after the calculation.

A1 ~	2.1 Des	ign by Load Case														
nput Data		Δ	В	C I	D	E				F					G	
General Data	Load-	A	Member	Location		E				F					Load	
Materials	ing	Description	No.	x [m]	Desian	1			Desi	an According to F	omula				Duration	
···· Cross-Sections	-	Ultimate Limit State Design														
Load Duration	RC10	1.4*LC1/p + 1.2*LC2/p + LC	4	1 980	0.86	≤1	303) Stab	ility - Compre	ssion parallel	to grain with buck	ding ab	out v-axi	s acc. to 7.5.5	5	Permaner	
Service Conditions - Members										-						
Service Conditions - Sets of Me		Serviceability Limit State Des	ign													
 Effective Lengths - Members 	CO1	LC1 + LC2 + LC3	6	1.980	0.91	≤1	401) Serv	iceability - De	esign situation	Permanent and	live act	tions acc	to 9.2.1 - Inn	er span,	Permaner	
Effective Lengths - Sets of Mer																
Serviceability Data																
sults Design by Load Case																
 Design by Load Case Design by Cross-Section 																
Design by Cross-Section Design by Set of Members																
Design by Member				Max:	0.91	< 1			9	2	>	1.0	V 7 6	2	\$	
Design by X-Location						1 - 1	•			¥ 6						
Governing Internal Forces by №	Details -	Details - Member 4 - x: 1.980 m - RC10														
Governing Internal Forces by S	Material Data - Hardwood Timber C 40															
Member Slendernesses	E Cross-section Data - T-Rectangle 260/160															
Parts List by Member	E Design Internal Forces															
· Parts List by Set of Members	🖃 Desig															
		npressive Force					No.d	-2.00					. 26	0.0		
	- Wic					b		260.0			- 11		1			
	Dep					d	4	160.0			- 11	t-				
		a of Cross-Section						416.00	cm ² kN/cm ²		_					
		npressive Stress active Length					No,d	4.000			_	160.0			∭…,	
		dius of Gyration					-0.y y	4.000			_	-)))	
		ndemess Degree					y ly	86 603		751	_	↓				
		aracteristic Compressive Streng	th				-y c0.k		kN/cm ²	Table 8						
		dulus of Elasticity for Compress					Ec0.m		kN/cm ²	Table 8						
		dification Eactor					Smod.1	0.600	iere oni	644	-			•		
	Mo	dification Factor					Cmod 2	1.000		6.4.4				~		
	Mo	dification Factor					Cmod.3	1.000		6.4.4						
	Mo	dification Factor					Cmod	0.600		6.4.4						
	Res	sistance Factor				7	/wc	1.400		6.4.5						
	Dee	sign Compressive Strength Par	allel to Gra	in		F	c0,d		kN/cm ²	7.2.6					[m	
	000	ective modulus of elasticity for	compressio	on parallel to g	grain	E	c0,ef		kN/cm ²	6.4.9						
	Effe					1	v	8874.67	cm4		~	0		X	T 0	
,	Effe	ment of Inertia					y I	0074.07	0.00		· ·	-			*	

Figure 4.1: Result window with design results and details

The design results are shown in the result Windows 2.1 through 2.5, sorted by different criteria.

Windows 3.1 and 3.2 list the governing internal forces. Window 3.3 informs you about the member slendernesses. The last two result Windows 4.1 and 4.2 show the parts lists sorted by member and set of members.



Every window can be selected by clicking the corresponding entry in the navigator. To set the previous or next input window, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] window.



To save the results, click [OK]. You exit RF-/TIMBER NBR and return to the main program.

Chapter 4 describes the different result windows one by one. The evaluation and checking of the results is described in Chapter 5 starting on page 40.





The upper table provides a summary of the results, sorted by load cases, load and result combinations of the governing designs. Furthermore, the list is split into *Ultimate Limit State Design* and *Serviceability Limit State Design* results.

Δ

The *Details* section below contains detailed information on the cross-section properties, design internal forces, and design details of the load case or combination selected in the upper table.

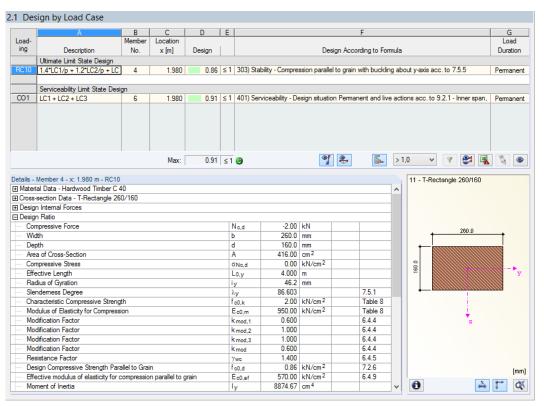


Figure 4.2: Window 2.1 Design by Load Case

Description

This column shows the descriptions of each designed load case, load or result combination.

Member No.

In this column, the number of each member is given that has the maximum design ratio of the respective loading.

Location **x**

The column shows the x-location of the member at which the maximum ratio occurs. For the tabular output, the program uses the following member locations *x*:

- Start and end nodes
- Division points according to optionally defined member division (see RFEM Table 1.16 or RSTAB Table 1.6)
- Member division according to specification for member results (see RFEM/RSTAB dialog box *Calculation parameters*, tab *Global Calculation Parameters*)
- Extreme values of internal forces

Design

Columns D and E show the design conditions according to NBR 7190:1997 [1].

The lengths of the colored scales graphically represent the respective design ratios.

Design According to Formula

This column lists the references in [1] according to which the different types of design have been performed.

Load Duration

Column H shows the load duration classes as defined in Window 1.4 (see Chapter 2.4, page 16).

4.2 Design by Cross-Section

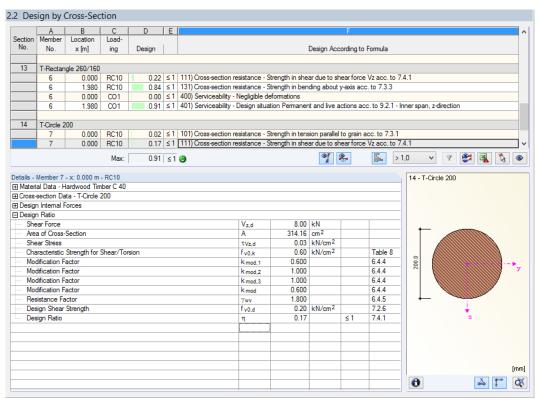


Figure 4.3: Window 2.2 Design by Cross-Section

This window lists the maximum ratios of all members and loadings selected for design, sorted by cross-section. For each section, the results are given for cross-section design, stability analysis, and serviceability limit state design.

4.3 Design by Set of Members

2.3 Design by Set of Members

	A	B	С	D	E					F						
Set	Member	Location	Load-													
No.	No.	x [m]	ing	Design					Design Ad	ccording	to Formula					
2	(Member															
	3	0.000	RC10	0.22		111) Cross-sect							.1			
	3	1.980	RC10			131) Cross-sect										
	3	0.040	RC10			321) Stability -				ut y-axis a	icc. to 7.5.6					
	3	0.000	CO1			400) Serviceab										
	3	1.980	CO1	0.36	≤1	401) Serviceab	ility - Design si	tuation Perma	anent and live	e actions i	acc. to 9.2.	1 - Inr	ier span, z	-direction		
3	(Member															
	4	0.000	RC10			102) Cross-sect										
	4	0.000	RC10	0.22	≤1	111) Cross-sect	tion resistance	- Strength in	shear due to	shear for	ce Vz acc. t	o 7.4	.1			
			Max:	0.91	≤1	۲			9	۰		> 1,	0 ~	9	1	\$
tails -	Member 3	- x: 0.040 m	- RC10										10 - T-Re	ctangle 160	260	
Mater	ial Data - S	oftwood Tim	per C 25									~		g		
Cross	-section Da	ta - T-Rectar	ngle 160/2	260												
Desig	n Internal F	orces	-													
Desig	n Ratio													. 160.0		
Ber	nding Mom	ent					M _{V,d}	0.31	kNm					100.0		
Ela	stic Section	n Modulus					Sy,c	-1802.67	cm ³				+	_		
Ber	nding Stres	s (Compressiv	ve Bending	g Strength)			ØMy.c.d	-0.02	kN/cm ²							
Effe	ective Leng	th for LTB					L1	4.000	m							
— Wio	dth						b	160.0	mm							
Dep	pth						d	260.0	mm				260.0			
Fac	tor						βE	4.000					2			У
Fac	tor						γf	1.400								
Cor	rection Coe	fficient					βм	7.264								
Cha	aracteristic	Compressive	Strength				fc0,k	2.50	kN/cm ²		Table 8					
		sticity for Co					E _{c0,m}	850.00	kN/cm ²		Table 8		,			
Mo	dification F	actor					k mod.1	0.600			6.4.4			z		
Mo	dification F	actor					k mod.2	1.000			6.4.4					
Мо	dification F	actor					k mod,3	0.800			6.4.4					
Мо	dification F	actor					kmod	0.480			6.4.4					
Re	sistance Fa	ctor					7wc	1.400			6.4.5					ler
De	sian Compr	essive Streng	th Paralle	to Grain			f c0.d	0.86	kN/cm ²		7.2.6					[mr

Figure 4.4: Window 2.3 Design by Set of Members

This result window is displayed when you have selected at least one set of members for the design. It lists the maximum design ratios sorted by set of members.

The *Member No.* column shows the number of the member within the set which has the maximum ratio with respect to the specific design criterion.

The output by set of members clearly presents the design for structural groups, e.g. chords.

2.4 Design by Member

	A	B	C	D				E					^
ember	20000000	Load											
No.	x [m]	Case	Design					Design Acc.	to Formula				
5	Cross-sectio	n No. 12											
	0.000	RC10	0.01	≤1	102) Cross-section resista	ance - Strengtł	h in compressi	on parallel to g	rain acc. to 7.3.2				
	0.000	RC10			111) Cross-section resista								
	1.980	RC10			151) Cross-section resista					to 7.	3.6		
	1.980	RC10			302) Stability - Compressi								
	0.000	RC10			313) Stability - Compressi								
	0.040	RC10			321) Stability - Lateral-tor			bout y-axis acc	c. to 7.5.6				
	0.000	CO1			400) Serviceability - Neg								
	1.980	CO1	0.16	≤1	401) Serviceability - Desi	gn situation Pe	ermanent and	live actions ac	c. to 9.2.1 - Inner s	pan,	z-direction		
													~
		Max:	0.91	< 1	A			9	۹.	>1	0 ¥	7 😂 🕱	%
		Max.	0.01	21	9			4				· – A	9 2
etails -	Member 5 - x	1 980 m	- RC10								12 T.R.	ctangle 160/260	
1 Materi	ial Data - Har	lwood Tin	ber C 40								12-140	orangio roorzoo	
	section Data			D									
	n Internal For			-									
- Desig													
Cor	npressive For	ce				N c.d	-2.00	kN				t 160.0	
— Wio	th					b	160.0	mm			+		
Dep	oth					d	260.0	mm					
- Are	a of Cross-Se	ction				A	416.00	cm ²					
Cor	npressive Stre	SS				ØNc,d	0.00	kN/cm ²					
Effe	ective Length					Lo.y	4.000	m			260.0		
Rad	dius of Gyratic	n				iy	75.1	mm			2		А
Slei	ndemess Deg	ree				λγ	53.294		7.5.1				
Cha	aracteristic Co	mpressive	Strength			fc0,k	2.00	kN/cm ²	Table 8				
- Mo	dulus of Elasti	city for Co	mpression			E _{c0,m}	950.00	kN/cm ²	Table 8				
- Mo	dification Fact	or				k mod,1	0.600		6.4.4				
Mo	dification Fact	or				k mod,2	1.000		6.4.4			z	
- Mo	dification Fact	or				k mod,3	1.000		6.4.4				
- Mo	dification Fact	or				kmod	0.600		6.4.4				
Res	sistance Facto	or				7wc	1.400		6.4.5				
Des	sign Compress	ive Streng	gth Parallel to	Grain	ı	fc0,d	0.86	kN/cm ²	7.2.6				(mm)
		م مراجع م	the for comp	eeeio	n parallel to grain	Ec0.ef	570.00	kN/cm ²	6.4.9				fuuri
- Effe	scrive modulu	s or elastic	ary for compr	Casio	i paraller to grain						0		T a

Figure 4.5: Window 2.4 Design by Member

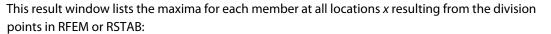
This result window presents the maximum ratios of the individual designs for each member. The columns are described in Chapter 4.1 on page 31.

4.5 Design by x-Location

5 Des	ign by x-l	Locatio	n													
	A	В	С	D					E							^
Member	Location	Load														
No.	x [m]	Case	Design					De	esign Acc. t	o Formul	а					
3	Cross-sectio	n No. 10	- T-Rectangl	e 160	0/260											
	0.000	RC10				tion resistance		hear due to s	hear force	Vz acc. t	o 7.4.1					
	0.000	CO1				oility - Negligible										
	0.000	RC10				tion resistance		hear due to s	hear force	Vz acc. t	o 7.4.1					
	0.000	CO1				bility - Negligible										
	0.040					tion resistance ·										
	0.040					tion resistance										_
	0.040	RC10				Lateral-torsional										
	0.040	CO1				bility - Design siti						z-direction				
	0.079	RC10	0.21	≤1	111) Cross-sed	tion resistance ·	 Strength in s 	hear due to s	hear force	Vz acc. t	o 7.4.1					
		Max:	0.91	≤1	3				9	e -	>1	.0 \	7	2	X	۲
E Cross-	al Data - Soft section Data n Internal For	- T-Recta		D												
	ar Force						Vz.d	8.00	LIN				+	160.0	-+	
- Wid							vz,a b	160.0								
- Dec							d	260.0				†				
	ar Stress						U TVz.d		kN/cm ²							
	racteristic St	enath for	Shear/Toreid	-			fv0.k		kN/cm ²		Table 8					
	dification Fac		Shour Toraid				Kmod 1	0.600	KIN/ GIT		6.4.4	260.0				
	dification Fac						kmod.2	1.000			6.4.4	26				Y
	dification Fac						Kmod 3	0.800			6.4.4					
Mor	dification Fac	tor					kmod	0.480			6.4.4					
	istance Fact						Ywy	1.800			6.4.5					
	ign Shear St						fv0.d		kN/cm ²		7.2.6	-				
	ign Ratio	-					η	0.22		≤1	7.4.1			*		
	-															
										1						
																(mm
							1			-		0		2		ă

Figure 4.6: Window 2.5 Design by x-Location

4



- Start and end nodes
- Division points according to optionally defined member division (see RFEM Table 1.16 or RSTAB Table 1.6)
- Member division according to specification for member results (see RFEM/RSTAB dialog box *Calculation Parameters*, tab *Global Calculation Parameters*)
- Extreme values of internal forces

4.6 Governing Internal Forces by Member

	A	B	С	D	E	F	G	Н	<u> </u>
No.	Location	Load		Forces [kN]		Moments [kNm]			
	x [m]	Case	N	Vy	Vz	MT	My	Mz	Design According to Formula
3	Cross-section No. 10 - T-Rectangle 160/260								
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	
	0.040	RC10	-2.00	0.00	7.84	0.00	0.31	0.00	
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to
4	Cross-section No. 11 - T-Rectangle 260/160								
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	
	1.980	C01	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to
6	Cross-section No. 12 - T-Rectangle 160/260								
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	102) Cross-section resistance - Strength in compression parallel to grain a
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz a
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	302) Stability - Compression parallel to grain with buckling about y-axis ac
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	313) Stability - Compression parallel to grain with buckling about z-axis ac
	0.040	RC10	-2.00	0.00	7.84	0.00	0.31	0.00	321) Stability - Lateral-torsional buckling in bending about y-axis acc. to 7
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to
	Cross-section No. 13 - T-Rectangle 260/160								
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz ad
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	-
	0.000	C01	0.00	0.00	0.00	0.00	0.00	0.00	
	1.980	C01	0.00	0.00	0.00	0.00	0.00	0.00	
7	Cross-sectio	- No. 14	T Cirolo 200						
/		RC10	1-Circle 200 12.00	0.00	8.00	0.00	0.00	0.00	101) Cross-section resistance - Strength in tension parallel to grain acc. to
	0.000	nulu	12.00	0.00	0.00	0.00	0.00	0.00	To the organisation realation of a birth girl in tension parallel to grain acc. to

Figure 4.7: Window 3.1 Governing Internal Forces by Member

For all designed members, this window displays the governing internal forces, i.e. those internal forces that produce the maximum ratio of each design.

Location x

At this x-location of the member, the respective maximum design ratio occurs.

Load Case

This column displays the numbers of the load case, load or result combination whose internal forces result in the maximum design ratios.

Forces / Moments

For each member, these columns present the axial and shear forces as well as the torsional and bending moments which give the maximum ratios in the respective cross-section designs, stability analyses, and serviceability limit state designs.

Design According to Formula

The final column informs you about the design types and equations by which the designs have been performed according to the Standard [1].

4.7 Governing Internal Forces by Set of Members

	A	В	С	D	E	F	G	H	I
Set	Location	Load		Forces [kN]			oments [kNm]		
No.	x [m]	Case	N	Vy	Vz	Мт	My	Mz	Design According to Formula
2	(Member No	o. 3)							
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00		131) Cross-section resistance - Strength in bending about y-axis acc. to 7.
	0.040	RC10	-2.00	0.00	7.84	0.00	0.31		321) Stability - Lateral-torsional buckling in bending about y-axis acc. to 7.8
	0.000	CO1	0.00	0.00	0.00	0.00	0.00		400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to 9
3	(Member No								
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	
	0.000	RC10	0.00	0.00	8.00	0.00	0.00		111) Cross-section resistance - Strength in shear due to shear force Vz ac
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00		151) Cross-section resistance - Strength in bending about y-axis and comp
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00		303) Stability - Compression parallel to grain with buckling about y-axis acc
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00		312) Stability - Compression parallel to grain with buckling about z-axis acc
	0.000	CO1	0.00	0.00	0.00	0.00	0.00		400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to 9

3.2 Governing Internal Forces by Set of Members

Figure 4.8: Window 3.2 Governing Internal Forces by Set of Members

This window contains the individual internal forces that result in the maximum ratios of the respective design for each set of members.

4.8 Member Slendernesses

Details...

Window 3.3 is shown when you have selected the respective check box in the *Other* tab of the *Details* dialog box (see Figure 3.4, page 28).

2.2.14

	А	B	С	D	E	F	G	н
lember		Length		Major Axis y			Minor Axis z	
No.	Under Stress	L [m]	ky[-]	iy [mm]	λy[-]	k _z [-]	iz [mm]	λz [-]
3	Compression / Flexure	4.000	1.000	75.1	53.294	1.000	46.2	86.60
4	Compression / Flexure	4.000	1.000	46.2	86.603	1.000	75.1	53.29
5	Compression / Flexure	4.000	1.000	75.1	53.294	1.000	46.2	86.60
6	Compression / Flexure	4.000	1.000	46.2	86.603	1.000	75.1	53.29
7	Compression / Flexure	4.000	1.000	50.0	80.000	1.000	50.0	80.00
		Members with compression Max λy: 86.603 ≤	140 🙂					
		Max λ_z : 86.603 \leq	140 🙂				F	Š

Figure 4.9: Window 3.3 Member Slendernesses

The table lists the effective slenderness ratios of the designed members for both directions of the principal axes. They depend on the load type.

Below the list, you find a comparison of the most unfavorable values with the limit values. The latter are managed in the *Other* tab of the *Details* dialog box (see Figure 3.4, page 28).

Members of the member type "Tension" or "Cable" are not included in this table.



Details...

This window is displayed only informative. It does not provide any stability analysis of slendernesses. 4.1 Danta List by Manala

4.9 Parts List by Member

Finally, RF-/TIMBER NBR provides a summary of all cross-sections contained in the design case.

	А	B	C	D	E	F	G	H I	
art	Cross-Section	Number of	Length	Total Length	Surface Area	Volume	Unit Weight	Weight	Total Weight
No.	Description	Members	[m]	[m]	[m ²]	[m ³]	[kg/m]	[kg]	[t]
1	10 - T-Rectangle 160/260	1	4.00	4.00	3.36	0.17	22.88	91.52	0.09
2	11 - T-Rectangle 260/160	1	4.00	4.00	3.36	0.17	27.04	108.16	0.10
3	12 - T-Rectangle 160/260	1	4.00	4.00	3.36	0.17	27.04	108.16	0.10
4	13 - T-Rectangle 260/160	1	4.00	4.00	3.36	0.17	22.88	91.52	0.09
5	14 - T-Circle 200	1	4.00	4.00	2.51	0.13	29.85	119.38	0.11
Gum		5		20.00	15.95	0.79			0.51

Figure 4.10: Window 4.1 Parts List by Member

Details...

By default, this list contains only designed members. If you need a parts list of all members of the model, select the corresponding option in the *Details* dialog box, tab *Other* (see Figure 3.4, page 28).

Part No.

The program automatically assigns item numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

Column B shows how many similar members are used for each part.

Length

This column displays the respective length of an individual member.

Total Length

In this column, the product determined from the two previous columns is given.

Surface Area



For each part, the program specifies the surface area relative to the total length. This area is determined from the *Surface Area* of the cross-sections. It can be checked in Window 1.3 and Windows 2.1 to 2.5 (see Figure 2.15, page 15).



The volume of a part is determined from the cross-sectional area and the total length.

Unit Weight

The unit mass of the cross-section is related to the length of one meter.

Weight

The values of this column represent the products of the entries in columns C and G.

Total Weight

The final column gives the total mass of each part.

Sum

At the end of the list, you find a sum of the values in columns B, D, E, F, and I. The last row of the column *Total Weight* gives information about the total amount of timber required.

4.10 Parts List by Set of Members

Figure 4.11: Window 4.2 Parts List by Set of Members

The last result window is displayed when you have selected at least one set of members for design. It represent the parts list of structural groups, e.g. chords.

Details on the various columns can be found in Chapter 4.9. If there are different cross-sections in the set of members, the program averages the surface area, the volume, and the cross-section weight.

5 Results Evaluation

You can evaluate the design results in different ways. The buttons below the upper table are very useful for that.

~ ~	- ·		
22	1)ecian	by	Cross-Section

	A	B	С	D	E					F				
	Member	Location	Load-											
No.	No.	x [m]	ing	Design					Design Acc	ording to	Formula			
10	T-Rectan	gle 160/260												
	3	0.000	RC10	0.22	≤1	111) Cross-section r	esistance - Si	trength in she	ar due to sh	near force	Vz acc. to 7.4	4.1		
	3	1.980	RC10			131) Cross-section r								
	3	0.040	RC10			321) Stability - Later			ding about ;	y-axis aco	c. to 7.5.6			
	3	0.000	CO1			400) Serviceability -								
	3	1.980	CO1	0.36	≤1	401) Serviceability -	Design situat	ion Permaner	nt and live a	ections ac	c. to 9.2.1 - In	ner span,	z-direction	
11	T-Rectan	gle 260/160												
	4	0.000	RC10			102) Cross-section r								
	4	0.000	RC10	0.22	≤1	111) Cross-section r	esistance - Si	trength in she	ar due to sh	near force	Vz acc. to 7.4	4.1		
			Max:	0.91	21	0			9	2.	国 >1	.0 🗸	v 🤻 😂 🛙	🔹 🖏 🤇
			Max.	0.51	21	9			4	Ψ.		,0 ,		V A A
otaila	Mombor 2	- x: 1.980 m	DC10									40 T D	Rectangle 160/260	
		oftwood Tim										10 - 1-R	Cectangle 100/200	
		ta - T-Recta		000										
	n Internal F		igie 100/	200										
	n Ratio	UICES												
	nding Mom	ent					M _{y,d}	8.00	kNm				+ 160.0	+
	stic Section						S _{v.t}	1802.67						
	stic Section						Sy.c	-1802.67						
		s (Tensile Be	ndina Stre	nath)			σ _{My,t,d}		kN/cm ²					
		s (Compressi					σ _{My,c,d}		kN/cm ²					
		Tensile Strer		gouonguny			ft0,k		kN/cm ²		6.3.3	260.0		
		Compressive					fc0.k		kN/cm ²		Table 8	26		У
	dification F						Kmod.1	0.600			644			
	dification F	actor					Kmod 2	1.000			6.4.4			
Mo		actor					k mod.3	0.800			6.4.4			
	dification H						kmod	0.480			6.4.4	-		
- Mo	dification F dification F	actor								1			*	
Mo Mo							7wt	1.800			6.4.5		Z	
Mo Mo Res	dification F	ictor					γwt	1.800			6.4.5		z	
Mo Mo Res Res	dification F sistance Fa sistance Fa	ictor ictor	irallel to G	rain			γwt γwc	1.400	kN/cm ²				Z	
Mo Mo Res Res Des	dification F sistance Fa sistance Fa sign Tensile	ictor ictor e Strength Pa					γwt γwc ft0,d	1.400 0.87	kN/cm ²		6.4.5		Z	
Moi Moi Res Res Des	dification F sistance Fa sistance Fa sign Tensile sign Compr	ictor ictor	th Paralle				γwt γwc	1.400 0.87		<u>≤1</u>	6.4.5 7.2.6		z	ſm

Figure 5.1: Buttons for results evaluation

Button	Description	Function
Y	Ultimate Limit State Design	Shows or hides the results of the ULS design
*	Serviceability Limit State Designs	Shows or hides the results of the SLS design
	Color Bars	Shows or hides the colored relation scales in the tables
> 1,0 • > 1,0 Max Define	Filter Parameters	Describes the filter criterion for the output in the tables: Design ratios greater than 1, maximum value or user-defined limit
V	Apply Filter	Displays only rows where the filter parameters are valid (ratio > 1, maximum, user-defined value)
2	Result Diagrams	Opens the Result Diagram on Member Window \rightarrow Chapter 5.2, page 43
	Excel Export	Exports the table to MS Excel \rightarrow Chapter 7.4.3, page 53
₹₹	Member Selection	Allows you to graphically select a member in the work window to display its results in the table
۲	View Mode	Jumps to the work window of RFEM or RSTAB to check or change the view

Table 5.1: Buttons in result Windows 2.1 through 2.5



You can also evaluate the design results in the work window of RFEM or RSTAB.

Background graphic and view mode

The work window of RFEM or RSTAB in the background is useful for you to find the position of a particular member in the model: There, the member selected in the RF-/TIMBER NBR result window is highlighted. Furthermore, an arrow indicates the relevant location x of this member.

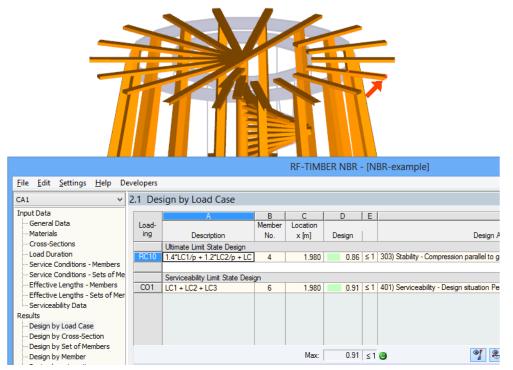


Figure 5.2: Indication of member and relevant *Location x* in RFEM model



If you cannot improve the display by moving the RF-/TIMBER NBR module window, click the solution to activate the *view mode*. Thus, you hide the module window so that you can change the view in the user interface of RFEM or RSTAB. In the view mode, you can use the functions of the *View* menu, e.g. zoom, move, or rotate the view. The pointer remains visible.

Click [Back] to return to the RF-/TIMBER NBR module.

RFEM/RSTAB work window

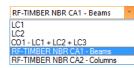
Graphics

You can also check the design ratios graphically in the RFEM/RSTAB model: Click [Graphics] to quit the design module. In the work window of RFEM or RSTAB, the design ratios are now displayed like the internal forces of a load case.



To turn the display of the design ratios on or off, use the [Show Results] button which is familiar from the display of internal forces. To switch the result values on or off, click the [Show Values] button next to it.

The tables of RFEM or RSTAB are of no relevance for the timber design results.



The design cases can be selected in the list of the menu bar.

The graphical representation of the design results can be controlled in the *Display* navigator, item **Results** \rightarrow **Members**. The ratios are shown *Two-Colored* by default.

5

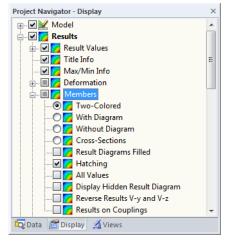


Figure 5.3: *Display* navigator – Results \rightarrow Members

When you have selected a multicolor display (options *With/Without Diagram* or *Cross-Sections*), the color panel is available. It provides the usual control functions. They are described in detail in the RFEM/RSTAB manual, Chapter 3.4.6.

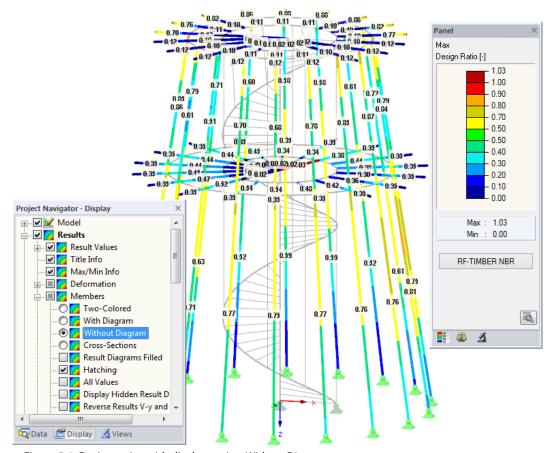


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 46).

RF-TIMBER NBR

To return to the add-on module, use the [RF-/TIMBER NBR] button in the panel.

5.2 Result Diagram

You can graphically evaluate the design ratios of members or sets of members in a result diagram, without using the work window of RFEM or RSTAB.

Select the member (or set of members) in the RF-/TIMBER NBR result window by clicking in the relevant table row. Then click the geal button to open the *Result Diagram on Member* dialog box. This button is located below the upper table (see Figure 5.1, page 40).

In the work window of RFEM or RSTAB, the result diagram can be accessed from the menu

Results ightarrow Result Diagrams for Selected Members



or via the toolbar button shown on the left.

A new window opens. It presents the distribution of the maximum design values on the member or set of members.

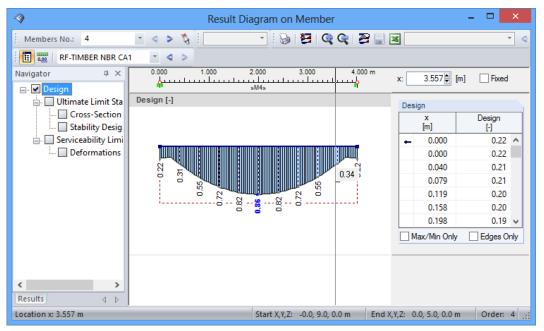


Figure 5.5: Dialog box Result Diagram on Member

You can switch the ULS and SLS results on or off in the *Results* navigator.

Use the list in the toolbar to select the relevant RF-/TIMBER NBR design case.

RF-TIMBER NBR CA1 - Beams LC1 LC2 CO1 - LC1 + LC2 + LC3 RF-TIMBER NBR CA2 - Columns RF-TIMBER NBR CA2 - Columns

The Result Diagram on Member dialog box is described in the RFEM or RSTAB manual, Chapter 9.5.

5.3 Filter for Results

The RF-/TIMBER NBR result windows allow you to sort the results by various criteria. In addition, you can use the filter options for the tables (see Figure 5.1, page 40) to reduce the numerical output according to specific ratios. This function is described in the *Knowledge Base* at our Web site:

https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000733

Furthermore, you can apply the filter options described in Chapter 9.9 of the RFEM manual or Chapter 9.7 of the RSTAB manual to evaluate the results graphically.



You can also use the *Visibility* options for RF-/TIMBER NBR to filter the members and evaluate them (see RFEM manual, Chapter 9.9.1 or RSTAB manual, Chapter 9.7.1).

Filtering design ratios

Graphics

The design ratios can be used as filter criteria in the RFEM/RSTAB work window which you access by clicking [Graphics]. To apply this filter function, the panel must be displayed. If it is not, select.

View ightarrow Control Panel (Color scale, Factors, Filter)



or use the toolbar button shown on the left.

The panel is described in the RFEM/RSTAB manual, Chapter 3.4.6. The filter settings for the results can be defined in the first tab (Color scale). As this tab is not available for the two-colored results, you haveset the display option *Colored With/Without Diagram* or *Cross-Sections* in the *Display* navigator (see Figure 5.3, page 42).

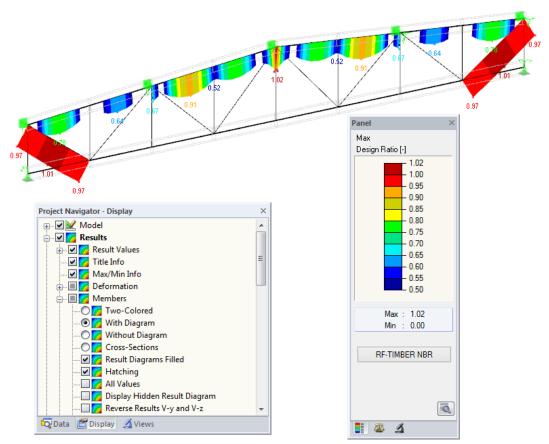


Figure 5.6: Filtering design ratios with adjusted color spectrum

As seen in Figure 5.6, the color spectrum can be set in such a way that only ratios greater than 0.50 are shown in the color ranges between blue and red.

1

Filtering members

In the *Filter* tab of the control panel, you can specify the numbers of particular members to display their results exclusively, i.e. filtered. This function is described in the RFEM manual, Chapter 9.9.3 or RSTAB manual, Chapter 9.7.3.

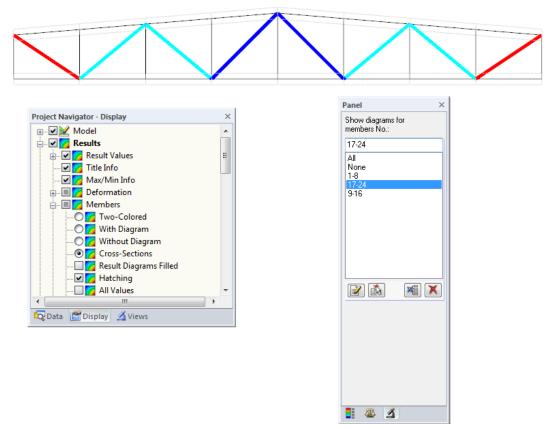


Figure 5.7: Member filter for ratios of diagonals

Unlike the *Visibility* function, the entire model is displayed. Figure 5.7 shows the ratios of the truss diagonals. All other members are displayed in the model, but they have no design ratios.

6 Printout

6.1 Printout Report

Similarly to RFEM or RSTAB, the program generates a printout report for the RF-/TIMBER NBR results which can be supplemented by graphics and descriptions. The selection in the printout report determines what data from the design module are to be included in the final printout.



The printout report is described in the RFEM or RSTAB manual. In particular, Chapter 10.1.3.5 *Selecting Data of Add-on Modules* describes how the input and output data of add-on modules can be selected.

For complex models with many design cases, it is recommended to split the data into several printout reports, thus allowing for a clearly-arranged documentation.

6.2 Graphic Printout



In RFEM or RSTAB, you can add any picture of the work window to the printout report or send it directly to a printer. In this way, the design ratios shown on the RFEM/RSTAB model can be used for the documentation.

The printing of graphics is described in the RFEM or RSTAB manual, Chapter 10.2.

Design ratios in RFEM/RSTAB model

To print the currently displayed design ratios, click

File ightarrow Print Graphic

B

or use the toolbar button shown on the left.

<u>6</u> 7						5.04.195		
:4≥	<u>F</u> ile	<u>E</u> dit	<u>V</u> iew	<u>I</u> nsert	<u>C</u> alculate	<u>R</u> esults	Tools	Ta <u>b</u>
					t Graphic			

Figure 6.1: [Print Graphic] button in RFEM toolbar

Result diagrams



You can also transfer the *Result Diagram on Member* images to the printout report or print them directly by using the [Print] button.

9	Result Diagram on Member
Members No.: B	🔽 < 🔉 🍾 i 🔜 🔽 i 🔂 🛂 i
RC10: 1.4*LC1/p +	1.2*LC 🔨 🔷 🕨 Print
Navigator 🛛 🕂 🗙	0.000 4.000 m
🖃 🔳 Global Deformatio 🔺	эМЗэ

Figure 6.2: [Print] button in *Result Diagram on Member* dialog box

The Graphic Printout dialog box appears (see Figure 6.3).

	Graphic Printou	it.	
eneral Options Factors Border and	Stretch Factors		
Graphic Picture	Window To Print	Graphic Size	
Directly to a printer	Current only	As screen v	iew
To a printout report: PR	✓ <u>M</u> ore	🔍 🔿 Window fillin	ng
◯ To the Clipboard	O Mass print	🔍 🔿 To scale	<u>1</u> : 100 ¥
○ To <u>3</u> D PDF			
Dearbia Distura Cine and Detetion	Options		
Graphic Picture Size and Rotation			
✓ Use whole page width	diagram	or selected <u>x</u> -location in result	
Use whole page <u>h</u> eight	Lock graphic p	victure (without update)	
Height: 100 🖨 [½ of page]			
	Show printout r	report on [OK]	
Rotation: 0+ [°]			
leader of Graphic Picture			
NBR - Members Design Ratio, CA1			

Figure 6.3: Dialog box Graphic Printout, tab General

This dialog box is described in the RFEM or RSTAB manual, Chapter 10.2. That chapter also describes the other tabs of the dialog box.

You can move a graphic anywhere within the printout report by using the drag-and-drop function.

If you want to modify an image in the printout report, right-click the relevant entry in the navigator of the printout report. The *Properties* option in the shortcut menu opens the *Graphic Printout* dialog box again. It offers you several options to adjust the image.

	Graphic Pri	ntout		×
General Options Factors Border a	and Stretch Factors			
Script	Symbols		Frame	
O Proportional	Proportional		○ <u>N</u> one	
Constant	◯ C <u>o</u> nstant		• Fra <u>m</u> ed	
Eactor: 1 🖨	F <u>a</u> ctor: 1 €		Title box	
Print Quality		Color		
Standard (max 1000 x 1000 pixels)		O <u>G</u> rayscale	e	
<u>Maximum (max 5000 x 5000 pixels</u>))	O Texts and	d lines in <u>b</u> lack	
<u>U</u> ser-defined		All colore	d	
	1000 🜩			
D			OK Cancel	

Figure 6.4: Dialog box Graphic Printout, tab Options

 Remove from Printout Report

 Start with New Page

 Selection...

 Properties...

7 General Functions

This chapter describes useful menu functions and export options for the design results.

7.1 Design Cases

Design cases allow you to arrange members for specific analyses. In this way, you can combine groups of structural components or analyze members with particular design specifications, e.g. modified materials, partial safety factors, cross-sections.

It is no problem to analyze the same member or set of members in different design cases.

RF-TIMBER NBR CA1 - Beams LC1 LC2 CO1 - LC1 + LC2 + LC3 RF-TIMBER NBR CA1 - Beams RF-TIMBER NBR CA2 - Columns To calculate a RF-/TIMBER NBR design case, you can also use the load case list in the toolbar of RFEM or RSTAB.

Create design case

To create a new design case, use the RF-/TIMBER NBR menu and select

```
File 
ightarrow New Case.
```

The following dialog box appears.

	New RF-TIMBER NBR Case	×
<u>N</u> o. 2	Description Design According to NBR	~
D	OK Cance	əl

Enter a *No*. (one that is still available) for the new design case and an optional *Description*. It facilitates the selection in the load case list.

Then click [OK] to open the 1.1 General Data Window of RF-/TIMBER NBR where you can enter the new design data.

Rename design case

To change the description of a design case, use the RF-/TIMBER NBR menu and select

$\textbf{File} \rightarrow \textbf{Rename Case}.$

The following dialog box appears.

Rename RF-TIMBER NBR Case							
<u>N</u> o. 2	Description New Description	~					
D	OK Cance	1					

Figure 7.2: Dialog box Rename RF-TIMBER NBR-Case

You can specify a different Description as well as a different No. for the design case.

Figure 7.1: Dialog box New RF-TIMBER NBR-Case

Copy design case

To copy the input data of the current design case, use the RF-/TIMBER NBR menu and select

File ightarrow Copy Case.

The following dialog box appears.

Copy RF-TIMBER NBR Case								
Copy from Case								
CA2 - New Description V								
New Case								
No.: Description:								
3 Design According to NBR								
	_							
DK Cance	ł							

Figure 7.3: Dialog box Copy RF-TIMBER NBR-Case

Define the No. and, if necessary, a Description of the new case.

Delete design case

To delete a design case, use the RF-/TIMBER NBR menu and select

```
\textbf{File} \rightarrow \textbf{Delete Case}.
```

The following dialog box appears.

Delete Cases ×								
Available Cases								
Description								
New Description								
Design According to NBR								
OK Cancel								

Figure 7.4: Dialog box Delete Cases

Select the design case in the list of Available Cases. To delete this case, click [OK].



7.2 Cross-Section Optimization

The design module offers you the option to optimize overstressed or little utilized cross-sections. To do this, select *Yes* for the relevant cross-section(s) in Window *1.3 Cross-Sections*, column C resp. D (see Figure 2.13, page 13).

You can also start the optimization in the result windows via the shortcut menu.

	A	В	С	D	Е	F	
Section	Member	Location	Load-				
No.	No.	x [m]	ing	Design		Design According to Formula	
10	T-Rectan	gle 160/260					
	3	0.000	CO1	0.22		111) Cross-section resistance - Strength in shear due to shear force Vz acc. to 7.4.1	
	3 4000 004 000 41 10100					section resistance - Strength in bending about y-axis acc. to 7.3.3	
	Info About Cross-Section				Dou	bleclick ty - Lateral torsional buckling in bending about y-axis acc. to 7.5.6	
11	T-Re	<u>O</u> ptimize	Cross-Se	ction		section resistance - Strength in shear due to shear force Vz acc. to 7.4.1	
						eters section resistance - Strength in bending about y-axis acc. to 7.3.3	
12 T-Rectangle 160/260							

Figure 7.5: Shortcut menu for cross-section optimization

Details...

During the optimization, the module determines the cross-section that fulfills the analysis requirements in the "optimal" way, i.e. comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.4, page 28). The required cross-section properties are calculated with the internal forces of RFEM or RSTAB. If a different cross-section proves to be more favorable, it will be used for the design. In this case, the graphic in Window 1.3 shows two cross-sections – the original section from RFEM or RSTAB and the optimized one (see Figure 7.7).

For a parametric cross-section, the following dialog box appears when you have selected *Yes* from the drop-down list.

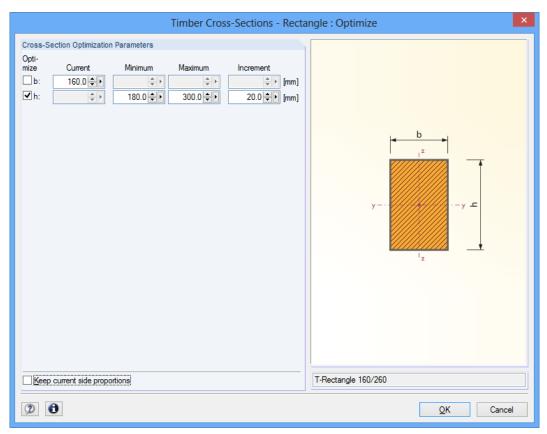


Figure 7.6: Dialog box Timber Cross-Sections - Rectangle : Optimize

By selecting the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. This enables the *Minimum* and *Maximum* columns where you can specify the upper and lower limits of the parameter. The *Increment* column determines the interval in which the size of the parameter varies during the optimization.

If you want to *Keep current side proportions*, select the corresponding check box. In addition, you have to select two parameters for the optimization.



Please note that the internal forces are not automatically recalculated with the modified cross-sections during the optimization: It is up to you to decide which sections should be transferred to RFEM or RSTAB for a new calculation. As a result of optimized cross-sections, the internal forces may vary considerably because of the changed stiffnesses of the model. Therefore, it is recommended to recalculate the internal forces of the modified cross-sections after the first optimization, and then to optimize the sections once again.

To export the modified cross-section(s) to RFEM or RSTAB, go to Window 1.3 Cross-Sections and select

$\textbf{Edit} \rightarrow \textbf{Export All Cross-Sections to RFEM/RSTAB}.$

The shortcut menu of Window 1.3 also provides some options to export optimized cross-sections to RFEM or RSTAB.

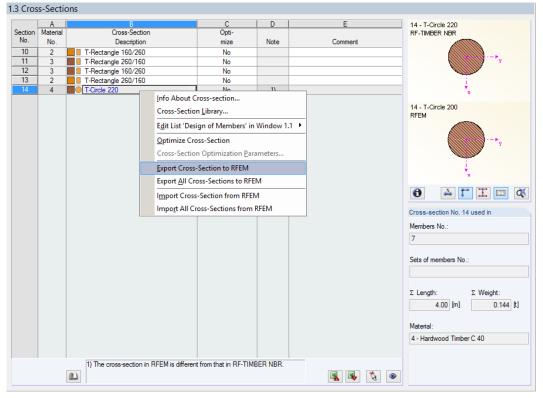


Figure 7.7: Shortcut menu in Window 1.3 Cross-Sections

Before the modified cross-sections are transferred to RFEM or RSTAB, a query appears as to whether the RFEM/RSTAB results should be deleted.

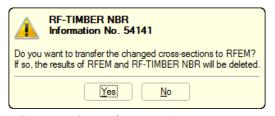


Figure 7.8: Query when exporting cross-sections

7 General Functions

Calculation

By confirming the query and then starting the [Calculation] in the RF-/TIMBER NBR module, the internal forces of RFEM or RSTAB as well as the design ratios will be determined in one single calculation run.

If the modified cross-sections have not been exported to RFEM or RSTAB yet, you can reimport the original sections in the design module by using the last two options shown in Figure 7.7. Please note that this shortcut menu is only available in Window *1.3 Cross-Sections*.

7.3 Units and Decimal Places

The units and decimal places of RFEM or RSTAB and of all add-on modules are managed in one dialog box. To define the units for RF-/TIMBER NBR, select the menu

Settings \rightarrow Units and Decimal Places.

The dialog box which is familiar from RFEM or RSTAB appears. RF-/TIMBER NBR is preset in the *Program / Module* list.

			Units and Decim	al Places	5		×
Program / Module	^	RF-TIMBER NBR			Parts List		
RF-TIMBER Pro RF-TIMBER AWC RF-TIMBER CSA RF-TIMBER SANS RF-TIMBER SANS RF-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RX-TIMBER RF-OTIMBER RF-OTIMBER RF-OTIMBER RF-OTIMBER RF-OTIMBER RF-OTIMBER RF-FOUNDATION RF-MOVE		Output Data Stresses: Design ratios: Dimensionless:	Unit De kN/cm^2 v	c. places 2 ÷ 2 ÷	Parts List Lengths: Tgtal lengths: Surface greas: Volumes: Weights per length: Weights: Total weights:	Unit m m^2 m^3 kg/m kg t	Dec. places
	*						
۵ 📔	œ					OK	Cancel

Figure 7.9: Dialog box Units and Decimal Places

۱

You can save the settings as a user-defined profile to reuse them in other models. Those functions are described in the RFEM or RSTAB manual, Chapter 11.1.3.

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

If you have modified the materials in RF-/TIMBER NBR for the design, you can export those materials to RFEM or RSTAB in a similar way as you export cross-sections: Open the *1.2 Materials* Window and then select

```
Edit \rightarrow Export All Materials to RFEM/RSTAB.
```

You can also export the modified materials to RFEM or RSTAB by using the shortcut menu of Window 1.2.

	Material Library					
	Export Material to RFEM					
	Export <u>A</u> ll Materials to RFEM					
Import Material from RFEM						
	Import All Materials from RFEM					

Figure 7.10: Shortcut menu of Window 1.2 Materials

Calculation

Before the modified materials are transferred to RFEM or RSTAB, a query appears as to whether the results of the main program should be deleted. When you have confirmed the query and then start the [Calculation] in RF-/TIMBER NBR, the new internal forces and the design ratios are determined in one single calculation run.

If the modified materials have not been exported to RFEM or RSTAB yet, you can transfer the original materials to the design module with the last two options shown in Figure 7.10. Please note that this shortcut menu is only available in Window *1.2 Materials*.

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

If you have adjusted the effective lengths in RF-/TIMBER NBR for the design, you can export the modified values to RFEM or RSTAB in a similar way as you export cross-sections: Got to the *1.7 Effective Lengths - Members* Window and then select

```
Edit \rightarrow Export All Effective Lengths to RFEM/RSTAB.
```

You can also use the corresponding option on the shortcut menu of Window 1.7.

Import Effective Length from RFEM	
Import <u>A</u> ll Effective Lengths from RFE	М
Export Effective Length to RFEM	
Export All Effective Lengths to RFEM	2

Figure 7.11: Shortcut menu of Window 1.7 Effective Lengths - Members

Before the modified effective lengths are transferred to RFEM or RSTAB, a query appears as to whether the results of the main program should be deleted.

7.4.3 Exporting Results

The RF-/TIMBER NBR results can also be used by other programs.

Clipboard

To copy selected cells of the result windows to the Clipboard, use the keys [Ctrl]+[C]. Press [Ctrl]+[V] to insert the cells, for example in a word processing program. The headers of the table columns will not be transferred.

Printout Report

You can print the data of RF-/TIMBER NBR into the global printout report (see Chapter 6.1, page 46). To export the tables and graphics, then select the printout report menu

```
\textbf{File} \rightarrow \textbf{Export to RTF}.
```

This function is described in the RFEM or RSTAB manual, Chapter 10.1.11.

Excel

RF-/TIMBER NBR provides a function for the direct data export to MS Excel or the CSV file format. To open the corresponding dialog box, select the menu

File \rightarrow Export Tables.

Export of Tables								
Table Parameters	Application Microsoft Excel CSV file format							
Transfer Parameters Export table to active workbook Export table to active worksheet Rewrite existing worksheet								
Selected Tables Active table Export hidden columns All tables Export tables with details Input tables Regult tables 								
Ø	OK Cancel							

Figure 7.12: Dialog box *Export of Tables*

When you have selected the relevant options, you can start the export by clicking [OK]. Excel will be started automatically, i.e. you do not have to open the program before.

🖹 🚽 🔊 - 🗘 - 📮 Sheet2 - Microsoft Excel															
File Home Insert Page Layout Formulas Data Review View Foxit Reader PDF PDF Architect Team 🛆 😮 📼 🗗								er XX							
$ \begin{array}{c c} & & & \\ & & & \\ & & & \\ Paste \\ & & \\ & & \\ & & \\ \end{array} $ Calibri $ \begin{array}{c} & & & 8 \\ & & & \\ B \\ & & & \\ \end{array} $ $ \begin{array}{c} & & & \\ & & \\ B \\ & & \\ \end{array} $ $ \begin{array}{c} & & \\ & & \\ & & \\ \end{array} $						E E Sort & Find &									
Clip	board 5	-	For		Gi .	-	Alignment 🛛 🖓	Number	- Fa	Styles	Cells		Editing)	
	A	1	• (*		fx Section										×
	А	В	С	D	E	F				G					A
1	Section	Member	Location	Load-											
2	No.	No.	x [m]	ing	Design					Design According to Form	ula				
3	10		le 160/260			10									
5		3	0,000	RC10 RC10	0,16	-				th in shear due to shear force Vz ac th in bending about y-axis acc. to 7					
6		3	1,980 0.040	RC10	0,39					ing in bending about y-axis acc. to 7					
7		5	0,040	NCIO	0,50		521) Stability - Ca	terai-torsional	DUCKI	ing in bending about y-axis acc. to	7.5.0				_
8	11	T-Rectang	le 260/160												
9		4	0,000	RC10	0,16	≤1	111) Cross-section resistance - Strength in shear due to shear force Vz acc. to 7.4.1					_			
10		4	1,980	RC10	0,63	≤1									
11															
12	12	T-Rectang	le 160/260												
13		5	0,000	RC10	0,16	≤1	111) Cross-sectio	n resistance - S	treng	th in shear due to shear force Vz ac	c. to 7.4.1				
14		5	1,980	RC10	0,39				-	th in bending about γ-axis acc. to 7					
15		5	0,040	RC10	0,27	≤1	321) Stability - La	teral-torsional	buckl	ing in bending about y-axis acc. to	7.5.6				_
16															
17 18	13		le 260/160			10									-
18		6	0,000	RC10 RC10	0,16				-	th in shear due to shear force Vz ac th in bending about y-axis acc. to 7					+
	N NI	-													
Rea		2.1 De	sign by Loa	d Case	2.2 De	sign	by Cross-Sect	$10n \neq 2.3$	Desi	an by S(I ◀	 ⊞□□□	100% (▶ <u> </u> _(+) .:
L ACC	□ □ 100% - + .:														



8 Example: Timber Beam

A timber beam with rectangular cross-section is designed according to NBR 7190:1997 [1]. The beam is restrained and subjected to compression and bending.

8

8.1 System and Loads

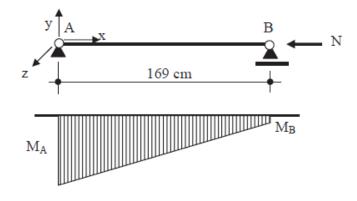


Figure 8.1: System and loads according to [2]

Model

Cross-section:	b/d = 6/12 cm
Material:	Hardwood C40
Moisture Class:	1
Category of Timber:	Second

Loads and Combinations

l = 15 kN
$\rm M_A=33.4~kN/cm$
$M_{\rm B}=$ 2.2 kN/cm
DC permanent
I = 10 kN
$\rm M_A=22.3~kN/cm$
$M_{\rm B}=1.4~{ m kN/cm}$
DC long-term
.4 LC1 + 1.4 LC2
.0 LC1 + 0.2 LC2

8.2 Calculation with RFEM/RSTAB

The system as well as the loads in all load cases is modeled in RFEM/RSTAB as a 3D model. We deactivate the automatic consideration of the self-weight when we create LC1.

We create load combination CO1 with the respective factors from the defined load cases. Then we calculate the model according to the linear static analysis.

For the SLS combination CO2, it is necessary to consider the *Effective Modulus of Elasticity*, $E_{c0,ef}$. This value can be calculated according to the formula in [1] 6.4.9 as follows:

$$k_{\text{mod}} = k_{\text{mod},1} \cdot k_{\text{mod},2} \cdot k_{\text{mod},3}$$
$$k_{\text{mod}} = 0.7 \cdot 1.0 \cdot 0.8 = 0.56$$
$$E_{c0,ef} = k_{\text{mod}} \cdot E_{c0,m} = 0.56 \cdot 1,950 = 1,092 \text{ MPa}$$

We define the factor $k_{mod} = 0.56$ in the *Edit Load Cases and Combinations* dialog box, *Modify Stiffness* tab. It can be assigned individually to each material. As there is only one material in our example, however, we can modify the stiffness globally.

Edit Load Cases and Combinations									
Load Cases Load Combinations Result Combinations	Super Combinations								
Existing Load Combinations	CO No. Load Combination Description								
CO1 1.4*LC1 + 1.4*LC2	2								
CO2 LC1 + 0.2*LC2									
	General Calculation Parameters Modify Stiffness								
	Materials and Cross-Sections								
	✓ Materials (E, G)								
	Multiply all with factor: 0.560 - [-]								
	O Ingividually								
	Cross-sections (A, Ay, Az, J, Iy, Iz)								
	Multiply all with factor:								
	🔿 Indi <u>v</u> idually 💽								

Figure 8.2: Modifying Modulus of Elasticity in the *Edit Load Cases and Combinations* dialog box



There would be even more options to modify the stiffness in the *Edit Load Cases and Combinations* dialog box: For example, we could allocate specific stiffness factors to members and consider the modified stiffness in the *Edit Member* dialog box separately for each member.

RFEM/RSTAB determines the diagrams of internal forces and deformation as seen in Figure 8.3.

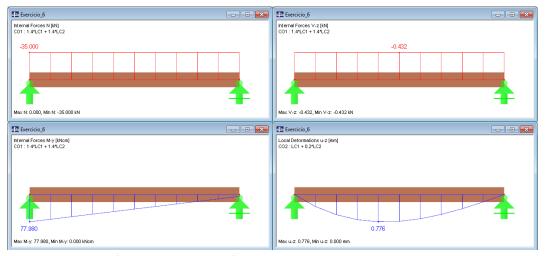


Figure 8.3: Internal forces N, M_y, V_z, and deformation u_z

8.3 Design with RF-/TIMBER NBR

8.3.1 Ultimate Limit State Design

In the 1.1 General Data window, we select **CO1** for the Ultimate Limit State design.

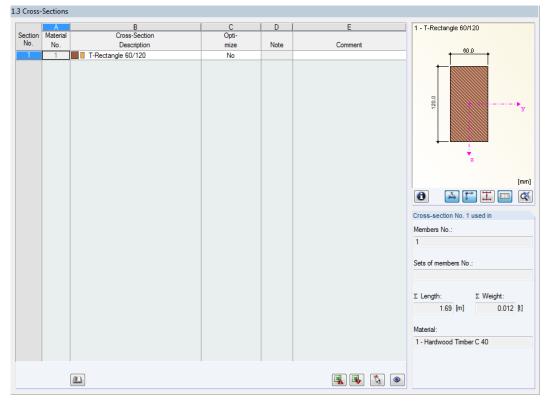
1.1 General Da	ta					
Design of						
Members:	1		X			
Sets:			\$ X	🞦 🗌 Aļ		
	State Serviceability Limit State		Selected for D	laging		æ
G LC1 A LC2 C02	LC1 + 0.2°LC2	∧ ∧ ₹	CO1	1.4°LC1 + 1.4°LC2		RF-TIMBER NBR Brazilian Standard
						Timber design of members and sets of members according to - NBR 7190:1997
All (3)	- 24 83					I New Col Lar
Comment						
				*		

Figure 8.4: Window 1.1 General Data

The next windows contain information on the selected material and cross-section.

	A	B		С	
aterial	Material	Type of Timbe	er		
No.	Description			Comment	
1	Hardwood Timber C 40 NBR 7190:1997	Solid Timber			
					•
	Properties Properties				
	dulus of Elasticity	E	19500.000	MPa	
	ar Modulus	G	975.000		
	cific Weight	7		kN/m ³	
	fficient of Thermal Expansion	α	5.0000E-06		
	tial Safety Factor	7M	1.00		
	onal Properties	1.00	1.00	1	
	ing Shear Strength	fR.k	0.10	kN/cm ²	Material No. 1 used in
	dulus of Elasticity for Compression	Ec0.m	19500.000		Cross-sections No.:
	dulus of Elasticity for Compression Perpendicular	Ec90.m	975.000		1
	racteristic Strength for Bending	ftM.k		kN/cm ²	
	racteristic Strength for Tension	ft0,k		kN/cm ²	
	aracteristic Strength for Tension Perpendicular	ft90.k		kN/cm ²	Members No.:
	racteristic Strength for Compression	fc0,k		kN/cm ²	1
	racteristic Strength for Compression Perpendicular	fc90,k	1.00	kN/cm ²	
Cha	racteristic Strength for Shear/Torsion	fv0,k	0.60	kN/cm ²	Sets of members No.:
Bas	ic Density	ρbas,m		kg/m ³	
					Σ Length: Σ Weight: 1.69 [m] 0.012 [t

Figure 8.5: Window 1.2 Materials



8

Figure 8.6: Window 1.3 Cross-Sections

In Window 1.4 Load Duration, factor $k_{mod,1}$ of CO1 is determined according to the shortest load duration within the combination. As the stability analysis for members defined by slenderness $80 < \lambda \le 140$ is performed according to [1] 7.5.5, it is also necessary to define the coefficients ψ_1 and ψ_2 . The criterion 'Permanent' or 'Variable' for the stability analysis is applied according to the LDC settings.

Load L	Ouration							
	A	В	C	D	E	F	G	Load Duration - Explanatory Notes
Load-		Load	Load Duration	Modification Factor	Combination	Coefficient		
ing	Description	Туре	Class	k mod, 1	Ψ1	Ψ2	Comment	Permanent:
LC1		Permanent	Permanent	0.600				Design working life
LC2		Accidental	Long-term	0.700				Long-term:
CO1	1.4*LC1 + 1.4*LC2	-	Long-term	0.700	0.300	0.200		More than six months
								Medium-term:
								One week to six months
								Short-term:
								Less than one week
								Instantanenous:
								Very short

Figure 8.7: Window 1.4 Load Duration

In Window 1.5 Service Conditions - Members, we specify the moisture class and the timber category. They determine the modification factors $k_{mod,2}$ and $k_{mod,3}$.

Service	Conditions - Members					
	А	B	C	D	E	Moisture Class
/lember	Moisture Class	Classification	Modificatio			Moisture Class (1):
No.		of Timber	k mod,2	k mod,3	Comment	Relative humidity of environment:
1	(1)	Second Category	1.000	0.800		$U_{amb} \leq 65\%$
						Average moisture content of timber: U _{eq} = 12%
						Moisture Class (2):
						Relative humidity of environment:
						$65\% < U_{amb} \le 75\%$ Average moisture content of timber: $U_{eq} = 15\%$
						Moisture Class (3):
						Relative humidity of environment:
						75% < U _{amb} ≤ 85%
						Average moisture content of timber: U _{eq} = 18%
						Moisture Class (4):
						Relative humidity of environment: $U_{amb} > 85\%$ for long periods Average moisture content of timber: $U_{ep} \ge 25\%$
						Classification of Timber
						First Category:
						The condition of first category timber can only be assumed if all the structural members are without defects. This has to be ensured by a visual method and by mechanical tests, which guarantees the homogenity of the applied timber.
						Second Category:
Set inp	out for members No.:					All the types of timber which are not of first category.
		1	V Al		🖳 🐧	S

Figure 8.8: Window 1.5 Service Conditions - Members

In Window 1.7 Effective Lengths - Members, we define the buckling lengths of the beam. The default buckling length coefficients $k_{cr,y} = k_{cr,z} = 1.0$ are adequate for our example. Likewise, the effective length for lateral-torsional buckling is equal to the member length, $L_1 = 169$ cm.

	Δ	B	С	D	E	F	G	Н		J	K
Member	Buckling		uckling About A			uckling About A			eral-Torsional Bu		N
No.	Possible	Possible	k _{or,y}	Lo.y [m]	Possible	k _{or,z}	Lo,z [m]	Possible	L ₁ Manually	L1[m]	Comment
1	V	J	1.000	1.690	J	1.000	1.690			1.690	

Figure 8.9: Window 1.7 Effective Lengths - Members

Calculation

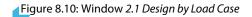
Then we can start the [Calculation].

Results

After the calculation, Window 2.1 Design by Load Case presents the governing designs.

8

	A	B	С	D	E				F				G
oad-		Member	Location										Load
ng	Description	No.	x [m]	Design			De	sign Acc	ording to For	mula			Duration
	Ultimate Limit State Design												
01	1.4*LC1 + 1.4*LC2	1	0.000	0.78	≤ 1 302) Sta	ability - Compre	ssion parall	el to grair	n with bucklir	ig abo	out y-axis acc	. to 7.5.4	Long-ter
			Max:	0.78	≤1 🕲		9			> 1	.0 🔻	7 😫 🖪	
ails -	Member 1 - x: 0.000 m - CO	1									1 - T-Recta	ngle 60/120	
	n Ratio												
	mpressive Force				N _{c,d}	-35.000							
Wi					b	60.0							
De					d	120.0				_			
	a of Cross-Section				A	7200.0							
	mpressive Stress				ØNc,d		kN/cm ²						
	ective Length				Lo.y	1.690							
Ra	dius of Gyration	s of Gyration				34.6	mm					. 60.0 .	
	Slendemess Degree				λγ	48.786			7.5.1			+ 00.0	
Characteristic Compressive Strength				fc0,k		kN/cm ²		Table 8		+			
	dulus of Elasticity for Compre	ssion			E _{c0,m}		kN/cm ²		Table 8				
	dification Factor				k mod,1	0.700			6.4.4				
	dification Factor				k mod,2	1.000			6.4.4				
Mo	dification Factor				k mod,3	0.800			6.4.4		20.0		
Мо	dification Factor				k mod	0.560			6.4.4		÷.		
	sistance Factor				7wc	1.400			6.4.5				
De	sign Compressive Strength P	arallel to Grai	in		fc0,d		kN/cm ²		7.2.6	Ξ			
Effe	ective modulus of elasticity fo	r compressio	n parallel to g	grain	E _{c0,ef}		kN/cm ²		6.4.9		∔		
	ment of Inertia				ly	8640000.0						1	
Crit	ical Force				FE.y	326.034	kN		7.5.4			z	
-	ical Force Check				Nc,d/FE,y	0.107							
	tual Design Bending Moment				M _{1.y.d}	77.980							
	centricity (Minimum Accident	al)			ea		mm		7.5.2				
Ec	centricity (Initial)				ei	22.3			7.5.4				
Ec	centricity (First Order)				e1	27.9			7.5.4				
Eco	centricity (Design)				ed	31.3	mm		7.5.4				
Ber	nding Moment from Eccentric	ities			Md	109.446	kNcm		7.5.4				
Ela	stic Section Modulus				Sy.c	-144000.0	mm ³						ſr
De	sign Strength for Compressio	n due to Ben	ding Moment	Md	σMd	-0.76	kN/cm ²		7.5.4		_		
De	sian Ratio				n	0.78		≤1	7.5.4	-	0	X	1



The Details in the lower section correspond to the designs in [1].

Design Strengths

Design compressive strength

$$\begin{split} f_{c0,d} &= k_{\text{mod}\,,1} \cdot k_{\text{mod}\,,2} \cdot k_{\text{mod}\,,3} \cdot \frac{f_{c0,k}}{\gamma_{wc}} \\ f_{c0,d} &= 0.70 \cdot 1.00 \cdot 0.80 \cdot \frac{4.00}{1.40} \\ f_{c0,d} &= 1.60 \text{ kN/m}^2 \end{split}$$

Design tensile strength

$$f_{t0,d} = k_{\text{mod},1} \cdot k_{\text{mod},2} \cdot k_{\text{mod},3} \cdot \frac{f_{t0,k}}{\gamma_{wt}}$$
$$f_{t0,d} = 0.70 \cdot 1.00 \cdot 0.80 \cdot \frac{5.20}{1.80}$$
$$f_{t0,d} = 1.62 \text{ kN/m}^2$$

Design shear strength

$$f_{v0,d} = k_{\text{mod},1} \cdot k_{\text{mod},2} \cdot k_{\text{mod},3} \cdot \frac{f_{v0,k}}{\gamma_{wv}}$$
$$f_{v0,d} = 0.70 \cdot 1.00 \cdot 0.80 \cdot \frac{0.60}{1.80}$$
$$f_{v0,d} = 0.19 \text{ kN/m}^2$$

Effective modulus of elasticity

$$E_{c0,ef} = k_{\text{mod},1} \cdot k_{\text{mod},2} \cdot k_{\text{mod},3} \cdot E_{c0,m}$$
$$E_{c0,ef} = 0.70 \cdot 1.00 \cdot 0.80 \cdot 1,950$$
$$E_{c0,ef} = 1,092 \text{ kN/m}^2$$

Stress Analysis

Compressive stress

$$\sigma_{N_{c,d}} = \frac{N_{c,d}}{A} = \frac{35 \text{ kN}}{72 \text{ cm}^2} = 0.49 \text{ kN/cm}^2$$

Bending stress

$$\sigma_{M_y,c,d} = \sigma_{M_y,t,d} = \frac{M_{y,d}}{S_y} = \frac{77.98 \text{ kNcm}}{144 \text{ cm}^3} = 0.54 \text{ kN/cm}^2$$

Shear stress

$$\tau_{V_{z,d}} = \frac{V_{z,d} \cdot Q_y}{I_y \cdot b} = \frac{0.432 \text{ kN} \cdot 108 \text{ cm}^3}{864 \text{ cm}^4 \cdot 6 \text{ cm}} = 0.01 \text{ kN/cm}^2$$

Buckling Design

The verification of buckling requires the calculation of additional moments due to eccentricities, as specified in [1] 7.5.

Axis y

$$\lambda_y = \frac{L_{0,y}}{i_y} = \frac{169 \text{ cm}}{3.464 \text{ cm}} = 48.79$$

As the slenderness is 40 $<\lambda \leq$ 80, the buckling design about the y-axis follows [1] 7.5.4.

Critical force

$$F_{E,y} = \frac{\pi^2 \cdot E_{c0,ef} \cdot I_y}{L_{0,y}^2} = \frac{\pi^2 \cdot 1,092 \text{ kN/m}^2 \cdot 864 \text{ cm}^4}{(169 \text{ cm})^2} = 326.03 \text{ kN}$$

Eccentricity (minimum accidental)

$$e_a = \frac{L_{0,y}}{300} = \frac{169 \text{ cm}}{300} = 0.563 \text{ cm}$$

Eccentricity (initial)

$$e_i = \max\left(\frac{M_{1,y,d}}{N_{c,d}}; \frac{h}{30}\right) = \max\left(\frac{77.98 \text{ kNcm}}{35.00 \text{ kN}}; \frac{12 \text{ cm}}{30}\right) = 2.228 \text{ cm}$$

Eccentricity (first order)

$$e_1 = e_a + e_i = 0.563 \text{ cm} + 2.228 \text{ cm} = 2.791 \text{ cm}$$

Eccentricity (design)

$$e_d = e_1 \cdot \left(\frac{F_{E,y}}{F_{E,y} - N_{c,d}}\right) = 2.791 \text{ cm} \cdot \left(\frac{326.03 \text{ kN}}{326.03 \text{ kN} - 35.00 \text{ kN}}\right) = 3.127 \text{ cm}$$

Bending moment from eccentricities

 $M_d = N_{c,d} \cdot e_d = 35.00 \text{ kN} \cdot 3.127 \text{ cm} = 109.44 \text{ kNcm}$

Design stress for compression due to bending moment from eccentricities

$$\sigma_{M_d} = \frac{M_d}{S_y} = \frac{109.44 \text{ kNcm}}{144 \text{ cm}^3} = 0.76 \text{ kN/cm}^2$$

Axis z

$$\lambda_z = rac{L_{0,z}}{i_z} = rac{169 \text{ cm}}{1.732 \text{ cm}} = 97.572$$

As the slenderness is 80 $< \lambda \leq$ 140, the buckling design about the z-axis follows [1] 7.5.5.

Critical Force

$$F_{E,z} = \frac{\pi^2 \cdot E_{c0,ef} \cdot I_z}{L_{0,z}^2} = \frac{\pi^2 \cdot 1,092 \text{ kN/m}^2 \cdot 216 \text{ cm}^4}{(169 \text{ cm})^2} = 81.51 \text{ kN}$$

Eccentricity (minimum accidental)

$$e_a = \max\left(\frac{L_{0,z}}{300}; \frac{b}{30}\right) = \max\left(\frac{169 \text{ cm}}{300}; \frac{6 \text{ cm}}{30}\right) = 0.563 \text{ cm}$$

Eccentricity, permanent loads (first order)

$$e_{ig} = rac{M_{1g,z,d}}{N_{g,c,d}} = rac{0 \text{ kNcm}}{21.00 \text{ kN}} = 0$$

Eccentricity (initial)

$$e_i = \frac{M_{1,z,d}}{N_{c,d}} = \frac{0 \text{ kNcm}}{35.00 \text{ kN}} = 0$$

Eccentricity (creep)

$$\begin{aligned} e_c &= (e_a + e_i) \cdot \left\{ \exp\left[\frac{\phi \cdot \left[N_{g,c,k} + (\psi_1 + \psi_2) \cdot N_{q,c,k}\right]}{F_{E,z} - \left[N_{g,c,k} + (\psi_1 + \psi_2) \cdot N_{q,c,k}\right]}\right] - 1 \right\} \\ e_c &= (0 + 0.56\overline{3}) \cdot \left\{ \exp\left[\frac{0.8 \cdot \left[15.00 + (0.3 + 0.2) \cdot 10.00\right]}{81.51 - \left[15.00 + (0.3 + 0.2) \cdot 10.00\right]}\right] - 1 \right\} = 0.1674 \, \mathrm{cm} \end{aligned}$$

Eccentricity (first order, effective)

$$e_{1,ef} = e_i + e_a + e_c = 0 + 0.563 \text{ cm} + 0.1674 \text{ cm} = 0.7307 \text{ cm}$$

Bending moment from eccentricities

$$M_d = N_{c,d} \cdot e_{1,ef} \left(\frac{F_{E,z}}{F_{E,z} - N_{c,d}} \right) = 35.00 \text{kN} \cdot 0.7307 \text{cm} \left(\frac{81.51 \text{ kN}}{81.51 \text{ kN} - 35.00 \text{ kN}} \right) = 44.82 \text{kNcm}$$

Design stress for compression due to bending moment from eccentricities

$$\sigma_{M_d} = \frac{M_d}{S_z} = \frac{44.82 \text{ kNcm}}{72 \text{ cm}^3} = 0.62 \text{ kN/cm}^2$$

Combined Bending and Axial Compression Design

The bending stresses are determined separately for the tension and compression zones according to [1] 7.3.6.

Design 1 - tension zone

$$-\left(\frac{\sigma_{N_{c,d}}}{f_{c0,d}}\right)^{2} + \frac{\sigma_{M_{y,t,d}}}{f_{t0,d}} + k_{M} \cdot \frac{\sigma_{M_{z,t,d}}}{f_{t0,d}} \le 1.00$$
$$-\left(\frac{0.49}{1.60}\right)^{2} + \frac{0.54}{1.62} + 0.5 \cdot \frac{0}{1.62} = 0.24 \le 1.00$$

Design 2 – compression zone

$$\left(\frac{\sigma_{N_{c,d}}}{f_{c0,d}}\right)^2 + \frac{\sigma_{M_{y,c,d}}}{f_{t0,d}} + k_M \cdot \frac{\sigma_{M_{z,c,d}}}{f_{t0,d}} \le 1.00$$
$$\left(\frac{0.49}{1.60}\right)^2 + \frac{0.54}{1.60} + 0.5 \cdot \frac{0}{1.60} = 0.43 \le 1.00$$

Shear Design

The design of shear is performed according to [1] 7.4.1.

Design

$$\frac{\tau_{V_{z,d}}}{f_{v0,d}} = \frac{0.01}{0.19} = 0.05 \le 1.00$$

Compression with Buckling Design

The design of compression with buckling follows [1] 7.5:

Design 1 – buckling about y-axis according to 7.5.4

$$\begin{aligned} &\frac{\sigma_{N_{c,d}}}{f_{t0,d}} + \frac{\sigma_{M_d}}{f_{t0,d}} \leq 1.00\\ &\frac{0.49}{1.60} + \frac{0.76}{1.60} = 0.78 \leq 1.00 \end{aligned}$$

Design 2 – buckling about z-axis according to 7.5.5

$$\begin{aligned} &\frac{\sigma_{N_{c,d}}}{f_{t0,d}} + \frac{\sigma_{M_d}}{f_{t0,d}} \leq 1.00\\ &\frac{0.49}{1.60} + \frac{0.62}{1.60} = 0.69 \leq 1.00 \end{aligned}$$

Lateral-Torsional Buckling Design

The lateral-torsional buckling design is performed according to [1] 7.5.6.

At first, the correction factor, $\beta_{\rm M}$, must be determined.

$$\beta_{M} = \frac{1}{0.26 \pi} \cdot \frac{\beta_{E}}{\gamma_{f}} \cdot \frac{\left(\frac{h}{b}\right)^{\frac{3}{2}}}{\left(\frac{h}{b} - 0.63\right)^{\frac{1}{2}}} = \frac{1}{0.26 \pi} \cdot \frac{4}{1.4} \cdot \frac{\left(\frac{12 \text{ cm}}{6 \text{ cm}}\right)^{\frac{3}{2}}}{\left(\frac{12 \text{ cm}}{6 \text{ cm}} - 0.63\right)^{\frac{1}{2}}} = 8.453$$

Design

$$\frac{L_1}{b} = \frac{169 \text{ cm}}{6 \text{ cm}} = 28.17 \le \frac{E_{c0,ef}}{\beta_M \cdot f_{c,0,d}} = \frac{1,092 \text{ kN/m}^2}{8.453 \cdot 1.60 \text{ kN/m}^2} = 80.74$$

8.3.2 Serviceability Limit State Design

For the SLS design, we created load combination **CO2**. We select that load combination in the *Serviceability Limit State* tab of Window *1.1 General Data* for design, and we make sure that the deflection limit for *Permanent and live actions* for usual constructions according to [1] 9.2.1 is set.

.1 General D	ata					
Design of <u>M</u> embers: Sets:	1		X	All I		
	ad Cases / Combinations	A 8	Selected for <u>D</u> CO2	esign LC1 + 0.2°LC2	Permanent and live actions	RE-TINBER NBR Brazilian Standard Imber design of members according to - NBR 7190:1997
AII (3) 🗸 🛃					
<u>C</u> omment				×		

Figure 8.11: Window 1.1 General Data

Then we enter member 1 in Window 1.9 Serviceability Data.

eability Data								
A	В	С	D	E	F	G	Н	
	Member	Referen	ce Length	Direc-	Preca	amber		
Reference to	No.	Manually	L [m]	tion	w _{c,y} [mm]	w _{c,z} [mm]	Beam Type	Comment
Member	1		1.690	z 💌		0.000	Beam	
	A Reference to	A B Member Reference to No.	A B C Member Referen Reference to No. Manually	A B C D Member Reference Length No. Manually L [m]	A B C D E Member Reference Length Direc- Reference to No. Manually L [m] tion	A B C D E F Member Reference Length Direc- Preca Reference to No. Manually L [m] tion w _{c.y} [mm]	A B C D E F G Member Reference Length Direc- Precamber Reference to No. Manually L [m] tion w _{o,y} [mm] w _{o,z} [mm]	Member Reference Length Direc- Precamber Reference to No. Manually L [m] tion w c.y [mm] w c.z [mm] Beam Type

Figure 8.12: Window 1.9 Serviceability Data

We do not modify the reference length, but we select only the direction z. The type of beam is set as *Beam* by default, which confines the deformation to L/200.

For the calculation, we change one setting in the *Details* dialog box, tab *Serviceability*: The deformation is to be relative to the **Undeformed system**.

8

Details	
Resistance Stability Serviceability Other	
Servicability (Deflections)	
Deformation relative to:	
Shifted members ends / set of member ends	
Undeformed system	

Figure 8.13: Dialog box Details, tab Serviceability

As mentioned in Chapter 8.2, the effective modulus of elasticity $E_{c0,ef} = 1,092$ MPa is defined in RFEM or RSTAB.

The calculation of RFEM or RSTAB gives the maximum deformation in the local z-direction as $u_z = 0.776$ mm.

Design

$$\frac{u_z}{\frac{L}{200}} = \frac{0.776 \,\mathrm{mm}}{\frac{169 \,\mathrm{cm}}{200}} = 0.09 \le 1.00$$

Calculation

After the [Calculation], this result of the deformation analysis is displayed in Window 2.1 Design by Load Case for the Serviceability Limit State Design table item.

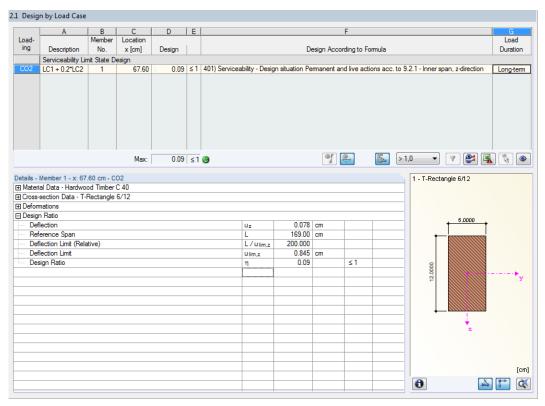


Figure 8.14: Window 2.1 Design by Load Case



- [1] NBR 7190:1997: Design of wooden structures. Associação Brasileira de Normas Técnicas, 1997.
- [2] Notas de Aula de Estruturas de Madeira. FRANCISCO A. R. GESUALDO, 2003.

Dlub

B

Index

В

-	
Background graphic	41
Beam type	24
Buckling	21
Buckling length 21,	59
Buckling length coefficient	22
Button	40

С

-
Calculation
Cantilever
Clipboard
Color scale
Combination coefficient
Comment
Connection
Control Panel
Criterion
Cross-section
Cross-section design
Cross-section info15
Cross-section library
Cross-section optimization

D

Decimal places	. 11, 52
Deformation	27
Deformation analysis	24
Design	. 30, 32
Design case	, 48, 49
Design colored	44
Design ratio	. 14, 32
Detail settings	25
Display navigator	. 42, 44

Ε

Effective length
Equivalent member
Equivalent member length 22
Excel
Exit RF-/TIMBER NBR
Export
Export cross-section
Export effective length
Export material

F

Filter
Filtering members
Flexural buckling

G

General data
Glued laminated timber
Graphic
Graphic printout
Graphical representation

I

Imperfections	6
Installation	3
Internal forces	1

L

Lateral-torsional buckling
Length
Length for lateral-torsional buckling 22
Limit tension stress
Limit values
List of members
Load case
Load combination7
Load duration
Load duration factor
Load type
Location x

Μ

Material 11,	, 53
Material description	.11
Material library	12
Material properties	.11
Member	. 6
Member slendernesses	. 36
Modification factor	9
Moisture class	. 18

Ν

Navigator
O Optimization 15, 28, 50, 51
P Panel

⊿ Dlubal

Parametric cross-section	, 50
Part	. 38
Particle board	. 12
Parts list	, 39
Permanent	. 17
Plywood	. 12
Precamber	. 24
Print	. 15
Printout report	, 47

R

Reference length8Relation scales40
Rendering 44
Resistance
Result combination7
Result diagram
Result values
Result window
Results evaluation 40
RF-STABILITY
RFEM/RSTAB graphic
RSBUCK

S

Second order analysis	26
Selecting windows	. 5
Service condition	18
Serviceability	24
Serviceability limit state	40

Set of members 6, 19, 23, 24, 33, 36, 39
Slenderness
Solid timber
Stability analysis
Standard
Start calculation
Start program
Start RF-/TIMBER NBR 4
Stress point
Sum
Surface Area

B

U

Ultimate limit state	40, 57
Units	11, 52

V

Variable	17
View mode	19, 40, 41
Visibility	
Volume	

Х

x-location																				3	1
------------	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	---	---

W

Weight
Windows
Work window