Version
August 2017

Add-on Module

RF-/STEEL NBR

Desing of Steel Members According to
ABNT NBR 8800

Program Description

© Dlubal Software GmbH 2017
Am Zellweg 2
93464 Tiefenbach
Germany

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

Contents

1 Introduction

1.1 Add-on Module RF-/STEEL NBR

The Standard ABNT NBR 8800 *Design of steel and composite structures for buildings* [1] describes the design of steel buildings in the Federative Republic of Brazil. With the add-on modules RF-STEEL NBR (for RFEM) and STEEL NBR (for RSTAB), DLUBAL provides powerful tools for the design of steel beam models according to this Standard.

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

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

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

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

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

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

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

Your DLUBAL team

1.2 Using the Manual

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

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

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

1.3 Starting RF-/STEEL NBR

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

Menu

To start the program from the menu bar, select

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

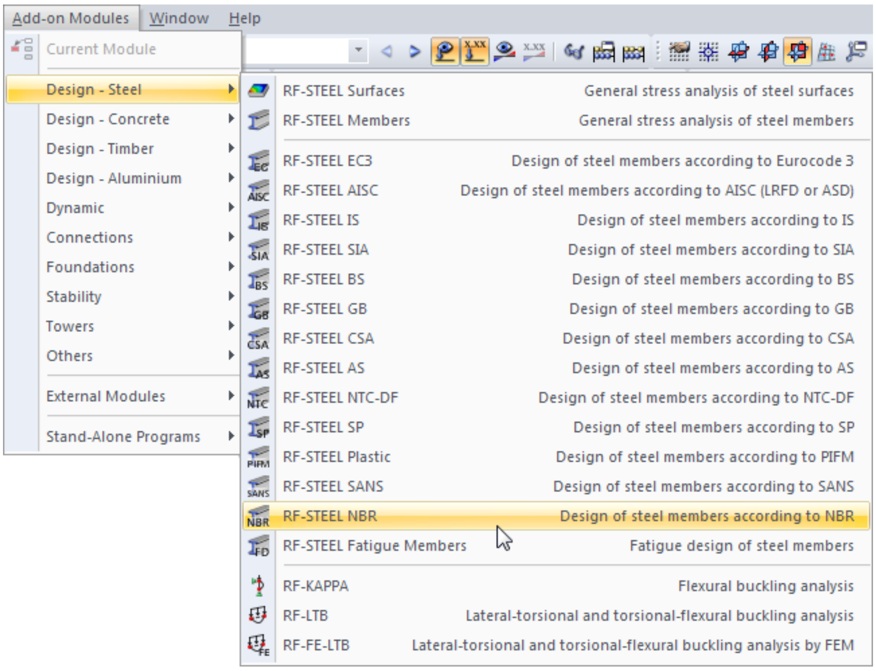


Figure 1.1: Menu *Add-on Modules → Design - Steel → RF-STEEL NBR*

Navigator

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

Add-on Modules → RF-/STEEL NBR.

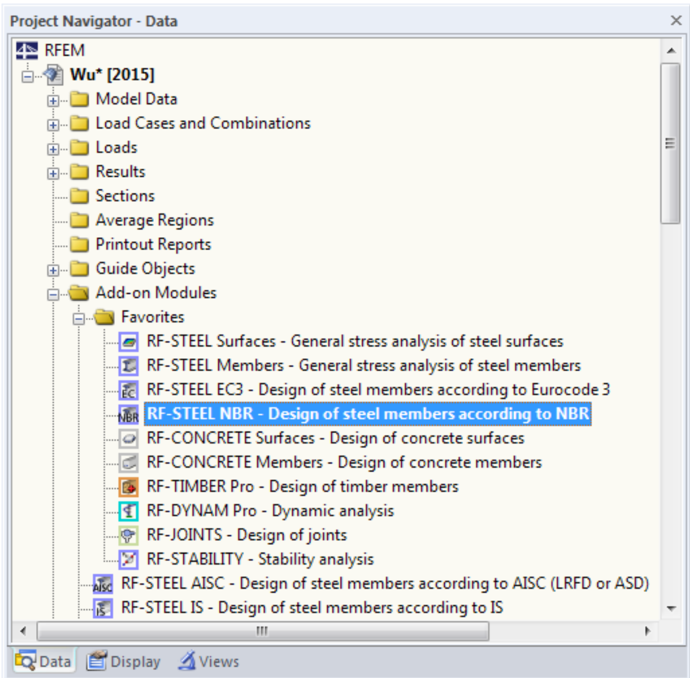


Figure 1.2: *Data* navigator *Add-on Modules → RF-STEEL NBR*

2 Input Data

When you have started the add-on module, a new window appears. In this window, a *Navigator* is displayed on the left. It manages the different window that can be currently selected. The drop-down list above the navigator contains the design cases (see Chapter 7.1, page 39).

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

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

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

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

2.1 General Data

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

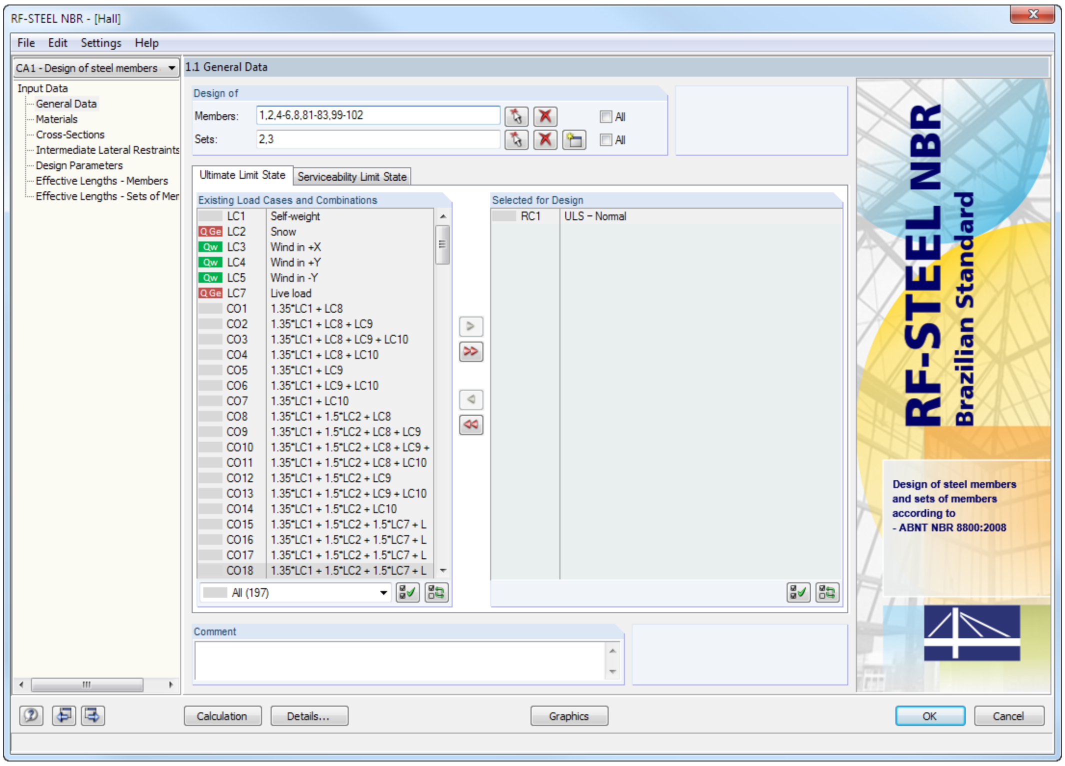


Figure 2.1: Window *1.1 General Data*

Design of

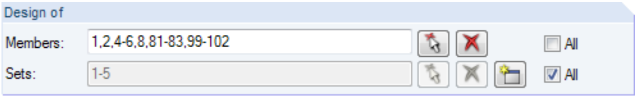


Figure 2.2: Design of members and sets of members

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

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

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

Comment

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

2.1.1 Ultimate Limit State

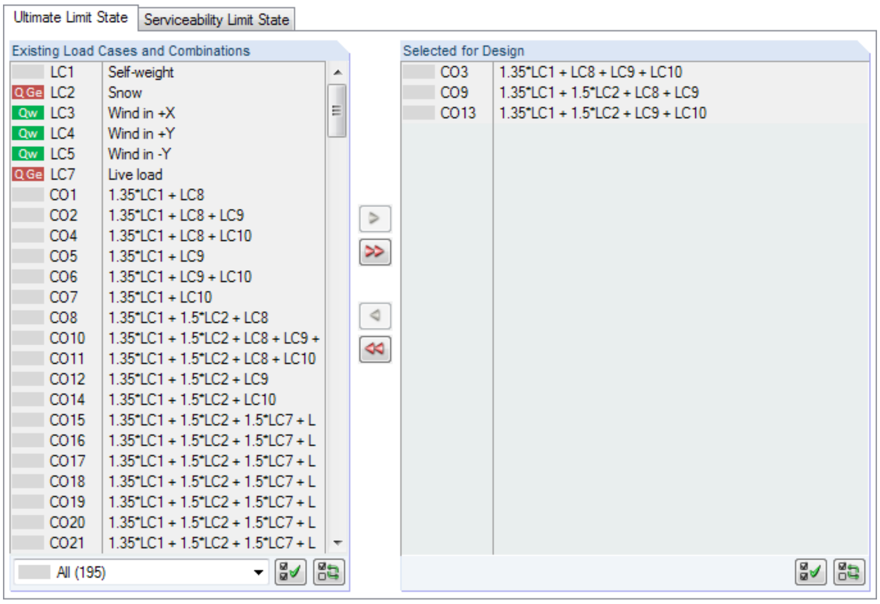


Figure 2.3: Window *1.1 General Data*, tab *Ultimate Limit State*

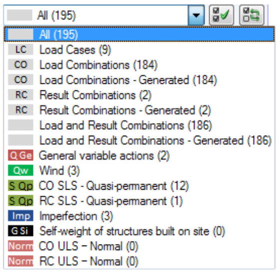
Existing Load Cases and Combinations

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

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

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

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



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

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

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

Selected for Design

The column on the right lists the load cases, load and result combinations selected for design. To remove an item from the list, click [◄] or double-click the item. To transfer the entire list to the left, click [◄◄].

Result combination

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

2.1.2 Serviceability Limit State

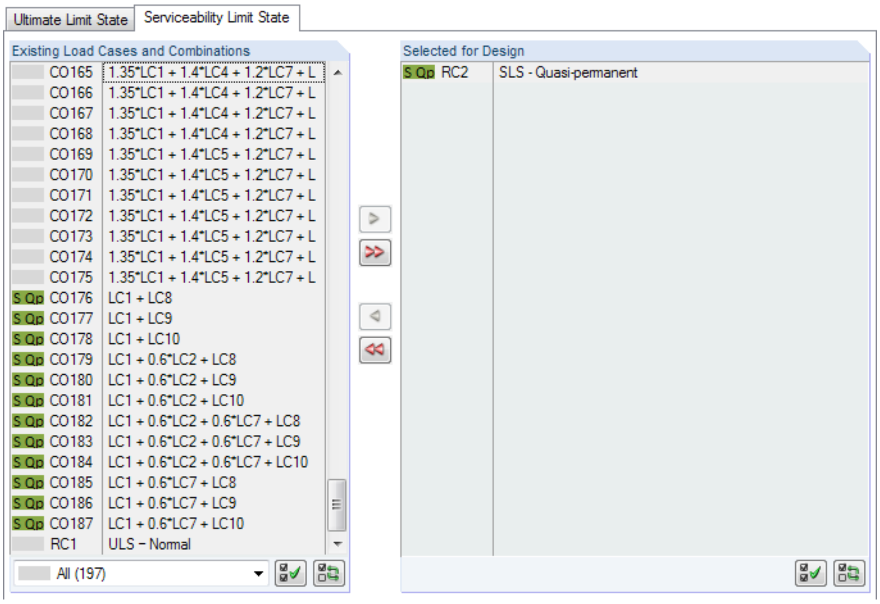


Figure 2.4: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

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

Details...

The limit values of the deflections are preset in the *Details* dialog box (see Figure 3.1, page 19). To adjust those values, click the [Details] button.

In the *1.8 Serviceability Data* Window, you can specify the reference lengths of the deformation analysis (see Chapter 2.8, page 18).

2.2 Materials

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

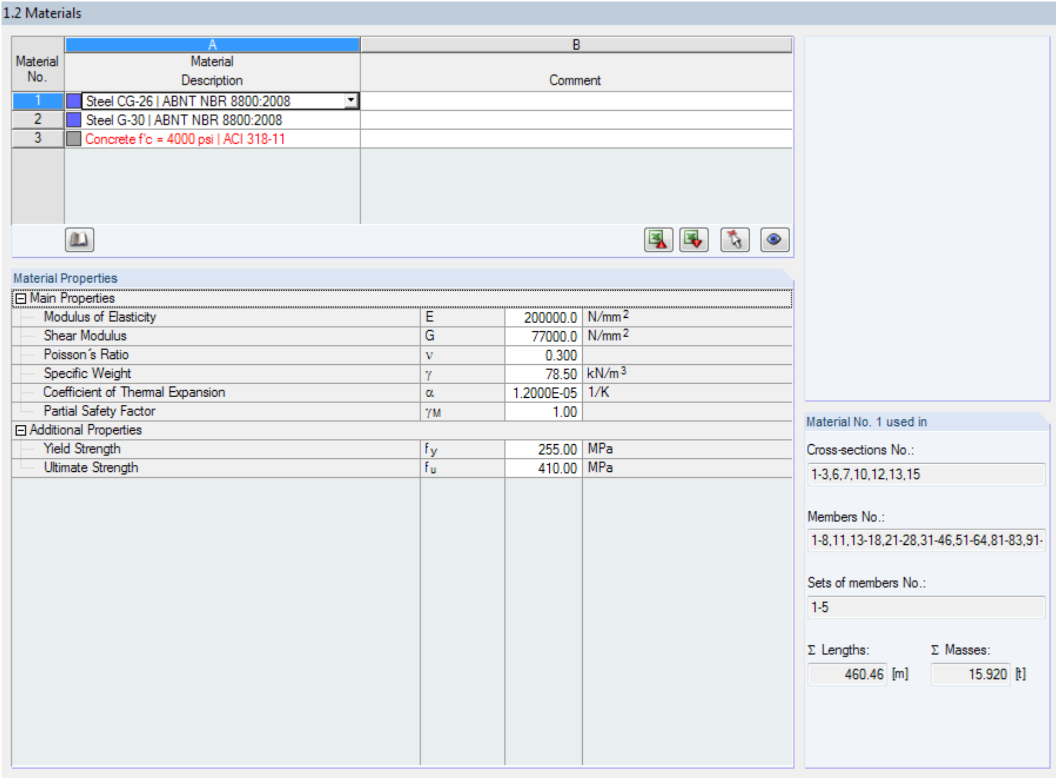


Figure 2.5: Window *1.2 Materials*

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

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

To adjust the units and decimal places of the material properties and stresses, select **Settings** → **Units and Decimal Places** from the menu bar of the module (see Chapter 7.3, page 43).

Material Description

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

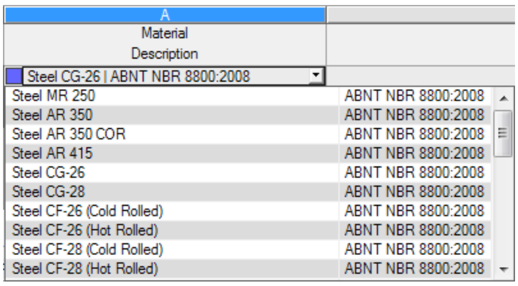


Figure 2.6: List of materials

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

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

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

Material Library

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

Edit → Material Library

or use the [Liberary] button.

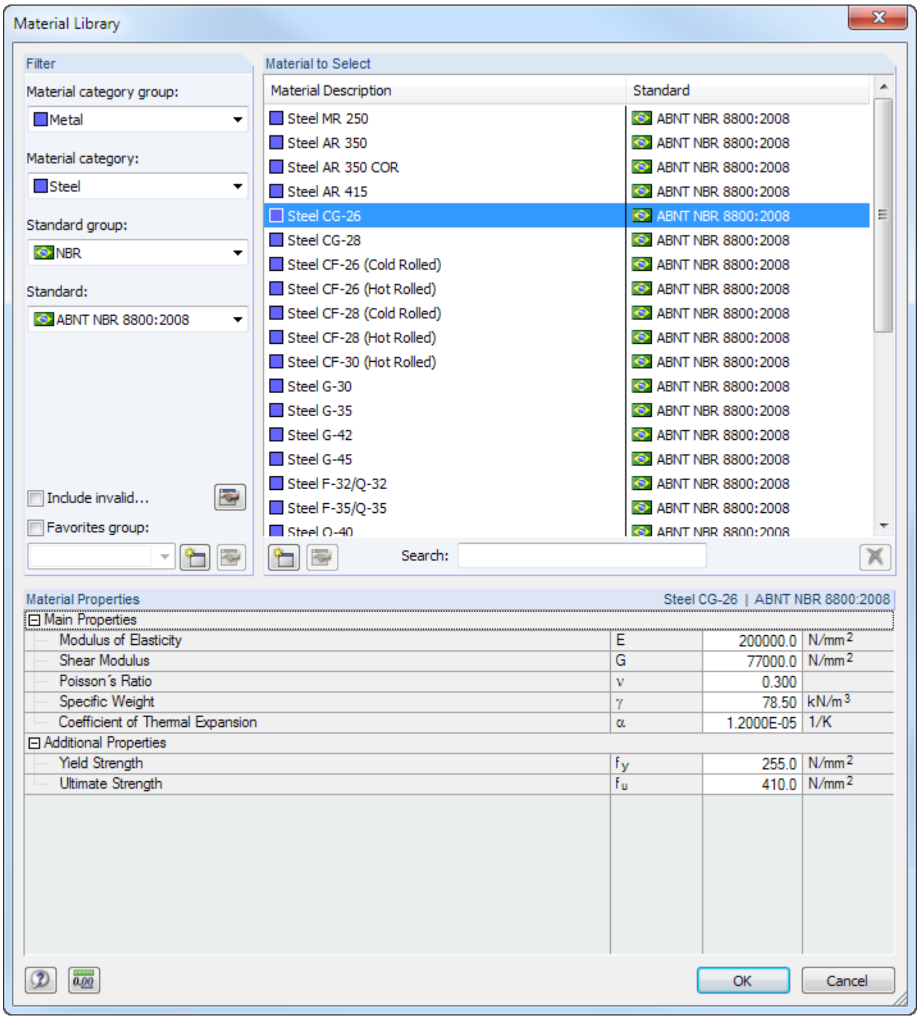


Figure 2.7: Dialog box *Material Library*

In the *Filter* section, *ABNT NBR 8800:2008* is the default Standard. Select the material quality that you want to use for the design in the *Material to Select* list. You can check the corresponding properties in the dialog section below.

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

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

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

2.3 Cross-Sections

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

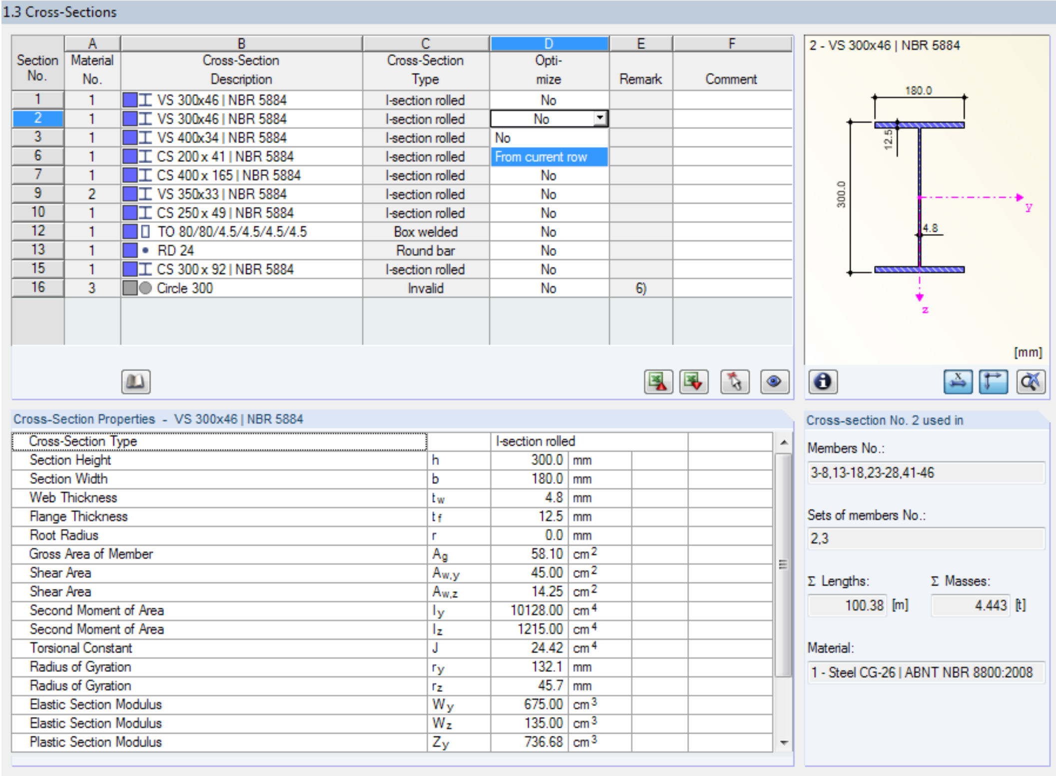


Figure 2.8: Window *1.3 Cross-Sections*

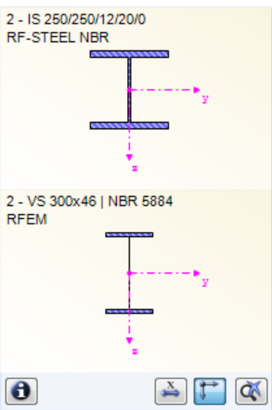
Cross-Section Description

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

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

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

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



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

A modified cross-section will be highlighted in blue.

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

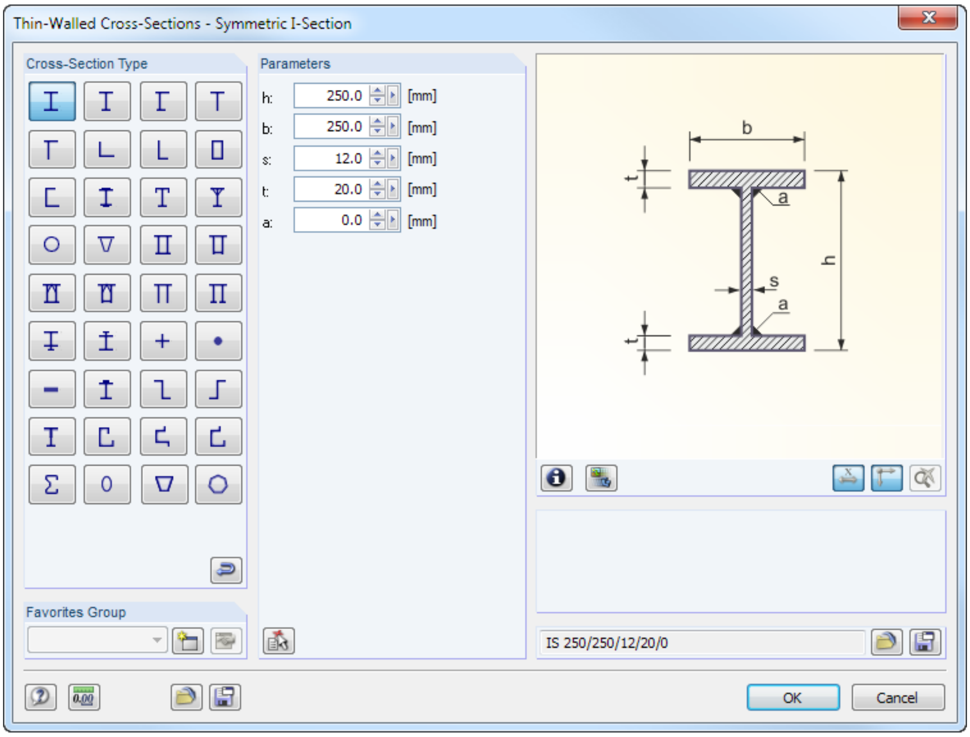


Figure 2.9: IS cross-section types in the cross-section library

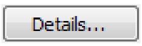
Cross-Section Type

The program displays the type of cross-section that will be used for the classification according to [1] Clause 5.1.2.1.

Max. Design Ratio

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

Optimize



It is possible to optimize every cross-section from the library. The program searches the cross-section that comes as close as possible to a user-defined maximum utilization ratio. You can specify this maximum ratio in the *Details* dialog box (see Figure 3.1, page 19).

To optimize a cross-section, open the drop-down list in column D (resp. E) and select *From current row*. Recommendations on the optimization can be found in Chapter 7.2 on page 41.

Remark

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

Member with Tapered Cross-Section

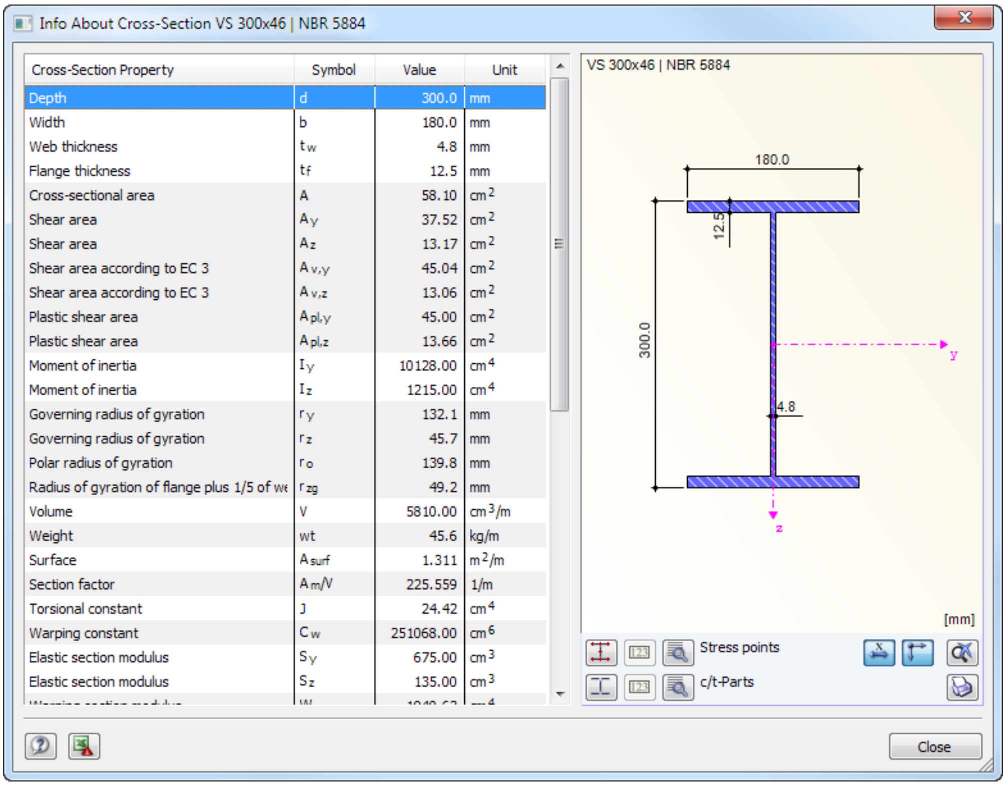
For tapered members with different cross-sections at the member start and member ends, the module displays both section numbers in two rows, according to the definition in RFEM or RSTAB.

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

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

Info About Cross-Section

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

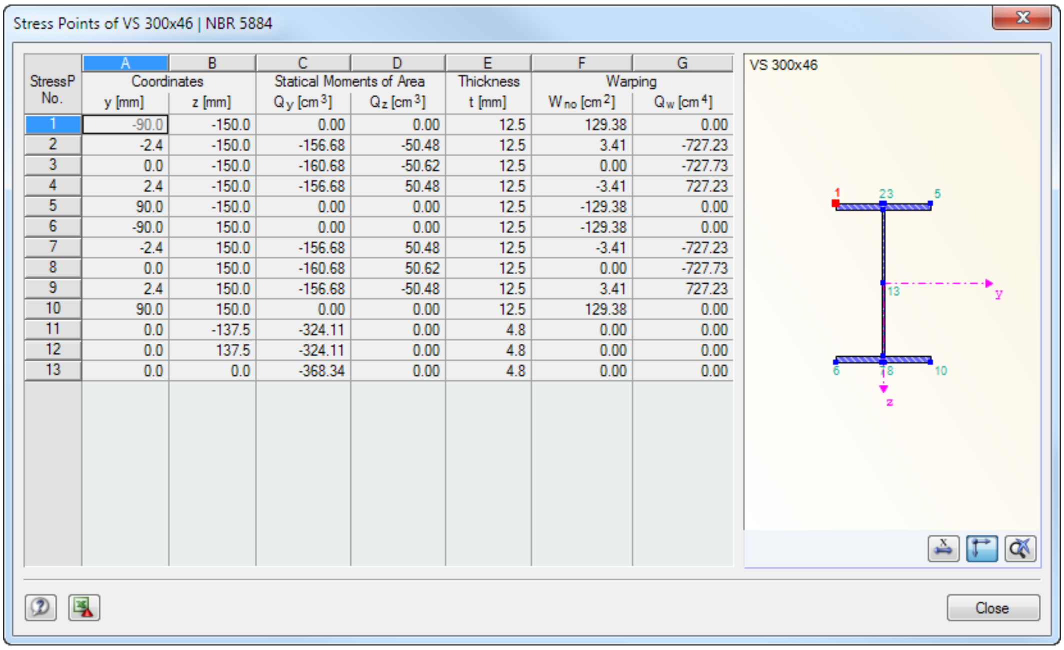


Cross-Section Property	Symbol	Value	Unit
Depth	d	300.0	mm
Width	b	180.0	mm
Web thickness	tw	4.8	mm
Flange thickness	tf	12.5	mm
Cross-sectional area	A	58.10	cm2
Shear area	Ay	37.52	cm2
Shear area	Az	13.17	cm2
Shear area according to EC 3	Av,y	45.04	cm2
Shear area according to EC 3	Av,z	13.06	cm2
Plastic shear area	Apl,y	45.00	cm2
Plastic shear area	Apl,z	13.66	cm2
Moment of inertia	Iy	10128.00	cm4
Moment of inertia	Iz	1215.00	cm4
Governing radius of gyration	ry	132.1	mm
Governing radius of gyration	rz	45.7	mm
Polar radius of gyration	ro	139.8	mm
Radius of gyration of flange plus 1/5 of we	rzg	49.2	mm
Volume	V	5810.00	cm3/m
Weight	wt	45.6	kg/m
Surface	Asurf	1.311	m2/m
Section factor	Am/V	225.559	1/m
Torsional constant	J	24.42	cm4
Warping constant	Cw	251068.00	cm6
Elastic section modulus	Sy	675.00	cm3
Elastic section modulus	Sz	135.00	cm3

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

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

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



Stress P No.	A Coordinates y [mm]	B z [mm]	C Statical Moments of Area Qy [cm3]	D Qz [cm3]	E Thickness t [mm]	F Warping Wno [cm2]	G Qw [cm4]
1	-90.0	-150.0	0.00	0.00	12.5	129.38	0.00
2	-2.4	-150.0	-156.68	-50.48	12.5	3.41	-727.23
3	0.0	-150.0	-160.68	-50.62	12.5	0.00	-727.73
4	2.4	-150.0	-156.68	50.48	12.5	-3.41	727.23
5	90.0	-150.0	0.00	0.00	12.5	-129.38	0.00
6	-90.0	150.0	0.00	0.00	12.5	-129.38	0.00
7	-2.4	150.0	-156.68	50.48	12.5	-3.41	-727.23
8	0.0	150.0	-160.68	50.62	12.5	0.00	-727.73
9	2.4	150.0	-156.68	-50.48	12.5	3.41	727.23
10	90.0	150.0	0.00	0.00	12.5	129.38	0.00
11	0.0	-137.5	-324.11	0.00	4.8	0.00	0.00
12	0.0	137.5	-324.11	0.00	4.8	0.00	0.00
13	0.0	0.0	-368.34	0.00	4.8	0.00	0.00

Figure 2.11: Dialog box *Stress Points of VS 300x46*

2.4 Intermediate Lateral Restraints

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

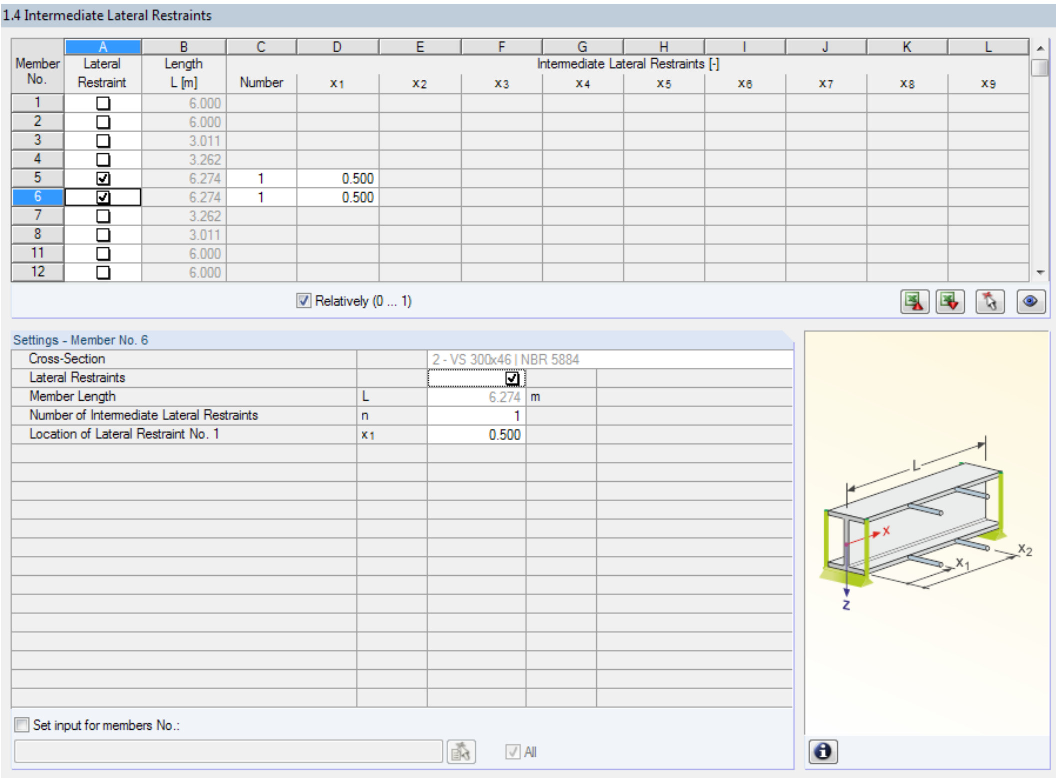


Figure 2.12: Window *1.4 Intermediate Lateral Restraints*

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

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

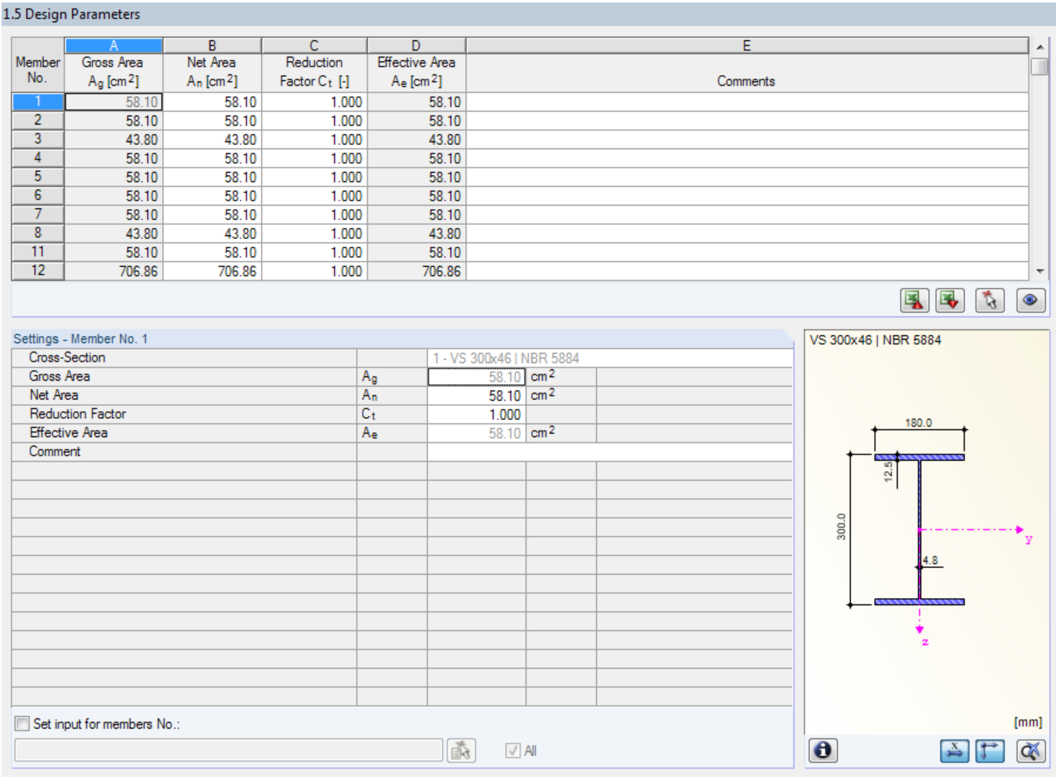
In column C, you can specify the *Number* of the intermediate restraints. Depending on the specification, one or more of the following *Intermediate Lateral Restraints* columns will be available for the definition of the x-locations.

Relatively (0 ... 1)

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

2.5 Design Parameters

This window controls specific parameters that are relevant for the design according to [1].



Member No.	A Gross Area A_g [cm²]	B Net Area A_n [cm²]	C Reduction Factor C_t [-]	D Effective Area A_e [cm²]	E Comments
1	58.10	58.10	1.000	58.10	
2	58.10	58.10	1.000	58.10	
3	43.80	43.80	1.000	43.80	
4	58.10	58.10	1.000	58.10	
5	58.10	58.10	1.000	58.10	
6	58.10	58.10	1.000	58.10	
7	58.10	58.10	1.000	58.10	
8	43.80	43.80	1.000	43.80	
11	58.10	58.10	1.000	58.10	
12	706.86	706.86	1.000	706.86	

Figure 2.13: Window *1.5 Design Parameters*

For each member, the *Gross Area* of the section is listed. The values can be modified in the *Net Area* column, e.g. when there are holes in the section. The net area A_n is required for the design of members for tension according [1] Clause 5.2.

The *Reduction Factor* C_t is related to the plastic behavior of the section. It can be specified according to [1] Clause 5.2.5.

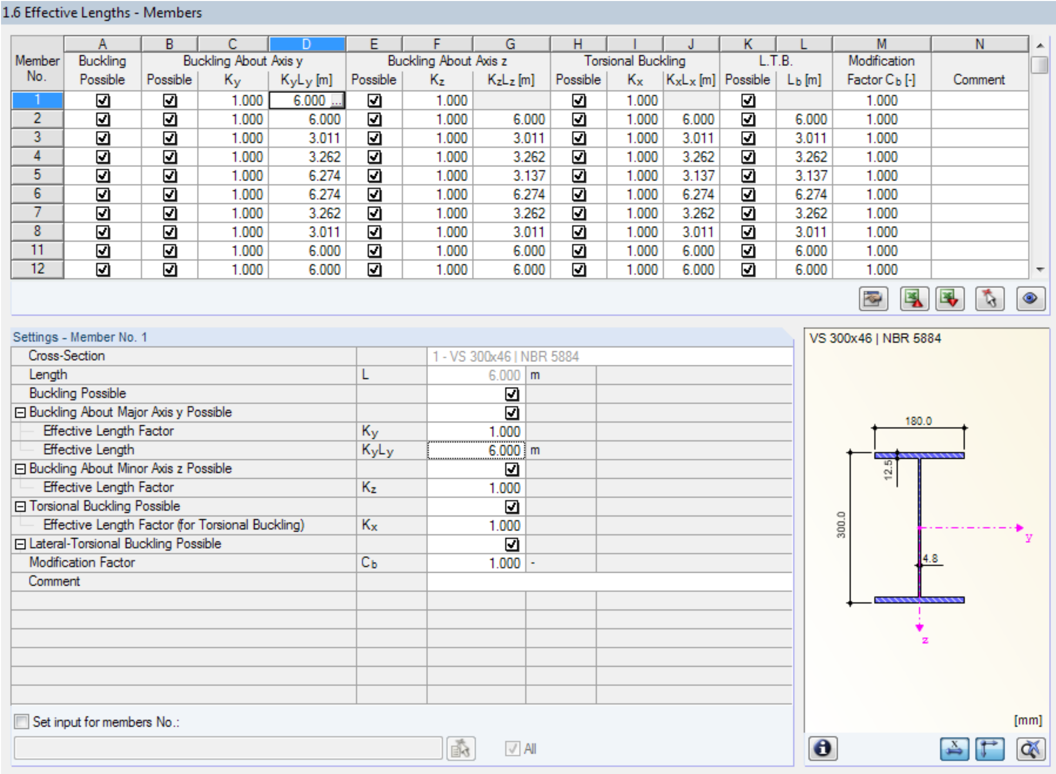
In the last column, the *Effective Area* A_e of the cross-section is shown for each member. The values are determined from the data of the two previous columns.

2.6 Effective Lengths - Members

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

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

Click to select a member graphically and show its row.


1.6 Effective Lengths - Members

Member No.	A Buckling Possible	B Buckling About Axis y Possible	C K_y	D K_yL_y [m]	E Buckling About Axis z Possible	F K_z	G K_zL_z [m]	H Torsional Buckling Possible	I K_x	J K_xL_x [m]	K L.T.B. Possible	L L_b [m]	M Modification Factor C_b [-]	N Comment
1	☑	☑	1.000	6.000	☑	1.000		☑	1.000		☑		1.000	
2	☑	☑	1.000	6.000	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	1.000	
3	☑	☑	1.000	3.011	☑	1.000	3.011	☑	1.000	3.011	☑	3.011	1.000	
4	☑	☑	1.000	3.262	☑	1.000	3.262	☑	1.000	3.262	☑	3.262	1.000	
5	☑	☑	1.000	6.274	☑	1.000	3.137	☑	1.000	3.137	☑	3.137	1.000	
6	☑	☑	1.000	6.274	☑	1.000	6.274	☑	1.000	6.274	☑	6.274	1.000	
7	☑	☑	1.000	3.262	☑	1.000	3.262	☑	1.000	3.262	☑	3.262	1.000	
8	☑	☑	1.000	3.011	☑	1.000	3.011	☑	1.000	3.011	☑	3.011	1.000	
11	☑	☑	1.000	6.000	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	1.000	
12	☑	☑	1.000	6.000	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	1.000	

Figure 2.14: Window *1.6 Effective lengths - Members*

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

You can enter the effective lengths manually in the table and in the *Settings* tree, or define them graphically in the work window by clicking the [...] button. The button is active when you place the cursor in the text box (see Figure 2.14).

The *Settings* tree includes the following parameters:

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

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

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

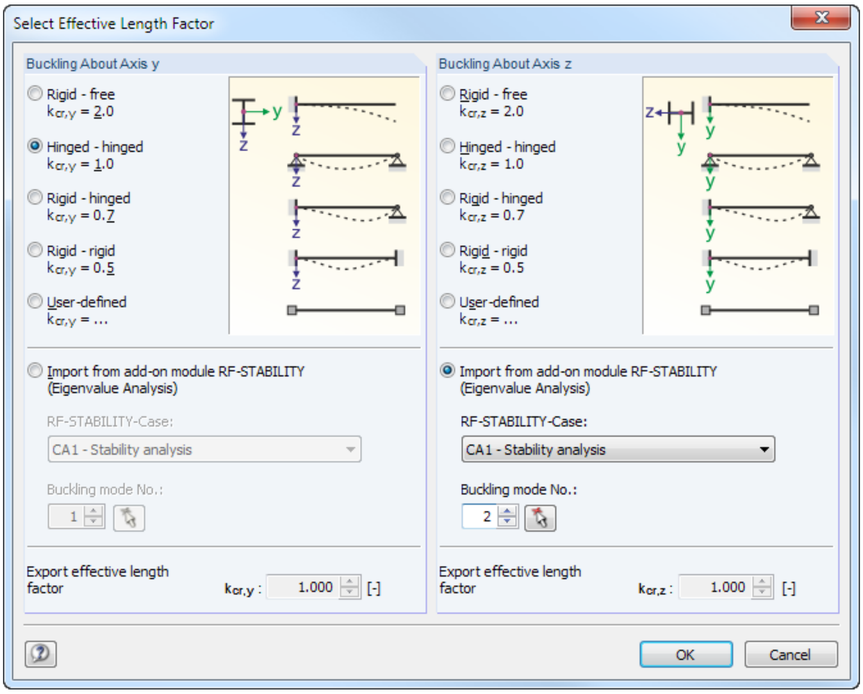


Figure 2.15: Dialog box *Select Effective Length Factor*

For each direction, you can select one of the four Euler buckling modes or apply a *User-defined* effective length factor k_{cr}. If an eigenvalue analysis has been performed in the RF-STABILITY or RSBUCK add-on module, you can import the *Buckling mode* in order to determine the factor.

Buckling Possible

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

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

Buckling About Axis y / Buckling about Axis z

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

You can check the position of the member axes in the cross-section graphic in the *1.3 Cross-Sections* Window (see Figure 2.8, page 9). To access the RFEM or RSTAB work window, click the [View Mode] button. There you can display the local member axes by using the shortcut menu of the member or the *Display* navigator (see Figure 2.16).

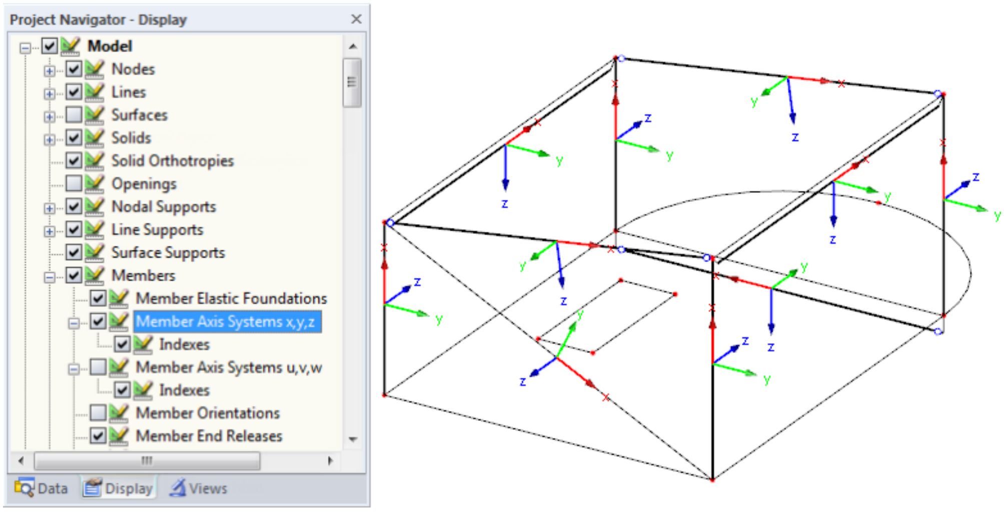


Figure 2.16: Displaying member axes in *Display* navigator of RFEM

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

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

To define the effective lengths graphically in the work window of RFEM or RSTAB, click [...]. This button becomes available when you place the cursor in a *KL* box (see Figure 2.14).

Torsional Buckling

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

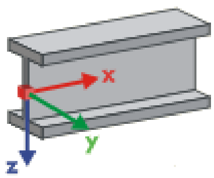


Figure 2.17: Member axes

L.T.B.

Column K controls whether a lateral-torsional buckling analysis is to be carried out. The lateral-torsional buckling lengths L_b can be defined in column L.

Modification Factor

In column M, the modification factor C_b for bending according to [1] Clauses 5.4.2.3 or 5.4.2.4 can be defined for each member.

Comment

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

Set Input for Members No.

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

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

2.7 Effective Lengths - Sets of Members

This window controls the effective lengths for sets of members. It is only displayed when you have selected one or more sets of members for design in the *1.1 General Data* Window (see Figure 2.2, page 5).

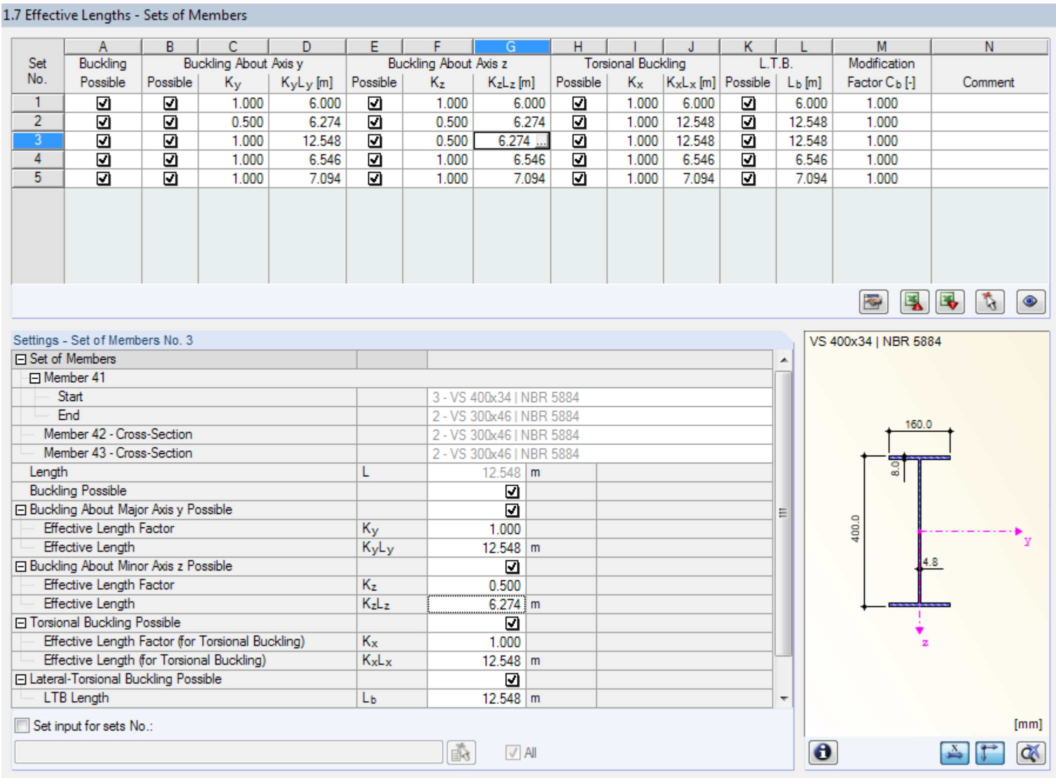
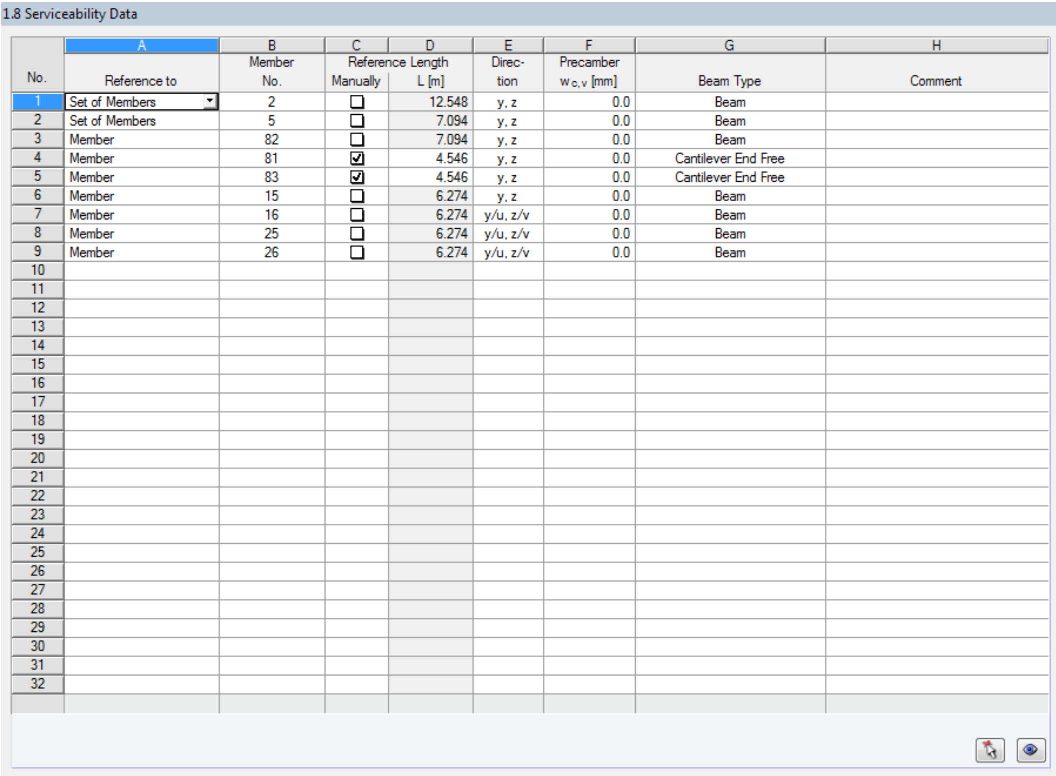


Figure 2.18: Window *1.7 Effective Lengths - Sets of Members*

The concept of the window is similar to the previous *1.6 Effective Lengths - Members* Window. Here you can define the effective lengths for buckling, torsional buckling and lateral-torsional buckling for sets of members, as described in Chapter 2.6.

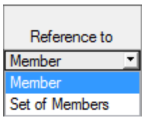
2.8 Serviceability Data

The last input window controls the settings for the serviceability limit state design of specific objects. It is available when you have selected one or more load cases or combinations in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 6).


1.8 Serviceability Data

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

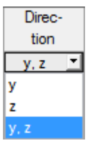
Figure 2.19: Window *1.8 Serviceability Data*



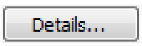
In Column A, you define whether the deformation refers to single members, lists of members, or sets of members.

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

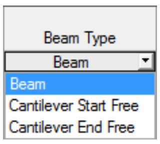
In column B, you can specify the numbers of the members or sets of members that are to be analyzed. The [...] button enables you to select the objects graphically in the work window. In column D, the *Reference Length* of each object is shown. The geometrical lengths of the members, lists or sets of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



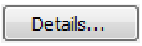
Column E controls the governing *Direction* for the deformation analysis. You can select the directions of the local member axes *y* and *z* (or *u* and *v* for unsymmetrical cross-sections).



You can consider a *Precamber* w_c in column F, if applicable. The reference to the axes is controlled by the specification in the *Details* dialog box (see Figure 3.1, page 19).



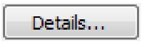
The *Beam Type* is important for the correct reference to the limit deformations. In column G, you can specify whether a beam or a cantilever is to be analyzed. For the latter, you can define which end has no support.



The *Details* dialog box controls whether the deformations are related to the undeformed system or the shifted ends of the members or sets of members (see Figure 3.1, page 19).

3 Calculation

3.1 Detail Settings



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

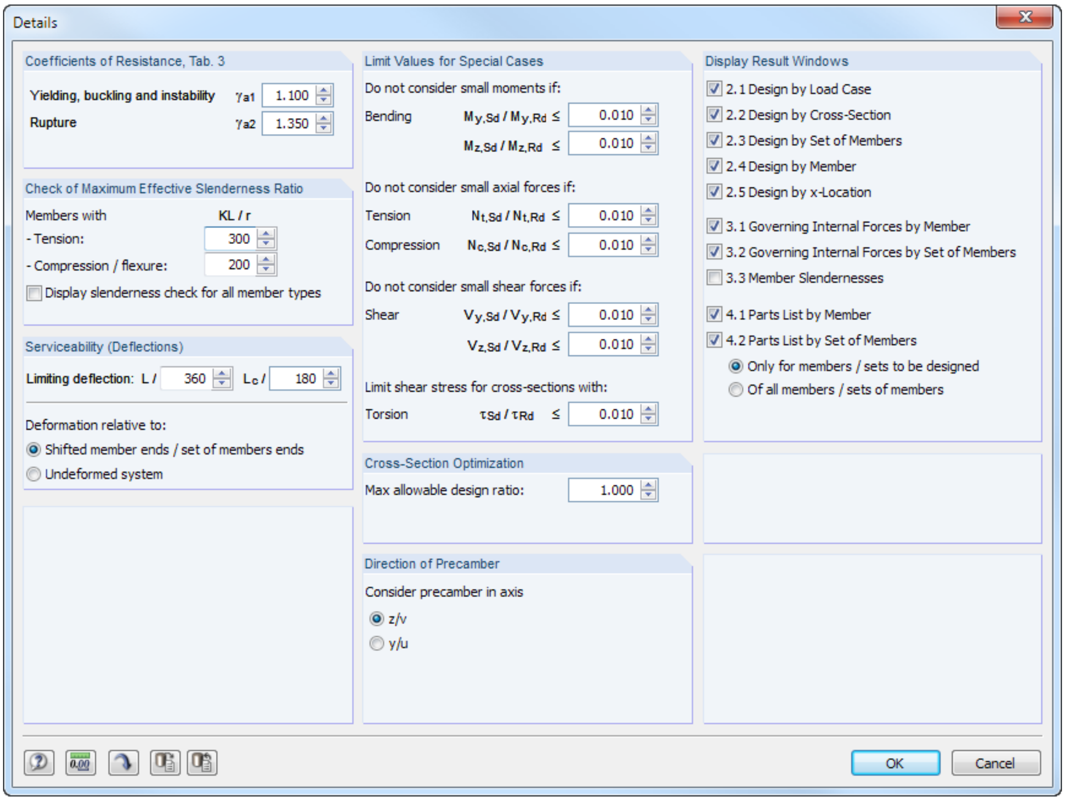


Figure 3.1: Dialog box *Details*

Coefficients of Resistance

The factors of the material resistance are preset according to [1] Table 3 regarding yielding, buckling, instability (γ_{a1}) and rupture (γ_{a2}). If required, those coefficients can be adjusted.

Check of Maximum Effective Slenderness Ratio

According to [1] Clause 5.2.8, the slenderness ratio *KL / r* preferably should not exceed 300 for tension members. For members with compression or flexure, the slenderness ratio should not exceed 200 (cf. [1] Clause 5.3.4). If required, the limit ratios can be adjusted.

By default, the slenderness ratios are not checked for specific member types, such as "Tension" and "Cable" members. It is possible, however, to activate the check for all member types.

The limit ratios are compared to the real member slendernesses in Window 3.3. That window is available after the calculation (see Chapter 4.8, page 28) when the corresponding option has been checked in the *Display Result Windows* section of the *Details* dialog box.

Serviceability (Deflections)

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

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

Limit Values for Special Cases

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

Those limit settings are not part of the Standard [1]. Changing the limits is in the responsibility of the user.

Cross-Section Optimization

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

Direction of Precamber

When you specify two directions in column E of the 1.8 *Serviceability Data* Window and apply a precamber in column F, you can determine for which direction the precamber is to be considered.

Display Result Windows

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

Window *3.3 Member Slendernesses* is deactivated by default.

3.2 Starting Calculation

Calculation

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

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

You can follow the process of the design in a separate dialog box.

4 Results

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

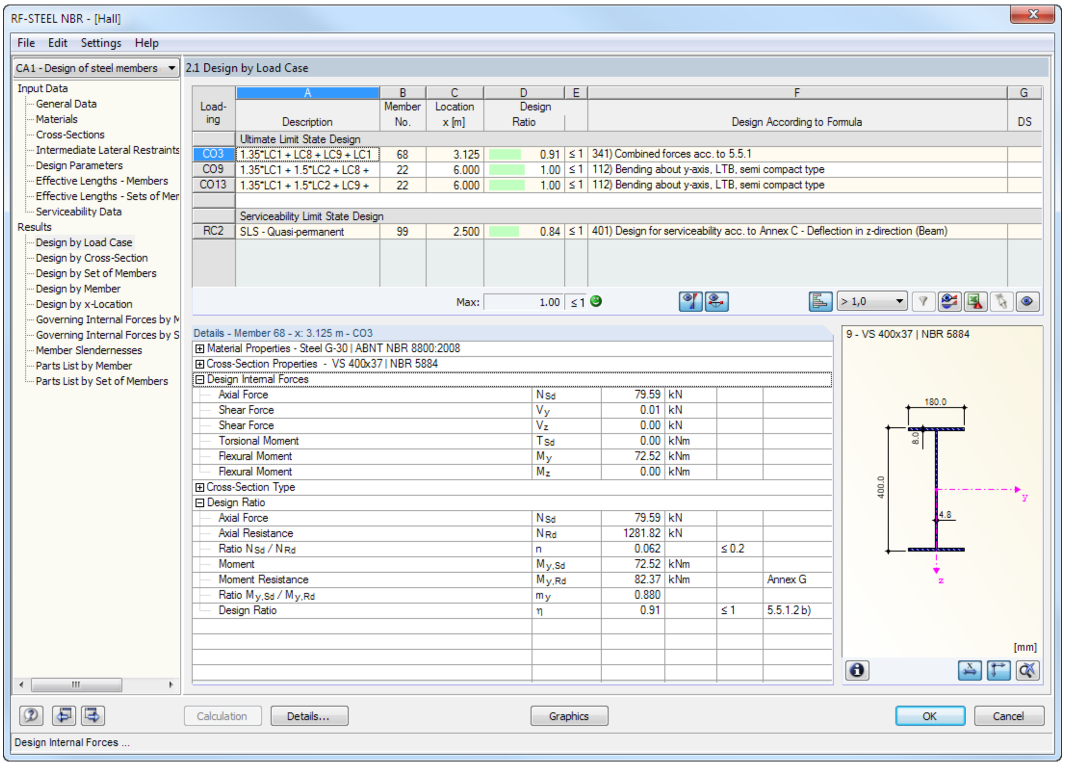
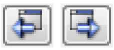


Figure 4.1: Result window with design results and details

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

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

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

Chapter 4 describes the different result windows one by one. The evaluation and checking of the results is described in Chapter 5 starting on page 31.

4.1 Design by Load Case

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

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

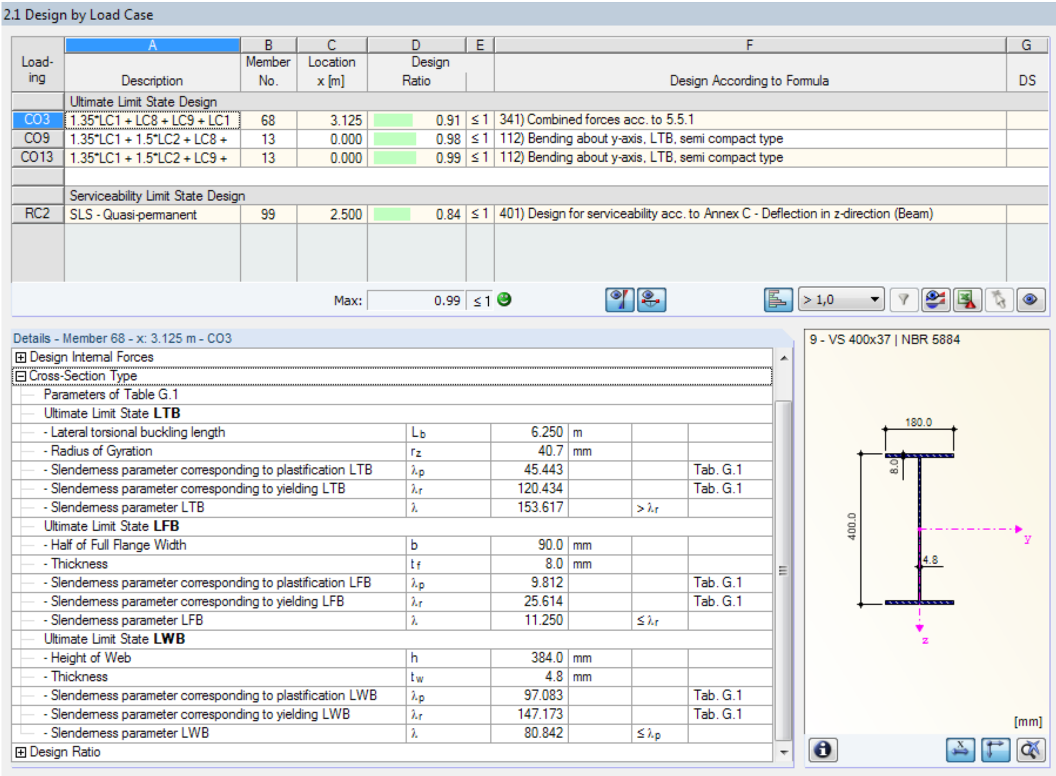


Figure 4.2: Window *2.1 Design by Load Case*

Description

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

Member No.

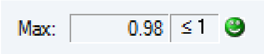
In this column, the number of each member is given that has the maximum design ratio of the respective loading.

Location x

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

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

Design Ratio



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

The lengths of colored bars represent the respective design ratios.

Design According to Formula

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

DS

The last column provides information on the respective design situations.

4.2 Design by Cross-Section

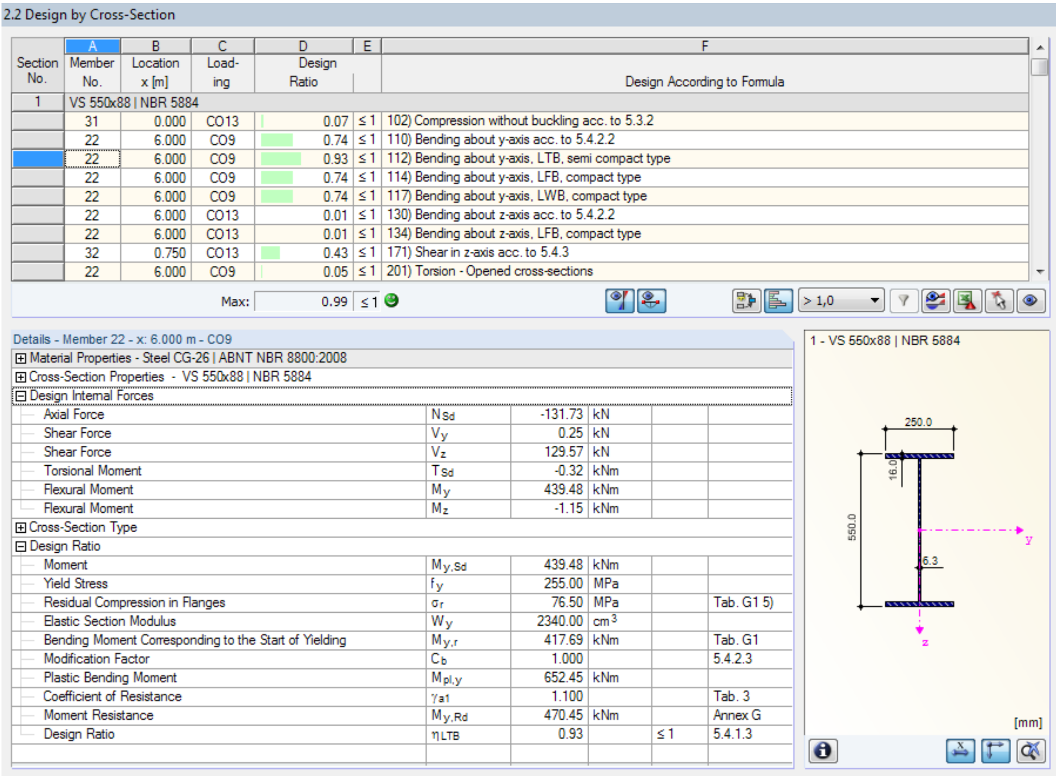


Figure 4.3: Window *2.2 Design by Cross-Section*

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

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

4.3 Design by Set of Members

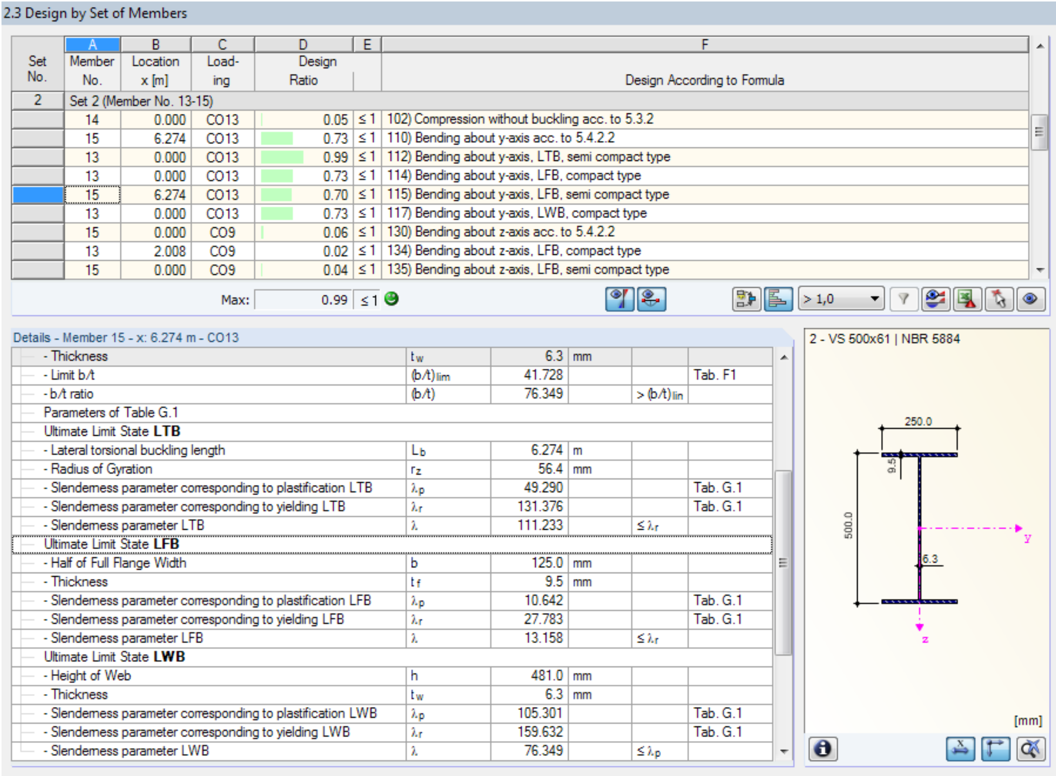


Figure 4.4: Window *2.3 Design by Set of Members*

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

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

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

4.4 Design by Member

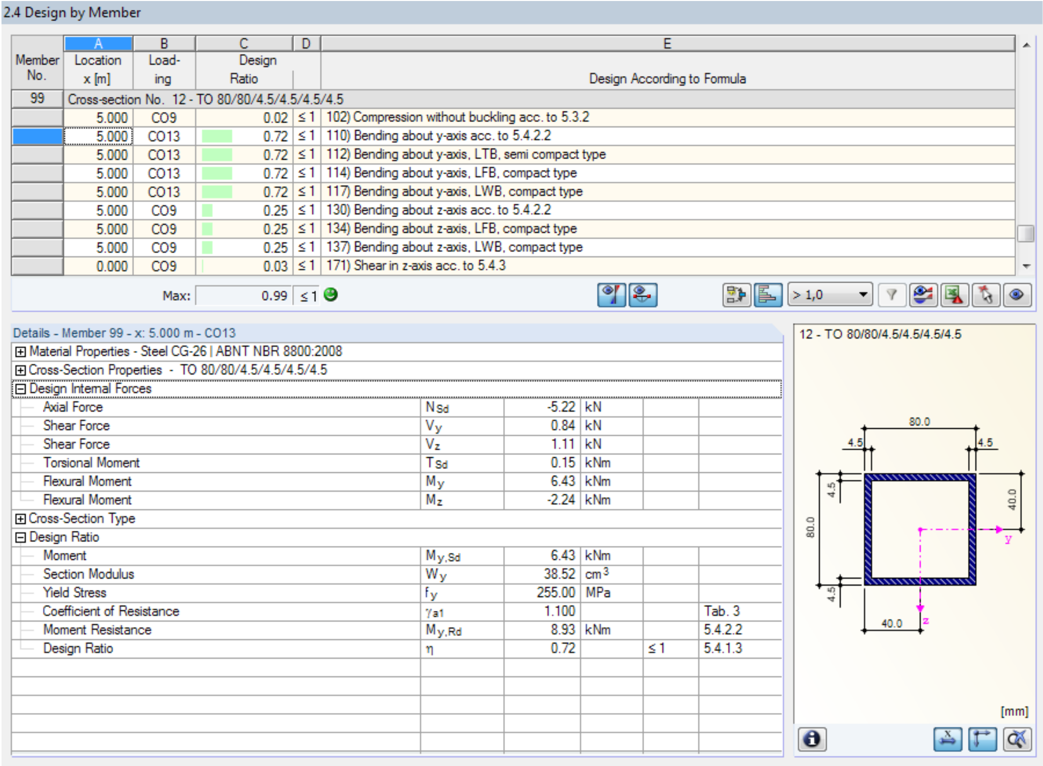


Figure 4.5: Window *2.4 Design by Member*

This result window lists the maximum ratios of the individual designs for each member. The columns are described Chapter 4.1 on page 22.

4.5 Design by x-Location

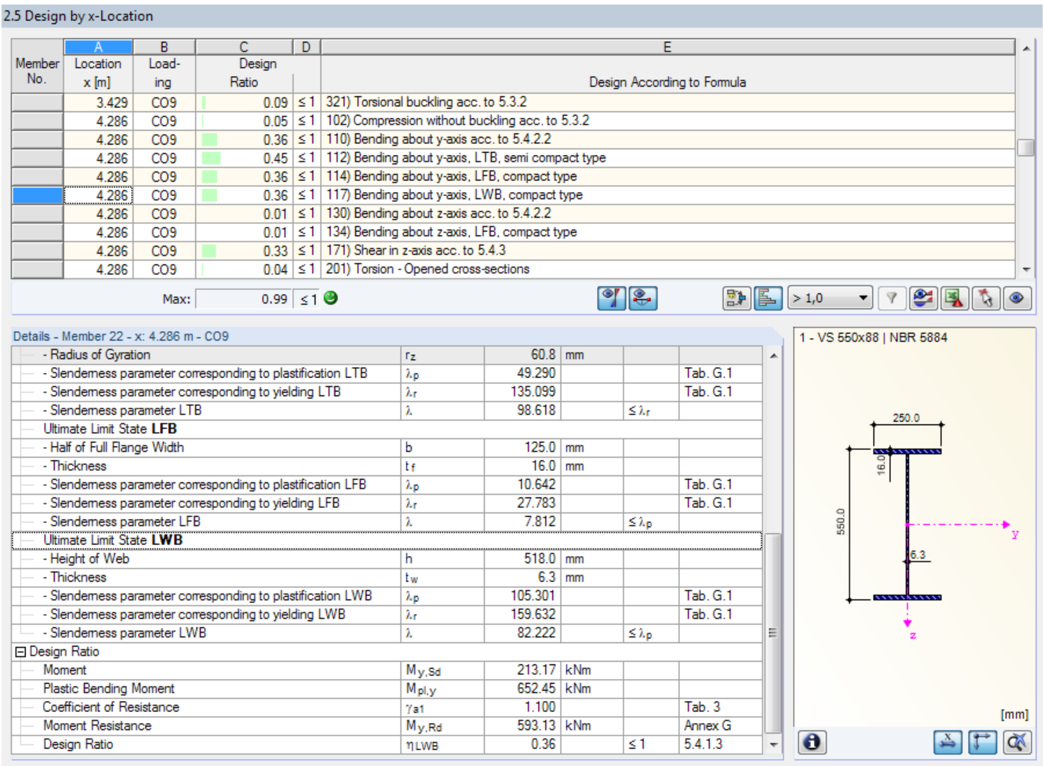
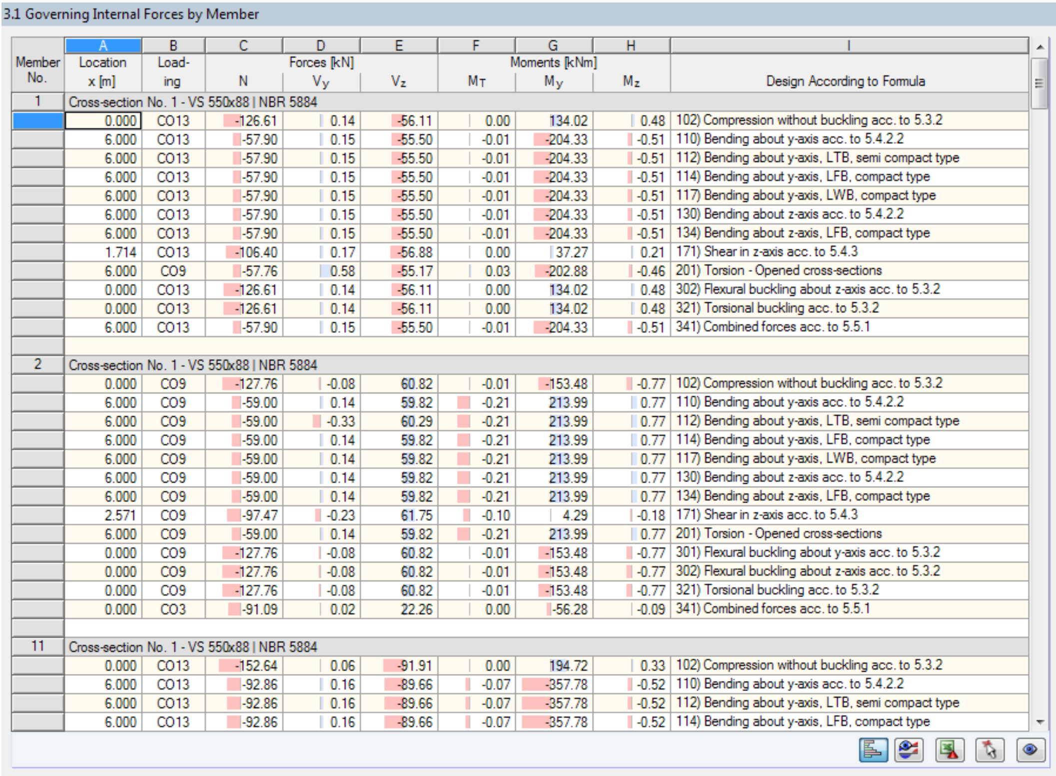


Figure 4.6: Window *2.5 Design by x-Location*

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

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

4.6 Governing Internal Forces by Member


3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Load-ing	C Forces [kN] N	D Vy	E Vz	F Moments [kNm] MT	G My	H Mz	I Design According to Formula
1	Cross-section No. 1 - VS 550x88 | NBR 5884								
	0.000	CO13	-126.61	0.14	-56.11	0.00	134.02	0.48	102) Compression without buckling acc. to 5.3.2
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	110) Bending about y-axis acc. to 5.4.2.2
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	112) Bending about y-axis, LTB, semi compact type
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	114) Bending about y-axis, LFB, compact type
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	117) Bending about y-axis, LWB, compact type
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	130) Bending about z-axis acc. to 5.4.2.2
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	134) Bending about z-axis, LFB, compact type
	1.714	CO13	-106.40	0.17	-56.88	0.00	37.27	0.21	171) Shear in z-axis acc. to 5.4.3
	6.000	CO9	-57.76	0.58	-55.17	0.03	-202.88	-0.46	201) Torsion - Opened cross-sections
	0.000	CO13	-126.61	0.14	-56.11	0.00	134.02	0.48	302) Flexural buckling about z-axis acc. to 5.3.2
	0.000	CO13	-126.61	0.14	-56.11	0.00	134.02	0.48	321) Torsional buckling acc. to 5.3.2
	6.000	CO13	-57.90	0.15	-55.50	-0.01	-204.33	-0.51	341) Combined forces acc. to 5.5.1
2	Cross-section No. 1 - VS 550x88 | NBR 5884								
	0.000	CO9	-127.76	-0.08	60.82	-0.01	-153.48	-0.77	102) Compression without buckling acc. to 5.3.2
	6.000	CO9	-59.00	0.14	59.82	-0.21	213.99	0.77	110) Bending about y-axis acc. to 5.4.2.2
	6.000	CO9	-59.00	-0.33	60.29	-0.21	213.99	0.77	112) Bending about y-axis, LTB, semi compact type
	6.000	CO9	-59.00	0.14	59.82	-0.21	213.99	0.77	114) Bending about y-axis, LFB, compact type
	6.000	CO9	-59.00	0.14	59.82	-0.21	213.99	0.77	117) Bending about y-axis, LWB, compact type
	6.000	CO9	-59.00	0.14	59.82	-0.21	213.99	0.77	130) Bending about z-axis acc. to 5.4.2.2
	6.000	CO9	-59.00	0.14	59.82	-0.21	213.99	0.77	134) Bending about z-axis, LFB, compact type
	2.571	CO9	-97.47	-0.23	61.75	-0.10	4.29	-0.18	171) Shear in z-axis acc. to 5.4.3
	6.000	CO9	-59.00	0.14	59.82	-0.21	213.99	0.77	201) Torsion - Opened cross-sections
	0.000	CO9	-127.76	-0.08	60.82	-0.01	-153.48	-0.77	301) Flexural buckling about y-axis acc. to 5.3.2
	0.000	CO9	-127.76	-0.08	60.82	-0.01	-153.48	-0.77	302) Flexural buckling about z-axis acc. to 5.3.2
	0.000	CO9	-127.76	-0.08	60.82	-0.01	-153.48	-0.77	321) Torsional buckling acc. to 5.3.2
	0.000	CO3	-91.09	0.02	22.26	0.00	-56.28	-0.09	341) Combined forces acc. to 5.5.1
11	Cross-section No. 1 - VS 550x88 | NBR 5884								
	0.000	CO13	-152.64	0.06	-91.91	0.00	194.72	0.33	102) Compression without buckling acc. to 5.3.2
	6.000	CO13	-92.86	0.16	-89.66	-0.07	-357.78	-0.52	110) Bending about y-axis acc. to 5.4.2.2
	6.000	CO13	-92.86	0.16	-89.66	-0.07	-357.78	-0.52	112) Bending about y-axis, LTB, semi compact type
	6.000	CO13	-92.86	0.16	-89.66	-0.07	-357.78	-0.52	114) Bending about y-axis, LFB, compact type

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

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

Location x

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

Loading

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

Forces / Moments

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

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [m]	B Loading	C Forces [kN] N	D V_y	E V_z	F Moments [kNm] M_T	G M_y	H M_z	I Design According to Formula
1	Set 1 (Member No. 51,52)								
	0.000	CO13	-136.26	-0.04	0.27	-0.02	-0.45	1.06	102) Compression without buckling acc. to 5.3.2
	0.000	CO9	-136.25	-0.70	2.19	-0.02	-8.36	-0.71	110) Bending about y-axis acc. to 5.4.2.2
	0.000	CO9	-136.25	-0.70	2.19	-0.02	-8.36	-0.71	112) Bending about y-axis, LTB, semi compact type
	0.000	CO9	-136.25	-0.70	2.19	-0.02	-8.36	-0.71	114) Bending about y-axis, LFB, compact type
	0.000	CO9	-136.25	-0.70	2.19	-0.02	-8.36	-0.71	117) Bending about y-axis, LWB, compact type
	3.000	CO9	-99.71	-0.10	1.63	-0.02	-1.73	1.39	130) Bending about z-axis acc. to 5.4.2.2
	3.000	CO9	-99.71	-0.10	1.63	-0.02	-1.73	1.39	134) Bending about z-axis, LFB, compact type
	3.000	CO9	-99.71	-0.69	2.22	-0.02	-1.73	1.39	171) Shear in z-axis acc. to 5.4.3
	0.000	CO9	-52.10	-0.10	0.60	-0.11	-1.95	1.08	201) Torsion - Opened cross-sections
	0.000	CO13	-136.26	-0.04	0.27	-0.02	-0.45	1.06	301) Flexural buckling about y-axis acc. to 5.3.2
	0.000	CO13	-136.26	-0.04	0.27	-0.02	-0.45	1.06	302) Flexural buckling about z-axis acc. to 5.3.2
	0.000	CO13	-136.26	-0.04	0.27	-0.02	-0.45	1.06	321) Torsional buckling acc. to 5.3.2
	0.000	CO3	-129.34	-0.14	1.90	-0.01	-6.93	0.07	341) Combined forces acc. to 5.5.1
2	Set 2 (Member No. 13-15)								
	0.000	CO13	-93.87	-0.95	68.85	-0.13	-134.68	-1.07	102) Compression without buckling acc. to 5.3.2
	6.274	CO13	-86.75	0.64	-1.96	-0.45	234.50	1.71	110) Bending about y-axis acc. to 5.4.2.2
	0.000	CO13	-96.86	-0.25	78.87	-0.37	-358.16	-0.45	112) Bending about y-axis, LTB, semi compact type
	0.000	CO13	-96.86	-0.25	78.87	-0.37	-358.16	-0.45	114) Bending about y-axis, LFB, compact type
	6.274	CO13	-86.75	0.64	-1.96	-0.45	234.50	1.71	115) Bending about y-axis, LFB, semi compact type
	0.000	CO13	-96.86	-0.25	78.87	-0.37	-358.16	-0.45	117) Bending about y-axis, LWB, compact type
	0.000	CO9	-88.67	0.05	54.47	0.20	67.90	2.64	130) Bending about z-axis acc. to 5.4.2.2
	2.008	CO9	-94.27	-0.87	72.46	-0.16	-204.52	-1.65	134) Bending about z-axis, LFB, compact type
	0.000	CO9	-88.67	0.05	54.47	0.20	67.90	2.64	135) Bending about z-axis, LFB, semi compact type
	0.000	CO13	-96.86	-0.25	78.87	-0.37	-358.16	-0.45	171) Shear in z-axis acc. to 5.4.3
	3.262	CO9	-92.91	-1.03	54.72	0.17	69.10	2.41	181) Shear in y-axis acc. to 5.4.3
	6.274	CO9	-87.39	0.64	-2.12	-0.45	234.43	1.74	201) Torsion - Opened cross-sections
	0.000	CO13	-93.87	-0.95	68.85	-0.13	-134.68	-1.07	301) Flexural buckling about y-axis acc. to 5.3.2
	0.000	CO13	-93.87	-0.95	68.85	-0.13	-134.68	-1.07	302) Flexural buckling about z-axis acc. to 5.3.2
	0.000	CO13	-93.87	-0.95	68.85	-0.13	-134.68	-1.07	321) Torsional buckling acc. to 5.3.2
	0.000	RC2	0.00	0.00	0.00	0.00	0.00	0.00	400) Design for serviceability - Negligible deflections
	3.137	RC2	0.00	0.00	0.00	0.00	0.00	0.00	401) Design for serviceability acc. to Annex C - Deflection in z-c

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

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

4.8 Members Slendernesses

Details...

Window 3.3 is shown when you have selected the respective check box in the *Details* dialog box (see Figure 3.1, page 19).

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C k_y [-]	D Major Axis y i_y [mm]	E λ_y [-]	F k_z [-]	G Minor Axis z i_z [mm]	H λ_z [-]	I
1	Compression / Flexure	6.000	1.000	239.0	25.099	1.000	60.8	98.618	
2	Compression / Flexure	6.000	1.000	239.0	25.099	1.000	60.8	98.618	
11	Compression / Flexure	6.000	1.000	239.0	25.099	1.000	60.8	98.618	
16	Compression / Flexure	6.274	1.000	210.3	29.830	1.000	56.4	111.236	
17	Compression / Flexure	3.262	1.000	210.3	15.512	1.000	56.4	57.843	
18	Compression / Flexure	3.011	1.000	210.3	14.318	1.000	56.4	53.390	
21	Compression / Flexure	6.000	1.000	239.0	25.099	1.000	60.8	98.618	
22	Compression / Flexure	6.000	1.000	239.0	25.099	1.000	60.8	98.618	
31	Compression / Flexure	3.000	1.000	239.0	12.550	1.000	60.8	49.309	
32	Compression / Flexure	3.000	1.000	239.0	12.550	1.000	60.8	49.309	
39	Compression / Flexure	3.000	1.000	239.0	12.550	1.000	60.8	49.309	
66	Compression / Flexure	6.250	1.000	168.3	37.144	1.000	40.7	153.617	
67	Compression / Flexure	6.250	1.000	168.3	37.144	1.000	40.7	153.617	
68	Compression / Flexure	6.250	1.000	168.3	37.144	1.000	40.7	153.617	
69	Compression / Flexure	6.250	1.000	168.3	37.144	1.000	40.7	153.617	
81	Compression / Flexure	6.546	1.000	85.6	76.481	1.000	49.1	133.250	
82	Compression / Flexure	7.094	1.000	85.6	82.883	1.000	49.1	144.405	
83	Compression / Flexure	6.546	1.000	85.6	76.481	1.000	49.1	133.250	
99	Compression / Flexure	5.000	1.000	30.9	161.930	1.000	30.9	161.930	
100	Compression / Flexure	5.000	1.000	30.9	161.930	1.000	30.9	161.930	

Members with compression / flexure:

Max K_yL / r_y:	161.930	≤ 200
Max K_zL / r_z:	161.930	≤ 200

Figure 4.9: Window *3.3 Member Slendernesses*

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

Details...

Below the list, you find a comparison of the most unfavorable values with the limit values that have been defined in the *Details* dialog box (see Figure 3.1, page 19).

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

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

4.9 Parts List by Member

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

4.1 Parts List by Member

	A	B	C	D	E	F	G	H	I
Part No.	Cross-Section Description	Number of Members	Length [m]	Total Length [m]	Surface Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1	1 - VS 550x88 | NBR 5884	5	6.00	30.00	62.62	0.34	88.39	530.35	2.652
2	2 - VS 500x61 | NBR 5884	1	6.27	6.27	12.47	0.05	61.07	383.17	0.383
3	2 - VS 500x61 | NBR 5884	1	3.26	3.26	6.48	0.03	61.07	199.25	0.199
4	2 - VS 500x61 | NBR 5884 ... 3 - VS 550x75 |	1	3.01	3.01	6.14	0.03	68.06	204.95	0.205
5	1 - VS 550x88 | NBR 5884	3	3.00	9.00	18.79	0.10	88.39	265.17	0.796
6	9 - VS 400x37 | NBR 5884	4	6.25	25.00	37.76	0.12	36.90	230.59	0.922
7	6 - CS 200 x 41 | NBR 5884	2	6.55	13.09	15.50	0.07	41.21	269.78	0.540
8	6 - CS 200 x 41 | NBR 5884	1	7.09	7.09	8.40	0.04	41.21	292.36	0.292
9	12 - TO 80/80/4.5/4.5/4.5/4.5	2	5.00	10.00	3.20	0.01	10.67	53.34	0.107
10	13 - RD 24	2	7.81	15.62	1.18	0.01	3.55	27.71	0.055
Sum		22		122.35	172.54	0.78			6.151

Figure 4.10: Window *4.1 Parts List by Member*

Details...

By default, this list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box (see Figure 3.1, page 19).

Part No.

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

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column shows the respective length of an individual member.

Total Length

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

Surface Area

For each item, the program gives the surface area relative to the total length. This area is determined from the *Surface Area* of the cross-sections. It can be checked in Windows 1.3 and 2.1 to 2.5 in the cross-section properties (see Figure 2.10, page 11).

Volume

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

Unit Weight

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

Weight

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

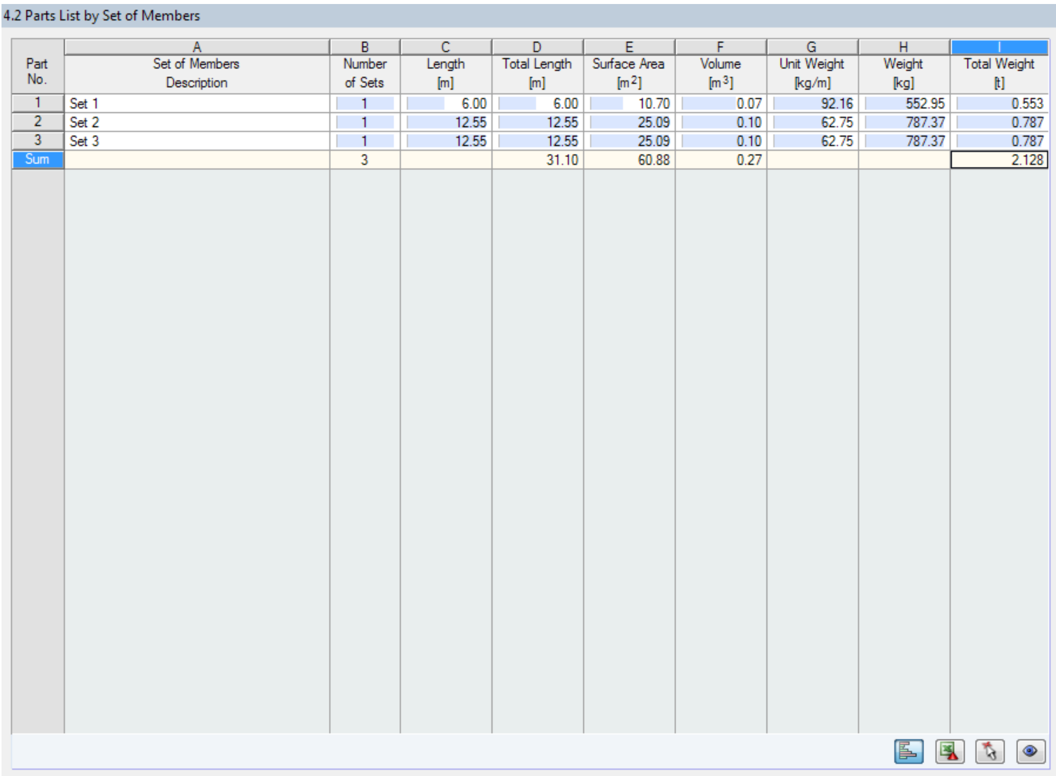
Total Weight

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

Sum

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

4.10 Parts List by Set of Members



4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Set 1	1	6.00	6.00	10.70	0.07	92.16	552.95	0.553
2	Set 2	1	12.55	12.55	25.09	0.10	62.75	787.37	0.787
3	Set 3	1	12.55	12.55	25.09	0.10	62.75	787.37	0.787
Sum		3		31.10	60.88	0.27			2.128

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

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

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

5 Results Evaluation

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

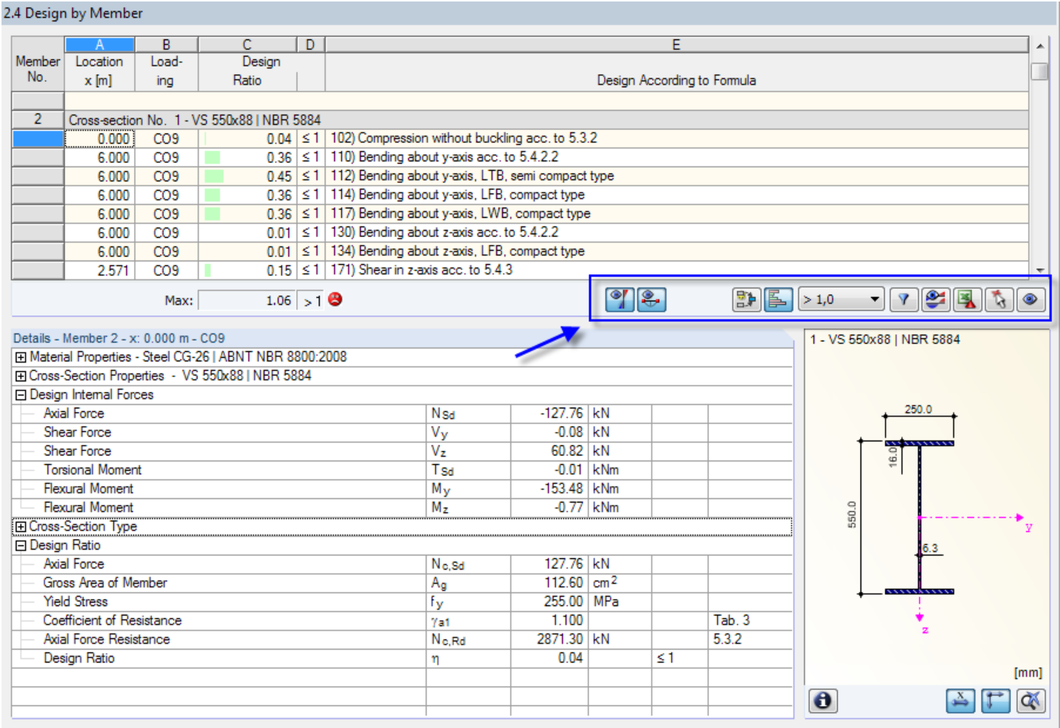


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Ultimate Limit State Design	Shows or hides the results of the ULS design
	Serviceability Limit State Design	Shows or hides the results of the SLS design
	Result Combination	Creates a new result combination from the governing load cases and load combinations
	Color Bars	Shows or hides the colored relation scales in the tables
> 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 limit)
	Result Diagrams	Opens the *Result Diagram on Member* Window → Chapter 5.2, page 35
	Excel Export	Exports the table to MS Excel → Chapter 7.4.3, page 44
	Member Selection	Option to select a member graphically for tabular results
	View Mode	Jumps to the RFEM/RSTAB work window to change the view

Table 5.1: Buttons in Windows 2.1 to 2.5

5.1 Results in RFEM/RSTAB Model

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

Background graphic and view mode

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

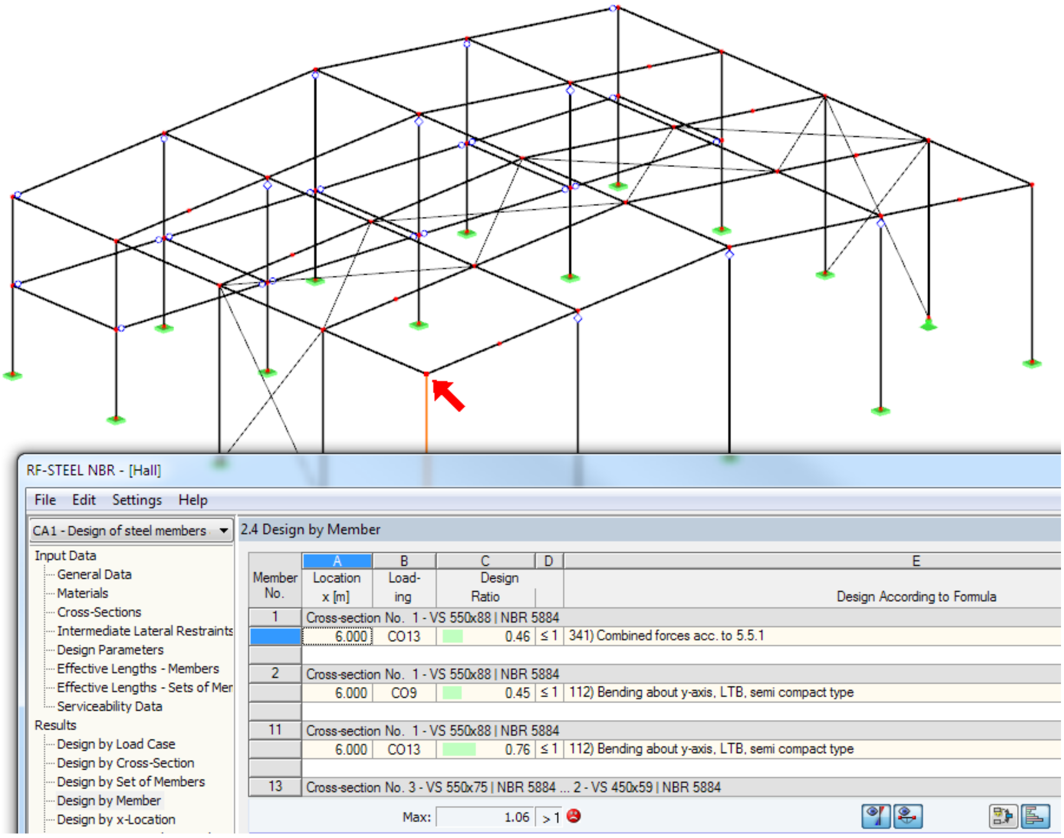
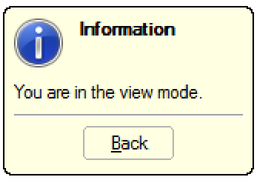


Figure 5.2: Indication of member and relevant *Location x* in RFEM model



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

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

RFEM/RSTAB work window

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

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

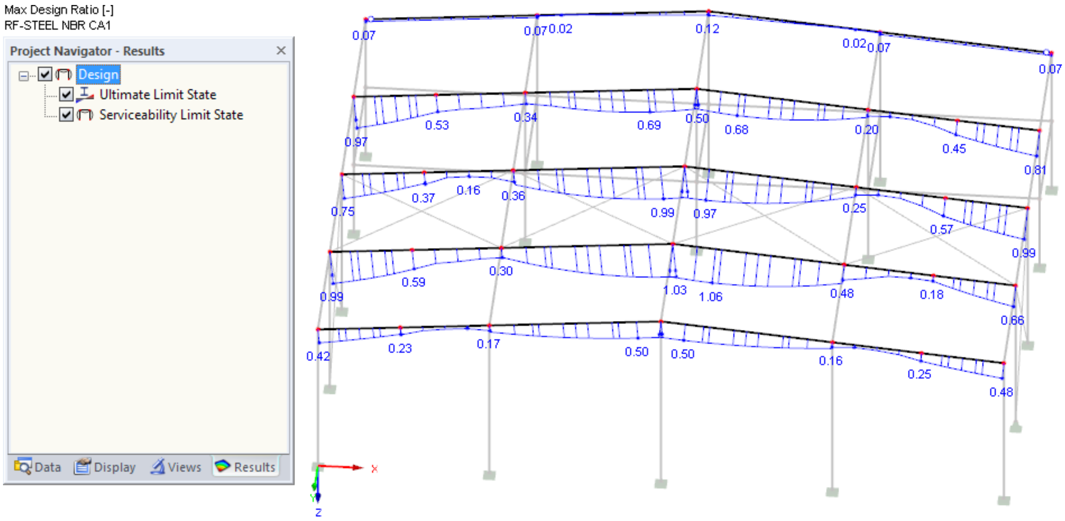
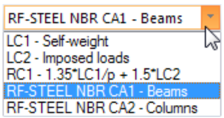


Figure 5.3: *Results* navigator for *Ultimate Limit State* and *Serviceability Limit State* designs

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

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



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

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

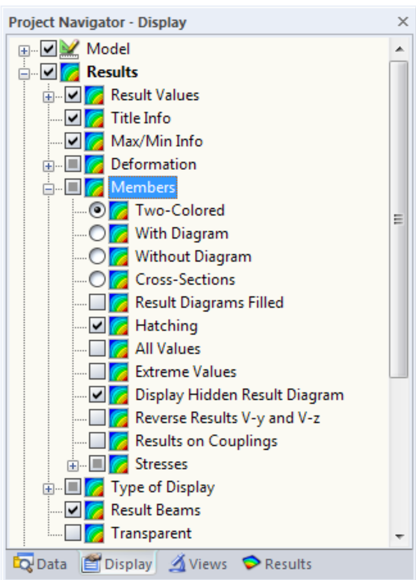


Figure 5.4: *Display* navigator: *Results* → *Members*

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

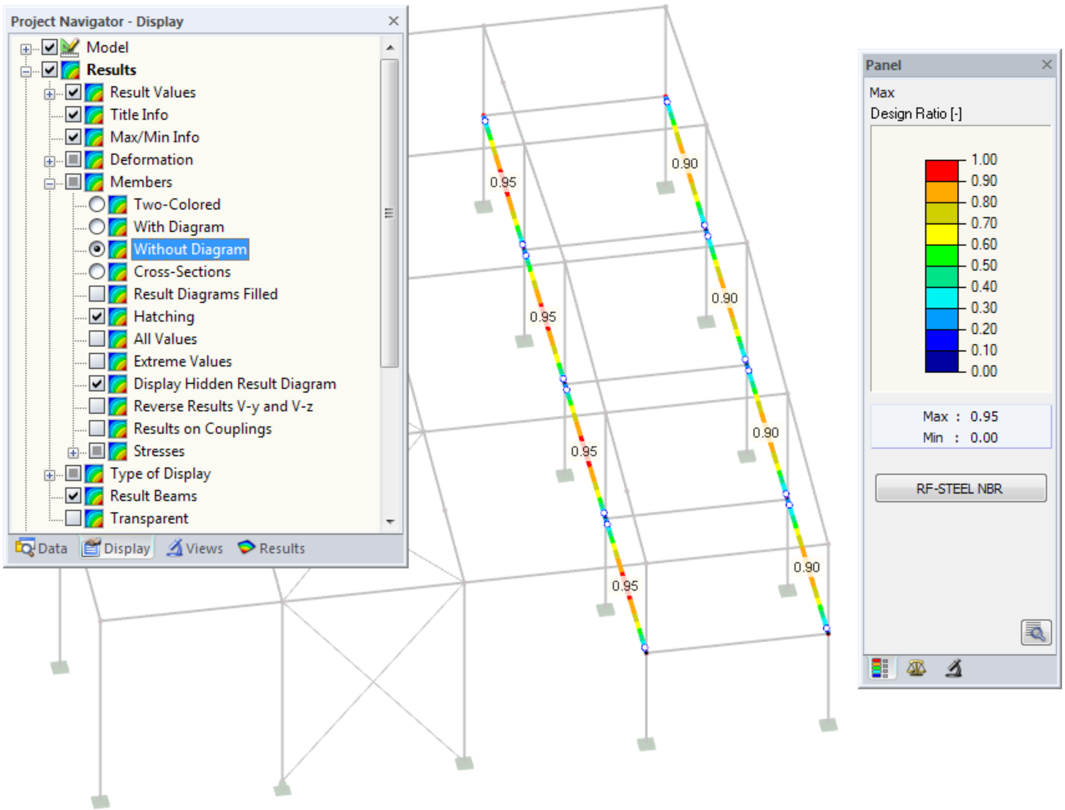


Figure 5.5: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 38).

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

5.2 Result Diagram

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

Select the member (or set of members) in the RF-/STEEL NBR result window by clicking in the relevant table row. Then click the  button to open the *Result Diagram on Member* dialog box. This button is located below the upper table (see Figure 5.1, page 31).

In the work window of RFEM or RSTAB, the result diagram can be accessed from the menu

Results → Result Diagrams for Selected Members

or via the toolbar button shown on the left.

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

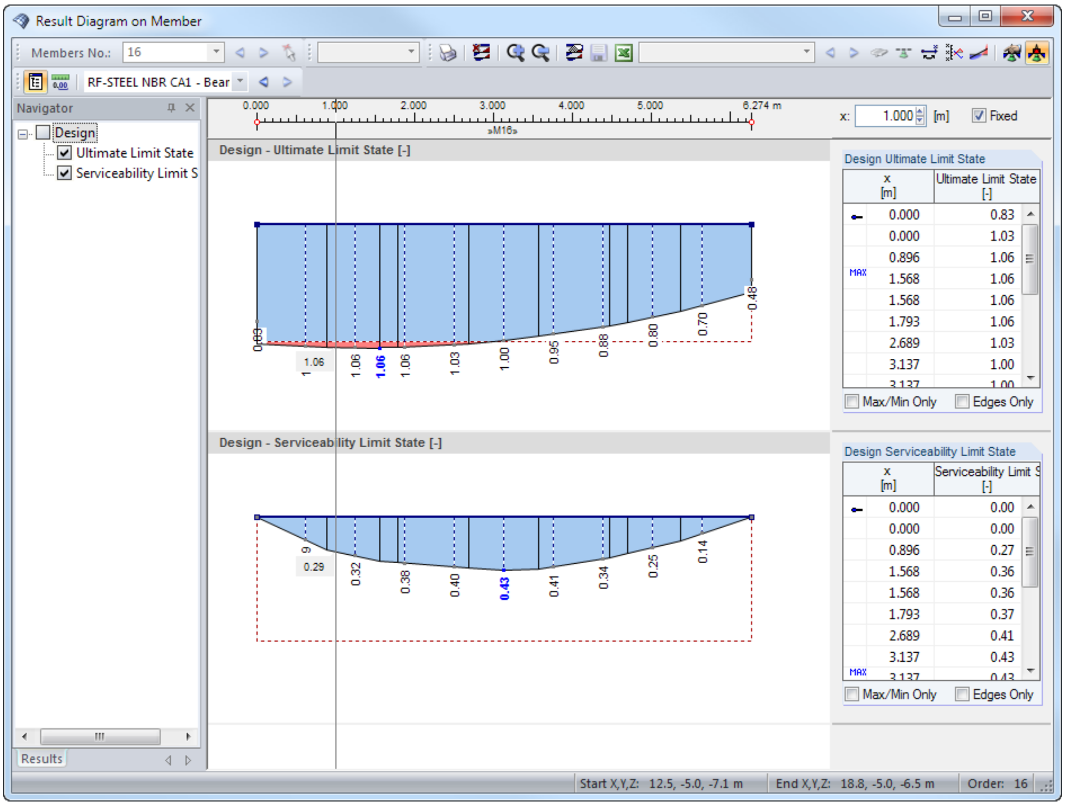
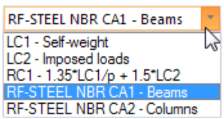


Figure 5.6: Dialog box *Result Diagram on Member*

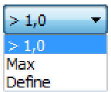


You can switch the ULS and SLS results on or off in the *Results* navigator.

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

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

5.3 Filter for Results



The RF-/STEEL NBR result windows allow you to sort the results by various criteria. In addition, you can use the filter options for the tables (see Figure 5.1, page 31) to reduce the numerical output according to specific ratios. This function is described in the *Knowledge Base* at our Web site: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000733

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

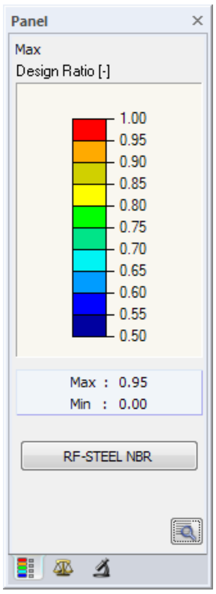
You can also use the *Visibility* options for RF-/STEEL NBR to filter the members and evaluate them (see RFEM manual, Chapter 9.9.1 or RSTAB manual, Chapter 9.7.1).

Filtering design ratios

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

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

or use the toolbar button shown on the left.



The panel is described in the RFEM/RSTAB manual, Chapter 3.4.6. The filter settings for the results can be defined in the first tab (Color scale). As this tab is not available for the two-colored results display, you have to set the display option *Colored With/Without Diagram* or *Cross-Sections* in the *Display* navigator (see Figure 5.4, page 33).

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

Filtering members

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

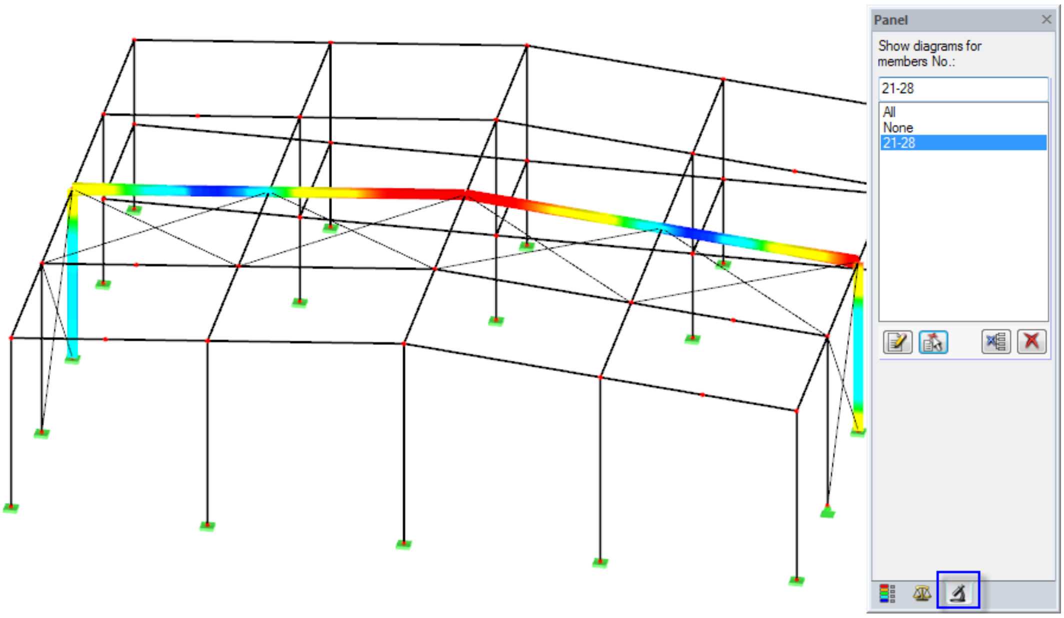


Figure 5.7: Filtering design ratios of one frame

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

6 Printout

6.1 Printout Report

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

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

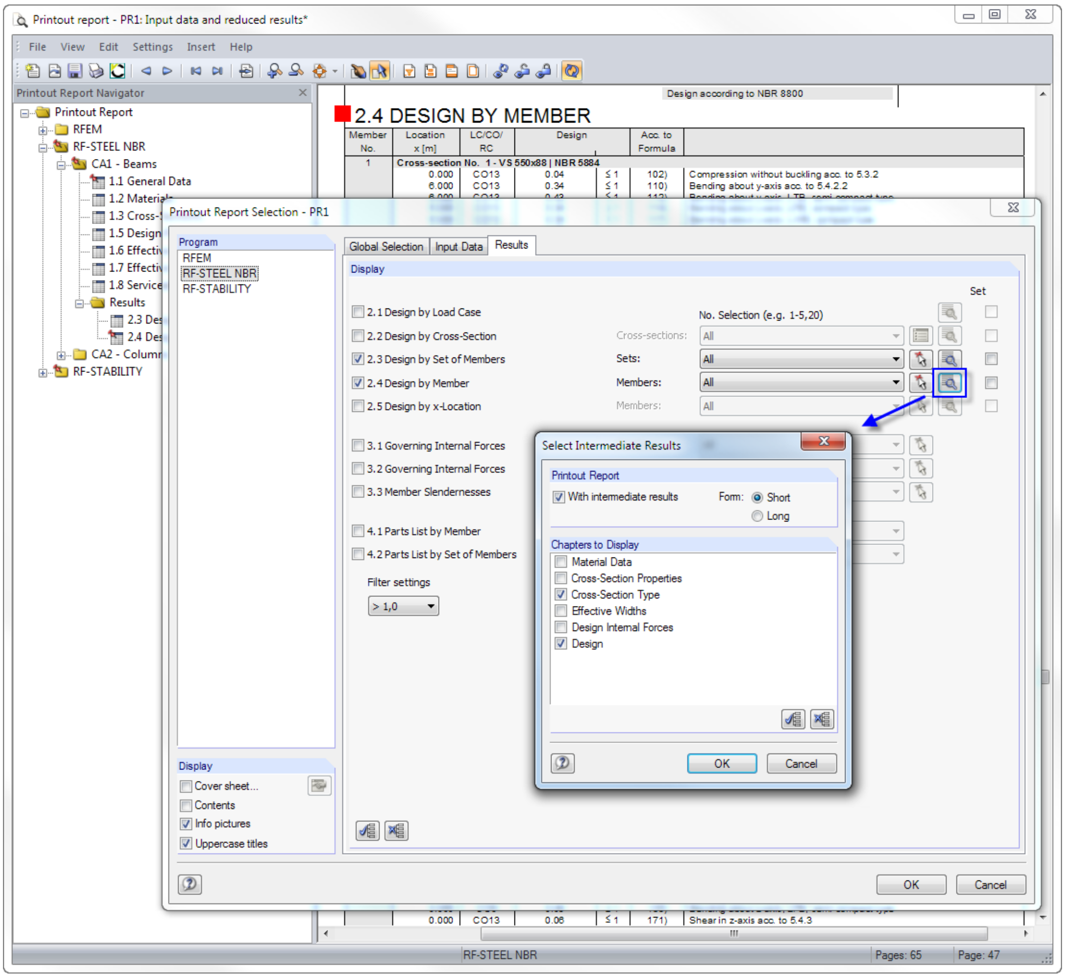


Figure 6.1: Selecting designs and intermediate results in printout report

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

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

6.2 Graphic Printout

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

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

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

File → Print Graphic

or use the toolbar button shown on the left.

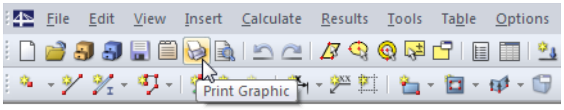


Figure 6.2: [Print Graphic] button in RFEM toolbar

The *Graphic Printout* dialog box appears.

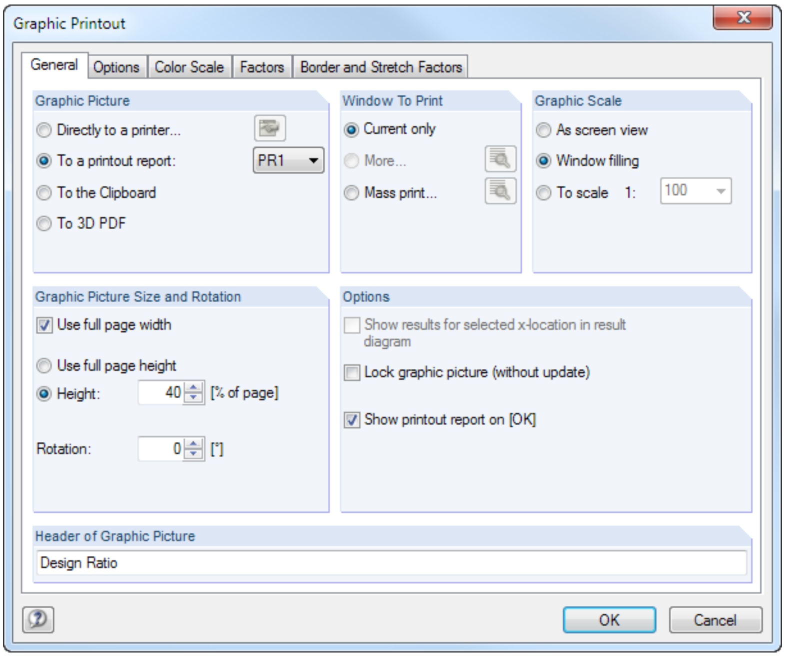
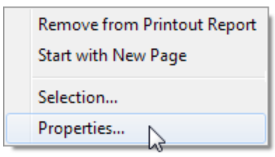


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

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



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

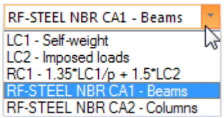
7 General Functions

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

7.1 Design Cases

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

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



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

Create design case

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

File → **New Case**.

The following dialog box appears.

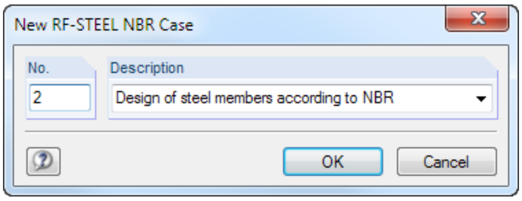


Figure 7.1: Dialog box *New RF-STEEL NBR Case*

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

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

Rename design case

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

File → **Rename Case**.

The following dialog box appears.

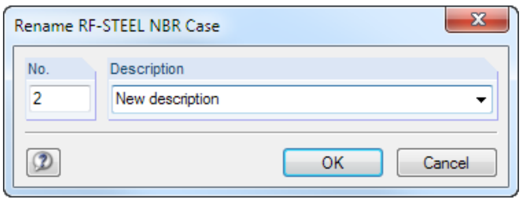


Figure 7.2: Dialog box *Rename RF-STEEL NBR Case*

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

Copy design case

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

File → **Copy Case**.

The following dialog box appears.

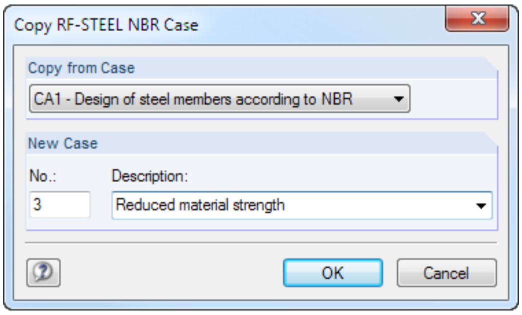


Figure 7.3: Dialog box *Copy RF-STEEL NBR Case*

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

Delete design case

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

File → **Delete Case**.

The following dialog box appears.

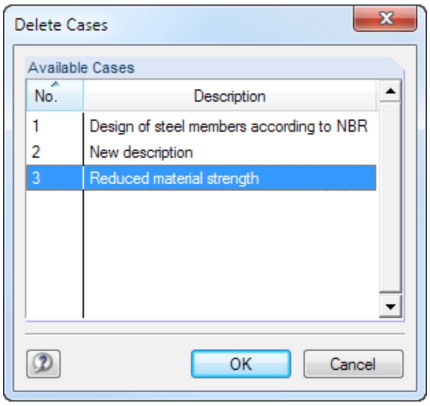
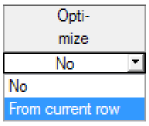


Figure 7.4: Dialog box *Delete Case*

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

7.2 Cross-Section Optimization



The design module offers you the option to optimize overstressed or little utilized cross-sections. Open the drop-down list in column D resp. E in Window *1.3 Cross-Sections* (see Figure 2.8, page 9) and select the optimization option *From current row*.

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

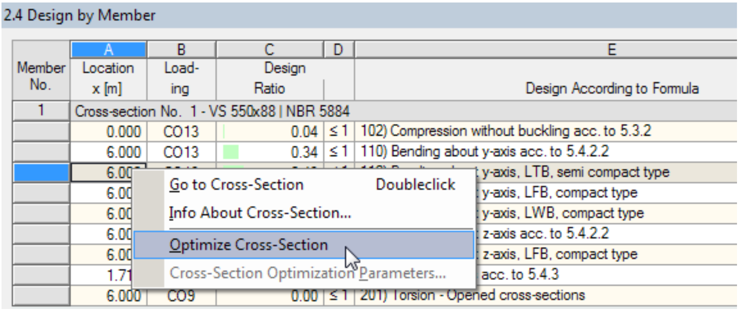


Figure 7.5: Shortcut menu to *Optimize Cross-Section*

During the optimization, the module determines the section that fulfills the analysis requirements in the "optimal" way, i.e. comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.1, page 19). The required cross-sectional properties are calculated with the internal forces of RFEM or RSTAB. If a different cross-section proves to be more favorable, it will be used for the design. In this case, the graphic in Window 1.3 shows two cross-sections – the original section from RFEM or RSTAB and the optimized one (see Figure 7.7).

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

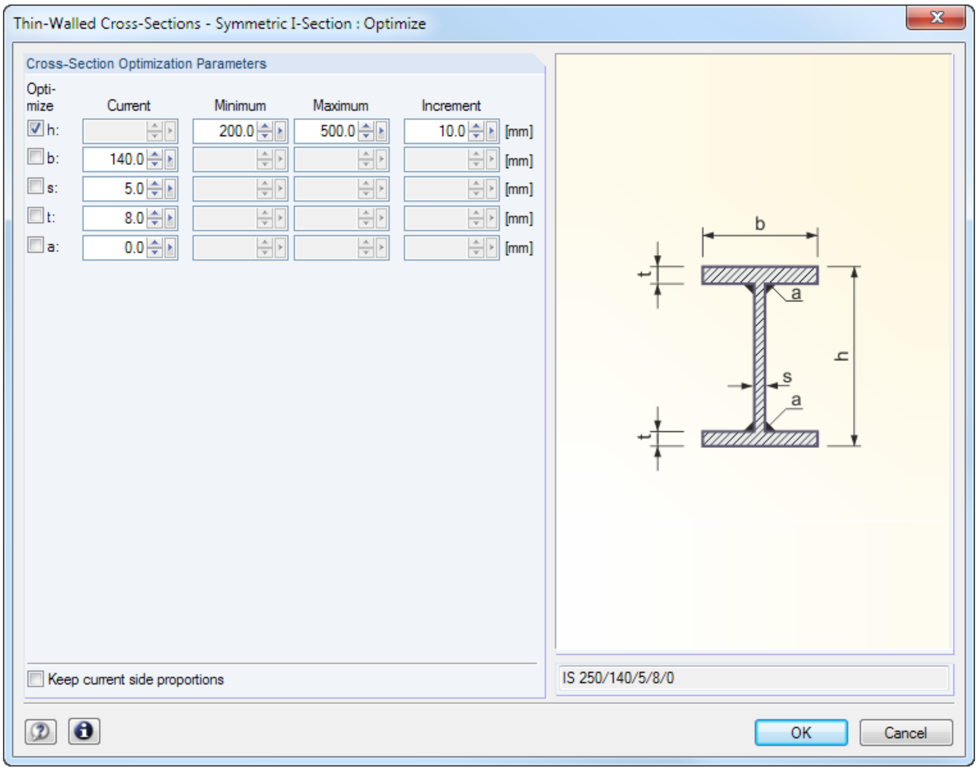


Figure 7.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section: Optimize*

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

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

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

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

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

Edit → Export All Cross-Sections to RFEM.

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

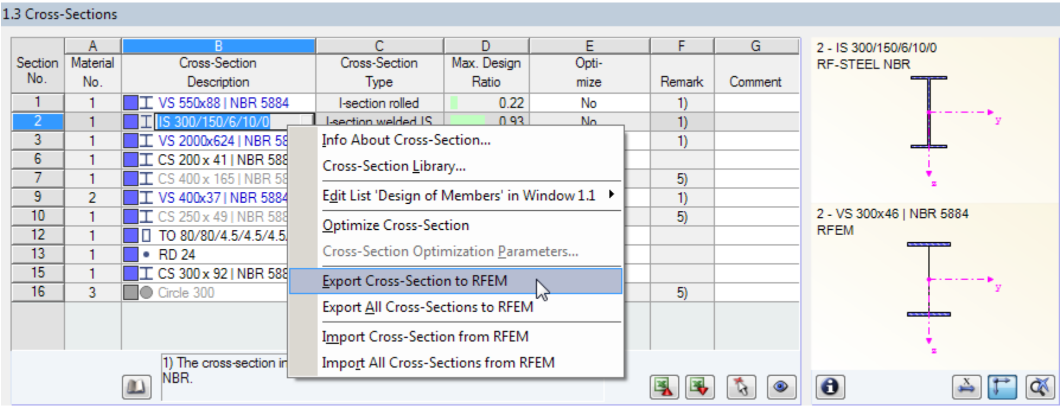


Figure 7.7: Shortcut menu in Window *1.3 Cross-Sections*

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

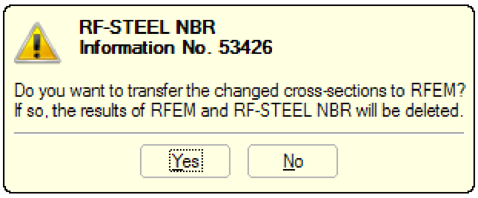


Figure 7.8: Confirmation when exporting cross-sections

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

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

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

7.3 Units and Decimal Places

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

Settings → Units and Decimal Places.

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

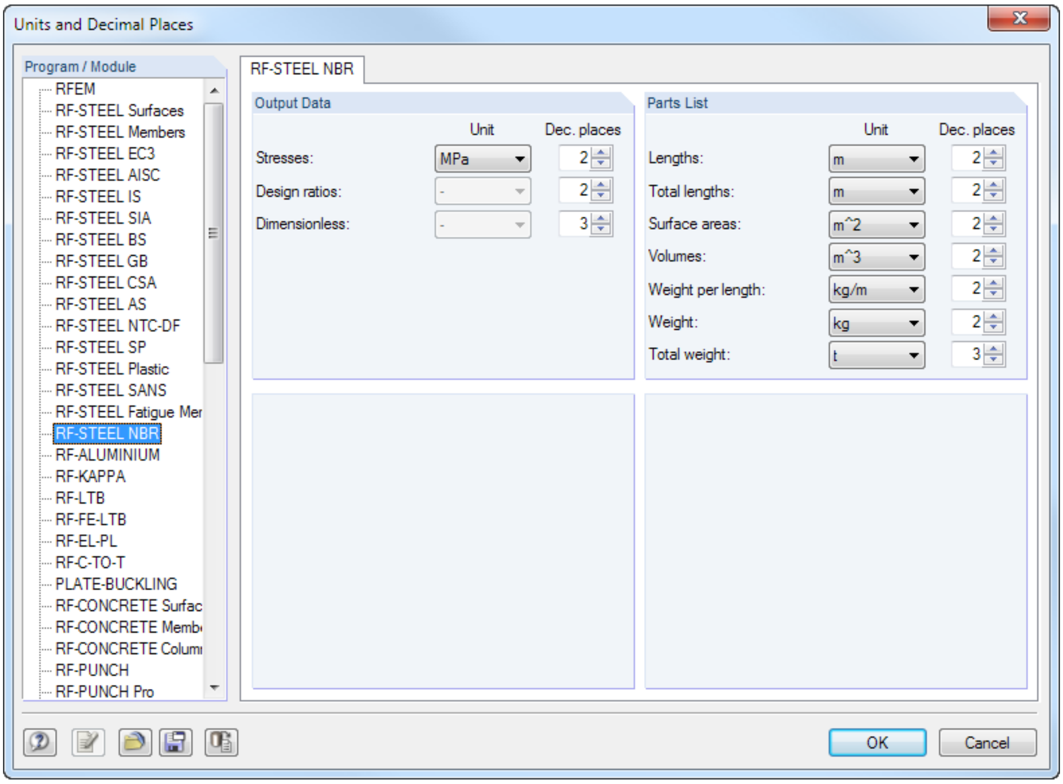


Figure 7.9: Dialog box *Units and Decimal Places*

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

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

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

Edit → Export All Materials to RFEM/RSTAB.

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

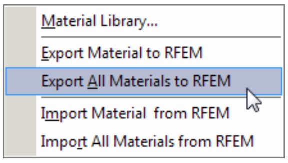
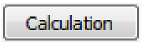


Figure 7.10: Shortcut menu of Window *1.2 Materials*



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

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

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

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

Edit → Export All Effective Lengths to RFEM/RSTAB.

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

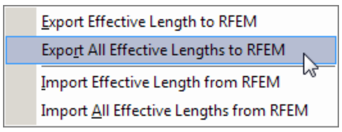


Figure 7.11: Shortcut menu of Window *1.5 Effective Lengths - Members*

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

7.4.3 Exporting Results

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

Clipboard

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

Printout Report

You can print the data of RF-/STEEL NBR to the printout report (see Chapter 6.1, page 37). To export the tables and graphics, then select the printout report menu

File → Export to RTF.

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

Excel

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

File → Export Tables.

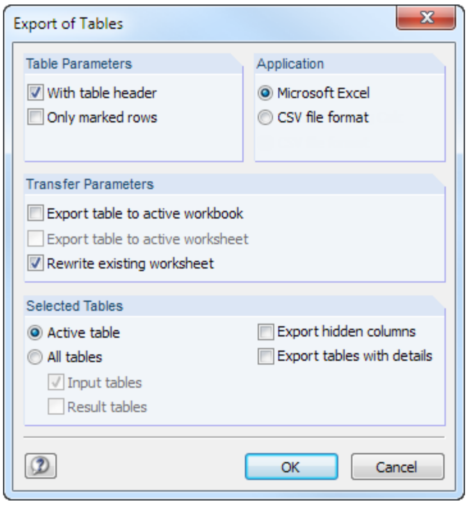
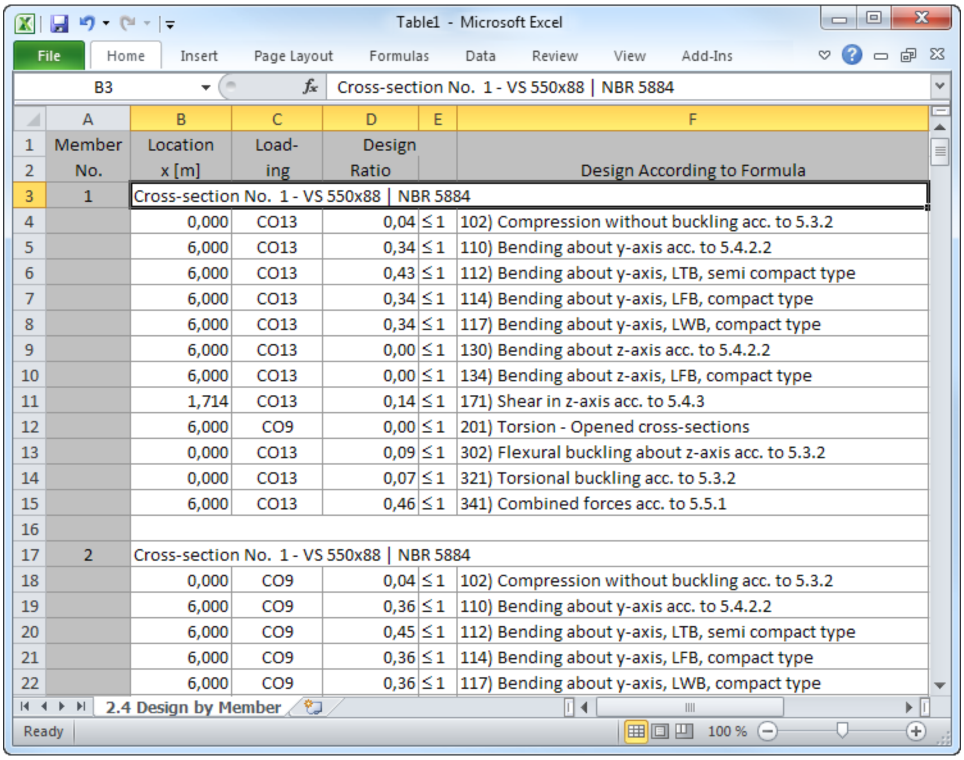


Figure 7.12: Dialog box *Export of Tables*

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



Member No.	Location x [m]	Loading	Design Ratio		Design According to Formula
1	Cross-section No. 1 - VS 550x88 | NBR 5884				
	0,000	CO13	0,04	≤ 1	102) Compression without buckling acc. to 5.3.2
	6,000	CO13	0,34	≤ 1	110) Bending about y-axis acc. to 5.4.2.2
	6,000	CO13	0,43	≤ 1	112) Bending about y-axis, LTB, semi compact type
	6,000	CO13	0,34	≤ 1	114) Bending about y-axis, LFB, compact type
	6,000	CO13	0,34	≤ 1	117) Bending about y-axis, LWB, compact type
	6,000	CO13	0,00	≤ 1	130) Bending about z-axis acc. to 5.4.2.2
	6,000	CO13	0,00	≤ 1	134) Bending about z-axis, LFB, compact type
	1,714	CO13	0,14	≤ 1	171) Shear in z-axis acc. to 5.4.3
	6,000	CO9	0,00	≤ 1	201) Torsion - Opened cross-sections
	0,000	CO13	0,09	≤ 1	302) Flexural buckling about z-axis acc. to 5.3.2
	0,000	CO13	0,07	≤ 1	321) Torsional buckling acc. to 5.3.2
	6,000	CO13	0,46	≤ 1	341) Combined forces acc. to 5.5.1
2	Cross-section No. 1 - VS 550x88 | NBR 5884				
	0,000	CO9	0,04	≤ 1	102) Compression without buckling acc. to 5.3.2
	6,000	CO9	0,36	≤ 1	110) Bending about y-axis acc. to 5.4.2.2
	6,000	CO9	0,45	≤ 1	112) Bending about y-axis, LTB, semi compact type
	6,000	CO9	0,36	≤ 1	114) Bending about y-axis, LFB, compact type
	6,000	CO9	0,36	≤ 1	117) Bending about y-axis, LWB, compact type

Figure 7.13: Results in Excel

Literature

[1] *NBR 8800:2008: Design of steel and composite structures for buildings*. Associação Brasileira de Normas Técnicas, 2008.

Index