Dlubal

Version
September 2013

Add-on Module

RF-STEEL NTC-DF

Ultimate Limit State, Serviceability and Stability Design According to NTC-RCDF 2004

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

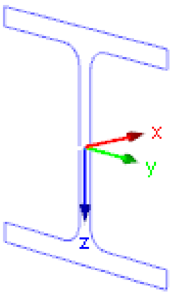
	Contents	Page
1.	**Introduction**	**4**
1.1	Add-on Module RF-STEEL NTC-DF	4
1.2	RF-STEEL NTC-DF - Team	5
1.3	Using the Manual	6
1.4	Open the Add-on Module RF-STEEL NTC-DF	6
2.	**Input Data**	**8**
2.1	General Data	8
2.1.1	Ultimate Limit State	9
2.1.2	Serviceability	10
2.2	Materials	11
2.3	Cross-sections	13
2.4	Intermediate Lateral Restraints	17
2.5	Effective Lengths - Members	18
2.6	Effective Lengths - Sets of Members	21
2.7	Nodal Supports - Sets of Members	22
2.8	Member End Releases - Sets of Members	24
2.9	Serviceability Data	25
2.10	Parameters - Members	26
3.	**Calculation**	**27**
3.1	Detail Settings	27
3.1.1	Ultimate Limit State	27
3.1.2	Stability	28
3.1.3	Serviceability	30
3.1.4	Other	31
3.2	Start Calculation	32
4.	**Results**	**33**
4.1	Design by Load Case	34
4.2	Design by Cross-Section	35
4.3	Design by Set of Members	36
4.4	Design by Member	37
4.5	Design by x-Location	37
4.6	Governing Internal Forces by Member	38
4.7	Governing Internal Forces by Set of Members	39
4.8	Member Slendernesses	40
4.9	Parts List by Member	41
4.10	Parts List by Set of Members	42
5.	**Results Evaluation**	**43**
5.1	Results in the RFEM Model	44
5.2	Result Diagrams	46
5.3	Filter for Results	47
6.	**Printout**	**49**
6.1	Printout report	49
6.2	RF-STEEL NTC-DF Graphic Printout	49
7.	**General Functions**	**51**
7.1	Design Cases	51
7.2	Cross-Section Optimization	53
7.3	Units and Decimal Places	55
7.4	Data Transfer	56
7.4.1	Export Material to RFEM	56
7.4.2	Export Effective Lengths to RFEM	56
7.4.3	Export Results	56
8.	**Example**	**58**
A	**Literature**	**65**
B	**Index**	**66**

1. Introduction

1.1 Add-on Module RF-STEEL NTC-DF

The Mexican standard (NTC-DF) describes the design, analysis and construction of steel structures in Mexico. With the RFEM add-on module RF-STEEL NTC-DF, DLUBAL ENGINEERING SOFTWARE provides a powerful tool for designing steel framework models.

RF-STEEL NTC-DF performs all typical ultimate limit state designs as well as stability and deformation analyses. The program is able to take into account various actions for the ultimate limit state design. Furthermore, you can choose between the interaction formulae mentioned in the code. In accordance with the code, RF-STEEL NTC-DF divides the cross-sections to be designed into the cross-section slenderness types. In this way, you can check the limitation of the design capacity and of the rotational capacity due to local buckling for cross-section parts. Moreover, RF-STEEL NTC-DF determines the c/t-ratios of the cross-section elements subjected to compression and classifies the cross-sections completely automatically.



Axis system

The axial system of members in RF-STEEL NTC-DF is different from the indices used in the Mexican standard: The index of the longitudinal member axis "z" is denoted as "x" in the program; "y" and "z" refer to the axes in the cross-section plane as seen in the image to the left.

For the stability analysis, you can specify for each member or set of members whether flexural buckling occurs in y- and/or z-direction. Furthermore, you can define additional lateral supports in order to represent the model close to reality. RF-STEEL NTC-DF determines the slenderness and elastic critical buckling loads from the boundary conditions. The ideal critical moment for lateral torsional buckling required for the lateral torsional buckling design can be determined automatically. In addition to that, it is possible to take into account the load application point of transverse loads, which is affecting the torsional resistance considerably.

For models with extremely slender cross-sections, the serviceability limit state represents an important design. The limit deformations are preset by default settings and can be adjusted, if necessary. In addition, it is possible to specify reference lengths and precambers that are considered accordingly in the design.

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

RF-STEEL NTC-DF 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 determination of internal forces to the design.

We hope you will enjoy working with RF-STEEL NTC-DF.

Your DLUBAL Team

1.2 RF-STEEL NTC-DF - Team

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

Program coordination

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

Programming

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

Cross-section and material database

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

Program design, dialog figures, and icons

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

Program supervision

Ing. Ph.D. Martin Čudejko
Dipl.-Ing. (FH) Sebastian Hawranke

Localization, manual

Ing. Fabio Borriello
Ing. Dmitry Bystrov
Eng.º Rafael Duarte
Ing. Jana Duníková
Ing. Lara Freyer
Bc. Chelsea Jennings
Ing. Ladislav Kábrt
Ing. Aleksandra Kociołek
Ing. Roberto Lombino
Eng.º Nilton Lopes
Mgr. Ing. Ph.D. Hana Macková
Ing. Téc. Ind. José Martínez
Mgr. Petra Pokorná
Ing. Marcela Svitáková
Dipl.-Ing. (FH) Robert Vogl
Ing. Marcin Wardyn

Technical support and quality management

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

1.3 Using the Manual

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

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

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

1.4 Open the Add-on Module RF-STEEL NTC-DF

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

Menu

To start the program in the RFEM menu bar, click

Add-on Modules → **Design - Steel** → **RF-STEEL NTC-DF**.

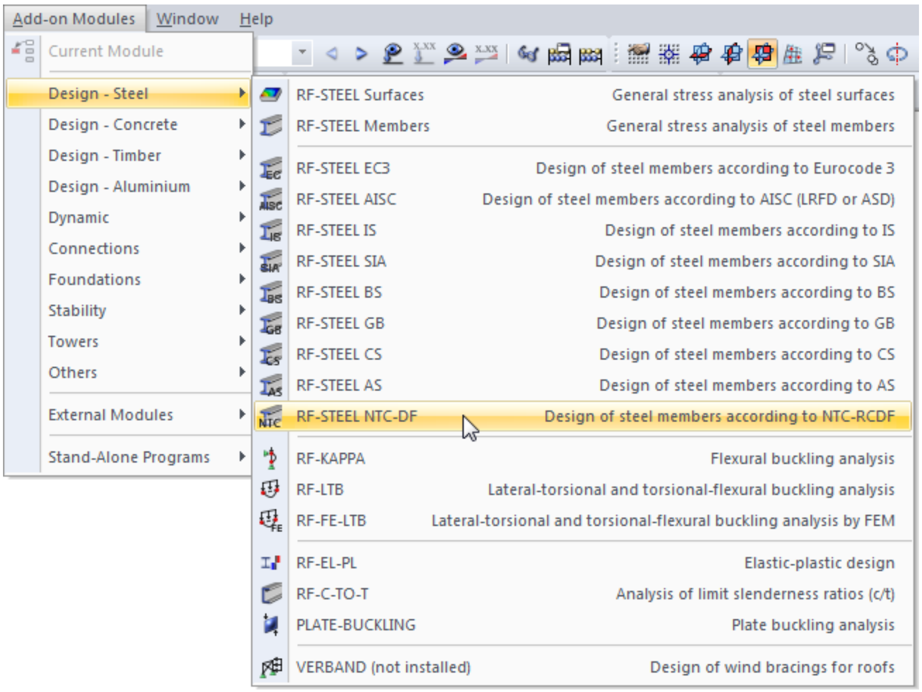


Figure 1.1: Menu: *Add-on Modules* → *Design - Steel* → *RF-STEEL NTC-DF*

Navigator

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

Add-on Modules → RF-STEEL NTC-DF.

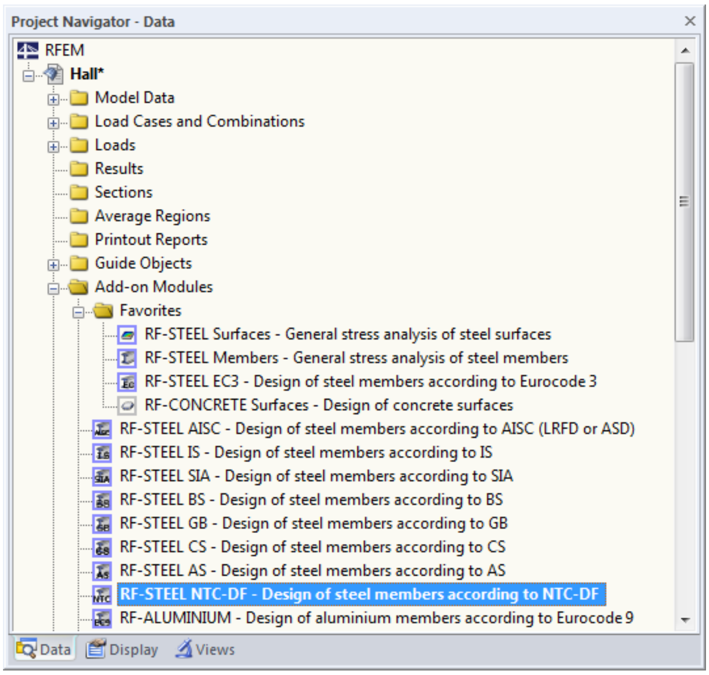
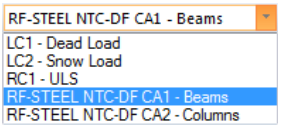
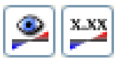


Figure 1.2: Data navigator: *Add-on Modules → RF-STEEL NTC-DF*

Panel





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

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

When the results display is activated, the panel is available, too. Click [RF-STEEL NTC-DF] in the panel to open the module.

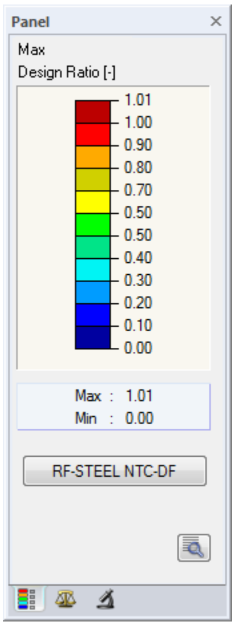


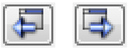
Figure 1.3: Panel button [RF-STEEL NTC-DF]

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 51).

The design relevant data is defined in several input windows. When you open RF-STEEL NTC-DF 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 NTC-DF and return to the main program. To exit the module without saving the data, click [Cancel].

2.1 General Data

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

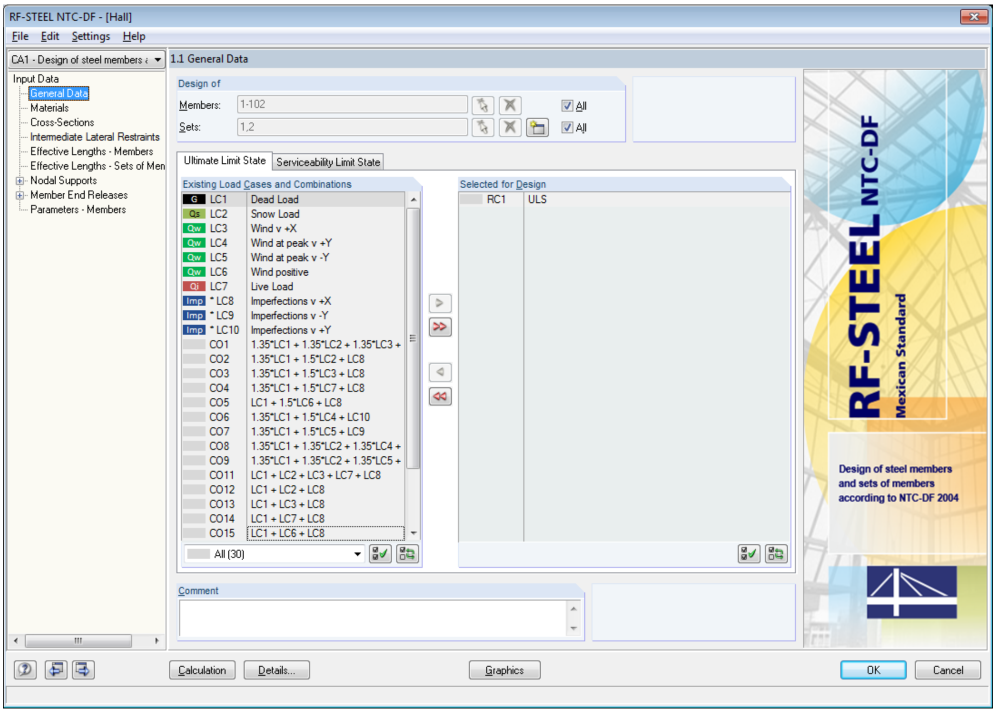


Figure 2.1: Window 1.1 *General Data*

Design of

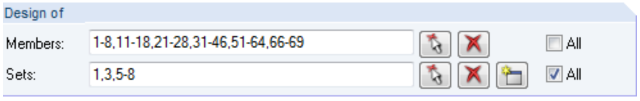


Figure 2.2: Design of members and sets of members

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 input fields to enter the numbers of the relevant members or sets of members. The list of the numbers preset in the field can be selected by double-clicking and overwritten by entering the data manually. 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 analyses of all members contained in the set of members and takes into account the boundary conditions of connected members for the stability analysis. The results are shown in the results windows 2.3 *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

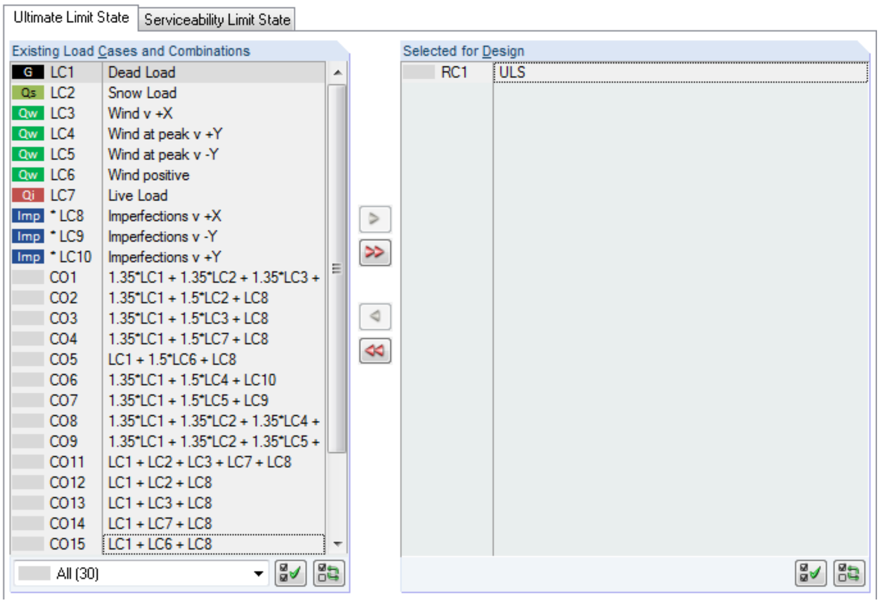


Figure 2.3: Window 1.1 *General Data*, tab *Ultimate Limit State*

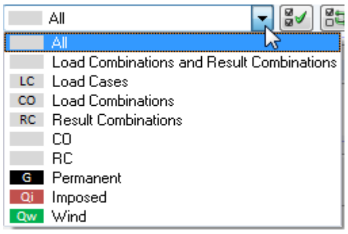
Existing Load Cases and Combinations

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

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

To transfer multiple entries of load cases, select them while pressing the [Ctrl] key, as common for Windows applications. Then use the button [▶] to transfer them simultaneously.

Load cases marked by an asterisk (*), like load case 8 in Figure 2.3, cannot be designed: This happens when the load cases are defined without any load data or the load cases contain only imperfections. When you transfer the load cases, a corresponding warning appears.



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

	Selects all cases in the list
	Inverts selection of load cases

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

Selected for Design

The column on the right lists the load cases as well as the 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 [◀◀].

The analysis of an enveloping max/min result combination is performed faster than the analysis of all load cases and load combinations that have been globally set. However, when analyzing a result combination, the influence of the contained loads is difficult to discern.

2.1.2 Serviceability

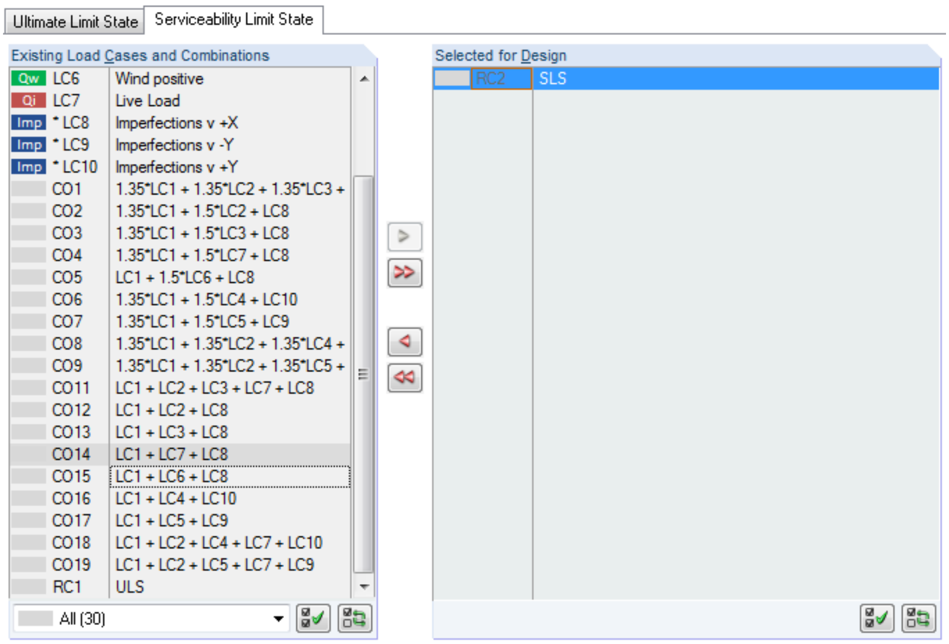


Figure 2.4: Window 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

This section lists all load cases, load combinations, and result combinations 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.

You can assign different limit values for deflection to the individual load cases, load combinations, and result combinations. Those limit values of the deformations can be adjusted, if necessary: Click [Details] to open the dialog box *Details* (see Figure 3.3, page 30).

In the 1.9 *Serviceability Data* window, the reference lengths that are governing for the deformation check are managed (see chapter 2.9, page 25).

2.2 Materials

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

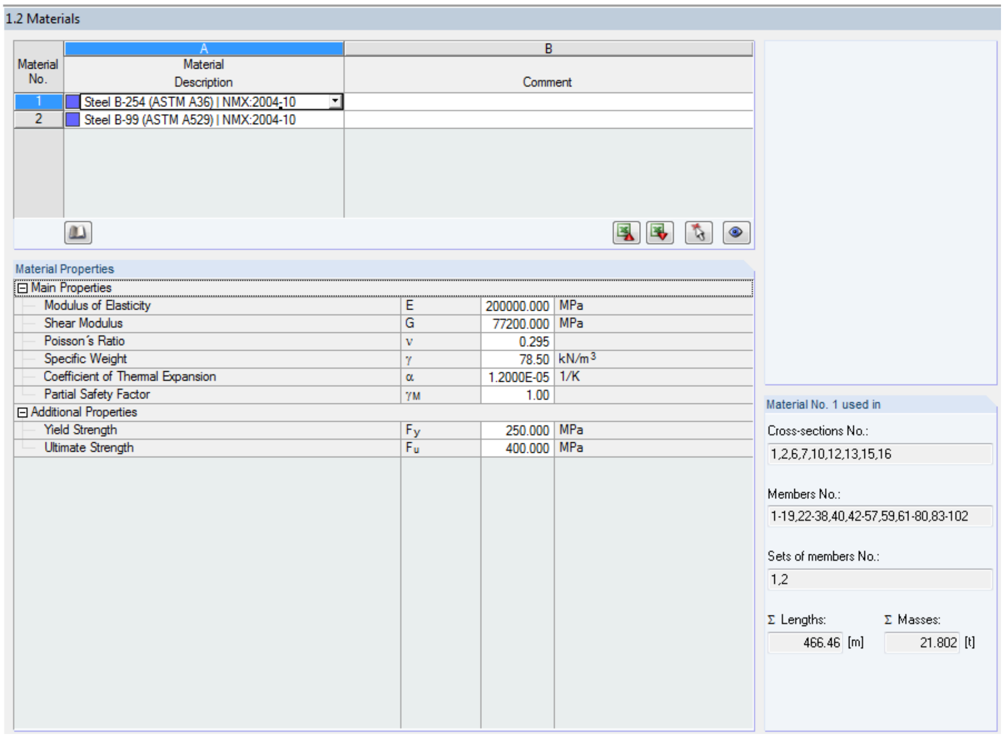


Figure 2.5: Window 1.2 *Materials*

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 from the module's menu **Settings** → **Units and Decimal Places** (see chapter 7.3, page 55).

Material Description

The materials defined in RFEM are already preset, but you can always modify them: To select the field, click the material in column A. Then click [▼] or press function key [F7] to open the material list.

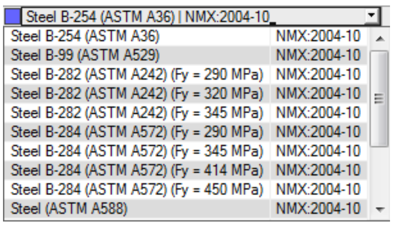


Figure 2.6: List of materials

According to the design concept of the standard [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 NTC-DF will import the material properties, too.

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

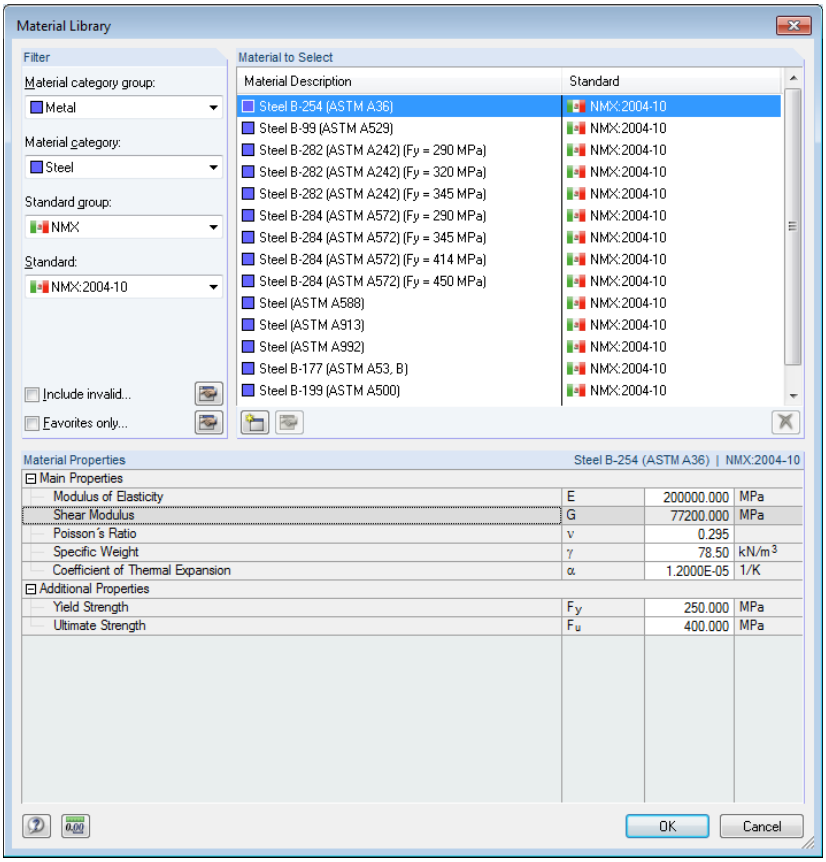


Figure 2.7: Dialog box *Material Library*

In the *Filter* section, *Steel* is preset as material category. Select the material quality that you want to use for the design in the *Material to Select* list. 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 RF-STEEL NTC-DF module.

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 allowed 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.

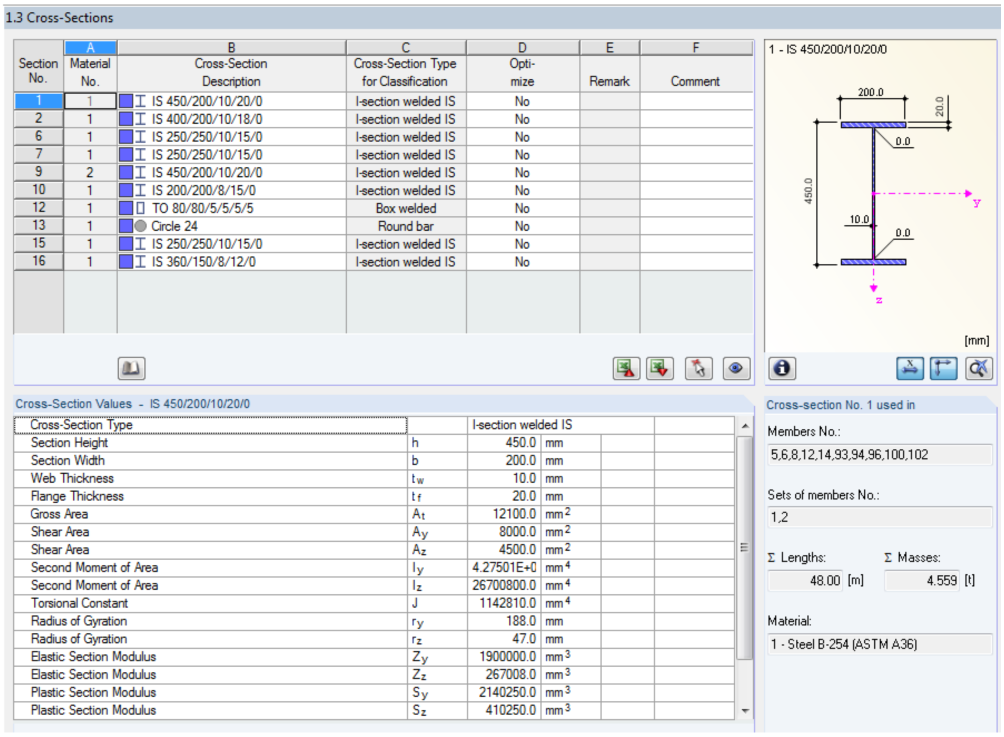


Figure 2.8: Window 1.3 *Cross-sections*

Cross-Section Description

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

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

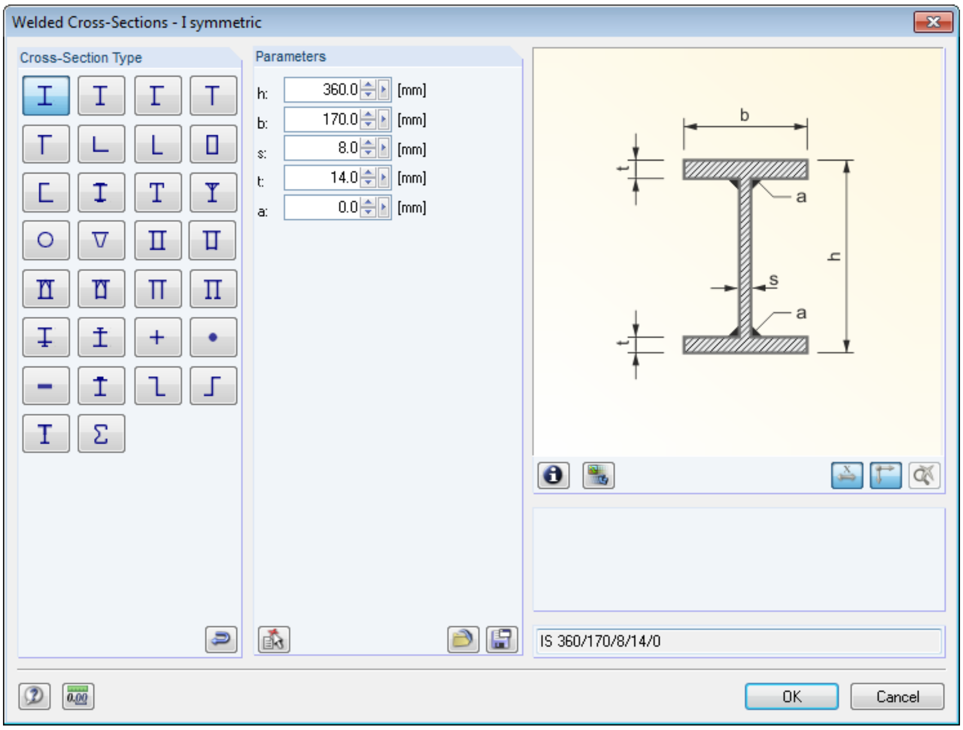
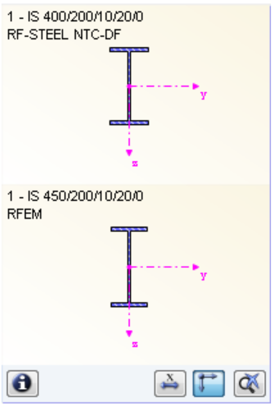


Figure 2.9: IS cross-sections in the cross-section library



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

A modified cross-section will be highlighted in blue.

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

Cross-Section Type for Classification

The cross-section type used for the classification is displayed. The cross-sections listed in [1] Table 5.2 can be designed plastically or elastically depending on the Class. Cross-sections that are not covered by this table 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 that comes as close as possible to a user-defined maximum utilization ratio. You can define the maximum ratio in the *Other* tab of the *Details* dialog box, (see Figure 3.4, page 31).

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 53.

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 stored 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 in Figure 2.8).

Member with tapered cross-section

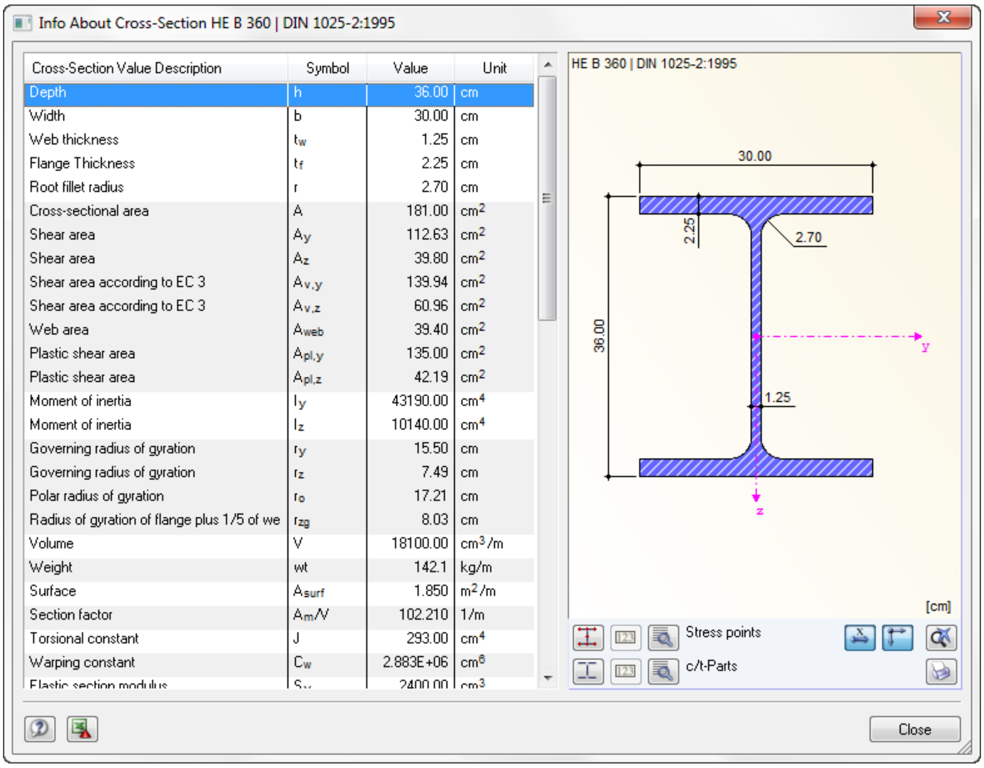
For tapered members with different cross-sections at the member start and member end, the module displays both cross-section numbers in two tables, in accordance with the definition in RFEM.

RF-STEEL NTC-DF also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the member end. For example, the normal stresses, are determined from the moments of inertia and the centroidal distances of the stress points. If the cross-sections at the start and the end 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 NTC-DF.

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.



Cross-Section Value Description	Symbol	Value	Unit
Depth	h	36.00	cm
Width	b	30.00	cm
Web thickness	t_w	1.25	cm
Flange Thickness	t_f	2.25	cm
Root fillet radius	r	2.70	cm
Cross-sectional area	A	181.00	cm^2
Shear area	A_y	112.63	cm^2
Shear area	A_z	39.80	cm^2
Shear area according to EC 3	$A_{v,y}$	139.94	cm^2
Shear area according to EC 3	$A_{v,z}$	60.96	cm^2
Web area	A_{web}	39.40	cm^2
Plastic shear area	$A_{pl,y}$	135.00	cm^2
Plastic shear area	$A_{pl,z}$	42.19	cm^2
Moment of inertia	I_y	43190.00	cm^4
Moment of inertia	I_z	10140.00	cm^4
Governing radius of gyration	r_y	15.50	cm
Governing radius of gyration	r_z	7.49	cm
Polar radius of gyration	r_o	17.21	cm
Radius of gyration of flange plus 1/5 of we	r_{zg}	8.03	cm
Volume	V	18100.00	cm^3/m
Weight	wt	142.1	kg/m
Surface	A_{surf}	1.850	m^2/m
Section factor	A_m/V	102.210	1/m
Torsional constant	J	293.00	cm^4
Warping constant	C_w	2.883E+06	cm^6
Elastic section modulus	S_y	2400.00	cm^3

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 are reserved for the following functions:

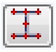
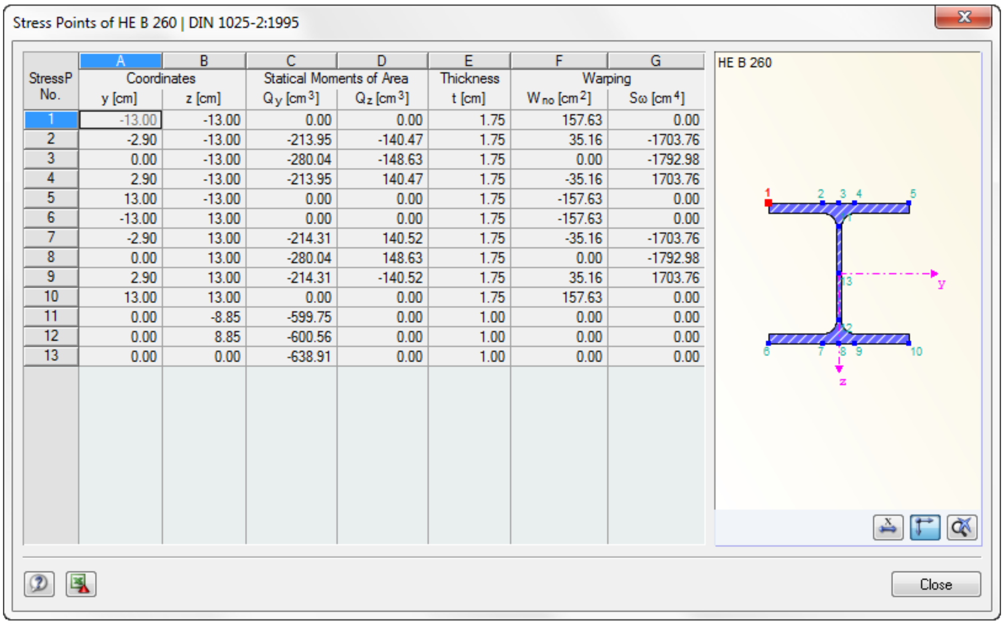
Button	Function
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the numbering of stress points or c/t-parts
	Displays or hides the details of the stress points or c/t-parts (see Figure 2.11)
	Displays or hides the dimensions of the cross-section
	Displays or hides 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.



StressP No.	A Coordinates y [cm]	B z [cm]	C Statical Moments of Area Q_y [cm³]	D Q_z [cm³]	E Thickness t [cm]	F Warping W_{no} [cm²]	G S_ω [cm⁴]
1	-13.00	-13.00	0.00	0.00	1.75	157.63	0.00
2	-2.90	-13.00	-213.95	-140.47	1.75	35.16	-1703.76
3	0.00	-13.00	-280.04	-148.63	1.75	0.00	-1792.98
4	2.90	-13.00	-213.95	140.47	1.75	-35.16	1703.76
5	13.00	-13.00	0.00	0.00	1.75	-157.63	0.00
6	-13.00	13.00	0.00	0.00	1.75	-157.63	0.00
7	-2.90	13.00	-214.31	140.52	1.75	-35.16	-1703.76
8	0.00	13.00	-280.04	148.63	1.75	0.00	-1792.98
9	2.90	13.00	-214.31	-140.52	1.75	35.16	1703.76
10	13.00	13.00	0.00	0.00	1.75	157.63	0.00
11	0.00	-8.85	-599.75	0.00	1.00	0.00	0.00
12	0.00	8.85	-600.56	0.00	1.00	0.00	0.00
13	0.00	0.00	-638.91	0.00	1.00	0.00	0.00

Figure 2.11: Dialog box *Stress Points of HE B 260*

2.4 Intermediate Lateral Restraints

In window 1.4, you can define intermediate lateral restraints for members. RF-STEEL NTC-DF always assumes this kind of support to be perpendicular to the cross-section's minor axis z (see Figure 2.10). Thus, it is possible to influence the members' effective lengths which are important for the stability analyses concerning flexural buckling and lateral-torsional buckling.

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

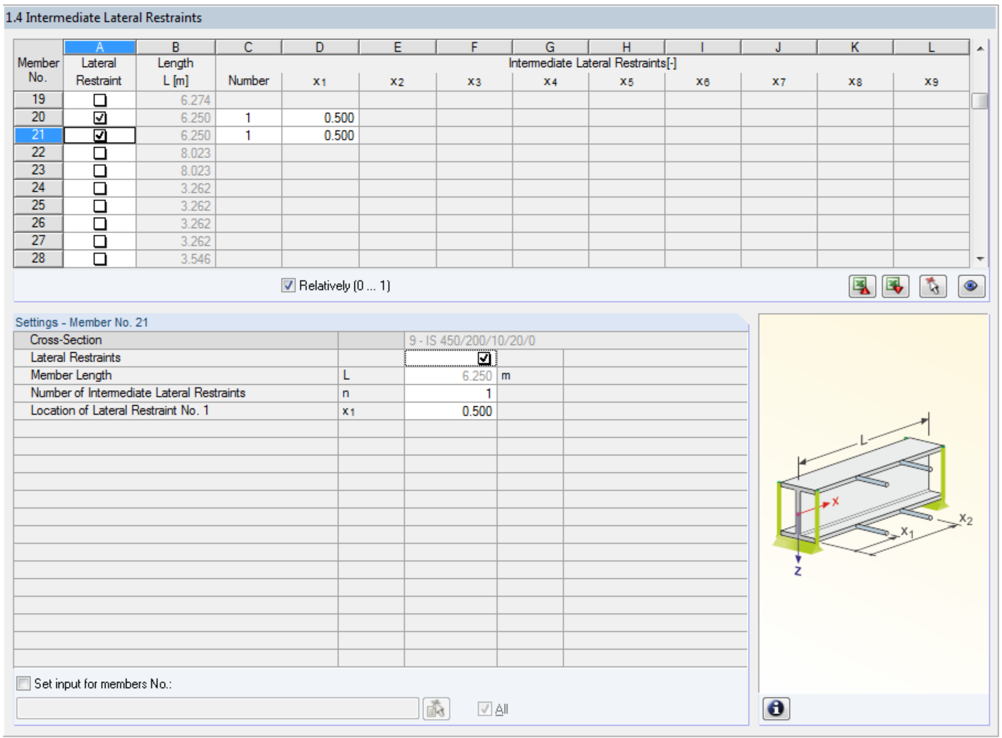


Figure 2.12: Window 1.4 *Intermediate Lateral Restraints*

In the upper part of the window, you can assign up to nine lateral restraints for each member. The *Settings* section shows the input as column overview for 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.

Relatively (0 ... 1)

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

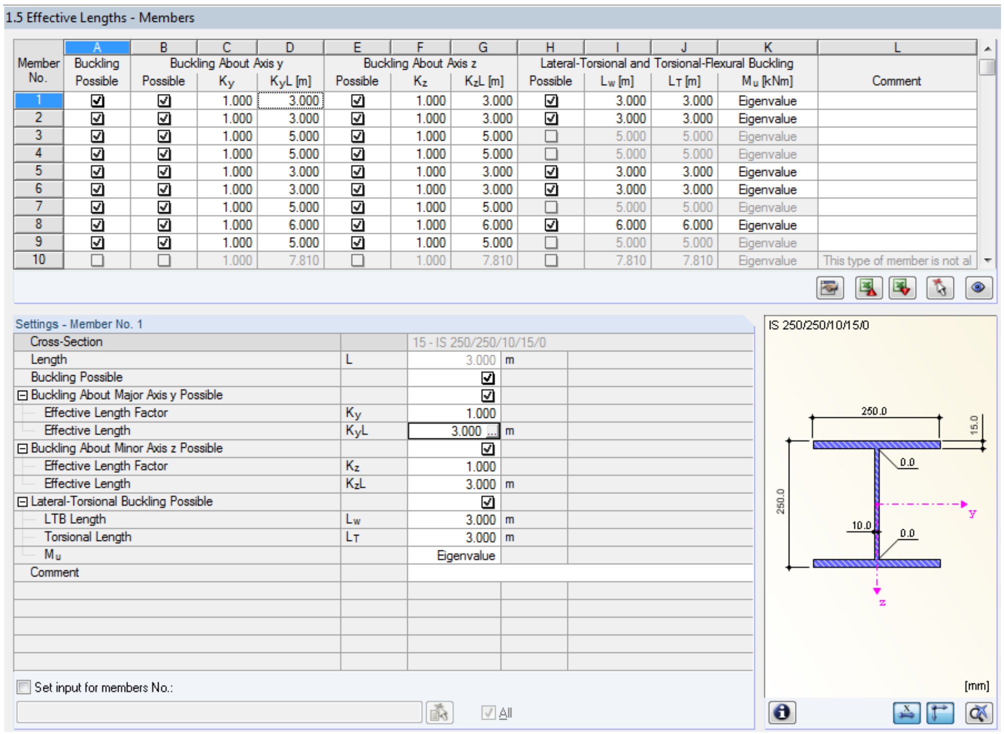
In case of cantilevers, avoid intermediate restraints because such supports divide the member into segments. For cantilevered beams, this would result in segments with lateral torsional restraints on one end each that are statically underdetermined.

2.5 Effective Lengths - Members

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



1.5 Effective Lengths - Members

Member No.	A Buckling Possible	B Buckling About Axis y Possible	C K_y	D K_yL [m]	E Buckling About Axis z Possible	F K_z	G K_zL [m]	H Lateral-Torsional and Torsional-Flexural Buckling Possible	I L_w [m]	J L_T [m]	K M_u [kNm]	L Comment
1	☑	☑	1.000	3.000	☑	1.000	3.000	☑	3.000	3.000	Eigenvalue	
2	☑	☑	1.000	3.000	☑	1.000	3.000	☑	3.000	3.000	Eigenvalue	
3	☑	☑	1.000	5.000	☑	1.000	5.000	☐	5.000	5.000	Eigenvalue	
4	☑	☑	1.000	5.000	☑	1.000	5.000	☐	5.000	5.000	Eigenvalue	
5	☑	☑	1.000	3.000	☑	1.000	3.000	☑	3.000	3.000	Eigenvalue	
6	☑	☑	1.000	3.000	☑	1.000	3.000	☑	3.000	3.000	Eigenvalue	
7	☑	☑	1.000	5.000	☑	1.000	5.000	☐	5.000	5.000	Eigenvalue	
8	☑	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	6.000	Eigenvalue	
9	☑	☑	1.000	5.000	☑	1.000	5.000	☐	5.000	5.000	Eigenvalue	
10	☐	☐	1.000	7.810	☐	1.000	7.810	☐	7.810	7.810	Eigenvalue	This type of member is not al

Figure 2.13: Window 1.5 *Effective Lengths - Members*

The effective lengths for buckling about the minor z-axis are aligned automatically with the entries of the 1.4 *Intermediate Lateral Restraints* window. If intermediate restraints divide the member into member segments of different lengths, the program displays no values in the table columns G, I, 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 input field (see figure above).

The *Settings* tree manages the following parameters:

- *Cross-Section*
- *Member Length*
- *Buckling Possible* for member (cf columns B and E)
- *Buckling about Axis y Possible* (cf columns C and D)
- *Buckling about Axis z Possible* (cf columns F and G)
- *Lateral-Torsional Buckling Possible* (cf columns I - K)

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.

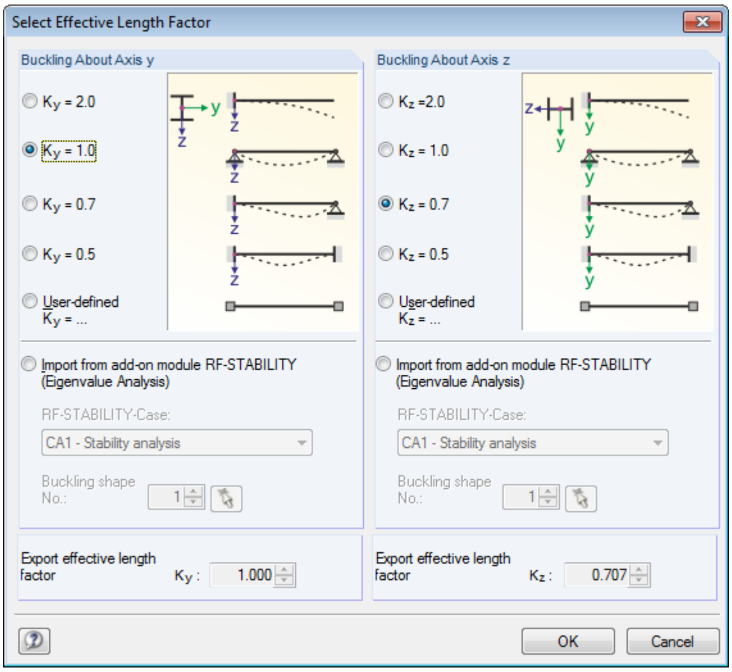


Figure 2.14: Dialog box *Select Effective Length Factor*

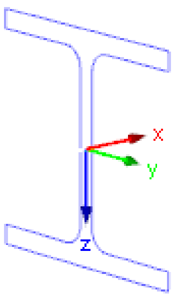
For each direction, the buckling length can be defined according to one of the Euler buckling modes or *User-defined*. If a RF-STABILITY case calculated according to the eigenvalue analysis is already available, you can also define a *Buckling Shape* to determine the factor.

Buckling Possible

A stability analysis for flexural 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 displayed in the *Comment* column.

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

Buckling about Axis y or Axis z



Axis system

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 buckling length coefficients K_y and K_z for buckling about the major or the minor axis can be selected freely.

You can check the position of the member axes in the cross-section graphic in the 1.3 *Cross-Sections* window (see Figure 2.8, page 13). 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 (see figure below).

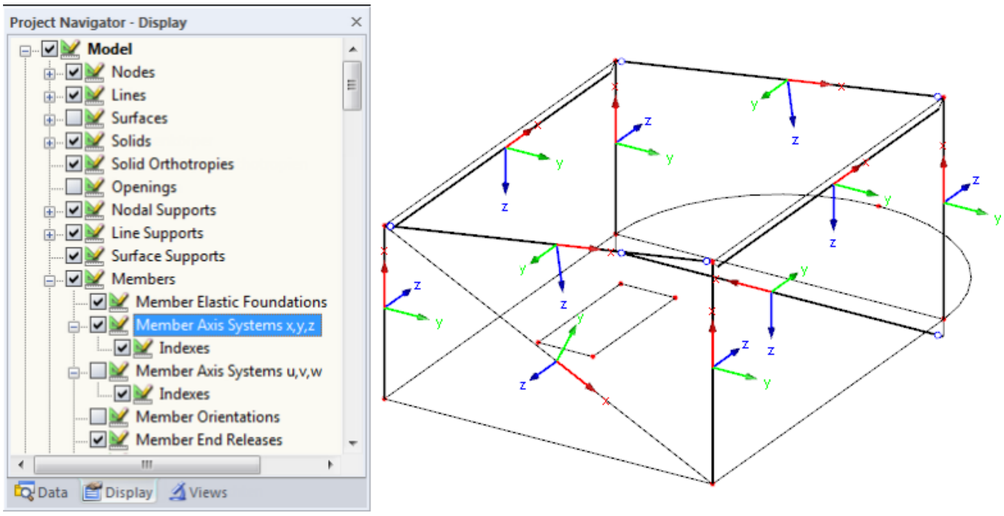


Figure 2.15: Selecting the member axis systems in the *Display* navigator of RFEM

If buckling is possible about one or even both member axes, you can enter the effective length factors as well as the buckling lengths in the columns C and D or 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*-input field (see Figure 2.13).

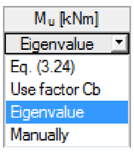
When you specify the effective length factor *K*, the program determines the effective length *KL* by multiplying the member length *L* by the effective length factor *K*. The input fields *K* and *KL* are interactive.

Lateral-Torsional and Torsional-Flexural Buckling Possible

Table column H shows you for which members the program performs an analysis of lateral-torsional and torsional-flexural buckling.

With the check box in the *Possible* columns, you decide whether a member is susceptible to torsional buckling. The LTB lengths L_w and the torsional lengths L_T in columns I and J can be edited by the user.

Elastic Buckling Moment M_u



The list in column K includes four options for the calculation of the nominal bending resistance moment M_u. It can be accessed by clicking the button [▼] which appears after clicking in a cell of this table column.

- **Eq. (3.24)**

  M_u is calculated according to [1] clause 3.3.2.2.

$$M_u = \frac{\pi}{CL}\sqrt{E \cdot I_z \cdot G \cdot J + \left(\frac{\pi \cdot E}{L}\right)^2 \cdot I_z \cdot C_a}$$

- **Use factor C_b**

  The modification factor C_b is calculated according to the AISC standard [2] equation F1-1.

$$C_b = \frac{12.5 M_{max}}{2.5 M_{max} + 3 M_A + 4 M_B + 3 M_C}$$

where

M_{max}	absolute value of maximum bending moment in unbraced segment
M_A	absolute value of bending moment at quarter point of segment
M_B	absolute value of bending moment at center of member or segment
M_C	absolute value of bending moment at three-quarter point of segment

The nominal bending resistance moment is then calculated with this modification factor.

$$M_u = C_b \frac{\pi}{CL} \sqrt{E \cdot I_z \cdot G \cdot J + \left(\frac{\pi \cdot E}{L}\right)^2 \cdot I_z \cdot C_a}$$

- **Eigenvalue**

  This option is set as default. It uses the general eigenvalue solver to determinate the elastic buckling moment.

- **Manually**

  The value of M_u can be defined individually.

Comment

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

2.6 Effective Lengths - Sets of Members

This window appears only if you have selected at least one set of members for design in the 1.1 *General Data* window (see Figure 3.2, page 28).

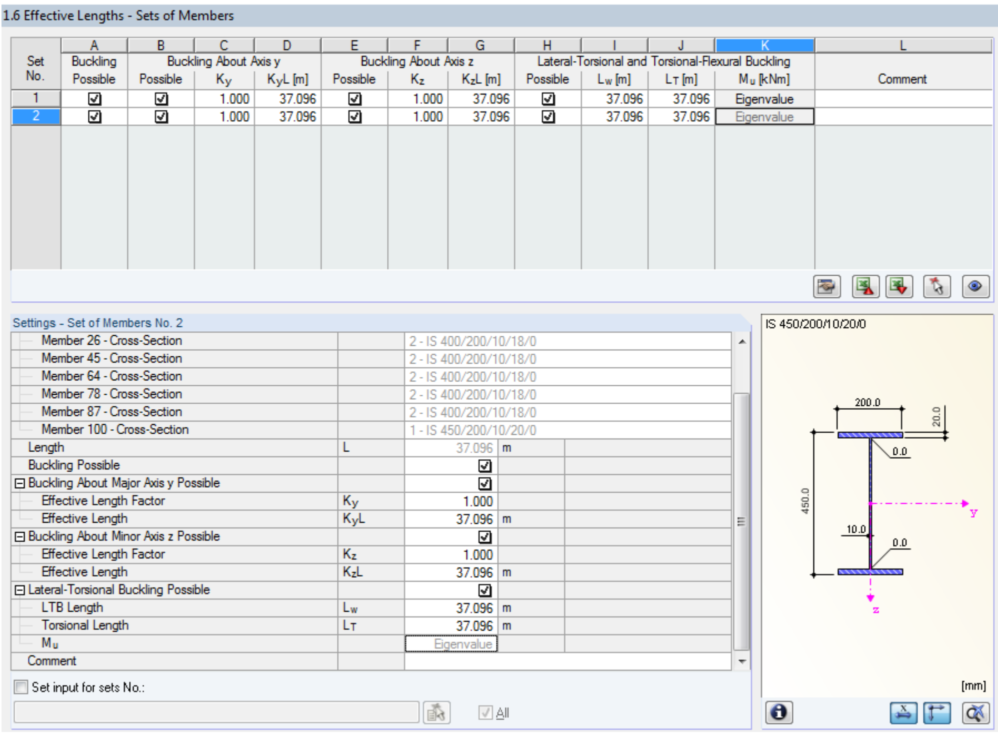


Figure 2.16: Window 1.6 *Effective Lengths - Sets of Members*

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 for the buckling about the two principal axes of the set of members 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.

If the *Member-Like Input* is selected for sets of members in the dialog box *Details* dialog box, tab *Stability* (see Figure 3.2, page 28), window 1.7 will not be displayed. In that case, you can define the intermediate lateral restraints by using division points in window 1.4.

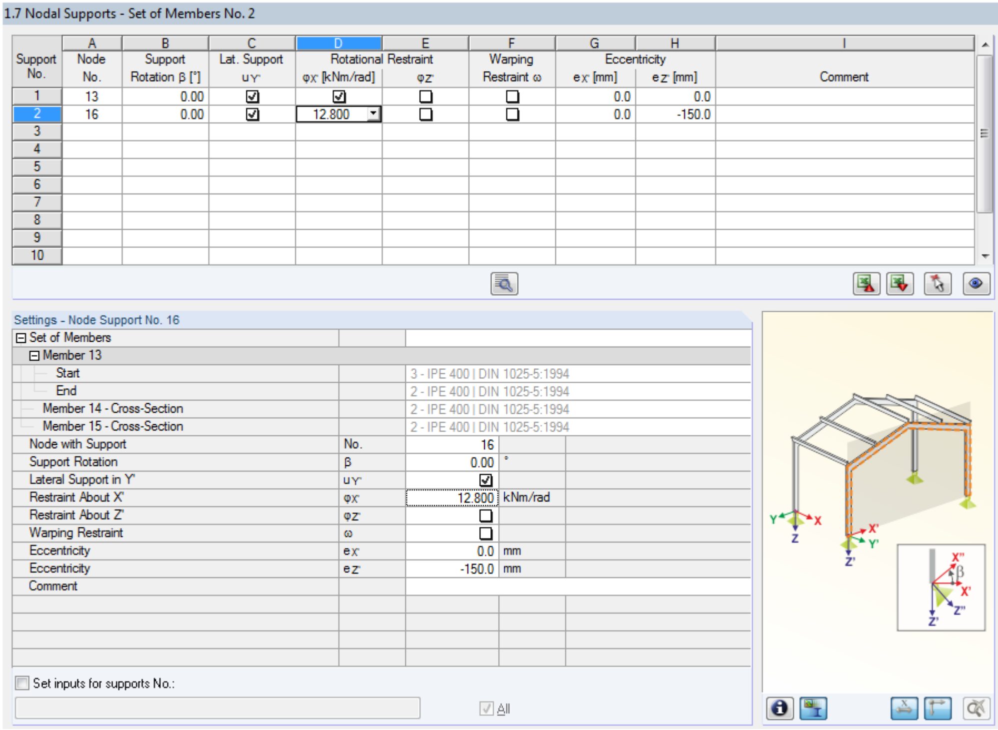


Figure 2.17: Window 1.7 *Nodal Supports - Set of Members*

To determine critical buckling factor of lateral-torsional buckling, a planar framework is created with four degrees of freedom for each node, which you have to define in window 1.7. This window refers to the current set of members (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 to Figure 2.21, the axes of the nodal supports for window 1.7.

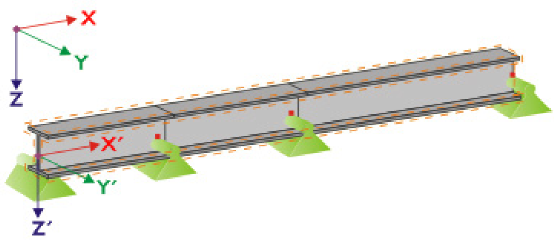


Figure 2.18: Auxiliary coordinate system for nodal supports – straight set of members

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.

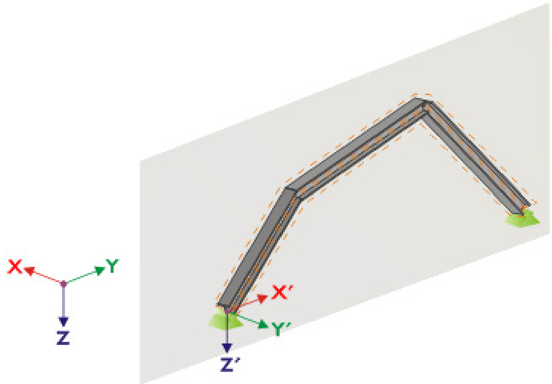


Figure 2.19: Auxiliary coordinate system for nodal supports – set of members in vertical plane

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 axis X' is horizontal and aligned in direction of the plane. The axis Y' is horizontal as well and defined perpendicular to the axis X'. The axis Z' is directed perpendicularly downwards.

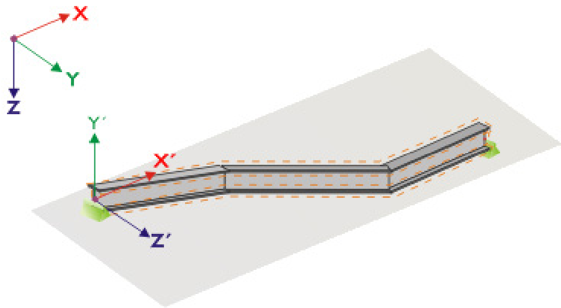


Figure 2.20: Auxiliary coordinate system for nodal supports – set of members in horizontal plane

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 axis Z' is directed parallel to the global Y-axis.

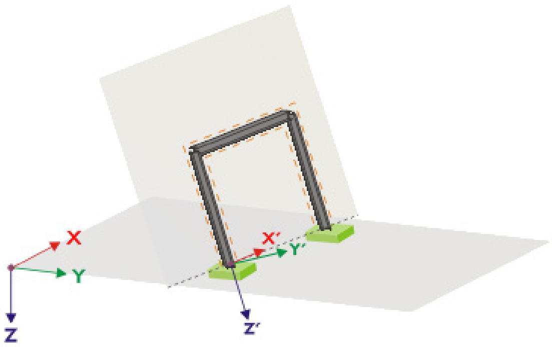


Figure 2.21: Auxiliary coordinate system for nodal supports – set of members in inclined plane

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 axis X' 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 will not be displayed when in the *Details* dialog box (see Figure 3.2, page 28) the *Member-Like Input* is selected for sets of members.

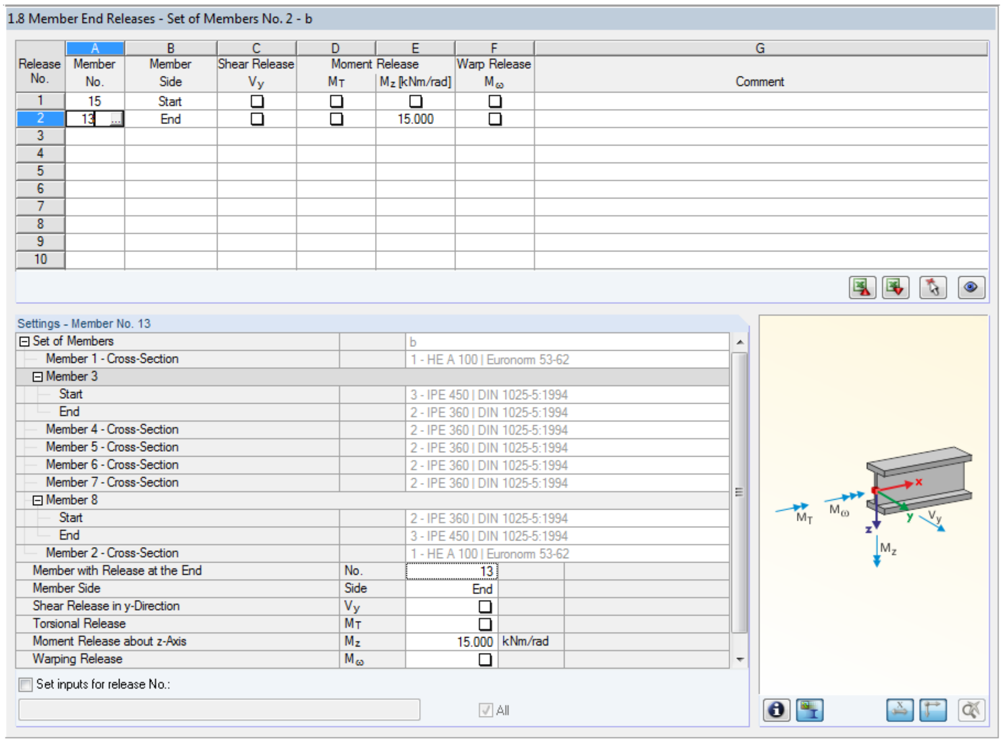
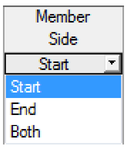


Figure 2.22: Window 1.8 *Member Releases - Set of Members*

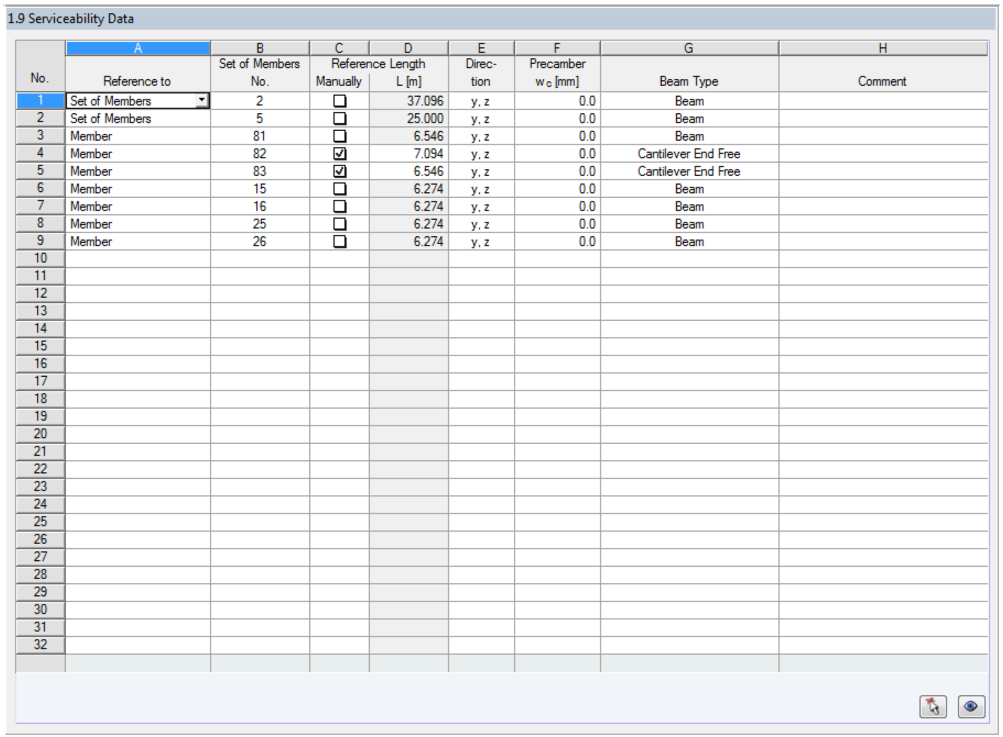


In table column B, you define the *Member Side* to which the release should be assigned. You can also connect the releases to both member sides.

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.

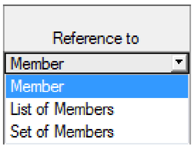
2.9 Serviceability Data

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 (see chapter 2.1.2, page 10).


1.9 Serviceability Data

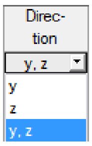
No.	A Reference to	B Set of Members No.	C Reference Length Manually	D Reference Length L [m]	E Direc-tion	F Precamber w_c [mm]	G Beam Type	H Comment
1	Set of Members	2	☐	37.096	y, z	0.0	Beam	
2	Set of Members	5	☐	25.000	y, z	0.0	Beam	
3	Member	81	☐	6.546	y, z	0.0	Beam	
4	Member	82	☑	7.094	y, z	0.0	Cantilever End Free	
5	Member	83	☑	6.546	y, z	0.0	Cantilever End Free	
6	Member	15	☐	6.274	y, z	0.0	Beam	
7	Member	16	☐	6.274	y, z	0.0	Beam	
8	Member	25	☐	6.274	y, z	0.0	Beam	
9	Member	26	☐	6.274	y, z	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 2.23: Window 1.9 *Serviceability Data*



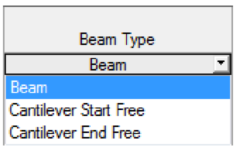
In column A, you decide whether you want to apply the deformation to single members, lists of members, or sets of members.

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.



In table column E, you define 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).

In column F, you can consider a *precamber* w_c.



The *Beam Type* is of vital importance for the correct application of limit deformations. In column G, you can specify whether there is a beam or a cantilever and which end should have no support.

The settings in the *Serviceability* tab of the *Details* dialog box decide whether the deformations are related to the undeformed initial model or to the shifted ends of members or sets of members (see Figure 3.3, page 30).

2.10 Parameters - Members

The last input window controls additional design parameters for members.

If the *Cross-Sectional Area* of a member is to be considered by specific parameters, set a check in relevant line of column A. You can then define the *Cross-sectional areas for tension design* in the *Settings* table below (net and effective areas, shear lag factor U).

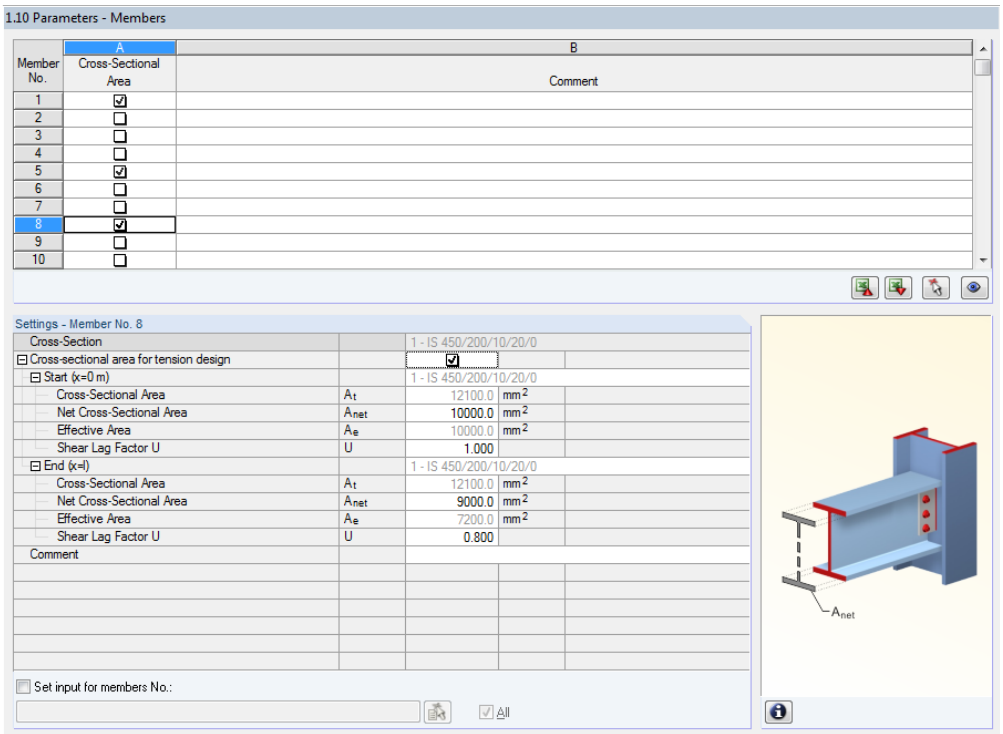


Figure 2.24: Window 1.10 *Parameters - Members*

Those parameters are relevant for the design of member connections and of cross-sections with tension.

3. Calculation

3.1 Detail Settings

Calculation

Details...

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

The dialog box *Details* contains the following tabs:

- Ultimate Limit State
- Stability
- Serviceability
- Other

3.1.1 Ultimate Limit State

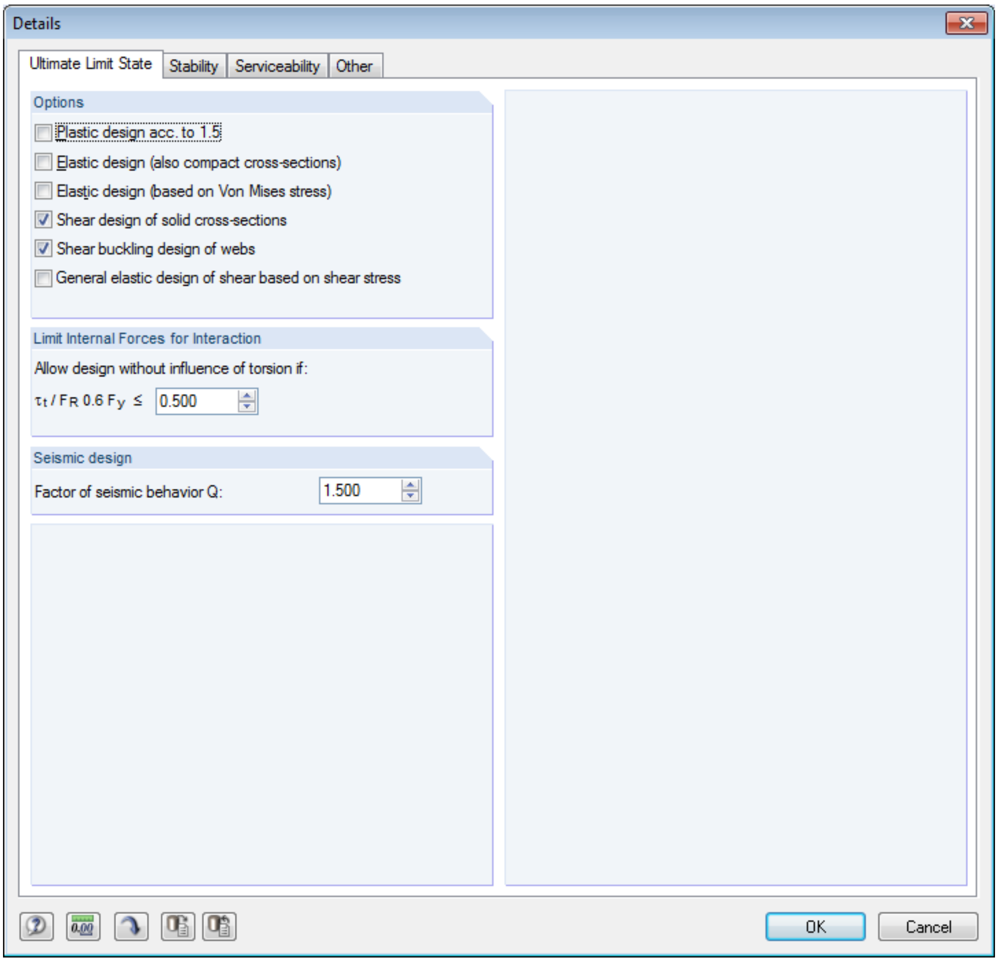


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

Options

According to [1] clause 1.5, there is also a *Plastic design* possible for members. If the requirements given in clause 1.5(a) are satisfied, hot-formed doubly symmetric I-sections can be designed according to this option. Cross-sections that are assigned to type 1 or 2 ("compact") will be designed plastically in RF-STEEL NTC-DF. If you do not want to perform a plastic design, you can activate the *Elastic design* for these cross-sections, too. Then all cross-sections will be considered as type 3 ("non-compact").

Alternatively, a conservative general elastic design based on stress analysis in stress points and VON MISES equivalent stresses can be applied. This option is useful for cross-sections with complex shapes or for members with torsional moments etc.

If the *Shear design* of solid flat or round bars or *Shear buckling design* of webs is not required in special cases, this design option can be deactivated.

The conservative *General elastic design of shear* based on the shear stress analysis in stress points can be activated additionally.

Limit Internal Forces for Interaction

The standard [1] offers no exact procedure how to design cross-sections under the torsion. Therefore, there is an option to ignore shear stress due to torsion for the cross-section design. You can enter the maximum ratio of torsional shear stress and shear strength so that the design is possible in spite of small torsional moments.

Seismic Design

The factor of seismic behavior Q can be edited, if necessary. This value is used to determine the cross-section type according to [1] Table 2.1.

3.1.2 Stability

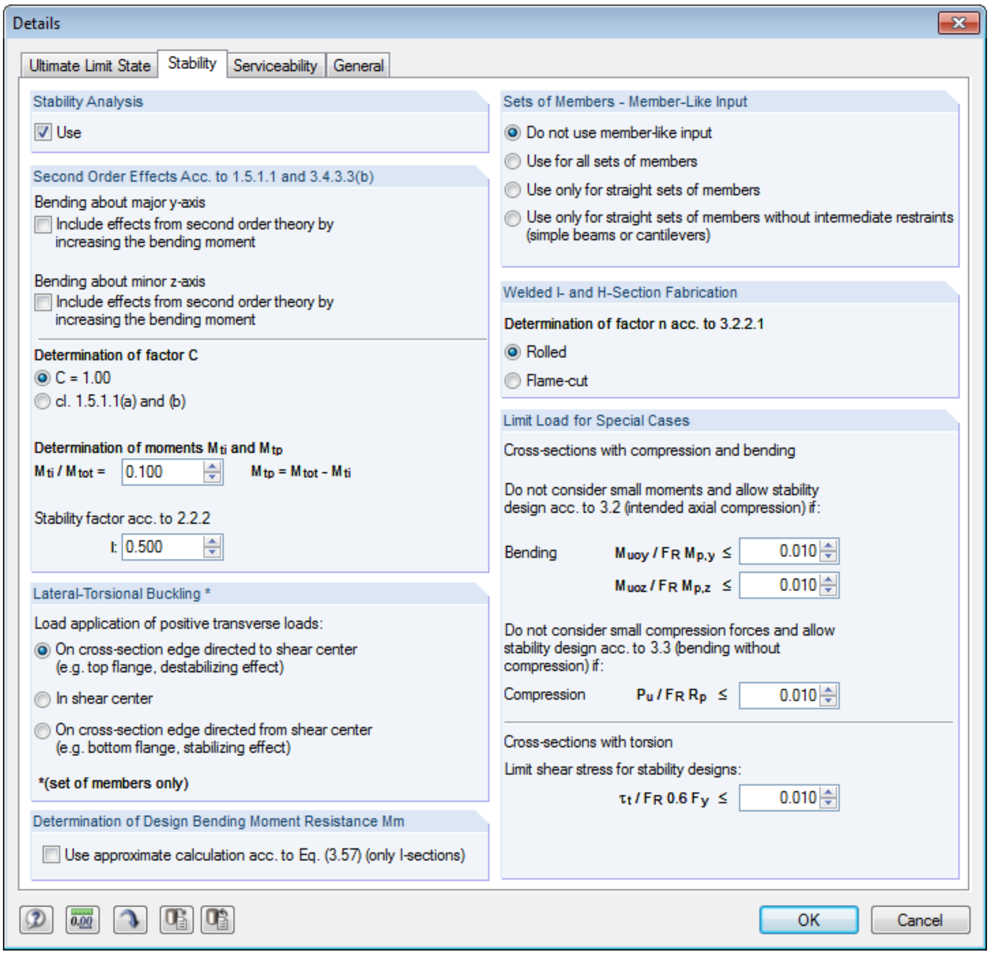


Figure 3.2: Dialog box *Details*, tab *Stability*

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 through 1.8 will not be displayed.

Second Order Effects

If a load case is calculated according to a linear static analysis, you can consider the *effects from 2nd order theory by increasing the bending moment* about the major and/or minor axis, according to [1] clauses 1.5.1.1 and 3.4.3.3(b). 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 by the appropriate factors. If you increase the bending moment, it does not affect the flexural-buckling analysis according to [1] as it is performed by using the axial forces.

The *Determination of factor C* can be by default C = 1.00 or according to clause 1.5.1.1(a) and (b). The bending *moments M_{ti} and M_{tp}* are determined in very simple way – as constant ratio to the total bending moment value. This ratio is constant for all members. If second order effects are included, the factors B1 and B2 are used to calculate the final design values M_{uo} and M^*_{uo} according to [1] Eq. (1.1) and (1.2) for plastic design and also according to Eq. (3.59) to (3.62). The *Stability factor* I of the entire frame (model) according to clause 2.2.2 can be edited, if necessary.

Lateral-Torsional Buckling

If transverse loads are present, it is important to define where these forces are acting on the cross-section: Depending on the *Load application* point, transverse loads can be stabilizing or destabilizing, and thus can decisively influence the ideal critical moment. The determination of the buckling factor α_{cr} for set of members is based on those settings. Please note that the load application point is only taken into account for sets of members.

Determination of Bending Design Resistance M_m

The value of the bending design resistance M_m is required for cross-section types 1 or 2 according to [1] clause 3.4.3.2, Eq. (3.56). This value can be calculated according to clause 3.3.2 or, approximately, according to Eq. (3.57) for I-sections. When this option is checked, M_m is determined, the *approximate calculation* is applied according to the Eq. (3.57) for I-sections, and according to clause 3.3.2 for all other cross-sections.

Set of Members - Member-Like Input

It is recommended to apply the RF-STEEL NTC-DF design only for straight sets of members. The stability data can be defined as member-like in window 1.6 (to treat a set of member like one single member) or as general in windows 1.7 and 1.8 (default). If the latter option *Do not use member-like input* is set, the support conditions have to be defined in window 1.7 for the sets of members.

With the option *Use for all sets of members*, you can define all stability data for sets of members in window 1.6 analogically to window 1.5 for single members. In this case, windows 1.7 and 1.8 are not displayed. The default simple girder values are used to determine the support conditions β, u_y, φ_x, φ_z and ω.

It is possible to use the member-like input *only for straight sets of members* with equal cross-section parameters. Windows 1.7 and 1.8 won't be displayed for straight sets. This option can be used e.g. for continuous beams.

The fourth option applies the member-like input only to *straight sets of members without intermediate restraints* modeled in RFEM. Thus, only sets of members which have RFEM supports/restraints at their ends will be considered for the member-like input. This option can be used to design e.g. simple beams or cantilevers. The connection of transverse beams to the intermediate nodes of the set is not accounted for, however. Windows 1.7 and 1.8 won't be displayed for straight sets that have no intermediate restraints.

Welded I- and H-Section Fabrication

For I- and H-sections, *Rolled* or *Flame-cut* fabrication methods are possible. The member section constant *n* as specified in [1] clause 3.2.2.1(a) depends on this type of fabrication. The selected type is then applied to <u>all</u> I and H sections of the design case.

Limit Load for Special Cases

To design cross-sections for intended axial compression according to [1] clause 3.2, it is possible to neglect *small moments* about the major and the minor axes by 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 P_u to F_R R_p.

Intended *torsion* is not clearly specified in [1]. If a torsional stress is available that is not exceeding the shear stress ratio of 1 % preset by default, it is not considered in the stability design. In this case, the output shows results 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. Nevertheless, the cross-section checks are run independently. These limit settings are not part of the Mexican standard. Changing the limits is in the responsibility of the program user.

3.1.3 Serviceability

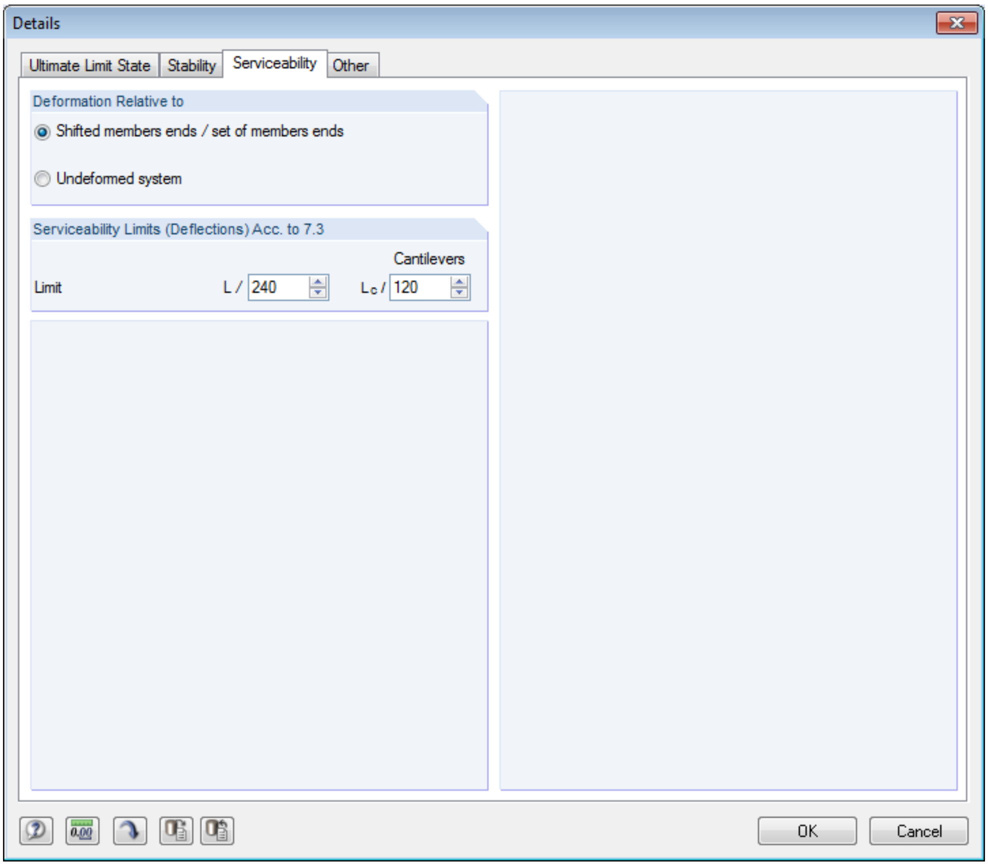


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Relative to

The 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 have to be checked relative to the displacements in the entire structural system.

Serviceability Limits (Deflections)

Here you can check and, if necessary, adjust the limit deformations of beams and cantilevers.

3.1.4 Other

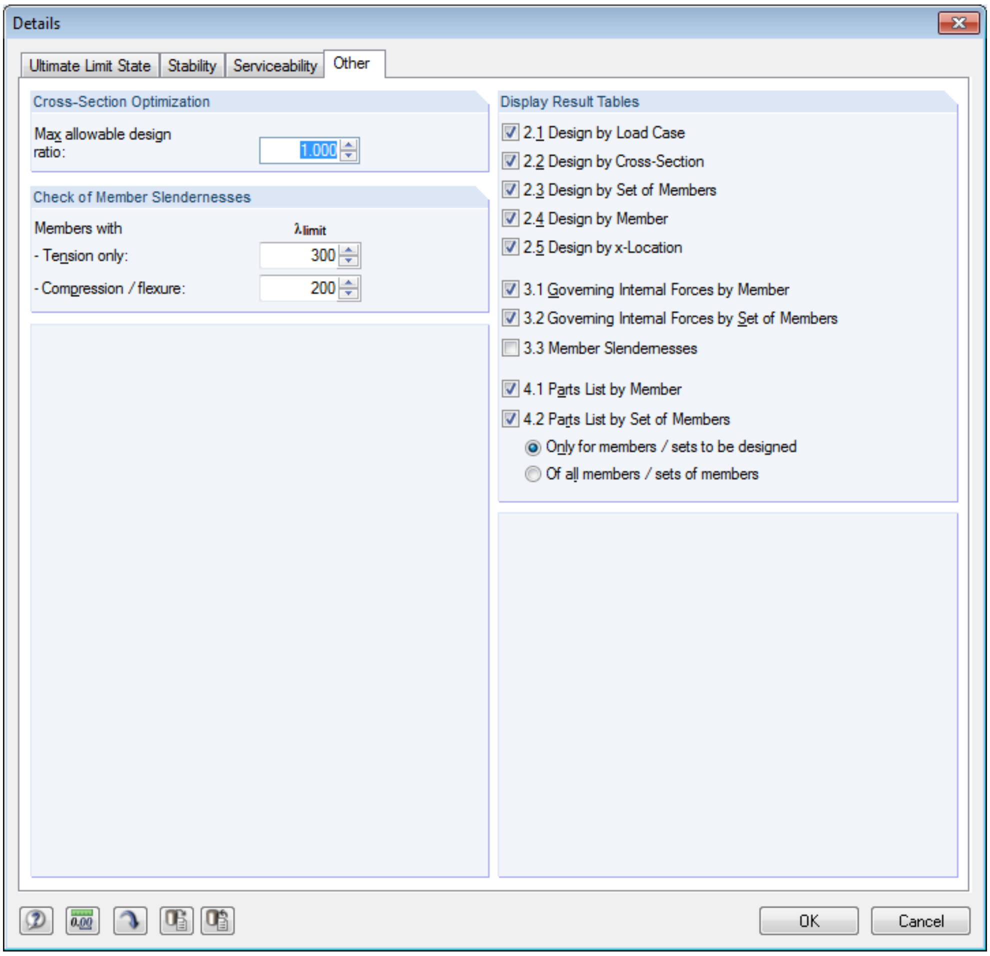


Figure 3.4: Dialog box *Details*, tab *Other*

Cross-Section Optimization

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

Check of Member Slendernesses

In the two input fields, you can specify the limit values λ_{limit} (the ratios $K \cdot L / r$) in order to define member slendernesses. It is possible to enter specifications separately for members with pure tension forces and members with bending and compression. The default values are given in [1] clause 2.2.3.

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 40) 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 be displayed. 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 NTC-DF add-on module.

RF-STEEL NTC-DF searches for the results of the load cases, load combinations, and result combinations to be designed. If these 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: The dialog box *To Calculate* (menu *Calculate* → *To Calculate*) lists design cases of the add-on modules like load cases and load combinations are listed.

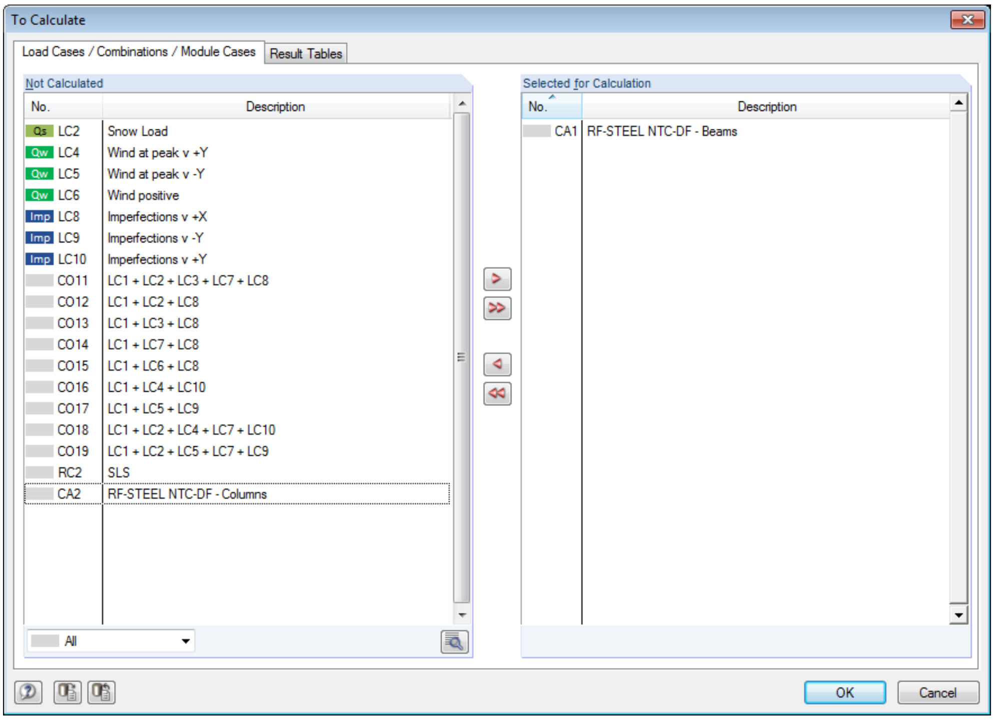
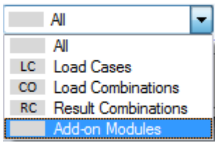


Figure 3.5: Dialog box *To Calculate*



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

To transfer the selected RF-STEEL NTC-DF 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 NTC-DF case in the toolbar list, and then click [Show Results].

Figure 3.6: Direct calculation of a RF-STEEL NTC-DF 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.

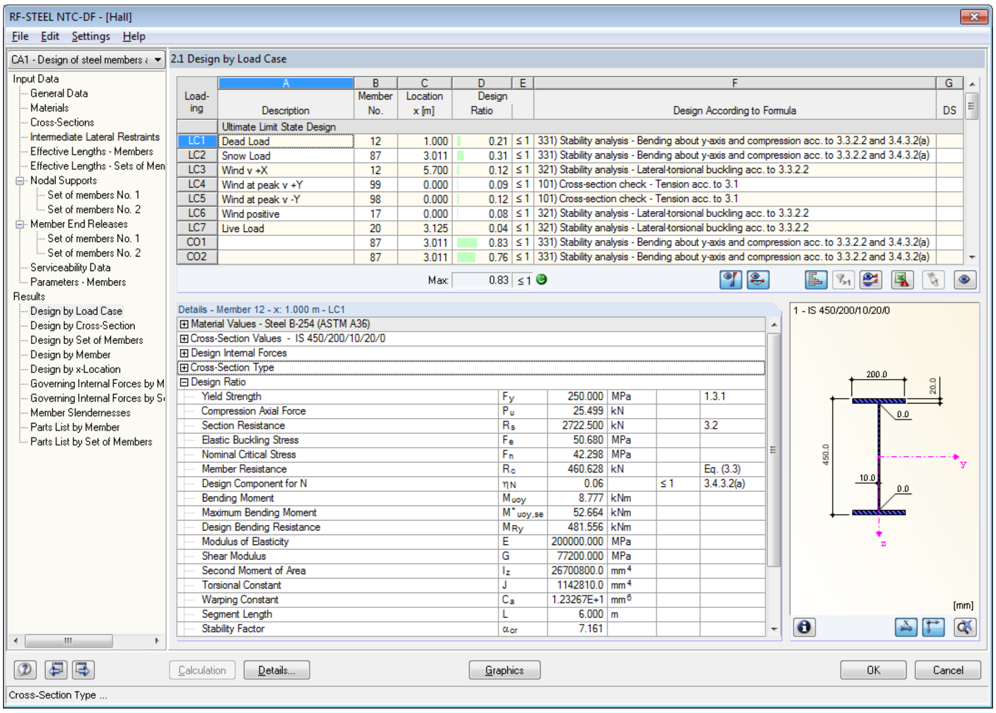
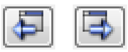


Figure 4.1: Results window with designs and intermediate values

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

The 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. You exit RF-STEEL NTC-DF and return to the main program.

Chapter 4 *Results* describes the different results windows one by one. Evaluating and checking results is described in chapter 5 *Results Evaluation*, page 43.

Dlubal

4.1 Design by Load Case

The upper part of the window provides a summary, sorted by load cases, load combinations, and result combinations of the governing designs. Furthermore, the list is divided in ultimate limit state, serviceability and stability designs.

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

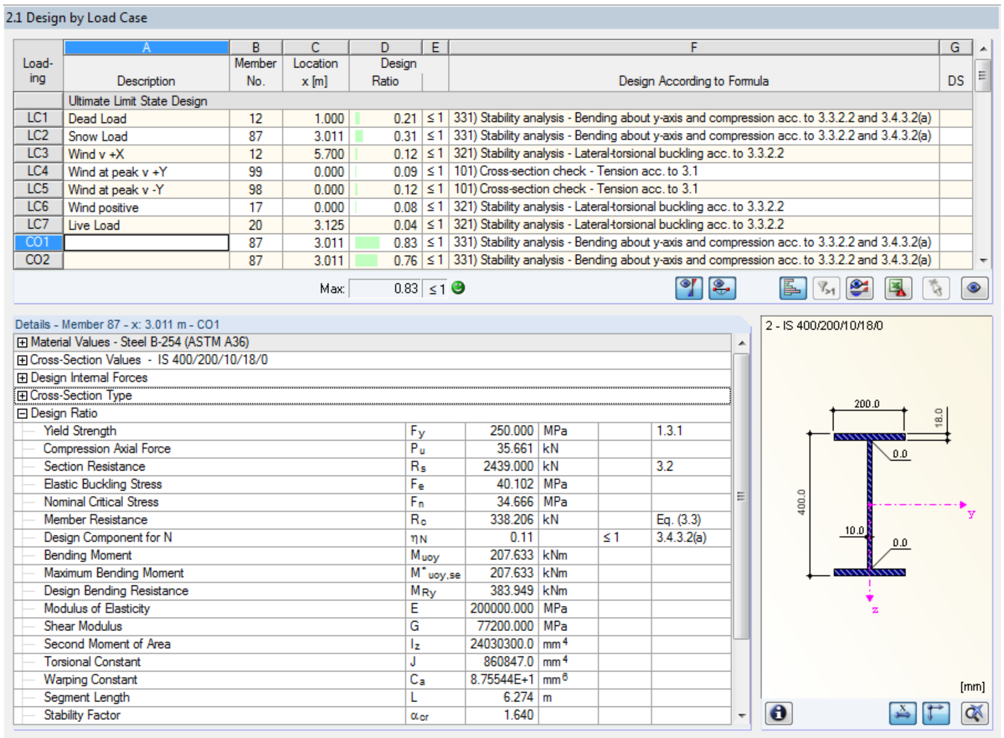


Figure 4.2: Window 2.1 *Design by Load Case*

Description

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

Member No.

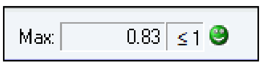
This column shows the number of the member that bears the maximum stress ratio of the designed loading.

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 Register Parameters*)
- Extreme values of internal forces

Design



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

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

Design according to Formula

This column lists the code's equations by which the designs have been performed.

4.2 Design by Cross-Section

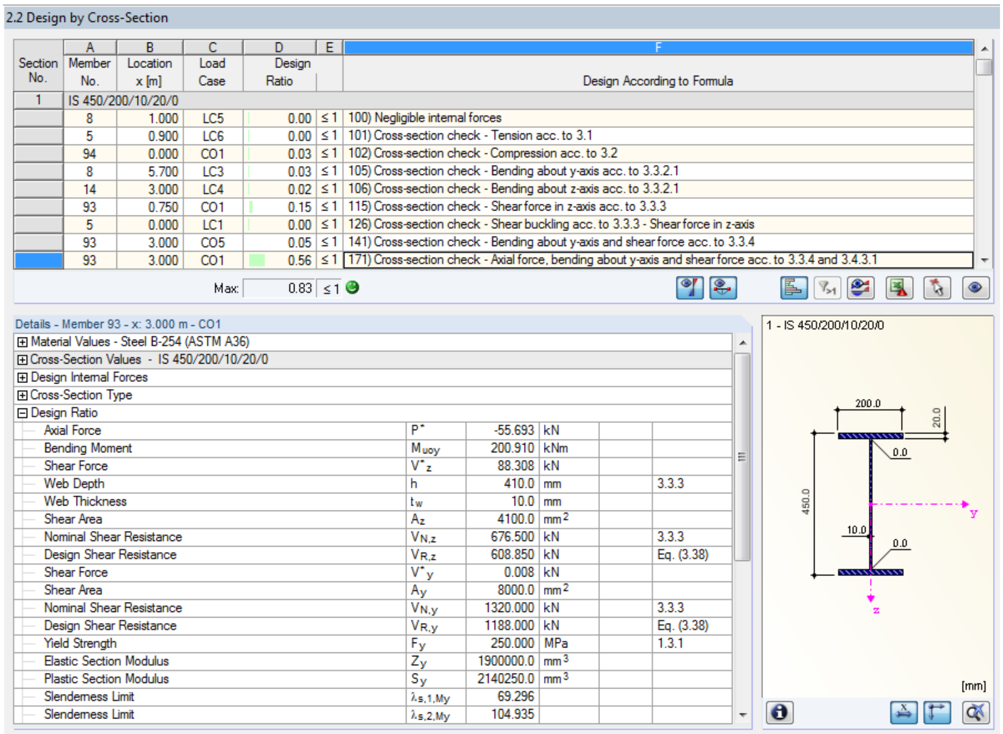


Figure 4.3: Window 2.2 *Design by Cross-Section*

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, stability analysis and serviceability limit state design.

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

4.3 Design by Set of Members

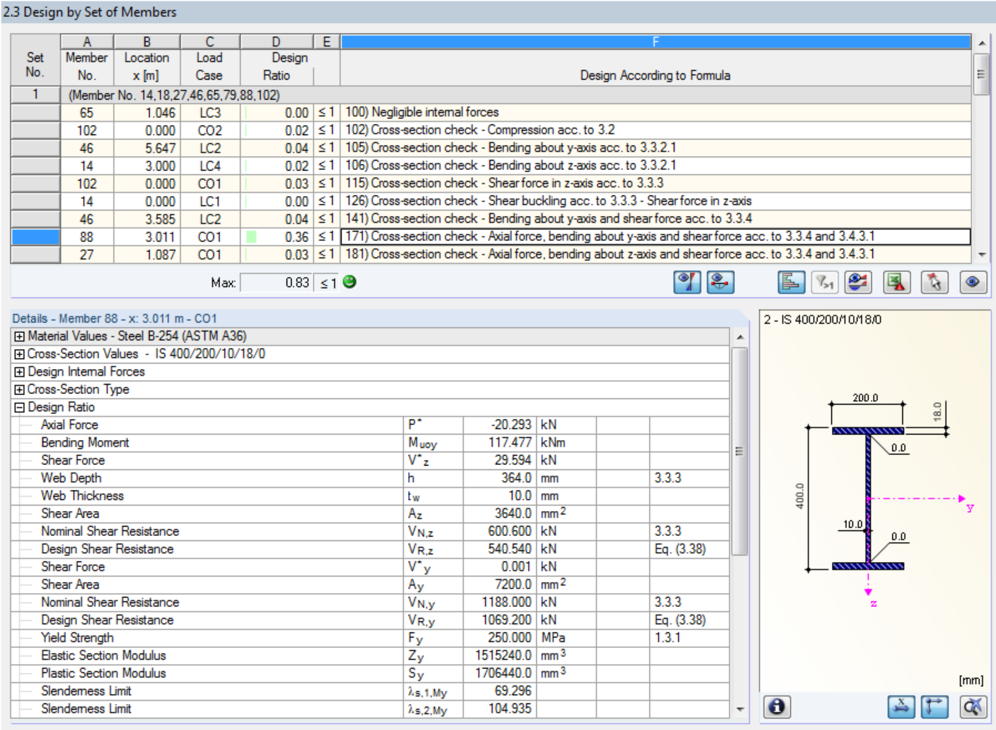


Figure 4.4: Window 2.3 *Design by Set of Members*

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

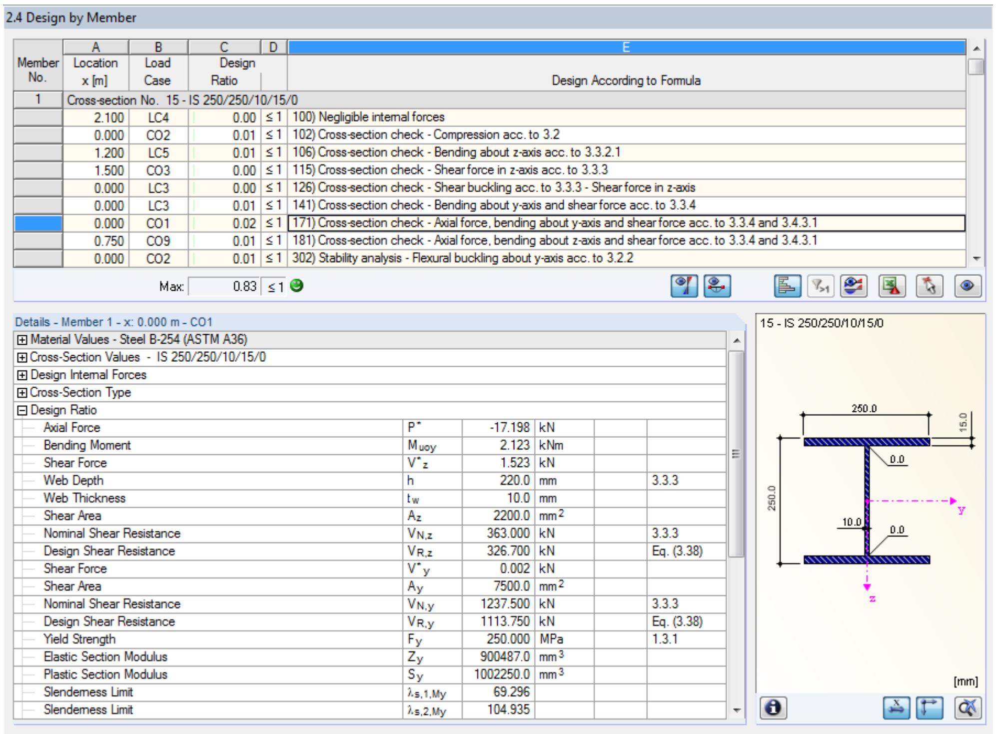


Figure 4.5: Window 2.4 *Design by Member*

This results window presents the maximum utilization ratios for the individual designs sorted by member number. The columns are described in detail in chapter 4.1 on page 34.

4.5 Design by x-Location

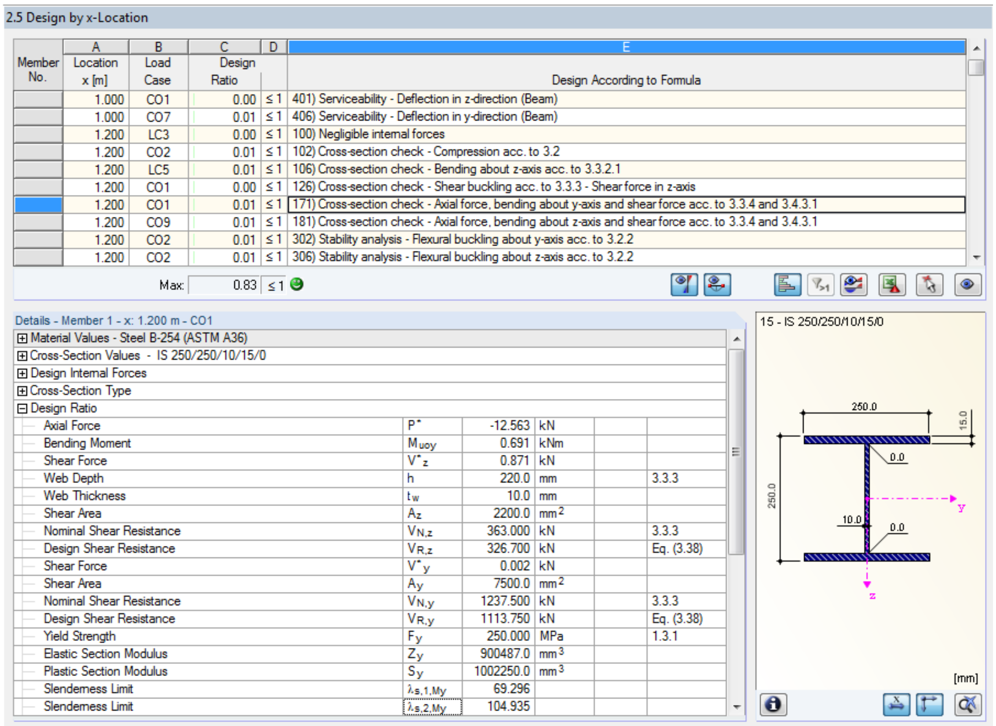
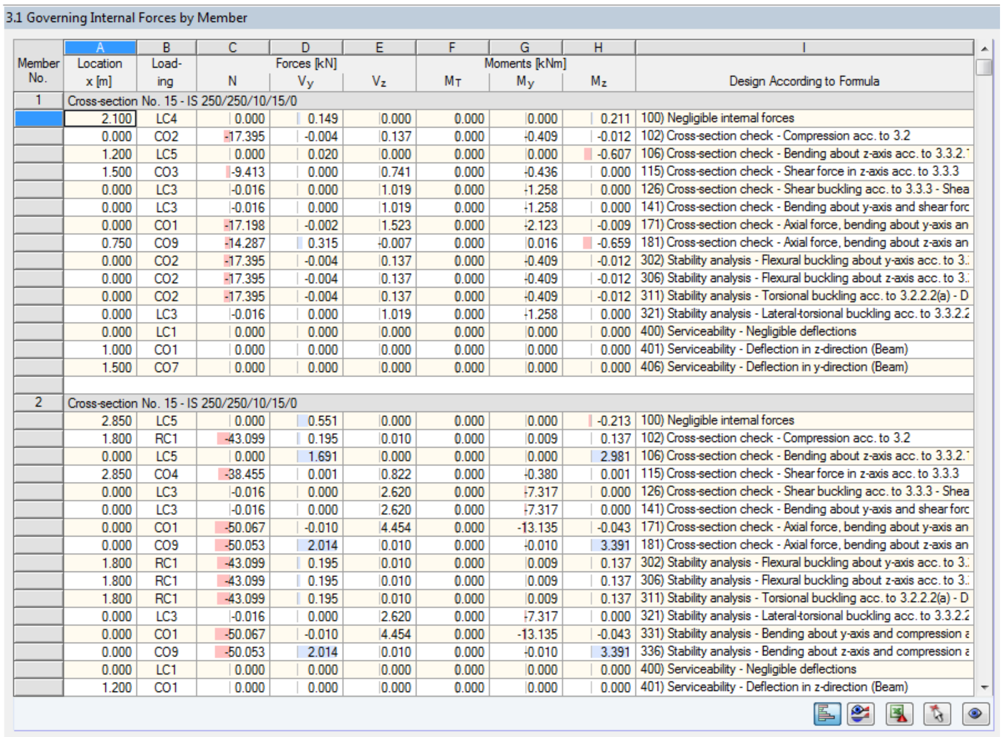


Figure 4.6: Window 2.5 *Design by x-Location*

This results window lists the maxima for each member at the locations ***x*** resulting from the division points 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 Register Parameters)*
- Extreme values of internal forces

4.6 Governing Internal Forces by Member


3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Loading	C Forces [kN] N	D V_y	E V_z	F Moments [kNm] M_T	G M_y	H M_z	I Design According to Formula
1	Cross-section No. 15 - IS 250/250/10/15/0								
	2.100	LC4	0.000	0.149	0.000	0.000	0.000	0.211	100) Negligible internal forces
	0.000	CO2	-17.395	-0.004	0.137	0.000	-0.409	-0.012	102) Cross-section check - Compression acc. to 3.2
	1.200	LC5	0.000	0.020	0.000	0.000	0.000	-0.607	106) Cross-section check - Bending about z-axis acc. to 3.3.2.1
	1.500	CO3	-9.413	0.000	0.741	0.000	-0.436	0.000	115) Cross-section check - Shear force in z-axis acc. to 3.3.3
	0.000	LC3	-0.016	0.000	1.019	0.000	-1.258	0.000	126) Cross-section check - Shear buckling acc. to 3.3.3 - Shea
	0.000	LC3	-0.016	0.000	1.019	0.000	-1.258	0.000	141) Cross-section check - Bending about y-axis and shear forc
	0.000	CO1	-17.198	-0.002	1.523	0.000	-2.123	-0.009	171) Cross-section check - Axial force, bending about y-axis an
	0.750	CO9	-14.287	0.315	-0.007	0.000	0.016	-0.659	181) Cross-section check - Axial force, bending about z-axis an
	0.000	CO2	-17.395	-0.004	0.137	0.000	-0.409	-0.012	302) Stability analysis - Flexural buckling about y-axis acc. to 3.
	0.000	CO2	-17.395	-0.004	0.137	0.000	-0.409	-0.012	306) Stability analysis - Flexural buckling about z-axis acc. to 3.
	0.000	CO2	-17.395	-0.004	0.137	0.000	-0.409	-0.012	311) Stability analysis - Torsional buckling acc. to 3.2.2.2(a) - D
	0.000	LC3	-0.016	0.000	1.019	0.000	-1.258	0.000	321) Stability analysis - Lateral-torsional buckling acc. to 3.3.2.2
	0.000	LC1	0.000	0.000	0.000	0.000	0.000	0.000	400) Serviceability - Negligible deflections
	1.000	CO1	0.000	0.000	0.000	0.000	0.000	0.000	401) Serviceability - Deflection in z-direction (Beam)
	1.500	CO7	0.000	0.000	0.000	0.000	0.000	0.000	406) Serviceability - Deflection in y-direction (Beam)
2	Cross-section No. 15 - IS 250/250/10/15/0								
	2.850	LC5	0.000	0.551	0.000	0.000	0.000	-0.213	100) Negligible internal forces
	1.800	RC1	-43.099	0.195	0.010	0.000	0.009	0.137	102) Cross-section check - Compression acc. to 3.2
	0.000	LC5	0.000	1.691	0.000	0.000	0.000	2.981	106) Cross-section check - Bending about z-axis acc. to 3.3.2.1
	2.850	CO4	-38.455	0.001	0.822	0.000	-0.380	0.001	115) Cross-section check - Shear force in z-axis acc. to 3.3.3
	0.000	LC3	-0.016	0.000	2.620	0.000	-7.317	0.000	126) Cross-section check - Shear buckling acc. to 3.3.3 - Shea
	0.000	LC3	-0.016	0.000	2.620	0.000	-7.317	0.000	141) Cross-section check - Bending about y-axis and shear forc
	0.000	CO1	-50.067	-0.010	4.454	0.000	-13.135	-0.043	171) Cross-section check - Axial force, bending about y-axis an
	0.000	CO9	-50.053	2.014	0.010	0.000	-0.010	3.391	181) Cross-section check - Axial force, bending about z-axis an
	1.800	RC1	-43.099	0.195	0.010	0.000	0.009	0.137	302) Stability analysis - Flexural buckling about y-axis acc. to 3.
	1.800	RC1	-43.099	0.195	0.010	0.000	0.009	0.137	306) Stability analysis - Flexural buckling about z-axis acc. to 3.
	1.800	RC1	-43.099	0.195	0.010	0.000	0.009	0.137	311) Stability analysis - Torsional buckling acc. to 3.2.2.2(a) - D
	0.000	LC3	-0.016	0.000	2.620	0.000	-7.317	0.000	321) Stability analysis - Lateral-torsional buckling acc. to 3.3.2.2
	0.000	CO1	-50.067	-0.010	4.454	0.000	-13.135	-0.043	331) Stability analysis - Bending about y-axis and compression a
	0.000	CO9	-50.053	2.014	0.010	0.000	-0.010	3.391	336) Stability analysis - Bending about z-axis and compression a
	0.000	LC1	0.000	0.000	0.000	0.000	0.000	0.000	400) Serviceability - Negligible deflections
	1.200	CO1	0.000	0.000	0.000	0.000	0.000	0.000	401) Serviceability - Deflection in z-direction (Beam)

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

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

Location x

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

Loading

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

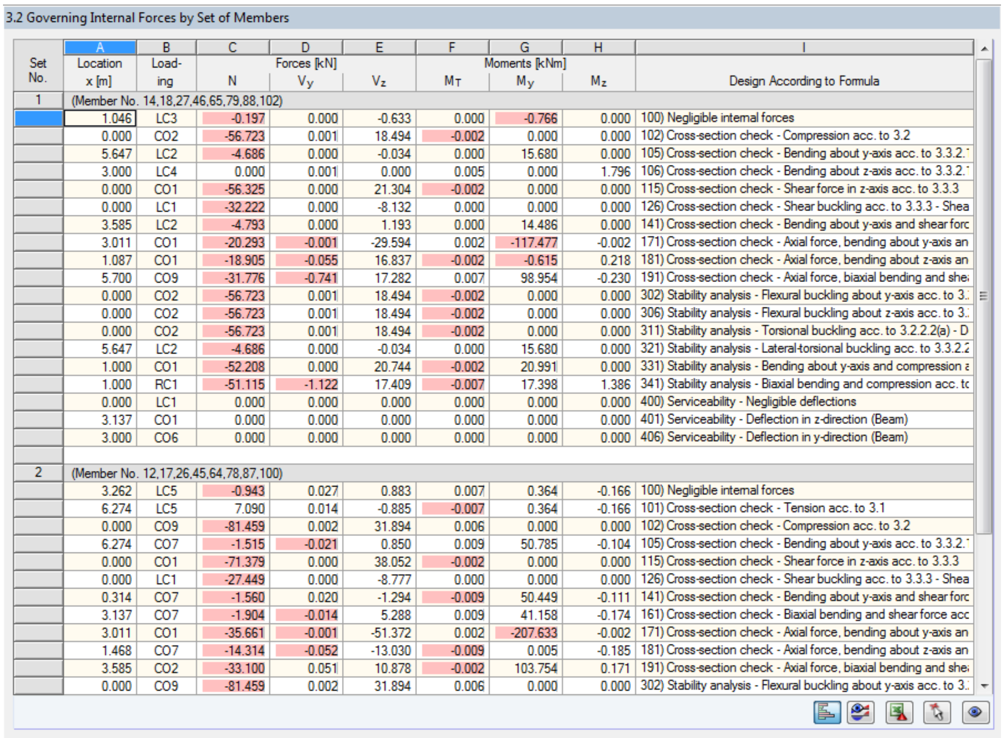
Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing maximum 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 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 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	(Member No. 14,18,27,46,65,79,88,102)								
	1.046	LC3	-0.197	0.000	-0.633	0.000	-0.766	0.000	100) Negligible internal forces
	0.000	CO2	-56.723	0.001	18.494	-0.002	0.000	0.000	102) Cross-section check - Compression acc. to 3.2
	5.647	LC2	-4.686	0.000	-0.034	0.000	15.680	0.000	105) Cross-section check - Bending about y-axis acc. to 3.3.2.1
	3.000	LC4	0.000	0.001	0.000	0.005	0.000	1.796	106) Cross-section check - Bending about z-axis acc. to 3.3.2.1
	0.000	CO1	-56.325	0.000	21.304	-0.002	0.000	0.000	115) Cross-section check - Shear force in z-axis acc. to 3.3.3
	0.000	LC1	-32.222	0.000	-8.132	0.000	0.000	0.000	126) Cross-section check - Shear buckling acc. to 3.3.3 - Shea
	3.585	LC2	-4.793	0.000	1.193	0.000	14.486	0.000	141) Cross-section check - Bending about y-axis and shear forc
	3.011	CO1	-20.293	-0.001	-29.594	0.002	-117.477	-0.002	171) Cross-section check - Axial force, bending about y-axis an
	1.087	CO1	-18.905	-0.055	16.837	-0.002	-0.615	0.218	181) Cross-section check - Axial force, bending about z-axis an
	5.700	CO9	-31.776	-0.741	17.282	0.007	98.954	-0.230	191) Cross-section check - Axial force, biaxial bending and she
	0.000	CO2	-56.723	0.001	18.494	-0.002	0.000	0.000	302) Stability analysis - Flexural buckling about y-axis acc. to 3.
	0.000	CO2	-56.723	0.001	18.494	-0.002	0.000	0.000	306) Stability analysis - Flexural buckling about z-axis acc. to 3.
	0.000	CO2	-56.723	0.001	18.494	-0.002	0.000	0.000	311) Stability analysis - Torsional buckling acc. to 3.2.2.2(a) - D
	5.647	LC2	-4.686	0.000	-0.034	0.000	15.680	0.000	321) Stability analysis - Lateral-torsional buckling acc. to 3.3.2.2
	1.000	CO1	-52.208	0.000	20.744	-0.002	20.991	0.000	331) Stability analysis - Bending about y-axis and compression a
	1.000	RC1	-51.115	-1.122	17.409	-0.007	17.398	1.386	341) Stability analysis - Biaxial bending and compression acc. to
	0.000	LC1	0.000	0.000	0.000	0.000	0.000	0.000	400) Serviceability - Negligible deflections
	3.137	CO1	0.000	0.000	0.000	0.000	0.000	0.000	401) Serviceability - Deflection in z-direction (Beam)
	3.000	CO6	0.000	0.000	0.000	0.000	0.000	0.000	406) Serviceability - Deflection in y-direction (Beam)
2	(Member No. 12,17,26,45,64,78,87,100)								
	3.262	LC5	-0.943	0.027	0.883	0.007	0.364	-0.166	100) Negligible internal forces
	6.274	LC5	7.090	0.014	-0.885	-0.007	0.364	-0.166	101) Cross-section check - Tension acc. to 3.1
	0.000	CO9	-81.459	0.002	31.894	0.006	0.000	0.000	102) Cross-section check - Compression acc. to 3.2
	6.274	CO7	-1.515	-0.021	0.850	0.009	50.785	-0.104	105) Cross-section check - Bending about y-axis acc. to 3.3.2.1
	0.000	CO1	-71.379	0.000	38.052	-0.002	0.000	0.000	115) Cross-section check - Shear force in z-axis acc. to 3.3.3
	0.000	LC1	-27.449	0.000	-8.777	0.000	0.000	0.000	126) Cross-section check - Shear buckling acc. to 3.3.3 - Shea
	0.314	CO7	-1.560	0.020	-1.294	-0.009	50.449	-0.111	141) Cross-section check - Bending about y-axis and shear forc
	3.137	CO7	-1.904	-0.014	5.288	0.009	41.158	-0.174	161) Cross-section check - Biaxial bending and shear force acc
	3.011	CO1	-35.661	-0.001	-51.372	0.002	-207.633	-0.002	171) Cross-section check - Axial force, bending about y-axis an
	1.468	CO7	-14.314	-0.052	-13.030	-0.009	0.005	-0.185	181) Cross-section check - Axial force, bending about z-axis an
	3.585	CO2	-33.100	0.051	10.878	-0.002	103.754	0.171	191) Cross-section check - Axial force, biaxial bending and she
	0.000	CO9	-81.459	0.002	31.894	0.006	0.000	0.000	302) Stability analysis - Flexural buckling about y-axis acc. to 3.

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	B	C	D	E	F	G	H	I
	Under Stress	Length L [m]	Major Axis y k_y [-]	i_y [mm]	λ_y [-]	Minor Axis z k_z [-]	i_z [mm]	λ_z [-]	
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	
14	Compression / Flexure	6.000	1.000	188.0	31.921	1.000	47.0	127.727	
15	Compression / Flexure	3.011	1.000	167.2	18.010	1.000	47.1	63.958	
16	Compression / Flexure	3.011	1.000	167.2	18.010	1.000	47.1	63.958	
17	Compression / Flexure	3.011	1.000	167.2	18.010	1.000	47.1	63.958	
18	Compression / Flexure	3.011	1.000	167.2	18.010	1.000	47.1	63.958	
19	Compression / Flexure	6.274	1.000	107.7	58.240	1.000	63.5	98.840	
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	167.2	19.512	1.000	47.1	69.292	
25	Compression / Flexure	3.262	1.000	167.2	19.512	1.000	47.1	69.292	
26	Compression / Flexure	3.262	1.000	167.2	19.512	1.000	47.1	69.292	
27	Compression / Flexure	3.262	1.000	167.2	19.512	1.000	47.1	69.292	
28	Compression / Flexure	3.546	1.000	107.7	32.918	1.000	63.5	55.865	
29	Compression / Flexure	3.000	1.000	107.7	27.849	1.000	63.5	47.263	
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	

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 appears only if you select the respective check box in the *Other* tab of the *Details* dialog box (see Figure 3.4, page 31).

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, tab *Other* (see Figure 3.4, page 31).

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

This window is displayed only for information. No stability design of slendernesses is intended.

4.9 Parts List by Member

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

4.1 Parts List by Member

	A	B	C	D	E	F	G	H	I
Part No.	Cross-Section Description	Number of Members	Length [m]	Total Length [m]	Surface Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1	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	11.77	58.88	1.472
3	1 - IS 450/200/10/20/0	4	3.00	12.00	20.16	0.15	94.98	284.95	1.140
4	1 - IS 450/200/10/20/0	6	6.00	36.00	60.48	0.44	94.98	569.91	3.419
5	13 - Circle 24	4	7.81	31.24	2.36	0.01	3.55	27.74	0.111
6	2 - IS 400/200/10/18/0	8	3.01	24.09	38.06	0.26	85.09	256.25	2.050
7	7 - IS 250/250/10/15/0	4	6.27	25.10	37.14	0.24	76.15	477.72	1.911
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 - Circle 24	8	8.02	64.18	4.84	0.03	3.55	28.49	0.228
10	2 - IS 400/200/10/18/0	8	3.26	26.10	41.24	0.28	85.09	277.62	2.221
11	6 - IS 250/250/10/15/0	2	3.55	7.09	10.50	0.07	76.15	270.01	0.540
12	6 - IS 250/250/10/15/0	3	3.00	9.00	13.32	0.09	76.15	228.44	0.685
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 400/200/10/18/0	8	6.27	50.19	79.30	0.54	85.09	533.88	4.271
17	6 - IS 250/250/10/15/0	1	4.09	4.09	6.06	0.04	76.15	311.74	0.312
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 - IS 250/250/10/15/0	1	7.09	7.09	10.50	0.07	76.14	540.17	0.540
20	6 - IS 250/250/10/15/0	1	6.55	6.55	9.69	0.06	76.15	498.45	0.498
Sum		102		516.46	507.84	3.38			26.551

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, tab *Other* (see Figure 3.4, page 31).

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 through 2.5 in the cross-section information (see Figure 2.10, page 15).

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.

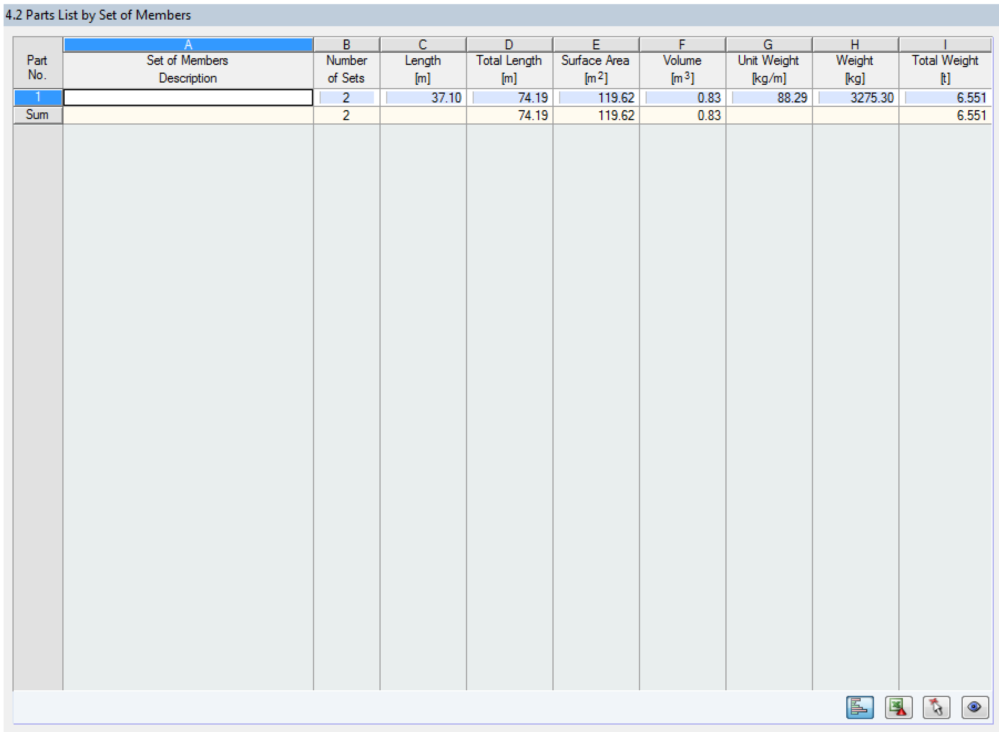
Total Weight

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 data field 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

	A	B	C	D	E	F	G	H	I
Part No.	Set of Members Description	Number of Sets	Length [m]	Total Length [m]	Surface Area [m^2]	Volume [m^3]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1		2	37.10	74.19	119.62	0.83	88.29	3275.30	6.551
Sum		2		74.19	119.62	0.83			6.551

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 window 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 there are different cross-sections in a set of members, the program averages the surface area, the volume, and the cross-section weight.

5. Results Evaluation

You can evaluate the design results in different ways. The buttons below the first window part can help you to evaluate the results.

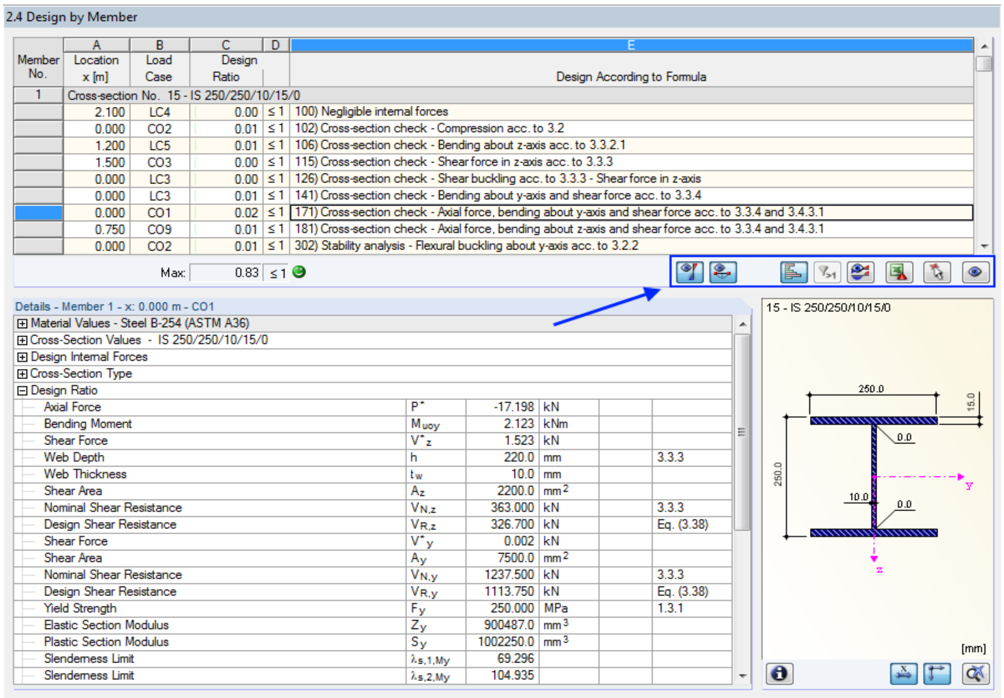


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Shows or hides the results of the ultimate limit state design
	Serviceability Limit State Designs	Shows or hides the results of the serviceability limit state design
	Show Color Bars	Shows or hides the colored relation scales in the results windows
	Show Rows with Ratio > 1	Displays only the rows where the ratio is greater than 1, and thus the design is failed
	Result Diagrams	Opens the window *Result Diagram on Member* → chapter 5.2, page 46
	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 to 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 NTC-DF 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.

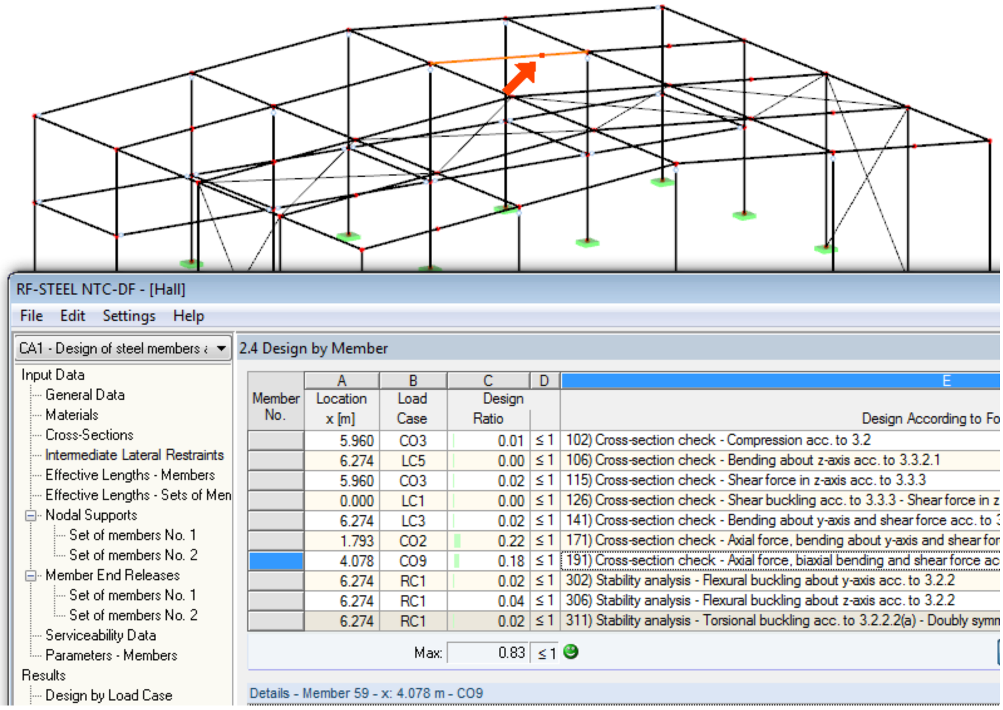


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model

If you can't improve the display by moving the RF-STEEL NTC-DF 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.

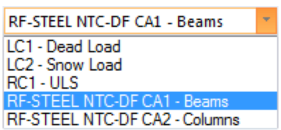
Click [Back] to return to the add-on module RF-STEEL NTC-DF.

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 and ultimate limit state or fire resistance 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, click the [Show Values] toolbar button to the right.



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

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

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.

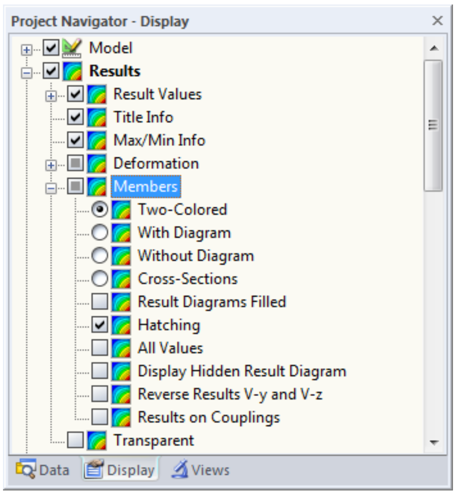


Figure 5.3: *Display* navigator: Results → Members

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.

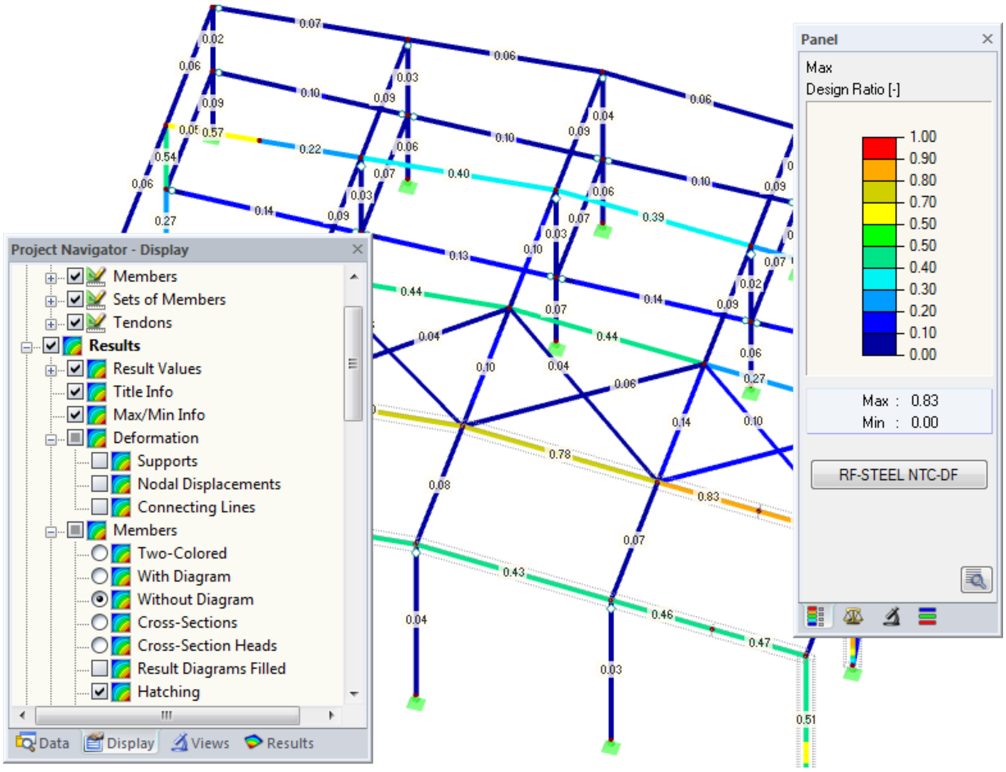


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see chapter 6.2, page 49).

To return to the RF-STEEL NTC-DF module, click [RF-STEEL NTC-DF] 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 NTC-DF 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 43).

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.

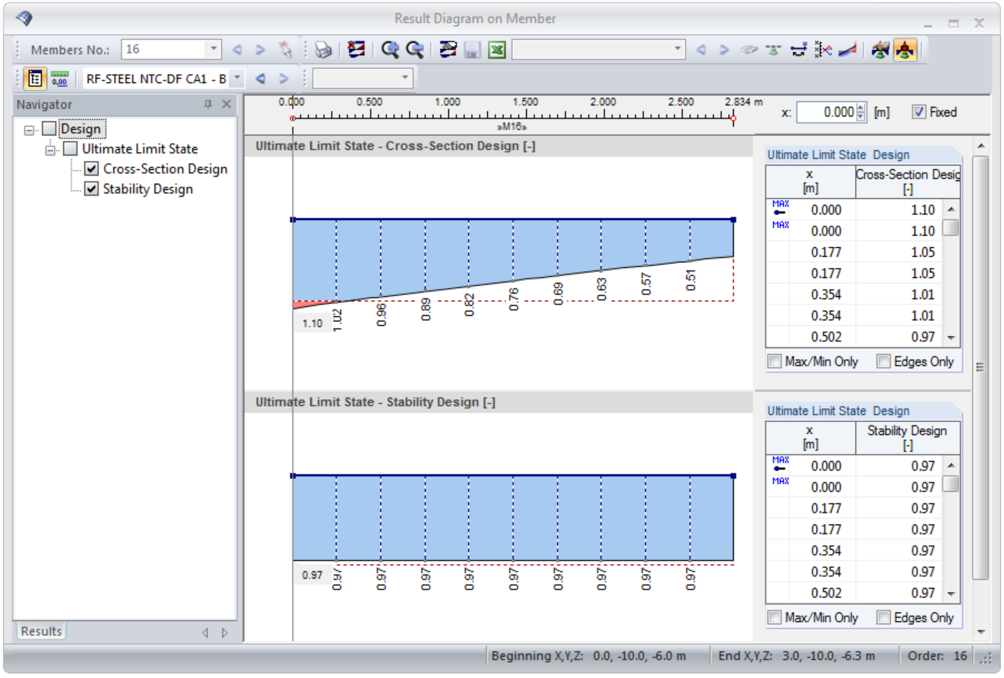
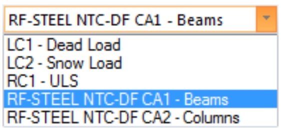


Figure 5.5: Dialog box *Result Diagram on Member*



Use the list in the toolbar above to choose the relevant RF-STEEL NTC-DF 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 NTC-DF 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* option also for RF-STEEL NTC-DF (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.

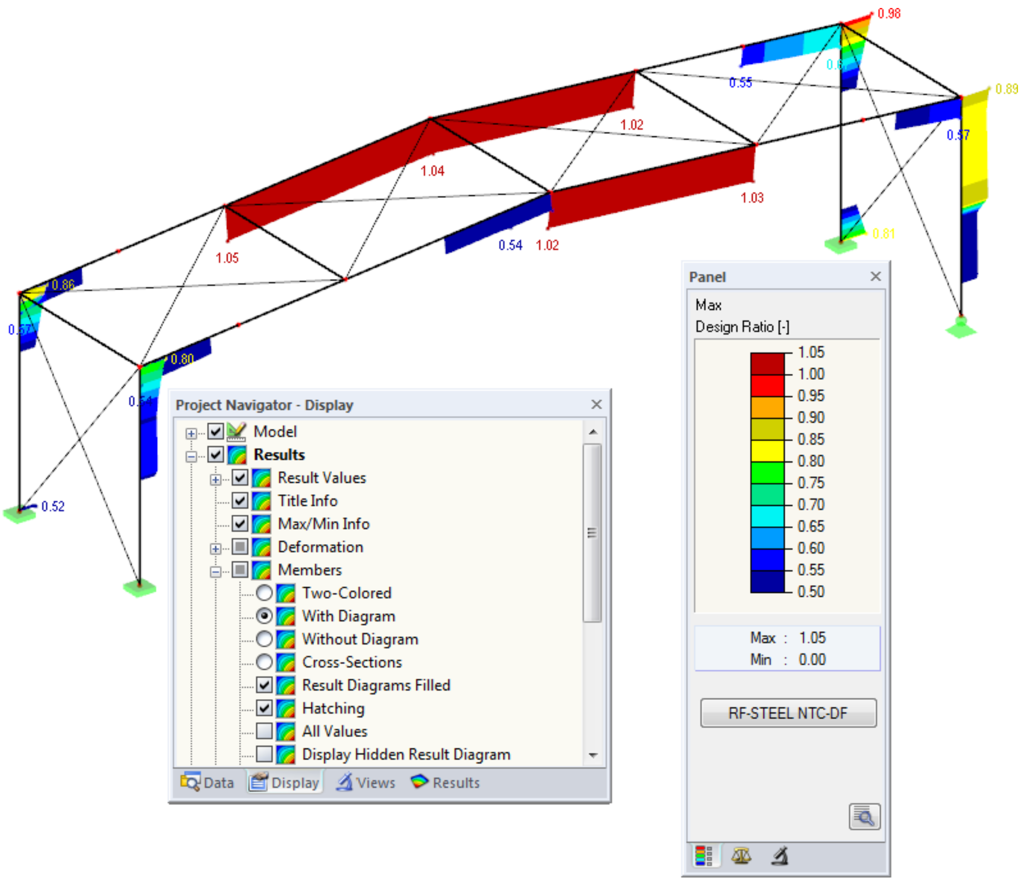


Figure 5.6: Filtering design ratios with adjusted color spectrum

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 design ratio diagrams that are not covered by the color spectrum. Those diagrams are represented by dotted lines.

Filtering members

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

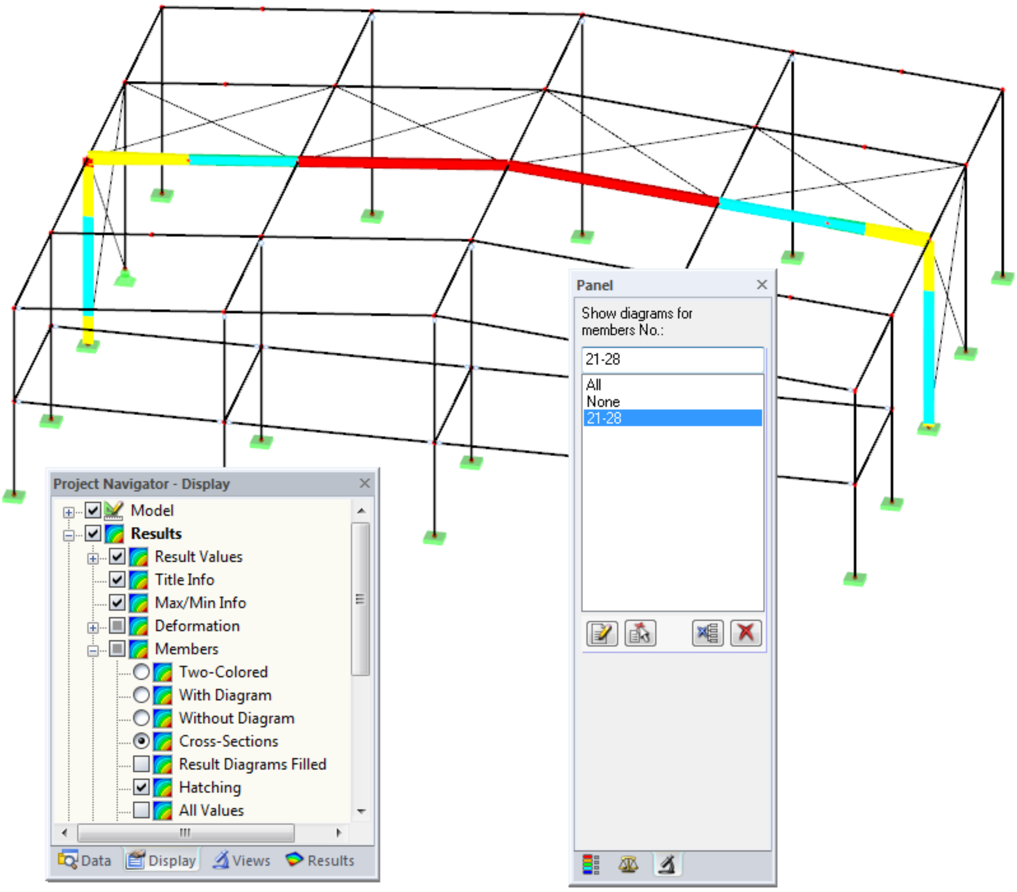


Figure 5.7: Member filter for the design ratios of a hall frame

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 NTC-DF 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 from add-on modules for the printout report.

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

6.2 RF-STEEL NTC-DF 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 on 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.

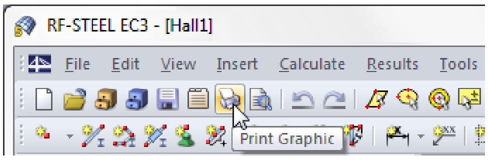


Figure 6.1: Button *Print Graphic* in RFEM toolbar

Result Diagrams

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

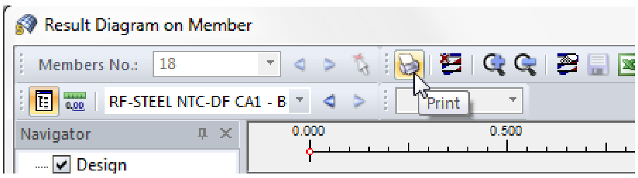


Figure 6.2: Button *Print Graphic* in the dialog box *Result Diagram on Member*

The *Graphic Printout* dialog box appears (see the following page).

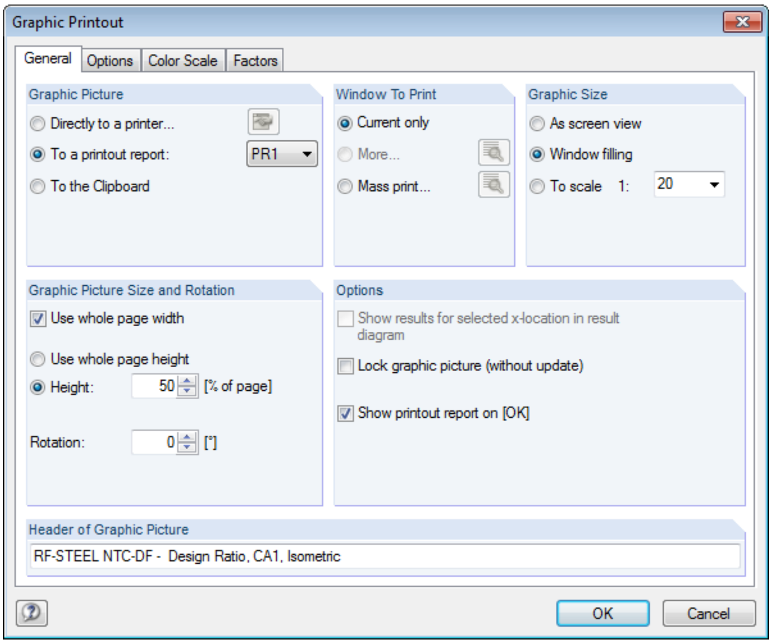
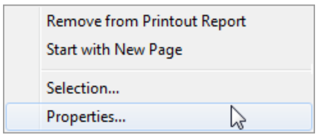


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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 *Properties* option in the context menu opens the *Graphic Printout* dialog box, offering various options for adjustment.

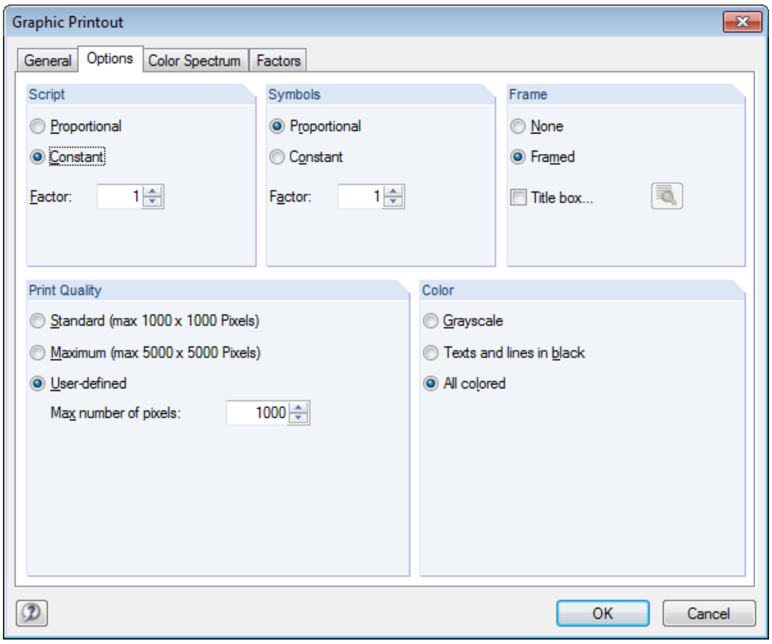


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

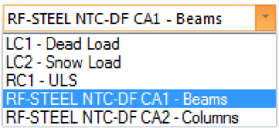
7. General Functions

The final 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 NTC-DF 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 NTC-DF menu and click

File → New Case.

The following dialog box appears:

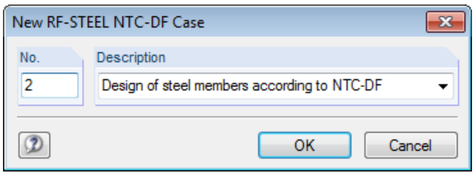


Figure 7.1: Dialog box *New RF-STEEL NTC-DF Case*

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 NTC-DF window 1.1 *General Data* where you can enter the design data.

Rename Design Case

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

File → Rename Case.

The following dialog box appears:

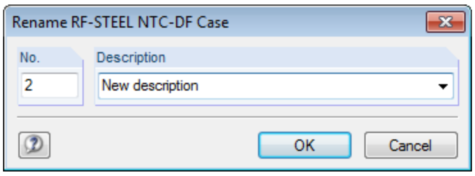


Figure 7.2: Dialog box *Rename RF-STEEL NTC-DF Case*

In this dialog box, you can define 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, select from the RF-STEEL NTC-DF menu

File → Copy Case.

The following dialog box appears:

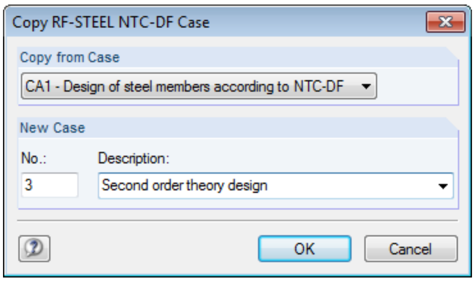


Figure 7.3: Dialog box *Copy RF-STEEL NTC-DF Case*

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

Delete Design Case

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

File → Delete Case.

The following dialog box appears:

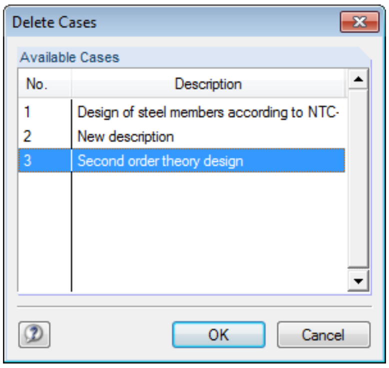
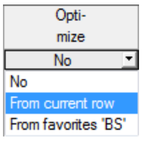


Figure 7.4: Dialog box *Delete Cases*

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 13). You can also start the cross-section optimization in the results windows by using the context menu.

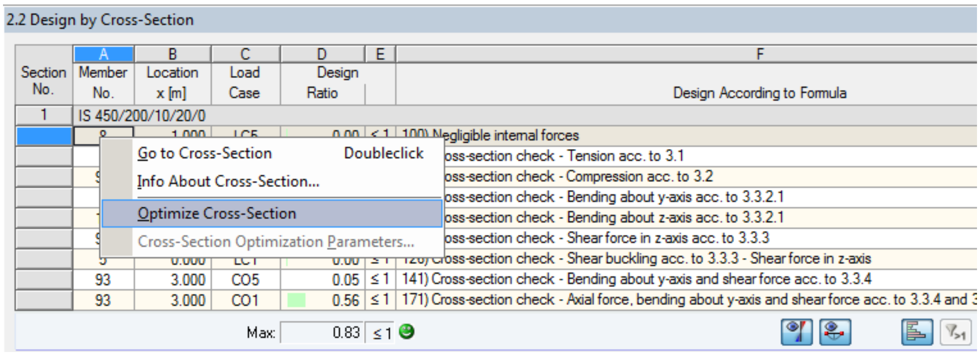


Figure 7.5: Context menu for cross-section optimization

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 31). 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.

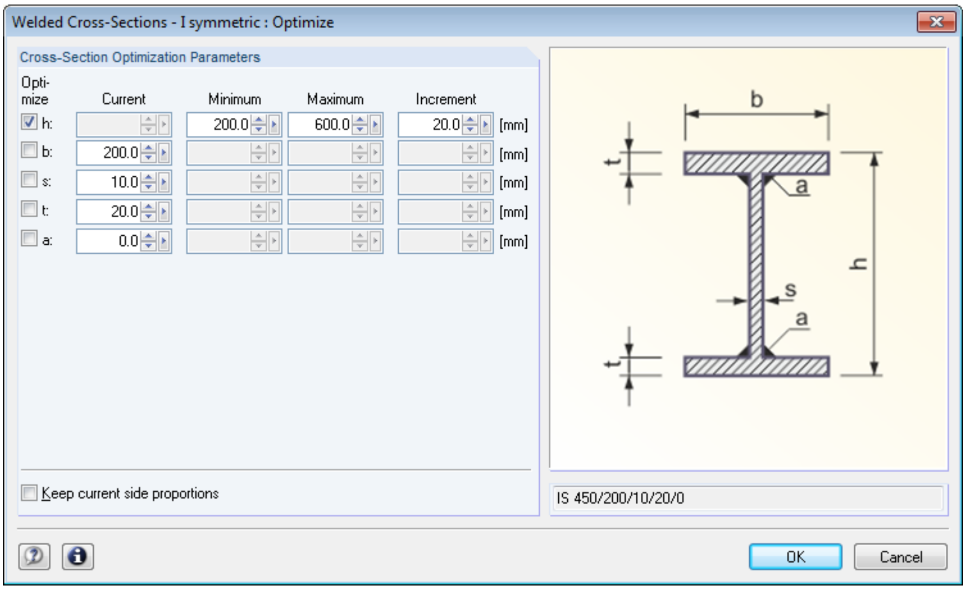


Figure 7.6: Dialog box *Welded Cross-Sections - I symmetric : Optimize*

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.

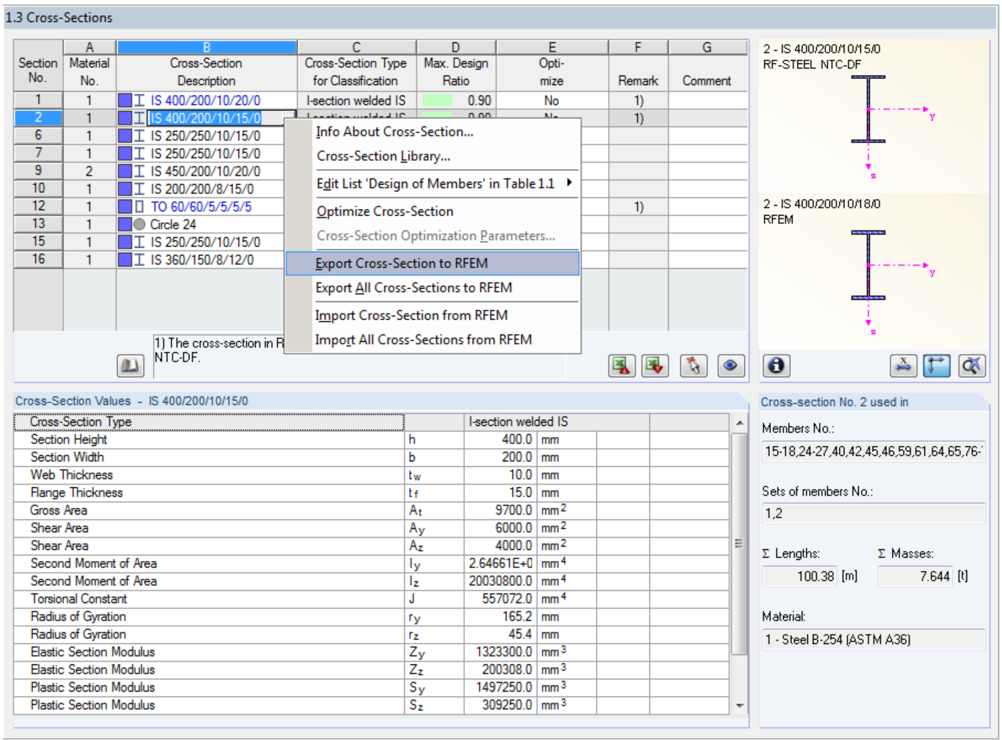


Figure 7.7: Context menu in window 1.3 *Cross-Sections*

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

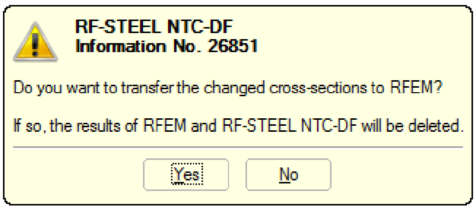


Figure 7.8: Query before transfer of modified cross-sections to RFEM

By confirming the query and then starting the [Calculation] in the RF-STEEL NTC-DF 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 reimport the original cross-sections in the design module by using the options shown in Figure 7.7. Please note that this option is only available in 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 modeling the taper layout manually.

7.3 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed in one dialog box. To define the units in RF-STEEL NTC-DF, use the menu and click

Settings → **Units and Decimal Places**.

The program opens the dialog box that is familiar from RFEM. RF-STEEL NTC-DF is preset in the *Program / Module* list.

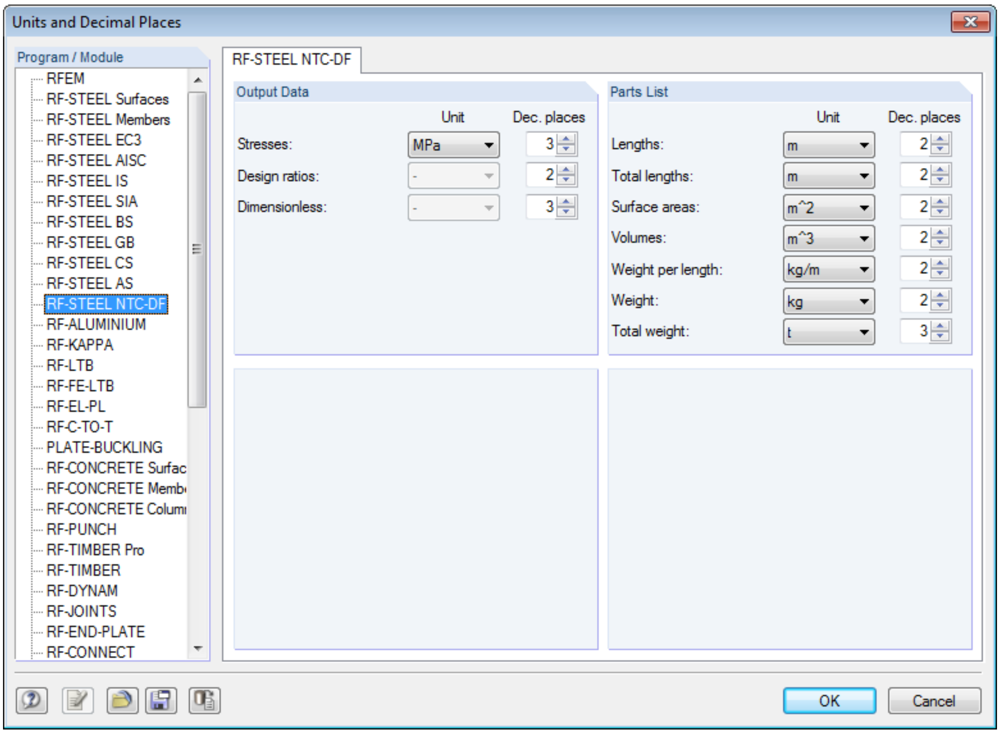


Figure 7.9: Dialog box *Units and Decimal Places*

You can save the settings as 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 NTC-DF for design, you can export the modified materials to RFEM in a similar manner as you can export cross-sections: Open the 1.2 *Materials* window, and then click

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

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

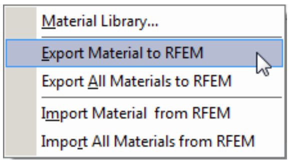


Figure 7.10: Context menu of window 1.2 *Materials*

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 NTC-DF, 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 the 1.2 *Materials* window.

7.4.2 Export Effective Lengths to RFEM

If you have adjusted the materials in RF-STEEL NTC-DF for design, you can export the modified materials to RFEM in a similar manner as you can export cross-sections: Open the 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.

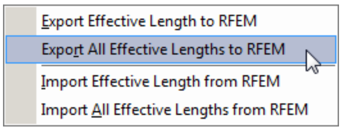


Figure 7.11: Context menu of window 1.5 *Effective Lengths - Members*

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 reimport 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 the windows 1.5 *Effective Lengths - Members* and 1.6 *Effective Lengths - Sets of Members*.

7.4.3 Export Results

The RF-STEEL NTC-DF 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 NTC-DF add-on module into the global printout report (see chapter 6.1, page 49) for export. Then, in the printout report, click

File → Export to RTF.

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

Excel / OpenOffice

RF-STEEL NTC-DF 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.

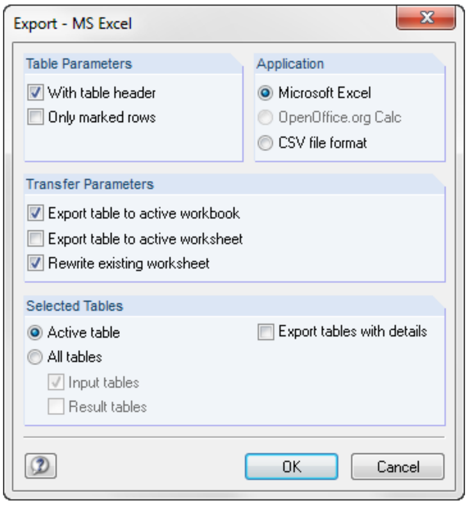


Figure 7.12: Dialog box *Export - MS Excel*

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

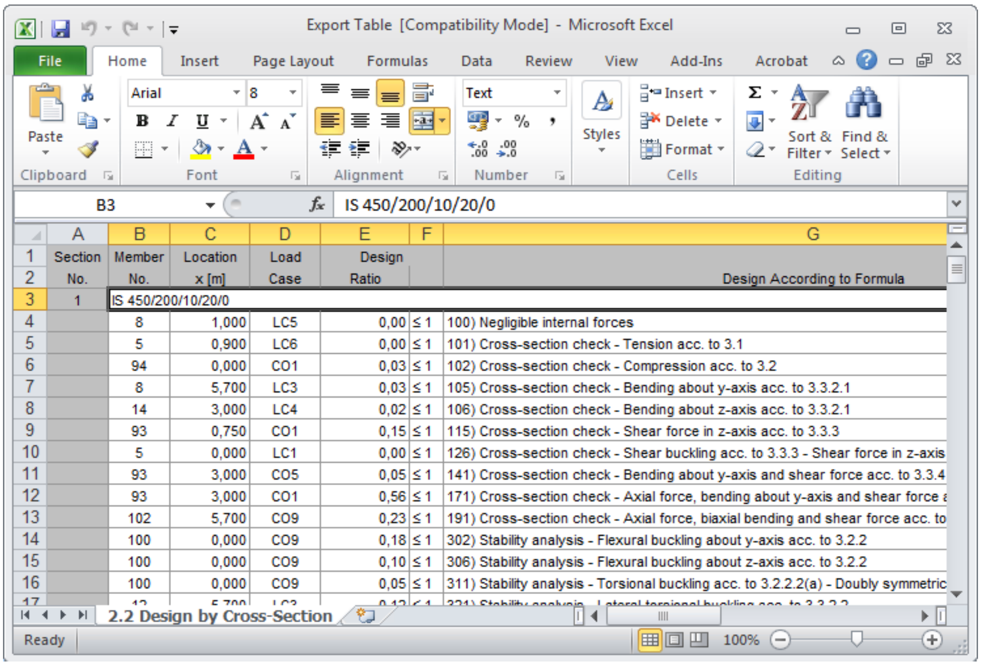


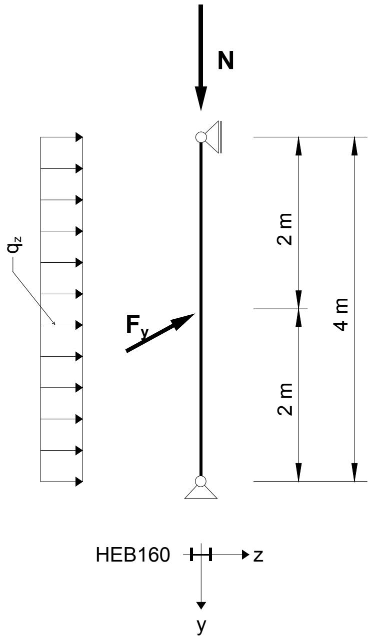
Figure 7.13: Result in *Excel*

8. Example

In our example, we perform the stability analyses of flexural buckling and lateral-torsional buckling for a column with double-bending, taking into account the interaction conditions.

Design values

System and loads



Design values of the static loads

N_d = 300 kN

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

$F_{y,d}$ = 7.5 kN

Figure 8.1: System and design loads (γ times)

Internal forces according to linear static analysis

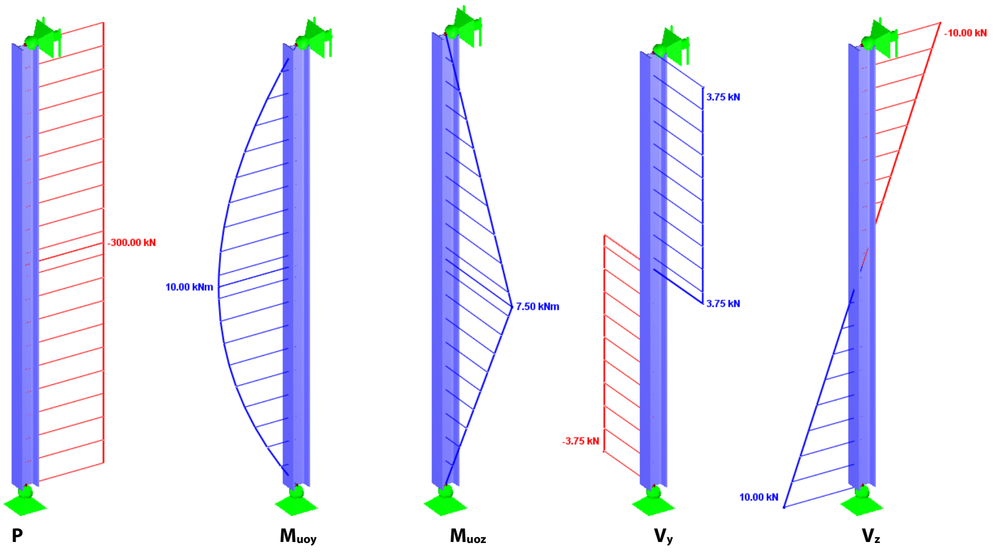


Figure 8.2: Internal forces

Design location (decisive x-location)

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

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

Cross-Section Properties HE-B 160, Steel B-254 (A36)

Property	**Symbol**	**Value**	**Unit**
Cross-section area	A_t	5425.00	mm^2
Moment of inertia	I_y	24920000.00	mm^4
Moment of inertia	I_z	8892000.00	mm^4
Governing radius of gyration	r_y	67.80	mm
Governing radius of gyration	r_z	40.50	mm
Polar radius of gyration	r_o	78.97	mm
Polar radius of gyration	$r_{o,M}$	419.00	mm
Cross-section mass	M	42.63	kg/m
Torsional constant	J	312400.00	mm^4
Warping constant	C_a	4.794E+10	mm^6
Elastic section modulus	S_y	311500.00	mm^3
Elastic section modulus	S_z	111200.00	mm^3
Plastic section modulus	Z_y	354000.00	mm^3
Plastic section modulus	Z_z	169960.00	mm^3

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

Flexural buckling critical stress

$$F_{E,z} = \frac{\pi^2 \cdot E}{(K_z \cdot L / r_z)^2} = \frac{\pi^2 \cdot 200000}{(1 \cdot 4000 / 40.50)^2} = 202.358 \text{MPa}$$

Flexural buckling critical load

$$P_{E,z} = A_t \frac{\pi^2 \cdot E}{(K_z \cdot L / r_z)^2} = 5425 \frac{\pi^2 \cdot 200000}{(1 \cdot 4000 / 40.50)^2} = 1097.79 \text{ kN}$$

Cross-section type acc. to [1] Table 2.1

"2 - compact": $A_e = A_t$

Slenderness parameter

$$\lambda_z = \frac{K_z \cdot L}{r_z} \sqrt{\left(\frac{F_y}{\pi^2 \cdot E}\right)} = \frac{1 \cdot 4000}{40.50} \sqrt{\left(\frac{250}{\pi^2 \cdot 200000}\right)} = 1.112$$

Constant *n* acc. to [1] clause 3.2.2.1(1)

n = 2.00 for hot-rolled I-sections ($F_y \le 414$ MPa, $t_f \le 50$ mm)

Design resistance in compression

$$R_{c,z}=\frac{F_y}{\left(1+\lambda_z^{2n}-0.15^{2n}\right)^{1/n}}A_t\cdot F_R=\frac{250}{\left(1+1.112^4-0.15^4\right)^{1/2}}5425\cdot 0.9=767.71\text{ kN}$$

Design ratio

$$\frac{P}{R_{c,z}}=\frac{300}{767.71}=\underline{\underline{0.39\leq 1}}$$

Result values from RF-STEEL NTC-DF calculation

Compression Axial Force	P_u	300.00	kN		
Section Resistance	R_s	1220.63	kN		3.2
Form Factor	A_e/A_t	1.000			2.1.3
Yield Strength	F_y	250.000	MPa		2.1
Modulus of Elasticity	E	200000	MPa		
Gross Area	A_t	5425	mm^2		
Second Moment of Area	I_z	8892000	mm^4		
Radius of Gyration	r_z	40.486	mm		
Effective Length	L_z	4000	mm		
Slenderness	λ_z	1.112			3.2.2
Elastic Flexural Buckling Stress	$F_{e,z}$	202.213	MPa		Eq. (3.12)
Factor	n	2.000			3.2.2
Nominal Critical Stress	$F_{n,z}$	157.236	MPa		
Resistance Factor	F_R	0.900			
Member Resistance	$R_{c,z}$	767.71	kN		Eq. (3.3)
Design Ratio	η	0.390		< 1	3.2.2

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

Flexural buckling critical stress

$$F_{E,y}=\frac{\pi^2\cdot E}{\left(K_y\cdot L/r_y\right)^2}=\frac{\pi^2\cdot 200000}{\left(1\cdot 4000/67.80\right)^2}=567.112\text{MPa}$$

Flexural buckling critical load

$$P_{E,y}=A_t\frac{\pi^2\cdot E}{\left(K_y\cdot L/r_y\right)^2}=5425\frac{\pi^2\cdot 200000}{\left(1\cdot 4000/67.80\right)^2}=3076.585\text{ kN}$$

Cross-section type acc. to Table 2.1

"2 - compact": $A_e = A_t$

Slenderness parameter

$$\lambda_y=\frac{K_y\cdot L}{r_y}\sqrt{\left(\frac{F_y}{\pi^2\cdot E}\right)}=\frac{1\cdot 4000}{67.80}\sqrt{\left(\frac{250}{\pi^2\cdot 200000}\right)}=0.664$$

Constant *n* acc. to clause 3.2.2.1(1)

n = 2.00 for hot-rolled I-section ($F_y \leq 414$ MPa, $t_f \leq 50$ mm)

Design resistance in compression

$$R_{c,y} = \frac{F_y}{\left(1+\lambda_y^{2n} - 0.15^{2n}\right)^{1/n}} A_t \cdot F_R = \frac{250}{\left(1+0.664^4 - 0.15^4\right)^{1/2}} 5425 \cdot 0.9 = 1117.02 \text{ kN}$$

Design ratio

$$\frac{P}{R_{c,y}} = \frac{300}{1117.02} = \underline{\underline{0.27 \leq 1}}$$

Result values from RF-STEEL NTC-DF calculation

Compression Axial Force	P_u	300.00	kN		
Section Resistance	R_s	1220.63	kN		3.2
Form Factor	A_e/A_t	1.000			2.1.3
Yield Strength	F_y	250.000	MPa		2.1
Modulus of Elasticity	E	200000	MPa		
Gross Area	A_t	5425.000	mm^2		
Second Moment of Area	I_y	24920000	mm^4		
Radius of Gyration	R_y	67.776	mm		
Effective Length	L_y	4000.000	mm		
Slenderness	λ_y	0.664			3.2.2
Elastic Flexural Buckling Stress	$F_{e,y}$	566.706	MPa		Eq. (3.12)
Factor	n	2.000			3.2.2
Nominal Critical Stress	$F_{n,y}$	228.781	MPa		
Resistance Factor	F_R	0.900			
Member Resistance	$R_{c,y}$	1117.02	kN		Eq. (3.3)
Design Ratio	η	0.27		< 1	3.2.2

Lateral-torsional buckling

Nominal and design buckling moment resistance according to [1] clause 3.3.2.2

The **nominal buckling moment resistance** for lateral torsional buckling will be determined for this example according to Eq. (3.24), taking into account pinned supports free to warp.

$$M_u = \frac{\pi}{CL}\sqrt{E \cdot I_z \cdot G \cdot J + \left(\frac{\pi \cdot E}{L}\right)^2 \cdot I_z \cdot C_a}$$

$$M_u = \frac{\pi}{4000}\sqrt{2.0e5 \cdot 8.892e6 \cdot 77200 \cdot 312400 + \left(\frac{\pi \cdot 2.0e5}{4000}\right)^2 \cdot 8.892e6 \cdot 4.794e10} = 181.507\ \text{kNm}$$

Limits of unsupported length

$$X_u = 4.293C\frac{Z_yF_y}{GJ}\sqrt{\frac{C_a}{I_z}} = 4.293 \cdot 1\frac{354000 \cdot 250}{77200 \cdot 312400}\sqrt{\frac{4.794e10}{8.892e6}} = 1.157$$

$$X_r = \frac{4}{3}C\frac{Z_yF_y}{GJ}\sqrt{\frac{C_a}{I_z}} = \frac{4}{3}1\frac{354000 \cdot 250}{77200 \cdot 312400}\sqrt{\frac{4.794e10}{8.892e6}} = 0.359$$

$$L_u = \frac{\sqrt{2}\pi}{X_u}\sqrt{\frac{EC_a}{GJ}}\sqrt{1+\sqrt{1+X_u^2}} = \frac{\sqrt{2}\pi}{1.157}\sqrt{\frac{2.e5 \cdot 4.794e10}{77200 \cdot 312400}}\sqrt{1+\sqrt{1+1.157^2}} = 3851\ \text{mm}$$

$$L_r = \frac{\sqrt{2}\pi}{X_r}\sqrt{\frac{EC_a}{GJ}}\sqrt{1+\sqrt{1+X_r^2}} = \frac{\sqrt{2}\pi}{0.359}\sqrt{\frac{2.e5 \cdot 4.794e10}{77200 \cdot 312400}}\sqrt{1+\sqrt{1+0.359^2}} = 11198\ \text{mm}$$

The **nominal member moment plastic resistance** is determined according to clause 3.3.2.1.

HEB-160: The cross-section type acc. to Table 2.1 is "2 - compact".

$$M_{p,y} = Z_y \cdot F_y = 354000 \cdot 250 = 88.50\text{kNm}$$

$$M_{p,z} = Z_z \cdot F_y = 170000 \cdot 250 = 42.50\text{kNm}$$

Design resistance acc. to 3.3.2.2(a) ($L > L_u$)

If

$$M_u > \frac{2}{3}M_{py} \qquad M_{Ry} = 1.15F_RM_{py}\left(1 - \frac{0.28M_{py}}{M_u}\right) \le F_RM_{py}$$

$$181.507 > \frac{2}{3} \cdot 88.5 = 59 \qquad M_{Ry} = \min\left\{1.15 \cdot 0.9 \cdot 88.50\left(1 - \frac{0.2888.50}{181.507}\right), 0.9 \cdot 88.50\right\}$$

$$M_{Ry} = 79.092\ \text{kNm}$$

Interaction of biaxial bending and compression

The design ratio is determined according to [1] clause 3.4.3.2(a). To calculate the final design ratio resistance acc. to Eq. (3.56), we need to determinate the value of M_m. This value can be calculated according to clause 3.3.2 or approximately according to Eq. (3.57) for I-sections. We use Eq. (3.57) in this example.

$$M_m = \min\left(F_R \left(1.07 - \frac{(L/r_z)\sqrt{F_y/E}}{18.55} \right) M_{py} ; F_R \cdot M_{py} \right)$$

$$M_m = \min\left(0.9 \left(1.07 - \frac{(4000/40.50)\sqrt{250/200000}}{18.55} \right) 88.50 ; 0.9 \cdot 88.50 \right) = 70.232 \text{ kNm}$$

Interaction design ratio acc. to 3.4.3.2(a), Eq. (3.56)

The value of R_c is the minimum design resistance in compression.

$$\frac{P_u}{R_c} + \frac{M^*_{uoy}}{M_m} + \frac{M^*_{uoz}}{F_R M_{pz}} \leq 1$$

$$\frac{300.00}{767.71} + \frac{10.00}{70.232} + \frac{7.50}{0.9 \cdot 42.50} = 0.391 + 0.142 + 0.196 = \underline{\underline{0.73 \leq 1}}$$

Result values from RF-STEEL NTC-DF calculation

Yield Strength	F_y	250	MPa		1.3.1
Compression Axial Force	P_u	300.00	kN		
Section Resistance	R_s	1220.63	kN		3.2
Elastic Buckling Stress	F_e	202.213	MPa		
Nominal Critical Stress	F_n	157.236	MPa		
Member Resistance	R_c	767.71	kN		Eq. (3.3)
Design Component for N	η_N	0.39		< 1	3.4.3.2(a)
Bending Moment	M_{uoy}	3.60	kNm		
Maximum Bending Moment	$M^*_{uoy,segm}$	10.00	kNm		
Nominal Section Resistance	M_{py}	88.50	kNm		
Design Bending Resistance	M_{Ry}	79.65	kNm		
Modulus of Elasticity	E	200000.000	MPa		
Shear Modulus	G	77200.000	MPa		
Second Moment of Area	I_z	8892000.000	mm^4		
Torsional Constant	J	312400.000	mm^4		
Warping Constant	C_a	4.79400E+10	mm^6		
Effective Length	L_w	4000	mm		3.3.2
Auxiliary Factor	X_r	0.359			Eq. (3.28)
Auxiliary Factor	X_u	1.157			Eq. (3.27)
Limit Unsupported Length	L_r	11198.700	mm		Eq. (3.26)
Limit Unsupported Length	L_u	3851.390	mm		Eq. (3.25)

Modification Factor	C_b	1.000			
Equivalent Moment Factor	C	1.000			3.3.2.2
Elastic Critical Moment	M_u	181.51	kNm		3.3.2.2
Design Bending Resistance	M_{my}	70.23	kNm		Eq. (3.57)
Design Component for My	η_{My}	0.142		< 1	3.4.3.2(a)
Bending Moment	M_{uoz}	1.50	kNm		
Maximum Bending Moment	$M^*_{uoz,segm}$	7.50	kNm		
Nominal Section Resistance	M_{pz}	42.50	kNm		
Design Bending Resistance	M_{Rz}	37.53	kNm		
Design Component for Mz	η_{Mz}	0.196		< 1	3.4.3.2(a)
Resistance Factor	F_R	0.900			
Design Ratio	η	0.73		< 1	3.4.3.2(a)

A Literature

[1] Normas Técnicas Complementarias para Diseño y Construcción de Estructuras Metálicas, Gaceta Oficial del Distrito Federal, Gobierno del Distrito Federal, México, La Ciudad de la Esperanza, 2004

[2] Specification for Structural Steel Buildings ANSI/AISC 360-10, U.S. Standard, June 22, 2010

B Index

A

Axis ... 19

B

Background graphic ... 44
Beam type ... 25
Bending design resistance M_m ... 29
Buckling ... 19
Buckling length ... 18
Buckling length coefficient ... 20
Button ... 43

C

Calculation ... 27
Cantilever ... 17, 25
Clipboard ... 56
Color spectrum ... 47
Colored design ... 47
Control panel ... 47
Cross-section ... 13, 53
Cross-section design ... 35
Cross-section library ... 13
Cross-section optimization ... 53
Cross-section type ... 14

D

Decimal places ... 11, 55
Deflection ... 10
Deflections ... 30
Deformation analysis ... 25
Design ... 9, 14, 33, 34, 35
Design case ... 44, 51, 52
Detail settings ... 27
Display navigator ... 45, 47

E

Effective length ... 18, 19, 21, 56
Eigenvalue ... 21
Elastic buckling moment ... 20
Equivalent member length ... 18
Excel ... 57
Exit RF-STEEL NTC-DF ... 8
Export ... 56
Export cross-section ... 54
Export effective length ... 56
Export Material ... 56

F

Favorites ... 53
Filter ... 47
Filtering members ... 48
Flexural buckling ... 17, 19

G

General data ... 8
Graphic ... 44
Graphic printout ... 49

H

Hidden result diagram ... 47

I

I- and H-section fabrication ... 29
Info about cross-section ... 15
Installation ... 6
Interaction ... 28
Intermediate lateral restraints ... 17
Internal forces ... 38, 54

L

Lateral restraint ... 17
Lateral-torsional buckling ... 17, 20, 29
Length ... 18, 41
Limit deformation ... 30
Limit internal forces ... 28
Limit load ... 30
Limit values ... 10
List of members ... 25
Load application ... 29
Load case ... 9, 10, 38
Load combination ... 9
Location x ... 34

M

Material ... 11, 56
Material library ... 12
Material properties ... 11
Member end releases ... 24
Member slendernesses ... 31, 40
Member-like input ... 29
Members ... 9
M_u ... 20

N

Navigator 8

Net area 26

Nodal support 22

Non-linear method (second order theory) 29

O

OpenOffice 57

Optimization 14, 31, 53, 54

Options 27

P

Panel 7, 45, 47

Parameterized cross-section 53

Parameters 26

Part 41

Parts list 41, 42

Plastic design 27

Precamber 25

Print 49

Printout report 49, 50

R

Ratio 34

Reference length 10

Reference scales 43

Relatively 17

Remark 15

Rendering 47

Result combination 9, 10

Result diagram 46, 49

Results evaluation 43

Results representation 45

Results values 44

Results window 33

RFEM graphic 49

RFEM work window 44

RF-STABILITY 19

S

Second order effects 29

Seismic design 28

Selecting windows 8

Serviceability 10, 25, 30

Serviceability limit state 25, 43

Set of members 9, 21, 22, 24, 25, 29, 36, 39, 42

Shear design 28

Shear lag factor 26

Shifted ends of members 30

Slenderness 31, 40

Special cases 30

Stability analysis 17, 28, 35

Stability factor 29

Start calculation 32

Start program 6

Start RF-STEEL NTC-DF 6

Stress point 16

Sum 42

Surface area 41

T

Tapered member 15, 35, 55

Tension design 26

Torsion 30

Torsional length 20

Torsional support 17

Torsional-flexural buckling 20

Transverse load 29

U

Ultimate limit state 9, 27, 43

Undeformed system 30

Units 11, 55

User profile 55

V

View mode 43, 44

Visibilities 47

Volume 42

W

Weight 42

X

x-location 34, 38