Version
October 2017

Add-on Module

RF-/STEEL CSA

Design of Steel Members
According to CSA S16

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 2017
Am Zellweg 2
93464 Tiefenbach
Germany

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

Contents

	Contents	Page
1.	**Introduction**	**3**
1.1	Add-on Module RF-/STEEL CSA	3
1.2	Using the Manual	3
1.3	Starting RF-/STEEL CSA	4
2.	**Input Data**	**5**
2.1	General Data	5
2.1.1	Ultimate Limit State	6
2.1.2	Serviceability Limit State	7
2.2	Materials	8
2.3	Cross-Sections	10
2.4	Intermediate Lateral Restraints	13
2.5	Effective Lengths - Members	14
2.6	Effective Lengths - Sets of Members	17
2.7	Nodal Supports - Sets of Members	18
2.8	Member Hinges - Sets of Members	21
2.9	Serviceability Data	22
3.	**Calculation**	**23**
3.1	Detail Settings	23
3.2	Starting Calculation	25
4.	**Results**	**26**
4.1	Design by Load Case	27
4.2	Design by Cross-Section	28
4.3	Design by Set of Members	29
4.4	Design by Member	30
4.5	Design by x-Location	30
4.6	Governing Internal Forces by Member	31
4.7	Governing Internal Forces by Set of Members	32
4.8	Member Slendernesses	33
4.9	Parts List by Member	34
4.10	Parts List by Set of Members	35
5.	**Results Evaluation**	**36**
5.1	Results in RFEM/RSTAB Model	37
5.2	Result Diagram	40
5.3	Filter for Results	41
5.4	Mode Shapes	42
6.	**Printout**	**43**
6.1	Printout Report	43
6.2	Graphic Printout	44
7.	**General Functions**	**45**
7.1	Design Cases	45
7.2	Cross-Section Optimization	47
7.3	Units and Decimal Places	49
7.4	Data Transfer	50
7.4.1	Exporting Materials to RFEM/RSTAB	50
7.4.2	Exporting Effective Lengths to RFEM/RSTAB	50
7.4.3	Exporting Results	50

© DLUBAL SOFTWARE 2017

8.	**Example**	**52**
8.1	Design Values	52
8.2	Cross-Section Properties - W 310x52	53
8.3	Material Properties - 260W	53
8.4	Classification of Cross-Section	54
8.5	Flexural Buckling	55
8.5.1	Buckling about y-Axis (Strong Axis)	55
8.5.2	Buckling about z-Axis (Weak Axis)	56
8.6	Lateral-Torsional Buckling	57
8.7	Member Strength and Stability	58
8.7.1	Cross-Sectional Strength	58
8.7.2	Overall Member Strength	59
8.7.3	Lateral-Torsional Buckling Strength	60
A.	**Literature**	**62**
B.	**Index**	**63**

© Dlubal Software 2017

1 Introduction

1.1 Add-on Module RF-/STEEL CSA

The Standard CSA S16, *Design of steel structures* [1] describes the design of steel buildings in Canada. With the add-on modules RF-STEEL CSA (for RFEM) and STEEL CSA (for RSTAB), DLUBAL provides powerful tools for the design of steel beam models according to this Standard.

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

RF-/STEEL CSA performs all typical ultimate limit state designs as well as stability and deformation analyses. The program takes into account various actions for the ultimate limit state design. The allocation of designed cross-sections into four classes makes an important part of the design according to the Canadian Standard. The purpose of this classification is to determine the range in which the local buckling in cross-section parts limits the load capacity so that the rotational capacity of cross-sections can be verified. RF-/STEEL CSA determines the limiting width-to-thickness ratios of compressed parts and carries out the classification automatically.

For the stability analysis, you can determine for every single member or set of members whether buckling and lateral-torsional buckling is possible. Lateral restraints can be added for a realistic representation of the structural model.

For models with slender cross-sections, the serviceability limit state has become an essential aspect of the design. The deflection limits are preset according to the Standard, but can be adjusted. In addition, you can specify the reference lengths and precambers, if necessary.

The program allows you to optimize cross-sections and to export them to RFEM or RSTAB. Using the design cases, it is possible to design separate structural components of complex structures or analyze alternatives with different sections or materials.

Since RF-/STEEL CSA is integrated in the main program, the design relevant input data is preset when the module is called up. After the analysis, the design results can be evaluated graphically in the work window of RFEM or RSTAB. Last but not least, it is possible to keep records of the analysis in the global printout report which includes the internal forces and the design results.

We hope you will enjoy working with RF-/STEEL CSA.

Your DLUBAL team

1.2 Using the Manual

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

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

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

1.3 Starting RF-/STEEL CSA

RFEM and RSTAB provide the following options to open the RF-/STEEL CSA add-on module.

Menu

To start the program from the menu bar, select

Add-on Modules → Design - Steel → RF-/STEEL CSA.

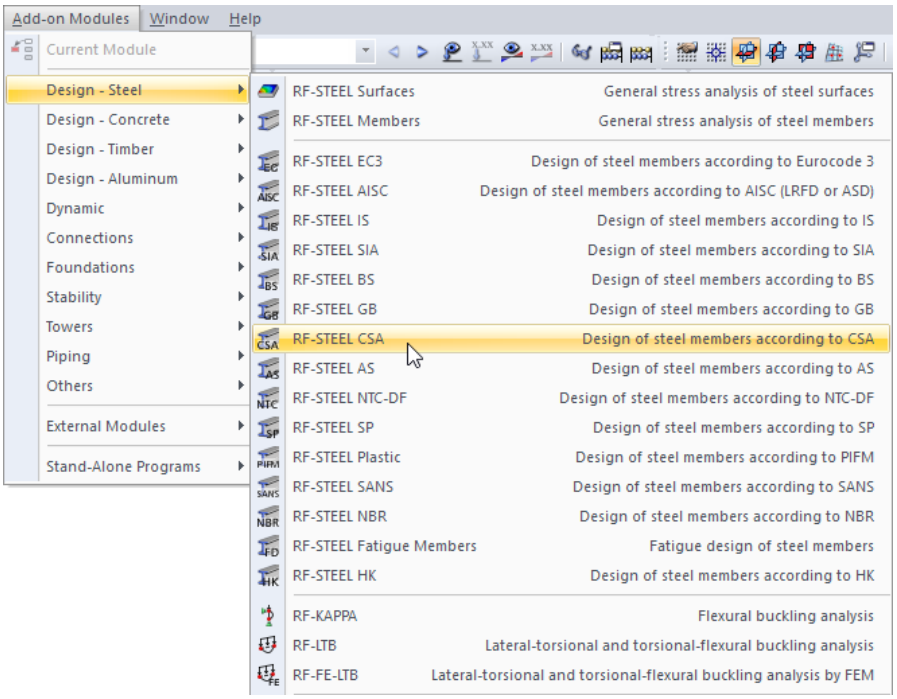


Figure 1.1: Menu *Add-on Modules → Design - Steel → RF-STEEL CSA*

Navigator

You can also start the add-on module in the *Data* navigator by selecting

Add-on Modules → RF-/STEEL CSA.

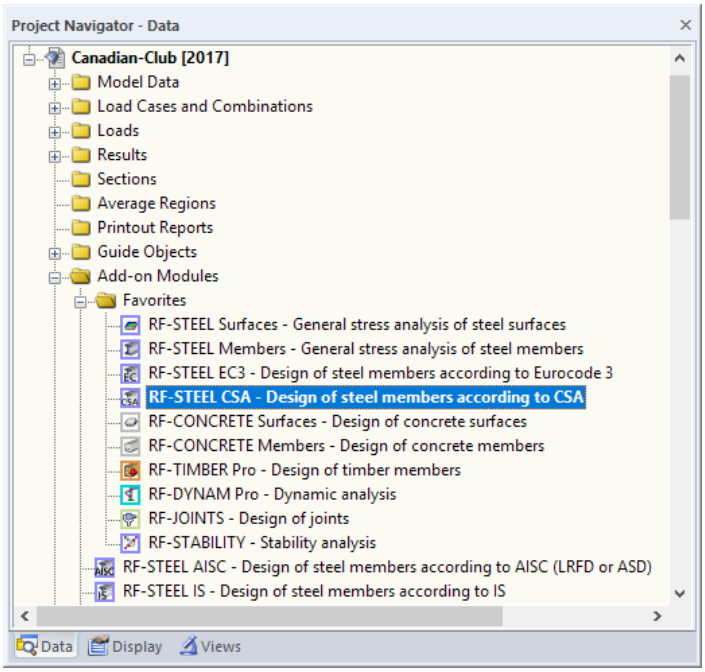


Figure 1.2: *Data* navigator *Add-on Modules → RF-STEEL CSA*

2 Input Data

When you have started the add-on module, a new window appears. In this window, a *Navigator* is displayed on the left. It manages the different window that can be currently selected. The drop-down list above the navigator contains the design cases (see Chapter 7.1, page 45).

The design-relevant data is to be defined in several input windows. When you open RF-/STEEL CSA for the first time, the following parameters are imported automatically:

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

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

[OK] saves the results. Thus, you quit RF-/STEEL CSA and return to RFEM or RSTAB. To exit the add-on module without saving any changes, click [Cancel].

2.1 General Data

In the *1.1 General Data* Window, you can select the objects and the loading relevant for the design. The two tabs manage the load cases, load combinations, and result combinations for the different types of design.

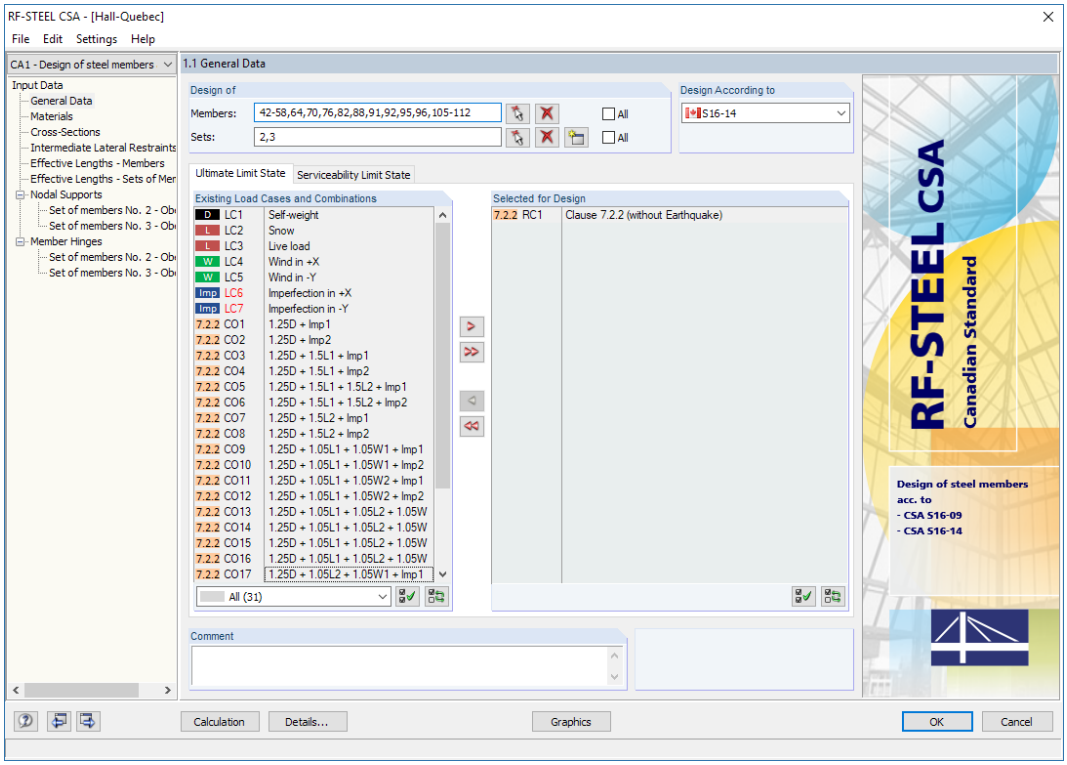


Figure 2.1: Window *1.1 General Data*

Design of

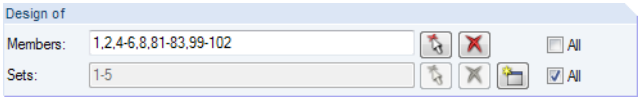


Figure 2.2: Design of members and sets of members

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

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

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

Comment

In this text box, you can enter notes, for example to describe the current design case.

2.1.1 Ultimate Limit State

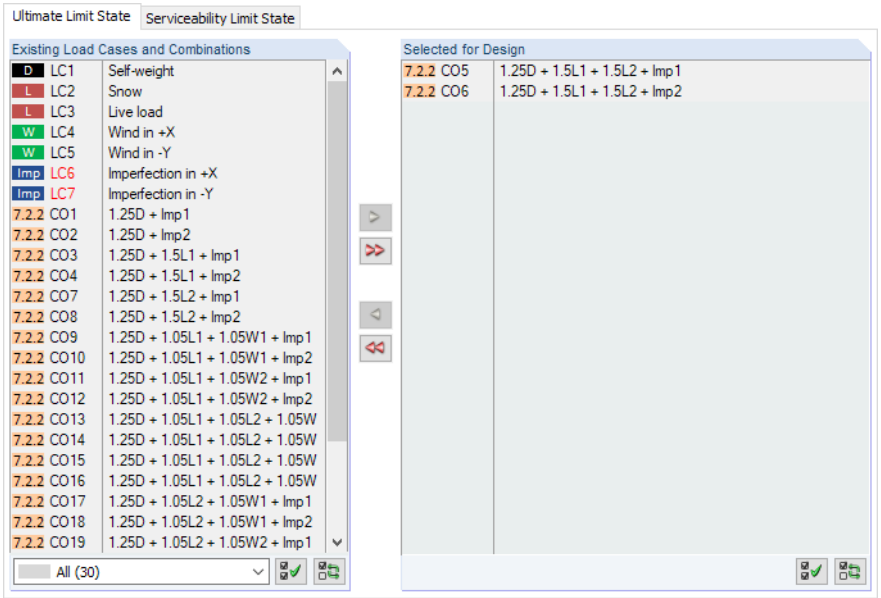


Figure 2.3: Window *1.1 General Data*, tab *Ultimate Limit State*

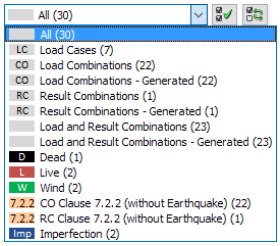
Existing Load Cases and Combinations

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

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

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

If a load case is highlighted in red, it cannot be designed. This happens when a load case has no loads or if it contains only imperfections (see Figure 2.3).



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

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

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

Selected for Design

The column on the right lists the load cases, load and result combinations selected for design. To remove an item from the list, select it and click . You can also double-click the item. To transfer the entire list to the left, click .

Result combination

The design of an enveloping max/min result combination, *RC*, is faster than the design of all contained load cases and load combinations. However, the influence of the contained actions is difficult to check afterwards.

2.1.2 Serviceability Limit State

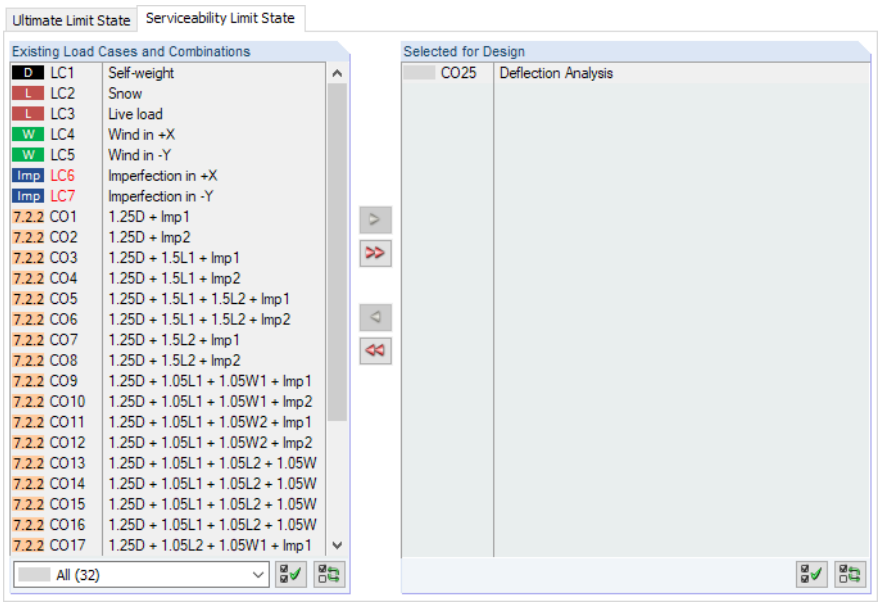


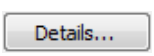
Figure 2.4: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

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



The limit values of the deflections are preset in the *Details* dialog box (see Figure 3.1, page 23). To adjust those values, click the [Details] button.

In the *1.9 Serviceability Data* Window, you can specify the reference lengths of the deformation analysis (see Chapter 2.9, page 22).

2.2 Materials

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

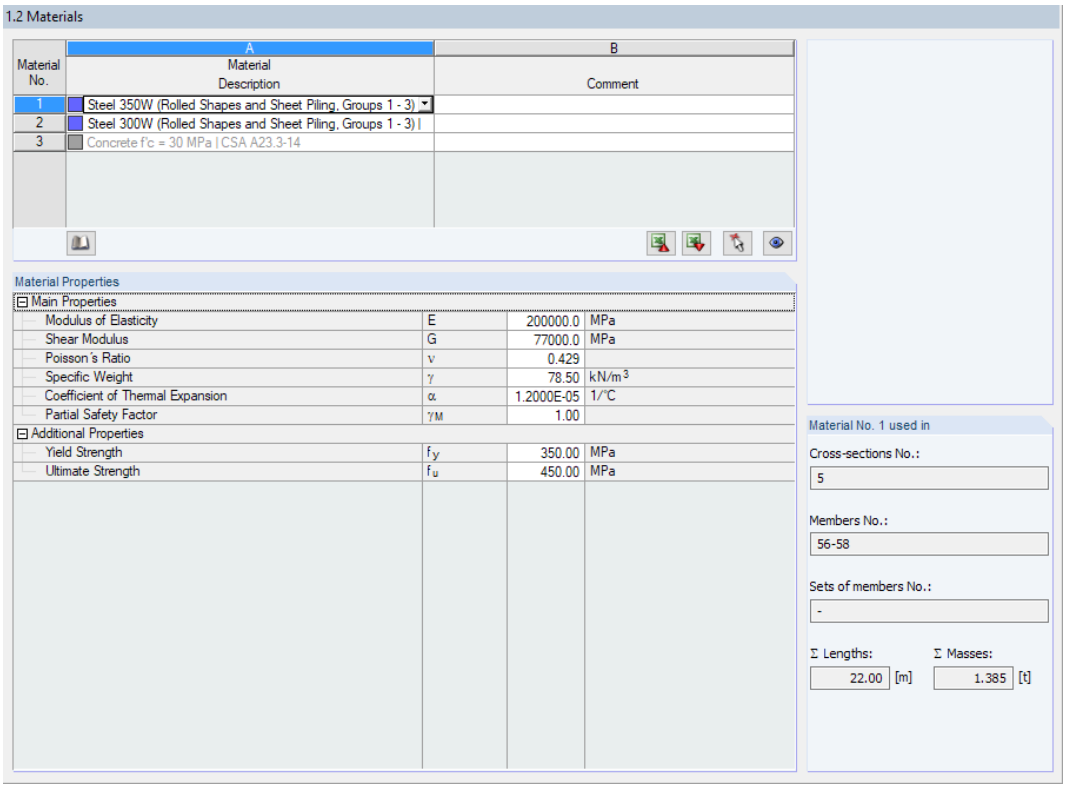


Figure 2.5: Window *1.2 Materials*

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

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

To adjust the units and decimal places of the material properties and stresses, select **Settings** → **Units and Decimal Places** from the menu bar of the module (see Chapter 7.3, page 49).

Material Description

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

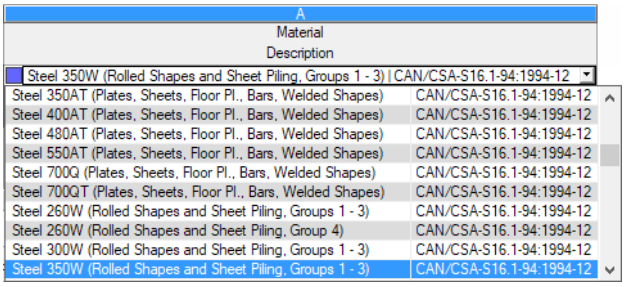


Figure 2.6: List of materials

According to the design concept of the Standard [1], only materials of the Canadian *Steel* category are available in the list.

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

As a matter of principle, the material properties cannot be edited in the RF-/STEEL CSA module.

Material Library

Alternatively, you can use the material library to change a material. To open the library, select

Edit → Material Library

or use the [Library] button.

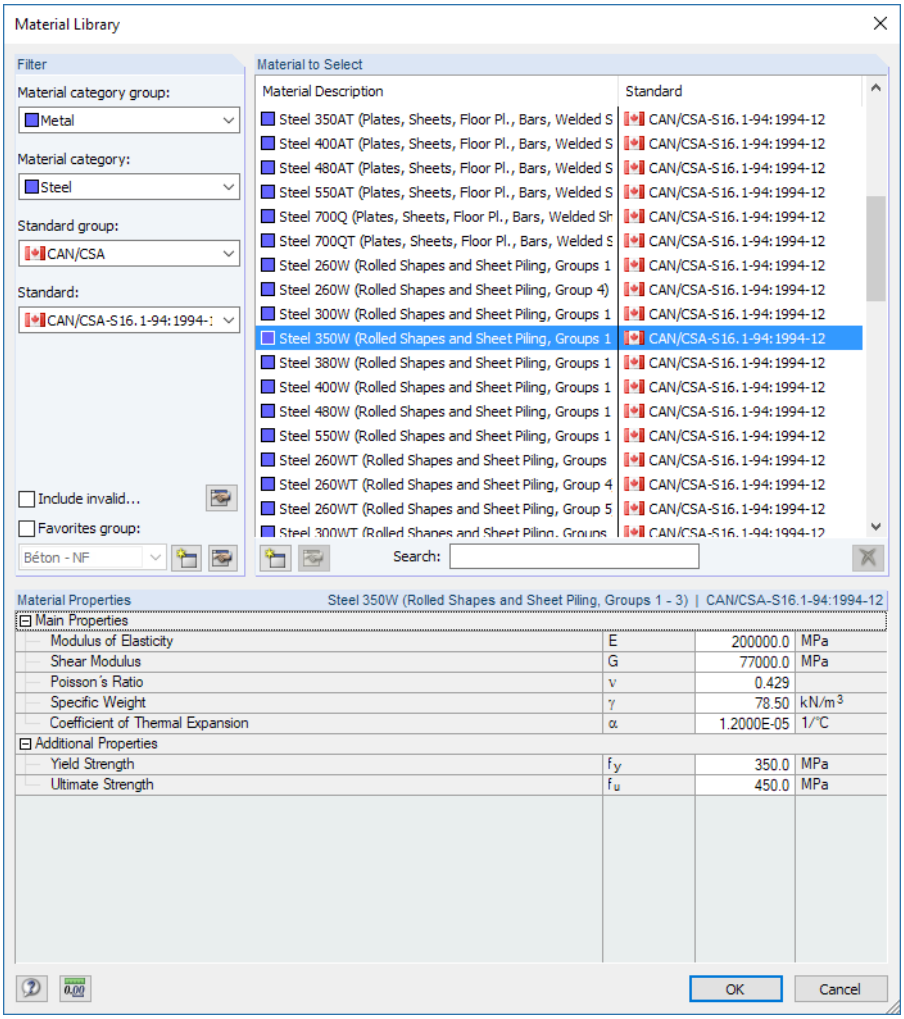
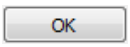


Figure 2.7: Dialog box *Material Library*

In the *Filter* section, *CAN/CSA-S16* is the default Standard. Select the material quality that you want to use for the design in the *Material to Select* list. You can check the corresponding properties in the dialog section below.



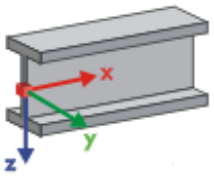
Click [OK] or press [↵] to transfer the selected material to Window 1.2 of RF-/STEEL CSA.

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

In the library, you can also select materials of categories *Cast Iron* and *Stainless Steel*. Please check, however, whether those materials are covered by the design concept of the Standard [1].

2.3 Cross-Sections

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



Coordinate System

The sectional coordinate system **yz** of RF-/STEEL CSA corresponds to the one of RFEM or RSTAB: The **y**-axis represents the strong principal axis of the cross-section, the **z**-axis is its weak axis. This coordinate system is used for both the input data and the results.

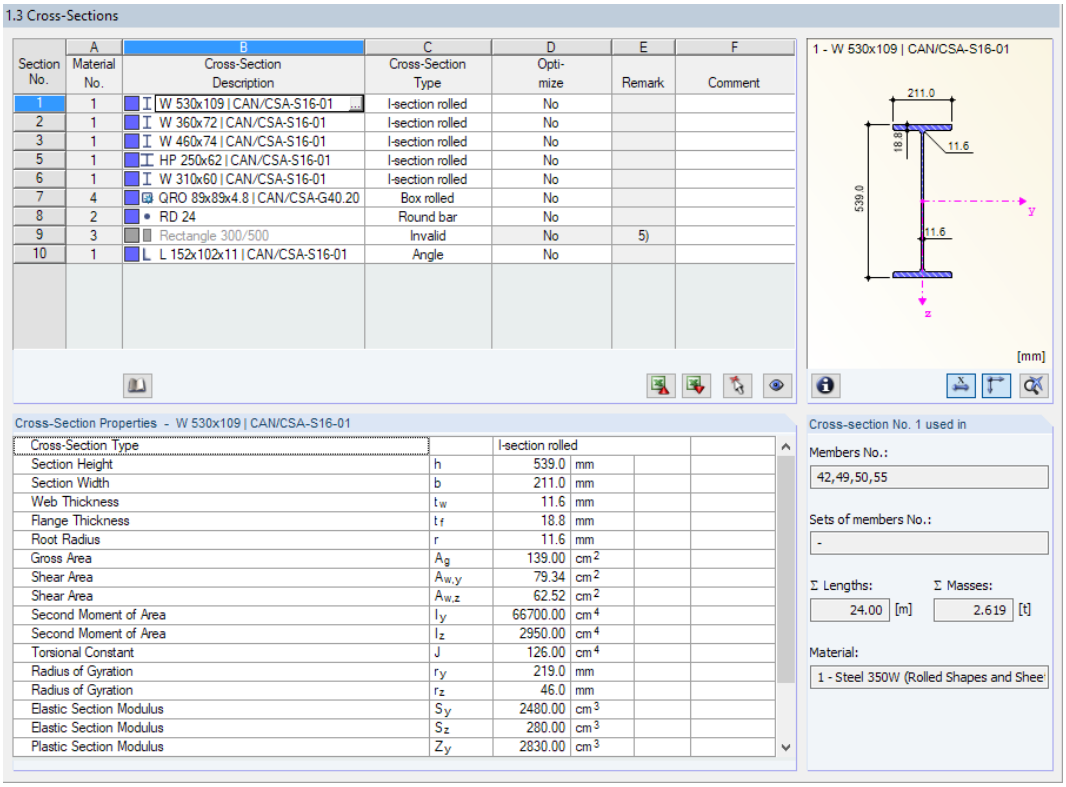
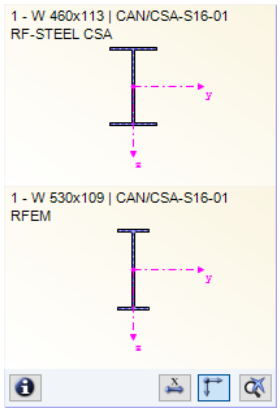


Figure 2.8: Window *1.3 Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM or RSTAB are preset with their material numbers.

If you want to modify a cross-section, select the entry in column B. Click the button or in the box, or press the function key [F7] to open the cross-section table of the current cross-section type (see Figure 2.9).



In this dialog box, you can select a different cross-section. To select a different section category, click to access the global library of cross-sections.

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

You can directly enter the new cross-section description in the text box. If the entry is listed in the database, RF-/STEEL CSA imports the cross-section parameters.

A modified cross-section will be highlighted in blue.

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

© Dlubal Software 2017

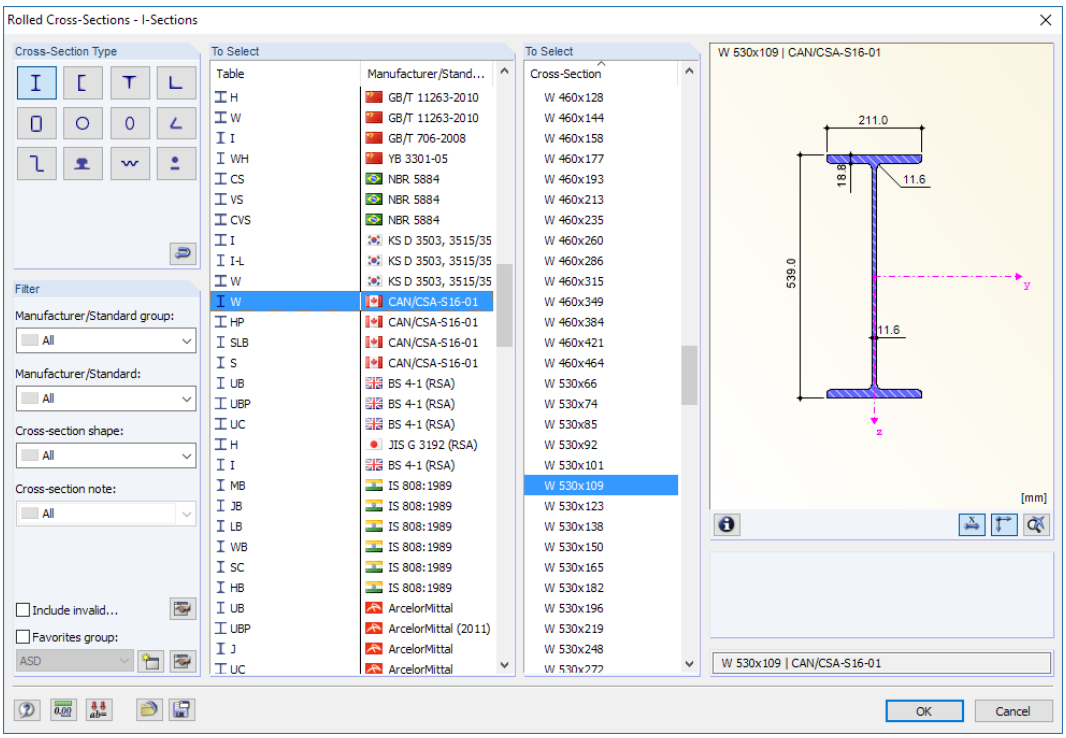


Figure 2.9: Rolled I-sections in cross-section library

Cross-Section Type
I-section rolled
I-section rolled
I-section rolled
I-section rolled
I-section rolled
Box rolled
Round bar
Invalid
Angle

Cross-Section Type

The program displays the type of cross-section that will be used for the classification according to [1] Clause 11.

Max. Design Ratio

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

Optimize

Details...

It is possible to optimize every cross-section from the library. The program searches the cross-section that comes as close as possible to a user-defined maximum utilization ratio. You can specify this maximum ratio in the *Details* dialog box (see Figure 3.1, page 23).

To optimize a cross-section, open the drop-down list in column D (resp. E) and select *From current row*. Recommendations on the optimization can be found in Chapter 7.2 on page 47.

Remark

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

Member with Tapered Cross-Section

For tapered members with different cross-sections at the member start and member ends, the module displays both section numbers in two rows, according to the definition in RFEM or RSTAB.

RF-/STEEL CSA also designs tapered members, provided that the section at the member start has the same number of stress points as the section at the member end. If the two sections have different numbers of stress points, the intermediate values cannot be interpolated. In this case, the calculation is neither possible in RFEM/RSTAB nor in RF-/STEEL CSA.

The stress points including their numbering can be checked graphically: Select the cross-section in Window 1.3 and click the [Info] button. The dialog box shown in Figure 2.10 appears.

Info About Cross-Section

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

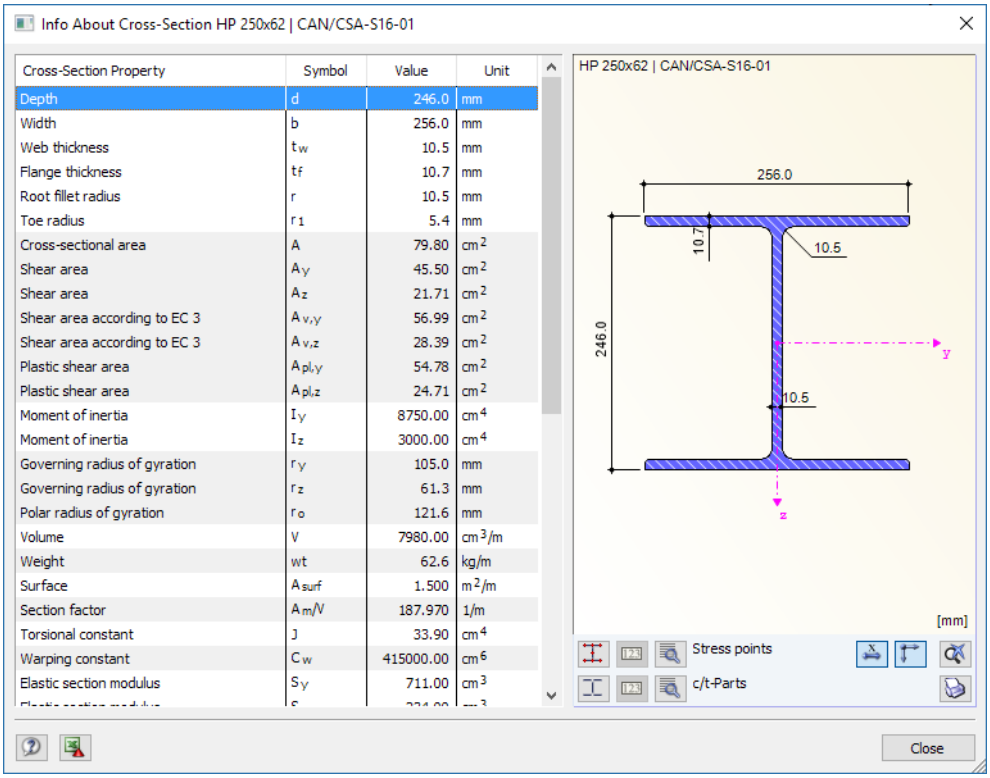


Figure 2.10: Dialog box *Info About Cross-Section*

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

Click the buttons find detailed information on the *Stress points* (centroidal distances, statical moments of area, warping constants etc.) or *c/t-Parts*, respectively.

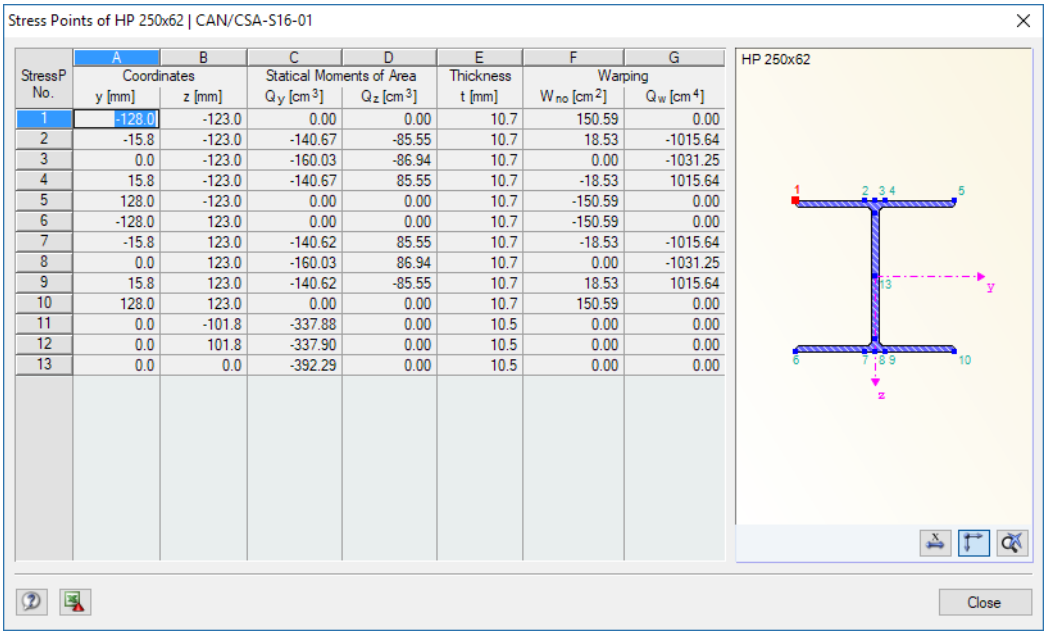


Figure 2.11: Dialog box *Stress Points of HP 250x62*

2.4 Intermediate Lateral Restraints

In Window 1.4, you can define lateral restraints for members. In RF-/STEEL CSA, this kind of support acts perpendicular to the z-axis of the cross-section (the minor axis, see Figure 2.10). Thus, you can manipulate the effective lengths of the members for the stability design concerning flexural buckling and lateral-torsional buckling.

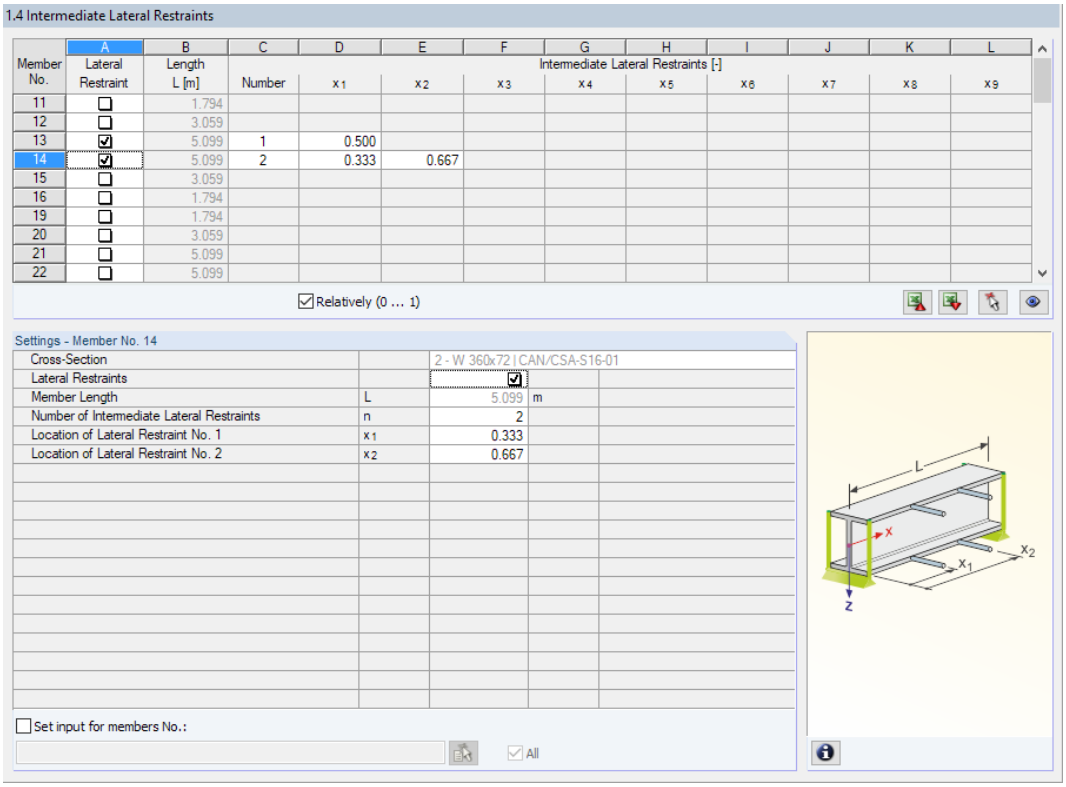


Figure 2.12: Window *1.4 Intermediate Lateral Restraints*

In the upper table, you can assign up to nine lateral restraints to each member. The *Settings* section below shows a column overview for the member selected above.

To define the restraints of a specific member, select the *Lateral Restraints* check box in column A. Then the other columns will be accessible where you can enter the parameters. To graphically select the member, click .

In column C, you can specify the *Number* of the intermediate restraints. Depending on the specification, one or more of the following *Intermediate Lateral Restraints* columns will be available for the definition of the x-locations.

Relatively (0 ... 1)

When the *Relatively (0 ... 1)* check box is activated, you can define the support points by their relative spacings. The positions of the intermediate restraints result from the member length and the relative distances from the member start. When you clear the *Relatively (0 ... 1)* check box, you can define the absolute distances.

2.5 Effective Lengths - Members

The window consists of two parts. The upper table presents a summary of all length factors of buckling, torsional buckling, and lateral-torsional buckling as well as the respective member lengths. The effective lengths defined in RFEM or RSTAB are preset. In the *Settings* section, additional information about the member selected in the upper table is given.

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

Click to select a member graphically and show its row.

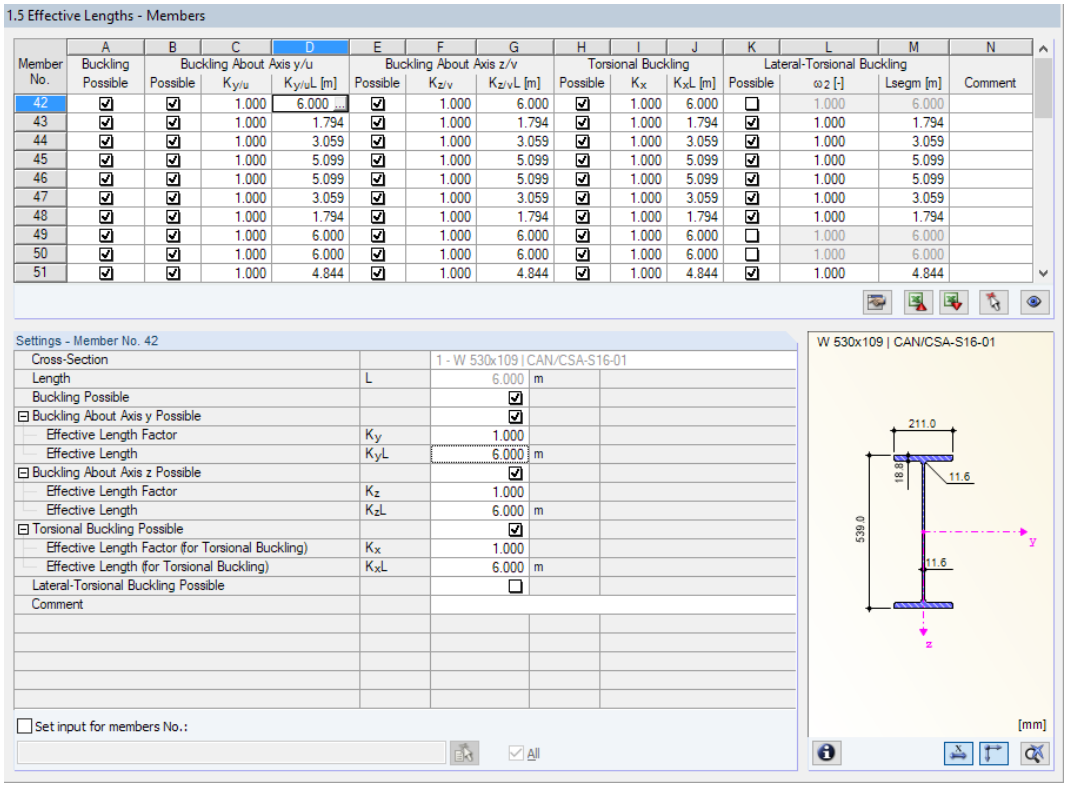


Figure 2.13: Window *1.5 Effective lengths - Members*

The effective lengths for buckling about the minor z-axis and torsional as well as lateral-torsional buckling are aligned automatically with the settings of Window *1.4 Intermediate Lateral Restraints* (see Chapter 2.4). If intermediate lateral restraints divide the member into segments of different lengths, no values are displayed in the table columns G, J, and L.

You can enter the effective lengths manually in the table and in the *Settings* tree, or define them graphically in the work window by clicking the ... button. The button is active when you place the cursor in the text box (see Figure 2.13).

The *Settings* tree includes the following parameters:

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

© Dlubal Software 2017

The table controls for which members an analysis of buckling, torsional or lateral-torsional buckling is to be performed. In addition, the *Effective Length Factor* can be adjusted for the respective designs. If you modify the factor, the equivalent member length is adjusted automatically, and vice versa.

You can also define the effective length of a member in a separate dialog box. To open it, click the button below the upper table.

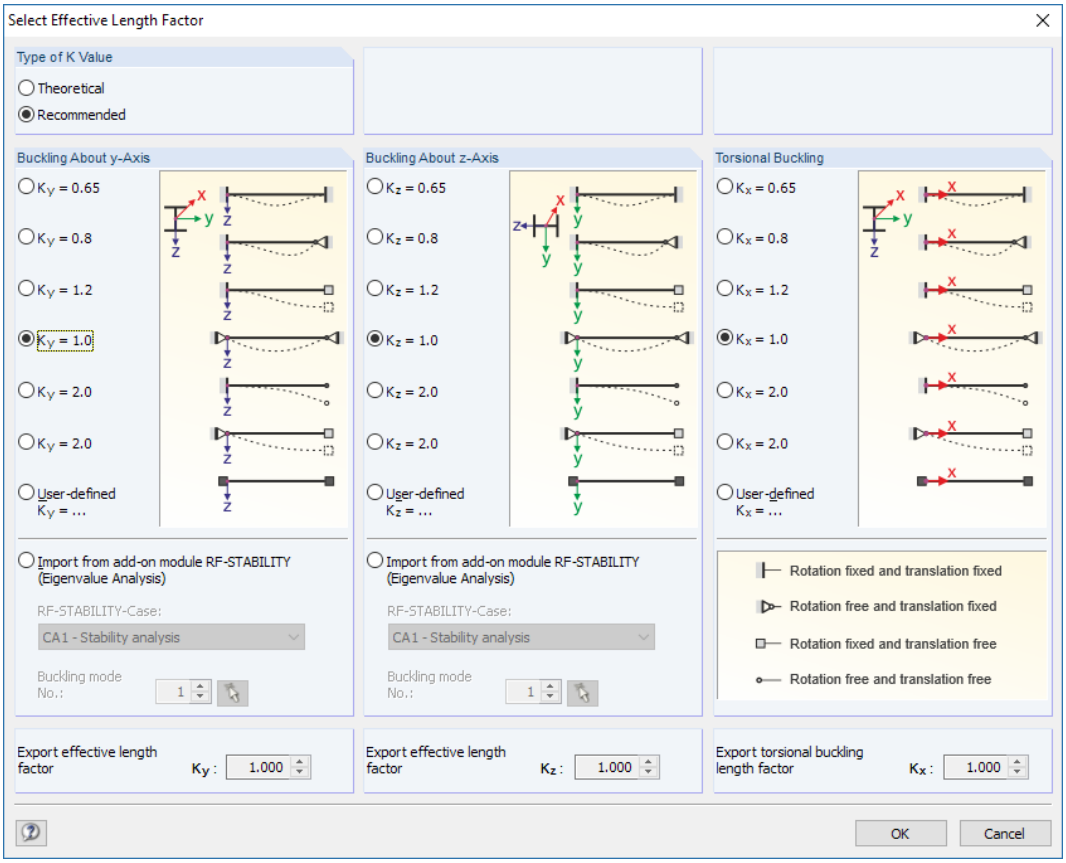


Figure 2.14: Dialog box *Select Effective Length Factor*

In this dialog box, the *Theoretical* or *Recommended* values of the factor, *K*, can be defined that are to be assigned to the selected member. The recommended values are described in [1] Annex F. For each direction, you can also select one of the theoretical Euler buckling modes or apply a *User-defined* factor.

If an eigenvalue analysis has been performed in the RF-STABILITY or RSBUCK add-on module, you can import the *Buckling mode* in order to determine the relevant factor.

Buckling Possible

The stability analysis for flexural and lateral-torsional buckling requires compressive forces to be included. Members for which this is not possible due to their member types (tension members, elastic foundations, rigid couplings) are disabled by default. The corresponding rows are dimmed, and a note appears in the *Comment* column.

The *Buckling possible* check boxes in table row A and in the *Settings* tree allow you to control the stability analyses: They determine whether the analyses for a member are to be performed or not.

Buckling About Axis y / Buckling about Axis z

The *Possible* columns control whether there is a buckling risk about the y-axis and/or z-axis. Those axes represent the local member axes, where the *y*-axis is the strong and the *z*-axis is the weak member axis. You can freely define the effective length factors, K_y, and K_z, for buckling about the strong or the weak axis.

You can check the position of the member axes in the cross-section graphic in the *1.3 Cross-Sections* Window (see Figure 2.8, page 10). To access the RFEM or RSTAB work window, click the [View Mode] button. There you can display the local member axes by using the shortcut menu of the member or the *Display* navigator.

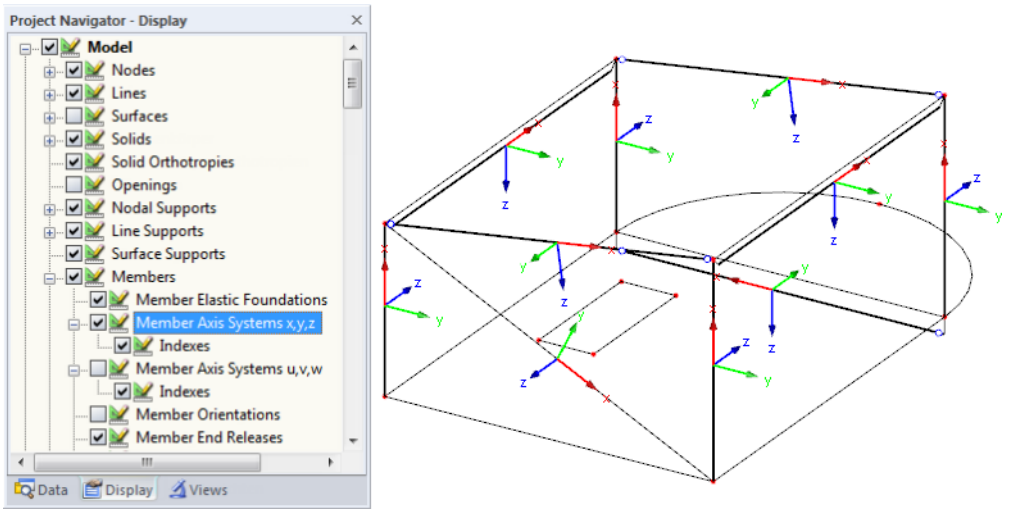


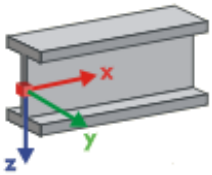
Figure 2.15: Displaying member axes in *Display* navigator of RFEM

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

When specifying the effective length factor *K*, the program determines the effective length *KL* by multiplying the member length *L* by the effective length factor. The *K* and *KL* boxes are interactive.

To define the effective lengths graphically in the work window of RFEM or RSTAB, click ... . This button becomes available when you place the cursor in a *KL* box (see Figure 2.13).

Torsional Buckling



Column H controls whether a torsional buckling design is to be performed. The effective length factors, K_x, and the torsional buckling lengths, K_xL_x, can be defined in columns I and J. The *x*-axis represents the center line of a member.

Lateral-Torsional Buckling

Column K controls whether a lateral-torsional buckling analysis is to be carried out.

Lateral-Torsional Buckling		
Possible	ω_2 [-]	Lsegm [m]
☑	1.000	1.794
☑	1.0	3.059
☑	acc. to 13.6	5.099
☑	manually	5.099

In column L, there is the possibility to choose among three options of the moment coefficient, ω_2, as seen to the left. The default value is 1.0. The coefficient can also be defined manually or determined by the program according to [1] Clause 13.6.

The lateral-torsional buckling lengths, L_{segm}, are shown in column M. To define the relevant lengths graphically in the work window of RFEM or RSTAB, click ... .

Comment

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

Set Input for Members No.

Below the *Settings* table, you find the *Set input for members No.* check box. If you select this check box, the subsequent settings will be applied to the selected members or *All* members (you can enter the member numbers manually or select them graphically with the button). This option is useful when you want to assign identical boundary conditions to several members (see https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000726).

With this function, you cannot change the settings you have already made.

2.6 Effective Lengths - Sets of Members

This window controls the effective lengths for sets of members. It is only displayed when you have selected one or more sets of members for design in the *1.1 General Data* Window (see Figure 2.2, page 6).

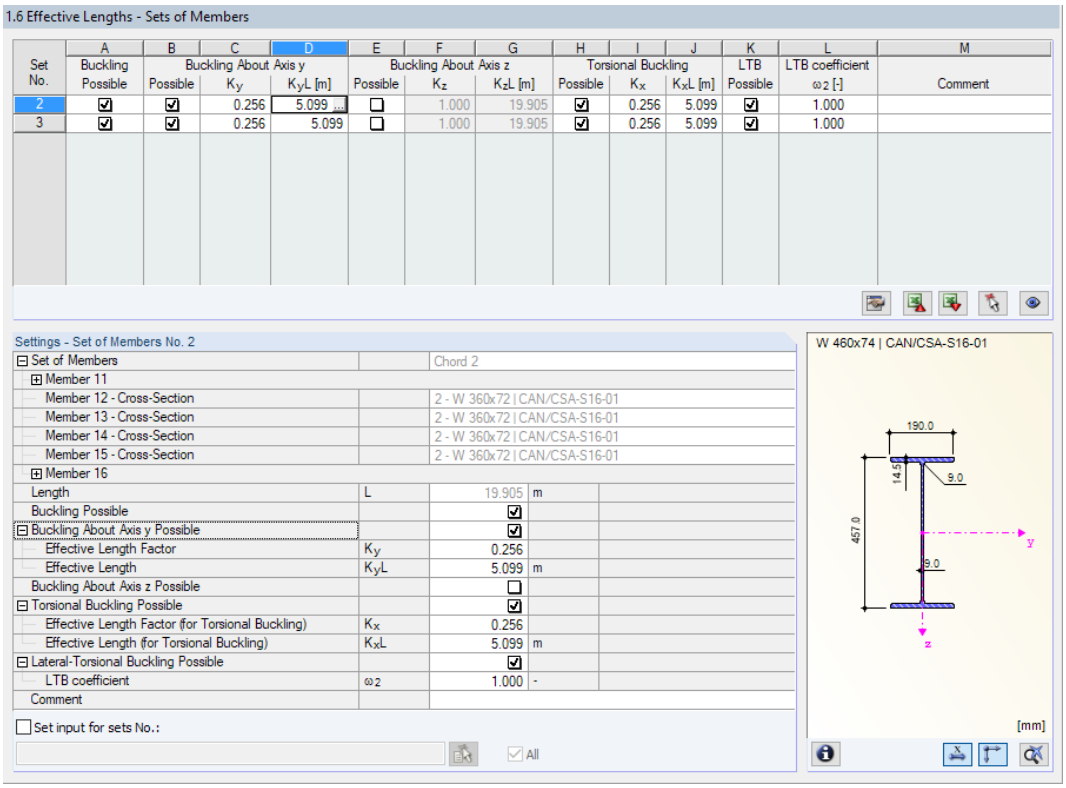


Figure 2.16: Window *1.6 Effective Lengths - Sets of Members*

The concept of the window is very similar to the previous *1.6 Effective Lengths - Members* Window. Here you can define the buckling lengths for sets of members, as described in Chapter 2.5.

There are differences, however, with respect to the parameters for torsional and lateral-torsional buckling. Those are to be defined by means of boundary conditions in Windows *1.7 Nodal Supports* (see Chapter 2.7) and *1.8 Member Hinges* (see Chapter 2.8).

2.7 Nodal Supports - Sets of Members

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

The stability design of each set of members is based on the respective loads and the boundary conditions. They are essential for the determination of the critical elastic moment, M_u, which is required for the design of the entire set of members.

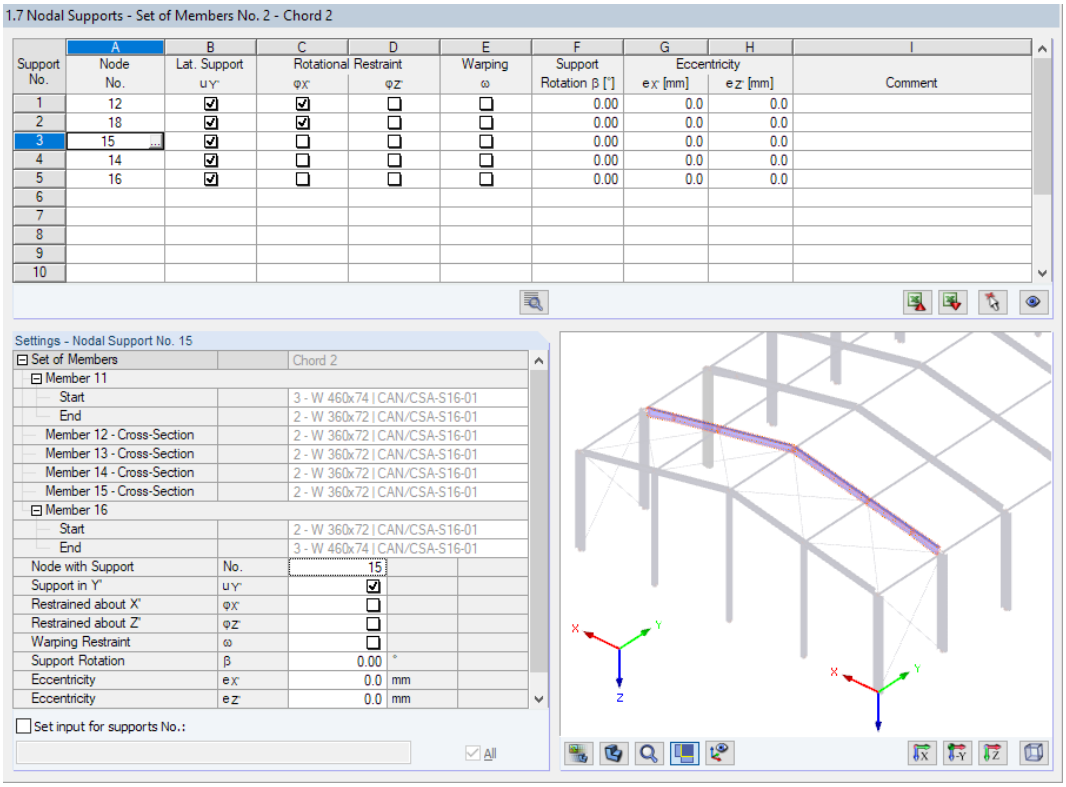


Figure 2.17: Window *1.7 Nodal Supports - Set of Members*

The current table manages the boundary conditions of the set of members that is selected in the navigator on the left!

The supports defined in RFEM or RSTAB (for example, in *Z* for a continuous beam) are not relevant in this table: The distributions of moments and shear forces for the determination of the critical elastic moment are automatically imported from RFEM/RSTAB. Here, you define the support conditions with respect to the failure modes of stability (buckling and lateral-torsional buckling).

Supports on the start and end nodes of each set of members are preset. Any other supports, for example due to connected members, must be added manually. Use the button to select nodes graphically in the RFEM/RSTAB work window.

According to [1] Clause 13.6, it is possible to design singly symmetric Class 1, 2, or 3 I-sections and T-sections that are subjected to uniaxial strong axis bending. In order to determine the critical elastic moment of the entire set of members, the program creates a planar framework model with four degrees of freedom for each node.

For the nodal supports, the orientation of axes within a set of members is important. The program checks the position of the nodes and internally defines the axes of the nodal supports for Window 1.7 according to Figure 2.18 through Figure 2.21. The [Local Coordinate System] button below the model graphic can help you with the orientation: Use it to display the set of members exclusively, with the axes being clearly visible (see also Knowledge Base article on our website).

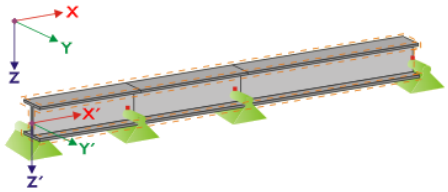


Figure 2.18: Auxiliary coordinate system for nodal supports – straight set of members

If all members of a set of members lie on a straight line, as shown in Figure 2.18, the local coordinate system of the first member within this set of members is applied to entire set of members.

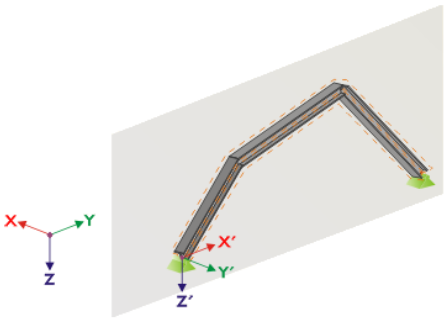


Figure 2.19: Auxiliary coordinate system for nodal supports – set of members in vertical plane

If the members do not lie on a straight line, they nevertheless must be located in one plane. We can see a vertical plane in Figure 2.19. In this case, the X′-axis is horizontal and oriented in the plane direction. The Y′-axis is horizontal as well, but perpendicular to the X′-axis. The Z′-axis points downwards.

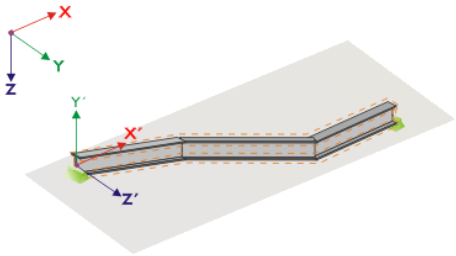


Figure 2.20: Auxiliary coordinate system for nodal supports – set of members in horizontal plane

If the members are located in a horizontal plane, the X′-axis is defined parallel to the X-axis of the global coordinate system. The Y′-axis then points in the opposite direction to the global Z-axis. The Z′-axis is parallel to the global Y-axis.

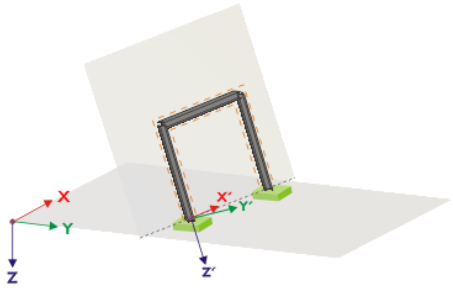


Figure 2.21: Auxiliary coordinate system for nodal supports – set of members in inclined plane

Figure 2.21 shows the general case: The members do not lie on a straight line, but are located in an inclined plane. The definition of the X′-axis results from the intersection line between the inclined and the horizontal plane. The Y′-axis is perpendicular to the X′-axis (and also perpendicular to the inclined plane). The Z′-axis is perpendicular to the X′- and Y′-axes.

© Dlubal Software 2017

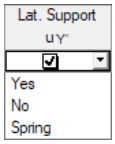
The buttons below the graphic have the following functions:

Button	Function
	Shows model or schematic sketch
	Shows members as 3D rendering or wire-frame model
	Shows current set of members or entire model
	Displays irrelevant members of model as transparent or opaque
	Shows set of members with local coordinate system or entire model
	Shows view in direction of X-axis
	Shows view in opposite direction of Y-axis
	Shows view in direction of Z-axis
	Shows isometric view

Table 2.2: Buttons for graphic window

If the set of members is laterally supported in specific locations, these nodes must be added manually in Window *1.7 Nodal Supports*. In this way, it is possible to consider, for example, the effect of a purlin which is connected in the spatial model of RFEM or RSTAB. If this support is neglected in the reduced set of members model, instabilities may occur.

The [...] button (see Figure 2.17) enables you to define the supported nodes graphically in the work window of RFEM or RSTAB.



In columns B to N of the table, you specify the support conditions of the selected nodes. To activate or deactivate the supports and restraints for the corresponding degrees of freedom, select the check boxes. Alternatively, you can enter the constants of a translational or rotational *Spring* manually.

The *Support rotation* and *Eccentricity* parameters allow for modeling support conditions close to reality.

With the [Edit warp stiffener] button (below the table) it is possible to make the program determine the constant of a warp spring.

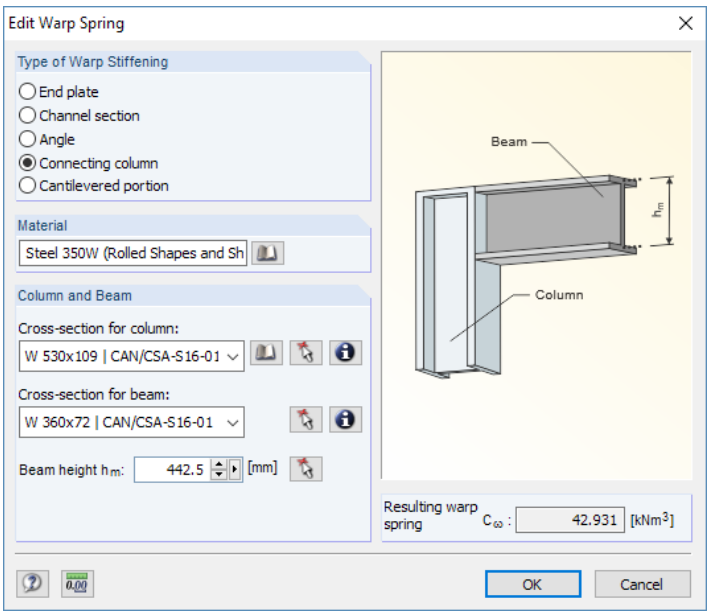


Figure 2.22: Dialog box *Edit Warp Spring*

The following warp stiffening types are available in the *Edit Warp Spring* dialog box:

- End plate
- Channel section
- Angle
- Connecting column
- Cantilevered portion

Materials and cross-sections can be selected via the lists and [Library] buttons. With the button, you can select them also graphically in the model of RFEM or RSTAB.

Based on the parameters, RF-/STEEL CSA determines the *Resulting warp spring*, C_ω, which can then be transferred to Window 1.7 with [OK].

2.8 Member Hinges - Sets of Members

This window is displayed if at least one set of members has been selected in the *1.1 General Data* Window. Here, you can define hinges for members within the set of members that, for structural reasons, do not transfer the degrees of freedom locked in Window 1.7 as internal forces. Make sure that no double hinges are generated in coaction with Window 1.7!

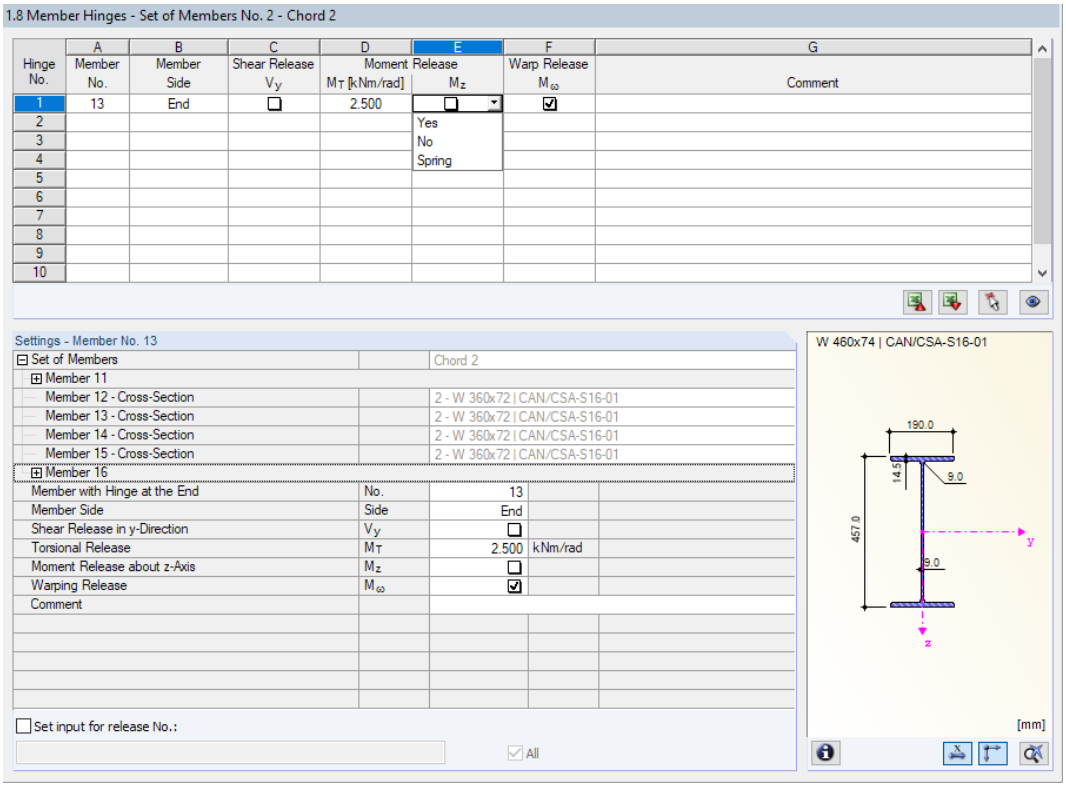
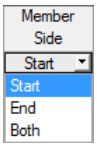


Figure 2.23: Window *1.8 Member Hinges*

The current table manages the hinges of the set of members that is selected in the navigator on the left.



In column B, you specify the *Member Side* where the hinge is located, or if there are hinges on both member sides.

In columns C to F, you can define the releases or spring constants in order to adjust the set of members model to the support conditions defined in Window 1.7.

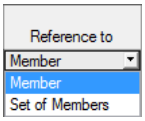
2.9 Serviceability Data

The last input window controls the settings for the serviceability limit state design of specific objects. It is available when you have selected one or more load cases or combinations in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 7).

1.9 Serviceability Data

No.	A Reference to	B Member No.	C Reference Length Manually	D Reference Length L [m]	E Direction	F Precamber $w_{c,v}$ [mm]	G Beam Type	H Comment
1	Set of Members	2	☐	12.548	y, z	0.0	Beam	
2	Set of Members	5	☐	7.094	y, z	0.0	Beam	
3	Member	82	☐	7.094	y, z	0.0	Beam	
4	Member	81	☑	4.546	y, z	0.0	Cantilever End Free	
5	Member	83	☑	4.546	y, z	0.0	Cantilever End Free	
6	Member	15	☐	6.274	y, z	0.0	Beam	
7	Member	16	☐	6.274	y/u, z/v	0.0	Beam	
8	Member	25	☐	6.274	y/u, z/v	0.0	Beam	
9	Member	26	☐	6.274	y/u, z/v	0.0	Beam	
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

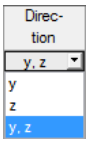
Figure 2.24: Window *1.9 Serviceability Data*



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

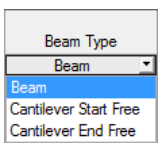
For a list or set of members, the orientation and rotation of all contained members must be identical. This will guarantee that the components of the deformation are taken into account correctly.

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

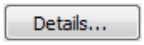


Column E controls the governing *Direction* for the deformation analysis. You can select the directions of the local member axes *y* and *z* (or *u* and *v* for unsymmetrical cross-sections).

You can consider a *Precamber*, w_c, in column F, if applicable. The reference to the axes is controlled by the specification in the *Details* dialog box (see Figure 3.1, page 23).



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



The *Details* dialog box controls whether the deformations are related to the undeformed system or the shifted ends of the members or sets of members (see Figure 3.1, page 23).

© Dlubal Software 2017

3 Calculation

3.1 Detail Settings

Details...

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

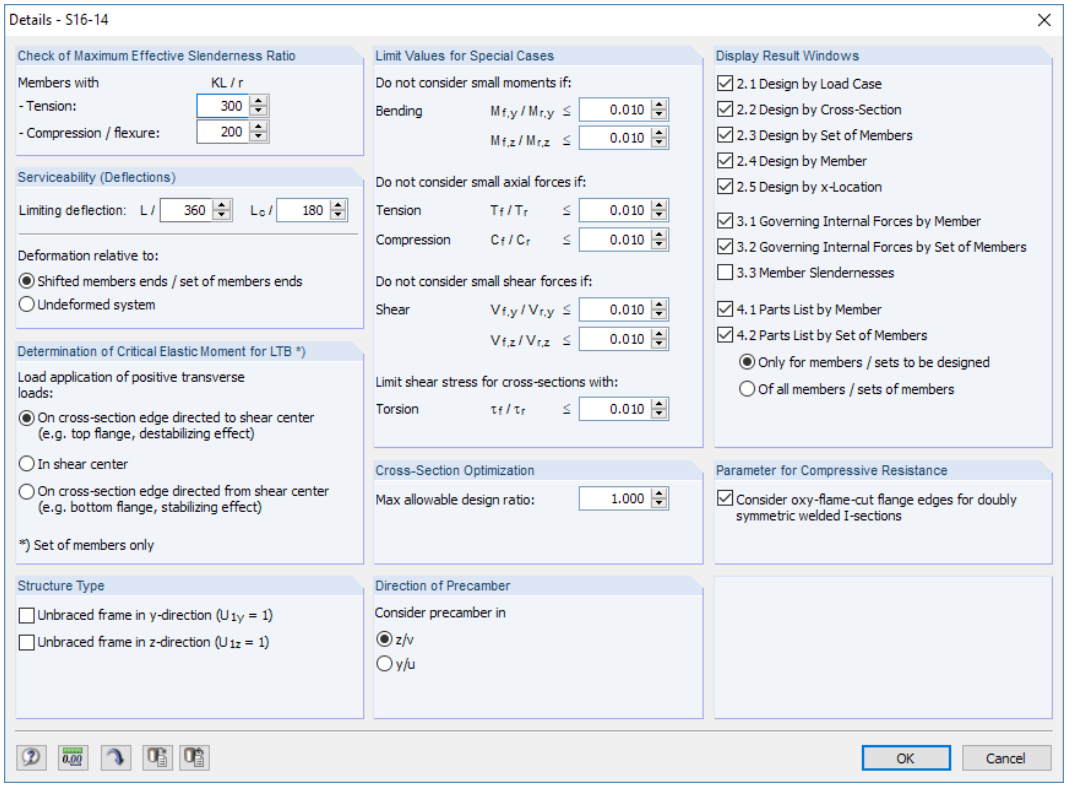


Figure 3.1: Dialog box *Details*

Check of Maximum Effective Slenderness Ratio

According to [1] Clause 10.4.2.2, the slenderness ratio, *KL / r*, preferably should not exceed 300 for members in tension. For members in compression or flexure, the slenderness ratio should not exceed 200 (cf. [1] Clause 10.4.2.1). If required, the limit ratios can be adjusted.

The limit ratios are compared to the real member slendernesses in Window 3.3. That window is available after the calculation (see Chapter 4.8, page 33) when the corresponding option has been checked in the *Display Result Windows* section of the *Details* dialog box.

Serviceability (Deflections)

For the SLS design, the limiting deflections can be separately defined for beams (default: L/360) and cantilevers (default: L_c/180). Annex D of the Standard [1] gives recommendations on the maximum values for deflections.

The options below specify whether the deformations are related to the shifted member ends or set of members ends (line between start and end nodes of deformed model) or to the undeformed original system. The difference is illustrated by an example in the Knowledge Base on our website: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001081

© Dlubal Software 2017

Determination of Critical Elastic Moment for LTB

The critical elastic moment, M_u, is calculated automatically for sets of members.

If there are transverse loads, it is important to define the location where those forces are acting: Depending on the location of *Load application*, they can have stabilizing or destabilizing effects due to their eccentricities. Thus, they have a major impact on the critical elastic moment. This setting is globally applied to all loads and all sets of members of the design case.

The signs of the eccentricities are related to the shear center, *M*, of each cross-section. The following article on our website provides more information about the sign convention for transverse loads: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000880

Structure Type

According to [1] Clause 13.8, a distinction is made between braced and unbraced frames. Both check boxes are cleared by default. If one or both check boxes are selected, the program assumes the factor U_1 to be 1.0 for unbraced frames.

Please note that the settings are globally applied to all members and sets of members. If there are both types of frames in the model, the objects can be designed in separate design cases with specific settings (see Chapter 7.1, page 45).

Limit Values for Special Cases

For a simplified design, it is possible to neglect small bending moments, axial forces, or shear forces, as well as shear stresses due to torsion. The limit ratios of the moments, forces, or stresses can be defined separately in this dialog box section.

The preset limits are not part of the Standard [1]. Changing the values is in the responsibility of the user.

Cross-Section Optimization

By default, the optimization is targeted on the maximum design ratio of 100%. If required, you can change this limit value.

Direction of Precamber

When you specify two directions in column E of the *1.9 Serviceability Data* Window and apply a precamber in column F, you can determine for which direction the precamber is to be considered.

Display Result Windows

In this dialog section, you can select which result windows including parts list are to be displayed in the output. Those windows are described in Chapter 4.

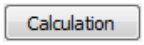
Window *3.3 Member Slendernesses* is deactivated by default.

Parameter for Compressive Resistance

This check box enables you to define the type of fabrication for doubly symmetric cross-sections. It has an effect on the parameter, *n*, which controls the value of the compressive resistance, C_r (see [1] Clause 13.3.1).

© Dlubal Software 2017

3.2 Starting Calculation



In all input windows of RF-/STEEL CSA, you can start the design via the [Calculation] button.

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

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

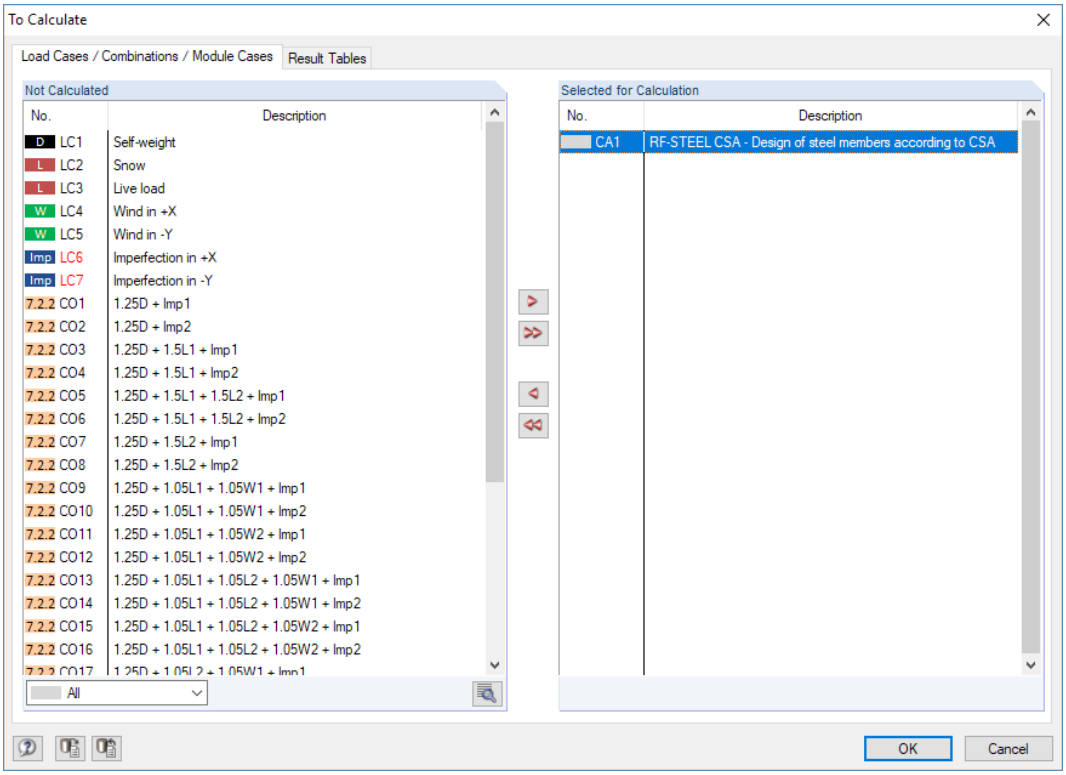
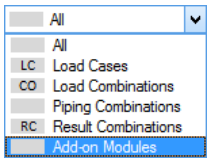


Figure 3.2: Dialog box *To Calculate*



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

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

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

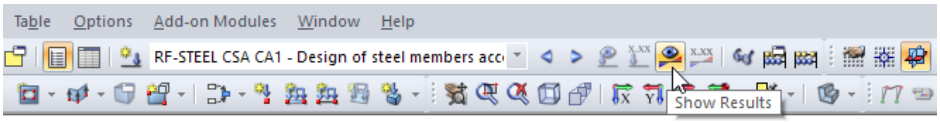


Figure 3.3: Direct calculation of an RF-STEEL CSA design case in RFEM

Subsequently, you can follow the calculation in a separate dialog box.

© Dlubal Software 2017

4 Results

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

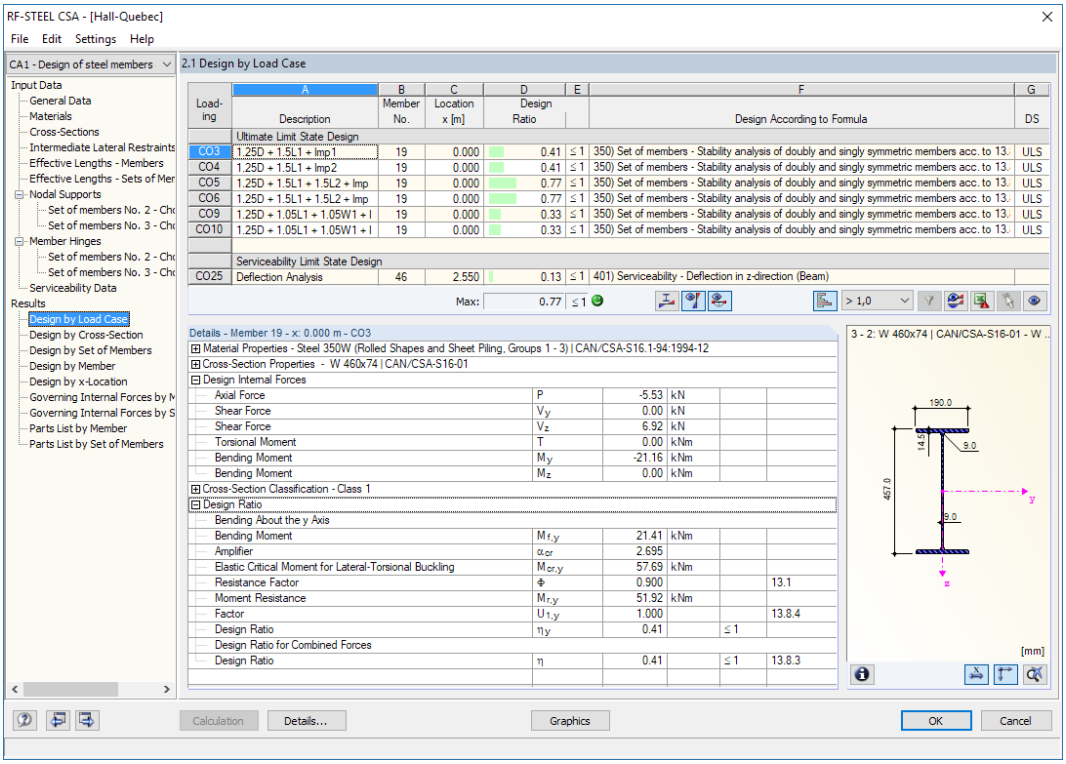
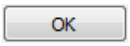


Figure 4.1: Result window with design results and details

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

Windows 3.1 and 3.2 list the governing internal forces, Window 3.3 gives information on the member slendernesses. The last two Windows 4.1 and 4.2 show the parts lists of members and sets 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.



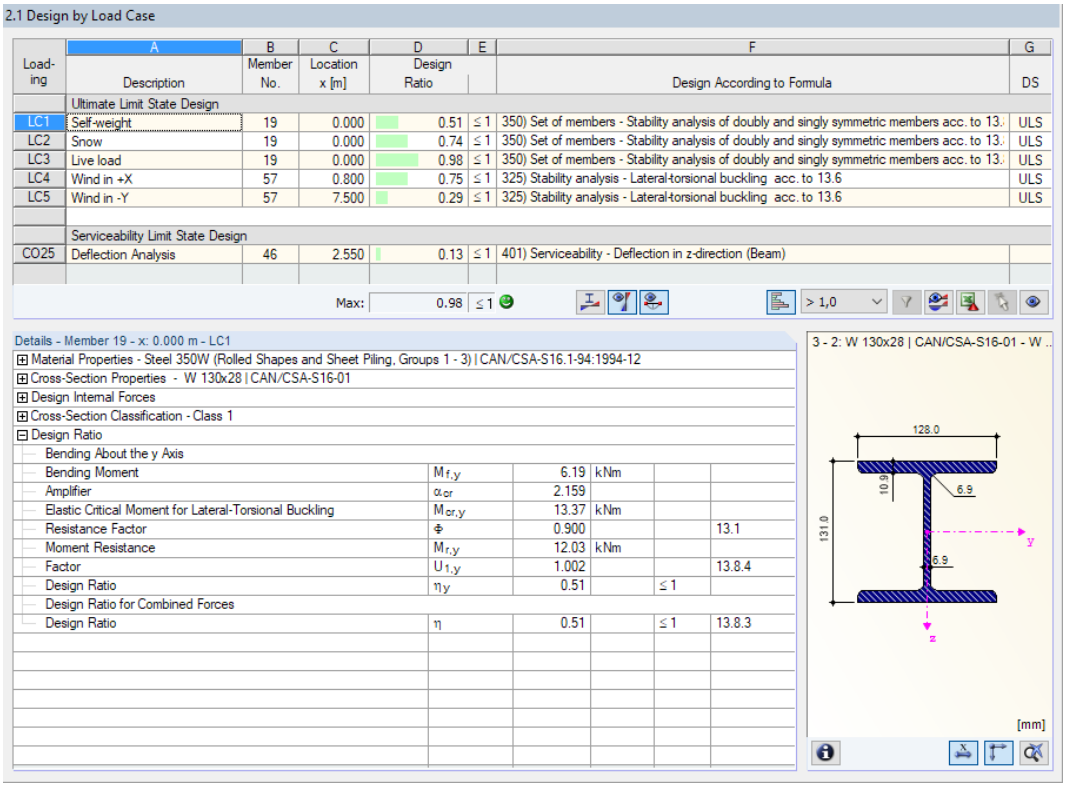
[OK] saves the results. RF-/STEEL CSA is closed and you return to the main program.

Chapter 4 describes the different result windows one by one. The evaluation and checking of the results is described in Chapter 5 starting on page 36.

4.1 Design by Load Case

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

The *Details* section below includes specific information on the cross-section properties, internal forces, and design parameters for the load case or combination selected in the upper table.



2.1 Design by Load Case

Load-ing	A Description	B Member No.	C Location x [m]	D Design Ratio	E	F Design According to Formula	G DS
	Ultimate Limit State Design						
LC1	Self-weight	19	0.000	0.51	≤ 1	350) Set of members - Stability analysis of doubly and singly symmetric members acc. to 13.	ULS
LC2	Snow	19	0.000	0.74	≤ 1	350) Set of members - Stability analysis of doubly and singly symmetric members acc. to 13.	ULS
LC3	Live load	19	0.000	0.98	≤ 1	350) Set of members - Stability analysis of doubly and singly symmetric members acc. to 13.	ULS
LC4	Wind in +X	57	0.800	0.75	≤ 1	325) Stability analysis - Lateral-torsional buckling acc. to 13.6	ULS
LC5	Wind in -Y	57	7.500	0.29	≤ 1	325) Stability analysis - Lateral-torsional buckling acc. to 13.6	ULS
	Serviceability Limit State Design						
CO25	Deflection Analysis	46	2.550	0.13	≤ 1	401) Serviceability - Deflection in z-direction (Beam)	
			Max:	0.98	≤ 1		

Details - Member 19 - x: 0.000 m - LC1

⊞ Material Properties - Steel 350W (Rolled Shapes and Sheet Piling, Groups 1 - 3) | CAN/CSA-S16.1-94:1994-12					
⊞ Cross-Section Properties - W 130x28 | CAN/CSA-S16-01					
⊞ Design Internal Forces					
⊞ Cross-Section Classification - Class 1					
⊟ Design Ratio					
Bending About the y Axis					
Bending Moment	$M_{f,y}$	6.19	kNm		
Amplifier	α_{cr}	2.159			
Elastic Critical Moment for Lateral-Torsional Buckling	$M_{cr,y}$	13.37	kNm		
Resistance Factor	Φ	0.900			13.1
Moment Resistance	$M_{r,y}$	12.03	kNm		
Factor	$U_{1,y}$	1.002			13.8.4
Design Ratio	η_y	0.51		≤ 1	
Design Ratio for Combined Forces					
Design Ratio	η	0.51		≤ 1	13.8.3

Figure 4.2: Window *2.1 Design by Load Case*

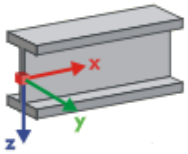
Description

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

Member No.

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

Location x



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

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

Design Ratio

Max: 0.98 ≤ 1

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

The lengths of colored bars represent the respective design ratios.

Design According to Formula

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

DS

The last column provides information on the respective design situations.

4.2 Design by Cross-Section

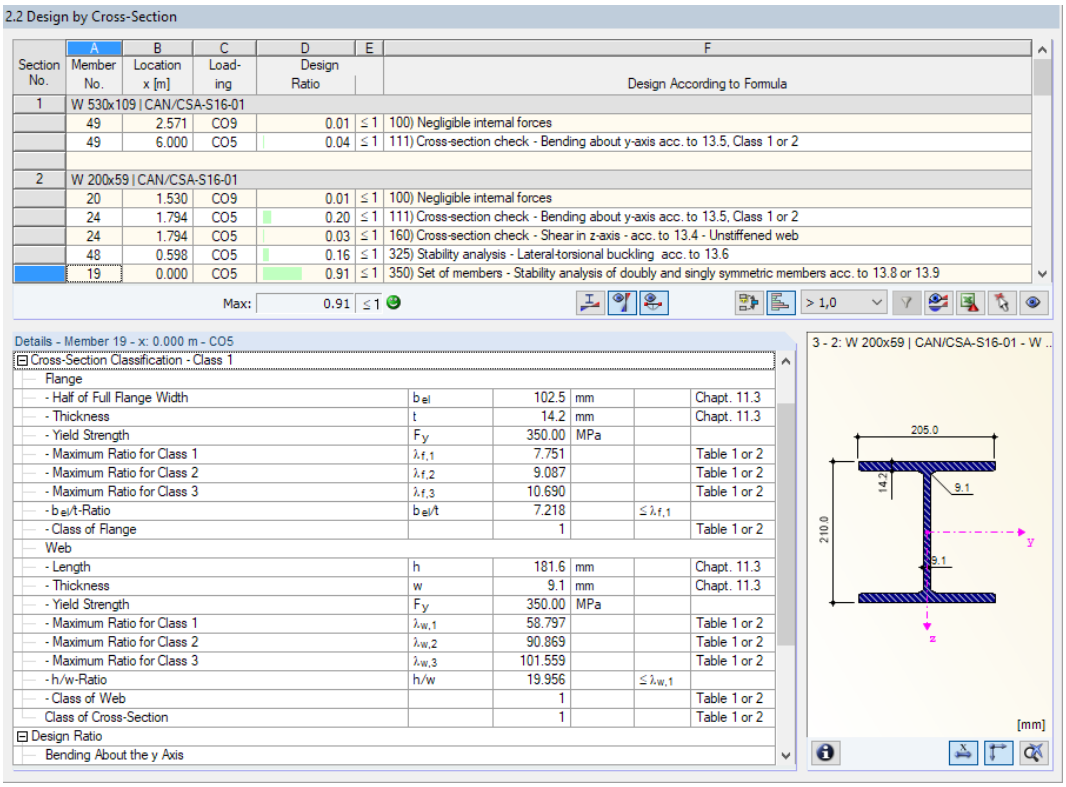


Figure 4.3: Window *2.2 Design by Cross-Section*

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

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

4.3 Design by Set of Members

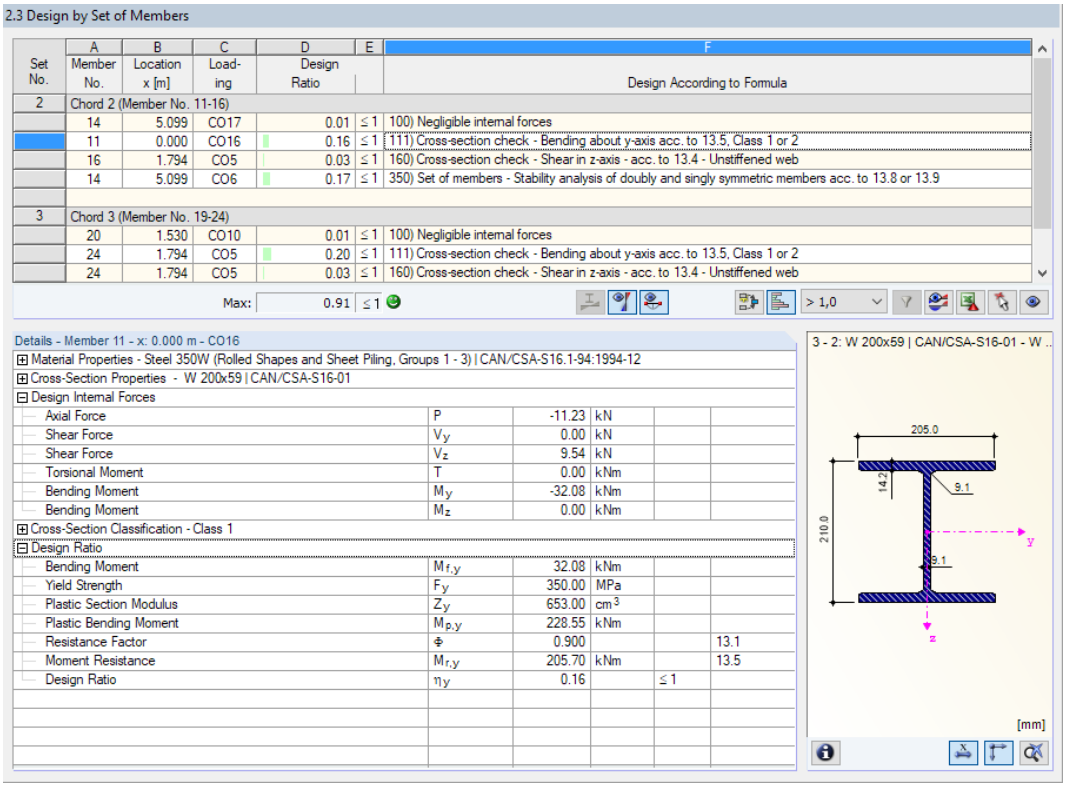


Figure 4.4: Window *2.3 Design by Set of Members*

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

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

The output by set of members clearly presents the design for an entire structural group, e.g. a frame.

© Dlubal Software 2017

4.4 Design by Member

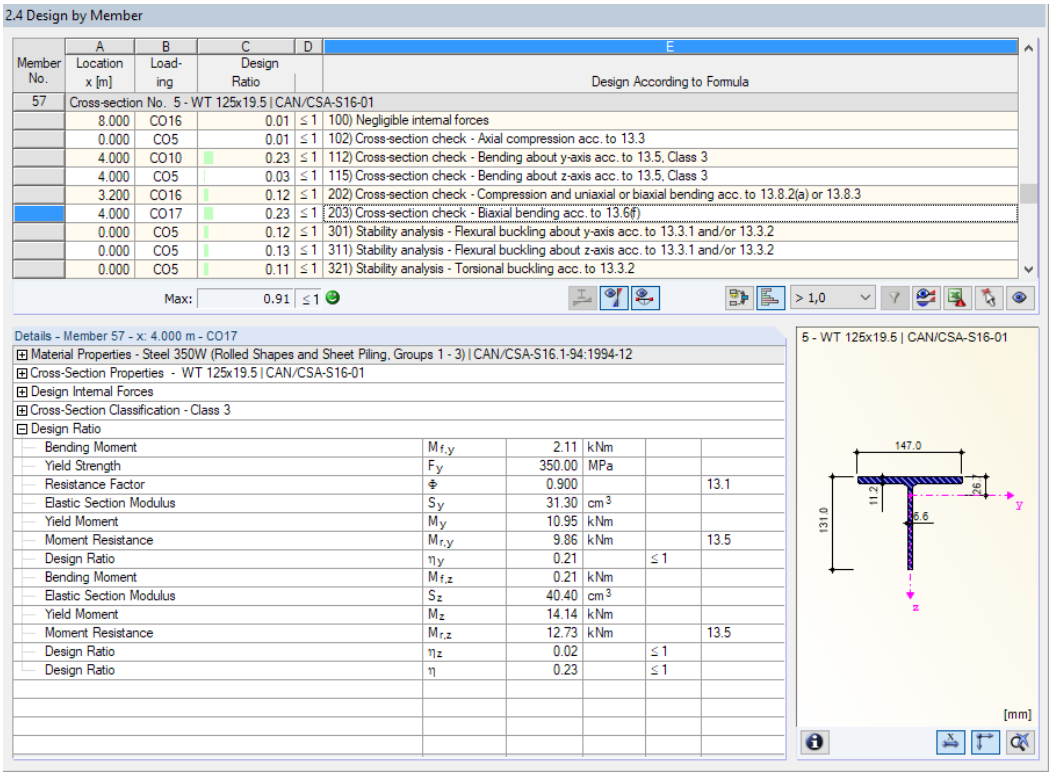


Figure 4.5: Window *2.4 Design by Member*

This result window lists the maximum ratios of the individual designs for each member. The columns are described Chapter 4.1 on page 27.

4.5 Design by x-Location

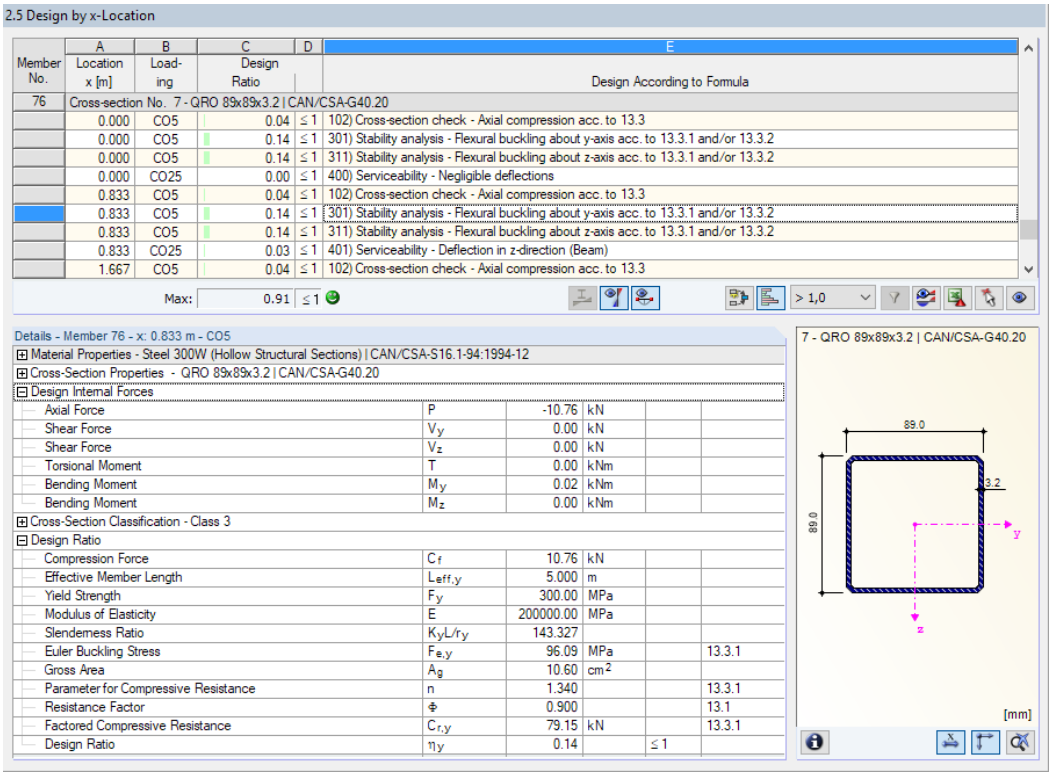


Figure 4.6: Window *2.5 Design by x-Location*

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

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

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Loading	C Forces [kN] P	D Forces [kN] $V_{y/u}$	E Forces [kN] $V_{z/v}$	F Moments [kNm] T	G Moments [kNm] $M_{y/u}$	H Moments [kNm] $M_{z/v}$	I Design According to Formula
11	Cross-section No. 3 - W 200x59 | CAN/CSA-S16-01 ... 2 - W 200x59 | CAN/CSA-S16-01								
	0.000	CO16	-11.23	0.00	9.54	0.00	-32.08	0.00	111) Cross-section check - Bending about y-axis acc. to 13.5, (
	0.000	CO6	-15.12	-0.01	11.40	0.00	-31.39	0.00	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unsti
	0.000	CO6	-15.12	-0.01	11.40	0.00	-31.39	0.00	350) Set of members - Stability analysis of doubly and singly sym
12	Cross-section No. 2 - W 200x59 | CAN/CSA-S16-01								
	1.530	CO17	-8.14	-0.01	3.23	0.00	0.52	0.02	100) Negligible internal forces
	0.000	CO16	-10.73	0.00	7.53	0.00	-17.01	0.01	111) Cross-section check - Bending about y-axis acc. to 13.5, (
	0.000	CO6	-14.52	-0.01	8.72	0.00	-13.66	0.02	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unsti
	0.000	CO6	-14.52	-0.01	8.72	0.00	-13.66	0.02	350) Set of members - Stability analysis of doubly and singly sym
13	Cross-section No. 2 - W 200x59 | CAN/CSA-S16-01								
	2.550	CO5	-19.12	0.01	0.14	0.00	11.39	0.01	111) Cross-section check - Bending about y-axis acc. to 13.5, (
	0.000	CO6	-19.54	0.01	4.03	0.00	5.96	0.05	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unsti
	0.000	CO6	-19.54	0.01	4.03	0.00	5.96	0.05	350) Set of members - Stability analysis of doubly and singly sym
14	Cross-section No. 2 - W 200x59 | CAN/CSA-S16-01								
	5.099	CO17	-11.10	-0.01	-2.26	0.00	2.02	0.03	100) Negligible internal forces
	2.550	CO6	-19.15	-0.01	-0.16	0.00	11.30	0.01	111) Cross-section check - Bending about y-axis acc. to 13.5, (
	5.099	CO5	-19.51	-0.01	-4.06	0.00	5.81	0.04	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unsti
	5.099	CO6	-19.54	-0.01	-4.03	0.00	5.97	0.05	350) Set of members - Stability analysis of doubly and singly sym
15	Cross-section No. 2 - W 200x59 | CAN/CSA-S16-01								
	0.000	CO17	-8.25	0.00	-2.34	0.00	2.02	0.03	100) Negligible internal forces
	3.059	CO5	-14.52	0.01	-8.76	0.00	-13.92	0.02	111) Cross-section check - Bending about y-axis acc. to 13.5, (
	3.059	CO5	-14.52	0.01	-8.76	0.00	-13.92	0.02	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unsti
	0.000	CO6	-14.06	0.01	-4.10	0.00	5.97	0.05	350) Set of members - Stability analysis of doubly and singly sym
16	Cross-section No. 2 - W 200x59 | CAN/CSA-S16-01 ... 3 - W 200x59 | CAN/CSA-S16-01								
	1.794	CO5	-15.13	0.01	-11.43	0.00	-31.71	0.00	111) Cross-section check - Bending about y-axis acc. to 13.5, (
	1.794	CO5	-15.13	0.01	-11.43	0.00	-31.71	0.00	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unsti
	0.000	CO6	-14.73	0.01	-8.36	0.00	-13.66	0.02	350) Set of members - Stability analysis of doubly and singly sym
19	Cross-section No. 3 - W 200x59 | CAN/CSA-S16-01 ... 2 - W 200x59 | CAN/CSA-S16-01								

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

For all designed members, the internal forces are listed that effectuate the maximum ratios of each type of design.

Location x

This column shows the x-locations where the maximum design ratios occur.

Loading

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

Forces / Moments

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

© Dlubal Software 2017

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [m]	B Loading	C Forces [kN] P	D Forces [kN] V_y	E Forces [kN] V_z	F Moments [kNm] T	G Moments [kNm] M_y	H Moments [kNm] M_z	I Design According to Formula
2	Chord 2 (Member No. 11-16)								
	5.099	CO17	-11.10	-0.01	-2.26	0.00	2.02	0.03	100) Negligible internal forces
	0.000	CO16	-11.23	0.00	9.54	0.00	-32.08	0.00	111) Cross-section check - Bending about y-axis acc. to 13.5, Clas
	1.794	CO5	-15.13	0.01	-11.43	0.00	-31.71	0.00	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffen
	5.099	CO6	-19.54	-0.01	-4.03	0.00	5.97	0.05	350) Set of members - Stability analysis of doubly and singly symme
3	Chord 3 (Member No. 19-24)								
	1.530	CO10	-4.42	0.00	3.48	0.00	-1.05	0.01	100) Negligible internal forces
	1.794	CO5	-10.67	-0.01	-13.49	0.00	-41.03	0.05	111) Cross-section check - Bending about y-axis acc. to 13.5, Clas
	1.794	CO5	-10.67	-0.01	-13.49	0.00	-41.03	0.05	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffen
	0.000	CO5	-9.52	0.01	5.98	0.00	6.38	0.02	325) Stability analysis - Lateral-torsional buckling acc. to 13.6
	0.000	CO5	-10.65	0.00	13.44	0.00	-40.56	0.00	350) Set of members - Stability analysis of doubly and singly symme
4	Chord 4 (Member No. 27-32)								
	5.099	CO17	-5.79	0.00	-3.25	0.00	2.03	0.00	100) Negligible internal forces
	1.794	CO5	-10.76	0.00	-13.45	0.00	-41.62	0.00	111) Cross-section check - Bending about y-axis acc. to 13.5, Clas
	1.794	CO5	-10.76	0.00	-13.45	0.00	-41.62	0.00	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffen
	0.000	CO5	-9.63	0.00	5.96	0.00	5.57	0.00	325) Stability analysis - Lateral-torsional buckling acc. to 13.6
	0.000	CO6	-10.76	0.00	13.43	0.00	-41.43	0.00	350) Set of members - Stability analysis of doubly and singly symme
6	Chord 6 (Member No. 43-48)								
	5.099	CO17	-11.10	0.01	-2.26	0.00	2.02	-0.03	100) Negligible internal forces
	1.794	CO5	-15.12	-0.01	-11.44	0.00	-31.72	0.00	111) Cross-section check - Bending about y-axis acc. to 13.5, Clas
	1.794	CO5	-15.12	-0.01	-11.44	0.00	-31.72	0.00	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffen
	0.598	CO5	-14.86	-0.01	-9.41	0.00	-19.26	-0.01	325) Stability analysis - Lateral-torsional buckling acc. to 13.6
	0.000	CO6	-15.09	0.01	11.41	0.00	-31.45	0.00	350) Set of members - Stability analysis of doubly and singly symme
	0.000	CO25	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deflections
	2.550	CO25	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Deflection in z-direction (Beam)

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

For each set of members, this window shows the internal forces that result in the maximum design ratios. The respective equations according to [1] are referred to in the last column.

© Dlubal Software 2017

4.8 Member Slendernesses

Details...

Window 3.3 is shown when you have selected the respective check box in the *Details* dialog box (see Figure 3.1, page 23).

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C Major Axis y/u $k_{y/u}$ [-]	D Major Axis y/u $i_{y/u}$ [mm]	E Major Axis y/u $\lambda_{y/u}$ [-]	F Minor Axis z/v $k_{z/v}$ [-]	G Minor Axis z/v $i_{z/v}$ [mm]	H Minor Axis z/v $\lambda_{z/v}$ [-]	I
21	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
22	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
29	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
30	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
37	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
38	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
43	Compression / Flexure	1.794	1.000	89.9	19.958	1.000	51.9	34.540	
44	Compression / Flexure	3.059	1.000	89.9	34.031	1.000	51.9	58.896	
45	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
46	Compression / Flexure	5.099	1.000	89.9	56.719	1.000	51.9	98.160	
47	Compression / Flexure	3.059	1.000	89.9	34.031	1.000	51.9	58.896	
48	Compression / Flexure	1.794	1.000	89.9	19.958	1.000	51.9	34.540	
49	Compression / Flexure	6.000	1.000	219.1	27.390	1.000	46.1	130.241	
50	Compression / Flexure	6.000	1.000	219.1	27.390	1.000	46.1	130.241	
51	Compression / Flexure	4.844	1.000	61.7	78.504	1.000	22.5	214.905	
52	Compression / Flexure	5.099	1.000	61.7	82.636	1.000	22.5	226.216	
53	Compression / Flexure	5.099	1.000	61.7	82.636	1.000	22.5	226.216	
54	Compression / Flexure	4.844	1.000	61.7	78.504	1.000	22.5	214.905	
55	Compression / Flexure	6.000	1.000	219.1	27.390	1.000	46.1	130.241	
56	Compression / Flexure	7.000	1.000	36.4	192.290	1.000	34.7	201.459	
57	Compression / Flexure	8.000	1.000	36.4	219.760	1.000	34.7	230.239	
58	Compression / Flexure	7.000	1.000	36.4	192.290	1.000	34.7	201.459	
64	Compression / Flexure	5.000	1.000	34.9	143.327	1.000	34.9	143.327	
70	Compression / Flexure	5.000	1.000	34.9	143.327	1.000	34.9	143.327	
76	Compression / Flexure	5.000	1.000	34.9	143.327	1.000	34.9	143.327	
82	Compression / Flexure	5.000	1.000	34.9	143.327	1.000	34.9	143.327	
88	Compression / Flexure	5.000	1.000	34.9	143.327	1.000	34.9	143.327	

Members with compression / flexure:

Max $K_{y/u}L / r_{y/u}$: 219.760 < 200

Max $K_{z/v}L / r_{z/v}$: 230.239 < 200

Figure 4.9: Window *3.3 Member Slendernesses*

The table lists the effective slenderness ratios of the designed members for both directions of the principal axes. They are determined in compliance with the load type.

Details...

Below the list, you find a comparison of the most unfavorable values with the limit values that have been defined in the *Details* dialog box (see Figure 3.1, page 23).

Members of the types 'tension' or 'cable' are not included in this table.

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

4.9 Parts List by Member

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

4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	Total Weight [t]
1	2 - W 200x59 | CAN/CSA-S16-01	8	5.10	40.79	49.77	0.31	59.35	302.61	2.421
2	2 - W 200x59 | CAN/CSA-S16-01 ... 3 - W 20	2	1.79	3.59	4.38	0.03	59.35	106.48	0.213
3	2 - W 200x59 | CAN/CSA-S16-01	2	3.06	6.12	7.46	0.05	59.35	181.56	0.363
4	1 - W 530x109 | CAN/CSA-S16-01	3	6.00	18.00	34.20	0.25	109.11	654.69	1.964
5	6 - W 150x13 | CAN/CSA-S16-01	2	4.84	9.69	6.68	0.02	12.64	61.22	0.122
6	6 - W 150x13 | CAN/CSA-S16-01	2	5.10	10.20	7.04	0.02	12.64	64.44	0.129
7	5 - WT 125x19.5 | CAN/CSA-S16-01	2	7.00	14.00	7.64	0.03	19.31	135.18	0.270
8	5 - WT 125x19.5 | CAN/CSA-S16-01	1	8.00	8.00	4.36	0.02	19.31	154.49	0.154
9	7 - QRO 89x89x3.2 | CAN/CSA-G40.20	5	5.00	25.00	8.75	0.03	8.32	41.60	0.208
10	10 - L 102x76x6.4 | CAN/CSA-S16-01	4	7.81	31.24	10.92	0.03	8.56	66.83	0.267
11	8 - RD 24	8	7.14	57.13	4.31	0.03	3.55	25.34	0.203
Sum		39		223.76	145.51	0.80			6.315

Figure 4.10: Window *4.1 Parts List by Member*

Details...

By default, this list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box (see Figure 3.1, page 23).

Part No.

The program automatically assigns item numbers to members with identical features.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column shows the respective length of an individual member.

Total Length

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

Surface Area

For each item, the program gives the surface area relative to the total length. This area is determined from the *Surface Area* of the cross-sections. It can be checked in Windows 1.3 and 2.1 to 2.5 in the cross-section properties (see Figure 2.10, page 12).

Volume

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

Unit Weight

The unit mass of a cross-section is related to the length of one meter. For tapered sections, the program averages both cross-section masses.

Weight

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

Total Weight

The final column gives the total mass of each sectional part.

Sum

At the end of the list, you find a summary of the values in the columns B, D, E, F, and I. The last row of the *Total Weight* column shows the total amount of required steel.

4.10 Parts List by Set of Members

4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Chord 2	1	19.91	19.91	24.28	0.15	59.35	1181.30	1.181
2	Chord 3	1	19.91	19.91	24.28	0.15	59.35	1181.30	1.181
3	Chord 4	1	19.91	19.91	24.28	0.15	59.35	1181.30	1.181
4	Chord 6	1	19.91	19.91	24.28	0.15	59.35	1181.30	1.181
5	Column A	1	8.00	8.00	4.36	0.02	19.31	154.49	0.154
Sum		5		87.62	101.50	0.62			4.880

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

The last result window is displayed when you have selected at least one set of members for design. It represents the parts list of structural groups, for example horizontal beams.

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

© DLUBAL SOFTWARE 2017

5 Results Evaluation

You can evaluate the design results in different ways. For this, the buttons below the tables are very useful.

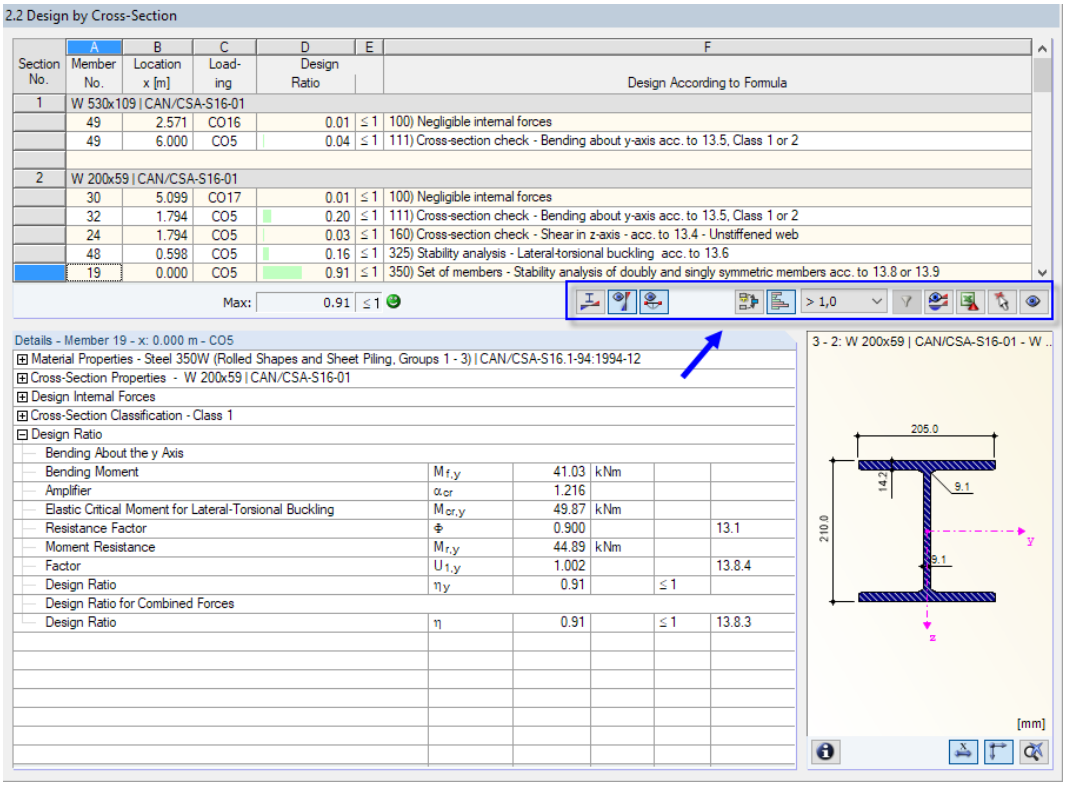
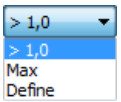


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Mode Shape	Opens *Mode Shape View* Window showing buckling shape → Chapter 5.4, page 42
	Ultimate Limit State Design	Shows or hides the results of the ULS design
	Serviceability Limit State Design	Shows or hides the results of the SLS design
	Result Combination	Creates a new result combination from the governing load cases and load combinations
	Color Bars	Shows or hides the colored relation scales in the tables
	Filter Parameters	Describes the filter criterion for the output in the tables: Design ratios greater than 1, maximum value or user-defined limit
	Apply Filter	Displays only rows where the filter parameters are valid (ratio > 1, maximum, user-defined limit)
	Result Diagrams	Opens the *Result Diagram on Member* Window → Chapter 5.2, page 40
	Excel Export	Exports the table to MS Excel → Chapter 7.4.3, page 50

	Member Selection	Option to select a member graphically to display its results in the table
	View Mode	Jumps to the work window of RFEM or RSTAB to view objects in model

Table 5.1: Buttons in Windows 2.1 to 2.5

5.1 Results in RFEM/RSTAB Model

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

Background graphic and view mode

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

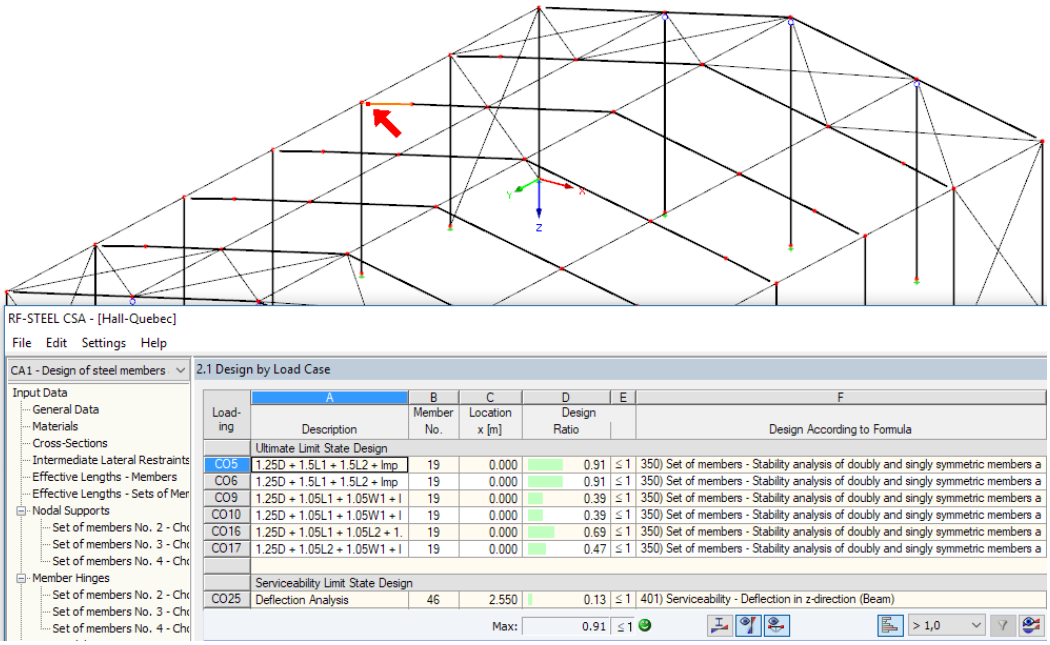
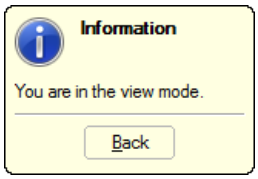


Figure 5.2: Indication of member and relevant *Location x* in RFEM model



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

Click [Back] to return to the RF-/STEEL CSA module.

RFEM/RSTAB work window

Graphics

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

In the *Results* navigator, you can select whether the ratios of the ULS and/or SLS designs are to be displayed.

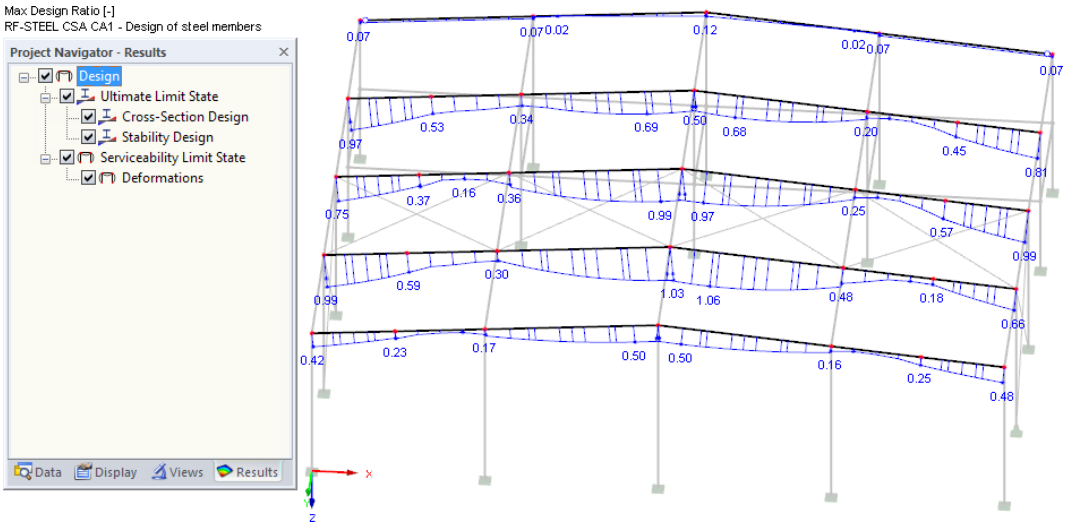
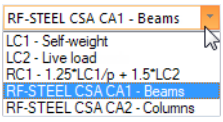


Figure 5.3: *Results* navigator for *Ultimate Limit State* and *Serviceability Limit State* designs

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

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



You can set the relevant RF-/STEEL CSA design case in the list of the toolbar.

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

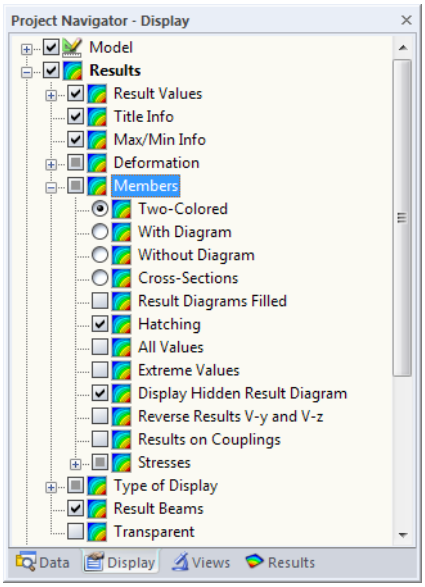


Figure 5.4: *Display* navigator: *Results* → *Members*

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

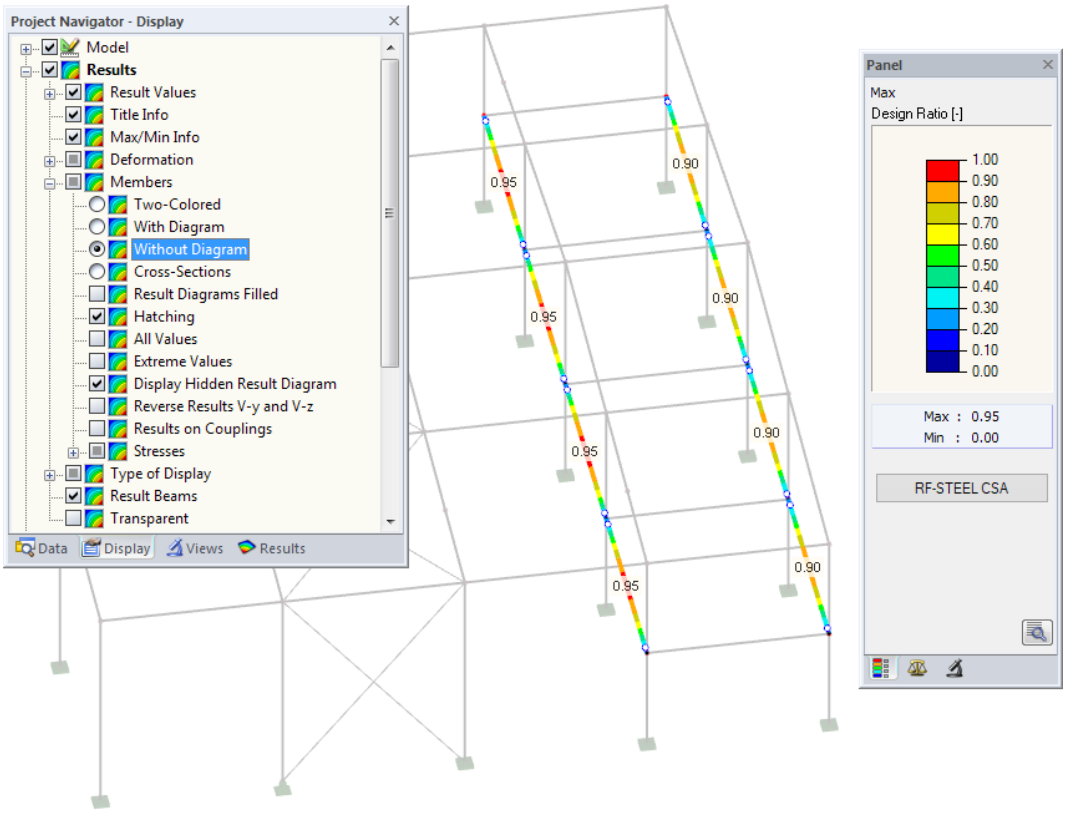


Figure 5.5: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 44).

RF-STEEL CSA

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

© Dlubal Software 2017

5.2 Result Diagram

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

Select the member (or set of members) in the RF-/STEEL CSA result window by clicking in the relevant table row. Then click the button to open the *Result Diagram on Member* dialog box. This button is located below the upper table (see Figure 5.1, page 36).

If you want to open the result diagram in the work window of RFEM or RSTAB, select the menu

Results → Result Diagrams for Selected Members

or the toolbar button shown on the left.

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

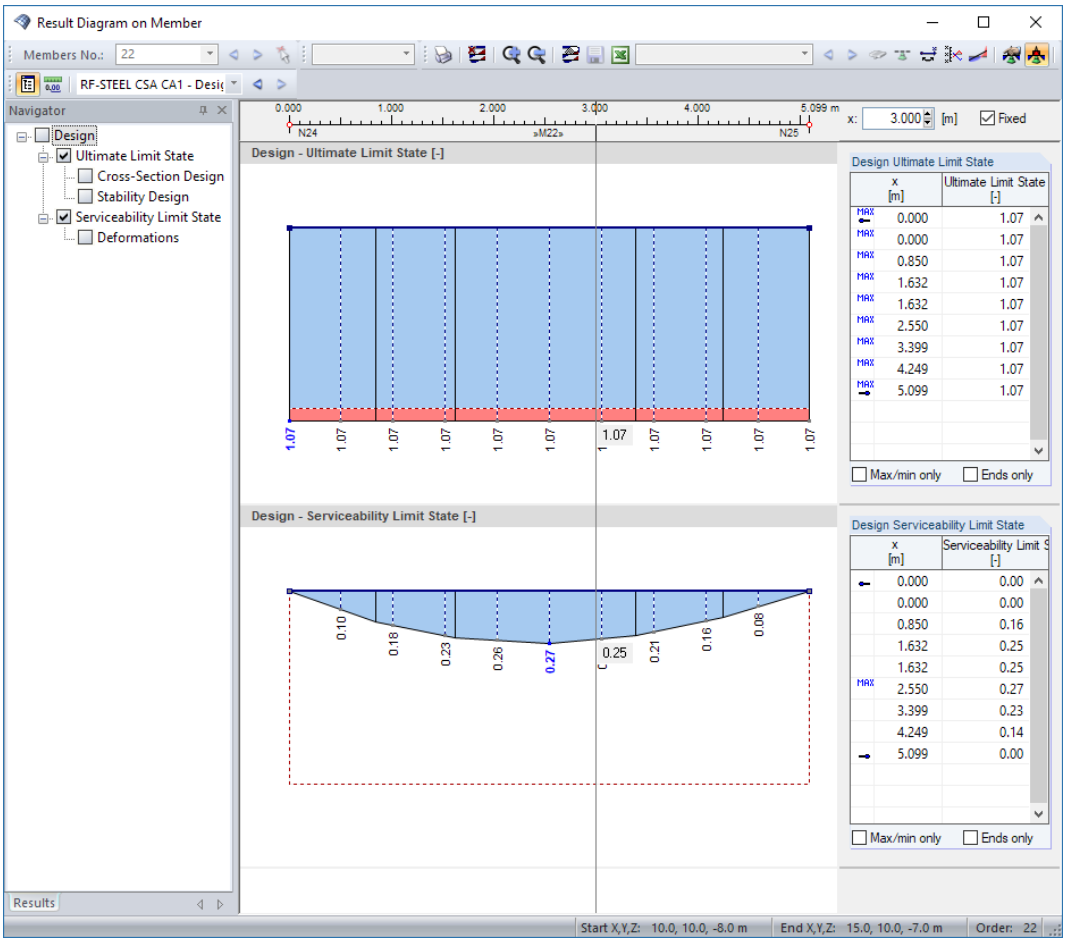
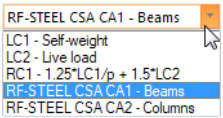


Figure 5.6: Dialog box *Result Diagram on Member*

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

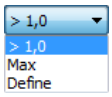


Use the list in the toolbar to select the relevant RF-/STEEL CSA design case.

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

© Dlubal Software 2017

5.3 Filter for Results

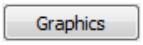


The RF-/STEEL CSA result windows allow you to sort the results by various criteria. In addition, you can use the filter options for the tables (see Figure 5.1, page 36) to reduce the numerical output according to specific ratios. This function is described in the *Knowledge Base* on our website: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000733

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

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

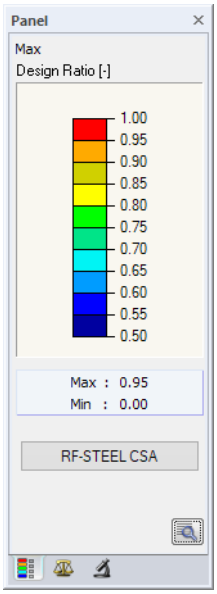
Filtering design ratios



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

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

or use the toolbar button shown on the left.



The panel is described in the RFEM/RSTAB manual, Chapter 3.4.6. The filter settings for the results can be defined in the first tab (Color scale). As this tab is not available for the two-colored results display, you have to set the display option *Colored With/Without Diagram* or *Cross-Sections* in the *Display* navigator (see Figure 5.4, page 38).

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

Filtering members

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

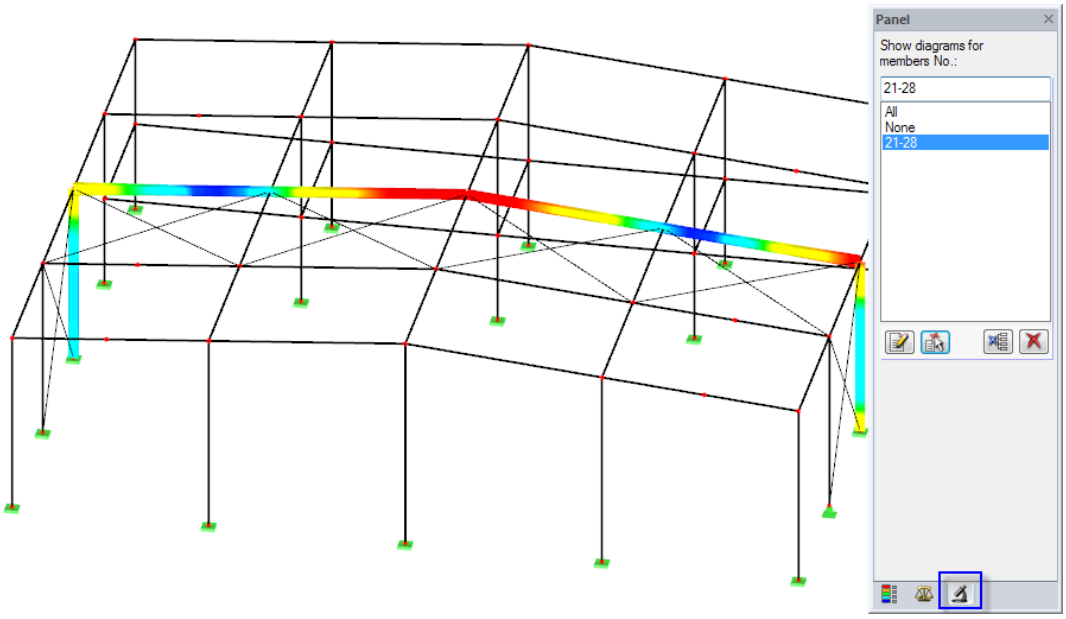


Figure 5.7: Filtering design ratios of one frame

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

5.4 Mode Shapes

The mode shapes of the sets of members can be checked graphically in a separate window: Go to the relevant set of members in the result window, and then select its row of *Set of members - Stability analysis* results. Now you can click the [Mode Shapes] button.

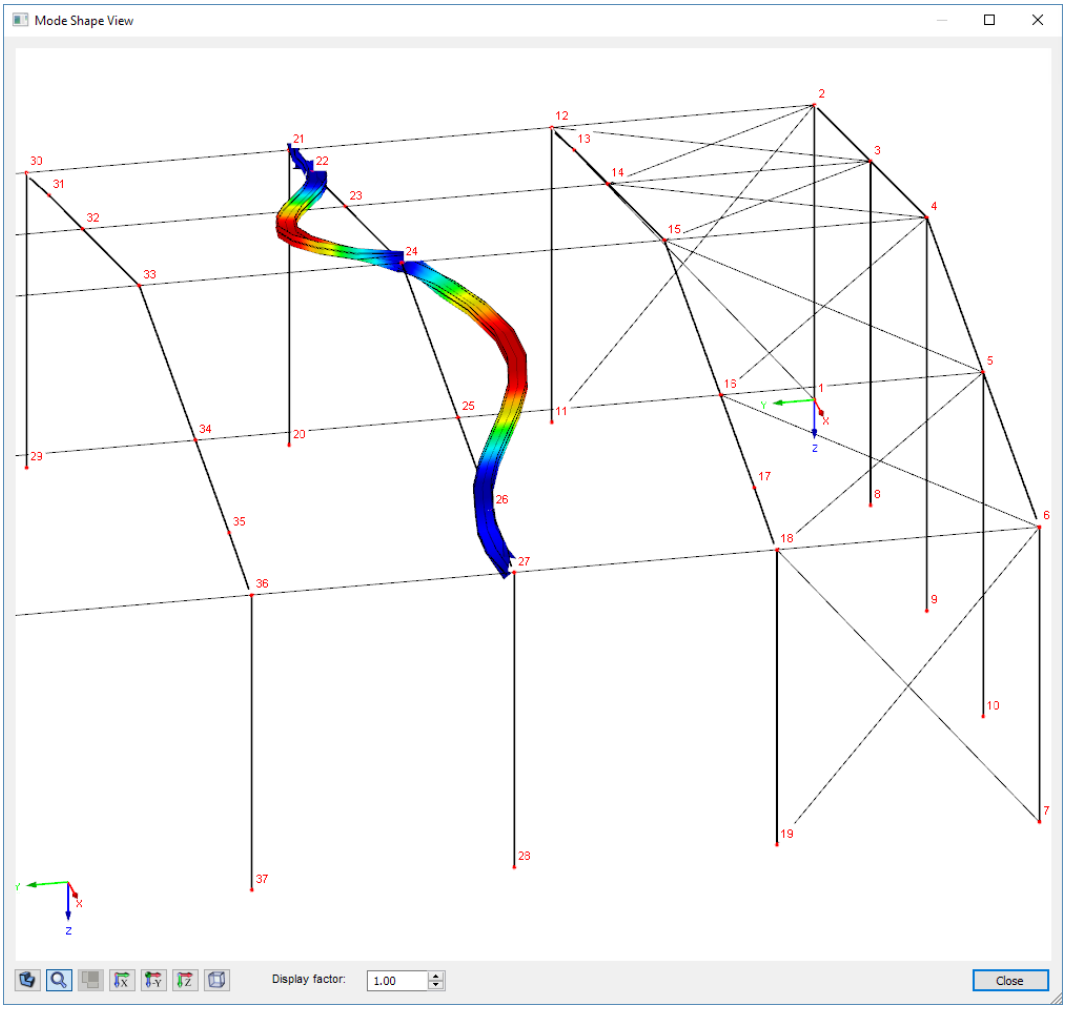


Figure 5.8: Mode shape of a set of members

The mode shapes of the sets of members are created automatically when determining the critical elastic moment, M_u. The mode shapes are not available in numerical form.

The buttons below the graphic are described in Table 2.2 on page 20.

Use the *Display factor* to scale the representation of the mode shape.

© Dlubal Software 2017

6 Printout

6.1 Printout Report

Similarly to RFEM or RSTAB, the program generates a printout report for the RF-/STEEL CSA results. It can be supplemented by graphics and descriptions. The selection in the printout report controls which data of the design module are included in the final printout.

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

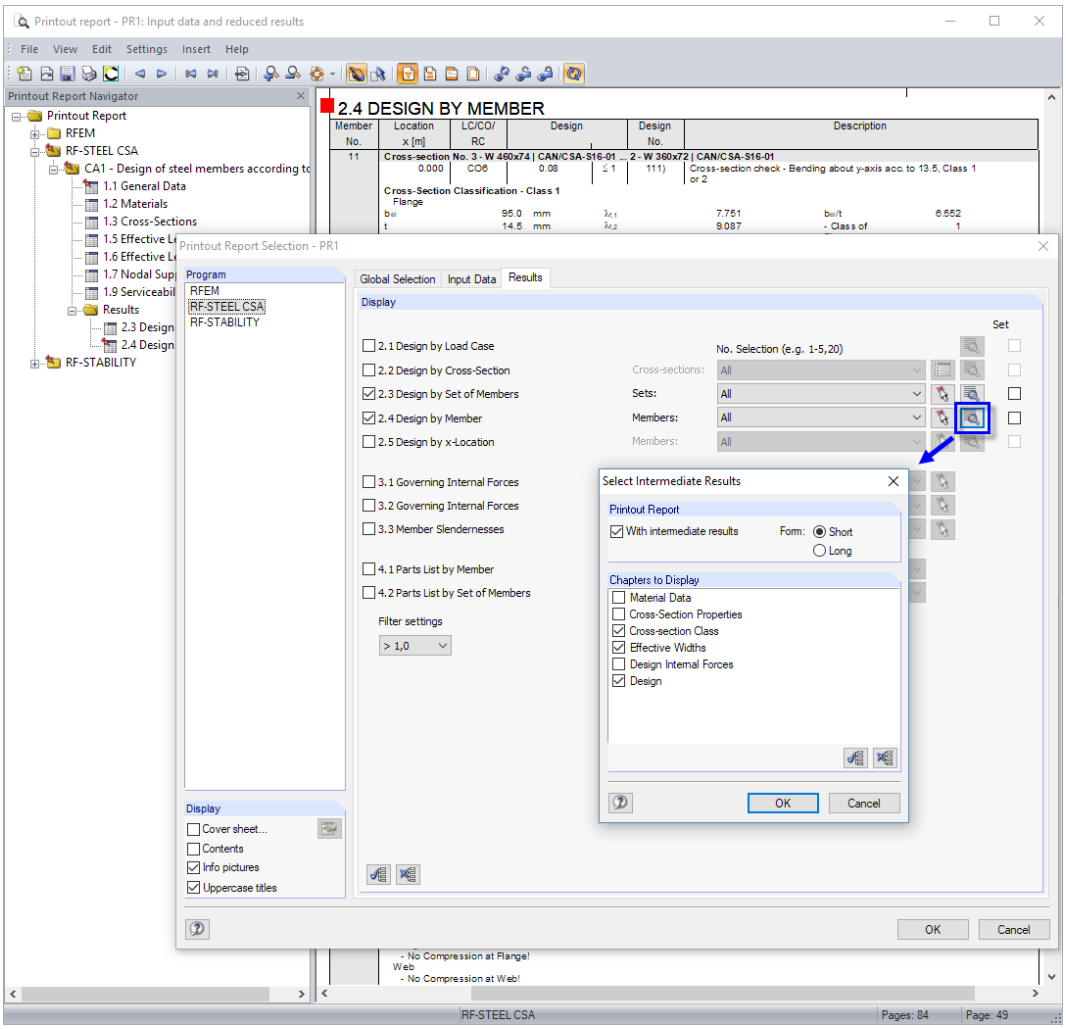


Figure 6.1: Selecting designs and intermediate results in printout report

Click the [Details] button when you want to include all or specific intermediate results in the printout report. They can be documented in a *Short* (compact list) or *Long* (descriptive list) form.

If you work on complex models featuring many design cases, you can split the data into several printout reports. This allows for a clearly arranged documentation.

© Dlubal Software 2017

6.2 Graphic Printout

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

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

To print the currently displayed graphic of the design ratios, click

File → Print Graphic

or use the toolbar button shown on the left.

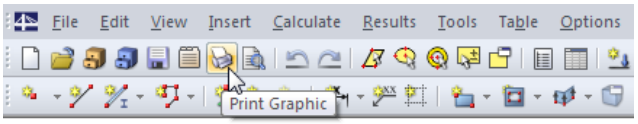


Figure 6.2: [Print Graphic] button in RFEM toolbar

The *Graphic Printout* dialog box appears.

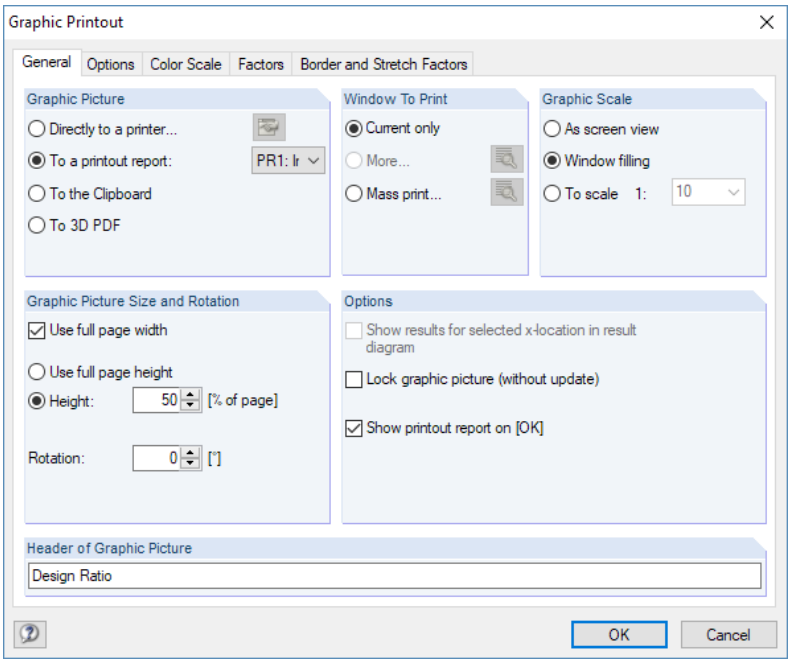
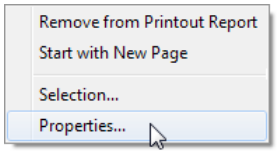


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

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



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

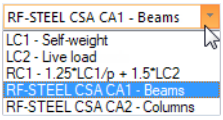
7 General Functions

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

7.1 Design Cases

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

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



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

Create design case

To create a new design case, use the RF-/STEEL CSA menu and select

File → **New Case**.

The following dialog box appears.

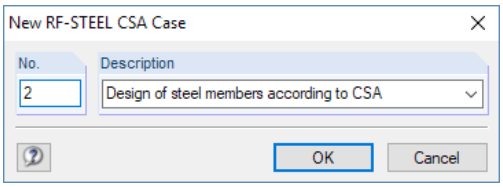


Figure 7.1: Dialog box *New RF-STEEL CSA Case*

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

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

Rename design case

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

File → **Rename Case**.

The following dialog box appears.

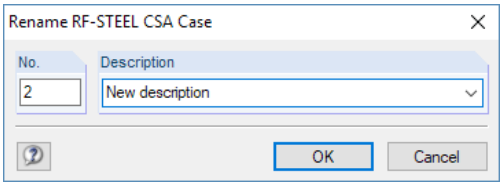


Figure 7.2: Dialog box *Rename RF-STEEL CSA Case*

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

Copy design case

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

File → **Copy Case**.

The following dialog box appears.

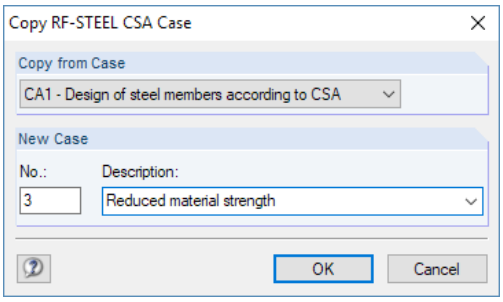


Figure 7.3: Dialog box *Copy RF-STEEL CSA Case*

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

Delete design case

To delete a design case, use the RF-/STEEL CSA menu and select

File → **Delete Case**.

The following dialog box appears.

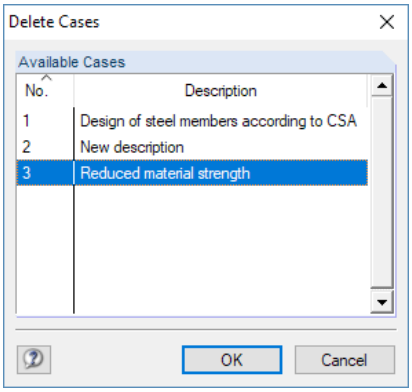
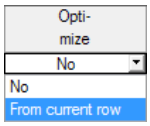


Figure 7.4: Dialog box *Delete Cases*

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

7.2 Cross-Section Optimization



The design module offers you the option to optimize overstressed or little utilized cross-sections. Open the drop-down list in column D resp. E in Window *1.3 Cross-Sections* (see Figure 2.8, page 10) and select the optimization option *From current row*.

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

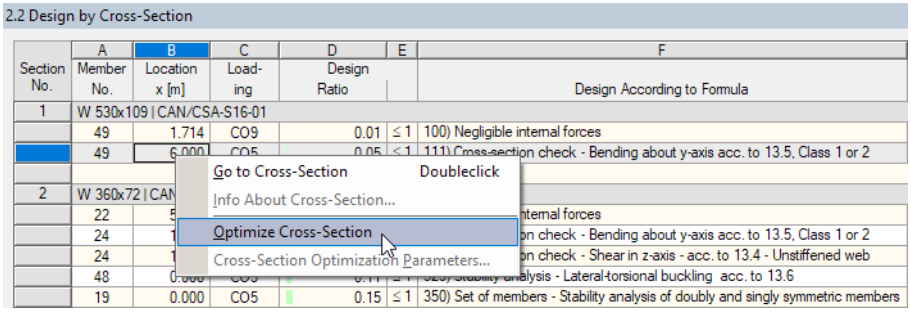


Figure 7.5: Shortcut menu to *Optimize Cross-Section*

During the optimization, the module determines the section that fulfills the analysis requirements in the "optimal" way, i.e. comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.1, page 23). The required cross-sectional properties are calculated with the internal forces of RFEM or RSTAB. If a different cross-section proves to be more favorable, it will be used for the design. In this case, the graphic in Window 1.3 shows two cross-sections – the original section from RFEM or RSTAB and the optimized one (see Figure 7.7).

When you optimize a parametric cross-section, the following dialog box appears:

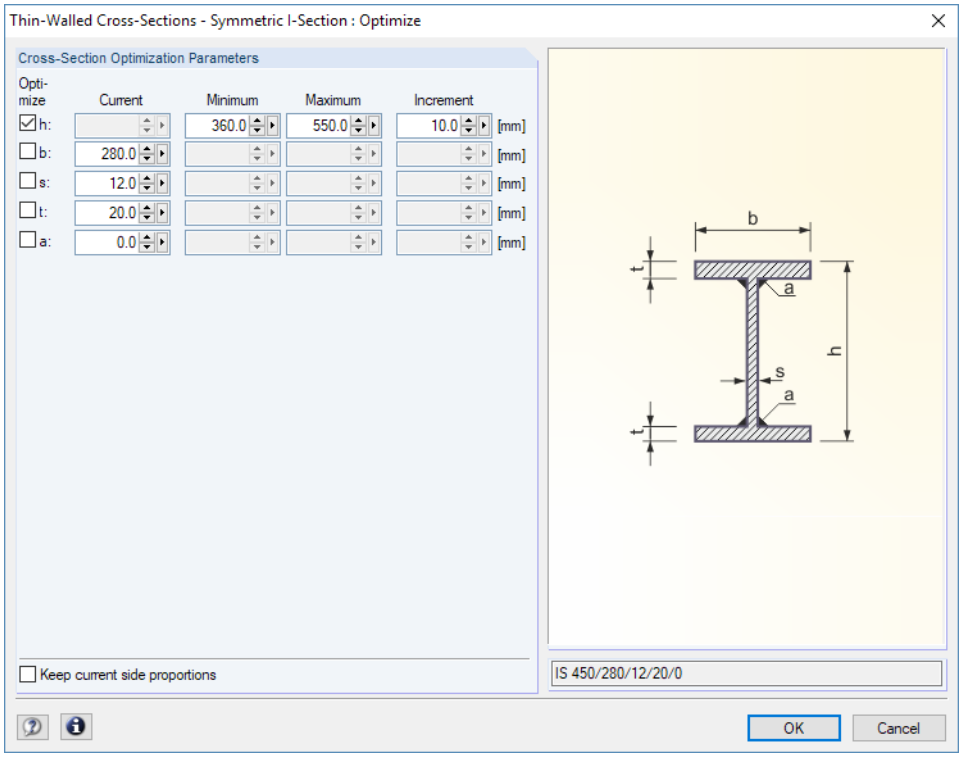


Figure 7.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section: Optimize*

By selecting the check box(es) in the *Optimize* column, you decide which parameter(s) you want to modify. They activate the *Minimum* and *Maximum* columns where you can specify the upper and lower limits of each parameter. The *Increment* column controls the interval in which the value of the parameter varies during the optimization.

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

Cross-sections composed of combined rolled cross-sections cannot be optimized.

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

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

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

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

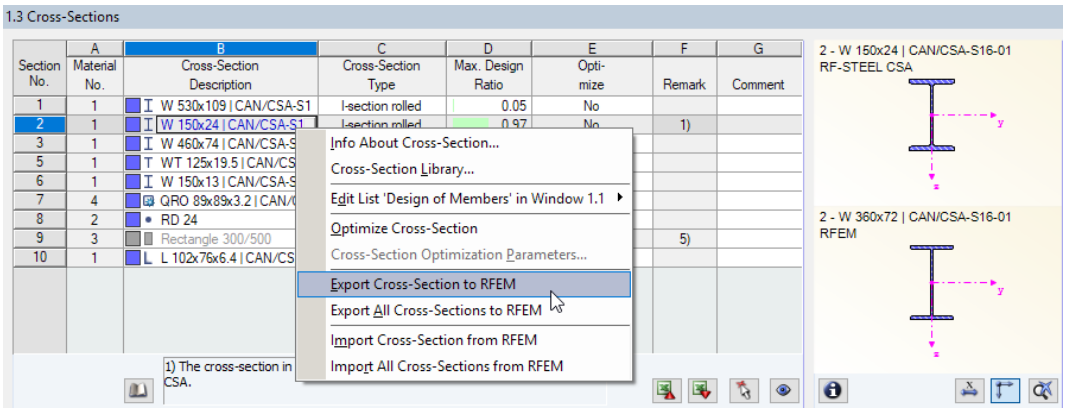


Figure 7.7: Shortcut menu in Window *1.3 Cross-Sections*

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

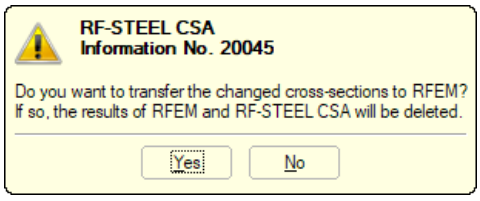


Figure 7.8: Confirmation when exporting cross-sections

Calculation

By approving the confirmation and starting the [Calculation] in the RF-/STEEL CSA module, the internal forces of RFEM or RSTAB as well as the design ratios will be determined in one single calculation run.

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

When optimizing a tapered member, the program modifies the cross-sections of the member start and member end. For the intermediate locations, the second moments of area are linearly interpolated. Since those values are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-sections differ considerably. It is then recommended to divide the taper into several members, thus modeling the taper layout manually.

© Dlubal Software 2017

7.3 Units and Decimal Places

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

Settings → Units and Decimal Places.

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

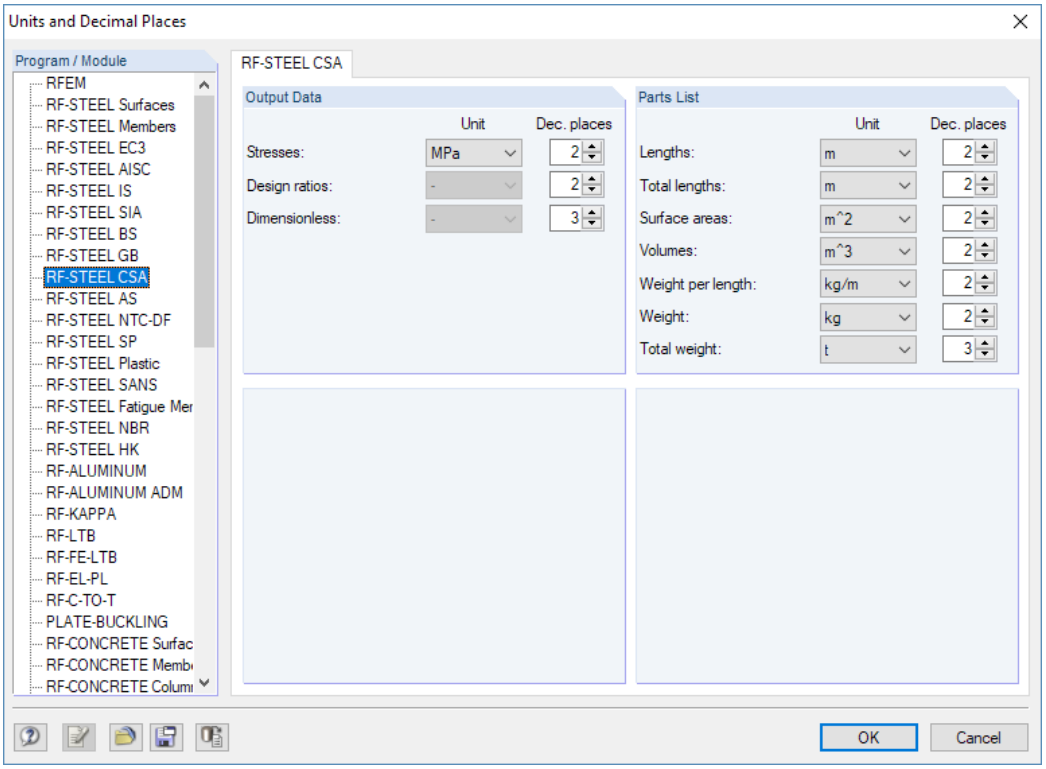


Figure 7.9: Dialog box *Units and Decimal Places*

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

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

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

Edit → Export All Materials to RFEM/RSTAB.

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

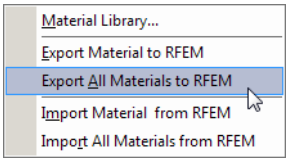
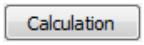


Figure 7.10: Shortcut menu of Window *1.2 Materials*



Before the modified materials are transferred to RFEM or RSTAB, a confirmation is required as to whether the results of the main program should be deleted. When you approve this confirmation and then start the [Calculation] in RF-/STEEL CSA, the new internal forces and design ratios will be determined in one single calculation run.

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

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

If you have adjusted the effective lengths in RF-/STEEL CSA for the design, you can export the modified values to RFEM or RSTAB in a similar way as you export cross-sections: Go to Window *1.5 Effective Lengths - Members* and then select

Edit → Export All Effective Lengths to RFEM/RSTAB.

You can also use the corresponding option on the shortcut menu of Window 1.5.

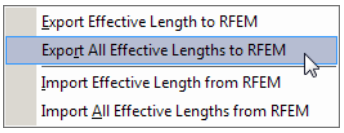


Figure 7.11: Shortcut menu of Window *1.5 Effective Lengths - Members*

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

7.4.3 Exporting Results

The RF-/STEEL CSA results can also be used by other programs.

Clipboard

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

© DLUBAL SOFTWARE 2017

Printout Report

You can print the data of RF-/STEEL CSA to the printout report (see Chapter 6.1, page 43). To export the tables and graphics, then select the printout report menu

File → Export to RTF.

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

Excel

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

File → Export Tables.

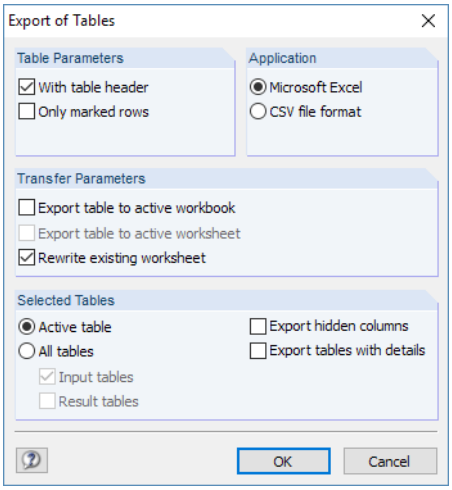
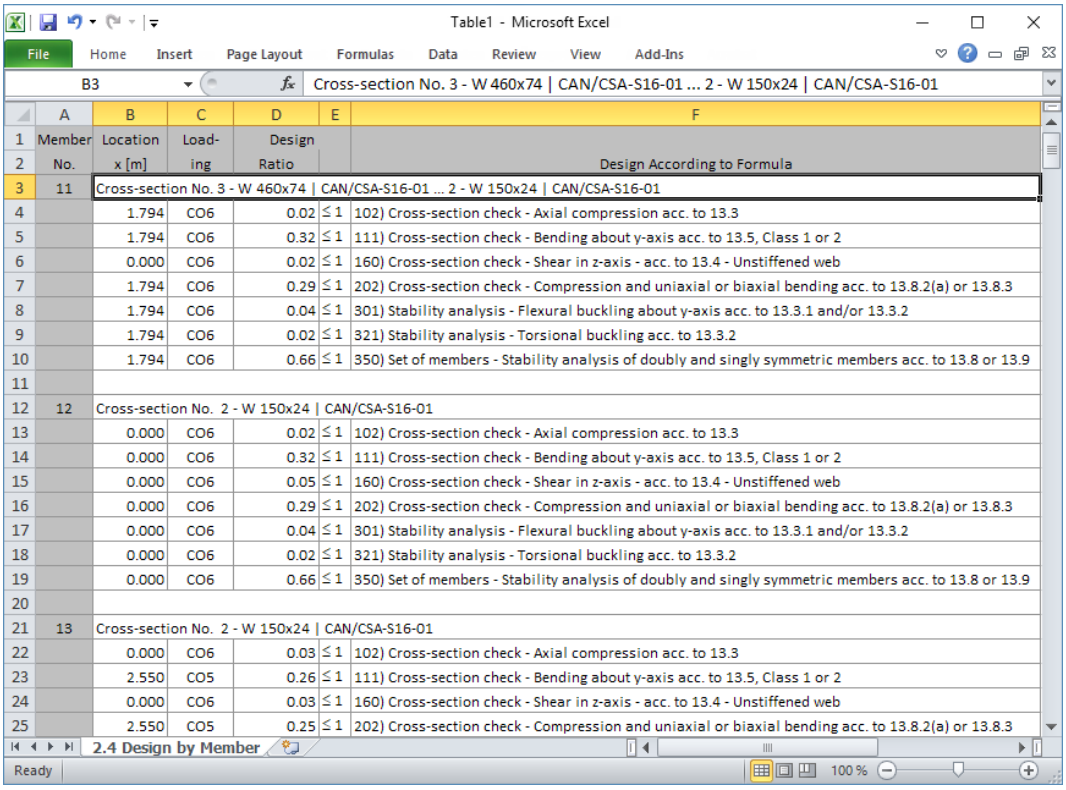


Figure 7.12: Dialog box *Export of Tables*

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



	A	B	C	D	E	F
1	Member	Location	Load-	Design		
2	No.	x [m]	ing	Ratio		Design According to Formula
3	11	Cross-section No. 3 - W 460x74 | CAN/CSA-S16-01 ... 2 - W 150x24 | CAN/CSA-S16-01				
4		1.794	CO6	0.02	≤ 1	102) Cross-section check - Axial compression acc. to 13.3
5		1.794	CO6	0.32	≤ 1	111) Cross-section check - Bending about y-axis acc. to 13.5, Class 1 or 2
6		0.000	CO6	0.02	≤ 1	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffened web
7		1.794	CO6	0.29	≤ 1	202) Cross-section check - Compression and uniaxial or biaxial bending acc. to 13.8.2(a) or 13.8.3
8		1.794	CO6	0.04	≤ 1	301) Stability analysis - Flexural buckling about y-axis acc. to 13.3.1 and/or 13.3.2
9		1.794	CO6	0.02	≤ 1	321) Stability analysis - Torsional buckling acc. to 13.3.2
10		1.794	CO6	0.66	≤ 1	350) Set of members - Stability analysis of doubly and singly symmetric members acc. to 13.8 or 13.9
11						
12	12	Cross-section No. 2 - W 150x24 | CAN/CSA-S16-01				
13		0.000	CO6	0.02	≤ 1	102) Cross-section check - Axial compression acc. to 13.3
14		0.000	CO6	0.32	≤ 1	111) Cross-section check - Bending about y-axis acc. to 13.5, Class 1 or 2
15		0.000	CO6	0.05	≤ 1	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffened web
16		0.000	CO6	0.29	≤ 1	202) Cross-section check - Compression and uniaxial or biaxial bending acc. to 13.8.2(a) or 13.8.3
17		0.000	CO6	0.04	≤ 1	301) Stability analysis - Flexural buckling about y-axis acc. to 13.3.1 and/or 13.3.2
18		0.000	CO6	0.02	≤ 1	321) Stability analysis - Torsional buckling acc. to 13.3.2
19		0.000	CO6	0.66	≤ 1	350) Set of members - Stability analysis of doubly and singly symmetric members acc. to 13.8 or 13.9
20						
21	13	Cross-section No. 2 - W 150x24 | CAN/CSA-S16-01				
22		0.000	CO6	0.03	≤ 1	102) Cross-section check - Axial compression acc. to 13.3
23		2.550	CO5	0.26	≤ 1	111) Cross-section check - Bending about y-axis acc. to 13.5, Class 1 or 2
24		0.000	CO6	0.03	≤ 1	160) Cross-section check - Shear in z-axis - acc. to 13.4 - Unstiffened web
25		2.550	CO5	0.25	≤ 1	202) Cross-section check - Compression and uniaxial or biaxial bending acc. to 13.8.2(a) or 13.8.3

Figure 7.13: Results in Excel

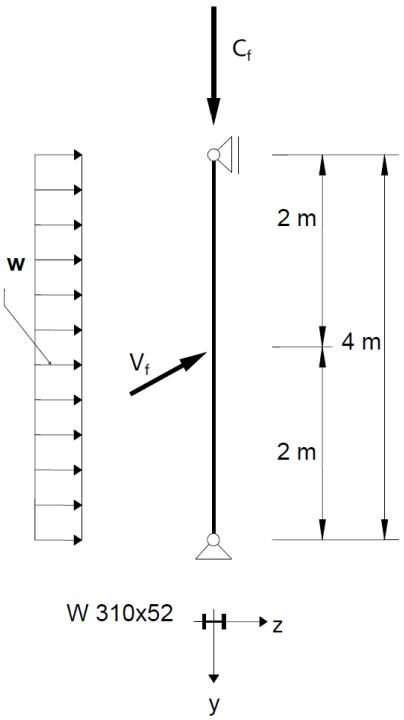
8 Example

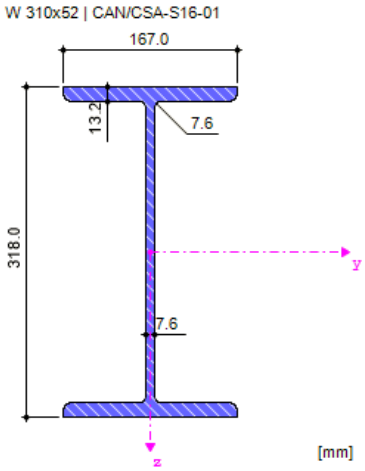
Column with Compression and Biaxial Bending

This example illustrates the stability design of a column with regard to flexural buckling and lateral torsional buckling. Furthermore, the interaction of bending moments and axial compressive forces is examined. The analysis follows the *Limit State Design* provisions.

8.1 Design Values

Model and loads



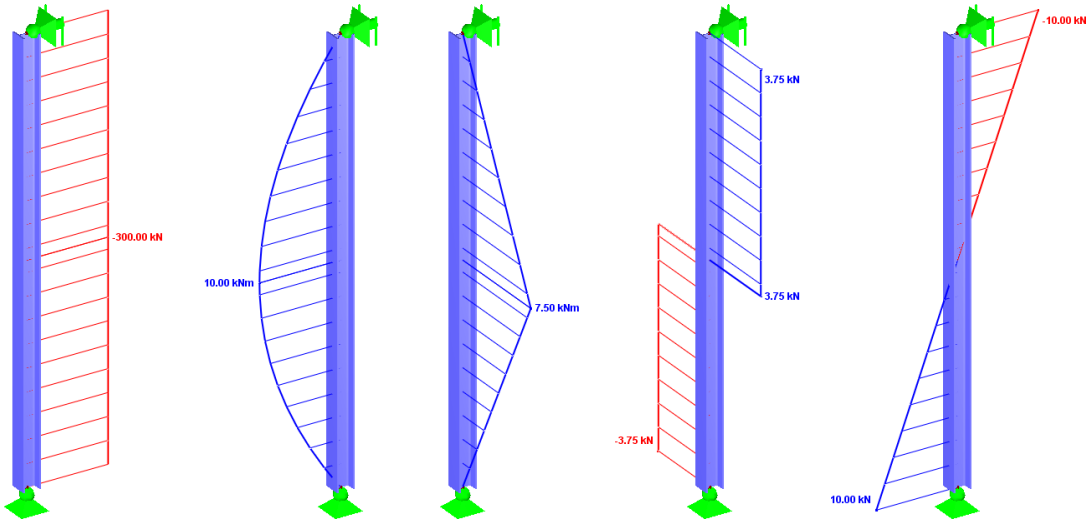


Material: Steel 260W

Design values of loads:
$C_f = 300$ kN
$w = 5.0$ kN/m
$V_f = 7.5$ kN

Figure 8.1: Model and loads

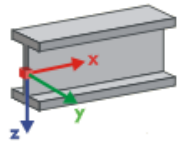
Internal forces due to linear analysis



N M_x M_y F_{vy} F_{vz}

Figure 8.2: Internal forces

Design location



The design proceeds according to x-locations, i.e. for specific positions *x* of the member. The governing location is at x = 2.00 m in the middle of the member, with the following internal forces:

N = −300.00 kN M_y = 10.00 kNm M_z = 7.50 kNm V_y = 3.75 kN V_z = 0.00 kN

8.2 Cross-Section Properties - W 310x52

Cross-Section Property	Symbol	Value	Unit
Gross area	A_g	66.70	cm^2
Moment of inertia about strong axis	I_y	11,900.00	cm^4
Moment of inertia about weak axis	I_z	1,030.00	cm^4
Radius of gyration about strong axis	r_y	13.40	cm
Radius of gyration about weak axis	r_z	3.90	mm
Cross-section weight	wt	52.40	kg/m
Torsional constant	J	30.80	cm^4
Warping constant	C_w	238,000.00	cm^6
Elastic section modulus about strong axis	S_y	750.00	cm^3
Elastic section modulus about weak axis	S_z	123.00	cm^3
Plastic section modulus about strong axis	Z_y	841.00	cm^3
Plastic section modulus about weak axis	Z_z	189.00	cm^3

8.3 Material Properties - 260W

Material Property	Symbol	Value	Unit
Modulus of elasticity	E	200,000	MPa
Shear modulus	G	77,000	MPa
Yield strength	F_y	260	MPa

© DLUBAL SOFTWARE 2017

8.4 Classification of Cross-Section

Flange

$b_{el} = 83.5$ mm

$t = 13.2$ mm

$$\lambda_{f,1} = \frac{145}{\sqrt{F_y}} = \frac{145}{\sqrt{260}} = 8.99$$

$$\lambda_{f,2} = \frac{170}{\sqrt{F_y}} = \frac{170}{\sqrt{260}} = 10.54$$

$$\lambda_{f,3} = \frac{200}{\sqrt{F_y}} = \frac{200}{\sqrt{260}} = 12.40$$

$$\frac{b_{el}}{t} = \frac{83.5}{13.2} = 6.33 \leq 8.99 = \lambda_{f,1} \quad \rightarrow \text{Class 1}$$

Web

$h = 291.6$ mm

$w = 7.6$ mm

$$\lambda_{w,1} = \frac{1{,}100}{\sqrt{F_y}} \left(1 - 0.39 \frac{C_f}{\phi C_y}\right) = \frac{1{,}100}{\sqrt{260}} \left(1 - 0.39 \frac{300{,}000}{0.9 \cdot 260 \cdot 6{,}670}\right) = 63.11$$

$$\lambda_{w,2} = \frac{1{,}700}{\sqrt{F_y}} \left(1 - 0.61 \frac{C_f}{\phi C_y}\right) = \frac{1{,}700}{\sqrt{260}} \left(1 - 0.61 \frac{300{,}000}{0.9 \cdot 260 \cdot 6{,}670}\right) = 93.07$$

$$\lambda_{w,3} = \frac{1{,}900}{\sqrt{F_y}} \left(1 - 0.65 \frac{C_f}{\phi C_y}\right) = \frac{1{,}900}{\sqrt{260}} \left(1 - 0.65 \frac{300{,}000}{0.9 \cdot 260 \cdot 6{,}670}\right) = 103.11$$

$$\frac{h}{w} = \frac{291.6}{7.6} = 38.37 \leq 63.11 = \lambda_{w,1} \quad \rightarrow \text{Class 1}$$

$\Rightarrow$ **Class of section: 1**

Classification in RF-/STEEL CSA

Cross-Section Classification - Class 1					
Flange					
- Half of Full Flange Width	b_{el}	83.5	mm		Chapt. 11.3
- Thickness	t	13.2	mm		Chapt. 11.3
- Yield Strength	F_y	260.00	MPa		
- Maximum Ratio for Class 1	$\lambda_{f,1}$	8.993			Table 1 or 2
- Maximum Ratio for Class 2	$\lambda_{f,2}$	10.543			Table 1 or 2
- Maximum Ratio for Class 3	$\lambda_{f,3}$	12.403			Table 1 or 2
- b_{el}/t-Ratio	b_{el}/t	6.326		$\leq \lambda_{f,1}$	
- Class of Flange		1			Table 1 or 2
Web					
- Length	h	291.6	mm		Chapt. 11.3
- Thickness	w	7.6	mm		Chapt. 11.3
- Yield Strength	F_y	260.00	MPa		
- Compression Force	C_f	300.00	kN		
- Gross Area	A_g	66.70	cm^2		
- Resistance factor	Φ	0.900			Chapt. 13.1
- Compressive Load at Yield Stress	C_y	1734.20	kN		
- Maximum Ratio for Class 1	$\lambda_{w,1}$	63.105			Table 1 or 2
- Maximum Ratio for Class 2	$\lambda_{w,2}$	93.068			Table 1 or 2
- Maximum Ratio for Class 3	$\lambda_{w,3}$	103.111			Table 1 or 2
- h/w-Ratio	h/w	38.368		$\leq \lambda_{w,1}$	
- Class of Web		1			Table 1 or 2
Class of Cross-Section		1			Table 1 or 2

Figure 8.3: Classification in RF-/STEEL CSA

8.5 Flexural Buckling

8.5.1 Buckling about y-Axis (Strong Axis)

Effective length:

$$L_{eff} = K_y L = 4.0 \text{ m}$$

Parameter for compressive resistance:

$$n = 1.34$$

Euler buckling stress:

$$F_{e,y} = \frac{\pi^2 E}{\left(\frac{L_{eff}}{r_y}\right)^2} = \frac{\pi^2 \cdot 200{,}000}{\left(\frac{4{,}000}{13.4}\right)^2} = 2{,}201.05 \text{ MPa}$$

Slenderness:

$$\lambda_y = \sqrt{\frac{F_y}{F_{e,y}}} = \sqrt{\frac{260}{2{,}201.05}} = 0.34$$

Factored axial compressive resistance:

$$C_{r,y} = \frac{\phi A F_y}{\left(1 + \lambda_y^{2n}\right)^{1/n}} = \frac{0.9 \cdot 6{,}670 \cdot 260}{\left(1 + 0.34^{2 \cdot 1.34}\right)^{1/1.34}} = 1{,}497.38 \text{ kN}$$

Design ratio:

$$\eta = \frac{C_f}{C_{r,y}} = \frac{300.00}{1{,}497.38} = 0.20 \leq 1.00 \qquad \Rightarrow \textbf{OK}$$

Buckling design in RF-/STEEL CSA

Design Ratio					
Compression Force	C_f	300.00	kN		
Effective Member Length	$L_{eff,y}$	4.000	m		
Yield Strength	F_y	260.00	MPa		
Modulus of Elasticity	E	200000.00	MPa		
Slenderness Ratio	K_yL/r_y	29.947			
Euler Buckling Stress	$F_{e,y}$	2201.05	MPa		13.3.1
Gross Area	A_g	66.70	cm^2		
Parameter for Compressive Resistance	n	1.340			13.3.1
Resistance Factor	Φ	0.900			13.1
Factored Compressive Resistance	$C_{r,y}$	1497.38	kN		13.3.1
Design Ratio	η_y	0.20		≤ 1	

Figure 8.4: Design details in RF-/STEEL CSA (check no. 301)

8.5.2 Buckling about z-Axis (Weak Axis)

Effective length:

$L_{eff} = K_z L = 4.0 \text{ m}$

Parameter for compressive resistance:

$n = 1.34$

Euler buckling stress:

$$F_{e,z} = \frac{\pi^2 E}{\left(\frac{L_{eff}}{r_z}\right)^2} = \frac{\pi^2 \cdot 200{,}000}{\left(\frac{4{,}000}{3.9}\right)^2} = 190.51 \text{ MPa}$$

Slenderness:

$$\lambda_z = \sqrt{\frac{F_y}{F_{e,z}}} = \sqrt{\frac{260}{190.51}} = 1.17$$

Factored axial compressive resistance:

$$C_{r,z} = \frac{\phi A F_y}{\left(1 + \lambda_z^{2n}\right)^{1/n}} = \frac{0.9 \cdot 6{,}670 \cdot 260}{\left(1 + 1.17^{2 \cdot 1.34}\right)^{1/1.34}} = 783.76 \text{ kN}$$

Design ratio:

$$\eta = \frac{C_f}{C_{r,z}} = \frac{300.00}{783.76} = 0.38 \leq 1.00 \qquad \Rightarrow \textbf{ OK – governing for flexural buckling}$$

Buckling design in RF-/STEEL CSA

Design Ratio					
Compression Force	C_f	300.00	kN		
Effective Member Length	$L_{eff,z}$	4.000	m		
Yield Strength	F_y	260.00	MPa		
Modulus of Elasticity	E	200000.00	MPa		
Slenderness Ratio	K_zL/r_z	101.790			
Euler Buckling Stress	$F_{e,z}$	190.51	MPa		13.3.1
Gross Area	A_g	66.70	cm^2		
Parameter for Compressive Resistance	n	1.340			13.3.1
Resistance Factor	Φ	0.900			13.1
Factored Compressive Resistance	$C_{r,z}$	783.76	kN		13.3.1
Design Ratio	η_z	0.38		≤ 1	

Figure 8.5: Design details in RF-/STEEL CSA (check no. 311)

8.6 Lateral-Torsional Buckling

Effective length:

$L_{eff} = K_y L_{LTB} = 4.0 \text{ m}$

Coefficient to account for increased moment resistance:

$$\omega_2 = \frac{4 M_{max}}{\sqrt{M_{max}^2 + 4 M_a^2 + 7 M_b^2 + 4 M_c^2}} = \frac{4 \cdot 10}{\sqrt{10^2 + 4 \cdot 7.5^2 + 7 \cdot 10^2 + 4 \cdot 7.5^2}} = 1.13$$

Critical elastic moment:

$$M_u = \frac{\omega_2 \pi}{L_{LTB}} \sqrt{E I_z G J + \left(\frac{\pi E}{L_{LT}}\right)^2 I_z C_w}$$

$$= \frac{1.13 \cdot \pi}{4{,}000} \sqrt{200{,}000 \cdot 10{,}300{,}000 \cdot 308{,}000 \cdot 77{,}000 + \left(\frac{\pi \cdot 200{,}000}{4{,}000}\right)^2 \cdot 10{,}300{,}000 \cdot 2.38 \cdot 10^{11}}$$

$= 293.82 \text{ kNm}$

Plastic moment resistance:

$M_p = 218.66 \text{ kNm}$

$M_u = 293.82 \text{ kNm} \geq 0.67 \cdot M_p = 146{,}50 \text{ kNm}$

Factored moment resistance:

$$M_r = 1.15 \phi M_p \left(1 - \frac{0.28 M_p}{M_u}\right) \leq \phi M_p$$

$$M_r = 1.15 \cdot 0.9 \cdot 218.66 \left(1 - \frac{0.28 \cdot 218.66}{293.82}\right) \leq 0.9 \cdot 218.66$$

$M_r = 179.16 \text{ MPa} \leq 196.79 \text{ MPa}$

Design ratio:

$$\eta = \frac{M_{f,y}}{M_r} = \frac{10.00}{179.16} = 0.06 \leq 1.00 \qquad \Rightarrow \textbf{ OK}$$

Lateral-torsional buckling design in RF-/STEEL CSA

Design Ratio					
Maximum Moment	$M_{y,max}$	10.00	kNm		
Coefficient	ω_2	1.131			13.6
Length of the Segment	L	4.000	m		
Modulus of Elasticity	E	200000.00	MPa		
Moment of Inertia	I_z	1030.00	cm^4		
Shear Modulus	G	77000.00	MPa		
Warping Torsional Constant	C_w	238000.00	cm^6		
St. Venant Torsional Constant	J	30.80	cm^4		
Resistance Factor	Φ	0.900			13.1
Elastic Critical Moment	M_u	293.82	kNm		13.6
Plastic Bending Moment	$M_{p,y}$	218.66	kNm		
Moment Resistance	$M_{r,y}$	179.16	kNm	$M_u > 0{,}67 M_{p,y}$	13.6
Design Ratio	η_{LTB}	0.06		≤ 1	

Figure 8.6: Design details in RF-/STEEL CSA (check no. 325)

8.7 Member Strength and Stability

Finally, the capacity to resist both bending moments and the axial compressive force is examined. The design follows the rules specified in [1] Clause 13.8.2.

8.7.1 Cross-Sectional Strength

Coefficient for bending in beam-columns:

$\beta = 0.6$

Factors to account for moment gradient and second-order effects:

$$U_{1,y} = \frac{\omega_{1,y}}{1 - \frac{C_f}{C_{e,y}}} = \frac{1.0}{1 - \frac{300}{14{,}680.20}} = 1.02$$

where

$\omega_{1,y} = 1.0$ for members subjected to distributed loads

$$C_{e,y} = \frac{\pi^2 E I_y}{L^2} = \frac{\pi^2 \cdot 200{,}000 \cdot 119{,}000{,}000}{4{,}000^2} = 14{,}680.20 \text{ kN}$$

$$U_{1,z} = \frac{\omega_{1,z}}{1 - \frac{C_f}{C_{e,z}}} = \frac{0.85}{1 - \frac{300}{1{,}270.63}} = 1.11$$

where

$\omega_{1,z} = 0.85$ for members subjected to a concentrated load

$$C_{e,z} = \frac{\pi^2 E I_z}{L^2} = \frac{\pi^2 \cdot 200{,}000 \cdot 10{,}300{,}000}{4{,}000^2} = 1{,}270.63 \text{ kN}$$

Compressive resistance:

$C_r = \phi A_g F_y = 0.9 \cdot 6{,}670 \cdot 260 = 1{,}560.78$ kN

Moment resistances (Class 1):

$M_{r,y} = \phi Z_y F_y = 0.9 \cdot 841{,}000 \cdot 260 = 196.79$ kNm

$M_{r,z} = \phi Z_z F_y = 0.9 \cdot 189{,}000 \cdot 260 = 44.22$ kNm

Design according to [1] Clause 13.8.2 a):

$$\frac{C_f}{C_r} + \frac{0.85\, U_{1,y} M_{f,y}}{M_{r,y}} + \frac{\beta\, U_{1,z} M_{f,z}}{M_{r,z}} \le 1.0$$

$$\frac{300}{1{,}560.78} + \frac{0.85 \cdot 1.02 \cdot 10}{196.79} + \frac{0.6 \cdot 1.11 \cdot 7.5}{44.22} = 0.19 + 0.05 + 0.16 \le 1.0$$

Design ratio:

$0.34 \le 1.00 \quad \Rightarrow$ **OK**

Cross-sectional strength design in RF-/STEEL CSA

Design Ratio					
Compression Force	C_f	300.00	kN		
Yield Strength	F_y	260.00	MPa		
Gross Area	A_g	66.70	cm^2		
Resistance Factor	Φ	0.900			13.1
Factored Compressive Resistance	C_r	1560.78	kN		13.3.1
Design Ratio	η_c	0.19		≤ 1	
Bending Moment	$M_{f,y}$	10.00	kNm		
Plastic Section Modulus	Z_y	841.00	cm^3		
Plastic Bending Moment	$M_{p,y}$	218.66	kNm		
Coefficient	$\omega_{1,y}$	1.000			13.8.5
Factor	$U_{1,y}$	1.021			13.8.4
Moment Resistance	$M_{r,y}$	196.79	kNm		13.5
Design Ratio	η_y	0.05		≤ 1	
Bending Moment	$M_{f,z}$	7.50	kNm		
Plastic Section Modulus	Z_z	189.00	cm^3		
Plastic Bending Moment	$M_{p,z}$	49.14	kNm		
Coefficient	$\omega_{1,z}$	0.850			13.8.5
Coefficient	β	0.600			13.8.2
Factor	$U_{1,z}$	1.113			13.8.4
Moment Resistance	$M_{r,z}$	44.23	kNm		13.5
Design Ratio	η_z	0.17		≤ 1	
Design Ratio	η	0.34		≤ 1	

Figure 8.7: Design details in RF-/STEEL CSA (check no. 202)

8.7.2 Overall Member Strength

Coefficient for bending in beam-columns:

$$\beta = 0.6 + 0.4\,\lambda_z = 0.6 + 0.4 \cdot 1.17 = 1.07 > 0.85 \;\rightarrow \beta = 0.85$$

where

$$\lambda_z = \sqrt{\frac{F_y}{F_{e,z}}} = \sqrt{\frac{260}{190.51}} = 1.17$$ (see Chapter 8.5.2)

The values of the factors, U_1, are the same as for the cross-sectional strength (see Chapter 8.7.1):

$$U_{1,y} = 1.02$$

$$U_{1,z} = 1.11$$

Compressive resistance:

$$C_r = \min\left(C_{r,y}; C_{r,z}\right) = \min\left(1{,}497.38; 783.76\right) = 783.76 \text{ kN}$$ (see Chapters 8.5.1 and 8.5.2)

Moment resistances (Class 1):

$$M_{r,y} = \phi\, Z_y\, F_y = 0.9 \cdot 841{,}000 \cdot 260 = 196.79 \text{ kNm}$$

$$M_{r,z} = \phi\, Z_z\, F_y = 0.9 \cdot 189{,}000 \cdot 260 = 44.22 \text{ kNm}$$

Design according to [1] Clause 13.8.2 b):

$$\frac{C_f}{C_r} + \frac{0.85\,U_{1,y}\,M_{f,y}}{M_{r,y}} + \frac{\beta\,U_{1,z}\,M_{f,z}}{M_{r,z}} \le 1.0$$

$$\frac{300}{783.76} + \frac{0.85 \cdot 1.02 \cdot 10}{196.79} + \frac{0.85 \cdot 1.11 \cdot 7.5}{44.22} = 0.38 + 0.05 + 0.16 \le 1.0$$

Design ratio:

$0.59 \le 1.00 \quad \Rightarrow$ **OK**

Overall member strength design in RF-/STEEL CSA

Design Ratio					
Compression Force	C_f	300.00	kN		
Yield Strength	F_y	260.00	MPa		
Gross Area	A_g	66.70	cm^2		
Resistance Factor	Φ	0.900			13.1
Factored Compressive Resistance	C_r	783.76	kN		13.3.1
Design Ratio	η_c	0.38		≤ 1	
Maximum Moment	$M_{y,max}$	10.00	kNm		
Plastic Section Modulus	Z_y	841.00	cm^3		
Plastic Bending Moment	$M_{p,y}$	218.66	kNm		
Coefficient	$\omega_{1,y}$	1.000			13.8.5
Factor	$U_{1,y}$	1.021			13.8.4
Moment Resistance	$M_{r,y}$	196.79	kNm		13.5
Design Ratio	η_y	0.05		≤ 1	
Maximum Moment	$M_{z,max}$	7.50	kNm		
Plastic Section Modulus	Z_z	189.00	cm^3		
Plastic Bending Moment	$M_{p,z}$	49.14	kNm		
Coefficient	$\omega_{1,z}$	0.850			13.8.5
Coefficient	β	0.850			13.8.2
Factor	$U_{1,z}$	1.113			13.8.4
Moment Resistance	$M_{r,z}$	44.23	kNm		13.5
Design Ratio	η_z	0.17		≤ 1	
Design Ratio	η	0.59		≤ 1	13.8.2.(b)

Figure 8.8: Design details in RF-/STEEL CSA (check no. 333)

8.7.3 Lateral-Torsional Buckling Strength

Coefficient for bending in beam-columns:

$$\beta = 0.6 + 0.4\,\lambda_z = 0.6 + 0.4 \cdot 1.17 = 1.07 > 0.85 \;\rightarrow \beta = 0.85$$

where

$$\lambda_z = \sqrt{\frac{F_y}{F_{e,z}}} = \sqrt{\frac{260}{190.51}} = 1.17$$ (see Chapter 8.5.2)

The values of the factors, U_1, are the same as for the cross-sectional strength (see Chapter 8.7.1):

$U_{1,y} = 1.02$

$U_{1,z} = 1.11$

Compressive resistance:

$C_r = \min\left(C_{r,y}; C_{r,z}\right) = \min\left(1{,}497.38; 783.76\right) = 783.76$ kN (see Chapters 8.5.1 and 8.5.2)

Moment resistances:

$M_{r,y} = 179.16$ kNm (see Chapter 8.6)

$M_{r,z} = \phi Z_z F_y = 0.9 \cdot 189{,}000 \cdot 260 = 44.22$ kNm

Design according to [1] Clause 13.8.2 c):

$$\frac{C_f}{C_r} + \frac{0.85\,U_{1,y}\,M_{f,y}}{M_{r,y}} + \frac{\beta\,U_{1,z}\,M_{f,z}}{M_{r,z}} \le 1.0$$

$$\frac{300}{783.76} + \frac{0.85 \cdot 1.02 \cdot 10}{179.16} + \frac{0.85 \cdot 1.11 \cdot 7.5}{44.22} = 0.38 + 0.05 + 0.16 \le 1.0$$

Design ratio:

$0.59 \le 1.00 \quad \Rightarrow$ **OK**

Lateral-torsional buckling strength design in RF-/STEEL CSA

Design Ratio					
Compression Force	C_f	300.00	kN		
Yield Strength	F_y	260.00	MPa		
Gross Area	A_g	66.70	cm^2		
Resistance Factor	Φ	0.900			13.1
Factored Compressive Resistance	C_r	783.76	kN		13.3.1
Design Ratio	η_c	0.38		≤ 1	
Maximum Moment	$M_{y,max}$	10.00	kNm		
Coefficient	ω_2	1.131			13.6
Length of the Segment	L	4.000	m		
Modulus of Elasticity	E	200000.00	MPa		
Moment of Inertia	I_z	1030.00	cm^4		
Shear Modulus	G	77000.00	MPa		
Warping Torsional Constant	C_w	238000.00	cm^6		
St. Venant Torsional Constant	J	30.80	cm^4		
Resistance Factor	Φ	0.900			13.1
Elastic Critical Moment	M_u	293.82	kNm		13.6
Plastic Bending Moment	$M_{p,y}$	218.66	kNm		
Factor	$U_{1,y}$	1.021			13.8.4
Moment Resistance	$M_{r,y}$	179.16	kNm	$M_u > 0,$	13.6
Design Ratio	η_{LTB}	0.06		≤ 1	
Maximum Moment	$M_{z,max}$	7.50	kNm		
Plastic Section Modulus	Z_z	189.00	cm^3		
Plastic Bending Moment	$M_{p,z}$	49.14	kNm		
Coefficient	β	0.850			13.8.2
Factor	$U_{1,z}$	1.113			13.8.4
Moment Resistance	$M_{r,z}$	44.23	kNm		13.5
Design Ratio	η_z	0.17		≤ 1	
Design Ratio	η	0.59		≤ 1	13.8.2.(c)

Figure 8.9: Design details in RF-/STEEL CSA (check no. 335)

In addition, the following requirement is to be met:

$$\frac{M_{f,y}}{M_{r,y}} + \frac{M_{f,z}}{M_{r,z}} \leq 1.0$$

$$\frac{10}{196.79} + \frac{7.5}{44.22} = 0.05 + 0.17 \leq 1.0$$

Design ratio:

$0.22 \leq 1.00 \quad \Rightarrow$ **OK**

Design Ratio					
Bending Moment	$M_{f,y}$	10.00	kNm		
Yield Strength	F_y	260.00	MPa		
Resistance Factor	Φ	0.900			13.1
Plastic Section Modulus	Z_y	841.00	cm^3		
Plastic Bending Moment	$M_{p,y}$	218.66	kNm		
Moment Resistance	$M_{r,y}$	196.79	kNm		13.5
Design Ratio	η_y	0.05		≤ 1	
Bending Moment	$M_{f,z}$	7.50	kNm		
Plastic Section Modulus	Z_z	189.00	cm^3		
Plastic Bending Moment	$M_{p,z}$	49.14	kNm		
Moment Resistance	$M_{r,z}$	44.23	kNm		13.5
Design Ratio	η_z	0.17		≤ 1	
Design Ratio	η	0.22		≤ 1	

Figure 8.10: Design details in RF-/STEEL CSA (check no. 203)

© Dlubal Software 2017

Literature

[1] *CSA S16: Design of steel structures.* Canadian Standards Association, Ontario, 2014.

© Dlubal Software 2017

Index

A
Axis . . . 16

B
Background graphic . . . 37
Beam type . . . 22
Buckling . . . 15, 16
Buttons . . . 36

C
Calculation . . . 23
Cantilever . . . 22, 23
Clipboard . . . 50
Color bars . . . 36
Color scale . . . 41
Comment . . . 6, 17
Compressive resistance . . . 24
Control Panel . . . 41
Coordinate system . . . 10
Critical elastic moment . . . 24
Cross-section . . . 10, 34, 47
Cross-section design . . . 28
Cross-section info . . . 12
Cross-section library . . . 10
Cross-section optimization . . . 24, 47
Cross-section type . . . 11

D
Decimal places . . . 8, 49
Deflection . . . 7, 23
Deformation analysis . . . 7, 22
Design . . . 7, 26, 27, 29, 30, 32
Design case . . . 38, 45, 46
Design colored . . . 41
Design ratio . . . 11, 27, 28
Detail settings . . . 23
Display navigator . . . 38, 41

E
Effective length . . . 14, 15, 17
Excel . . . 51
Exit RF-/STEEL CSA . . . 5
Export . . . 50
Export cross-section . . . 48
Export effective length . . . 50
Export material . . . 50

F
Filter . . . 36, 41
Filtering members . . . 41
Flexural buckling . . . 13, 15

G
General data . . . 5
Graphic printout . . . 44
Graphics . . . 38

H
Hinge . . . 21

I
Installation . . . 3
Intermediate lateral restraints . . . 13
Internal forces . . . 31, 48

L
Lateral restraints . . . 13
Lateral-torsional buckling . . . 13, 16, 24
Length . . . 14, 34
Limit values . . . 24
List of member . . . 22
Load application . . . 24
Load case . . . 7, 31
Load combination . . . 7
Loading . . . 31
Location x . . . 27

M
Material . . . 8, 50
Material description . . . 8
Material library . . . 9
Material properties . . . 8, 9
Member . . . 6
Member slenderness . . . 33
Mode shape . . . 36, 42

N
Navigator . . . 5
Nodal support . . . 18

O
Optimization . . . 11, 24, 47, 48

P
Panel . . . 39, 41
Parametric cross-section . . . 47
Part . . . 34
Parts list . . . 34, 35

© DLUBAL SOFTWARE 2017

Precamber 22, 24
Print graphic 44
Printout report 43, 44
Program start 4

R

Reference length 7
Relatively 13
Remark 11
Rendering 41
Result combination 7, 36
Result diagram 40
Result values 38
Result window 26
Results display 38
Results evaluation 36
RF-STABILITY 15
RFEM/RSTAB graphic 44
RSBUCK 15

S

Selecting windows 5
Serviceability limit state 7, 22, 36
Set of members 6, 17, 18, 21, 22, 29, 32, 35
Shifted members ends 23
Slenderness 33
Slenderness ratio 23
Special cases 24
Stability analysis 15, 28
Stainless steel 9
Start calculation 25
Start RF-/STEEL CSA 4
Stress point 12
Structure type 24
Sum 35
Surface area 34

T

Taper 11, 28, 48
Torsional buckling 16
Transverse load 24

U

Ultimate limit state 6, 36
Unbraced frame 24
Undeformed system 23
Units 8, 49
User profile 49

V

View mode 37
Visibility 41
Volume 35

X

x-Location 27, 31

W

Warp spring 20
Weight 35
Window 5
Work window 37, 38, 41

© Dlubal Software 2017