Dlubal

Version
October 2014

Add-on Module

RF-TIMBER CSA

Design of Timber Members
According to CSA O86-09

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 CSA

The Canadian Standard CAN/CSA O86-09 Engineering Design in Wood [1] provides the most comprehensive set of wood design requirements for Canadian conditions, and is referenced by building codes across the country. This Standard employs the limit states design method. With the RFEM add-on module RF-TIMBER CSA from the company DLUBAL all users obtain a powerful tool for the design of timber structures modeled with member elements according to this Standard.

RF-TIMBER CSA 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 analyses require the definition of imperfections in RFEM. The design is performed with unit stability factors for compression with buckling.

In timber construction, the serviceability limit state is an important design. In this connection, you can assign load cases, load combinations, and result combinations individually to various design situations. 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 CSA 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 CSA.

Your DLUBAL Team

1.2 RF-TIMBER CSA - Team

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

Program coordination

Georg Dlubal
Myroslava Petronyuk
Younes El Frem

Programming

Tomáš Drtina
Darya Shatalova
Jiří Patrák

Cross-section and material database

Jan Rybín
Petr Oulehle
Jiří Kubíček
Peter Roth

Program design, dialog figures, and icons

Georg Dlubal
Robert Kolouch
Jan Miléř

Program supervision

Jiří Hanzálek
Gerhard Rehm
Ondřej Švorc
Bastian Kuhn

Localization, manual

Fabio Borriello
Dmitry Bystrov
Rafael Duarte
Lara Freyer
Ladislav Kábrt
Nilton Lopes
José Martínez
Myroslava Petronyuk
Jagoda Podgórna
Chelsea Prokop
Marcela Svitáková
Robert Vogl

Technical support and quality management

Cosme Asseya
Markus Baumgärtel
Sonja von Bloh
Frank Faulstich
René Flori
Walter Fröhlich
Bastian Kuhn
Ulrich Lex
Andreas Niemeier
Gerhard Rehm
Walter Rustler
Frank Sonntag

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 CSA 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 the DLUBAL blogs at **https://www.dlubal.de/blog/en** where you can search the articles by specific terms.

1.4 Open the Add-on Module RF-TIMBER CSA

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

Menu

To start the program in the RFEM menu bar, click

Add-on Modules → Design - Timber → RF-TIMBER CSA.

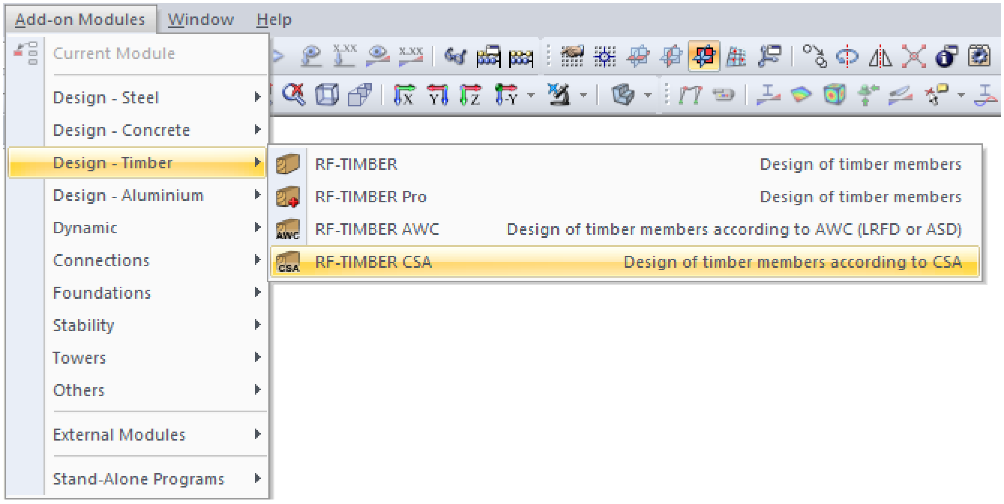


Figure 1.1: Menu: *Add-on Modules → Design - Timber → RF-TIMBER CSA*

Navigator

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

Add-on Modules → RF-TIMBER CSA.

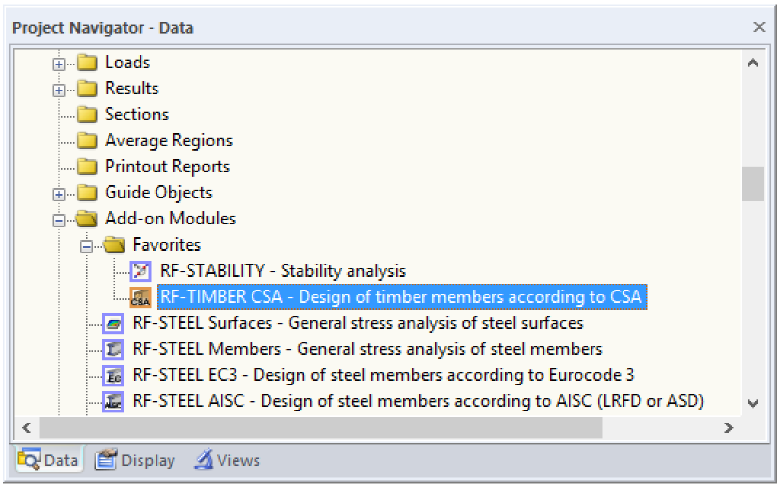
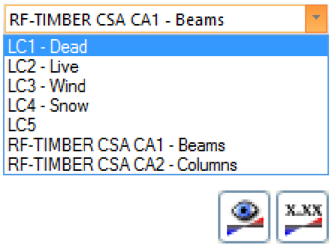


Figure 1.2: Data navigator: *Add-on Modules → RF-TIMBER CSA*



Panel

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

Set the relevant RF-TIMBER CSA 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 CSA] in the panel to open the module.

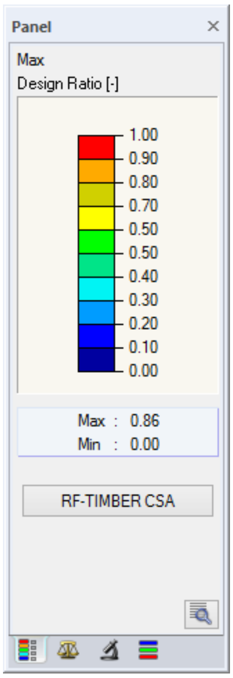


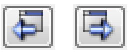
Figure 1.3: Panel button [RF-TIMBER CSA]

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 57).

The design relevant data is defined in several input windows. When you start RF-TIMBER CSA 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 CSA 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.

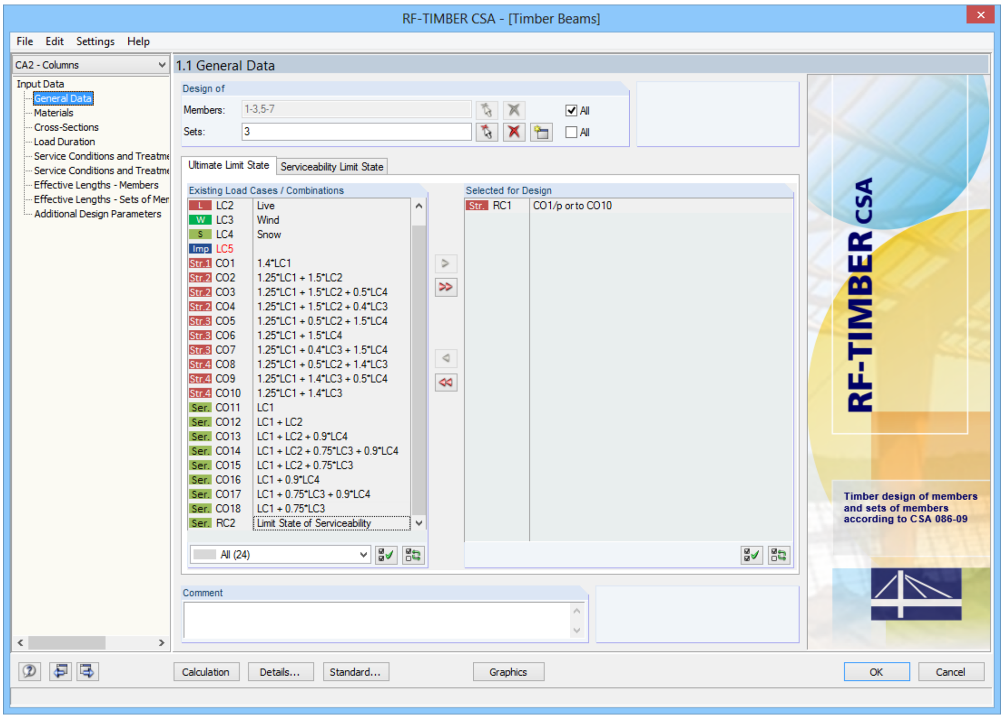


Figure 2.1: Window 1.1 *General Data*

Design of

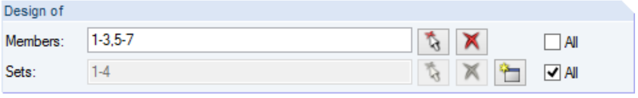


Figure 2.2: Design of members and sets of members

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.

Comment

Figure 2.3: User-defined comment

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

2.1.1 Ultimate Limit State

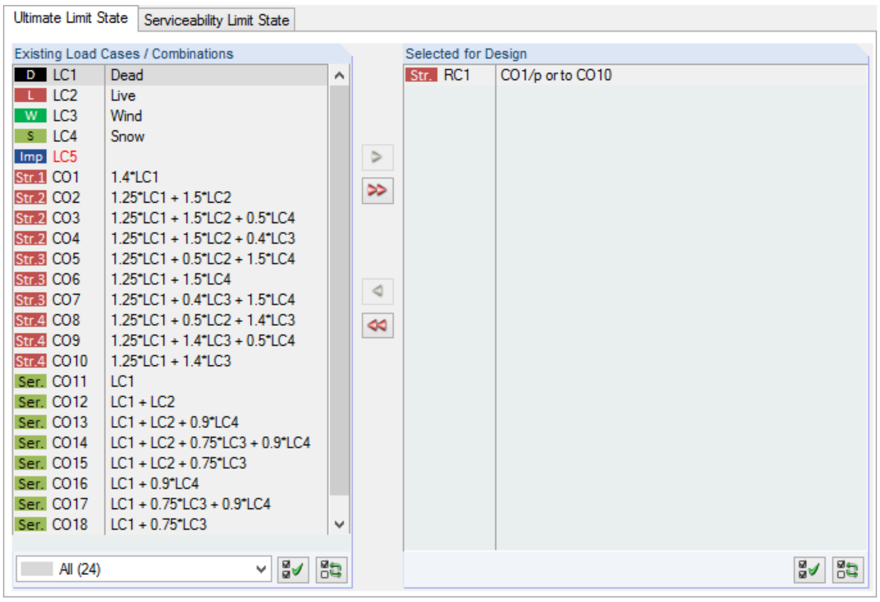


Figure 2.4: Window 1.1 *General Data*, tab *Ultimate Limit State*

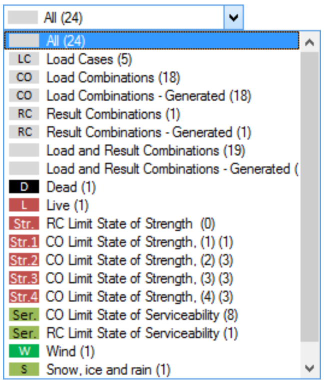
Existing Load Cases / Combinations

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 5 in Figure 2.4, 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.

However, the analysis of a result combination has also disadvantages: First, the influence of the contained actions is difficult to discern. Second, for the determination of the size factor for bending (K_{Zbg}) for glued-laminated timber, the envelope of the moment distributions is analyzed. From that, the most unfavorable distribution (max or min) is applied. However, this distribution only rarely reflects the moment distribution in the individual load combinations. Thus, in the case of RC design, more unfavorable values of the factor K_{Zbg} are to be expected, leading to higher ratios.

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

2.1.2 Serviceability Limit State

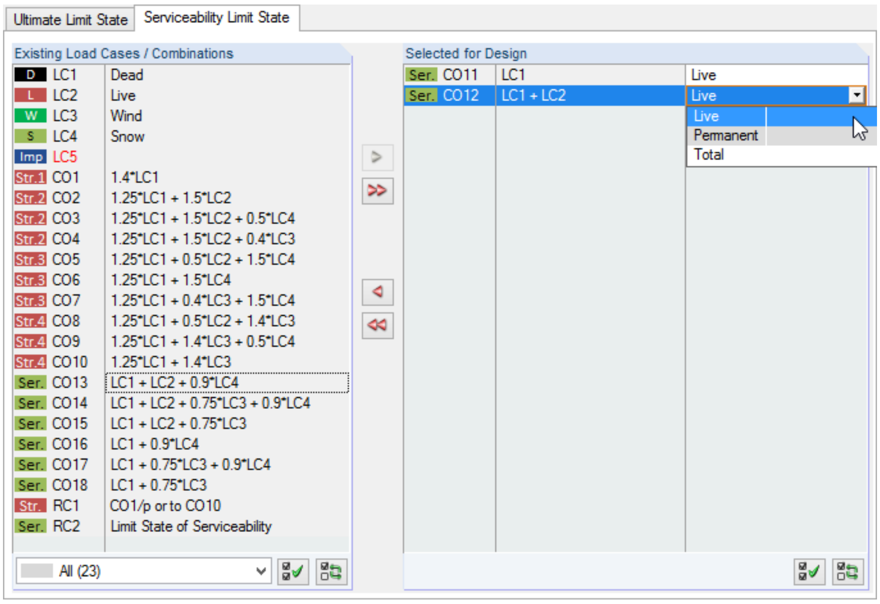


Figure 2.5: Window 1.1 *General Data*, tab *Serviceability Limit State*

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).

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

- Live
- Permanent
- Total

You can change the design situation by using the drop-down list that you can open by clicking [▼] at the end of the box (see Figure 2.5).

Standard...

The limit values of the deformations are specified in the *Standard* dialog box (see Figure 2.8, page 14) which you can call up by clicking the [Standard] button.

In the 1.12 *Serviceability Data* window, the reference lengths decisive for the deformation check are managed (see Chapter 2.12, page 61).

2.1.3 Standard

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 four tabs.

General Factors

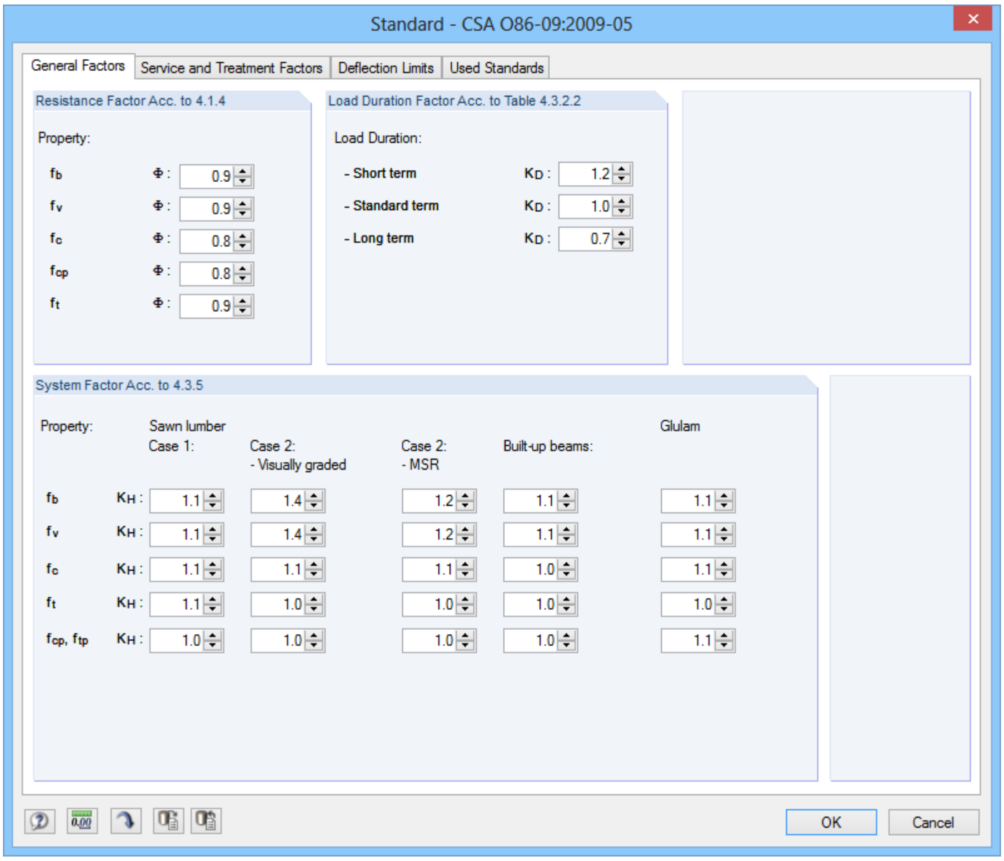


Figure 2.6: Dialog box *Standard*, tab *General Factors*

In the dialog box sections, you can check or, if necessary, modify the *Resistance Factors*, the *Load Duration Factors* or the *System Factors*.

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

Service and Treatment Factors

In the second tab of the *Standard* dialog box, you find *Service Condition Factors* and the *Treatment Factors*. These can be also modified, if necessary.

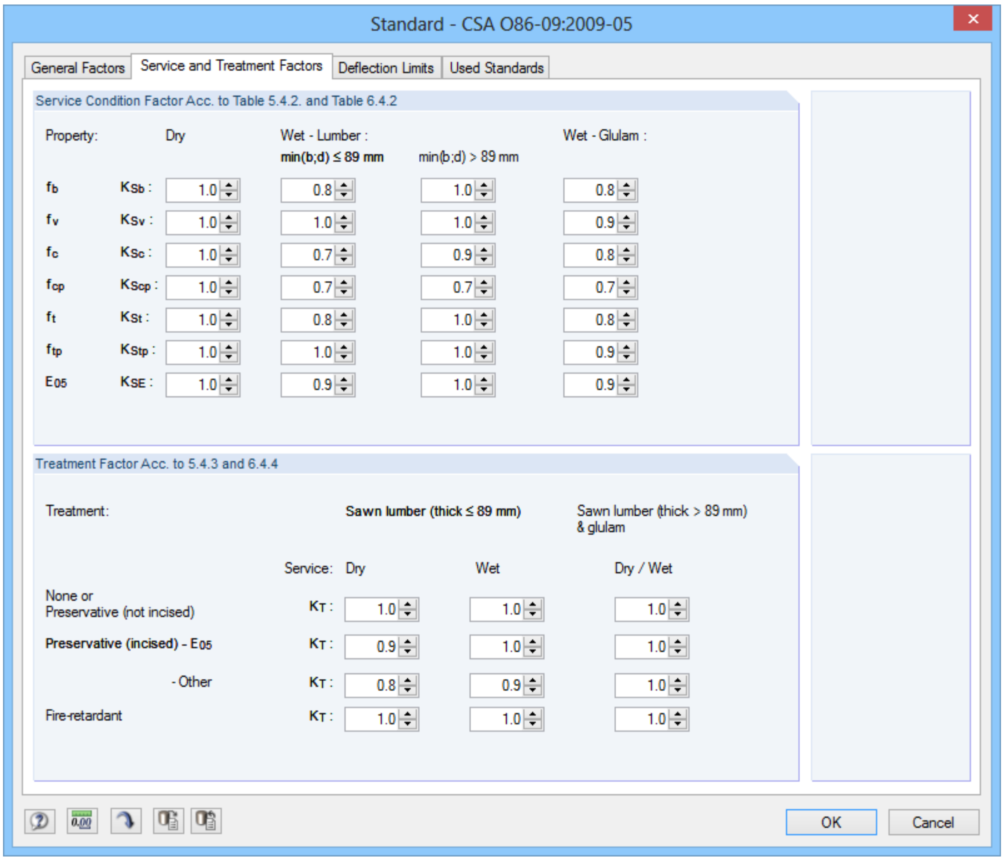


Figure 2.7: Dialog box *Standard*, tab *Service and Treatment Factors*

Deflection Limits

In the third tab, you can define the *Limit Values of Deflection* for the SLS design, depending on the design situation.

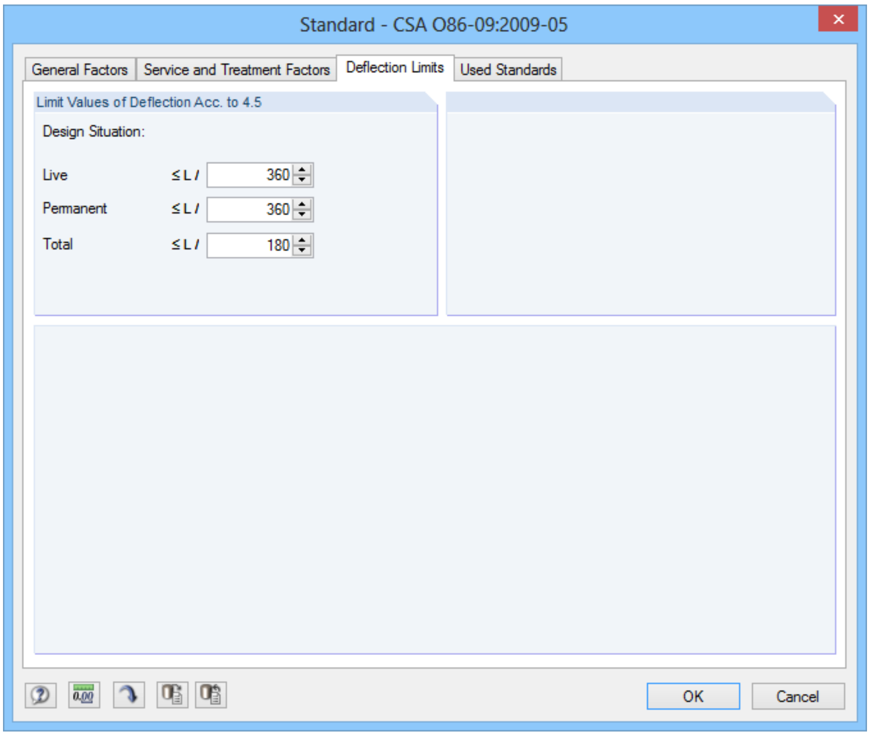


Figure 2.8: Dialog box *Standard*, tab *Deflection Limits*

Used Standards

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

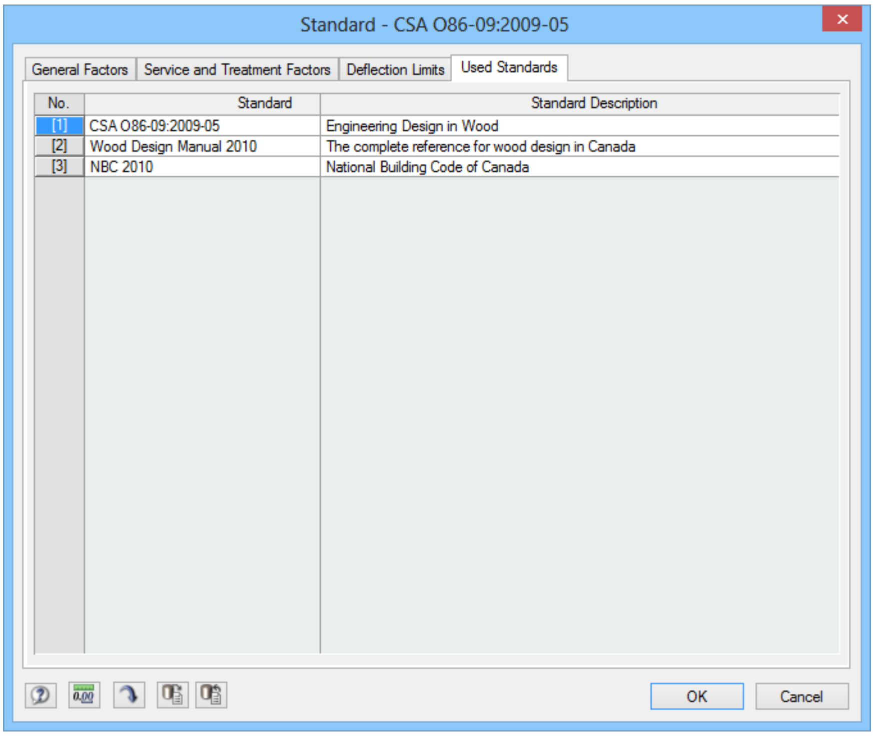


Figure 2.9: Dialog box *Standard*, tab *Used Standards*

2.2 Materials

The window 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.

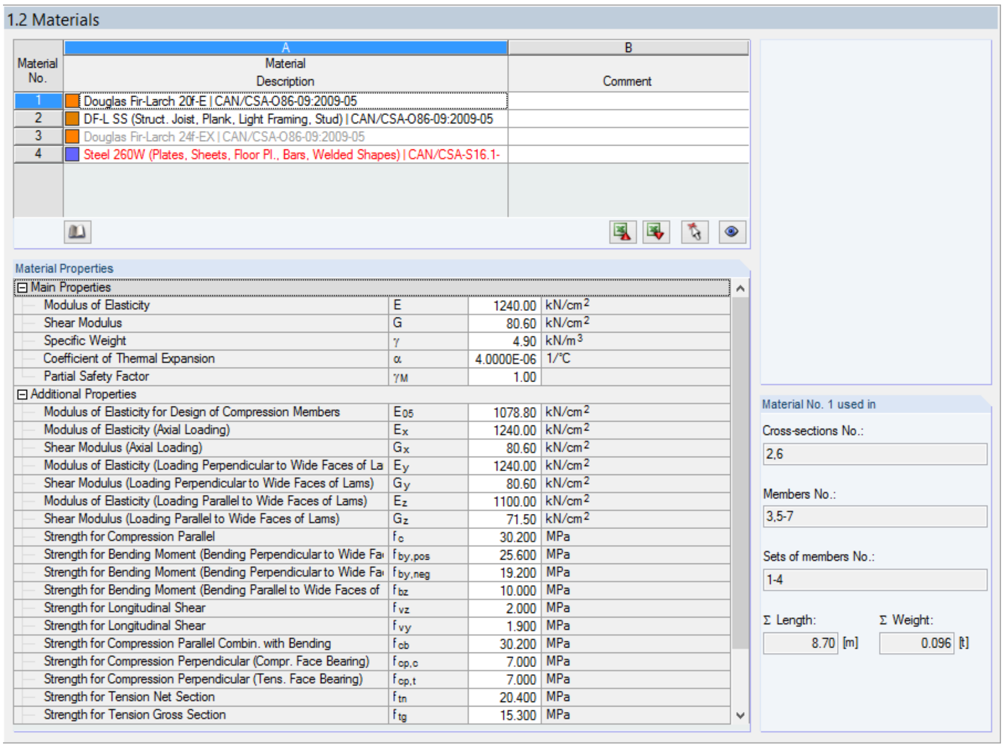


Figure 2.10: Window 1.2 *Materials*

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 61).

Material Description

The materials defined in RFEM are already 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.

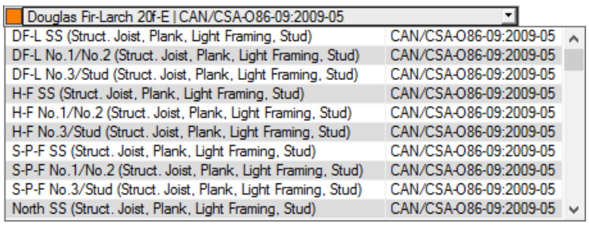


Figure 2.11: List of materials

According to the design concept of CAN/CSA-O86-09:2009-05 [1], the list includes only materials of the Canadian Standard.

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

If you change the material description manually and the entry is stored in the material library, RF-TIMBER CSA will import the material properties, too.

Principally, it is not possible to edit the material properties in the RF-TIMBER CSA module.

Material Library

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

Edit → Material Library

or click the button shown on the left.

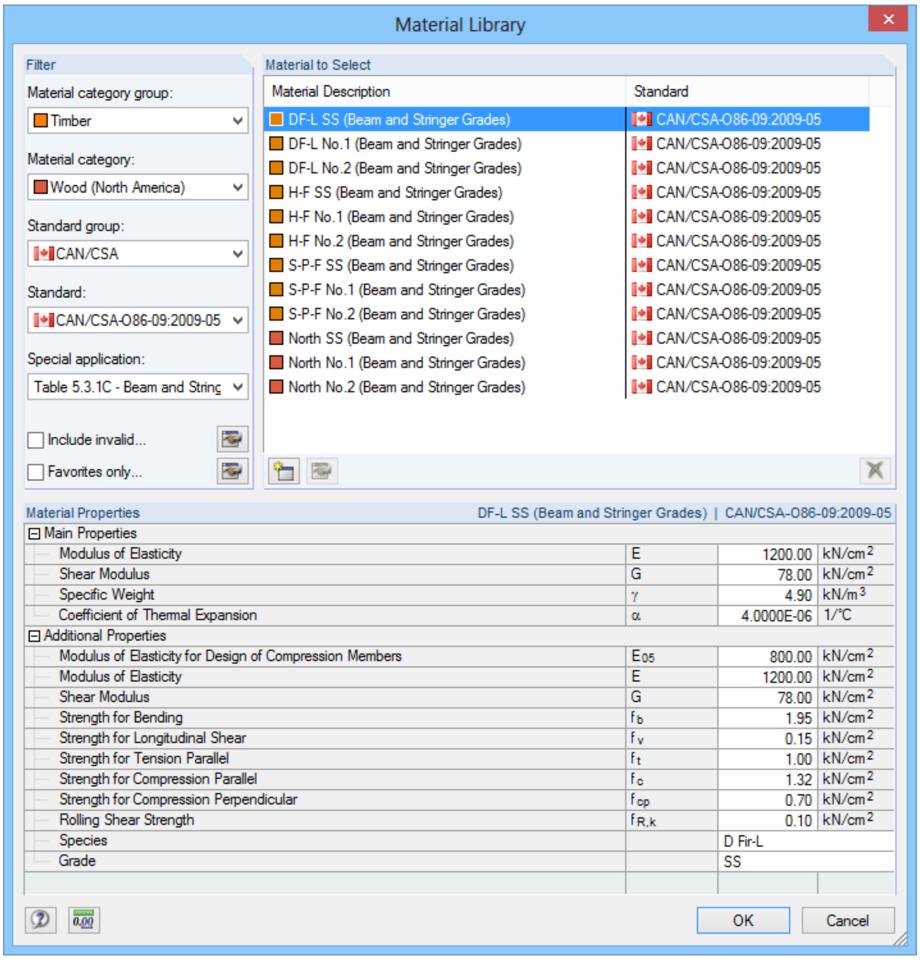


Figure 2.12: Dialog box *Material Library*

In the *Filter* section, *CAN/CSA* 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.

OK

Click [OK] or press [↵] to transfer the selected material to window 1.2 of the module RF-TIMBER CSA.

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

Material Properties

The lower section of window 1.2 contains the specified strength values for bending f_b, tension parallel f_t, tension perpendicular f_{tp}, shear f_v, compression parallel f_c, compression perpendicular f_{cp}, as well as modulus of elasticity for design of compression members E_{05}.

The specified bending and shear strength values and modulus of elasticity for glued-laminated timber (material of [1] Table 6.3 special application) are extended for cases of bending perpendicular and parallel to the wide faces of laminations and bending to f_{by} and f_{bz}. If the narrow faces of laminations are normal to the direction of load, the bending resistance and stiffness for glulam are based on No. 2 grade lumber properties. When structural laminated timber material stressed primarily in bending is selected, the value f_{by} is doubled for the case of positive $f_{by,pos}$ and negative $f_{by,neg}$ bending. In addition, the value f_{cp} is doubled for compression $f_{cp,c}$ and tension $f_{cp,t}$ face of cross-section.

For beams of [1] Table 5.3.1C with special application materials loaded to the wide face, the specified strengths for bending and the specified modulus of elasticity are automatically multiplied by the relevant wide face factor. Please note that this multiplication is used only in the add-on module. The internal forces are not recalculated in RFEM.

Standard...

The design values of the material strengths are to be determined with the modification factors. Those factors can be modified in the *Standard Settings* dialog box (see Figure 2.6 and Figure 2.7, pages 12 and 13).

2.3 Cross-Sections

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

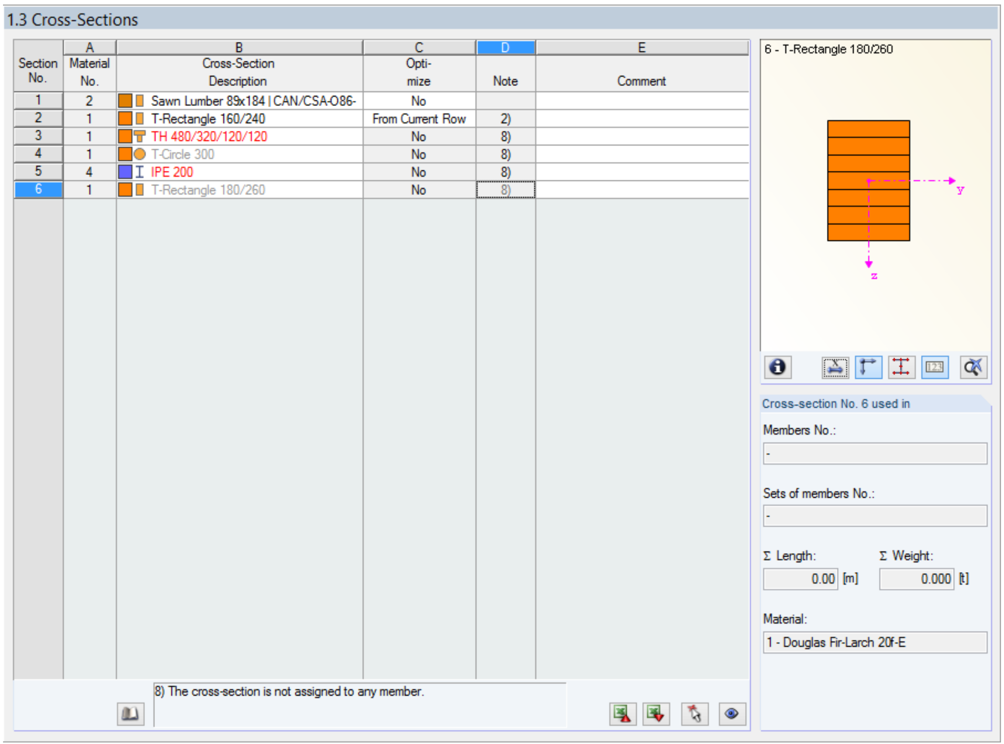


Figure 2.13: Window 1.3 *Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM are preset together with the assigned material numbers.

The design is possible for the parametric timber rectangular and circular cross-section and for standardized timber rectangular cross-section according to the Canadian Standard CAN/CSA O86-09.

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.

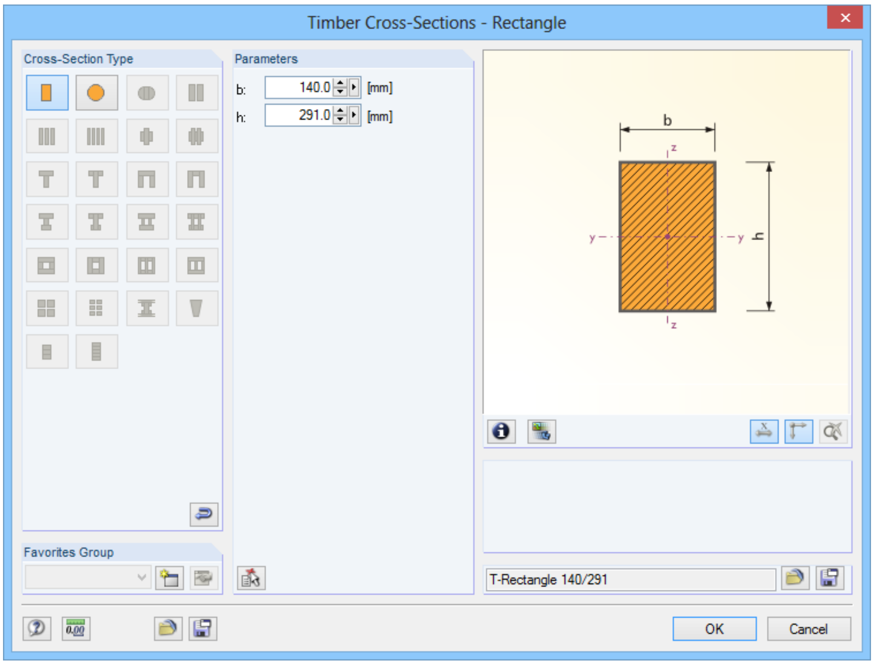


Figure 2.14: Parametric timber cross-sections of the library

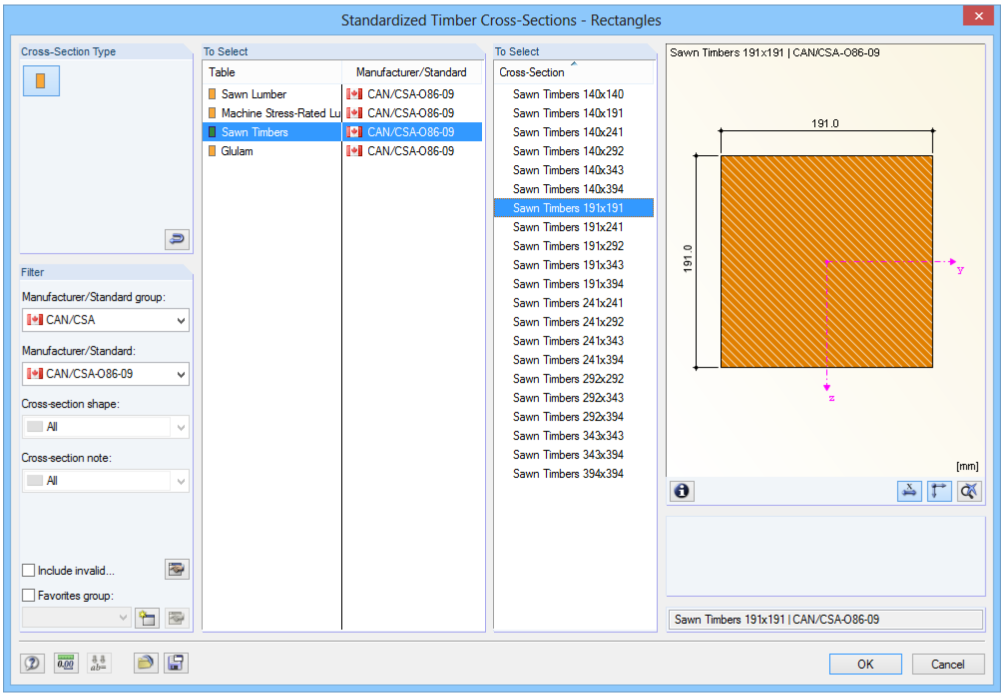
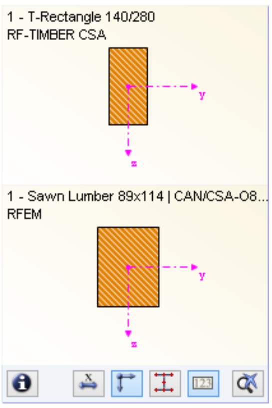


Figure 2.15: Standardized timber cross-sections of the library



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

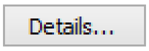
A modified cross-section will be highlighted in blue.

If cross-sections specified in RF-TIMBER CSA 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 CSA.

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



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 3.4, page 37).

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 58.

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.

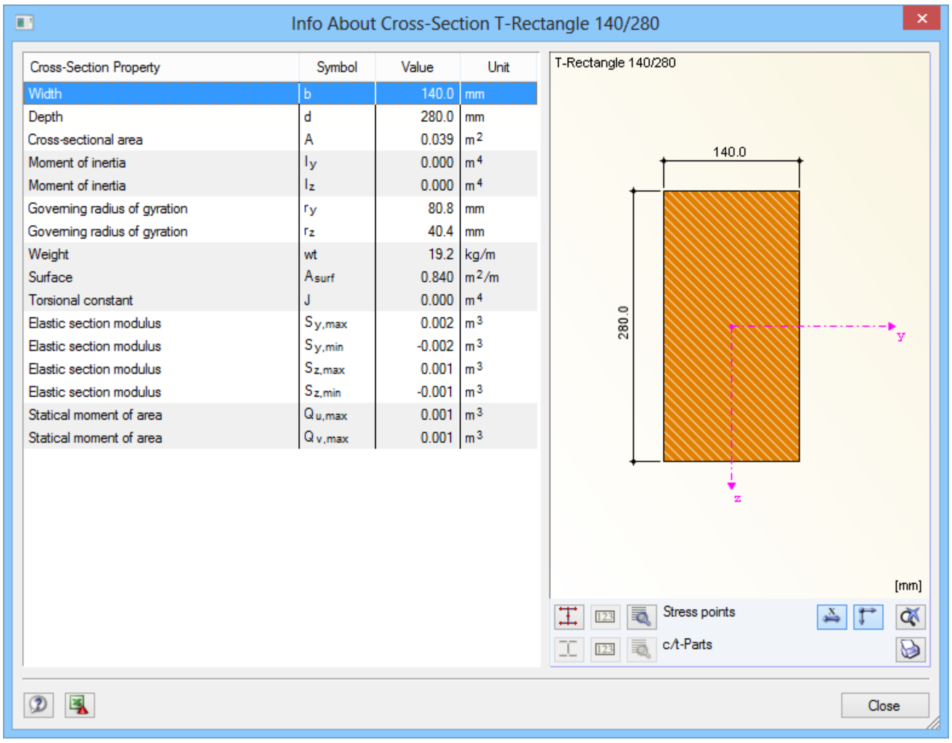


Figure 2.16: Dialog box *Info About Cross-Section*

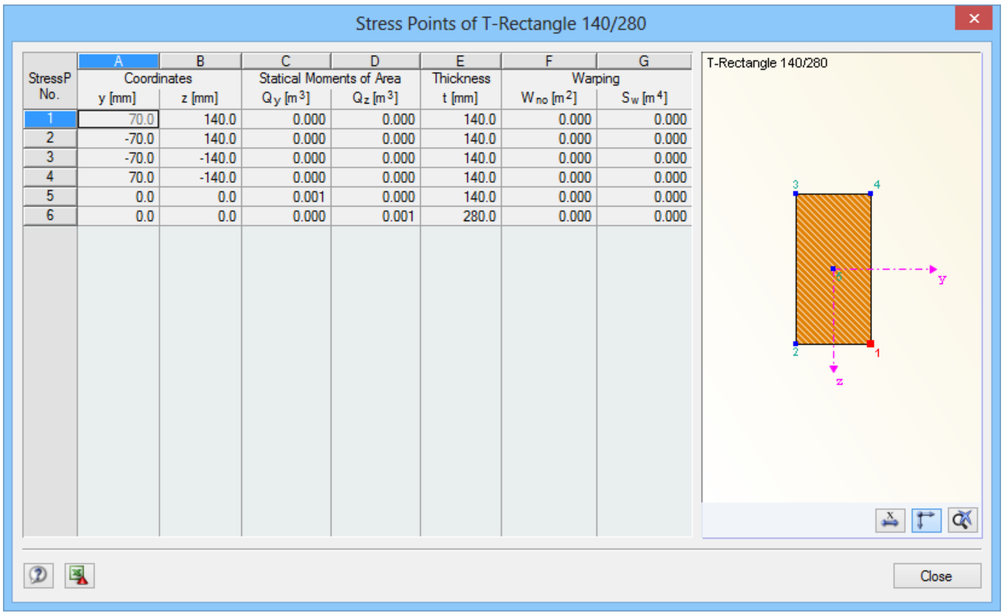
The right part of the dialog box shows the currently selected cross-section.

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 2.17)
	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.).


Stress Points of T-Rectangle 140/280

StressP No.	A Coordinates y [mm]	B z [mm]	C Statical Moments of Area Q_y [m³]	D Q_z [m³]	E Thickness t [mm]	F Warping W_{no} [m²]	G S_w [m⁴]
1	70.0	140.0	0.000	0.000	140.0	0.000	0.000
2	-70.0	140.0	0.000	0.000	140.0	0.000	0.000
3	-70.0	-140.0	0.000	0.000	140.0	0.000	0.000
4	70.0	-140.0	0.000	0.000	140.0	0.000	0.000
5	0.0	0.0	0.001	0.000	140.0	0.000	0.000
6	0.0	0.0	0.000	0.001	280.0	0.000	0.000

Figure 2.17: Dialog box *Stress Points*

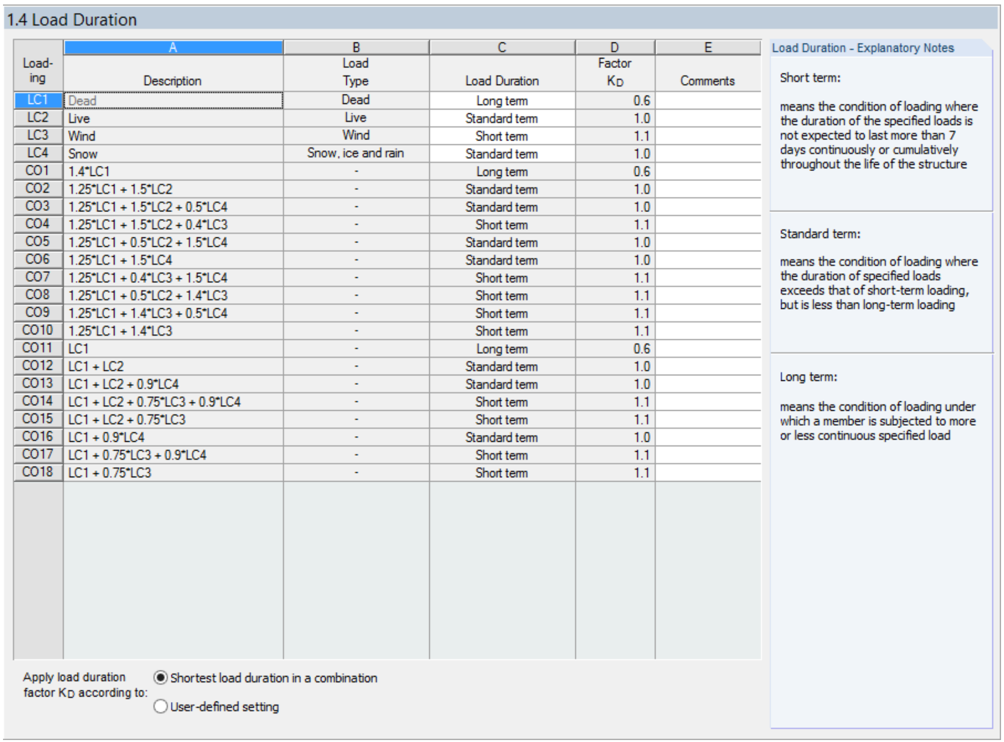
Member with tapered cross-section

For tapered members with different cross-sections at both member ends, the module displays the two cross-sections numbers in separate table rows, in accordance with the definition in RFEM.

RF-TIMBER CSA is also able to design glued-laminated tapered members if the same cross-section type is defined for the start and the end cross-section. Additional specifications are required in Window 1.10 (see Chapter 2.10, page 31).

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, load combinations, result combination, and dynamic combinations.



1.4 Load Duration

Loading	A Description	B Load Type	C Load Duration	D Factor K_D	E Comments
LC1	Dead	Dead	Long term	0.6	
LC2	Live	Live	Standard term	1.0	
LC3	Wind	Wind	Short term	1.1	
LC4	Snow	Snow, ice and rain	Standard term	1.0	
CO1	1.4*LC1	-	Long term	0.6	
CO2	1.25*LC1 + 1.5*LC2	-	Standard term	1.0	
CO3	1.25*LC1 + 1.5*LC2 + 0.5*LC4	-	Standard term	1.0	
CO4	1.25*LC1 + 1.5*LC2 + 0.4*LC3	-	Short term	1.1	
CO5	1.25*LC1 + 0.5*LC2 + 1.5*LC4	-	Standard term	1.0	
CO6	1.25*LC1 + 1.5*LC4	-	Standard term	1.0	
CO7	1.25*LC1 + 0.4*LC3 + 1.5*LC4	-	Short term	1.1	
CO8	1.25*LC1 + 0.5*LC2 + 1.4*LC3	-	Short term	1.1	
CO9	1.25*LC1 + 1.4*LC3 + 0.5*LC4	-	Short term	1.1	
CO10	1.25*LC1 + 1.4*LC3	-	Short term	1.1	
CO11	LC1	-	Long term	0.6	
CO12	LC1 + LC2	-	Standard term	1.0	
CO13	LC1 + LC2 + 0.9*LC4	-	Standard term	1.0	
CO14	LC1 + LC2 + 0.75*LC3 + 0.9*LC4	-	Short term	1.1	
CO15	LC1 + LC2 + 0.75*LC3	-	Short term	1.1	
CO16	LC1 + 0.9*LC4	-	Standard term	1.0	
CO17	LC1 + 0.75*LC3 + 0.9*LC4	-	Short term	1.1	
CO18	LC1 + 0.75*LC3	-	Short term	1.1	

Figure 2.18: Window 1.4 *Load Duration*

Loading

All actions selected in the 1.1 *General Data* window are listed here. For combinations, included load cases are listed, too.

Description

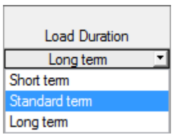
The load case descriptions make the classification easier.

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 Duration

Loads and their superpositions must be assigned to classes of load duration. The classification of actions is specified in [1] Table 2.3.2, for example.



For load cases and variable result combinations, the load duration can be changed by using the list shown on the left: Click the cell in column C, thus selecting the field. The [▼] button becomes available. For load combinations and *Or* result combinations, RF-TIMBER CSA performs the classification automatically taking into account the shortest load duration action of included load cases. When the bottom-side button is switched to *User-defined settings*, load combinations and *Or* result combinations are user-changeable as well.

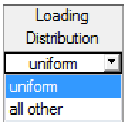
The class of the load duration is required for the determination of the load duration factor, K_D.

Factor K_D

The impact of the load duration on the material strengths is taken into account by means of the load duration factor (see [1] Table 4.3.2.2).

The default values of the factors K_D can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.6, page 12).

Loading Distribution



This column will be added when a radial stress design is to be carried out for tapered or curved members. Those settings can be made in Window 1.10 or 1.11 (see Chapters 2.10 and 2.11).

For the design of tension perpendicular to grain, the distribution of the loading has to be assigned so that the size factor, K_{Ztp}, can be determined (see [1] Table 6.5.6.6.1).

2.5 Service Conditions and Treatment - Members

The determination of moisture and wood treatment makes it possible to assign the service condition factors, K_S, and the treatment factors, K_T, to each member. The moisture service conditions can be specified individually for sawn lumber or glued-laminated timber according to [1] Table 5.4.2 and Table 6.4.2, the treatment factor according to [1] Table 5.4.3 and Chapter 6.4.4.

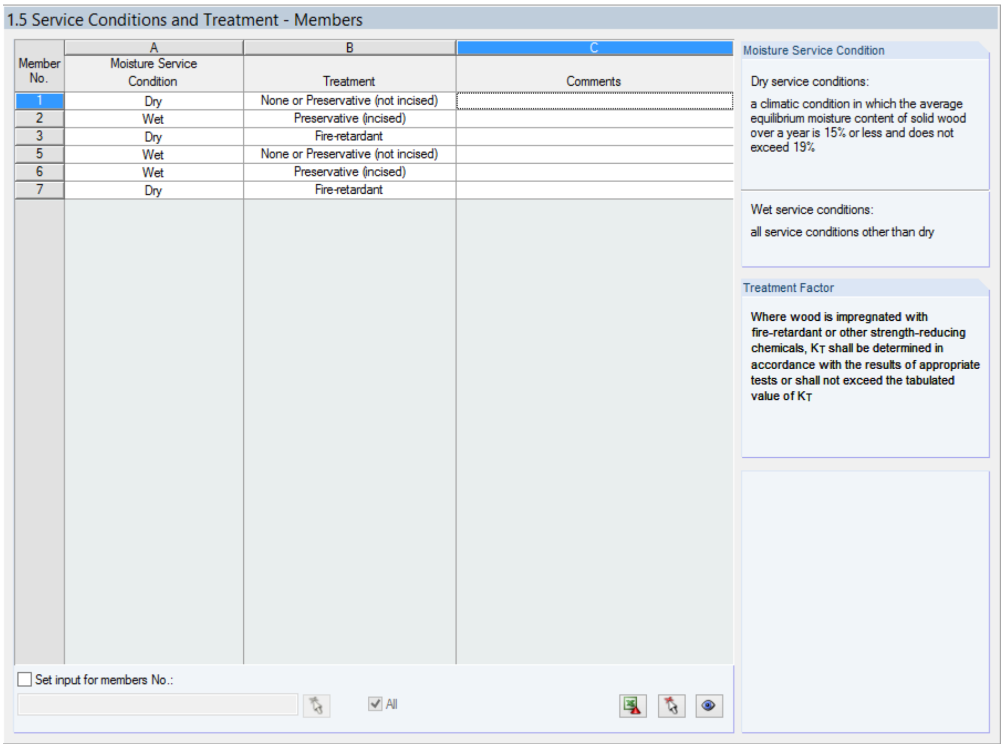
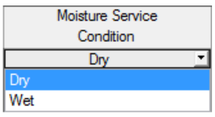
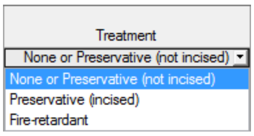


Figure 2.19: Window 1.5 *Service Conditions and Treatment - Members*





By default, RF-TIMBER CSA assigns *Dry* service conditions and *None or Preservative (not incised)* treatment. If you want to define different moisture conditions or treatment to the selected members, use the [▼] button.

The default values of the factors K_S and K_T can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.7, page 13).

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.org Calc
	Directs to the row of member graphically selected in RFEM work window
	Switches into RFEM work window

Table 2.4: Buttons in the window *Service Conditions and Treatment - Members*

2.6 Service Conditions and Treatment - Set of Members

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

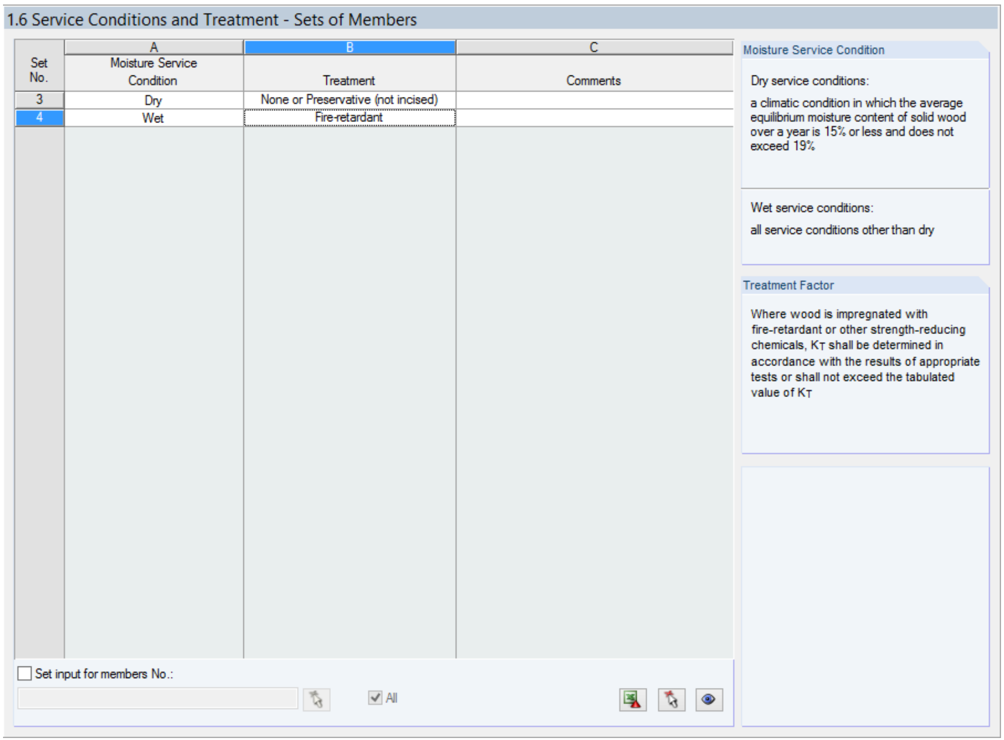


Figure 2.20: Window 1.6 *Service Conditions and Treatment - Sets of Members*

The concept of this window is similar to the one in the previous window 1.5 *Service Conditions and Treatment - Members* window. In this window, you can assign the moisture service conditions and the treatment of the wood to each set of members.

2.7 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 35). 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.

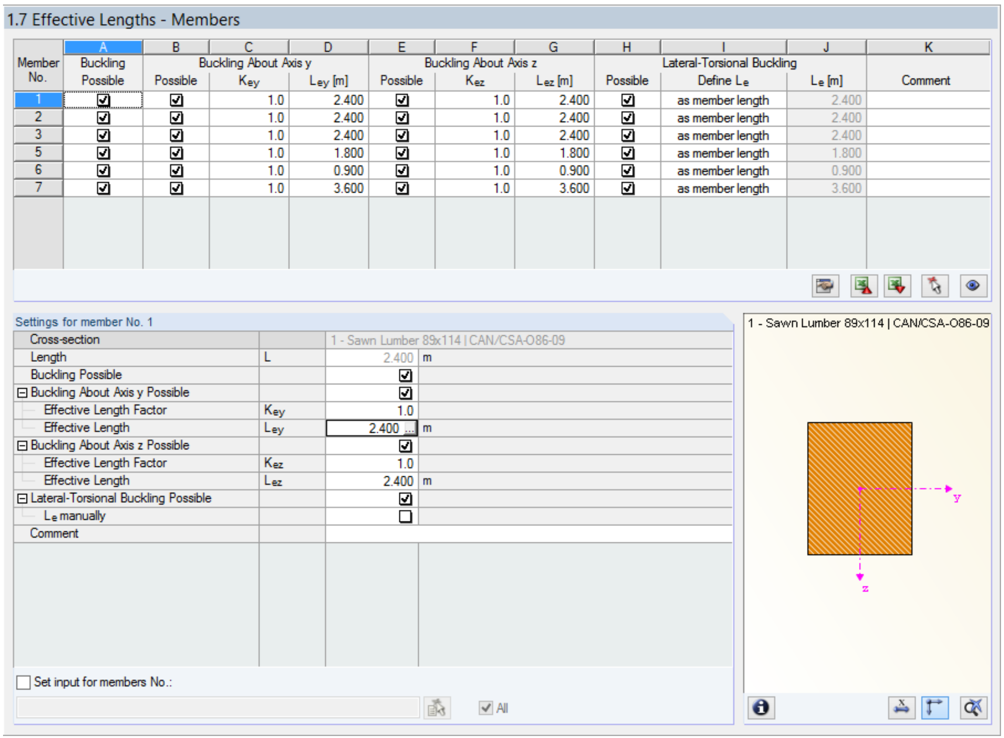


Figure 2.21: Window 1.7 *Effective Lengths - Members* for equivalent member method

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 J)

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.

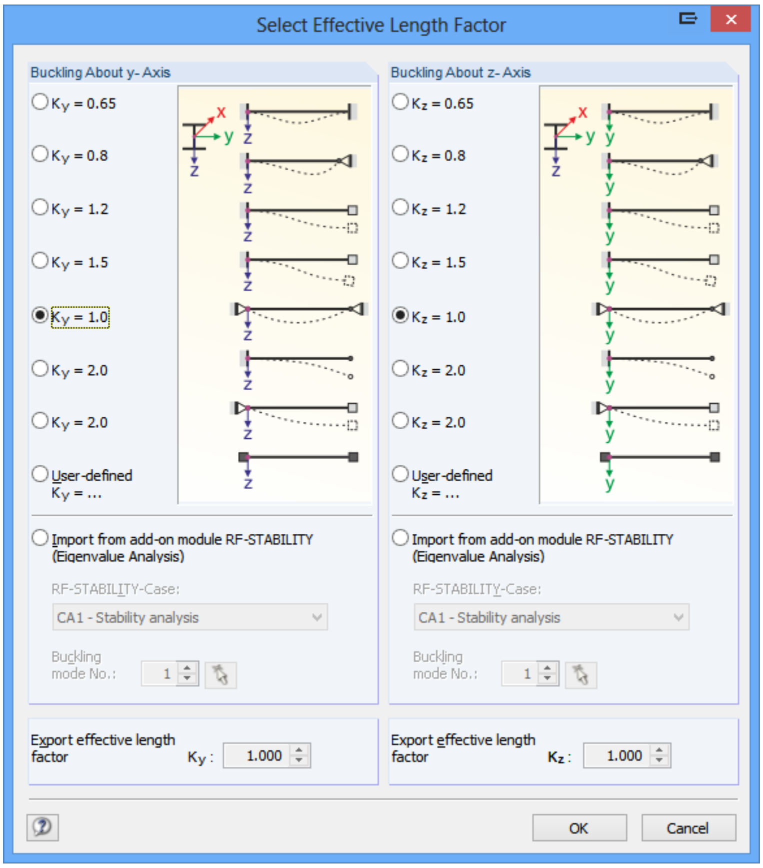


Figure 2.22: Dialog box *Select Effective Length Factor*

For each direction, you can select one of the buckling modes (recommended and theoretical values of buckling length factors as specified in [1] Table A.5.5.6.1) or enter a *User-defined* effective length coefficient *K*.

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

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 design according to [1].

Buckling About Axis y or Axis z

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

You can check the position of the member axes in the cross-section graphic in window 1.3 *Cross-Sections* (see Figure 2.13, page 18). 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 context menu or the *Display* navigator.

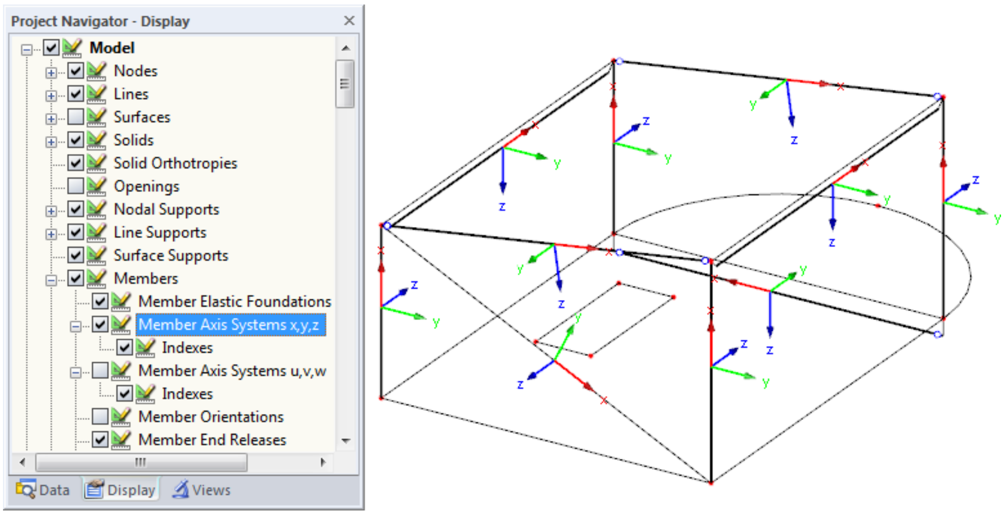


Figure 2.23: Selecting the member axis systems in the *Display* navigator of RFEM

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.

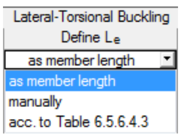
To specify the buckling lengths in the work window graphically, click [...]. This button becomes available when you click in a L_e text box (see Figure 2.21).

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 buckling length coefficient. The text boxes for K_e and L_e are interactive.

Lateral-Torsional Buckling

Table column H controls for which members a lateral-torsional buckling analysis is to be carried out.

Define L_e



The member lengths are preset in column I as equivalent member lengths relevant for lateral-torsional buckling. When you set the option *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 clicking […] 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.

When the option *acc. to Table 6.5.6.4.3* is selected, you can determine the lateral-torsional buckling length in accordance with [1] Table 6.5.6.4.3. A new dialog box is opened in which you can select the effective length according to the loading conditions (see figure below).

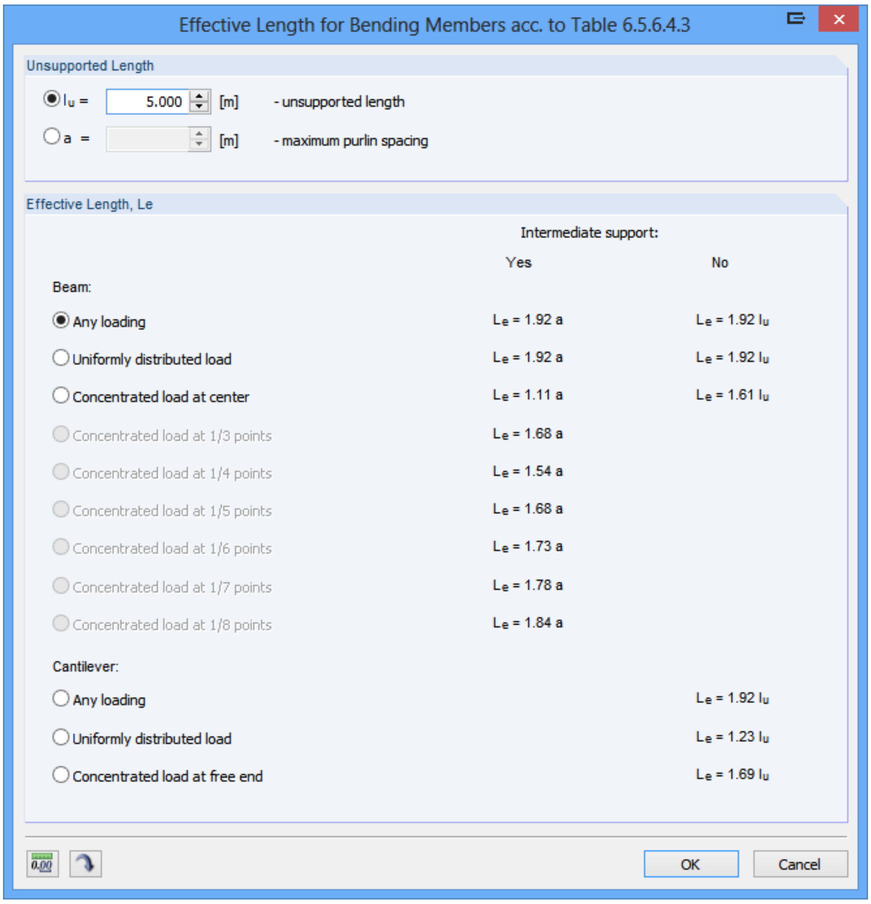


Figure 2.24: Dialog box *Effective Length for Bending Members acc. to Table 6.5.6.4.3*

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.

2.8 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 35).

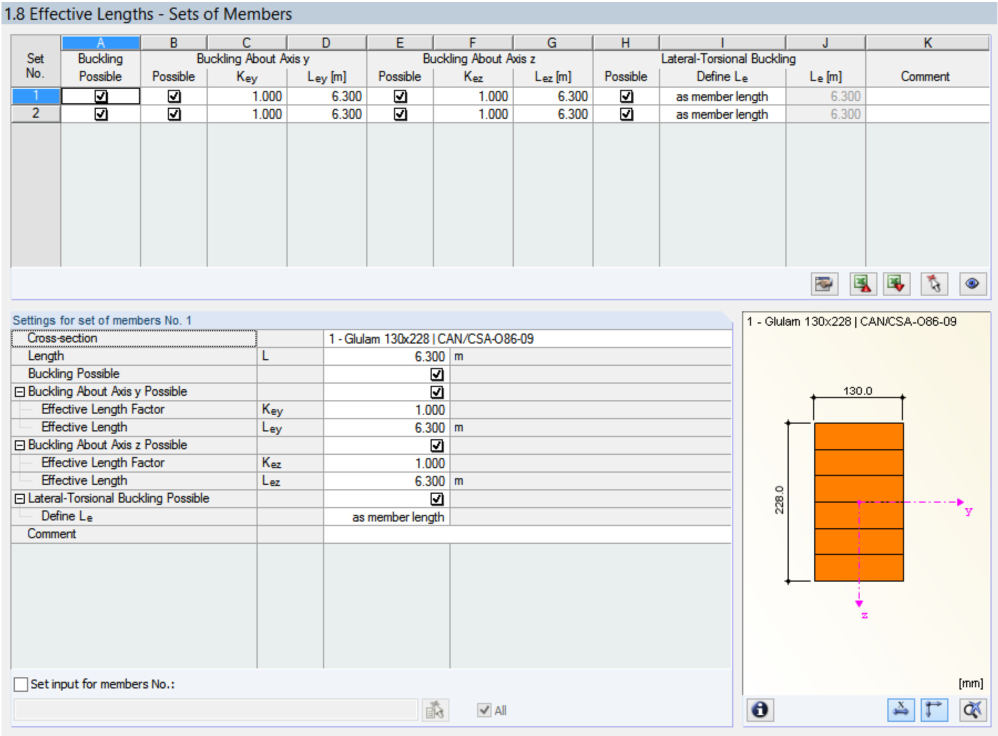
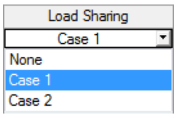


Figure 2.25: Window 1.8 *Effective Lengths - Sets of Members*

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 2.7. They determine the boundary conditions of the entire set of members which is to be treated as an equivalent member.

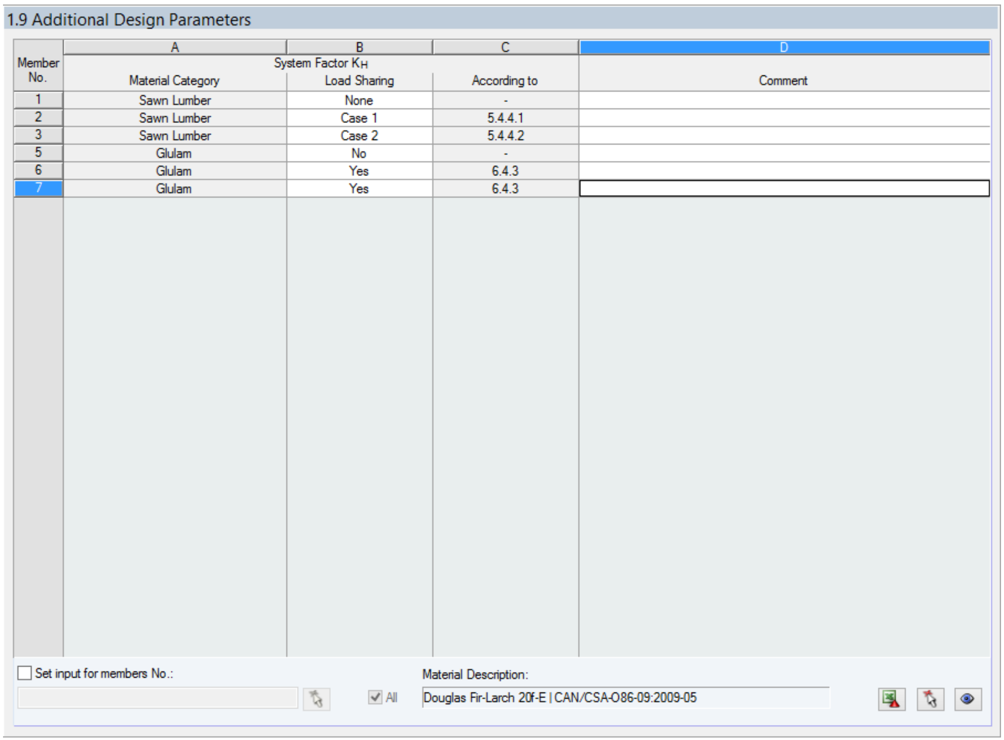
Please note that curved sets of members are excluded from the stability analysis.

2.9 Additional Design Parameters



This window allows you to allocate the system factor, K_H, for each member. This factor depends on the interaction between members in the model and on the material. By clicking the [▼] button in column B, the *Case* according to [1] Chapter 5.4.4 can be selected.

By default, the program assigns *None/No* system factor. The factor can be checked and, if necessary, adjusted in the *Standard* dialog box (see Figure 2.6, page 12).



Member No.	A Material Category	B System Factor K_H Load Sharing	C According to	D Comment
1	Sawn Lumber	None	-	
2	Sawn Lumber	Case 1	5.4.4.1	
3	Sawn Lumber	Case 2	5.4.4.2	
5	Glulam	No	-	
6	Glulam	Yes	6.4.3	
7	Glulam	Yes	6.4.3	

Figure 2.26: Window 1.9 *Additional Design Parameters*

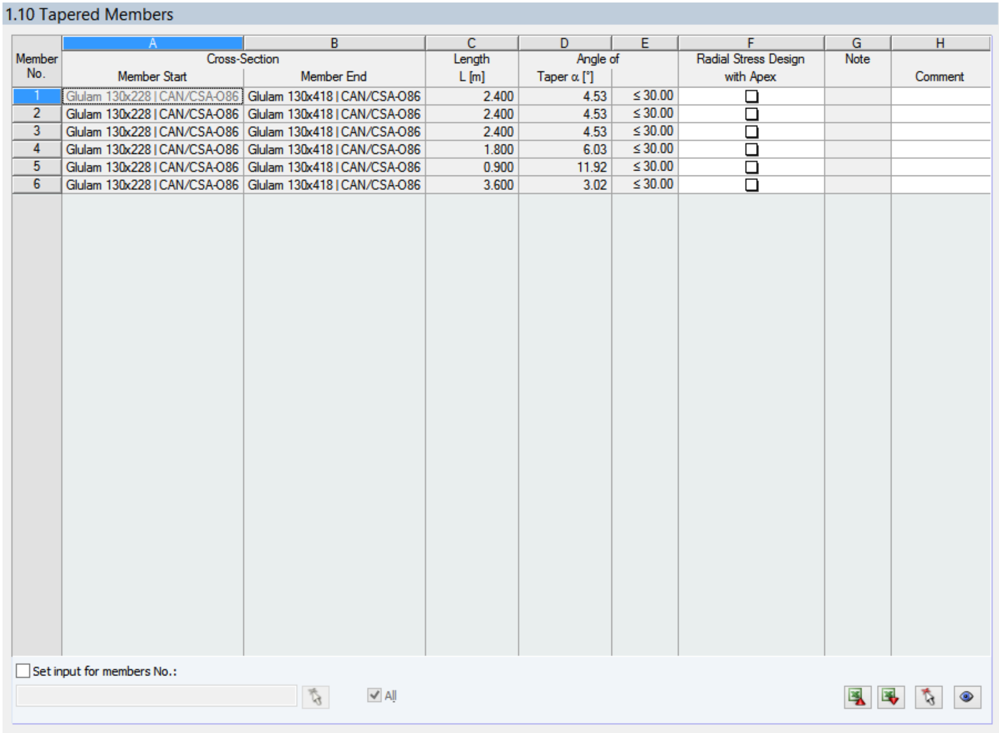
The *Set input for members No.* check box below the table enables you to assign the settings to selected or *All* members (see Chapter 2.5 and Chapter 2.7).

The *Material Description* (also below the table) informs you which material is used for the current member. The buttons to the right of this text box are described in Table 2.4 on page 24.

2.10 Tapered Members

This window is only available when you have selected at least one member with different cross-sections at both member ends for the design in Window 1.1 *General Data*. This window manages criteria such as the angle of taper of variable cross-sections, for example.

The design of tapered members is only possible for **rectangular** cross-sections whose material is **glued-laminated** timber according to [1] Table 6.3.


1.10 Tapered Members

Member No.	A Cross-Section Member Start	B Member End	C Length L [m]	D Angle of Taper α [°]	E	F Radial Stress Design with Apex	G Note	H Comment
1	Glulam 130x228 | CAN/CSA-O86	Glulam 130x418 | CAN/CSA-O86	2.400	4.53	≤ 30.00	☐		
2	Glulam 130x228 | CAN/CSA-O86	Glulam 130x418 | CAN/CSA-O86	2.400	4.53	≤ 30.00	☐		
3	Glulam 130x228 | CAN/CSA-O86	Glulam 130x418 | CAN/CSA-O86	2.400	4.53	≤ 30.00	☐		
4	Glulam 130x228 | CAN/CSA-O86	Glulam 130x418 | CAN/CSA-O86	1.800	6.03	≤ 30.00	☐		
5	Glulam 130x228 | CAN/CSA-O86	Glulam 130x418 | CAN/CSA-O86	0.900	11.92	≤ 30.00	☐		
6	Glulam 130x228 | CAN/CSA-O86	Glulam 130x418 | CAN/CSA-O86	3.600	3.02	≤ 30.00	☐		

Figure 2.27: Window 1.10 *Tapered Members*

Cross-Section

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

Length L

In this column, you can check the length of each tapered member.

Angle of Taper α

RF-TIMBER CSA determines the inclination of the taper on the basis of geometric conditions. That angle is displayed for your information.

The maximum angle of taper can be defined in the *Details* dialog box, tab *Other* (see Figure 3.4, page 37).

Radial Stress Design with Apex

If this check box has been ticked, RF-TIMBER CSA performs the design for the maximum tension stresses perpendicular to the grain in the ridge cross-section according to [1] Clause 6.5.6.6.2.

If this design is activated, it will be necessary to check the *Loading Distribution* in Window 1.4 *Load Duration* (see Chapter 2.4, page 22). The distribution of the loading has an effect on the size factor for tension perpendicular to grain, K_{Ztp}.

2.11 Curved Members

This window is available when at least one member with a curved shape has been selected for design in Window 1.1 *General Data*. Curved members can be defined, for example, by using the line types "arc" or "circle", i.e. lines with constant radius of curvature along the entire length.

The design of curved members is only possible for **rectangular** cross-sections whose material is **glued-laminated** timber according to [1] Table 6.3.

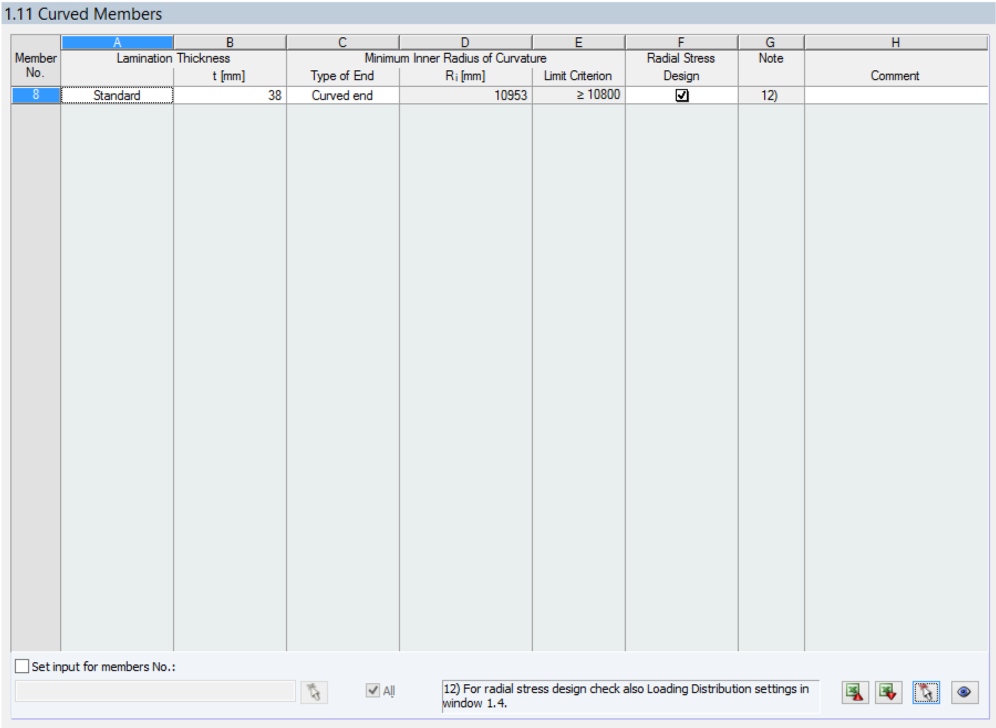
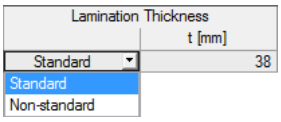


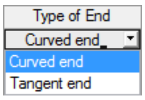
Figure 2.28: Window 1.11 *Curved Members*

Lamination Thickness



In those two columns, you can specify the type and the thickness *t* of the lamellas. The various *Standard* and *Non-standard* lamination thicknesses are listed in [1] Table A.6.5.5.

Minimum Inner Radius of Curvature



The program checks the inner radius of curvature. The design is only allowed for members whose inner radius, R_i, meets the limit criterion as described in [1] Table A.6.5.5. For that, the *Type of End* can be selected in the list box.

The *Limit Criterion* is displayed in column E. This value depends on the lamination thickness and the end type of the member.

Radial Stress Design

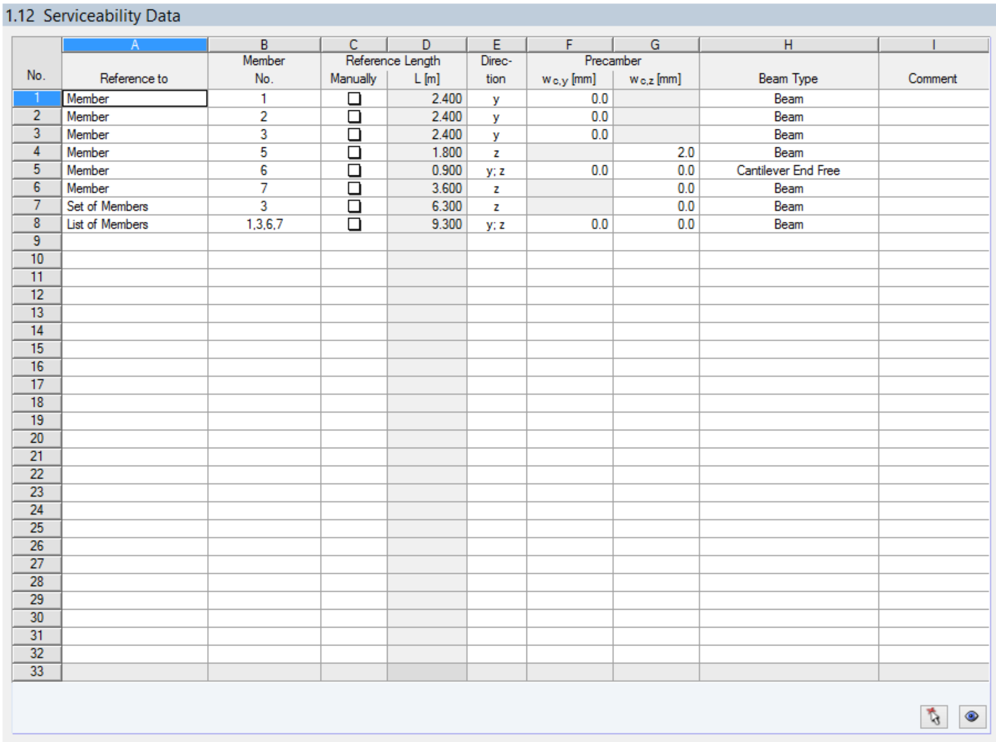
Optionally RF-TIMBER CSA performs a check of the tension stresses perpendicular to the grain in each location of the member where the bending moment tends to decrease the curvature (increase the radius) of the member.

If this design is activated, it will be necessary to check the *Loading Distribution* in Window 1.4 *Load Duration* (see Chapter 2.4, page 22). The distribution of the loading has an effect on the size factor for tension perpendicular to grain, K_{Ztp}.

The enclosed angle, β, can be determined only for members that are located on an undivided curved line, i.e. a line without additional nodes. Thus, the radial stress design is not possible for sets of members either.

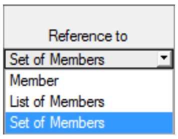
2.12 Serviceability Data

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).


1.12 Serviceability Data

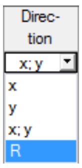
No.	A Reference to	B Member No.	C Reference Length Manually	D L [m]	E Direc-tion	F Precamber $w_{c,y}$ [mm]	G $w_{c,z}$ [mm]	H Beam Type	I Comment
1	Member	1	☐	2.400	y	0.0		Beam	
2	Member	2	☐	2.400	y	0.0		Beam	
3	Member	3	☐	2.400	y	0.0		Beam	
4	Member	5	☐	1.800	z		2.0	Beam	
5	Member	6	☐	0.900	y; z	0.0	0.0	Cantilever End Free	
6	Member	7	☐	3.600	z		0.0	Beam	
7	Set of Members	3	☐	6.300	z		0.0	Beam	
8	List of Members	1,3,6,7	☐	9.300	y; z	0.0	0.0	Beam	
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 2.29: Window 1.12 *Serviceability Data*



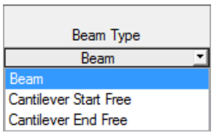
In column A, you decide whether you want to apply the deformation to single members, lists of members, or sets of members.

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.

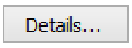


Column E controls the governing *Direction* for the deformation analysis. You can select the directions of the local member axes x and y and the resultant direction R.

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



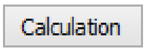
The *Beam Type* is important to correctly determine the limit deformations. Column H controls whether there is a beam or a cantilever and which end is not supported.



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 36).

3. Calculation

3.1 Detail Settings



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

The *Details* dialog box contains the following tabs:

- *Resistance*
- *Stability*
- *Serviceability*
- *Other*

3.1.1 Resistance

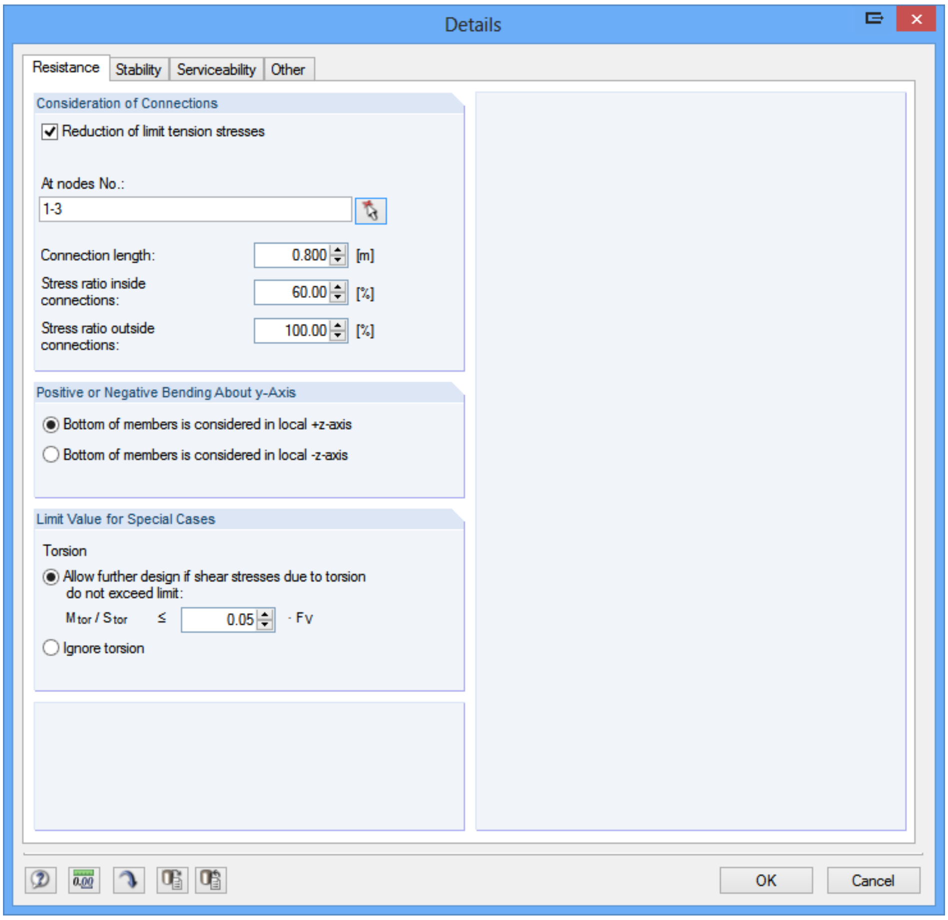


Figure 3.1: Dialog box *Details*, tab *Resistance*

Consideration of Connections

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.

Positive or Negative Bending About y-Axis

Structural glued laminated timber members stressed in bending have different reference bending design values for positive bending (bottom of beam is stressed in tension) and negative bending (top of beam is stressed in tension), see [4] Table 5A and Table 5C.

For RF-TIMBER CSA to apply the correct bending design value, you have to specify whether the bottom side of members is located in the direction of the local z-axis or opposite.

Limit Value for Special Cases

Torsion design is not specified in CAN/CSA O86-09. 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]. Changing the limit is the responsibility of the user.

It is also possible to completely *Ignore torsion*.

3.1.2 Stability

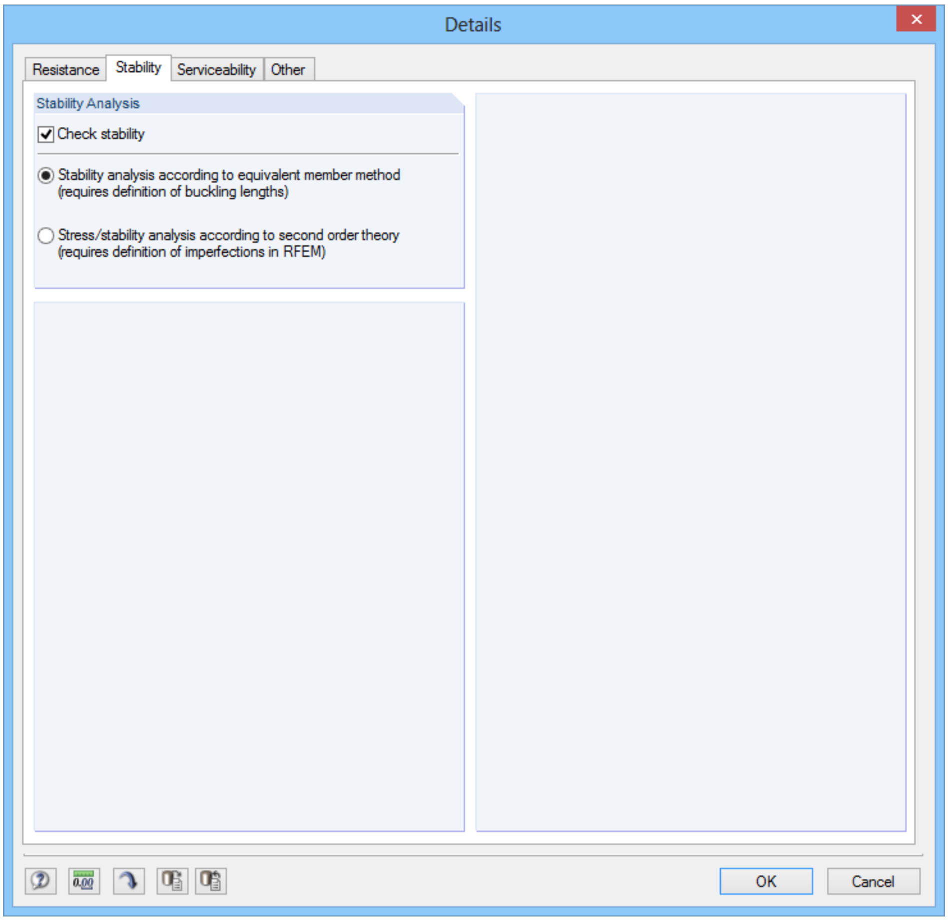
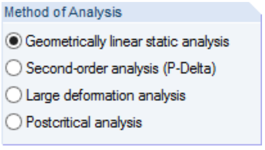


Figure 3.2: Dialog box *Details*, tab *Stability*

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.



Equivalent member method:
Specifying method of analysis in RFEM

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

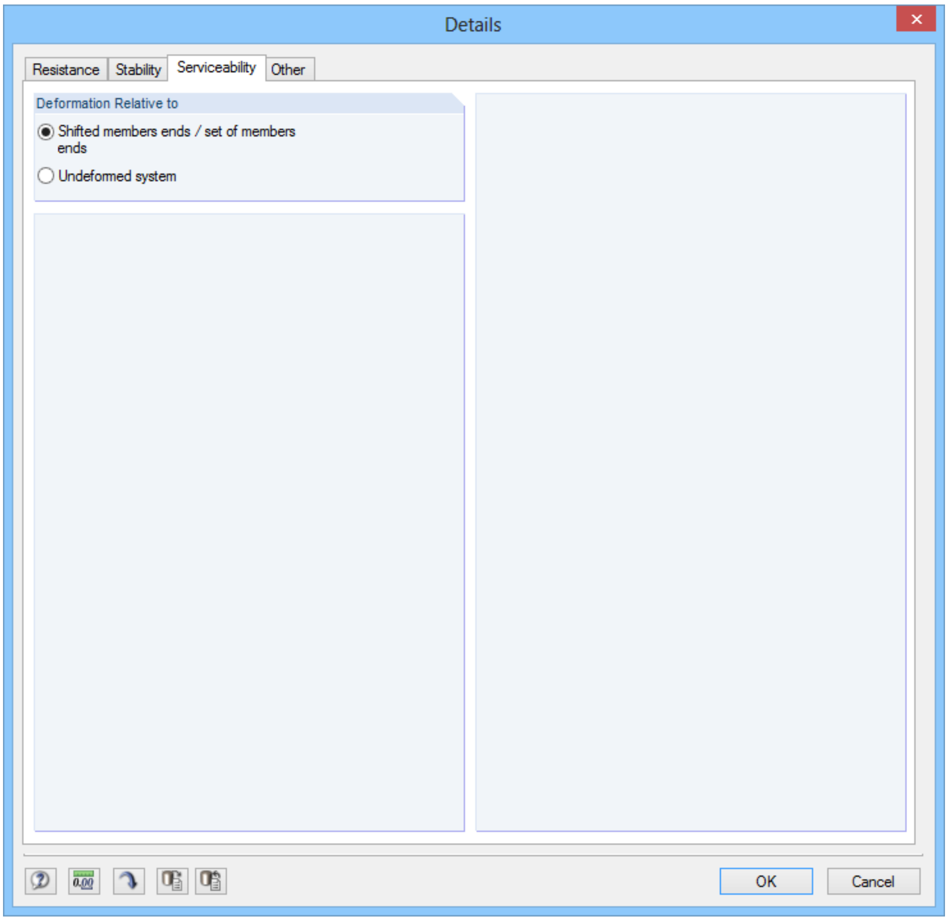


Figure 3.3: Dialog box *Details*, tab *Serviceability*

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.

3.1.4 Other

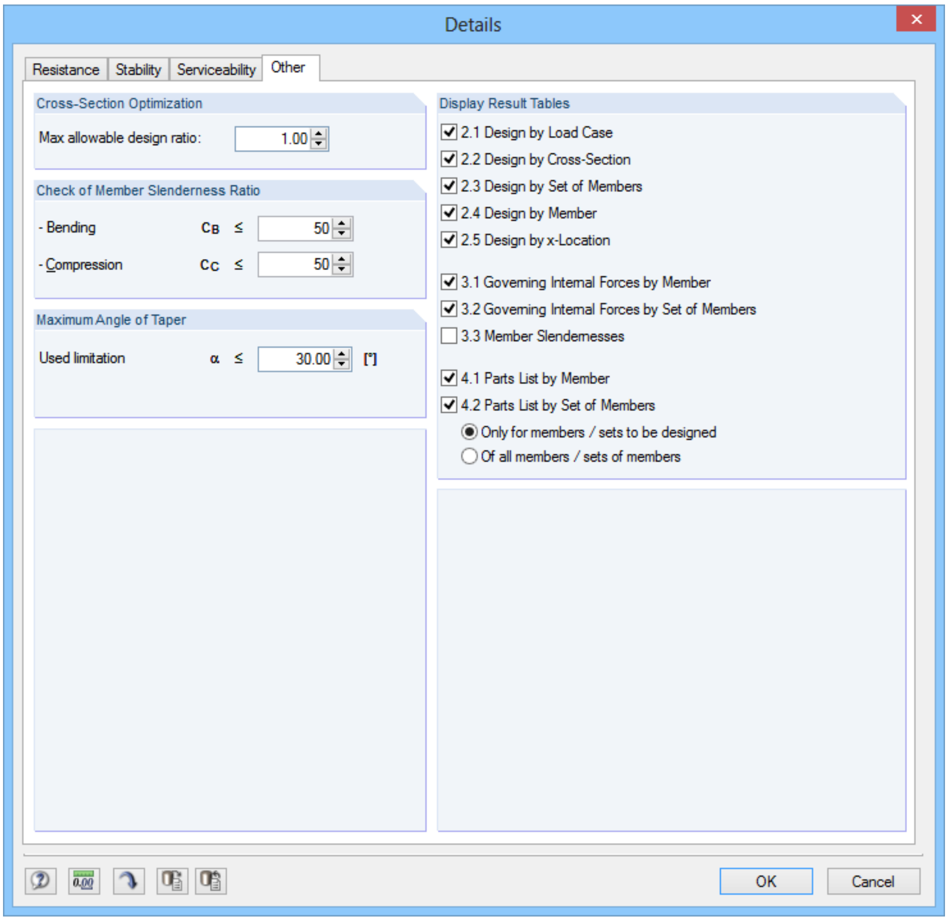


Figure 3.4: Dialog box *Details*, tab *Other*

Cross-Section Optimization

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

Check of Member Slenderness Ratio

In the two text boxes, you can specify the limit values of the member slendernesses. You can define the ratios separately for members with bending, C_B, and for members with compression, C_C.

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 46) if the corresponding check box is selected in the *Display Result Tables* dialog box section.

Maximum Angle of Taper

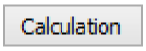
The taper angle α is described in [1] Figure 6.5.6.6.3. According [1] Table 6.5.6.6.3, the maximum value of that angle is 30°. The *Used limitation* can be set to a smaller angle, if necessary.

Display Result Tables

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

Window 3.3 *Member Slendernesses* 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 CSA add-on module.

RF-TIMBER CSA 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.

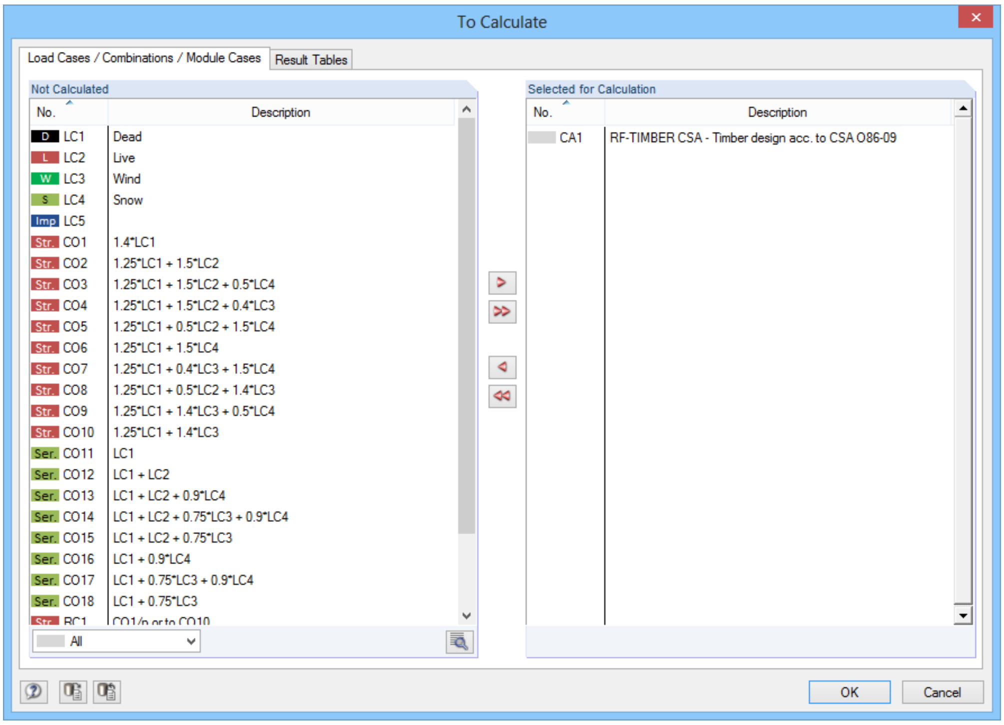
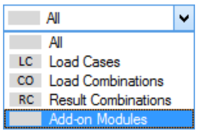


Figure 3.5: Dialog box *To Calculate*



If the RF-TIMBER CSA 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 CSA 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 CSA design case in the toolbar list, and then click [Show Results].

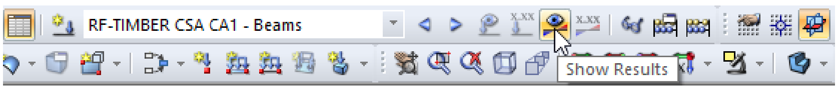


Figure 3.6: Direct calculation of a RF-TIMBER CSA design case in RFEM

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.

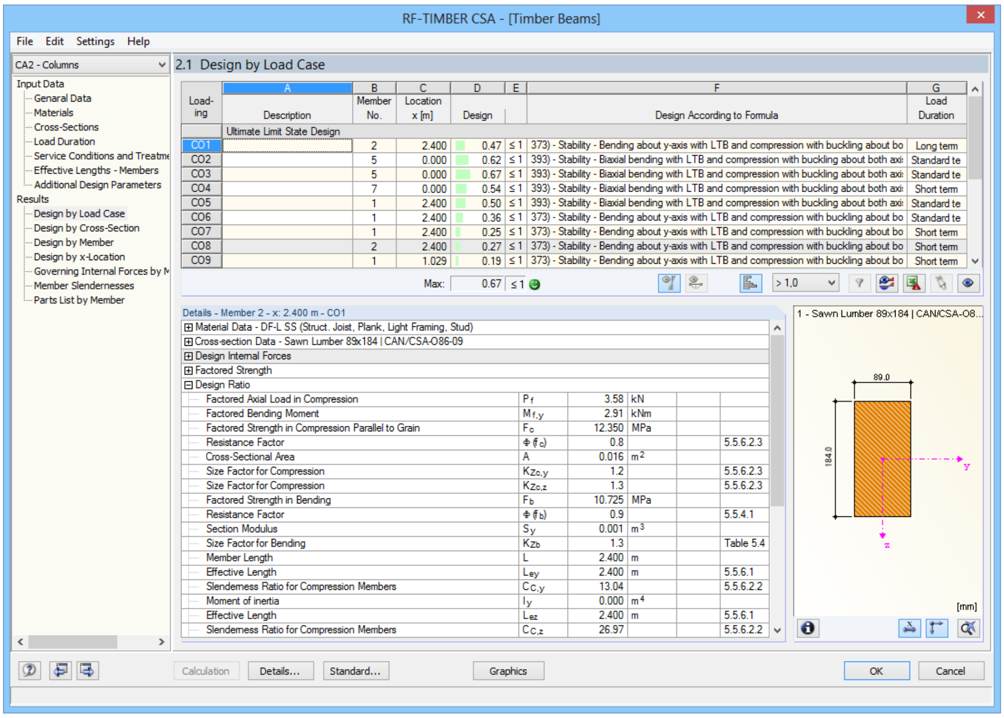
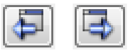


Figure 4.1: Results window with designs and intermediate values

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 CSA 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 49.

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* and *Serviceability Limit State 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.

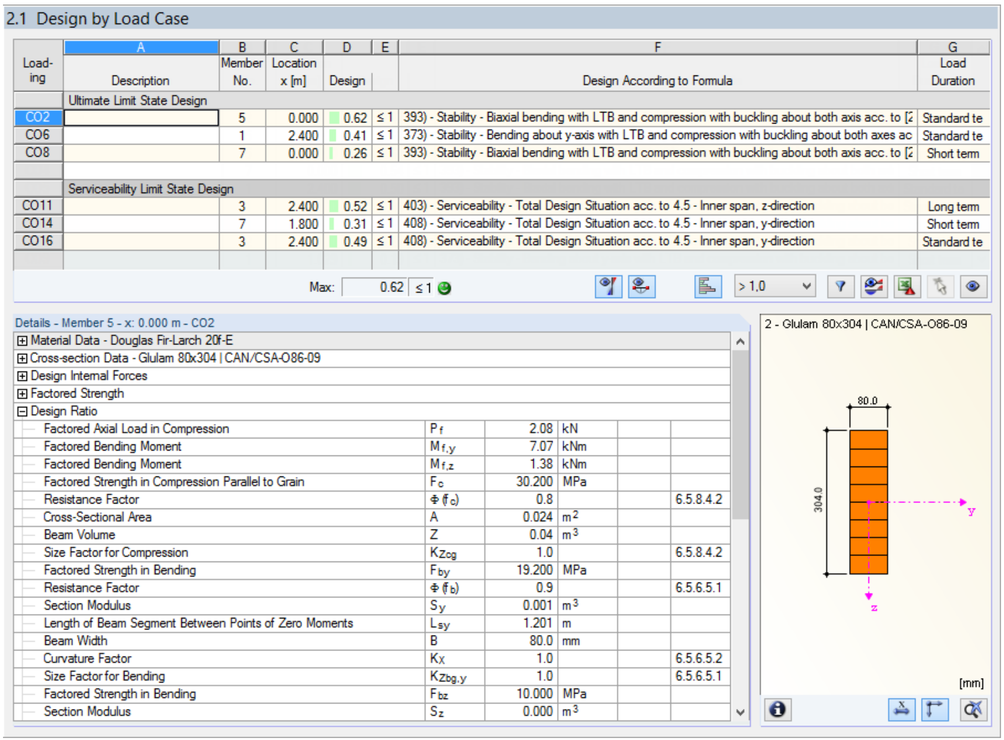


Figure 4.2: Window 2.1 *Design by Load Case*

Description

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

Member No.

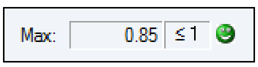
This column shows the number of the member that bears the maximum design ratio of every designed loading.

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 CAN/CSA O86-09.

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 (see Chapter 2.4, page 22).

4.2 Design by Cross-Section

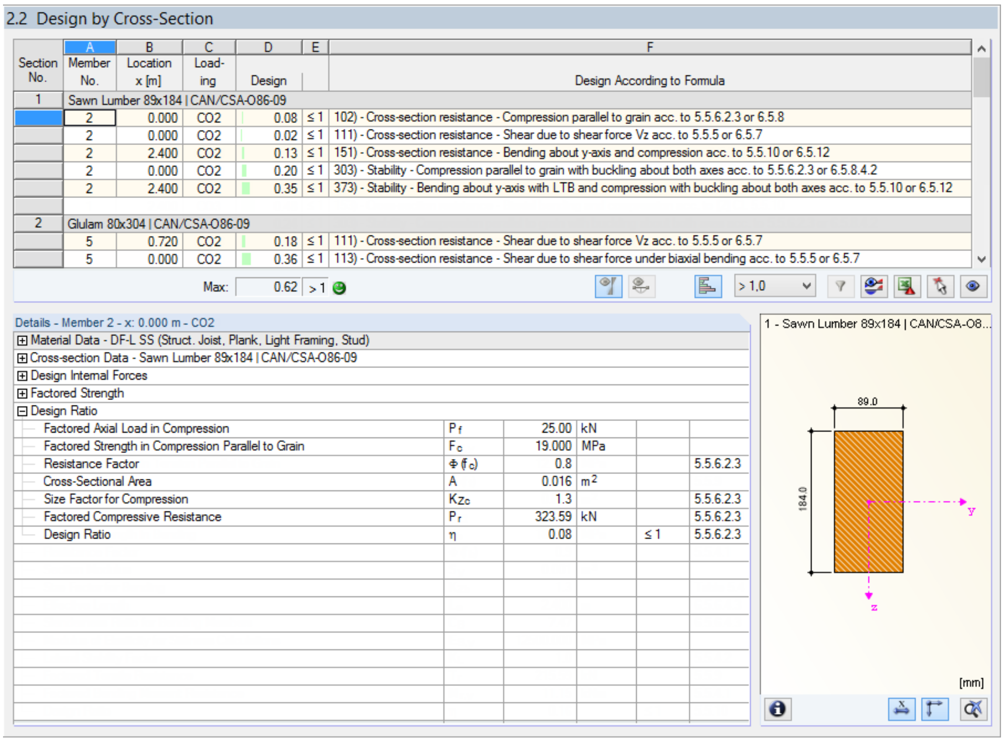


Figure 4.3: Window 2.2 *Design by Cross-Section*

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, and serviceability limit state design.

For tapered members, both cross-section descriptions are displayed in the table row next to the cross-section number.

4.3 Design by Set of Members

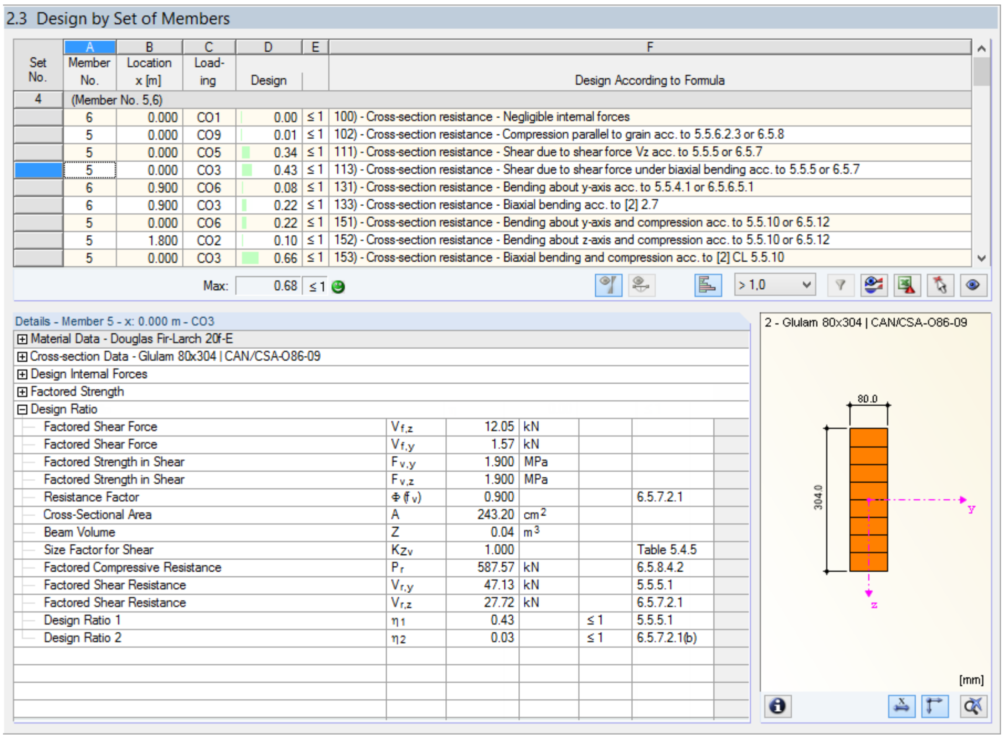


Figure 4.4: Window 2.3 *Design by Set of Members*

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

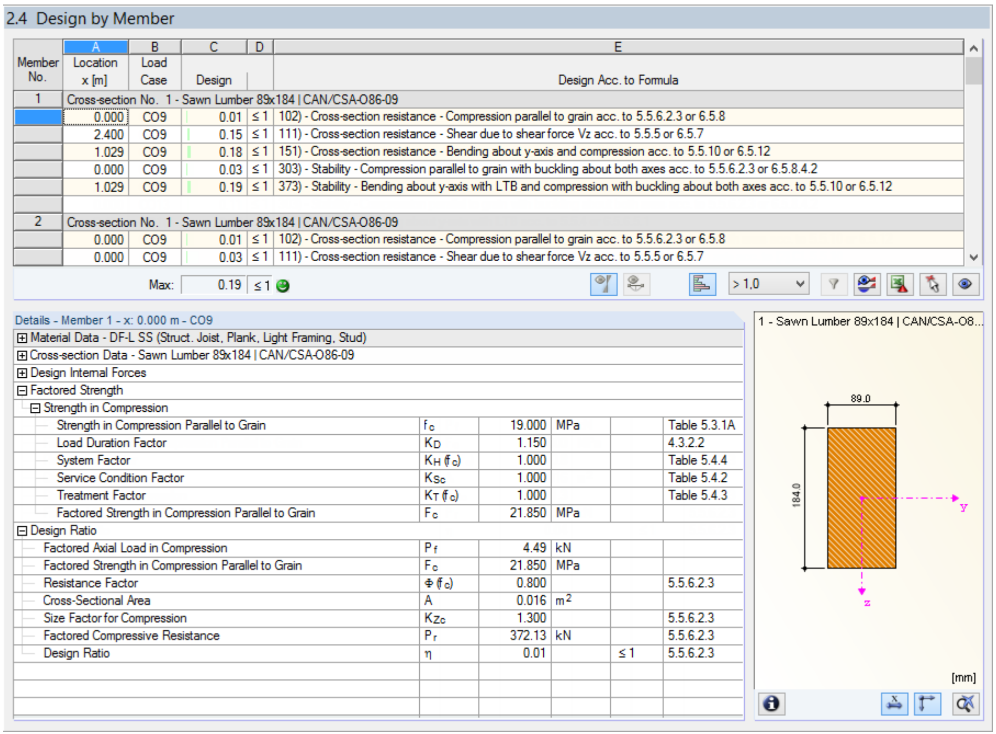


Figure 4.5: Window 2.4 *Design by Member*

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 40.

4.5 Design by x-Location

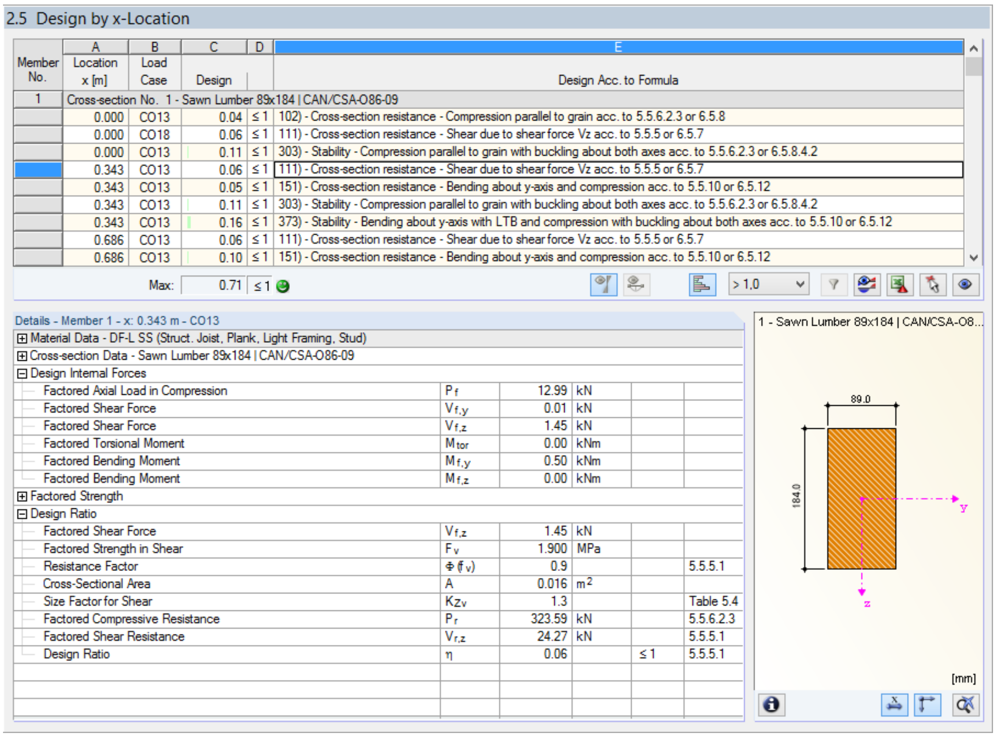


Figure 4.6: Window 2.5 *Design by x-Location*

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 - Sawn Lumber 89x184 | CAN/CSA-O86-09								
	0.000	CO9	-4.49	0.00	-3.60	0.00	0.00	0.00	102) - Cross-section resistance - Compression parallel to grain acc. to 5.5.
	2.400	CO9	-4.27	0.00	4.15	0.00	0.67	0.00	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.!
	1.029	CO9	-4.41	0.00	-0.28	0.00	-2.00	0.00	151) - Cross-section resistance - Bending about y-axis and compression a
	0.000	CO9	-4.49	0.00	-3.60	0.00	0.00	0.00	303) - Stability - Compression parallel to grain with buckling about both axe
	1.029	CO9	-4.41	0.00	-0.28	0.00	-2.00	0.00	373) - Stability - Bending about y-axis with LTB and compression with buc
2	Cross-section No. 1 - Sawn Lumber 89x184 | CAN/CSA-O86-09								
	0.000	CO9	-5.31	0.00	-0.80	0.00	0.00	0.00	102) - Cross-section resistance - Compression parallel to grain acc. to 5.5.
	0.000	CO9	-5.31	0.00	-0.80	0.00	0.00	0.00	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.!
	2.400	CO9	-5.07	0.00	-0.78	0.00	-1.89	0.00	151) - Cross-section resistance - Bending about y-axis and compression a
	0.000	CO9	-5.31	0.00	-0.80	0.00	0.00	0.00	303) - Stability - Compression parallel to grain with buckling about both axe
	2.400	CO9	-5.07	0.00	-0.78	0.00	-1.89	0.00	373) - Stability - Bending about y-axis with LTB and compression with buc
3	Cross-section No. 1 - Sawn Lumber 89x184 | CAN/CSA-O86-09								
	0.000	CO9	-6.02	0.00	-0.65	0.00	0.00	0.00	102) - Cross-section resistance - Compression parallel to grain acc. to 5.5.
	0.000	CO9	-6.02	0.00	-0.65	0.00	0.00	0.00	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.!
	2.400	CO9	-5.79	0.00	-0.63	0.00	-1.53	0.00	151) - Cross-section resistance - Bending about y-axis and compression a
	0.000	CO9	-6.02	0.00	-0.65	0.00	0.00	0.00	303) - Stability - Compression parallel to grain with buckling about both axe
	2.400	CO9	-5.79	0.00	-0.63	0.00	-1.53	0.00	373) - Stability - Bending about y-axis with LTB and compression with buc
5	Cross-section No. 2 - Glulam 80x304 | CAN/CSA-O86-09								
	0.000	CO9	-3.37	0.00	0.13	0.00	0.01	0.00	102) - Cross-section resistance - Compression parallel to grain acc. to 5.5.
	1.800	CO9	-3.37	0.00	-2.48	0.00	-2.11	0.00	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.!
	1.800	CO9	-3.37	0.00	-2.48	0.00	-2.11	0.00	151) - Cross-section resistance - Bending about y-axis and compression a
	1.800	CO9	-3.37	0.00	-2.48	0.00	-2.11	0.00	303) - Stability - Compression parallel to grain with buckling about both axe
	1.800	CO9	-3.37	0.00	-2.48	0.00	-2.11	0.00	373) - Stability - Bending about y-axis with LTB and compression with buc
6	Cross-section No. 2 - Glulam 80x304 | CAN/CSA-O86-09								
	0.000	CO9	0.00	0.00	0.00	0.00	0.00	0.00	100) - Cross-section resistance - Negligible internal forces
	0.900	CO9	-2.75	0.00	-1.30	0.00	-0.59	0.00	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.!
	0.900	CO9	0.00	0.00	-1.30	0.00	-0.59	0.00	131) - Cross-section resistance - Bending about y-axis acc. to 5.5.4.1 or 6
	0.900	CO9	-2.75	0.00	-1.30	0.00	-0.59	0.00	151) - Cross-section resistance - Bending about y-axis and compression a
	0.900	CO9	-2.75	0.00	-1.30	0.00	-0.59	0.00	303) - Stability - Compression parallel to grain with buckling about both axe

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, this column displays the axial and shear forces as well as the torsional and bending moments producing maximum design ratios in the respective cross-section, stability and serviceability limit state 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] have been performed.

4.7 Governing Internal Forces by Set of Members

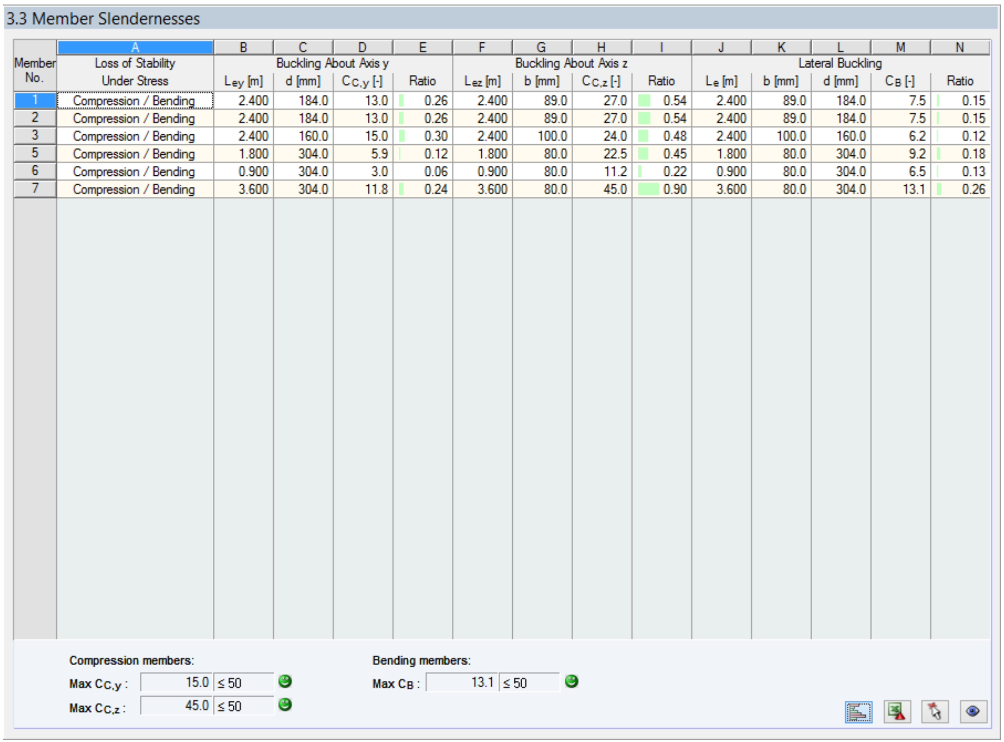
3.2 Governing Internal Forces by Set of Members

	A	B	C	D	E	F	G	H	I
Set No.	Location x [m]	Load Case	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
3	Continuous beam 1.1 (Member No. 5-7)								
	0.000	CO2	0.00	0.00	0.00	0.00	0.00	0.00	100) - Cross-section resistance - Negligible internal forces
	0.720	CO3	-1.93	-0.71	5.83	0.00	-1.73	-0.56	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.5 or
	0.000	CO3	-1.56	2.55	14.85	-0.03	-7.20	1.38	113) - Cross-section resistance - Shear due to shear force under biaxial bendi
	3.600	CO2	-1.44	-1.79	-11.02	0.04	-3.66	0.00	131) - Cross-section resistance - Bending about y-axis acc. to 5.5.4.1 or 6.5.6
	0.000	CO2	-1.45	2.55	12.21	-0.02	-5.80	1.38	133) - Cross-section resistance - Biaxial bending acc. to [2] 2.7
	3.600	CO3	-1.55	-1.79	-13.06	0.04	-3.98	0.00	151) - Cross-section resistance - Bending about y-axis and compression acc.
	1.800	CO2	-1.92	0.58	-2.67	0.00	0.06	-0.49	152) - Cross-section resistance - Bending about z-axis and compression acc.
	0.000	CO3	-1.94	-1.57	12.06	-0.03	-8.17	-1.38	153) - Cross-section resistance - Biaxial bending and compression acc. to [2]
	0.900	CO2	-2.75	-1.09	-4.45	0.01	-2.00	0.49	303) - Stability - Compression parallel to grain with buckling about both axes a
	3.600	CO2	-1.44	-1.79	-11.02	0.04	-3.66	0.00	311) - Stability - Bending about y-axis with LTB acc. to 5.5.4 or 6.5.6.5.1
	0.000	CO2	-1.45	2.55	12.21	-0.02	-5.80	1.38	313) - Stability - Biaxial bending with LTB acc. to [2] 2.7
	1.800	CO2	-1.92	0.58	-2.67	0.00	0.06	-0.49	343) - Stability - Bending about z-axis without LTB and compression with buck
	3.600	CO3	-1.55	-1.79	-13.06	0.04	-3.98	0.00	373) - Stability - Bending about y-axis with LTB and compression with bucklin
	0.000	CO3	-1.94	-1.57	12.06	-0.03	-8.17	-1.38	393) - Stability - Biaxial bending with LTB and compression with buckling abo
	1.800	CO12	0.00	0.00	0.00	0.00	0.00	0.00	403) - Serviceability - Total Design Situation acc. to 4.5 - Inner span, z-directi
	1.800	CO12	0.00	0.00	0.00	0.00	0.00	0.00	408) - Serviceability - Total Design Situation acc. to 4.5 - Inner span, y-directi
	0.180	CO12	0.00	0.00	0.00	0.00	0.00	0.00	413) - Serviceability - Total Design Situation acc. to 4.5 - Cantilever, z-directic
	0.900	CO12	0.00	0.00	0.00	0.00	0.00	0.00	418) - Serviceability - Total Design Situation acc. to 4.5 - Cantilever, y-directic

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.	Loss of Stability Under Stress	Buckling About Axis y: L_{ey} [m]	d [mm]	$C_{C,y}$ [-]	Ratio	Buckling About Axis z: L_{ez} [m]	b [mm]	$C_{C,z}$ [-]	Ratio	Lateral Buckling: L_e [m]	b [mm]	d [mm]	C_B [-]	Ratio
1	Compression / Bending	2.400	184.0	13.0	0.26	2.400	89.0	27.0	0.54	2.400	89.0	184.0	7.5	0.15
2	Compression / Bending	2.400	184.0	13.0	0.26	2.400	89.0	27.0	0.54	2.400	89.0	184.0	7.5	0.15
3	Compression / Bending	2.400	160.0	15.0	0.30	2.400	100.0	24.0	0.48	2.400	100.0	160.0	6.2	0.12
5	Compression / Bending	1.800	304.0	5.9	0.12	1.800	80.0	22.5	0.45	1.800	80.0	304.0	9.2	0.18
6	Compression / Bending	0.900	304.0	3.0	0.06	0.900	80.0	11.2	0.22	0.900	80.0	304.0	6.5	0.13
7	Compression / Bending	3.600	304.0	11.8	0.24	3.600	80.0	45.0	0.90	3.600	80.0	304.0	13.1	0.26

Compression members:
Max $C_{C,y}$: 15.0 ≤ 50
Max $C_{C,z}$: 45.0 ≤ 50

Bending members:
Max C_B : 13.1 ≤ 50

Figure 4.9: Window 3.3 *Member Slendernesses*

Details...

This results window is shown only when you have selected the respective check box in the *Other* tab of the *Details* dialog box (see Figure 3.4, page 37).

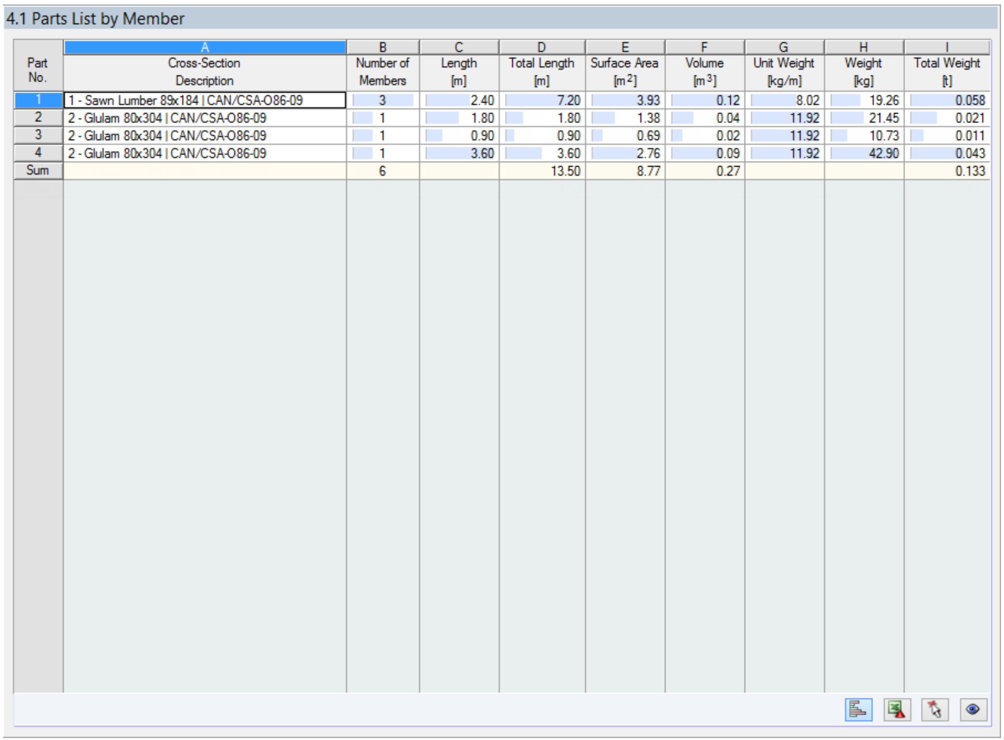
Details...

The table lists the effective slendernesses of the designed members which can lose their stability as compression members, bending members or combinations of both. They were determined depending on the type of load and occurrence of buckling or lateral-torsional buckling. 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 3.4, page 37).

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

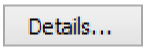
4.9 Parts List by Member

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



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	1 - Sawn Lumber 89x184 | CAN/CSA-O86-09	3	2.40	7.20	3.93	0.12	8.02	19.26	0.058
2	2 - Glulam 80x304 | CAN/CSA-O86-09	1	1.80	1.80	1.38	0.04	11.92	21.45	0.021
3	2 - Glulam 80x304 | CAN/CSA-O86-09	1	0.90	0.90	0.69	0.02	11.92	10.73	0.011
4	2 - Glulam 80x304 | CAN/CSA-O86-09	1	3.60	3.60	2.76	0.09	11.92	42.90	0.043
Sum		6		13.50	8.77	0.27			0.133

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 3.4, page 37).

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 2.16, page 20).

Dlubal

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.

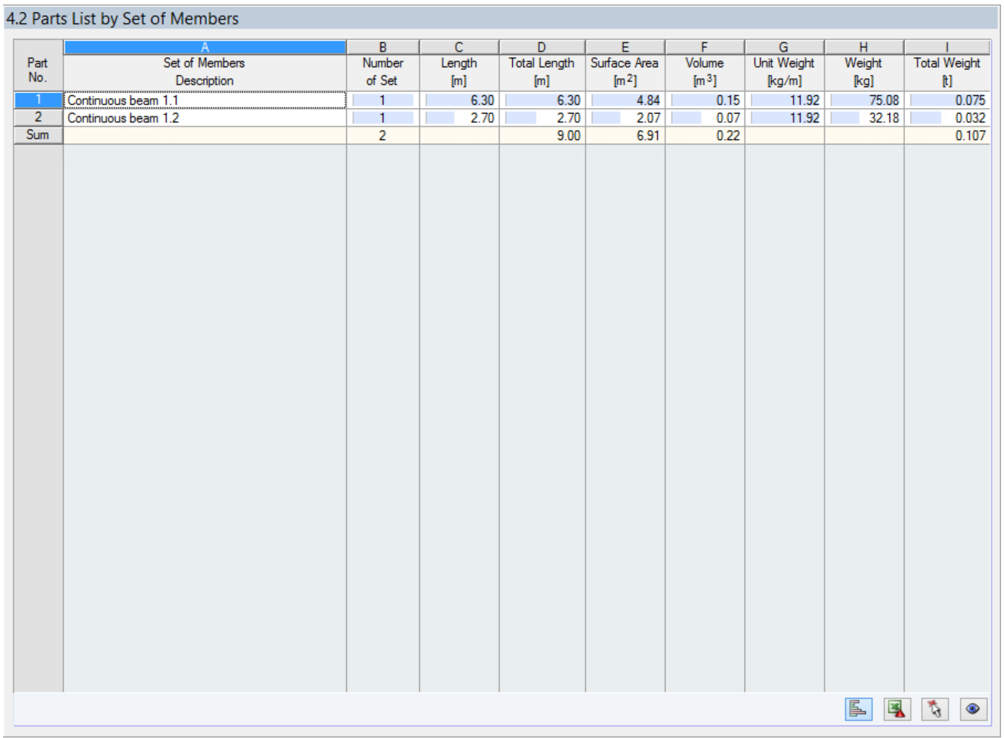
Total Weight

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 [m^2]	F Volume [m^3]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Continuous beam 1.1	1	6.30	6.30	4.84	0.15	11.92	75.08	0.075
2	Continuous beam 1.2	1	2.70	2.70	2.07	0.07	11.92	32.18	0.032
Sum		2		9.00	6.91	0.22			0.107

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. For this, the buttons located below the upper results tables are very useful.

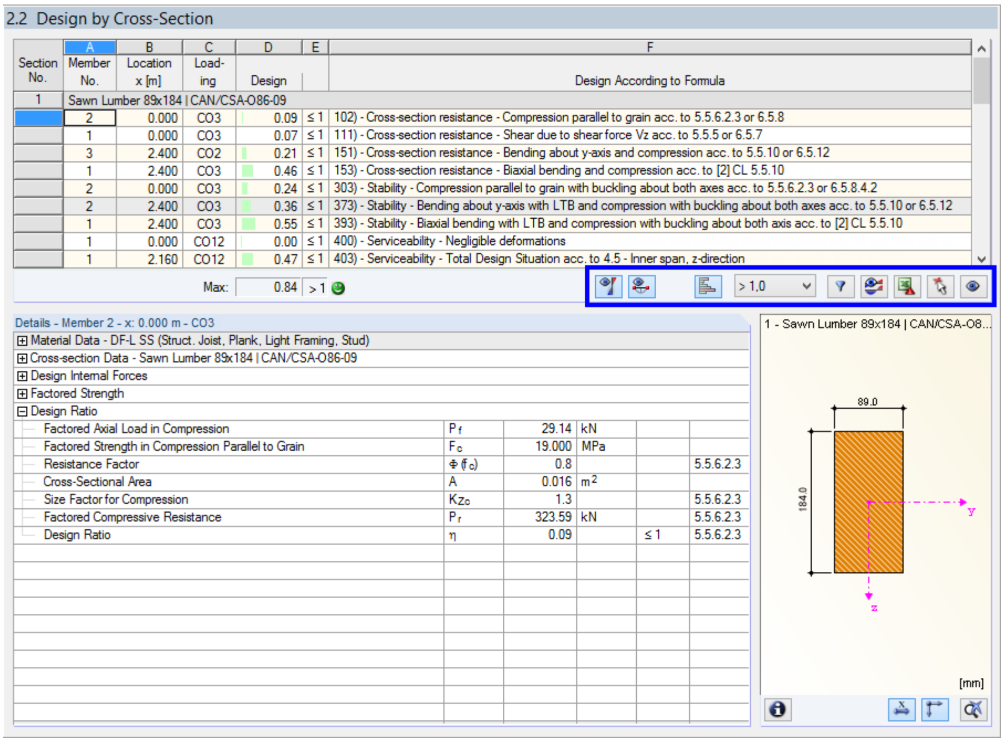


Figure 5.1: Buttons for evaluation of results

The buttons have the following functions:

Button	Description	Function
	ULS Design	Shows or hides the results of the ultimate limit state design
	SLS Design	Shows or hides the results of the serviceability limit state 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 52
	Excel Export	Exports the table to MS Excel / OpenOffice → Chapter 7.4.3, page 62
	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 results 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 CSA 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.

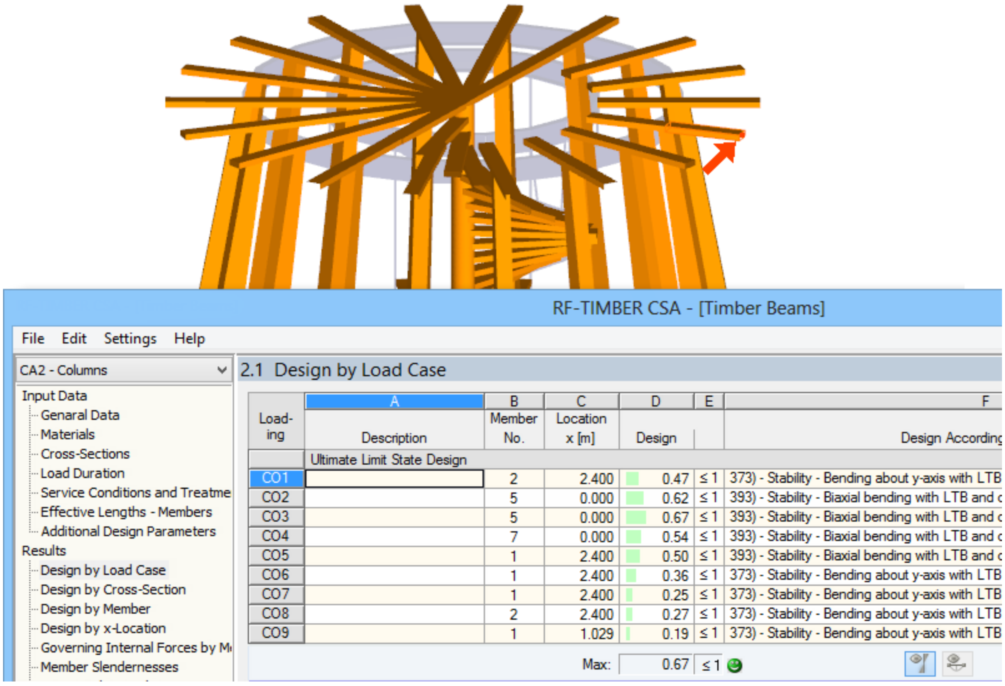
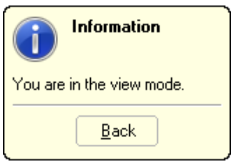


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model



If you cannot improve the display by moving the RF-TIMBER CSA 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.

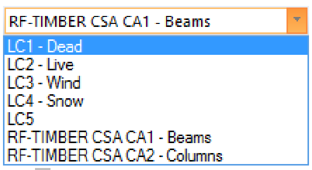
Click [Back] to return to the add-on module RF-TIMBER CSA.

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.

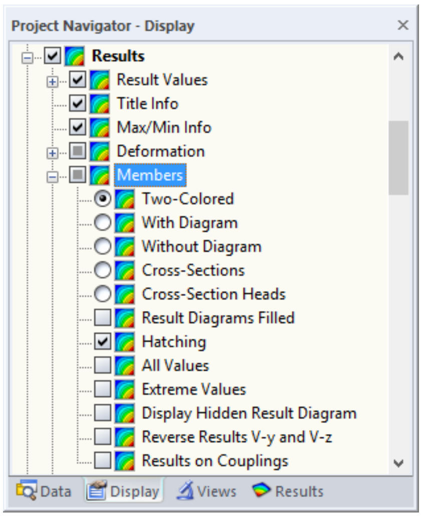


Figure 5.3: *Display* navigator: Results → Members

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.

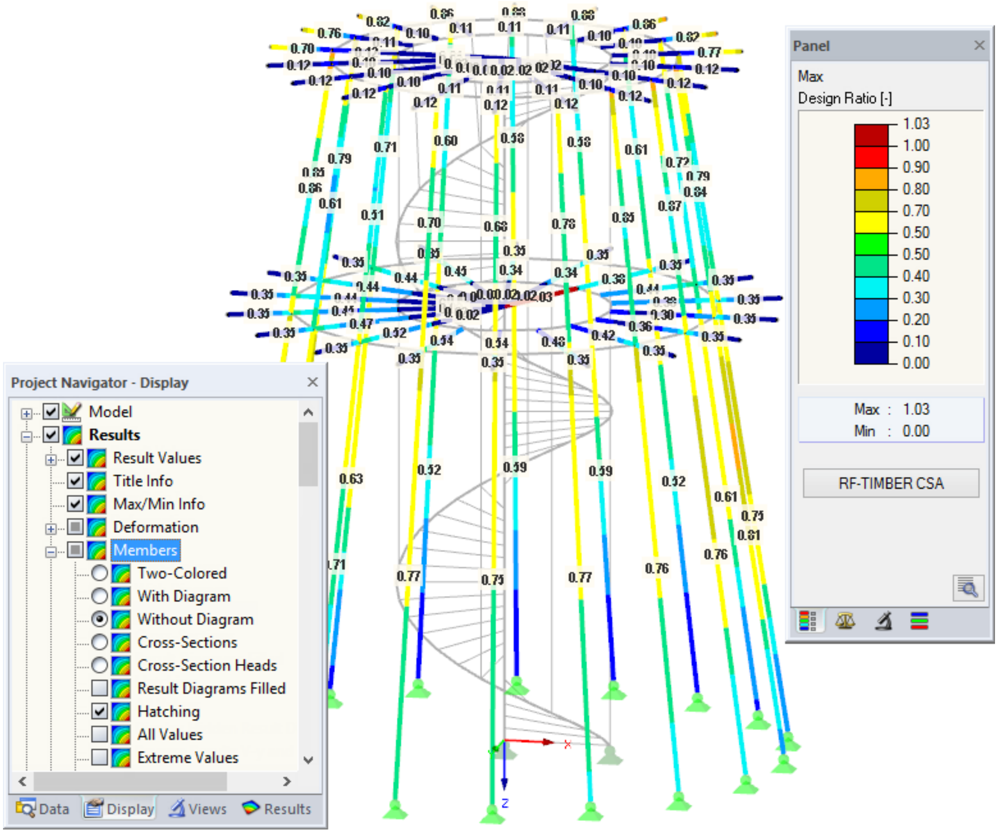


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 55).

To return to the add-on module, click the [RF-TIMBER CSA] 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 CSA 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 49).

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.

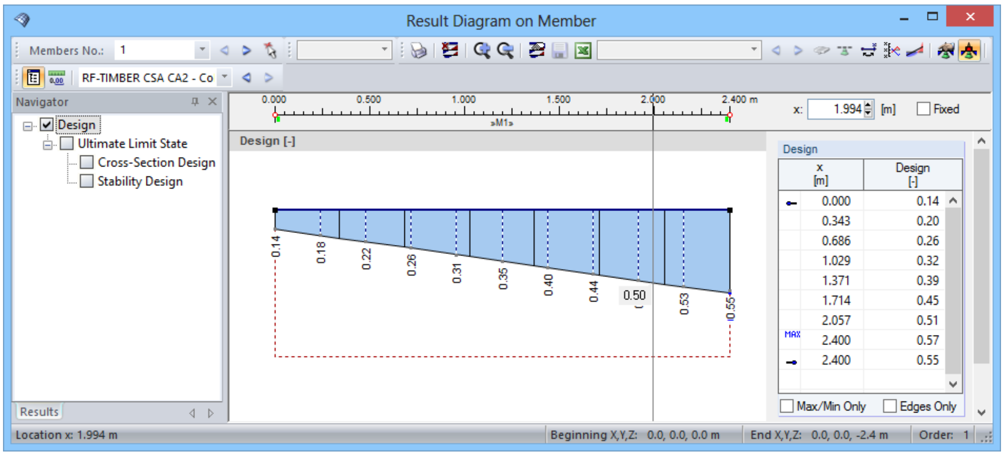
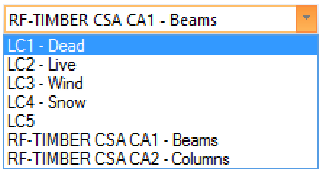


Figure 5.5: Dialog box *Result Diagram on Member*



Use the list in the toolbar above to select the relevant RF-TIMBER CSA 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 CSA 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 CSA (see RFEM manual, Chapter 9.9.1) to filter the members in order to evaluate them.

Filtering designs

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.

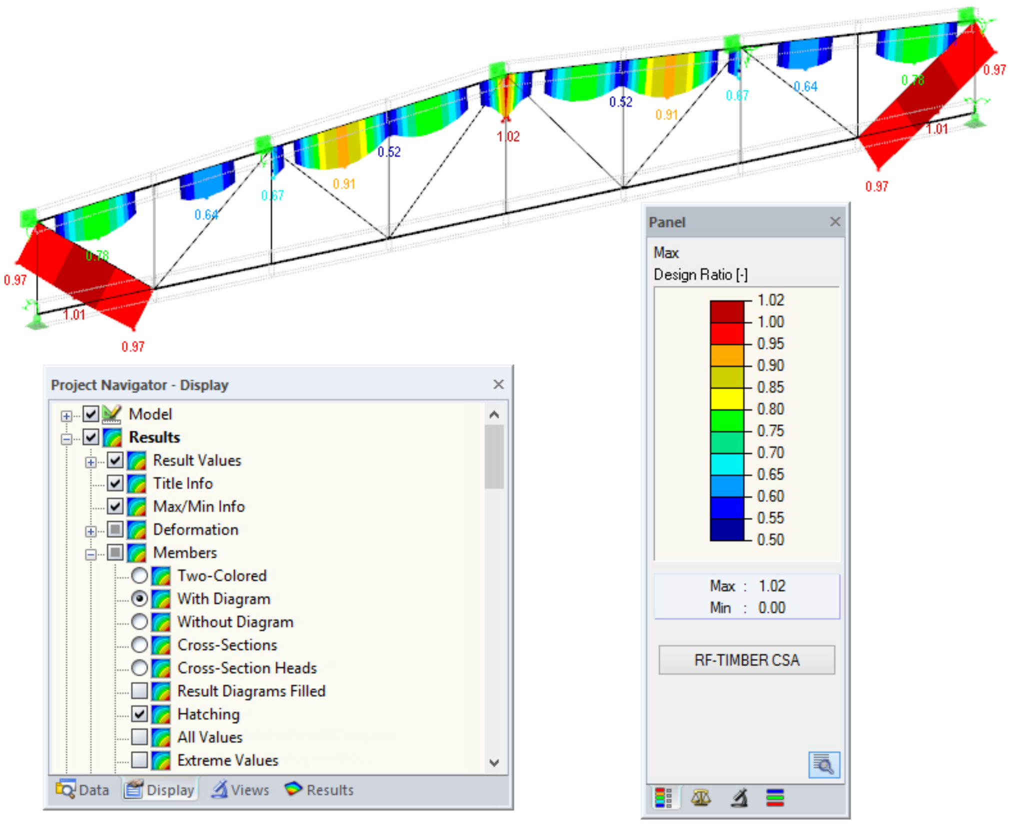
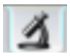


Figure 5.6: Filtering design ratios with adjusted color spectrum

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.

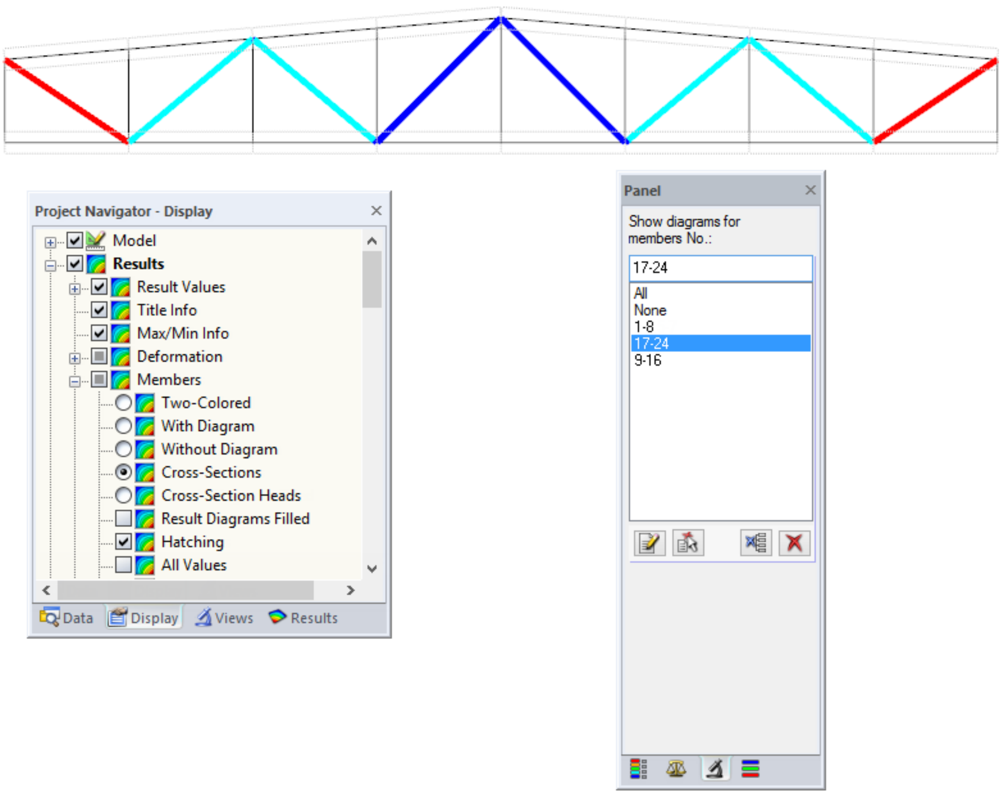


Figure 5.7: Member filter for ratios of diagonals

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 CSA 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 10.1.3.4 *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.

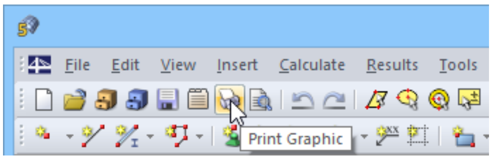


Figure 6.1: Button *Print* in RFEM toolbar

Result diagrams

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

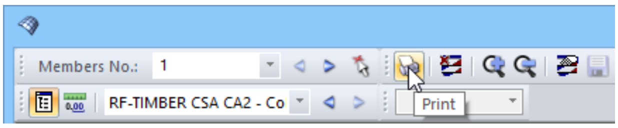


Figure 6.2: Button *Print* in the *Result Diagram on Member*

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

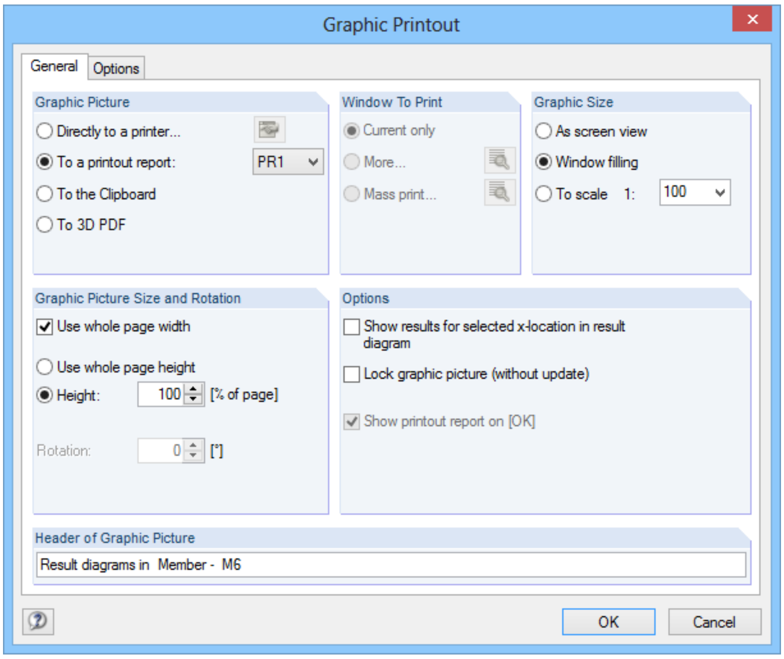
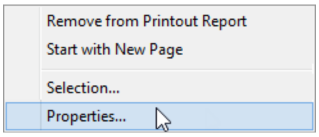


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in the RFEM manual, Chapter 10.2. The RFEM manual also describes the *Options* and *Color Spectrum* tab.

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 context menu opens the *Graphic Printout* dialog box, offering various possibilities for adjustment.

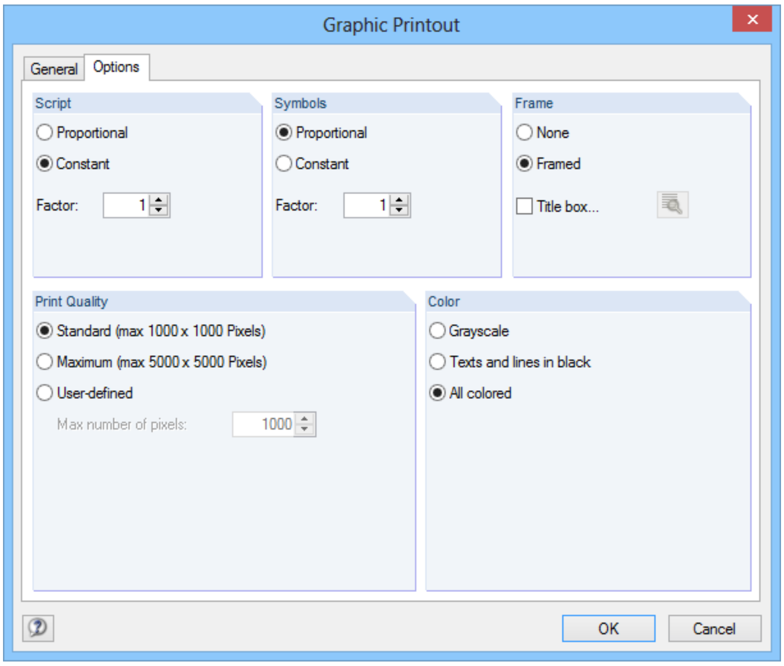


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

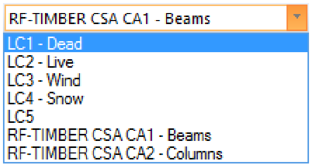
7. General Functions

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 CSA 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 CSA menu and click

File → New Case.

The following dialog box appears:

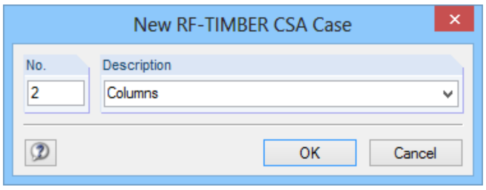


Figure 7.1: Dialog box *New RF-TIMBER CSA-Case*

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 CSA 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 CSA menu and click

File → Rename Case.

The following dialog box appears:

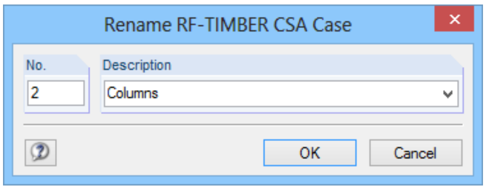


Figure 7.2: Dialog box *Rename RF-TIMBER CSA-Case*

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 CSA menu

File → Copy Case

The following dialog box appears:

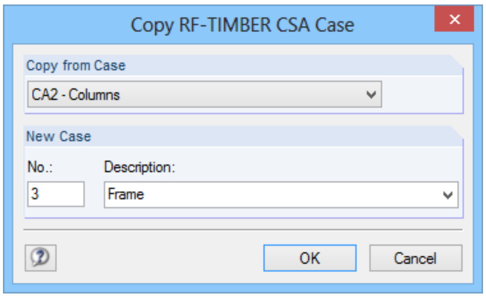


Figure 7.3: Dialog box *Copy RF-TIMBER CSA-Case*

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

Delete design case

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

File → Delete Case

The following dialog box appears:

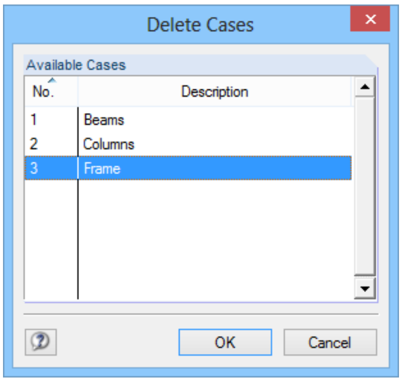


Figure 7.4: Dialog box *Delete Cases*

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 2.15 on page 19).

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

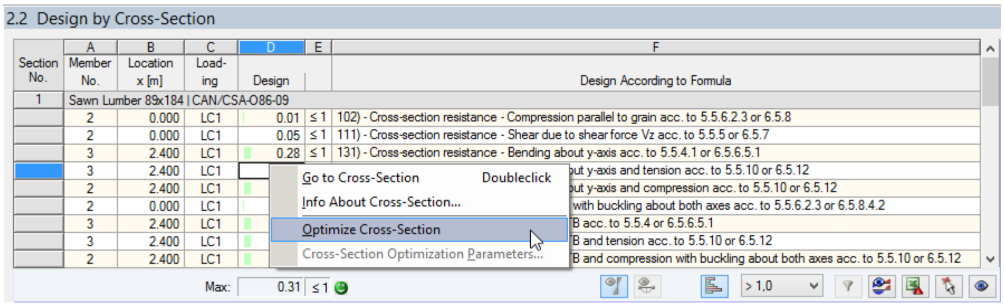


Figure 7.5: Context-menu for cross-section optimization

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 3.4, page 37). 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.

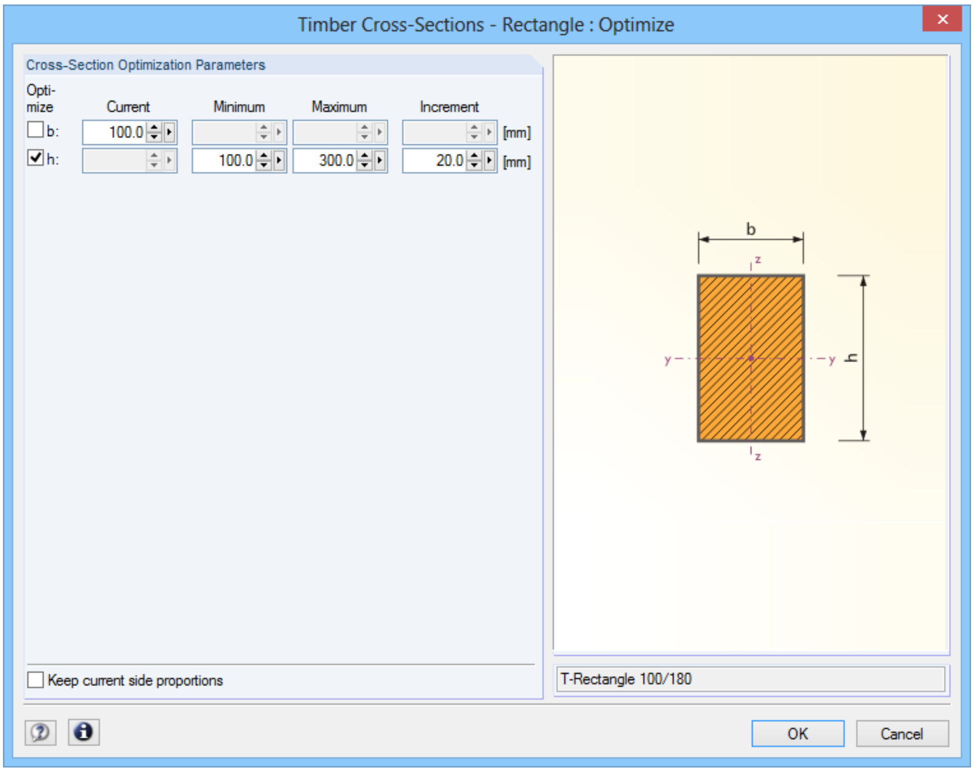


Figure 7.6: Dialog box *Timber Cross-Sections - Rectangle : Optimize*

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 context menu in window 1.3 to export optimized cross-sections to RFEM.

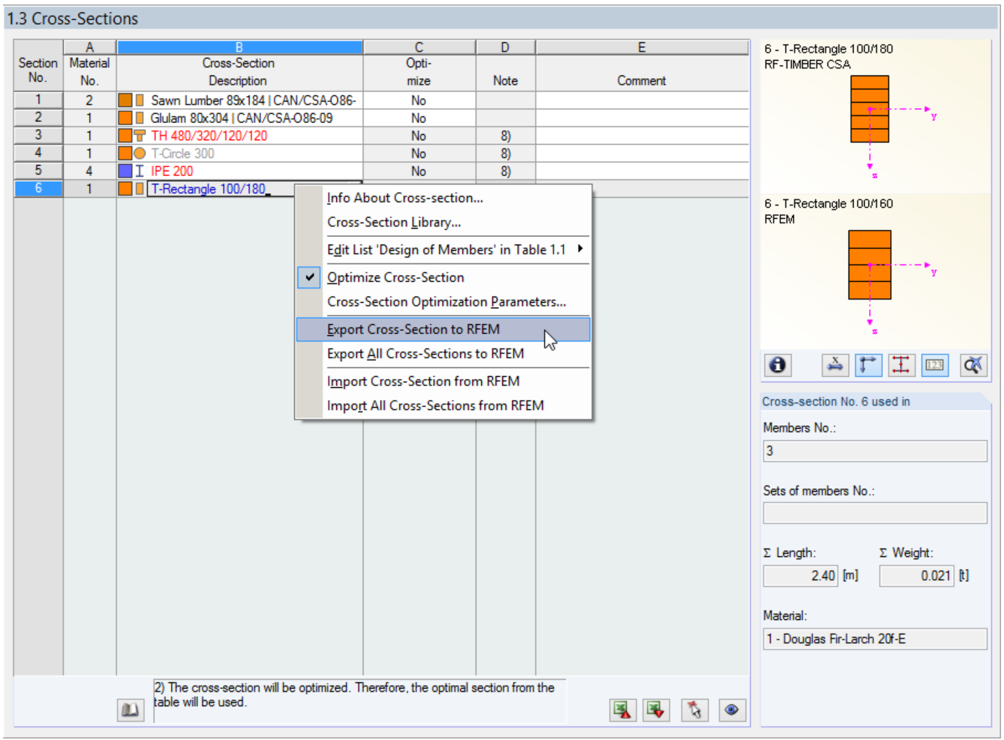


Figure 7.7: Context menu in window 1.3 *Cross-Sections*

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

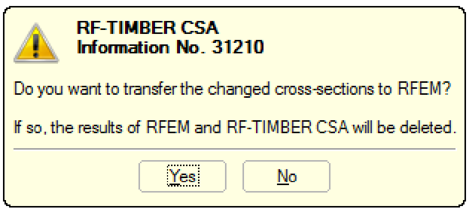


Figure 7.8: Query before transfer of modified cross-sections to RFEM

Calculation

By confirming the query and then starting the [Calculation] in the RF-TIMBER CSA 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 CSA, select menu

Settings → **Units and Decimal Places**.

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

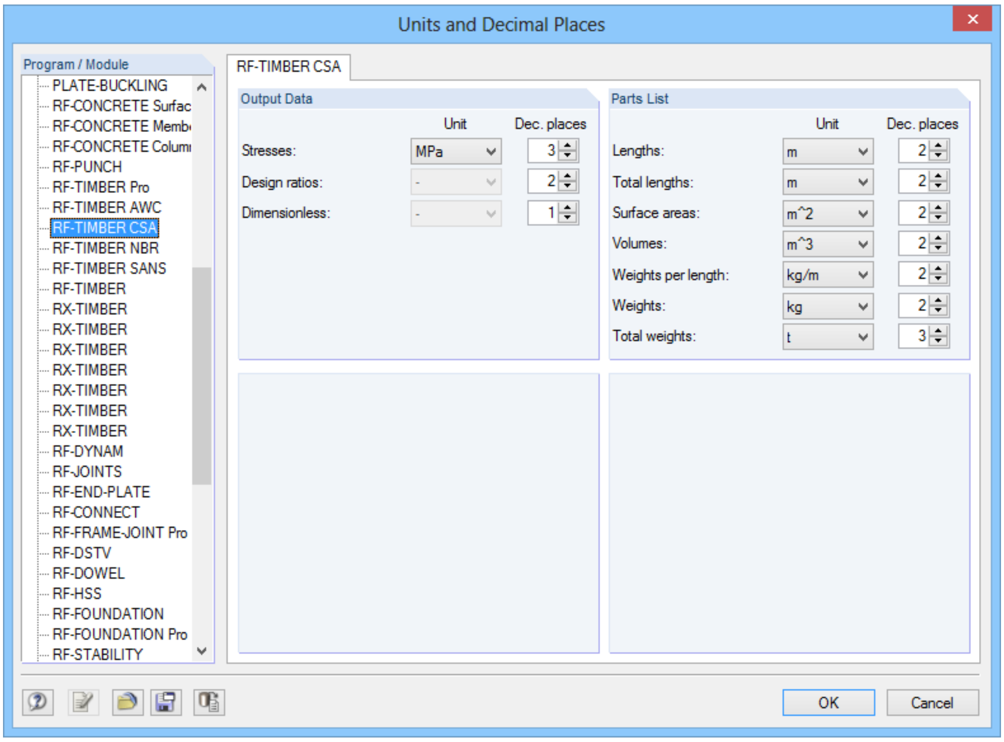


Figure 7.9: Dialog box *Units and Decimal Places*

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 CSA 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 context menu of window 1.2.

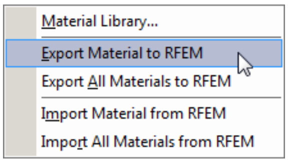
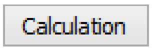


Figure 7.10: Context menu of window 1.2 *Materials*



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 CSA, 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 CSA 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 context menu of window 1.7.

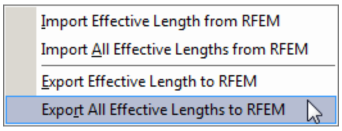


Figure 7.11: Context menu of window 1.7 *Effective Lengths - Members*

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 CSA 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 CSA add-on module into the global printout report (see Chapter 6.1, page 55) 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 CSA provides a function for the direct data export to MS Excel, OpenOffice.org Calc, or the file format CSV. To open the corresponding dialog box, click

File → Export Tables

The following export dialog box appears.

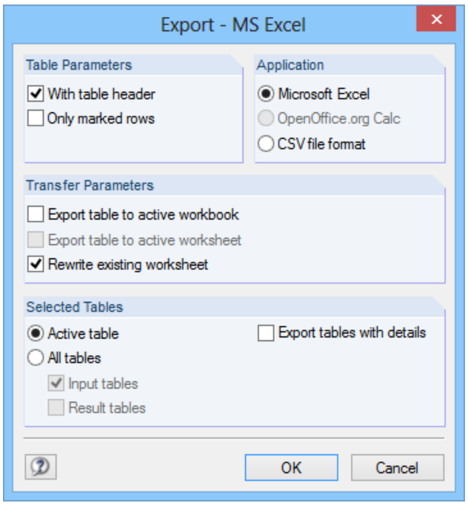


Figure 7.12: Dialog box *Export - MS Excel*

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.

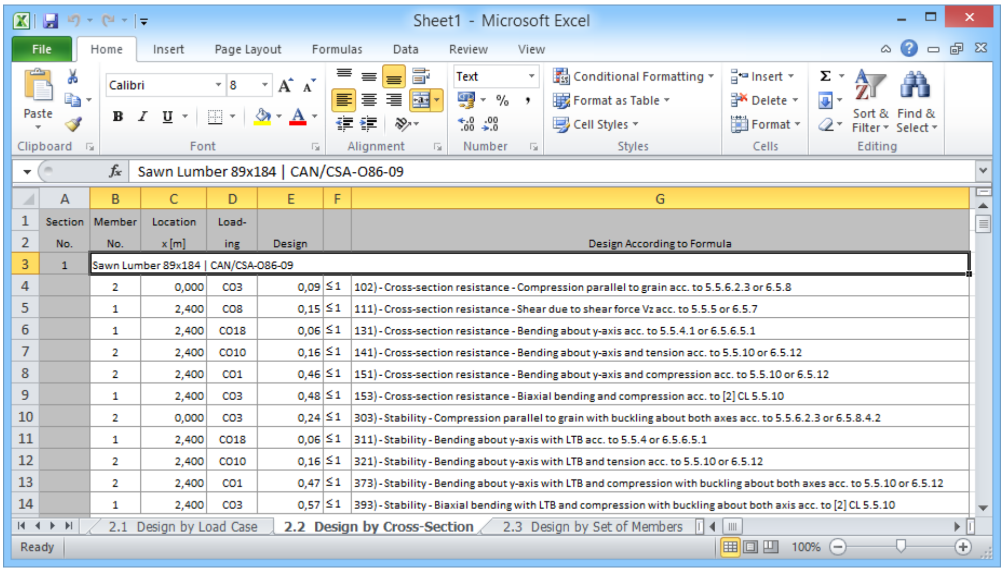


Figure 7.13: Results in *Excel*

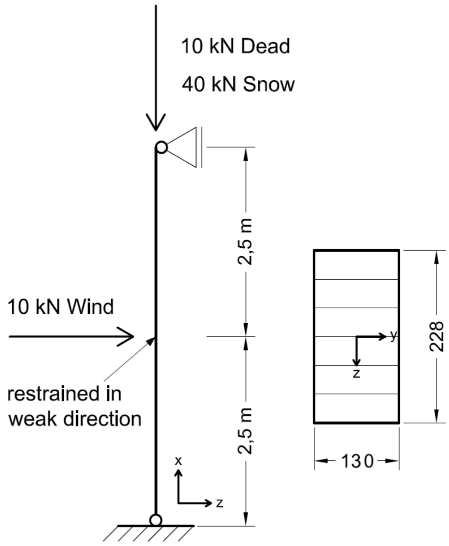
8. Examples

8.1 Glulam Column

We perform the design according to CAN/CSA O86-09 for a glulam 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 material is a Spruce-Lodgepole Pine-Jack Pine 20f-EX glulam in dry service conditions.

That example is described in [2].

8.1.1 System and Loads



Model

Cross-section:	b = 130 mm d = 228 mm
Material:	Spruce-Lodgepole Pine-Jack Pine 20f-EX Glulam
Moisture Condition:	Dry

Load

LC 1 Dead:	10 kN
LC 2 Snow:	40 kN
LC 3 Wind:	10 kN
Load combinations ULS:	1.25D + 0.5S + 1.4W 1.25D + 1.5S + 0.4W
Load combination SLS:	0.75W

Figure 8.1: System and loads according to [2]

If beams are divided by nodes, it is necessary to verify them as a set of members. In this example, the beam is not divided because the wind load is defined as a *Concentrated Member Load*. If a nodal load was used, we would have to design a set of members instead of a member (see chapter 2.1).

8.1.2 Calculation with RFEM

The system as well as the loads in all load cases is modeled in RFEM as a 2D-XZ 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**.

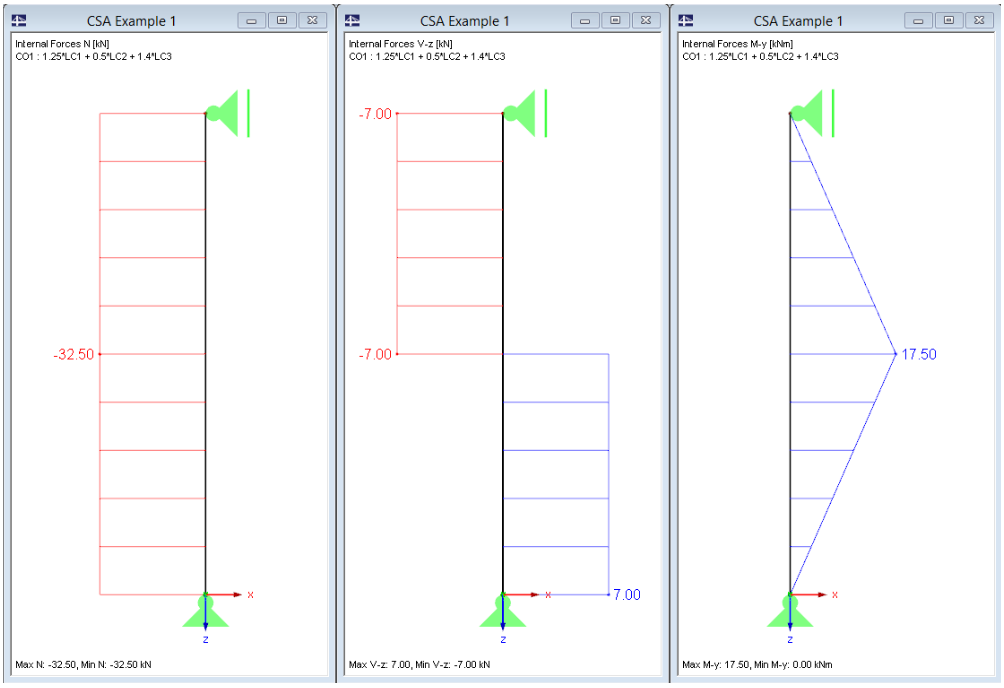


Figure 8.2: Internal forces N, V_z and M_y (ULS)

8.1.3 Design with RF-TIMBER CSA

8.1.3.1 Ultimate Limit State Design

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

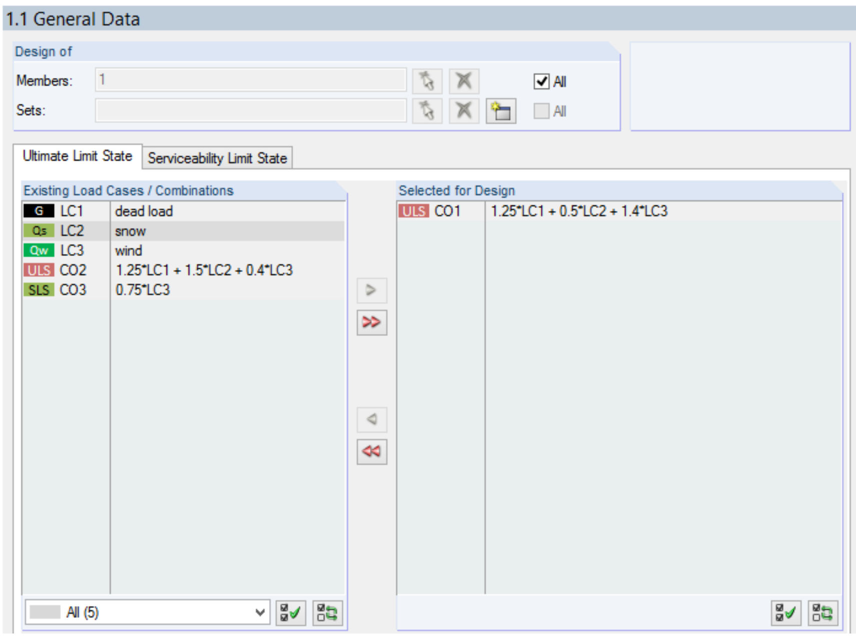


Figure 8.3: Window 1.1 *General Data*

The 1.2 *Materials* window presents the characteristic strengths of the selected material.

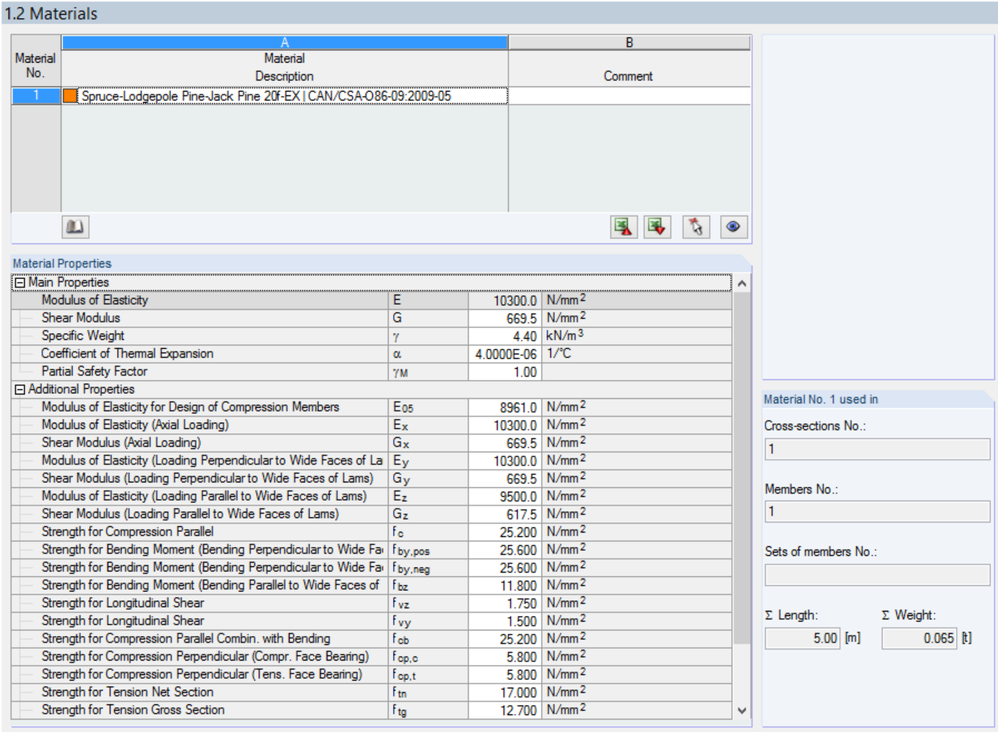


Figure 8.4: Window 1.2 *Materials*

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

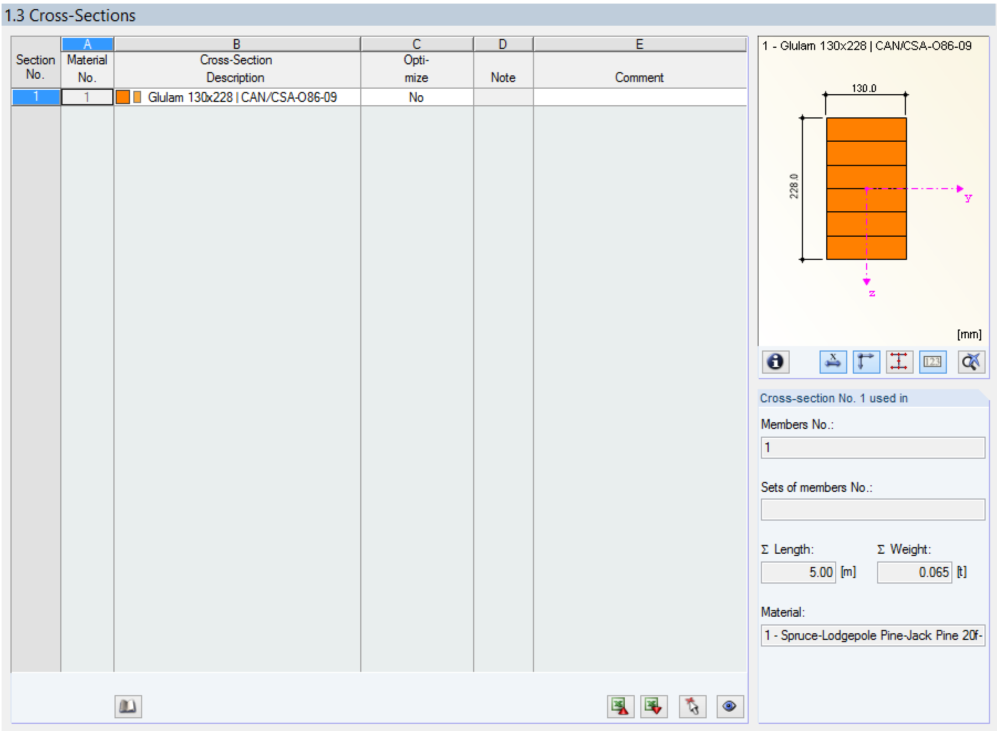


Figure 8.5: Window 1.3 *Cross-Sections*

In window 1.4 *Load Duration*, we define the load duration. For the dead load, we leave *Long term*, for snow we change it to **Standard term** and for wind to **Short term**. According to [1] Chapter 4.3.2.4, we choose **Shortest load duration in a combination** so the load duration factor K_D of the shortest duration will be considered for the load combination.

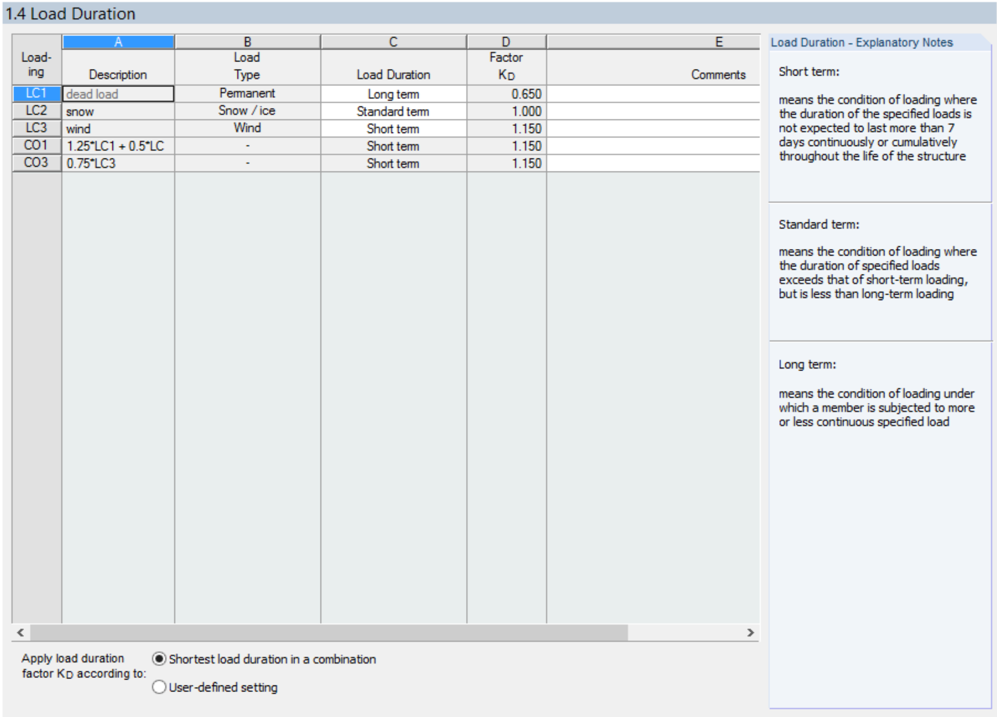


Figure 8.6: Window 1.4 *Load Duration*

In window 1.5 *Service Conditions and Treatment - Members*, we define the moisture and the treatment parameters. The treatment factor K_T will be taken as unity in this case.

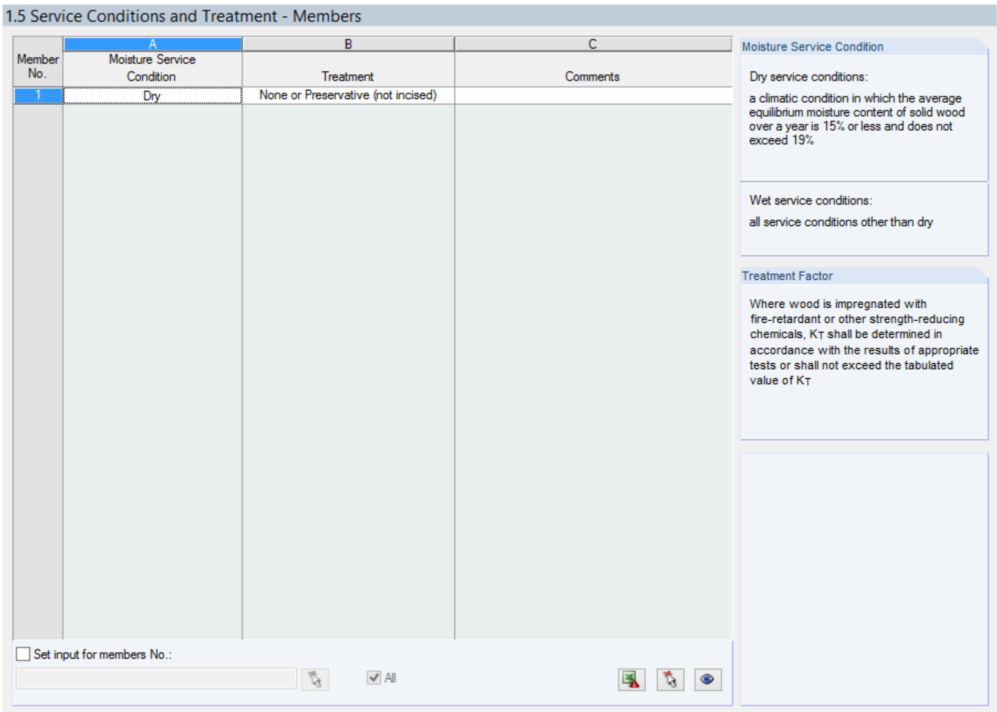


Figure 8.7: Window 1.5 *Service Conditions and Treatment - Members*

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

About the main axis (*Buckling About Axis y*), we can leave the default value of 5.00 m (see [1] Table A.5.5.6.1). For the minor axis (*Buckling About Axis z*), however, we define the buckling length $\mathbf{L_{ez} = 2.50}$ **m** as the column is restrained in the middle of the member (see Figure 8.1). In this case, we can deactivate *Lateral-Torsional Buckling* because the slenderness ratio C_B does not exceed 10 (see [1] Chapter 6.5.6.4.4) so that we get the lateral stability factor K_L = 1.0.

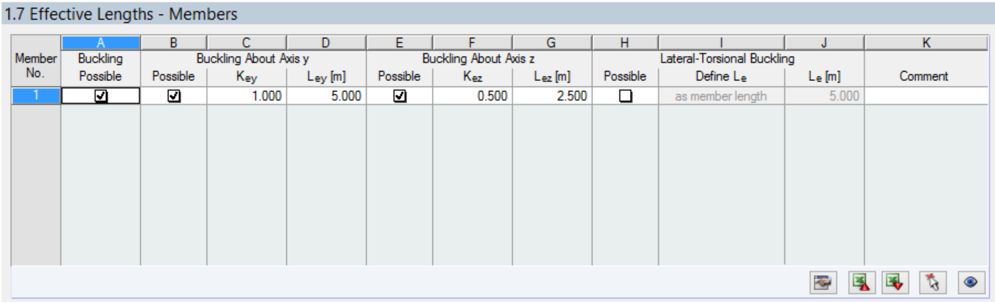


Figure 8.8: Window 1.7 *Effective Lengths – Members*

In window 1.9 *Additional Design Parameters*, the use of the system factor K_H is not required.

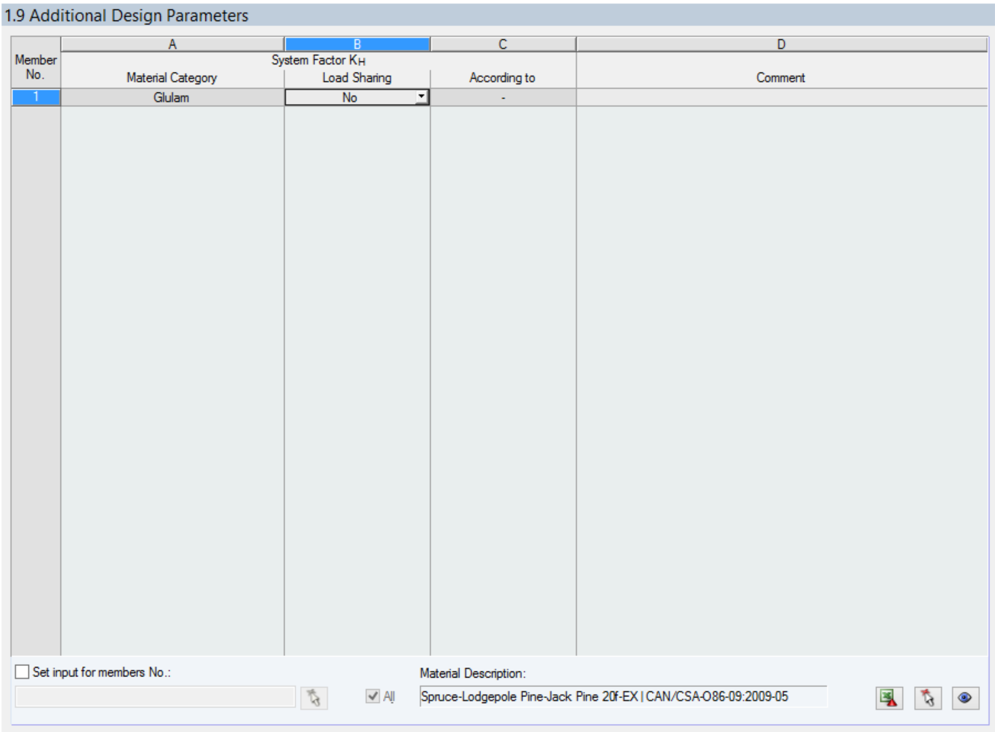
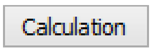


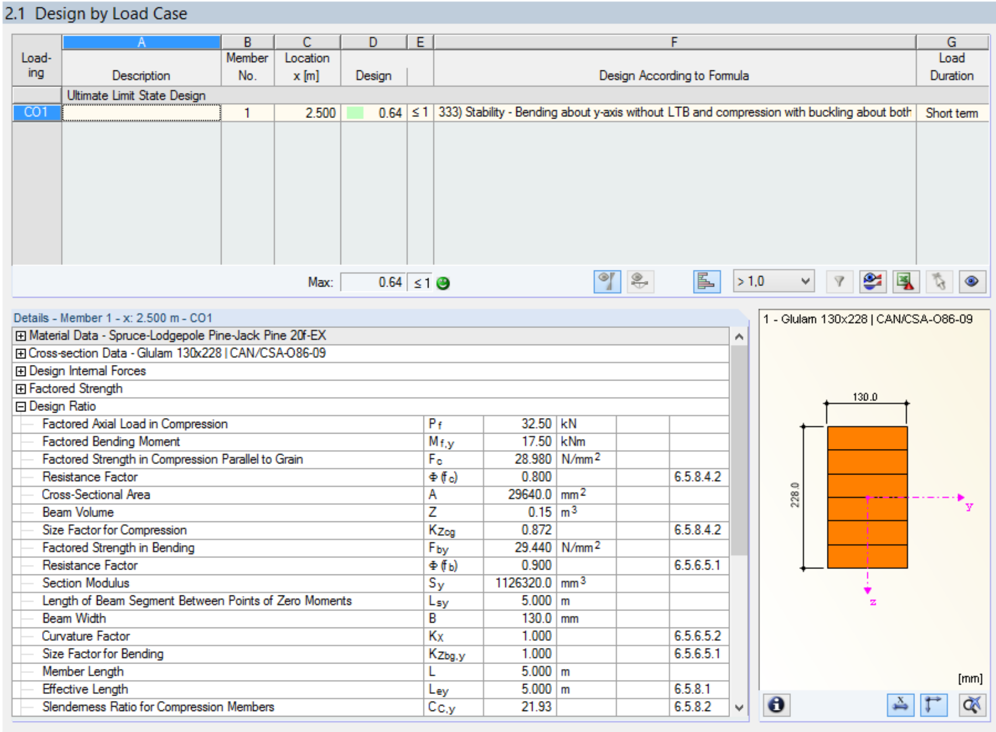
Figure 8.9: Window 1.9 *Additional Design Parameters*



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

Results

After the calculation, the governing design is presented in the 2.1 *Design by Load Case* window.



2.1 Design by Load Case

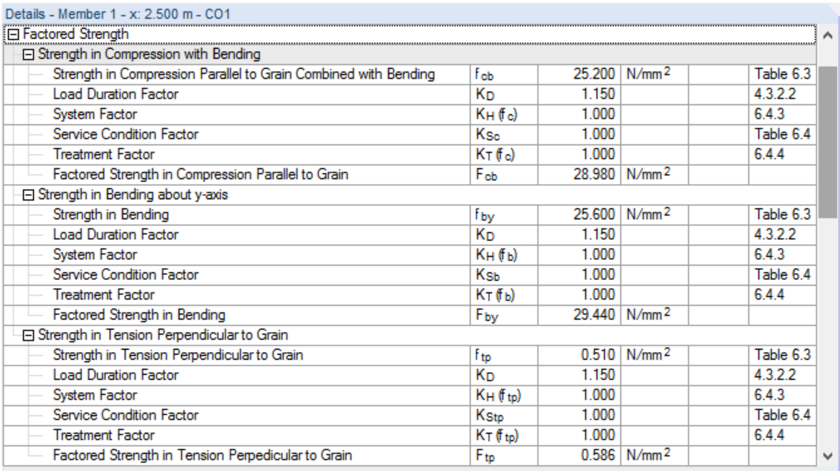
Load-ing	A Description	B Member No.	C Location x [m]	D Design	E	F Design According to Formula	G Load Duration
	Ultimate Limit State Design						
CO1		1	2.500	0.64	≤ 1	333) Stability - Bending about y-axis without LTB and compression with buckling about both	Short term
			Max:	0.64	≤ 1		

Details - Member 1 - x: 2.500 m - CO1

⊞ Material Data - Spruce-Lodgepole Pine-Jack Pine 20f-EX				
⊞ Cross-section Data - Glulam 130x228 | CAN/CSA-O86-09				
⊞ Design Internal Forces				
⊞ Factored Strength				
⊟ Design Ratio				
Factored Axial Load in Compression	P_f	32.50	kN	
Factored Bending Moment	$M_{f,y}$	17.50	kNm	
Factored Strength in Compression Parallel to Grain	F_c	28.980	N/mm²	
Resistance Factor	Φ (f c)	0.800		6.5.8.4.2
Cross-Sectional Area	A	29640.0	mm²	
Beam Volume	Z	0.15	m³	
Size Factor for Compression	K_{Zcg}	0.872		6.5.8.4.2
Factored Strength in Bending	F_{by}	29.440	N/mm²	
Resistance Factor	Φ (f b)	0.900		6.5.6.5.1
Section Modulus	S_y	1126320.0	mm³	
Length of Beam Segment Between Points of Zero Moments	L_{sy}	5.000	m	
Beam Width	B	130.0	mm	
Curvature Factor	K_X	1.000		6.5.6.5.2
Size Factor for Bending	$K_{Zbg,y}$	1.000		6.5.6.5.1
Member Length	L	5.000	m	
Effective Length	L_{ey}	5.000	m	6.5.8.1
Slenderness Ratio for Compression Members	$C_{c,y}$	21.93		6.5.8.2

Figure 8.10: Window 2.1 *Design by Load Case*

The *Details* in the lower part of the window correspond to the design results in [2]. All values you need to comprehend the calculation are listed there.



Details - Member 1 - x: 2.500 m - CO1

⊟ Factored Strength				
⊟ Strength in Compression with Bending				
Strength in Compression Parallel to Grain Combined with Bending	f_{cb}	25.200	N/mm²	Table 6.3
Load Duration Factor	K_D	1.150		4.3.2.2
System Factor	K_H (f c)	1.000		6.4.3
Service Condition Factor	K_{Sc}	1.000		Table 6.4
Treatment Factor	K_T (f c)	1.000		6.4.4
Factored Strength in Compression Parallel to Grain	F_{cb}	28.980	N/mm²	
⊟ Strength in Bending about y-axis				
Strength in Bending	f_{by}	25.600	N/mm²	Table 6.3
Load Duration Factor	K_D	1.150		4.3.2.2
System Factor	K_H (f b)	1.000		6.4.3
Service Condition Factor	K_{Sb}	1.000		Table 6.4
Treatment Factor	K_T (f b)	1.000		6.4.4
Factored Strength in Bending	F_{by}	29.440	N/mm²	
⊟ Strength in Tension Perpendicular to Grain				
Strength in Tension Perpendicular to Grain	f_{tp}	0.510	N/mm²	Table 6.3
Load Duration Factor	K_D	1.150		4.3.2.2
System Factor	K_H (f tp)	1.000		6.4.3
Service Condition Factor	K_{Stp}	1.000		Table 6.4
Treatment Factor	K_T (f tp)	1.000		6.4.4
Factored Strength in Tension Perpedicular to Grain	F_{tp}	0.586	N/mm²	

Figure 8.11: Details - *Factored Strength*

Factored Compressive Resistance

$$P_r = \Phi \times F_c \times A \times K_{Zcg} \times K_C$$

$$F_c = f_{c,b} \times (K_D \times K_H \times K_{Sc} \times K_T)$$
$$F_c = 25.2 \times (1.15 \times 1.0 \times 1.0 \times 1.0)$$
$$F_c = 28.98\ N/mm^2$$

$$K_{Zcg} = 0.68 \times Z^{-0.13} \le 1.0$$
$$K_{Zcg} = 0.68 \times 0.1482^{-0.13} \le 1.0$$
$$K_{Zcg} = 0.872$$

$$K_C = \left[1.0 + \frac{F_C \times K_{Zcg} \times C_c^3}{35 \times E_{05} \times K_{SE} \times K_T}\right]^{-1}$$
$$C_{c,y} = \frac{l_{e,y}}{d} = \frac{5.0}{0.228} = 21.93$$
$$C_{c,z} = \frac{l_{e,z}}{d} = \frac{2.5}{0.13} = 19.23$$
$$E_{05} = 0.87 \times 10300 = 8961\ N/mm^2$$
$$K_{C,y} = \left[1.0 + \frac{28.98 \times 0.872 \times 21.93^3}{35 \times 8961 \times 1.0 \times 1.0}\right]^{-1}$$
$$K_{C,y} = 0.541$$
$$K_{C,z} = \left[1.0 + \frac{28.98 \times 0.872 \times 19.23^3}{35 \times 8961 \times 1.0 \times 1.0}\right]^{-1}$$
$$K_{C,z} = 0.636$$

$$P_{r,y} = 0.8 \times 28.98 \times 29640 \times 0.872 \times 0.541$$
$$P_{r,y} = 323.85\ kN$$

$$P_{r,z} = 0.8 \times 28.98 \times 29640 \times 0.872 \times 0.636$$
$$P_{r,z} = 380.81 kN$$

Factored Bending Moment Resistance

$$M_{r1} = \varphi \times F_b \times S \times K_x \times K_{Zbg}$$
$$M_{r2} = \varphi \times F_b \times S \times K_x$$
LTB is switched off in mask 1.7
→ no factor K_L for M_{r2}

$$F_b = f_{b,y} \times (K_D \times K_H \times K_{Sb} \times K_T)$$
$$F_b = 25.6 \times (1.15 \times 1.0 \times 1.0 \times 1.0)$$
$$F_b = 29.44\ N/mm^2$$

$$K_{Zbg,y} = 1.03 \times (b \times L_{sy})^{-0.18} \le 1.0$$
$$K_{Zbg,y} = 1.03 \times (0.13 \times 5.0)^{-0.18} \le 1.0$$
$$K_{Zbg,y} = 1.0$$

$$M_{r1,y} = 0.9 \times 29.44 \times 1126320 \times 1.0 \times 1.0$$
$$M_{r1,y} = 29.84\ kNm$$

$$M_{r2,y} = 0.9 \times 29.44 \times 1126320 \times 1.0$$
$$M_{r2,y} = 29.84\ kNm$$

Euler Buckling Load

$$P_E = \frac{\pi^2 \times E_{05} \times K_{SE} \times K_T \times I}{L_e^2}$$
$$P_{E,y} = \frac{\pi^2 \times 8961 \times 1.0 \times 1.0 \times 1.284 \times 10^8}{5000^2}$$
$$P_{E,y} = 454.24\ kN$$

Combined Bending and Axial Load

$$\left(\frac{P_f}{P_r}\right)^2 + \frac{M_f}{M_r} \times \left[\frac{1}{1 - \frac{P_f}{P_E}}\right] \le 1.00$$

Buckling about axis y

$$\left(\frac{32.50}{323.85}\right)^2 + \frac{17.50}{29.84} \times \left[\frac{1}{1 - \frac{32.50}{454.24}}\right] = 0.64 \le 1.00$$

Buckling about axis z

$$\left(\frac{32.50}{380.81}\right)^2 + \frac{17.50}{29.84} \times \left[\frac{1}{1 - \frac{32.50}{454.24}}\right] = 0.64 \le 1.00$$

Factored Shear Resistance

$$V_r = \Phi \times F_v \times \frac{2 \times A}{3}$$

$$F_v = f_v \times (K_D \times K_H \times K_{Sv} \times K_T)$$
$$F_v = 1.75 \times (1.15 \times 1.0 \times 1.0 \times 1.0)$$
$$F_v = 2.01\, N/mm^2$$

$$V_{r,z} = 0.9 \times 2.01 \times \frac{2 \times 29640}{3}$$
$$V_{r,z} = 35{,}79\ kN$$

Shear Design

$$\frac{V_{f,z}}{V_{r,z}} = \frac{7.00}{35.79} = 0{,}20 \leq 1.00$$

8.1.3.2 Serviceability Limit State Design

We check the deflection limit for the wind load.

In the **Serviceability Limit State** tab of the 1.1 *General Data* window, we select the load combination **CO3** for design and assign the design situation **Total**.

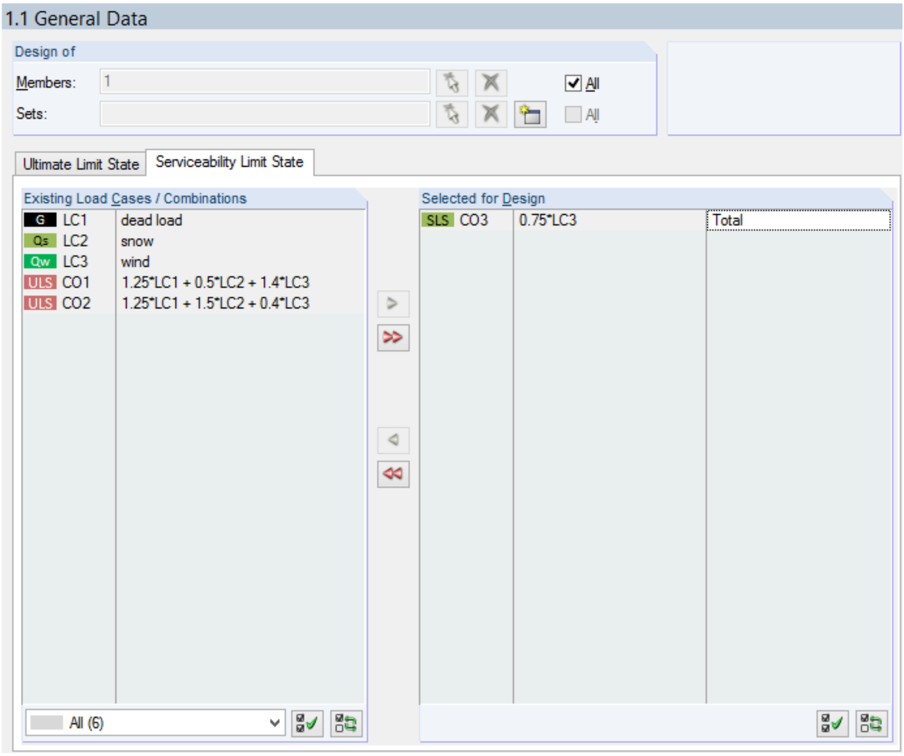
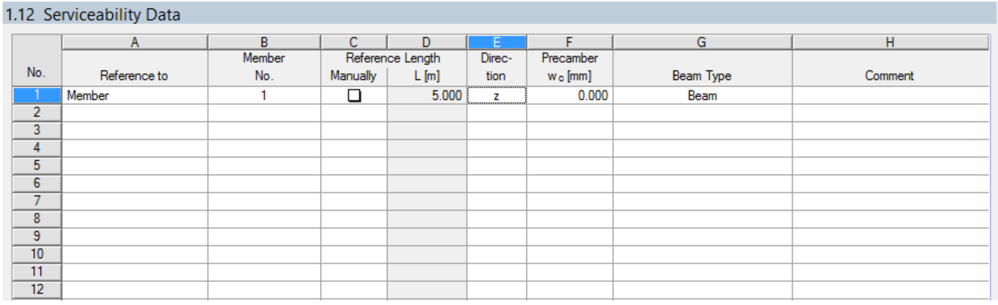


Figure 8.12: Window 1.1 *General Data, tab Serviceability Limit State*

Then we enter member No. **1** in the 1.12 *Serviceability Data* window.


1.12 Serviceability Data

No.	A Reference to	B Member No.	C Reference Length Manually	D L [m]	E Direction	F Precamber w_c [mm]	G Beam Type	H Comment
1	Member	1	☐	5.000	z	0.000	Beam	
2								
3								
4								
5								
6								
7								
8								
9								
10								
11								
12								

Figure 8.13: Window 1.12 *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/180* according to the deflection limits in the *Standard* dialog box.

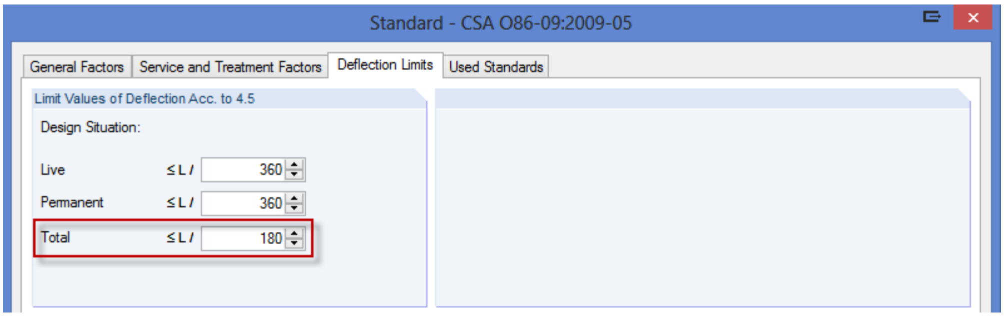


Figure 8.14: Dialog box *Standard, tab Serviceability*

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.

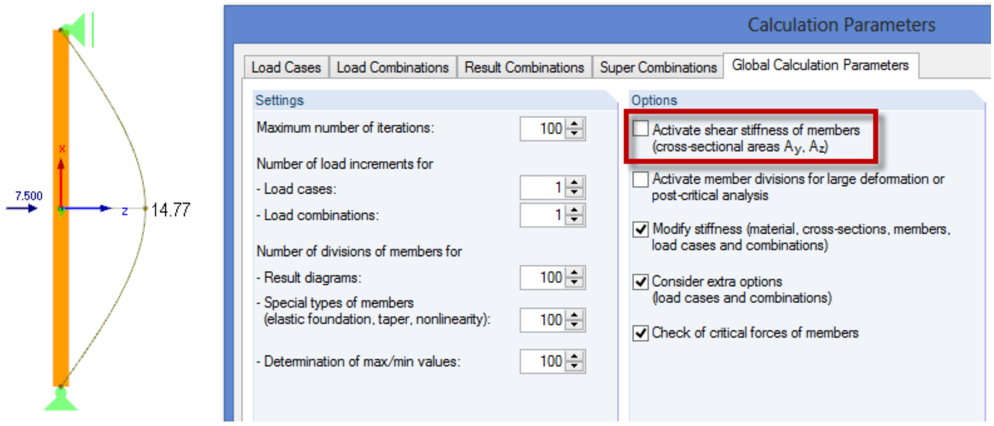


Figure 8.15: Dialog box *Calculation Parameters* in RFEM and deflection of CO3

Deflection Design

$$\frac{\Delta z}{\Delta max,z} \leq 1.0$$

$$\Delta max,z = \frac{L}{180} = \frac{5000}{180} = 27.78\,mm$$

$$\frac{\Delta z}{\Delta max,z} = \frac{14.77}{27.78} = 0.53 \leq 1.0$$

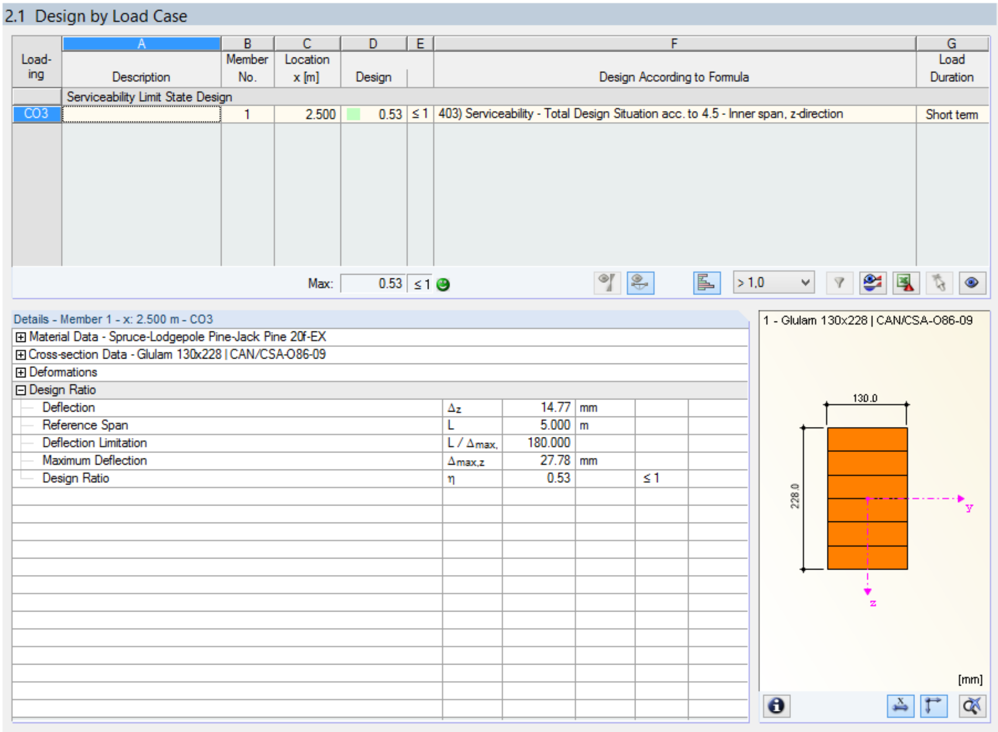


Figure 8.16: Window 2.1 *Design by Load Case*

8.2 Double-Tapered Beam

This example presents the design of a double-tapered beam that is laterally supported over its entire length.

8.2.1 System and Loads

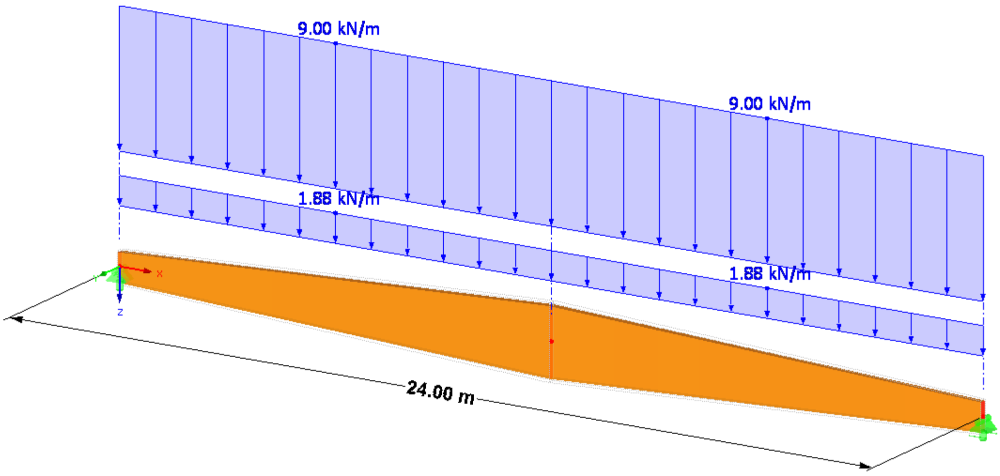


Figure 8.17: System and loads

Model

Material:	Douglas Fir-Larch 20f-E
Length:	24.0 m
Cross-section 1:	180 x 750 mm
Cross-section 2:	180 x 1800 mm
Moisture Condition:	Wet

Loads

LC1 Dead:	1.50 kN/m
LC2 Snow:	6.00 kN/m
Load Combination ULS:	1.25D + 1.5S

8.2.2 Calculation with RFEM

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

We create the considered load combination with relevant factors from 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.

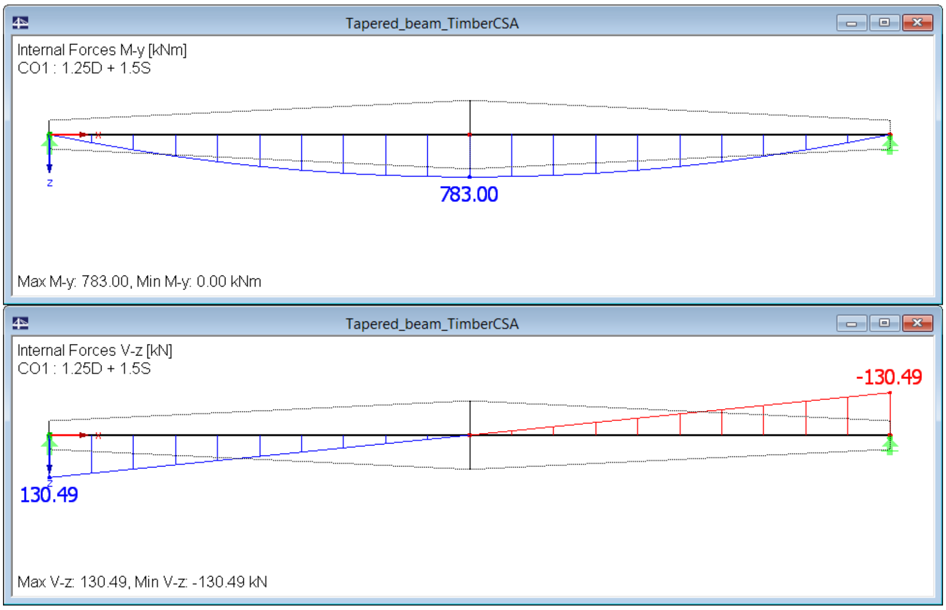


Figure 8.18: Internal forces M_y and V_z

8.2.3 Design with RF-TIMBER CSA

8.2.3.1 Ultimate Limit State Design

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

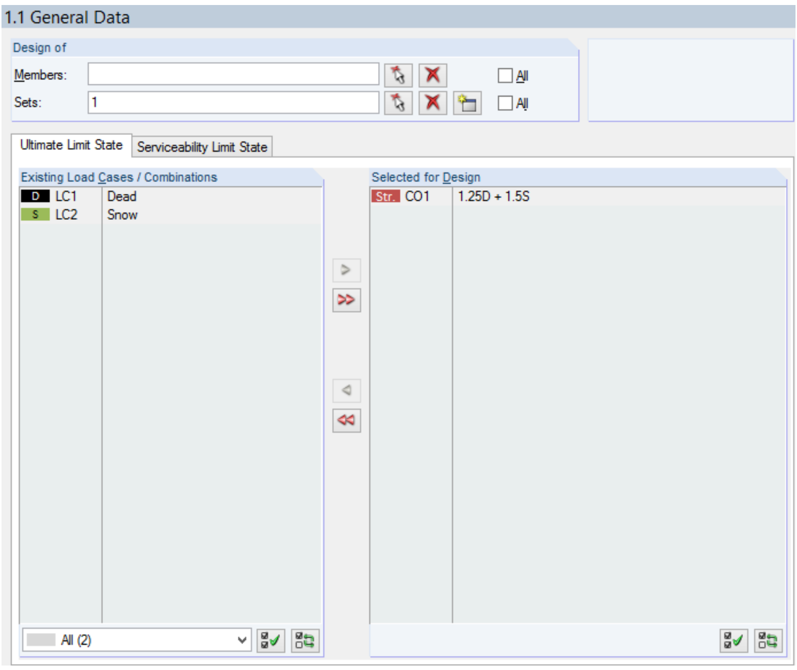


Figure 8.19: Window 1.1 *General Data*

Window 1.2 *Materials* presents the characteristic strengths of the material.

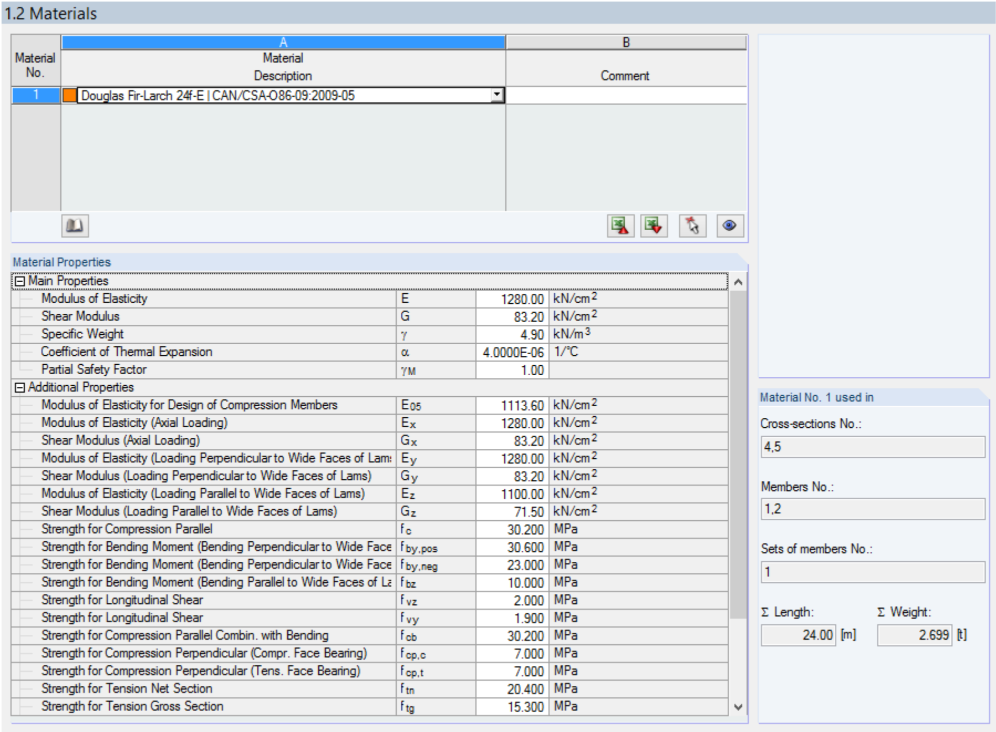


Figure 8.20: Window 1.2 *Materials*

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

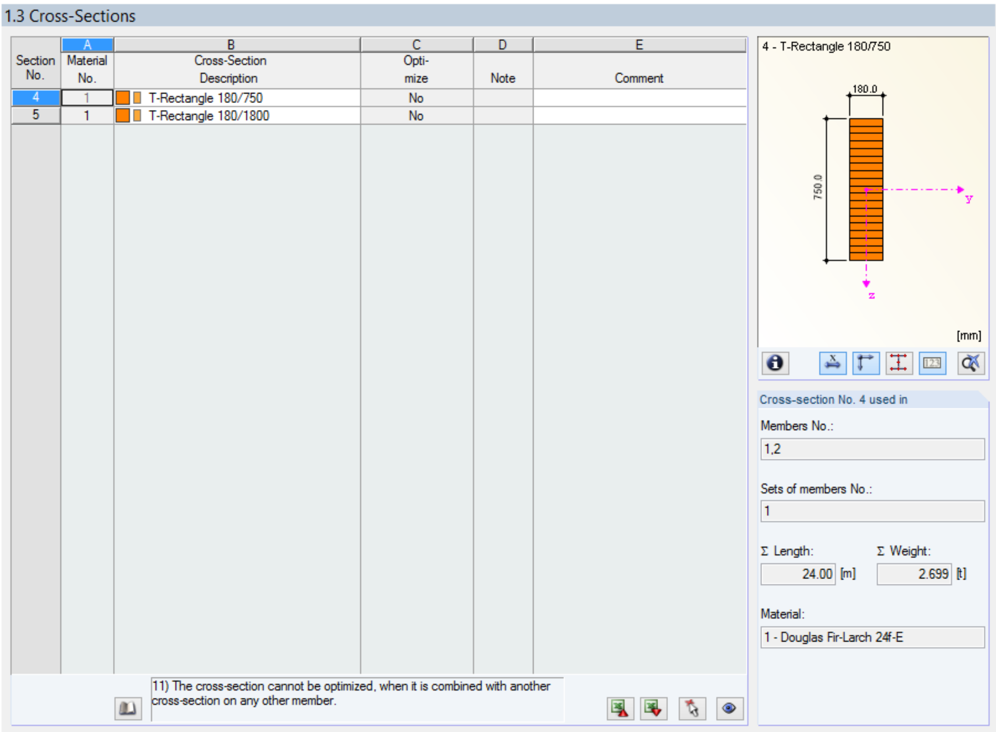


Figure 8.21: Window 1.3 *Cross-Sections*

In Window 1.4 *Load Duration*, we define the load duration. The dead and snow loads are uniformly distributed. Therefore, the loading distribution to determine the size factor, K_{Ztp}, for tension perpendicular to grain is to be set to **Uniform**.

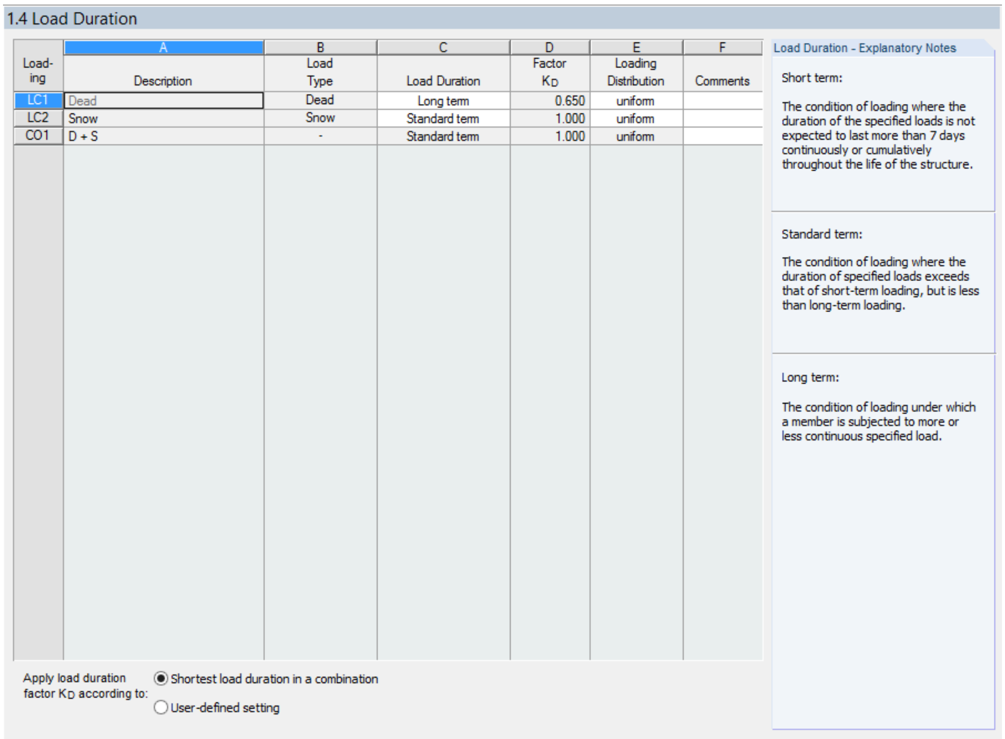


Figure 8.22: Window 1.4 *Load Duration*

In Window 1.6 *Service Conditions and Treatment - Sets of Members*, we define the moisture and treatment parameters. The treatment factor, K_T, will be taken as unity in this case.

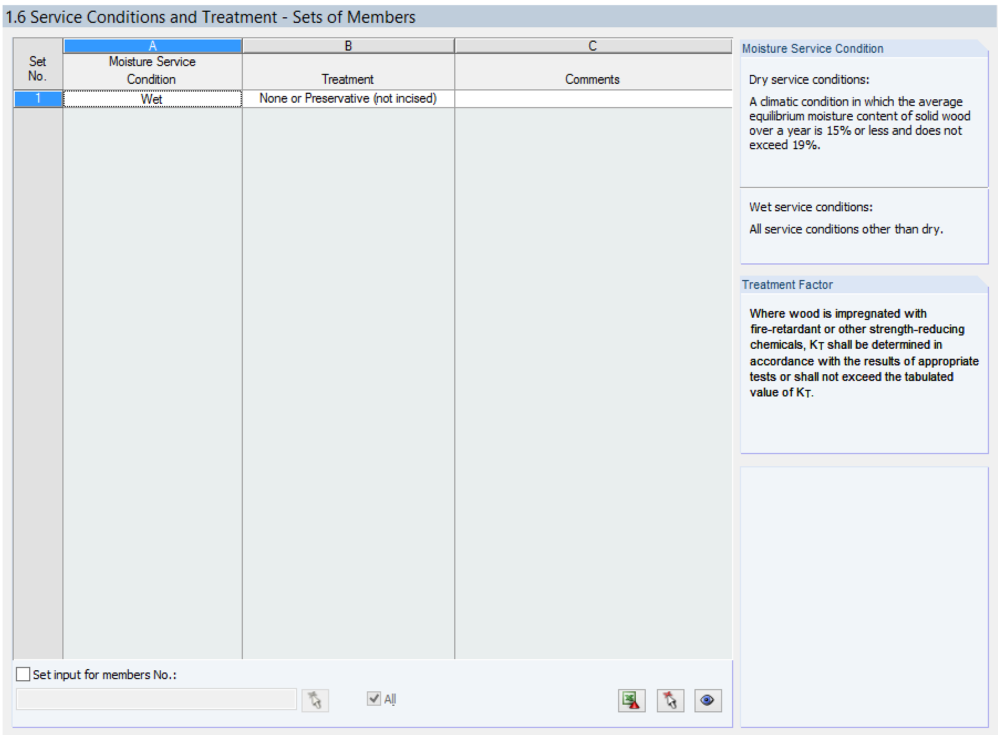


Figure 8.23: Window 1.6 *Service Condition and Treatment - Sets of Members*

In Window 1.8 *Effective Lengths - Sets of Members* we specify the buckling lengths. The set of members is laterally supported over its entire length. Thus, there is no risk of instability. We clear the two check boxes **Buckling Possible** and **Lateral-Torsional Buckling Possible**.

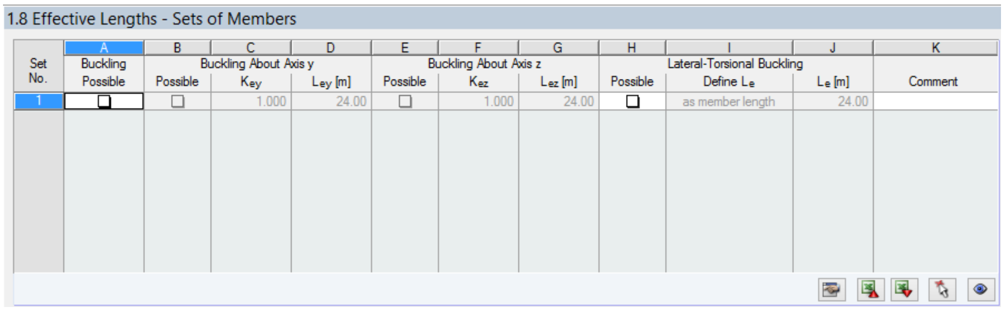


Figure 8.24: Window 1.8 *Effective Lengths - Sets of Members*

In Window 1.9 *Additional Design Parameters*, the application of the system factor, K_H, is not required.

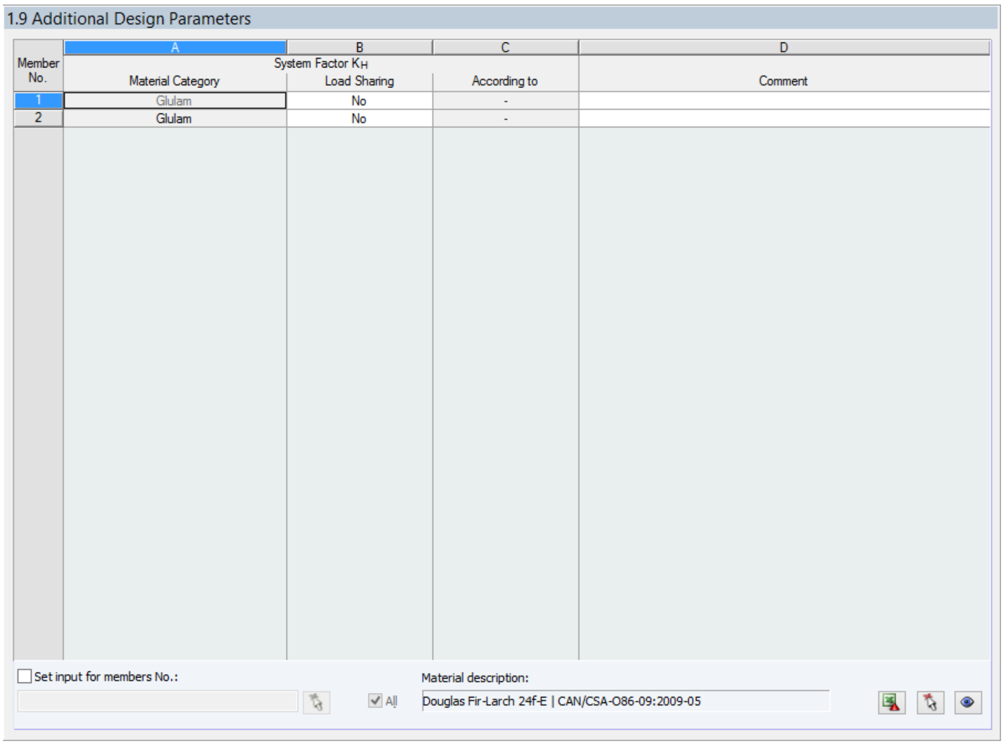


Figure 8.25: Window 1.9 *Additional Design Parameters*

In Window 1.10 *Tapered Members*, the two cross-sections are listed automatically, including the preset angle of taper which is 5°. We activate the **Radial Stress Design** for both cross-sections.

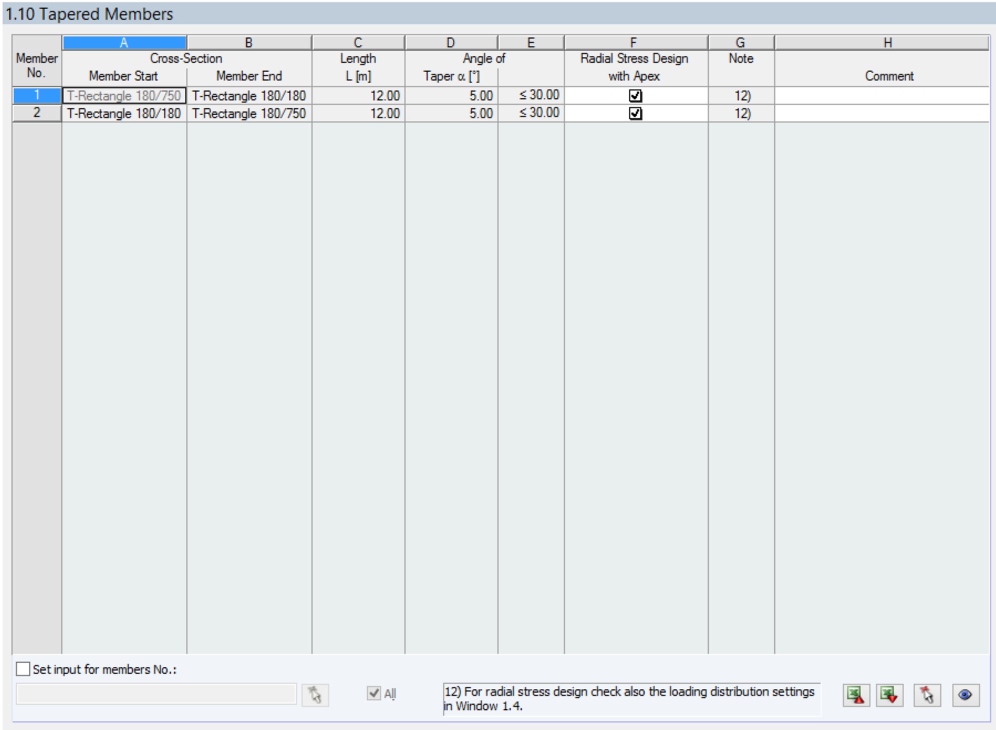


Figure 8.26: Window 1.10 *Tapered Members*

Calculation

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

After the calculation, the governing design ratios can be checked in Window 2.3 *Design by Set of Members*.

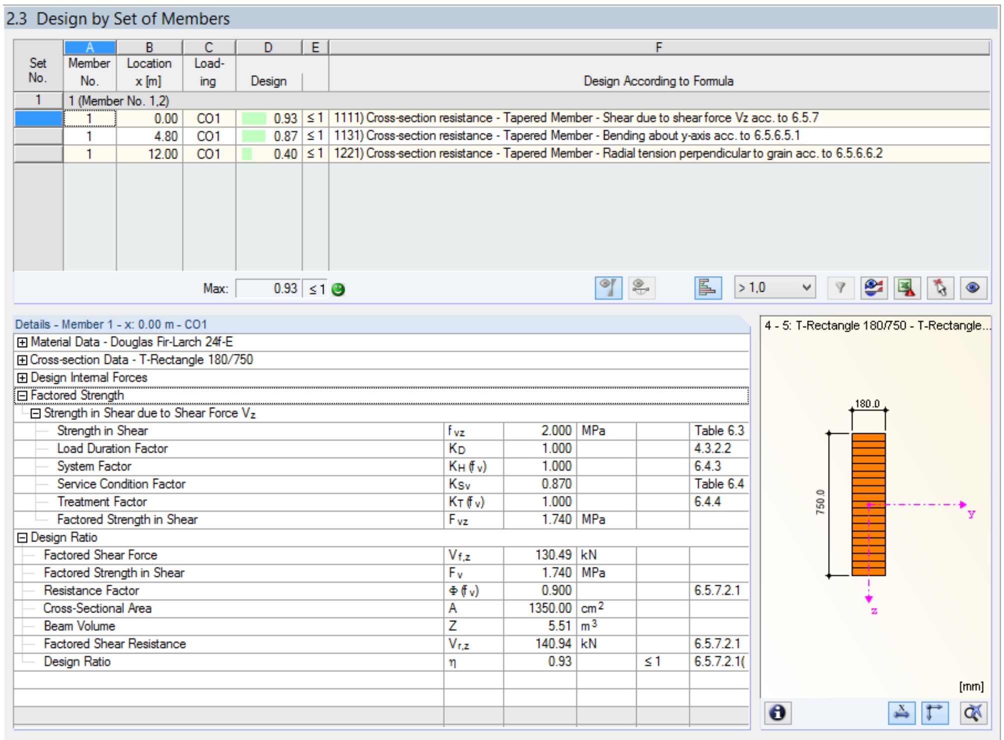


Figure 8.27: Window 2.3 *Design by Set of Members*

Factored Shear Resistance

$$V_r = \varphi \times F_v \times \frac{2}{3} \times A$$

$$F_v = f_v \times (K_D \times K_H \times K_{Sv} \times K_T)$$
$$F_v = 2.0 \times (1.0 \times 1.0 \times 0.87 \times 1.0)$$
$$F_v = 1.74\,\text{MPa}$$

$$V_{r,z} = 0.9 \times 1.74 \times \frac{2}{3} \times 135000$$
$$V_{r,z} = 140.94\,\text{kN}$$

Factored Bending Moment Resistance

$$M_r = \varphi \times F_b \times S \times K_X \times K_{Zbg} \times \frac{1}{1 + 2.7\tan\alpha}$$

$$F_b = f_{b,y} \times (K_D \times K_H \times K_{Sb} \times K_T)$$
$$F_b = 30.6 \times (1.0 \times 1.0 \times 0.8 \times 1.0)$$
$$F_b = 24.48\,\text{MPa}$$

$$K_{Zbg,y} = 1.03 \times (B \times L_{sy})^{-0.18} \leq 1.0$$
$$K_{Zbg,y} = 1.03 \times (0.18 \times 24.00)^{-0.18} \leq 1.0$$
$$K_{Zbg,y} = 0.791$$

$$\frac{1}{1 + 2.7\tan\alpha} = \frac{1}{1 + 2.7\tan 5°} = 0.809$$

$$M_{r,y} = 0.9 \times 24.48 \times 41067000 \times 1.0 \times 0.791 \times 0.809$$
$$M_{r,y} = 579.28\,\text{kNm}$$

Factored Bending Moment Resistance in Tension Perpendicular to Grain

$$M_{rt} = \varphi \times F_{tp} \times S \times K_{Ztp} \times K_R$$

$$F_{tp} = f_{tp} \times (K_D \times K_H \times K_{Stp} \times K_T)$$
$$F_{tp} = 0.83 \times (1.0 \times 1.0 \times 0.85 \times 1.0)$$
$$F_{tp} = 0.706\,\text{MPa}$$

$$K_{Ztp} = \frac{36}{(A \times d)^{0.2}} = \frac{36}{(324000 \times 1800)^{0.2}} = 0.636$$

$$K_R = \left[A + B \times \left(\frac{d}{R}\right) + C \times \left(\frac{d}{R}\right)^2\right]^{-1} = [0.02 + 0 + 0]^{-1} = 50$$

$$M_{rt,y} = 0.9 \times 0.706 \times 97200000 \times 0.636 \times 50$$
$$M_{rt,y} = 1960.92\,\text{kNm}$$

Shear Design

$$\frac{V_{f,z}}{V_{r,z}} = \frac{130.49}{140.94} = 0.93 \leq 1.0$$

Bending Design

$$\frac{M_{f,y}}{M_{r,y}} = \frac{501.12}{579.28} = 0.87 \leq 1.0$$

Radial Stress Design

$$\frac{M_{f,y}}{M_{rt,y}} = \frac{783.0}{1960.92} = 0.40 \leq 1.0$$

8.3 Curved Beam

According to the current standard, the cross-section resistance design is performed also for curved portions of bending members with constant cross-section. In the module, the member of this type is not allowed for stability calculations.

8.3.1 System and Loads

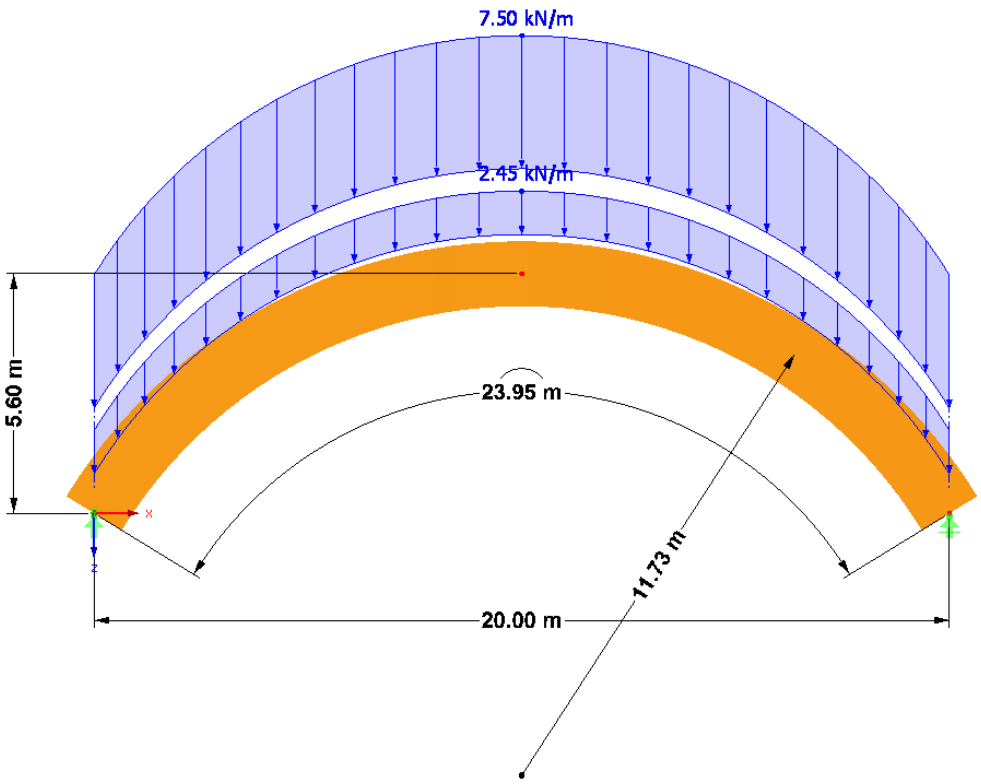


Figure 8.28: System and loads

Model

Material:	Douglas Fir-Larch 20f-EX
Length:	20.0 m
Radius of Curvature at Centerline	11.73m
Cross-section:	b = 215 mm
	d = 1520 mm
Thickness of Lamination:	38 mm
Moisture Condition:	Dry

Loads

LC 1 Dead:	1.96 kN/m
LC 2 Snow:	5.00 kN/m
Load Combination ULS:	1.25D + 1.5S

8.3.2 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 considered load combination with relevant factors from 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.

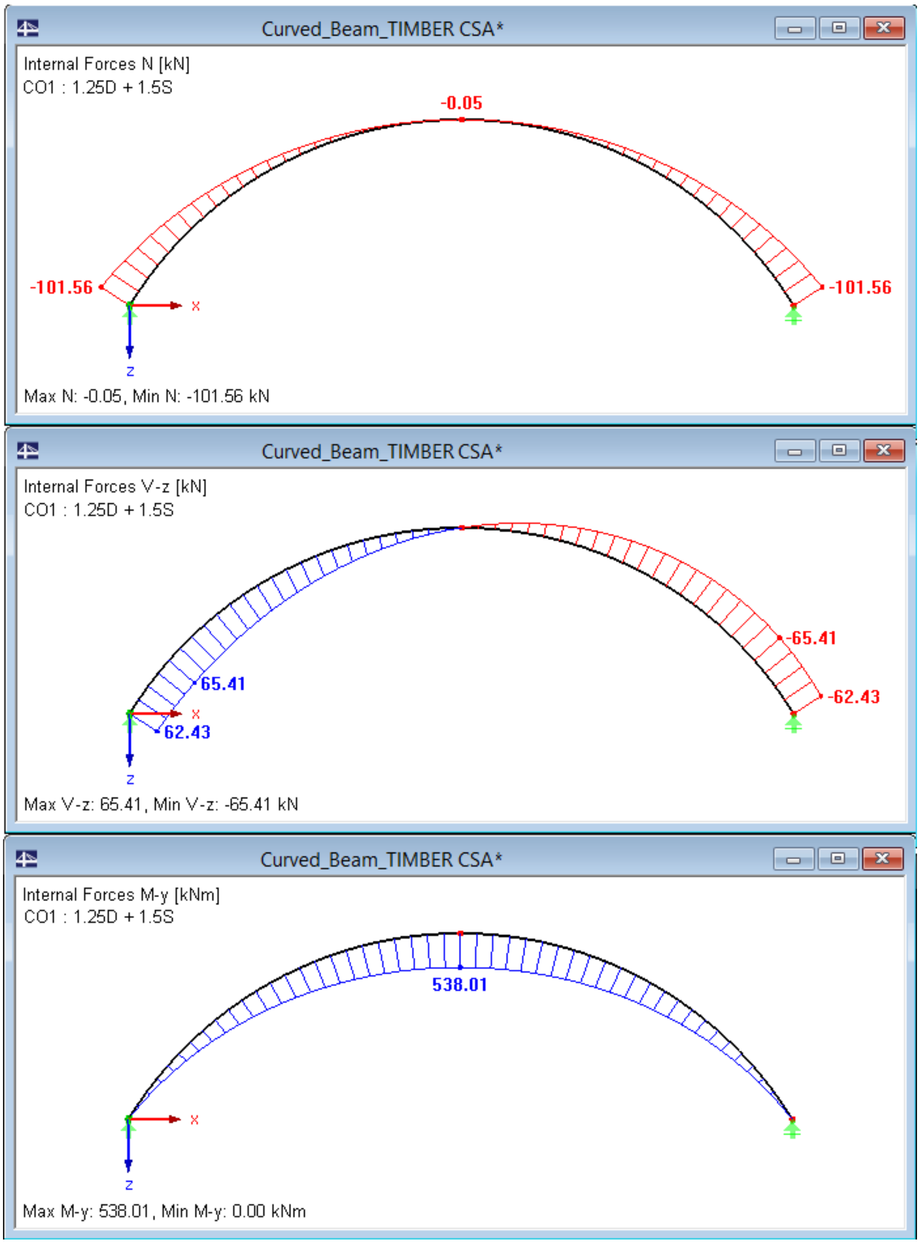


Figure 8.29: Internal forces N, V_z, M_y

8.3.3 Design with RF-TIMBER CSA

8.3.3.1 Ultimate Limit State Design

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

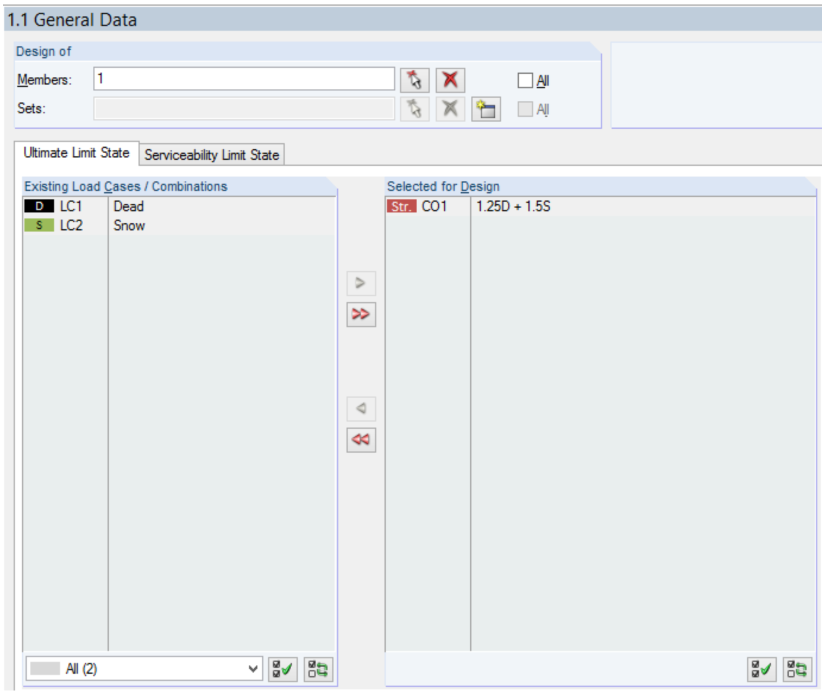


Figure 8.30: Window 1.1 *General Data*

Window 1.2 *Materials* presents the characteristic strengths of the selected material.

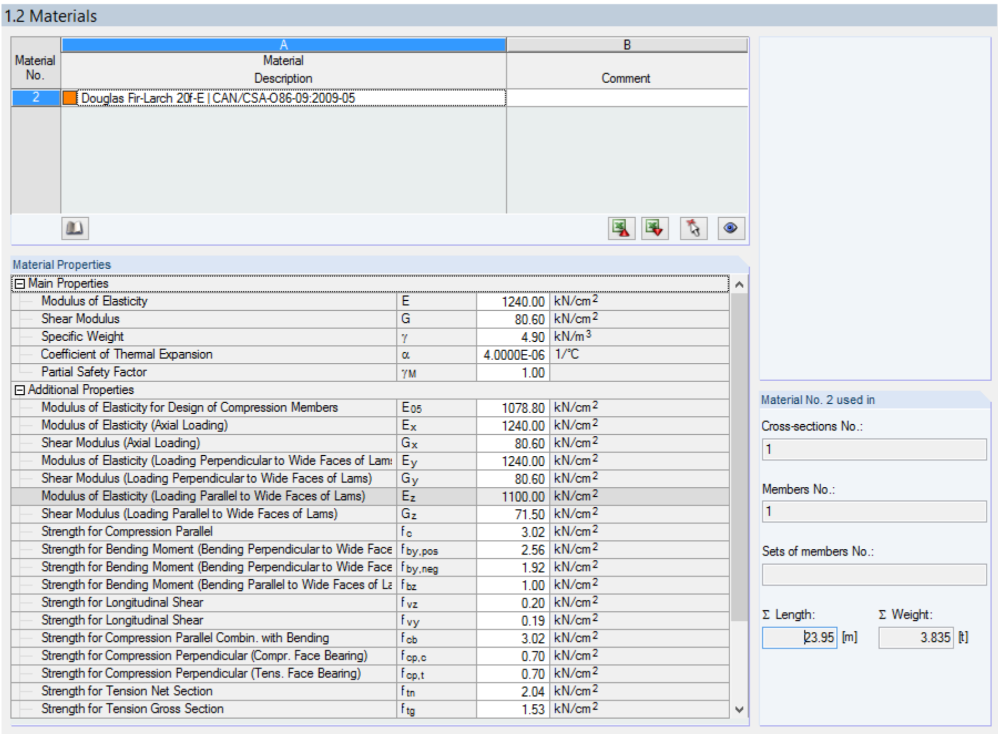


Figure 8.31: Window 1.2 *Materials*

In Window 1.4 *Load Duration*, we allocate **Long term** to the dead load and **Standard term** to the snow load. Both loads have a **uniform** distribution.

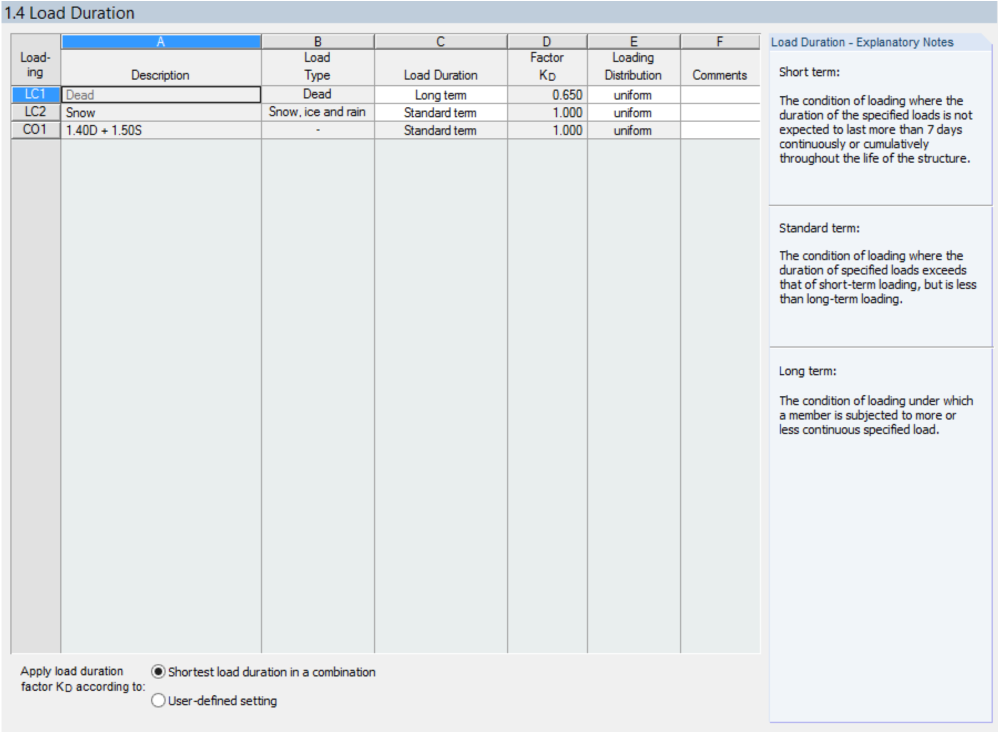


Figure 8.32: Window 1.4 *Load Duration*

In window 1.5 *Service Conditions and Treatment - Members*, we define the moisture and treatment parameters. The treatment factor, K_T, will be taken as unity in this case.

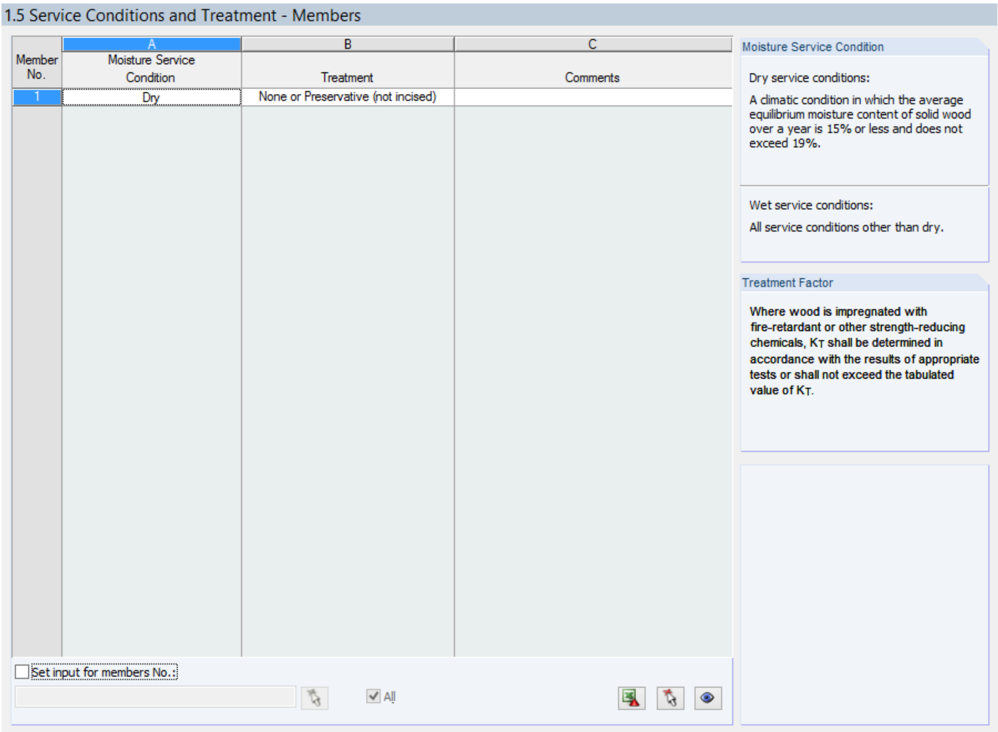
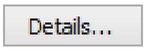


Figure 8.33: Window 1.5 *Service Conditions and Treatment - Members*



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

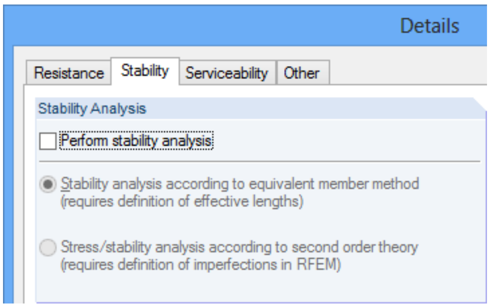


Figure 8.34: Dialog box *Details*, tab *Stability*

In Window 1.9 *Additional Design Parameters*, the application of the system factor, K_H, is not required.

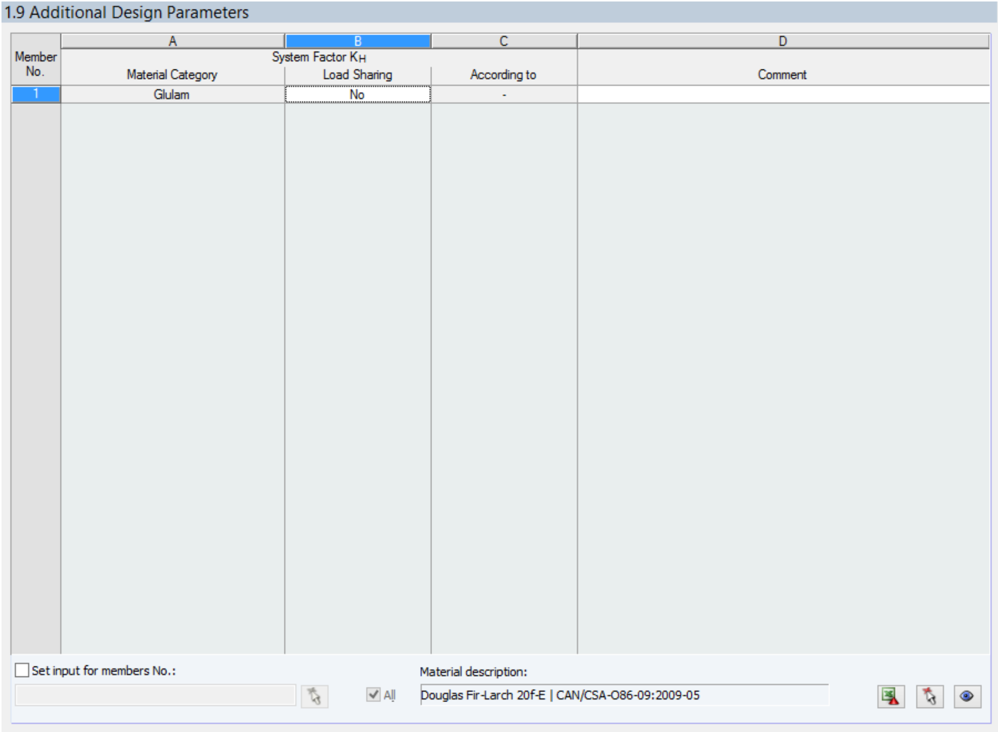
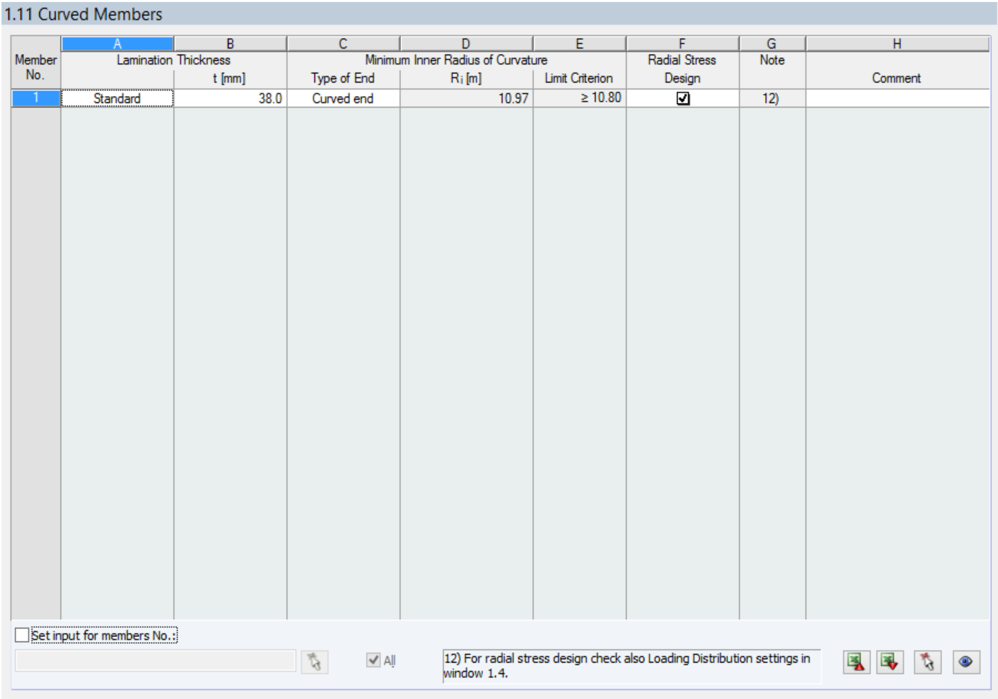


Figure 8.35: Window 1.9 *Additional Design Parameters*

In Window 1.11 *Curved Members*, the **Standard** lamination thickness of 38 mm and the **Curved** type of end are preset. These values are directly reflected in the check of the minimum inner radius of curvature.



1.11 Curved Members

Member No.	A Lamination Thickness	B t [mm]	C Minimum Inner Radius of Curvature Type of End	D R_i [m]	E Limit Criterion	F Radial Stress Design	G Note	H Comment
1	Standard	38.0	Curved end	10.97	≥ 10.80	☑	12)	

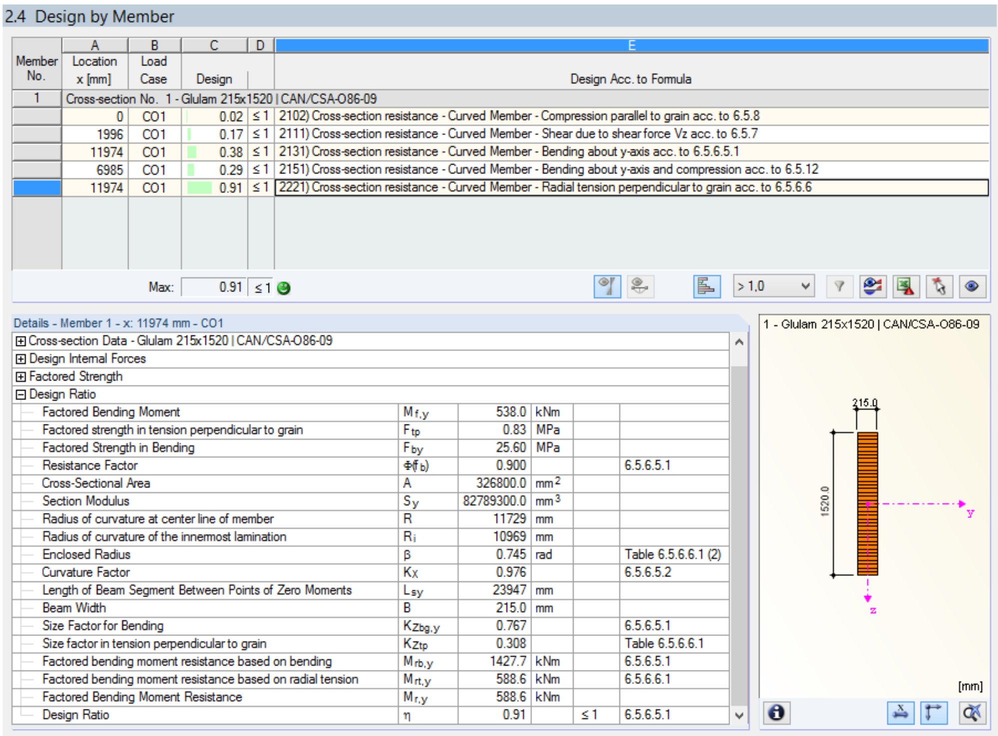
Figure 8.36: Window 1.11 *Curved Members*

To perform the **Radial Stress Design**, we check the corresponding check box in column F.

Calculation

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

The governing design ratios can then be checked in Window 2.4 *Design by Member*.



2.4 Design by Member

Member No.	A Location x [mm]	B Load Case	C Design	D	E Design Acc. to Formula
1	Cross-section No. 1 - Glulam 215x1520 | CAN/CSA-O86-09				
	0	CO1	0.02	≤ 1	2102) Cross-section resistance - Curved Member - Compression parallel to grain acc. to 6.5.8
	1996	CO1	0.17	≤ 1	2111) Cross-section resistance - Curved Member - Shear due to shear force Vz acc. to 6.5.7
	11974	CO1	0.38	≤ 1	2131) Cross-section resistance - Curved Member - Bending about y-axis acc. to 6.5.6.5.1
	6985	CO1	0.29	≤ 1	2151) Cross-section resistance - Curved Member - Bending about y-axis and compression acc. to 6.5.12
	11974	CO1	0.91	≤ 1	2221) Cross-section resistance - Curved Member - Radial tension perpendicular to grain acc. to 6.5.6.6
	Max:		0.91	≤ 1	

Details - Member 1 - x: 11974 mm - CO1

⊞ Cross-section Data - Glulam 215x1520 | CAN/CSA-O86-09					
⊞ Design Internal Forces					
⊞ Factored Strength					
⊟ Design Ratio					
Factored Bending Moment	$M_{f,y}$	538.0	kNm		
Factored strength in tension perpendicular to grain	F_{tp}	0.83	MPa		
Factored Strength in Bending	F_{by}	25.60	MPa		
Resistance Factor	$\Phi(f_b)$	0.900			6.5.6.5.1
Cross-Sectional Area	A	326800.0	mm^2		
Section Modulus	S_y	82789300.0	mm^3		
Radius of curvature at center line of member	R	11729	mm		
Radius of curvature of the innermost lamination	R_i	10969	mm		
Enclosed Radius	β	0.745	rad		Table 6.5.6.6.1 (2)
Curvature Factor	K_X	0.976			6.5.6.5.2
Length of Beam Segment Between Points of Zero Moments	L_{sy}	23947	mm		
Beam Width	B	215.0	mm		
Size Factor for Bending	$K_{Zbg,y}$	0.767			6.5.6.5.1
Size factor in tension perpendicular to grain	K_{Ztp}	0.308			Table 6.5.6.6.1
Factored bending moment resistance based on bending	$M_{rb,y}$	1427.7	kNm		6.5.6.5.1
Factored bending moment resistance based on radial tension	$M_{rt,y}$	588.6	kNm		6.5.6.6.1
Factored Bending Moment Resistance	$M_{r,y}$	588.6	kNm		
Design Ratio	η	0.91		≤ 1	6.5.6.5.1

Figure 8.37: Window 2.4 *Design by Member*

Factored Compressive Resistance

$$P_r = \varphi \times F_c \times A \times K_{Zcg}$$

$$F_c = f_c \times (K_D \times K_H \times K_{Sc} \times K_T)$$
$$F_c = 30.2 \times (1.0 \times 1.0 \times 1.0 \times 1.0)$$
$$F_c = 30.2\,\text{MPa}$$

$$K_{Zcg} = 0.68 \times (Z)^{-0.13} \leq 1.0$$
$$K_{Zcg} = 0.68 \times (7.83)^{-0.13} \leq 1.0$$
$$K_{Zcg} = 0.520$$

$$P_r = 0.8 \times 30.2 \times 326800 \times 0.520$$
$$P_r = 4108.9\,\text{kN}$$

Factored Shear Resistance

$$V_r = \varphi \times F_v \times \frac{2}{3} \times A$$

$$F_v = f_v \times (K_D \times K_H \times K_{Sv} \times K_T)$$
$$F_v = 2.0 \times (1.0 \times 1.0 \times 1.0 \times 1.0)$$
$$F_v = 2.0\,\text{MPa}$$

$$V_{r,z} = 0.9 \times 2.0 \times \frac{2}{3} \times 326800$$
$$V_{r,z} = 392.2\,\text{kN}$$

Factored Bending Moment Resistance Based on Bending Strength

$$M_{rb} = \varphi \times F_b \times S \times K_X \times K_{Zbg}$$

$$F_b = f_{b,y} \times (K_D \times K_H \times K_{Sb} \times K_T)$$
$$F_b = 25.6 \times (1.0 \times 1.0 \times 1.0 \times 1.0)$$
$$F_b = 25.6\,\text{MPa}$$

$$K_X = 1 - 2000\left(\frac{t}{R_i}\right)^2$$
$$K_X = 1 - 2000\left(\frac{38}{10969}\right)^2$$
$$K_X = 0.976$$

$$K_{Zbg,y} = 1.03 \times (B \times L_{sy})^{-0.18} \leq 1.0$$
$$K_{Zbg,y} = 1.03 \times (0.215 \times 23.947)^{-0.18} \leq 1.0$$
$$K_{Zbg,y} = 0.767$$

$$M_{rb,y} = 0.9 \times 25.6 \times 82789300 \times 0.976 \times 0.767$$
$$M_{rb,y} = 1427.7\,\text{kNm}$$

Factored Bending Moment Resistance Based on Radial Tension Strength

$$M_{rt} = \varphi \times F_{tp} \times \frac{2}{3} \times A \times R \times K_{Ztp}$$

$$F_{tp} = f_{tp} \times (K_D \times K_H \times K_{Stp} \times K_T)$$
$$F_{tp} = 0.83 \times (1.0 \times 1.0 \times 1.0 \times 1.0)$$
$$F_{tp} = 0.83\,\text{MPa}$$

$$K_{Ztp} = \frac{24}{(A \times R \times \beta)^{0.2}}$$

The enclosed angle β is measured between those points where the factored bending moment is 85% of the maximum factored bending moment.

If we define the angle β manually, we follow the next steps:

In our example, the angle is between the x-locations with the bending moment of $538.0\,\text{kNm} \times 0.85 = 457.3\,\text{kNm}$.

We can find those locations with the moment 457.3 kNm in the *Result Diagram*.

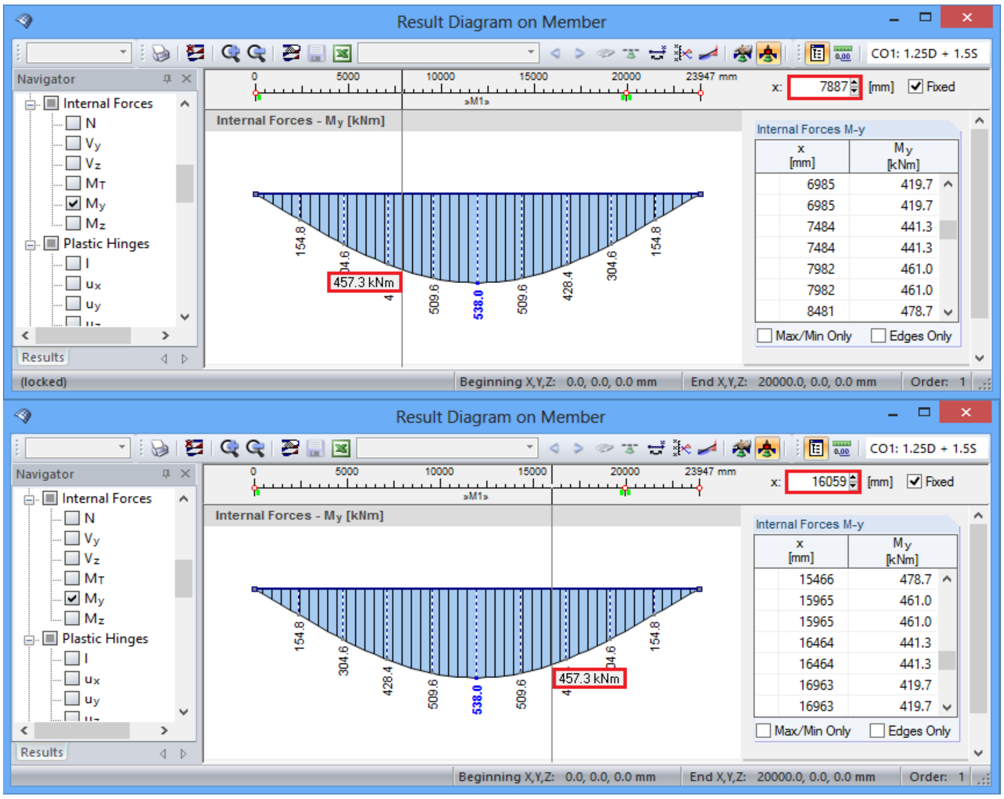


Figure 8.38: Result Diagram with internal force M_y

The two x-locations are

$$x_1 = 7.89\,\text{m}$$
$$x_2 = 16.06\,\text{m}$$

Thus, the arc length of the circular sector is

$$l = x_2 - x_1 = 16.06 - 7.89 = 8.17\,\text{m}$$

Now we can define the enclosed angle in radians as follows.

$$l = R \times \beta$$

$$\beta = \frac{l}{R} = \frac{8.17}{11.73} = 0.697\,\text{rad}$$

In RFEM, the internal division for members is set to 10 by default. Therefore, the value of the angle **β** is **0.745 rad** in RF-TIMBER CSA. For more accurate results, it would be possible to increase the number of division for members in the RFEM dialog box *Calculation Parameters* (see RFEM manual, Chapter 7.3.3).

$$K_{Ztp} = \frac{24}{(215 \times 1520 \times 11729 \times 0.745)^{0.2}} = 0.308$$

$$M_{rt,y} = 0.9 \times 0.83 \times \frac{2}{3} \times 326800 \times 11729 \times 0.308$$

$$M_{rt,y} = 588.6\,\text{kNm}$$

Combined Bending and Axial Load

$$\left[\frac{P_f}{P_r}\right]^2 + \frac{M_{f,y}}{M_{rb,y}} = \left[\frac{20.5}{4108.9}\right]^2 + \frac{419.7}{1427.7} = 0.29 \leq 1.0$$

Shear Design

$$\frac{V_{f,z}}{V_{r,z}} = \frac{65.4}{392.2} = 0.17 \leq 1.0$$

Radial Stress Design

$$\frac{M_{f,y}}{\min(M_{rb,y};M_{rt,y})} = \frac{538.0}{\min(1427.7;588.6)} = \frac{538.0}{588.6} = 0.91 \leq 1.0$$

A Literature

[1] CAN/CSA O86-09: Engineering Design in Wood; May, 2009

[2] Wood Design Manual, Canadian Wood Council, 2010

[3] National Building Code of Canada, 2010

[4] National Design Specification (NDS) Supplement: Design Values for Wood Construction, 2012 Edition

B Index