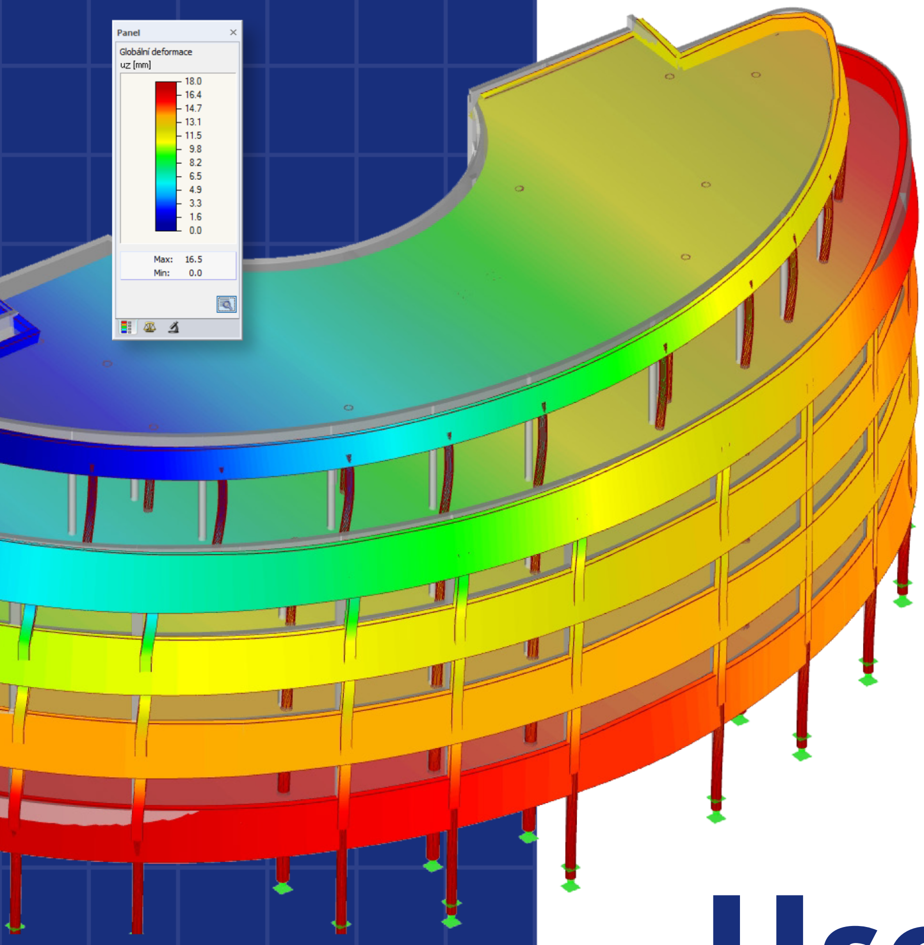
RF-/STEEL EC3

Ultimate limit state and serviceability limit state design, fire resistance design, and stability analyses according to Eurocode 3



User Manual

Version

October 2020

Dlubal Software

Short Overview

Dlubal Software GmbH
Am Zellweg 2
93464 Tiefenbach
Germany

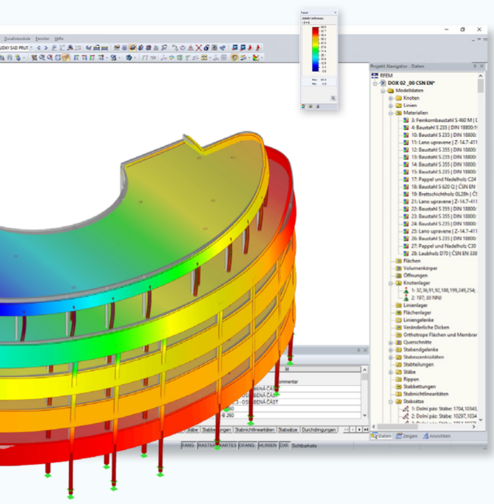
Telephone: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com

Dlubal Software, Inc.
The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Phone: +1 267 702-2815
E-mail: info@dlubal.com

Using the Manual

The program description is organized in chapters which follow the order and structure of the input and result tables. The chapters present the individual tables column by column. They help to better understand the functioning of the add-on module. General functions are described in the manuals of the main programs RFEM and RSTAB.
This manual also deals with the module extensions RF-/STEEL Warping Torsion, RF-/STEEL Plasticity, and RF-/STEEL Cold-Formed Sections.



Hint

The text of the manual shows the described buttons in square brackets, for example [OK]. In addition, they are pictured on the left. Expressions appearing in dialog boxes, tables, and menus are set in *italics* to clarify the explanation. You can also use the search function for the Knowledge Base and FAQs to find a solution in the posts about the RF-/STEEL EC3 add-on module.

Topicality

The high quality standards placed on the software are guaranteed by a continuous development of the program versions. This may result in differences between program description and the current software version you are using. Thank you for your understanding that no claims can be derived from the figures and descriptions. We always try to adapt the documentation to the current state of the software.

Table of Contents

1 Introduction

1.1 Add-on Module RF-/STEEL EC3

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

In the following, the add-on modules of both main programs are described in one manual and are referred to as **RF-/STEEL EC3**.

RF-/STEEL EC3 performs all typical ultimate limit state designs as well as stability and deformation analyses. In the ultimate limit state design, the add-on module considers the effect of various loadings. It is possible to choose among the interaction formulas given in the standard. An essential part of the verification according to Eurocode 3 is the classification of the cross-sections to be designed into the classes 1 to 4. This way, the limitation of the capacity to withstand stresses as well as the rotational capacity due to local buckling of cross-section parts is checked. RF-/STEEL EC3 determines the c/t-ratios of the cross-section parts subjected to compression and carries out the classification automatically.

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

RF-/STEEL EC3 performs 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. The precautions against fire can be selected in the form of encasements with different physical properties.

The serviceability limit state represents an important design for structures with slender cross-sections. 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, it is possible to 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 [4].

If you have a license for the "RF-/STEEL Cold-Formed Sections" module extension, the design of cold-formed components according to EN 1993-1-3 is also possible.

The add-on module provides an automatic cross-section optimization with the possibility to export modified cross-sections to RFEM or RSTAB. Separate design cases allow for the flexibility to analyze individual structural components in complex structures.

Like other add-on modules, RF-/STEEL EC3 is completely integrated in the RFEM or RSTAB program. Thus, the design-relevant input data is preset when you have started the add-on module. After the design, you can use the graphical user interface of the main program to evaluate the results. As they are also included in the global printout report, the entire verification can be presented in a consistent and appealing form.

This manual also describes the following two module extensions for RF-/STEEL EC3:

- **RF-/STEEL Warping Torsion**
- **RF-/STEEL Plasticity**
- **RF-/STEEL Cold-Formed Sections**

We wish you a pleasant and successful working with RF-/STEEL EC3.

Your Dlubal Software team

1.2 Using the Manual

Topics like installation, graphical user interface, results evaluation, and printout are described in detail in the manuals of the main programs RFEM and RSTAB. The present manual focuses on typical features of the RF-/STEEL EC3 add-on module.

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

In the PDF manual, you can perform a full-text search as usual with [Ctrl]+[F]. However, if you cannot find what you are looking for, you can also go to the Knowledge Base on our website to find related articles about the steel add-on modules. You can also consult the FAQs on our website.

1.3 Opening RF-/STEEL EC3 Add-on Module

RFEM and RSTAB provide the following options to open the RF-/STEEL EC3 add-on module.

Menu

To start the program on the RFEM or RSTAB menu bar, select

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

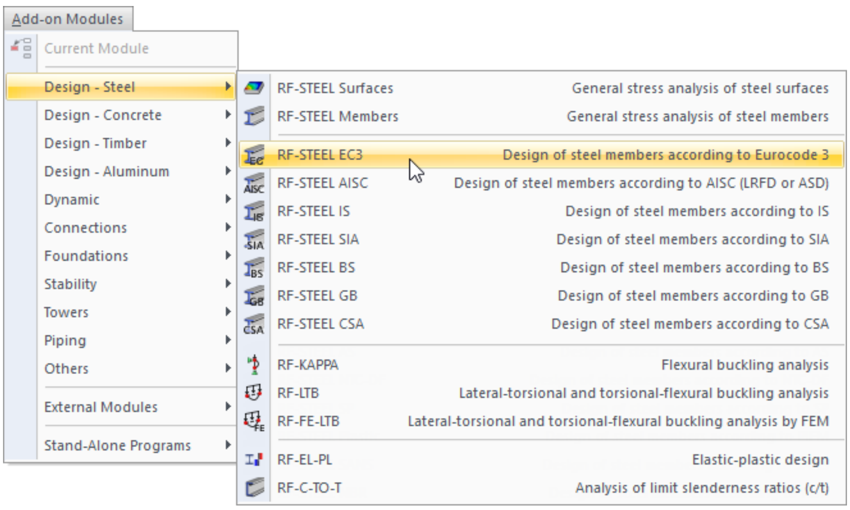


Figure 1.1 Menu *Add-on Modules → Design - Steel → RF-STEEL EC3*

Navigator

To start RF-/STEEL EC3 in the *Data* navigator, select

Add-on Modules ⟶ RF-/STEEL EC3.

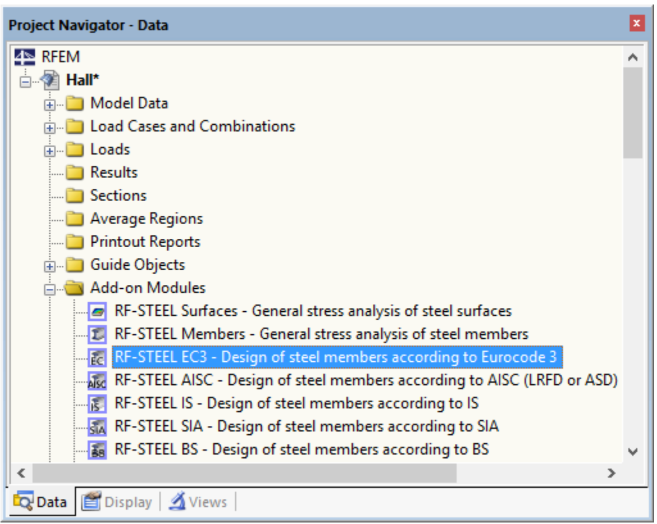
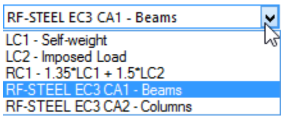


Figure 1.2 Data navigator: *Add-on Modules ⟶ RF-STEEL EC3*

Panel



If any results from RF-/STEEL EC3 are already available in the model, you can open the design module in the panel:

Set the relevant design case in the load case list of the menu bar. Click the [Show Results] button to display the design criterion graphically on the members.

When the results display is activated, the panel appears showing the [RF-/STEEL EC3] button which you can use to open the add-on module.

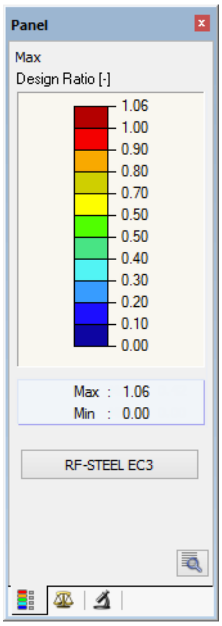


Figure 1.3 Panel with [RF-STEEL EC3] button

2 Input Data

When you have started the add-on module, a new window appears. In this window, a navigator is displayed on the left, managing the available module windows. The drop-down list above the navigator contains the design cases (see Chapter 7.1).

The design-relevant data must be defined in several input windows. The following parameters are imported automatically when you open RF-/STEEL EC3 for the first time:

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

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

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

2.1 General Data

In Window *1.1 General Data*, you select the members, sets of members, and actions that you want to design. The three tabs manage the load cases, load combinations, and result combinations for the ultimate limit state, the serviceability limit state, and the fire protection design.

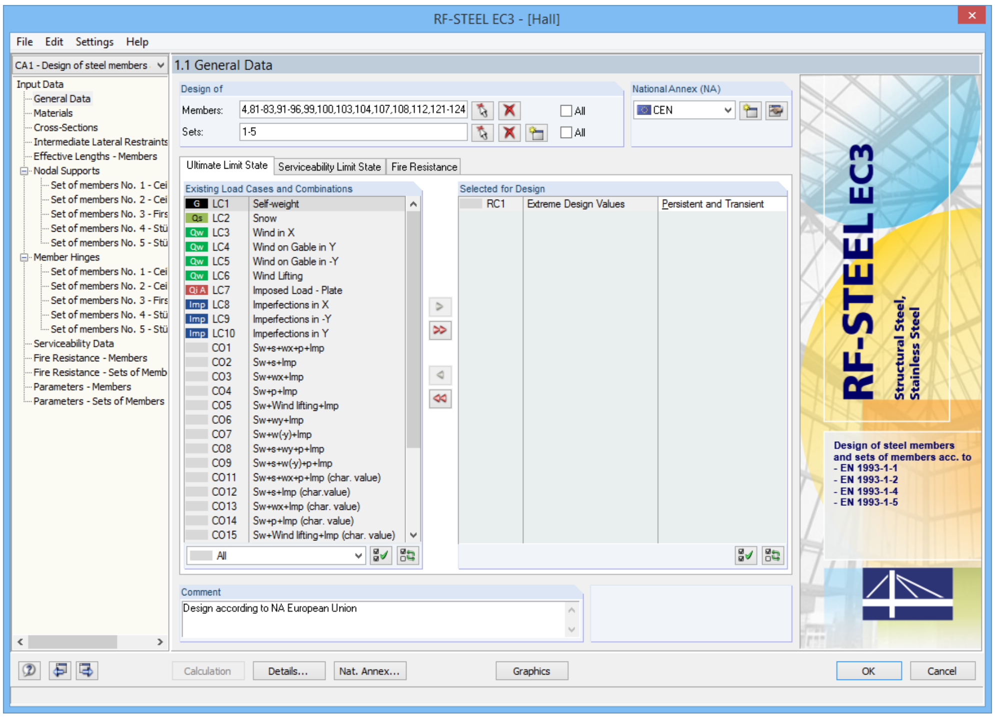


Figure 2.1 Window *1.1 General Data*

Design of

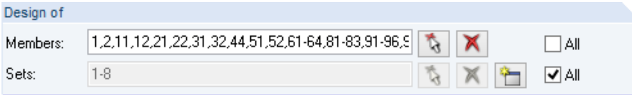


Figure 2.2 Design of members and sets of members

You can design *Members* as well as *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. Use the [Delete] button to clear the list of preset numbers. Use the [Select] button to define objects graphically in the RFEM or RSTAB work window.

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

To define a new set of members, click the [New] button. The dialog box known from RFEM or RSTAB appears where you can enter the parameters for the set of members.

National Annex (NA)

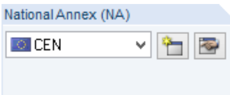


Figure 2.3 National Annex

In the drop-down list in the upper-right corner of the window, you can select the National Annex whose parameters apply to the design and to the deformation's limit values .

Use the [Edit] button to open a dialog box where you can check and, if necessary, adjust the parameters of the current NA. The dialog box is described in Chapter 2.1.4.

Comment

Figure 2.4 User-defined comment

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

2.1.1 Ultimate Limit State

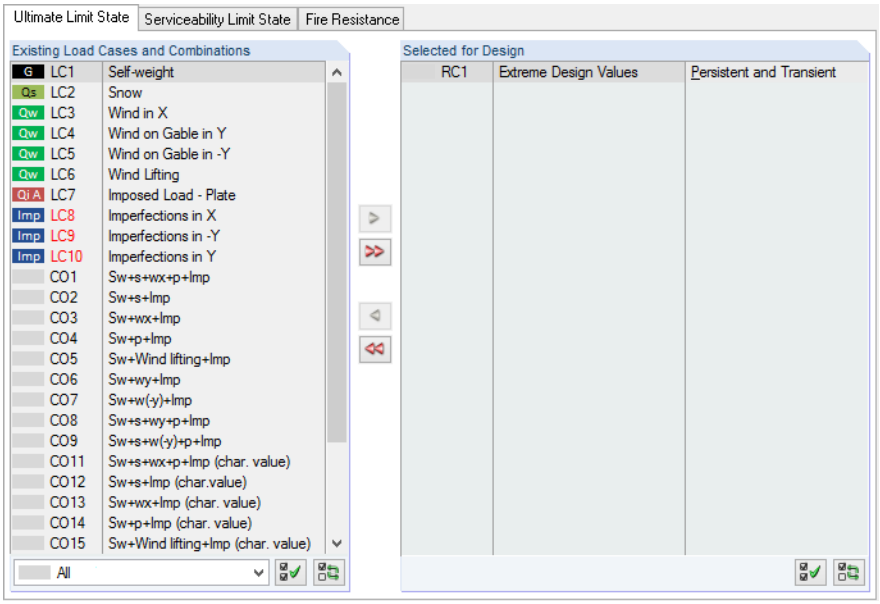


Figure 2.5 Window *1.1 General Data*, tab *Ultimate Limit State*

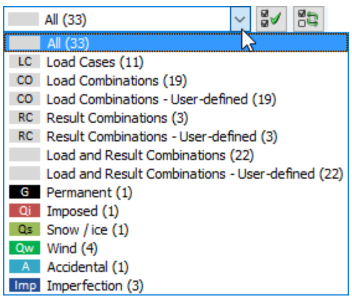
Existing Load Cases and Combinations

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

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

As common for Windows applications, selecting several load cases is possible by clicking them one by one while holding down the [Ctrl] key. Thus, you can transfer several load cases at the same time.

If a load case's number is marked in red such as LC8 in Figure 2.5, you cannot design it: It indicates a load case without load data, or a load case that contains imperfections. A warning appears if you try to transfer it.



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

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

Table 2.1 Buttons in *Ultimate Limit State* tab

Selected for Design

The column on the right lists the load cases as well as load and result combinations that have been selected for the design. To remove selected items from the list, click or double-click the entries. To empty the entire list, click .

You can assign the load cases, load and result combinations 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).

To change the design situation, use the list which you can access by clicking the button at the end of the text box.

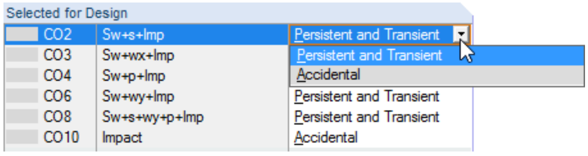


Figure 2.6 Assigning the design situation

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

Result combination

Designing an enveloping max/min result combination (RC) is performed faster than designing all contained load cases and load combinations, but 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. This distribution, however, only rarely reflects the moment distribution that is available in the individual load combinations. Thus, for an RC design, more unfavorable values for M_{cr} are expected, leading to higher ratios.

Result combinations should be selected for design only in case of dynamic combinations. For "usual" combinations, it is recommended to use load combinations because here the actual moment distributions are applied for the determination of M_{cr}.

Details...

In the *General* tab of the *Details* dialog box, you can define how result combinations of the 'OR' type are handled in the design (see Chapter 3.1.8).

2.1.2 Serviceability Limit State

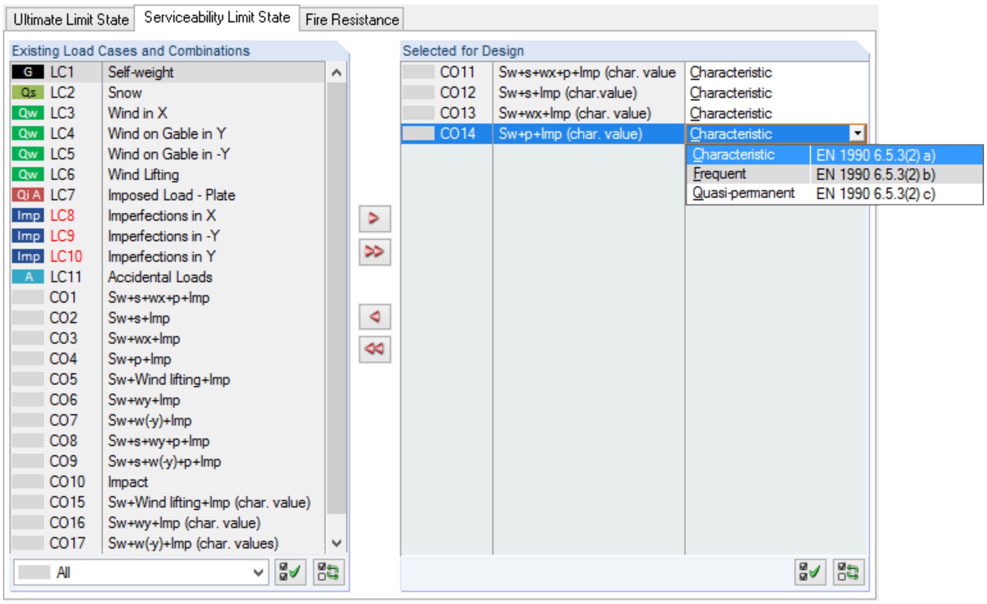


Figure 2.7 Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

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

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

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

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

Nat. Annex...

The limit values of the deformations are defined in the National Annex. They can be adjusted for the design situations in the *National Annex Settings* dialog box (see Figure 2.10) that you open with the [Nat. Annex] button.

In Window *1.9 Serviceability Data*, the reference lengths applying to the deformation analysis are managed (see Chapter 2.9).

2.1.3 Fire Resistance

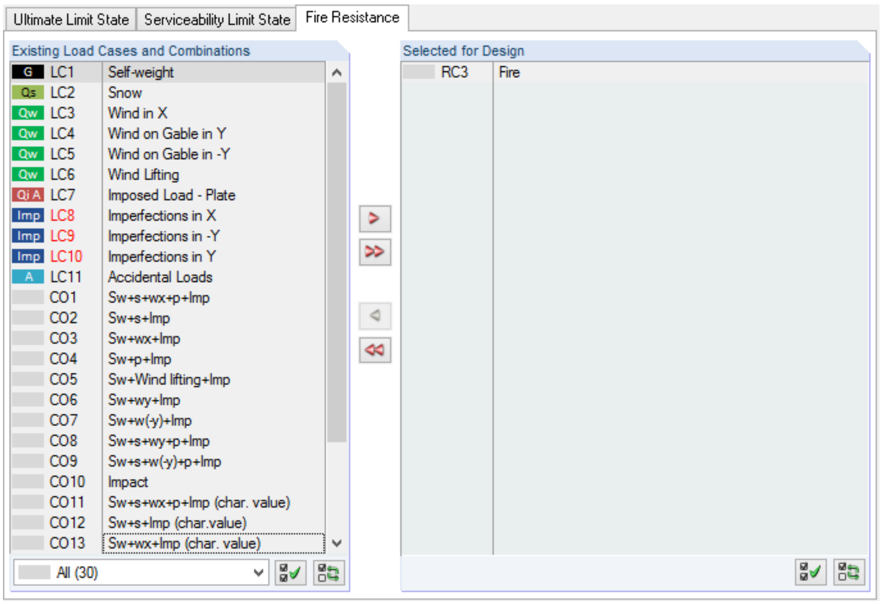


Figure 2.8 Window *1.1 General Data*, tab *Fire Resistance*

Existing Load Cases and Combinations

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

Selected for Design

You can add or remove load cases as well as load and result combinations, as described in Chapter 2.1.1. Here, you should select the actions that have been determined according to EN 1991-1-2 [2].

Fire protection designs are also possible for combined cross-sections.

2.1.4 National Annex (NA)

In the list in the upper right corner of Window *1.1 General Data*, you can select the National Annex whose parameters you want to apply to the design and the limit values of the deformation.

Figure 2.9 Selecting the National Annex

If necessary, to check and adjust the preset parameters (see Figure 2.10), click the button. Use the button to create a user-defined annex.

Nat. Annex...

Moreover, in every input window you find the [Nat. Annex] button which you also can use to open the *National Annex Settings* dialog box that consists of three tabs.

Base

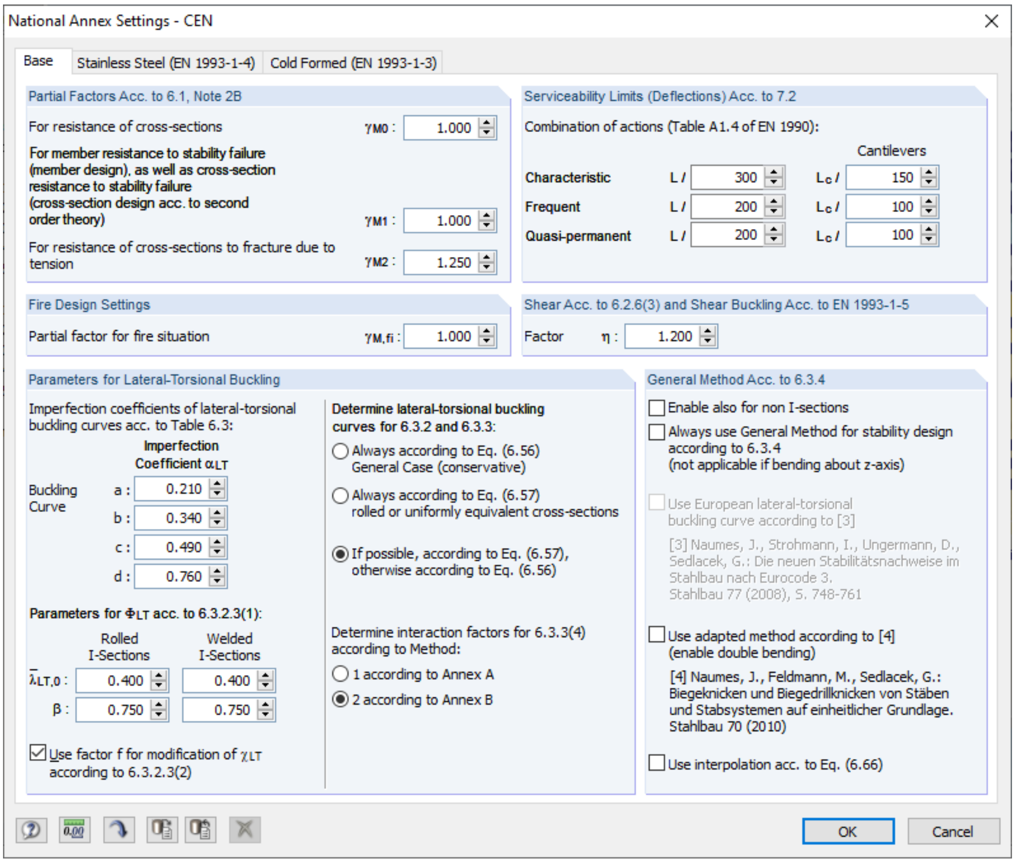


Figure 2.10 Dialog box *National Annex Settings - CEN*, tab *Base*

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

In the *General Method Acc. to 6.3.4* section, you can specify if the stability analyses are always performed according to [1] clause 6.3.4. According to the German National Annex, the general method is only allowed for I-shaped cross-sections. The option *Enable also for non I-sections* allows you to use the method for other cross-sections, too.

In addition, you can perform a stability analysis using the *European lateral-torsional buckling curve* by Naumes [6]. In his dissertation [7], Naumes expanded 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. The *adapted method* is also available for designing unsymmetrical cross-sections as well as 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

(a) as the minimum value of the values for 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

(b) as a value that is interpolated between χ and χ_{LT} — see also [1] equation (6.66).

As the method by Naumes is based on the standardized European lateral-torsional buckling curve taking into account the modified imperfection factor α^*, the interaction between flexural buckling and lateral-torsional buckling according to [1] equation (6.66) can be omitted.

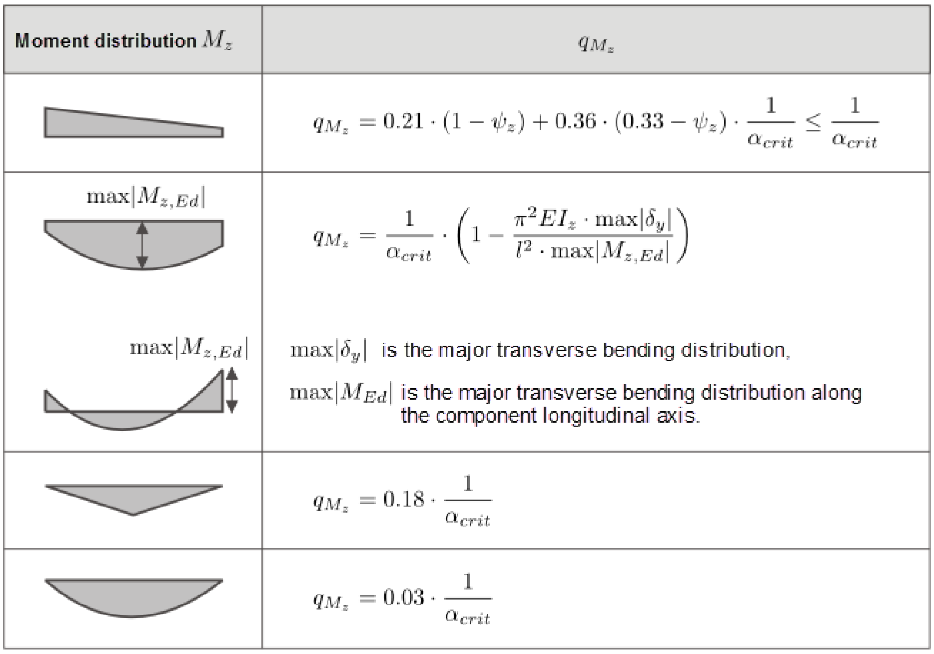
Calculation	
Main plane	Secondary plane
$\alpha_{Ed}(x) = \frac{\chi_{LT}(x) \cdot \alpha_{ult,k}(x)}{\gamma_{M1}} \geq 1$	$\beta_z(x) = \frac{M_{z,Ed}(x)}{M_{z,Rd}(x)} \cdot (1 - q_{Mz})$
Design	
simplified	exact
$\Delta n_R = 0.9$	$\Delta n_R = 1 - \frac{1}{\alpha_{Ed}(x)} \cdot \left[1 - \frac{1}{\alpha_{Ed}(x)}\right]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

Figure 2.11 Calculation run for method by Naumes

In the first step, the calculation is carried out separately for the main and 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 of the main and secondary load-bearing plane and comparing them to the design criterion Δn_R.



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

Figure 2.12 Determination of moment factor q_{Mz}

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

Button	**Function**
	Resets the program's presettings
	Imports user-defined default settings
	Saves modified settings as default
	Deletes user-defined National Annex

Table 2.2 Buttons in *National Annex Settings* dialog box

Stainless Steel (EN 1993-1-4)

RF-/STEEL EC3 allows for the design of structural components made of stainless steel according to EN 1993-1-4 [4].

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

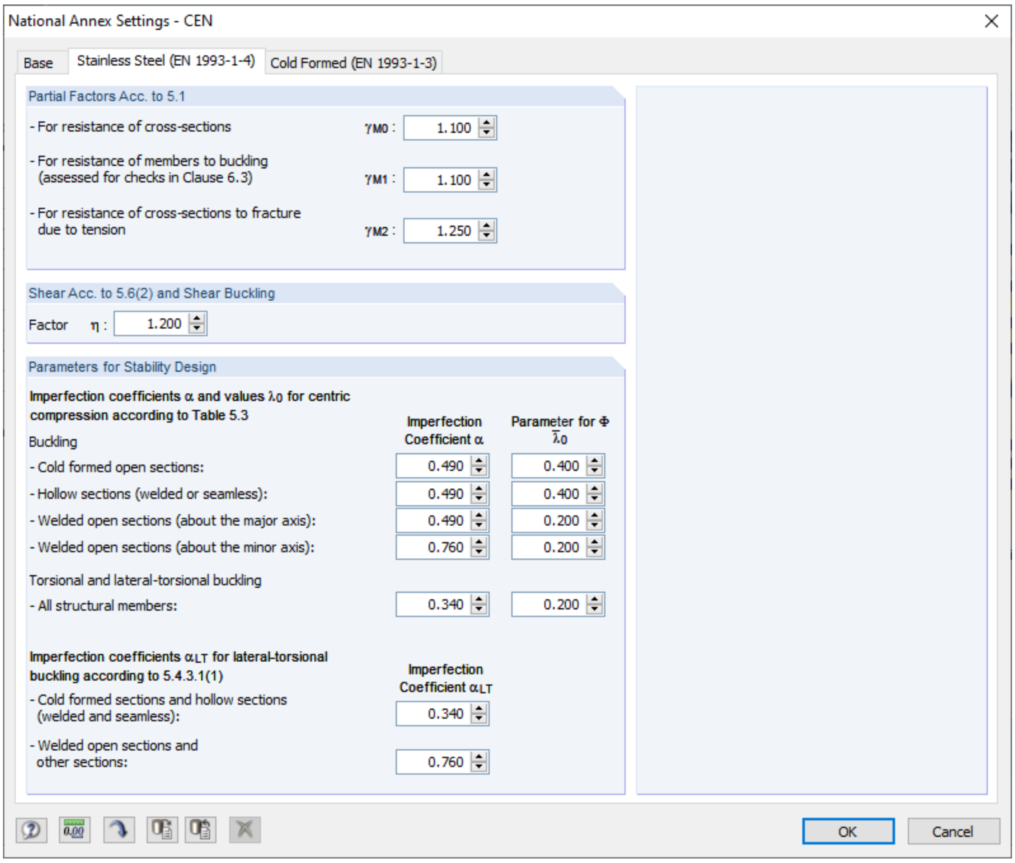


Figure 2.13 Dialog box *National Annex Settings - CEN*, tab *Stainless Steel (EN 1993-1-4)*

Cold Formed (EN 1993-1-3)

In this tab, you can define the partial factors for designing cold-formed sections according to EN 1993-1-3 [3]. A license of the **RF-/STEEL Cold-Formed Sections** module extension is required for the design.

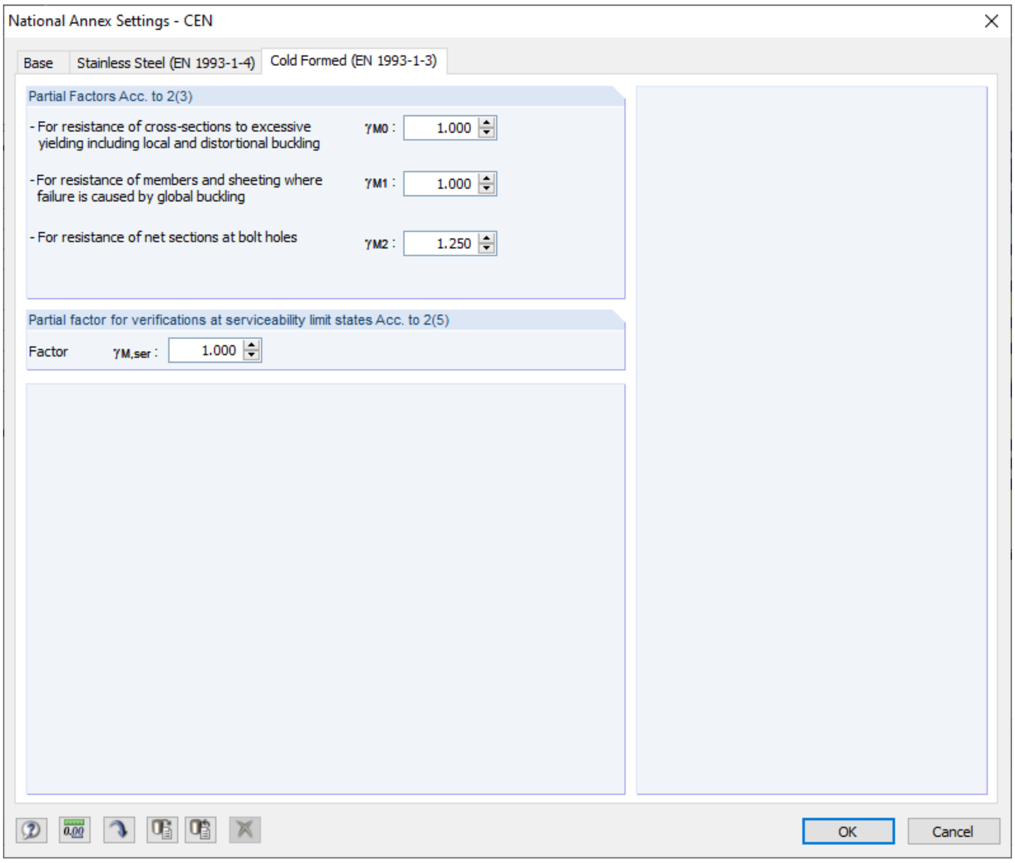


Figure 2.14 Dialog box *National Annex Settings - CEN*, tab *Cold-Formed (EN 1993-1-3)*

The *Partial Factors Acc. to 2(3)* section manages the factors γ_M, which are to be applied for the ultimate limit state design according to [3] 2(3). The recommended values or the values defined in the National Annex are preset.

If necessary, you can adjust the factor $\gamma_{M,ser}$ in the *Partial factor for verifications at serviceability limit states Acc. to 2(5)* section.

According to [3] Table 3.1a, Note 1, a *Reduction Factor of* f_{yb} *and* f_u should be considered for steel plates with a thickness of less than 3 mm. This factor for reducing the basic yield strength and tensile strength is preset with the recommended value of 0.9.

2.2 Materials

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

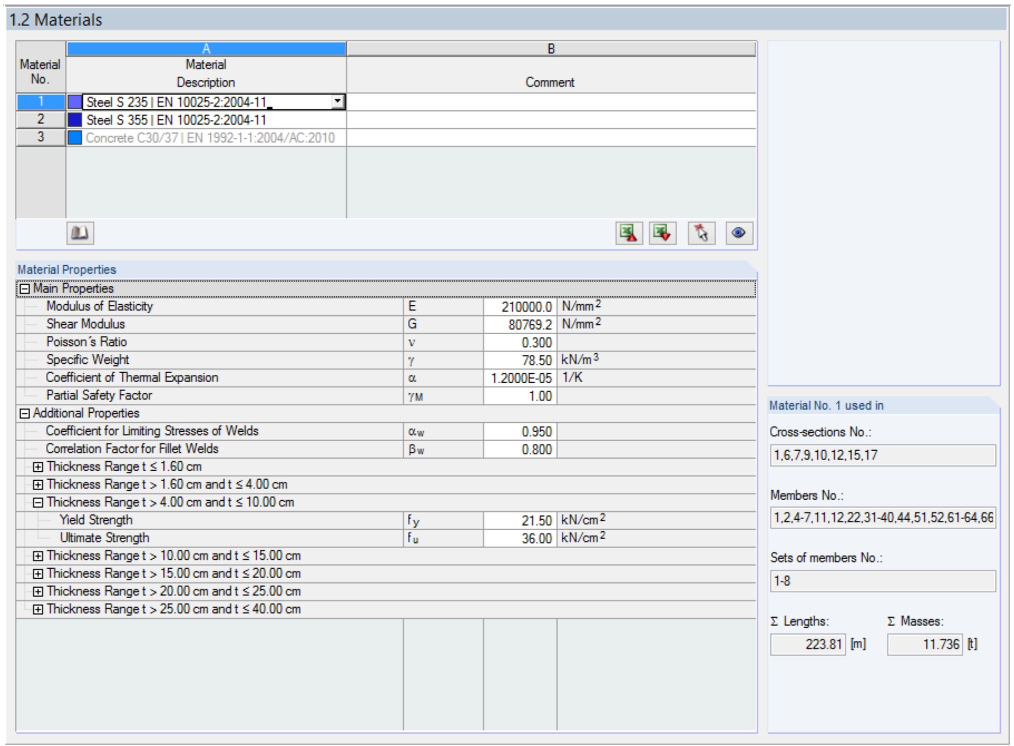


Figure 2.15 Window *1.2 Materials*

Materials that won't be used in the design are grayed out. Materials that are not allowed are highlighted in red. Modified materials are displayed in blue.

Chapter 4.3 of the RFEM manual, or Chapter 4.2 of the RSTAB manual, describes the material properties that are used for the determination of the internal forces (*Main Properties*). The properties of the materials that are required for the design are also stored in the global material library. These values are preset (*Additional Properties*).

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

Material Description

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

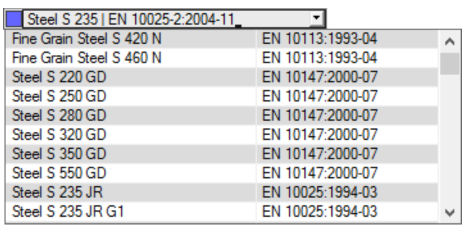


Figure 2.16 List of materials

According to the design concept of the Standard [1], only materials of the *Steel* category can be selected.

After the material transfer, the design-relevant *Material Properties* are updated.

If you change the material description manually and the new entry is already listed in the material library, RF-/STEEL EC3 will import the material properties as well.

The material properties are generally not editable in the RF-/STEEL EC3 add-on module.

Material Library

Many materials are stored in the database. To open the material library, click on the module menu

Edit → Material Library

or use the button shown on the left.

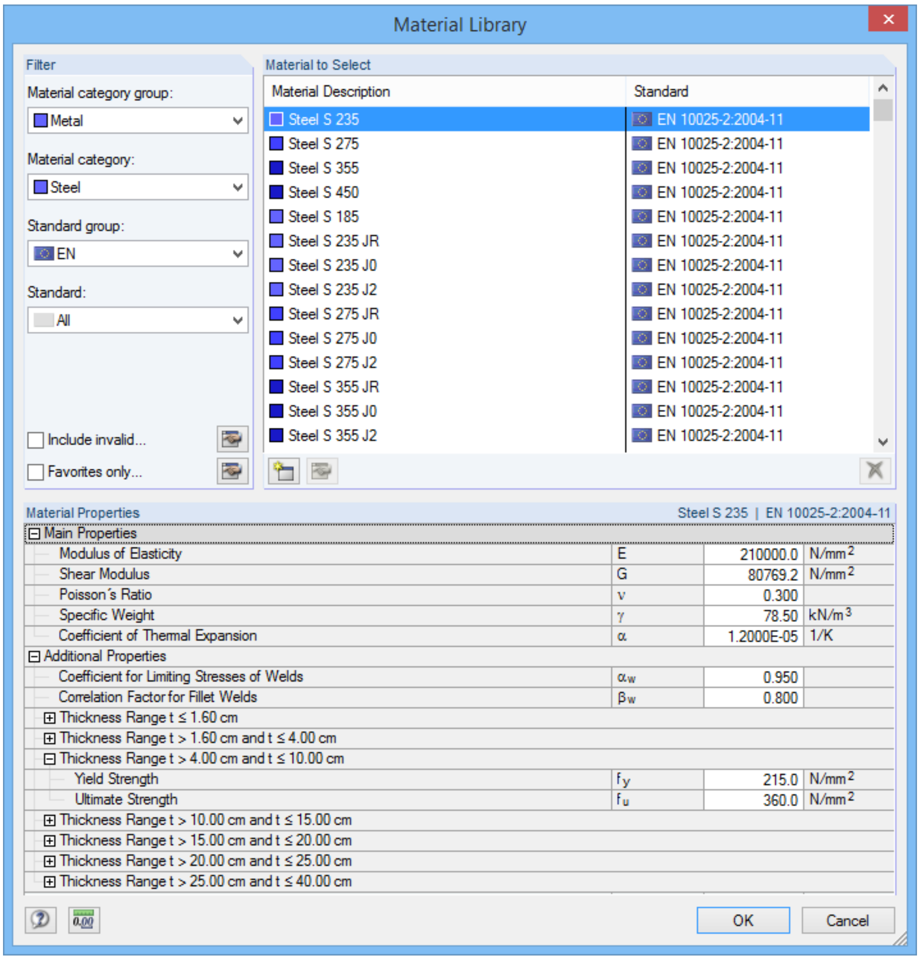


Figure 2.17 Dialog box *Material Library*

The *Steel* material category is preset in the *Filter* section. You can select the desired material grade from the *Material to Select* list; then you can check the properties in the dialog section below.

OK

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

Chapter 4.3 of the RFEM manual, or Chapter 4.2 of the RSTAB manual, describes how to filter, add, or reorganize materials.

In the library, you can also select materials of the categories *Cast Iron* and *Stainless Steel*. However, please check 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 the design. In addition, you can specify optimization parameters.

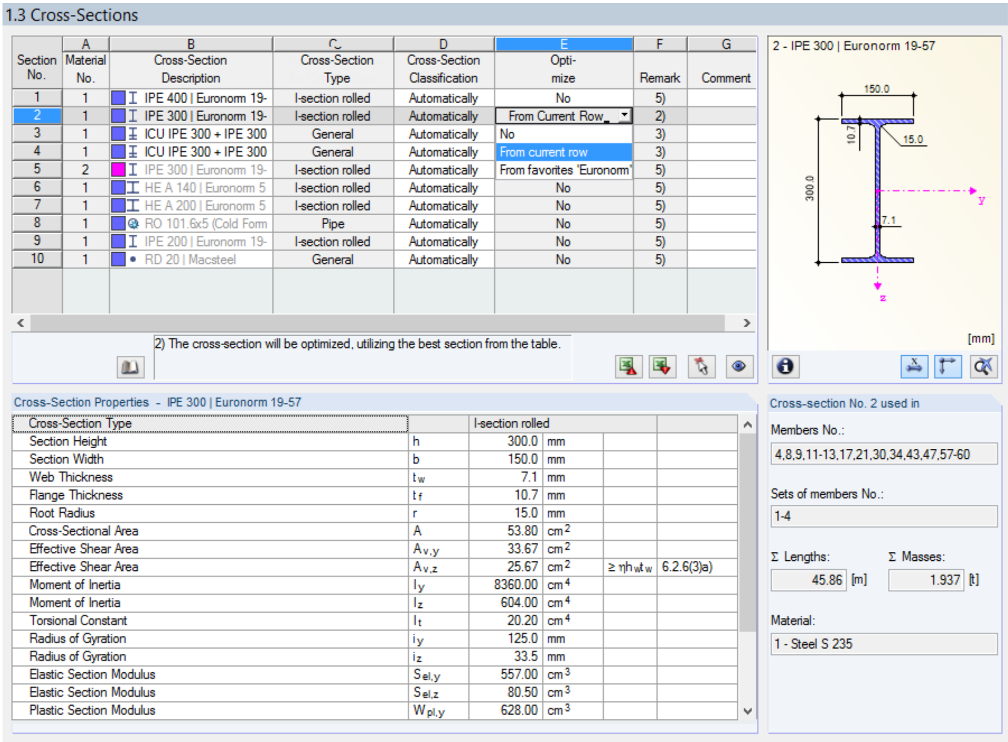


Figure 2.18 Window *1.3 Cross-Sections*

Cross-Section Description

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

To modify a cross-section, click the entry in column B. Thus, you set the field active. Then, open the cross-section table of the current input field by clicking the [Cross-Section Library] button or at the end of the box. You can also use the function key [F7] (see Figure 2.19).

In this dialog box, you can choose a different cross-section or even a different cross-section table. If you want to select a completely different cross-section category, click the [Back to Cross-Section Library] button. Then, the general cross-section library opens.

Chapter 4.13 of the RFEM manual, or Chapter 4.3 of the RSTAB manual, describes how to select cross-sections from the library.

You can also enter a new cross-section description directly into the input field in column B. If the entry is already listed in the database, RF-/STEEL EC3 will import the cross-section properties. A modified cross-section is highlighted in blue.

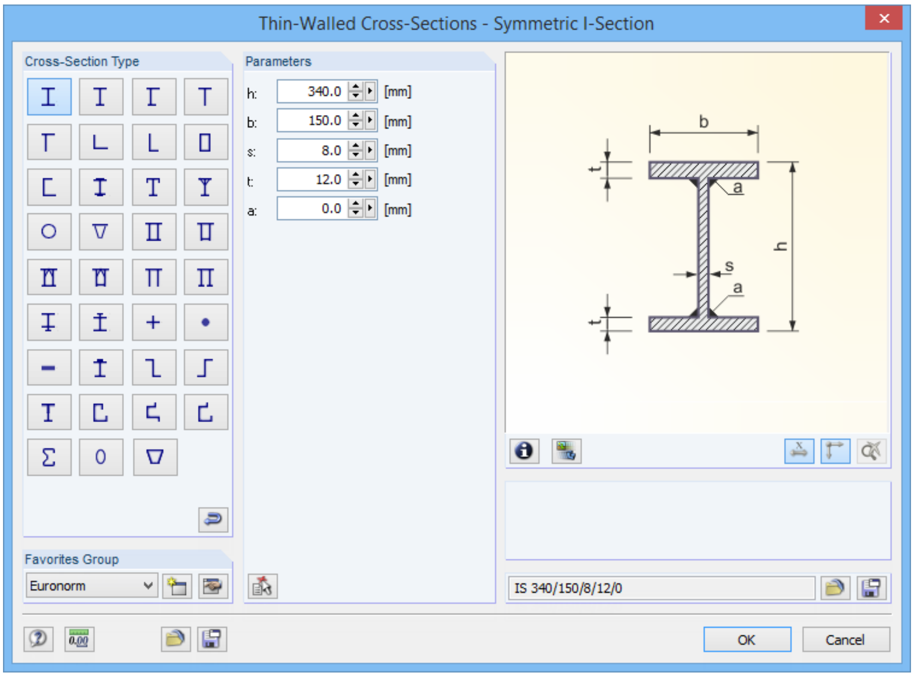
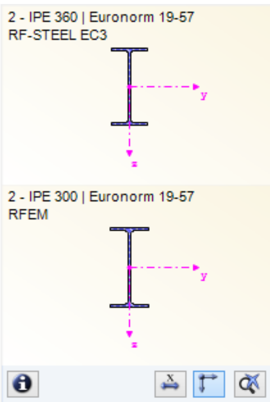


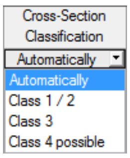
Figure 2.19 IS cross-section types of cross-section library



If cross-sections set in RF-/STEEL EC3 are different from the ones used in RFEM or RSTAB, both cross-sections are displayed in the graphic to the right. The designs will be performed with the internal forces from RFEM or RSTAB for the cross-section selected in RF-/STEEL EC3.

Cross-Section Type

This column shows the cross-section type that is 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, which means class 3 or 4.



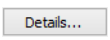
Classification

RF-/STEEL EC3 performs the classification *Automatically*. If this is not desired, you can define the cross-section class manually in the drop-down list. For example, an I-section embedded in concrete cannot buckle locally. By classifying it manually into class 3 it is possible to perform the design without taking into account the effective widths.

Max. Design Ratio

This column is displayed only after the calculation. It is intended to be a decision support for the optimization: Looking at the design ratios and colored relation scales, you can clearly see which cross-sections are hardly utilized and thus oversized, or extremely stressed and thus undersized.

Optimize



Each cross-section of the library can pass through an optimization process: For the internal forces from RFEM or RSTAB, the program searches the cross-section that comes as close as possible to a user-defined maximum ratio that can be defined in the *General* tab of the *Details* dialog box (see Figure 3.14).

To optimize a cross section, open the drop-down list in column E or F, and select the relevant entry: *From current row* or, if available, *From favorites 'Description'*. Recommendations for optimizing cross-sections can be found in Chapter 7.2.

Remark

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

If the warning *Incorrect type of cross-section!* appears before calculating, a cross-section is set which is not listed in the database. This may be a user-defined cross-section or a SHAPE-THIN cross-section that has not yet been calculated. To select an appropriate cross-section for the design, click the [Library] button (see description below Figure 2.18).

Member with tapered cross-section

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

RF-/STEEL EC3 also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the end. Normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the cross-sections at the start and end of a tapered member have different numbers of stress points, the intermediate values cannot be interpolated. The calculation is neither possible in RFEM or RSTAB nor in RF-/STEEL EC3.

The cross-section's stress points including numbering can be checked graphically: Select the cross-section in Window 1.3, and then click the [Info] button. The dialog box shown in Figure 2.20 appears.

Info About Cross-Section

Below the cross-section graphic, you find the [Info] button. Click it to open the *Info About Cross-Section* dialog box where you can see the cross-section properties, stress points and c/t-parts.

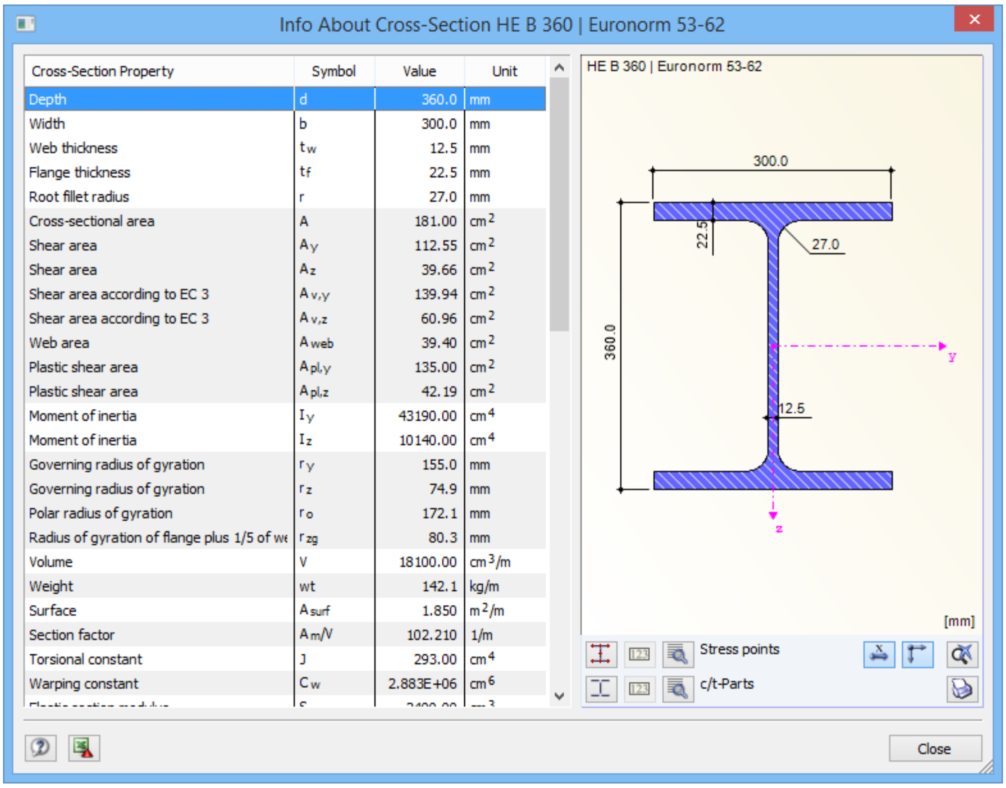


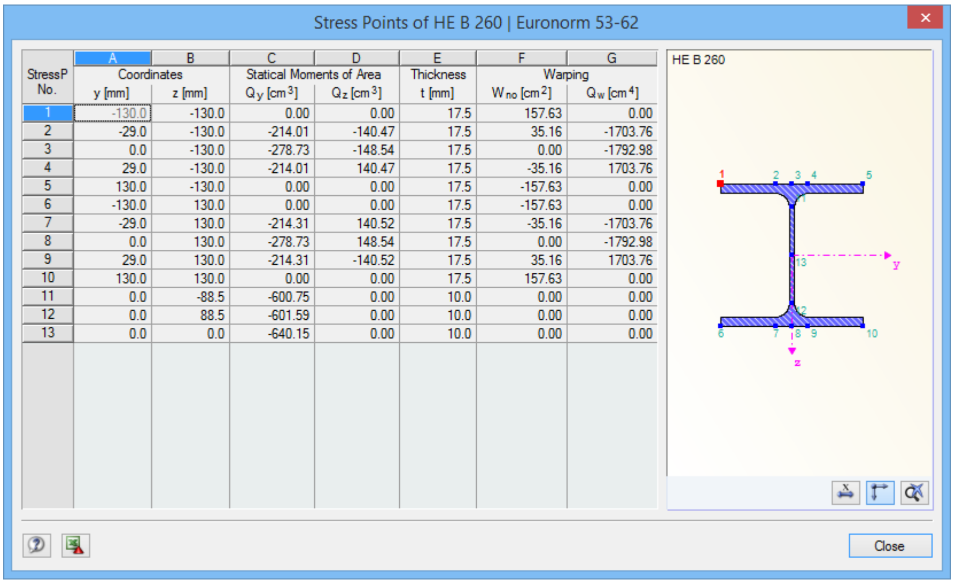
Figure 2.20 Dialog box *Info About Cross-Section*

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

Button	Function
	Displays or hides stress points
	Displays or hides c/t-parts
	Displays or hides numbers of stress points or c/t-parts
	Shows details of stress points or c/t-parts (see Figure 2.21)
	Displays or hides dimensions of cross-section
	Displays or hides principal axes of cross-section
	Resets full view of cross-section

Table 2.3 Buttons of cross-section graphic

Use the [Details] buttons to call up specific information about stress points (centroid distances, statical moments of area, warping ordinates, etc.) and c/t-parts.


Stress Points of HE B 260 | Euronorm 53-62

StressP No.	A Coordinates y [mm]	B Coordinates z [mm]	C Statical Moments of Area Q_y [cm³]	D Statical Moments of Area Q_z [cm³]	E Thickness t [mm]	F Warping W_{no} [cm²]	G Warping Q_w [cm⁴]
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

Figure 2.21 Dialog box *Stress Points of HE B 260*

Buckling curves of cross-section

If the situation requires it, you can change the buckling curves in the *Cross-Section Properties* table (lower part of Window *1.3 Cross-Sections*).

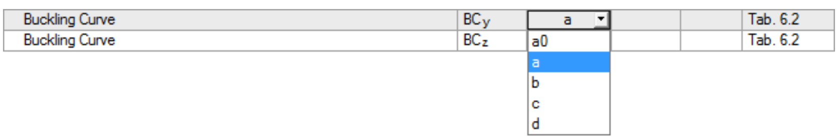


Figure 2.22 Changing buckling curve BC_y

2.4 Intermediate Lateral Restraints

In Window 1.4, you can define lateral supports for members. RF-/STEEL EC3 always assumes these intermediate supports to be perpendicular to the cross-section's minor axis z (see Figure 2.20). Thus, it is possible to influence the members' effective lengths (only for *Lateral and torsional* restraint type) which are important for the stability analyses concerning flexural buckling and lateral-torsional buckling.

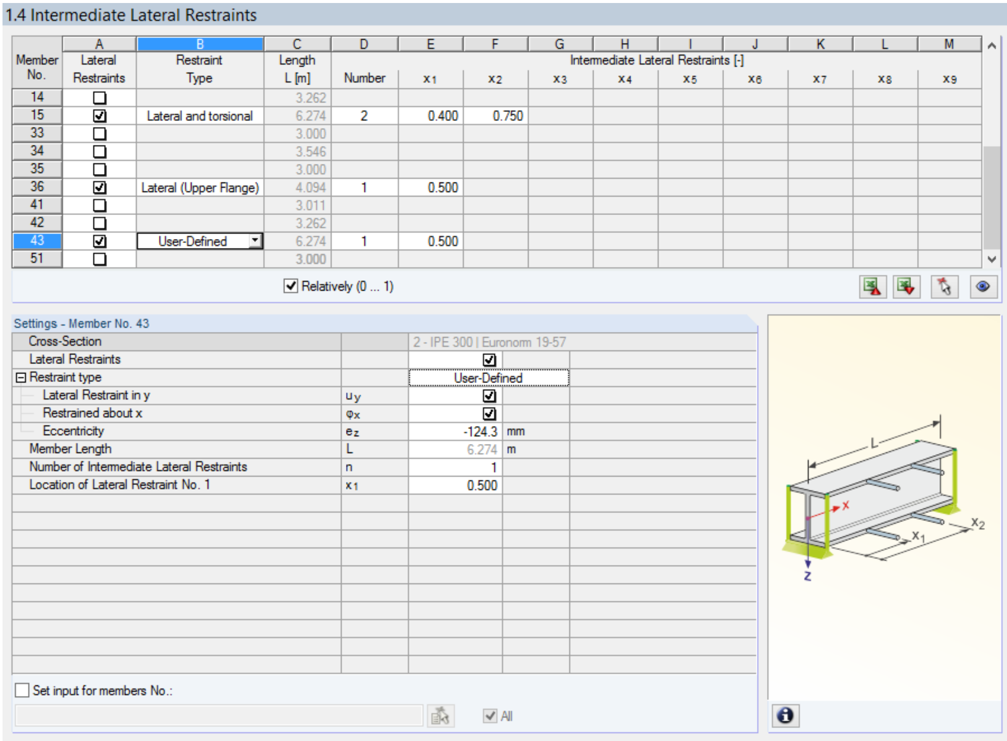
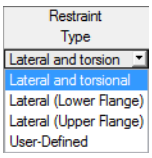


Figure 2.23 Window *1.4 Intermediate Lateral Restraints*

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

To define the intermediate restraints for a member, select the check box for *Lateral Restraints* in column A. With the button you can select the member graphically to activate its row in the table. When the check box has been selected, the fields where you can enter the parameters are available.



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

In column D, you can define the *Number* of intermediate restraints. Depending on the setting, you can access one or more of the following *Intermediate Lateral Restraints* columns for defining the x-locations.

☑ Relatively (0 ... 1)

When the check box for *Relatively (0 ... 1)* is selected, you must define the support points by relative input: The locations of the intermediate supports result from the member length and the relative distances to the member start. It is also possible to define the distances manually in the table, if the *Relatively (0 ... 1)* check box is cleared.

Set of members

If sets of members are designed according to the equivalent member method (see Chapter 3.1.2), supports of the lateral and torsional restraint type within the set of members must be defined here. Such restrained regions must **not** be described by effective lengths in Window 1.6!

You can find an example of using intermediate lateral restraints in our Knowledge Base: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001557

2.5 Effective Lengths - Members

This module window is subdivided into two parts. The table in the upper part lists the buckling length factors and equivalent member lengths for buckling and lateral-torsional buckling of all members to be designed. The effective lengths defined in RFEM or RSTAB are preset. In the *Settings* section, you can see additional information about the member whose table row is selected in the upper part.

With the button you can select a member graphically to activate its row in the table.

Changing entries is possible in the table as well as the *Settings* tree.

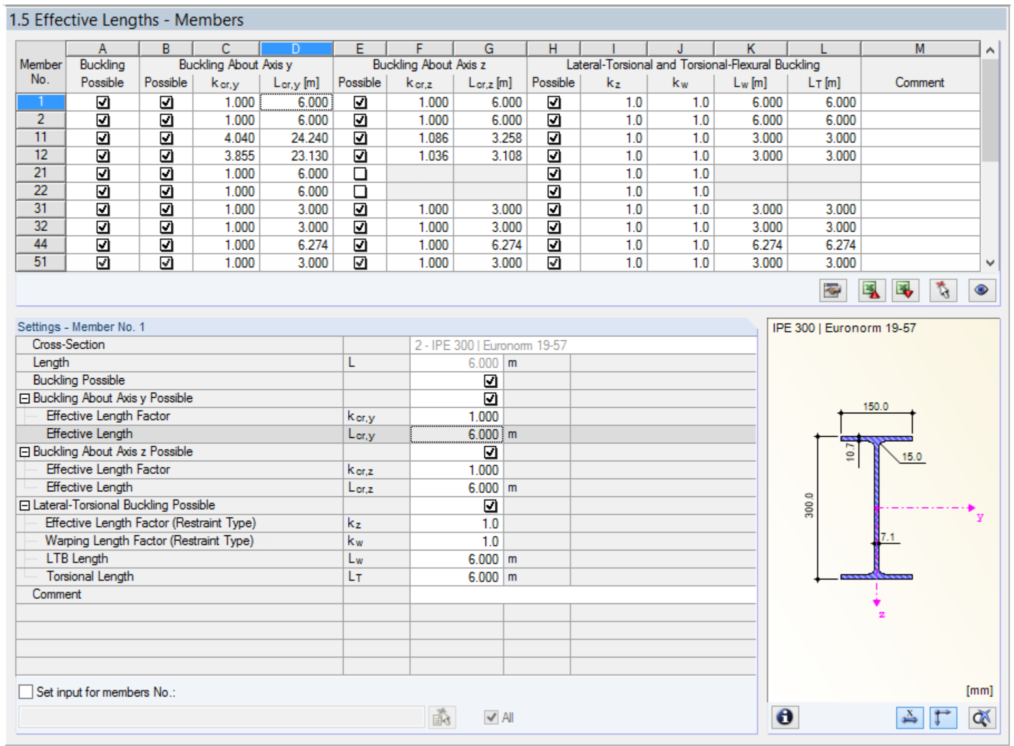


Figure 2.24 Window *1.5 Effective Lengths - Members*

The effective lengths for buckling about the minor axis z are aligned automatically with Window *1.4 Intermediate Lateral Restraints*. If the intermediate supports divide the member into segments of different lengths, no values are displayed in columns G, K, and L of Window 1.5.

You can enter the effective lengths manually in the table and the *Settings* tree. You can also define them graphically in the work window by using the button. This button becomes active when the cursor is placed in the text box (see Figure 2.24).

The *Settings* tree includes the following parameters:

- *Cross-Section*
- *Length* of member
- *Buckling Possible* for member (corresponds to columns B, E, and H)

- *Buckling About Axis y* (corresponds to columns C and D)
- *Buckling About Axis z* (corresponds to columns F and G)
- *Lateral-Torsional Buckling* (corresponds to columns I to K)

For the selected member, you can define whether a buckling or a lateral-torsional buckling analysis is generally to be carried out. In addition, you can adjust the *Effective Length Factor* and the *Warping Length Factor* for the respective directions. When changing a factor, the equivalent member length will be adjusted automatically, and vice versa.

It is also possible to define the effective length of a member in a dialog box that you open with the button [Select effective length factor]. You can find the button below the table.

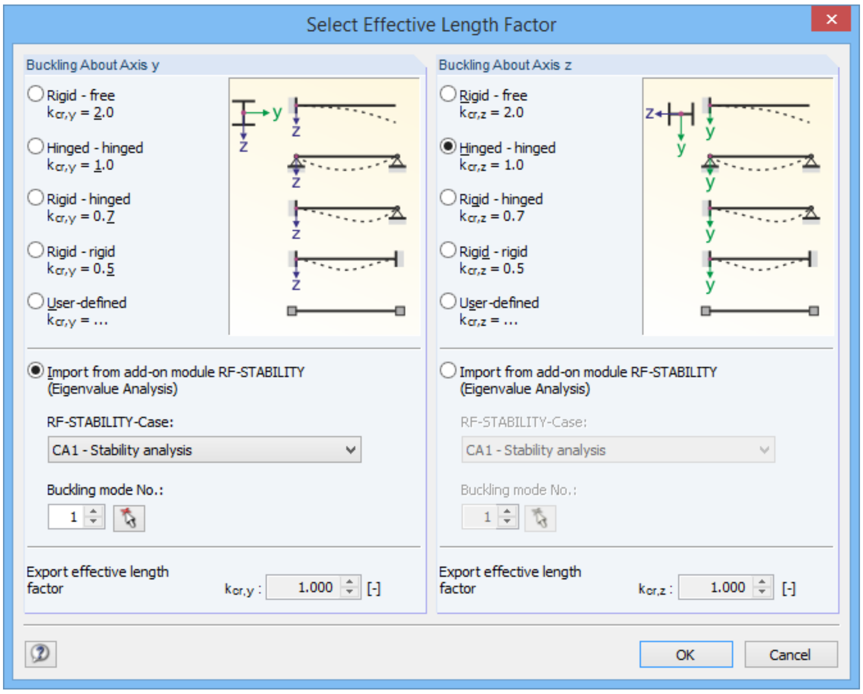


Figure 2.25 Dialog box *Select Effective Length Factor*

For each direction, you can select one of the four Euler buckling modes. You can also set a *User-defined* effective length factor. If an eigenvalue analysis has been carried out by the RF-STABILITY or RSBUCK add-on module, it is also possible to define a *Buckling mode* for the determination of the factor.

Buckling Possible

The stability analyses for flexural and lateral-torsional buckling require the ability to absorb compressive forces. Therefore, members for which such an absorption is not possible due to the member type (for example, tension members, elastic foundations, rigid connections) are excluded from the outset. The rows are grayed out in the table, and a corresponding note is shown in the *Comment* column.

The *Buckling Possible* check boxes in table row A and in the *Settings* tree offer a control option for the stability analyses: They determine if these analyses are performed or omitted for the member.

Buckling About Axis y / Buckling about Axis z

With the check box in the *Possible* column, you decide if a member has the risk of buckling about the axis y and/or z. These axes represent the local member axes, with axis y being the "major" and axis z the "minor" member axis. The effective length factors $k_{cr,y}$ and $k_{cr,z}$ for buckling about the major or minor axis can be selected freely.

In the *1.3 Cross-Sections* window, you can check the position of the member axes in the cross-section graphic (see Figure 2.18). With the [Jump to graphic] button you can also access the work window of the main program. There, you can display the local member axes by using the member's shortcut menu or the *Display* navigator.

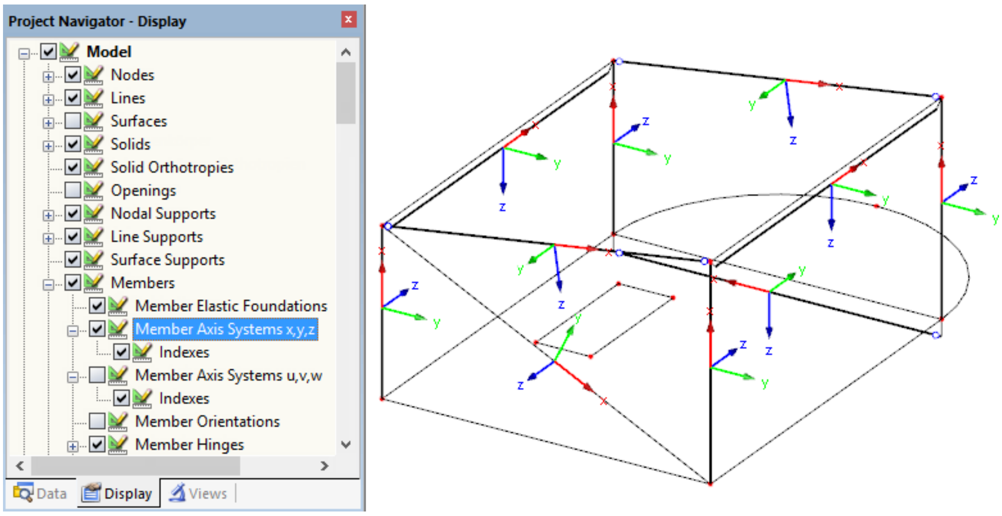


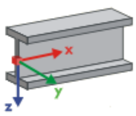
Figure 2.26 Activating the member axis systems in *Display* navigator of RFEM

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

To define the effective lengths graphically in the work window, use the button. This button becomes available when the cursor is placed in a L_{cr} input field (see Figure 2.24).

When you specify the effective length factor k_{cr}, the program determines the effective length L_{cr} by multiplying the member length *L* by this factor. The input fields k_{cr} and L_{cr} are interactive.

Lateral-Torsional and Torsional-Flexural Buckling Possible

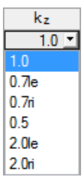


Axis definition for k_z and k_w

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

To determine M_{cr} by the eigenvalue calculation method, an internal member model with four degrees of freedom is created. These degrees must be defined by the factors k_z and k_w. With both factors interacting, it is possible to determine the support conditions for lateral-torsional buckling (for example, lateral and torsional restraint).

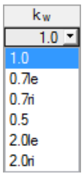
Effective Length Factor k_z



The factor k_z controls the lateral displacement u_y and the rotation φ_z at the member ends.

- k_z = 1.0 restrained against lateral displacement u_y on both member ends
- k_z = 0.7le restrained against displacement u_y on both ends and restraint about z left
- k_z = 0.7ri restrained against displacement u_y on both ends and restraint about z right
- k_z = 0.5 restrained against displacement u_y and restraint about z on both member ends
- k_z = 2.0le restrained against displacement u_y and restraint about z left; right end free
- k_z = 2.0ri restrained against displacement u_y and restraint about z right; left end free

Warping Length Factor k_w



The factor k_w controls the torsion about the member's longitudinal axis $_x$ and the warping ω.

- k_w = 1.0 restrained against rotation about x on both member ends; free to warp on both sides
- k_w = 0.7le restrained against rotation about x on both ends and warping restraint left
- k_w = 0.7ri restrained against rotation about x on both ends and warping restraint right
- k_w = 0.5 torsion and warping restraint on both member ends
- k_w = 2.0le restrained against rotation about x and warping ω left; right end free
- k_w = 2.0ri restrained against rotation about x and warping ω right; left end free

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

You can model a lateral and torsional restraint by using the factors k_z = 1.0 (support in y with free rotation about z) and k_w = 1.0 (restrained against torsion about x with free warping). Because the internal member model requires only four degrees of freedom, defining other boundary conditions is not necessary.

If the lateral-torsional buckling length L_w or the torsional buckling length L_T differs from the length of the member or the buckling length, you can define the lengths L_w and L_T also manually in columns K and L or graphically with the [...] button.

Comment

In the final column, you can enter user-defined notes to describe, for example, the equivalent member lengths.

Set input for members No.

Below the *Settings* table, you find the check box *Set input for members No.* When you select it, the **subsequent** settings will be applied to *All* members or to selected members (enter the member numbers manually or select them graphically with [button]). This option is useful if you want to assign the same boundary conditions to several members. You can find an example in the Knowledge Base on our website.

Settings which have already been defined cannot be changed subsequently with this function.

2.6 Effective Lengths - Sets of Members

Details...

This window appears only if at least one set of members is set for design in the *1.1 General Data* window and the *Equivalent Member Method* for sets of members is selected in the *Details* dialog box (see Figure 3.2). Then, Windows 1.7 and 1.8 are not displayed. In this case, lateral intermediate supports can be defined in Window 1.4 by using division points.

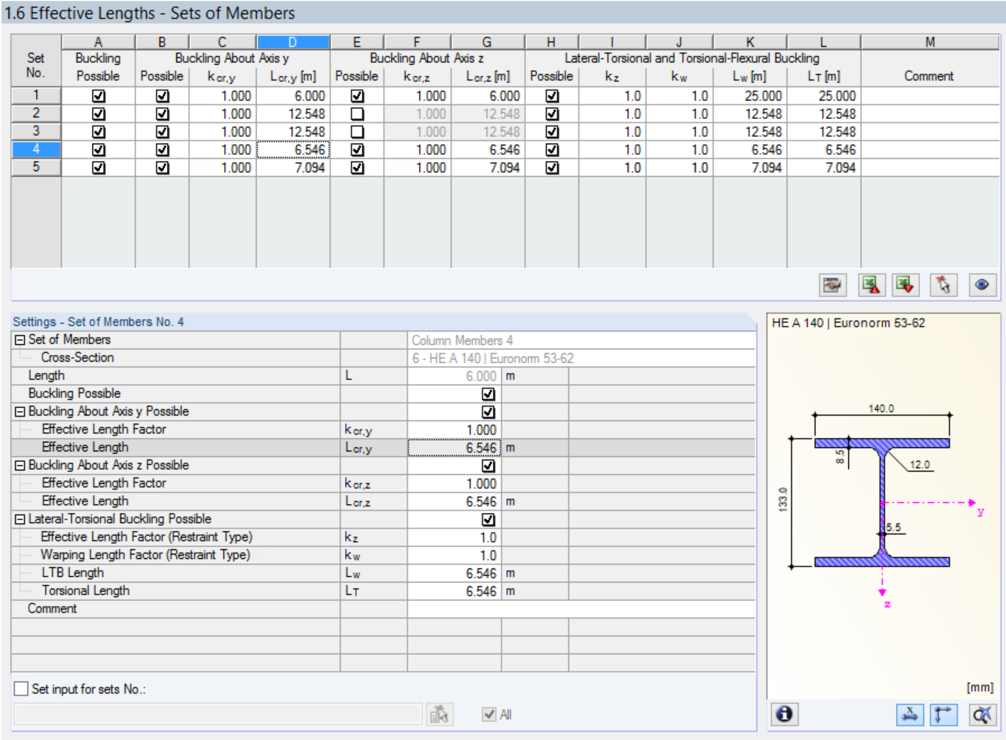


Figure 2.27 Window *1.6 Effective Lengths - Sets of Members*

The concept of this window is similar to the previous Window *1.5 Effective Lengths - Members*. Here, you can enter the effective lengths for buckling about both principal axes of the set of members, as described in Chapter 2.5. They define the boundary conditions of the set of members that is handled in its entirety as an equivalent member.

In our Knowledge Base, you can find an example of entering effective lengths for sets of members: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001557

2.7 Nodal Supports - Sets of Members

This window is displayed if at least one set of members is selected for design in window *1.1 General Data* and the stability analysis is performed as per the general method according to [1] clause 6.3.4 (default setting).

Details...

If the *Equivalent Member Method* for sets of members is selected in the *Details* dialog box (see Figure 3.2), Window 1.7 is not displayed. In this case, lateral intermediate supports can be defined in Window 1.4 by using division points.

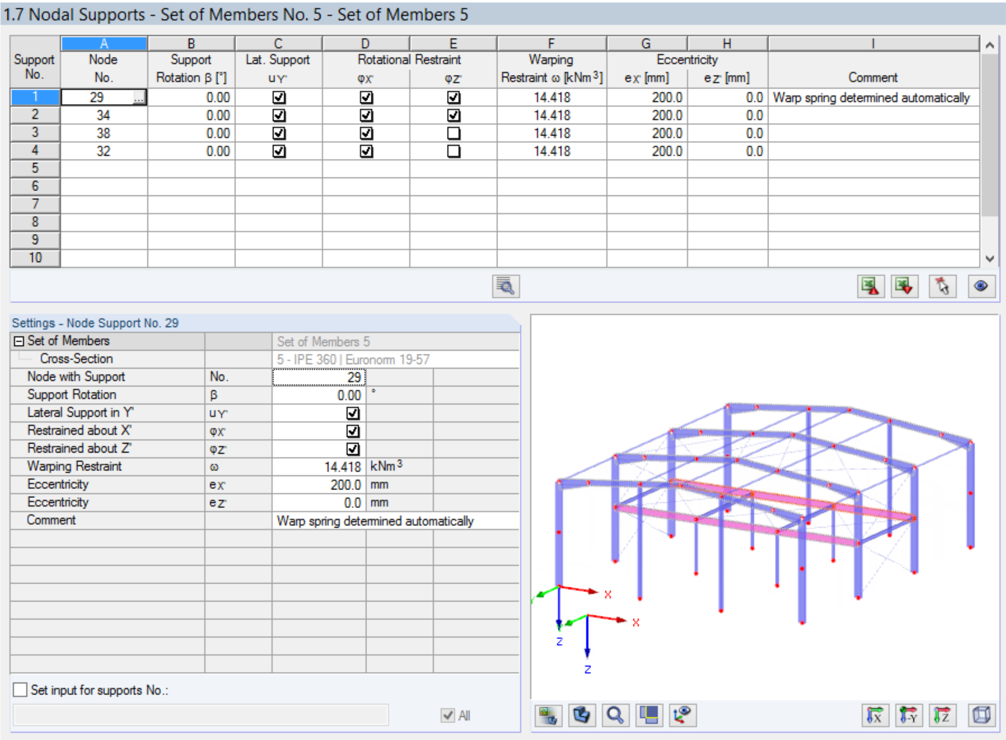


Figure 2.28 Window *1.7 Nodal Supports - Set of Members*

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

The supports defined in RFEM or RSTAB (for example, in Z for a continuous beam) are not relevant in this window: The distributions of moments and shear forces for the determination of the amplification factor are automatically imported from RFEM/RSTAB. Here, you define the support conditions affecting the stability failure (buckling, lateral-torsional buckling).

Supports on the start and end nodes of the set of members are preset. Any other supports, for example due to connected members, must be added manually. Use the button to select nodes graphically in the main program's work window.

According to [1] clause 6.3.4 (1), it is possible to design singly symmetric cross-sections that are stressed 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 orientation of the axes in the set of members is important for the nodal support definition. The program checks the position of the nodes and internally defines the axes of the nodal supports for Window 1.7 according to Figure 2.29 to Figure 2.32. The [Local Coordinate System] button below the model graphic can help you with the orientation: Use it to display the set of members in a partial view where the axes are clearly visible. You can find an example in the Knowledge Base on our website.

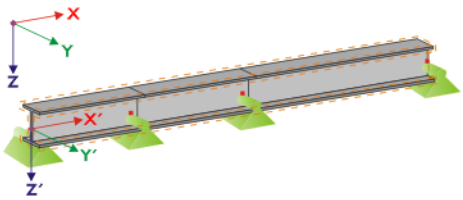


Figure 2.29 Auxiliary coordinate system for nodal supports - straight set of members

If all members of a set of members rest on a straight line, as shown in Figure 2.29, 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.

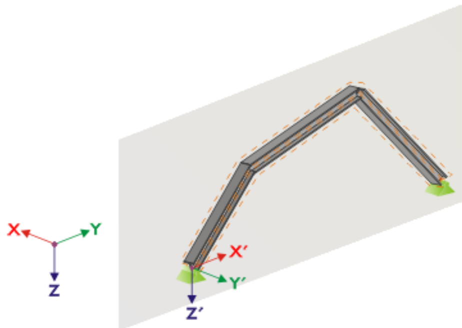


Figure 2.30 Auxiliary coordinate system for nodal supports - set of members in vertical plane

If the 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 2.30, the members rest in a vertical plane. In this case, the X'-axis is horizontal and oriented in the direction of the plane. The Y'-axis is horizontal as well and defined perpendicular to the X'-axis. The Z'-axis is oriented perpendicular downwards.

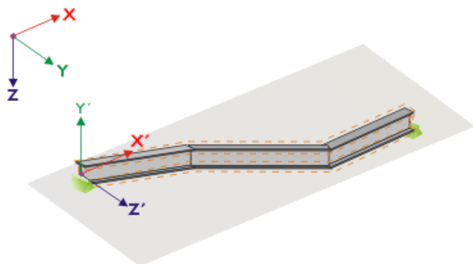


Figure 2.31 Auxiliary coordinate system for nodal supports - set of members in horizontal plane

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

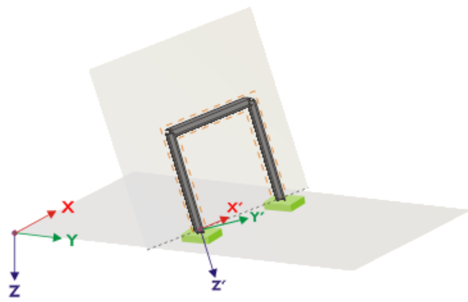


Figure 2.32 Auxiliary coordinate system for nodal supports - set of members in inclined plane

Figure 2.32 shows the general case of a buckled set of members: The members do not rest on a straight line, but in an inclined plane. The definition of the X'-axis results from the intersection line between the inclined and the horizontal plane. Thus, the Y'-axis is perpendicular to the X'-axis and in vertical position 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
	Shows model or system sketch
	Shows members as 3D rendering or wire-frame model
	Shows current set of members or entire model
	Displays irrelevant members of model as transparent or opaque
	Shows set of members with local coordinate system or entire model
	Shows view in direction of X-axis
	Shows view in opposite direction of Y-axis
	Shows view in direction of Z-axis
	Sets isometric view

Table 2.4 Buttons for cross-section graphic

With the [Edit warp stiffener] button it is possible to determine the constant of a warp spring by the program.

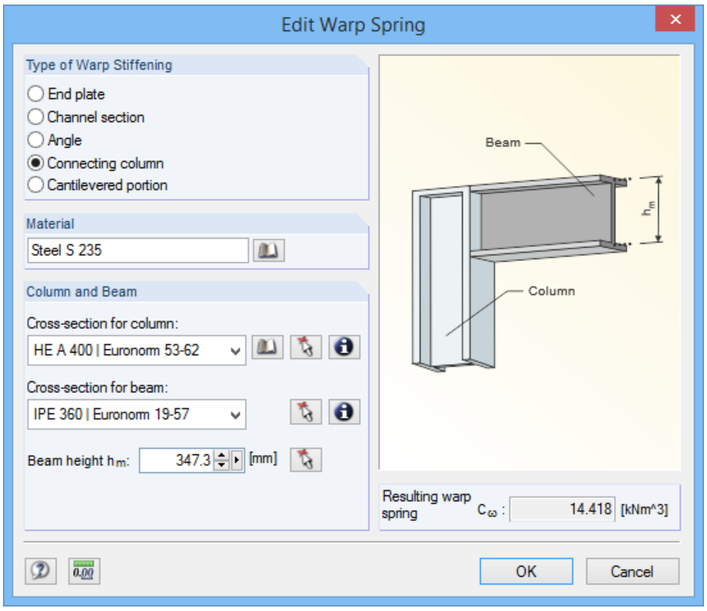


Figure 2.33 Dialog box *Edit Warp Spring*

The following warp stiffening types are available in the *Edit Warp Spring* dialog box:

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

Materials and cross-sections can be selected by using the lists and [Library] buttons. With the button you can select them also graphically in the RFEM/RSTAB model.

Based on the parameters, RF-/STEEL EC3 determines the *Resulting warp spring* C_ω which can then be imported with [OK] in Window 1.7.

Warping analysis with seven degrees of freedom

Details...

To analyze sets of members according to the second-order analysis for flexural-torsional buckling with warping torsion, select the corresponding check box in the *Warping Torsion* tab of the *Details* dialog box (see Figure 3.9). The table titles of Window 1.7 will be adjusted accordingly.

The warping analysis requires a license of the **RF-/STEEL Warping Torsion** module extension.

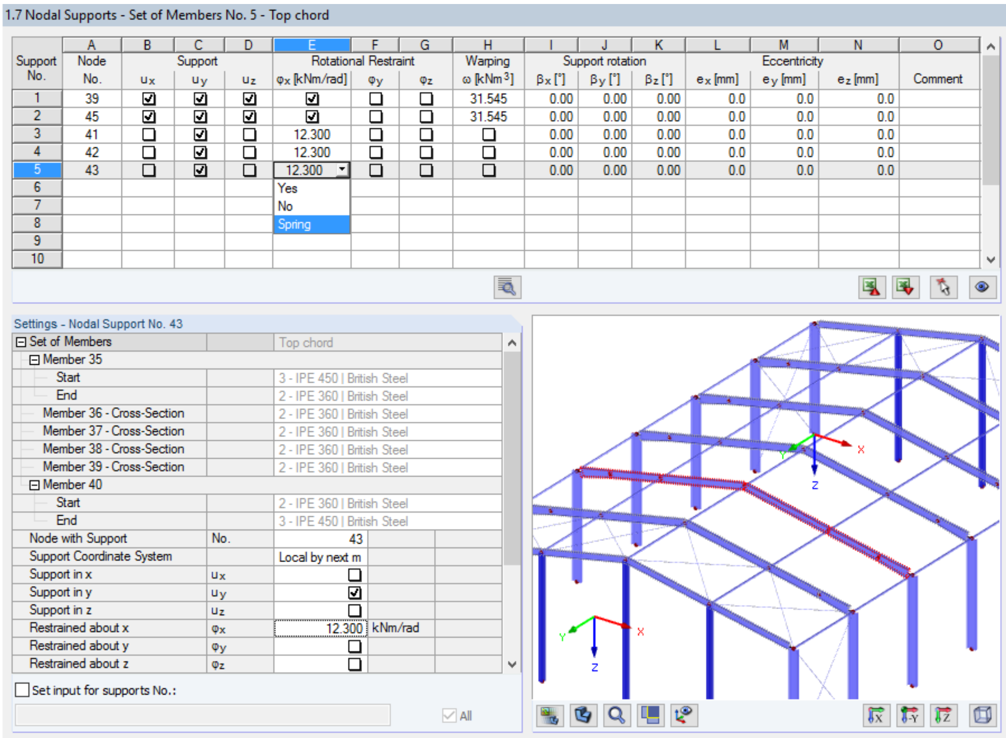


Figure 2.34 Window *1.7 Nodal Supports - Set of Members* for warping analysis with seven degrees of freedom

Here, you define the support conditions of the set of members singled out from the system, which are available on the nodes of the members involved. Nodal supports defined in RFEM or RSTAB are preset, as well as supports on both ends of the set of members.

Lateral supports of the set of members must be added in the form of additional supports. In this way, you can represent, for example, a purlin's effect given in the spatial model of RFEM or RSTAB. If this support is missing in the model of the singled out set of members, instabilities are possible.

With the button, the supported nodes can be selected graphically in the RFEM/RSTAB work window.

In columns B to N, you specify the support conditions of the selected nodes. To activate or deactivate the supports and restraints for the corresponding degrees of freedom, click in the check boxes. Alternatively, you can enter the constants of the translational and rotational springs manually.

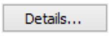
The *Support rotation* and *Eccentricity* parameters allow for modeling support conditions close to reality.

You can find an example describing the warping torsion analysis of a tapered single-span beam in an article of our Knowledge Base on our website.

The warping analysis of a frame is also the topic of a webinar, which you can watch or download on Youtube.

2.8 Member Hinges - Sets of Members

This window is displayed if at least one set of members is selected for design in the *1.1 General Data* window. Here, you can define hinges for members within the set of members that, for structural reasons, don't transfer the degrees of freedom locked in Window 1.7 as internal forces. Make sure that no double hinges are generated in coaction with Window 1.7.



Window 1.8 is not displayed if the *Equivalent Member Method* for sets of members is selected in the *Details* dialog box (see Figure 3.2).

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

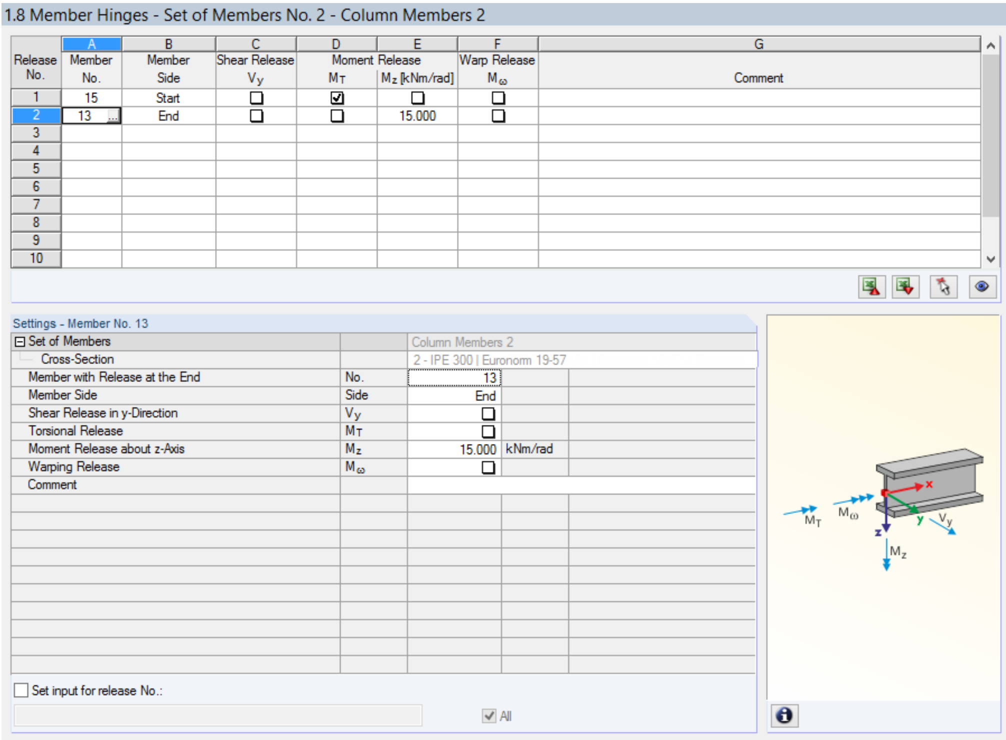
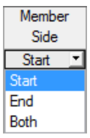


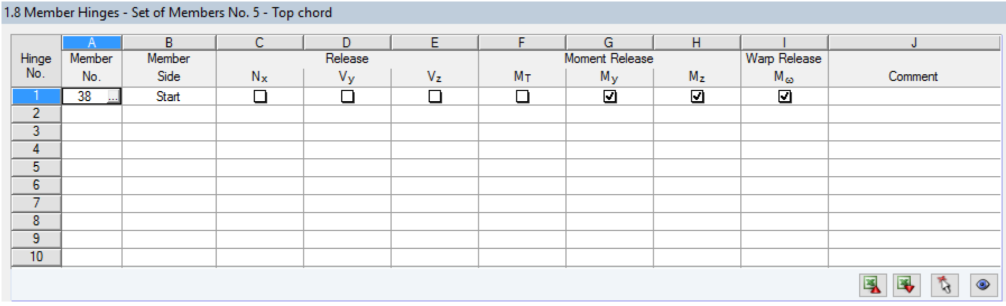
Figure 2.35 Window *1.8 Member Hinges - Set of Members*



In column B, you specify the *Member Side* where the hinge is located, or if there are hinges on both member sides.

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

If the warping analysis with seven degrees of freedom is selected in the *Warping Torsion* tab of the *Details* dialog box (requires license of **RF-/STEEL Warping Torsion** module extension), the columns are extended by corresponding input options.



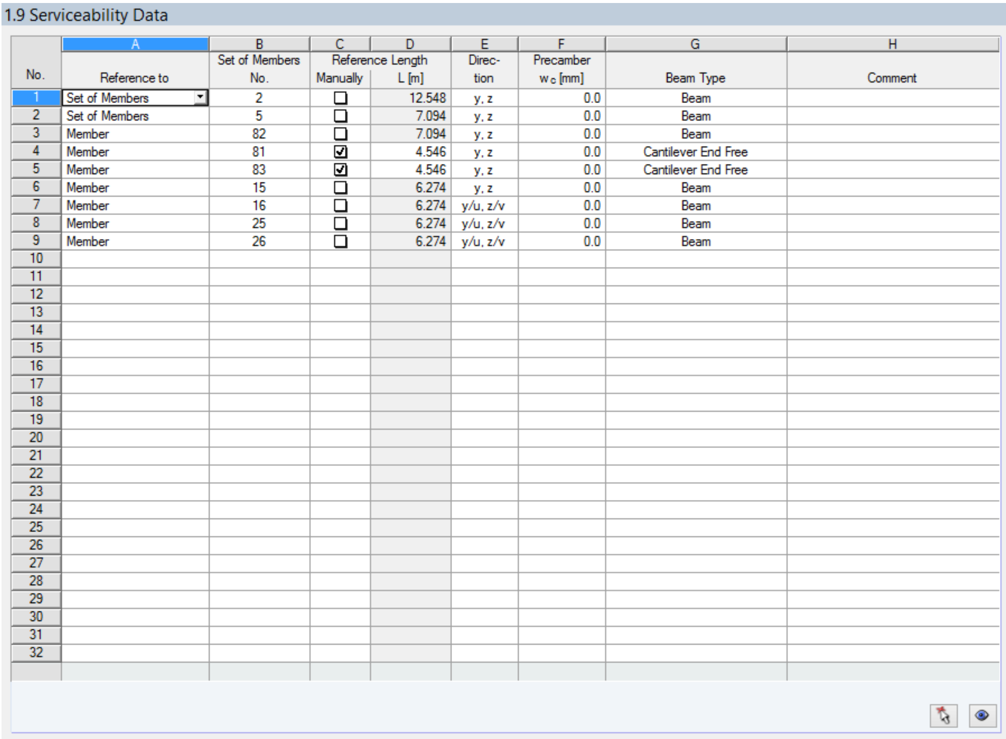
1.8 Member Hinges - Set of Members No. 5 - Top chord

Hinge No.	A Member No.	B Member Side	C Release Nx	D Release Vy	E Release Vz	F Moment Release MT	G Moment Release My	H Moment Release Mz	I Warp Release Mω	J Comment
1	38	Start	☐	☐	☐	☐	☑	☑	☑	
2										
3										
4										
5										
6										
7										
8										
9										
10										

Figure 2.36 Window 1.8 *Member Hinges - Set of Members* for warping analysis with seven degrees of freedom

2.9 Serviceability Parameters

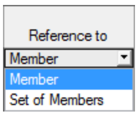
This window controls various settings for the serviceability limit state design. It is displayed if corresponding data has been set in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2).



1.9 Serviceability Data

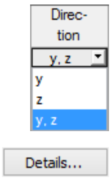
No.	A Reference to	B Set of Members No.	C Reference Length Manually	D Reference Length L [m]	E Direction	F Precamber wc [mm]	G Beam Type	H Comment
1	Set of Members	2	☐	12.548	y, z	0.0	Beam	
2	Set of Members	5	☐	7.094	y, z	0.0	Beam	
3	Member	82	☐	7.094	y, z	0.0	Beam	
4	Member	81	☑	4.546	y, z	0.0	Cantilever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

Figure 2.37 Window 1.9 *Serviceability Data*



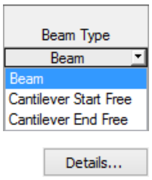
In column A, you decide whether the deformation refers to single members or sets of members. For a set of members, it is necessary that a uniform member orientation and rotation of all included members is given. Only in this way, the deformation components will be determined correctly.

In column B, you enter the numbers of the members or sets of members that you want to design. You can also use the button to select them graphically in the main program's work window. Then, the *Reference Length* appears automatically in column D. The column presets the lengths of the members and sets of members. You can adjust the values *Manually* after having selected the check box in column C.



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

In column F, a *Precamber* can be taken into account. The precamber's general direction is defined in the *Serviceability* tab of the *Details* dialog box (see Figure 3.4). If the precamber is related to the "major" principal axis y or u, the column title changes to $w_{c,y}$ or $w_{c,u}$. A positive precamber must be entered, if it is directed **against** the local member axis z (as a rule when global axis Z is downwards oriented). The precamber is only considered for quasi-permanent design situations (see Chapter 2.1.2)!



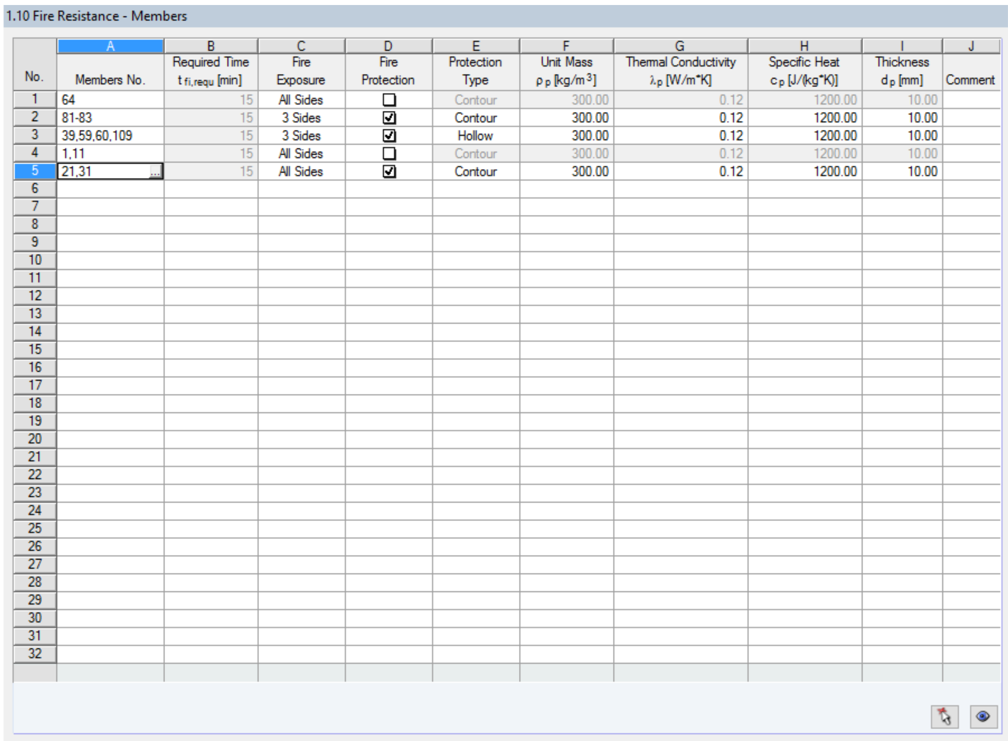
For a correct application of limit deformations, the *Beam Type* is also important. In column G, you can specify whether a beam or a cantilever is to be designed and which end is free of support.

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 3.4).

2.10 Fire Resistance - Members

This window manages the fire protection parameters for members. It is displayed if corresponding data has been set in the *Fire Resistance* tab of Window 1.1 (see Chapter 2.1.3).

The fire resistance design performed with RF-/STEEL EC3 is described in the following article of the Knowledge Base on our website: In Chapter 8.2, you can find an example describing the fire protection design of a steel column.



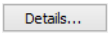
1.10 Fire Resistance - Members

No.	A Members No.	B Required Time $t_{fi,requ}$ [min]	C Fire Exposure	D Fire Protection	E Protection Type	F Unit Mass ρ_p [kg/m³]	G Thermal Conductivity λ_p [W/m*K]	H Specific Heat c_p [J/(kg*K)]	I Thickness d_p [mm]	J Comment
1	64	15	All Sides	☐	Contour	300.00	0.12	1200.00	10.00	
2	81-83	15	3 Sides	☑	Contour	300.00	0.12	1200.00	10.00	
3	39,59,60,109	15	3 Sides	☑	Hollow	300.00	0.12	1200.00	10.00	
4	1,11	15	All Sides	☐	Contour	300.00	0.12	1200.00	10.00	
5	21,31	15	All Sides	☑	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 2.38 Window 1.10 *Fire Resistance - Members*

In column A, you decide for which members you want to perform a fire resistance design. With the [...] button you can select the members graphically in the work window of RFEM or RSTAB.

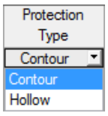
Designs are only possible for members that have been selected for design in the *1.1 General Data* window (see Figure 2.2).



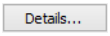
The *Required time* $t_{fi,requ}$ of the fire resistance is set according to the specification in the *Details* dialog box (see Figure 3.5). If the option *Define individually for each member* is set there, the fields of column B become accessible for user-defined fire durations.

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

The following article in the Knowledge Base describes how the shadowing effect can be considered:



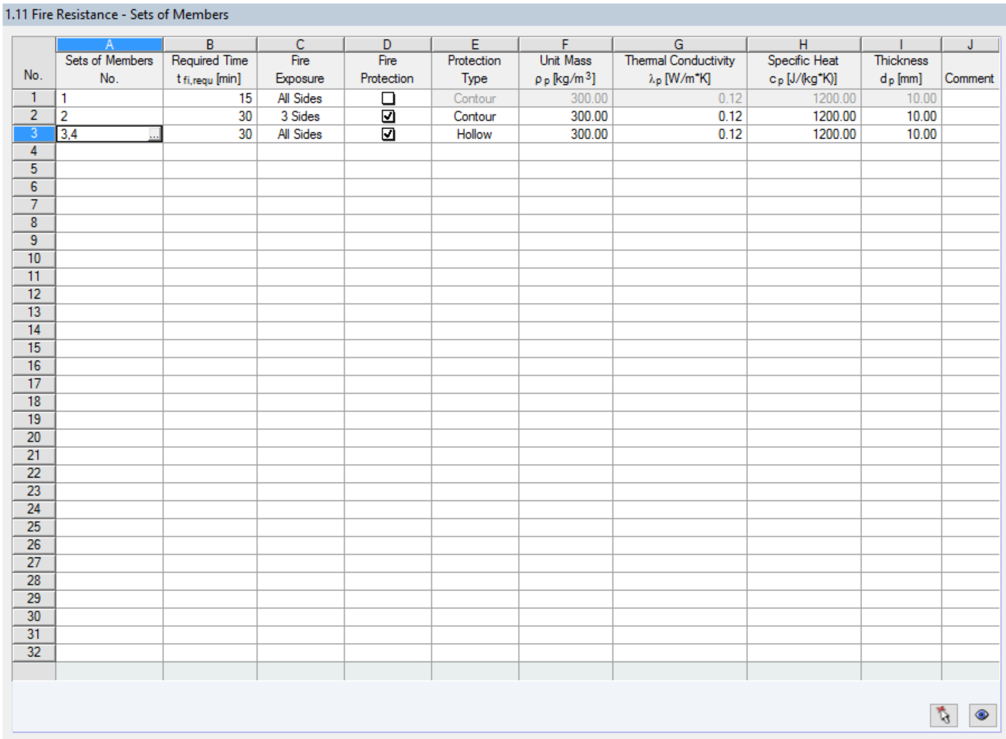
If there is an encasement for fire protection, you can select the *Protection Type* in column E. You can choose between a contour-type coating that follows the geometry of the cross-section (plaster or panel coatings, for example) and a hollow-type board encasement of the cross-section. Then, the parameters must be specified in columns F to I.



The general parameters for the fire resistance design are managed in the *Fire Resistance* tab of the *Details* dialog box (see Figure 3.5).

2.11 Fire Resistance - Sets of Members

This window manages the fire protection parameters for sets of members. It is displayed if at least one set of members is selected for design in the *1.1 General Data* window and corresponding data is set in the *Fire Resistance* tab (see Chapter 2.1.3).



1.11 Fire Resistance - Sets of Members

No.	A Sets of Members No.	B Required Time $t_{fi,requ}$ [min]	C Fire Exposure	D Fire Protection	E Protection Type	F Unit Mass ρ_p [kg/m³]	G Thermal Conductivity λ_p [W/m*K]	H Specific Heat c_p [J/(kg*K)]	I Thickness d_p [mm]	J Comment
1	1	15	All Sides	☐	Contour	300.00	0.12	1200.00	10.00	
2	2	30	3 Sides	☑	Contour	300.00	0.12	1200.00	10.00	
3	3,4	30	All Sides	☑	Hollow	300.00	0.12	1200.00	10.00	
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 2.39 Window *1.11 Fire Resistance - Sets of Members*

The concept of the window is similar to the previous Window *1.10 Fire Resistance - Members*. Here, you can enter the fire protection parameters of the relevant 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 [3] clause 10.1 and 10.3).

The upper section lists the members selected for design together with the parameters affecting the lateral-torsional buckling analysis. The parameters are interactive with the specifications in the *Settings - Member No.* section below.

To the right of the *Settings* table, you can see information or options for selection in the form of a graphic facilitating the definition of boundary conditions. This graphic is aligned with the current parameter.

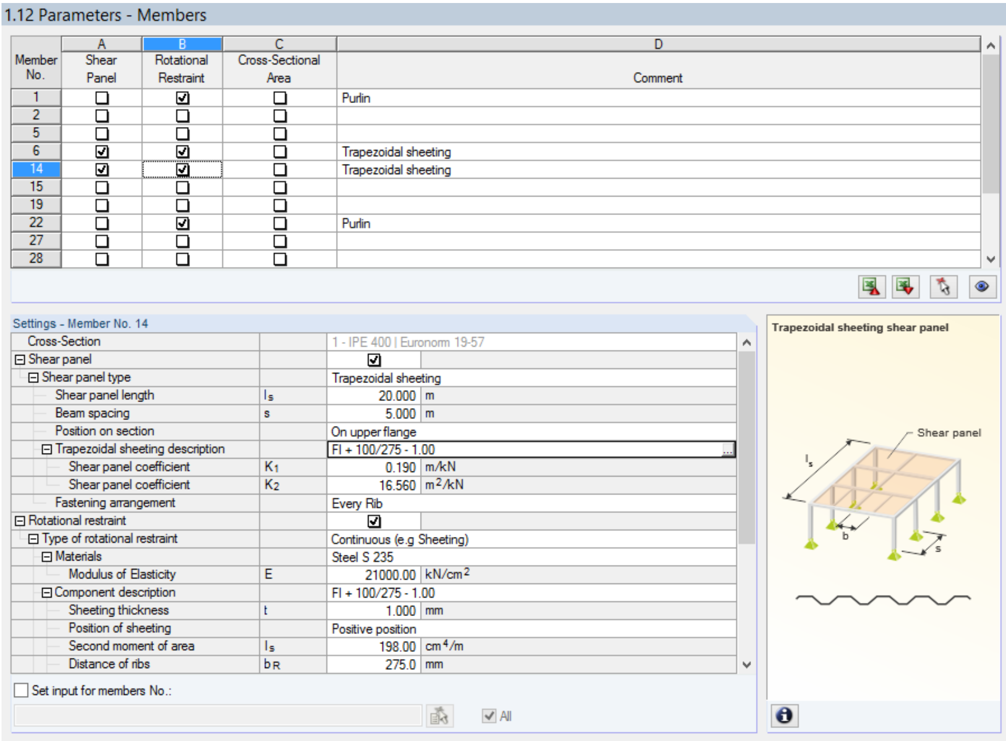


Figure 2.40 Window *1.12 Parameters - Members*

Below the *Settings* table, you find the check box *Set input for members No.* When you select it, the **subsequent** settings will be applied to *All* members or to selected members (enter the member numbers manually or select them graphically with [button]). This option is useful if you want to assign the same boundary conditions to several members.

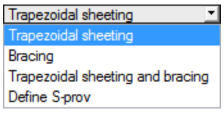
In the *Comment* column, you can enter user-defined notes, for example, to describe the parameters of a member, which are relevant for lateral-torsional buckling.

Cross-section

This row shows the cross-section description for information. For a tapered member, the descriptions of the start and end cross-section are displayed.

Shear Panel

To enter the shear panel parameters, select the check box in column A or the *Settings* table.



You can select the shear panel type in the drop-down list.

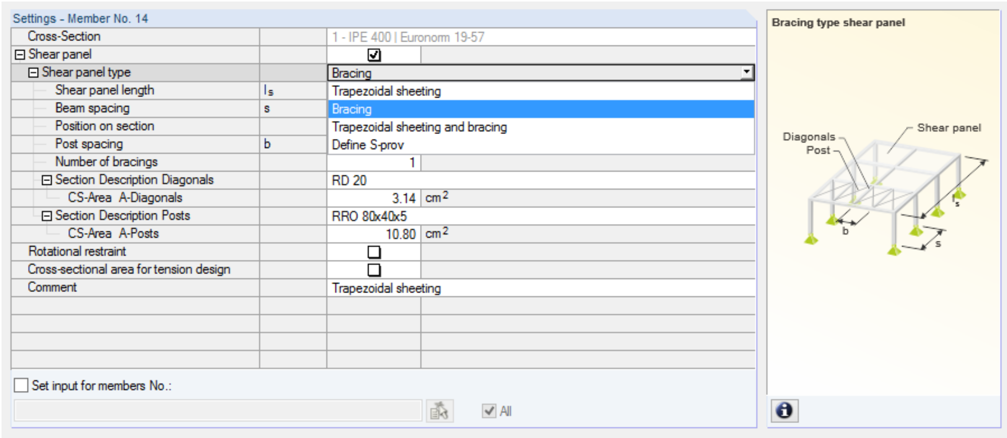
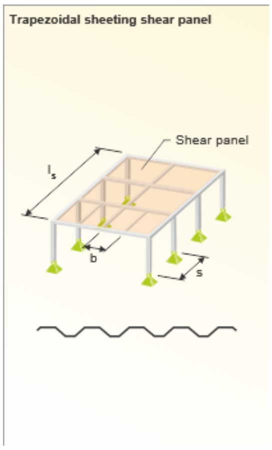


Figure 2.41 Selecting the shear panel type

Trapezoidal sheeting

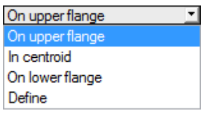


The application of a continuous lateral support is described in EN 1993-1-1 [1], Annex BB.2.1 and EN 1993-1-3 [3] clause 10.1.5.1.

To determine the shear panel stiffness of a trapezoidal sheeting (corrugated sheet), the following specifications are required (see Figure 2.40):

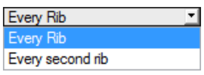
- Shear panel length l_S
- Beam spacing s
- Position of trapezoidal sheeting on section
- Trapezoidal sheeting description
- Fastening arrangement

You can enter the *Shear panel length* and the *Beam spacing* manually. You can also select them graphically with [button]. This button becomes available as soon as the cursor is placed in one of these text boxes. Then, you can select two snap points in the RFEM/RSTAB 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. If the entered data is user-defined, the distance *d* is related to the centroid; the sign results from the orientation of the cross-section's z-axis.

To access the corrugated sheets library, click the [button] button that becomes available after clicking in the *Trapezoidal sheeting description* text box (see Figure 2.40). The cross-section library of RFEM or RSTAB appears (see Figure 2.42) where you can select the trapezoidal sheeting by double-click or with [OK]. Thus, the *Shear panel coefficients* K_1 and K_2 (according to the approval certificate) will be automatically entered in the *Settings* table. The basic width *b* of the trapezoidal sheeting indicated in the cross-section database has no influence on these coefficients.



The *Fastening arrangement* for 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 the factor 5.

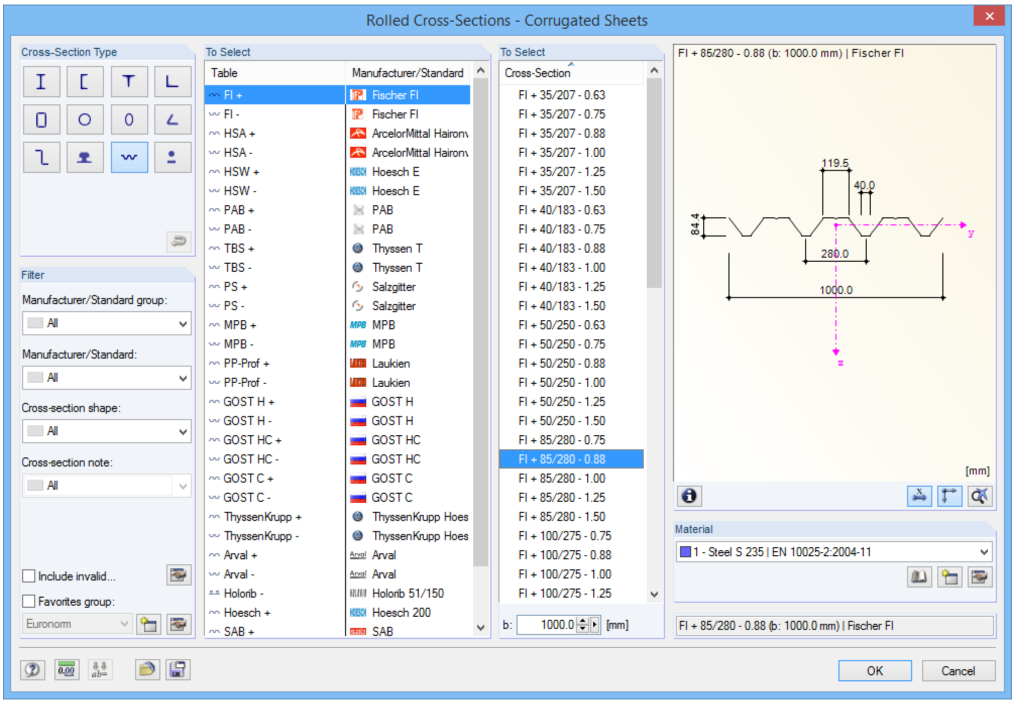


Figure 2.42 Cross-section library for *Rolled Cross-Sections - Corrugated Sheets*

Bracing

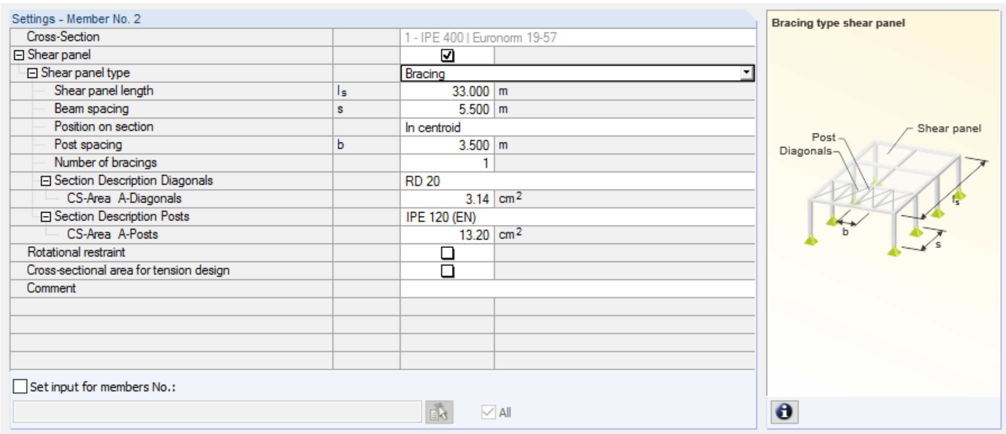
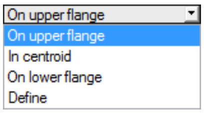


Figure 2.43 Shear panel type *Bracing*

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

- Shear panel length l_S
- Beam spacing s
- Position of 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. You can also select them graphically with . This button becomes available as soon as the cursor is placed in one of these text boxes. Then, you can select two points in the RFEM/RSTAB work window, defining the shear panel or the spacings.



The bracing's *Position on section* can be considered in different ways by means of the list shown on the left. If the entered data is user-defined, the distance *d* is related to the centroid; the sign results from the orientation of the cross-section's z-axis.

The easiest way to define the cross-sectional areas of the diagonals and posts is to select the *Section Description* in the cross-section library of RFEM/RSTAB. You can access this library with the button available at the end of the text box. Then, the *CS-Area* will be imported automatically. But you can also enter the value directly.

Trapezoidal sheeting and bracing

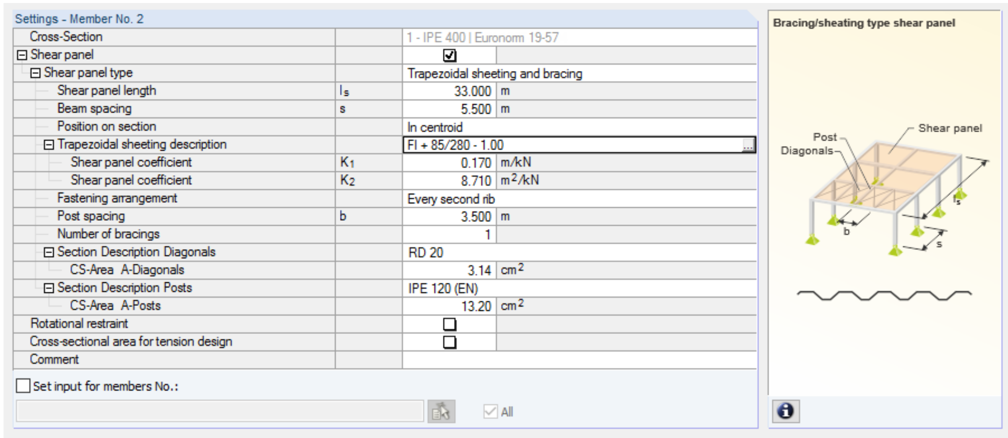


Figure 2.44 Shear panel type *Trapezoidal sheeting and bracing*

To determine the provided shear panel stiffness due to the 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* .

Define S-prov

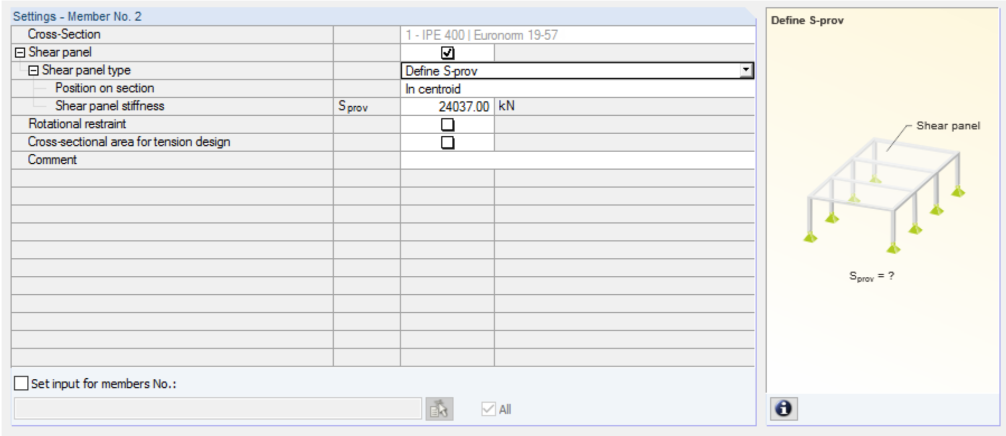
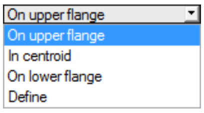


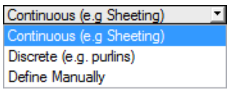
Figure 2.45 Shear panel stiffness *Define S-prov*



The value of the provided *Shear panel stiffness* S_{prov} can be entered also directly.

In addition, the shear panel's *Position on section* must be specified.

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 the rotational restraint in the drop-down list.

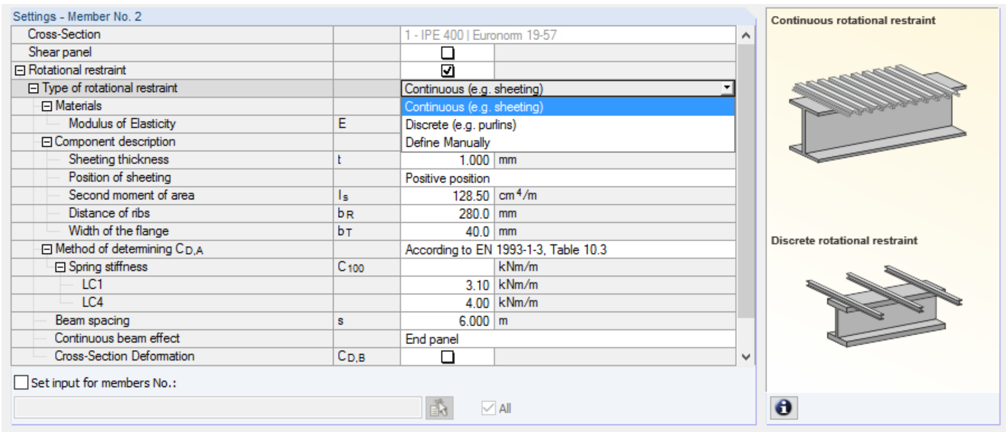
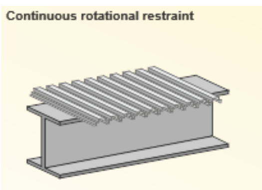


Figure 2.46 Selecting the type of rotational restraint

Continuous rotational restraint



To determine the stiffness components from a corrugated sheet and the connection deformation, the following specifications are required (see Figure 2.46):

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

To access the corrugated sheets library, click the [...] button that is displayed after clicking in the *Component description* text box (see Figure 2.44). The cross-section library of RFEM or RSTAB appears (see Figure 2.42) where you can select the corrugated sheet by double-clicking or with [OK]. The section parameters of *Sheeting thickness t, Position of sheeting*, effective *Second moment of area* for the downward loading direction, *Distance of ribs* b_R (corrugation width), and *Width of the flange* b_T are imported automatically.

When the continuous rotational restraint is set, you also have to consider the deformation of the connection. In the entry for *Method of determining* $C_{D,A}$, you can enter the spring stiffness C_{100} for the individual load cases and combinations. It is also possible to determine it by the program according to [3] Table 10.3. To determine it automatically, use the [...] button that appears after clicking in the text box of the row C_{100}. A dialog box opens where you can select the appropriate coefficient.

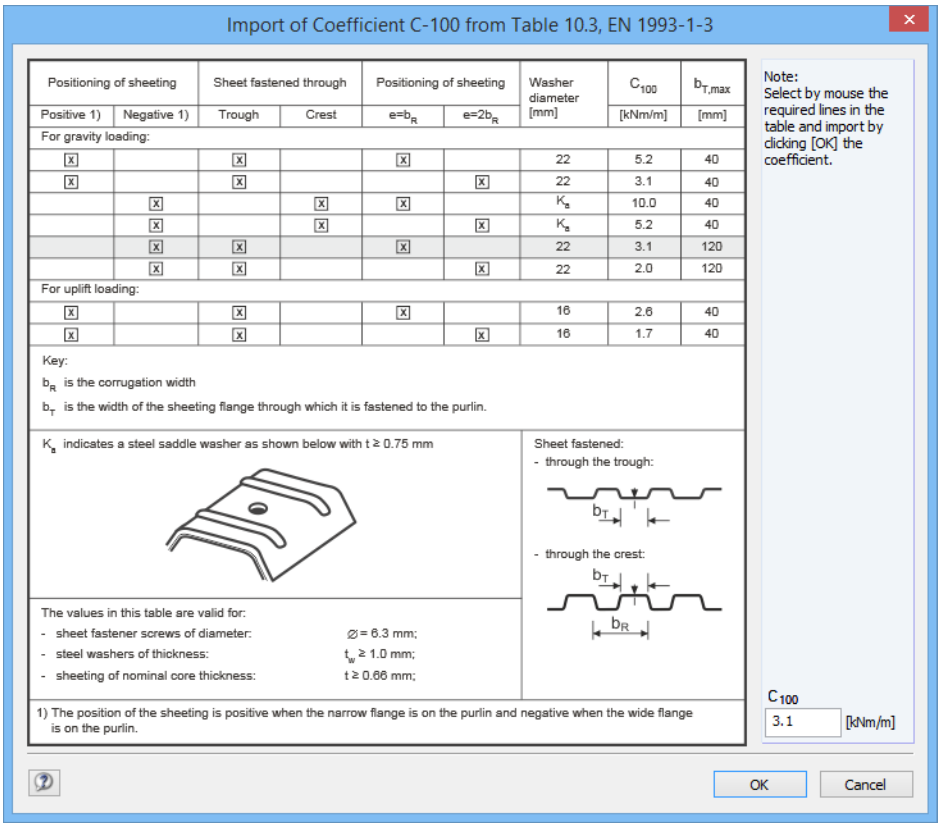
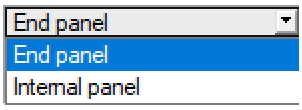


Figure 2.47 Dialog box *Import of Coefficient C-100 from Table 10.3, EN 1993-1-3*

When you click [OK], this value will be assigned to all load cases and combinations that are selected for design. If you want to assign the coefficient by load case, you have to open the *Import of Coefficient* dialog box via the C_{100} text boxes of the individual load cases and combinations.

You can also define the *Beam spacing* manually or graphically with the [...] button. In the work window of RFEM or RSTAB, click two nodes defining the distance between the beams.



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

Discrete rotational restraint

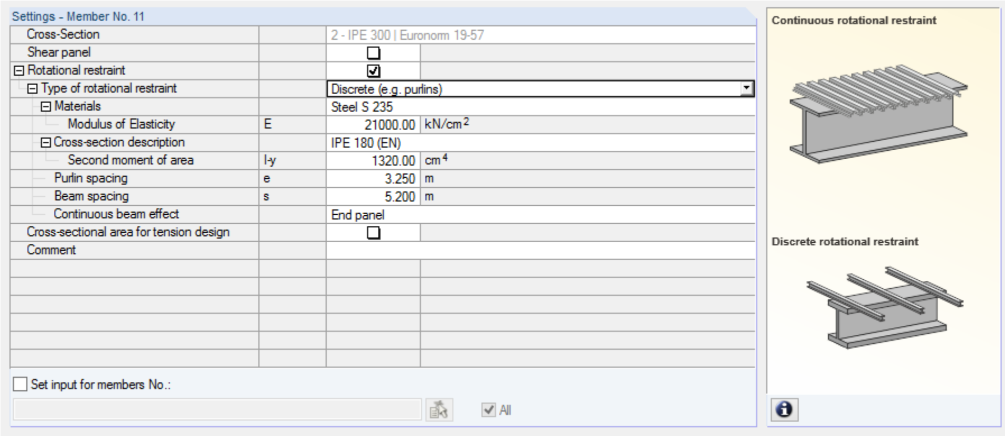


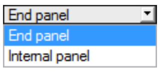
Figure 2.48 *Discrete* type of rotational restraint

To determine the stiffness component from single supports, for example purlins, the following specifications are required:

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

The *Material* and the *Cross-section description* can each be selected in the library of RFEM or RSTAB that you access with the button. Activate the relevant text box by clicking in it.

You can enter the *Purlin spacing* and the *Beam spacing* manually or graphically with the button. In the RFEM/RSTAB work window, click two nodes defining the distance between the purlins or horizontal beams.



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

Define Manually

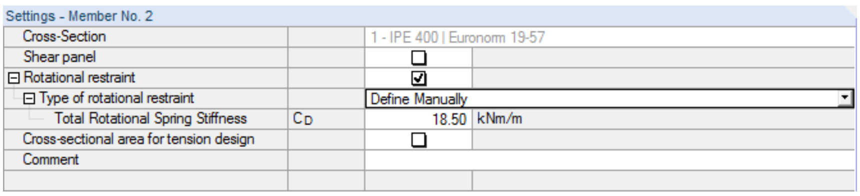


Figure 2.49 *Define Manually* the rotational spring stiffness

You can also directly enter the value of the provided *Total Rotational Spring Stiffness* C_D.

Cross-sectional area for tension design

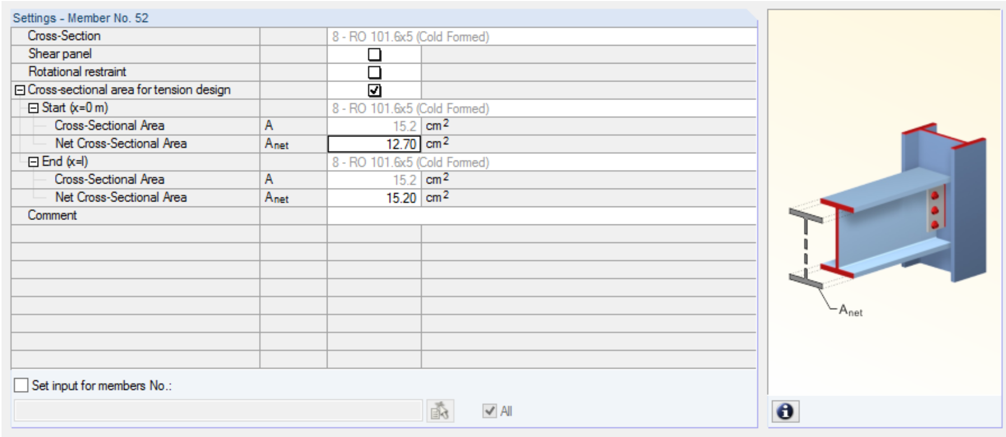


Figure 2.50 Defining the *Cross-sectional area for tension design*

According to [1] clause 6.2.3, section reductions due to holes must be considered in the 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 if at least one set of members is selected for design in the *1.1 General Data* window.

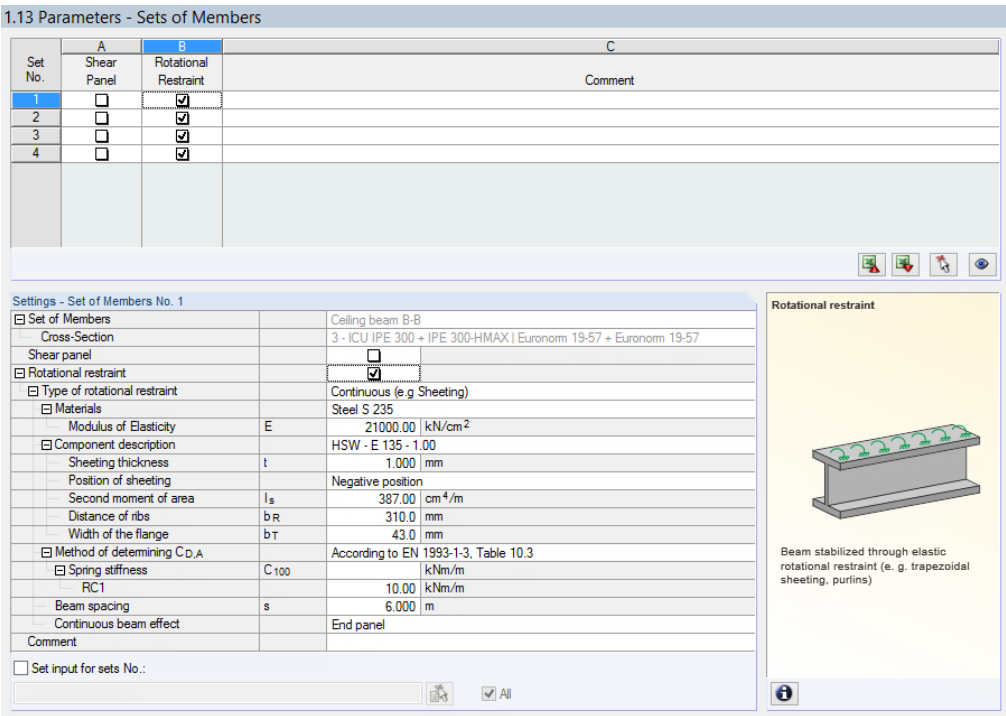
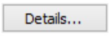


Figure 2.51 Window *1.13 Parameters - Sets of Members*

The concept of this window is similar to the previous Window *1.12 Parameters - Members*. Here, you can define the parameters for shear panel and rotational restraint for each set of members as described in Chapter 2.12.

Warping analysis with seven degrees of freedom



If sets of members are analyzed according to the second-order analysis for flexural-torsional buckling with warping torsion (see *Warping Torsion* tab of *Details* dialog box), you can define precambers additionally in this window: The eigenshape has a strong impact on the warping analysis that is performed by taking into account seven degrees of freedom. Window 1.13 is extended accordingly for this input.

The warping analysis requires a license of the **RF-/STEEL Warping Torsion** module extension.

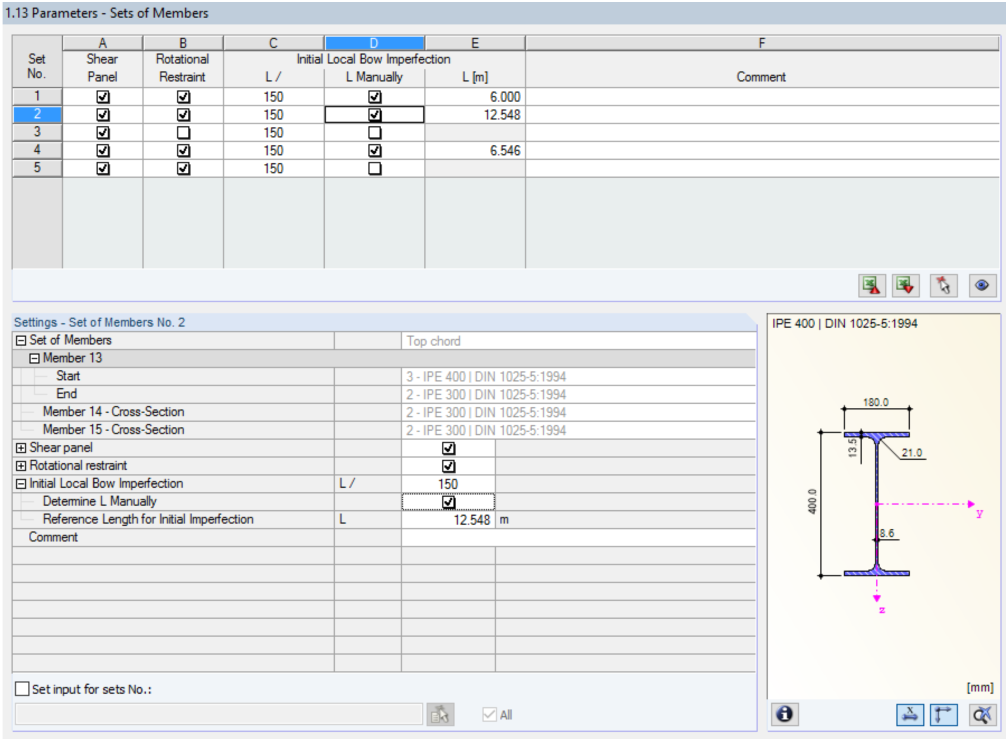


Figure 2.52 Window *1.13 Parameters - Sets of Members* for warping analysis with seven degrees of freedom

Based on the boundary conditions, RF-/STEEL EC3 determines the eigenshapes of the sets of members before the actual calculation is carried out. They will be considered accordingly for further analysis.

In column C, you enter the *Initial Local Bow Imperfection* relative to the length of the set of members. The most unfavorable value L/150 according to [1] Table 5.1 is preset. This value can be adjusted to the cross-section's buckling curve by taking into account the coefficient *k* for member imperfections according to [1] 5.3.4 (3).

The length of the entire set of members is preset as the reference length. As soon as the *L Manually* check box has been selected, the length can be user-defined in column E, for example to consider lateral supports.

Find more information about the application of bow imperfections in the RF-FE-LTB manual on our website.

2.14 Local Transverse Forces

This input screen is displayed when the two check boxes in the *Web Stiffening and Local Transverse Forces* section in the *Cold-Formed* section of the *Details* dialog box (see Chapter 3.1.7) are selected. In Window 1.14, you can define the parameters that are required for the local transverse force design according to EN 1993-1-3 [3] 6.1.7 for cold-formed sections. This design ensures that there is no compression or buckling of the web due to local transverse forces through the flange into the web. In case of a combined loading from bending and local transverse forces, it is also examined whether the conditions specified in EN 1993-1-3 [3] 6.1.11 are met.

The local transverse force design is only possible for cross-sections with non-stiffened webs according to [3] 6.1.7.2 and 6.1.7.3. Web cross-sections with longitudinal stiffenings **cannot** be designed according to [3] 6.1.7.4.

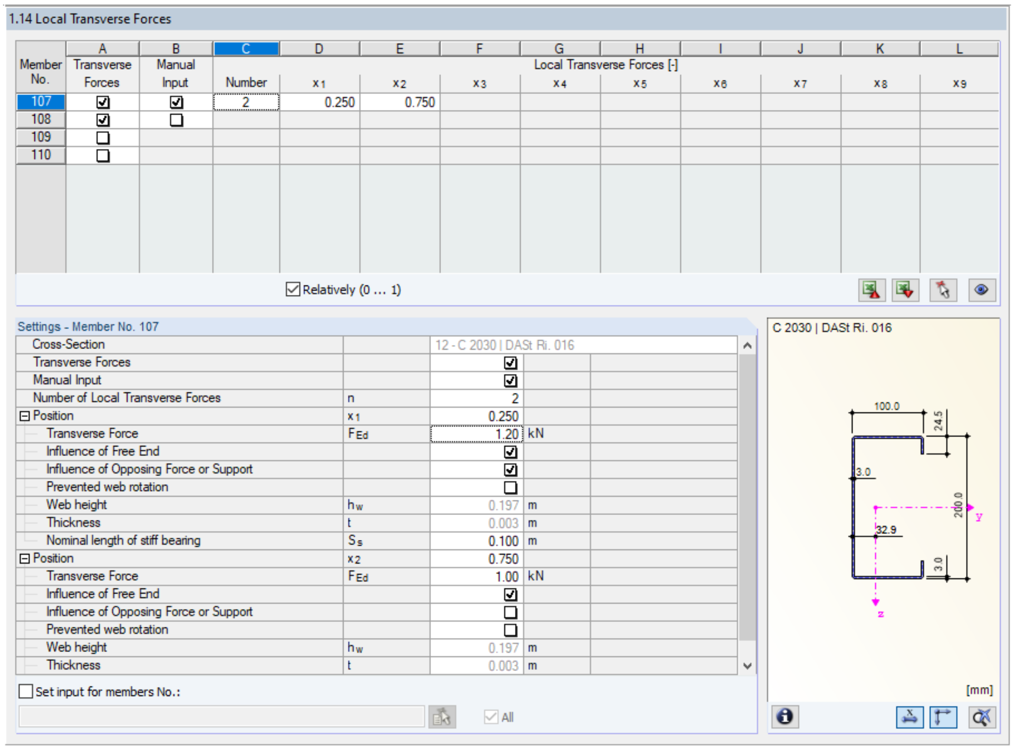


Figure 2.53 Window 1.14 *Local Transverse Forces*

In the upper part of the window, you can specify for which members the local transverse force design should be performed and which length of the stiff bearing is to be applied in each case. If you enter the transverse forces manually, you can set specific settings for the member selected above in the *Settings* section.

The consideration of the *Transverse Forces* is activated for all members by default. Thus, the shear force distribution is used for the design of the web loading for local load application. If the check box in column A is cleared in one row, this design is omitted for the corresponding member.

During the design, the locations of discontinuity resulting from the shear force distribution are analyzed. All parameters for determining the web resistance $R_{w,Rd}$ are determined automatically; the *Nominal length of stiff bearing* s_s is preset to 0.10 m, but can be adjusted globally for each member. If the internal force distribution of a member does not represent the actual conditions, the introduced load can be defined individually. To do this, activate *Manual input* in column B. In doing so, additional fields in the table and in the *Settings* become accessible.

The *Number* of shear loads determines how many design locations exist on the member. In the *Local Transverse Forces* columns, you can define positions *x* for which you want to perform an analysis. These locations can be arranged anywhere on the member: The design only uses the parameters that have to be defined for each position in the *Settings* section.

When entering the data manually, specify the design value of the *Transverse Force* F_{Ed}. If there is *Influence of Free End*, the resistance of the web $R_{w,Rd}$ for the geometry condition $c \leq 1.5\ h_w$ is determined, for example, according to [3] Equation (6.15a), (6.15b), or (6.15c); for cross-sections with two or more stiffened webs, category 1 according to [3] Figure 6.9 is applied. When the check box is cleared, [3] Equation (6.15d) or (6.15e) is used. The *Influence of Opposing Force or Support* check box controls whether the design is performed according to [3] Figure 6.1 a) or Figure 6.7 b) for cross-sections with only one web. If it is selected, a distance $e \leq 1.5\ h_w$ is assumed. Cross-sections with two webs are classified in category 1 (see [3] Figure 6.9). The *Prevented web rotation* check box controls whether the resistance of the web $R_{w,Rd}$ is determined with equations according to [3] 6.1.7.2(4). For each position, the *Nominal length of stiff bearing* s_s can be specified by the user. When designing a cross-section with two webs, the *Ratio of shear forces* $V_{Ed,2}/V_{Ed,1}$ with which the effective support length l_a is calculated according to [3] 6.1.7.3(4) for category 2, has to be specified as well. β_v is determined as follows:

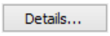
$$\beta_v = \frac{1 - \frac{V_{Ed,2}}{V_{Ed,1}}}{1 + \frac{V_{Ed,2}}{V_{Ed,1}}}$$

An example of local load introduction design is presented in a technical article that can be found in the Knowledge Base on our website.

If a continuous beam exists, the design must be performed on the **set of members**: The force F_{Ed}, which is governing for designing local transverse forces, is thus correctly determined for the intermediate supports.

3 Calculation

3.1 Detailed Settings



The designs are based on the internal forces determined in RFEM or RSTAB.

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
- Warping torsion
- Plasticity
- Cold-Formed
- General

3.1.1 Ultimate Limit State

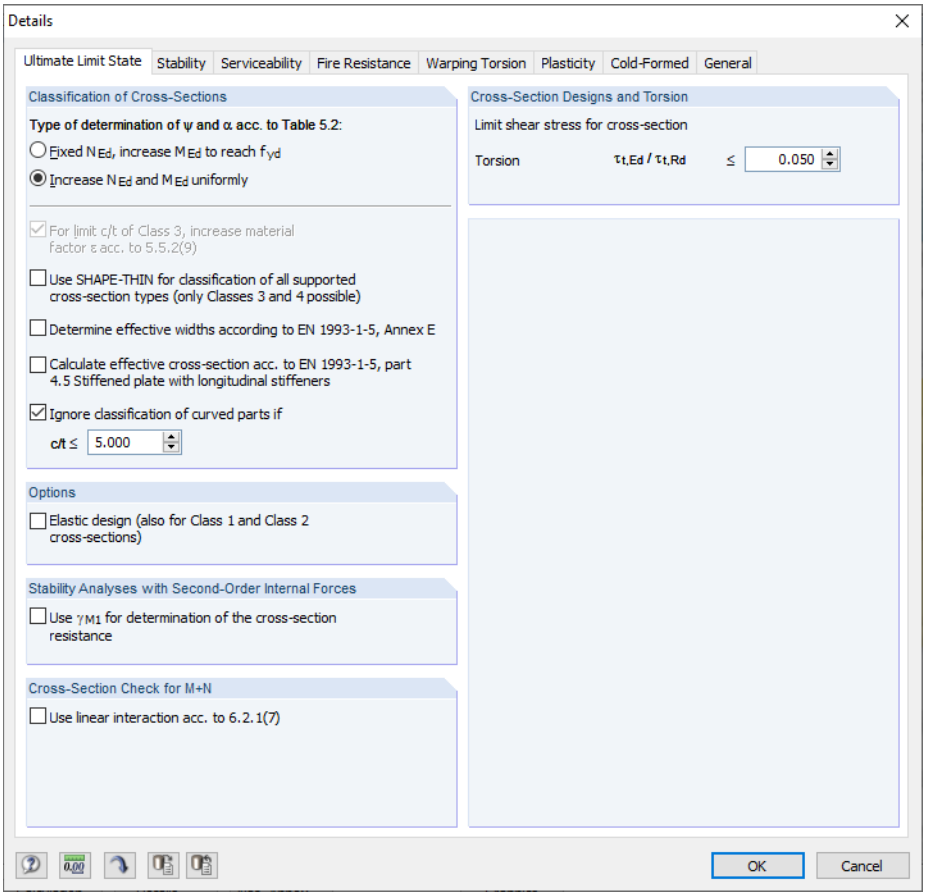


Figure 3.1 Dialog box *Details*, tab *Ultimate Limit State*

Classification of Cross-Sections

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

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

You can access the check box *For limit c/t of Class 3, increase material factor ε acc. to 5.5.2(9)* if the stability analysis in the *Stability* tab is deactivated. This is based on the specifications for classification in [1] clause 5.5.2 (10). With the stability analysis being deactivated, it is possible to handle cross-sections classified as class 4 like cross-sections of class 3 by increasing the factor ε.

If you select the option *Use SHAPE-THIN for classification of all supported cross-section types*, the effective cross-section values of class 4 sections will be calculated by the method used in the cross-section program SHAPE-THIN. For cross-sections classified as 'General' (belonging neither to a rolled nor a parametrized cross-section table), the classification will generally be performed with SHAPE-THIN. You can design these cross-sections only elastically as class 3 or class 4 sections.

Optionally, you can *Determine effective widths according to EN 1993-1-5, Annex E*. Annex E of [5] describes alternative methods for determining the effective cross-section areas for stresses below the yield strength (see also the article in the Knowledge Base on our website).

If you want to *Calculate effective cross-section acc. to EN 1993-1-5, part 4.5* for stiffened buckling panels, select the corresponding check box. This option only affects SHAPE-THIN sections for which buckling panels and stiffeners have been defined.

The width/thickness ratios relevant for the classification can cause problems for cross-sections with curved elements from SHAPE-THIN. The check box *Ignore classification of curved parts* allows you to exclude short fillet arcs from the classification as soon as the user-defined c/t-ratio is below the limit (see the article in the Knowledge Base). Then, longitudinal ribs or folds and grooves of sheeting have no influence on the design.

Options

Cross-sections assigned to class 1 or 2 are designed plastically by RF-/STEEL EC3. If this is not desired, you can activate the *Elastic design* also for those 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 the internal forces calculated according to the second order theory, you can use this check box to decide if factor γ_{M1} (instead of γ_{M0}) is used for the cross-section designs.

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, if necessary, modify it in the *National Annex Settings* dialog box (see Chapter 2.10).

Find more information on stability analyses in an article of the Knowledge Base at our website.

Cross-Section Check for M+N

The check box *Use linear interaction acc. to 6.2.1(7)* determines whether to use a 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.

Cross-Section with Class 4 and Torsion

In the input field, you can define the shear stress component from torsion up to which the torsional stresses are neglected in the cross-section design. So it is possible to suppress warnings of oversized torsional stresses for cross-sections of class 4.

3.1.2 Stability

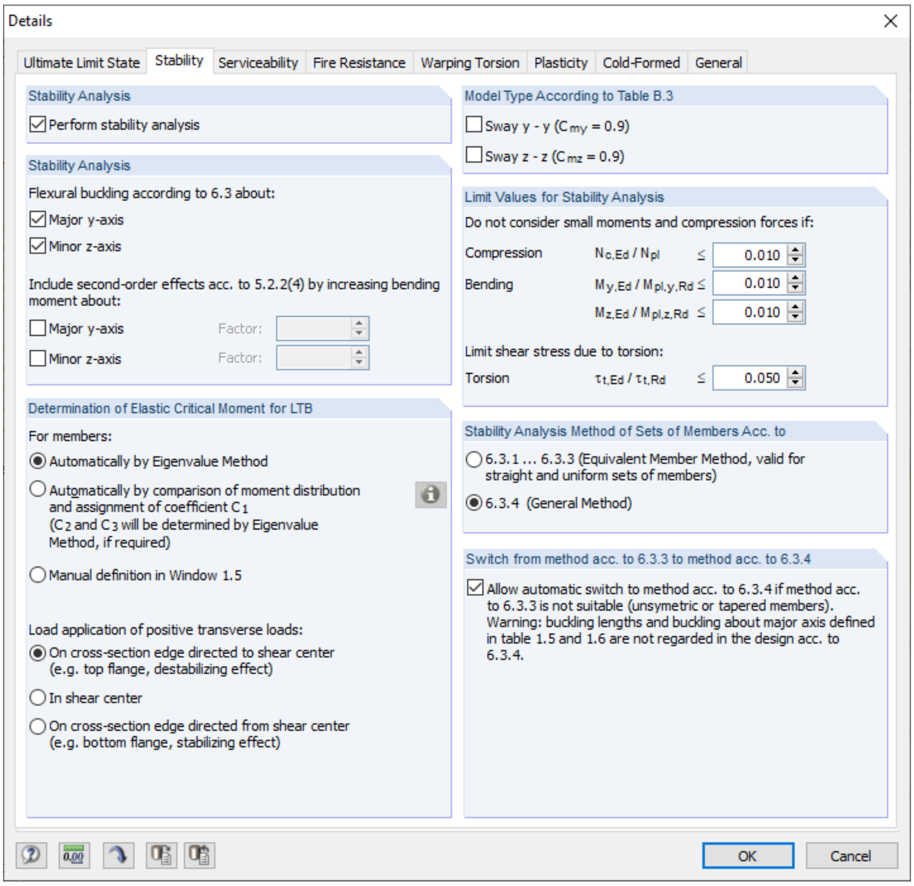


Figure 3.2 Dialog box *Details*, tab *Stability*

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, Windows 1.4 to 1.8 are not displayed.

When the check box is selected, you can define the axes relevant for the analysis of *Flexural buckling according to* 6.3 of [1].

Furthermore, it is possible to consider *second-order effects acc. to 5.2.2(4)* by a factor for bending moments that can be defined manually. In this way, for example, when designing a frame with its governing buckling mode represented by lateral displacement, you can determine the internal forces according to linear static analysis and increase them by appropriate factors. Increasing the bending moments does not affect the flexural-buckling analysis according to [1] clause 6.3.1, which is performed with the axial forces.

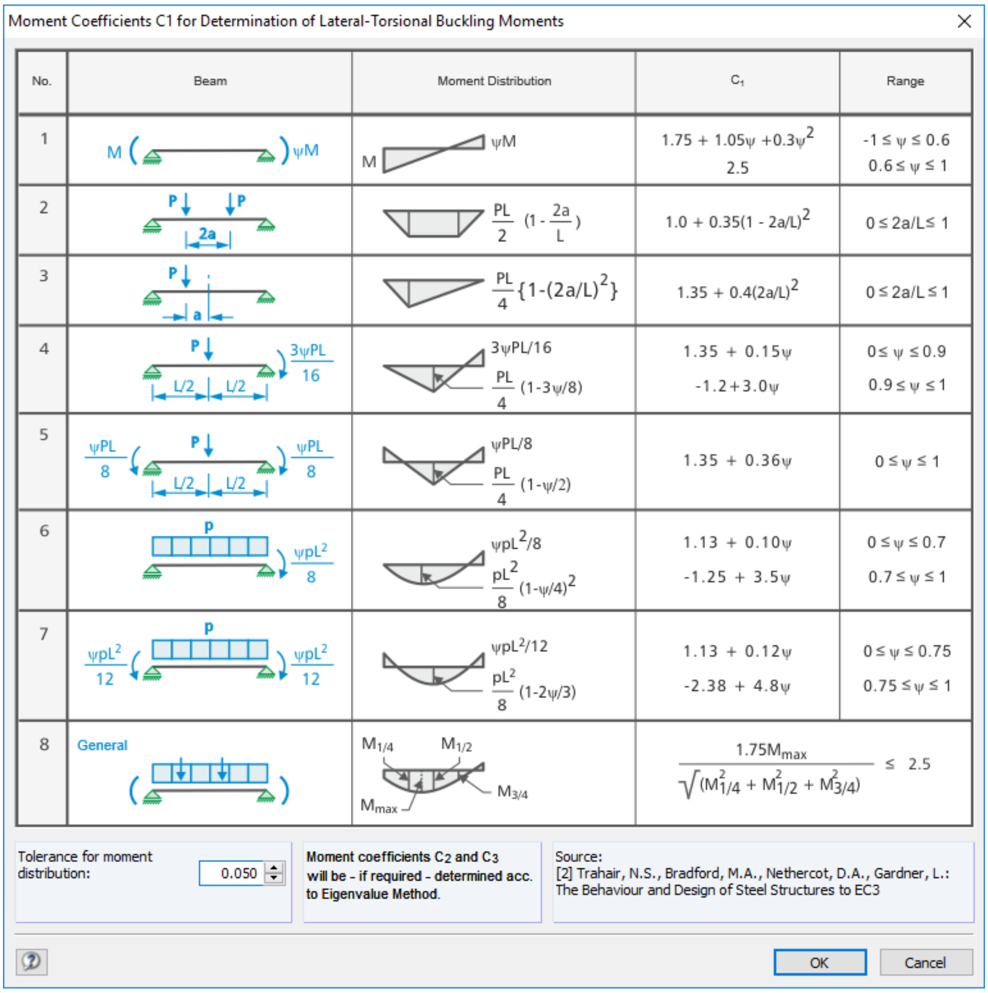
Determination of Elastic Critical Moment for LTB

By default, RF-/STEEL EC3 determines the elastic critical moment *Automatically by Eigenvalue Method*. For the calculation, the program uses a finite member model to determine M_{cr} taking into account 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).

When determining the elastic critical moment *Automatically by comparison of moment distribution*, factor C_1 is determined by means of the moment distribution. Click the [Info] button to open a dialog box showing the load and moment distributions.


Moment Coefficients C1 for Determination of Lateral-Torsional Buckling Moments

No.	Beam	Moment Distribution	C₁	Range
1	M, ψM	M, ψM	$1.75 + 1.05\psi + 0.3\psi^2$ 2.5	$-1 \le \psi \le 0.6$ $0.6 \le \psi \le 1$
2	P, P, 2a	$\frac{PL}{2}(1 - \frac{2a}{L})$	$1.0 + 0.35(1 - 2a/L)^2$	$0 \le 2a/L \le 1$
3	P, a	$\frac{PL}{4}\{1-(2a/L)^2\}$	$1.35 + 0.4(2a/L)^2$	$0 \le 2a/L \le 1$
4	P, L/2, L/2, $\frac{3\psi PL}{16}$	$3\psi PL/16$ $\frac{PL}{4}(1-3\psi/8)$	$1.35 + 0.15\psi$ $-1.2+3.0\psi$	$0 \le \psi \le 0.9$ $0.9 \le \psi \le 1$
5	$\frac{\psi PL}{8}$, P, L/2, L/2, $\frac{\psi PL}{8}$	$\psi PL/8$ $\frac{PL}{4}(1-\psi/2)$	$1.35 + 0.36\psi$	$0 \le \psi \le 1$
6	p, $\frac{\psi pL^2}{8}$	$\psi pL^2/8$ $\frac{pL^2}{8}(1-\psi/4)^2$	$1.13 + 0.10\psi$ $-1.25 + 3.5\psi$	$0 \le \psi \le 0.7$ $0.7 \le \psi \le 1$
7	$\frac{\psi pL^2}{12}$, p, $\frac{\psi pL^2}{12}$	$\psi pL^2/12$ $\frac{pL^2}{8}(1-2\psi/3)$	$1.13 + 0.12\psi$ $-2.38 + 4.8\psi$	$0 \le \psi \le 0.75$ $0.75 \le \psi \le 1$
8	General	$M_{1/4}$, $M_{1/2}$, $M_{3/4}$, M_{max}	$\frac{1.75 M_{max}}{\sqrt{(M_{1/4}^2 + M_{1/2}^2 + M_{3/4}^2)}} \le 2.5$	

Tolerance for moment distribution: 0.050

Moment coefficients C_2 and C_3 will be - if required - determined acc. to Eigenvalue Method.

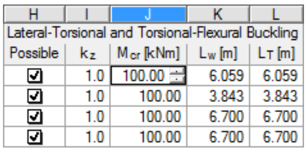
Source:
[2] Trahair, N.S., Bradford, M.A., Nethercot, D.A., Gardner, L.: The Behaviour and Design of Steel Structures to EC3

OK Cancel

Figure 3.3 Dialog box *Moment Coefficients C1 for Determination of Lateral-Torsional Buckling Moments*

The *Tolerance for moment distribution* in this dialog box allows you to control the degree up to which deviations are acceptable for the 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]
☑	1.0	100.00	6.059	6.059
☑	1.0	100.00	3.843	3.843
☑	1.0	100.00	6.700	6.700
☑	1.0	100.00	6.700	6.700

M_{cr} user-defined

With the option *Manual definition in Window 1.5*, the title of column J in Window 1.5 changes to $\mathbf{M_{cr}}$ so that you can enter the elastic critical moment for LTB directly.

If *transverse loads* are available, it is important to define the location where these forces are acting on the cross-section: Depending on the load application, transverse loads can be stabilizing or destabilizing, and thus have a major impact on the elastic critical moment.

The signs of the eccentricities are related to the cross-section's shear center M. An article in our Knowledge Base provides more information about the sign convention for transverse loads.

Model Type According to Table B.3

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

Limit Values for Stability Analysis

Small compression forces do not allow for a check of pure bending according to [1] clause 6.3.2. A user-defined limit ratio of $N_{c,Ed}$ / N_{pl} allows you to hide small compression forces for this design.

To design asymmetric cross-sections with the intended axial compression according to [1] clause 6.3.1, you can analogously neglect *small moments* about the major and the minor axis with the settings defined in this dialog section.

According to [1] clause 6.3.4, the general method is allowed for unsymmetric cross-sections or tapered members only if they are subjected to compression and/or uniaxial bending in the principal plane. In order to neglect a minor moment loading about the minor axis, you can define a limit for the moments ratio $M_{z,Ed}$ / $M_{pl,z,Rd}$.

Scheduled Torsion is not clearly specified in [1]. If there is a torsional stress that does not exceed the predefined ratio of existing shear stress to *Limit shear stress due to torsion* of 5%, it is neglected for the stability analysis. In this case, results for flexural buckling and lateral-torsional buckling are displayed.

If one of the limit values in this dialog section is exceeded, a note appears in the results window and the program won't perform any stability analysis. However, the cross-section designs are performed independently. These limit settings are **not** part of the Standard [1] or any National Annex. Modifying the limits is the user's responsibility.

In our Knowledge Base, you can find an example explaining the application of limit values: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001498

Stability Analysis Method of Sets of Members

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

With the default setting of *6.3.4 (General Method)*, the program performs a general analysis according to [1] clause 6.3.4 which is based on the coefficient α_{cr}. In Windows *1.7 Nodal Supports* and *1.8 Member Hinges*, the boundary conditions must be defined with regard to the stability failure (buckling and lateral-torsional buckling) separately for each set of members. The factors k_z and k_w from Window 1.5 are not used.

Find more information about the general method in an article of the Knowledge Base.

The options are locked if the stability analysis with warping torsion is set (see Chapter 3.1.5).

Switch from method acc. to 6.3.3 to method acc. to 6.3.4

For structural components with bending and compression, the method according to [1] 6.3.3 is only applicable for double-symmetric cross-sections. However, components with single-symmetrical cross-sections or a tapered member diagram can be designed with the General Method according to [1] 6.3.4. In Germany, only I-sections are allowed, which is why a warning appears during calculation. In this case, it would be possible to deviate from the National Annex in the *National Annex Settings* dialog box and also allow the General Method for non-I-sections (see Figure 2.10).

If the check box is selected, RF-/STEEL automatically selects the appropriate design method. The buckling lengths defined in Window 1.5 and 1.6 for buckling about the principal axis are not considered in the design according to Section 6.3.4.

3.1.3 Serviceability

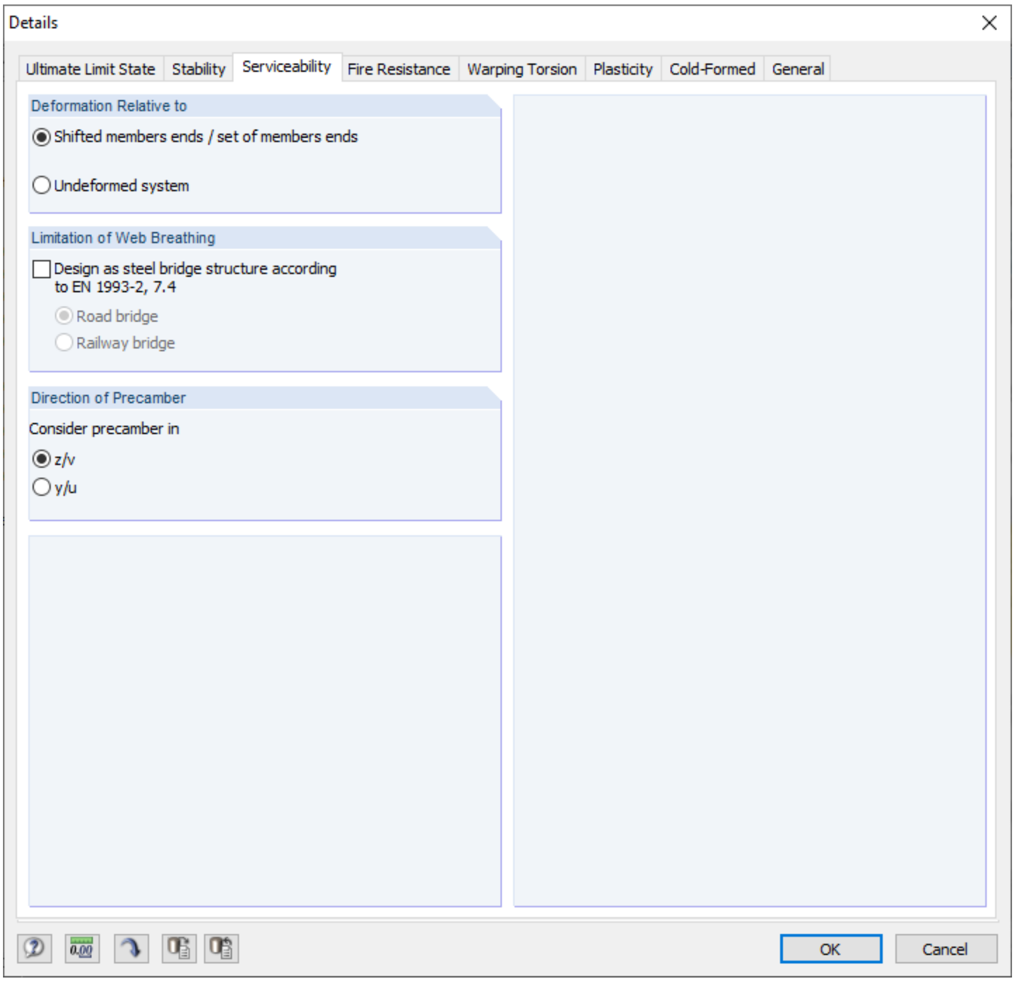


Figure 3.4 Dialog box *Details*, tab *Serviceability*

Deformation Relative to

The options control whether the maximum deformations are related to the shifted ends of members or sets of members (connection line between start and end nodes of the deformed system) or to the undeformed initial system. Generally, the deformations are designed relative to the displacements in the entire structural system.

An article of the Knowledge Base on our website presents an example describing the relation of deformations.

Nat. Annex...

You can check and adjust, if necessary, the limit deformations in the *National Annex Settings* dialog box (see Figure 2.10).

Limitation of Web Breathing

In 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 of plates ("web breathing") as well as stiffness reductions due to plate buckling. The check box *Design as steel bridge structure according to EN 1993-2, 7.4* [8] allows you to analyze the breathing (repeated out-of-plane deformation), which can result in fatigue problems at the web-to-flange connections.

You have to specify if a *Road bridge* or a *Railway bridge* is designed because there are different criteria for each case.

Direction of Precamber

With the two options you can decide in which of the local member axes a precamber ("rise"), if applicable, is available. Depending on the setting, column F of Window 1.9 shows $w_{c,v}$ or $w_{c,u}$ as the title (see Figure 2.37).

3.1.4 Fire Resistance

This tab manages detailed settings for the fire resistance design.

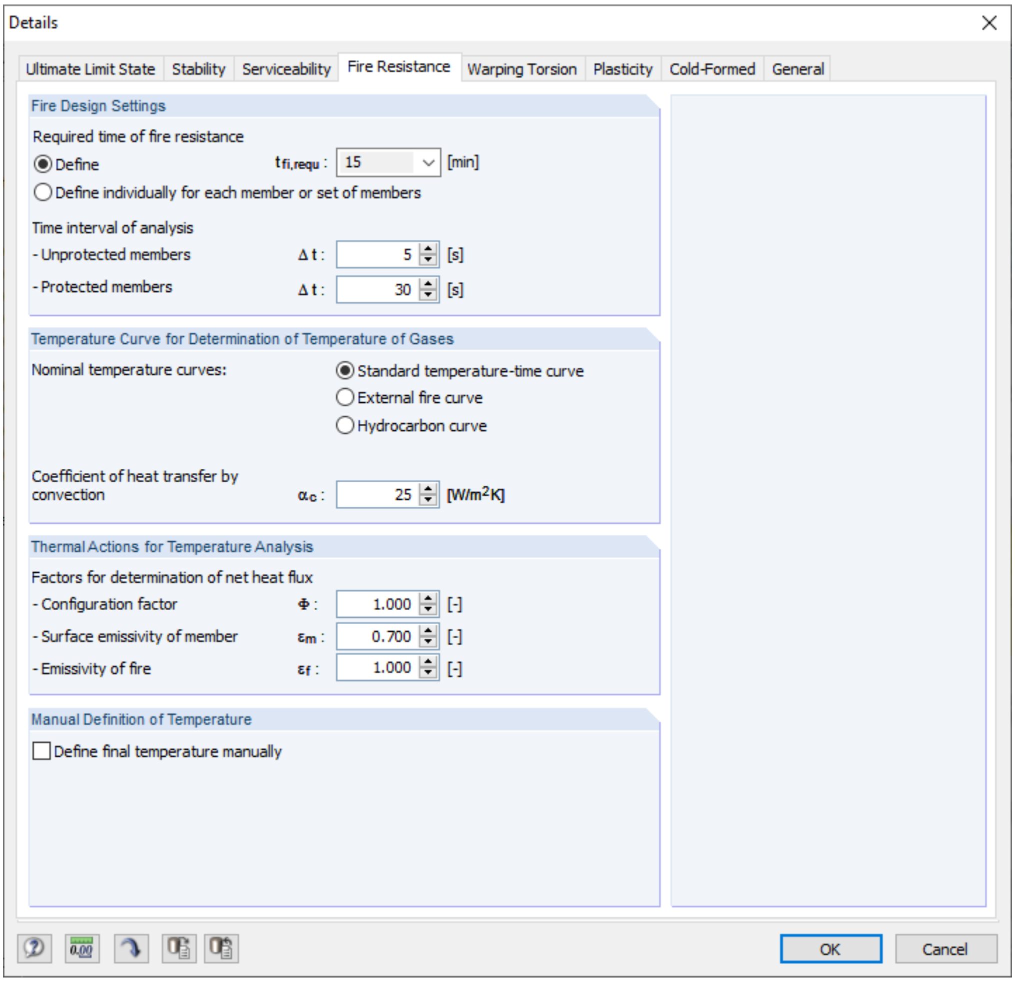


Figure 3.5 Dialog box *Details*, tab *Fire Resistance*

In addition to the *Required time of fire resistance* and the *Time interval* needed for the determination of the temperature change, you have to define the *Temperature Curve* to determine the gas temperature. Three curves are available for selection (see Figure 3.6 to Figure 3.8).

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

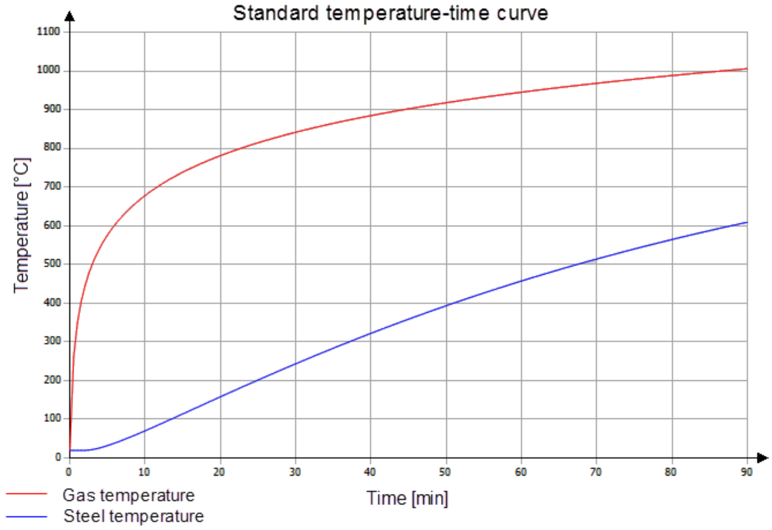


Figure 3.6 Standard temperature-time curve

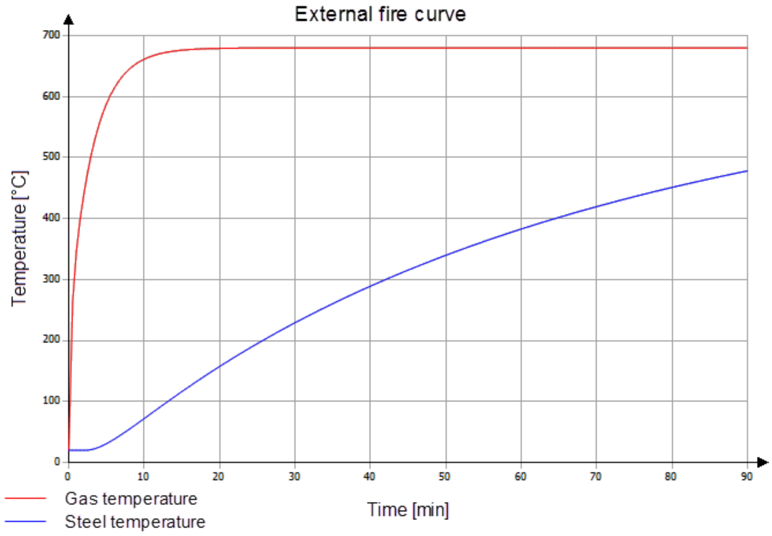


Figure 3.7 External fire curve

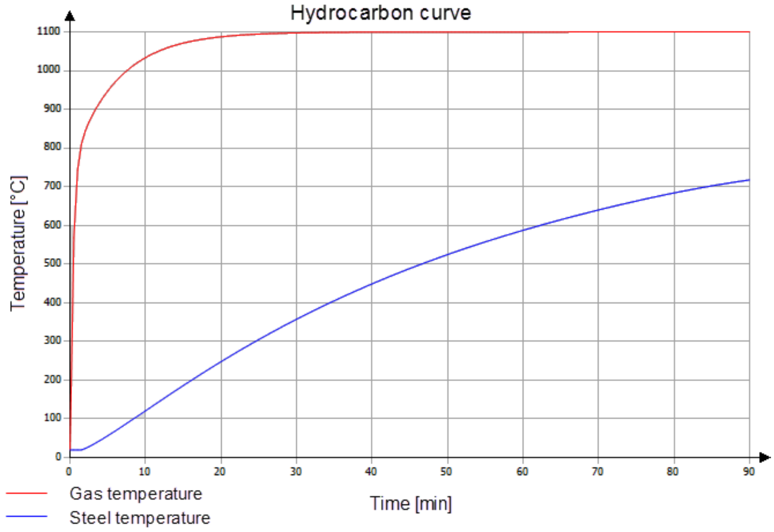


Figure 3.8 Hydrocarbon curve

1.10 Fire Resistance - Members

	A	B
No.	Members No.	Temperature Θ_a [°C]
1	1	300.00

When you select the check box for *Define final temperature manually*, you can enter the temperature Θ_a in Windows 1.10 and 1.11 individually.

3.1.5 Warping Torsion

This tab allows for settings of the warping analysis for sets of members. You can access the entries if the **RF-/STEEL Warping Torsion** module extension is licensed.

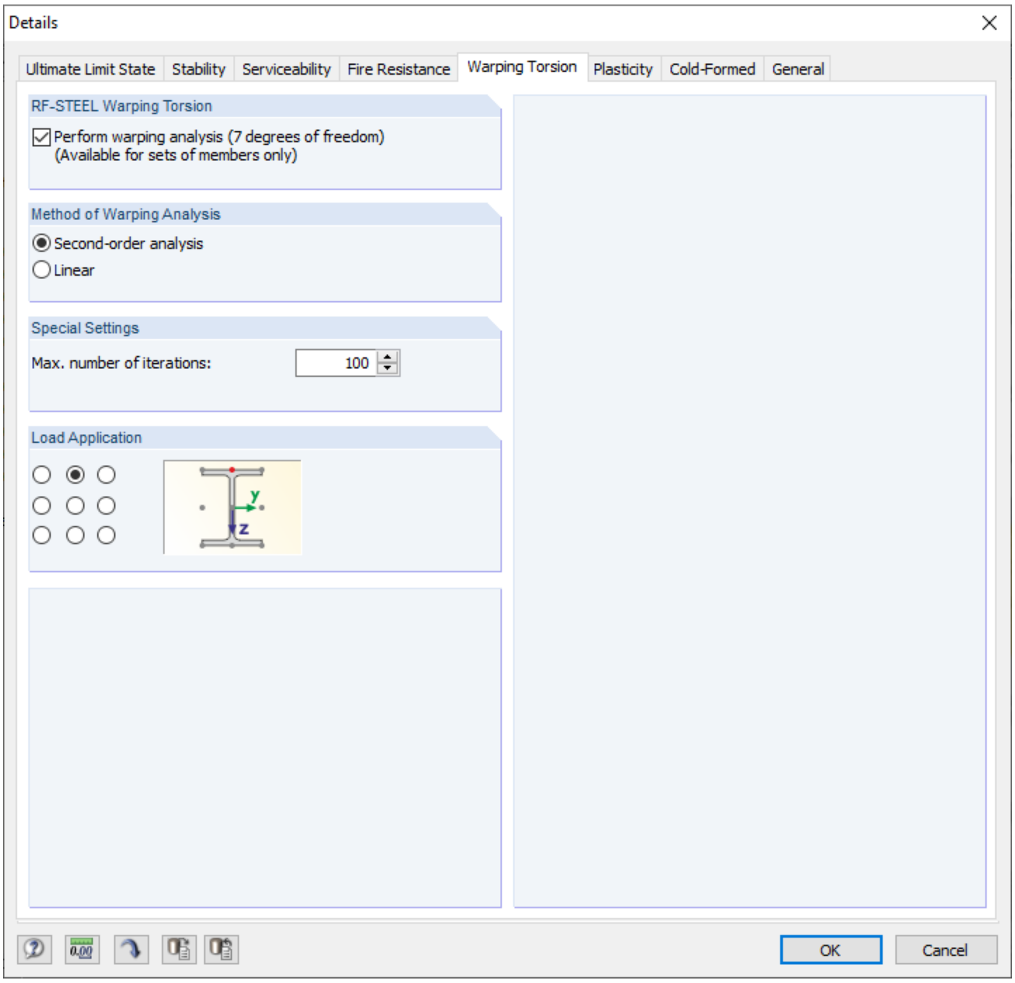


Figure 3.9 Dialog box *Details*, tab *Warping Torsion*

If you want RF-/STEEL EC3 to *Perform a warping analysis*, select this check box. Thus, the other dialog sections become accessible. At the same time, the corresponding options for the stability analysis of sets of members will be locked in the *Stability* tab.

In the method with seven degrees of freedom, the stability calculation is carried out according to the second-order analysis for flexural-torsional buckling taking into account warping torsion and imperfections affine to mode shapes. The freedom degrees of the displacements and rotations in and about the three axes X', Y' and Z' as well as warping can be user-defined in Windows 1.7 and 1.8 (see Figure 2.34 and Figure 2.36). In Window 1.13, you have to define the initial rise of the precamber (see Figure 2.52).

Find a detailed description of the second-order analysis for flexural-torsional buckling in the RF-/FE-LTB manual on our website.

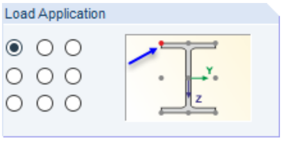
Find some examples describing the principles of this method in the following articles:

- https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001298
- https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001377

In addition to the second-order analysis, the *Method of Warping Analysis* can be a linear calculation.

Then, the set of members will be analyzed according to linear static analysis. This way, it is possible, for example, to analyze effects of warping and to deduce consequences on the stability behavior.

The warping analysis is performed iteratively, with the stiffness matrix *K* changing due to already computed internal forces and deformations. The *Max number of iterations* prevents the calculation from entering an infinite loop when there are convergence problems.



The *Load Application* has an important role for the stability analysis with seven degrees of freedom. Depending on the application point, the load has a stabilizing or destabilizing effect on the stability behavior. Use the nine setting options to define the location where the load is acting on the cross-section. The current point is highlighted red in the cross-section sketch.

Determination of loading in RF-/STEEL Warping Torsion

The loadings applied for the warping analysis are based on the results from RFEM or RSTAB. The module extension uses the member deformations in order to determine the moment distributions and thus the loads. It is therefore necessary to ensure, when defining the boundary conditions in Window 1.7, that the singled out member set model corresponds to the conditions of the RFEM/RSTAB model. For example, if rotations are enabled for a horizontal beam node, which are restricted in the model by a connected column, different internal force distributions in RFEM or RSTAB and RF-/STEEL EC3 will be the result.

The following article describes how the loading is determined in RF-/STEEL Warping Torsion: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001417

3.1.6 Plasticity

This tab allows for settings of the advanced plastic analysis for cross-sections. You can access the entries if the **RF-/STEEL Plasticity** module extension is licensed.

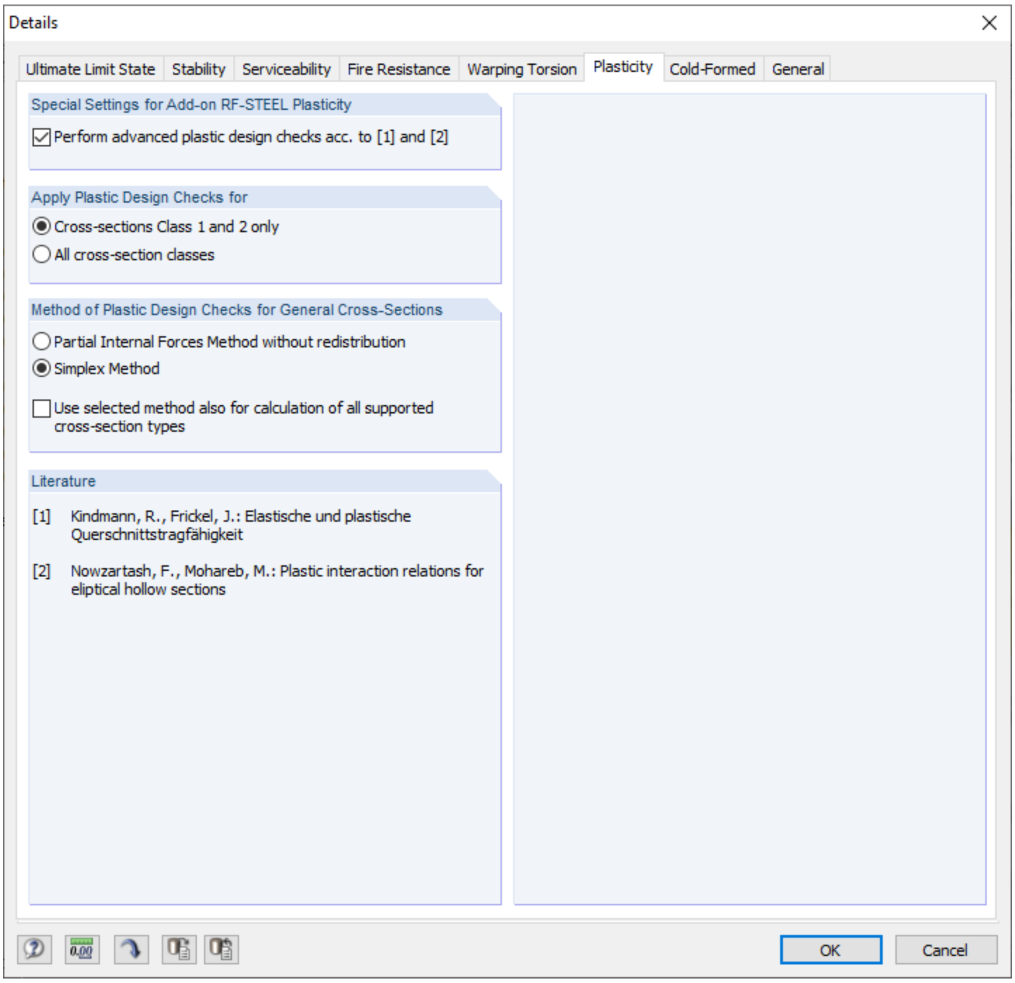


Figure 3.10 Dialog box *Details*, tab *Plasticity*

In the cross-section design performed by the elastic-plastic design method, linear elastic material behavior is assumed for the calculation of the internal forces S_d, and linear elastic-perfectly plastic material behavior for the calculation of the resistances R_d. Thus, the cross-section's reserves are used but the possibly available plastic reserves of the system are not considered. When the limit internal forces are reached in the fully plastic state, the ultimate limit state is reached.

If you want the program to *Perform advanced plastic design checks acc. to [1] and [2]*, select the check box (literature in manual: [10] and [11]). Then, the other dialog sections become accessible.

Find a detailed description of the plastic cross-section designs in the manual for RF-/STEEL Plastic on our website.

3.1.7 Cold-Formed

This tab allows you to analyze cold-formed sections according to EN 1993-1-3 [3]. The check boxes are only accessible if the **RF-/STEEL Cold-Formed Sections** module extension is licensed.

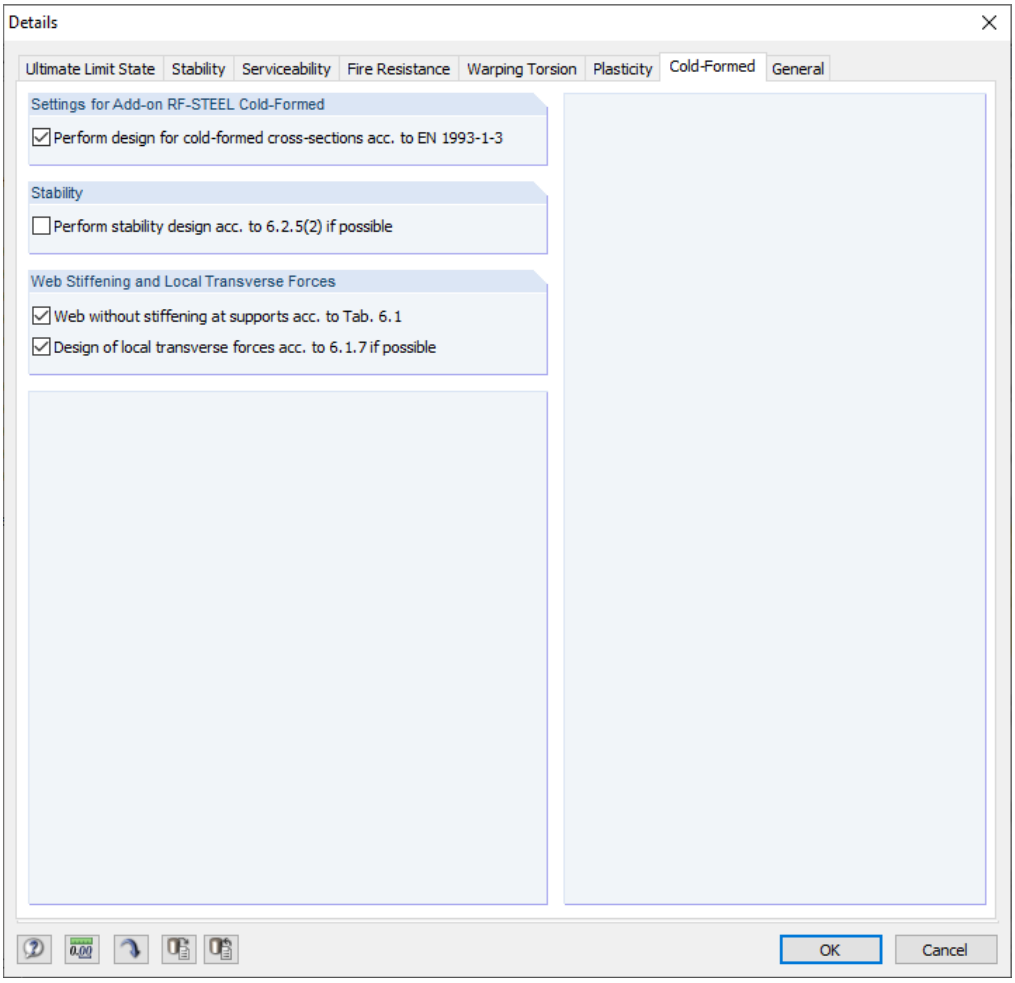


Figure 3.11 Dialog box *Details*, tab *Cold-Formed*

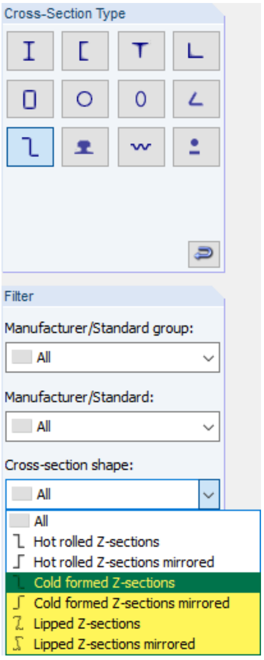
If you want RF-/STEEL EC3 to *Perform design for cold-formed cross-sections acc. to EN 1993-1-3*, select the corresponding check box. Then the other sections of this tab become accessible as well.

An example of the design of a cold-formed C-section is presented in a technical article that you can find in the Knowledge Base on our website. There is also a webinar in which the design of cold-formed sections according to EN 1993-1-3 is discussed.

The design according to EN 1993-1-3 covers sections that are "cold-formed". These are cold-rolled steel products made of thin-walled sheet metal that has been cold-formed by roll-forming or bending methods. Typical cross-section shapes of cold-formed sections are shown in [3] Figure 1.1. The cross-section information in the program also includes the longitudinal stiffenings of the sections and thus the respective buckling panels.

The design of cold-formed sections is described in detail in the [3] standard.

The cross-section library contains various series of C, U, L, and Z-sections that are automatically recognized as "cold-formed" with the corresponding buckling panels and stiffeners. These sections can be filtered according to the *Cross-section shape*.



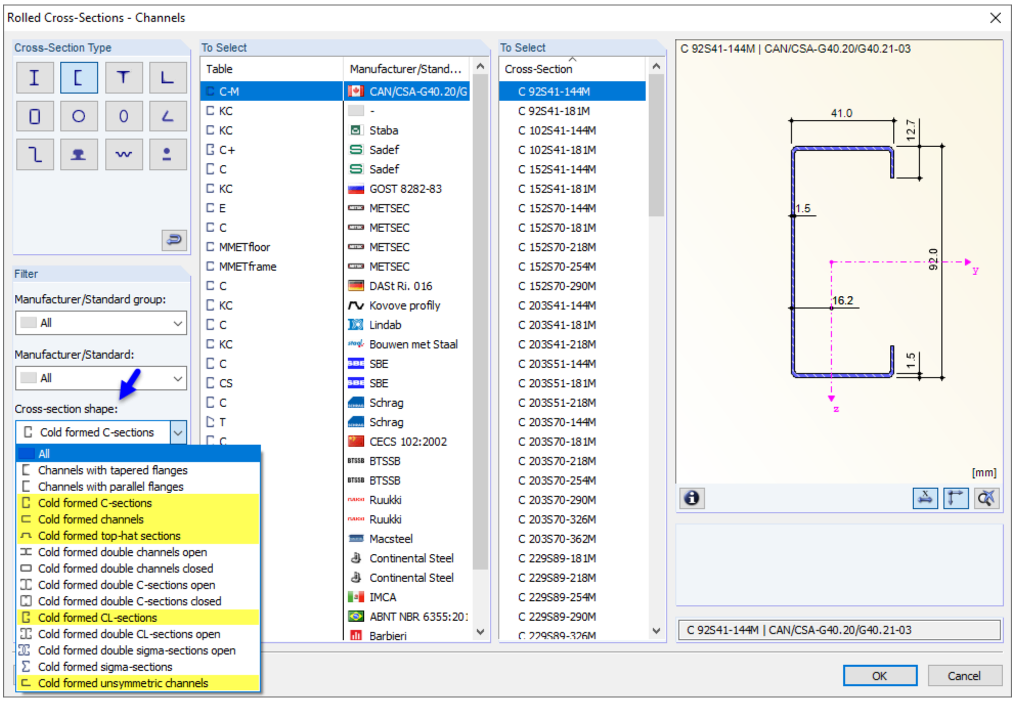


Figure 3.12 Designable cold-formed sections in cross-section library

For the series mentioned above, it is also possible to design parametrized cross-sections. They can be defined using the [Parametric Input] button.

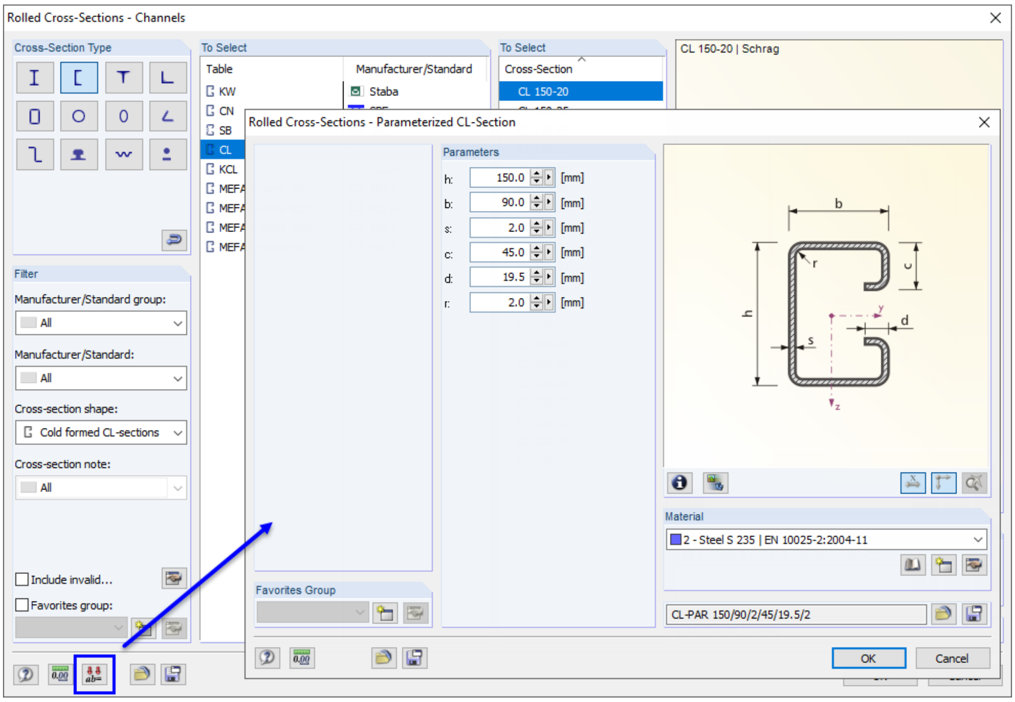


Figure 3.13 Parametric cross-section with buckling panels and stiffeners

Furthermore, it is possible to design cross-sections that were calculated in SHAPE-THIN 9 with the corresponding buckling panel and stiffener definitions according to EN 1993-1-3.

Cold-formed circular and rectangular sections are not part of the EN 1993-1-3 standard.

All cross-sections of the design case that do not fulfill the "cold-formed" criterion are analyzed according to EN 1993-1-1 [1].

If the *Perform stability design acc. to 6.2.5(2) if possible* check box is selected, the stability analysis for bending and axial compression force is performed according to the following interaction relation:

$$\left(\frac{N_{Ed}}{N_{b,Rd}}\right)^{0.8} + \left(\frac{M_{Ed}}{M_{b,Rd}}\right)^{0.8} \leq 1.0$$

where

$N_{b,Rd}$ Resistance of a component subjected to compression according to [3] 6.2.2

$M_{b,Rd}$ Moment resistance according to [3] 6.2.4

This alternative mentioned in [3] 6.2.5 (2) replaces a structural component calculation according to the second-order analysis as per EN 1993-1-1 with the effective cross-sections according to [3] 5.5. For biaxial bending, however, a calculation of structural components as per [3] 6.2.5 (1) according to the second-order analysis is required in order to determine the interaction between axial force and bending moment. This has to be taken into account when selecting load cases and combinations in Window *1.1 General Data*.

The *Web without stiffening at supports acc. to Tab. 6.1* check box affects the value of the shear buckling strength f_{bv}. According to [3] 6.1.5, starting from a web slenderness ratio of 1.4, the geometrical conditions in the form of stiffeners on the support have to be considered accordingly so that web deformations (local buckling) are avoided and the support forces are absorbed.

With the *Design of local transverse forces acc. to 6.1.7 if possible* check box, you can control whether the program also analyzes local failure modes in the web that occur due to support forces or local transverse forces through the flange into the web. [3] 6.1.7 describes different cases and design conditions that have to be fulfilled for the web loading. The boundary conditions, such as the length of the stiff bearing, can be defined in Window *1.14 Local Transverse Forces* (see Chapter 2.14).

3.1.8 General

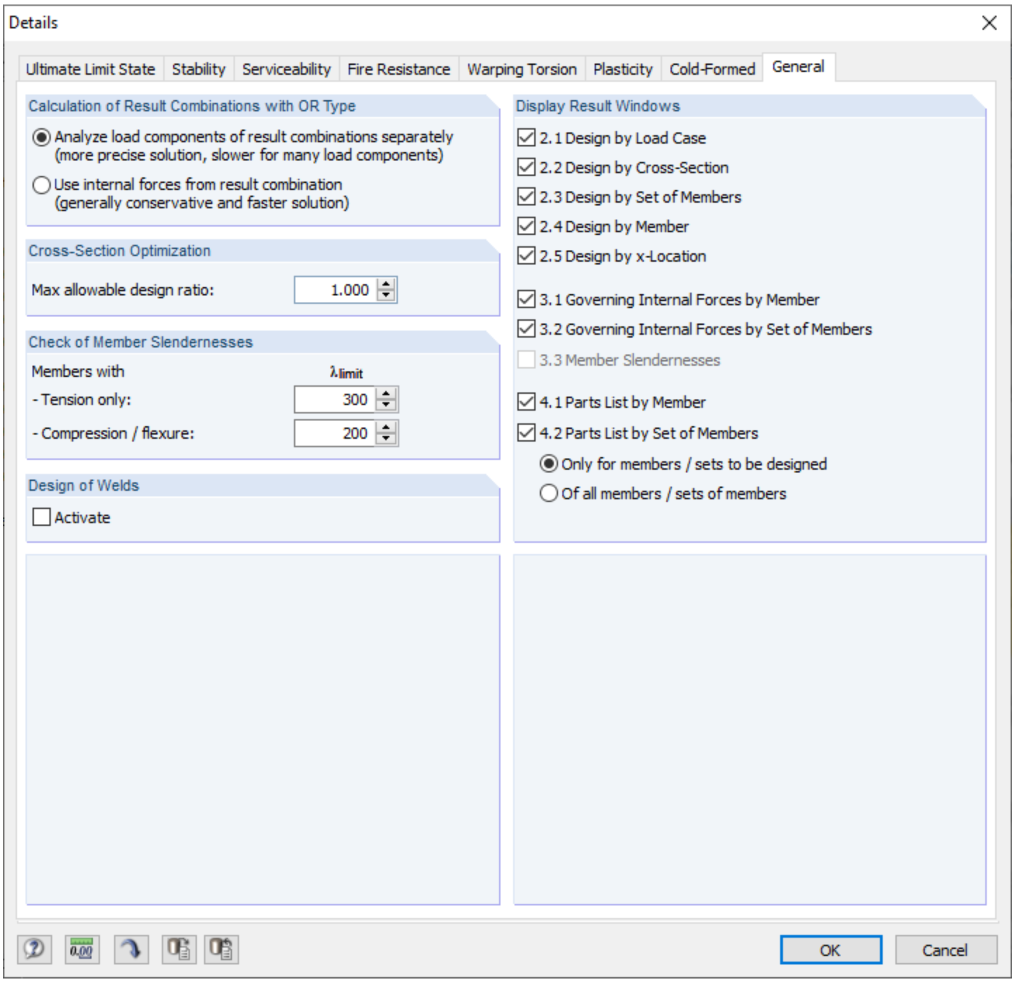


Figure 3.14 Dialog box *Details*, tab *General*

Calculation of Result Combinations with OR Type

If combinations are created automatically, usually many load combinations (CO) are produced. Generally, these combinations are summarized in a result combination (RC) as alternatively acting in an 'OR' connection which provides the envelope: CO1/p or CO2/p or CO3/p or CO4/p etc. For the design of these result combinations, you have two possibilities in RF-/STEEL EC3.

The *load components* of the contained combinations can be analyzed *separately*. Thus, the elastic critical moments for lateral-torsional buckling are determined separately for each constellation, and the designs are performed accordingly. This approach provides exact results. However, it is very time-consuming and requires a high computational effort.

Alternatively, it is possible to *Analyze result combinations generally*. This calculation runs considerably faster because RF-/STEEL EC3 uses only the extreme values with the corresponding internal forces for the design. However, the result may be incorrect if the RC includes a combination where several internal forces (such as N and M_y) are together just below the extreme values.

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 values λ_{limit} in order to define the member slendernesses. Separate specifications are possible for members with tension forces only and for members with bending and compression.

In Window 3.3, the limit values are compared to the real member slendernesses. That window is available after the calculation (see Chapter 4.8) when the corresponding check box in the *Display Result Windows* section to the right has been selected.

Design of Welds

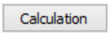
When you select this check box, weld designs will be performed in the course of the analysis. The program performs the typical designs according to EN 1993-1-8 [12]. After the calculation, you find the results listed among the cross-section designs (see also the article in the Knowledge Base on our website).

Display Result Windows

In this dialog section, you can select which result windows including parts list are displayed. The windows are described in Chapter 4.

Window 3.3 *Member Slendernesses* is deactivated by default.

3.2 Starting the Calculation



In all input windows of the RF-/STEEL EC3 add-on module, you can start the calculation by clicking 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 or RSTAB calculation to determine the design-relevant internal forces.

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

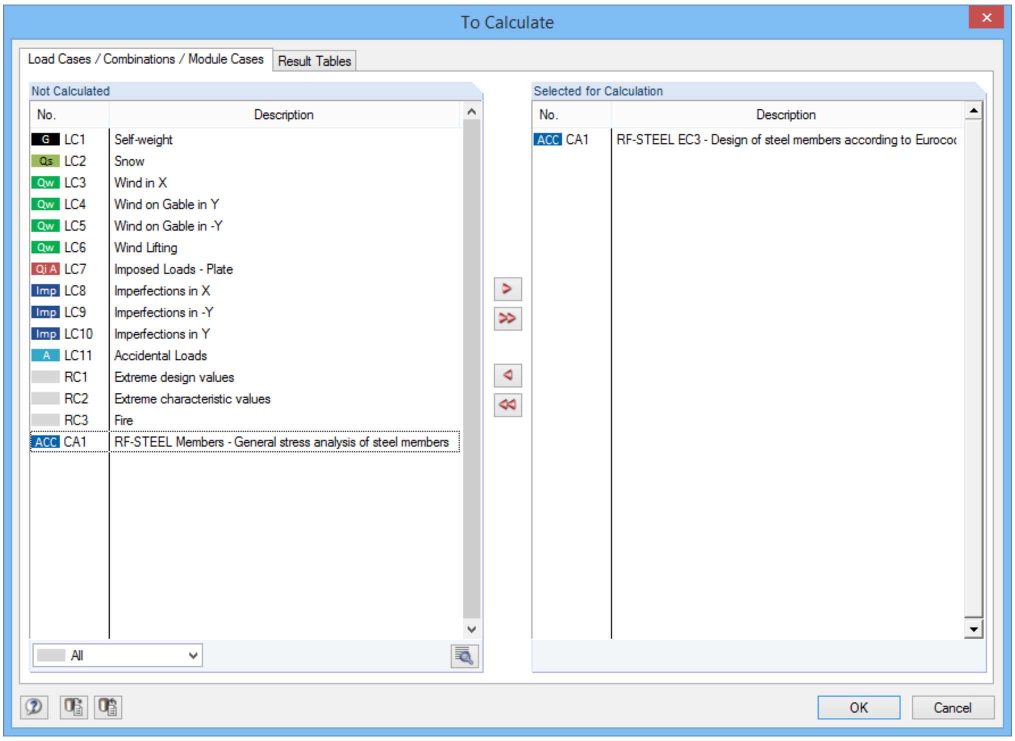
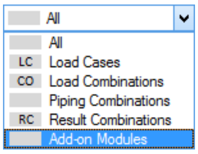


Figure 3.15 Dialog box *To Calculate*



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 list.

To transfer the selected RF-/STEEL EC3 cases to the list on the right, use the [▷] button. Then, click [OK] to start the calculation.

You can also calculate a design case directly by using the list in the toolbar: Set the RF-/STEEL EC3 case and click the [Show Results] button.

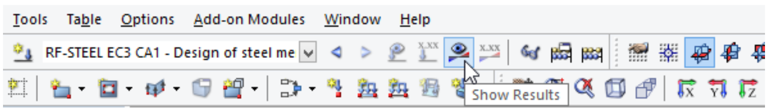


Figure 3.16 Direct calculation of an RF-STEEL EC3 design case in RFEM

Subsequently, you can observe the calculation process in the solver dialog box.

4 Results

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

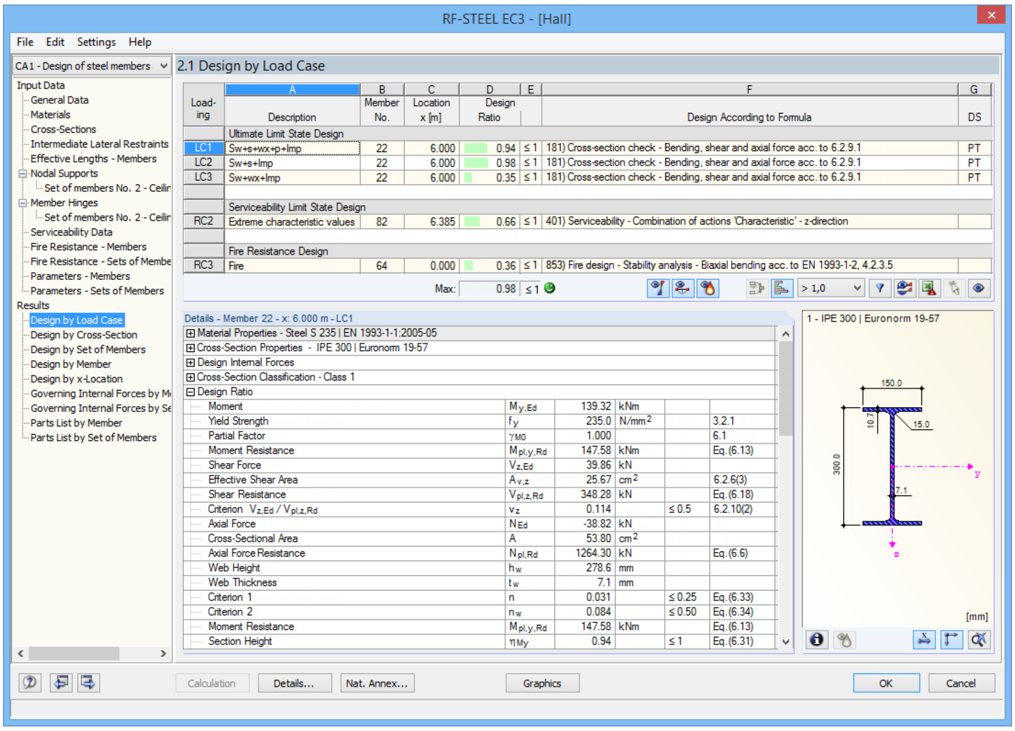


Figure 4.1 Result window with designs and intermediate values

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.

Windows 4.1 and 4.2 show the parts lists by members and sets 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 [F2] and [F3] to go through the windows.

Click [OK] to save the results. Then, you exit RF-/STEEL EC3 and return to the main program.

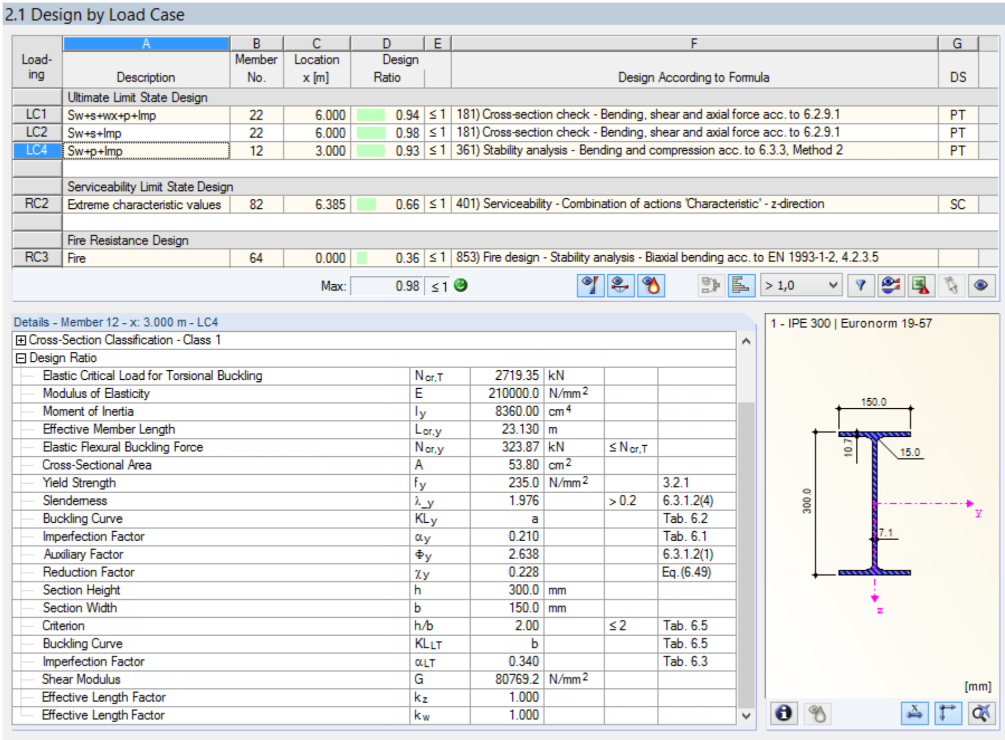
Chapter 4 describes the result windows one by one. Evaluating and checking results is described in Chapter 5.

4.1

Design by Load Case

The upper part of the window shows a summary of the governing designs (i.e. the maximum design ratios for each action), sorted by load case, load combination, and result combination. The table is divided into ultimate as well as serviceability limit state and fire resistance designs.

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



2.1 Design by Load Case

Load-ing	A Description	B Member No.	C Location x [m]	D Design Ratio	E	F Design According to Formula	G DS
	Ultimate Limit State Design						
LC1	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/mm²		
Moment of Inertia	I_y	8360.00	cm⁴		
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	cm²		
Yield Strength	f_y	235.0	N/mm²		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/mm²		
Effective Length Factor	k_z	1.000			
Effective Length Factor	k_w	1.000			

Figure 4.2 *Window 2.1 Design by Load Case*: Maximum Design Ratios for Ultimate Limit State, Serviceability Limit State, and Fire Resistance Designs

Description

This column shows the descriptions of the load cases, load and result combinations for which the designs have been performed.

Member No.

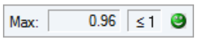
This column shows the number of the member with the maximum design ratio for the designed action.

Location x

This column shows the respective x-location of the member where the maximum design ratio occurs. The following member locations x are used for the table output:

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

Design Ratio



Columns D and E show the design conditions according to [1], [2], and [4].

The length of the colored bar represents graphically the respective design ratio.

Design According to Formula

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

In the *Details* table, the design formulas are shown with the design conditions that are relevant for the selected design.

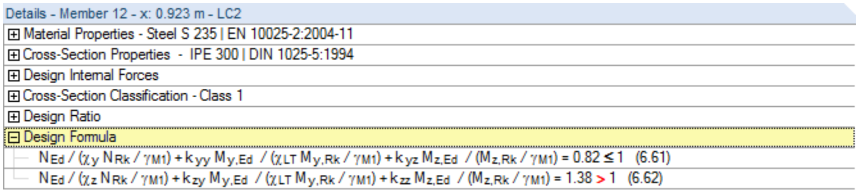

Details - Member 12 - x: 0.923 m - LC2
⊞ Material Properties - Steel S 235 | EN 10025-2:2004-11
⊞ Cross-Section Properties - IPE 300 | DIN 1025-5:1994
⊞ Design Internal Forces
⊞ Cross-Section Classification - Class 1
⊞ Design Ratio
⊟ Design Formula
$N_{Ed} / (\chi_y N_{Rk} / \gamma_{M1}) + k_{yy} M_{y,Ed} / (\chi_{LT} M_{y,Rk} / \gamma_{M1}) + k_{yz} M_{z,Ed} / (M_{z,Rk} / \gamma_{M1}) = 0.82 \leq 1$ (6.61)
$N_{Ed} / (\chi_z N_{Rk} / \gamma_{M1}) + k_{zy} M_{y,Ed} / (\chi_{LT} M_{y,Rk} / \gamma_{M1}) + k_{zz} M_{z,Ed} / (M_{z,Rk} / \gamma_{M1}) = 1.38 > 1$ (6.62)

Figure 4.3 *Design Formula* shown in *Details* table

This function is also described in the Knowledge Base on our website.

DS

Column G provides information on the design relevant situations (DS): *PT* or *AC* for the ultimate limit state, or one of the three design situations for serviceability (*SC, SF, SQ*) according to the specifications in the *1.1 General Data* window (see Figure 2.1).

4.1.1 Warping Torsion

If the *Perform warping analysis* check box in the *Details* dialog box has been selected (see Chapter 3.1.5), the results of the ultimate limit state designs considering seven degrees of freedom appear in this window. The table below shows the intermediate values of the warping torsion analysis.

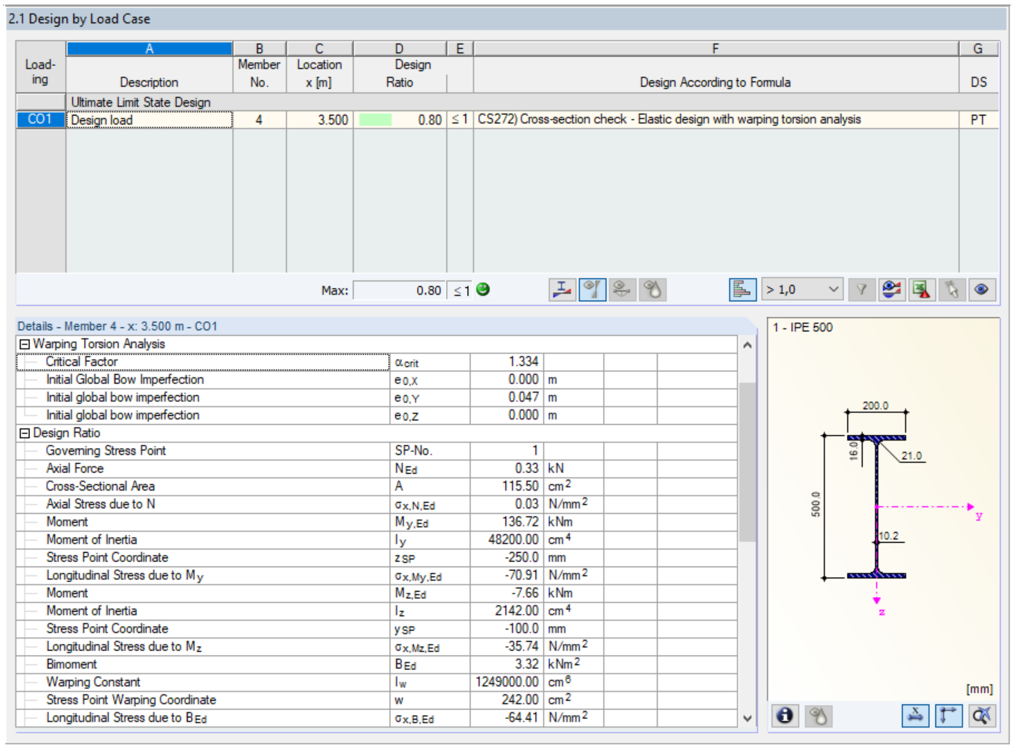


Figure 4.4 Window 2.1 *Design by Load Case* with intermediate values for *Warping Torsion Analysis*

The details contain information about the *Design Internal Forces* of the equivalent loads on the deformed system (see Knowledge Base article on our website). Among other things, the *Critical Factor* α_{crit}, the *Longitudinal stress due to bimoment* B_{Ed}, and the *Secondary Torsional Shear Stress* $\tau_{t,sec,Ed}$ are output as well.

You can use the button to graphically check the mode shapes of the set of members (see Chapter 5.4).

4.1.2 Plasticity

If the *Perform advanced plastic design checks* check box has been activated in the *Details* dialog box (see Chapter 3.1.6), the upper table shows the designs of shear flow in the cross-section parts as well as the designs of the allowable bending moments and axial forces.

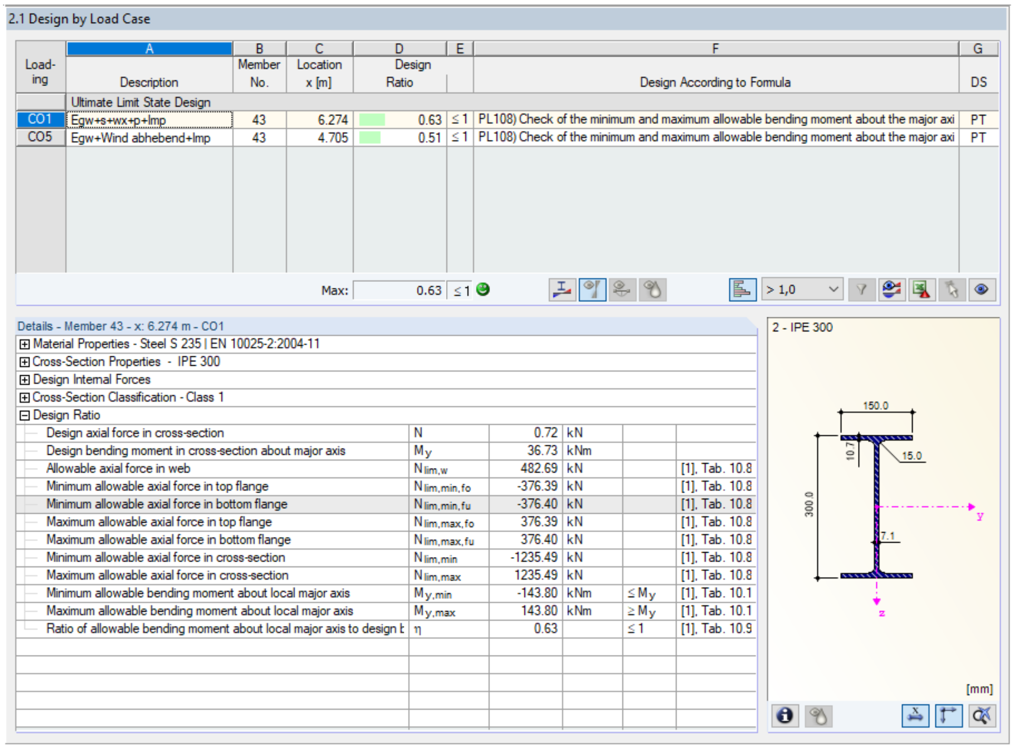
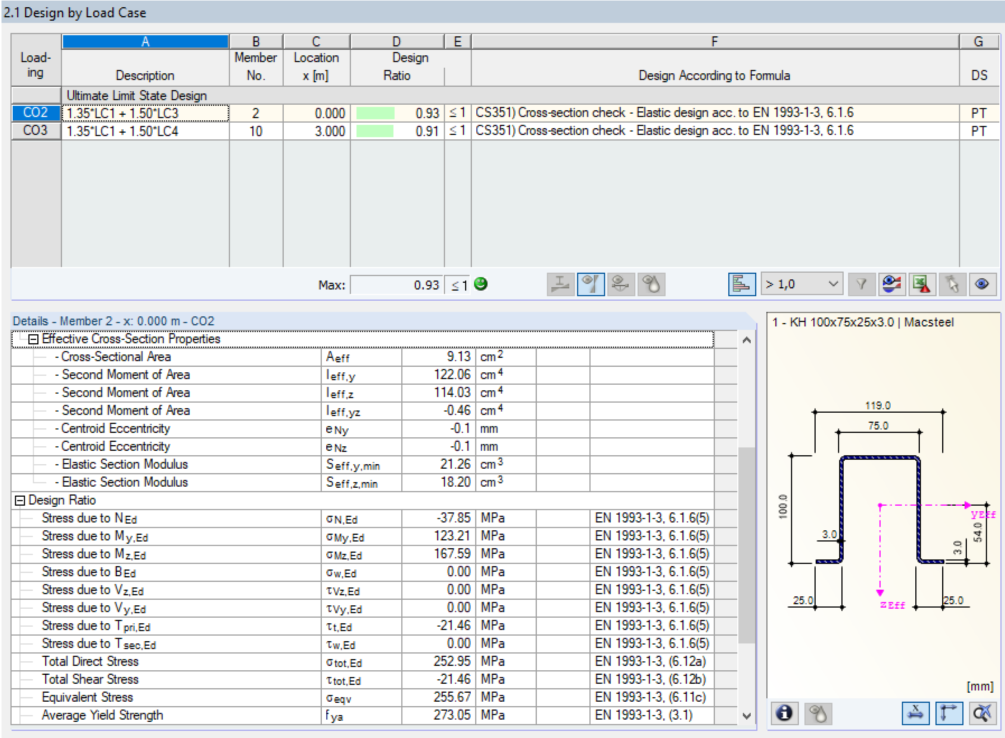


Figure 4.5 Window *2.1 Design by Load Case* with intermediate values for advanced plastic design checks

The table below lists the intermediate values of the plastic design that are available according to the partial internal forces method or the simplex method. These include, for example, the yield strengths of structural components subjected to shear force, the plastic shear force, torsional, and bending resistances, as well as the plastic axial load bearing capacities.

4.1.3 Cold-Formed

If the *Perform design for cold-formed cross-sections* check box in the *Details* dialog box has been selected (see Chapter 3.1.7), the intermediate values required for the design of cold-formed sections according to [3] appear in the table below.



2.1 Design by Load Case

Load-ing	A Description	B Member No.	C Location x [m]	D Design Ratio	E	F Design According to Formula	G DS
	Ultimate Limit State Design						
CO2	1.35*LC1 + 1.50*LC3	2	0.000	0.93	≤ 1	CS351) Cross-section check - Elastic design acc. to EN 1993-1-3, 6.1.6	PT
CO3	1.35*LC1 + 1.50*LC4	10	3.000	0.91	≤ 1	CS351) Cross-section check - Elastic design acc. to EN 1993-1-3, 6.1.6	PT
			Max:	0.93	≤ 1		

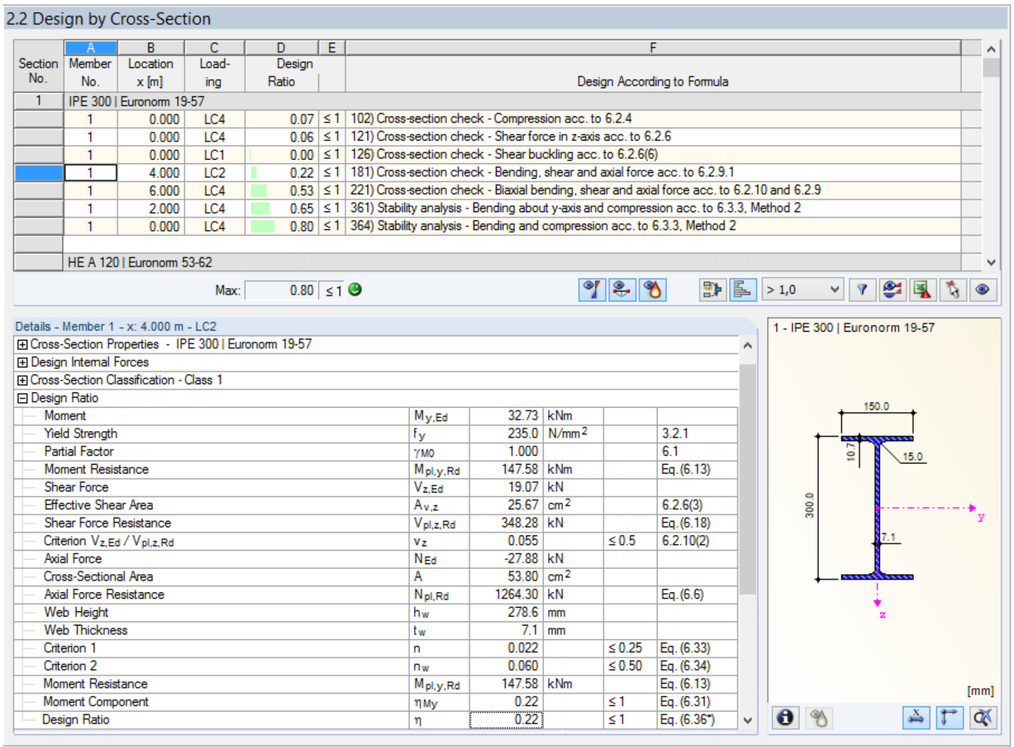
Details - Member 2 - x: 0.000 m - CO2

Effective Cross-Section Properties				
- Cross-Sectional Area	A_{eff}	9.13	cm^2	
- Second Moment of Area	$I_{eff,y}$	122.06	cm^4	
- Second Moment of Area	$I_{eff,z}$	114.03	cm^4	
- Second Moment of Area	$I_{eff,yz}$	-0.46	cm^4	
- Centroid Eccentricity	e_{Ny}	-0.1	mm	
- Centroid Eccentricity	e_{Nz}	-0.1	mm	
- Elastic Section Modulus	$S_{eff,y,min}$	21.26	cm^3	
- Elastic Section Modulus	$S_{eff,z,min}$	18.20	cm^3	
Design Ratio				
Stress due to N_{Ed}	$\sigma_{N,Ed}$	-37.85	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to $M_{y,Ed}$	$\sigma_{My,Ed}$	123.21	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to $M_{z,Ed}$	$\sigma_{Mz,Ed}$	167.59	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to B_{Ed}	$\sigma_{w,Ed}$	0.00	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to $V_{z,Ed}$	$\tau_{Vz,Ed}$	0.00	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to $V_{y,Ed}$	$\tau_{Vy,Ed}$	0.00	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to $T_{pri,Ed}$	$\tau_{t,Ed}$	-21.46	MPa	EN 1993-1-3, 6.1.6(5)
Stress due to $T_{sec,Ed}$	$\tau_{w,Ed}$	0.00	MPa	EN 1993-1-3, 6.1.6(5)
Total Direct Stress	$\sigma_{tot,Ed}$	252.95	MPa	EN 1993-1-3, (6.12a)
Total Shear Stress	$\tau_{tot,Ed}$	-21.46	MPa	EN 1993-1-3, (6.12b)
Equivalent Stress	σ_{eqv}	255.67	MPa	EN 1993-1-3, (6.11c)
Average Yield Strength	f_{ya}	273.05	MPa	EN 1993-1-3, (3.1)

Figure 4.6 Window 2.1 *Design by Load Case* with intermediate values for *Effective Cross-Section Properties* of a cold-formed section

The effective cross-section properties are output separately for the effects of axial force, bending about y, and bending about z. For the stability analysis, the interaction between axial force and bending moment with the effective cross-section values according to EN 1993-1-1 [1] or the interaction relation [3] (6.36) is used, depending on the specification.

4.2 Design by Cross-Section



In this results 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

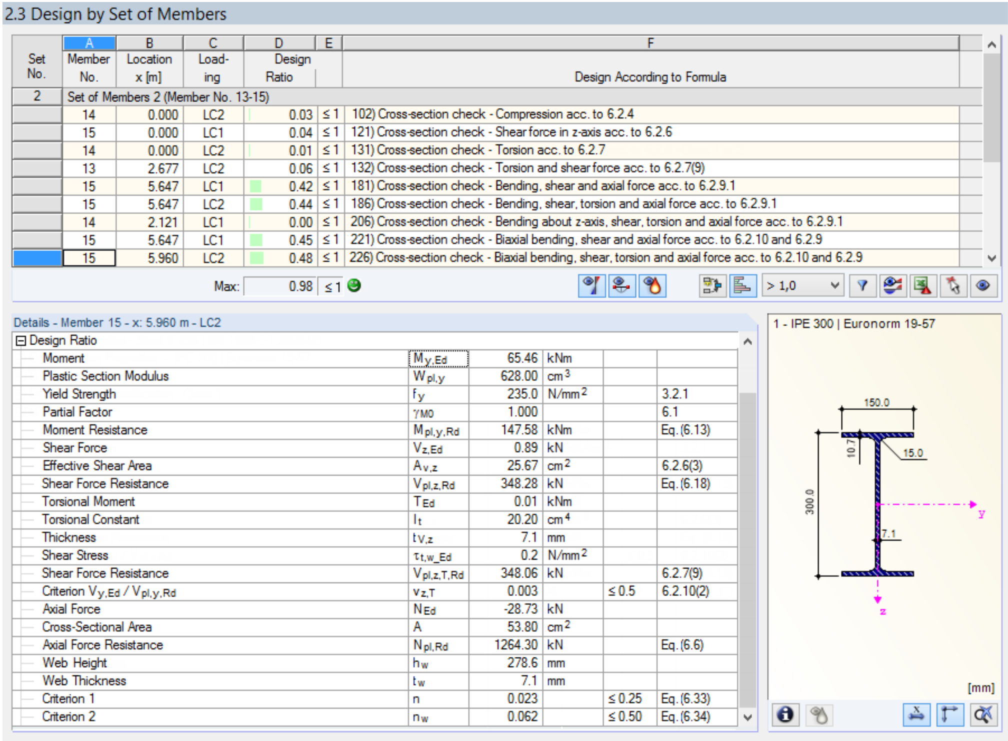


Figure 4.8 Window 2.3 *Design by Set of Members*

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

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

The output by set of members allows you to clearly present the design of an entire structural group (a frame, for example).

4.4 Design by Member

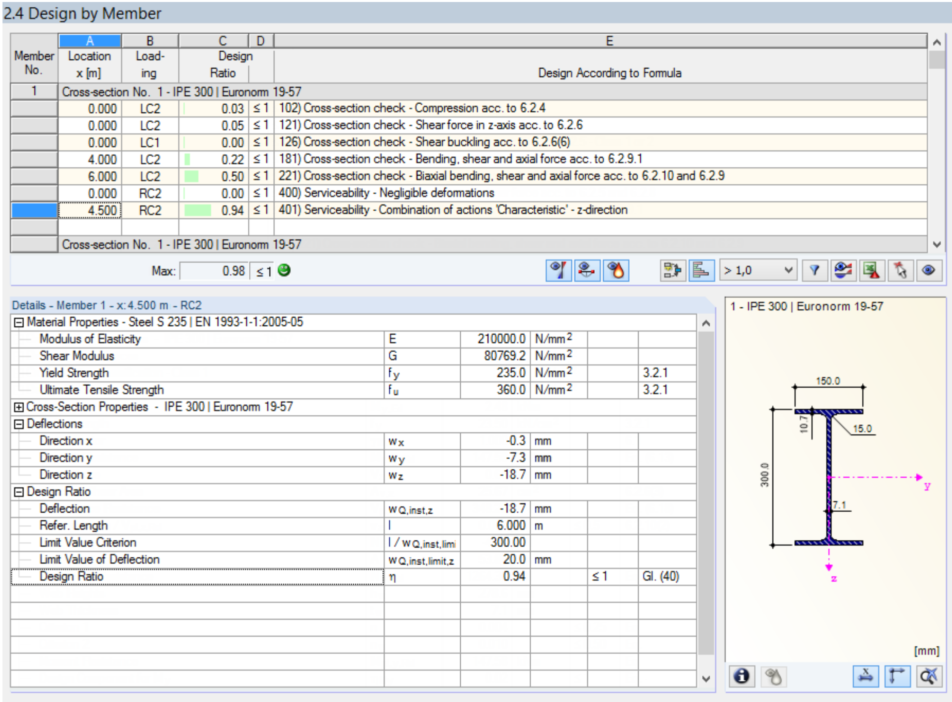


Figure 4.9 Window 2.4 *Design by Member*

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

4.5 Design by x-Location

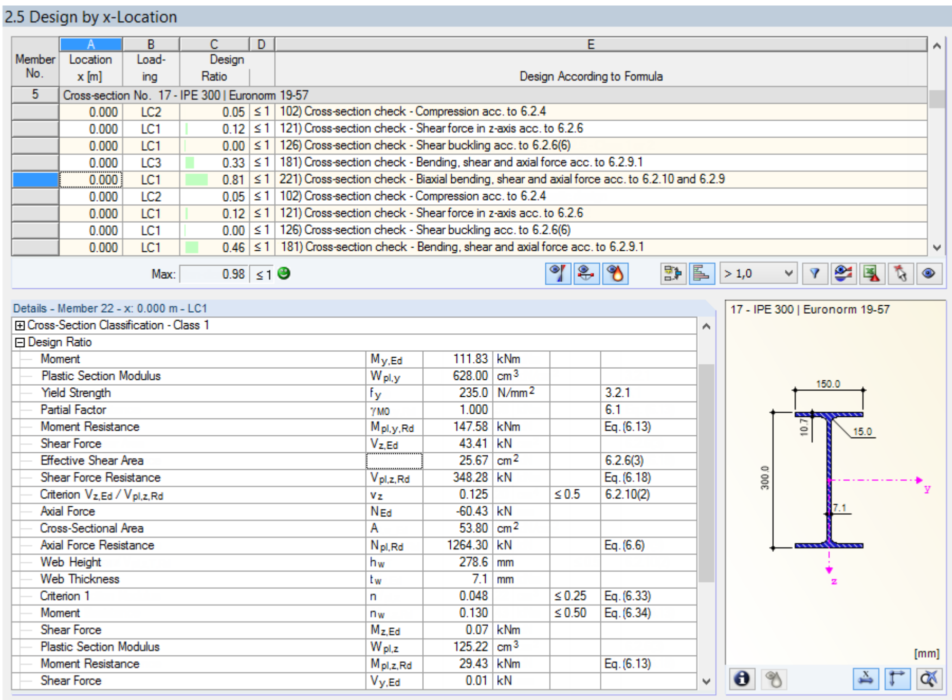


Figure 4.10 Window 2.5 *Design by x-Location*

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

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

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

Member No.	A Location x [m]	B Loading	C Forces [kN] N_{Ed}	D $V_{y,Ed}$	E $V_{z,Ed}$	F Moments [kNm] T_{Ed}	G $M_{y,Ed}$	H $M_{z,Ed}$	I Design According to Formula
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 forc
	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 axi
	6.000	EK3	-21.43	0.04	20.95	0.00	75.65	-0.18	721) Fire design - Cross-section check - Biaxial bending, shear a
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 forc
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 forc
13	Cross-section No. 3 - IPE 400 | Euronorm 19-57 ... 2 - IPE 300 | Euronorm 19-57								

Figure 4.11 Window 3.1 *Governing Internal Forces by Member*

For each member, this window displays the governing internal forces, that is, the forces and moments that result in the maximum utilization in the individual designs.

Location x

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

Loading

This column shows the numbers of the load case as well as the load or result combination whose internal forces result in the maximum design ratio.

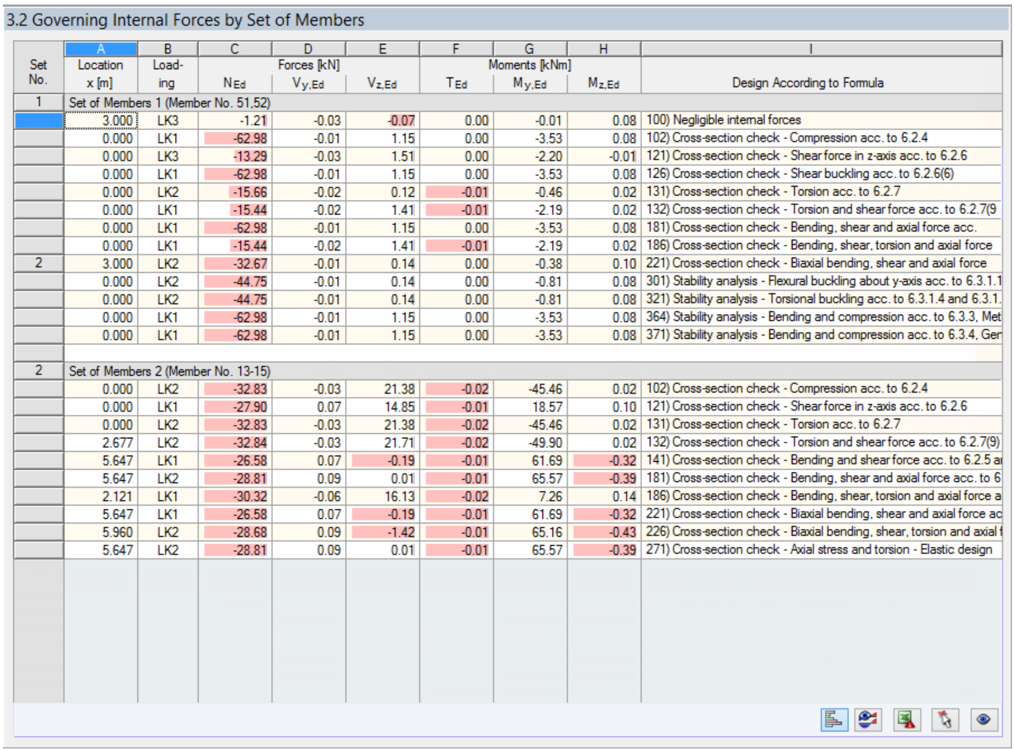
Forces / Moments

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 final column gives information on the design types and equations used for performing the designs according to [1], [2] or [4].

4.7 Governing Internal Forces by Set of Members

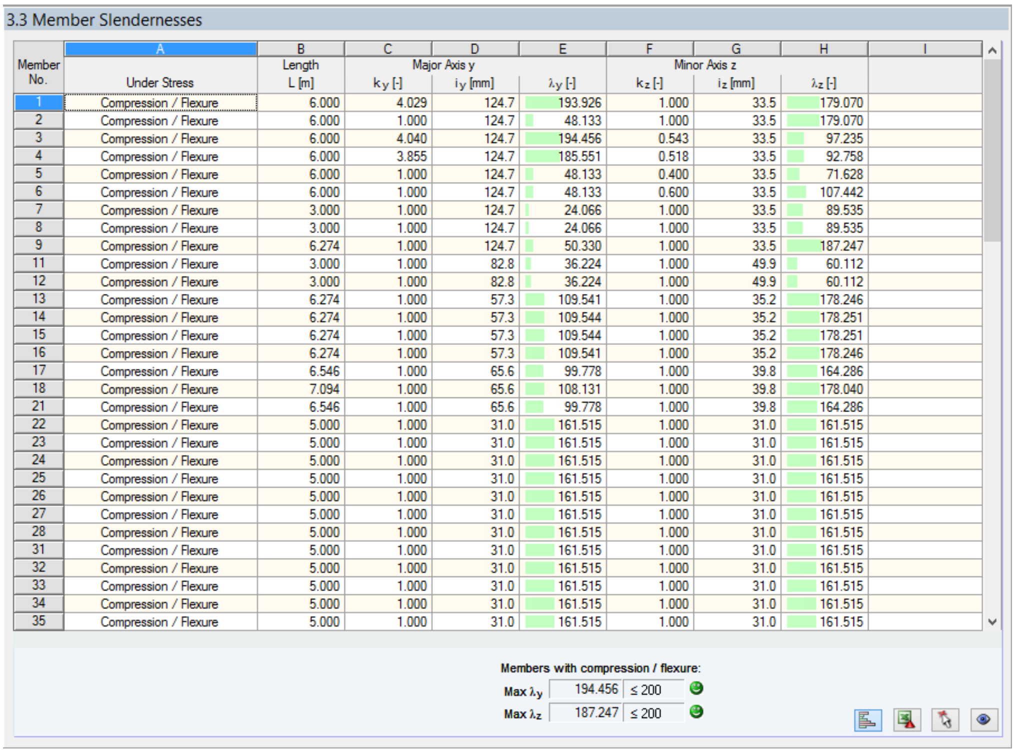

3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [m]	B Load-ing	C Forces [kN] N_{Ed}	D $V_{y,Ed}$	E $V_{z,Ed}$	F Moments [kNm] T_{Ed}	G $M_{y,Ed}$	H $M_{z,Ed}$	I 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	-15.66	-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.
	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 a
	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
	2.121	LK1	-30.32	-0.06	16.13	-0.02	7.26	0.14	186) Cross-section check - Bending, shear, torsion and axial force a
	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 ac
	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 f
	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 4.12 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 for the individual designs.

4.8 Member Slendernesses


3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C Major Axis y k_y [-]	D i_y [mm]	E λ_y [-]	F Minor Axis z k_z [-]	G i_z [mm]	H λ_z [-]	I
1	Compression / Flexure	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 4.13 *Window 3.3 Member Slendernesses*

Details...

This result window is displayed when the corresponding check box has been selected in the *General* tab of the *Details* dialog box (see Figure 3.14).

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They have been determined as a function of the load type. Below the list, you can see a comparison with the limit values defined in the *General* tab of the *Details* dialog box (see Figure 3.14).

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

This window is only of an informative nature. 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

	A	B	C	D	E	F	G	H	I
Part No.	Cross-Section Description	Number of Members	Length [m]	Total Length [m]	Surface Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Total Weight [t]
1	1 - 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	0.399
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 4.14 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, you can set it in the *General* tab of the *Details* dialog box (see Figure 3.14).

Part No.

The program 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

The values in this column are the product from the previous two columns.

Surface Area

For each part, the program displays the surface areas relative to the total length. The surface area is determined from the *Surface* of the cross-section that can be found in Windows 1.3 and 2.1 to 2.5 in the cross-section info (see Figure 2.20).

Volume

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

Unit Weight

The *Unit Weight* represents the cross-section weight 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 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 shown in columns B, D, E, F, and I. The last row of the *Total Weight* column shows the required total amount of steel.

4.10 Parts List by Set of Members

4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Set 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 4.15 Window *4.2 Parts List by Set of Members*

The last result window is displayed if at least one set of members has been selected for design. It gives an overview of the steel parts of entire structural groups such as horizontal beams.

The columns are described in the previous chapter. If there are different cross-sections within a set of members, the program averages the surface area, the volume and the cross-section weight.

5 Results Evaluation

You can evaluate the design results in different ways. The buttons below the upper table may help you.

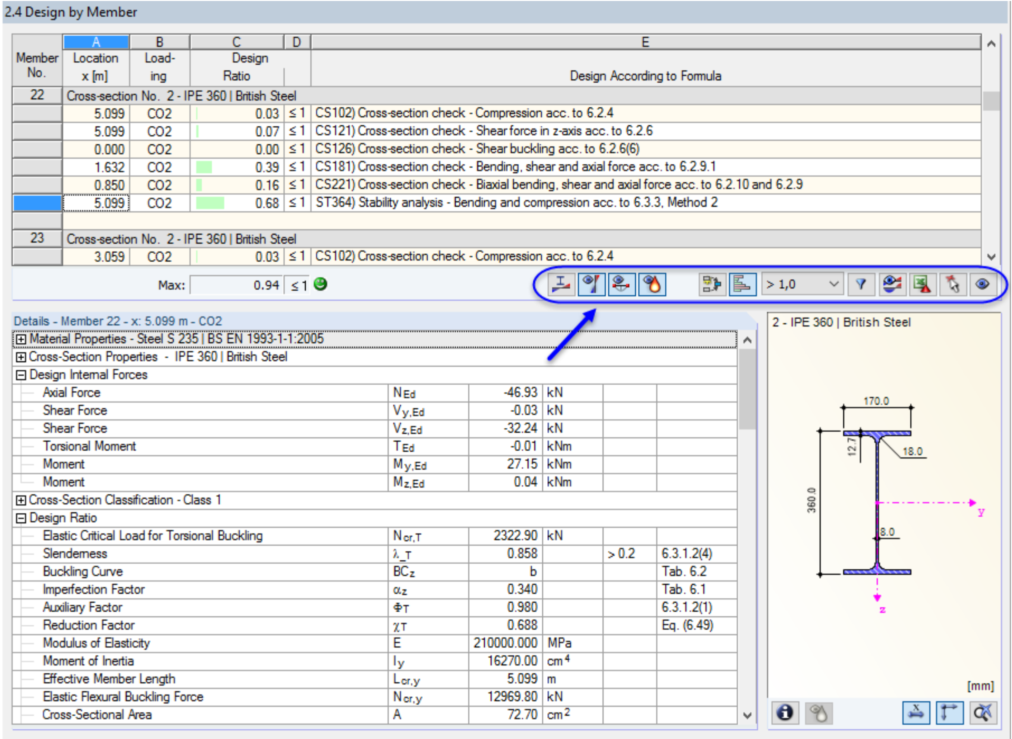


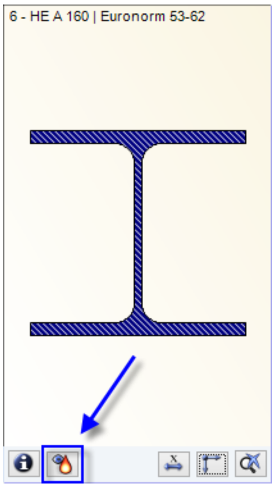
Figure 5.1 Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Mode shapes	Opens *Mode Shape View* window → Chapter 5.4
	Ultimate limit state	Displays or hides results of ultimate limit state design
	Serviceability	Displays or hides results of serviceability limit state design
	Fire resistance	Displays or hides results of fire protection design
	Result combination	Creates a new result combination from governing load cases and load combinations
	Color bars	Displays or hides colored relation scales in result windows

> 1,0 / > 1,0 / Max / Define	Filter parameters	Describes criterion by which results are filtered in tables: ratios greater than 1, maximum value, or user-defined limit
	Apply filter	Shows only rows to which filter parameters apply (ratio > 1, maximum, defined value)
	Result diagrams	Opens *Result Diagram on Member* window → Chapter 5.2
	Excel export	Exports table to MS Excel → Chapter 7.4.3
	Select member	Selects a member graphically to display its results in the table
	View mode	Jumps to RFEM or RSTAB work window to change the view

Table 5.1 Buttons in the result Windows 2.1 to 2.5



When evaluating fire resistance designs, you can have a look at the used temperature-time curve: Click the button below the cross-section graphic to open the *Fire Curves* diagram, as shown in Figure 3.6 to Figure 3.8.

5.1 Results on RFEM/RSTAB Model

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

Background graphic and view mode

The RFEM/RSTAB 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 color in the background graphic. Moreover, an arrow indicates the member's x-location selected in the active table row.

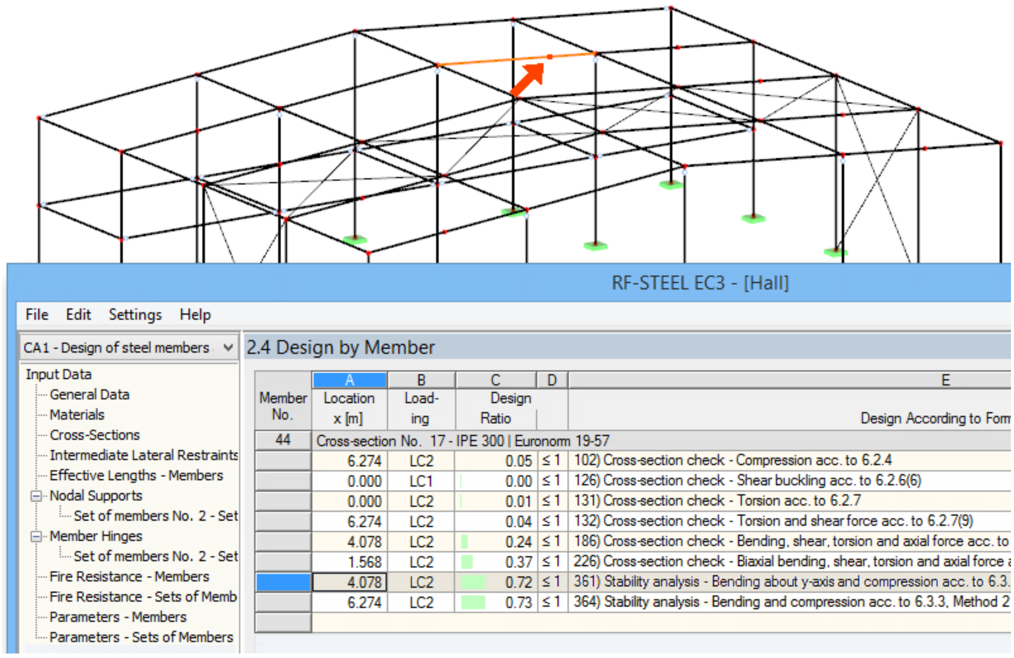
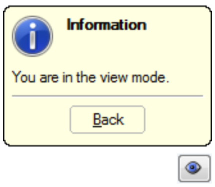


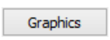
Figure 5.2 Indication of member and current *Location x* in RFEM model



In case you cannot improve the model display by moving the RF-/STEEL EC3 module window, click the [Jump to graphic] button to activate the *view mode*: The program hides the module window so that you can adjust the view in the RFEM/RSTAB work window. The view mode provides the functions of the *View* menu, for example, zooming, moving or rotating the model view. The indicating arrow remains visible.

Click [Back] to return to the RF-/STEEL EC3 add-on module.

RFEM/RSTAB work window



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

In the *Results* navigator, you can select the design ratios separately for the ultimate and the serviceability limit state design as well as the fire protection design. It is also possible to check the classifications of cross-sections.

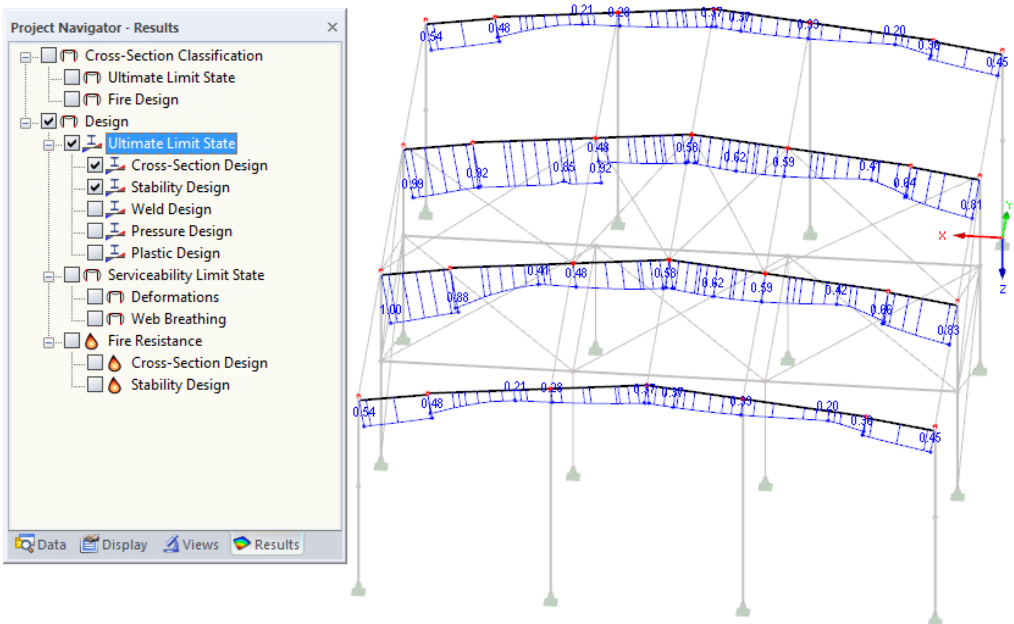
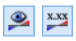
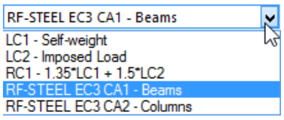


Figure 5.3 *Results* navigator for RF-/STEEL EC3



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

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



You can set the design cases in the drop-down list of the RFEM/RSTAB menu bar.

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

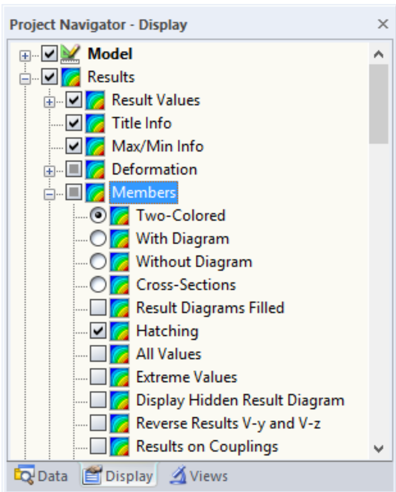


Figure 5.4 *Display* navigator: *Results → Members*

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

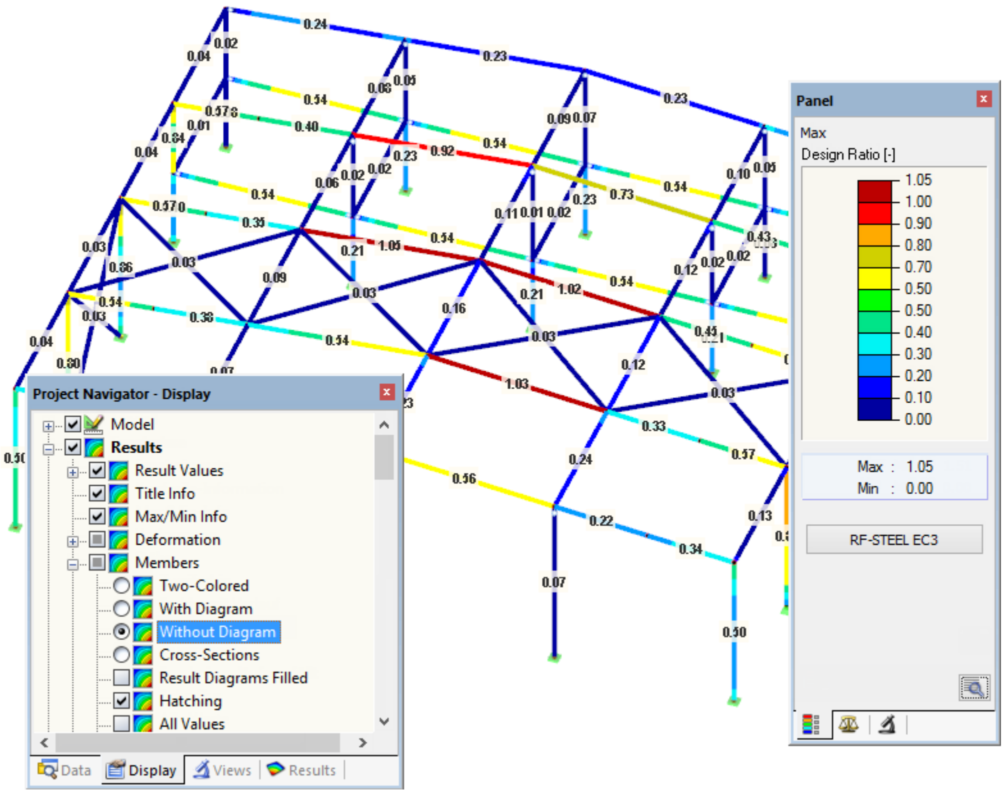


Figure 5.5 Design ratios with display option *Without Diagram*

It is possible to transfer the graphics of design results to the printout report (see Chapter 6.2).

RF-STEEL EC3

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

If results of the **RF-/STEEL Warping Torsion** module extension are available, the corresponding internal forces can also be checked on the model. The *Results* navigator provides additional entries for this.

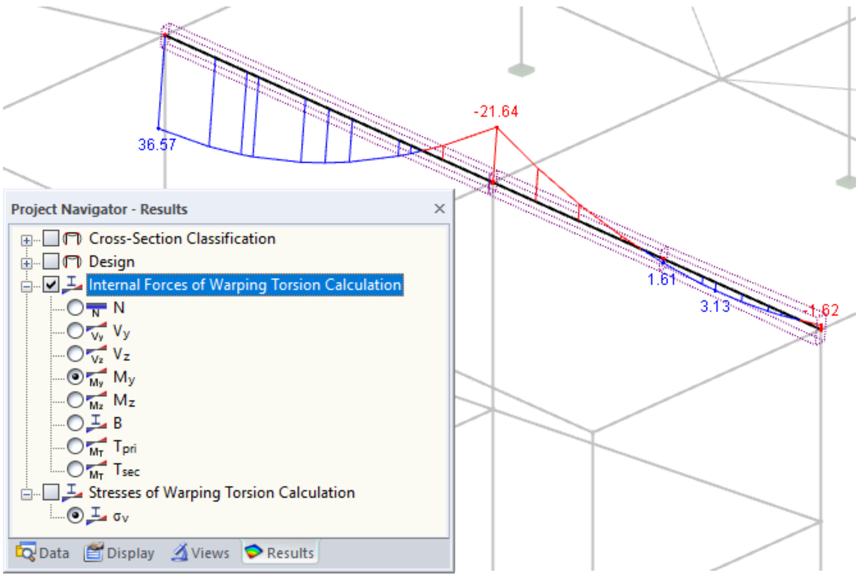


Figure 5.6 Internal Forces of Warping Torsion Calculation

5.2 Result Diagrams

You can also evaluate the member results graphically in the form of result diagrams.

Select the member (or set of members) in the RF-/STEEL EC3 result window by clicking in the member's table row. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. You can find it below the upper result table (see Figure 5.1).

To access the result diagrams in the RFEM/RSTAB graphic, select on the menu

Results → Result Diagrams for Selected Members

or use the corresponding button in the toolbar of RFEM or RSTAB.

A window opens which shows graphically the distribution of the design values on the member or set of members.

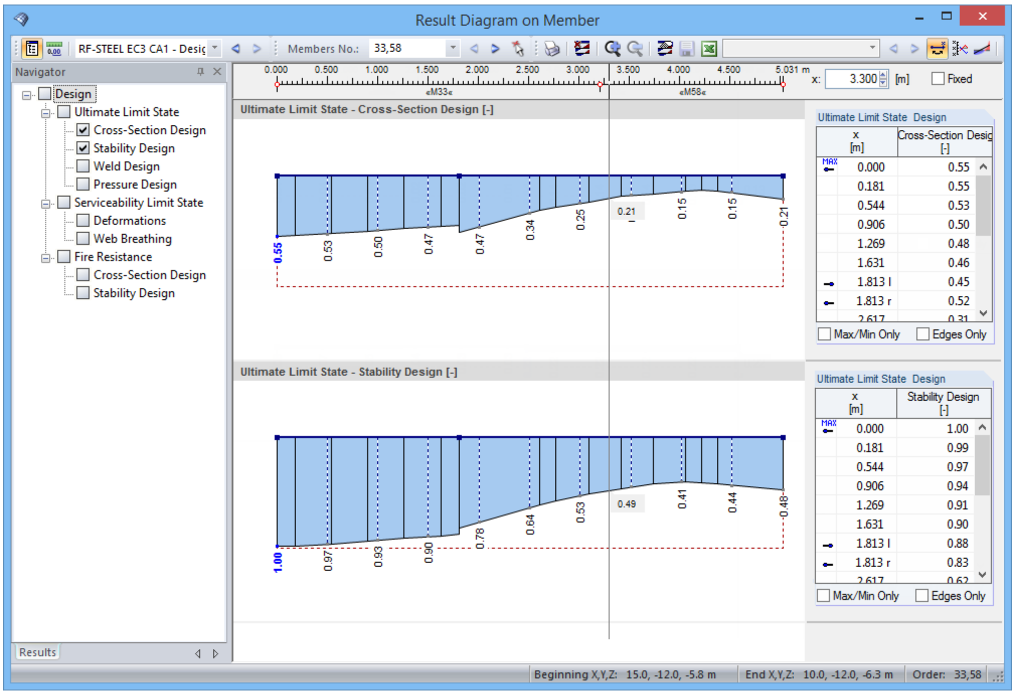
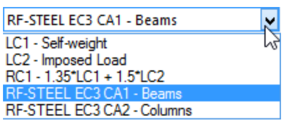


Figure 5.7 Dialog box *Result Diagram on Member*

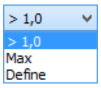


Again, the *Results* navigator allows for a targeted selection among classifications and the designs of the ultimate and the serviceability limit state as well as of fire resistance.

Use the list in the toolbar to switch between the RF-/STEEL EC3 design cases.

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

5.3 Filter for Results

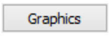


The arrangement of the RF-/STEEL EC3 result windows already provides a selection by various criteria. In addition, there are filter options for the tables (see Figure 5.1) to limit the numerical output by design ratios. This function is also described in the Knowledge Base on our website.

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

The possibilities offered by the *Visibility* function (see Chapter 9.9.1 in RFEM manual, or Chapter 9.7.1 in RSTAB manual) are also available for RF-/STEEL EC3 to filter the members for the evaluation.

Filtering designs



The design ratios can easily be used as filter criteria in the work window of RFEM or RSTAB that you can access with the [Graphics] button. To apply this function, the panel must be displayed. If it is not active, select on the RFEM/RSTAB menu **View → Control Panel (Color Scale, Factors, Filter)** or use the button in the toolbar.

The panel is described in Chapter 3.4.6 of the RFEM or RSTAB manual. 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 options *With/Without Diagram* or *Cross-Sections* in the *Display* navigator.

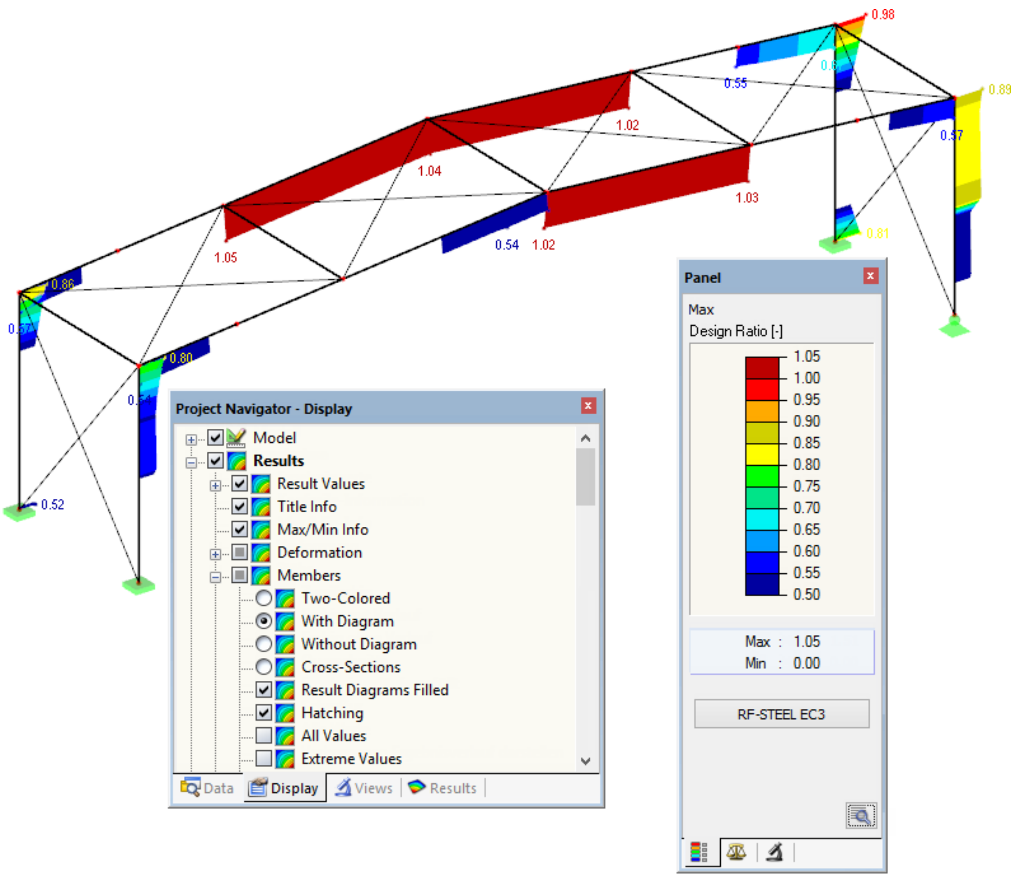


Figure 5.8 Filtering design ratios with adjusted color scale

As shown in Figure 5.8, the panel's scale of values can be set in such a way that only design ratios greater than 0.50 are displayed in a color range between blue and red.

The function *Display Hidden Result Diagram* in the *Display* navigator (**Results → Members**) shows all design ratios which are beyond 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. The function is described in Chapter 9.9.3 of the RFEM manual or in Chapter 9.7.3 of the RSTAB manual.

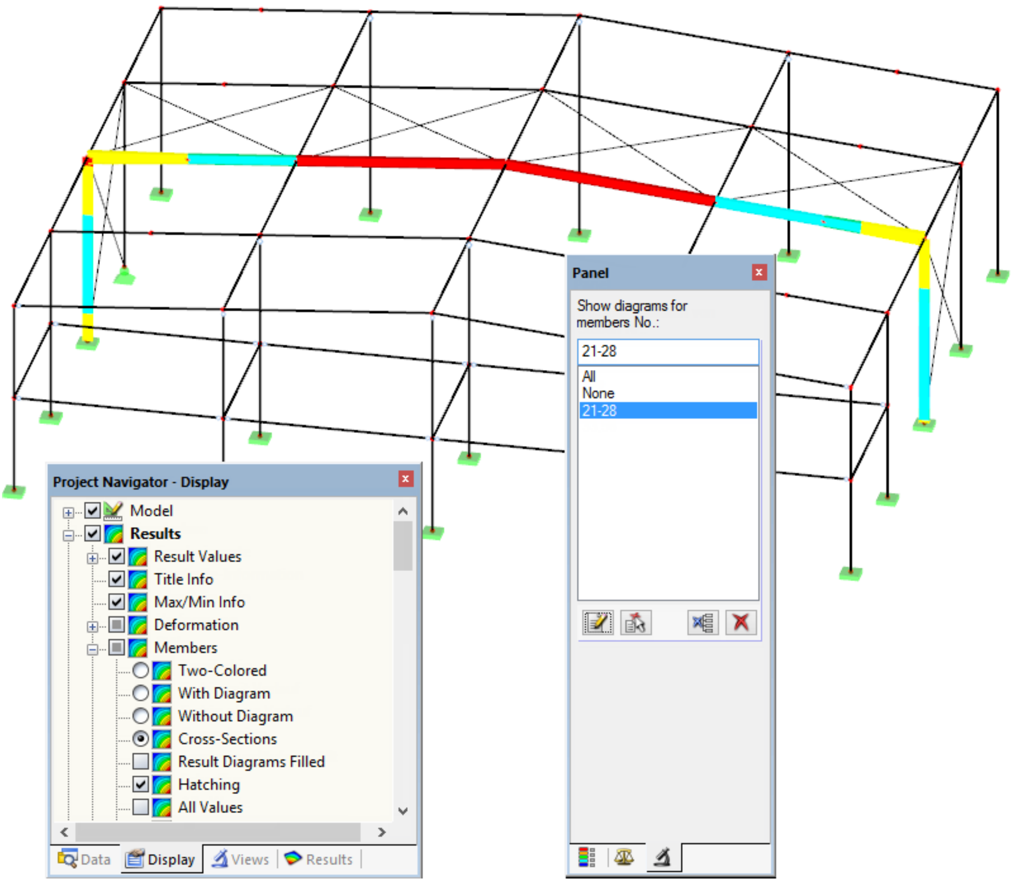


Figure 5.9 Member filter for design ratios of a hall frame

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.

5.4 Mode Shapes

The mode shapes of the sets of members can be checked graphically in a separate window: Select the relevant set of members in the result window, and then click the [Mode Shapes] button.

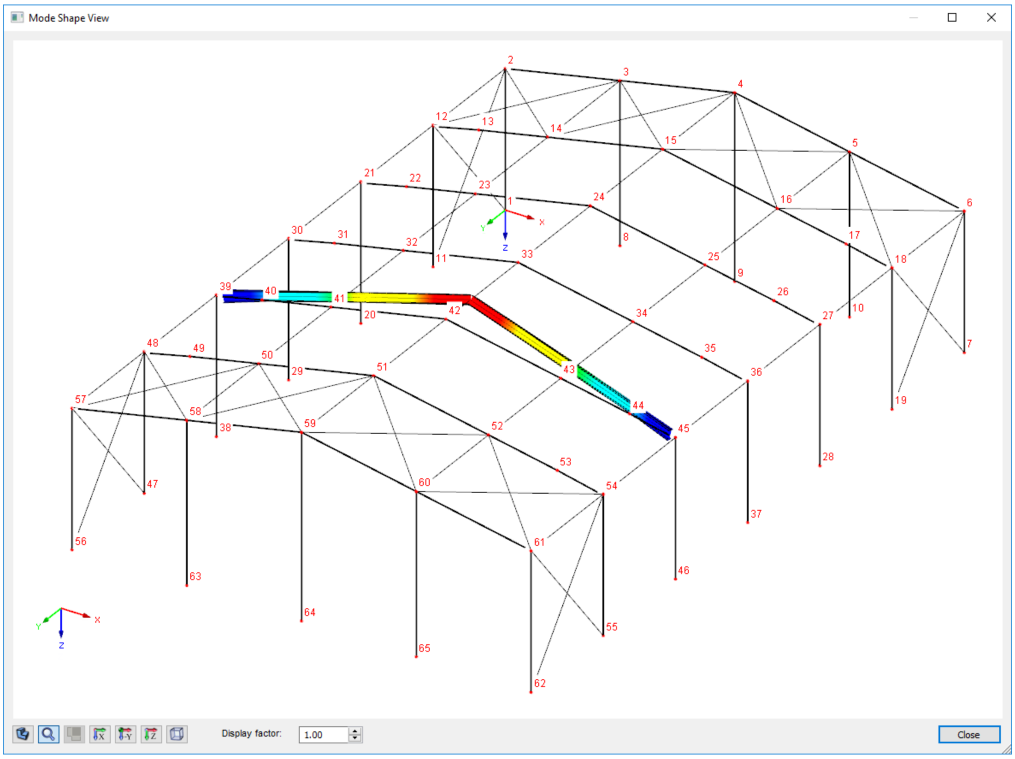


Figure 5.10 Mode shape of a set of members

The mode shapes of the sets of members are created automatically when determining the critical factor α_{cr}. If the *warping analysis* with seven degrees of freedom has been selected (see Chapter 3.1.5), the mode shapes have been considered in the calculation with the rise of the precamber set in Window 1.13 (see Figure 2.52).

The mode shapes are not available in numerical form.

The buttons below the graphic are described in Table 2.4.

Use the *Display factor* to display the mode shape in an exaggerated graphical representation.

With the [Print] button, you can directly print the current graphic or transfer it to the printout report.

The following article presents an example for the mode shape of a tapered steel frame:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001156

6 Printout

6.1 Printout Report

A printout report is generated for the data of the RF-/STEEL EC3 add-on module, like in RFEM or RSTAB, to which you can add graphics and descriptions. The selection in the printout report determines which data from the design module will finally be included in the printout.

The printout report is described in the RFEM or RSTAB manual. Chapter 10.1.3.4 *Selecting Data of Add-on Modules* explains how to prepare input and output data of add-on modules for the printout.

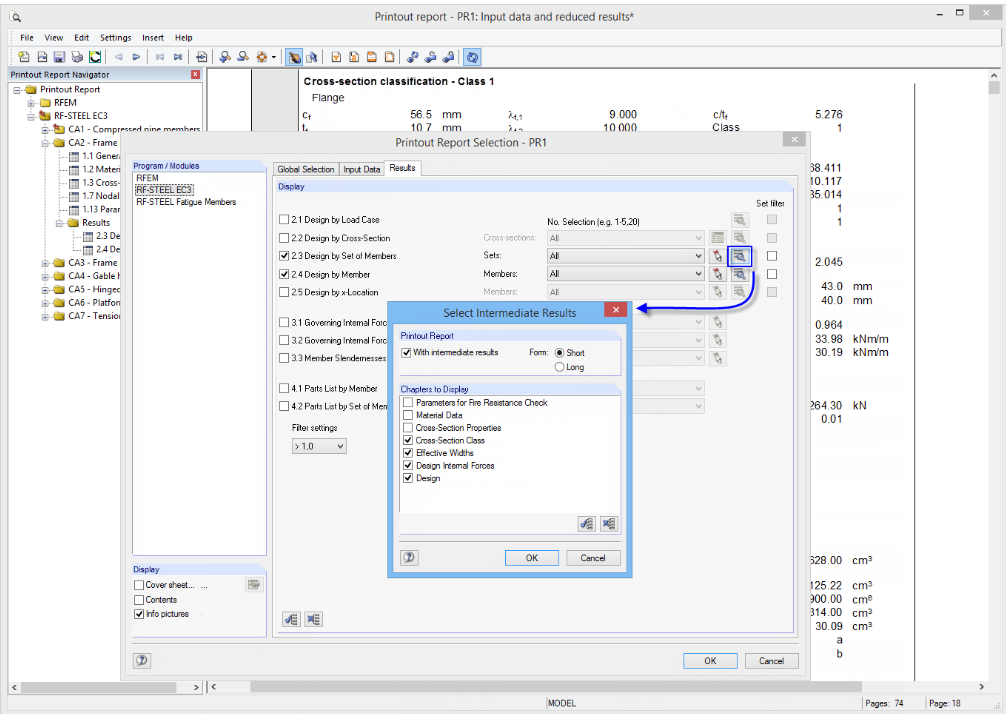


Figure 6.1 Selecting designs and intermediate results in the printout report

Click the [Details] button to specify if the printout also includes intermediate results. They can be defined in a list and documented in a *Short* (compact representation) or *Long* form (list representation).

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

6.2 Graphic Printout

In RFEM and RSTAB, you can transfer every image displayed in the work window to the printout report. It is also possible to send it directly to the printer. Thus, the design ratios displayed in the model can be prepared for the printout, too.

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

Designs in RFEM/RSTAB model

To print the current graphic of design ratios, select on the menu

File → Print Graphic

or use the toolbar button shown on the left.

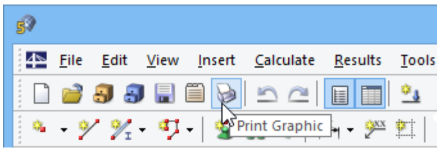


Figure 6.2 *Print Graphic* button in RFEM toolbar

Result diagrams

Also in the *Result Diagram on Member* dialog box, you can send the graphic with design values to the report by clicking the [Print] button. Alternatively, you can print it directly.

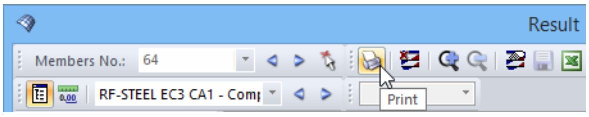


Figure 6.3 *Print* button in *Result Diagram on Member* dialog box

The following dialog box opens:

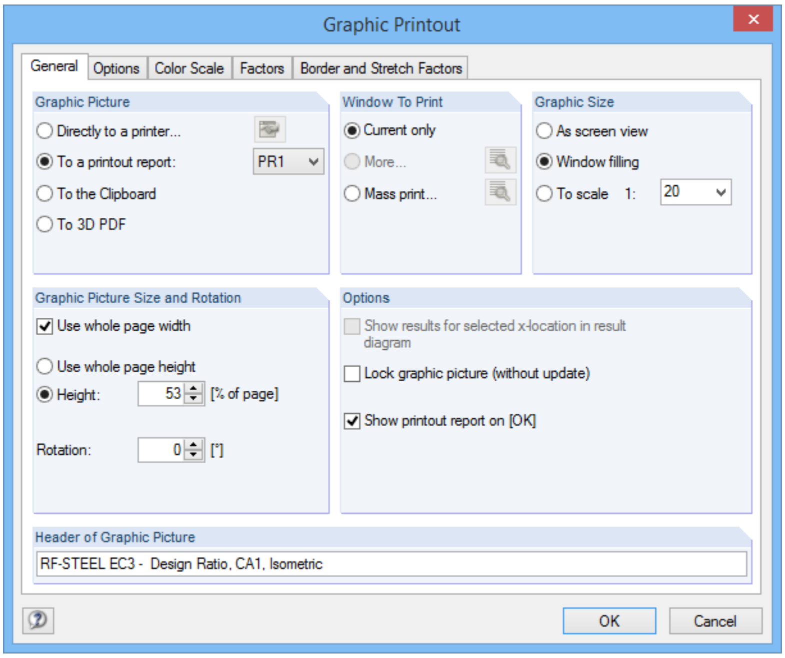
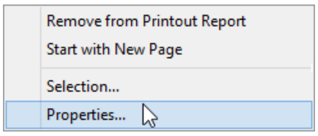


Figure 6.4 Dialog box *Graphic Printout*, tab *General*

The *Graphic Printout* dialog box is described in Chapter 10.2 of the RFEM or RSTAB manual. There, you find also descriptions of the remaining dialog tabs.

To move a graphic within the printout report to another position, use the drag-and-drop function.



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

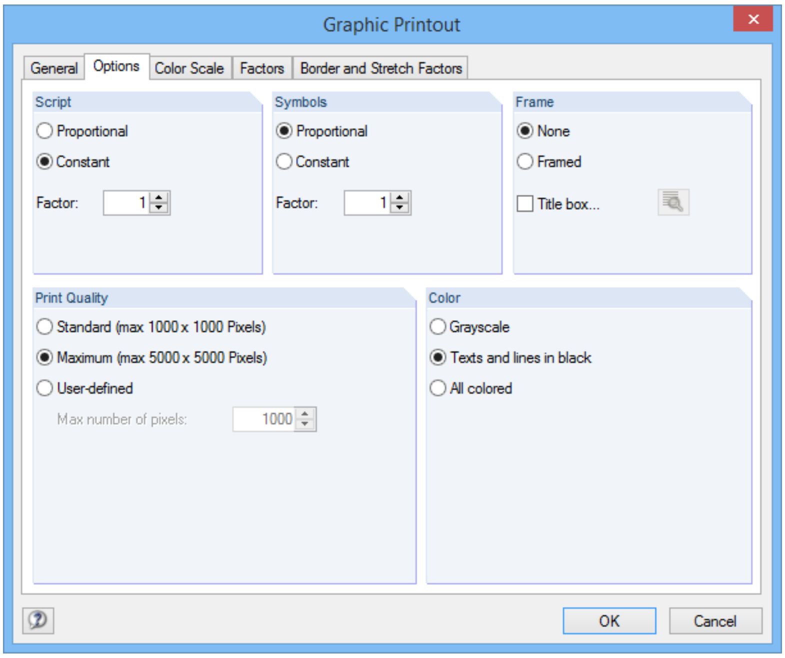


Figure 6.5 Dialog box *Graphic Printout*, tab *Options*

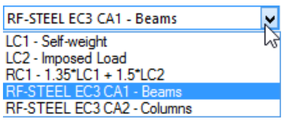
7 General Functions

This chapter describes useful menu functions as well as export options.

7.1 Design Cases

Design cases allow you to group members for the designs. This way, you can consider groups of structural components or analyze members with particular design specifications (for example, changed materials, partial safety factors, optimization).

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



You can access the design cases of RF-/STEEL EC3 also in RFEM or RSTAB by using the load case list of the toolbar.

Create a new design case

To create a new design case, select on the RF-/STEEL EC3 menu

File → New Case.

The following dialog box appears.

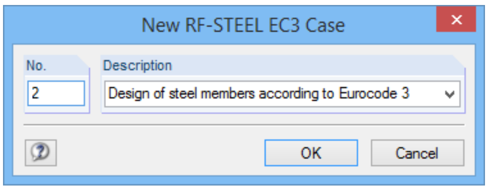


Figure 7.1 Dialog box *New RF-STEEL EC3 Case*

In this dialog box, enter a *No.* (one that is not yet assigned) for the new design case. A *Description* will make the selection in the load case list easier.

After clicking [OK], the RF-/STEEL EC3 Window *1.1 General Data* opens for entering the design data.

Rename a design case

To change the description of a design case, select on the RF-/STEEL EC3 menu

File → Rename Case.

The following dialog box appears.

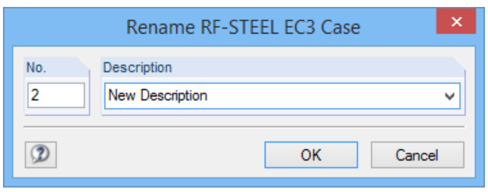


Figure 7.2 Dialog box *Rename RF-STEEL EC3 Case*

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

Copy a design case

To copy the input data of the current design case, select on the RF-/STEEL EC3 menu

File → Copy Case.

The following dialog box appears.

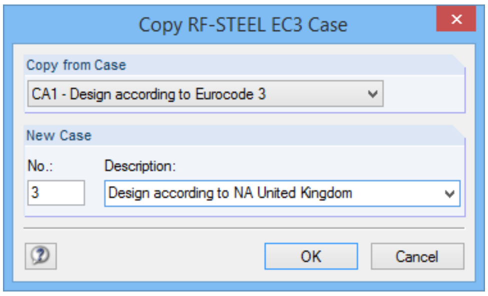


Figure 7.3 Dialog box Copy *RF-STEEL EC3 Case*

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

Delete a design case

To delete a design case, select on the RF-/STEEL EC3 menu

File → Delete Case.

The following dialog box appears.

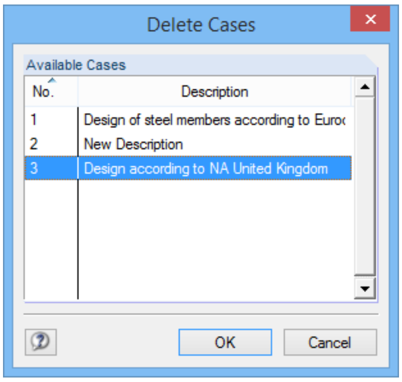
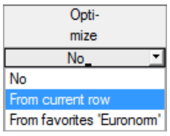


Figure 7.4 Dialog box *Delete Cases*

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 design module offers you the possibility to optimize overloaded or little utilized cross-sections: Define the relevant sections in window *1.3 Cross-Sections* by opening the drop-down list in column E or F where you decide if the cross-sections are to be determined *From current row* or from user-defined *Favorites* (see Figure 2.18). You can also start the optimization in the result windows by using the shortcut menu.

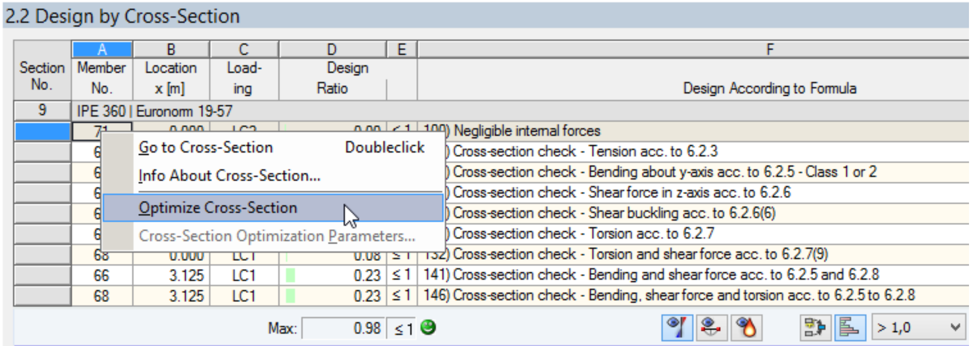
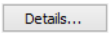


Figure 7.5 Shortcut menu for cross-section optimization



During the optimization process, RF-/STEEL EC3 determines the cross-section that fulfills the ultimate limit state (!) design in the most "optimal" way, that is, comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.14). The required cross-section properties are determined with the internal forces as available from RFEM or RSTAB. If another cross-section proves to be more favorable, it is used for the design. Then, the graphic in Window 1.3 shows two cross-sections — the original cross-section from RFEM or RSTAB and the optimized cross-section (see Figure 7.7).

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

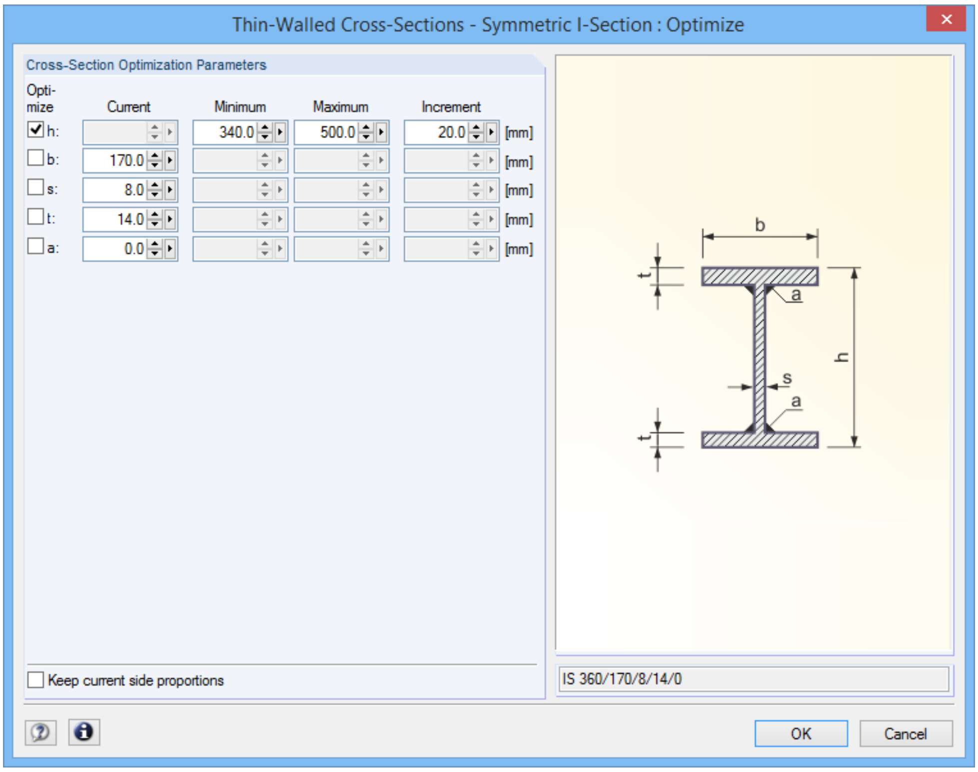


Figure 7.6 Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section : Optimize*

You determine the parameter(s) that you want to modify by selecting the *Optimize* check box(es). This enables the *Minimum* and *Maximum* columns where you can define the upper and lower limits of the parameter. The *Increment* column controls the interval in which the size of the parameter varies during the optimization process.

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

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

Please note that during the optimization the internal forces won't be automatically recalculated with the modified cross-sections: It is up to you to decide which cross-sections should be transferred to RFEM or RSTAB 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 with the modified cross-sections after the first optimization, and then to optimize the cross-sections once again.

You can export the modified cross-sections to RFEM or RSTAB: Go to the *1.3 Cross-Sections* window and select on the menu

Edit → Export All Cross-Sections to RFEM/RSTAB.

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

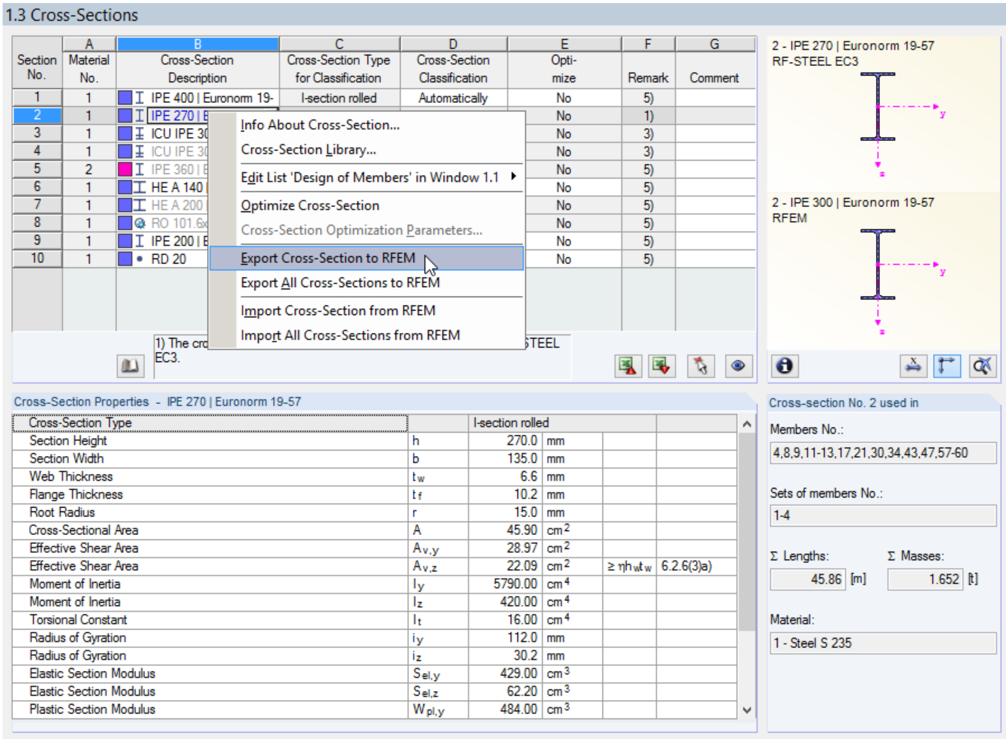


Figure 7.7 Shortcut menu in Window *1.3 Cross-Sections*

Before the modified cross-sections are transferred, a query appears asking if the results of RFEM or RSTAB should be deleted.

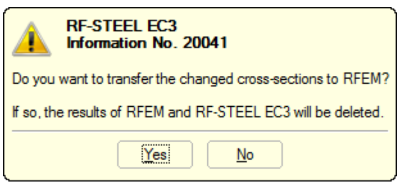


Figure 7.8 Query before transfer of modified cross-sections to RFEM

After starting the [Calculation] in RF-/STEEL EC3, the internal forces and design ratios are determined in one calculation run.

If the modified cross-sections have not yet been exported to RFEM or RSTAB, you can reimport the original cross-sections to the design module by using the options shown in Figure 7.7. Please note that this possibility 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

The units and decimal places are managed for RFEM/RSTAB and the add-on modules in one dialog box. In RF-/STEEL EC3, you can access this dialog box for adjusting the units by selecting on the menu

Settings → Units and Decimal Places.

The dialog box known from RFEM or RSTAB appears. The RF-/STEEL EC3 add-on module is preset in the *Program / Module* list.

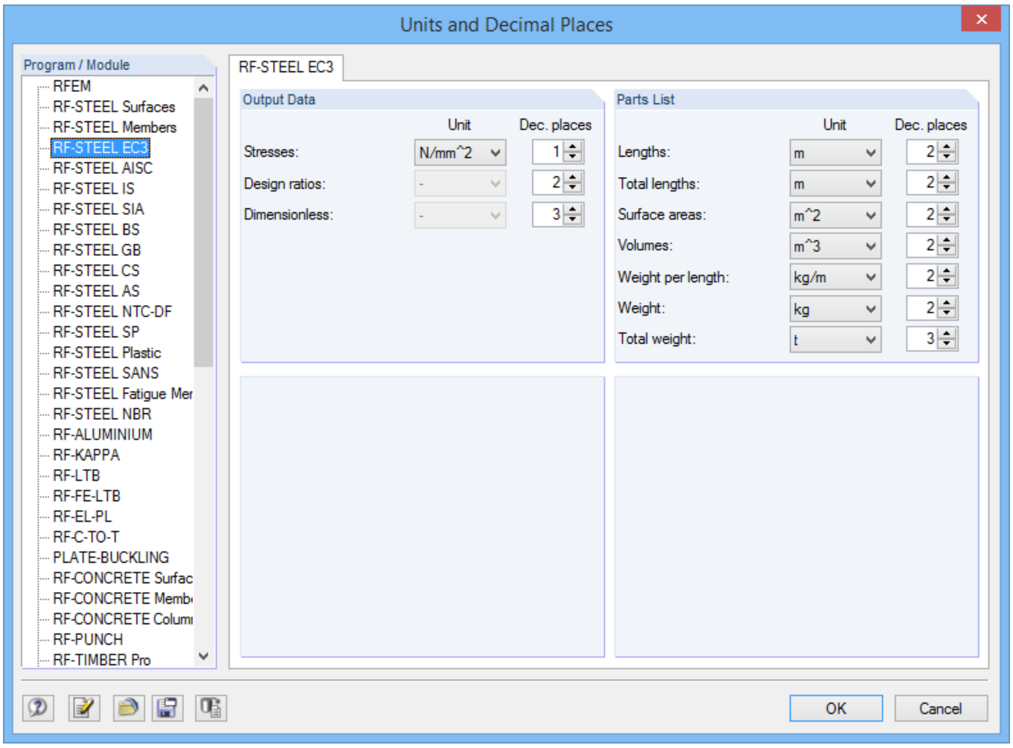


Figure 7.9 Dialog box *Units and Decimal Places*

The modified settings can be saved as user profile and reused in other models. The functions are described in Chapter 11.1.3 of the RFEM or RSTAB manual.

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

If the materials have been adjusted in RF-/STEEL EC3 for the design, you can export the modified materials to RFEM or RSTAB in a similar way as you export cross-sections: Open the *1.2 Materials* window, and then select on the menu

Edit → Export All Materials to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.2 to export materials to RFEM/RSTAB.

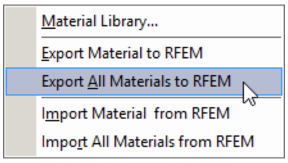
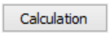


Figure 7.10 Shortcut menu of Window *1.2 Materials*



Before the modified cross-sections are transferred, a query appears asking if the results of RFEM or RSTAB should be deleted. After starting the [Calculation] in RF-/STEEL EC3, the internal forces and design ratios are determined in one calculation run.

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

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

If the effective lengths have been adjusted in RF-/STEEL EC3 for the designs, you can also export the modified effective lengths to RFEM or RSTAB: Go to the *1.5 Effective Lengths - Members* window and select on the menu

Edit → Export All Effective Lengths to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.5 to export effective lengths to RFEM/RSTAB.

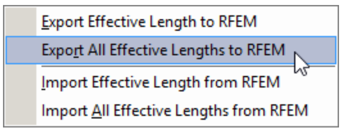


Figure 7.11 Shortcut menu of Window *1.5 Effective Lengths - Members*

Before the modified effective lengths are transferred, a query appears asking if the results of RFEM/RSTAB should be deleted.

If the modified effective lengths have not yet been exported to RFEM or RSTAB, you can reimport the original effective lengths to the design module by using the options shown in Figure 7.11.

7.4.3 Export of Results

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

Clipboard

To copy cells selected in the results windows to the clipboard, use the keys [Ctrl]+[C]. To insert them, for example, in a word processing program, press [Ctrl]+[V]. The headers of the table columns won't be transferred.

Printout report

The data of RF-/STEEL EC3 can be printed into the printout report (see Chapter 6.1) where they can be exported. Then, in the printout report, select on the menu

File → Export to RTF.

This function is described in Chapter 10.1.11 of the RFEM or RSTAB manual.

Excel

RF-/STEEL EC3 provides a function for directly exporting data to MS Excel or the CSV file format. To access this function, select on the menu

File → Export Tables.

The following export dialog box opens.

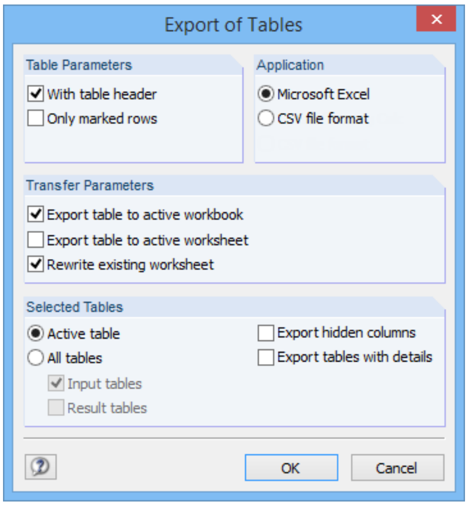
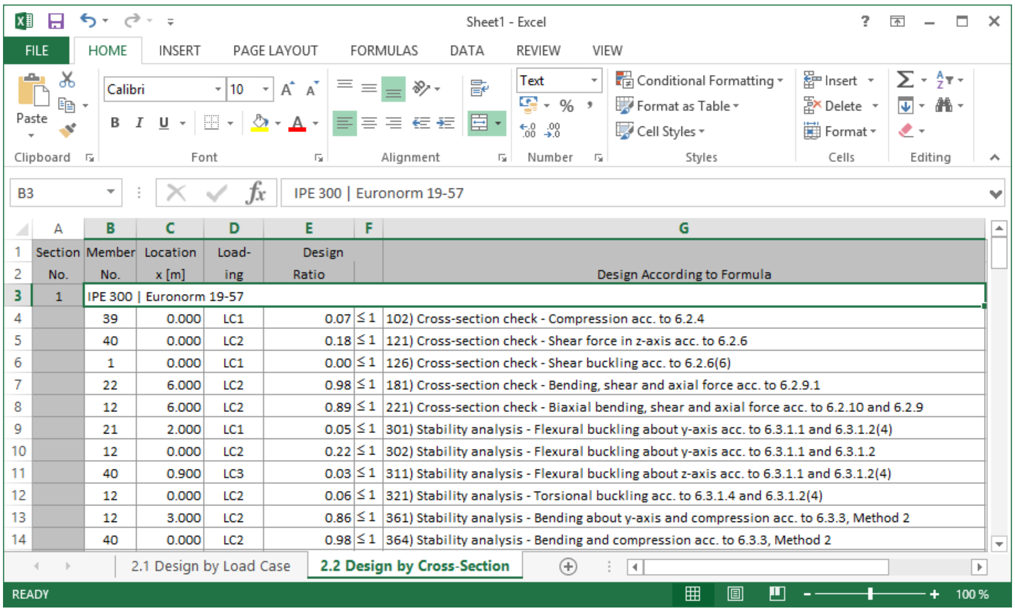


Figure 7.12 Dialog box *Export of Tables*

When you have selected the relevant data, you can start the export with [OK]. Excel will be started automatically, that is, you do not need to open the program first.



Section No.	Member No.	Location x [m]	Loading	Design Ratio		Design According to Formula
1	IPE 300 | Euronorm 19-57					
	39	0.000	LC1	0.07	≤ 1	102) Cross-section check - Compression acc. to 6.2.4
	40	0.000	LC2	0.18	≤ 1	121) Cross-section check - Shear force 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)
	22	6.000	LC2	0.98	≤ 1	181) Cross-section check - Bending, shear and axial force acc. to 6.2.9.1
	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
	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)
	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
	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)
	12	0.000	LC2	0.06	≤ 1	321) Stability analysis - Torsional buckling acc. to 6.3.1.4 and 6.3.1.2(4)
	12	3.000	LC2	0.86	≤ 1	361) Stability analysis - Bending about y-axis and compression acc. to 6.3.3, Method 2
	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

This chapter presents two examples describing the design with RF-/STEEL EC3. Other examples are described in the following articles on our website:

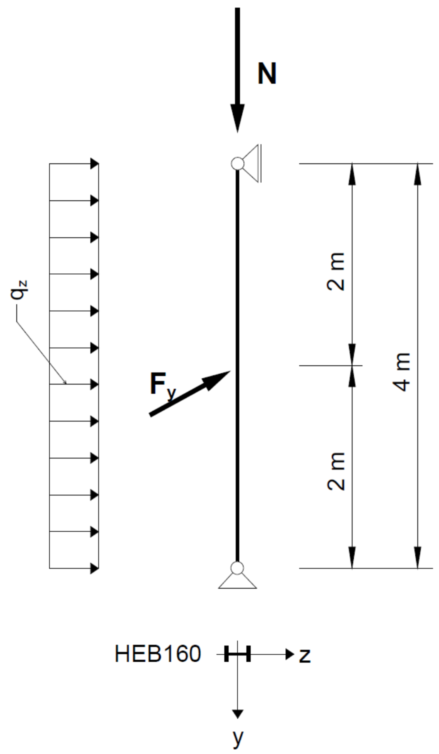
- https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001377
- https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001447
- https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001600
- https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001622

8.1 Stability

We will perform some 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 kN
$q_{z,d}$ = 5 kN/m
$F_{y,d}$ = 7.5 kN

Figure 8.1 System and Loading

Internal forces according to linear static analysis

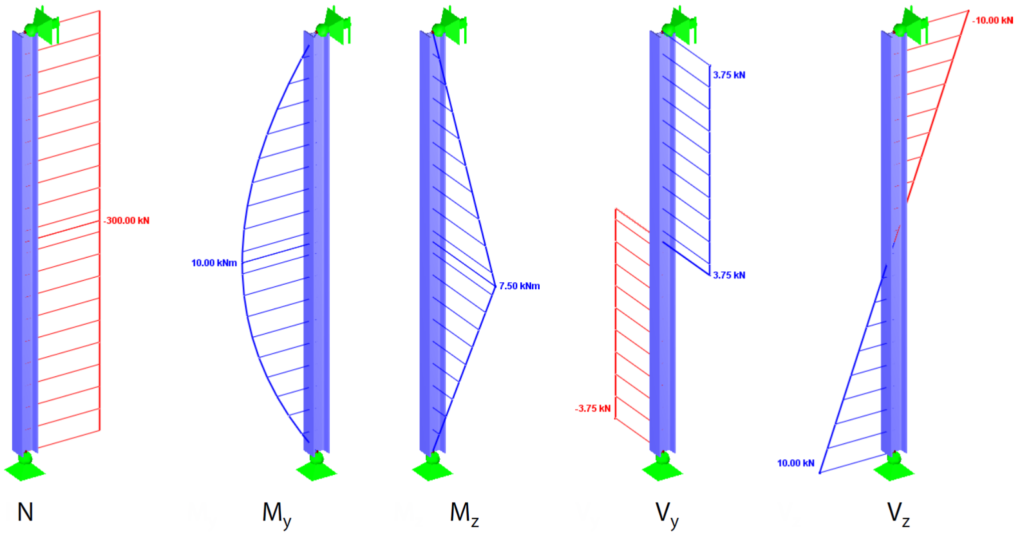


Figure 8.2 Internal forces

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 or RSTAB determines the following internal forces:

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

Table 8.1 Internal forces

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	2490.00	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	I_ω	47940.00	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 8.2 Cross-section properties HE-B 160, S 235

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

$$N_{cr,z} = \frac{21000 \cdot 889.00 \cdot \pi^2}{400.00^2} = 1151.60 \text{ kN}$$

$$\bar{\lambda}_z = \sqrt{\frac{A \cdot f_y}{N_{cr,z}}} = \sqrt{\frac{54.30 \cdot 23.5}{1151.60}} = 1.053 > 0.2$$

⟶ Design for flexural buckling must be performed.

cross-sectional geometry: h/b = 1.00 ≤ 1.2; structural steel S 235; t ≤ 100 mm

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

⇒ α_z = 0.49 ([1] Table 6.1)

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

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

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

Results of RF-/STEEL EC3 calculation

Second moment of area	I_z	889.00	cm^4		
Effective member length	$L_{cr,z}$	4.000	m		
Elastic flexural buckling force	$N_{cr,z}$	1151.60	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 8.3 Results of RF-/STEEL EC3 calculation

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

$$N_{cr,y} = \frac{21000 \cdot 2490.00 \cdot \pi^2}{400.00^2} = 3225.51 \text{ kN}$$

$$\bar{\lambda}_y = \sqrt{\frac{A \cdot f_y}{N_{cr,y}}} = \sqrt{\frac{54.30 \cdot 23.5}{3225.51}} = 0.629 > 0.2$$

⟶ Design for flexural buckling must be performed.

cross-sectional geometry: h/b = 1.00 ≤ 1.2; structural steel S 235; t ≤ 100 mm

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

⇒ α_y = 0.34 ([1] Table 6.1)

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

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

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

Results of RF-/STEEL EC3 calculation

Second moment of area	I_y	2490.00	cm^4		
Effective member length	$L_{cr,y}$	4.000	m		
Elastic flexural buckling force	$N_{cr,y}$	3225.51	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 8.4 Results 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 assumed to be in the shear center (you can adjust the application point for transverse loads in the *Details* dialog box, cf. Chapter 3.1.2).

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

$$M_{cr} = 1.13 \cdot \frac{\pi^2 \cdot 21000 \cdot 889}{400^2} \cdot \sqrt{\frac{47940}{889} + \frac{400^2 \cdot 8100 \cdot 31.40}{\pi^2 \cdot 21000 \cdot 889}} = 215.71\ \text{kNm}$$

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 $M_{cr,x}$ values. Those are the elastic critical moments at the x-locations relative to the elastic critical moment at the location of the maximum moment. Using $M_{cr,x}$, the program then 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: $W_y = W_{pl,y} = 354.00\ cm^3$

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

Reduction factor χ_{LT}

Calculation according to [1] clause 6.3.2.3

HEB-160: h/b = 1.0 < 2.0 ⇒ buckling curve *b* according to [1] Table 6.5

Auxiliary factor:

$$\Phi_{LT} = 0.5 \cdot \left[1 + \alpha_{LT} \cdot \left(\bar{\lambda}_{LT} - \bar{\lambda}_{LT,0}\right) + \beta \cdot \bar{\lambda}_{LT}^2\right]$$

$$\Phi_{LT} = 0.5 \cdot [1 + 0.34 \cdot (0.621 - 0.40) + 0.75 \cdot 0.621^2] = 0.682$$

Limiting slenderness:

$$\bar{\lambda}_{LT,0} = 0.40$$

Parameter (minimum value):

$$\beta = 0.75$$

Imperfection factor according to [1] Table 6.3:

$$\alpha_{LT} = 0.34$$

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

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

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

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

Correction factor k_c according to [1] Table 6.6 for a parabolic moment diagram:

$$k_c = 0.94$$

$$f = 1 - 0.5 \cdot (1 - 0.94) \cdot [1 - 2.0 \cdot (0.621 - 0.8)^2] = 0.972$$

Interaction factors k_{yy} and k_{yz}

Determination according to [6] Annex B, Table B.2, for structural components susceptible to torsional deformations

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

$$C_{my} = C_{mLT} = 0.95 + 0.05 \cdot \alpha_h = 0.95 \text{ where } \alpha_h = \frac{M_h}{M_s} = \frac{0}{10} = 0$$

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

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

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

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} is obtained according to Table B.3 for ψ = 0 as:

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

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

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

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

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

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

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

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

According to [1] Eq. (6.61) the following requirement must be fulfilled:

$$\frac{N_{Ed}}{\chi_y \cdot N_{Rk}/\gamma_{M1}} + k_{yy} \cdot \frac{M_{y,Ed}}{\chi_{LT} \cdot M_{y,Rk}/\gamma_{M1}} + k_{yz} \cdot \frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M1}} \leq 1$$

where

$$M_{y,Rk} = W_{pl,y} \cdot f_y = 354 \cdot 23.5 = 8319 \text{ kNcm} = 83.19 \text{ kNm}$$

$$M_{z,Rk} = W_{pl,z} \cdot f_y = 169.96 \cdot 23.5 = 3994.1 \text{ kNcm} = 39.94 \text{ kNm}$$

$$\frac{300}{0.822 \cdot 1276.05/1.0} + 1.067 \cdot \frac{10.0}{0.908 \cdot 83.19/1.0} + 0.888 \cdot \frac{7.50}{39.94/1.0} = 0.594 \leq 1$$

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

According to EN1993-1-1 Eq. (6.62) the following requirement must be fulfilled:

$$\frac{N_{Ed}}{\chi_z \cdot N_{Rk}/\gamma_{M1}} + k_{zy} \cdot \frac{M_{y,Ed}}{\chi_{LT} \cdot M_{y,Rk}/\gamma_{M1}} + k_{zz} \cdot \frac{M_{z,Ed}}{M_{z,Rk}/\gamma_{M1}} \leq 1$$

$$\frac{300}{0.510 \cdot 1276.05/1.0} + 0.934 \cdot \frac{10.0}{0.908 \cdot 83.19/1.0} + 1.481 \cdot \frac{7.50}{39.94/1.0} = 0.863 \leq 1$$

Results of RF-/STEEL EC3 calculation

Section depth	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	8100.00	kN/cm^3		
Length factor	k_z	1.000			
Length factor	k_w	1.000			
Length	L	4.000	m		
Warping constant	I_w	47940.00	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	Com-ponent	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		
Governing cross-sectional area	A_i	54.30	cm^2		Tab. 6.7

Compression resistance	N_{Rk}	1276.05	kN		Tab. 6.7
Partial safety 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 8.5 Results of RF-/STEEL EC3 calculation

8.2 Fire Resistance

This example describes the fire resistance design for a steel column, using the National Annex of Germany.

System and loads

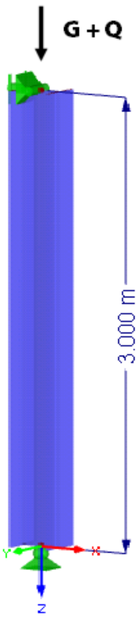


Figure 8.3

Column cross-section: HE-B 300, steel S 235
System: hinged column, β = 1.0
System height: 3 m
Loading: G_K = 1200 kN, Q_K = 600 kN

Ultimate limit state design for room temperature

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

$$N_{cr,z} = \frac{21000 \cdot 8560.00 \cdot \pi^2}{300.00^2} = 19712.90 \text{ kN}$$

$$\bar{\lambda}_z = \sqrt{\frac{A \cdot f_y}{N_{cr,z}}} = \sqrt{\frac{149.0 \cdot 23.5}{19712.90}} = 0.422 > 0.2$$

⟶ Design for flexural buckling must be performed.

cross-sectional geometry: h/b = 1.00 ≤ 1.2; structural steel S 235; t ≤ 100 mm

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

⇒ α_z = 0.49 ([1] Table 6.1)

$$\Phi = 0.5 \cdot [1 + 0.49 \cdot (0.422 - 0.2) + 0.422^2] = 0.643$$

$$\chi_z = \frac{1}{0.643 + \sqrt{0.643^2 - 0.422^2}} = 0.886$$

$$N_{Ed} = 1.35 \cdot G_K + 1.5 \cdot Q_K = 1.35 \cdot 1200 + 1.5 \cdot 600 = 2520 \text{ kN}$$

Design ratio

$$\frac{N_{Ed}}{\chi_z \cdot A \cdot f_y / \gamma_{M1}} = \frac{2520}{0.886 \cdot 149.0 \cdot 23.5/1.1} = 0.894 \leq 1.0$$

Results of RF-/STEEL EC3 calculation

Second moment of area	I_z	8560.00	cm^4		
Effective member length	$L_{cr,z}$	3.000	m		
Elastic flexural buckling force	$N_{cr,z}$	19712.9	kN		
Slenderness	λ_z	0.4215		> 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	0.643			6.3.1.2(1)
Reduction factor	χ_z	0.886			Eq. (6.49)
Flexural buckling resistance	$N_{b,z,Rd}$	2821.80	kN		Eq. (6.47)
Design ratio	η	0.893		≤ 1.0	Eq. (6.46)

Table 8.6 Results of RF-/STEEL EC3 calculation

Fire resistance design

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

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

Specific weight:	$\rho_p = 945\ kg/m^3$
Thermal conductivity:	$\lambda_p = 0.2\ W/K$
Specific heat capacity:	$c_p = 1700\ J/kgK$
Thickness:	$d_p = 18\ mm$

Determination of reduction factors

$k_{y,\Theta} = 0.704$ — [10] Table 3.1

$k_{E,\Theta} = 0.528$ — [10] Table 3.1

Design in fire situation according to [2] clause 4.2.3.2

Imperfection factor α:

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

Non-dimensional relative slenderness $\bar{\lambda}_\Theta$:

$$\bar{\lambda}_\theta = \bar{\lambda} \cdot \left[\frac{k_{y,\theta}}{k_{E,\theta}}\right]^{0.5} = 0.422 \cdot \left[\frac{0.704}{0.528}\right]^{0.5} = 0.486$$

Auxiliary factor:

$$\Phi_\theta = \frac{1}{2} \cdot [1 + \alpha \cdot \bar{\lambda}_\theta + \bar{\lambda}_\theta^2] = \frac{1}{2} \cdot [1 + 0.65 \cdot 0.486 + 0.486^2] = 0.776$$

Reduction factor for flexural buckling in fire situation:

$$\chi_{fi} = \frac{1}{\varphi_\theta + \sqrt{\varphi_\theta^2 - \bar{\lambda}_\theta^2}} = \frac{1}{0.776 + \sqrt{0.776^2 - 0.486^2}} = 0.724$$

Buckling resistance of structural component subjected to compression:

$$N_{b,fi,Rd} = \frac{\chi_{fi} \cdot A \cdot k_{y,\theta} \cdot f_y}{\gamma_{M,fi}} = \frac{0.724 \cdot 149.0 \cdot 0.704 \cdot 23.5}{1.0} = 1784.7 \text{ kN}$$

Loading in case of fire:

$$N_{fi,Ed} = 1.0 \cdot G_k + 0.9 \cdot Q_k = 1.0 \cdot 1200 + 0.9 \cdot 600 = 1740 \text{ kN}$$

Design ratio

$$\eta = \frac{N_{fi,Ed}}{N_{b,fi,Rd}} = \frac{1740}{1784.7} = 0.975 \leq 1.0$$

Results of RF-/STEEL EC3 calculation

Reduction factor	$k_{y,\theta}$	0.704			[2], Tab. 3.1
Reduction factor	$k_{E,\theta}$	0.528			[2], Tab. 3.1
Slenderness	$\lambda_{z,\theta}$	0.486			[2], Eq. (4.7)
Imperfection factor	α	0.650			[2], 4.2.3.2(2)
Auxiliary factor	$\Phi_{z,\Theta}$	0.776			[2], 4.2.3.2(2)
Reduction factor	$\chi_{z,fi}$	0.724			[2], Eq. (4.6)
Partial safety factor	$\gamma_{M,fi}$	1.000			[2], 2.3 (1)
Flexural buckling resistance	$N_{b,fi,z,\Theta,Rd}$	1784.4	kN		[2], Eq. (4.5)
Design ratio	η	0.975		≤ 1.0	[2], Eq. (4.1)

Table 8.7 Results of RF-/STEEL EC3 calculation

You can find more information about the thermal behavior of steel as a material for structural fire design in our Knowledge Base:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001496

Another article describes the fire resistance design by means of parametric temperature-time curves:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001613

9 Literature

[1] Eurocode 3: Design of steel structures - Part 1–1: General rules and rules for buildings; EN 1993–1–1:2010–12

[2] EN 1993-1-2 (2005): Eurocode 3: Design of steel structures - Part 1-2: General rules - Structural fire design [Authority: The European Union Per Regulation 305/2011, Directive 98/34/EC, Directive 2004/18/EC]

[3] Eurocode 3: Design of steel structures - Part 1–3: General rules - Supplementary rules for cold-formed members and sheeting; EN 1993–1–3:2010–12

[4] EN 1993-1-4 (2006): Eurocode 3: Design of steel structures - Part 1-4: General rules - Supplementary rules for stainless steels [Authority: The European Union Per Regulation 305/2011, Directive 98/34/EC, Directive 2004/18/EC]

[5] Eurocode 3: Design of steel structures - Part 1-5: General rules - Plated structural elements; EN 1993-1-5:2006 + AC:2009

[6] Johannes Naumes, Isabell Strohmann, Dieter Ungermann and Gerhard Sedlacek. Die neuen Stabilitätsnachweise im Stahlbau nach Eurocode 3. Stahlbau, 77, 2008.

[7] Johannes Naumes, Markus Feldmann and Gerhard Sedlacek. Biegeknicken und Biegedrillknicken von Stäben auf einheitlicher Grundlage, Band 70, Shaker Verlag 2010

[8] EN 1993-2 (2006): Eurocode 3: Design of steel structures - Part 2: Steel bridges [Authority: The European Union Per Regulation 305/2011, Directive 98/34/EC, Directive 2004/18/EC]

[9] EN 1991-1-2 (2002): Eurocode 1: Actions on structures - Part 1-2: General actions - Actions on structures exposed to fire [Authority: The European Union Per Regulation 305/2011, Directive 98/34/EC, Directive 2004/18/EC]

[10] Kindmann, R.; Frickel, J.: Elastische und plastische Querschnittstragfähigkeit. Berlin: Ernst & Sohn, 2002

[11] F. Nowzartash and M. Mohareb. Plastic interaction relations for elliptical hollow sections. Thin-Walled Structures, 47, 2009.

[12] Eurocode 3: Design of steel structures - Part 1-8: Design of joints; EN 1993–1–8:2005 + AC:2009

[13] Manual SHAPE-THIN. Tiefenbach: Dlubal Software, January 2012.