Version
December 2014

Add-on Module

RF-STEEL Fatigue Members

Fatigue Analysis of Steel Members

Program Description

Am Zellweg 2
D-93464 Tiefenbach
Germany

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

Contents

1 Introduction

1.1 Add-on Module RF-STEEL Fatigue Members

The fatigue designs according to EN 1993-1-9 [1] are based on the so-called nominal stress concept. This method analyzes the influence of material fatigue on a structure. Many damages in a steel structure are affiliated to recurring effects stressing the particular points of the construction until the fatigue failure occurs. The RF-STEEL Fatigue Members add-on module performs the fatigue designs using the method of damage equivalent factors.

The following useful features facilitate the work with the add-on module:

- Determination of stress ranges for selected load cases, load or result combinations
- Free assignment of detail categories to the stress points of cross-sections
- User-defined specification of the damage equivalent factors

Since RF-STEEL Fatigue Members is integrated in the main program, the general input data and internal forces are already available for the design. It is possible to evaluate designs and stress ranges graphically in the RFEM work window and to associate them in the global printout report.

The analysis determines the maximal design ratios of sets of members. Furthermore, RF-STEEL Fatigue Members allows an automatic cross-section optimization including the export of modified cross-section to RFEM.

Using the design cases, you can analyze different types of stress designs. A parts list with quantity surveying completes the design.

We wish you enjoyment and success with RF-STEEL Fatigue Members.

Your DLUBAL Team

1.2 Using the Manual

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

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

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

1.3 Opening RF-STEEL Fatigue Members Add-on Module

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

Menu

To open the program from the RFEM menu bar, click

Add-on Modules → **Design - Steel** → **RF-STEEL Fatigue Members**.

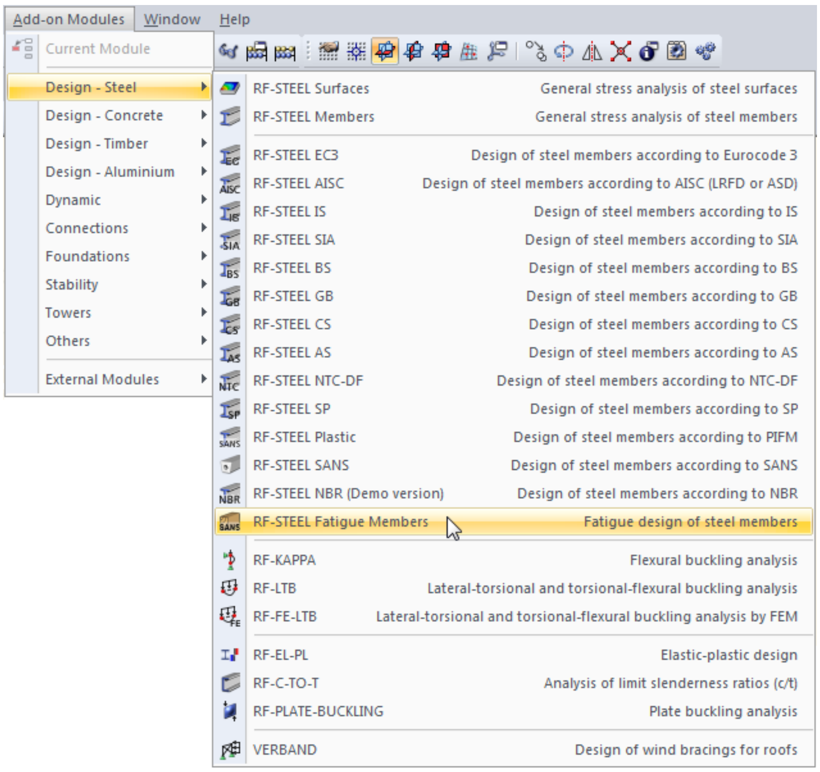


Figure 1.1: Menu: *Add-on Modules* → *Design - Steel* → *RF-STEEL Fatigue Members*

Navigator

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

Add-on Modules → RF-STEEL Fatigue Members.

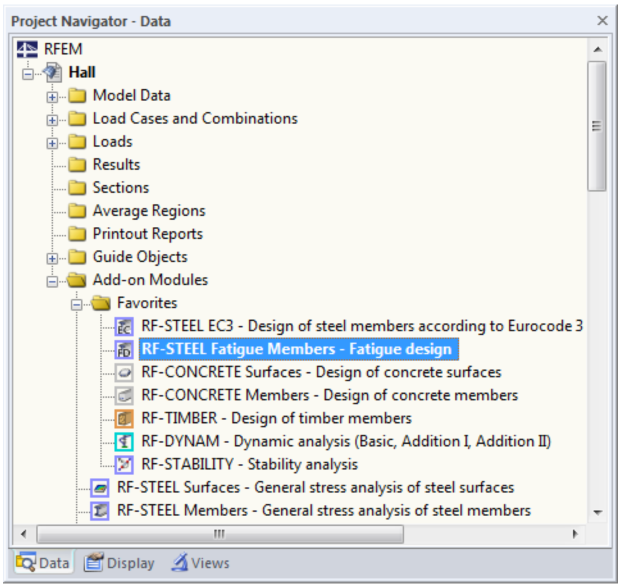
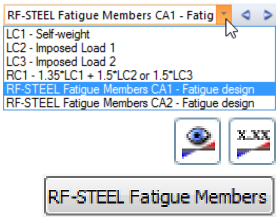


Figure 1.2: Data navigator: *Add-on Modules → RF-STEEL Fatigue Members*

Panel



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

Set the relevant design case in the load case list of the RFEM toolbar. Click [Show Results] button to graphically display the stresses and stress ratios.

When the result display is activated, the panel is available, too. Now you can use the [RF-STEEL Fatigue Members] button in the panel to open the module.

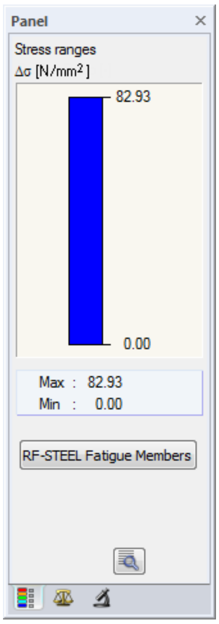


Figure 1.3: Panel button [RF-STEEL Fatigue Members]

2 Theoretical Principles

2.1 Design Concept

The fatigue designs according to EN 1993-1-9 [1] are based on the so-called nominal stress concept. The internal forces are determined according to the structural analysis for members; the stresses are calculated according to the mechanics of materials at the location where the crack formation is expected. In fatigue design, the nominal stress ranges $\Delta\sigma$ and $\Delta\tau$, as a result of the actions, are compared with design values of the fatigue strength $\Delta\sigma_R$ and $\Delta\tau_R$. Furthermore, the concept of the partial safety factors is valid in [1]. The partial safety factor γ_{Mf} for the fatigue strength is graded according to the reliability concept and the possible consequences of failure. The following table shows the recommendations:

Table 3.1: Recommended values for partial factors for fatigue strength

Assesment method	Consequence of failure	
	Low	High
Damage tolerant	1.00	1.15
Safe life	1.15	1.35

Figure 2.1: EN 1993-1-9, Table 3.1

The stresses on the action side are to be determined by a serviceability level. For the partial safety factor γ_{Ff}, the value $\gamma_{Mf} = 1.0$ applies to the actions.

The Standard [1] provides the following design methods:

- Design with fatigue strength
- Design by means of damage accumulation
- Design with damage equivalent factors

In the RF-STEEL Fatigue Members add-on module, the design is carried out by means of damage equivalent factors. Therefore, only this design method is described in the manual.

2.2 Design with Damage Equivalent Factors

This method represents the standard method according to [1]. The damage equivalent stress ranges $\Delta\sigma_{E,2}$ and $\Delta\tau_{E,2}$ related to $2 \cdot 10^6$ stress cycles are to be contrasted with the limit values of the fatigue strength $\Delta\sigma_C$ or $\Delta\tau_C$ for $2 \cdot 10^6$ cycles of the corresponding detail category under consideration of partial safety factors.

2.2.1 Design Equations

Limitation of stress ranges

According to [1], Eq. 8.1, the stress ranges for normal and shear stresses are limited as follows:

$$\Delta\sigma \leq 1.5 \cdot f_y \tag{2.1}$$

$$\Delta\tau \leq \frac{1.5 \cdot f_y}{\sqrt{3}} \tag{2.2}$$

Fatigue designs

Fatigue designs for normal and shear stresses are limited according to [1], Eq. 8.2 as follows:

$$\frac{\gamma_{Ff} \cdot \Delta\sigma_{E,2}}{\Delta\sigma_C / \gamma_{Mf}} \leq 1.0 \tag{2.3}$$

$$\frac{\gamma_{Ff} \cdot \Delta\tau_{E,2}}{\Delta\tau_C / \gamma_{Mf}} \leq 1.0 \tag{2.4}$$

Simultaneous effect of normal and shear stresses

According to [1], Eq. 8.3, also the following design criteria are to be satisfied:

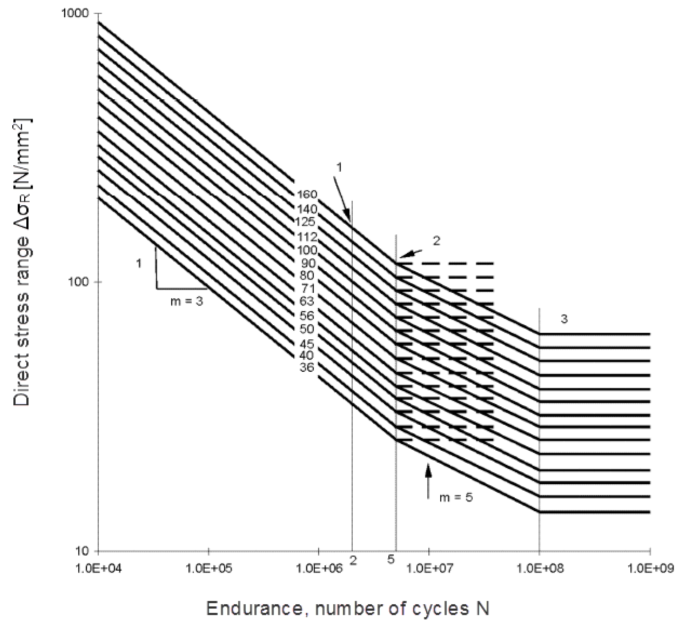
$$\left(\frac{\gamma_{Ff} \cdot \Delta\sigma_{E,2}}{\Delta\sigma_C / \gamma_{Mf}}\right)^3 + \left(\frac{\gamma_{Ff} \cdot \Delta\tau_{E,2}}{\Delta\tau_C / \gamma_{Mf}}\right)^3 \leq 1.0 \tag{2.5}$$

Nominal stresses: $\gamma_{Ff} \cdot \Delta\sigma_{E,2} = \lambda \cdot \Delta\sigma$

$\gamma_{Ff} \cdot \Delta\tau_{E,2} = \lambda \cdot \Delta\tau$

Determination of partial safety factors γ_{Mf} and γ_{Ff} is described in Chapter 2.1.

The fatigue strength $\Delta\sigma_C$ or $\Delta\tau_C$ for $2 \cdot 10^6$ cycles are assigned to the respective detail categories in [1]. You can deduce the normal or shear stress ranges from the following diagrams.



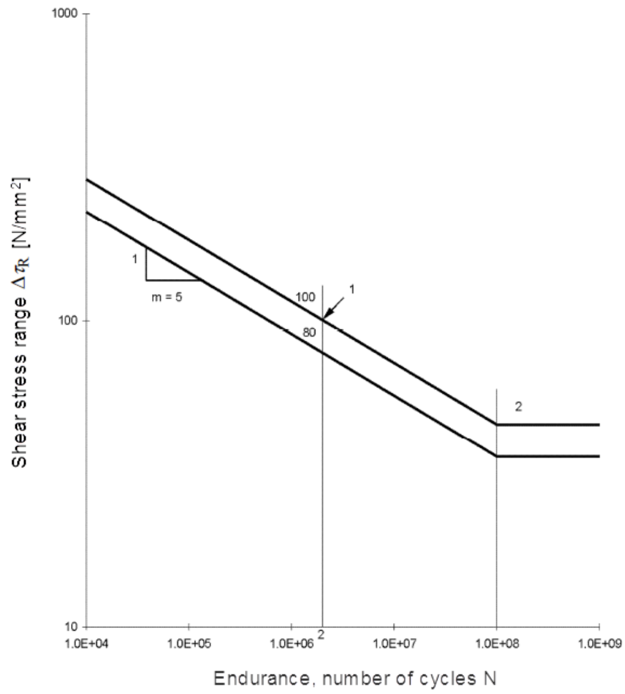
1 Detail category $\Delta\sigma_C$

2 Constant amplitude fatigue limit $\Delta\sigma_D$

3 Cut-off limit $\Delta\sigma_L$

Figure 7.1: Fatigue strength curves for direct stress ranges

Figure 2.2: EN 1993-1-9, Figure 7.1



1 Detail category $\Delta\tau_C$

2 Cut-off limit $\Delta\tau_L$

Figure 7.2: Fatigue strength curves for shear stress ranges

Figure 2.3: EN 1993-1-9, Figure 7.2

2.2.2 Stress Range $\Delta\sigma$

The stress range describes the difference between the maximum and the minimum normal or shear stress (also called top and bottom stress). These are defined as follows:

$$\Delta\sigma = \sigma_{max} - \sigma_{min} \tag{2.6}$$

σ_{max} : Maximum stress (with sign)

σ_{min} : Minimum stress (with sign)

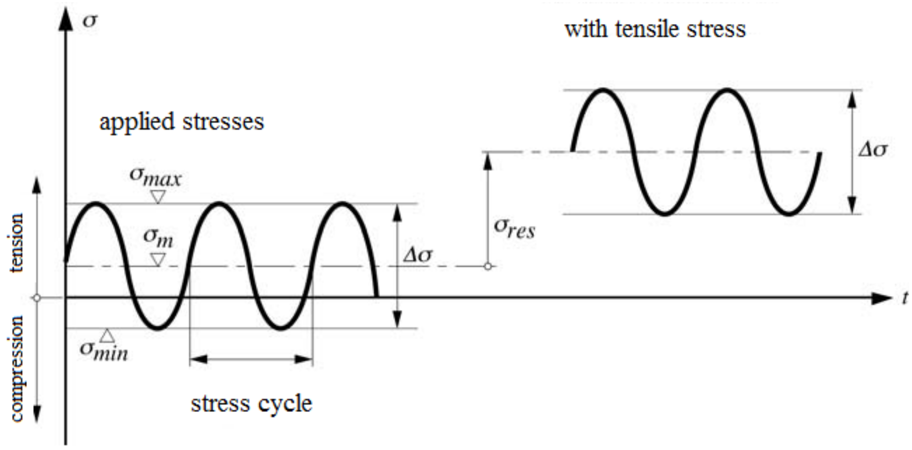


Figure 2.4: Stress ranges and influence of internal tensile stresses according to [2]

2.2.3 Damage Equivalent Factor λ

The damage equivalent factors depend on the designed structural elements. They are described in the respective standards. The following table shows the standards where you can find the rules for calculation of the damage equivalent factors.

EN 1993-2	Steel bridges
EN 1993-3-1	Towers, masts and chimneys
EN 1993-3-2	Towers, masts and chimneys
EN 1993-4-1	Tank structures
EN 1993-4-2	Tank structures
EN 1993-4-3	Pipelines
EN 1993-6	Craneways

Table 2.1: Standards including damage equivalent factors

If there is no information on the damage equivalent factors λ, the design requirements according to [1], Annex A are applied. Then the **nominal stress** results from the damage equivalent factors and the stress range.

$$\gamma_{Ff} \cdot \Delta\sigma_{E,2} = \lambda \cdot \Delta\sigma \tag{2.7}$$

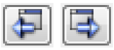
$$\gamma_{Ff} \cdot \Delta\tau_{E,2} = \lambda \cdot \Delta\tau \tag{2.8}$$

3 Input Data

After you open the add-on module, a new window appears. In this window, a navigator is displayed on the left, managing the tables that can be selected currently. The drop-down list above the navigator contains the design cases (see Chapter 8.1, page 37).

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

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



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

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

3.1 General Data

In the 1.1 *General Data* window, you can select the members, sets of members and actions you want to design, as well as the relevant standard.

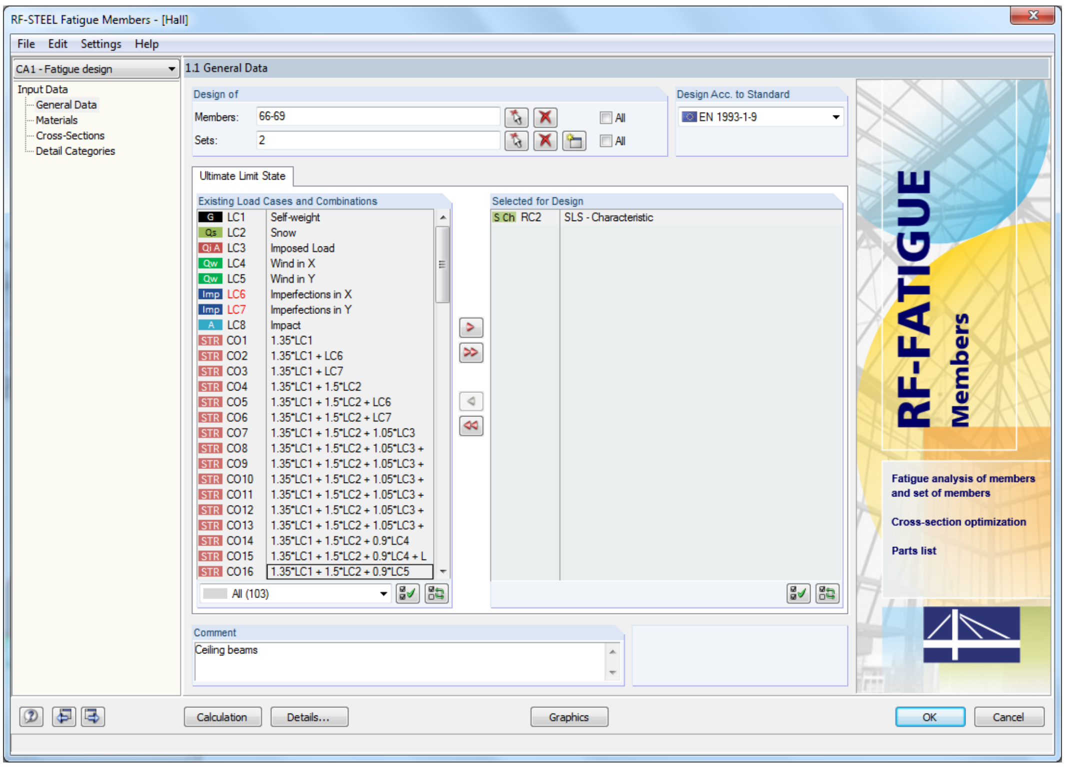


Figure 3.1: Window 1.1 *General Data*

Design of

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

When you design a set of members, the program determines the maximum stress ratio of all members contained in the set of members. The results are shown in result windows 2.2 *Design by Set of Members* and 4.2 *Parts List by Set of Members*.

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

Design Acc. to Standard

Currently, the design method is according to EN 1993-1-9 [1].

Existing Load Cases and Combinations

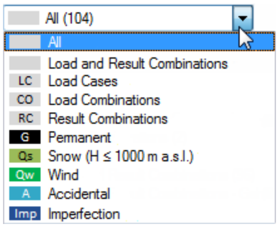
This column lists all load cases as well as load and result combinations that have been created in RFEM.

RF-STEEL Fatigue Members can also design combinations of the RF-DYNAM module.

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

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

Load cases marked in red, like LC 6 or LC 7 in Figure 2.1, cannot be designed: This happens when the load cases are defined without any load data or the load cases contain only imperfections. Then, when you transfer the load cases, a corresponding warning appears.



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

	Select all cases in the list
	Invert selection of load cases

Table 3.1: Buttons in tab *Existing Load Cases*

Selected for Design

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

It is necessary to select several load combinations for the design in order to analyze the stress differences (stress ranges)!

Comment

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

3.2 Materials

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

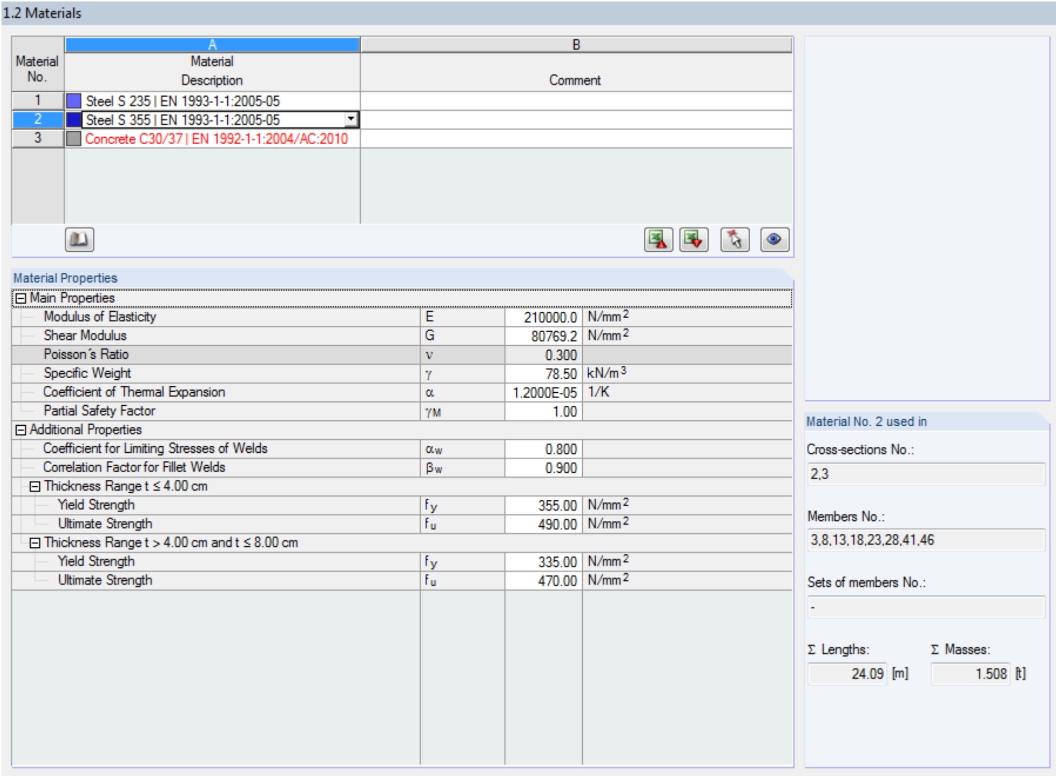


Figure 3.2: Window 1.2 *Materials*

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

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

You can adjust the units and decimal places of material properties and stresses using the menu **Settings** → **Units and Decimal Places** (see Chapter 8.3, page 41).

Material Description

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

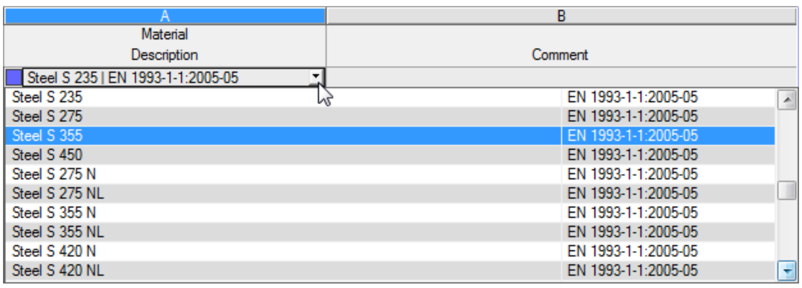


Figure 3.3: List of materials

Only materials of the "Steel" category are available in the list.

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

If you change the material description manually and the entry is stored in the material library, RF-STEEL Fatigue Members will import the material properties, too.

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

Material Library

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

Edit → Material Library

or use the button shown on the left.

The following *Material Library* dialog box appears.

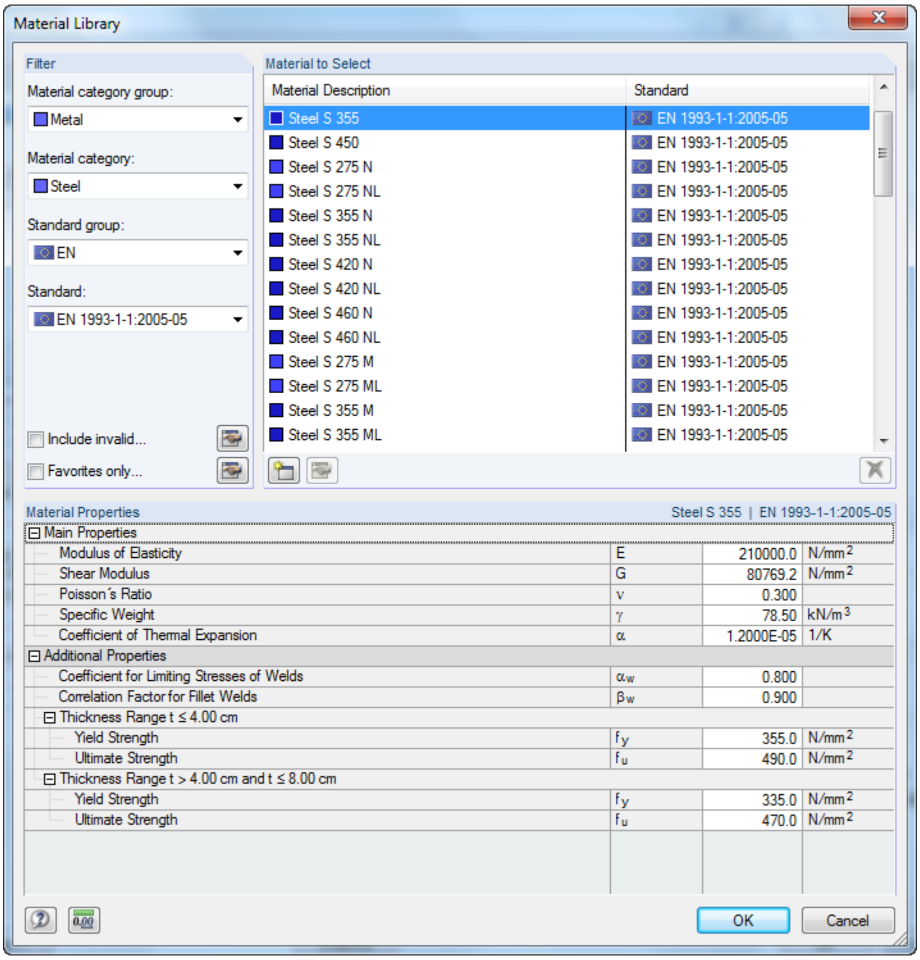


Figure 3.4: Dialog box *Material Library*

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

Click [OK] or use the [↵] button to transfer the selected material to Window 1.2 of RF-STEEL Fatigue Members.

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

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

3.3 Cross-Sections

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

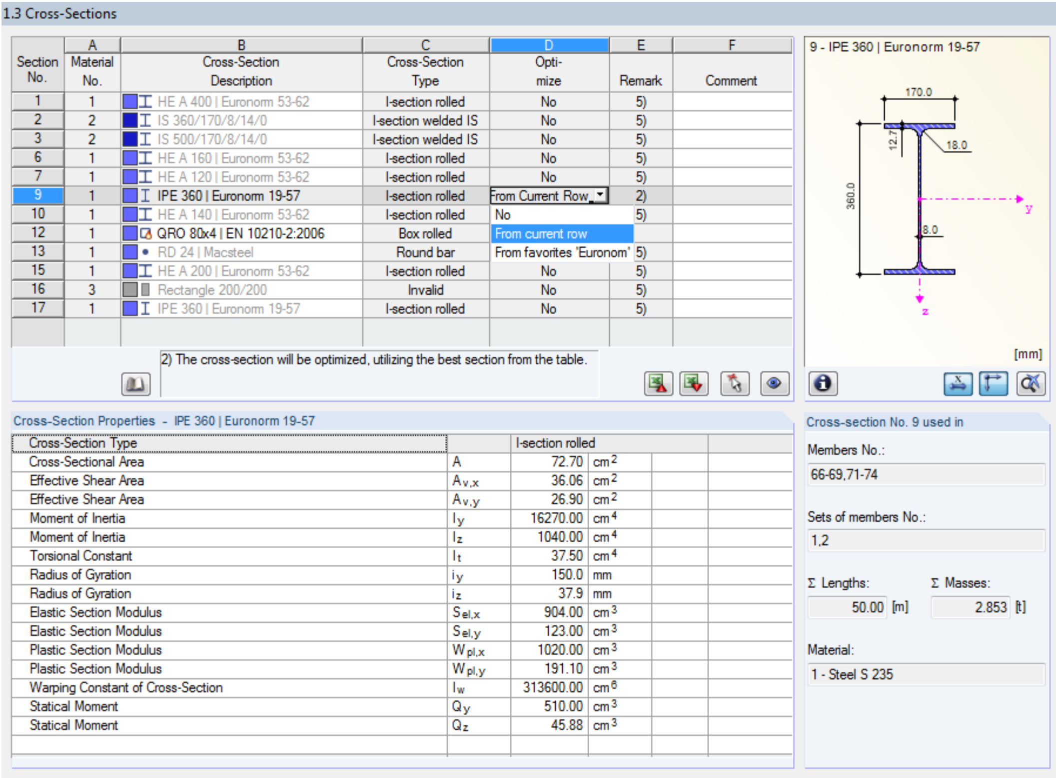


Figure 3.5: Window 1.3 *Cross-Sections*

Cross-Section Description

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

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

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

Chapter 4.13 of the RFEM manual describes how cross-sections can be selected from the library.

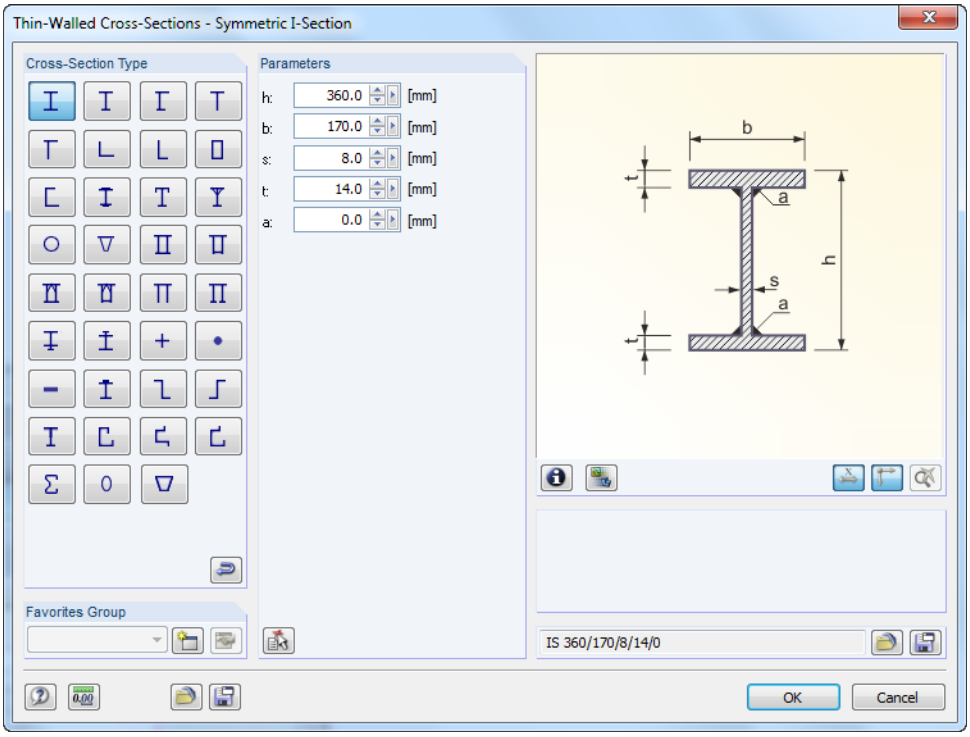
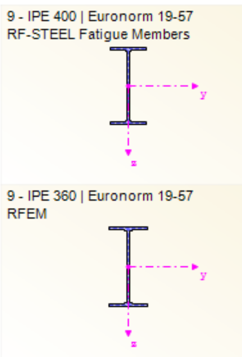


Figure 3.6: IS cross-section types in the cross-section library



You can directly enter a new cross-section description in the input window. If the entry is listed in the database, RF-STEEL Fatigue Members imports these cross-sections parameters, too. A modified cross-section will be highlighted in blue.

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

Max. Design Ratio

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

Optimize

You can optimize every cross-section from the library: For the RFEM internal forces, the program searches the cross-section that comes as close as possible to a user-defined maximum ratio. You can define the maximum ratio in the *Details* dialog box (see Figure 4.2, page 20).

If you want to optimize a cross-section, open the corresponding drop-down list in column D or E and select the desired entry: *From current row* or, if available, *From favorites* 'Description'. Recommendations for the cross-section optimization can be found in Chapter 8.2 on page 38.

Remark

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

A warning might appear before the calculation: *Incorrect type of cross-section!* It means that there is a cross-section that is not stored in the database. This may be a user-defined cross-section or a SHAPE-THIN cross-section that has not been calculated yet. To select an appropriate cross-section for the design, click the [Library] button (see description below Figure 3.5).

Member with Tapered Cross-Section

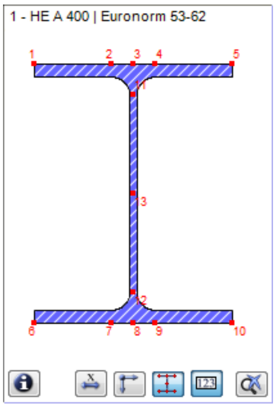
For tapered members with different cross-sections at the member start and member end, the module displays both cross-section numbers in two rows, in accordance with the definition in RFEM.

RF-STEEL Fatigue Members also designs tapered members, provided that the cross-section at the member start has the same number of stress points as the cross-section at the member end. For example, the normal stresses are determined from the moments of inertia and the centroidal distances of the stress points. If the cross-section at the start and the end of a tapered member have a different number of stress points, the intermediate values cannot be interpolated. The calculation is possible neither in RFEM nor in RF-STEEL Fatigue Members.

To produce the same number of stress points, you can, for example, define the second cross-section as a copy of the cross-section start with adjusted geometry parameters. The easiest way to do this is to describe both cross-sections as parametric profiles. In such case, the cross-section type *IVU I-Section Plus Lower Flange* is recommended.

Stress points of the cross-section including numbering can also be checked graphically: Select the cross-section in Window 1.3 and click the [Info] button. The dialog box shown in Figure 3.7 appears.

Cross-Section Graphic



The right part of the window shows the currently selected cross-section.

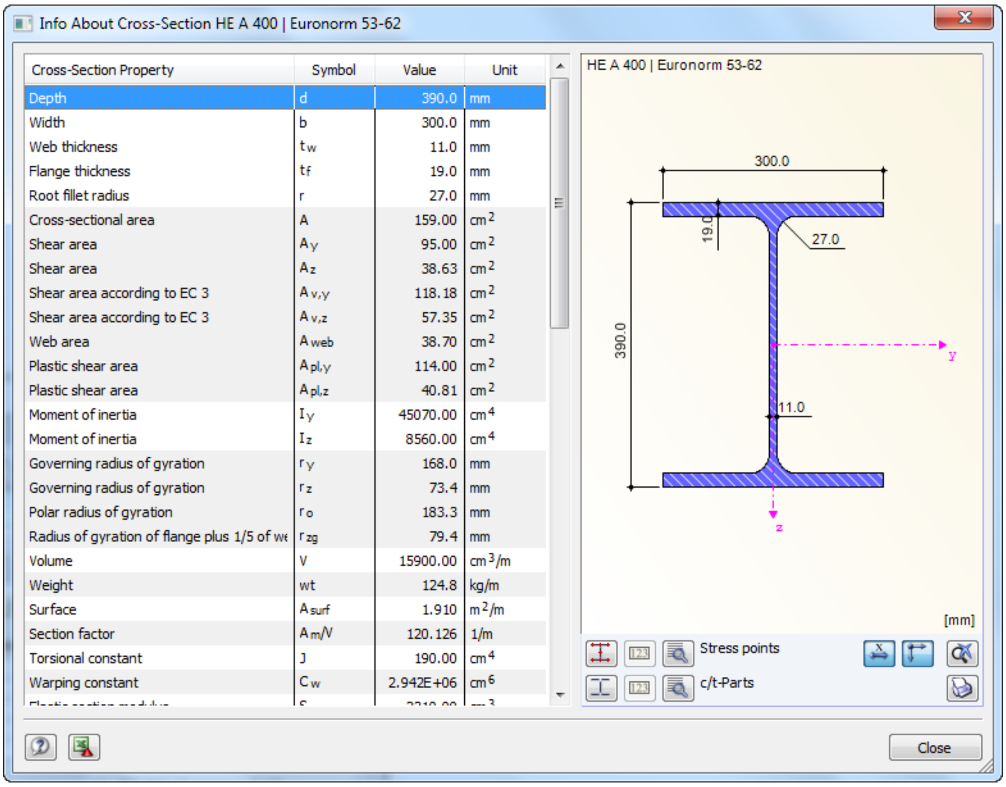
The buttons below the graphic have the following functions:

Button	Function
	Open the *Info About Cross-Section* dialog box (see Figure 3.7)
	Display or hide the cross-section dimensions
	Display or hide the principal axes of the cross-section
	Display or hide the stress points
	Display or hide the numbering of stress points
	Reset full view of the cross-section graphic

Table 3.2: Buttons of cross-section graphic

Info About Cross-Section

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

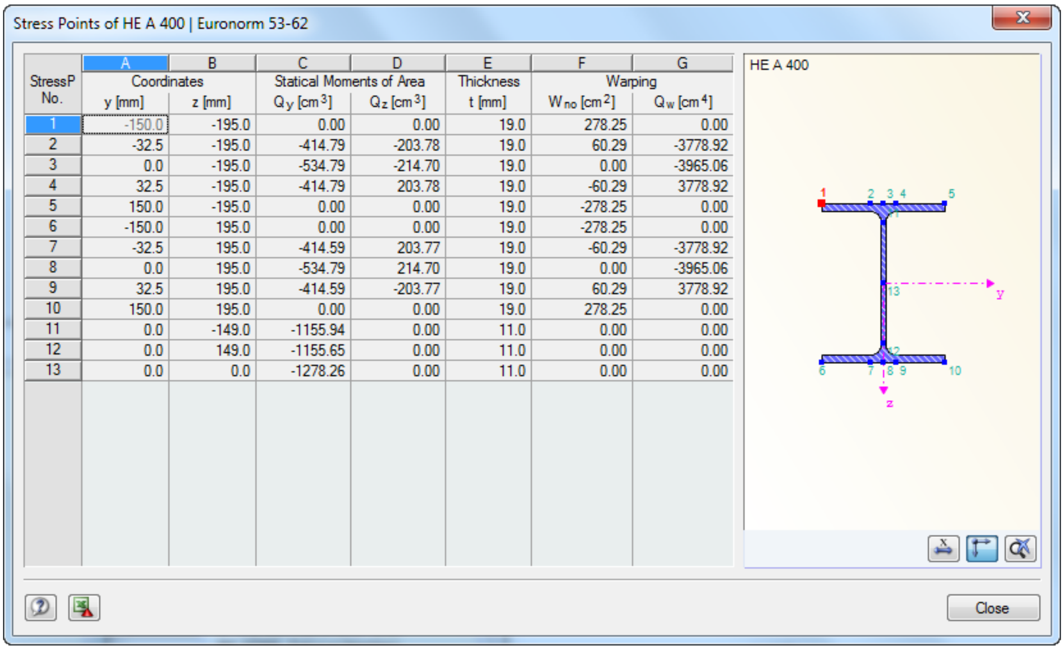


Info About Cross-Section HE A 400 | Euronorm 53-62

Cross-Section Property	Symbol	Value	Unit
Depth	d	390.0	mm
Width	b	300.0	mm
Web thickness	t_w	11.0	mm
Flange thickness	t_f	19.0	mm
Root fillet radius	r	27.0	mm
Cross-sectional area	A	159.00	cm^2
Shear area	A_y	95.00	cm^2
Shear area	A_z	38.63	cm^2
Shear area according to EC 3	$A_{v,y}$	118.18	cm^2
Shear area according to EC 3	$A_{v,z}$	57.35	cm^2
Web area	A_{web}	38.70	cm^2
Plastic shear area	$A_{pl,y}$	114.00	cm^2
Plastic shear area	$A_{pl,z}$	40.81	cm^2
Moment of inertia	I_y	45070.00	cm^4
Moment of inertia	I_z	8560.00	cm^4
Governing radius of gyration	r_y	168.0	mm
Governing radius of gyration	r_z	73.4	mm
Polar radius of gyration	r_o	183.3	mm
Radius of gyration of flange plus 1/5 of we	r_{zg}	79.4	mm
Volume	V	15900.00	cm^3/m
Weight	wt	124.8	kg/m
Surface	A_{surf}	1.910	m^2/m
Section factor	A_m/V	120.126	1/m
Torsional constant	J	190.00	cm^4
Warping constant	C_w	2.942E+06	cm^6

Figure 3.7: Dialog box *Info About Cross-Section*

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



Stress Points of HE A 400 | Euronorm 53-62

StressP No.	A: Coordinates y [mm]	B: Coordinates z [mm]	C: Statical Moments of Area Q_y [cm^3]	D: Statical Moments of Area Q_z [cm^3]	E: Thickness t [mm]	F: Warping W_{no} [cm^2]	G: Warping Q_w [cm^4]
1	-150.0	-195.0	0.00	0.00	19.0	278.25	0.00
2	-32.5	-195.0	-414.79	-203.78	19.0	60.29	-3778.92
3	0.0	-195.0	-534.79	-214.70	19.0	0.00	-3965.06
4	32.5	-195.0	-414.79	203.78	19.0	-60.29	3778.92
5	150.0	-195.0	0.00	0.00	19.0	-278.25	0.00
6	-150.0	195.0	0.00	0.00	19.0	-278.25	0.00
7	-32.5	195.0	-414.59	203.77	19.0	-60.29	-3778.92
8	0.0	195.0	-534.79	214.70	19.0	0.00	-3965.06
9	32.5	195.0	-414.59	-203.77	19.0	60.29	3778.92
10	150.0	195.0	0.00	0.00	19.0	278.25	0.00
11	0.0	-149.0	-1155.94	0.00	11.0	0.00	0.00
12	0.0	149.0	-1155.65	0.00	11.0	0.00	0.00
13	0.0	0.0	-1278.26	0.00	11.0	0.00	0.00

Figure 3.8: Dialog box *Stress Points of HE A 400*

3.4 Detail Categories

In the 1.4 *Detail Categories* window, you can define the fatigue strength of members previously selected for the design in Window 1.1.

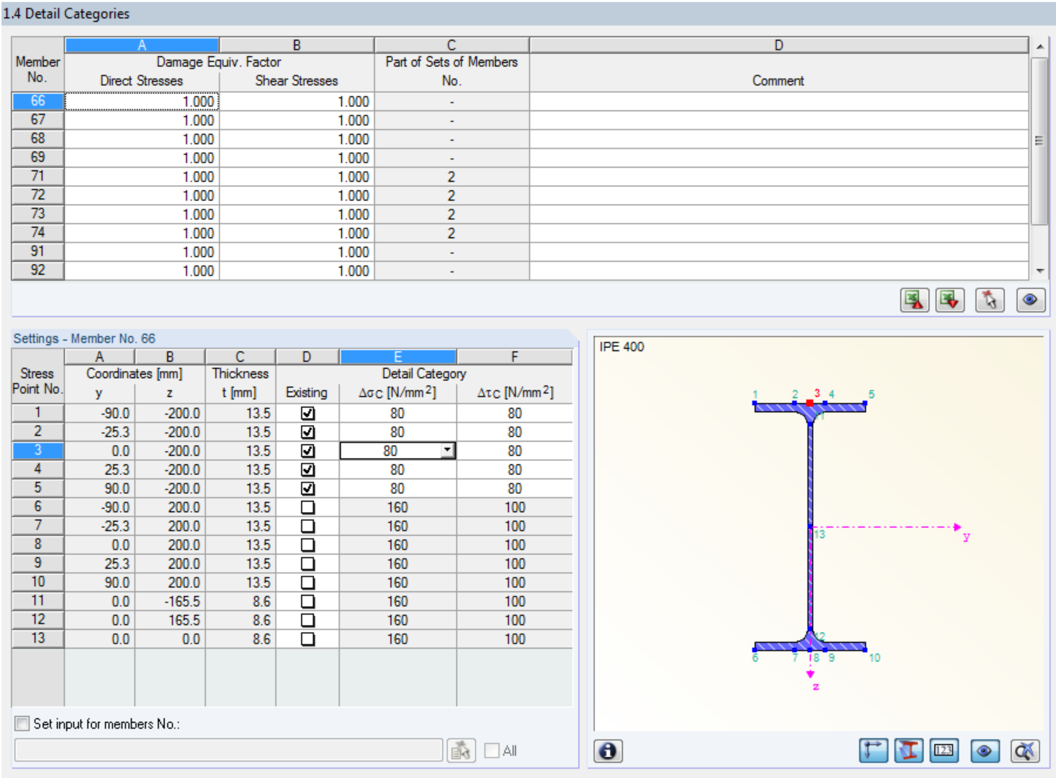


Figure 3.9: Window 1.4 *Detail Categories*

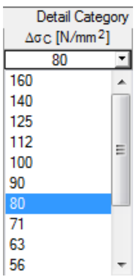
Damage Equivalent Factor

Using the damage equivalent factors λ, which you can define separately for the *Normal stresses* and *Shear stresses*, and the stress range $\Delta\sigma$ or $\Delta\tau$ resulting from the existing load, the design-relevant nominal stress is determined. The determination of the damage equivalent factors is subjected to different standards (see Chapter 2.2.3, page 9).

Part of Set of Members No.

This column shows to which set of members a member belongs.

Settings - Member No.



In the bottom section of the window, you can define detail categories for the relevant stress points of the cross-section. The selected stress point (cursor position in a table row) is highlighted in red in the cross-section graphic. By clicking a stress point in the graphic, you select the relevant row in the table.

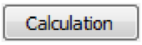
Assigning detail categories facilitates the calculated determination of strength. The number of detail categories represents the reference value of the fatigue strength $\Delta\sigma_C$ or $\Delta\tau_C$ in N/mm. You can obtain the detail categories with strengths from the tables 8.1 to 8.10 of the Eurocode [1].

It is also possible to use the list for a selection of the detail categories (Wöhler curves).

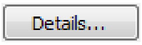
Below the *Settings* table, there is the *Set input for members No.* check box. If selected, the subsequent settings apply to the selected members or *All* members (you can enter the member numbers manually or select them graphically using the [Select] button). This option is useful when assigning several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

4 Calculation

4.1 Detail Settings



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



The *Details* dialog box consists of the following tabs:

- Ultimate Limit State
- General

4.1.1 Ultimate Limit State

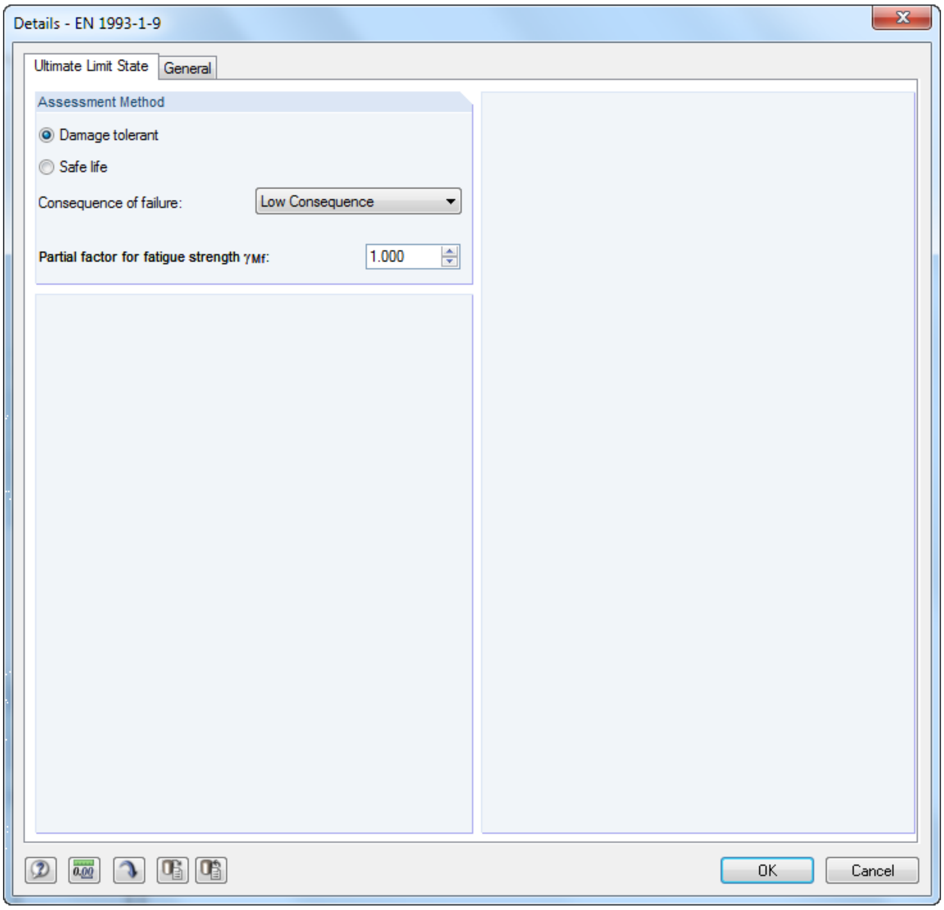
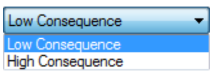


Figure 4.1: Dialog box *Details*, tab *Ultimate Limit State*

Assessment Method



The reliability concept and possible consequence of failure affect the partial safety factor γ_{Mf} considered when determining the fatigue strength. You can set this partial safety factor using the option *Damage tolerant* or *Safe life* together with the *Consequence of failure* (low or high).

You can also specify the partial factor γ_{Mf} for fatigue strength directly.

[1], Table 3.1 shows recommendations for the factor γ_{Mf} (see Figure 2.1, page 6).

4.1.2 General

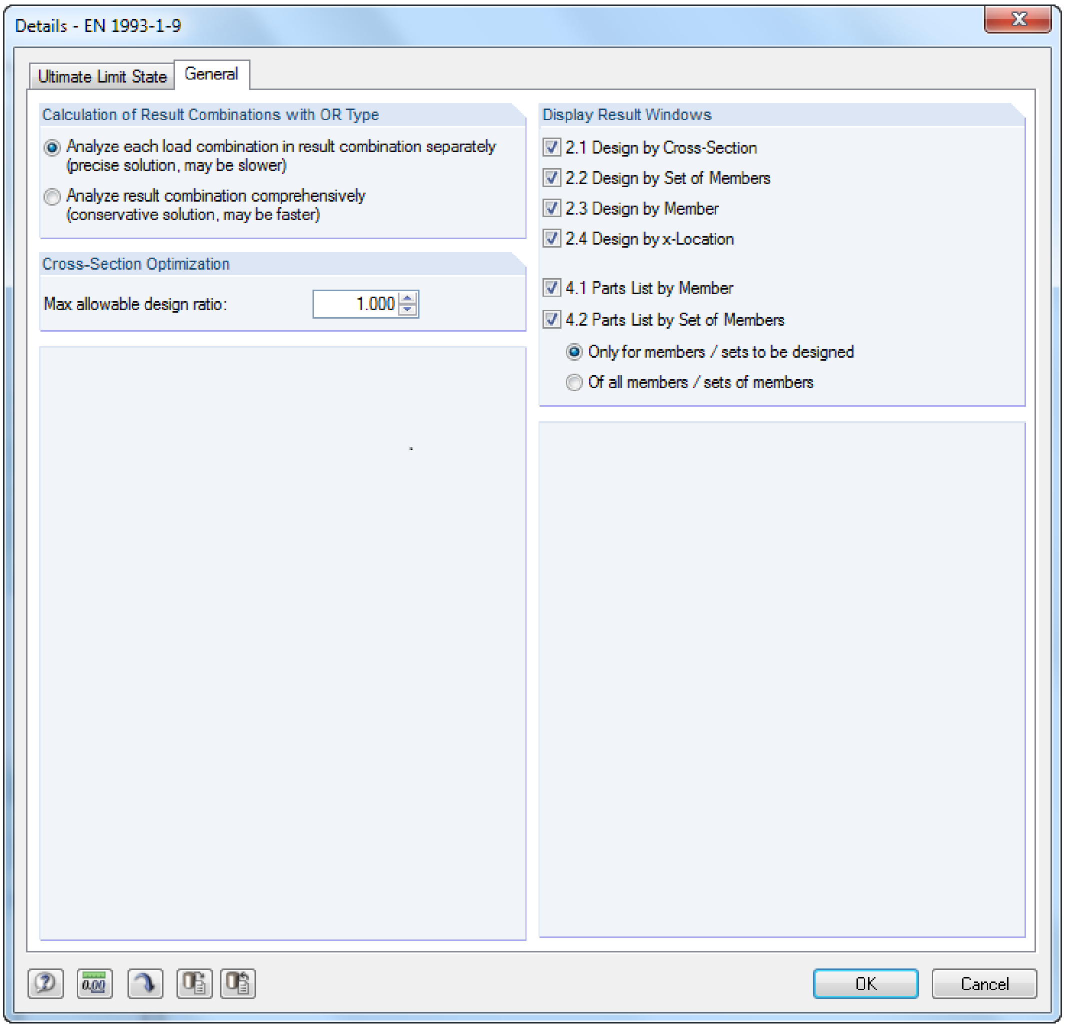


Figure 4.2: Dialog box *Details*, tab *General*

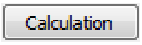
Cross-Section Optimization

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

Display Result Windows

In this dialog section, you can select the result tables including parts list that you want to be displayed. Those windows are described in Chapter 5.

4.2 Calculation Start



In all input windows of the RF-STEEL Fatigue Members add-on module, you can start the calculation using the [Calculation] button.
RF-STEEL Fatigue Members searches for the results of the load cases, load combinations and result combinations to be designed. If these cannot be found, the program starts the RFEM calculation to determine the design-relevant internal forces.
You can also start the calculation in the RFEM user interface: The *To Calculate* dialog box (menu ***Calculate*** → ***To Calculate***) lists design cases of the add-on modules like load cases and load combinations.

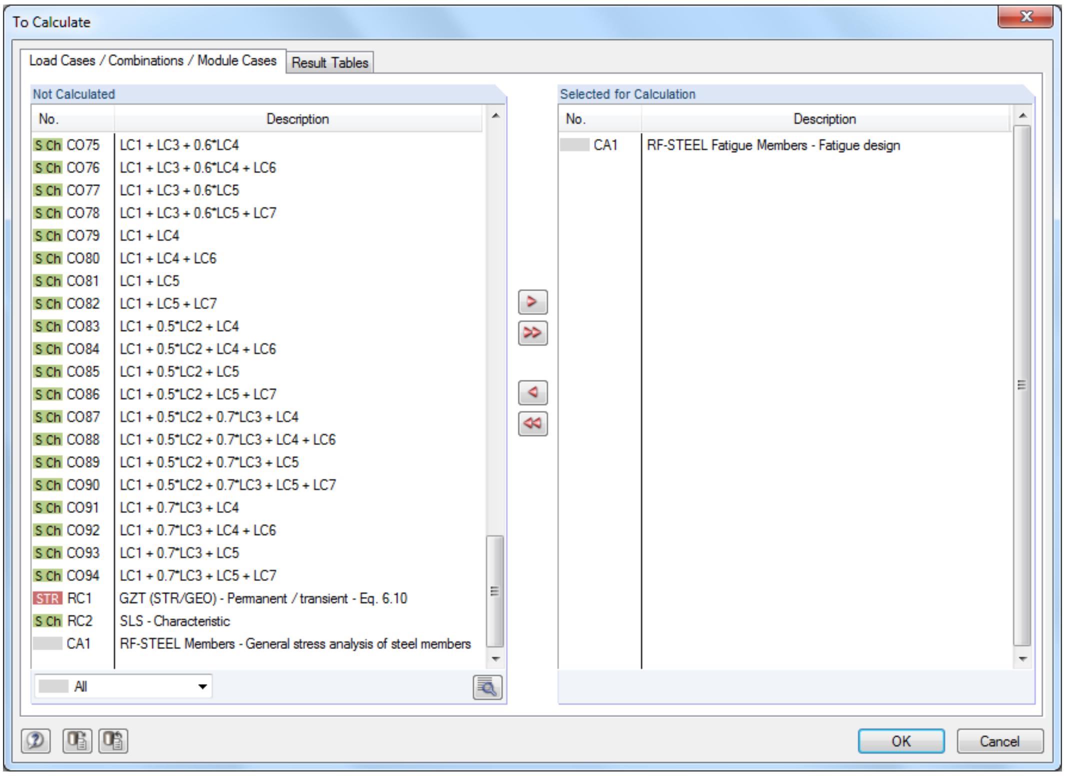
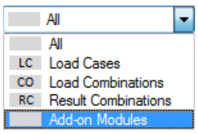


Figure 4.3: Dialog box *To Calculate*



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

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

To calculate a design case directly, you can also use the list in the toolbar: Select the RF-STEEL Fatigue Members design case in the toolbar list and click [Show Results].

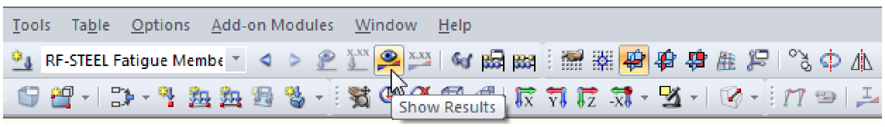


Figure 4.4: Direct calculation of RF-STEEL Fatigue Members case in RFEM

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

5 Results

The 2.1 *Design by Cross-Section* window is displayed immediately after the calculation.

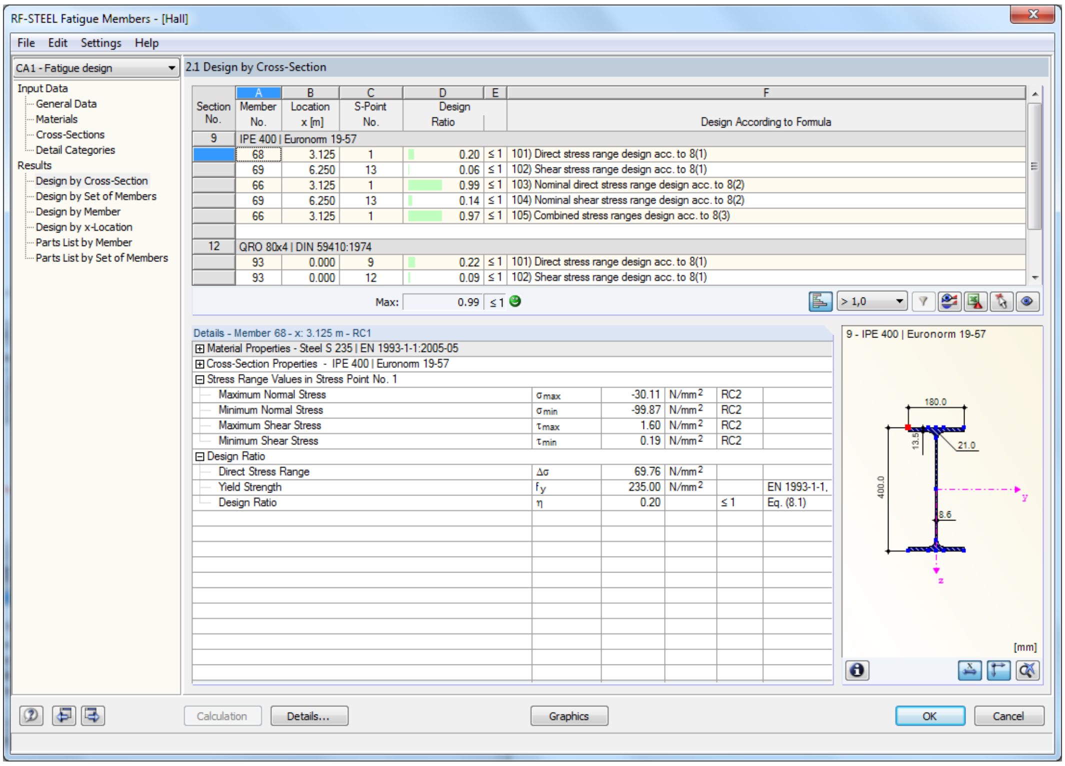
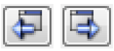


Figure 5.1: Result windows with designs and design ratios

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

Windows 4.1 and 4.2 show the parts lists by member and set of members.



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

[OK] saves the results. You exit RF-STEEL Fatigue Members and return to the main program.

Chapter 5 - *Results* describes the different result windows one by one. The evaluation of and checking results is described in Chapter 6 - Results Evaluation, page 29 ff.

5.1 Design by Cross-Section

In this result window, the maximum design ratios of all designed members and actions are listed by cross-section. The ratios for the internal forces of the governing load cases and combinations are sorted by stress type.

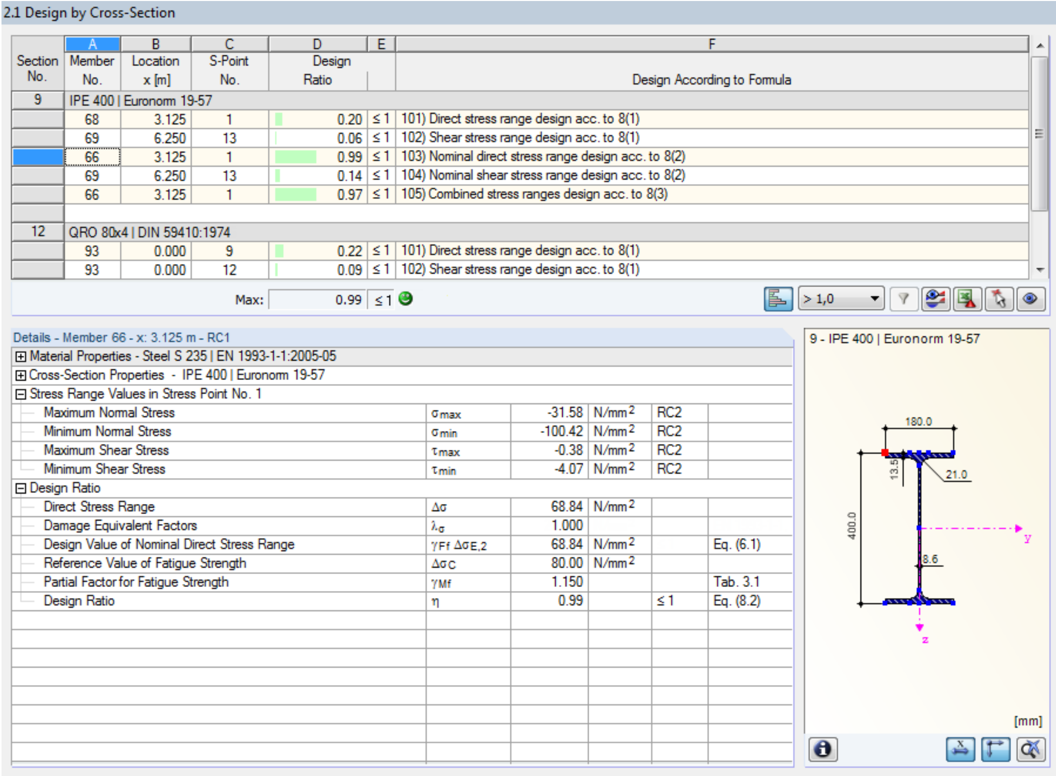


Figure 5.2: Window 2.1 *Design by Cross-Section*

Section No.

The results are listed by cross-section number. The description of the cross-section is displayed on the right of the cross-section number.

Member No.

It shows the number of the member with the maximum design ratio of the design type indicated in column F.

Location x

This column shows the respective x-location where the member's maximum design ratio occurs. For the output in form of tables, the program uses the following member locations *x*:

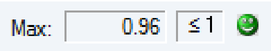
- Start and end node
- Division points according to possibly defined member division (see RFEM Table 1.16)
- Member division according to specification for member results (*Calculation Parameters* dialog box, *Global Register Parameters* tab in RFEM)
- Extreme values of internal forces

S-point No.

The design is carried out on certain stress points of the cross-section. These points are defined by centroidal distances, statical moments, and cross-section thicknesses, which allow for design according to [1]. The cross-section dialog graphic in the bottom right shows the currently selected stress point (that means the stress point of the table row where the pointer is placed) highlighted in red.

To check the stress point properties, click the [Info] button (see Chapter 3.2, page 17).

Design Ratio



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

The length of colored bars represents the respective design ratios.

Design According to Formula

This column displays the individual fatigue designs according to [1], Clause 8.

5.2 Design by Set of Members

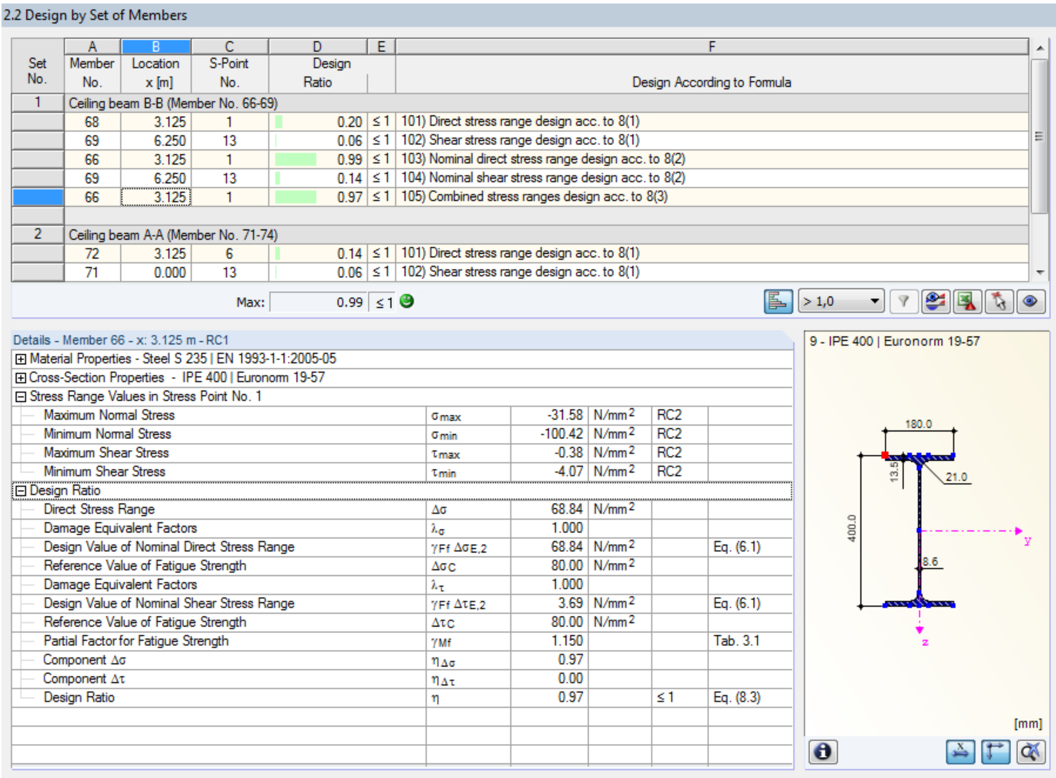


Figure 5.3: Window 2.2 *Design by Set of Members*

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

The column *Member No.* shows the number of the one member within the set of members that bears the maximum stress ratio for the respective stress types.

The output by sets of members clearly presents the design for an entire structural group (for example a continuous beam).

5.3 Design by Member

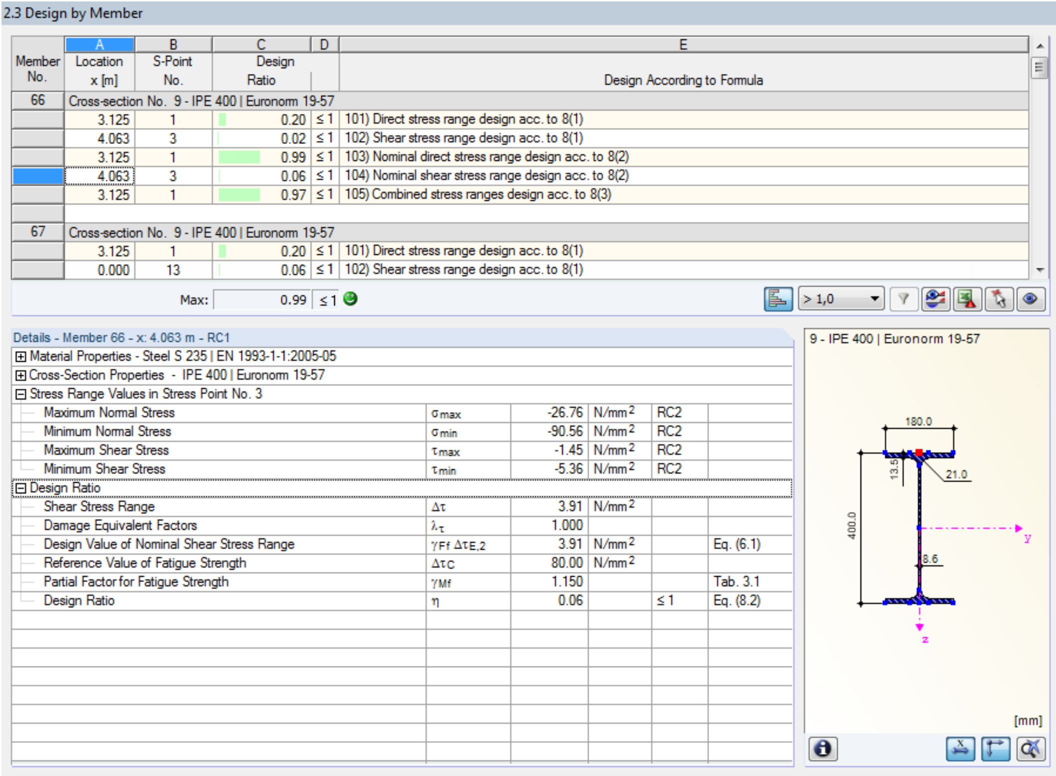


Figure 5.4: Window 2.3 *Design by Member*

This result window presents the maximum design ratios for different stress types sorted by member number. The columns are described in detail in Chapter 5.1 on page 23.

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

5.4 Design by x-Location

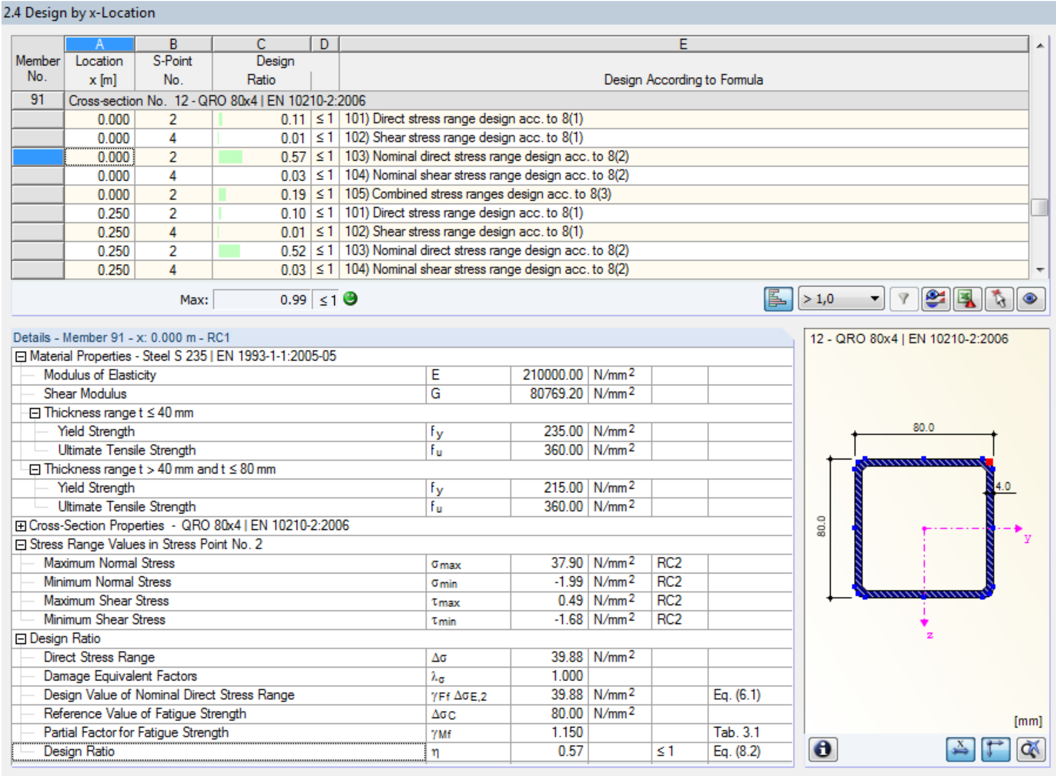


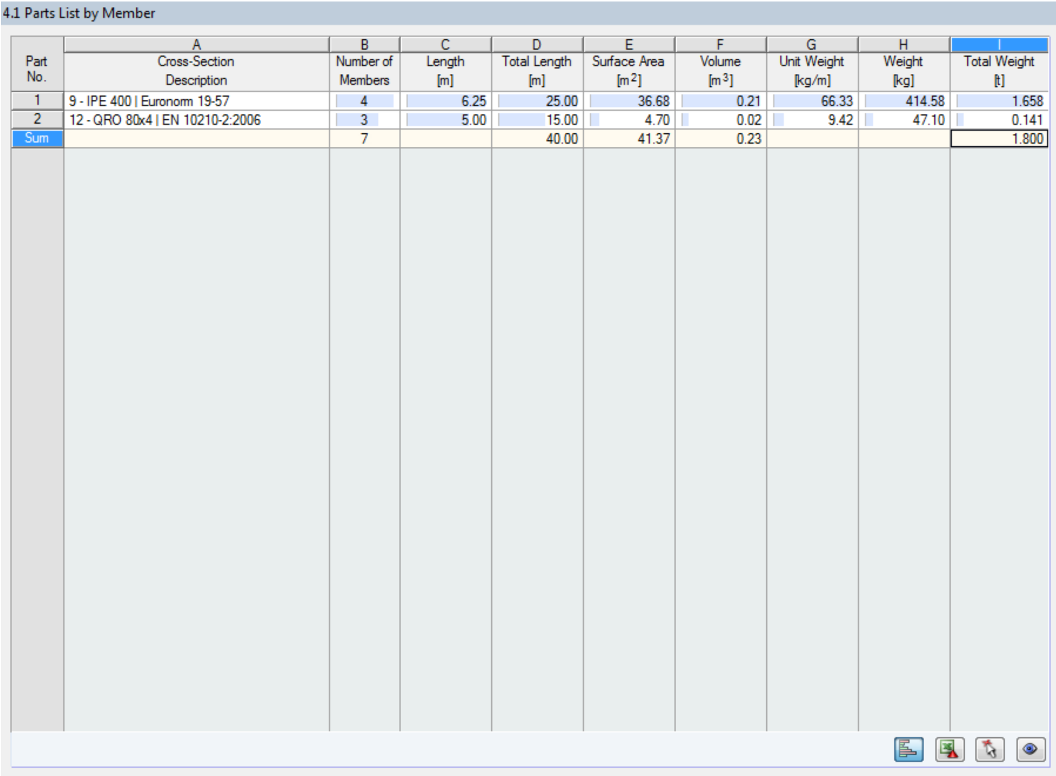
Figure 5.5: Window 2.4 *Design by x-Location*

This result window lists the maximum stresses for each member at all location **x** resulting from the division points defined in RFEM:

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

5.5 Parts List by Member

Finally, there is a summary of all cross-sections included in the design case.



4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	9 - IPE 400 | Euronorm 19-57	4	6.25	25.00	36.68	0.21	66.33	414.58	1.658
2	12 - QRO 80x4 | EN 10210-2:2006	3	5.00	15.00	4.70	0.02	9.42	47.10	0.141
Sum		7		40.00	41.37	0.23			1.800

Figure 5.6: Window 4.1 *Parts List by Member*

Details...

By default, this list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box (see Figure 4.2, page 20).

Part No.

The program automatically assigns part numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column shows the respective length of an individual member.

Total Length

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

Surface Area

For each part, the program indicates the surface area relative to the total length. The surface area is determined from the *Surface Area* of the cross-sections that can be seen in Windows 1.3 and 2.1 to 2.5 in the cross-section properties (see Figure 3.7, page 17).

Volume

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

Unit Weight

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

Weight

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

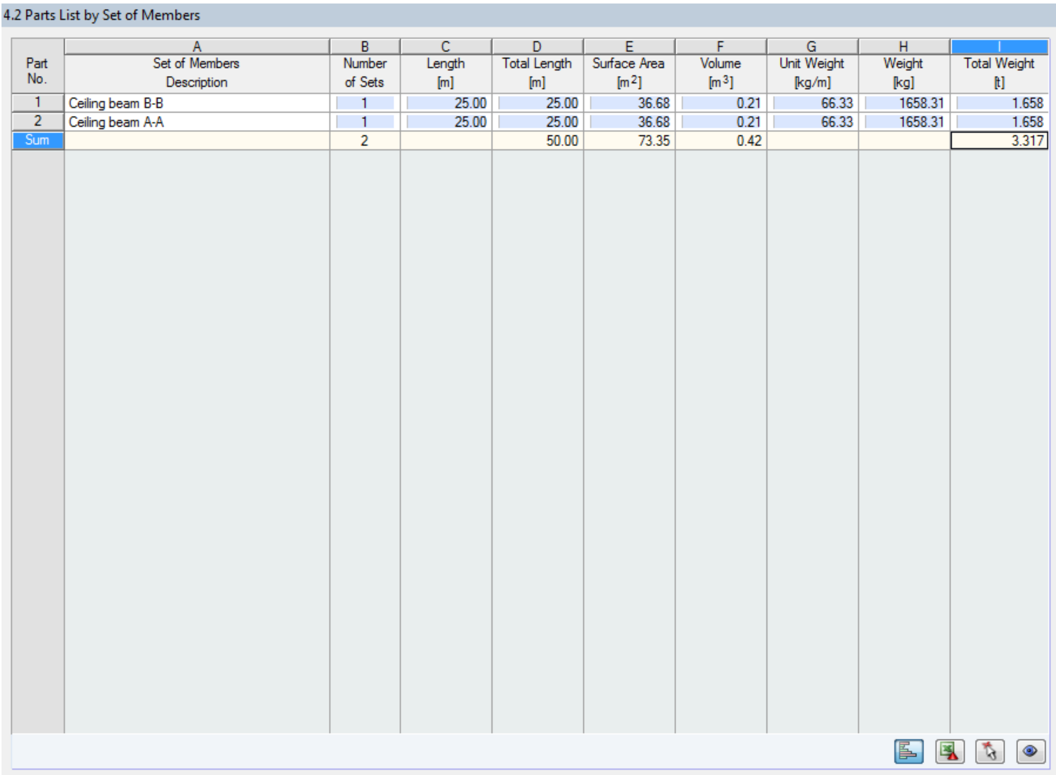
Total Weight

The final column indicates the total weight of each part.

Sum

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

5.6 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	Ceiling beam B-B	1	25.00	25.00	36.68	0.21	66.33	1658.31	1.658
2	Ceiling beam A-A	1	25.00	25.00	36.68	0.21	66.33	1658.31	1.658
Sum		2		50.00	73.35	0.42			3.317

Figure 5.7: Window 4.2 *Parts List by Set of Members*

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

Details on the various columns can be found in the previous Chapter 5.5. If there are different cross-sections used in a set of members, the program averages the surface area, the volume and the cross-section weight.

6 Results Evaluation

For the evaluation process, you can use the buttons below the table.

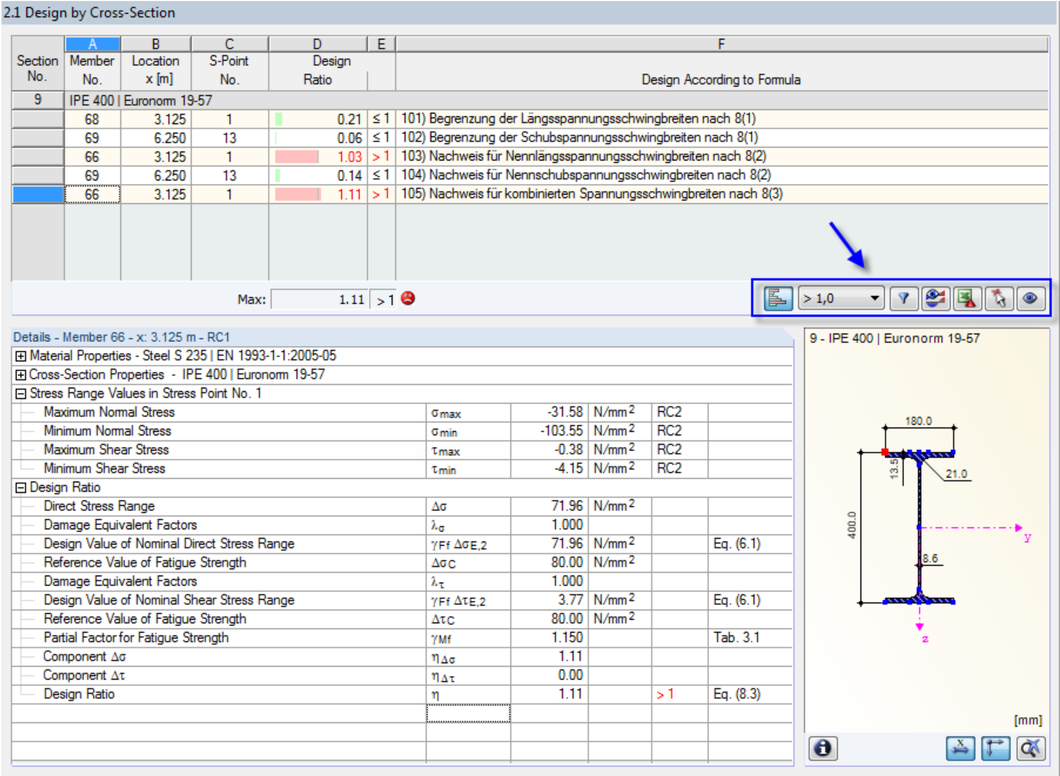


Figure 6.1: Buttons for results evaluation in tables

The buttons have the following functions:

Button	Description	Function
	Color Bars	Display or hide colored relation scales in the result windows
> 1,0 > 1,0 Max Define	Filter Parameter	Describe the criterion for filtering results in tables: ratios greater than 1, maximum or user-defined limit
	Apply Filter	Display only the rows with the filter parameter applied (design ratio > 1, maximum, user-defined limit)
	Result Diagrams	Open the *Result Diagram on Member* window → Chapter 6.2, page 31
	Excel Export	Export the table to MS Excel / OpenOffice → Chapter 8.4.2, page 42
	Member Selection	Select a member graphically to display its results in the table
	View Mode	Jump to the RFEM work window to change the view

Table 6.1: Buttons in the result windows 2.1 to 2.4

6.1 Results in RFEM Model

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

RFEM Background Graphic and View Mode

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

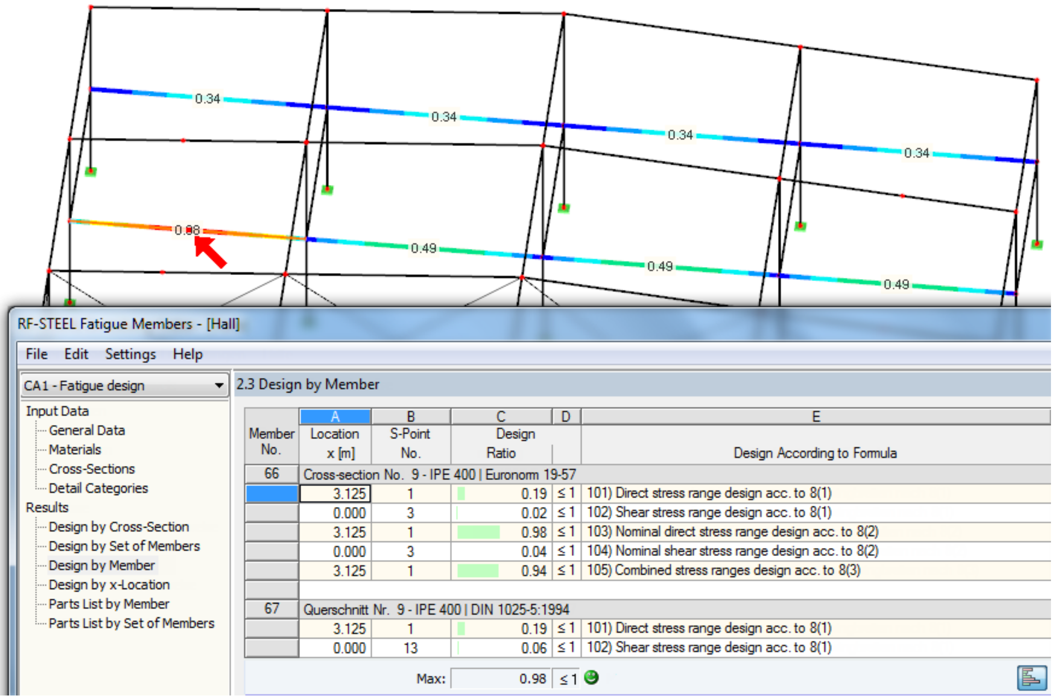
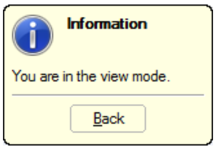


Figure 6.2: Indication of member and current *Location x* in the RFEM model



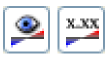
In case you cannot improve the display by moving the RF-STEEL EC3 module window, click [Jump to Graphic] to activate the *View mode*: The program hides the window so that you can modify the display in the RFEM user interface. The view mode provides the functions of the *View* menu, for example zooming, moving, or rotating the display. The pointer remains visible.

Click [Back] to return to the RF-STEEL Fatigue Members add-on module.

RFEM Work Window

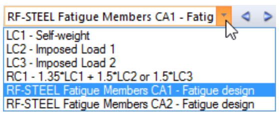
You can also graphically check the design ratios and stress ranges in the RFEM model: Click [Graphics] to exit the design module. In the RFEM work window, the design ratios are now displayed like the internal forces of a load case.

In the *Results* navigator, you can specify which design ratios or stress ranges you want to display in the graphic.



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

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



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

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

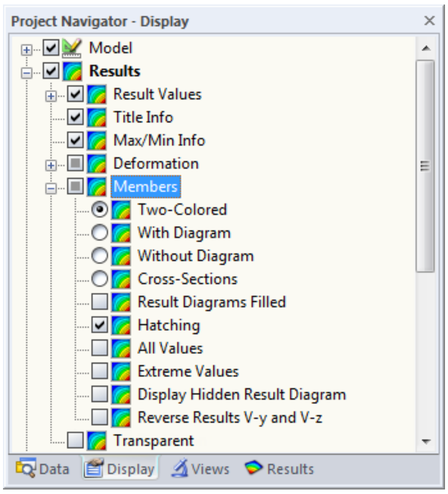


Figure 6.3: *Display* navigator: *Results* → *Members*

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

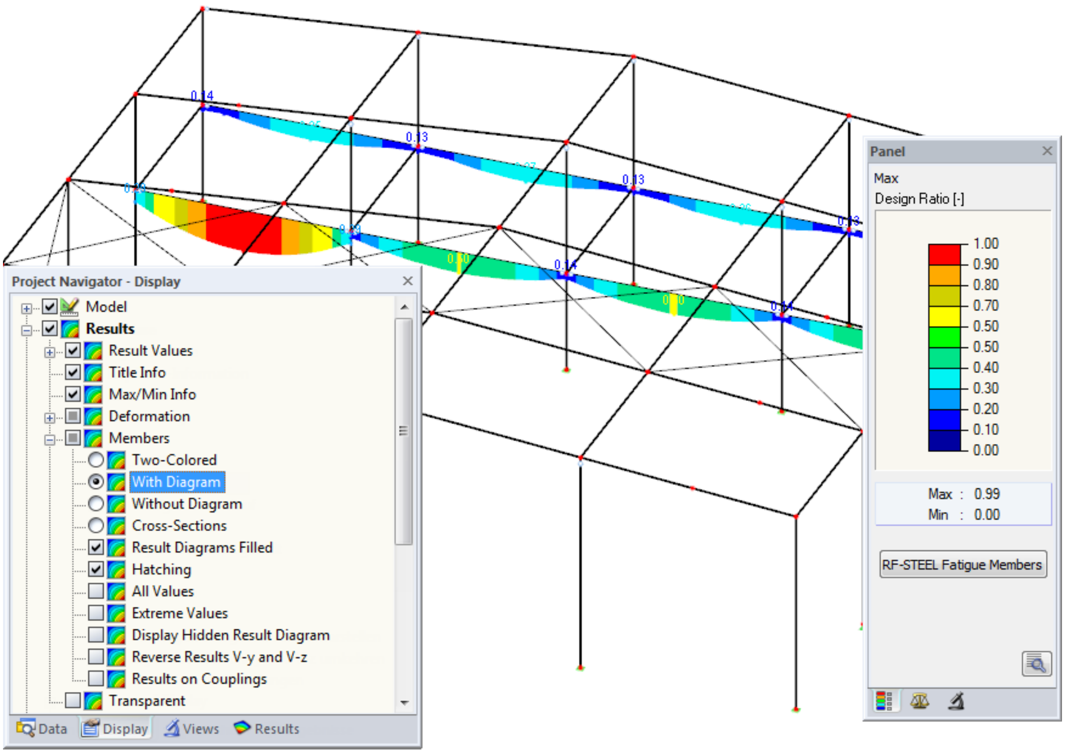
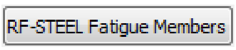


Figure 6.4: Design ratios with display options *With Diagram* and *Result Diagrams Filled*

You can transfer the graphics of stresses and design ratios to the printout report (see Chapter 7.2, page 35).



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

6.2 Result Diagrams

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

To do this, select the member (or set of members) in the RF-STEEL Fatigue Members result window by clicking in the table row of the member. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. The button is located below the upper result table (see Figure 6.1, page 29).

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

Results → Result Diagrams for Selected Members

or use the corresponding button in the RFEM toolbar.

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

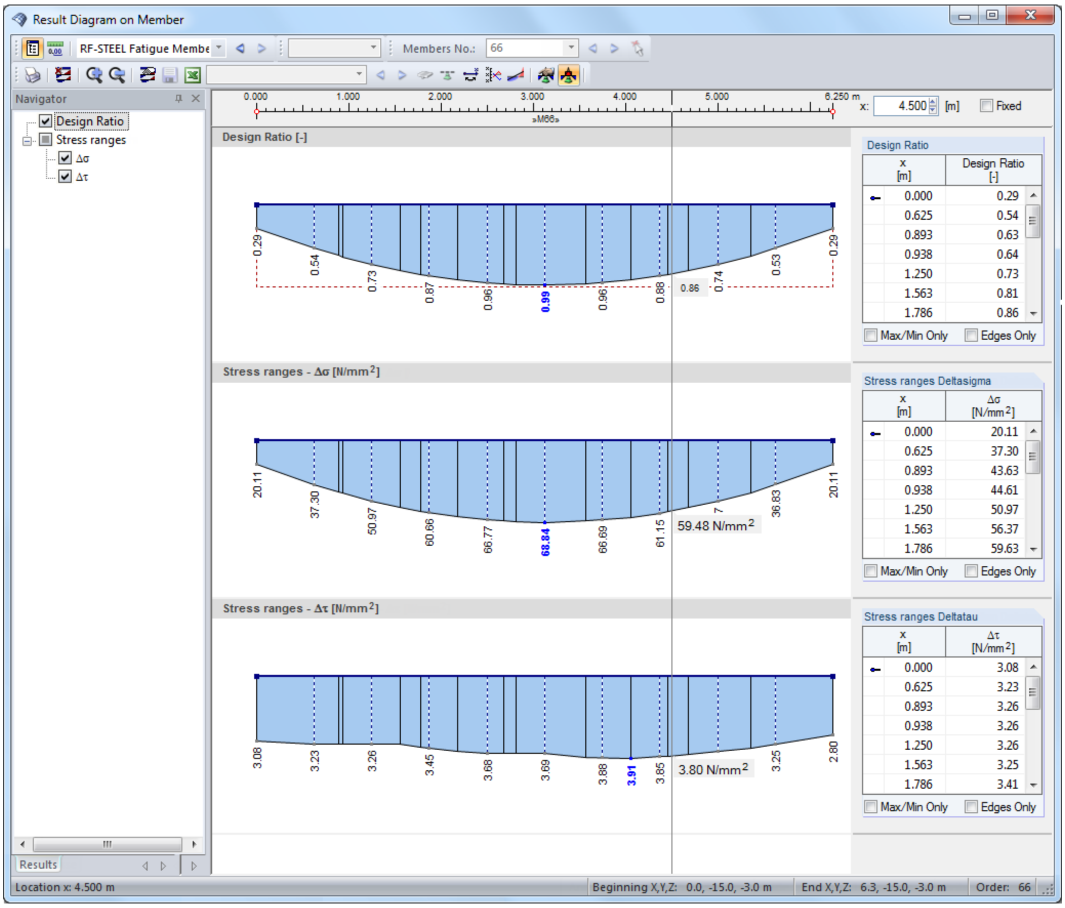
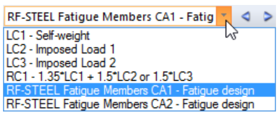


Figure 6.5: Dialog box *Result Diagram on Member*



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

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

6.3 Filter for Results

The RF-STEEL Fatigue Member result windows allow you to sort the results by various criteria. In addition, you can use the filter options described in Chapter 9.9 of the RFEM manual to evaluate the design results graphically.

You can use the *Visibility* option also for RF-STEEL Fatigue Members (see RFEM manual, Chapter 9.9.1) to filter the surfaces and members in order to evaluate them.

Filtering Designs

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

View → Control Panel (Color Scale → Factors → Filter)

or use the toolbar button shown on the left.

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

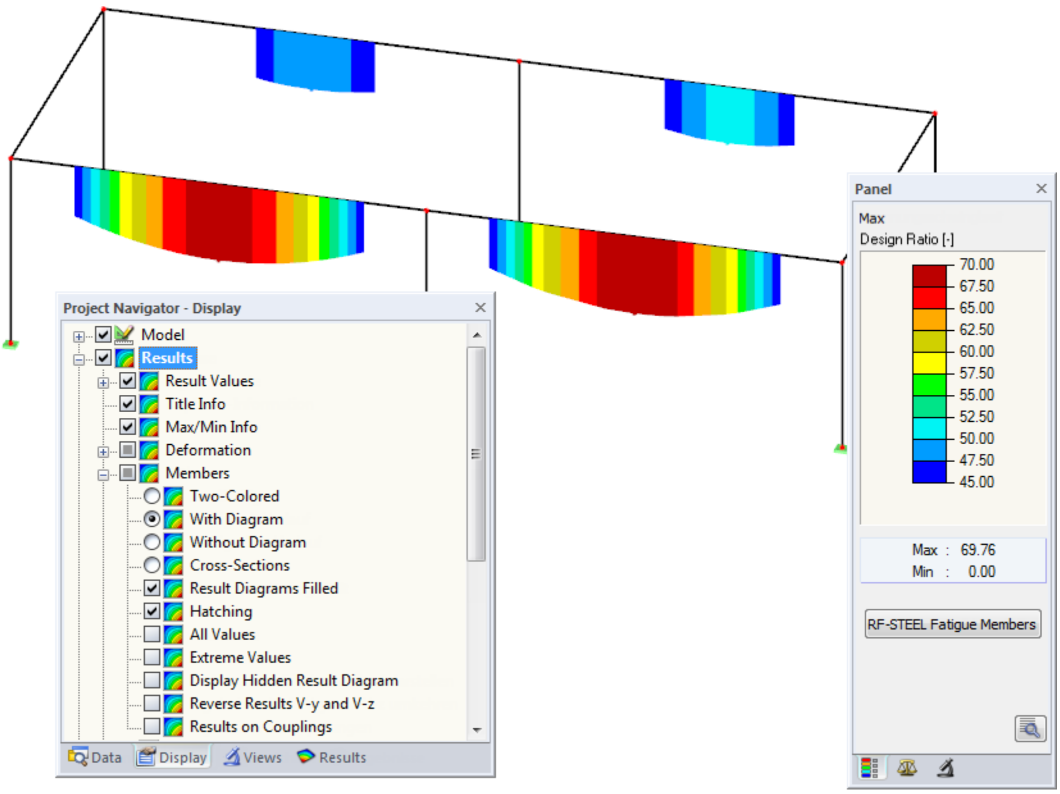


Figure 6.6: Filtering stress ranges with adjusted color scale

As the figure above shows, the color scale can be set in such a way that only stress ranges greater than 45 N/mm^2 are shown in a color range between blue and red. Furthermore, the color scale can be adjusted in such a way that one color range covers exactly 2.5 N/mm^2, for example.

The *Display Hidden Result Diagram* function in the *Display* navigator (**Results** → **Members**) shows all stress diagrams that are not within this value spectrum. Those diagrams are represented by dotted lines.

Filtering Members

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

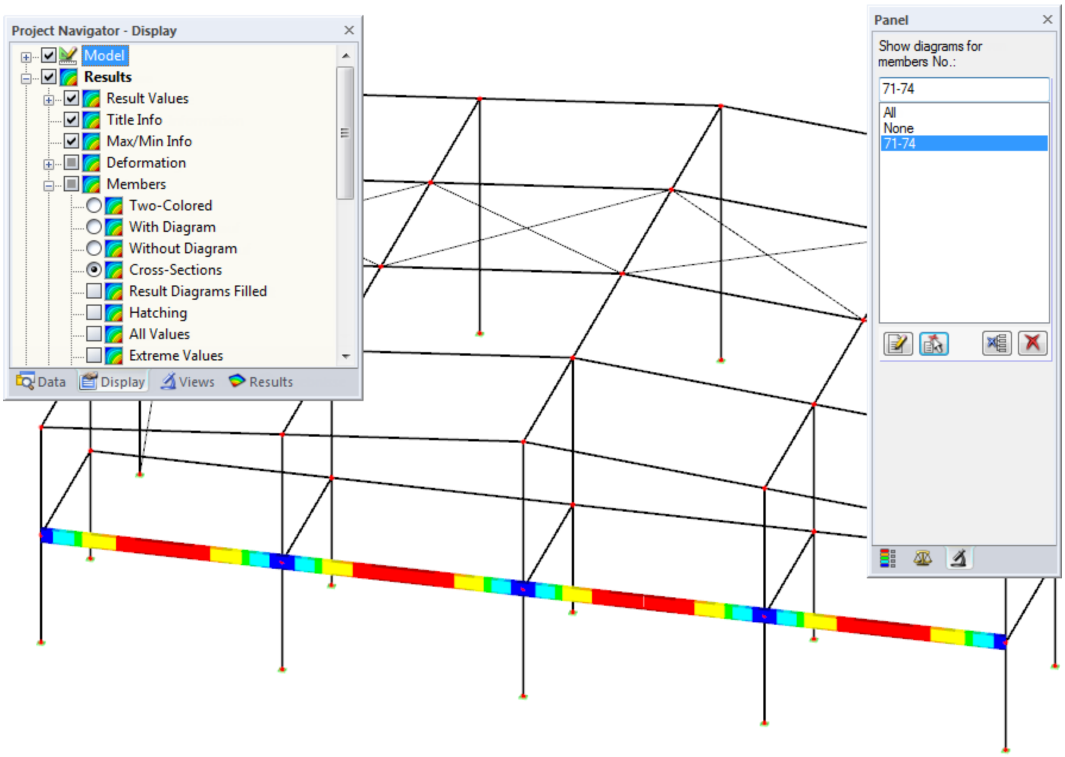


Figure 6.7: Member filter for stress ranges of a beam

In contrast to the visibility function, the model will be displayed completely in the graphic. The figure above shows the stress ranges of a beam. The remaining designed members are displayed in the model but without $\Delta\sigma$ results.

7 Printout

7.1 Printout Report

Similar to RFEM, the program generates a printout report for the RF-STEEL Fatigue Members results, to which you can add graphics and descriptions. The selection in the printout report determines what data from the design module will be included in the printout.

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

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

7.2 Graphic Printout

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

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

7.2.1 Results in RFEM Model

To print the currently displayed graphic of the design ratios or stress ranges, click

File → Print Graphic

or use the toolbar button shown on the left.

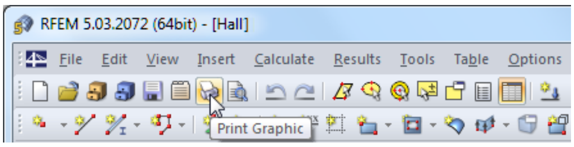


Figure 7.1: Button *Print Graphic* in RFEM toolbar

7.2.2 Result Diagrams

Also in the *Result Diagram on Member* dialog box (see Figure 6.5, page 32), you can click [Print] to transfer the graphic to the printout report or print it directly.

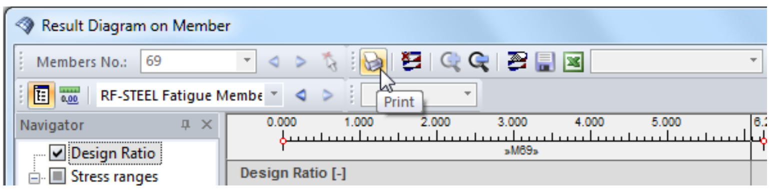


Figure 7.2: Button *Print* in the dialog box *Result Diagram on Member*

The following dialog box appears:

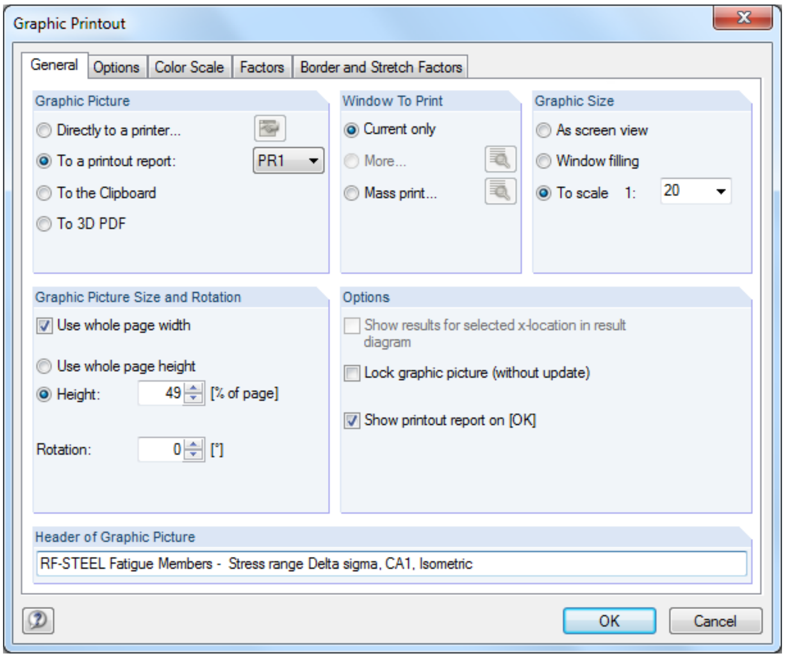
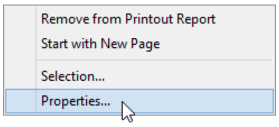


Figure 7.3: Dialog box *Graphic Printout*, tab *General*

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

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



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

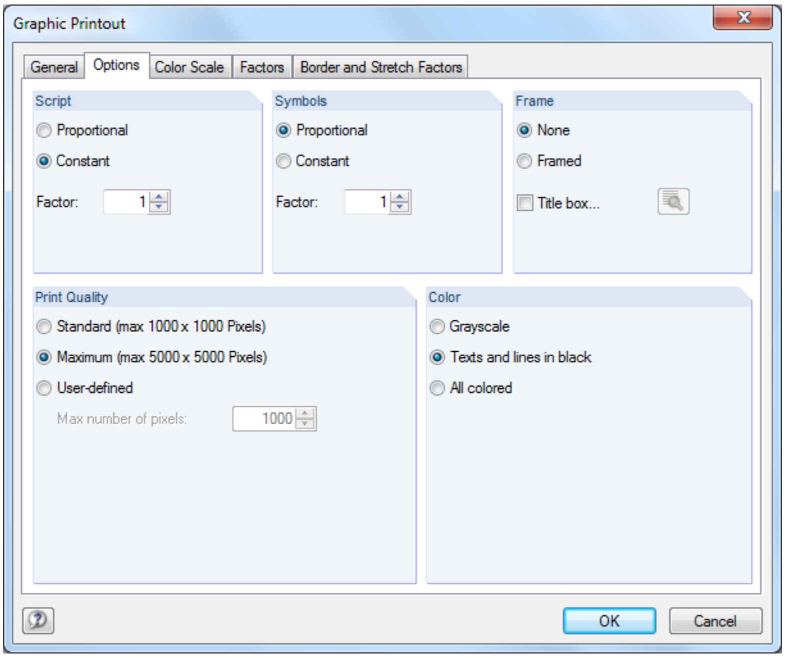


Figure 7.4: Dialog box *Graphic Printout*, tab *Options*

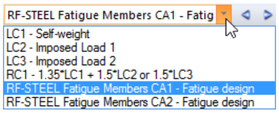
8 General Functions

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

8.1 Design Cases

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

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



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

Create New Design Case

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

File → New Case.

The following dialog box appears:

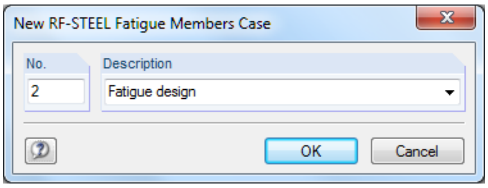


Figure 8.1: Dialog box *New RF-STEEL Fatigue Members-Case*

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

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

Rename Design Case

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

File → Rename Case.

The following dialog box appears:

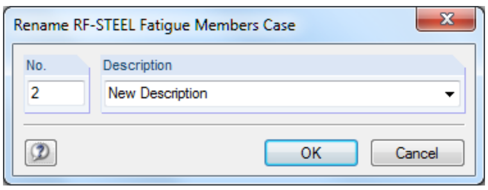


Figure 8.2: Dialog box *Rename RF-STEEL Fatigue Members-Case*

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

Copy Design Case

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

File → Copy Case.

The following dialog box appears:

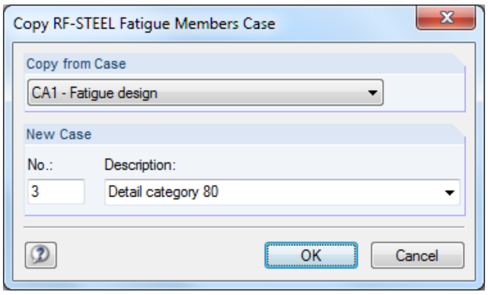


Figure 8.3: Dialog box *Copy RF-STEEL Fatigue Members-Case*

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

Delete Design Case

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

File → Delete Case.

The following dialog box appears:

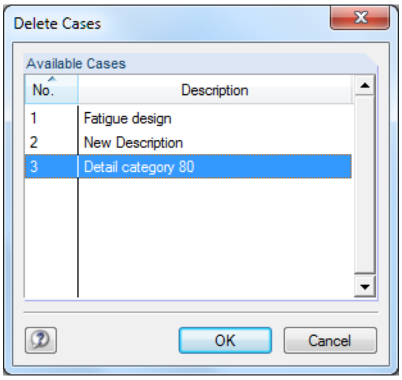
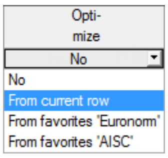


Figure 8.4: Dialog box *Delete Cases*

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

8.2 Cross-Section Optimization



The RF-STEEL Fatigue Members design module provides an option to optimize overloaded or little utilized cross-sections. To do this, open the drop-down lists in columns D or E of the relevant cross-sections in the 1.3 *Cross-Sections* window and select whether the cross-sections should be determined *From the current row* or from the user-defined *Favorites* (see Figure 3.5, page 14). You can also start the cross-section optimization in the result windows by using the shortcut menu.

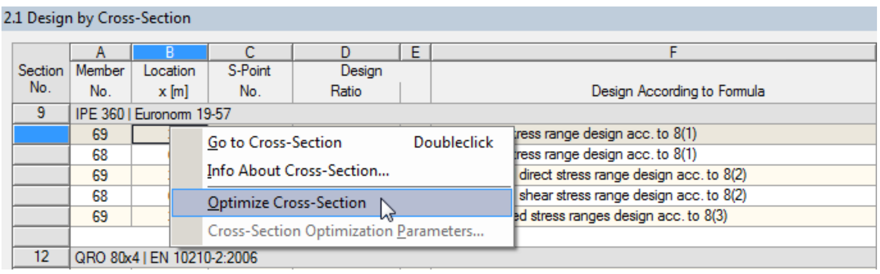


Figure 8.5: Shortcut menu for cross-section optimization

During the optimization process, RF-STEEL Fatigue Members determines the cross-section that fulfills the analysis requirements in the most "optimal" way, that is, it comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 4.2, page 20). The required cross-section properties are determined with the internal forces from RFEM. If another cross-section proves to be more favorable, this cross-section is used for the design. Then, the graphic in Window 1.3 shows two cross-sections: the original cross-section from RFEM and the optimized one (see Figure 8.7).

When optimizing a parametrized cross-section, the *Optimization* dialog box appears.

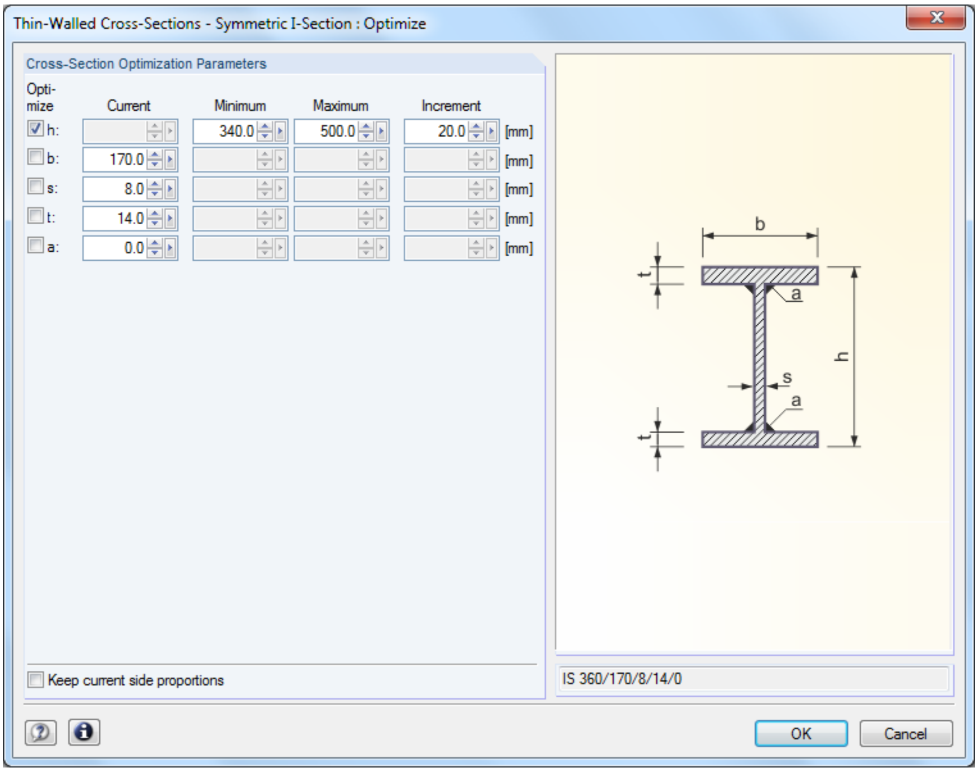


Figure 8.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section: Optimize*

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

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

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

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

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

Edit → **Export All Cross-Sections to RFEM**.

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

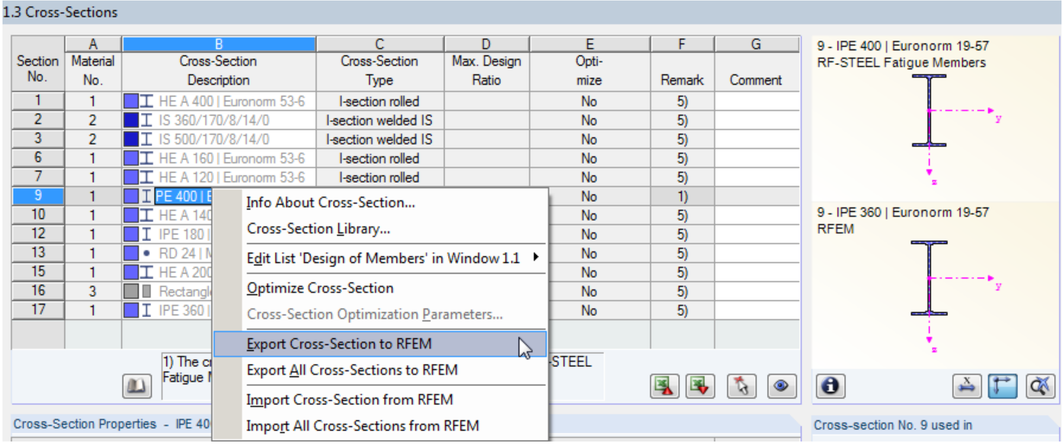


Figure 8.7: Shortcut menu in window 1.3 *Cross-Sections*

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

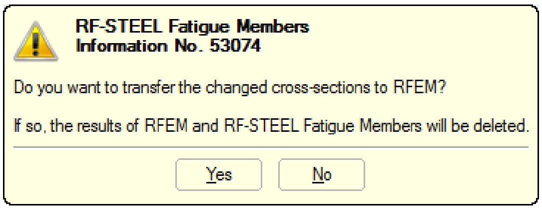


Figure 8.8: Query before transfer of modified cross-sections to RFEM

Calculation

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

If the modified cross-sections have not been exported to RFEM yet, you can reimport the original cross-sections in the design module by using the options shown in Figure 8.7. Please note that this option is only available in the 1.3 *Cross-Sections* window.

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

8.3 Units and Decimal Places

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

Settings → Units and Decimal Places.

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

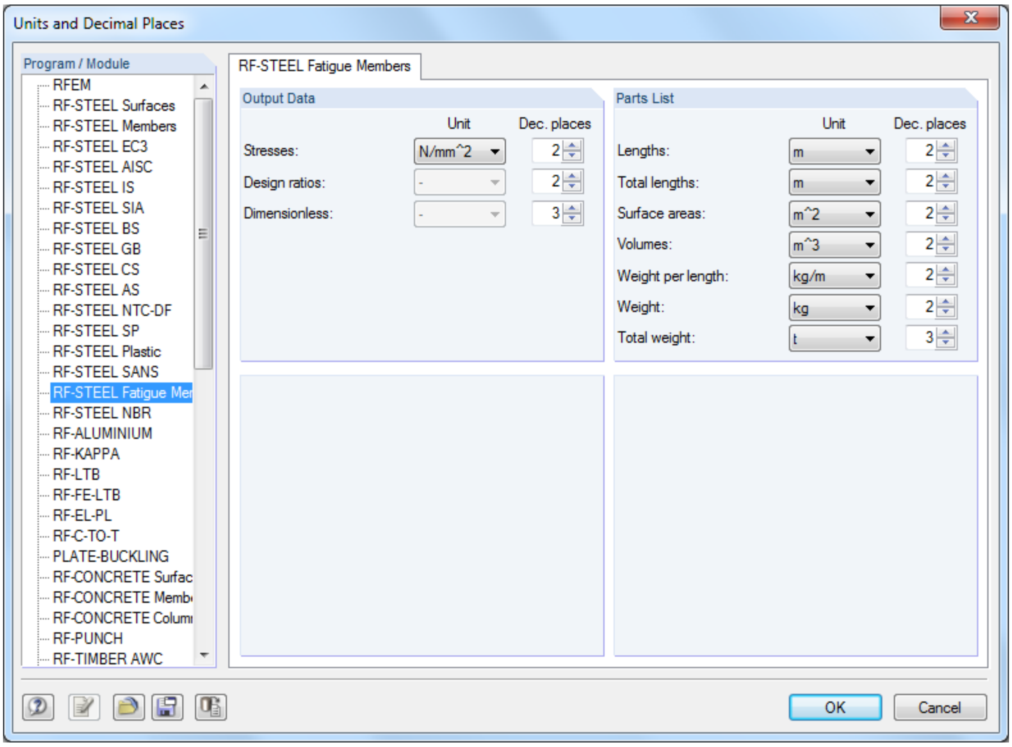


Figure 8.9: Dialog box *Units and Decimal Places*

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

8.4 Data Transfer

8.4.1 Exporting Materials to RFEM

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

Edit → Export All Materials to RFEM.

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

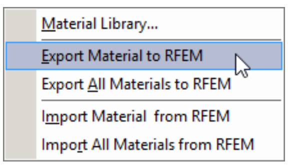
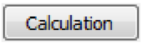


Figure 8.10: Shortcut menu in the window 1.2 *Materials*



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

If the modified cross-sections have not been exported to RFEM yet, you can reimport the original cross-sections in the design module by using the options shown in Figure 8.10. Please note, however, that this option is only available in the 1.2 *Materials* window.

8.4.2 Exporting Results

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

Clipboard

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

Printout report

You can print the data of RF-STEEL Fatigue Members in the global printout report (see Chapter 7.1, page 35) to export them subsequently. Then, in the printout report, click

File → **Export to RTF**.

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

Excel / OpenOffice

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

File → **Export Tables**.

The following export dialog box appears:

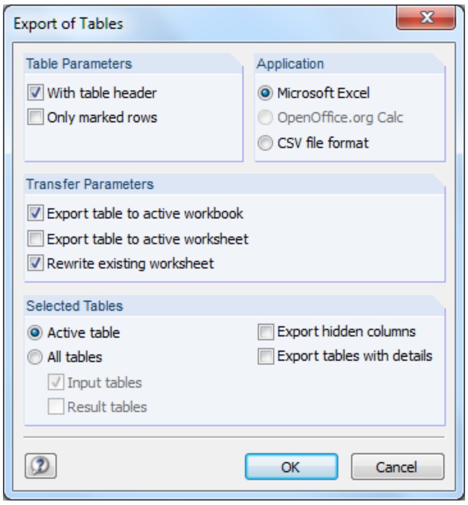
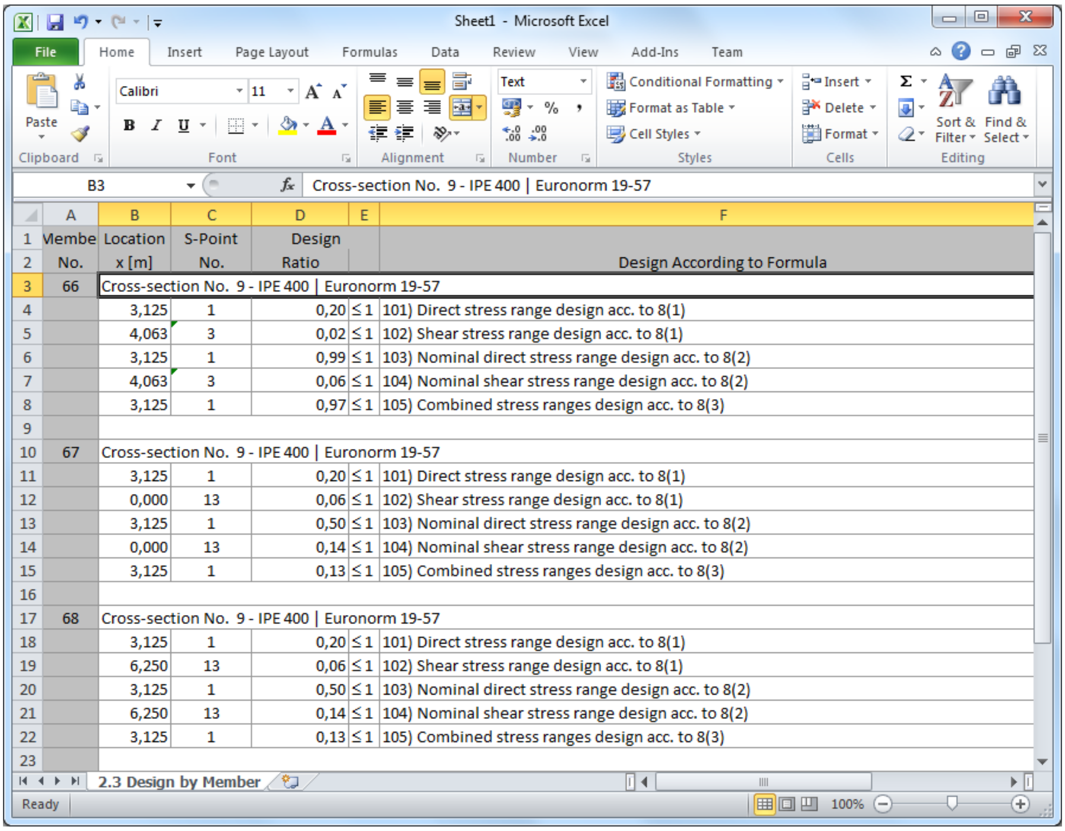


Figure 8.11: Dialog box *Export - MS Excel*

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



Membe No.	Location x [m]	S-Point No.	Design Ratio		Design According to Formula
66	Cross-section No. 9 - IPE 400 | Euronorm 19-57				
	3,125	1	0,20	≤ 1	101) Direct stress range design acc. to 8(1)
	4,063	3	0,02	≤ 1	102) Shear stress range design acc. to 8(1)
	3,125	1	0,99	≤ 1	103) Nominal direct stress range design acc. to 8(2)
	4,063	3	0,06	≤ 1	104) Nominal shear stress range design acc. to 8(2)
	3,125	1	0,97	≤ 1	105) Combined stress ranges design acc. to 8(3)
67	Cross-section No. 9 - IPE 400 | Euronorm 19-57				
	3,125	1	0,20	≤ 1	101) Direct stress range design acc. to 8(1)
	0,000	13	0,06	≤ 1	102) Shear stress range design acc. to 8(1)
	3,125	1	0,50	≤ 1	103) Nominal direct stress range design acc. to 8(2)
	0,000	13	0,14	≤ 1	104) Nominal shear stress range design acc. to 8(2)
	3,125	1	0,13	≤ 1	105) Combined stress ranges design acc. to 8(3)
68	Cross-section No. 9 - IPE 400 | Euronorm 19-57				
	3,125	1	0,20	≤ 1	101) Direct stress range design acc. to 8(1)
	6,250	13	0,06	≤ 1	102) Shear stress range design acc. to 8(1)
	3,125	1	0,50	≤ 1	103) Nominal direct stress range design acc. to 8(2)
	6,250	13	0,14	≤ 1	104) Nominal shear stress range design acc. to 8(2)
	3,125	1	0,13	≤ 1	105) Combined stress ranges design acc. to 8(3)

Figure 8.12: Results in *Excel*

9 Example: Tubular Truss

In the following example from [2], we examine a 36 m long ceiling beam of a machine hall formed as a planar truss. The beam is to be variably stressed by machine loads. These loads represent the upper loads, that means the loading varies between zero and the maximum value. The result is a constant stress range corresponding to the magnitude of the represented loads.

9.1 System and Loads

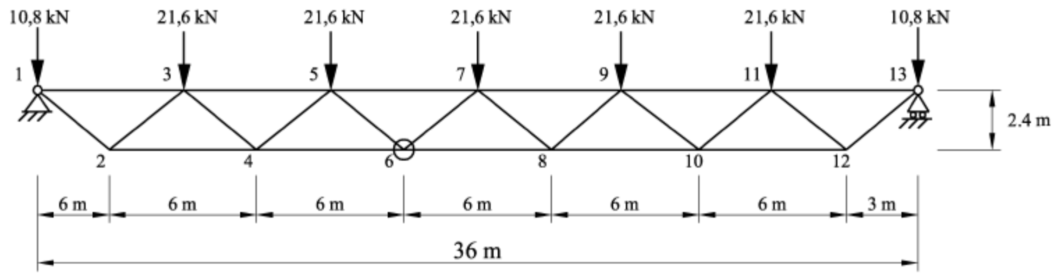


Figure 9.1: Planar truss, constant stress range

Cross-Sections

Top chord:	RO 219.1x7.1 (DIN EN 10220)
Struts:	RO 88.9x4 (DIN EN 10220)
Bottom chord:	RO 177.8x7.1 (DIN EN 10220)

Material

Structural steel S 235 (EN 10025-2:2004-11)

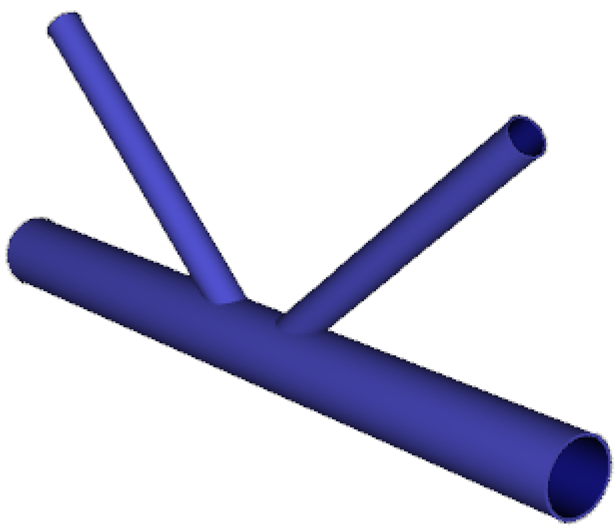


Figure 9.2: Node 6 in bottom chord of the truss

Determination of Member Loading

The beam is modeled with continuous chords and flexibly connected struts. The calculation of the structure provides the following normal forces and bending moments for the members on Node 6.

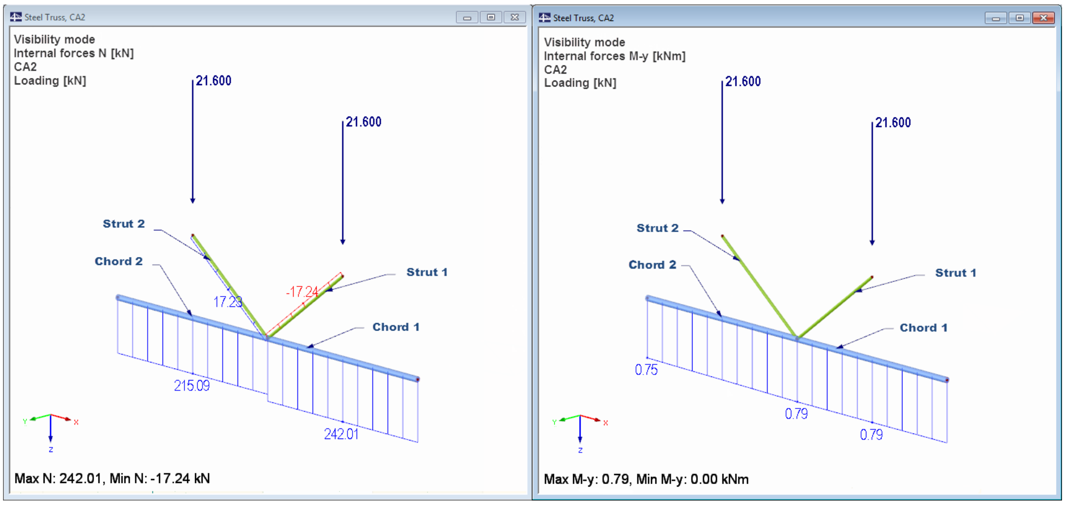


Figure 9.3: Internal forces N and M_y on Node 6

The internal forces can be presented as a load combination consisting of two load cases:

Load case 1: Normal forces in equilibrium

Load case 2: Remaining internal forces in chord (normal forces and bending)

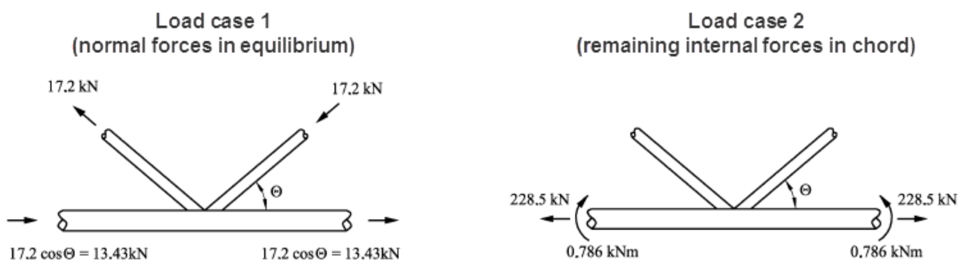


Figure 9.4: Load cases for Node 6

Nominal Stress Range of Critical Components

As pictured in Figure 9.3, the governing stress is in Chord 1, since there is the major tension force. Only Strut 2 of both diagonals is subjected to the tension force.

Load case 1: Normal forces in equilibrium

$$\sigma_{b,ax} = \frac{17.2}{1070} \cdot 10^3 = 16.07\,\mathrm{N/mm^2}$$

Load case 2: Remaining internal forces in chord

$$\sigma_{ch,ch} = \frac{228.50}{3807} \cdot 10^3 - \frac{0.786 \cdot 10^6}{0.156 \cdot 10^6} = 55.0\,\mathrm{N/mm^2}$$

Bending moments in the chord reduce the tension force of the chord on the side where the struts are connected.

In order to consider the actions of secondary bending moments, the nominal stress range must be multiplied by the increase factors k_1 according to [1], Tab 4.1. A factor of 1.3 applies for the stuts and a factor of 1.5 for the chords.

Load case 1: Normal forces in equilibrium

$$\sigma_{b,ax} = 1.3 \cdot 16.07 = 20.90\,\mathrm{N/mm^2}$$

Load case 2: Remaining internal forces in chord

$$\sigma_{ch,ch} = 1.5 \cdot 55.0 = 82.50\,\mathrm{N/mm^2}$$

Selection of Detail Category Class

When applying the nominal stress concept, the construction details for welded nodes of trusses are covered in detail categories of [1], Tab. 8.7. Application of the detail categories is possible only in compliance with specific geometrical boundary conditions. Parameters of the nodes in the example are within the allowable limits.

For the wall thicknesses ratio of tubes $t_0/t_i = 1.775$, the detail category 45 is applicable.

Fatigue Design and Lifetime of Node 6

With the detail category $\sigma_C = 45\ \mathrm{N/mm^2}$ for $2 \cdot 10^6$ cycles and the partial safety factor $\gamma_{Mf} = 1.15$, a number of the maximum possible stress cycles ensues from the general form of the fatigue strength curve or the S-N curve (Wöhler curve) until the fatigue failure occurs.

For the existing structural details according to [1], Tab. 8.7, the S-N curve inclinations is m = 5. The governing location is the main chord with the maximum stress range $\sigma_{ch,ch} = 83\ \mathrm{N/mm^2}$.

$$N_R = 2 \cdot 10^6 \cdot \frac{\left(\Delta\sigma_C/\gamma_{M\,f}\right)^m}{\Delta\sigma^m_{ch,ch}} = 2 \cdot 10^6 \cdot \frac{(45/1.15)^5}{83^5} = 46660 \text{ cycles}$$

The damage equivalent factor λ is obtained from the general formula as:

$$\lambda = \left(\frac{1}{2 \cdot 10^6} \cdot \sum_i^n \left(\frac{\Delta\sigma_{ch,ch}}{\max\,\Delta\sigma}\right)^m \cdot n_E\right)^{\frac{1}{m}} = \left(\frac{1}{2 \cdot 10^6} \cdot \sum_i^n \left(\frac{83}{83}\right)^5 \cdot 46660\right)^{\frac{1}{5}} = 0.471$$

9.2 Design in RF-STEEL Fatigue Members

9.2.1 Definition of Detail Category

Similarly to the failure graphic of the detail category 45 in [1], Tab. 8.7, the governing design area is the upper area of the bottom chord member (Chord 1). The design-relevant stress points are implemented for each cross-section in the cross-section library. In our example, tubular stress points 8 to 12 are relevant for the design. You can assign the corresponding detail category for these stress points in the 1.4 *Design Categories* window.

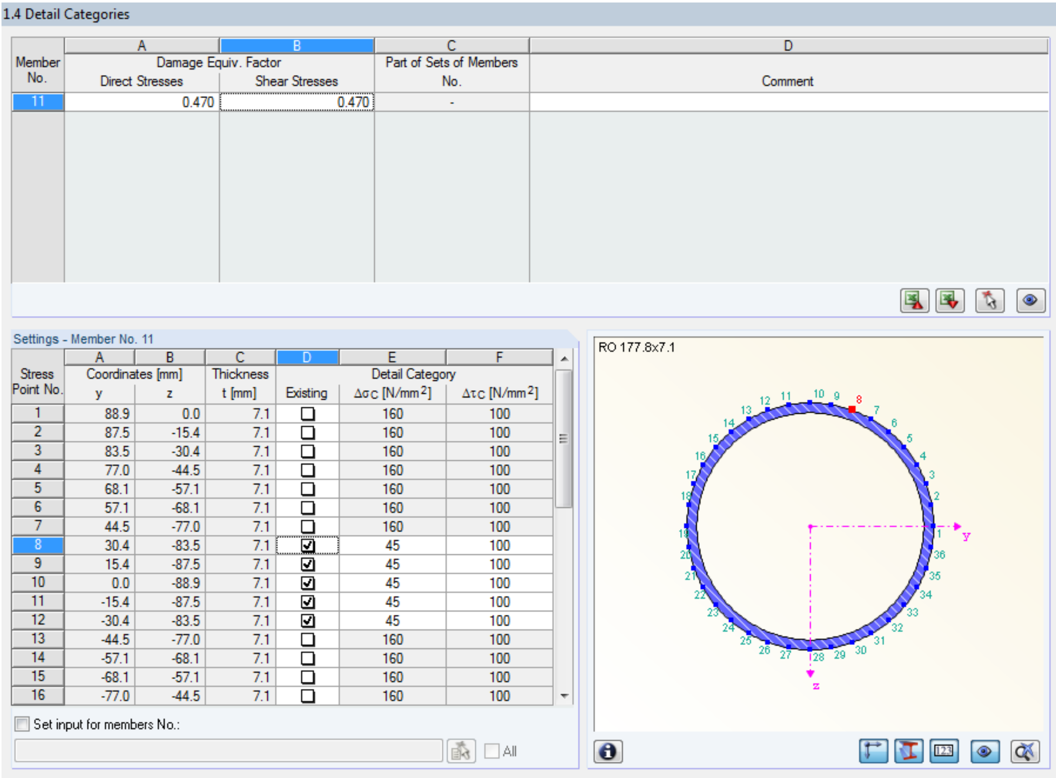


Figure 9.5: Window 1.4 *Detail Categories*

In Window 1.4, you can also define the damage equivalent factor for the bottom chord member. Due to preliminary considerations with the determination of the maximum load cycle as well as retroactive calculation of the respective damage equivalent factors, the design ratio is set to 100 % in RF-STEEL Fatigue Members.

9.2.2 Limitation of Direct Stress Ranges

In the 2.1 *Design by Cross-Section* window, the design ratio for the limitation of direct stress range is set to 24 %.

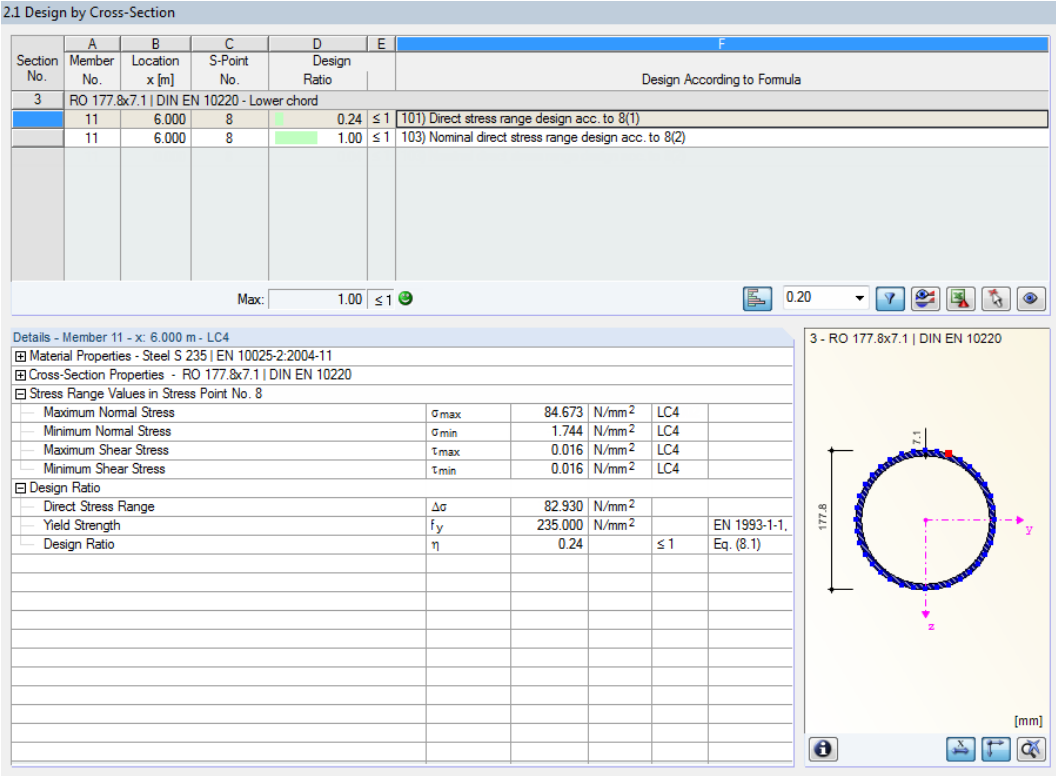


Figure 9.6: Window 2.1 *Design by Cross-Section* - Limitation of direct stress range

Limitation of stress ranges

$$\Delta\sigma \leq 1.5 \cdot f_y$$

$$82.93 \leq 1.5 \cdot 235$$

$$82.93 \leq 352.5$$

Check:

$$\frac{82.93}{352.5} = 0.24 \leq 1.00$$

The serviceability limit state design is thus fulfilled.

9.2.3 Design of Nominal Direct Stress Range

For the design of nominal direct stress range according to [1], Clause 8(2), the design ratio is set to 100%.

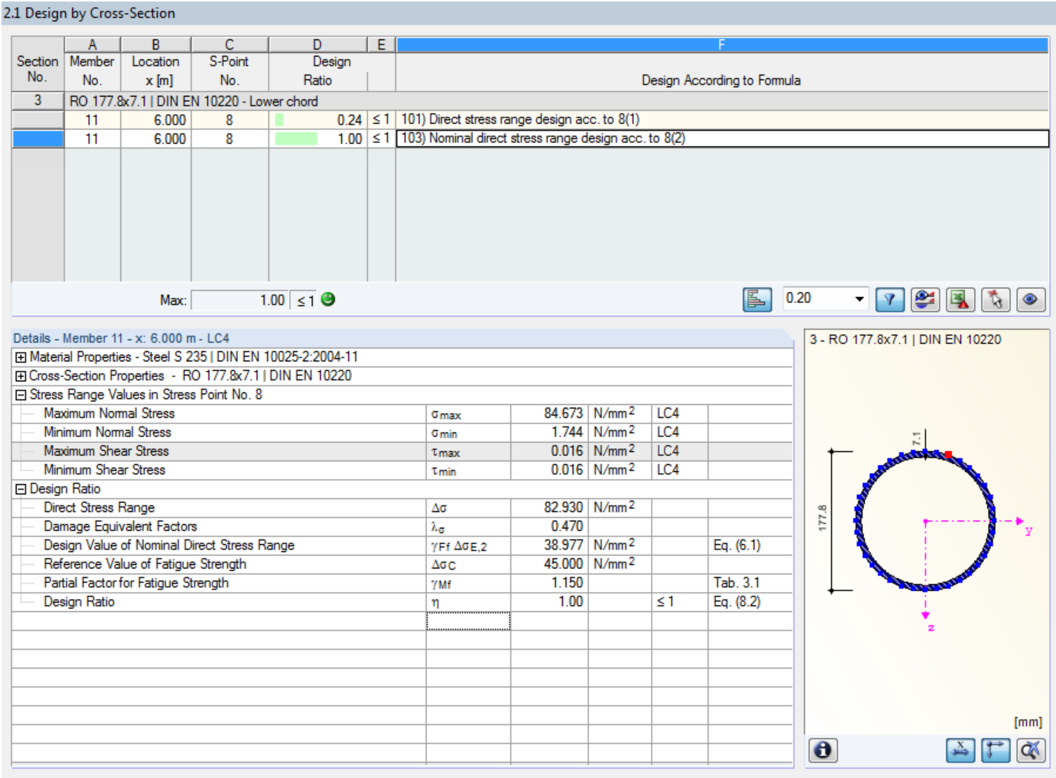


Figure 9.7: Window 2.1 *Design by Cross-Section* - Design for nominal direct stress range

Direct stresses

$$\gamma_{Ff} \cdot \Delta\sigma_{E,2} = \lambda \cdot \Delta\sigma$$

$$\gamma_{Ff} \cdot \Delta\sigma_{E,2} = 0.471 \cdot 82.93$$

$$\gamma_{Ff} \cdot \Delta\sigma_{E,2} = 38.977 \,\mathrm{N/mm^2}$$

Check:

$$\frac{\gamma_{Ff} \cdot \Delta\sigma_{E,2}}{\Delta\sigma_C / \gamma_{Mf}} \leq 1.00$$

$$\frac{38.977}{45/1.15} \leq 1.00$$

$$1.00 \leq 1.00$$

Literature

[1] *EN 1993-1-9: Bemessung und Konstruktion von Stahlbauten Teil 1-9: Ermüdung*. Beuth Verlag GmbH, Berlin, 2005.

[2] *Grundlagen und Erläuterung der neuen Ermüdungsnachweise nach Eurocode 3*. Nussbaumer A., Günther H.-P., Stahlbau-Kalender, Ernst & Sohn, Berlin, 2006.

Index