

**Version
October 2014**

Add-on Module

TIMBER CSA

**Design of Timber Members
According to CSA O86-09**

Program Description

All rights, including those of translations, are reserved.

No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of DLUBAL SOFTWARE GMBH.

© **Dlubal Software GmbH**

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

Contents	Page	Contents	Page
1. Introduction	4	4.7 Governing Internal Forces by Set of Members	44
1.1 Add-on Module TIMBER CSA	4	4.8 Member Slendernesses	45
1.2 TIMBER CSA - Team	5	4.9 Parts List by Member	46
1.3 Using the Manual	5	4.10 Parts List by Set of Members	47
1.4 Open the Add-on Module TIMBER CSA	6	5. Evaluation of Results	48
2. Input Data	8	5.1 Results in RSTAB Model	49
2.1 General Data	8	5.2 Result Diagrams	51
2.1.1 Ultimate Limit State	9	5.3 Filter for Results	52
2.1.2 Serviceability Limit State	11	6. Printout	54
2.1.3 Standard	12	6.1 Printout Report	54
2.2 Materials	15	6.2 Graphic Printout	54
2.3 Cross-Sections	18	7. General Functions	56
2.4 Load Duration	22	7.1 Design Cases	56
2.5 Service Conditions and Treatment - Members	23	7.2 Cross-Section Optimization	58
2.6 Service Conditions and Treatment - Set of Members	24	7.3 Units and Decimal Places	60
2.7 Effective Lengths - Members	25	7.4 Data Transfer	61
2.8 Effective Lengths - Sets of Members	29	7.4.1 Export Material to RSTAB	61
2.9 Additional Design Parameters	30	7.4.2 Export Effective Lengths to RSTAB	61
2.10 Tapered Members	31	7.4.3 Export Results	61
2.11 Serviceability Data	32	8. Examples	63
3. Calculation	33	8.1 Glulam Column	63
3.1 Detail Settings	33	8.1.1 System and Loads	63
3.1.1 Resistance	33	8.1.2 Calculation with RSTAB	64
3.1.2 Stability	34	8.1.3 Design with TIMBER CSA	64
3.1.3 Serviceability	35	8.1.3.1 Ultimate Limit State Design	64
3.1.4 Other	36	8.1.3.2 Serviceability Limit State Design	70
3.2 Start Calculation	37	8.2 Double-Tapered Beam	73
4. Results	38	8.2.1 System and Loads	73
4.1 Design by Load Case	39	8.2.2 Calculation with RSTAB	74
4.2 Design by Cross-Section	40	8.2.3 Design with TIMBER CSA	74
4.3 Design by Set of Members	41	8.2.3.1 Ultimate Limit State Design	74
4.4 Design by Member	42	A Literature	81
4.5 Design by x-Location	42	B Index	82
4.6 Governing Internal Forces by Member	43		

1. Introduction

1.1 Add-on Module 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 RSTAB add-on module TIMBER CSA from the company DLUBAL all users obtain a powerful tool for the design of timber structures according to this Standard.

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 RSTAB. 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 RSTAB. Separate design cases allow for a separate design of large systems or analysis of variants.

TIMBER CSA is one of the add-on modules integrated in the RSTAB environment. Thus, the design-relevant input data is preset when you open the module. Subsequent to the design, you can use the graphical RSTAB 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 TIMBER CSA.

Your DLUBAL Team

1.2 TIMBER CSA - Team

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

Program coordination

Georg Dluba
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 Dluba
Robert Kolouch

Jan Milér

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 RSTAB. The present manual focuses on typical features of the 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.dluba.de/blog/en> where you can search the articles by specific terms.

1.4 Open the Add-on Module TIMBER CSA

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

Menu

To start the program in the RSTAB menu bar, click

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

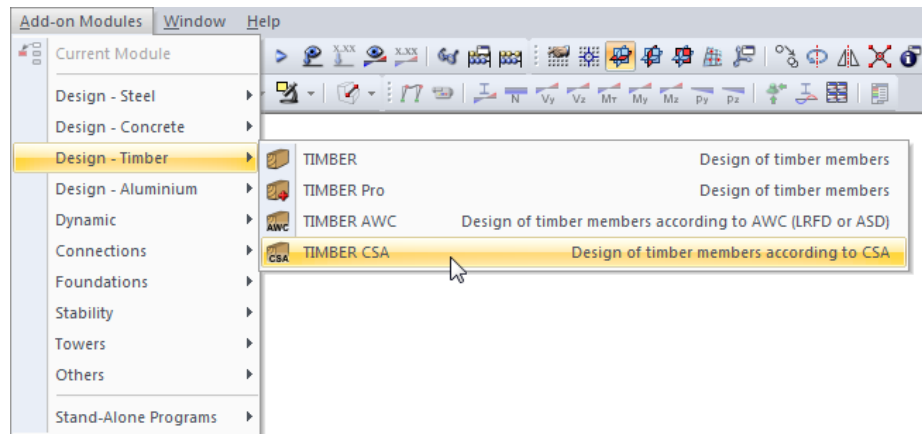


Figure 1.1: Menu: *Add-on Modules* → *Design - Timber* → *TIMBER CSA*

Navigator

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

Add-on Modules → TIMBER CSA.

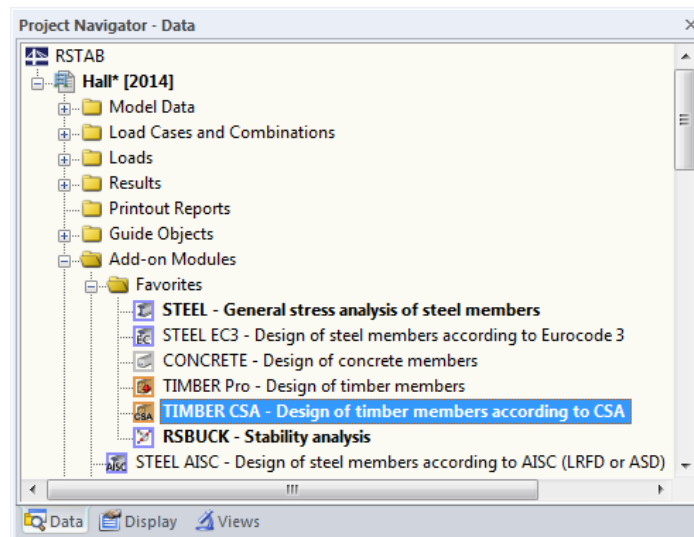
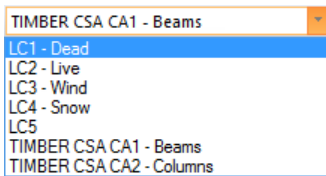


Figure 1.2: Data navigator: *Add-on Modules* → *TIMBER CSA*



TIMBER CSA

Panel

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

Set the relevant TIMBER CSA design case in the load case list of the RSTAB 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 [TIMBER CSA] in the panel to open the module.

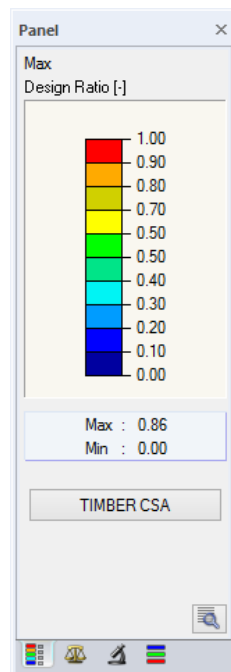


Figure 1.3: Panel button [TIMBER CSA]

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 56).

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

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



OK Cancel

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 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, result combinations, and super combinations for the different designs.

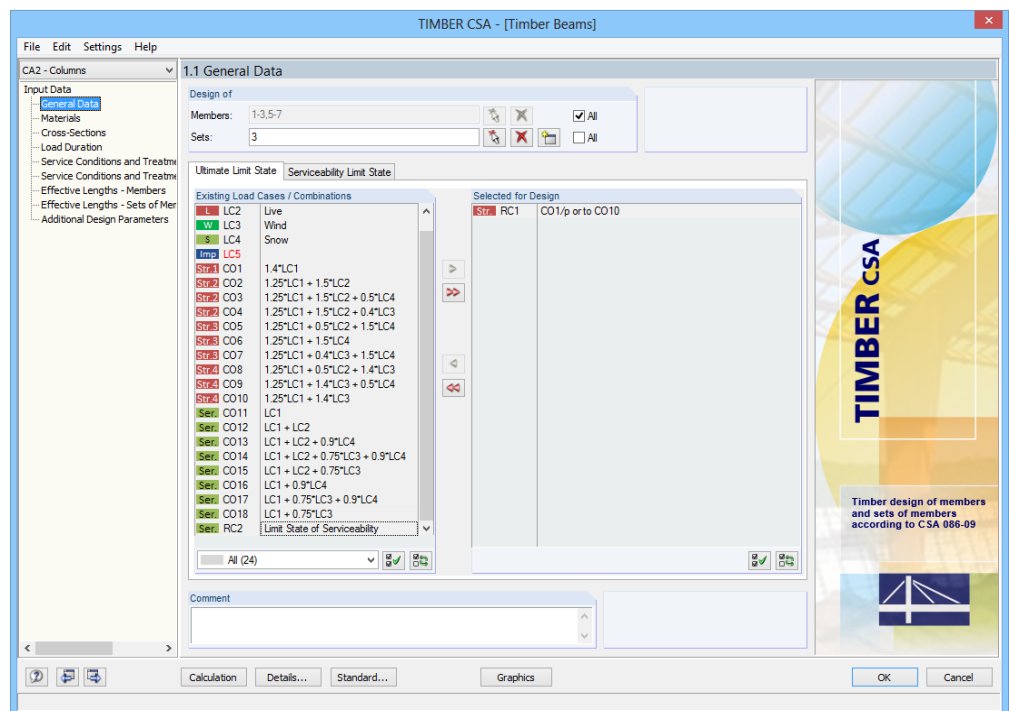


Figure 2.1: Window 1.1 *General Data*

Design of

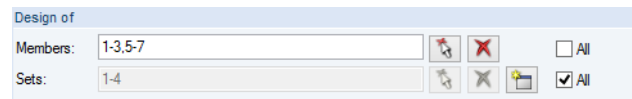


Figure 2.2: Design of members and sets of members

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 RSTAB work window after clicking [\wedge].

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 RSTAB 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

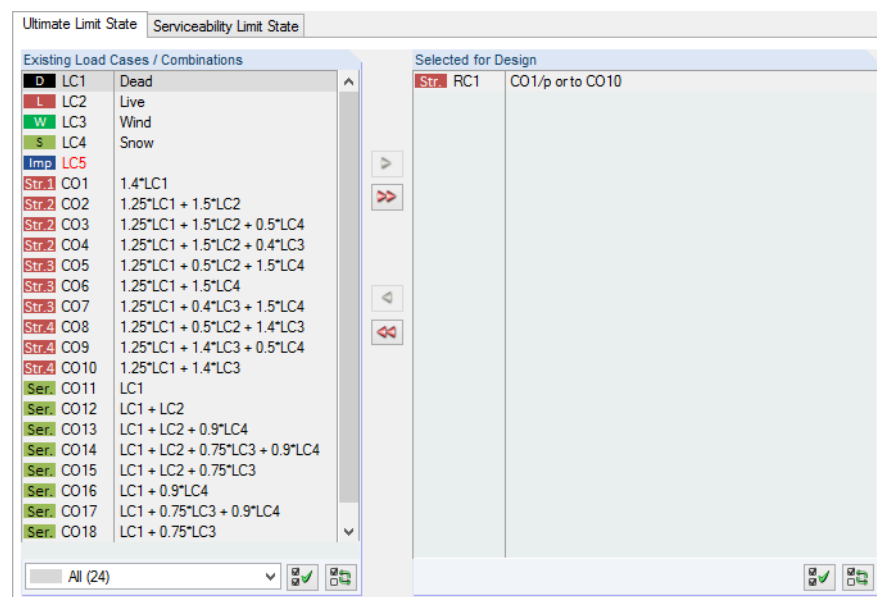


Figure 2.4: Window 1.1 General Data, tab Ultimate Limit State

Existing Load Cases / Combinations

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

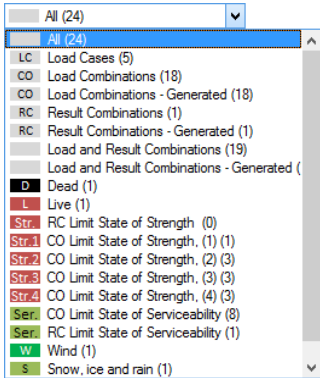


To transfer selected entries to the *Selected for Design* list on the right, click [►]. Alternatively, you can double-click the items. To transfer the complete list to the right, click [►►].

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

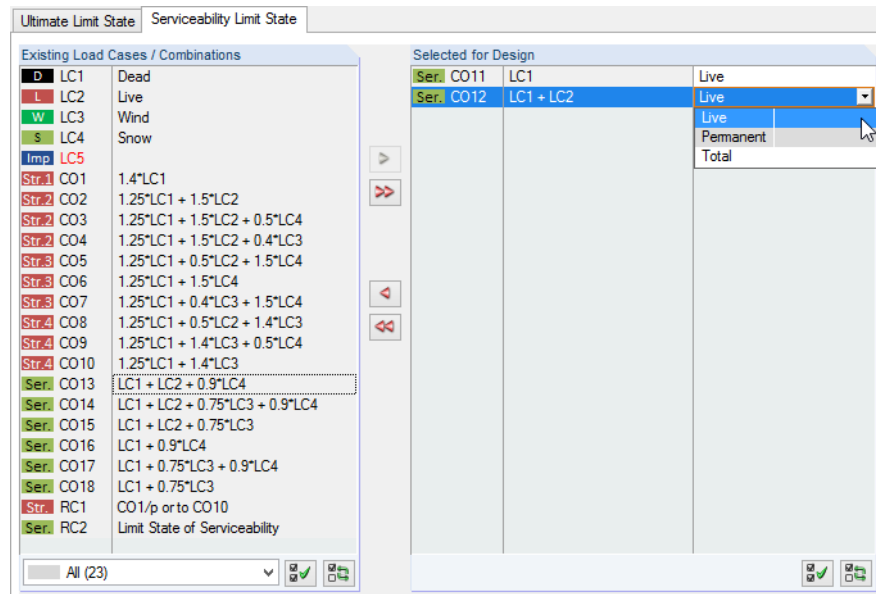


Figure 2.5: Window 1.1 General Data, tab Serviceability Limit State

Existing Load Cases and Combinations

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

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 2.5Figure 2.4).

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 2.11, page 60).



Standard...

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.

Standard...

General Factors

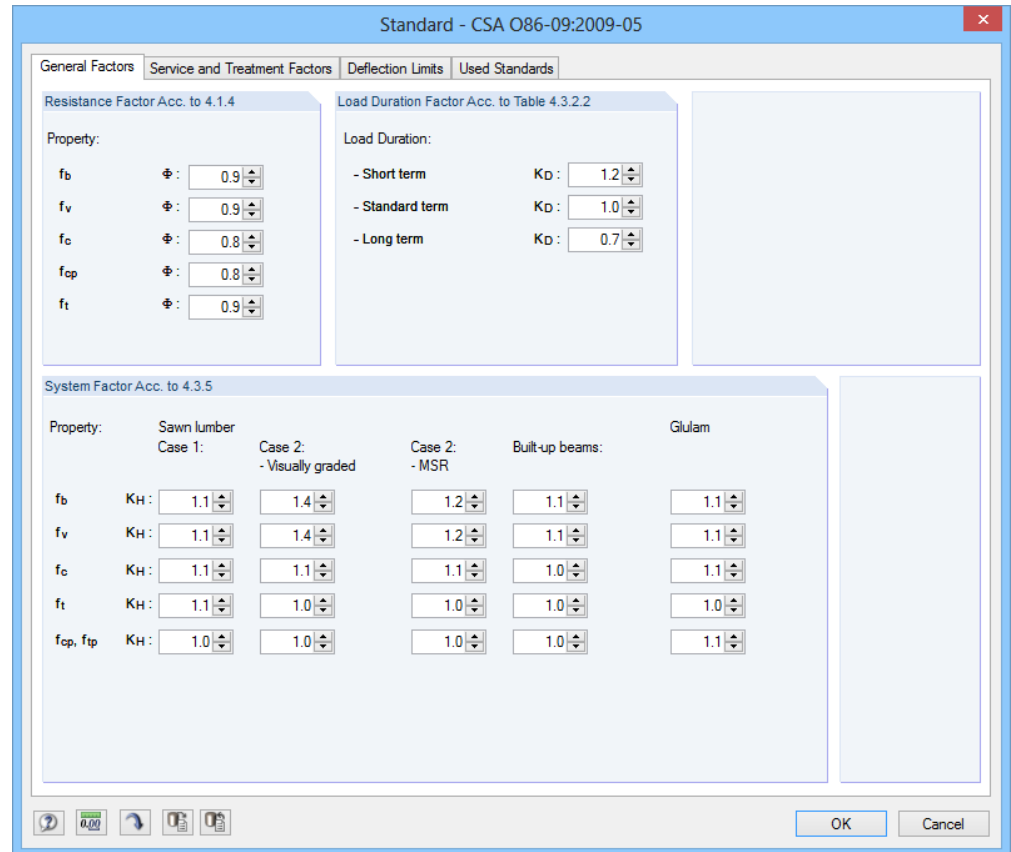


Figure 2.6: Dialog box *Standard*, tab *General Factors*

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:





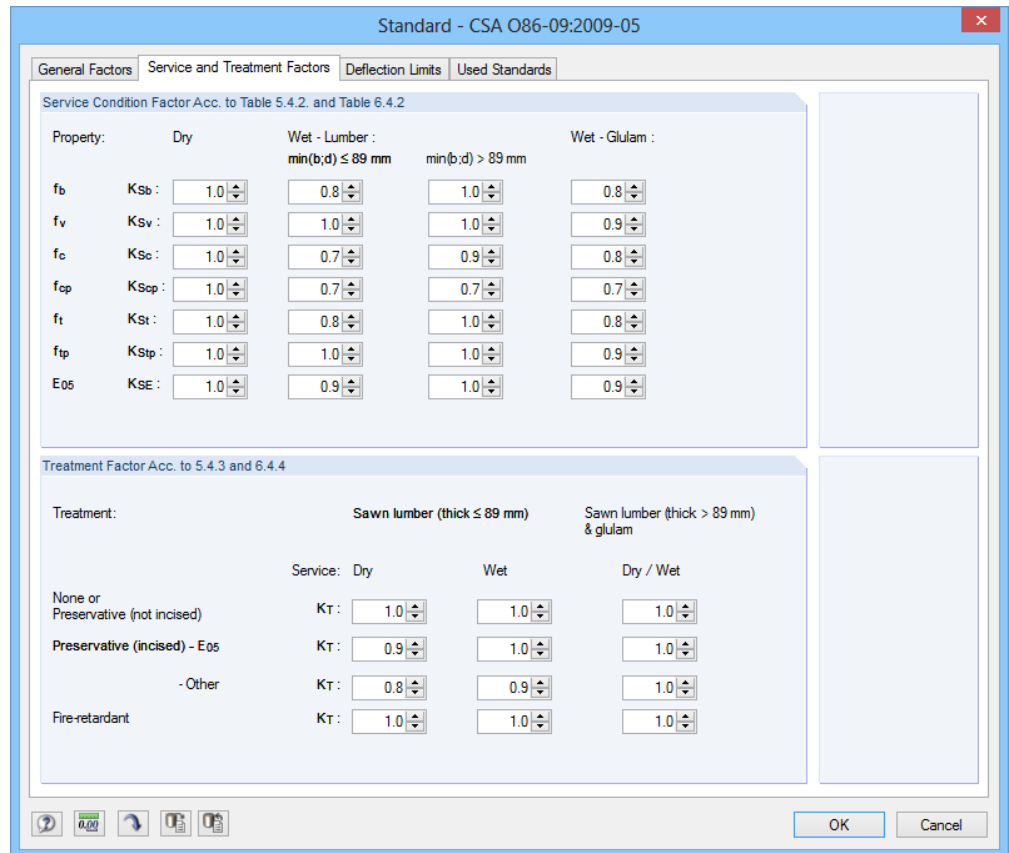
Button	Function
	Resets the program's default settings
	Imports user-defined standard settings
	Saves the current settings as default
	Deletes a user-defined Standard

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.



Standard - CSA O86-09:2009-05

General Factors | **Service and Treatment Factors** | Deflection Limits | Used Standards

Service Condition Factor Acc. to Table 5.4.2. and Table 6.4.2

Property:	Dry	Wet - Lumber :		Wet - Glulam :
		min(b,d) ≤ 89 mm	min(b,d) > 89 mm	
fb	KSb : 1.0	0.8	1.0	0.8
fv	KSv : 1.0	1.0	1.0	0.9
fc	KSc : 1.0	0.7	0.9	0.8
fcp	KSop : 1.0	0.7	0.7	0.7
ft	KSt : 1.0	0.8	1.0	0.8
ftp	KStp : 1.0	1.0	1.0	0.9
E05	KSE : 1.0	0.9	1.0	0.9

Treatment Factor Acc. to 5.4.3 and 6.4.4

Treatment:	Sawn lumber (thick ≤ 89 mm)		Sawn lumber (thick > 89 mm) & glulam
	Service: Dry	Wet	Dry / Wet
None or Preservative (not incised)	KT : 1.0	1.0	1.0
Preservative (incised) - E05	KT : 0.9	1.0	1.0
- Other	KT : 0.8	0.9	1.0
Fire-retardant	KT : 1.0	1.0	1.0

OK Cancel

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.

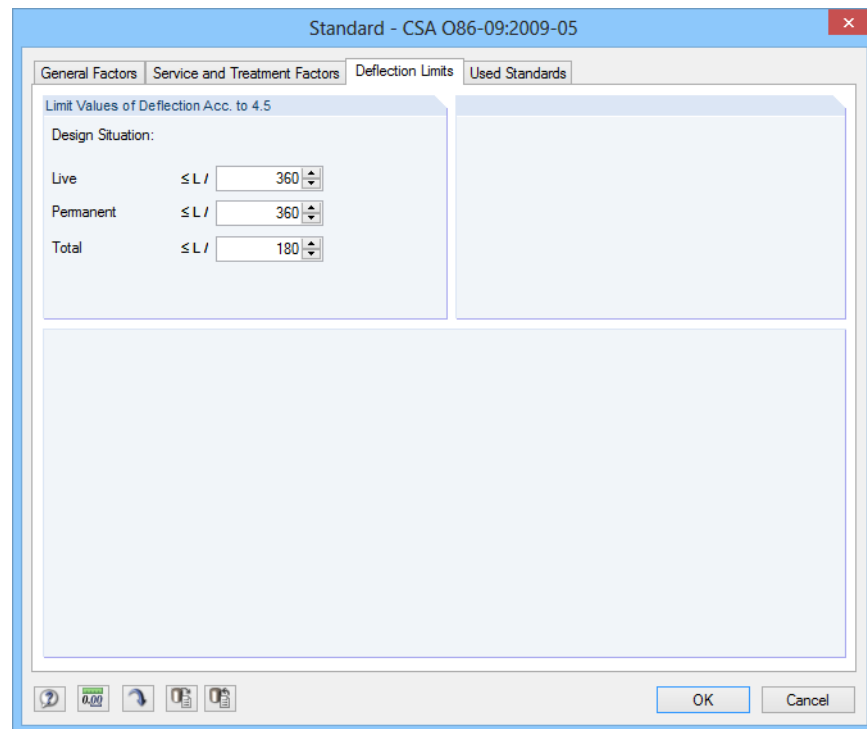


Figure 2.8: Dialog box *Standard*, tab *Deflection Limits*

Used Standards

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

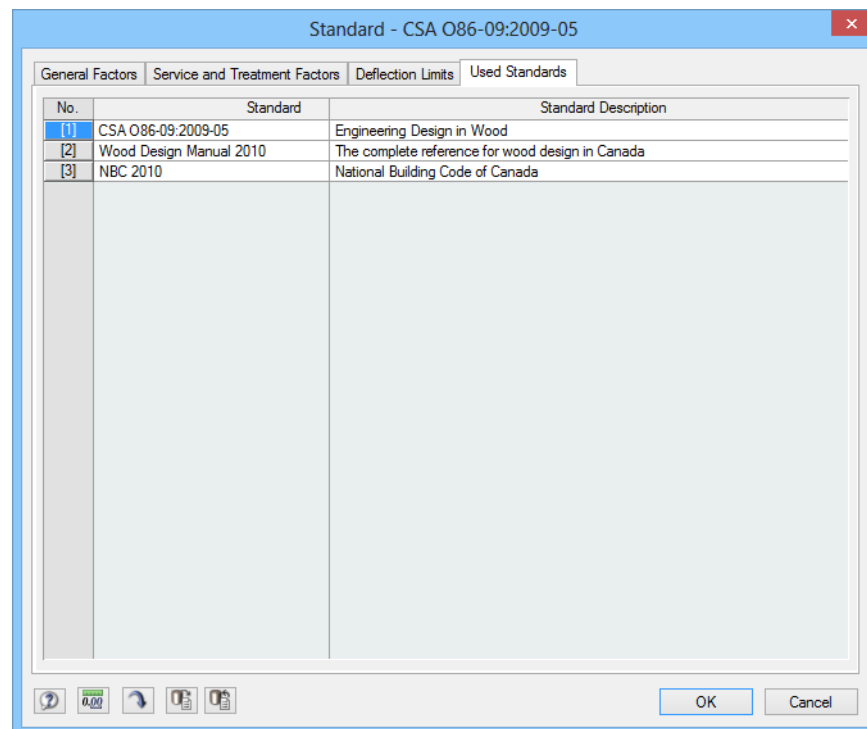


Figure 2.9: Dialog box *Standard*, tab *Used Standards*

2.2 Materials

The window consists of two parts. In the upper part, all materials created in RSTAB 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.

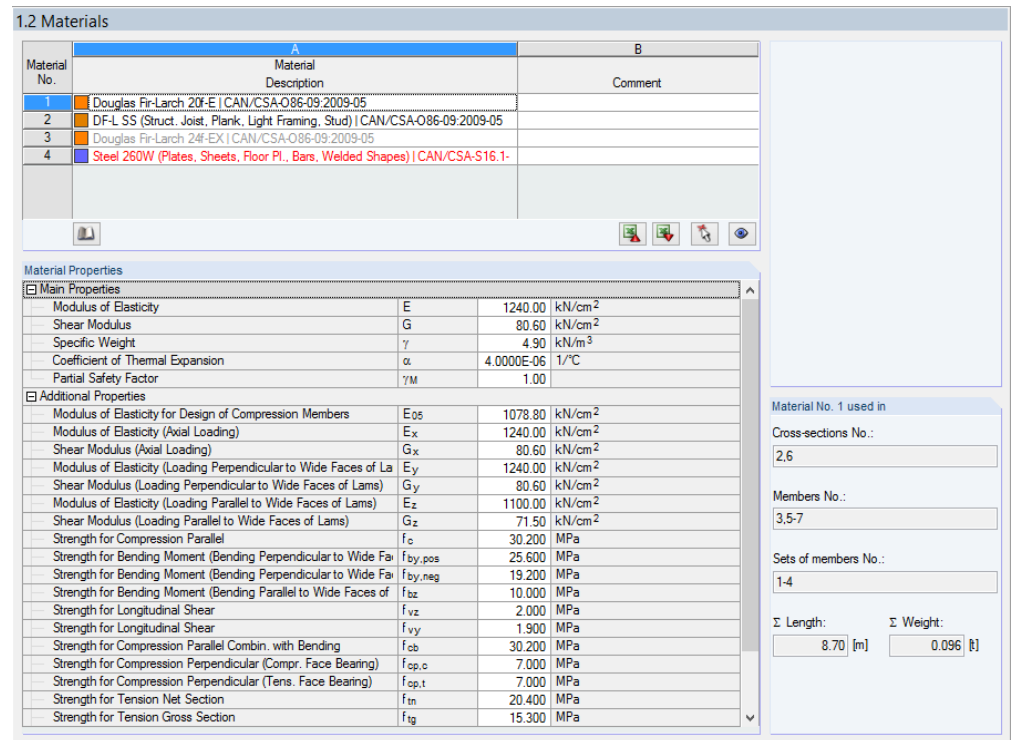


Figure 2.10: Window 1.2 Materials

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 4.2 of the RSTAB 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 60).

Material Description

The materials defined in RSTAB 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.

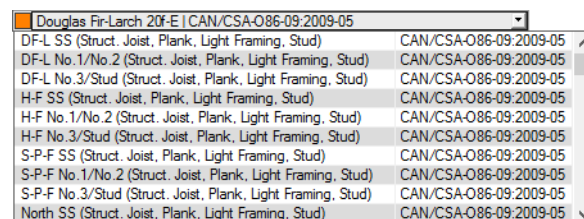


Figure 2.11: List of materials

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, TIMBER CSA will import the material properties, too.

Principally, it is not possible to edit the material properties in the 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.

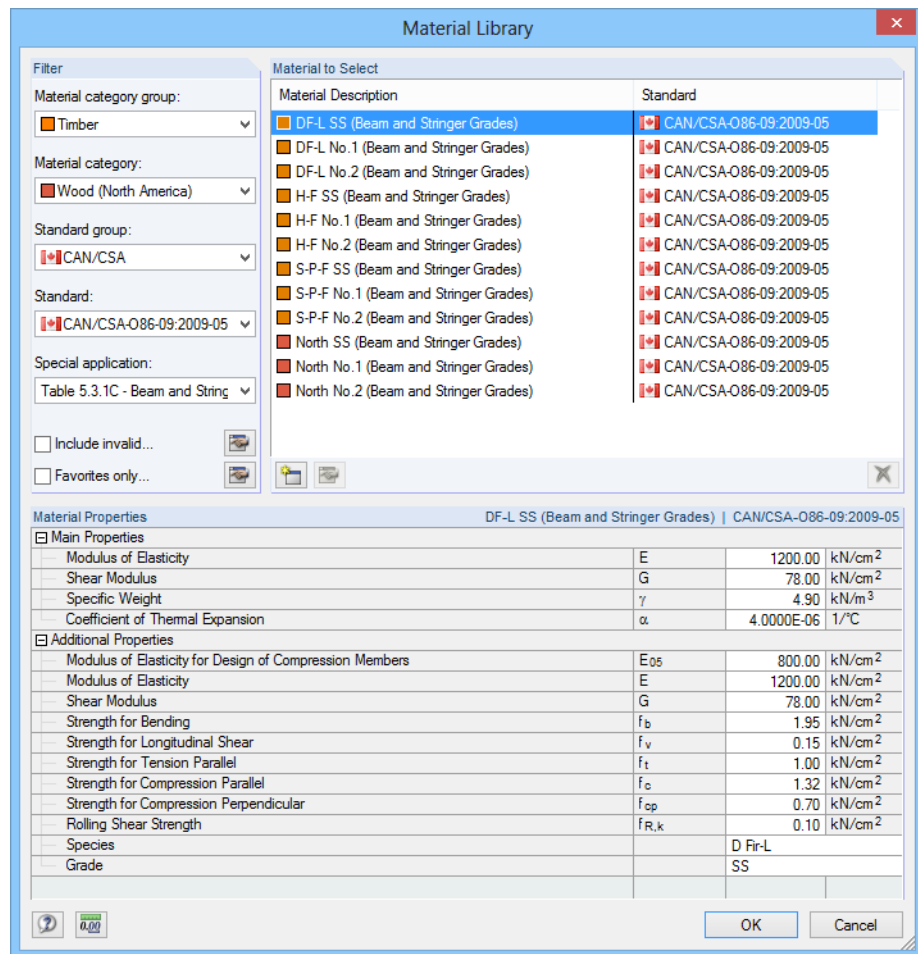


Figure 2.12: Dialog box *Material Library*

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.

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

Chapter 4.2 of the RSTAB manual describes in detail how materials can be filtered, added, or rearranged.

OK

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 RSTAB.

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.

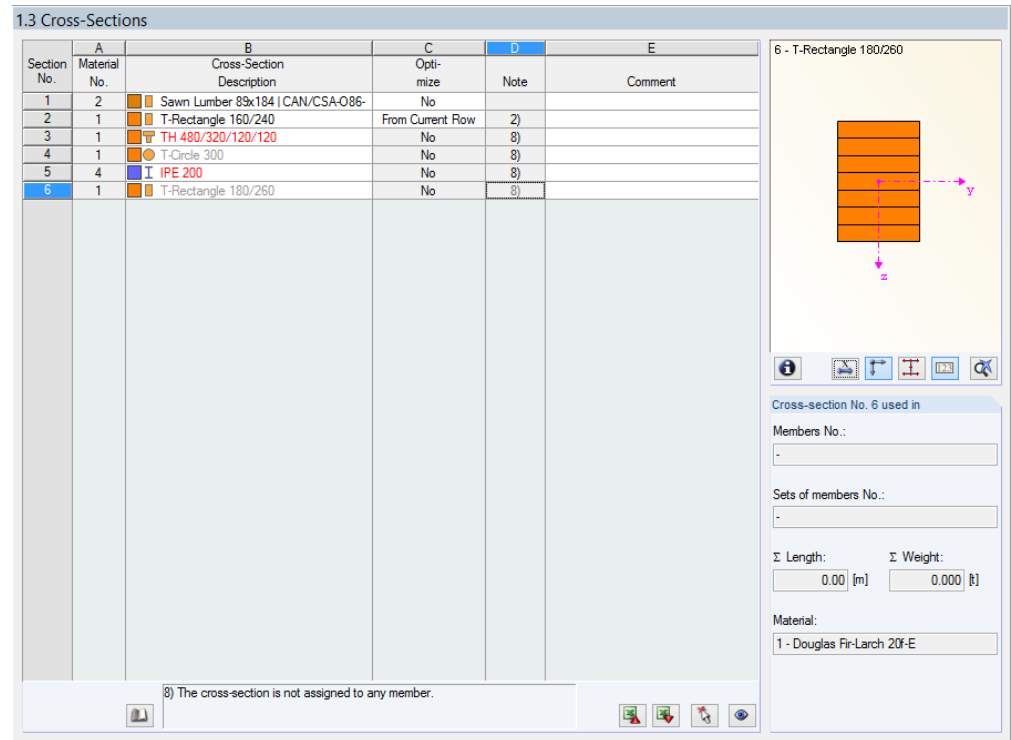


Figure 2.13: Window 1.3 Cross-Sections

Cross-Section Description

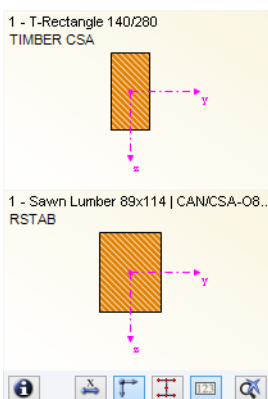
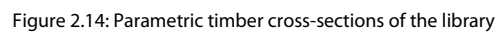
The cross-sections defined in RSTAB are preset together with the assigned material numbers.

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 4.3 of the RSTAB manual describes how cross-sections can be selected from the library.



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

If cross-sections specified in TIMBER CSA are different from the ones used in RSTAB, 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 RSTAB for the cross-section selected in 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 RSTAB 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 36).

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 57.

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.

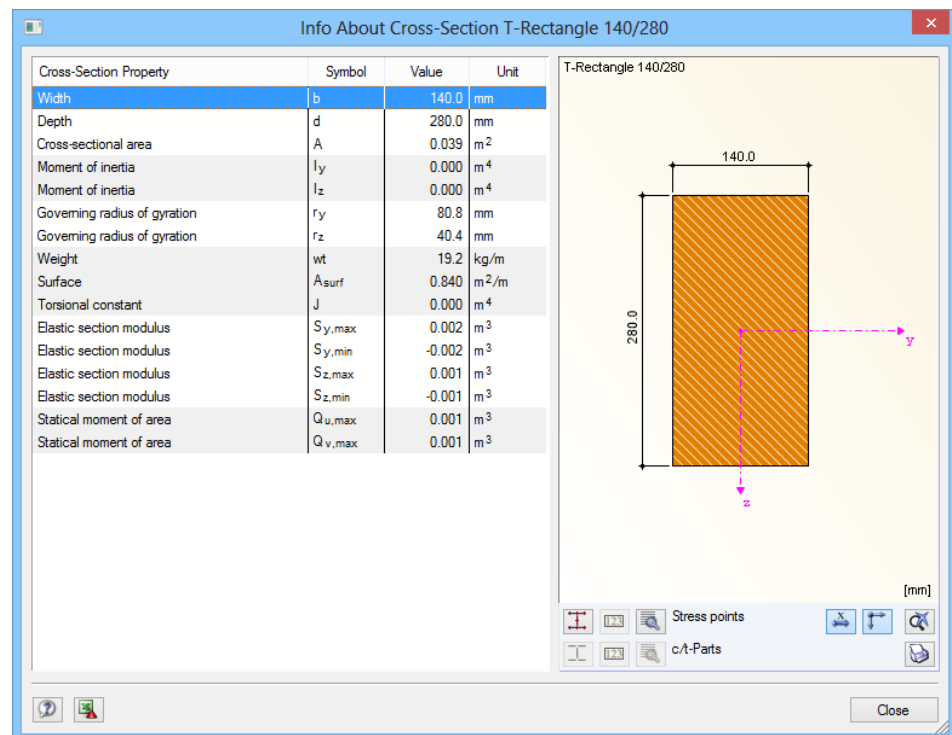


Figure 2.16: Dialog box *Info About Cross-Section*

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.).

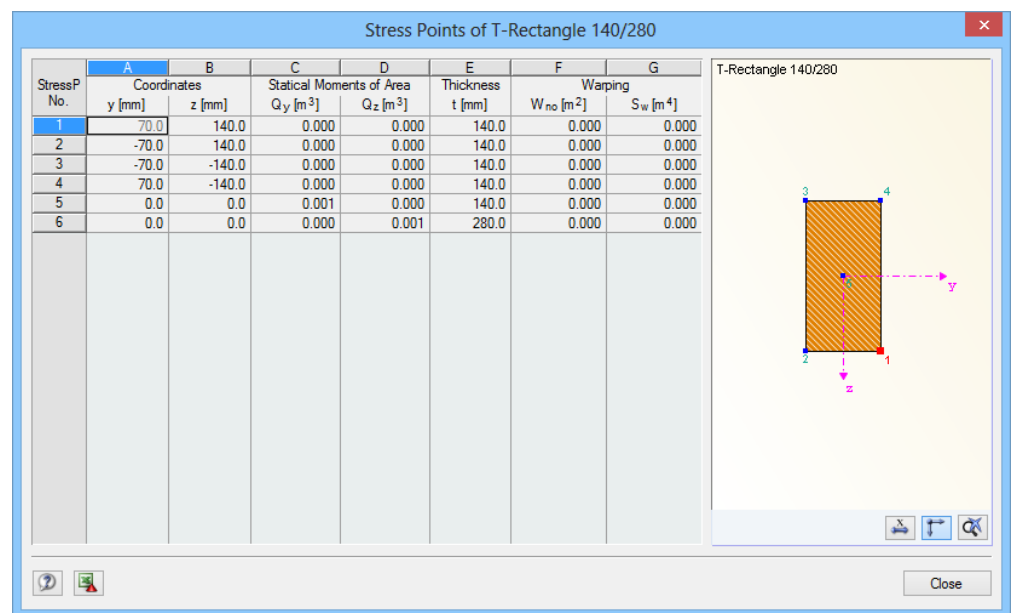


Figure 2.17: Dialog box *Stress Points*

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 RSTAB.

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

Load- ing	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	

Load Duration - Explanatory Notes

Short term:
means the condition of loading where the duration of the specified loads is not expected to last more than 7 days continuously or cumulatively throughout the life of the structure

Standard term:
means the condition of loading where the duration of specified loads exceeds that of short-term loading, but is less than long-term loading

Long term:
means the condition of loading under which a member is subjected to more or less continuous specified load

Apply load duration factor K_D according to: ☒ Shortest load duration in a combination ☐ User-defined setting

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 RSTAB. 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, 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.

Load Duration

Long term ▼

Short term

Standard term

Long term

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).

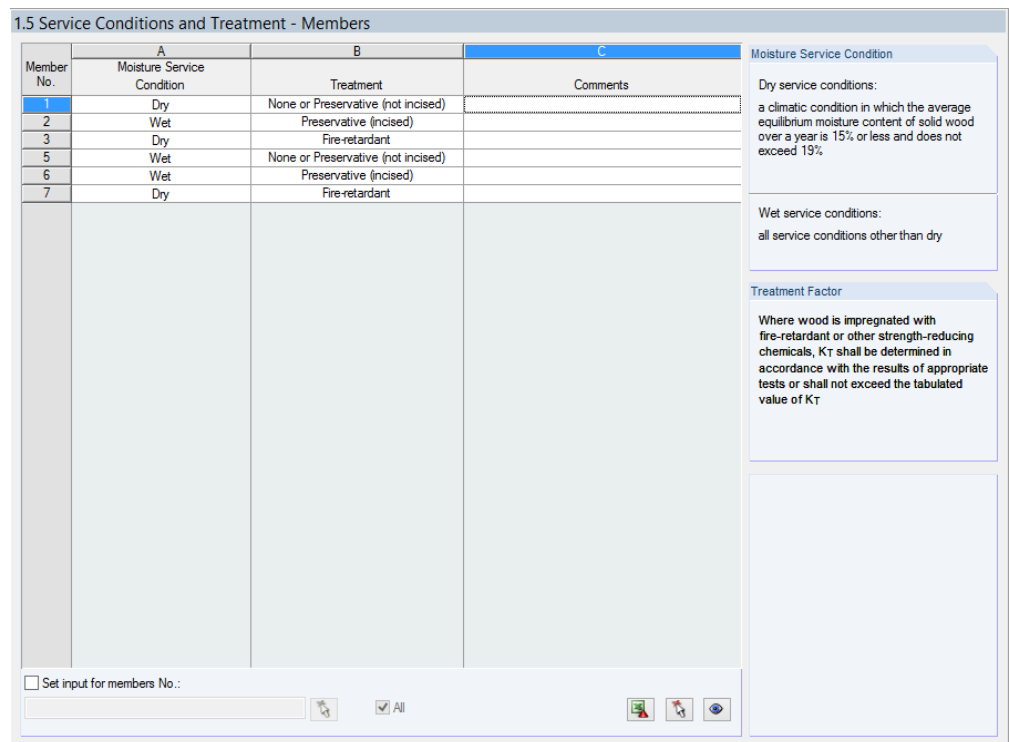
Loading Distribution

This column will be added when a radial stress design is to be carried out for tapered members. Those settings can be made in Window 1.10 *Tapered Members* (see Chapter 2.10).

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.



Member No.	A Moisture Service Condition	B Treatment	C Comments
1	Dry	None or Preservative (not incised)	
2	Wet	Preservative (incised)	
3	Dry	Fire-retardant	
5	Wet	None or Preservative (not incised)	
6	Wet	Preservative (incised)	
7	Dry	Fire-retardant	

Moisture Service Condition

Dry service conditions:
a climatic condition in which the average equilibrium moisture content of solid wood over a year is 15% or less and does not exceed 19%.

Wet service conditions:
all service conditions other than dry

Treatment Factor

Where wood is impregnated with fire-retardant or other strength-reducing chemicals, K_T shall be determined in accordance with the results of appropriate tests or shall not exceed the tabulated value of K_T .

☐ Set input for members No.:

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

By default, 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).

Standard...

Loading
Distribution

uniform

uniform

all other

Moisture Service
Condition

Dry

Dry

Wet

Treatment

None or Preservative (not incised)

None or Preservative (not incised)

Preservative (incised)

Fire-retardant

Standard...



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 RSTAB work window
	Switches into RSTAB 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*.

1.6 Service Conditions and Treatment - Sets of Members

Set No.	A Moisture Service Condition	B Treatment	C Comments
3	Dry	None or Preservative (not incised)	
4	Wet	Fire-retardant	

Moisture Service Condition

Dry service conditions:
a climatic condition in which the average equilibrium moisture content of solid wood over a year is 15% or less and does not exceed 19%

Wet service conditions:
all service conditions other than dry

Treatment Factor

Where wood is impregnated with fire-retardant or other strength-reducing chemicals, K_T shall be determined in accordance with the results of appropriate tests or shall not exceed the tabulated value of K_T





☐ Set input for members No.:  ☒ All   

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 34). 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 RSTAB 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.

1.7 Effective Lengths - Members

Member No.	A Buckling Possible	Buckling About Axis y		Buckling About Axis z		Lateral-Torsional Buckling		J L _e [m]	K Comment	
		B Possible	C Key	D L _{ey} [m]	E Possible	F K _{ez}	G L _{ez} [m]			H Possible
1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.0	2.400	<input checked="" type="checkbox"/>	1.0	2.400	<input checked="" type="checkbox"/>	as member length	2.400
2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.0	2.400	<input checked="" type="checkbox"/>	1.0	2.400	<input checked="" type="checkbox"/>	as member length	2.400
3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.0	2.400	<input checked="" type="checkbox"/>	1.0	2.400	<input checked="" type="checkbox"/>	as member length	2.400
5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.0	1.800	<input checked="" type="checkbox"/>	1.0	1.800	<input checked="" type="checkbox"/>	as member length	1.800
6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.0	0.900	<input checked="" type="checkbox"/>	1.0	0.900	<input checked="" type="checkbox"/>	as member length	0.900
7	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.0	3.600	<input checked="" type="checkbox"/>	1.0	3.600	<input checked="" type="checkbox"/>	as member length	3.600

Settings for member No. 1

Cross-section	1 - Sawn Lumber 89x114 CAN/CSA-O86-09		
Length	L	2.400	m
Buckling Possible		<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/> Buckling About Axis y Possible		<input checked="" type="checkbox"/>	
Effective Length Factor	K _{ey}	1.0	
Effective Length	L _{ey}	2.400	m
<input checked="" type="checkbox"/> Buckling About Axis z Possible		<input checked="" type="checkbox"/>	
Effective Length Factor	K _{ez}	1.0	
Effective Length	L _{ez}	2.400	m
<input checked="" type="checkbox"/> Lateral-Torsional Buckling Possible		<input checked="" type="checkbox"/>	
L _e manually		<input type="checkbox"/>	
Comment			

☐ Set input for members No.:

1 - Sawn Lumber 89x114 | CAN/CSA-O86-09

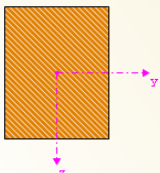


Figure 2.21: Window 1.7 *Effective Lengths - Members* for equivalent member method



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.

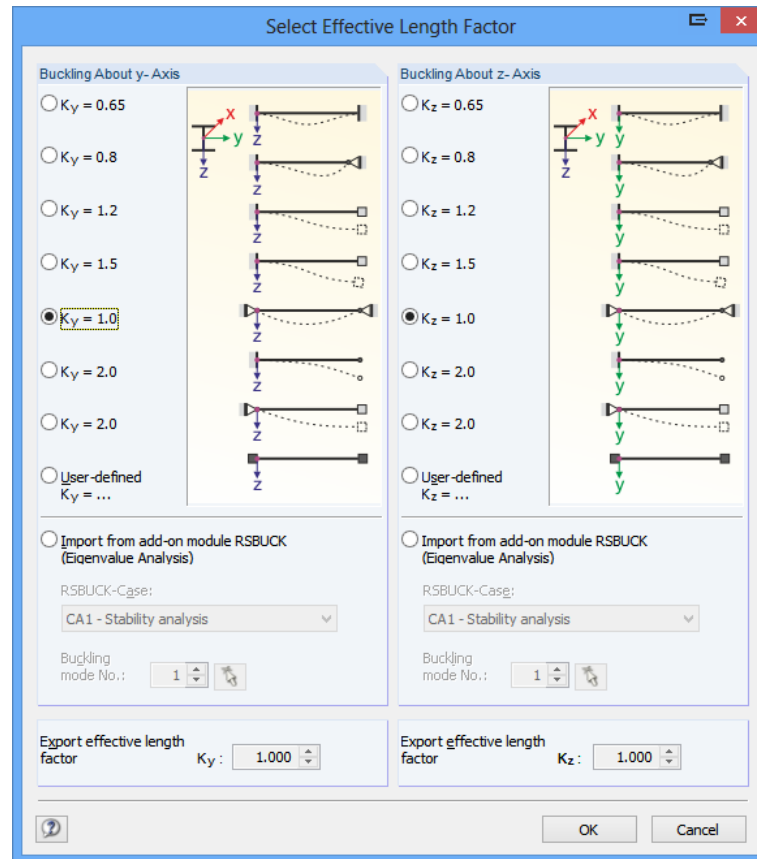


Figure 2.22: Dialog box *Select Effective Length Factor*

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 RSBUCK, 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 RSTAB 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.

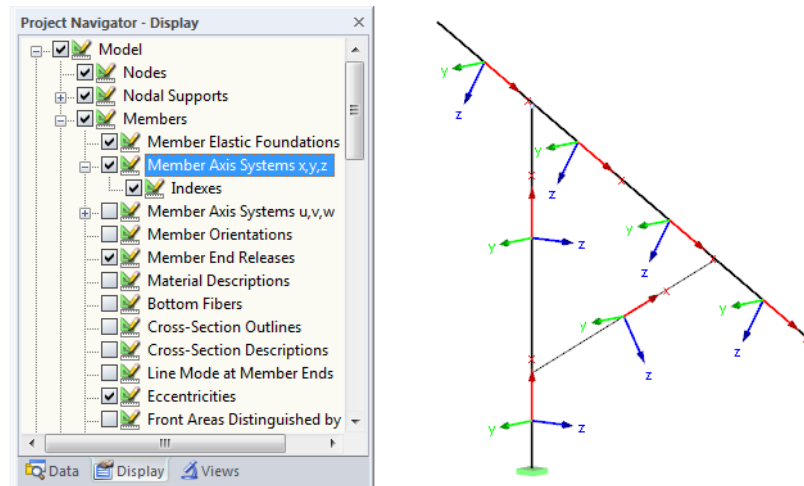


Figure 2.23: Selecting the member axis systems in the *Display* navigator of RSTAB

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.

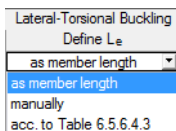
Lateral-Torsional Buckling

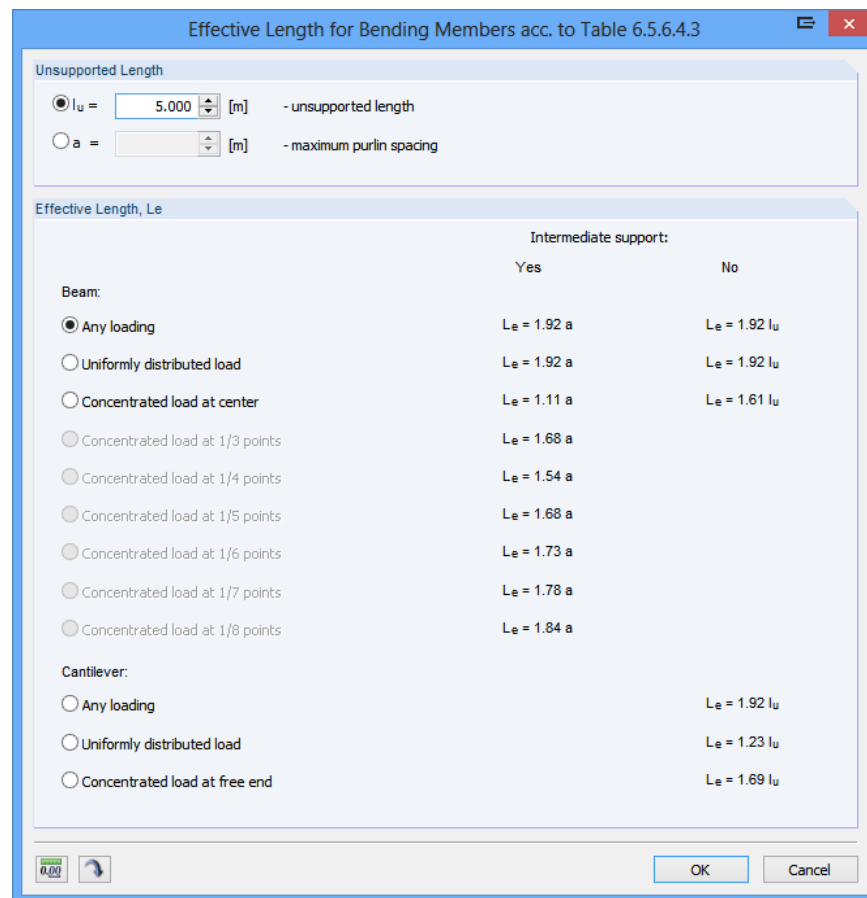
Table column H controls for which members a lateral-torsional buckling analysis is to be carried out.

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).





Effective Length for Bending Members acc. to Table 6.5.6.4.3

Unsupported Length

☒ $l_u = 5.000$ [m] - unsupported length

☐ $a =$ [] [m] - maximum purlin spacing

Effective Length, L_e

Beam:	Intermediate support:	
	Yes	No
<input checked="" type="radio"/> Any loading	$L_e = 1.92 a$	$L_e = 1.92 l_u$
<input type="radio"/> Uniformly distributed load	$L_e = 1.92 a$	$L_e = 1.92 l_u$
<input type="radio"/> Concentrated load at center	$L_e = 1.11 a$	$L_e = 1.61 l_u$
<input type="radio"/> Concentrated load at 1/3 points	$L_e = 1.68 a$	
<input type="radio"/> Concentrated load at 1/4 points	$L_e = 1.54 a$	
<input type="radio"/> Concentrated load at 1/5 points	$L_e = 1.68 a$	
<input type="radio"/> Concentrated load at 1/6 points	$L_e = 1.73 a$	
<input type="radio"/> Concentrated load at 1/7 points	$L_e = 1.78 a$	
<input type="radio"/> Concentrated load at 1/8 points	$L_e = 1.84 a$	
Cantilever:		
<input type="radio"/> Any loading		$L_e = 1.92 l_u$
<input type="radio"/> Uniformly distributed load		$L_e = 1.23 l_u$
<input type="radio"/> Concentrated load at free end		$L_e = 1.69 l_u$

OK Cancel

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 [\wedge] 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 your 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 34).

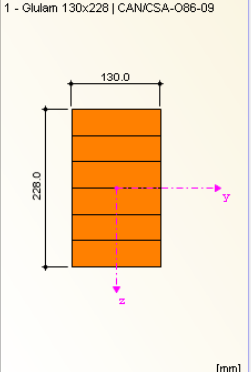
1.8 Effective Lengths - Sets of Members

Set No.	Buckling Possible	Buckling About Axis y Possible Key	Key	L _{ey} [m]	Buckling About Axis z Possible Key	Key	L _{ez} [m]	Lateral-Torsional Buckling Possible Define L _e	L _e [m]	Comment
1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	6.300	<input checked="" type="checkbox"/>	1.000	6.300	<input checked="" type="checkbox"/>	as member length	6.300
2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	6.300	<input checked="" type="checkbox"/>	1.000	6.300	<input checked="" type="checkbox"/>	as member length	6.300

Settings for set of members No. 1

Cross-section	1 - Glulam 130x228 CAN/CSA-O86-09		
Length	L	6.300 m	
Buckling Possible		<input checked="" type="checkbox"/>	
Buckling About Axis y Possible		<input checked="" type="checkbox"/>	
Effective Length Factor	K _{ey}	1.000	
Effective Length	L _{ey}	6.300 m	
Buckling About Axis z Possible		<input checked="" type="checkbox"/>	
Effective Length Factor	K _{ez}	1.000	
Effective Length	L _{ez}	6.300 m	
Lateral-Torsional Buckling Possible		<input checked="" type="checkbox"/>	
Define L _e		as member length	
Comment			

☐ Set input for members No.: ☒ All



1 - Glulam 130x228 | CAN/CSA-O86-09

[mm]

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.

2.9 Additional Design Parameters

Load Sharing

Case 1 ▼

None

Case 1

Case 2

Standard...

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).

1.9 Additional Design Parameters

Member No.	A	B	C	D
	Material Category	System Factor K_H Load Sharing	According to	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	

☐ Set input for members No.: ☒ All

Material Description: Douglas Fir-Larch 20F-E | CAN/CSA-O86-09:2009-05

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 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	<input type="checkbox"/>		
2	Glulam 130x228 CAN/CSA-O86	Glulam 130x418 CAN/CSA-O86	2.400	4.53	≤ 30.00	<input type="checkbox"/>		
3	Glulam 130x228 CAN/CSA-O86	Glulam 130x418 CAN/CSA-O86	2.400	4.53	≤ 30.00	<input type="checkbox"/>		
4	Glulam 130x228 CAN/CSA-O86	Glulam 130x418 CAN/CSA-O86	1.800	6.03	≤ 30.00	<input type="checkbox"/>		
5	Glulam 130x228 CAN/CSA-O86	Glulam 130x418 CAN/CSA-O86	0.900	11.92	≤ 30.00	<input type="checkbox"/>		
6	Glulam 130x228 CAN/CSA-O86	Glulam 130x418 CAN/CSA-O86	3.600	3.02	≤ 30.00	<input type="checkbox"/>		

☐ Set input for members No.:

☒ All





   

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

TIMBER CSA determines the inclination of the taper on the basis of geometric conditions. That angle is displayed for your information. The equations used in the program are valid only for angles smaller than 30° (see [1] Table 6.5.6.6.3).

The maximum angle of taper can be defined in the *Details* dialog box, tab *Other* (see Figure 3.4, page 36).

Radial Stress Design with Apex

If this check box has been ticked, 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 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 Manually	D Length L [m]	E Direction	F Precamber $w_{c,y}$ [mm]	G Precamber $w_{c,z}$ [mm]	H Beam Type	I Comment
1	Member	1	<input type="checkbox"/>	2.400	y	0.0		Beam	
2	Member	2	<input type="checkbox"/>	2.400	y	0.0		Beam	
3	Member	3	<input type="checkbox"/>	2.400	y	0.0		Beam	
4	Member	5	<input type="checkbox"/>	1.800	z		2.0	Beam	
5	Member	6	<input type="checkbox"/>	0.900	y; z	0.0	0.0	Cantilever End Free	
6	Member	7	<input type="checkbox"/>	3.600	z		0.0	Beam	
7	Set of Members	3	<input type="checkbox"/>	6.300	z		0.0	Beam	
8	List of Members	1,3,6,7	<input type="checkbox"/>	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 2.28: 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.

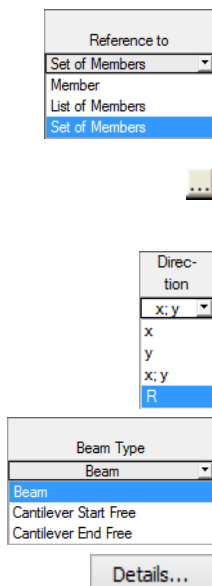
In table column B, you enter the numbers of the members or sets of members that you want to design. You can also click [...] to select them graphically in the RSTAB work window. Then, the *Reference Length* appears in column D automatically. This column presets the lengths of the members, sets of members, 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 35).



3. Calculation

3.1 Detail Settings

Calculation

Details...

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

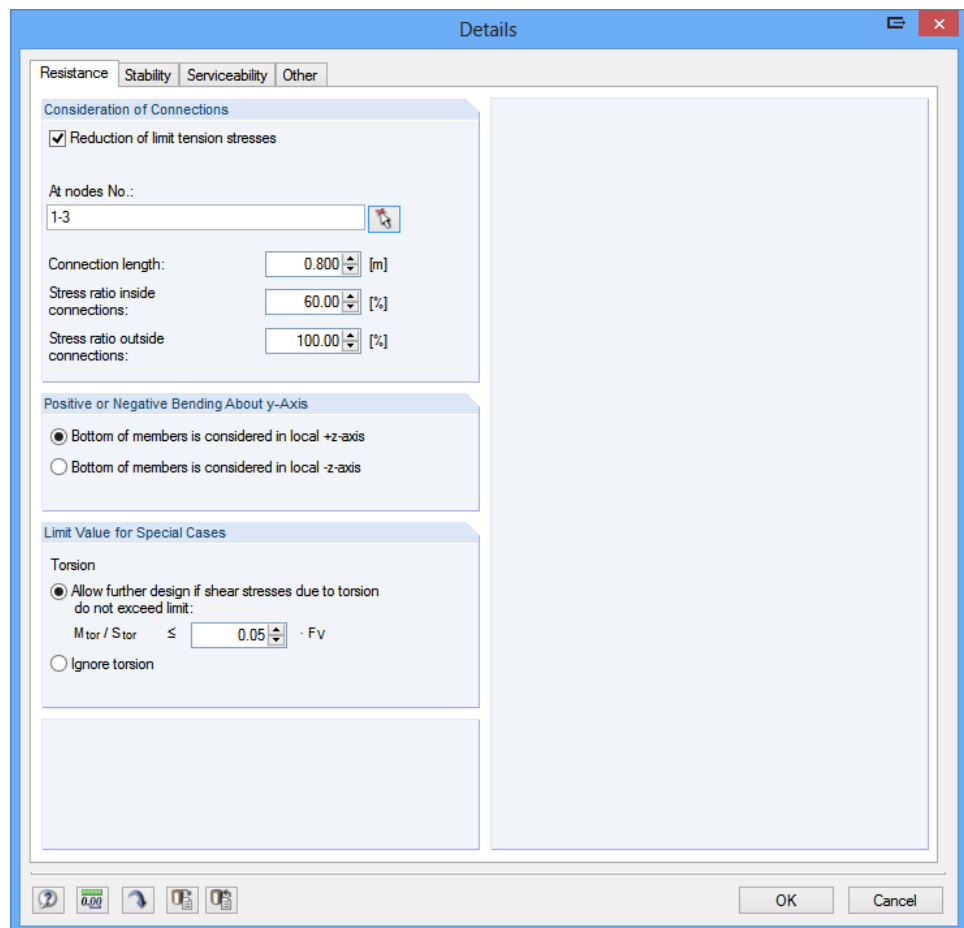


Figure 3.1: Dialog box *Details*, tab *Resistance*

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 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

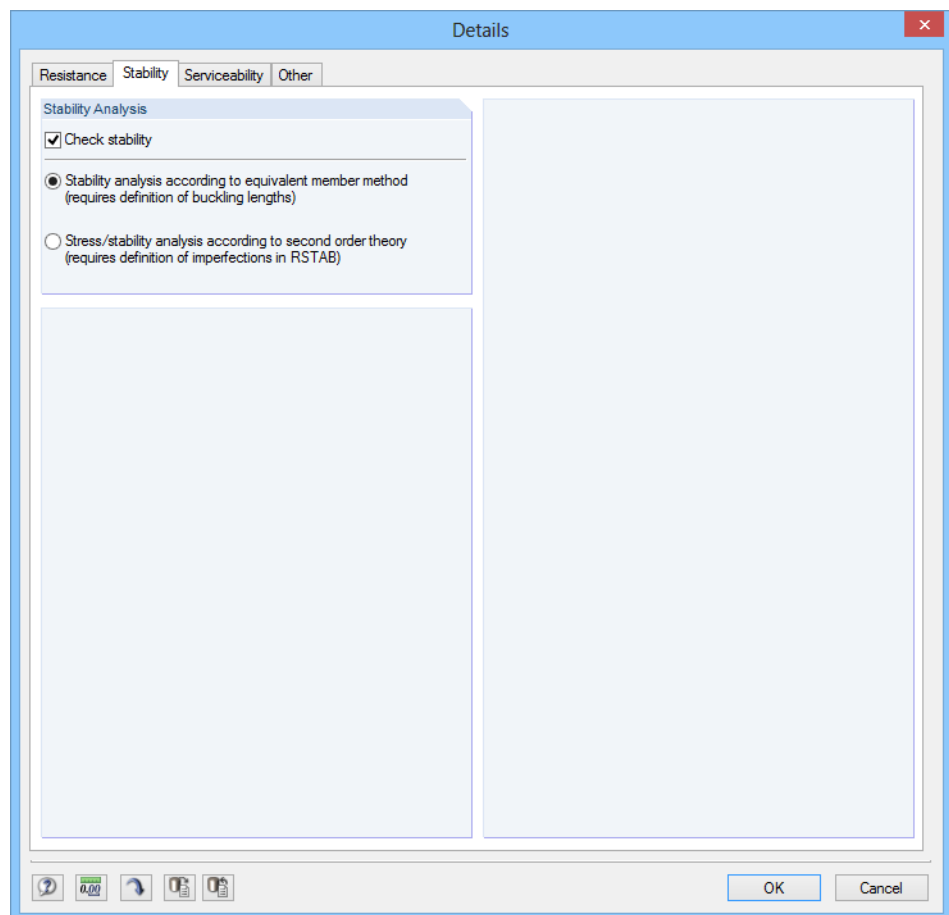
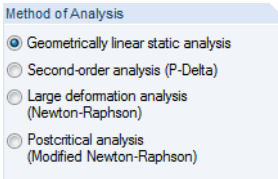


Figure 3.2: Dialog box *Details*, tab *Stability*

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 RSTAB



The *equivalent member method* uses the internal forces determined in RSTAB. 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 RSTAB and their consideration for the load combinations. The flexural buckling analysis is carried out during the calculation of the load combinations in RSTAB.

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

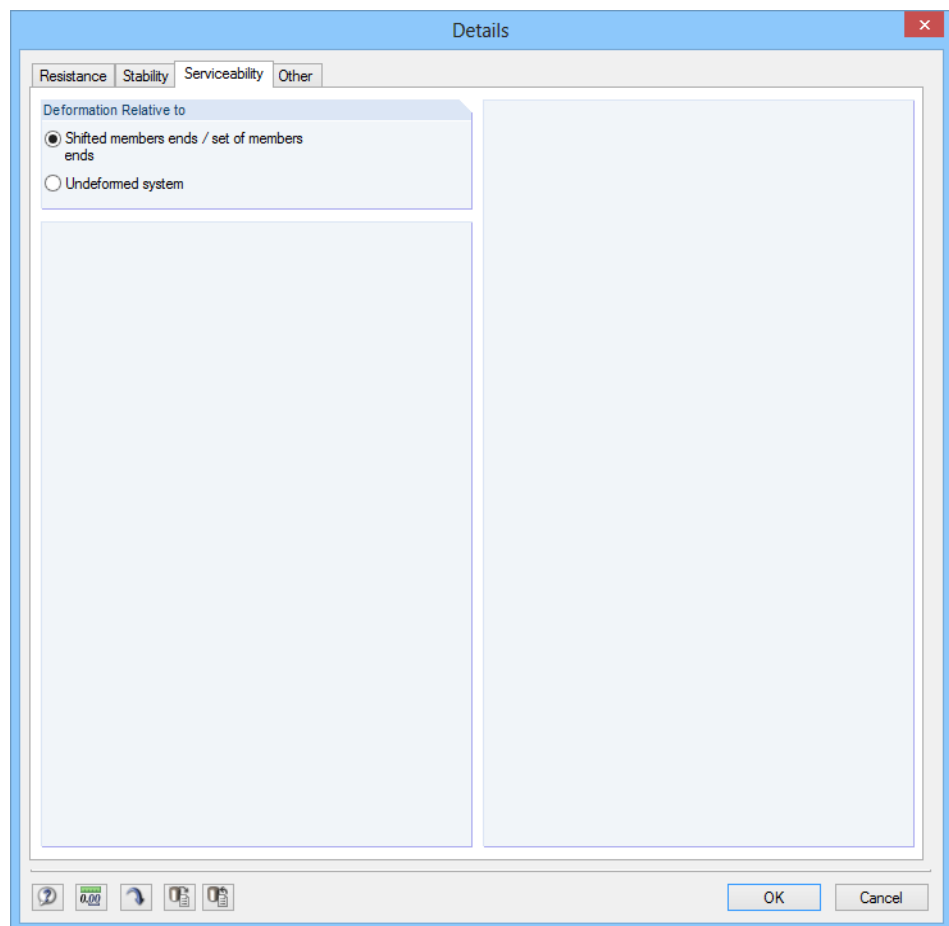


Figure 3.3: Dialog box *Details*, tab *Serviceability*

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

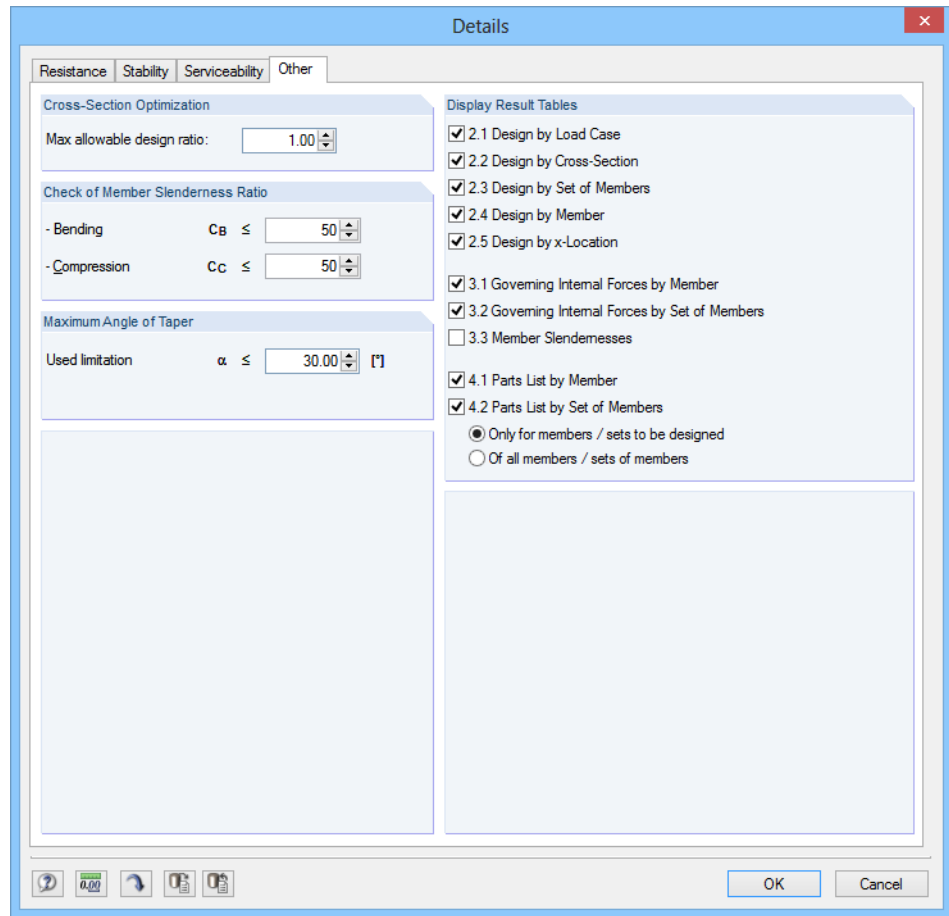


Figure 3.4: Dialog box *Details*, tab *Other*

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 45) 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

Calculation

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

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 RSTAB calculation to determine the design relevant internal forces.

You can also start the calculation in the user interface of RSTAB: 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.

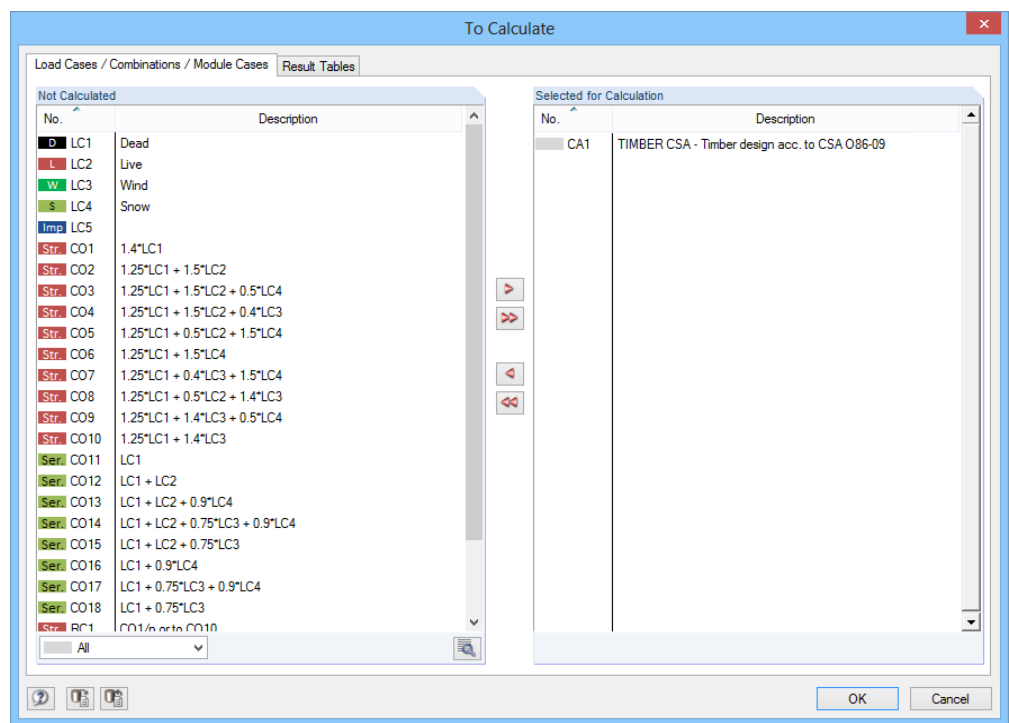


Figure 3.5: Dialog box *To Calculate*

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

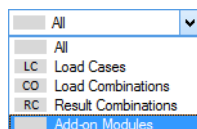
To transfer the selected TIMBER CSA cases to the list on the right, use the [►] button. Click [OK] to start the calculation.

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



Figure 3.6: Direct calculation of a TIMBER CSA design case in RSTAB

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.

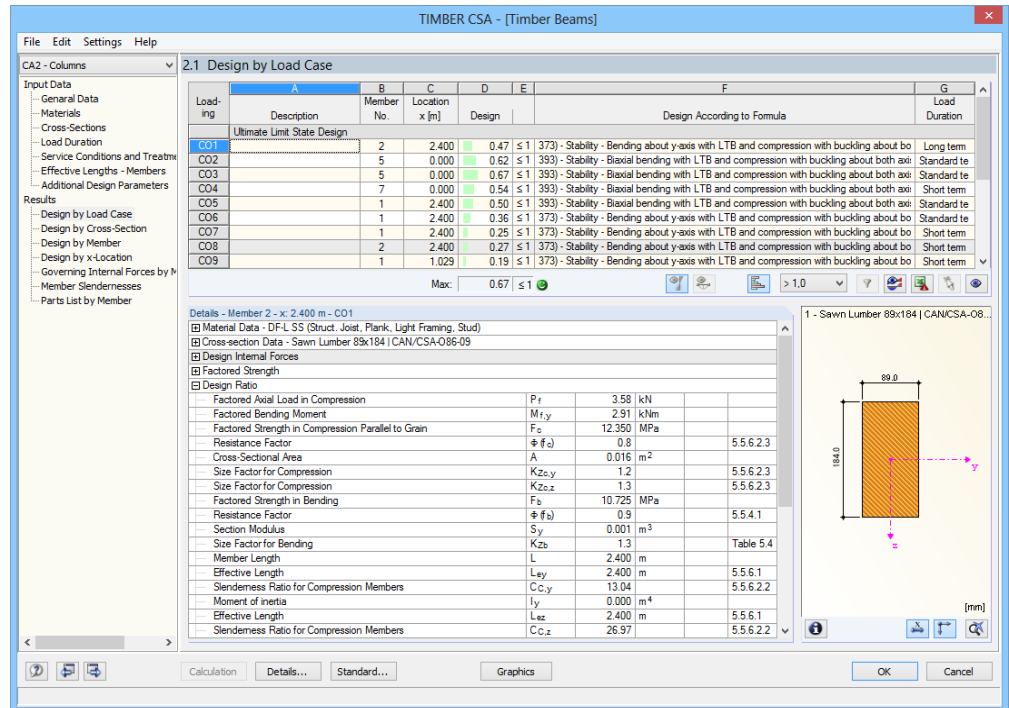


Figure 4.1: Results window with designs and intermediate values

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 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 48.



OK

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.

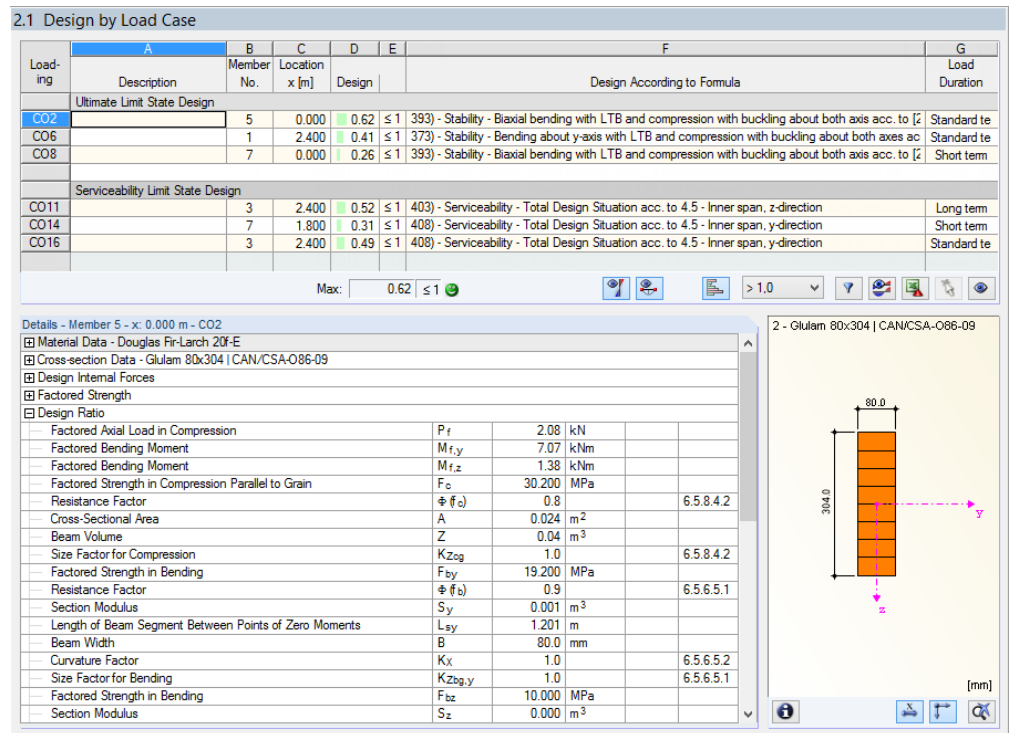


Figure 4.2: Window 2.1 Design by Load Case

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

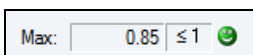
The column shows the x-location at which the maximum design ratio of each member occurs. For the table output, the program uses the following member locations x:

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

Design

Columns D and E show the design conditions according to 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

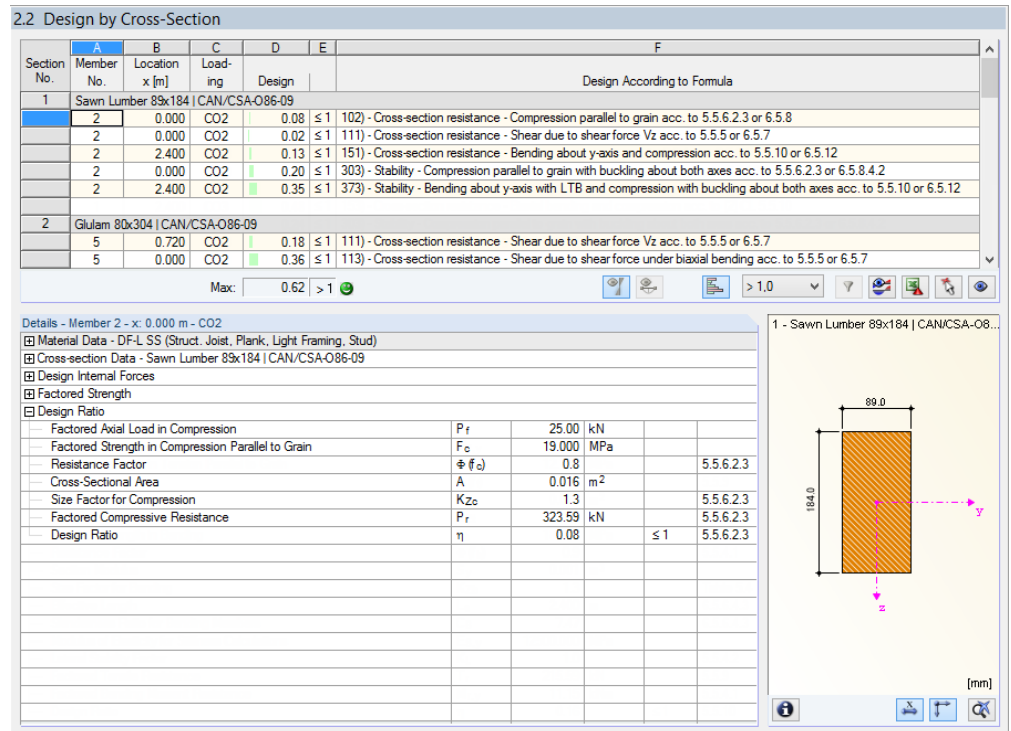


Figure 4.3: Window 2.2 *Design by Cross-Section*

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

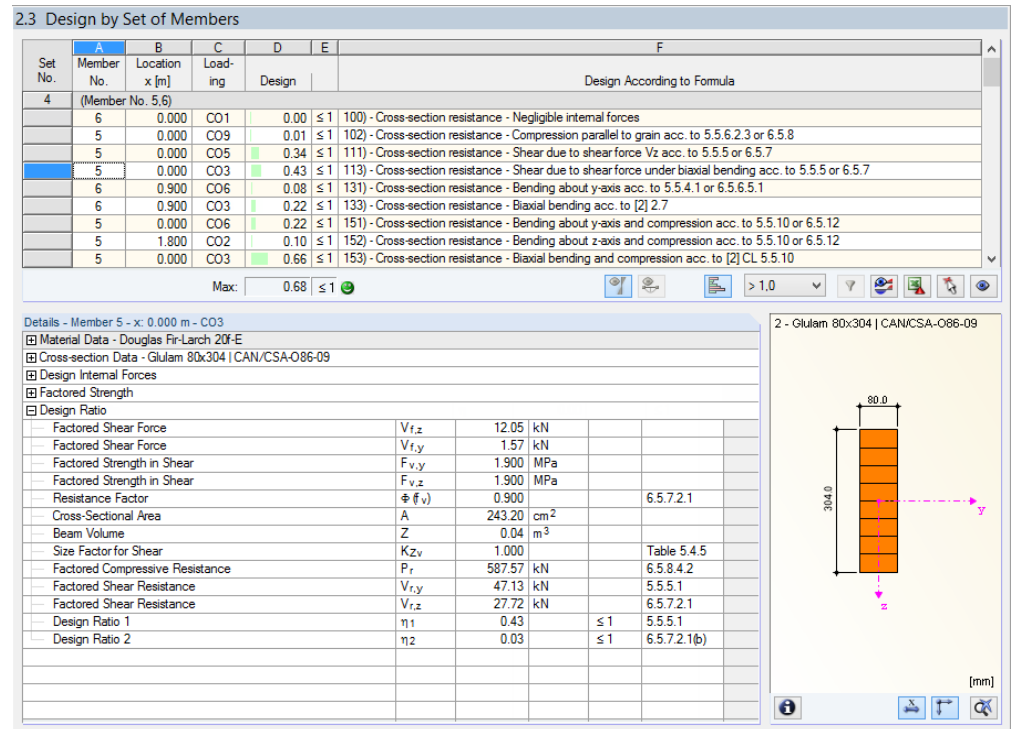


Figure 4.4: Window 2.3 *Design by Set of Members*

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

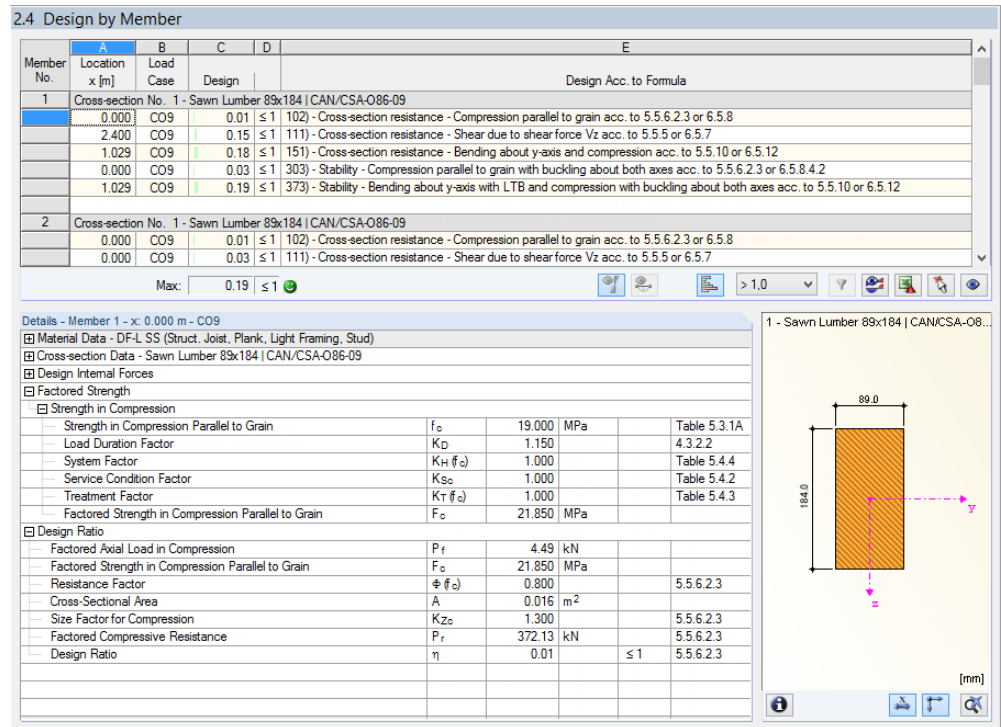


Figure 4.5: Window 2.4 Design by Member

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 39.

4.5 Design by x-Location

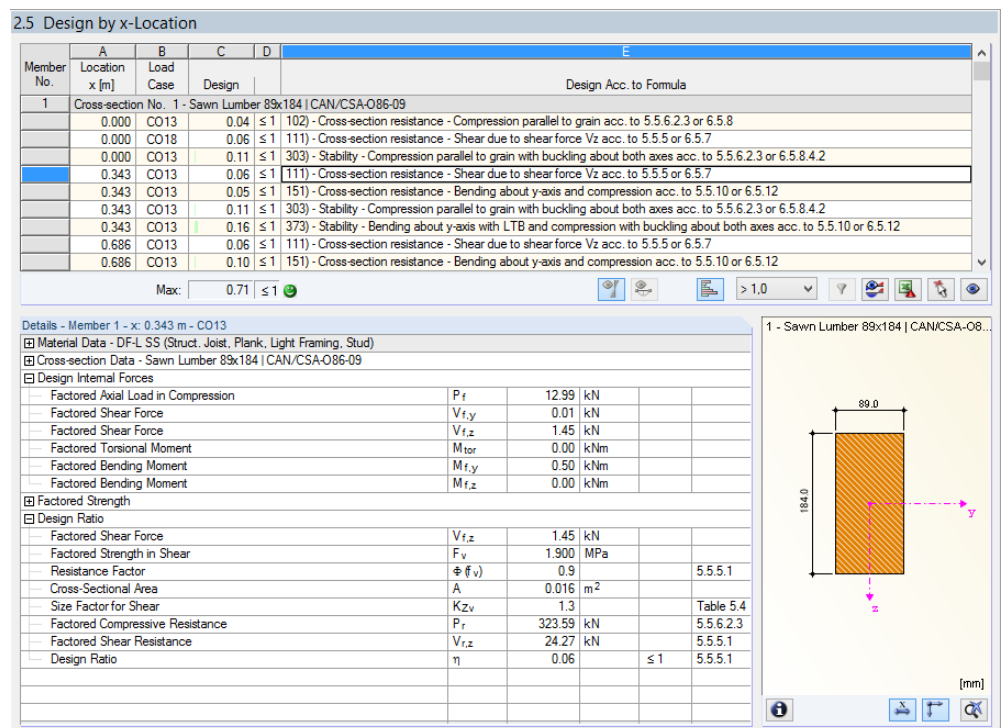


Figure 4.6: Window 2.5 Design by x-Location

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

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

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Load Case	C N	D Forces [kN] V _y	E V _z	F M _T	G Moments [kNm] M _y	H M _z	I 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 5.5.1
	2.400	CO9	-4.27	0.00	4.15	0.00	0.67	0.00	111) - Cross-section resistance - Shear due to shear force V _z acc. to 5.5.1
	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 acc. to 5.5.1
	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 axes
	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 buckling
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 5.5.1
	0.000	CO9	-5.31	0.00	-0.80	0.00	0.00	0.00	111) - Cross-section resistance - Shear due to shear force V _z acc. to 5.5.1
	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 acc. to 5.5.1
	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 axes
	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 buckling
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 5.5.1
	0.000	CO9	-6.02	0.00	-0.65	0.00	0.00	0.00	111) - Cross-section resistance - Shear due to shear force V _z acc. to 5.5.1
	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 acc. to 5.5.1
	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 axes
	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 buckling
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
	1.800	CO9	-3.37	0.00	-2.48	0.00	-2.11	0.00	111) - Cross-section resistance - Shear due to shear force V _z acc. to 5.5.1
	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 acc. to 5.5.1
	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 axes
	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 buckling
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 V _z acc. to 5.5.1
	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 acc. to 5.5.1
	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 axes

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

Set No.	A	B	C	D	E	F	G	H	I
	Location x [m]	Load Case	N	Forces [kN] V _y	V _z	M _T	Moments [kNm] 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 V _z 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. to
	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. to
	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 buckl
	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 abou
	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-directi
	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-directi

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

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

4.8 Member Slendernesses

3.3 Member Slendernesses

Member No.	A Loss of Stability Under Stress	B L_{ey} [m]	C Buckling About Axis y d [mm]	D C_{Cy} [-]	E Ratio	F L_{ez} [m]	G Buckling About Axis z b [mm]	H C_{Cz} [-]	I Ratio	J L_e [m]	K b [mm]	L Lateral Buckling d [mm]	M C_B [-]	N 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:		Bending members:	
Max C_{Cy} :	15.0 ≤ 50	Max C_B :	13.1 ≤ 50
Max C_{Cz} :	45.0 ≤ 50		

Figure 4.9: Window 3.3 Member Slendernesses

Details...

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 36).

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 36).

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

4.9 Parts List by Member

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

4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [m]	D Total Length [m]	E Surface Area [m ²]	F Volume [m ³]	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*

Details...

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 36).

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).



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

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

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

The final column indicates the total weight of each part.

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

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.

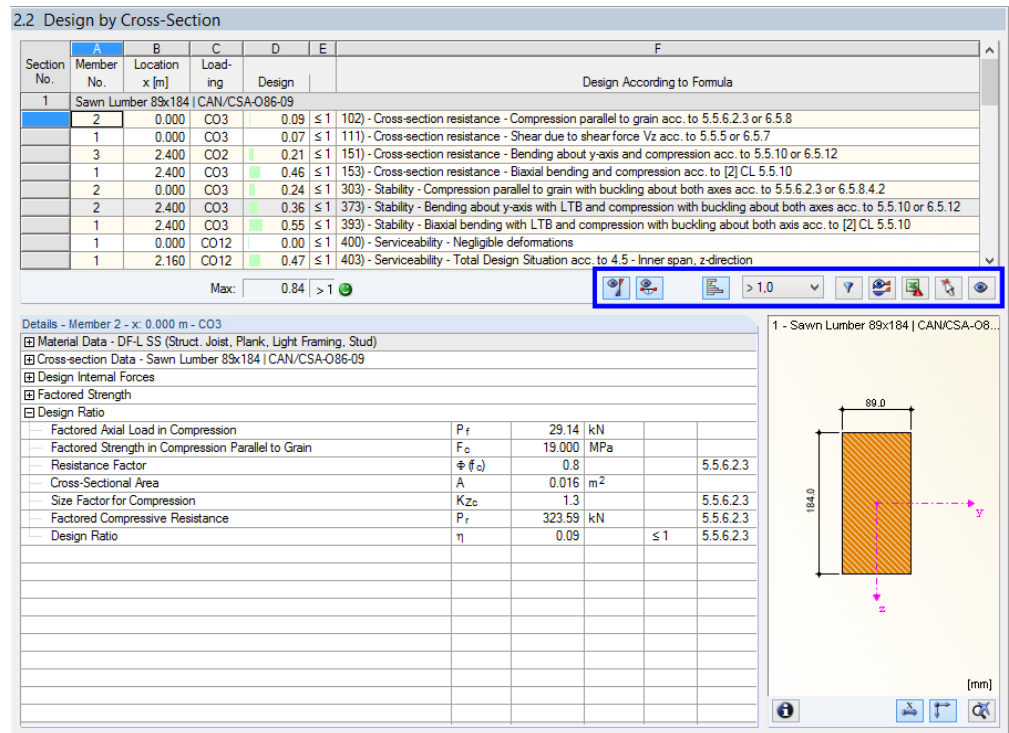


Figure 5.1: Buttons for evaluation of results

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
	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 <i>Result Diagram on Member</i> → Chapter 5.2, page 51
	Excel Export	Exports the table to MS Excel / OpenOffice → Chapter 7.4.3, page 61
	Member Selection	Allows you to graphically select a member to display its results in the table
	View Mode	Jumps to the RSTAB work window to change the view

Table 5.1: Buttons in results windows 2.1 through 2.5

5.1 Results in RSTAB Model

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

RSTAB background graphic and view mode

The RSTAB work window in the background is useful for finding the position of a particular member in the model: The member selected in the 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.

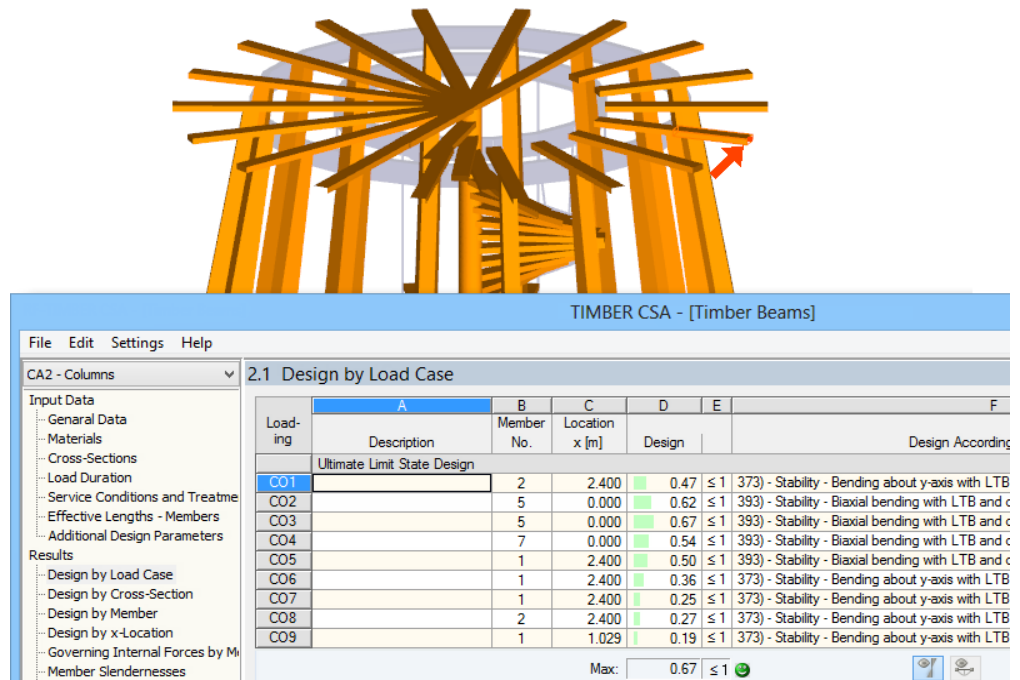


Figure 5.2: Indication of the member and the current Location x in the RSTAB model

If you cannot improve the display by moving the TIMBER CSA module window, click the [Jump to Graphic] button to activate the View Mode: Thus, you hide the module window so that you can modify the display in the RSTAB 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 TIMBER CSA.

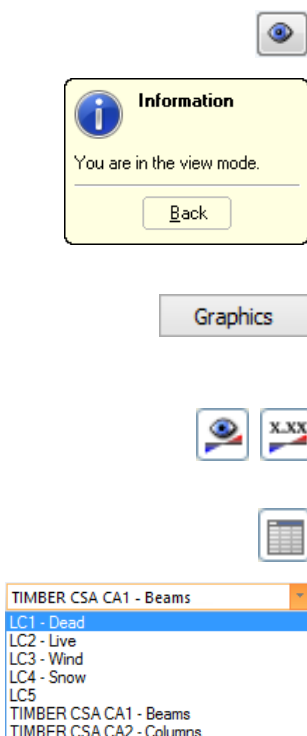
RSTAB work window

You can also graphically check the design ratios in the RSTAB model: Click [Graphics] to exit the design module. In the RSTAB 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 RSTAB. To display the result values, click the [Show Values] toolbar button to the right.

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

The design cases can be set by means of the list in the RSTAB 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.

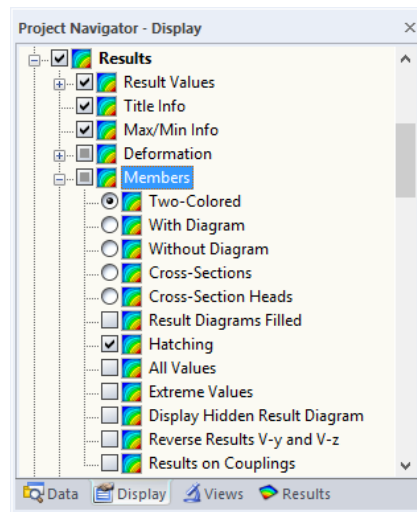


Figure 5.3: Display navigator: Results → Members



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 RSTAB manual, Chapter 3.4.6.

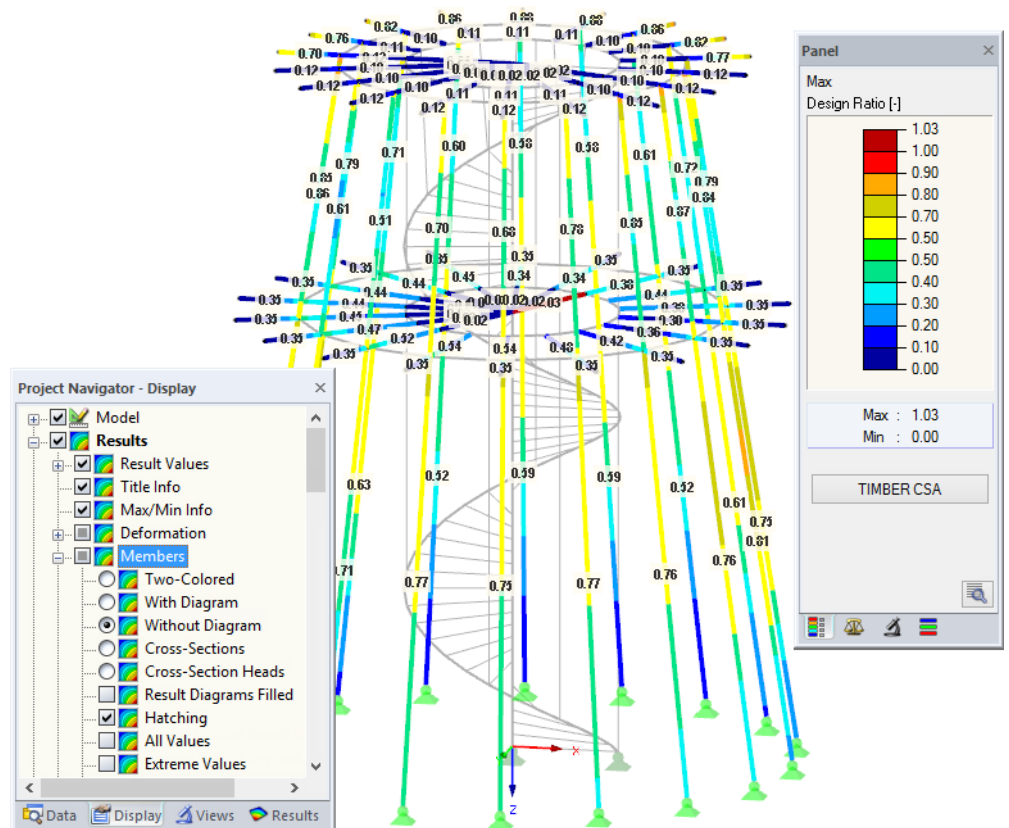


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 54).

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

TIMBER CSA

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 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 48).



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

Results → Result Diagrams for Selected Members

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

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

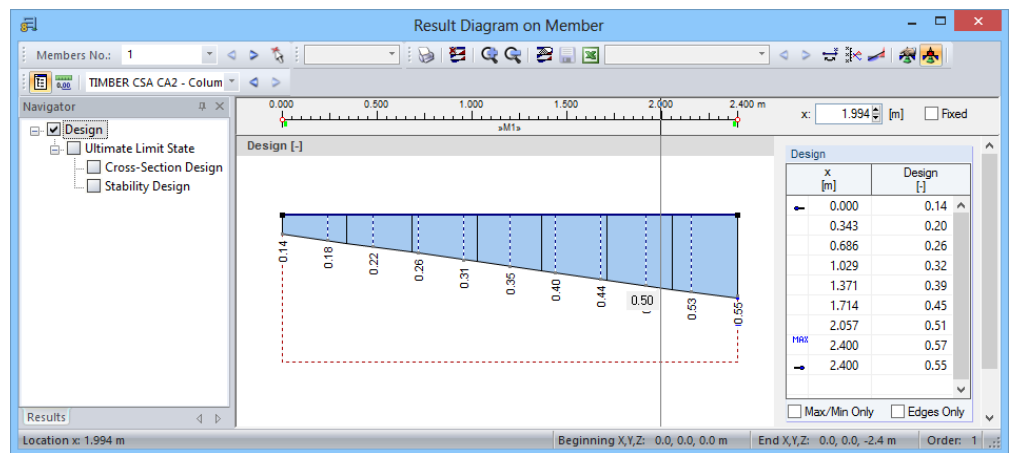
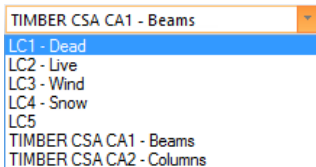


Figure 5.5: Dialog box *Result Diagram on Member*



Use the list in the toolbar above to select the relevant TIMBER CSA design case.

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

5.3 Filter for Results

The 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 9.7 of the RSTAB manual.

You can use the *Visibility* option also for TIMBER CSA (see RSTAB manual, Chapter 9.7.1) to filter the members in order to evaluate them.

Filtering designs

The design ratios can easily be used as filter criteria in the RSTAB 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 RSTAB 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.

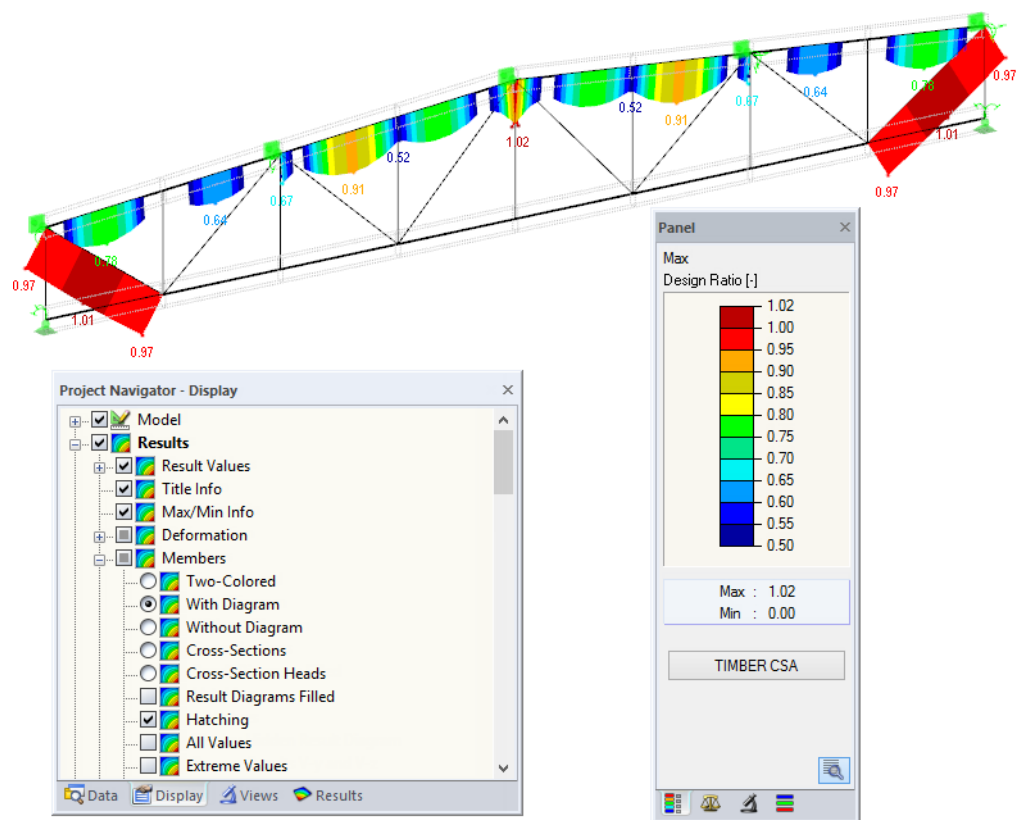


Figure 5.6: Filtering design ratios with adjusted color spectrum

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 RSTAB manual, Chapter 9.7.3.

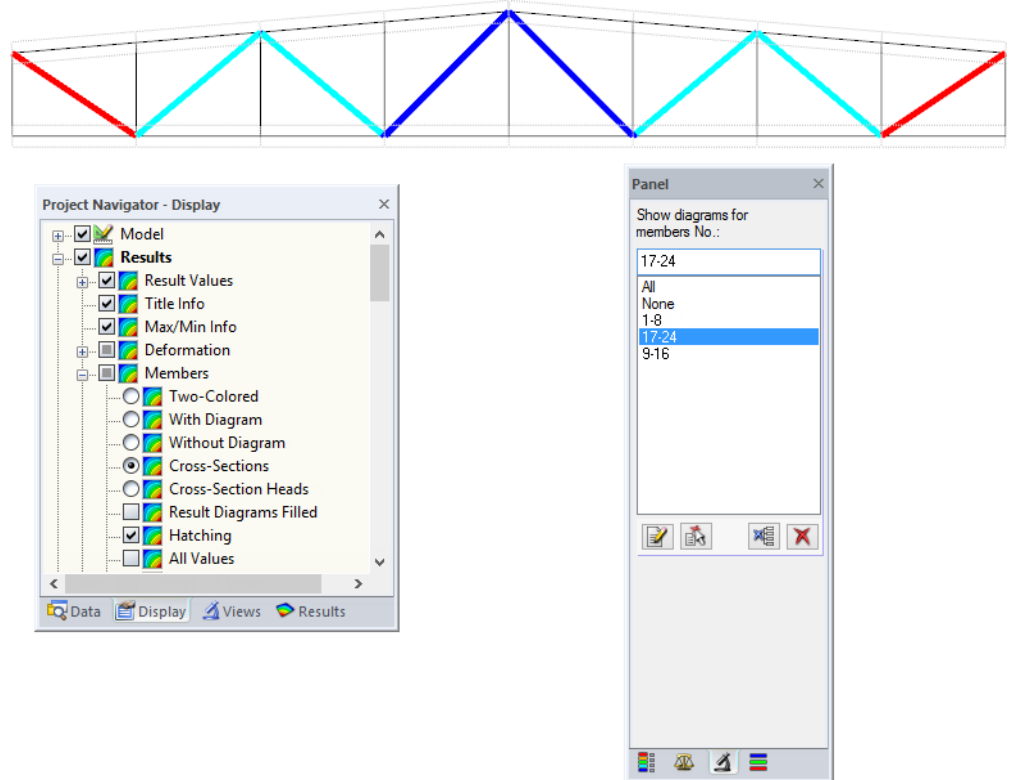


Figure 5.7: Member filter for ratios of diagonals

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 RSTAB, the program generates a printout report for the 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 RSTAB manual. In particular, Chapter 10.1.3.5 *Selecting Data of Add-on Modules* describes how to select input and output data from add-on modules for the printout report.

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

6.2 Graphic Printout

In RSTAB, 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 RSTAB model for the printout, too.



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

Designs in the RSTAB model

To print the currently displayed design ratios, click

File → Print Graphic

or use the toolbar button shown on the left.

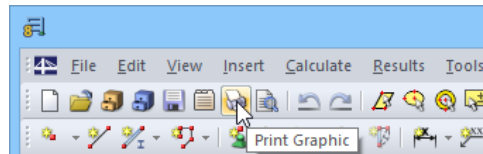


Figure 6.1: Button *Print* in RSTAB toolbar

Result diagrams

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

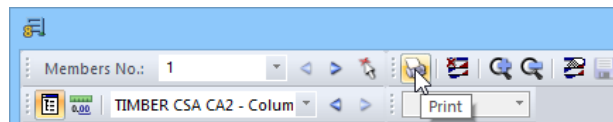
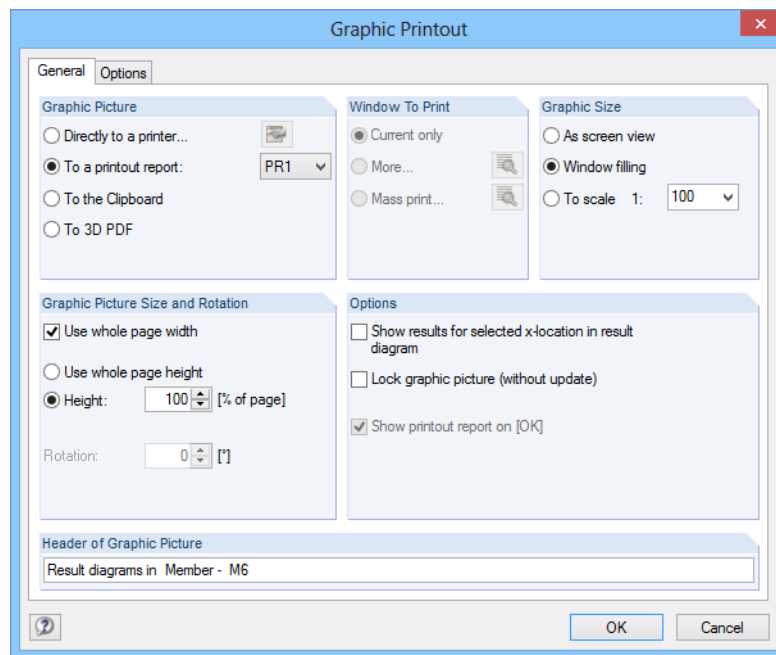


Figure 6.2: Button *Print* in the *Result Diagram on Member*

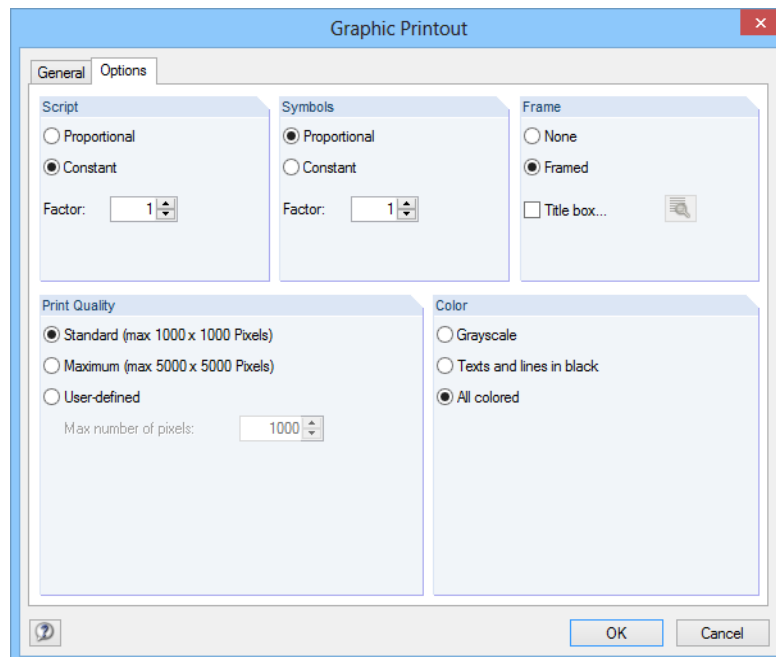
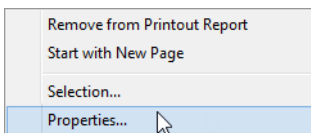
The *Graphic Printout* dialog box appears (see figure on next page).

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

This dialog box is described in the RSTAB manual, Chapter 10.2. The RSTAB 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 TIMBER CSA design case, you can also use the load case list in the RSTAB toolbar.

Create new design case

To create a new design case, use the TIMBER CSA menu and click

File → New Case.

The following dialog box appears:

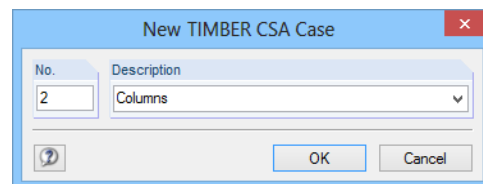


Figure 7.1: Dialog box *New TIMBER CSA-Case*

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 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 TIMBER CSA menu and click

File → Rename Case.

The following dialog box appears:

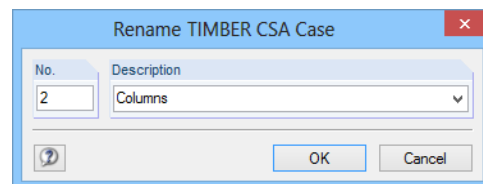


Figure 7.2: Dialog box *Rename TIMBER CSA-Case*

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 TIMBER CSA menu

File → Copy Case

The following dialog box appears:

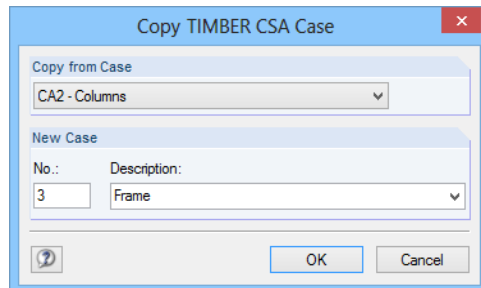


Figure 7.3: Dialog box *Copy TIMBER CSA-Case*

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

Delete design case

To delete design cases, use the TIMBER CSA menu

File → Delete Case

The following dialog box appears:

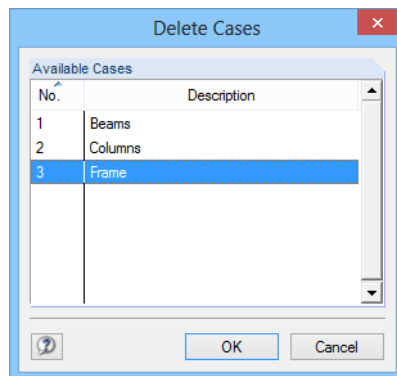


Figure 7.4: Dialog box *Delete Cases*

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.

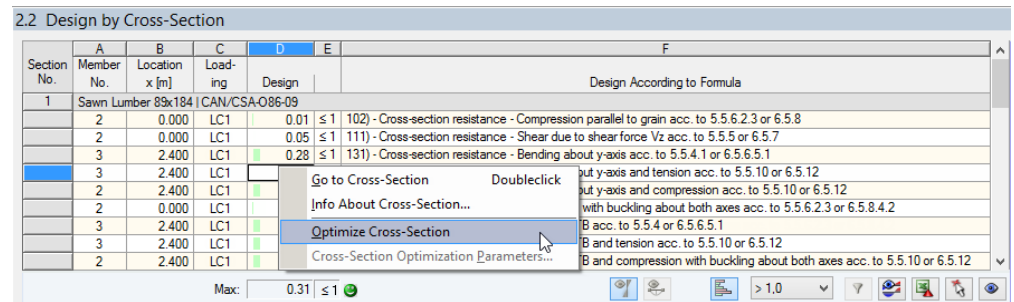


Figure 7.5: Context-menu for cross-section optimization

Details...

During the optimization process, the module determines the cross-section that fulfills the analysis requirements in the most optimal way, that is, comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.4, page 36). The required cross-section properties are determined with the internal forces from RSTAB. 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 RSTAB 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.

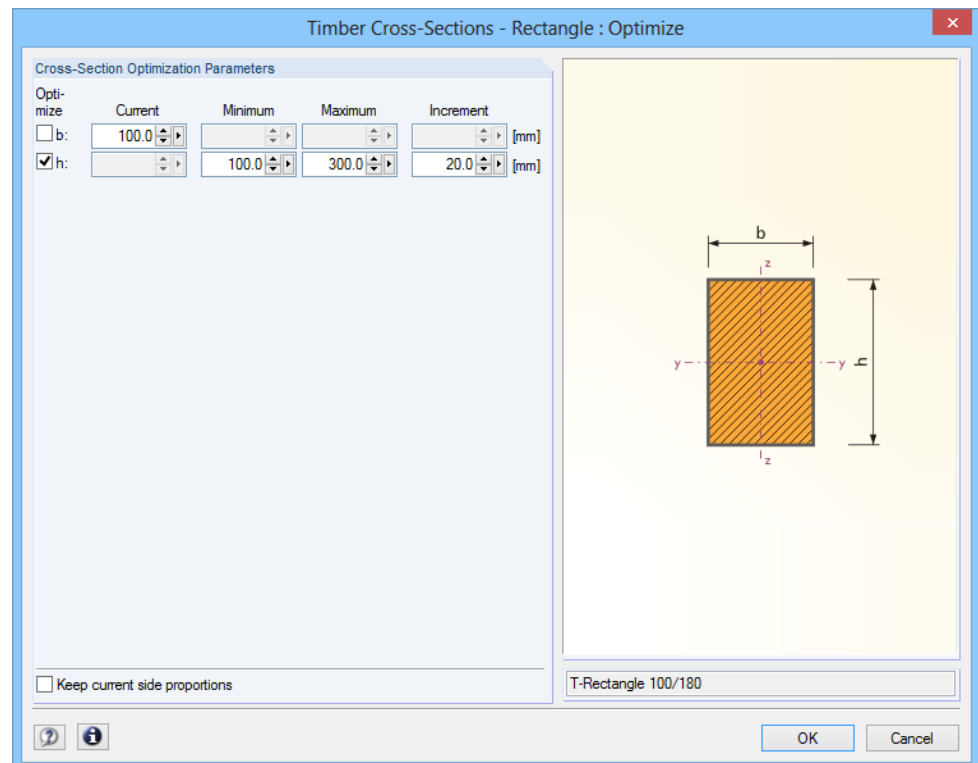


Figure 7.6: Dialog box *Timber Cross-Sections - Rectangle : Optimize*

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 RSTAB 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 RSTAB: Go to the 1.3 *Cross-Sections* window, and then click

Edit → Export All Cross-Sections to RSTAB

Alternatively, you can use the context menu in window 1.3 to export optimized cross-sections to RSTAB.

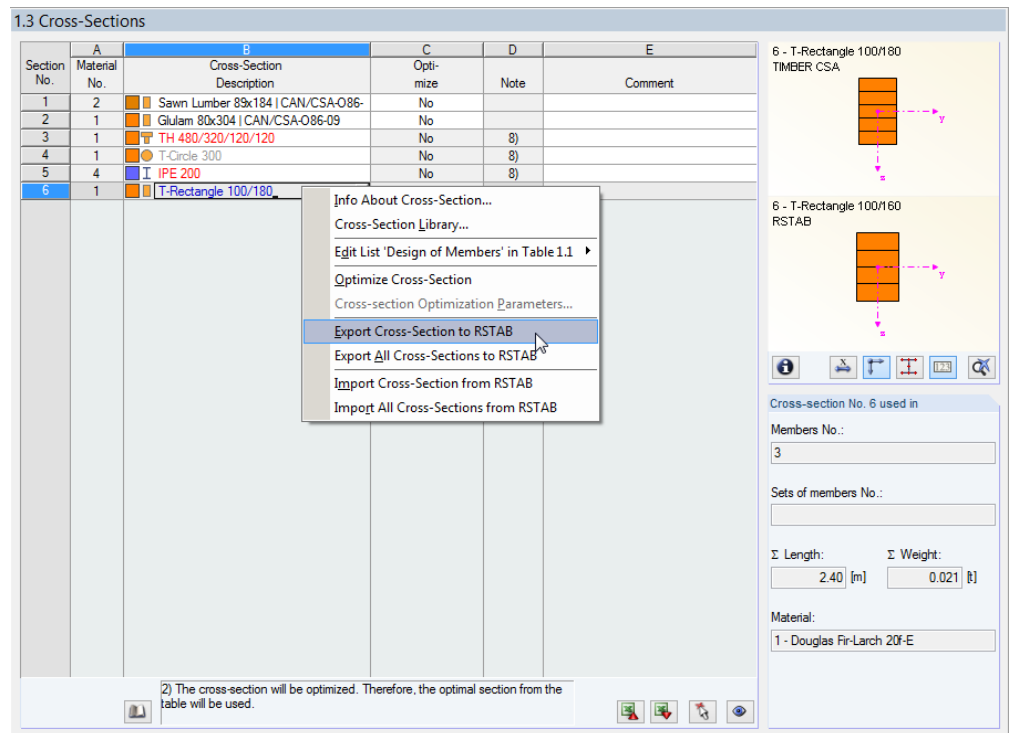


Figure 7.7: Context menu in window 1.3 *Cross-Sections*

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

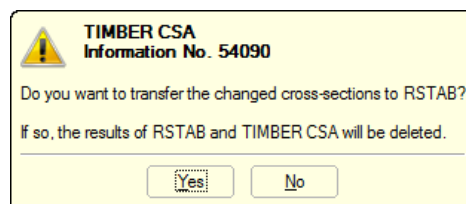


Figure 7.8: Query before transfer of modified cross-sections to RSTAB

Calculation

By confirming the query and then starting the [Calculation] in the TIMBER CSA module, the RSTAB 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 RSTAB 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 RSTAB and the add-on modules are managed in one dialog box. To define the units in TIMBER CSA, select menu

Settings → Units and Decimal Places.

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

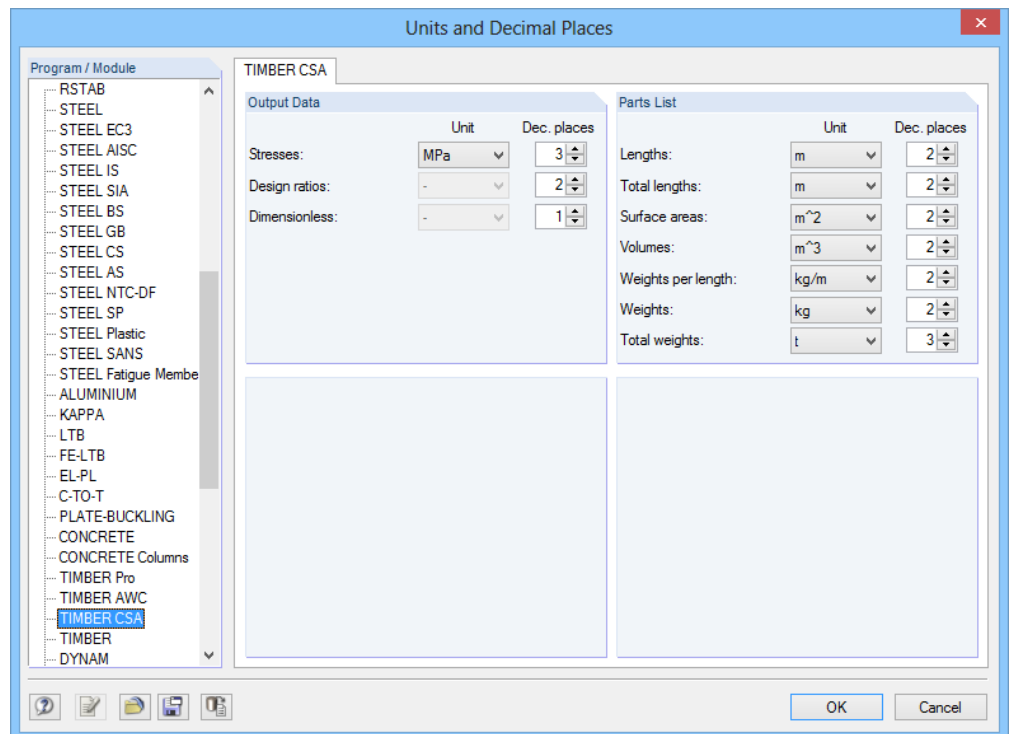


Figure 7.9: Dialog box *Units and Decimal Places*



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

7.4 Data Transfer

7.4.1 Export Material to RSTAB

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

Edit → Export All Materials to RSTAB.

You can also export the modified materials to RSTAB using the context menu of window 1.2.

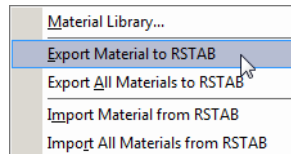


Figure 7.10: Context menu of window 1.2 *Materials*

Calculation

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

If the modified materials have not been exported to RSTAB 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 RSTAB

If you have adjusted the materials in TIMBER CSA for design, you can export the modified materials to RSTAB 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 RSTAB

or use the corresponding option on the context menu of window 1.7.

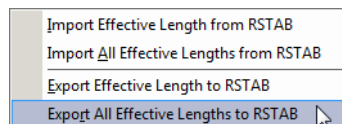


Figure 7.11: Context menu of window 1.7 *Effective Lengths - Members*

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

If the modified effective lengths have not been exported to RSTAB 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 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 TIMBER CSA add-on module into the global printout report (see Chapter 6.1, page 54) for export. Then, in the printout report, click

File → Export to RTF

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

Excel / OpenOffice

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.

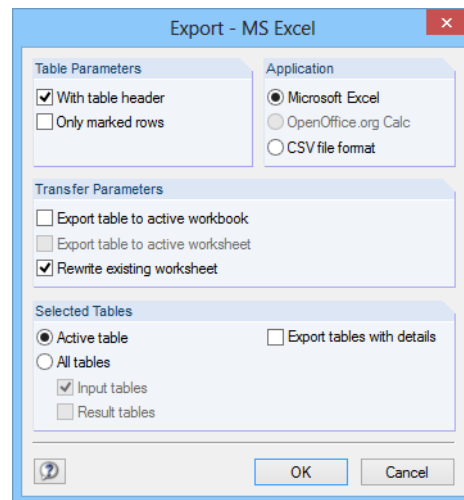
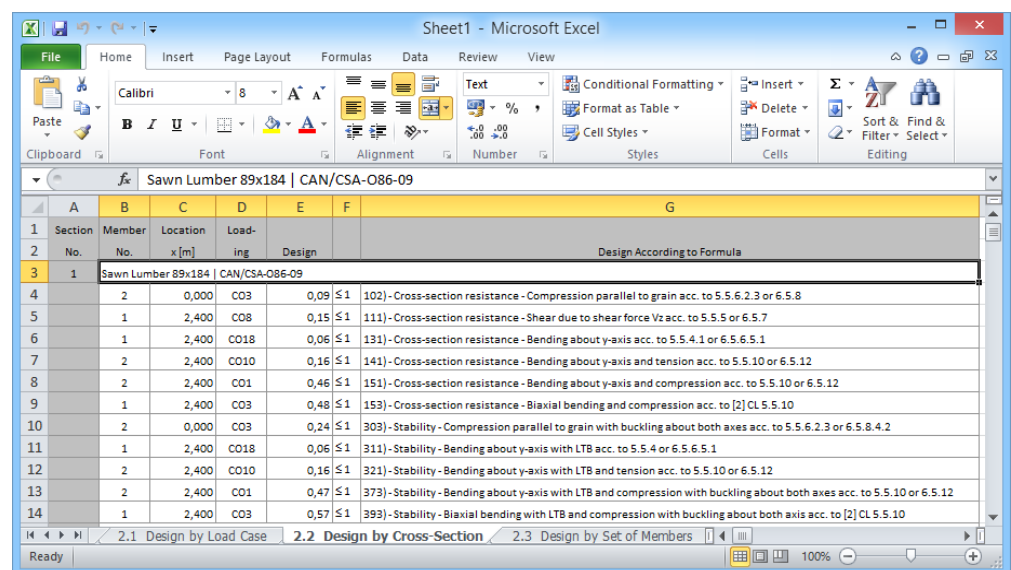


Figure 7.12: Dialog box *Export - MS Excel*

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.



Section	Member	Location	Load	Design	
1	No.	No.	x [m]	Design	Design According to Formula
3	1	Sawn Lumber 89x184 CAN/CSA-O86-09			
4	2	0,000	CO3	0,09 ≤ 1	102) - Cross-section resistance - Compression parallel to grain acc. to 5.5.6.2.3 or 6.5.8
5	1	2,400	CO8	0,15 ≤ 1	111) - Cross-section resistance - Shear due to shear force Vz acc. to 5.5.5 or 6.5.7
6	1	2,400	CO18	0,06 ≤ 1	131) - Cross-section resistance - Bending about y-axis acc. to 5.5.4.1 or 6.5.6.5.1
7	2	2,400	CO10	0,16 ≤ 1	141) - Cross-section resistance - Bending about y-axis and tension acc. to 5.5.10 or 6.5.12
8	2	2,400	CO1	0,46 ≤ 1	151) - Cross-section resistance - Bending about y-axis and compression acc. to 5.5.10 or 6.5.12
9	1	2,400	CO3	0,48 ≤ 1	153) - Cross-section resistance - Biaxial bending and compression acc. to 2] CL 5.5.10
10	2	0,000	CO3	0,24 ≤ 1	303) - Stability - Compression parallel to grain with buckling about both axes acc. to 5.5.6.2.3 or 6.5.8.4.2
11	1	2,400	CO18	0,06 ≤ 1	311) - Stability - Bending about y-axis with LTB acc. to 5.5.4 or 6.5.6.5.1
12	2	2,400	CO10	0,16 ≤ 1	321) - Stability - Bending about y-axis with LTB and tension acc. to 5.5.10 or 6.5.12
13	2	2,400	CO1	0,47 ≤ 1	373) - Stability - Bending about y-axis with LTB and compression with buckling about both axes acc. to 5.5.10 or 6.5.12
14	1	2,400	CO3	0,57 ≤ 1	393) - Stability - Biaxial bending with LTB and compression with buckling about both axes acc. to 2] CL 5.5.10

Figure 7.13: Results in *Excel*

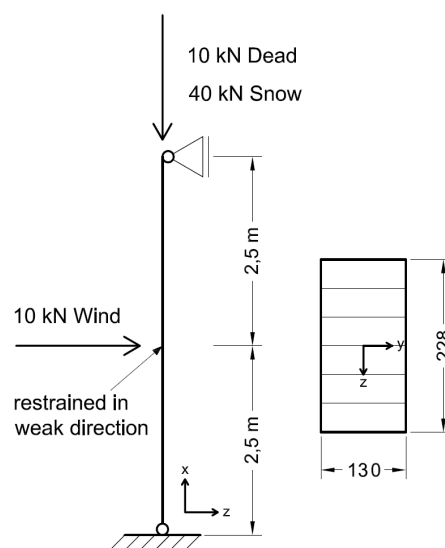
8. Examples

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: $b = 130 \text{ mm}$
 $d = 228 \text{ mm}$

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 RSTAB

The system as well as the loads in all load cases is modeled in RSTAB 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**.

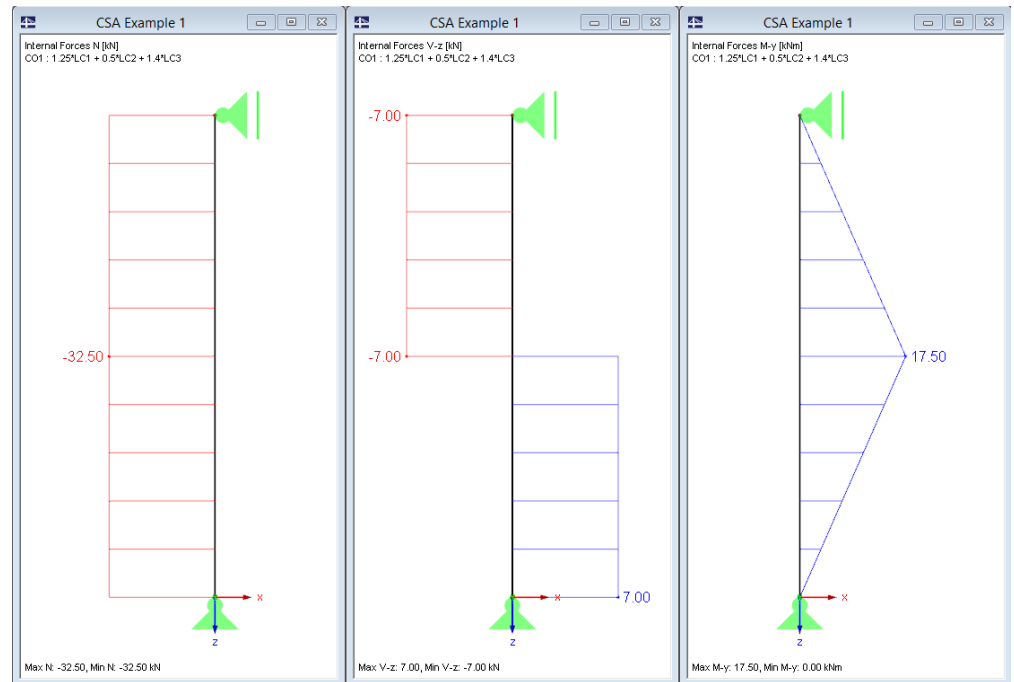


Figure 8.2: Internal forces N, V_z and M_y (ULS)

8.1.3 Design with 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.

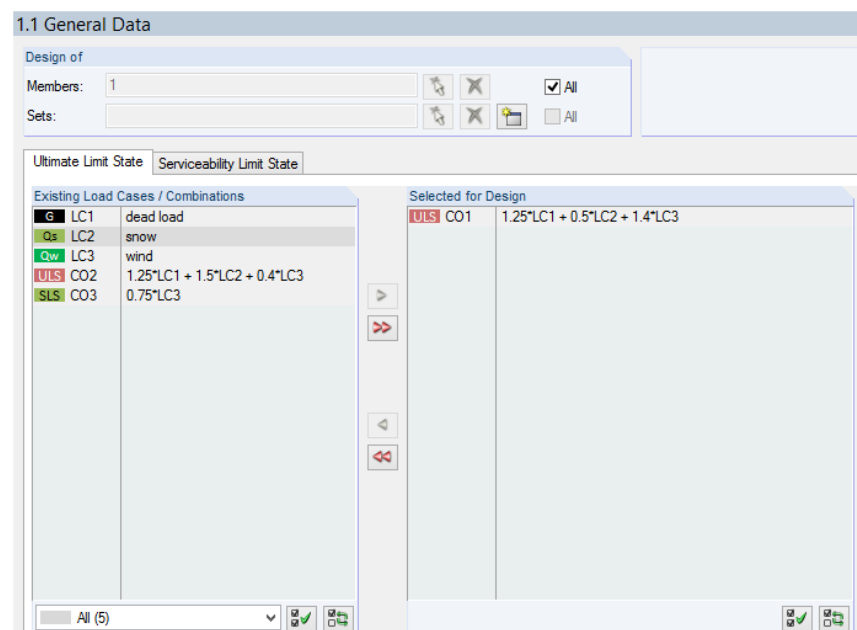


Figure 8.3: Window 1.1 *General Data*

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

1.2 Materials

Material No.	Material Description	Comment
1	Spruce-Lodgepole Pine-Jack Pine 20F-EX CAN/CSA-O86-09:2009-05	

Material Properties

Main Properties

Property	Symbol	Value	Unit
Modulus of Elasticity	E	10300.0	N/mm ²
Shear Modulus	G	669.5	N/mm ²
Specific Weight	γ	4.40	kN/m ³
Coefficient of Thermal Expansion	α	4.0000E-06	1/°C
Partial Safety Factor	γ_M	1.00	

Additional Properties

Property	Symbol	Value	Unit
Modulus of Elasticity for Design of Compression Members	E ₀₅	8961.0	N/mm ²
Modulus of Elasticity (Axial Loading)	E _x	10300.0	N/mm ²
Shear Modulus (Axial Loading)	G _x	669.5	N/mm ²
Modulus of Elasticity (Loading Perpendicular to Wide Faces of Lams)	E _y	10300.0	N/mm ²
Shear Modulus (Loading Perpendicular to Wide Faces of Lams)	G _y	669.5	N/mm ²
Modulus of Elasticity (Loading Parallel to Wide Faces of Lams)	E _z	9500.0	N/mm ²
Shear Modulus (Loading Parallel to Wide Faces of Lams)	G _z	617.5	N/mm ²
Strength for Compression Parallel	f _c	25.200	N/mm ²
Strength for Bending Moment (Bending Perpendicular to Wide Faces of Lams)	f _{by,pos}	25.600	N/mm ²
Strength for Bending Moment (Bending Perpendicular to Wide Faces of Lams)	f _{by,neg}	25.600	N/mm ²
Strength for Bending Moment (Bending Parallel to Wide Faces of Lams)	f _{bz}	11.800	N/mm ²
Strength for Longitudinal Shear	f _{vz}	1.750	N/mm ²
Strength for Longitudinal Shear	f _{vy}	1.500	N/mm ²
Strength for Compression Parallel Combin. with Bending	f _{cb}	25.200	N/mm ²
Strength for Compression Perpendicular (Compr. Face Bearing)	f _{cp,c}	5.800	N/mm ²
Strength for Compression Perpendicular (Tens. Face Bearing)	f _{cp,t}	5.800	N/mm ²
Strength for Tension Net Section	f _{tn}	17.000	N/mm ²
Strength for Tension Gross Section	f _{tg}	12.700	N/mm ²

Material No. 1 used in

Cross-sections No.: 1

Members No.: 1

Sets of members No.:

Σ Length: 5.00 [m] Σ Weight: 0.065 [t]

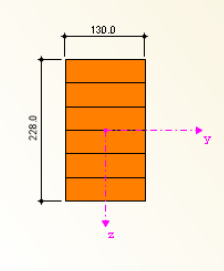
Figure 8.4: Window 1.2 *Materials*

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

1.3 Cross-Sections

Section No.	Material No.	Cross-Section Description	Optimize	Note	Comment
1	1	Glulam 130x228 CAN/CSA-O86-09	No		

1 - Glulam 130x228 | CAN/CSA-O86-09



[mm]

Cross-section No. 1 used in

Members No.: 1

Sets of members No.:

Σ Length: 5.00 [m] Σ Weight: 0.065 [t]

Material: 1 - Spruce-Lodgepole Pine-Jack Pine 20F-EX

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.

1.4 Load Duration

Load- ing	A Description	B Load Type	C Load Duration	D Factor K_D	E Comments
LC1	dead load	Permanent	Long term	0.650	
LC2	snow	Snow / ice	Standard term	1.000	
LC3	wind	Wind	Short term	1.150	
CO1	1.25*LC1 + 0.5*LC	-	Short term	1.150	
CO3	0.75*LC3	-	Short term	1.150	

Load Duration - Explanatory Notes

Short term:
means the condition of loading where the duration of the specified loads is not expected to last more than 7 days continuously or cumulatively throughout the life of the structure

Standard term:
means the condition of loading where the duration of specified loads exceeds that of short-term loading, but is less than long-term loading

Long term:
means the condition of loading under which a member is subjected to more or less continuous specified load

Apply load duration factor K_D according to: ☒ Shortest load duration in a combination ☐ User-defined setting

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.

1.5 Service Conditions and Treatment - Members

Member No.	A Moisture Service Condition	B Treatment	C Comments
1	Dry	None or Preservative (not incised)	

Moisture Service Condition

Dry service conditions:
a climatic condition in which the average equilibrium moisture content of solid wood over a year is 15% or less and does not exceed 19%

Wet service conditions:
all service conditions other than dry

Treatment Factor

Where wood is impregnated with fire-retardant or other strength-reducing chemicals, K_T shall be determined in accordance with the results of appropriate tests or shall not exceed the tabulated value of K_T

☐ Set input for members No.: ☒ All

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 $L_{ez} = 2.50$ 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 $K_L = 1.0$.

1.7 Effective Lengths - Members

Member No.	A		B		C		D		E		F		G		H		I		J		K
	Buckling Possible	Possible	Key	L_{ey} [m]	Possible	K_{ez}	L_{ez} [m]	Possible	Define L_e	L_e [m]	Comment										
1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	5.000	<input checked="" type="checkbox"/>	0.500	2.500	<input type="checkbox"/>	as member length	5.000											

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.

1.9 Additional Design Parameters

Member No.	A		B		C		D
	Material Category	System Factor K_H	Load Sharing	According to	Comment		
1	Glulam	No	-	-			

☐ Set input for members No.:
☒ All

Material Description: Spruce-Lodgepole Pine-Jack Pine 20F-EX | CAN/CSA-O86-09:2009-05

Figure 8.9: Window 1.9 *Additional Design Parameters*

Calculation

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.

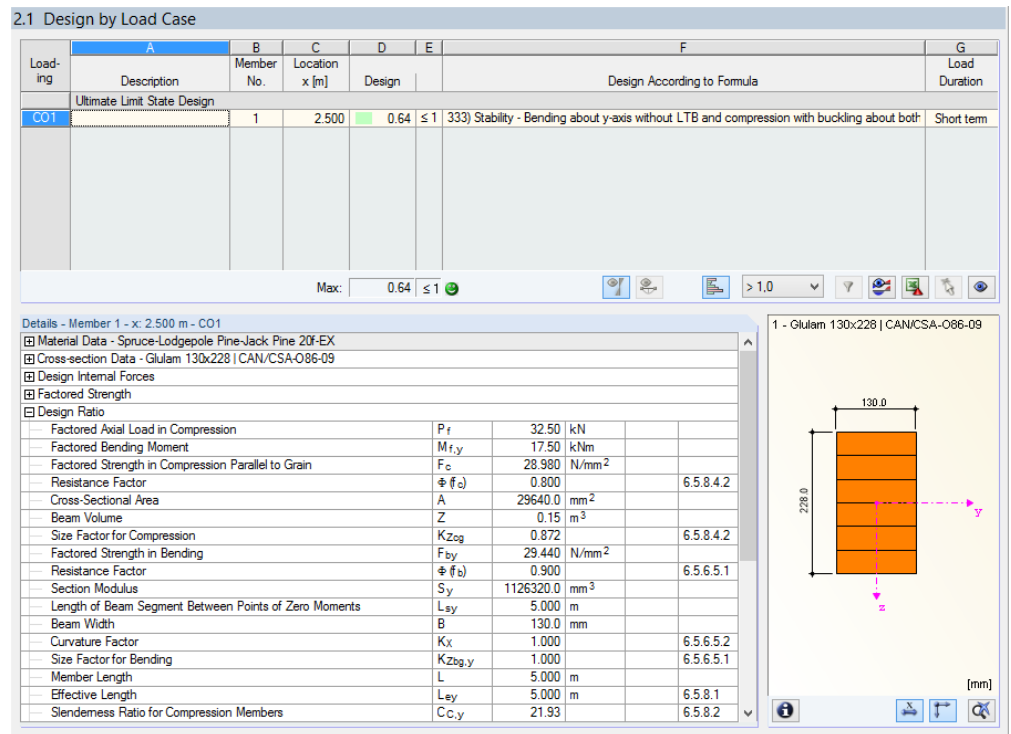


Figure 8.10: Window 2.1 *Design by Load Case*

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.

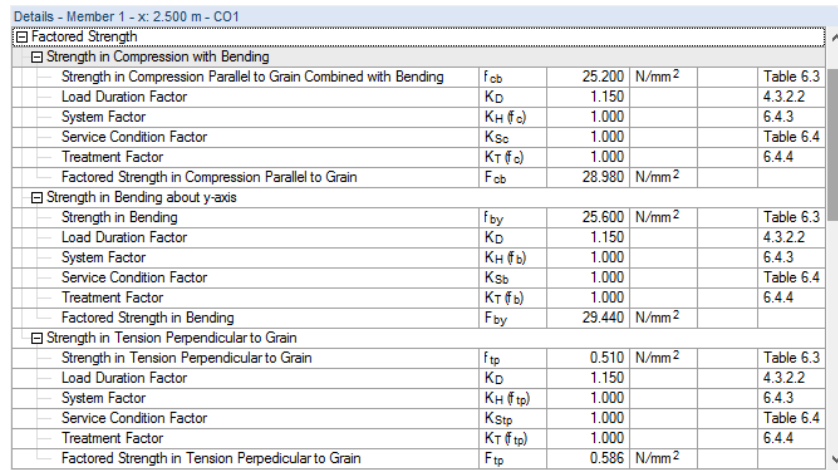


Figure 8.11: Details - *Factored Strength*

Factored Compressive Resistance

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

$$F_c = f_{c,b} \times (K_D \times K_H \times K_{Sc} \times K_T)$$

$$F_c = 25.2 \times (1.15 \times 1.0 \times 1.0 \times 1.0)$$

$$F_c = 28.98 \text{ N/mm}^2$$

$$K_{Zcg} = 0.68 \times Z^{-0.13} \leq 1.0$$

$$K_{Zcg} = 0.68 \times 0.1482^{-0.13} \leq 1.0$$

$$K_{Zcg} = 0.872$$

$$K_C = \left[1.0 + \frac{F_c \times K_{Zcg} \times C_c^3}{35 \times E_{05} \times K_{SE} \times K_T} \right]^{-1}$$

$$C_{c,y} = \frac{l_{e,y}}{d} = \frac{5.0}{0.228} = 21.93$$

$$C_{c,z} = \frac{l_{e,z}}{d} = \frac{2.5}{0.13} = 19.23$$

$$E_{05} = 0.87 \times 10300 = 8961 \text{ N/mm}^2$$

$$K_{C,y} = \left[1.0 + \frac{28.98 \times 0.872 \times 21.93^3}{35 \times 8961 \times 1.0 \times 1.0} \right]^{-1}$$

$$K_{C,y} = 0.541$$

$$K_{C,z} = \left[1.0 + \frac{28.98 \times 0.872 \times 19.23^3}{35 \times 8961 \times 1.0 \times 1.0} \right]^{-1}$$

$$K_{C,z} = 0.636$$

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

$$P_{r,y} = 323.85 \text{ kN}$$

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

$$P_{r,z} = 380.81 \text{ kN}$$

Combined Bending and Axial Load

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

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}

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

$$F_b = 25.6 \times (1.15 \times 1.0 \times 1.0 \times 1.0)$$

$$F_b = 29.44 \text{ N/mm}^2$$

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

$$K_{Zbg,y} = 1.03 \times (0.13 \times 5.0)^{-0.18} \leq 1.0$$

$$K_{Zbg,y} = 1.0$$

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

$$M_{r1,y} = 29.84 \text{ kNm}$$

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

$$M_{r2,y} = 29.84 \text{ kNm}$$

Euler Buckling Load

$$P_E = \frac{\pi^2 \times E_{05} \times K_{SE} \times K_T \times I}{L_e^2}$$

$$P_{E,y} = \frac{\pi^2 \times 8961 \times 1.0 \times 1.0 \times 1.284 \times 10^8}{5000^2}$$

$$P_{E,y} = 454.24 \text{ kN}$$

Buckling about axis y

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

Buckling about axis z

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

Factored Shear Resistance

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

$$F_v = f_v \times (K_D \times K_H \times K_{Sv} \times K_T)$$

$$F_v = 1.75 \times (1.15 \times 1.0 \times 1.0 \times 1.0)$$

$$F_v = 2.01 \text{ N/mm}^2$$

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

$$V_{r,z} = 35,79 \text{ kN}$$

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**.

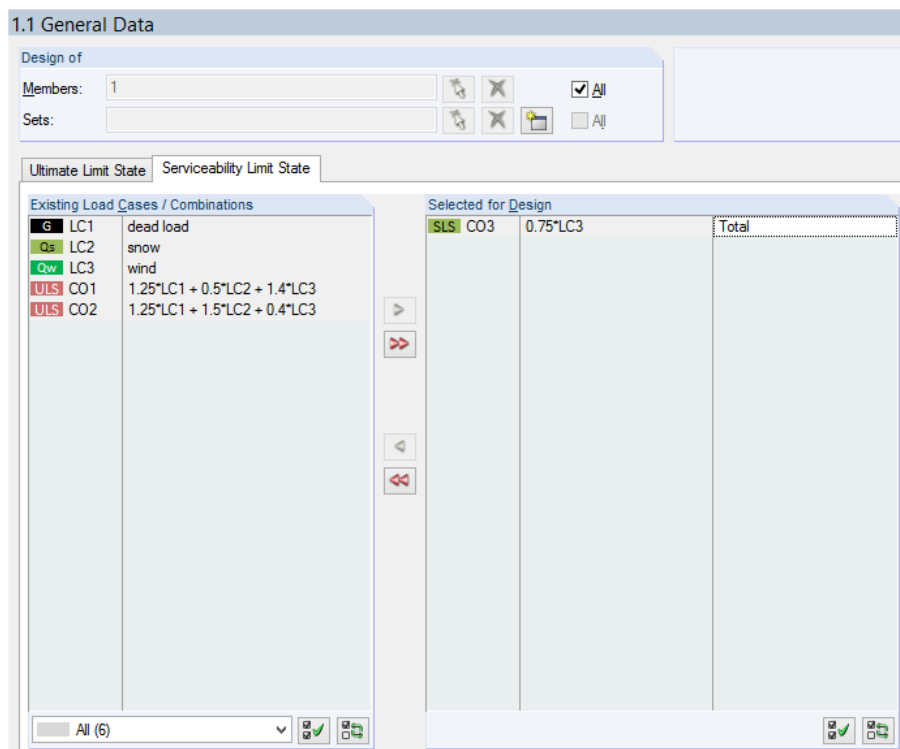


Figure 8.12: Window 1.1 *General Data*, tab *Serviceability Limit State*

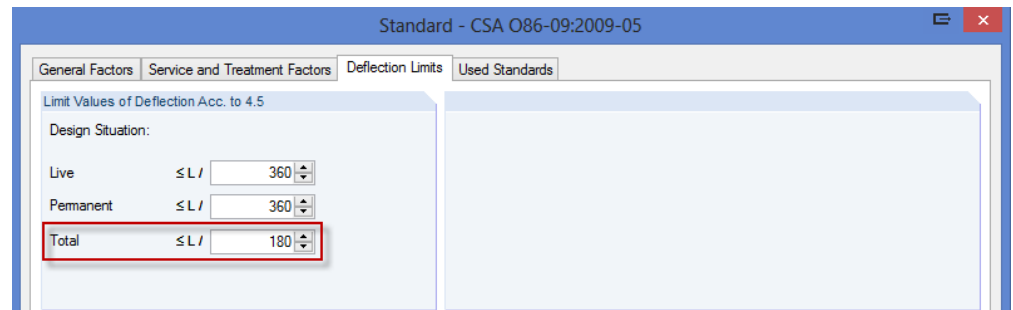
Then we enter member No. **1** in the 1.12 *Serviceability Data* window.

1.12 Serviceability Data								
No.	A Reference to	B Member No.	C Reference Manually	D Length L [m]	E Direction	F Precamber w ₀ [mm]	G Beam Type	H Comment
1	Member	1	<input type="checkbox"/>	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.



Standard - CSA O86-09:2009-05

General Factors | Service and Treatment Factors | Deflection Limits | Used Standards

Limit Values of Deflection Acc. to 4.5

Design Situation:

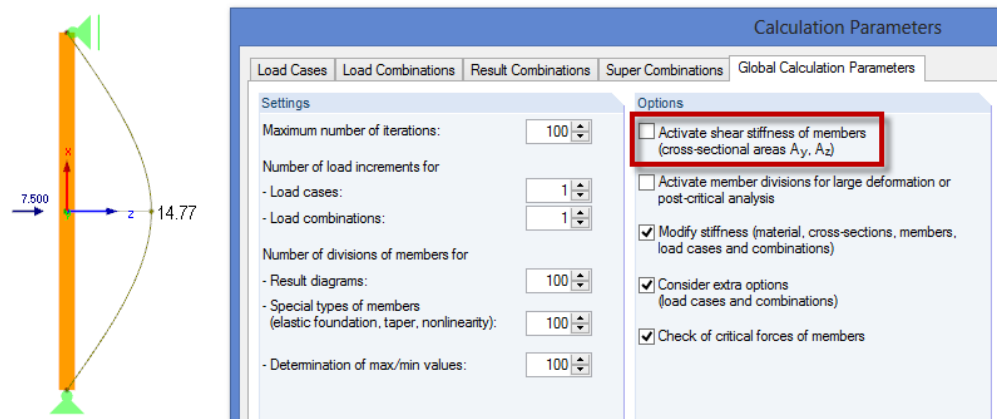
Live $\leq L / 360$

Permanent $\leq L / 360$

Total $\leq L / 180$

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 RSTAB.



Calculation Parameters

Load Cases | Load Combinations | Result Combinations | Super Combinations | Global Calculation Parameters

Settings

Maximum number of iterations: 100

Number of load increments for:

- Load cases: 1
- Load combinations: 1

Number of divisions of members for:

- Result diagrams: 100
- Special types of members (elastic foundation, taper, nonlinearity): 100
- Determination of max/min values: 100

Options

- ☐ Activate shear stiffness of members (cross-sectional areas A_y, A_z)
- ☐ Activate member divisions for large deformation or post-critical analysis
- ☒ Modify stiffness (material, cross-sections, members, load cases and combinations)
- ☒ Consider extra options (load cases and combinations)
- ☒ Check of critical forces of members

Deflection diagram of CO3: A vertical beam of length 7.500 m is shown with a deflection curve. The deflection at the top is 14.77 mm. The beam is supported at the bottom by a green triangle.

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

Deflection Design

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

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

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

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
	Serviceability Limit State Design						
CO3		1	2.500	0.53	≤ 1	403) Serviceability - Total Design Situation acc. to 4.5 - Inner span, z-direction	Short term

Max: 0.53 ≤ 1

Details - Member 1 - x: 2.500 m - CO3

- Material Data - Spruce-Lodgepole Pine-Jack Pine 20f-EX
- Cross-section Data - Glulam 130x228 | CAN/CSA-O86-09
- Deformations
- Design Ratio

Deflection	Δz	14.77	mm	
Reference Span	L	5.000	m	
Deflection Limitation	L / Δ_{\max}	180.000		
Maximum Deflection	$\Delta_{\max,z}$	27.78	mm	
Design Ratio	η	0.53		≤ 1

1 - Glulam 130x228 | CAN/CSA-O86-09

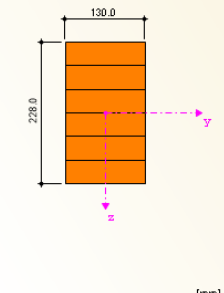


Figure 8.16: Window 2.1 Design by Load Case

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

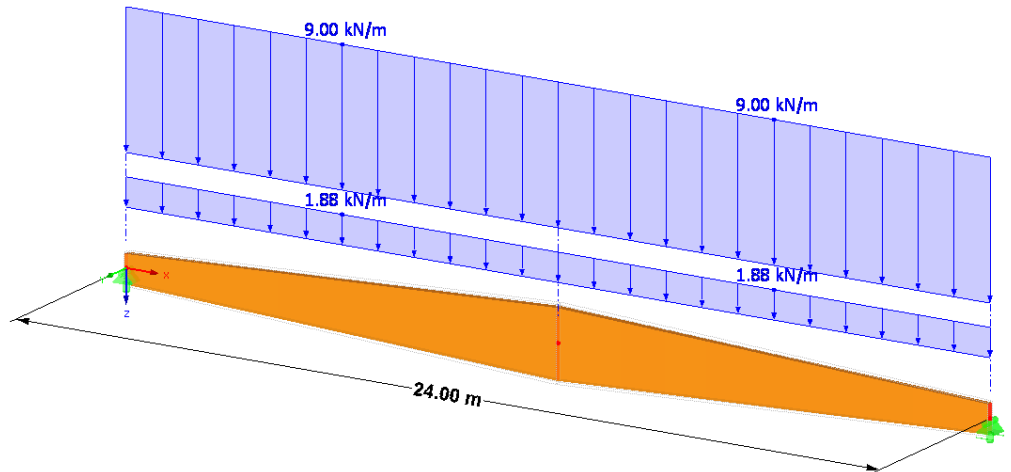


Figure 8.17: System and loads

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 RSTAB

The system as well as the loads in all load cases is modeled as a 3D model in RSTAB. 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. RSTAB determines the diagrams of internal forces shown in the following figure.

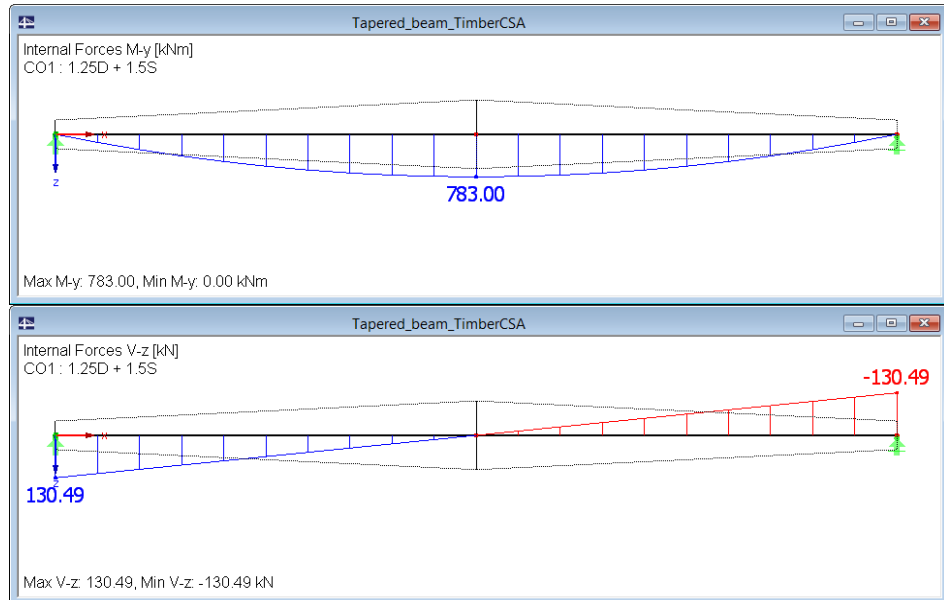


Figure 8.18: Internal forces M_y and V_z

8.2.3 Design with 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.

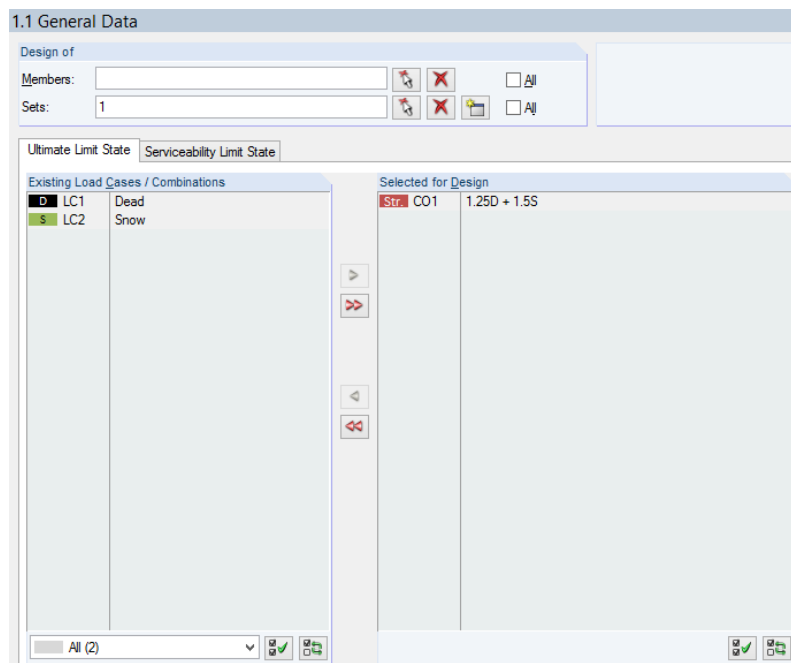


Figure 8.19: Window 1.1 *General Data*

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

1.2 Materials

Material No.	A Material Description	B Comment
1	Douglas Fir-Larch 24F-E CAN/CSA-O86-09:2009-05	

Material Properties

Main Properties

Property	Value	Unit
Modulus of Elasticity	E	1280.00 kN/cm ²
Shear Modulus	G	83.20 kN/cm ²
Specific Weight	γ	4.90 kN/m ³
Coefficient of Thermal Expansion	α	4.0000E-06 1/°C
Partial Safety Factor	γ_M	1.00

Additional Properties

Property	Value	Unit
Modulus of Elasticity for Design of Compression Members	E ₀₅	1113.60 kN/cm ²
Modulus of Elasticity (Axial Loading)	E _x	1280.00 kN/cm ²
Shear Modulus (Axial Loading)	G _x	83.20 kN/cm ²
Modulus of Elasticity (Loading Perpendicular to Wide Faces of Lams)	E _y	1280.00 kN/cm ²
Shear Modulus (Loading Perpendicular to Wide Faces of Lams)	G _y	83.20 kN/cm ²
Modulus of Elasticity (Loading Parallel to Wide Faces of Lams)	E _z	1100.00 kN/cm ²
Shear Modulus (Loading Parallel to Wide Faces of Lams)	G _z	71.50 kN/cm ²
Strength for Compression Parallel	f _c	30.200 MPa
Strength for Bending Moment (Bending Perpendicular to Wide Face)	f _{by,pos}	30.600 MPa
Strength for Bending Moment (Bending Perpendicular to Wide Face)	f _{by,neg}	23.000 MPa
Strength for Bending Moment (Bending Parallel to Wide Faces of Lams)	f _{bz}	10.000 MPa
Strength for Longitudinal Shear	f _{vz}	2.000 MPa
Strength for Longitudinal Shear	f _{vy}	1.900 MPa
Strength for Compression Parallel Combin. with Bending	f _{cb}	30.200 MPa
Strength for Compression Perpendicular (Compr. Face Bearing)	f _{cp,c}	7.000 MPa
Strength for Compression Perpendicular (Tens. Face Bearing)	f _{cp,t}	7.000 MPa
Strength for Tension Net Section	f _{tn}	20.400 MPa
Strength for Tension Gross Section	f _{tg}	15.300 MPa

Material No. 1 used in

Cross-sections No.: 4,5

Members No.: 1,2

Sets of members No.: 1

Σ Length: 24.00 [m] Σ Weight: 2.699 [t]

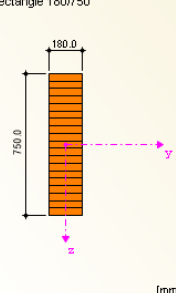
Figure 8.20: Window 1.2 *Materials*

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

1.3 Cross-Sections

Section No.	A Material No.	B Cross-Section Description	C Max. Design Ratio	D Optimize	E Note	F Comment
4	1	T-Rectangle 180/750	0.93	No		
5	1	T-Rectangle 180/1800	0.93	No		

4 - T-Rectangle 180/750



[mm]

Cross-section No. 4 used in

Members No.: 1,2

Sets of members No.: 1

Σ Length: 24.00 [m] Σ Weight: 2.699 [t]

Material: 1 - Douglas Fir-Larch 24F-E

11) The cross-section cannot be optimized, when it is combined with another cross-section on any other member.

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**.

1.4 Load Duration

Load- ing	A Description	B Load Type	C Load Duration	D Factor K_D	E Loading Distribution	F Comments
LC1	Dead	Dead	Long term	0.650	uniform	
LC2	Snow	Snow	Standard term	1.000	uniform	
CD1	D + S	-	Standard term	1.000	uniform	

Load Duration - Explanatory Notes

Short term:
The condition of loading where the duration of the specified loads is not expected to last more than 7 days continuously or cumulatively throughout the life of the structure.

Standard term:
The condition of loading where the duration of specified loads exceeds that of short-term loading, but is less than long-term loading.

Long term:
The condition of loading under which a member is subjected to more or less continuous specified load.

Apply load duration factor K_D according to:
☒ Shortest load duration in a combination
☐ User-defined setting

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.

1.6 Service Conditions and Treatment - Sets of Members

Set No.	A Moisture Service Condition	B Treatment	C Comments
1	Wet	None or Preservative (not incised)	

Moisture Service Condition

Dry service conditions:
A climatic condition in which the average equilibrium moisture content of solid wood over a year is 15% or less and does not exceed 19%.

Wet service conditions:
All service conditions other than dry.

Treatment Factor

Where wood is impregnated with fire-retardant or other strength-reducing chemicals, K_T shall be determined in accordance with the results of appropriate tests or shall not exceed the tabulated value of K_T .

☐ Set input for members No.:

☒ All

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**.

1.8 Effective Lengths - Sets of Members

Set No.	A Buckling Possible	B Possible	C Buckling About Axis y Key	D L _{ey} [m]	E Possible	F Buckling About Axis z Key	G L _{ez} [m]	H Possible	I Lateral-Torsional Buckling Define L _e	J L _e [m]	K Comment
1	<input type="checkbox"/>	<input type="checkbox"/>	1.000	24.00	<input type="checkbox"/>	1.000	24.00	<input type="checkbox"/>	as member length	24.00	

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.

1.9 Additional Design Parameters

Member No.	A Material Category	B System Factor K_H Load Sharing	C According to	D Comment
1	Glulam	No	-	
2	Glulam	No	-	

☐ Set input for members No.:

Material description: ☒ All

Douglas Fir-Larch 24f-E | CAN/CSA-O86-09:2009-05

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.

1.10 Tapered Members

Member No.	Cross-Section		Length L [m]	Angle of Taper α [°]	Radial Stress Design with Apex	Note	Comment
	Member Start	Member End					
1	T-Rectangle 180/750	T-Rectangle 180/180	12.00	5.00	<input checked="" type="checkbox"/>	12)	
2	T-Rectangle 180/180	T-Rectangle 180/750	12.00	5.00	<input checked="" type="checkbox"/>	12)	

☐ Set input for members No.:

☒ All

12) For radial stress design check also the loading distribution settings in Window 1.4.

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*.

2.3 Design by Set of Members

Set No.	Member No.	Location x [m]	Load- ing	Design	
1	1 (Member No. 1.2)				
	1	0.00	CO1	0.93	1111) Cross-section resistance - Tapered Member - Shear due to shear force Vz acc. to 6.5.7
	1	4.80	CO1	0.87	1131) Cross-section resistance - Tapered Member - Bending about y-axis acc. to 6.5.6.5.1
	1	12.00	CO1	0.40	1221) Cross-section resistance - Tapered Member - Radial tension perpendicular to grain acc. to 6.5.6.6.2

Max: 0.93 ≤ 1

Details - Member 1 - x: 0.00 m - CO1

- Material Data - Douglas Fir-Larch 24f-E
- Cross-section Data - T-Rectangle 180/750
- Design Internal Forces
- Factored Strength
 - Strength in Shear due to Shear Force Vz

	f_{vz}	2.000 MPa	Table 6.3
Load Duration Factor	K_D	1.000	4.3.2.2
System Factor	$K_H(f_v)$	1.000	6.4.3
Service Condition Factor	K_{Sv}	0.870	Table 6.4
Treatment Factor	$K_T(f_v)$	1.000	6.4.4
Factored Strength in Shear	F_{vz}	1.740 MPa	
 - Design Ratio

	$V_{t,z}$	130.49 kN	
Factored Shear Force	F_v	1.740 MPa	
Resistance Factor	$\phi(f_v)$	0.900	6.5.7.2.1
Cross-Sectional Area	A	1350.00 cm ²	
Beam Volume	Z	5.51 m ³	
Factored Shear Resistance	$V_{t,z}$	140.94 kN	6.5.7.2.1
Design Ratio	η	0.93	≤ 1 6.5.7.2.1

4 - 5: T-Rectangle 180/750 - T-Rectangle...

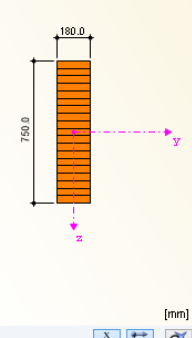


Figure 8.27: Window 2.3 *Design by Set of Members*

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

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

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

$$F_b = 30.6 \times (1.0 \times 1.0 \times 0.8 \times 1.0)$$

$$F_b = 24.48 \text{ MPa}$$

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

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

$$K_{Zbg,y} = 0.791$$

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

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

$$M_{r,y} = 579.28 \text{ kNm}$$

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

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

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

A

Additional design parameters	30
Angle of taper.....	31
Apex.....	31

B

Background graphic.....	49
Beam type.....	32
Buckling.....	26
Buckling about axis	27
Buckling length.....	25, 27
Buckling length coefficient.....	27
Buttons	48

C

Calculation.....	33
Cantilever.....	32
Case.....	30
Clipboard	61
Color spectrum	52
Colored design.....	52
Connections.....	33
Control panel.....	52
Cross-section	18, 31, 58
Cross-section design	40
Cross-section info.....	20
Cross-section library.....	18
Cross-section optimization	58

D

Decimal places	15, 60
Deflection limits.....	14
Deformation analysis.....	32
Design	9, 20, 38, 39, 40
Design case	49, 56, 57
Design ratio.....	39
Design situation.....	14
Details	33
Display navigator.....	50, 52

E

Effective length.....	25, 29, 61
Equivalent member.....	29
Equivalent member length.....	25, 27
Equivalent member method.....	35
Evaluation of results.....	48

Excel	62
Exit TIMBER CSA	8
Export	61
Export cross-section.....	59
Export effective length	61
Export material	61

F

Factor K_D	23
Filter	48, 52
Filtering members	53
Flexural buckling.....	26

G

General Data.....	8
Graphic	49
Graphic printout.....	54
Graphical representation	50

H

Hidden result diagram	52
-----------------------------	----

I

Imperfections.....	35
Installation	5
Internal forces.....	43, 59

L

Lateral-torsional buckling	27, 35
Length	25, 46
Limit tension stress	33
Limit values.....	11
Load case	9, 11, 43
Load combination	9
Load duration.....	22
Load duration factor	12, 22, 23
Load type.....	22
Loading distribution	23
Location x.....	39, 42

M

Material	15, 61
Material description.....	15
Material library.....	16
Material properties.....	15
Member list	32
Member slendernesses.....	36, 45

Members	9	Service conditions	23, 24
Moisture	23	Serviceability	32, 35
N		Serviceability limit state	11, 48
Navigator	8	Set of members	9, 24, 29, 32, 41, 44, 47
Negative bending	34	Shifted ends of members	35
O		Slenderness	45
OpenOffice	62	Stability analysis	34, 40
Optimization	20, 36, 58, 59	Standard	14
P		Standardized cross-section	18
Panel	7, 50, 52	Start calculation	37
Parametrized cross-section	18, 58	Start program	6
Part	46	Start TIMBER CSA	6
Parts list	46, 47	Stress point	21
Positive bending	34	Sum	47
Precamber	32	Surface area	46
Print	21, 54	System factor	12, 30
Printout report	54, 55	T	
R		Taper	36
Radial stress design	31	Tapered member	21, 31, 40
Reference design value	17	Torsion design	34
Reference length	11	Treatment factor	13, 23
Relation scales	48	U	
Remark	20	Ultimate limit state	9, 48
Rendering	52	Undeformed system	35
Resistance	33	Units	15, 60
Resistance factor	12	User profile	60
Result combination	9, 10	V	
Result diagram	51, 54	View mode	48, 49
Results values	49	Visibilities	52
Results window	38	Volume	47
RSBUCK	26	W	
RSTAB graphic	54	Weight	47
RSTAB work window	49	Wide face	17
S		Windows	8
Second-order analysis	35	X	
Selecting windows	8	x-location	43
Service condition factor	13, 23		