Version
September 2017

Add-on Module

RF-/TIMBER NBR

Design of Timber Members According to NBR 7190:1997

Program Description

Am Zellweg 2
93464 Tiefenbach
Germany

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

Contents

1 Introduction

1.1 Add-on Module RF-/TIMBER NBR

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

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

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

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

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

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

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

Your DLUBAL team

1.2 Using the Manual

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

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

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

1.3 Open RF-/TIMBER NBR

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

Menu

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

Add-on Modules → **Design - Timber** → **RF-/TIMBER NBR**.

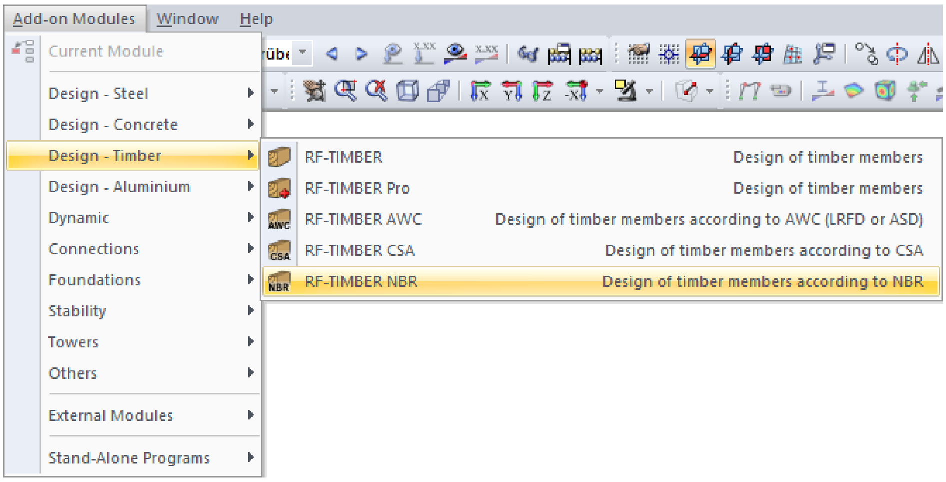


Figure 1.1: Menu *Add-on Modules → Design - Timber → RF-TIMBER NBR*

Navigator

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

Add-on Modules → **RF-/TIMBER NBR**.

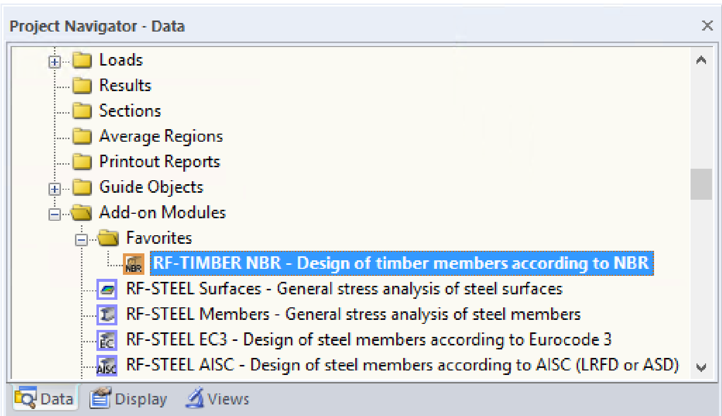
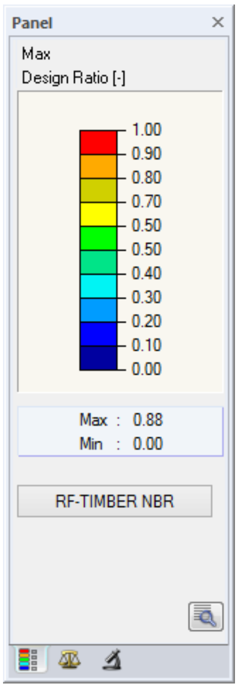


Figure 1.2: Data navigator: *Add-on Modules → RF-TIMBER NBR*



Panel

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

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

2 Input Data

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

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

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

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

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

2.1 General Data

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

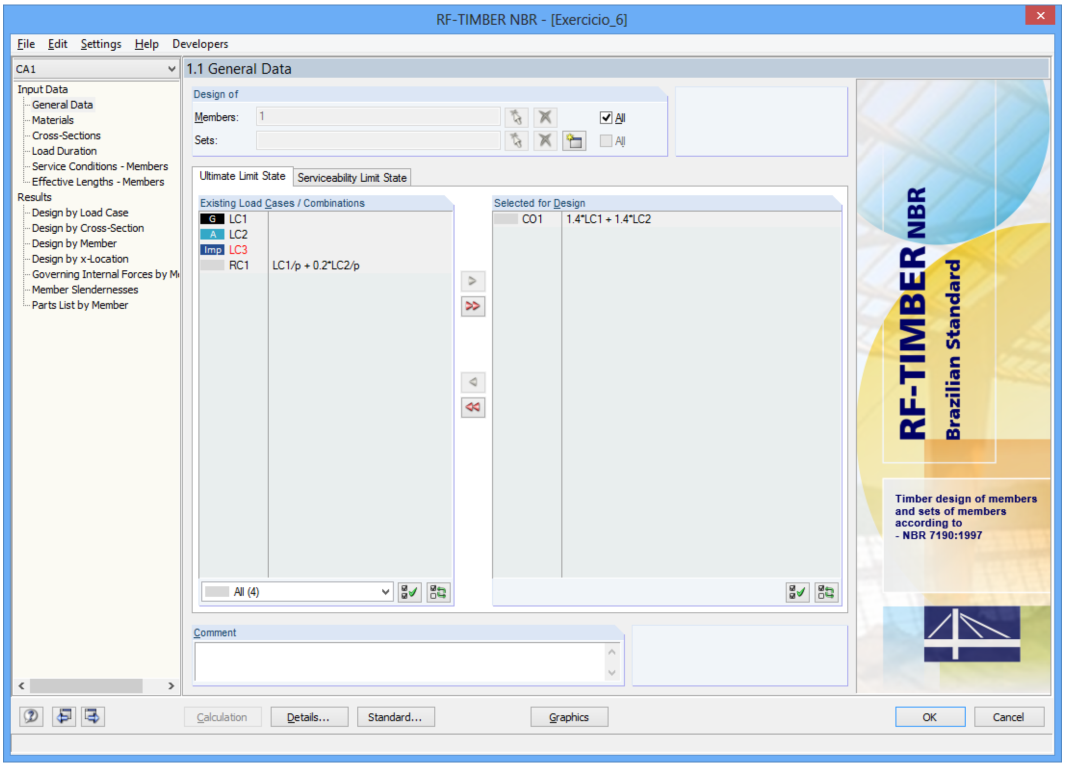


Figure 2.1: Window *1.1 General Data*

Design of

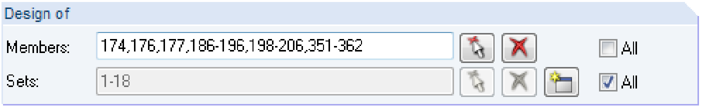


Figure 2.2: Design of members and sets of members

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

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

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

Comment

Figure 2.3: User-defined comment

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

2.1.1 Ultimate Limit State

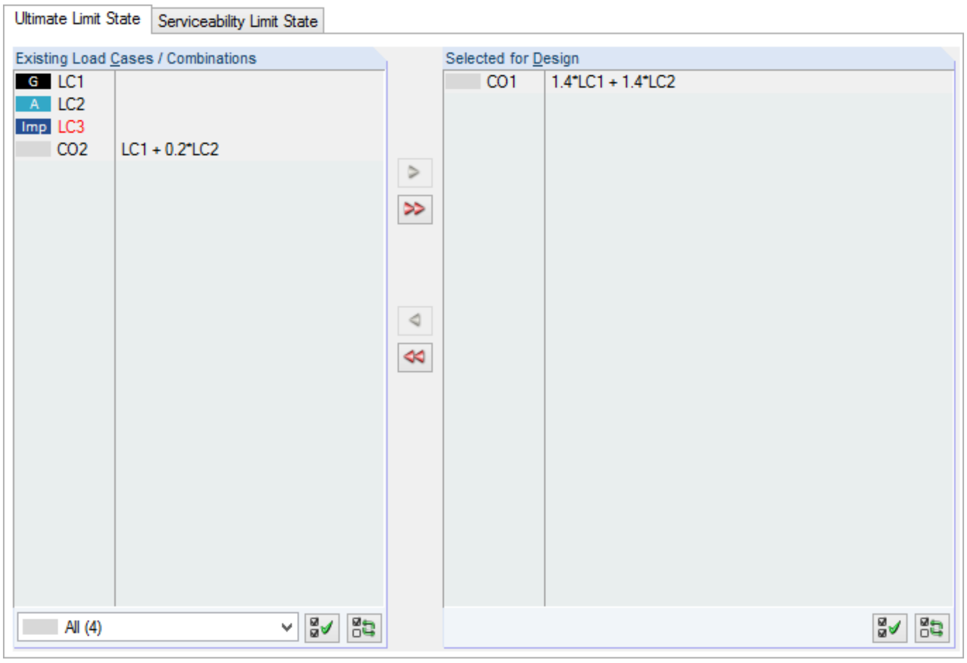


Figure 2.4: Window *1.1 General Data*, tab *Ultimate Limit State*

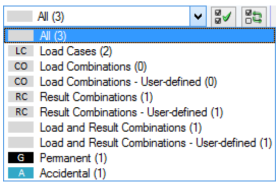
Existing Load Cases and Combinations

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

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

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

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



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

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

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

Selected for Design

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

Result combination

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

2.1.2 Serviceability Limit State

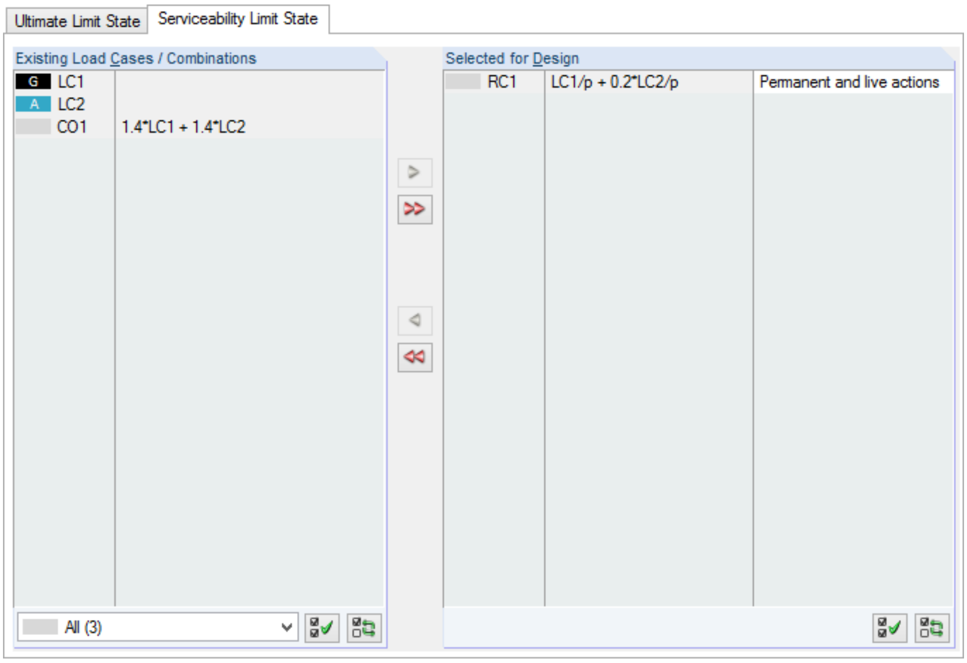


Figure 2.5: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

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

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

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

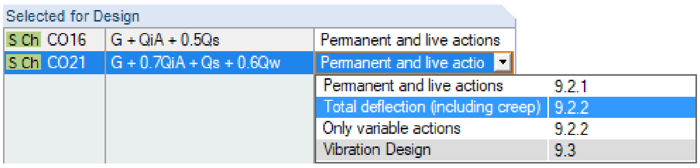


Figure 2.6: Selecting design situtation

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

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

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

2.1.3 Standard

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

Modification Factors

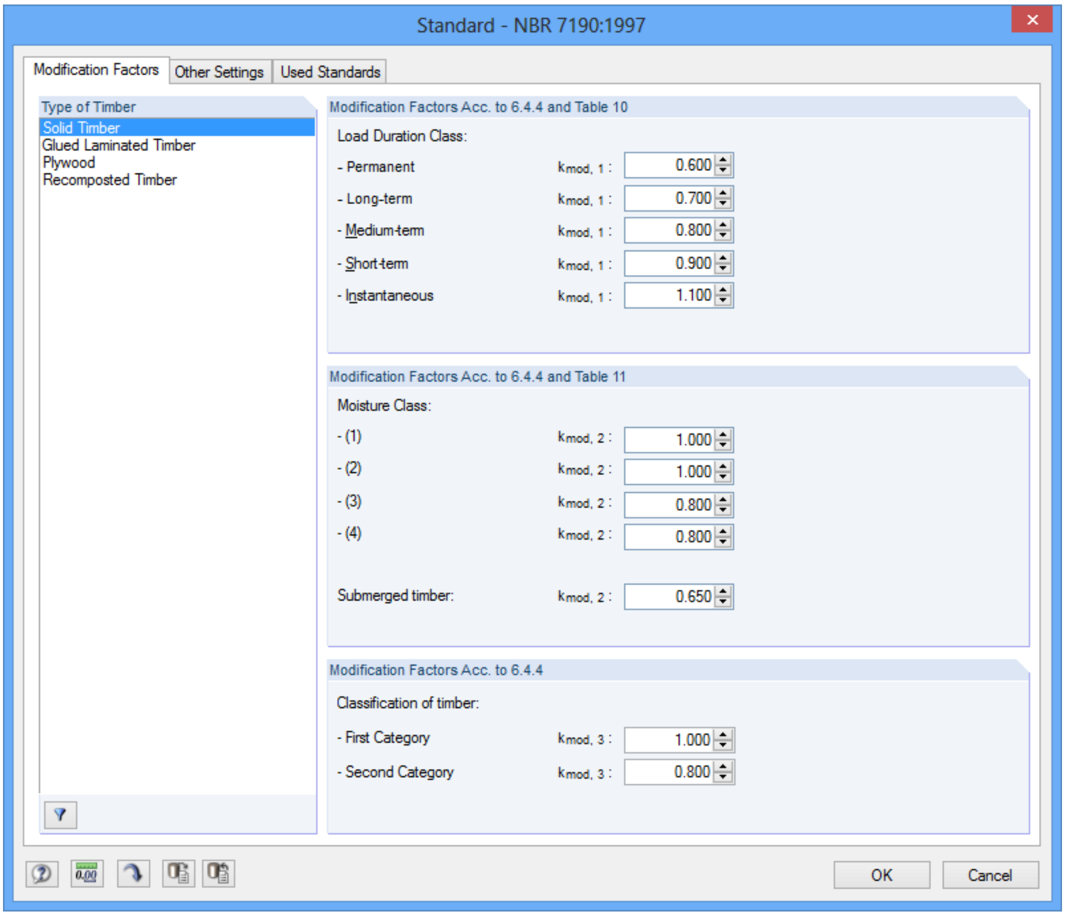


Figure 2.7: Dialog box *Standard*, tab *Modification Factors*

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

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

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

Table 2.2: Buttons in the *Standard* dialog box

Other Settings

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

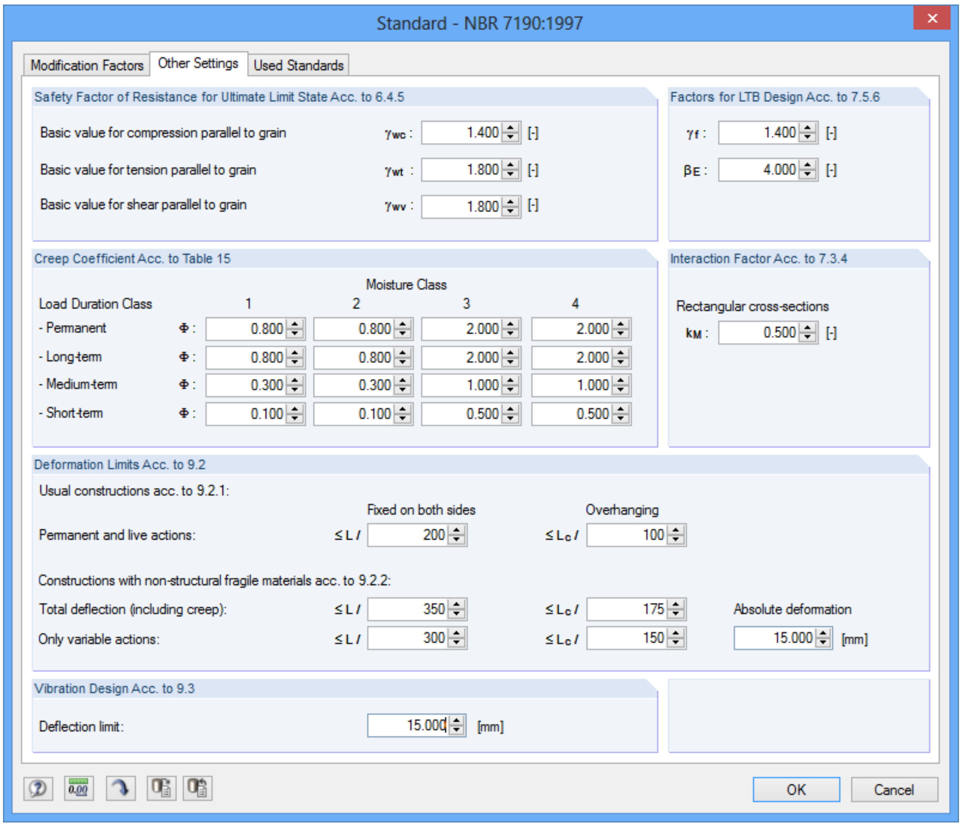


Figure 2.8: Dialog box *Standard*, tab *Others*

Used Standards

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

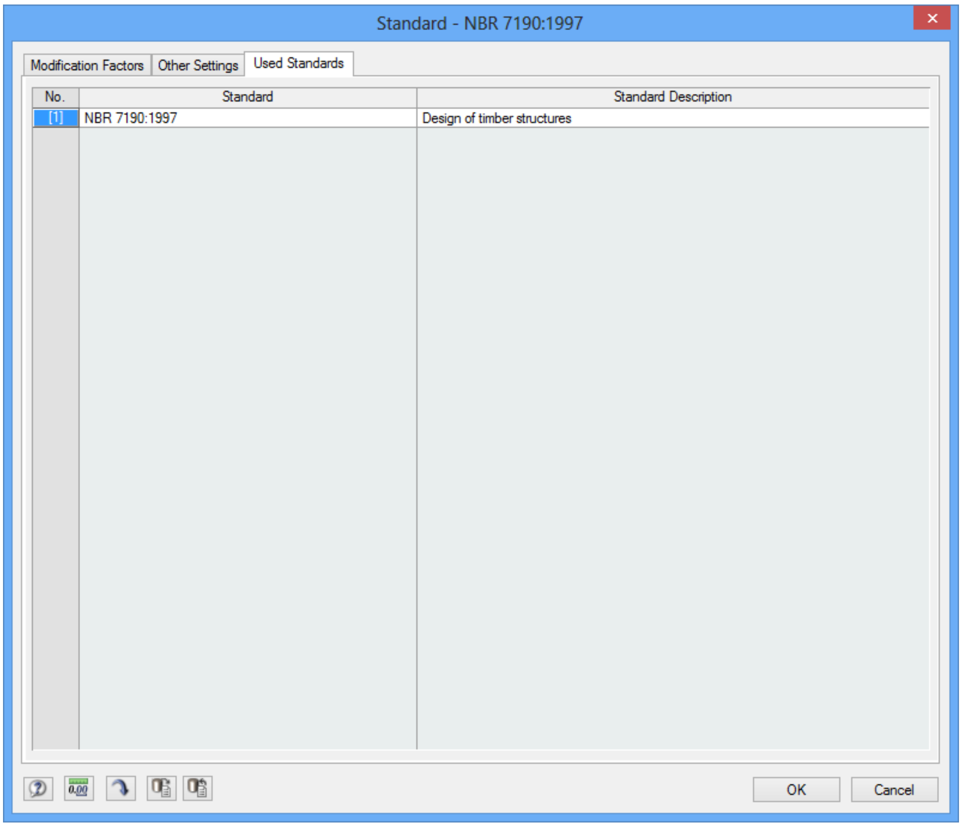


Figure 2.9: Dialog box *Standard*, tab *Used Standards*

2.2 Materials

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

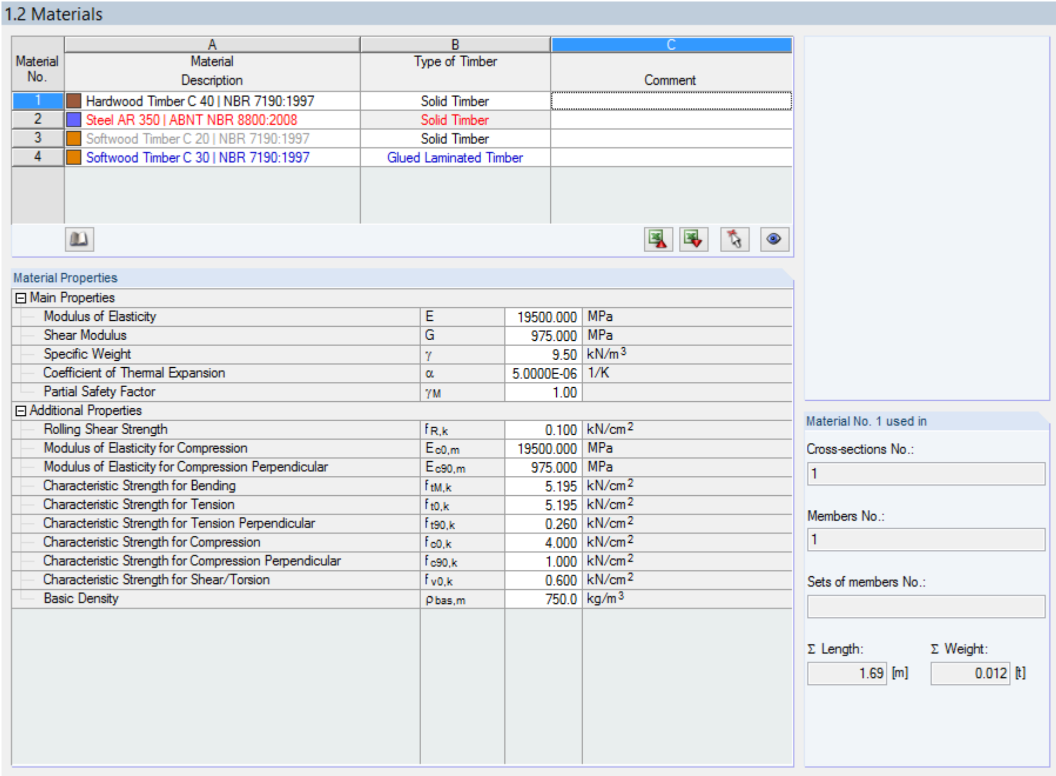


Figure 2.10: Window *1.2 Materials*

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

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

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

Material Description

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

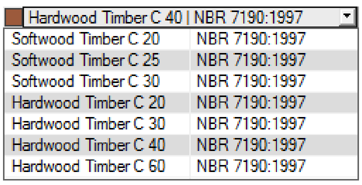


Figure 2.11: List of materials

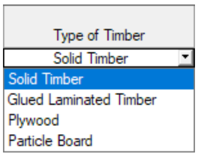
According to the design concept of the NBR 7190:1997 [1], the list includes only materials of the *NBR* standard group.

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

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

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

Type of Timber



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

Material Library

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

Edit → Material Library

or click the corresponding button.

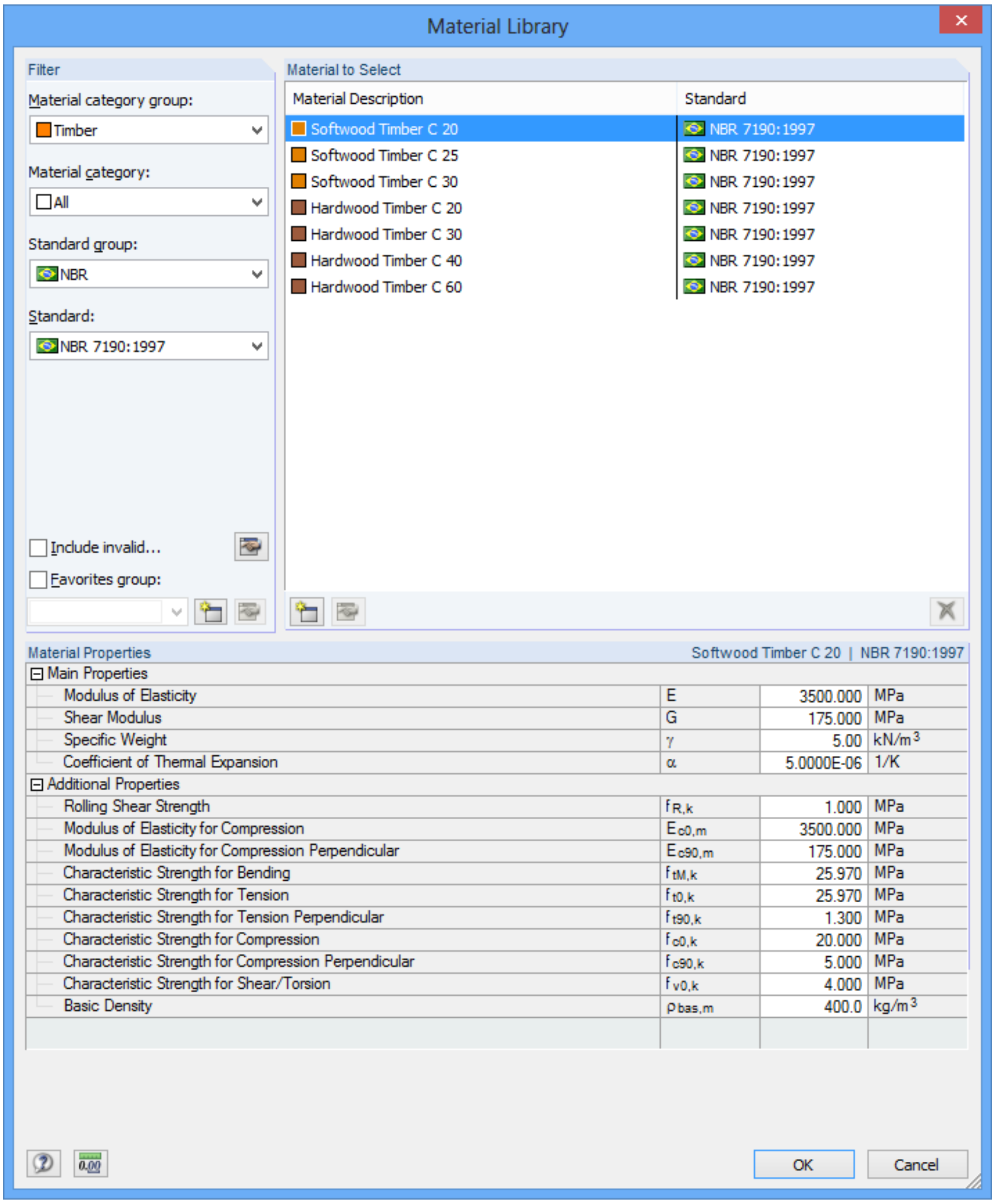


Figure 2.12: Dialog box *Material Library*

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

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

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

Material Properties

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

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

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

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

2.3 Cross-Sections

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

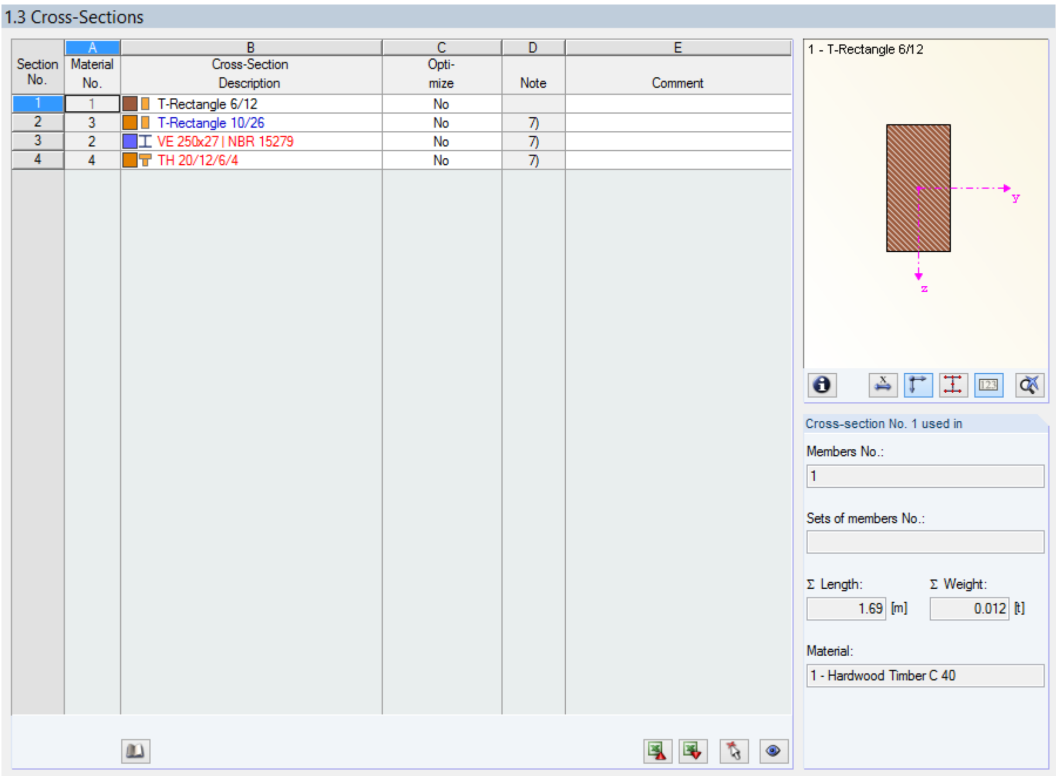
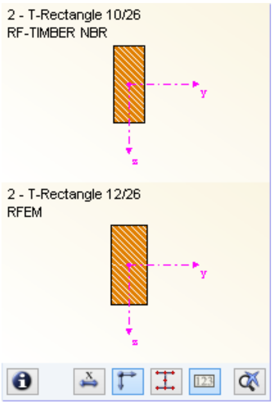


Figure 2.13: Window *1.3 Cross-Sections*

Cross-Section Description



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

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

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

Modified cross-sections are highlighted in blue.

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

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

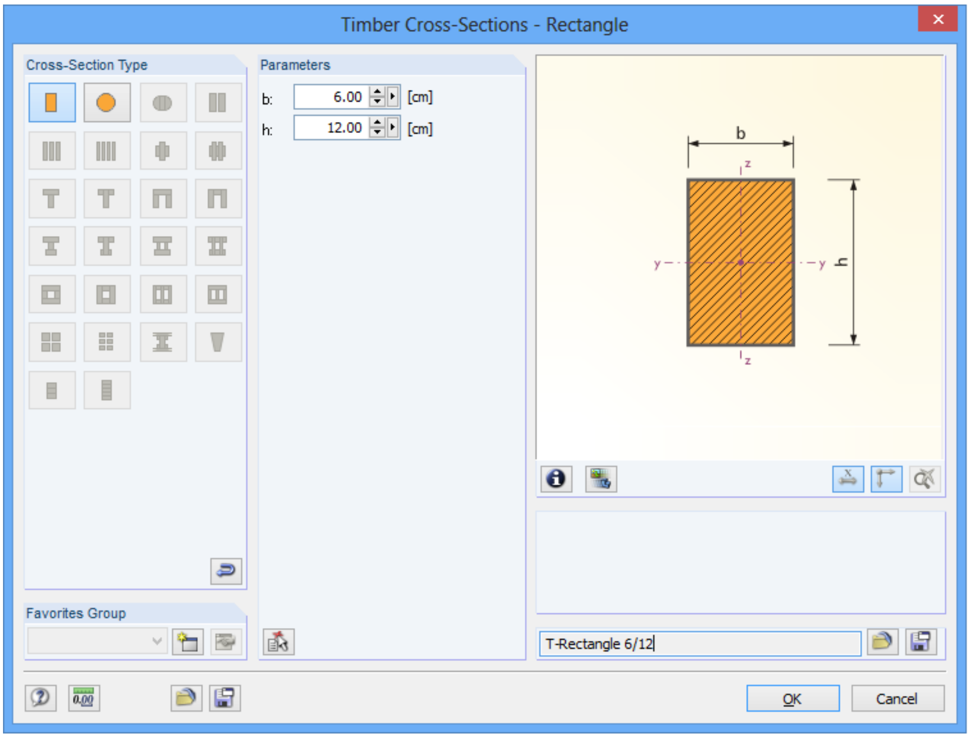


Figure 2.14: Parametric *Timber Cross-Sections* of the library

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

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

Max. Design Ratio

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

Optimize

Details...

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

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

Note

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

Info About Cross-Section

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

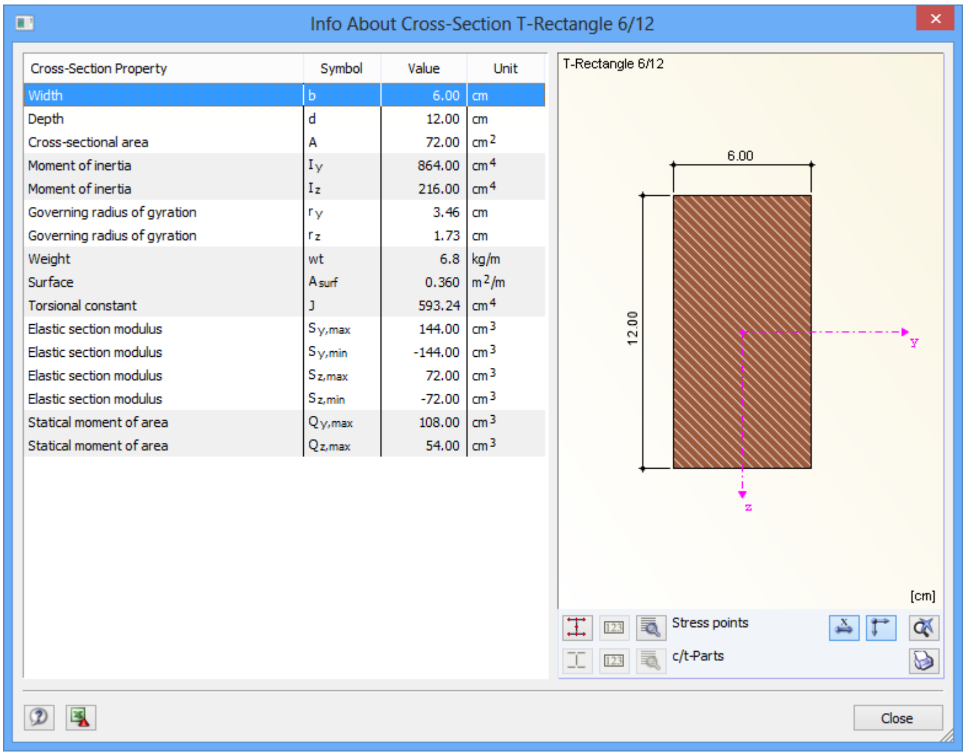


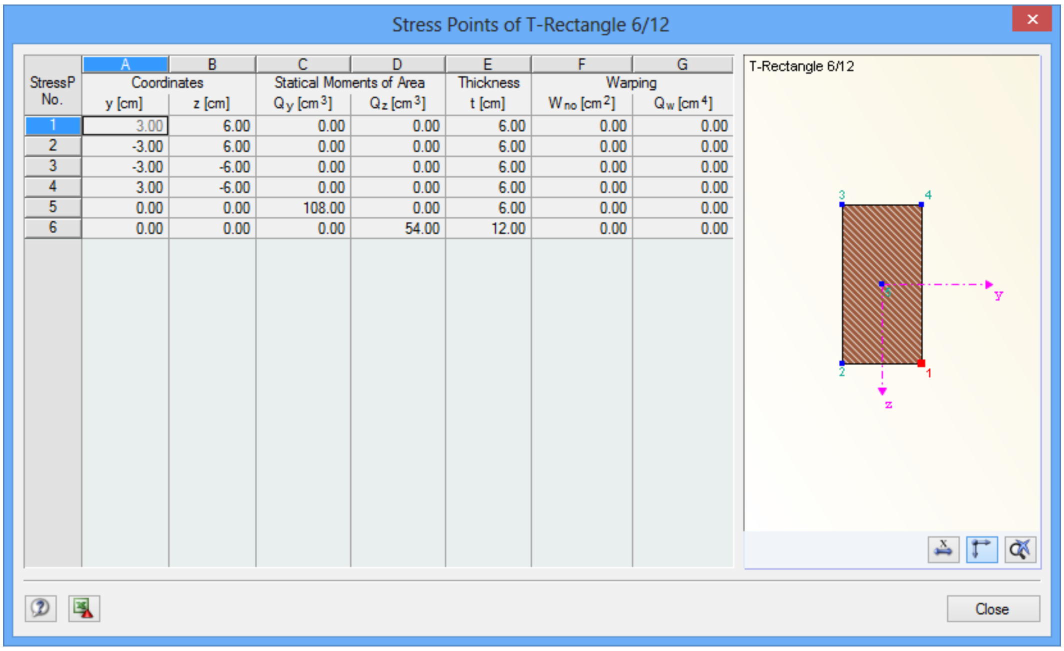
Figure 2.15: Dialog box *Info About Cross-Section*

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

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

Table 2.3: Buttons of cross-section graphic

To show specific information on the stress points (distances to center of gravity, statical moments etc.), click [Details] and use the appearing dialog box.

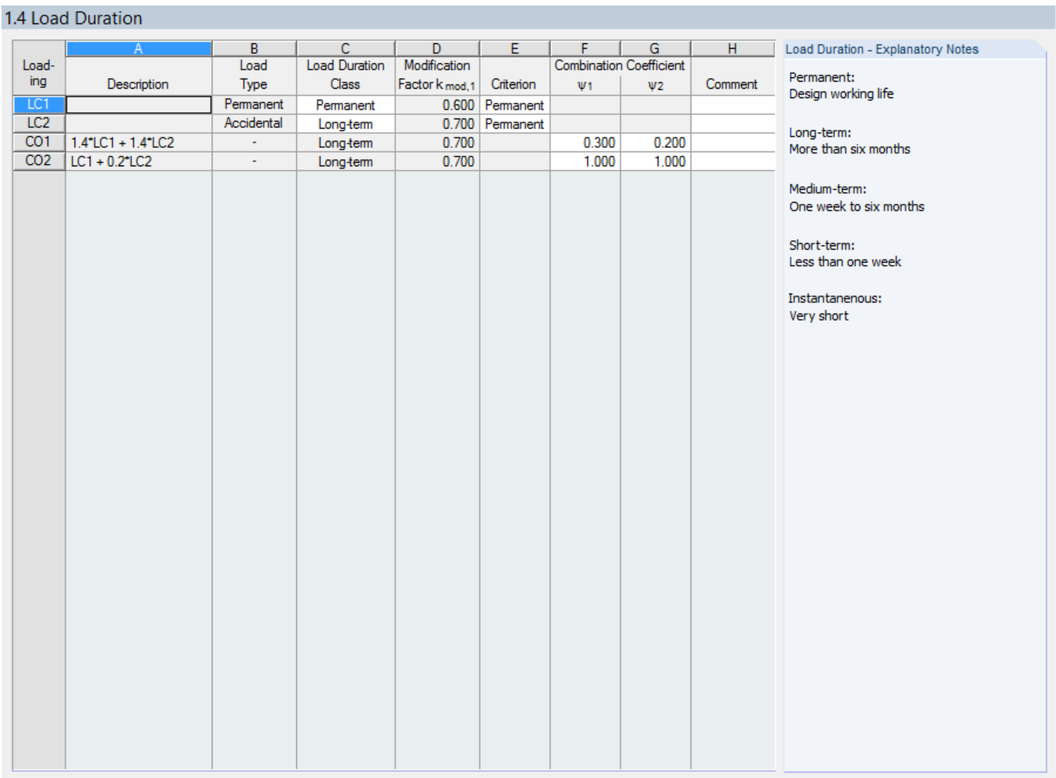


StressP No.	A Coordinates y [cm]	B z [cm]	C Statical Moments of Area Q_y [cm³]	D Q_z [cm³]	E Thickness t [cm]	F Warping W_{no} [cm²]	G Q_w [cm⁴]
1	3.00	6.00	0.00	0.00	6.00	0.00	0.00
2	-3.00	6.00	0.00	0.00	6.00	0.00	0.00
3	-3.00	-6.00	0.00	0.00	6.00	0.00	0.00
4	3.00	-6.00	0.00	0.00	6.00	0.00	0.00
5	0.00	0.00	108.00	0.00	6.00	0.00	0.00
6	0.00	0.00	0.00	54.00	12.00	0.00	0.00

Figure 2.16: Dialog box *Stress Points*

2.4 Load Duration

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



Loading	A Description	B Load Type	C Load Duration Class	D Modification Factor $k_{mod,1}$	E Criterion	F Combination Coefficient ψ_1	G ψ_2	H Comment
LC1		Permanent	Permanent	0.600	Permanent			
LC2		Accidental	Long-term	0.700	Permanent			
CO1	1.4*LC1 + 1.4*LC2	-	Long-term	0.700		0.300	0.200	
CO2	LC1 + 0.2*LC2	-	Long-term	0.700		1.000	1.000	

Figure 2.17: Window *1.4 Load Duration*

Loading

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

Description

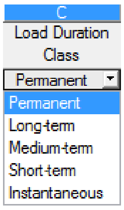
The descriptions of the load cases help you to classify them.

Load Type

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

Load Duration Class

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



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

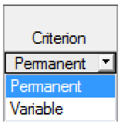
The load duration class is required to determine the modification factor, $k_{mod,1}$. This factor is also considered for the material stiffness.

Modification Factor $k_{mod,1}$

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

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

Criterion



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

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

Combination Coefficient ψ_1 / ψ_2

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

2.5 Service Conditions - Members

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

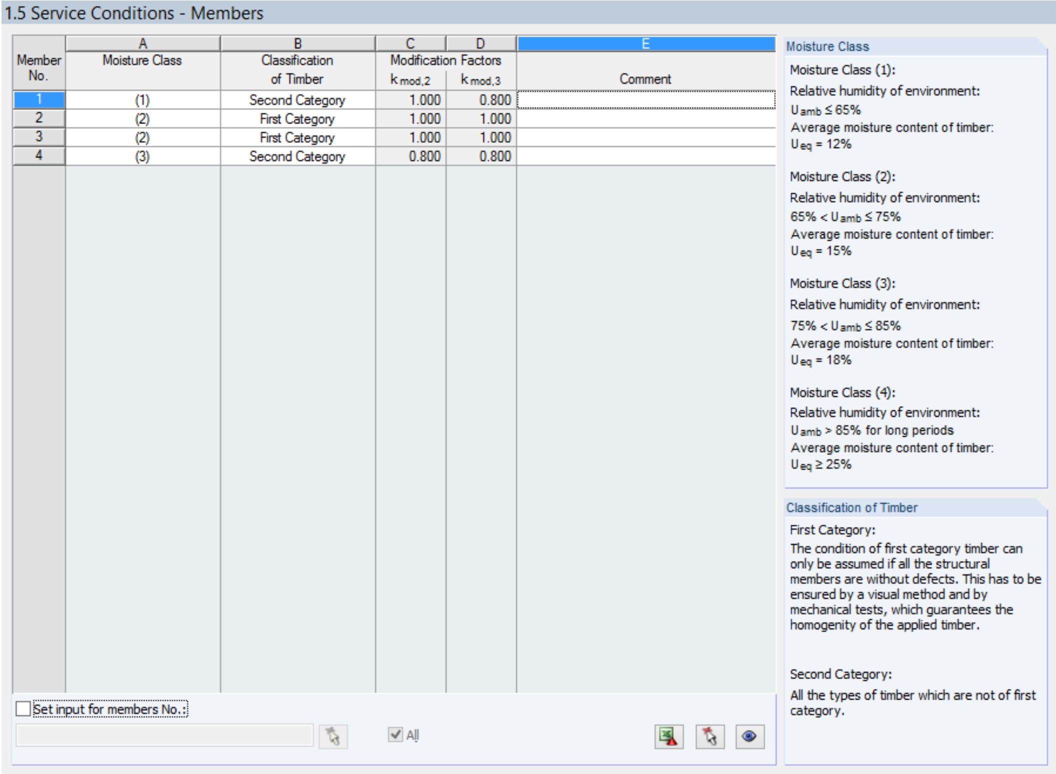
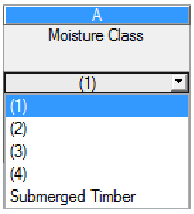
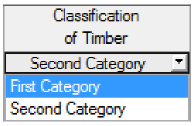


Figure 2.18: Window *1.5 Service Conditions – Members*



Moisture Class

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



Classification of Timber

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

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

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

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

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

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

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

Table 2.4: Buttons in Window *1.5 Service Conditions – Members*

2.6 Service Conditions - Set of Members

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

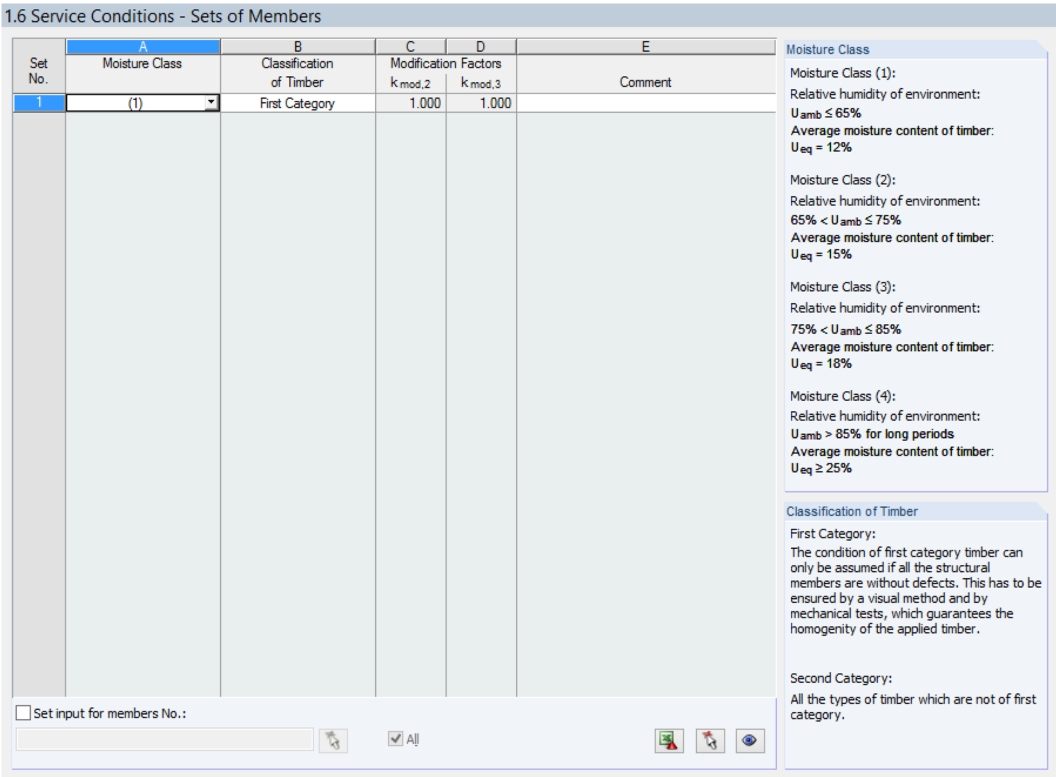


Figure 2.19: Window *1.6 Service Conditions - Sets of Members*

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

2.7 Effective Lengths - Members

Details...

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

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

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

Click to select a member graphically and show its row.

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

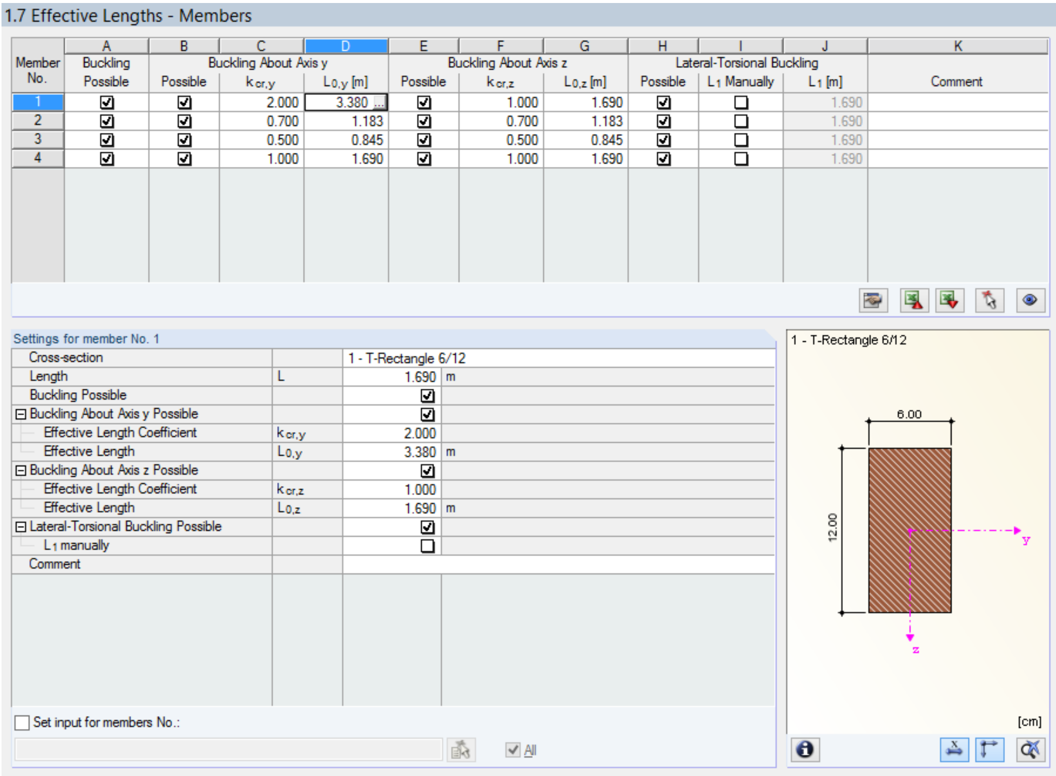


Figure 2.20: Window *1.7 Effective Lengths - Members* for stability analysis according to [1] 7.5

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

The *Settings* tree manages the following parameters:

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

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

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

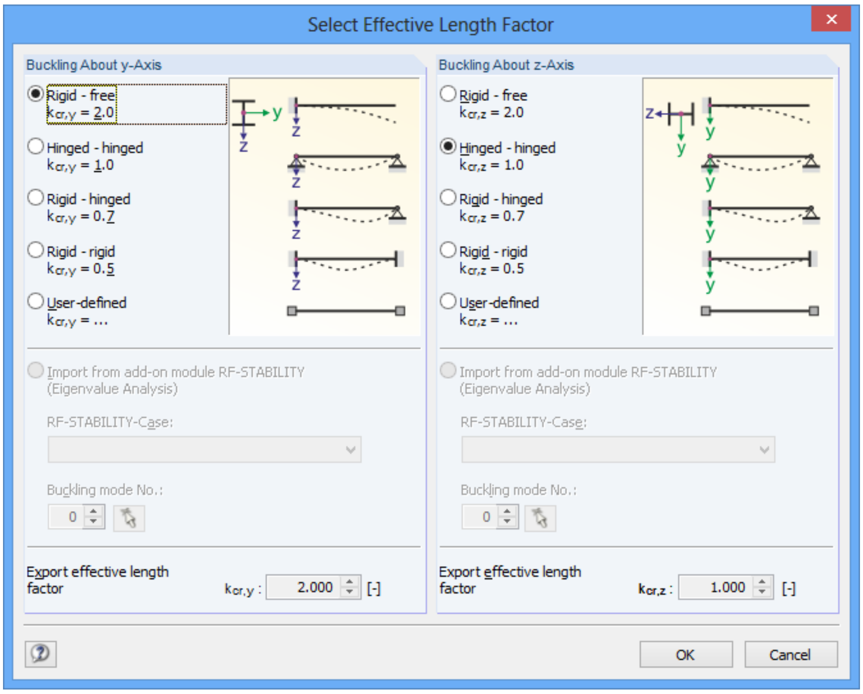


Figure 2.21: Dialog box *Select Effective Length Factor*

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

Buckling Possible

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

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

Buckling About Axis y or Axis z

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

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

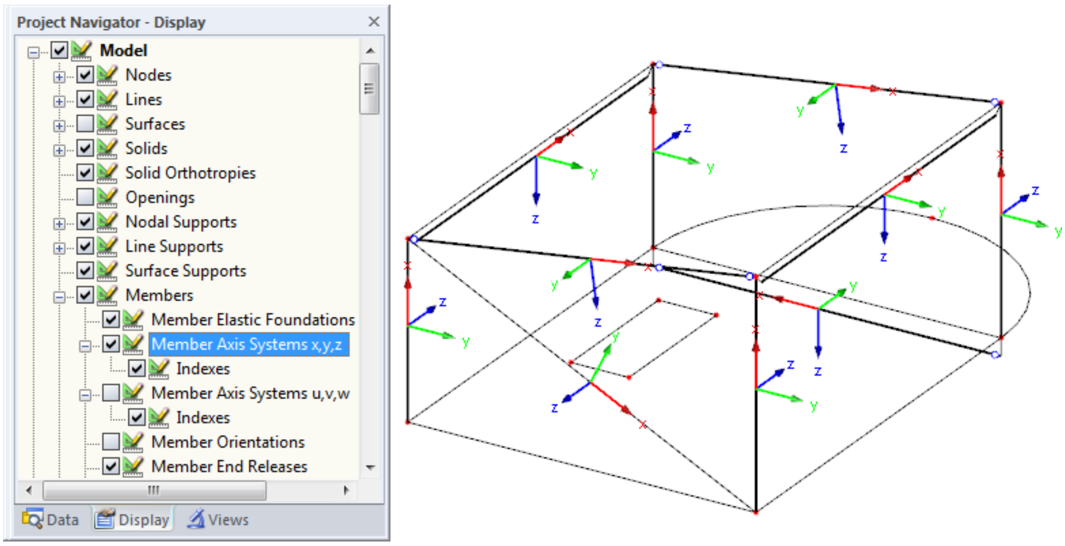


Figure 2.22: Activating the member axis systems in the *Display* navigator of RFEM

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

To specify the buckling lengths in the work window graphically, click . This button is available when you have selected a cell of the L_0 column (see Figure 2.20).

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

Lateral-Torsional Buckling Possible

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

L_1 Manually

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

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

Comment

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

2.8 Effective Lengths - Sets of Members

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

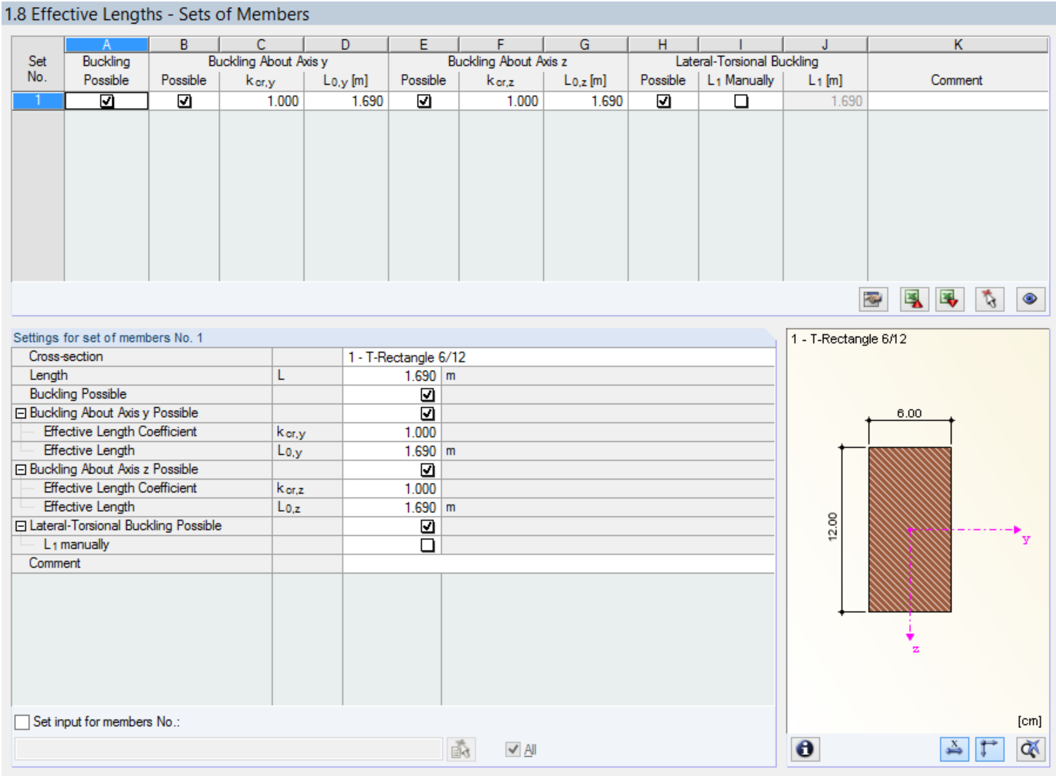
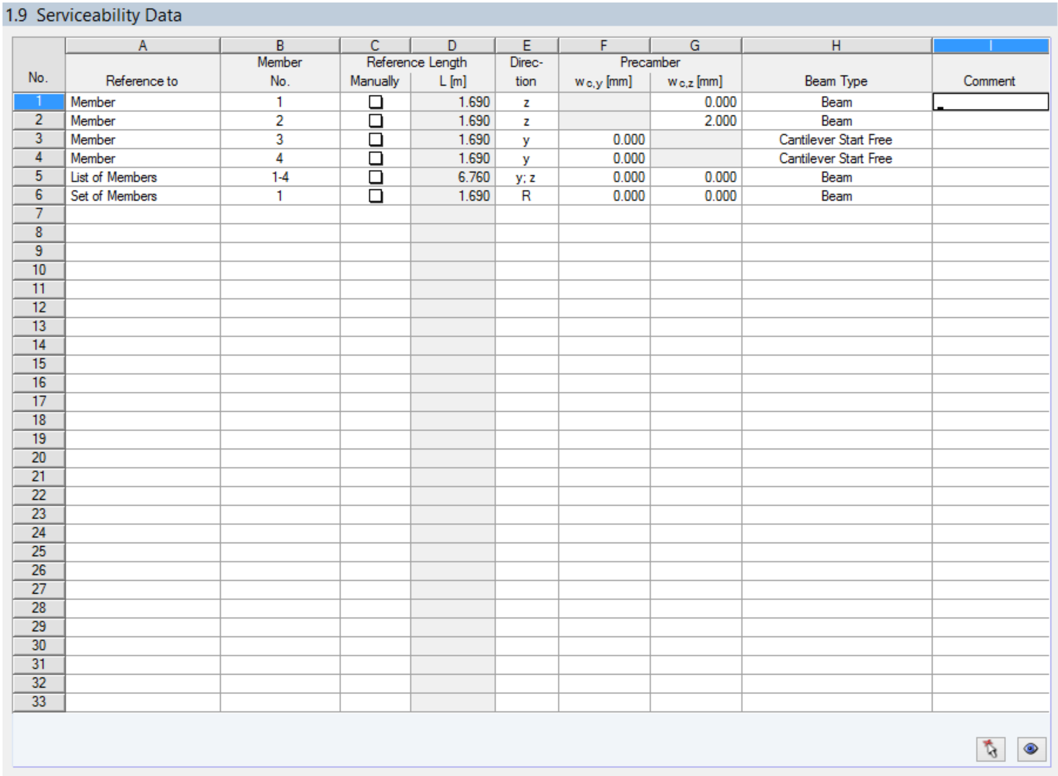


Figure 2.23: Window *1.8 Effective Lengths - Sets of Members*

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

2.9 Serviceability Data

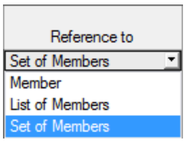
The last input window controls the settings for the serviceability limit state design of specific objects. It is available when one or more load cases or combinations have been selected in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 8).



1.9 Serviceability Data

No.	A Reference to	B Member 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	Member	1	☐	1.690	z		0.000	Beam	
2	Member	2	☐	1.690	z		2.000	Beam	
3	Member	3	☐	1.690	y	0.000		Cantilever Start Free	
4	Member	4	☐	1.690	y	0.000		Cantilever Start Free	
5	List of Members	1-4	☐	6.760	y; z	0.000	0.000	Beam	
6	Set of Members	1	☐	1.690	R	0.000	0.000	Beam	
7									
8									
9									
10									
11									
12									
13									
14									
15									
16									
17									
18									
19									
20									
21									
22									
23									
24									
25									
26									
27									
28									
29									
30									
31									
32									
33									

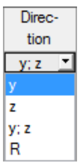
Figure 2.24: Window *1.9 Serviceability Data*



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

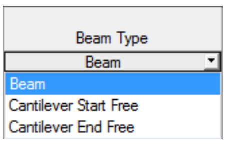
For a list or set of members, the orientation and rotation of all contained members must be identical. This will guarantee that the components of the deformation are taken into account correctly.

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

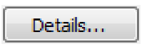


Column E controls the governing *Direction* for the deformation analysis. You can select the directions of the local member axes *y* and *z* or the resultant *R*.

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



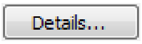
The *Beam Type* is important for the correct reference to the limit deformations. In column G, you can specify a beam or a cantilever is to be analyzed. For the latter, you can define which end has no support.



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

3 Calculation

3.1 Detail Settings



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

The *Details* dialog box contains the following tabs:

- Resistance
- Stability
- Serviceability
- Other

3.1.1 Resistance

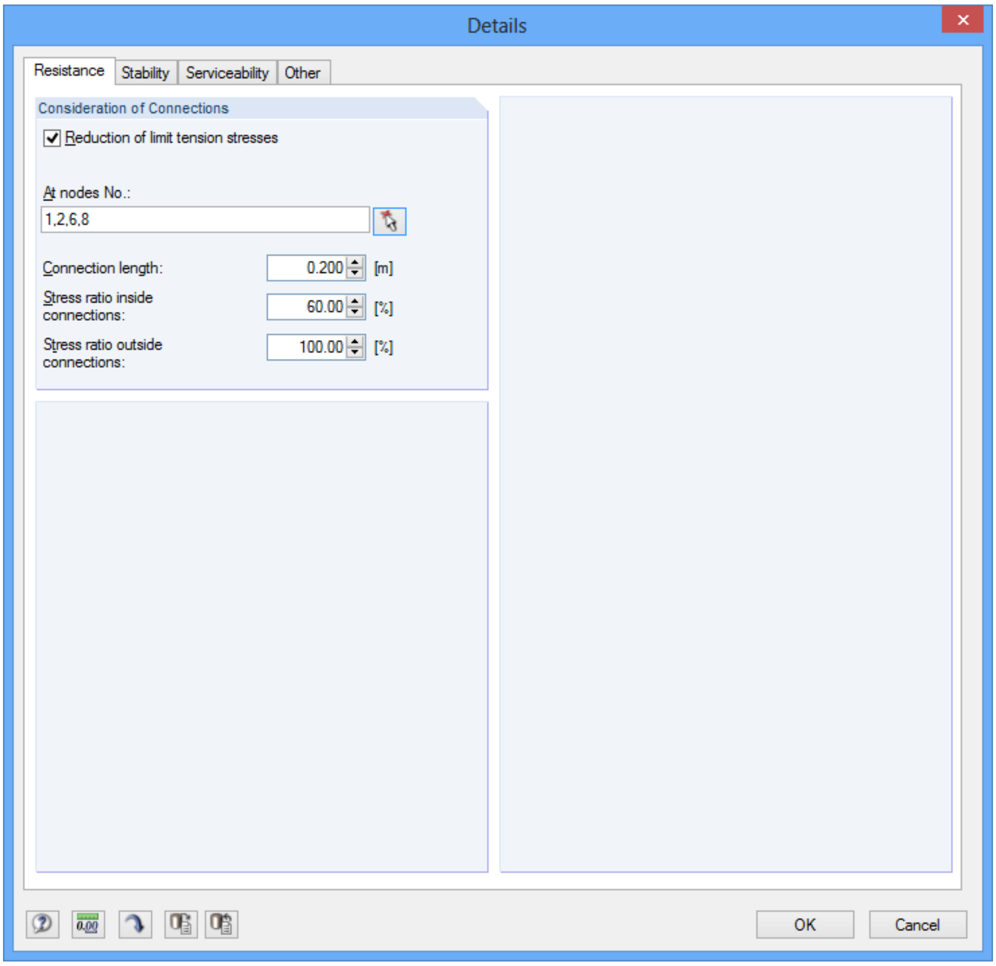


Figure 3.1: Dialog box *Details*, tab *Resistance*

Consideration of Connections

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

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

3.1.2 Stability

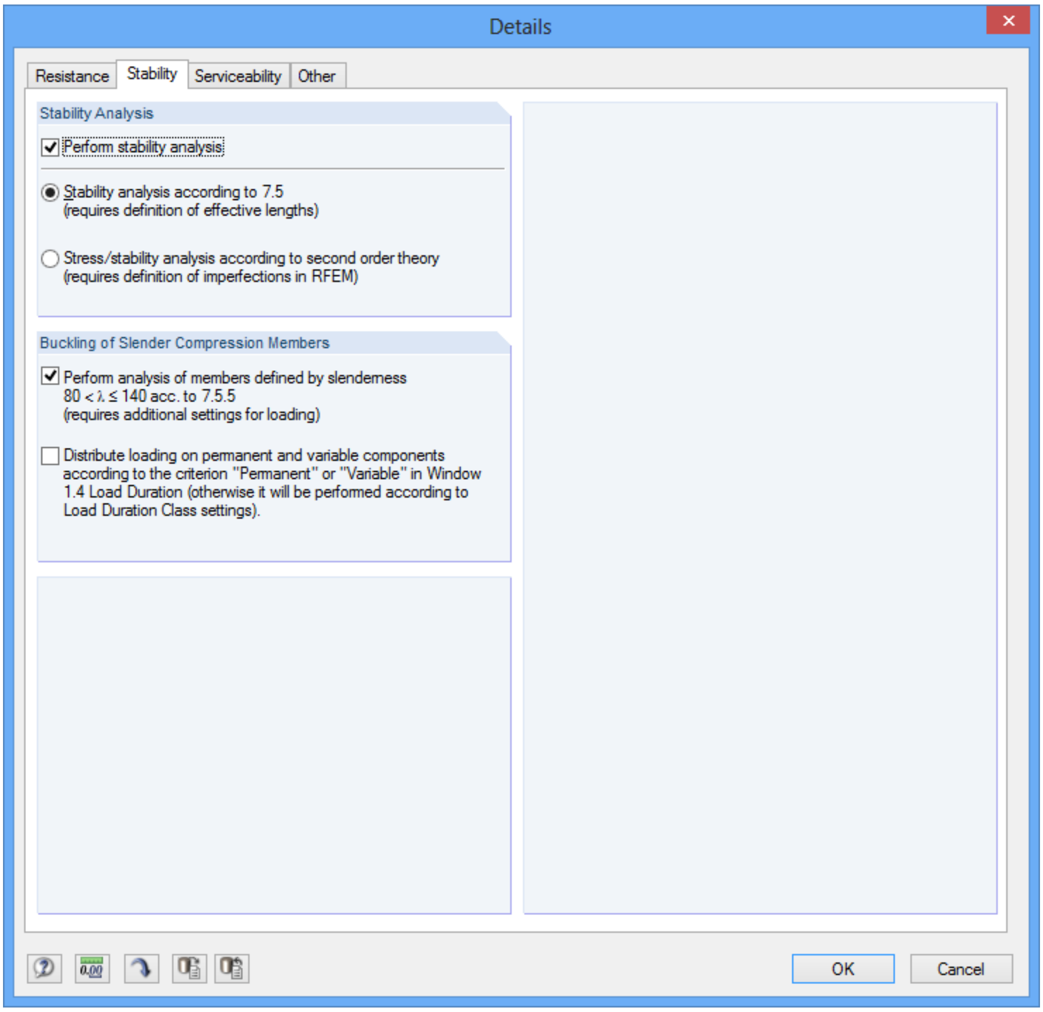
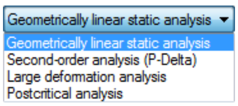


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

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



Method of analysis in RFEM/RSTAB

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

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

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

Buckling of Slender Compression Members

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

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

3.1.3 Serviceability

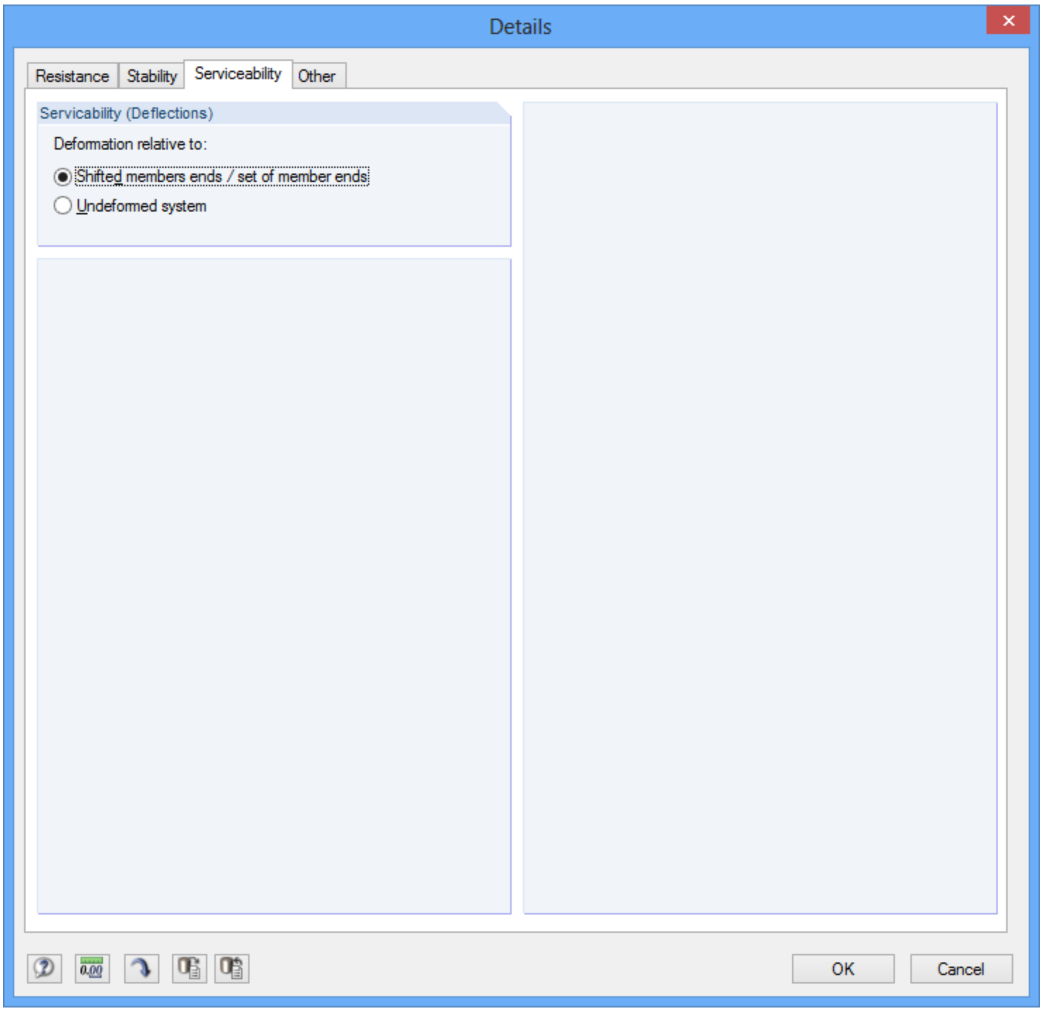


Figure 3.3: Dialog box *Details*, tab *Serviceability*

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

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

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

3.1.4 Other

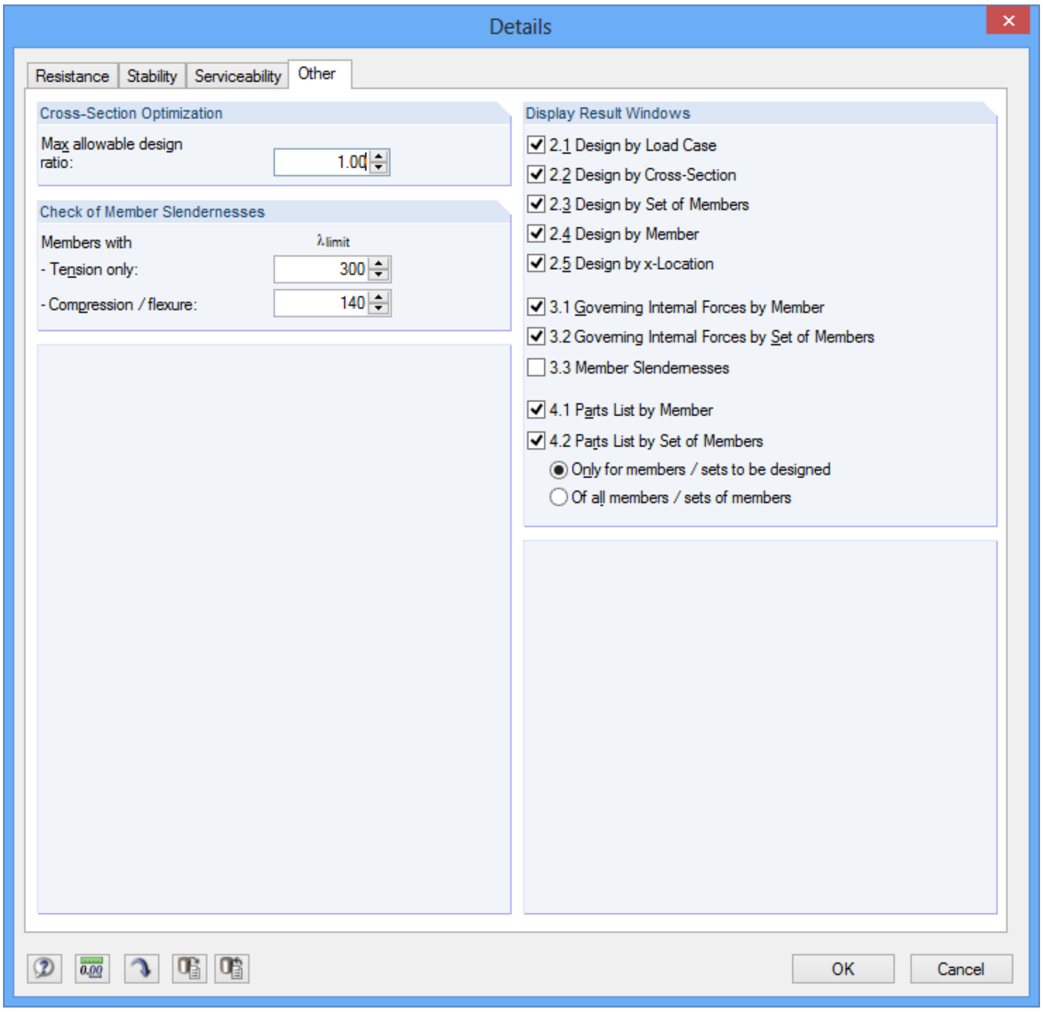


Figure 3.4: Dialog box *Details*, tab *Other*

Cross-Section Optimization

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

Check of Member Slendernesses

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

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

Display Result Windows

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

Window *3.3 Member Slendernesses* is deactivated by default.

3.2 Start Calculation

Calculation

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

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

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

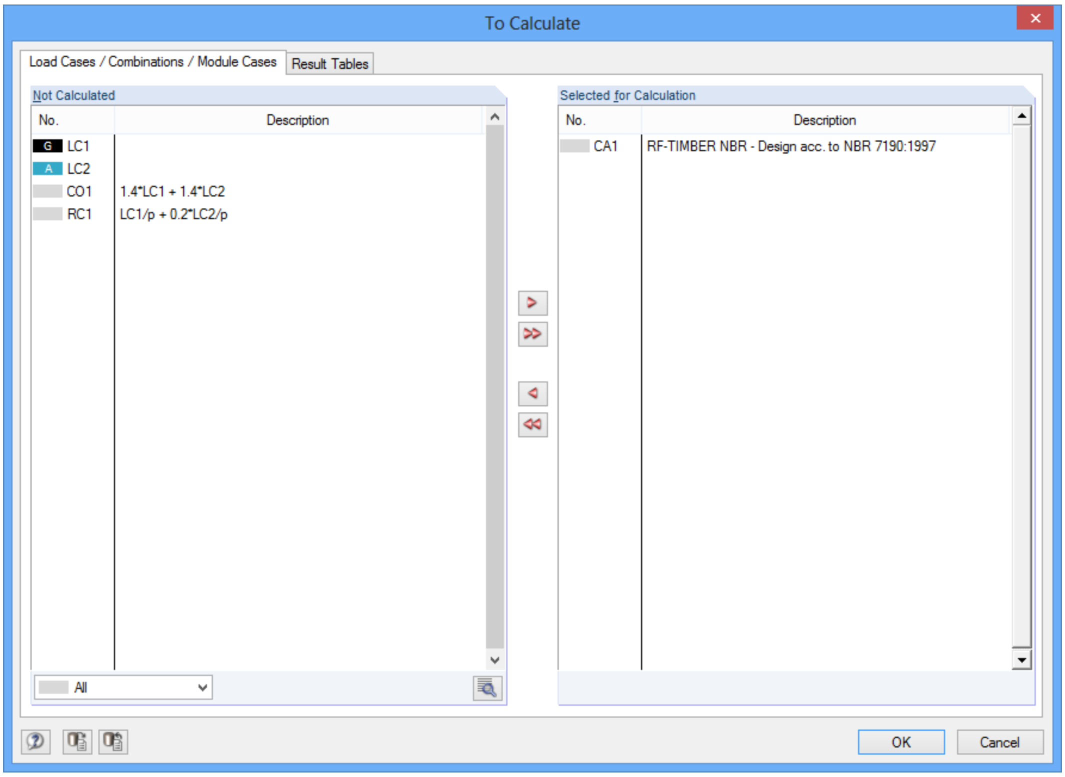
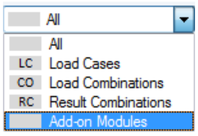


Figure 3.5: Dialog box *To Calculate* of RFEM



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

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

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

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

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

4 Results

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

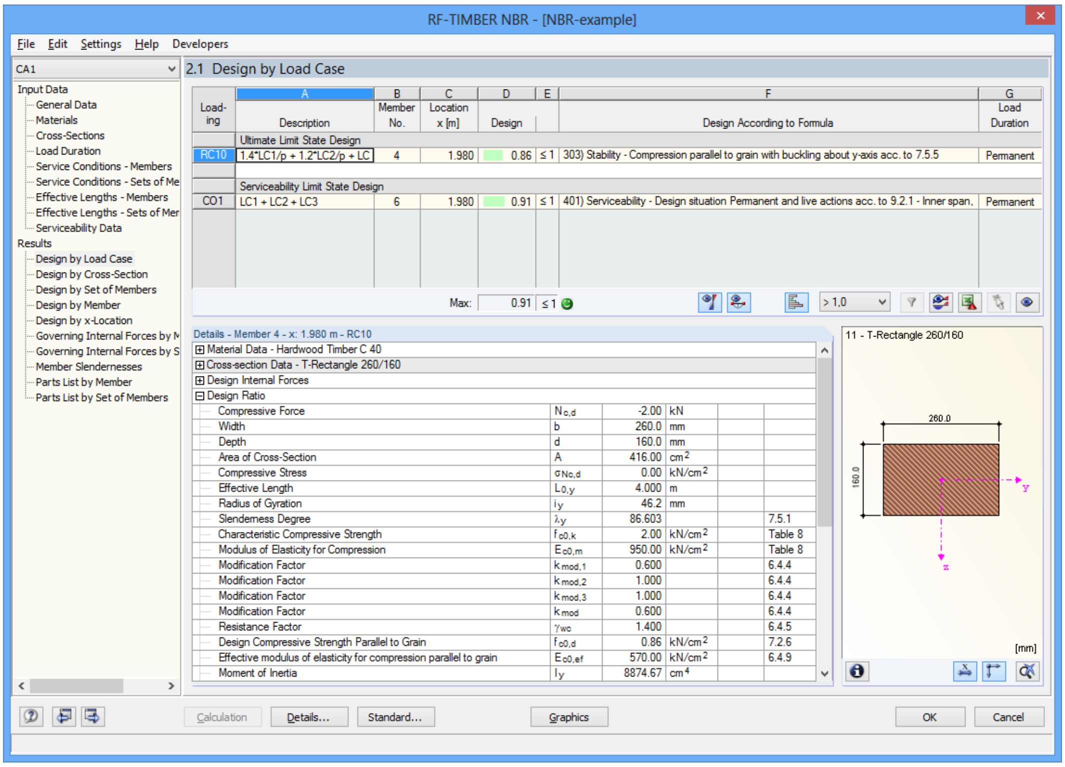
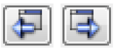


Figure 4.1: Result window with design results and details

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

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



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

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

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

4.1 Design by Load Case

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

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

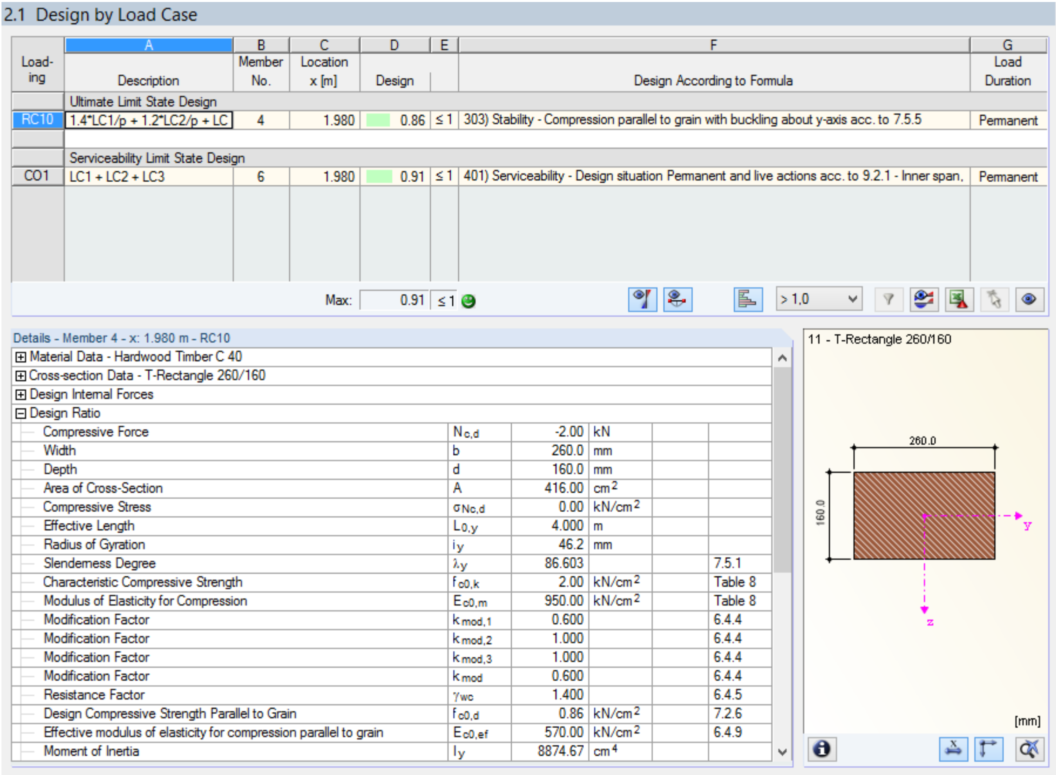


Figure 4.2: Window *2.1 Design by Load Case*

Description

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

Member No.

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

Location x

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

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

Design

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

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

Design According to Formula

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

Load Duration

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

4.2 Design by Cross-Section

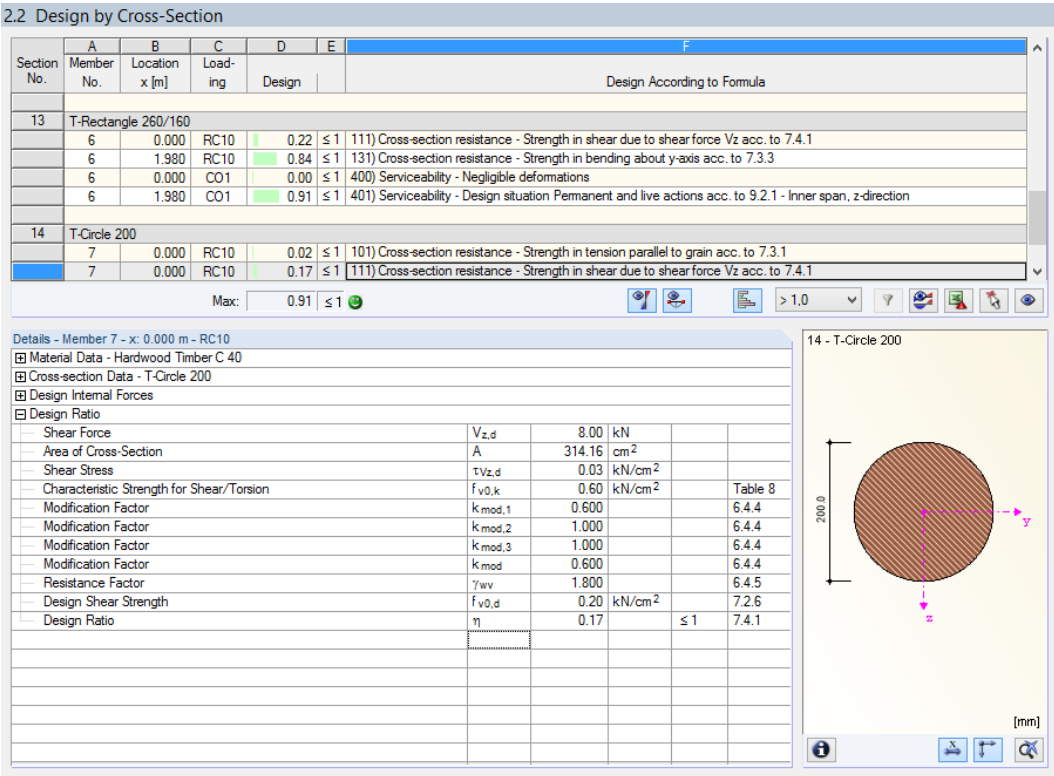


Figure 4.3: Window *2.2 Design by Cross-Section*

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

4.3 Design by Set of Members

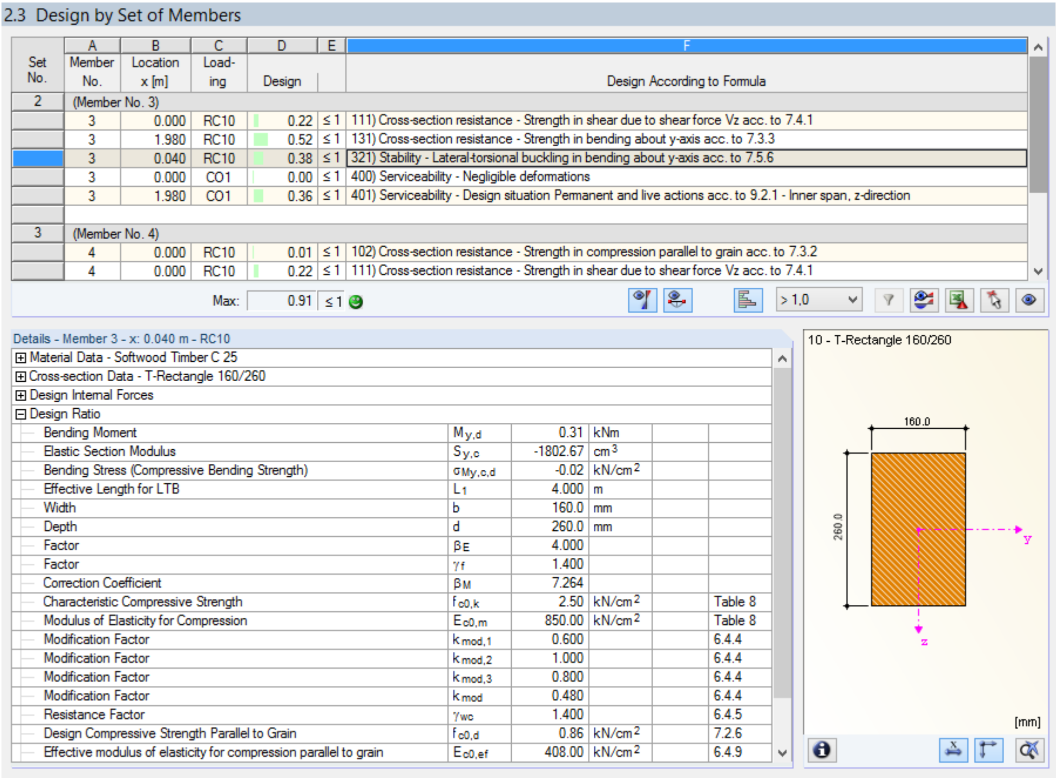


Figure 4.4: Window *2.3 Design by Set of Members*

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

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

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

4.4 Design by Member

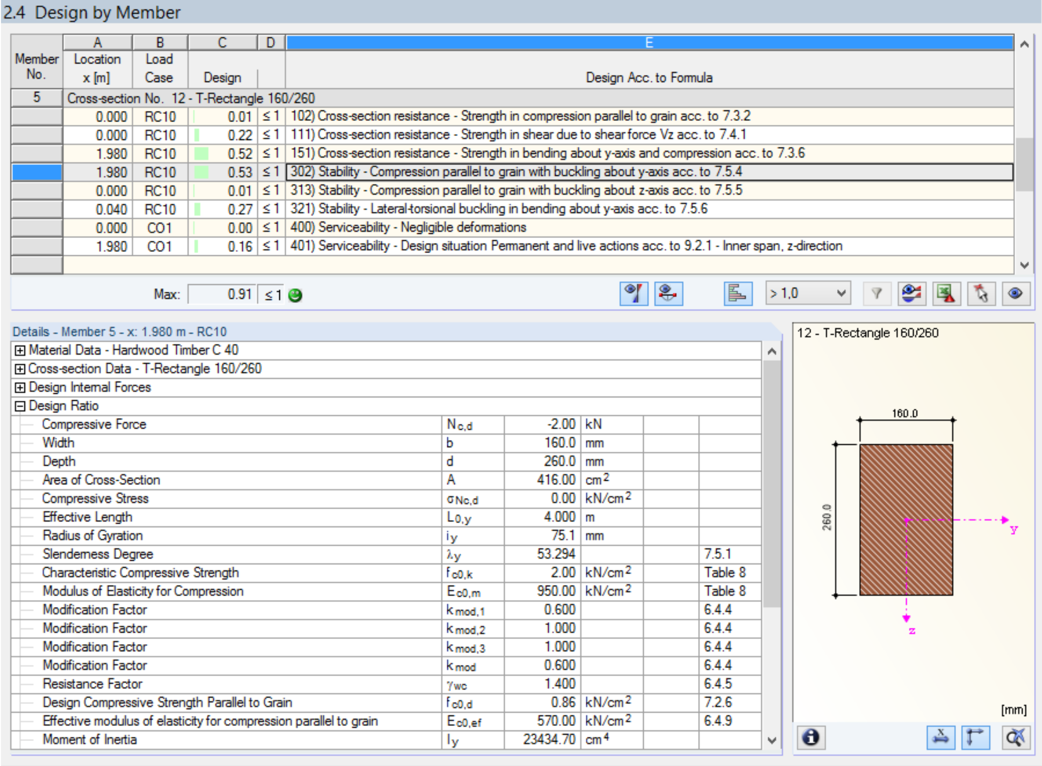


Figure 4.5: Window *2.4 Design by Member*

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

4.5 Design by x-Location

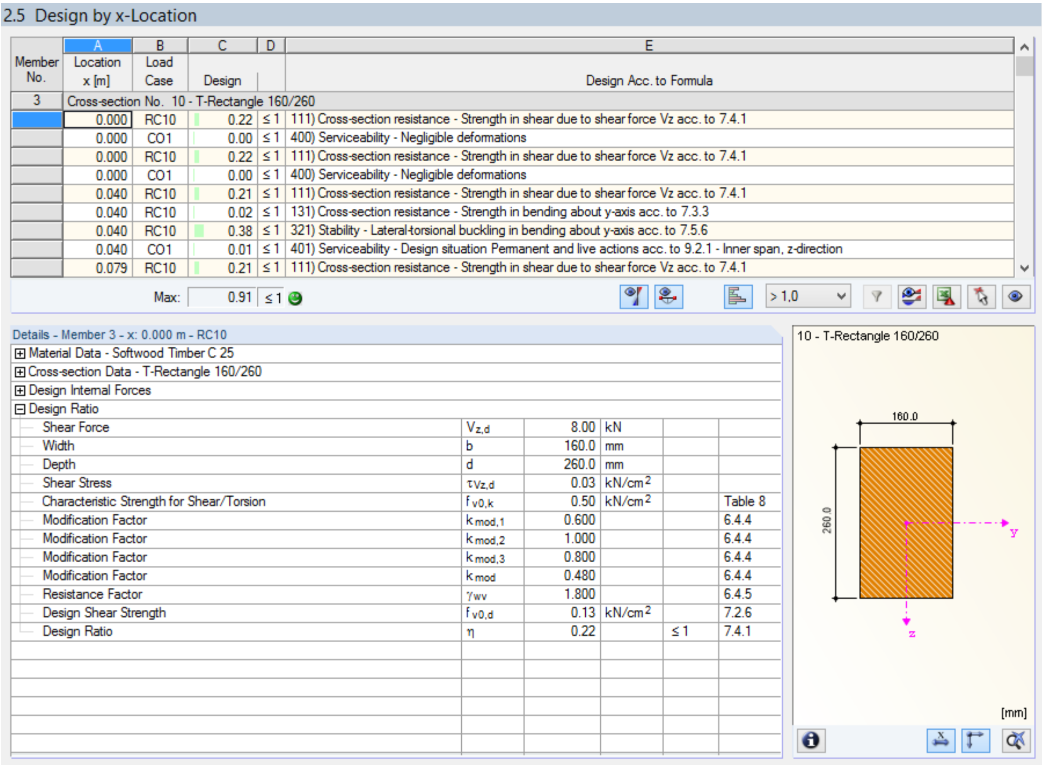


Figure 4.6: Window *2.5 Design by x-Location*

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

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

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

	A	B	C	D	E	F	G	H	I
Member No.	Location x [m]	Load Case	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
3	Cross-section No. 10 - T-Rectangle 160/260								
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz ac
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	131) Cross-section resistance - Strength in bending about y-axis acc. to 7
	0.040	RC10	-2.00	0.00	7.84	0.00	0.31	0.00	321) Stability - Lateral-torsional buckling in bending about y-axis acc. to 7.
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to S
4	Cross-section No. 11 - T-Rectangle 260/160								
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	102) Cross-section resistance - Strength in compression parallel to grain a
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz ac
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	151) Cross-section resistance - Strength in bending about y-axis and comp
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	303) Stability - Compression parallel to grain with buckling about y-axis acc
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	312) Stability - Compression parallel to grain with buckling about z-axis acc
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to S
5	Cross-section No. 12 - T-Rectangle 160/260								
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	102) Cross-section resistance - Strength in compression parallel to grain a
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz ac
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	151) Cross-section resistance - Strength in bending about y-axis and comp
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	302) Stability - Compression parallel to grain with buckling about y-axis acc
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	313) Stability - Compression parallel to grain with buckling about z-axis acc
	0.040	RC10	-2.00	0.00	7.84	0.00	0.31	0.00	321) Stability - Lateral-torsional buckling in bending about y-axis acc. to 7.
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to S
6	Cross-section No. 13 - T-Rectangle 260/160								
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz ac
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	131) Cross-section resistance - Strength in bending about y-axis acc. to 7
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Permanent and live actions acc. to S
7	Cross-section No. 14 - T-Circle 200								
	0.000	RC10	12.00	0.00	8.00	0.00	0.00	0.00	101) Cross-section resistance - Strength in tension parallel to grain acc. to

Figure 4.7: Window *3.1 Governing Internal Forces by Member*

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

Location x

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

Load Case

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

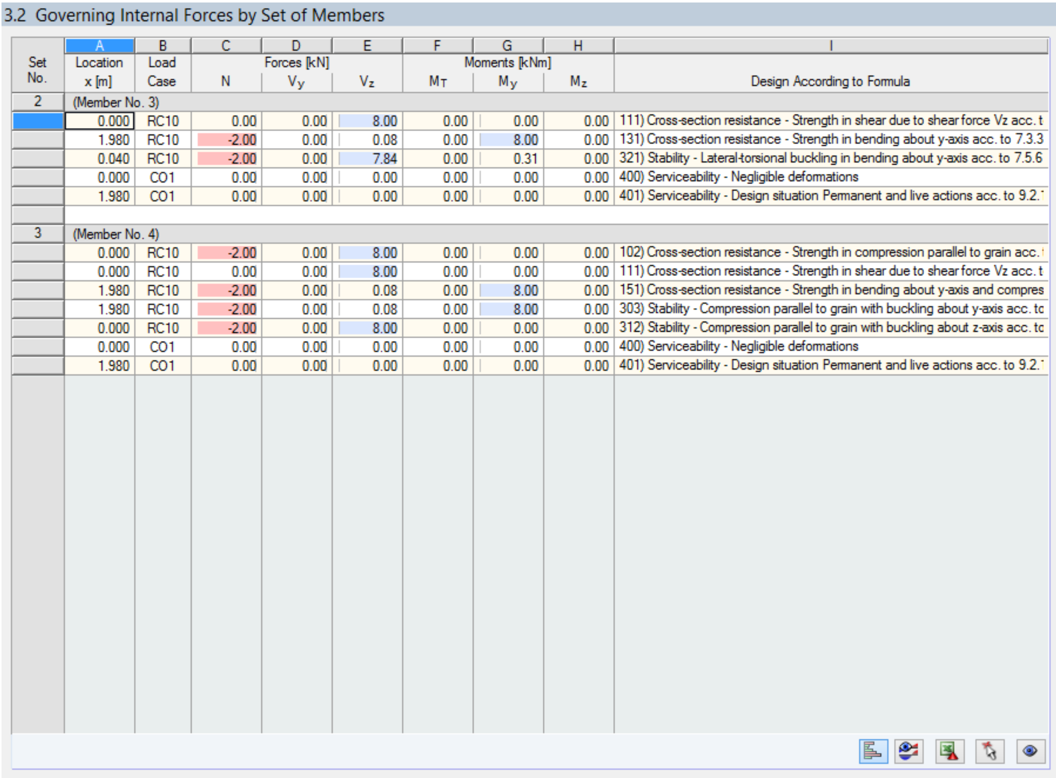
Forces / Moments

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

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members



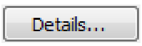
3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [m]	B Load Case	C Forces [kN] N	D Vy	E Vz	F Moments [kNm] MT	G My	H Mz	I Design According to Formula
2	(Member No. 3)								
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz acc. t
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	131) Cross-section resistance - Strength in bending about y-axis acc. to 7.3.3
	0.040	RC10	-2.00	0.00	7.84	0.00	0.31	0.00	321) Stability - Lateral-torsional buckling in bending about y-axis acc. to 7.5.6
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible defomations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Pemanent and live actions acc. to 9.2.
3	(Member No. 4)								
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	102) Cross-section resistance - Strength in compression parallel to grain acc.
	0.000	RC10	0.00	0.00	8.00	0.00	0.00	0.00	111) Cross-section resistance - Strength in shear due to shear force Vz acc. t
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	151) Cross-section resistance - Strength in bending about y-axis and compres
	1.980	RC10	-2.00	0.00	0.08	0.00	8.00	0.00	303) Stability - Compression parallel to grain with buckling about y-axis acc. tc
	0.000	RC10	-2.00	0.00	8.00	0.00	0.00	0.00	312) Stability - Compression parallel to grain with buckling about z-axis acc. tc
	0.000	CO1	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible defomations
	1.980	CO1	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design situation Pemanent and live actions acc. to 9.2.

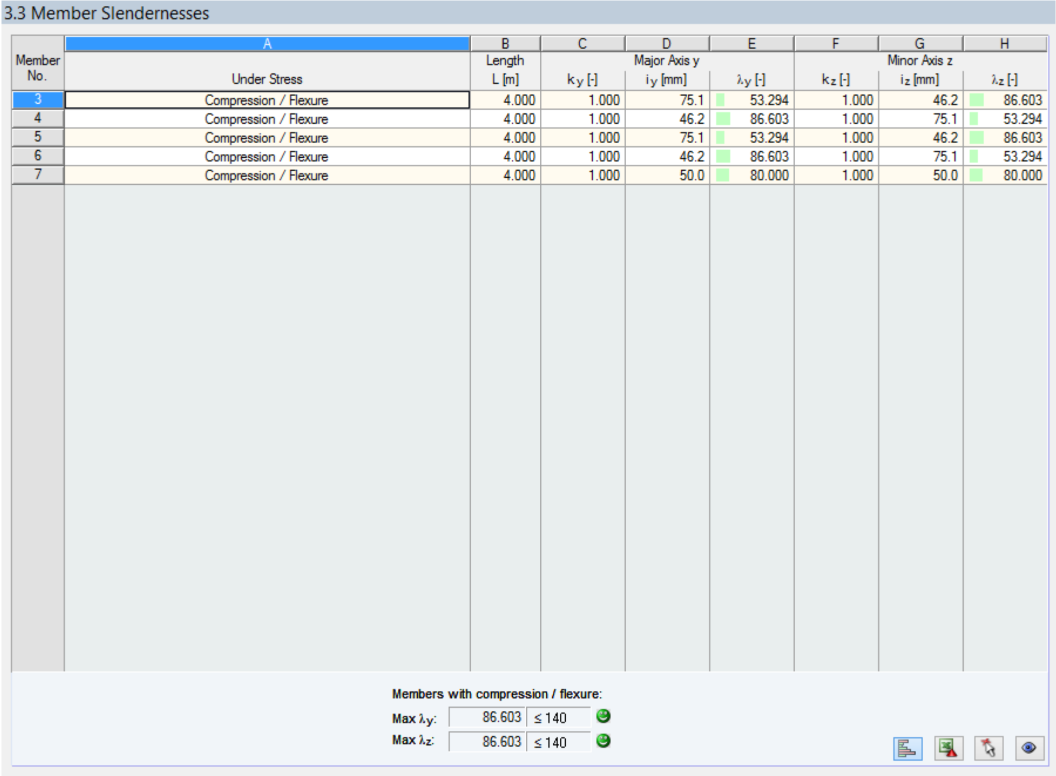
Figure 4.8: Window *3.2 Governing Internal Forces by Set of Members*

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

4.8 Member Slendernesses



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



3.3 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 [-]
3	Compression / Flexure	4.000	1.000	75.1	53.294	1.000	46.2	86.603
4	Compression / Flexure	4.000	1.000	46.2	86.603	1.000	75.1	53.294
5	Compression / Flexure	4.000	1.000	75.1	53.294	1.000	46.2	86.603
6	Compression / Flexure	4.000	1.000	46.2	86.603	1.000	75.1	53.294
7	Compression / Flexure	4.000	1.000	50.0	80.000	1.000	50.0	80.000

Members with compression / flexure:
Max λ_y: 86.603 ≤ 140
Max λ_z: 86.603 ≤ 140

Figure 4.9: Window *3.3 Member Slendernesses*

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

Details...

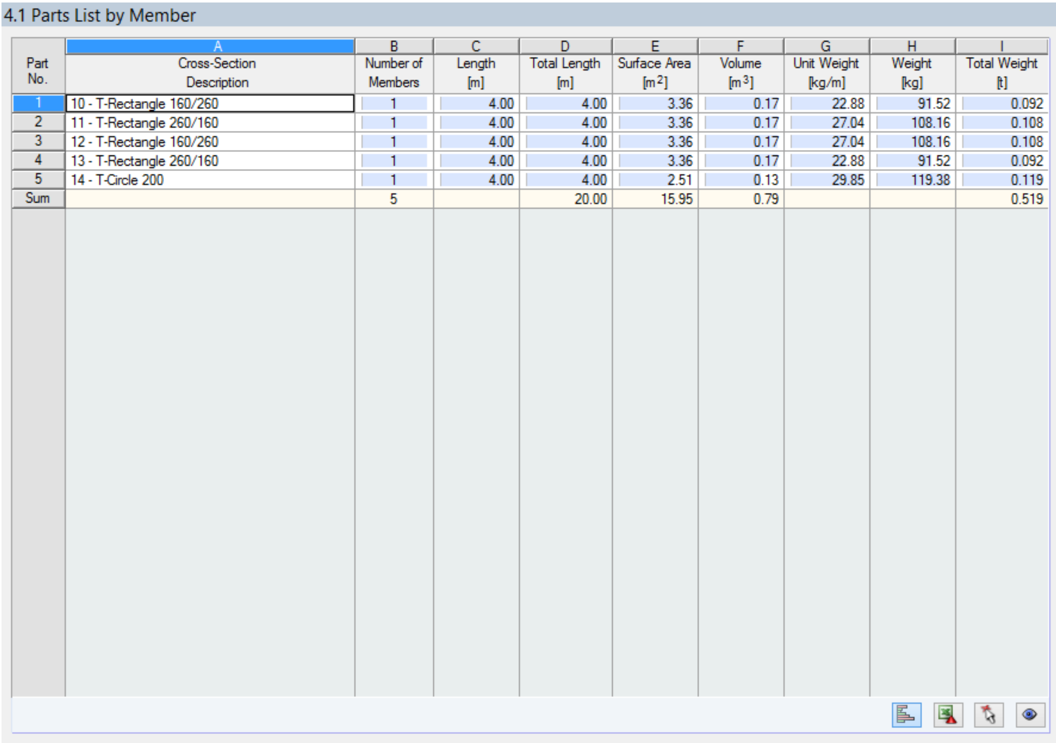
Below the list, you find a comparison of the most unfavorable values with the limit values. The latter are managed in the *Other* tab of the *Details* dialog box (see Figure 3.4, page 28).

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

This window is displayed only informative. It does not provide any stability analysis of slendernesses.

4.9 Parts List by Member

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



4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	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	10 - T-Rectangle 160/260	1	4.00	4.00	3.36	0.17	22.88	91.52	0.092
2	11 - T-Rectangle 260/160	1	4.00	4.00	3.36	0.17	27.04	108.16	0.108
3	12 - T-Rectangle 160/260	1	4.00	4.00	3.36	0.17	27.04	108.16	0.108
4	13 - T-Rectangle 260/160	1	4.00	4.00	3.36	0.17	22.88	91.52	0.092
5	14 - T-Circle 200	1	4.00	4.00	2.51	0.13	29.85	119.38	0.119
Sum		5		20.00	15.95	0.79			0.519

Figure 4.10: Window *4.1 Parts List by Member*

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

Part No.

The program automatically assigns item numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column displays the respective length of an individual member.

Total Length

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

Surface Area

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

Volume

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

Unit Weight

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

Weight

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

Total Weight

The final column gives the total mass of each part.

Sum

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

4.10 Parts List by Set of Members

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		1	4.00	4.00	3.36	0.17	22.88	91.52	0.092
2		1	4.00	4.00	3.36	0.17	27.04	108.16	0.108
Sum		2		8.00	6.72	0.33			0.200

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

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

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

5 Results Evaluation

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

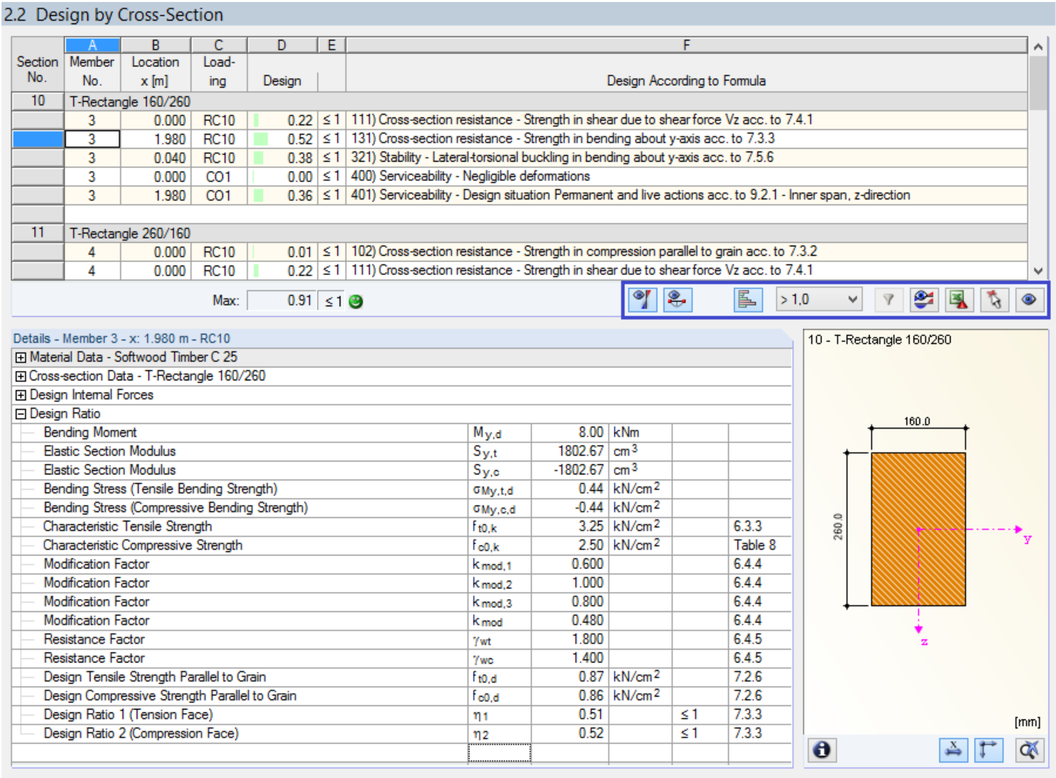


Figure 5.1: Buttons for results evaluation

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

Table 5.1: Buttons in result Windows 2.1 through 2.5

5.1 Results in RFEM/RSTAB Model

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

Background graphic and view mode

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

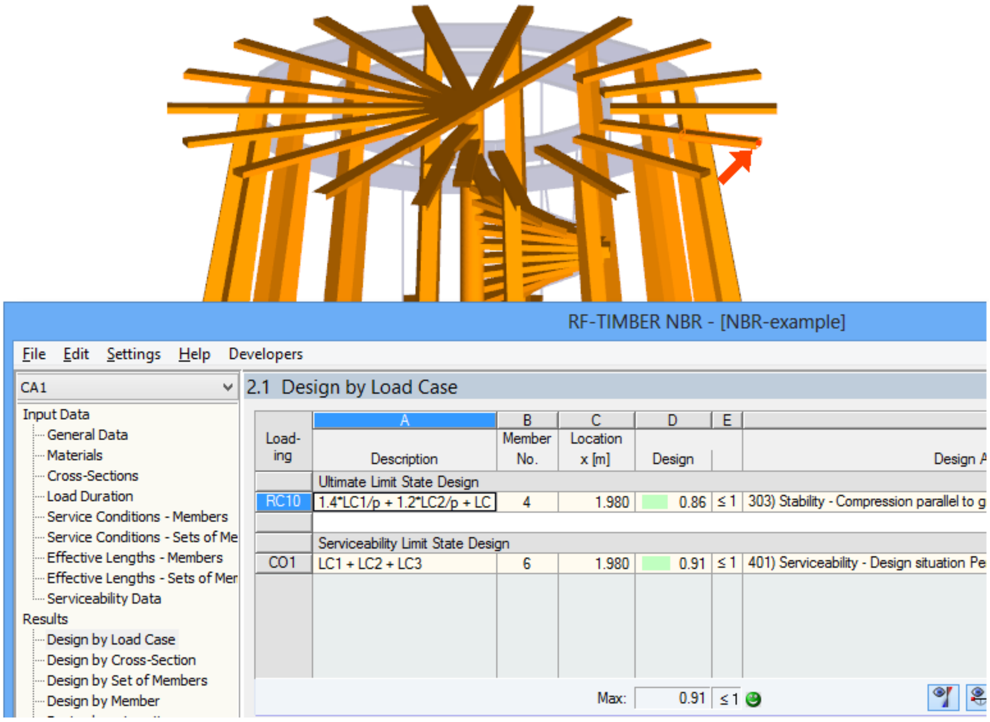


Figure 5.2: Indication of member and relevant *Location x* in RFEM model

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

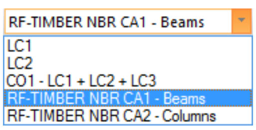
Click [Back] to return to the RF-/TIMBER NBR module.

RFEM/RSTAB work window

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

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

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



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

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

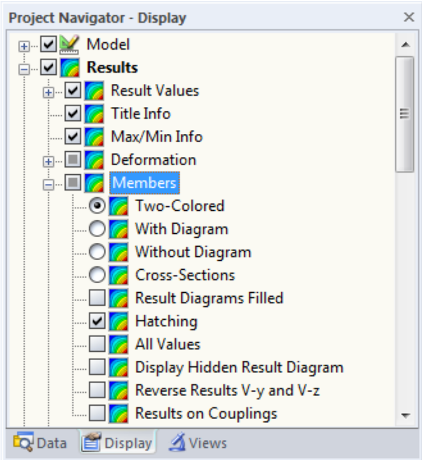


Figure 5.3: *Display* navigator – Results → Members

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

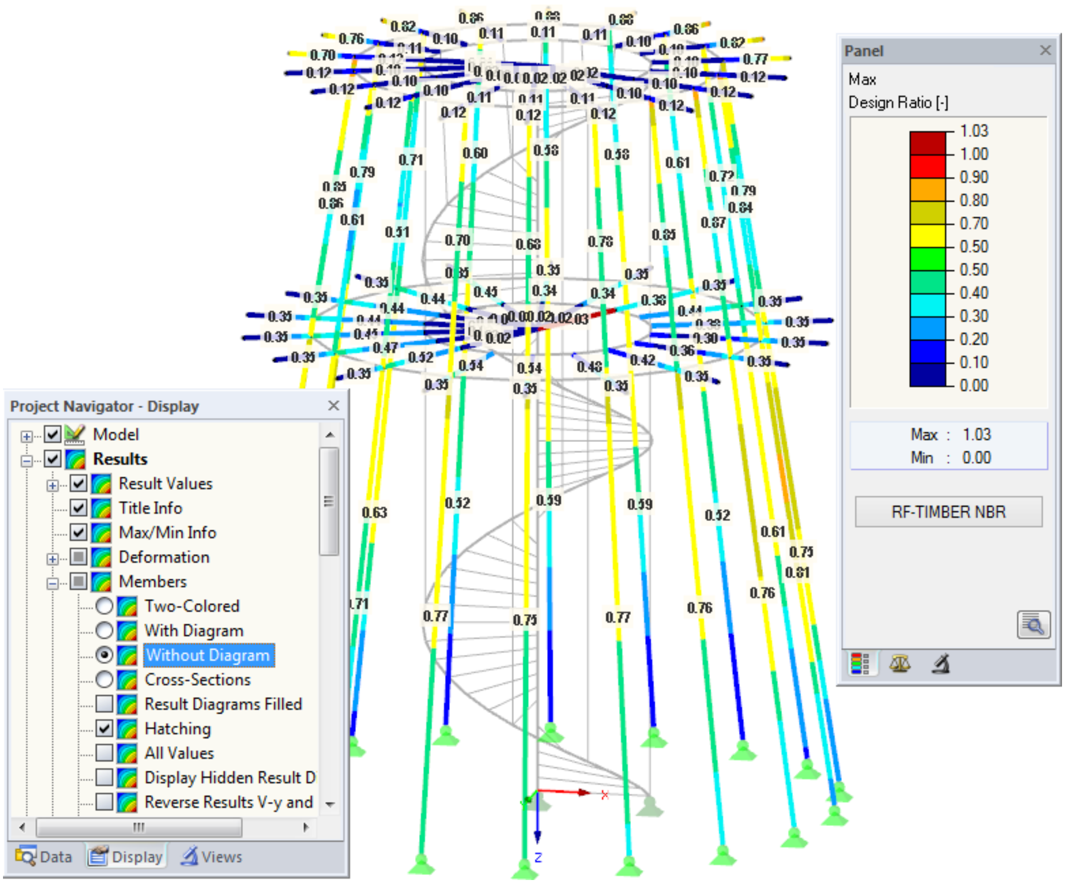
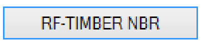


Figure 5.4: Design ratios with display option *Without Diagram*

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



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

5.2 Result Diagram

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

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

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

Results → Result Diagrams for Selected Members

or via the toolbar button shown on the left.

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

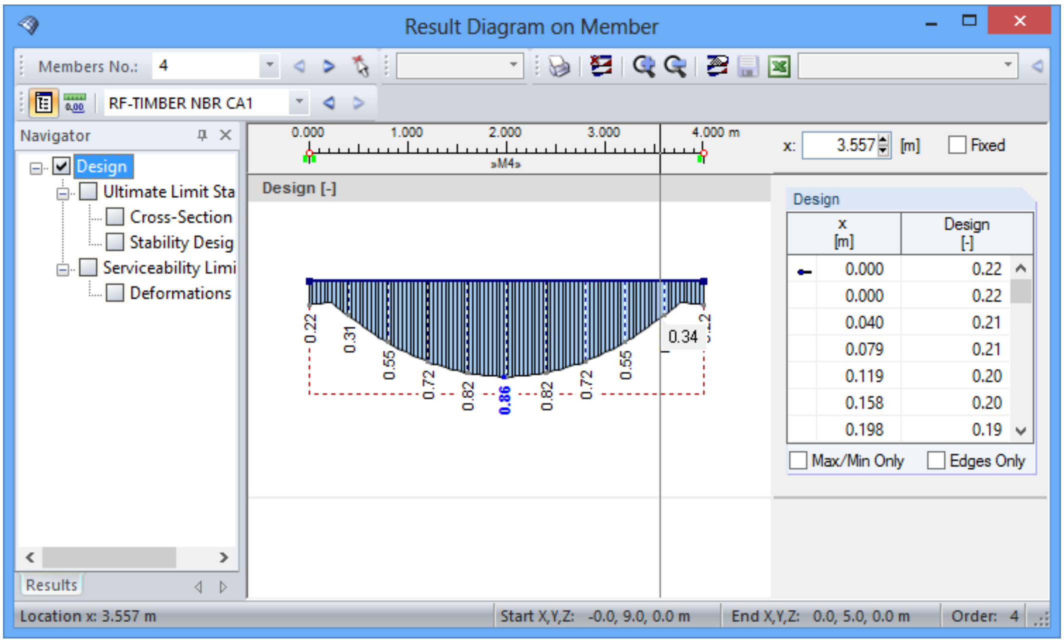
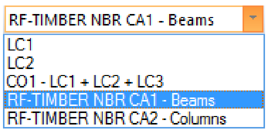


Figure 5.5: Dialog box *Result Diagram on Member*

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



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

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

5.3 Filter for Results

The RF-/TIMBER NBR result windows allow you to sort the results by various criteria. In addition, you can use the filter options for the tables (see Figure 5.1, page 40) to reduce the numerical output according to specific ratios. This function is described in the *Knowledge Base* at our Web site:
https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000733

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

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

Filtering design ratios

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

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

or use the toolbar button shown on the left.

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

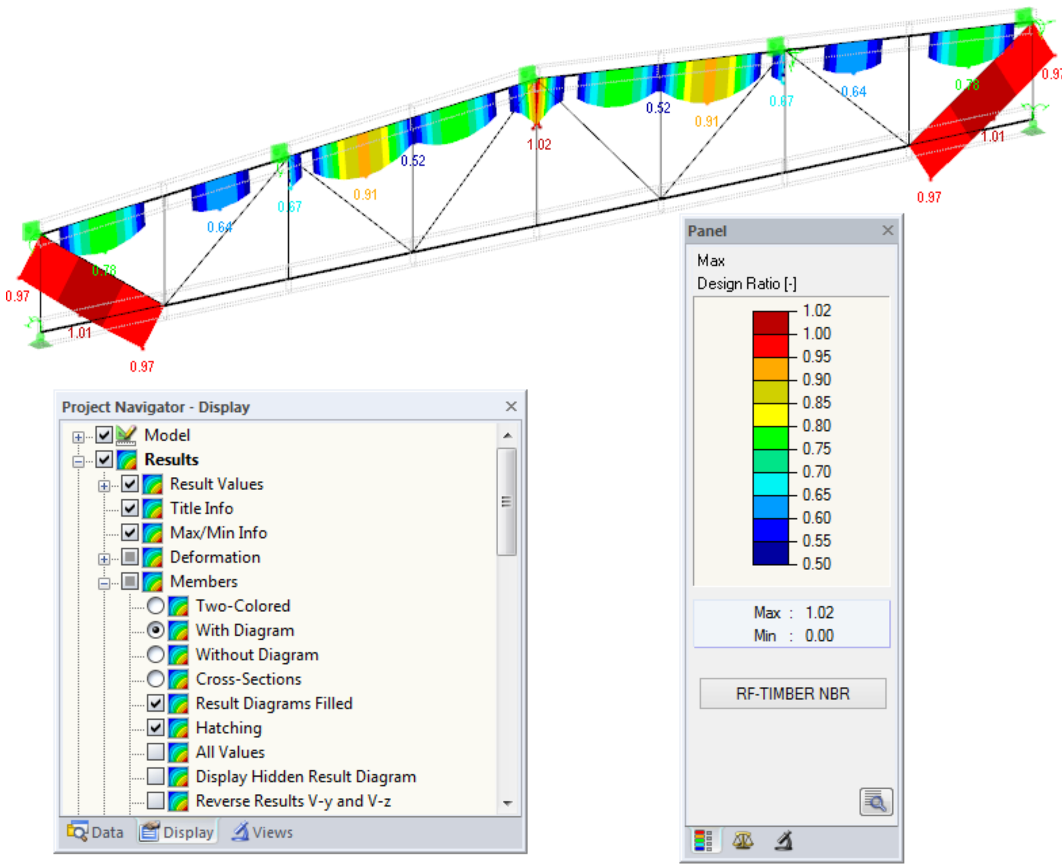


Figure 5.6: Filtering design ratios with adjusted color spectrum

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

Filtering members

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

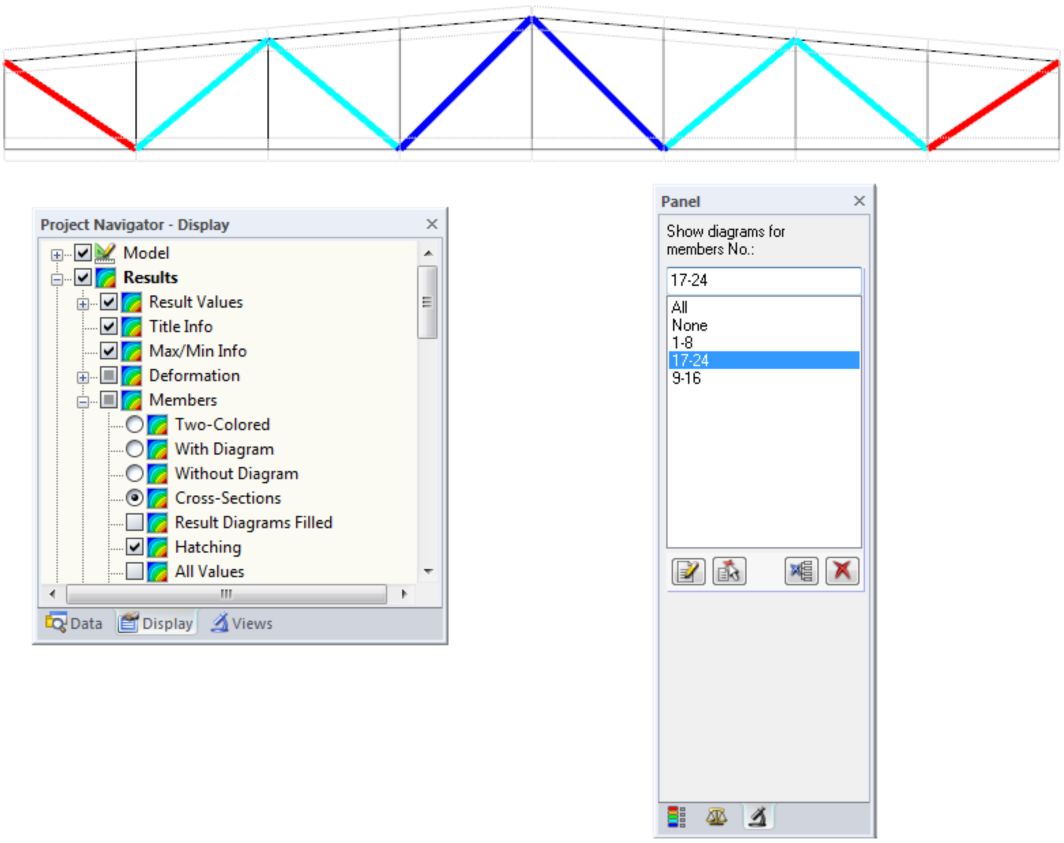


Figure 5.7: Member filter for ratios of diagonals

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

6 Printout

6.1 Printout Report

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

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

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

6.2 Graphic Printout

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

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

Design ratios in RFEM/RSTAB model

To print the currently displayed design ratios, click

File → Print Graphic

or use the toolbar button shown on the left.

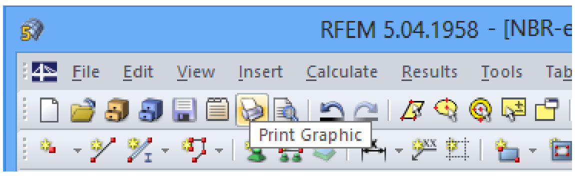


Figure 6.1: [Print Graphic] button in RFEM toolbar

Result diagrams

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

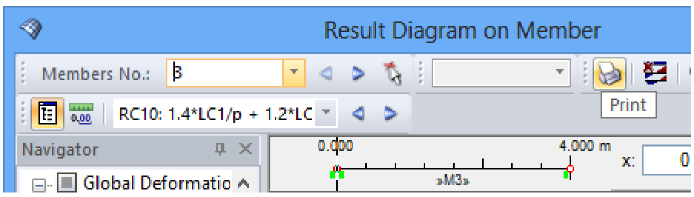


Figure 6.2: [Print] button in *Result Diagram on Member* dialog box

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

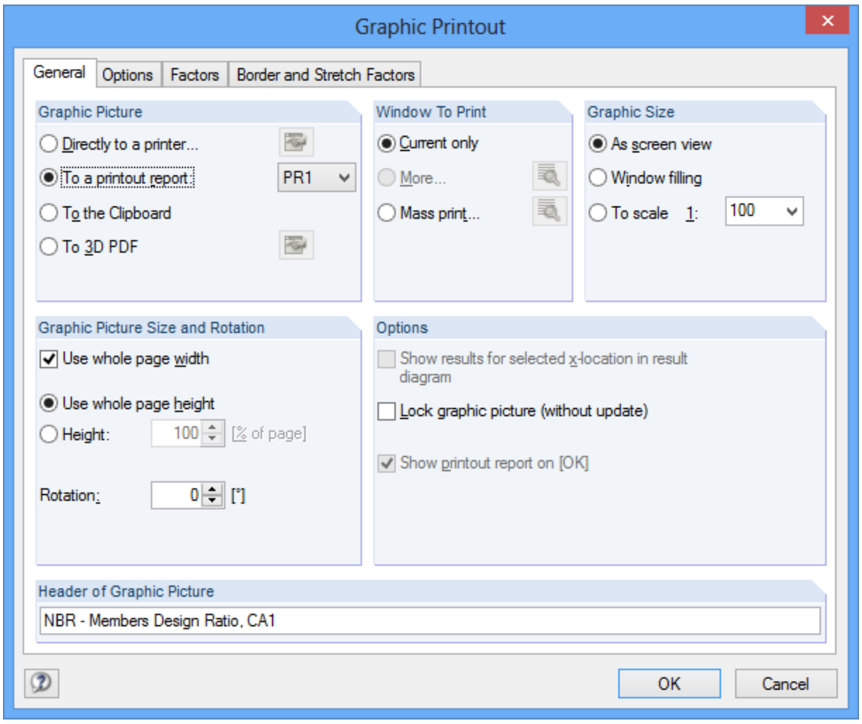
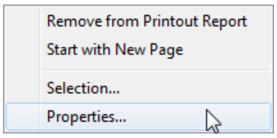


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

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



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

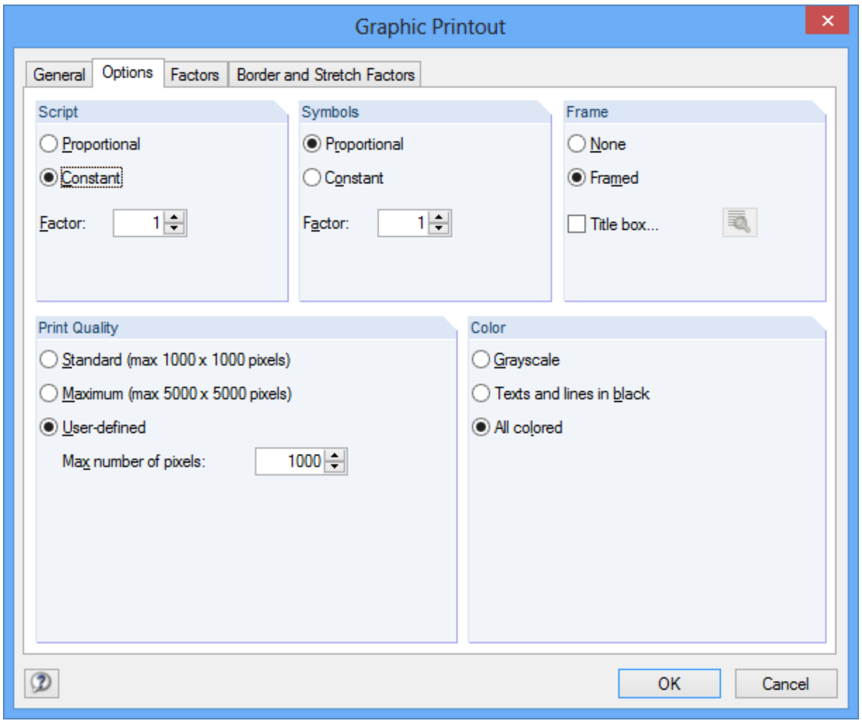


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

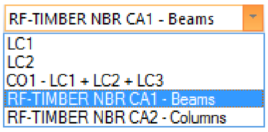
7 General Functions

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

7.1 Design Cases

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

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



To calculate a RF-/TIMBER NBR design case, you can also use the load case list in the toolbar of RFEM or RSTAB.

Create design case

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

File → **New Case**.

The following dialog box appears.

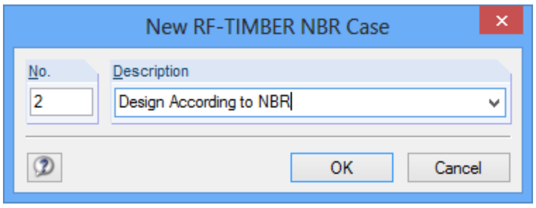


Figure 7.1: Dialog box *New RF-TIMBER NBR-Case*

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

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

Rename design case

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

File → **Rename Case**.

The following dialog box appears.

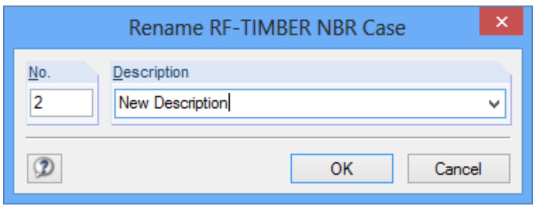


Figure 7.2: Dialog box *Rename RF-TIMBER NBR-Case*

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

Copy design case

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

File → **Copy Case**.

The following dialog box appears.

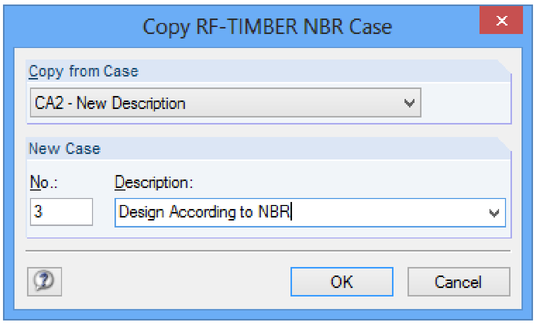


Figure 7.3: Dialog box *Copy RF-TIMBER NBR-Case*

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

Delete design case

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

File → **Delete Case**.

The following dialog box appears.

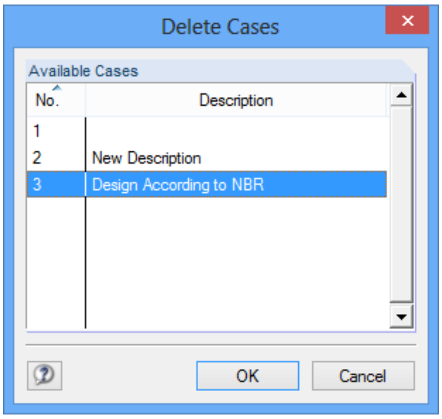
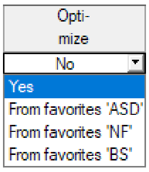


Figure 7.4: Dialog box *Delete Cases*

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

7.2 Cross-Section Optimization



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

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

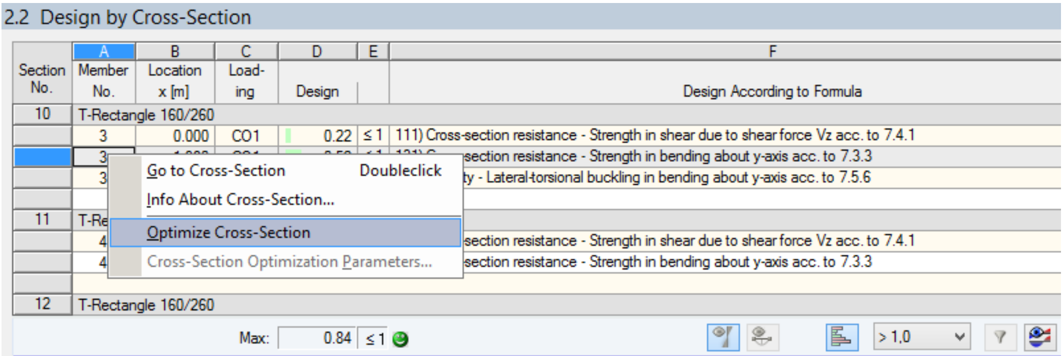
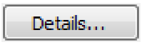


Figure 7.5: Shortcut menu for cross-section optimization



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

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

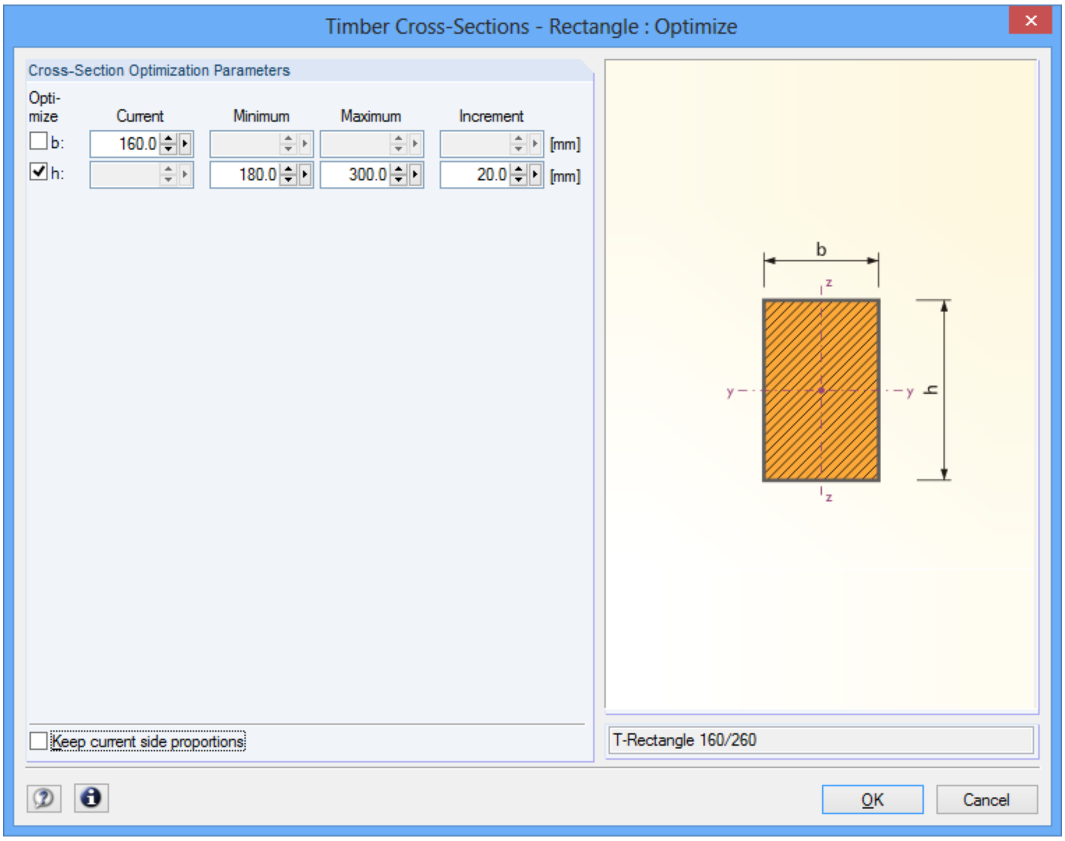


Figure 7.6: Dialog box *Timber Cross-Sections - Rectangle : Optimize*

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

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

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

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

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

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

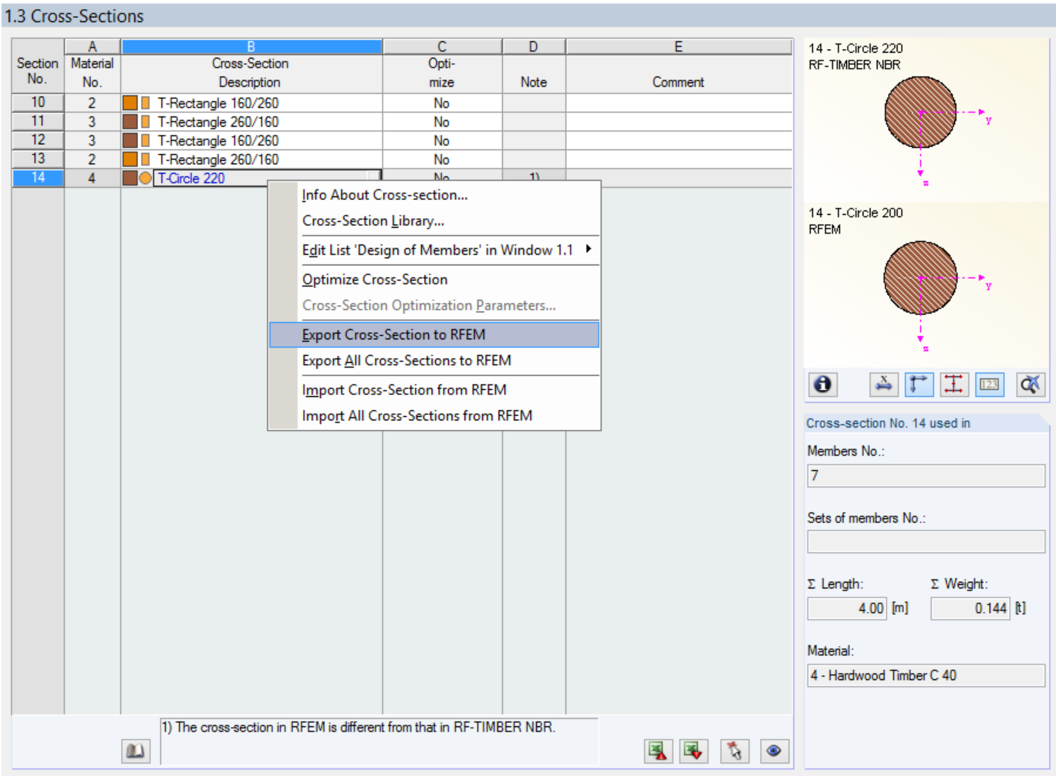


Figure 7.7: Shortcut menu in Window *1.3 Cross-Sections*

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

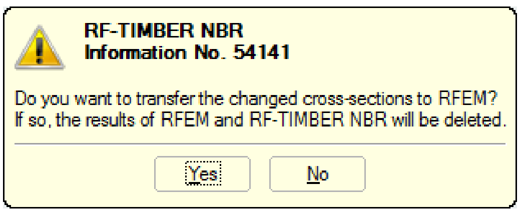
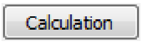


Figure 7.8: Query when exporting cross-sections



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

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

7.3 Units and Decimal Places

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

Settings → **Units and Decimal Places**.

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

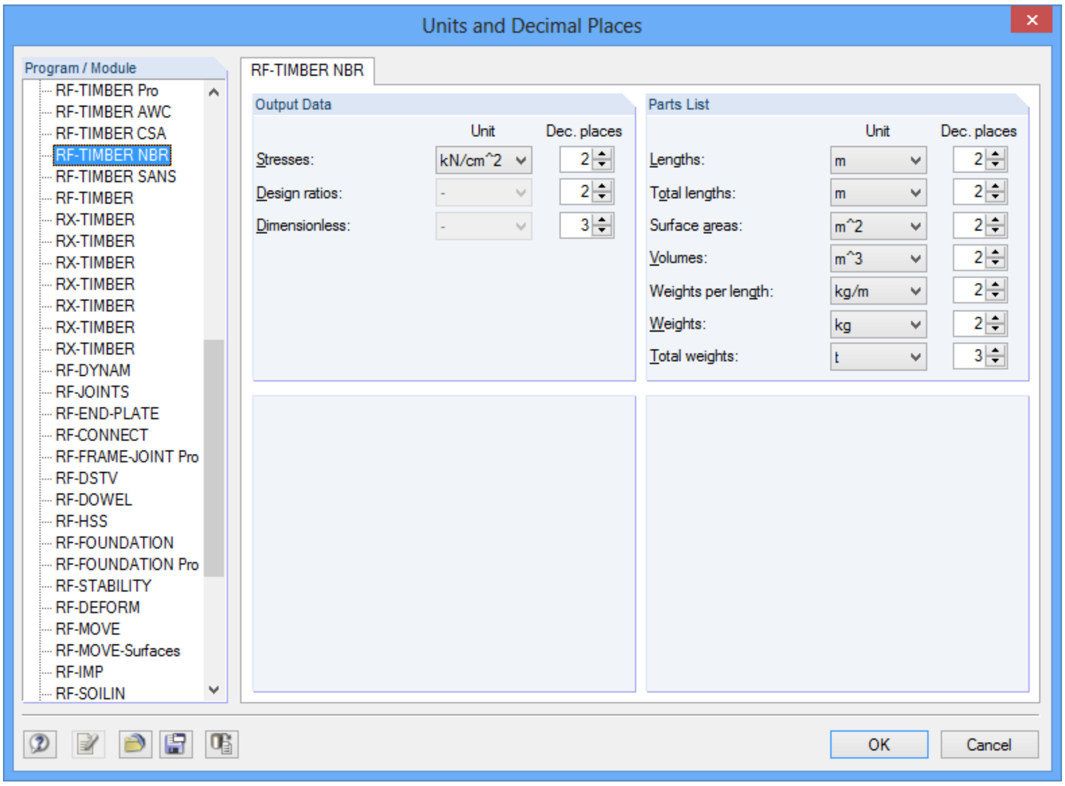


Figure 7.9: Dialog box *Units and Decimal Places*

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

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

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

Edit → Export All Materials to RFEM/RSTAB.

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

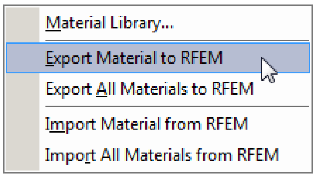
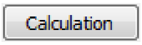


Figure 7.10: Shortcut menu of Window *1.2 Materials*



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

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

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

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

Edit → Export All Effective Lengths to RFEM/RSTAB.

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

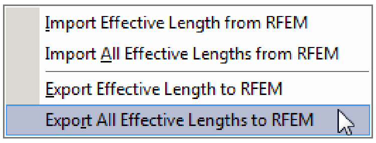


Figure 7.11: Shortcut menu of Window *1.7 Effective Lengths - Members*

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

7.4.3 Exporting Results

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

Clipboard

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

Printout Report

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

File → Export to RTF.

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

Excel

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

File → Export Tables.

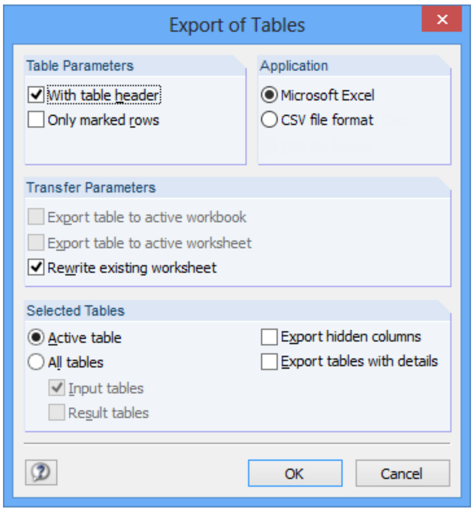


Figure 7.12: Dialog box *Export of Tables*

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

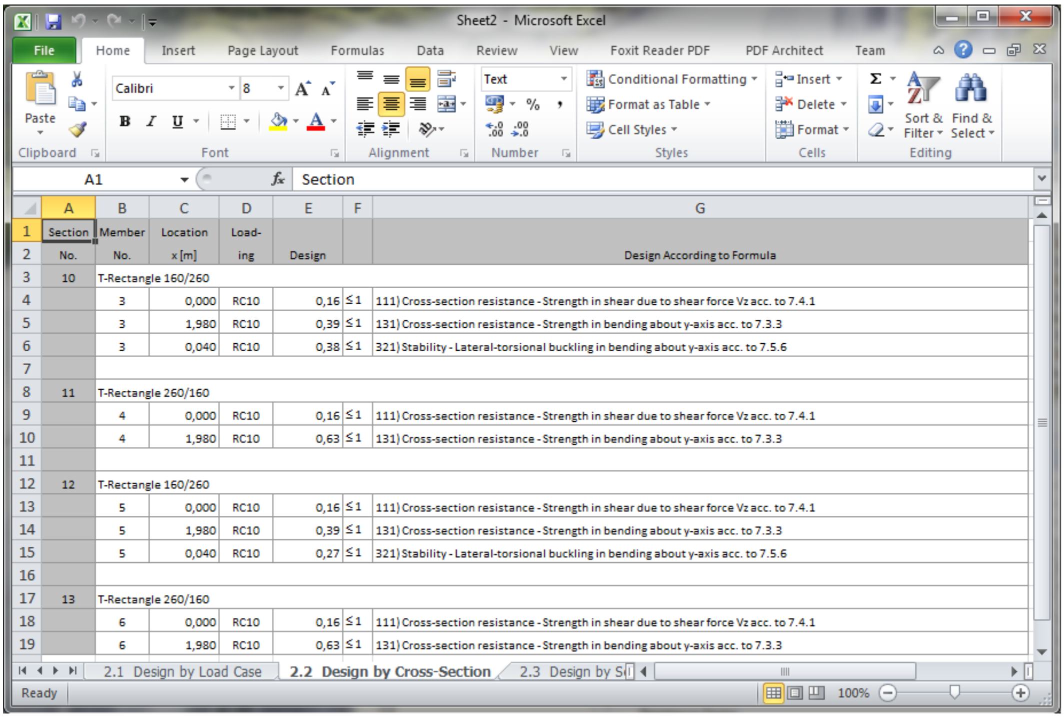


Figure 7.13: Results in Excel

8 Example: Timber Beam

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

8.1 System and Loads

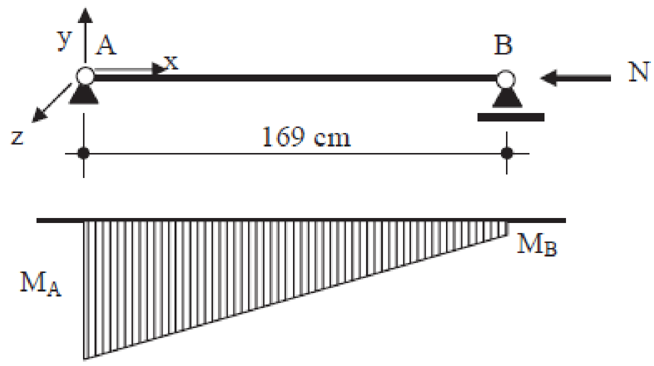


Figure 8.1: System and loads according to [2]

Model

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

Loads and Combinations

LC1:	N = 15 kN
	M_A = 33.4 kN/cm
	M_B = 2.2 kN/cm
	LDC permanent
LC2:	N = 10 kN
	M_A = 22.3 kN/cm
	M_B = 1.4 kN/cm
	LDC long-term
CO1 (ULS):	1.4 LC1 + 1.4 LC2
CO2 (SLS):	1.0 LC1 + 0.2 LC2

8.2 Calculation with RFEM/RSTAB

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

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

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

$$k_{mod} = k_{mod,1} \cdot k_{mod,2} \cdot k_{mod,3}$$

$$k_{mod} = 0.7 \cdot 1.0 \cdot 0.8 = 0.56$$

$$E_{c0,ef} = k_{mod} \cdot E_{c0,m} = 0.56 \cdot 1{,}950 = 1{,}092 \text{ MPa}$$

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

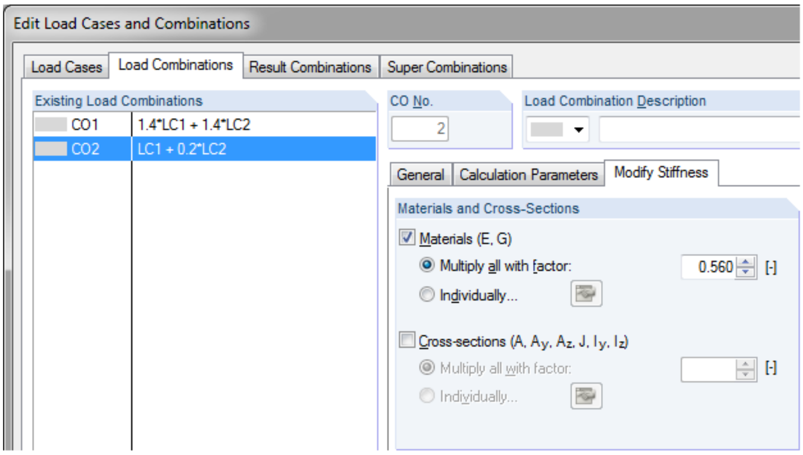


Figure 8.2: Modifying Modulus of Elasticity in the *Edit Load Cases and Combinations* dialog box

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

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

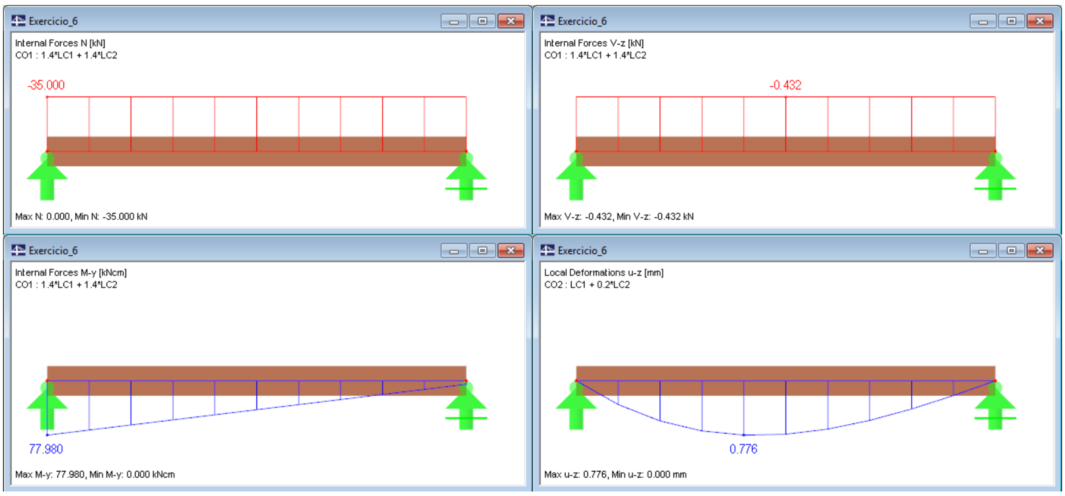


Figure 8.3: Internal forces N, M_y, V_z, and deformation u_z

8.3 Design with RF-/TIMBER NBR

8.3.1 Ultimate Limit State Design

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

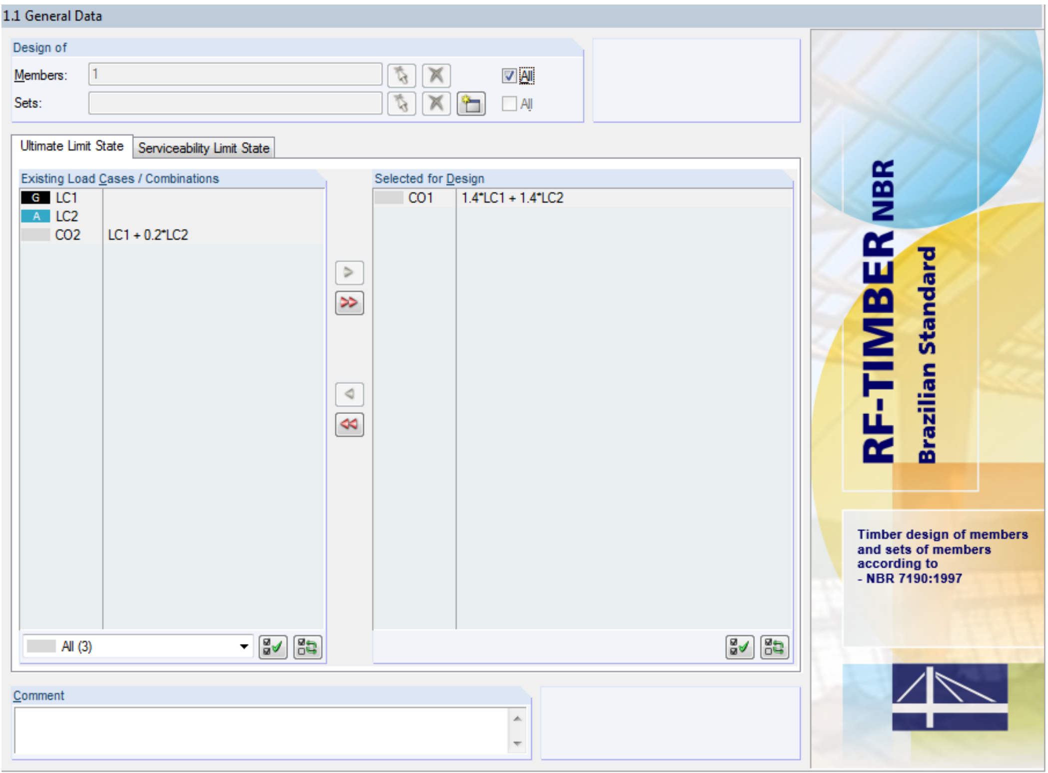


Figure 8.4: Window *1.1 General Data*

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

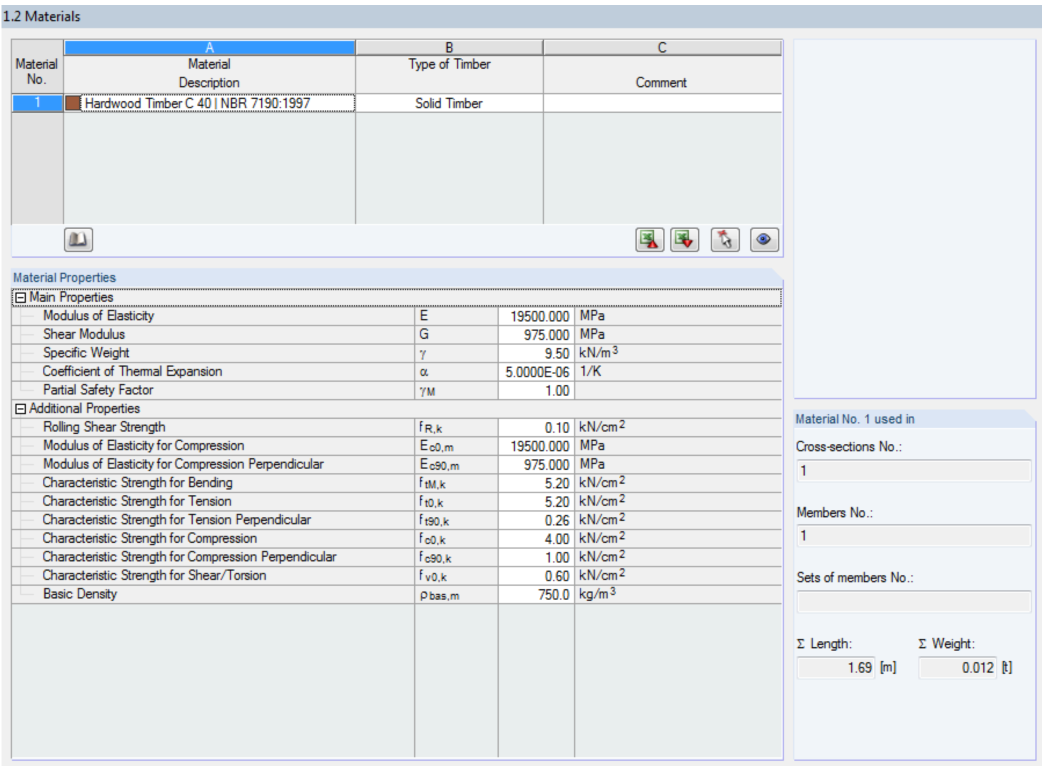


Figure 8.5: Window *1.2 Materials*

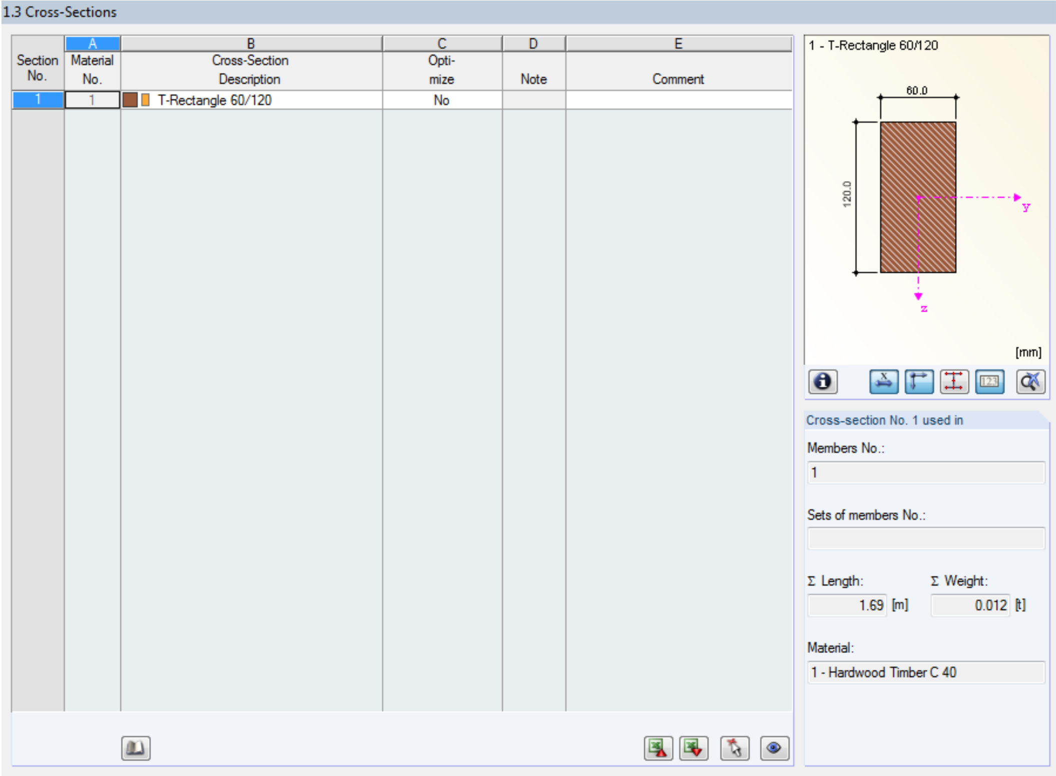


Figure 8.6: Window *1.3 Cross-Sections*

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

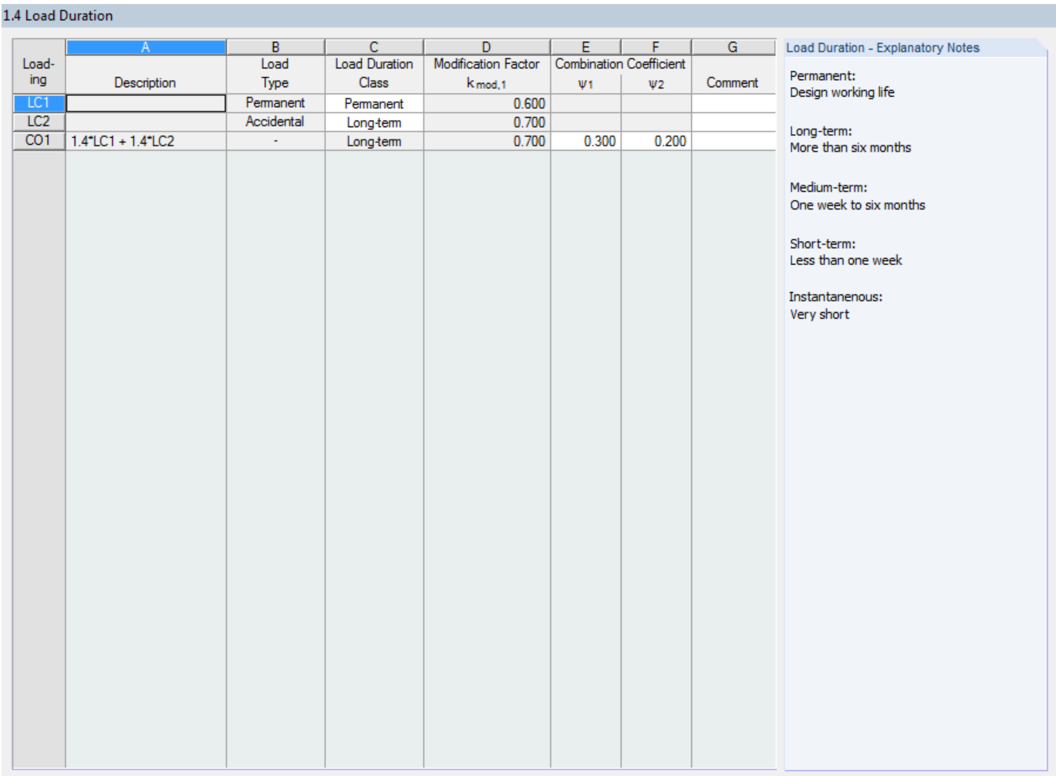


Figure 8.7: Window *1.4 Load Duration*

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

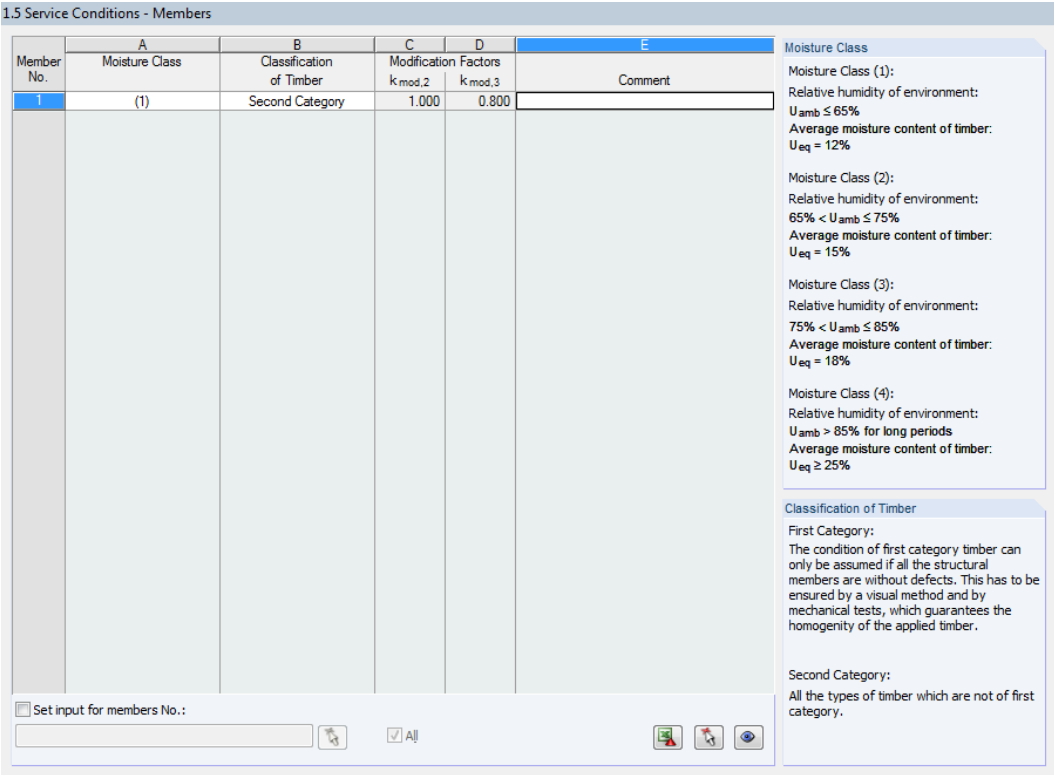


Figure 8.8: Window *1.5 Service Conditions - Members*

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

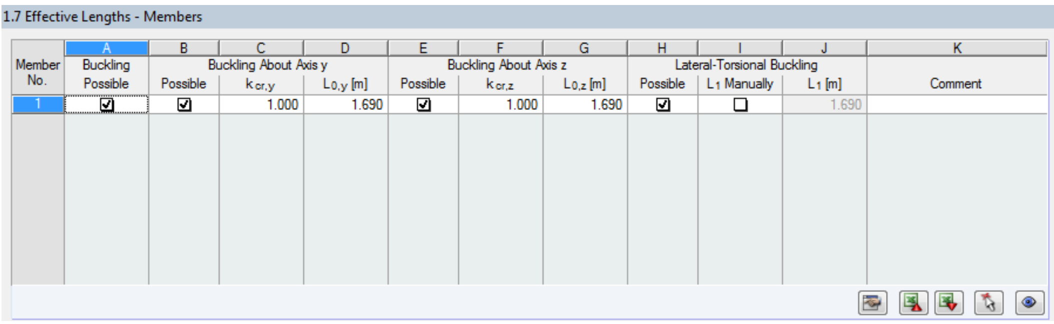


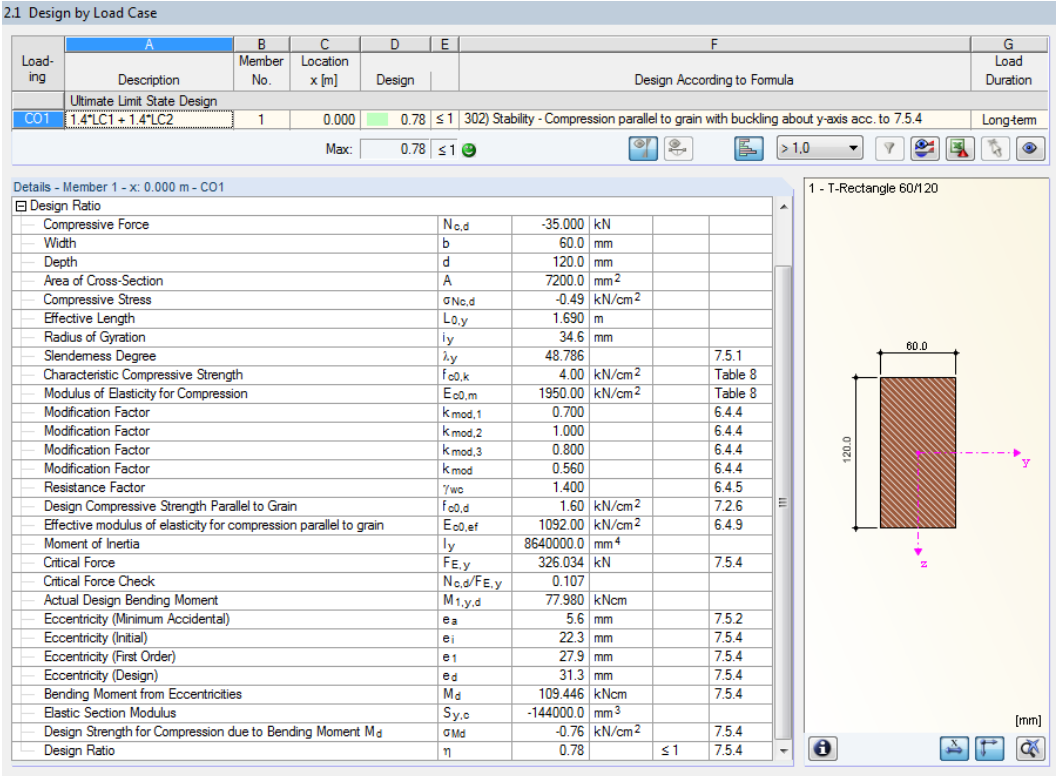
Figure 8.9: Window *1.7 Effective Lengths - Members*

Calculation

Then we can start the [Calculation].

Results

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


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 Load Duration
	Ultimate Limit State Design						
CO1	1.4*LC1 + 1.4*LC2	1	0.000	0.78	≤ 1	302) Stability - Compression parallel to grain with buckling about y-axis acc. to 7.5.4	Long-term
			Max:	0.78	≤ 1		

Details - Member 1 - x: 0.000 m - CO1

Design Ratio					
Compressive Force	$N_{c,d}$	-35.000	kN		
Width	b	60.0	mm		
Depth	d	120.0	mm		
Area of Cross-Section	A	7200.0	mm^2		
Compressive Stress	$\sigma_{Nc,d}$	-0.49	kN/cm^2		
Effective Length	$L_{0,y}$	1.690	m		
Radius of Gyration	i_y	34.6	mm		
Slenderness Degree	λ_y	48.786			7.5.1
Characteristic Compressive Strength	$f_{c0,k}$	4.00	kN/cm^2		Table 8
Modulus of Elasticity for Compression	$E_{c0,m}$	1950.00	kN/cm^2		Table 8
Modification Factor	$k_{mod,1}$	0.700			6.4.4
Modification Factor	$k_{mod,2}$	1.000			6.4.4
Modification Factor	$k_{mod,3}$	0.800			6.4.4
Modification Factor	k_{mod}	0.560			6.4.4
Resistance Factor	γ_{wc}	1.400			6.4.5
Design Compressive Strength Parallel to Grain	$f_{c0,d}$	1.60	kN/cm^2		7.2.6
Effective modulus of elasticity for compression parallel to grain	$E_{c0,ef}$	1092.00	kN/cm^2		6.4.9
Moment of Inertia	I_y	8640000.0	mm^4		
Critical Force	$F_{E,y}$	326.034	kN		7.5.4
Critical Force Check	$N_{c,d}/F_{E,y}$	0.107			
Actual Design Bending Moment	$M_{1,y,d}$	77.980	kNcm		
Eccentricity (Minimum Accidental)	e_a	5.6	mm		7.5.2
Eccentricity (Initial)	e_i	22.3	mm		7.5.4
Eccentricity (First Order)	e_1	27.9	mm		7.5.4
Eccentricity (Design)	e_d	31.3	mm		7.5.4
Bending Moment from Eccentricities	M_d	109.446	kNcm		7.5.4
Elastic Section Modulus	$S_{y,c}$	-144000.0	mm^3		
Design Strength for Compression due to Bending Moment M_d	σ_{Md}	-0.76	kN/cm^2		7.5.4
Design Ratio	η	0.78		≤ 1	7.5.4

Figure 8.10: Window *2.1 Design by Load Case*

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

Design Strengths

Design compressive strength

$$f_{c0,d} = k_{mod,1} \cdot k_{mod,2} \cdot k_{mod,3} \cdot \frac{f_{c0,k}}{\gamma_{wc}}$$

$$f_{c0,d} = 0.70 \cdot 1.00 \cdot 0.80 \cdot \frac{4.00}{1.40}$$

$$f_{c0,d} = 1.60 \text{ kN/m}^2$$

Design tensile strength

$$f_{t0,d} = k_{mod,1} \cdot k_{mod,2} \cdot k_{mod,3} \cdot \frac{f_{t0,k}}{\gamma_{wt}}$$

$$f_{t0,d} = 0.70 \cdot 1.00 \cdot 0.80 \cdot \frac{5.20}{1.80}$$

$$f_{t0,d} = 1.62 \text{ kN/m}^2$$

Design shear strength

$$f_{v0,d} = k_{mod,1} \cdot k_{mod,2} \cdot k_{mod,3} \cdot \frac{f_{v0,k}}{\gamma_{wv}}$$

$$f_{v0,d} = 0.70 \cdot 1.00 \cdot 0.80 \cdot \frac{0.60}{1.80}$$

$$f_{v0,d} = 0.19 \text{ kN/m}^2$$

Effective modulus of elasticity

$$E_{c0,ef} = k_{mod,1} \cdot k_{mod,2} \cdot k_{mod,3} \cdot E_{c0,m}$$

$$E_{c0,ef} = 0.70 \cdot 1.00 \cdot 0.80 \cdot 1{,}950$$

$$E_{c0,ef} = 1{,}092 \text{ kN/m}^2$$

Stress Analysis

Compressive stress

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

Bending stress

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

Shear stress

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

Buckling Design

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

Axis y

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

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

Critical force

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

Eccentricity (minimum accidental)

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

Eccentricity (initial)

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

Eccentricity (first order)

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

Eccentricity (design)

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

Bending moment from eccentricities

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

Design stress for compression due to bending moment from eccentricities

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

Axis z

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

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

Critical Force

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

Eccentricity (minimum accidental)

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

Eccentricity, permanent loads (first order)

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

Eccentricity (initial)

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

Eccentricity (creep)

$$e_c = (e_a + e_i) \cdot \left\{\exp\left[\frac{\phi \cdot [N_{g,c,k} + (\psi_1 + \psi_2) \cdot N_{q,c,k}]}{F_{E,z} - [N_{g,c,k} + (\psi_1 + \psi_2) \cdot N_{q,c,k}]}\right] - 1\right\}$$

$$e_c = (0 + 0.56\overline{3}) \cdot \left\{\exp\left[\frac{0.8 \cdot [15.00 + (0.3 + 0.2) \cdot 10.00]}{81.51 - [15.00 + (0.3 + 0.2) \cdot 10.00]}\right] - 1\right\} = 0.1674\ \text{cm}$$

Eccentricity (first order, effective)

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

Bending moment from eccentricities

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

Design stress for compression due to bending moment from eccentricities

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

Combined Bending and Axial Compression Design

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

Design 1 – tension zone

$$-\left(\frac{\sigma_{N_{c,d}}}{f_{c0,d}}\right)^2 + \frac{\sigma_{M_{y,t,d}}}{f_{t0,d}} + k_M \cdot \frac{\sigma_{M_{z,t,d}}}{f_{t0,d}} \leq 1.00$$

$$-\left(\frac{0.49}{1.60}\right)^2 + \frac{0.54}{1.62} + 0.5 \cdot \frac{0}{1.62} = 0.24 \leq 1.00$$

Design 2 – compression zone

$$\left(\frac{\sigma_{N_{c,d}}}{f_{c0,d}}\right)^2 + \frac{\sigma_{M_{y,c,d}}}{f_{t0,d}} + k_M \cdot \frac{\sigma_{M_{z,c,d}}}{f_{t0,d}} \leq 1.00$$

$$\left(\frac{0.49}{1.60}\right)^2 + \frac{0.54}{1.60} + 0.5 \cdot \frac{0}{1.60} = 0.43 \leq 1.00$$

Shear Design

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

Design

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

Compression with Buckling Design

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

Design 1 – buckling about y-axis according to 7.5.4

$$\frac{\sigma_{N_{c,d}}}{f_{t0,d}} + \frac{\sigma_{M_d}}{f_{t0,d}} \leq 1.00$$

$$\frac{0.49}{1.60} + \frac{0.76}{1.60} = 0.78 \leq 1.00$$

Design 2 – buckling about z-axis according to 7.5.5

$$\frac{\sigma_{N_{c,d}}}{f_{t0,d}} + \frac{\sigma_{M_d}}{f_{t0,d}} \leq 1.00$$

$$\frac{0.49}{1.60} + \frac{0.62}{1.60} = 0.69 \leq 1.00$$

Lateral-Torsional Buckling Design

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

At first, the correction factor, β_M, must be determined.

$$\beta_M = \frac{1}{0.26\,\pi} \cdot \frac{\beta_E}{\gamma_f} \cdot \frac{\left(\frac{h}{b}\right)^{\frac{3}{2}}}{\left(\frac{h}{b} - 0.63\right)^{\frac{1}{2}}} = \frac{1}{0.26\,\pi} \cdot \frac{4}{1.4} \cdot \frac{\left(\frac{12\text{ cm}}{6\text{ cm}}\right)^{\frac{3}{2}}}{\left(\frac{12\text{ cm}}{6\text{ cm}} - 0.63\right)^{\frac{1}{2}}} = 8.453$$

Design

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

8.3.2 Serviceability Limit State Design

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

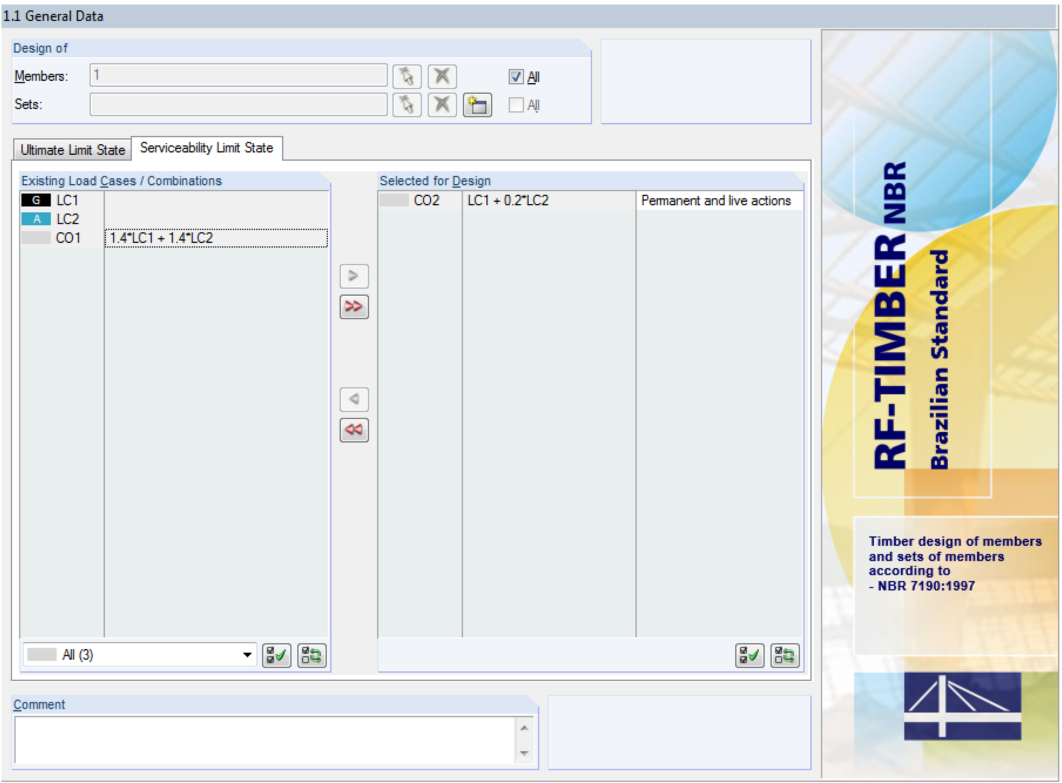


Figure 8.11: Window *1.1 General Data*

Then we enter member **1** in Window *1.9 Serviceability Data*.

1.9 Serviceability Data

	A	B	C	D	E	F	G	H	I
No.	Reference to	Member No.	Reference Length Manually	L [m]	Direc-tion	Precamber $w_{c,y}$ [mm]	$w_{c,z}$ [mm]	Beam Type	Comment
1	Member	1	☐	1.690	z		0.000	Beam	
2									
3									
4									
5									
6									
7									
8									
9									
10									

Figure 8.12: Window *1.9 Serviceability Data*

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

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

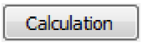
Figure 8.13: Dialog box *Details*, tab *Serviceability*

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

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

Design

$$\frac{u_z}{\frac{L}{200}} = \frac{0.776\text{ mm}}{\frac{169\text{ cm}}{200}} = 0.09 \leq 1.00$$



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

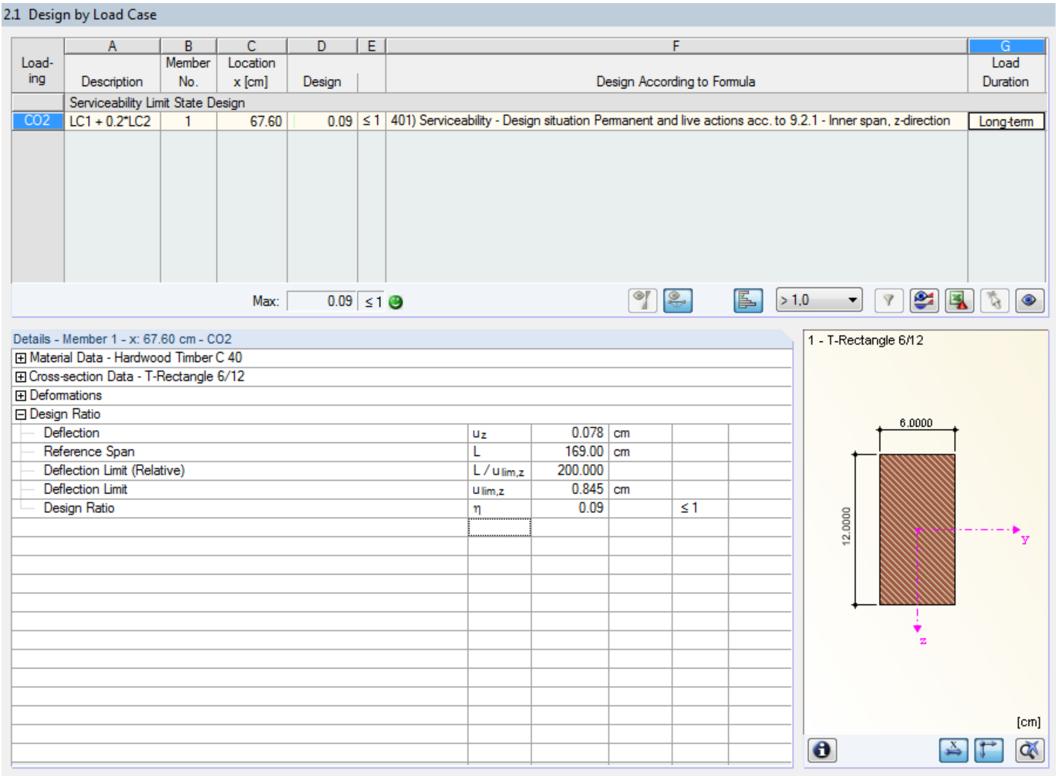


Figure 8.14: Window *2.1 Design by Load Case*

Literature

[1] *NBR 7190:1997: Design of wooden structures*. Associação Brasileira de Normas Técnicas, 1997.

[2] *Notas de Aula de Estruturas de Madeira*. Francisco A. R. Gesualdo, 2003.

Index