Dlubal

Version
February 2016

Add-on Module

RF-TIMBER SANS

Design of Timber Members Acc. to SANS 10163-1 and 10163-2

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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 SANS

The South African Standards SANS 10163-1 [1] and SANS 10163-2 [2] incorporate design provision for limit-states design (LSD) and allowable stress design (ASD). With the RFEM add-on module RF-TIMBER SANS from the company DLUBAL all users obtain a powerful tool for the design of timber structures modeled with member elements according to those Standards.

RF-TIMBER SANS performs all cross-section resistance designs, stability analyses, and deformation analyses provided by the standard. The stability analysis is carried out according to the equivalent member method or the second-order analysis. When the equivalent member method is applied, the program considers stability factors based on effective buckling lengths and effective lengths for lateral buckling. Second order analysis requires definition of imperfections in RFEM and calculates with unit stability factors for compression with buckling. In additional to this, the fire resistance design is possible.

In timber construction, the serviceability limit state is an important design. In this connection, selected load cases, load combinations, and allowed result combinations can be checked for limit deflection. The limit deformations are preset according to the Standard, and can be adjusted, if necessary. In addition to this, it is possible to specify reference lengths and precambers that will be considered accordingly in the design.

If necessary, you can optimize standardized or parametric cross-sections and export them to RFEM. Separate design cases allow for a separate design of large systems or analysis of variants.

RF-TIMBER SANS is one of the add-on modules integrated in the RFEM environment. Thus, the design-relevant input data is preset when you open the module. Subsequent to the design, you can use the graphical RFEM user interface to evaluate the results. Last but not least, you can document the checks from the analysis of internal forces to the design in the global printout report.

We hope you will enjoy working with RF-TIMBER SANS.

Your DLUBAL Team

1.2 RF-TIMBER SANS - Team

The following people were involved in the development of RF-TIMBER SANS:

Program coordination

Dipl.-Ing. Georg Dlubal
Ing. Myroslava Petronyuk
Dipl.-Ing. (FH) Gerhard Rehm

Programming

Ing. Tomáš Drtina
Darya Shatalova
Ing. Jiří Patrák

Cross-section and material database

Ing. Ph.D. Jan Rybín
Mgr. Petr Oulehle
Ing. Jiří Kubíček
Peter Roth, Bc.

Program design, dialog figures, icons

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

Program supervision

Ing. Myroslava Petronyuk
Dipl.-Ing. (FH) Gerhard Rehm
Ing. Ondřej Švorc
Dipl.-Ing. (FH) Bastian Kuhn

Localization, manual

Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Markéta Fišerová, Bc.
Ing. Lara Freyer
Ing. Alessandra Grosso, Ph.D.
Ing. Ladislav Kábrt
Eng.º Nilton Lopes
Mgr. Ing. Hana Macková
Ing. Téc. Ind. José Martínez
Ing. Myroslava Petronyuk
Mgr. Jagoda Podgórna
Mgr. Petra Pokorná
Chelsea Prokop, BSc Eng
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
M.Sc. Sonja von Bloh
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Gerhard Rehm
Dipl.-Ing. (FH) Lukas Sühnel

1.3 Using the Manual

Topics like installation, graphical user interface, results evaluation, and printout are described in detail in the manual of the main program RFEM. The present manual focuses on typical features of the RF-TIMBER SANS 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 find the index. However, if you still cannot find what you are looking for, please check our website **www.dlubal.com** where you can go through our *FAQ* pages by selecting particular criteria.

1.4 Starting RF-TIMBER SANS

RFEM provides the following options to start the add-on module RF-TIMBER SANS.

Menu

To start the program in the RFEM menu bar, click

Add-on Modules → **Design - Timber** → **RF-TIMBER SANS**.

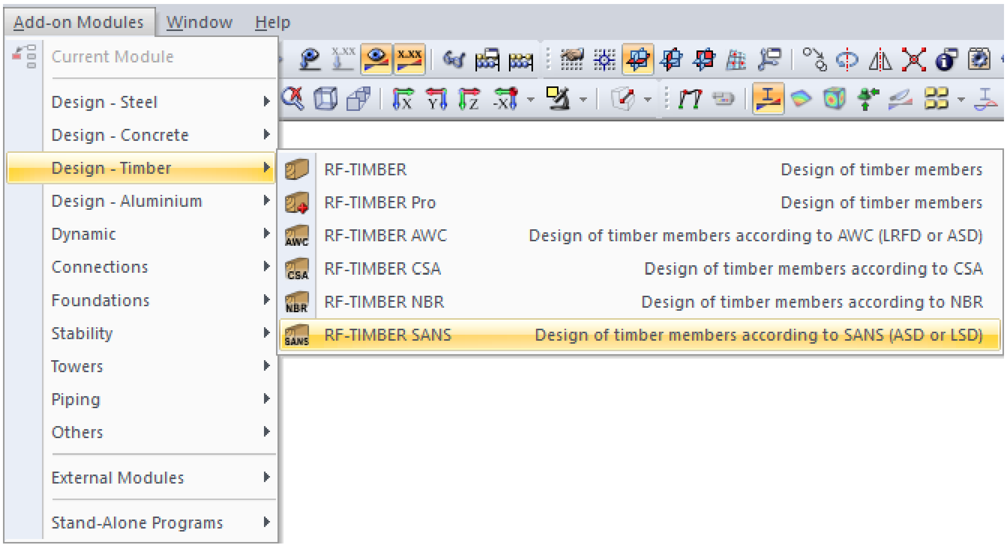


Figure 1.1: Menu: Add-on Modules → Design – Timber → RF-TIMBER SANS

Navigator

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

Add-on Modules → RF-TIMBER SANS.

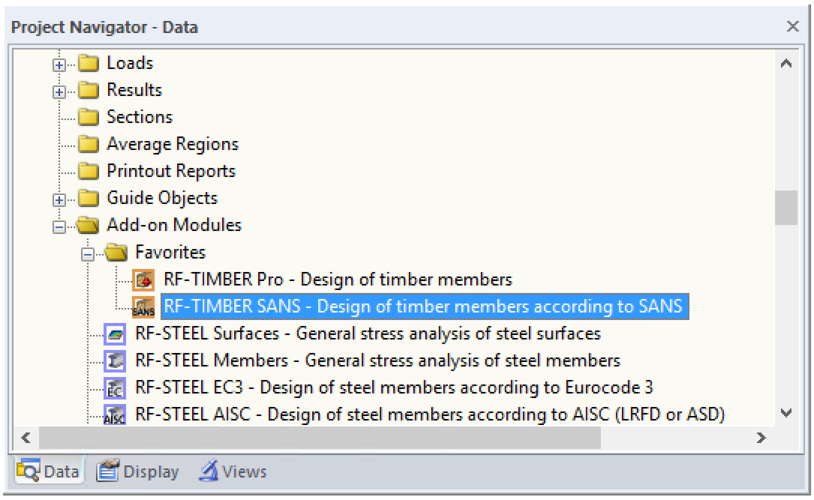
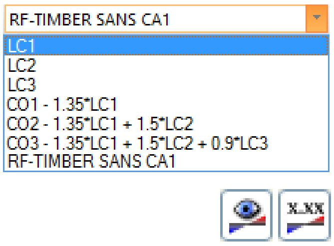


Figure 1.2: Data navigator: *Add-on Modules → RF-TIMBER SANS*



Panel

If results from RF-TIMBER SANS are already available in the RFEM model, you can also open the design module in the panel:

Set the relevant RF-TIMBER SANS design case in the load case list of the RFEM toolbar. Then click the [Show results] button to graphically display the design criterion on the members.

When the results display is activated, the panel is available, too. Now you can click the button [RF-TIMBER SANS] in the panel to open the module.

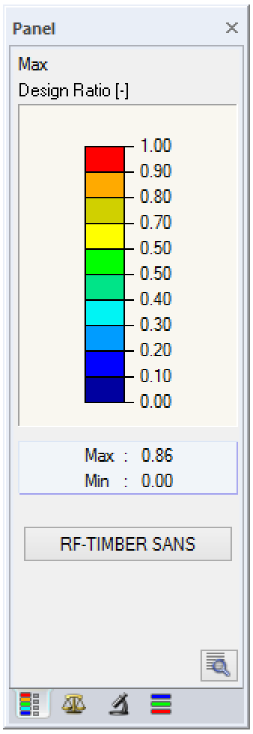


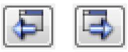
Figure 1.3: Panel button [RF-TIMBER SANS]

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 available windows that can be currently selected. The drop-down list above the navigator contains the design cases (see Chapter 7.1, page 63).

The design relevant data is defined in several input windows. When you start RF-TIMBER SANS for the first time, the following parameters are imported automatically:

- Members and sets of members
- Load cases, load combinations, 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].

To save the results, click [OK]. Thus, you exit RF-TIMBER SANS and return to the main program. To exit the module without saving the new data, click [Cancel].

2.1 General Data

In the 1.1 *General Data* Window, you select the members, sets of members, and actions that you want to design. The tabs are managing the load cases, load combinations, and result combinations for the different designs.

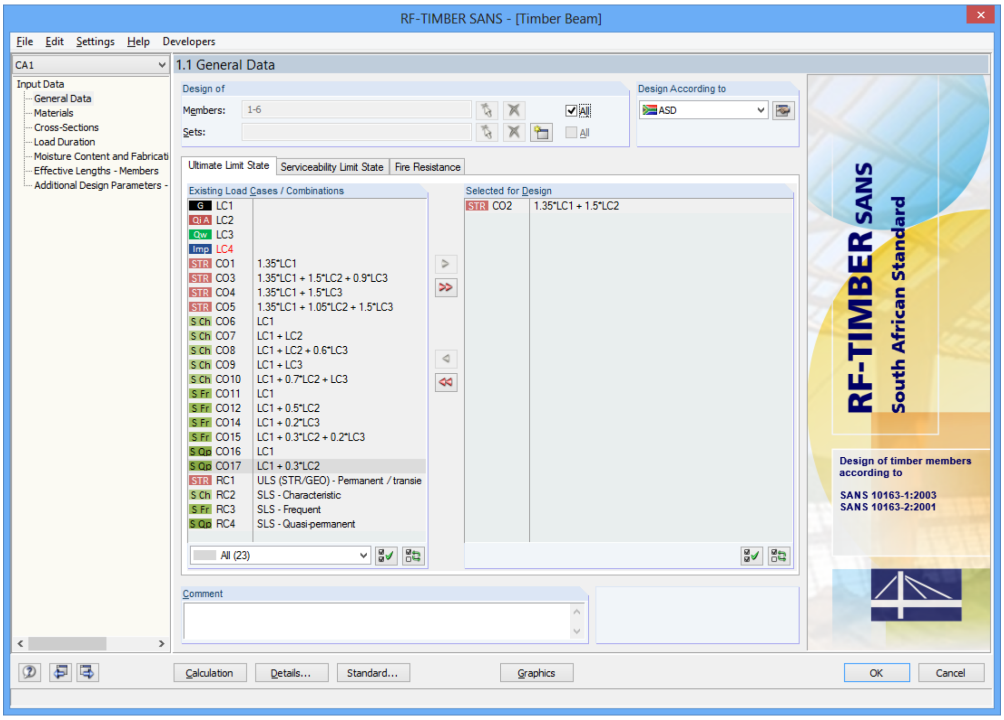


Figure 2.1: Window 1.1 *General Data*

Design of

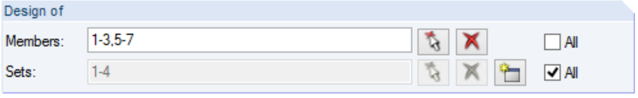


Figure 2.2: Design of members and sets of members

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 list of the numbers preset in the field can be cleared by clicking the [Delete] button. Alternatively, you can select the objects graphically in the RFEM work window after clicking [↖].

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 the results 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 appears where you can specify the parameters of set of members.

If beams are divided by nodes, it is necessary to verify them as a set of members.

Design according to

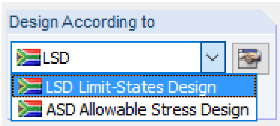


Figure 2.3: Design according to LSD or ASD

The options of the list box control whether the analysis is carried out according to the provisions of the *Limit-States Design* (LSD) or the *Allowable Stress Design* (ASD).

Comment

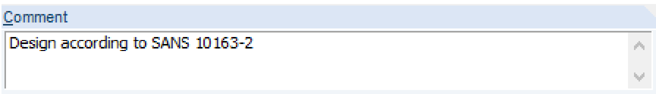


Figure 2.4: User-defined comment

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

2.1.1 Ultimate Limit State

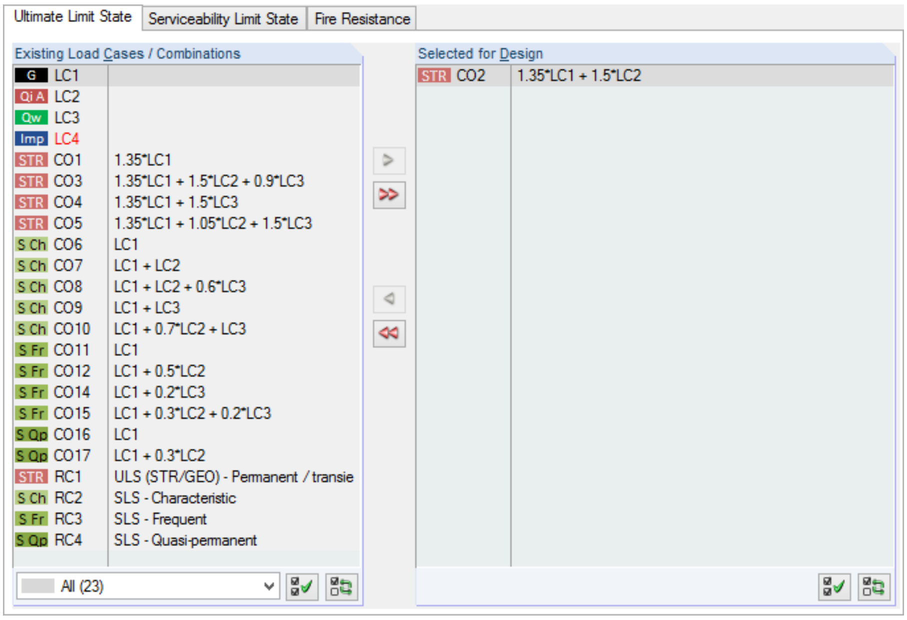


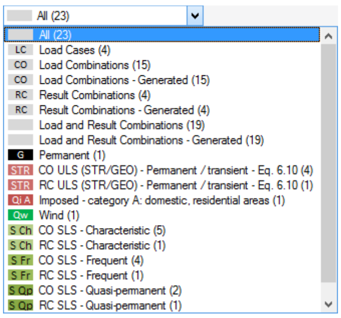
Figure 2.5: Window 1.1 *General Data*, tab *Ultimate Limit State*

Existing Load Cases / Combinations

This column lists all load cases, load combinations, and result combinations created in RFEM.

To transfer selected entries 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 transfer multiple entries at once, select them while pressing the [Ctrl] key, as common for Windows applications.



Load cases highlighted in red, like LC 4 in Figure 2.5, cannot be designed. This happens when the load cases are defined without any load data or the load cases contain only imperfections. When you transfer the load cases, a corresponding warning appears.

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

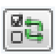
	Selects all load cases in the list
	Inverts the selection of load cases

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

Selected for Design

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

The design of an enveloping max/min result combination is performed faster than the design of all contained load cases and load combinations.

Result combinations should be selected for design only for dynamic combinations. For "usual" combinations, load combinations are recommended.

2.1.2 Serviceability Limit State

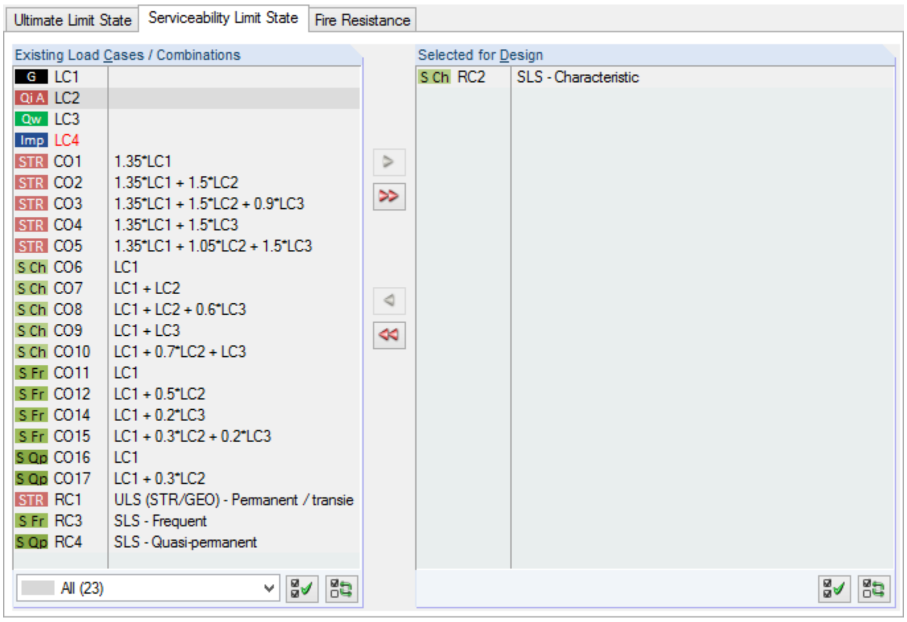


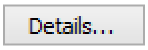
Figure 2.6: Window 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

This section lists all load cases, load combinations, and result combinations created in RFEM.

Selected for Design

Load cases, load combinations, and result combinations can be added or removed (see Chapter 2.1.1).



The limit value of the deformation is controlled by the settings in the *Details* dialog box, tab *Serviceability* (see Figure 3.3, page 42) which you can call up by clicking the [Details] button.

In the 1.11 *Serviceability Data* Window, the reference lengths decisive for the deformation check are managed (see Chapter 2.13, page 67).

2.1.3 Fire Resistance

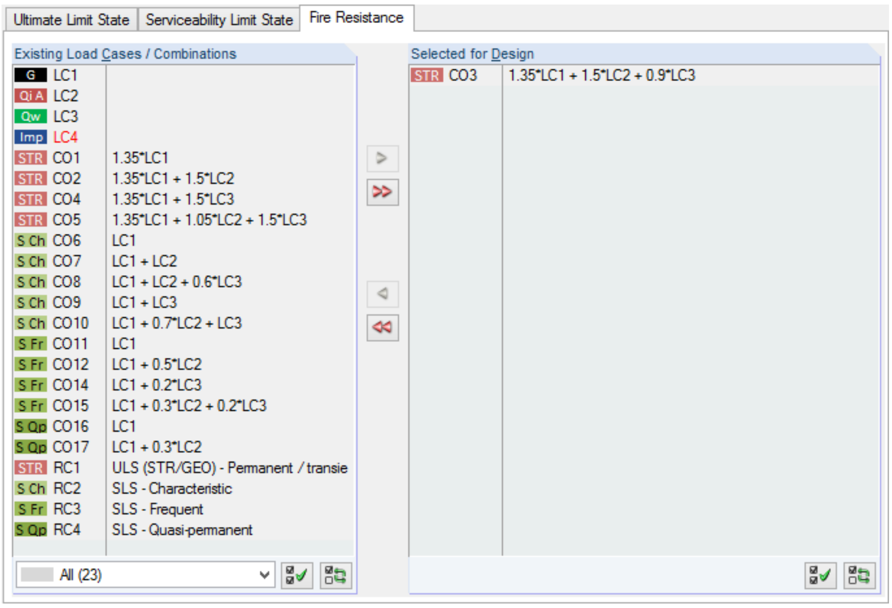


Figure 2.7: Window 1.1 *General Data*, tab *Fire Resistance*

Existing Load Cases and Combinations

All load cases, load combinations, and result combinations created in RFEM are listed here.

Selected for Design

Load cases, load combinations and allowed result combinations can be added or removed (see Chapter 2.1.1).

The fire resistance design is performed by means of reduced cross-sections. The general specifications for the fire resistance design are managed in the dialog boxes *Standard* (see Chapter 2.1.4) and *Details,* tab *Fire Resistance* (see Figure 3.4, page 43).

2.1.4 Standard

The drop-down lists in the 1.1 *General Data* Window allow you to select the method of design whose parameters are to be applied to the design.

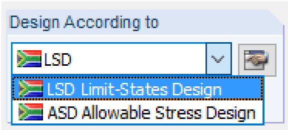


Figure 2.8: Selecting design method

All preset parameters which are applied to the design are specified in the *Standard* dialog box.

To check and, if necessary, adjust these parameters, you can use the [Standard] button in all input windows in order to open the *Standard* dialog box. It consists of three tabs. The content of the dialog box depends on the selected design method.

Load Duration Factors

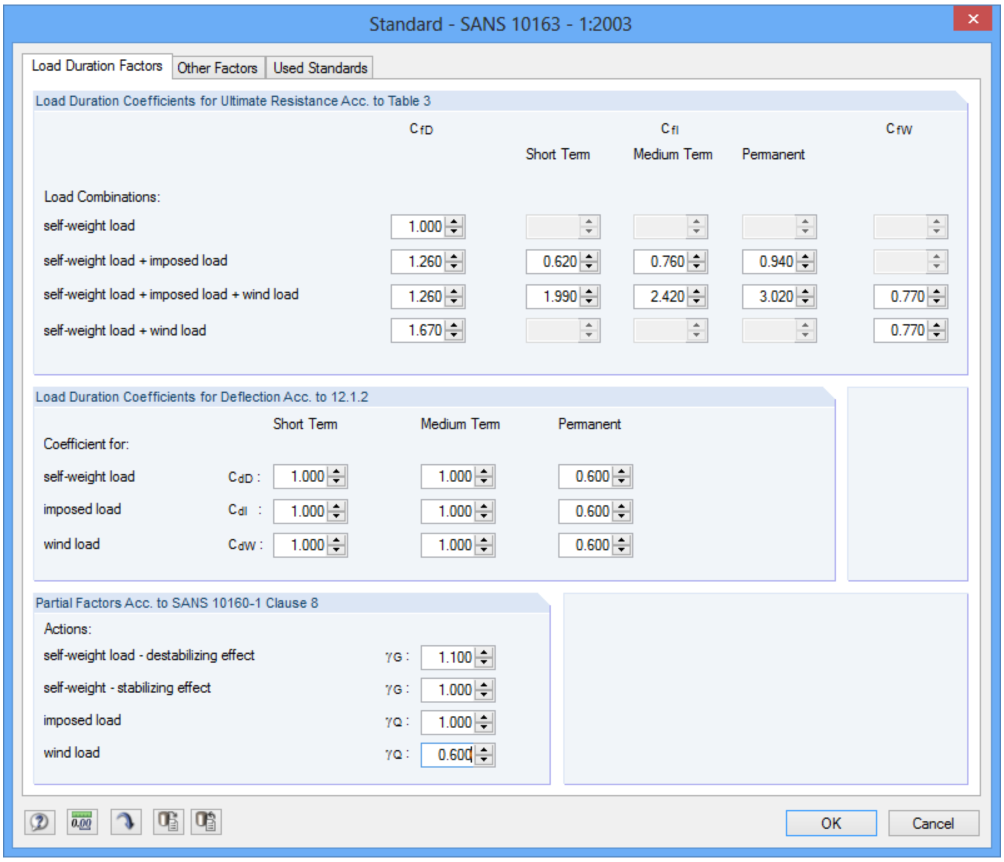


Figure 2.9: Dialog box *Standard*, tab *Load Duration Factors* for **LSD** design method

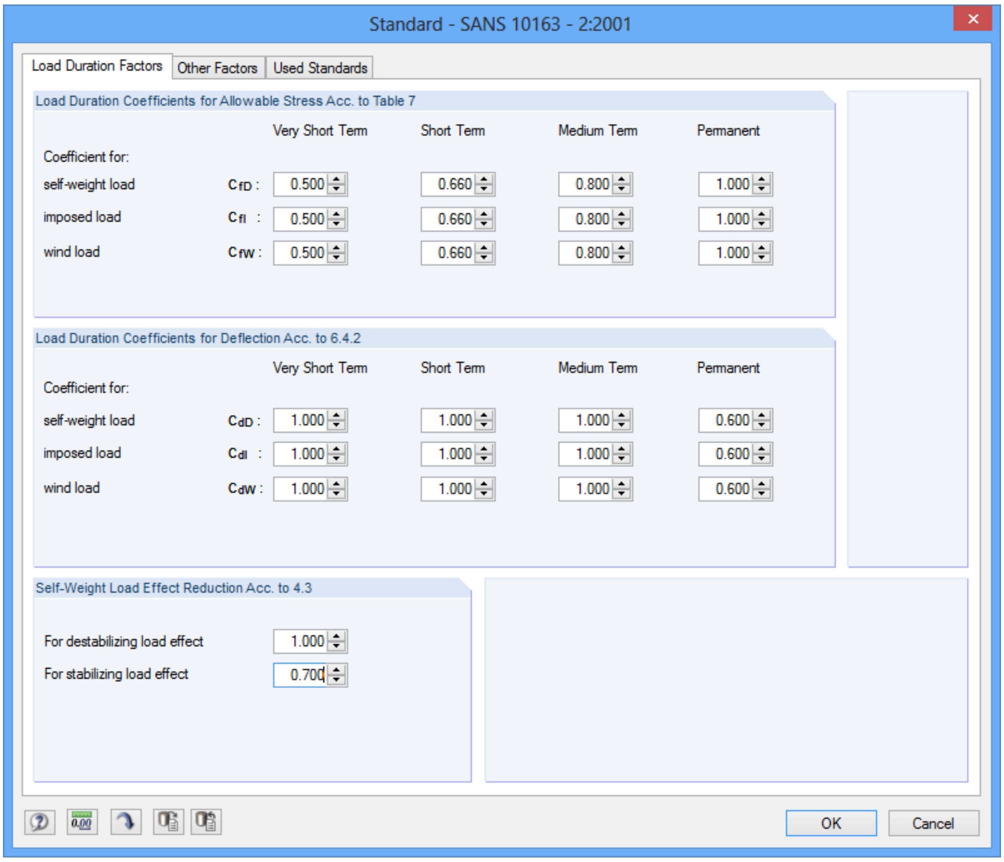


Figure 2.10: Dialog box *Standard*, tab *Load Duration Factors* for **ASD** design method

In the dialog box sections, you can check or, if necessary, modify the *Load Duration Coefficients* or the *Self-Weight Load Effect Reduction*.

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

Button	**Function**
	Resets the program's default settings
	Imports user-defined standard settings
	Saves the current settings as default
	Deletes a user-defined Standard

Table 2.2: Buttons in the *Standard* dialog box

Other Factors

In the second tab of the *Standard* dialog box, you find more factors significant for the design.

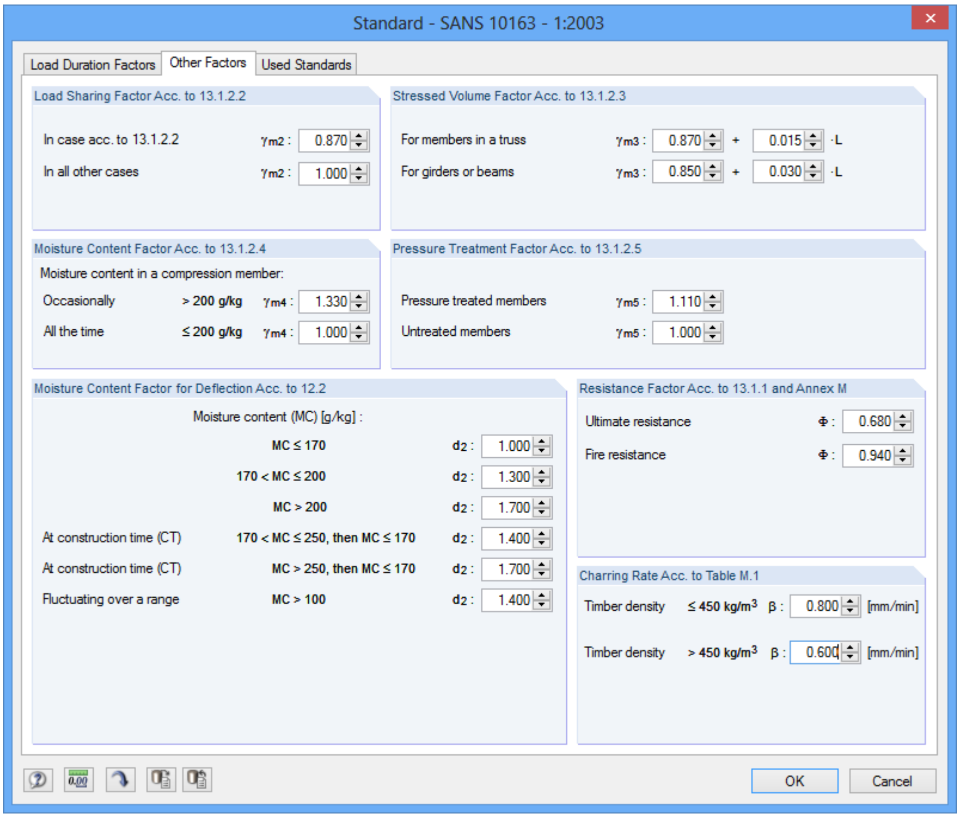


Figure 2.11: Dialog box *Standard*, tab *Other Factors* for LSD design method

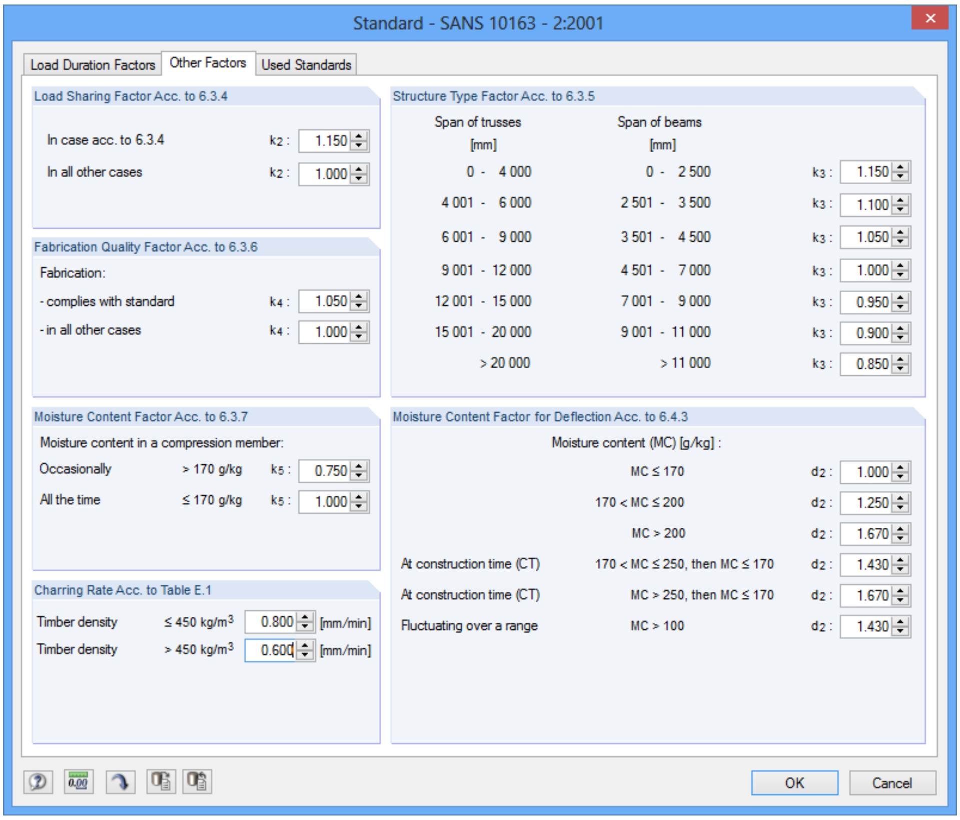


Figure 2.12: Dialog box *Standard*, tab *Other Factors* for ASD design method

Used Standards

The last tab informs you about the Standards according to which the design is performed.

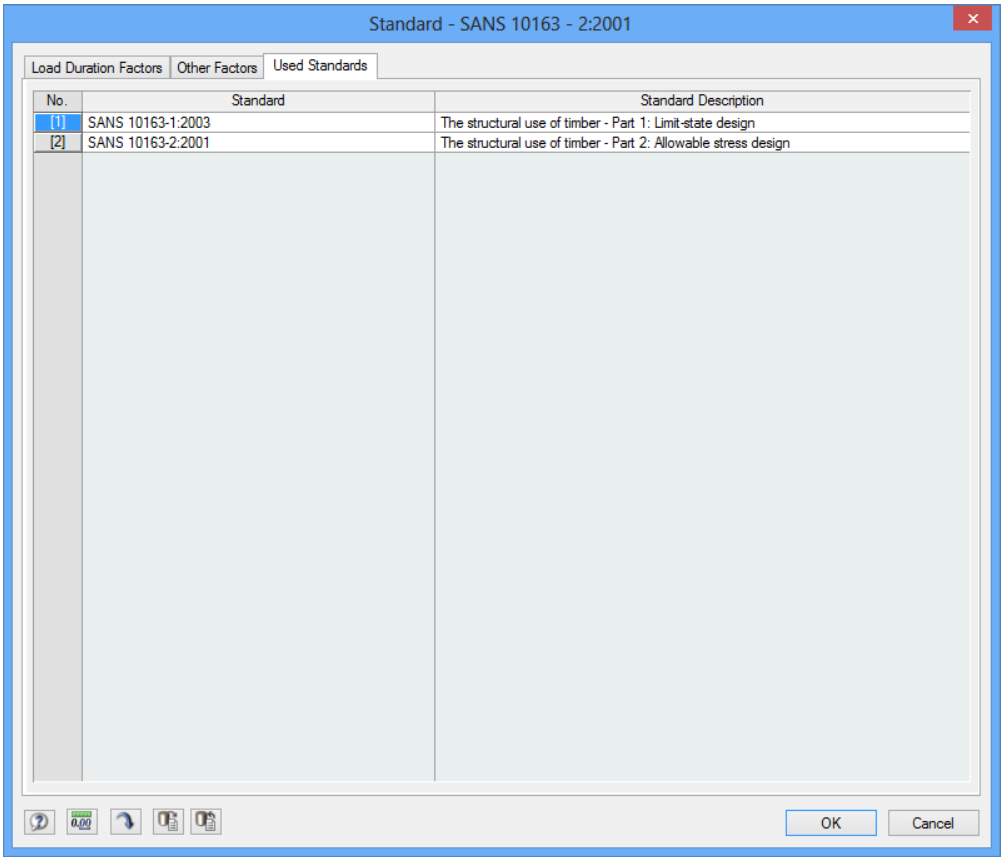


Figure 2.13: Dialog box *Standard*, tab *Used Standards*

2.2 Materials

The window consists of two parts. In the upper part, all materials created in RFEM are listed. In the *Material Properties* section, the properties of the current material, that is, the table row currently selected in the upper section, are displayed.

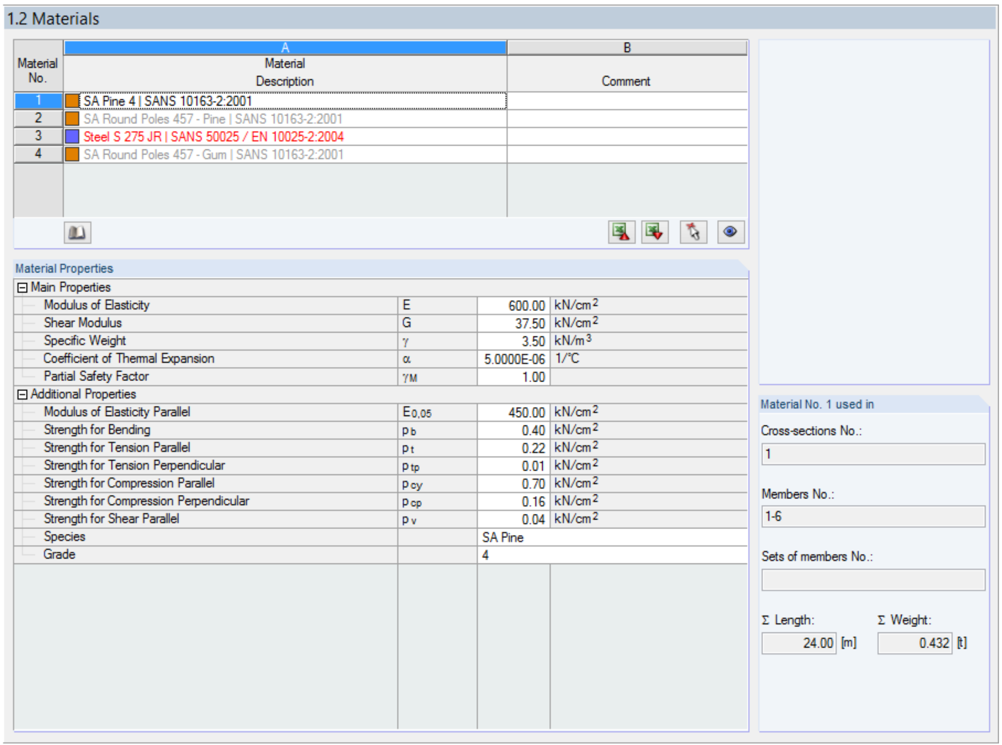


Figure 2.14: Window 1.2 *Materials*

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

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

To adjust the units and decimal places of material properties and stresses, select in the module's menu **Settings** → **Units and Decimal Places** (see Chapter 7.3, page 67).

Material Description

The materials defined in RFEM are preset, but it is always possible to modify them: To do this, click the material in column A. Then click [▼] or press function key [F7] to open the material list.

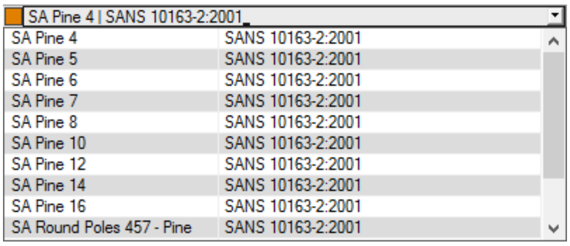


Figure 2.15: List of materials for ASD design method

According to the design concept of SANS 10163-1 [1] and SANS 10163-2 [2], the list includes only materials of the South African Standard.

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

It is not possible to edit the material properties in the RF-TIMBER SANS module.

Material Library

Many materials are available in the library. To open the corresponding dialog box, select

Edit → Material Library

or click the button shown on the left.

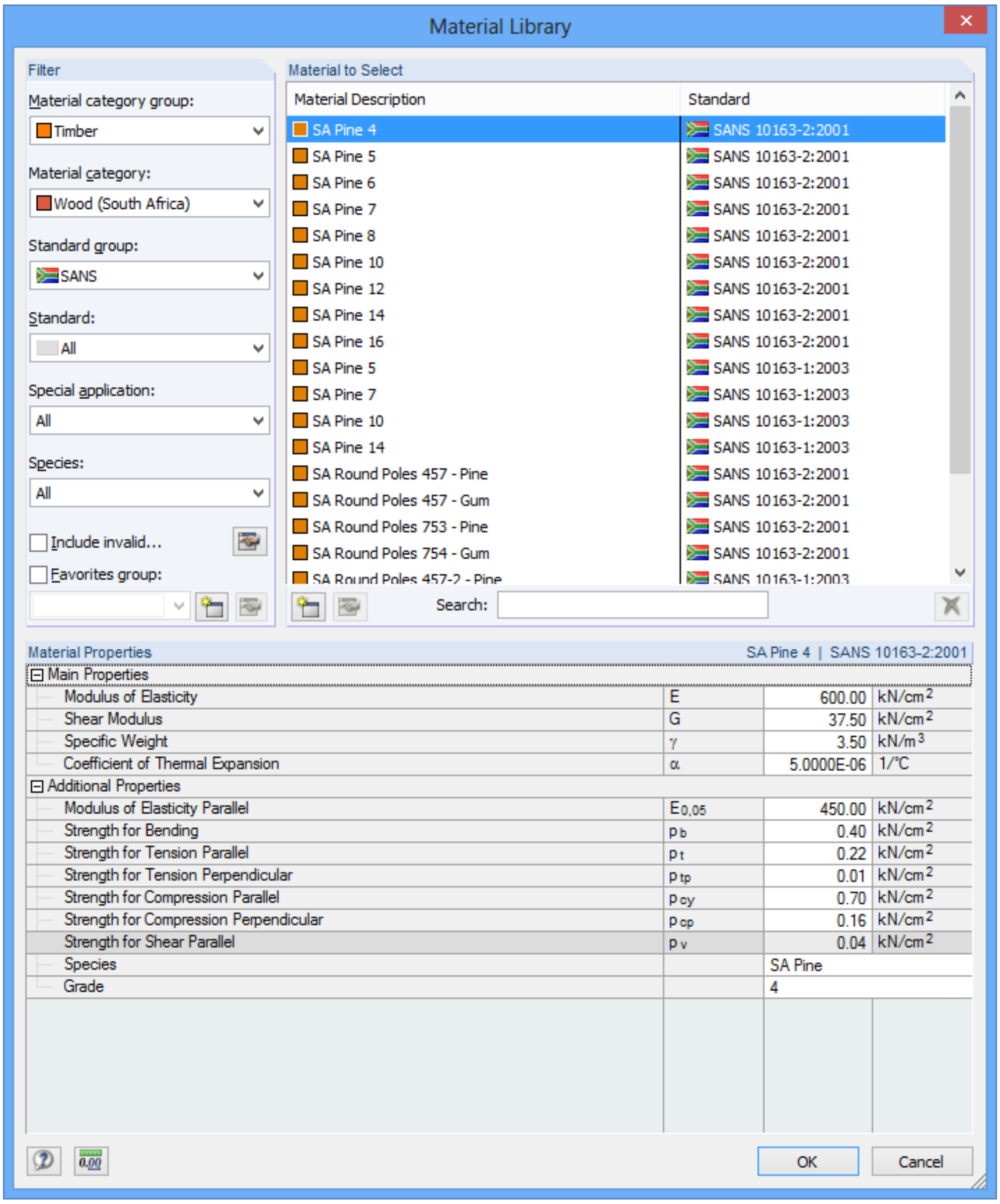


Figure 2.16: Dialog box *Material Library*

In the *Filter* section, *SANS* is preset as 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 the module RF-TIMBER SANS.

Chapter 4.3 of the RFEM manual describes how materials can be filtered, added, or rearranged.

Material Properties

The lower section of Window 1.2 contains the specified strength values for bending, tension parallel, tension perpendicular, compression parallel, compression perpendicular, shear, as well as modulus of elasticity for design of compression members.

The compressive stress parallel to the grain, p_{cy}, in material database is defined for the short columns, which slenderness ratio is $l_e/b \leq 10$. For columns with slenderness ratio $l_e/b > 10$ the compressive stress parallel is calculated in the module according to [2] Chapter 7.4.1.

Standard...

The design values of the material strengths are to be determined with the modification factors. The factors can be modified in the *Standard Settings* dialog box (see Figure 2.10 and Figure 2.12).

2.3 Cross-Sections

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

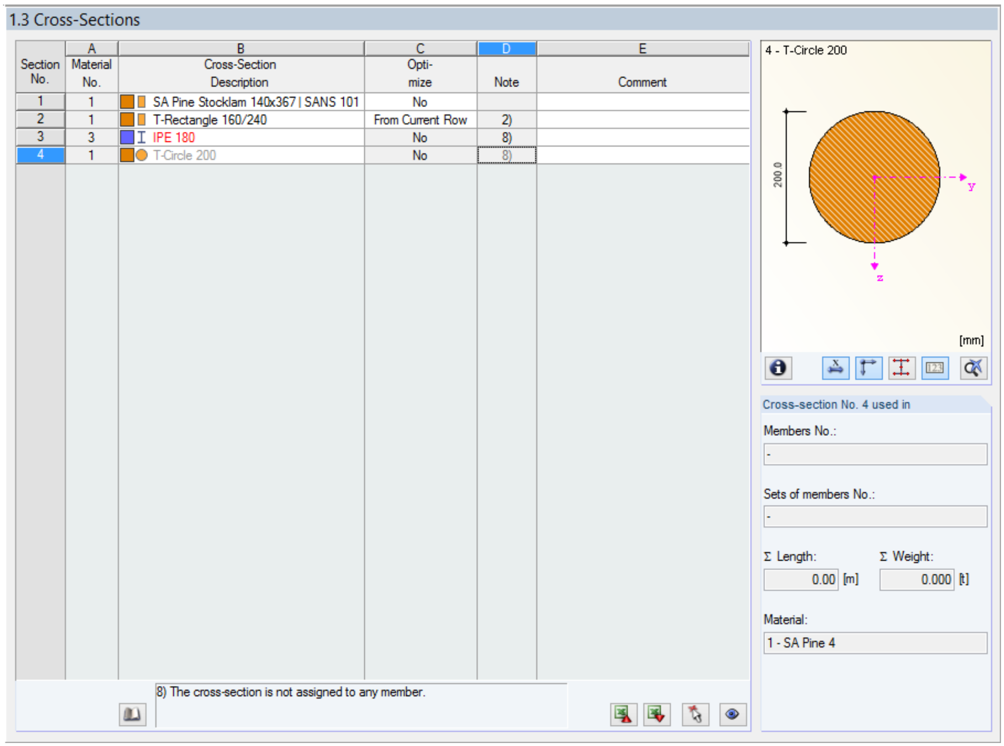


Figure 2.17: Window 1.3 *Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM are preset, as well as the assigned material numbers.

The design is possible for parametric rectangular and circular timber cross-sections and for standardized rectangular timber cross-sections according to the South African Standard SANS 10163.

To modify a cross-section, click the entry in column B selecting this field. Click [Cross-section Library] or [...] in the field or press function key [F7] to open the cross-section table of the current input field (see the following two figures).

In those dialog boxes, you can select a different cross-section or a different cross-section table. To select a different cross-section category, click [Back to cross-section library] to access the general cross-section library.

Chapter 4.13 of the RFEM manual describes how cross-sections can be selected from the library.

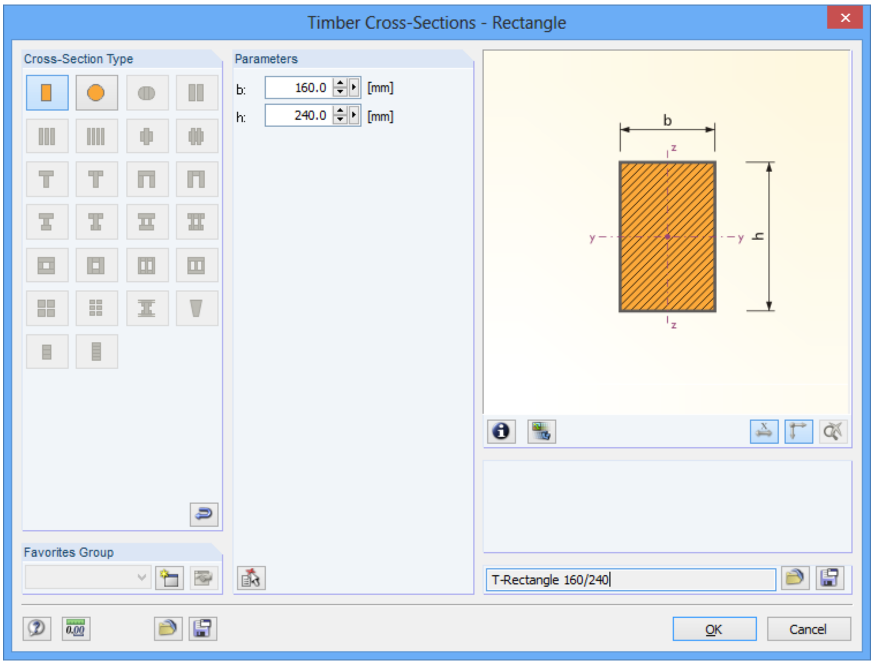


Figure 2.18: Parametric timber cross-sections of the library

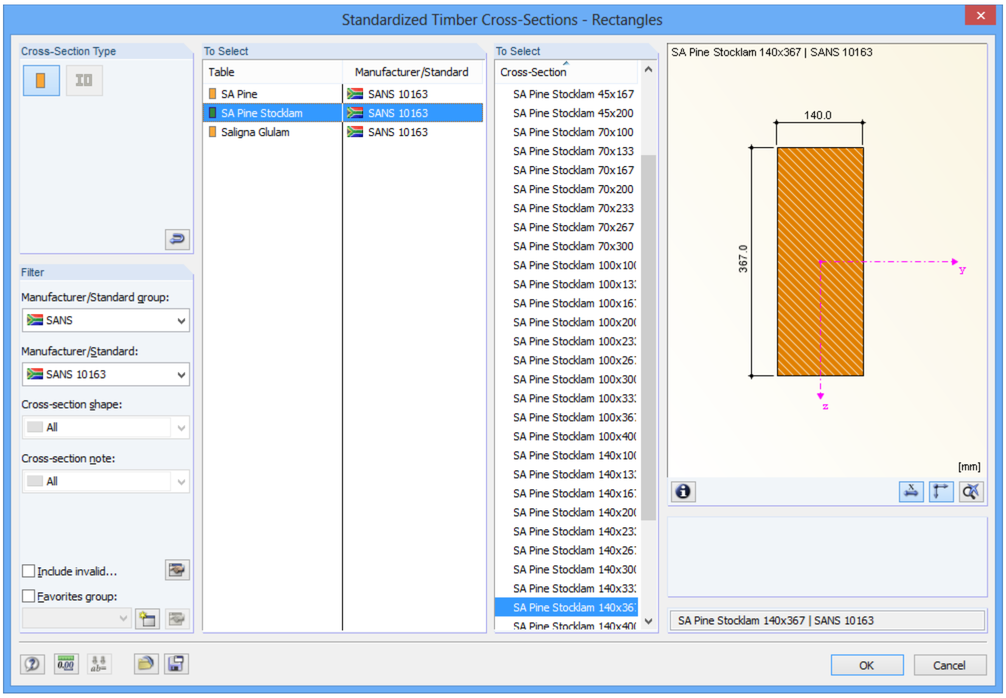
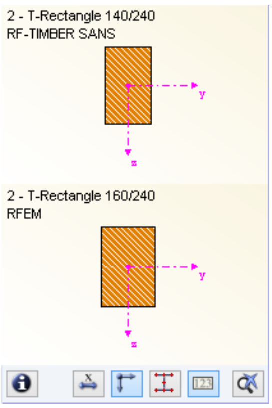


Figure 2.19: Standardized timber cross-sections of the library



The new cross-section description can be entered in the text box directly. If the database contains an entry, RF-TIMBER SANS imports these cross-section parameters, too.

A modified cross-section will be highlighted in blue.

If cross-sections specified in RF-TIMBER SANS are different from the ones used in RFEM, both cross-sections are displayed in the graphic in the right part of the window. The designs will be performed with the internal forces from RFEM for the cross-section selected in RF-TIMBER SANS.

Max. Design Ratio

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

Optimize

Details...

You can optimize all rectangular and circular cross-sections: For the RFEM internal forces, the program searches the cross-section that comes as close as possible to a user-defined maximum utilization ratio. You can define the maximum ratio in the *Other* tab of the *Details* dialog box, (see Figure 3.5, page 44).

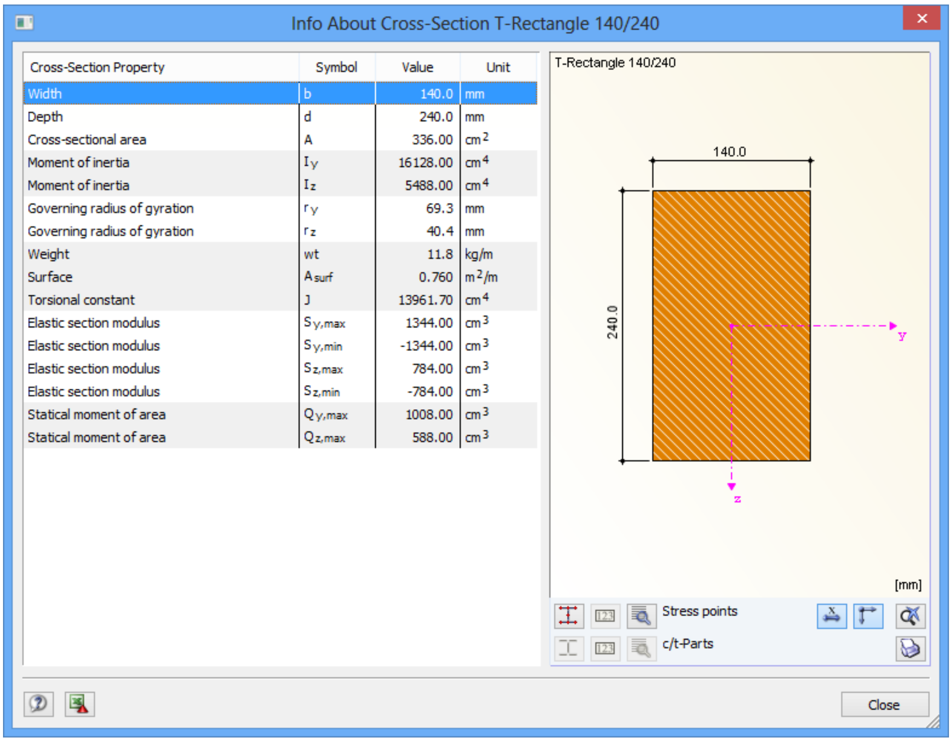
If you want to optimize a cross-section, open the drop-down list in column D or E. Recommendations for optimizing cross-sections can be found in Chapter 7.2 on page 64.

Remark

This column shows remarks in the form of footers that are described in detail below the cross-section list.

Info About Cross-Section

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



Cross-Section Property	Symbol	Value	Unit
Width	b	140.0	mm
Depth	d	240.0	mm
Cross-sectional area	A	336.00	cm^2
Moment of inertia	I_y	16128.00	cm^4
Moment of inertia	I_z	5488.00	cm^4
Governing radius of gyration	r_y	69.3	mm
Governing radius of gyration	r_z	40.4	mm
Weight	wt	11.8	kg/m
Surface	A_{surf}	0.760	m^2/m
Torsional constant	J	13961.70	cm^4
Elastic section modulus	$S_{y,max}$	1344.00	cm^3
Elastic section modulus	$S_{y,min}$	-1344.00	cm^3
Elastic section modulus	$S_{z,max}$	784.00	cm^3
Elastic section modulus	$S_{z,min}$	-784.00	cm^3
Statical moment of area	$Q_{y,max}$	1008.00	cm^3
Statical moment of area	$Q_{z,max}$	588.00	cm^3

Figure 2.20: Dialog box *Info About Cross-Section*

The right part of the dialog box shows the currently selected cross-section.

The buttons below the graphic have the following functions:

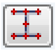
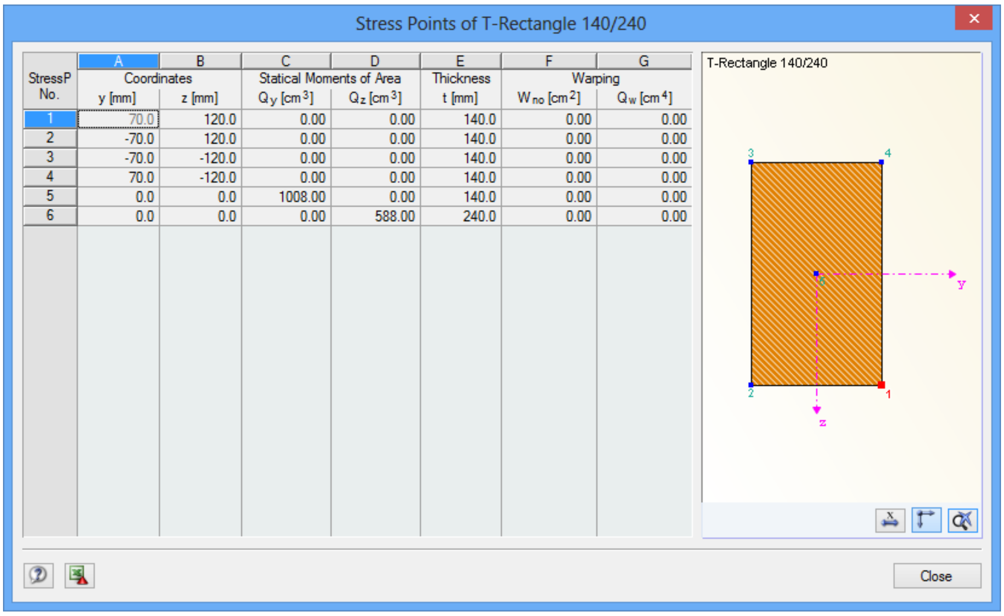
Button	Function
	Displays or hides the stress points
	Displays or hides the numbers of stress points
	Shows the details of the stress points (see Figure 2.21)
	Displays or hides the dimensions of the cross-section
	Displays or hides the principal axes of the cross-section
	Resets the full view of the cross-section graphic
	Prints the cross-section values and cross-section graphic

Table 2.3: Buttons of cross-section graphic

Click [Details] to call up specific information on the stress points (distances to center of gravity, statical moments of area, etc.).

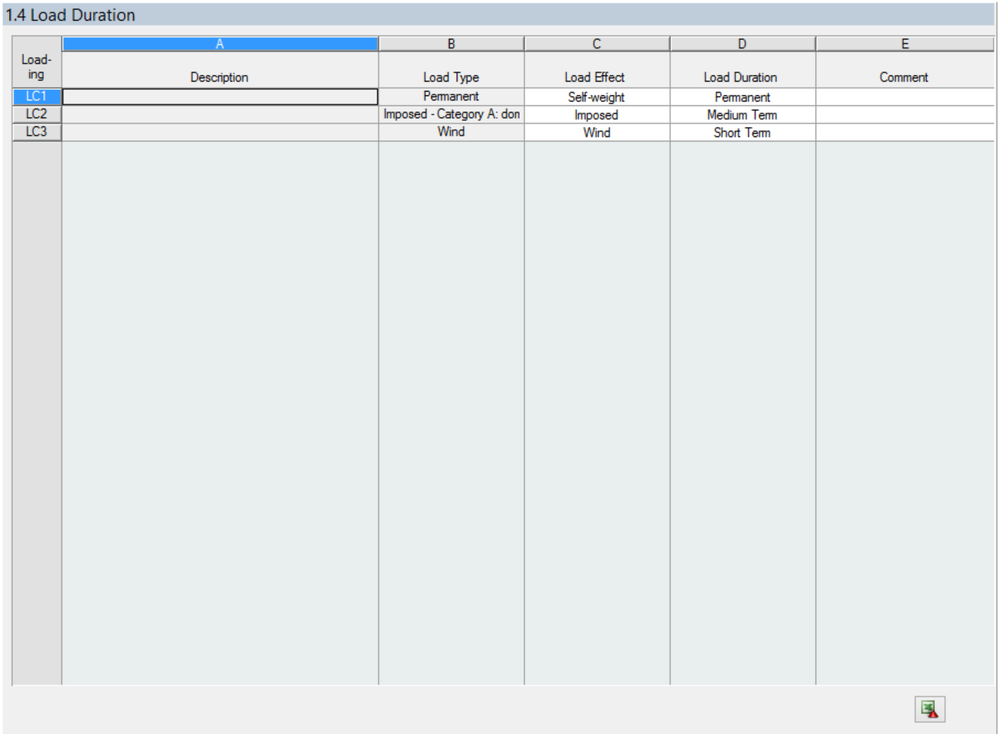


	A	B	C	D	E	F	G
StressP No.	Coordinates		Statical Moments of Area		Thickness	Warping	
	y [mm]	z [mm]	Q_y [cm³]	Q_z [cm³]	t [mm]	W_{no} [cm²]	Q_w [cm⁴]
1	70.0	120.0	0.00	0.00	140.0	0.00	0.00
2	-70.0	120.0	0.00	0.00	140.0	0.00	0.00
3	-70.0	-120.0	0.00	0.00	140.0	0.00	0.00
4	70.0	-120.0	0.00	0.00	140.0	0.00	0.00
5	0.0	0.0	1008.00	0.00	140.0	0.00	0.00
6	0.0	0.0	0.00	588.00	240.0	0.00	0.00

Figure 2.21: Dialog box *Stress Points*

2.4 Load Duration

In Window 1.4, you define the load duration to consider factors reflecting the different load duration for all chosen load cases.



1.4 Load Duration

Load-ing	A Description	B Load Type	C Load Effect	D Load Duration	E Comment
LC1		Permanent	Self-weight	Permanent	
LC2		Imposed - Category A: don	Imposed	Medium Term	
LC3		Wind	Wind	Short Term	

Figure 2.22: Window 1.4 *Load Duration*

Loading

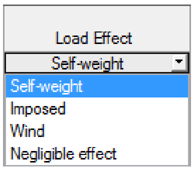
All load cases selected in the 1.1 *General Data* Window are listed here.

Description

The load case descriptions make the classification easier.

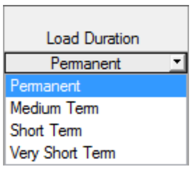
Load Type

This table column shows the load cases' types of action as defined while creating them in RFEM. They are the basis for the presetting in the subsequent table column.



Load Effect

The load effect defines the type of loading which the construction is loaded. In this column you can define the load effect of each load case: By clicking the cell in column C, the [▼] button becomes available. If there is a load case you do not want to take into account when determining the load duration factor, you can choose *Negligible load effect* in the list box.

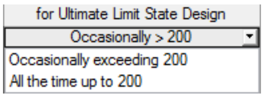


Load Duration

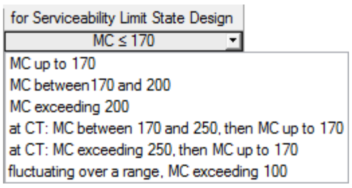
Loads must be assigned to classes of load duration. The classification of actions is specified in [1] Table 3 or in [1] Table 7, for example. The load duration can be changed as follows: Click the cell in column D to select the box, and then use the [▼] button to open the list.

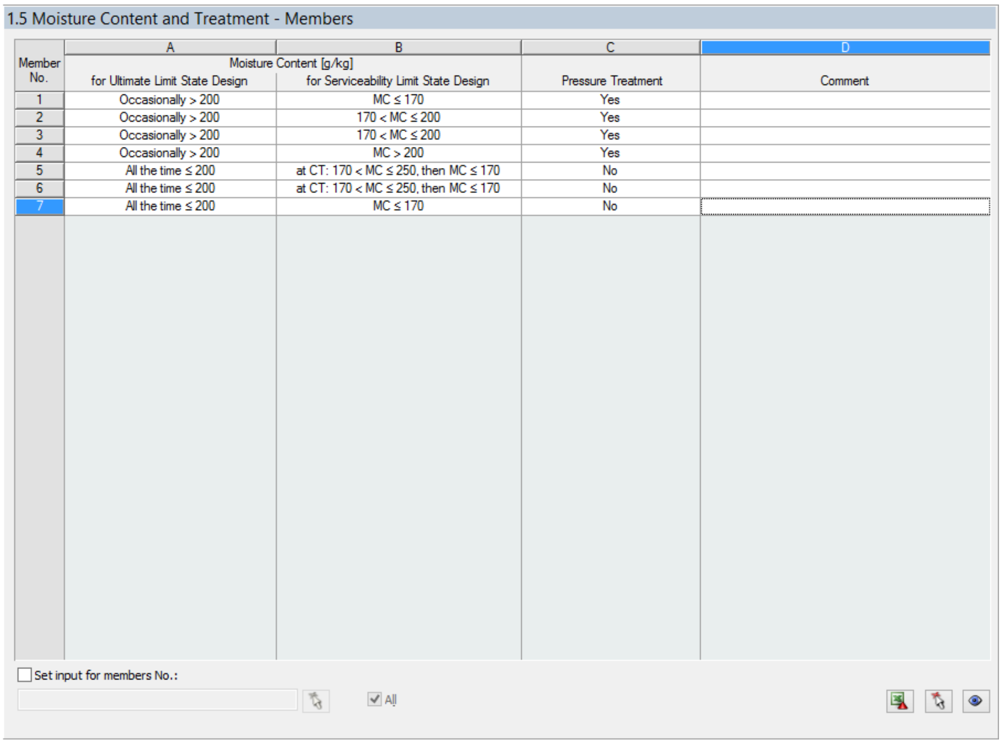
The classes of the load effect and load duration are required for the determination of the load duration factors, γ_{m1} and d_1 (for LSD method), and the factors, k_1 and d_1 (for ASD method). The default values of the load duration coefficients C_d, C_f or C_l can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.9 and Figure 2.10, page 13 f.).

2.5 Moisture Content and Treatment - Members (LSD)



The determination of moisture makes it possible to assign the material factor for moisture content for ultimate limit state design, γ_{m4}, and for serviceability limit state design, d_2. The columns A or B are only available if you have set the relevant entries in the *Ultimate Limit State* or *Serviceability Limit State* tab of Window 1.1 *General Data*.

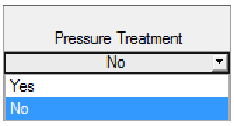




Member No.	A: Moisture Content [g/kg] for Ultimate Limit State Design	B: Moisture Content [g/kg] for Serviceability Limit State Design	C: Pressure Treatment	D: Comment
1	Occasionally > 200	MC ≤ 170	Yes	
2	Occasionally > 200	170 < MC ≤ 200	Yes	
3	Occasionally > 200	170 < MC ≤ 200	Yes	
4	Occasionally > 200	MC > 200	Yes	
5	All the time ≤ 200	at CT: 170 < MC ≤ 250, then MC ≤ 170	No	
6	All the time ≤ 200	at CT: 170 < MC ≤ 250, then MC ≤ 170	No	
7	All the time ≤ 200	MC ≤ 170	No	

Figure 2.23: Window 1.5 *Moisture Content and Treatment - Members* (LSD)

The default values of the factors γ_{m4}, d_2 and γ_{m5} can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.11, page 15).



By default, pressure treatment is not used. If you want to define treatment to the selected members, use the [▼] button.

Below the *Settings* table, you find the *Set input for members No.* check box. If it is selected, the settings entered afterward will be applied to the selected or even to *All* members. Members can be selected by entering the member number or by selecting them graphically with the [↖] button. This option is useful when you want to assign identical conditions to several members. Please note that settings that have been already defined cannot be changed subsequently by using this function.

The other buttons below the table have the following functions:

Button	**Function**
	Exports table into MS Excel or OpenOffice Calc
	Directs to the row of member graphically selected in RFEM work window
	Switches into RFEM work window

Table 2.4: Buttons in Window 1.5 *Moisture Content and Fabrication Quality - Members*

2.6 Moisture Content and Treatment - Sets of Members (LSD)

This window is only available if one or more sets of members have been selected in Window 1.1 *General Data*.

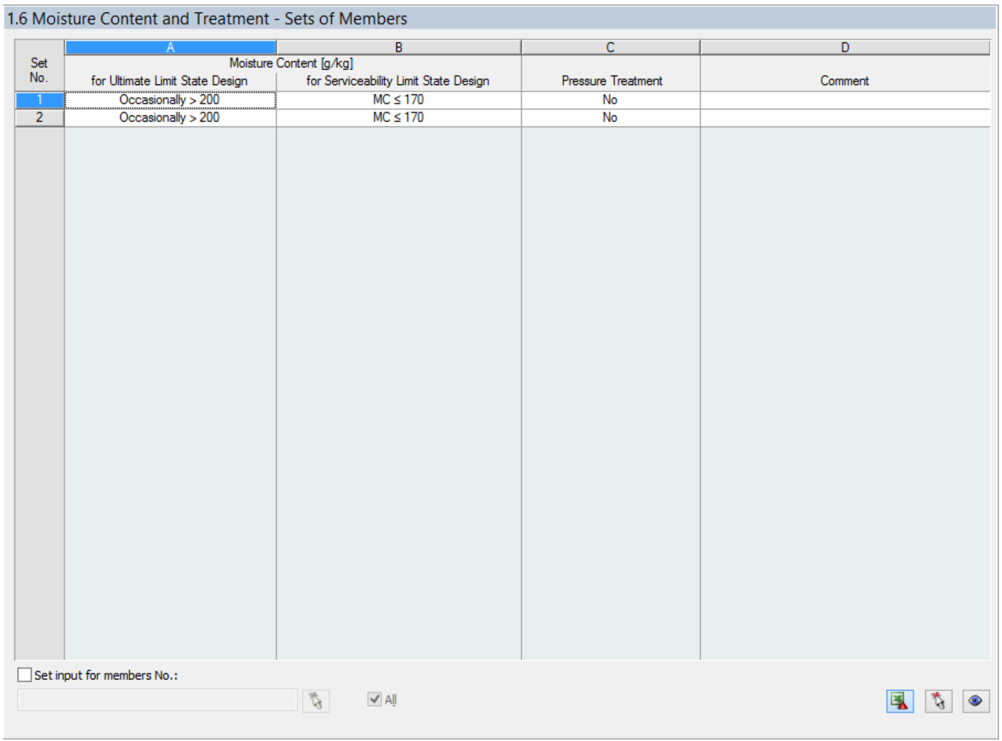
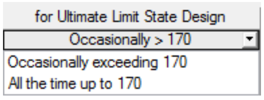


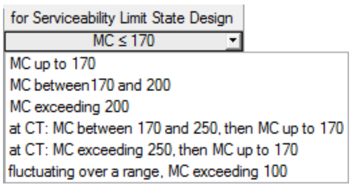
Figure 2.24: Window 1.6 *Moisture Content and Treatment – Set of Members* (LSD)

The concept of this window is similar to the one in the previous Window 1.6 *Moisture Content and Treatment - Members*. In this window, you can assign the moisture content and the fabrication quality of the wood to each set of members.

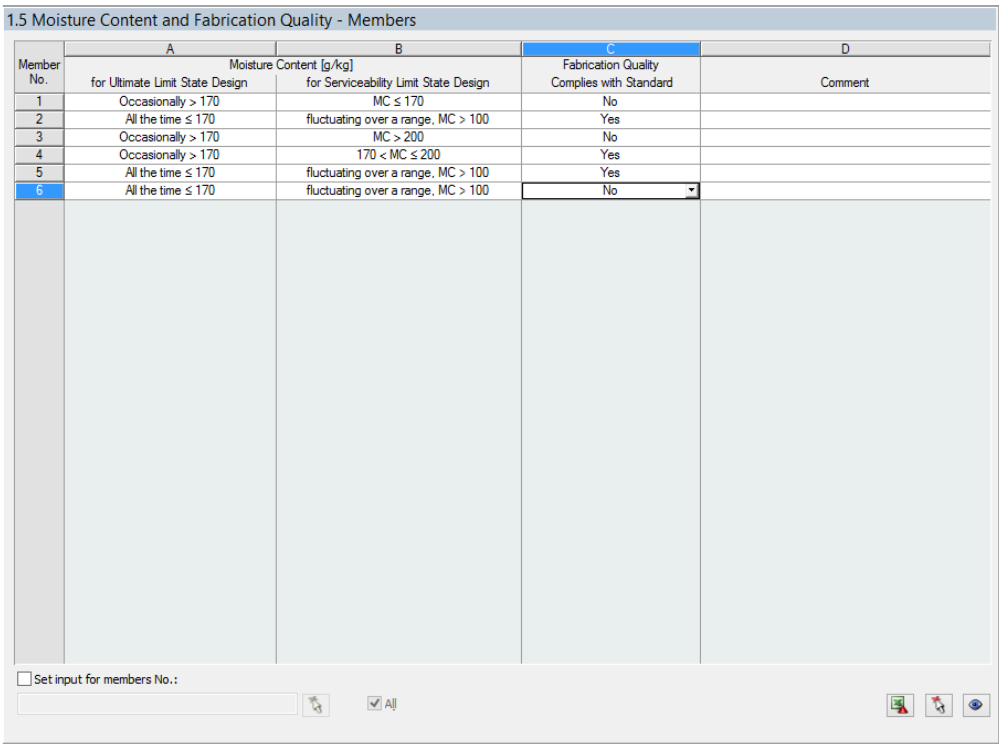
2.7 Moisture Content and Fabrication Quality - Members (ASD)

This window is similar to the 1.5 *Moisture Content and Treatment - Members* Window above (see Figure 2.23).





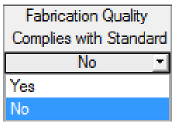
In this window it is possible to assign the moisture content factors for the *Ultimate Limit State Design*, k_5, and for the *Serviceability Limit State Design*, d_2.



Member No.	A: Moisture Content [g/kg] for Ultimate Limit State Design	B: for Serviceability Limit State Design	C: Fabrication Quality Complies with Standard	D: Comment
1	Occasionally > 170	MC ≤ 170	No	
2	All the time ≤ 170	fluctuating over a range, MC > 100	Yes	
3	Occasionally > 170	MC > 200	No	
4	Occasionally > 170	170 < MC ≤ 200	Yes	
5	All the time ≤ 170	fluctuating over a range, MC > 100	Yes	
6	All the time ≤ 170	fluctuating over a range, MC > 100	No	

Figure 2.25: Window 1.5 *Moisture Content and Fabrication Quality - Members* (ASD)

The default values of the factors k_5, d_2 and k_4 can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.12, page 15).



When a component is complying with a SANS Standard, you can apply the modification factor for *Fabrication Quality*, k_4 in column C.

2.8 Moisture Content and Fabrication Quality - Sets of Members (ASD)

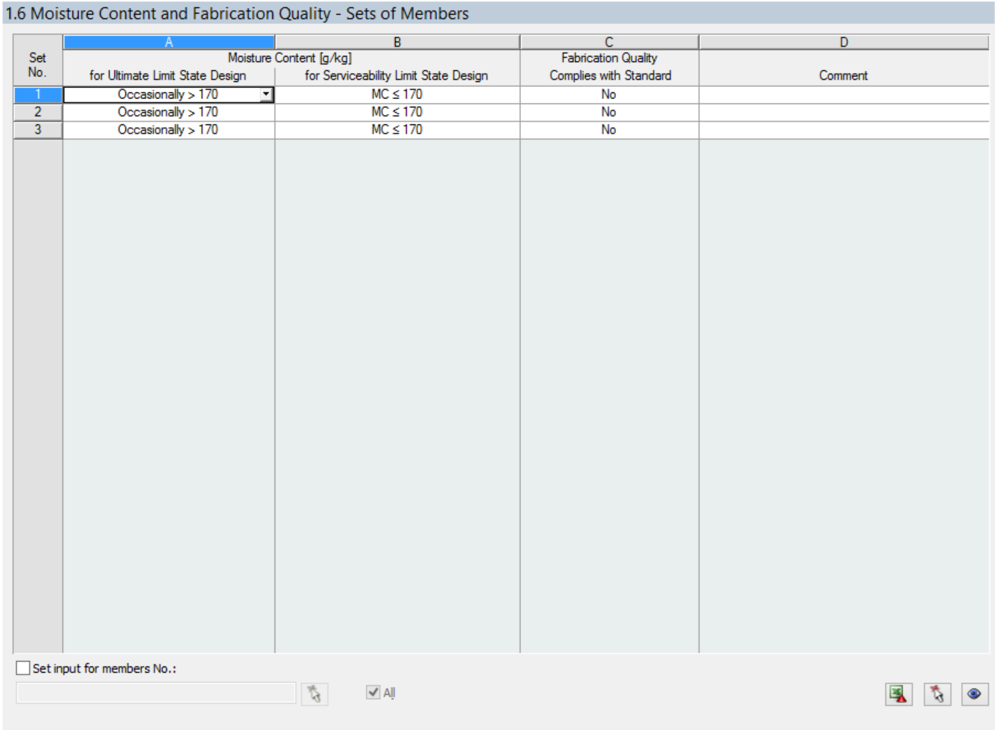


Figure 2.26: Window 1.6 *Moisture Content and Fabrication Quality - Sets of Members* (ASD)

The concept of this window is similar to the one in the previous Window 1.*5 Moisture Content and Fabrication Quality - Members* window. In this window, you can assign the moisture content and the fabrication quality of the wood to each set of members.

2.9 Effective Lengths - Members

Details...

The appearance of the window depends on whether the stability analysis is carried out according to the equivalent member method or according to second-order analysis. You can specify that method in the *Stability* tab of the *Details* dialog box (see Figure 3.2, page 41). The following description refers to the equivalent member default method. For that, the parameters of buckling and lateral-torsional buckling must be defined.

If the stability analysis is deactivated in the *Stability* tab of the *Details* dialog box, Window 1.7 is not shown.

The window consists of two parts. The table in the upper part provides summarized information about the factors for the lengths of buckling and lateral-torsional buckling as well as the equivalent member lengths of the members to be designed. The effective lengths defined in RFEM are preset. In the *Settings* section, you can see further information about the member whose row is selected in the upper section.

Click [↖] to select a member graphically and to show its row.

Changes can be made in the table as well as in the *Settings* tree.

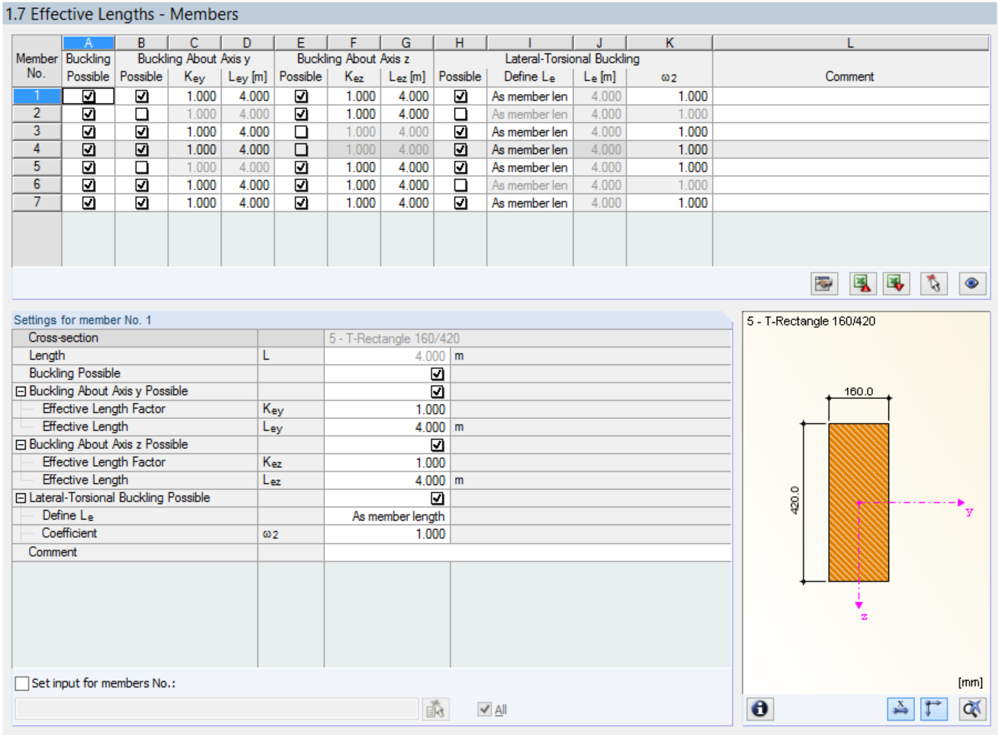


Figure 2.27: Window 1.7 *Effective Lengths - Members (LSD)* for equivalent member method

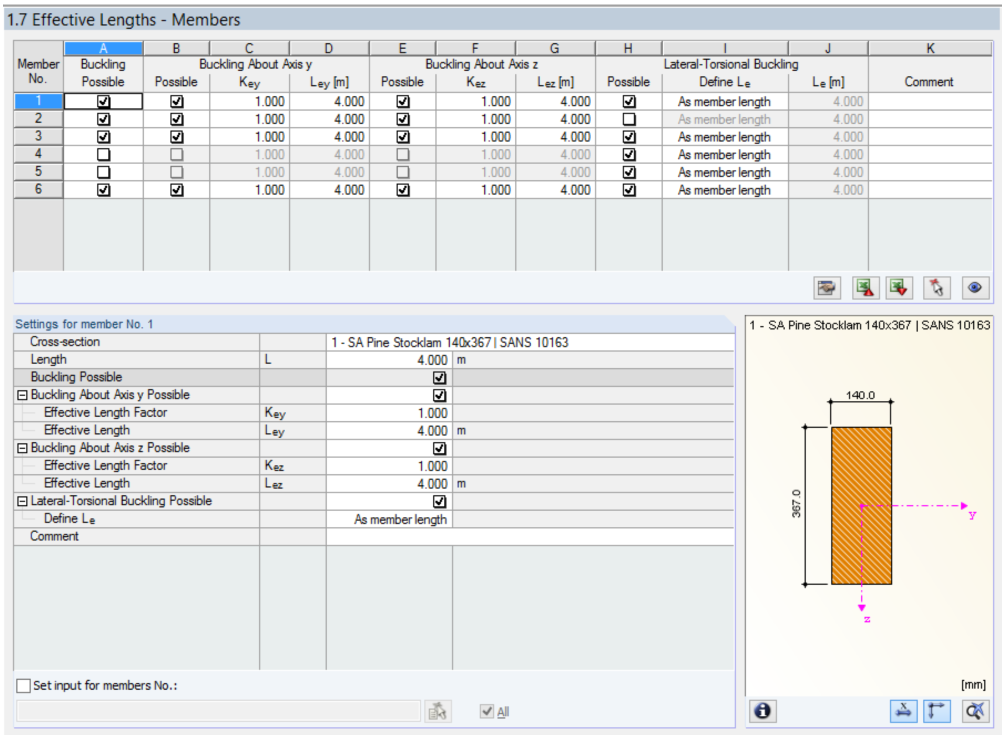


Figure 2.28: Window 1.7 *Effective Lengths - Members (ASD)* for equivalent member method

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 text box (see figure above).

The *Settings* tree manages the following parameters:

- *Cross-section*
- *Length* (actual length of the member)
- *Buckling Possible* (corresponds to column A)
- *Buckling About Axis y Possible* (corresponds to columns B to D)
- *Buckling About Axis z Possible* (corresponds to columns E to G)
- *Lateral-Torsional Buckling Possible* (corresponds to columns H to K)

In this table, you can specify for the currently selected member whether to carry out a buckling or a lateral-torsional buckling analysis. In addition to this, you can adjust the *Effective Length Factor* for the respective lengths. 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 of the window.

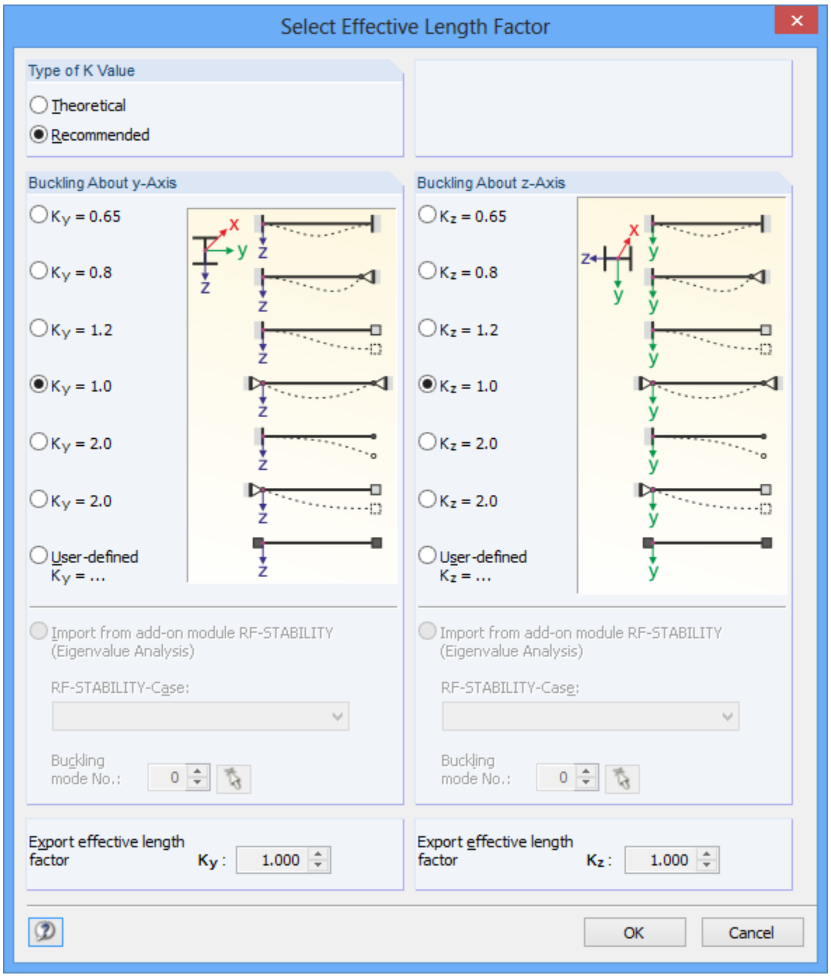


Figure 2.29: Dialog box *Select Effective Length Factor* for LSD method

For each direction, you can select one of the buckling modes (theoretical and recommended values of buckling length factors according to [1] Table B.1).

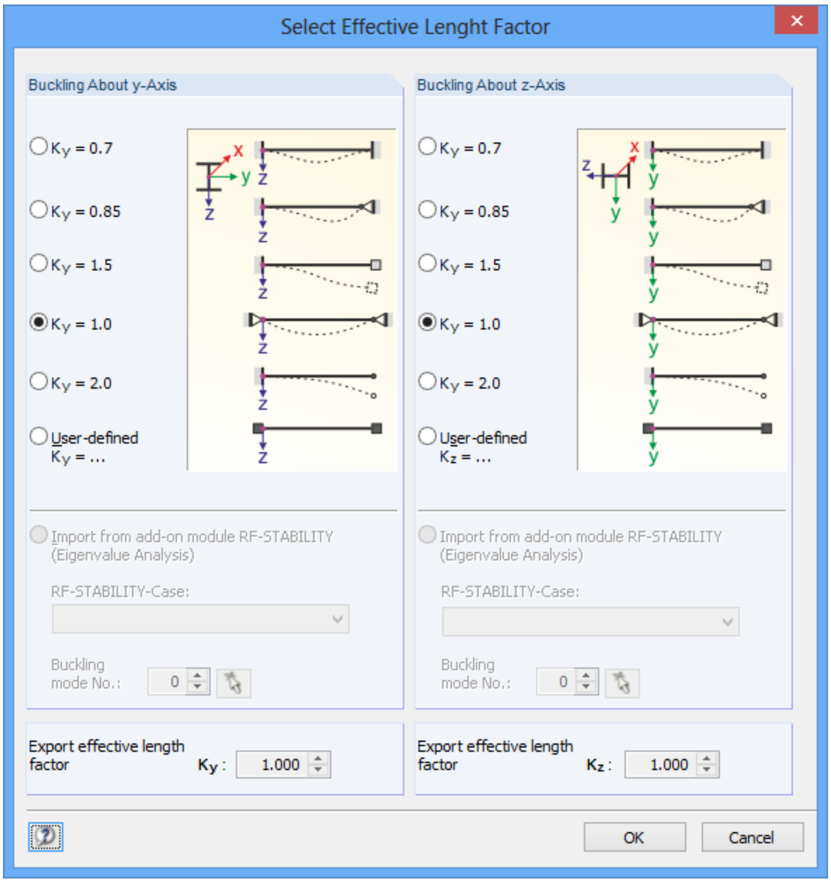


Figure 2.30: Dialog box *Select Effective Length Factor* for ASD method

If an eigenvalue analysis was carried out in the add-on module RF-STABILITY, you can also select a *Buckling mode* to determine the coefficient.

Buckling Possible

A stability analysis for flexural buckling and lateral-torsional buckling requires the ability of members to resist compressive forces. Therefore, members for which such resistance is not possible because of the member type (for example tension members, elastic foundations, rigid couplings) are excluded from design in the first place. 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 enable you to classify specific members as compression members or, alternatively, to exclude them from the stability analysis..

Buckling About Axis y / Buckling About 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, with axis y being the "major" and axis z the "minor" member axis. The buckling length factors K_{ey} and K_{ez} 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 Window 1.3 *Cross-Sections* (see Figure 2.17, page 20). To access the RFEM work window, click [View mode]. In the work window, you can display the local member axes by using the member's shortcut menu or the *Display* navigator.

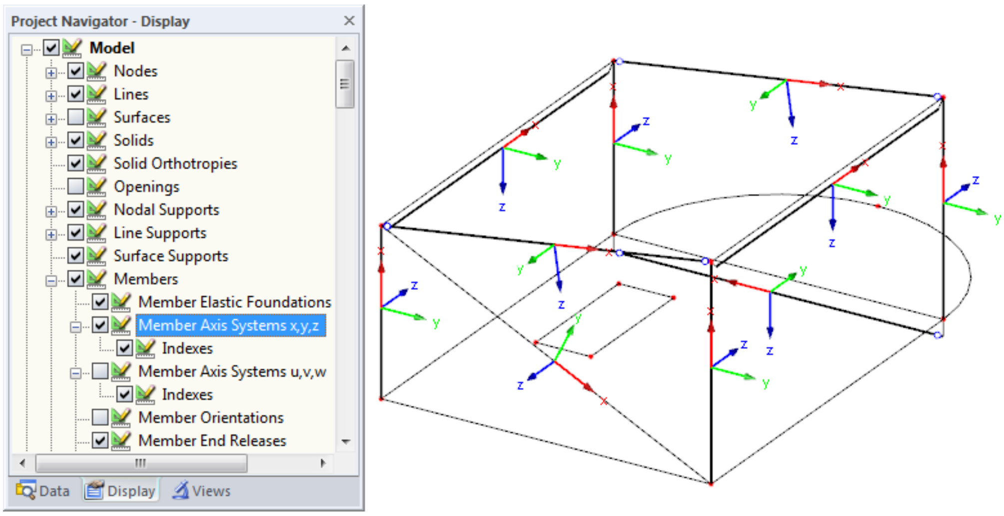


Figure 2.31: Selecting the member axis systems in the *Display* navigator of RFEM

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

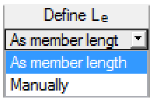
To specify the buckling lengths in the work window graphically, click [...]. This button becomes available when you click in a L_e text box.

When you define the effective length factor K_e, the program determines the effective length L_e by multiplying the member length L by this effective length factor. The text boxes for K_e and L_e are interactive.

Lateral-Torsional Buckling Possible

Table column H controls whether a lateral-torsional buckling analysis is to be carried out or not.

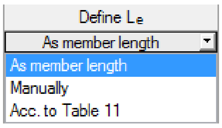
Define L_e (LSD)



The member lengths are preset in column I as equivalent member lengths relevant for lateral-torsional buckling. When you set the options manually in the list box, you can specify the length for lateral-torsional buckling L_e in column J. You can also define it graphically after click-ing [...] as the distance of the lateral supports. Thus, you can adjust the boundary conditions of a structural component if it consists of several members between the supports.

In column K, the coefficient for calculating the moment resistance of a laterally unsupported beam, ω_2, is indicated. By default it is preset to 1.0, but it can be adjusted if necessary.

Define L_e (ASD)



The definition of L_e is similar to the LSD method. In addition, it is possible to define lateral-torsional buckling length in accordance with [2] Table 11. A new dialog is opened in which you can select the effective length according to the loading conditions (see Figure 2.32).

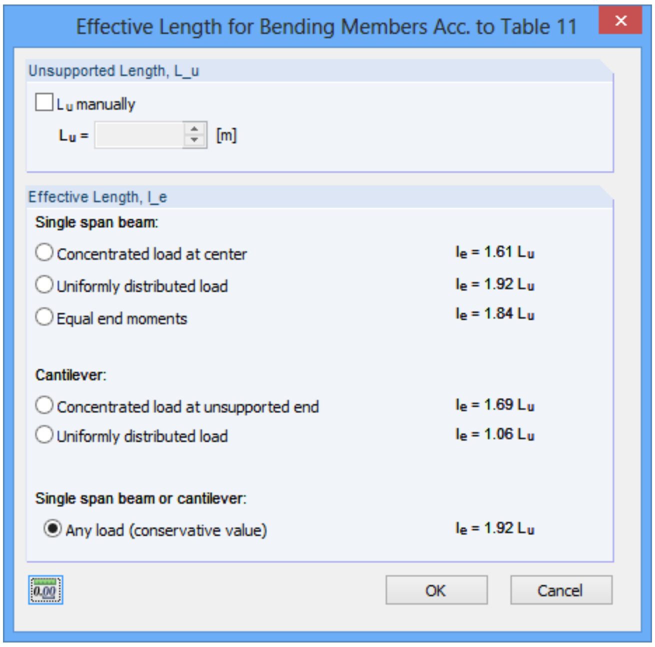


Figure 2.32: Dialog box *Effective Length for Bending Members Acc. to Table 11*

Below the *Settings* table, you find the *Set inputs for members No.* check box. If selected, the settings entered afterward will be applied to the selected or to *All* members. Members can be selected by typing the member number or by selecting them graphically using the [↖] button. This option is useful when you want to assign the same boundary conditions to several members. Please note that already defined settings cannot be changed subsequently with this function.

Comment

In the last table column, you can enter you own comments for each member to describe, for example, the selected equivalent member lengths.

2.10 Effective Lengths - Sets of Members

This window is only available if one or more sets of members have been selected in Window 1.1 *General Data*. Additionally, the stability check must have been activated in the dialog box *Details*, tab *Stability* (see Figure 3.2, page 41).

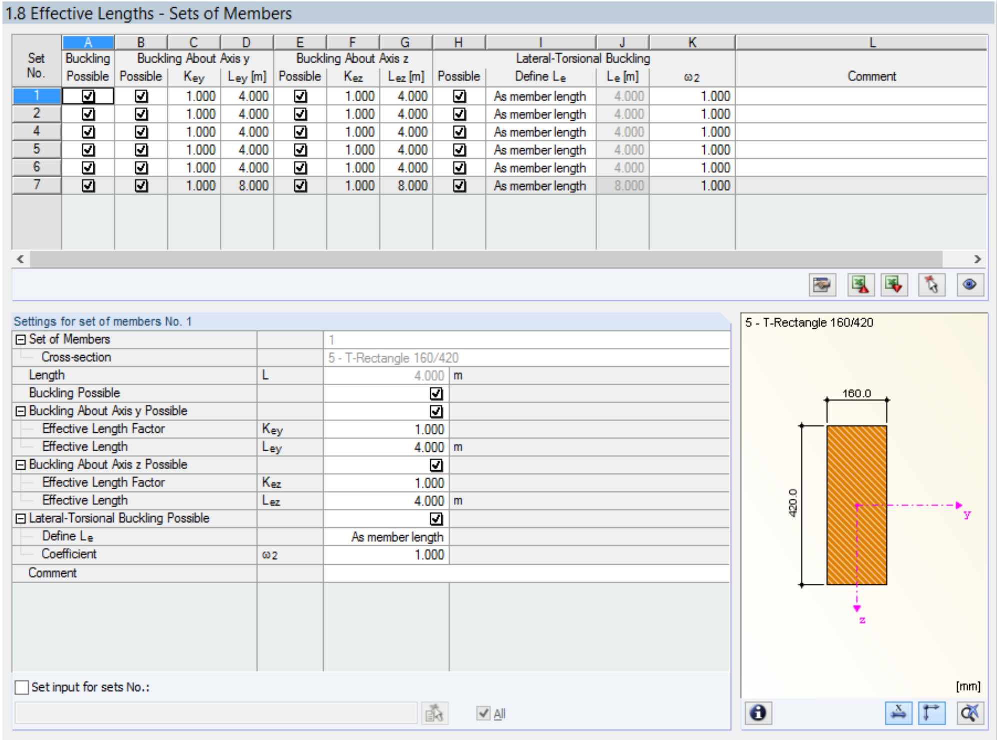


Figure 2.33: Window 1.8 *Effective Lengths - Sets of Members* (LSD)

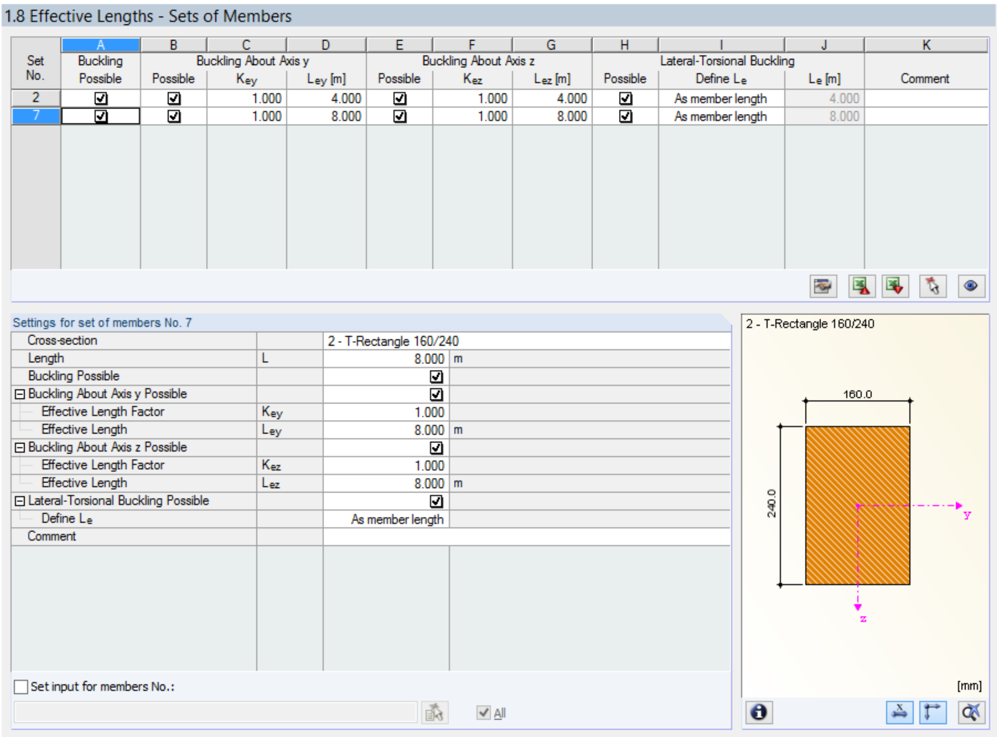
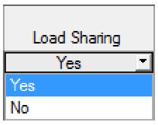
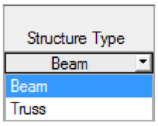


Figure 2.34: Window 1.8 *Effective Lengths - Sets of members* (ASD) for equivalent member method

The concept of this window is similar to the one in the previous Window 1.7 *Effective Lengths - Members*. In this window, you can enter the effective lengths for buckling as well as for lateral-torsional buckling as described in Chapter 2.9. They determine the boundary conditions of the entire set of members which is to be treated as an equivalent member.

2.11 Additional Design Parameters - Members





This window allows you to allocate the load sharing factor, γ_{m2} (LSD) or k_2 (ASD), to each member. This factor depends on the interaction between members in the model. By clicking the [▼] button in column A, the *Yes* option (according to [1] Chapter 13.1.2.2 or Chapter 6.3.4) can be selected.

You can define in column B whether the member is beam or truss, which controls the structure type factor, γ_{m3} (LSD) or k_3 (ASD).

By default, the program assigns *No* load sharing factor and the *Beam* structure type factor. The factors can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.11 and Figure 2.12, page 15Figure).

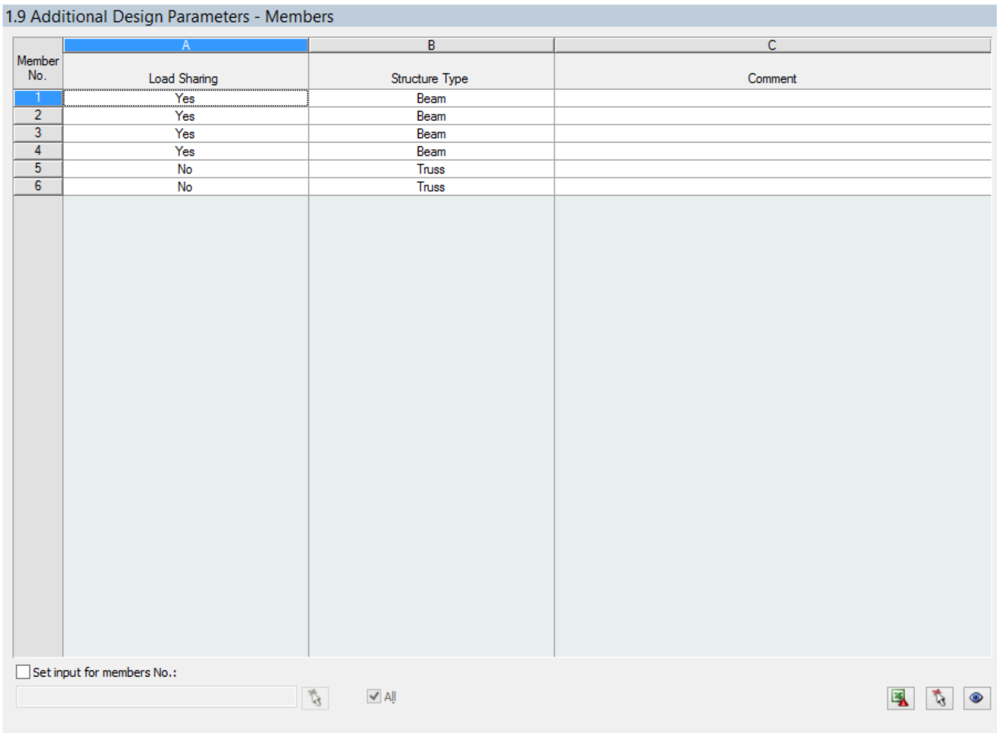
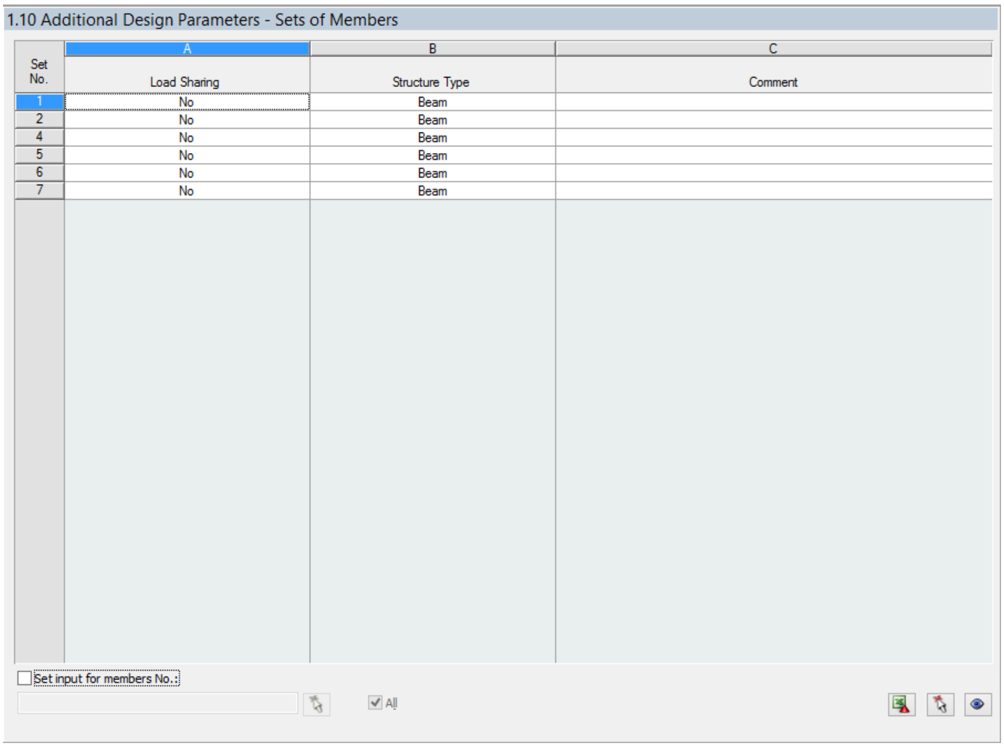


Figure 2.35: Window 1.9 *Additional Design Parameters*

The *Set input for members No.* check box below the table enables you to assign the settings to selected or *All* members (see Chapter 2.7 and Chapter 2.9).

2.12 Additional Design Parameters - Sets of Members

This window manages the additional design parameters of sets of members. It is displayed when you have selected one or more sets of members in Window 1.1 *General Data*.



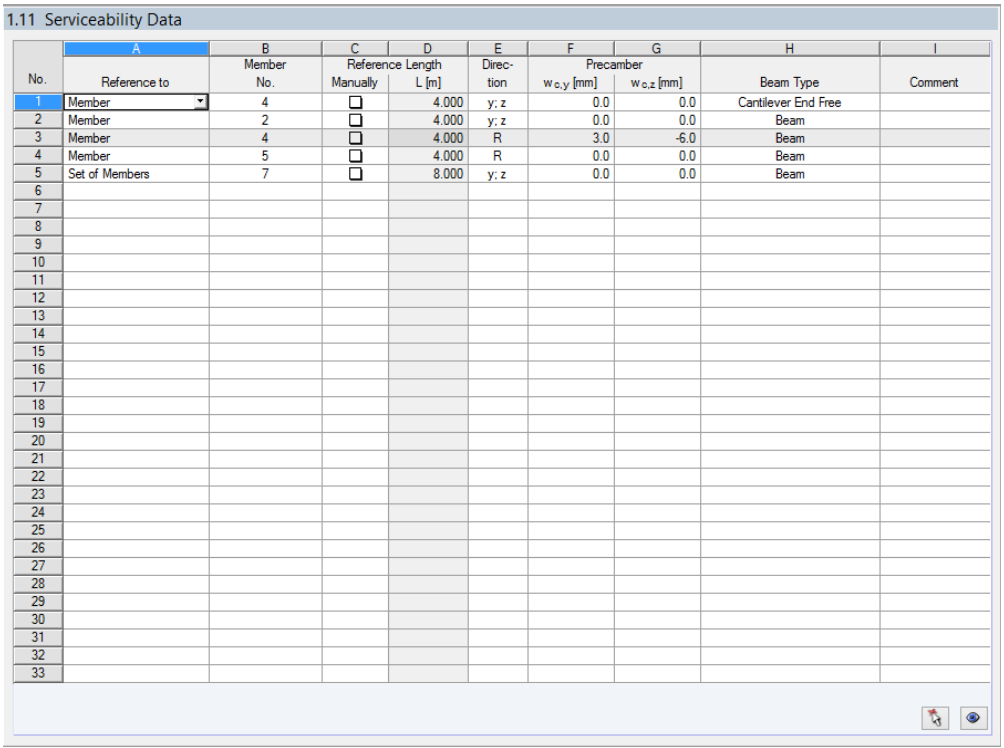
Set No.	A Load Sharing	B Structure Type	C Comment
1	No	Beam	
2	No	Beam	
4	No	Beam	
5	No	Beam	
6	No	Beam	
7	No	Beam	

Figure 2.36: Window 1.10 *Additional Design Parameters - Sets of Members*

The set-up of this window is similar to the one of the previous window (see Chapter 2.11).

2.13 Serviceability Data

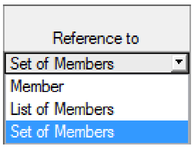
This input window controls several settings for the serviceability limit state design. It is only available if you have set the relevant entries in the *Serviceability Limit State* tab of Window 1.1 *General Data* (see Chapter 2.1.2, page 11).



1.11 Serviceability Data

No.	A Reference to	B Member No.	C Reference Length Manually	D Reference Length L [m]	E Direc-tion	F Precamber $w_{c,y}$ [mm]	G Precamber $w_{c,z}$ [mm]	H Beam Type	I Comment
1	Member	4	☐	4.000	y; z	0.0	0.0	Cantilever End Free	
2	Member	2	☐	4.000	y; z	0.0	0.0	Beam	
3	Member	4	☐	4.000	R	3.0	-6.0	Beam	
4	Member	5	☐	4.000	R	0.0	0.0	Beam	
5	Set of Members	7	☐	8.000	y; z	0.0	0.0	Beam	
6									
7									
8									
9									
10									
11									
12									
13									
14									
15									
16									
17									
18									
19									
20									
21									
22									
23									
24									
25									
26									
27									
28									
29									
30									
31									
32									
33									

Figure 2.37: Window 1.11 *Serviceability Data*



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

In table column B, you enter the numbers of the members or sets of members that you want to design. You can also click [...] to select them graphically in the RFEM work window. Then, the *Reference Length* appears in column D automatically. This column presets the lengths of the members, sets of members, or member lists. If required, you can adjust these values after selecting the *Manually* check box in column C.

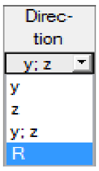
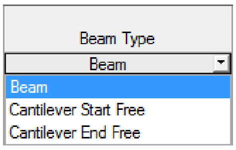


Table column E defines the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z and the resultant direction R.

In column F and G, you can allow for some *Precamber* $w_{c,y}$ and $w_{c,z}$..

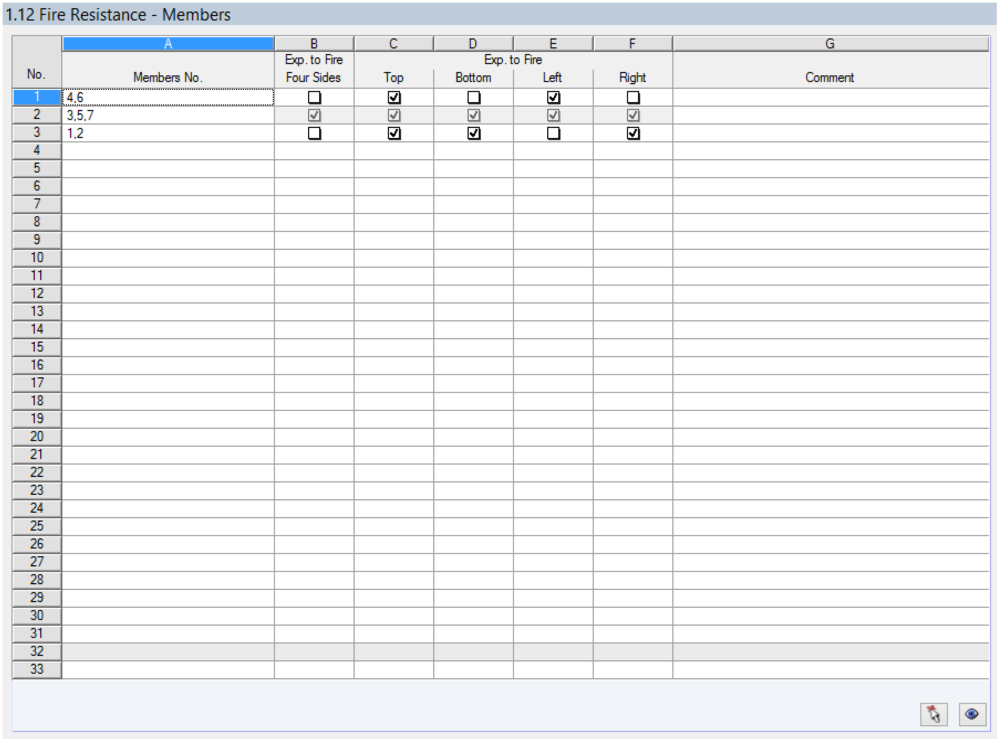


The *Beam Type* is of crucial importance for the correct application of limit deformations. Column H controls whether is a beam or cantilever and which ends is not supported.

The settings in the *Serviceability* tab of the *Details* dialog box decide whether the deformations are related to the undeformed initial model or to the shifted ends of members or sets of members (see Figure 3.3, page 42).

2.14 Fire Resistance - Members

This window manages the different fire resistance parameters. It is available when you have set the relevant entries in the *Fire Resistance* tab of Window 1.1 *General Data* (see Chapter 2.1.3, page 12).



1.12 Fire Resistance - Members

No.	A Members No.	B Exp. to Fire Four Sides	C Exp. to Fire Top	D Bottom	E Left	F Right	G Comment
1	4,6	☐	☑	☐	☑	☐	
2	3,5,7	☑	☑	☑	☑	☑	
3	1,2	☐	☑	☑	☐	☑	
4							
5							
6							
7							
8							
9							
10							
11							
12							
13							
14							
15							
16							
17							
18							
19							
20							
21							
22							
23							
24							
25							
26							
27							
28							
29							
30							
31							
32							
33							

Figure 2.38: Window 1.12 *Fire Resistance - Members*

...

Table column A contains the members that are to be taken into account for the fire resistance design. Click […] to graphically select the members in the RFEM work window.

In column B you specify if there is an *Exposure to Fire on Four Sides*. If the cross-section is not exposed to fire on all sides, clear the selection of the check box. Thus, the following columns become available in which you can specify the sides that are exposed to fire. The ideal remaining cross-section is then computed with those assumptions.

Details...

The general parameters of the fire resistance analysis are managed in the *Details* dialog box, tab *Fire Resistance Design* (see Figure 3.4, page 43).

2.15 Fire Resistance - Sets of Members

This window manages the fire resistance parameters of sets of members. It is displayed when you have selected one or more sets of members in Window 1.1 *General Data* and when you have allocated specific load cases or combinations in the *Fire Resistance* tab of that window.

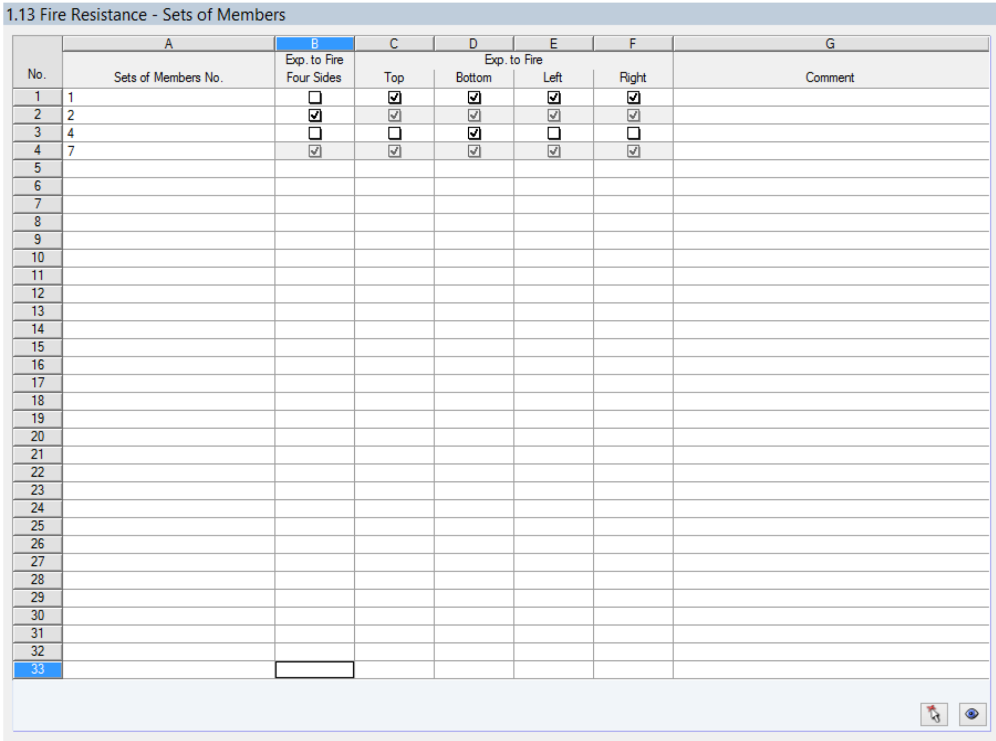


Figure 2.39: Window 1.13 *Fire Resistance - Set of Members*

The set-up of this window is similar to the one of the previous window (see Chapter 2.14). Here you can define the sides of the cross-section that are exposed to fire for the relevant set of members.

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*
- *Fire Resistance*
- *Other*

3.1.1 Resistance

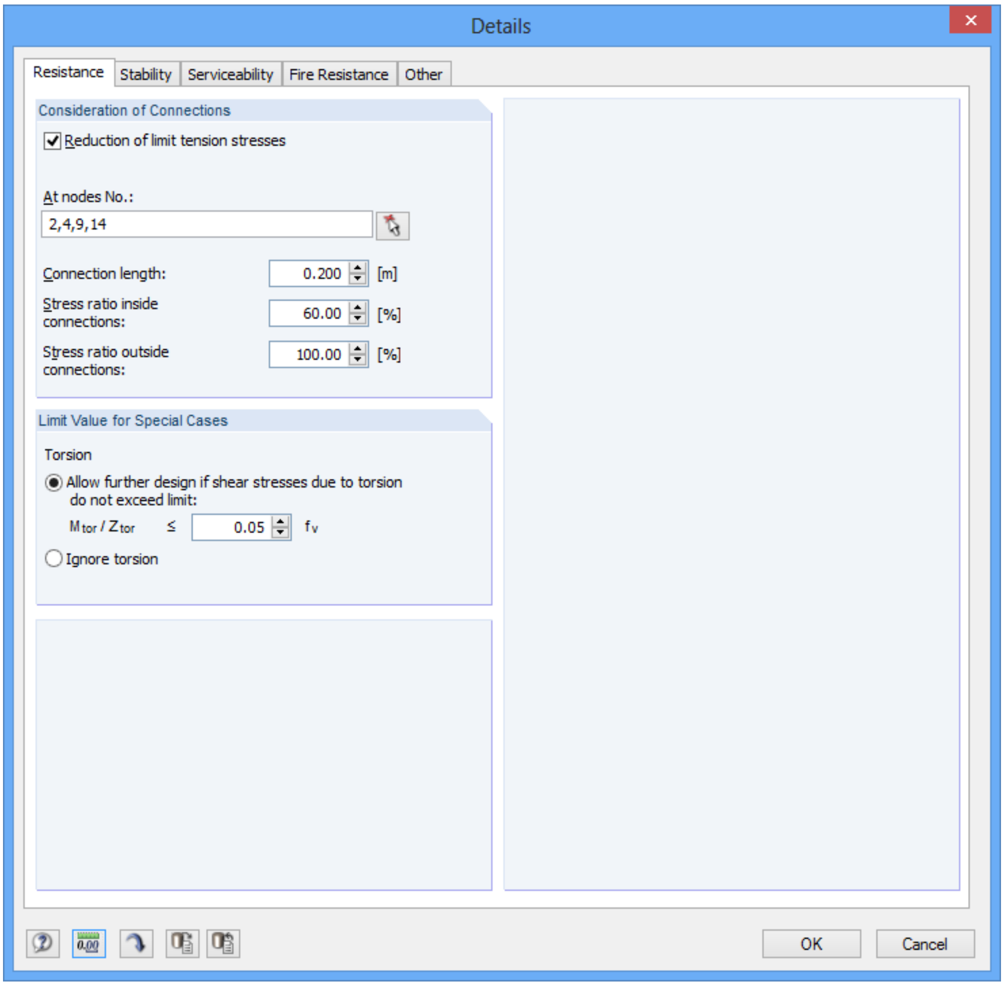


Figure 3.1: Dialog box *Details*, tab *Resistance*

Consideration of Connections

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

The numbers of the relevant *nodes* can be entered manually or selected graphically by clicking the [↖] button.

The *Connection length* defines the zone on the member where reduced stresses are considered. In the text box below, enter the allowable stress ratio for *Inside connections* in percent. If required, you can also change the maximum ratio for *Outside connections* of the connection zone.

Limit Value for Special Cases

Torsion design is not specified in SANS 10163. It is possible to ignore shear stresses due to torsion if a user-defined ratio of the torsional shear resistance is not exceeded (default: 5 %).

If the limit is exceeded, a note appears in the result window. This limit setting is not part of the Standard [1] or [2][1]. Changing the limit is the responsibility of the user. It is also possible to completely *Ignore torsion*.

3.1.2 Stability

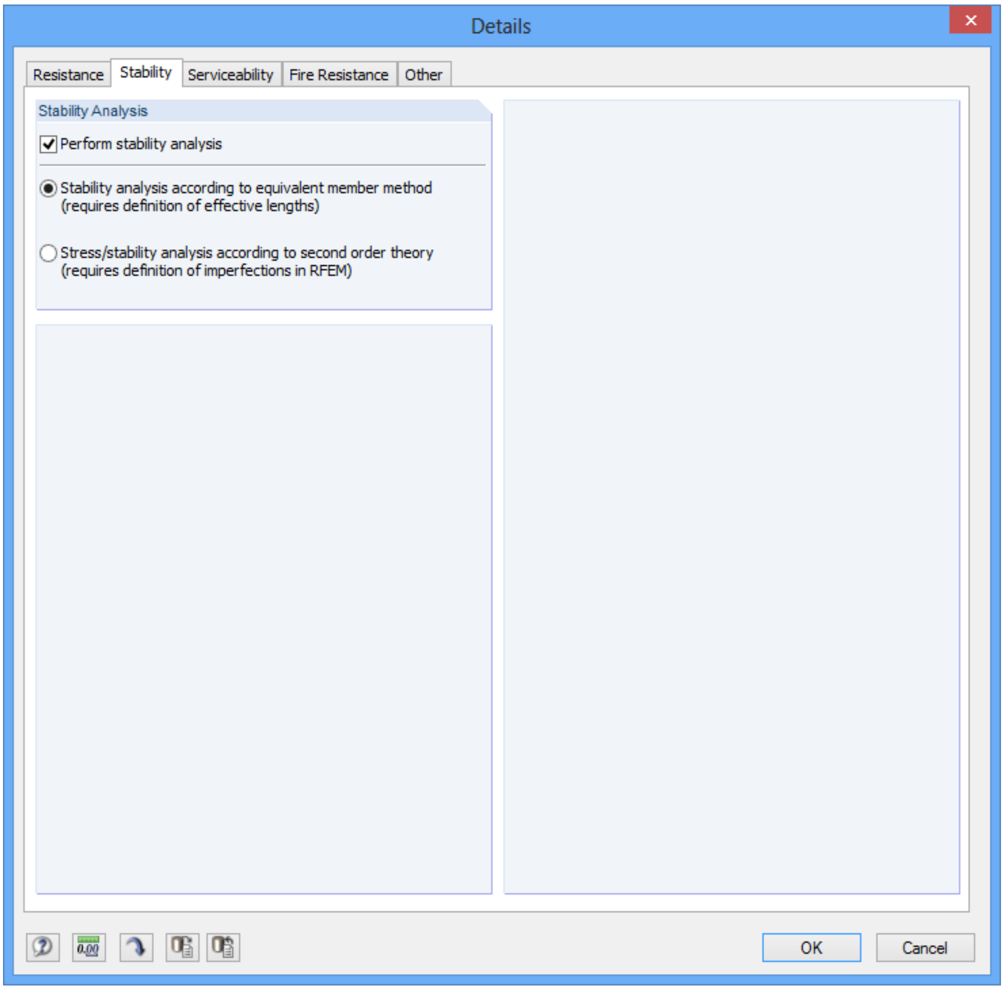
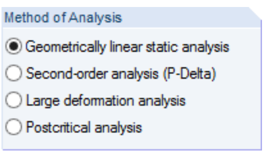


Figure 3.2: Dialog box *Details*, tab *Stability*



Equivalent member method:
Specifying method of analysis in RFEM

The *Check stability* check box controls whether to run, in addition the cross-section design, a stability analysis. If you clear the check box, the input Windows 1.7 and 1.8 will not be shown.

The *equivalent member method* uses the internal forces determined in RFEM. In this method, make sure that the **Geometrically linear static analysis** has been set (the default setting is the 2nd order analysis). When you perform the stability analysis according to the equivalent member method, the effective lengths of the members and sets of members subject to compression or compression and bending must be specified in Windows 1.7 and 1.8.

If the bearing capacity of the model is significantly affected by its deformations, we recommend selecting a calculation according to the *second order theory*. This approach additionally requires the definition of imperfections in RFEM and their consideration for the load combinations. The flexural buckling analysis is carried out during the calculation of the load combinations in RFEM.

The lateral-torsional buckling design must also be carried out for second order calculations. Thus, the lateral-torsional buckling lengths of members or sets of members are to be specified in Windows 1.7 or 1.8 *Effective Lengths* manually. In this way, we can make sure that the lateral-torsional buckling analysis is performed with the appropriate factors (for example 1.0).

3.1.3 Serviceability

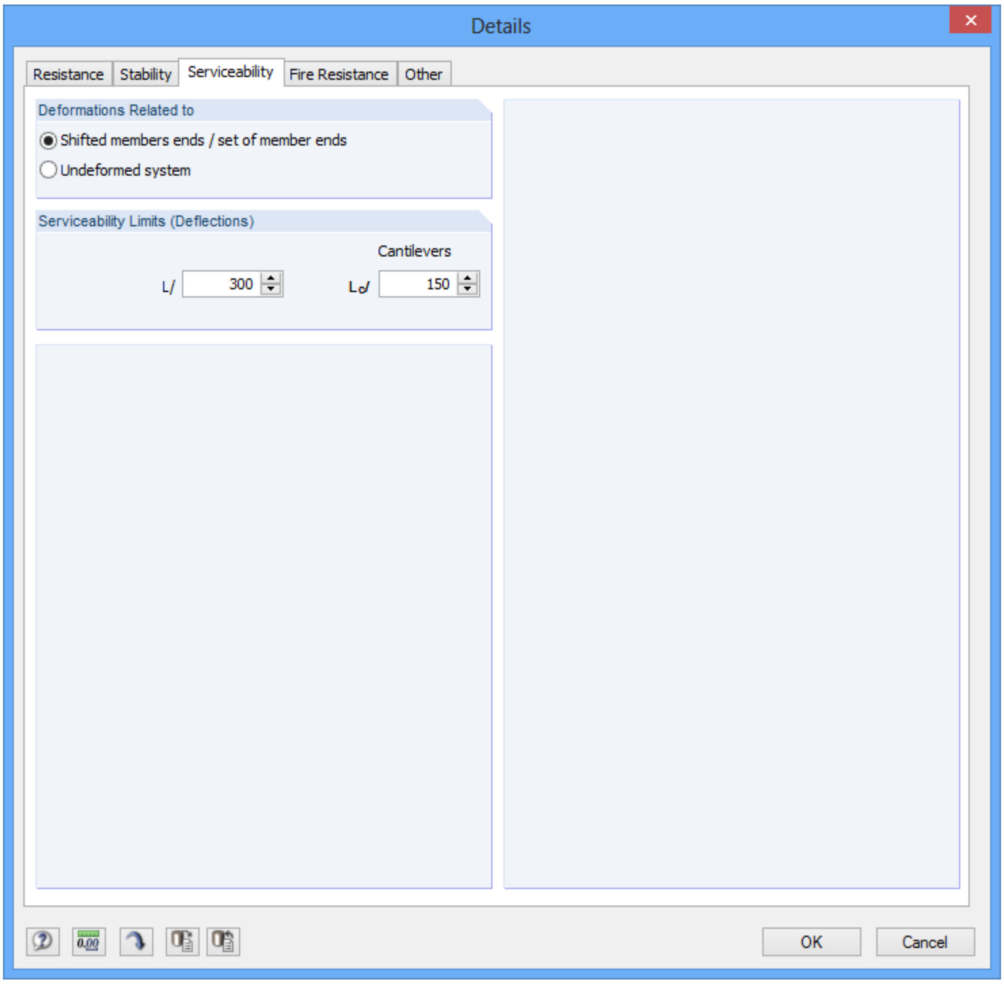


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformations Related to

The option fields control whether the maximum deformations are related to the shifted ends of members or sets of members (connection line between start and end nodes of the deformed system) or to the undeformed initial system. As a rule, the deformations are to be checked relative to the displacements in the entire structural system.

Serviceability Limits (Deflections)

In this tab, it is possible to change the allowable deflection for the serviceability limit state design if the default values *L/300* (for inner span) and L_c*/150* (for cantilever) are not appropriate.

3.1.4 Fire Resistance

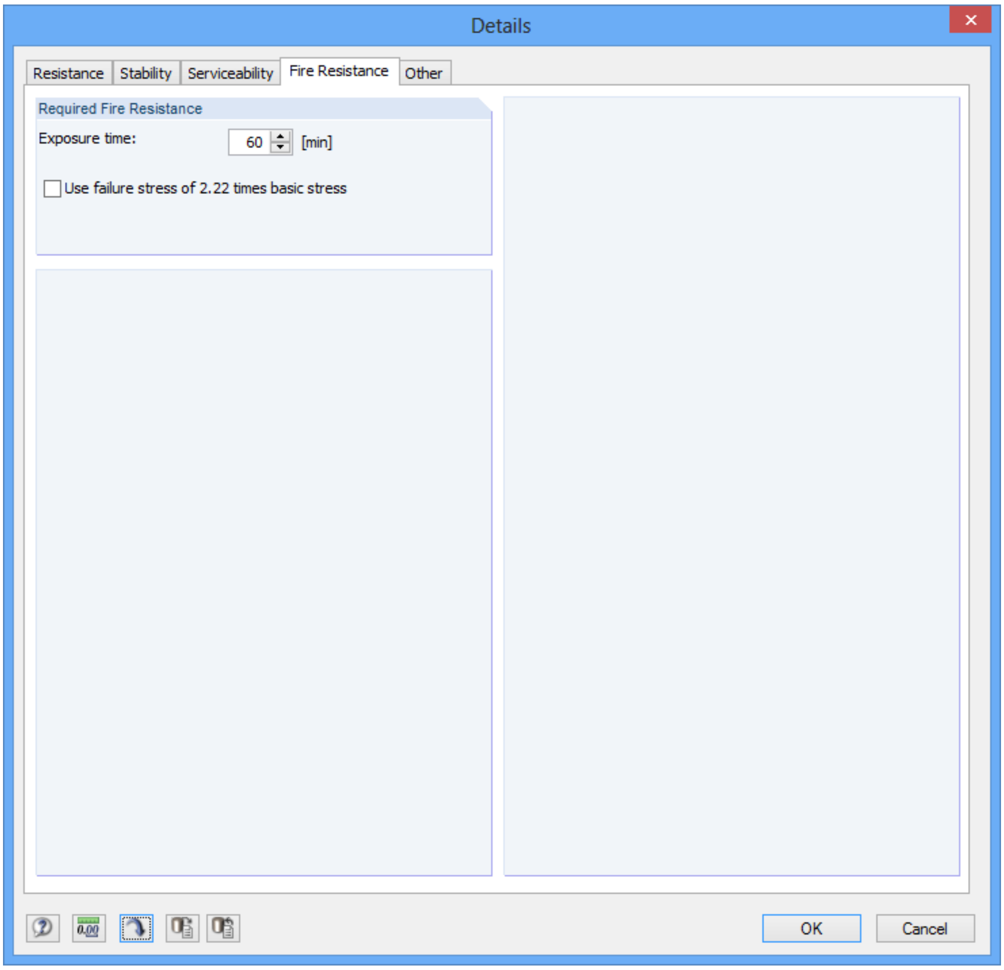


Figure 3.4: Dialog box *Details*, tab *Fire Resistance (ASD)*

The *Exposure time* can be defined by specifying the duration of the fire in the text box.

In [2] for fire design, the grade stresses of members are used by default. If required, it is possible to analyze the fire resistance with the characteristic stresses by selecting the check box *Use failure stress of 2.22 times basic stress* (characteristic stress is equal to 2.22 times of grade stress).

Additionally, some standard-specific parameters that are significant for the fire resistance design can be set in the *Standard* dialog box (see Figure 2.11 or Figure 2.12, page 15).

3.1.5 Other

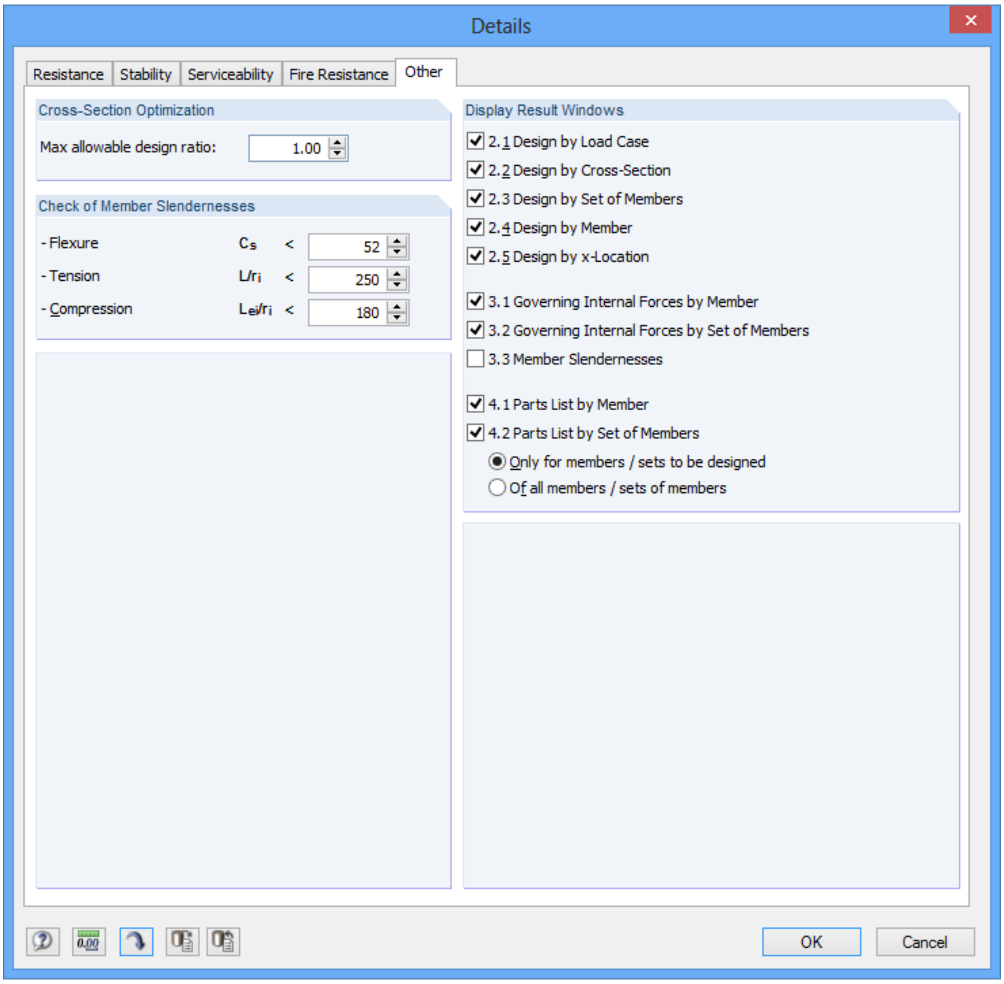


Figure 3.5: Dialog box *Details*, tab *Other*

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 Slenderness

In the three text boxes, you can specify the limit values of the member slendernesses. You can define the ratios separately for members with *Flexure* (C_S), for members with *Tension* (L/r_i), and for members with *Compression* (L_{ei}/r_i).

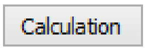
The limit values are compared to the real member slendernesses in Window 3.3. This window is available after the calculation (see Chapter 4.8, page 52) if the corresponding check box is selected in the *Display Result Windows* dialog box section.

Display Result Windows

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

The 3.3 *Member Slendernesses* Window is inactive by default.

3.2 Start Calculation



To start the calculation, click the [Calculation] button which is available in all input windows of the RF-TIMBER SANS add-on module.

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

You can also start the calculation in the user interface of RFEM: In the dialog box *To Calculate* (menu *Calculate* → *To Calculate*), design cases of the add-on modules are listed like load cases and load combinations.

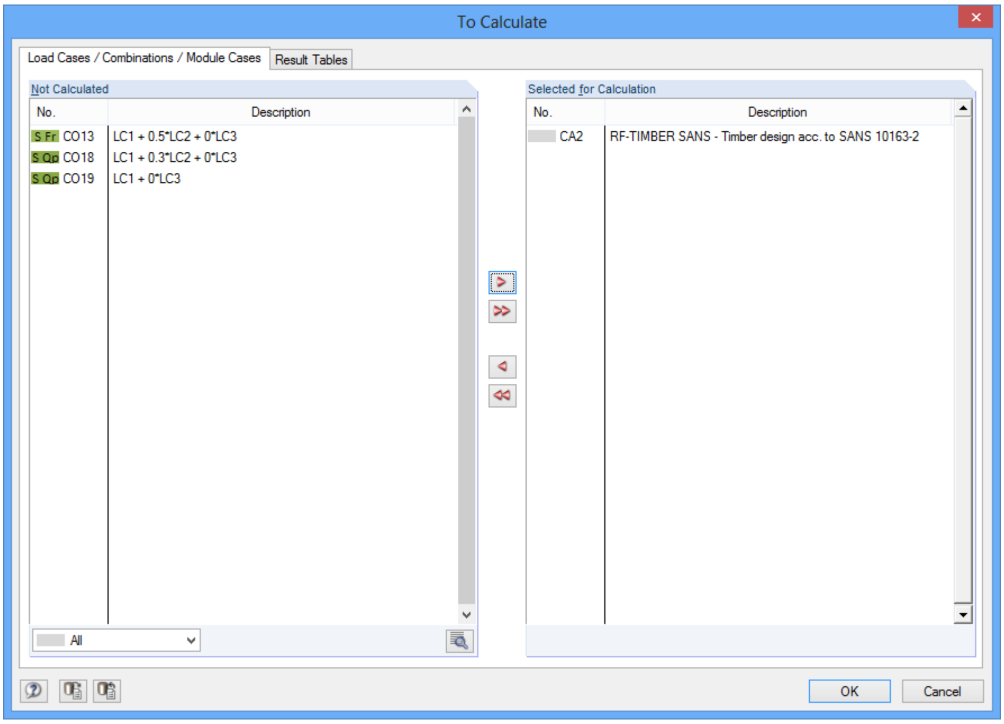
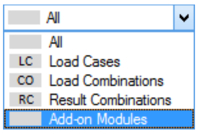


Figure 3.6: Dialog box *To Calculate*



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

To transfer the selected RF-TIMBER SANS 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 SANS design case in the toolbar list, and then click [Show Results].

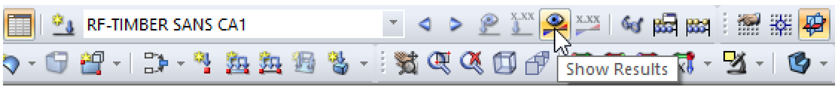


Figure 3.7: Direct calculation of a RF-TIMBER SANS design case in RFEM

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

4. Results

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

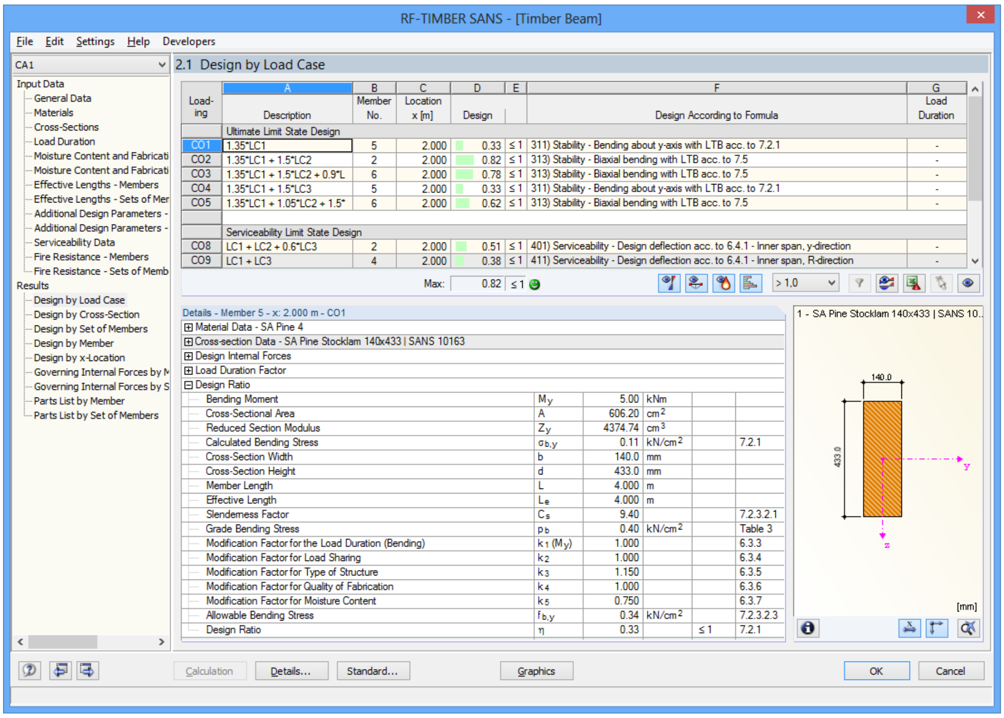
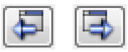


Figure 4.1: Results window with designs and intermediate values

The designs are shown in the results 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 results 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.

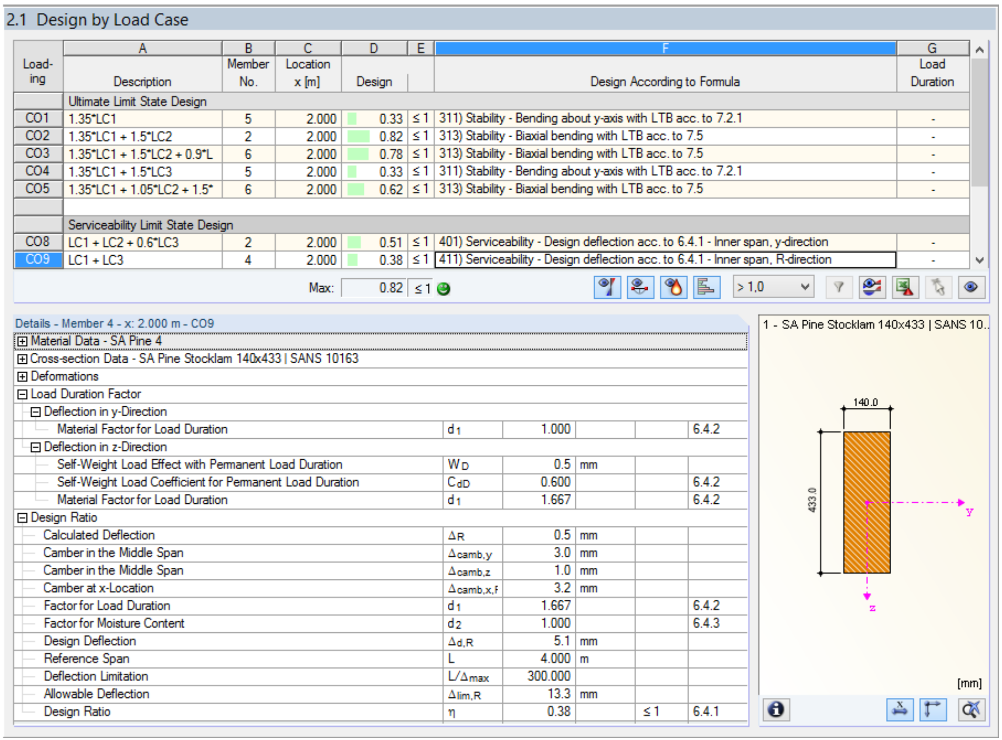
To save the results, click [OK]. You exit RF-TIMBER SANS and return to the main program.

Chapter 4 *Results* describes the different results windows one by one. Evaluating and checking results is described in Chapter 5 *Evaluation of Results* starting on page 51.

4.1 Design by Load Case

The upper part of the window provides a summary, sorted by load cases, load combinations, and result combinations of the governing designs. Furthermore, the list is divided in *Ultimate Limit State Design, Serviceability Limit State Design* and *Fire Resistance Design* results.

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



2.1 Design by Load Case

Load-ing	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	1.35*LC1	5	2.000	0.33	≤ 1	311) Stability - Bending about y-axis with LTB acc. to 7.2.1	-
CO2	1.35*LC1 + 1.5*LC2	2	2.000	0.82	≤ 1	313) Stability - Biaxial bending with LTB acc. to 7.5	-
CO3	1.35*LC1 + 1.5*LC2 + 0.9*L	6	2.000	0.78	≤ 1	313) Stability - Biaxial bending with LTB acc. to 7.5	-
CO4	1.35*LC1 + 1.5*LC3	5	2.000	0.33	≤ 1	311) Stability - Bending about y-axis with LTB acc. to 7.2.1	-
CO5	1.35*LC1 + 1.05*LC2 + 1.5*	6	2.000	0.62	≤ 1	313) Stability - Biaxial bending with LTB acc. to 7.5	-
	Serviceability Limit State Design						
CO8	LC1 + LC2 + 0.6*LC3	2	2.000	0.51	≤ 1	401) Serviceability - Design deflection acc. to 6.4.1 - Inner span, y-direction	-
CO9	LC1 + LC3	4	2.000	0.38	≤ 1	411) Serviceability - Design deflection acc. to 6.4.1 - Inner span, R-direction	-

Max: 0.82 ≤ 1

Details - Member 4 - x: 2.000 m - CO9

⊞ Material Data - SA Pine 4					
⊞ Cross-section Data - SA Pine Stocklam 140x433 | SANS 10163					
⊞ Deformations					
⊟ Load Duration Factor					
⊟ Deflection in y-Direction					
Material Factor for Load Duration	d_1	1.000			6.4.2
⊟ Deflection in z-Direction					
Self-Weight Load Effect with Permanent Load Duration	W_D	0.5	mm		
Self-Weight Load Coefficient for Permanent Load Duration	C_{dD}	0.600			6.4.2
Material Factor for Load Duration	d_1	1.667			6.4.2
⊟ Design Ratio					
Calculated Deflection	Δ_R	0.5	mm		
Camber in the Middle Span	$\Delta_{camb,y}$	3.0	mm		
Camber in the Middle Span	$\Delta_{camb,z}$	1.0	mm		
Camber at x-Location	$\Delta_{camb,x,F}$	3.2	mm		
Factor for Load Duration	d_1	1.667			6.4.2
Factor for Moisture Content	d_2	1.000			6.4.3
Design Deflection	$\Delta_{d,R}$	5.1	mm		
Reference Span	L	4.000	m		
Deflection Limitation	L/Δ_{max}	300.000			
Allowable Deflection	$\Delta_{lim,R}$	13.3	mm		
Design Ratio	η	0.38		≤ 1	6.4.1

Figure 4.2: Window 2.1 *Design by Load Case*

Description

This column shows the descriptions of the load cases, load and result combinations used for the design.

Member No.

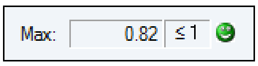
This column shows the number of the member that bears the maximum design ratio of every designed loading.

Location x

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

- Start and end node
- Division points according to possibly defined member division (see RFEM table 1.16)
- Member division according to specification for member results (RFEM dialog box *Calculation parameters*, tab *Global Calculation Parameters*)
- Extreme values of internal forces

Design



Columns D and E show the design conditions according to [1]or [2].

The lengths of the colored bars represent the respective utilizations.

Design According to Formula

This column lists the equations of the Standard by which the designs have been performed.

Load Duration

In column G, the load duration classes as defined in Window 1.4 are listed for load cases (see Chapter 2.4, page 24).

4.2 Design by Cross-Section

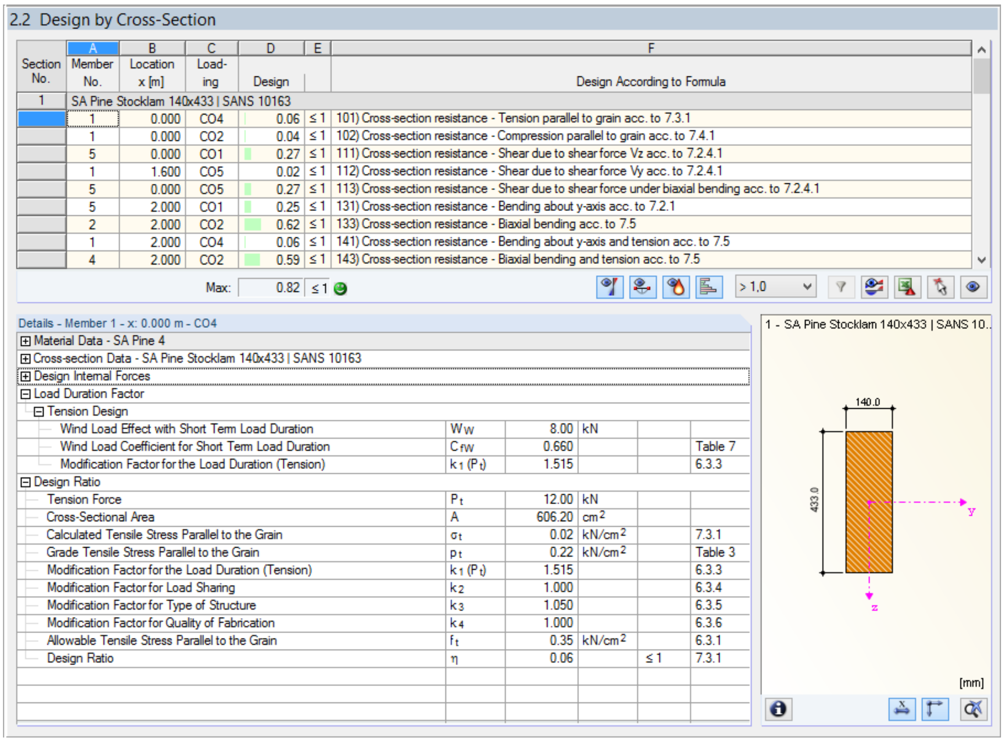


Figure 4.3: Window 2.2 *Design by Cross-Section*

This window lists the maximum ratios of all members and actions selected for design, sorted by cross-sections.

The results are sorted by cross-section design, stability design, serviceability limit state design and fire resistance design.

4.3 Design by Set of Members

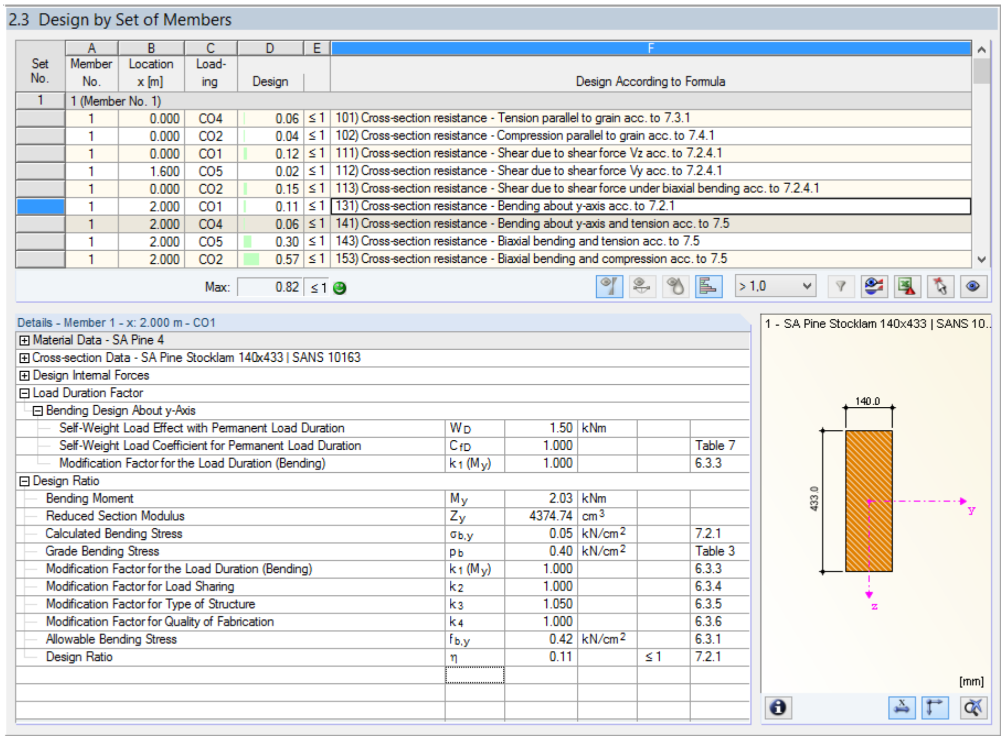


Figure 4.4: Window 2.3 *Design by Set of Members*

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

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

The output by sets of members clearly presents the design for an entire structural group (for example a chord).

4.4 Design by Member

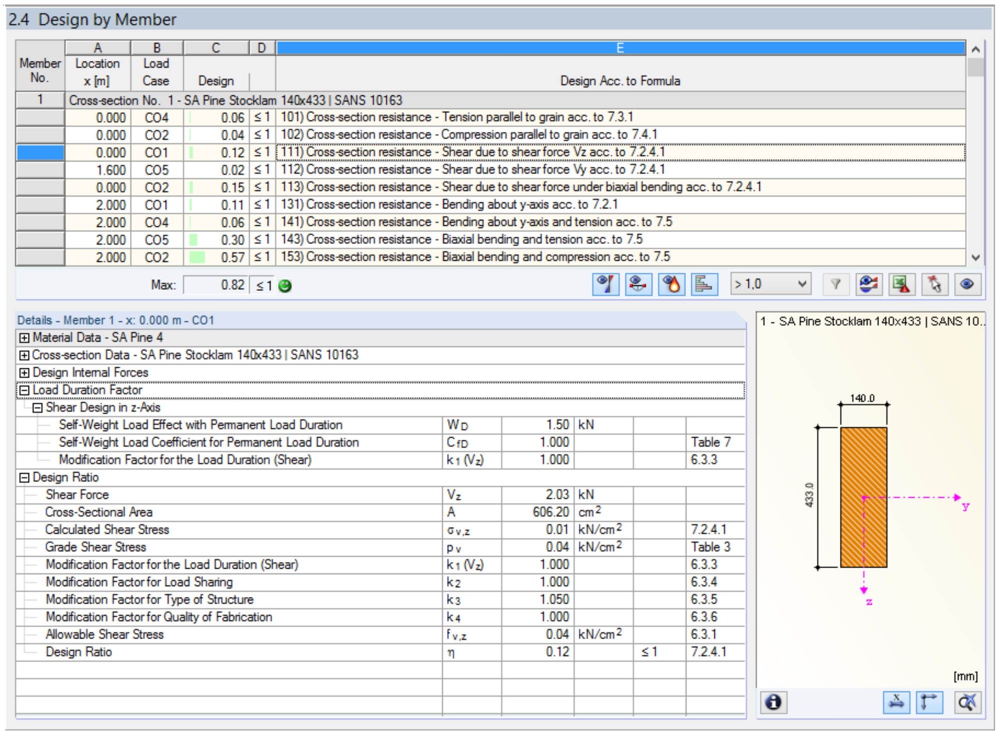


Figure 4.5: Window 2.4 *Design by Member*

This results window presents the maximum ratios for the individual designs sorted by member number. The columns are described in detail in Chapter 4.1 on page 47.

4.5 Design by x-Location

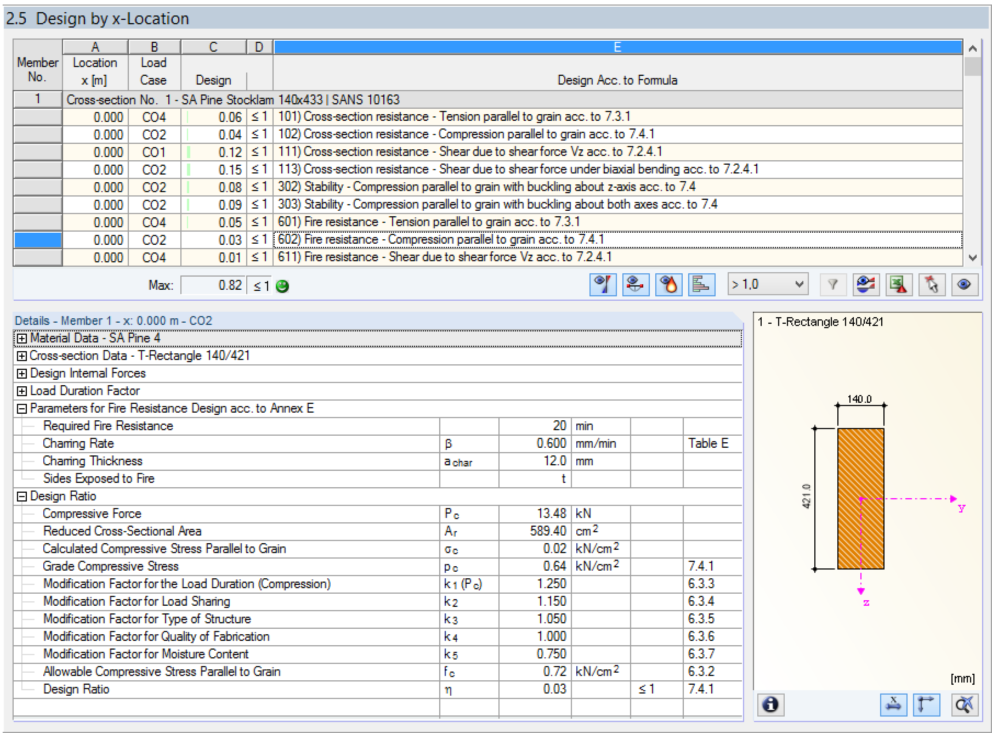
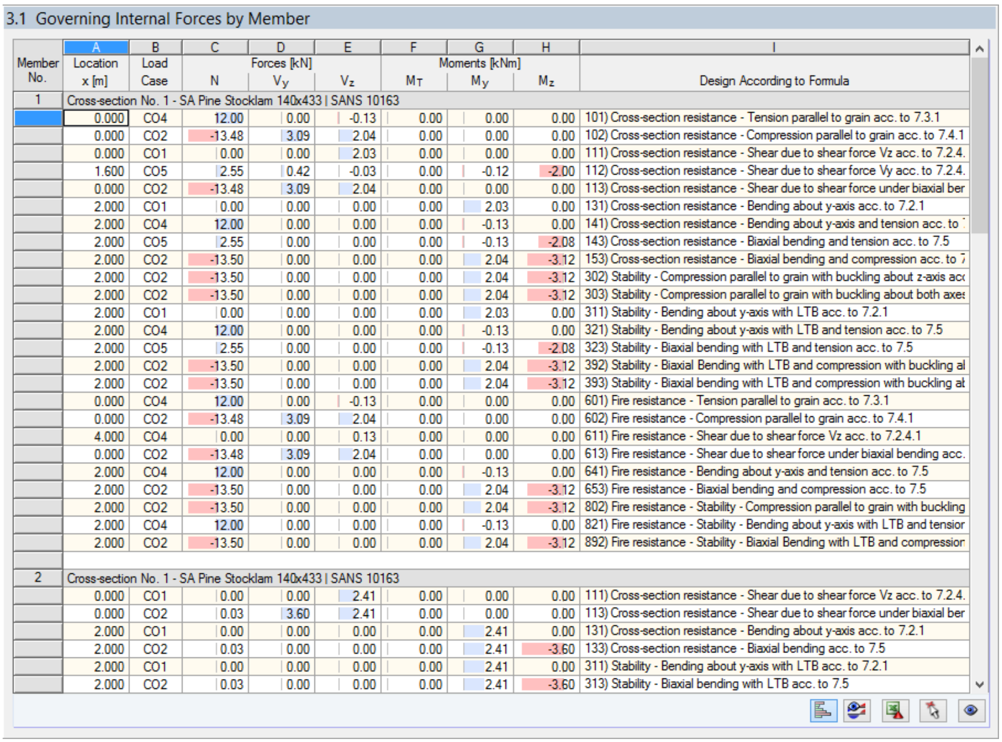


Figure 4.6: Window 2.5 *Design by x-Location*

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

- Start and end node
- Division points according to possibly defined member division (see RFEM table 1.16)
- Member division according to specification for member results (RFEM 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
1	Cross-section No. 1 - SA Pine Stocklam 140x433 | SANS 10163								
	0.000	CO4	12.00	0.00	-0.13	0.00	0.00	0.00	101) Cross-section resistance - Tension parallel to grain acc. to 7.3.1
	0.000	CO2	-13.48	3.09	2.04	0.00	0.00	0.00	102) Cross-section resistance - Compression parallel to grain acc. to 7.4.1
	0.000	CO1	0.00	0.00	2.03	0.00	0.00	0.00	111) Cross-section resistance - Shear due to shear force Vz acc. to 7.2.4.
	1.600	CO5	2.55	0.42	-0.03	0.00	-0.12	-2.00	112) Cross-section resistance - Shear due to shear force Vy acc. to 7.2.4.
	0.000	CO2	-13.48	3.09	2.04	0.00	0.00	0.00	113) Cross-section resistance - Shear due to shear force under biaxial ber
	2.000	CO1	0.00	0.00	0.00	0.00	2.03	0.00	131) Cross-section resistance - Bending about y-axis acc. to 7.2.1
	2.000	CO4	12.00	0.00	0.00	0.00	-0.13	0.00	141) Cross-section resistance - Bending about y-axis and tension acc. to
	2.000	CO5	2.55	0.00	0.00	0.00	-0.13	-2.08	143) Cross-section resistance - Biaxial bending and tension acc. to 7.5
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	153) Cross-section resistance - Biaxial bending and compression acc. to 7
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	302) Stability - Compression parallel to grain with buckling about z-axis acc
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	303) Stability - Compression parallel to grain with buckling about both axes
	2.000	CO1	0.00	0.00	0.00	0.00	2.03	0.00	311) Stability - Bending about y-axis with LTB acc. to 7.2.1
	2.000	CO4	12.00	0.00	0.00	0.00	-0.13	0.00	321) Stability - Bending about y-axis with LTB and tension acc. to 7.5
	2.000	CO5	2.55	0.00	0.00	0.00	-0.13	-2.08	323) Stability - Biaxial bending with LTB and tension acc. to 7.5
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	392) Stability - Biaxial Bending with LTB and compression with buckling al
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	393) Stability - Biaxial bending with LTB and compression with buckling al
	0.000	CO4	12.00	0.00	-0.13	0.00	0.00	0.00	601) Fire resistance - Tension parallel to grain acc. to 7.3.1
	0.000	CO2	-13.48	3.09	2.04	0.00	0.00	0.00	602) Fire resistance - Compression parallel to grain acc. to 7.4.1
	4.000	CO4	0.00	0.00	0.13	0.00	0.00	0.00	611) Fire resistance - Shear due to shear force Vz acc. to 7.2.4.1
	0.000	CO2	-13.48	3.09	2.04	0.00	0.00	0.00	613) Fire resistance - Shear due to shear force under biaxial bending acc.
	2.000	CO4	12.00	0.00	0.00	0.00	-0.13	0.00	641) Fire resistance - Bending about y-axis and tension acc. to 7.5
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	653) Fire resistance - Biaxial bending and compression acc. to 7.5
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	802) Fire resistance - Stability - Compression parallel to grain with buckling
	2.000	CO4	12.00	0.00	0.00	0.00	-0.13	0.00	821) Fire resistance - Stability - Bending about y-axis with LTB and tensior
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	892) Fire resistance - Stability - Biaxial Bending with LTB and compressior
2	Cross-section No. 1 - SA Pine Stocklam 140x433 | SANS 10163								
	0.000	CO1	0.00	0.00	2.41	0.00	0.00	0.00	111) Cross-section resistance - Shear due to shear force Vz acc. to 7.2.4.
	0.000	CO2	0.03	3.60	2.41	0.00	0.00	0.00	113) Cross-section resistance - Shear due to shear force under biaxial ber
	2.000	CO1	0.00	0.00	0.00	0.00	2.41	0.00	131) Cross-section resistance - Bending about y-axis acc. to 7.2.1
	2.000	CO2	0.03	0.00	0.00	0.00	2.41	-3.60	133) Cross-section resistance - Biaxial bending acc. to 7.5
	2.000	CO1	0.00	0.00	0.00	0.00	2.41	0.00	311) Stability - Bending about y-axis with LTB acc. to 7.2.1
	2.000	CO2	0.03	0.00	0.00	0.00	2.41	-3.60	313) Stability - Biaxial bending with LTB acc. to 7.5

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

For each member, this window displays the governing internal forces, that is, those internal forces that result in the maximum utilization in each design.

Location x

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

Load Case

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

Forces / Moments

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

Design According to Formula

The final column provides information on the types of design and the equations by which the designs according to [1] or [2] have been performed.

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	2 (Member No. 2)								
	0.000	CO1	0.00	0.00	2.41	0.00	0.00	0.00	111) Cross-section resistance - Shear due to shear force Vz acc. to 7.2.4.
	0.000	CO2	0.03	3.60	2.41	0.00	0.00	0.00	113) Cross-section resistance - Shear due to shear force under biaxial ber
	2.000	CO1	0.00	0.00	0.00	0.00	2.41	0.00	131) Cross-section resistance - Bending about y-axis acc. to 7.2.1
	2.000	CO2	0.03	0.00	0.00	0.00	2.41	-3.60	133) Cross-section resistance - Biaxial bending acc. to 7.5
	2.000	CO1	0.00	0.00	0.00	0.00	2.41	0.00	311) Stability - Bending about y-axis with LTB acc. to 7.2.1
	2.000	CO2	0.03	0.00	0.00	0.00	2.41	-3.60	313) Stability - Biaxial bending with LTB acc. to 7.5
	0.000	CO8	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	2.000	CO8	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Design deflection acc. to 6.4.1 - Inner span, y-directic
	2.000	CO14	0.00	0.00	0.00	0.00	0.00	0.00	402) Serviceability - Design deflection acc. to 6.4.1 - Inner span, z-directic
	0.000	CO4	0.00	0.00	0.61	0.00	0.00	0.00	611) Fire resistance - Shear due to shear force Vz acc. to 7.2.4.1
	0.000	CO2	0.03	3.60	2.41	0.00	0.00	0.00	613) Fire resistance - Shear due to shear force under biaxial bending acc.
	2.000	CO4	0.00	0.00	0.00	0.00	0.61	0.00	631) Fire resistance - Bending about y-axis acc. to 7.2.1
	2.000	CO2	0.03	0.00	0.00	0.00	2.41	-3.60	633) Fire resistance - Biaxial bending acc. to 7.5
	2.000	CO4	0.00	0.00	0.00	0.00	0.61	0.00	811) Fire resistance - Stability - Bending about y-axis with LTB acc. to 7.2
	2.000	CO2	0.03	0.00	0.00	0.00	2.41	-3.60	813) Fire resistance - Stability - Biaxial bending with LTB acc. to 7.5
4	1 (Member No. 1)								
	0.000	CO4	12.00	0.00	-0.13	0.00	0.00	0.00	101) Cross-section resistance - Tension parallel to grain acc. to 7.3.1
	0.000	CO2	-13.48	3.09	2.04	0.00	0.00	0.00	102) Cross-section resistance - Compression parallel to grain acc. to 7.4.1
	0.000	CO1	0.00	0.00	2.03	0.00	0.00	0.00	111) Cross-section resistance - Shear due to shear force Vz acc. to 7.2.4.
	1.600	CO5	2.55	0.42	-0.03	0.00	-0.12	-2.00	112) Cross-section resistance - Shear due to shear force Vy acc. to 7.2.4.
	0.000	CO2	-13.48	3.09	2.04	0.00	0.00	0.00	113) Cross-section resistance - Shear due to shear force under biaxial ber
	2.000	CO1	0.00	0.00	0.00	0.00	2.03	0.00	131) Cross-section resistance - Bending about y-axis acc. to 7.2.1
	2.000	CO4	12.00	0.00	0.00	0.00	-0.13	0.00	141) Cross-section resistance - Bending about y-axis and tension acc. to
	2.000	CO5	2.55	0.00	0.00	0.00	-0.13	-2.08	143) Cross-section resistance - Biaxial bending and tension acc. to 7.5
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	153) Cross-section resistance - Biaxial bending and compression acc. to 7
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	302) Stability - Compression parallel to grain with buckling about z-axis acc
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	303) Stability - Compression parallel to grain with buckling about both axes
	2.000	CO1	0.00	0.00	0.00	0.00	2.03	0.00	311) Stability - Bending about y-axis with LTB acc. to 7.2.1
	2.000	CO4	12.00	0.00	0.00	0.00	-0.13	0.00	321) Stability - Bending about y-axis with LTB and tension acc. to 7.5
	2.000	CO5	2.55	0.00	0.00	0.00	-0.13	-2.08	323) Stability - Biaxial bending with LTB and tension acc. to 7.5
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	392) Stability - Biaxial Bending with LTB and compression with buckling al
	2.000	CO2	-13.50	0.00	0.00	0.00	2.04	-3.12	393) Stability - Biaxial bending with LTB and compression with buckling at

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 design for each set of members.

4.8 Member Slendernesses

3.3 Member Slendernesses

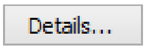
Member No.	A Loss of Stability Under Stress	B Buckling About Axis Y $L_{e,y}/r_y$ [-]	C Ratio	D Buckling About Axis Z $L_{e,z}/r_z$ [-]	E Ratio	F Lateral Buckling C_s [-]	G Ratio	H Tension L/r_y [-]	I Ratio	J L/r_z [-]	K Ratio
1	Compression / Bending / Tension	32.913	0.18	59.385	0.33	9.400	0.18	32.913	0.13	98.974	0.40
2	Bending / Tension	-	-	-	-	9.400	0.18	32.913	0.13	98.974	0.40
3	Tension	-	-	-	-	-	-	90.909	0.36	90.909	0.36
4	Bending / Tension	-	-	-	-	9.400	0.18	32.913	0.13	98.974	0.40

Compression members:
Max $L_{e,y}/r_y$: 32.913 ≤ 180
Max $L_{e,z}/r_z$: 59.385 ≤ 180

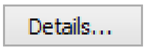
Bending members:
Max C_s : 9.400 ≤ 52

Tension members:
Max L/r_y : 90.909 ≤ 250
Max L/r_z : 98.974 ≤ 250

Figure 4.9: Window 3.3 *Member Slendernesses*



This results window is displayed when you have selected the respective check box in the *Other* tab of the *Details* dialog box (see Figure 3.5, page 44).

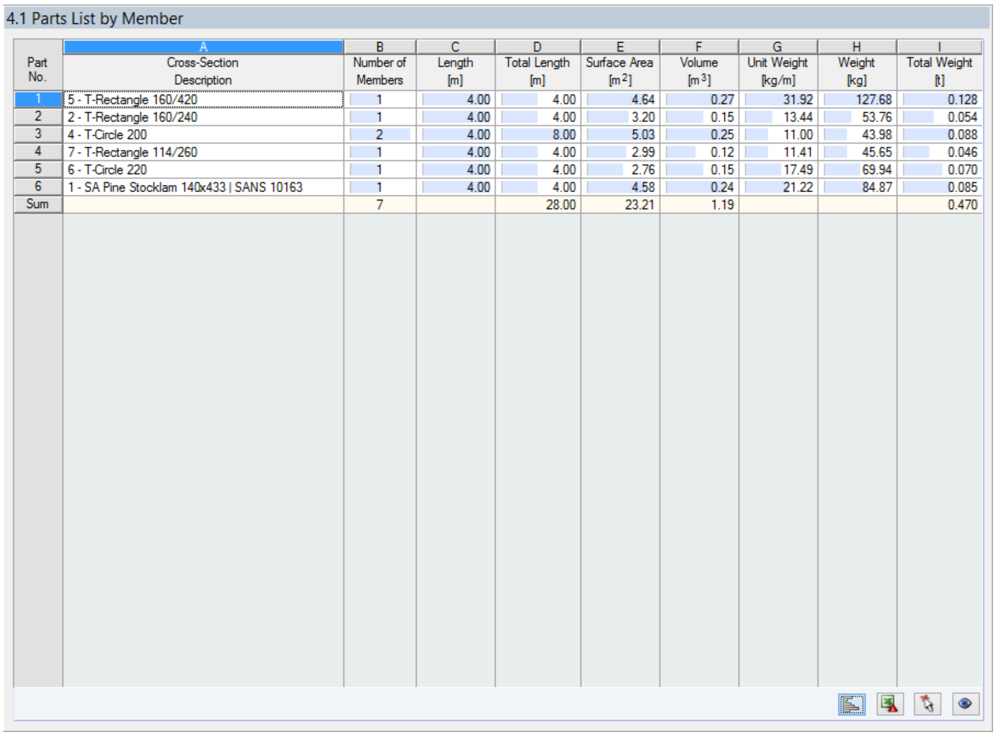


The table lists all effective slendernesses ratios of the designed members for both principal axes directions. They are determined depending on the load type. At the end of the list, you find a comparison with the limit values that have been defined in the *Details* dialog box, tab *Other* (see Figure 3.5, page 44).

This window is displayed only for information. No design of the slendernesses is carried out.

4.9 Parts List by Member

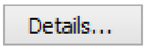
RF-TIMBER SANS provides a summary of all cross-sections that are included in the design case.



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	5 - T-Rectangle 160/420	1	4.00	4.00	4.64	0.27	31.92	127.68	0.128
2	2 - T-Rectangle 160/240	1	4.00	4.00	3.20	0.15	13.44	53.76	0.054
3	4 - T-Circle 200	2	4.00	8.00	5.03	0.25	11.00	43.98	0.088
4	7 - T-Rectangle 114/260	1	4.00	4.00	2.99	0.12	11.41	45.65	0.046
5	6 - T-Circle 220	1	4.00	4.00	2.76	0.15	17.49	69.94	0.070
6	1 - SA Pine Stocklam 140x433 | SANS 10163	1	4.00	4.00	4.58	0.24	21.22	84.87	0.085
Sum		7		28.00	23.21	1.19			0.470

Figure 4.10: Window 4.1 *Parts List by Member*



By default, the list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box, tab *Other* (see Figure 3.5, page 44).

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

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

Length

This column displays the respective length of an individual member.

Total Length

This column shows the product determined from the two previous columns.

Surface Area

For each part, the program indicates the surface area relative to the total length. The surface area is determined from the *Surface Area* of the cross-sections that can be seen in Windows 1.3 and 2.1 through 2.5 (see Figure 2.20, page 22).

Volume

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

Unit Weight

The *Unit Weight* of the cross-section is relative to the length of one meter.

Weight

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

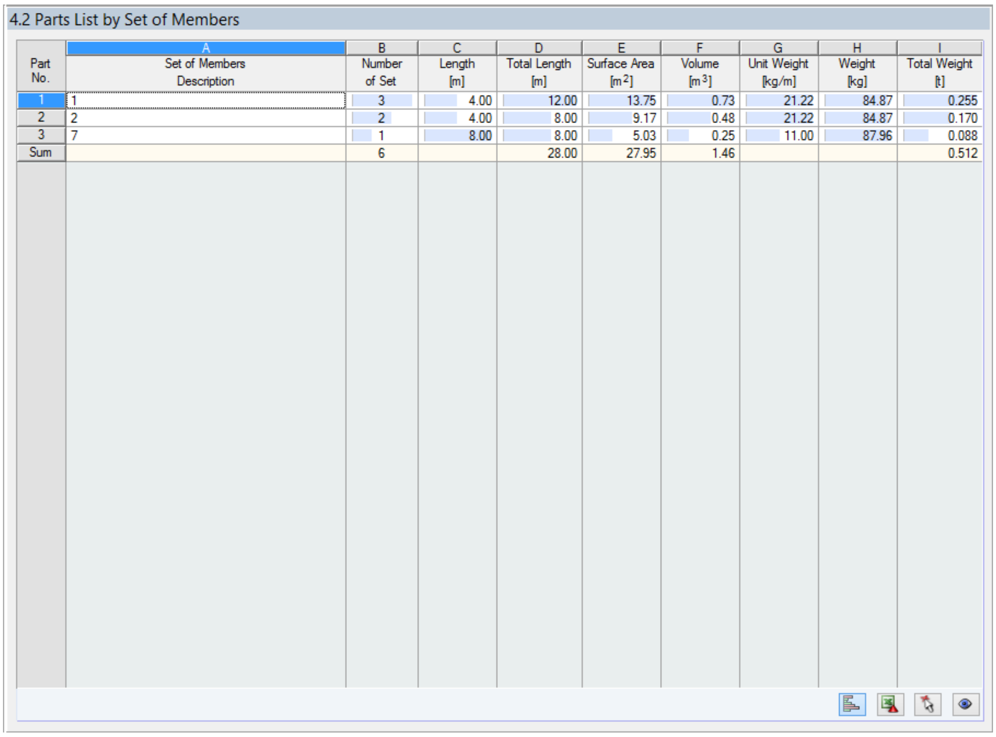
Total Weight

The final column indicates the total weight of each part.

Sum

At the bottom of the list, you find a sum of the values in the columns B, D, E, F, and I. The last cell 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	3	4.00	12.00	13.75	0.73	21.22	84.87	0.255
2	2	2	4.00	8.00	9.17	0.48	21.22	84.87	0.170
3	7	1	8.00	8.00	5.03	0.25	11.00	87.96	0.088
Sum		6		28.00	27.95	1.46			0.512

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

The last results window is displayed if you have selected at least one set of members for design. It summarizes an entire structural group (for example a chord) in a parts list.

Details on the various columns can be found in the previous chapter. 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. Evaluation of Results

You can evaluate the design results in different ways.

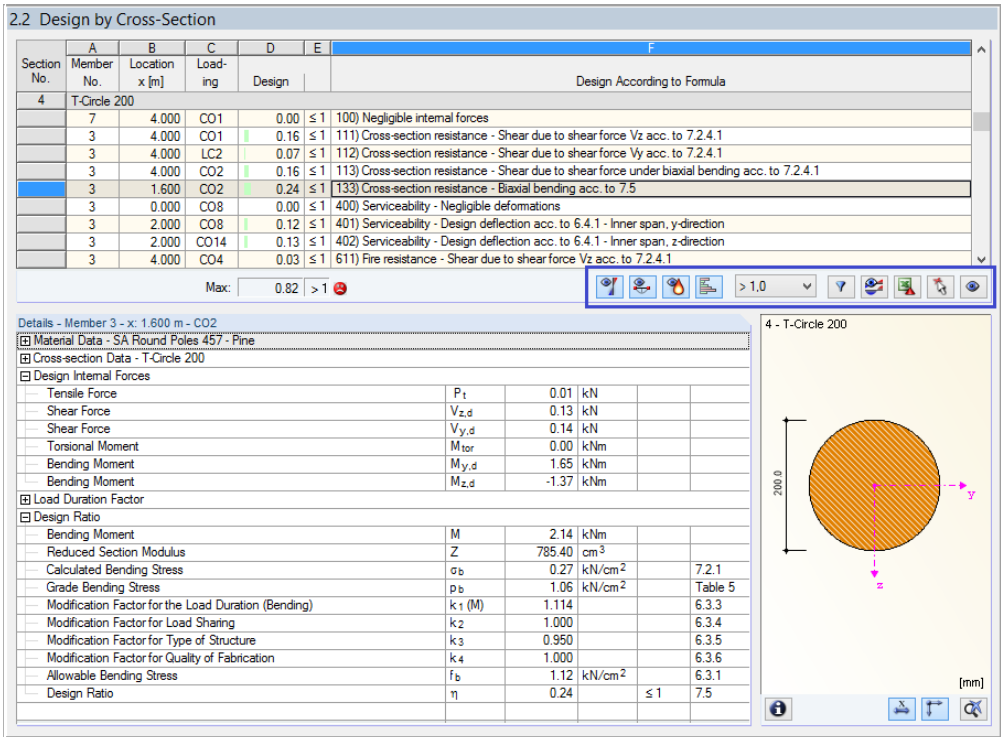


Figure 5.1: Buttons for evaluation of results

The buttons below the upper results tables are very useful for the evaluation of the results.

Button	Description	Function
	Ultimate Limit State Designs	Shows or hides the results of the ultimate limit state design
	Serviceability Limit State Designs	Shows or hides the results of the serviceability limit state design
	Fire Resistance Design	Shows or hides the results of the fire resistance design
	Show Color Bars	Shows or hides the colored relation scales in the results windows
> 1,0	Filter Options	Displays only rows with ratios greater than the filter criterion set in text box: design ratios > 1, maximum or user-defined limit
	Result Diagrams	Opens the Window *Result Diagram on Member* → Chapter 5.2, page 58
	Excel Export	Exports the table to MS Excel / OpenOffice → Chapter 7.4.3, page 68
	Member Selection	Allows you to graphically select a member to display its results in the table
	View Mode	Jumps to the RFEM work window to change the view

Table 5.1: Buttons in Windows 2.1 through 2.5

5.1 Results in RFEM Model

To evaluate the design results, you can also use the RFEM work window.

RFEM background graphic and view mode

The RFEM work window in the background is useful for finding the position of a particular member in the model: The member selected in the RF-TIMBER SANS results window is highlighted in the selection color in the background graphic. Furthermore, an arrow indicates the member's x-location that is displayed in the selected window row.

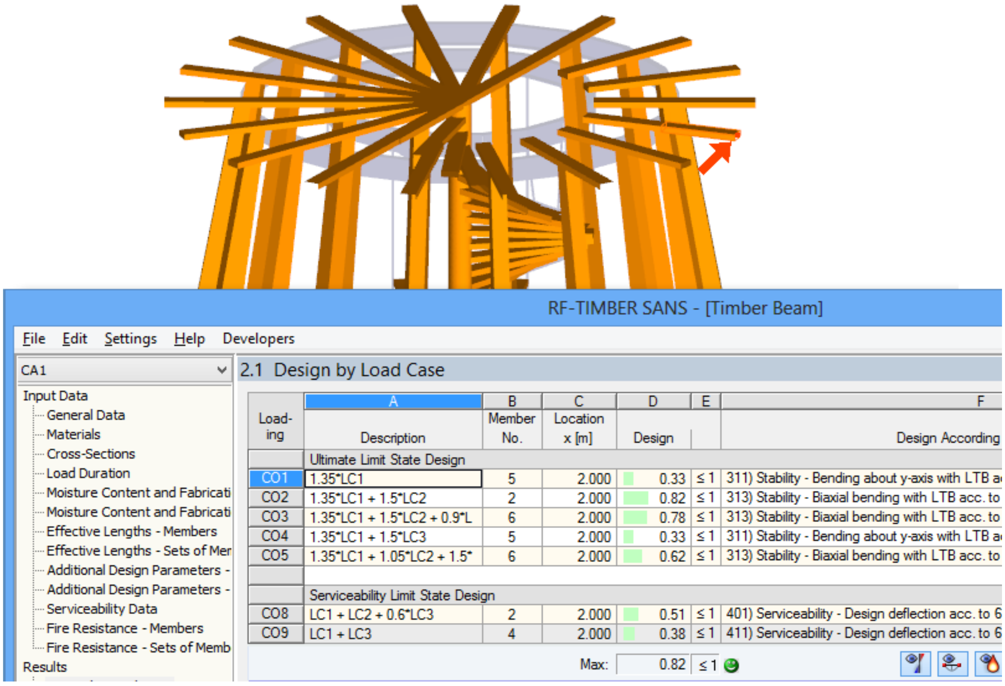
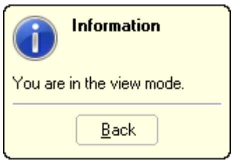


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model



If you cannot improve the display by moving the RF-TIMBER SANS module window, click [Jump to Graphic] to activate the *View Mode*: Thus, you hide the module window so that you can modify the display in the RFEM user interface. In the view mode, you can use the functions of the *View* menu, for example zooming, moving, or rotating the display. The pointer remains visible.

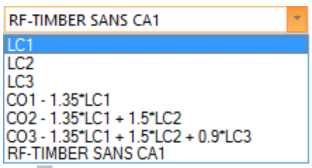
Click [Back] to return to the add-on module RF-TIMBER SANS.

RFEM work window

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

To turn the display of design results on or off, use the [Show Results] button known from the display of internal forces in RFEM. To display the result values, click the [Show Values] toolbar button to the right.

The RFEM tables are of no relevance for the evaluation of design results.



The design cases can be set by means of the list in the RFEM menu bar.

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

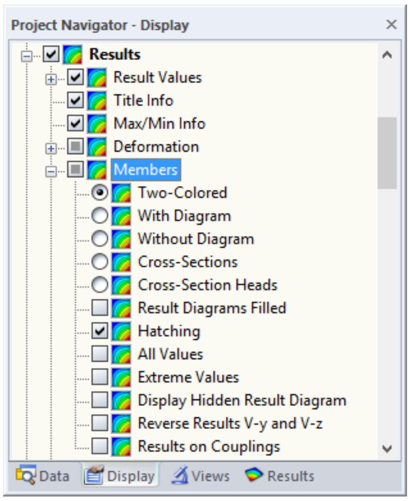


Figure 5.3: *Display* navigator: Results → Members

When you select a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color panel becomes available. It provides common control functions that are described in detail in the RFEM manual, Chapter 3.4.6.

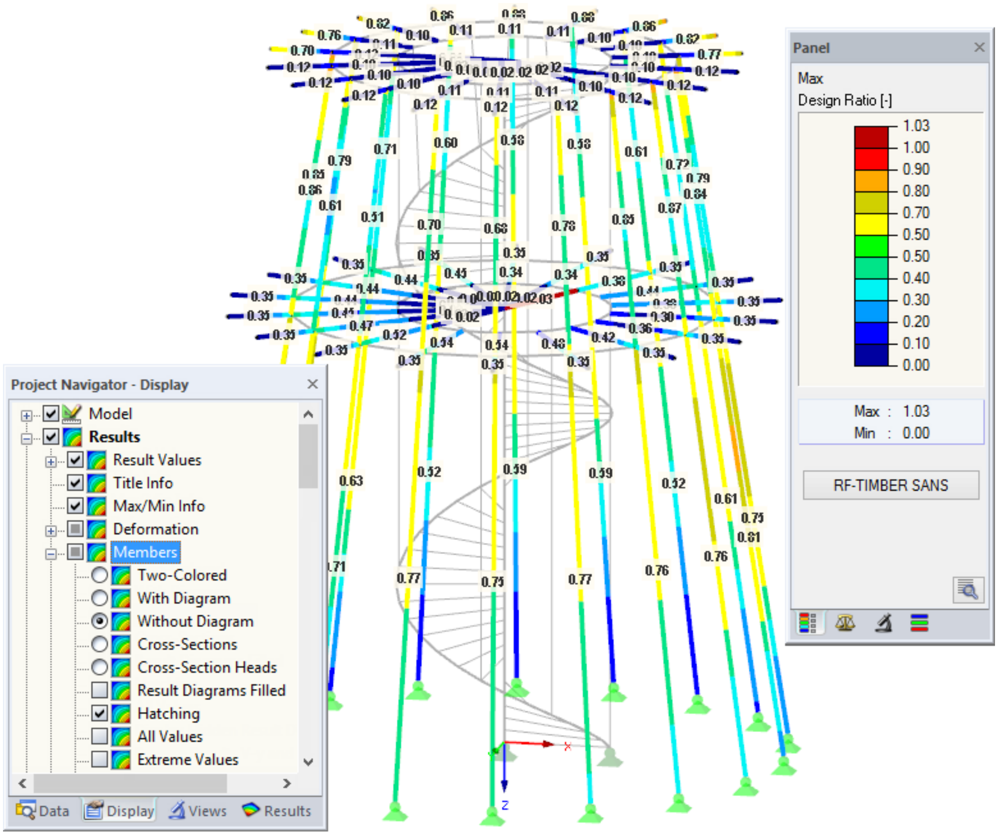


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 61).

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

5.2 Result Diagrams

You can also evaluate the member results graphically in the result diagram.

To do this, select the member (or set of members) in the RF-TIMBER SANS results window by clicking in the table row of the member. Then open the *Result Diagram on Member* dialog box by clicking the button shown on the left. The button is located below the upper results table (see Figure 5.1, page 55).

To display the result diagrams, select the command from the RFEM menu

Results → Result Diagrams for Selected Members

or use the button in the RFEM toolbar shown on the left.

A window opens, graphically showing the distribution of the maximum design values on the member or set of members.

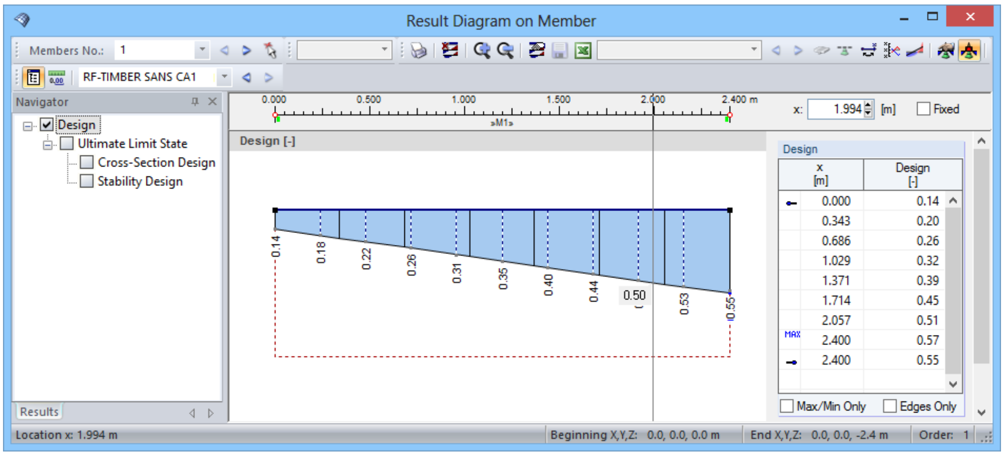
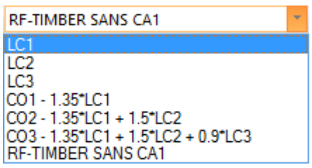


Figure 5.5: Dialog box *Result Diagram on Member*



Use the list in the toolbar above to select the relevant RF-TIMBER SANS design case.

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

5.3 Filter for Results

The RF-TIMBER SANS results windows allow you to sort the results by various criteria. In addition, you can use the filter options for graphical evaluation of the results as described in Chapter 9.9 of the RFEM manual.

You can use the *Visibility* option also for RF-TIMBER SANS (see RFEM manual, Chapter 9.9.1) to filter the members in order to evaluate them.

Filtering designs

Graphics

The design ratios can easily be used as filter criteria in the RFEM work window, which you can open 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 manual, Chapter 3.4.6. The filter settings for the results must be defined in the first panel tab (Color spectrum). As this register is not available for the two-colored results display, you have to use the *Display* navigator and set the display options *Colored With/Without Diagram* or *Cross-Sections* first.

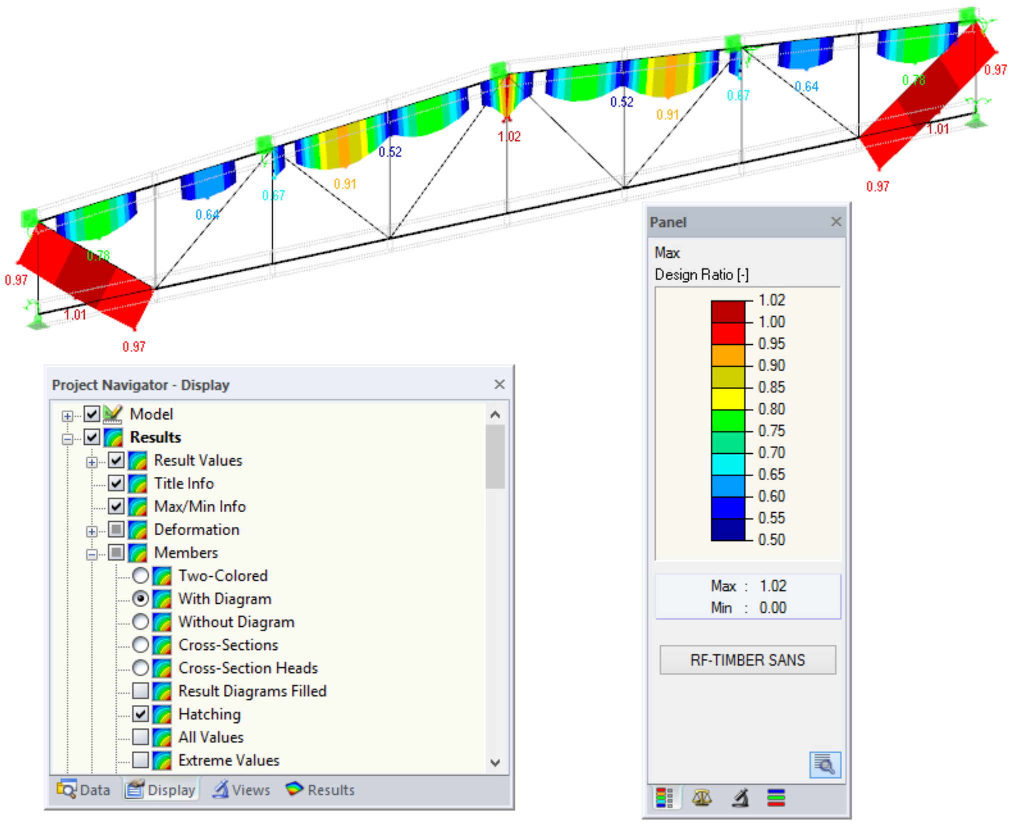
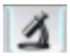


Figure 5.6: Filtering design ratios with adjusted color spectrum

As the figure above shows, the color spectrum can be set in such a way that only ratios higher than 0.50 are shown in a color range between blue and red.

If you select the *Display Hidden Result Diagram* option in the *Display* navigator (*Results → Members*), you can display all design ratio diagrams that are not covered by the color spectrum. Those diagrams are represented by dotted lines.

Filtering members



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

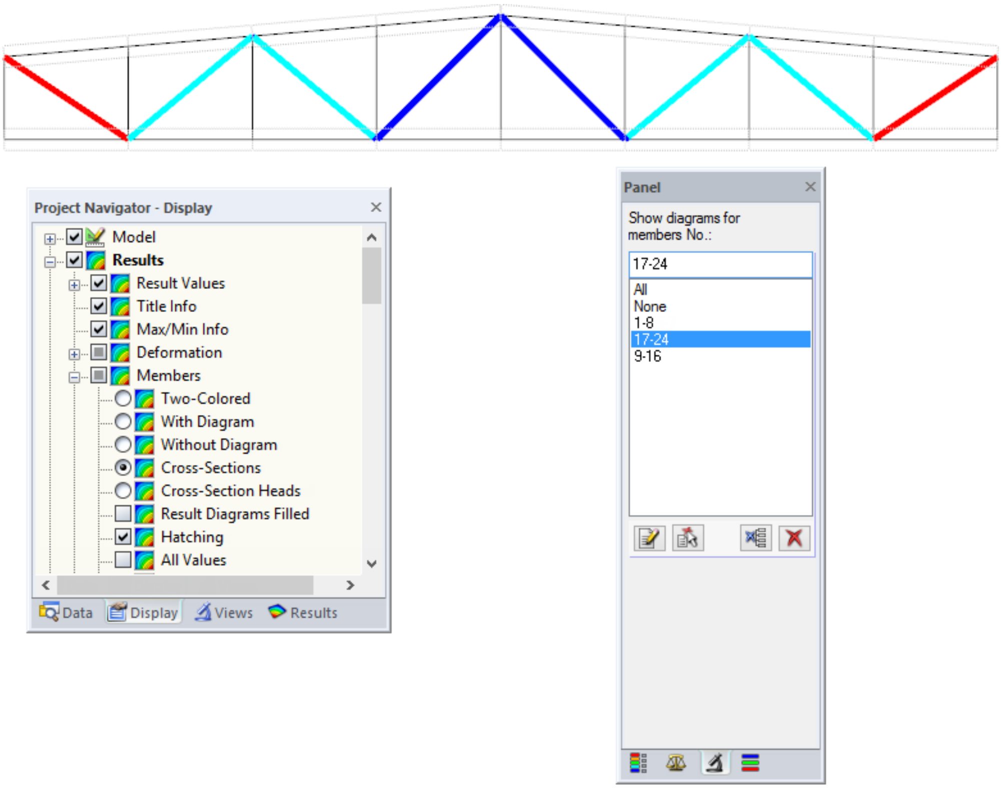


Figure 5.7: Member filter for ratios of diagonals

Unlike the partial view function (*Visibilities*), the graphic displays the entire model. The figure above shows the ratios in the diagonals of a truss girder. The remaining members are displayed in the model but are shown without design ratios.

6. Printout

6.1 Printout Report

Similar to RFEM, the program generates a printout report for the RF-TIMBER SANS results, to which you can add graphics and descriptions. The selection in the printout report determines what data from the design module will be included in the printout.

The printout report is described in the RFEM manual. In particular, Chapter 10.1.3.5 *Selecting Data of Add-on Modules* describes how to select input and output data from add-on modules for the printout report.

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

6.2 Graphic Printout

In RFEM, you can add every picture that is displayed in the work window to the printout report or send it directly to a printer. In this way, you can prepare the design ratios displayed on the RFEM model for the printout, too.

Printing of graphics is described in the RFEM manual, Chapter 10.2.

Designs in the RFEM model

To print the currently displayed design ratios, click

File → Print Graphic

or use the toolbar button shown on the left.

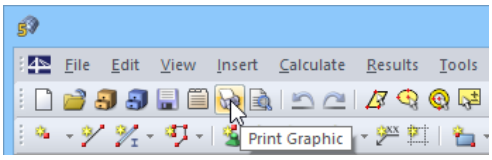


Figure 6.1: Button *Print* in RFEM toolbar

Result diagrams

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

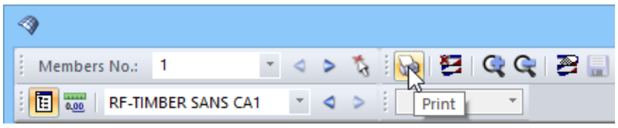


Figure 6.2: Button *Print* in the *Result Diagram on Member*

The *Graphic Printout* dialog box appears (see figure on next page).

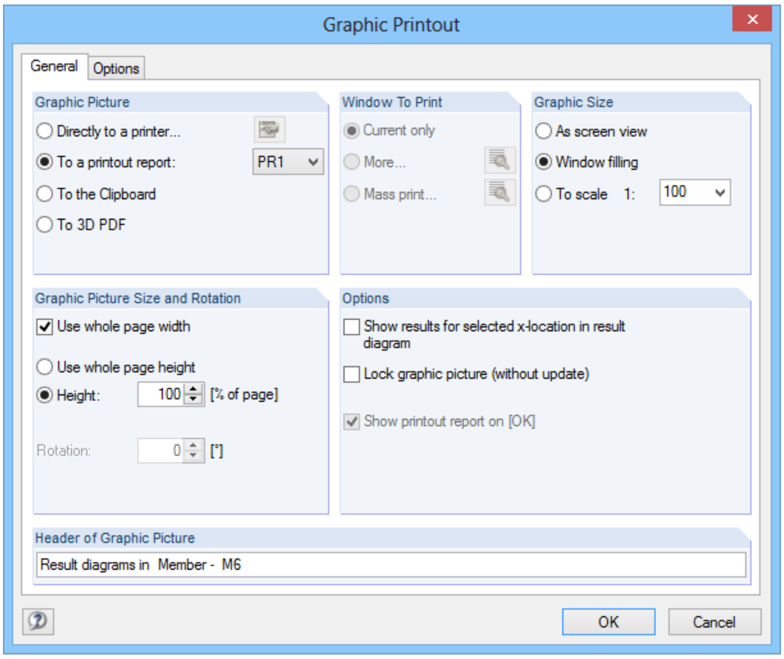
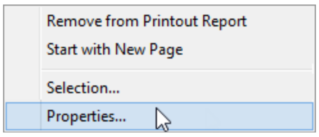


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in the RFEM manual, Chapter 10.2, including the other tabs of this dialog box.

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



To adjust a graphic subsequently 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, offering various possibilities for adjustment.

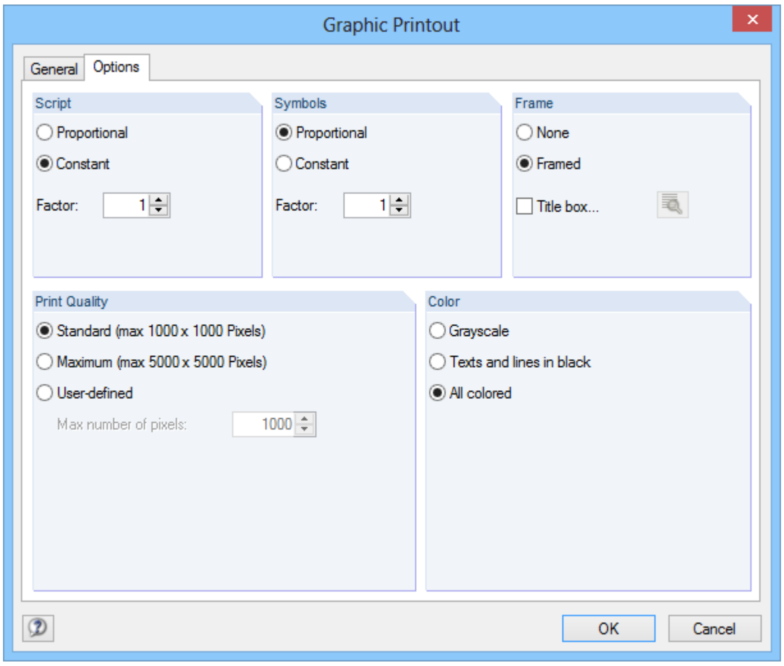


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

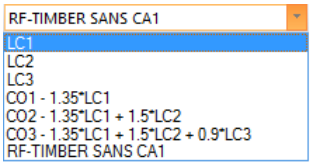
7. General Functions

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

7.1 Design Cases

Design cases allow you to group members for the design: In this way, you can combine groups of structural components or analyze members with particular design specifications (for example changed materials, load duration or system factors, optimization).

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



To calculate a RF-TIMBER SANS design case, you can also use the load case list in the RFEM toolbar.

Create new design case

To create a new design case, use the RF-TIMBER SANS menu and click

File → New Case.

The following dialog box appears:

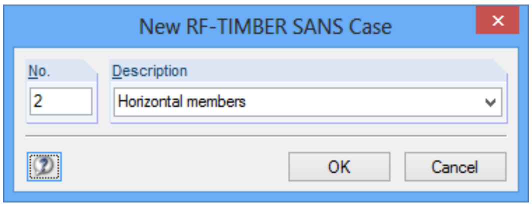


Figure 7.1: Dialog box *New RF-TIMBER SANS Case*

In this dialog box, enter a *No.* (one that is still available) for the new design case. The corresponding *Description* will make the selection in the load case list easier.

Click [OK] to open the RF-TIMBER SANS Window 1.1 *General Data* where you can enter the design data.

Rename design case

To change the description of a design case, use the RF-TIMBER SANS menu and click

File → Rename Case.

The following dialog box appears:

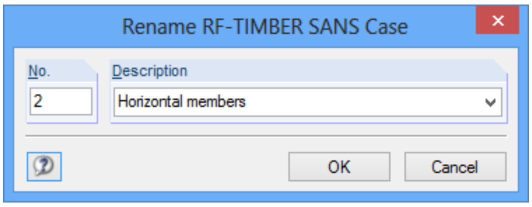


Figure 7.2: Dialog box *Rename RF-TIMBER SANS Case*

In this dialog box, you can enter a different *Description* and also a different *No.* of the design case.

Copy design case

To copy the input data of the current design case, use the RF-TIMBER SANS menu

File → Copy Case

The following dialog box appears:

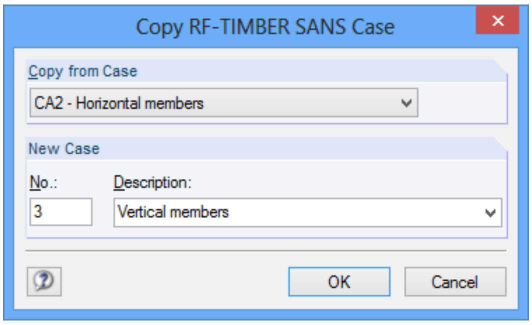


Figure 7.3: Dialog box *Copy RF-TIMBER SANS-Case*

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

Delete design case

To delete design cases, use the RF-TIMBER SANS menu

File → Delete Case

The following dialog box appears:

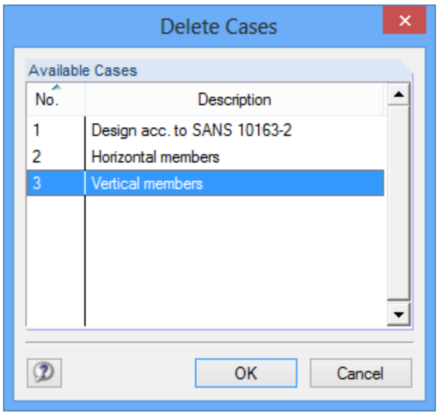


Figure 7.4: Dialog box *Delete Cases*

The design case can be selected in the list *Available Cases*. To delete the selected case, click [OK].

7.2 Cross-Section Optimization

The design module offers you the option to optimize overloaded or little utilized cross-sections. To do this, select in the column D or E of the relevant cross-sections in the 1.3 *Cross-Section* Window the option *Yes* (for parametric rectangular and circular sections) or *From current row* (for standardized sections according to [1], see Figure 2.19 on page 21).

You can also start the cross-section optimization in the results windows by using the shortcut menu.

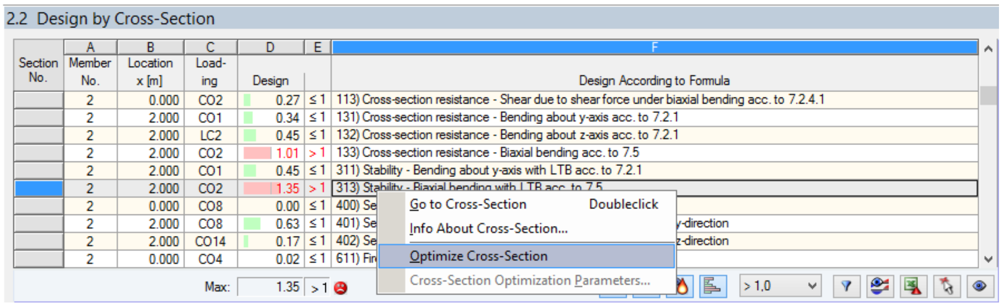


Figure 7.5: Shortcut menu for cross-section optimization

Details...

During the optimization process, the module determines the cross-section that fulfills the analysis requirements in the most optimal way, that is, comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.5, page 44). The required cross-section properties are determined with the internal forces from RFEM. If another cross-section proves to be more favorable, this cross-section is used for the design. Then, the graphic in Window 1.3 shows two cross-sections: the original cross-section from RFEM and the optimized one (see Figure 7.7).

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

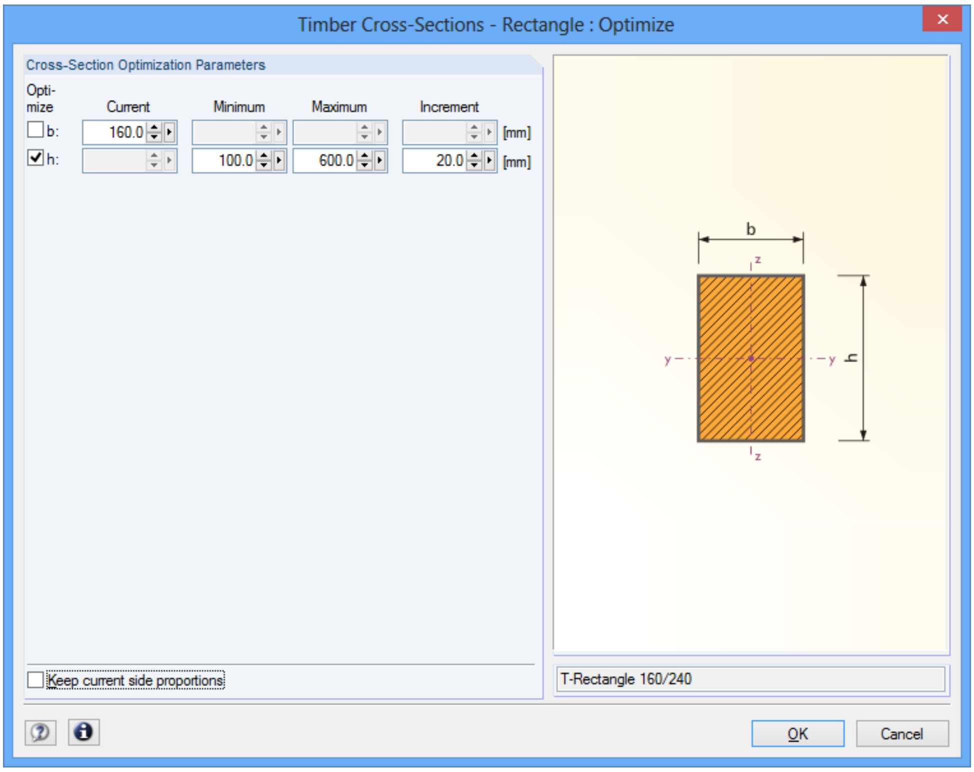


Figure 7.6: Dialog box *Timber Cross-Sections - Rectangle : Optimize*

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 process.

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

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

You can export the modified cross-sections to RFEM: Go to the 1.3 *Cross-Sections* Window, and then click

Edit → Export All Cross-Sections to RFEM

Alternatively, you can use the shortcut menu in Window 1.3 to export optimized cross-sections to RFEM.

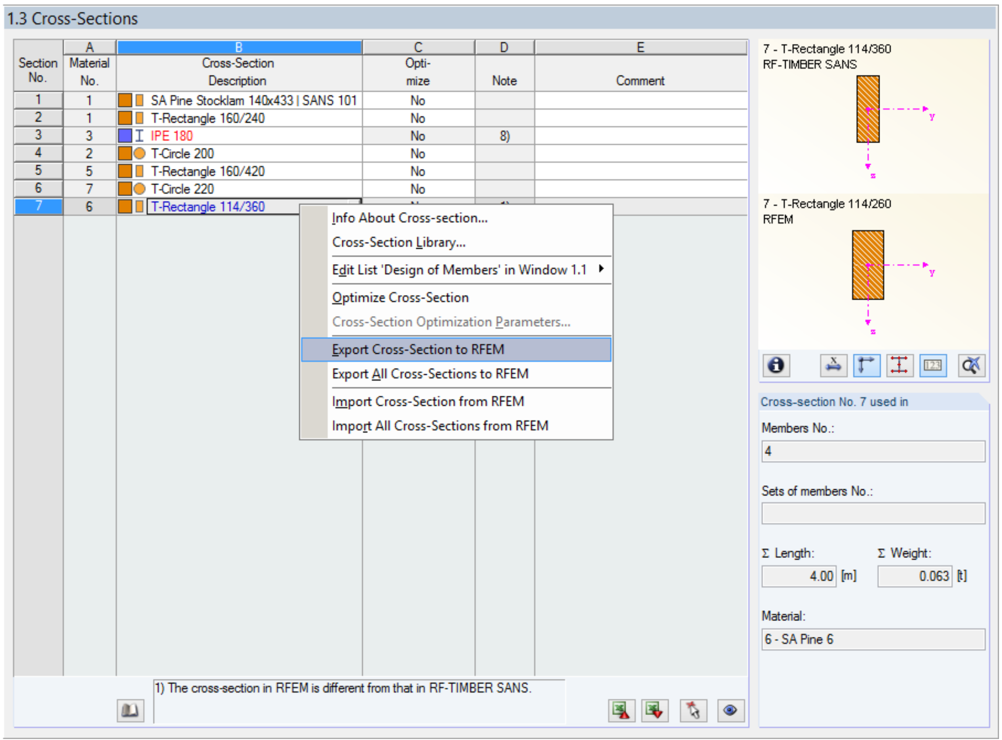


Figure 7.7: Shortcut menu in Window 1.3 *Cross-Sections*

Before the changed materials are transferred to RFEM, a security query appears as to whether the RFEM results should be deleted.

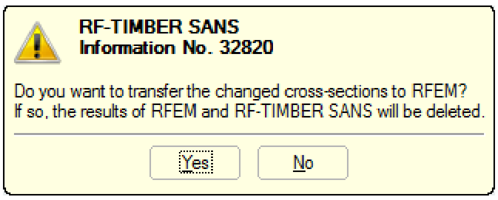


Figure 7.8: Query before transfer of modified cross-sections to RFEM

Calculation

By confirming the query and then starting the [Calculation] in the RF-TIMBER SANS module, the RFEM internal forces as well as the designs will be determined in one single calculation run.

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

7.3 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed in one dialog box. To define the units in RF-TIMBER SANS, select menu

Settings → Units and Decimal Places.

The following dialog box appears that is familiar from RFEM. RF-TIMBER SANS is preset in the *Program / Module* list.

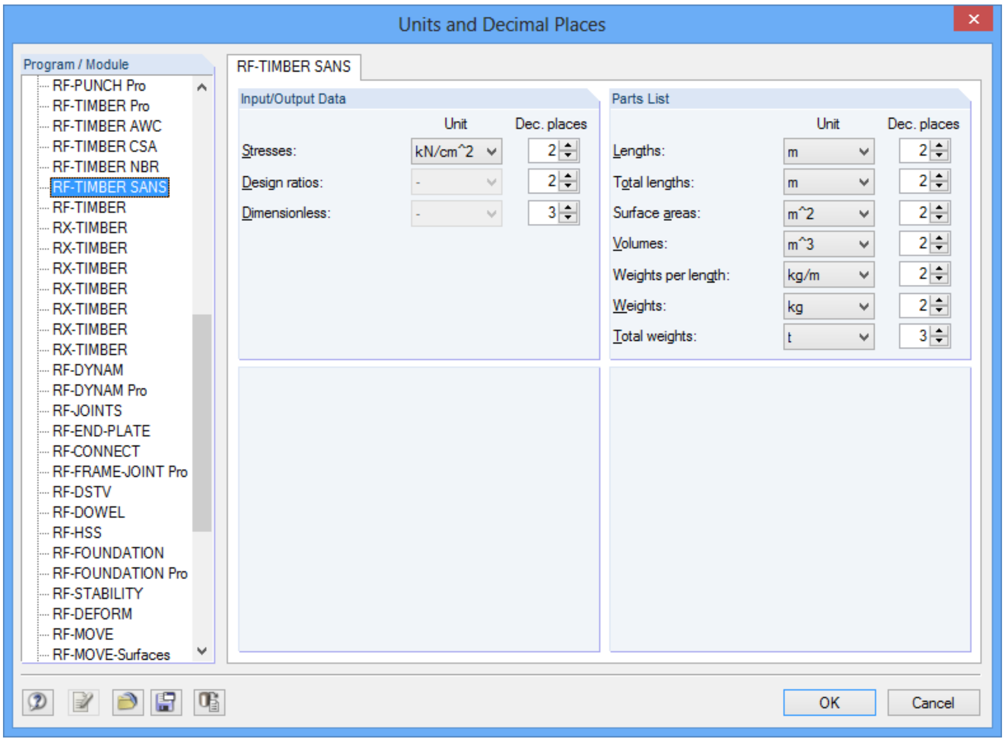


Figure 7.9: Dialog box *Units and Decimal Places*

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

7.4 Data Transfer

7.4.1 Export Material to RFEM

If you have modified the materials in RF-TIMBER SANS for design, you can export the modified materials to RFEM in a similar way as you export cross-sections: Open the 1.2 *Materials* Window, and then click

Edit → Export All Materials to RFEM.

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

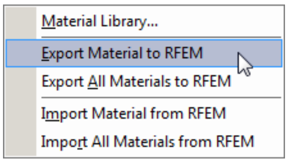
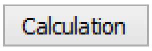


Figure 7.10: Shortcut menu of Window 1.2 *Materials*



Before the changed materials are transferred to RFEM, a security query appears as to whether the results of RFEM should be deleted. When you have confirmed the query and then start the [Calculation] in RF-TIMBER SANS, the RFEM internal forces and designs are determined in one single calculation run.

If the modified materials have not been exported to RFEM yet, you can transfer the original materials to the design module, using the options shown in Figure 7.10. Please note, however, that this option is only available in the 1.2 *Materials* Window.

7.4.2 Export Effective Lengths to RFEM

If you have adjusted the materials in RF-TIMBER SANS for design, you can export the modified materials to RFEM in a similar way as you export cross-sections: Open the 1.7 *Effective Lengths - Members* Window, and then select

Edit → Export All Effective Lengths to RFEM

or use the corresponding option on the shortcut menu of Window 1.7.

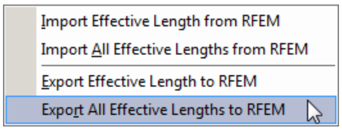


Figure 7.11: Shortcut menu of Window 1.7 *Effective Lengths - Members*

Before the modified materials are transferred to RFEM, a security query appears as to whether the results of RFEM should be deleted.

If the modified effective lengths have not been exported to RFEM yet, you can reimport the original effective lengths to the design module by using the options shown in Figure 7.11. Please note, however, that this option is only available in the Windows 1.7 *Effective Lengths - Members* and 1.8 *Effective Lengths - Sets of Members*.

7.4.3 Export Results

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

Clipboard

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

Printout Report

You can print the data of the RF-TIMBER SANS add-on module into the global printout report (see Chapter 6.1, page 61) for export. Then, in the printout report, click

File → Export to RTF

The function is described in the RFEM manual, Chapter 10.1.11.

Excel / OpenOffice

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

File → Export Tables

The following export dialog box appears.

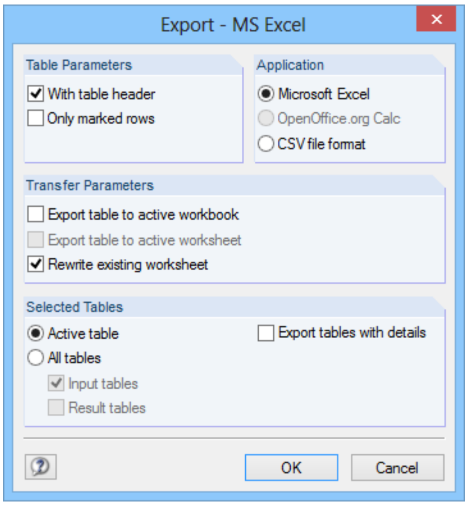


Figure 7.12: Dialog box *Export - MS Excel*

When you have selected the relevant options, you can start the export by clicking [OK]. Excel or OpenOffice will be started automatically, that is, the programs do not have to be opened first.

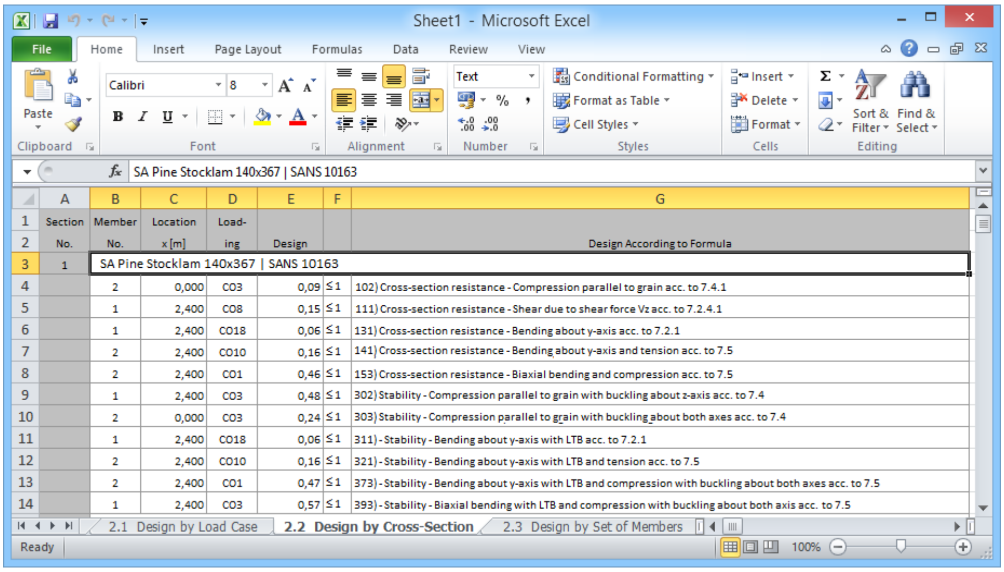


Figure 7.13: Results in *Excel*

8. Examples

8.1 Beam Column (ASD Solution)

We perform the design according to SANS 10163-2 ASD [2] for wood column that is subjected to compression and bending. At its base, it provides a non-movable hinged support. At the top, there is a hinged support which is movable along the local x-axis. The column is braced at the mid-height in the narrow dimension. The material is a SA Pine 4 and moisture content will not exceed 170 g/kg.

8.1.1 System and Loads

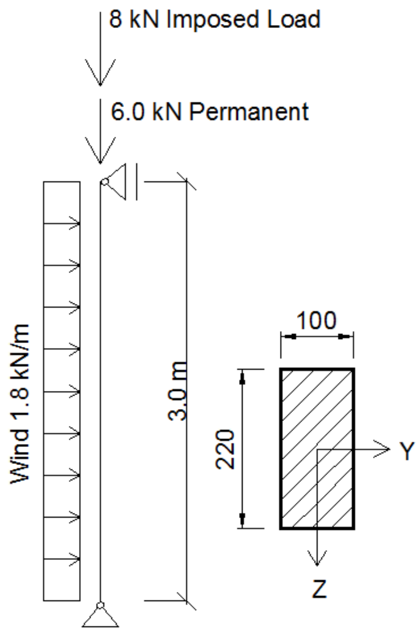


Figure 8.1: System and loads

Model

Cross-section:	b = 100 mm
	d = 220 mm
Material:	SA Pine 4
Moisture Content:	≤ 170 g/kg

Load

LC 1 Permanent:	6.0 kN
LC 2 Wind:	1.6 kN/m
LC 3 Imposed Load:	8.0 kN

Load combinations

ULS:	1.35LC1 + 1.5LC2 + 1.05LC3
SLS:	LC1 + LC2 + 0.7LC3

8.1.2 Calculation with RFEM

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

We create the load combinations for ULS and SLS with the relevant factors for the defined load cases. Then we calculate the model according to the **linear static analysis**.

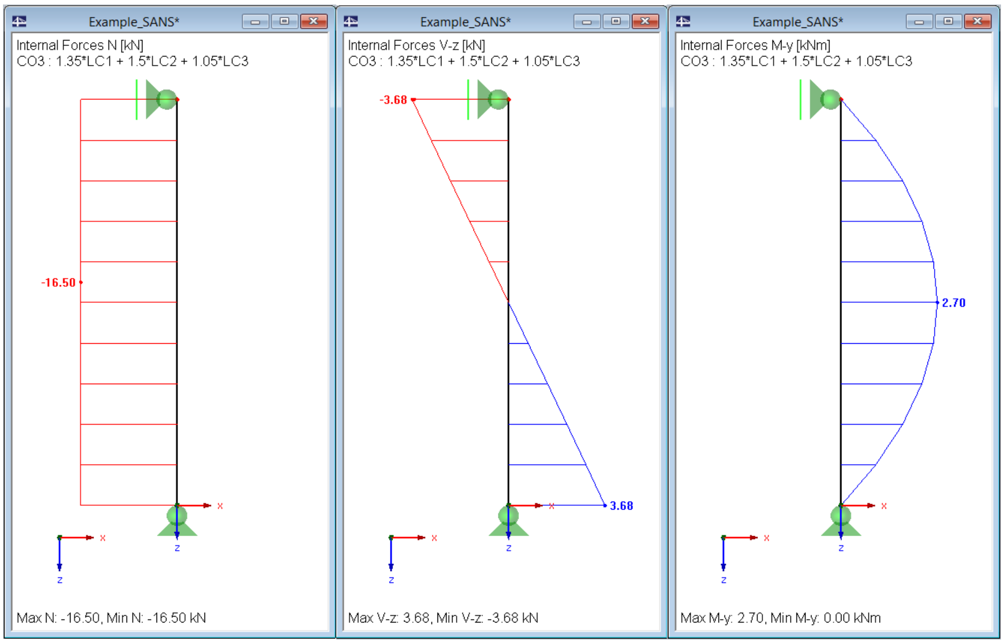


Figure 8.2: Internal forces N, V_z and M_y (ULS)

8.1.3 Design with RF-TIMBER SANS

8.1.3.1 Ultimate Limit State Design

In window 1.1 *General Data*, we select combination **CO3** for the **Ultimate Limit State** design.

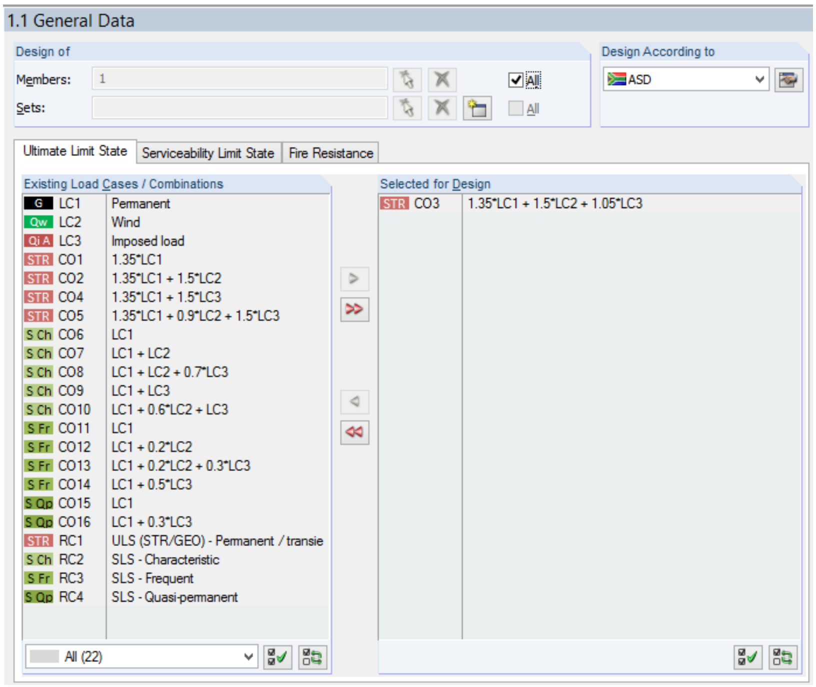


Figure 8.3: Window 1.1 *General Data*

The 1.2 *Materials* window presents the grade stresses of the selected material.

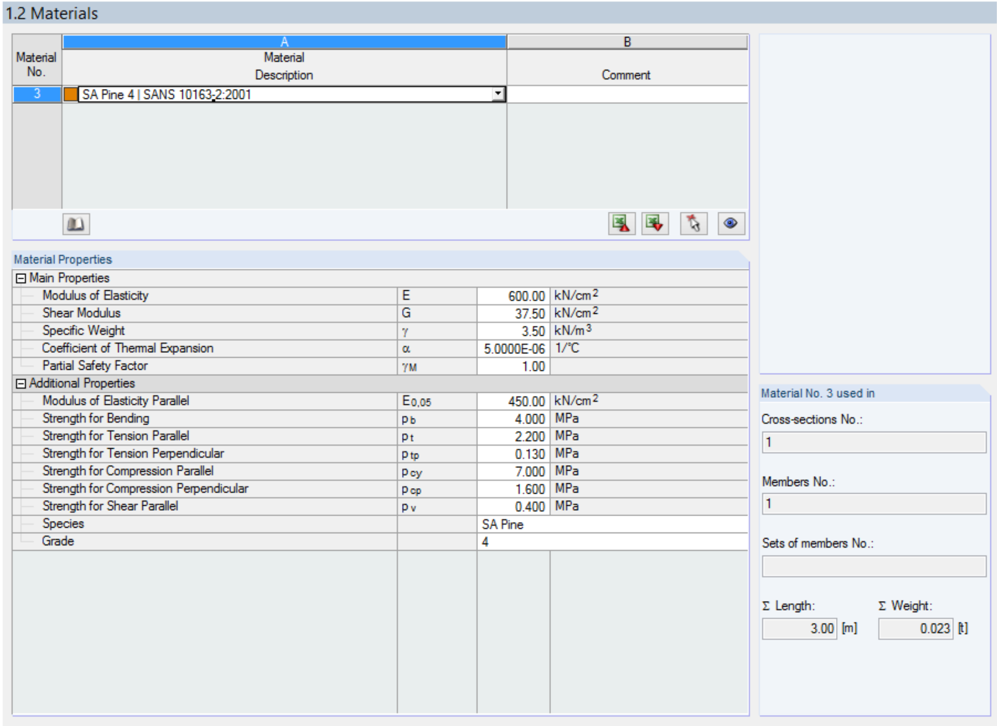


Figure 8.4: Window 1.2 *Materials*

In the 1.3 *Cross-Sections* window, the parameters of the cross-section can be checked.

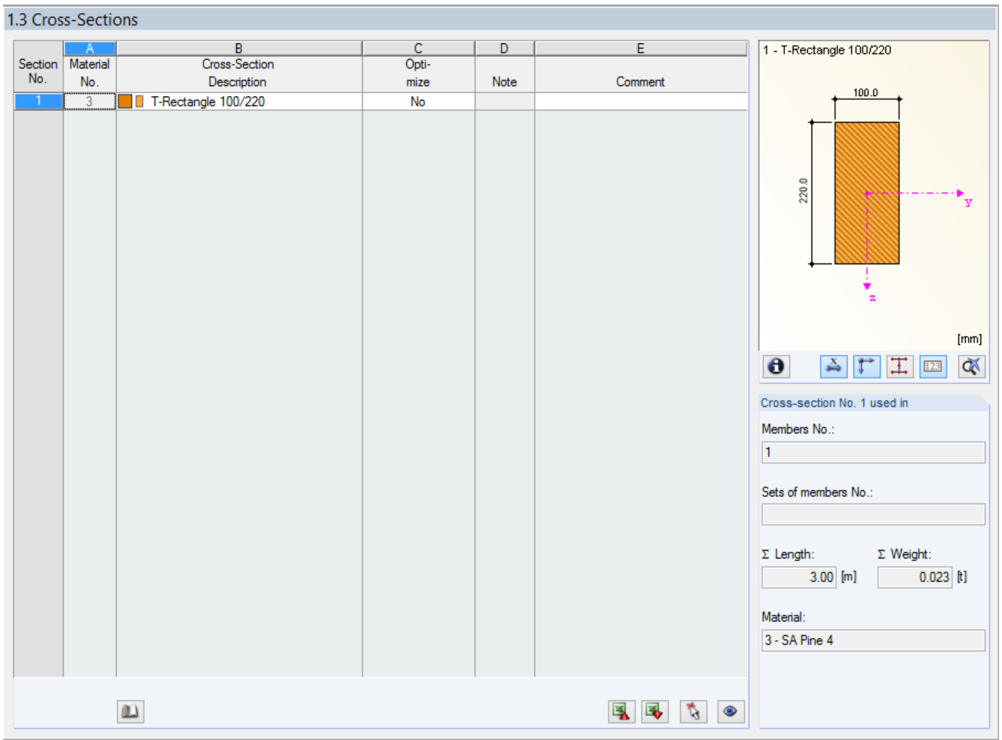


Figure 8.5: Window 1.3 *Cross-Sections*

In window 1.4 *Load Duration*, we define the load effect and load duration. The load duration factor, k_1, is determined from the ratio of internal forces at the current location. It may cause different values of the factor at each x-location.

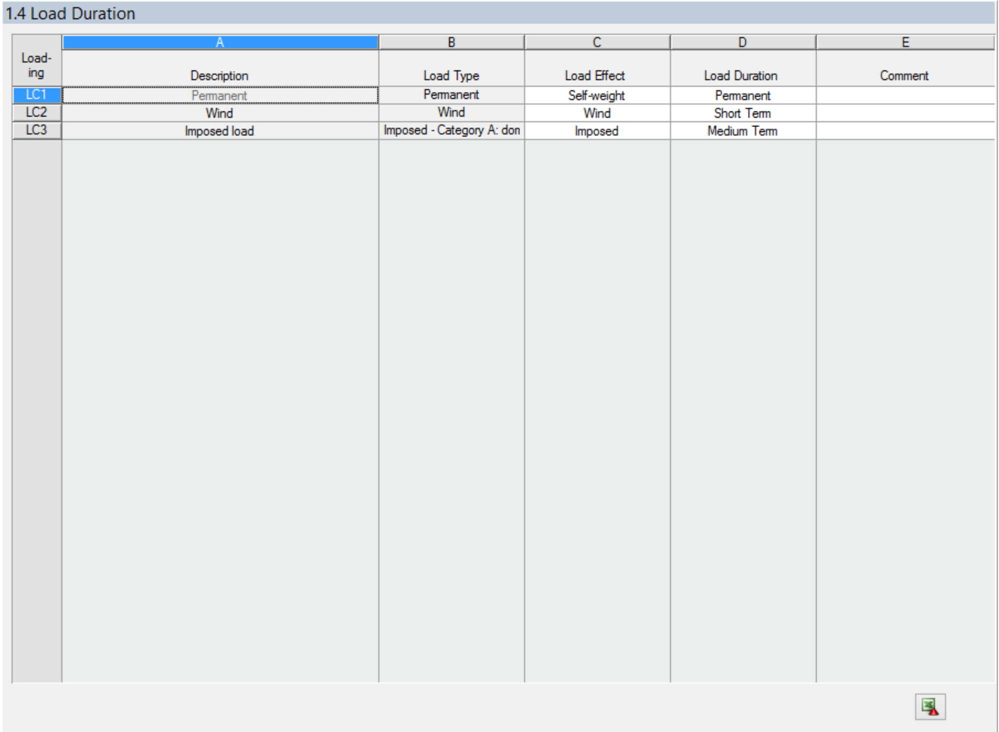


Figure 8.6: Window 1.4 *Load Duration*

In window 1.5 *Moisture Content and Fabrication Quality - Members*, we define the moisture content for ULS and SLS design. In our case the column fabrication does not comply with a SANS standard, therefore the fabrication quality factor, k_4, is taken as 1.0.

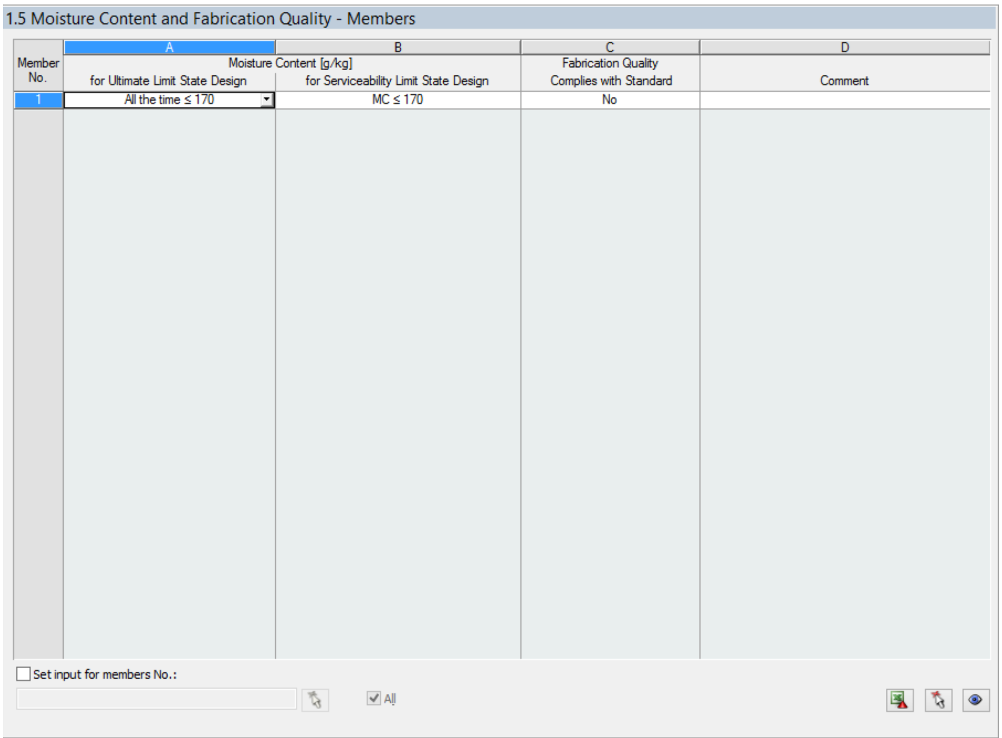
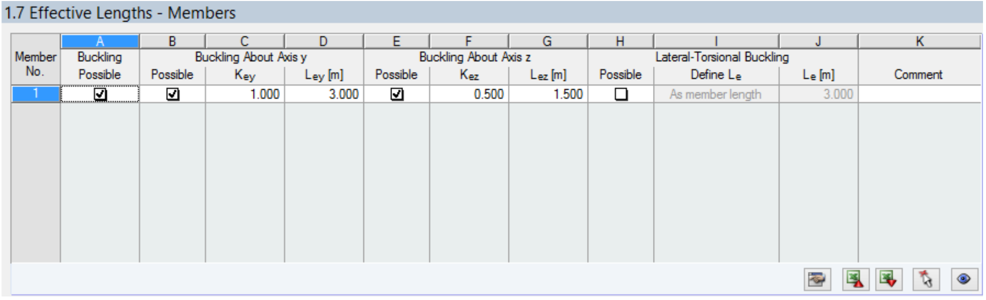


Figure 8.7: Window 1.5 *Moisture Content and Fabrication Quality - Members*

In the 1.7 *Effective Lengths - Members* window we specify the buckling lengths of the column.

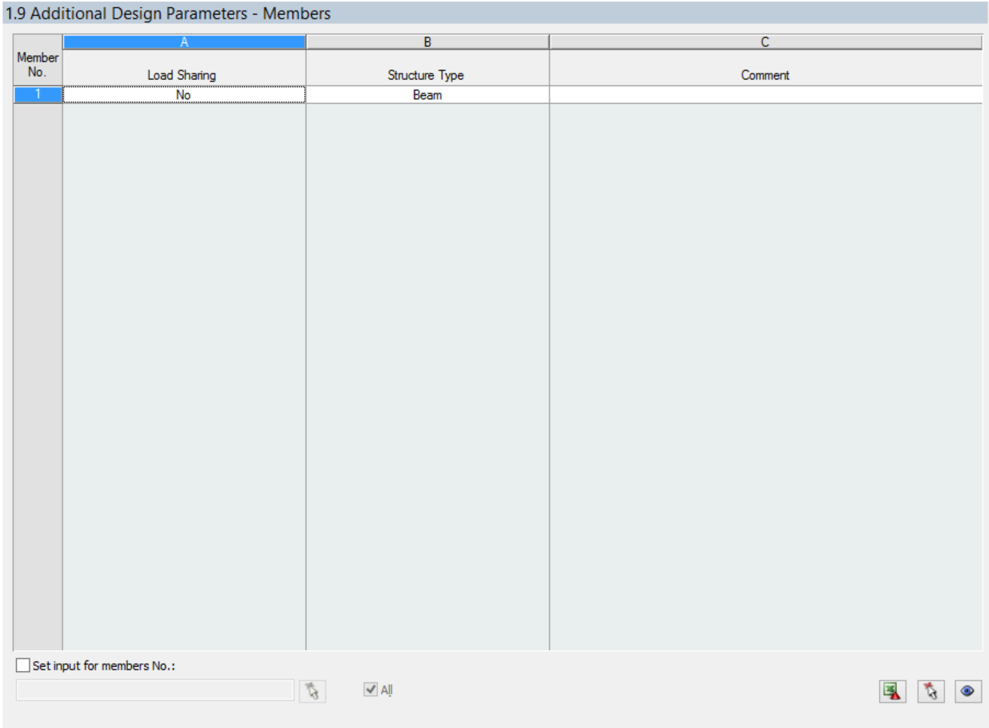
As far as the main axis is concerned (*Buckling About Axis y*), we can leave the default value 3.0 m (see [1] Table 15). For the minor axis (*Buckling About Axis z*), however, we can reduce the buckling length to $\mathbf{L_{ez} = 1.50}$ **m** as the column is restrained in the middle of the member. We can also deactivate *Lateral-Torsional Buckling* because the slenderness factor C_S does not exceed 10 (see [1] Chapter 7.2.3.2.3 a)).


1.7 Effective Lengths - Members

Member No.	A Buckling Possible	B Buckling About Axis y Possible	C K_{ey}	D L_{ey} [m]	E Buckling About Axis z Possible	F K_{ez}	G L_{ez} [m]	H Lateral-Torsional Buckling Possible	I Define L_e	J L_e [m]	K Comment
1	☑	☑	1.000	3.000	☑	0.500	1.500	☐	As member length	3.000	

Figure 8.8: Window 1.7 *Effective Lengths - Members*

In window 1.9 *Additional Design Parameters - Members*, the use of the load sharing factor k_2 is not required. The structure type of the column is a *Beam*.


1.9 Additional Design Parameters - Members

Member No.	A Load Sharing	B Structure Type	C Comment
1	No	Beam	

Figure 8.9: Window 1.9 *Additional Design Parameters - Members*

Calculation

Then we start the [Calculation].

Results

After the calculation, the governing design is presented in the 2.1 *Design by Load Case* Window. All values you need to comprehend the calculation are listed in the lower part of this window.

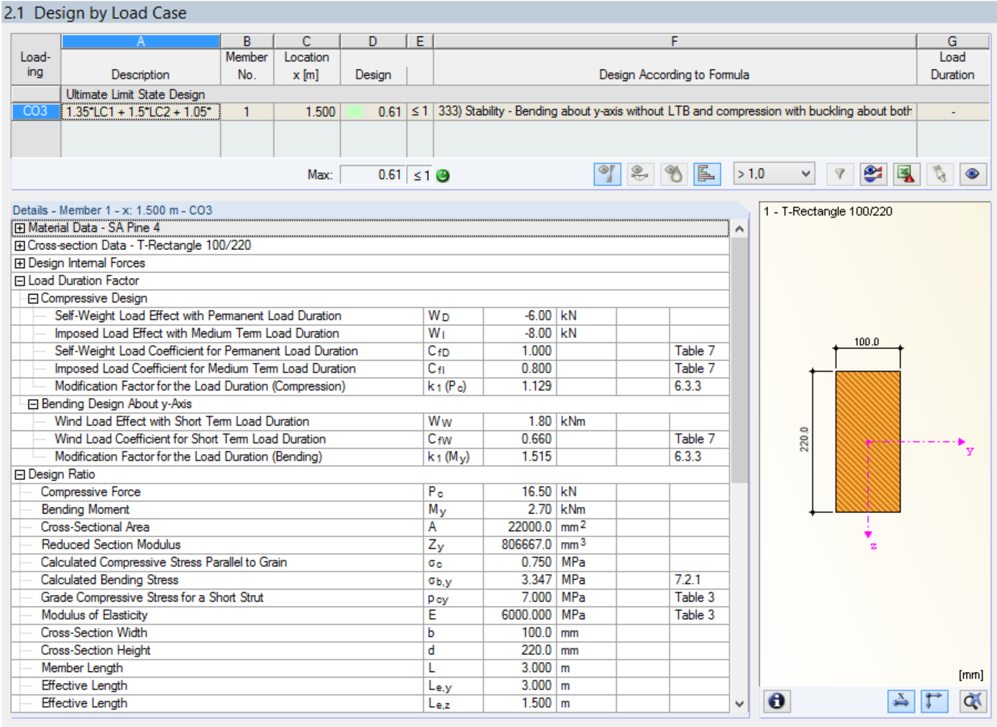


Figure 8.10: Window 2.1 *Design by Load Case*

Load Duration Factor

The table below shows the internal forces of the load cases, without partial safety factors, that are included in combination CO3:

Loading	Load Effect	Load Duration	Coefficients		N [kN]	V_z [kN]	M_y [kNm]
LC1	Self-weight	Permanent	C_{fD}	1.00	**-6.0**	0	0
LC2	Wind	Short Term	C_{fW}	0.66	0	**2.40**	**1.8**
LC3	Imposed	Medium Term	C_{fI}	0.80	**-8.0**	0	0

$$k_1(P_c) = \frac{W_D + W_I}{C_{fD} \times W_D + C_{fI} \times W_I} = \frac{6.0 + 8.0}{1.0 \times 6.0 + 0.8 \times 8.0}$$

$$k_1(P_c) = 1.129$$

$$k_1(M_y) = \frac{W_W}{C_{fW} \times W_W} = \frac{1.80}{0.66 \times 1.80}$$

$$k_1(M_y) = 1.515$$

$$k_1(V_z) = \frac{W_W}{C_{fW} \times W_W} = \frac{2.40}{0.66 \times 2.40}$$

$$k_1(V_z) = 1.515$$

Allowable Compressive Stress

$$f_c = p_c \times k_1(P_c) \times k_2 \times k_3 \times k_4 \times k_5$$

Euler Stress

$$p_{e,y} = \frac{\pi^2 \times \frac{E}{2.22}}{\lambda_y^2} \qquad p_{e,z} = \frac{\pi^2 \times \frac{E}{2.22}}{\lambda_z^2}$$

$$p_{e,y} = \frac{\pi^2 \times \frac{6000}{2.22}}{47.24^2} \qquad p_{e,z} = \frac{\pi^2 \times \frac{6000}{2.22}}{51.96^2}$$

$$p_{e,y} = 11.954 \text{ MPa} \qquad p_{e,z} = 9.879 \text{ MPa}$$

Grade Compressive Stress

$$p_c = \frac{p_{cy} + (\eta + 1) \times p_e}{2} - \sqrt{\left[\frac{p_{cy} + (\eta + 1) \times p_e}{2}\right]^2 - p_{cy} \times p_e}$$

$$p_{c,y} = \frac{7.0 + (0.09 + 1) \times 11.954}{2} - \sqrt{\left[\frac{7.0 + (0.09 + 1) \times 11.954}{2}\right]^2 - 7.0 \times 11.954}$$

$$p_{c,z} = \frac{7.0 + (0.10 + 1) \times 9.879}{2} - \sqrt{\left[\frac{7.0 + (0.10 + 1) \times 9.879}{2}\right]^2 - 7.0 \times 9.879}$$

$$p_{c,y} = 5.900 \text{ MPa}$$

$$p_{c,z} = 5.636 \text{ MPa}$$

$$f_{c,y} = 5.900 \times 1.129 \times 1.0 \times 1.10 \times 1.0 \times 1.0$$

$$f_{c,z} = 5.636 \times 1.129 \times 1.0 \times 1.10 \times 1.0 \times 1.0$$

$$f_{c,y} = 7.327 \text{ MPa}$$

$$f_{c,z} = 7.000 \text{ MPa}$$

Allowable Bending Stress

$$f_{b,y} = p_b \times k_1(M_y) \times k_2 \times k_3 \times k_4$$

$$f_{b,y} = 4.0 \times 1.515 \times 1.0 \times 1.1 \times 1.0$$

$$f_{b,y} = 6.667 \text{ MPa}$$

Allowable Shear Stress

$$f_{v,z} = p_v \times k_1(V_z) \times k_2 \times k_3 \times k_4$$
$$f_{v,z} = 0.4 \times 1.515 \times 1.0 \times 1.1 \times 1.0$$
$$f_{v,z} = 0.667 \, \text{MPa}$$

Combined Bending and Axial Load

Buckling about axis y

$$\frac{\sigma_{b,y}}{f_{b,y}} + \frac{\sigma_c}{f_{c,y}} = \frac{3.347}{6.667} + \frac{0.750}{7.327} = 0.604 \leq 1.0$$

Buckling about axis z

$$\frac{\sigma_{b,y}}{f_{b,y}} + \frac{\sigma_c}{f_{c,z}} = \frac{3.347}{6.667} + \frac{0.750}{7.000} = 0.609 \leq 1.0$$

Shear Design

$$\frac{\sigma_{v,z}}{f_{v,z}} = \frac{0.245}{0.667} = 0.367 \leq 1.0$$

8.1.3.2 Serviceability Limit State Design

In the **Serviceability Limit State** tab of the 1.1 *General Data* Window, we select the load combination **CO8** for design.

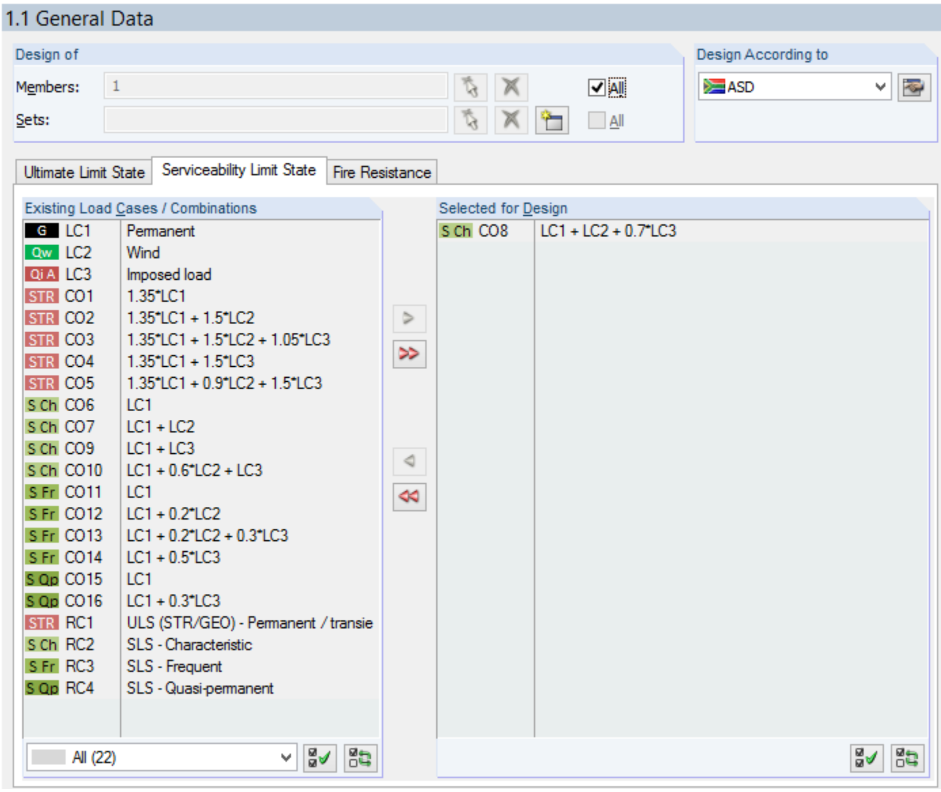


Figure 8.11: Window 1.1 *General Data, tab Serviceability Limit State*

Then we enter member No. **1** in the 1.1 *Serviceability Data* Window.

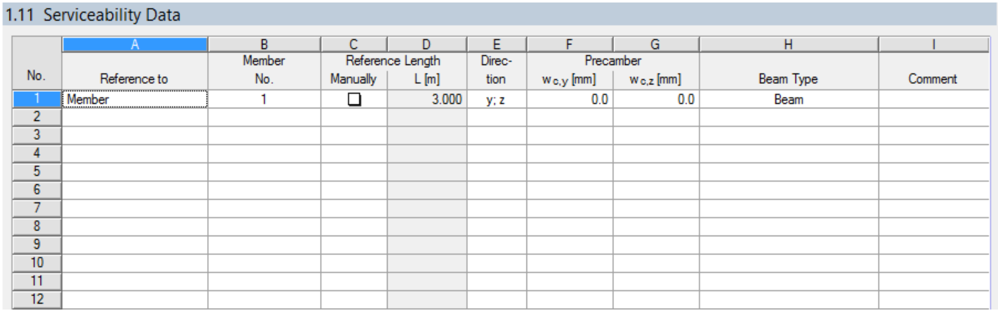


Figure 8.12: Window 1.11 *Serviceability Data*

We do not modify the *Reference Length* but we restrict the *Direction* to **z** (in this case we could leave *y,z* as well). As it is a simple span beam, we select the **Beam** in the *Beam Type* list.

The limiting deflection is *L/300* according to the deflection limits in the *Details* dialog box.

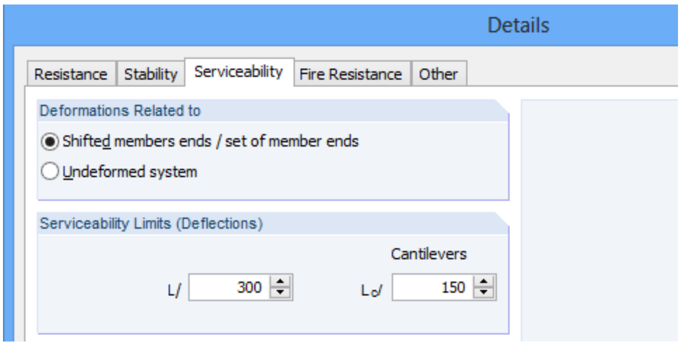


Figure 8.13: Dialog box *Details, tab Serviceability*

In this example, we do not consider the deflection due to shear. Therefore, we deactivate the **shear stiffness** in the *Calculation Parameters* dialog box of RFEM.

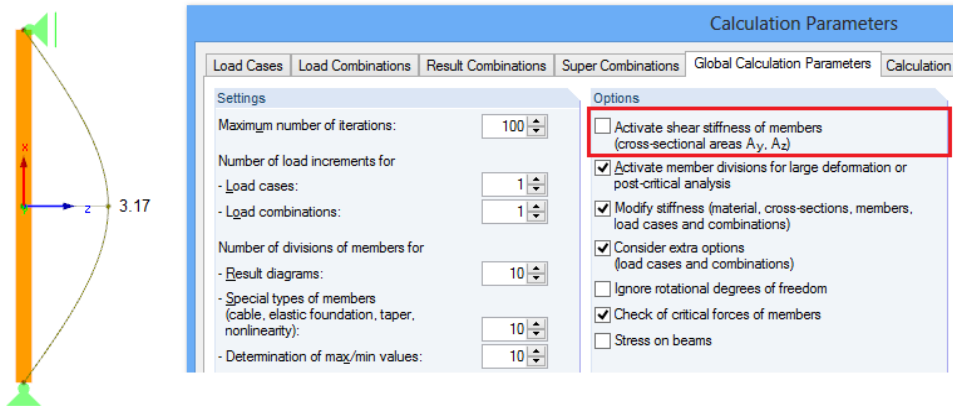


Figure 8.14: Dialog box *Calculation Parameters* in RFEM and deflection of CO8

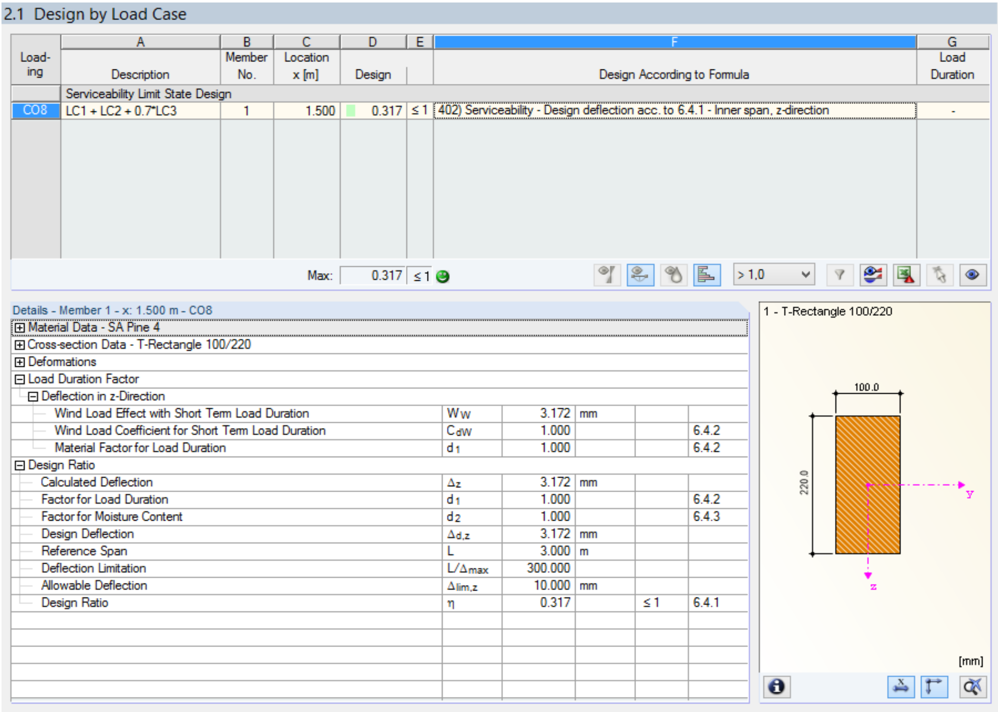
Deflection Design

$$\frac{\Delta_{d,z}}{\Delta_{\lim,z}} \leq 1.0$$

$$\Delta \lim, z = \frac{L}{300} = \frac{3000}{300} = 10.0\,\text{mm}$$

$$\Delta_{d,z} = \Delta_z \times d_1 \times d_2 = 3.17 \times 1.0 \times 1.0 = 3.17\,\text{mm}$$

$$\frac{\Delta_{d,z}}{\Delta_{\lim,z}} = \frac{3.17}{10.0} = 0.317 \leq 1.0$$



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
	Serviceability Limit State Design						
CO8	LC1 + LC2 + 0.7*LC3	1	1.500	0.317	≤ 1	402) Serviceability - Design deflection acc. to 6.4.1 - Inner span, z-direction	-

Max: 0.317 ≤ 1

Details - Member 1 - x: 1.500 m - CO8

⊞ Material Data - SA Pine 4					
⊞ Cross-section Data - T-Rectangle 100/220					
⊞ Deformations					
⊟ Load Duration Factor					
⊟ Deflection in z-Direction					
Wind Load Effect with Short Term Load Duration	W_W	3.172	mm		
Wind Load Coefficient for Short Term Load Duration	C_{dW}	1.000			6.4.2
Material Factor for Load Duration	d_1	1.000			6.4.2
⊟ Design Ratio					
Calculated Deflection	Δ_z	3.172	mm		
Factor for Load Duration	d_1	1.000			6.4.2
Factor for Moisture Content	d_2	1.000			6.4.3
Design Deflection	$\Delta_{d,z}$	3.172	mm		
Reference Span	L	3.000	m		
Deflection Limitation	L/Δ_{max}	300.000			
Allowable Deflection	$\Delta_{lim,z}$	10.000	mm		
Design Ratio	η	0.317		≤ 1	6.4.1

Figure 8.15: Window 2.1 *Design by Load Case*

8.2 Timber Beam (LSD Solution)

We perform the design according to SANS 10163-1 LSD [1] for timber roof purlin that is restrained and subjected to biaxial bending. The beam is laterally supported at this ends only, and there are no intermediate lateral supports. The material is a SA Pine 7.

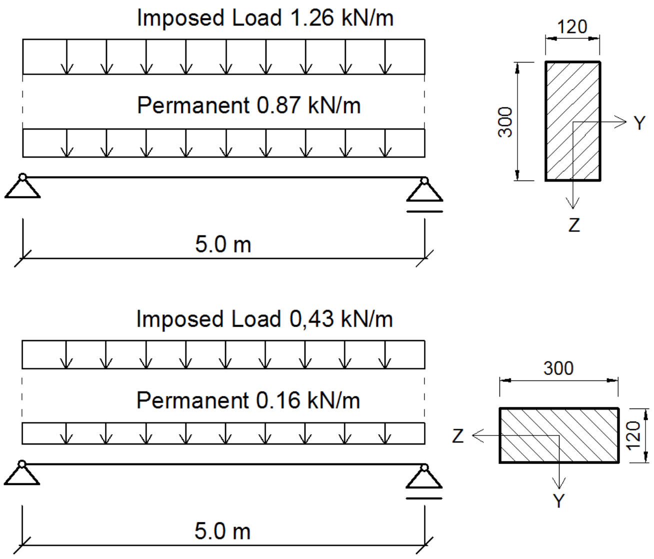


Figure 8.16: System and loads

Model

Cross-section:	b = 100 mm
	d = 220 mm
Material:	SA Pine 7

Load

LC 1 Permanent:	1.3 kN/m
LC 2 Imposed Load:	2.1 kN/m
Load combination ULS:	1.35LC1 + 1.5LC3

8.2.1 Calculation with RFEM

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

We create the load combinations for ULS design with the relevant factors for the defined load cases. Then we calculate the model according to the **linear static analysis**.

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

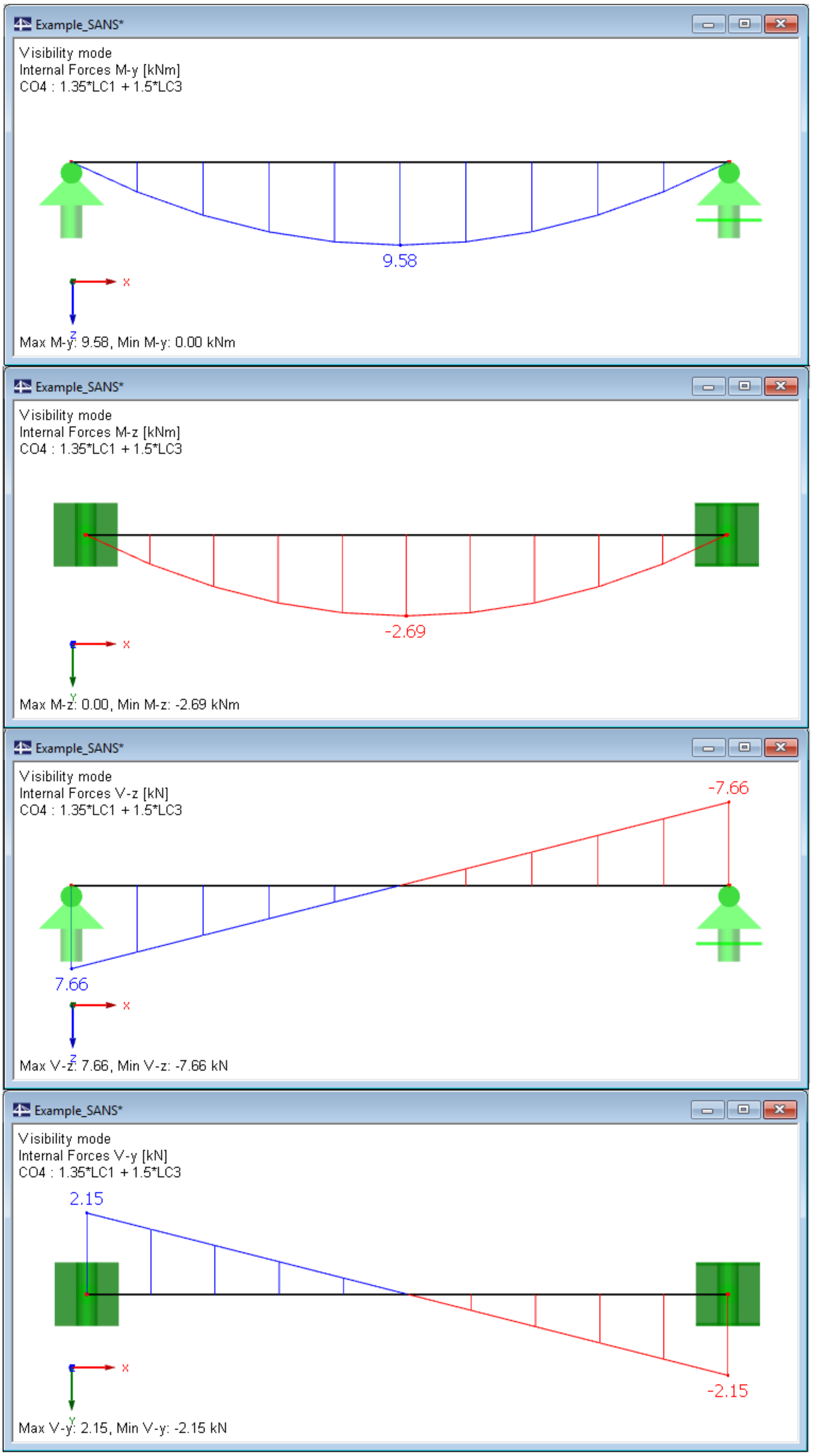


Figure 8.17: Internal forces M_y, M_z, V_z and V_y

8.2.2 Design with RF-TIMBER SANS

8.2.2.1 Ultimate Limit State Design

In Window 1.1 *General Data*, we select combination **CO4** for the **Ultimate Limit State** design.

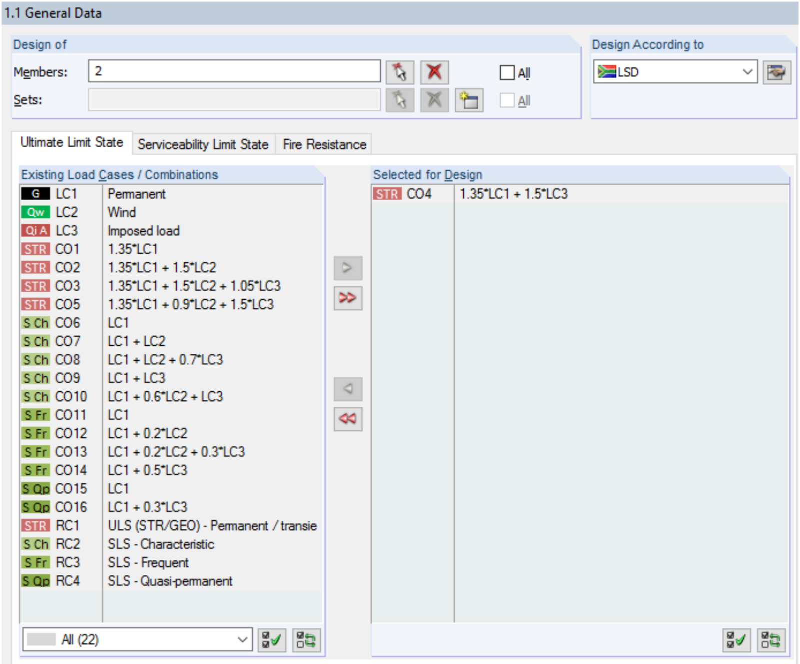


Figure 8.18: Window 1.1 *General Data*

The 1.2 *Materials* Window presents the grade stresses of the selected material.

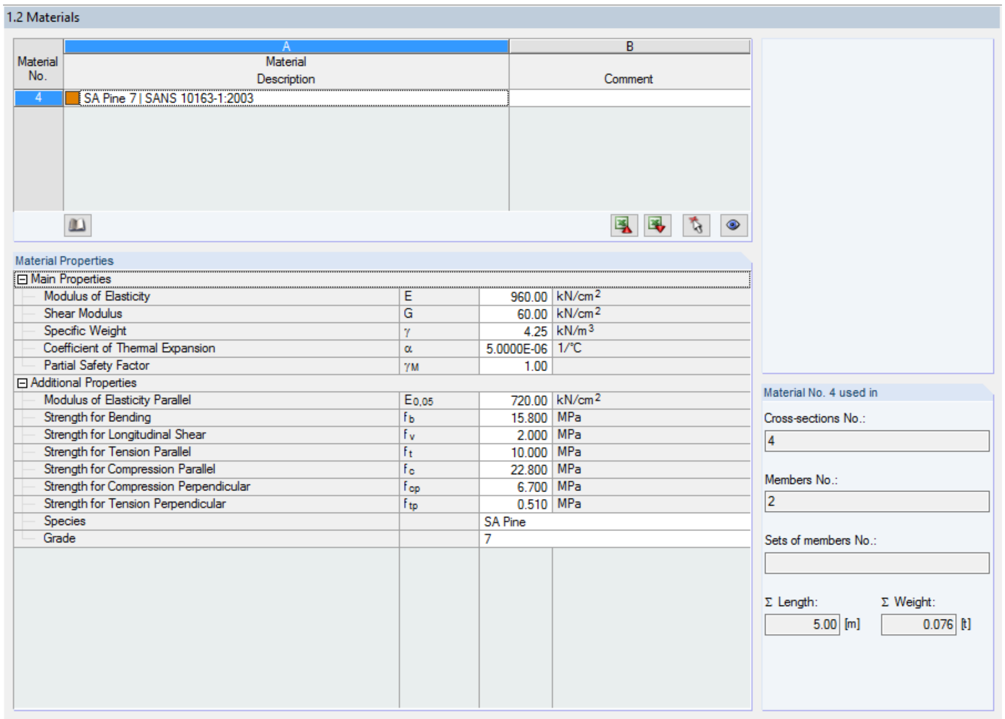


Figure 8.19: Window 1.2 *Materials*

In the 1.3 *Cross-Sections* Window, the parameters of the cross-section can be checked.

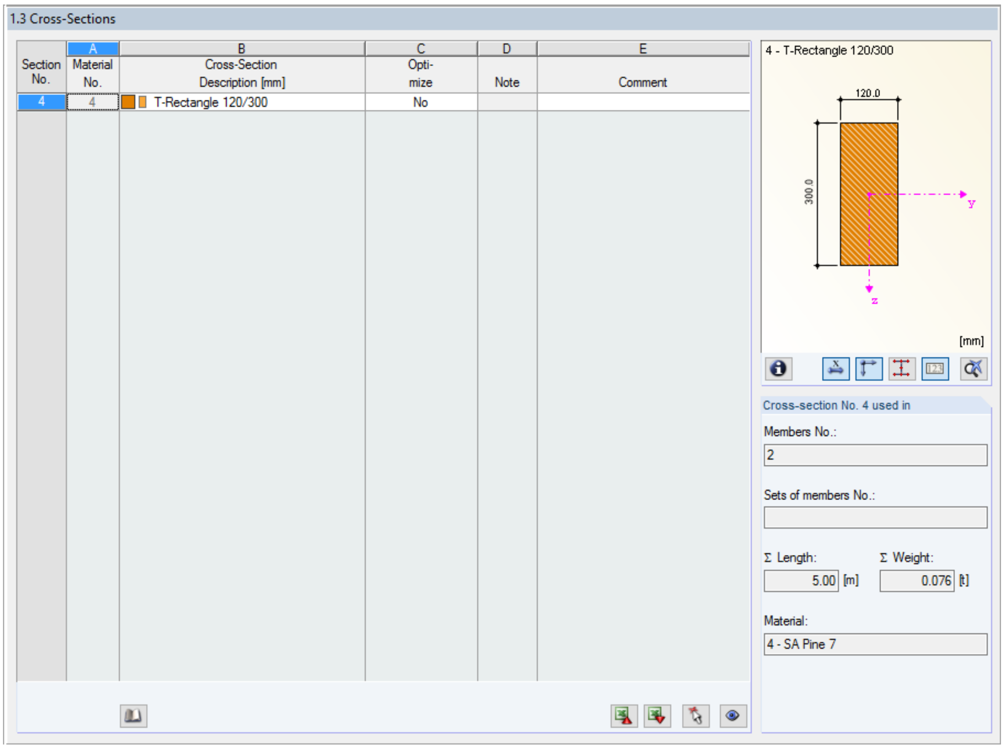


Figure 8.20: Window 1.3 *Cross-Sections*

In Window 1.4 *Load Duration*, we define the load effect and load duration. The load duration factor, γ_{m1}, is determined from the ratio of internal forces at the current location. It may cause different values of the factor at each x-location.

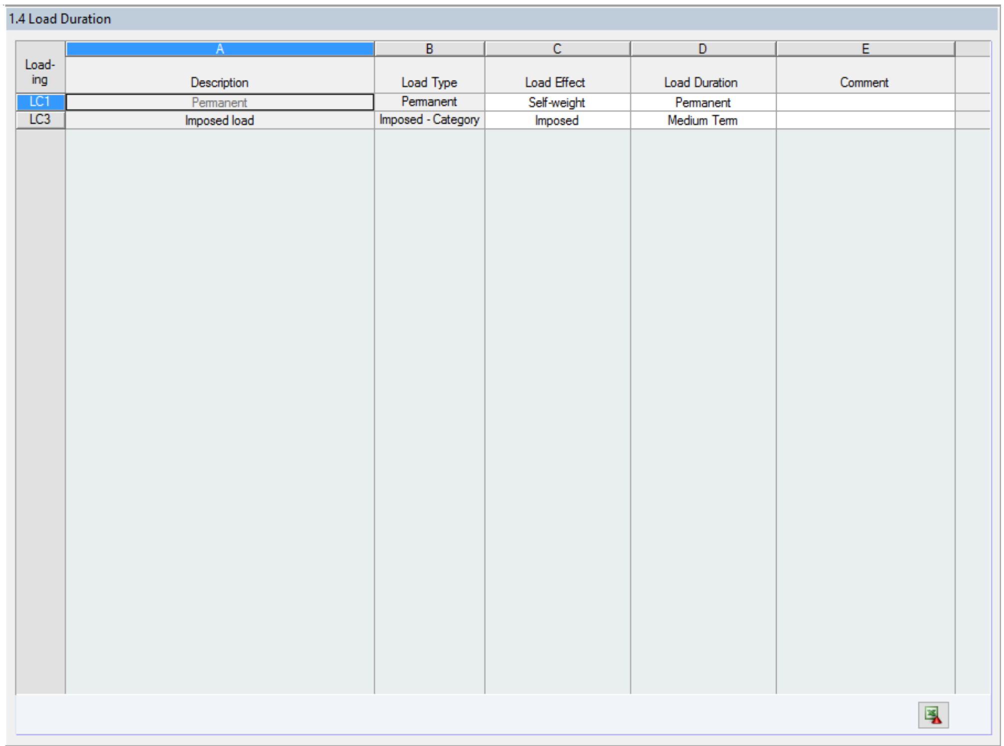
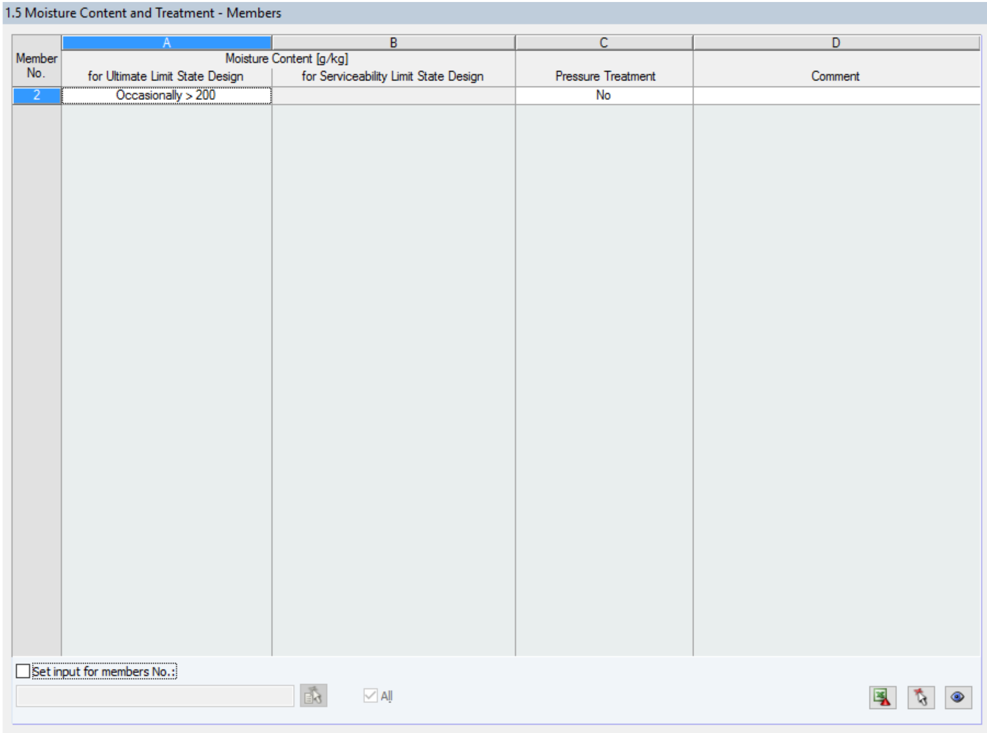


Figure 8.21: Window 1.4 *Load Duration*

In window 1.5 *Moisture Content and Treatment - Members*, we define the moisture content for ULS design. In our case the roof purlin is not pressure treated, therefore the treatment factor, Y_{m5}, is taken as 1.0.



1.5 Moisture Content and Treatment - Members

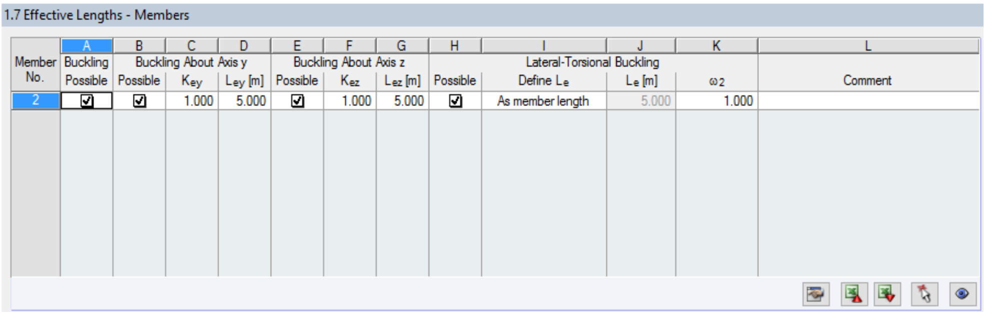
Member No.	A Moisture Content [g/kg] for Ultimate Limit State Design	B for Serviceability Limit State Design	C Pressure Treatment	D Comment
2	Occasionally > 200		No	

Set input for members No.:

All

Figure 8.22: Window 1.5 *Moisture Content and Treatment - Members*

In the 1.7 *Effective Lengths - Members* Window we specify the buckling lengths of the purlin. The beam is braced only at the ends. The effective buckling length for lateral-torsional buckling is defined as member length.



1.7 Effective Lengths - Members

Member No.	A Buckling Possible	B Buckling About Axis y Possible	C K_{ey}	D L_{ey} [m]	E Buckling About Axis z Possible	F K_{ez}	G L_{ez} [m]	H Lateral-Torsional Buckling Possible	I Define L_e	J L_e [m]	K ω_2	L Comment
2	☑	☑	1.000	5.000	☑	1.000	5.000	☑	As member length	5.000	1.000	

Figure 8.23: Window 1.7 *Effective Lengths - Members*

In Window 1.9 *Additional Design Parameters - Members*, the use of the load sharing factor, Υ_{m2}, is not required. The structure type of the purlin is a *Beam*.

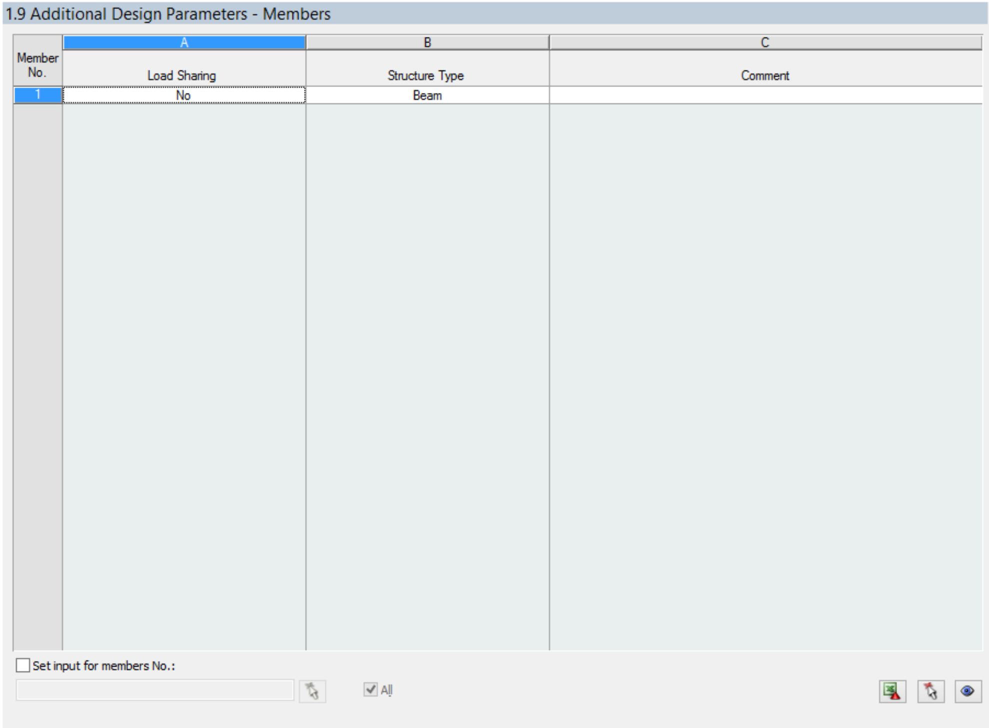


Figure 8.24: Window 1.9 *Additional Design Parameters - Members*

Calculation

Then we start the [Calculation].

Results

After the calculation, the governing design is presented in the 2.1 *Design by Load Case* Window. All values you need to comprehend the calculation are listed in the lower part of this window.

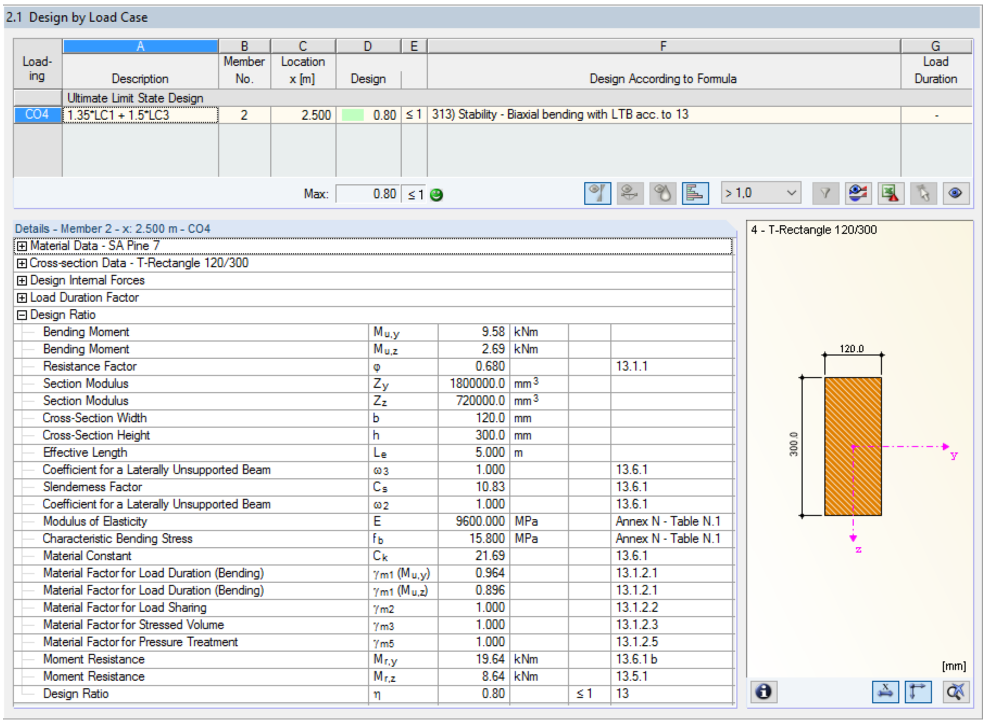


Figure 8.25: Window 2.1 *Design by Load Case*

Load Duration Factor

The table below shows the internal forces of the load cases, without partial safety factors, that are included in combination CO4:

Loading	Load Effect	Load Duration	Coefficients		M_y [kNm]	M_z [kNm]	V_z [kN]	V_y [kN]
LC1	Self-weight	Permanent	C_{fD}	1.26	2.72	-0.50	2.18	0.40
LC3	Imposed	Medium Term	C_{fI}	0.76	3.94	-1.34	3.15	1.08

$$\gamma_{m1}(M_{u,y}) = \frac{C_{fD} \times W_{DU} + C_{fI} \times W_{IU}}{W_{DU} + W_{IU}} = \frac{1.26 \times 2.72 + 0.76 \times 3.94}{2.72 + 3.94}$$

$$\gamma_{m1}(M_{u,y}) = 0.964$$

$$\gamma_{m1}(M_{u,z}) = \frac{C_{fD} \times W_{DU} + C_{fI} \times W_{IU}}{W_{DU} + W_{IU}} = \frac{1.26 \times (-0.50) + 0.76 \times (-1.34)}{-0.50 - 1.34}$$

$$\gamma_{m1}(M_{u,z}) = 0.896$$

$$\gamma_{m1}(V_{u,z}) = \frac{C_{fD} \times W_{DU} + C_{fI} \times W_{IU}}{W_{DU} + W_{IU}} = \frac{1.26 \times 2.18 + 0.76 \times 3.15}{2.18 + 3.15}$$

$$\gamma_{m1}(V_{u,z}) = 0.964$$

$$\gamma_{m1}(V_{u,y}) = \frac{C_{fD} \times W_{DU} + C_{fI} \times W_{IU}}{W_{DU} + W_{IU}} = \frac{1.26 \times 0.40 + 0.76 \times 1.08}{0.40 + 1.08}$$

$$\gamma_{m1}(V_{u,y}) = 0.896$$

Moment resistance for laterally unsupported members

Slenderness Factor

$$C_s = \sqrt{\frac{L_e \times h}{(1-0.1125 \times \omega_3) \times b^2}} = \sqrt{\frac{5.0 \times 0.30}{(1-0.1125 \times 1.0) \times 0.12^2}}$$

$$C_s = 10.83$$

$$C_k = \sqrt{\frac{0.774 \times \omega_2 \times E}{f_b}} = \sqrt{\frac{0.774 \times 1.0 \times 9600}{15.80}}$$

$$C_k = 21.69$$

Moment Resistance M_y :

$$10 < C_s \leq C_k$$

$$M_{r,y} = \varphi \times Z_y \times \left(1 - 0.333 \times \left(\frac{C_s}{C_k}\right)^4\right) \times \frac{f_b}{\gamma_{m1} \times \gamma_{m2} \times \gamma_{m3} \times \gamma_{m5}}$$

$$M_{r,y} = 0.68 \times 1.80 \times \left(1 - 0.333 \times \left(\frac{10.83}{21.69}\right)^4\right) \times \frac{15.80}{0.964 \times 1.0 \times 1.0 \times 1.0}$$

$$M_{r,y} = 19.64\ \text{kNm}$$

Moment Resistance M_z :

$$M_{r,z} = \varphi \times Z_z \times \frac{f_b}{\gamma_{m1} \times \gamma_{m2} \times \gamma_{m3} \times \gamma_{m5}}$$

$$M_{r,z} = 0.68 \times 0.72 \times \frac{15.80}{0.896 \times 1.0 \times 1.0 \times 1.0}$$

$$M_{r,z} = 8.64\ \text{kNm}$$

Shear resistance

$$V_r = 0.67 \times \varphi \times A \times \frac{f_v}{\gamma_{m1} \times \gamma_{m2} \times \gamma_{m3} \times \gamma_{m5}}$$

$$V_{r,z} = 0.67 \times 0.68 \times 36.00 \times \frac{2.0}{0.964 \times 1.0 \times 1.0 \times 1.0}$$

$$V_{r,y} = 0.67 \times 0.68 \times 36.00 \times \frac{2.0}{0.896 \times 1.0 \times 1.0 \times 1.0}$$

$$V_{r,z} = 34.02\ \text{kN}$$

$$V_{r,y} = 36.63\ \text{kN}$$

Biaxial bending design

$$\frac{M_{u,y}}{M_{r,y}}+\frac{M_{u,z}}{M_{u,z}}=\frac{9.58}{19.64}+\frac{2.69}{8.64}=0.80\leq 1.00$$

Shear design

$$\frac{V_{u,z}}{V_{r,z}}=\frac{7.66}{34.02}=0.23\ \leq 1.00$$

$$\frac{V_{u,y}}{V_{u,y}}=\frac{2.15}{36.63}=0.06\ \leq 1.00$$

A Literature

[1] SANS 10163-1: The structural use of timber; Part 1: Limit-states design
Edition 2.3, 2003

[2] SANS 10163-2: The structural use of timber; Part 2: Allowable stress design
Edition 1.1, 2001

B Index