Dlubal

Version
March 2014

Add-on Module

RF-STEEL SANS

Ultimate Limit State and Serviceability Limit State Design According to SANS 10162-1:2011

Program Description

© Dlubal Software GmbH
Am Zellweg 2 D-93464 Tiefenbach

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

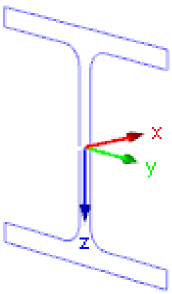
Contents

1. Introduction

1.1 Additional Module RF-STEEL SANS

The South African National Standard *SANS 10162-1* determines rules for the design, analysis and construction of steel buildings in South Africa. With the add-on module RF-STEEL SANS from the DLUBAL SOFTWARE GMBH company all users obtain a highly efficient and universal tool to design steel structures modeled with member elements according to this standard.

All typical designs of load capacity, stability and deformation are carried out in the module RF-STEEL SANS. Different actions are taken into account during the load capacity design. It is also possible to choose among the interaction formulae mentioned in the code. The slenderness types of the cross-sections are taken into account automatically. Thus, it is possible to check the limitation of the design capacity and of the rotational capacity due to local buckling for cross-section parts. RF-STEEL SANS automatically calculates the limiting width-to-thickness ratios of compressed parts and classifies the cross-section.



Axis system

The sectional coordinate system in RF-STEEL SANS is different from the indices used in the South African standard. It corresponds to the one used in RFEM: The indices "y" and "z" refer to the axes in the cross-section plane as seen in the image to the left.

For the stability design, you can determine for every single member or set of members whether buckling is possible in the direction of y-axis and/or z-axis. Lateral supports can be added for a realistic representation of the structural model. The specific ratios of slendernesses and critical stresses are automatically determined by RF-STEEL SANS on the basis of the boundary conditions. The location where the loads are applied, which influences the lateral torsional design, can be defined in the detailed settings.

The serviceability limit state has become important for the static calculation of modern civil engineering as more and more slender cross-sections are being used. In RF-STEEL SANS, load cases, load combinations and result combinations can be arranged individually to cover the various design situations. It is also possible to specify reference lengths and precambers for the design.

If required, you can optimize the cross-sections and export the modified cross-sections to RFEM. Using the design cases, you can design separate structural components in complex structures or analyze variants.

RF-STEEL SANS is an add-on module integrated in RFEM. For this reason, the design relevant input data is already preset when you have started the 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, from the calculation of the internal forces to all design details.

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

Your DLUBAL Team

1.2 RF-STEEL SANS Team

The following people participated in the development of the RF-STEEL SANS module:

Program coordinators

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

Programmers

Ing. Zdeněk Kosáček
Dipl.-Ing. Georg Dlubal
Dr.-Ing. Jaroslav Lain
Ing. Martin Budáč
Mgr. Petr Oulehle
Dr.-Ing. Jaroslav Lain
Ing. Zbyněk Zámečník
Ing. Jakub Melka

Library of cross-sections and materials

Ing. Ph.D. Jan Rybín
Stanislav Krytinář
Jan Brnušák

Design of program, dialog boxes and icons

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

Testing and technical support

Ing. Ph.D. Martin Čudejko
Ing. Martin Hlavačka
Dipl.-Ing. (BA) Markus Baumgärtel
M.Sc. Sonja von Bloh
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Wieland Götzler
Dipl.-Ing. (FH) Ulrich Lex
Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

Manual

Ing. Ph.D. Martin Čudejko
Dipl.-Ing. (FH) Robert Vogl
Bc. Chelsea Jennings

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 SANS.

The descriptions in this manual follow the sequence and structure of the module's input and results windows. 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, windows, 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 Starting RF-STEEL SANS

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

Menu

To start the program in the RFEM menu bar, click

Add-on Modules → Design - Steel → RF-STEEL SANS.

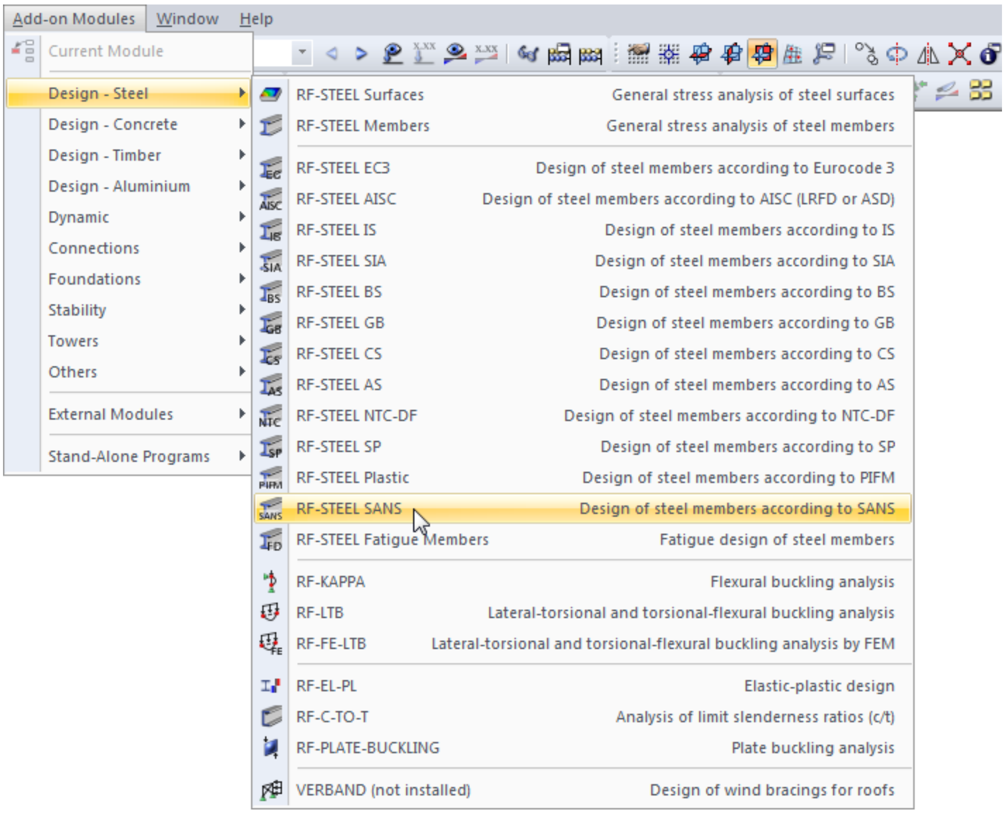


Figure 1.1: Main Menu: *Additional Modules → Design - Steel → RF-STEEL SANS*

Navigator

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

Add-on Modules → RF-STEEL SANS.

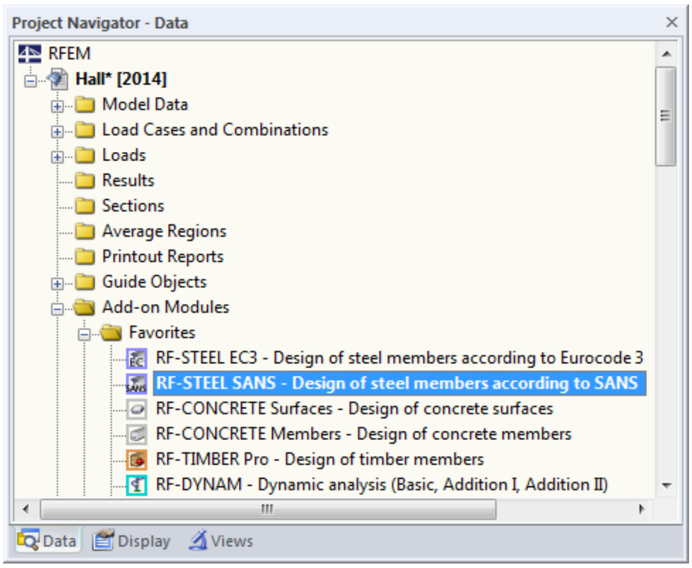
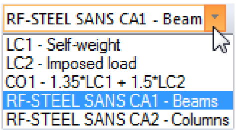


Figure 1.2: Data Navigator: *Add-on Modules → RF-STEEL SANS*

Panel



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

Set the relevant RF-STEEL SANS design case in the load case list of the RFEM toolbar. Then click 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 SANS] in the panel to open the module.

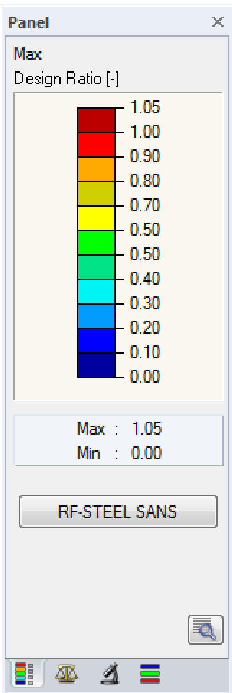


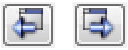
Figure 1.3: Panel button [RF-STEEL SANS]

2. Input Data

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

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

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



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

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

2.1 General Data

In 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.

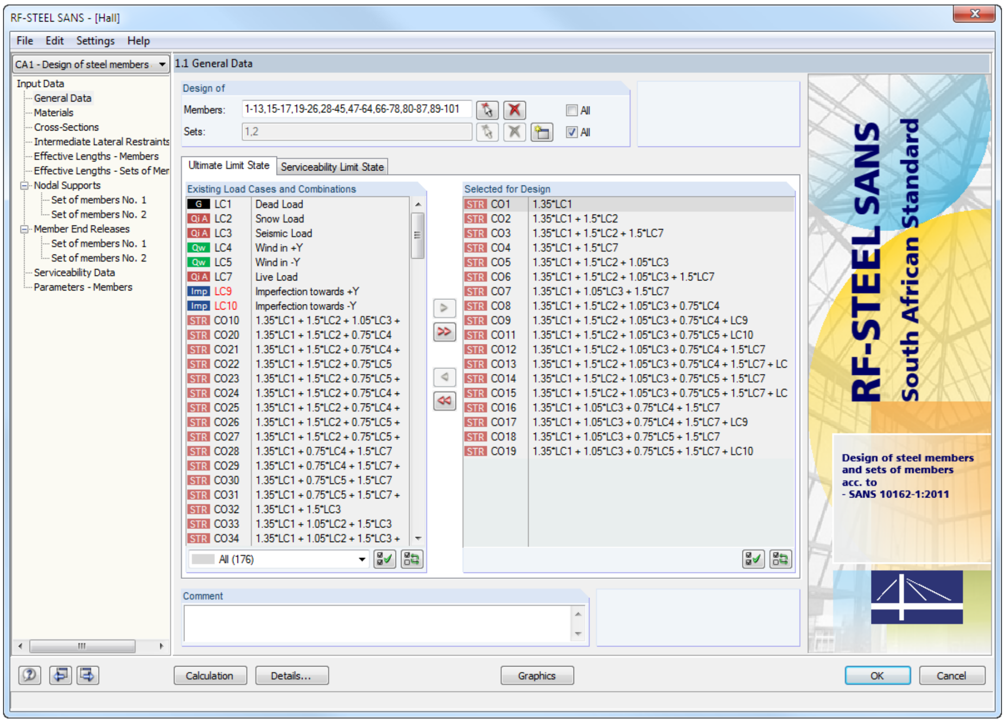


Figure 2.1: Window 1.1 *General Data*

Design of

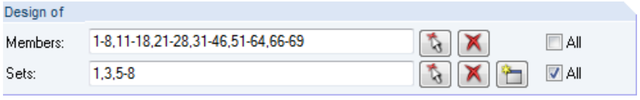


Figure 2.2: Design of members and sets of members

The design can be carried out for *Members* as well as for *Sets of Members*. If you want to design only selected objects, clear the *All* check box: Then you can access the text boxes to enter the numbers of the relevant members or sets of members. The list of the numbers preset in the field can be cleared by clicking the [Delete] button. Alternatively, you can select the objects graphically in the RFEM work window after clicking [↖].

When you design a set of members, the program determines the extreme values of the 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 result 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.

2.1.1 Ultimate Limit State

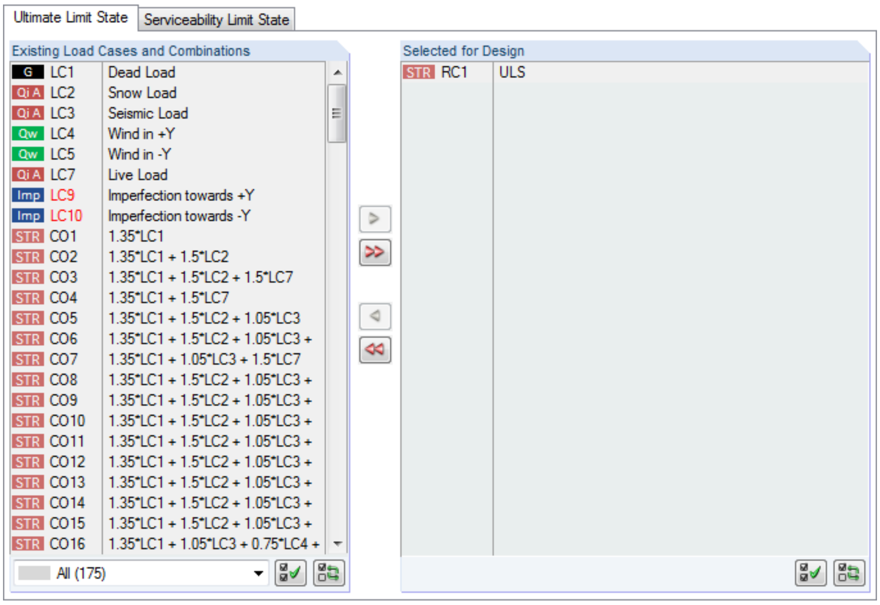


Figure 2.3: Window 1.1 *General Data*, *Ultimate Limit State* tab

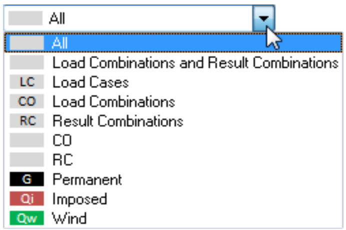
Existing Load Cases and Combinations

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

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 at once, select them while pressing the [Ctrl] key, as common for Windows applications.

Load cases written in red like load cases 9 and 10 in Figure 2.3 cannot be designed: Those load cases are defined without any load data, or they contain only imperfections. When you transfer such a load case, a warning appears.



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

	Select all cases in the list.
	Invert selection of load cases.

Table 2.1: Buttons in tab *Ultimate Limit State*

Selected for Design

The column on the right lists the load cases, load and result combinations 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 load combinations. However, the analysis of a result combination has also disadvantages: First, the influence of the contained loads is difficult to discern.

Second, for the determination of the critical elastic moment for lateral-torsional buckling, the envelope of the moment distributions is analyzed, from which the most unfavorable distribution (max or min) is taken. However, this distribution only rarely reflects the moment distribution in the individual load combinations. Thus, in the case of a RC design, more unfavorable values for the critical elastic moment are to be expected, leading to higher ratios.

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

2.1.2 Serviceability Limit State

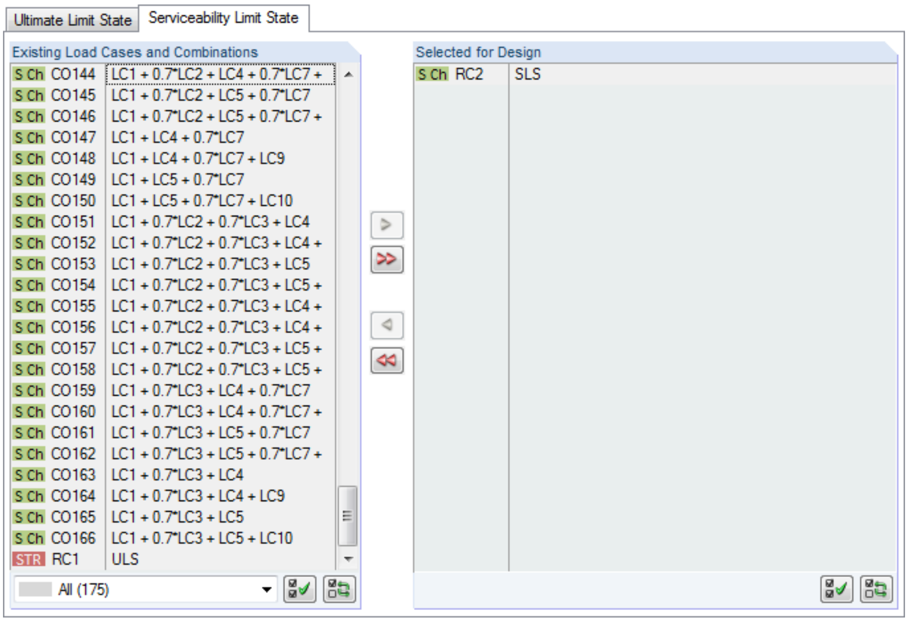


Figure 2.4: Window 1.1 *General Data*, *Serviceability Limit State* tab

Existing Load Cases and Combinations

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

Selected for Design

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

The limit values of the deformations are controlled by the settings in the *Details* dialog box (see Figure 3.3, page 31) which you can call up by clicking the [Details] button.

In the window 1.9 *Serviceability Data*, the reference lengths decisive for the deformation check are managed (see chapter 2.9, page 26).

2.2 Materials

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

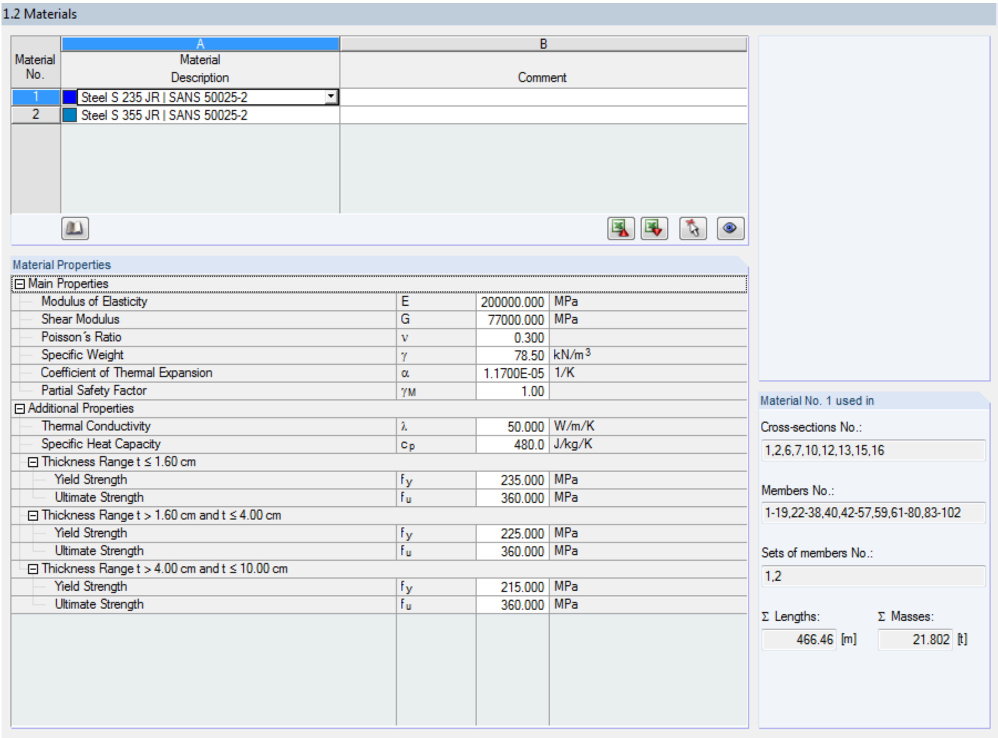


Figure 2.5: Window 1.2 *Materials*

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

The material properties required for the determination of internal forces are described in chapter 4.3 of the RFEM manual (*Main Properties*). The material properties required for design are stored in the global material library. 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 already preset, but it is always possible to modify them. To do this, click the material in column A. Then click [▼] or press function key [F7] to open the material list.

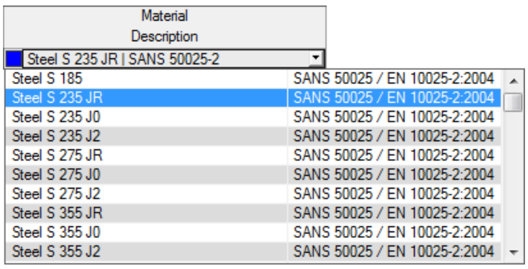


Figure 2.6: List of materials

According to the design concept of SANS 10162-1 [1], you can select only materials of the "Steel" category.

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

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

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

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.

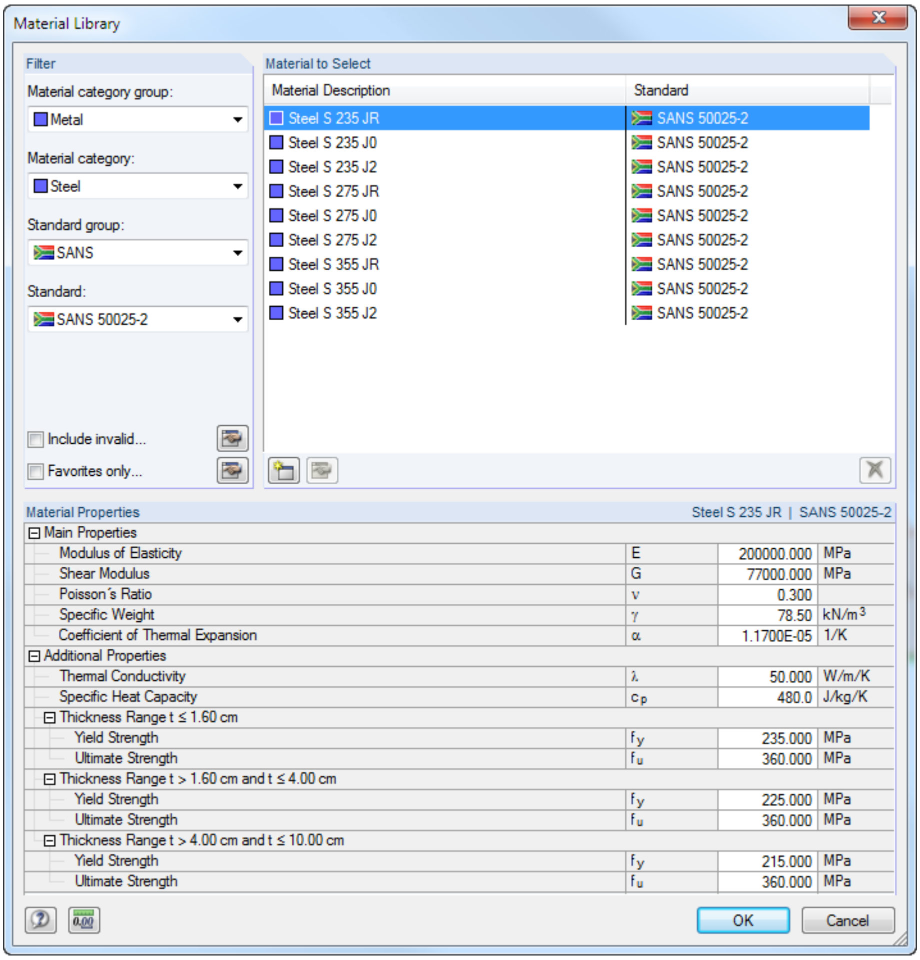


Figure 2.7: Dialog box *Material Library*

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

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

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

You can also select material categories like *Cast Iron* or *Stainless Steel*. Please check, however, whether these materials are covered by the design concept of the Code [1].

2.3 Cross-Sections

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

Coordinate System

The sectional coordinate system in RF-STEEL SANS is different from the indices used in the South African standard. It corresponds to the one used in RFEM (see image in Figure 2.8): The **y**-axis is the major principal axis of the cross-section, the **z**-axis the minor axis. This coordinate system is used for both the input data and the results.

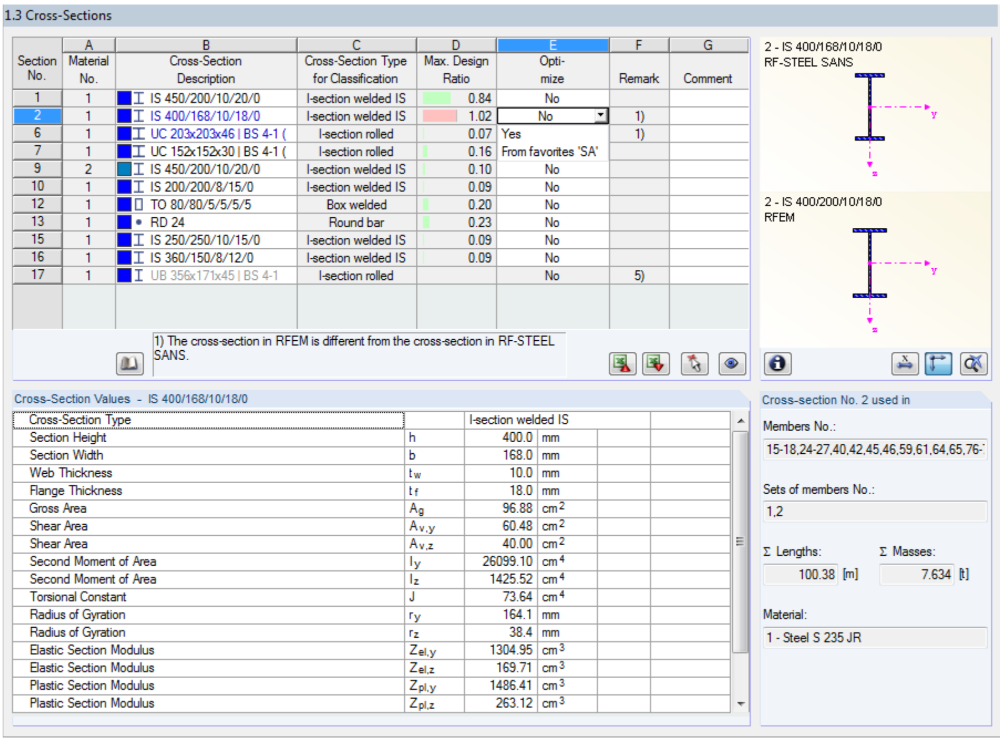


Figure 2.8: Window 1.3 *Cross-Sections*

Cross-Section Description

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

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

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

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

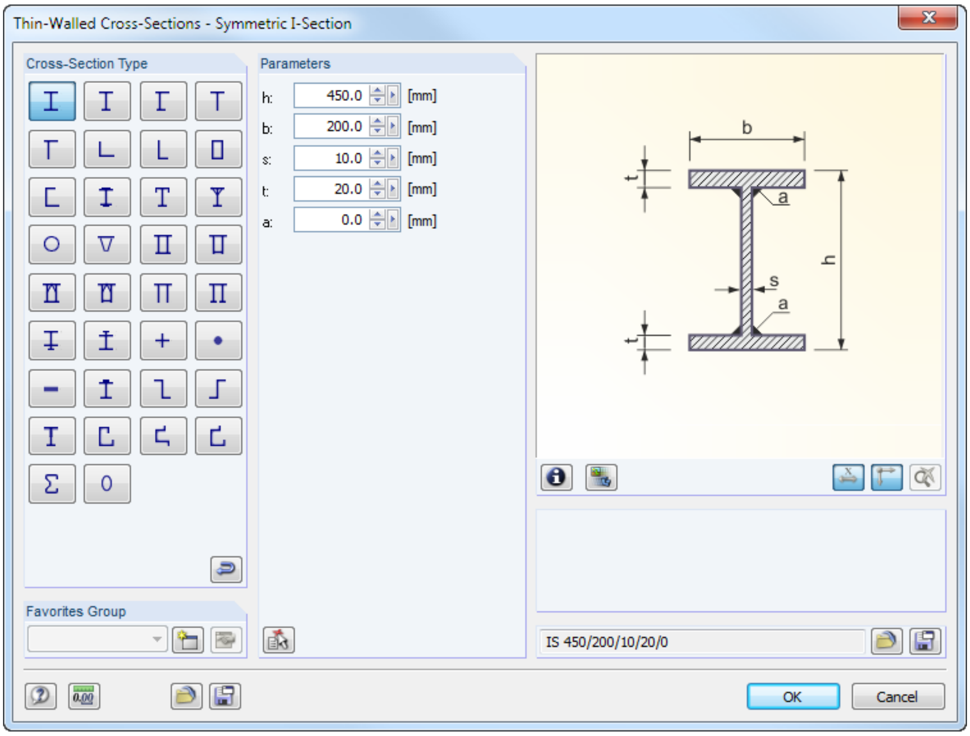
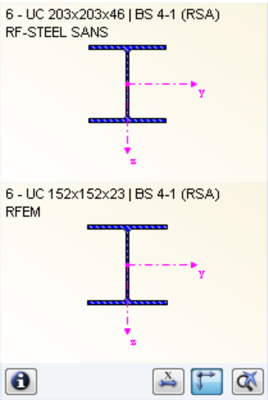


Figure 2.9: IS cross-sections in the cross-section library



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

A modified cross-section will be highlighted in blue.

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

Cross-Section Type for Classification

The cross-section type used for the classification is displayed, e.g. I-shape rolled, welded, box, round bar, etc. The cross-sections listed in [1] tables 3 and 4 can be designed plastically or elastically depending on the class. Cross-sections that are not covered by the standard are classified as *General*.

Max. Design Ratio

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

Optimize

You can optimize every cross-section from the library: For the RFEM internal forces, the program searches the cross-section in the same table that comes as close as possible to a user-defined maximum ratio. The maximum ratio can be defined in the *Details* dialog box (see Figure 3.4, page 32).

If you want 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.

A warning might appear before the calculation: *Incorrect type of cross-section!* This means that there is a cross-section that is not registered in the data base. This may be a user-defined cross-section, or a SHAPE-THIN cross-section that has not been calculated yet. To select an appropriate cross-section for design, click [Library] (see description after Figure 2.8).

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 rows, in accordance with the definition in RFEM.

RF-STEEL SANS also designs tapered members, provided that the cross-section at the start of the member has the same number of stress points as the cross-section at the end of the member. For example, the normal stresses 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 have a different number of stress points, the intermediate values cannot be interpolated. The calculation is possible neither in RFEM nor in RF-STEEL SANS.

The cross-section's stress points including numbering can also be checked graphically: Select the cross-section in window 1.3 and click [Info]. The dialog box shown in Figure 2.10 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.

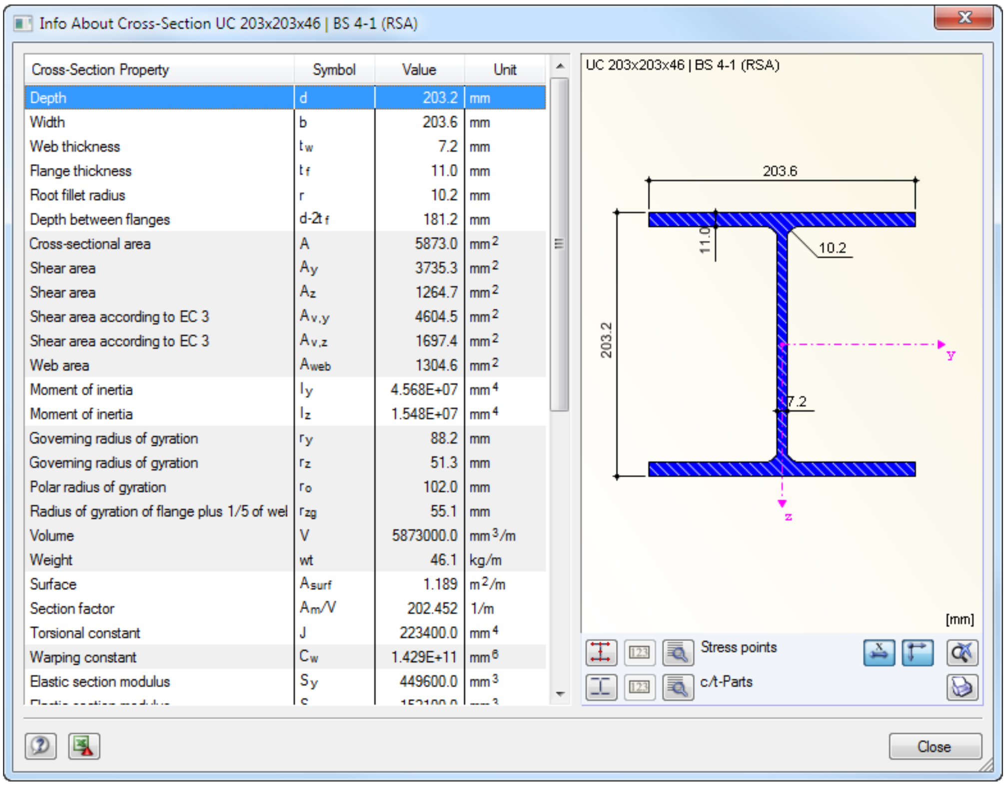


Figure 2.10: Dialog box *Info about Cross-Section*

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

The buttons below the graphic have the following functions:

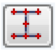
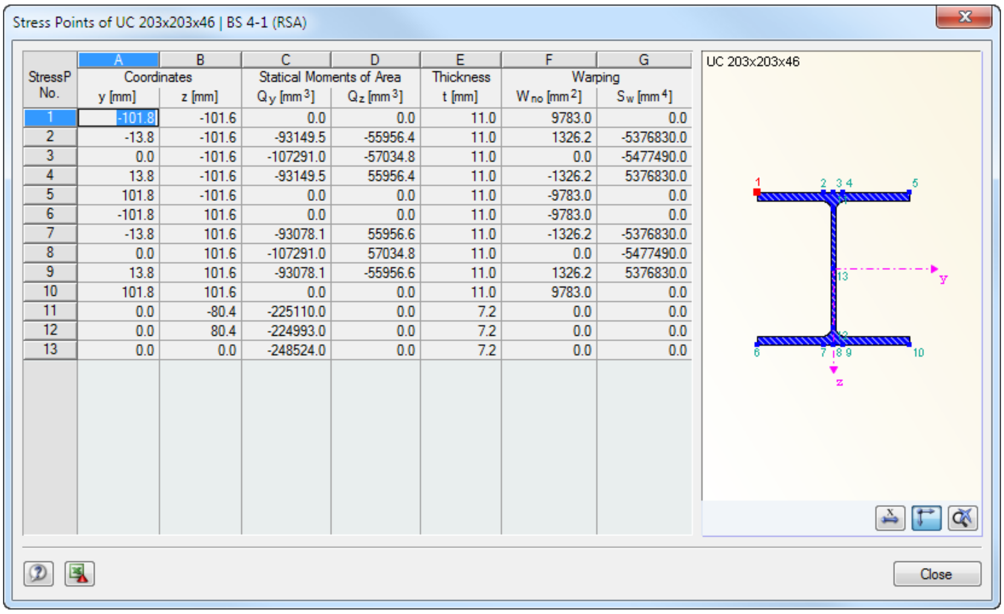
Button	Function
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the numbering of stress points or c/t-parts
	Displays or hides the details of the stress points or c/t-parts (see Figure 2.11)
	Displays or hides the dimensions of the cross-section
	Displays or hides 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 [Details] to call up detailed information on stress points (distance to center of gravity, statical moments of area, normalized warping constants etc.) and c/t-parts.



Stress Points of UC 203x203x46 | BS 4-1 (RSA)

	A	B	C	D	E	F	G
StressP No.	Coordinates		Statical Moments of Area		Thickness	Warping	
	y [mm]	z [mm]	Q_y [mm³]	Q_z [mm³]	t [mm]	W_{no} [mm²]	S_w [mm⁴]
1	-101.8	-101.6	0.0	0.0	11.0	9783.0	0.0
2	-13.8	-101.6	-93149.5	-55956.4	11.0	1326.2	-5376830.0
3	0.0	-101.6	-107291.0	-57034.8	11.0	0.0	-5477490.0
4	13.8	-101.6	-93149.5	55956.4	11.0	-1326.2	5376830.0
5	101.8	-101.6	0.0	0.0	11.0	-9783.0	0.0
6	-101.8	101.6	0.0	0.0	11.0	-9783.0	0.0
7	-13.8	101.6	-93078.1	55956.6	11.0	-1326.2	-5376830.0
8	0.0	101.6	-107291.0	57034.8	11.0	0.0	-5477490.0
9	13.8	101.6	-93078.1	-55956.6	11.0	1326.2	5376830.0
10	101.8	101.6	0.0	0.0	11.0	9783.0	0.0
11	0.0	-80.4	-225110.0	0.0	7.2	0.0	0.0
12	0.0	80.4	-224993.0	0.0	7.2	0.0	0.0
13	0.0	0.0	-248524.0	0.0	7.2	0.0	0.0

Figure 2.11: Dialog box *Stress Points of UC 203x203x46*

2.4 Intermediate Lateral Restraints

In window 1.4, you can define intermediate lateral restraints for members. RF-STEEL SANS always assumes this kind of support to be perpendicular to the cross-section's minor axis z (see Figure 2.8). Thus, it is possible to influence the members' effective lengths which are important for the design of column buckling and lateral torsional buckling.

For the calculation, all intermediate lateral restraints are considered as torsional supports.

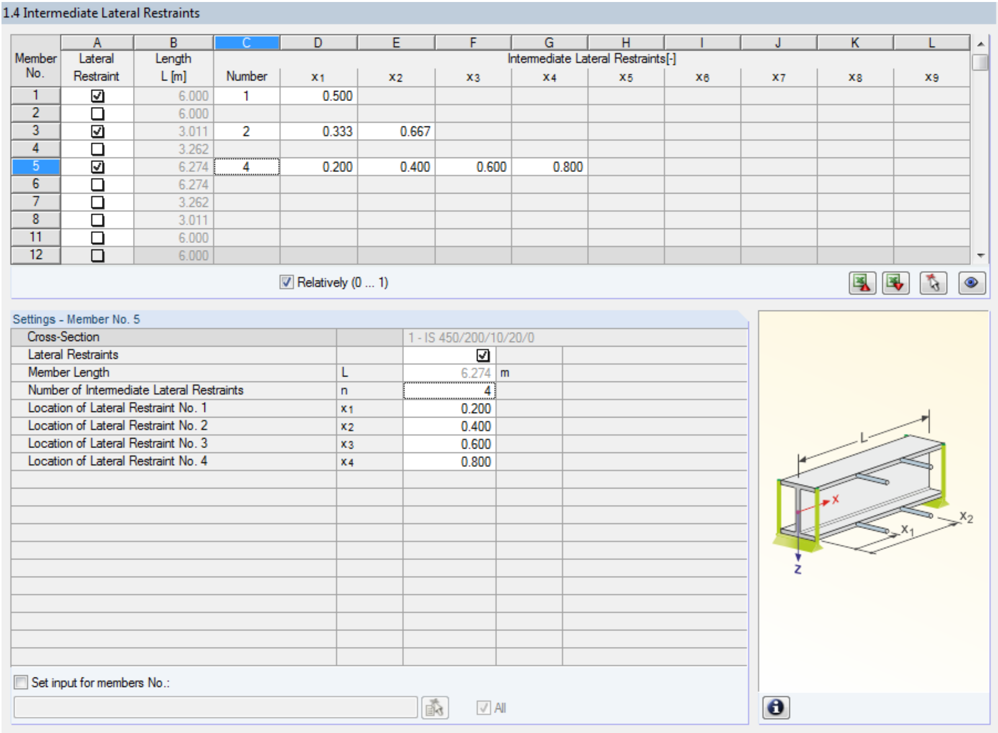


Figure 2.12: Window 1.4 *Intermediate Lateral Restraints*

In the upper part of the window, you can assign up to nine lateral supports for each member. The *Settings* section arranges the input in a column for an overview of the member selected above.

To define the intermediate restraints of a member, select the *Lateral Restraint* check box in column A. To graphically select the member and to activate its row, click [↖]. By selecting the check box, the other columns become available for you to enter the parameters.

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

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

In case of cantilevers, avoid intermediate restraints because such supports divide the member into segments. For cantilevered beams, this would results in segments with torsional restraints on one end each that are statically underdetermined.

2.5 Effective Lengths - Members

Window 1.5 consists of two parts. The table in the upper part provides summarized information on the effective length factors K_y and K_z, the effective lengths K_yL and K_zL, the lengths L_w and L_T for torsional or torsional-flexural buckling and the member types of the beams to be designed. The effective lengths defined in RFEM are preset. In the *Settings* section, further information is shown about the member whose row is selected in the upper section.

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

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

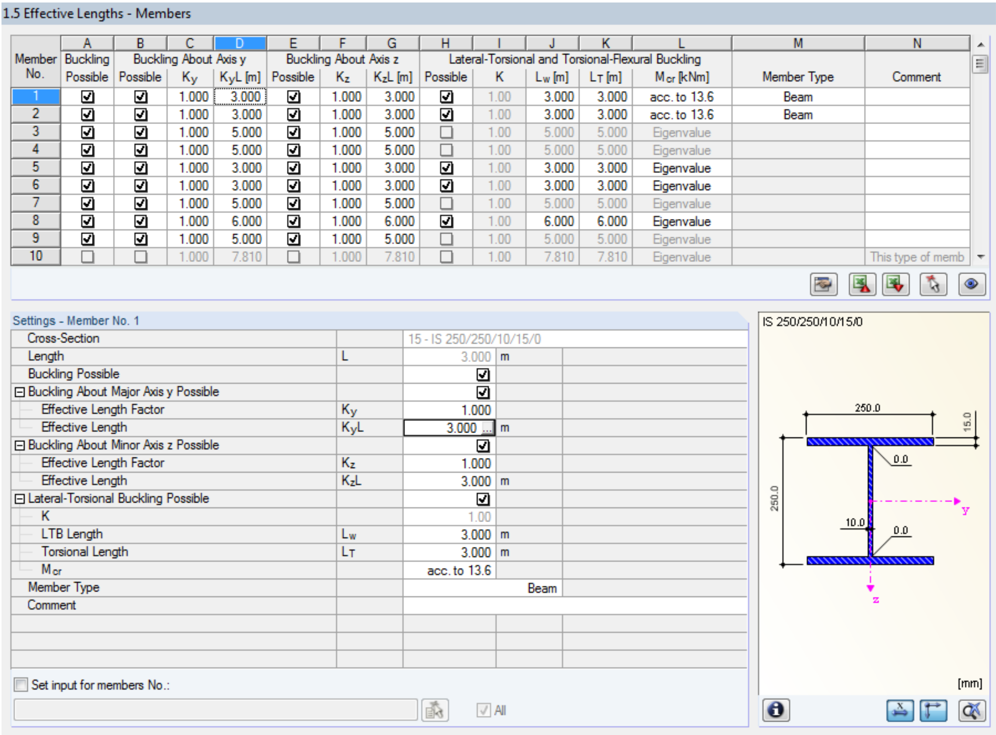


Figure 2.13: Window 1.5 *Effective Lengths - Members*

The effective lengths for the column buckling about the minor axis z and the effective lengths for lateral-torsional buckling are aligned automatically with the entries of window 1.4 *Intermediate Lateral Restraints*. If intermediate restraints divide the member into member segments of different lengths, the program displays no values in the table columns G and J of window 1.5.

The effective lengths can be entered manually in the table and in the *Settings* tree, or defined graphically in the work window after clicking [...]. This button is enabled when you click in the effective length box.

The *Settings* tree manages the following parameters:

- *Cross-section*
- *Length* (actual length of the member)
- *Buckling Possible* (cf column A)
- *Buckling About Major Axis y Possible* (buckling lengths, cf columns B - D)
- *Buckling About Minor Axis z Possible* (buckling lengths, cf columns E - G)
- *Lateral-Torsional Buckling Possible* (LTB and torsional lengths, cf columns H - L)
- *Member Type* (cf column M)

In this table, you can specify for the currently selected member whether to carry out a buckling or a lateral-torsional buckling analysis. In addition to this, you can adjust the *Effective Length Factor* for the respective lengths. When a coefficient is modified, the equivalent member length is adjusted automatically, and vice versa.

You can also define the buckling length of a member in a dialog box. To open it, click the button shown on the left. It is located on the right below the upper table of the window.

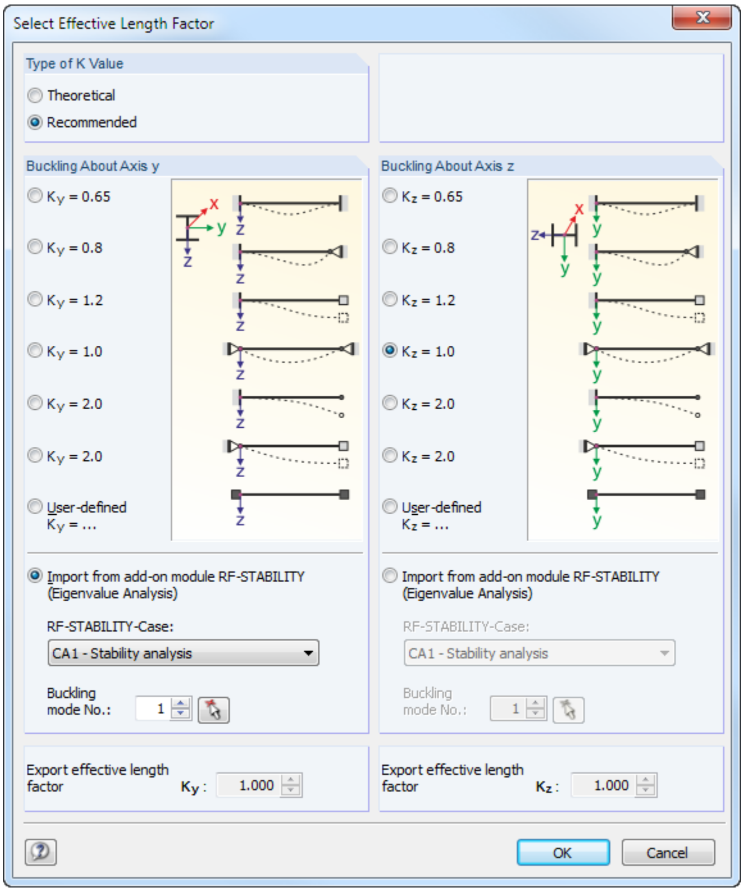


Figure 2.14: Dialog box *Select Effective Length Coefficient*

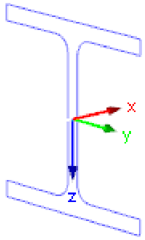
In this dialog box, the values of the coefficient *K* can be defined that are to be assigned to the selected member(s). The *Theoretical* and *Recommended* values are described in [1], Annex E. Generally, it is possible to select predefined coefficients or to enter *User-defined* values.

If a RF-STABILITY case calculated according to the eigenvalue analysis is already available, you can also select a *Buckling mode* to determine the factor.

Buckling Possible

A stability analysis for buckling and lateral buckling requires the ability of members to absorb compressive forces. Therefore, members for which such absorption is not possible because of the member type (for example tension members, elastic foundations, rigid connections) are excluded from design in the first place. The corresponding rows appear dimmed and a note is indicated in the *Comment* column.

The *Buckling Possible* check boxes in table row A and in the *Settings* tree enable you to classify specific members as compression members or, alternatively, to exclude them from the design according to [1].



Axis system

Buckling About Axis y or Axis z

With the check boxes in the *Possible* table columns, you decide whether a member is susceptible to buckling about the y-axis and/or z-axis. These axes represent the local member axes, with axis y being the "major" and axis z the "minor" member axis. The effective length coefficients K_y and K_z for buckling about the major or the minor axes can be selected freely.

You can check the position of the member axes in the cross-section graphic in window 1.3 *Cross-Sections* (see Figure 2.8, page 14). To access the RFEM work window, click [View mode]. In the work window, you can display the local member axes by using the member's context menu or the *Display* navigator.

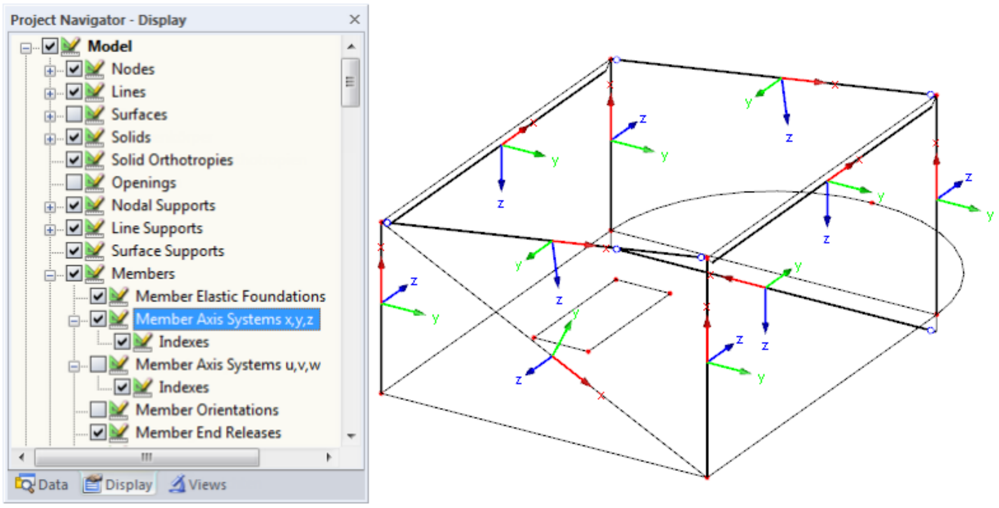


Figure 2.15: Displaying the member axes in the *Display* navigator of RFEM

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

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

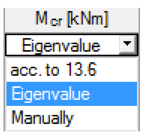
When you define the effective length coefficient *K*, the program determines the effective length *KL* by multiplying the member length *L* by this buckling length coefficient.

Lateral-Torsional and Torsional-Flexural Buckling

Table column H controls whether a lateral-torsional and torsional-flexural buckling design is to be carried out.

The effective lengths L_W and L_T depend on the settings of window 1.4 *Intermediate Lateral Restraints*. It is also possible to enter user-defined values into the columns J and K. The factor *K* (column I) is calculated from the lateral-torsional buckling length L_w of each member or member segment.

The effective length factors are to be determined according to [1], Table 1 or Table 2.



In column L, three options are available for the calculation of the critical elastic moment M_{cr}.

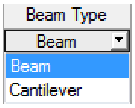
M_{cr} can be calculated according to the equation as specified in [1], clause 13.6 a) ii):

$$M_{cr} = \frac{\omega_2 \pi}{KL} \sqrt{E \cdot I_z \cdot G \cdot J + \left(\frac{\pi \cdot E}{KL}\right)^2 \cdot I_z \cdot C_w}$$

For that, it is necessary to define the *Member Type* in column M.

By default, M_{cr} is determined by the *Eigenvalue* solver which takes into account the specific boundary conditions by using a finite model. The value of M_{cr} can also be entered *Manually*.

Member Type



Column M provides two options to allocate the *Member Type* according to [1], clause 13.6.

Comment

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

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

2.6 Effective Lengths - Sets of Members

Input window 1.6 controls the effective lengths for sets of members. It is only available if one or more sets of members have been selected in window 1.1 *General Data.*

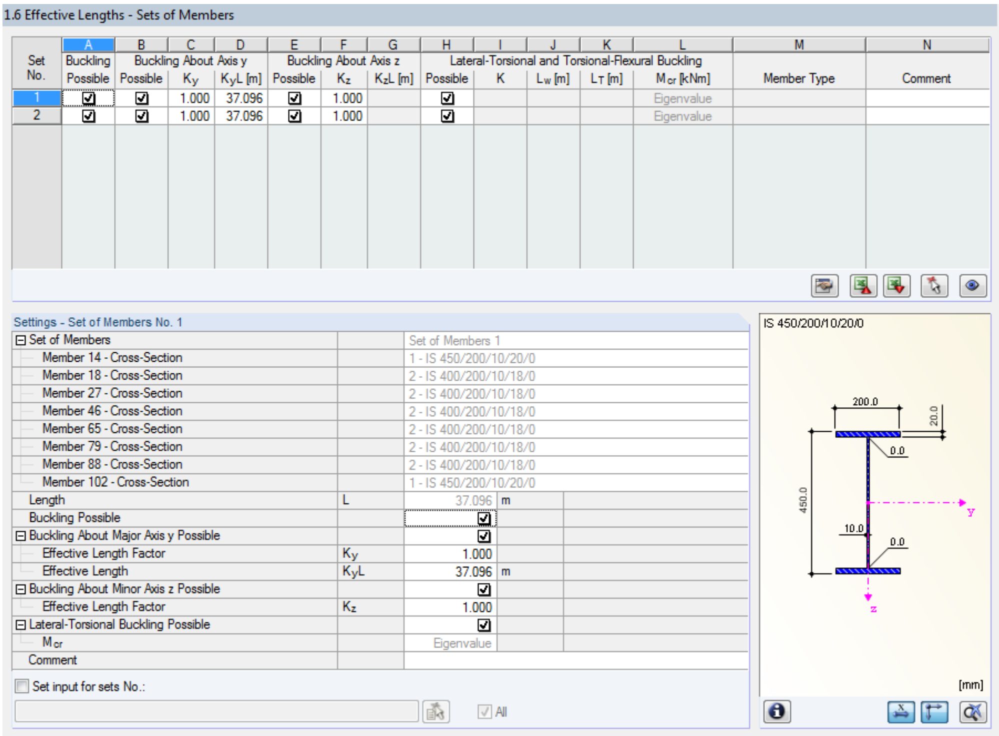


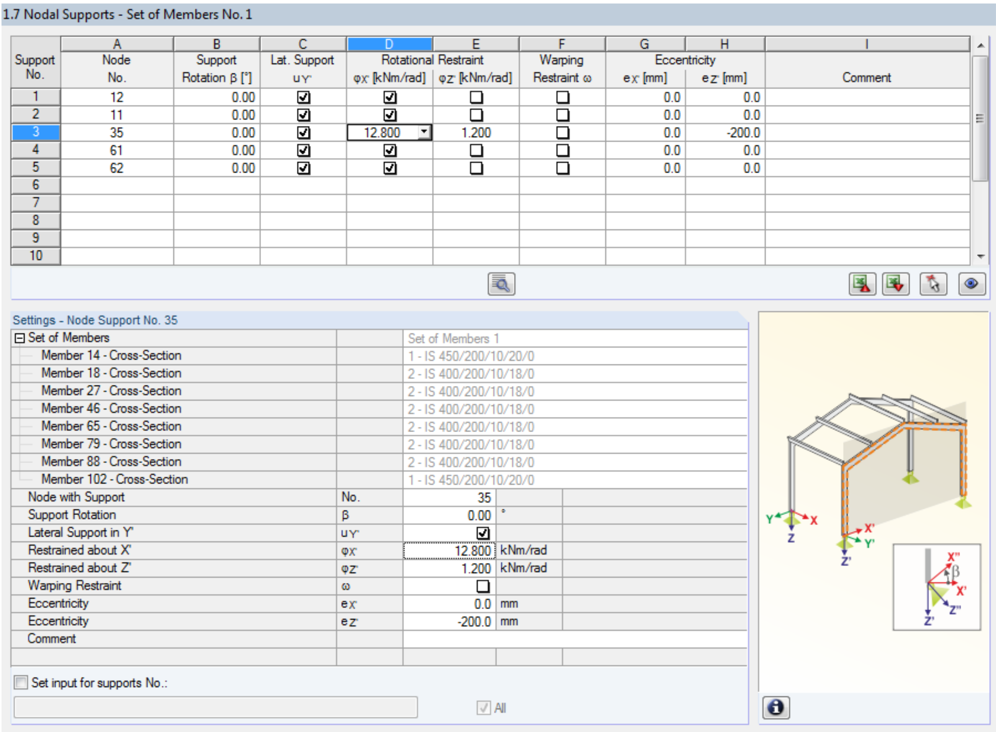
Figure 2.16: Window 1.6 *Effective Lengths - Set of Members*

The concept of this window is similar to the one in the previous 1.5 *Effective Lengths - Members* window. In this window, you can enter the effective lengths of the set of members for buckling and lateral-torsional buckling as described in chapter 2.5.

2.7 Nodal Supports - Sets of Members

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

In RF-STEEL SANS, the stability analysis for sets of members is performed with specific parameters. If, however, the *Member-Like Input* is set in the *Details* dialog box (see Figure 3.2, page 29), window 1.7 will not be displayed. In that case, the intermediate lateral restraints can be defined in window 1.4 by division points.



Support No.	A Node No.	B Support Rotation β [°]	C Lat. Support uY'	D Rotational Restraint φX' [kNm/rad]	E Rotational Restraint φZ' [kNm/rad]	F Warping Restraint ω	G Eccentricity eX' [mm]	H Eccentricity eZ' [mm]	I Comment
1	12	0.00	☑	☑	☐	☐	0.0	0.0	
2	11	0.00	☑	☑	☐	☐	0.0	0.0	
3	35	0.00	☑	12.800	1.200	☐	0.0	-200.0	
4	61	0.00	☑	☑	☐	☐	0.0	0.0	
5	62	0.00	☑	☑	☐	☐	0.0	0.0	
6									
7									
8									
9									
10									

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

To determine the critical factor of lateral-torsional buckling, a planar framework is created with four degrees of freedom for each node. The parameters are to be defined in window 1.7. The settings in the table refer to the current set of members which is selected in the add-on module's navigator on the left.

The orientation of the axes in the set of members is important for the definition of nodal supports. The program checks the position of the nodes and internally defines, according to Figure 2.18 through Figure 2.21, the axes of the nodal supports for window 1.7.

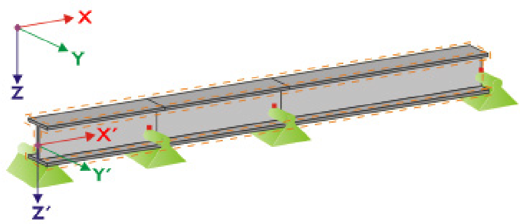


Figure 2.18: Auxiliary coordinate system for nodal supports – straight set of members

If all members of a set of members lie in a straight line as shown in Figure 2.18, 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.

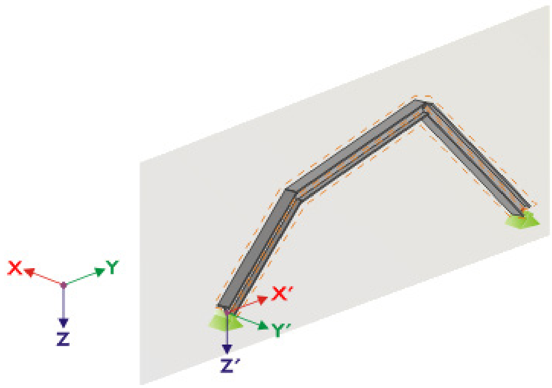


Figure 2.19: Auxiliary coordinate system for nodal supports – set of members in vertical plane

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 2.19, they are lying in a vertical plane. In this case, the X'-axis is horizontal and oriented in direction of the plane. The Y'-axis is horizontal as well and defined perpendicular to the X'-axis. The Z'-axis is oriented perpendicularly downwards.

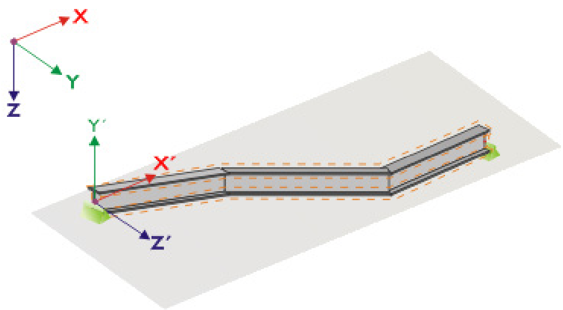


Figure 2.20: Auxiliary coordinate system for nodal supports – set of members in horizontal plane

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

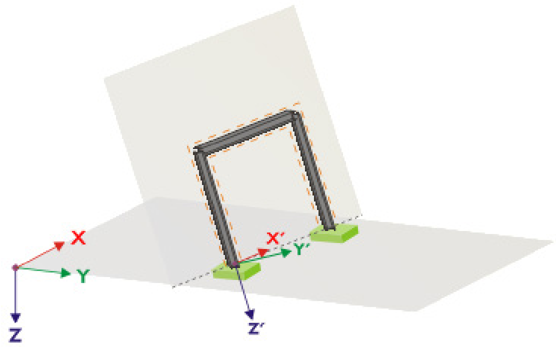


Figure 2.21: Auxiliary coordinate system for nodal supports – set of members in inclined plane

Figure 2.21 shows the general case of a buckled set of members: The members are not lying in one straight line but in an inclined plane. The definition of the X'-axis arises out of the intersection line of the inclined plane with the horizontal plane. Thus, the Y'-axis is defined perpendicular to the X'-axis and directed perpendicular to the inclined plane. The Z'-axis is defined perpendicular to the X'- and Y'-axes.

2.8 Member End Releases - Sets of Members

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

Window 1.8 is not displayed when the *Member-Like Input* is set in the dialog box *Details* (see Figure 3.2, page 29) for sets of members.

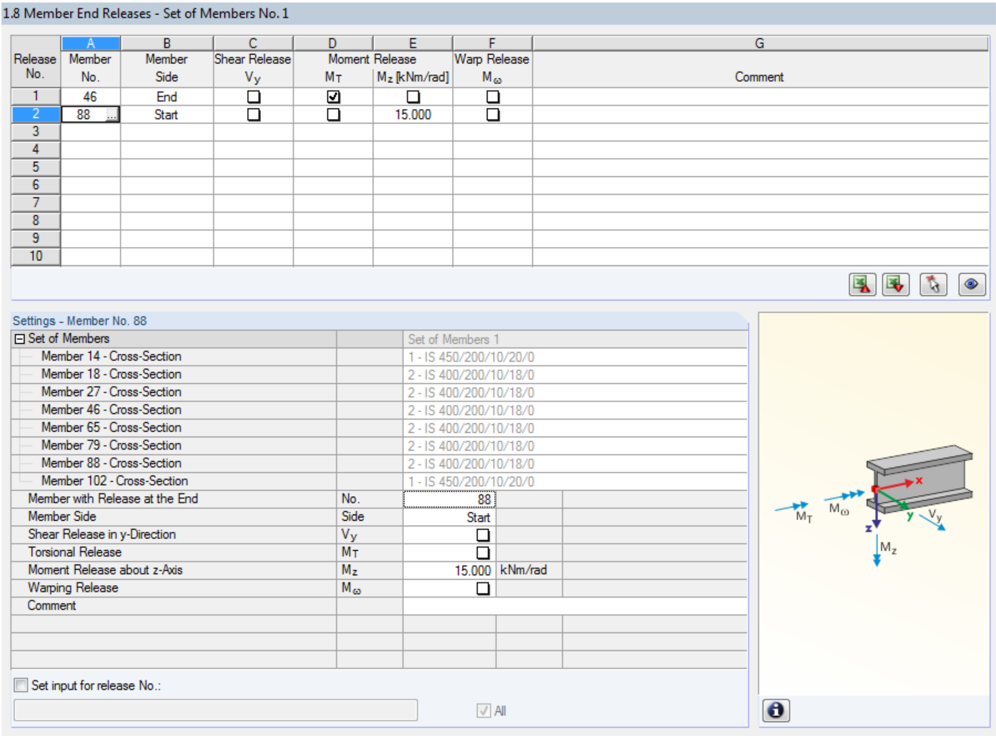
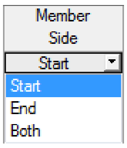


Figure 2.22: Window 1.8 *Member End Releases - Set of Members*

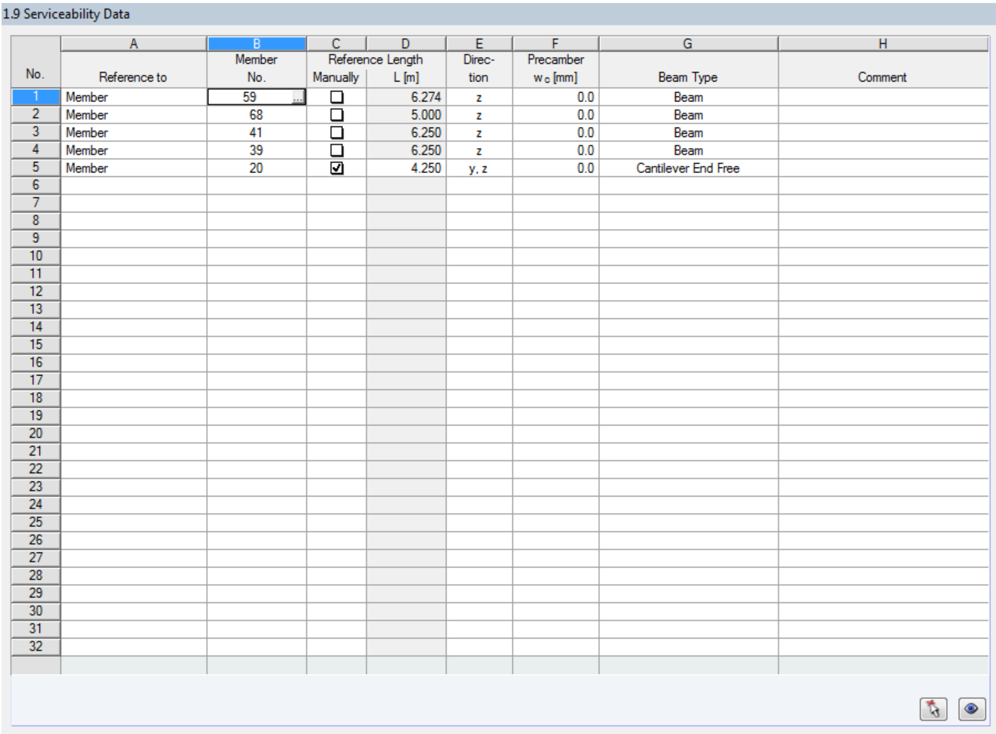


In table column B, you define the *Member Side* to which the release should be assigned. You can also connect the releases to both member sides.

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

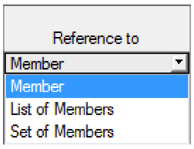
2.9 Serviceability Data

This input window controls several settings for the serviceability limit state design. It is only available if you have set the according entries in the *Serviceability Limit State* tab of window 1.1 *General Data* (see Figure 2.4, page 11).



No.	A Reference to	B Member No.	C Manually	D Reference Length L [m]	E Direc-tion	F Precamber w_c [mm]	G Beam Type	H Comment
1	Member	59	☐	6.274	z	0.0	Beam	
2	Member	68	☐	5.000	z	0.0	Beam	
3	Member	41	☐	6.250	z	0.0	Beam	
4	Member	39	☐	6.250	z	0.0	Beam	
5	Member	20	☑	4.250	y, z	0.0	Cantilever End Free	
6								
7								
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

Figure 2.23: 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 these values after selecting the *Manually* check box in column C.

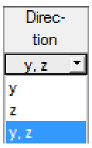
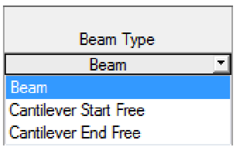


Table column E defines the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z (or u and v for unsymmetrical cross-sections).

A *precamber* w_c can be taken into account by using entries specified in column F.



The *Beam Type* is of vital importance for the correct application of limit deformations. In table column G, you can select the girder to be a beam or a cantilever and decide which end should have no support.

The settings of the *Details* dialog box determine whether the deformations are related to the undeformed initial model or to the shifted ends of members or sets of members (see Figure 3.3, page 31).

2.10 Parameters - Members

The last input window controls additional design parameters for members.

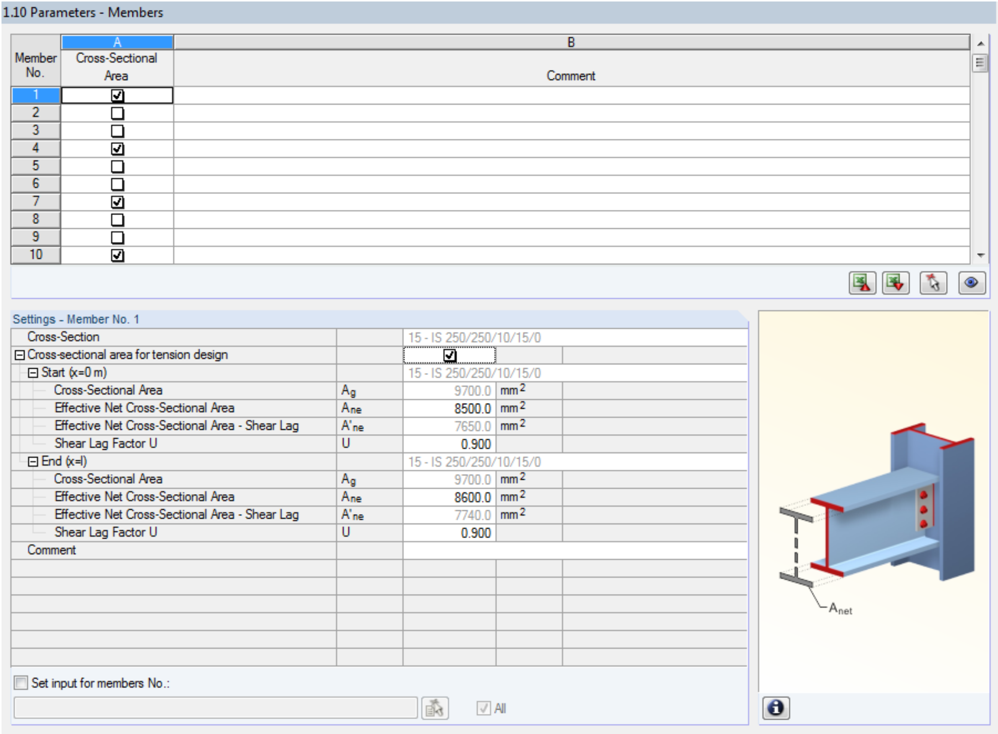


Figure 2.24: Window 1.10 *Parameters - Members*

Cross-Sectional Area

This column provides an option to reduce the cross-sectional area of specific members due to connections. In the *Settings* section below, the effective net area according to [1], clause 12.3.1 and the shear lag factor according to [1], clause 12.3.3 can be defined. Those parameters can be set for the start and end nodes of each member.

The effective cross-sectional areas are taken into account for the cross-section design of members in tension.

3. Calculation

3.1 Details

Calculation

Details...

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

The *Details* dialog box contains the following tabs:

- Ultimate Limit State
- Stability
- Serviceability
- General

3.1.1 Ultimate Limit State

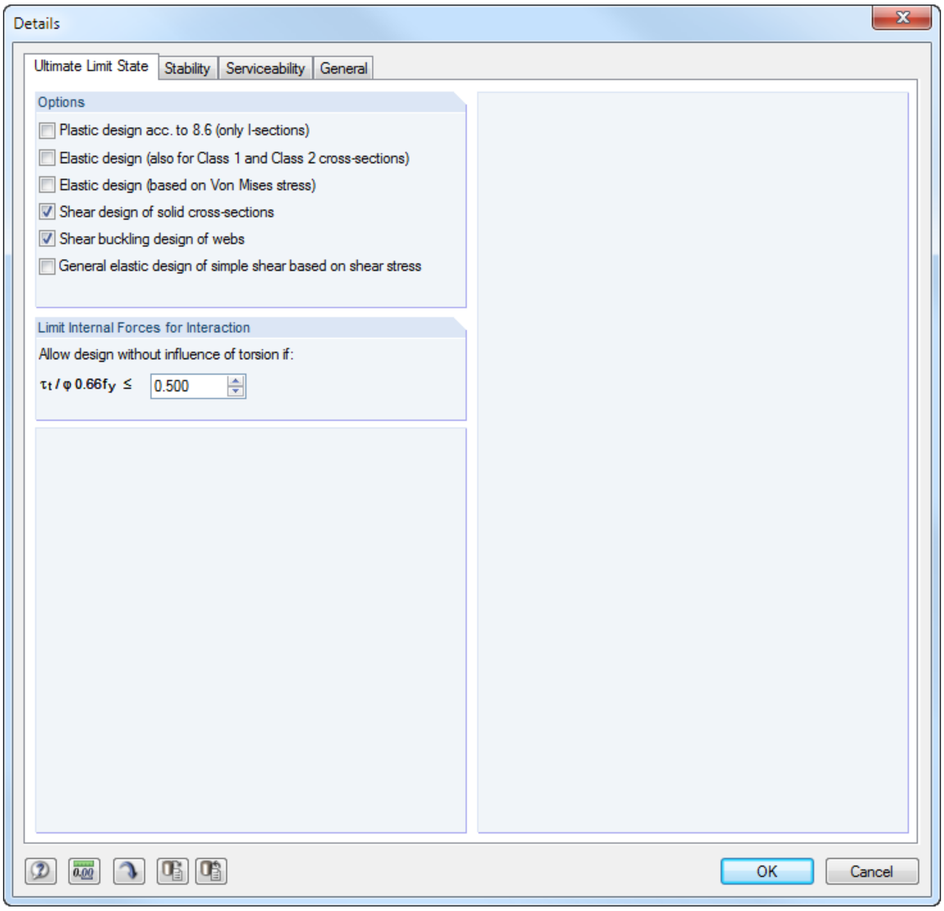


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

Options

The *Plastic design acc. to 8.6* of hot-formed doubly symmetric I-sections (class 1) is possible, provided that the requirement given in [1], clause 8.6 a) is satisfied.

Cross-sections that are assigned to class 1 or 2 are designed plastically in RF-STEEL SANS. If you do not want to perform a plastic design, you can activate the *Elastic Design* for these cross-section classes, too.

The conservative general *Elastic design* is based on the analysis of stresses in stress points inclusive of VON MISES stresses. This method can be useful for cross-sections of complicated shapes or members with torsion.

If the *Shear design of solid cross-sections* (flat or round bars) or the *Shear buckling design of webs* is not required, it can be disabled.

The conservative *General elastic design based on simple shear stress* in the stress points can be activated to determine the stresses of x-locations where shear without bending occurs.

Limit Internal Forces for Interaction

The South African standard provides no exact recommendation how to design cross-sections with torsional moments. Therefore, it possible to ignore shear stress due to torsion if a user-defined ratio of torsional shear stress τ_t and shear strength (product of resistance factor φ and ultimate shear stress) is not exceeded.

3.1.2 Stability

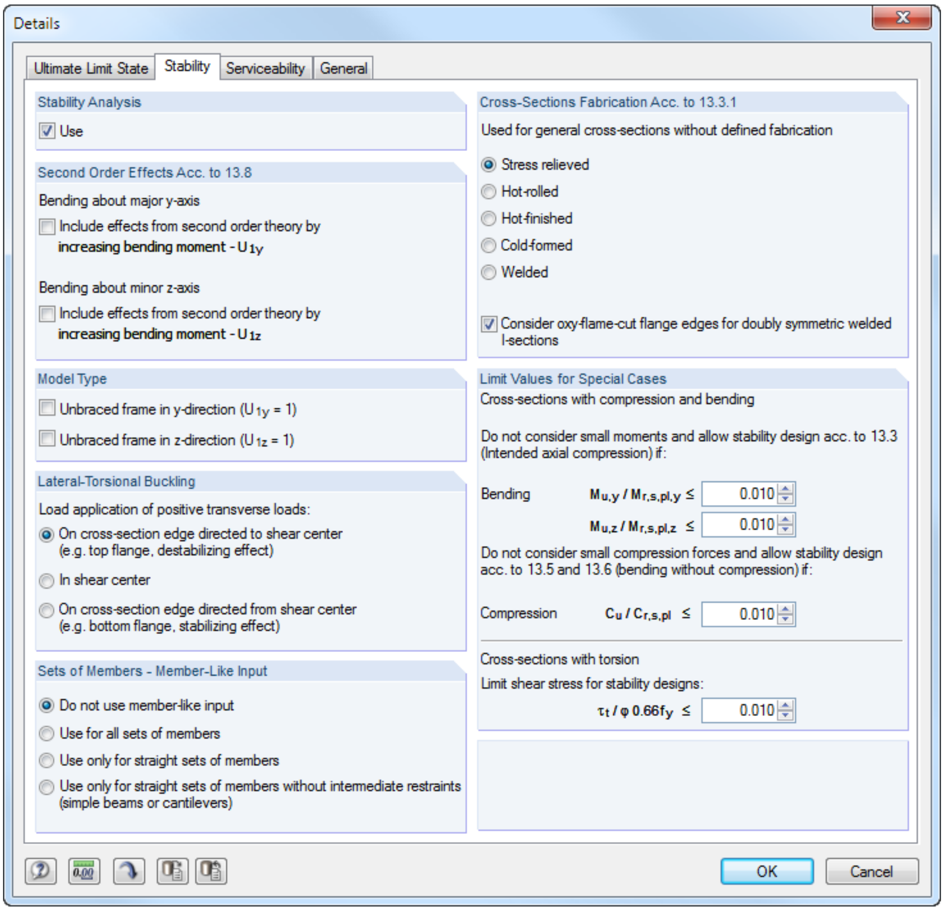


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

The *Use* check box controls whether to run, in addition to the cross-section checks, a stability analysis. If you clear the check box, the input windows 1.4, 1.5 and 1.6 will not be displayed.

Second Order Effects Acc. to 13.8

It is possible to *Include effects from second order theory* according to [1], clause 13.8 by increasing the bending moments about the major and/or minor axes. Thus when you design, for example, a frame whose governing buckling mode is represented by lateral displacement, you can determine the internal forces according to linear static analysis and increase them with the factors according to [1], clause 13.8.2 and 13.8.4. If you increase the bending moment, it does not affect the flexural-buckling analysis which is performed by using the axial forces.

Model Type

According to [1], clause 13.8.2, the design of axial compression and bending depends on the fact whether the members are part of unbraced or braced frames. Please note that the settings of this check box are applied to all members of the design case.

Lateral-Torsional Buckling

If transverse loads are present, it is important to define where those forces are acting on the cross-section: Depending on the *Load application* point, transverse loads can be stabilizing or destabilizing, and, thus, can influence the critical elastic moment decisively.

Sets of Members - Member-Like Input

By default, the boundary conditions of every set of members are to be defined in windows 1.6, 1.7 and 1.8. Alternatively, the "member-like input" can be applied: all sets of members are managed in the same manner as beams. Then, the input data of window 1.6 is internally transformed to the boundary conditions required for the stability design. Windows 1.7 and 1.8 are not existent for the member-like input. As a rule, the member-like input is recommended for straight sets of members only.

If the default *Do not use member-like input* is set, a general analysis is carried out. The support conditions have to be defined for each set of members in table 1.7.

The option *Use for all sets of members* makes it possible to design sets of members in the same way as single members. This approach is applicable when every set of members corresponds to a single member model. Then the default values for simple girders are used to determine the support conditions β, u_y, φ_x, φ_z and ω.

It is possible to apply the member-like input *only for straight sets of members* which have the same cross-section (e.g. continuous beams).

The last option to use the member-like input *only for straight sets of members without intermediate restraints*, according to the definition in RFEM: Only sets of members which have RFEM supports or restraints at their ends will be considered. This option can be used to design e.g. simple beams or cantilevers. Tables 1.7 and 1.8 are not displayed. Transverse beams that are connected at the intermediate nodes cannot be accounted for.

Cross-Sections Fabrication Acc. to 13.3.1

The factor *n* is relevant for the flexural buckling design according to [1], clause 13.3.1. This factor depends on the fabrication of the sections (state of residual stresses). Rolled or oxy-flame-cut plates can be used to fabricate welded I and H sections.

Please note that the type of fabrication which is set in this dialog section is applied to all cross-sections of the design case.

Limit Values for Special Cases

To design non-symmetrical cross-sections for intended axial compression according to [1], clause 13.3, you can neglect *small moments* about the major and the minor axes using the settings defined in this dialog section.

In the same way, you can switch off *small compression forces* for the pure design of bending by defining a limit ratio for C_u to $C_{r,s,pl}$.

Intended *torsion* is not clearly specified in SANS 10162-1. If the torsional stresses do not exceed the shear stress ratio of 1 % (default), they are not considered in the stability design. The design is then carried for flexural buckling and lateral-torsional 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. However, the cross-section checks are run independently. These limit settings are not part of the code [1]. Changing the limits is in the responsibility of the user.

3.1.3 Serviceability

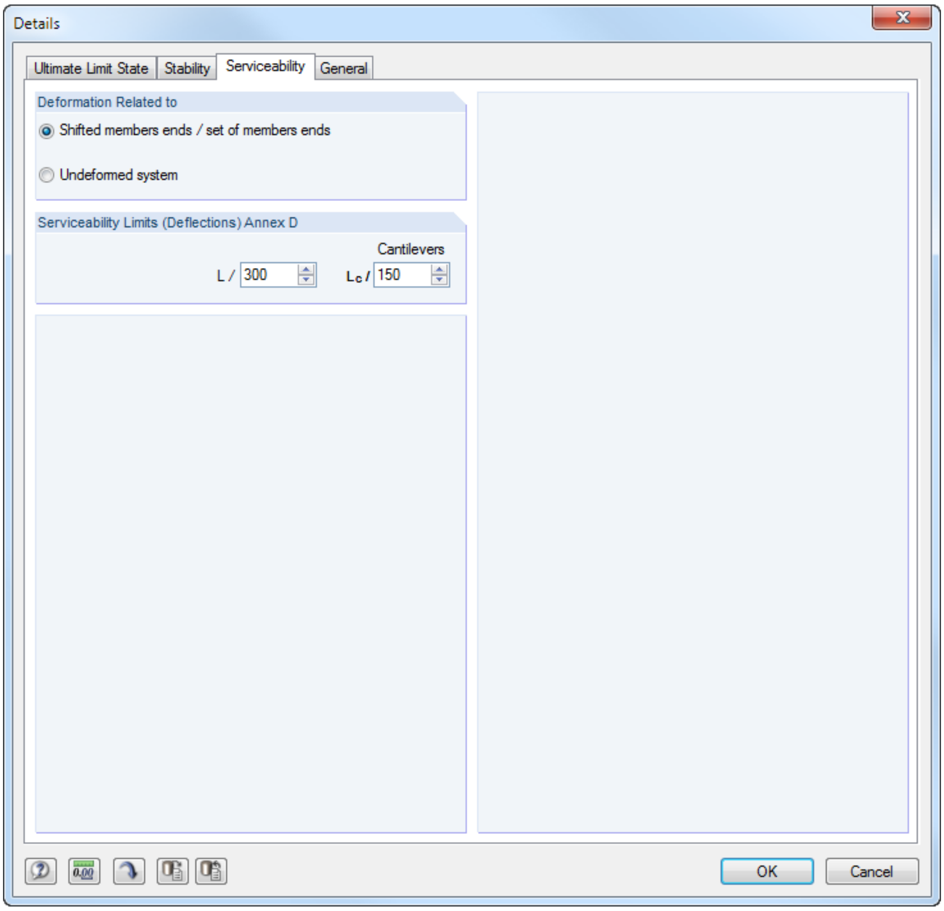


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Related to

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

Serviceability Limits (Deflections) Annex D

In this dialog section, you can check and, if necessary, adjust the limit deformations of simple beams and cantilevers. The recommended maximum values for deflections at serviceability are contained in [1], Table D.1.

3.1.4 General

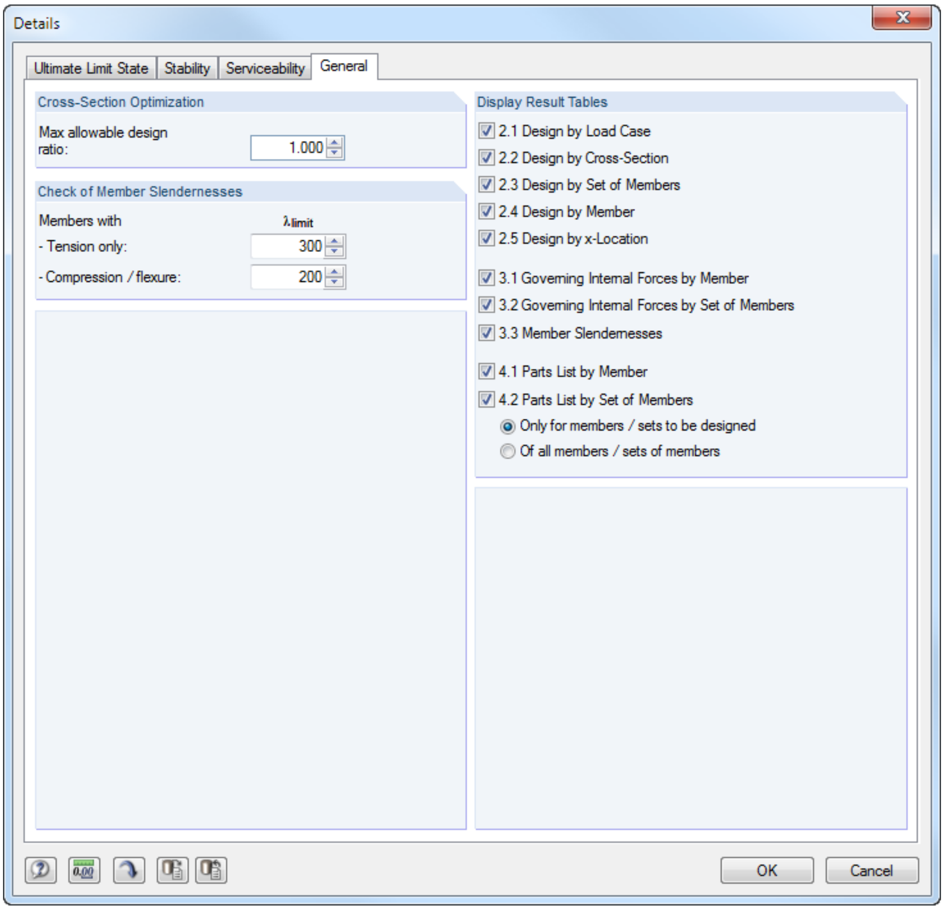


Figure 3.4: Dialog box *Details*, tab *General*

Cross-Section Optimization

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

Check of Member Slendernesses

In the two text boxes, you can specify the limit values λ_{limit} in order to define member slendernesses. You can enter specifications separately for members with pure tension forces and members with bending and compression.

The limit values are compared to the real member slendernesses in window 3.3. This window is available after the calculation (see chapter 4.8, page 41) if the corresponding check box is selected in the *Display Result Tables* dialog box section.

Display Result Tables

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

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

3.2 Start Calculation

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

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

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

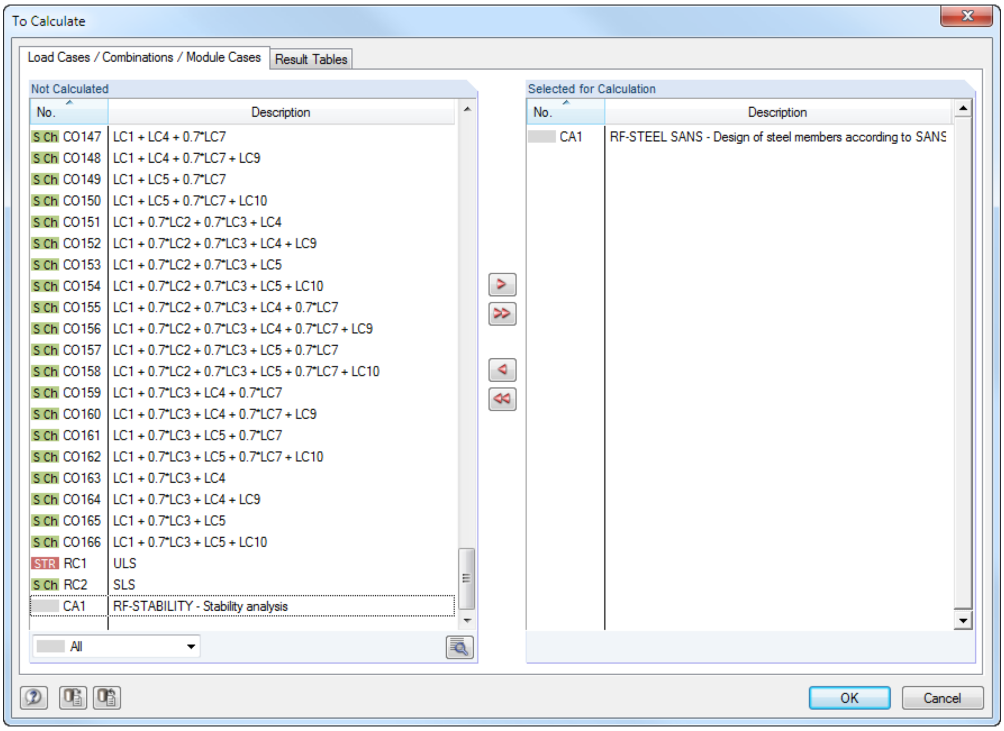
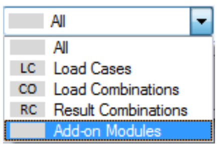


Figure 3.5: *To Calculate* dialog box



If the RF-STEEL SANS design cases are missing in the *Not Calculated* list, select *All* or *Add-on Modules* in the drop-down list at the end of the list.

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

To calculate a design case directly, use the list in the toolbar. Select the RF-STEEL SANS design case in the toolbar list, and then click [Show Results].

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

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

4. Results

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

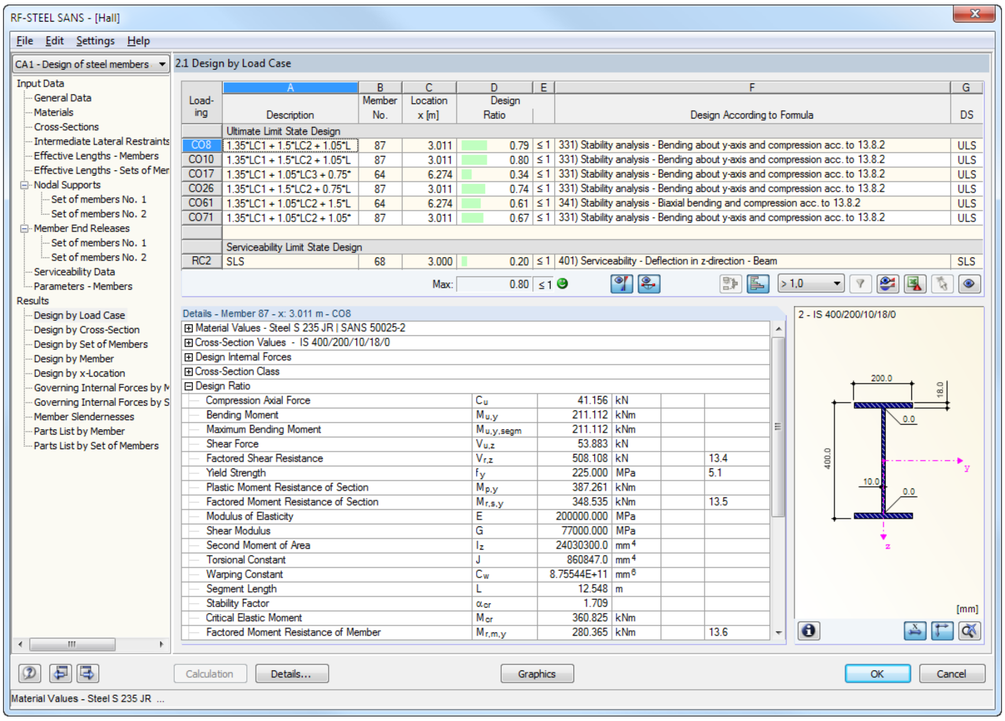
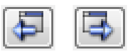


Figure 4.1: Results window with designs and intermediate values

The designs are shown in the results windows 2.1 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. The last two results windows 4.1 and 4.2 show the parts lists sorted by member and set of members.



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

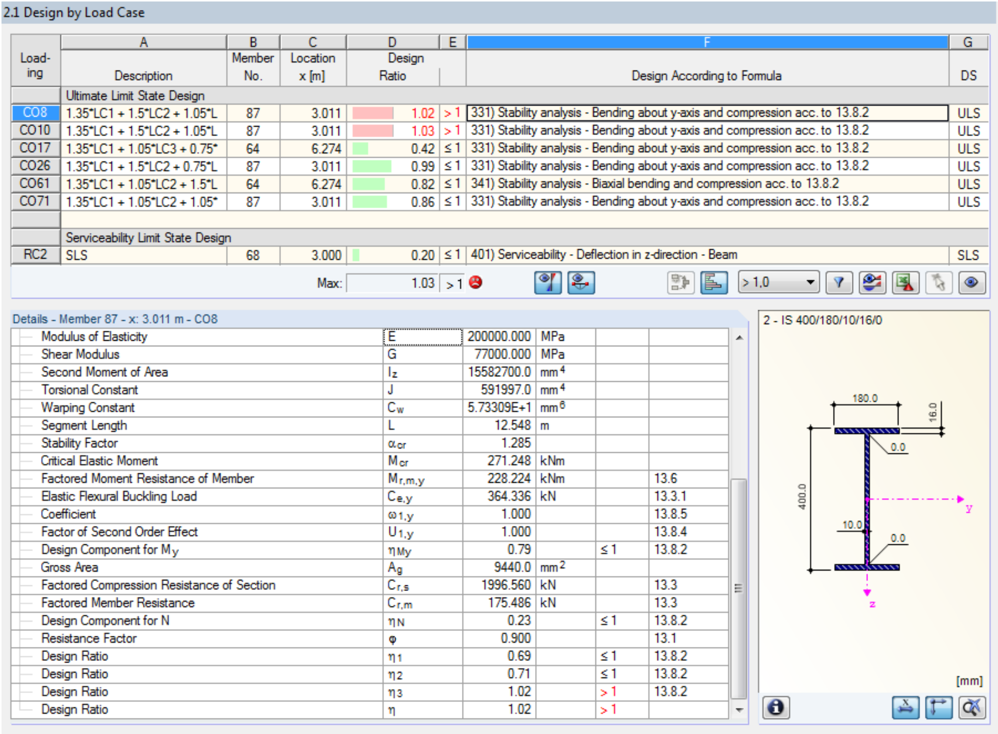
Click [OK] to save the results. Thus you exit RF-STEEL SANS and return to the main program.

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

4.1 Design by Load Case

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

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



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						
CO8	1.35*LC1 + 1.5*LC2 + 1.05*L	87	3.011	1.02	> 1	331) Stability analysis - Bending about y-axis and compression acc. to 13.8.2	ULS
CO10	1.35*LC1 + 1.5*LC2 + 1.05*L	87	3.011	1.03	> 1	331) Stability analysis - Bending about y-axis and compression acc. to 13.8.2	ULS
CO17	1.35*LC1 + 1.05*LC3 + 0.75*	64	6.274	0.42	≤ 1	331) Stability analysis - Bending about y-axis and compression acc. to 13.8.2	ULS
CO26	1.35*LC1 + 1.5*LC2 + 0.75*L	87	3.011	0.99	≤ 1	331) Stability analysis - Bending about y-axis and compression acc. to 13.8.2	ULS
CO61	1.35*LC1 + 1.05*LC2 + 1.5*L	64	6.274	0.82	≤ 1	341) Stability analysis - Biaxial bending and compression acc. to 13.8.2	ULS
CO71	1.35*LC1 + 1.05*LC2 + 1.05*	87	3.011	0.86	≤ 1	331) Stability analysis - Bending about y-axis and compression acc. to 13.8.2	ULS
	Serviceability Limit State Design						
RC2	SLS	68	3.000	0.20	≤ 1	401) Serviceability - Deflection in z-direction - Beam	SLS
			Max:	1.03	> 1		

Details - Member 87 - x: 3.011 m - CO8

Modulus of Elasticity	E	200000.000	MPa		
Shear Modulus	G	77000.000	MPa		
Second Moment of Area	I_z	15582700.0	mm^4		
Torsional Constant	J	591997.0	mm^4		
Warping Constant	C_w	5.73309E+1	mm^6		
Segment Length	L	12.548	m		
Stability Factor	α_{cr}	1.285			
Critical Elastic Moment	M_{cr}	271.248	kNm		
Factored Moment Resistance of Member	$M_{r,m,y}$	228.224	kNm		13.6
Elastic Flexural Buckling Load	$C_{e,y}$	364.336	kN		13.3.1
Coefficient	$\omega_{1,y}$	1.000			13.8.5
Factor of Second Order Effect	$U_{1,y}$	1.000			13.8.4
Design Component for M_y	η_{My}	0.79		≤ 1	13.8.2
Gross Area	A_g	9440.0	mm^2		
Factored Compression Resistance of Section	$C_{r,s}$	1996.560	kN		13.3
Factored Member Resistance	$C_{r,m}$	175.486	kN		13.3
Design Component for N	η_N	0.23		≤ 1	13.8.2
Resistance Factor	φ	0.900			13.1
Design Ratio	η_1	0.69		≤ 1	13.8.2
Design Ratio	η_2	0.71		≤ 1	13.8.2
Design Ratio	η_3	1.02		> 1	13.8.2
Design Ratio	η	1.02		> 1	

Figure 4.2: Window 2.1 *Design by Load Case*

Description

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

Member No.

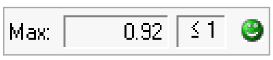
This column shows the number of the member that bears the maximum stress ratio of the designed loading.

Location x

This column shows the respective x-location where the member's maximum stress ratio occurs. For the table output, the program uses the following member locations *x*:

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

Design Ratio



Columns D and E display the design conditions according to SANS 10165-1 [1].

The lengths of the colored bars represent the respective utilizations.

Design according to Formula

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

DS

The final column provides information on the respective design-relevant design situation *(DS)*: *ULS* (Ultimate Limit State) or *SLS* (Serviceability Limit State).

4.2 Design by Cross-Section

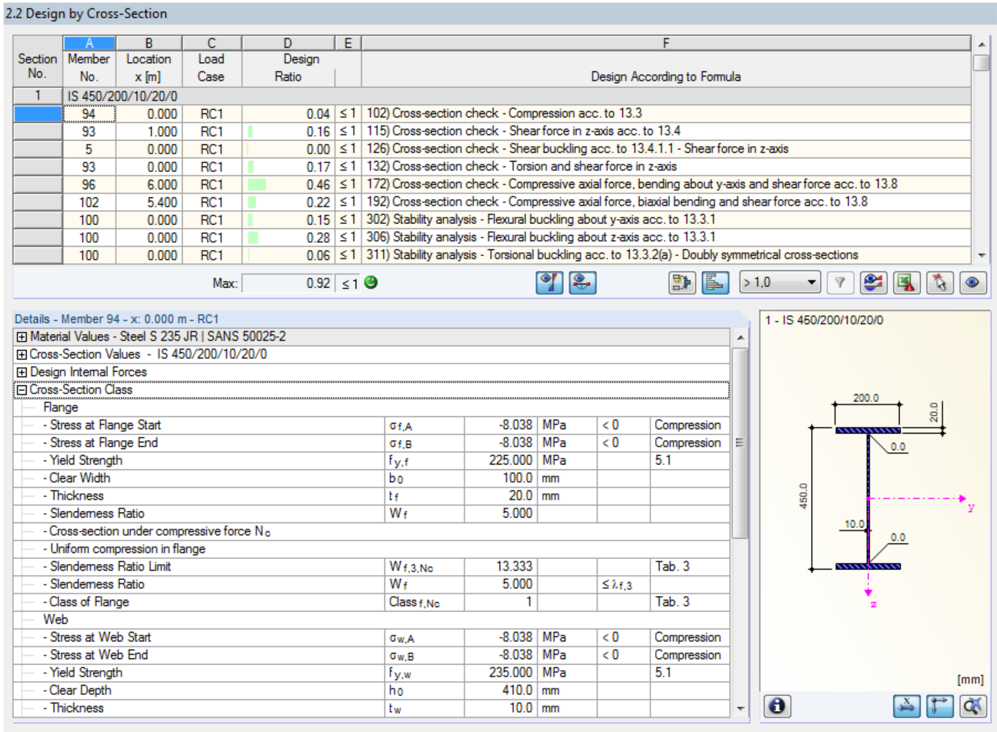


Figure 4.3: Window 2.2 *Design by Cross-Section*

This window lists the maximum ratios of all members and actions selected for design, sorted by cross-section. The results are sorted by cross-section design and serviceability limit state design.

If there is a tapered member, the results of both cross-section numbers are listed.

4.3 Design by Set of Members

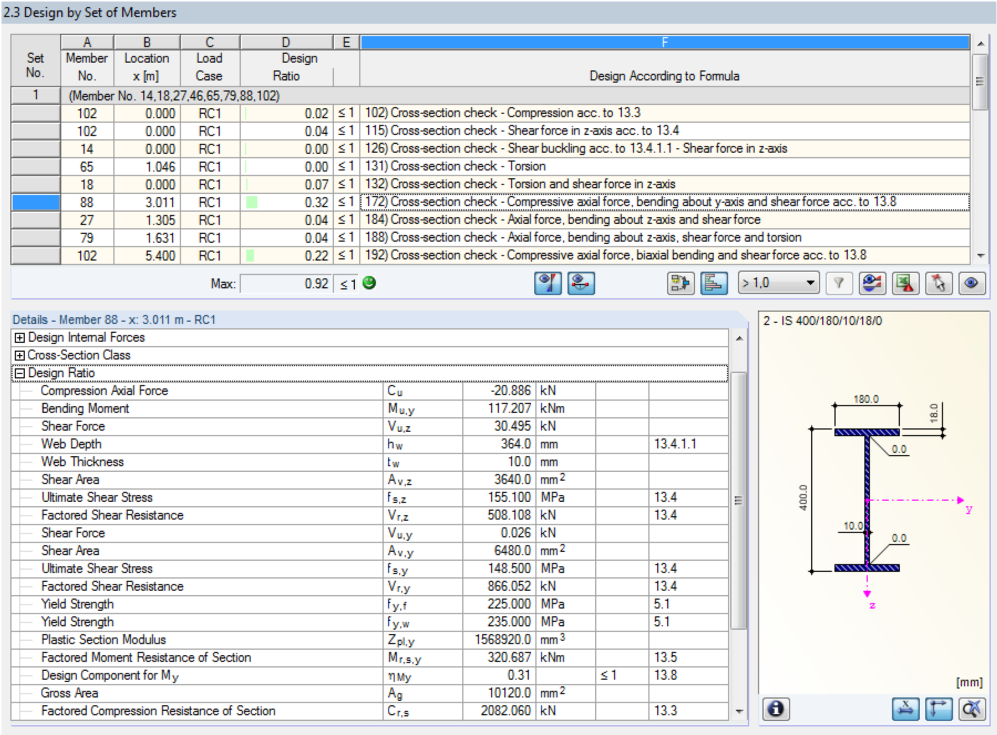


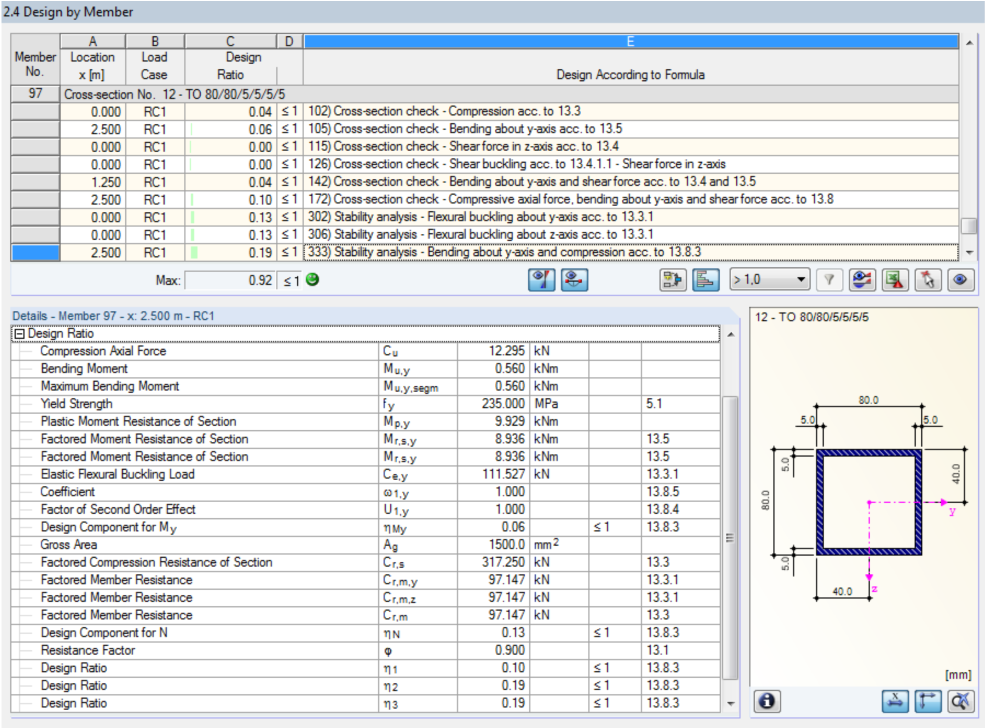
Figure 4.4: Window 2.3 *Design by Set of Members*

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

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

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

4.4 Design by Member



2.4 Design by Member

Member No.	A Location x [m]	B Load Case	C Design Ratio	D	E Design According to Formula
97	Cross-section No. 12 - TO 80/80/5/5/5/5				
	0.000	RC1	0.04	≤ 1	102) Cross-section check - Compression acc. to 13.3
	2.500	RC1	0.06	≤ 1	105) Cross-section check - Bending about y-axis acc. to 13.5
	0.000	RC1	0.00	≤ 1	115) Cross-section check - Shear force in z-axis acc. to 13.4
	0.000	RC1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shear force in z-axis
	1.250	RC1	0.04	≤ 1	142) Cross-section check - Bending about y-axis and shear force acc. to 13.4 and 13.5
	2.500	RC1	0.10	≤ 1	172) Cross-section check - Compressive axial force, bending about y-axis and shear force acc. to 13.8
	0.000	RC1	0.13	≤ 1	302) Stability analysis - Flexural buckling about y-axis acc. to 13.3.1
	0.000	RC1	0.13	≤ 1	306) Stability analysis - Flexural buckling about z-axis acc. to 13.3.1
	2.500	RC1	0.19	≤ 1	333) Stability analysis - Bending about y-axis and compression acc. to 13.8.3
	Max:		0.92	≤ 1	

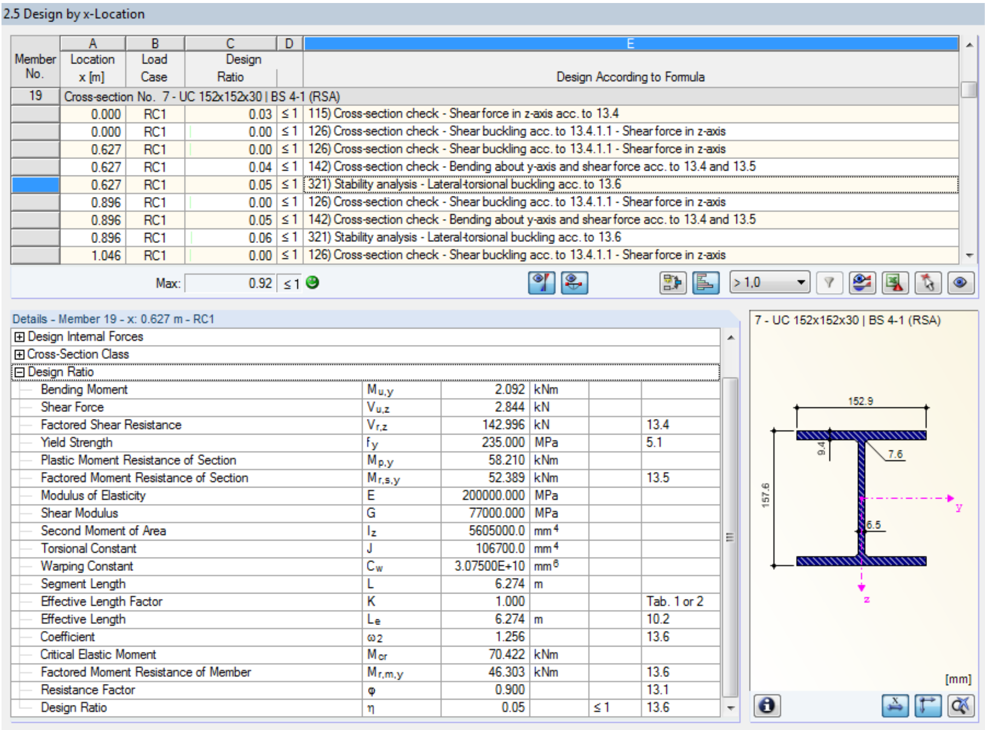
Details - Member 97 - x: 2.500 m - RC1

Design Ratio					
Compression Axial Force	C_u	12.295	kN		
Bending Moment	$M_{u,y}$	0.560	kNm		
Maximum Bending Moment	$M_{u,y,segm}$	0.560	kNm		
Yield Strength	f_y	235.000	MPa		5.1
Plastic Moment Resistance of Section	$M_{p,y}$	9.929	kNm		
Factored Moment Resistance of Section	$M_{r,s,y}$	8.936	kNm		13.5
Factored Moment Resistance of Section	$M_{r,s,y}$	8.936	kNm		13.5
Elastic Flexural Buckling Load	$C_{e,y}$	111.527	kN		13.3.1
Coefficient	$\omega_{1,y}$	1.000			13.8.5
Factor of Second Order Effect	$U_{1,y}$	1.000			13.8.4
Design Component for M_y	η_{My}	0.06		≤ 1	13.8.3
Gross Area	A_g	1500.0	mm^2		
Factored Compression Resistance of Section	$C_{r,s}$	317.250	kN		13.3
Factored Member Resistance	$C_{r,m,y}$	97.147	kN		13.3.1
Factored Member Resistance	$C_{r,m,z}$	97.147	kN		13.3.1
Factored Member Resistance	$C_{r,m}$	97.147	kN		13.3
Design Component for N	η_N	0.13		≤ 1	13.8.3
Resistance Factor	φ	0.900			13.1
Design Ratio	η_1	0.10		≤ 1	13.8.3
Design Ratio	η_2	0.19		≤ 1	13.8.3
Design Ratio	η_3	0.19		≤ 1	13.8.3

Figure 4.5: Window 2.4 *Design by Member*

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

4.5 Design by x-Location



2.5 Design by x-Location

Member No.	A Location x [m]	B Load Case	C Design Ratio	D	E Design According to Formula
19	Cross-section No. 7 - UC 152x152x30 | BS 4-1 (RSA)				
	0.000	RC1	0.03	≤ 1	115) Cross-section check - Shear force in z-axis acc. to 13.4
	0.000	RC1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shear force in z-axis
	0.627	RC1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shear force in z-axis
	0.627	RC1	0.04	≤ 1	142) Cross-section check - Bending about y-axis and shear force acc. to 13.4 and 13.5
	0.627	RC1	0.05	≤ 1	321) Stability analysis - Lateral-torsional buckling acc. to 13.6
	0.896	RC1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shear force in z-axis
	0.896	RC1	0.05	≤ 1	142) Cross-section check - Bending about y-axis and shear force acc. to 13.4 and 13.5
	0.896	RC1	0.06	≤ 1	321) Stability analysis - Lateral-torsional buckling acc. to 13.6
	1.046	RC1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shear force in z-axis
	Max:		0.92	≤ 1	

Details - Member 19 - x: 0.627 m - RC1

⊞ Design Internal Forces
⊞ Cross-Section Class

Design Ratio					
Bending Moment	$M_{u,y}$	2.092	kNm		
Shear Force	$V_{u,z}$	2.844	kN		
Factored Shear Resistance	$V_{r,z}$	142.996	kN		13.4
Yield Strength	f_y	235.000	MPa		5.1
Plastic Moment Resistance of Section	$M_{p,y}$	58.210	kNm		
Factored Moment Resistance of Section	$M_{r,s,y}$	52.389	kNm		13.5
Modulus of Elasticity	E	200000.000	MPa		
Shear Modulus	G	77000.000	MPa		
Second Moment of Area	I_z	5605000.0	mm^4		
Torsional Constant	J	106700.0	mm^4		
Warping Constant	C_w	3.07500E+10	mm^6		
Segment Length	L	6.274	m		
Effective Length Factor	K	1.000			Tab. 1 or 2
Effective Length	L_e	6.274	m		10.2
Coefficient	ω_2	1.256			13.6
Critical Elastic Moment	M_{cr}	70.422	kNm		
Factored Moment Resistance of Member	$M_{r,m,y}$	46.303	kNm		13.6
Resistance Factor	φ	0.900			13.1
Design Ratio	η	0.05		≤ 1	13.6

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

This results window lists the maxima for each member at the 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]	Load-ing	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
1	Cross-section No. 15 - IS 250/250/10/15/0								
	2.500	RC1	-3.953	0.214	0.168	0.000	-0.011	0.126	100) Negligible internal forces
	0.000	RC1	-13.642	-0.401	-0.051	0.000	0.152	0.158	102) Cross-section check - Compression acc. to 13.3
	2.500	RC1	-3.987	-0.821	-0.049	0.000	0.025	-0.486	106) Cross-section check - Bending about z-axis acc. to 13.5
	0.000	RC1	-13.608	-0.196	1.675	0.001	-2.313	0.090	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Sł
	0.000	RC1	-13.608	-0.164	1.675	0.001	-2.313	0.189	172) Cross-section check - Compressive axial force, bending at
	1.000	RC1	-9.781	0.082	-0.049	0.000	0.099	-1.041	184) Cross-section check - Axial force, bending about z-axis an
	0.000	RC1	-13.621	0.687	1.150	-0.002	-1.551	-0.657	192) Cross-section check - Compressive axial force, biaxial ben
	0.000	RC1	-15.836	-0.140	-0.115	0.000	0.344	0.262	302) Stability analysis - Flexural buckling about y-axis acc. to 1:
	0.000	RC1	-15.836	-0.140	-0.115	0.000	0.344	0.262	306) Stability analysis - Flexural buckling about z-axis acc. to 1:
	0.000	RC1	-15.836	-0.140	-0.115	0.000	0.344	0.262	311) Stability analysis - Torsional buckling acc. to 13.3.2(a) - Dc
	0.000	RC1	-13.608	-0.164	1.675	0.001	-2.313	0.189	331) Stability analysis - Bending about y-axis and compression a
	0.000	RC1	-13.643	0.686	-0.049	0.000	0.148	-0.659	336) Stability analysis - Bending about z-axis and compression a
	0.000	RC1	-13.621	0.687	1.150	-0.002	-1.551	-0.657	341) Stability analysis - Biaxial bending and compression acc. tc
2	Cross-section No. 15 - IS 250/250/10/15/0								
	1.000	RC1	-46.248	-0.036	0.058	0.000	0.226	-0.058	102) Cross-section check - Compression acc. to 13.3
	0.000	RC1	-33.822	-0.683	3.760	0.000	-10.935	-1.299	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Sł
	2.500	RC1	-35.542	-0.587	2.327	0.001	-3.287	-0.035	172) Cross-section check - Compressive axial force, bending at
	0.000	RC1	-45.233	2.786	0.060	0.000	0.080	5.077	184) Cross-section check - Axial force, bending about z-axis an
	0.000	RC1	-45.201	-0.951	3.800	0.000	-10.956	-1.964	192) Cross-section check - Compressive axial force, biaxial ben
	1.000	RC1	-46.248	-0.036	0.058	0.000	0.226	-0.058	302) Stability analysis - Flexural buckling about y-axis acc. to 1:
	1.000	RC1	-46.248	-0.036	0.058	0.000	0.226	-0.058	306) Stability analysis - Flexural buckling about z-axis acc. to 1:
	1.000	RC1	-46.248	-0.036	0.058	0.000	0.226	-0.058	311) Stability analysis - Torsional buckling acc. to 13.3.2(a) - Dc
	2.500	RC1	-35.542	-0.587	2.327	0.001	-3.287	-0.035	331) Stability analysis - Bending about y-axis and compression a
	0.000	RC1	-45.233	2.786	0.060	0.000	0.080	5.077	336) Stability analysis - Bending about z-axis and compression a
	0.500	RC1	-43.278	2.494	2.463	-0.001	-6.323	3.747	341) Stability analysis - Biaxial bending and compression acc. tc
3	Cross-section No. 12 - TO 80/80/5/5/5/5								
	0.000	RC1	-1.178	0.000	0.401	0.000	0.000	0.000	102) Cross-section check - Compression acc. to 13.3
	2.500	RC1	-0.616	0.000	0.000	0.000	0.500	0.000	105) Cross-section check - Bending about y-axis acc. to 13.5
	0.000	RC1	-1.178	0.000	0.401	0.000	0.000	0.000	115) Cross-section check - Shear force in z-axis acc. to 13.4
	0.000	RC1	0.618	0.000	0.396	0.000	0.000	0.000	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Sł
	1.250	RC1	-0.615	0.000	0.200	0.000	0.375	0.000	142) Cross-section check - Bending about y-axis and shear forc

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

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

Location x

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

Load Case

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

Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing maximum design ratios in the respective ultimate limit state and serviceability limit state designs.

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

Set No.	A 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	(Member No. 14,18,27,46,65,79,88,102)								
	0.000	RC1	-57.207	0.468	20.895	0.004	0.000	0.000	102) Cross-section check - Compression acc. to 13.3
	0.000	RC1	-57.207	0.468	20.895	0.004	0.000	0.000	115) Cross-section check - Shear force in z-axis acc. to 13.4
	0.000	RC1	-42.500	0.463	-6.671	-0.004	0.000	0.000	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shea
	1.046	RC1	-16.050	-0.005	-0.951	0.023	55.086	0.232	131) Cross-section check - Torsion
	0.000	RC1	-20.634	-0.080	29.806	0.053	-108.370	-0.004	132) Cross-section check - Torsion and shear force in z-axis
	3.011	RC1	-20.886	-0.026	-30.495	0.027	-117.207	0.006	172) Cross-section check - Compressive axial force, bending about
	1.305	RC1	-19.653	-0.082	17.401	0.001	-0.319	0.351	184) Cross-section check - Axial force, bending about z-axis and sh
	1.631	RC1	-17.237	0.032	-15.394	0.012	-0.325	0.129	188) Cross-section check - Axial force, bending about z-axis, shear
	5.400	RC1	-35.045	-0.368	18.441	0.010	106.174	-0.253	192) Cross-section check - Compressive axial force, biaxial bending
	0.000	RC1	-57.207	0.468	20.895	0.004	0.000	0.000	302) Stability analysis - Flexural buckling about y-axis acc. to 13.3.
	0.000	RC1	-57.207	0.468	20.895	0.004	0.000	0.000	306) Stability analysis - Flexural buckling about z-axis acc. to 13.3.
	0.000	RC1	-57.207	0.468	20.895	0.004	0.000	0.000	311) Stability analysis - Torsional buckling acc. to 13.3.2(a) - Doubl
	3.011	RC1	-20.886	-0.026	-30.495	0.027	-117.207	0.006	331) Stability analysis - Bending about y-axis and compression acc.
	1.087	RC1	-19.645	-0.027	-16.810	-0.002	3.612	-0.128	341) Stability analysis - Biaxial bending and compression acc. to 13
2	(Member No. 12,17,26,45,64,78,87,100)								
	0.000	RC1	-84.639	0.002	39.046	0.003	0.000	0.000	102) Cross-section check - Compression acc. to 13.3
	6.274	RC1	-0.457	-0.025	0.952	0.019	51.371	-0.221	107) Cross-section check - Biaxial bending acc. to 13.5 and 13.6e)
	0.000	RC1	-84.639	0.002	39.046	0.003	0.000	0.000	115) Cross-section check - Shear force in z-axis acc. to 13.4
	0.000	RC1	-34.015	-0.001	-2.730	0.003	0.000	0.000	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shea
	1.046	RC1	-37.613	-0.017	-0.160	0.026	89.796	0.270	131) Cross-section check - Torsion
	3.011	RC1	-38.348	-0.042	-55.005	0.053	-217.686	0.064	132) Cross-section check - Torsion and shear force in z-axis
	0.000	RC1	-0.891	0.021	-2.304	-0.019	51.151	-0.220	162) Cross-section check - Biaxial bending and shear force acc. to
	3.011	RC1	-38.348	-0.042	-55.005	0.053	-217.686	0.064	172) Cross-section check - Compressive axial force, bending about
	1.631	RC1	-29.938	-0.070	-27.843	-0.005	0.527	-0.209	184) Cross-section check - Axial force, bending about z-axis and sh
	5.019	RC1	-32.749	0.081	3.206	0.015	115.918	0.138	192) Cross-section check - Compressive axial force, biaxial bending
	0.000	RC1	-84.639	0.002	39.046	0.003	0.000	0.000	302) Stability analysis - Flexural buckling about y-axis acc. to 13.3.
	0.000	RC1	-84.639	0.002	39.046	0.003	0.000	0.000	306) Stability analysis - Flexural buckling about z-axis acc. to 13.3.
	0.000	RC1	-84.639	0.002	39.046	0.003	0.000	0.000	311) Stability analysis - Torsional buckling acc. to 13.3.2(a) - Doubl
	6.274	RC1	-0.457	-0.025	0.952	0.019	51.371	-0.221	323) Stability analysis - Biaxial bending acc. to 13.6(e)
	3.011	RC1	-41.798	0.029	-53.808	-0.060	-210.692	-0.051	331) Stability analysis - Bending about y-axis and compression acc.
	6.274	RC1	-40.650	0.019	-25.340	0.016	41.848	0.132	341) Stability analysis - Biaxial bending and compression acc. to 13

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

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

4.8 Member Slendernesses

3.3 Member Slendernesses

Member No.	A 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	3.000	1.000	107.7	27.849	1.000	63.5	47.263	
2	Compression / Flexure	3.000	1.000	107.7	27.849	1.000	63.5	47.263	
3	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
4	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
5	Compression / Flexure	3.000	1.000	188.0	15.960	1.000	47.0	63.863	
6	Compression / Flexure	3.000	1.000	188.0	15.960	1.000	47.0	63.863	
7	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
8	Compression / Flexure	6.000	1.000	188.0	31.921	1.000	47.0	127.727	
9	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
12	Compression / Flexure	6.000	1.000	188.0	31.921	1.000	47.0	127.727	
13	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
15	Compression / Flexure	3.011	1.000	165.4	18.209	1.000	41.6	72.361	
16	Compression / Flexure	3.011	1.000	165.4	18.209	1.000	41.6	72.361	
17	Compression / Flexure	3.011	1.000	165.4	18.209	1.000	41.6	72.361	
19	Compression / Flexure	6.274	1.000	67.6	92.818	1.000	38.3	163.914	
20	Compression / Flexure	6.250	1.000	188.0	33.251	1.000	47.0	133.049	
21	Compression / Flexure	6.250	1.000	188.0	33.251	1.000	47.0	133.049	
24	Compression / Flexure	3.262	1.000	165.4	19.728	1.000	41.6	78.396	
25	Compression / Flexure	3.262	1.000	165.4	19.728	1.000	41.6	78.396	
26	Compression / Flexure	3.262	1.000	165.4	19.728	1.000	41.6	78.396	
28	Compression / Flexure	3.546	1.000	88.2	40.207	1.000	51.3	69.069	
29	Compression / Flexure	3.000	1.000	88.2	34.016	1.000	51.3	58.434	
30	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
31	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
32	Compression / Flexure	3.546	1.000	86.2	41.123	1.000	52.1	68.012	
33	Compression / Flexure	3.000	1.000	86.2	34.791	1.000	52.1	57.540	
34	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
35	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
36	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938	
37	Compression / Flexure	6.546	1.000	146.2	44.787	1.000	32.8	199.581	

Members with compression / flexure:

Max K_yL / r_y:	162.938	≤ 200
Max K_zL / r_z:	199.581	≤ 200

Figure 4.9: Window 3.3 *Member Slendernesses*

Details...

This results window is shown only when you have selected the respective check box in the *Details* dialog box (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 depending on the type of load. At the end of the list, you find a comparison with the limit values that have been defined in the *Details* dialog box (see Figure 3.4, page 32).

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

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

4.9 Parts List by Member

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

4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	15 - IS 250/250/10/15/0	4	3.00	12.00	17.76	0.12	76.15	228.44	0.914
2	12 - TO 80/80/5/5/5/5	25	5.00	125.00	40.00	0.19	11.78	58.88	1.472
3	1 - IS 450/200/10/20/0	4	3.00	12.00	20.16	0.15	94.99	284.96	1.140
4	1 - IS 450/200/10/20/0	4	6.00	24.00	40.32	0.29	94.99	569.91	2.280
5	13 - RD 24	4	7.81	31.24	2.36	0.01	3.55	27.71	0.111
6	2 - IS 400/180/10/18/0	6	3.01	18.07	27.10	0.18	79.44	239.23	1.435
7	7 - UC 152x152x30 | BS 4-1 (RSA)	4	6.27	25.10	22.61	0.10	30.03	188.43	0.754
8	9 - IS 450/200/10/20/0	8	6.25	50.00	84.00	0.61	94.99	593.66	4.749
9	13 - RD 24	8	8.02	64.18	4.84	0.03	3.55	28.47	0.228
10	2 - IS 400/180/10/18/0	6	3.26	19.57	29.36	0.20	79.44	259.18	1.555
11	6 - UC 203x203x46 | BS 4-1 (RSA)	2	3.55	7.09	8.43	0.04	46.10	163.48	0.327
12	6 - UC 203x203x46 | BS 4-1 (RSA)	3	3.00	9.00	10.70	0.05	46.10	138.31	0.415
13	10 - IS 200/200/8/15/0	2	3.55	7.09	8.40	0.05	57.78	204.87	0.410
14	10 - IS 200/200/8/15/0	3	3.00	9.00	10.66	0.07	57.78	173.33	0.520
15	16 - IS 360/150/8/12/0	1	6.55	6.55	8.54	0.04	49.36	323.12	0.323
16	2 - IS 400/180/10/18/0	6	6.27	37.64	56.47	0.38	79.44	498.42	2.991
17	6 - UC 203x203x46 | BS 4-1 (RSA)	1	4.09	4.09	4.87	0.02	46.10	188.75	0.189
18	10 - IS 200/200/8/15/0	1	4.09	4.09	4.85	0.03	57.78	236.53	0.237
19	6 - UC 203x203x46 | BS 4-1 (RSA)	1	7.09	7.09	8.43	0.04	46.10	327.05	0.327
20	6 - UC 203x203x46 | BS 4-1 (RSA)	1	6.55	6.55	7.78	0.04	46.10	301.79	0.302
Sum		94		479.36	417.63	2.63			20.676

Figure 4.10: Window 4.1 *Parts List by Member*

Details...

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

Part No.

The program automatically assigns item numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column displays the respective length of an individual member.

Total Length

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

Surface Area

For each part, the program indicates the surface area related to the total length. The surface area is determined from the *Surface Area* of the cross-sections that can be seen in windows 1.3 and 2.1 to 2.5 in the cross-section information (see Figure 2.10, page 16).

Volume

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

Unit Weight

The *Unit Weight* of the cross-section is relative to the length of one meter. For tapered cross-sections, the program averages both cross-section masses.

Weight

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

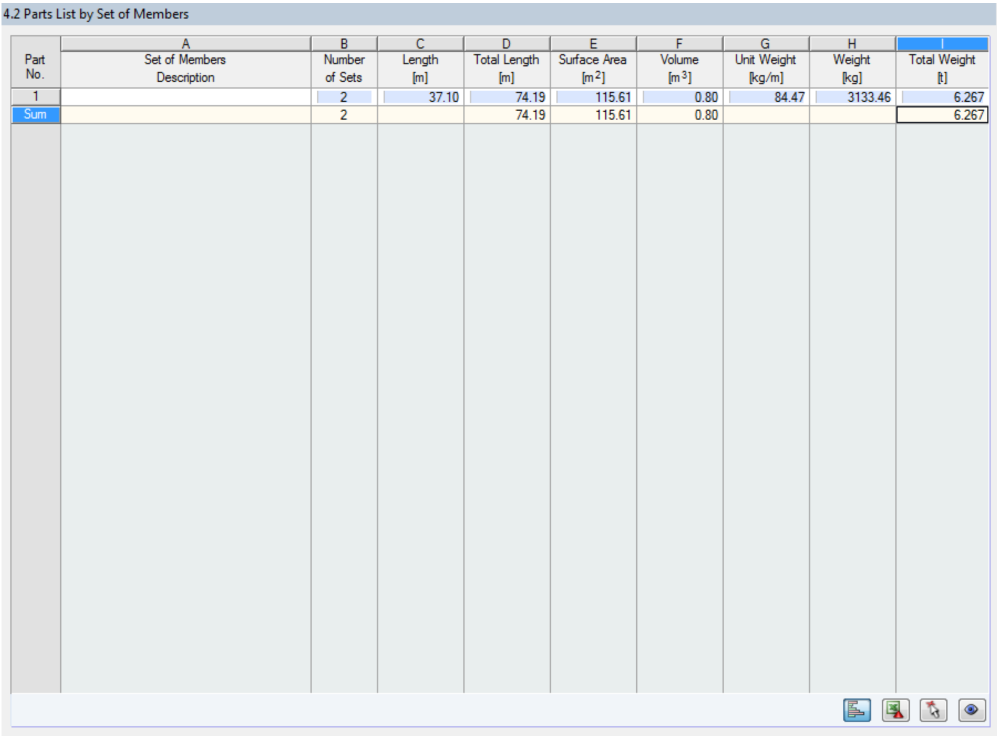
Total Weight

The final column indicates the total mass of each part.

Sum

At the bottom of the list, you find a sum of the values in the columns B, D, E, F, and I. The last cell of the column *Total Weight* gives information about the total amount of steel required.

4.10 Parts List by Set of Members


4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of 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		2	37.10	74.19	115.61	0.80	84.47	3133.46	6.267
Sum		2		74.19	115.61	0.80			6.267

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

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

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

5. Evaluation of Results

The design results can be evaluated in different ways. For this, the buttons in the results windows are very useful which are located below the upper tables.

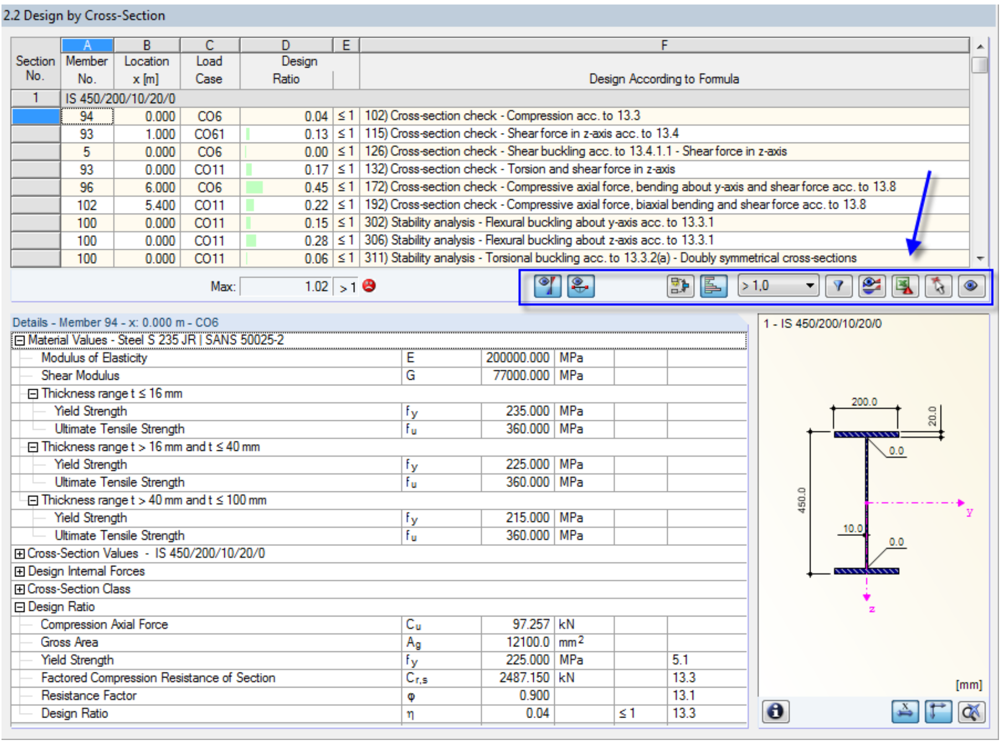


Figure 5.1: Buttons for evaluation of results

These buttons have the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Turns on and off the results of the ultimate limit state design
	Serviceability Limit State Designs	Turns on and off the results of the serviceability limit state design
	New Result Combination	Creates a result combination from the governing load cases and load combinations
	Show Color Bars	Turns on and off the colored reference scales in the results windows
> 1,0 > 1,0 Max Define	Filter Parameters	Describes the filter criterion for the output in the tables: Design ratios greater than 1, maximum value or user-defined limit
	Apply Filter	Displays only rows where the filter parameters are valid (ratio > 1, maximum, user-defined value)
	Result Diagrams	Opens the window *Result Diagram on Member* → chapter 5.2, page 47
	Excel Export	Exports the table to MS Excel / OpenOffice → chapter 7.4.3, page 57

	Member Selection	Allows you to graphically select a member to display its results in the table
	View Mode	Jumps to the RFEM work window to change the view

Table 5.1: Buttons in results windows 2.1 through 2.5

5.1 Results in the 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 when you want to find the position of a particular member in the model: The member selected in the RF-STEEL SANS results window is highlighted in the selection color in the background graphic. Furthermore, an arrow indicates the member's x-location that is displayed in the selected table row.

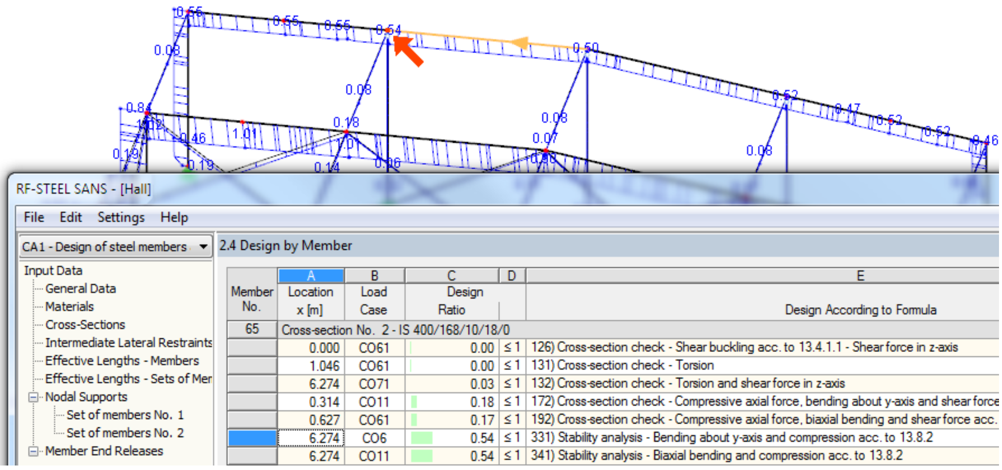


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model

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

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

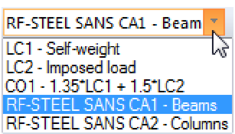
RFEM work window

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

In the *Results* navigator, you can specify which design ratios of the service or ultimate limit state design you want to display 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 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 means of the list in the RFEM menu bar.

To adjust the graphical representation of the results, you can select *Results → Members* in the *Display* navigator. The display of the design ratios is *Two-Colored* by default.

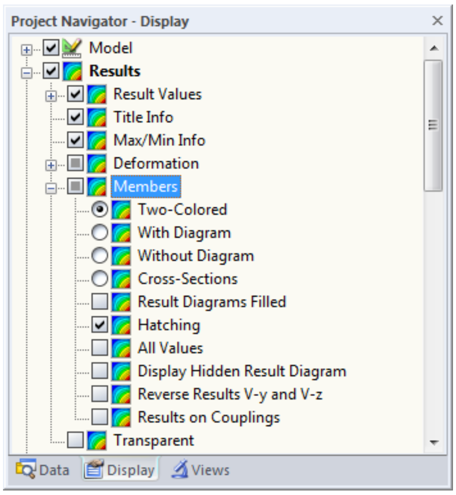


Figure 5.3: *Display* navigator: Results → Members

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

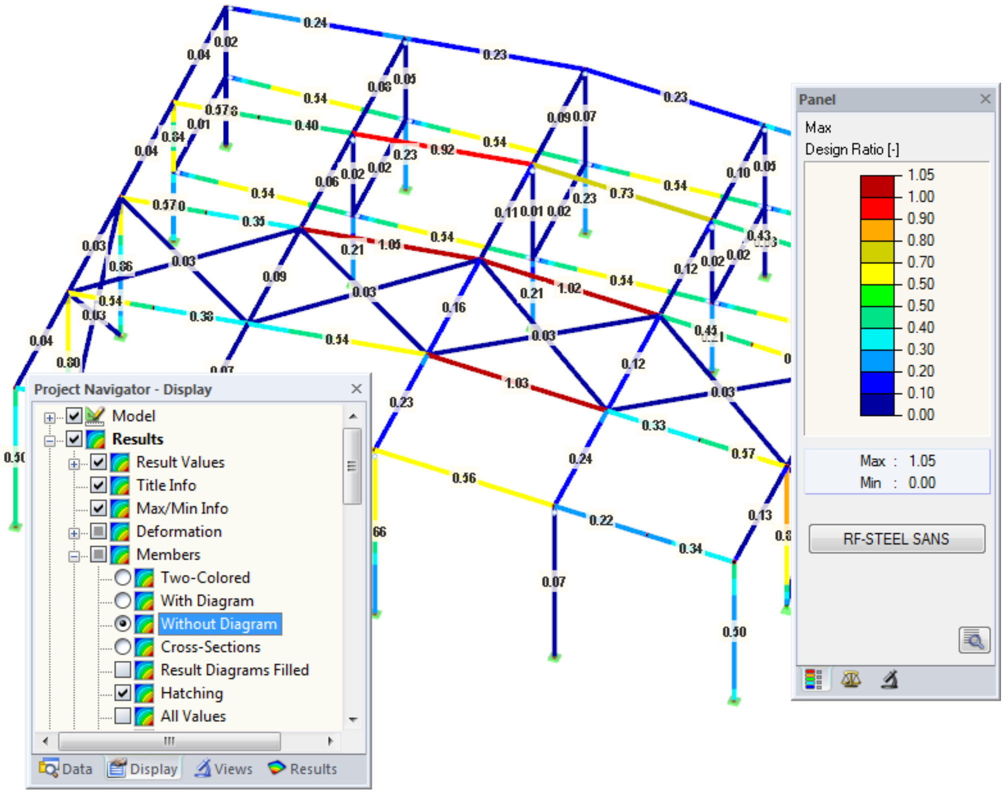


Figure 5.4: Design ratios with display option *Without Diagram*

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

To return to the RF-STEEL SANS module, click [RF-STEEL SANS] in the panel.

5.2 Result Diagrams

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

To do this, select the member (or set of members) in the RF-STEEL SANS results window by clicking in the table row of the member. Then open the *Result Diagram on Member* dialog box by clicking the button shown on the left. The button is located below the upper results table (see Figure 5.1, page 44).

The result diagrams are also available in 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, graphically showing the distribution of the maximum design values on the member or set of members.

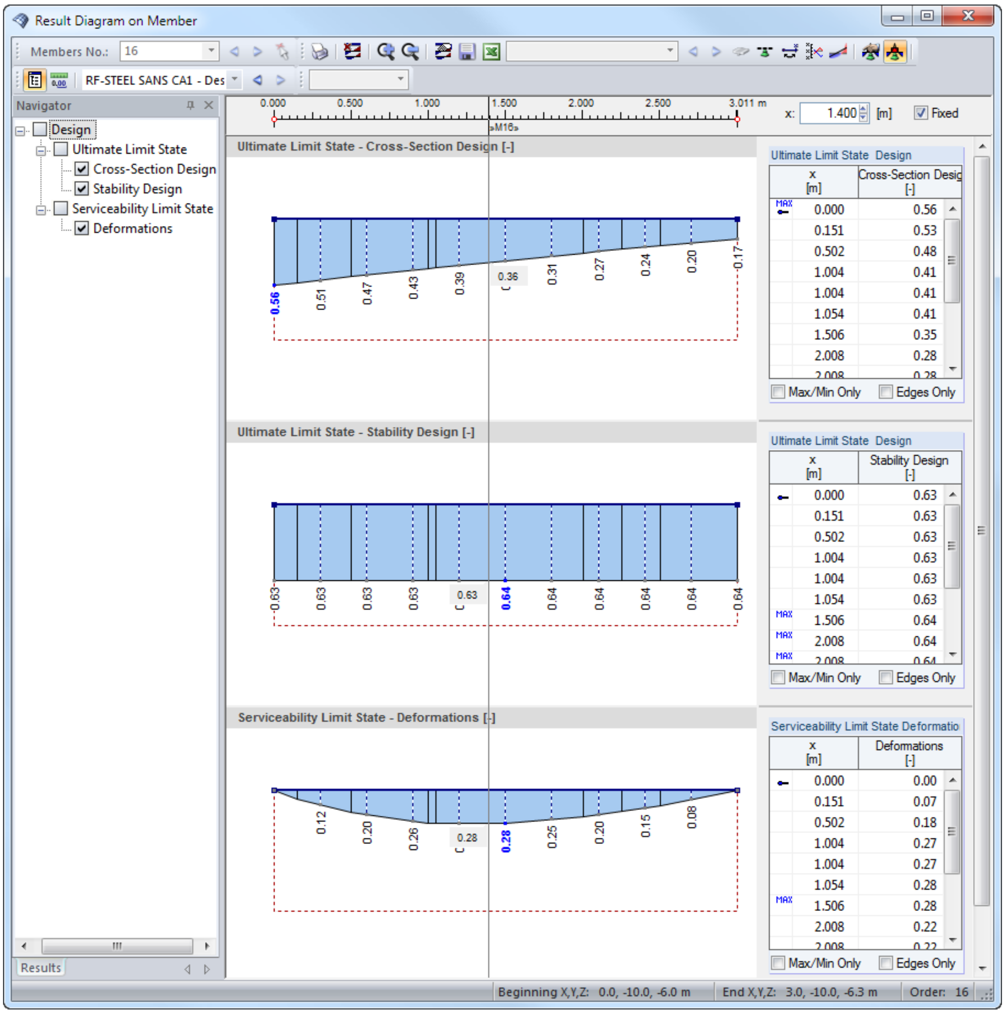
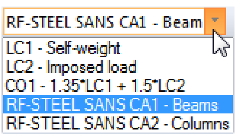


Figure 5.5: Dialog box *Result Diagram on Member*



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

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

5.3 Filter for Results

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

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

Filtering designs

The design ratios can easily be used as filter criteria in the RFEM work window which you can access by clicking [Graphics]. To apply this filter function, the panel must be displayed. If it is not shown, 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 register is not available for the two-colored results display, you have to use the *Display* navigator and set the display options *Colored With/Without Diagram* or *Cross-Sections* first.

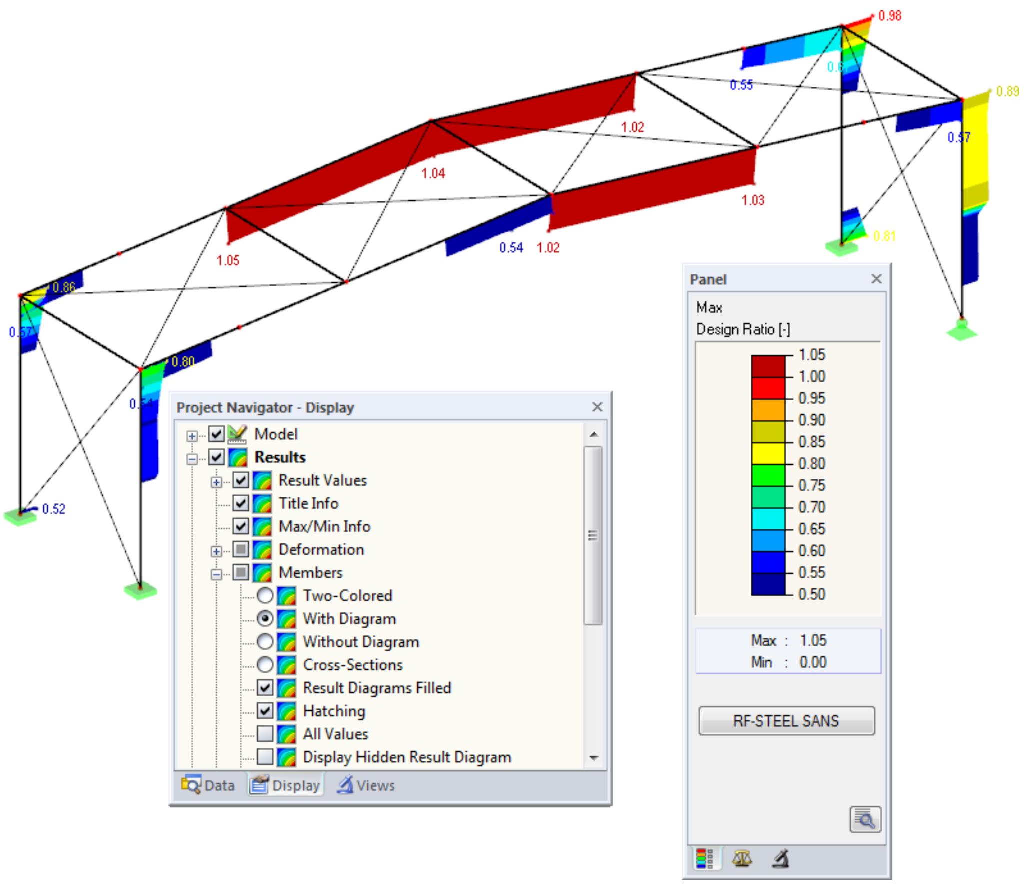


Figure 5.6: Filtering design ratios with adjusted color spectrum

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

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

Filtering members

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

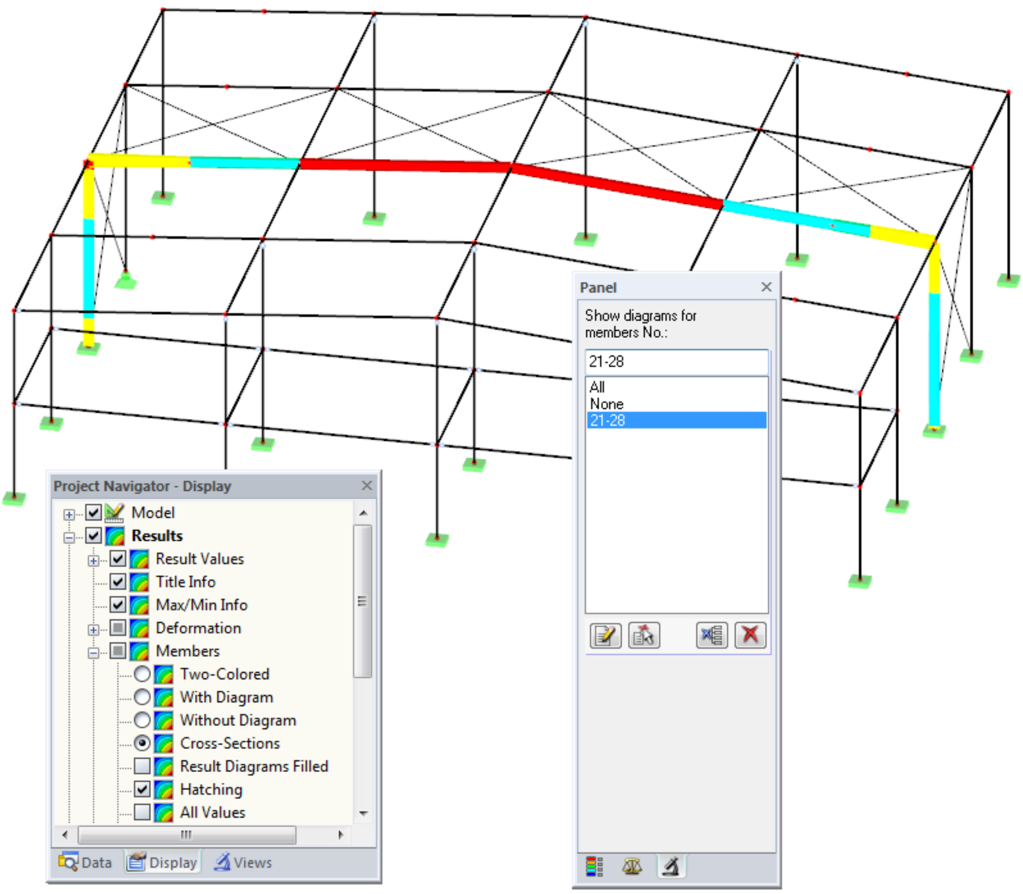


Figure 5.7: Member filter for the stress ratios of a hall frame

Unlike the partial view function (*Visibilities*), the graphic displays the entire model. 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

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

The printout report is described in the RFEM manual. In particular, chapter 10.1.3.4 *Selecting Data of Add-on Modules* describes how to select input and output data of add-on modules for the printout.

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

6.2 RF-STEEL SANS Graphic Printout

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

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.

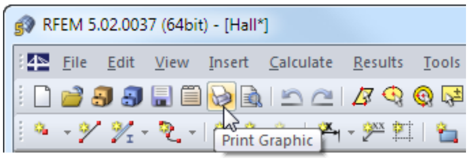


Figure 6.1: Button *Print Graphic* in RFEM toolbar

Result Diagrams

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

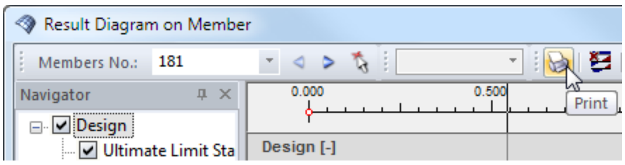


Figure 6.2: Button *Print Graphic* in the dialog box *Result Diagram on Member*

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

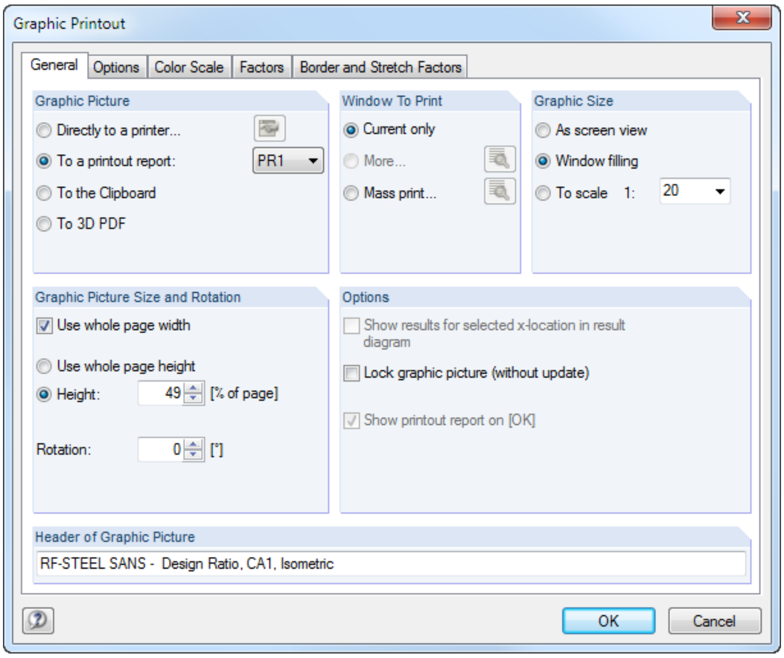
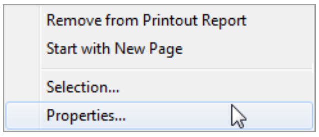


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

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



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

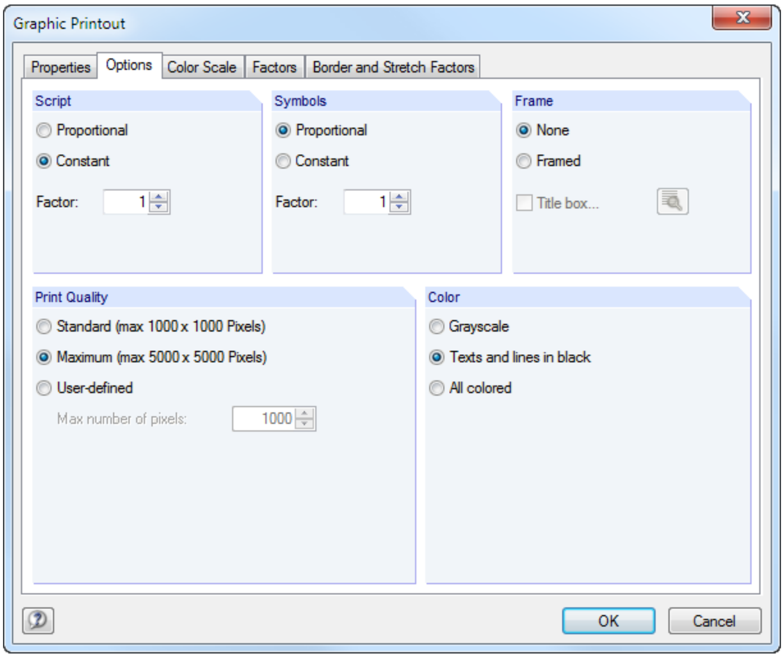


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

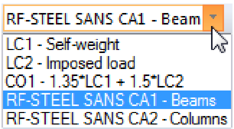
7. General Functions

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 changed materials, partial safety factors, optimization).

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



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

Create New Design Case

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

File → New Case.

The following dialog box appears:

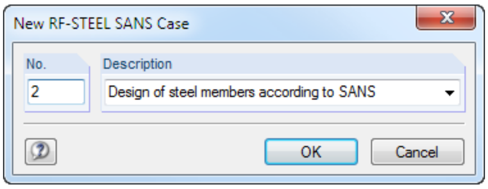


Figure 7.1: Dialog box *New RF-STEEL SANS Case*

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

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

Rename a Design Case

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

File → Rename Case.

The following dialog box appears:

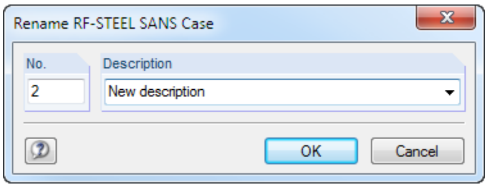


Figure 7.2: Dialog box *Rename RF-STEEL SANS Case*

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 SANS menu and click

File → Copy Case.

The following dialog box appears:

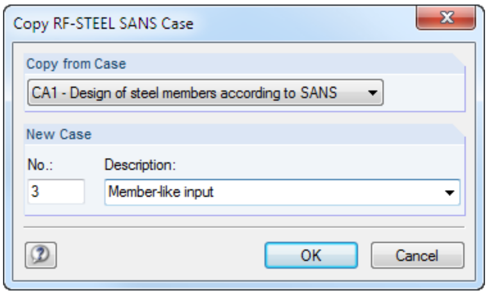


Figure 7.3: Dialog box *Copy RF-STEEL SANS Case*

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

Delete a Design Case

To delete design cases, use the RF-STEEL SANS menu and click

File → Delete Case.

The following dialog box appears:

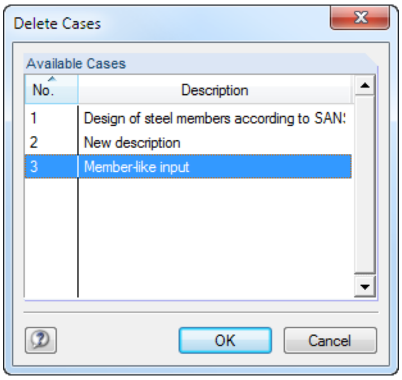
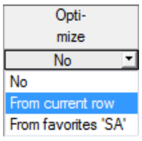


Figure 7.4: Dialog box *Delete Cases*

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 option to optimize overloaded or little utilized cross-sections. To do this, select in column D or E of the relevant cross-sections in the 1.3 *Cross-Sections* window whether to determine the cross-section *From the current row* or the user-defined *Favorites* (see Figure 2.8, page 14). You can also start the cross-section optimization in the results windows by using the context menu

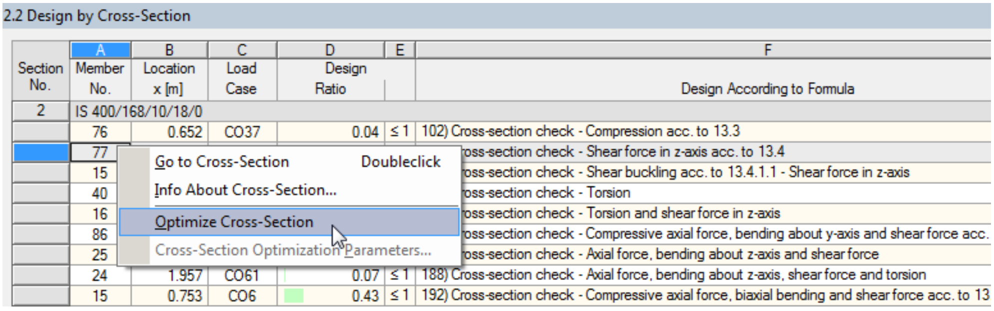


Figure 7.5: Context menu for cross-section optimization

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

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

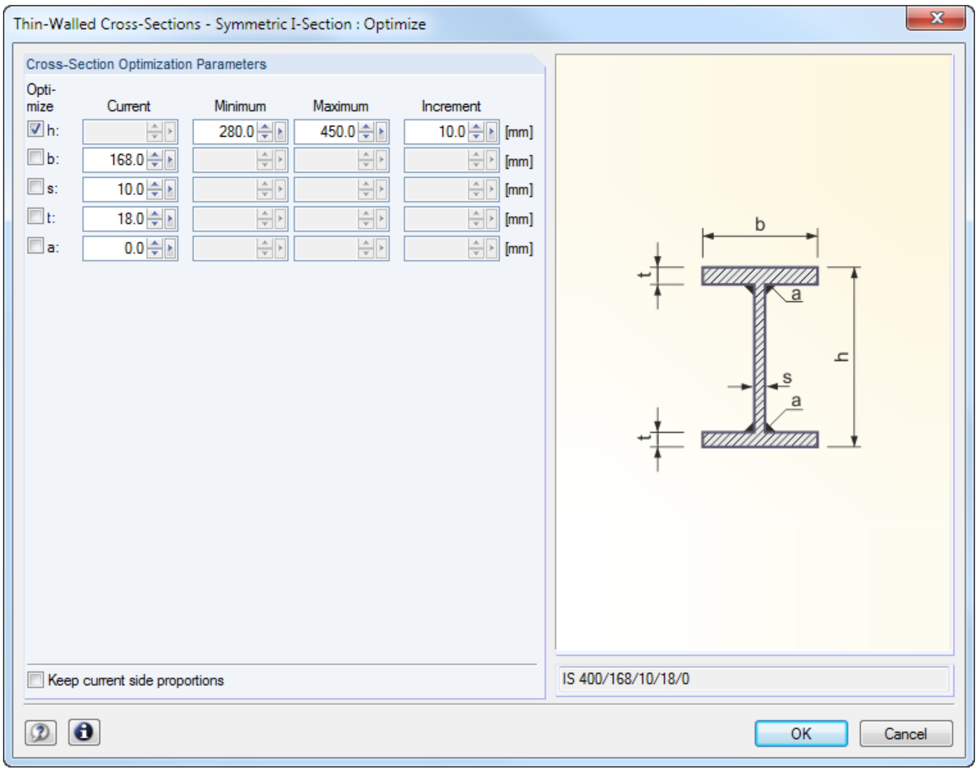


Figure 7.6: Dialog box *Welded Cross-Sections - I symmetric : Optimize*

By selecting the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. This enables the *Minimum* and *Maximum* columns, where you can specify the upper and lower limits of the parameter. The *Increment* column determines the interval in which the size of the parameter varies during the optimization process.

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

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

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

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

Edit → Export All Cross-Sections to RFEM.

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

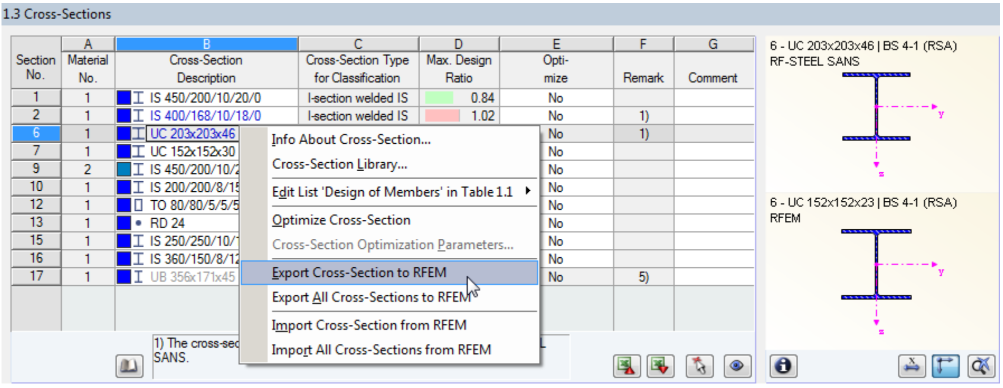


Figure 7.7: Context menu in window 1.3 *Cross-Sections*

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

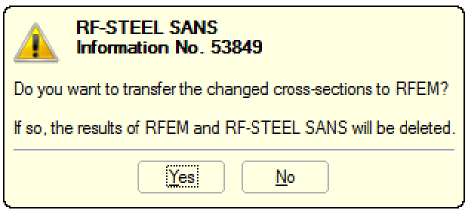


Figure 7.8: Query before transfer of modified cross-sections to RFEM

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

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

If you optimize a tapered member, the program modifies the member start and end. 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, thus manually modeling the taper layout.

7.3 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed in one dialog box. In RF-STEEL SANS, you can use the menu to define the units. To open the corresponding dialog box, click

Settings → **Units and Decimal Places**.

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

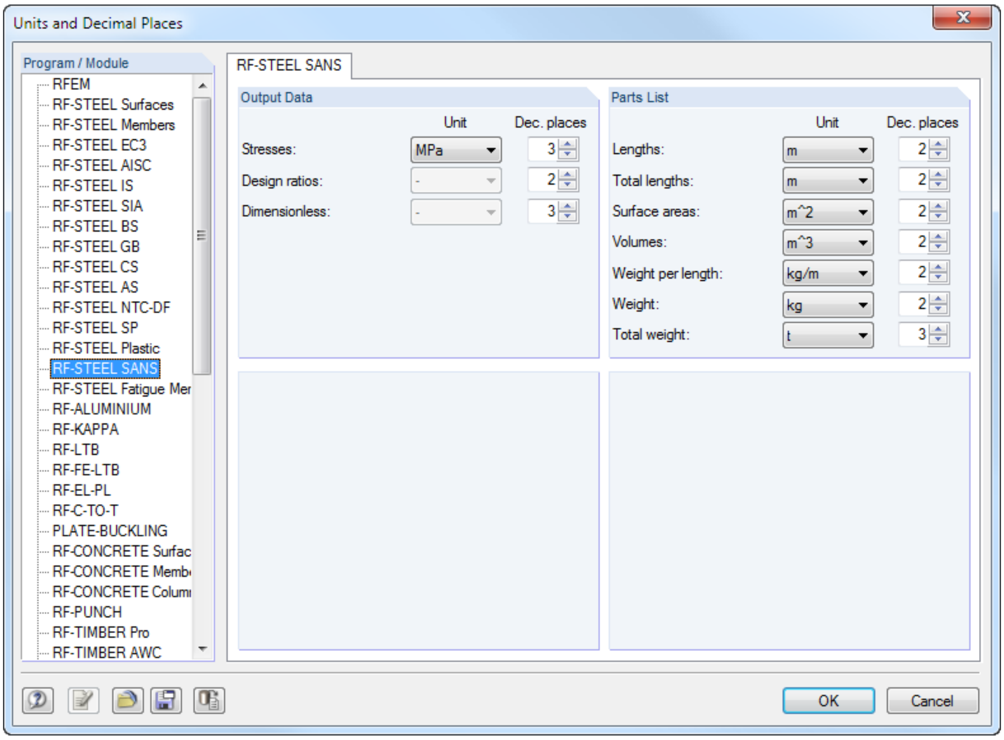


Figure 7.9: Dialog box *Units and Decimal Places*

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

7.4 Data Transfer

7.4.1 Export Material to RFEM

If you have adjusted the materials in RF-STEEL SANS for design, you can export the modified materials to RFEM in a similar manner as you export members and cross-sections: Open window 1.2 *Materials*, and then click

Edit → **Export All Materials to RFEM**.

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

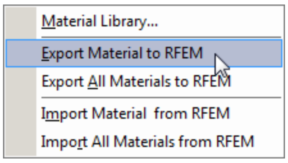


Figure 7.10: Context menu of window 1.2 *Materials*

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

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

7.4.2 Export Effective Lengths to RFEM

If you have adjusted the materials in RF-STEEL SANS for design, you can export the modified materials to RFEM in a similar manner as you export cross-sections: Open window 1.5 *Effective Lengths - Members*, and then click

Edit → **Export All Effective Lengths to RFEM**.

or use the corresponding option on the context menu of window 1.5.

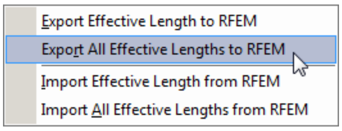


Figure 7.11: Context menu of window 1.5 *Effective Lengths - Members*

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

If the modified effective lengths have not been exported to RFEM yet, you can retransfer the original effective lengths to the design module, using the options shown in Figure 7.11. Please note, however, that this option is only available in window 1.5 *Effective Lengths - Members* and 1.6 *Effective Lengths - Sets of Members*.

7.4.3 Export Results

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

Clipboard

To copy cells selected in the results windows to the Clipboard, press the 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

You can print the data of the RF-STEEL SANS add-on module into the global printout report (see chapter 6.1, page 50) for export. Then, in the printout report, click

File → Export to RTF.

The function is described in the RFEM manual, 10.1.11.

Excel / OpenOffice

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

File → Export Tables.

The following export dialog box appears

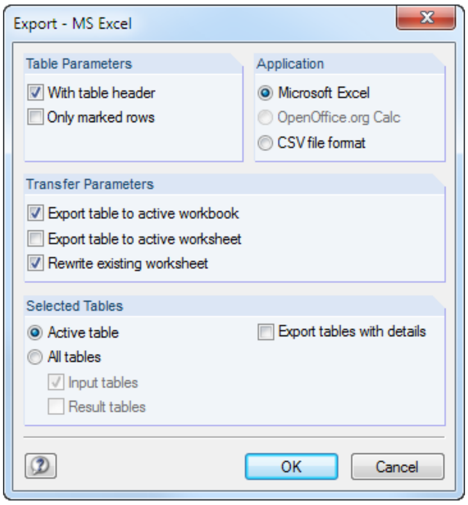
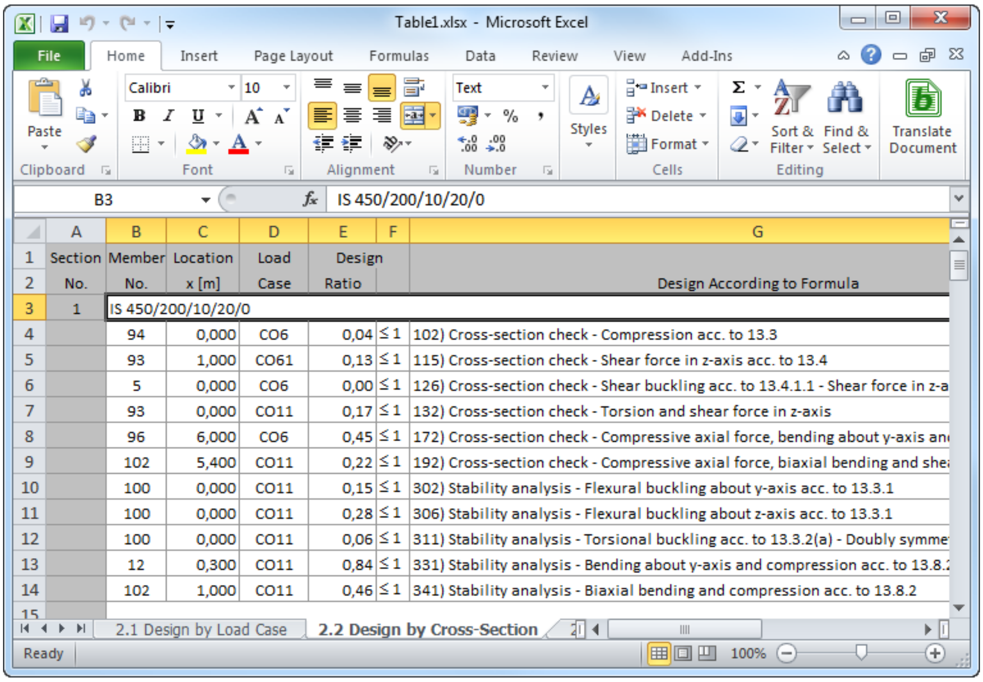


Figure 7.12: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, you can start the export by clicking [OK]. Excel or OpenOffice will be started automatically, i.e. they do not have to be opened first.



A	B	C	D	E	F	G
Section No.	Member No.	Location x [m]	Load Case	Design Ratio		Design According to Formula
1	IS 450/200/10/20/0					
	94	0,000	CO6	0,04	≤ 1	102) Cross-section check - Compression acc. to 13.3
	93	1,000	CO61	0,13	≤ 1	115) Cross-section check - Shear force in z-axis acc. to 13.4
	5	0,000	CO6	0,00	≤ 1	126) Cross-section check - Shear buckling acc. to 13.4.1.1 - Shear force in z-a
	93	0,000	CO11	0,17	≤ 1	132) Cross-section check - Torsion and shear force in z-axis
	96	6,000	CO6	0,45	≤ 1	172) Cross-section check - Compressive axial force, bending about y-axis an
	102	5,400	CO11	0,22	≤ 1	192) Cross-section check - Compressive axial force, biaxial bending and she
	100	0,000	CO11	0,15	≤ 1	302) Stability analysis - Flexural buckling about y-axis acc. to 13.3.1
	100	0,000	CO11	0,28	≤ 1	306) Stability analysis - Flexural buckling about z-axis acc. to 13.3.1
	100	0,000	CO11	0,06	≤ 1	311) Stability analysis - Torsional buckling acc. to 13.3.2(a) - Doubly symme
	12	0,300	CO11	0,84	≤ 1	331) Stability analysis - Bending about y-axis and compression acc. to 13.8.2
	102	1,000	CO11	0,46	≤ 1	341) Stability analysis - Biaxial bending and compression acc. to 13.8.2

Figure 7.13: Results in Excel

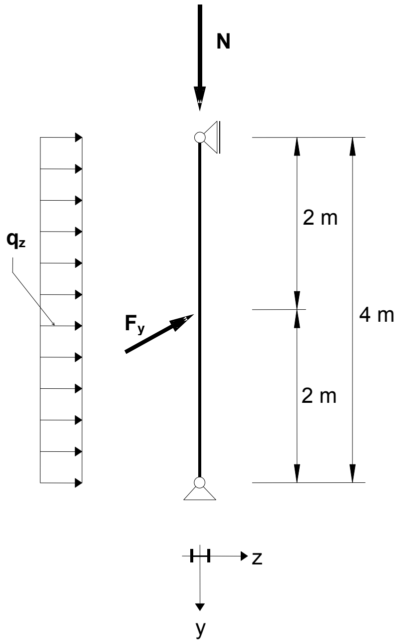
8. Example

Column with Biaxial Bending

In this example, the stability design of flexural buckling and lateral-torsional buckling is carried out by analyzing the relevant interaction conditions according to [1].

Design values

System and loads



Design values of static loads:

N_d = 300 kN
$q_{z,d}$ = 5.0 kN/m
$F_{y,d}$ = 7.5 kN

Cross-section: UC 152x152x37

Material: Steel S 235

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

Internal forces according to linear static analysis

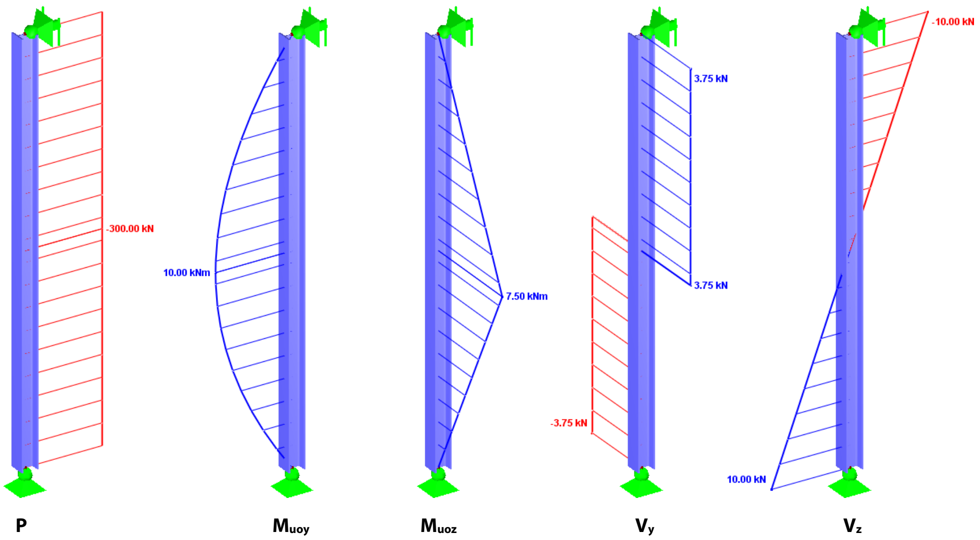


Figure 8.2: Internal Forces

Design location (decisive location x)

The design is performed for all locations x (see chapter 4.5) of the member. The following internal forces act in the decisive location at x = 2.00 m:

P = –300.00 kN M_{uoy} = 10.00 kNm M_{uoz} = 7.50 kNm V_y = 3.75 kN V_z = 0.00 kN

Cross-section properties UC 152x152x37, Steel S 235

Property	Symbol	Value	Unit
Area of cross-section	A_g	47.11	cm^2
Moment of inertia	I_y	2210.00	cm^4
Moment of inertia	I_z	7060.00	cm^4
Radius of inertia	r_y	6.85	cm
Radius of inertia	r_z	3.87	cm
Polar radius of gyration	r_o	7.86	cm
Polar radius of gyration	$r_{o,M}$	41.90	cm
Cross-section weight	G	37.00	kg/m
Torsional constant	J	19.70	cm^4
Warping constant	C_a	399000.00	cm^6
Elastic section modulus	S_y	274.00	cm^3
Elastic section modulus	S_z	91.50	cm^3
Plastic section modulus	Z_y	309.00	cm^3
Plastic section modulus	Z_z	140.00	cm^3

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

Elastic flexural buckling stress

$$f_{ez} = \frac{\pi^2 \cdot E}{(K_z \cdot L / r_z)^2} = \frac{\pi^2 \cdot 200000}{(1 \cdot 4000 / 38.71)^2} = 184.866\,\text{MPa}$$

Elastic flexural buckling load

$$C_{e,z} = A_g \frac{\pi^2 \cdot E}{(K_z \cdot L / r_z)^2} = 4711 \frac{\pi^2 \cdot 200000}{(1 \cdot 4000 / 38.71)^2} = 870.90\,\text{kN}$$

Cross-section type acc. to Table 3 is “compact”: $A_e = A_g$

Slenderness ratio

$$\lambda_z = \frac{K_z \cdot L}{r_z} \sqrt{\left(\frac{f_y}{\pi^2 \cdot E}\right)} = \frac{1 \cdot 4000}{38.71} \sqrt{\left(\frac{235}{\pi^2 \cdot 200000}\right)} = 1.127$$

The appropriate constant *n* is given in [1], clause 13.3.1.

n = 1.34 for hot-rolled I-section

Factored member resistance in compression

$$C_{r,z} = \phi A \cdot f_y \left(1 + \lambda_z^{2n}\right)^{-1/n} = 0.9 \cdot 4711 \cdot 235 \left(1 + 1.127^{2 \cdot 1.34}\right)^{-1/1.34} = 522.04\,\text{kN}$$

Design ratio

$$\frac{C_u}{C_{r,z}} = \frac{300}{522.04} = \underline{\underline{0.575 \le 1}}$$

Result values from RF-STEEL SANS calculation

Compression Axial Force	C_u	300.00	kN		
Factored Compression Resistance of Section	$C_{r,s}$	996.38	kN		13.3
Yield Strength	f_y	235	MPa		5.1
Modulus of Elasticity	E	200000	MPa		
Gross Area	A_g	4711	mm^2		
Second Moment of Area	I_z	7060000	mm^4		
Radius of Gyration	r_z	38.712	mm		
Effective Length	$L_{e,z}$	4000	mm		
Geometrical Slenderness Ratio	$\lambda_{g,z}$	103.327			
Elastic Flexural Buckling Load	$C_{e,z}$	870.99	kN		13.3.1
Elastic Flexural Buckling Stress	$f_{e,z}$	184.885	MPa		13.3.1
Slenderness Ratio	λ_z	1.127			13.3.1
Cross-Section Fabrication Factor	n	1.340			13.3.1
Resistance Factor	Φ	0.900			13.1
Factored Member Resistance	$C_{r,m,z}$	521.82	kN		13.3.1
Design Ratio	η	0.575		≤ 1	13.3.1

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

Elastic flexural buckling stress

$$f_{e,y} = \frac{\pi^2 \cdot E}{(K_y \cdot L / r_y)^2} = \frac{\pi^2 \cdot 200000}{(1 \cdot 4000 / 68.49)^2} = 578.714\,\text{MPa}$$

Elastic flexural buckling load

$$C_{e,y} = A_g \frac{\pi^2 \cdot E}{(K_y \cdot L / r_y)^2} = 4711 \frac{\pi^2 \cdot 200000}{(1 \cdot 4000 / 68.49)^2} = 2726.32\,\text{kN}$$

Cross-section type acc. to Table 3 is "compact": $A_e = A_g$

Slenderness ratio

$$\lambda_y = \frac{K_y \cdot L}{r_y} \sqrt{\left(\frac{f_y}{\pi^2 \cdot E}\right)} = \frac{1 \cdot 4000}{68.49} \sqrt{\left(\frac{235}{\pi^2 \cdot 200000}\right)} = 0.637$$

The appropriate constant *n* is given in [1], clause 13.3.1.

n = 1.34 for hot-rolled I-section

Factored member resistance in compression

$$C_{r,y} = \phi A \cdot f_y \left(1 + \lambda_y^{2n}\right)^{-1/n} = 0.9 \cdot 4711 \cdot 235 \left(1 + 0.637^{2 \cdot 1.34}\right)^{-1/1.34} = 819.86\text{kN}$$

Design ratio

$$\frac{C_u}{C_{r,y}} = \frac{300}{819.86} = \underline{\underline{0.366 \le 1}}$$

Result values from RF-STEEL SANS calculation

Compression Axial Force	C_u	300.00	kN		
Factored Compression Resistance of Section	$C_{r,s}$	996.38	kN		13.3
Yield Strength	f_y	235	MPa		5.1
Modulus of Elasticity	E	200000	MPa		
Gross Area	A_g	4711	mm^2		
Second Moment of Area	I_y	22100000	mm^4		
Radius of Gyration	r_y	68.492	mm		
Effective Length	$L_{e,y}$	4000	mm		
Geometrical Slenderness Ratio	$\lambda_{g,y}$	58.401			
Elastic Flexural Buckling Load	$C_{e,y}$	2726.48	kN		13.3.1
Elastic Flexural Buckling Stress	$f_{e,y}$	578.747	MPa		13.3.1
Slenderness Ratio	λ_y	0.637			13.3.1
Cross-Section Fabrication Factor	N	1.340			13.3.1
Resistance Factor	Φ	0.900			13.1
Factored Member Resistance	$C_{r,m,y}$	819.73	kN		13.3.1
Design Ratio	η	0.366		≤ 1	13.3.1

Lateral-torsional buckling

Critical elastic moment acc. to [1], clause 13.6

The critical elastic moment of the unbraced member is given by the formula in clause 13.6 a) ii). The cross-section shape is doubly symmetric – class 1. The member is considered as a simply supported beam.

$$M_{cr} = \frac{\omega_2 \cdot \pi}{KL} \sqrt{E \cdot I_z \cdot G \cdot J + \left(\frac{\pi \cdot E}{KL}\right)^2 \cdot I_z \cdot C_w}$$

$$M_{cr} = \frac{1.00 \cdot \pi}{1 \cdot 4000} \sqrt{2.0e5 \cdot 7.06e6 \cdot 77000 \cdot 197000 + \left(\frac{\pi \cdot 2.0e5}{1 \cdot 4000}\right)^2 \cdot 7.06e6 \cdot 3.99e10} = 132.285 \text{ kNm}$$

where $\kappa = 0.00$ and $\omega_2 = 1.00$.

Factored moment resistance acc. to [1], clause 13.5

Nominal moment plastic resistances acc. to [1], clause 13.5 a)

UC 152x152x37, cross-section class acc. to Table 4 is "1":

$$M_{p,y} = Z_{pl,y} \cdot f_y = 309000 \cdot 235 = 72.62 \text{ kNm}$$

$$M_{p,z} = Z_{pl,z} \cdot f_y = 140000 \cdot 235 = 32.9 \text{ kNm}$$

Factored moment resistance of member

If

$M_{cr} > 0.67 \cdot M_{py}$ $\quad M_{ry} = 1.15\phi M_{py}\left(1 - \frac{0.28 M_{py}}{M_{cr}}\right) \leq \phi M_{py}$

$132.285 > 0.67 \cdot 72.62 = 48.65$ $\quad M_{ry} = \min\left\{1.15 \cdot 0.9 \cdot 72.62\left(1 - \frac{0.28 \cdot 72.62}{132.285}\right), 0.9 \cdot 72.62\right\}$

$M_{ry} = 63.61 \text{kNm}$

Interaction of biaxial bending and compression

The determination of the design ratio is given in [1], clause 13.8.2. To calculate the final design ratio, we need to determinate the values of the interaction factors U_{1y}, U_{1z} and β. These values are calculated according to [1], clause 13.8.4 and 13.8.5. We consider the member as "braced frame".

$$\beta = \min\{0.6 + 0.4 \cdot \lambda_z ; 0.85\} = \min\{0.6 + 0.4 \cdot 1.127; 0.85\} = 0.85$$

The member is subjected to a distributed load in z-axis (ω_{1y} = 1.00) and a concentrated load in y-axis (ω_{1z} = 0.85), see [1] clause 13.8.5 b) and c).

$$U_{1y} = \frac{\omega_{1y}}{1 - C_u / C_{e,y}} = \frac{1.00}{1 - 300/2726.32} = 1.124$$

$$U_{1z} = \frac{\omega_{1z}}{1 - C_u / C_{e,z}} = \frac{0.85}{1 - 300/870.90} = 1.297$$

Interaction design ratio acc. to 13.8.2 (a) – cross-sectional strength

$$C_{r,s} = \phi \cdot A_g \cdot f_y = 0.9 \cdot 4711 \cdot 235 = 996.38 \text{ kN}$$

$$\frac{C_u}{C_{r,s}} + \frac{0.85 \cdot U_{1y} \cdot M_{uy}}{M_{ry}} + \frac{\beta \cdot U_{1z} \cdot M_{uz}}{M_{rz}} \leq 1$$

$$\frac{300.00}{996.38} + \frac{0.85 \cdot 1.124 \cdot 10.00}{0.9 \cdot 72.62} + \frac{0.6 \cdot 1.297 \cdot 7.50}{0.9 \cdot 32.9} = 0.301 + 0.146 + 0.197 = \underline{\underline{0.644 \leq 1}}$$

Interaction design ratio acc. to 13.8.2 (b) – overall member strength

$$\frac{C_u}{C_r} + \frac{0.85 \cdot U_{1y} \cdot M_{uy}}{M_{ry}} + \frac{\beta \cdot U_{1z} \cdot M_{uz}}{M_{rz}} \leq 1$$

$$\frac{300.00}{522.04} + \frac{0.85 \cdot 1.124 \cdot 10.00}{0.9 \cdot 72.62} + \frac{0.85 \cdot 1.297 \cdot 7.50}{0.9 \cdot 32.9} = 0.575 + 0.146 + 0.279 = \underline{\underline{1.000 \leq 1}}$$

Interaction design ratio acc. to 13.8.2 (c) – lateral-torsional buckling strength

$$\frac{C_u}{C_r} + \frac{0.85 \cdot U_{1y} \cdot M_{uy}}{M_{ry}} + \frac{\beta \cdot U_{1z} \cdot M_{uz}}{M_{rz}} \leq 1$$

$$\frac{300.00}{522.04} + \frac{0.85 \cdot 1.124 \cdot 10.00}{63.61} + \frac{0.85 \cdot 1.297 \cdot 7.50}{0.9 \cdot 32.9} = 0.575 + 0.150 + 0.279 = \underline{\underline{1.004 \geq 1}}$$

Dlubal

Result values from RF-STEEL SANS calculation

Compression Axial Force	C_u	300.00	kN		
Bending Moment	$M_{u,y}$	10.00	kNm		
Maximum Bending Moment	$M_{u,y,segm}$	10.00	kNm		
Bending Moment	$M_{u,z}$	7.50	kNm		
Maximum Bending Moment	$M_{u,z,segm}$	7.50	kNm		
Yield Strength	f_y	235	MPa		5.1
Plastic Moment Resistance of Section	$M_{p,y}$	72.62	kNm		
Factored Moment Resistance of Section	$M_{r,s,y}$	65.35	kNm		13.5
Plastic Moment Resistance of Section	$M_{p,z}$	32.90	kNm		
Factored Moment Resistance of Section	$M_{r,s,z}$	29.61	kNm		13.5
Modulus of Elasticity	E	200000	MPa		
Shear Modulus	G	77000	MPa		
Second Moment of Area	I_z	7060000	mm^4		
Torsional Constant	J	197000	mm^4		
Warping Constant	C_w	3.99E+10	mm^6		
Segment Length	L	4000	mm		
Effective Length Factor	K	1.000			Tab. 1 or 2
Effective Length	L_e	4000	mm		10.2
Coefficient	ω_2	1.000			13.6
Critical Elastic Moment	M_{cr}	132.29	kNm		
Factored Moment Resistance of Member	$M_{r,m,y}$	63.61	kNm		13.6
Elastic Flexural Buckling Load	$C_{e,y}$	2726.48	kN		13.3.1
Coefficient	$\omega_{1,y}$	1.000			13.8.5
Factor of Second Order Effect	$U_{1,y}$	1.124			13.8.4
Design Component for M_y	η_{My}	0.150		≤ 1	13.8.2
Factor	β	0.850			13.8.2
Elastic Flexural Buckling Load	$C_{e,z}$	870.99	kN		13.3.1
Coefficient	$\omega_{1,z}$	0.850			13.8.5
Factor of Second Order Effect	$U_{1,z}$	1.297			13.8.4
Design Component for M_z	η_{Mz}	0.279		≤ 1	
Gross Area	A_g	4711	mm^2		
Factored Compression Resistance of Section	$C_{r,s}$	996.38	kN		13.3
Factored Member Resistance	$C_{r,m,y}$	819.73	kN		13.3.1
Factored Member Resistance	$C_{r,m,z}$	521.82	kN		13.3.1
Factored Member Resistance	$C_{r,m,T}$	850.35	kN		13.3.2
Factored Member Resistance	$C_{r,m}$	521.82	kN		13.3
Design Component for N	η_N	0.575		≤ 1	13.8.2
Resistance Factor	Φ	0.900			13.1
Design Ratio	η_1	0.644		≤ 1	13.8.2
Design Ratio	η_2	1.000		≤ 1	13.8.2
Design Ratio	η_3	1.004		≥ 1	13.8.2
Design Ratio	η	1.004		≥ 1	

A Literature

[1] SABS Standards Division: South African National Standard SANS 10162-1:2011
The structural use of steel, Part 1: Limit-states design of hot-rolled steelwork,
Pretoria 2011

[2] Southern African Institute of steel construction: Southern African Steel Construction Handbook, 8th edition, 2013

[3] PARROTT, G.: Structural steel design according to SANS 10162:1-2005, 2nd edition, Durban 2005

B Index