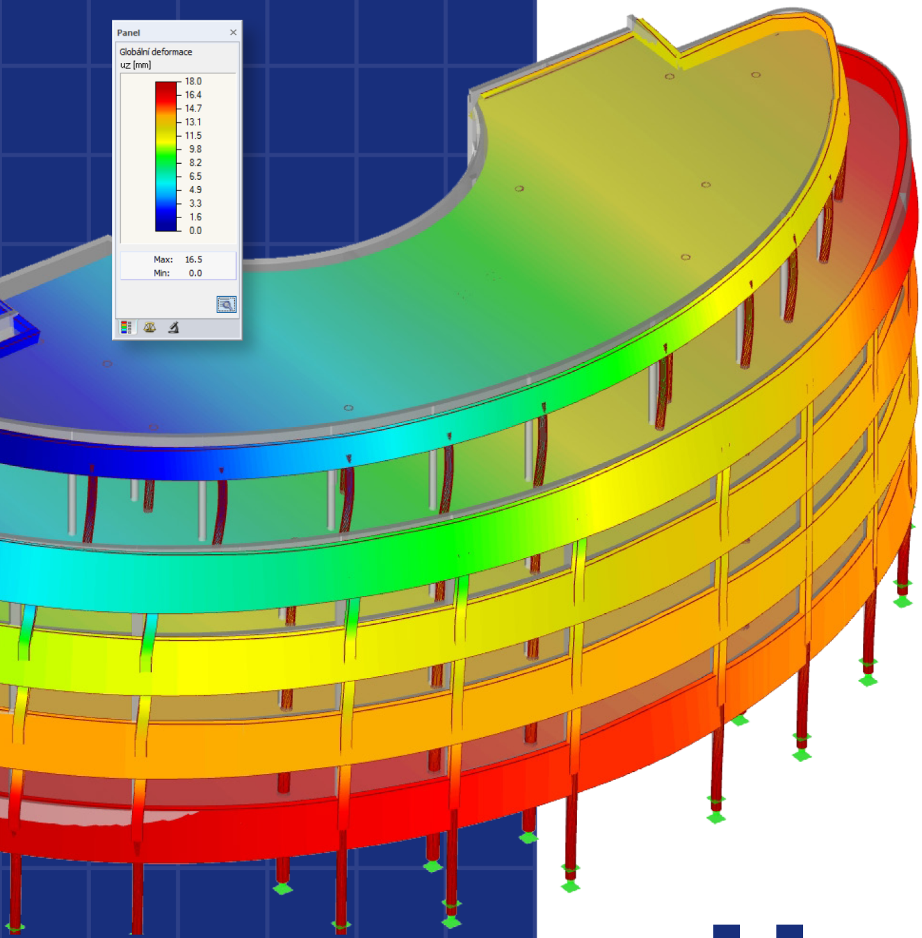
RF-/TIMBER Pro

Design of Timber Members According to EN 1995, DIN 1052, and SIA 265



User Manual

Version

June 2018

Dlubal Software

Short Overview

Dlubal Software GmbH
Am Zellweg 2
93464 Tiefenbach
Germany

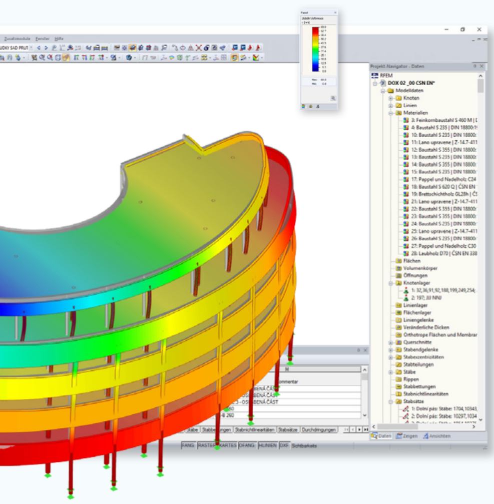
Telephone: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com

Dlubal Software, Inc.
The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Phone: +1 267 702-2815
E-mail: info@dlubal.com

Using the Manual

The program description is organized in chapters which follow the order and structure of the input and result tables. The chapters present the individual tables column by column. They help to better understand the functioning of the add-on module. General functions are described in the manuals of the main program RFEM or RSTAB.



Hint

The text of the manual shows the described buttons in square brackets, for example [OK]. In addition, they are pictured on the left. Expressions appearing in dialog boxes, tables, and menus are set in *italics* to clarify the explanation. You can also use the search function for the Knowledge Base and FAQs to find a solution in the posts about add-on modules.

Topicality

The high quality standards placed on the software are guaranteed by a continuous development of the program versions. This may result in differences between program description and the current software version you are using. Thank you for your understanding that no claims can be derived from the figures and descriptions. We always try to adapt the documentation to the current state of the software.

Table of Contents

1 Introduction

1.1 Add-on Module RF-/TIMBER Pro

Eurocode 5 (EN 1995-1-1:2010-12 [1] + A1:2008) regulates the draft, design, and construction of timber structures in the member states of the European Union. With the add-on modules RF-TIMBER Pro (for RFEM) and TIMBER Pro (for RSTAB) Dlubal Software offers you powerful tools for the design of timber structures modeled with beam elements. Country-specific regulations are taken into account by different National Annexes (NA). In addition to the parameters included in the program, you can define your own limit values or create new National Annexes. Moreover, it is possible to perform designs according to DIN 1052:2008 [2] and SIA 265:2012 [3] in RF-/TIMBER Pro.

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

RF-/TIMBER Pro performs the ultimate limit state designs, stability analyses, and deformation analyses provided by the standards. The stability analysis can be carried out according to the equivalent member method or a second-order analysis. If the equivalent member method is applied, the program considers regular axial compression parallel to the grain, bending without compression force, bending and compression, shear due to shear force, as well as bending and tension. Furthermore, the fire resistance design according to EN 1995-1-2 [5], DIN 4102-4 [4] or SIA 265 [3] is possible.

In timber construction, the serviceability limit state represents an important design. Load cases, load combinations, and result combinations can be assigned to different design situations. The limit deformations are preset by the National Annex and can be adjusted, if necessary. In addition, it is possible to specify reference lengths and precambers that will be considered accordingly in the design.

The program provides an automatic cross-section optimization with the possibility to export modified cross-sections to RFEM or RSTAB. Separate design cases allow for a flexible analysis of structural components in complex models.

Like other add-on modules, RF-/TIMBER Pro is completely integrated in RFEM or RSTAB. Thus, the design-relevant input data is preset when you start the add-on module. Subsequent to the design, you can use the graphical user interface of the main program to evaluate the results. As they can be 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 Pro.

Your Dlubal Software 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 Pro add-on module.

The descriptions in this manual follow the sequence and structure of the input and result windows. In the text, the described **buttons** are given in square brackets, for example [Apply]. 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 explanation.

In the PDF manual, you can perform a full-text search as usual with [Ctrl]+[F]. However, if you cannot find what you are looking for, you can also go to the Knowledge Base on our website to find related articles about the timber add-on modules. Or consult the FAQs on our website.

1.3 Open the Add-on Module RF-/TIMBER Pro

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

Menu

To start the program on the RFEM or RSTAB menu bar, select

Add-on Modules → Design - Timber → RF-/TIMBER Pro.

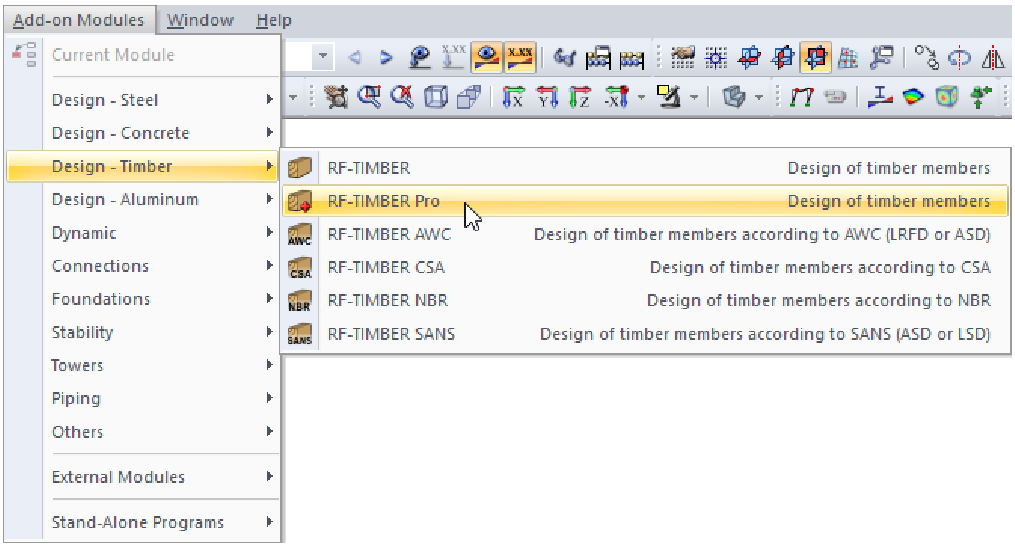


Figure 1.1 Menu *Add-on Modules → Design - Timber → RF-TIMBER Pro*

Navigator

To start RF-/TIMBER Pro in the *Data* navigator, select

Add-on Modules → RF-/TIMBER Pro.

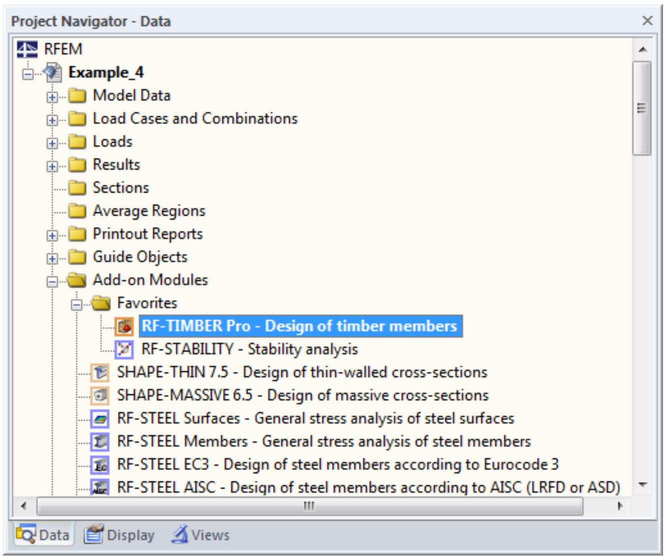
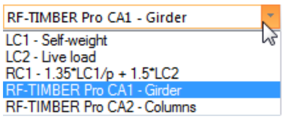


Figure 1.2 *Data navigator: Add-on Modules → RF-TIMBER Pro*

Panel



If any results from RF-/TIMBER Pro are already available in the model, you can open the design module on the panel:

Set the relevant design case in the load case list of the menu bar. Click the [Show Results] button to display the design criterion graphically on the members.

When the results display is activated, the panel appears showing the [RF-/TIMBER Pro] button which you can use to open the add-on module.

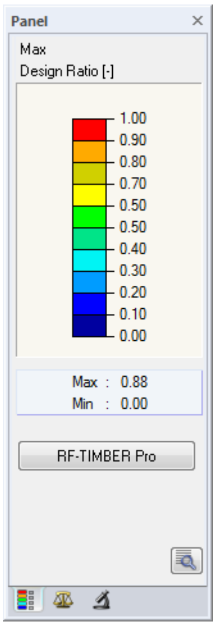


Figure 1.3 Panel with [RF-TIMBER Pro] button

2 Input Data

When you have started the add-on module, a new window appears. In this window, a navigator is displayed on the left, managing the available module windows. The drop-down list above the navigator contains the design cases (see Chapter 7.1).

The design-relevant data must be defined in several input windows. When you open RF-/TIMBER Pro for the first time, the following parameters will be imported automatically:

- Members and sets of members
- Load cases, load combinations, and result combinations
- Materials
- Cross-sections
- Buckling lengths
- Internal forces (in background, if calculated)

To select a window, click the corresponding entry in the navigator. To go to the previous or subsequent module 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 entered data, click [OK]. You will exit RF-/TIMBER Pro and return to the main program. Click [Cancel] to exit the add-on module without saving the new data.

2.1 General Data

In the *1.1 General Data* window, you select the members, sets of members, and actions that you want to design. Three tabs are managing the load cases, load combinations, and result combinations for the ultimate limit state, the serviceability limit state, and the fire protection design.

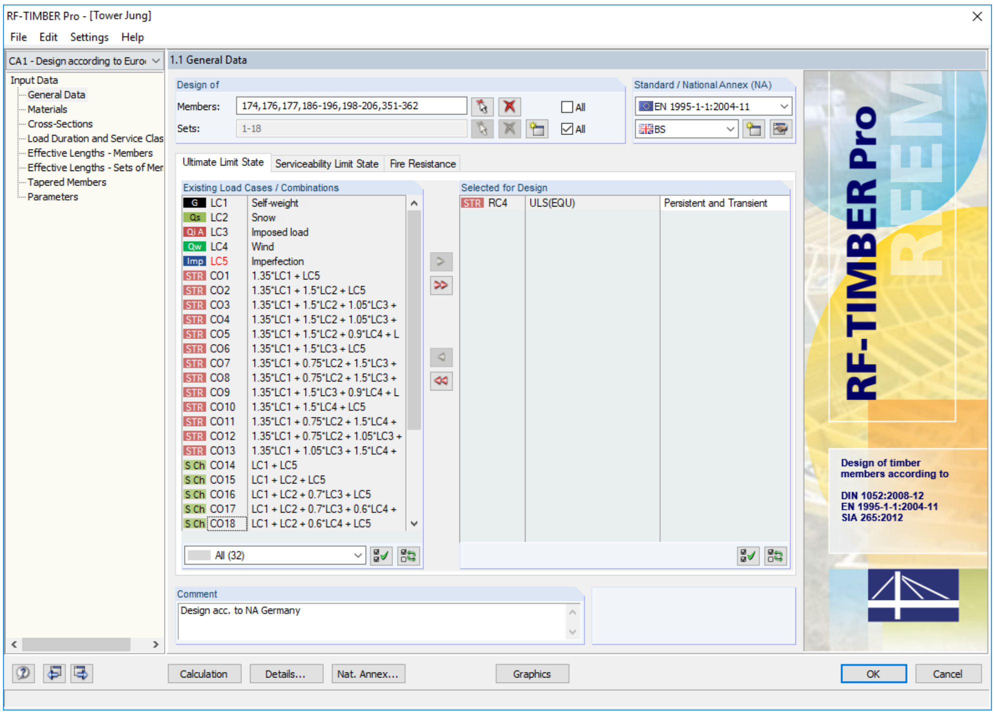


Figure 2.1 Window *1.1 General Data*

Design of

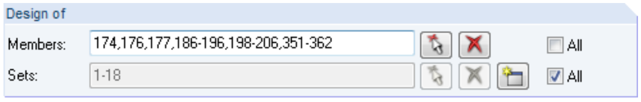


Figure 2.2 Design of members and sets of members

You can design *Members* as well as *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. Use the [Delete] button to clear the list of preset numbers. Use the [Select] button to define the objects graphically in the RFEM or RSTAB work window.

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

To define a new set of members, click the [New] button. The dialog box known from RFEM or RSTAB appears where you can enter the parameters for the set of members.

Standard / National Annex (NA)

In the drop-down list in the upper-right corner of the window, you can select the standard whose parameters apply to the design and to the deformation's limit values. You can select from:

- DIN 1052:2008-12 [2]
- EN 1995-1-1:2004-11 [1]
- SIA 265:2012 [3]

If you select EN 1995-1-1, you also have to specify the National Annex.

Figure 2.3 National Annexes for EN 1995-1

Use the [Edit] button to open a dialog box where you can check and adjust, if necessary, the parameters of the current NA. The dialog box is described in Chapter 2.1.4.

Comment

Figure 2.4 User-defined comment

In this text box, you can enter user-defined notes.

2.1.1 Ultimate Limit State

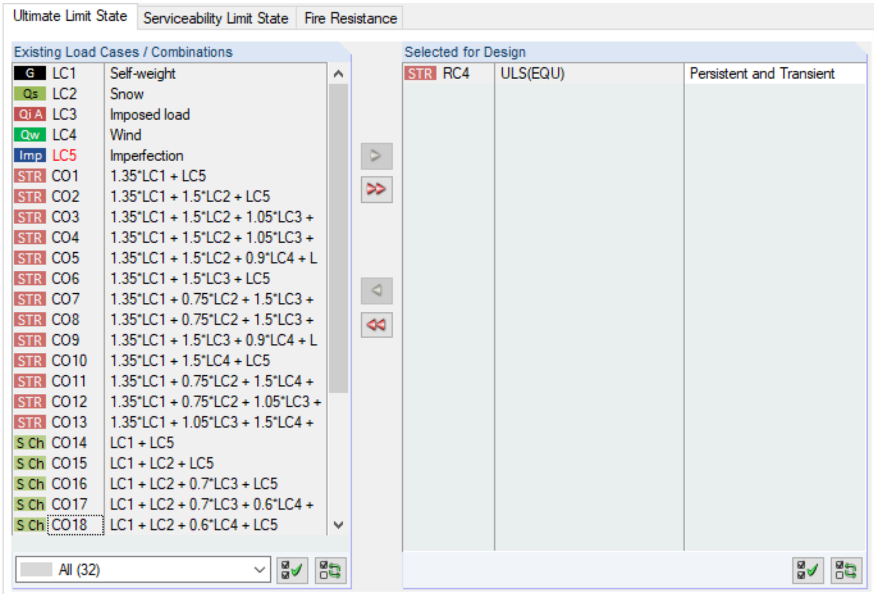


Figure 2.5 Window *1.1 General Data*, tab *Ultimate Limit State*

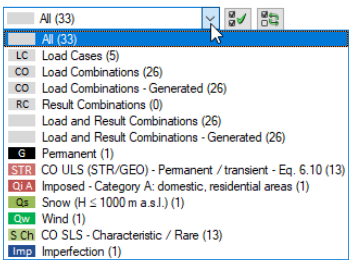
Existing Load Cases / Combinations

This column lists all load cases, load combinations, and result combinations that have been created in RFEM or RSTAB.

To transfer selected entries to the *Selected for Design* list on the right, click the [▷] button. Alternatively, you can double-click the entries. To transfer the entire list to the right, use the [▷▷] button.

As common for Windows applications, selecting several load cases is possible by clicking them one by one while holding down the [Ctrl] key. Thus, you can transfer several load cases all at once.

If a load case's number is marked in red such as LC5 in Figure 2.5, you cannot design it: It indicates a load case without load data, or a load case that contains imperfections. A warning appears if you try to transfer it.



Below the list, several filter options are available. They help you assign the entries sorted by load case, load combination, or action category. The buttons have the following functions:

[Select all button]	Selects all load cases in the list
[Invert selection button]	Inverts selection of load cases

Table 2.1 Buttons in *Ultimate Limit State* tab

Selected for Design

The column on the right lists the load cases as well as load and result combinations that have been selected for the design. To remove selected items from the list, click or double-click the entries. To empty the entire list, click .

You can assign the load cases as well as load and result combinations to the following design situations:

- *Persistent and Transient*
- *Accidental*

This classification controls the γ_M factor that is included in the determination of the R_d resistances for the cross-section designs and stability analyses (see Figure 2.9).

To change a design situation, use the list which you can access by clicking the button at the end of the text box.

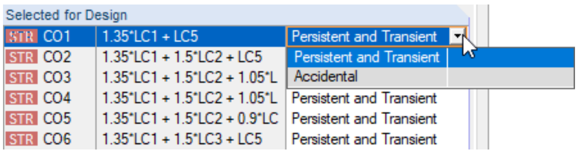


Figure 2.6 Assigning the design situation

For a multiple selection, press the [Ctrl] key and click the corresponding entries. Thus, you can change several entries at once.

The analysis of an enveloping max/min result combination is faster than the analysis of all load cases and load combinations indiscriminately selected for design. However, when analyzing a result combination it is difficult to discern the influence of the included actions.

2.1.2 Serviceability Limit State

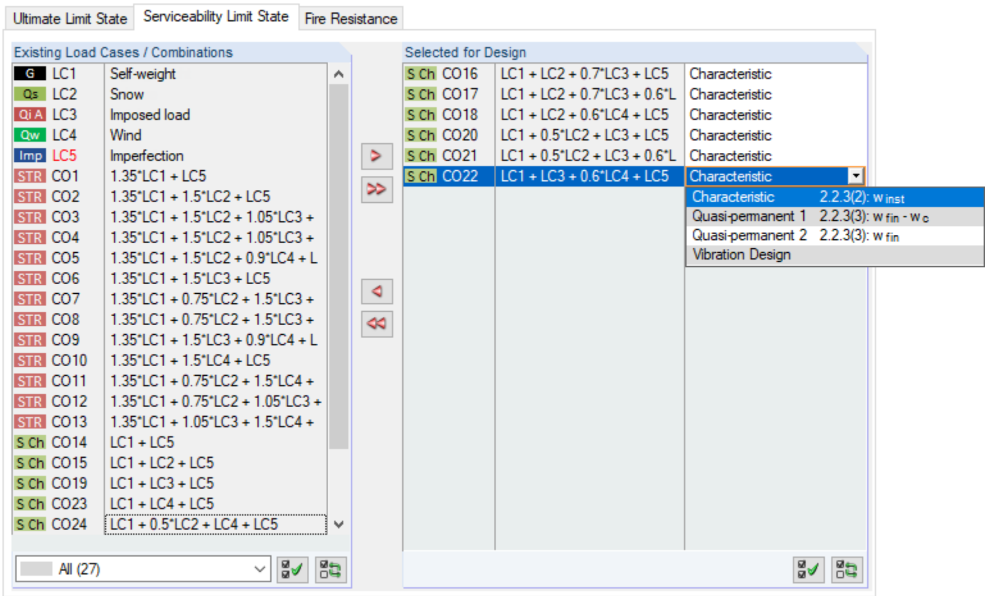


Figure 2.7 Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases / Combinations

This column lists all load cases and combinations that have been created in RFEM or RSTAB.

Selected for Design

You can add or remove load cases as well as load and result combinations as described in Chapter 2.1.1.

It is possible to assign different limit values for deflection to the load cases, load and result combinations. The following design situations for EN 1995-1-1 are available for selection:

- *Characteristic* 2.2.3(2):

  w_{inst} : characteristic combination without creep component

- *Quasi-permanent* 2.2.3(3):

  w_{fin} - w_c : quasi-permanent combination with camber

- *Quasi-permanent* 2.2.3(3):

  w_{fin} : quasi-permanent combination

- *Vibration Design*:

  Verification of natural frequency by means of limit value w_{inst}

To change the design situation, use the list which you can access by clicking the button at the end of the text box (see Figure 2.7).

Standard...
Nat. Annex...

The limit values of the deformations are defined in the National Annex. They can be adjusted for the design situations in the *Standard* or *National Annex Settings* dialog box (see Figure 2.9) that you can open with the [Standard] or [Nat. Annex] button.

In Window *1.9 Serviceability Data*, the reference lengths applying to the deformation analysis are managed (see Chapter 2.10).

2.1.3 Fire Resistance

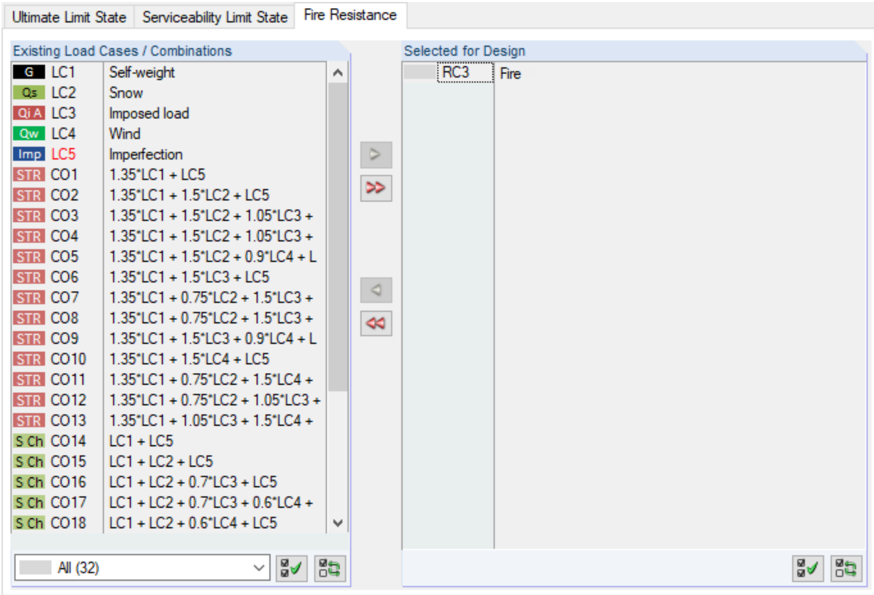


Figure 2.8 Window *1.1 General Data*, tab *Fire Resistance*

Existing Load Cases / Combinations

This column lists all load cases and combinations that have been created in RFEM or RSTAB.

Selected for Design

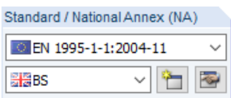
You can add or remove load cases as well as load and result combinations as described in Chapter 2.1.1. Here, you should select the actions that have been determined according to EN 1995-1-2 [5].

Nat. Annex...

The fire resistance design is performed by means of a reduced cross-section. The general specifications for the fire resistance design are managed in the *Fire Factors* tab of the *National Annex Settings* dialog box (see Figure 2.13) as well as in the *Fire Resistance* tab of the *Details* dialog box (see Figure 3.4).

Fire protection designs are not possible for combined cross-sections: Since the neutral axis will be shifted during the cross-section reduction, the stiffness of the cross-section would have to be recalculated in each charring calculation. In addition, when determining the internal forces in RFEM or RSTAB, this change of stiffness would have to be taken into account by recalculating it.

2.1.4 Standard / National Annex



In the list in the upper right corner of the *1.1 General Data* window, you can select the standard or, if EN 1995-1-1 is set, the National Annex whose parameters you want to apply to the design and the limit values of the deformation (see Figure 2.3).

Click the button to check and adjust, if necessary, the preset parameters (see Figure 2.9). Use the button to create a user-defined annex.

Standard...
Nat. Annex...

Moreover, in every input window you can find the [Nat. Annex] button (for EN 1995-1-1) or [Standard] (for DIN 1052 and SIA 265) which also opens the *National Annex Settings* or *Standard* dialog box. This dialog box consists of several tabs.

Material Factors

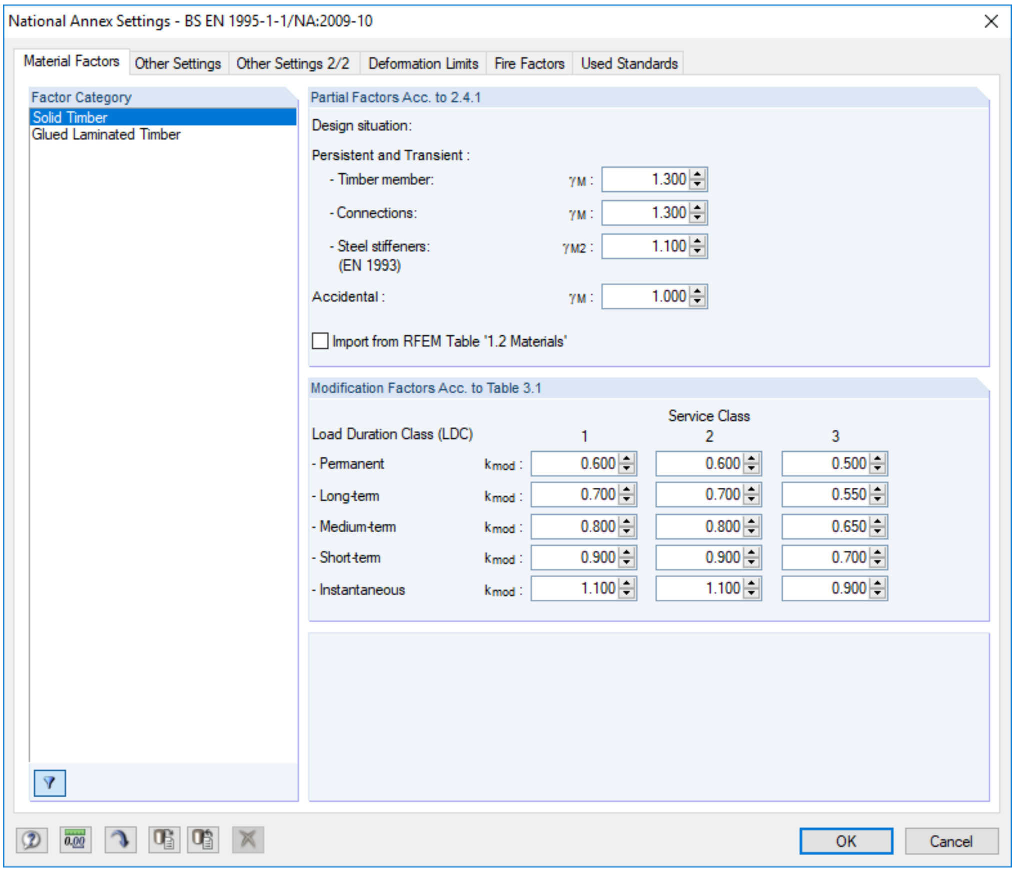


Figure 2.9 Dialog box *National Annex Settings - BS*, tab *Material Factors*

For each *Factor Category* (solid timber, glued laminated timber) the *Partial Factors* γ_M and *Modification Factors* k_{mod} are preset according to the standard for the different design situations as well as load duration and service classes. These values can be adjusted as needed.

Use the [Include Unused Material Categories] button to show all available timber material grades in the *Factor Category* column.

The buttons in the lower left corner of this dialog box have the following functions:

Button	Function
	Resets the program's default settings
	Imports user-defined default settings
	Saves modified settings as default
	Deletes user-defined National Annex

Table 2.2 Buttons in *National Annex Settings* dialog box

Other Settings

The second tab (and also the third for EN 1995-1-1) of this dialog box provides various coefficients that are significant for the verification. They can be adjusted by selecting the *User-defined* option.

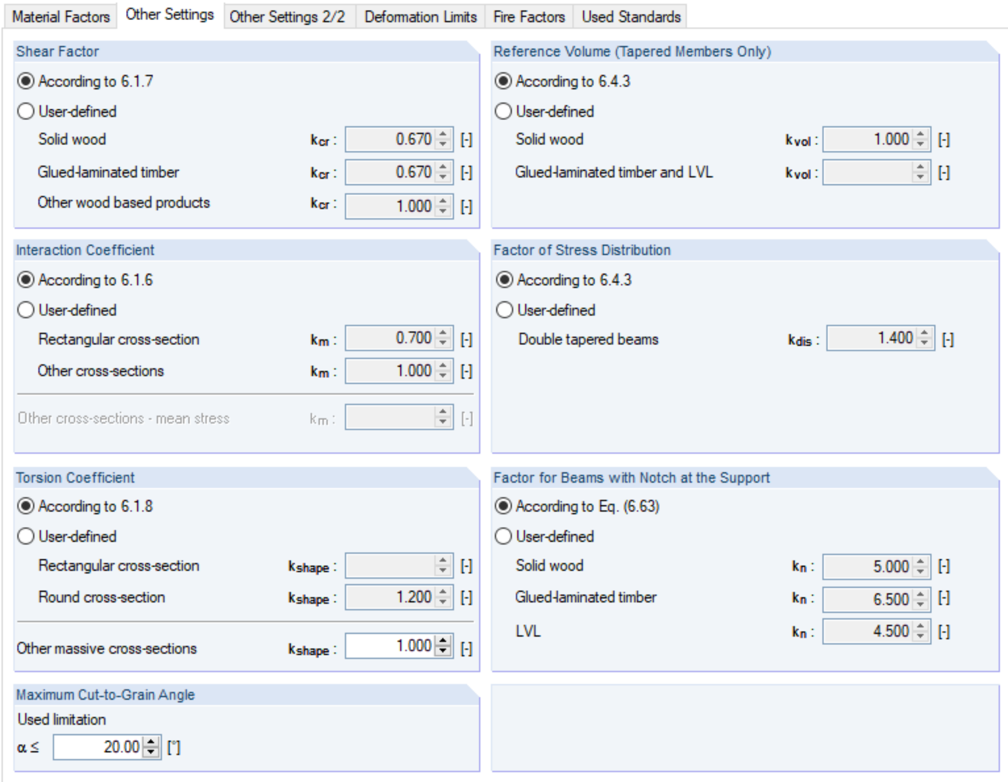


Figure 2.10 Dialog box *National Annex Settings - BS*, tab *Other Settings*

In this tab, it is also possible to adjust the *Maximum Cut-to-Grain Angle* α.

If there are *Beams with Notch at the Support*, the coefficient k_n for sheathing material according to [1], expression (6.63) will affect the design (see Chapter 2.12).

Figure 2.11 Dialog box *National Annex Settings - BS*, tab *Other Settings 2/2*

With the *Other Settings 2/2* tab you can control if an *Increase of Bending, Shear and Tensile Strength* according to [1] will be performed for small cross-sections: For those sections it is assumed, from a statistical point of view, that timber of a superior grade is distributed over the section. The strengths for the tension design (referring to the cross-section width) and the bending stress design (referring to the section height) can be increased by the k_h factors as follows.

- Solid timber with h < 150 mm for bending or b < 150 mm for tension:

$$k_h = \min \begin{cases} \left(\frac{150}{h}\right)^{0.2} \\ 1.3 \end{cases}$$

Equation 2.1

- Glulam timber with h < 600 mm for bending or b < 600 mm for tension:

$$k_h = \min \begin{cases} \left(\frac{600}{h}\right)^{0.1} \\ 1.1 \end{cases}$$

Equation 2.2

RF-/TIMBER Pro recognizes the available material and automatically increases the strengths for activated options.

In accordance with the German annex for [1], it is possible to increase the flexural strength of the lamellas by 20% if they are edgewise subjected to bending.

Deformation Limits

In this dialog tab, you can check and adjust, if necessary, the *Limit Values of Deformation* for the different design situations and support conditions.

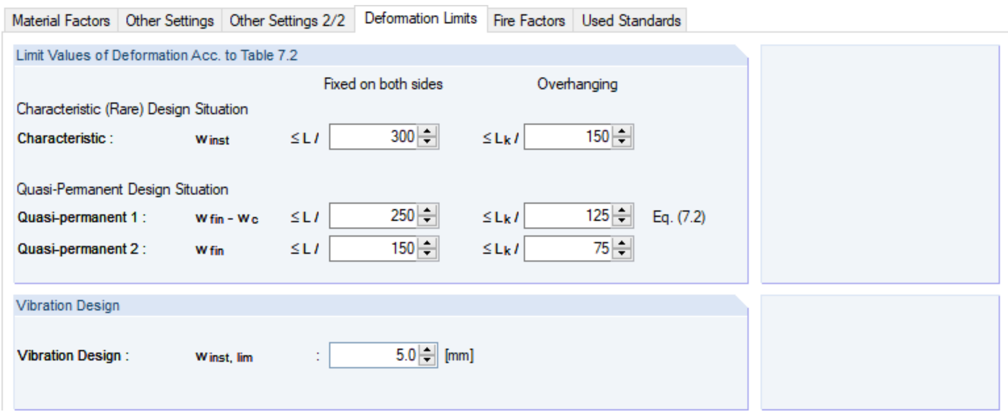


Figure 2.12 Dialog box *National Annex Settings - BS*, tab *Deformation Limits*

The value $w_{inst,lim}$ for the *Vibration Design* is explained by means of an example in the following technical article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000717

Fire Factors

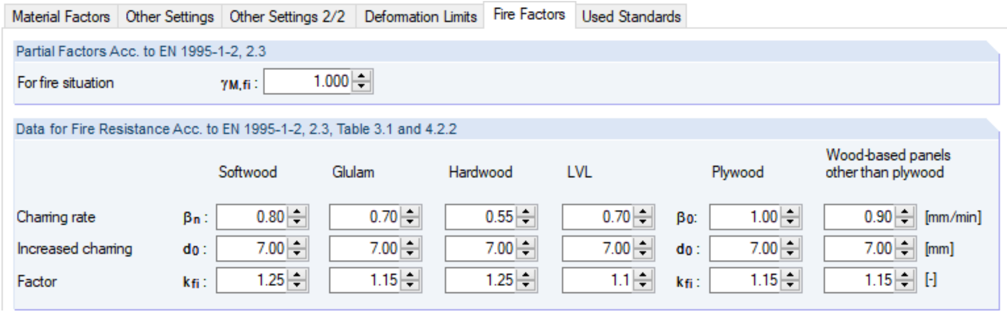


Figure 2.13 Dialog box *National Annex Settings - BS*, tab *Fire Factors*

This dialog tab manages the *Partial Factors* $\gamma_{M,fi}$ for the case of fire according to [5] as well as additional *Data for Fire Resistance* (charring rate β_n, increased charring $d_{0)}$ for different types of timber. They are required for the determination of the ideal residual cross-section.

The Factor k_{fi} is required to determine the 20% fractile value of the strength and stiffness from the 5% fractile value.

Used Standards

The final tab of the *National Annex Settings* dialog box informs you about the standards according to which the designs will be performed.

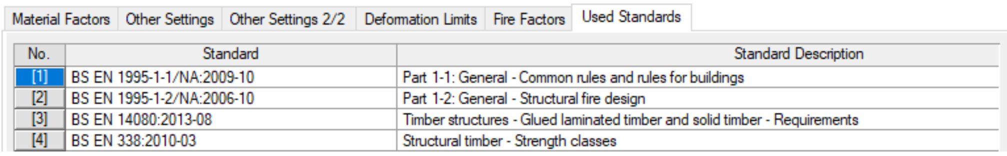


Figure 2.14 Dialog box *National Annex Settings - BS*, tab *Used Standards*

2.2 Materials

This module window is subdivided into two parts. The upper part lists all materials created in RFEM or RSTAB. In the *Material Properties* section, the properties of the current material, i.e. the table row selected in the upper section, are shown.

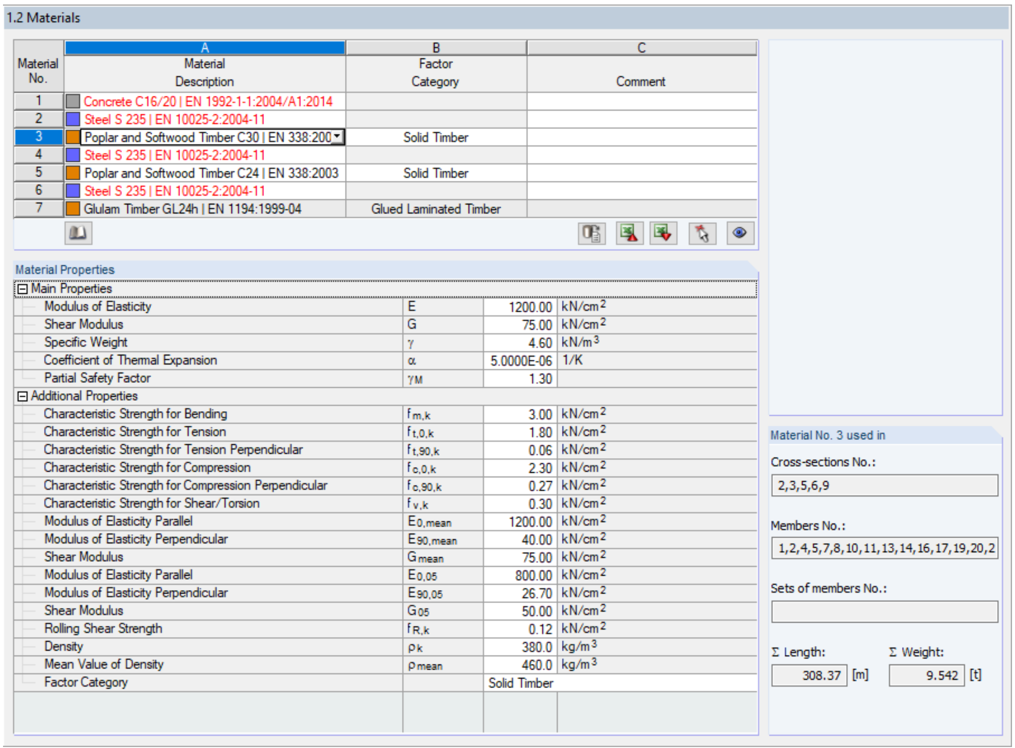


Figure 2.15 Window *1.2 Materials*

Materials that won't be used in the design are grayed out. Materials that are not allowed are highlighted in red. Modified materials are displayed in blue.

Chapter 4.3 of the RFEM manual and Chapter 4.2 of the RSTAB manual describe the material properties that are used for the determination of the internal forces (*Main Properties*). The properties of the materials that are required for the design are also stored in the global material library. These values are preset (*Additional Properties*).

To adjust the units and decimal places of the characteristic values and stresses, select on the module menu **Settings → Units and Decimal Places** (see Chapter 7.3).

Material Description

The materials defined in RFEM or RSTAB are preset but you can modify them anytime: Click the material in column A to activate the field. Then, click the button, or press the function key [F7] to open the material list.

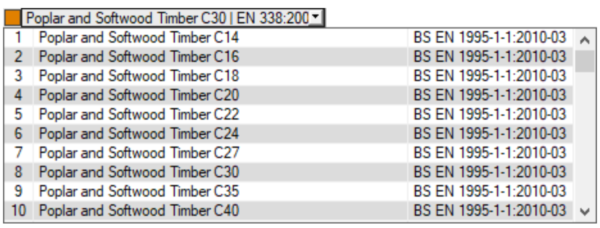


Figure 2.16 List of materials

In accordance with the design concept of the timber standards, the list includes only materials of the *Timber* category.

After the material transfer, the design-relevant *Material Properties* are updated.

If you change the material description manually and the new entry is already listed in the material library, RF-/TIMBER Pro will import the material properties as well.

The material properties are generally not editable in the RF-/TIMBER Pro add-on module.

Material Library

Many materials are stored in the database. To open the material library, select on the module menu

Edit → Material Library

or use the button shown on the left.

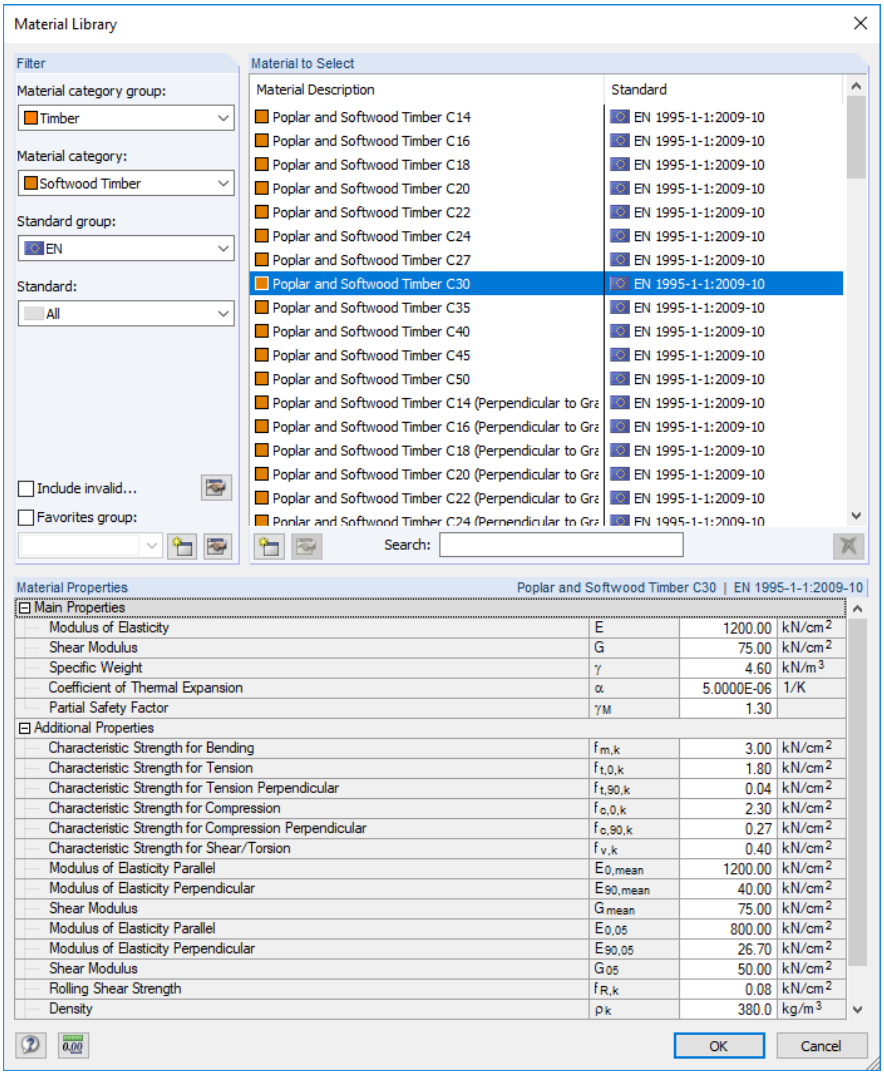


Figure 2.17 Dialog box *Material Library*

The *Timber* material category is preset in the *Filter* section. You can select the desired material grade from the *Material to Select* list; then you can check the properties in the dialog section below.

Click [OK] or use [↵] to transfer the selected material to Window 1.2 of RF-/TIMBER Pro.

Chapter 4.3 of the RFEM manual and Chapter 4.2 of the RSTAB manual describe how to filter, add, or reorganize materials.

Material Properties

In the lower section of Window 1.2, the characteristic strength values for bending $f_{m,k}$, tension parallel $f_{t,0,k}$, tension perpendicular $f_{t,90,k}$, compression parallel $f_{c,0,k}$, compression perpendicular $f_{c,90,k}$ as well as for shear and torsion $f_{v,k}$ are specified.

The design values of the material strengths must be determined, as shown e.g. in [1], Eq. (2.14), with the modification factors k_{mod} and the partial safety factors γ_M.

$$X_d = k_{mod} \cdot \frac{X_k}{\gamma_M}$$

Equation 2.3

Nat. Annex...

The modification and partial safety factors can be adjusted in the *National Annex Settings* dialog box (see Figure 2.9).

2.3 Cross-Sections

This window lists the cross-sections used for the design. In addition, you can specify optimization parameters.

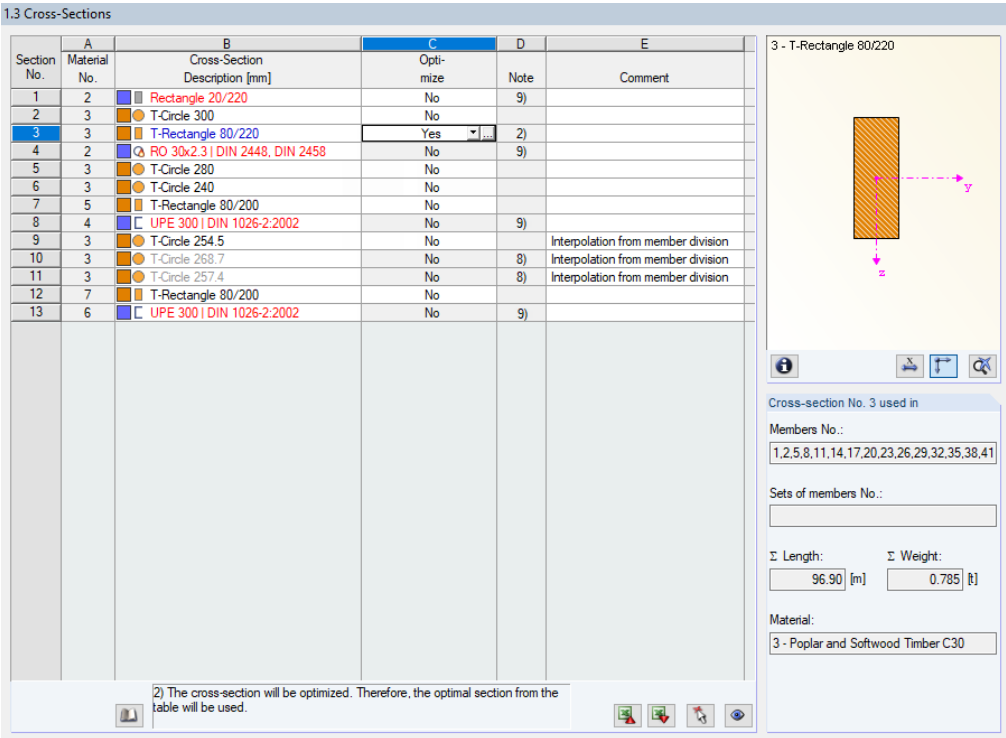


Figure 2.18 Window 1.3 *Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM or RSTAB are preset together with the assigned material numbers. The design is possible for parametric timber and solid sections of the library.

To modify a cross-section, click the entry in column B, setting the field active. Then, open the cross-section table of the current input field by clicking the [Cross-Section Library] button or the button at the end of the text box. You can also use the function key [F7] (see Figure 2.19).

In this dialog box, you can choose a different cross-section or even a different cross-section table. If you want to select a completely different cross-section category, click the [Back to Cross-Section Library] button that opens the general cross-section library.

Chapter 4.13 of the RFEM manual and Chapter 4.3 of the RSTAB manual describe how to select cross-sections from the library.

You can also enter a new cross-section description directly into the input field in column B. If the entry is already listed in the database, RF-/TIMBER Pro will import the cross-section properties. A modified cross-section is highlighted in blue.

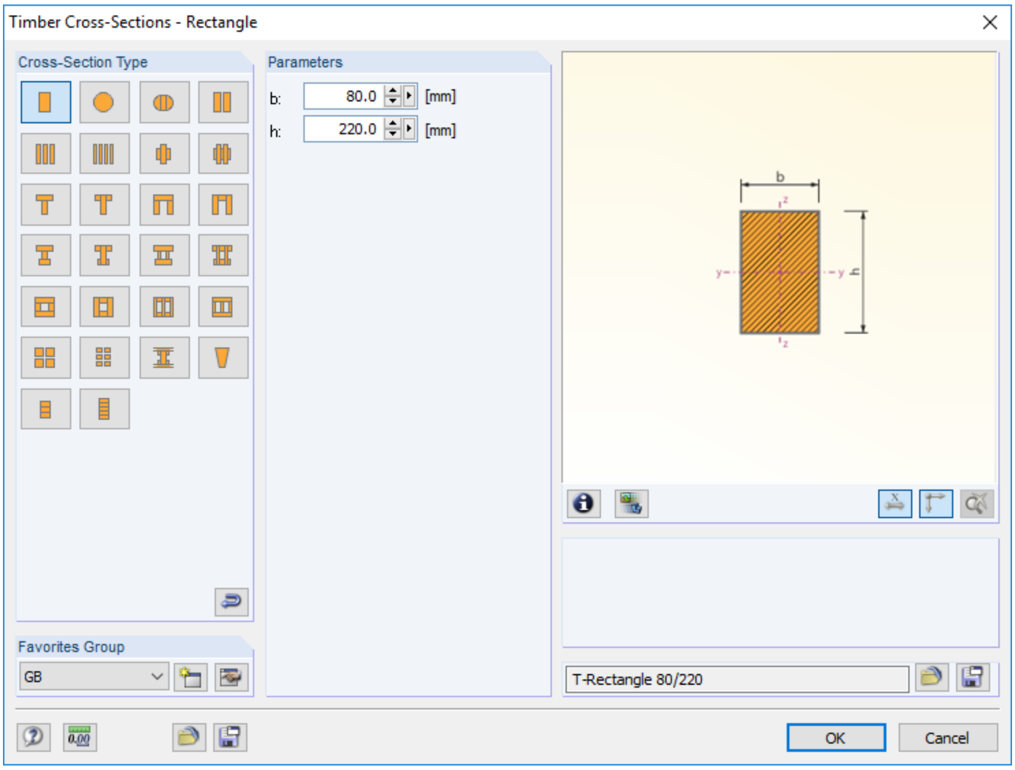
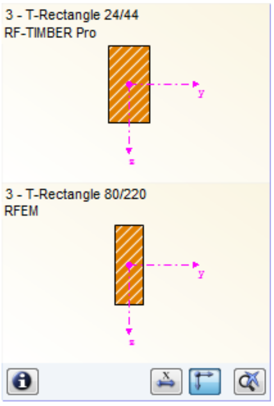


Figure 2.19 Timber sections of cross-section library



If cross-sections set in RF-/TIMBER Pro are different from the ones used in RFEM or RSTAB, both cross-sections are shown in the window graphic to the right. The designs will be performed with the internal forces from RFEM or RSTAB for the cross-section selected in RF-/TIMBER Pro.

If the cross-section is a built-up cross-section, the slip in the joint (*Coefficient of compliance*) due to fasteners can be taken into account.

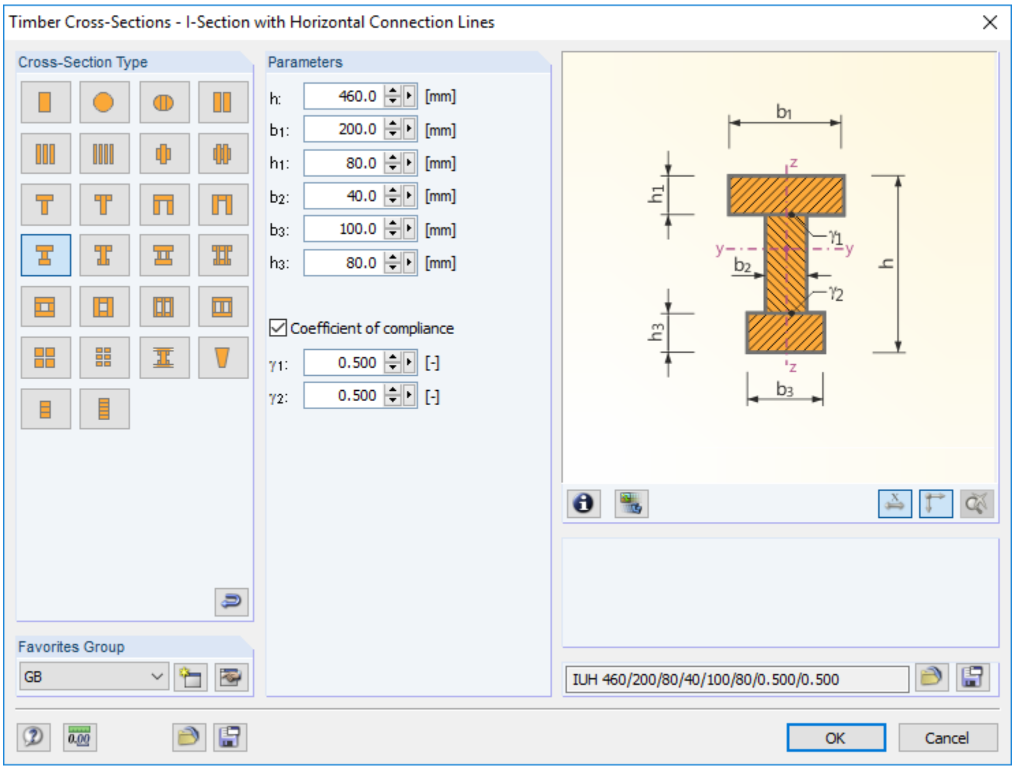
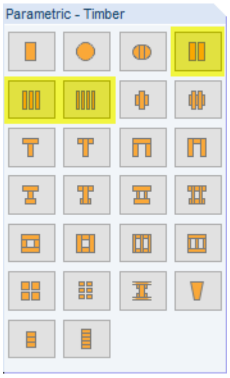


Figure 2.20 Considering the slip of a built-up cross-section

The cross-section properties are computed by the so-called γ-procedure according to [1] Annex B.2. The following is assumed:

- A single-span or a continuous beam with hinged supports is used.
- The cross-section values are constant over the member length (that is, no tapered member).
- The loading is sinusoidal.
- Torsion of the cross-section is excluded.
- Lateral-torsional buckling is not analyzed.



Non built-up cross-sections (collar tie connection)

Further restrictions exist for timber cross-sections that are non built-up sections. These concern the cross-section tables *H-2B*, *H-3B* and *4B*.

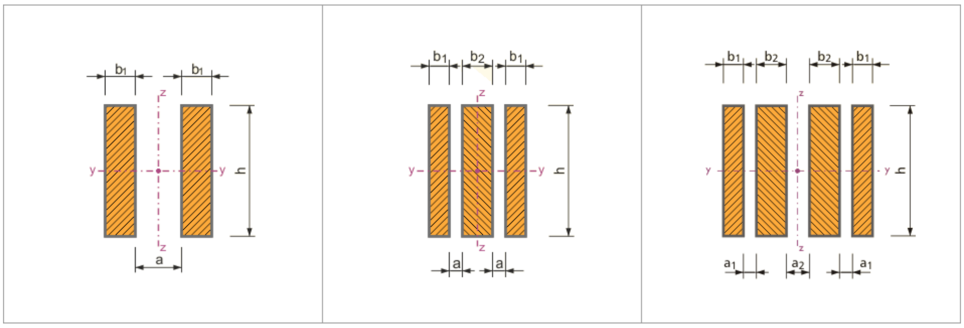
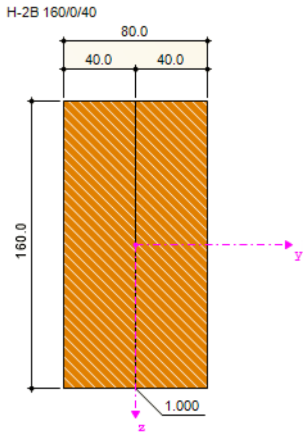
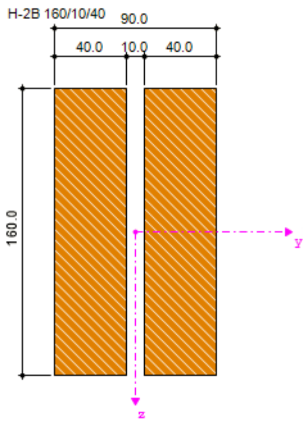


Figure 2.21 Non built-up timber sections – cross-section tables *H-2B*, *H-3B* and *4B*



If the distance between these cross-sections is a=0, the restrictions of the γ-procedure listed above apply. But if the distance is greater than 0, these cross-sections are considered to be non-contiguous sections and the cross-section values are calculated, for example, without the parallel axis theorem. The restrictions according to [1] clause B.1.2 still apply!



For better understanding, the cross-section values of a collar tie connection are compared below: without spacing (cross-section *2B 160/0/40*) and with spacing (cross-section *2B 160/10/40*).

2B 160/0/40	**2B 160/10/40**
$A = 2 \cdot h \cdot b = 2 \cdot 4 \cdot 16 = 128\ cm^2$	$A = 128\ cm^2$
$A_y = 5/6 \cdot A = 106.7\ cm^2$	$A_y = 5/6 \cdot 16 \cdot 4 = 53.3\ cm^2$
$I_{z,i} = 16 \cdot 4^3 / 12 = 85.3\ cm^4$	$I_{z,i} = 16 \cdot 4^3 / 12 = 85.3\ cm^3$
$I_{z,eff} = \Sigma(I_i + \gamma_i \cdot A_i . a_i^2) = 2 \cdot (85.3 + 64 \cdot 2^2) = 682.7\ cm^4$	$I_{z,eff} = 2 \cdot I_{z,i} = 170.7\ cm^4$
$a_i = \dfrac{A_1(h_1+h_2)-0}{2\,\Sigma\gamma_i \cdot A_i} = \dfrac{128 \cdot 8}{2 \cdot (128+128)} = 2$	$a_i = 0$

Table 2.3 Comparison of cross-section values

A factor γ=1 (glued) is assumed for a cross-section without spacing. Calculations with the factor γ=0 would result in the same values like for the cross-section with spacing.

Max. Design Ratio

This table column is displayed only after the calculation. It is intended to be a decision support for optimization: Looking at the design ratios and colored relation scales, you can clearly see which cross-sections are hardly utilized and thus oversized, or extremely stressed and thus undersized.

Optimize

Details...

Rectangular and circular cross-sections can undergo an optimization process: For the internal forces from RFEM or RSTAB, the program searches the cross-section that comes as close as possible to a user-defined maximum ratio. This ratio can be defined in the *Other* tab of the *Details* dialog box (see Figure 3.5).

To optimize a cross section, open the drop-down list in column D or E and select the desired entry: *Yes* or *From favorites 'Description'*. Recommendations for optimizing cross-sections can be found in Chapter 7.2.

Note

This column shows remarks in the form of footnotes. They are explained below the cross-section list.

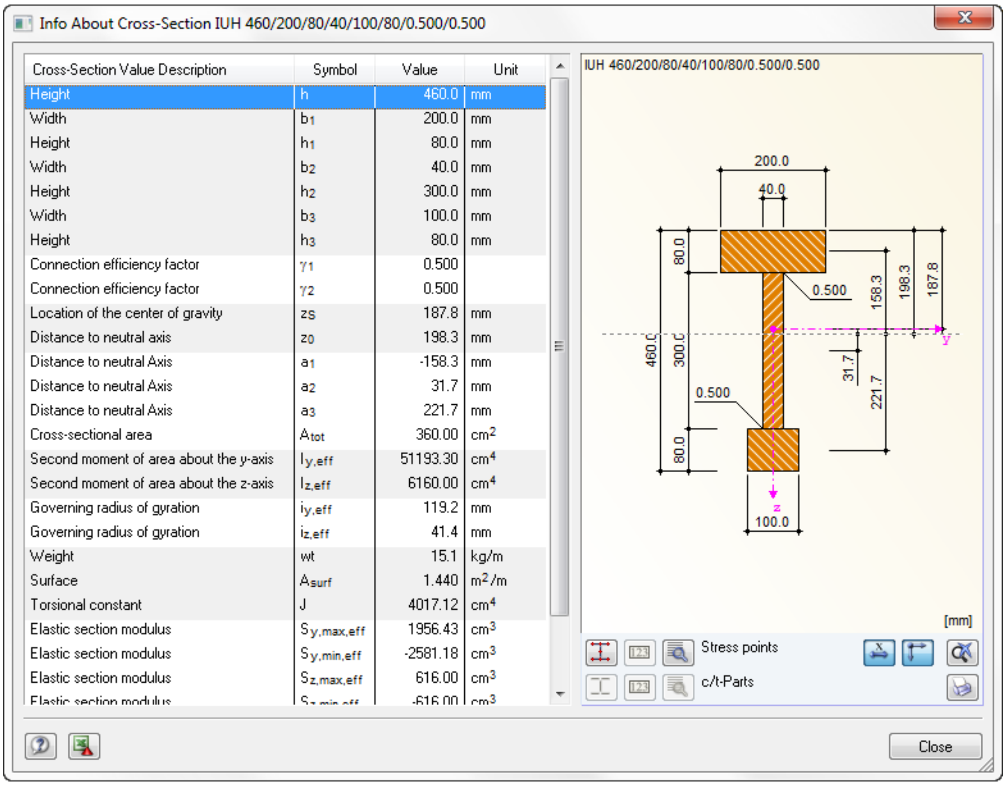
Member with tapered cross-section

For tapered members with different cross-sections at the member start and end, both cross-section numbers are shown in two rows, in accordance with the definition in RFEM or RSTAB.

RF-/TIMBER Pro also designs tapered members if the start and end cross-section have the same cross-section type. This requires further specifications in module Window 1.7 (see Chapter 2.7).

Info About Cross-Section

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



Cross-Section Value Description	Symbol	Value	Unit
Height	h	460.0	mm
Width	b_1	200.0	mm
Height	h_1	80.0	mm
Width	b_2	40.0	mm
Height	h_2	300.0	mm
Width	b_3	100.0	mm
Height	h_3	80.0	mm
Connection efficiency factor	γ_1	0.500	
Connection efficiency factor	γ_2	0.500	
Location of the center of gravity	z_S	187.8	mm
Distance to neutral axis	z_0	198.3	mm
Distance to neutral Axis	a_1	-158.3	mm
Distance to neutral Axis	a_2	31.7	mm
Distance to neutral Axis	a_3	221.7	mm
Cross-sectional area	A_{tot}	360.00	cm^2
Second moment of area about the y-axis	$I_{y,eff}$	51193.30	cm^4
Second moment of area about the z-axis	$I_{z,eff}$	6160.00	cm^4
Governing radius of gyration	$i_{y,eff}$	119.2	mm
Governing radius of gyration	$i_{z,eff}$	41.4	mm
Weight	wt	15.1	kg/m
Surface	A_{surf}	1.440	m^2/m
Torsional constant	J	4017.12	cm^4
Elastic section modulus	$S_{y,max,eff}$	1956.43	cm^3
Elastic section modulus	$S_{y,min,eff}$	-2581.18	cm^3
Elastic section modulus	$S_{z,max,eff}$	616.00	cm^3
Elastic section modulus	$S_{z,min,eff}$	-616.00	cm^3

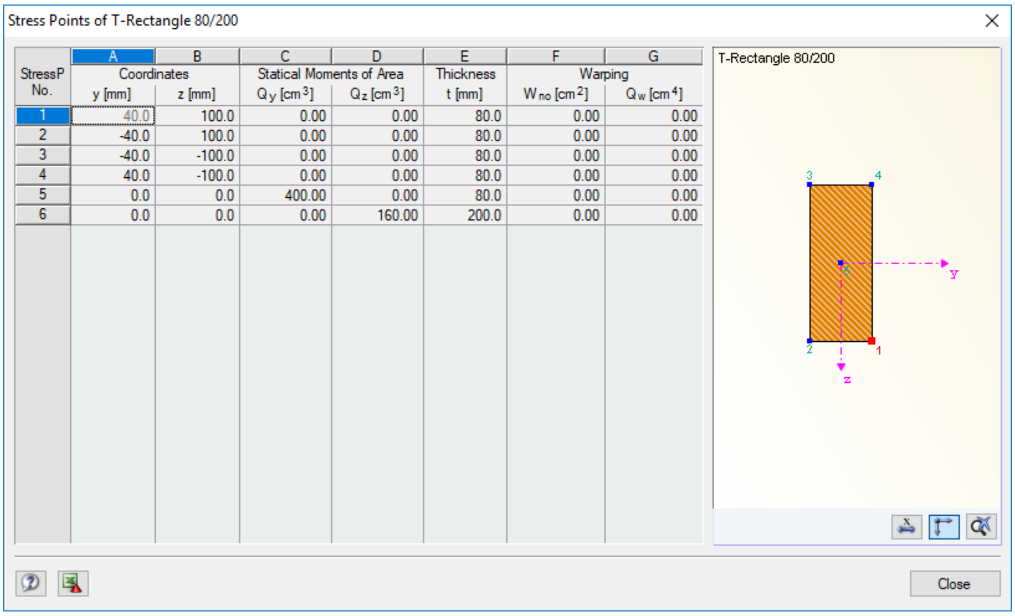
Figure 2.22 Dialog box *Info About Cross-Section*

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

Button	Function
	Displays or hides stress points
	Displays or hides numbers of stress points
	Shows details of stress points (see Figure 2.23)
	Displays or hides dimensions of cross-section
	Displays or hides principal axes of cross-section
	Resets full view of cross-section
	Prints values and graphic of cross-section

Table 2.4 Buttons of cross-section graphic

Use the [Details] button to call up specific information about the stress points (centroid distances, statical moments of area, etc.).



StressP No.	A Coordinates y [mm]	B z [mm]	C Statical Moments of Area Q_y [cm³]	D Q_z [cm³]	E Thickness t [mm]	F Warping W_{no} [cm²]	G Q_w [cm⁴]
1	40.0	100.0	0.00	0.00	80.0	0.00	0.00
2	-40.0	100.0	0.00	0.00	80.0	0.00	0.00
3	-40.0	-100.0	0.00	0.00	80.0	0.00	0.00
4	40.0	-100.0	0.00	0.00	80.0	0.00	0.00
5	0.0	0.0	400.00	0.00	80.0	0.00	0.00
6	0.0	0.0	0.00	160.00	200.0	0.00	0.00

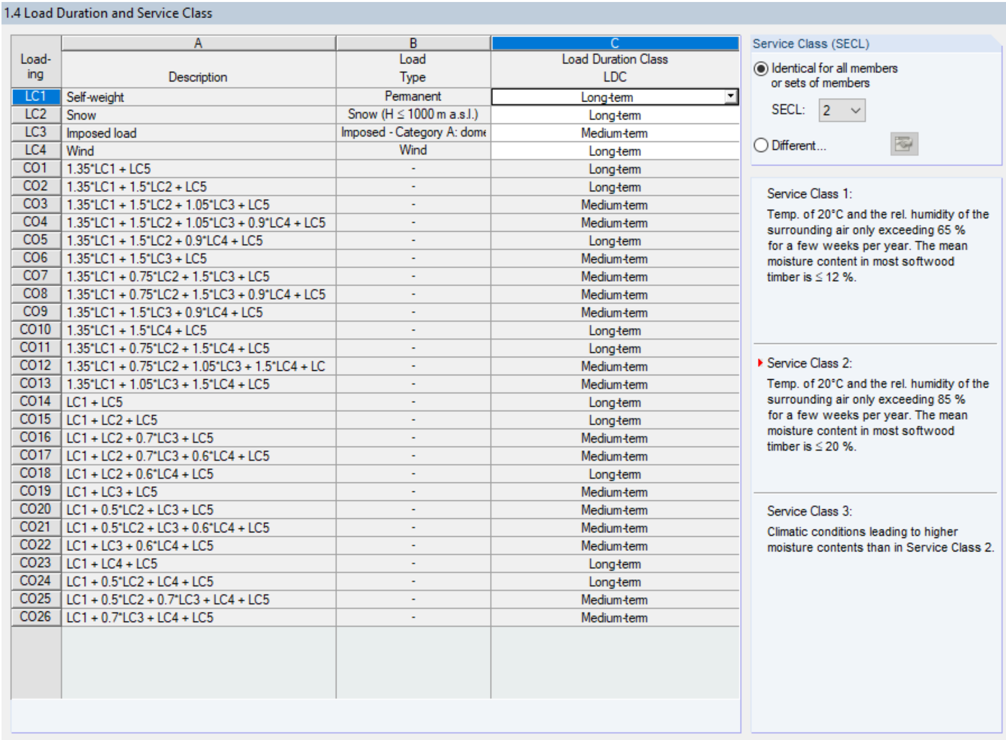
Figure 2.23 Dialog box *Stress Points*

Information about stress points with regard to the determination of shear stresses can be found in Chapter 5.2.

2.4

Load Duration and Service Class

In Window 1.4, you can define the load duration and the service classes of members and sets of members in order to determine the climatic conditions for the designs.



1.4 Load Duration and Service Class

Load-ing	A Description	B Load Type	C Load Duration Class LDC
LC1	Self-weight	Permanent	Long-term
LC2	Snow	Snow (H ≤ 1000 m a.s.l.)	Long-term
LC3	Imposed load	Imposed - Category A: dome	Medium-term
LC4	Wind	Wind	Long-term
CO1	1.35*LC1 + LC5	-	Long-term
CO2	1.35*LC1 + 1.5*LC2 + LC5	-	Long-term
CO3	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + LC5	-	Medium-term
CO4	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + 0.9*LC4 + LC5	-	Medium-term
CO5	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + LC5	-	Long-term
CO6	1.35*LC1 + 1.5*LC3 + LC5	-	Medium-term
CO7	1.35*LC1 + 0.75*LC2 + 1.5*LC3 + LC5	-	Medium-term
CO8	1.35*LC1 + 0.75*LC2 + 1.5*LC3 + 0.9*LC4 + LC5	-	Medium-term
CO9	1.35*LC1 + 1.5*LC3 + 0.9*LC4 + LC5	-	Medium-term
CO10	1.35*LC1 + 1.5*LC4 + LC5	-	Long-term
CO11	1.35*LC1 + 0.75*LC2 + 1.5*LC4 + LC5	-	Long-term
CO12	1.35*LC1 + 0.75*LC2 + 1.05*LC3 + 1.5*LC4 + LC	-	Medium-term
CO13	1.35*LC1 + 1.05*LC3 + 1.5*LC4 + LC5	-	Medium-term
CO14	LC1 + LC5	-	Long-term
CO15	LC1 + LC2 + LC5	-	Long-term
CO16	LC1 + LC2 + 0.7*LC3 + LC5	-	Medium-term
CO17	LC1 + LC2 + 0.7*LC3 + 0.6*LC4 + LC5	-	Medium-term
CO18	LC1 + LC2 + 0.6*LC4 + LC5	-	Long-term
CO19	LC1 + LC3 + LC5	-	Medium-term
CO20	LC1 + 0.5*LC2 + LC3 + LC5	-	Medium-term
CO21	LC1 + 0.5*LC2 + LC3 + 0.6*LC4 + LC5	-	Medium-term
CO22	LC1 + LC3 + 0.6*LC4 + LC5	-	Medium-term
CO23	LC1 + LC4 + LC5	-	Long-term
CO24	LC1 + 0.5*LC2 + LC4 + LC5	-	Long-term
CO25	LC1 + 0.5*LC2 + 0.7*LC3 + LC4 + LC5	-	Medium-term
CO26	LC1 + 0.7*LC3 + LC4 + LC5	-	Medium-term

Service Class (SECL)
Identical for all members or sets of members
SECL: 2
Different...

Service Class 1:
Temp. of 20°C and the rel. humidity of the surrounding air only exceeding 65 % for a few weeks per year. The mean moisture content in most softwood timber is ≤ 12 %.

Service Class 2:
Temp. of 20°C and the rel. humidity of the surrounding air only exceeding 85 % for a few weeks per year. The mean moisture content in most softwood timber is ≤ 20 %.

Service Class 3:
Climatic conditions leading to higher moisture contents than in Service Class 2.

Figure 2.24 Module Window *1.4 Load Duration and Service Class*

Loading

The table column lists all actions that have been selected for design in the *1.1 General Data* window. In case of combinations, included load cases are also displayed.

Description

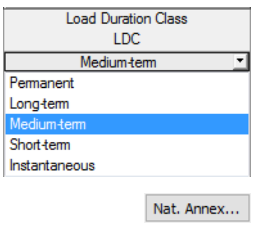
The load case descriptions make the classification easier.

Load Type

This table column shows the action types of the load cases as they were defined in RFEM or RSTAB during their creation. They form the basis for the settings in the next table column.

Load Duration Class LDC

The designs require the assignment of loads and their superpositions to particular load duration classes. Rules for the classification of actions can be found, for example, in [2] Table 4 or [1] Table 2.1.



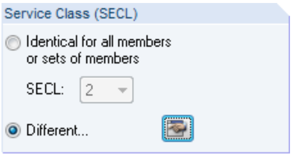
For load cases and result combinations you can change the load duration with the list shown on the left: Click into the cell of column C to activate the field. The ▾ button will be enabled. In case of load and Or-result combinations, RF-/TIMBER Pro carries out the classification automatically taking into account the respective governing action or the contained load cases.

The class of the load duration is required for the determination of the modification factor k_{mod} which affects the strength properties of the material (see [1] Table 3.1). The k_{mod} factors can be checked and adjusted, if necessary, in the *National Annex Settings* dialog box (see Figure 2.9).

If an automatic combination of actions was set in RFEM or RSTAB, the load duration classes are automatically taken into account according to the specifications in RFEM or RSTAB. Thus, a redefinition in RF-/TIMBER Pro is not necessary. However, you can adjust the classification of load cases in this table.

Service Class (SECL)

The classification into service classes makes it possible to assign strength parameters and to calculate deformations by taking into account environmental conditions. The service classes are specified, for example, in [1] clause 2.3.1.3.



By default, all members and sets of members are assigned to the same service class. To classify objects into different service classes, activate the *Different* option. Use the button to open the following dialog box.

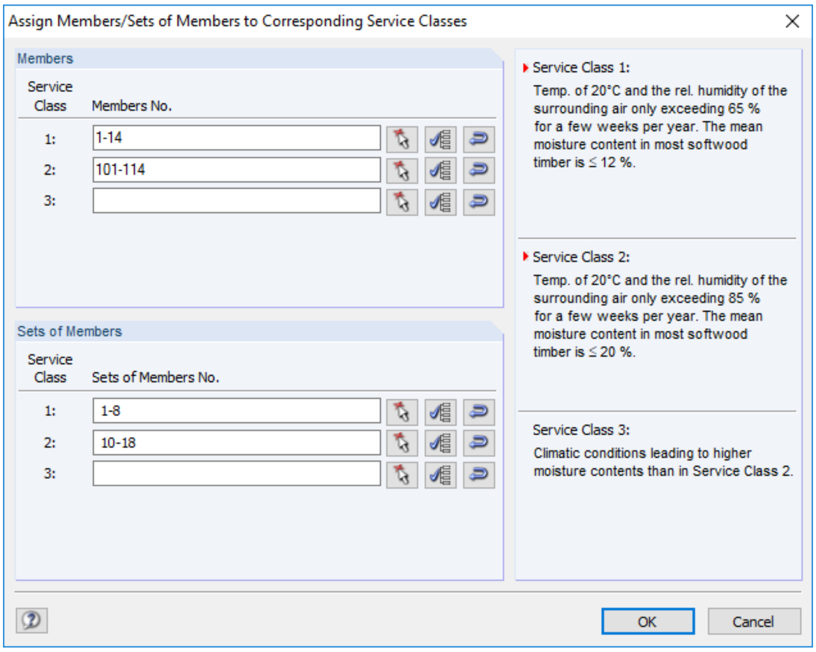


Figure 2.25 Dialog box *Assign Members/Sets of Members to Corresponding Service Classes*

Here, you can individually classify *Members* and *Sets of Members* into service classes. The buttons next to the text boxes facilitate the selection. They have the following functions:

Button	Function
	Allows for graphical selection of objects in RFEM/RSTAB work window
	Assigns all members/sets of members to respective service class
	Assigns all members/sets of members not yet assigned to respective service class

Table 2.5 Buttons in dialog box *Assign Members/Sets of Members to Corresponding Service Classes*

2.5 Effective Lengths - Members

Details...

The layout of this module window depends on whether the stability analysis is carried out according to the equivalent member method or a second-order analysis. The method is to be defined in the *Stability* tab of the *Details* dialog box (see Figure 3.2) The following refers to the **equivalent member method** for which the parameters of buckling and lateral-torsional buckling must be defined.

If the stability analysis is deactivated in the *Stability* tab of the *Details* dialog box, Window 1.5 is not shown.

The window is subdivided into two parts. The table in the upper part shows compressed information on the buckling length factors and equivalent member lengths for buckling and lateral-torsional buckling of all members that are to be designed. The effective lengths defined in RFEM or RSTAB are preset. In the *Settings* section, you can see additional information on the member whose table row is selected in the upper part.

With the button you can select a member graphically to activate its row in the table.

Changing entries is possible in the table as well as the *Settings* tree.

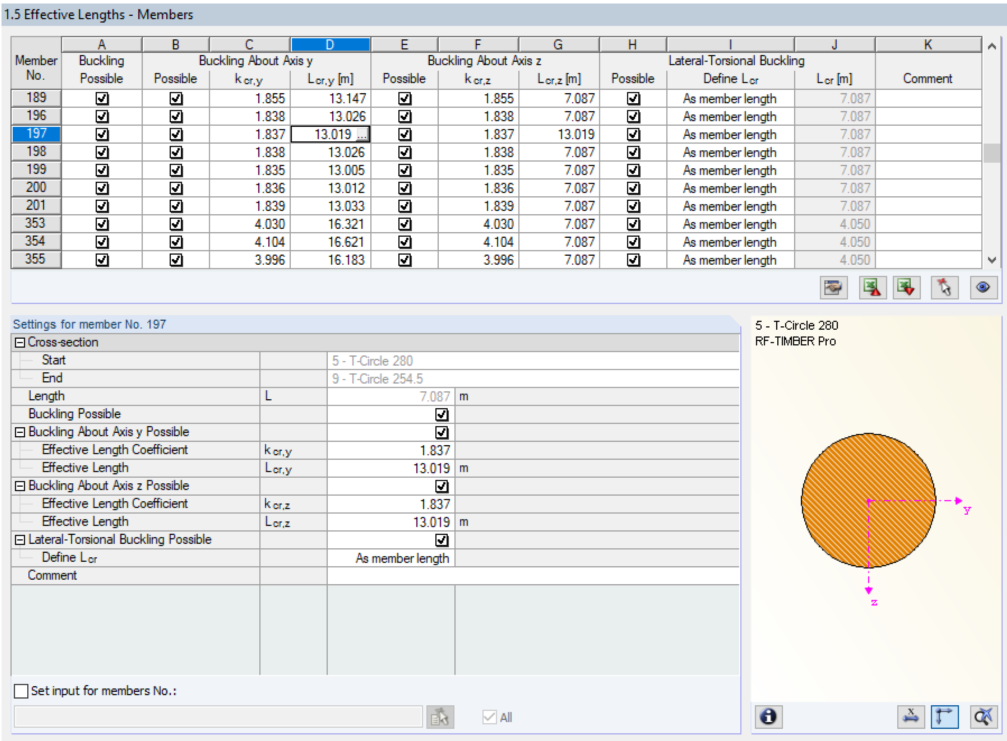


Figure 2.26 Window 1.5 *Effective Lengths - Members*

In both window parts, you can enter the effective lengths manually. You can also define them graphically in the work window by using the [...] button. It becomes active when the cursor is placed in the text box (see Figure 2.26).

The *Settings* tree includes the following parameters:

- *Cross-section*
- *Length* of member
- *Buckling Possible* for member (corresponds to columns B, E, and H)
- *Buckling About Axis y* (corresponds to columns C and D)
- *Buckling About Axis z* (corresponds to columns F and G)
- *Lateral-Torsional Buckling* (corresponds to columns I to K)

In the *Settings*, you can define for the member selected above whether a buckling or a lateral-torsional buckling analysis is generally to be carried out. In addition, you can adjust the effective length coefficient for the respective directions. When changing this factor, the equivalent member length will be adjusted automatically, and vice versa.

It is also possible to define the effective length of a member in a dialog box that you open with the [Select effective length factor] button. You can find the button below the table.

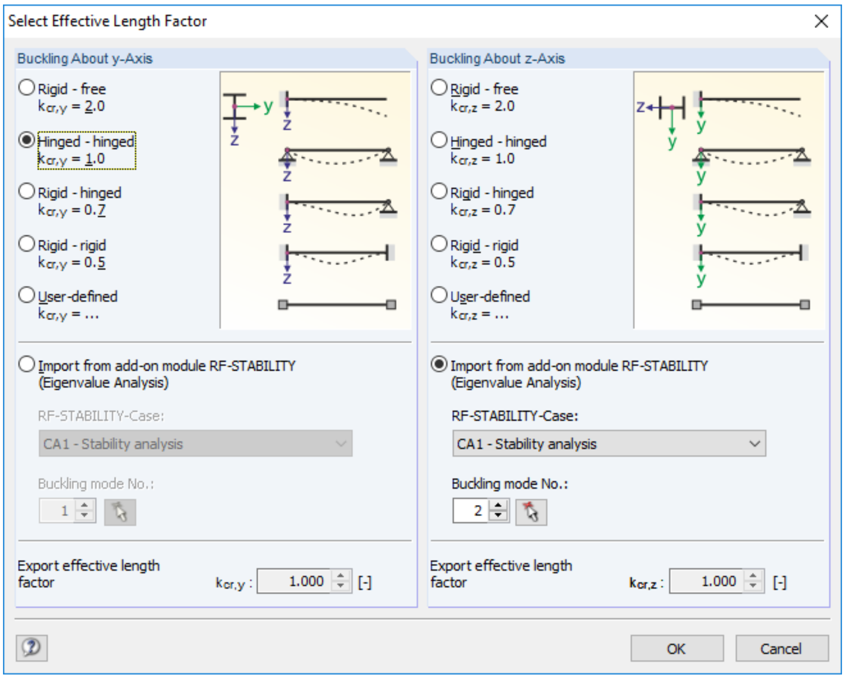


Figure 2.27 Dialog box *Select Effective Length Factor*

For each direction, you can select one of the four Euler buckling modes. You can also set a *User-defined* effective length factor. If an eigenvalue analysis has been carried out by the RF-STABILITY or RSBUCK add-on module, it is also possible to define a *Buckling mode* for the determination of the factor.

Buckling Possible

The stability analyses for flexural and lateral-torsional buckling require the ability to absorb compressive forces. Therefore, members for which such an absorption is not possible due to the member type (for example, tension members, elastic foundations, rigid connections) are excluded from the outset. The rows are grayed out in the table, and a corresponding note is shown in the *Comment* column.

Curved members (only RF-TIMBER Pro) are excluded as well from the stability analysis: Analyses performed according to the equivalent member method require for curved members a definition of the buckling length in the member third-points. In addition, stability analyses of curved members, for example according to [1] clause 6.3.3, are valid only for single-span beams. Statically indeterminate systems or models with several supports require further analyses.

The *Buckling Possible* check boxes in table row A and in the *Settings* tree offer a control option for the stability analyses: They determine if these analyses are performed or omitted for the member.

Buckling About Axis y / Buckling about Axis z

With the check boxes in the *Possible* columns, you decide if a member has the risk of buckling about the axis y and/or z. These axes represent the local member axes, with axis y being the "major" and axis z the "minor" member axis. The effective length factors $k_{cr,y}$ and $k_{cr,z}$ for buckling about the major or minor axis can be selected freely.

The position of the member axes can be checked in the cross-section graphic of the *1.3 Cross-Sections* window (see Figure 2.18). With the [Jump to graphic] button you can also access the RFEM or RSTAB work window where you can display the local member axes by using the member's shortcut menu or the *Display* navigator.

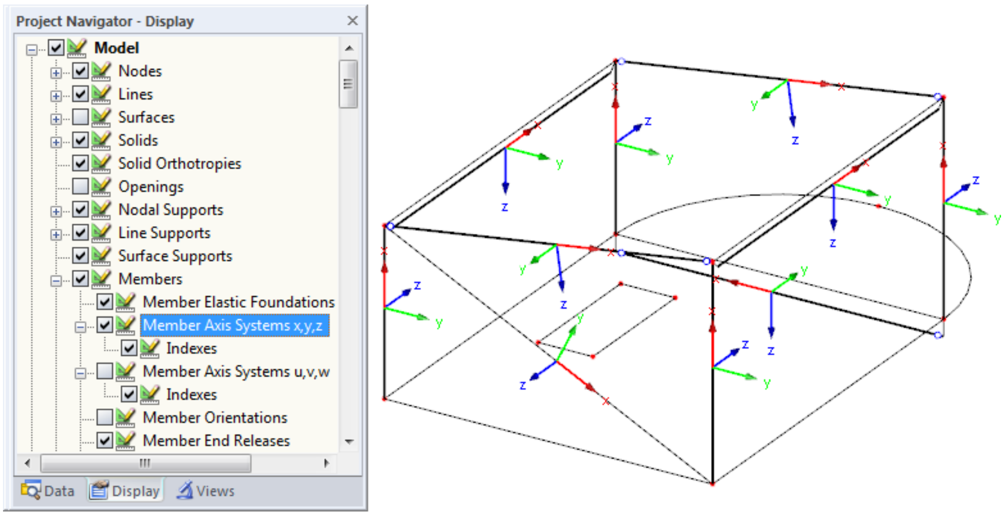


Figure 2.28 Activating the member axis systems in *Display* navigator of RFEM

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

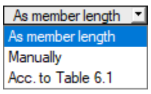
To define the effective lengths graphically in the work window, use the button. This button becomes available when the cursor is placed in a L_{cr} input field (see Figure 2.26).

When you specify the effective length factor k_{cr}, the program determines the effective length L_{cr} by multiplying the member length *L* by this factor. The input fields k_{cr} and L_{cr} are interactive.

Lateral-Torsional Buckling Possible

Column H shows which members are included in the analysis of lateral-torsional buckling.

Define L_{cr}



The *member length* is set by default as equivalent member length for the lateral-torsional buckling analysis. More options become accessible when you click into a field in column H.

The lateral-torsional buckling length can be defined *Manually* by entering L_{cr} in column J. When clicking into this table column, the function becomes available, and you can graphically determine the distance between the lateral supports. A manual adjustment may be useful for any structural component consisting of several members between the supports.

If you select the *Acc. to Table 6.1* option (only available for DIN), a dialog box opens where you can define the lateral-torsional buckling parameters according to [1] Table 6.1 (see Figure 2.29). In the DIN standard, the effective length is described as a quotient of the span length which depends on the type of bending member and the type of loading.

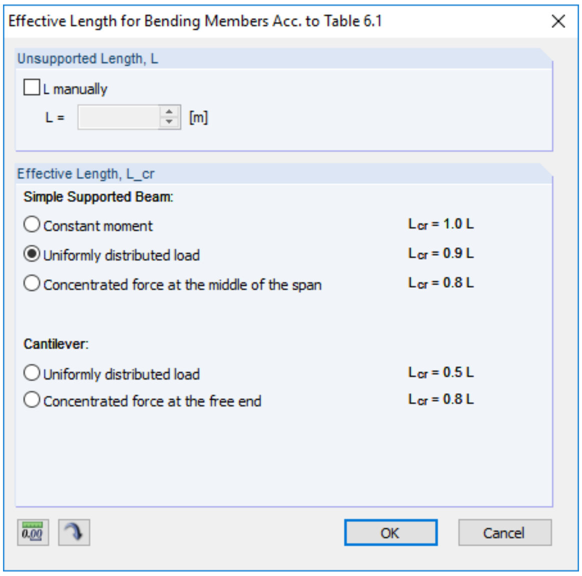


Figure 2.29 Dialog box *Effective Length for Bending Members Acc. to Table 6.1*

Comment

In the final column, you can enter user-defined notes to describe, for example, the equivalent member lengths.

Set input for members No.

Below the *Settings* table, you can find the check box *Set input for members No.* If you tick it, the **subsequently** made settings will apply to *All* members or to selected members (enter the member numbers manually or select them graphically with). This option may help you when assigning the same boundary conditions to several members (see Dlubal article https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000726).

Settings which have already been defined cannot be changed subsequently with this function.

2.6 Effective Lengths - Sets of Members

Details...

This window appears only if at least one set of members is set for design in the *1.1 General Data* window and the stability analysis is activated in the *Stability* tab of the *Details* dialog box (see Figure 3.2).

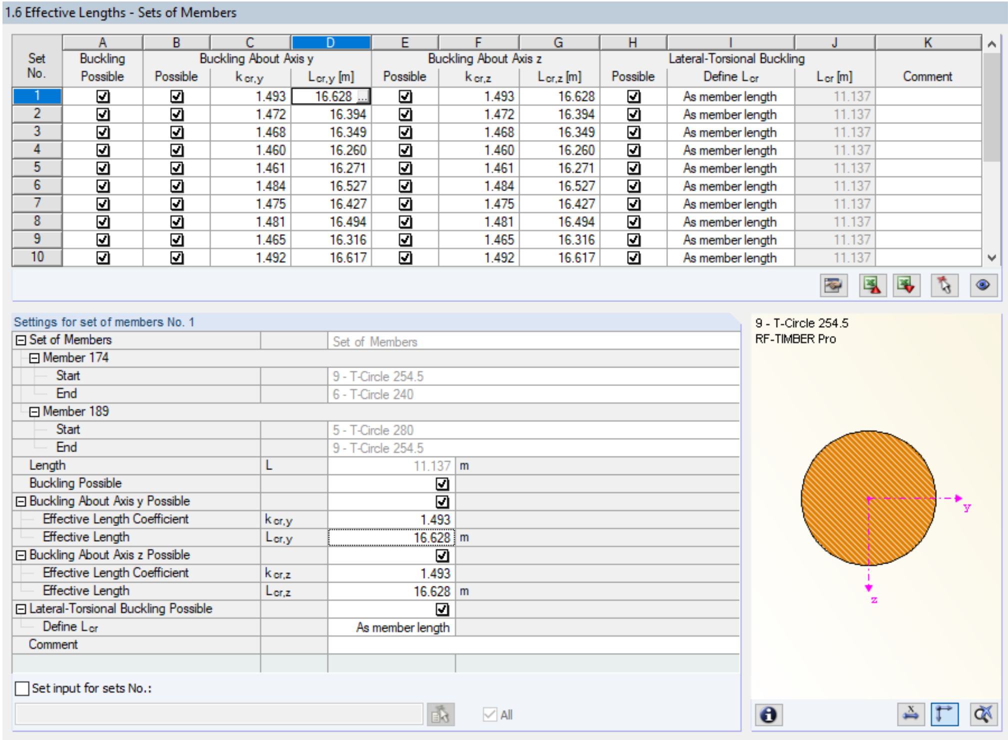


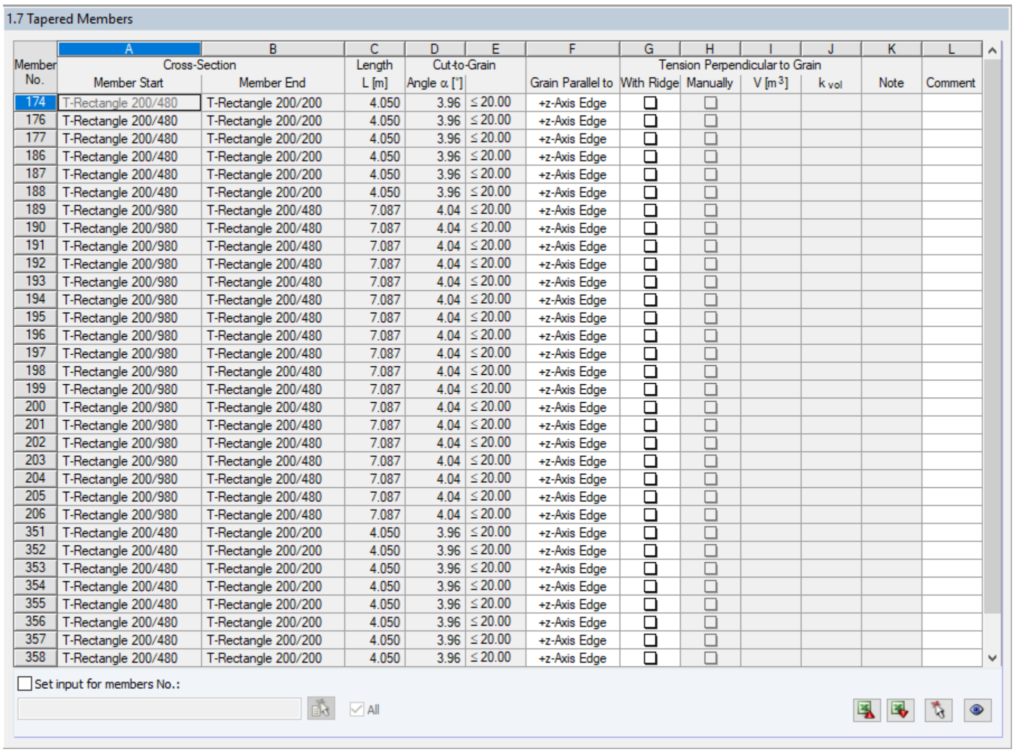
Figure 2.30 Window *1.6 Effective Lengths - Sets of Members*

The concept of this window is similar to the previous Window *1.5 Effective Lengths - Members*. Here, you can enter the effective lengths for buckling about both principal axes of the set of members, as described in Chapter 2.5. They define the boundary conditions of the set of members that is handled in its entirety as an equivalent member.

Please note that curved sets of members are excluded from the stability analysis: Analyses performed according to the equivalent member method require for curved beams a definition of the buckling length in the third-points. In addition, stability analyses of curved beams, for example according to [1] clause 6.3.3, are valid only for single-span beams. Statically indeterminate systems or models with several supports require further analyses.

2.7 Tapered Members

This window is displayed if at least one member with different cross-sections at both member ends is selected for design in the *1.1 General Data* window. The window manages criteria like the cut-to-grain angle of the variable cross-sections.



1.7 Tapered Members

Member No.	A Cross-Section Member Start	B Member End	C Length L [m]	D Cut-to-Grain Angle α [°]	E	F Grain Parallel to	G Tension Perpendicular to Grain With Ridge	H Manually	I V [m³]	J k vol	K Note	L Comment
174	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
176	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
177	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
186	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
187	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
188	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
189	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
190	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
191	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
192	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
193	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
194	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
195	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
196	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
197	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
198	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
199	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
200	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
201	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
202	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
203	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
204	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
205	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
206	T-Rectangle 200/980	T-Rectangle 200/480	7.087	4.04	≤ 20.00	+z-Axis Edge	☐	☐				
351	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
352	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
353	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
354	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
355	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
356	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
357	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				
358	T-Rectangle 200/480	T-Rectangle 200/200	4.050	3.96	≤ 20.00	+z-Axis Edge	☐	☐				

Set input for members No.:

All

Figure 2.31 Window *1.7 Tapered Members*

Cross-Section

The first two table columns list the cross-sections that are defined at the *Member Start* and *Member End*.

Length

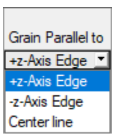
The length of the tapered member is displayed for checking reasons.

Cut-to-Grain Angle α

RF-/TIMBER Pro determines the cut-to-grain angle from the geometric conditions. The equations used in the program are only valid for cutting angles of α ≤ 24° (for EN 1995-1-1 [1] and SIA 265 [3]) or α ≤ 10° (for DIN 1052 [2]).

The limit values given in column E can be checked and, if necessary, adjusted in the *National Annex Settings* dialog box (see Figure 2.9).

Grain Parallel to



In table column F, you can specify the member edge to which the timber's grain direction is running parallel. The "top" or "bottom" edge is clearly defined by the orientation of the local member axis z (see Figure 2.28). Alternatively, it is possible to align the grain with the *Center line*.

In most cases, the grain runs parallel to the edge that is located on the member side in the direction of the +z-axis ("bottom"). This means that the beam is cut at the top side.

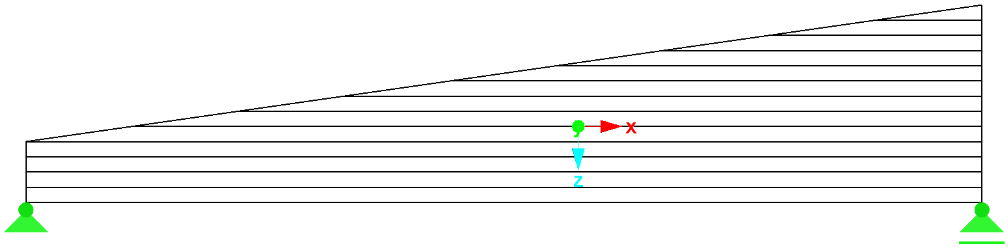


Figure 2.32 Grain parallel to edge in direction of +z-axis

If the grain runs parallel to the —z-axis ("top"), the tapered beam is cut at the bottom side. This case is an exception because cutting a grain in the bending tension area is avoided.

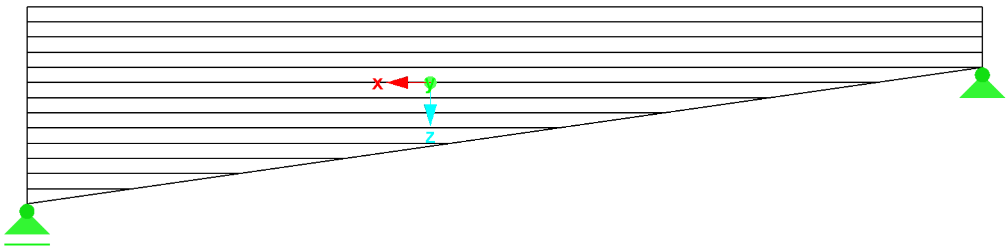


Figure 2.33 Grain parallel to edge in direction of —z-axis

Both images apply to an orientation of the member axis in accordance with the global coordinate system.

Tension Perpendicular to Grain

If the check box is ticked, designs for the maximum tensile stresses perpendicular to the grain, for example according to [2] condition (85) or [1] condition (6.50), and for shear due to shear force will be performed in the ridge cross-section.

The volume *V* required for the transversal tension analysis is determined by RF-/TIMBER Pro on the basis of the geometric conditions according to [1] Figure 6.9(a). The portion of the small "wedge" due to α_{ap} is considered in a simplified way so that the volume subjected to transversal tension in the ridge zone is greater. However, because of the mostly low taper slope, this simplification has almost no impact on the analysis.

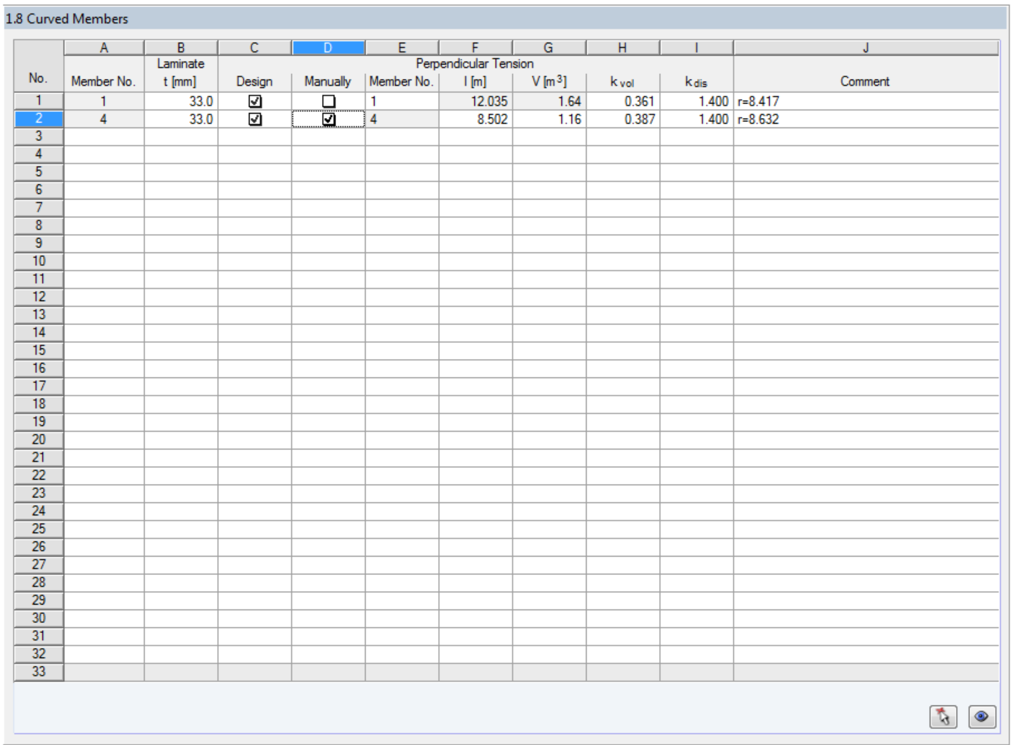
Alternatively, you can enter the specifications *Manually*. After selecting the check box, the fields for entering the volume *V* and the volume factor k_{vol} according to [1] expression (6.51) become accessible.

2.8 Curved Members

Module Window *1.8 Curved Members* is only available in the RFEM add-on module **RF**-TIMBER Pro: RSTAB does not allow for curved lines.

This window appears if at least one member with a curved shape is selected for design in the *1.1 General Data* window. Curved members can be defined in RFEM, for example, by using the line types "spline" or "arc."

According to SIA 265 [3], the design of curved members is not possible.


1.8 Curved Members

No.	A Member No.	B Laminate t [mm]	C Design	D Manually	E Perpendicular Tension Member No.	F l [m]	G V [m³]	H k_{vol}	I k_{dis}	J Comment
1	1	33.0	☑	☐	1	12.035	1.64	0.361	1.400	r=8.417
2	4	33.0	☑	☑	4	8.502	1.16	0.387	1.400	r=8.632
3										
4										
5										
6										
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 2.34 Window *1.8 Curved Members*

Member

This table column lists the numbers of all members that are aligned to curved lines and have a uniform cross-section.

Laminate

If a glulam material is used, you can specify the thickness t of the lamellas.

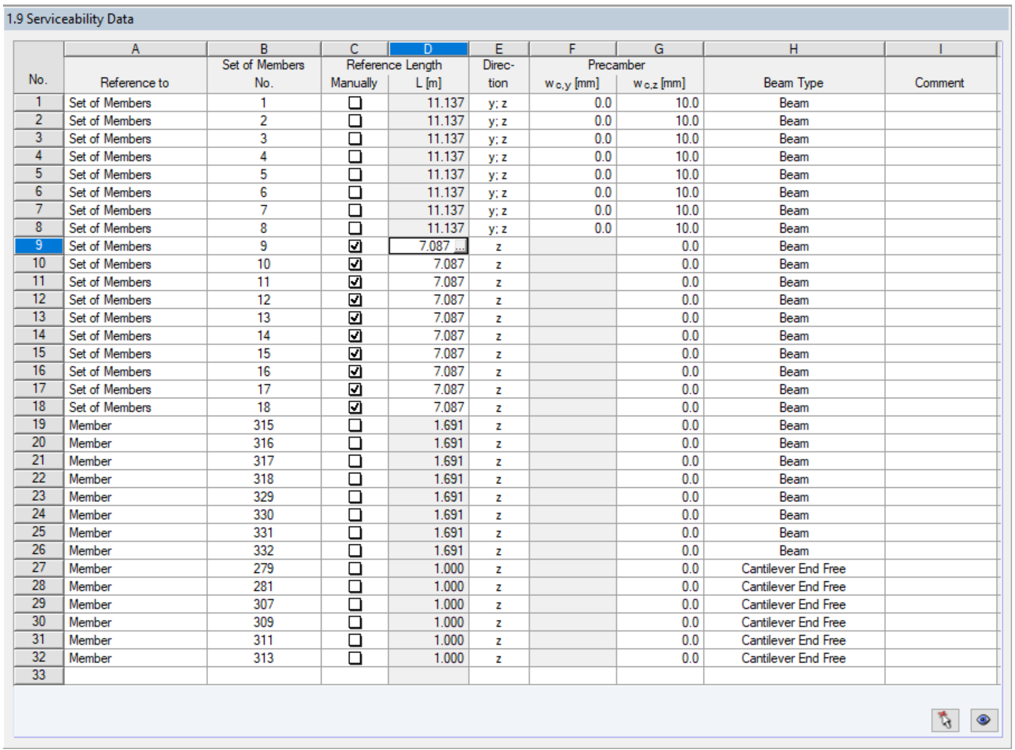
Perpendicular Tension

If you select the *Design* check box, RF-TIMBER Pro will perform an analysis of transversal tension. The factors k_{dis} and k_{vol} are preset according to [1] expression (6.51) or (6.52), but you can adjust them.

For EN 1995-1-1, columns F and G for adjusting the length *l* and the volume *V* are accessible after selecting the *Manually* check box.

2.9 Serviceability Data

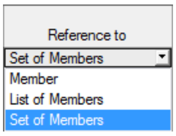
This window controls various settings for the serviceability limit state design. It is displayed if corresponding data has been set in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2).



1.9 Serviceability Data

No.	A Reference to	B Set of Members No.	C Reference Length Manually	D L [m]	E Direc-tion	F Precamber $w_{c,y}$ [mm]	G $w_{c,z}$ [mm]	H Beam Type	I Comment
1	Set of Members	1	☐	11.137	y; z	0.0	10.0	Beam	
2	Set of Members	2	☐	11.137	y; z	0.0	10.0	Beam	
3	Set of Members	3	☐	11.137	y; z	0.0	10.0	Beam	
4	Set of Members	4	☐	11.137	y; z	0.0	10.0	Beam	
5	Set of Members	5	☐	11.137	y; z	0.0	10.0	Beam	
6	Set of Members	6	☐	11.137	y; z	0.0	10.0	Beam	
7	Set of Members	7	☐	11.137	y; z	0.0	10.0	Beam	
8	Set of Members	8	☐	11.137	y; z	0.0	10.0	Beam	
9	Set of Members	9	☑	7.087	z		0.0	Beam	
10	Set of Members	10	☑	7.087	z		0.0	Beam	
11	Set of Members	11	☑	7.087	z		0.0	Beam	
12	Set of Members	12	☑	7.087	z		0.0	Beam	
13	Set of Members	13	☑	7.087	z		0.0	Beam	
14	Set of Members	14	☑	7.087	z		0.0	Beam	
15	Set of Members	15	☑	7.087	z		0.0	Beam	
16	Set of Members	16	☑	7.087	z		0.0	Beam	
17	Set of Members	17	☑	7.087	z		0.0	Beam	
18	Set of Members	18	☑	7.087	z		0.0	Beam	
19	Member	315	☐	1.691	z		0.0	Beam	
20	Member	316	☐	1.691	z		0.0	Beam	
21	Member	317	☐	1.691	z		0.0	Beam	
22	Member	318	☐	1.691	z		0.0	Beam	
23	Member	329	☐	1.691	z		0.0	Beam	
24	Member	330	☐	1.691	z		0.0	Beam	
25	Member	331	☐	1.691	z		0.0	Beam	
26	Member	332	☐	1.691	z		0.0	Beam	
27	Member	279	☐	1.000	z		0.0	Cantilever End Free	
28	Member	281	☐	1.000	z		0.0	Cantilever End Free	
29	Member	307	☐	1.000	z		0.0	Cantilever End Free	
30	Member	309	☐	1.000	z		0.0	Cantilever End Free	
31	Member	311	☐	1.000	z		0.0	Cantilever End Free	
32	Member	313	☐	1.000	z		0.0	Cantilever End Free	
33									

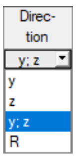
Figure 2.35 Window 1.9 *Serviceability Data*



In column A, you decide whether the deformation refers to single members, lists of members, or sets of members.

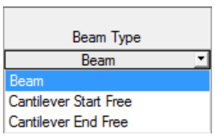
For a set or list of members, it is necessary that a uniform member orientation and rotation of all included members is given. Only in this way, the deformation components will be determined correctly.

In column B, you can enter the numbers of the members or sets of members that you want to design. You can also use the button to select them graphically in the RFEM/RSTAB work window. Then, the *Reference Length* appears automatically in column D. The column presets the lengths of the members, sets of members, or lists of members. You can adjust the values *Manually* after selecting the check box in column C.



In column E, you define the governing *Direction* for the deformation analysis. You can choose the directions of the local member axes y and z as well as the resulting deformation R.

In columns F and G, a *Precamber* w_c can be taken into account. The two input options are related to the directions of the member axes y and z (see Figure 2.28).

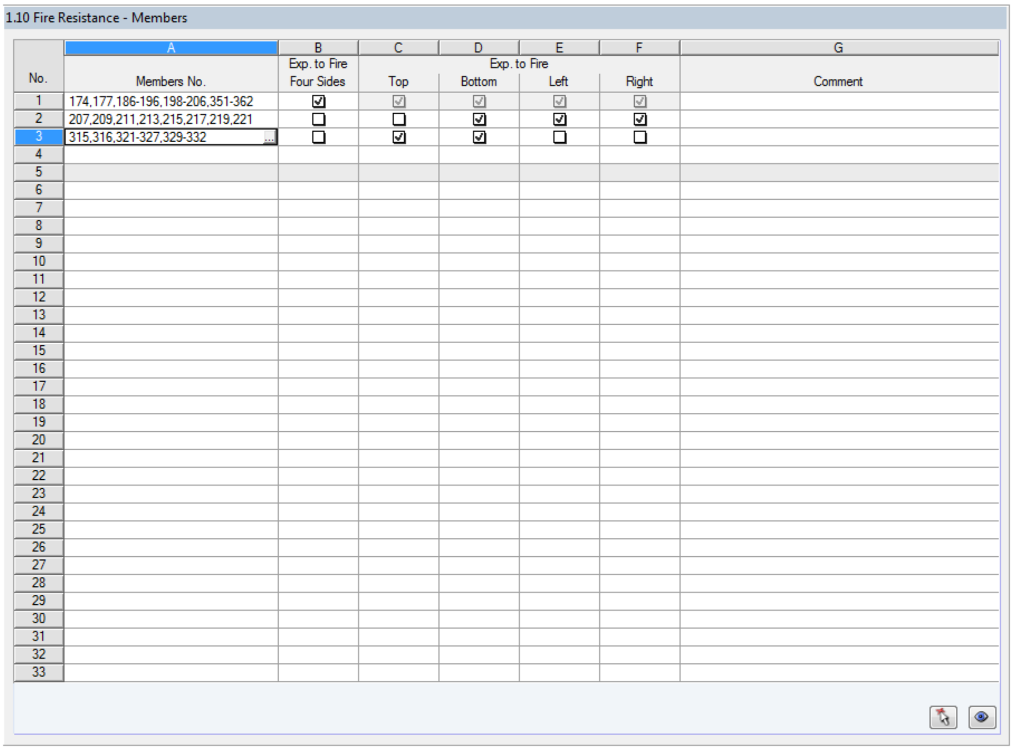


For a correct application of limit deformations, the *Beam Type* is of vital importance. In column H, you can specify whether a beam or a cantilever is to be designed and which end is free of support.

The setting in the *Serviceability* tab of the *Details* dialog box indicates whether the deformations are related to the undeformed system or to shifted members ends/set of members ends (see Figure 3.3).

2.10 Fire Resistance - Members

This window manages the fire protection parameters for members. It is displayed if corresponding data has been set in the *Fire Resistance* tab of Window 1.1 (see Chapter 2.1.3).


1.10 Fire Resistance - Members

No.	A Members No.	B Exp. to Fire Four Sides	C Exp. to Fire Top	D Bottom	E Left	F Right	G Comment
1	174,177,186-196,198-206,351-362	☑	☑	☑	☑	☑	
2	207,209,211,213,215,217,219,221	☐	☐	☑	☑	☑	
3	315,316,321-327,329-332	☐	☑	☑	☐	☐	

Figure 2.36 Window *1.10 Fire Resistance - Members*

In column A, you decide for which members you want to perform a fire resistance design. With the button you can select the members graphically in the work window of RFEM or RSTAB.

In column B, you can specify if they are *Exp. to Fire on Four Sides*. If the cross-section is not exposed to fire on all sides, clear the selection of the check-box. Thus, the following table columns become available where you can specify the sides of the cross-section that are exposed to fire by ticking the boxes. Based on your settings, the ideal remaining cross-section is computed.

Details...

The general parameters for the fire resistance design are managed in the *Fire Resistance* tab of the *Details* dialog box (see Figure 3.4).

2.11 Fire Resistance - Sets of Members

This window manages the fire protection parameters for sets of members. It is displayed if at least one set of members is selected for design in the *1.1 General Data* window and corresponding data is set in the *Fire Resistance* tab.

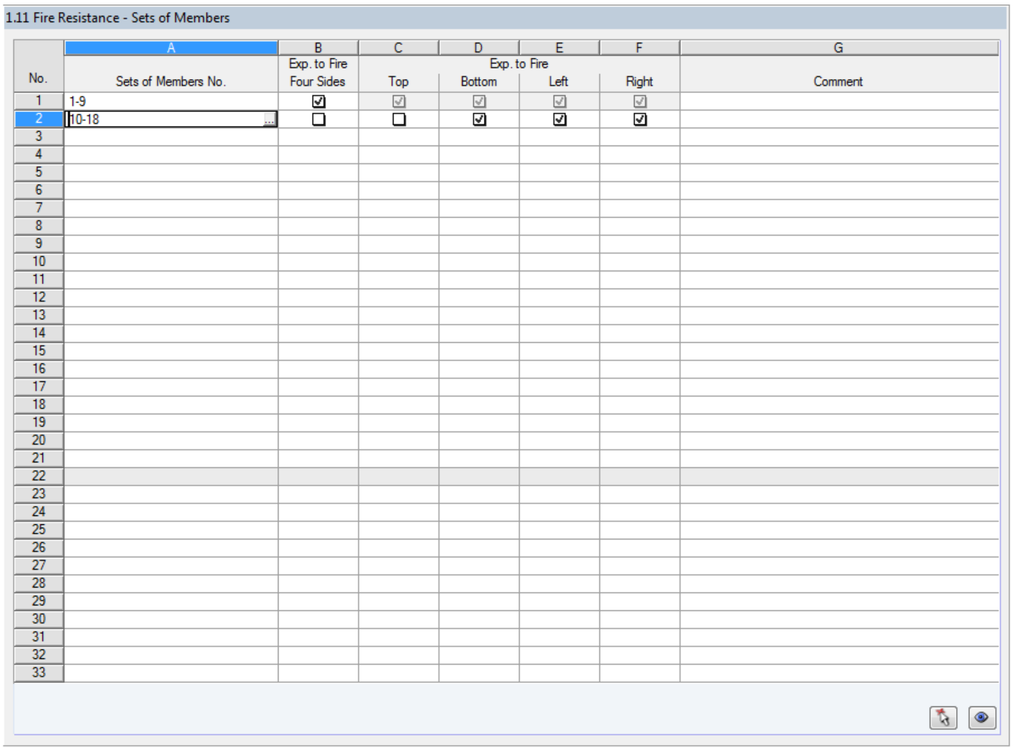


Figure 2.37 Window *1.11 Fire Resistance - Sets of Members*

The concept of the window is similar to the previous Window *1.10 Fire Resistance - Members*. Here, you can define the cross-section's sides of charring as described in Chapter 2.10.

2.12 Parameters

In the final input window, you can define weakenings of the cross-section due to notches as well as parameters for the shear force reduction on supports.

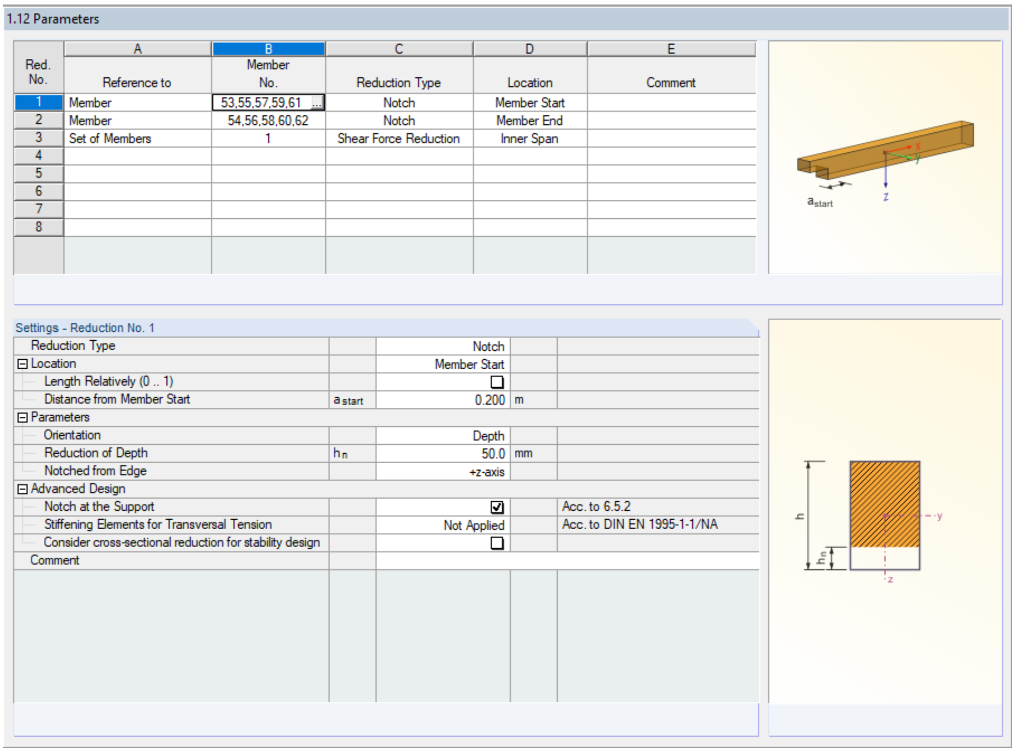
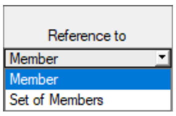


Figure 2.38 Window *1.12 Parameters*

Reference to



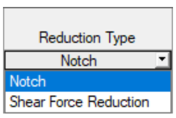
In column A, you decide whether the next settings refer to single members or sets of members.

Member/Set of Members No.

In column B, you can manually enter the numbers of the members or sets of members or select them graphically in the RFEM/RSTAB work window with the button.

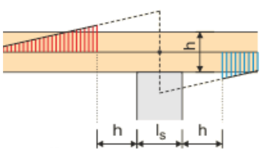
Please note for a set of members that all included members have the same member orientation and rotation. Moreover, the orientation of the set of members must be identical with the orientation of the members. These conditions apply to notches as well as shear force reductions.

Reduction Type



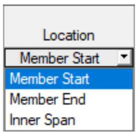
The list offers a choice between *Notch* and *Shear Force Reduction*. Notches represent weakenings of the cross-section as they occur, for example, on supports or are caused by birdsmouth cuts for beam joints.

The parameters for shear force reduction in the support zone are specified in the German annex for [1] (NA.5) clause 6.1.7 as follows:



For bending beams with supports at the lower beam edge and a load application on the upper beam edge, it is allowed to perform the design of shear stresses and, if applicable, of shear connectors in the zone of end and intermediate supports, providing that there are no notches or cut-outs, with the governing shear force. The shear force in a distance *h* (*h* = beam depth over support center) to the support edge may be assumed.

Location



For the selected elements you have to define the position where the notch or the reduction zone is located. This location can refer to the start, the end, or the inner span of the member or set of members. A list with the corresponding options becomes accessible when you click into the input field.

The distances of the cross-section weakening or the shear force reduction along the member axis refer to the structural system, not to the real beam.

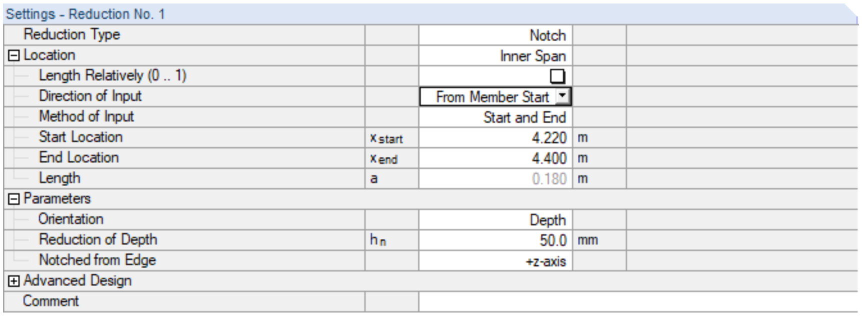
Comment

The final table column offers the possibility to enter user-defined notes.

Settings

In the window section below the table, you can define the details of the notch or the shear force reduction. The available parameters also depend on the specified *Location*. Figure 2.39 shows the parameters entered for a notch in the inner span of a set of members.

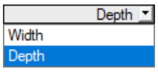
Notch



Settings - Reduction No. 1

Reduction Type		Notch	
⊟ Location		Inner Span	
Length Relatively (0 .. 1)		☐	
Direction of Input		From Member Start	
Method of Input		Start and End	
Start Location	x_{start}	4.220	m
End Location	x_{end}	4.400	m
Length	a	0.180	m
⊟ Parameters			
Orientation		Depth	
Reduction of Depth	h_n	50.0	mm
Notched from Edge		+z-axis	
⊞ Advanced Design			
Comment			

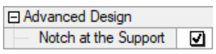
Figure 2.39 Window section *Settings* for notch in inner span



The cross-section reduction itself can be defined by means of *Parameters*. You have to specify the axis direction where the weakening occurs. The *Depth* corresponds to the local member axis z, the *Width* to the axis y (see Figure 2.28).

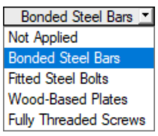
The cross-section's reduction can then be entered in the input field for h_n or b_n.

If you want to reduce the cross-section in both its width and depth, the member in the table above must be selected twice. Then, you can define the orientation separately.



If the cross-section weakening is at the same time a notch on a support, it is possible to activate the *Notch at the Support* option (not available for inner span). In this way, a reduced shear strength, which depends on the geometry of the notch as the cross-section weakening, is taken into account on the reduced cross-section for the shear force design.

The depth factors k_n for beams with notches at the support can be set in the *Other Settings* tab of the *National Annex Settings* dialog box by user-defined specifications for solid wood, glued-laminated timber, and laminated veneer lumber (LVL) (see Chapter 2.1.4).



For the design according to EN 1995-1-1 [1] it is possible to define additionally *Stiffening Elements for Transversal Tension* according to DIN EN 1995-1-1 NCI NA.6.8.3 or ÖNORM B 1995-1-1 Annex G.3 This possibility is mentioned only in these two annexes but is applicable in RF-/TIMBER Pro for each National Annex. The list offers different options for transversal tension stiffenings.

When calculating it is assumed that a crack occurs in the notch zone and the subjacent part is completely suspended on the upper part by fasteners.

Furthermore, you can *Consider cross-sectional reduction for stability design* to determine the slenderness ratio.

Settings - Reduction No. 1

⊟ Advanced Design				
Notch at the Support		☑		Acc. to 6.5.2
⊟ Stiffening Elements for Transversal Tension		Bonded Steel Bars		Acc. to DIN EN 1995-1-1/NA
Number of Stiffening Elements		2		
Distance from Edge to Fastener	t	40.0	mm	
⊟ Size		M12		
Outside Diameter	d_r	12.0	mm	
Tensile Stress Area	A_s	0.84	cm^2	
⊟ Grade		4.6		
Ultimate Strength	f_{ub}	40.00	kN/cm	
Consider cross-sectional reduction for stability design		☑		
⊟ Set Different Data for Fire Design		☑		
Exposed to Fire				
Four Sides		☐		
Top		☑		
Bottom		☐		
Left		☑		
Right		☑		
Comment				

Figure 2.40 Window section *Settings* for transversal tension stiffenings, stability design, and fire resistance

If a fire resistance design is performed, you can *Set Different Data for Fire Design* in order to correctly determine the sides of the reduced cross-sections that are exposed to fire.

Shear Force Reduction

For reducing the shear force at the start of a member RF-/TIMBER Pro provides the input parameters shown in Figure 2.39. Thus, you can define the geometric area which should be irrelevant for the shear force design.

The possibility of a shear force reduction is given in the following National Annexes:

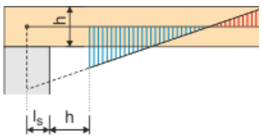
- DIN: NCI to 6.1.7 (NA.5)
- ÖNORM: 6.1.7(2)
- SFS: RIL 205-1-2009, 6.1.7

In case the design is performed according to a different annex, the rules according to DIN EN apply.

Settings - Reduction No. 4

Reduction Type		Shear Force Reduction		
⊟ Location		Member Start		Acc. DIN EN 1995-1-1/NA NCI Zu 6.1.7 (NA.5)
Length Relatively (0 .. 1)		☐		
Distance from Member Start until Support Edge	l_s	0.150	m	
⊟ Internal Force				
Reduce Vy / Vu		☐		
Reduce Vz / Vv		☑		
Comment				

Figure 2.41 Window section *Settings* for shear force reduction on support



It is possible to determine the width of the support by means of the *Distance from Member Start to Support Edge* l_s. As described above, this distance refers to the structural system, not to the real beam.

Furthermore, it must be specified if only the *Internal Force* V_z or V_v in the direction of the "weak" axis (default) or also the shear force in the direction of the member axis y or u is to be considered for the reduction.

3 Calculation

3.1 Detailed Settings

Details...

The designs are carried out with the internal forces and moments determined in RFEM or RSTAB.

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

The *Details* dialog box has the following tabs:

- Resistance
- Stability
- Serviceability
- Fire Resistance
- Other

3.1.1 Resistance

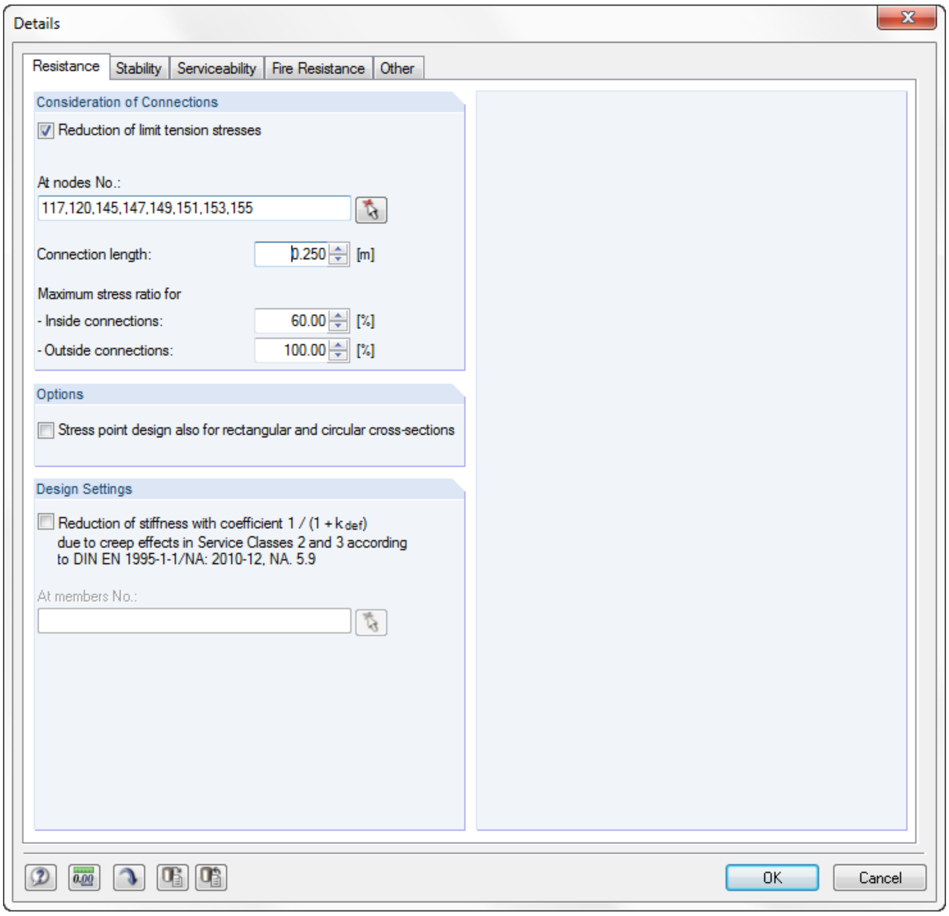


Figure 3.1 Dialog box *Details*, tab *Resistance*

Consideration of Connections

Often, zones near member connections show weakenings of the cross-section. It is possible to take this effect into account by a *Reduction of limit tension stresses*.

The numbers of the relevant *nodes* can be entered manually or selected graphically with the button.

The *Connection length* defines the zone of the members where reduced stresses are considered. In the input field below, you must specify the *Maximum stress ratio* within the connection zone in percent. If required, you can also limit the allowable design ratio outside the connection.

Options

For biaxially bended rectangular cross-sections, the design standards specify a reduction of stresses: The loading is usually less than for uniaxial bending, for which the stresses are maximal over the entire cross-section width.

If you want to compare the stresses of different cross-sections, you can deactivate this reduction by selecting the option *Stress point design also for rectangular and circular cross-sections*. Then, the design will be carried out for each stress point of the cross-section.

The stress points of cross-sections are described in Chapter 2.3.

Design Settings

According to German rules of NCI NA.5.9, a *Reduction of stiffness* has to be done in service classes 2 and 3 for permanent and quasi-permanent load contributions greater than 70% in order to take into account the influence of creeping:

$$f_{c,0,d} \cdot \frac{1}{1 + k_{def}}$$

Equation 3.1

A modulus of elasticity of 1100 kN/cm^2 is reduced in SECL 2 to 1100 / (1 + 0.8) = 611.1 kN/cm^2.

This reduced stiffness is considered according to the equivalent member method in the buckling analysis.

3.1.2 Stability

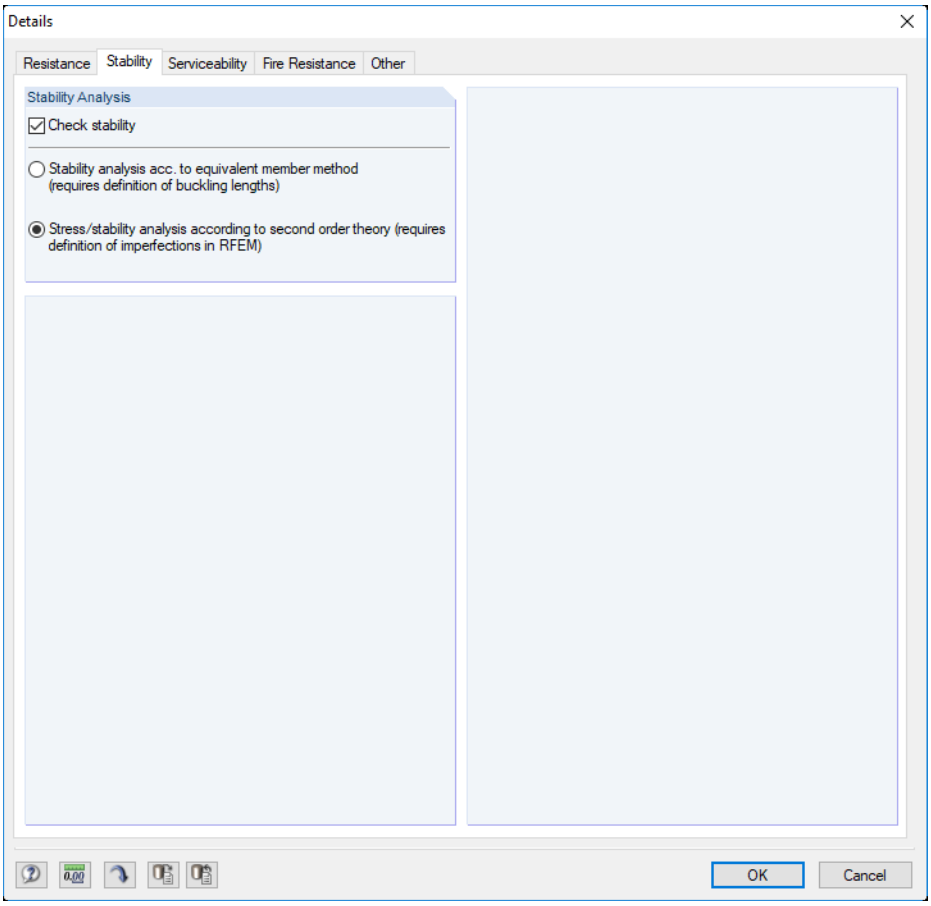
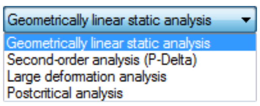


Figure 3.2 Dialog box *Details*, tab *Stability*

Stability Analysis

The *Check stability* check box controls whether or not a stability analysis is performed in addition to the cross-section designs. If you clear it, the input Windows 1.5 and 1.6 won't be shown.



Specifying mode of calculation in RFEM/RSTAB

The *equivalent member method* uses the internal forces determined in RFEM or RSTAB. When applying this method, make sure that the **geometrically linear static analysis** has been set for load combinations (default setting is 2^{nd} order analysis)! Then, to perform the stability analysis, the effective lengths of the members and sets of members subjected to compression or compression and bending must be specified in Windows 1.5 and 1.6.

If the load bearing capacity of a structural system is significantly affected by its deformations, it is recommended to select a calculation according to the *Second-order analysis*. This approach requires the definition of imperfections in RFEM or RSTAB and their consideration for load combinations. The flexural buckling analysis is carried out in the course of the successful calculation of these load combinations in RFEM or RSTAB.

The lateral-torsional buckling analysis must be carried also when calculating according to the second-order analysis. For this you must specify the lengths L_{cr} of the members or sets of members manually in Window 1.5 or 1.6 *Effective Lengths*. In this way, we can make sure that the lateral-torsional buckling analysis will be performed with the appropriate factors (for example 1.0).

3.1.3 Serviceability

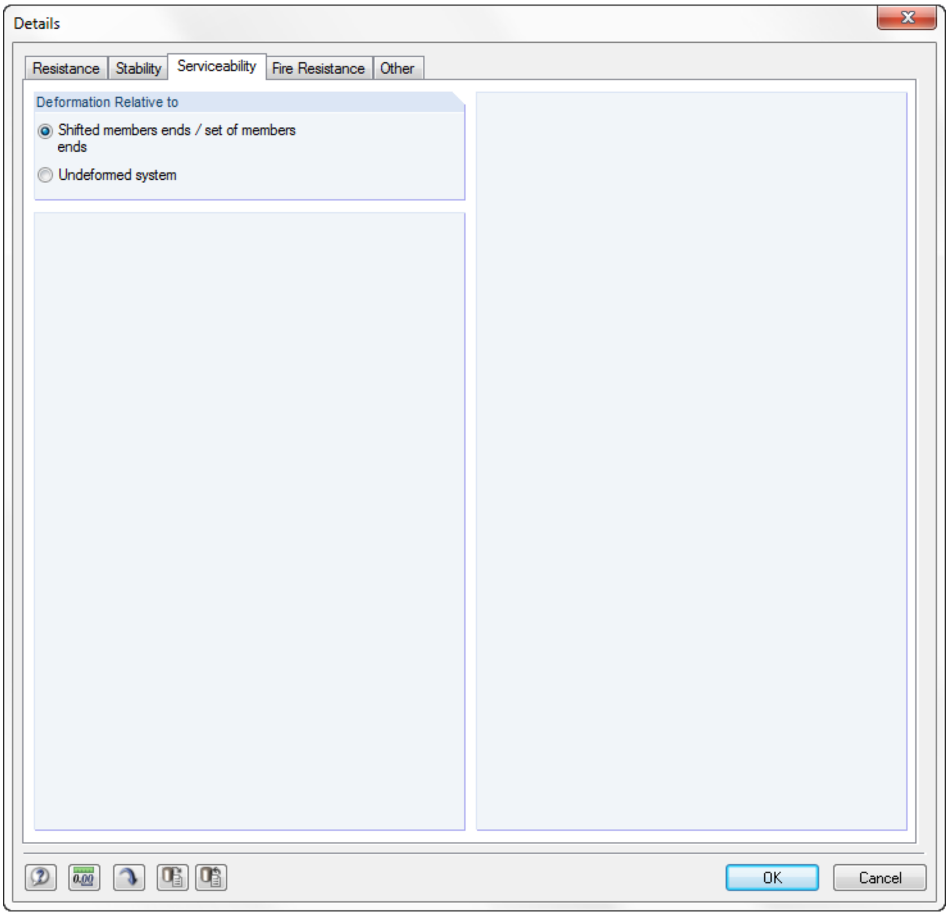


Figure 3.3 Dialog box *Details*, tab *Serviceability*

The options control whether the maximum deformations are related to the shifted ends of members or sets of members (connection line between start and end nodes of the deformed system) or to the undeformed initial system. Generally, deformations must be designed relative to the displacements in the entire structural system.

You can find an example for relating deformations in the following Dlubal article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001081

Nat. Annex...

You can check and, if necessary, adjust the limit deformations in the *National Annex Settings* dialog box (see Figure 2.9).

3.1.4 Fire Resistance

This tab manages detailed settings for the fire resistance design.

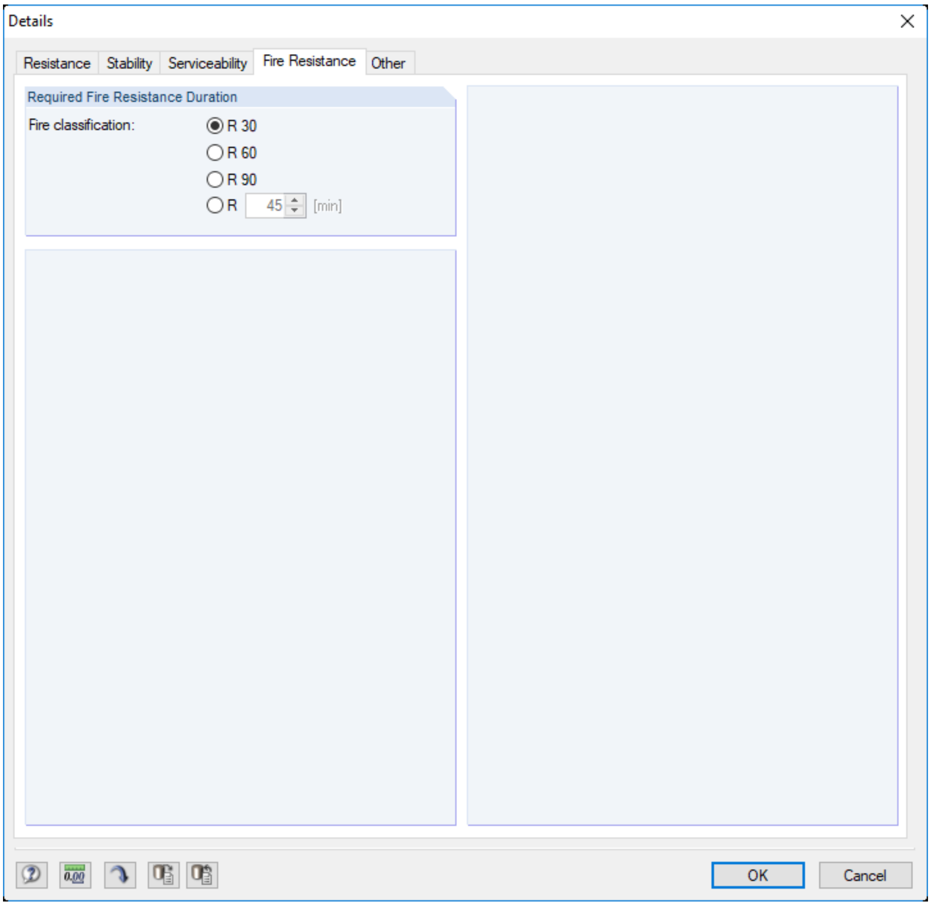


Figure 3.4 Dialog box *Details*, tab *Fire Resistance*

The *Fire classification* can be selected directly or defined individually by specifying a time for the fire duration.

Nat. Annex...

The *National Annex Settings* dialog box manages the standard-specific parameters that are significant for the fire resistance design (see Figure 2.9).

3.1.5 Other Settings

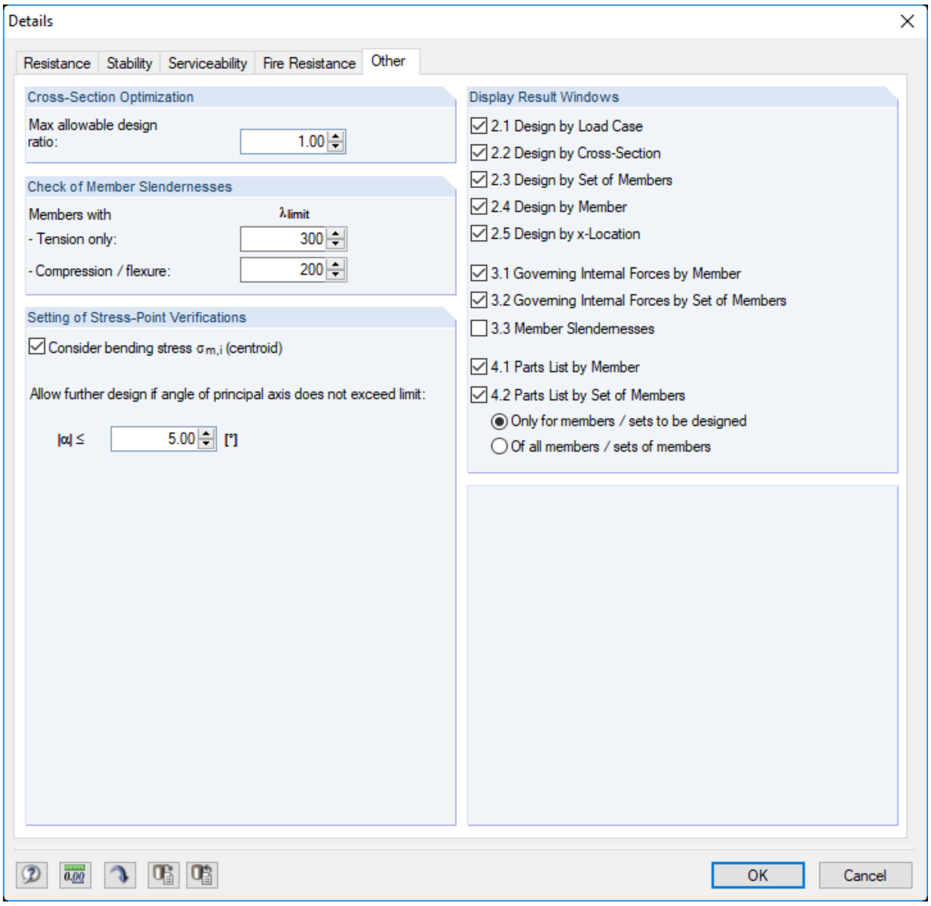


Figure 3.5 Dialog box *Details*, tab *Other*

Cross-Section Optimization

By default, the optimization is targeted on the maximum design ratio of 100%. If necessary, you can set a different upper limit in this field.

Check of Member Slendernesses

The two fields define the limit values λ_{limit} to control the member slendernesses. Separate specifications are possible for members with tension forces only and for members with bending and compression.

In Window 3.3, the limit values are compared to the real member slendernesses. This window is available after the calculation (see Chapter 4.8) if the corresponding check box in the *Display Result Windows* dialog section to the right is ticked.

Setting of Stress-Point Verifications

The check box *Consider bending stress $\sigma_{m,i}$ (centroid)* controls whether the bending stress component available in the centroid is also considered when analyzing compression and bending (stress analysis and stability analysis for buckling).

An example described in Chapter 8.2 illustrates how the stresses from bending are divided into tensile, compressive, and bending stresses.

The design of cross-sections of the "Parametric - Massive" category also include doubly unsymmetrical sections. When performing the shear stress design according to Equation 5.1, the thickness *t* may result for distinctive unbalances in stresses which are too low because it is related to the axes y and z.

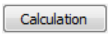
Therefore, it is possible to limit the angle α with the check box *Allow further design if angle of principal axis does not exceed limit*.

Display Result Windows

In this dialog section, you can select which result windows including parts list are shown. The windows are described in Chapter 4.

Window 3.3 *Member Slendernesses* is deactivated by default.

3.2 Starting the Calculation



In each input window of the RF-/TIMBER Pro add-on module, you can start the calculation by clicking the [Calculation] button.

RF-/TIMBER Pro searches for the results of the load cases, load combinations, and result combinations to be designed. If they cannot be found, the program starts the RFEM or RSTAB calculation to determine the design-relevant internal forces.

You can also start the RF-/TIMBER Pro calculation in the RFEM or RSTAB user interface: The *To Calculate* dialog box (menu **Calculate → To Calculate**) lists the design cases of the add-on modules like load cases or load combinations.

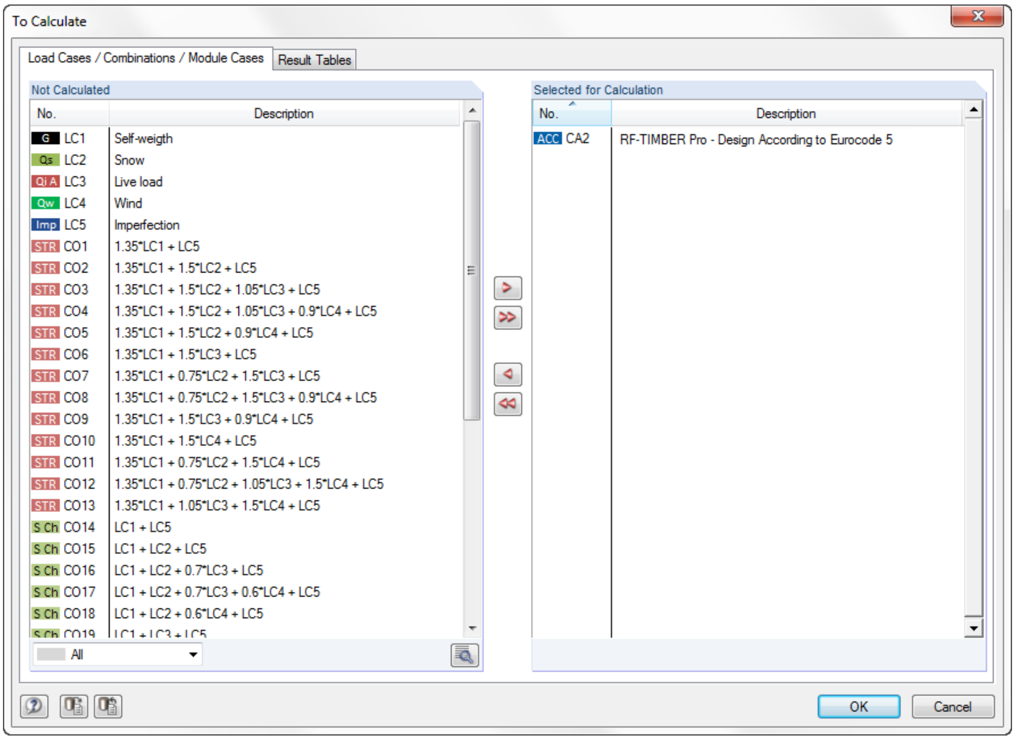
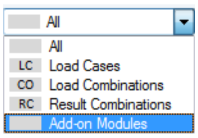


Figure 3.6 Dialog box *To Calculate*



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

To transfer the selected RF-/TIMBER Pro cases to the list on the right, use the [>] button. Then, click [OK] to start the calculation.

You can also calculate a design case directly by using the list in the toolbar: Set the RF-/TIMBER Pro case and click the [Show Results] button.

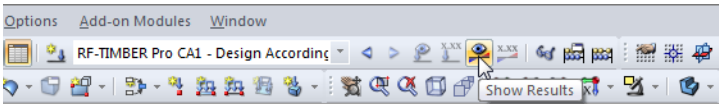


Figure 3.7 Direct calculation of an RF-TIMBER Pro design case in RFEM

Subsequently, you can observe the calculation process in the solver dialog box.

4 Results

Window 2.1 *Design by Load Case* appears immediately after the calculation.

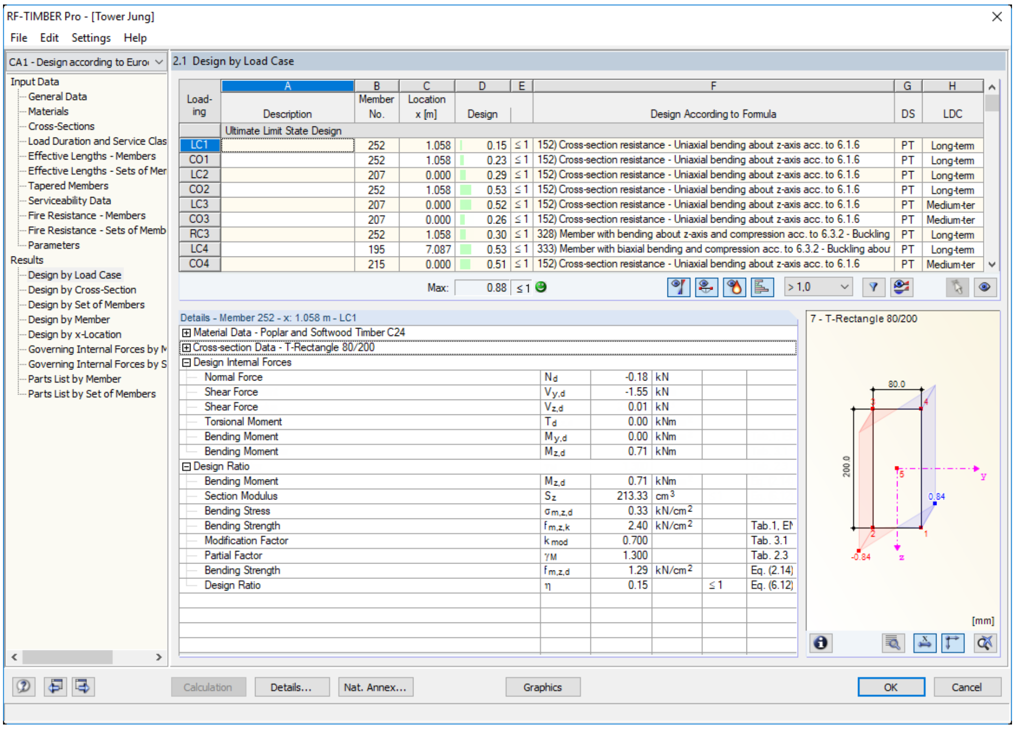


Figure 4.1 Result window with designs and intermediate values

The designs are shown in the result Windows 2.1 to 2.5, sorted by different criteria.

Windows 3.1 and 3.2 list the governing internal forces; Window 3.3 gives information on member slendernesses.

Windows 4.1 and 4.2 show the parts lists by members and sets of members.

Every window can be selected by clicking the corresponding entry in the navigator. To go to the previous or subsequent module window, use the buttons shown on the left. You can also use the function keys [F2] and [F3] to go through the windows.

Click [OK] to save the results. You will exit RF-/TIMBER Pro and return to the main program.

Chapter 4 describes the result windows one by one. Evaluating and checking results is described in Chapter 5.

4.1 Design by Load Case

The upper part of the window shows a summary of the governing designs, sorted by load case, load combination, and result combination. In addition, the table is subdivided into ultimate as well as serviceability limit state and fire resistance design.

The lower part includes detailed information on the cross-section properties, analyzed internal forces, and design parameters for the load case selected above.

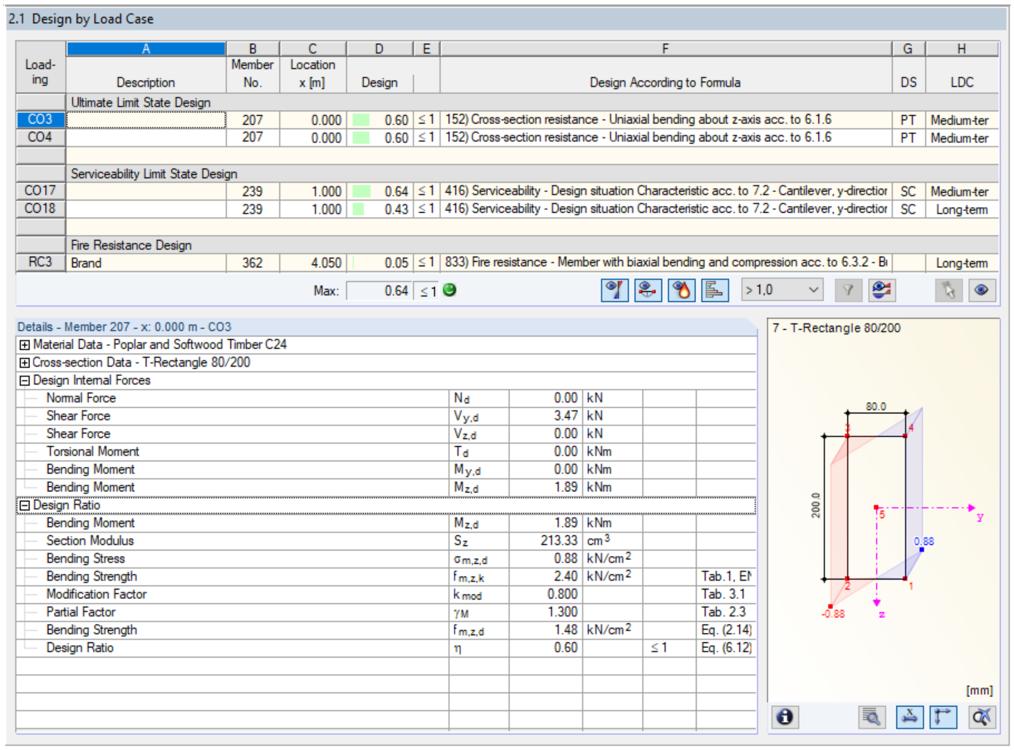


Figure 4.2 Window *2.1 Design by Load Case*

Description

This column shows the descriptions of the load cases, load and result combinations for which the designs have been performed.

Member No.

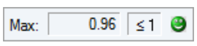
This column shows the number of the member with the maximum ratio for the designed action.

Location x

This column shows the respective x-location of the member where the maximum design ratio occurs. The following member locations *x* are used for the table output:

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

Design



Columns D and E show the design conditions conforming to standards ([1], [2] or [3]).

The length of the colored bar represents graphically the respective design ratio.

Design According to Formula

This column displays the expressions of the standard from which the designs have been performed.

DS

Column G provides information on the design relevant situations (DS): *PT* or *AC* for the ultimate limit state (see Figure 2.6), or one of the design situations *SC*, *SF*, *SQ* or *SQ1* for serviceability according to the specifications in the *1.1 General Data* window (see Figure 2.7).

LDC

Column H shows the load duration classes that have been defined in Window 1.4 (see Chapter 2.4). They affect the modification factors k_{mod}.

4.2 Design by Cross-Section

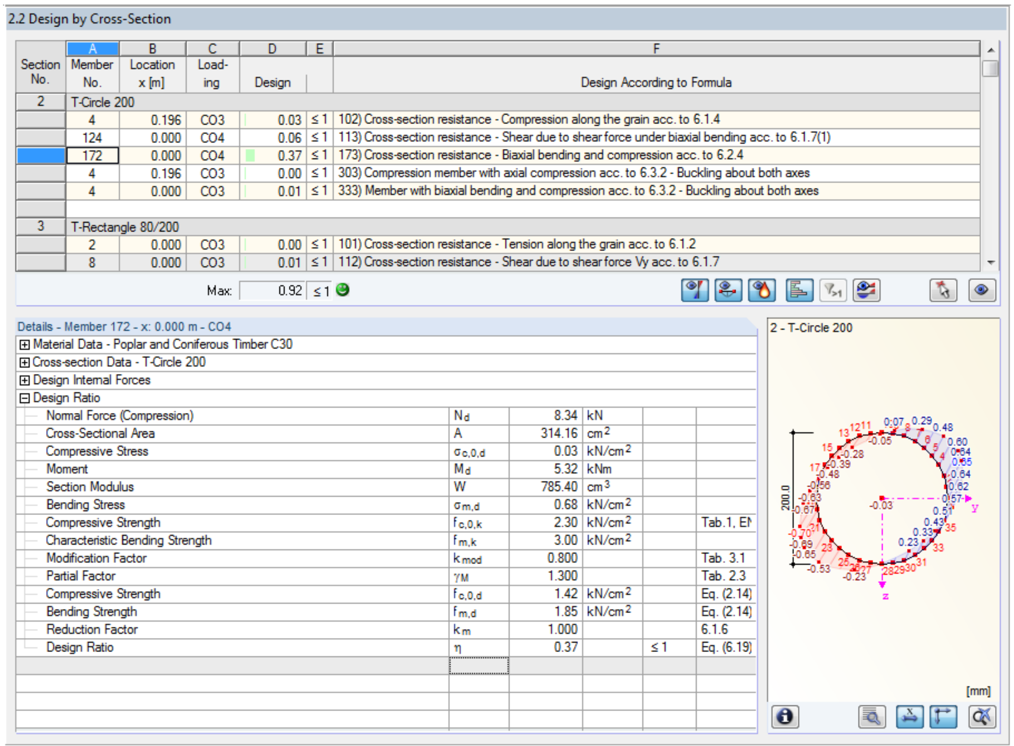


Figure 4.3 Window 2.2 *Design by Cross-Section*

In this results window, the maximum design ratios of all members and actions selected for design are listed by cross-section. The results are sorted by cross-section design and stability analysis as well as serviceability limit state design and fire resistance design.

If there is a tapered member, the cross-sections of the member start and end are listed separately.

4.3 Design by Set of Members

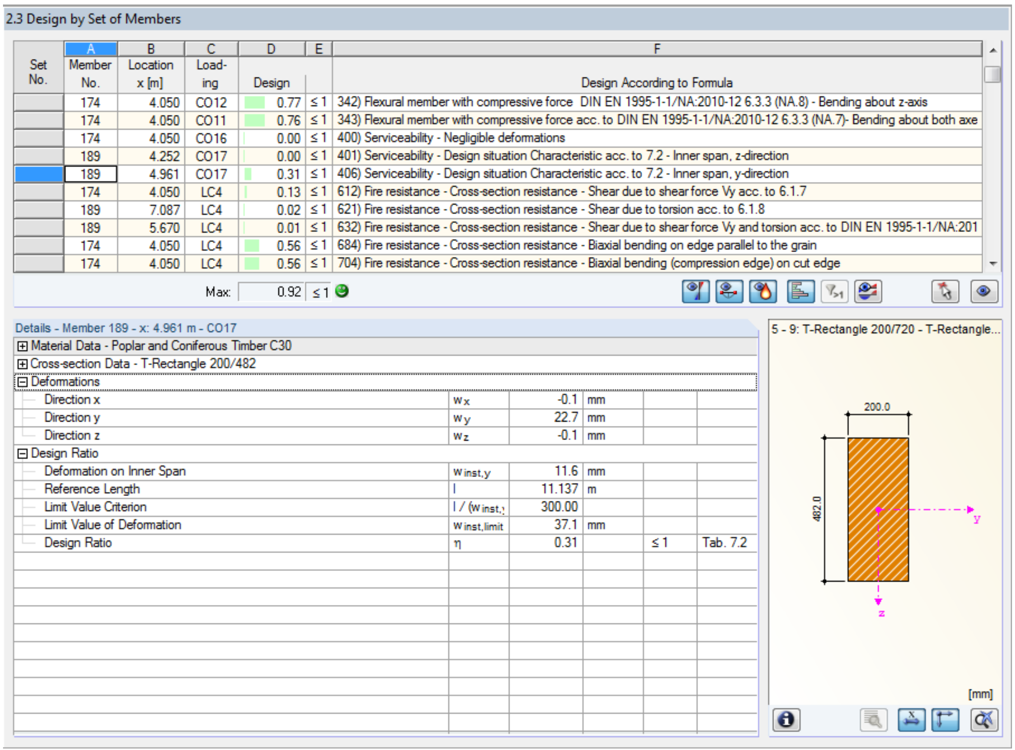


Figure 4.4 Window 2.3 *Design by Set of Members*

This results window is displayed if at least one set of members has been selected for design. The window lists the maximum design ratios sorted by set of members.

The *Member No.* column shows the number of the member within the set of members that bears the maximum ratio for the individual design criteria.

The output by set of members allows you to clearly present the design of an entire structural group (a chord, for example).

4.4 Design by Member

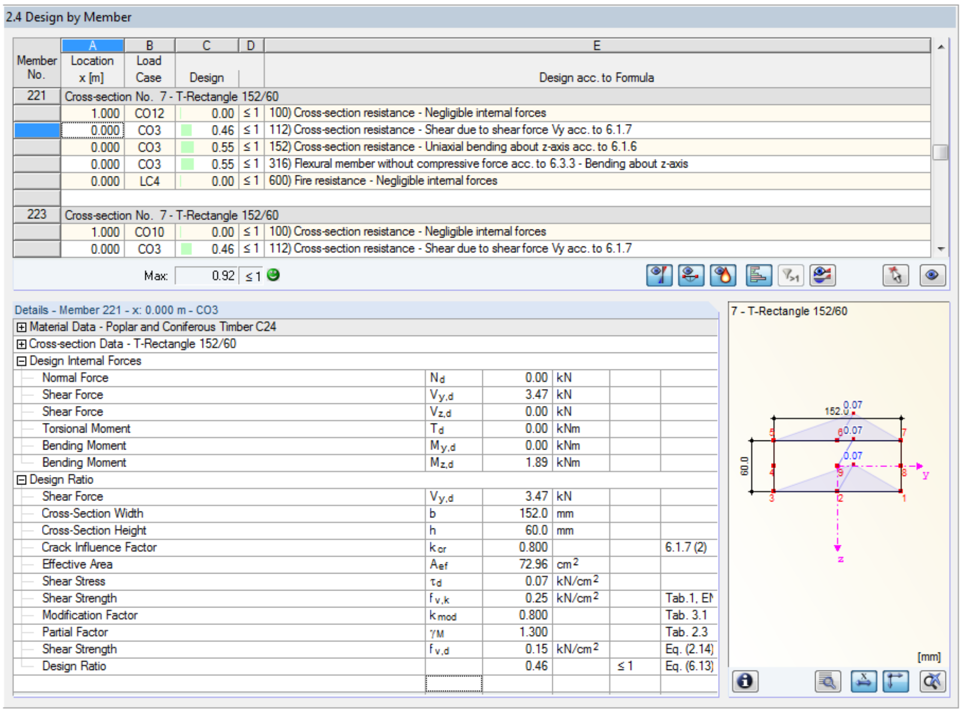


Figure 4.5 Window 2.4 *Design by Member*

This results window shows the maximum design ratios for the individual designs sorted by member number. The columns are described in detail in Chapter 4.1.

4.5 Design by x-Location

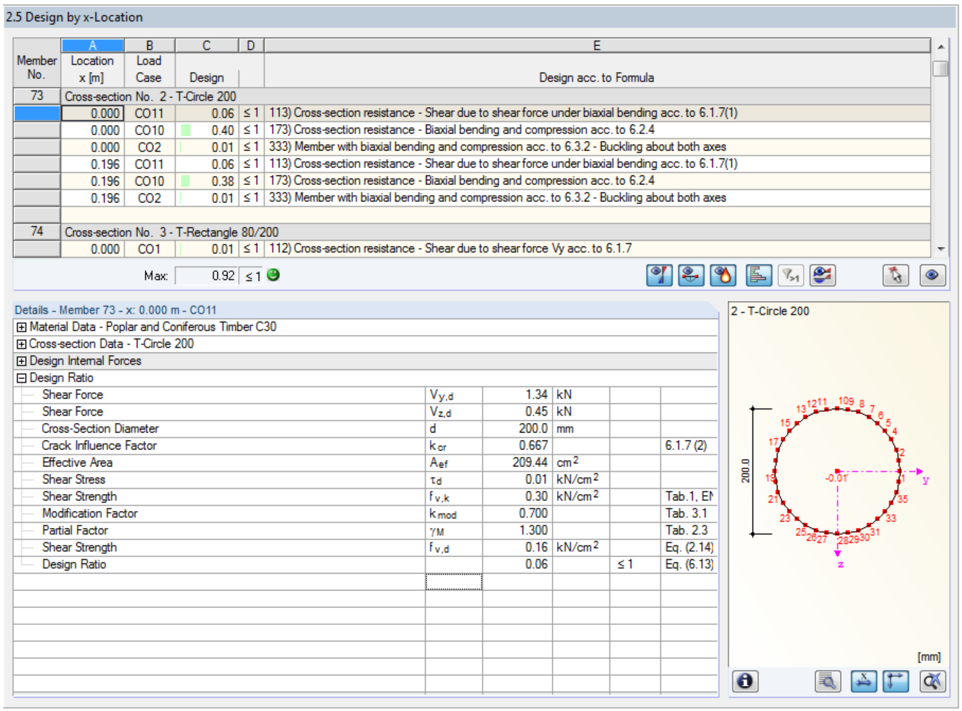


Figure 4.6 Window 2.5 *Design by x-Location*

This results window lists the maxima for each member at all locations *x*, resulting from the division points defined in RFEM or RSTAB:

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

4.6 Governing Internal Forces by Member

For each member, this window displays the governing internal forces, that is, the forces and moments that result in the maximum utilization in the individual designs.

3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Load Case	C Forces [kN] N	D V_y	E V_z	F Moments [kNm] M_T	G M_y	H M_z	I Design According to Formula
248	Cross-section No. 7 - T-Rectangle 152/60								
	1.058	CO12	-0.70	-3.11	0.01	0.00	0.00	1.36	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
	1.058	CO7	-0.42	-3.91	0.01	0.01	0.00	1.68	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7
	1.058	CO7	-0.42	-3.91	0.01	0.01	0.00	1.68	132) Cross-section resistance - Shear due to shear force Vy and torsion a
	1.058	CO7	-0.42	-3.91	0.01	0.01	0.00	1.68	172) Cross-section resistance - Uniaxial bending about z-axis and compre
	0.000	CO7	-0.42	-1.73	0.01	0.01	-0.02	-1.22	173) Cross-section resistance - Biaxial bending and compression acc. to 6
	1.058	CO7	-0.42	-3.91	0.01	0.01	0.00	1.68	328) Member with bending about z-axis and compression acc. to 6.3.2 - E
	1.058	CO7	-0.42	-3.91	0.01	0.01	0.00	1.68	346) Flexural member with compressive force acc. to 6.3.3 - Bending abo
	0.000	CO7	-0.42	-1.73	0.01	0.01	-0.02	-1.22	347) Flexural member with compressive force acc. to DIN EN 1995-1-1/N
249	Cross-section No. 7 - T-Rectangle 152/60								
	1.058	CO12	-0.45	-2.91	0.00	0.00	0.00	1.28	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
	1.058	CO7	-0.39	-3.56	0.01	0.00	0.00	1.53	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7
	1.058	CO7	-0.39	-3.56	0.01	0.00	0.00	1.53	132) Cross-section resistance - Shear due to shear force Vy and torsion a
	1.058	CO7	-0.39	-3.56	0.01	0.00	0.00	1.53	172) Cross-section resistance - Uniaxial bending about z-axis and compre
	0.000	CO7	-0.39	-1.38	0.01	0.00	-0.02	-1.00	173) Cross-section resistance - Biaxial bending and compression acc. to 6
	1.058	CO7	-0.39	-3.56	0.01	0.00	0.00	1.53	328) Member with bending about z-axis and compression acc. to 6.3.2 - E
	1.058	CO7	-0.39	-3.56	0.01	0.00	0.00	1.53	346) Flexural member with compressive force acc. to 6.3.3 - Bending abo
	0.000	CO7	-0.39	-1.38	0.01	0.00	-0.02	-1.00	347) Flexural member with compressive force acc. to DIN EN 1995-1-1/N
250	Cross-section No. 7 - T-Rectangle 152/60								
	1.058	CO8	-0.43	-3.47	0.01	0.00	0.00	1.49	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
	1.058	CO7	-0.39	-3.54	0.01	0.00	0.00	1.52	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7
	1.058	CO7	-0.39	-3.54	0.01	0.00	0.00	1.52	132) Cross-section resistance - Shear due to shear force Vy and torsion a
	1.058	CO1	-0.21	-1.63	0.01	0.00	0.00	0.79	152) Cross-section resistance - Uniaxial bending about z-axis acc. to 6.1.6
	0.000	CO1	-0.21	-1.05	0.01	0.00	-0.01	-0.61	153) Cross-section resistance - Biaxial bending acc. to 6.1.6
	1.058	CO7	-0.39	-3.54	0.01	0.00	0.00	1.52	172) Cross-section resistance - Uniaxial bending about z-axis and compre
	0.000	CO7	-0.39	-1.36	0.01	0.00	-0.01	-0.99	173) Cross-section resistance - Biaxial bending and compression acc. to 6
	1.058	CO1	-0.21	-1.63	0.01	0.00	0.00	0.79	316) Flexural member without compressive force acc. to 6.3.3 - Bending a
	0.000	CO1	-0.21	-1.05	0.01	0.00	-0.01	-0.61	317) Flexural member without compressive force acc. to 6.3.3 - Bending a
	1.058	CO7	-0.39	-3.54	0.01	0.00	0.00	1.52	328) Member with bending about z-axis and compression acc. to 6.3.2 - E
	1.058	CO7	-0.39	-3.54	0.01	0.00	0.00	1.52	346) Flexural member with compressive force acc. to 6.3.3 - Bending abo
	0.000	CO7	-0.39	-1.36	0.01	0.00	-0.01	-0.99	347) Flexural member with compressive force acc. to DIN EN 1995-1-1/N

Figure 4.7 Window 3.1 *Governing Internal Forces by Member*

Location x

This column shows the respective x-location of the member where the maximum design ratio occurs.

Load Case

This column shows the numbers of the load case as well as the load or result combination whose internal forces result in the maximum design ratio.

Forces / Moments

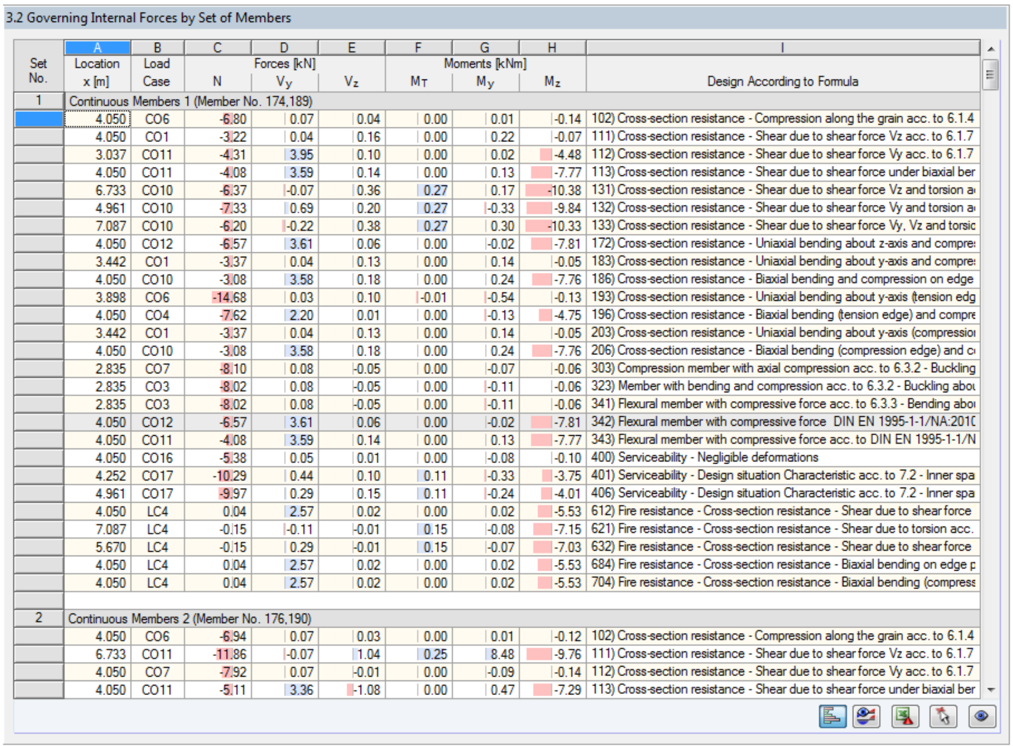
For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing the maximum ratios in the respective cross-section designs, stability analyses, serviceability limit state designs, and fire resistance designs.

Design According to Formula

The final column gives information on the design types and equations used for performing the designs according to the selected standard.

4.7

Governing Internal Forces by Set of Members



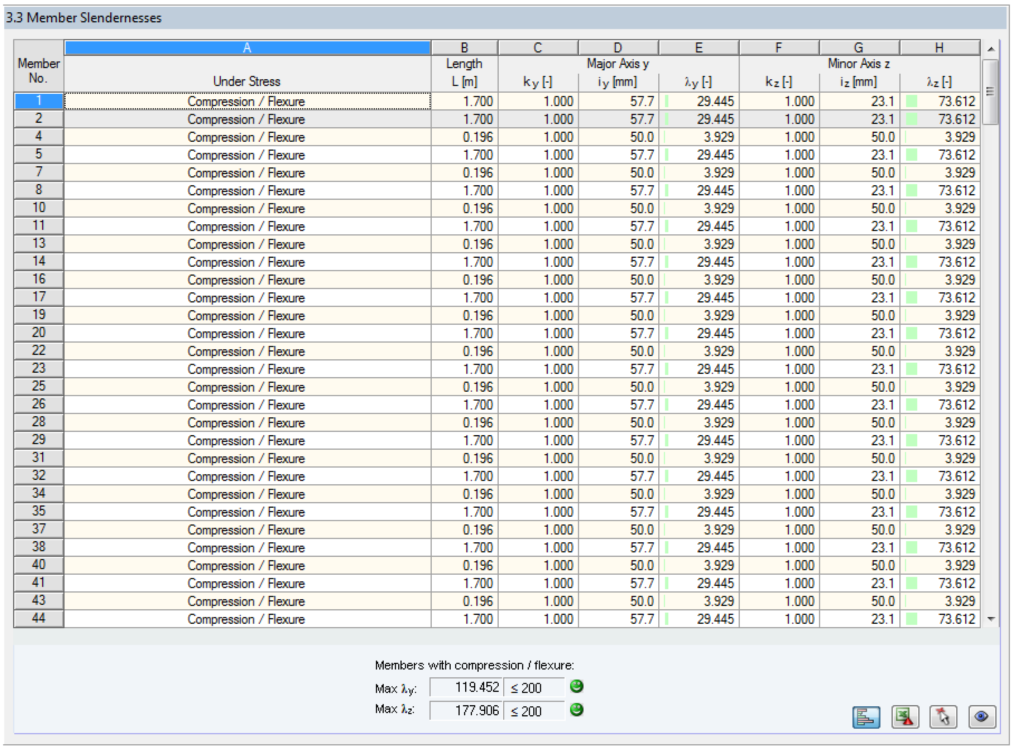
3.2 Governing Internal Forces by Set of Members

	A	B	C	D	E	F	G	H	I
Set No.	Location x [m]	Load Case	Forces [kN] N	Vy	Vz	Moments [kNm] MT	My	Mz	Design According to Formula
1	Continuous Members 1 (Member No. 174,189)								
	4.050	CO6	-6.80	0.07	0.04	0.00	0.01	-0.14	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
	4.050	CO1	-3.22	0.04	0.16	0.00	0.22	-0.07	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
	3.037	CO11	-4.31	3.95	0.10	0.00	0.02	-4.48	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7
	4.050	CO11	-4.08	3.59	0.14	0.00	0.13	-7.77	113) Cross-section resistance - Shear due to shear force under biaxial ber
	6.733	CO10	-6.37	-0.07	0.36	0.27	0.17	-10.38	131) Cross-section resistance - Shear due to shear force Vz and torsion a
	4.961	CO10	-7.33	0.69	0.20	0.27	-0.33	-9.84	132) Cross-section resistance - Shear due to shear force Vy and torsion a
	7.087	CO10	-6.20	-0.22	0.38	0.27	0.30	-10.33	133) Cross-section resistance - Shear due to shear force Vy, Vz and torsic
	4.050	CO12	-6.57	3.61	0.06	0.00	-0.02	-7.81	172) Cross-section resistance - Uniaxial bending about z-axis and compre:
	3.442	CO1	-3.37	0.04	0.13	0.00	0.14	-0.05	183) Cross-section resistance - Uniaxial bending about y-axis and compre:
	4.050	CO10	-3.08	3.58	0.18	0.00	0.24	-7.76	186) Cross-section resistance - Biaxial bending and compression on edge
	3.898	CO6	-14.68	0.03	0.10	-0.01	-0.54	-0.13	193) Cross-section resistance - Uniaxial bending about y-axis (tension edg
	4.050	CO4	-7.62	2.20	0.01	0.00	-0.13	-4.75	196) Cross-section resistance - Biaxial bending (tension edge) and compre
	3.442	CO1	-3.37	0.04	0.13	0.00	0.14	-0.05	203) Cross-section resistance - Uniaxial bending about y-axis (compressio
	4.050	CO10	-3.08	3.58	0.18	0.00	0.24	-7.76	206) Cross-section resistance - Biaxial bending (compression edge) and c
	2.835	CO7	-8.10	0.08	-0.05	0.00	-0.07	-0.06	303) Compression member with axial compression acc. to 6.3.2 - Buckling
	2.835	CO3	-8.02	0.08	-0.05	0.00	-0.11	-0.06	323) Member with bending and compression acc. to 6.3.2 - Buckling abou
	2.835	CO3	-8.02	0.08	-0.05	0.00	-0.11	-0.06	341) Flexural member with compressive force acc. to 6.3.3 - Bending abo
	4.050	CO12	-6.57	3.61	0.06	0.00	-0.02	-7.81	342) Flexural member with compressive force DIN EN 1995-1-1/NA:201(
	4.050	CO11	-4.08	3.59	0.14	0.00	0.13	-7.77	343) Flexural member with compressive force acc. to DIN EN 1995-1-1/N
	4.050	CO16	-5.38	0.05	0.01	0.00	-0.08	-0.10	400) Serviceability - Negligible deformations
	4.252	CO17	-10.29	0.44	0.10	0.11	-0.33	-3.75	401) Serviceability - Design situation Characteristic acc. to 7.2 - Inner spa
	4.961	CO17	-9.97	0.29	0.15	0.11	-0.24	-4.01	406) Serviceability - Design situation Characteristic acc. to 7.2 - Inner spa
	4.050	LC4	0.04	2.57	0.02	0.00	0.02	-5.53	612) Fire resistance - Cross-section resistance - Shear due to shear force
	7.087	LC4	-0.15	-0.11	-0.01	0.15	-0.08	-7.15	621) Fire resistance - Cross-section resistance - Shear due to torsion acc.
	5.670	LC4	-0.15	0.29	-0.01	0.15	-0.07	-7.03	632) Fire resistance - Cross-section resistance - Shear due to shear force
	4.050	LC4	0.04	2.57	0.02	0.00	0.02	-5.53	684) Fire resistance - Cross-section resistance - Biaxial bending on edge p
	4.050	LC4	0.04	2.57	0.02	0.00	0.02	-5.53	704) Fire resistance - Cross-section resistance - Biaxial bending (compress
2	Continuous Members 2 (Member No. 176,190)								
	4.050	CO6	-6.94	0.07	0.03	0.00	0.01	-0.12	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
	6.733	CO11	-11.86	-0.07	1.04	0.25	8.48	-9.76	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
	4.050	CO7	-7.92	0.07	-0.01	0.00	-0.09	-0.14	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7
	4.050	CO11	-5.11	3.36	-1.08	0.00	0.47	-7.29	113) Cross-section resistance - Shear due to shear force under biaxial ber

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

For each set of members, this window shows the internal forces and moments that result in the maximum ratios for the individual designs.

4.8 Member Slendernesses



Member No.	A Under Stress	B Length L [m]	C Major Axis y k_y [-]	D i_y [mm]	E λ_y [-]	F Minor Axis z k_z [-]	G i_z [mm]	H λ_z [-]
1	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
2	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
4	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
5	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
7	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
8	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
10	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
11	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
13	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
14	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
16	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
17	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
19	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
20	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
22	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
23	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
25	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
26	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
28	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
29	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
31	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
32	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
34	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
35	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
37	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
38	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
40	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
41	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612
43	Compression / Flexure	0.196	1.000	50.0	3.929	1.000	50.0	3.929
44	Compression / Flexure	1.700	1.000	57.7	29.445	1.000	23.1	73.612

Figure 4.9 Window 3.3 *Member Slendernesses*

Details...

This result window is displayed if the corresponding check box is ticked in the *Other* tab of the *Details* dialog box (see Figure 3.5).

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They have been determined as a function of the load type. Below the list, you see a comparison with the limit values defined in the *Other* tab of the *Details* dialog box (see Figure 3.5).

Members of the "Tension" or "Cable" type are not displayed in this window.

This window is only of an informative nature. It provides no stability analysis of slendernesses.

4.9 Parts List by Member

Finally, there is a summary of all cross-sections included in the design case.

4.1 Parts List by Member

	A	B	C	D	E	F	G	H	I
Part No.	Cross-Section Description	Number of Members	Length [m]	Total Length [m]	Surface Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1	3 - T-Rectangle 80/220	57	1.70	96.90	58.14	1.71	6.16	10.47	0.597
2	2 - T-Circle 300	55	0.20	10.80	10.18	0.76	24.74	4.86	0.267
3	2 - T-Circle 300	1	0.07	0.07	0.07	0.01	24.74	1.77	0.002
4	2 - T-Circle 300	1	0.13	0.13	0.12	0.01	24.74	3.09	0.003
5	7 - T-Rectangle 180/200	18	1.00	18.00	13.68	0.65	12.60	12.60	0.227
6	7 - T-Rectangle 180/200	18	1.06	19.04	14.47	0.69	12.60	13.33	0.240
7	12 - T-Rectangle 80/200	18	1.00	18.00	10.08	0.29	5.92	5.92	0.107
8	12 - T-Rectangle 80/200	18	1.69	30.44	17.05	0.49	5.92	10.01	0.180
9	7 - T-Rectangle 180/200	19	1.70	32.30	24.55	1.16	12.60	21.42	0.407
Sum		205		225.68	148.33	5.75			2.030

Figure 4.10 Window *4.1 Parts List by Member*

Details...

By default, this list contains only the designed members. If you need a parts list for all members of the model, you can set it in the *Other* tab of the *Details* dialog box (see Figure 3.5).

Part No.

The program assigns part numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

This column shows how many similar members are used for each part.

Length

This column shows the respective length of an individual member.

Total Length

The values in this column are the product from the previous two columns.

Surface Area

For each part, the program displays the surface areas relative to the total length. They are determined from the *Surface* of the cross-sections, which can be found in Windows 1.3 and 2.1 to 2.5 in the *Info About Cross-Section* (see Figure 2.22).

Volume

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

Unit Weight

The *Unit Weight* represents the cross-section weight relative to the length of one meter. For tapered cross-sections, the program averages both cross-section weights.

Weight

The values of this column are determined from the product of the entries in columns C and G.

Total Weight

The final column indicates the total weight of each part.

Sum

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

4.10 Parts List by Set of Members

4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Set	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Set of Members 1	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
2	Set of Members 2	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
3	Set of Members 3	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
4	Set of Members 4	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
5	Set of Members 5	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
6	Set of Members 6	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
7	Set of Members 7	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
8	Set of Members 8	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
9	Set of Members 9	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
10	Set of Members 10	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
11	Set of Members 11	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
12	Set of Members 12	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
13	Set of Members 13	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
14	Set of Members 14	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
15	Set of Members 15	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
16	Set of Members 16	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
17	Set of Members 17	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
18	Set of Members 18	1	11.14	11.14	14.60	1.01	31.88	355.07	0.355
Sum		18		200.47	262.80	18.26			6.391

Figure 4.11 Window 4.2 *Parts List by Set of Members*

The last result window is displayed if at least one set of members has been selected for design. It gives an overview of the timber parts of entire structural groups such as chords.

The columns are described in the previous chapter. If there are different cross-sections within a set of members, the program averages the surface area, the volume, and the unit weight.

5 Results Evaluation

The buttons below the upper table may help you to evaluate the results.

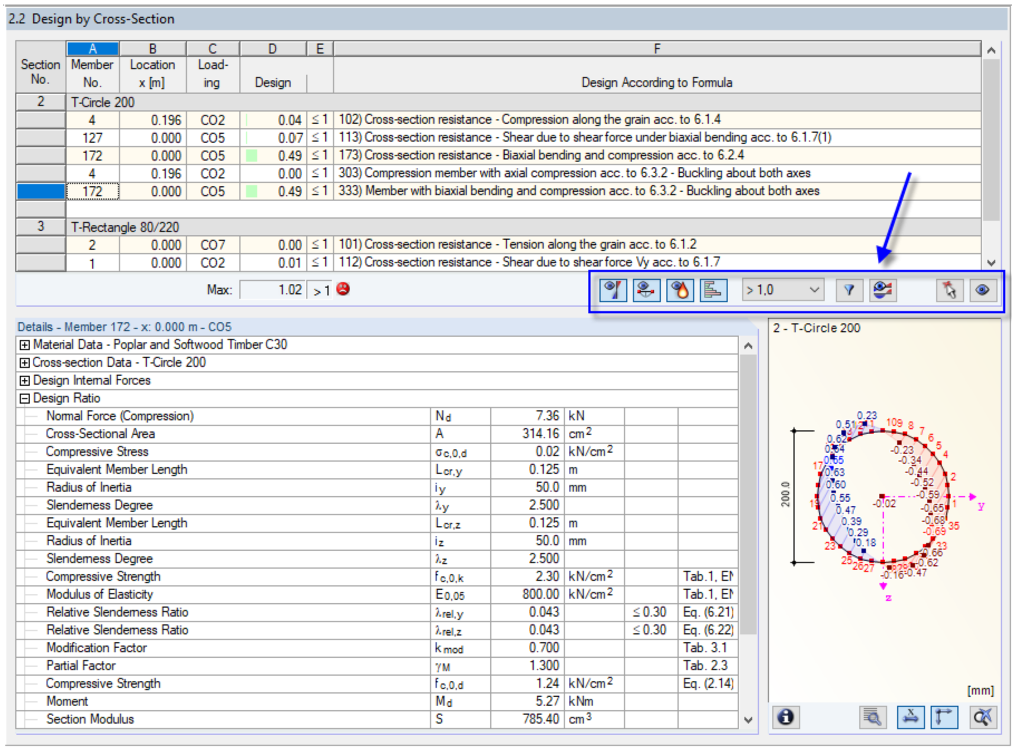


Figure 5.1 Buttons for results evaluation

Button	Description	Function
	Ultimate limit state	Displays or hides results of ultimate limit state design
	Serviceability limit state	Displays or hides results of serviceability limit state design
	Fire resistance	Displays or hides results of fire resistance design
	Color bars	Displays or hides colored relation scales in result windows
> 1,0 / > 1,0 / Max / Define	Filter parameters	Describes criterion by which results are filtered in tables: ratios greater than 1, maximum value, or user-defined limit
	Apply filter	Shows only rows to which filter parameters apply (ratios > 1, maximum, defined value)

	Result diagrams	Opens *Result Diagram on Member* window (see Chapter 5.3)
	Select member	Selects a member graphically to display its results in the table
	View mode	Jumps to RFEM or RSTAB work window to change the view

Table 5.1 Buttons in result Windows 2.1 to 2.5

5.1 Results on RFEM/RSTAB Model

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

Background graphic and view mode

The RFEM/RSTAB work window in the background is useful when you want to find the position of a particular member in the model: The member selected in the result window of RF-/TIMBER Pro is highlighted in color in the background graphic. Moreover, an arrow indicates the member's x-location selected in the active table row.

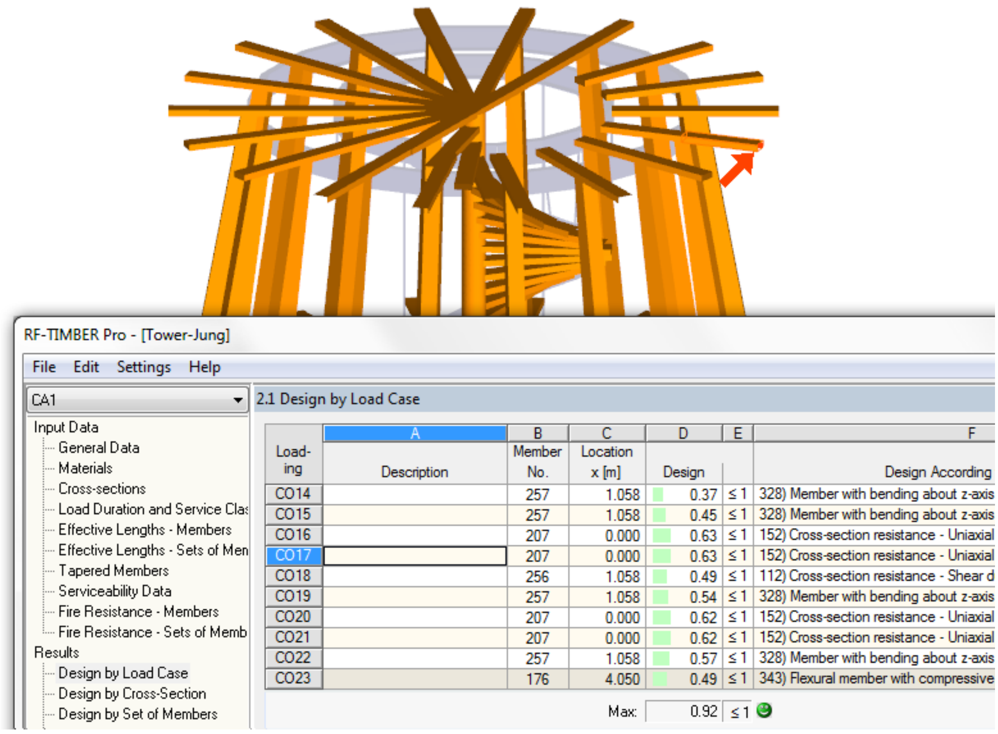
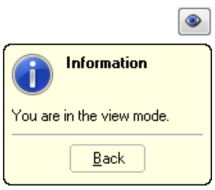


Figure 5.2 Indication of member and current *Location x* in RFEM model



In case you cannot improve the model display by moving the RF-/TIMBER Pro module window, click the [Jump to graphic] button to activate the view mode: The function will hide the module window so that you can adjust the view in the RFEM/RSTAB work window. The view mode provides the functions of the *View* menu, for example, zooming, moving or rotating the model view. The indicating arrow remains visible.

Click [Back] to return to the RF-/TIMBER Pro add-on module.

RFEM/RSTAB work window

You can check the design ratios also graphically in the model: Click the [Graphics] button to exit 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.

In the *Results* navigator, you can select the design ratios separately for the ultimate and the serviceability limit state design as well as the fire protection design.

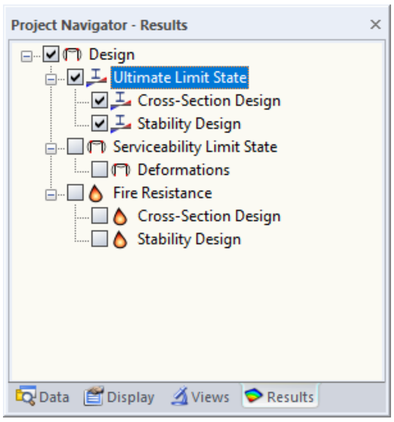
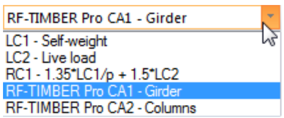


Figure 5.3 *Results* navigator for RF-/TIMBER Pro

Similar to the display of internal forces, the [Show Results] button shows or hides the display of the design results. Click the [Show Result Values] button to the right in order to display the result values.

The RFEM/RSTAB tables are not relevant for the evaluation of the design results.



You can set the design cases in the drop-down list of the RFEM/RSTAB menu bar.

To adjust the graphical representation of results, you can use the **Results → Members** entry in the *Display* navigator. The display of design ratios is *Two-Colored* by default.

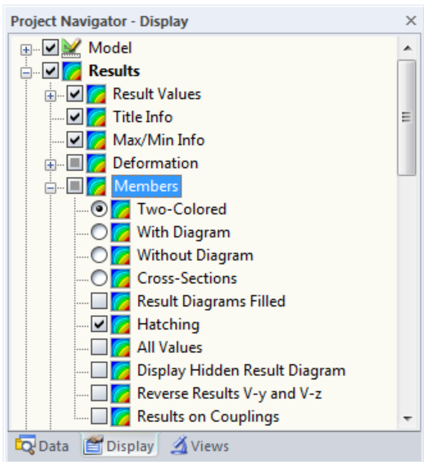


Figure 5.4 *Display* navigator: *Results → Members*

If you select a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color panel becomes available, providing common control functions (see Figure 5.5). The functions are described in Chapter 3.4.6 of the RFEM or RSTAB manual.

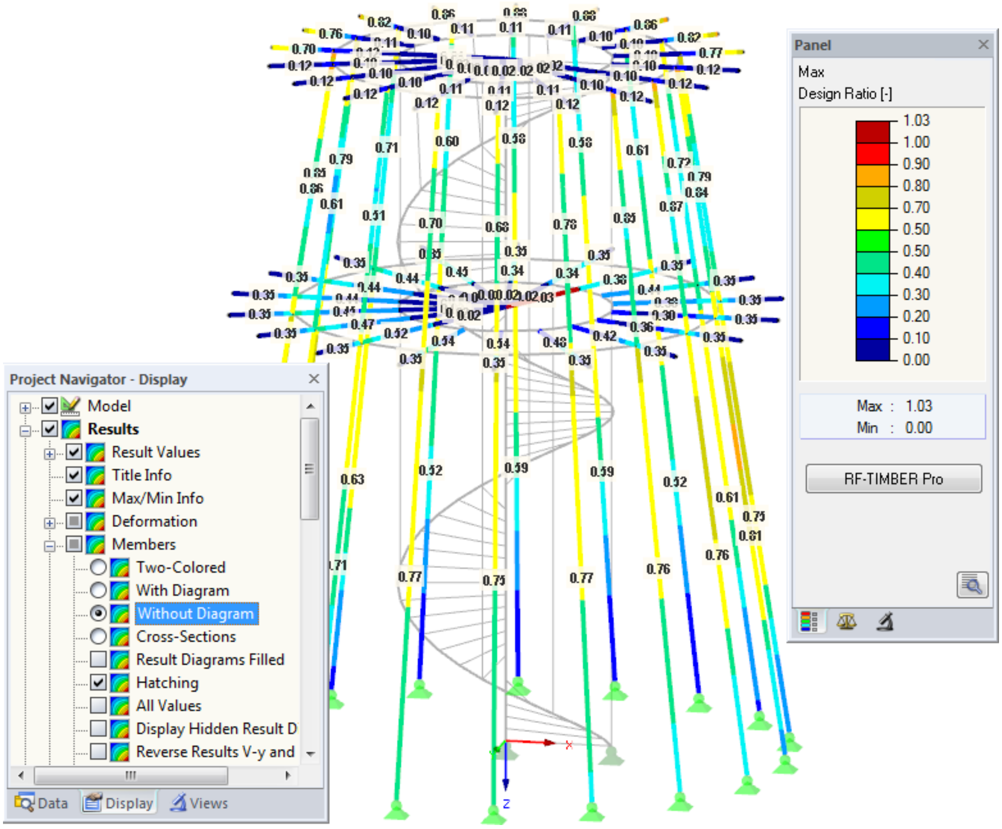


Figure 5.5 Design ratios with display option *Without Diagram*

It is possible to transfer the graphics of design results to the printout report (see Chapter 6.1).

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

5.2 Results on Cross-Section

In result Windows 2.1 to 2.5, the table results are illustrated by a dynamic stress graphic: This graphic shows the stress diagram on the cross-section at the current x-location for the selected design type. If a different x-location or design type is selected in the table, the display is updated.

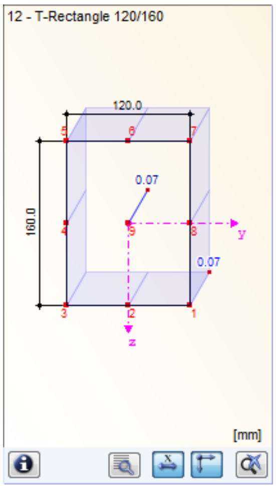


Figure 5.6 Diagram of normal stresses on cross-section

The display can be zoomed in or out by using the wheel button of the mouse. To move the stress graphic, use the drag-and-drop function. To reset the full view, click [Show All Graphic].

Extended display of stresses and ratios

The [Show or Print] button allows for a specific evaluation of the results for each stress point. It opens the *Cross-section* dialog box.

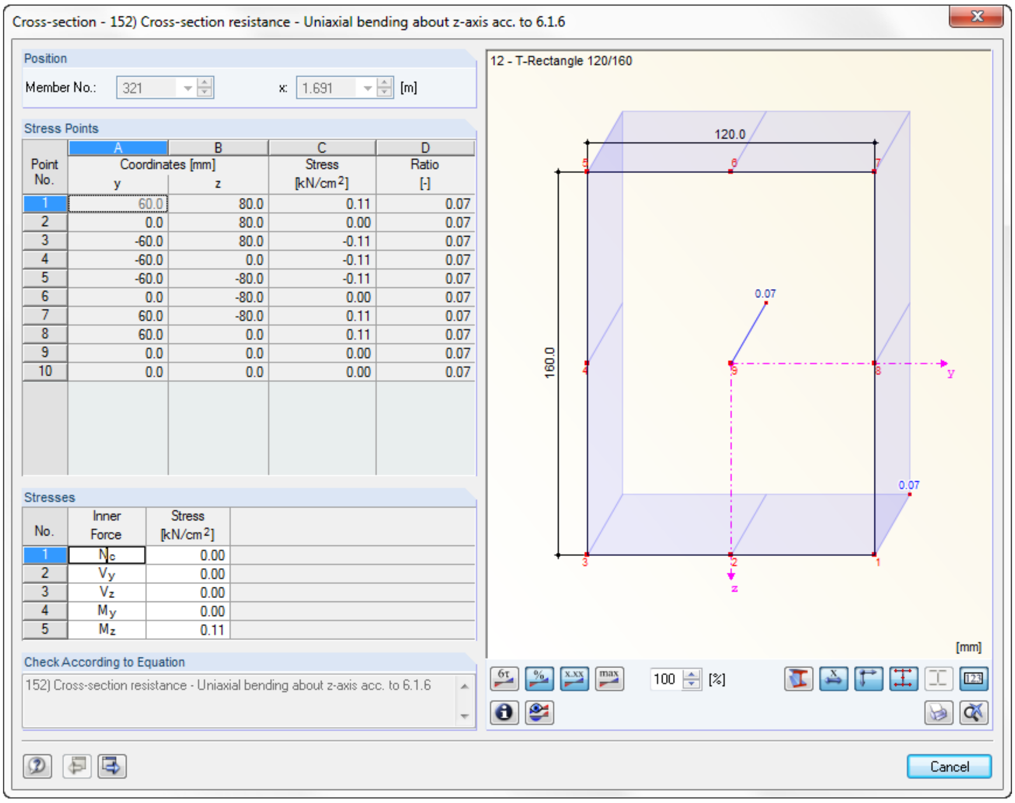


Figure 5.7 Dialog box *Cross-section*

The *Position* dialog section shows the current member number and location *x* on the member.

The *Stress Points* dialog section lists all stress points of the cross-section. A point selected in this table is highlighted in red in the graphic. The *Coordinates* columns show the centroidal distances *y* and *z*. The *Stress* column informs you about the stresses in the stress points.

The *Stresses* dialog section shows the stress components resulting from the internal forces at the current stress point (selected in the dialog section above).

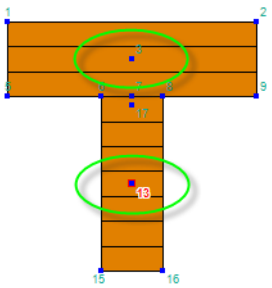
In the *Design Ratio* dialog section, the maximum ratio of the available stress to the limit stress is shown for the current location x.

Determination of shear stresses

For thin-walled cross-sections, we can assume as a simplification that the shear stress runs parallel to the wall of the cross-section. Therefore, we add the parts of the shear stresses resulting from both components of the shear forces. The sign of the statical moment defines here which parts are applied positively and which negatively.

The shear stress due to the torsional moment is to be considered differently for the total shear stress, depending on whether it is an open or closed cross-section. In the case of an open section, the torsional shear stress is added with the sign to that sum from the individual shear stresses that results in the greater absolute value of the sum.

For a closed section, however, the torsional shear stress is simply added to the sum resulting from the individual shear stresses. Here, the signs for core area and statical moments are set in such a way that they correspond to the program-specific sign conventions of the shear stress that is dependent on the loading.



Twin stress points

Stress points lying within the cross-section do not permit the assumption mentioned above that the shear stress runs parallel to the wall of the cross-section. Here, a special method with twin stress points is used creating two stress points with identical coordinates in the cross-section. The one stress point considers the statical moment about the **y**-axis (parameter for shear stress due to vertical shear force according to Equation 5.1), the other considers the statical moment about the **z**-axis (parameter for shear stress due to horizontal shear force). For these stress points, the complementary statical moment is equal to zero.

$$\tau = \frac{V_z \cdot S_y}{I_y \cdot t}$$

Equation 5.1 Shear stress due to shear force V_z

Different thicknesses can be assigned to the twin stress points, which also have an influence on the calculation of the shear stress. The shear stresses are considered as interdependent components acting perpendicular to each other: they are two components of one stress state. For the determination of the total shear stress, both parts are quadratically added. The shear stress due to the torsional moment is not considered in these points.

The shear stresses of result combinations available in the twin stress points may **not** be combined linearly. Therefore, the extreme values of both components are evaluated with the corresponding complementary shear stresses in order to determine the greatest total shear stress.

In the graphic, it is possible to show stresses as well as stress ratios.

The buttons below the graphic (see Figure 5.7) have the following functions:

Button	Description	Function
	Stress diagram	Shows or hides the display of stresses
	Stress ratio	Shows or hides the display of ratios
	Values	Shows or hides the results values
	Maximum values	Shows only extreme values or values in all points
100 [%]	Exaggeration	Enables scaling of the result diagrams
	Cross-section	Shows or hides the filled cross-section
	Dimension	Shows or hides the dimension lines
	Axes	Shows or hides the principle axes of the cross-section
	Stress points	Shows or hides the stress points
	Numbering	Shows or hides the numbers of stress points
	Info	Opens the *Info About Cross-Section* dialog box
	Result diagrams	Opens the *Result Diagram on Member* window
	Print	Allows for printing current result graphic
	Show all graphic	Resets the full view of result graphic

Table 5.2 Buttons in Cross-section dialog box

5.3 Result Diagrams

You can evaluate the member results also graphically in the form of result diagrams.

Select the member (or set of members) in the RF-/TIMBER Pro result window by clicking in the member's table row. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. You can find it below the upper result table (see Figure 5.1).

To access the result diagrams in the RFEM/RSTAB graphic, select on the menu

Results → Result Diagrams for Selected Members

or use the corresponding button in the toolbar of RFEM or RSTAB.

A window opens which graphically shows the distribution of the design values on the member or set of members.

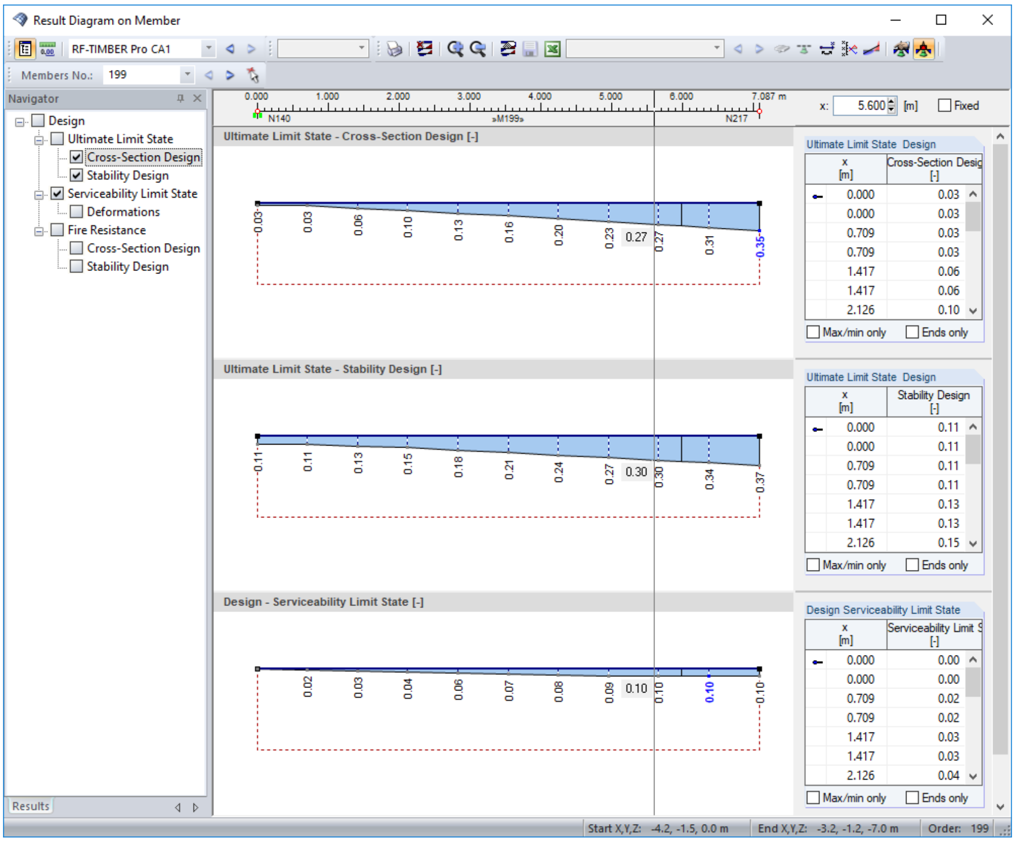
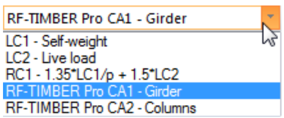


Figure 5.8 Dialog box *Result Diagram on Member*

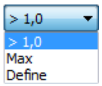
Again, the *Results* navigator allows for a targeted selection among the designs of the ultimate and the serviceability limit state as well as of fire resistance.



Use the list in the toolbar to switch between the RF-/TIMBER Pro design cases.

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

5.4 Filter for Results

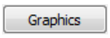


The arrangement of the RF-/TIMBER Pro result windows already provides a selection by various criteria. In addition, there are filter options for the tables (see Figure 5.1) in order to limit the numerical output by design ratios. This function is also described in the Knowledge Base on our website.

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

The possibilities offered by the *Visibility* function (see Chapter 9.9.1 in RFEM manual and Chapter 9.7.1 in RSTAB manual) are also available for RF-/TIMBER Pro in order to filter the members for the evaluation.

Filtering designs



The design ratios can easily be used as filter criteria in the work window of RFEM or RSTAB that you can access with the [Graphics] button. To apply this function, the panel must be displayed. If it is not active, select on the RFEM/RSTAB menu

View → Control Panel (Color Scale, Factors, Filter)

or use the corresponding toolbar button.

The panel is described in Chapter 3.4.6 of the RFEM or RSTAB manual. The filter settings for the results must be defined in the first panel tab (Color Scale). As this tab is not available for the two-colored results display, you have to set the display options *With/Without Diagram* or *Cross-Sections* in the *Display* navigator.

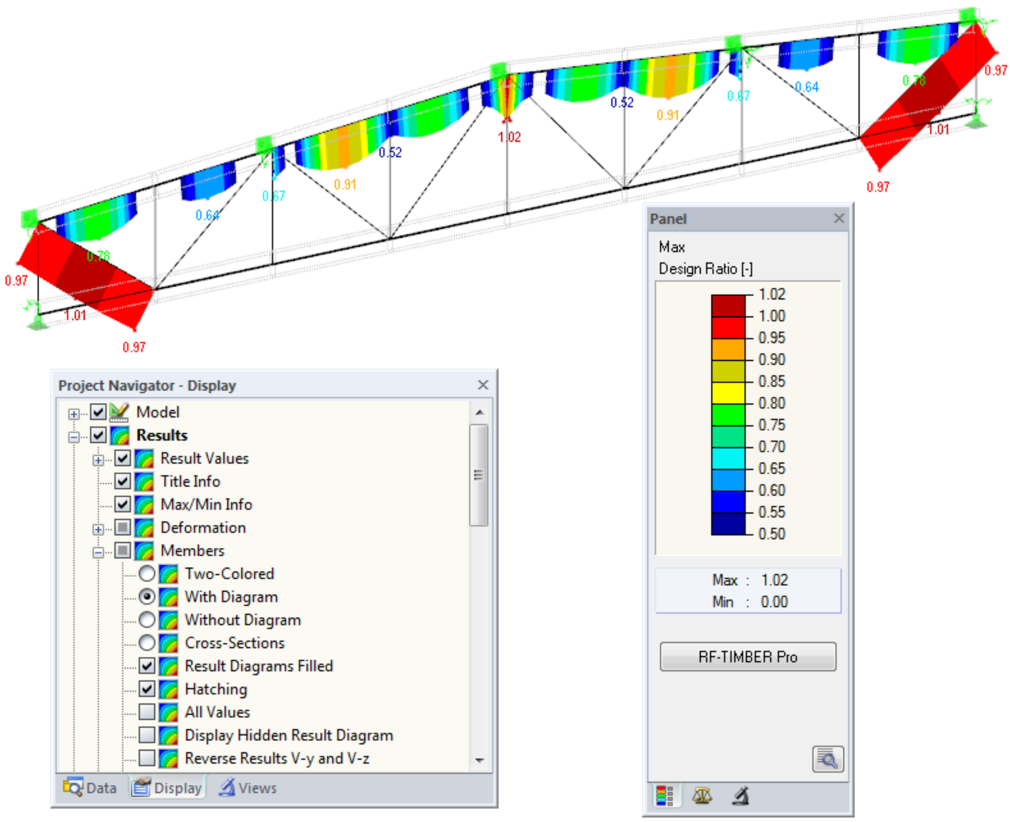


Figure 5.9 Filtering design ratios with adjusted color scale

As shown in Figure 5.9, the panel's scale of values can be set in such a way that only design ratios greater than 0.50 are displayed in a color range between blue and red.

The function *Display Hidden Result Diagram* in the *Display* navigator (**Results → Members** shows all design ratios which are beyond the value spectrum. Those diagrams are represented by dotted lines.

Filtering members

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

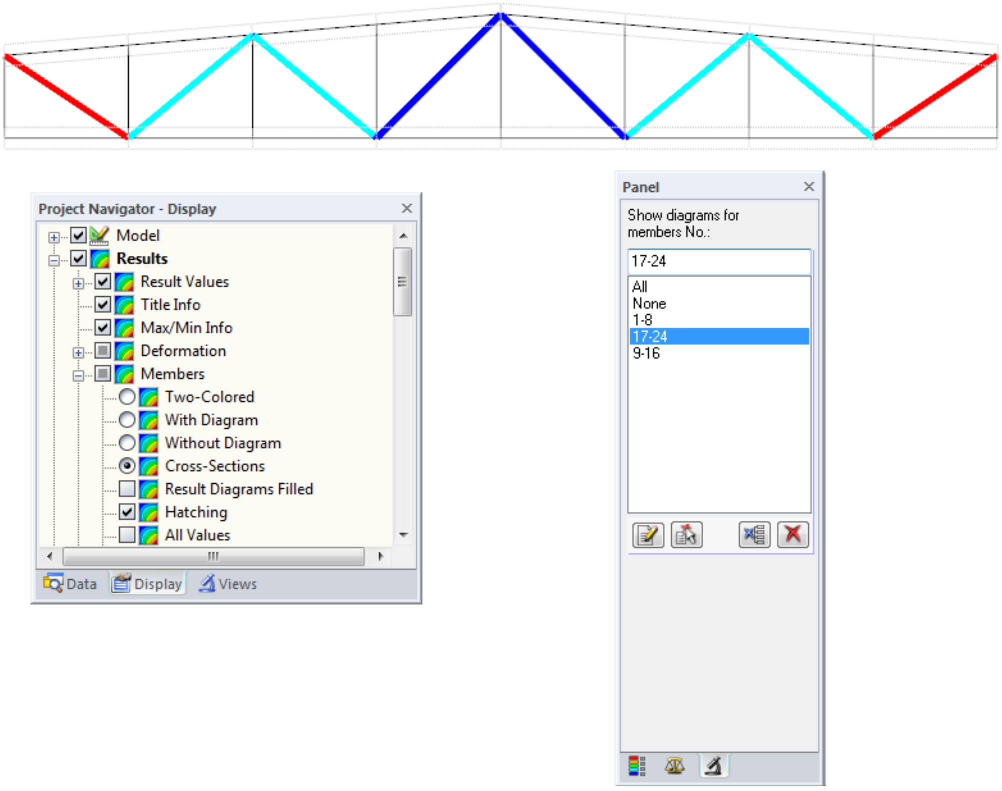


Figure 5.10 Member filter for ratios of diagonals

In contrast to the visibility function, the model will be displayed completely in the graphic. The figure above shows the design ratios for the diagonals of a truss. The remaining members are shown in the model but are displayed without design ratios.

6 Printout

6.1 Printout Report

A printout report is generated for the data of the RF-/TIMBER Pro add-on module, like in RFEM or RSTAB, to which you can add graphics and descriptions. The selection in the printout report determines which data from the design module will be included in the printout.

The printout report is described in the RFEM and RSTAB manual. Chapter 10.1.3.5 *Selecting Data of Add-on Modules* explains how to prepare input and output data of add-on modules for the printout.

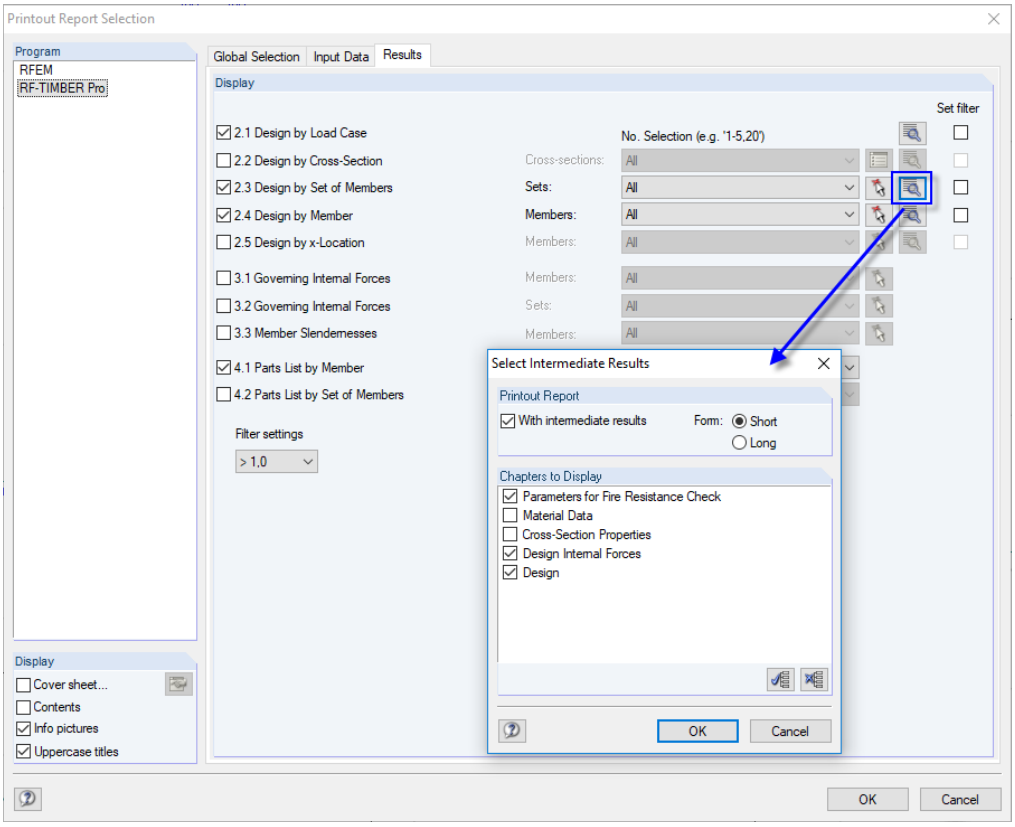


Figure 6.1 Selecting designs and intermediate results in the printout report

Click the [Details] button to specify if the printout also includes intermediate results. They can be defined in a list and documented in a *Short* (compact representation) or *Long* form (list representation).

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

6.2 Graphic Printout

In RFEM and RSTAB, you can transfer every image displayed in the work window to the printout report. It is also possible to send it directly to the printer. Thus, the design ratios displayed in the model can be prepared for the printout, too.

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

Designs in RFEM/RSTAB model

To print the current graphic of design ratios, select on the menu

File → Print Graphic

or use the corresponding toolbar button.

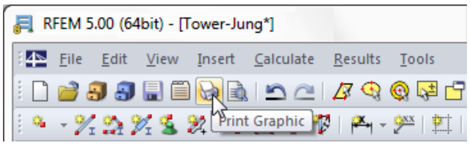


Figure 6.2 Button *Print Graphic* in RFEM toolbar

Result diagrams

Also in the *Result Diagram on Member* dialog box, you can send the graphic with design values to the report by clicking the [Print] button. Alternatively, you can print it directly.

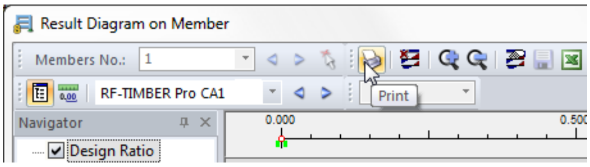


Figure 6.3 *Print* button in *Result Diagram on Member* dialog box

The following dialog box opens:

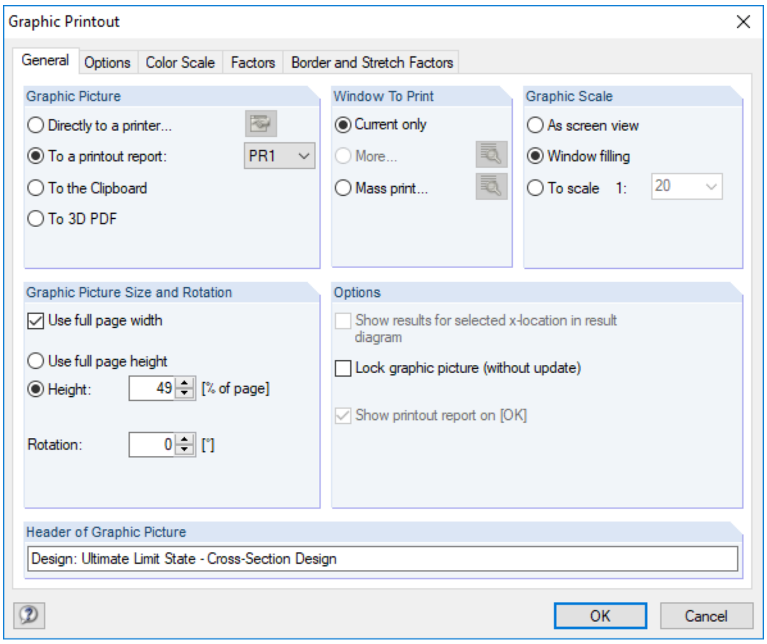
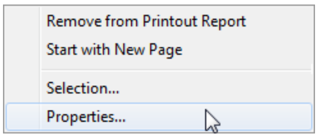


Figure 6.4 Dialog box *Graphic Printout*, tab *General*



The *Graphic Printout* dialog box is described in Chapter 10.2 of the RFEM or RSTAB manual. There, you also find descriptions of the remaining dialog tabs.

To move a graphic within the printout report to another position, use the drag-and-drop function.

To adjust a graphic subsequently in the printout report, right-click the relevant entry in the report navigator. The *Properties* option in the shortcut menu opens again the *Graphic Printout* dialog box where you can adjust the settings.

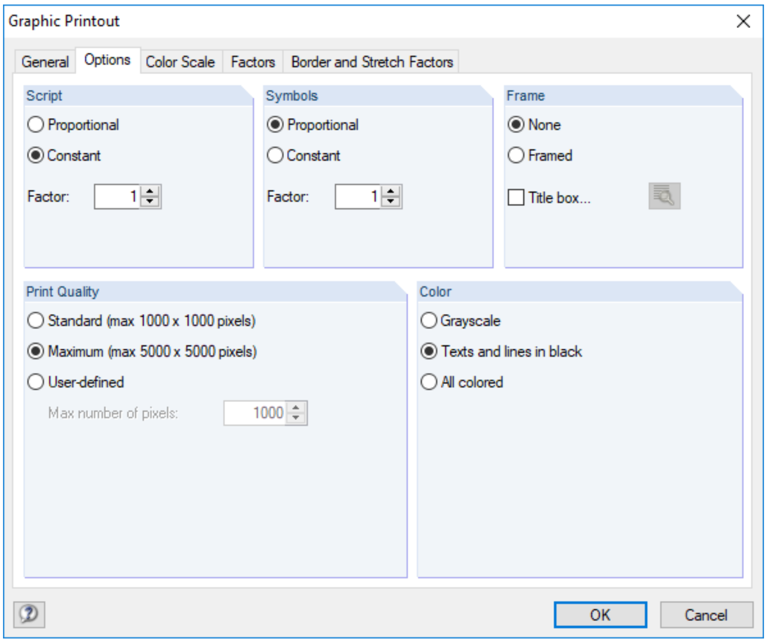


Figure 6.5 Dialog box *Graphic Printout*, tab *Options*

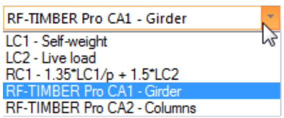
7 General Functions

This chapter describes useful menu functions as well as export options for the designs.

7.1 Design Cases

Design cases allow you to group members for the designs. This way, you can consider groups of structural components or analyze members with particular design specifications (for example, changed materials, partial safety factors, optimization).

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



You can access the design cases of RF-/TIMBER Pro also in RFEM or RSTAB by using the load case list of the toolbar.

Create a new design case

To create a new design case, select on the RF-/TIMBER Pro menu

File → New Case.

The following dialog box appears.

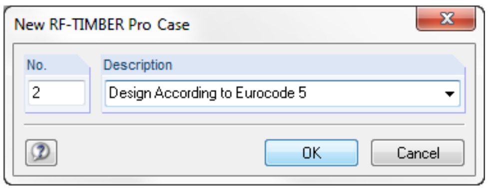


Figure 7.1 Dialog box *New RF-TIMBER Pro Case*

In this dialog box, enter a *No.* (one that is not yet assigned) for the new design case. A *Description* will make the selection in the load case list easier.

After clicking [OK], the RF-/TIMBER Pro Window *1.1 General Data* opens for entering the design data.

Rename a design case

To change the description of a design case, select on the RF-/TIMBER Pro menu

File → Rename Case.

The following dialog box appears.

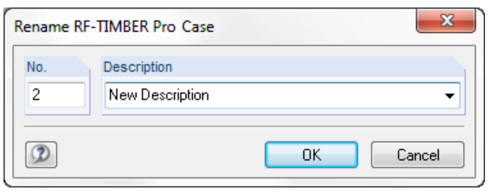


Figure 7.2 Dialog box *Rename RF-TIMBER Pro Case*

In this dialog box, you can specify a different *Description* as well as a different *No.* for the design case.

Copy a design case

To copy the input data of the current design case, select on the RF-/TIMBER Pro menu

File → Copy Case.

The following dialog box appears.

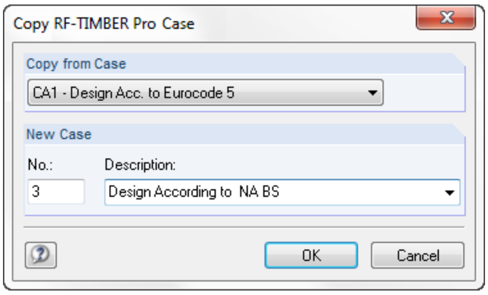


Figure 7.3 Dialog box *Copy RF-TIMBER Pro Case*

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

Delete a design case

To delete a design case, select on the RF-/TIMBER Pro menu

File → Delete Case.

The following dialog box appears.

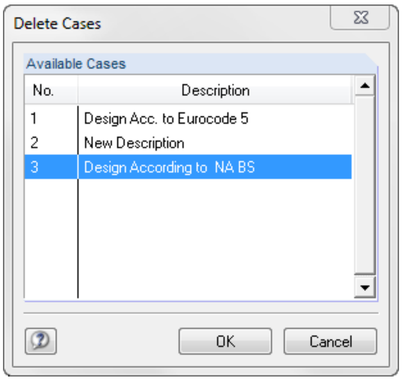


Figure 7.4 Dialog box *Delete Cases*

You can select the design case in the list of *Available Cases*. To delete the selected case, click [OK].

7.2 Cross-Section Optimization

RF-/TIMBER Pro allows you to optimize overloaded or little utilized cross-sections. However, this is possible **only** for rectangular and circular cross-sections. In the case of built-up cross-sections, an automatic optimization would not be economical due to the large number of parameters, and problematic due to the slips.

In the *1.3 Cross-Sections* window, you can define the sections for optimization by selecting the *Yes* entry in the drop-down list box of column C (or D) (see Figure 2.18). You can also start the optimization in the result windows by using the shortcut menu.

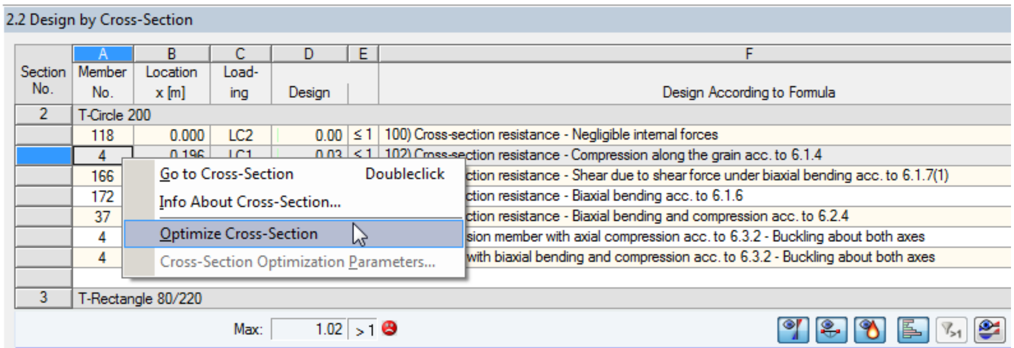


Figure 7.5 Shortcut menu for cross-section optimization

Details...

During the optimization process, the program finds the cross-section that fulfills the **ultimate limit state** design in the most "optimal way", that means comes as close as possible to the maximum allowable ratio specified in the *Details* dialog box (see Figure 3.5). The required cross-section properties are determined with the internal forces and moments as available in RFEM or RSTAB. If another cross-section proves to be more favorable, this new cross-section is used for the design. Then, the graphic in Window 1.3 shows two cross-sections — the original cross-section from RFEM or RSTAB and the optimized cross-section (see Figure 7.7).

After activating the *Optimize* function, the following dialog box appears:

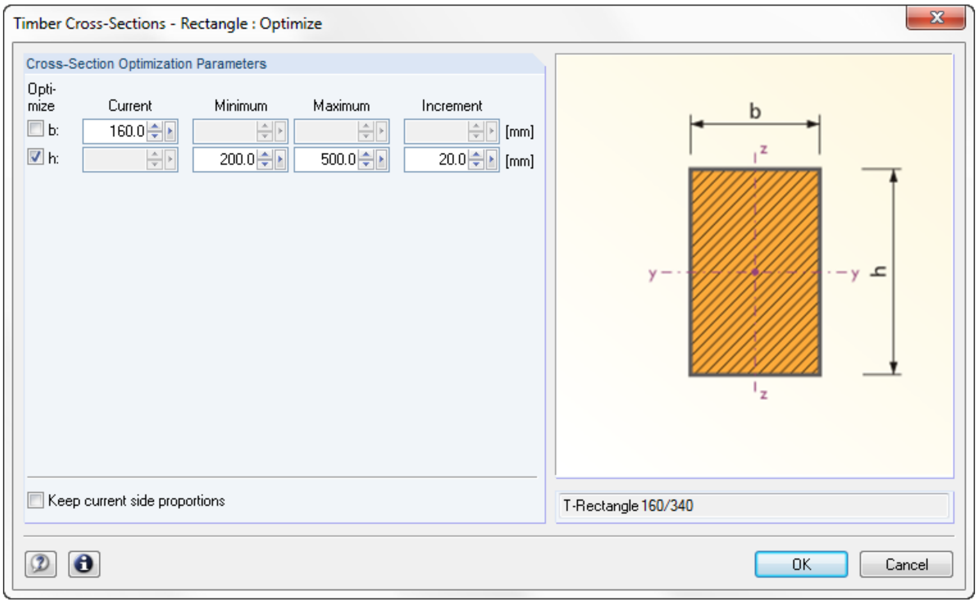


Figure 7.6 Dialog box *Timber Cross-Sections - Rectangle : Optimize*

You can determine the parameter(s) that you want to modify by ticking the *Optimize* check box(es). This enables the *Minimum* and *Maximum* columns where you can define the upper and lower limits of the parameter. The *Increment* column controls the interval in which the size of the parameter varies during the optimization process.

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

Please note that during the optimization the internal forces won't be automatically recalculated with the modified cross-sections: It is up to you to decide which cross-sections should be transferred to RFEM or RSTAB for recalculation. As a result of optimized cross-sections, the internal forces may differ significantly because of the modified stiffnesses in the structural system. Therefore, it is recommended to recalculate the internal forces with the modified cross-sections after the first optimization, and then to optimize the cross-sections once again.

You can export the modified cross-sections to RFEM or RSTAB: Go to the *1.3 Cross-Sections* window and select on the menu

Edit → Export All Cross-Sections to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.3 to export optimized cross-sections to RFEM or RSTAB.

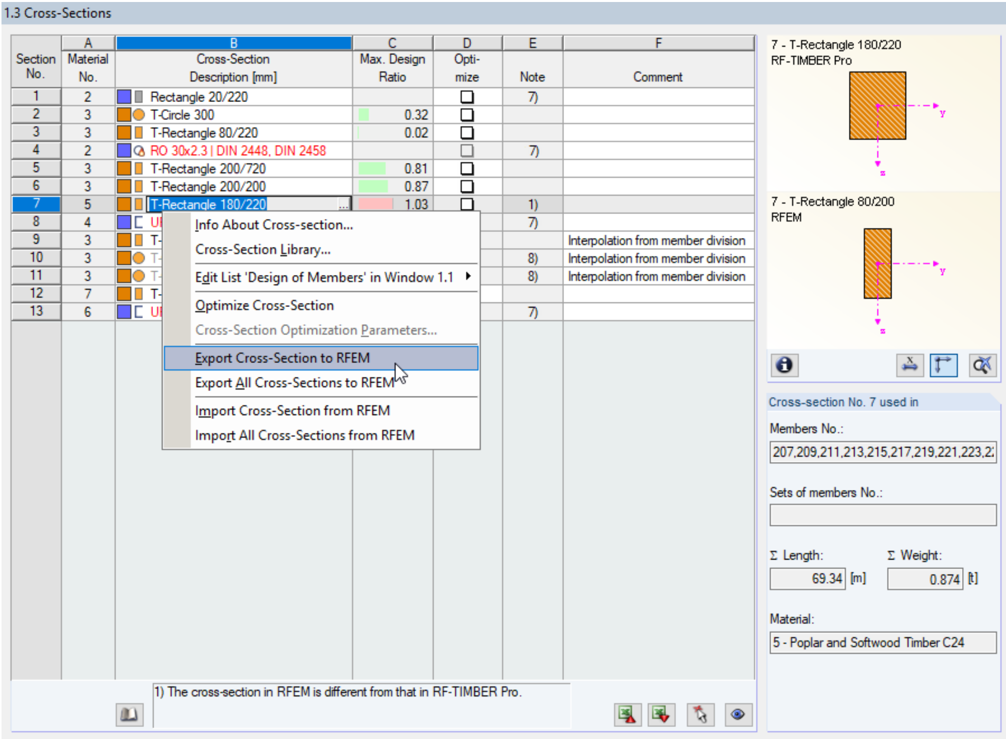


Figure 7.7 Shortcut menu in Window *1.3 Cross-Sections*

Before the modified cross-sections are transferred, a query appears asking if the results of RFEM or RSTAB should be deleted.

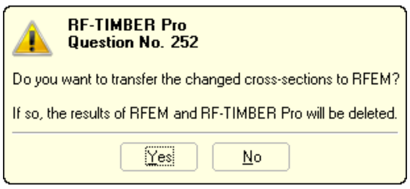


Figure 7.8 Query before transfer of modified cross-sections to RFEM

Calculation

After starting the [Calculation] in RF-/TIMBER Pro, the internal forces and design ratios are determined in one calculation run.

If the modified cross-sections have not yet been exported to RFEM or RSTAB, it is possible to reimport the original cross-sections to the design module by using the options shown in Figure 7.7. Please note that this possibility is only available in the *1.3 Cross-Sections* window.

If you optimize a tapered member, the program modifies the member start and end. Then, it linearly interpolates the second moments of area for the intermediate locations. Since these moments are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-section differ considerably. In such a case, it is recommended to divide the taper into several members, thus modeling the taper layout manually.

7.3 Units and Decimal Places

The units and decimal places are managed for RFEM/RSTAB and the add-on modules in one dialog box. In RF-/TIMBER Pro, you can access this dialog box for adjusting the units by selecting on the menu

Settings → Units and Decimal Places.

The dialog box known from RFEM or RSTAB appears. The RF-/TIMBER Pro add-on module is preset in the *Program / Module* list.

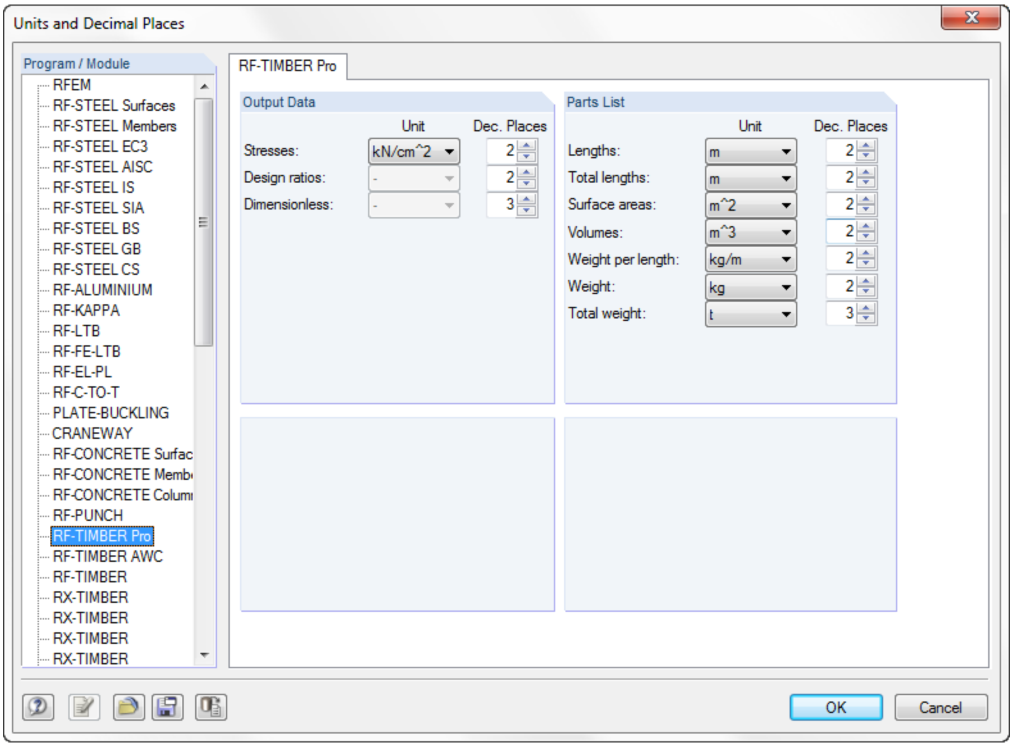


Figure 7.9 Dialog box *Units and Decimal Places*

The modified settings can be saved as user profile and reused in other models. The functions are described in Chapter 11.1.3 of the RFEM or RSTAB manual.

7.4 Data Exchange

7.4.1 Exporting Materials to RFEM/RSTAB

If the materials have been adjusted for the design in RF-/TIMBER Pro, you can export the modified materials to RFEM or RSTAB in a similar way as you export cross-sections: Open the *1.2 Materials* window, and then select on the menu

Edit → Export All Materials to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.2 to export materials to RFEM/RSTAB.

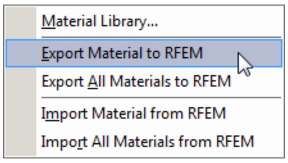
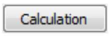


Figure 7.10 Shortcut menu of Window *1.2 Materials*



Before the modified materials are transferred, a query appears asking if the results of RFEM or RSTAB should be deleted. After starting the [Calculation] in RF-/TIMBER Pro, the internal forces and design ratios are determined in one calculation run.

If the modified materials have not yet been exported to RFEM or RSTAB, it is possible to reimport the original materials to the design module by using the options shown in Figure 7.10. Please note that this possibility is only available in the *1.2 Materials* window.

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

If the effective lengths have been adjusted for the designs in RF-/TIMBER Pro, you can also export these modified lengths to RFEM or RSTAB: Go to the *1.5 Effective Lengths - Members* window, and then select on the menu

Edit → Export All Effective Lengths to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.5 to export effective lengths to RFEM/RSTAB.

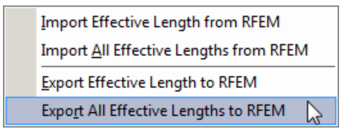


Figure 7.11 Shortcut menu of Window *1.5 Effective Lengths - Members*

Before the modified effective lengths are transferred, a query appears asking if the results of RFEM/ RSTAB should be deleted.

If the modified effective lengths have not yet been exported to RFEM or RSTAB, it is possible to reimport the original effective lengths to the design module by using the options shown in Figure 7.11.

7.4.3 Exporting Results

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

Clipboard

To copy cells selected in the results windows to the clipboard, use the keys [Ctrl]+[C]. To insert them, for example, in a word processing program, press [Ctrl]+[V]. The headers of the table columns won't be transferred.

Printout report

The data of RF-/TIMBER Pro can be printed into the printout report (see Chapter 6.1) where it can be exported. Then, in the printout report, select on the menu

File → Export to RTF.

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

Excel

RF-/TIMBER Pro provides a function for directly exporting data to MS Excel or the CSV file format. To access this function, select on the menu

File → Export Tables.

The following export dialog box opens.

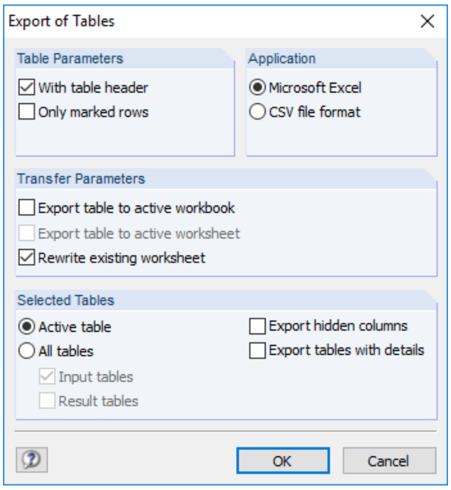
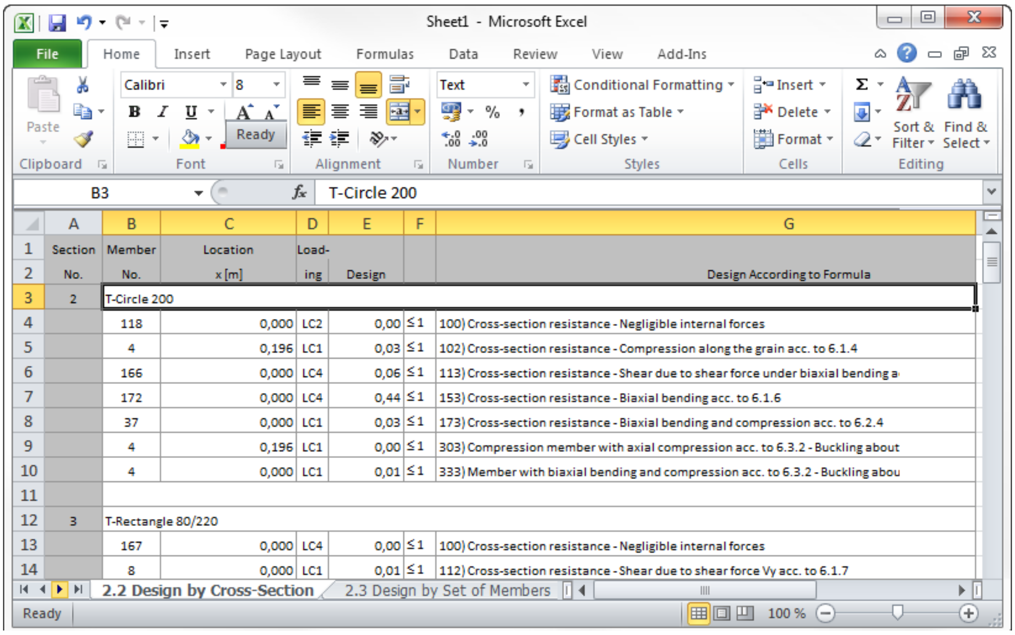


Figure 7.12 Dialog box *Export of Tables*

When you have selected the relevant data, you can start the export with [OK]. Excel will be started automatically, that is, you do not need to open the program first.



Section No.	Member No.	Location x [m]	Load-ing	Design		Design According to Formula
2	T-Circle 200					
	118	0,000	LC2	0,00	≤ 1	100) Cross-section resistance - Negligible internal forces
	4	0,196	LC1	0,03	≤ 1	102) Cross-section resistance - Compression along the grain acc. to 6.1.4
	166	0,000	LC4	0,06	≤ 1	113) Cross-section resistance - Shear due to shear force under biaxial bending a
	172	0,000	LC4	0,44	≤ 1	153) Cross-section resistance - Biaxial bending acc. to 6.1.6
	37	0,000	LC1	0,03	≤ 1	173) Cross-section resistance - Biaxial bending and compression acc. to 6.2.4
	4	0,196	LC1	0,00	≤ 1	303) Compression member with axial compression acc. to 6.3.2 - Buckling about
	4	0,000	LC1	0,01	≤ 1	333) Member with biaxial bending and compression acc. to 6.3.2 - Buckling abou
3	T-Rectangle 80/220					
	167	0,000	LC4	0,00	≤ 1	100) Cross-section resistance - Negligible internal forces
	8	0,000	LC1	0,01	≤ 1	112) Cross-section resistance - Shear due to shear force Vy acc. to 6.1.7

Figure 7.13 Results in *Excel*

8 Examples

8.1 Timber Column

We perform designs according to EN 1995-1-1 [1] for a timber column that is restrained as well as subjected to compression and bending, and supported on the free end in direction Y.

The example is described in the German timber construction book [6], page 236.

8.1.1 System and Loads

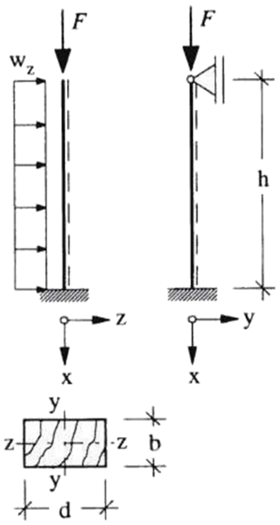


Figure 8.1 System and loads according to [6]

Model

Cross-section:	b/d = 14/22 cm
Material:	Softwood C24
Height:	h = 3.20 m
Service class:	1
LDC:	Permanent

Load

LC1 self-weight:	F = 45 kN
LC2 wind:	w = 1.5 kN/m

Design values

$N_d = 1.35 \cdot F = 1.35 \cdot 45 \text{ kN} = 60.75 \text{ kN} \; (k_{mod} = 0.6)$

$q_d = 1.5 \cdot w = 1.5 \cdot 1.5 \text{ kN/m} = 2.25 \text{ kN/m} \; (k_{mod} = 0.9)$

8.1.2 Calculation with RFEM/RSTAB

The system as well as the loads in both load cases are modeled as a 3D model in RFEM or RSTAB. We deactivate the automatic consideration of the self-weight when we create LC1 because it is also neglected in the example of the German timber construction book.

We superimpose the load cases for the fundamental combination according to the geometrically linear analysis with the corresponding partial safety factors in a result combination. It is important for the designs in RF-/TIMBER Pro to define both load cases with the "permanent" criterion.

RFEM or RSTAB determines the diagrams of internal forces shown in the following figure.

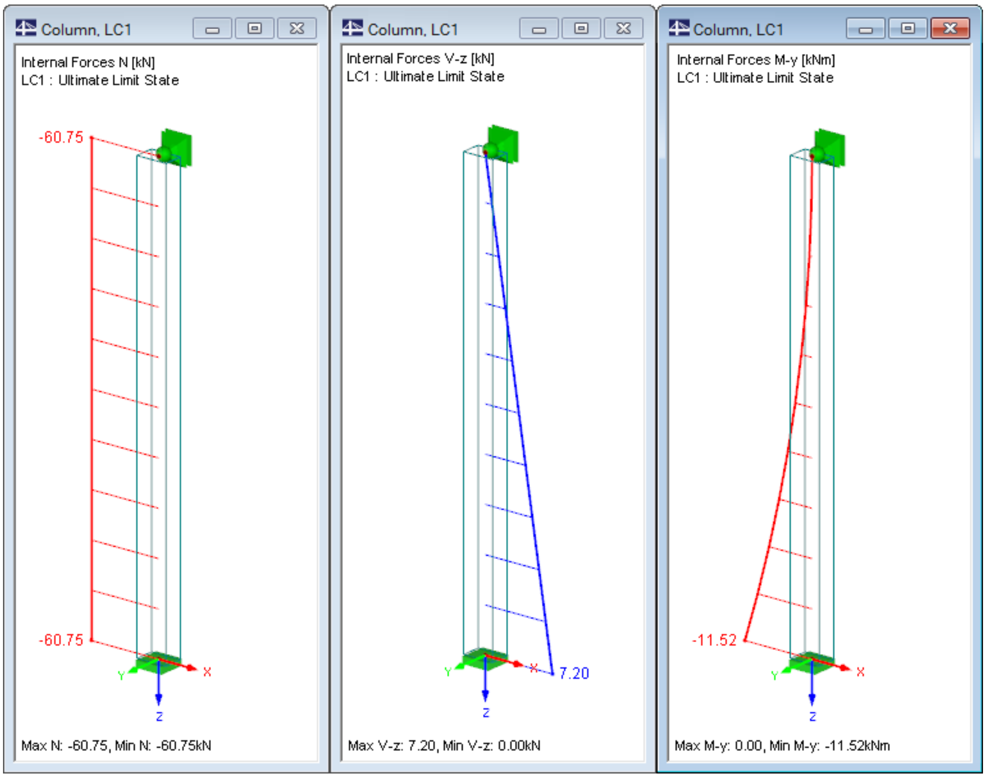


Figure 8.2 Internal forces N, V_z and M_y

The analyzed internal forces are equivalent to the ones mentioned in [6], page 237.

8.1.3 Design with RF-/TIMBER Pro

8.1.3.1 Ultimate Limit State Design

In the *1.1 General Data* window, we select the result combination **RC1** for the *Ultimate Limit State* design.

We perform the design according to **EN 1995-1-1** with the German National Annex **DIN** (see Figure 8.3).

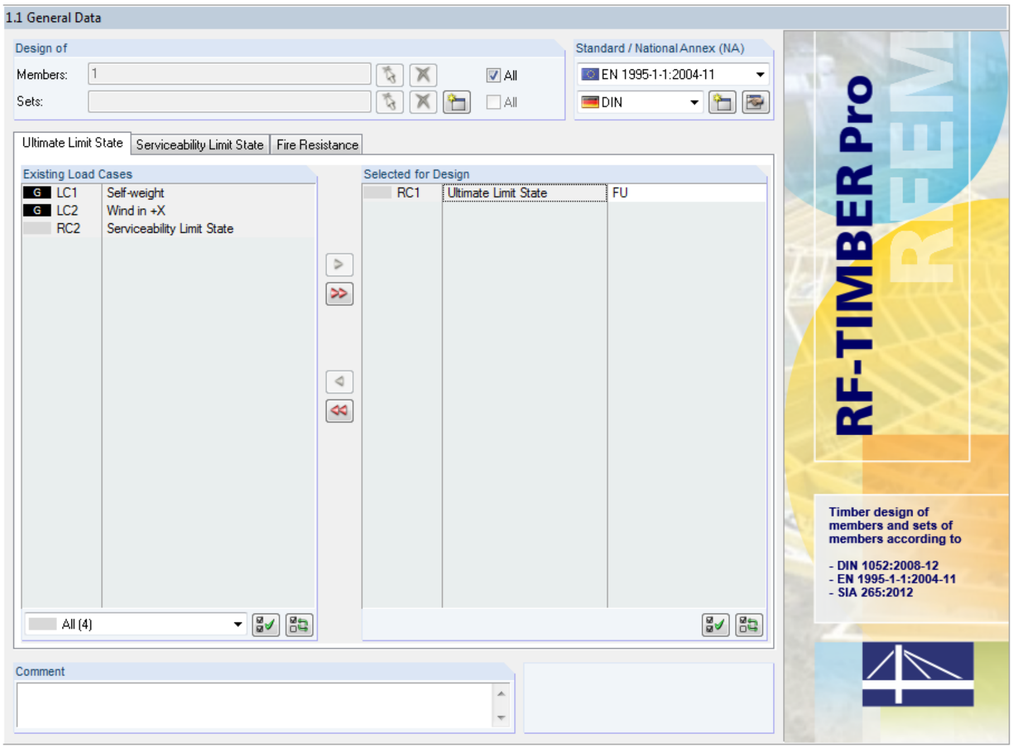


Figure 8.3 Window *1.1 General Data*

Windows *1.2 Materials* and *1.3 Cross-Sections* present the characteristic strengths of the selected material and the available cross-section.

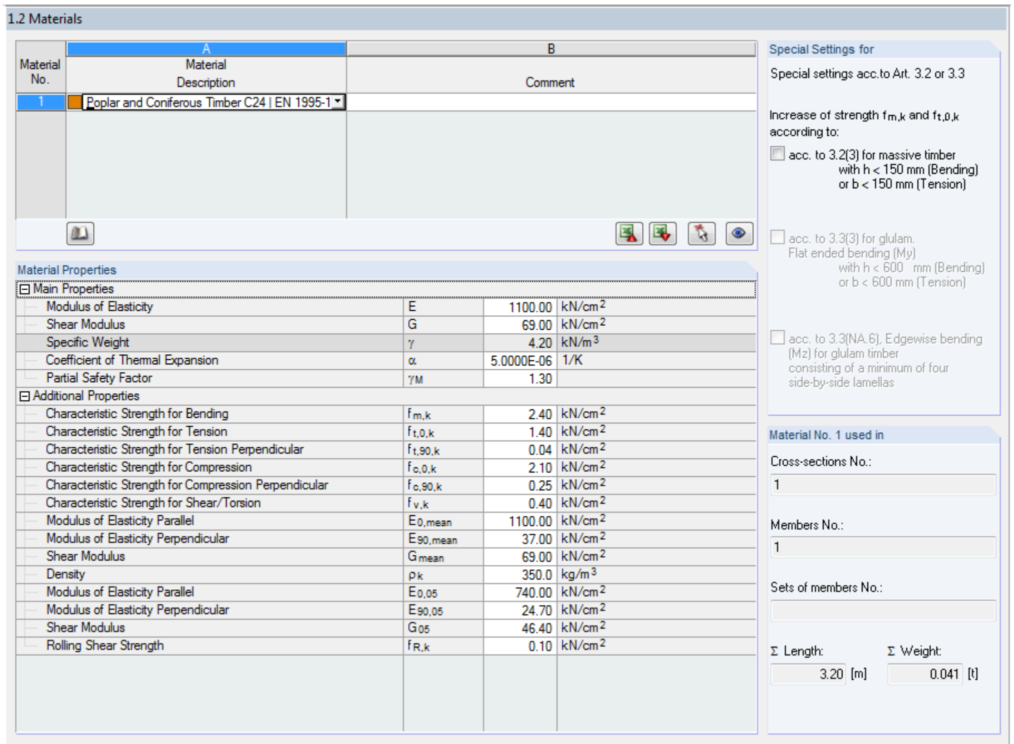


Figure 8.4 Window *1.2 Materials*

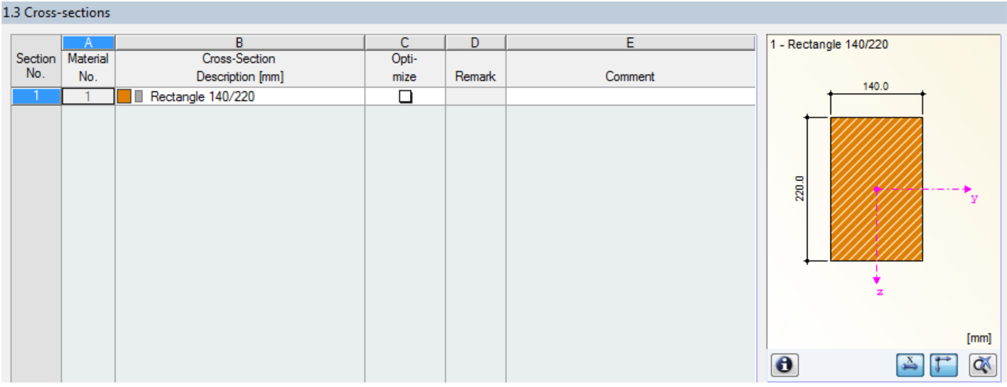


Figure 8.5 Window *1.3 Cross-sections*

In Window 1.4, we define load duration and service class. The factor k_{mod} of RC1 will be calculated from the load duration class (LDC) of the contained load cases by taking into account the service class (SECL).

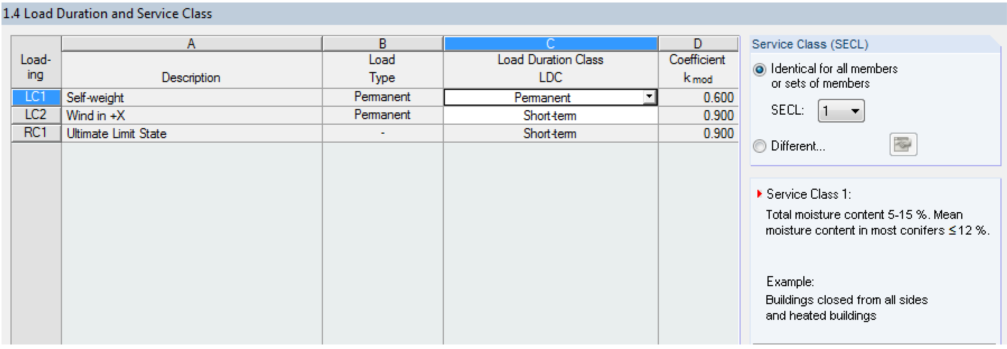


Figure 8.6 Window *1.4 Load Duration and Service Class*

In the *1.5 Effective Lengths - Members* window, we specify the buckling lengths of the column. The example provides the Euler buckling modes 1 and 3 with the buckling length coefficients $k_{cr,y}$ = **2.0** and $k_{cr,z}$ = **0.7**.

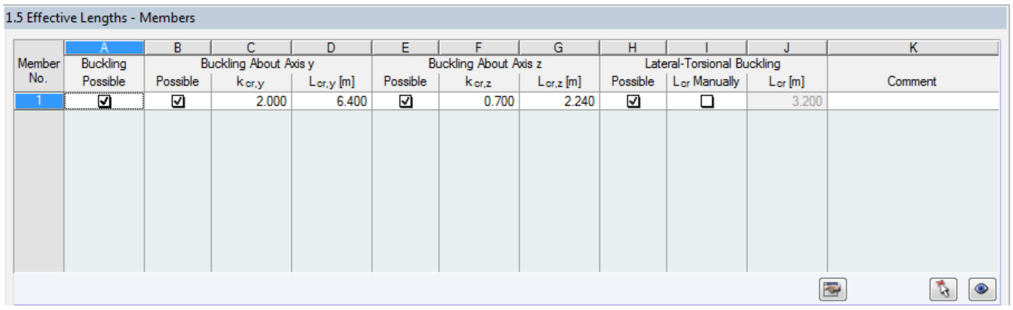


Figure 8.7 Window *1.5 Effective Lengths - Members*

We start the calculation by clicking the [Calculation] button.

After the calculation, the *2.1 Design by Load Case* window appears showing the governing designs.

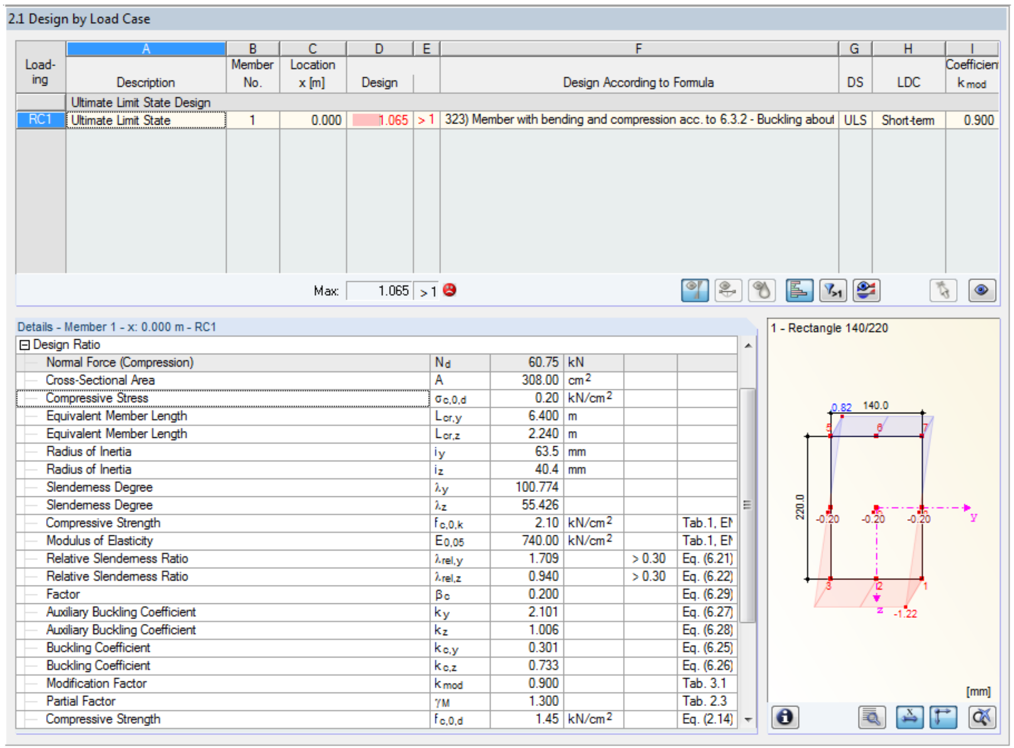


Figure 8.8 Window *2.1 Design by Load Case*

The *Details* displayed in the lower part of the window correspond to the designs described in [6].

$$\sigma_{m,d} = \frac{M_y}{W_y} = \frac{1152\ \text{kNcm}}{1129.33\ \text{cm}^3} = 1.02\ \text{kN/cm}^2$$

$$\sigma_{c,0,d} = \frac{N}{A} = \frac{60{,}75\ \text{kN}}{308\ \text{cm}^2} = 0{,}197\ \text{kN/cm}^2$$

According to [1], we have to reduce the allowable compressive stress by the buckling coefficient k_c for the stability analysis (buckling design). This coefficient depends on the slenderness ratio λ.

$$i_y = \frac{d}{\sqrt{12}} = \frac{22\ \text{cm}}{\sqrt{12}} = 6.35\ \text{cm}$$

$$i_z = \frac{b}{\sqrt{12}} = \frac{14\ \text{cm}}{\sqrt{12}} = 4.04\ \text{cm}$$

The slenderness ratio is given as:

$$\lambda_z = \frac{s_k}{i_z} = \frac{224\ \text{cm}}{4.04\ \text{cm}} = 55.4$$

$$\lambda_y = \frac{s_k}{i_y} = \frac{640\ \text{cm}}{6.35\ \text{cm}} = 100.8$$

The buckling coefficient k_c according to [1], clause 6.3.2 is (intermediate values may be interpolated linearly):

$$k_{c,z} = 0.733$$

$$k_{c,y} = 0.301$$

Stability analysis

Design according to [1] Eq. (6.23):

$$f_{m,d} = \frac{f_{m,k} \cdot k_{mod}}{\gamma_M} = \frac{2.4 \cdot 0.9}{1.3} = 1.66\ \text{kN/cm}^2$$

Design:

$$\eta_1 = \frac{\frac{N}{A}}{k_{c,y} \cdot f_{c,o,d}} + \frac{\frac{M}{W}}{k_m \cdot f_{m,y,d}} = \frac{0.197}{0.301 \cdot 1.45} + \frac{1.02}{1.66} = 1.066 > 1$$

Shear design

Design of shear from transverse force according to [1] clause 6.1.7:

$$f_{v,d} = \frac{f_{v,k} \cdot k_{mod}}{\gamma_M} = \frac{0.4 \cdot 0.9}{1.3} = 0.277\ \text{kN/cm}^2$$

The shear stresses can be determined by the stress point details with the corresponding statical moments of area (see Figure 2.23).

$$\tau_d = \frac{V_y \cdot S_{z,i}}{I_z \cdot t_i} + \frac{V_z \cdot S_{y,i}}{I_y \cdot t_i} = \frac{7.2\ \text{kN} \cdot 847\ \text{cm}^3}{5030.67\ \text{cm}^4 \cdot 22\ \text{cm}} = 0.055\ \text{kN/cm}^2$$

Design:

$$\frac{\tau_d}{f_{v,d}} = \frac{0.055}{0.277} = 0.199 \leq 1$$

8.1.3.2 Serviceability Limit State Design

We create another result combination in RFEM or RSTAB with different partial safety factors for the serviceability limit state design:

RC2 = 1.0 · LC1/s + 1.0 · LC2/s

In RF-/TIMBER Pro, we select the result combination **RC2** for the design in the *Serviceability Limit State* tab of the *1.1 General Data* window and assign the design combination **Characteristic** according to 2.2.3(2).

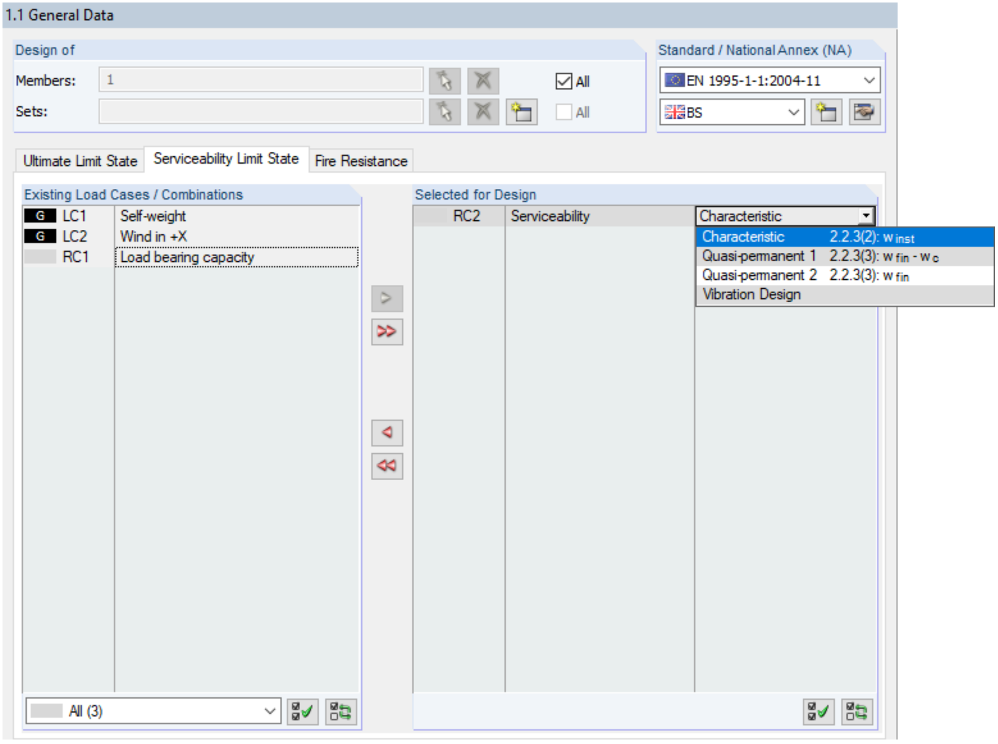


Figure 8.9 Window *1.1 General Data*, tab *Serviceability Limit State*

Then, we enter member **1** in the *1.9 Serviceability Data* window.

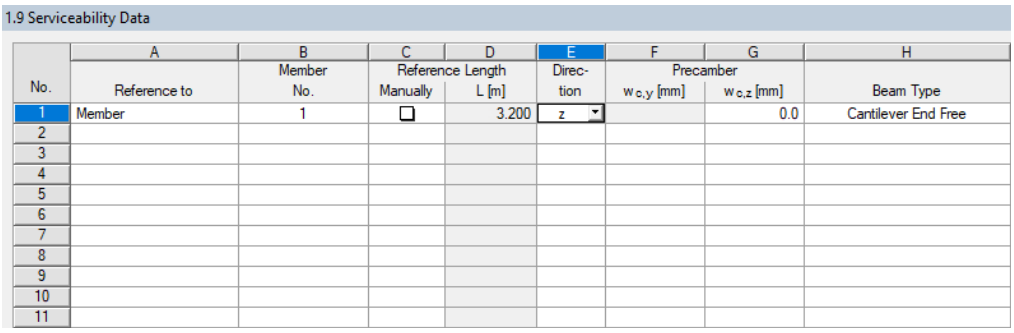


Figure 8.10 Window *1.9 Serviceability Data*

We do not modify the reference length but we restrict the direction to **z**. As the beam has no support in this direction, we select **Cantilever End Free** in the *Beam Type* list.

Details...

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

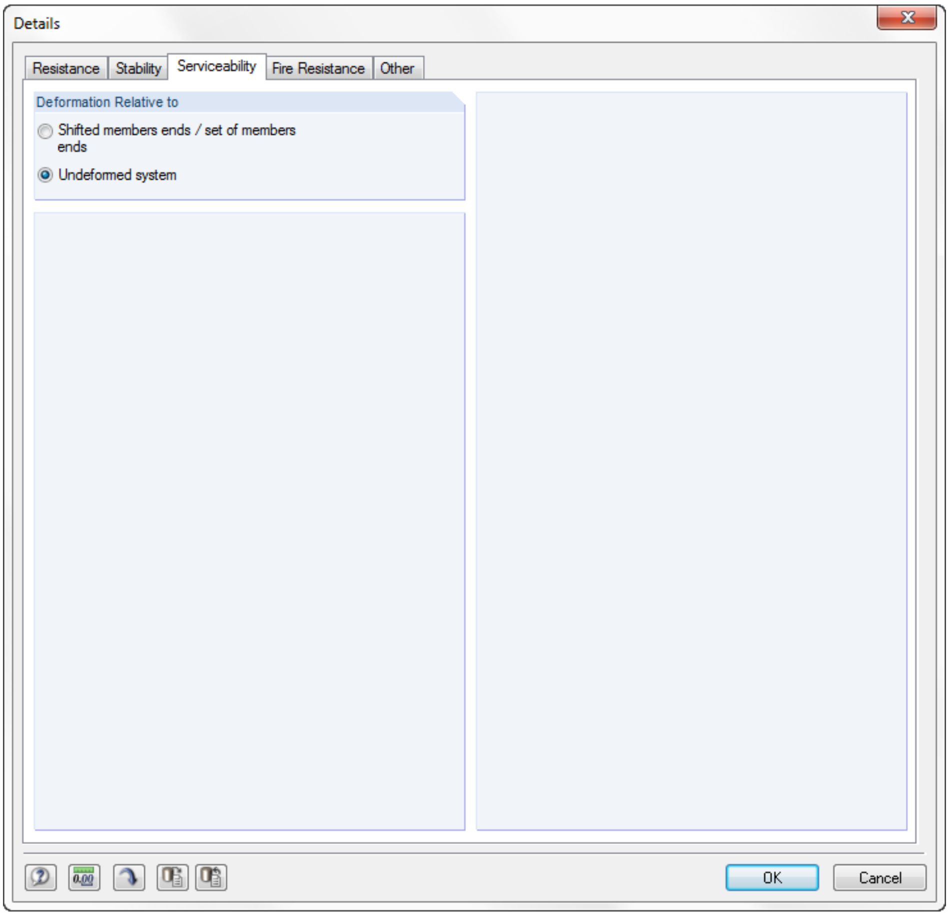


Figure 8.11 Dialog box *Details*, tab *Serviceability*

In [6], a modulus of elasticity of 10 000 MN/m^2 is applied. Thus, a new material with corresponding characteristics would have to be defined in RFEM or RSTAB.

However, to simplify matters, we use the default value of 11 000 MN/m^2 for the following equation.

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

$$w_{inst} = \frac{1.5 \cdot 3.2^4}{8 \cdot 11000 \cdot 12422.70} \cdot \frac{10^{-1}}{10^{-8}} = 1.44\ \text{cm} < 2.13\ \text{cm} = \frac{320}{150}$$

Design:

$$\frac{w_{inst}}{w_{zul}} = \frac{1.44\ \text{cm}}{2.13\ \text{cm}} = 0.676 < 1$$

The result of this deformation analysis is also displayed after the [Calculation] in the *2.1 Design by Load Case* result window under the table entry *Serviceability Limit State Design*.

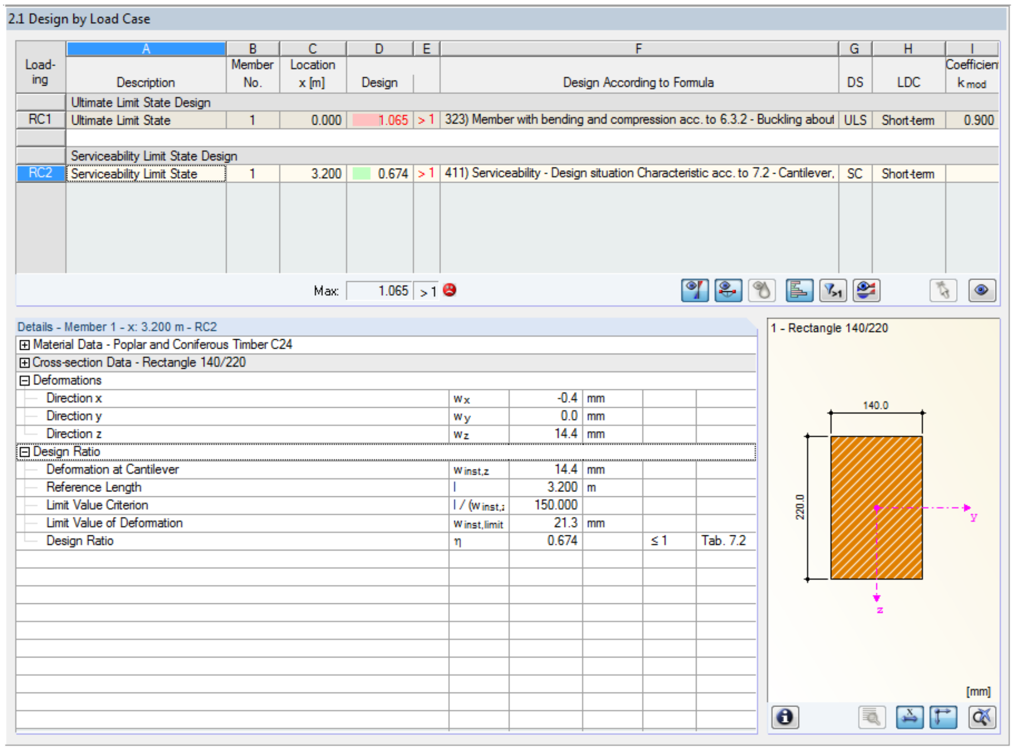


Figure 8.12 Window 2.1 *Design by Load Case*

8.2 Built-up Cross-Section

We perform the designs according to EN 1995-1-1 for a single-span beam with a length of 6.50 m. This example is taken from the lecture notes of the University of Wismar [7] (example 5.1).

8.2.1 System and Loads

The beam consists of softwood C30 and is built up of three identical square timbers 80 mm / 180 mm with nails. The parts of the cross-section subjected to compression are held in the middle against lateral displacement ($l_{ef,z}$ = 3.25 m).

Model

Material:	Softwood C30
Span length:	l = 6.50 m
Service class:	1
LDC:	Medium-term

Loading

LC1 imposed load:	q = 2.70 kN/m

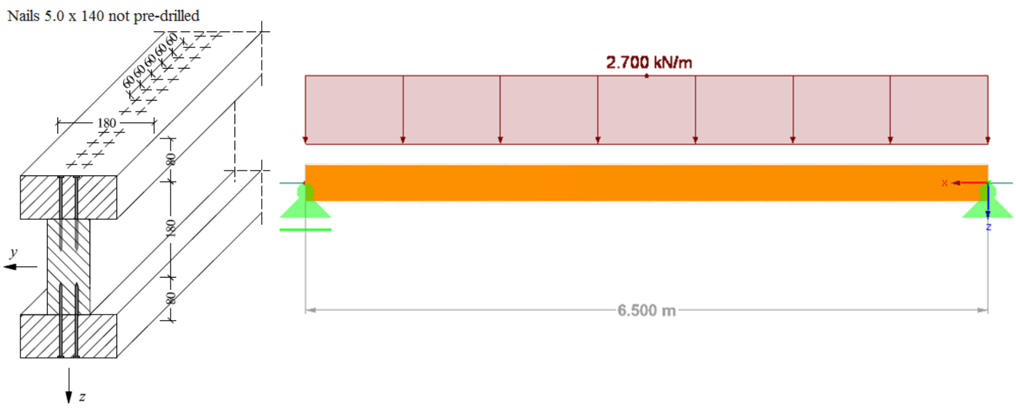


Figure 8.13 System and loads according to [7]

Slip - ultimate limit state

Due to the high compliance (slip) of the fasteners, very different stiffnesses of the cross-section result at the initial and the final state of loading. These can affect the ultimate and the serviceability limit state design.

Modulus of elasticity	$E_{0,mean}$ = 12000 N/mm^2
Slip modulus	$k_{1/3} = {}^2/_3 \cdot k_{ser} = {}^2/_3 \cdot 895 = 600$ N/mm
Area of cross-section	$A_{1\text{-}3} = 180 \cdot 80 = 14.4 \cdot 10^3$ mm^2
Second moment of area	$I_{y,1/3} = 180 \cdot 80^3 / 12 = 7.68 \cdot 10^6$ mm^4
Second moment of area	$I_{y,2} = 80 \cdot 180^3 / 12 = 38.88 \cdot 10^6$ mm^4
Effective distance fasteners	s_{ef} = 60 mm

Slip in joint 1:

$$\gamma_1 = \frac{1}{1 + \frac{\pi^2 \cdot E_1 \cdot A_1 \cdot s_1}{k_1 \cdot l^2}} = \frac{1}{1 + \frac{\pi^2 \cdot 12 \cdot 10^3 \cdot 14.4 \cdot 10^3 \cdot \frac{60}{2}}{600 \cdot (6.5 \cdot 10^3)^2}} = 0.331$$

$$\gamma_2 = 1$$

Slip in joint 2:

$$\gamma_3 = \frac{1}{1 + \frac{\pi^2 \cdot E_3 \cdot A_3 \cdot s_3}{k_3 \cdot l^2}} = \frac{1}{1 + \frac{\pi^2 \cdot 12 \cdot 10^3 \cdot 14.4 \cdot 10^3 \cdot \frac{60}{2}}{600 \cdot (6.5 \cdot 10^3)^2}} = 0.331$$

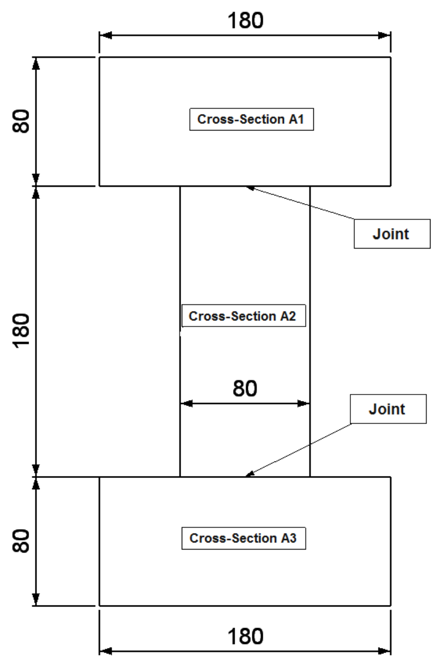


Figure 8.14 Description of cross-section and slip

Slip - serviceability limit state

In the limit state of loading, the creep deformation of timber according to EN 1995-1-1 must be considered for the quasi-permanent design situation. The creep coefficient k_{def} is to be taken as 0.6 in service class 1. The slip of the joint changes as follows:

Modulus of elasticity

$$E_i = \frac{E_{0,mean}}{1 + \psi_2 \cdot k_{def,i}} = 12000\ N/mm^2 = 8824\ N/mm^2$$

Slip modulus

$$k_{1/3} = \frac{2}{3} \cdot k_{ser} = \frac{2}{3} \cdot 895 = 600\ N/mm$$

Area of cross-section

$$A_{1-3} = 180 \cdot 80 = 14.4 \cdot 10^3\ mm^2$$

Second moment of area

$$I_{y,1/3} = \frac{180 \cdot 80^3}{12} = 7.68 \cdot 10^6\ mm^4$$

Second moment of area

$$I_{y,2} = \frac{80 \cdot 180^3}{12} = 38.88 \cdot 10^6\ mm^4$$

Effective distance fasteners

$$s_{ef} = 60\ mm$$

Slip in joint 1

$$\gamma_1 = \frac{1}{1 + \frac{\pi^2 \cdot 8824 \cdot 14.4 \cdot 10^3 \cdot \frac{60}{2}}{350 \cdot (6.5 \cdot 10^3)^2}} = 0.282$$

$$\gamma_2 = 1$$

Slip in joint 2

$$\gamma_3 = 0.282$$

Stiffnesses

The difference in the slip is 0.331 to 0.282 and is, therefore, not very serious.

Now we calculate the stiffness values for the ultimate limit state only. The effects due to different slips are described in Chapter 8.2.3.2.

Bending stiffness about Y:

$$(E \cdot I_y)_{ef} = \sum_1^3 (E_i \cdot I_{i,y} + \gamma_i \cdot E - i \cdot A_i \cdot a_i^2) = 2.586 \cdot 10^{12}\ Nmm^2$$

Bending stiffness about Z:

$$E \cdot I_y = 12000 \cdot \left(2 \cdot \frac{80 \cdot 180^3}{12} + \frac{180 \cdot 80^3}{12}\right) = 1.025 \cdot 10^{12}\ \text{Nmm}^2$$

RF-/TIMBER Pro uses the following cross-section values:

Cross-section description	Symbol	Value	Unit
Width	b_1	18.00	cm
Height	h_1	8.00	cm
Width	b_2	8.00	cm
Height	h_2	18.00	cm
Width	b_3	18.00	cm
Height	h_3	8.00	cm
Slip in joint	$\gamma_{Joint\ 1}$	0.331	
Slip in joint	$\gamma_{Joint\ 2}$	0.331	
Position of centroid	z_S	17.00	cm
Distance to stress lines	z_0	17.00	cm
Distance to stress lines	a_1	-13.00	cm
Distance to stress lines	a_2	0.00	cm
Distance to stress lines	a_3	13.00	cm
Second moment of area about y-axis	$I_{y,eff}$	21,534.40	cm^4
Second moment of area about z-axis	$I_{z,eff}$	8,544.00	cm^4

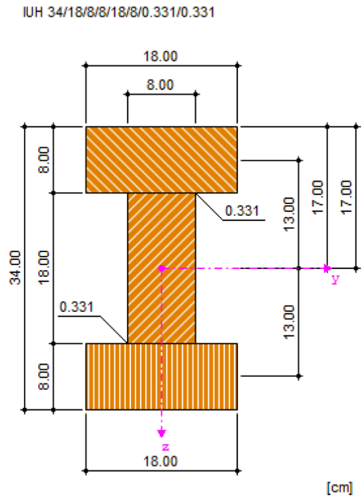


Figure 8.15 Cross-section values in RF-/TIMBER Pro

With a modulus of elasticity of 12,000 N/mm², the effective stiffness is given as:

Bending stiffness about Y:

$$(E \cdot I_y)_{ef} = 1200 \cdot 21534.4 = 2.58 \cdot 10^7\ \text{kNcm}^2$$

Bending stiffness about Z:

$$E \cdot I_y = 12000 \cdot 8544 = 1.025 \cdot 10^{12}\ \text{kNcm}^2$$

Hence, the stiffnesses are identical.

8.2.2 Calculation with RFEM/RSTAB

The system and the loads are created in RFEM or RSTAB as a 3D model.

We specify the standard **EN 1990 + EN 1995** and the National Annex for **DIN** in the *Classification of Loads and Combinations* section of the *New Model - General Data* dialog box. The automatic generation of combinations is of no importance for this example.

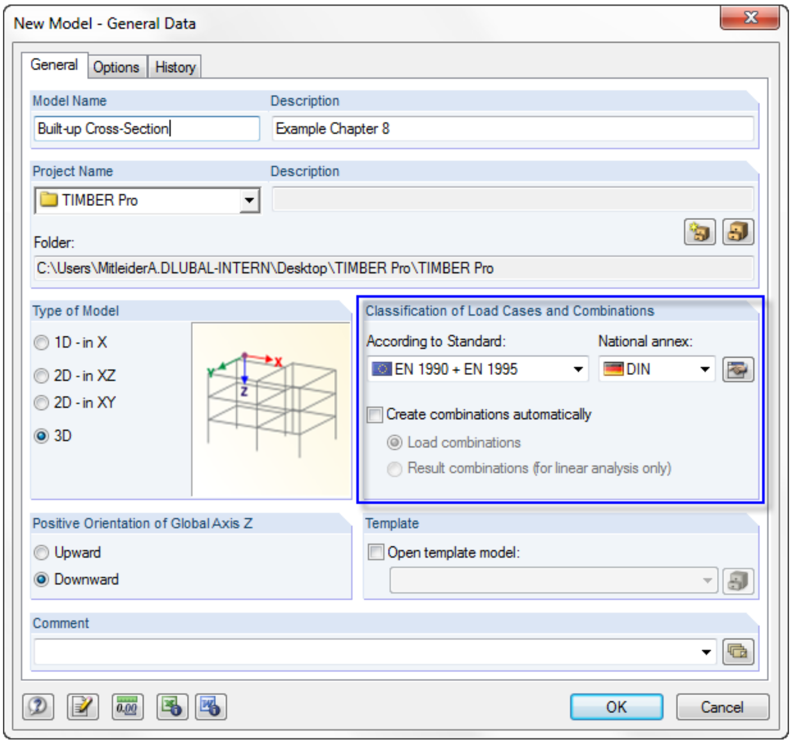


Figure 8.16 *New Model - General Data* dialog box in RFEM/RSTAB

For the system shown in Figure 8.13, we define the cross-section in the library, taking into account the *Coefficient of compliance* (slip). We apply $\gamma_1 = \gamma_2 =$ **0.331** for the ultimate limit state design.

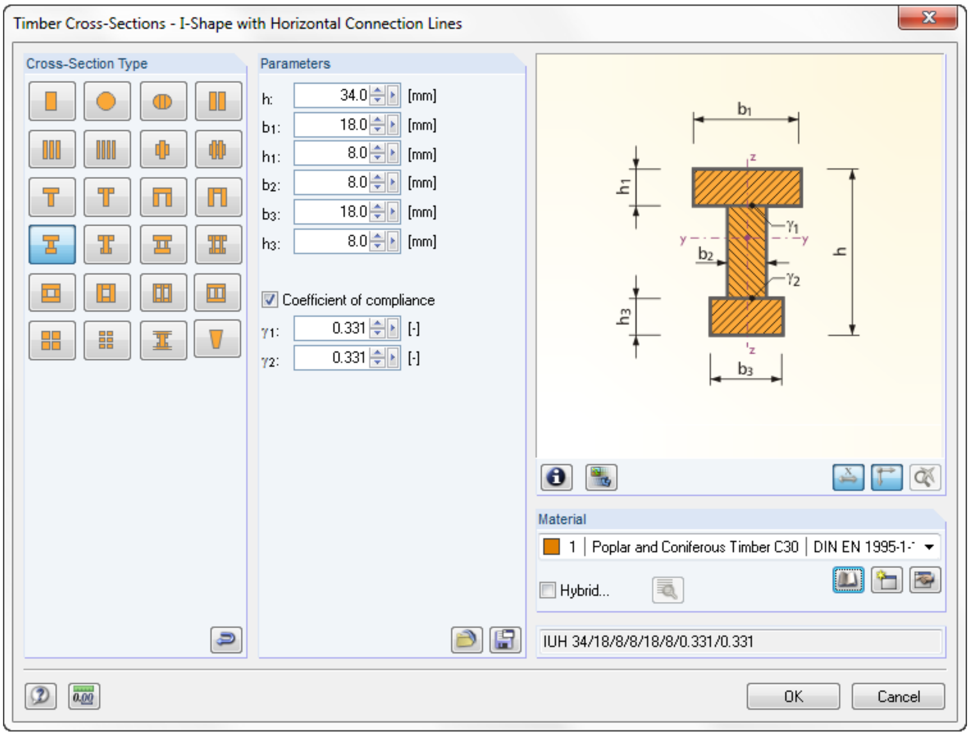


Figure 8.17 Library dialog box *Timber Cross-Sections*

We deactivate the automatic consideration of the self-weight when we create the load case because it is also neglected in the example from [7]. The *Load Duration Class* is **Medium-term**. The specification is also valid for RF-/TIMBER Pro.

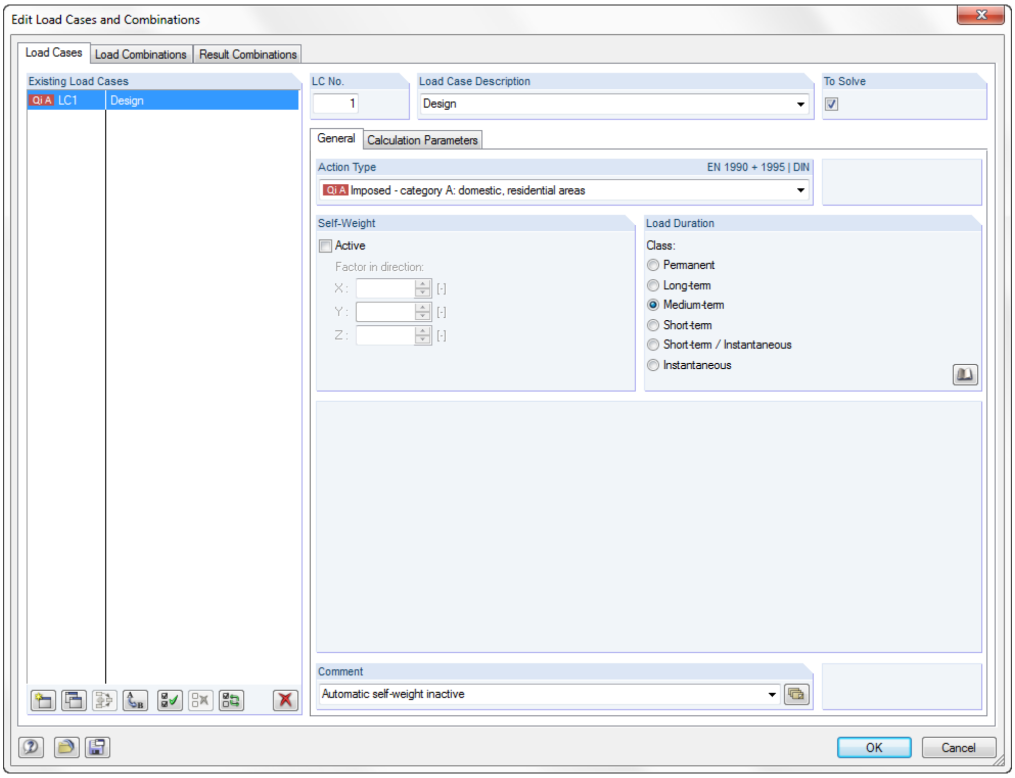


Figure 8.18 RFEM dialog box *Edit Load Cases and Combinations*

We define a member load of **2.7** kN/m in the direction **Global Z**.

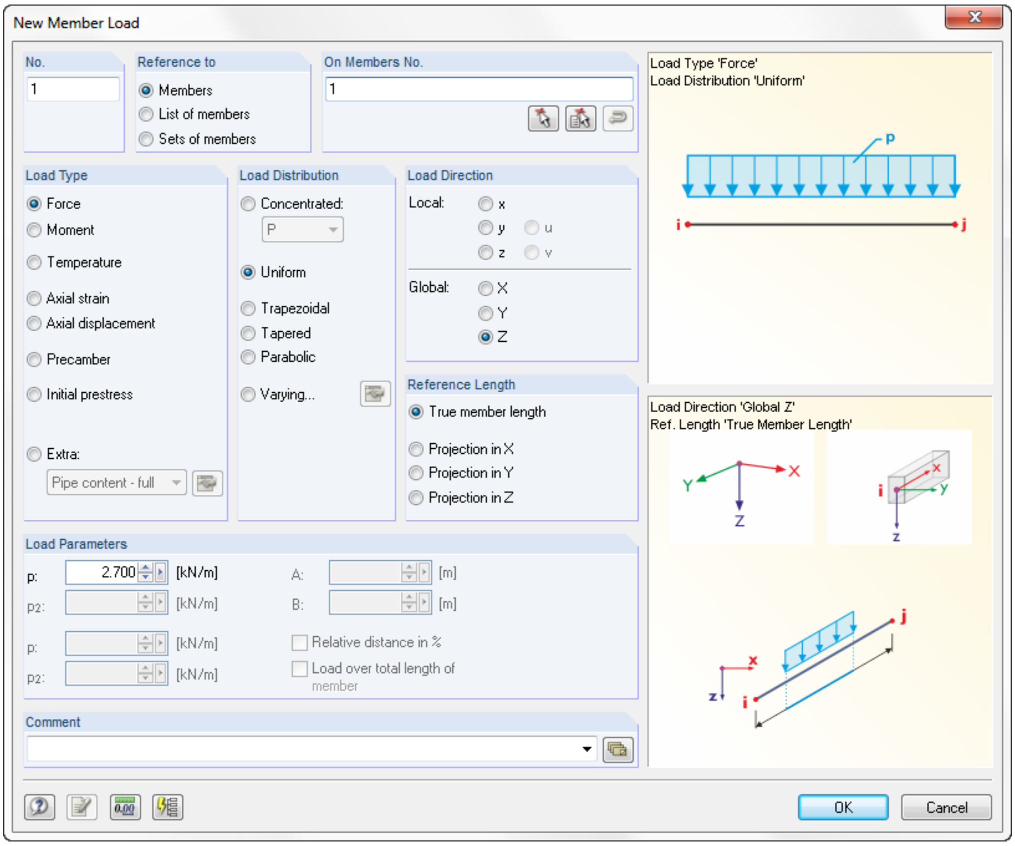


Figure 8.19 Dialog box *New Member Load* in RFEM/RSTAB

8.2.3 Design with RF-/TIMBER Pro

8.2.3.1 Ultimate Limit State Design

In the *1.1 General Data* window, we select the load case **LC1** for the *Ultimate Limit State* design.

We perform the design according to **EN 1995-1-1** with the German National Annex **DIN**.

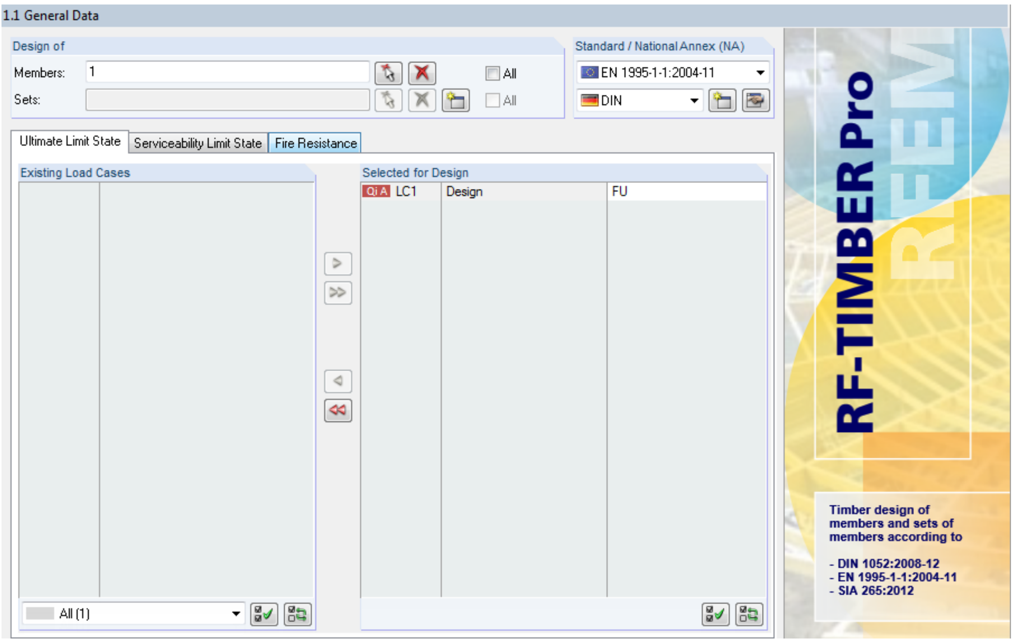


Figure 8.20 Window *1.1 General Data*

We check in the *1.4 Load Duration and Service Class* window whether or not the LDC of the load case is preset as **Medium-term**.

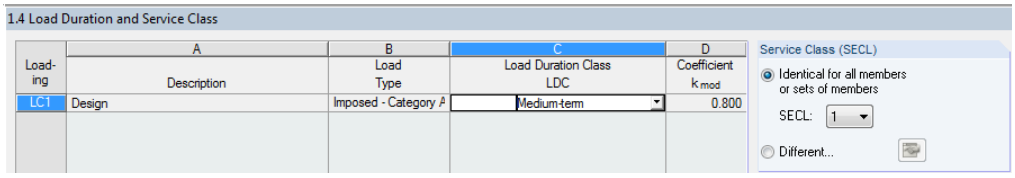


Figure 8.21 Window *1.4 Load Duration and Service Class*

In the *1.5 Effective Lengths - Members* window, we change the buckling lengths of the beam to **3.25** m. As the lateral-torsional buckling analysis for built-up cross-sections is not specified in the codes, the columns H to J are not shown for built-up cross-sections.

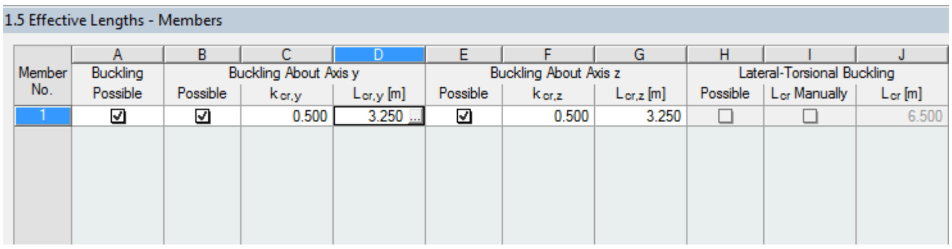
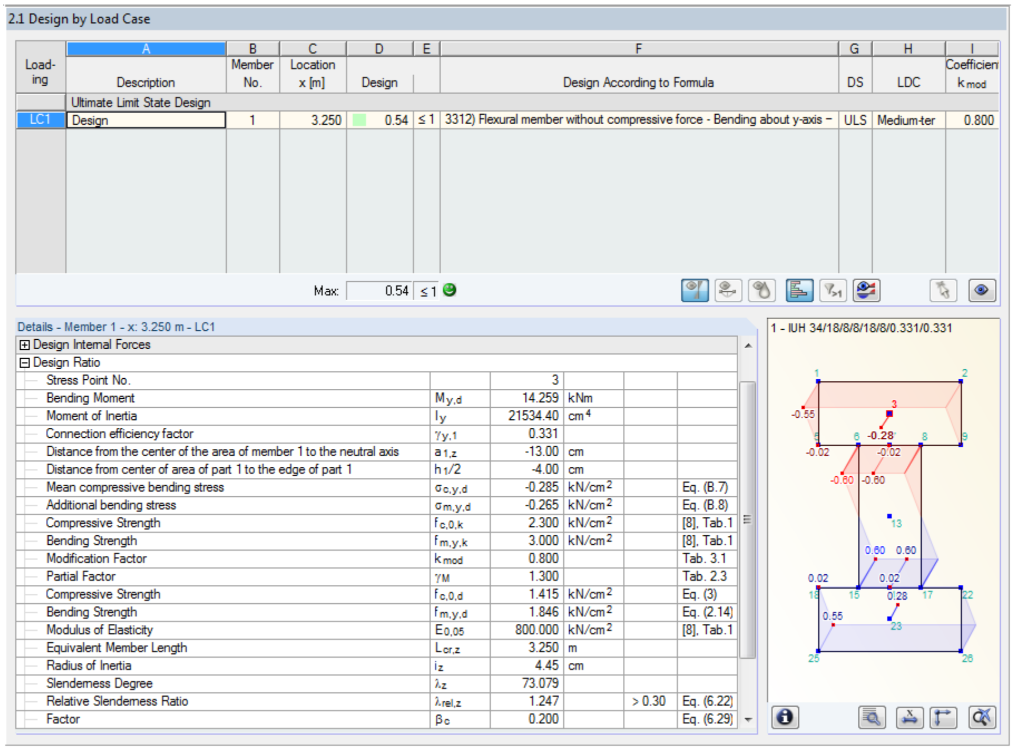


Figure 8.22 Window *1.5 Effective Lengths - Members*

Calculation

After the [Calculation], the *2.1 Design by Load Case* window appears showing the governing designs.


2.1 Design by Load Case

Loading	A Description	B Member No.	C Location x [m]	D Design		E	F Design According to Formula	G DS	H LDC	I Coefficient k_{mod}
	Ultimate Limit State Design									
LC1	Design	1	3.250		0.54	≤ 1	331) Flexural member without compressive force - Bending about y-axis –	ULS	Medium-ter	0.800
			Max:		0.54	≤ 1				

Details - Member 1 - x: 3.250 m - LC1

Design Internal Forces					
Design Ratio					
Stress Point No.		3			
Bending Moment	$M_{y,d}$	14.259	kNm		
Moment of Inertia	I_y	21534.40	cm⁴		
Connection efficiency factor	$\gamma_{y,1}$	0.331			
Distance from the center of the area of member 1 to the neutral axis	$a_{1,z}$	-13.00	cm		
Distance from center of area of part 1 to the edge of part 1	$h_1/2$	-4.00	cm		
Mean compressive bending stress	$\sigma_{c,y,d}$	-0.285	kN/cm²		Eq. (B.7)
Additional bending stress	$\sigma_{m,y,d}$	-0.265	kN/cm²		Eq. (B.8)
Compressive Strength	$f_{c,0,k}$	2.300	kN/cm²		[8], Tab.1
Bending Strength	$f_{m,y,k}$	3.000	kN/cm²		[8], Tab.1
Modification Factor	k_{mod}	0.800			Tab. 3.1
Partial Factor	γ_M	1.300			Tab. 2.3
Compressive Strength	$f_{c,0,d}$	1.415	kN/cm²		Eq. (3)
Bending Strength	$f_{m,y,d}$	1.846	kN/cm²		Eq. (2.14)
Modulus of Elasticity	$E_{0,05}$	800.000	kN/cm²		[8], Tab.1
Equivalent Member Length	$L_{cr,z}$	3.250	m		
Radius of Inertia	i_z	4.45	cm		
Slenderness Degree	λ_z	73.079			
Relative Slenderness Ratio	$\lambda_{rel,z}$	1.247		> 0.30	Eq. (6.22)
Factor	β_c	0.200			Eq. (6.29)

Figure 8.23 Window 2.1 *Design by Load Case*

Details...

The governing design is the flexural buckling design with a design ratio of 54%. This value is obtained only if the option *Consider bending stress $\sigma_{m,i}$ (centroid)* has been activated (see Figure 3.5) in the *Other* tab of the *Details* dialog box.

The designs are additionally checked by manual calculation.

Design of normal stresses according to expression (B.7)

$$N_{i,d} = \frac{M_{y,d}}{(EI)_{ef}} \cdot E_i \cdot \gamma_i \cdot a_i \cdot A_i$$

$$N_{1,d} = N_{3,d} = \frac{1426\ \text{kNcm}}{2.58 \cdot 10^7\ \text{kNcm}} \cdot 1200\ \text{kN/cm}^2 \cdot 0.331 \cdot 13\ \text{cm} \cdot 144\ \text{cm}^2 = 41.4\ \text{kN}$$

$$\sigma_{c,1/3,d} = \frac{N}{A} = \frac{41.1\ \text{kN}}{144\ \text{cm}^2} = 0.285\ \text{kN/cm}^2$$

Design of compression

$$\frac{\sigma_{c,1,d}}{f_{c,0,d}} = \frac{0.285\ \text{kN/cm}^2}{1.42\ \text{kN/cm}^2} = 0.20 < 1$$

Design of tension

$$\frac{\sigma_{c,3,d}}{f_{t,0,d}} = \frac{0.285\ \text{kN/cm}^2}{1.11\ \text{kN/cm}^2} = 0.26 < 1$$

The distribution of compressive and tensile stresses is as follows:

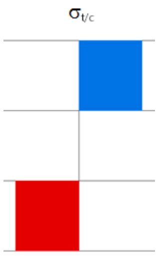


Figure 8.24 Qualitative distribution of stresses

Design of edge stresses according to expression (B.8)

$$\sigma_{m,i,d} = \frac{M_{y,d}}{(EI)_{ef}} \cdot E_i \cdot \frac{h_i}{2}$$

$$\sigma_{m,1,d} = \sigma_{m,3,d} = \frac{1426\ \text{kNcm}}{2.58 \cdot 10^7\ \text{kNcm}} \cdot 1200\ \text{kN/cm}^2 \cdot \frac{8\ \text{cm}}{2} = 0.265\ \text{kN/cm}^2$$

$$\sigma_{m,2,d} = \frac{1426\ \text{kNcm}}{2.58 \cdot 10^7\ \text{kNcm}} \cdot 1200\ \text{kN/cm}^2 \cdot \frac{18\ \text{cm}}{2} = 0.596\ \text{kN/cm}^2$$

Design of web

$$\frac{\sigma_{m,2,d}}{f_{m,d}} = \frac{0.596}{1.85} = 0.32 < 1$$

The stresses are graphically displayed in the *Cross-section* dialog box in RF-/TIMBER Pro which you can open by clicking [Show or Print Cross-section Values].

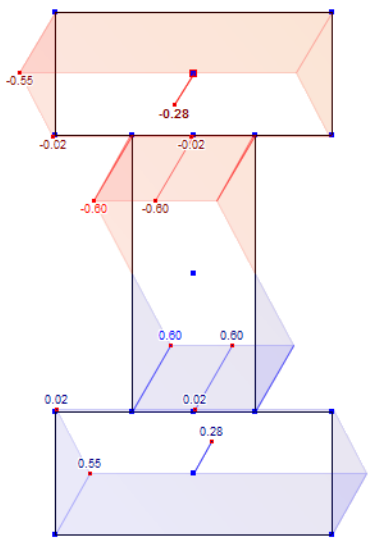


Figure 8.25 Display of flexural edge stress in RF-TIMBER Pro

Design of shear stresses in the neutral plane of the web

$$\tau_{2,max,d} = \frac{V_{max,d} \cdot (\gamma_3 \cdot E_3 \cdot A_3 \cdot a_3 + 0.5 \cdot E_2 \cdot b_2 \cdot h^2)}{(EI)_{ef} \cdot b_2} =$$

$$= \frac{8.78\text{ kN} \cdot \left(0.331 \cdot 1200\text{ kN/cm}^2 \cdot 144\text{ cm}^2 \cdot 13\text{ cm} + 0.5 \cdot 1200\text{ kN/cm}^2 \cdot 8\text{ cm} \cdot \left(\frac{18}{2} + 0\right)^2\right)}{2.58 \cdot 10^7\text{ kNcm}^2 \cdot 8\text{ cm}}$$

$$= 0.048\text{ kN/cm}^2$$

$$\frac{\tau_{2,max,d}}{f_{v,d}} = \frac{0.048}{0.123} = 0.39 < 1$$

Shear force in connecting joint

$$F_{1,v,Ed} = \frac{V_{max,d} \cdot \gamma_1 \cdot E_1 \cdot A_1 \cdot a_1 \cdot s_{1,min}}{(E \cdot I)_{ef}} =$$

$$= \frac{8.78\text{ kN} \cdot 0.331 \cdot 1200\text{ kN/cm}^2 \cdot 144\text{ cm}^3 \cdot 13\text{ cm} \cdot 3\text{ cm}}{2.58 \cdot 10^7\text{ kNcm}^2} = 0.76\text{ kN}$$

The shear force in the connecting joint is not shown in RF-/TIMBER Pro because the distances of the fasteners cannot be defined in the program.

Buckling coefficient

According to [1], we have to reduce the allowable compressive stress for the stability analysis (buckling design) by the buckling coefficient k_c. This coefficient depends on the slenderness ratio λ.

$$i_y = \sqrt{\frac{I_y}{A}} = \sqrt{\frac{21534.4\text{ cm}^4}{432\text{ cm}^2}} = 7.06\text{ cm}$$

$$i_z = \sqrt{\frac{I_z}{A}} = \sqrt{\frac{8544\text{ cm}^4}{432\text{ cm}^2}} = 4.45\text{ cm}$$

The slenderness ratio is given as:

$$\lambda_z = \frac{s_k}{i_z} = \frac{325\text{ cm}}{4.45\text{ cm}} = 73.03$$

The buckling coefficient k_c according to [1], clause 6.3.2 is (intermediate values may be interpolated linearly):

$$k_{c,z} = \frac{1}{k_z + \sqrt{k_z^2 - \lambda_{rel,z}^2}} = \frac{1}{1.37 + \sqrt{1.37^2 - 1.25^2}} = 0.51$$

$$\lambda_{rel,z}^2 = \frac{\lambda_z}{\pi} \cdot \sqrt{\frac{f_{c,0,k}}{E_{0.05}}} = \frac{73.03}{\pi} \cdot \sqrt{\frac{2.3\text{ kN/cm}^2}{800\text{ kN/cm}^2}} = 1.25$$

$$k_z = 0.5 \cdot (1 + \beta_c(\lambda_{rel,z} - 0.3) + \lambda_{rel,y}^2) = 0.5 \cdot (1 + 0.2(1.25 - 0.3) + 1.25^2) = 1.37$$

Stability analysis

Design according to [1] Eq. (6.24):

$$\eta_1 = \frac{\sigma_{c,y,d}}{k_{c,z} \cdot f_{c,0,d}} + k_m \cdot \frac{\sigma_{m,y,d}}{f_{m,d}} = \frac{0.285}{0.51 \cdot 1.42} + 1.0 \cdot \frac{0.265}{1.85} = 0.54 < 1$$

Nat. Annex...

For this design, where the stresses from bending are additionally divided into compressive bending stresses, it can be legitimate to reduce the coefficient k_m in the *National Annex Settings* to 0.7 (see Figure 2.10). The design ratio thus becomes a little smaller. On the other hand, the component of compressive bending has now a lesser strength.

8.2.3.2 Serviceability Limit State Design

The serviceability limit state design is usually carried out for a respective load combination. In this example, however, we want to illustrate the effects of the different stiffnesses. In Chapter 8.2.1, the γ-factors for the stiffnesses at the initial and final state of loading are determined as 0.331 or 0.282. In order to consider the resulting differences in the stiffnesses correctly, we must perform another calculation where also the ultimate limit state design is carried out with the end stiffness - the stiffness from the serviceability limit state calculation.

Thus, it becomes clear why no fire resistance design is possible for built-up cross-sections: The stiffnesses would also have to be recalculated for the fire resistance design. Furthermore, in the fire resistance design, the changed position of the neutral axis becomes computable only upon reducing the cross-section.

Adjusting the model

We exit RF-/TIMBER Pro with [OK].

In RFEM or RSTAB, we create a copy of the model including loading: We select the member and loading and copy the selection by clicking on the menu

Edit → Move/Copy.

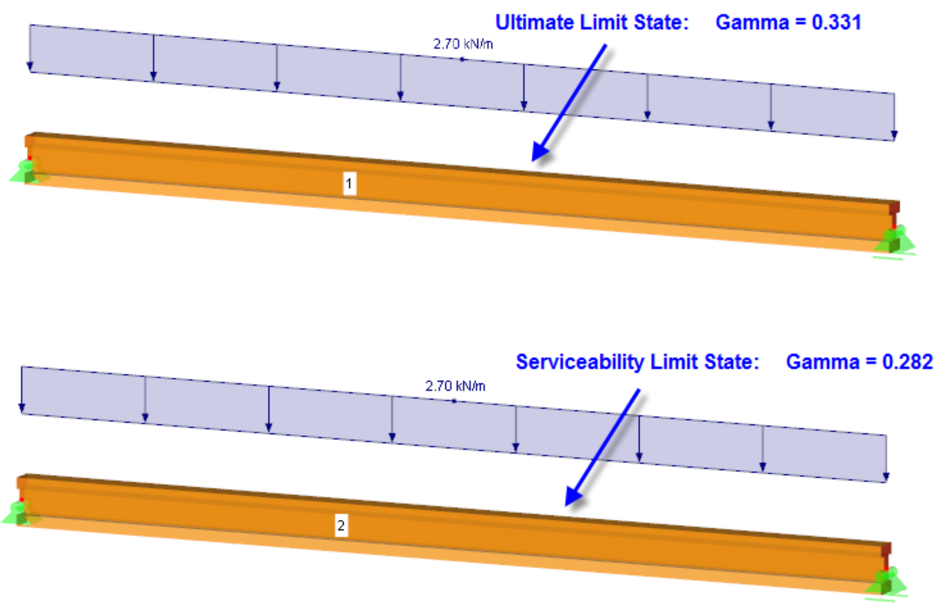


Figure 8.26 Model and copy in RFEM or RSTAB

To consider the changed slip, we define a new cross-section with modified γ slip factors for the copied member.

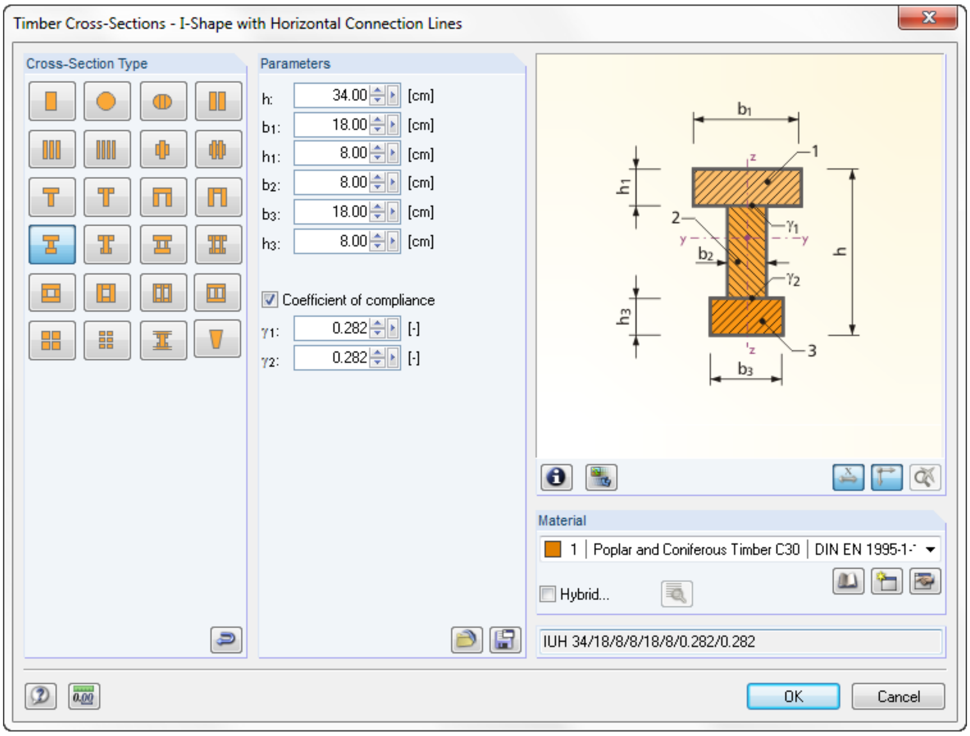


Figure 8.27 Library dialog box *Timber Cross-Sections* with $\gamma_1 = \gamma_2 = 0.282$

We reopen the RF-/TIMBER Pro add-on module and copy design case 1 by clicking on the menu

File → Copy Case.

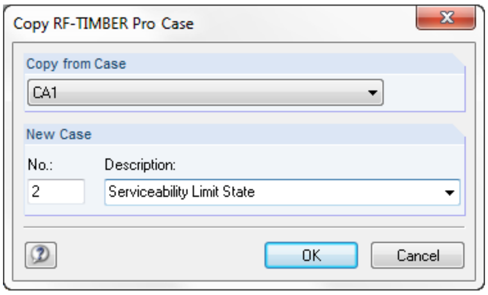


Figure 8.28 Dialog box *Copy RF-TIMBER Pro Case*

In design case 2, we specify member **2** for the design (see Figure 8.29).

We do not change LC1, which is preset for the design of the *Ultimate Limit State*.

In the *Serviceability Limit State* tab, we transfer **LC1** to the *Selected for Design* list (strictly speaking, a separate action combination should be defined). We assign this load case to the **Quasi-permanent** design combination: Thus, the deformation is compared with the limit value of the final deformation of l/300 = 21.7 mm.

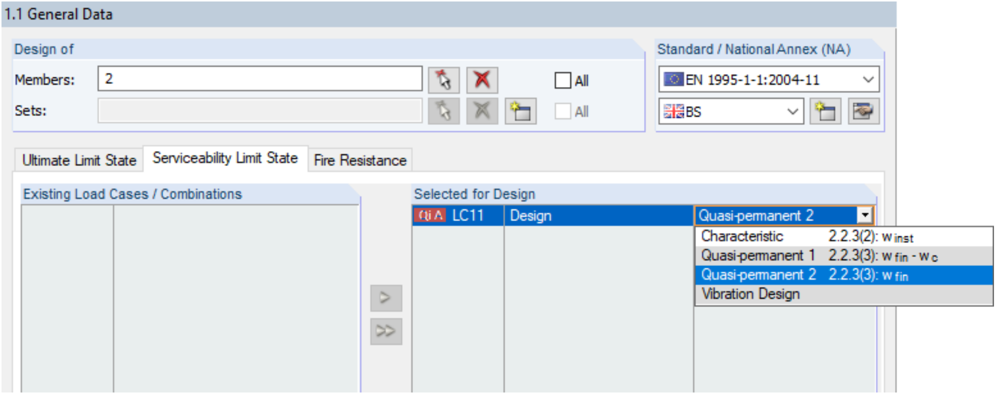


Figure 8.29 Window *1.1 General Data*, tab *Serviceability Limit State*

Next, in the *1.5 Effective Lengths - Members* window, we reduce the buckling lengths for member 2 to **3.25** m (see Figure 8.22).

In the *1.9 Serviceability Data* window, we specify member **2**.

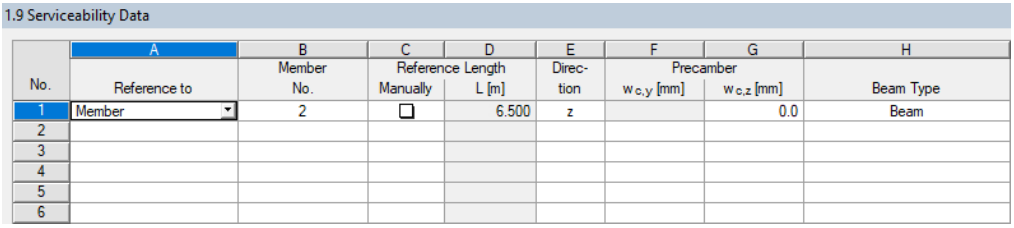


Figure 8.30 Window *1.9 Serviceability Data*

The [Calculation] shows that the deflection analysis is exceeded by 26%. The design ratio in the ultimate limit state designs also increases slightly: Due to the greater deformation/the smaller stiffness of the cross-section, each single cross-section part is more affected by bending.

Section	Location x [m]	Load Case	De-sign		Design According to Formula
1	IUH 34/18/8/8/18/8/**0.331/0.331**				
	0.00	LC1	0.39	≤ 1	Shear due to shear force V_z acc. to 6.1.7
	3.25	LC1	0.32	≤ 1	Extreme compressive bending stress M_y acc. to 6.1.6
	3.25	LC1	0.18	≤ 1	Mean compressive bending stress M_y acc. to 6.1.4
	3.25	LC1	0.40	≤ 1	Mean tensile bending stress M_y acc. to 6.1.2
	3.25	LC1	0.32	≤ 1	Extreme tensile bending stress M_y acc. to 6.1.6
	3.25	LC1	0.54	≤ 1	Bending about y-axis; mean compressive bending stress M_y acc. to 6.3.2

Figure 8.31 Design ratios with γ-factors **0.331** in design case 1

Section No.	Location x [m]	Load Case	De-sign		Design According to Formula
2	IUH 34/18/8/8/18/8/**0.282/0.282**				
	0.00	LC1	0.40	≤ 1	Shear due to shear force V_z acc. to 6.1.7
	3.25	LC1	0.36	≤ 1	Extreme compressive bending stress M_y acc. to 6.1.6
	3.25	LC1	0.20	≤ 1	Mean compressive bending stress M_y acc. to 6.1.4
	3.25	LC1	0.41	≤ 1	Mean tensile bending stress M_y acc. to 6.1.2
	3.25	LC1	0.36	≤ 1	Extreme tensile bending stress M_y acc. to 6.1.6
	3.25	LC1	0.54	≤ 1	Bending about y-axis; mean compressive bending stress M_y acc. to 6.3.2
	3.25	LC1	1.26	> 1	Serviceability - Quasi-permanent acc. to 7.2 - Inner span, z-direction

Figure 8.32 Design ratios with γ-factors **0.282** in design case 2

8.3 Monopitch Roof Beam

According to current timber standards, the strength of a tapered beam must be reduced depending on the loading (tensile bending or compressive bending). This usually applies to monopitch roof beams.

8.3.1 System and Loads

Model

Material:	Softwood timber GL24h
Span length:	l = 11.0 m
Cantilever:	l = 3.0 m
Service class:	2
LDC:	Short-term
Cross-section 1:	18/16 cm (start)
Cross-section 2:	18/110 cm (end)

Beam laterally supported (no stability problem)

Load

LC1 self-weight:	g = 2.7 kN/m
LC2 snow:	q = 5.7 kN/m
RC1:	1.35 · LC1 + 1.5 · LC2

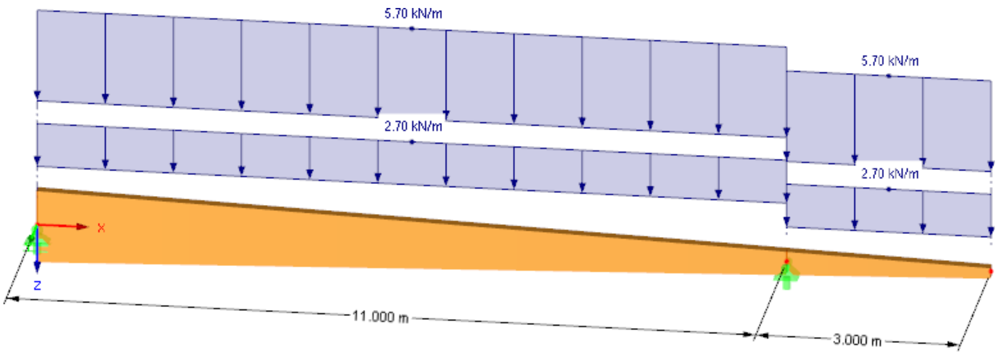


Figure 8.33 System and loads

8.3.2 Calculation with RFEM/RSTAB

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

We superimpose the load cases for the fundamental combination according to the geometrically linear analysis with the corresponding partial safety factors in a result combination.

We obtain the following moment distribution:

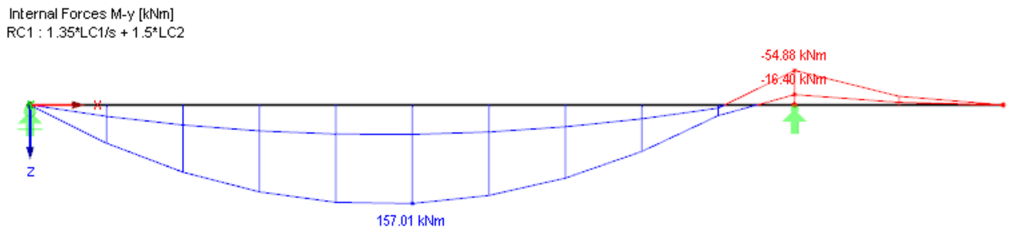
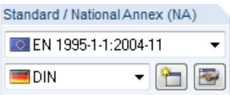


Figure 8.34 Moment distribution

8.3.3 Design with RF-/TIMBER Pro

In the *1.1 General Data* window, we select the result combination **RC1** for the *Ultimate Limit State* design.



We perform the design according to **EN 1995-1-1** with the German National Annex **DIN**.

In the *1.4 Load Duration and Service Class* window, we select the **Short-term** LDC for **RC1**.

1.4 Load Duration and Service Class

Loading	A Description	B Load Type	C Load Duration Class LDC	D Coefficient k_{mod}
LC1	Self-weight	Permanent	Permanent	0.600
LC2	Snow	Snow / ice	Short-term	0.900
RC1	1.35*LC1/p + 1.5*LC2	-	Short-term	0.900

Permanent
Long-term
Medium-term
Short-term
Short-term / Instantaneous
Instantaneous

Service Class (SECL)
Identical for all members or sets of members
SECL: 2
Different...

Service Class 1:
Total moisture content 5-15 %. Mean moisture content in most conifers ≤ 12 %.

Figure 8.35 Window *1.4 Load Duration and Service Class*

RF-/TIMBER Pro recognizes tapered members in the model. The cut-to-grain angle (taper angle) of 3.84° is preset in the *1.7 Tapered Members* window.

1.7 Tapered Members

Member No.	A Cross-Section Member Start	B Cross-Section Member End	C Length L [m]	D Cut-to-Grain Angle α [°]	E	F Grain Parallel to	G Tension Perpendicular to Grain With Ridge	H Manually	I V [m³]	J k_{vol}	K Note
1	T-Rectangle 18/16	T-Rectangle 18/36.1	3.000	3.84	≤ 24.00	+z-Axis Edge	☐	☐			
2	T-Rectangle 18/36.1	T-Rectangle 18/110	11.000	3.84	≤ 24.00	+z-Axis Edge	☐	☐			

Figure 8.36 Window *1.7 Tapered Members*

The grain runs parallel to the edge that is located in the direction of the positive z-axis (that is the member bottom side). Thus, the cut grains run through the compressive bending area (field) or the tensile bending area (cantilever column). Cut grains with tension have a significantly unfavorable effect on the bearing capacity of timber.

The limitation of the cut-to-grain angle is not specified in the global Eurocode but only in the National Application Documents. New studies show that an insufficient safety results only starting from a cut-to-grain angle of 24°. The limitation of this angle to 10° was handled more strictly in DIN 1052, but the design used there was slightly different, too. For further information, see [6].

Details...

Since the beam is not prone to instability risk, we deactivate the stability analysis in the *Details* dialog box.

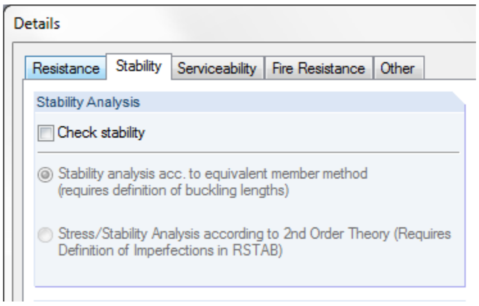
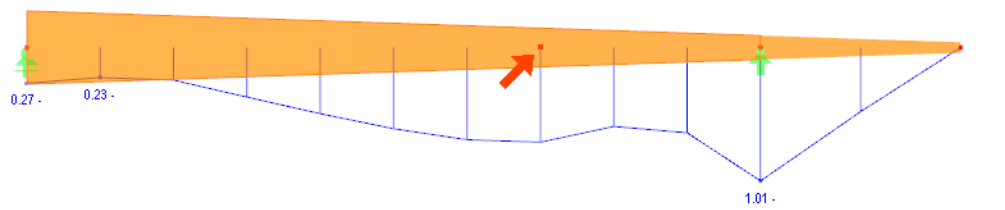


Figure 8.37 Dialog box *Details*, tab *Stability*

Calculation

After the [Calculation], you can evaluate the reduction at the tensile and compressive edge in the *2.5 Design by x-Location* window.



2.5 Design by x-Location

Member No.	A Location x [m]	B Load Case	C Design	D	E Design acc. to Formula
	1.100	RC1	0.65	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
	1.100	RC1	0.18	≤ 1	181) Cross-section resistance - Uniaxial bending about y-axis on edge parallel to the grain acc. to 6.4.2 and 6.1.6
	1.100	RC1	0.19	≤ 1	201) Cross-section resistance - Uniaxial bending about y-axis (compression edge) on cut edge
	2.200	RC1	0.43	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
	2.200	RC1	0.57	≤ 1	181) Cross-section resistance - Uniaxial bending about y-axis on edge parallel to the grain acc. to 6.4.2 and 6.1.6
	2.200	RC1	0.60	≤ 1	201) Cross-section resistance - Uniaxial bending about y-axis (compression edge) on cut edge
	3.300	RC1	0.26	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
	3.300	RC1	0.69	≤ 1	181) Cross-section resistance - Uniaxial bending about y-axis on edge parallel to the grain acc. to 6.4.2 and 6.1.6
	3.300	RC1	0.72	≤ 1	201) Cross-section resistance - Uniaxial bending about y-axis (compression edge) on cut edge
	4.400	RC1	0.13	≤ 1	111) Cross-section resistance - Shear due to shear force Vz acc. to 6.1.7
		Max:	1.01	> 1	

Figure 8.38 Window *2.5 Design by x-Location*

Design for compressive stresses of member 2, location x = 3.30 m

The design is carried out according to [1] clause 6.4.2.

$$\sigma_{m,\alpha,d} = \frac{M_y}{W_y} = \frac{11652\ \text{kNcm}}{10196.7\ \text{cm}^3} = 1.14\ \text{kN/cm}^2$$

$$f_{m,d} = \frac{f_{m,k} \cdot k_{mod}}{\gamma_M} = \frac{2.4 \cdot 0.9}{1.3} = 1.66\ \text{kN/cm}^2$$

$$k_{m,\alpha} = \frac{1}{\sqrt{1 + \left(\frac{f_{m,d}}{1.5 \cdot f_{v,d}} \cdot \tan\alpha\right)^2 + \left(\frac{f_{m,d}}{f_{c,90,d}} \cdot \tan^2\alpha\right)^2}} = 0.955$$

Design:

$$\frac{\sigma_{m,\alpha d}}{k_{m,\alpha} \cdot f_{m,d}} = \frac{1.14}{0.955 \cdot 1.66} = 0.72 < 1$$

With 4.5%, the strength reduction by the factor $k_{m,\alpha}$ is small.

Design for tensile stresses of member 2, location x = 0.00 m

The design location is above the support at the cantilever.

The design according to [1] clause 6.4.2 is as follows.

$$\sigma_{m,\alpha,d} = \frac{M_y}{W_y} = \frac{5488\ \text{kNcm}}{3918.9\ \text{cm}^3} = 1.40\ \text{kN/cm}^2$$

$$f_{m,d} = \frac{f_{m,k} \cdot k_{mod}}{\gamma_M} = \frac{2.4 \cdot 0.9}{1.3} = 1.66\ \text{kN/cm}^2$$

$$k_{m,\alpha} = \frac{1}{\sqrt{1 + \left(\frac{f_{m,d}}{0.75 \cdot f_{v,d}} \cdot \tan\alpha\right)^2 + \left(\frac{f_{m,d}}{f_{c,90,d}} \cdot \tan^2\alpha\right)^2}} = 0.83$$

Design:

$$\frac{\sigma_{m,\alpha,d}}{k_{m,\alpha} \cdot f_{m,d}} = \frac{1.40}{0.83 \cdot 1.66} = 1.01 \approx 1$$

By the higher reduction of 17% the design in the cut area becomes governing.

A solution for this beam could be to move the cut grains to the bottom side. Find the corresponding input option in table column F of the *1.7 Tapered Members* window (see Figure 8.36).

8.4 Curved Beam

The designs are performed according to DIN EN 1995-1-1 for a beam curved in an S-shape. Due to this geometrical form, the beam has a very distinctive stability problem.

Beams of this shape are not explicitly specified in the standards. The characteristics for the stability analysis are not covered there. For this reason, the stability analysis was deactivated in this example.

Designing curved members is only possible in the add-on module **RF-**TIMBER Pro. RSTAB does not allow for curved lines.

8.4.1 System and Loads

Model

Material:	Softwood timber GL28h
Span width field 1:	l = 11.90 m
Span width field 2:	l = 9.13 m
Service class:	1
LDC:	Short-term
k_{mod}:	0.9
Cross-section:	20/68 cm
Beam laterally supported	⇒ no stability problem

Load

LC1 self-weight:	g = 2.4 kN/m
LC2 snow:	q = 3.2 kN/m
RC1:	1.35 · LC1 + 1.5 · LC2

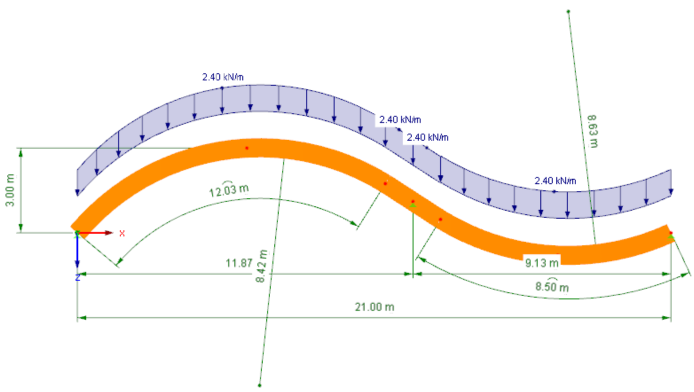


Figure 8.39 System and loads

8.4.2 Calculation with RFEM

The system as well as the loads of both load cases are modeled as a 3D model in RFEM. We deactivate the automatic consideration of the self-weight for LC1.

We superimpose the load cases for the fundamental combination according to the geometrically linear analysis with the corresponding partial safety factors in a result combination.

RFEM determines the diagrams of internal forces shown in Figure 8.40.

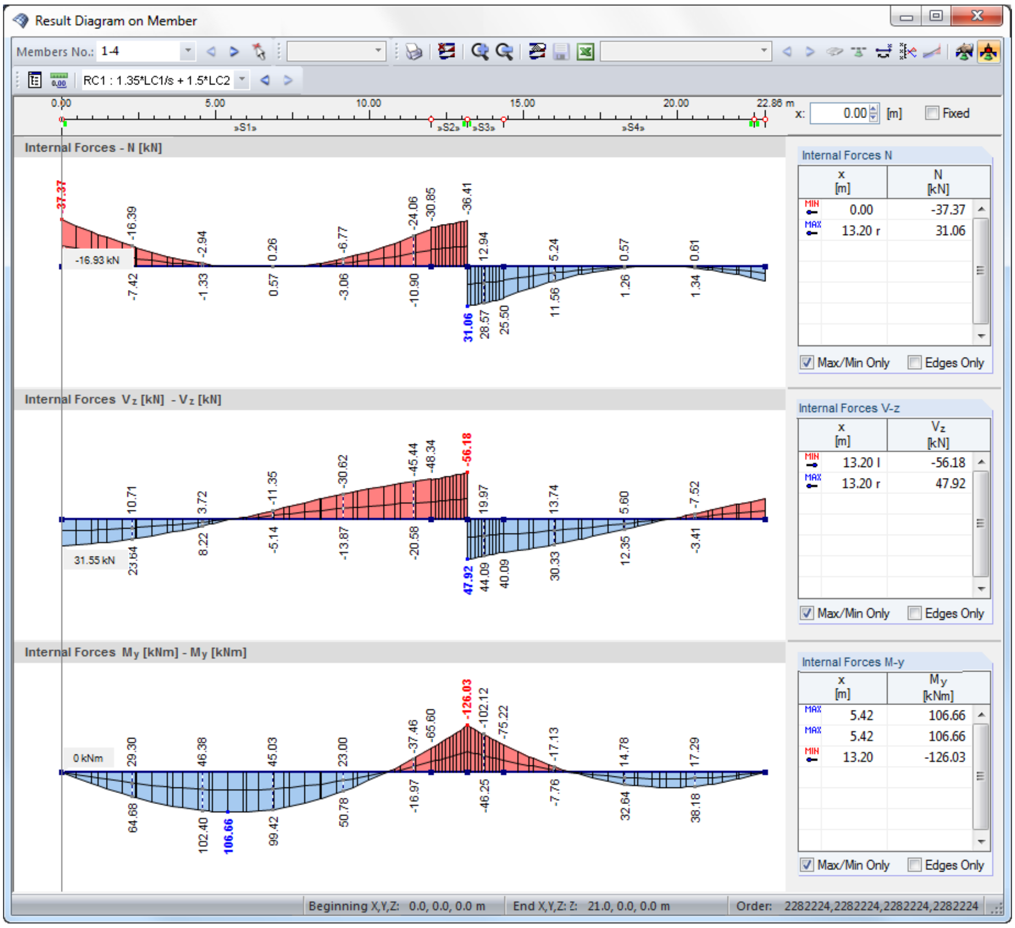


Figure 8.40 Internal forces N, V_z and M_y

8.4.3 Design with RF-TIMBER Pro

Ultimate limit state design

In the *1.1 General Data* window, we select the result combination **RC1** for the *Ultimate Limit State* design.

We perform the design according to **EN 1995-1-1** with the German National Annex **DIN**.

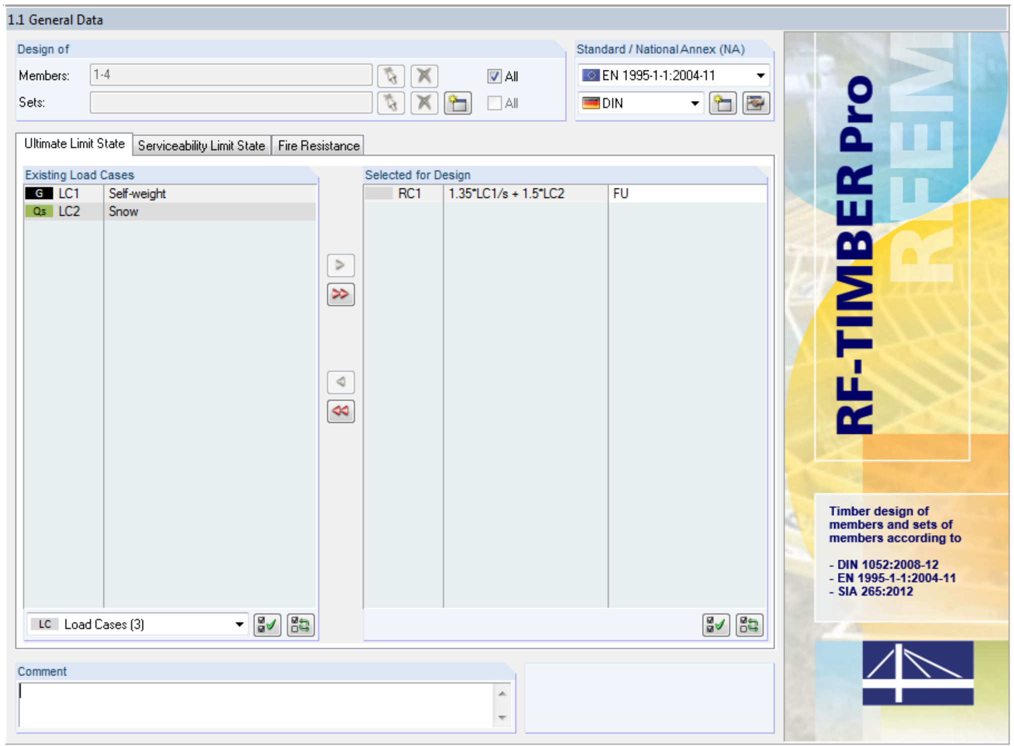


Figure 8.41 Window *1.1 General Data*

The Windows 1.2 and 1.3 are not different from the ones in the previous examples.

In the *1.4 Load Duration and Service Class* window, LDC and SECL of the load cases are preset based on the *Load Duration* defined in RFEM. We change the LDC of RC1 to **Short-term**.

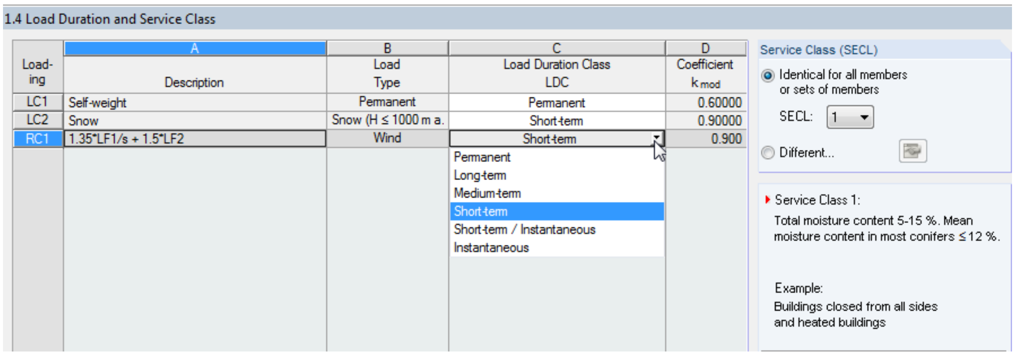


Figure 8.42 Window *1.4 Load Duration and Service Class*

The *1.5 Effective Lengths* window for the input of buckling and lateral-torsional buckling lengths for curved members is not shown, as the lateral-torsional design for this beam types is not clearly specified in the Standard [1]. There is a method only for single-span beams by which curved beams can be designed at a distance of one third of the smallest cross-section height.

Since the beam is supported on all sides, we deactivate the stability analysis in the *Details* dialog box.

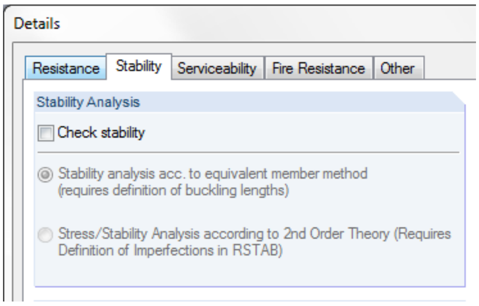
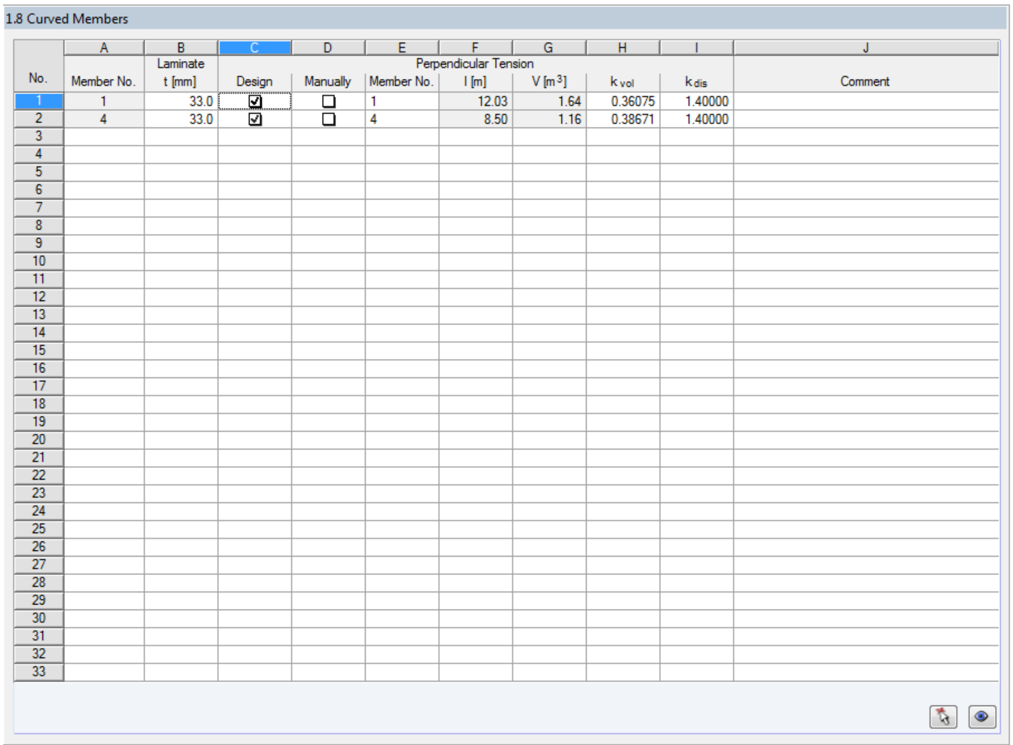


Figure 8.43 Dialog box *Details*, tab *Stability*

Then, we set the input Window *1.8 Curved Members*.



1.8 Curved Members

	A	B	C	D	E	F	G	H	I	J
		Laminate			Perpendicular Tension					
No.	Member No.	t [mm]	Design	Manually	Member No.	l [m]	V [m³]	k_{vol}	k_{dis}	Comment
1	1	33.0	☑	☐	1	12.03	1.64	0.36075	1.40000	
2	4	33.0	☑	☐	4	8.50	1.16	0.38671	1.40000	
3										
4										
5										
6										
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 8.44 Window *1.8 Curved Members*

We check whether the thickness t of the lamellas is **33 mm**, limiting the beam's radius of curvature.

RF-TIMBER Pro performs the check of *Perpendicular Tension*, if the check box in the **Design** column is selected.

Calculation

We start the [Calculation]. Then, we select the *2.4 Design by Member* window.

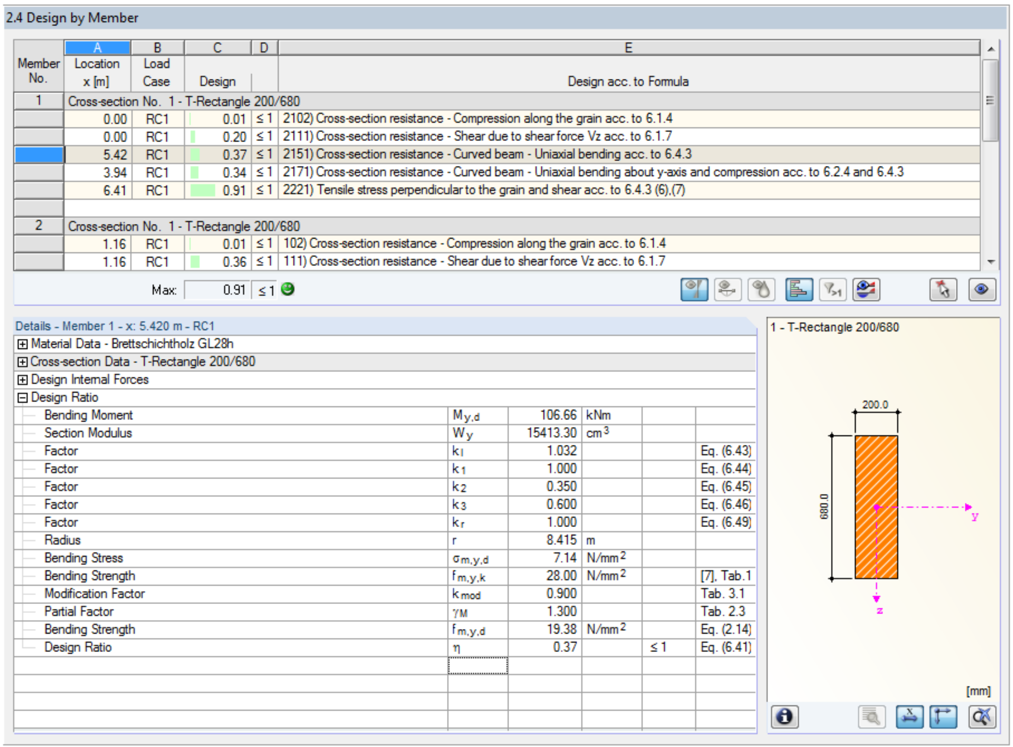


Figure 8.45 Window 2.4 *Design by Member*

For *Member No.* 1, the greatest design ratio due to bending is 0.37.

In the *Details* table, we can check the radius *r* among the design parameters for member 1. The program imports the member's curvature from the RFEM arc parameters.

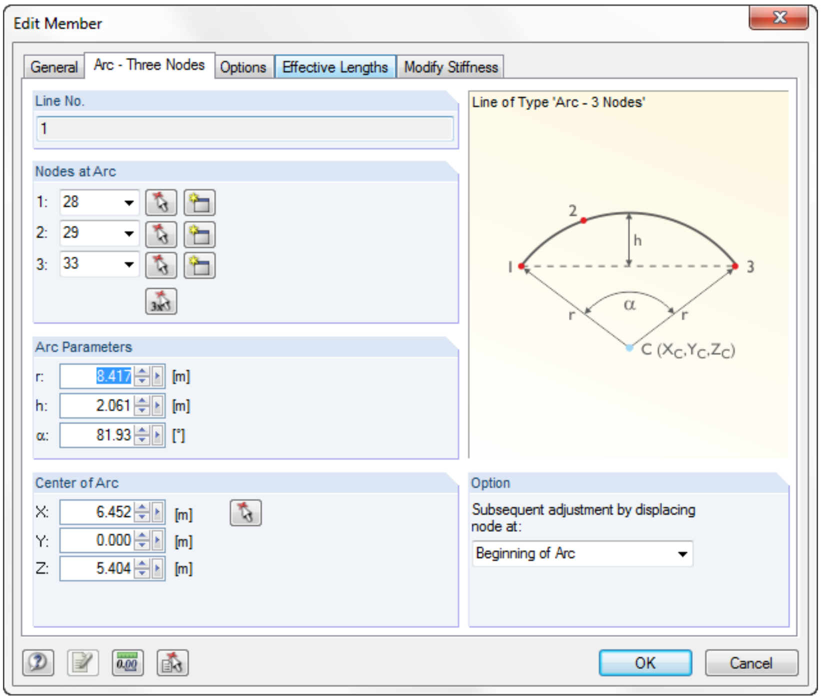


Figure 8.46 RFEM dialog box *Edit Member*

The designs are additionally checked by manual calculation.

Check of bending stress

Design according to [1] condition (6.41) for location x = 5.91 m:

$$\sigma_{m,y,d} = k_l \cdot \frac{M_{y,d}}{W_y} = 1.03 \cdot \frac{10666\ \text{kNcm}}{15413\ \text{cm}^3} = 0.71\ \text{kN/cm}^2$$

where

$$k_l = k_1 + k_2\left(\frac{h}{r}\right) + k_3\left(\frac{h}{r}\right)^2 + k_4\left(\frac{h}{r}\right)^3 = 1.0 + 0.35 \cdot \left(\frac{0.68\ \text{m}}{8.42\ \text{m}}\right) + 0.6 \cdot \left(\frac{0.68\ \text{m}}{8.42\ \text{m}}\right)^2 = 1.03$$

- $k_1 = 1.0$
- $k_2 = 0.35$
- $k_3 = 0.6$
- $k_4 = 0$

$$k_r = 0.76 + 0.001 \cdot \frac{r_{inside}}{t} = 0.76 + 0.001 \cdot \frac{8.075\ \text{m}}{0.033\ \text{m}} = 1.004$$

where

$$r_{inside} = r - 0.5 \cdot h = 8.417 - 0.5 \cdot 0.680 = 8.075\ \text{m}$$

As

$$\frac{r_{inside}}{t} > 240 \cdot k_r \quad \Rightarrow k_r = 1.0$$

$$f_{m,y,k} = 2.8\ \text{kN/cm}^2$$

$$f_{m,y,d} = \frac{2.8\ \text{kN/cm}^2 \cdot 0.9}{1.3} = 1.94\ \text{kN/cm}^2$$

Design:

$$\eta = \frac{\sigma_{m,y,d}}{k_r \cdot f_{m,y,d}} = \frac{0.71\ \text{kN/cm}^2}{1.0 \cdot 1.94\ \text{kN/cm}^2} = 0.36 < 1$$

Design of perpendicular tension stress

Design according to [1] condition (6.53) for location x = 6.406 m:

$$\sigma_{t,90,d} = k_p \cdot \frac{6 \cdot M_y}{b \cdot h^2} = 0.02 \cdot \frac{6 \cdot 10371\ \text{kNcm}}{20\ \text{cm} \cdot (68\ \text{cm})^2} = 0.0135\ \text{kN/cm}^2$$

where

$$k_p = k_5 + k_6\left(\frac{h}{r}\right) + k_7\left(\frac{h}{r}\right) = 0{,}25 \cdot \frac{0{,}68\ \text{m}}{8{,}42\ \text{m}} = 0{,}0202$$

- $k_5 = k_7 = 0$
- $k_2 = 0.25$

$$k_{dis} = 1.4$$

$$k_{vol} = \left(\frac{V_0}{V}\right)^2 = \left(\frac{0.01}{1.63}\right)^2 = 0.361$$

$$f_{t,90,k} = 0.045\ \text{kN/cm}^2$$

$$f_{t,90,d} = \frac{0.045\ \text{kN/cm}^2 \cdot 0.9}{1.3} = 0.031\ \text{kN/cm}^2$$

$$b_{eff} = k_{cr} \cdot b = 0.781 \cdot 20 = 15.62\ \text{cm}$$

$$\tau_d = 1.5 \cdot \frac{V_{z,d}}{b_{eff} \cdot h} = 1.5 \cdot \frac{7.31}{15.62 \cdot 68} = 0.010\ \text{kN/cm}^2$$

$$f_{v,k} = 0.35\ \text{kN/cm}^2$$

$$f_{v,d} = \frac{0.35\ \text{kN/cm}^2 \cdot 0.9}{1.3} = 0.24\ \text{kN/cm}^2$$

Design:

$$\eta = \frac{\tau_d}{f_{v,d}} + \frac{\sigma_{t,90,d}}{k_{dis} \cdot k_{vol} \cdot f_{t,90,d}} = \frac{0.010\ \text{kN/cm}^2}{0.24\ \text{kN/cm}^2} + \frac{0.0135\ \text{kN/cm}^2}{1.4 \cdot 0.361 \cdot 0.031\ \text{kN/cm}^2} = 0.91 < 1$$

9 Literature

[1] Eurocode 5: Design of timber structures - Part 1-1: General - Common rules and rules for buildings; EN 1995-1-1:2010-12

[2] DIN 1052:2008-12: Design of timber structures - General rules and rules for buildings. Beuth Verlag GmbH, Berlin, 2008.

[3] SIA 265:2012: Holzbau. Schweizerischer Ingenieur- und Architektenverein, Zürich, 2012

[4] DIN 4102-22: Fire behaviour of building materials and building components - Part 22: Application standard for DIN 4102-4 based on the design of partial safety factors. Beuth Verlag GmbH, Berlin, 11, 2004.

[5] Eurocode 5: Design of timber structures - Part 1-2: General - Structural fire design; German version EN 1995-1-2:2004 + AC:2009. Beuth Verlag GmbH, Berlin, 10, 2006.

[6] Holzbau-Taschenbuch, Band 1: Grundlagen, Entwurf, Bemessung und Konstruktionen. Ernst & Sohn, 9. Auflage 1996

[7] Boddenberg, R.-W.: Vorlesung Holzbau II Sommersemester 2015. Wismar: Hochschule Wismar, Fachgebiet Baustatik und Holzbau, 2015

[8] Blass, H., Ehlbeck, J., Kreuzinger, H., & Steck, G. (2005). *Erläuterungen zu DIN 1052: Entwurf, Berechnung und Bemessung von Holzbauwerken* (2nd ed.). Karlsruhe: Bruderverlag.

[9] Albert, A.: Schneider - Bautabellen für Ingenieure mit Berechnungshinweisen und Beispielen, 23. Auflage. Köln: Bundesanzeiger, 2018

[10] Bauen mit Holz. Bruderverlag Albert Bruder GmbH Co. KG, 04, 2012.