Version
June 2013

Add-on Module

RF-STEEL SIA

Ultimate Limit State Design, Serviceability Limit State Design, Stability Analysis According to SIA 263:2003

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Add-on Module RF-STEEL SIA

The RFEM add-on module RF-STEEL SIA is a powerful tool for designing steel structures that are modeled by beam elements according to the Swiss standard SIA:2003. You can carry out all typical designs for ultimate limit state, stability and deformation. RF-STEEL SIA is able to take into account the effect of various actions for the ultimate limit state design. Furthermore, you can choose between the interaction formulae mentioned in the standard.

In accordance with SIA 263:2003, the program divides the cross-sections to be designed into the classes 1 to 4. Thus, the limitation of the design capacity and the rotational capacity due to local buckling of cross-section parts is checked. Moreover, RF-STEEL SIA determines the c/t-ratios of the cross-section elements subjected to compression and classifies the cross-sections automatically.

For the stability analysis you can specify separately for each member or set of members whether flexural buckling is possible in y- and/or z-direction. Furthermore, you can define additional lateral supports in order to represent the model close to reality. RF-STEEL SIA determines the slendernesses and ideal buckling loads from the boundary conditions. The ideal moment for lateral buckling required for the lateral buckling analysis can be determined automatically or specified manually. RF-STEEL SIA takes the load application point of transverse loads into account, which is affecting the torsional resistance considerably. Possible torsional instabilities (lateral-torsional buckling) of compression members are not examined, however.

The serviceability limit state represents an important design for structures with slender cross-sections. Load cases, load combinations and result combinations can be assigned to different design situations. The limit deformations are preset according to SIA 260, annex A, but can be adjusted if necessary. In addition, it is possible to specify reference lengths and precambers that are considered accordingly in the design.

If required, RF-STEEL SIA optimizes cross-sections and exports the modified sections to RFEM. With the help of design cases it is possible to design structural components in complex structural systems separately or to analyze variants.

RF-STEEL SIA is an add-on module integrated in RFEM. Thus, the design relevant input data is preset after starting the steel module. Subsequent to the design, you can use the graphical RFEM user interface to evaluate the results. Finally, the design process can be documented in the global printout report of RFEM, from determination of internal forces to design.

We hope you will enjoy working with RF-STEEL SIA.

Your DLUBAL Team

1.2 RF-STEEL SIA - Team

The following people were involved in the development of RF-STEEL SIA:

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem

Programming

Ing. Zdeněk Kosáček
Dipl.-Ing. Georg Dlubal
Dr.-Ing. Jaroslav Lain
Ing. Martin Budáč
Mgr. Petr Oulehle
Zbyněk Zámečník
DiS. Jiří Šmerák

Cross-section and material database

Ing. Ph.D. Jan Rybín
Mgr. Petr Oulehle
Ing. Jiří Kubíček

Program design, dialog figures and icons

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

Program supervision

Ing. Martin Vasek
Dipl.-Ing. (FH) Wieland Götzler

Localization, manual

Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Ing. Jana Duníková
Dipl.-Ing. (FH) René Flori
Ing. Lara Freyer
Alessandra Grosso
Bc. Chelsea Jennings
Jan Jeřábek
Ing. Ladislav Kábrt
Ing. Aleksandra Kociołek
Ing. Roberto Lombino
Eng.º Nilton Lopes
Mgr. Ing. Hana Macková
Ing. Téc. Ind. José Martínez
MA SKT Anton Mitleider
Dipl.-Ü. Gundel Pietzcker
Mgr. Petra Pokorná
Ing. Michaela Prokopová
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl
Ing. Marcin Wardyn

Technical support and quality management

M.Eng. Cosme Asseya
Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. Moritz Bertram
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. (FH) Ulrich Lex
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

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

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

At the end of the manual, you find the index. However, if you don’t find what you are looking for, please check our website www.dlubal.com where you can go through our comprehensive *FAQ* pages by selecting particular criteria.

1.4 Open the Add-on Module RF-STEEL SIA

RFEM provides the following options to start the add-on module RF-STEEL SIA.

Menu

To start the program in the RFEM menu bar, click

Add-on Modules → **Design - Steel** → **RF-STEEL SIA**.

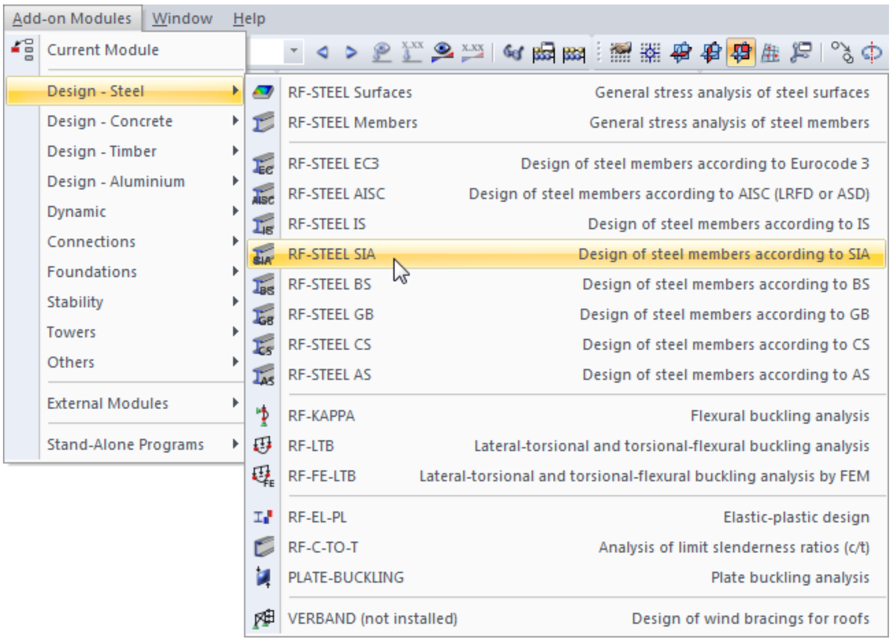


Figure 1.1: Menu: *Add-on Modules → Design - Steel → RF-STEEL SIA*

Navigator

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

Add-on Modules → RF-STEEL SIA.

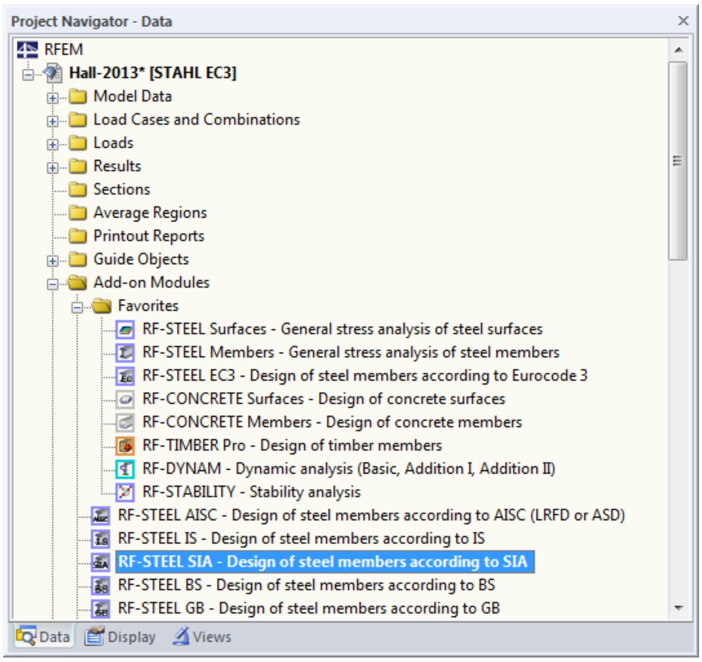
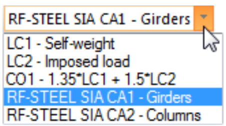
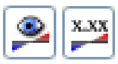


Figure 1.2: Data navigator: *Add-on Modules → RF-STEEL SIA*

Panel





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

Set the relevant RF-STEEL SIA design case in the load case list of the RFEM toolbar. Use the button [Show Results] to display the design criterion on the members graphically.

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

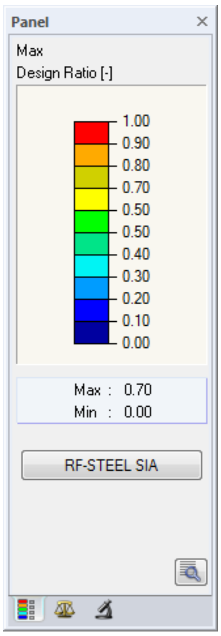


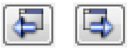
Figure 1.3: Panel button [RF-STEEL SIA]

2. Input Data

When you have started the add-on module, a new window opens. In this window, a Navigator is displayed on the left, managing the module windows that can be currently selected. The drop-down list above the navigator contains the design cases (see chapter 7.1, page 52).

You have to enter and define the design relevant data in several input windows. When you open RF-STEEL SIA for the first time, the following parameters are imported automatically:

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



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

Click [OK] to save the input. You exit RF-STEEL SIA and return to the main program. If you click [Cancel], you exit the module but without saving the data.

2.1 General Data

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

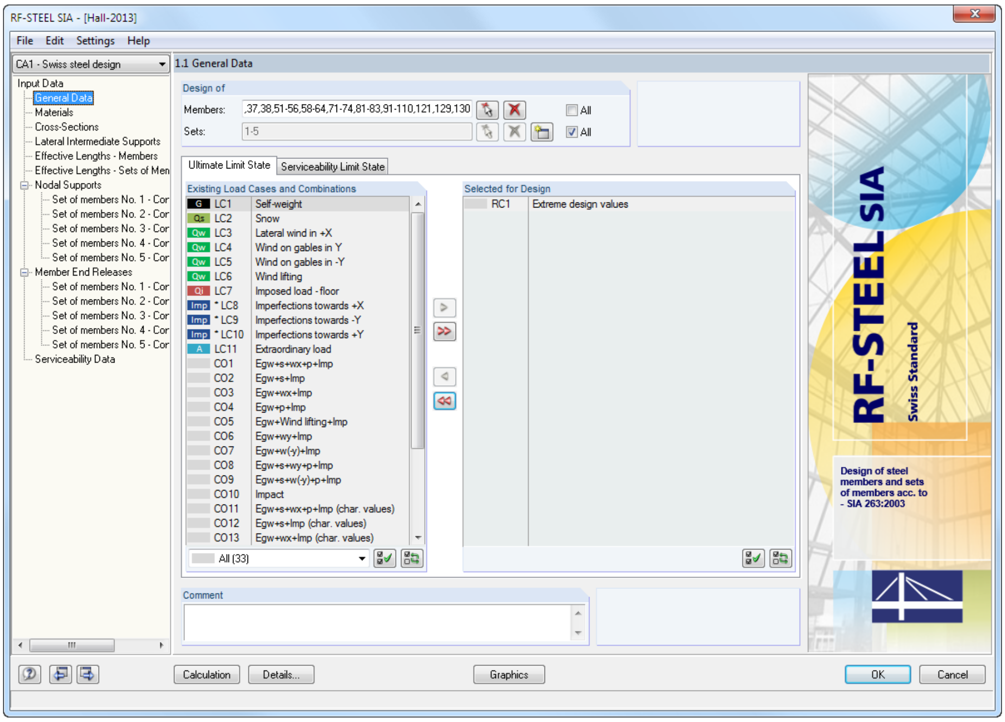


Figure 2.1: Window 1.1 *General Data*

Design of

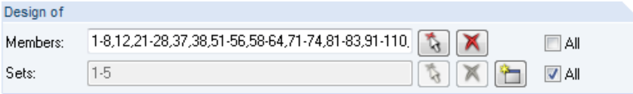


Figure 2.2: Design of members and sets of members

The design can be carried out for *Members* and *Sets* of members. If you want to design only selected objects, clear the *All* check box: Then you can access the input fields to enter the numbers of the relevant members or sets of members. The list of numbers preset in the field can be selected by double-clicking and overwritten by entering the data manually. Click [↖] to select the objects graphically in the RFEM work window.

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

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

Comment

Figure 2.3: User-defined comment

In this input field, you can type user-defined notes describing for example the current design case.

2.1.1 Ultimate Limit State

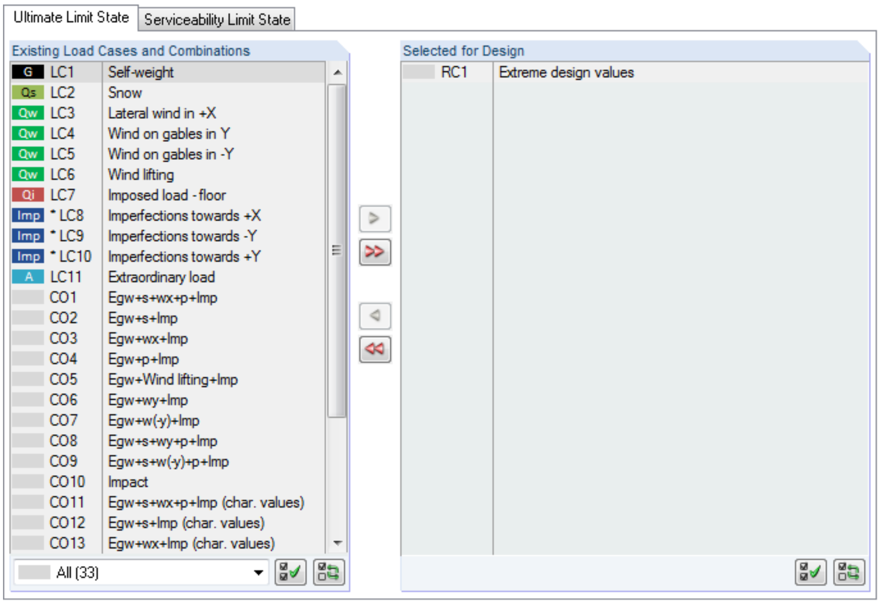


Figure 2.4: Window 1.1 *General Data*, tab *Ultimate Limit State*

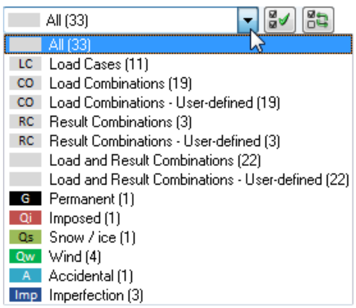
Existing Load Cases and Combinations

In this column, all load cases, load combinations, and result combinations created in RFEM are listed.


Click [▶] to transfer selected entries to the list *Selected for Design* on the right side. You can also double-click the items. To transfer the complete list to the right, click [▶▶].

To transfer multiple entries of load cases, click the entries while pressing the [Ctrl] key, as common for Windows applications. In this way, you can transfer several load cases simultaneously.

Load cases marked by an asterisk (*), like load case 8 in Figure 2.4, cannot be designed: This happens when the load cases are defined without any load data, or the load cases contain only imperfections. When you want to transfer such a load case, a corresponding warning appears.



Several filter options are available below the list. They will help you assign the entries sorted by to load cases, load combinations, or action categories. The buttons have the following functions:

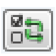
	Selects all cases in the list.
	Inverts selection of load cases.

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

Selected for Design


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

Result combination

The design of an enveloping max/min result combination is performed faster than the design of all contained load cases and combinations. But analyzing a result combination has also disadvantages: First, the influence of the contained loads is difficult to discern. Second, for determining the ideal lateral buckling moment M_{cr} the program analyzes the envelope of the moment distributions from which the more unfavorable distribution (max or min) is applied.

However, this distribution only rarely reflects the moment distribution in the individual load combinations. Thus, in the case of an RC design, more unfavorable values for M_{cr} leading to higher ratios are to be expected.

Result combinations should be selected for design only in case of dynamic combinations. For "usual" combinations, load combinations are recommended because here the actual moment distributions are applied for the determination of M_{cr}.

2.1.2 Serviceability Limit State

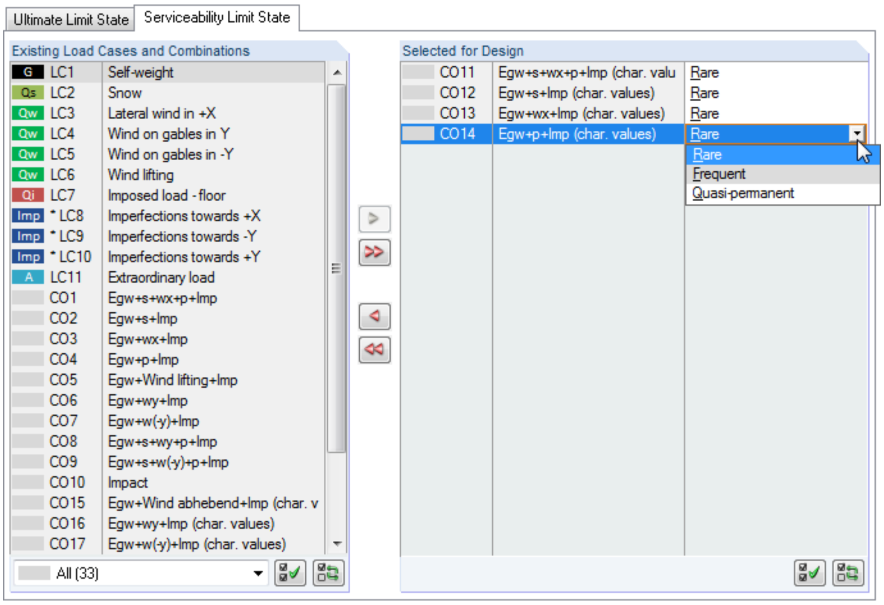


Figure 2.5: Window 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

In this window section, all load cases, load combinations, and result combinations created in RFEM are listed.

Selected for Design

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

It is possible to assign different limit values for deflection to the load cases, load and result combinations. The following design situations can be selected:

- *Rare*
- *Frequent*
- *Quasi-permanent*

To modify the design situation, use the list which can be accessed at the end of the input field by clicking [▼] (see Figure 2.5).

The limit values of the deformations are defined in the *Serviceability* tab of the *Details* dialog box (see Figure 3.3, page 31) where they can be adjusted for the design situations.

In module window 1.9 *Serviceability Data*, the reference lengths that are decisive for the deformation analysis are managed (see chapter 2.9, page 27).

2.2 Materials

The module window is subdivided into two parts. In the upper part, all materials created in RFEM are listed. In the *Material Properties* section below, the properties of the current material whose table row is selected in the upper section, are displayed.

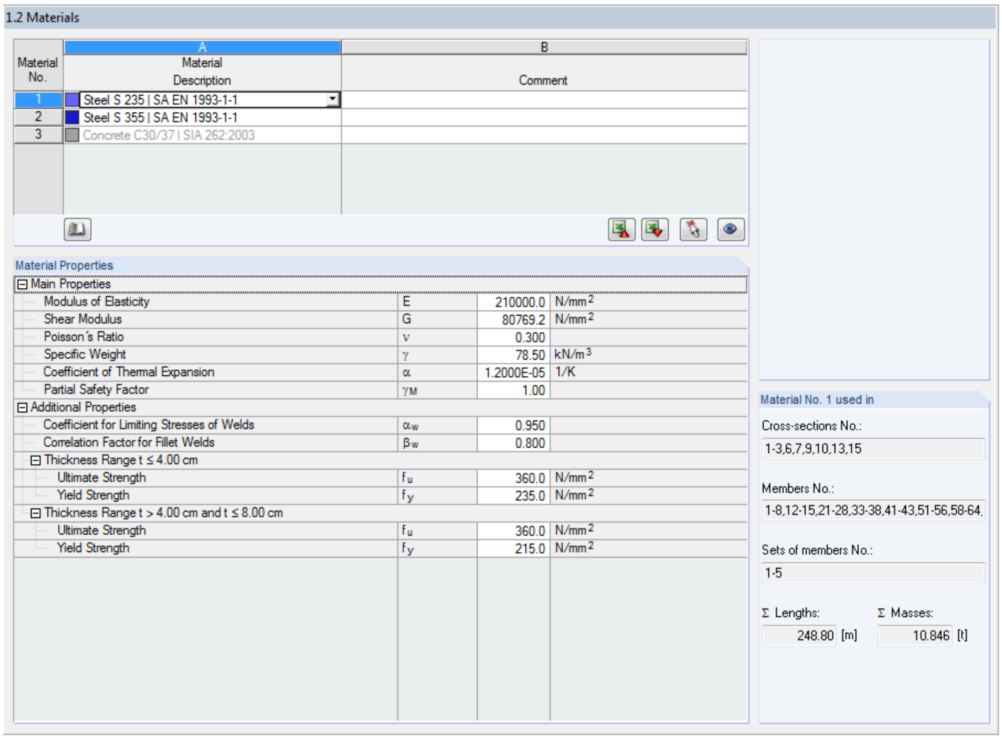


Figure 2.6: Window 1.2 *Materials*

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

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

To adjust the units and decimal places of material properties and stresses, select in the module's menu **Settings** → **Units and Decimal Places** (see chapter 7.3, page 56).

Material Description

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

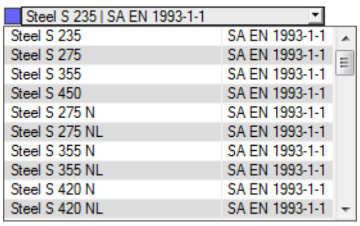


Figure 2.7: List of materials

According to the design concept of the Standard [1], you can select only materials of the "Steel" category.

The design relevant *Material Properties* will be refreshed after the import.

If you change the material description manually, and the entry is stored in the material library, RF-STEEL SIA will import the corresponding material properties.

Editing the material properties in the add- on module RF-STEEL SIA is basically not possible.

Material Library

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

Edit → Material Library

or use the button shown on the left.

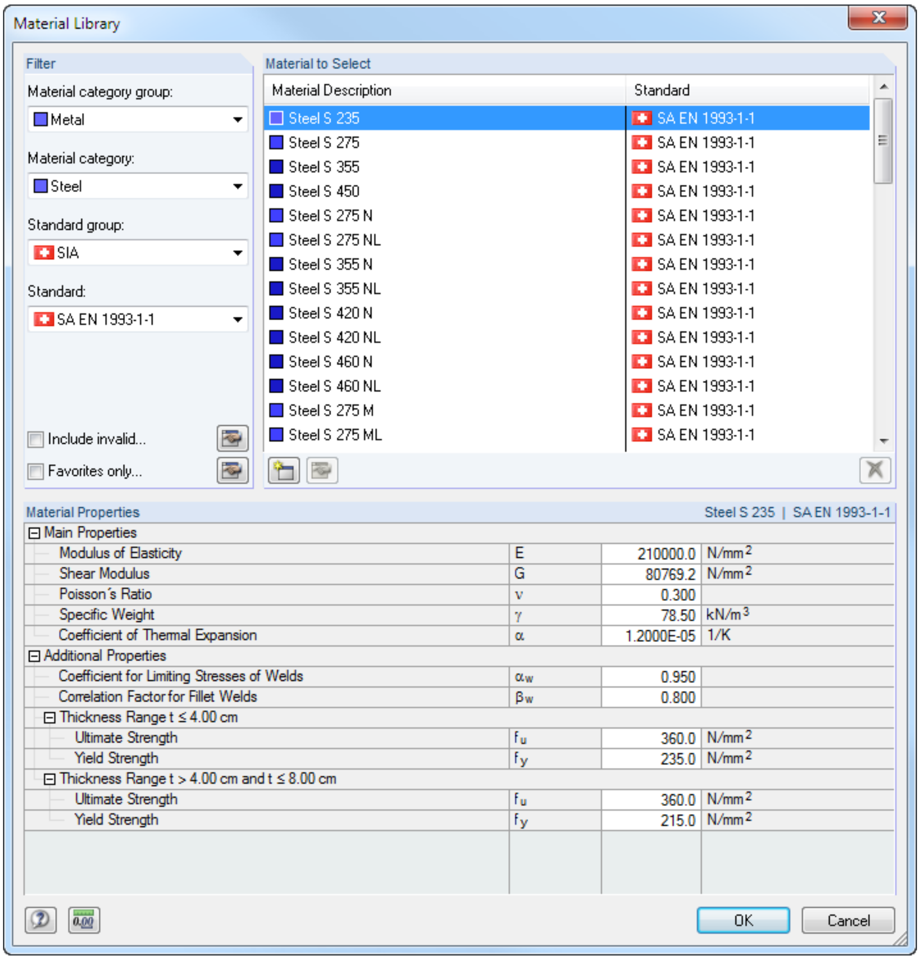


Figure 2.8: Dialog box *Material Library*

In the *Filter* section, *Steel* is preset as material category. In the list *Material to Select* to the right, select the material grade that you want to use for the design. The corresponding properties can be checked in the dialog section below.

Click [OK] or [↵] to transfer the selected material to module window 1.2 of RF-STEEL SIA.

Chapter 4.3 in the RFEM manual describes in detail how materials can be filtered, added, or re-arranged.

2.3 Cross-sections

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

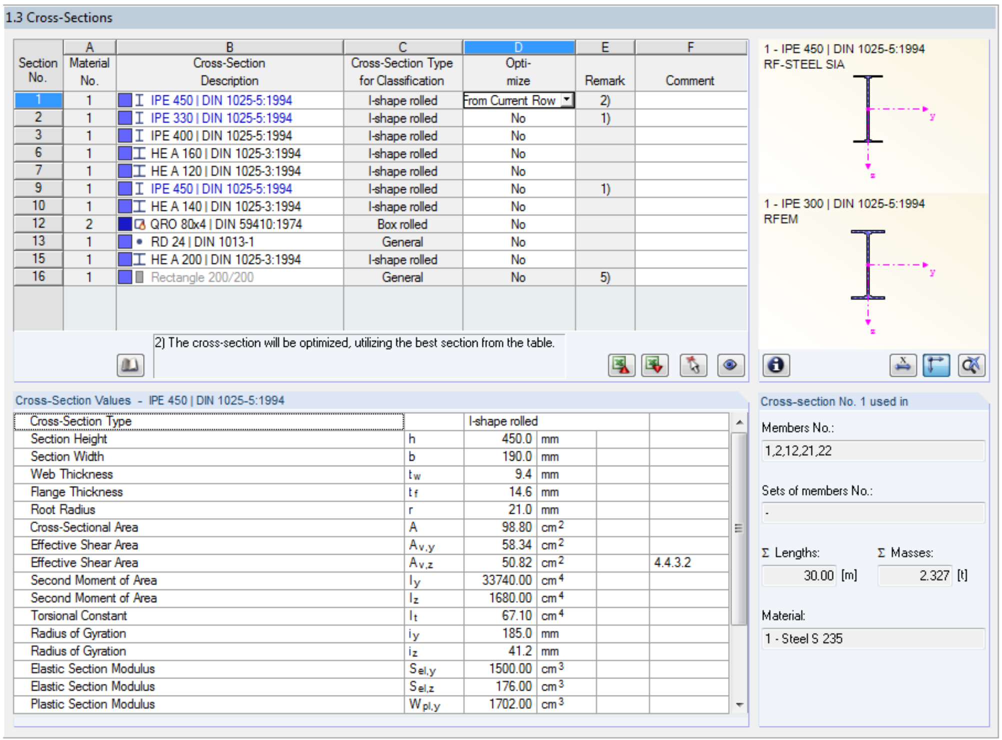


Figure 2.9: Window 1.3 *Cross-Sections*

Cross-Section Description

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

If you want to modify a cross-section, click the entry in table column B to activate its table field. Now, click the button [Import Cross-Section] below, the button [...] in the activated field, or use the function key [F7] to open the cross-section table of the current input field (see the following figure).

A dialog box appears where you can select another cross-section or even a different cross-section table. If you want to use a completely different cross-section category, click the button [Back to Cross-Section Library] to access the general cross-section database.

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

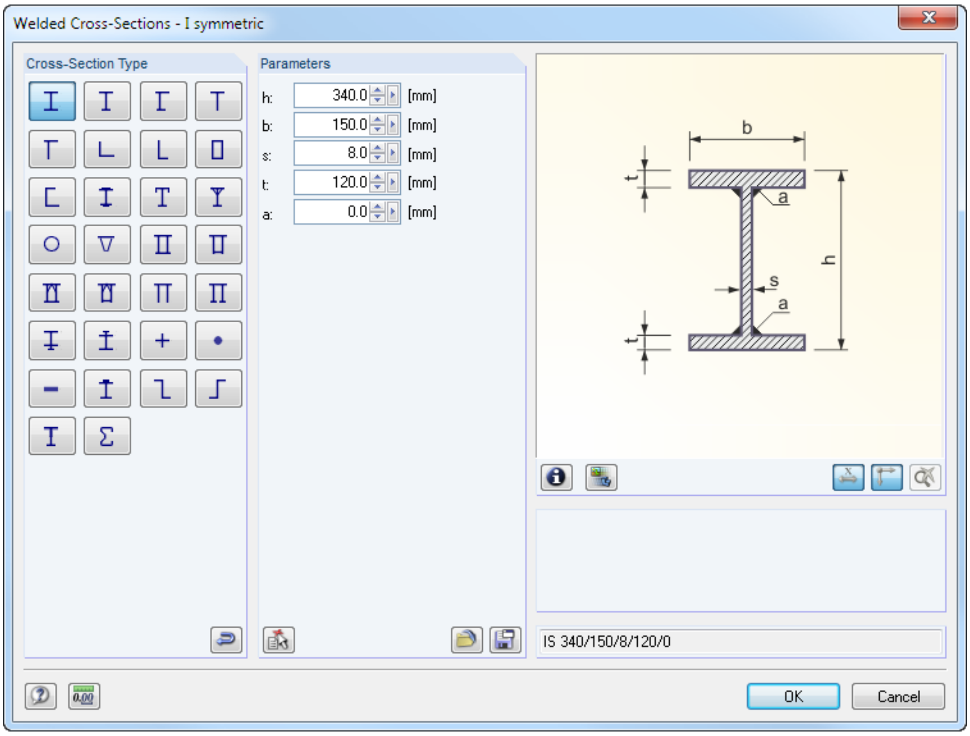
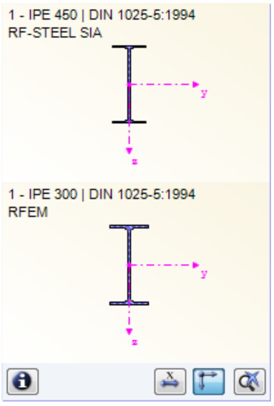


Figure 2.10: IS cross-section table in the cross-section library



It is also possible to enter the new cross-section description directly into the input field. If the entry is recorded in the data base, RF-STEEL SIA imports its cross-section properties.

A modified cross-section will be highlighted in blue.

If cross-sections set in RF-STEEL SIA are different from the ones used in RFEM, both cross-sections are displayed in the graphic on the right in the module window. The designs will be performed with the RFEM internal forces for the cross-section selected in RF-STEEL SIA.

Cross-Section Type for Classification

The cross-section type used for the classification is displayed. The cross-sections listed in [1] table 5a and 5b can be designed plastically or elastically depending on the class. Cross-sections that are not covered by these tables are classified as *General*. They can be designed only elastically (cross-section class 3 or 4).

Max. Design Ratio

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

Optimize

Each cross-section of the library can run through an optimization process: For the RFEM internal forces RF-STEEL SIA finds the cross-section that comes as close as possible to a user-defined maximum ratio. This ratio can be defined in the dialog box *Details*, tab *Other* (see Figure 3.4, page 32).

To optimize a cross-section, open the drop-down list in column D or E and select the desired entry: *From current row* or, if available, *From favorites 'Description'*. Recommendations for the cross-section optimization can be found in chapter 7.2 on page 54.

Remark

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

If the program displays the message *Cross-section No. XX is not permitted!* when starting the [Calculation], you have set a cross-section that is not registered in the cross-section database. This may be a user-defined cross-section, or a SHAPE-THIN section which has not yet been calculated. To select an appropriate cross-section for the design, click [Library] (see description after Figure 2.9).

Member with tapered cross-section

For tapered members with different cross-sections at the member start and member end, the module displays both cross-section numbers in two tables, in accordance with the definition in RFEM.

RF-STEEL SIA also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the member end. The normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the start and the end cross-section of a tapered member do not have the same number of stress points, the intermediate values cannot be interpolated. The calculation is neither possible in RFEM nor in RF-STEEL SIA.

The cross-section's stress points including numbering can be checked graphically: Select the cross-section in module window 1.3 and click the [Info] button. The dialog box shown in Figure 2.11 appears.

Info About Cross-Section

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

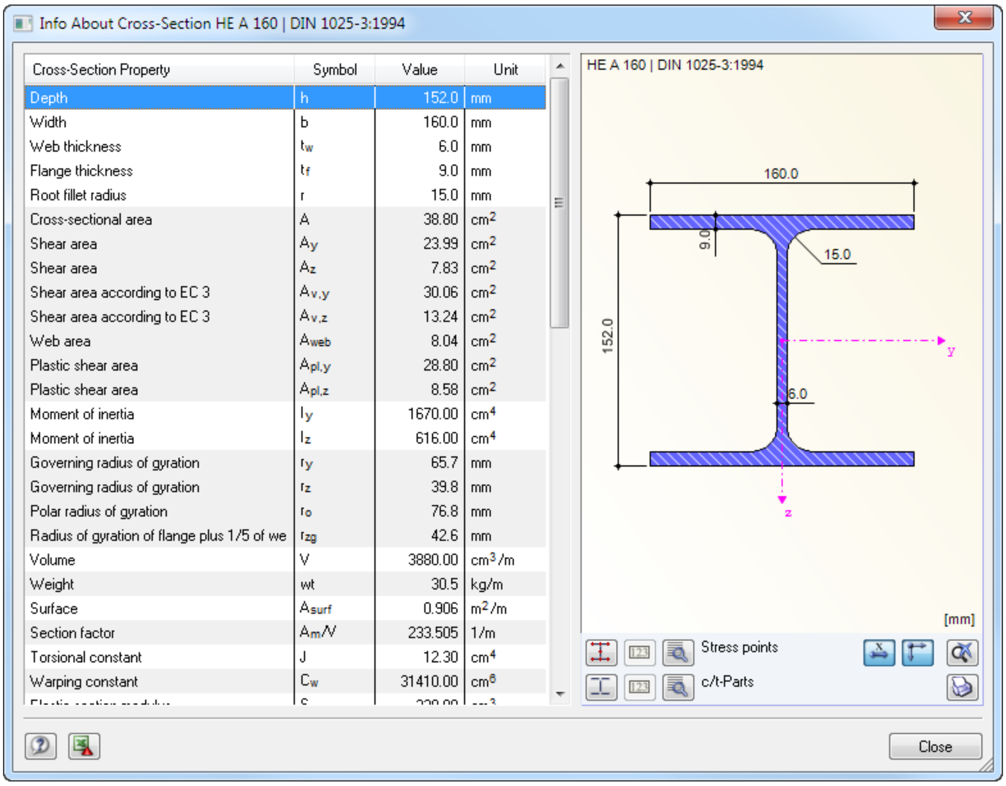


Figure 2.11: Dialog box *Info About Cross-Section*

In the right part of the dialog box, the currently selected cross-section is displayed.

The buttons below the graphic have the following functions:

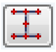
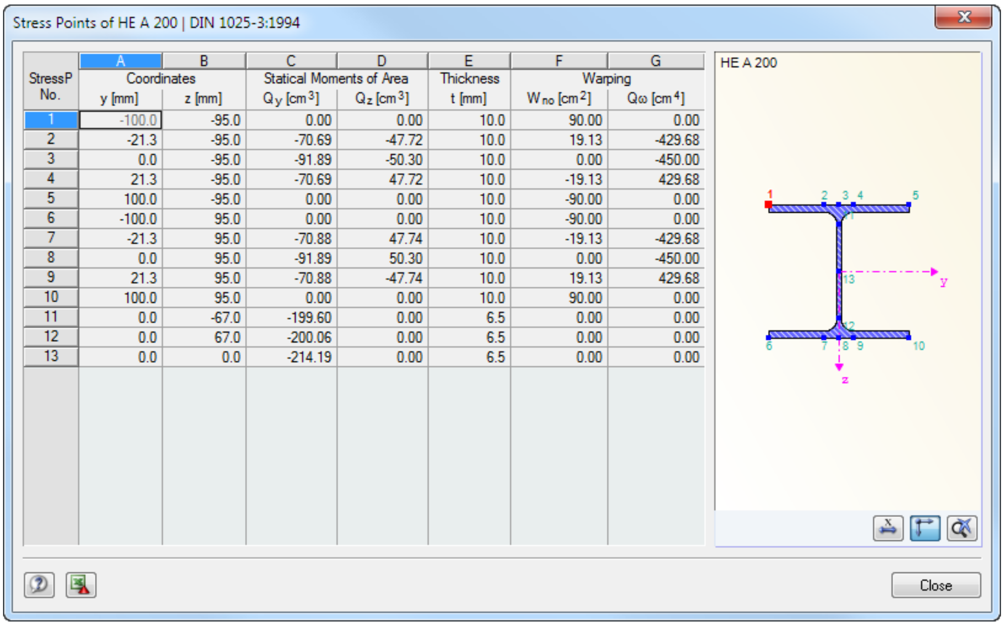
Button	Function
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the numbering of stress points or c/t-parts
	Displays or hides the details of the stress points or c/t-parts (see Figure 2.12)
	Displays or hides the dimensions of the cross-section
	Displays or hides the principal axes of the cross-section
	Resets the full view of the cross-section graphic

Table 2.2: Buttons of cross-section graphic

Click the [Details] button shown on the left to call up detailed information on stress points (distances to center of gravity, statical moments of area, warping ordinates etc.) and c/t-parts.



Stress Points of HE A 200 | DIN 1025-3:1994

StressP No.	A	B	C	D	E	F	G
	Coordinates		Statical Moments of Area		Thickness	Warping	
	y [mm]	z [mm]	Q_y [cm³]	Q_z [cm³]	t [mm]	W_{no} [cm²]	Q_ω [cm⁴]
1	-100.0	-95.0	0.00	0.00	10.0	90.00	0.00
2	-21.3	-95.0	-70.69	-47.72	10.0	19.13	-429.68
3	0.0	-95.0	-91.89	-50.30	10.0	0.00	-450.00
4	21.3	-95.0	-70.69	47.72	10.0	-19.13	429.68
5	100.0	-95.0	0.00	0.00	10.0	-90.00	0.00
6	-100.0	95.0	0.00	0.00	10.0	-90.00	0.00
7	-21.3	95.0	-70.88	47.74	10.0	-19.13	-429.68
8	0.0	95.0	-91.89	50.30	10.0	0.00	-450.00
9	21.3	95.0	-70.88	-47.74	10.0	19.13	429.68
10	100.0	95.0	0.00	0.00	10.0	90.00	0.00
11	0.0	-67.0	-199.60	0.00	6.5	0.00	0.00
12	0.0	67.0	-200.06	0.00	6.5	0.00	0.00
13	0.0	0.0	-214.19	0.00	6.5	0.00	0.00

Figure 2.12: Dialog box *Stress Points of HE A 200*

2.4 Lateral Intermediate Supports

In module window 1.4, you can define lateral intermediate supports for members. RF-STEEL SIA always assumes this kind of support to be perpendicular to the cross-section's minor axis z (see Figure 2.11). Thus, it is possible to influence the members' effective lengths which are important for the stability analyses concerning flexural buckling and lateral buckling.

All lateral intermediate supports are considered as forked supports for the calculation.

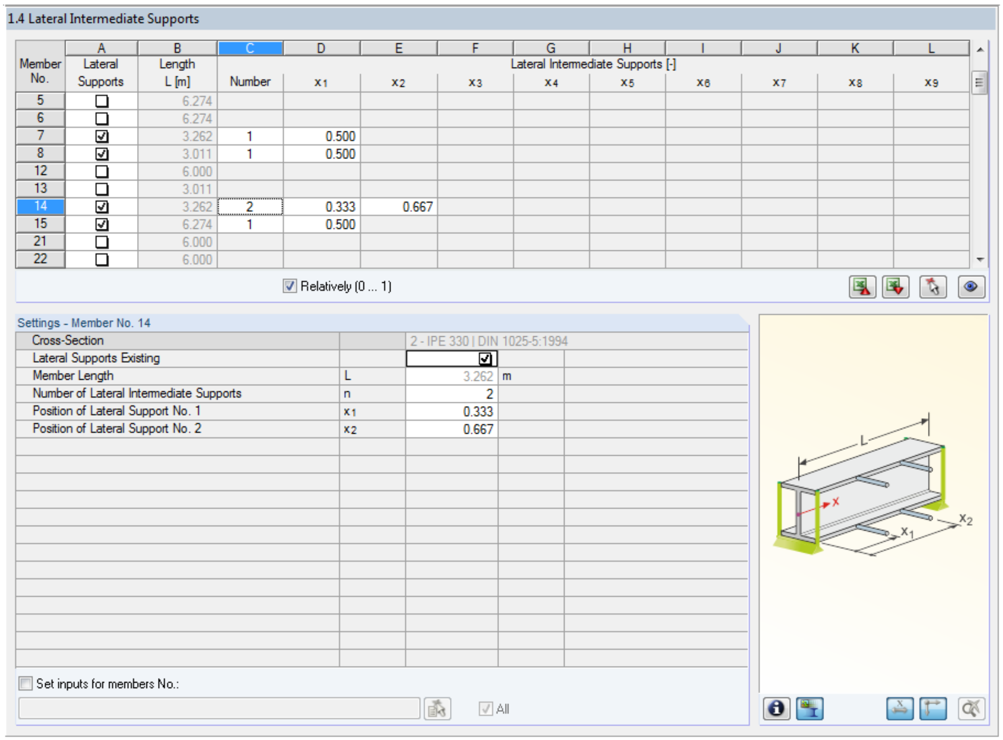


Figure 2.13: Module window 1.4 *Lateral Intermediate Supports*

In the upper part of the window, you can assign up to nine lateral supports for each member. The lower window part shows you a summary of the data entered for the member selected above.

If the check box *Relatively (0 … 1)* is ticked, the support points can be defined by relative-input: The positions of the intermediate supports result from the member length and the relative distances set from the member start. When the check box *Relatively (0 ... 1)* is cleared, you can define the distances also by entering distances in the upper table.

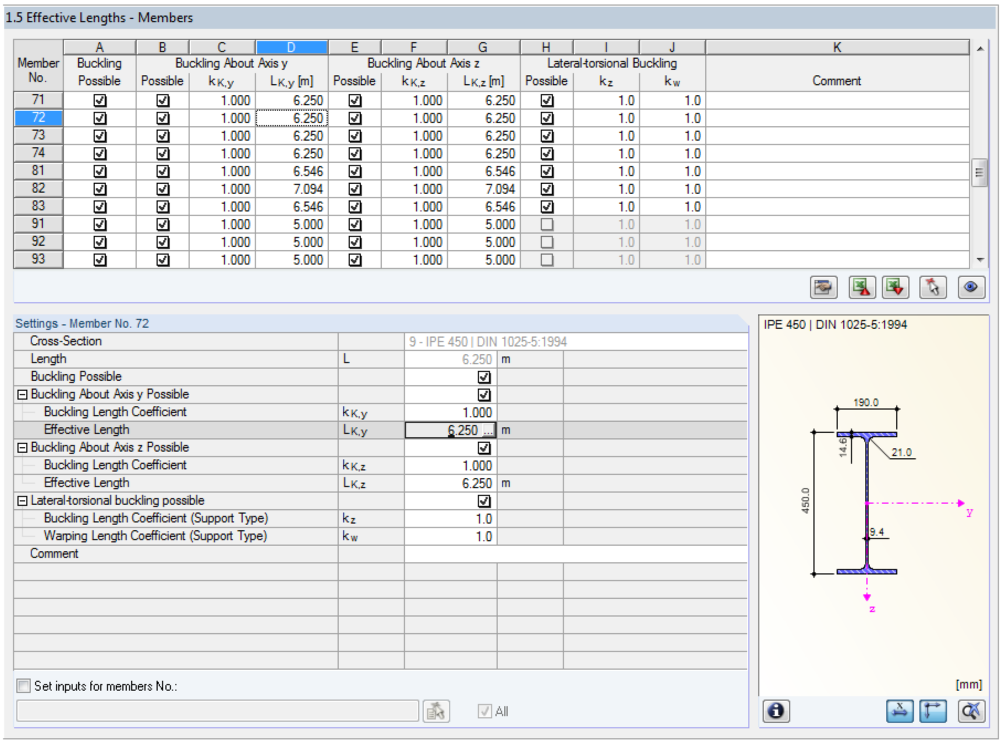
In case of cantilevers, avoid using intermediate supports, as such supports divide the member in segments for the calculation. The result for cantilevered beams would be segments that are forked-supported on one side and thus statically underdetermined (forked support respectively on one end only).

2.5 Effective Lengths - Members

The module window is subdivided into two parts. The table in the upper part contains summarized information about the factors for the lengths of buckling and lateral-torsional buckling as well as the equivalent member lengths of the members to be designed. The effective lengths defined in RFEM are already preset. In the *Settings* part below, further information about the member whose row is selected in the upper window section is shown.

Click the [↖] button for selecting a member graphically to show its row in the module table.

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


1.5 Effective Lengths - Members

Member No.	A Buckling Possible	B Buckling About Axis y Possible	C $k_{K,y}$	D $L_{K,y}$ [m]	E Buckling About Axis z Possible	F $k_{K,z}$	G $L_{K,z}$ [m]	H Lateral-torsional Buckling Possible	I k_z	J k_w	K Comment
71	☑	☑	1.000	6.250	☑	1.000	6.250	☑	1.0	1.0	
72	☑	☑	1.000	6.250	☑	1.000	6.250	☑	1.0	1.0	
73	☑	☑	1.000	6.250	☑	1.000	6.250	☑	1.0	1.0	
74	☑	☑	1.000	6.250	☑	1.000	6.250	☑	1.0	1.0	
81	☑	☑	1.000	6.546	☑	1.000	6.546	☑	1.0	1.0	
82	☑	☑	1.000	7.094	☑	1.000	7.094	☑	1.0	1.0	
83	☑	☑	1.000	6.546	☑	1.000	6.546	☑	1.0	1.0	
91	☑	☑	1.000	5.000	☑	1.000	5.000	☐	1.0	1.0	
92	☑	☑	1.000	5.000	☑	1.000	5.000	☐	1.0	1.0	
93	☑	☑	1.000	5.000	☑	1.000	5.000	☐	1.0	1.0	

Figure 2.14: Module window 1.5 *Effective Lengths - Members*

The effective lengths for buckling about the minor axis z are aligned automatically with the entries in module window 1.4 *Lateral Intermediate Supports*. If lateral intermediate supports are dividing the member into member segments of different lengths, RF-STEEL SIA does not display any value in table columns G, K and L of module window 1.5.

The effective lengths can be entered manually in the table and the *Settings* tree, or defined graphically in the work window by clicking the button [...]. Access to this button is enabled when the cursor is placed in the input field (see figure above).

The *Settings* tree manages the following parameters:

- *Cross-Section*
- Member *Length*
- *Buckling Possible* for member (corresponds to column A)
- *Buckling About Axis y Possible* (corresponds to columns B to D)
- *Buckling About Axis z Possible* (corresponds to columns E to G)
- *Lateral-torsional buckling possible* (corresponds to columns H to J)

For the currently selected member you can specify whether to carry out a buckling or a lateral buckling analysis. In addition, it is possible to adjust the *Buckling Length Coefficient* and the *Warping Length Coefficient* for the respective directions. When a coefficient is modified, the equivalent member length will be adjusted automatically, and vice versa.

The buckling length of a member can also be defined in a separate dialog box that can be accessed by clicking the button shown on the left. The button is located below the table.

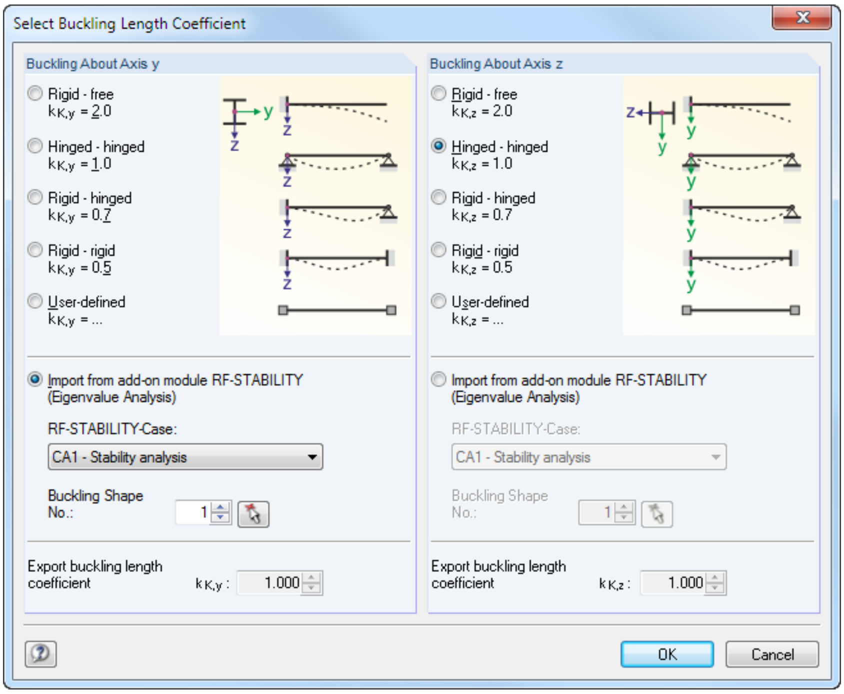


Figure 2.15: Dialog box *Select Buckling Length Coefficient*

For each direction you can select one of the four Euler buckling modes or set a *User-defined* buckling length coefficient. If an eigenvalue analysis has been carried out in the add-on module RF-STABILITY, you can also define a *Buckling Shape* to determine the coefficient.

Buckling possible

The stability analyses for flexural buckling and lateral buckling require the ability of members to absorb compressive forces. Therefore, members for which the absorption is not possible due to the member type (such as tension members, elastic foundations, rigid connections) are excluded from the steel design in the first place. The corresponding rows are grayed out in the table, and a note is indicated in the *Comment* column.

The check boxes for *Buckling Possible* in table A and the *Settings* tree offer you control options for the stability analysis: They determine whether the analysis should or should not be performed for a member.

Buckling about axis y or axis z

With the check boxes in the *Possible* table columns, you decide whether a member has a risk of buckling about axis y and/or z. These axes represent the local member axes, with axis y being the major (strong) and axis z the minor (weak) member axis. The buckling length coefficients $k_{K,y}$ and $k_{K,z}$ for buckling about the major or minor axis can be selected freely.

The position of the member axes can be checked in the cross-section graphic of module window 1.3 *Cross-Sections* (see Figure 2.9, page 14). With a click on the button [Jump to graphic] it is also possible to access the RFEM work window where the local member axes can be displayed with the helpf of the member's context menu or the *Display* navigator.

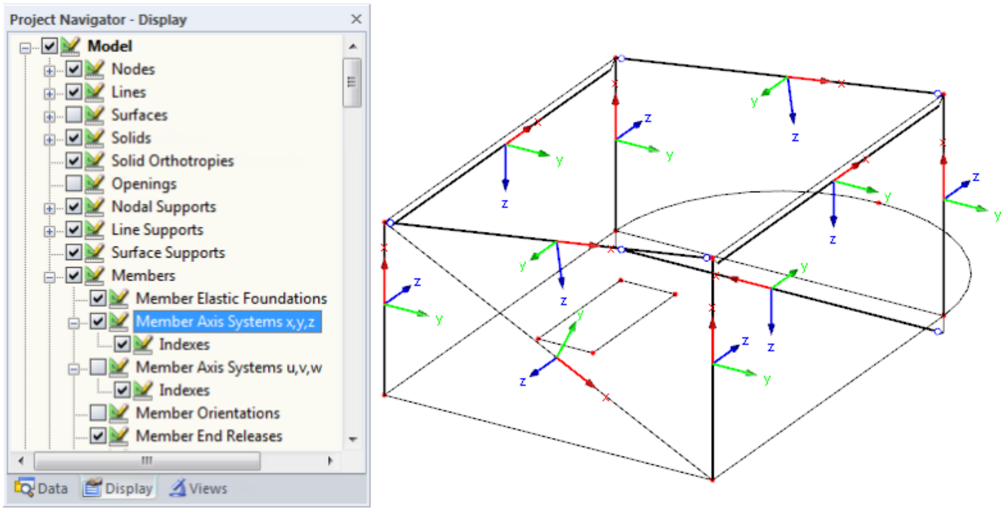


Figure 2.16: Selecting the member axis systems in the *Display* navigator of RFEM

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

To define the buckling lengths graphically in the work window, click the [...] button. The button becomes available when you click in an L_K input field (see Figure 2.14).

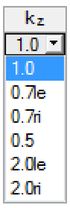
When you enter a value for the buckling length coefficient k_K, RF-STEEL SIA determines the effective length L_K by multiplying the member length L with the coefficient. The input fields k_K and L_K are interactive.

Lateral-torsional buckling possible

Table column H controls the performance of a lateral buckling analysis for the individual members.

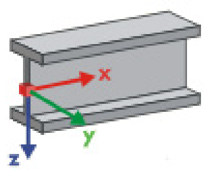
Buckling length coefficient k_z

To determine M_{cr} according to the eigenvalue calculation method, an internal member model with four degrees of freedom is created in the program's background. The following definitions of k_z and k_w (see page 22) are used to represent the degrees of freedom on the supports of the model:



$k_z = 1.0$	forked support on both girder ends
$k_z = 0.7$le	restrained on the left and forked support on the right
$k_z = 0.7$ri	restrained on the right and forked support on the left
$k_z = 0.5$	restraint on both girder ends
$k_z = 2.0$le	restrained on the left and free member end on the right
$k_z = 2.0$ri	restrained on the right and free member end on the left

A forked support with $k_z = 1.0$ results in a support with a retention in direction of the y-axis and a constraint of the torsion about the x-axis (longitudinal axis) of the member. In case of a restraint, the cross-section's torsion is additionally prevented about the z-axis. The abbreviations *le* and *ri* refer to the left and right side. The description *le* always refers to the support conditions at the start of the member.



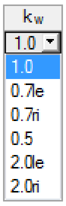
Axis definition for k_z and k_w

As the definitions for k_z and k_w always refer to member start and member end, particular attention must be paid when intermediate supports are taken into account: They divide the member into individual segments for the calculation. Thus, for cantilevered beams the result would be segments that are forked supported on one side and statically underdetermined (forked support respectively on one end only).

Warping length coefficient k_w

With the warping length coefficient k_w you define the fourth degree of freedom on the support, which is also affecting the determination of the ideal moment for lateral buckling M_{cr}. It must be defined whether the cross-section can warp freely (support is free to warp) or if a warping restraint is set.

The definition follows the one of the buckling length coefficient k_z (see above) but now it is a restraint that describes the prevention of warping. By default, RF-STEEL SIA applies the member length as the length for lateral buckling. You can select another support type in the list.



$k_w = 1.0$	support free to warp on both girder ends
$k_w = 0.7le$	restrained on the left and forked support on the right
$k_w = 0.7ri$	restrained on the right and forked support on the left
$k_w = 0.5$	warping restraint on both girder ends
$k_w = 2.0le$	restrained on the left and free member end on the right
$k_w = 2.0ri$	restrained on the right and free member end on the left

As the internal member model needs only four degrees of freedom, a definition of the remaining degrees of freedom (displacement in x- and z-direction) is unnecessary.

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

Comment

In the final table column, you can enter user-defined notes to describe for example the equivalent member lengths.

2.6 Effective Lengths - Sets of Members

Data input in this module window is similar to the one of the previous window 1.5 *Effective Lengths - Members*. It is possible to enter the effective lengths for the buckling about both principal axes of the member set as described in chapter 2.5.

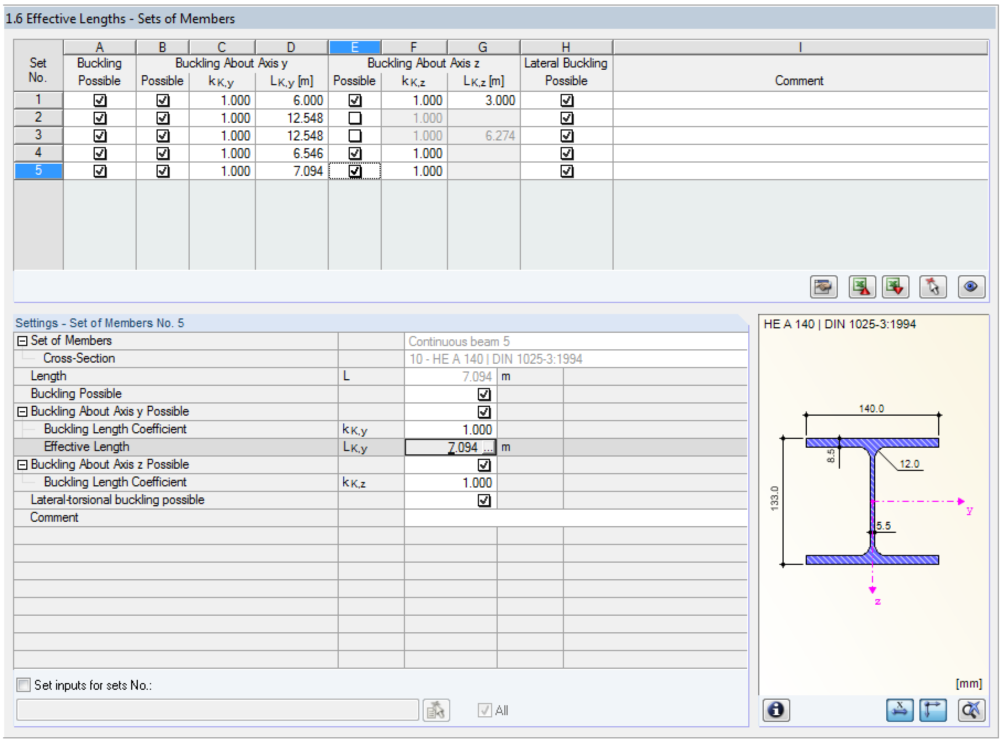


Figure 2.17: Module window 1.6 *Effective Lengths - Sets of Members*

2.7 Nodal Supports - Sets of Members

The module window is displayed only if you have selected at least one set of members for design in window 1.1 *General Data*.

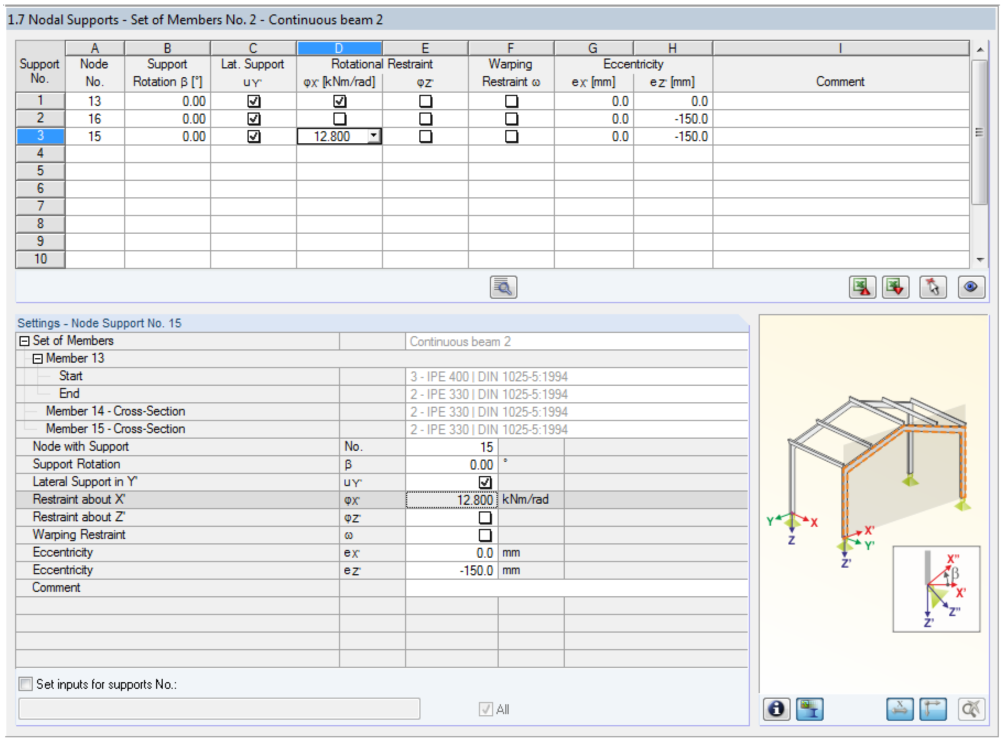


Figure 2.18: Module window 1.7 *Nodal Supports - Set of Members*

Stability analyses for sets of members are carried out according to [1], chapter 4.5.3. The analysis method requires to know the amplification factor α_{Mcr} of the entire member set. To determine the factor, a planar framework with four degrees of freedom per node is created. The degrees must be defined in module window 1.7 whose displayed table refers to the current set of members (selected in the navigator on the left).

The orientation of axes in the set of members is important for the definition of nodal supports. RF-STEEL SIA checks the position of the nodes and defines the axes of the nodal supports internally for window 1.7 in accordance with Figure 2.19 to Figure 2.22.

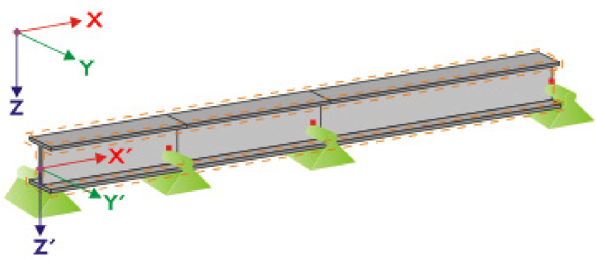


Figure 2.19: Auxiliary coordinate system for nodal supports - straight set of members

If all members of a set of members are lying in a straight line as shown in Figure 2.19, the local coordinate system of the first member in the set of members corresponds to the equivalent coordinate system of the entire set of members.

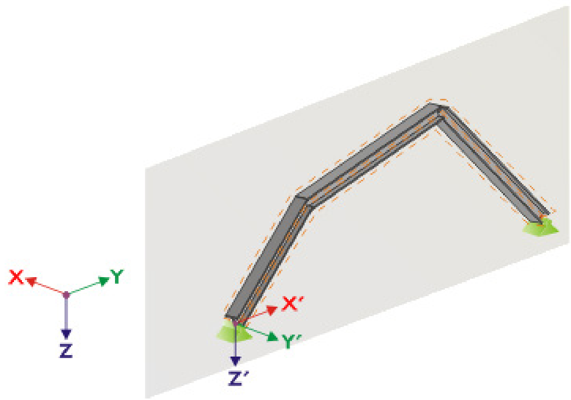


Figure 2.20: Auxiliary coordinate system for nodal supports - set of members in vertical plane

If members of a set of members are not lying in a straight line, they must at least lie in the same plane. In Figure 2.20, they are lying in a vertical plane. In this case, axis X' is horizontal and aligned in direction of the plane. Axis Y' is horizontal as well but defined perpendicular to axis X'. Axis Z' is directed perpendicularly downwards.

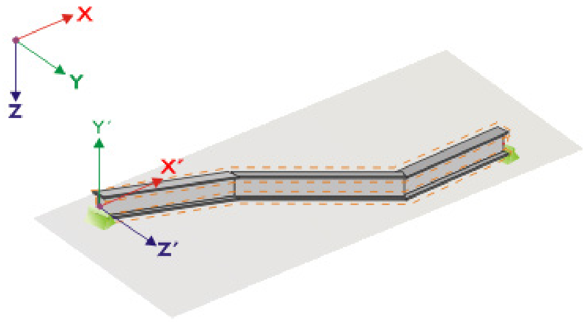


Figure 2.21: Auxiliary coordinate system for nodal supports - set of members in horizontal plane

If members of a buckled set of members are lying in a horizontal plane, axis X' is defined parallel to the X-axis of the global coordinate system. Thus, axis Y' is set in opposite direction to the global Z-axis, and axis Z' is directed parallel to the global Y-axis.

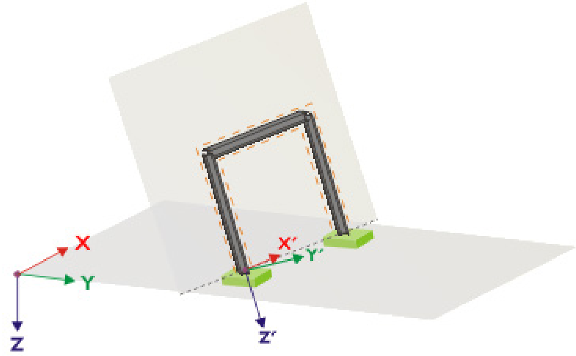


Figure 2.22: Auxiliary coordinate system for nodal supports - set of members in inclined plane

Figure 2.22 shows the general case of a buckled set of members: The members are not lying in one straight line but in an inclined plane. The definition of axis X' is based on the line created by the inclined plane intersecting the horizontal plane. Thus, axis Y' is defined right-angled to axis X' and directed perpendicular to the inclined plane. Axis Z' is defined perpendicular to the axes X' and Y'.

2.8 Member End Releases - Sets of Members

The module window is displayed only if you have selected at least one set of members for design in window 1.1 *General Data*. You can define releases for members in the set of members that do not transfer the locked degrees of freedom specified in module window 1.7 as internal forces due to structural reasons. The table refers to the current set of members (selected in the navigator on the left).

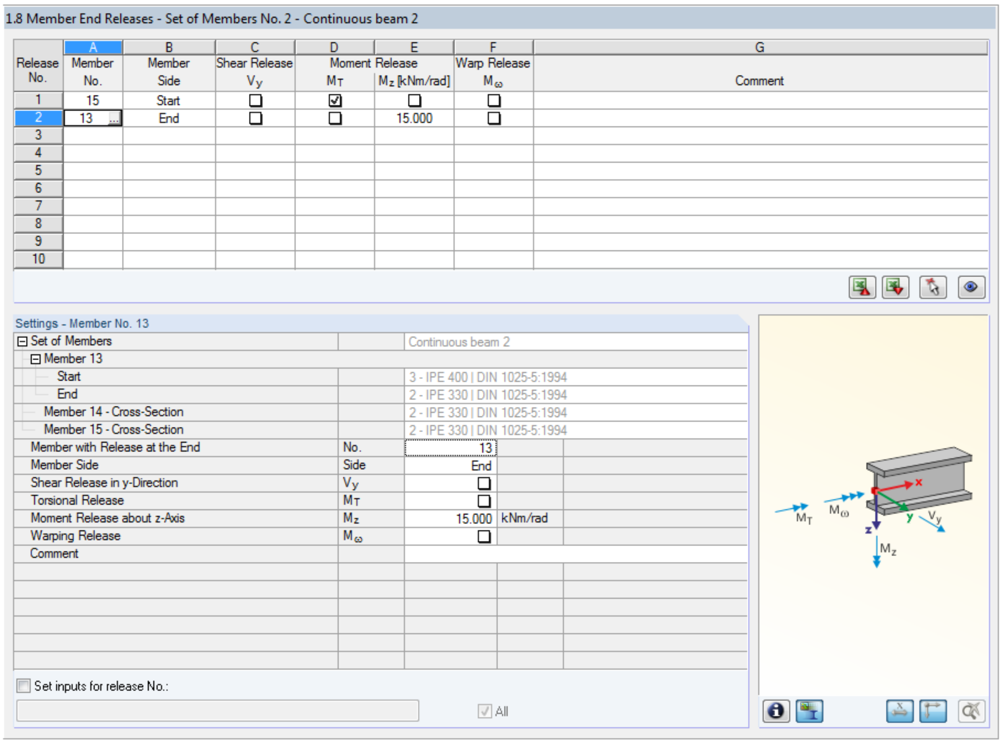
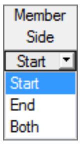


Table 2.23: Module window 1.8 *Member End Releases - Set of Members*



In table column B, you define the *Member Side* to which the release should be assigned. You can also connect the releases to both member sides.

In columns C to F, you can define releases or spring constants to align the member set model with the support conditions entered in module window 1.7.

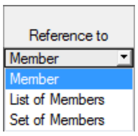
2.9 Serviceability Data

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

1.9 Serviceability Data

No.	A Reference to	B Set of Members No.	C Reference Length Manually	D L [m]	E Direc-tion	F Precamber w_c [mm]	G Beam Type	H Comment
1	Set of Members	1	☑	3.000	z	0.0	Beam	
2	Member	2	☐	12.548	y, z	0.0	Beam	
3	List of Members	3	☑	16.548	z	0.0	Beam	
4	Set of Members	4	☐	6.546	z	0.0	Beam	
5	Set of Members	5	☐	7.094	z	0.0	Beam	
6	Member	82	☐	7.094	y, z	0.0	Beam	
7	Member	83	☑	6.546	y, z	0.0	Cantilever End Free	
8	Member	81	☑	6.546	y, z	0.0	Cantilever End Free	
9	Member	25	☐	6.274	y, z	0.0	Beam	
10	Member	26	☐	6.274	y, z	0.0	Beam	
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

Table 2.24: Module window 1.9 *Serviceability Data*



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

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

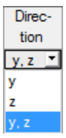
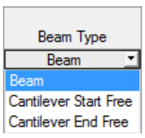


Table column E defines the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z (or u and v for unsymmetrical cross-sections).

A *Precamber* w_c can be taken into account by entering values in column F.



The *Beam Type* is of vital importance for the correct application of the limit deformations. In table column G, you can select the girder to be a beam or a cantilever and decide which end is without support.

Settings in the *Serviceability* tab of the *Details* dialog box determine whether the deformations are related to the undeformed initial structure or to the shifted ends of members or sets of members (see Figure 3.3, page 31).

3. Calculation

3.1 Detail Settings

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

The dialog box *Details* contains the following tabs:

- Ultimate Limit State
- Stability
- Serviceability
- Other

3.1.1 Ultimate Limit State

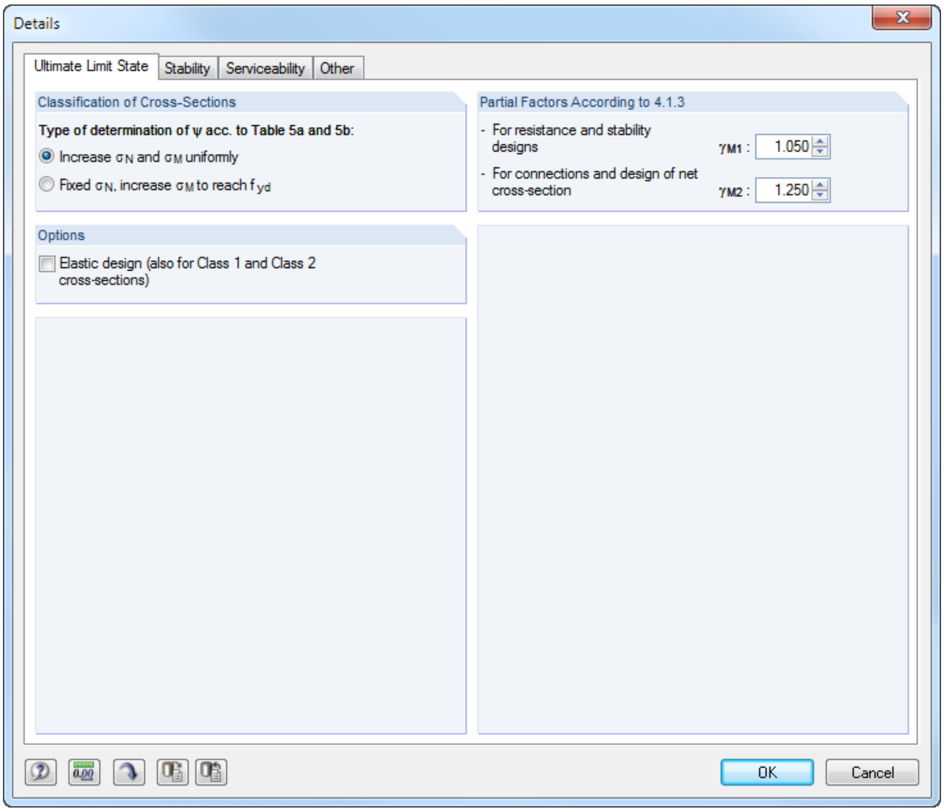


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

Classification of Cross-Sections

If stresses from compression and bending occur together in a cross-section, you can determine the stress-deformation ratio ψ in two ways (factor ψ is required for the determination of the c/t-ratio according to [1] table 5a and 5b):

- Increase σ_N and σ_M uniformly
  The stress components from axial force and bending are increased uniformly until the yield strength f_{yd} is reached.
- Fixed σ_N, increase σ_M to reach f_{yd}
  Only the stress component from bending is increased to reach the yield strength.

Options

Cross-sections that are assigned to class 1 or 2 will be designed plastically in RF-STEEL SIA. If you do not want to perform a plastic design, it is possible to activate the *Elastic design* also for these cross-section classes.

Partial Factors According to 4.1.3

The resistance coefficients of material can be defined separately for *resistance and stability designs* (γ_{M1}) and *connections and design of net cross-section* (γ_{M2}). The values recommended in [1], chapter 4.1.3 are preset.

3.1.2 Stability

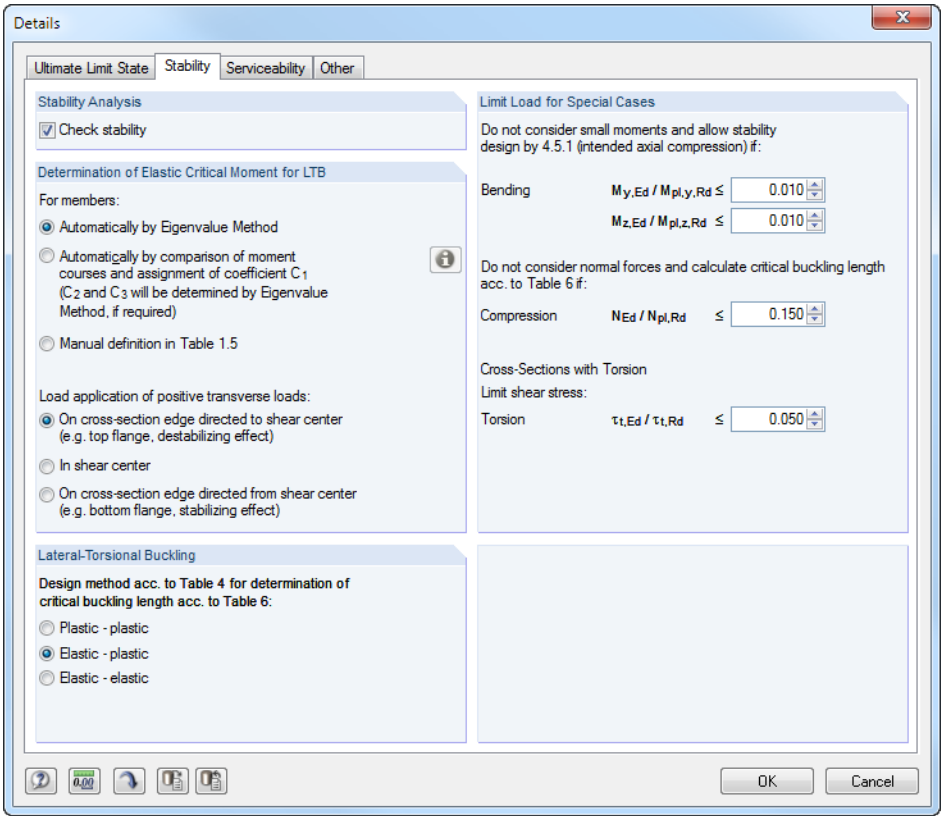


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

The check box *Check stability* allows you to perform a stability analysis in addition to the cross-section designs. If you clear the check box, the input windows 1.4 to 1.8 are not displayed.

Determination of Elastic Critical Moment for LTB

By default, RF-STEEL SIA determines the ideal moment for lateral buckling *Automatically by Eigenvalue Method*. The program uses a finite member model to determine M_{cr}, taking into account the following items:

- Dimensions of gross cross-section
- Load type and position of load application point
- Effective distribution of moments
- Lateral restraints (by support conditions)
- Effective boundary conditions

The degrees of freedom can be controlled by the factors k_z and k_w (see chapter 2.5, page 21).

If the ideal critical moment is determined *Automatically by comparison of moment courses*, the coefficient C_1 is determined by means of the moment diagram. To view the load courses and moment distributions, open the corresponding dialog box by clicking [Info]. The coefficients C_2 and C_3 are determined automatically by the eigenvalue method, if required.

H	I	J
Lateral-torsional Buckling		
Possible	k_z	M_{cr} [kNm]
☑	1.0	100.00
☑	1.0	100.00
☑	1.0	100.00
☑	1.0	100.00

M_{cr} user-defined

If you select the option *Manual definition in Table 1.5*, the title of table column J in module window 1.5 changes to M_{cr} so that the ideal moment for lateral buckling can be entered directly.

In case transverse loads are available, it is important to define where these forces are acting on the cross-section: Depending on the *Load application*, transverse loads can be stabilizing or destabilizing, and thus have a major impact on the ideal critical moment.

Lateral-Torsional Buckling

[1] table 6 describes the determination of the critical lengths for lateral buckling that are required for the lateral buckling analysis and depend on the design method (procedure PP, EP or EE). The design methods are described in [1] table 4 describing the cross-section classification.

The design method *Elastic - plastic* is preset.

Limit Load for Special Cases

To design unsymmetrical cross-sections for intended axial compression according to [1], chapter 4.5.1, it is possible to neglect *small moments* about the major and the minor axis by settings defined in this dialog section.

In the same way, it is possible to *not consider normal forces* for the pure design of bending by defining a limit ratio for N to N_{pl}. In this case, the length for lateral buckling is determined according to [1], table 6.

Intended *Torsion* is not clearly specified in [1]. If a torsional stress is available which does not exceed the shear stress ratio of 5 % preset by default, it is neglected for the stability analysis. The output will show results for flexural buckling and lateral buckling.

If one of the limits in this dialog section is exceeded, a note appears in the results window. No stability analysis is carried out. Nevertheless, the cross-section designs are performed. These limit settings are <u>not</u> part of the Swiss standard SIA 263:2003. Changing the limits is in the responsibility of the program user.

3.1.3 Serviceability

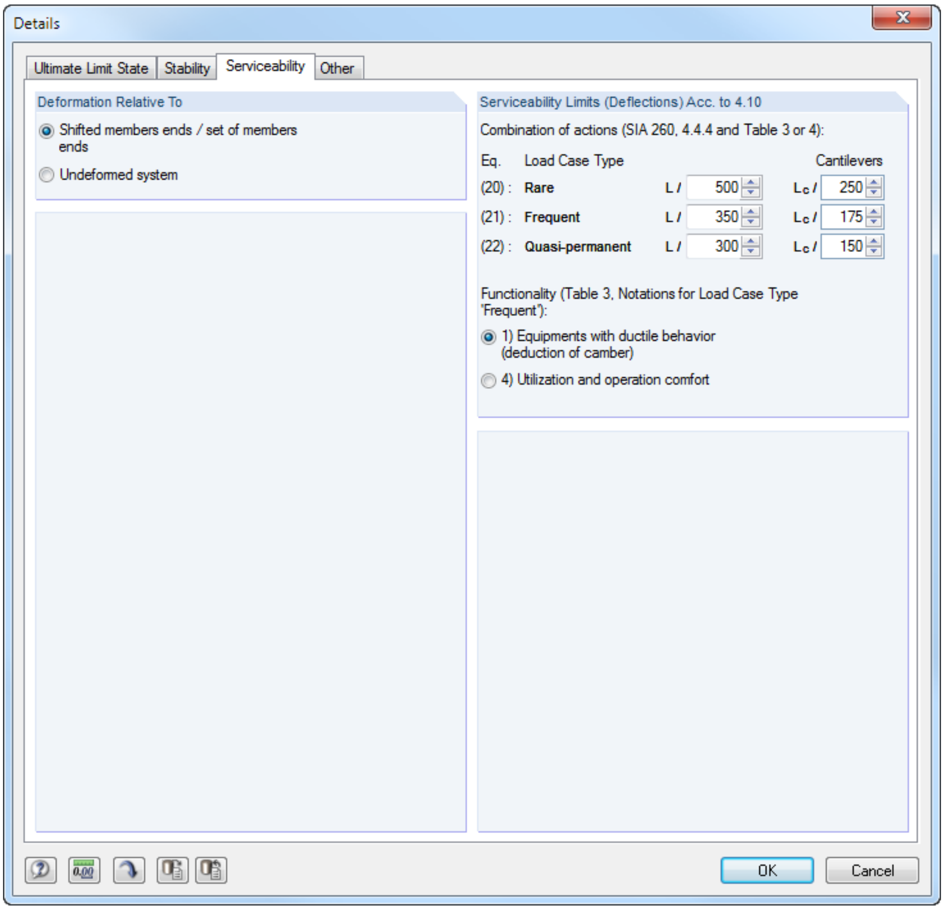


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Relative To

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

Serviceability Limits (Deflections) According to 4.10

The dialog section manages the deformation limits that must be observed for the serviceability limit state design. The limit values of deformations depend on the type of load case (see [2], chapter 4.4.4) that can be assigned in the tab *Serviceability Limit State* of module window 1.1 *General Data* (see Figure 2.5, page 11). In accordance with [2], tables 3 and 4, it is allowed to use larger deflections for cantilevers than for floors and beams.

For the load case type 'frequent' you have to specify the *Functionality* according to remarks of [2], table 3.

3.1.4 Other

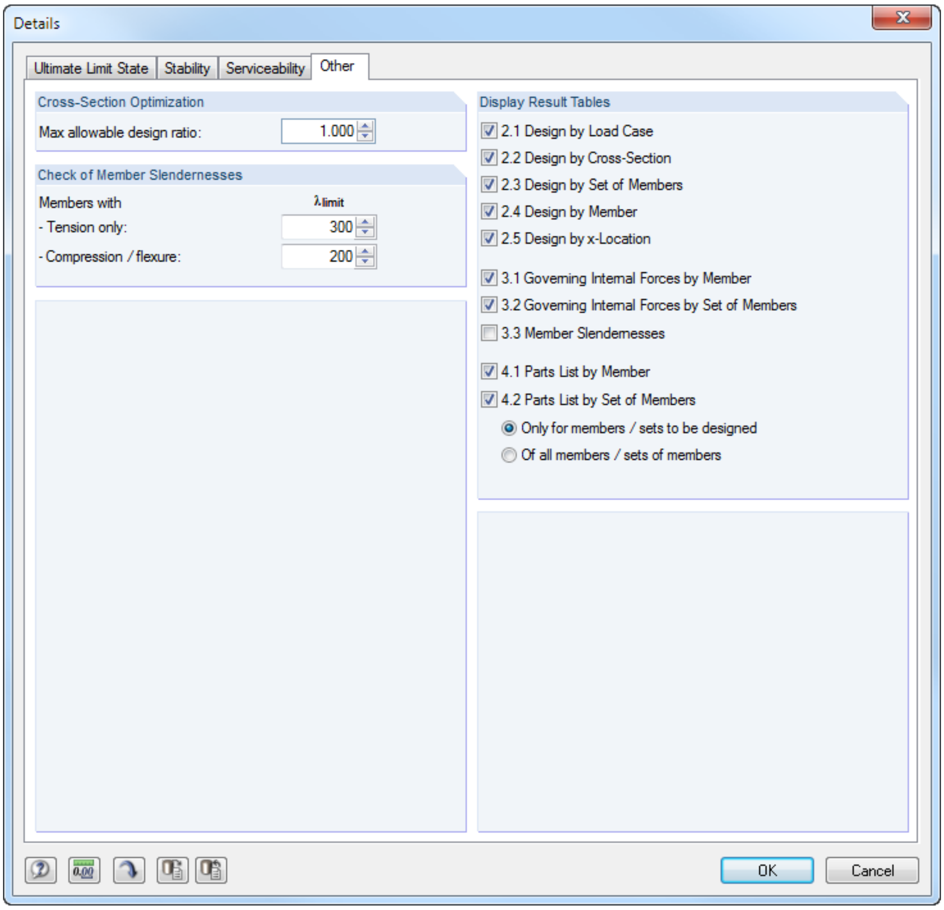


Figure 3.4: Dialog box *Details*, tab *Other*

Cross-Section Optimization

The optimization is targeted on the maximum design ratio of 100 %. If necessary, you can specify another upper limit in this input field.

Check of Member Slendernesses

Both input fields show the limit values λ_{limit} used to check the member slendernesses. It is possible to enter specifications separately for members with pure tension forces and members with bending and compression.

In module window 3.3, the limit values are compared to the real member slendernesses. The window is available only after the calculation is done (see chapter 4.8, page 41) and when the corresponding check box is ticked in the dialog section *Display Result Tables*.

Display Result Tables

In this dialog section, you can select the results windows including parts list that you want to be displayed. The windows are described in chapter 4 *Results*.

Module window 3.3 *Member Slendernesses* is inactive by default.

3.2 Start Calculation

To start the calculation, click the [Calculation] button that is available in all input windows of the add-on module RF-STEEL SIA.

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

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

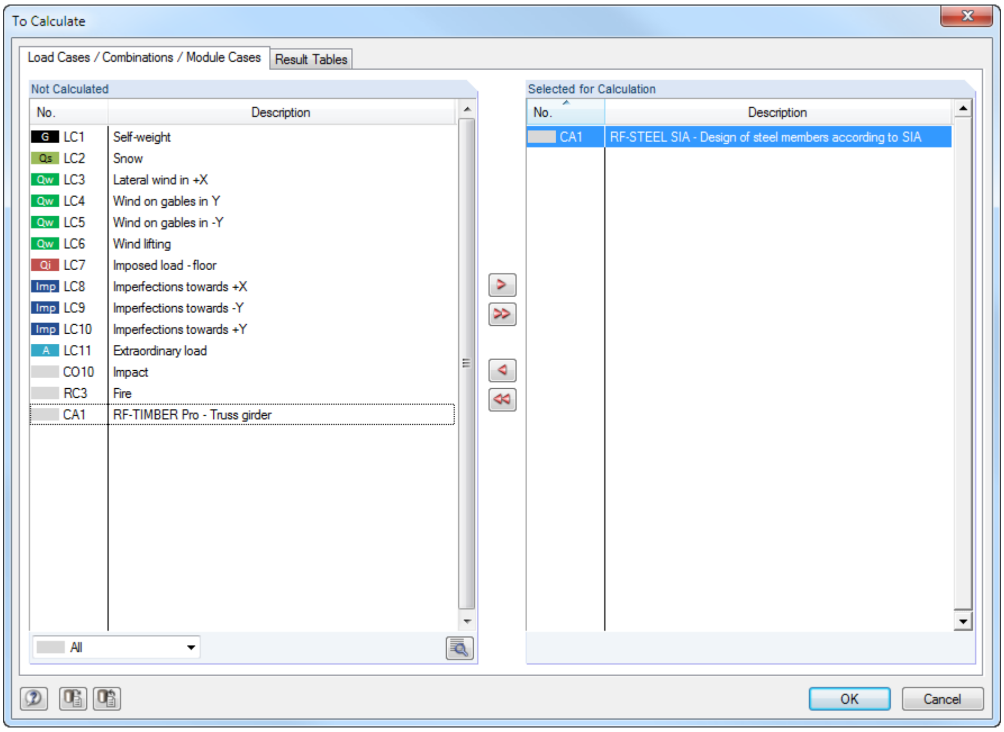
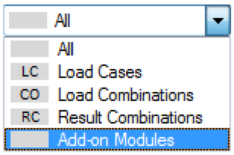


Figure 3.5: Dialog box *To Calculate*



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

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

It is also possible to calculate a design case directly by using the list in the toolbar. Select the relevant RF-STEEL SIA case in the toolbar list, and then click [Show Results].

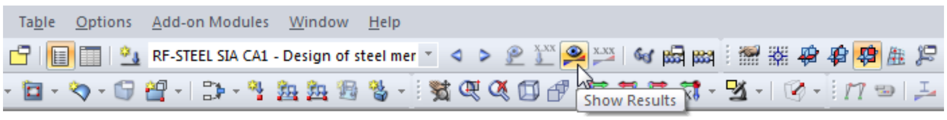


Figure 3.6: Direct calculation of a RF-STEEL SIA case in RFEM

Now, a separate dialog box appears where you can observe the design process.

4. Results

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

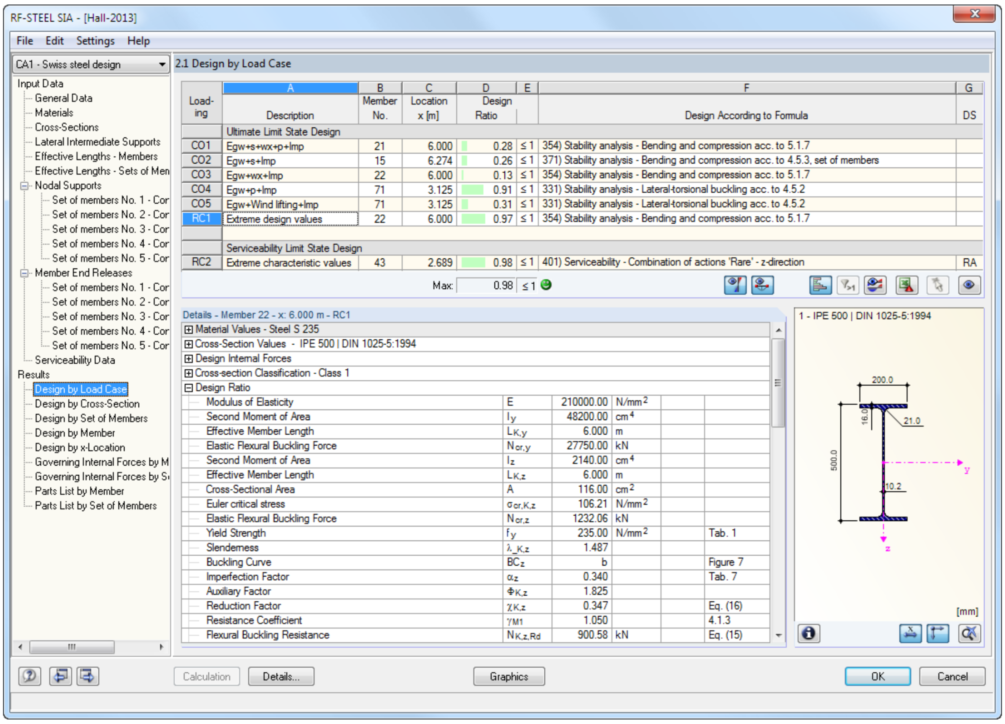
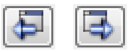


Figure 4.1: Results window with designs and intermediate values

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

Windows 3.1 and 3.2 list the governing internal forces. Window 3.3 informs you about the member slendernesses. In the final two results windows 4.1 and 4.2, parts lists are displayed by member and set of members.



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

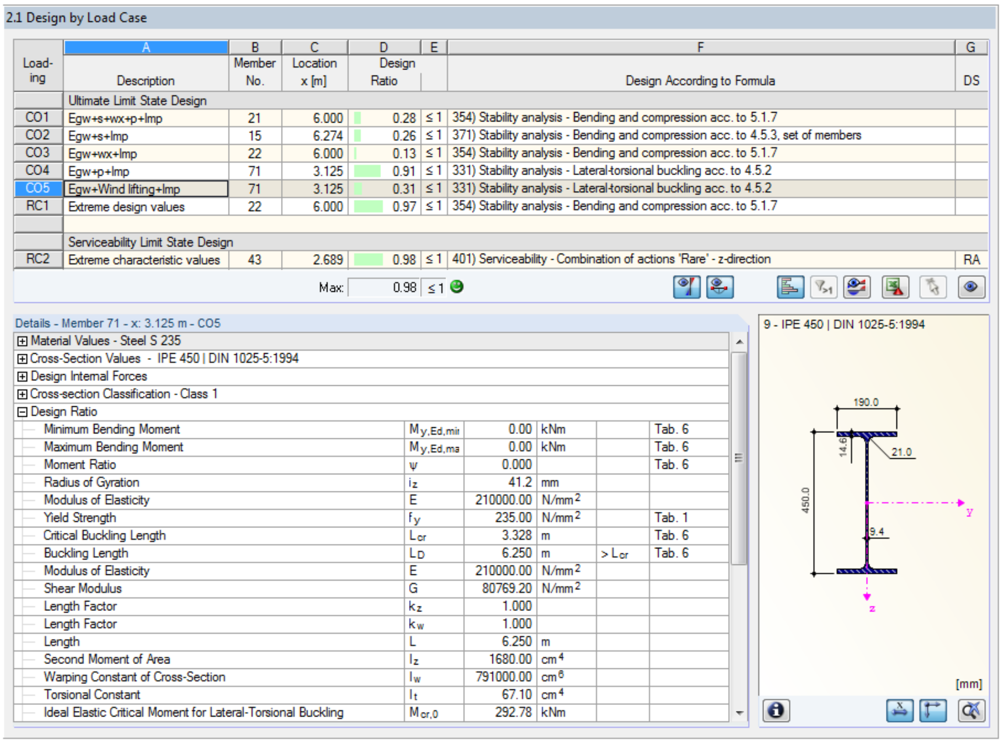
Click [OK] to save the results. You exit RF-STEEL SIA and return to the main program.

Chapter 4 *Results* describes the different results windows one by one. Evaluating and checking results is described in chapter 5 *Results Evaluation*, page 44.

4.1 Design by Load Case

The upper part of the window offers a summery, sorted by load cases, load combinations, and result combinations of the governing designs. The list is additionally subdivided into ultimate limit state and serviceability limit state design.

The lower part contains detailed information about the cross-section characteristics, analyzed internal forces, and design parameters for the load case selected above.


2.1 Design by Load Case

Loading	A Description	B Member No.	C Location x [m]	D Design Ratio	E	F Design According to Formula	G DS
	Ultimate Limit State Design						
CO1	Egw+s+wx+p+Imp	21	6.000	0.28	≤ 1	354) Stability analysis - Bending and compression acc. to 5.1.7	
CO2	Egw+s+Imp	15	6.274	0.26	≤ 1	371) Stability analysis - Bending and compression acc. to 4.5.3, set of members	
CO3	Egw+wx+Imp	22	6.000	0.13	≤ 1	354) Stability analysis - Bending and compression acc. to 5.1.7	
CO4	Egw+p+Imp	71	3.125	0.91	≤ 1	331) Stability analysis - Lateral-torsional buckling acc. to 4.5.2	
CO5	Egw+Wind lifting+Imp	71	3.125	0.31	≤ 1	331) Stability analysis - Lateral-torsional buckling acc. to 4.5.2	
RC1	Extreme design values	22	6.000	0.97	≤ 1	354) Stability analysis - Bending and compression acc. to 5.1.7	
	Serviceability Limit State Design						
RC2	Extreme characteristic values	43	2.689	0.98	≤ 1	401) Serviceability - Combination of actions 'Rare' - z-direction	RA
			Max:	0.98	≤ 1		

Details - Member 71 - x: 3.125 m - CO5

- Material Values - Steel S 235
- Cross-Section Values - IPE 450 | DIN 1025-5:1994
- Design Internal Forces
- Cross-section Classification - Class 1
- Design Ratio

Minimum Bending Moment	$M_{y,Ed,min}$	0.00	kNm		Tab. 6
Maximum Bending Moment	$M_{y,Ed,max}$	0.00	kNm		Tab. 6
Moment Ratio	ψ	0.000			Tab. 6
Radius of Gyration	i_z	41.2	mm		
Modulus of Elasticity	E	210000.00	N/mm²		
Yield Strength	f_y	235.00	N/mm²		Tab. 1
Critical Buckling Length	L_{cr}	3.328	m		Tab. 6
Buckling Length	L_D	6.250	m	> L_{cr}	Tab. 6
Modulus of Elasticity	E	210000.00	N/mm²		
Shear Modulus	G	80769.20	N/mm²		
Length Factor	k_z	1.000			
Length Factor	k_w	1.000			
Length	L	6.250	m		
Second Moment of Area	I_z	1680.00	cm⁴		
Warping Constant of Cross-Section	I_w	791000.00	cm⁶		
Torsional Constant	I_t	67.10	cm⁴		
Ideal Elastic Critical Moment for Lateral-Torsional Buckling	$M_{cr,0}$	292.78	kNm		

Figure 4.2: Module window 2.1 *Design by Load Case*

Description

The column shows the descriptions of the load cases, load and result combinations for which the designs have been performed.

Member No.

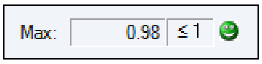
This column shows the number of the member that bears the maximum stress ratio of the designed loading.

Location x

The respective maximum stress ratio occurs on the displayed location x of the member. RF-STEEL SIA uses the following member locations *x* for the table output:

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

Design



Columns D and E display the design conditions according to SIA 263:2003 .

The length of the colored scale represents the respective design ratio.

RF-STEEL SIA does not perform the design of torsional instability (lateral-torsional buckling) as mentioned in [1] Clause 4.5.1.1.

Design According to Formula

The column lists the standard's equations by which the designs have been performed.

DS

The final column contains information about the design-relevant design situation (*DS*): Ultimate limit state (no entry) or one of the three design situations for serviceability (*RA*, *FR*, *QP*) according to the specification in module window 1.1 *General Data* (see Figure 2.5, page 11).

4.2 Design by Cross-Section

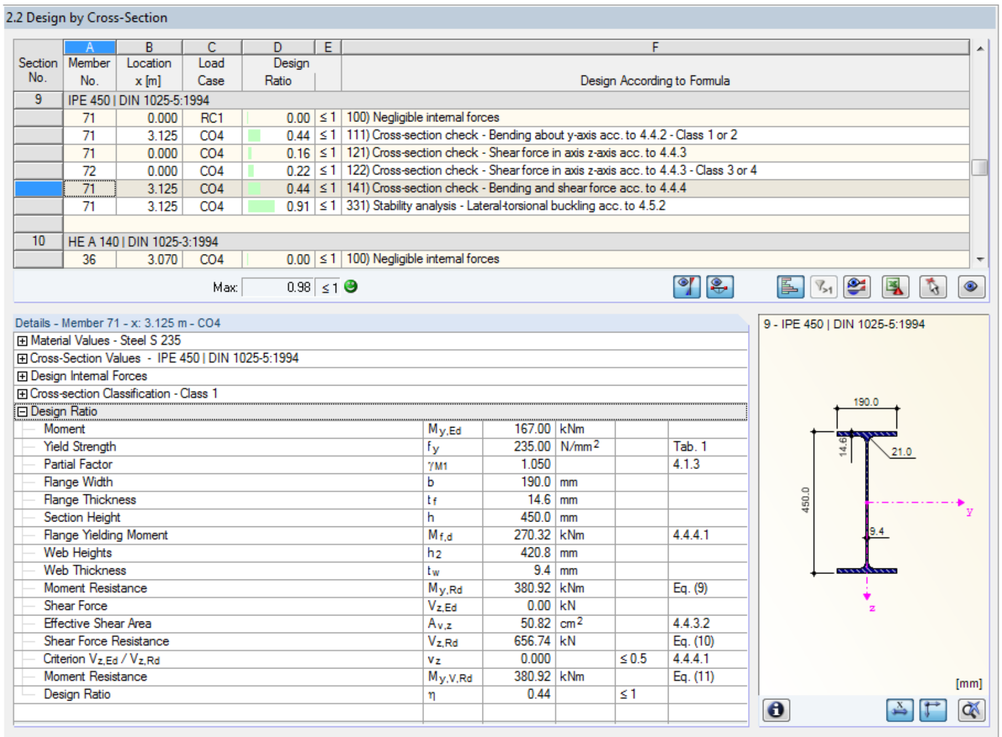


Figure 4.3: Module window 2.2 *Design by Cross-Section*

The window lists the maximum ratios of all members and actions selected for design, sorted by cross-section. The results are shown respectively for cross-section checks, stability analyses and designs of serviceability.

If a member is a tapered member, both cross-section descriptions are displayed in the table row next to the section number.

4.3 Design by Set of Members

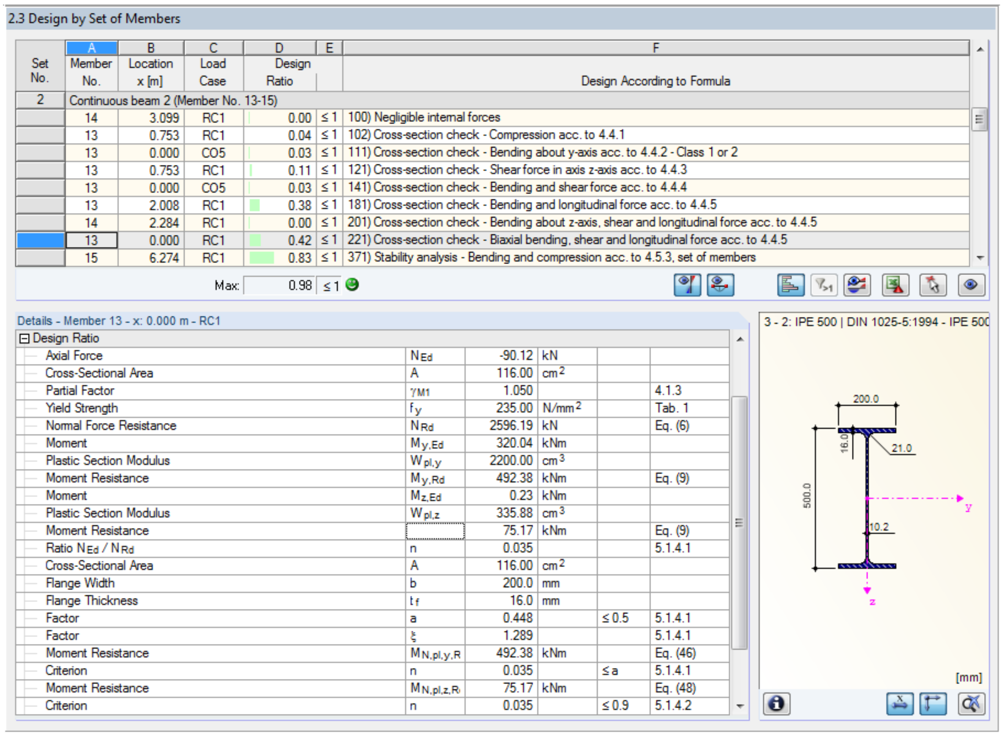


Figure 4.4: Module window 2.3 *Design by Set of Members*

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

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

The results output shown by set of members can clearly present the design for an entire structural group (for example a frame).

4.4 Design by Member

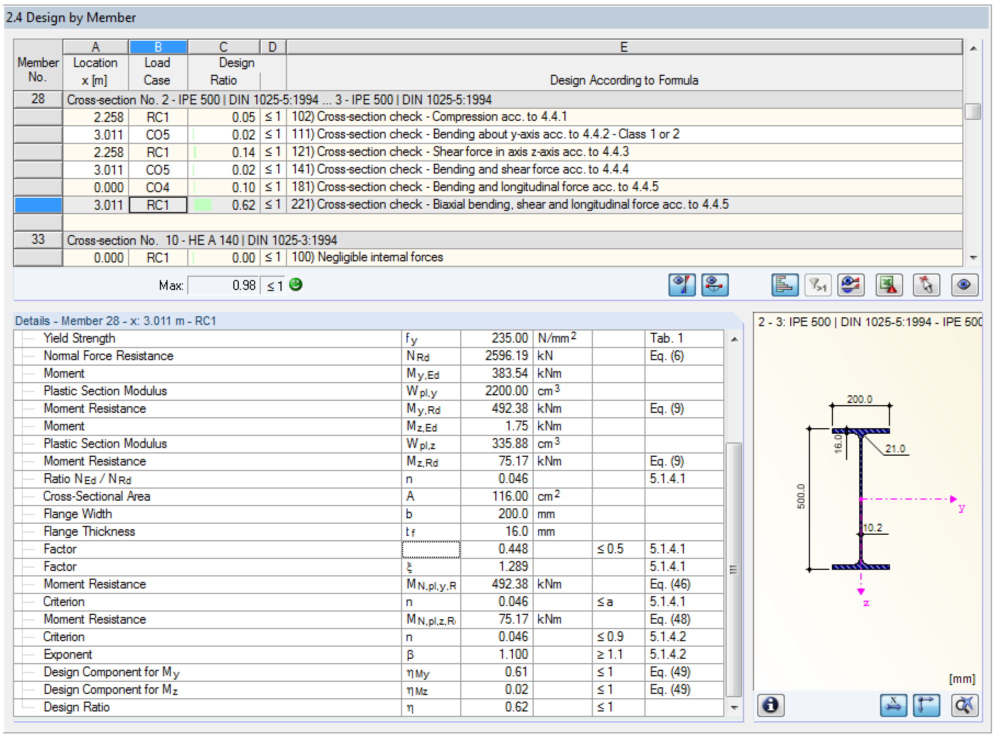


Figure 4.5: Module window 2.4 *Design by Member*

The results window presents the maximum ratios for the individual designs sorted by member number. The table columns are described in detail in chapter 4.1 on page 35.

4.5 Design by x-Location

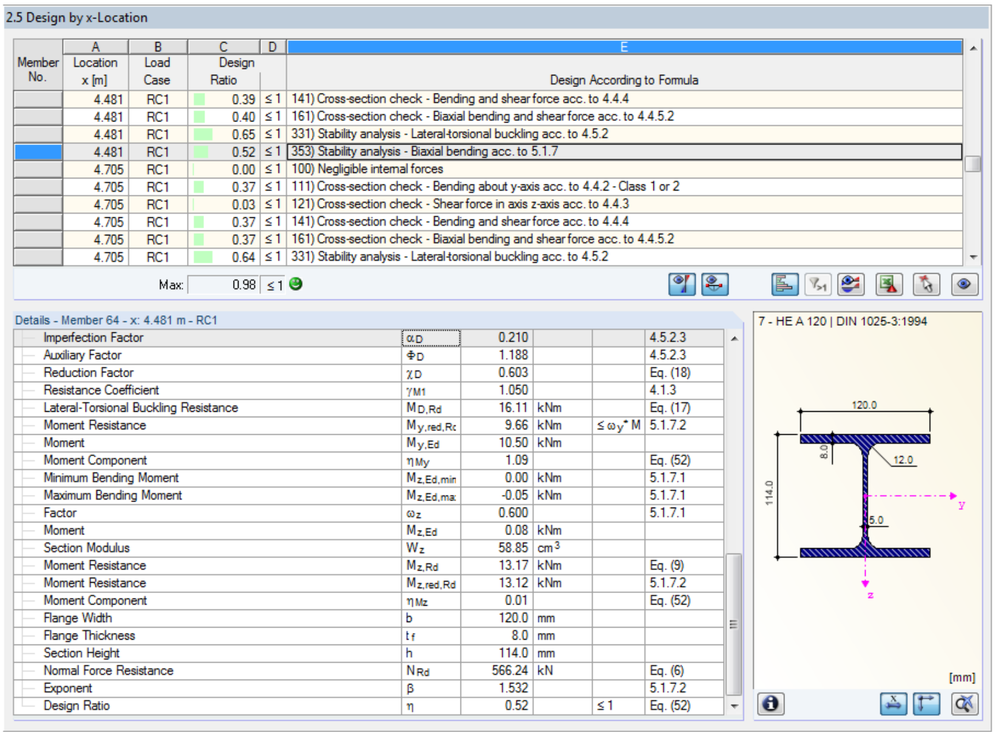


Figure 4.6: Module window 2.5 *Design by x-Location*

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

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

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

	A	B	C	D	E	F	G	H	I
Member No.	Location x [m]	Loading	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
1	Cross-section No. 1 - IPE 500 | DIN 1025-5:1994								
	0.000	RC1	0.00	0.00	0.00	0.00	0.00	0.00	100) Negligible internal forces
	0.000	RC1	-122.55	4.09	-51.77	0.00	120.47	5.39	102) Cross-section check - Compression acc. to 4.4.1
	6.000	CO3	-5.17	0.02	-4.91	0.00	-16.76	-0.08	111) Cross-section check - Bending about y-axis acc. to 4.4.2 -
	0.000	RC1	-122.55	4.09	-51.77	0.00	120.47	5.39	121) Cross-section check - Shear force in axis z-axis acc. to 4.4
	0.000	RC1	-90.22	-6.86	-19.84	0.00	46.25	-8.47	123) Cross-section check - Shear force in axis y-axis acc. to 4.4
	6.000	CO3	-5.17	0.02	-4.91	0.00	-16.76	-0.08	141) Cross-section check - Bending and shear force acc. to 4.4
	6.000	RC1	-21.46	-2.55	-20.46	0.00	-75.18	0.01	181) Cross-section check - Bending and longitudinal force acc.
	6.000	RC1	-53.52	-2.24	-51.77	0.00	-190.15	-0.16	221) Cross-section check - Biaxial bending, shear and longitudi
	0.000	CO5	-52.50	0.04	-0.66	0.00	1.03	0.07	302) Stability analysis - Buckling about y-axis acc. to 4.5.1
	0.000	CO5	-52.50	0.04	-0.66	0.00	1.03	0.07	312) Stability analysis - Buckling about z-axis acc. to 4.5.1
	0.000	CO5	-52.50	0.04	-0.66	0.00	1.03	0.07	322) Stability analysis - Torsional buckling acc. to 4.5.1
	6.000	CO3	-5.17	0.02	-4.91	0.00	-16.76	-0.08	331) Stability analysis - Lateral-torsional buckling acc. to 4.5.2
	6.000	RC1	-53.52	-2.24	-51.77	0.00	-190.15	-0.16	354) Stability analysis - Bending and compression acc. to 5.1.7
2	Cross-section No. 1 - IPE 500 | DIN 1025-5:1994								
	0.000	RC1	0.00	0.00	0.00	0.00	0.00	0.00	100) Negligible internal forces
	0.000	RC1	-123.22	4.02	53.99	0.00	-129.14	5.09	102) Cross-section check - Compression acc. to 4.4.1
	0.000	RC1	-123.22	4.02	53.99	0.00	-129.14	5.09	121) Cross-section check - Shear force in axis z-axis acc. to 4.4
	0.000	RC1	-90.42	-6.88	20.54	0.00	-48.97	-8.53	123) Cross-section check - Shear force in axis y-axis acc. to 4.4
	6.000	CO5	-1.49	0.04	1.30	0.00	4.17	-0.17	161) Cross-section check - Biaxial bending and shear force acc
	6.000	RC1	-54.19	-2.30	53.99	0.00	194.83	-0.06	181) Cross-section check - Bending and longitudinal force acc.
	2.400	RC1	-62.81	-2.38	20.54	0.00	0.32	2.35	201) Cross-section check - Bending about z-axis, shear and lon
	6.000	RC1	-53.63	3.84	53.29	-0.01	192.41	-0.60	221) Cross-section check - Biaxial bending, shear and longitudi
	0.000	CO5	-52.62	0.04	1.30	0.00	-3.60	0.09	302) Stability analysis - Buckling about y-axis acc. to 4.5.1
	0.000	CO5	-52.62	0.04	1.30	0.00	-3.60	0.09	312) Stability analysis - Buckling about z-axis acc. to 4.5.1
	0.000	CO5	-52.62	0.04	1.30	0.00	-3.60	0.09	322) Stability analysis - Torsional buckling acc. to 4.5.1
	6.000	CO5	-1.49	0.04	1.30	0.00	4.17	-0.17	353) Stability analysis - Biaxial bending acc. to 5.1.7
	6.000	RC1	-54.19	-2.30	53.99	0.00	194.83	-0.06	354) Stability analysis - Bending and compression acc. to 5.1.7
3	Cross-section No. 3 - IPE 500 | DIN 1025-5:1994 ... 2 - IPE 500 | DIN 1025-5:1994								
	2.677	RC1	-0.58	0.01	0.60	0.01	-0.91	-0.01	100) Negligible internal forces
	0.000	RC1	-56.12	-0.06	48.48	-0.10	-189.82	0.17	102) Cross-section check - Compression acc. to 4.4.1
	0.000	CO5	-0.88	0.01	1.07	0.01	-3.00	0.03	111) Cross-section check - Bending about y-axis acc. to 4.4.2 -

Figure 4.7: Module window 3.1 *Governing Internal Forces by Member*

The table displays for each member the governing internal forces that result in the maximum ratio for the individual designs.

Location x

The respective maximum ratio occurs on the displayed location x of the member.

Loading

The column shows the number of the load case, load combination or result combination whose internal forces result in the maximum design ratio.

Forces / Moments

For each member the table column shows the axial and shear forces as well as the torsional and bending moments resulting in the maximum design ratio for the individual cross-section checks, stability analyses and serviceability limit state designs.

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [m]	B Load-ing	C Forces [kN] N	D V_y	E V_z	F Moments [kNm] M_T	G M_y	H M_z	I Design According to Formula
1	Continuous beam 1 (Member No. 51,52)								
	2.700	CO3	-1.73	-0.02	0.06	0.00	-0.01	0.05	100) Negligible internal forces
	0.000	RC1	-180.10	-2.95	0.14	0.00	-0.18	-3.68	102) Cross-section check - Compression acc. to 4.4.1
	3.000	CO5	0.22	-0.09	0.12	0.00	0.02	0.18	116) Cross-section check - Bending about z-axis acc. to 4.4.2 -
	0.000	CO3	-9.49	-0.02	1.07	0.00	-1.53	-0.01	121) Cross-section check - Shear force in axis z-axis acc. to 4.4
	0.000	RC1	-120.95	5.45	0.06	0.00	-0.07	6.78	123) Cross-section check - Shear force in axis y-axis acc. to 4.4
	3.000	CO5	0.22	-0.09	0.12	0.00	0.02	0.18	151) Cross-section check - Bending about z-axis and shear forc
	0.000	CO4	-179.65	-0.07	0.83	0.00	-3.12	-0.03	181) Cross-section check - Bending and longitudinal force acc.
	0.000	RC1	-120.95	5.45	0.06	0.00	-0.07	6.78	201) Cross-section check - Bending about z-axis, shear and lon
	3.000	RC1	-145.33	-0.37	0.14	0.00	0.25	-0.63	221) Cross-section check - Biaxial bending, shear and longitudi
	1.950	RC1	-157.67	-0.89	0.14	0.00	0.10	0.05	302) Stability analysis - Buckling about y-axis acc. to 4.5.1
	1.950	RC1	-157.67	-0.89	0.14	0.00	0.10	0.05	312) Stability analysis - Buckling about z-axis acc. to 4.5.1
	1.950	RC1	-157.67	-0.89	0.14	0.00	0.10	0.05	322) Stability analysis - Torsional buckling acc. to 4.5.1
	0.000	RC1	-120.95	5.45	0.06	0.00	-0.07	6.78	354) Stability analysis - Bending and compression acc. to 5.1.7
	0.000	RC1	-120.95	5.45	0.06	0.00	-0.07	6.78	371) Stability analysis - Bending and compression acc. to 4.5.3,
	0.000	RC2	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations
	2.250	RC2	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Combination of actions 'Rare' - z-direction
2	Continuous beam 2 (Member No. 13-15)								
	3.099	RC1	0.64	0.00	-0.15	0.00	-0.91	0.00	100) Negligible internal forces
	0.753	RC1	-98.89	-0.17	65.46	-0.04	-250.74	-0.36	102) Cross-section check - Compression acc. to 4.4.1
	0.000	CO5	2.60	0.01	-6.27	0.00	15.16	0.03	111) Cross-section check - Bending about y-axis acc. to 4.4.2 -
	0.753	RC1	-89.84	0.05	68.40	-0.05	-267.31	0.19	121) Cross-section check - Shear force in axis z-axis acc. to 4.4
	0.000	CO5	2.60	0.01	-6.27	0.00	15.16	0.03	141) Cross-section check - Bending and shear force acc. to 4.4
	2.008	RC1	-89.37	0.05	63.04	-0.05	-184.87	0.13	181) Cross-section check - Bending and longitudinal force acc.
	2.284	RC1	-34.03	0.10	19.05	-0.02	0.08	-0.18	201) Cross-section check - Bending about z-axis, shear and lon
	0.000	RC1	-90.12	0.05	71.69	-0.05	-320.04	0.23	221) Cross-section check - Biaxial bending, shear and longitudi
	6.274	RC1	-58.72	0.32	-2.73	-0.01	178.20	-1.54	371) Stability analysis - Bending and compression acc. to 4.5.3,
3	Continuous beam 3 (Member No. 41-43)								
	1.794	RC1	-0.46	-0.02	0.21	0.00	0.95	0.01	100) Negligible internal forces
	6.274	CO5	6.89	-0.01	0.87	0.00	-1.91	0.02	101) Cross-section check - Tension acc. to 4.4.1
	0.502	RC1	-167.51	-0.12	78.90	0.09	-286.81	-0.49	102) Cross-section check - Compression acc. to 4.4.1
	0.000	CO5	4.69	-0.03	-2.41	0.00	9.61	-0.10	111) Cross-section check - Bending about y-axis acc. to 4.4.2 -

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

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

4.8 Member Slendernesses

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C Major Axis y k_y [-]	D i_y [mm]	E λ_y [-]	F Minor Axis z k_z [-]	G i_z [mm]	H λ_z [-]	I
1	Compression / Flexure	6.000	1.000	203.8	29.435	1.000	43.0	139.693	
2	Compression / Flexure	6.000	1.000	203.8	29.435	1.000	43.0	139.693	
3	Compression / Flexure	3.011	1.000	203.8	14.773	1.000	43.0	70.110	
4	Compression / Flexure	3.262	1.000	203.8	16.005	1.000	43.0	75.957	
5	Compression / Flexure	6.274	1.000	203.8	30.779	1.000	43.0	146.071	
6	Compression / Flexure	6.274	1.000	203.8	30.779	1.000	43.0	146.071	
7	Compression / Flexure	3.262	1.000	203.8	16.005	1.000	43.0	75.957	
8	Compression / Flexure	3.011	1.000	203.8	14.773	1.000	43.0	70.110	
12	Compression / Flexure	6.000	1.000	203.8	29.435	1.000	43.0	139.693	
21	Compression / Flexure	6.000	1.000	203.8	29.435	1.000	43.0	139.693	
22	Compression / Flexure	6.000	1.000	203.8	29.435	1.000	43.0	139.693	
23	Compression / Flexure	3.011	1.000	203.8	14.773	1.000	43.0	70.110	
24	Compression / Flexure	3.262	1.000	203.8	16.005	1.000	43.0	75.957	
25	Compression / Flexure	6.274	1.000	203.8	30.779	1.000	43.0	146.071	
26	Compression / Flexure	6.274	1.000	203.8	30.779	1.000	43.0	146.071	
27	Compression / Flexure	3.262	1.000	203.8	16.005	1.000	43.0	75.957	
28	Compression / Flexure	3.011	1.000	203.8	14.773	1.000	43.0	70.110	
37	Compression / Flexure	3.000	1.000	57.3	52.380	1.000	35.2	85.234	
38	Compression / Flexure	3.546	1.000	57.3	61.913	1.000	35.2	100.746	
51	Compression / Flexure	3.000	1.000	82.8	36.224	1.000	49.9	60.112	
52	Compression / Flexure	3.000	1.000	82.8	36.224	1.000	49.9	60.112	
53	Compression / Flexure	3.000	1.000	74.4	40.303	1.000	45.2	66.390	
54	Compression / Flexure	3.546	1.000	74.4	47.638	1.000	45.2	78.472	
55	Compression / Flexure	3.000	1.000	74.4	40.303	1.000	45.2	66.390	
56	Compression / Flexure	4.094	1.000	74.4	55.000	1.000	45.2	90.600	
58	Compression / Flexure	3.546	1.000	74.4	47.638	1.000	45.2	78.472	
59	Compression / Flexure	3.000	1.000	82.8	36.224	1.000	49.9	60.112	
60	Compression / Flexure	3.000	1.000	82.8	36.224	1.000	49.9	60.112	
61	Compression / Flexure	6.274	1.000	48.9	128.190	1.000	30.2	207.628	
62	Compression / Flexure	6.274	1.000	48.9	128.194	1.000	30.2	207.633	

Members with compression / flexure:

Max $\lambda_{K,y}$: 128.194 ≤ 200

Max $\lambda_{K,z}$: 207.633 > 200

Figure 4.9: Module window 3.3 *Member Slendernesses*

Details...

This results window is shown only if you select the respective check box in the *Details* dialog box, tab *Other* (see Figure 3.4, page 32).

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They were determined as a function of the type of load. Below the table, you find a comparison with the limit values defined in the *Details* dialog box, tab *Other* (see Figure 3.4, page 32).

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

The table is displayed only for information. No stability design of slendernesses is intended.

4.9 Parts List by Member

Finally, RF-STEEL SIA provides a summary of the cross-sections included in the design case.

4.1 Parts List by Member

	A	B	C	D	E	F	G	H	I
Part No.	Cross-Section Description	Number of Members	Length [m]	Total Length [m]	Surface Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1	1 - IPE 500 | DIN 1025-5:1994	5	6.00	30.00	52.32	0.35	91.06	546.36	2.732
2	2 - IPE 500 | DIN 1025-5:1994 ... 3 - IPE 500 |	4	3.01	12.05	21.01	0.14	91.06	274.21	1.097
3	2 - IPE 500 | DIN 1025-5:1994	4	3.26	13.05	22.76	0.15	91.06	297.08	1.188
4	2 - IPE 500 | DIN 1025-5:1994	4	6.27	25.10	43.77	0.29	91.06	571.31	2.285
5	10 - HE A 140 | DIN 1025-3:1994	1	3.00	3.00	2.38	0.01	24.65	73.95	0.074
6	10 - HE A 140 | DIN 1025-3:1994	1	3.55	3.55	2.82	0.01	24.65	87.41	0.087
7	15 - HE A 200 | DIN 1025-3:1994	4	3.00	12.00	13.68	0.06	42.23	126.70	0.507
8	6 - HE A 180 | DIN 1025-3:1994	2	3.00	6.00	6.12	0.03	35.56	106.68	0.213
9	6 - HE A 180 | DIN 1025-3:1994	2	3.55	7.09	7.23	0.03	35.56	126.10	0.252
10	6 - HE A 180 | DIN 1025-3:1994	1	4.09	4.09	4.18	0.02	35.56	145.58	0.146
11	7 - HE A 120 | DIN 1025-3:1994	4	6.27	25.10	16.99	0.06	19.86	124.60	0.498
12	9 - IPE 450 | DIN 1025-5:1994	4	6.25	25.00	40.13	0.25	77.56	484.74	1.939
13	6 - HE A 180 | DIN 1025-3:1994	2	6.55	13.09	13.35	0.06	35.56	232.78	0.466
14	6 - HE A 180 | DIN 1025-3:1994	1	7.09	7.09	7.24	0.03	35.56	252.27	0.252
15	12 - QRO 120x4.5 | DIN 59410:1974	20	5.00	100.00	46.90	0.20	16.09	80.46	1.609
16	13 - RD 24 | DIN 1013-1	1	7.81	7.81	0.59	0.00	3.55	27.71	0.028
17	13 - RD 24 | DIN 1013-1	2	8.02	16.05	1.21	0.01	3.55	28.47	0.057
Sum		62		310.06	302.66	1.71			13.431

Figure 4.10: Module window 4.1 *Parts List by Member*

Details...

By default, only the designed members are contained in the list. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box, tab *Other* (see Figure 3.4, page 32).

Part No.

RF-STEEL SIA automatically assigns part numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column displays the respective length of an individual member.

Total Length

The values in this column represent the product determined from the two previous columns.

Surface Area

RF-STEEL SIA indicates the surface areas related to the total length part by part. The areas are determined from the *Surface* of the cross-sections that can be seen in the cross-section information (see Figure 2.11, page 16) available in module windows 1.3 and 2.1 to 2.5.

Volume

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

Unit Weight

The *Unit Weight* represents the cross-section mass related to the length of one meter. For tapered cross-sections both cross-section properties are averaged by the program.

Weight

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

Total Weight

The final column indicates the total mass of each part.

Sum

At the bottom of the list, you find a summary with the summed up values of columns B, D, E, F, and I. The last data field of the *Total Weight* column gives information about the required total amount of 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	Continuous beam 1	1	6.00	6.00	6.84	0.03	42.23	253.40	0.253
2	Continuous beam 2	1	12.55	12.55	21.88	0.15	91.06	1142.60	1.143
3	Continuous beam 3	1	12.55	12.55	21.88	0.15	91.06	1142.60	1.143
4	Continuous beam 4	1	6.55	6.55	5.20	0.02	24.65	161.35	0.161
5	Continuous beam 5	1	7.09	7.09	5.63	0.02	24.65	174.86	0.175
Sum		5		44.74	61.44	0.37			2.875

Figure 4.11: Module window 4.2 *Parts List by Set of Members*

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

Details on the table columns can be found in the previous chapter. If different cross-sections are used in the set of members, the program averages surface area, volume and cross-section mass.

5. Results Evaluation

You can evaluate the design results in various manners. The buttons below the table in the upper window part can be helpful for the evaluation process.

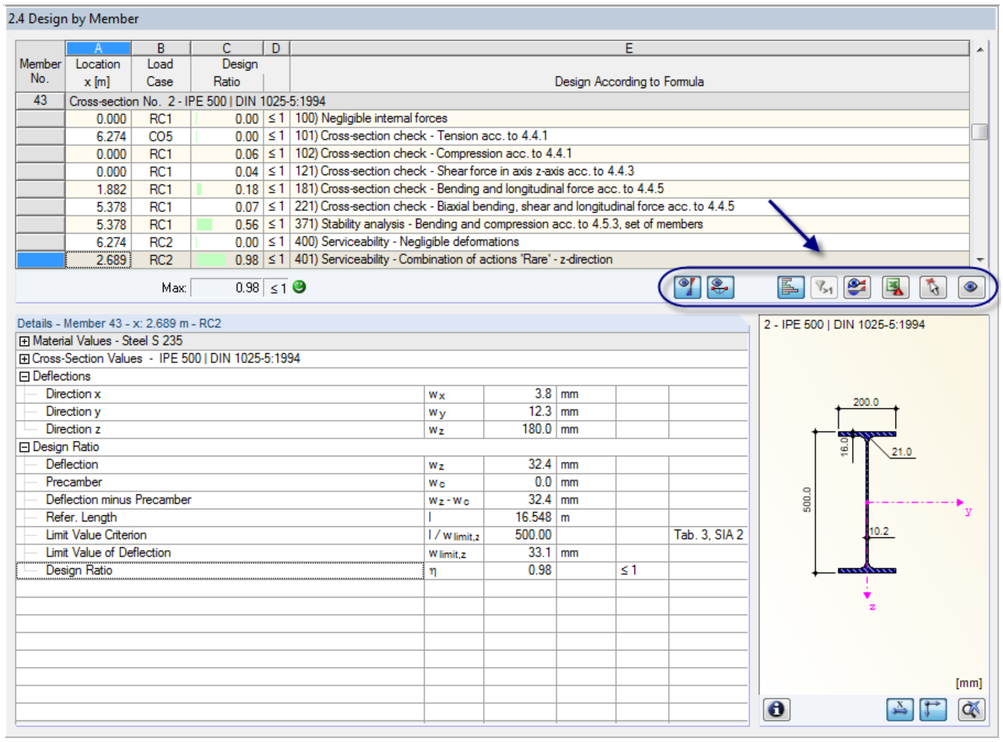


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Ultimate limit state design	Turns on and off the results of the ultimate limit state design
	Serviceability limit state design	Turns on and off the results of the serviceability limit state design
	Show color bars	Turns on and off the colored reference scales in the results windows
	Show rows with ratio > 1	Displays only the table rows where the ratio is more than 1, thus the design is failed
	Show result diagrams	Opens the window *Result Diagram on Member* → chapter 5.2, page 47
	Export to MS Excel	Exports the table to MS Excel / OpenOffice → chapter 7.4.3, page 58
	Select member	Allows you to select a member graphically to display its results in the table
	Jump to graphic	Jumps to the RFEM work window to change the view

Table 5.1: Buttons in results windows 2.1 to 2.5

5.1 Results in RFEM Model

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

RFEM background graphic and view mode

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

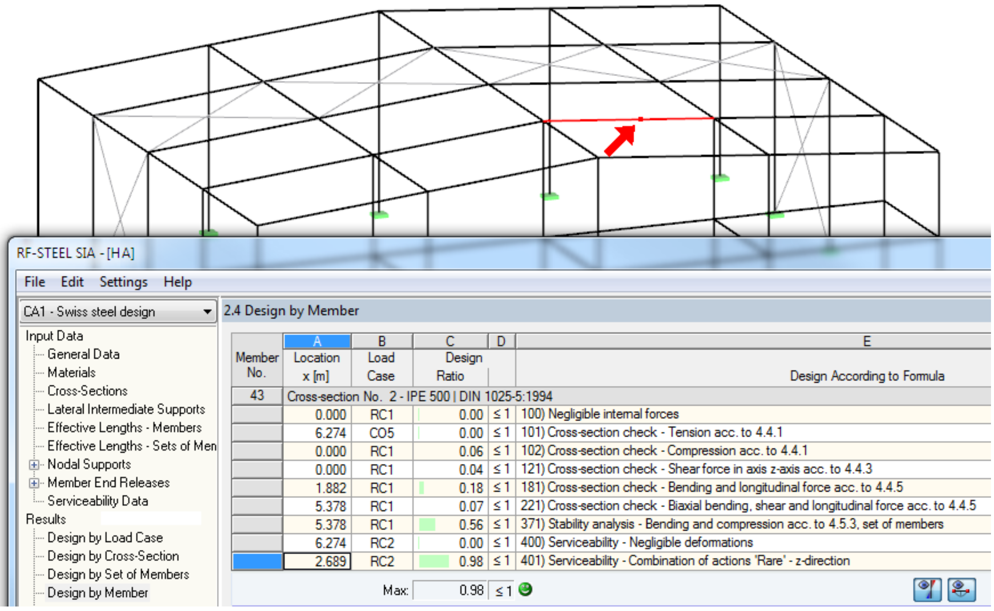


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model

In case you cannot improve the graphic display by shifting the RF-STEEL SIA program window, click [Jump to graphic] to activate the *view mode*: The window will be hidden so that you are able to adjust the view in the RFEM workspace. The view mode provides the functions of the *View* menu, for example zooming, moving, or rotating the view. The marking arrow remains visible.

Click [Back] to return to the add-on module RF-STEEL SIA.

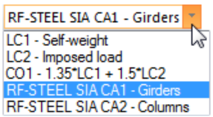
RFEM work window

The design ratios can also be checked graphically in the RFEM model. Click [Graphics] to exit the design module. Now, the ratios are displayed in the RFEM work window like internal forces of a load case.

In the *Results* navigator, you have the possibility to define which ratios of the ultimate or serviceability limit state design you want to be represented graphically.

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

As the RFEM tables are of no relevance for the evaluation of design results, you can hide them.



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

The graphical representation of results can be controlled in the *Display* navigator by selecting *Results* → *Members*. The display of ratios is *Two-Colored* by default.

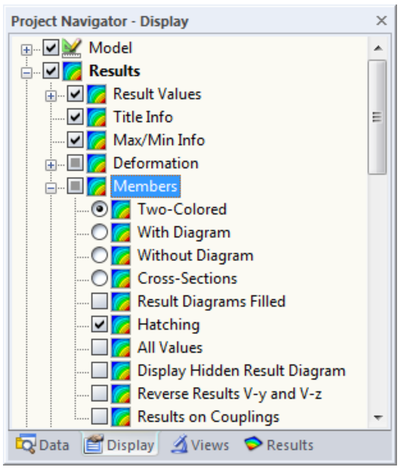


Figure 5.3: *Display* navigator: Results → Members

In case of a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color panel is available, providing common control functions. The functions are described in detail in the RFEM manual, chapter 3.4.6.

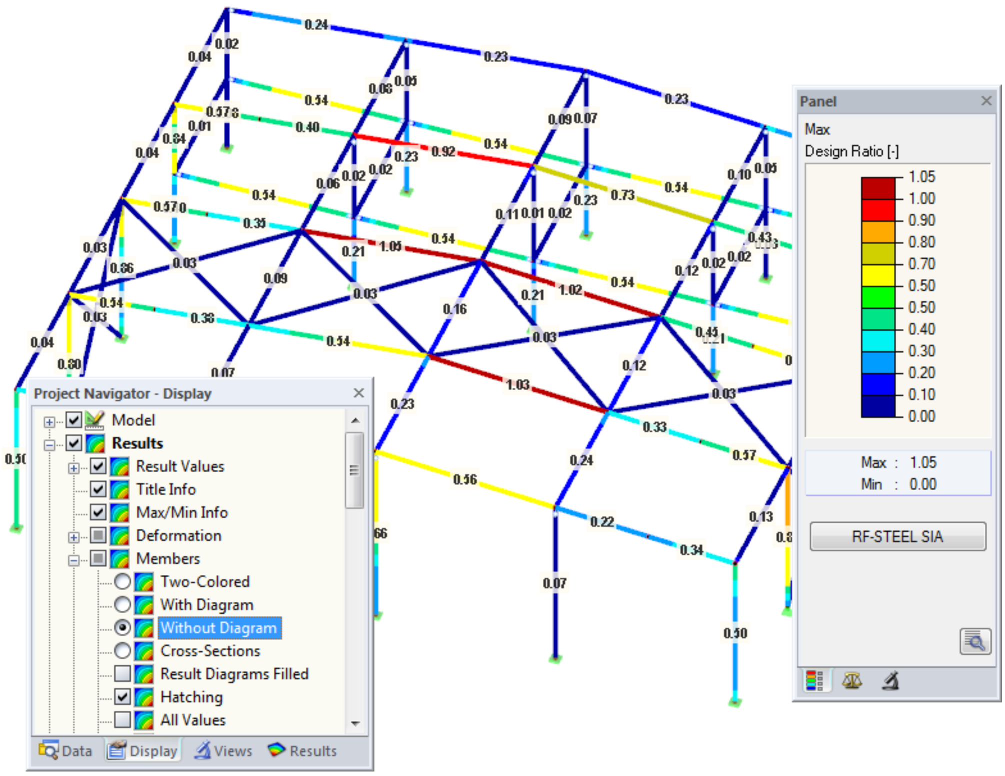


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see chapter 6.2, page 50).

To return to the add-on module, click [RF-STEEL SIA] in the panel.

5.2 Result Diagrams

You can evaluate a member's result distributions also graphically in the result diagram.

Select the member (or set of members) in the RF-STEEL SIA results window by clicking in the table row of the relevant member. Then, open the dialog box *Result Diagram on Member* by clicking the button shown on the left. You find the button below the upper results table (see Figure 5.1, page 44).

It is also possible to acces the result diagrams from the RFEM graphic. To display the diagrams, click

Results → Result Diagrams for Selected Members

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

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

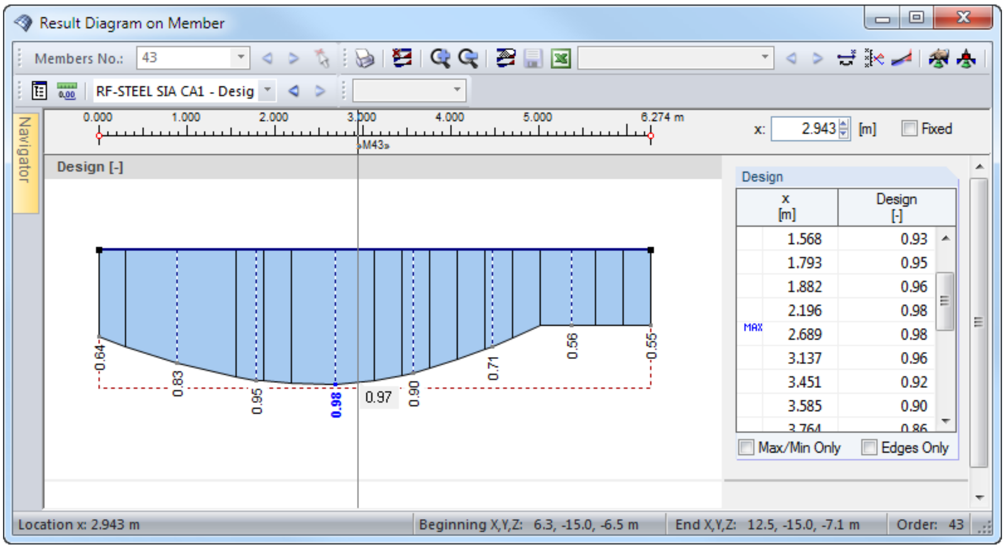
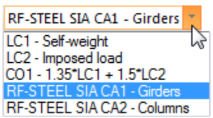


Figure 5.5: Dialog box *Result Diagram on Member*



Use the list in the toolbar to choose the relevant RF-STEEL SIA design case.

The dialog box *Result Diagram on Member* is described in detail in the RFEM manual, chapter 9.5.

5.3 Filter for Results

The RF-STEEL SIA results windows allow you to sort the results by various criteria. In addition, you can use the filter functions described in chapter 9.9 of the RFEM manual to evaluate the design results graphically.

Furthermore, you can use the *Visibility* settings for RF-STEEL SIA (see RFEM manual, chapter 9.9.1) to filter the members for your evaluation.

Filtering designs

The design ratios can be well used as filter criteria in the RFEM work window that you can access by clicking [Graphics]. To apply the filter function, the panel must be displayed. If the panel is not active, go to the RFEM menu bar and click

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

or use the toolbar button shown on the left.

The panel is described in the RFEM manual, chapter 3.4.6. The filter settings for the results must be defined in the first panel tab (color spectrum). As this tab is not available for the two-colored results display, you have to use the *Display* navigator and set the display options colored *With/Without Diagram* or *Cross-Sections* first.

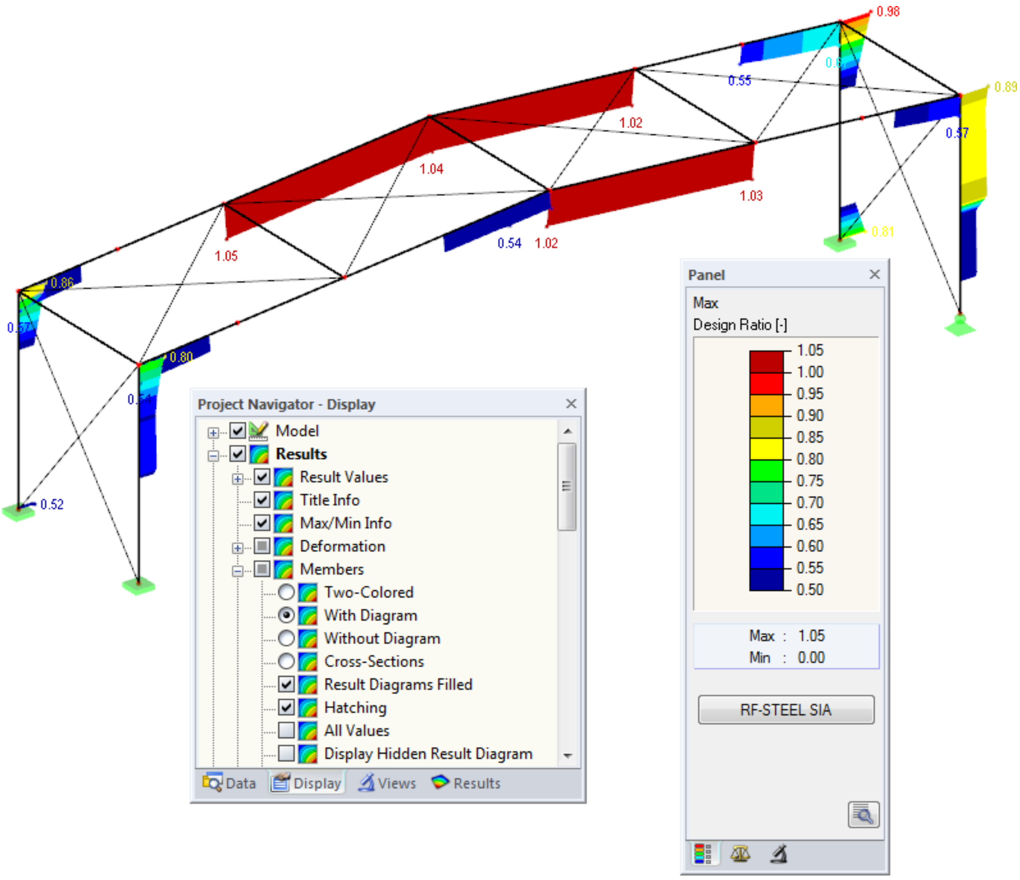
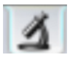


Figure 5.6: Filtering design ratios with adjusted color spectrum

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

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

Filtering members



In the *Filter* tab of the control panel, you can specify the numbers of selected members to show their results in a filtered display. This function is described in detail in the RFEM manual, chapter 9.9.3.

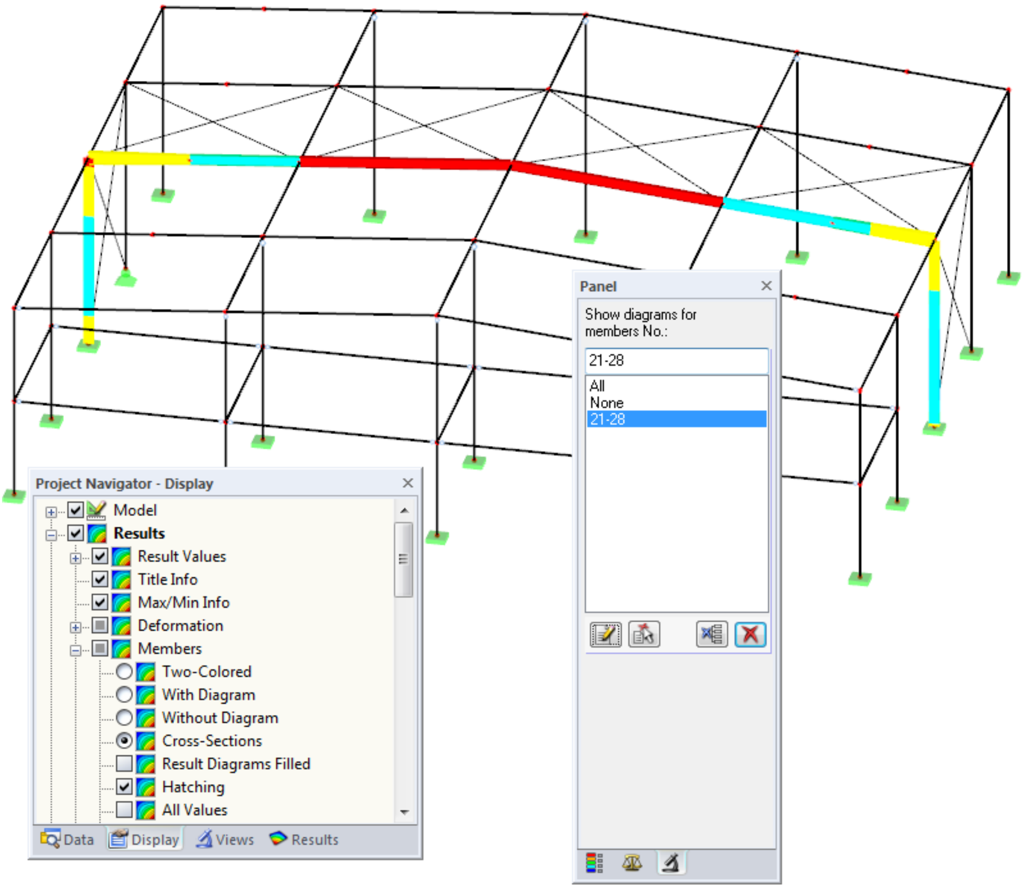


Figure 5.7: Member filter for design ratios of a hall frame

In contrast to the visibility function, the model is now displayed completely in the graphic. The figure above shows the design ratios of a hall frame. The remaining members are displayed in the model but are shown without design ratios.

6. Printout

6.1 Printout Report

Printout reports are created like in RFEM. First, a printout is generated for the data of the RF-STEEL SIA add-on module to which you can add graphics and explanations. Then, in the printout report you select the data of the design module that will finally appear in the report.

The printout report is described in the RFEM manual. In particular, chapter 10.1.3.4 *Selecting Data of Add-on Modules* provides information about the selection of input and output data in add-on modules for the printout.

For complex structural systems with a high number of design cases, it is recommended to split data into several printout reports which makes data arrangement clearer.

6.2 Graphic Printout

In RFEM every picture that is displayed in the work window can be included in the printout report or sent directly to a printer. Thus, also design ratios displayed in the RFEM model can be prepared for the printout.

The printing of graphics is described in the RFEM manual, chapter 10.2.

Designs in the RFEM model

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

File → Print Graphic

or use the toolbar button shown on the left.

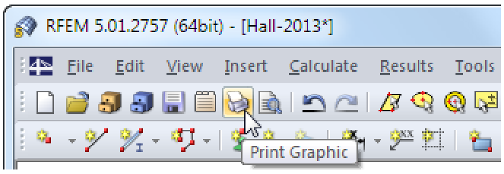


Figure 6.1: Button *Print Graphic* in RFEM toolbar

Result diagrams

In the dialog box *Result Diagram on Member* you can also use the [Print] button to transfer the graphic of design values to the report or to print it directly.

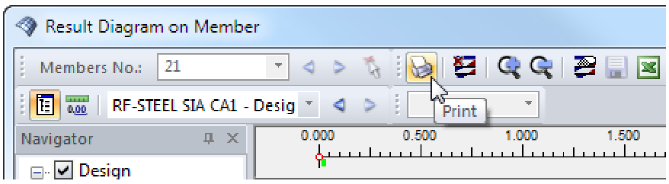


Figure 6.2: Button *Print* in the dialog box *Result Diagram on Member*

The dialog box *Graphic Printout* shown on the following page opens.

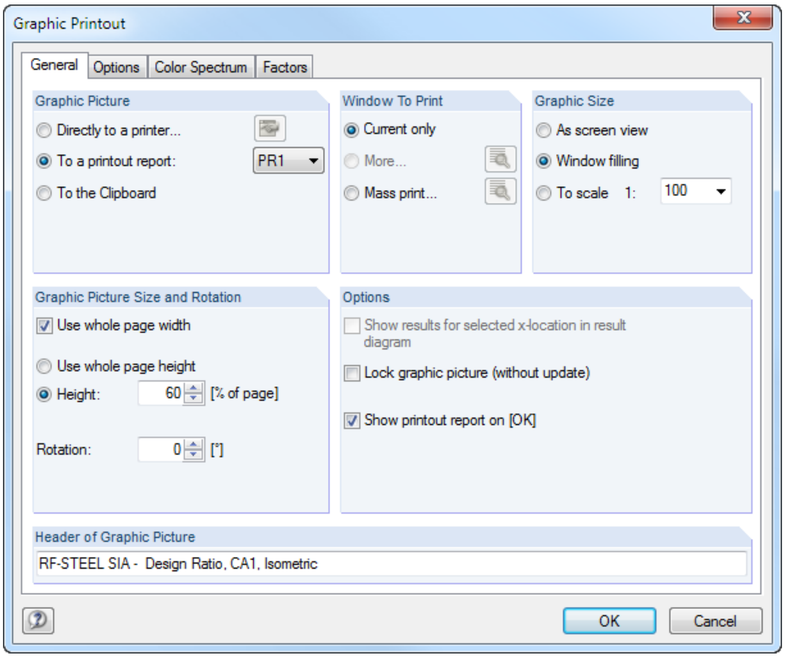
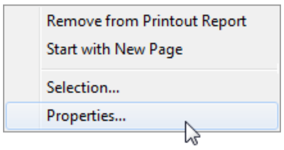


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

A graphic can be moved anywhere within the printout report by using the drag-and-drop function.



To adjust a graphic subsequently in the printout report, right-click the relevant entry in the report navigator. The option *Properties* in the context menu opens the dialog box *Graphic Printout*, offering various options for adjustment.

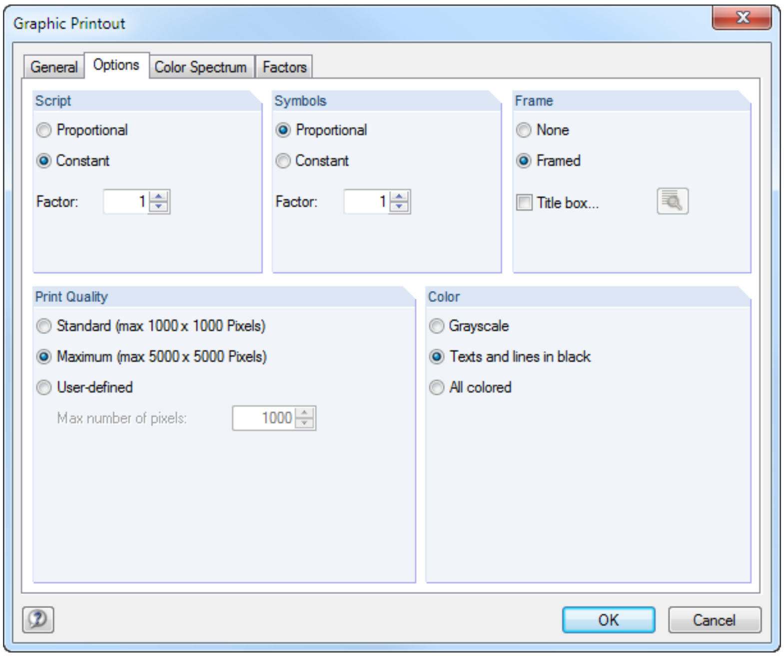


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

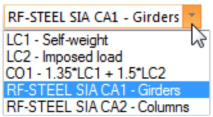
7. General Functions

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

7.1 Design Cases

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

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



Access to the design cases of RF-STEEL SIA is also available in RFEM by using the load case list of the toolbar.

Create a new design case

To create a new design case, use the RF-STEEL SIA menu and click

File → New Case.

The following dialog box appears:

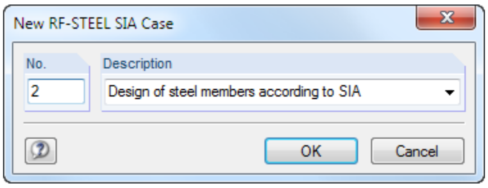


Figure 7.1: Dialog box *New RF-STEEL SIA Case*

In the dialog box, enter a *No.* (which is not yet assigned) for the new design case. The *Description* makes the selection in the load case list easier.

When you click [OK], module window 1.1 *General Data* opens where you can enter the new design data.

Rename a design case

To change the description of a design case, use the RF-STEEL SIA menu and click

File → Rename Case.

The following dialog box appears:

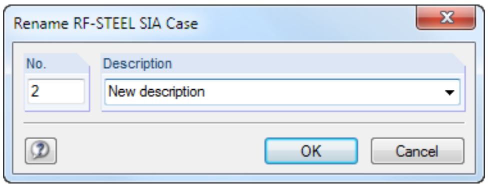


Figure 7.2: Dialog box *Rename RF-STEEL SIA Case*

In this dialog box, you can define a different *Description* as well as a different *No.* for the design case.

Copy a design case

To copy the input data of the current design case, use the RF-STEEL SIA menu and click

File → Copy Case.

The following dialog box appears:

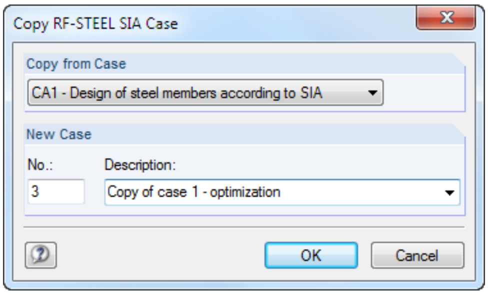


Figure 7.3: Dialog box *Copy RF-STEEL SIA Case*

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

Delete a design case

To delete a design case, use the RF-STEEL SIA menu and click

File → Delete Case.

The following dialog box appears:

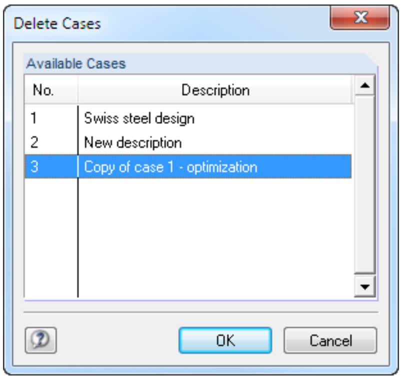
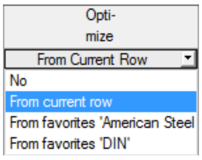


Figure 7.4: Dialog box *Delete Cases*

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

7.2 Cross-Section Optimization



The design module offers you the possibility to optimize overloaded or little utilized cross-sections: Select the relevant cross-sections in module window 1.3 *Cross-Sections* where you can use the list in table column D or E to decide whether the cross-sections *From current row* or the user-defined *favorites* will be determined (see Figure 2.9, page 14). You can also start the cross-section optimization in the results windows by using the context menu.

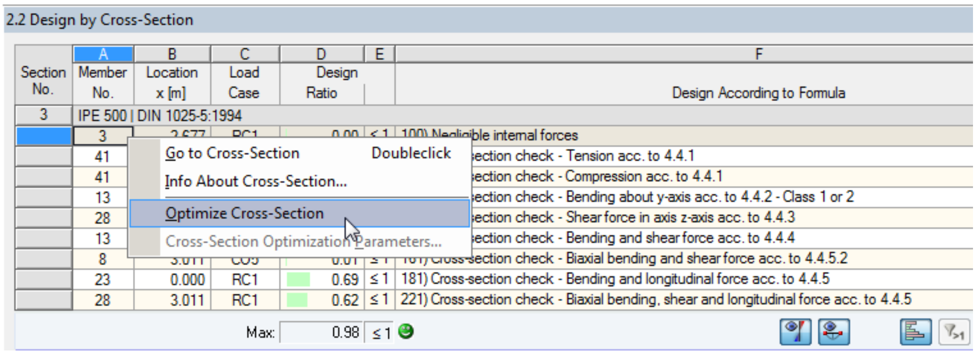


Figure 7.5: Context menu for cross-section optimization

During the optimization the program finds the cross-section that fulfills the analysis requirements in the most optimal way, which means comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.4, page 32). The required cross-section properties are determined with the internal forces from RFEM. If a cross-section proves to be more favorable, it will be used for the design. Then, the graphic in module window 1.3 shows two cross-sections: the original cross-section from RFEM and the optimized one (see Figure 7.7).

When you want to *Optimize* a parametric cross-section, the following dialog box appears.

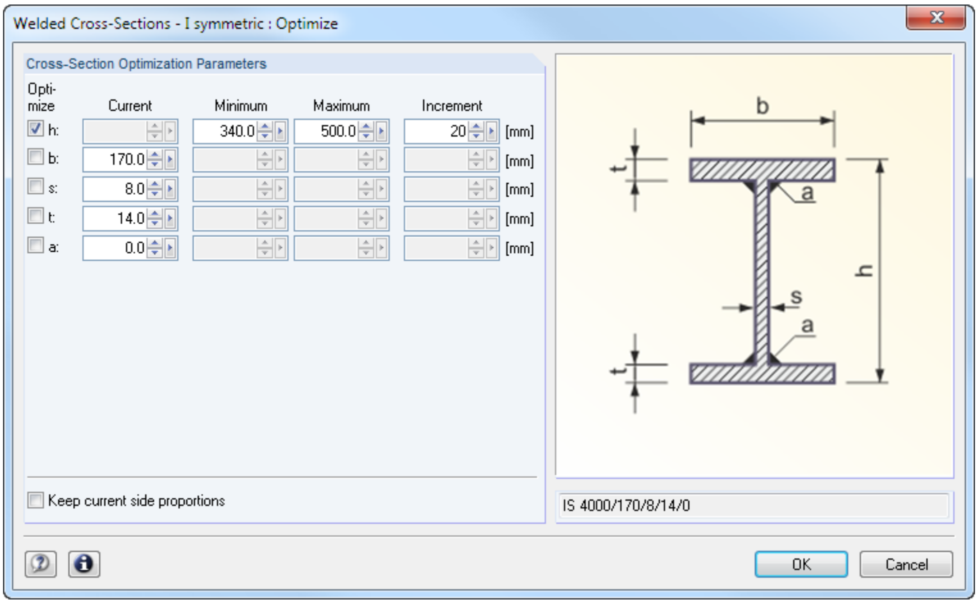


Figure 7.6: Dialog box *Welded Cross-Sections - I symmetric : Optimize*

By ticking the check box(es) in the *Optimize* column, you decide which parameter(s) you want to modify. A ticked box enables the *Minimum* and *Maximum* columns where you can define the upper and lower limits of the parameter. The *Increment* column determines the interval in which the dimensions of the parameter may vary during the optimization process.

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

Cross-sections based on combined rolled cross-sections cannot be optimized.

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

The modified cross-sections can be exported to RFEM: Set module window 1.3 *Cross-Sections*, and then click

Edit → Export All Cross-Sections to RFEM.

Also the context menu in window 1.3 provides options to export optimized cross-sections to RFEM.

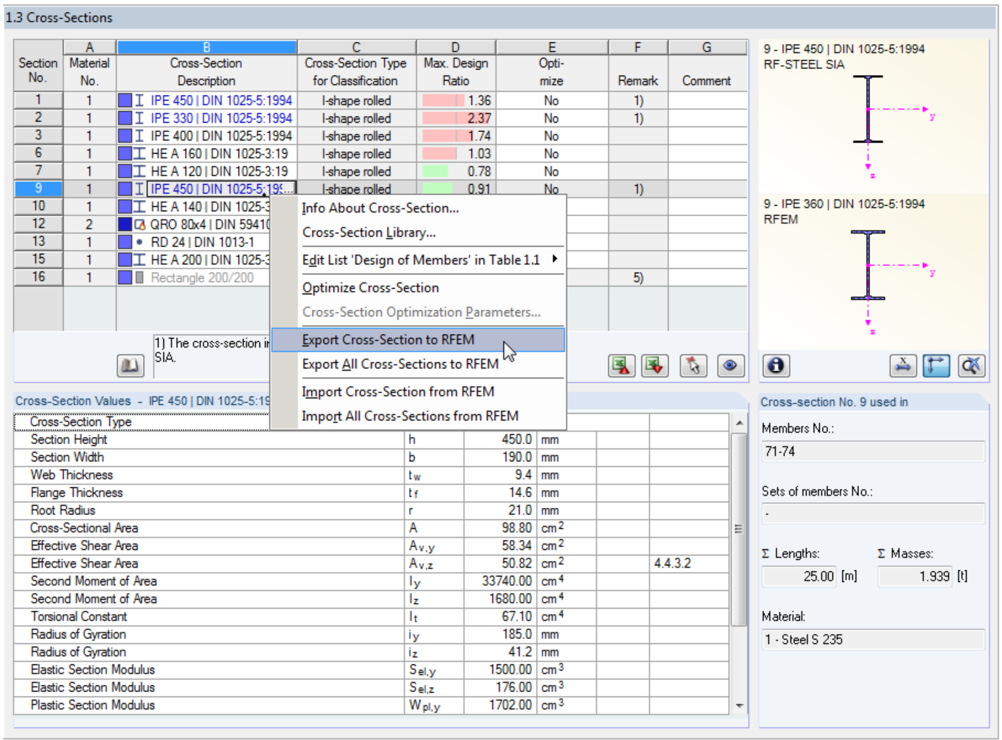


Figure 7.7: Context menu in module window 1.3 *Cross-Sections*

Before the modified cross-sections are transferred to RFEM, a query appears asking if you want to delete the results of RFEM.

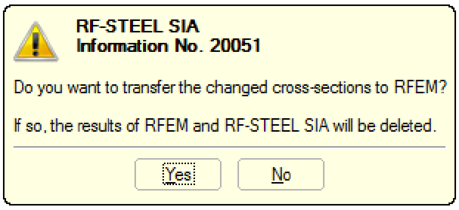


Figure 7.8: Query before transfer of modified cross-sections to RFEM

After starting the [Calculation] in RF-STEEL SIA, the RFEM internal forces and designs will be determined in one calculation run.

If modified cross-sections have not yet been exported to RFEM, it is possible to import the original cross-sections to the design module by using the options shown in Figure 7.7. Please note that this possibility is only available in module window 1.3 *Cross-Sections*.

If you optimize a tapered member, the program optimizes the member start and end locations. Then, it linearly interpolates the second moments of area for the intermediate locations. As these moments are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-section differ considerably. In such a case, it is recommended to divide the taper into several members and thus to represent the taper layout manually.

7.3 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed together. To access the dialog box for adjusting the units, use the RF-STEEL SIA menu and click

Settings → Units and Decimal Places.

The program opens the following dialog box that you already know from RFEM. RF-STEEL SIA is preset in the list *Program / Module*.

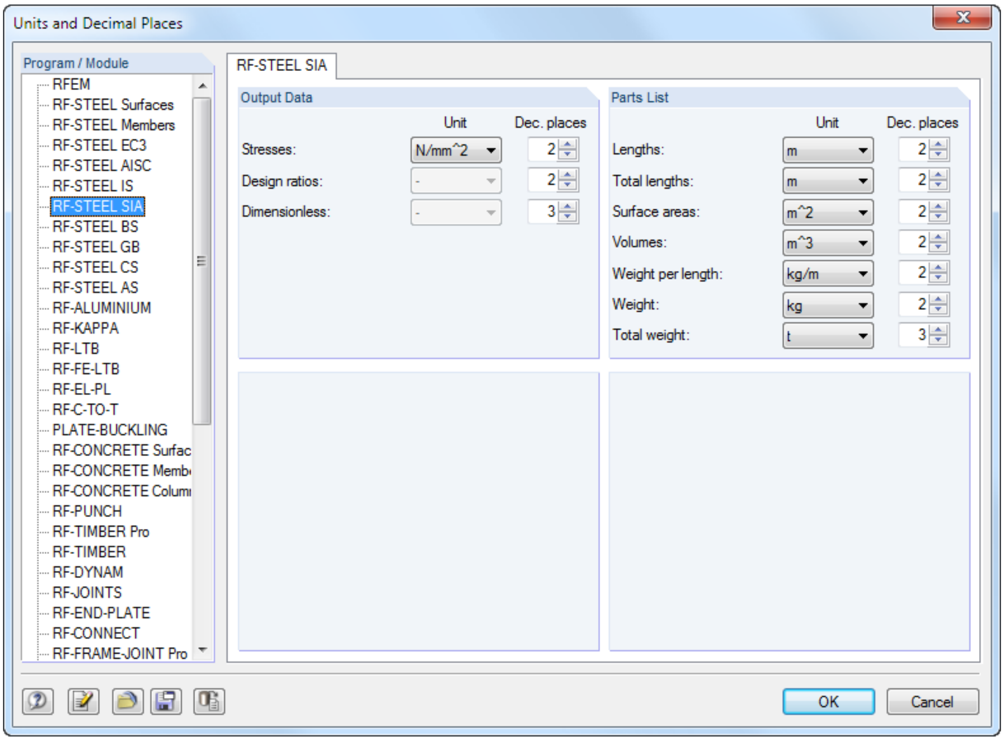


Figure 7.9: Dialog box *Units and Decimal Places*

The settings can be saved as user profile to reuse them in other models. The corresponding functions are described in the RFEM manual, chapter 11.1.3.

7.4 Data Transfer

7.4.1 Export of Material to RFEM

If you adjust materials for the design in RF-STEEL SIA, it is possible to export the modified materials to RFEM like optimized cross-sections: Open module window 1.2 *Materials*, and then click

Edit → Export All Materials to RFEM.

Materials can also be exported to RFEM by using the context menu of module window 1.2.

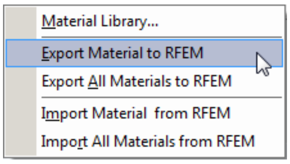


Figure 7.10: Context menu of module window 1.2 *Materials*

Before the modified materials are transferred to RFEM, a query appears asking if you want to delete the results of RFEM. After starting the [Calculation] in RF-STEEL SIA, the RFEM internal forces and designs will be determined in one calculation run.

If modified materials have not yet been exported to RFEM, it is possible to import the original materials to the design module by using the options shown in Figure 7.10. Please note that this possibility is only available in module window 1.2 *Materials*.

7.4.2 Export of Effective Lengths to RFEM

If you adjust the buckling lengths for the designs in RF-STEEL SIA, it is also possible to export the modified effective lengths to RFEM: Open module window 1.5 *Effective Lengths - Members*, and then click

Edit → Export All Effective Lengths to RFEM.

Effective lengths can also be exported to RFEM by using the context menu of window 1.5.

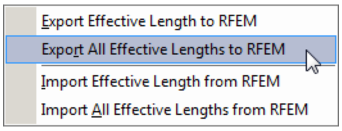


Figure 7.11: Context menu of module window 1.5 *Effective Lengths - Members*

Before the modified buckling lengths are transferred to RFEM, a query appears asking if you want to delete the results of RFEM.

If modified effective lengths have not yet been exported to RFEM, it is possible to import the original effective lengths to the design module by using the options shown in Figure 7.11. Please note that this possibility is only available in module windows 1.5 *Effective Lengths - Members* and 1.6 *Effective Lengths - Sets of Members*.

7.4.3 Export of Results

The results of RF-STEEL SIA can also be used in other programs.

Clipboard

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

Printout report

Data of the add-on module RF-STEEL SIA can be printed in the printout report (see chapter 6.1, page 50) where it can be exported. Then, use the printout report menu and click

File → Export to RTF.

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

Excel / OpenOffice

RF-STEEL SIA provides a function for the direct data export to MS Excel, OpenOffice.org Calc or the file format CSV. To open the corresponding dialog box, use the RF-STEEL SIA menu and click

File → Export Tables.

The following export dialog box appears.

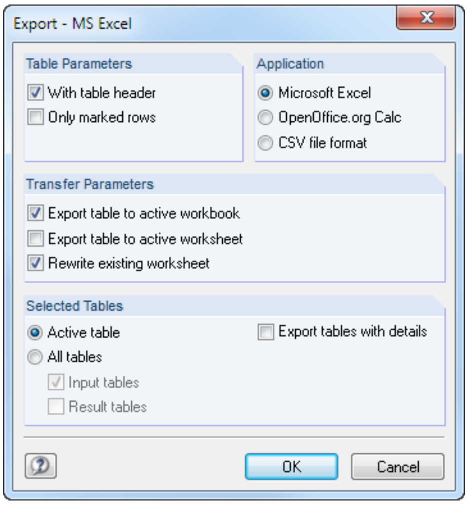


Figure 7.12: Dialog box *Export - MS Excel*

When the settings are selected, you can start the export by clicking [OK]. Excel or OpenOffice will start automatically. It is not necessary to run the programs in the background.

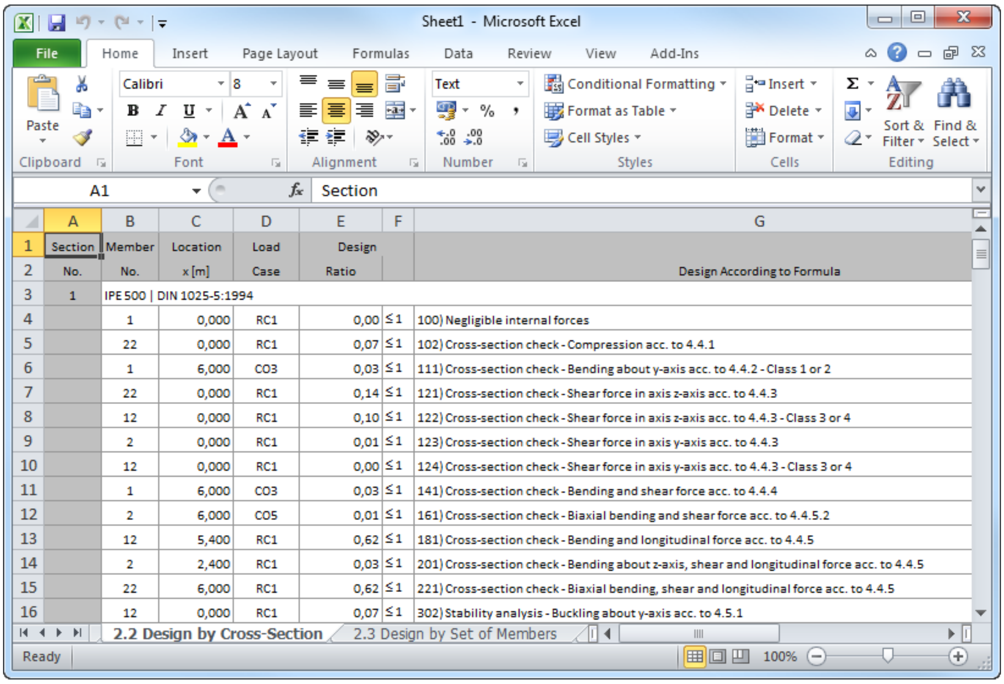


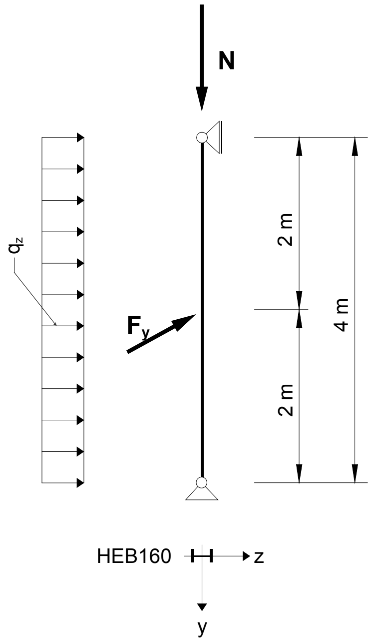
Figure 7.13: Result in *Excel*

8. Example

We have a column with double-bending for which we perform stability designs for flexural buckling and lateral buckling considering the interaction conditions.

Design values

System and loads



Design values of the static loads

N_d = 300 kN

$q_{z,d}$ = 5.0 kN/m

$F_{y,d}$ = 7.5 kN

Figure 8.1: System and design loads (γ-fold)

Internal forces according to linear static analysis

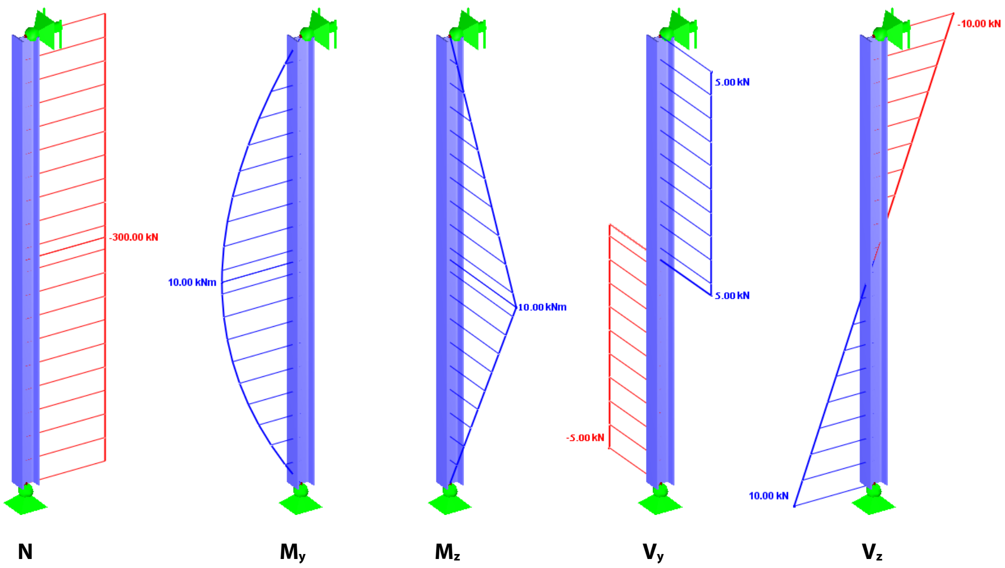


Figure 8.2: Internal forces

Design location (decisive x-location)

The design is performed for all x-locations (see chapter 4.5) of the equivalent member. The decisive location is x = 2.00 m. RFEM determines the following internal forces:

$N = -300.00$ kN $M_y = 10.00$ kNm $M_z = 10.00$ kNm $V_y = 5.00$ kN $V_z = 0.00$ kN

Cross-section properties HE-B 160, S 235

Property	Symbol	Value	Unit
Cross-sectional area	A	54.30	cm²
Moment of inertia	I_y	2490.00	cm⁴
Moment of inertia	I_z	889.00	cm⁴
Governing radius of gyration	r_y	6.78	cm
Governing radius of gyration	r_z	4.05	cm
Polar radius of gyration	r_o	7.90	cm
Weight	wt	42.63	kg/m
Torsional constant	J	31.40	cm⁴
Warping constant	C_w	47940.00	cm⁶
Elastic section modulus	S_y	311.00	cm³
Elastic section modulus	S_z	111.00	cm³
Plastic section modulus	Z_y	354.00	cm³
Plastic section modulus	Z_z	169.96	cm³
Buckling curve	BC_y	b	
Buckling curve	BC_z	c	

Flexural buckling about minor axis (⊥ to z-z axis)

$$N_{cr,z} = \frac{21000 \cdot 889.00 \cdot \pi^2}{400.00^2} = 1151.60\,\text{kN}$$

$$\sigma_{cr,K,z} = \frac{1151.6}{54.30} = 21.21\,\text{kN/cm}^2$$

$$\bar{\lambda}_{K,z} = \sqrt{\frac{f_y}{\sigma_{cr,K,z}}} = \sqrt{\frac{23.5}{21.21}} = 1.053$$

Structural steel S 235 t ≤ 100 mm

[1] Figure 7: buckling stress curve c $\Rightarrow$ $\alpha_z = 0.49$ (table 7)

$$\Phi = 0.5 \cdot \left[1 + 0.49 \cdot (1{,}053 - 0.2) + 1.053^2\right] = 1.263$$

$$\chi_{K,z} = \frac{1}{1.263 + \sqrt{1.263^2 - 1.053^2}} = 0.510$$

$$N_{K,z,Rd} = 0.510 \cdot 54.30 \cdot \frac{23.5}{1.05} = 619.81\,\text{kN}$$

$$\frac{N_{Ed}}{N_{K,z,Rd}} = \frac{300}{619.81} = 0{,}484$$

Result values from RF-STEEL SIA calculation

Second moment of area	I_z	889.00	cm^4	
Effective member length	$L_{K,z}$	4.000	m	
Euler critical stress	$\sigma_{cr,K,z}$	21.21	kN/cm^2	
Related buckling slenderness	$\lambda_{\_K,z}$	1.053		
Buckling curve	BC_z	c		Figure 7
Imperfection factor	α_z	0.490		
Auxiliary factor	$\Phi_{K,z}$	1.263		
Reduction factor	$\chi_{K,z}$	0.510		Eq. (16)
Flexural buckling resistance	$N_{K,z,Rd}$	619.81	kN	Eq. (15)

Flexural buckling about the major axis (⊥ to y-y axis)

$$N_{cr,y} = \frac{21000 \cdot 2490.00 \cdot \pi^2}{400.00^2} = 3225.51\,kN$$

$$\sigma_{cr,K,y} = \frac{3225.51}{54.30} = 59.40\ kN \cdot cm^{-2}$$

$$\overline{\lambda}_{K,y} = \sqrt{\frac{f_y}{\sigma_{cr,K,y}}} = \sqrt{\frac{23.5}{59.40}} = 0.629$$

Structural steel S 235 $\quad$ t ≤ 100 mm

[1] Figure 7: buckling stress curve b $\quad \Rightarrow \quad \alpha_y = 0.34$ (table 7)

$$\Phi = 0.5 \cdot \left[1 + 0.34 \cdot (0{,}629 - 0.2) + 0.629^2\right] = 0.771$$

$$\chi_{K,y} = \frac{1}{0.771 + \sqrt{0.771^2 - 0.629^2}} = 0.822$$

$$\frac{N_{Ed}}{\chi_{K,y} \cdot A \cdot f_y / \gamma_{M1}} = \frac{300}{0.822 \cdot 54.30 \cdot 23.5 / 1.05} = 0.300$$

Result values from RF-STEEL SIA calculation

Second moment of area	I_y	2490.00	cm^4	
Effective member length	$L_{K,y}$	4.000	m	
Euler critical stress	$\sigma_{cr,K,y}$	59.40	kN/cm^2	
Cross-sectional area	A	54.30	cm^2	
Yield strength	f_y	23.50	kN/cm^2	Table 1
Related buckling slenderness	$\lambda_{\_K,y}$	1.053		
Buckling curve	BC_y	b		Figure 7
Imperfection factor	α_y	0.340		
Auxiliary factor	$\Phi_{K,y}$	0.7713		
Reduction factor	$\chi_{K,y}$	0.822		
Flexural buckling resistance	$N_{K,y,Rd}$	999.27	kN	Eq. (15)

Lateral buckling

Ideal moment for lateral buckling

The ideal moment for lateral buckling is determined by assuming the support to be hinged and free to warp.

The point of load application is assumed to be in the shear center (the application point for transverse loads can be adjusted in *Details* dialog box, see chapter 3.1.2, page 29).

$$M_{cr.0} = \frac{\pi^2 \cdot E \cdot I_z}{L^2} \cdot \sqrt{\frac{I_\omega}{I_z} + \frac{L^2 \cdot G \cdot I_t}{\pi^2 \cdot E \cdot I_z}}$$

$$M_{cr.0} = \frac{\pi^2 \cdot 21000 \cdot 889}{400^2} \cdot \sqrt{\frac{47940}{889} + \frac{400^2 \cdot 8100 \cdot 31.40}{\pi^2 \cdot 21000 \cdot 889}} = 190.90 \text{ kNm}$$

$$M_{cr} = C_1 \cdot M_{cr.0} = 1.13 \cdot 190.90 = 215.71 \text{kNm}$$

The program also shows $M_{cr,0}$, which is determined assuming a constant moment distribution.

For the results evaluated by x-location, the program also shows the values $M_{cr,x}$. They represent the ideal moments for lateral buckling on the locations x in relation to the moments for lateral buckling on the location of the maximum moment. With $M_{cr,x}$ the program calculates the relative slenderness $\overline{\lambda}_{LT}$.

Slenderness ratio for lateral buckling

Calculation according to [1], chapter 4.5.2 for location with maximum moment at x = 2.00 m:

HEB-160, cross-section class 1: $W_y \Rightarrow W_{pl,y} = 354.0 \text{ cm}^3$

$$\overline{\lambda}_D = \sqrt{\frac{W_y \cdot f_y}{M_{cr}}} = \sqrt{\frac{354 \cdot 23.5}{215.71}} = 0.621$$

Design value of moments for lateral buckling

Calculation according to [1], chapter 4.5.2

HEB-160: Rolled cross-section $\Rightarrow$ α_D = 0.21 (corresponds to buckling stress curve a)

- **$M_{D,Rd}$**

$$\Phi_D = 0.5 \cdot \left[1 + \alpha_D \cdot \left(\overline{\lambda}_D - 0.2\right) + \overline{\lambda}_D{}^2\right] = 0.5 \cdot \left[1 + 0.21 \cdot (0.621 - 0.2) + 0.621^2\right] = 0.737$$

$$\chi_D = \frac{1}{\Phi_D + \sqrt{\Phi_D{}^2 - \overline{\lambda}_D{}^2}} = \frac{1}{0.737 + \sqrt{0.737^2 - 0.621^2}} = 0.882$$

$$M_{D,Rd} = \chi_D \cdot W_y \cdot \frac{f_y}{\gamma_{M1}} = 0.882 \cdot 354.0 \cdot \frac{235}{1.05} = 69.87 \text{ kNm}$$

- **$M_{D,Rd,min}$**

$$\overline{\lambda}_{D,min} = \sqrt{\frac{W_y \cdot f_y}{M_{cr,0}}} = \sqrt{\frac{354 \cdot 23.5}{190,90}} = 0,660$$

$$\Phi_{D,min} = 0.5 \cdot \left[1 + 0.21 \cdot (0.660 - 0.2) + 0.660^2\right] = 0.766$$

$$\chi_{D,min} = \frac{1}{0.766 + \sqrt{0.766^2 - 0.660^2}} = 0.866$$

$$M_{D,Rd,min} = 0.866 \cdot 354.0 \cdot \frac{235}{1.05} = 68.59\,\text{kNm}$$

Stability of single members with double symmetrical I-sections

Determination according to [1], chapter 5.1.7

Factors to consider distribution of moments: $\omega_y = \omega_z = 1.00$

$$M_{z,Rd} = W_z \cdot \frac{f_y}{\gamma_{M1}} = 169.96 \cdot \frac{23.5}{1.05} = 38.04\,\text{kNm}$$

$$N_{Rd} = A \cdot \frac{f_y}{\gamma_{M1}} = 54.30 \cdot \frac{23.5}{1.05} = 1215.29\,\text{kN}$$

$$M_{y,red,Rd} = M_{D,Rd,min} \cdot \left(1 - \frac{N_{Ed}}{N_{Kz,Rd}}\right) \cdot \left(1 - \frac{N_{Ed}}{N_{cr,y}}\right)$$

$$M_{y,red,Rd} = 68.59 \cdot \left(1 - \frac{300}{619.81}\right) \cdot \left(1 - \frac{300}{3225.51}\right) = 32.10\,\text{kNm} \leq \omega_y \cdot M_{D,Rd} = 69.87\,\text{kNm}$$

$$M_{z,red,Rd} = M_{z,Rd} \cdot \left(1 - \frac{N_{Ed}}{N_{Kz,Rd}}\right) \cdot \left(1 - \frac{N_{Ed}}{N_{cr,z}}\right)$$

$$M_{z,red,Rd} = 38.04 \cdot \left(1 - \frac{300}{619.81}\right) \cdot \left(1 - \frac{300}{1151.6}\right) = 14.52\,\text{kNm}$$

$$\beta = 0.4 + \frac{N_{Ed}}{N_{Rd}} + \frac{b}{h - t_f} = 0.4 + \frac{300}{1215.29} + \frac{16}{16 - 1.3} = 1.735$$

$$\left(\frac{M_{y,Ed}}{M_{y,red,Rd}}\right)^{\beta} + \left(\frac{M_{z,Ed}}{M_{y,red,Rd}}\right)^{\beta} = \left(\frac{10}{32.10}\right)^{1.735} + \left(\frac{10}{14.52}\right)^{1.735} = \underline{\underline{0.66 < 1.00}}$$

Result values from RF-STEEL SIA calculation

Modulus of elasticity	E	21000.00	kN/cm²	
Second moment of area	I_y	2490.00	cm⁴	
Effective member length	$L_{K,y}$	4.000	m	
Ideal elastic flexural buckling force	$N_{cr,y}$	3225.510	kN	
Second moment of area	I_z	889.00	cm⁴	
Effective member length	$L_{K,z}$	4.000	m	
Cross-sectional area	A	54.30	cm²	
Euler critical stress	$\sigma_{cr,K,z}$	21.21	kN/cm²	
Ideal elastic flexural buckling force	$N_{cr,z}$	1151.600	kN	
Yield strength	f_y	23.50	kN/cm²	Tab. 1
Related buckling slenderness	$\lambda_{\_K,z}$	1.053		
Buckling curve	BC_z	c		Figure 7
Imperfection factor	α_z	0.490		Tab. 7
Auxiliary factor	$\Phi_{K,z}$	1.263		
Reduction factor	$\chi_{K,z}$	0.510		Eq. (16)
Resistance coefficient	γ_{M1}	1.050		4.1.3
Design value for flexural buckling resistance	$N_{K,z,Rd}$	619.810	kN	Eq. (15)
Axial force (compression)	N_{Ed}	300.000	kN	
Minimum member end moment	$M_{y,Ed,min}$	0.000	kNm	5.1.7.1
Maximum member end moment	$M_{y,Ed,max}$	0.000	kNm	5.1.7.1
Factor	ω_y	1.000		5.1.7.1
Modulus of elasticity	E	21000.00	kN/cm²	
Shear modulus	G	8100.00	kN/cm²	
Length factor	k_z	1.000		
Length factor	k_w	1.000		
Length	L	4.000	m	
Warping constant	C_w	47940.00	cm⁶	
Torsional constant	J	31.40	cm⁴	
Ideal elastic critical moment for lateral-torsional buckling to determine related slenderness	$M_{cr,x}$	190.896	kNm	
Elastic section modulus	S_y	354.00	cm³	
Yield strength	f_y	23.50	kN/cm²	Tab. 1
Related slenderness for lateral buckling	$\lambda_{\_D,min}$	0.660		4.5.2.3
Auxiliary factor	$\Phi_{D,min}$	0.766		4.5.2.3
Reduction factor	$\chi_{D,min}$	0.866		
Resistance coefficient	γ_{M1}	1.050		4.1.3
Lateral-torsional buckling resistance	$M_{D,Rd,min}$	68.586	kNm	Eq. (17)

Moment distribution	Diagr M_y	6) Parabola		
Max. field moment	$M_{y,max}$	10.000	kNm	
Boundary moment	$M_{y,A}$	0.000	kNm	
Moment ratio	ψ	0.000		
Moment coefficient	C_1	1.130		
Ideal critical moment for lateral buckling	M_{cr}	215.712	kNm	
Elastic section modulus	S_y	354.00	cm^3	
Yield strength	f_y	23.50	kN/cm^2	Tab. 1
Related slenderness for lateral buckling	λ_D	0.621		4.5.2.3
Imperfection factor	α_D	0.210		4.5.2.3
Auxiliary factor	Φ_D	0.737		4.5.2.3
Reduction factor	χ_D	0.882		Eq. (18)
Resistance coefficient	γ_{M1}	1.050		4.1.3
Lateral-torsional buckling resistance	$M_{D,Rd}$	69.868	kNm	Eq. (17)
Moment resistance	$M_{y,red,Rd}$	32.098	kNm	5.1.7.2
Moment	$M_{y,Ed}$	10.000	kNm	
Moment component	η_{My}	0.31		Eq. (52)
Minimum member end moment	$M_{z,Ed,min}$	0.000	kNm	5.1.7.1
Maximum member end moment	$M_{z,Ed,max}$	0.000	kNm	5.1.7.1
Factor	ω_z	1.000		5.1.7.1
Moment	$M_{z,Ed}$	10.000	kNm	
Elastic section modulus	S_z	169.96	cm^3	
Moment resistance	$M_{z,Rd}$	38.039	kNm	Eq. (9)
Moment resistance	$M_{z,red,Rd}$	14.515	kNm	5.1.7.2
Moment component	η_{Mz}	0.69		Eq. (52)
Section width	b	160.0	mm	
Flange thickness	t_f	13.0	mm	
Section height	h	160.0	mm	
Normal force resistance	N_{Rd}	1215.290	kN	Eq. (6)
Exponent	β	1.735		5.1.7.2
Design ratio	η	**0.66**	**< 1**	Eq. (52)

A Literature

[1] SIA 263:2003 Bauwesen – Stahlbau

[2] SIA 260:2003 Bauwesen – Grundlagen der Projektierung von Bauwerken

[3] Stahlbau: Grundbegriffe und Bemessungsverfahren, Hirt M., Bez R., Ernst & Sohn 1998

[4] Tragwerke aus Stahl nach Eurocode 3, Werner, 1. Auflage 1996

[5] The Behaviour and Design of Steel Structures to EC3 , Trahair N.S., Bradford M.A., Nethercot D.A., Gardner L., Taylor & Francis Ltd 2007

[6] Rules for Member Stability in EN 1993-1-1, ECCS Technical Committee 8 – Stability

B Index