



Version
December 2014

Add-on Module

RF-STEEL EC3

Ultimate Limit State, Serviceability, Fire Resistance, and Stability Analyses According to Eurocode 3

Program Description

All rights, including those of translations, are reserved.
No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of DLUBAL SOFTWARE GMBH.



© Dlubal Software GmbH 2015
Am Zellweg 2
D-93464 Tiefenbach
Germany

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



Contents

	Contents	Page
1.	Introduction	3
1.1	Add-on Module RF-STEEL EC3	3
1.2	Using the Manual	4
1.3	Opening RF-STEEL EC3 Add-on Module	4
2.	Input Data	6
2.1	General Data	6
2.1.1	Ultimate Limit State	8
2.1.2	Serviceability Limit State	10
2.1.3	Fire Resistance	11
2.1.4	National Annex (NA)	11
2.2	Materials	15
2.3	Cross-Sections	17
2.4	Intermediate Lateral Restraints	21
2.5	Effective Lengths - Members	22
2.6	Effective Lengths - Sets of Members	26
2.7	Nodal Supports - Set of Members	27
2.8	Member Hinges - Set of Members	30
2.9	Serviceability Data	31
2.10	Fire Resistance - Members	32
2.11	Fire Resistance - Sets of Members	33
2.12	Parameters - Members	34
2.13	Parameters - Sets of Members	41
3.	Calculation	42
3.1	Detail Settings	42
3.1.1	Ultimate Limit State	42
3.1.2	Stability	44
3.1.3	Serviceability	46
3.1.4	Fire Resistance	47
3.1.5	Other	49
3.2	Start Calculation	50
4.	Results	51
4.1	Design by Load Case	52
4.2	Design by Cross-Section	53
4.3	Design by Set of Members	54
4.4	Design by Member	55
4.5	Design by x-Location	56
4.6	Governing Internal Forces by Member	57
4.7	Governing Internal Forces by Set of Members	58
4.8	Members Slendernesses	59
4.9	Parts List by Member	60
4.10	Parts List by Set of Members	61
5.	Results Evaluation	63
5.1	Results in RFEM Model	64
5.2	Result Diagrams	67
5.3	Filter for Results	68
6.	Printout	70



6.1	Printout Report	70
6.2	Graphic Printout	71
7.	General Functions	73
7.1	Design Cases	73
7.2	Cross-Section Optimization	75
7.3	Units and Decimal Places	77
7.4	Data Transfer	78
7.4.1	Exporting Materials to RFEM	78
7.4.2	Exporting Effective Lengths to RFEM	78
7.4.3	Exporting Results	78
8.	Examples	81
8.1	Stability	81
8.2	Fire Resistance	88

1 Introduction

1.1 Add-on Module RF-STEEL EC3



Eurocode 3 [1] describes design, analysis, and construction of steel structures in the member states of the European Union. With the RFEM add-on module RF-STEEL EC3, DLUBAL provides a powerful tool for designing steel structures modeled with member elements. Country-specific regulations are taken into account by National Annexes (NA). In addition to the parameters included in the program, you can define your own limit values or create new National Annexes.

RF-STEEL EC3 can carry out all typical ultimate limit state designs as well as stability and deformation analyses. The program takes into account various actions for the ultimate limit state design. Furthermore, you can choose between the interaction formulas given in the standard. An essential part of the analysis according to Eurocode 3 is the classification of the designed cross-sections into the Classes 1 to 4. In this way, you can check the limitation of the design and rotational capacity by means of the local buckling of cross-section parts. RF-STEEL EC3 determines the c/t -ratios of the cross-section parts subjected to compression stress and performs the classification automatically.

For the stability analysis, you can specify for each member or set of members whether flexural buckling occurs in the y - and/or z -direction. You can also define additional lateral restraints in order to represent the model close to reality. In addition, the stabilizing effect of purlins and sheeting can be taken into account by rotational restraints and shear panels. RF-STEEL EC3 determines the slenderness ratios and elastic critical buckling loads on the basis of boundary conditions. The elastic critical moment for lateral-torsional buckling required for the lateral-torsional buckling analysis can be determined automatically or specified manually. There is the load application point of transverse loads considered, effecting the torsional resistance considerably.

RF-STEEL EC3 can also perform the fire resistance design according to EN 1993-1-2 [2]. The design is carried out on the bearing capacity level according to the simplified calculation method. As fire protection, you can select encasements with various physical properties.

For structures with slender cross-sections, the serviceability limit state represents an important design. The load cases, load combinations, and result combinations can be assigned to different design situations. The limit deformations are preset by the National Annexes and can be adjusted, if necessary. In addition, you can specify reference lengths and precambers that are considered accordingly in the design.

RF-STEEL EC3 also allows you to design structural components made of stainless steel according to EN 1993-1-4 [3].

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

Since RF-STEEL EC3 is integrated in the main program RFEM, the design relevant input data is already available when you open the module. After the calculation, you can evaluate the designs graphically in the RFEM work window. Last but not least, it is possible to cover the analysis progress in the global printout report, from determination of internal forces up to the results of the design.

We wish you enjoyment and success with RF-STEEL EC3.

Your DLUBAL team

1.2 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 RF-STEEL EC3 add-on module.



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

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

1.3 Opening RF-STEEL EC3 Add-on Module

RFEM provides the following options to open the RF-STEEL EC3 add-on module.

Menu

To open the program from the RFEM menu bar, click

Add-on Modules → **Design - Steel** → **RF-STEEL EC3**.

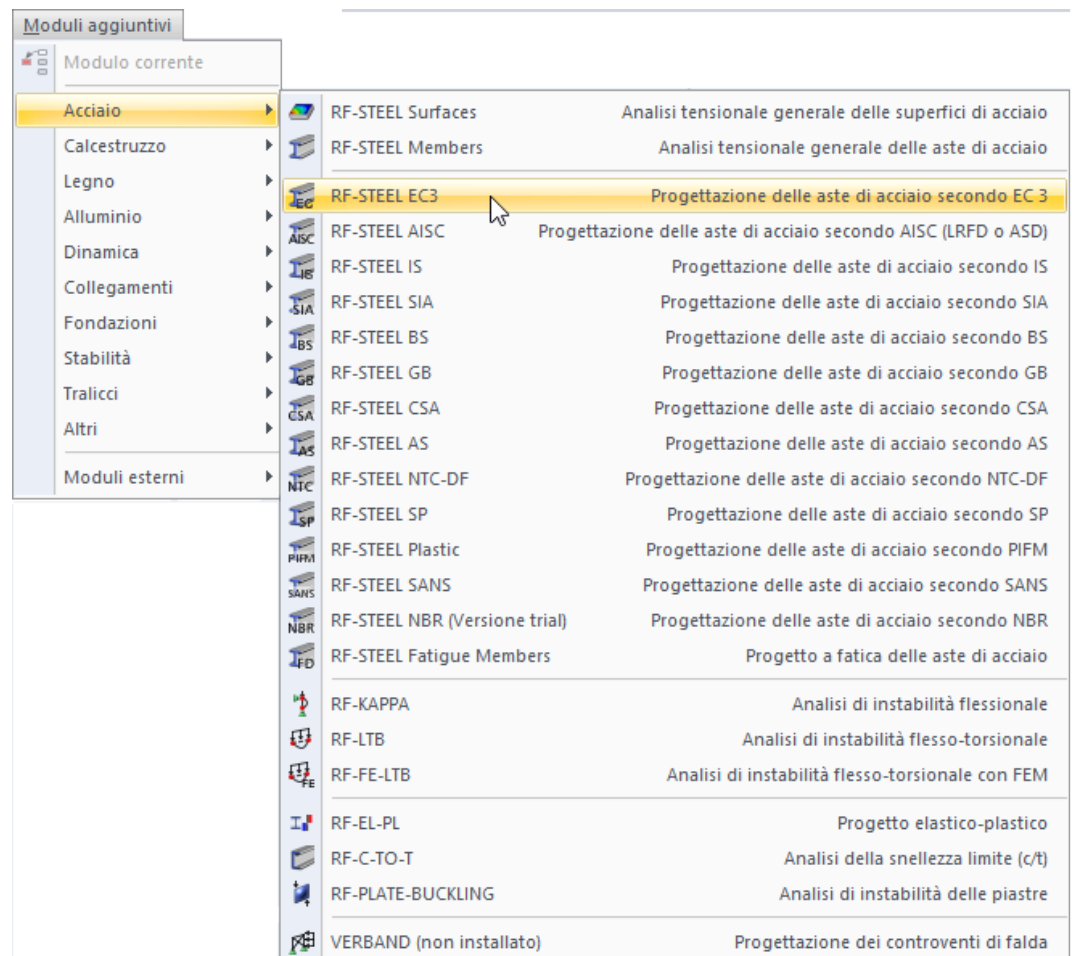


Figure 1.1: Menu: *Add-on Modules* → *Design - Steel* → *RF-STEEL EC3*

Navigator

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

Add-on Modules → **RF-STEEL EC3**.

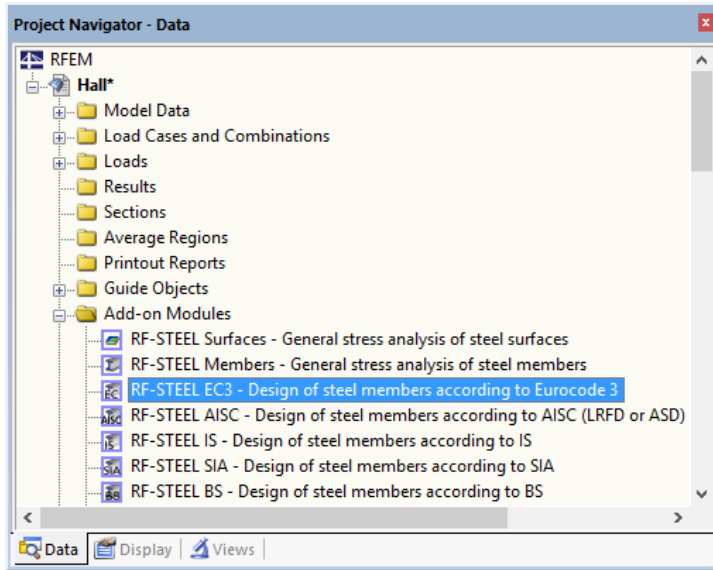
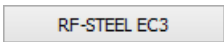
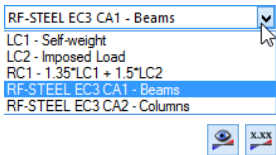


Figure 1.2: Data navigator: *Add-on Modules* → *RF-STEEL EC3*

Panel



If results from RF-STEEL EC3 are already available in the RFEM model, you can also open the design modules from the panel:

Set the relevant RF-STEEL EC3 design case in the load case list of the RFEM toolbar. Click the [Show Results] button to graphically display the design criterion on the members.

Now you can use the [RF-STEEL EC3] button in the panel to open the module.

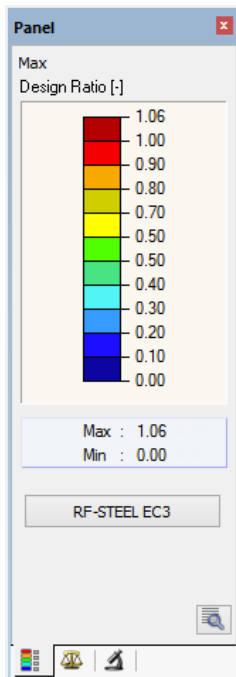


Figure 1.3: Panel button [RF-STEEL EC3]

2 Input Data

After you open the add-on module, a new window appears. In this window, a navigator is displayed on the left, managing the tables that can be selected currently. The drop-down list above the navigator contains the design cases (see [Chapter 7.1, page 73](#)).

The design-relevant data is defined in several input windows. When you open RF-STEEL EC3 for the first time, the following parameters are imported automatically:

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



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



[OK] saves the results. Thus, you exit RF-STEEL EC3 and return to the main program. To exit the add-on module without saving the new data, click [Cancel].

2.1 General Data

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

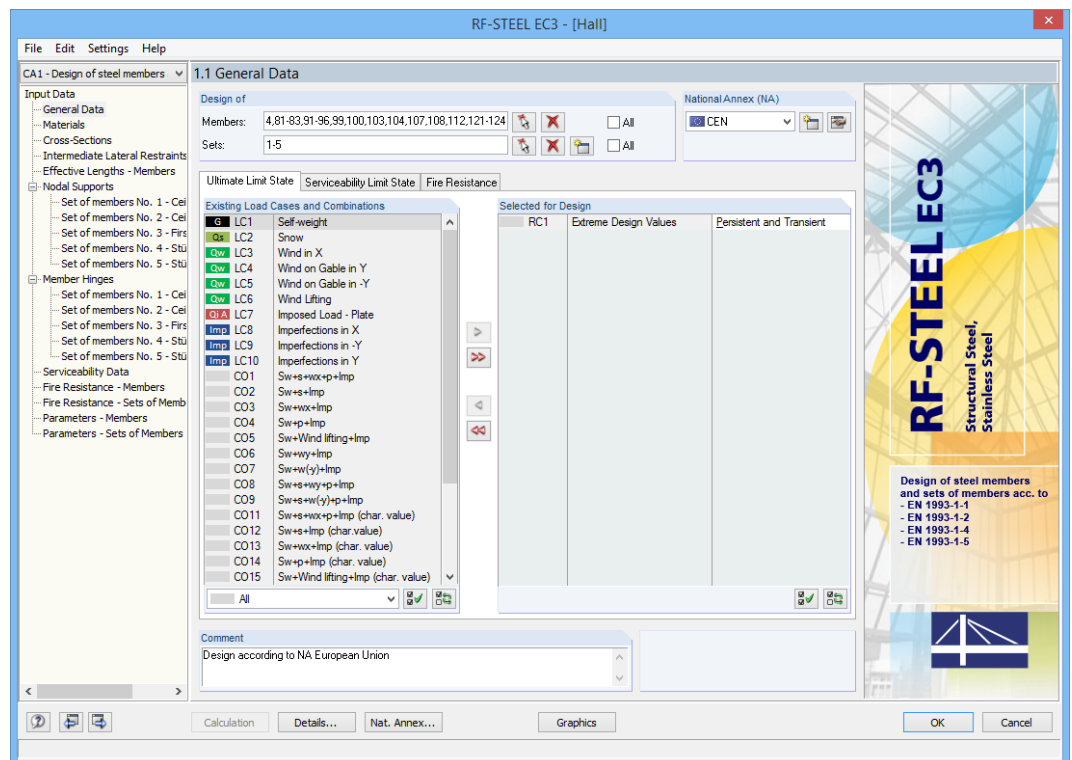


Figure 2.1: Window 1.1 *General Data*

Design of

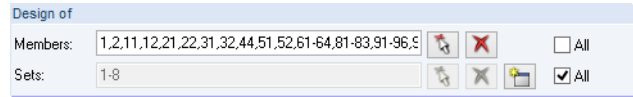


Figure 2.2: Design of members and sets of members



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

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



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

National Annex (NA)

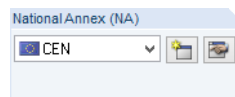


Figure 2.3: National Annex

In the drop-down list box in the upper-right corner of the window, you can define the National Annex. The annex parameters apply for the design and the limit values of the deformation.



Use the [Edit] button to open a dialog box where you can check and/or adjust the parameters of the selected National Annex. This dialog box is described in [Chapter 2.1.4, page 11](#).

Comment



Figure 2.4: User-defined comment

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

2.1.1 Ultimate Limit State

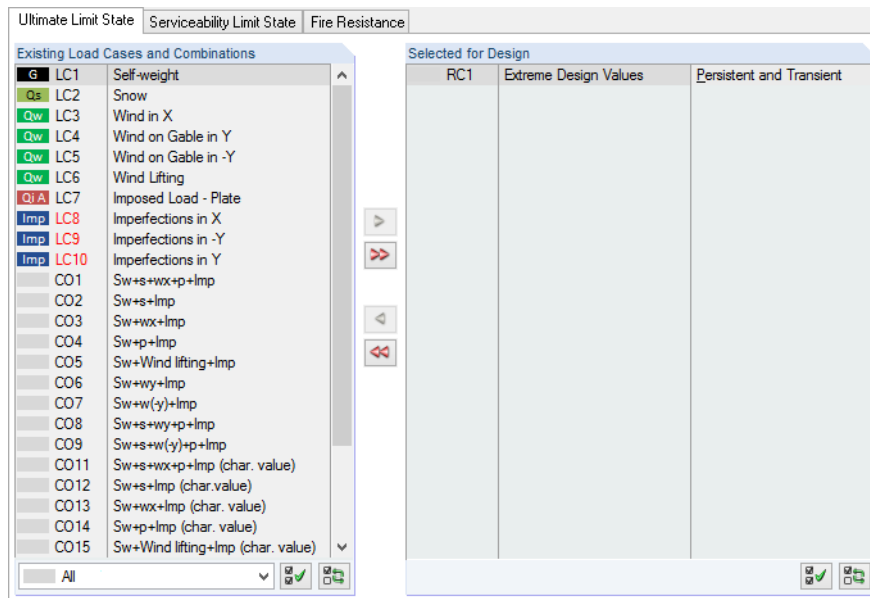


Figure 2.5: Window 1.1 *General Data*, tab *Ultimate Limit State*

Existing Load Cases and Combinations

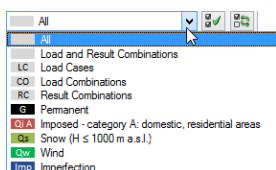
This column lists all load cases as well as load and result combinations that have been created in RFEM.

Use the button to transfer the selected entries to the *Selected for Design* table on the right. Alternatively, you can double-click the entries. To transfer the entire list to the right, use the button.

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 in red, like LC 8 to 10 in [Figure 2.5](#), cannot be designed: This happens when the load cases are defined without any load data or contain only imperfections. Then, when you transfer the load cases, a corresponding warning appears.

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



	Select all cases in the list
	Invert selection of load cases

Table 2.1: Buttons in tab *Ultimate Limit State*

Selected for Design

The column on the right lists the load cases as well as the load and result combinations selected for design. Use the button or double-click the entries to remove selected entries from the list. Use the button to transfer the entire list to the left.

The load cases, load combinations, and result combinations can be assigned to the following design situations:

- *Persistent and Transient*
- *Accidental*

This classification controls the partial factors γ_{M0} , γ_{M1} and γ_{M2} that are included in the determination of the resistances R_d for the cross-section and stability analyses (see [Figure 2.10, page 12](#)).

To change the design situation, use the list at the end of the text box, which you can open by clicking the drop-down arrow .

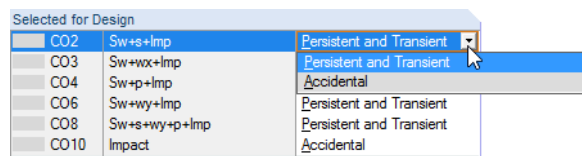


Figure 2.6: Assigning design situation

For multiple selection, press [Ctrl] and click the corresponding entries. Thus, you can change several entries at once.



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

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

2.1.2 Serviceability Limit State

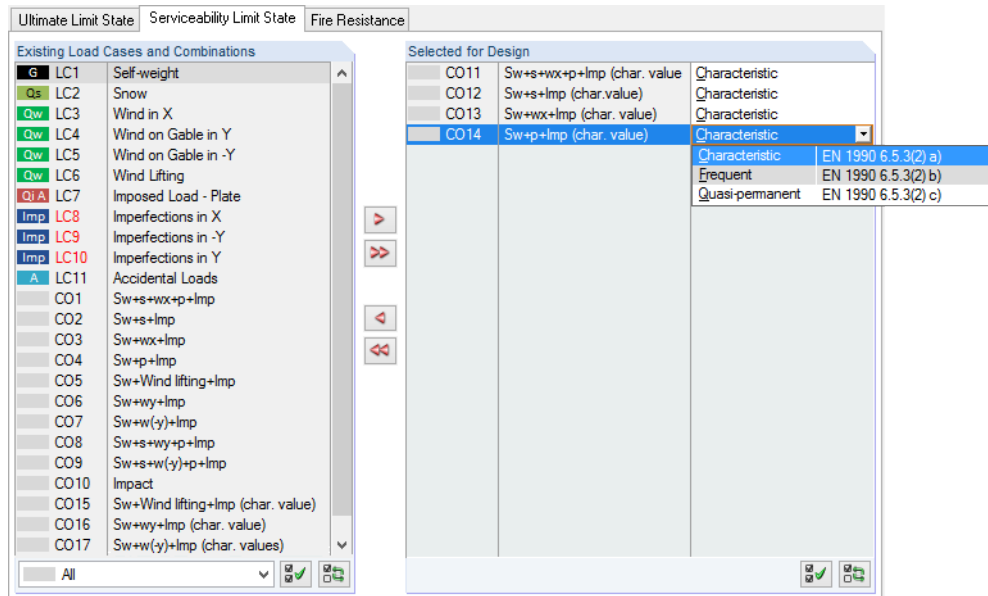


Figure 2.7: Window 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design



You can add or remove load cases as well as load and result combinations as described in [Chapter 2.1.1](#).

You can assign different deflection limit values to the individual load cases, load combinations and result combinations. You can select from the following design situations:

- *Characteristic*
- *Frequent*
- *Quasi-permanent*

To change the design situation, use the list at the end of the text box, which you can open by clicking (see [Figure 2.7](#)).

Nat. Annex...

The limit values of the deformations are defined in the National Annex. To adjust these values according to the design situation, click the [Nat. Annex] button. The *National Annex Settings* dialog box appears (see [Figure 2.10](#), page 12).

In the 1.9 *Serviceability Data* window, you can specify the reference lengths that are governing for the deformation analysis (see [Chapter 2.9](#), page 31).

2.1.3 Fire Resistance

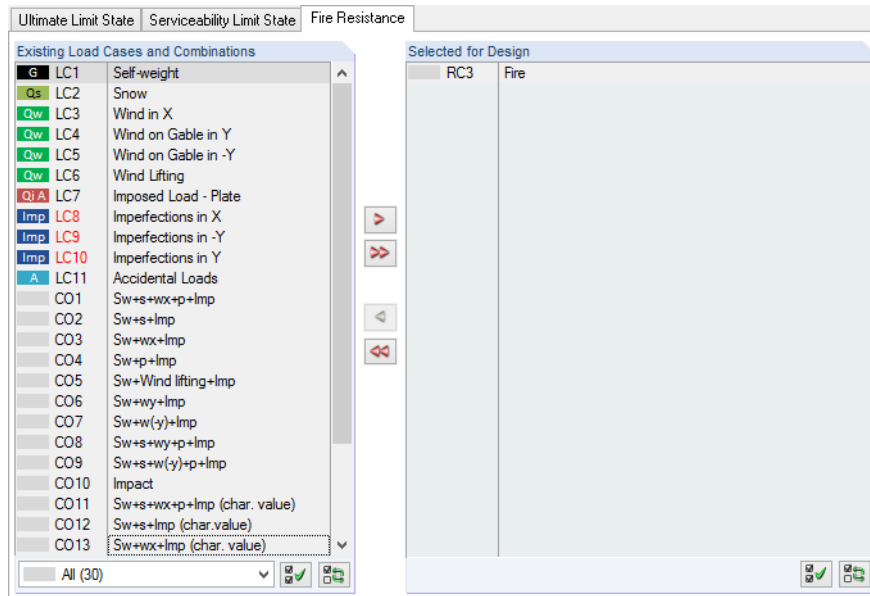


Figure 2.8: Window 1.1 *General Data*, tab *Fire resistance*

Existing Load Cases and Combinations

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

Selected for Design



You can add or remove load cases as well as load and result combinations, as described in [Chapter 2.1.1](#). In this dialog section, you can select the actions that have been determined according to EN 1991-1-2 [2].

2.1.4 National Annex (NA)

In the upper-right list of the 1.1 *General Data* window, you can select the National Annex with parameters valid for the design and limit values of the deformation.



Figure 2.9: Selecting the National Annex



Click the [Edit] button to check and, if necessary, adjust the preset parameters (see the following figure).



Nat. Annex...

Click the [New] button to create a user-defined National Annex.

In addition, you can use the [Nat. Annex] button available in all input windows. Thus, you open the *National Annex Settings* dialog box, that consists of two tabs.

Base

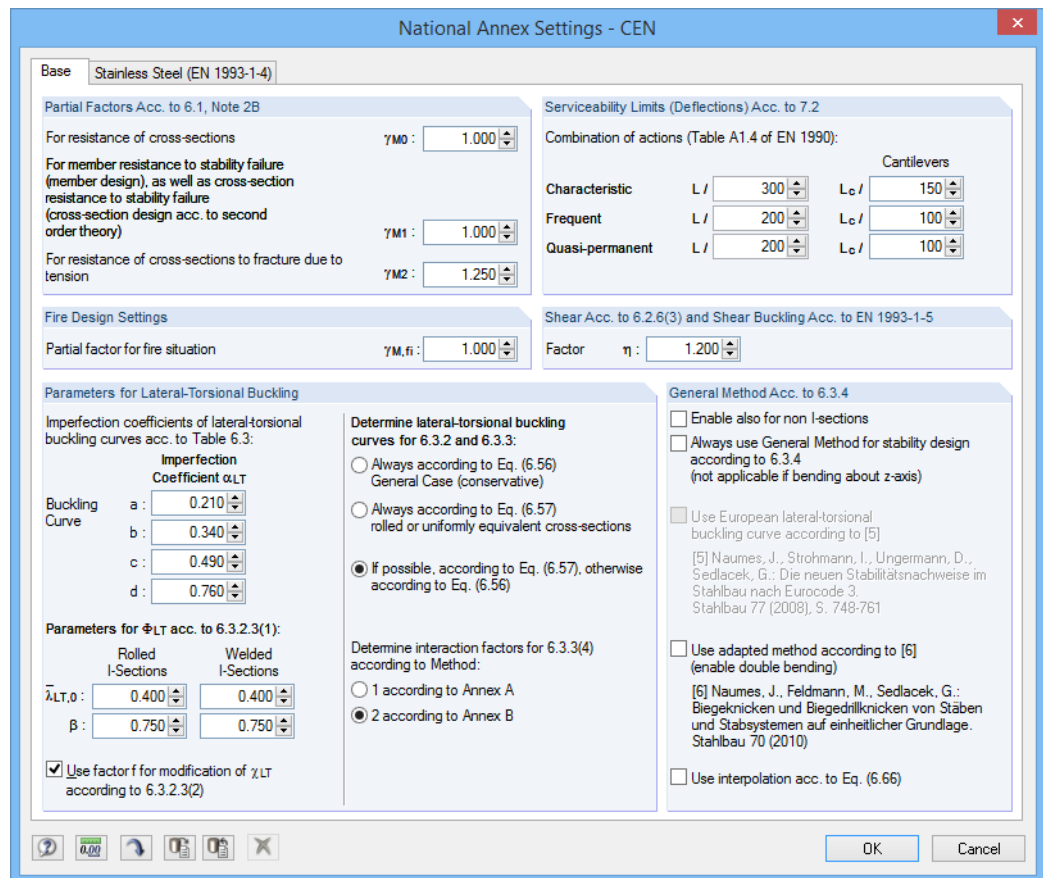


Figure 2.10: Dialog box *National Annex Settings - CEN*, tab *Base*

In the individual sections of this tab, you can check the *Partial Factors*, the *Serviceability Limits (Deflections)*, as well as the *Parameters for Lateral-Torsional Buckling* and adjust them, if necessary.



In the *General Method Acc. to 6.3.4* section, you can determine the stability analysis to be always performed according to [1], Clause 6.3.4. The German National Annex sets the General Method applicable only for I-shaped cross-sections. The option *Enable also for non I-sections* allows you to use the method also for other cross-sections.

In addition, you can perform the stability analysis using the *European lateral-torsional buckling curve* according to NAUMES [4]. In his dissertation [5], NAUMES adapted the “General Method for buckling and lateral-torsional buckling designs of structural components” according to [1], Clause 6.3.4 for additional transverse bending and torsion. This *Adapted Method* is now available for designing unsymmetrical cross-sections as well as for tapered members and sets of members with biaxial bending (torsion is currently not considered in RF-STEEL EC3).

According to [1], Clause 6.3.4 (4), the reduction factor χ_{op} is to be calculated either

- as the minimum value of buckling according to 6.3.1 or χ_{LT} for lateral-torsional buckling according to 6.3.2 using the slenderness ratio λ_{op} , or
- as the value interpolated between χ and χ_{LT} - see also [1], Equation (6.66).

Since the method according to NAUMES is based on the standardized European lateral-torsional buckling curve considering the modified imperfection factor α^* , the interaction between local buckling and lateral-torsional buckling according to [1], Equation (6.66) is not applied.

Calculation	
Main plane	Secondary plane
$\alpha_{Ed}(x) = \frac{\chi_{LT}(x) \cdot \alpha_{ult,k}(x)}{\gamma_{M1}} \geq 1$	$\beta_z(x) = \frac{M_{z,Ed}(x)}{M_{z,Rd}(x)} \cdot (1 - q_{Mz})$
Design	
simplified	exact
$\Delta n_R = 0.9$	$\Delta n_R = 1 - \frac{1}{\alpha_{Ed}(x)} \cdot \left[1 - \frac{1}{\alpha_{Ed}(x)} \right] \cdot \chi_{LT}^2(x) \cdot \bar{\lambda}_{LT}^2(x)$
$\frac{1}{\alpha_{Ed}(x)} + \beta_z(x) \leq \Delta n_R$	

Figure 2.11: Calculation for the method according to NAUMES

In the first step, the calculation is carried out separately for the main and the secondary load-bearing plane. Simultaneously, the moment factor q_{mz} is determined according to Figure 2.12.

In the second step, the design criterion Δn_R is determined.

Finally, the design is performed by summing up the design ratios for the main and the secondary load-bearing plane and compared to the design criterion Δn_R .

Moment distribution M_z	q_{Mz}
	$q_{Mz} = 0.21 \cdot (1 - \psi_z) + 0.36 \cdot (0.33 - \psi_z) \cdot \frac{1}{\alpha_{crit}} \leq \frac{1}{\alpha_{crit}}$
	$q_{Mz} = \frac{1}{\alpha_{crit}} \cdot \left(1 - \frac{\pi^2 EI_z \cdot \max \delta_y }{l^2 \cdot \max M_{z,Ed} } \right)$
	$\max \delta_y $ is the major transverse bending distribution, $\max M_{Ed} $ is the major transverse bending distribution along the component longitudinal axis.
	$q_{Mz} = 0.18 \cdot \frac{1}{\alpha_{crit}}$
	$q_{Mz} = 0.03 \cdot \frac{1}{\alpha_{crit}}$

Figure 2.12: Determining the moment factor q_{Mz}

The buttons in the *National Annex Settings* dialog box have the following functions:

Button	Function
	Reset original settings of the program
	Import user-defined default settings
	Save modified settings as default
	Delete user-defined National Annex

Table 2.2: Buttons in the dialog box *National Annex Settings*

Stainless Steel (EN 1993-1-4)

In RF-STEEL EC3, it is possible to design also structural components made of stainless steel according to EN 1993-1-4 [3].

In the second tab of the *National Annex Settings* dialog box, you can find the relevant *Partial Factors* and *Parameters for Stability Design*.

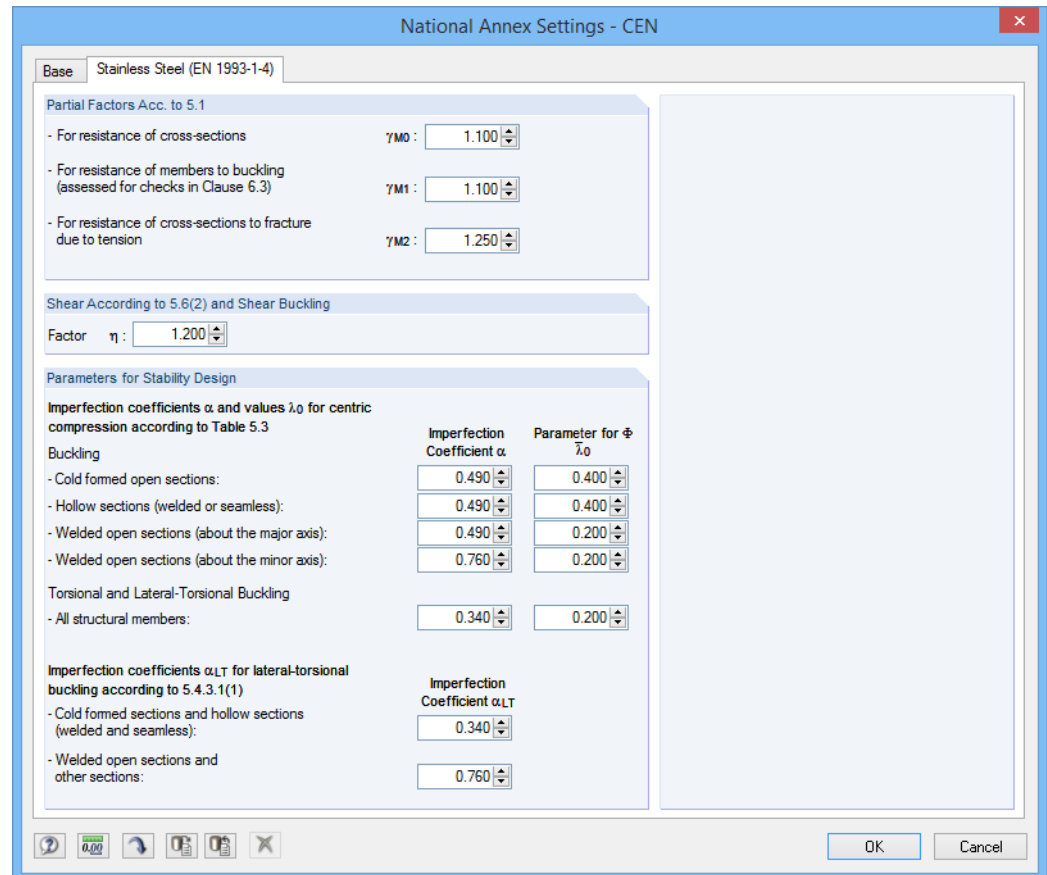


Figure 2.13: Dialog box *National Annex Settings - CEN*, tab *Stainless Steel (EN 1993-1-4)*

2.2 Materials

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

Figure 2.14: Window 1.2 *Materials*

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

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

You can adjust the units and decimal places of material properties and stresses using the menu **Settings** → **Units and Decimal Places** (see [Chapter 7.3, page 77](#)).

Material Description

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

Figure 2.15: List of Materials

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

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 EC3 will import the material properties, too.

Generally, the material properties are not editable in the RF-STEEL EC3 add-on module.

Material Library

Numerous materials are already available in the library. To open the material library, click

Edit → Material Library



or use the button shown on the left.

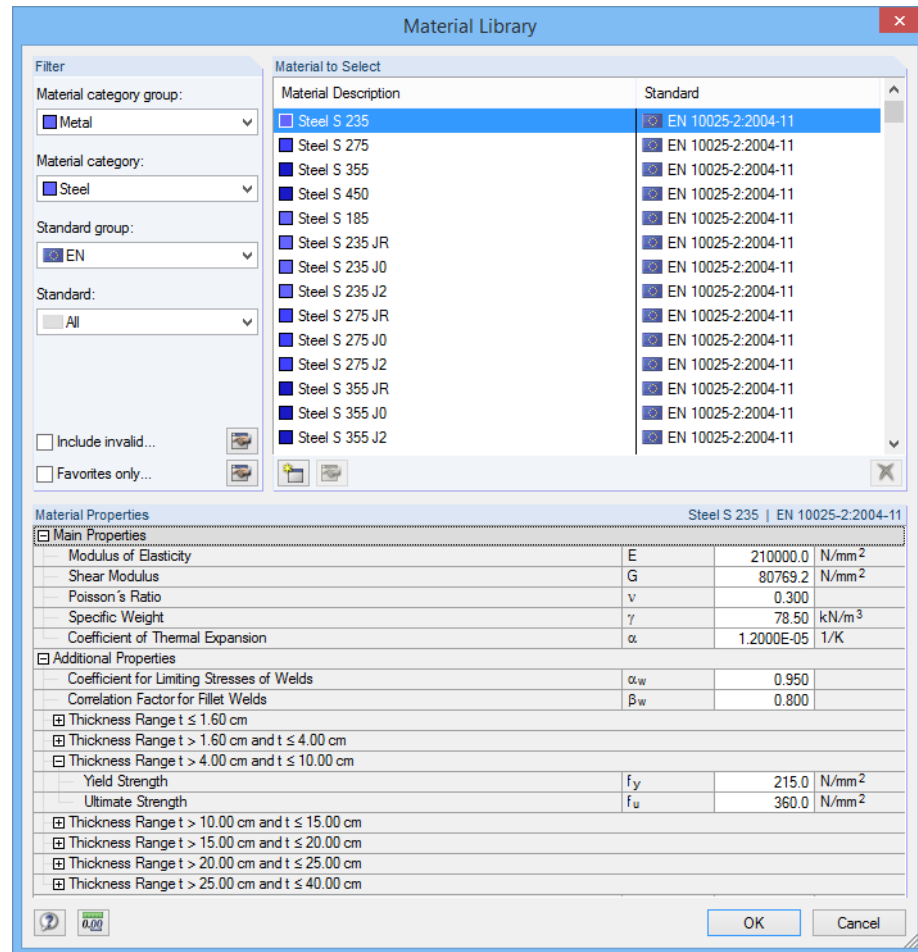
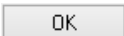


Figure 2.16: Dialog box *Material Library*

In the *Filter* section, the current *Steel* material category is preset. Select the specific material in the *Material to Select* list. You can check the corresponding properties in the dialog section below.



Click [OK] or use the [←] button to transfer the selected material to Window 1.2 of RF-STEEL EC3.

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

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

2.3 Cross-Sections

This window lists the cross-sections used for design. In addition, you can specify optimization parameters here.

1.3 Cross-Sections

Section No.	A	B	C	D	E	F	G
Section No.	Material No.	Cross-Section Description	Cross-Section Type for Classification	Cross-Section Classification	Optimize	Remark	Comment
1	1	I IPE 400 Euronorm 19-	I-section rolled	Automatically	No	5)	
2	1	I IPE 300 Euronorm 19-	I-section rolled	Automatically	From Current Row	2)	
3	1	I ICU IPE 300 + IPE 300	General	Automatically	No	3)	
4	1	I ICU IPE 300 + IPE 300	General	Automatically	From current row	3)	
5	2	I IPE 300 Euronorm 19-	I-section rolled	Automatically	From favorites 'Euronorm'	5)	
6	1	I HE A 140 Euronorm 5	I-section rolled	Automatically	No	5)	
7	1	I HE A 200 Euronorm 5	I-section rolled	Automatically	No	5)	
8	1	RO 101.6x5 (Cold Form)	Pipe	Automatically	No	5)	
9	1	I IPE 200 Euronorm 19-	I-section rolled	Automatically	No	5)	
10	1	RD 20 Macsteel	General	Automatically	No	5)	

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

Cross-Section Properties - IPE 300 | Euronorm 19-57

Cross-Section Type		I-section rolled		
Section Height	h	300.0	mm	
Section Width	b	150.0	mm	
Web Thickness	t _w	7.1	mm	
Flange Thickness	t _f	10.7	mm	
Root Radius	r	15.0	mm	
Cross-Sectional Area	A	53.80	cm ²	
Effective Shear Area	A _{v,y}	33.67	cm ²	
Effective Shear Area	A _{v,z}	25.67	cm ²	≥ η _h at w 6.2.6(3)a)
Moment of Inertia	I _y	8360.00	cm ⁴	
Moment of Inertia	I _z	604.00	cm ⁴	
Torsional Constant	I _t	20.20	cm ⁴	
Radius of Gyration	i _y	125.0	mm	
Radius of Gyration	i _z	33.5	mm	
Elastic Section Modulus	S _{el,y}	557.00	cm ³	
Elastic Section Modulus	S _{el,z}	80.50	cm ³	
Plastic Section Modulus	W _{pl,y}	628.00	cm ³	

Cross-section No. 2 used in

Members No.: 4,8,9,11-13,17,21,30,34,43,47,57-60

Sets of members No.: 1-4

Σ Lengths: 45.86 [m] Σ Masses: 1.937 [t]

Material: 1 - Steel S 235

Figure 2.17: Window 1.3 Cross-Sections

Cross-Section Description

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



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



In this dialog box, you can select a different cross-section type. 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.

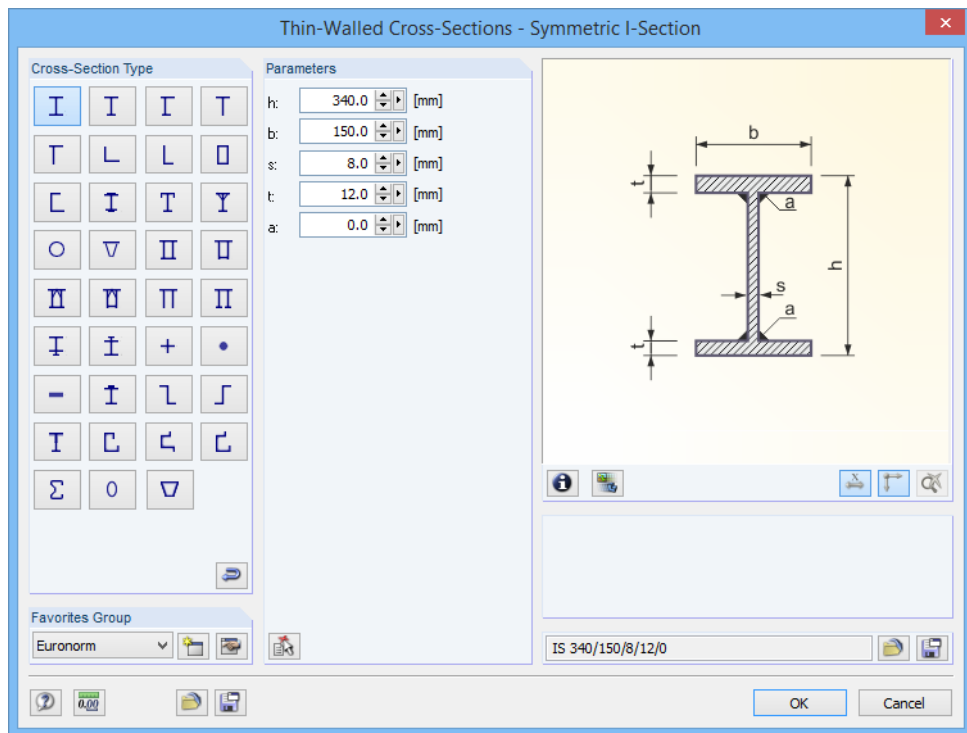
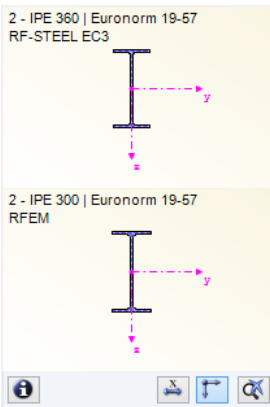


Figure 2.18: IS cross-section types in the cross-section library



You can directly enter the new cross-section description in the input window. If the entry is listed in the database, RF-STEEL EC3 imports these cross-section parameters, too.

A modified cross-section will be highlighted in blue.

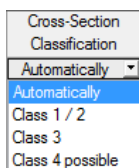
If the cross-sections in RF-STEEL EC3 are different from the ones used in RFEM, both cross-sections are displayed in the graphic on the right. The designs will be performed for the cross-section selected in RF-STEEL EC3, using the internal forces from RFEM.

Cross-Section Type for Classification

The program displays a cross-section type that will be used for the classification. The cross-sections listed in [1], Table 5.2 can be designed plastically or elastically depending on the Class. Cross-sections not included in this table are classified as *General* and can only be designed elastically (Class 3 or 4).

Classification

RF-STEEL EC3 performs the classification *Automatically*. If this is not desired, you can determine the cross-section class manually using the drop-down list.



Max. Design Ratio

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

Optimize

You can optimize every cross-section from the library: For the RFEM internal forces, the program searches the cross-section that comes as close as possible to a user-defined maximum ratio. You can define the maximum ratio in the *Details* dialog box, tab *Others* (see Figure 2.8, page 49).

If you want to optimize a cross-section, open the corresponding drop-down list in column D or E and select the desired entry: *From current row* or, if available, *From favorites 'Description'*. Recommendations for the cross-section optimization can be found in Chapter 7.2 on page 75.

Remark

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



A warning might appear before the calculation: *Incorrect type of cross-section!* It means that there is a cross-section that is not stored in the database. 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 the design, click the [Library] button (see description below [Figure 2.17](#)).

Member with Tapered Cross-Section

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

RF-STEEL EC3 also designs tapered members, provided that the cross-section at the member 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 cross-section at the start and the end of a tapered member have a different number of stress points, the intermediate values cannot be interpolated. Then, the calculation is possible neither in RFEM nor in RF-STEEL EC3.



The cross-section's stress points including numbering can also be checked graphically: Select the cross-section in Window 1.3 and click the [Info] button. The dialog box shown in [Figure 2.19](#) appears.

Info About Cross-Section



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

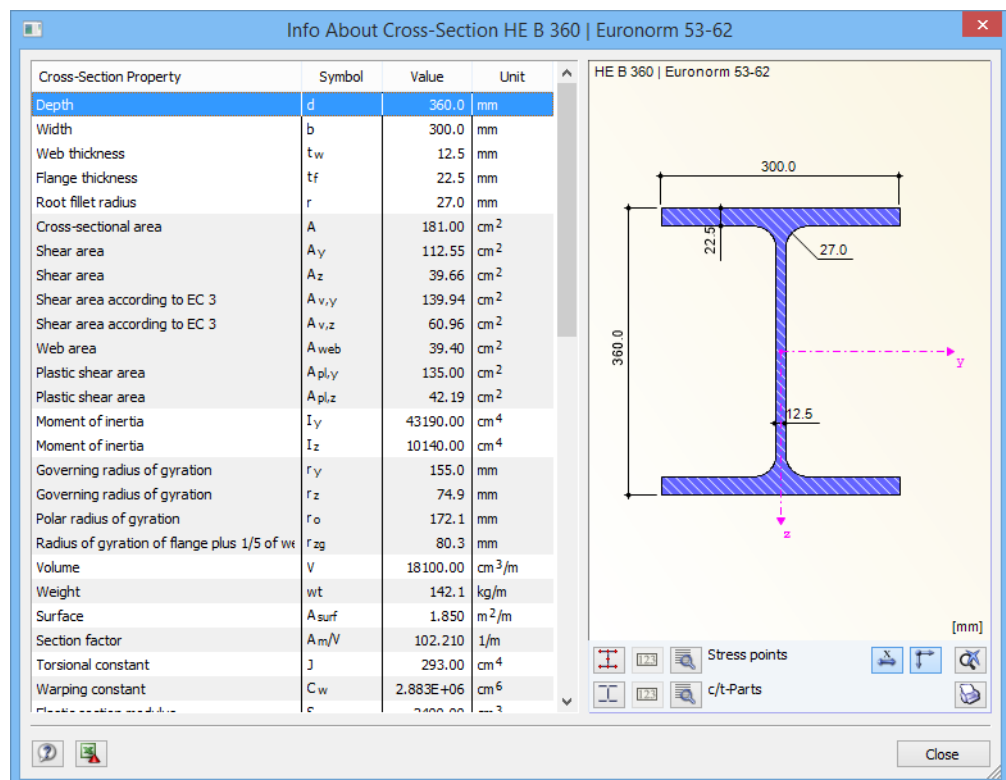


Figure 2.19: Dialog box *Info About Cross-Section*

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

Button	Function
	Display or hide the stress points
	Display or hide the cross-section c/t-parts.
	Display or hide the numbering of stress points or c/t-parts
	Display the details of stress points or c/t-parts (see Figure 2.20)
	Display or hide the cross-section dimensions
	Display or hide the principal axes of the cross-section
	Reset the full view of the cross-section graphic

Table 2.3: Buttons of cross-section graphic



Click [Details] to call up detailed information on stress points (centroidal distances, statical moments of area, warping constants etc.) and c/t-parts.

Stress Points of HE B 260 | Euronorm 53-62

StressP No.	Coordinates		Statical Moments of Area		Thickness t [mm]	Warping	
	y [mm]	z [mm]	Q _y [cm ³]	Q _z [cm ³]		W _{no} [cm ²]	Q _w [cm ⁴]
1	-130.0	-130.0	0.00	0.00	17.5	157.63	0.00
2	-29.0	-130.0	-214.01	-140.47	17.5	35.16	-1703.76
3	0.0	-130.0	-278.73	-148.54	17.5	0.00	-1792.98
4	29.0	-130.0	-214.01	140.47	17.5	-35.16	1703.76
5	130.0	-130.0	0.00	0.00	17.5	-157.63	0.00
6	-130.0	130.0	0.00	0.00	17.5	-157.63	0.00
7	-29.0	130.0	-214.31	140.52	17.5	-35.16	-1703.76
8	0.0	130.0	-278.73	148.54	17.5	0.00	-1792.98
9	29.0	130.0	-214.31	-140.52	17.5	35.16	1703.76
10	130.0	130.0	0.00	0.00	17.5	157.63	0.00
11	0.0	-88.5	-600.75	0.00	10.0	0.00	0.00
12	0.0	88.5	-601.59	0.00	10.0	0.00	0.00
13	0.0	0.0	-640.15	0.00	10.0	0.00	0.00

Close

Figure 2.20: Dialog box *Stress Points of HE B 260*



If circumstances require it, you can change the buckling curves in the *Cross-Section Properties* table of the 1.3 *Cross-Sections* window. This function is described in the following article of the DLUBAL Blog: www.dlubal.com/blog/8684

2.4 Intermediate Lateral Restraints

In Window 1.4, you can define intermediate lateral restraints for members. In RF-STEEL EC3, this kind of support is always perpendicular to the minor z-axis of the cross-section (see Figure 2.19). Thus, you can influence the members' effective lengths which are important for the stability analysis of flexural buckling and lateral-torsional buckling (only for the *Lateral and torsional* restraint type).

1.4 Intermediate Lateral Restraints

Member No.	A Lateral Restraints	B Restraint Type	C Length L [m]	D Number	E x ₁	F x ₂	G, H, I, J, K, L, M Intermediate Lateral Restraints [-]										
							x ₃	x ₄	x ₅	x ₆	x ₇	x ₈	x ₉				
14	<input type="checkbox"/>		3.262														
15	<input checked="" type="checkbox"/>	Lateral and torsional	6.274	2	0.400	0.750											
33	<input type="checkbox"/>		3.000														
34	<input type="checkbox"/>		3.546														
35	<input type="checkbox"/>		3.000														
36	<input checked="" type="checkbox"/>	Lateral (Upper Flange)	4.094	1	0.500												
41	<input type="checkbox"/>		3.011														
42	<input type="checkbox"/>		3.262														
43	<input checked="" type="checkbox"/>	User-Defined	6.274	1	0.500												
51	<input type="checkbox"/>		3.000														

Relatively (0 ... 1)

Settings - Member No. 43

Cross-Section	2 - IPE 300 Euronorm 19-57	
Lateral Restraints	<input checked="" type="checkbox"/>	
Restraint type	User-Defined	
Lateral Restraint in y	<input checked="" type="checkbox"/>	u_y
Restrained about x	<input checked="" type="checkbox"/>	ϕ_x
Eccentricity	-124.3	mm
Member Length	6.274	m
Number of Intermediate Lateral Restraints	1	n
Location of Lateral Restraint No. 1	0.500	x ₁

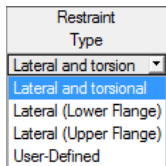
Set input for members No.:

All

Figure 2.21: Window 1.4 *Intermediate Lateral Restraints*

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

To define the intermediate restraints of a member, select the *Lateral Restraints* check box in column A. To graphically select the member, click . If you select the check box, the other columns become available and you can enter the parameters.



In column B, you can select the *Restraint Type* from the drop-down list. The lateral and torsional restraint is preset. Furthermore, you can place the intermediate restraints also at the lower or upper flange. The *User-defined* restraint type allows you to individually specify the restraint parameters in the *Settings* section (lateral support in the direction of the member axis y, restrained about longitudinal member axis x, eccentricity of a restraint).

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

Relatively (0 ... 1)

If you select the *Relatively (0 ... 1)* check box, you can define the support points by relative input. The positions of the intermediate restraints are determined by the member length and the relative distances of the member start. If you clear the *Relatively (0 ... 1)* check box, it is possible to define the distances manually in the upper table.



In case of cantilevers, you should avoid intermediate restraints as they divide the member into segments. For cantilevered beams, this would result in segments with lateral and torsional restraints (on one end each) that are statically underdetermined.

2.5 Effective Lengths - Members

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

Click to select a member graphically and show its row.

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

1.5 Effective Lengths - Members

Member No.	Buckling About Axis y		Buckling About Axis z		Lateral-Torsional and Torsional-Flexural Buckling			Comment				
	Possible	$k_{cr,y}$	$L_{cr,y}$ [m]	Possible	$k_{cr,z}$	$L_{cr,z}$ [m]	Possible		k_z	k_w	L_w [m]	L_T [m]
1	<input checked="" type="checkbox"/>	1.000	6.000	<input checked="" type="checkbox"/>	1.000	6.000	<input checked="" type="checkbox"/>	1.0	1.0	6.000	6.000	
2	<input checked="" type="checkbox"/>	1.000	6.000	<input checked="" type="checkbox"/>	1.000	6.000	<input checked="" type="checkbox"/>	1.0	1.0	6.000	6.000	
11	<input checked="" type="checkbox"/>	4.040	24.240	<input checked="" type="checkbox"/>	1.086	3.258	<input checked="" type="checkbox"/>	1.0	1.0	3.000	3.000	
12	<input checked="" type="checkbox"/>	3.855	23.130	<input checked="" type="checkbox"/>	1.036	3.108	<input checked="" type="checkbox"/>	1.0	1.0	3.000	3.000	
21	<input checked="" type="checkbox"/>	1.000	6.000	<input type="checkbox"/>			<input checked="" type="checkbox"/>	1.0	1.0			
22	<input checked="" type="checkbox"/>	1.000	6.000	<input type="checkbox"/>			<input checked="" type="checkbox"/>	1.0	1.0			
31	<input checked="" type="checkbox"/>	1.000	3.000	<input checked="" type="checkbox"/>	1.000	3.000	<input checked="" type="checkbox"/>	1.0	1.0	3.000	3.000	
32	<input checked="" type="checkbox"/>	1.000	3.000	<input checked="" type="checkbox"/>	1.000	3.000	<input checked="" type="checkbox"/>	1.0	1.0	3.000	3.000	
44	<input checked="" type="checkbox"/>	1.000	6.274	<input checked="" type="checkbox"/>	1.000	6.274	<input checked="" type="checkbox"/>	1.0	1.0	6.274	6.274	
51	<input checked="" type="checkbox"/>	1.000	3.000	<input checked="" type="checkbox"/>	1.000	3.000	<input checked="" type="checkbox"/>	1.0	1.0	3.000	3.000	

Settings - Member No. 1

Cross-Section		2 - IPE 300 Euronorm 19-57
Length	L	6.000 m
Buckling Possible		<input checked="" type="checkbox"/>
<input type="checkbox"/> Buckling About Axis y Possible		<input checked="" type="checkbox"/>
Effective Length Factor	$k_{cr,y}$	1.000
Effective Length	$L_{cr,y}$	6.000 m
<input type="checkbox"/> Buckling About Axis z Possible		<input checked="" type="checkbox"/>
Effective Length Factor	$k_{cr,z}$	1.000
Effective Length	$L_{cr,z}$	6.000 m
<input type="checkbox"/> Lateral-Torsional Buckling Possible		<input checked="" type="checkbox"/>
Effective Length Factor (Restraint Type)	k_z	1.0
Warping Length Factor (Restraint Type)	k_w	1.0
LTB Length	L_w	6.000 m
Torsional Length	L_T	6.000 m
Comment		

IPE 300 | Euronorm 19-57

[mm]

Figure 2.22: Window 1.5 Effective lengths - Members

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

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

The *Settings* tree includes the following parameters:

- *Cross-Section*
- *Length* of the member
- *Buckling possible* for the member (cf. columns B, E and H)
- *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 to K)

In this table, you can determine a buckling analysis or lateral-torsional buckling analysis to be performed for the selected member. In addition, you can adjust the *Effective Length Factor* and the *Warping Length Factor* for the respective lengths. If you modify the factor, the equivalent member length is adjusted automatically, and vice versa.



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

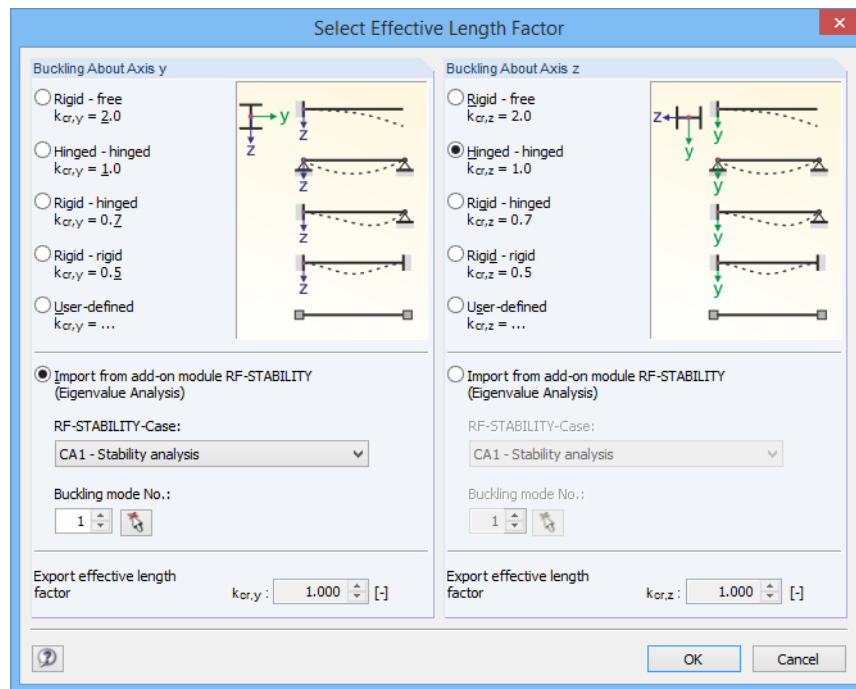


Figure 2.23: Dialog box *Select Effective Length Factor*

For each direction, you can select one of the four Euler buckling modes or set the effective length factor as *User-defined*. If the eigenvalue analysis has been performed in the RF-STABILITY add-on module, you can import the *Buckling mode* from there in order to determine the factor.

Buckling Possible

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

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

Buckling About Axis y or Axis z

The *Possible* columns control if there is a buckling risk about the y-axis and/or z-axis. These axes represent the local member axes, where the y-axis is the “major” and the z-axis is the “minor” member axis. You can freely select the effective length factors $k_{cr,y}$ and $k_{cr,z}$ for buckling about the major or the minor axis.



You can check the position of the member axes in the cross-section graphic in the 1.3 *Cross-Sections* window (see [Figure 2.17, page 17](#)). To access the RFEM work window, click the [View Mode] button.

There you can display the local member axes by using the member's shortcut menu or the *Display* navigator.

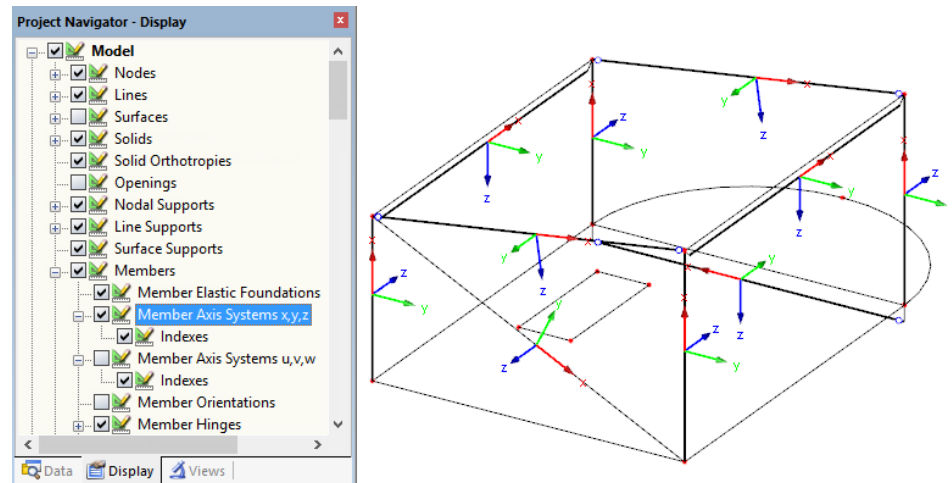
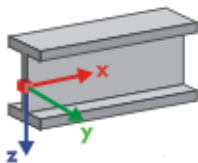


Figure 2.24: Selecting the member axis system in the *Display* navigator of RFEM

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

To specify the effective lengths in the work window graphically, click . You can click the button only if the cursor is placed in a L_{cr} text box (see Figure 2.22).

When specifying the effective length factor k_{cr} , the program determines the effective length L_{cr} by multiplying the member length L by the effective length factor. The text boxes k_{cr} and L_{cr} are interactive.



Axes definition for k_z and k_w

Lateral-Torsional Buckling Possible

Column H shows which members are included in the analysis of lateral-torsional buckling.

To determine M_{cr} by the eigenvalue calculation method, a member with four degrees of freedom is created in the program background. You can define these degrees of freedom using the factors k_z and k_w . Both factors determine the support conditions for lateral-torsional buckling (for example a lateral and torsional restraint).

Effective Length Factor k_z

k_z
1.0
0.7le
0.7ri
0.5
2.0le
2.0ri

The factor k_z specifies the lateral restraint u_y and the rotational restraint ϕ_z on member ends.

$k_z = 1.0$ lateral restraint u_y on both member ends

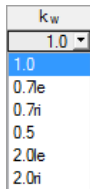
$k_z = 0.7le$ lateral restraint u_y on both ends, restrained about z on the left

$k_z = 0.7ri$ lateral restraint u_y on both ends, restrained about z on the right

$k_z = 0.5$ lateral restraint u_y , restrained about z on both ends

$k_z = 2.0le$ lateral restraint u_y , restrained about z on the left; free end on the right

$k_z = 2.0ri$ lateral restraint u_y , restrained about z on the right; free end on the left



Warping Length Factor k_w

The factor k_w specifies torsion ϕ_x about the member axis and the warping restraint ω .

$k_w = 1.0$ restrained about x on both ends; free to warp on both ends

$k_w = 0.7le$ restrained about x on both ends; warping restraint on the left

$k_w = 0.7ri$ restrained about x on both ends; warping restraint on the right

$k_w = 0.5$ torsion and warping restraint on both ends

$k_w = 2.0le$ restrained about x , warping restraint ω on the left; free to warp on the right


$k_w = 2.0ri$ restrained about x , warping restraint ω on the right; free to warp on the left



The abbreviations *le* and *ri* refer to the left and right side. The abbreviation *le* always describes the support conditions at the member start.




You can model a lateral and torsional restraint using the factors $k_z = 1.0$ (lateral restraint in y , free torsion about z) and $k_w = 1.0$ (restrained about x , free to warp). Since the internal member model requires only four degrees of freedom, the definition of the other boundary conditions is not necessary.

If the lateral-torsional length L_w or the torsional buckling length L_T differs from the member or buckling length, you can define the lengths L_w and L_T manually in columns K and L or determine them graphically using the  button.

Comment

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

Set Input for Members No.

Under the *Settings* table, you find the *Set input for members No.* check box. If you select the check box, the subsequent settings will be applied to the selected members or *All* members (you can enter the member numbers manually or select them graphically using the  button). This option is useful when assigning several members to the same boundary conditions (see also DLUBAL Blog www.dlubal.com/blog/11115/).



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

Details...

2.6 Effective Lengths - Sets of Members

This window appears only if you selected at least one set of members for design in the 1.1 *General Data* window (see [Figure 2.2](#), page 44) and selected the *Equivalent Member Method* for sets of members in the *Details* dialog box. If applied, Windows 1.7 and 1.8 will not be displayed. Then, you can define the intermediate lateral restraints by using division points in Window 1.4.

1.6 Effective Lengths - Sets of Members

Set No.	A Buckling Possible	Buckling About Axis y		Buckling About Axis z		Lateral-Torsional and Torsional-Flexural Buckling				M Comment			
		B Possible	C $k_{cr,y}$	D $L_{cr,y}$ [m]	E Possible	F $k_{cr,z}$	G $L_{cr,z}$ [m]	H Possible	I k_z		J k_w	K L_w [m]	L L_T [m]
1	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	6.000	<input checked="" type="checkbox"/>	1.000	6.000	<input checked="" type="checkbox"/>	1.0	1.0	25.000	25.000	
2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	12.548	<input type="checkbox"/>	1.000	12.548	<input checked="" type="checkbox"/>	1.0	1.0	12.548	12.548	
3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	12.548	<input type="checkbox"/>	1.000	12.548	<input checked="" type="checkbox"/>	1.0	1.0	12.548	12.548	
4	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	6.546	<input checked="" type="checkbox"/>	1.000	6.546	<input checked="" type="checkbox"/>	1.0	1.0	6.546	6.546	
5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1.000	7.094	<input checked="" type="checkbox"/>	1.000	7.094	<input checked="" type="checkbox"/>	1.0	1.0	7.094	7.094	

Settings - Set of Members No. 4

<input checked="" type="checkbox"/> Set of Members	Column Members 4
Cross-Section	6 - HE A 140 Euronorm 53-62
Length	L 6.000 m
Buckling Possible	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/> Buckling About Axis y Possible	<input checked="" type="checkbox"/>
Effective Length Factor	$k_{cr,y}$ 1.000
Effective Length	$L_{cr,y}$ 6.546 m
<input checked="" type="checkbox"/> Buckling About Axis z Possible	<input checked="" type="checkbox"/>
Effective Length Factor	$k_{cr,z}$ 1.000
Effective Length	$L_{cr,z}$ 6.546 m
<input checked="" type="checkbox"/> Lateral-Torsional Buckling Possible	<input checked="" type="checkbox"/>
Effective Length Factor (Restraint Type)	k_z 1.0
Warping Length Factor (Restraint Type)	k_w 1.0
LTB Length	L_w 6.546 m
Torsional Length	L_T 6.546 m
Comment	

HE A 140 | Euronorm 53-62

Figure 2.25: Window 1.6 *Effective Lengths - Sets of Members*

The concept of the window is similar to the previous 1.5 *Effective Lengths - Members* window. Here you can enter the effective lengths for buckling about both principal axes of the set of members, as described in [Chapter 2.5](#).

Details...

2.7 Nodal Supports - Set of Members

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

The stability analysis for sets of members is usually carried out in compliance with the General Method according to [1], Clause 6.3.4. However, if you select the *Equivalent Member Method* for sets of members in the *Details* dialog box (see Figure 2.2, page 44), Window 1.7 will not be displayed. Then, you can define the intermediate lateral restraints by using division points in Window 1.4.

Support No.	A Node No.	B Support Rotation β [°]	C Lat. Support u_y	D Rotational Restraint φ_x	E Restraint φ_z	F Warping Restraint ω [kNm ³]	G Eccentricity e_x [mm]	H Eccentricity e_z [mm]	I Comment
1	29	0.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	14.418	200.0	0.0	Warp spring determined automatically
2	34	0.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	14.418	200.0	0.0	
3	38	0.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	14.418	200.0	0.0	
4	32	0.00	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	14.418	200.0	0.0	
5									
6									
7									
8									
9									
10									

Figure 2.26: Window 1.7 *Nodal Supports - Set of Members*



The current table manages the boundary conditions of the set of members selected in the navigator on the left!

The supports defined in RFEM, such as the supports in Z of a continuous beam, are not relevant in this window. The program imports the distributions of moments and shear forces for determination of amplification factor automatically from RFEM. You can define several support conditions here that have an impact on stability failure (buckling, lateral-torsional buckling).

Supports on the start and end nodes of a set of members are preset. Any other supports, for example as a result of additional members, has to be added manually. You can select nodes graphically in RFEM work window using the button.



According to [1], Clause 6.3.4 (1), you can design monosymmetrical cross-sections that are loaded solely in their principal plane. For this analysis method, it is necessary to know the amplification factor $\alpha_{cr,op}$ of the entire set of members. In order to determine this factor, the program creates a planar framework with four degrees of freedom for each node.



The axes orientation in a set of members is important for the definition of nodal supports. The program checks the position of nodes and internally defines the axes of nodal supports for Window 1.7, according to Figure 2.27 to Figure 2.30. The [Local Coordinate System] button below the model graphic can help you with the orientation. With this button, you can display the set of members in a partial view where the axes are clearly visible (see DLUBAL Blog www.dlubal.com/blog/9763).

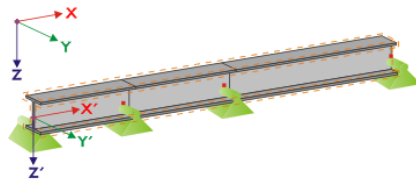


Figure 2.27: Auxiliary coordinate system of nodal supports - straight set of members

If all members of a set of members rest on a straight line, as shown in the [Figure 2.27](#), 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.

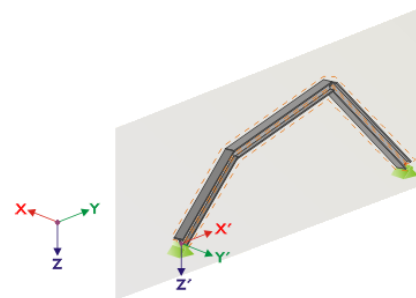


Figure 2.28: Auxiliary coordinate system of nodal supports - set of members on vertical plane

If members of a set of members do not rest on a straight line, they still have to be located in the same plane. In [Figure 2.28](#), the members rest on a vertical plane. In this case, the X' -axis is horizontal and it is oriented in the direction of the plane. The Y' -axis is horizontal as well and defined as perpendicular to the X' -axis. The Z' -axis is oriented perpendicularly downwards.

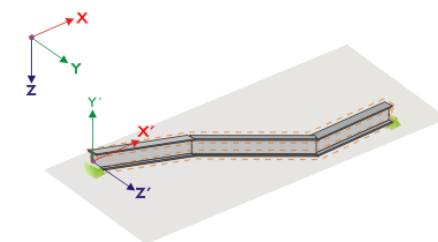


Figure 2.29: Auxiliary coordinate of nodal supports - set of members on horizontal plane

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

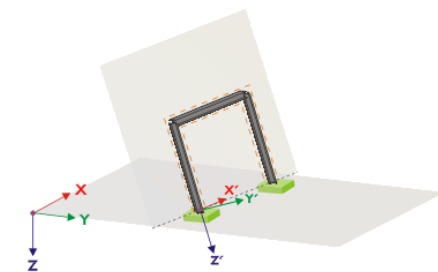


Figure 2.30: Auxiliary coordinate system of nodal supports - set of members on inclined plane

[Figure 2.30](#) shows the general case of a buckled set of members: The members do not rest on a straight line, but on an inclined plane. The definition of the X' -axis results from the intersection line between the inclined plane and the horizontal plane. Thus, the Y' -axis is perpendicular to the X' -axis and directed perpendicularly to the inclined plane. The Z' -axis is defined perpendicular to the X' -axis and Y' -axis.

The buttons below the graphic have the following functions:

Button	Function
	Display the model or the system sketch
	Display the members as a 3D rendering or a wireframe model
	Display the current set of members or the entire model
	Display the irrelevant members of the model transparent or opaque
	Display the set of members in the local coordinate system or the entire model
	Show the view in the direction of the X-axis
	Show the view against the Y-axis
	Show the view in the direction of the Z-axis
	Display the isometric view

Table 2.4: Buttons of cross-section graphic



Click the [Edit warp stiffener and import the warp spring] button to determine the constant of a warp spring.

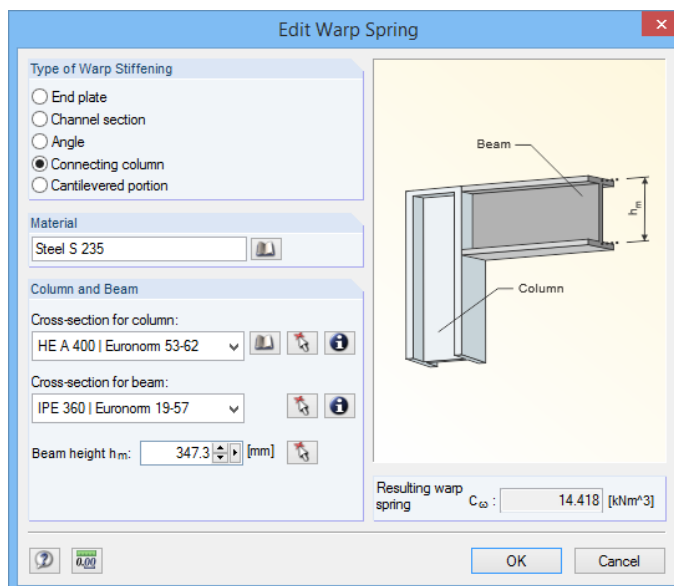


Figure 2.31: Dialog box *Edit Warp Spring*

In the *Edit Warp Spring* dialog box, you can select one of the following types of warp springs:

- End plate
- Channel section
- Angle
- Connecting column
- Cantilevered portion



You can select the materials and cross-sections using the lists and [Library] buttons. Click the button for a graphical selection in the RFEM model.

RF-STEEL EC3 determines the *Resulting warp spring* C_{ω} from the parameters and then imports the result in Window 1.7 by clicking [OK].

2.8 Member Hinges - Set of Members

This window appears only if you have selected at least one set of members for the design in the 1.1 *General Data* window. In this window, you can define hinges for members and sets of members that do not transfer locked degrees of freedom specified in Window 1.7 as internal forces for structural reasons. Please note, that no double hinges are generated in the interaction with Window 1.7.



Details...

The table manages the hinge parameters of a set of members selected in the navigator on the left.

If the *Equivalent Member Method* is selected in the *Details* dialog box (see Figure 2.2, page 44), Window 1.8 is not displayed. You can define intermediate lateral restraints by using division points in Window 1.4.

1.8 Member Hinges - Set of Members No. 2 - Column Members 2

Release No.	A Member No.	B Member Side	C Shear Release V_y	D Moment Release M_T	E Moment Release M_z [kNm/rad]	F Warp Release M_ω	G Comment
1	15	Start	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	
2	13	End	<input type="checkbox"/>	<input type="checkbox"/>	15,000	<input type="checkbox"/>	
3							
4							
5							
6							
7							
8							
9							
10							

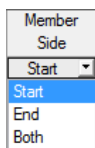
Settings - Member No. 13

<input checked="" type="checkbox"/> Set of Members	Column Members 2
Cross-Section	2 - IPE 300 Euronorm 19-57
Member with Release at the End	No. 13
Member Side	End
Shear Release in y-Direction	<input type="checkbox"/> V_y
Torsional Release	<input type="checkbox"/> M_T
Moment Release about z-Axis	15,000 kNm/rad M_z
Warping Release	<input type="checkbox"/> M_ω
Comment	

Set input for release No.:

All

Figure 2.32: Window 1.8 *Member Hinges - Set of Members*



Column B shows the *Member Side* on which a hinge is located or if there are hinges on both member sides.

In columns C to F, you can define hinges or spring constants to adjust the set of members model to the support conditions in Window 1.7.

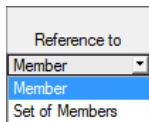
2.9 Serviceability Data

This input window controls various settings for the serviceability limit state design. The window is available only if you have entered the corresponding data in the *Serviceability Limit State* tab in Window 1.1 (see [Chapter 2.1.2, page 10](#)).

1.9 Serviceability Data

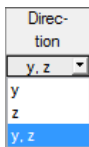
No.	A Reference to	B Set of Members No.	C Reference Length Manually	D Reference Length L [m]	E Direc- tion	F Precamber w_c [mm]	G Beam Type	H Comment
1	Set of Members	2	<input type="checkbox"/>	12.548	y, z	0.0	Beam	
2	Set of Members	5	<input type="checkbox"/>	7.094	y, z	0.0	Beam	
3	Member	82	<input type="checkbox"/>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

Figure 2.33: Window 1.9 *Serviceability Data*



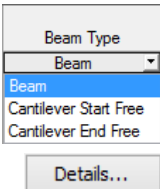
Column A shows if the deformation refers to single members, lists of members, or to sets of members.

Column B shows the numbers of members or sets of members to be designed. You can select the numbers graphically in the RFEM work window using the button. Then, the *Reference Length* automatically appears in column D. You can define the values *Manually* if you select the check box in column C.



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

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



For the correct application of limit deformations, the *Beam Type* is especially crucial. In column G, you can specify whether a beam or a cantilever is to be designed and which end is free of support.

The setting in the *Serviceability* tab of the *Details* dialog box indicates whether the deformations are related to the undeformed system or to shifted members ends / set of members ends (see [Figure 2.3, page 46](#)).

2.10 Fire Resistance - Members

This input window includes the fire resistance parameters. It appears only if you have entered the corresponding data in the *Fire Resistance* tab of Window 1.1 (see [Chapter 2.1.3, page 11](#)).

1.10 Fire Resistance - Members

No.	A	B	C	D	E	F	G	H	I
	Members No.	Fire Exposure	Fire Protection	Protection Type	Unit Mass ρ_p [kg/m ³]	Thermal Conductivity λ_p [W/m*K]	Specific Heat c_p [J/(kg*K)]	Thickness d_p [mm]	Comment
1	64	All Sides	<input type="checkbox"/>	Contour	300.00	0.12	1200.00	10.00	
2	81-83	3 Sides	<input checked="" type="checkbox"/>	Contour	300.00	0.12	1200.00	10.00	
3	39,59,60,109	3 Sides	<input checked="" type="checkbox"/>	Hollow	300.00	0.12	1200.00	10.00	
4	1.11	All Sides	<input type="checkbox"/>	Contour	300.00	0.12	1200.00	10.00	
5	21,31	All Sides	<input checked="" type="checkbox"/>	Contour	300.00	0.12	1200.00	10.00	
6									
7									
8									
9									
10									
11									
12									
13									
14									
15									
16									
17									
18									
19									
20									
21									
22									
23									
24									
25									
26									
27									
28									
29									
30									
31									
32									

Figure 2.34: Window 1.10 *Fire Resistance - Members*

Column A contains the members considered in the fire resistance design. Click to graphically select the members in the RFEM work window.



It is possible to perform analyses only for members selected for design in the 1.1 *General Data* window (see [Figure 2.2, page 7](#)).

Fire Exposure
 All Sides ▾
 3 Sides
 All Sides

Protection Type
 Contour ▾
 Contour
 Hollow

Details...

In column B, you can define the number of cross-section sides exposed to fire. The *Fire Exposure* affects the determination of the section factors according to [2], Table 4.2 and Table 4.3.

If there is an encasement for fire resistance, you can select the *Protection Type* in column D. You can choose between a contour encasement, following the geometry of the cross-section (for example intumescent coating) and a hollow encasement of the cross section. Then, you can specify the corresponding parameters in columns E to H.

The general parameters for the fire resistance design are managed in the *Fire Resistance* tab of the *Details* dialog box (see [Figure 2.4, page 47](#)).

The fire resistance design in RF-STEEL EC3 is described in the following article of DLUBAL Blog: www.dlubal.com/blog/1242

Details...

2.11 Fire Resistance - Sets of Members

This window appears only if you have selected at least one set of members for design in the 1.1 *General Data* window and entered the corresponding data in the *Fire Resistance* tab of Window 1.1 (see [Chapter 2.1.3, page 11](#)).

1.11 Fire Resistance - Sets of Members

No.	A Sets of Members No.	B Fire Exposure	C Fire Protection	D Protection Type	E Unit Mass ρ_p [kg/m ³]	F Thermal Conductivity λ_p [W/m*K]	G Specific Heat c_p [J/(kg*K)]	H Thickness d_p [mm]	I Comment
1	1	All Sides	<input type="checkbox"/>	Contour	300.00	0.12	1200.00	10.00	
2	2	3 Sides	<input checked="" type="checkbox"/>	Contour	300.00	0.12	1200.00	10.00	
3	3.4	All Sides	<input checked="" type="checkbox"/>	Hollow	300.00	0.12	1200.00	12.00	
4									
5									
6									
7									
8									
9									
10									
11									
12									
13									
14									
15									
16									
17									
18									
19									
20									
21									
22									
23									
24									
25									
26									
27									
28									
29									
30									
31									
32									

Figure 2.35: Window 1.1 *Fire Resistance - Sets of Members*

The concept of the window is similar to the previous 1.10 *Fire Resistance - Members* window. You can enter the fire protection parameters of the corresponding sets of members, as described in [Chapter 2.10](#).

2.12 Parameters - Members

This window allows you to enter specifications for beams that are laterally supported by sheeting or purlins (see [6], Clause 10.1 and 10.3).

The upper section lists the members selected for design together with the parameters relevant for the lateral-torsional buckling design. These parameters interact with the specifications in the *Settings - Member No.* section below.

To the right of the *Settings* table, you can see information or selecting options in the form of graphics facilitating the definition of boundary conditions. The views vary depending on the currently selected parameter.

1.12 Parameters - Members

Member No.	A		B		C		D	
	Shear Panel	Rotational Restraint	Cross-Sectional Area				Comment	
1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>				Purlin	
2	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
5	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>				Trapezoidal sheeting	
14	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>				Trapezoidal sheeting	
15	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
19	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
22	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>				Purlin	
27	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					
28	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>					

Settings - Member No. 14

Cross-Section		1 - IPE 400 Euronorm 19-57
<input type="checkbox"/> Shear panel		<input checked="" type="checkbox"/>
<input type="checkbox"/> Shear panel type		Trapezoidal sheeting
Shear panel length	l_s	20.000 m
Beam spacing	s	5.000 m
Position on section		On upper flange
<input type="checkbox"/> Trapezoidal sheeting description		FI + 100/275 - 1.00
Shear panel coefficient	K_1	0.190 m/kN
Shear panel coefficient	K_2	16.560 m ² /kN
Fastening arrangement		Every Rib
<input type="checkbox"/> Rotational restraint		<input checked="" type="checkbox"/>
<input type="checkbox"/> Type of rotational restraint		Continuous (e.g. Sheeting)
<input type="checkbox"/> Materials		Steel S 235
Modulus of Elasticity	E	21000.00 kN/cm ²
<input type="checkbox"/> Component description		FI + 100/275 - 1.00
Sheeting thickness	t	1.000 mm
Position of sheeting		Positive position
Second moment of area	I_s	198.00 cm ⁴ /m
Distance of ribs	b_R	275.0 mm

Set input for members No. : All

Trapezoidal sheeting shear panel

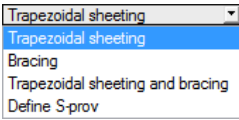
Figure 2.36: Window 1.12 Parameters - Members

Below the *Settings* table, there is the *Set input for members No.* check box. If selected, the subsequent settings applies to the selected members or *All* members (you can enter the member numbers manually or select them graphically using the button). This option is useful when assigning several members to the same boundary conditions.

In the *Comment* column, you can enter user-defined comments for each member, for example to describe a member's parameters relevant for lateral-torsional buckling.

Cross-Section

This column shows the cross-section description. For a tapered member, the description of the start and end cross-section is displayed.



Shear Panel

To enter the shear panel parameters, select the check box in column A or in the *Settings* table. You can select the shear panel type in the drop-down list.

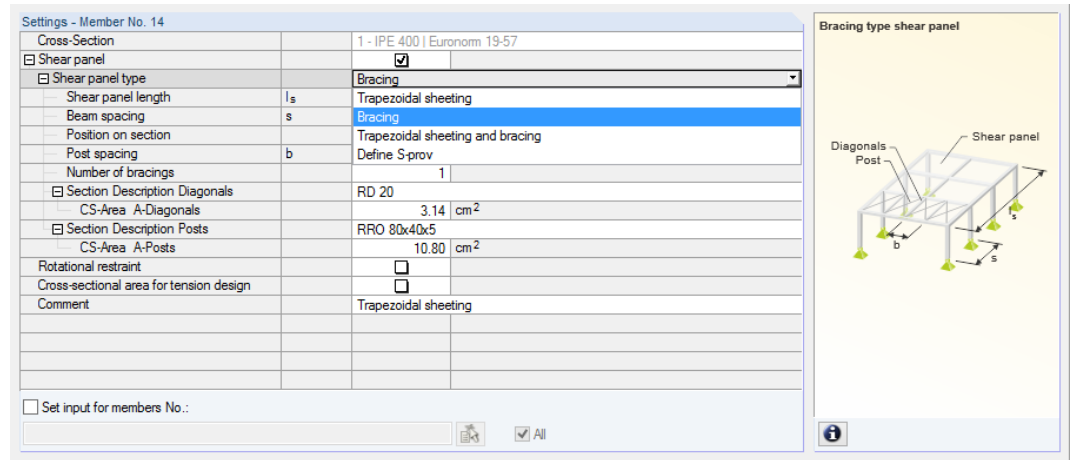


Figure 2.37: Selecting shear panel type

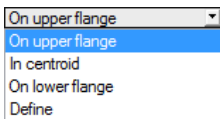
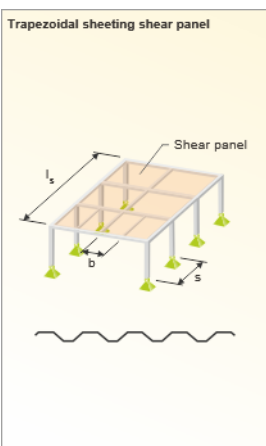
Trapezoidal sheeting

The application of a continuous lateral support is described in EN 1993-1-1 [1], Annex BB.2.1 and EN 1993-1-3 [6], Clause 10.1.5.1.

To determine the shear panel stiffness of a trapezoidal sheeting (corrugated sheet), the following specifications are required (see Figure 2.36):

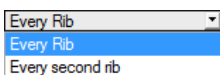
- Shear panel length l_s
- Beam spacing s
- Position of trapezoidal sheeting on section
- Trapezoidal sheeting description
- Fastening arrangement

You can enter the **Shear panel length** and the **Beam spacing** manually or select them graphically by clicking . This button becomes available when the cursor is placed in one of these text boxes. Then, you can select two snap points in the RFEM work window defining the shear panel or the beam spacing.



The **Position on section** of the trapezoidal sheeting can be considered in different ways by means of the list shown on the left. The selected point of torsion D is marked in the cross-section graphic, even for user-defined entries. The distance d is relative to the centroid here; the sign results from the z -axis of the cross-section.

To access the corrugated sheets library, click the button displayed after you click in the **Trapezoidal sheeting description** text box (see Figure 2.36). The RFEM cross-section library appears (see Figure 2.38), where you can select the trapezoidal sheeting by double-clicking or clicking [OK]. Thus, the *shear panel coefficients* K_1 and K_2 (according to the approval certificate) are automatically entered in the *Settings* table. The basic width b of the trapezoidal sheeting has no influence on these coefficients.



The **Fastening arrangement** of the trapezoidal section affects the shear stiffness that the sheeting provides to the beam. If the trapezoidal sheeting is fastened only in every second rib, the shear stiffness to be applied is reduced by a factor of 5.

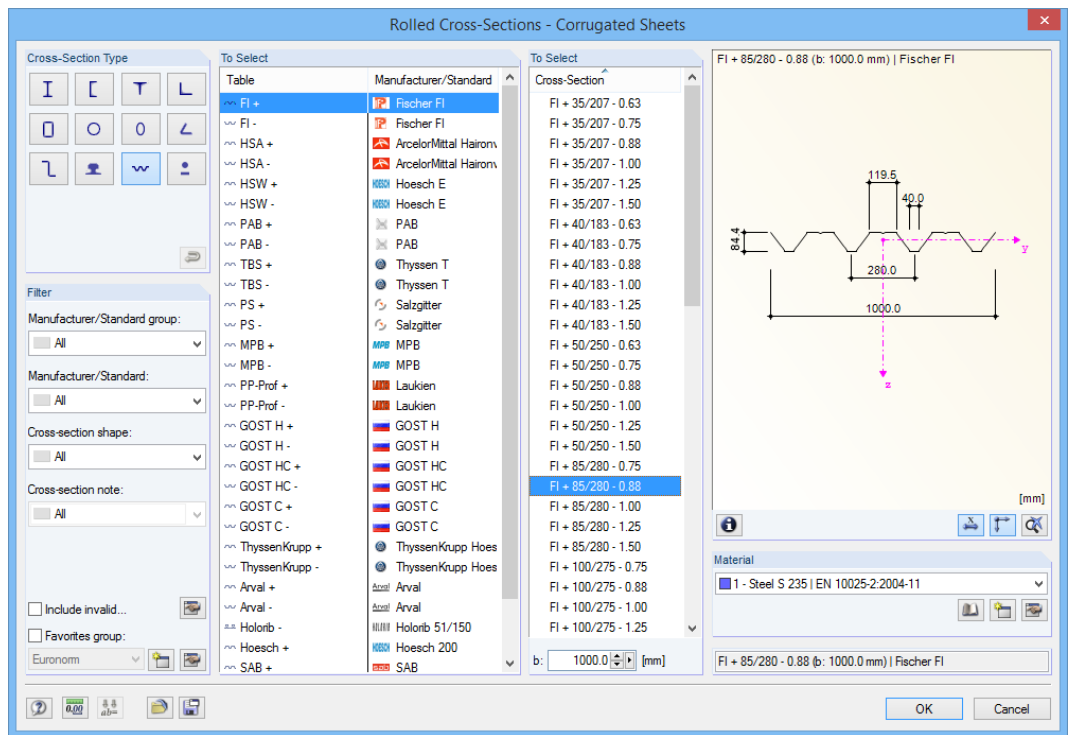


Figure 2.38: Cross-section library *Rolled Cross-Sections - Corrugated Sheets*

Bracing

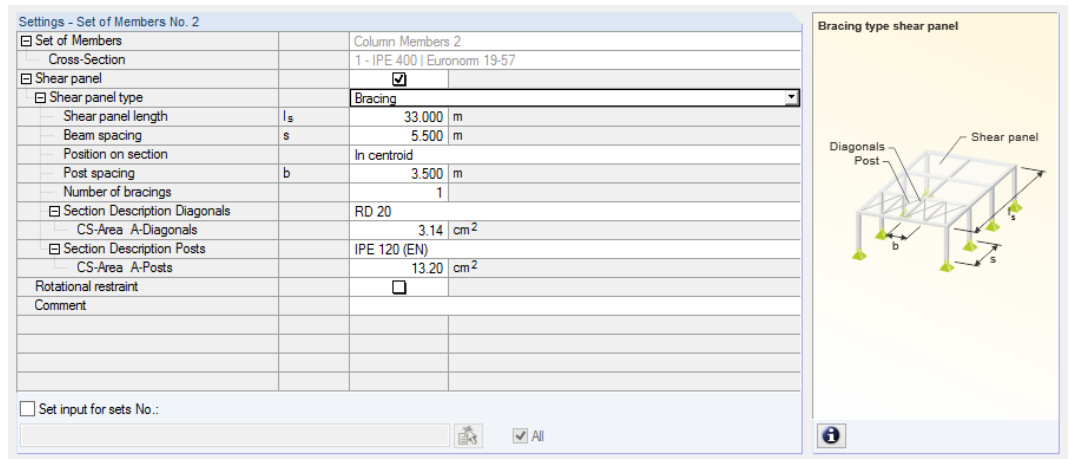


Figure 2.39: Shear panel type *Bracing*

To determine the provided shear panel stiffness, the following specifications are required:

- Shear panel length l_s
- Beam spacing s
- Position of the bracing on section
- Post spacing b
- Number of bracings
- Section of diagonals
- Section of posts



You can enter the **Shear panel length**, the **Beam spacing**, and the **Post spacing** manually or select it graphically by clicking . This button becomes available when the cursor is placed in one of these text boxes. Then, you can select two points in the RFEM work window, defining the shear panel or the spacing.

The bracing's **Position on section** can be considered in different ways using the list shown on the left. The selected point of torsion D is marked in the cross-section graphic, even for user-defined entries. The distance d is relative to the centroid here; the sign results from the z-axis of the cross-section.

You can easily define the cross-sectional area of the diagonals and posts by selecting the **Section Description** in the RFEM library. To access the library, click the button at the end of the text box. Then, the **CS-Area** is imported automatically. You can also enter the value directly.

Trapezoidal sheeting and bracing

Figure 2.40: Shear panel type *Trapezoidal sheeting and bracing*

To determine the provided shear panel stiffness due to trapezoidal sheeting and bracing, the following specifications are required:

- Shear panel length l_s
- Beam spacing s
- Position of shear panel on section
- Trapezoidal sheeting description
- Fastening arrangement
- Post spacing b
- Number of bracings
- Section of diagonals
- Section of posts

This way of defining the shear panel combines the parameters of the aforementioned options *Trapezoidal sheeting and Bracing*.

Defining S-prov

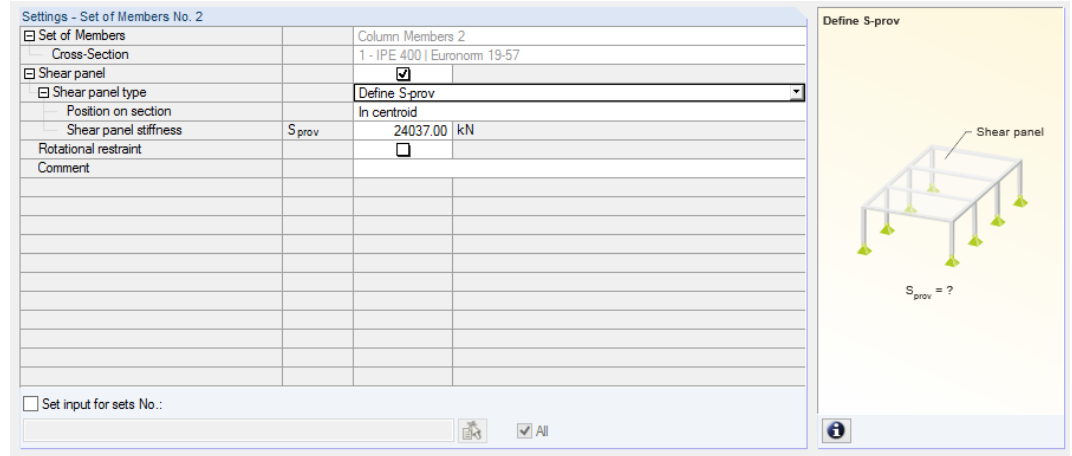
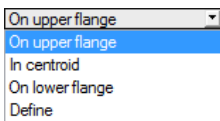


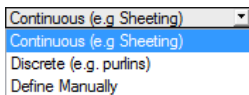
Figure 2.41: Defining shear panel stiffness S_{prov}



You can also enter the value of the provided **Shear panel stiffness** S_{vorh} directly. In addition, you have to specify the shear panel's **Position on section**.

Rotational Restraint

To enter the rotational restraint parameters, select the check box in column B or in the *Settings* table.



You can select the type of rotational restraint in the drop-down list.

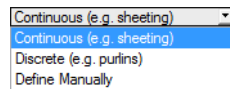


Figure 2.42: Selecting the type of rotational restraint

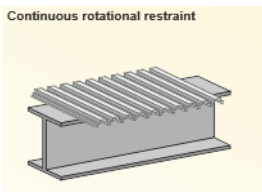
Continuous rotational restraint

In order to determine the stiffness components from trapezoidal sheeting and the connection deformation, the following specifications are necessary (see [Figure 2.42](#)):

- Material and description of the trapezoidal sheeting
- Method of determining $C_{D,A}$
- Beam spacing s
- Continuous beam effect

To access the corrugated sheet library, click the button displayed after you click in the **Component description** text box. The RFEM cross-section library appears (see [Figure 2.38](#), page 36), where you can select a corrugated sheet by double-clicking or clicking [OK]. The section parameters *Sheeting thickness* t , *Position of sheeting*, effective *Second moment of area* I_s for the downward loading direction, *Distance of ribs* b_R (corrugation width), and *Width of the flange* b_T are imported automatically.

In the case of continuous rotational restraint, you have to consider the deformation of the connection. You can specify the rotational spring stiffness C_{100} for the individual load cases and combinations in the **Method of determining** $C_{D,A}$ entry, or the program determines it according to [6], Table 10.3. For automatic calculation, click the button shown after clicking in the text box of the row C_{100} . A dialog box appears where you can select the appropriate coefficient.




Import of Coefficient C-100 from Table 10.3, EN 1993-1-3

Positioning of sheeting		Sheet fastened through		Positioning of sheeting		Washer diameter [mm]	C ₁₀₀ [kNm/m]	b _{T,max} [mm]
Positive 1)	Negative 1)	Trough	Crest	e=b _R	e=2b _R			
For gravity loading:								
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		22	5.2	40
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>	22	3.1	40
	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	K _s	10.0	40
	<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>	K _s	5.2	40
	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>	22	3.1	120
	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>	22	2.0	120
For uplift loading:								
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>		16	2.6	40
<input checked="" type="checkbox"/>		<input checked="" type="checkbox"/>			<input checked="" type="checkbox"/>	16	1.7	40

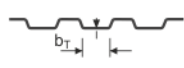
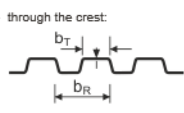
Note: Select by mouse the required lines in the table and import by clicking [OK] the coefficient.

Key:
 b_R is the corrugation width
 b_T is the width of the sheeting flange through which it is fastened to the purlin.

K_s indicates a steel saddle washer as shown below with t ≥ 0.75 mm



The values in this table are valid for:
 - sheet fastener screws of diameter: $\varnothing = 6.3$ mm;
 - steel washers of thickness: $t_w \geq 1.0$ mm;
 - sheeting of nominal core thickness: $t \geq 0.66$ mm;

Sheet fastened:
 - through the trough:

 - through the crest:



1) The position of the sheeting is positive when the narrow flange is on the purlin and negative when the wide flange is on the purlin.

C₁₀₀
 3.1 [kNm/m]

OK Cancel

Figure 2.43: Dialog box *Import of Coefficient C-100 from Table 10.3, EN 1993-1-3*

Click [OK] to assign this value to all load cases and combinations selected for the design. In order to assign the coefficient by load case, you have to open the *Import of Coefficient* dialog box via the C₁₀₀ text boxes of the individual load cases and combinations.

You can also define the **Beam spacing** manually or graphically by clicking the  button. For this, click two nodes in the RFEM work window defining the distance between the beams.

End panel
 End panel
 Internal panel

The **Continuous beam effect** has an impact on the coefficient k of the rotational restraint C_{D,C}, definable by means of the list in this row (*End panel*: k = 2, *Internal panel*: k = 4).

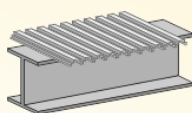
Discrete rotational restraint

Settings - Member No. 11

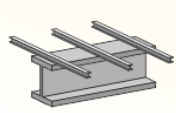
Cross-Section	2 - IPE 300 Euronorm 19-57	
Shear panel	<input type="checkbox"/>	
Rotational restraint	<input checked="" type="checkbox"/>	
Type of rotational restraint	Discrete (e.g. purlins)	
Materials	Steel S 235	
Modulus of Elasticity	E	21000.00 kN/cm ²
Cross-section description	IPE 180 (EN)	
Second moment of area	I _y	1320.00 cm ⁴
Purlin spacing	e	3.250 m
Beam spacing	s	5.200 m
Continuous beam effect	End panel	
Cross-sectional area for tension design	<input type="checkbox"/>	
Comment		

Set input for members No.:

Continuous rotational restraint



Discrete rotational restraint



OK All

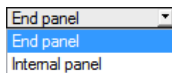
Figure 2.44: Type of rotational restraint *Discrete*

In order to determine the stiffness component from isolated columns (for example purlins), the following specifications are required:

- Material and description of the cross-section
- Purlin spacing e
- Beam spacing s
- Continuous beam effect

Material and *Cross-section description* can be selected from the RFEM library using the button. Select the relevant text box and click in it.

You can enter the *Purlin spacing* and the *Beam spacing* manually or graphically by clicking the button. For this, select two nodes defining the spacing of the purlins or beams by clicking them in the RFEM work window.



The *Continuous beam effect* has an impact on the coefficient k of the rotational restraint $C_{D,C}$, definable by means of the list in this row (*End panel*: $k = 2$, *Internal panel*: $k = 4$).

Define Manually

Settings - Member No. 2		
Cross-Section		1 - IPE 400 Euronorm 19-57
Shear panel		<input type="checkbox"/>
Rotational restraint		<input checked="" type="checkbox"/>
<input type="checkbox"/> Type of rotational restraint		Define Manually
<input type="checkbox"/> Total Rotational Spring Stiffness	C_D	18.50 kNm/m
Cross-sectional area for tension design		<input type="checkbox"/>
Comment		

Figure 2.45: Rotational Spring Stiffness - Define Manually

You can also directly enter the existing **Total Rotational Spring Stiffness** C_D .

Cross-Sectional Area for Tension Design

Settings - Member No. 52		
Cross-Section		8 - RO 101.6x5 (Cold Formed)
Shear panel		<input type="checkbox"/>
Rotational restraint		<input type="checkbox"/>
<input checked="" type="checkbox"/> Cross-sectional area for tension design		<input checked="" type="checkbox"/>
<input type="checkbox"/> Start ($x=0$ m)		8 - RO 101.6x5 (Cold Formed)
<input type="checkbox"/> Cross-Sectional Area	A	15.2 cm ²
<input type="checkbox"/> Net Cross-Sectional Area	A_{net}	12.70 cm ²
<input type="checkbox"/> End ($x=l$)		8 - RO 101.6x5 (Cold Formed)
<input type="checkbox"/> Cross-Sectional Area	A	15.2 cm ²
<input type="checkbox"/> Net Cross-Sectional Area	A_{net}	15.20 cm ²
Comment		

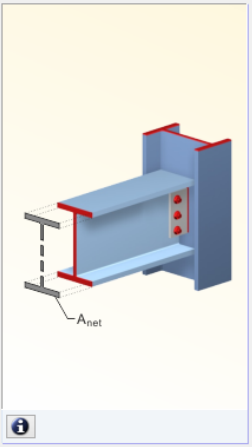


Figure 2.46: Cross-sectional area for tension design

According to [1], Clause 6.2.3, holes for fasteners are considered in tensile stress design. You can define the **Net Cross-Sectional Area** A_{net} separately for the *Start* and *End* of the member - fasteners are usually located at these two x -locations. The table also shows the gross cross-sectional area A .

2.13 Parameters - Sets of Members

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

1.13 Parameters - Sets of Members

Set No.	A Shear Panel	B Rotational Restraint	C Comment
1	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
2	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
3	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
4	<input type="checkbox"/>	<input checked="" type="checkbox"/>	

Settings - Set of Members No. 1

<input type="checkbox"/> Set of Members	Ceiling beam B-B
Cross-Section	3 - ICL IPE 300 + IPE 300-HMAX Euronorm 19-57 + Euronorm 19-57
Shear panel	<input type="checkbox"/>
<input type="checkbox"/> Rotational restraint	<input checked="" type="checkbox"/>
Type of rotational restraint	Continuous (e.g. Sheeting)
Materials	Steel S 235
Modulus of Elasticity	E 21000.00 kN/cm ²
<input type="checkbox"/> Component description	HSW - E 135 - 1.00
Sheeting thickness	t 1.000 mm
Position of sheeting	Negative position
Second moment of area	I _s 387.00 cm ⁴ /m
Distance of ribs	b _R 310.0 mm
Width of the flange	b _T 43.0 mm
<input type="checkbox"/> Method of determining C _{D,A}	According to EN 1993-1-3, Table 10.3
Spring stiffness	C ₁₀₀
RC1	10.00 kNm/m
Beam spacing	s 6.000 m
Continuous beam effect	End panel
Comment	

Set input for sets No.: All

Rotational restraint

Beam stabilized through elastic rotational restraint (e.g. trapezoidal sheeting, purlins)

Figure 2.47: Window 1.13 *Parameters - Sets of Members*

The concept of this window is similar to the previous 1.12 *Parameters - Members* window. In this window, you can define the parameters of shear panels and rotational restraints for each set of members as described in [Chapter 2.12](#).

3 Calculation

3.1 Detail Settings

Details...

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

The *Details* dialog box has the following tabs:

- Ultimate Limit State
- Stability
- Serviceability
- Fire Resistance
- Other

3.1.1 Ultimate Limit State

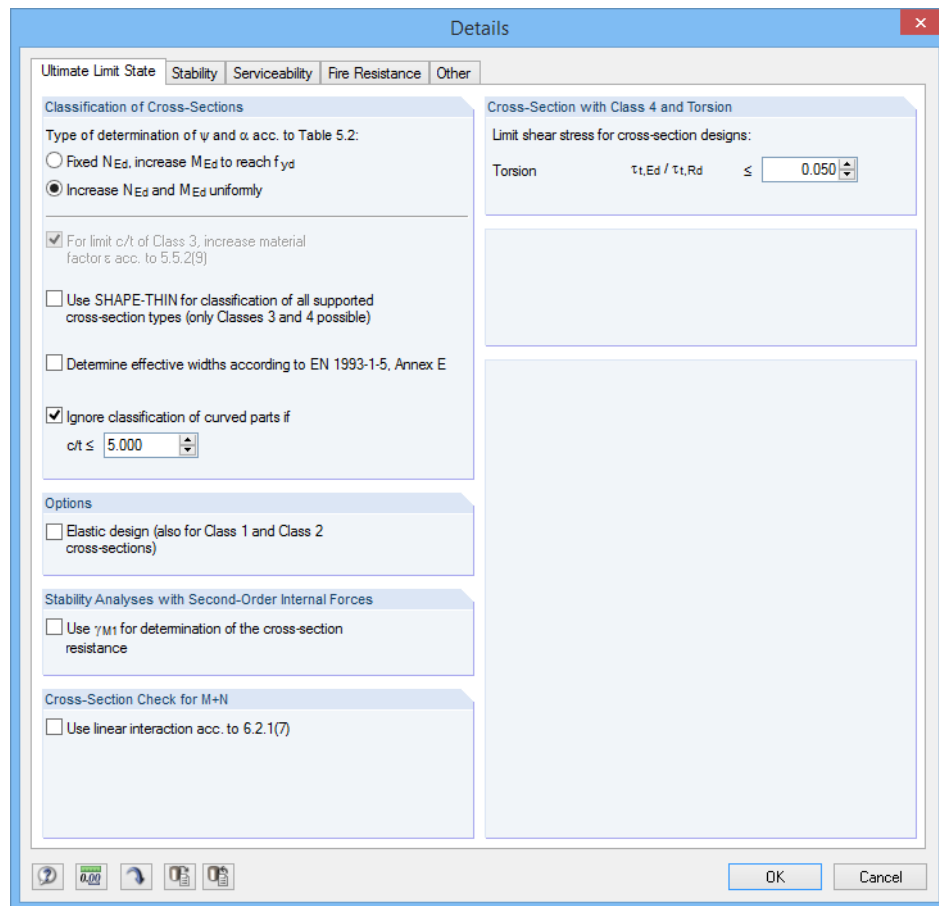


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

Classification of Cross-Sections

If stresses from compression and bending occur together in a cross-section, you can determine the stress deformation ratio ψ with regard to the compression zone factor α in two ways (the factor ψ is required for the determination of the appropriate c/t -ratio according to [1], Table 5.2):

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

The *For limit c/t of Class 3, increase material factor ε acc. to 5.5.2(9)* check box appears if the stability analysis has been deactivated in the *Stability* tab. This is based on the specifications for classification in [1], Clause 5.5.2 (10). If the stability analysis is deactivated you can consider the cross-sections classified as Class 4 for cross-sections of Class 3 by increasing the factor ε .

If you select the *Use SHAPE-THIN for Classification of all supported cross-section types* check box, the effective cross-section properties of Class 4 sections will be calculated according to the method used in SHAPE-THIN. For cross-sections classified as 'General' (that is, they belong neither to a rolled nor a parametrized cross-section table), the classification will be generally performed with SHAPE-THIN. You can design these cross-sections only elastically as Class 3 or Class 4 cross-sections.

Optionally, you can *Determine effective widths according to EN 1993-1-5, Annex E. [7]*, Annex E describes the alternative methods for determining effective cross-section areas for stresses below the yield strength (see also DLUBAL Blog www.dlubal.com/blog/5535).

The width/thickness ratios relevant for the classification could cause problems with section in case of SHAPE-THIN curved elements. Using the *Ignore classification of curved parts* check box, you can exclude short fillet arcs from the classification once the user-defined c/t -ratio is below the limit (see DLUBAL Blog www.dlubal.com/blog/11166). Longitudinal ribs or slopes of thin sheetings has no influence on the design.

Options

Cross-sections assigned to Class 1 or 2 are designed plastically in RF-STEEL EC3. If you do not want to perform a plastic design, you can activate the *Elastic Design* also for these cross-section classes.

Stability Analyses with Second-Order Internal Forces

If the stability analyses are not performed with the equivalent member method according to [1], Clause 6.3, but with second-order internal forces, you can use this check box to specify whether to use the partial safety factor γ_{M1} (instead of γ_{M0}) for the cross-section design.

Nat. Annex...

The partial safety factor γ_{M1} is relevant for the resistance determination in case of instability (structural component check). You can check and/or modify the safety factor in the *National Annex Settings* dialog box (see [Chapter 3.10, page 12](#)).

Cross-Section Check for M+N

The *Use linear interaction acc. to 6.2.1(7)* check box determines whether to use the linear addition of the utilization ratios for the moments and axial forces according to [1], Eq. (6.2), or Eq. (6.44) as conservative approximation for the resistance verification of the cross-section.

3.1.2 Stability

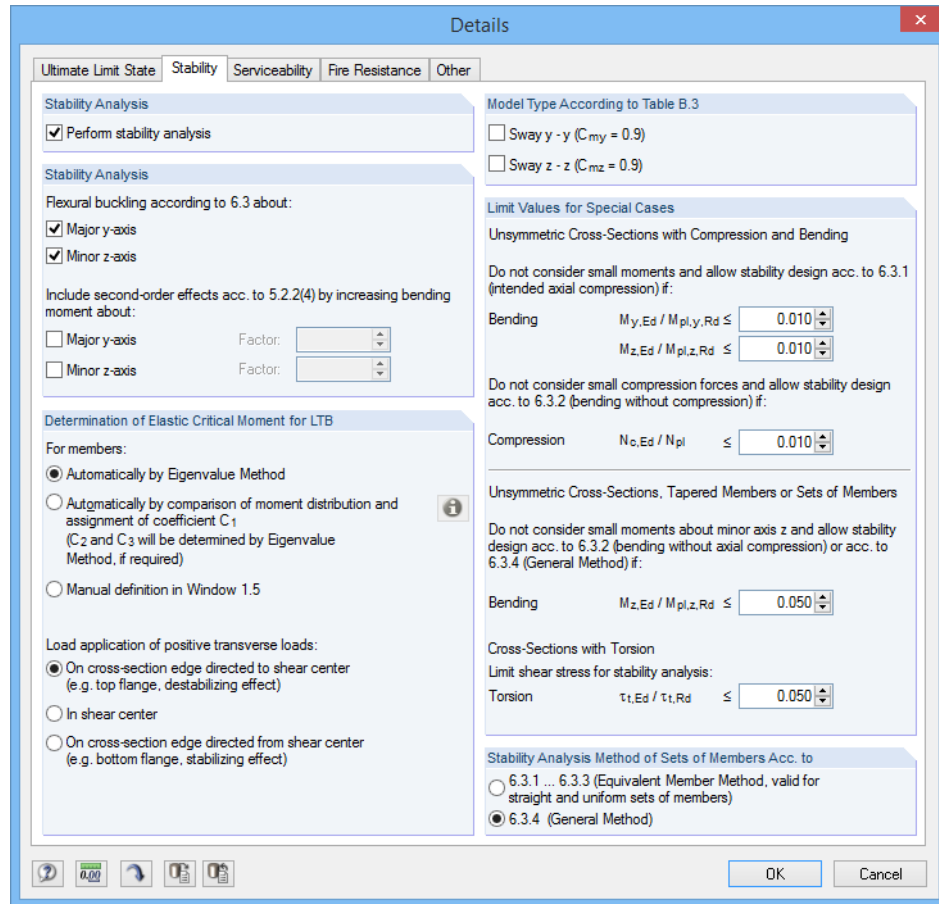


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

The *Perform stability analysis* check box controls whether to run a stability analysis in addition to the cross-section designs. If you clear the check box, the Windows 1.4 to 1.8 will not appear.

If you select the check box, you can define the axes relevant for the determination of *Flexural buckling according to 6.3* of [1].

In addition, you can *Include second-order effects acc. to 5.2.2(4) by increasing bending moment about* an axis that you can define manually. In this way, when designing for example a frame with governing buckling mode represented by lateral displacement, you can determine the internal forces according to linear static analysis and increase them with the appropriate factors. The bending moment increment has no effect on the flexural-buckling analysis according to [1], Clause 6.3.1, which is performed by using the axial forces.

Determination of Elastic Critical Moment for LTB

By default, RF-STEEL EC3 determines the elastic critical moment for lateral-torsional buckling *Automatically by Eigenvalue Method*. For the calculation, the program uses a finite member model to determine M_{cr} with regard to the following items:

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

You can specify the degrees of freedom by the factors k_z and k_w (see [Chapter 2.5, page 22](#)).



You can also determine the elastic critical moment *Automatically by comparison of moment distribution* and assignment of the coefficient C_1 . Click the [Info] button to open the corresponding dialog box and view the load and moment distributions. The coefficients C_2 and C_3 will be determined automatically by the Eigenvalue Method, if required.

H	I	J	K	L
Lateral-Torsional and Torsional-Flexural Buckling				
Possible	k_z	M_{cr} [kNm]	L_w [m]	L_T [m]
<input checked="" type="checkbox"/>	1.0	100.00 →	6.059	6.059
<input checked="" type="checkbox"/>	1.0	100.00	3.843	3.843
<input checked="" type="checkbox"/>	1.0	100.00	6.700	6.700
<input checked="" type="checkbox"/>	1.0	100.00	6.700	6.700

M_{cr} user-defined

Select the *Manual definition in Window 1.5* check box to change the name of column J to M_{cr} , so you can directly enter the elastic critical moment for LTB.

If there are *transverse loads*, it is necessary to define the location where these forces are acting on the cross-section: Depending on the load application point, transverse loads can be stabilizing or destabilizing, and thus influence the elastic critical moment.

Model Type According to Table B.3

According to [1], Annex B, Table B.3, the equivalent uniform moment factor for structural components with buckling in the form of lateral deflection should be taken as $C_{my} = 0.9$ or $C_{mz} = 0.9$. Both check boxes are cleared by default. If you select the check boxes, the program determines the factors C_{my} and C_{mz} according to the criteria defined in Table B.3.

Limit Values for Special Cases

To design unsymmetric cross-sections with the intended axial compression according to [1], Clause 6.3.1, you can neglect *small moments* about the major and the minor axes using the settings defined in this dialog box section.

For the pure check of bending according to [1], Clause 6.3.2, you can similarly neglect *small compression forces* where the limit ratio of $N_{c,Ed} / N_{pl}$ is defined.

According to [1], Clause 6.3.4, the design of *Unsymmetric Cross-Sections, Tapered Members or Sets of Members* is only possible for uniaxial bending in the principal plane and/or compression. In order to neglect the small moments about the minor axis, you can define a limit for the moment ratios $M_{z,Ed} / M_{pl,z,Rd}$.

The intended *torsion* is not clearly specified in [1]. If there is a torsional stress not exceeding the shear stress ratio of 5 % preset by default, it is not considered in the stability design; only results for flexural and lateral-torsional buckling are displayed.



If one of the limits in this dialog section is exceeded, a note appears in the result window and no stability analysis is performed. However, the cross-section designs are performed independently. These limit settings are not part of the Standard [1] or any National Annex. Modification of the limits is in the user's responsibility.

Stability Analysis Method of Sets of Members



The stability behavior of sets of members can be analyzed according to two methods.

According to 6.3.1 ... 6.3.3 (*Equivalent Member Method*), it is possible to treat sets of members as one large member. For this, define the factors k_z and k_w in the 1.6 *Effective Lengths - Sets of Members* window. These factors are used to determine the support conditions β , u_y , φ_x , φ_z , and ω . If applied, Windows 1.7 and 1.8 will not be displayed. Please note that the factors k_z and k_w are identical for each section or member of the set of members. In general, you should use the equivalent member method only for straight sets of members.

If you select 6.3.4 (*General Method*), the program performs a general analysis according to [1], Clause 6.3.4, based on the coefficient α_{cr} . In the 1.7 *Nodal Support - Set of Members* window, you can define the support conditions with regard to the stability failure (buckling and lateral-torsional buckling) for each set of members individually. The factors k_z and k_w from Window 1.5 are not used.

3.1.3 Serviceability

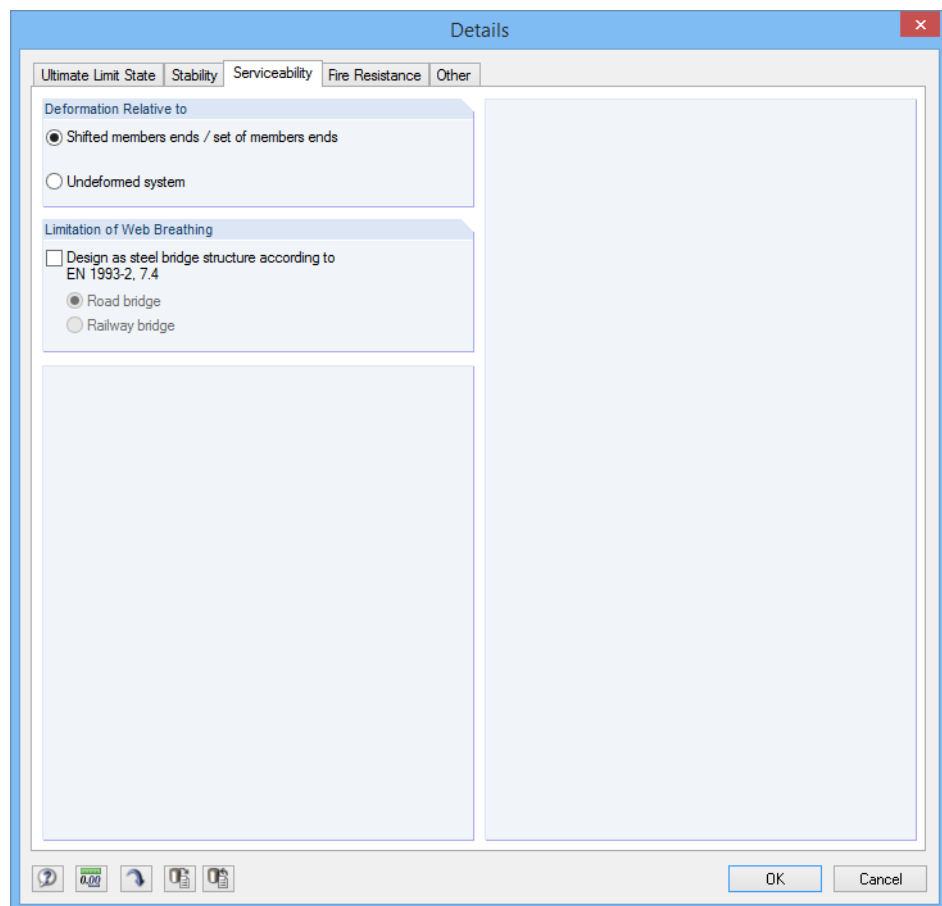


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Relative To

These options specify whether the maximum deformations are relative to the shifted members ends / set of members ends (connection line between start and end nodes of a deformed system) or to the undeformed initial system. Generally, the deformations are designed as relative to the displacements in the entire structural system.

Nat. Annex...

You can check and, if necessary, adjust the limit deformations in the *National Annex Settings* dialog box (see [Figure 3.10](#), page 12).

Limitation of Web Breathing

In the case of the serviceability limit state design of steel bridges, it is necessary to check the slenderness ratio of web plates in order to avoid excessive rippling and breathing of plates as well as stiffness reductions due to plate buckling. The *Design as steel bridge structure according to EN 1993-2, 7.4* [8] check box allows you to analyze the breathing (repeated out-of-plane deformation), which can result in fatigue problems at the web-to-flange connections.

It is necessary to define whether you design a *Road bridge* or a *Railway bridge* as there are different criteria for each case.

3.1.4 Fire Resistance

In this tab, you can control detail settings for the fire resistance design.

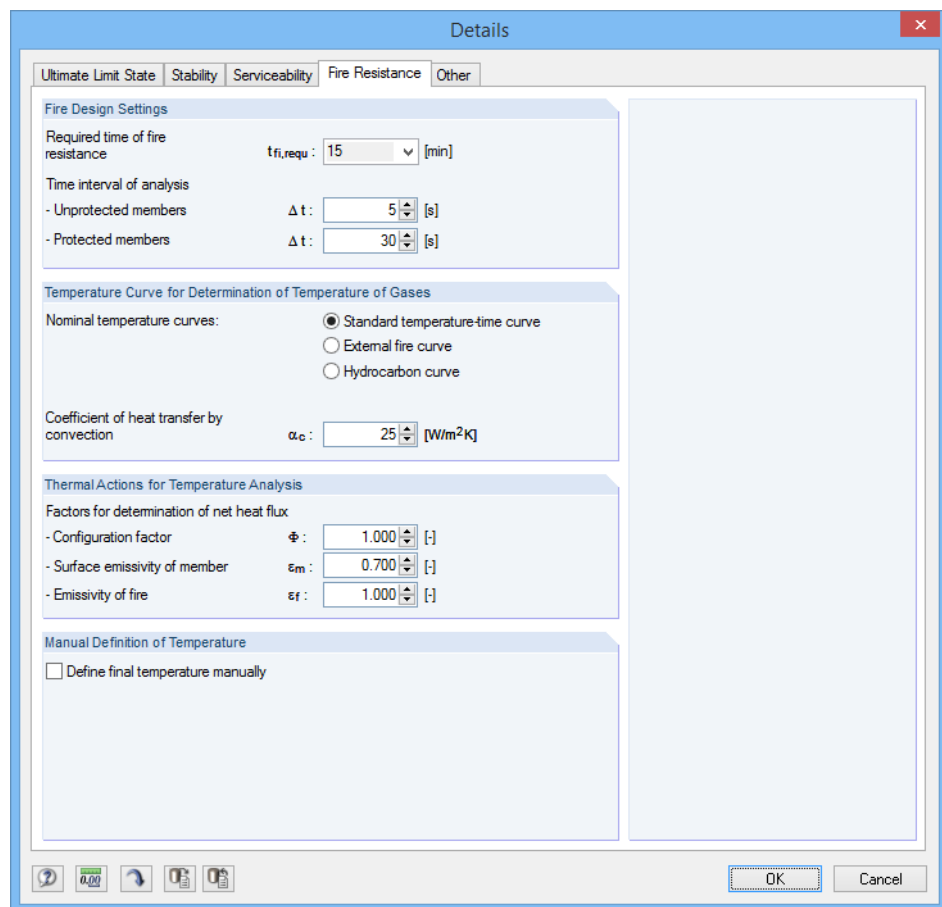


Figure 3.4: Dialog box *Details*, tab *Fire resistance*

In order to determine the temperature change, you should define the *Required time of fire resistance* and the *Time interval of analysis* as well as the *Temperature Curve for Determination of Temperature of Gases*. You can select one of three nominal temperature curves illustrated in [Figure 3.5](#) to [Figure 3.7](#) on the next page.

The *Factors for determination of net heat flux* are preset in accordance with [9] and [2]. However, you can adjust them according to specific conditions.

If you select the *Define final temperature manually* check box, then it is possible to individually define the temperature θ_a in Window 1.10 and 1.11.

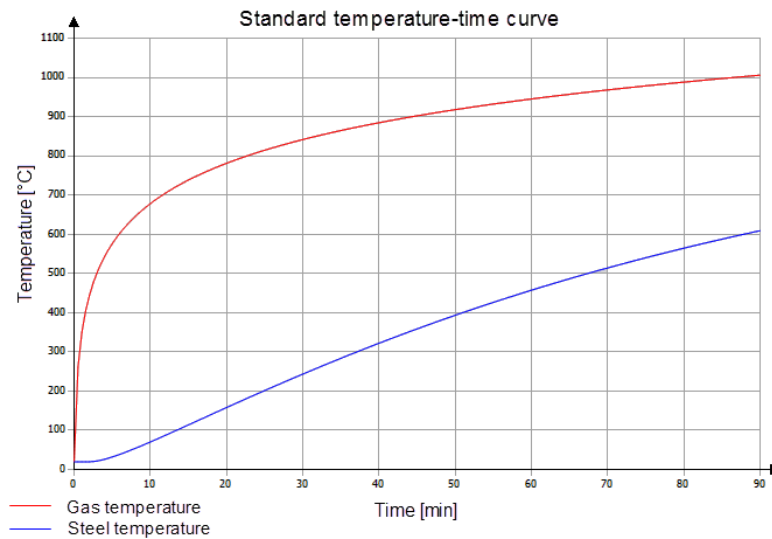


Figure 3.5: Standard temperature-time curve

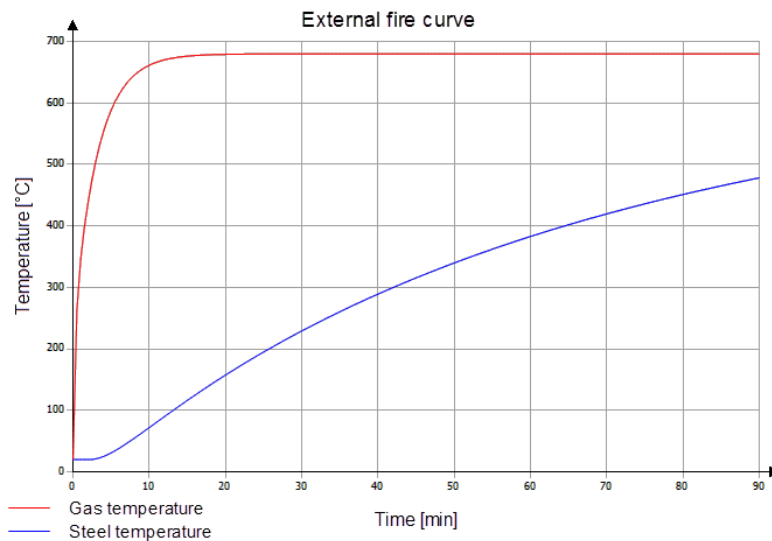


Figure 3.6: External fire curve

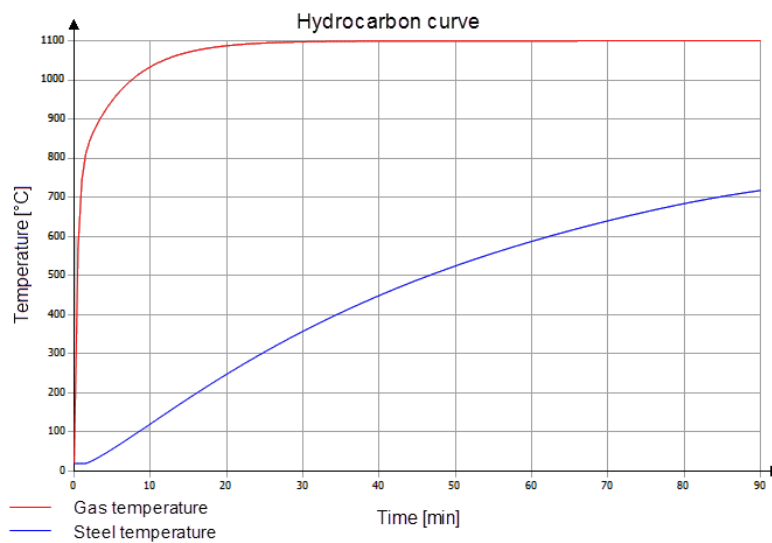


Figure 3.7: Hydrocarbon curve

3.1.5 Other

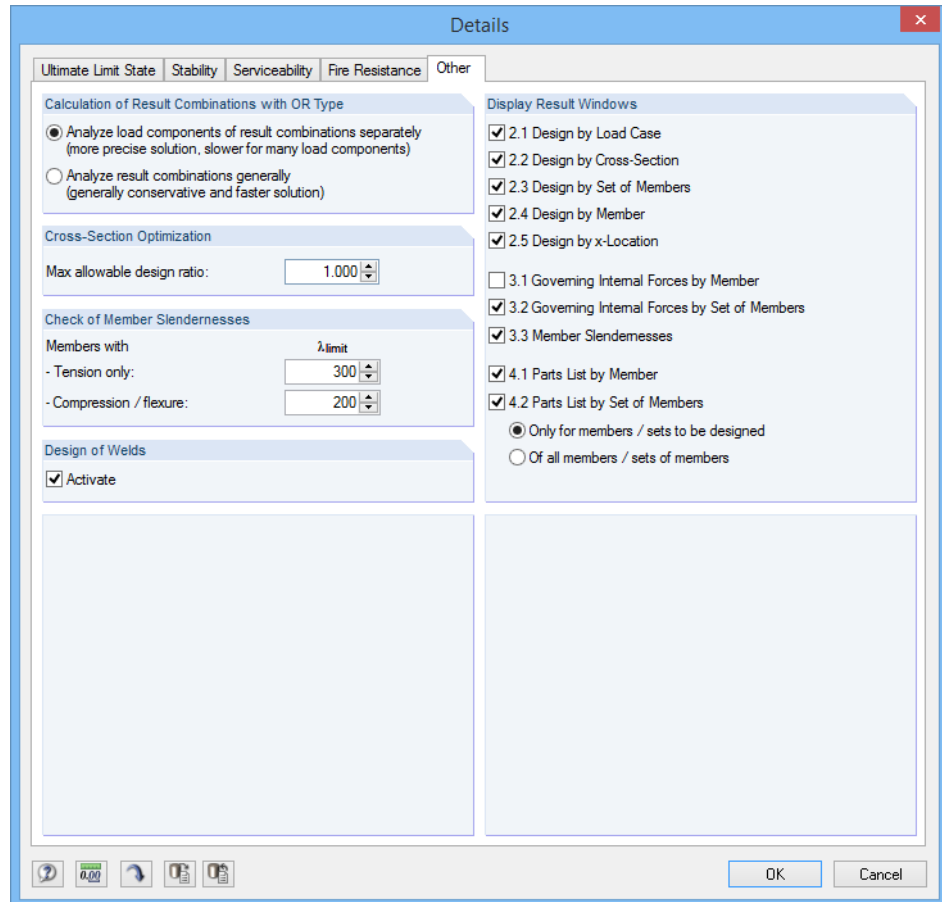


Figure 3.8: Dialog box *Details*, tab *Other*

Cross-Section Optimization

By default, the optimization is targeted on the maximum allowable design ratio of 100 %. If necessary, you can set a different design ratio in this text box.

Check of Member Slendernesses

In both text boxes, you can specify the limit value λ_{limit} in order to define member slendernesses. You can enter specifications separately for members with tension forces only and for members with flexure and compression.

The limit values are compared to the real member slendernesses in Window 3.3. This result window is available after the calculation (see [Chapter 4.8, page 59](#)), if you have selected the corresponding check box in the *Display Result Tables* dialog box section.

Design of Welds

If you select this check box, the weld designs are performed in the course of the analysis. The program performs the typical designs according to EN 1993-1-8 [10]. After the calculation, you can find the results under the cross-section designs (see also the following DLUBAL Blog: www.dlubal.com/blog/3675).

Display Result Windows

In this dialog section, you can select which result windows including the parts list should be displayed. These windows are described in [Chapter 4 - Results](#).

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

3.2 Start Calculation

Calculation

In all input windows of the RF-STEEL EC3 add-on module, you can start the calculation using the [Calculation] button.

RF-STEEL EC3 searches for the results of the load cases, load combinations and result combinations to be designed. If they cannot be found, the program starts the RFEM calculation to determine the design-relevant internal forces.

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

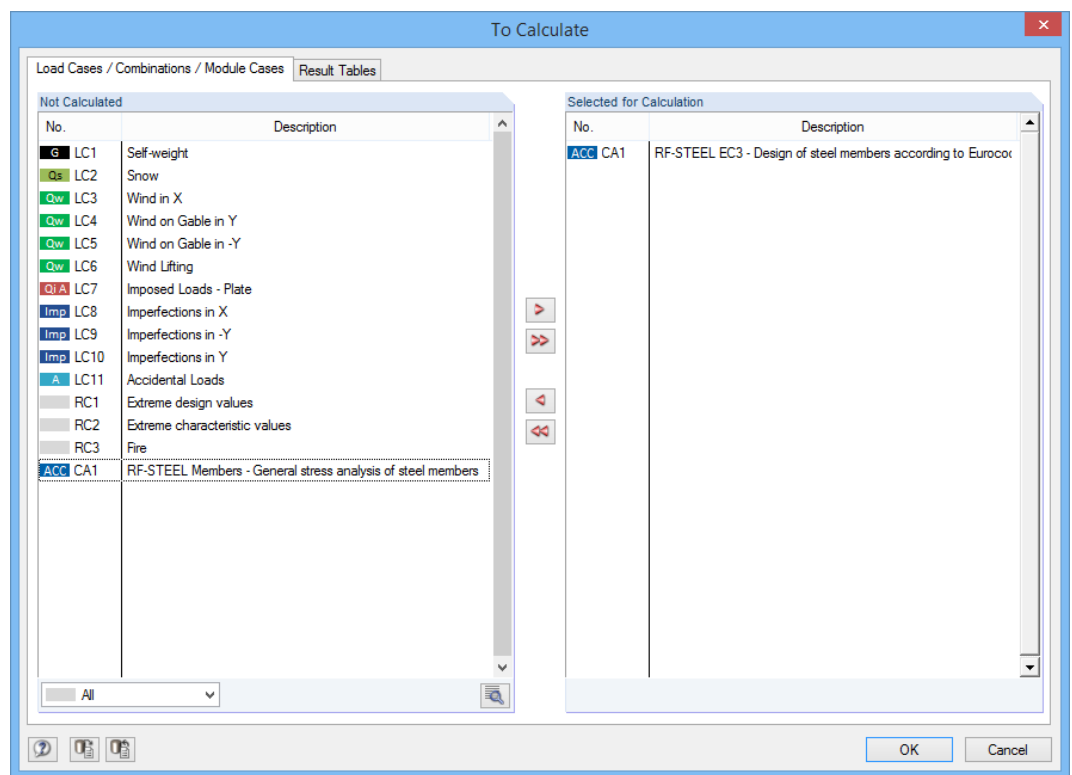
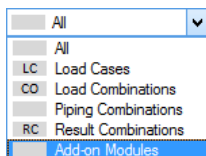


Figure 3.9: Dialog box *To Calculate*



If the RF-STEEL EC3 design cases are missing in the *Not Calculated* section, select *All* or *Add-on Modules* in the drop-down list below the section.

Use the button to transfer the selected RF-STEEL EC3 cases to the list on the right. Click [OK] to start the calculation.

To calculate a design case directly, use the list in the toolbar. Select the RF-STEEL EC3 design case in the toolbar list and click [Show Results].



Figure 3.10: Direct calculation of a RF-STEEL EC3 case in RFEM

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

4 Results

Chapter 4 - Results describes the different results windows one by one. The evaluation of and checking results is described in Chapter 5 - Results Evaluation, page 63 ff.

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

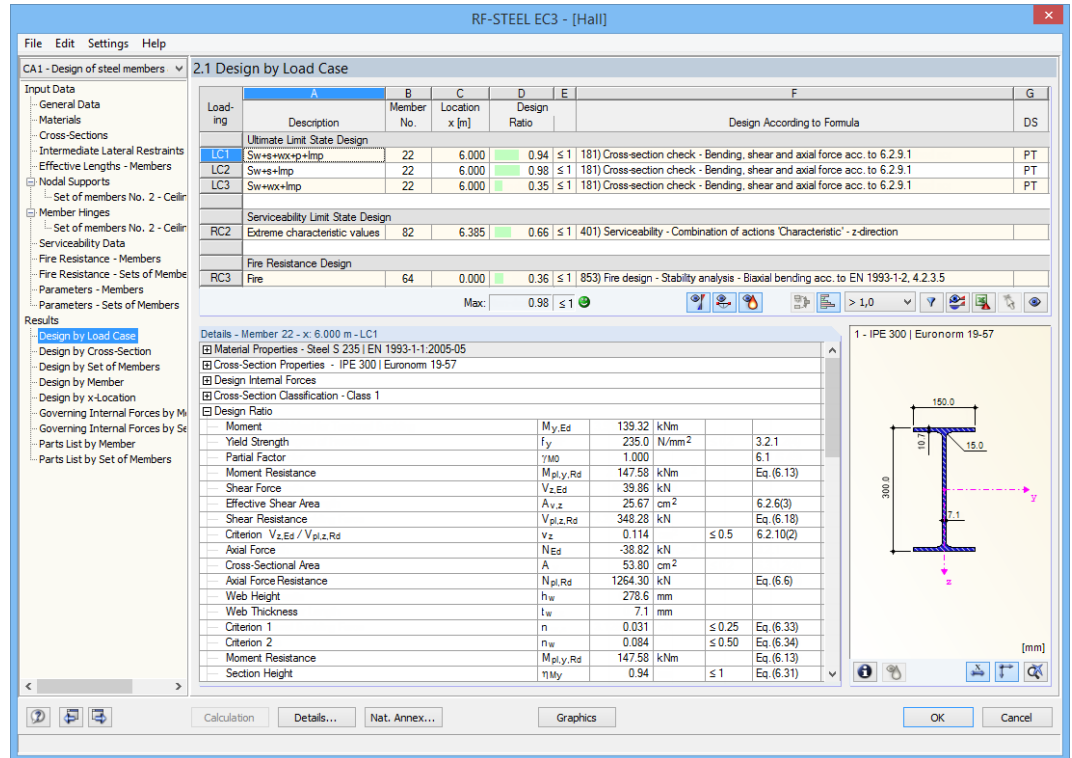


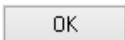
Figure 4.1: Result window with designs and intermediate values

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

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



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



[OK] saves the results. RF-STEEL EC3 is closed and you return to the main program.

4.1 Design by Load Case



The upper part of the window shows a summary of the governing designs, sorted by load case, load combination, and result combination. Furthermore, the list is divided in ultimate limit state design, serviceability limit state design, and fire resistance design.

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

2.1 Design by Load Case

Load-ing	A	B	C	D	E	F	G
	Description	Member No.	Location x [m]	Design Ratio		Design According to Formula	DS
Ultimate Limit State Design							
LC1	Sw+s+wx+p+Imp	22	6.000	0.94	≤ 1	181) Cross-section check - Bending, shear and axial force acc. to 6.2.9.1	PT
LC2	Sw+s+Imp	22	6.000	0.98	≤ 1	181) Cross-section check - Bending, shear and axial force acc. to 6.2.9.1	PT
LC4	Sw+p+Imp	12	3.000	0.93	≤ 1	361) Stability analysis - Bending and compression acc. to 6.3.3, Method 2	PT
Serviceability Limit State Design							
RC2	Extreme characteristic values	82	6.385	0.66	≤ 1	401) Serviceability - Combination of actions 'Characteristic' - z-direction	SC
Fire Resistance Design							
RC3	Fire	64	0.000	0.36	≤ 1	853) Fire design - Stability analysis - Biaxial bending acc. to EN 1993-1-2, 4.2.3.5	
				Max:	0.98	≤ 1	

Details - Member 12 - x: 3.000 m - LC4

□ Cross-Section Classification - Class 1

□ Design Ratio

— Elastic Critical Load for Torsional Buckling	$N_{cr,T}$	2719.35	kN		
— Modulus of Elasticity	E	210000.0	N/mm ²		
— Moment of Inertia	I_y	8360.00	cm ⁴		
— Effective Member Length	$L_{cr,y}$	23.130	m		
— Elastic Flexural Buckling Force	$N_{cr,y}$	323.87	kN	≤ $N_{cr,T}$	
— Cross-Sectional Area	A	53.80	cm ²		
— Yield Strength	f_y	235.0	N/mm ²		3.2.1
— Slenderness	λ_{y}	1.976		> 0.2	6.3.1.2(4)
— Buckling Curve	KL_y	a			Tab. 6.2
— Imperfection Factor	α_y	0.210			Tab. 6.1
— Auxiliary Factor	Φ_y	2.638			6.3.1.2(1)
— Reduction Factor	χ_y	0.228			Eq. (6.49)
— Section Height	h	300.0	mm		
— Section Width	b	150.0	mm		
— Criterion	h/b	2.00		≤ 2	Tab. 6.5
— Buckling Curve	KL_{LT}	b			Tab. 6.5
— Imperfection Factor	α_{LT}	0.340			Tab. 6.3
— Shear Modulus	G	80769.2	N/mm ²		
— Effective Length Factor	k_z	1.000			
— Effective Length Factor	k_w	1.000			

1 - IPE 300 | Euronorm 19-87

Figure 4.2: Window 2.1 Design by Load Case

Description

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

Member No.

This column shows the number of the member with the maximum design ratio of the designed actions.

Location x

This column shows the respective x-location where the member's maximum design 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 (*Calculation Parameters* dialog box, *Global Calculation Parameters* tab in RFEM)
- Extreme values of internal forces

Design Ratio

Max: 0.98 ≤ 1

Columns D and E show the design conditions according to [1], [2] and [3]. The length of colored bars represents the respective design ratios.

Design According to Formula

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

DS

The last column provides information on the respective check-relevant design situations (*DS*): *PT* or *AC* for the ultimate limit state or one of three design situations for serviceability (*CH*, *FR*, *QP*) according to the specifications in the 1.1 *General Data* window (see [Figure 4.7, page 10](#)).

4.2 Design by Cross-Section

2.2 Design by Cross-Section

Section No.	Member No.	Location x [m]	Load-ing	Design Ratio	E	F
IPE 300 Euronorm 19-57						
1	1	0.000	LC4	0.07	≤ 1	102) Cross-section check - Compression acc. to 6.2.4
	1	0.000	LC4	0.06	≤ 1	121) Cross-section check - Shearforce in z-axis acc. to 6.2.6
	1	0.000	LC1	0.00	≤ 1	126) Cross-section check - Shear buckling acc. to 6.2.6(6)
	1	4.000	LC2	0.22	≤ 1	181) Cross-section check - Bending, shear and axial force acc. to 6.2.9.1
	1	6.000	LC4	0.53	≤ 1	221) Cross-section check - Biaxial bending, shear and axial force acc. to 6.2.10 and 6.2.9
	1	2.000	LC4	0.65	≤ 1	361) Stability analysis - Bending about y-axis and compression acc. to 6.3.3, Method 2
	1	0.000	LC4	0.80	≤ 1	364) Stability analysis - Bending and compression acc. to 6.3.3, Method 2
HE A 120 Euronorm 53-62						

Max: 0.80 ≤ 1

Details - Member 1 - x: 4.000 m - LC2

- Cross-Section Properties - IPE 300 | Euronorm 19-57
- Design Internal Forces
- Cross-Section Classification - Class 1
- Design Ratio

Moment	$M_{y,Ed}$	32.73	kNm		
Yield Strength	f_y	235.0	N/mm ²		3.2.1
Partial Factor	γ_{M0}	1.000			6.1
Moment Resistance	$M_{pl,y,Rd}$	147.58	kNm		Eq. (6.13)
Shear Force	$V_{z,Ed}$	19.07	kN		
Effective Shear Area	$A_{v,z}$	25.67	cm ²		6.2.6(3)
Shear Force Resistance	$V_{pl,z,Rd}$	348.28	kN		Eq. (6.18)
Criterion $V_{z,Ed} / V_{pl,z,Rd}$	v_z	0.055		≤ 0.5	6.2.10(2)
Axial Force	N_{Ed}	-27.88	kN		
Cross-Sectional Area	A	53.80	cm ²		
Axial Force Resistance	$N_{pl,Rd}$	1264.30	kN		Eq. (6.6)
Web Height	h_w	278.6	mm		
Web Thickness	t_w	7.1	mm		
Criterion 1	n	0.022		≤ 0.25	Eq. (6.33)
Criterion 2	n_w	0.060		≤ 0.50	Eq. (6.34)
Moment Resistance	$M_{pl,y,Rd}$	147.58	kNm		Eq. (6.13)
Moment Component	η_{My}	0.22		≤ 1	Eq. (6.31)
Design Ratio	η	0.22		≤ 1	Eq. (6.36*)

Figure 4.3: Window 2.2 Design by Cross-Section

In this result window, the maximum design ratios of all members and actions selected for design are listed by cross-section. The results are sorted by cross-section design and stability analysis as well as serviceability limit state design and fire resistance design.

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

4.3 Design by Set of Members

2.3 Design by Set of Members

Set No.	A Member No.	B Location x [m]	C Load-ing	D Design Ratio	E	F Design According to Formula
2	Set of Members 2 (Member No. 13-15)					
	14	0.000	LC2	0.03	≤ 1	102) Cross-section check - Compression acc. to 6.2.4
	15	0.000	LC1	0.04	≤ 1	121) Cross-section check - Shear force in z-axis acc. to 6.2.6
	14	0.000	LC2	0.01	≤ 1	131) Cross-section check - Torsion acc. to 6.2.7
	13	2.677	LC2	0.06	≤ 1	132) Cross-section check - Torsion and shear force acc. to 6.2.7(9)
	15	5.647	LC1	0.42	≤ 1	181) Cross-section check - Bending, shear and axial force acc. to 6.2.9.1
	15	5.647	LC2	0.44	≤ 1	186) Cross-section check - Bending, shear, torsion and axial force acc. to 6.2.9.1
	14	2.121	LC1	0.00	≤ 1	206) Cross-section check - Bending about z-axis, shear, torsion and axial force acc. to 6.2.9.1
	15	5.647	LC1	0.45	≤ 1	221) Cross-section check - Biaxial bending, shear and axial force acc. to 6.2.10 and 6.2.9
	15	5.960	LC2	0.48	≤ 1	226) Cross-section check - Biaxial bending, shear, torsion and axial force acc. to 6.2.10 and 6.2.9

Max: 0.98 ≤ 1

Details - Member 15 - x: 5.960 m - LC2

Design Ratio	Value	Unit	Reference
Moment	65.46	kNm	
Plastic Section Modulus	628.00	cm ³	
Yield Strength	235.0	N/mm ²	3.2.1
Partial Factor	1.000		6.1
Moment Resistance	147.58	kNm	Eq. (6.13)
Shear Force	0.89	kN	
Effective Shear Area	25.67	cm ²	6.2.6(3)
Shear Force Resistance	348.28	kN	Eq. (6.18)
Torsional Moment	0.01	kNm	
Torsional Constant	20.20	cm ⁴	
Thickness	7.1	mm	
Shear Stress	0.2	N/mm ²	
Shear Force Resistance	348.06	kN	6.2.7(9)
Criterion $V_{y,Ed} / V_{pl,y,Rd}$	0.003		≤ 0.5 6.2.10(2)
Axial Force	-28.73	kN	
Cross-Sectional Area	53.80	cm ²	
Axial Force Resistance	1264.30	kN	Eq. (6.6)
Web Height	278.6	mm	
Web Thickness	7.1	mm	
Criterion 1	0.023		≤ 0.25 Eq. (6.33)
Criterion 2	0.062		≤ 0.50 Eq. (6.34)

Figure 4.4: Window 2.3 Design by Set of Members

This result window is displayed if you have selected at least one set of members for the design. The window lists the maximum design 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 stress ratio for the respective design criteria.

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

4.4 Design by Member

2.4 Design by Member

Member No.	A Location x [m]	B Load-ing	C Design Ratio	D	E Design According to Formula
1	Cross-section No. 1 - IPE 300 Euronorm 19-57				
	0.000	LC2	0.03 ≤ 1	102)	Cross-section check - Compression acc. to 6.2.4
	0.000	LC2	0.05 ≤ 1	121)	Cross-section check - Shear force in z-axis acc. to 6.2.6
	0.000	LC1	0.00 ≤ 1	126)	Cross-section check - Shear buckling acc. to 6.2.6(6)
	4.000	LC2	0.22 ≤ 1	181)	Cross-section check - Bending, shear and axial force acc. to 6.2.9.1
	6.000	LC2	0.50 ≤ 1	221)	Cross-section check - Biaxial bending, shear and axial force acc. to 6.2.10 and 6.2.9
	0.000	RC2	0.00 ≤ 1	400)	Serviceability - Negligible deformations
	4.500	RC2	0.94 ≤ 1	401)	Serviceability - Combination of actions 'Characteristic' - z-direction
	Cross-section No. 1 - IPE 300 Euronorm 19-57				

Max: 0.98 ≤ 1

Details - Member 1 - x: 4.500 m - RC2

Material Properties - Steel S 235 | EN 1993-1-1:2005-05

Modulus of Elasticity	E	210000.0	N/mm ²	
Shear Modulus	G	80769.2	N/mm ²	
Yield Strength	f _y	235.0	N/mm ²	3.2.1
Ultimate Tensile Strength	f _u	360.0	N/mm ²	3.2.1

Cross-Section Properties - IPE 300 | Euronorm 19-57

Deflections

Direction x	w _x	-0.3	mm	
Direction y	w _y	-7.3	mm	
Direction z	w _z	-18.7	mm	

Design Ratio

Deflection	w _{Q,inst,z}	-18.7	mm	
Refer. Length	l	6.000	m	
Limit Value Criterion	l / w _{Q,inst,lim}	300.00		
Limit Value of Deflection	w _{Q,inst,limit,z}	20.0	mm	
Design Ratio	η	0.94	≤ 1	Gl. (40)

Figure 4.5: Window 2.4 Design by Member

This result window shows the maximum design ratios for individual designs sorted by member number. The columns are described in detail in Chapter 4.1 on page 52.

4.5 Design by x-Location

2.5 Design by x-Location

Member No.	A Location x [m]	B Load-ing	C Design Ratio	D	E Design According to Formula
5	Cross-section No. 17 - IPE 300 Euronorm 19-57				
	0.000	LC2	0.05 ≤ 1	102)	Cross-section check - Compression acc. to 6.2.4
	0.000	LC1	0.12 ≤ 1	121)	Cross-section check - Shear force in z-axis acc. to 6.2.6
	0.000	LC1	0.00 ≤ 1	126)	Cross-section check - Shear buckling acc. to 6.2.6(6)
	0.000	LC3	0.33 ≤ 1	181)	Cross-section check - Bending, shear and axial force acc. to 6.2.9.1
	0.000	LC1	0.81 ≤ 1	221)	Cross-section check - Biaxial bending, shear and axial force acc. to 6.2.10 and 6.2.9
	0.000	LC2	0.05 ≤ 1	102)	Cross-section check - Compression acc. to 6.2.4
	0.000	LC1	0.12 ≤ 1	121)	Cross-section check - Shear force in z-axis acc. to 6.2.6
	0.000	LC1	0.00 ≤ 1	126)	Cross-section check - Shear buckling acc. to 6.2.6(6)
	0.000	LC1	0.46 ≤ 1	181)	Cross-section check - Bending, shear and axial force acc. to 6.2.9.1

Max: 0.98 ≤ 1

Details - Member 22 - x: 0.000 m - LC1

Cross-Section Classification - Class 1

Design Ratio

Moment	$M_{y,Ed}$	111.83	kNm		
Plastic Section Modulus	$W_{pl,y}$	628.00	cm ³		
Yield Strength	f_y	235.0	N/mm ²		3.2.1
Partial Factor	γ_{M0}	1.000			6.1
Moment Resistance	$M_{pl,y,Rd}$	147.58	kNm		Eq. (6.13)
Shear Force	$V_{z,Ed}$	43.41	kN		
Effective Shear Area		25.67	cm ²		6.2.6(3)
Shear Force Resistance	$V_{pl,z,Rd}$	348.28	kN		Eq. (6.18)
Criterion $V_{z,Ed} / V_{pl,z,Rd}$	v_z	0.125		≤ 0.5	6.2.10(2)
Axial Force	N_{Ed}	-60.43	kN		
Cross-Sectional Area	A	53.80	cm ²		
Axial Force Resistance	$N_{pl,Rd}$	1264.30	kN		Eq. (6.6)
Web Height	h_w	278.6	mm		
Web Thickness	t_w	7.1	mm		
Criterion 1	n	0.048		≤ 0.25	Eq. (6.33)
Moment	n_w	0.130		≤ 0.50	Eq. (6.34)
Shear Force	$M_{z,Ed}$	0.07	kNm		
Plastic Section Modulus	$W_{pl,z}$	125.22	cm ³		
Moment Resistance	$M_{pl,z,Rd}$	29.43	kNm		Eq. (6.13)
Shear Force	$V_{y,Ed}$	0.01	kN		

Figure 4.6: Window 2.5 Design by x-Location

This result window lists the maxima for each member at all locations x, resulting from the division points 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 (*Calculation Parameters* dialog box, *Global Calculation Parameters* tab in RFEM)
- Extreme values of internal forces

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Load- ing	C			D			E			F			G			H			I Design According to Formula
			N _{Ed}	V _{y,Ed}	V _{z,Ed}	T _{Ed}	M _{y,Ed}	M _{z,Ed}	N _{Ed}	V _{y,Ed}	V _{z,Ed}	T _{Ed}	M _{y,Ed}	M _{z,Ed}	N _{Ed}	V _{y,Ed}	V _{z,Ed}	T _{Ed}	M _{y,Ed}	M _{z,Ed}	
1 Cross-section No. 1 - IPE 300 Euronorm 19-57																					
	0.000	LK2	-43.98	0.04	-19.07	0.00	43.84	0.12	102) Cross-section check - Compression acc. to 6.2.4												
	0.000	LK2	-43.98	0.04	-19.07	0.00	43.84	0.12	121) Cross-section check - Shear force in z-axis acc. to 6.2.6												
	0.000	LK1	-42.61	0.04	-15.08	0.00	34.58	0.12	126) Cross-section check - Shear buckling acc. to 6.2.6(6)												
	4.000	LK2	-27.88	0.04	-19.07	0.00	-32.73	-0.05	181) Cross-section check - Bending, shear and axial force acc.												
	6.000	LK2	-19.82	0.04	-19.07	0.00	-70.79	-0.14	221) Cross-section check - Biaxial bending, shear and axial force												
	0.000	EK2	0.00	0.00	0.00	0.00	0.00	0.00	400) Serviceability - Negligible deformations												
	4.500	EK2	0.00	0.00	0.00	0.00	0.00	0.00	401) Serviceability - Combination of actions 'Characteristic' - z-d												
2 Cross-section No. 1 - IPE 300 Euronorm 19-57																					
	0.000	LK2	-44.35	0.00	20.68	0.00	-50.30	-0.04	102) Cross-section check - Compression acc. to 6.2.4												
	0.000	LK1	-43.25	0.00	21.47	0.00	-52.70	-0.05	121) Cross-section check - Shear force in z-axis acc. to 6.2.6												
	0.000	LK1	-43.25	0.00	21.47	0.00	-52.70	-0.05	126) Cross-section check - Shear buckling acc. to 6.2.6(6)												
	6.000	LK2	-20.19	0.00	20.68	0.00	74.06	-0.04	181) Cross-section check - Bending, shear and axial force acc.												
	0.000	EK3	-90.47	0.04	20.95	0.00	-50.04	0.04	602) Fire design - Cross-section check - Compression acc. to E												
	3.900	EK3	-45.60	0.04	20.95	0.00	31.66	-0.11	621) Fire design - Cross-section check - Shear force												
	0.000	EK3	-90.47	0.04	20.95	0.00	-50.04	0.04	681) Fire design - Cross-section check - Bending, shear and axial force												
	6.000	EK3	-21.43	0.04	20.95	0.00	75.65	-0.18	721) Fire design - Cross-section check - Biaxial bending, shear and axial force												
11 Cross-section No. 1 - IPE 300 Euronorm 19-57																					
	0.000	LK2	-52.82	0.02	-30.65	0.00	65.91	0.07	102) Cross-section check - Compression acc. to 6.2.4												
	0.000	LK2	-52.82	0.02	-30.65	0.00	65.91	0.07	121) Cross-section check - Shear force in z-axis acc. to 6.2.6												
	0.000	LK1	-50.76	0.02	-23.01	0.00	48.02	0.07	126) Cross-section check - Shear buckling acc. to 6.2.6(6)												
	6.000	LK2	-31.21	0.02	-30.65	0.00	-118.19	-0.05	181) Cross-section check - Bending, shear and axial force acc.												
	0.000	LK2	-52.82	0.02	-30.65	0.00	65.91	0.07	221) Cross-section check - Biaxial bending, shear and axial force												
12 Cross-section No. 1 - IPE 300 Euronorm 19-57																					
	0.000	LK2	-56.98	-0.01	20.93	0.00	0.00	0.00	102) Cross-section check - Compression acc. to 6.2.4												
	0.000	LK1	-54.76	-0.01	21.68	0.00	0.00	0.00	121) Cross-section check - Shear force in z-axis acc. to 6.2.6												
	0.000	LK1	-54.76	-0.01	21.68	0.00	0.00	0.00	126) Cross-section check - Shear buckling acc. to 6.2.6(6)												
	4.500	LK2	-40.77	-0.01	20.93	0.00	95.72	0.06	181) Cross-section check - Bending, shear and axial force acc.												
	6.000	LK2	-35.37	-0.01	20.93	0.00	126.74	0.07	221) Cross-section check - Biaxial bending, shear and axial force												
13 Cross-section No. 3 - IPE 400 Euronorm 19-57 ... 2 - IPE 300 Euronorm 19-57																					

Figure 4.7: Window 3.1 Governing Internal Forces by Member

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

Location x

This column shows the respective x-location where the member's maximum design ratio occurs.

Loading

This column shows numbers of the load case as well as load and result combination with internal forces resulting in the maximum design ratio.

Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing the maximum ratios in the respective cross-section designs, stability analyses, serviceability limit state designs, and fire resistance designs.

Design According to Formula

The last column shows the design types and equations used for the designs according to [1], [2], or [3].

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

Set No.	A		B		C		D		E		F		G		H		I
	Location x [m]	Load-ing	N _{Ed}	V _{y,Ed}	V _{z,Ed}	T _{Ed}	M _{y,Ed}	M _{z,Ed}	Design According to Formula								
1	Set of Members 1 (Member No. 51.52)																
	3.000	LK3	-1.21	-0.03	-0.07	0.00	-0.01	0.08	100) Negligible internal forces								
	0.000	LK1	-62.98	-0.01	1.15	0.00	-3.53	0.08	102) Cross-section check - Compression acc. to 6.2.4								
	0.000	LK3	-13.29	-0.03	1.51	0.00	-2.20	-0.01	121) Cross-section check - Shear force in z-axis acc. to 6.2.6								
	0.000	LK1	-62.98	-0.01	1.15	0.00	-3.53	0.08	126) Cross-section check - Shear buckling acc. to 6.2.6(6)								
	0.000	LK2	-15.86	-0.02	0.12	-0.01	-0.46	0.02	131) Cross-section check - Torsion acc. to 6.2.7								
	0.000	LK1	-15.44	-0.02	1.41	-0.01	-2.19	0.02	132) Cross-section check - Torsion and shear force acc. to 6.2.7(9)								
	0.000	LK1	-62.98	-0.01	1.15	0.00	-3.53	0.08	181) Cross-section check - Bending, shear and axial force acc.								
	0.000	LK1	-15.44	-0.02	1.41	-0.01	-2.19	0.02	186) Cross-section check - Bending, shear, torsion and axial force								
2	3.000	LK2	-32.67	-0.01	0.14	0.00	-0.38	0.10	221) Cross-section check - Biaxial bending, shear and axial force								
	0.000	LK2	-44.75	-0.01	0.14	0.00	-0.81	0.08	301) Stability analysis - Flexural buckling about y-axis acc. to 6.3.1.1								
	0.000	LK2	-44.75	-0.01	0.14	0.00	-0.81	0.08	321) Stability analysis - Torsional buckling acc. to 6.3.1.4 and 6.3.1.5								
	0.000	LK1	-62.98	-0.01	1.15	0.00	-3.53	0.08	364) Stability analysis - Bending and compression acc. to 6.3.3, Met								
	0.000	LK1	-62.98	-0.01	1.15	0.00	-3.53	0.08	371) Stability analysis - Bending and compression acc. to 6.3.4, Ger								
2	Set of Members 2 (Member No. 13-15)																
	0.000	LK2	-32.83	-0.03	21.38	-0.02	-45.46	0.02	102) Cross-section check - Compression acc. to 6.2.4								
	0.000	LK1	-27.90	0.07	14.85	-0.01	18.57	0.10	121) Cross-section check - Shear force in z-axis acc. to 6.2.6								
	0.000	LK2	-32.83	-0.03	21.38	-0.02	-45.46	0.02	131) Cross-section check - Torsion acc. to 6.2.7								
	2.677	LK2	-32.84	-0.03	21.71	-0.02	-49.90	0.02	132) Cross-section check - Torsion and shear force acc. to 6.2.7(9)								
	5.647	LK1	-26.58	0.07	-0.19	-0.01	61.69	-0.32	141) Cross-section check - Bending and shear force acc. to 6.2.5 and 6.2.6								
	5.647	LK2	-28.81	0.09	0.01	-0.01	65.57	-0.39	181) Cross-section check - Bending, shear and axial force acc. to 6.2.7								
	2.121	LK1	-30.32	-0.06	16.13	-0.02	7.26	0.14	186) Cross-section check - Bending, shear, torsion and axial force acc. to 6.2.7								
	5.647	LK1	-26.58	0.07	-0.19	-0.01	61.69	-0.32	221) Cross-section check - Biaxial bending, shear and axial force acc. to 6.2.5 and 6.2.6								
	5.960	LK2	-28.68	0.09	-1.42	-0.01	65.16	-0.43	226) Cross-section check - Biaxial bending, shear, torsion and axial force acc. to 6.2.5 and 6.2.6								
	5.647	LK2	-28.81	0.09	0.01	-0.01	65.57	-0.39	271) Cross-section check - Axial stress and torsion - Elastic design								

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

For each set of members, this window shows the internal forces that result in the maximum ratios of the design.

4.8 Members Slendernesses

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C Major Axis y		E λ_y [-]	F Minor Axis z		H λ_z [-]
			k _y [-]	i _y [mm]		k _z [-]	i _z [mm]	
1	Compression / Flexure	6.000	4.029	124.7	193.926	1.000	33.5	179.070
2	Compression / Flexure	6.000	1.000	124.7	48.133	1.000	33.5	179.070
3	Compression / Flexure	6.000	4.040	124.7	194.456	0.543	33.5	97.235
4	Compression / Flexure	6.000	3.855	124.7	185.551	0.518	33.5	92.758
5	Compression / Flexure	6.000	1.000	124.7	48.133	0.400	33.5	71.628
6	Compression / Flexure	6.000	1.000	124.7	48.133	0.600	33.5	107.442
7	Compression / Flexure	3.000	1.000	124.7	24.066	1.000	33.5	89.535
8	Compression / Flexure	3.000	1.000	124.7	24.066	1.000	33.5	89.535
9	Compression / Flexure	6.274	1.000	124.7	50.330	1.000	33.5	187.247
11	Compression / Flexure	3.000	1.000	82.8	36.224	1.000	49.9	60.112
12	Compression / Flexure	3.000	1.000	82.8	36.224	1.000	49.9	60.112
13	Compression / Flexure	6.274	1.000	57.3	109.541	1.000	35.2	178.246
14	Compression / Flexure	6.274	1.000	57.3	109.544	1.000	35.2	178.251
15	Compression / Flexure	6.274	1.000	57.3	109.544	1.000	35.2	178.251
16	Compression / Flexure	6.274	1.000	57.3	109.541	1.000	35.2	178.246
17	Compression / Flexure	6.546	1.000	65.6	99.778	1.000	39.8	164.286
18	Compression / Flexure	7.094	1.000	65.6	108.131	1.000	39.8	178.040
21	Compression / Flexure	6.546	1.000	65.6	99.778	1.000	39.8	164.286
22	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
23	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
24	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
25	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
26	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
27	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
28	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
31	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
32	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
33	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
34	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515
35	Compression / Flexure	5.000	1.000	31.0	161.515	1.000	31.0	161.515

Members with compression / flexure:
 Max λ_y 194.456 ≤ 200
 Max λ_z 187.247 ≤ 200

Figure 4.9: Window 3.3 Member slendernesses

Details...

This result window appears only if you have selected the respective check box in the *Details* dialog box, tab *Other* (see Figure 4.8, page 49).

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They are determined in compliance with the load type. At the end of the list, there is a comparison with the limit values that have been defined in the *Details* dialog box, tab *Other* (see Figure 4.8, page 49).

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

This window is only of an informative character. It provides no stability analysis of slendernesses.

4.9 Parts List by Member

Finally, there is a summary of all cross-sections 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 [m ²]	F Volume [m ³]	G Unit Weight [kg/m]	H Weight [kg]	Total Weight [t]
1	1 - IPE 300 Euronorm 19-57	6	6.00	36.00	41.72	0.19	42.23	253.40	1.520
2	2 - IPE 300 Euronorm 19-57 ... 3 - IPE 400	8	3.01	24.09	31.63	0.17	54.28	163.46	1.308
3	2 - IPE 300 Euronorm 19-57 ... 3 - IPE 400	8	3.26	26.10	30.25	0.14	42.23	137.78	1.102
4	2 - IPE 300 Euronorm 19-57 ... 3 - IPE 400	8	6.27	50.19	58.17	0.27	42.23	264.97	2.120
5	1 - IPE 300 Euronorm 19-57	4	3.00	12.00	13.91	0.06	42.23	126.70	0.507
6	10 - HE A 140 Euronorm 53-62	3	3.00	9.00	7.15	0.03	24.65	73.95	0.222
7	10 - HE A 140 Euronorm 53-62	2	3.55	7.09	5.63	0.02	24.65	87.41	0.175
8	10 - HE A 140 Euronorm 53-62	1	4.09	4.09	3.25	0.01	24.65	100.91	0.101
9	15 - HE A 200 Euronorm 53-62	4	3.00	12.00	13.68	0.06	42.23	126.70	0.507
10	6 - HE A 160 Euronorm 53-62	2	3.00	6.00	5.44	0.02	30.46	91.37	0.183
11	6 - HE A 160 Euronorm 53-62	2	3.55	7.09	6.43	0.03	30.46	108.00	0.216
12	6 - HE A 160 Euronorm 53-62	1	4.09	4.09	3.71	0.02	30.46	124.70	0.125
13	16 - Rectangle 200/200	1	3.00	3.00	2.40	0.12	314.00	942.00	0.942
14	7 - HE A 140 Euronorm 53-62	4	6.27	25.10	19.93	0.08	24.65	154.64	0.619
15	9 - IPE 360 Euronorm 19-57	8	6.25	50.00	67.65	0.36	57.07	356.68	2.853
16	6 - HE A 160 Euronorm 53-62	2	6.55	13.09	11.86	0.05	30.46	199.38	0.319
17	6 - HE A 160 Euronorm 53-62	1	7.09	7.09	6.43	0.03	30.46	216.07	0.216
18	12 - QRO 80x4 EN 10210-2:2006	25	5.00	125.00	39.13	0.15	9.42	47.10	1.178
19	13 - RD 24 Macsteel	4	7.81	31.24	2.36	0.01	3.55	27.71	0.111
20	13 - RD 24 Macsteel	8	8.02	64.18	4.84	0.03	3.55	28.47	0.228
Sum		102		516.46	375.55	1.86			14.630

Figure 4.10: Window 4.1 Parts List by Member

Details...

By default, this list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box, tab *Other* (see [Figure 4.8, page 49](#)).

Part No.

The program automatically assigns part numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column shows 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 relative to the total length. The surface area is determined from the *Surface* of the cross-sections that can be seen in Windows 1.3 and 2.1 to 2.5 in the cross-section properties (see [Figure 4.19, page 19](#)).

Volume

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

Unit Weight

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

Weight

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

Total Weight

The final column indicates the total weight of each part.

Sum

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

4.10 Parts List by Set of Members

4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m ²]	F Volume [m ³]	G Unit Weight [kg/m]	H Weight [kg]	Total Weight [t]
1	Set of Members 1	1	6.00	6.00	6.84	0.03	42.23	253.40	0.253
2	Set of Members 2	1	12.55	12.55	15.01	0.07	45.12	566.22	0.566
3	Set of Members 3	1	12.55	12.55	15.01	0.07	45.12	566.22	0.566
4	Set of Members 4	1	6.55	6.55	5.20	0.02	24.65	161.35	0.161
5	Set of Members 5	1	7.09	7.09	5.63	0.02	24.65	174.86	0.175
Sum		5		44.74	47.68	0.22			1.722

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

The last result window is displayed if you have selected at least one set of members for design. The window summarizes an entire structural group in a parts list, for example horizontal beams.

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

5 Results Evaluation

You can evaluate the design results in different ways. For this, you can use the buttons below the upper table.

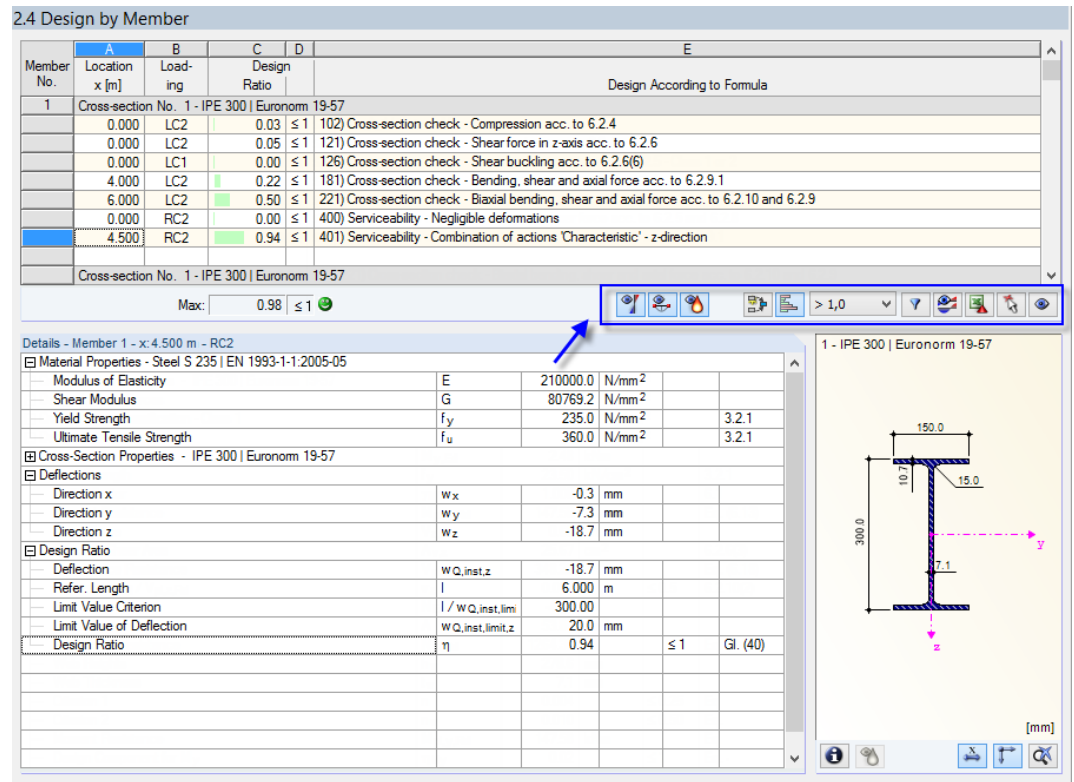


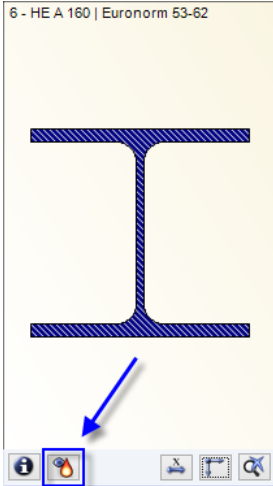
Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Display or hide the results of ULS design
	Serviceability Limit State Designs	Display or hide the results of SLS design
	Fire Protection Designs	Display or hide the results of the fire protection design
	Result Combination	Create a new result combination from the governing load cases and load combinations
	Color Bars	Display or hide the colored relation scales in the result windows
	Filter Parameter	Describe the criterion for filtering results in tables: ratios greater than 1, maximum or user-defined limit
	Apply Filter	Display only the rows with applied filter parameter (design ratio > 1, maximum, user-defined limit)
	Result Diagrams	Open the <i>Result Diagram on Member</i> window → Chapter 5.2, page 67
	Excel Export	Export the table to MS Excel / OpenOffice → Chapter 7.4.3, page 78

	Member Selection	Select a member graphically to display its results in the table
	View Mode	Jump to the RFEM work window to change the view

Table 5.1: Buttons in the result windows 2.1 to 2.5



When evaluating the fire resistance design, you can check the steel temperature development graphically: Click the button shown on the left (below the cross-section graphic in the result window) to open the *Temperature Curves* diagram, as shown in [Figure 5.5](#) to [Figure 5.7](#), page 48.

5.1 Results in RFEM Model

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

RFEM background graphic and view mode

The RFEM work window in the background is useful when you want to find the position of a particular member in the model: The member selected in the result window of RF-STEEL EC3 is highlighted in the selection color in the background graphic. An arrow indicates also the member's x-location that is displayed in the active table row.

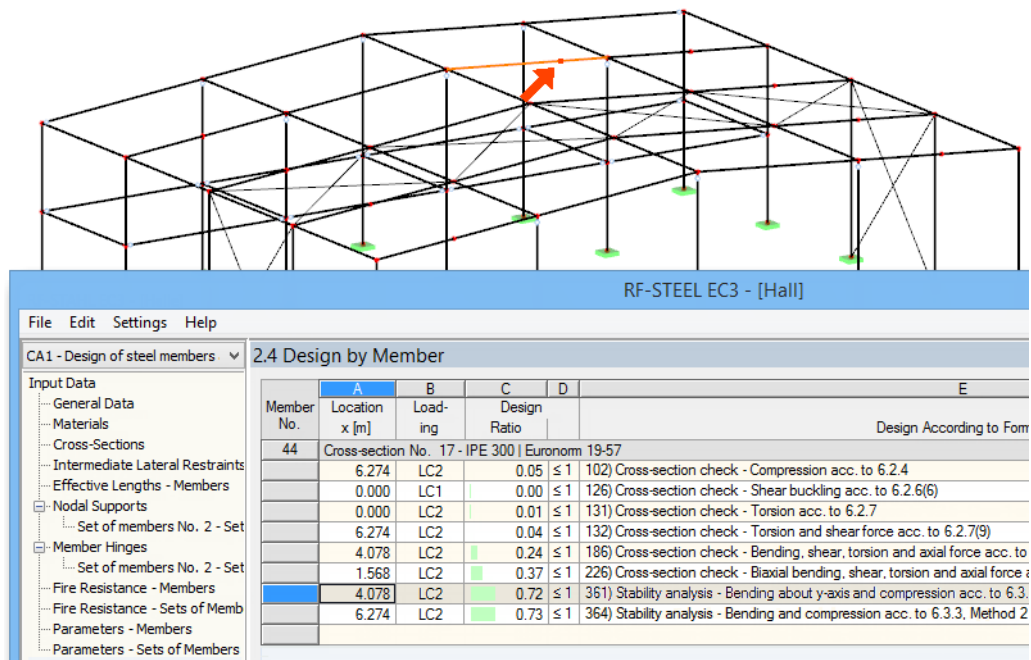
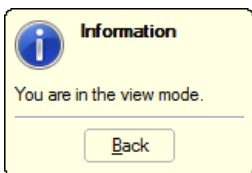


Figure 5.2: Indication of member and current *Location x* in the RFEM model



In case you cannot improve the display by moving the RF-STEEL EC3 module window, click [Jump to Graphic] to activate the *View mode*: The program hides the 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 RF-STEEL EC3 add-on module.

RFEM work window

Graphics

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

In the *Results* navigator, you can specify which design ratios of the serviceability and ultimate limit state or fire resistance design you want to display in the graphic.

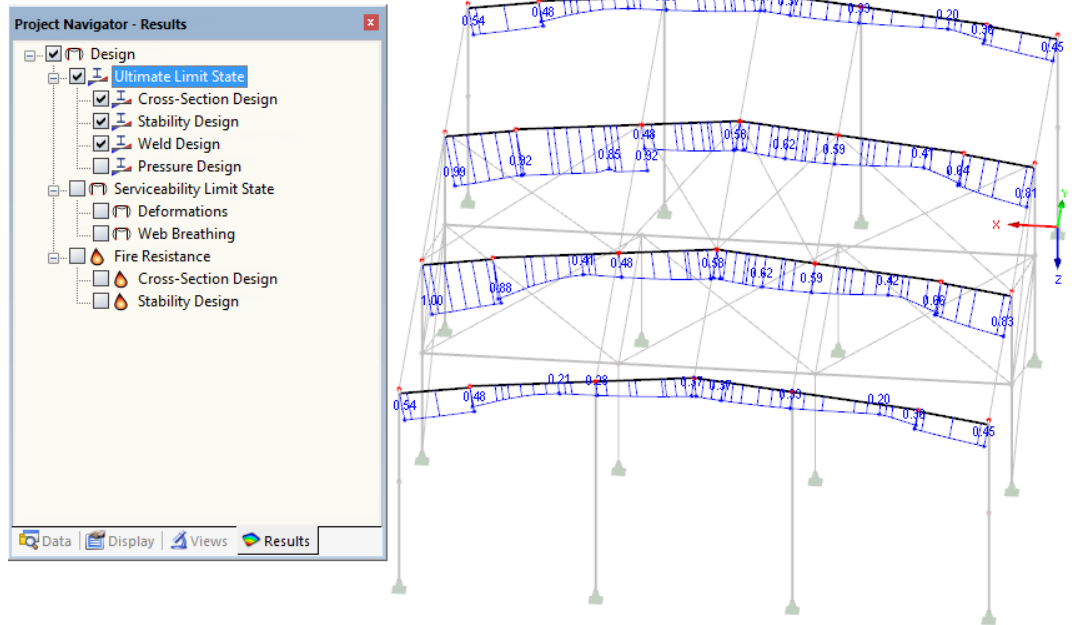


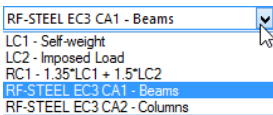
Figure 5.3: Results navigator for *Ultimate limit state*, *Serviceability* and *Fire resistance*



To turn the display of design results on or off, click the [Show Results] button, that you know from the display of internal forces in RFEM. To display the result values, click the [Show Values] button on the right.



The RFEM tables are not relevant for the evaluation of the design results.



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

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

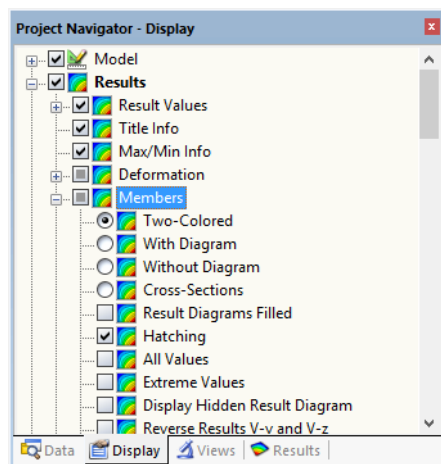


Figure 5.4: Display navigator: *Results* → *Members*

If you select a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color panel becomes available, providing common control functions. These functions are described in detail in the RFEM manual, Chapter 3.4.6.

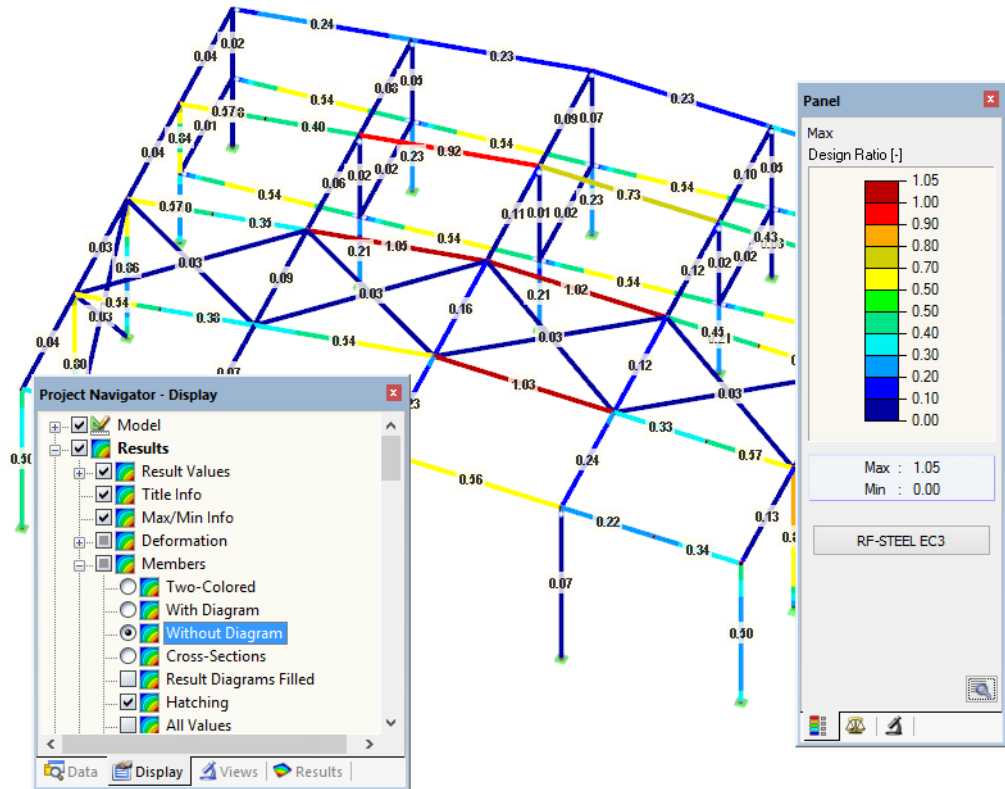


Figure 5.5: Design ratios with display option *Without Diagram*

You can transfer the graphics of design results to the printout report (see [Chapter 6.2, page 71](#)).

RF-STEEL EC3

To return to the add-on module, click [RF-STEEL EC3] in the panel.

5.2 Result Diagrams

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



To do this, select the member (or set of members) in the RF-STEEL EC3 result window by clicking in the table row of the member. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. The button is located below the upper result table (see [Figure 5.1, page 63](#)).

To display the result diagrams, select the command from the menu of RFEM graphic



Results → **Result Diagrams for Selected Members**

or use the corresponding button in the RFEM toolbar.

A window appears, graphically showing the distribution of the results on the member or set of members.

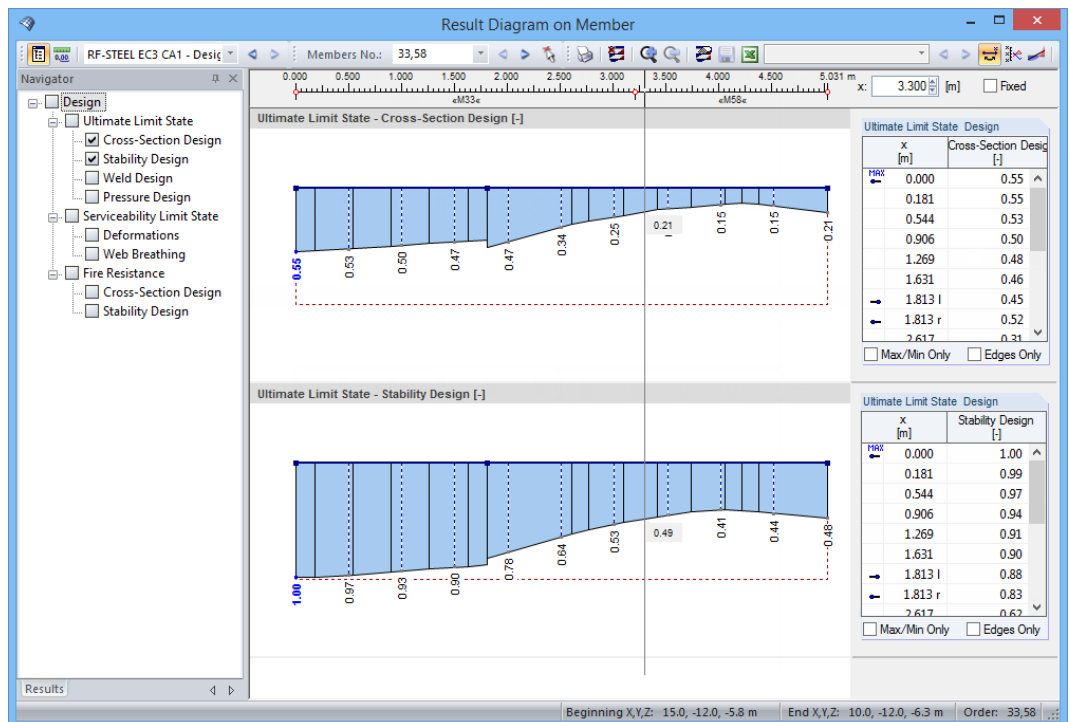
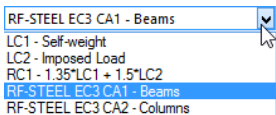


Figure 5.6: Dialog box *Result Diagram on Member*

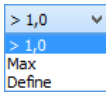
Also here, you can display the designs of ultimate limit state, serviceability and fire resistance using the *Results* navigator.



Use the list in the toolbar above to select the relevant RF-STEEL EC3 design case.

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

5.3 Filter for Results



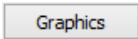
The RF-STEEL EC3 result windows allow you to sort the results by various criteria. In addition, you can use the filter options for the windows (see [Figure 5.1, page 63](#)) to limit the numerical output by design ratios. This function is described in the following article of the DLUBAL Blog: www.dlubal.com/blog/11217

Furthermore, you can use the filter options described in Chapter 9.9 of the RFEM manual to evaluate the results graphically.



For RF-STEEL EC3, you can also use the *Visibility* options to filter the members in order to evaluate them (see RFEM manual, Chapter 9.9.1).

Filtering designs



You can easily use the design ratios as filter criteria in the RFEM work window which can be accessed by clicking [Graphics]. To apply this function, the panel must be displayed. If the panel is not active, you can activate it in the RFEM menu by clicking



View → **Control panel**

or use the toolbar button shown on the left.

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

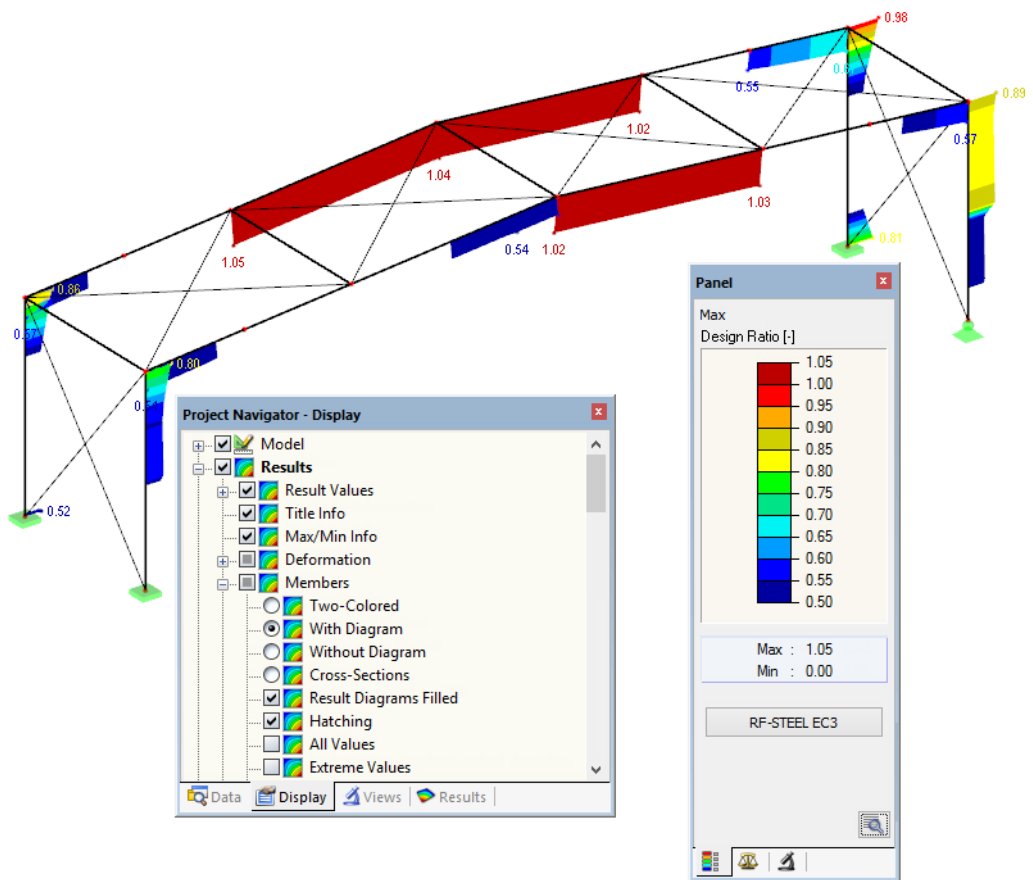


Figure 5.7: Filtering design ratios with adjusted color scale

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

The function *Display hidden result diagram* in the *Display* navigator (**Results** → **Members**) shows all design ratios out of the value spectrum. Those diagrams are represented by dotted lines.

Filtering members



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

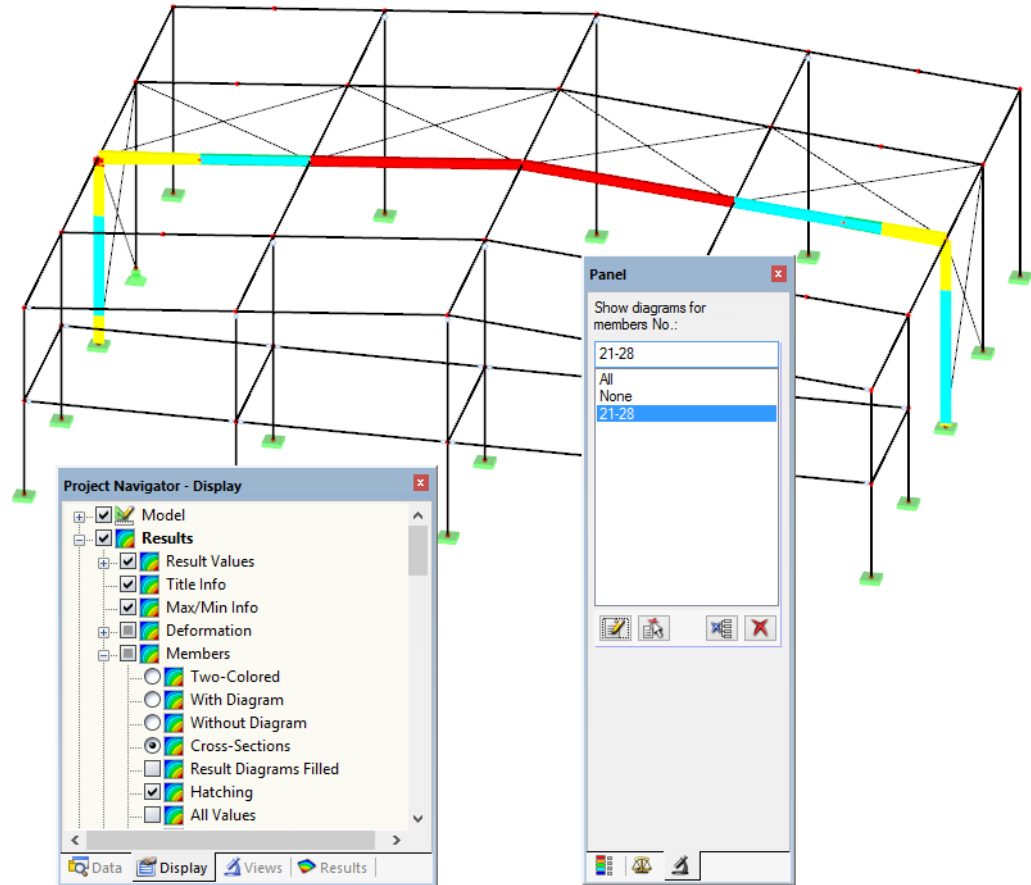


Figure 5.8: Member filter for the design ratios of a hall frame

In contrast to the visibility function, the model will be 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 EC3 results, to which you can add graphics and descriptions. The selection in the printout report determines what data from the design module will be included in the printout.



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

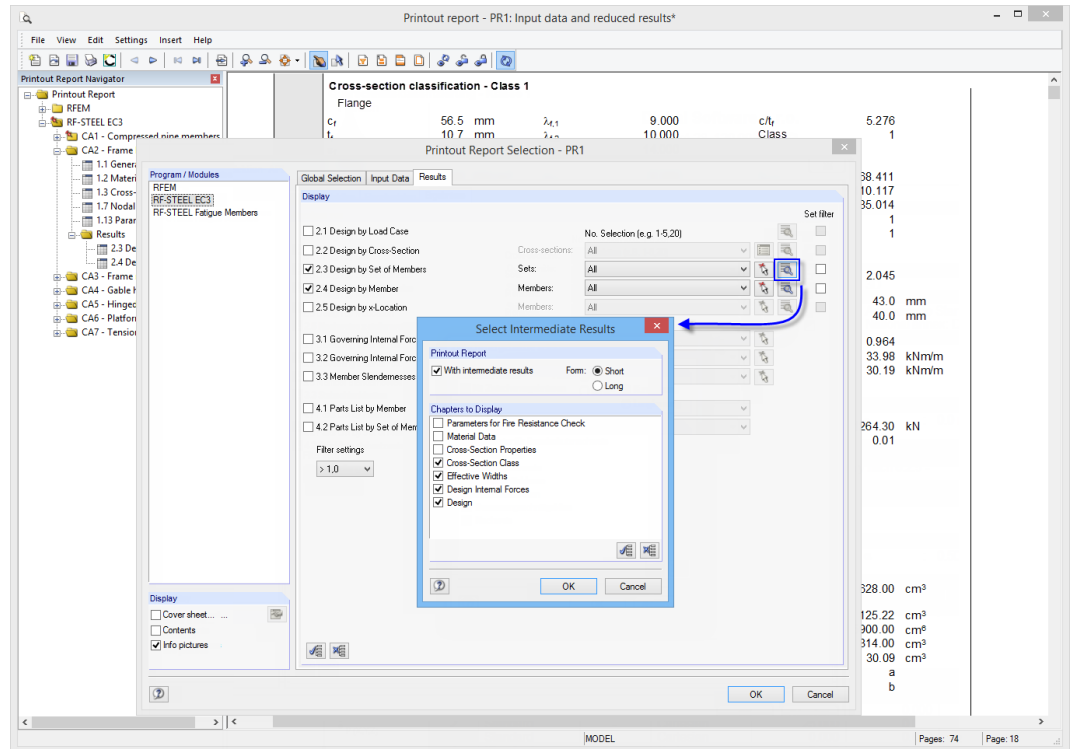


Figure 6.1: Selecting designs and intermediate results in the printout report



Click the [Details] button to specify if you want to include intermediate results in the printout report and document them as a list in a *Short form* (compact representation) or a *Long form* (list representation).

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

6.2 Graphic Printout

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



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

Designs in 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.

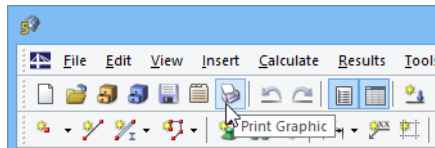


Figure 6.2: *Print Graphic* button in RFEM toolbar

Result Diagrams



Also in the *Result Diagram on Member* dialog box, you can click [Print] to transfer the graphic to the printout report or print it directly.

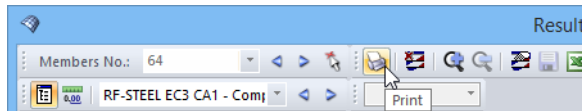


Figure 6.3: *Print* button in the dialog box *Result Diagram on Member*

The following dialog box appears:

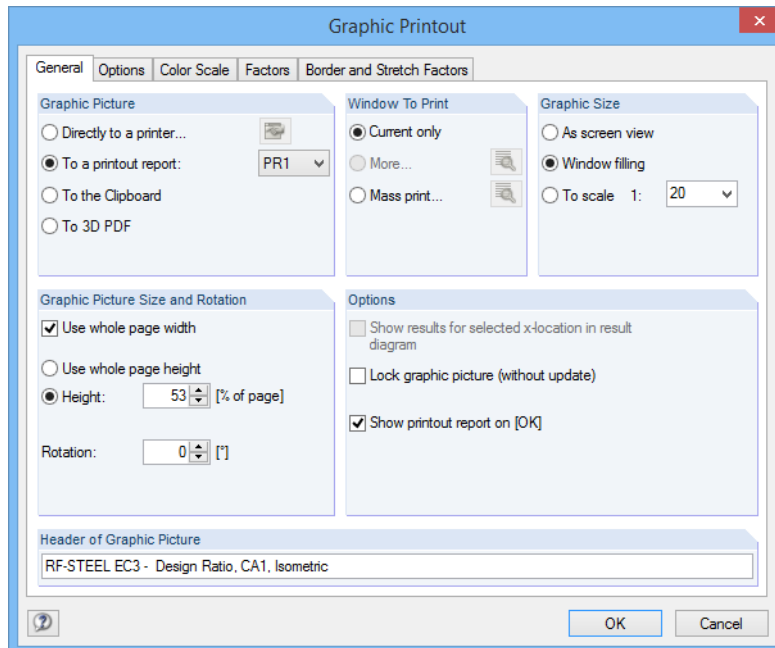


Figure 6.4: Dialog box *Graphic Printout*, tab *General*

The *Graphic Printout* dialog box is described in detail in the RFEM manual, Chapter 10.2. The RFEM manual also describes the *Options* and *Color Scale* tab.

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

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

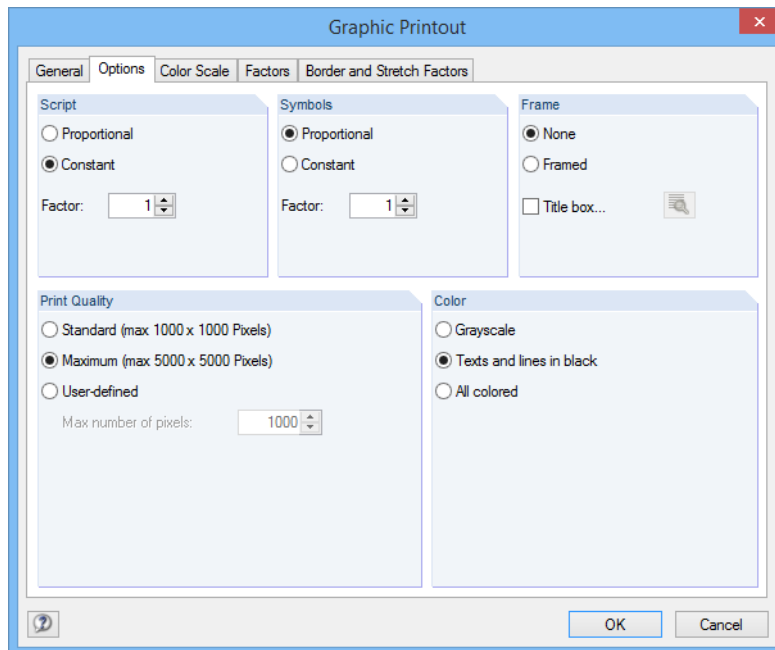
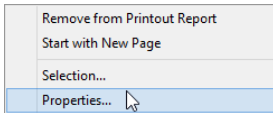


Figure 6.5: Dialog box *Graphic Printout*, tab *Options*

7 General Functions

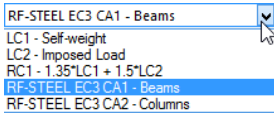
This chapter describes useful menu functions as well as export options for the designs.

7.1 Design Cases

Design cases allow you to group members for a design. In this way, you can combine groups of structural elements or analyze members with particular design specifications (for example modified materials, detail categories, optimization).

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

To calculate a RF-STEEL EC3 design case, you can also use the load case list in the RFEM toolbar.



Create New Design Case

To create a new design case, use the RF-STEEL EC3 menu and click

File → **New Case**.

The following dialog box appears:

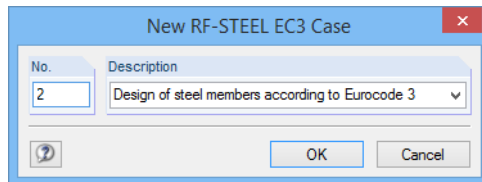


Figure 7.1: Dialog box *New RF-STEEL EC3 Case*

In this dialog box, enter a *Number* (that is still available) of the new design case. The corresponding *Description* makes the selection in the load case list easier.

Click [OK] to open the 1.1 *General Data* window in RF-STEEL EC3 where you can enter the design data.

Rename Design Case

To change the description of a design case, use the RF-STEEL EC3 menu and click

File → **Rename Case**.

The following dialog box appears:

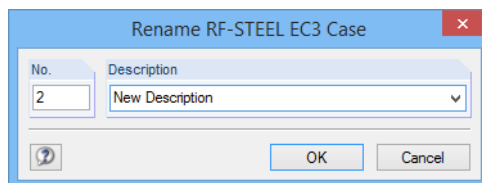


Figure 7.2: Dialog box *Rename RF-STEEL EC3 Case*

In this dialog box, you can specify a different *Description* as well as a different *Number* of the design case.

Copy Design Case

To copy the input data of the current design case, use the RF-STEEL EC3 menu and click

File → **Copy Case**.

The following dialog box appears:

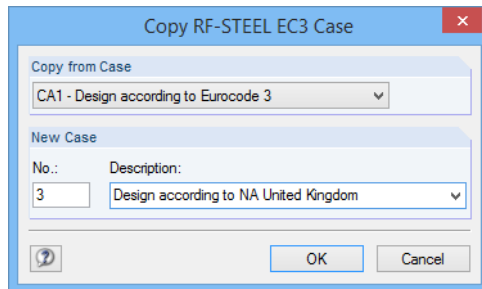


Figure 7.3: Dialog box *Copy RF-STEEL EC3 Case*

Define the *Number* and, if necessary, a *Description* for the new case.

Delete Design Case

To delete design cases, use the RF-STEEL EC3 menu and click

File → **Delete Case**.

The following dialog box appears:

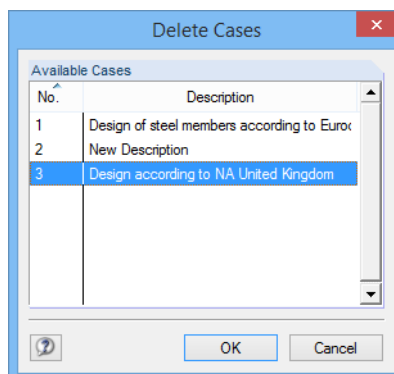
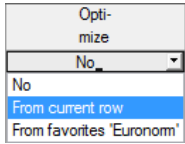


Figure 7.4: Dialog box *Delete Case*

You can select the design case in the list of *Available Cases*. To delete the selected case, click [OK].



7.2 Cross-Section Optimization

The RF-STEEL EC3 design module offers you the option to optimize overloaded or little utilized cross-sections. To do this, open the drop-down list in column D or E in the 1.3 *Cross-Sections* window and select whether the cross-section is to be determined *From current row* or from the user-defined *Favorites* (see [Figure 7.17, page 17](#)). You can also start the optimization in the result windows by using the shortcut menu.

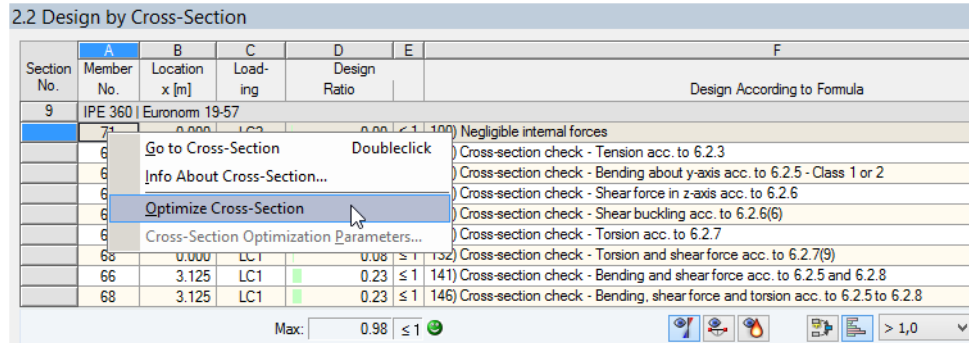


Figure 7.5: Shortcut menu for cross-section optimization

During the optimization process, RF-STEEL EC3 determines the cross-section that fulfills the re-sistance analysis requirements in the most “optimal” way, that is, it comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see [Figure 7.8, page 49](#)). The required cross-section properties are determined with the internal forces from RFEM. If another cross-section proves to be more favorable, this cross-section is used for the design. Then, the graphic in Window 1.3 shows two cross-sections: the original cross-section from RFEM and the optimized one (see [Figure 7.7](#)).

If you select the *Optimization* option for a parametrized cross-section, the following dialog box appears.

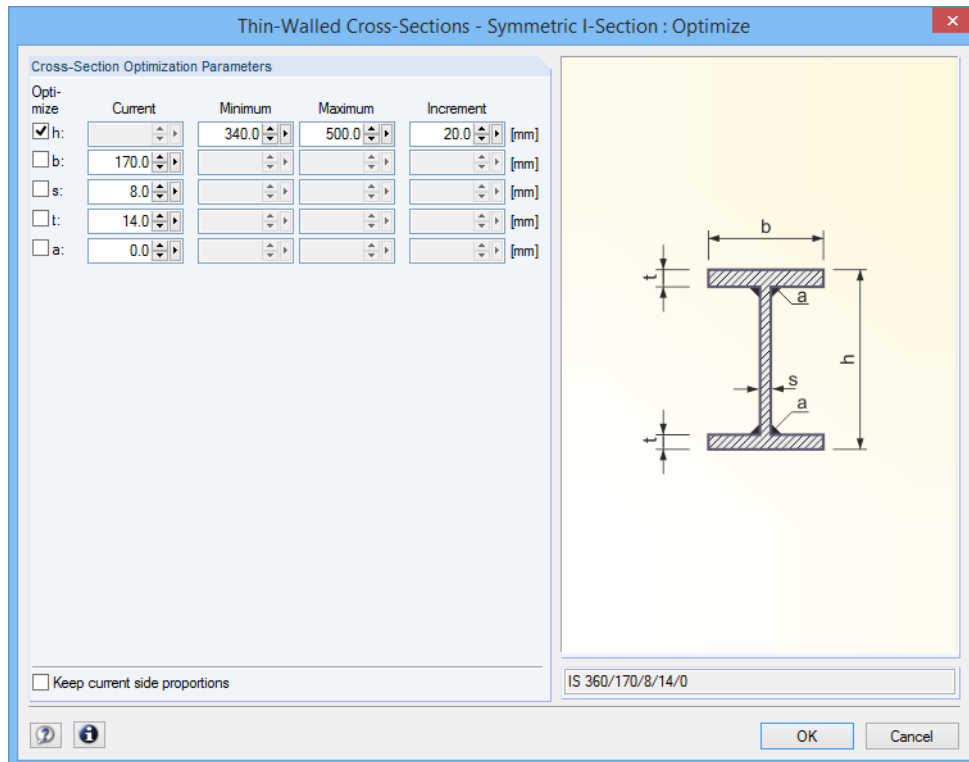


Figure 7.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section: Optimize*

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

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

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



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

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

Edit → Export All Cross-Sections to RFEM.

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

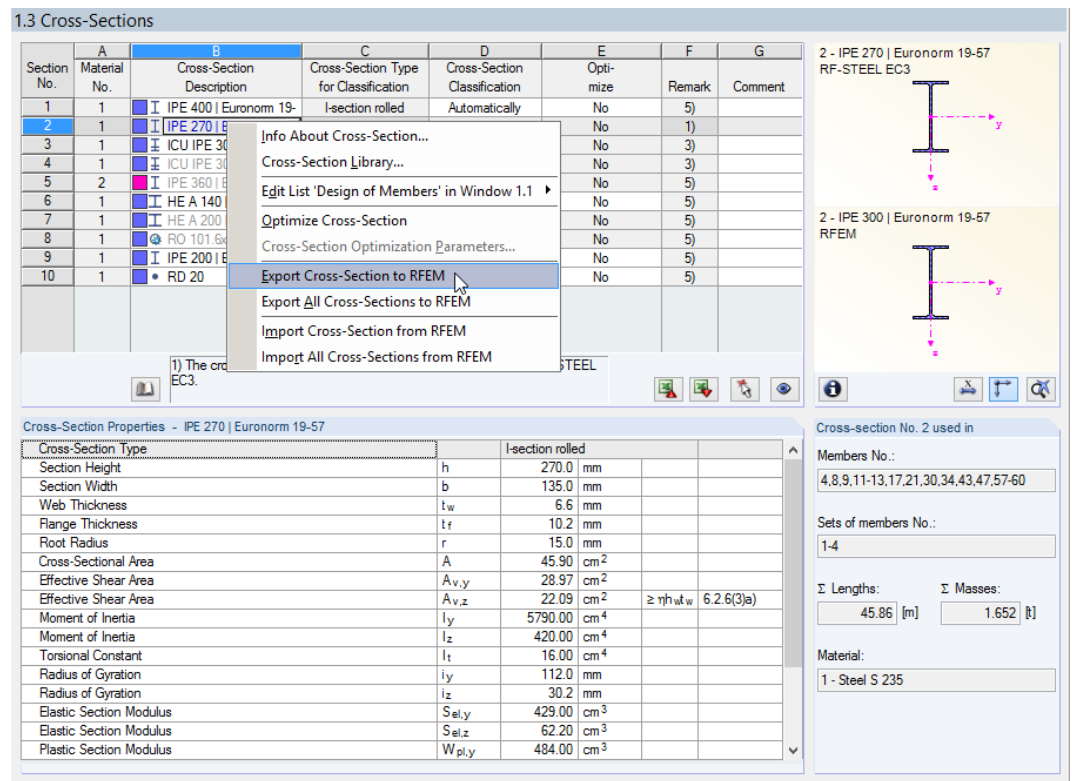


Figure 7.7: Shortcut menu in window 1.3 *Cross-Sections*

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

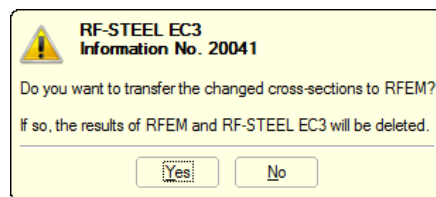


Figure 7.8: Query before transfer of modified cross-sections to RFEM

Calculation

By confirming the query and starting the [Calculation] in the RF-STEEL EC3 module, the RFEM internal forces as well as the design ratios will be determined in a single calculation run.

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



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

7.3 Units and Decimal Places

Units and decimal places for RFEM and the add-on modules are managed in one dialog box. In RF-STEEL EC3, you can use the menu to adjust the units. To open the corresponding dialog box, click

Settings → **Units and Decimal Places**.

The following dialog box appears which you already know from RFEM. RF-STEEL EC3 is preset in the *Program / Module* list.

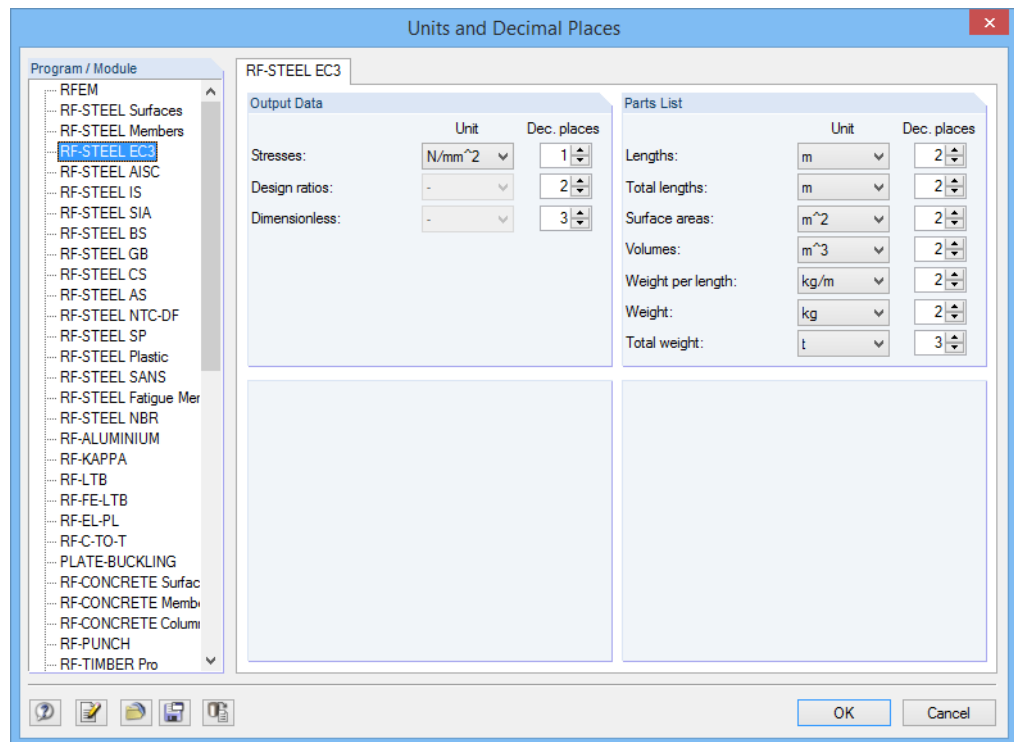


Figure 7.9: Dialog box *Units and Decimal Places*



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

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM

When you have adjusted the materials in RF-STEEL EC3 for design, you can export the modified materials to RFEM in a similar way as you export cross-sections: Open the 1.2 *Materials* window and then use the menu

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

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



Figure 7.10: Shortcut menu in the window 1.2 *Materials*

Calculation

Before the modified materials are transferred to RFEM, a security query appears as to whether the results of RFEM should be deleted. By confirming the query and starting the [Calculation] in the RF-STEEL EC3 module, the RFEM internal forces as well as the design ratios will be determined in a single calculation run.

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

7.4.2 Exporting Effective Lengths to RFEM

When you have adjusted the effective lengths in RF-STEEL EC3 for design, you can export the modified effective lengths to RFEM in a similar way as you export cross-sections: Go to the 1.5 *Effective Lengths - Members* window and click

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

or use the corresponding option in the shortcut menu of Window 1.5.

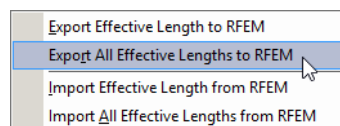


Figure 7.11: Shortcut menu of window 1.5 *Effective Lengths - Members*

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

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

7.4.3 Exporting Results

The RF-STEEL EC3 results can also be used by other programs.

Clipboard

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

Printout Report

You can print the data of RF-STEEL EC3 into the global printout report (see [Chapter 7.2, page 71](#)) to export them subsequently. Then, in the printout report, click

File → **Export to RTF**.

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

Excel / OpenOffice

RF-STEEL EC3 provides a function for directly exporting data to MS Excel, OpenOffice.org Calc, or the CSV file format. To open the corresponding dialog box, click

File → **Export Tables**.

The following export dialog box appears:

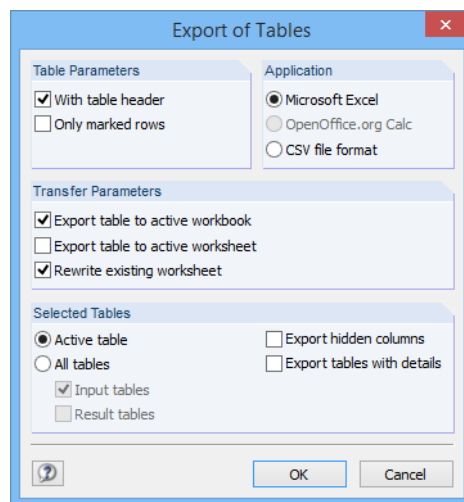


Figure 7.12: Dialog box *Export of Tables*

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

Section No.	Member No.	Location x [m]	Load-ing	Design Ratio	
1	39	0.000	LC1	0.07 ≤ 1	102) Cross-section check - Compression acc. to 6.2.4
2	40	0.000	LC2	0.18 ≤ 1	121) Cross-section check - Shear force in z-axis acc. to 6.2.6
3	1	0.000	LC1	0.00 ≤ 1	126) Cross-section check - Shear buckling acc. to 6.2.6(6)
4	22	6.000	LC2	0.98 ≤ 1	181) Cross-section check - Bending, shear and axial force acc. to 6.2.9.1
5	12	6.000	LC2	0.89 ≤ 1	221) Cross-section check - Biaxial bending, shear and axial force acc. to 6.2.10 and 6.2.9
6	21	2.000	LC1	0.05 ≤ 1	301) Stability analysis - Flexural buckling about y-axis acc. to 6.3.1.1 and 6.3.1.2(4)
7	12	0.000	LC2	0.22 ≤ 1	302) Stability analysis - Flexural buckling about y-axis acc. to 6.3.1.1 and 6.3.1.2
8	40	0.900	LC3	0.03 ≤ 1	311) Stability analysis - Flexural buckling about z-axis acc. to 6.3.1.1 and 6.3.1.2(4)
9	12	0.000	LC2	0.06 ≤ 1	321) Stability analysis - Torsional buckling acc. to 6.3.1.4 and 6.3.1.2(4)
10	12	3.000	LC2	0.86 ≤ 1	361) Stability analysis - Bending about y-axis and compression acc. to 6.3.3, Method 2
11	40	0.000	LC2	0.98 ≤ 1	364) Stability analysis - Bending and compression acc. to 6.3.3, Method 2

Figure 7.13: Results in Excel

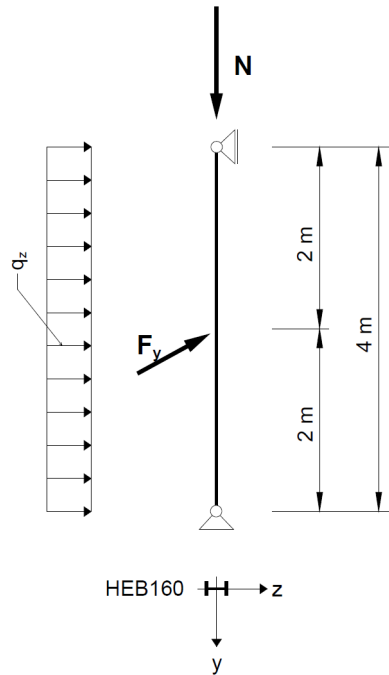
8 Examples

8.1 Stability

In the following example, we perform stability analyses for flexural buckling and lateral-torsional buckling for a column with double-bending, considering the interaction conditions.

Design Values

System and loads



Design values of static loads

$$N_d = 300 \text{ kN}$$

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

$$F_{y,d} = 7.5 \text{ kN}$$

Figure 8.1: System and design loads (γ times)

Internal forces according to linear static analysis

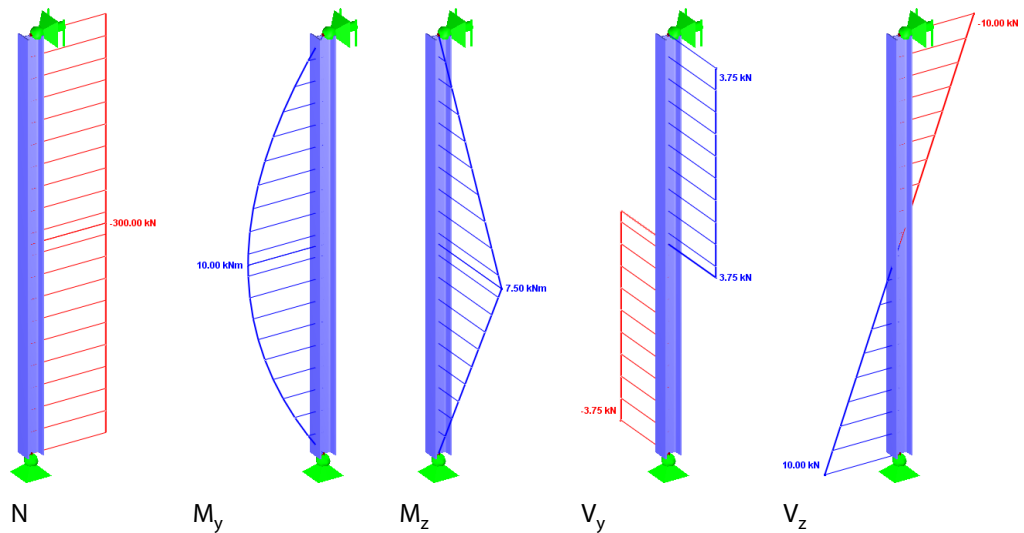


Figure 8.2: Internal forces

Design locations (governing x-location)

The design is performed for all x-locations (see [Chapter 4.5](#)) of the equivalent member. The governing location is $x = 2.00$ m. RFEM determines the following internal forces:

$$N = -300.00 \text{ kN} \quad M_y = 10.00 \text{ kNm} \quad M_z = 7.50 \text{ kNm} \quad V_y = 3.75 \text{ kN} \quad V_z = 0.00 \text{ kN}$$

Cross-Section Properties HE-B 160, S 235

Property	Symbol	Value	Unit
Cross-Sectional Area	A	54.30	cm ²
Moment of Inertia	I_y	2,490.00	cm ⁴
Moment of Inertia	I_z	889.00	cm ⁴
Radius of Gyration	i_y	6.78	cm
Radius of Gyration	i_z	4.05	cm
Polar Radius of Gyration	i_p	7.90	cm
Polar Radius of Gyration	$i_{p,M}$	41.90	cm
Section Weight	G	42.63	kg/m
Torsional Constant	I_T	31.40	cm ⁴
Warping Constant of Cross-Section	I_ω	47,940.00	cm ⁶
Elastic Section Modulus	W_y	311.00	cm ³
Elastic Section Modulus	W_z	111.00	cm ³
Plastic Section Modulus	$W_{pl,y}$	354.00	cm ³
Plastic Section Modulus	$W_{pl,z}$	169.96	cm ³
Buckling Curve	BC_y	b	
Buckling Curve	BC_z	c	

Table 8.1: Cross-Section Properties HE-B 160, S 235

Flexural buckling about the minor axis (\perp to z-z axis)

$$N_{cr,z} = \frac{21,000 \cdot 889.00 \cdot \pi^2}{400.00^2} = 1,151.60 \text{ kN}$$

$$\bar{\lambda}_z = \sqrt{\frac{A \cdot f_y}{N_{cr,z}}} = \sqrt{\frac{54.30 \cdot 23.5}{1,151.60}} = 1.053$$

$$\bar{\lambda}_z = 1.053 > 0.2 \quad \rightarrow \text{Design for flexural buckling must be performed.}$$

$$\text{Cross-sectional geometry } \frac{h}{b} = 1.00 \leq 1.2 \quad \text{Structural steel S 235} \quad t \leq 100 \text{ mm}$$

[1], Table 6.2, row 3, column 4: Buckling curve c

$$\Rightarrow \alpha_z = 0.49 \quad (\text{Table 6.1})$$

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

$$\chi_z = \frac{1}{1.263 + \sqrt{1.263^2 - 1.053^2}} = 0.510$$

$$\frac{N_{Ed}}{\chi_z \cdot A \cdot f_y / \gamma_{M1}} = \frac{300}{0.510 \cdot 54.30 \cdot 23.5 / 1.0} = 0.461$$

Result values of RF-STEEL EC3 calculation

Moment of Inertia	I_z	889.00	cm ⁴		
Effective Member Length	$L_{cr,z}$	4.000	m		
Elastic Flexural Buckling Force	$N_{cr,z}$	1,151.60	kN		
Slenderness	λ_z	1.053		> 0.2	6.3.1.2(4)
Buckling Curve	BC_z	c			Tab. 6.2
Imperfection Factor	α_z	0.490			Tab. 6.1
Auxiliary Factor	Φ_z	1.263			6.3.1.2(1)
Reduction Factor	χ_z	0.510			Eq. (6.49)

Table 8.2: Result values of RF-STEEL EC3 calculation

Flexural buckling about the major axis (\perp to y-y axis)

$$N_{cr,y} = \frac{21,000 \cdot 2,490.00 \cdot \pi^2}{400.00^2} = 3,225.51 \text{ kN}$$

$$\bar{\lambda}_y = \sqrt{\frac{A \cdot f_y}{N_{cr,y}}} = \sqrt{\frac{54.30 \cdot 23.5}{3,225.51}} = 0.629$$

$$\bar{\lambda}_y = 0.629 > 0.2 \quad \rightarrow \text{Design for flexural buckling must be performed.}$$

$$\text{Cross-sectional geometry } \frac{h}{b} = 1.00 \leq 1.2 \quad \text{Structural steel S 235} \quad t \leq 100 \text{ mm}$$

[1], Table 6.2, row 3, column 4: Buckling curve b

$$\Rightarrow \alpha_y = 0.34 \quad \text{Table 6.1}$$

$$\Phi = 0.5 \cdot [1 + 0.34 \cdot (0.629 - 0.2) + 0.629^2] = 0.771$$

$$\chi_y = \frac{1}{0.771 + \sqrt{0.771^2 - 0.629^2}} = 0.822$$

$$\frac{N_{Ed}}{\chi_y \cdot A \cdot f_y / \gamma_{M1}} = \frac{300}{0.822 \cdot 54.30 \cdot 23.5 / 1.0} = 0.286$$

Result values of RF-STEEL EC3 calculation

Moment of Inertia	I_y	2,490.00	cm ⁴		
Effective Member Length	$L_{cr,y}$	4.000	m		
Elastic Flexural Buckling Force	$N_{cr,y}$	3,225.51	kN		
Cross-Sectional Area	A	54.30	cm ²		
Yield Strength	f_y	23.50	kN/cm ²		3.2.1
Slenderness	λ_y	0.629		> 0.2	6.3.1.2(4)
Buckling Curve	BC_y	b			Tab. 6.2
Imperfection Factor	α_y	0.340			Tab. 6.1
Auxiliary Factor	Φ_y	0.771			6.3.1.2(1)
Reduction Factor	χ_y	0.822			Eq. (6.49)

Table 8.3: Result values of RF-STEEL EC3 calculation

Lateral-Torsional Buckling

Ideal elastic critical moment

In this example, the elastic critical moment for lateral-torsional buckling is determined according to the Austrian National Annex with assumption of hinged supports free to warp.

The load application point is in the shear center (for transverse loads, you can adjust the application point in the *Details* dialog box, see [Chapter 3.1.2, page 44](#)).

$$M_{cr} = C_1 \cdot \frac{\pi^2 \cdot E \cdot I_z}{L^2} \cdot \sqrt{\frac{I_\omega}{I_z} + \frac{L^2 \cdot G \cdot I_t}{\pi^2 \cdot E \cdot I_z}}$$

$$M_{cr} = 1.13 \cdot \frac{\pi^2 \cdot 21,000 \cdot 889}{400^2} \cdot \sqrt{\frac{47,940}{889} + \frac{400^2 \cdot 8,100 \cdot 31.40}{\pi^2 \cdot 21,000 \cdot 889}} = 215.71 \text{ kNm}$$

The program also shows $M_{cr,0}$, which is determined on the basis of a constant moment distribution.



For the results by x-location, the program also shows the value $M_{cr,x'}$, that is, the elastic critical moment at the x-locations relative to the elastic critical moment at the location of the maximum moment. Using $M_{cr,x'}$, the program calculates the relative slenderness $\bar{\lambda}_{LT}$.

Slenderness for lateral-torsional buckling

Calculation according to [1], Clause 6.3.2.2, for location with the maximum moment at $x = 2.00$ m:

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

$$\bar{\lambda}_{LT} = \sqrt{\frac{W_y \cdot f_y}{M_{cr}}} = \sqrt{\frac{354 \cdot 23.5}{215.71}} = 0.621$$

Reduction factor χ_{LT}

Calculation according to [1], Clause 6.3.2.3

HEB-160: $h/b = 1.0 < 2.0 \Rightarrow$ Buckling curve according to b, Table 6.5

$$\begin{aligned} \text{Auxiliary factor: } \Phi_{LT} &= 0.5 \cdot [1 + \alpha_{LT} \cdot (\bar{\lambda}_{LT} - \bar{\lambda}_{LT,0}) + \beta \cdot \bar{\lambda}_{LT}^2] \\ \Phi_{LT} &= 0.5 \cdot [1 + 0.34 \cdot (0.621 - 0.40) + 0.75 \cdot 0.621^2] = 0.682 \end{aligned}$$

$$\text{Limiting slenderness: } \bar{\lambda}_{LT,0} = 0.40$$

$$\text{Parameter (minimum value): } \beta = 0.75$$

$$\text{Imperfection factor: } \alpha_{LT} = 0.34 \quad (\text{Table 6.3})$$

$$\chi_{LT} = \frac{1}{\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta \cdot \bar{\lambda}_{LT}^2}} = \frac{1}{0.682 + \sqrt{0.682^2 - 0.75 \cdot 0.621^2}} = 0.908$$

According to [1], Clause 6.3.2.3, the reduction factor may be modified as follows:

$$\chi_{LT,mod} = \frac{\chi_{LT}}{f} \quad \text{where } f = 1 - 0.5 \cdot (1 - k_c) \cdot [1 - 2.0 \cdot (\bar{\lambda}_{LT} - 0.8)^2]$$

$$\chi_{LT,mod} = \frac{0.908}{0.972} = 0.934$$

For a parabolic moment diagram, we obtain the following correction factor k_c :

$$k_c = 0.94 \quad (\text{Table 6.6})$$

$$f = 1 - 0.5 \cdot (1 - k_c) \cdot [1 - 2.0 \cdot (\bar{\lambda}_{LT} - 0.8)^2] = 1 - 0.5 \cdot (1 - 0.94) \cdot [1 - 2.0 \cdot (0.621 - 0.8)^2]$$

$$f = 0.972$$

Interaction factors k_{yy} and k_{yz}

Determination according to [1], Annex B, Table B.2, for structural components susceptible to torsional deformations. The equivalent moment factor C_{mLT} according to Table B.3 for $\psi = 0$ is obtained as:

$$C_{m_y} = C_{m_{LT}} = 0.95 + 0.05 \cdot \alpha_h = 0.95 \quad \text{where} \quad \alpha_h = \frac{M_h}{M_s} = \frac{0}{10} = 0$$

$$k_{yy} = C_{m_y} \cdot \left(1 + (\bar{\lambda}_y - 0.2) \cdot \frac{N_{Ed}}{\chi_y \cdot N_{Rk}/\gamma_{M1}} \right) \leq C_{m_y} \cdot \left(1 + 0.8 \cdot \frac{N_{Ed}}{\chi_y \cdot N_{Rk}/\gamma_{M1}} \right)$$

$$k_{yy} = 0.95 \cdot (1 + (0.629 - 0.2) \cdot 0.286) \leq 0.95 \cdot (1 + 0.8 \cdot 0.286) = \underline{1.067} \leq 1.167$$

$$k_{yz} = 0.60 \cdot k_{zz} = 0.60 \cdot 1.481 = \underline{0.888}$$

Interaction factors k_{zy} and k_{zz}

Determination according to [1], Annex B, Table B.2, for structural components susceptible to torsional deformations

The equivalent moment factor C_{mLT} according to Table B.3 for $\psi = 0$ is obtained as:

$$C_{m_z} = 0.90 + 0.01 \cdot \alpha_h = 0.90 \quad \text{where} \quad \alpha_h = \frac{M_h}{M_s} = \frac{0}{10} = 0$$

$$k_{zy} = \left(1 - \frac{0.1 \cdot \bar{\lambda}_z}{C_{m_{LT}} - 0.25} \cdot \frac{N_{Ed}}{\chi_z \cdot N_{Rk}/\gamma_{M1}} \right) \geq \left(1 - \frac{0.1}{C_{m_{LT}} - 0.25} \cdot \frac{N_{Ed}}{\chi_z \cdot N_{Rk}/\gamma_{M1}} \right)$$

$$k_{zy} = \left(1 - \frac{0.1 \cdot 1.053}{0.95 - 0.25} \cdot 0.461 \right) \geq \left(1 - \frac{0.1}{0.95 - 0.25} \cdot 0.461 \right) = 0.892 \leq 0.934$$

$$k_{zy} = \underline{0.934}$$

$$k_{zz} = C_{m_z} \cdot \left(1 + (2 \cdot \bar{\lambda}_z - 0.6) \cdot \frac{N_{Ed}}{\chi_z \cdot N_{Rk}/\gamma_{M1}} \right) \leq C_{m_z} \cdot \left(1 + 1.4 \cdot \frac{N_{Ed}}{\chi_z \cdot N_{Rk}/\gamma_{M1}} \right)$$

$$k_{zz} = 0.90 \cdot (1 + (2 \cdot 1.053 - 0.6) \cdot 0.461) \leq 0.90 \cdot (1 + 1.4 \cdot 0.461) = 1.525 \geq 1.481$$

$$k_{zz} = \underline{1.481}$$

Interaction design for buckling about the major axis and lateral-torsional buckling

$$\frac{N_{Ed}}{\chi_y \cdot N_{Rk}/\gamma_{M1}} + k_{yy} \cdot \frac{M_{y,Ed}}{\chi_{LT} \cdot M_{y,Rk}/\gamma_{M1}} + k_{yz} \cdot \frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M1}} \leq 1 \quad \text{according to [1], Eq.(6.61)}$$

$$M_{y,Rk} = W_{pl,y} \cdot f_y = 354 \cdot 23.5 = 8,319 \text{ kNcm} = 83.19 \text{ kNm}$$

$$M_{z,Rk} = W_{pl,z} \cdot f_y = 169.96 \cdot 23.5 = 3,994.1 \text{ kNcm} = 39.94 \text{ kNm}$$

$$\frac{300}{0.822 \cdot 1,276.05/1.0} + 1.067 \cdot \frac{10.0}{0.908 \cdot 83.19/1.0} + 0.888 \cdot \frac{7.50}{39.94/1.0} = \underline{0.594} \leq 1$$

Interaction design for buckling about the minor axis and lateral-torsional buckling

$$\frac{N_{Ed}}{\chi_z \cdot N_{Rk}/\gamma_{M1}} + k_{zy} \cdot \frac{M_{y,Ed}}{\chi_{LT} \cdot M_{y,Rk}/\gamma_{M1}} + k_{zz} \cdot \frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M1}} \leq 1 \quad \text{according to [1], Eq.(6.62)}$$

$$\frac{300}{0.510 \cdot 1,276.05/1.0} + 0.934 \cdot \frac{10.0}{0.908 \cdot 83.19/1.0} + 1.481 \cdot \frac{7.50}{39.94/1.0} = \underline{0.863} \leq 1$$

Result values of RF-STEEL EC3 calculation

Section Height	h	160.0	mm		
Section Width	b	160.0	mm		
Criterion	h/b	1.00		≤ 2	Tab. 6.5
Buckling Curve	BC_{LT}	b			Tab. 6.5
Imperfection Factor	α_{LT}	0.340			Tab. 6.3
Shear Modulus	G	8,100.00	kN/cm ³		
Length Factor	k_z	1.000			
Length Factor	k_w	1.000			
Length	L	4.000	m		
Warping Constant of Cross-Section	I_w	47,940.00	cm ⁶		
Torsional Constant	I_t	31.40	cm ⁴		
Elastic Critical Moment for LTB for Determination of Related Slenderness	$M_{cr,0}$	190.90	kNm		
Moment Distribution	Diagr M_y	6) Parabola			
Maximum Sagging Moment	$M_{y,max}$	10.00	kNm		
Boundary Moment	$M_{y,A}$	0.00	kNm		
Moment Ratio	ψ	0.000			
Moment Factor	C_1	1.130			[2]
Ideal elastic critical moment	M_{cr}	215.71	kNm		
Elastic Section Modulus	W_y	354.00	cm ³		
Slenderness	λ_{LT}	0.621			6.3.2.2(1)
Parameter	$\lambda_{LT,0}$	0.400			6.3.2.3(1)
Parameter	β	0.750			6.3.2.3(1)
Auxiliary Factor	ϕ_{LT}	0.682			6.3.2.3(1)
Reduction Factor	χ_{LT}	0.908			Eq. (6.57)
Correction Factor	k_c	0.940			6.3.2.3(2)
Modification Factor	f	0.972			6.3.2.3(2)
Reduction Factor	$\chi_{LT,mod}$	0.934			Eq. (6.58)
Moment Distribution	Diagr M_y	3) Max in Span			Tab. B.3
Moment Factor	ψ_y	1.000			Tab. B.3
Moment	$M_{h,y}$	0.00	kNm		Tab. B.3
Moment	$M_{s,y}$	10.00	kNm		Tab. B.3
Ratio $M_{h,y}/M_{s,y}$	$\alpha_{h,y}$	0.000			Tab. B.3
Load Type	Load z	Uniform load			Tab. B.3
Moment Factor	C_{my}	0.950			Tab. B.3
Moment Distribution	Diagr M_z	3) Max in Span			Tab. B.3
Moment Factor	ψ_z	1.000			Tab. B.3
Moment	$M_{h,z}$	0.00	kNm		Tab. B.3
Moment	$M_{s,z}$	7.50	kNm		Tab. B.3
Ratio $M_{h,z}/M_{s,z}$	$\alpha_{h,z}$	0.000			Tab. B.3
Load Type	Load y	Concentrated load			Tab. B.3

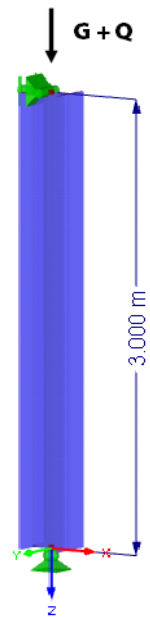
Moment Factor	C_{mz}	0.900			Tab. B.3
Moment Distribution	Diagr $M_{y,LT}$	3) Max in Span			Tab. B.3
Moment Factor	$\psi_{y,LT}$	1.000			Tab. B.3
Moment	$M_{h,y,LT}$	0.00	kNm		Tab. B.3
Moment	$M_{s,y,LT}$	10.00	kNm		Tab. B.3
Ratio $M_{h,y,LT}/M_{s,y,LT}$	$\alpha_{h,y,LT}$	0.000			Tab. B.3
Load Type	Load z	Uniform load			Tab. B.3
Moment Factor	C_{mLT}	0.950			Tab. B.3
Component Type	Component	Torsionally Weak			
Interaction Factor	k_{yy}	1.067			Tab. B.2
Interaction Factor	k_{yz}	0.888			Tab. A.1
Interaction Factor	k_{zy}	0.934			Tab. A.1
Interaction Factor	k_{zz}	1.481			Tab. A.1
Axial Force (Compression)	N_{Ed}	300.00	kN		
Cross-Sectional Area	A_i	54.30	cm ²		Tab. 6.7
Compression Resistance	N_{Rk}	1,276.05	kN		Tab. 6.7
Partial Factor	γ_{M1}	1.000			6.1
Design Component for N	γ_{Ny}	0.29		≤ 1	Eq. (6.61)
Design Component for N	h_{Nz}	0.46		≤ 1	Eq. (6.62)
Moment	$M_{y,Ed}$	10.00	kNm		
Moment Resistance	$M_{y,Rk}$	83.19	kNm		Tab. 6.7
Moment Component	η_{My}	0.13			Eq. (6.61)
Moment	$M_{z,Ed}$	7.50	kNm		
Elastic Section Modulus	W_z	169.96	cm ³		
Moment Resistance	$M_{z,Rk}$	39.94	kNm		Tab. 6.7
Moment Component	η_{Mz}	0.19			Eq. (6.61)
Design 1	η_1	0.59		≤ 1	Eq. (6.61)
Design 2	η_2	0.86		≤ 1	Eq. (6.62)

Table 8.4: Result values of RF-STEEL EC3 calculation

2.1 Fire Resistance

In this example, we perform the fire design of a steel column.

System and Loads



Column cross-section: HE-B 300, Steel S 235

System: Hinged column, $\beta = 1.0$

System height: 3.00 m

Loading: $G_K = 1,200$ kN

$Q_K = 600$ kN

Figure 8.3: System and loads

Ultimate Limit State Design for Room Temperature

Flexural buckling about the minor axis (\perp to z-z axis)

$$N_{cr,z} = \frac{21,000 \cdot 8,560.00 \cdot \pi^2}{300.00^2} = 19,712.90 \text{ kN}$$

$$\bar{\lambda}_z = \sqrt{\frac{A \cdot f_y}{N_{cr,z}}} = \sqrt{\frac{149.0 \cdot 24.0}{19,712.90}} = 0.426$$

$$\bar{\lambda}_z = 0.426 > 0.2 \quad \rightarrow \text{Design for flexural buckling must be performed.}$$

Cross-sectional geometry $h/b = 1.00 \leq 1.2$ Structural steel S 235 $t \leq 100$ mm

[1], Table 6.2, row 3, column 4: Buckling curve c

$$\Rightarrow \alpha_z = 0.49 \quad (\text{Table 6.1})$$

$$\Phi = 0.5 \cdot [1 + 0.49 \cdot (0.426 - 0.2) + 0.426^2] = 0.646$$

$$\chi_Z = \frac{1}{0.646 + \sqrt{0.646^2 - 0.426^2}} = 0.884$$

$$N_{Ed} = 1.35 \cdot G_K + 1.5 \cdot Q_K = 1.35 \cdot 1,200 + 1.5 \cdot 600 = 2,520 \text{ kN}$$

Design ratio

$$\frac{N_{Ed}}{\chi_Z \cdot A \cdot f_y / \gamma_{M1}} = \frac{2,520}{0.884 \cdot 149.0 \cdot 24.0 / 1.1} = 0.877 \leq 1.0$$

Result values of RF-STEEL EC3 calculation

Moment of Inertia	I_z	8,560.00	cm ⁴		
Effective Member Length	$L_{cr,z}$	3.000	m		
Elastic Flexural Buckling Force	$N_{cr,z}$	19,712.9	kN		
Slenderness	λ_z	0.4,259		> 0.2	6.3.1.2(4)
Buckling Curve	BC_z	c			Tab. 6.2
Imperfection Factor	α_z	0.490			Tab. 6.1
Auxiliary Factor	Φ_z	0.646			6.3.1.2(1)
Reduction Factor	χ_z	0.884			Eq. (6.49)
Buckling Resistance	$N_{b,z,Rd}$	2,872.27	kN		Eq. (6.47)
Design ratio	η	0.877		≤ 1.0	Eq. (6.46)

Table 8.5: Result values of RF-STEEL EC3 calculation

Fire Resistance Design

After a fire exposure of 90 min, the mean steel temperature is 524 °C, according to the standard temperature-time curve.

As a fire resistance material, a box-shaped GRP encasement (glass-reinforced plastic) is used, having the following properties:

Unit Mass:	$\rho_p = 945.0 \text{ kg/m}^3$
Thermal Conductivity:	$\lambda_p = 0.20 \text{ W/K}$
Specific Heat:	$c_p = 1,700 \text{ J/kgK}$
Thickness:	$d_p = 18 \text{ mm}$

Determination of reduction factors

$$k_{y,\theta} = 0.703 \quad \text{according to [2], Table 3.1}$$

$$k_{E,\theta} = 0.528 \quad \text{according to [2], Table 3.1}$$

Design in fire situation according to [2], 4.2.3.2

Imperfection factor α :

$$\alpha = 0.65 \cdot \sqrt{\frac{235}{f_y}} = 0.65 \cdot \sqrt{\frac{235}{240}} = 0.643$$

Non-dimensional slenderness ratio $\bar{\lambda}_{\theta}$:

$$\bar{\lambda}_{\theta} = \bar{\lambda} \cdot \left[\frac{k_{y,\theta}}{k_{E,\theta}} \right]^{0.5} = 0.426 \cdot \left[\frac{0.703}{0.528} \right]^{0.5} = 0.491$$

Auxiliary factor:

$$\Phi_{\theta} = \frac{1}{2} \cdot \left[1 + \alpha \cdot \bar{\lambda}_{\theta} + \bar{\lambda}_{\theta}^2 \right] = \frac{1}{2} \cdot \left[1 + 0.643 \cdot 0.491 + 0.491^2 \right] = 0.778$$

Reduction factor for flexural buckling in the fire design situation:

$$\chi_{fi} = \frac{1}{\varphi_{\theta} + \sqrt{\varphi_{\theta}^2 - \bar{\lambda}_{\theta}^2}} = \frac{1}{0.778 + \sqrt{0.778^2 - 0.491^2}} = 0.723$$

Buckling resistance of structural component subjected to compression:

$$N_{b,fi,Rd} = \frac{\chi_{fi} \cdot A \cdot k_{y,\theta} \cdot f_y}{\gamma_{M,fi}} = \frac{0.723 \cdot 149.0 \cdot 0.703 \cdot 24}{1.0} = 1,817.83$$

Loading in case of fire:

$$N_{fi,Ed} = 1.0 \cdot G_k + 0.9 \cdot Q_k = 1.0 \cdot 1,200 + 0.9 \cdot 600 = 1,740 \text{ kN}$$

Design ratio

$$\eta = \frac{N_{fi,Ed}}{N_{b,fi,Rd}} = \frac{1,740}{1,817.83} = 0.957 \leq 1.0$$

Result values of RF-STEEL EC3 calculation

Reduction Factor	$k_{y,\theta}$	0.703			[2], Tab. 3.1
Reduction Factor	$k_{E,\theta}$	0.528			[2], Tab. 3.1
Slenderness	$\lambda_{z,\theta}$	0.4,915			[2], Eq. (4.7)
Imperfection Factor	α	0.6,432			[2], 4.2.3.2(2)
Auxiliary Factor	$\Phi_{z,\theta}$	0.778			[2], 4.2.3.2(2)
Reduction Factor	$\chi_{z,fi}$	0.723			[2], Eq. (4.6)
Buckling Resistance	$N_{b,fi,z,\theta,Rd}$	1,817.83	kN		
Design ratio	η	0.957		≤ 1.0	[2], Eq. (4.1)

Table 8.6: Result values of RF-STEEL EC3 calculation

Literature

- [1] *EN 1993-1-1: Bemessung und Konstruktion von Stahlbauten Teil 1-1: Allgemeine Bemessungsregeln und Regeln für den Hochbau.* Beuth Verlag GmbH, Berlin, 2005.
- [2] *EN 1993-1-2: Eurocode 3: Bemessung und Konstruktion von Stahlbauten Teil 1-2: Allgemeine Regeln - Tragwerksbemessung für den Brandfall.* Beuth Verlag GmbH, Berlin, 2006.
- [3] *EN 1993-1-4: Bemessung und Konstruktion von Stahlbauten Teil 1-4: Ergänzende Regeln zur Anwendung von nichtrostenden Stählen.* Beuth Verlag GmbH, Berlin, 2006.
- [4] Johannes Naumes, Isabell Strohmann, Dieter Ungermann and Gerhard Sedlacek. Die neuen stabilitätsnachweise im stahlbau nach eurocode 3. *Stahlbau*, 77, 2008.
- [5] Johannes Naumes, Markus Feldmann and Gerhard Sedlacek. *Biegeknicke und Biegedrillknicken von Stäben auf einheitlicher Grundlage*, volume 70. Shaker Verlag, 2010.
- [6] *EN 1993-1-3: Bemessung und Konstruktion von Stahlbauten Teil 1-3: Allgemeine Regeln – Ergänzende Regeln für kaltgeformte dünnwandige Bauteile und Bleche.* Beuth Verlag GmbH, Berlin, 2006.
- [7] *EN 1993-1-5: Bemessung und Konstruktion von Stahlbauten Teil 1-5: Plattenförmige Bauteile.* Beuth Verlag GmbH, Berlin, 2006.
- [8] *EN 1993-2: Eurocode 3: Bemessung und Konstruktion von Stahlbauten - Teil 2: Stahlbrücken.* CEN, Brüssel, 2007.
- [9] *EN 1991-1-2: Eurocode 1: Einwirkungen auf Tragwerke - Teil 1-2: Allgemeine Einwirkungen - Brandeinwirkungen auf Tragwerke.* Beuth Verlag GmbH, Berlin, 2002.
- [10] *EN 1993-1-8: Bemessung und Konstruktion von Stahlbauten Teil 1-8: Bemessung von Anschlüssen.* Beuth Verlag GmbH, Berlin, 2005.

Index

A	
Accidental	9
Adapted method	12
Axis	23
B	
Background graphic	64
Beam spacing	35, 37, 38, 39, 40
Beam type	31
Boundary conditions	34
Bracing	36, 37
Buckling	23
Buckling curve	20
Buttons	63
C	
Calculation	42
Cantilever	22, 31
Characteristic	10
Classification	18, 43
Clipboard	79
Color bars	63
Color scale	68, 71
Comment	7, 25
Connection deformation	38
Continuous beam effect	39, 40
Continuous rotational restraint	38
Control panel	68
Corrugated sheet	38
Cross-section	17, 34, 60, 75
Cross-section class	43
Cross-section design	53
Cross-section info	19
Cross-section library	17
Cross-section optimization	49, 75
Cross-section type	18
Cross-sectional area	40
D	
Decimal places	15, 77
Deflection	10
Deformation analysis	10, 31
Design	7, 51, 52, 53, 54, 55, 57
Design case	65, 73, 74
Design colored	68
Design of welds	49
Design ratio	18, 52, 53
Design situation	9, 53
Detail settings	42
Diagonal	37
Discrete rotational restraint	39
Display hidden result diagram	69
Display navigator	65, 68, 69
E	
Effective length	22, 23, 26, 78
Effective length factor	24
Elastic critical moment for lateral-torsional buckling	45
End panel	39, 40
Equivalent member length	22
Equivalent member method	26, 27, 30, 46
European lateral-torsional buckling curve	12
Excel	79
Exit RF-STEEL EC3	6
Export	78
Exporting cross-section	76
Exporting effective lengths	78
Exporting materials	78
F	
Fastening arrangement	35
Favorite	75
Filter	63, 68, 69
Filtering members	69
Fire exposure	32
Fire protection design	63
Fire resistance	11, 47, 88
Fire resistance design	32, 47
Flexural buckling	21, 23, 44
Frequent	10
G	
General data	6
General method	46
Graphic printout	71, 72
Graphics	65
H	
Hollow encasement	32
I	
Increment factor	44
Installation	4
Interaction	43
Intermediate lateral restraints	21
Internal forces	57, 76
Internal panel	39, 40

L		Q	
Lateral and torsional restraint	21, 25	Quasi-permanent	10
Lateral restraints	21		
Lateral-torsional buckling	12, 21, 24, 81, 84	R	
Length	22, 60	Reference length	10
Limit deformation	46	Relatively	21
Limit values	7, 10, 11, 45	Remark	19
List of member	31	Rendering	68
Load application	45	Result combination	9, 10, 63
Load case	8, 9, 10	Result diagram	67, 71
Load combination	9	Result values	65
Loading	57	Result window	51
Location x	52	Results display	65
		Results evaluation	51, 63
M		RF-STABILITY	23
Material	15, 78	RFEM graphic	67, 71
Material description	15	RFEM work window	65, 68
Material library	16	Rotational restraint	38
Material properties	15, 16	Rotational spring	40
Member	7	Rotational spring stiffness C_{100}	38
Member hinge	30		
Member slenderness	49, 59	S	
Model type	45	Scrolling in windows	6
		Second-order analysis	43, 44
N		Serviceability	46
National Annex	7, 11	Serviceability limit state	10, 31, 63
Naumes	12	Set of members	7, 26, 27, 30, 31, 41, 46, 54, 58, 61
Navigator	6	Shear panel	35
Net cross-sectional area	40	Shear panel length	35, 37
Net heat flux	47	Shear panel stiffness	38
Nodal support	27	Shifted members ends	46
		SHAPE-THIN	43
O		Slenderness	59
Opening RF-STEEL EC3	4	Special cases	45
OpenOffice	79	Stability analysis	9, 12, 23, 43, 44, 53
Optimization	18, 49, 75, 76	Stability analysis method	46
		Stability design	14, 45
P		Stainless steel	14, 16
Panel	5, 66, 68	Starting calculation	50
Parameter	34, 41	Steel bridge	47
Parametrized cross-section	75	Stress point	20
Part	60	Sum	61
Partial factors	12	Surface area	61
Parts list	60, 61		
Persistent	9	T	
Post	37	Table	6
Precamber	31	Taper	19, 45, 53, 77
Print graphic	71	Temperature curve	47
Printout report	70, 72	Tension design	40
Program start	4	Torsion	45
Protection type	32	Transient	9
Purlin	40		

Transverse load	45	Visibility	68
Trapezoidal sheeting	35, 37	Volume	61
U			
Ultimate limit state	8, 42, 63	X	
Undeformed system	46	x-Location	52, 57
Units	15, 77	W	
User profile	77	Warping length factor	23, 25
V			
View mode	64	Web breathing	47
		Weight	61