Version
May 2013

Add-on Module

RF-STEEL AS

Ultimate Limit State, Serviceability and Stability Design According to AS 4100

Program Description

© Ing.-Software Dlubal
Am Zellweg 2 D-93464 Tiefenbach

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

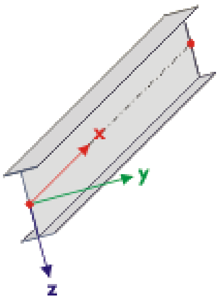
Contents

1. Introduction

1.1 Add-on Module RF-STEEL AS

The Australian Standard AS 4100 for Steel Structures describes the design, analysis, and construction of steel structures. With the RFEM add-on module RF-STEEL AS the DLUBAL company provides a powerful tool to design steel structures according to the Australian code.

RF-STEEL AS 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 Australian code, RF-STEEL AS 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 AS determines the c/t-ratios of the cross-section elements subjected to compression and classifies the cross-sections completely automatically.



In RF-STEEL AS, the indices of the member axes are different from those used in the code: The longitudinal axis is denoted by the index is **x** instead of *z*. For the axes in the cross-section plane, the axes **y** and **z** are used (see figure 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 AS determines the slendernesses and elastic critical buckling loads from the boundary conditions. The ideal critical moment for lateral torsional buckling required for the lateral torsional buckling design is 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 structures employing extremely slender cross-sections, the serviceability limit state represents an important design. The load cases, load and result combinations can be assigned to different design situations. 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 use the add-on module to optimise cross-sections and export the modified cross-sections to RFEM. Design cases enable you to separately design structural components of complex models or to analyse variants.

RF-STEEL AS 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 determination of internal forces to design.

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

Your DLUBAL Team

1.2 RF-STEEL AS - Team

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

Program coordination

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

Programming

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

Cross-section and material database

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

Program design, dialog figures, and icons

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

Program supervision

Ing. Ph.D. Martin Čudejko
Dipl.-Ing. (FH) Wieland Götzler

Localisation, manual

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

Technical support and quality management

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

1.3 Using the Manual

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

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

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

1.4 Open the Add-on Module RF-STEEL AS

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

Menu

To start the program in the RFEM menu bar, click

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

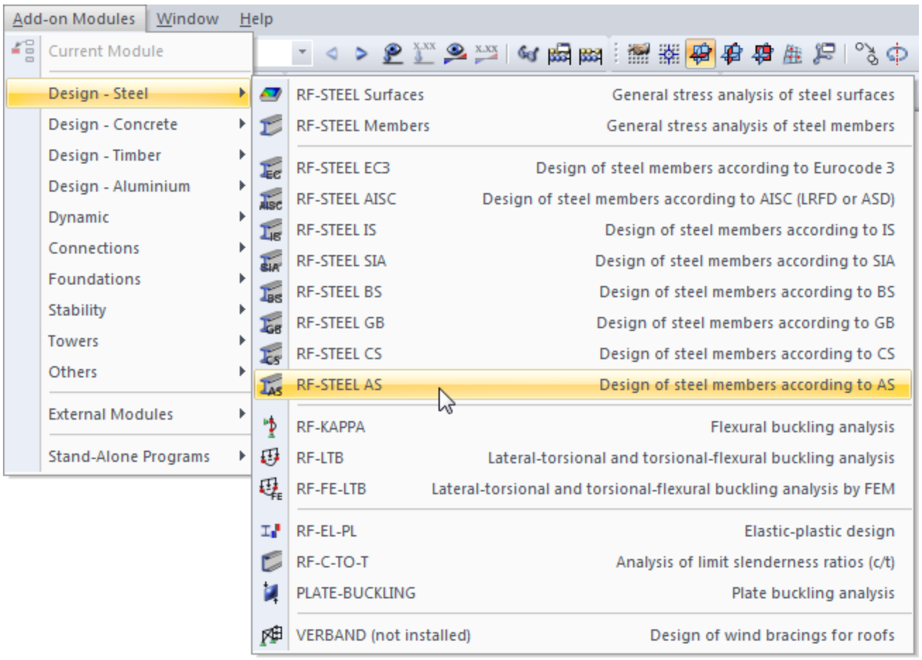


Figure 1.1: Menu: *Add-on Modules → Design - Steel → RF-STEEL AS*

Navigator

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

Add-on Modules → RF-STEEL AS.

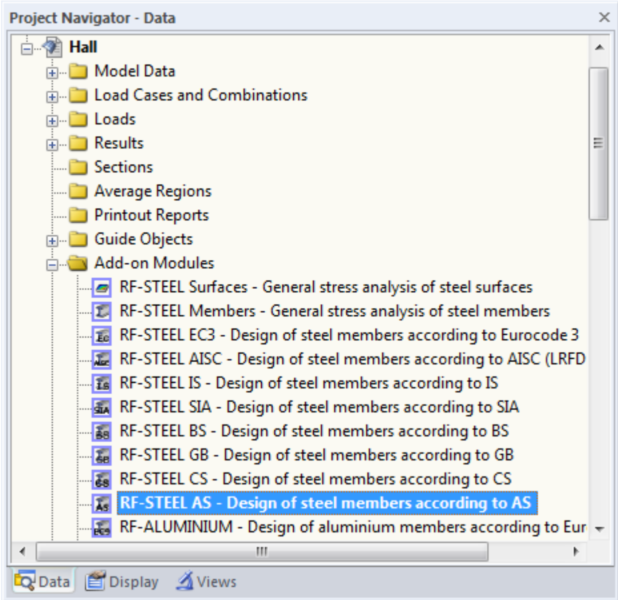
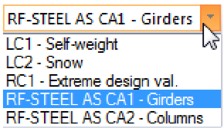
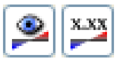


Figure 1.2: Data navigator: *Add-on Modules → RF-STEEL AS*

Panel





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

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

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

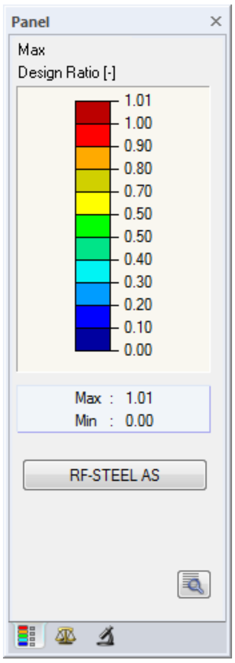


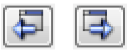
Figure 1.3: Panel button [RF-STEEL AS]

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 tables that can be currently selected. The drop-down list above the navigator contains the design cases (see chapter 7.1, page 52).

The design relevant data is defined in several input tables. When you open RF-STEEL AS 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 table, click the corresponding entry in the navigator. To set the previous or next input table, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] table.

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

2.1 General Data

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

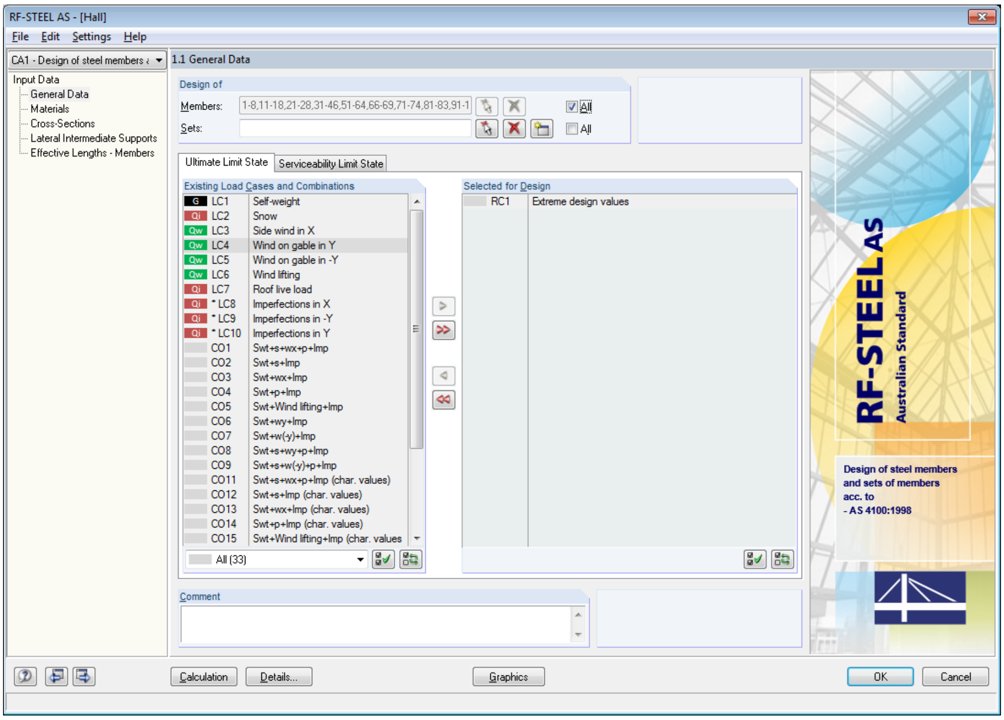


Figure 2.1: Table 1.1 *General Data*

Design of

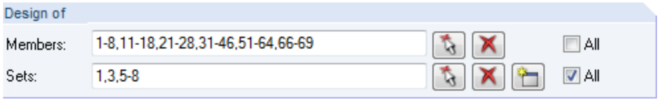


Figure 2.2: Design of members and sets of members

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. Click [↖] to select the objects graphically in the RFEM work window.

When you design a set of members, the program determines the extreme values of the designs of all members contained in the set of members and takes into account the boundary conditions of connected members for the stability analysis. The results are shown in the result table 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

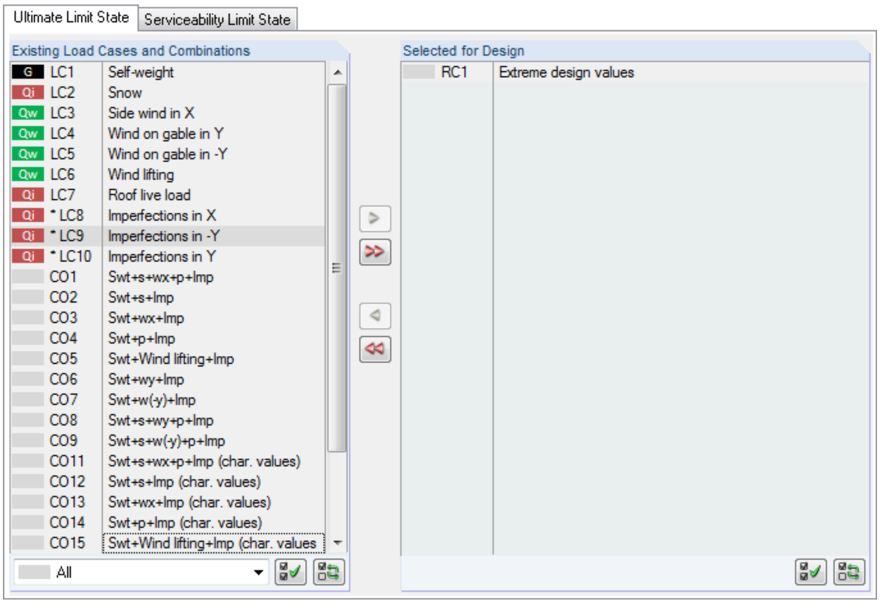


Figure 2.3: Table 1.1 *General Data*, tab *Ultimate Limit State*

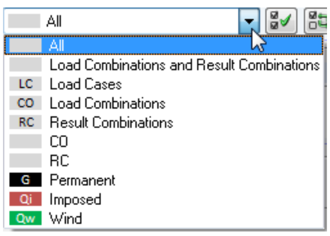
Existing Load Cases and Combinations

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

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

To transfer multiple entries of load cases, click the entries while pressing the [Ctrl] key, as common for Windows applications. Thus, you can transfer several load cases at the same time.

Load cases marked by an asterisk (*), like load case 9 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 try to transfer those 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 according to load cases, load combinations, or action categories. The buttons have the following functions:

[Select all button]	Select all load cases in the list.
[Invert selection button]	Invert 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 [◀◀].

Result combination

The design of an enveloping max/min result combination is performed faster than the design of all contained load cases and load combinations. However, the analysis of a result combination has also disadvantages: First, the influence of the contained loads is difficult to discern. Second, for the determination of the factor β_m according to clause 4.4.2.2 (b), the envelope of the moment distributions is analyzed, from which the most unfavorable distribution (max or min) is taken. However, this distribution only rarely reflects the moment distribution in the individual load combinations. Thus, in the case of a RC design, more unfavorable values for β_m are likely that lead to higher ratios.

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

2.1.2 Serviceability

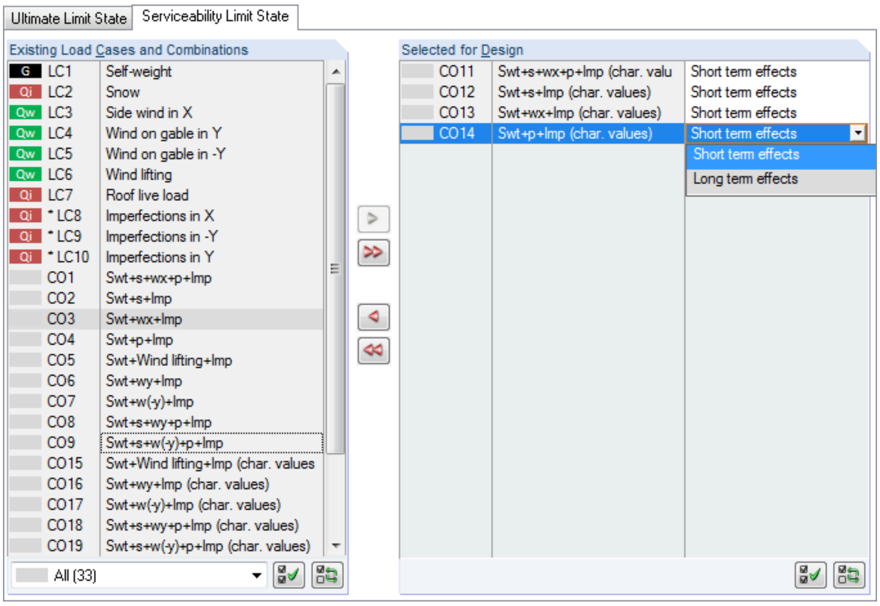


Figure 2.4: Table 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

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

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

- *Short term effects*
- *Long term effects*

To modify the design situation, use the list which can be accessed at the end of the input field by clicking [▼] (see Figure 2.4).

The limit values of the deformations are defined in the design details. To adjust these values according to the design situation, click [Details] to open the dialog box *Details*, tab *Serviceability* (see Figure 3.3, page 31).

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

2.2 Materials

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

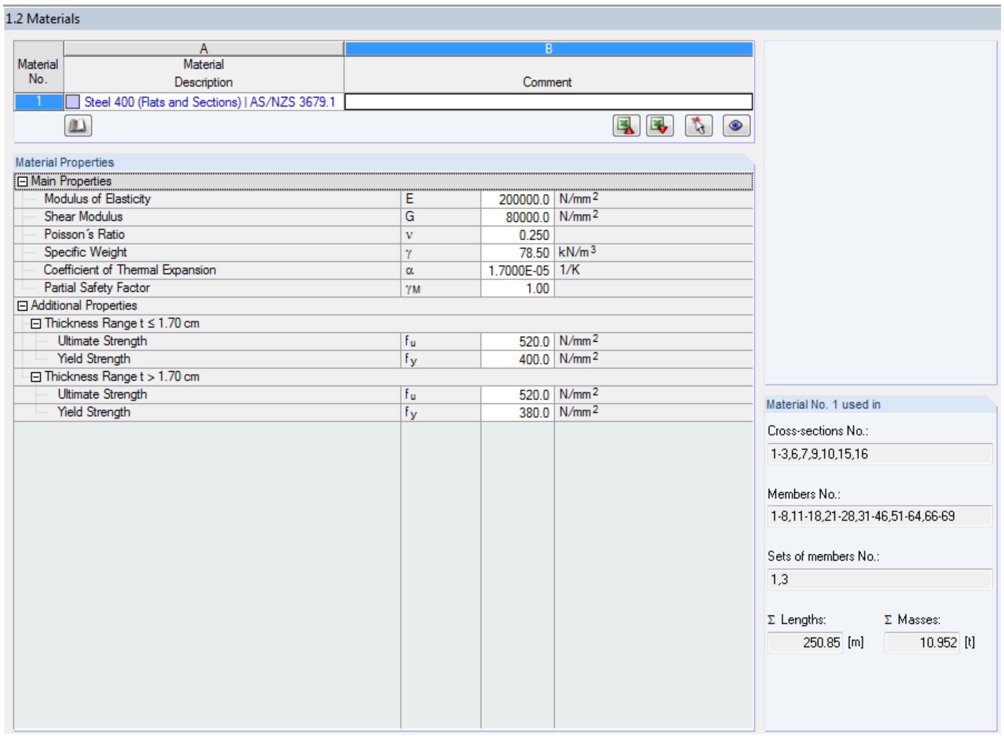


Figure 2.5: Table 1.2 *Materials*

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 4.2 of the RFEM manual (*Main Properties*). The material properties required for design are stored in the global material library. The values are preset (*Additional Properties*).

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

Material Description

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

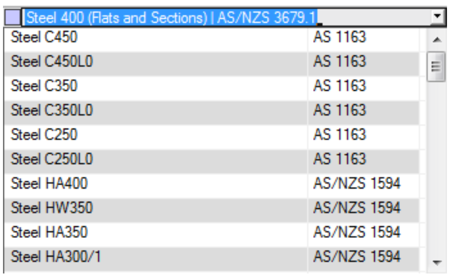


Figure 2.6: List of materials

According to the design concept of the Australian 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 AS will import the material properties, too.

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

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.

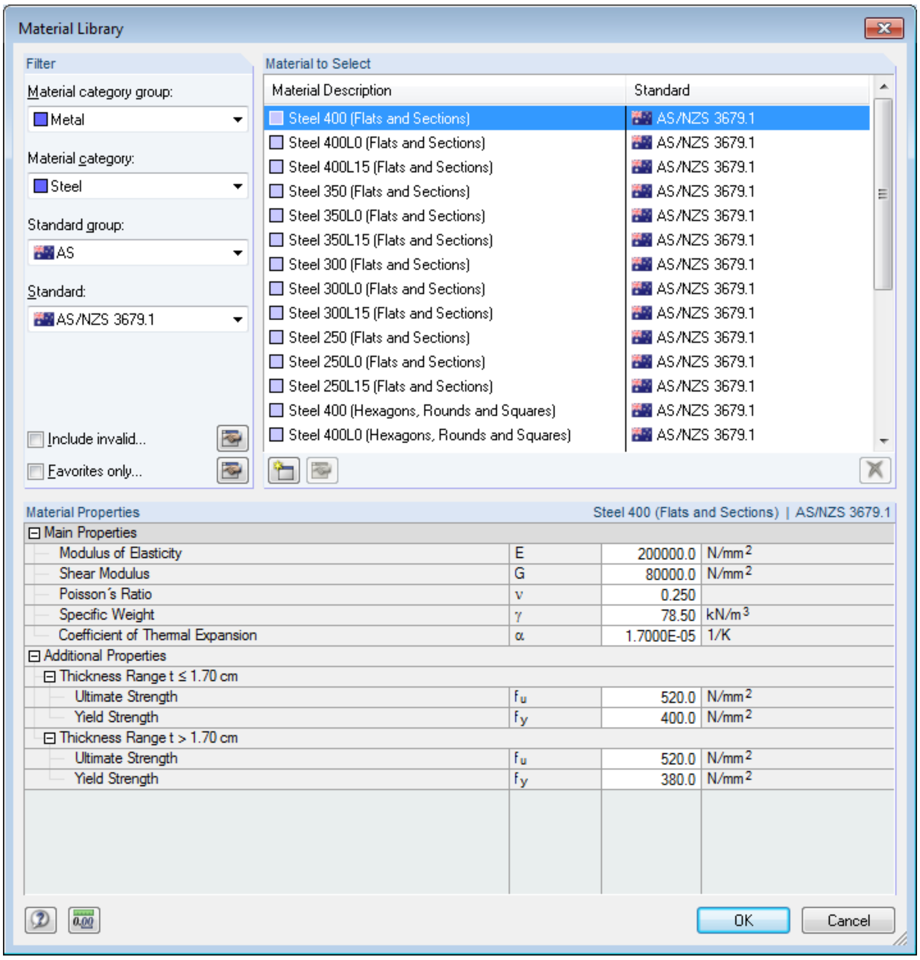


Figure 2.7: Dialog box *Material Library*

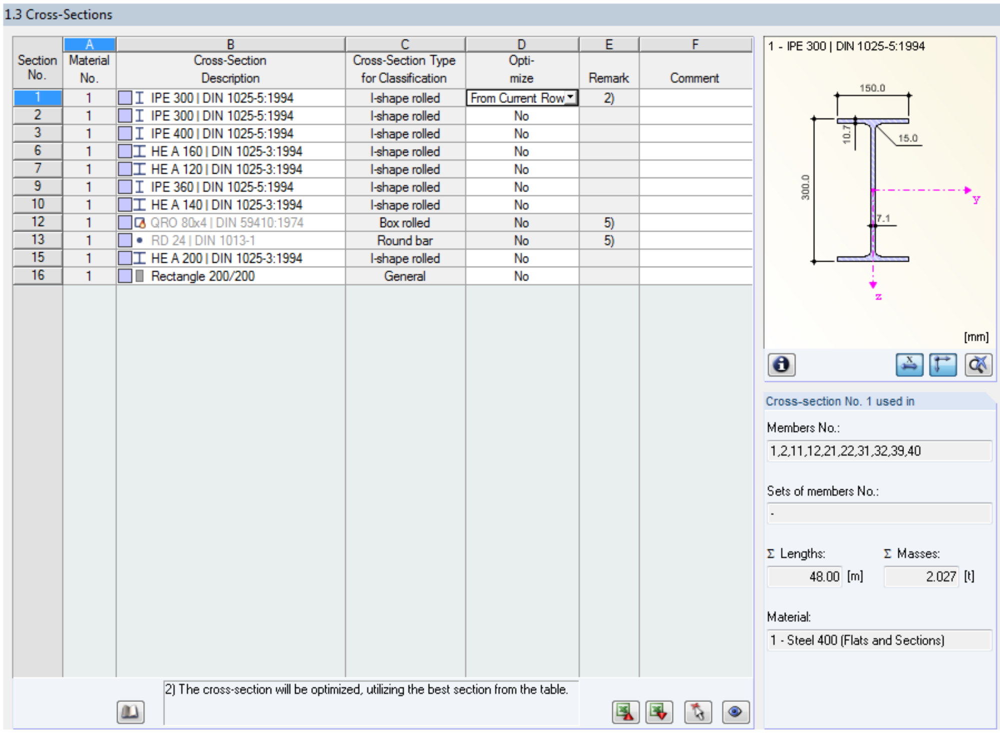
In the *Filter* section, *Steel* is preset as material category. Select the material quality that you want to use for the design in the list *Material to Select*. The corresponding properties can be checked in the dialog section below.

Click [OK] or [↵] to transfer the selected material to table 1.2 of the RF-STEEL AS module. Please check, however, whether these materials are allowed by the design concept of the Australian Standard [1].

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

2.3 Cross-sections

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



1.3 Cross-Sections

Section No.	A Material No.	B Cross-Section Description	C Cross-Section Type for Classification	D Opti-mize	E Remark	F Comment
1	1	IPE 300 | DIN 1025-5:1994	I-shape rolled	From Current Row	2)	
2	1	IPE 300 | DIN 1025-5:1994	I-shape rolled	No		
3	1	IPE 400 | DIN 1025-5:1994	I-shape rolled	No		
6	1	HE A 160 | DIN 1025-3:1994	I-shape rolled	No		
7	1	HE A 120 | DIN 1025-3:1994	I-shape rolled	No		
9	1	IPE 360 | DIN 1025-5:1994	I-shape rolled	No		
10	1	HE A 140 | DIN 1025-3:1994	I-shape rolled	No		
12	1	QRO 80x4 | DIN 59410:1974	Box rolled	No	5)	
13	1	RD 24 | DIN 1013-1	Round bar	No	5)	
15	1	HE A 200 | DIN 1025-3:1994	I-shape rolled	No		
16	1	Rectangle 200/200	General	No		

2) The cross-section will be optimized, utilizing the best section from the table.

Figure 2.8: Table 1.3 *Cross-sections*

Cross-Section Description

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

If you want to modify a cross-section, click the entry in column B to select 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.

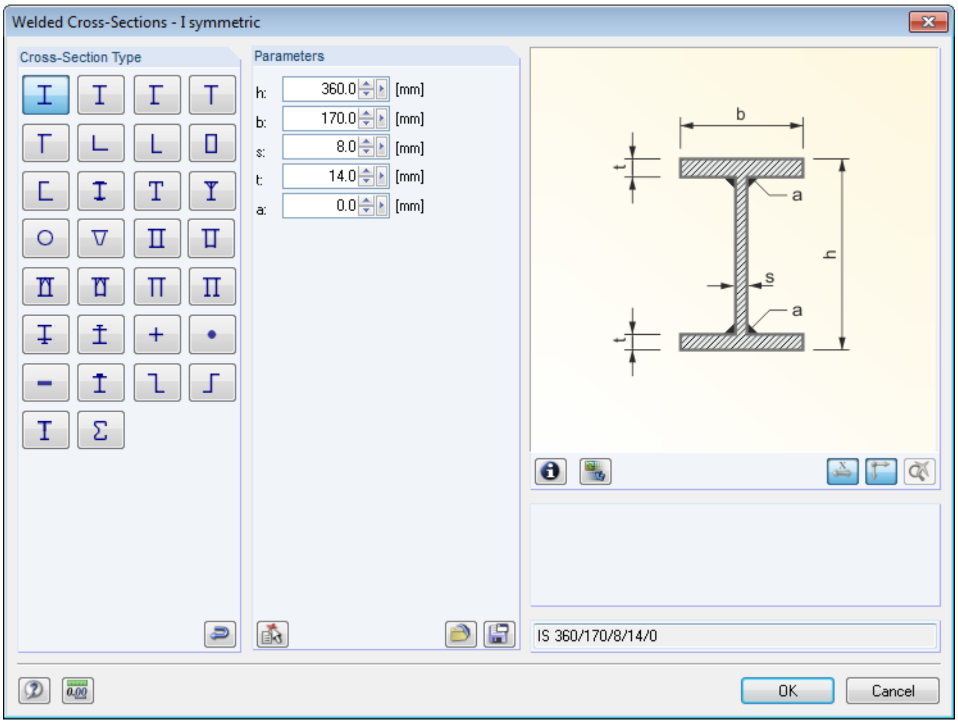
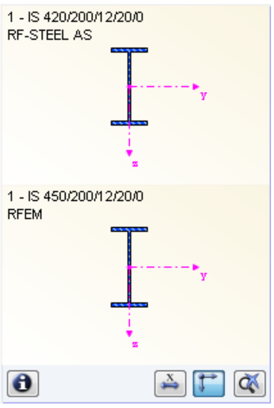


Figure 2.9: IS cross-sections in the cross-section library



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

A modified cross-section will be highlighted in blue.

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

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 optimisation. By means of the displayed design ratio and coloured relation scales, you can see which cross-sections are little utilised and thus oversized, or overloaded and thus undersized.

Optimise

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

If you want to optimize a cross-section, open the drop-down list in column D or E and select the desired entry: *From Current Row* or, if available, *From favourites 'Description'*. Recommendations for the cross-section optimisation can be found in chapter 7.2 on page 54.

Remark

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

A warning might appear before the calculation: *Incorrect type of cross-section!* This means that there is a cross-section that is not registered in the data base. This may be a user-defined cross-section, or a SHAPE-THIN cross-section that has not been calculated yet. To select an appropriate cross-section for design, click [Library] (see description below 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 AS also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the member end. The normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the start and the end cross-section of a tapered member have not the same number of stress points, the intermediate values cannot be interpolated. The calculation is neither possible in RFEM nor in RF-STEEL AS.

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

Info About Cross-Section

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

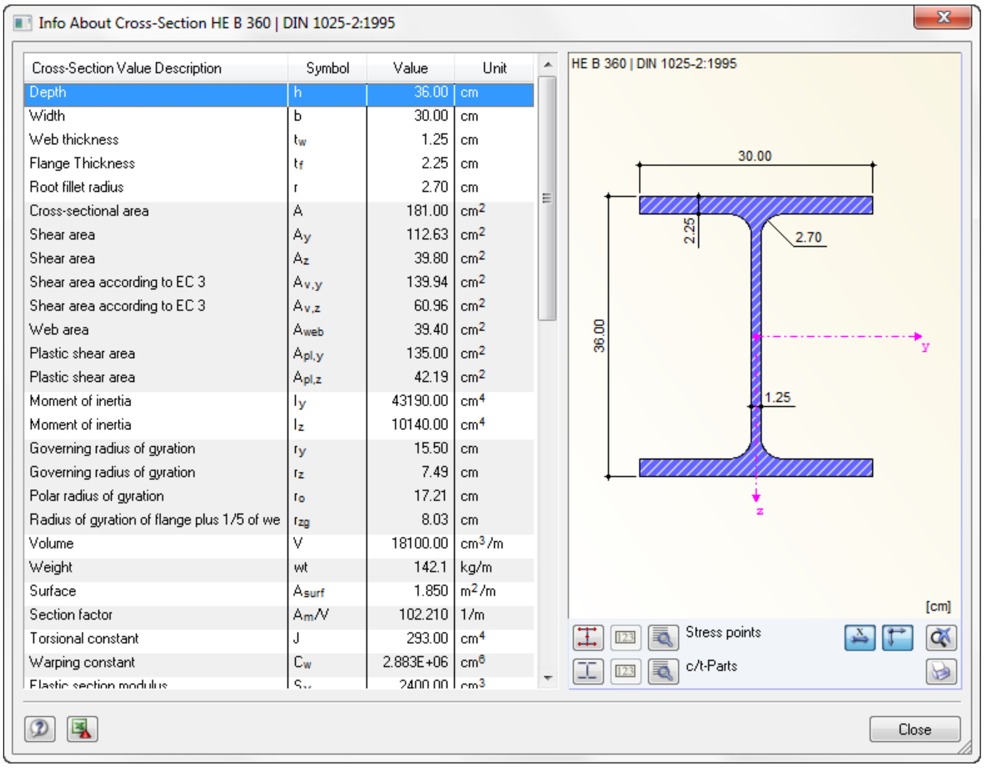


Figure 2.10: Dialog box *Info About Cross-Section*

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

In RF-STEEL AS, the indices of the member axes are different from those used in the code: The longitudinal axis is denoted by the index is **x** instead of *z*. For the axes in the cross-section plane, the axes **y** and **z** are used (see Figure 2.10).

The buttons below the graphic are reserved for the following functions:

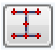
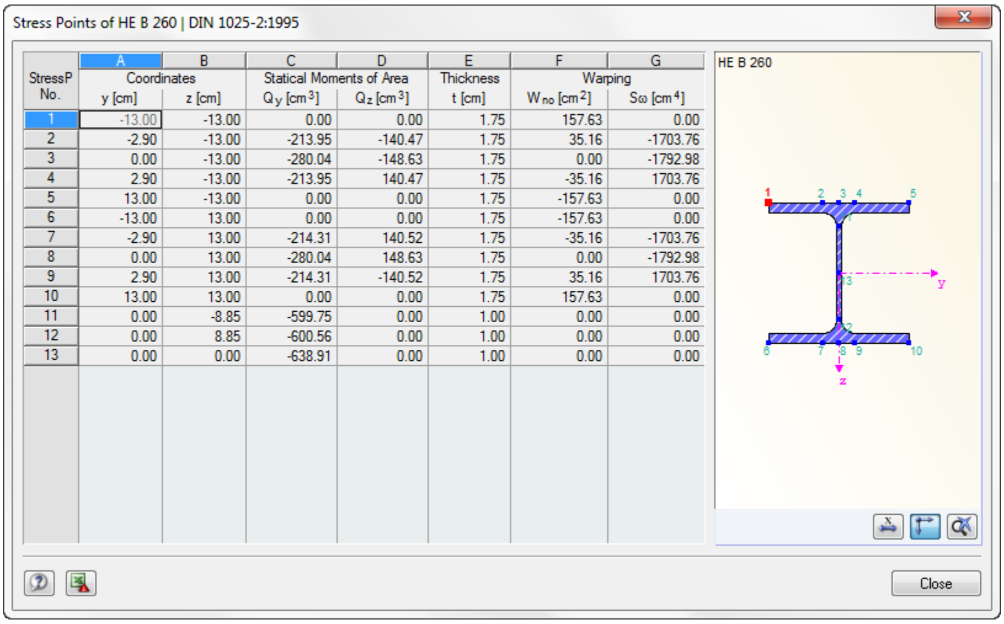
Button	Function
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the numbering of stress points or c/t-parts
	Displays or hides the details of the stress points or c/t-parts (see Figure 2.11)
	Displays or hides the dimensions of the cross-section
	Displays or hides 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 centre of gravity, statical moments of area, normalised warping constants etc.) and c/t-parts.



StressP No.	A Coordinates y [cm]	B Coordinates z [cm]	C Statical Moments of Area Q_y [cm³]	D Statical Moments of Area Q_z [cm³]	E Thickness t [cm]	F Warping W_{no} [cm²]	G Warping $S\omega$ [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 Lateral Intermediate Supports

In table 1.4, you can define lateral intermediate supports for members. RF-STEEL AS 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, all lateral intermediate supports are considered as forked supports.

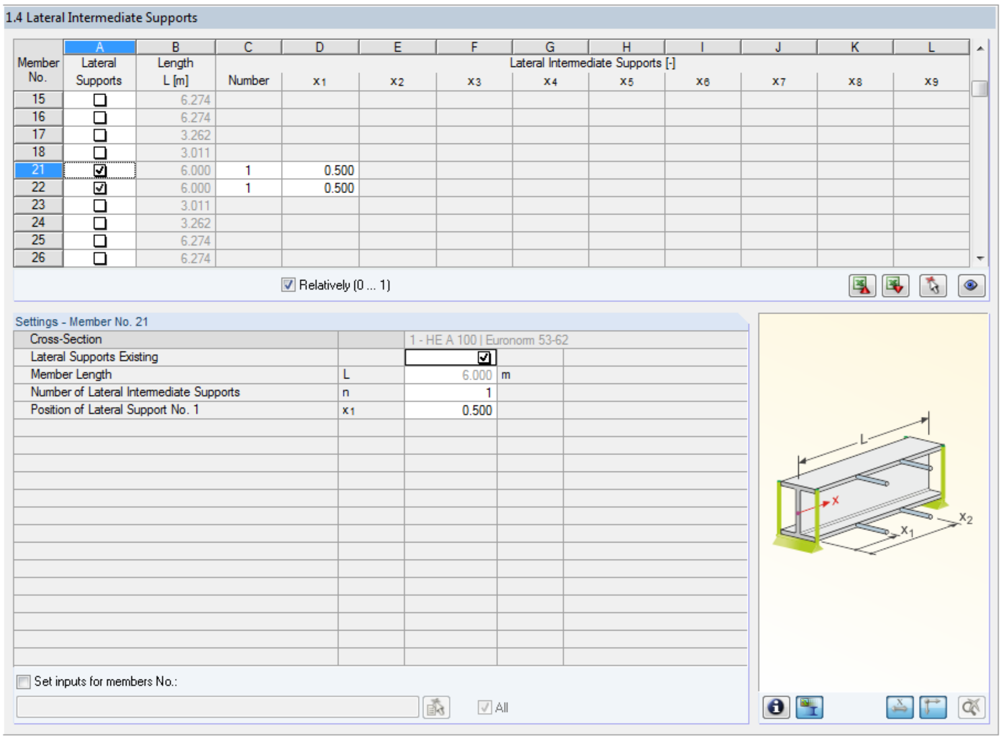


Figure 2.12: Table 1.4 *Lateral Intermediate Supports*

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

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 supports are determined from the member length and the relative distances from the member start. When the check box *Relatively (0 ... 1)* is cleared, you can define the distances manually in the upper table.

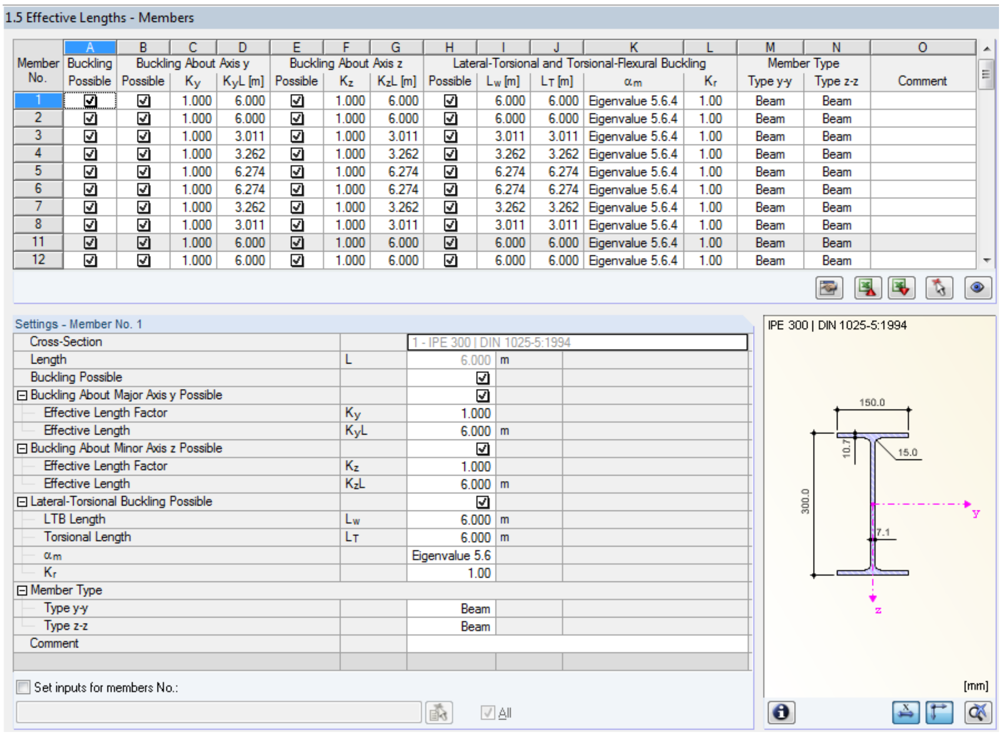
In case of cantilevers, avoid intermediate supports, as such supports divide the member in segments. Thus for cantilevered beams, this would results in segments that are forked supported on one side and thus statically underdetermined (forked support respectively on one end only).

2.5 Effective Lengths - Members

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

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

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


1.5 Effective Lengths - Members

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
Member No.	Buckling Possible	Buckling About Axis y Possible	K_y	K_yL [m]	Buckling About Axis z Possible	K_z	K_zL [m]	Lateral-Torsional and Torsional-Flexural Buckling Possible	L_w [m]	L_T [m]	α_m	K_r	Member Type Type y-y	Type z-z	Comment
1	☑	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	6.000	Eigenvalue 5.6.4	1.00	Beam	Beam	
2	☑	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	6.000	Eigenvalue 5.6.4	1.00	Beam	Beam	
3	☑	☑	1.000	3.011	☑	1.000	3.011	☑	3.011	3.011	Eigenvalue 5.6.4	1.00	Beam	Beam	
4	☑	☑	1.000	3.262	☑	1.000	3.262	☑	3.262	3.262	Eigenvalue 5.6.4	1.00	Beam	Beam	
5	☑	☑	1.000	6.274	☑	1.000	6.274	☑	6.274	6.274	Eigenvalue 5.6.4	1.00	Beam	Beam	
6	☑	☑	1.000	6.274	☑	1.000	6.274	☑	6.274	6.274	Eigenvalue 5.6.4	1.00	Beam	Beam	
7	☑	☑	1.000	3.262	☑	1.000	3.262	☑	3.262	3.262	Eigenvalue 5.6.4	1.00	Beam	Beam	
8	☑	☑	1.000	3.011	☑	1.000	3.011	☑	3.011	3.011	Eigenvalue 5.6.4	1.00	Beam	Beam	
11	☑	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	6.000	Eigenvalue 5.6.4	1.00	Beam	Beam	
12	☑	☑	1.000	6.000	☑	1.000	6.000	☑	6.000	6.000	Eigenvalue 5.6.4	1.00	Beam	Beam	

Figure 2.13: Table 1.5 *Effective Lengths - Members*

The effective lengths for local buckling about the minor axis are aligned automatically with the entries of table 1.4 *Lateral Intermediate Supports*. If lateral intermediate supports are dividing the member into member segments of different lengths, the program displays no value in the table columns G, I, and J of table 1.5.

The effective lengths can be entered manually in the table and in the *Settings* tree, or defined graphically by clicking the button [...] in the work window. 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 - L)
- *Member Type y-y*
- *Member Type z-z*

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 that, you can adjust the *Buckling Length Coefficient* and the *Warping Length Coefficient* for the respective lengths. When a coefficient is modified, the equivalent member length will be adjusted automatically, and vice versa.

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

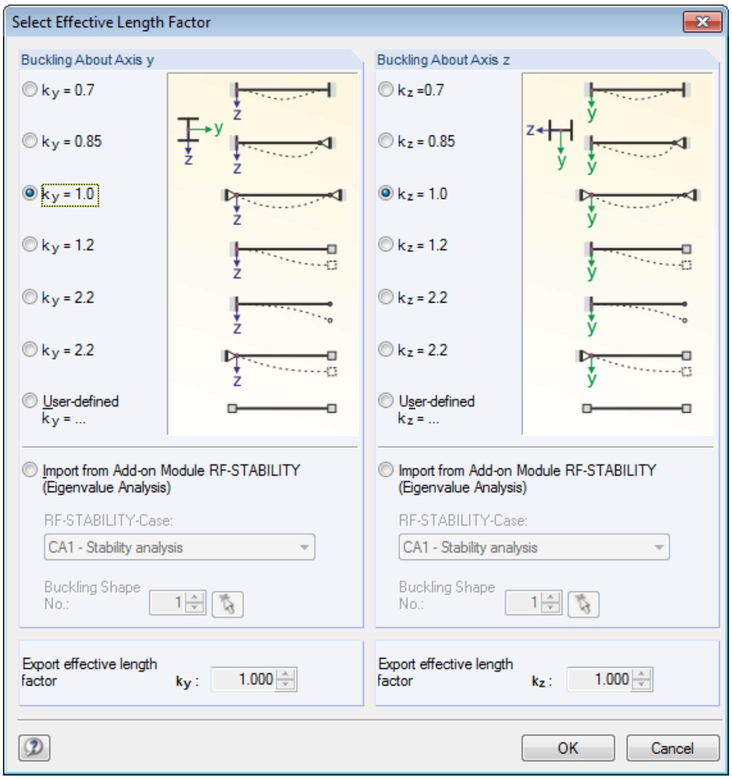


Figure 2.14: Dialog box *Select Buckling Length Coefficient*

For every direction, the buckling length can be defined according to one of the four 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 indicated in the *Comment* column.

The check boxes *Buckling Possible* 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

With the check boxes in the *Possible* table columns, you decide whether a member has a risk of buckling about the axis y and/or z. 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.

The position of the member axes can be checked in the cross-section graphic in table 1.3 *Cross-Sections* (see Figure 2.8, page 14).

The indices of the member axes are different from those used in the Australian Standard: The longitudinal axis is denoted by the index is **x** instead of *z*. For the axes in the cross-section plane, the axes **y** and **z** are used.

To access the RFEM work window, click [View mode]. To show the local member axes in the RFEM work window, you can use the member's context menu or the *Display* navigator.

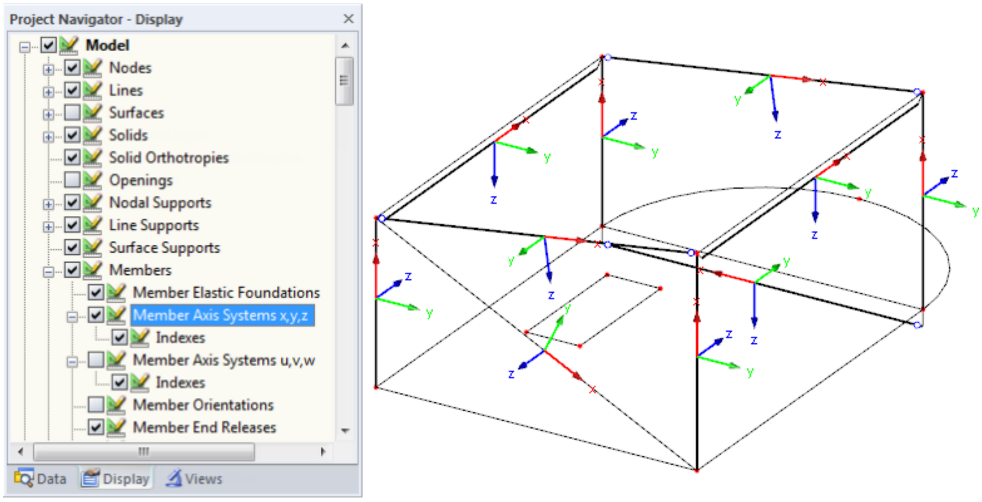


Figure 2.15: Selecting the member axis systems in the *Display* navigator of RFEM

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

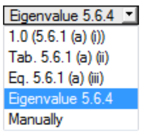
To specify the buckling lengths in the work window graphically, click [...]. This button becomes available when you click in a *KL* input field (see Figure 2.13).

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

Lateral-torsional Buckling Possible

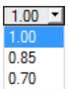
Table column H shows you for which members the program performs an analysis of lateral-torsional buckling.

With the check box in the *Possible* table columns, you decide whether a member has a risk of torsional-flexural and lateral-torsional buckling. When you have set the check, you can edit the buckling lengths L_W and L_T in columns I and J.



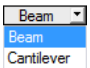
Moment Modification Factor α_m

The moment modification factor α_m can be calculated according to the options given in clause 5.6.1.1 of the standard. It is also possible to define this factor *Manually*.



Lateral Rotation Restraint Factor K_r

The lateral rotation restraint factor depending on the arrangement of the restraint is given in Table 5.6.3 (3) of the standard.



Member Type

Columns M and N define the restraint type of member segment according to clauses 5.6.1 and 5.6.2 of the standard: *Beam* or *Cantilever*. The member type is equal for all member segments.

Comment

In the last table column, you can enter user-defined comments for each member to describe, for example, the selected equivalent member lengths.

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

2.6 Effective Lengths - Sets of Members

This table appears only if at least one Set of Members has been selected for design in the table 1.1 *General Data*.

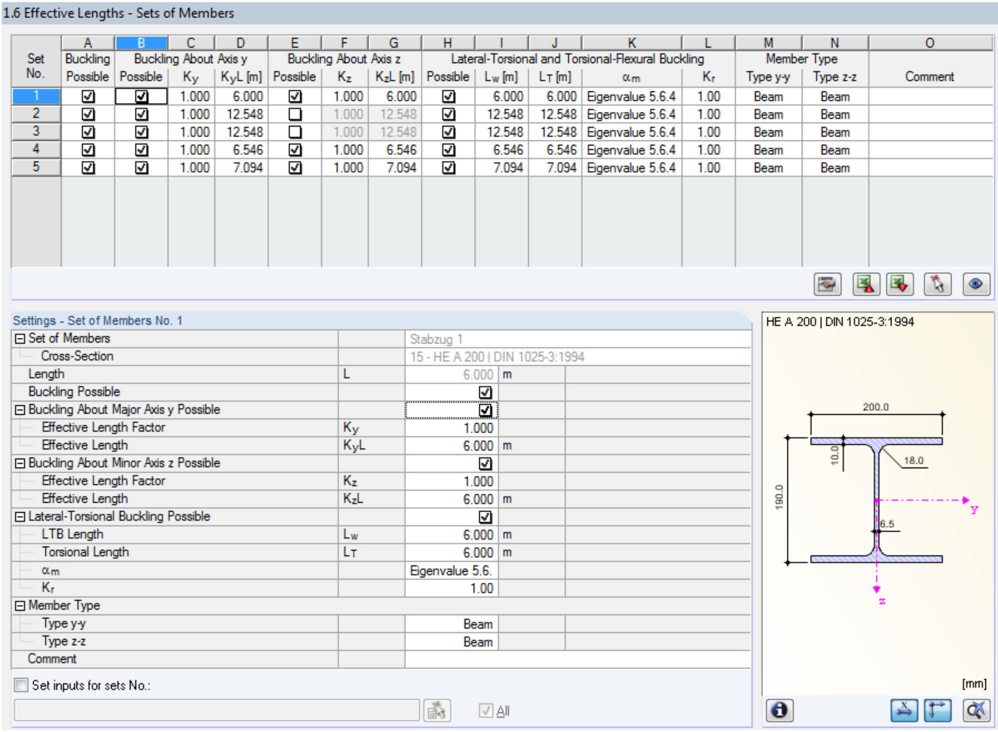


Figure 2.16: Table 1.6 *Effective Lengths - Sets of Members*

This table's concept is similar to the one in the previous table 1.5 *Effective Lengths - Members*. In this table, 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

Details...

This table is displayed when you have chosen at least one set of members for the design in table 1.1 *General Data*. Table 1.7 is not available if in the dialog box *Details*, tab *Stability* (see Figure 3.2, page 29) the *Member-like input* has been selected for sets of members. In this case, you can define the lateral intermediate supports by division points in table 1.4.

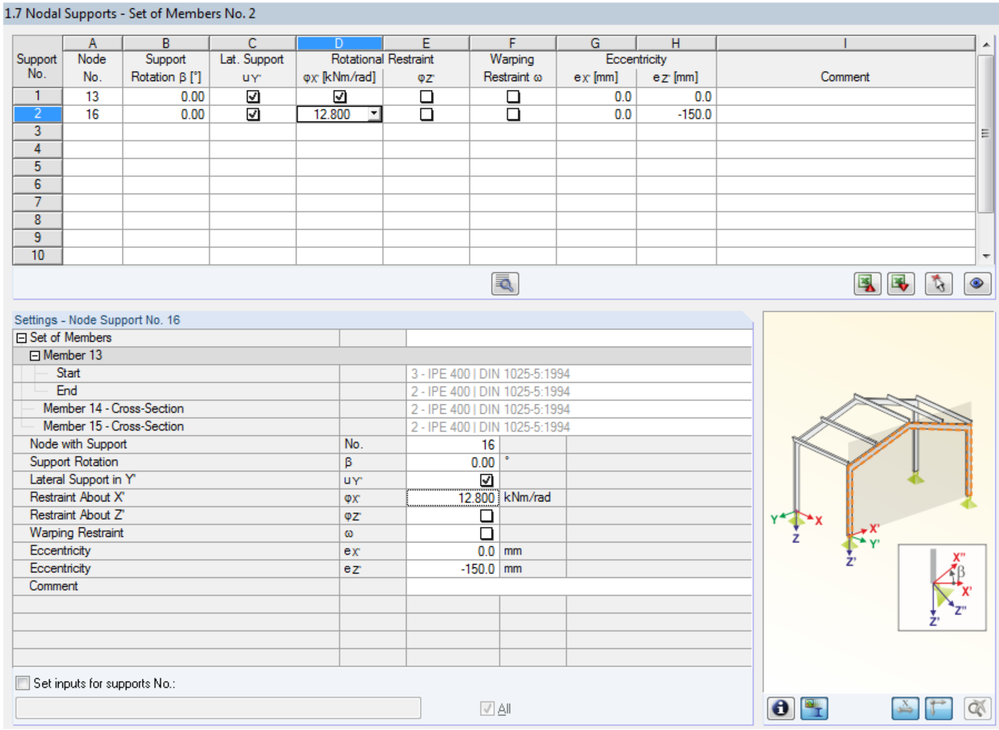


Figure 2.17: Table 1.7 *Nodal Supports - Set of Members*

To determine the critical factor for lateral-torsional buckling, a planar framework is created with four degrees of freedom for each node which you have to define in table 1.7. This table refers to the current set of members (selected in the add-on module's navigator on the left).

The orientation of 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 table 1.7.

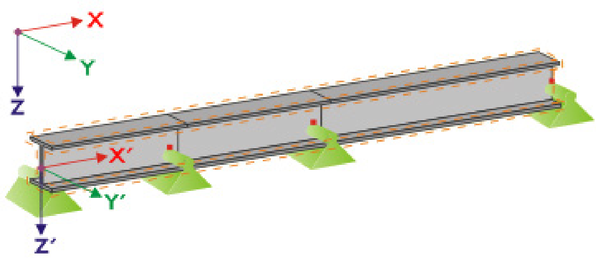


Figure 2.18: Auxiliary coordinate system for nodal supports - straight set of members

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

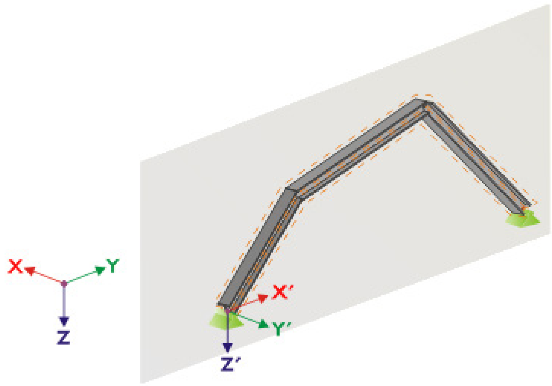


Figure 2.19: Auxiliary coordinate system for nodal supports - set of members in vertical plane

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.

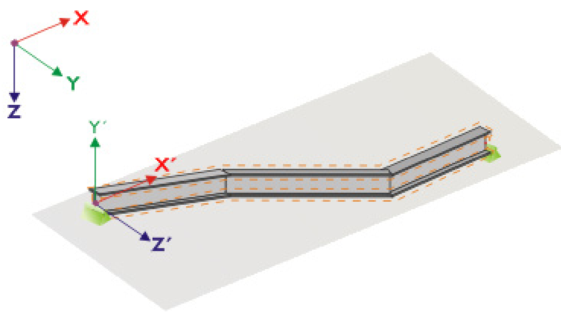


Figure 2.20: Auxiliary coordinate system for nodal supports - set of members in horizontal plane

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

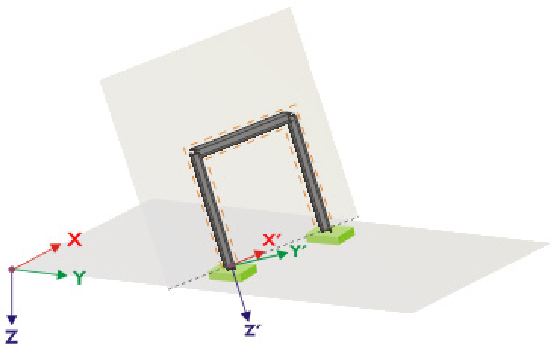


Figure 2.21: Auxiliary coordinate system for nodal supports - set of members in inclined plane

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 axis X' arises out of the intersection line of the inclined plane with the horizontal plane. Thus, the axis Y' is defined right-angled to the axis X' and directed perpendicular to the inclined plane. The axis Z' is defined perpendicular to the axis X' and Y'.

2.8 Member End Releases - Sets of Members

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

Table 1.8 is not available if in the dialog box *Details*, tab *Stability* (see Figure 3.2, page 29) the *Member-like input* has been selected for the sets of members.

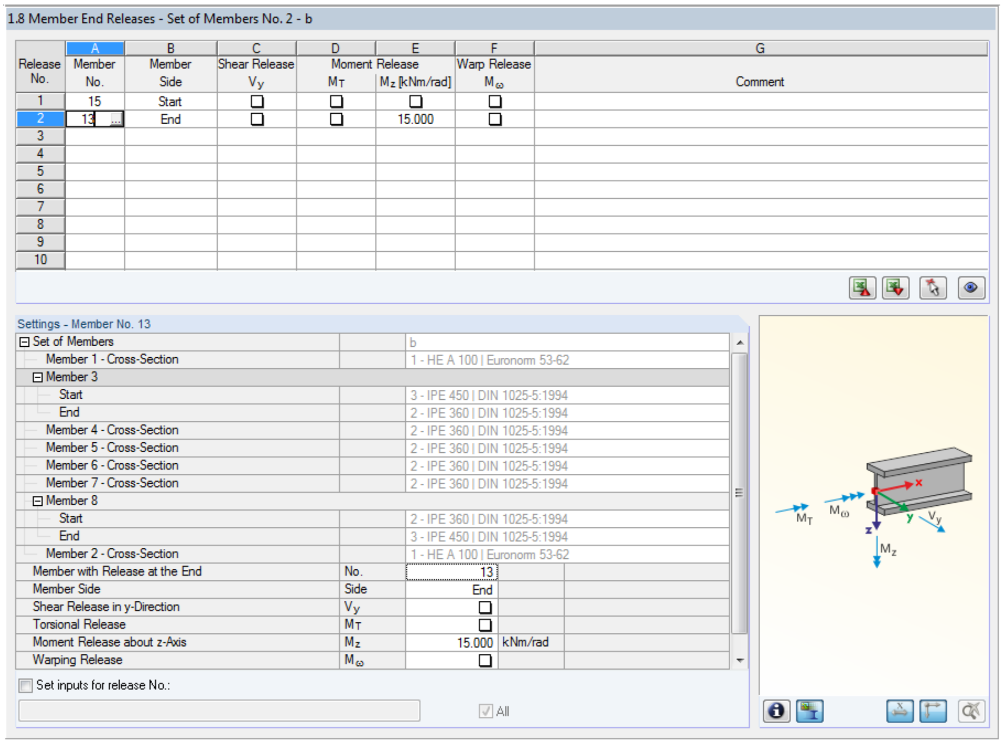
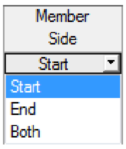


Table 2.22: Table 1.8 *Member Releases - Set of Members*



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 to F, you can define releases or spring constants to align the set of members model with the support conditions in table 1.7.

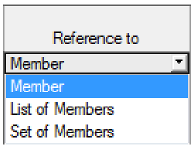
2.9 Serviceability Data

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

1.9 Serviceability Data

	A	B	C	D	E	F	G	H
No.	Reference to	Set of Members No.	Reference Length Manually	L [m]	Direc-tion	Precamber w_c [mm]	Beam Type	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								

Table 2.23: Table 1.9 *Serviceability Data*



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

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

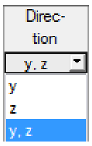
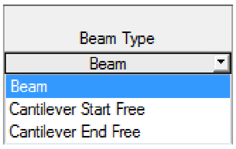


Table column E defines the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z (or u and v for unsymmetrical cross-sections).

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



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

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

3. Calculation

3.1 Detail Settings

Before you start the calculation by clicking [Calculation], it is recommended to check the design details. The corresponding dialog box can be accessed in all tables 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

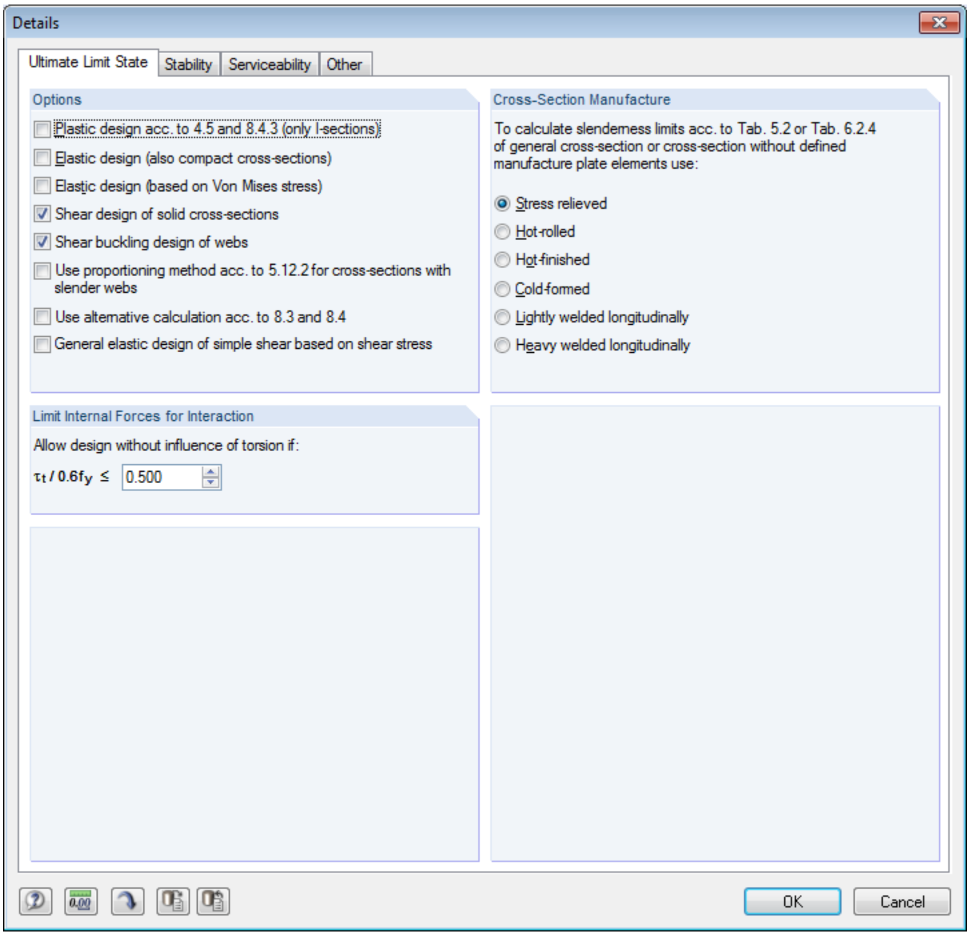


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

Options

The *Plastic design* according to clause 4.5 can be used I-shape sections. When the conditions described in clause 4.5 are satisfied, hot-formed doubly symmetric I-sections can also be designed according to clause 8.4.3.

Generally, cross-sections of the type "compact" are designed plastically. If you do not want to perform a plastic design, you can activate the *Elastic design* for compact cross-sections.

The conservative general *Elastic design* is based on a stress analysis in stress points and the VON MISES equivalent stresses. It can be useful for cross-section that have complicated shapes, for members with torsional moments etc.

The *Shear design* of solid flat or round bars or *Shear buckling design* of webs is not required in special cases. Those design options can be deactivated, if appropriate.

By default, the shear and bending interaction method is set according to clause 5.12.3. It is possible to set the proportioning method according to clause 5.12.2 for cross-sections where bending is assumed to be resisted only by the flanges.

The Australian standard offers alternative formulae for the compression and bending interaction design of doubly symmetric I-sections and rectangular hollow sections. They can be used to determine the cross-section capacity (clause 8.3) and member capacity (clause 8.4).

The conservative general elastic design based on shear stress analysis in stress points can be activated, if appropriate.

Limit Internal Forces for Interaction

The Australian standard gives no exact procedure how the design of torsional moments is to be handled. Therefore, RF-STEEL AS includes an option to ignore shear stresses due to torsion, which makes it possible to design those cross-sections. The ratio of stress and shear strength can be defined manually.

Cross-section Manufacture

According to the Commentary on the Australian standard [2] , residual stresses should be taken into account for the determination of element slenderness limits and cross-section types. In this dialog section, the type of manufacture can be specified that is to be applied for general and SHAPE-THIN cross-sections.

3.1.2 Stability

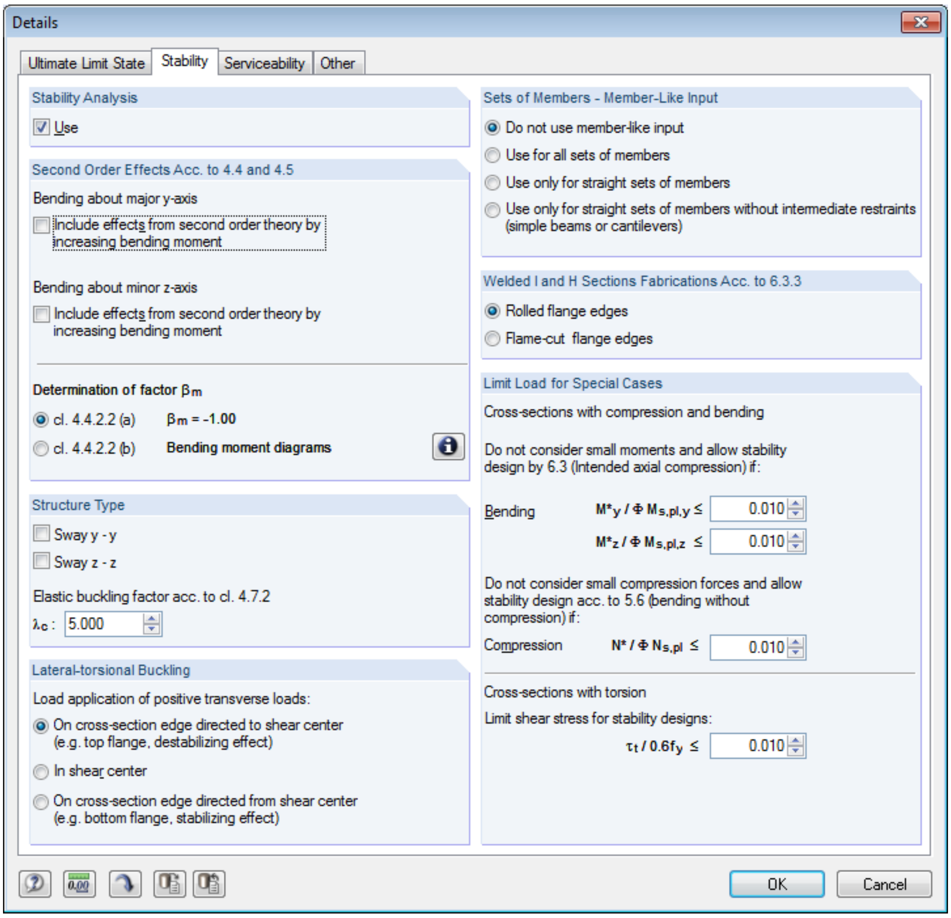


Figure 3.2: Dialog box *Details,* tab *Stability*

Stability Analysis

The check box *Use* controls whether to perform, in addition to the cross-section designs, a stability analysis. If you clear the check box, the input tables 1.4 to 1.8 will not be displayed.

Second Order Effects

You can *Include the effects from second order theory* according to clauses 4.4 and 4.5. When you design, for example, a frame whose governing buckling mode is represented by lateral displacement, you can determine the internal forces according to linear static analysis and increase them with the appropriate factors. If you increase the bending moment, this does not affect the flexural-buckling analysis according to [1], which is performed by using the axial forces.

The *Determination of factor* β_m can be conservative according to clause 4.4.2.2 a) or approximate according to clause 4.4.2.2 b) by matching the distribution of the bending moment along the member according to figure 4.4.2.2 in the Australian standard. The moment distributions can be displayed via the [Info] button.

Structure Type

According to clause 4.4.2, the type of member is required for determination of moment amplification factor. The check boxes of this dialog section define whether there are *Sway* or braced members in the model.

Additionally, the *Elastic buckling factor* λ_c of whole frame (model) according to clause 4.7.2 can be specified.

Lateral-torsional Buckling

If transverse loads exist, it is important to define where those forces are acting on the cross-section: Depending on the *Load Application* point, transverse loads can be stabilizing or destabilizing, and in this way they can critically influence the ideal critical moment. These settings have an effect on the determination of the load height factor K_l (see [1] table 5.6.3(2)).

Set of Members - Member-Like Input

The input of stability data of sets of members can be realised as member-like (table 1.6) or as general (tables 1.7 and 1.8). It is recommended to apply the RF-STEEL AS design only for straight sets of members. If the default *Do not use member-like input* is set, the support conditions have to be defined in table 1.7 for all sets of members. The factors K_r of table 1.6 will not be used.

The option *Use for all sets of members* makes it possible to define all stability data for sets of members in table 1.6 – analogically to table 1.5 for single members. In this case, tables 1.7 and 1.8 are not displayed. The factors K_r and member types y-y and z-z as defined in table 1.6 are used to determine the support conditions β, u_y, φ_x, φ_z, and ω.

It is possible to *Use only for Straight Sets of Members* the member-like input. Additionally, equal cross-section parameters must apply. The factors K_r as defined in table 1.6 are applied; tables 1.7 and 1.8 are not be displayed for straight sets. This option can be used e.g. for continuous beams. Please note that the factor K_r is identical for every segment or partial member of the set.

The last option *Use only to straight sets of members without intermediate restraints* applies the member-like input only to sets of members which have supports or restraints at their ends that have been defined in RFEM. Thus, it is possible 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. Tables 1.7 and 1.8 are not displayed for straight sets that have no intermediate supports.

Welded I and H Sections Fabrications Acc. to 6.3.3

For the fabrication of welded I and H sections, rolled or flame-cut plates can be used. The member section constant α_b depends on the fabrication of the plates. It is given in table 6.3.3 of the Australian standard. The type of fabrication which is set in this dialog box applies to <u>all</u> cross-sections of the design case.

Limit Load for Special Cases

To design unsymmetrical cross-sections for intended axial compression according to [1] 6.3, it is possible to neglect *small moments* about the major and the minor axis by settings defined in this dialog section.

In the same way, you can switch off small *compression forces* for the pure design of bending by defining a limit ratio for N^* to $\Phi N_{s,pl}$.

Intended *torsion* is not clearly specified in AS 4100. If a torsional stress is available that is not exceeding the shear stress ratio of 5 % 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 table. No stability analysis is carried out. Nevertheless, the cross-section designs are performed. These limit settings are <u>not</u> part of the Australian standard. Changing the limits is in the responsibility of the program user.

3.1.3 Serviceability

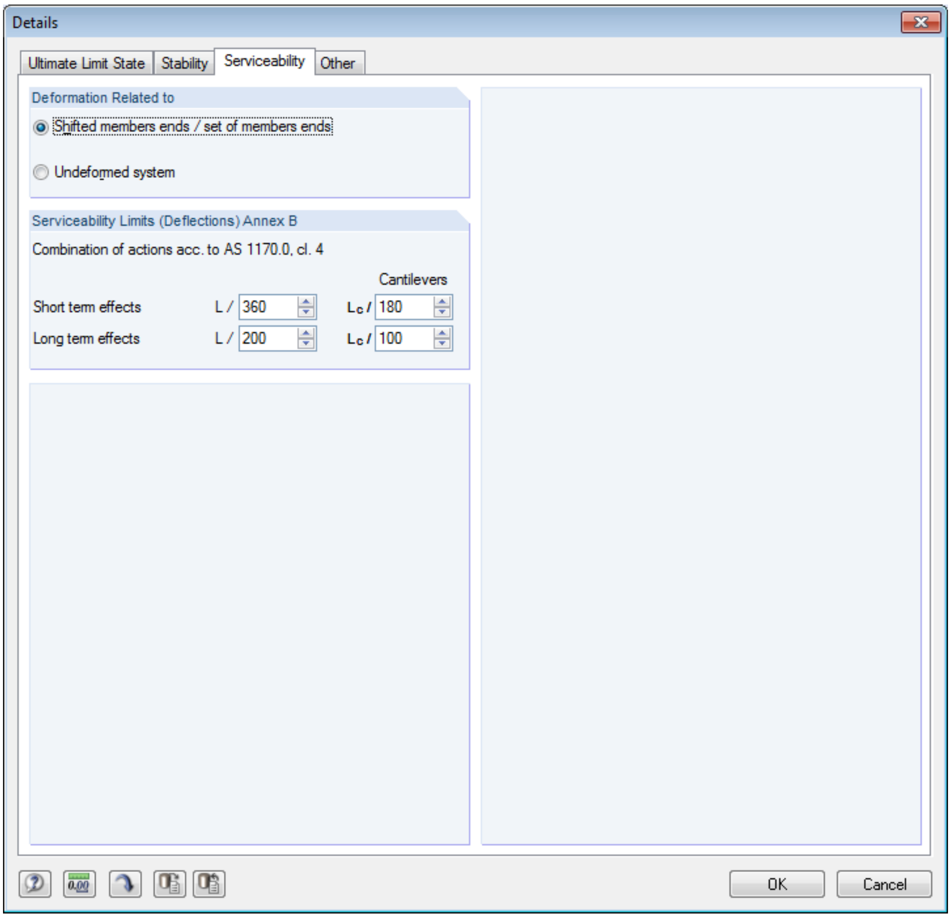


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Related to

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

Serviceability Limits (Deflections) Annex B

In this dialog section, the limit deformations can be checked and, if necessary, adjusted.

3.1.4 Other

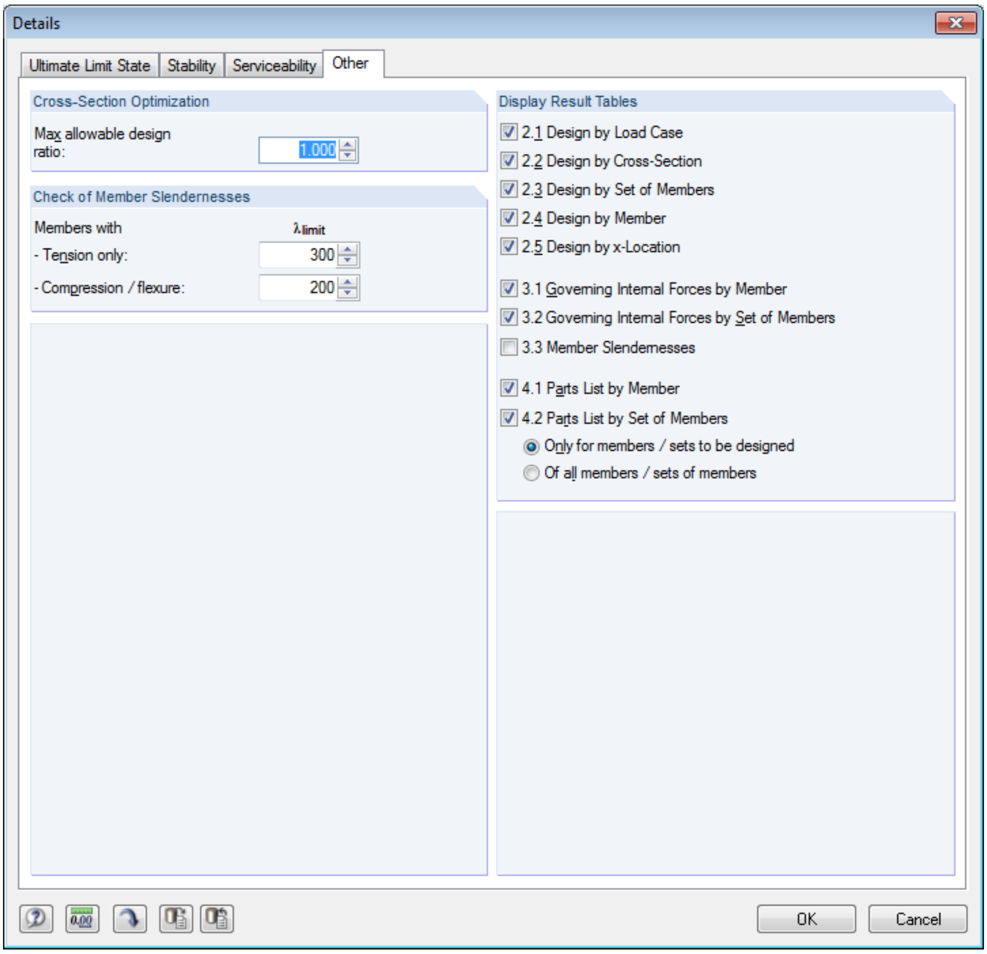


Figure 3.4: Dialog box *Details*, tab *Other*

Cross-Section Optimisation

The optimisation 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

The two fields allow for the input of limit values λ_{limit} 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.

In table 3.3, the limit values are compared to the real member slendernesses. This table is available only after the calculation (see chapter 4.8, page 40) and if the corresponding check box is selected in the dialog section *Display Result Tables*.

Display Result Tables

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

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

3.2 Start Calculation

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

RF-STEEL AS 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: In the dialog box *To Calculate* (menu *Calculate* → *To Calculate*), design cases of the add-on modules like load cases and load combinations are listed.

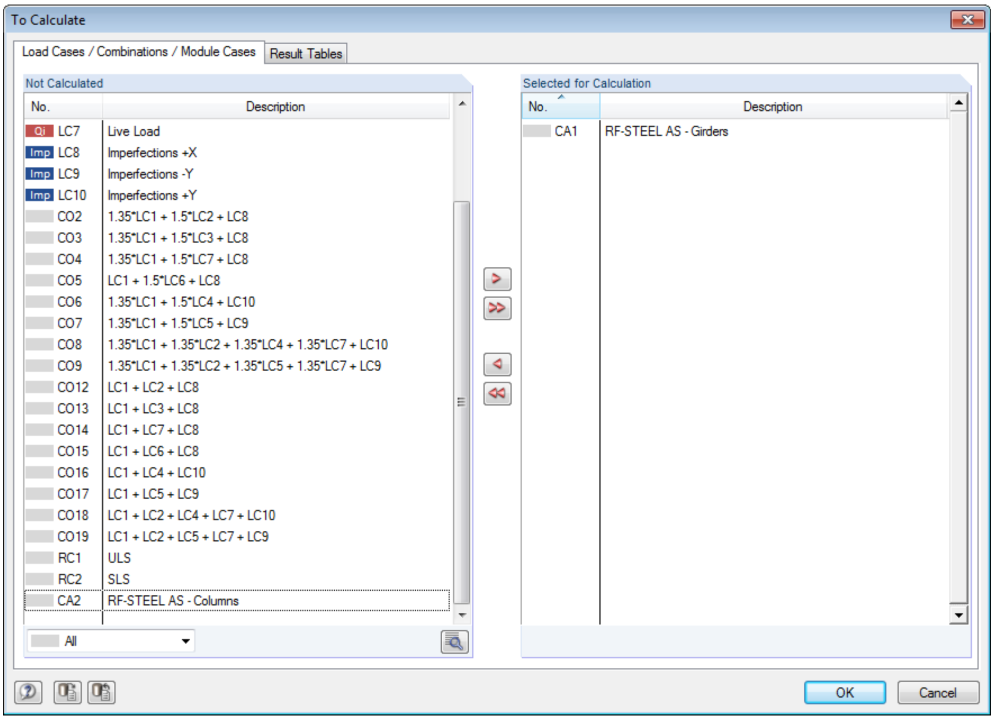
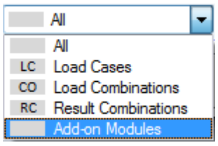


Figure 3.5: Dialog box *To Calculate*



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

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

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

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

4. Results

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

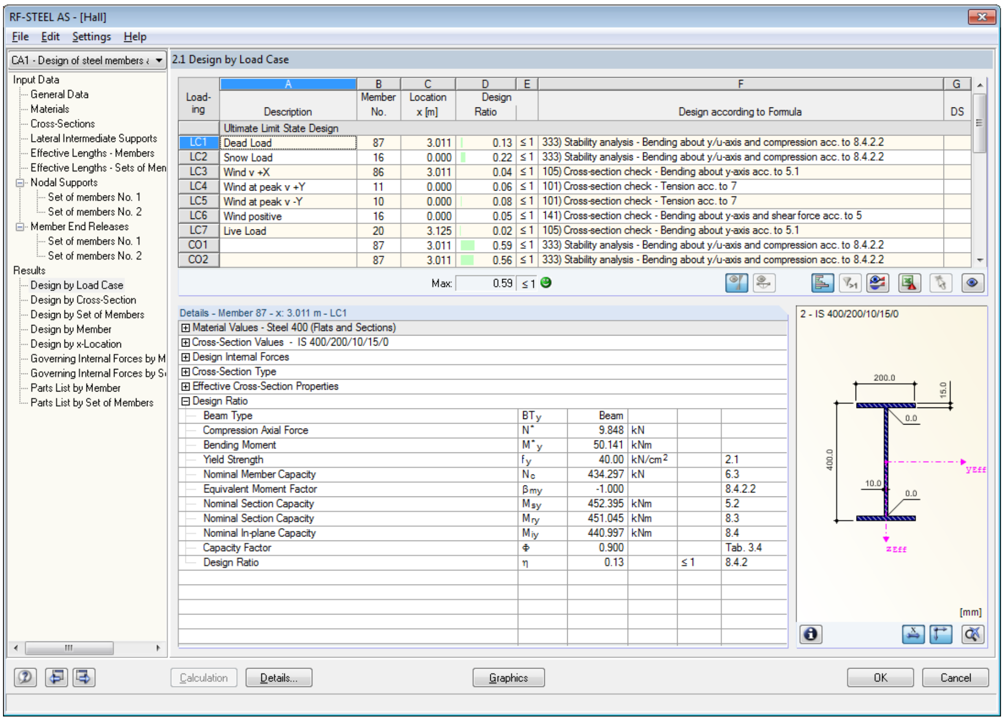


Figure 4.1: Results table with designs and intermediate values

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

The tables 3.1 and 3.2 list the governing internal forces. Table 3.3 informs you about the member slendernesses. In the last two results tables, 4.1 and 4.2, parts lists are displayed by member and set of members.

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

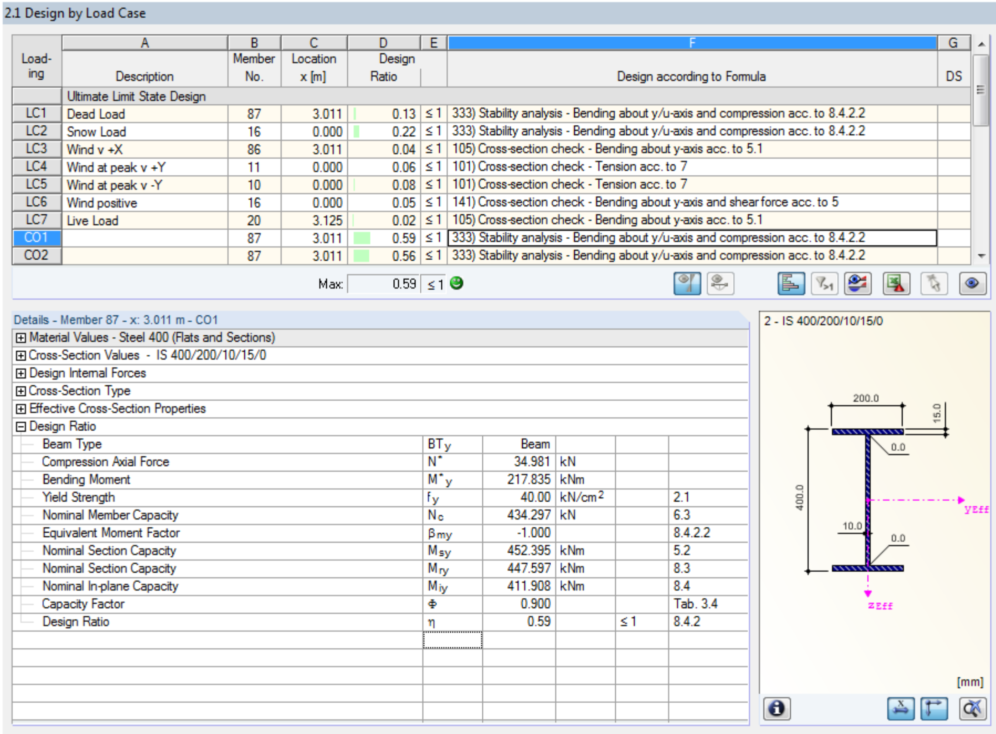
Click [OK] to save the results. Thus you exit RF-STEEL AS and return to the main program.

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

4.1 Design by Load Case

The upper part of this table offers a summery, sorted by load cases, load and result combinations of the governing designs. Furthermore, the list contains ultimate limit state, serviceability and stability design.

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



2.1 Design by Load Case

Loading	A Description	B Member No.	C Location x [m]	D Design Ratio	E	F Design according to Formula	G DS
	Ultimate Limit State Design						
LC1	Dead Load	87	3.011	0.13	≤ 1	333) Stability analysis - Bending about y/u-axis and compression acc. to 8.4.2.2	
LC2	Snow Load	16	0.000	0.22	≤ 1	333) Stability analysis - Bending about y/u-axis and compression acc. to 8.4.2.2	
LC3	Wind v +X	86	3.011	0.04	≤ 1	105) Cross-section check - Bending about y-axis acc. to 5.1	
LC4	Wind at peak v +Y	11	0.000	0.06	≤ 1	101) Cross-section check - Tension acc. to 7	
LC5	Wind at peak v -Y	10	0.000	0.08	≤ 1	101) Cross-section check - Tension acc. to 7	
LC6	Wind positive	16	0.000	0.05	≤ 1	141) Cross-section check - Bending about y-axis and shear force acc. to 5	
LC7	Live Load	20	3.125	0.02	≤ 1	105) Cross-section check - Bending about y-axis acc. to 5.1	
CO1		87	3.011	0.59	≤ 1	333) Stability analysis - Bending about y/u-axis and compression acc. to 8.4.2.2	
CO2		87	3.011	0.56	≤ 1	333) Stability analysis - Bending about y/u-axis and compression acc. to 8.4.2.2	
			Max:	0.59	≤ 1		

Details - Member 87 - x: 3.011 m - CO1

⊞ Material Values - Steel 400 (Flats and Sections)					
⊞ Cross-Section Values - IS 400/200/10/15/0					
⊞ Design Internal Forces					
⊞ Cross-Section Type					
⊞ Effective Cross-Section Properties					
⊟ Design Ratio					
Beam Type	BT_y	Beam			
Compression Axial Force	N^*	34.981	kN		
Bending Moment	M^*_y	217.835	kNm		
Yield Strength	f_y	40.00	kN/cm²		2.1
Nominal Member Capacity	N_c	434.297	kN		6.3
Equivalent Moment Factor	β_{my}	-1.000			8.4.2.2
Nominal Section Capacity	M_{sy}	452.395	kNm		5.2
Nominal Section Capacity	M_{ry}	447.597	kNm		8.3
Nominal In-plane Capacity	M_{iy}	411.908	kNm		8.4
Capacity Factor	Φ	0.900			Tab. 3.4
Design Ratio	η	0.59		≤ 1	8.4.2

Figure 4.2: Table 2.1 *Design by Load Case*

Description

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

Member No.

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

Location x

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

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

Design

Max: 0.98 ≤ 1

Columns D and E display the design conditions according to AS 4100-1998.

The length of the coloured scale represents the respective stress ratio.

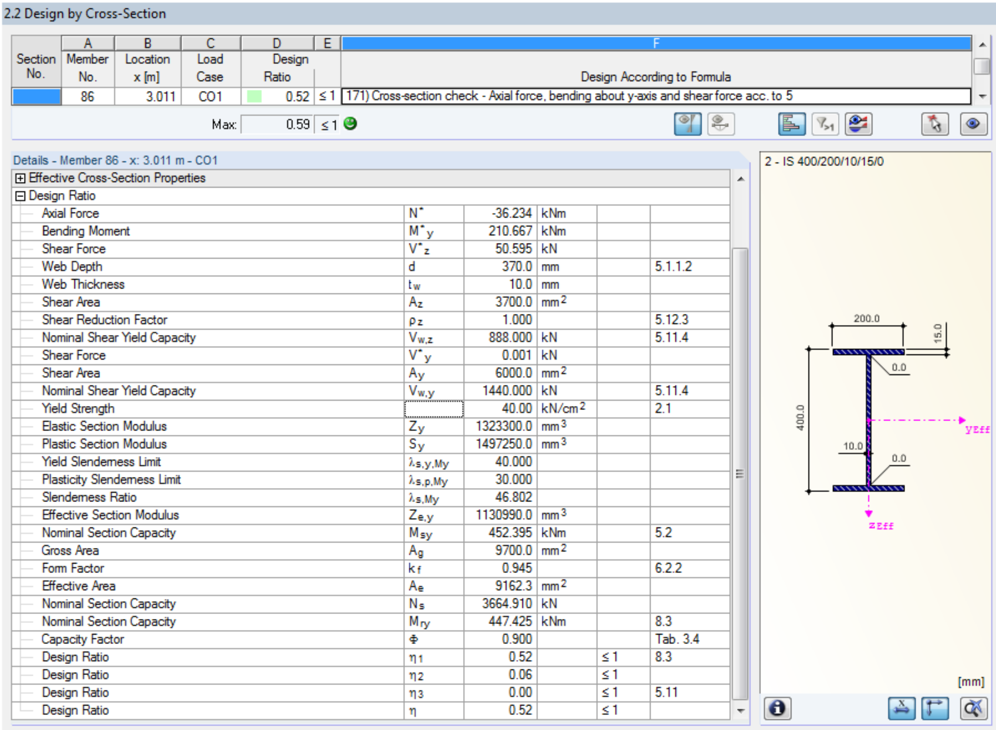
Design according to Formula

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

DS

The final column contains information about the respective design-relevant design situation (*DS*): *ULS* (ultimate limit state) or one of the two design situations for serviceability (*Short Term Effects, Long Term Effects*) according to the specification in table 1.1 *General Data* (see Figure 2.4, page 11).

4.2 Design by Cross-Section



2.2 Design by Cross-Section

Section No.	A Member No.	B Location x [m]	C Load Case	D Design Ratio	E	F Design According to Formula
	86	3.011	CO1	0.52	≤ 1	171) Cross-section check - Axial force, bending about y-axis and shear force acc. to 5
			Max:	0.59	≤ 1	

Details - Member 86 - x: 3.011 m - CO1

⊞ Effective Cross-Section Properties

⊟ Design Ratio

Axial Force	N^*	-36.234	kNm		
Bending Moment	M^*_y	210.667	kNm		
Shear Force	V^*_z	50.595	kN		
Web Depth	d	370.0	mm		5.1.1.2
Web Thickness	t_w	10.0	mm		
Shear Area	A_z	3700.0	mm²		
Shear Reduction Factor	ρ_z	1.000			5.12.3
Nominal Shear Yield Capacity	$V_{w,z}$	888.000	kN		5.11.4
Shear Force	V^*_y	0.001	kN		
Shear Area	A_y	6000.0	mm²		
Nominal Shear Yield Capacity	$V_{w,y}$	1440.000	kN		5.11.4
Yield Strength		40.00	kN/cm²		2.1
Elastic Section Modulus	Z_y	1323300.0	mm³		
Plastic Section Modulus	S_y	1497250.0	mm³		
Yield Slenderness Limit	$\lambda_{s,y,My}$	40.000			
Plasticity Slenderness Limit	$\lambda_{s,p,My}$	30.000			
Slenderness Ratio	$\lambda_{s,My}$	46.802			
Effective Section Modulus	$Z_{e,y}$	1130990.0	mm³		
Nominal Section Capacity	M_{sy}	452.395	kNm		5.2
Gross Area	A_g	9700.0	mm²		
Form Factor	k_f	0.945			6.2.2
Effective Area	A_e	9162.3	mm²		
Nominal Section Capacity	N_s	3664.910	kN		
Nominal Section Capacity	M_{ry}	447.425	kNm		8.3
Capacity Factor	Φ	0.900			Tab. 3.4
Design Ratio	η_1	0.52		≤ 1	8.3
Design Ratio	η_2	0.06		≤ 1	
Design Ratio	η_3	0.00		≤ 1	5.11
Design Ratio	η	0.52		≤ 1	

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

This table 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 you have 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

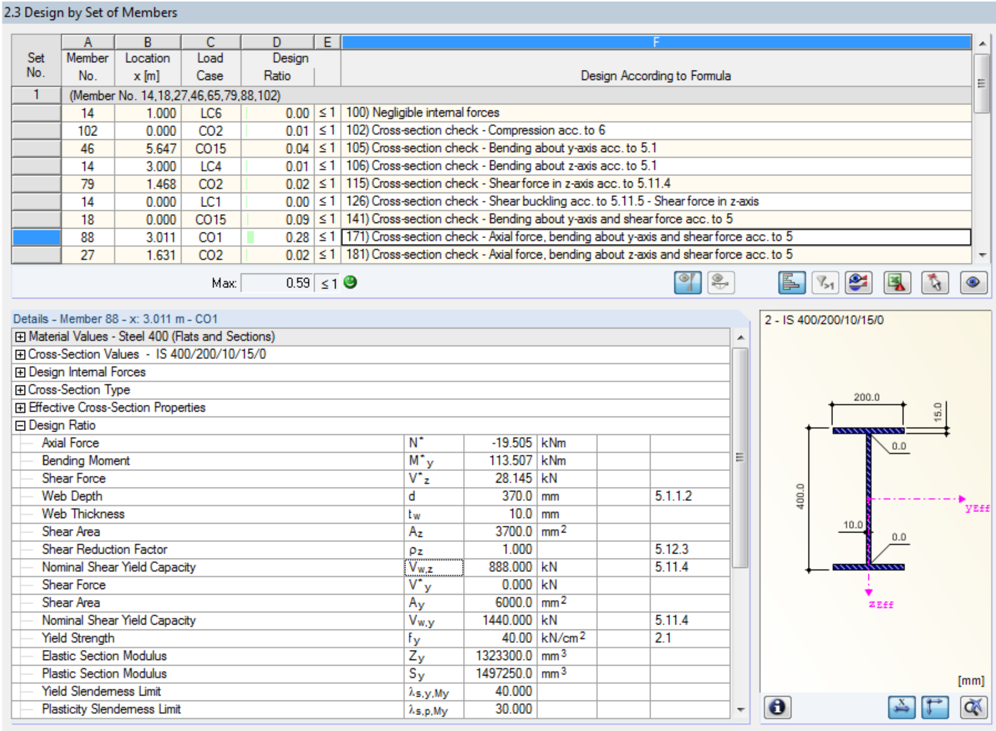


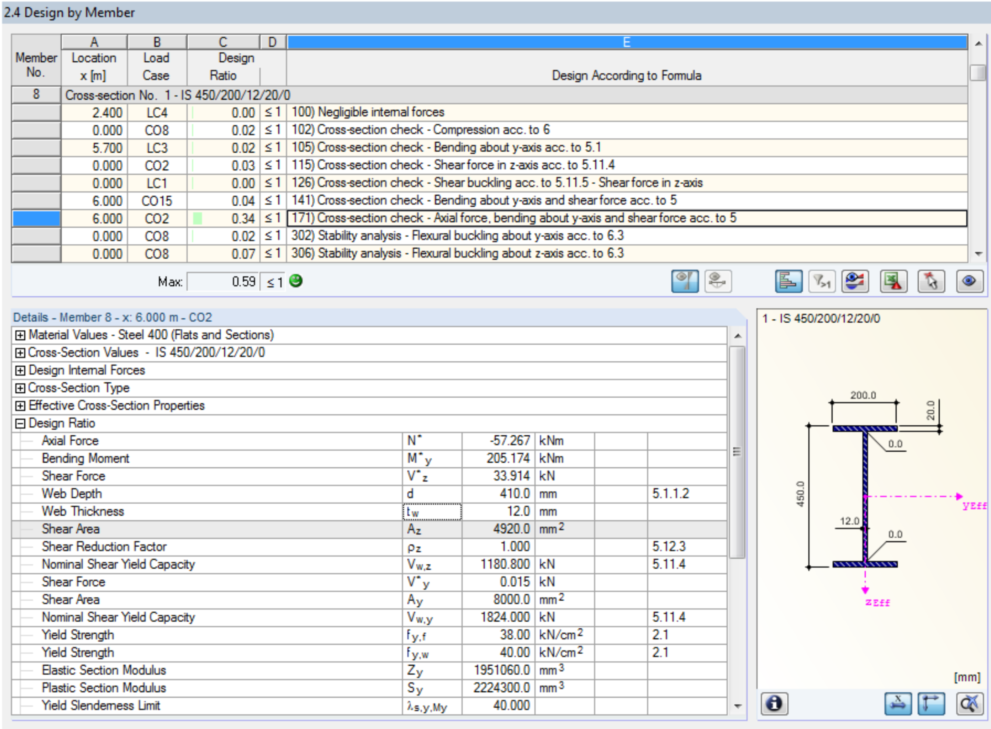
Figure 4.4: Table 2.3 *Design by Set of Members*

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

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

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

4.4 Design by Member



2.4 Design by Member

Member No.	A Location x [m]	B Load Case	C Design Ratio		D	E Design According to Formula
8	Cross-section No. 1 - IS 450/200/12/20/0					
	2.400	LC4	0.00	≤ 1		100) Negligible internal forces
	0.000	CO8	0.02	≤ 1		102) Cross-section check - Compression acc. to 6
	5.700	LC3	0.02	≤ 1		105) Cross-section check - Bending about y-axis acc. to 5.1
	0.000	CO2	0.03	≤ 1		115) Cross-section check - Shear force in z-axis acc. to 5.11.4
	0.000	LC1	0.00	≤ 1		126) Cross-section check - Shear buckling acc. to 5.11.5 - Shear force in z-axis
	6.000	CO15	0.04	≤ 1		141) Cross-section check - Bending about y-axis and shear force acc. to 5
	6.000	CO2	0.34	≤ 1		171) Cross-section check - Axial force, bending about y-axis and shear force acc. to 5
	0.000	CO8	0.02	≤ 1		302) Stability analysis - Flexural buckling about y-axis acc. to 6.3
	0.000	CO8	0.07	≤ 1		306) Stability analysis - Flexural buckling about z-axis acc. to 6.3
		Max:	0.59	≤ 1		

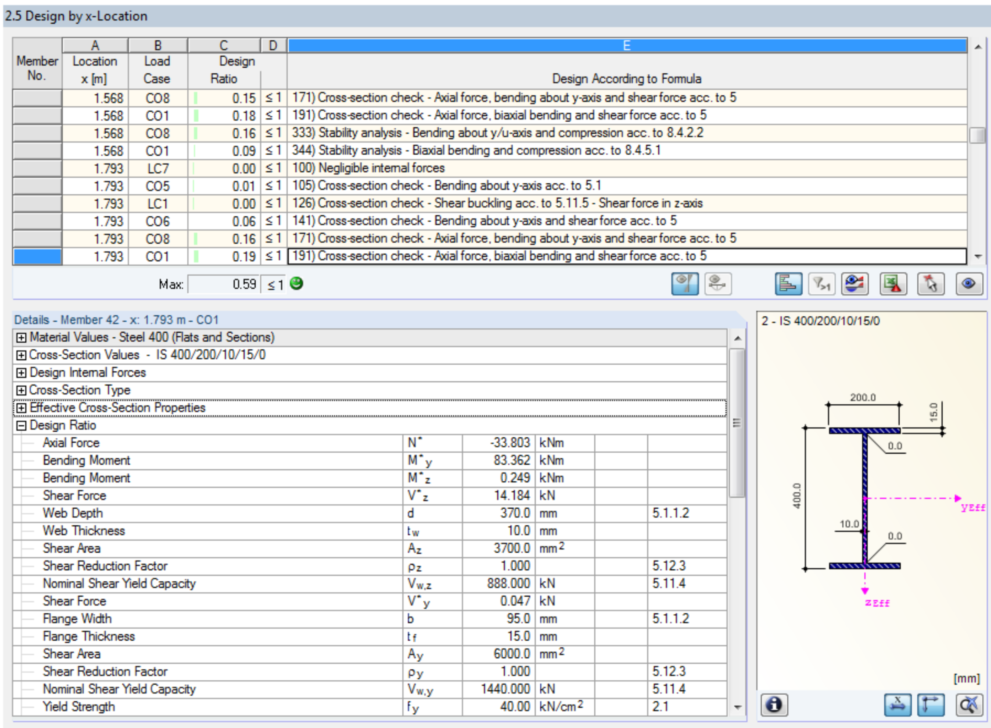
Details - Member 8 - x: 6.000 m - CO2

⊞ Material Values - Steel 400 (Flats and Sections)				
⊞ Cross-Section Values - IS 450/200/12/20/0				
⊞ Design Internal Forces				
⊞ Cross-Section Type				
⊞ Effective Cross-Section Properties				
⊟ Design Ratio				
Axial Force	N^*	-57.267	kNm	
Bending Moment	M^*_y	205.174	kNm	
Shear Force	V^*_z	33.914	kN	
Web Depth	d	410.0	mm	5.1.1.2
Web Thickness	t_w	12.0	mm	
Shear Area	A_z	4920.0	mm²	
Shear Reduction Factor	ρ_z	1.000		5.12.3
Nominal Shear Yield Capacity	$V_{w,z}$	1180.800	kN	5.11.4
Shear Force	V^*_y	0.015	kN	
Shear Area	A_y	8000.0	mm²	
Nominal Shear Yield Capacity	$V_{w,y}$	1824.000	kN	5.11.4
Yield Strength	$f_{y,f}$	38.00	kN/cm²	2.1
Yield Strength	$f_{y,w}$	40.00	kN/cm²	2.1
Elastic Section Modulus	Z_y	1951060.0	mm³	
Plastic Section Modulus	S_y	2224300.0	mm³	
Yield Slenderness Limit	$\lambda_{s,y,My}$	40.000		

Figure 4.5: Table 2.4 *Design by Member*

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

4.5 Design by x-Location



2.5 Design by x-Location

Member No.	A Location x [m]	B Load Case	C Design Ratio		D	E Design According to Formula
	1.568	CO8	0.15	≤ 1		171) Cross-section check - Axial force, bending about y-axis and shear force acc. to 5
	1.568	CO1	0.18	≤ 1		191) Cross-section check - Axial force, biaxial bending and shear force acc. to 5
	1.568	CO8	0.16	≤ 1		333) Stability analysis - Bending about y/u-axis and compression acc. to 8.4.2.2
	1.568	CO1	0.09	≤ 1		344) Stability analysis - Biaxial bending and compression acc. to 8.4.5.1
	1.793	LC7	0.00	≤ 1		100) Negligible internal forces
	1.793	CO5	0.01	≤ 1		105) Cross-section check - Bending about y-axis acc. to 5.1
	1.793	LC1	0.00	≤ 1		126) Cross-section check - Shear buckling acc. to 5.11.5 - Shear force in z-axis
	1.793	CO6	0.06	≤ 1		141) Cross-section check - Bending about y-axis and shear force acc. to 5
	1.793	CO8	0.16	≤ 1		171) Cross-section check - Axial force, bending about y-axis and shear force acc. to 5
	1.793	CO1	0.19	≤ 1		191) Cross-section check - Axial force, biaxial bending and shear force acc. to 5
		Max:	0.59	≤ 1		

Details - Member 42 - x: 1.793 m - CO1

⊞ Material Values - Steel 400 (Flats and Sections)				
⊞ Cross-Section Values - IS 400/200/10/15/0				
⊞ Design Internal Forces				
⊞ Cross-Section Type				
⊞ Effective Cross-Section Properties				
⊟ Design Ratio				
Axial Force	N^*	-33.803	kNm	
Bending Moment	M^*_y	83.362	kNm	
Bending Moment	M^*_z	0.249	kNm	
Shear Force	V^*_z	14.184	kN	
Web Depth	d	370.0	mm	5.1.1.2
Web Thickness	t_w	10.0	mm	
Shear Area	A_z	3700.0	mm²	
Shear Reduction Factor	ρ_z	1.000		5.12.3
Nominal Shear Yield Capacity	$V_{w,z}$	888.000	kN	5.11.4
Shear Force	V^*_y	0.047	kN	
Flange Width	b	95.0	mm	5.1.1.2
Flange Thickness	t_f	15.0	mm	
Shear Area	A_y	6000.0	mm²	
Shear Reduction Factor	ρ_y	1.000		5.12.3
Nominal Shear Yield Capacity	$V_{w,y}$	1440.000	kN	5.11.4
Yield Strength	f_y	40.00	kN/cm²	2.1

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

This results table lists the maxima for each member at the locations ***x*** resulting from the division points defined in RFEM:

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

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

	A	B	C	D	E	F	G	H	I
Member No.	Location x [m]	Load-ing	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
1	Cross-section No. 15 - IS 250/250/10/15/0								
	1.500	CO13	-6.973	0.000	0.467	0.000	-0.250	0.000	100) Negligible internal forces
	0.000	CO2	-17.393	-0.003	0.130	0.000	-0.389	-0.009	102) Cross-section check - Compression acc. to 6
	0.150	LC3	-0.018	0.000	0.932	0.000	-1.033	0.000	105) Cross-section check - Bending about y-axis acc. to 5.1
	1.200	LC5	0.000	0.019	0.000	0.001	0.000	-0.607	106) Cross-section check - Bending about z-axis acc. to 5.1
	0.000	LC3	-0.018	0.000	0.992	0.000	-1.177	0.000	126) Cross-section check - Shear buckling acc. to 5.11.5 - She
	0.000	LC3	-0.018	0.000	0.992	0.000	-1.177	0.000	141) Cross-section check - Bending about y-axis and shear forc
	0.000	CO3	-15.210	0.000	1.599	0.000	-2.083	0.001	171) Cross-section check - Axial force, bending about y-axis an
	1.350	CO7	-9.972	-0.038	-0.002	0.001	0.003	-0.873	181) Cross-section check - Axial force, bending about z-axis an
	0.000	CO2	-17.393	-0.003	0.130	0.000	-0.389	-0.009	302) Stability analysis - Flexural buckling about y-axis acc. to 6.
	0.000	CO2	-17.393	-0.003	0.130	0.000	-0.389	-0.009	306) Stability analysis - Flexural buckling about z-axis acc. to 6.
2	Cross-section No. 15 - IS 250/250/10/15/0								
	3.000	LC5	0.000	0.491	0.000	0.001	0.000	-0.292	100) Negligible internal forces
	2.250	CO9	-41.357	0.813	0.007	0.000	0.009	0.189	102) Cross-section check - Compression acc. to 6
	0.000	LC5	0.000	1.691	0.000	0.001	0.000	2.980	106) Cross-section check - Bending about z-axis acc. to 5.1
	0.000	LC3	-0.018	0.000	2.652	0.000	-7.334	0.000	126) Cross-section check - Shear buckling acc. to 5.11.5 - She
	0.000	LC3	-0.018	0.000	2.652	0.000	-7.334	0.000	141) Cross-section check - Bending about y-axis and shear forc
	0.000	CO1	-50.067	-0.007	4.520	0.000	-13.199	-0.034	171) Cross-section check - Axial force, bending about y-axis an
	0.000	CO7	-35.399	2.363	0.003	0.001	-0.003	4.052	181) Cross-section check - Axial force, bending about z-axis an
	0.000	CO4	-49.461	0.001	0.832	0.000	-2.755	0.004	302) Stability analysis - Flexural buckling about y-axis acc. to 6.
	0.000	CO4	-49.461	0.001	0.832	0.000	-2.755	0.004	306) Stability analysis - Flexural buckling about z-axis acc. to 6.
	0.000	CO1	-50.067	-0.007	4.520	0.000	-13.199	-0.034	333) Stability analysis - Bending about y/u-axis and compressio
	0.000	CO7	-35.399	2.363	0.003	0.001	-0.003	4.052	337) Stability analysis - Bending about z-axis and compression a
3	Cross-section No. 12 - TO 80/80/5/5/5/5								
	0.000	LC5	-0.748	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	2.500	CO7	-1.055	0.000	0.000	0.000	0.502	0.000	105) Cross-section check - Bending about y-axis acc. to 5.1
	0.000	CO7	-1.055	0.000	0.401	0.000	0.000	0.000	115) Cross-section check - Shear force in z-axis acc. to 5.11.4
	0.000	LC1	0.004	0.000	0.294	0.000	0.000	0.000	126) Cross-section check - Shear buckling acc. to 5.11.5 - She
4	Cross-section No. 12 - TO 80/80/5/5/5/5								
	0.000	LC4	-0.061	0.000	0.000	0.000	0.000	0.000	100) Negligible internal forces
	2.500	CO9	-0.159	0.000	0.000	0.000	0.497	0.000	105) Cross-section check - Bending about y-axis acc. to 5.1

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

This table displays for each member the governing internal forces that result in maximum stress ratios in each design.

Location x

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

Load Case

This column displays the number of the load case, the load or result combination whose internal forces produce maximum stresses.

Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing maximum stresses 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.	Location x [m]	Loading	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
1	(Member No. 14,18,27,46,65,79,88,102)								
	1.000	LC6	2.500	0.000	1.493	0.000	1.493	0.000	100) Negligible internal forces
	0.000	CO2	-55.735	0.000	17.829	-0.003	0.000	0.000	102) Cross-section check - Compression acc. to 6
	5.647	CO15	-5.856	0.002	0.014	0.000	20.694	0.001	105) Cross-section check - Bending about y-axis acc. to 5.1
	3.000	LC4	0.000	0.001	0.000	0.005	0.000	1.797	106) Cross-section check - Bending about z-axis acc. to 5.1
	1.468	CO2	-18.978	0.002	-16.282	0.001	-0.382	-0.003	115) Cross-section check - Shear force in z-axis acc. to 5.11.4
	0.000	LC1	-31.485	0.000	-7.565	0.000	0.000	0.000	126) Cross-section check - Shear buckling acc. to 5.11.5 - Shear f
	0.000	CO15	-6.813	-0.002	9.353	0.000	-35.294	0.000	141) Cross-section check - Bending about y-axis and shear force a
	3.011	CO1	-19.505	0.000	-28.145	0.001	-113.507	-0.003	171) Cross-section check - Axial force, bending about y-axis and sh
	1.631	CO2	-18.708	-0.060	16.302	-0.002	-0.296	0.272	181) Cross-section check - Axial force, bending about z-axis and sh
	5.700	CO8	-30.271	1.483	-16.594	0.008	-94.986	0.464	191) Cross-section check - Axial force, biaxial bending and shear fo
	0.000	CO2	-55.735	0.000	17.829	-0.003	0.000	0.000	302) Stability analysis - Flexural buckling about y-axis acc. to 6.3
	0.000	CO2	-55.735	0.000	17.829	-0.003	0.000	0.000	306) Stability analysis - Flexural buckling about z-axis acc. to 6.3
	3.011	CO1	-19.505	0.000	-28.145	0.001	-113.507	-0.003	333) Stability analysis - Bending about y/u-axis and compression ac
	5.700	CO8	-30.271	1.483	-16.594	0.008	-94.986	0.464	344) Stability analysis - Biaxial bending and compression acc. to 8.
2	(Member No. 12,17,26,45,64,78,87,100)								
	5.333	LC3	-0.167	0.000	-1.316	0.000	1.040	0.000	100) Negligible internal forces
	0.000	LC5	7.095	-0.013	0.885	0.005	0.291	-0.142	101) Cross-section check - Tension acc. to 7
	0.000	CO9	-80.491	0.002	31.414	0.007	0.000	0.000	102) Cross-section check - Compression acc. to 6
	6.274	CO7	-0.780	-0.018	0.914	0.007	46.520	-0.079	105) Cross-section check - Bending about y-axis acc. to 5.1
	1.631	CO2	-35.111	0.003	-32.055	0.001	-0.340	-0.004	115) Cross-section check - Shear force in z-axis acc. to 5.11.4
	0.000	LC1	-26.749	0.000	-8.219	0.000	0.000	0.000	126) Cross-section check - Shear buckling acc. to 5.11.5 - Shear f
	5.647	CO7	-0.851	-0.017	1.725	0.007	45.692	-0.091	141) Cross-section check - Bending about y-axis and shear force a
	3.011	CO1	-34.981	-0.001	-49.897	0.001	-204.707	-0.003	171) Cross-section check - Axial force, bending about y-axis and sh
	2.121	CO2	-33.550	-0.059	29.755	-0.001	-0.197	0.299	181) Cross-section check - Axial force, bending about z-axis and sh
	2.510	CO2	-33.016	0.051	15.258	-0.001	84.221	0.230	191) Cross-section check - Axial force, biaxial bending and shear fo
	0.000	CO9	-80.491	0.002	31.414	0.007	0.000	0.000	302) Stability analysis - Flexural buckling about y-axis acc. to 6.3
	0.000	CO9	-80.491	0.002	31.414	0.007	0.000	0.000	306) Stability analysis - Flexural buckling about z-axis acc. to 6.3
	3.011	CO1	-34.981	-0.001	-49.897	0.001	-204.707	-0.003	333) Stability analysis - Bending about y/u-axis and compression ac
	2.510	CO2	-33.016	0.051	15.258	-0.001	84.221	0.230	344) Stability analysis - Biaxial bending and compression acc. to 8.

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

This table contains the internal forces that result in the maximum ratios within the design of each set of members.

4.8 Member Slendernesses

3.3 Member Slendernesses

Member No.	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	184.3	16.275	1.000	45.5	65.961
6	Compression / Flexure	3.000	1.000	184.3	16.275	1.000	45.5	65.961
7	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938
8	Compression / Flexure	6.000	1.000	184.3	32.550	1.000	45.5	131.922
9	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938
12	Compression / Flexure	6.000	1.000	184.3	32.550	1.000	45.5	131.922
13	Compression / Flexure	5.000	1.000	30.7	162.938	1.000	30.7	162.938
14	Compression / Flexure	6.000	1.000	184.3	32.550	1.000	45.5	131.922
15	Compression / Flexure	3.011	1.000	165.2	18.231	1.000	45.4	66.267
16	Compression / Flexure	3.011	1.000	165.2	18.231	1.000	45.4	66.267
17	Compression / Flexure	3.011	1.000	165.2	18.231	1.000	45.4	66.267
18	Compression / Flexure	3.011	1.000	165.2	18.231	1.000	45.4	66.267
19	Compression / Flexure	6.274	1.000	107.7	58.240	1.000	63.5	98.841
20	Compression / Flexure	6.250	1.000	188.0	33.251	1.000	47.0	133.049
21	Compression / Flexure	6.250	1.000	188.0	33.251	1.000	47.0	133.049
24	Compression / Flexure	3.262	1.000	165.2	19.751	1.000	45.4	71.793
25	Compression / Flexure	3.262	1.000	165.2	19.751	1.000	45.4	71.793
26	Compression / Flexure	3.262	1.000	165.2	19.751	1.000	45.4	71.793
27	Compression / Flexure	3.262	1.000	165.2	19.751	1.000	45.4	71.793
28	Compression / Flexure	3.546	1.000	129.3	27.432	1.000	63.5	55.865
29	Compression / Flexure	3.000	1.000	129.3	23.208	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: 162.938 ≤ 200

Figure 4.9: Table 3.3 *Member Slendernesses*

Details...

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

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They were determined 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 32).

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

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

4.9 Parts List by Member

Finally, RF-STEEL AS Cross-sections provides a summary of all members that are included in the design case.

4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	15 - IS 250/250/10/15/0	4	3.00	12.00	17.76	0.12	76.15	228.44	0.914
2	12 - TO 80/80/5/5/5/5	25	5.00	125.00	40.00	0.19	11.78	58.88	1.472
3	1 - IS 450/200/12/20/0	4	3.00	12.00	20.11	0.16	101.42	304.27	1.217
4	1 - IS 450/200/12/20/0	6	6.00	36.00	60.34	0.47	101.42	608.53	3.651
5	13 - Circle 24	4	7.81	31.24	2.36	0.01	3.55	27.74	0.111
6	2 - IS 400/200/10/15/0	8	3.01	24.09	38.06	0.23	76.15	229.30	1.834
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 400/200/10/15/0	8	3.26	26.10	41.24	0.25	76.15	248.42	1.987
11	6 - IS 300/250/10/18/0	2	3.55	7.09	11.21	0.08	91.37	324.01	0.648
12	6 - IS 300/250/10/18/0	3	3.00	9.00	14.22	0.10	91.37	274.12	0.822
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	6 - IS 300/250/10/18/0	2	6.55	13.09	20.69	0.15	91.37	598.13	1.196
16	2 - IS 400/200/10/15/0	8	6.27	50.19	79.30	0.49	76.15	477.73	3.822
17	6 - IS 300/250/10/18/0	1	4.09	4.09	6.47	0.05	91.37	374.09	0.374
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 300/250/10/18/0	1	7.09	7.09	11.21	0.08	91.37	648.21	0.648
Sum		102		516.46	512.84	3.41			26.752

Figure 4.10: Table 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 32).

Part No.

The program automatically assigns item numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column displays the respective length of an individual member.

Total Length

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

Surface Area

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

Volume

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

Cross-Section Mass

The *Cross-Section Mass* is related to the length of one meter. For tapered cross-sections, the program averages both cross-section properties.

Mass

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

Total Mass

The final column indicates the total mass of each part.

Sum

At the bottom of the list, you find a summary of the summed up values of column B, D, E, F, and I. The last data field of the column *Total Mass* gives information about the total amount of steel required.

4.10 Parts List by Set of Members

4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1		2	37.10	74.19	119.53	0.80	84.32	3127.97	6.256
Sum		2		74.19	119.53	0.80			6.256

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

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

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

5. Results Evaluation

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

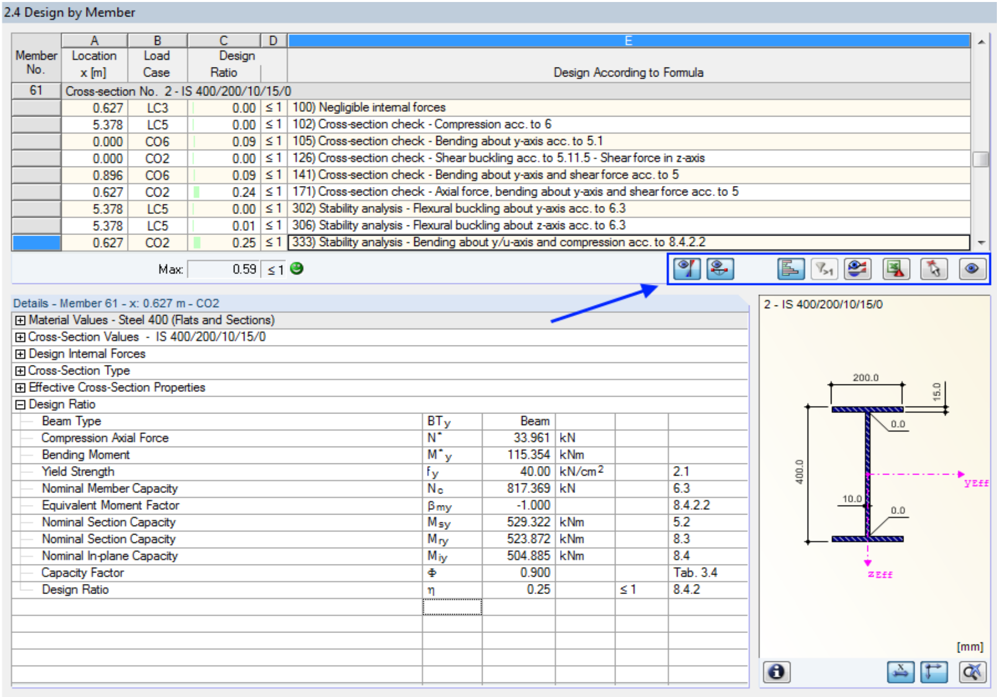


Figure 5.1: Buttons for results evaluation

The buttons are reserved for the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Turns on and off the results of the ultimate limit state design
	Serviceability Limit State Designs	Turns on and off the results of the serviceability limit state design
	Show Colour Bars	Turns on and off the coloured reference scales in the results tables
	Show Rows with Ratio > 1	Displays only the rows where the ratio is more than 1, and thus the design is failed
	Result Diagrams	Opens the window *Result Diagram on Member* ➙ chapter 5.2, page 47
	Excel Export	Exports the table to MS Excel / OpenOffice ➙ chapter 7.4.3, page 58
	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 tables 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 for finding the position of a particular member in the model: The member selected in the RF-STEEL AS results table is highlighted in the selection colour in the background graphic. Furthermore, an arrow indicates the member's x-location that is displayed in the selected table row.

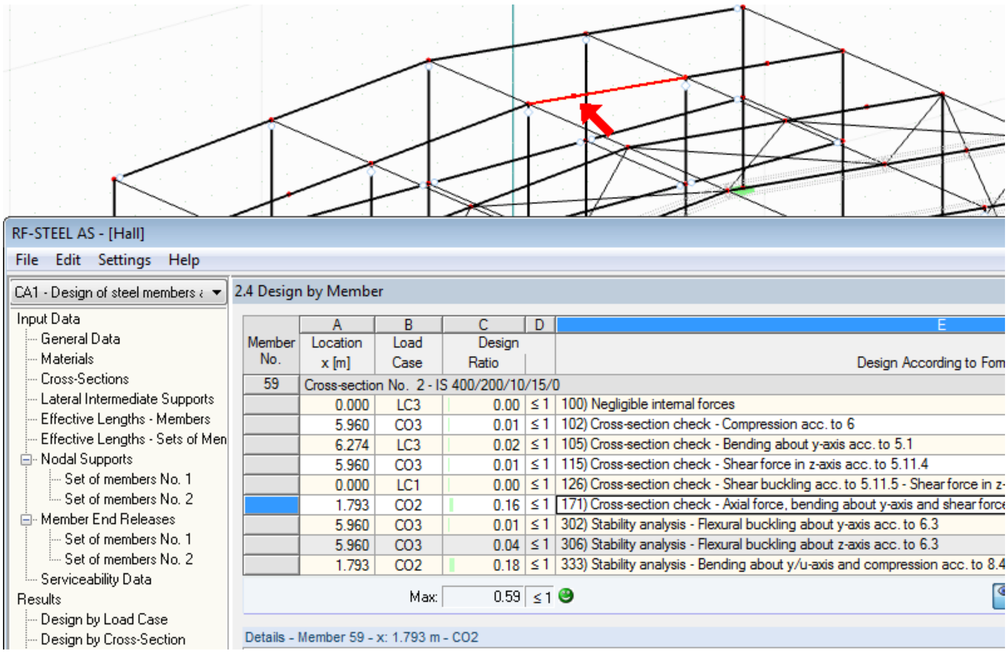


Figure 5.2: Indication of the member and the current *Location x* in the RFEM model

In case you cannot improve the display by moving the RF-STEEL AS 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 AS.

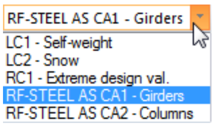
RFEM work window

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

In the *Results* navigator of RF-STEEL AS, you can choose which design ratios of the ULS or SLS analyses are to be displayed graphically.

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

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



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

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

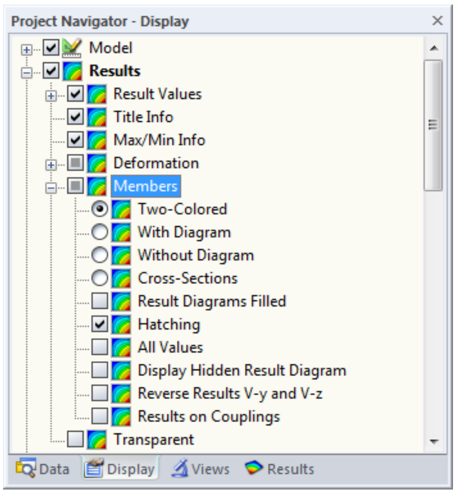


Figure 5.3: *Display* navigator: Results → Members

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

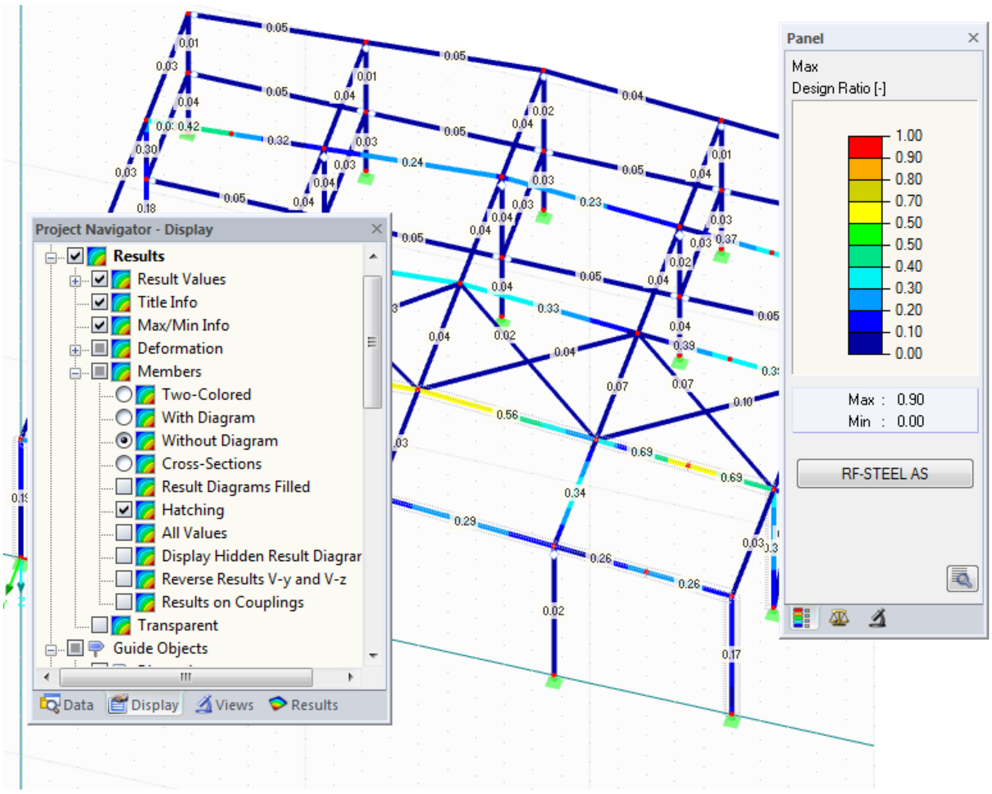


Figure 5.4: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see chapter 6.2, page 50) by clicking the panel button [RF-STEEL AS].

5.2 Result Diagrams

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

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

The result diagrams are 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.

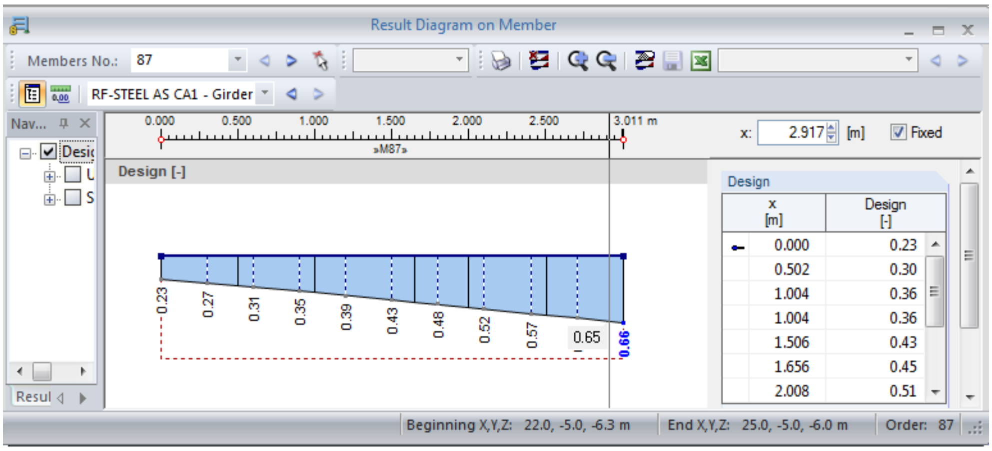
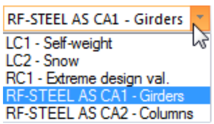


Figure 5.5: Dialog box *Result Diagram on Member*



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

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

5.3 Filter for Results

The RF-STEEL AS results tables 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 option *Visibility* also for RF-STEEL AS (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 can be accessed by clicking [Graphics]. To apply this filter function, the panel must be displayed. If the panel is not active, click

View → Control Panel (Colour 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 (Colour spectrum). As this register is not available for the two-coloured results display, you have to use the *Display* navigator and set the display options *Coloured With/Without Diagram* or *Cross-Sections* first.

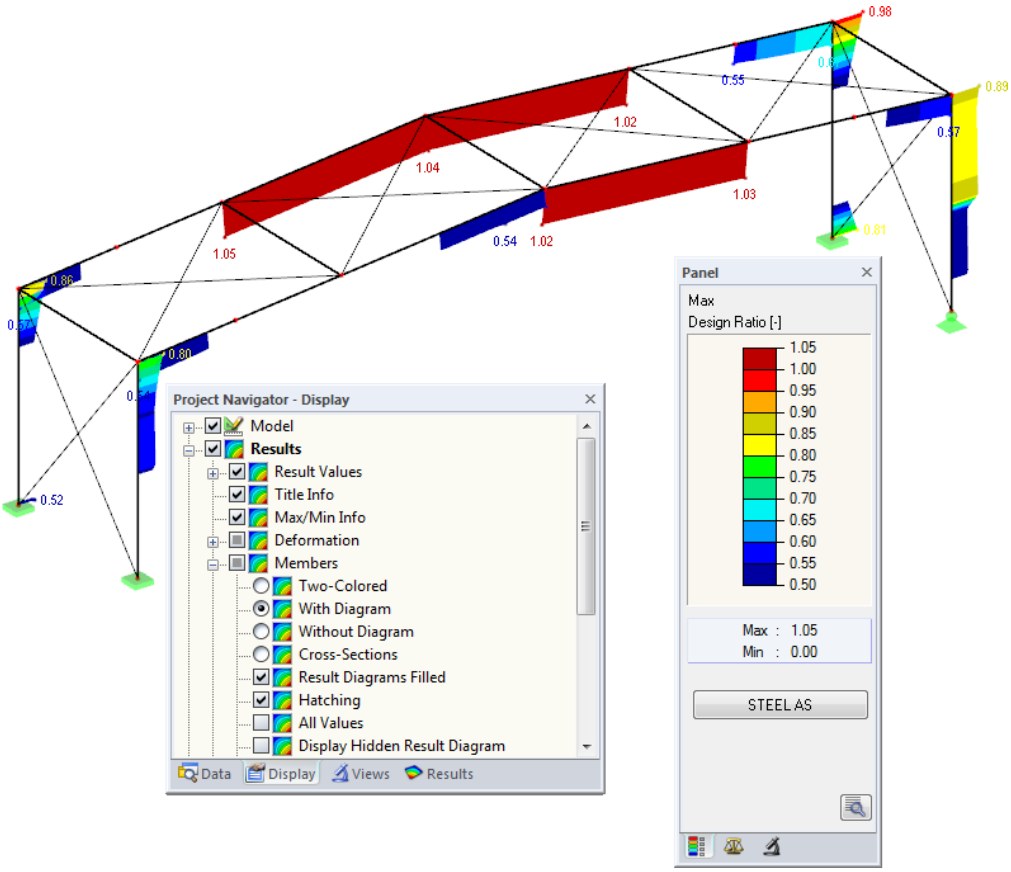


Figure 5.6: Filtering design ratios with adjusted colour spectrum

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

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

Filtering members

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

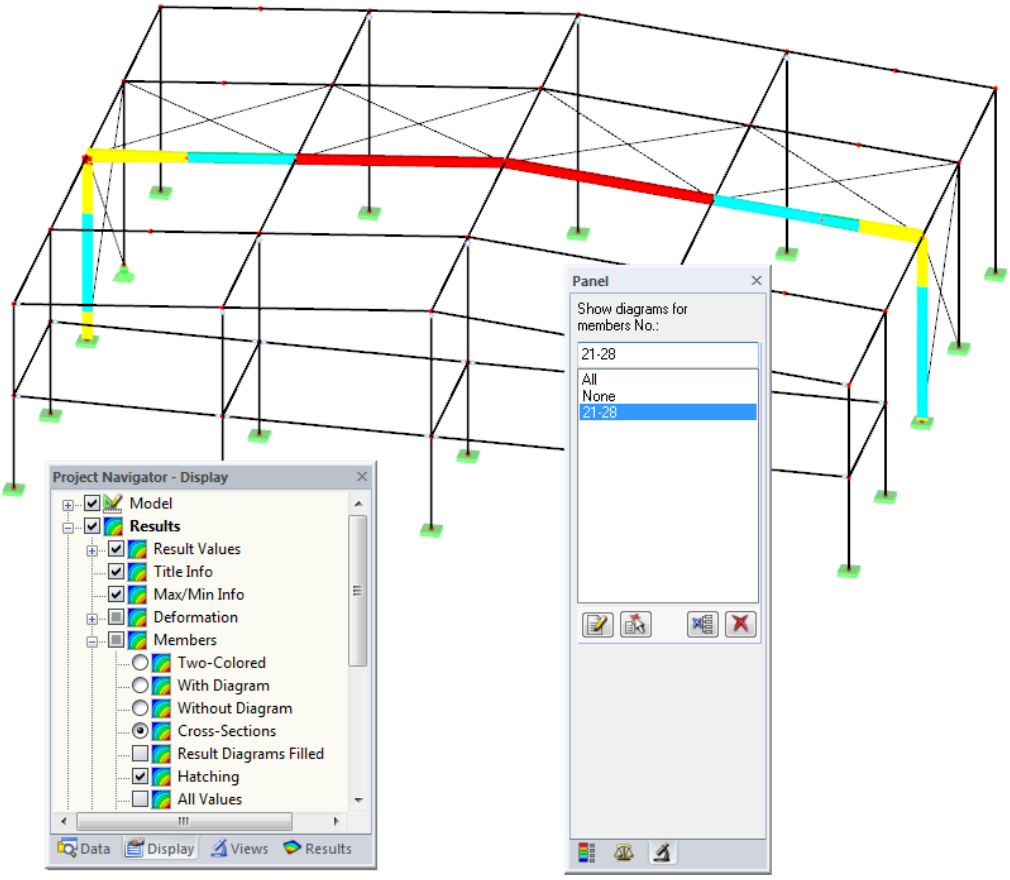


Figure 5.7: Member filter for the stress ratios of a hall frame

Unlike the partial view function, the model is now displayed completely in the graphic. The figure above shows the design ratios of a hall frame. The remaining members are displayed in the model but are shown without design ratios.

6. Printout

6.1 Printout report

Similar to RFEM, the program generates a printout report for the RF-STEEL AS results, to which graphics and descriptions can be added. In the printout report, you can select the data from the design module to 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* provides information concerning the selection of input and output data in add-on modules for the printout.

For large 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 AS Graphic Printout

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

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

Designs in the RFEM model

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

File → Print Graphic

or use the toolbar button shown on the left.

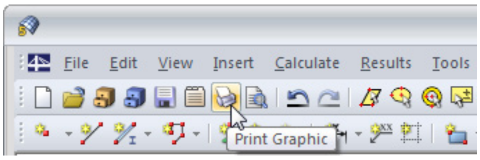


Figure 6.1: Button *Print Graphic* in RFEM toolbar

Result Diagrams

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

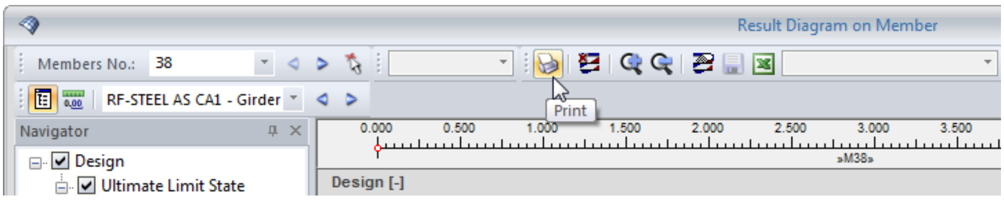


Figure 6.2: Button *Print Graphic* in the dialog box *Result Diagram on Member*

The following dialog box opens:

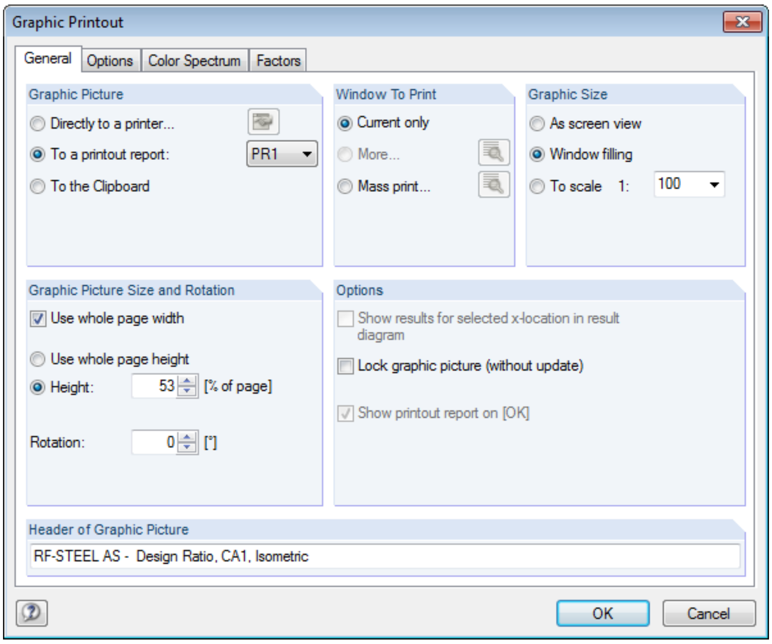
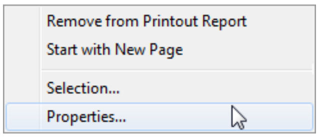


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

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

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



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

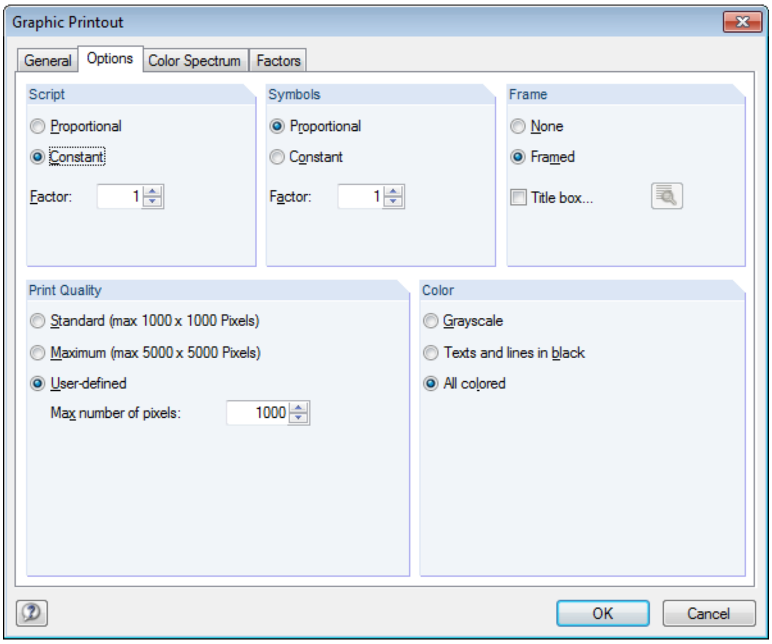


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

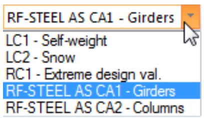
7. General Functions

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 analyse members with particular design specifications (for example changed materials, partial safety factors, optimisation).

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



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

Create a New Design Case

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

File → New Case.

The following dialog box appears:

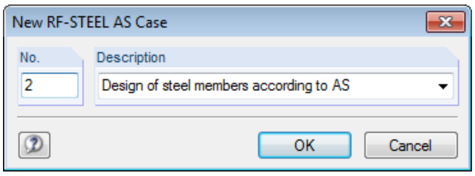


Figure 7.1: Dialog box *New RF-STEEL AS-Case*

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 AS table 1.1 *General Data* where you can enter the design data.

Rename a Design Case

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

File → Rename Case.

The following dialog box appears:

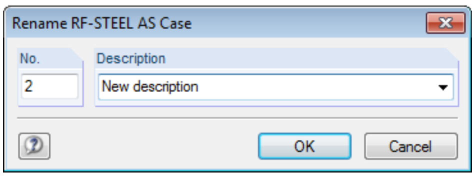


Figure 7.2: Dialog box *Rename RF-STEEL AS-Case*

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

Copy a Design Case

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

File → Copy Case.

The following dialog box appears:

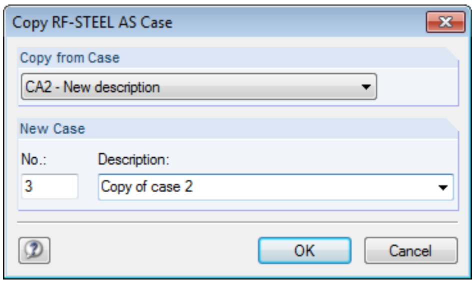


Figure 7.3: Dialog box *Copy RF-STEEL AS-Case*

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

Delete a Design Case

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

File → Delete Case.

The following dialog box appears:

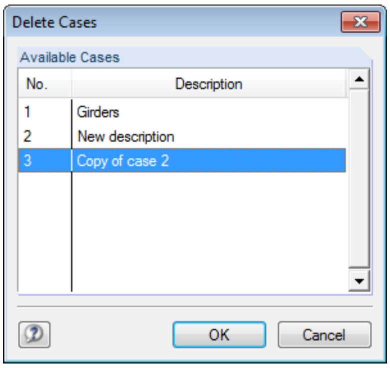
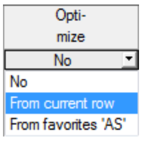


Figure 7.4: Dialog box *Delete Case*

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

7.2 Cross-Section Optimisation



The design module offers you the option to optimise overloaded or little utilised cross-sections. To do this, select in column D of the relevant cross-sections in table 1.3 *Cross-Sections* whether to determine the cross-section *From the current row* or the user-defined *Favorites* (see Figure 2.8, page 14). You can also start the cross-section optimisation out of the results tables by using the context menu.

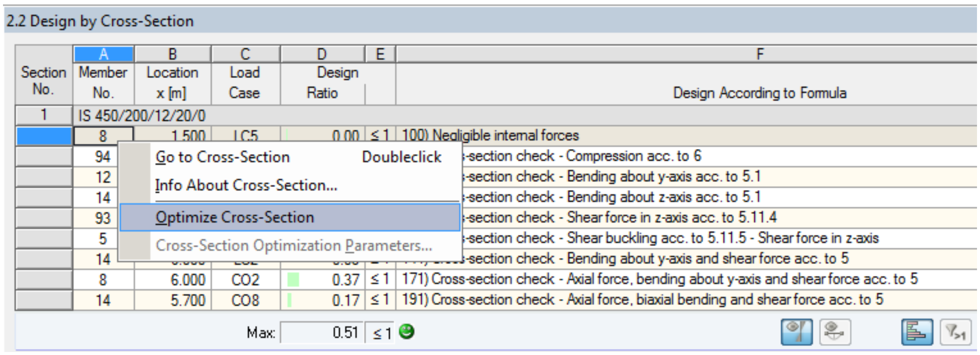


Figure 7.5: Context menu for cross-section optimisation

During the optimisation process, the module determines the cross-section within the same cross-section table that fulfils the analysis requirements in the most optimal way, that is, comes as close as possible to the maximum allowable stress ratio specified in the *Details* dialog box (see Figure 3.4, page 32). The required cross-section properties will be determined with the internal forces from RFEM. If another cross-section proves to be more favourable, this cross-section will be used for the design. Then, the graphic in table 1.3 will show two cross-sections: the original cross-section from RFEM and the optimised one (see Figure 7.7).

For a parameterised cross-section, the dialog box *Optimise* appears if you select the corresponding option.

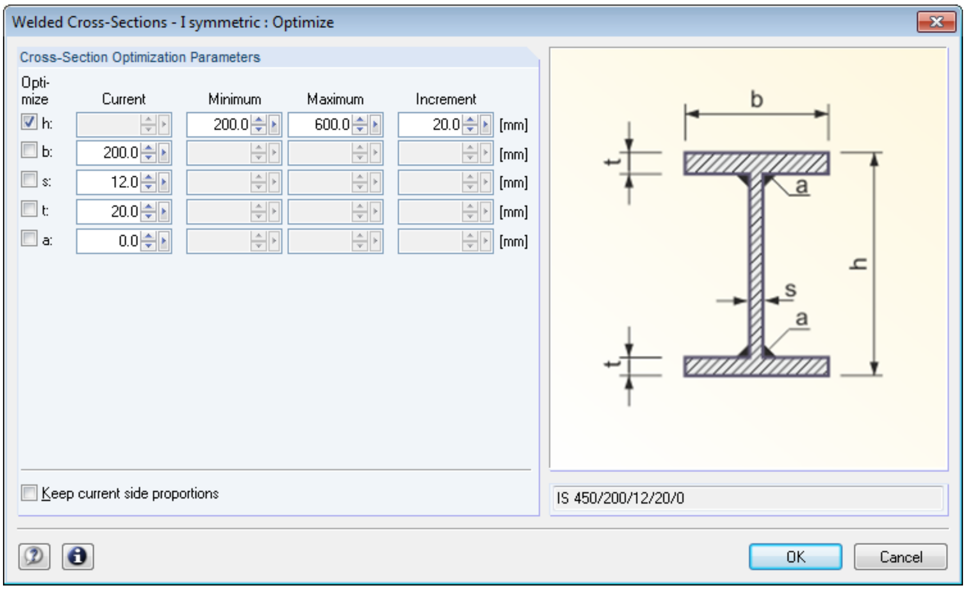


Figure 7.6: Dialog box *Welded Cross-Sections - I symmetric : Optimise*

By selecting the check boxes in the *Optimise* 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 optimisation 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 optimisation.

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

Please note for the optimisation process that the internal forces will not be recalculated automatically with the changed cross-sections: It is up to you to decide which thicknesses or cross-sections should be transferred to RFEM for recalculation. As a result of optimised cross-sections, the 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 optimisation, and then to optimise the cross-sections once again.

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

Edit → Export All Cross-Sections to RFEM.

The context menu in table 1.3 provides options to export optimised cross-sections to RFEM.

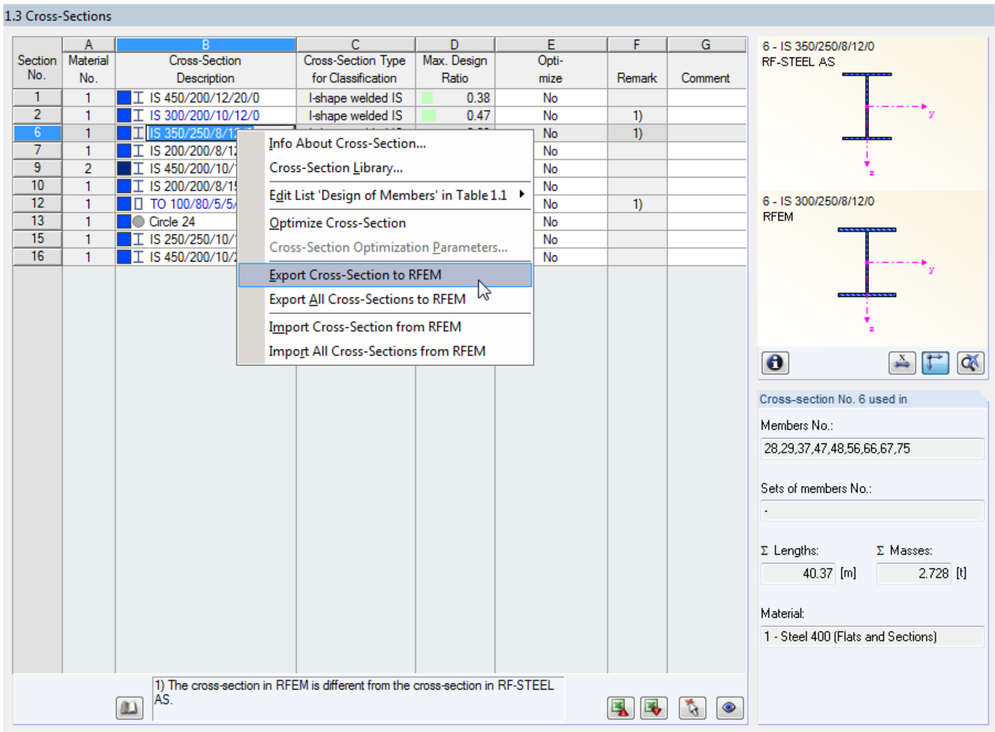


Figure 7.7: Context menu in table 1.3 *Cross-Sections*

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

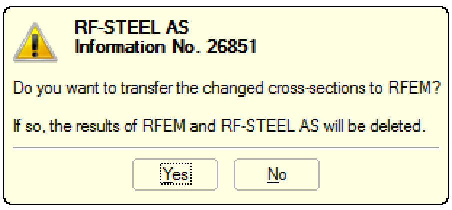


Figure 7.8: Query before transfer of modified cross-sections to RFEM

By confirming the query and starting the [Calculation] subsequently in the RF-STEEL AS module, the internal forces of RFEM and the designs will be determined in one single calculation run.

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

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

7.3 Units and Decimal Places

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

Settings → Units and Decimal Places.

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

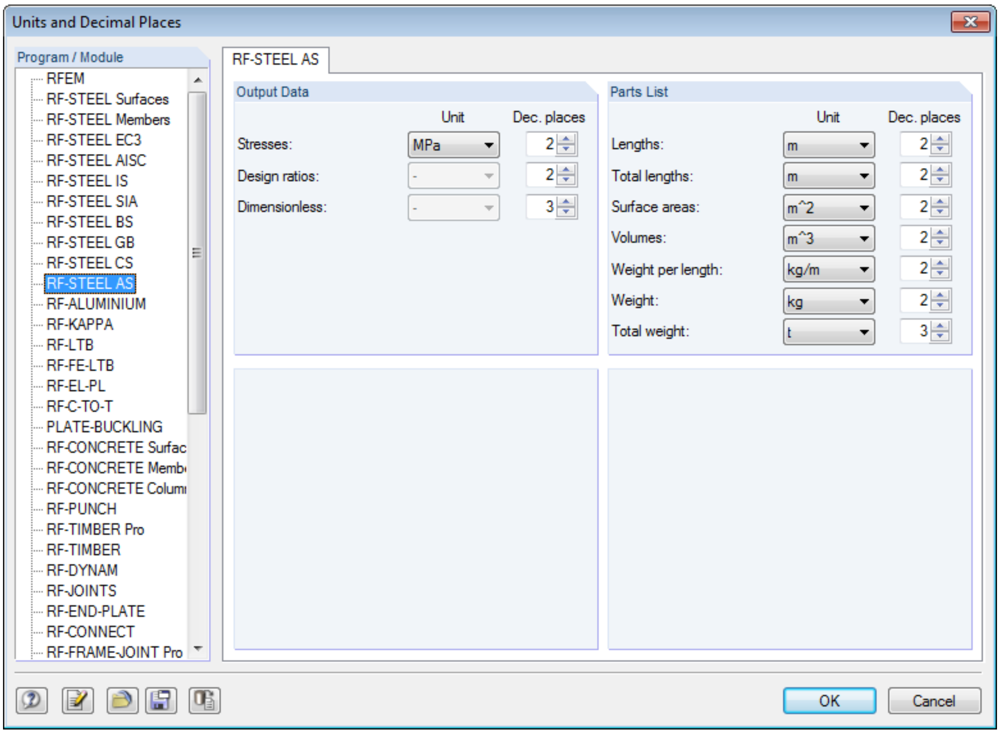


Figure 7.9: Dialog box *Units and Decimal Places*

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

7.4 Data Transfer

7.4.1 Material Export to RFEM

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

Edit → Export All Materials to RFEM.

The modified materials can also be exported to RFEM by using the context menu of table 1.2.

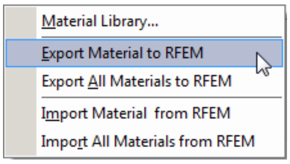


Figure 7.10: Context menu of table 1.2 *Materials*

Before the modified materials are transferred to RFEM, a security query appears as to whether the results of RFEM should be deleted. After you confirm the query and start the [Calculation] subsequently in RF-STEEL AS, the RFEM internal forces and designs will be 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 table 1.2 *Materials*.

7.4.2 Export Effective Lengths to RFEM

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

Edit → Export All Effective Lengths to RFEM.

The modified effective lengths can also be exported to RFEM via the context menu of table 1.5.

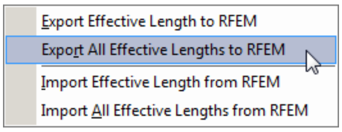


Figure 7.11: Context menu of table 1.5 *Effective Lengths - Members*

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

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

7.4.3 Export of Results

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

Clipboard

To copy cells selected in the results tables 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

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

File → Export to RTF.

The function is described in the RFEM manual, 10.1.11.

Excel / OpenOffice

RF-STEEL AS 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.

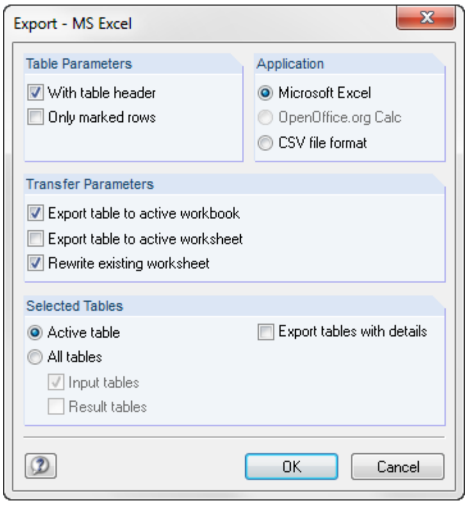


Figure 7.12: Dialog box *Export - MS Excel*

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.

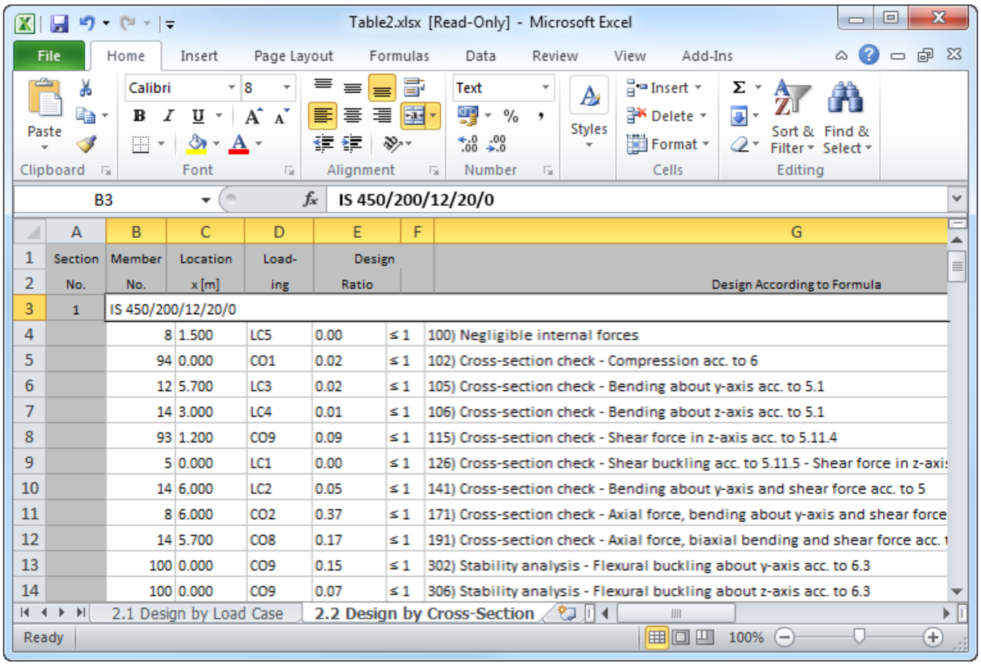


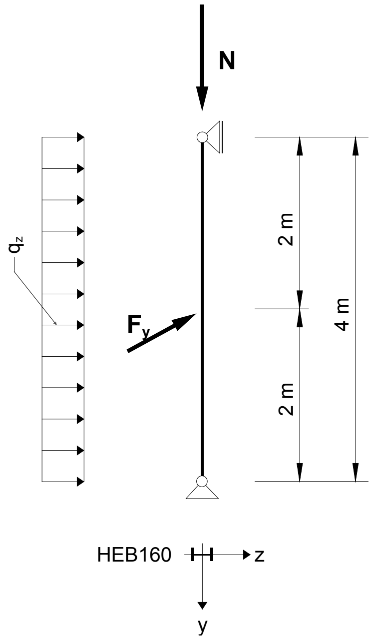
Figure 7.13: Result in *Excel*

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-fold)

Internal forces according to linear static analysis

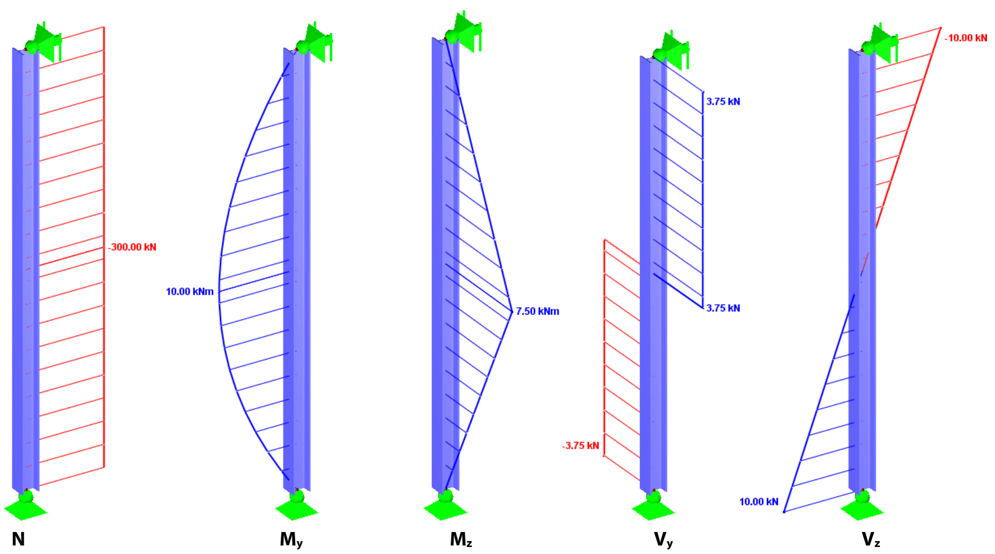


Figure 8.2: Internal forces

Design location (decisive x-location)

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

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

Cross-Section Properties HE-B 160, Steel 250

Property	Symbol	Value	Unit
Cross-section area	A	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_ω	47940000000.00	mm^6
Elastic section modulus	Z_y	311500.00	mm^3
Elastic section modulus	Z_z	111200.00	mm^3
Plastic section modulus	S_y	354000.00	mm^3
Plastic section modulus	S_z	169960.00	mm^3

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

$$N_{om,z} = \frac{\pi^2 \cdot 200000 \cdot 8892000}{4000^2} = 1097.01\,\text{kN}$$

Cross-section class is compact, $A_e = A$, $k_f = 1.0$

$$\lambda_{n,z} = \left(\frac{l_{e,z}}{r_z}\right)\sqrt{k_f}\sqrt{\left(\frac{f_y}{250}\right)} = \left(\frac{4000}{40.5}\right)\sqrt{1.0}\sqrt{\left(\frac{250}{250}\right)} = 98.765$$

$$\alpha_{a,z} = \frac{2100 \cdot (\lambda_{n,z} - 13.5)}{\lambda_{n,z}^2 - 15.3\lambda_{n,z} + 2050} = \frac{2100 \cdot (98.765 - 13.5)}{98.765^2 - 15.3 \cdot 98.765 + 2050} = 17.395$$

The appropriate section constant α_b is given in Table 6.3.3(1).

$\alpha_b = 0.00$ for hot-rolled I-section ($t_f \leq 40$ mm)

$$\lambda_z = \lambda_{n,z} + \alpha_{a,z} \cdot \alpha_b = 98.765 + 17.395 \cdot 0 = 98.765$$

$$\eta_z = 0.00326 \cdot (\lambda_z - 13.5) = 0.00326 \cdot (98.765 - 13.5) = 0.278$$

$$\xi_z = \frac{\left(\frac{\lambda_z}{90}\right)^2 + 1 + \eta_z}{2 \cdot \left(\frac{\lambda_z}{90}\right)^2} = \frac{\left(\frac{98.765}{90}\right)^2 + 1 + 0.278}{2 \cdot \left(\frac{98.765}{90}\right)^2} = 1.031$$

$$\alpha_{c,z} = \xi_z \left(1 - \sqrt{\left[1 - \left(\frac{90}{\xi_z \lambda_z}\right)^2\right]}\right) = 1.031\left(1 - \sqrt{\left[1 - \left(\frac{90}{1.031 \cdot 98.765}\right)^2\right]}\right) = 0.549$$

$$N_s = A_w \cdot f_{y,w} + A_f \cdot f_{y,f} = 1759 \cdot 260 + 3666 \cdot 250 = 1373.84 \text{ kN}$$

$$N_{c,z} = \alpha_{c,z} \cdot N_s = 0.549 \cdot 1373.84 = 754.50 \text{ kN}$$

$$\Phi = 0.9$$

$$\frac{N^*}{\Phi \cdot N_{c,z}} = \frac{300}{0.9 \cdot 754.50} = \underline{\underline{0.442 \le 1}}$$

Result values from RF-STEEL AS calculation

Manufacture		HR			
Compression Axial Force	N*	300	kN		
Nominal Section Capacity	N_s	1373.840	kN		
Form Factor	k_f	1.000			6.2.2
Yield Strength	f_y	250	MPa		2.1
Modulus of Elasticity	E	200000	MPa		
Gross Area	A_g	5425	mm^2		
Second Moment of Area	I_z	8892000	mm^4		
Radius of Gyration	r_z	40.486	mm		
Effective Length	$l_{e,z}$	4000	mm		
Geometrical Slenderness	$\lambda_{g,z}$	98.801			6.3.1
Elastic Flexural Buckling Load	$N_{om,z}$	1097.010	kN		4.6.2
Auxiliary Factor	$\alpha_{a,z}$	17.392			6.3.3
Section Constant	α_b	0			Tab. 6.3.3
Modified Slenderness	$\lambda_{n,z}$	98.801			6.3.3
Slenderness	λ_z	98.801			6.3.3
Auxiliary Factor	η_z	0.278			6.3.3
Auxiliary Factor	ξ_z	1.030			6.3.3
Reduction Factor	$\alpha_{c,z}$	0.549			6.3.3
Capacity Factor	Φ	0.900			Tab. 3.4
Nominal Member Capacity	$N_{c,z}$	754.172	kN		6.3
Design Ratio	η	0.442		≤ 1	6.1

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

$$N_{om,y} = \frac{\pi^2 \cdot 200000 \cdot 24920000}{4000^2} = 3074.38\ \text{kN}$$

Cross-section class is compact, $A_e = A$, $k_f = 1.0$

$$\lambda_{n,y} = \left(\frac{l_{e,y}}{r_y}\right)\sqrt{k_f}\sqrt{\left(\frac{f_y}{250}\right)} = \left(\frac{4000}{67.8}\right)\sqrt{1.0}\sqrt{\left(\frac{250}{250}\right)} = 58.997$$

$$\alpha_{a,y} = \frac{2100 \cdot (\lambda_{n,y} - 13.5)}{\lambda_{n,y}^2 - 15.3\lambda_{n,y} + 2050} = \frac{2100 \cdot (58.997 - 13.5)}{58.997^2 - 15.3 \cdot 58.997 + 2050} = 20.645$$

The appropriate section constant α_b is given in Table 6.3.3(1).

$\alpha_b = 0.00$ for hot-rolled I-section ($t_f \leq 40$ mm)

$$\lambda_y = \lambda_{n,y} + \alpha_{a,y} \cdot \alpha_b = 58.997 + 20.645 \cdot 0 = 58.997$$

$$\eta_y = 0.00326 \cdot (\lambda_y - 13.5) = 0.00326 \cdot (58.997 - 13.5) = 0.148$$

$$\xi_y = \frac{\left(\frac{\lambda_y}{90}\right)^2 + 1 + \eta_y}{2 \cdot \left(\frac{\lambda_y}{90}\right)^2} = \frac{\left(\frac{58.997}{90}\right)^2 + 1 + 0.148}{2 \cdot \left(\frac{58.997}{90}\right)^2} = 1.836$$

$$\alpha_{c,y} = \xi_y\left(1 - \sqrt{\left[1 - \left(\frac{90}{\xi_y \lambda_y}\right)^2\right]}\right) = 1.031\left(1 - \sqrt{\left[1 - \left(\frac{90}{1.836 \cdot 58.997}\right)^2\right]}\right) = 0.814$$

$$N_s = A_w \cdot f_{y,w} + A_f \cdot f_{y,f} = 1759 \cdot 260 + 3666 \cdot 250 = 1373.84\ \text{kN}$$

$$N_{c,y} = \alpha_{c,y} \cdot N_s = 0.814 \cdot 1373.84 = 1118.63\ \text{kN}$$

$$\Phi = 0.9$$

$$\frac{N^*}{\Phi \cdot N_{c,y}} = \frac{300}{0.9 \cdot 1118.63} = \underline{\underline{0.298 \leq 1}}$$

Result values from RF-STEEL AS calculation

Manufacture		HR			
Compression Axial Force	N*	300	kN		
Nominal Section Capacity	N_s	1373.840	kN		
Form Factor	k_f	1.000			6.2.2
Yield Strength	f_y	250	MPa		2.1
Modulus of Elasticity	E	200000	MPa		
Gross Area	A_g	5425	mm^2		

Second Moment of Area	I_y	24920000	mm^4		
Radius of Gyration	r_y	67.776	mm		
Effective Length	$l_{e,y}$	4000	mm		
Geometrical Slenderness	$\lambda_{g,y}$	59.018			6.3.1
Elastic Flexural Buckling Load	$N_{om,y}$	3074.380	kN		4.6.2
Auxiliary Factor	$\alpha_{a,y}$	20.645			6.3.3
Section Constant	α_b	0			Table 6.3.3
Modified Slenderness	$\lambda_{y,z}$	59.018			6.3.3
Slenderness	λ_y	59.018			6.3.3
Auxiliary Factor	η_y	0.148			6.3.3
Auxiliary Factor	ξ_y	1.835			6.3.3
Reduction Factor	$\alpha_{c,y}$	0.814			6.3.3
Capacity Factor	Φ	0.900			Table 3.4
Nominal Member Capacity	$N_{c,y}$	1118.470	kN		6.3
Design Ratio	η	0.298		≤ 1	6.1

Lateral-torsional buckling

Effective length

Twist restraint factor $k_t = 1.0$

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

Load height factor $k_l = 1.0$

Lateral rotation factor $k_r = 1.0$

Effective length according to clause 5.6.3

$$L_w = k_t \cdot k_l \cdot k_r \cdot L = 4000\text{mm}$$

Reference buckling moment

The reference buckling moment for lateral torsional buckling will be determined for this example according to Eq. 5.6.1.1(3), taking into account pinned supports free to warp.

$$M_o = M_{oa} = \sqrt{\left[\left(\frac{\pi^2 \cdot E \cdot I_z}{L_w^2}\right)\left[G \cdot J + \left(\frac{\pi^2 \cdot E \cdot I_w}{L_w^2}\right)\right]\right]}$$

$$M_o = \sqrt{\left[\left(\frac{\pi^2 \cdot 2.0e5 \cdot 8892000}{4000^2}\right)\left[8.0e4 \cdot 312400 + \left(\frac{\pi^2 \cdot 2.0e5 \cdot 4.794e10}{4000^2}\right)\right]\right]} = 184.132\,\text{kNm}$$

Nominal member moment capacity

The nominal member moment capacity of the segment without full lateral restraint, fully restrained at both ends of open cross-section with equal flanges constant along member is calculated according to Eq. 5.6.1.1(1).

The moment modification factor can be determined according to clauses 5.6.1.1 (a) (i) to (iv). We can use data from table 5.6.1 for a parabolic moment diagram: $\alpha_m = 1.13$.

The slenderness reduction factor is calculated according to Eq. 5.6.1.1(2).

The nominal section moment capacity is determined in accordance with clause 5.2 for the gross cross-section:

HEB-160, cross-section type “compact”: $S_y = 354000\text{mm}^3$

$$M_{s,y} = f_y \cdot \min(S_y ; 1.5 \cdot Z_y) = 250 \cdot \min(354000; 1.5 \cdot 311500) = 88.5\text{kNm}$$

RF-STEEL AS calculates value of $M_{s,y}$, taking into account different yield strengths of the material depending on the material thickness. Therefore, we can use:

$$M_{s,y} = 88.86\text{kNm}$$

Now we can calculate the slenderness reduction factor:

$$\alpha_s = 0.6\left[\sqrt{\left[\left(\frac{M_{s,y}}{M_{oa}}\right)^2 + 3\right]} - \left(\frac{M_{s,y}}{M_{oa}}\right)\right] = 0.6\left[\sqrt{\left[\left(\frac{88.86}{184.132}\right)^2 + 3\right]} - \left(\frac{88.86}{184.132}\right)\right] = 0.789$$

Finally, we determine the nominal member moment capacity:

$$M_{b,y} = \alpha_m \cdot \alpha_s \cdot M_{s,y} = 1.13 \cdot 0.789 \cdot 88.86 = 79.251\text{kNm}$$

Interaction of biaxial bending and compression

The design ratio is determined according to clause 8.4.5.1.

Calculation of $M_{i,y}$ $M_{i,z}$ and $M_{o,y}$

Nominal in-plane member moment capacity $M_{i,y}$ acc. to clause 8.4.2.2

$$M_{i,y} = M_{s,y} \cdot \left(1 - \frac{N^*}{\Phi N_{c,y}}\right) = 88.86 \cdot \left(1 - \frac{300}{0.9 \cdot 1118.63}\right) = 65.029\text{kNm}$$

We can use the alternative calculation for doubly symmetric compact I-sections with $k_f = 1.0$:

$$M_{i,y} = M_{s,y} \cdot \left\{\left[1 - \left(\frac{1+\beta_{m,y}}{2}\right)^3\right]\left(1 - \frac{N^*}{\Phi N_{c,y}}\right) + 1.18 \cdot \left(\frac{1+\beta_{m,y}}{2}\right)^3 \sqrt{\left(1 - \frac{N^*}{\Phi N_{c,y}}\right)}\right\}$$

In this equation, we use the equivalent moment factor $\beta_{m,y} = -1.0$ according to clause 4.4.2.2(a).

$$M_{i,y} = 88.86 \cdot \left\{\left[1 - \left(\frac{1-1}{2}\right)^3\right]\left(1 - \frac{300}{0.9 \cdot 1118.63}\right) + 1.18 \cdot \left(\frac{1-1}{2}\right)^3 \sqrt{\left(1 - \frac{300}{0.9 \cdot 1118.63}\right)}\right\}$$

$$M_{i,y} = 62.381\text{kNm}$$

Nominal in-plane member moment capacity $M_{i,z}$ acc. to clause 8.4.2.2

$$M_{s,z} = f_y \cdot \min(S_z ; 1.5 \cdot Z_z) = 250 \cdot \min(170000 ; 1.5 \cdot 111200) = 41.70 \text{kNm}$$

$$M_{i,z} = M_{s,z} \cdot \left(1 - \frac{N^*}{\Phi N_{c,z}}\right) = 41.70 \cdot \left(1 - \frac{300}{0.9 \cdot 754.50}\right) = 25.119 \text{kNm}$$

We can use the alternative calculation for doubly symmetric compact I-sections with $k_f = 1.0$:

$$M_{i,z} = M_{s,z} \cdot \left\{ \left[1 - \left(\frac{1+\beta_{m,z}}{2}\right)^3\right] \left(1 - \frac{N^*}{\Phi N_{c,z}}\right) + 1.18 \cdot \left(\frac{1+\beta_{m,z}}{2}\right)^3 \sqrt{\left(1 - \frac{N^*}{\Phi N_{c,z}}\right)} \right\}$$

In this equation, we use the equivalent moment factor $\beta_{m,z}$ = -1.0 according to clause 4.4.2.2(a).

$$M_{i,z} = 41.70 \cdot \left\{ \left[1 - \left(\frac{1-1}{2}\right)^3\right] \left(1 - \frac{300}{0.9 \cdot 754.50}\right) + 1.18 \cdot \left(\frac{1-1}{2}\right)^3 \sqrt{\left(1 - \frac{300}{0.9 \cdot 754.50}\right)} \right\}$$

$$M_{i,z} = 23.277 \text{kNm}$$

Nominal out-of-plane member moment capacity $M_{o,y}$ acc. to clause 8.4.4.1

$$M_{oy} = M_{b,y} \cdot \left(1 - \frac{N^*}{\Phi N_{c,z}}\right) = 79.251 \cdot \left(1 - \frac{300}{0.9 \cdot 754.50}\right) = 47.740 \text{kNm}$$

Interaction design ratio acc. to clause 8.4.5.1

For $M_{c,y}$, the lesser value of the nominal in-plane member capacity $M_{i,y}$ and the nominal out-of-plane capacity $M_{o,y}$ has to be applied. In our example, $M_{o,y}$ is relevant.

$$\left(\frac{M_y^*}{\Phi \cdot M_{c,y}}\right)^{1.4} + \left(\frac{M_z^*}{\Phi \cdot M_{i,z}}\right)^{1.4} \leq 1$$

$$\left(\frac{10.00}{0.9 \cdot 47.74}\right)^{1.4} + \left(\frac{7.50}{0.9 \cdot 23.277}\right)^{1.4} = \underline{\underline{0.367 \leq 1}}$$

Result values from RF-STEEL AS calculation

Beam Type		Beam			
Beam Type	BT_z	Beam			
Compression Axial Force	N*	300.000	kN		
Nominal Member Capacity	N_c	754.172	kN		6.3
Bending Moment	M^*_y	10.000	kNm		
Maximum Bending Moment	$M^*_{y,segm}$	10.000	kNm		
Modulus of Elasticity	E	200000	MPa		
Shear Modulus	G	80000	MPa		
Second Moment of Area	I_z	8892000	mm^4		
Torsional Constant	J	312400.000	mm^4		
Warping Constant	I_w	4.79400E+10	mm^6		
Segment Length	l	4000	mm		
Twist Restraint Factor	k_t	1.000			Table 5.6.3(1)
Load Height Factor	k_l	1.000			Table 5.6.3(2)
Lateral Rotation Restraint Factor	k_r	1.000			Table 5.6.3(3)
Effective Length	l_e	4000	mm		5.6.2
Section Constant	β	0.000			5.6.1.2 or H.4
Modification Factor	α_m	1.130			5.6
Slenderness Reduction Factor	α_s	0.789			5.6
Amended Elastic Buckling Moment	M_{oa}	184.132	kNm		
Nominal Out-of-plane Capacity	M_{by}	79.268	kNm		5.6
Equivalent Moment Factor	β_{my}	-1.000			8.4.2.2
Nominal Section Capacity	M_{sy}	88.859	kNm		5.2
Nominal Section Capacity	M_{ry}	67.299	kNm		8.3
Nominal In-plane Capacity	M_{iy}	62.377	kNm		8.4
Nominal Out-of-plane Capacity	M_{oy}	44.233	kNm		8.4.2
Nominal Moment Capacity	M_y	44.233	kNm		8.4
Design Component for M_y	η_{My}	0.251			
Bending Moment	M^*_z	7.500	kNm		
Equivalent Moment Factor	β_{mz}	-1.000			8.4.2.2
Nominal Section Capacity	M_{sz}	41.700	kNm		5.2
Nominal Section Capacity	M_{rz}	31.582	kNm		8.3
Nominal In-plane Capacity	M_{iz}	23.269	kNm		8.4
Design Component for M_z	η_{Mz}	0.358			
Capacity Factor	Φ	0.900			
Design Ratio	η	0.382		≤ 1	8.4.5.1

A Literature

[1] AS 4100-1998: Australian Standard, Steel Structures, Standards Australia, 1998

[2] AS 4100 Supplement 1-1999: Steel Structures – Commentary (Supplement to AS 4100-1998), Australian Standard, Steel Structures, Standards Australia, Second Edition, 1999

[3] Steel Structures, Design Manual to AS 4100, First Edition, KIRKE B., AL-JAMEL, I.H., 2004

B Index