

Version December 2014

Add-on Module

RF-STEEL Fatigue Members

Fatigue Analysis of Steel Members

Program Description

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



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

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



Contents

Contents

Page

1.	Introduction 3
1.1	Add-on Module RF-STEEL Fatigue Members
1.2	Using the Manual
1.3	Opening RF-STEEL Fatigue Members Add-on Module
2.	Theoretical Principles 6
2.1	Design Concept
2.2	Design with Damage Equivalent Factors
2.2.1	Design Equations 7
2.2.2	Stress Range $\Delta \sigma$
2.2.3	Damage Equivalent Factor λ
3.	Input Data
3.1	General Data
3.2	Materials
3.3	Cross-Sections
3.4	Detail Categories
4.	Calculation
4.1	Detail Settings
4.1.1	Ultimate Limit State
4.1.2	General
4.2	Calculation Start
5.	Results
5.1	Design by Cross-Section
5.2	Design by Set of Members
5.3	Design by Member
5.4	Design by x-Location
5.5	Parts List by Member
5.6	Parts List by Set of Members
6.	Results Evaluation
6.1	Results in RFEM Model
6.2	Result Diagrams
6.3	Filter for Results
7.	Printout
7.1	Printout Report
7.2	Graphic Printout
7.2.1	Results in RFEM Model
7.2.2	Result Diagrams
8.	General Functions
8.1	Design Cases
8.2	Cross-Section Optimization
8.3	Units and Decimal Places
8.4	Data Transfer
8.4.1	Exporting Materials to RFEM
8.4.2	Exporting Results
9.	Example: Tubular Truss
9.1	System and Loads
9.2	Design in RF-STEEL Fatigue Members

⊿ ■ Dlubal



9.2.1	Definition of Detail Category
9.2.2	Limitation of Direct Stress Ranges
9.2.3	Design of Nominal Direct Stress Range
Α.	Literature
B.	Index

© Dlubal Software GmbH 2015



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 DLUBALTeam

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

۲

To open the program from the RFEM menu bar, click

```
\textbf{Add-on Modules} \rightarrow \textbf{Design-Steel} \rightarrow \textbf{RF-STEEL Fatigue Members}.
```

Add-on Modules Window		Help	0	
Curre	ent Module	હ્ય	🛤 🛤 🛛 🚟 🕷	🖗 🕸 🕸 🖉 🛸 💠 🕼 🗙 🐨 🔯 📽
Desig	gn - Steel 🔶 🕨	🥏	RF-STEEL Surface	s General stress analysis of steel surfaces
Desig	gn - Concrete 🔹 🕨	Ľ	RF-STEEL Membe	rs General stress analysis of steel members
Desig	gn - Timber 🔹 🕨	Fe	RF-STEEL EC3	Design of steel members according to Eurocode 3
Desig	gn - Aluminium 🕨 🕨	Also	RF-STEEL AISC	Design of steel members according to AISC (LRFD or ASD)
Dyna	mic 🕨	LIS	RF-STEEL IS	Design of steel members according to IS
Conr	ections 🕨	SIA	RF-STEEL SIA	Design of steel members according to SIA
Foun	idations 🕨	BS	RF-STEEL BS	Design of steel members according to BS
Stabi	lity 🕨	168	RF-STEEL GB	Design of steel members according to GB
Othe	rs 🕨	Ics	RF-STEEL CS	Design of steel members according to CS
Othe	rs P	TAS	RF-STEEL AS	Design of steel members according to AS
Exter	nal Modules 🔹 🕨	NIC	RF-STEEL NTC-DF	Design of steel members according to NTC-DF
		Isp	RF-STEEL SP	Design of steel members according to SP
		SANS	RF-STEEL Plastic	Design of steel members according to PIFM
		•	RF-STEEL SANS	Design of steel members according to SANS
		NBR	RF-STEEL NBR (D	emo version) Design of steel members according to NBR
		SANS	RF-STEEL Fatigue	Members Fatigue design of steel members
		1	RF-KAPPA	Flexural buckling analysis
		U	RF-LTB	Lateral-torsional and torsional-flexural buckling analysis
		₽. FE	RF-FE-LTB	Lateral-torsional and torsional-flexural buckling analysis by FEM
		I,	RF-EL-PL	Elastic-plastic design
			RF-C-TO-T	Analysis of limit slenderness ratios (c/t)
		2	RF-PLATE-BUCKL	NG Plate buckling analysis
		₽ ₽	VERBAND	Design of wind bracings for roofs



Navigator

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

Add-on Modules \rightarrow RF-STEEL Fatigue Members.



Figure 1.2: Data navigator: Add-on Modules \rightarrow 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 $\Delta\sigma_{\rm R}$ and $\Delta\tau_{\rm R}$. Furthermore, the concept of the partial safety factors is valid in [1]. The partial safety factor $\gamma_{\rm Mf}$ 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

Assessment method	Consequence of failure		
Assesment method	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 $\gamma_{\text{Mf}} = 1.0$ 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.

This method represents the standard method according to [1]. The damage equivalent stress ranges $\Delta \sigma_{\rm E,2}$ and $\Delta \tau_{\rm E,2}$ related to $2 \cdot 10^6$ stress cycles are to be contrasted with the limit values of the fatigue strength $\Delta \sigma_{\rm C}$ or $\Delta \tau_{\rm C}$ 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:

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

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

Fatigue designs

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

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

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

Simultaneous effect of normal and shear stresses

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

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

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

Determination of partial safety factors $\gamma_{\rm Mf}$ and $\gamma_{\rm Ff}$ is described in Chapter 2.1.

The fatigue strength $\Delta \sigma_{\rm C}$ or $\Delta \tau_{\rm C}$ for 2 · 10⁶ 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 $\Delta \sigma_{\scriptscriptstyle D}$
- 3 Cut-off limit $\Delta \sigma_L$



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



1 Detail category $\Delta \tau_{\rm C}$

2 Cut-off limit $\Delta \tau_{\rm L}$



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:

$$\Delta \sigma = \sigma_{m \, a x} - \sigma_{m \, i \, n}$$

$$\sigma_{max} : \text{ Maximum stress (with sign)}$$

$$\sigma_{min} : \text{ Minimum stress (with sign)}$$
(2.6)



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

OK

Cancel

- 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.

RF-STEEL Fatigue Members - [Ha	0						×
File Edit Settings Help							
CA1 - Fatigue design 👻	1.1 General [Data					
Input Data	Design of					Design Acc. to Standard	
General Data	Members:	66-69			A	EN 1993-1-9 -	
- Cross-Sections	Sets:	2			<u>برا</u>		
Detail Categories		-					
	Ultimate Lir	nit State					
	Existing Lo	ad Cases and Combinations		Selected for Des	sign		
	G LC1	Self-weight	•	S Ch RC2 S	SLS - Characteristic		
	Os LC2	Snow					
	Qw LC4	Wind in X					
	Qw LC5	Wind in Y					
	Imp LC6	Imperfections in X					
	A LC8	Imperfections in T					
	STR CO1	1.35*LC1					
	STR CO2	1.35*LC1 + LC6	>>				
	STR CO3	1.35"LC1 + LC7 1.35"LC1 + 1.5"LC2					
	STR CO5	1.35*LC1 + 1.5*LC2 + LC6					
	STR CO6	1.35*LC1 + 1.5*LC2 + LC7					
	STR CO7	1.35*LC1 + 1.5*LC2 + 1.05*LC3					
	STR CO8	1.35*LC1 + 1.5*LC2 + 1.05*LC3 +					
	STR CO1	0 1.35*LC1 + 1.5*LC2 + 1.05*LC3 +					Fatigue analysis of members
	STR CO1	1 1.35*LC1 + 1.5*LC2 + 1.05*LC3 +					and set of members
	STR COT	2 1.35°LC1 + 1.5°LC2 + 1.05°LC3 + 3 1.35°LC1 + 1.5°LC2 + 1.05°LC3 +					Cross-section optimization
	STR CO1	4 1.35*LC1 + 1.5*LC2 + 0.9*LC4					
	STR CO1	5 1.35*LC1 + 1.5*LC2 + 0.9*LC4 + L					Parts list
	STR CO1	6 1.35*LC1 + 1.5*LC2 + 0.9*LC5					
	All (103) 👻 🖉					TINIT
	Ceiling bean	ne					
	Coming Dean	10			<u>_</u>		
					· · · ·		It is a second second
	Colubri			_	Line 1		
	Calculation	Details		Grap	phics		OK Cancel

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.



 Al

 Load and Result Combinations

 LC
 Load Cases

 CO
 Load Combinations

 R
 Result Combinations

 G
 Permanent

 GS
 Snow (H ≤ 1000 m a.s.l.)

 GW
 Wind

 Accidental

 Imp<</td>
 Imperfection

All (104)

⊿ √	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 selected items from the list. Use the selected items from the list. Use the selected items from the 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.

L2 Mater	ials					
	A		B			
Material	Material				-	
No.	Description		Comm	ient		
1	Steel S 235 EN 1993-1-1:2005-05			-		
2	Steel S 355 EN 1993-1-1:2005-05				-	
3	Concrete C30/371 EN 1992-1-1:2004/AC:2010				-	
				📲 🗣 🔕 🧕		
Material I	Properties					
🖃 Main I	Properties					
Mo	dulus of Elasticity	E	210000.0	N/mm ²		
- She	ar Modulus	G	80769.2	N/mm ²		
Poi	sson´s Ratio	ν	0.300			
- Spe	ecific Weight	γ	78.50	kN/m ³		
- Coe	efficient of Thermal Expansion	α	1.2000E-05	1/K		
- Par	tial Safety Factor	γM	1.00		Material No. 2 upod	in
🖃 Additio	onal Properties				material No. 2 used	
- Coe	efficient for Limiting Stresses of Welds	αω	0.800		Cross-sections No.:	
- Cor	relation Factor for Fillet Welds	βw	0.900		23	
🗆 🕀 Thi	ckness Range t ≤ 4.00 cm					
- N	field Strength	fy	355.00	N/mm ²	Manhan Na i	
- L	Jltimate Strength	fu	490.00	N/mm ²	Members No.:	
🗆 🗇 Thi	ckness Range t > 4.00 cm and t ≤ 8.00 cm				3,8,13,18,23,28,41	.46
- `	field Strength	fy	335.00	N/mm ²		
- L	Jltimate Strength	fu	470.00	N/mm ²	Sets of members No	0.2
					-	
					Σ Lengths:	Σ Masses:
					24.09 [m]	1.508 [t]

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** \rightarrow **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 I or press function key [F7] to open the material list.

A	В
Material	
Description	Comment
Steel S 235 EN 1993-1-1:2005-05	
Steel S 235	EN 1993-1-1:2005-05
Steel S 275	EN 1993-1-1:2005-05
Steel S 355	EN 1993-1-1:2005-05
Steel S 450	EN 1993-1-1:2005-05
Steel S 275 N	EN 1993-1-1:2005-05
Steel S 275 NL	EN 1993-1-1:2005-05
Steel S 355 N	EN 1993-1-1:2005-05
Steel S 355 NL	EN 1993-1-1:2005-05
Steel S 420 N	EN 1993-1-1:2005-05
Steel S 420 NL	EN 1993-1-1:2005-05

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 \rightarrow Material Library

or use the button shown on the left.

The following Material Library dialog box appears.

Material Library				×
Filter	Material to Select			
Material category group:	Material Description	Standard		*
Metal -	Steel S 355	🔯 EN 19	93-1-1:2005-05	
	Steel S 450	EN 19	93-1-1:2005-05	
Material category:	Steel S 275 N	EN 19	93-1-1:2005-05	E
Steel 👻	Steel S 275 NI	EN 19	93-1-1-2005-05	
	Steel S 255 N	EN 19	a3.1.1.2005.05	
Standard group:	Steel S 355 N	EN 19	03-1-1.2003-05	
🖸 EN 👻	Steel S 355 NL	EN 13:	03-1-1.2003-05	
	Steel S 420 N	EN 19	93-1-1:2005-05	
Standard:	Steel S 420 NL	EN 19	3-1-1:2005-05	
🔯 EN 1993-1-1:2005-05 🛛 👻	Steel S 460 N	EN 19	93-1-1:2005-05	
	Steel S 460 NL	EN 19	93-1-1:2005-05	
	Steel S 275 M	EN 19	93-1-1:2005-05	
	Steel S 275 ML	EN 19	93-1-1:2005-05	
	Steel S 355 M	🔯 EN 19	93-1-1:2005-05	
🔲 Include invalid 🔯	Steel S 355 ML	🔯 EN 199	93-1-1:2005-05	-
Favorites only	*	1		×
Material Properties		Stee	LS 355 L EN 199	3-1-1:2005-05
A Main Properties				
Modulus of Elasticity		E	210000.0	N/mm ²
- Shear Modulus		G	80769.2	N/mm ²
Poisson's Ratio		v	0.300	
Specific Weight		γ	78.50	kN/m ³
Coefficient of Thermal Expansion	1	α	1.2000E-05	1/K
Additional Properties	7347 L L			
Coefficient for Limiting Stresses of Complexing Fraction For Stresses of	of Welds	αω	0.800	
Diplocase Rappet 54.00 cm	5	Þw	0.900	
Yield Strength		fx	355.0	N/mm ²
Ultimate Strength		fu .	490.0	N/mm ²
☐ Thickness Range t > 4.00 cm ar	nd t ≤ 8.00 cm		400.0	
Yield Strength		fy	335.0	N/mm ²
Ultimate Strength		fu	470.0	N/mm ²
			ОК	Cancel

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.

OK

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.

	A	В	(C	D	E	F	9 - IPE 360 Euronorm 19-57
ction	Material	Cross-Section	Cross-Section	Opti-			
No.	No.	Description	Туре	mize	Remark	Comment	
1	1	T HE A 400 Euronorm 53-62	I-section rolled	No	5)		+ 170.0
2	2	I IS 360/170/8/14/0	I-section welded	IS No	5)		+
3	2	I IS 500/170/8/14/0	I-section welded	IS No	5)		N 180
6	1	T HE A 160 Euronorm 53-62	I-section rolled	No	5)		
7	1	T HE A 120 Euronorm 53-62	I-section rolled	No	5)		
9	1	I IPE 360 Euronorm 19-57	I-section rolled	From Current Rov	(<u> </u>		60.0
10	1	T HE A 140 Euronorm 53-62	I-section rolled	No	5)		en
12	1	QRO 80x4 EN 10210-2:2006	Box rolled	From current row	/		8.0
13	1	 RD 24 Macsteel 	Round bar	From favorites 'E	uronom' 5)		
15	1	HE A 200 Euronorm 53-62	I-section rolled	No	5)		
16	3	Rectangle 200/200	Invalid	No	5)		
17	1	IPE 360 Euronorm 19-57	I-section rolled	No	5)		z
	(2) The cross-section will be optimiz	zed, utilizing the best	section from the table) 🔹 🐧 💌	
oss-Se	ction Prop	2) The cross-section will be optimiz	zed, utilizing the best	section from the table) 🔹 🍾 💿	Cross-section No. 9 used in
oss-Se Cross-	ection Prop Section Ty	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe	zed, utilizing the best	section from the table	ed		Cross-section No. 9 used in Members No.:
oss-Se Cross- Cross-	ection Prop Section Ty Sectional A	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe	zed, utilizing the best	I-section rolle	ed		Cross-section No. 9 used in Members No.:
oss-Se Cross- Cross- Effecti	ection Prop Section Ty Sectional A ve Shear A	2) The cross-section will be optimiz 2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea	zed, utilizing the best	I-section rolle	ed cm ² cm ²		Cross-section No. 9 used in Members No.: 66-69.71-74
oss-Se Cross- Effecti Effecti	ection Prop Section Ty Sectional A ve Shear A ve Shear A	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea	A	I-section roll I-section roll 72.70 7.x 36.06 7.y 26.90	ed cm ² cm ² cm ²		Cross-section No. 9 used in Members No.: 66-69,71-74
oss-Se Cross- Effecti Effecti Mome	ection Prop Section Ty Sectional A ve Shear A ve Shear A nt of Inertia	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea vea vea vea vea vea vea ve	A A A I J	I-section from the table I-section rollo 72.70 (x 36.06 (x) 26.90 16270.00	ed cm ² cm ² cm ² cm ² cm ⁴		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.:
oss-Se Cross- Effecti Effecti Momen	ection Prop Section Ty Sectional A ve Shear A ve Shear A nt of Inertia nt of Inertia	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea vea	A A A A A A A A A A A A A A A A A A A	I-section rolle I-section rolle 72.70 7.x 36.06 1.y 26.90 16270.00 1040.00 1040.00	ed cm ² cm ² cm ² cm ⁴ cm ⁴		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1,2
oss-Se Cross- Effecti Effecti Momer Torsio	ection Prop Section Ty Sectional A ve Shear A ve Shear A nt of Inettia nt of Inettia nal Constar	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea trea trea trea trea trea trea trea tr	A A A A A I y I z I t	I-section roll 72.70 7.x 36.06 7.y 26.90 16270.00 1040.00 37.50	ed cm ² cm ² cm ² cm ⁴ cm ⁴ cm ⁴		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1,2
oss-Se Cross- Effecti Effecti Momei Torsio Radius	ection Prop Section Ty Sectional A ve Shear A ve Shear A nt of Inetia nt of Inetia nal Constar s of Gyratio	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea trea trea trea	A A A A A A A I y I z I t I y	Isection from the table 72.70 72.70 72.70 72.70 16270.00 1040.00 37.50 150.0	ed cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses:
oss-Se Cross- Effecti Effecti Momen Torsion Radius Radius	ection Prop Section Ty Sectional A ve Shear A ve Shear A ve Shear A ht of Inertia nal Constar s of Gyratio s of Gyratio s of Gyratio	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea trea trea trea trea trea trea trea tr	A A A A A I y I z I z I z I z C	Isection from the table 1-section roll 72.70 xx 36.06 xy 26.90 16270.00 16270.00 1040.00 37.50 150.0 37.90	ed cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses: 50.00 [m] 2.853
oss-Se Cross- Effecti Effecti Momen Torsion Radius Elastic	ection Prop Section Ty Sectional A ve Shear A ve Shear A ve Shear A not of Inertia nal Constar s of Gyratio s of Gyratio s Section M Section M	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea trea trea trea trea trea trea trea tr	A A A A I y I z S S C C	I-section rolle 1-section rolle 72.70 7.x 36.06 7.y 22.90 16270.00 1040.00 37.50 150.0 37.9 1.x 904.00 12.20 12.20 1.22	d cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ³		Cross-section No. 9 used in Members No.: 66-69.71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses: 50.00 [m] 2.853
Cross- Cross- Effecti Effecti Momer Torsion Radius Radius Elastic Elastic	ection Prop Section Ty Sectional A ve Shear A ve Shear A to f Inertia nal Constar s of Gyratio s of Gyratio s of Gyratio Section M Section M	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea trea trea trea trea trea trea trea tr	A A A A A A A A A A A A A A A A A A A	I-section roll I-section roll 72.70 72.70 16270.00 1620.00 1	ed cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ³ cm ³		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses: 50.00 [m] 2.853 Material: 2.853
oss-Se Cross- Effecti Effecti Mome Torsiol Radius Elastic Elastic Plastic	Ection Prop Section Ty Sectional A ve Shear A ve Shear A ve Shear A nt of Inertia nal Constar s of Gyratio s of Gyratio Section M Section M Section M	2) The cross-section will be optimiz erties PE 360 Euronorm 19-57 pe vea vea vea trea trea trea fi	A A A A A A A A A A A A A A A A A A A	Isection from the table 72.70 72.70 72.70 72.70 16270.00 1040.00 37.50 150.0 37.50 150.0 37.50 150.0 122.00 14.y 123.00 pl.x 1020.00 1020.	ad cm ² cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ⁴ cm ³ cm		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses: 50.00 [m] 2.853 Material: 1.0 0055
oss-Se Cross- Cross- Effecti Effecti Momel Torsiol Radius Bastic Elastic Plastic Plastic	ection Prop Section Ty Sectional A ve Shear A ve Shear A not of Inertia nal Constar s of Gyratio s of Gyratio s of Gyratio s of Gyratio s Section M S Section M S Section M	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea vea in n n n odulus lodulus lodulus lodulus lodulus	A A A A A A A A A A A A A A A A A B Y Iz Iz S S S S S S S S S S S S S S S S S	l-section rolle 72.70 (x 36.06 16270.00 16270.00 16270.00 1750.0 137.50 150.0 37.9 al.x 904.00 al.y 122.00 pl.x 1020.00 pl.x 1021.00 al.32500.00 1313600.00	d cm ² cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ³ cm ⁴ cm ⁴ cm ⁴ cm ³ cm ³ cm ³ cm ³ cm ³ cm ³ cm ³ cm ³ cm ⁵ cm ⁶ cm ⁶		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses: 50.00 [m] 2.853 Material: 1 - Steel S 235
oss-Se Cross- Effecti Effecti Momer Torsion Radius Elastic Elastic Plastic Varpin Statics	ection Prop Section Ty Sectional A ve Shear A to of Inertia nal Constar s of Gyratio s of Gyratio s of Gyratio s of Gyratio s of Gyratio s Section M s Section M s Section M s Section M s Section M	2) The cross-section will be optimiz erties - IPE 360 Euronorm 19-57 pe vea vea vea trea trea trea trea trea trea trea tr	A A A A A A A A A A A A A A A V V V V V	l-section roll 	ed cm ² cm ² cm ² cm ⁴ cm ⁴ cm ⁴ cm ³ cm ³ cm ³ cm ³ cm ³ cm ³		Cross-section No. 9 used in Members No.: 66-69,71-74 Sets of members No.: 1.2 Σ Lengths: Σ Masses: 50.00 [m] 2.853 Material: 1 - Steel S 235

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.



3

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.

Thin-Walled Cross-Sections - Symmetric I-Section							
Cross-Section Type $\begin{bmatrix} I & I & I & T \\ T & L & L & D \\ \hline I & I & T & Y \\ \hline O & \nabla & I & I \\ \hline I & I & T & I \\ \hline I & I & T & I \\ \hline I & I & I & I \\ \hline I & L & L & C \\ \hline S & O & \nabla \\ \hline Favorites Group \\ \end{bmatrix}$	Parameters h: 360.0 ⊕h [mm] b: 170.0 ⊕h [mm] s: 8.0 ⊕h [mm] t: 14.0 ⊕h [mm] a: 0.0 ⊕h [mm]						
	Ŕ	IS 360/170/8/14/0					
		OK Cancel					

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.



0

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

Table 3.2: Buttons of cross-section graphic



0



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

Cross-Section Property	Symbol	Value	Unit	-	HE A 400 Euronorm 53-62
)epth	d	390.0	mm		
Vidth	b	300.0	mm		
Veb thickness	tw	11.0	mm		
lange thickness	tf	19.0	mm		1 1 300.0
oot fillet radius	r	27.0	mm	=	• • • • • • • • • • • • • • • • • • •
ross-sectional area	Α	159.00	cm ²		
hear area	Ay	95.00	cm ²		27.0
hear area	Az	38.63	cm ²		
hear area according to EC 3	A _{V,Y}	118.18	cm ²		
hear area according to EC 3	A _{v,z}	57.35	cm ²		
/eb area	Aweb	38.70	cm ²		06
lastic shear area	Apl,y	114.00	cm ²		m y
lastic shear area	A pl,z	40.81	cm ²		
loment of inertia	Iy	45070.00	cm ⁴		11.0
loment of inertia	Iz	8560.00	cm ⁴		
overning radius of gyration	ry	168.0	mm		
overning radius of gyration	rz	73.4	mm		
olar radius of gyration	ro	183.3	mm		÷
adius of gyration of flange plus 1/5 of we	rzg	79.4	mm		z
olume	V	15900.00	cm ³ /m		
/eight	wt	124.8	kg/m		
urface	Asurf	1.910	m²/m		
ection factor	Am/V	120.126	1/m		
orsional constant	J	190.00	cm ⁴		井 🖂 Stress points
Varping constant	Cw	2.942E+06	cm ⁶		CT 23 C/t-Parts
	e.	2210.00	3	+	

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 Poi	ints of HE A 4	00 Euronorr	n 53-62					×
	A	В	С	D	E	F	G	HE A 400
StressP	Coordi	nates	Statical Mom	ents of Area	Thickness	Wan	ping	
No.	y [mm]	z [mm]	Qy [cm ³]	Q _z [cm ³]	t [mm]	W _{no} [cm ²]	Qw [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	1 2 3 4 5
5	150.0	-195.0	0.00	0.00	19.0	-278.25	0.00	ter and the second s
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	13 y
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	6 7 <mark>8 9 10</mark>
								÷
								2
	-							Close
	8							Close

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.

		A		В	C		D
lember		Damage	Equiv. Facto	r	Part of Sets of	of Members	
No.	Direct	Stresses	She	ar Stresses	No). I	Comment
66		1.00	0		1.000 -		
67		1.00	0		1.000 -		
68		1.00	0		1.000 -		
69		1.00	0		1.000 -		
71		1.00	0		1.000 2		
72		1.00	0		1.000 2		
73		1.00	0		1.000 2		
74		1.00	0		1.000 2		
91		1.00	0		1.000 -		
92		1.00	0		1.000 -		
tinas -	Member No.	66					
	A	B	C	D	E	F	IPE 400
ress	Coordinat	tes [mm]	Thickness		Detail Catego	iry	
nt No.	у	z	t [mm]	Existing	$\Delta \sigma c [N/mm^2]$	ATC IN/m	1 2 2 4 5
1	-90.0	-200.0	13.5	V	80	80	
1 2	-90.0 -25.3	-200.0 -200.0	13.5 13.5	√ √	80 80	80	turi fina
1 2 3	-90.0 -25.3 0.0	-200.0 -200.0 -200.0	13.5 13.5 13.5	7 7 7	80 80 80 80	80 80 80	
1 2 3 4	-90.0 -25.3 0.0 25.3	-200.0 -200.0 -200.0 -200.0	13.5 13.5 13.5 13.5 13.5	9 9 9	80 80 80 80 80	80 80 80 80 80	
1 2 3 4 5	-90.0 -25.3 0.0 25.3 90.0	-200.0 -200.0 -200.0 -200.0 -200.0	13.5 13.5 13.5 13.5 13.5 13.5	9 9 9 9	80 80 80 80 80 80 80	80 80 80 80 80 80	
1 2 3 4 5 6	-90.0 -25.3 0.0 25.3 90.0 -90.0	-200.0 -200.0 -200.0 -200.0 -200.0 200.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5	7 7 7 7	80 80 80 80 80 80 80 160	80 80 80 80 80 80 100	
1 2 3 4 5 6 7	-90.0 -25.3 0.0 25.3 90.0 -90.0 -25.3	-200.0 -200.0 -200.0 -200.0 -200.0 200.0 200.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	9 9 9 9 0	80 80 80 80 80 160 160	80 80 80 80 80 80 100 100	
1 2 3 4 5 6 7 8	-90.0 -25.3 0.0 25.3 90.0 -90.0 -25.3 0.0	-200.0 -200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	y y y y	80 80 80 80 80 160 160 160 160	80 80 80 80 80 100 100 100	
1 2 3 4 5 6 7 8 9	-90.0 -25.3 0.0 25.3 90.0 -90.0 -25.3 0.0 25.3	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	> > > > > - - - - - - - - - -	80 80 80 80 160 160 160 160 160	80 80 80 80 80 80 100 100 100 100	
1 2 3 4 5 6 7 8 9 9 10	-90.0 -25.3 0.0 25.3 90.0 -90.0 -25.3 0.0 25.3 90.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	> > > > - - - - - - - - - -	80 80 80 80 160 160 160 160 160	80 80 80 80 80 80 100 100 100 100 100	13
1 2 3 4 5 6 7 8 9 10 11	-90.0 -25.3 0.0 25.3 90.0 -90.0 -25.3 0.0 25.3 90.0 0.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 -165.5	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	> > > > -	80 80 80 80 160 160 160 160 160 160 160	80 80 80 80 80 80 100 100 100 100 100 10	
1 2 3 4 5 6 7 8 9 10 11 12	-90.0 -25.3 90.0 -90.0 -25.3 0.0 25.3 90.0 25.3 90.0 0.0 0.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 165.5 165.5	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	S S <t< td=""><td>80 80 80 80 80 160 160 160 160 160 160 160</td><td>80 80 80 80 80 100 100 100 100 100 100 1</td><td></td></t<>	80 80 80 80 80 160 160 160 160 160 160 160	80 80 80 80 80 100 100 100 100 100 100 1	
1 2 3 4 5 6 7 8 9 10 11 12 13	-90.0 -25.3 90.0 -90.0 -25.3 0.0 25.3 90.0 25.3 90.0 0.0 0.0 0.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 165.5 165.5 0.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	> > > > -	80 80 80 80 160 160 160 160 160 160 160 16	80 80 80 80 80 100 100 100 100 100 100 1	Tis
1 2 3 4 5 6 7 8 9 10 11 12 13	-90.0 -25.3 90.0 -90.0 -25.3 0.0 25.3 90.0 25.3 90.0 0.0 0.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 165.5 165.5	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	9 9 9 	80 80 ▼ 80 ▼ 80 80 160 160 160 160 160 160 160 16	80 80 80 80 80 100 100 100 100 100 100 1	113 113 113 113 113 113 113 113
1 2 3 4 5 6 7 8 9 10 11 12 13	-90.0 -25.3 0.0 25.3 90.0 -25.3 0.0 -25.3 90.0 0.0 0.0 0.0 0.0 0.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 -165.5 165.5 0.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	9 9 9 	80 80 80 80 80 160 160 160 160 160 160 160 16	80 80 80 80 80 80 100 100 100 100 100 10	13
1 2 3 4 5 6 7 8 9 10 11 12 13	-90.0 -25.3 0.0 25.3 90.0 -90.0 -25.3 90.0 0.0 25.3 90.0 0.0 0.0 0.0	-200.0 -200.0 -200.0 -200.0 200.0 200.0 200.0 200.0 200.0 200.0 200.0 -165.5 165.5 0.0	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5	9 9 9 	80 80 80 80 80 160 160 160 160 160 160 160 16	80 80 80 80 100 100 100 100 100 100 100	
1 2 3 4 5 6 7 8 9 10 11 12 13	-90.0 -25.3 90.0 -90.0 -90.0 -25.3 90.0 25.3 90.0 0.0 0.0 0.0 0.0 0.0 0.0 0.	-200.0 -200.0 -200.0 -200.0 -200.0 200.	13.5 13.5 13.5 13.5 13.5 13.5 13.5 13.5		80 80 80 80 80 160 160 160 160 160 160 160 16	80 80 80 100 100 100 100 100 100 100	y

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.

Detail C	ategory
Δσς [N/m	ım∠]
80	<u> </u>
160	
140	
125	
112	=
100	-
90	
71	
63	
56	-

8

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 <u>subsequent</u> 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

Calculation

Details...

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

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

- Ultimate Limit State
- General

4.1.1 Ultimate Limit State

Details - EN 1993-1-9	×
Ultimate Limit State General	
Assessment Method	
 Damage tolerant 	
Safe life	
Consequence of failure: Low Consequence	-
Partial factor for fatigue strength γ _{Mf} : 1.000	
	OK Cancel

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 $\gamma_{\rm Mf}$ for fatigue strength directly.

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

4.1.2 General

Details - EN 1993-1-9	x
Ultimate Limit State General	
Calculation of Result Combinations with OR Type	Display Result Windows
 Analyze each load combination in result combination separately (precise solution, may be slower) Analyze result combination comprehensively (conservative solution, may be faster) 	 ✓ 2.1 Design by Cross-Section ✓ 2.2 Design by Set of Members ✓ 2.3 Design by Member ✓ 2.4 Design by x-Location
Cross-Section Optimization	
Max allowable design ratio: 1.000	 ✓ 4.1 Parts List by Member ✓ 4.2 Parts List by Set of Members Only for members / sets to be designed
	Of all members / sets of members
	OK Cancel

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

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** \rightarrow **To Calculate**) lists design cases of the add-on modules like load cases and load combinations.

o Calculate							×
Load Cases / (Combinations / Module Cases Result Tables						
Not Calculated	1			Selected for	Calculation		
No.	Description	*		No.	Descripti	on	-
S Ch CO75	LC1 + LC3 + 0.6*LC4			CA1	RF-STEEL Fatigue Members - Fatig	jue design	
S Ch CO76	LC1 + LC3 + 0.6*LC4 + LC6						
S Ch CO77	LC1 + LC3 + 0.6*LC5						
S Ch CO78	LC1 + LC3 + 0.6*LC5 + LC7						
S Ch CO79	LC1 + LC4						
S Ch CO80	LC1 + LC4 + LC6						
S Ch CO81	LC1 + LC5						
S Ch CO82	LC1 + LC5 + LC7						
S Ch CO83	LC1 + 0.5*LC2 + LC4		>>				
S Ch CO84	LC1 + 0.5*LC2 + LC4 + LC6						
S Ch CO85	LC1 + 0.5*LC2 + LC5						
S Ch CO86	LC1 + 0.5*LC2 + LC5 + LC7		4				=
S Ch CO87	LC1 + 0.5*LC2 + 0.7*LC3 + LC4						
S Ch CO88	LC1 + 0.5*LC2 + 0.7*LC3 + LC4 + LC6						
S Ch CO89	LC1 + 0.5*LC2 + 0.7*LC3 + LC5						
S Ch CO90	LC1 + 0.5*LC2 + 0.7*LC3 + LC5 + LC7						
S Ch CO91	LC1 + 0.7*LC3 + LC4						
S Ch CO92	LC1 + 0.7*LC3 + LC4 + LC6	_					
S Ch CO93	LC1 + 0.7*LC3 + LC5						
S Ch CO94	LC1 + 0.7*LC3 + LC5 + LC7						
STR RC1	GZT (STR/GEO) - Permanent / transient - Eq. 6.10	=					
S Ch RC2	SLS - Characteristic						
CA1	RF-STEEL Members - General stress analysis of steel members	+					+
Al	- -	Q)					
2 B	6					OK Ca	ancel

Figure 4.3: Dialog box To Calculate



٩

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].

<u>T</u> ools Ta <u>b</u> l	Options Add-on Modules Window Help
⁰₃ RF-STEE	atigue Membe 🝸 \land 👂 🖉 🎬 🤗 🚧 🕼 🕼 📾 🛤 🗄 🗱 🥵 🤹 🏦 🎾 🚷 💠 🛝
🗇 😭 - I	🕽 - 🍟 🏨 🏨 👹 - 🗄 🕱 🖓 📶 🚮 🕅 Show Results 🕏 🛱 🛱 🛱 🛣 - 💆 - 1 🧭 - 1 🎢 🖘 1 🚑

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.

RF-STEEL Fatigue Members - [Ha	11]														×
File Edit Settings Help															
CA1 - Fatigue design 👻	2.1 Design	n by Cros	s-Section												
Input Data		A	В	C	D	E					F				
···· General Data	Section	Member	Location	S-Point	Design										
Materials	No.	No.	x [m]	No.	Ratio				De	esign Acc	ording to Formula				
Cross-Sections	9	IPE 400	Euronom 19	-57											
Regulte		68	3.125	1	0.20	≤1	101) Direct stress ra	nge design ac	c. to 8(1)						
Design by Cross-Section		69	6.250	13	0.06	≤1	102) Shear stress ra	nge design ac	c. to 8(1)						
Design by Set of Members		66	3.125	1	0.99	51	103) Nominal direct	stress range o	esign acc	. to 8(2)					
- Design by Member		69	6.250	13	0.14	21	104) Ivominal shear	stress range d	esign acc	. to 8(2)					
Design by x-Location		66	3.120		0.97	21	Tub) Combined stres	ss ranges desi	gri acc. to	0(3)					
Parts List by Member	12	000 00	ALDIN 5941	0.1974											_
Parts List by Set of Members	14	93	0.000	9	0.22	<1	101) Direct stress ra	nge design ag	c to 8(1)						
		93	0.000	12	0.02	<1	102) Shear stress ra	nge design ac	c to 8(1)						
		1 35	0.000	12	0.00			ngo acoign ac	0.10 0(1)		(F)				
				Max:	0.99	≤ 1	9					> 1,0	• Y	24	ৰ 🔍
	Details -	Member 6	8 - v: 3 125 n	n - RC1									400 LEuror	orm 19.57	
	I Mater	ial Properti	ies - Steel S 2	35 I EN 1993-1	-1-2005-05							0 - IP L	400 20101	Ionn 13-07	
	FI Cross	-Section Pr	roperties - IP	F 400 Eurono	m 19-57										
	⊟ Stress	Range Va	alues in Stres	s Point No. 1											
	Ma	ximum Non	mal Stress				σmax	-30.11	N/mm ²	RC2					
	- Min	nimum Nom	nal Stress				σmin	-99.87	N/mm ²	RC2			18	0.0	
	— Ma	ximum She	ear Stress				τ _{max}	1.60	N/mm ²	RC2			+	-	
	Min	nimum Shea	ar Stress				τmin	0.19	N/mm ²	RC2			3.5	21.0	
	🖃 Desig	n Ratio												<u> </u>	
	- Dire	ect Stress	Range				Δσ	69.76	N/mm ²						
	- Yie	ld Strength	1 I				fy	235.00	N/mm ²		EN 1993-1-1,		8		
	- De	sign Ratio					η	0.20		≤1	Eq. (8.1)		~		¥
														8.6	
														l I	
														Course .	
														÷	
														z	
										_					
															[mm]
														X	P) A
	Calculat	tion	Details	ר			Graphics	7					O		Cancel
								_							

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.

5

OK

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.



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.

L Design	n by Cross	s-Section													
	A	B	С	D	E					F					
Section	Member	Location	S-Point	Design											
No.	No.	x [m]	No.	Ratio				De	esign Acc	ording to Formula	l.				
9	IPE 400	Euronorm 19)-57												
	68	3.125	1	0.20	≤1	101) Direct stress ra	inge design ad	c. to 8(1)							
	69	6.250	13	0.06	≤1	102) Shear stress ra	inge design ad	c. to 8(1)							
	66	3.125	1	0.99	≤1	103) Nominal direct	03) Nominal direct stress range design acc. to 8(2)								
	69	6.250	13	0.14	≤1	104) Nominal shear	04) Nominal shear stress range design acc. to 8(2)								
	66	3.125	1	0.97	51	105) Combined stree	ss ranges desi	gn acc. to	8(3)						
12	QBO 80v	4 I DIN 5941	0.1974												
	93	0 000	9	0.22	≤1	101) Direct stress ra	inge design ag	c. to 8(1)							
	93	0.000	12	0.09	≤1	102) Shear stress ra	inge design ad	c. to 8(1)							
			Max:	0.99	≤1	•				F	> 1,0	- 7 😂 🖪 🗞 👁			
E Materia Cross- Stress	al Propertie Section Pr Range Va	es - Steel S 2 operties - IP alues in Stress	235 EN 1993- 9E 400 Euron s Point No. 1	-1-1:2005-05 10rm 19-57											
Max	dimum Norr	mal Stress				σmax	-31.58	N/mm ²	RC2			100.0			
— Mini	imum Norm	nal Stress				σmin	-100.42	N/mm ²	RC2			+ 180.0			
Max	amum She	ar Stress				τ _{max}	-0.38	N/mm ²	RC2						
Mini	imum Shea	ar Stress				τmin	-4.07	N/mm ²	RC2			₩ <u>21.0</u>			
] Design	n Ratio														
- Dire	ct Stress I	Kange				Δσ	68.84	N/mm ²							
- Dan	nage Equiv	of Naminal D	S ins at Chasas D			λ _σ	1.000	NI /mm 2		Ea. (C.1)	400	уу			
Bef	erence Va	lue of Estique	a Strength	ange		7FT 20E,2	90.04	N/mm ²		Eq. (0.1)		8.6			
- Part	tial Factor f	for Eatique St	trenath			ZOC	1 150	14/1111-		Tab 3.1					
Des	ion Ratio	ion naligate of	longan			n .	0.99		≤1	Eq. (8.2)					
	-														
												z			
												[mm			
											0	🎽 🚰 ⊄			

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

5 Results

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

.2 Desigr	2 Design by Set of Members														
	A	В	С	D	F					F					
Set	Member	Location	S-Point	Design	_										
No.	No.	x [m]	No.	Ratio				De	sign Acc	ording to Formula					
1	Ceiling be	am B-B (Men	nber No. 66-6	9)											
	68	3.125	1	0.20	≤1	101) Direct stress ra	nge design ac	c. to 8(1)							
	69	6.250	13	0.06	≤1	102) Shear stress ra	nge design ac	c. to 8(1)				E			
	66	3.125	1	0.99	≤1	103) Nominal direct	stress range d	lesign acc.	to 8(2)						
	69	6.250	13	0.14	≤1	104) Nominal shear) Nominal shear stress range design acc. to 8(2)								
	66	3.125	1	0.97	≤1	105) Combined stres	s ranges desi	gn acc. to	8(3)						
2 Ceiling beam A-A (Member No. 71-74)															
	72	3.125	6	0.14	≤1	101) Direct stress ra	nge design ac	c. to 8(1)							
	71	0.000	13	0.06	≤1	102) Shear stress ra	nge design ac	cc. to 8(1)				T			
	Max: 0.99 ≤ 1 🔮														
Details - Member 66 - x: 3.125 m - RC1 9 - IPE 400 Euronorm 19-57															
🕀 Materi	ial Properti	es - Steel S 2	35 EN 1993-	1-1:2005-05											
Cross-	Section Pr	operties - IP	E 400 Euron	om 19-57											
Stress	Range Va	lues in Stress	s Point No. 1												
— Max	ximum Non	mal Stress				σmax	-31.58	N/mm ²	RC2			180.0			
Min	imum Nom	nal Stress				σmin	-100.42	N/mm ²	RC2			+			
- Max	ximum She	ar Stress				τ _{max}	-0.38	N/mm ²	RC2		+	- total and the second			
Min	imum Shea	ar Stress				τmin	-4.07	N/mm ²	RC2			21.0			
Desig	n Ratio														
Dire	ect Stress I	Range				Δσ	68.84	N/mm ²			q				
Dar	mage Equir	valent Factor	s			λσ	1.000			5 (0.4)	400	·•			
Des	sign Value	of Nominal D	rect Stress R	ange		γFf ΔσE,2	68.84	N/mm ²		Eq. (6.1)		2.6			
Ret	erence Va	lue of Fatigue	e Strength			Δσς	80.00	N/mm ²				<u></u>			
Dar	nage Equi iaa Valua	valent Factor	S Lana Chana Di			Λ _τ	1.000	N /mm 2		E- (C 1)					
Des	sign value	of Nominal Si	near Stress Hi	ange		γFf ΔτE,2	3.63	N/mm ²		Eq. (6.1)	+				
- Nei Dad	tial Eastern	for Entirue Ct	e Strengtri			Arc	1 150	N/mm~	-	T=b 2.1		÷			
Cor	nan Factori	or raligue ol "	rengin			7Mt	0.97			1dD. 5.1		2			
Cor	mponent A	т				1140	0.00								
Des	sion Ratio					η <u>Δτ</u>	0.00		< 1	Fa (8.3)					
- 000	agir riduo					1	0.37			Eq. (0.0)					
<u> </u>												[mm]			
											0	🎽 📬 🍝			

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).



0

	Δ	B	C						F					
Member	Location	S-Point	Design	0										
No.	x [m]	No.	Ratio					Design	According	to Formula				
66	Cross-section	n No. 9 - IPE	400 Euronom 1	9-57				a see gri						
	3 125	1	0.20	<1	101) Direct str	ess range desig	n acc. to 8(1)							
_	4 063	3	0.02	<1	102) Shear str	ess range desig	n acc. to 8(1)							
	3 125	1	0.99	<1	103) Nominal of	direct stress ran	ne design acc	to 8(2)						
	4 063	3	0.06	≤1	104) Nominal s	shear stress ran	ne design acc	to 8(2)						
	3 125	1	0.97	≤1	105) Combined	mbined stress ranges design acc. to 8(3)								
				-										
67	Cross-sectio	n No. 9 - IPE	400 Euronorm 1	9-57										
	3.125	1	0.20	≤1	101) Direct str	ess range desig	n acc. to 8(1)							
	0.000	13	0.06	≤1	102) Shear str	ess range desig	n acc. to 8(1)							
		Max:	0.99	≤ 1	•						> 1,0	• 7 😂 🛃 🗞 👁		
etails - I	Member 66 -	x: 4.063 m - F	RC1								9 - IPE 40	0 Euronorm 19-57		
E Materi	al Properties	- Steel S 235	EN 1993-1-1:200)5-05										
+ Cross-	Section Prop	erties - IPE 4	400 Euronom 19-	-57										
∃ Stress	Range Value	es in Stress Po	pint No. 3											
— Max	kimum Normal	Stress				σ _{max}	-26.76	N/mm ²	RC2			190.0		
— Mini	imum Normal	Stress				σmin	-90.56	N/mm ²	RC2			100.0		
— Max	imum Shear	Stress				τ _{max}	-1.45	N/mm ²	RC2		+	- Contraction of the Contraction		
— Mini	imum Shear S	Stress				τmin	-5.36	N/mm ²	RC2			21.0		
Desigr	n Ratio													
She	ar Stress Rar	nge				Δτ	3.91	N/mm ²						
- Dan	nage Equival	ent Factors				λτ	1.000				100.			
- Des	ign Value of	Nominal Shea	ar Stress Range			γFf Δτε,2	3.91	N/mm ²		Eq. (6.1)	4	Y		
- Ref	erence Value	of Fatigue St	trength			Δτς	80.00	N/mm ²				8.6		
- Part	tial Factor for	Fatigue Stren	igth			γMf	1.150			Tab. 3.1				
Des	ign Ratio					η	0.06		≤1	Eq. (8.2)	+			
												z		
												Imm		
											0	🚔 🔚 🕰		

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

2.4 Desigr	by x-Locati	on										
	A	В	C	D	(F			
Member	Location	S-Point	Design									
No.	x [m]	No.	Ratio					Design	According	to Formula		
91	Cross-section	n No. 12 - QF	RO 80x4 EN 102	10-2:	2006							
	0.000	2	0.11	≤1	101) Direct stre	ess range desig	n acc. to 8(1)					
	0.000	4	0.01	≤1	102) Shear stre	ess range desig	n acc. to 8(1)					
	0.000	2	0.57	≤1	103) Nominal o	direct stress ran	ge design acc	. to 8(2)				
	0.000	4	0.03	≤1	104) Nominal s	shear stress ran	ge design acc	. to 8(2)				
	0.000	2	0.19	≤1	105) Combined	d stress ranges	design acc. to	8(3)				
	0.250	2	0.10	≤1	101) Direct stre	ess range desig	n acc. to 8(1)					
	0.250	4	0.01	≤1	102) Shear stre	ess range desig	n acc. to 8(1)					
	0.250	2	0.52	≤1	103) Nominal o	lominal direct stress range design acc. to 8(2)						
	0.250	4	0.03	≤1	104) Nominal s	shear stress ran	ge design acc	. to 8(2)				
	Max: 0.99 ≤ 1 ♥											
Details - I	Details - Member 91 - x: 0.000 m - RC1 12 - QRO 80x4 EN 10210-2:2005											
🕀 Materi	al Properties -	Steel S 235	EN 1993-1-1:200	5-05								
- Moo	dulus of Elasti	city				E	210000.00	N/mm ²				
- She	ar Modulus					G	80769.20	N/mm ²				
🖃 Thio	kness range	t ≤ 40 mm										
- Y	ield Strength					fy	235.00	N/mm ²				t 80.0
L	ltimate Tensil	e Strength				fu	360.00	N/mm ²			I +−	
- 🖂 Thio	ckness range	t > 40 mm an	d t ≤ 80 mm									8
- Y	ield Strength					fy	215.00	N/mm ²				4.0
	ltimate Tensil	e Strength				fu	360.00	N/mm ²				
Cross-	Section Prope	rties - QRO	80x4 EN 10210-	2:20	D6						80.0	
⊟ Stress	Range Value	s in Stress Po	bint No. 2									
Max	imum Normal	Stress				σmax	37.90	N/mm ²	RC2			
Mini	mum Normal	Stress				σmin	-1.99	N/mm ²	RC2			S
Max	timum Shear S	otress				τmax	0.49	N/mm ²	RC2		-	
Mini	mum Shear S	tress				τmin	-1.68	N/mm ²	RC2			*
E Design												
Dire	ct stress Han	ige int Eastern				20	39.88	N/mm ²				
Dan	iage Equivale	ant Pactors	t Strace Dance			Ag	1.000	NI/mm2	-	Eq. (6.1)		
Des Rof	arence Value	of Estique St	reports			7FT 40E,2	20.00	N/mm2		Eq. (0.1)		
Part	ial Factor for I	Fatique Stren	ath			200	1 150	10/1011-		Tab 3.1		[mm]
	ion Batio	augue Stren	gui			7 MI	0.57		<1	Fg. (8.2)	a	
Les	ign natio					1	0.57		2.1	Lq. (0.2)		

Figure 5.5: Window 2.4 Design by x-Location

This result window lists the maximum stresses for each member at all location \mathbf{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.

.1 Parts	List by Member								
	A	B	C	D	F	F	G	н	
Part	Cross-Section	Number of	Length	Total Length	Surface Area	Volume	Unit Weight	Weight	Total Weight
No.	Description	Members	[m]	[m]	[m ²]	[m ³]	[kg/m]	[kg]	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
									N 3 3

Figure 5.6: Window 4.1 Parts List by Member

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.

Details...

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

0

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).



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

rt).	Set of Members Description	Number	Length	T					
). ! 	Description		Lengui	Total Length	Surface Area	Volume	Unit Weight	Weight	Total Weigh
m		of Sets	[m]	[m]	[m ²]	[m ³]	[kg/m]	[kg]	[t]
m	Ceiling beam B-B	1	25.00	25.00	36.68	0.21	66.33	1658.31	1.6
m	Ceiling beam A-A	1	25.00	25.00	36.68	0.21	66.33	1658.31	1.0
		2		50.00	73.35	0.42			3.

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.

2.1 Desigr	h by Cross	s-Section										
	A	B	C	D	F						F	
Section	Member	Location	S-Point	Design								
No.	No.	x [m]	No.	Ratio					0	esign Ac	cording to Form	nula
9	IPE 400	Euronorm 19	-57									
	68	3.125	1	0.21	≤1	101) Begre	nzung der Längssp	annı	ungsschwi	ngbreiten	nach 8(1)	
	69	6.250	13	0.06	≤1	102) Begre	nzung der Schubsp	ann	ungsschwi	ingbreiten	nach 8(1)	
	66	3.125	1	1.03	>1	103) Nach	weis für Nennlängss	pan	nungssch	wingbreite	en nach 8(2)	
	69	6.250	13	0.14	≤1	104) Nach	weis für Nennschub	spa	nnungssch	nwingbreit	en nach 8(2)	
	66	3.125	1	1.11	>1	105) Nach	weis für kombinierte	n Sp	bannungss	chwingbr	eiten nach 8(3))
			Max:	1.11	>1	8						
Details -	Member 66	6 - x: 3.125 n	1 - RC1									9 - IPE 400 Euronorm 19-57
	al Properti	es - Steel S 2	35 EN 1993-	1-1:2005-05								
E Cross-	Section Pr	operties - IP	E 400 Euron	orm 19-57								
Stress	Range Va	lues in Stress	s Point No. 1									
— Max	dimum Nor	mal Stress				σmax	-31.	58	N/mm ²	RC2		180.0
- Min	imum Nom	nal Stress				σmin	-103.	55	N/mm ²	RC2		100.0
Max	amum She	ar Stress				τ _{max}	-0.	38	N/mm ²	RC2		
- Min	imum Shea	ar Stress				τmin	-4.	15	N/mm ²	RC2		21.0
Design	n Ratio											
Dire	ct Stress I	Range				Δσ	71.	96	N/mm ²			
- Dar	nage Equi	valent Factor	S			λσ	1.0	00				
Des	ign Value	of Nominal D	irect Stress Ra	ange		γFf Δ	σE,2 71.	96	N/mm ²		Eq. (6.1)	
Ref	erence Va	lue of Fatigue	e Strength			Δσς	80.	00	N/mm ²			
Dan	nage Equir	valent Factor	s			λτ	1.0	00				
Des	ign Value	of Nominal Si	hear Stress Ra	ange		γFf Δ	.τE,2 3.	//	N/mm ²		Eq. (6.1)	
Ref	erence Va	lue of Fatigue	Strength			Δτς	80.	00	N/mm ²			_
Part	tial Factor	for Fatigue St	rength			7Mf	1.1	50			Tab. 3.1	Z
Con	nponent ∆	5				ηΔσ	1.	11				_
- Con	nponent ∆	τ				ηΔτ	0.	00			E (0.0)	_
- Des	ign Ratio					η	1.	11		>1	Eq. (8.3)	_
								_		-		[mm]
												ð 📑 🕰

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	Filter Parameter	Describe the criterion for filtering results in tables: ratios greater than 1, maximum or user-defined limit
7	Apply Filter	Display only the rows with the filter parameter applied (design ratio > 1, maximum, user-defined limit)
2	Result Diagrams	Open the Result Diagram on Member window \rightarrow Chapter 6.2, page 31
	Excel Export	Export the table to MS Excel / OpenOffice \rightarrow Chapter 8.4.2, page 42
₹₹	Member Selection	Select a member graphically to display its results in the table
۲	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.

6

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.

6



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



Graphics

X_XX

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.

RF-STEEL Fatigue Members CA1 - Fatig	< > >
LC1 - Self-weight LC2 - Imposed Load 1 LC3 - Imposed Load 2 RC1 - 1.35*LC1 + 1.5*LC2 or 1.5*LC3	13
RF-STEEL Fatigue Members CA1 - Fatigue	design
RF-STEEL Fatique Members CA2 - Fatique	design

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** \rightarrow **Members** in the *Display navigator*. The display of the stresses and design ratios is *Two-Colored* by default.

6 Results Evaluation



Figure 6.3: Display navigator: Results \rightarrow 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).

RF-STEEL Fatigue Members

2

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

6

Ł

by clicking the button shown on the left. The button is located below the upper result table (see Figure 6.1, page 29).

6

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

Results ightarrow 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

Graphics

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 
ightarrow Control Panel (Color Scale 
ightarrow Factors 
ightarrow 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.

6



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/mm² 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/mm², for example.

The Display Hidden Result Diagram function in the Display navigator (**Results** \rightarrow **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.



6 Results Evaluation



6

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.



7.1 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 ightarrow Print Graphic

or use the toolbar button shown on the left.

ø	RFEM	5.03.20	72 (64bi	t) - [Hall]				
:4	<u>F</u> ile	e <u>E</u> dit	<u>V</u> iew	Insert	<u>C</u> alculate	<u>R</u> esul	ts <u>T</u> ools	Ta <u>b</u> le	<u>O</u> ptions
] 🖻	33		۵ 🖌 🖌	500	13 🤆	g 😳 🚰	-	🔲 🎱
1	5	/ 🎢	- 🦈 -	Print	t Graphic	환 📍	- 🗖 -	沟 🕩	- 🗊 😭

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.

ĺ	Result Diagram on Member	er				
	Members No.: 69	- < > 🏌	s i 🙀 📴 I G	R 🗣 🎅 🗐 🗷		Ŧ
	RF-STEEL Fatigue	√lembe * \land >	Print	*		
I	Navigator $ $ \times	0.000	1.000 2.000	3.000 4	.000 5.000	6.
I		ىلىيىيان 🔰			ليتبيانينيانيتيانيينا	Ч
I	Design Ratio	T		»M69»		ין
I	Stress ranges	Design Ratio	[-]			

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

The following dialog box appears:

Graphic Printout	×
General Options Color Scale Factors Borde	er and Stretch Factors
Graphic Picture ○ Directly to a printer ● To a printout report: ● To the Clipboard ○ To 3D PDF	Window To Print Graphic Size © Current only As screen view Window filling Mass print To scale 1: 20 •
Graphic Picture Size and Rotation Image: Size and Rotation	Options Control of the selected x-location in result diagram Lock graphic picture (without update) Show printout report on [OK]
Header of Graphic Picture RF-STEEL Fatigue Members - Stress range Delt	a sigma, CA1, Isometric

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.

Remove from Printout Report Start with New Page Selection... Properties... 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.

Graphic Printout				×		
General Options Color Scale Fac	tors Border and Stretch	Factors				
Script	Symbols		Frame			
Proportional	Proportional		None			
 Constant 	Constant		Framed			
Factor: 1	Factor: 1		Title box			
Print Quality		Color				
Standard (max 1000 x 1000 Pixels))	⊚ Grayscale				
Maximum (max 5000 x 5000 Pixels)	Texts and lines in black				
O User-defined		All colored				
Max number of pixels:	1000 丈					
			OK	Cancel		

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 ightarrow New Case.

The following dialog box appears:

New RF-ST	TEEL Fatigue Members Case
No. 2	Description Fatigue design
D	OK Cancel

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

$\textbf{File} \rightarrow \textbf{Rename Case.}$

The following dialog box appears:

Rename R	F-STEEL Fatigue Members Case
No. 2	Description Vew Description
D	OK Cancel

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 \rightarrow Copy Case.

8 General Functions

The following dialog box appears:

Copy RF-S	Copy RF-STEEL Fatigue Members Case										
Copy from	Copy from Case										
CA1 - Fa	tigue design 🔹										
New Cas	e										
No.:	Description:										
3	Detail category 80 🗸										
	OK Cancel										

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

$\mathbf{File} \rightarrow \mathbf{Delete} \ \mathbf{Case.}$

The following dialog box appears:

C	elete Ca	ases
ſ	Availab	le Cases
	No.	Description
	1	Fatigue design
	2	New Description
	3	Detail category 80
		_
	٢	OK Cancel

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.

2.1 Desigr	n by Cros	s-Sect	ion											
	A	F												
Section	Member	Loca	ation	S-Point	Design			· · · · · · · · · · · · · · · · · · ·						
No.	No.	x [m]		No.	Ratio			Design According to Formula						
9	IPE 360	Euron	orm 1	19-57										
	69			- Go to Cross-	Section	De	whleelick	ress range design acc. to 8(1)						
	68				Section	00	UDIECIICK	ress range design acc. to 8(1)						
	69			Info About C	Cross-Section			direct stress range design acc. to 8(2)						
	68			Ontimize Cr	oss-Section			shear stress range design acc. to 8(2)						
	69			<u>o</u> pennee en	635 55661011	-		ed stress ranges design acc. to 8(3)						
				Cross-Sectio	n Optimization	<u>P</u> arai	meters							
12	QRO 80x	(4 EN	1021	210-2:2006										

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.

R

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 \rightarrow Export All Cross-Sections to RFEM.

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

1.3 Cross-	Sections								
	A	В		С	D	E	F	G	9 - IPE 400 Europorm 19-57
Section	Material	Cross-Sec	tion	Cross-Section	Max. Design	Opti-			RF-STEEL Fatigue Members
No.	No.	Descriptio	on	Туре	Ratio	mize	Remark	Comment	
1	1	HE A 400 Eu	ronorm 53-6	I-section rolled		No	5)		
2	2	T IS 360/170/8	/14/0	I-section welded IS		No	5)		У
3	2	IS 500/170/8	/14/0	I-section welded IS		No	5)		
6	1	HE A 160 Eu	ronorm 53-6	I-section rolled		No	5)		
7	1	HE A 120 Eu	ronorm 53-6	I-section rolled		No	5)		*
9	1	I PE 400 E	Info About	Cross-Section		No	1)		
10	1	T HE A 140				No	5)		9 - IPE 360 Euronorm 19-57
12	1	I IPE 180	Cross-Sect	ion Library		No	5)		RFEM
13	1	RD 24 N	Edit List 'D	esign of Members' in	Window 1.1	No	5)		
15	1	HE A 200	-			No	5)		»
16	3	Rectangle	Optimize C	ross-Section		No	5)		y y
17	1	I IPE 360 I	Cross-Sect	ion Optimization <u>P</u> ar	ameters	No	5)		_ _ _
			Export Cros	ss-Section to RFEM	La				U ▼ x
1) The citizen Export All Citizen Export			Cross-Sections to RFE	STEEL					
Import Cro				ss-Section from RFE	M				
Cross-Section Properties - IPE 40 Import All C				Cross-Sections from	RFEM				Cross-section No. 9 used in

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.

RF-STEEL Fatigue Members Information No. 53074									
Do you want to transfer the changed cross-sections to RFEM?									
If so, the results of RFEM and RF-STEEL Fatigue Members will be deleted.									
<u>Y</u> es <u>N</u> o									

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 \rightarrow 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.

Units and Decimal Places							×
Program / Module		RF-STEEL Fatigue Members					
DE STEEL Surfaces		Output Data			Parts List		
DE STEEL Mombom			Unit	Dec places		Unit	Dec places
		~		olal			
		Stresses:	N/mm [*] 2 -	2 🤤	Lengths:	m 🔻	2 🖵
DE STEEL AISC		Design ratios:		2 🚔	Total lengths:	m 🔻	2 🚔
		Dimensionless:		3 🚔	Surface areas:	m^2 ▼	2 🌩
					Volumes:	m^3 ▼	2 🚔
					Weight per length:	kg/m ▼	2 🤤
					Weight:	ka 💌	2 🚔
RESTEEL Plastic					Total weight:	t 🔻	3 🌐
RE-STEEL Fatigue Mer							
RESTEEL NBR							
RE-ALLIMINIUM							
REKAPPA							
BE-I TB							
BE-FE-I TB							
BE-EL-PL							
BE-C-TO-T							
PLATE-BUCKLING							
BE-CONCRETE Surfac							
BE-CONCRETE Memb							
RF-TIMBER AWC	7						
	6					ОК	Cancel

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 \rightarrow 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



8 General Functions

Calculation

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

```
\textbf{File} \rightarrow \textbf{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

$\textbf{File} \rightarrow \textbf{Export Tables}.$

The following export dialog box appears:

Export of Tables	×
Table Parameters	Application
With table header	Microsoft Excel
Only marked rows	OpenOffice.org Calc
	CSV file format
Transfer Parameters	
Export table to active workbook	:
Export table to active workshee	t
Rewrite existing worksheet	
Selected Tables	
 Active table 	Export hidden columns
All tables	Export tables with details
✓ Input tables	
Result tables	
2	OK Cancel

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.

8

		01 - 1-				Sheet1 - Microcoft Evcel	- 0 X				
		· (- · -				Sheet - Microsoft Excel					
F	ile	Home	Insert Pa	ge Layout Fo	rmula	as Data Review View Add-Ins Team					
	1 👗	Calibri	Ŧ	11 × A A	≡	= 📄 Text 🔹 🔢 Conditional Formatting 🐑 🔐 Ins	sert * 🏾 🖍 👗				
	<u> </u>	-			E	🚍 🚍 📴 🛛 🛒 * % 🔹 🧱 Format as Table * 🛛 👬 De	elete - 💽 - 📶 🖬 🖬				
Pas	te 🝼	BI	<u>u</u> -	• 🖄 • <u>A</u> •	*	🖅 🗞 🐝 🗒 Cell Styles 🛪 🏢 Fo	ormat * 2 * Filter * Select *				
Clip	board (oard 🕫 Font 🕫 Alignment 🕫 Number 🕫 Styles Cells Editing									
	В	3	- (**	<i>f</i> ∗ Cross-	sect	ion No. 9 - IPE 400 Euronorm 19-57	*				
	А	В	С	D	E	F	-				
1	Vembe	Location	S-Point	Design			_				
2	No.	x [m]	No.	Ratio		Design According to Formula					
3	66	Cross-sec	tion No. 9	IPE 400 Euro	norr	m 19-57					
4		3,125	1	0,20 5	1 1	101) Direct stress range design acc. to 8(1)					
5		4,063	3	0,02 ≤	1 1	102) Shear stress range design acc. to 8(1)					
6		3,125	1	0,99 ≤	11	103) Nominal direct stress range design acc. to 8(2)					
7		4,063	3	0,06 ≤	1 1	104) Nominal shear stress range design acc. to 8(2)					
8		3,125	1	0,97 ≤	1 1	105) Combined stress ranges design acc. to 8(3)					
9							=				
10	67	Cross-sec	tion No. 9	- IPE 400 Euro	norr	m 19-57					
11		3,125	1	0,20 5	1 1	101) Direct stress range design acc. to 8(1)					
12		0,000	13	0,06 ≤	11	102) Shear stress range design acc. to 8(1)					
13		3,125	1	0,50 ≤	11	103) Nominal direct stress range design acc. to 8(2)					
14		0,000	13	0,14 ≤	11	104) Nominal shear stress range design acc. to 8(2)					
15		3,125	1	0,13 5	11	105) Combined stress ranges design acc. to 8(3)					
16											
17	68	Cross-sec	tion No. 9	IPE 400 Euro	norr	m 19-57					
18		3,125	1	0,20 ≤	1	101) Direct stress range design acc. to 8(1)					
19		6,250	13	0,06 ≤	11	102) Shear stress range design acc. to 8(1)					
20		3,125	1	0,50 ≤	11	103) Nominal direct stress range design acc. to 8(2)					
21		6,250	13	0,14 5	11	104) Nominal shear stress range design acc. to 8(2)					
22		3,125	1	0,13 5	1	105) Combined stress ranges design acc. to 8(3)					
23							×				
14 4	• •	2.3 Desigr	n by Membe	er / 🔁 🖊							
Rea	dy					III II	00% 🕘 🗸 🕂 🕂				

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

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)



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_v 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)





9

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

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



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

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

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 $\sigma_{\rm C} = 45 \text{ N/mm}^2$ for $2 \cdot 10^6$ cycles and the partial safety factor $\gamma_{\rm Mf} = 1.15$, 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 $\sigma_{ch,ch} = 83 \text{ N/mm}^2$.

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

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.

9





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 %.

2.1 Design by Cross-Section



9

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

Limitation of stress ranges

$$\begin{split} \Delta \sigma &\leq 1.5 \cdot f_y \\ 82.93 &\leq 1.5 \cdot 235 \\ 82.93 &\leq 352.5 \end{split}$$

Check: $\frac{82.93}{352.5} = 0.24 \le 1.00$

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%.

g

2.1 Design	n by Cros	-Section													
	A	B	С	D	I E						F				
Section	Member	Location	S-Point	Design											
No.	No.	x [m]	No.	Ratio					[Design Ac	cording to Form	nula			
3	RO 177.8	x7.1 DIN E	N 10220 - Loi	wer chord											
	11	6.000	8	0.2	4 ≤1	1 101) Direct stress ra	nge design ac	c. to 8(1)						
	11	6.000	8	1.0	0 ≤ 1	1 103) Nominal direct	stress range d	esign acc.	to 8(2)					
		Max:	1	.00 ≤1 🥹								0.20	•	7 😂 (i
Details -	Member 11	- x: 6.000 n	1 - LC4									3-1	RO 177.8x	(7.1 DIN E	EN 10220
Materi	ial Properti	es - Steel S 2	35 DIN EN 1	10025-2:2004-1											
	Section Pr	operties - R	0 177.8x7.1	DIN EN 10220											
Stress	Range Va	lues in Stress	s Point No. 8												
Max	ximum Non	nal Stress					σmax	84.673	N/mm ²	LC4					
Min	imum Nom	nal Stress					σmin	1.744	N/mm ²	LC4		_		5	
Max	ximum She	ar Stress					τ _{max}	0.016	N/mm ²	LC4			<u>←</u>	of the second	
Min	imum Shea	ar Stress					τmin	0.016	N/mm ²	LC4		_	ALTER O		
Desig	n Ratio										1	_	1		
Dire	ect Stress I	Kange					Δσ	82.930	N/mm ²			1.8	1		
Dar	mage Equir	alent Factor	s				λσ	0.470			5 (0.4)	- 5			2 Y
Des	sign Value	of Nominal D	rect Stress H	lange			γFf ΔσE,2	38.977	N/mm ²		Eq. (6.1)	_			<u></u>
Ret	erence va	iue of Fatigue	e Strengtn				Δσς	45.000	N/mm ²		T-1-0.1	_			ALL DE LE DE
Par	tial Factor	or Fatigue St	rengtn				γMf	1.100			Tap. 3.1	•	↓	10000	.
Des	sign Katio						η	1.00		51	Eq. (8.2)	_		- ÷	
												-		z	
												-			
<u> </u>															
<u> </u>															
<u> </u>												- 1			[mn
												0		[🍝 🔭 ⊄

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

Direct stresses

$$\begin{split} &\gamma_{F\,f} \cdot \varDelta \sigma_{E\,,2} = \lambda \cdot \varDelta \sigma \\ &\gamma_{F\,f} \cdot \varDelta \sigma_{E\,,2} = 0.471 \cdot 82.93 \\ &\gamma_{F\,f} \cdot \varDelta \sigma_{E\,,2} = 38.977 \, \mathrm{N} \, / \mathrm{mm}^2 \end{split}$$

Check:

$$\begin{aligned} &\frac{\gamma_{F\,f}\cdot\,\Delta\sigma_{E\,,2}}{\Delta\sigma_C/\gamma_{M\,f}} \leq 1.00\\ &\frac{38.977}{45/1.15} \leq 1.00 \end{aligned}$$

 $1.00 \le 1.00$



- [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.

∠ Dlubal

Index

В

Background graphic	30
Bottom stress	. 9
Buttons	29

С

-
Calculation start
Clipboard
Color bars
Color scale
Comment 11
Consequence of failure
Control panel
Cross-section 14, 38
Cross-section graphic
Cross-section library 14
Cross-section optimization
Cross-section properties

D

Damage equivalent factor 6, 7, 9, 18
Damage tolerant 19
Decimal places 12, 41
Design
Design case
Design ratio
Detail category
Detail settings
Display navigator

Ε

Excel	42
Exit RF-STEEL EC3	10
Export	42
Exporting cross-section	39
Exporting material	41

F

Fatigue design	. 7,	24
Fatigue strength	. 6,	18
Favorite		38
Filter	29,	32
Filtering members		33

G

General data 10)
Graphic printout	5
Graphics)

н

Hidden result diagram 33
I
Installation
Internal forces

B

L

Length	7
Load case	1
Load combination	1
Location x	б

М

Material	12, 41
Material description	12
Material library	13
Material properties	12
Member	11, 25

Ν

Navigator10
Nominal stress
Normal stress range7

0

Open RF-STEEL Fatigue Members	4
OpenOffice	42
Optimization	15, 38, 39

Ρ

Panel 5, 31, 33
Parametrized cross-section 39
Part
Partial safety factor
Parts list
Print graphic
Printout report
Program start

R

Remark	15
Result combination	11
Result diagram	35
Result display	30
Result values	30
Result windows	22
Results evaluation	29
Results navigator	30

Dluba

RF-DYNAM1	1
RFEM graphic	5
RFEM work window	9

S

-
Safe life 19
Scrolling in windows
Set of members
Shear stress range7
Stainless steel
Standard
Stress cycle
Stress point
Stress range
Sum
Surface area

Т

Top stress	9
U Units	2, 41 . 41
V View mode	, 30 . 32 . 28
X x-Location	s, 26
W Weight Window	. 28 . 10

B