

Version July 2013

Add-on Module



Stress Analysis Cross-section Optimization

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 ENGINEERING SOFTWARE.

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

 Tel.:
 +49 (0) 9673 9203-0

 Fax:
 +49 (0) 9673 9203-51

 E-mail:
 info@dlubal.com

 Web:
 www.dlubal.com



Content

Content	Page		Content	Page
Introduction	4	5.	Results Evaluation	39
Add-on Module STEEL	4	5.1	Selection of Stresses	40
STEEL Team	5	5.2	Results on Cross-section	41
Using the Manual	6	5.3	Results in RSTAB Model	43
Open the Add-on Module STEEL	6	5.4	Result Diagrams	46
Input Data	8	5.5	Filter for Results	47
General Data	8	6.	Printout	49
Materials	10	6.1	Printout Report	49
Cross-Sections	15	6.2	Printing STEEL Graphics	49
Calculation	19	6.2.1	Results on Cross-Section	49
Detail Settings	19	6.2.2	Results in the RSTAB Model	50
Stresses and Ratio	22	6.2.3	Result Diagrams	52
Start Calculation	27	7.	General Functions	53
Results	28	7.1	Design Cases	53
Stresses by Cross-Section	29	7.2	Cross-Section Optimization	55
Stresses by Set of Members	31	7.3	Units and Decimal Places	57
Stresses by Member	32	7.4	Data Transfer	58
Stresses by x-Location	33	7.4.1	Material Export to RSTAB	58
Stresses at Every Stress Point	34	7.4.2	Export of Results	58
Governing Internal Forces by Member	35	Α	Literature	60
Governing Internal Forces by Set of Members	36	В	Index	61
Parts List by Member	37			
Parts List by Set of Members	38			
	IntroductionAdd-on Module STEELSTEEL TeamUsing the ManualOpen the Add-on Module STEELInput DataGeneral DataMaterialsCross-SectionsCalculationDetail SettingsStresses and RatioStart CalculationResultsStresses by Cross-SectionStresses by Set of MembersStresses by X-LocationStresses by X-LocationStresses at Every Stress PointGoverning Internal Forces by MemberGoverning Internal Forces by Set of MembersParts List by Member	Introduction4Add-on Module STEEL4STEEL Team5Using the Manual6Open the Add-on Module STEEL6Input Data8General Data8Materials10Cross-Sections15Calculation19Detail Settings19Stresses and Ratio22Start Calculation27Results28Stresses by Cross-Section29Stresses by Set of Members31Stresses by X-Location33Stresses at Every Stress Point34Governing Internal Forces by Member35Governing Internal Forces by Set of Members36Parts List by Member37	Introduction45.Add-on Module STEEL45.1STEEL Team55.2Using the Manual65.3Open the Add-on Module STEEL65.4Input Data85.5General Data86.Materials106.1Cross-Sections156.2Calculation196.2.1Detail Settings196.2.2Stresses and Ratio226.2.3Start Calculation277.Results287.1Stresses by Cross-Section297.2Stresses by Set of Members317.3Stresses by X-Location337.4.1Stresses at Every Stress Point347.4.2Governing Internal Forces by Set of Members368Parts List by Member37*	Introduction45.Results EvaluationAdd-on Module STEEL45.1Selection of StressesSTEEL Team55.2Results on Cross-sectionUsing the Manual65.3Result in RSTAB ModelOpen the Add-on Module STEEL65.4Result DiagramsInput Data85.5Filter for ResultsGeneral Data86.PrintoutMaterials106.1Printout ReportCross-Sections156.2Printiout ReportCross-Sections196.2.1Results in the RSTAB ModelDetail Settings196.2.2Result bi on Cross-SectionDetail Settings196.2.3Result DiagramsStart Calculation277.General FunctionsResults287.1Design CasesStresses by Cross-Section297.2Cross-Section OptimizationStresses by Set of Members317.3Units and Decimal PlacesStresses by X-Location337.4.1Material Export to RSTABStresses at Every Stress Point347.4.2Export of ResultsGoverning Internal Forces by Set of Members36BIndexParts List by Member3737Index



I. Introduction

1.1 Add-on Module STEEL

The RSTAB add-on module STEEL performs general stress designs by calculating normal, shear, and equivalent stresses and comparing them with the limit stresses. The add-on module provides a comprehensive library for cross-sections and materials with standard-specific limit stresses that can be adjusted and expanded. All cross-sections are provided with design relevant stress points. The results of these points can also be used for graphical evaluations.

As STEEL is integrated in the graphical user interface of the main program, you can use all input data of the model as well as the internal forces for the design. You can evaluate the stresses and stress ratios in the RSTAB work window graphically and include them in the global printout report.

The internal forces that are found to be governing for the design are indicated for each member. During the stress analysis, you can also determine the maximum stresses of sets of members. In addition, STEEL offers you an automatic cross-section optimization including an export option for modified cross-sections to RSTAB.

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

The following useful features facilitate the work with STEEL:

- Display of maximum stress ratios in the cross-section table, helping you to decide how to optimize the cross-section
- Connection between STEEL tables and RSTAB work window, thus selecting, for example, the current member of the table in the background graphic
- View mode to change the view in the RSTAB work window
- Colored relation scales in results windows
- Info icon for successful or failed stress design
- Representation of STEEL stresses and ratios in the form of result diagrams
- Filter options for the stresses displayed in the RSTAB graphic
- Display of stresses and stress ratios on rendered model
- Export of modified cross-sections and materials to RSTAB
- Data export to MS Excel and OpenOffice.org Calc or as a CSV file

Therefore, STEEL is the appropriate program for general stress analyses. However, stability analyses, as required for example in EN 1993-1-1 or DIN 18000 part 2, <u>cannot</u> be performed. For such analyses, the add-on modules STEEL EC3 or KAPPA and LTB are recommended.

We hope you will enjoy working with the add-on module STEEL.

Your DLUBALTeam



1.2 STEEL Team

The following people were involved in the development of STEEL:

Program coordination

Dipl.-Ing. Georg Dlubal

Dipl.-Ing.(FH) Younes El Frem

Programming

Ing. Ing. Zdeněk Kosáček Dipl.-Ing. Georg Dlubal Ing. Martin Budáč Mgr. Petr Oulehle Zbyněk Zámečník

Cross-section and material database

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

Program design, dialog figures and icons

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

Program supervision

Ing. Martin Vasek

Localization, manual

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

M.Eng. Dipl.-Ing.(BA) Andreas Niemeier

Technical support and quality management

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



1.3 Using the Manual

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

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

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

1.4 Open the Add-on Module STEEL

RSTAB provides the following options to start the add-on module STEEL.

Menu

۲

To start the program in the RSTAB menu bar, click

Add-on Modules \rightarrow Design - Steel \rightarrow STEEL.

-	l-on Modules Windo	w	<u>H</u> elp	D	
* 8	Current Module		4	> 🖻 🕺 🛸 🎘	🖂 🖬 📾 🔚 🎆 📽 🤹 😰 🏦 🎾 💧 🖄 💠 🕼
	Shape Properties	÷	3 -	™ - ® - i	M 🖘 🗾 🐂 🗸 🙀 My 📈 My 📊 🏹 🖓 🛃
	Design - Steel	•	D	STEEL	General stress analysis of steel members
	Design - Concrete	×	Fee	STEEL EC3	Design of steel members according to Eurocode 3
	Design - Timber	×	Also	STEEL AISC	Design of steel members according to AISC (LRFD or ASD)
	Design - Composite	•	Lis	STEEL IS	Design of steel members according to IS
	Design - Aluminium	+	SIA	STEEL SIA	Design of steel members according to SIA
	Dynamic	ł	IBS	STEEL BS	Design of steel members according to BS
	Connections	ł	GB	STEEL GB	Design of steel members according to GB
	Foundations	ł	Is	STEEL CS	Design of steel members according to CS
	Stability	+	A'	STEEL AS (not in	stalled) Design of steel members according to AS
	Others		1	KAPPA	Flexural buckling analysis
	ouldis		1	LTB	Lateral-torsional and torsional-flexural buckling analysis
			€. Fe	FE-LTB L	ateral-torsional and torsional-flexural buckling analysis by FEM
			14	EL-PL (not instal	led) Elastic-plastic design
				C-TO-T	Analysis of limit slenderness ratios (c/t)
			2	PLATE-BUCKLING	Plate buckling analysis
			M	VERBAND	Design of wind bracings for roofs
			I	ASD	Design of steel members according to AISC (ASD)
			4	CRANEWAY	Design of crane runway girders

Figure 1.1: Menu: Add-on Modules \rightarrow Design - Steel \rightarrow STEEL



Navigator

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

Add-on Modules \rightarrow STEEL.

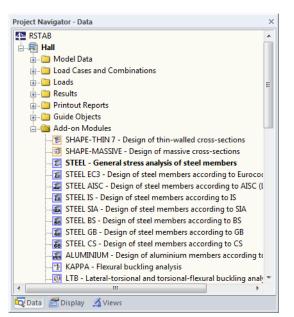


Figure 1.2: Data navigator: Add-on Modules \rightarrow STEEL

Panel

In case STEEL results are already available in the RSTAB model, you can also open the design module in the panel:

Set the relevant STEEL design case in the load case list of the RSTAB toolbar. To display the stresses and design ratios graphically, click [Show Results].

When the results display is activated, the panel is available, too. Now you can use the button [STEEL] to open the module.

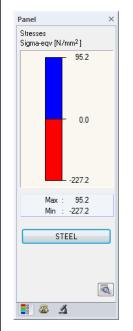
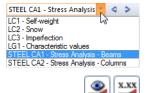


Figure 1.3: Panel button [STEEL]

STEEL





2. Input Data

When you have started the add-on module, a new window opens. In this window, a Navigator is displayed on the left, managing the tables that can be selected currently. The drop-down list above the navigator contains the design cases (see chapter7.1, page 53).

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

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

To select a table, click the corresponding entry in the navigator. To set the previous or next input window, use the button shown on the left. You can also use the function keys to select the previous [F3] or next [F2] window.

Click [OK] to save the results. Quit STEEL and return to the main program. When you click [Cancel], you exit the module but without saving the data.

2.1 General Data

In window 1.1 *General Data*, you select the members, sets of members, and actions that you want to design. The design standard will be specified in window 1.2 because the material properties are related to the standard.

	1.1 General Da	ta				
nput Data General Data Materials Cross-sections Secula Sitesses by Set of Members Sitesses by Set of Members Sitesses by Kucoation Sitesses at Every Sitess Point Governing Internal Fraces by M - Governing Internal Fraces by S Parts List by Member Parts List by Set of Members	Sets: Existing Load G LC1 G LC1 G LC2 G LC4 C2 C4 C4 C5 C7 C5 C7 C5 C0 C7 C5 C0 C7 C5 C0 C7 C5 C0 C7 C5 C0 C7 C5 C7 C5 C7 C5 C7 C7 C7 C7 C7 C7 C7 C7 C7 C7	Self-weight > Snow Live load Wind in X Wind in X Imperfections in X Imperfections in Y Shock bad 1.35*LC1 1.35*LC1 1.55*LC2 1.35*LC1 1.5*LC2 1.35*LC1 1.5*LC1 1.35*LC1 1.5*LC1 </th <th>s s</th> <th>✓ All ✓ All ■ ∠ All ■ Cold for Design IN FC1 0.27 (STR/GE0) - Permanent</th> <th>nt / varable - Gr 6.1</th> <th>STEEL members</th>	s s	✓ All ✓ All ■ ∠ All ■ Cold for Design IN FC1 0.27 (STR/GE0) - Permanent	nt / varable - Gr 6.1	STEEL members
	STR C012 STR C013 STR C014 STR C015 All	135/LC1 + 157LC2 + 1057LC3 + 057LC5 135/LC1 + 157LC2 + 1057LC3 + 057LC5 + 1357LC1 + 157LC2 + 057LC4 1357LC1 + 157LC2 + 057LC4 + LC6 ▼ ▼		Ĵ.	3 1	Stress Analysis of Members and Sets of Members Cross-Section Optimization Parts List

Figure 2.1: Window 1.1: General Data



Cancel

OK





Design of

Design of		
Members:	1-8,11-18,21-28,31-46,51-64,66-69,71-74,81-83,91-1	🔽 All
Sets:	1-8 🐧 🗙 🎦	🔽 All

Figure 2.2: Design of members and sets of members



2

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 boxes. Then you can access the input fields to enter the relevant members or sets of members. The list of the numbers preset in the field can be selected by double-clicking and overwritten by entering the data manually. Use the button [ⁿ] if you want to display the objects graphically.

When you design a set of members, STEEL determines the maximum stress ratio of all members contained in the set of members. The results are shown in the window 2.2 *Stresses by Set* of Members, 3.2 *Governing Internal Forces by Set of Members* and 4.2 *Parts List by Set of Members*.

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

Existing Load Cases

In this column, all load cases as well as load and result combinations that have been created in RSTAB are listed.

Furthermore, STEEL is able to design super combinations and dynamic load cases from the add-on module DYNAM.

Use the button [▶] to transfer selected entries to the list *Selected for Design* on the right. You can also double-click the entries. To transfer the complete 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 that are marked by an asterisk (*), like load case 6 and 7 in Figure 2.1, cannot be designed: This happens when the load cases are defined without any load data or the load case contains imperfections. Then, when you transfer the load cases, a corresponding warning appears.

At the end of the list, there are several filter options available. They will help you assign the entries sorted according to load cases and combinations or action categories. The buttons are reserved for the following functions:

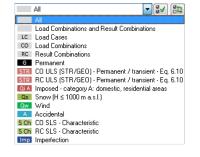
Select all cases in the list.
Invert selection of load cases.

Table 2.1: Buttons in dialog section 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 the design. Click [\blacktriangleleft] or double-click the entries to remove selected items from the list. With the button [$\triangleleft \triangleleft$], you can transfer the entire list to the left.

The design of an enveloping max/min result combination is performed faster than the analysis of all load cases and load combinations that have been globally set. However, when analyzing a result combination, the influence of the contained loads is difficult to infer.







Comment

~
-

In this input field, you can enter user-defined notes describing in detail, for example, the current STEEL design case.

2.2 Materials

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

1.2 Materi	als									
	A	В	С	D	1		F	G		
Material	Material	Safety Factor	Yield Strength		Lim	it Stresse	s [N/mm ²]			
No.	Description	γм [-]	fyk [N/mm ²]	Manual		tσx	Limit τ	Limit σ _{eqv}		
1	Steel S 235 DIN 18800:1990-1	1.10	240.0			218.2	126.0	218.2		
2	Steel S 355 DIN 18800:1990-11	1.10	360.0			327.3	189.0	327.3		
3	Concrete C30/37 EN 1992-1-1:	1.00	0.0			0.0	0.0	0.0		
							1	\$		
	Properties									
🖃 Main F										
	dulus of Elasticity		E		210000.0	N/mm ²				
	ar Modulus		G		81000.0	N/mm ²				
	Poisson's Ratio		ν		0.296 78.50 kN/m ³					
	cific Weight		γ							
	fficient of Thermal Expansion		o.	1	.2000E-05	1/K				
	ial Safety Factor		γM		1.10				Material No. 2 used	in
	nal Properties									
	fficient for Limiting Stresses of Welds		αw		0.800				Cross-sections No.:	
	ckness Range t ≤ 4.00 cm								2,3	
	litimate Strength		fu			N/mm ²				
Y	îeld Strength		fy		360.0	N/mm ²			Members No.:	
									3,8,13,18,23,28,41	,46
									Sets of members No	a.:
									3	
									Σ Lengths:	Σ Masses:
									24.09 [m]	1.508 [t]

Figure 2.4: Table 1.2 Materials

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

The material properties required for the determination of internal forces are described in chapter 4.2 of the RSTAB manual (*Main Properties*). The material properties required for design are stored in the global material library. The values are preset (*Additional Properties*).

To adjust the units and decimal places of material properties and stresses, select in the module's **Settings** \rightarrow menu **Units and Decimal Places** (see chapter 7.3, page 57).



Material Description

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

Steel S 235	DIN 18800:1990-11
Steel S 275	DIN 18800:1990-11
Steel S 355	DIN 18800:1990-11
Fine Grain Steel S 275 N	DIN 18800:1990-11
Fine Grain Steel S 275 M	DIN 18800:1990-11
Fine Grain Steel S 355 N	DIN 18800:1990-11
Fine Grain Steel S 355 M	DIN 18800:1990-11
Fine Grain Steel S 460 N	DIN 18800:1990-11
Fine Grain Steel S 460 M	DIN 18800:1990-11
Improved Steel C 35+N	DIN 18800:1990-11

Figure 2.5: List of materials

Only "Steel" materials are available in the list. In principle, you can carry out the design with different materials if their stress design concept is based on the comparison of existing normal, shear, and equivalent stresses with allowable stresses (for example the design of cross-sections consisting of aluminum or stainless steel). Of course, you must additionally take into account further standard specifications.

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

Limit stresses of a material that is not allowed (for example coniferous timber) can be defined by means of *Yield Strength* (column C) or by ticking the check box *Manually* (column D) and entering user-defined specifications. When you have defined the allowable stresses in the columns E to G, the red color will disappear.

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

The import of materials from the library is described later.

Safety Factor γ_M

This factor describes the safety factor contained in the design values of the material stiffnesses. Therefore, *M* is indicated. By means of the factor γ_{M} , the characteristic value of the yield strength f_{yk} is reduced in order to determine the limit normal stress $\sigma_{R,d}$ (see Equation 2.1) and the limit shear stress $\tau_{R,d}$ (Equation 2.2).



Thus, for example for DIN 18800, the factor γ_{M} is considered twice for the design if the calculation is carried out according to second-order or large deformation analysis: According to DIN 18800 part 2, el. (116) you must consider the influence of deformations by a stiffness that is reduced about 10 % when determining internal forces. Moreover, you must reduce the design values of the stiffnesses by the safety factor γ_{M} when you design the ultimate limit state.

Yield Strength f_{y,k}

The yield strength describes the limit to which the material can be strained without plastic deformation. The characteristic values of several steel grades can be found for example in EN 1993-1-1, section 3 or DIN 18800 part 1, section 4.

When modifying the yield strength, the limit stresses in columns E, F, and G are adjusted automatically.

For some materials, there is a relation between the characteristic yield strength $f_{y,k}$ and the thickness *t* of the relevant structural component. In the section *Material Properties*, the *Thickness Range* of the material selected above is shown with the corresponding yield strength.

2 Input Data



The zones of the yield strength are specified in the standards, for example DIN 18800 part 1, table 1. To control and, if required, adjust the thickness of structural components including yield strength, click [Edit] (see Figure 2.7, page 14).

Limit Stresses

The limit stresses of materials that are stored in the material library are preset automatically.

If you want to adjust the limit stresses, you can use the check box *Manually* or the button [Edit Material] (see Figure 2.7, page 14).

Manually

If the check box is ticked, you can define the limit stresses in columns E to G manually.

Modified material properties are marked by an asterisk in the column Material Description.

Limit σ_x

The limit normal stress represents the allowable stress for actions due to bending and axial force. According to DIN 18800 part 1, el. (746), it is determined by the characteristic value of the yield strength, reduced by the partial safety factor γ_{M} .

$$\sigma_{x,R,d} = \frac{f_{yk}}{\gamma_M}$$

Equation 2.1

Limit τ

The limit shear stress indicates the allowable shear stress due to shear and torsion. To determine the limit shear stress according to DIN 18800 part 1, el. (746), the partial safety factor γ_{M} is also taken into account by the equation used to determine the limit shear stress.

$$\tau_{\text{R,d}} = \frac{f_{yk}}{\gamma_{\text{M}} \cdot \sqrt{3}}$$

Equation 2.2

Limit σ_{eqv}

The limit equivalent stress represents the allowable equivalent stress for the simultaneous effect of several stresses. According to DIN 18800 part 1, el. (746), it is determined by Equation 2.1.

	A
Material	Material
No.	Description
1	Steel S 235 DIN 18800:1990

10

1



Material Library

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

Edit ightarrow Material Library

or use the button shown on the left.

Filter	Material to Select			
Material category group:	Material Description		Standard	
Metal	✓ Steel S 235		💻 DIN 18	800:1990-11
	Steel S 275		🧾 DIN 18	800:1990-11
Material category:	Steel S 355		🥅 DIN 18	800:1990-11
Steel	Eine Grain Steel S 275 N		🥅 DIN 18	800:1990-11
	Fine Grain Steel S 275 M			800:1990-11
Standard group:	Fine Grain Steel S 355 N			800:1990-11
💻 DIN	Fine Grain Steel S 355 M			800:1990-11
	Fine Grain Steel S 460 N			800:1990-11
Standard:				
DIN 18800:1990-11	Fine Grain Steel S 460 M			800:1990-11
	Improved Steel C 35+N			800:1990-11
	Improved Steel C 45+N			800:1990-11
	Steel St 37-2		🧮 DIN 18	800:1990-11
	Steel USt 37-2		🧮 DIN 18	800:1990-11
🔲 Include invalid	🐷 🔲 Steel RSt 37-2		🥅 DIN 18	800:1990-11
			1	
Favorites only				/
Material Properties			St	eel S 235 DIN 18800:1990-
■ Main Properties				
 Modulus of Elasticity 		E	210000.0	
Shear Modulus		G	81000.0	N/mm ²
Poisson's Ratio		v	0.296	
- Specific Weight		γ		kN/m ³
 Coefficient of Thermal Expa 	nsion	α	1.2000E-05	1/K
Partial Safety Factor Additional Properties		7M	1.10	
 Additional Properties Coefficient for Limiting Stres 	ses of Welds	aw	0.950	
☐ Thickness Range t ≤ 4.00 c		0.0	0.300	
Ultimate Strength		fu	360.0	N/mm ²
Yield Strength		fy		N/mm ²
□ Thickness Range t > 4.00 c	cm and t ≤ 10.00 cm			
Ultimate Strength		fu		N/mm ²
Yield Strength		fy	215.0	N/mm ²

Figure 2.6: Dialog box Material Library

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

Click [OK] or use the [$_{\rightarrow}$] key to transfer the selected material to window 1.2 of the add-on module STEEL.

Chapter 4.2 in the RSTAB manual describes in detail how materials can be filtered, added, or rearranged.

You can also select materials of categories like *Cast Iron* or *Stainless Steel*. Please check, however, whether these materials are allowed by the standard's design concept.



or	u



Edit Material

15

To adjust the yield strengths and limit stresses of the currently selected material, click the button shown on the left. The following dialog box appears:

dit Materi	ial					e
Material De	escription					
Steel S 2	35		- 📖 -			
Safety Fac	ctor					
үм:	1.10 🚔 [·]					
Yield Strer	ngths and Limi	t Stresses Sub	ject to Compone	nt Thicknesses		
	A	В	С	D	E	F
Range	Thicknes		Yield Strength		it Stresses [N/mn	
No.	from	to	f _{yk} [N/mm ²]	Limit σ _×	Limit τ	Limit σ _{eqv}
1	0.0	40.0	240.0	218.2	126.0	218
2	40.0	100.0	215.0	195.5	112.8	195.
Number of ranges:	f	2 🌲	🔲 User-	defined, independ	dent definition of li	imit stresses
Comment						
(M) 🕞	2				ОК	Cancel

Figure 2.7: Dialog box Edit Material

In the table *Yield Strengths and Limit Stresses Subject to Component Thicknesses* the limits of the *Component Thickness t* can be shifted by entering values manually in column B. The number of ranges is defined in the standards.

If the check box *User-defined, independent definition of limit stresses* is ticked, you can define the limit stresses for each range individually. If this check box is not active, the limit stresses are determined by the yield strength $f_{y,k}$ (column C) and the safety factor γ_M according to Equation 2.1 and Equation 2.2.



2.3 Cross-Sections

This window lists the cross-sections that are used for the design. In addition, the window allows you to specify optimization parameters.

Edit Settings Help							
General stress analysis of $ extsf{ heta}$	1.3 Cross-	sections					
Data		A	В	С	D	E	1 - IPE 500 DIN 1025-5:1994
eneral Data	Section		Cross-Section	Opti-			
aterials oss-sections	No.	No.	Description	mize	Note	Comment	. 20.00
oss-sections	1	1	I IPE 500 DIN 1025-5:1994	From Current Row	2)		+
	2	1	I IPE 360 DIN 1025-5:1994	No Lê			+
	3	1	I IPE 450 DIN 1025-5:1994	From current row			² / _{2.10}
	7	1	T HE A 160 DIN 1025-3:1994 T HE A 120 DIN 1025-3:1994	From favorites 'SIA' From favorites 'BS'			
	9	1	T IPE 360 DIN 1025-3:1994	No			20.00
	10	1	T HE A 140 DIN 1025-3:1994	No			20
	12	1	QRO 80x4 DIN 59410:1974	No			1.02
	13	1	Circle 2.4	No			
	15	1	T HE A 200 I DIN 1025-3:1994	No			
							Ť,
							0 🎽 🚰 🎞 🖾
							Cross-section No. 1 used in
							Members No.:
							1,2,11,12,21,22,31,32,39,40
							Sets of members No.:
							Σ Length: Σ Weight:
							48.00 [m] 4.371
							Material
							1 - Steel S 235
			2) The cross-section will be optimized table will be used.	d. Therefore, the optimal s	ection from the	i	
8	Calculati		Details	Graphics			OK Can

Figure 2.8: Window 1.3 Cross-Sections

Cross-Section Description

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



3

If you want to modify a cross-section, click the entry in column B to activate this field. Click [Cross-section Library] or [...] in the field or press function key [F7] to open the cross section table of the current input field (see figure on the left).

In this dialog box, you can select a different cross-section or a different row. To select a different cross-section category, click [Back to Cross-section Library] and access the general crosssection library.

Chapter 4.3 of the RSTAB manual describes how cross-sections can be selected from the library.



Welded Cross-Sections - I symmet	ric		.
Cross-Section Type I I Γ Τ Γ L L Π C I Τ Υ Ο ∇ Π Π Π Π Π Π I I + • - I L Γ Γ Σ	Parameters h: 360.0 m/m [mm] b: 170.0 m/m [mm] s: 8.0 m/m [mm] t: 14.0 m/m [mm] a: 0.0 m/m [mm]		
		IS 360/170/8/14/0	
			OK Cancel

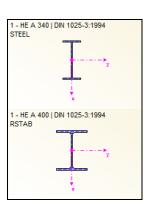


Figure 2.9: IS section types in the cross-section library

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

A modified cross-section will be highlighted in blue.

If the STEEL cross-sections are different from the ones used in RSTAB, both cross-sections are displayed in the graphic in the right part of the window. The stress analysis will be performed with the internal forces from RSTAB for the cross-section selected in STEEL.

Max. Design Ratio

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

Optimize

You can optimize every cross-section from the library: Search for the cross-section in the same row that comes as close as possible to a user-defined maximum ratio. The maximum ratio can be defined in the dialog box *Details* (see Figure 3.1, page 19).

If you want to optimize a cross-section, open the corresponding drop-down list in column D or E and select the relevant entry: *From current row* or, if available, *From favorites 'description'*. Recommendations for the cross-section optimization can be found in chapter 7.2 on page 55.

Remark

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

A warning might appear before the calculation: *Incorrect type of cross-section!* In this case, there is a cross-section that is not registered in the data base. This may be a user-defined section or a SHAPE-THIN section that has not been calculated yet. To select an appropriate section for the design, use the [Library] button (see description in Figure 2.8).



Member with tapered cross-section

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

STEEL also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the end of the member. The normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the start and the end cross-section of a tapered member have not the same number of stress points, the intermediate values cannot be interpolated. A calculation won't be possible neither in RSTAB nor in STEEL.

To produce the same number of stress points, you can, for example, define the second crosssection 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 cross-sections. In such a case, the cross-section group *IVU I-Section Plus Lower Flange* is recommended.

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

Cross-section graphic

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

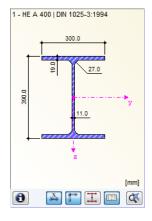
The buttons below the graphic are reserved for the following functions:

Button	Function
0	Opens the dialog box Info about Cross-section (see Figure 2.10)
X	Displays or hides the dimensions of the cross-section
1	Displays or hides the principal axes of the cross-section
Ŧ	Displays or hides the stress points
123	Displays or hides the numbering of stress points or c/t-parts
X	Resets the full view of the cross-section graphic

Table 2.2: Buttons of cross-section graphic

Info About Cross-Section

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



Ŧ

0

0



Cross-Section Value Description	Symbol	Value	Unit	*	HE A 400 DIN 1025-3:1994
Depth	h	390.0	mm		
Width	Ь	300.0	mm	-	
Web thickness	tw	11.0	mm		
Flange Thickness	tf	19.0	mm		300.0
Root fillet radius	r	27.0	mm	Ξ	
Cross-sectional area	А	159.00	cm ²	-	
Shear area	Ay	94.96	cm ²		27.0
Shear area	Az	38.77	cm ²		
Shear area according to EC 3	Av.y	118.18	cm ²		
Shear area according to EC 3	A _{v,z}	57.35	cm ²		
Web area	AWeb	38.70	cm ²		300 D
Plastic shear area	A _{pl,y}	114.00	cm ²		R Y
Plastic shear area	A _{pl,z}	40.81	cm ²		
Moment of inertia	ly	45070.00	cm ⁴		11.0
Moment of inertia	Iz	8560.00	cm ⁴		
Governing radius of gyration	ſy	168.0	mm		
Governing radius of gyration	٢z	73.4	mm		
Polar radius of gyration	ro	183.3	mm		↓ · · · · · · · · · · · · · · · · · · ·
Radius of gyration of flange plus 1/5 of we	rzg	79.4	mm		Z
Volume	V	15900.00	cm ³ /m		
Weight	wt	124.8	kg/m		
Surface	Asurf	1.910	m²/m		
Section factor	A _m /V	120.126	1/m		[m
Torsional constant	J	190.00	cm ⁴		🛄 🔝 🔍 Stress points 🔛 🚰 🖸
Warping constant	Cw	2.942E+06	cm ⁶		C/t-Parts
Elastic section modulus	S.,	2310.00	cm ³	Ŧ	

Figure 2.10: Dialog box Info About Cross-section

Click the button [Details] to receive detailed information on stress points (distance to center of gravity, statical moments of area, normalized warping constants etc.) and c/t-parts.

	A	B	C	D	E	F	G	HE A 400
itressP	Coordi		Statical Mom		Thickness	Wan		
No.	y [mm]	z [mm]	Q _y [cm ³]	Q _z [cm ³]	t [mm]	W _{no} [cm ²]	Sco [cm ⁴]	
1	-150.0	-195.0	0.00	0.00	19.0	278.25	0.00	
2	-32.5	-195.0	-414.53	-203.75	19.0	60.29	-3778.92	
3	0.0	-195.0	-537.44	-214.82	19.0	0.00	-3965.06	
4	32.5	-195.0	-414.53	203.75	19.0	-60.29	3778.92	
5	150.0	-195.0	0.00	0.00	19.0	-278.25	0.00	1 2 3 4 5
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	-537.44	214.82	19.0	0.00	-3965.06	
9	32.5	195.0	-414.59	-203.77	19.0	60.29	3778.92	3
10	150.0	195.0	0.00	0.00	19.0	278.25	0.00	
11	0.0	-149.0	-1153.46	0.00	11.0	0.00	0.00	
12	0.0	149.0	-1153.16	0.00	11.0	0.00	0.00	Series Series
13		0.0	-1275.73	0.00	11.0	0.00	0.00	6 7 8 9 10
								z

Figure 2.11: Dialog box Stress Points of HE A 400



3. Calculation

3.1 Detail Settings

Calculation

Details...

Before you start the calculation by clicking [Calculation], it is recommended to check the design details. The corresponding dialog box can be accessed in all windows of the add-on module by clicking the [Details] button.

Details					
Display Result Tables	Consider Plastification				
I Stresses by Cross-Section 2.2 Stresses by Set of Members 2.3 Stresses by Member 2.4 Stresses by x-Location 2.5 Stresses by Stress Point	 Allow local plastification according to DIN 18800 Part 1, Element (749) (Exceed Sigma-eqv by 10%) Calculate normal stresses with plastic shape factors Alpha-pl according to DIN 18800 Part 1, El (750) (Only considered with respect to rolled I-sections) 				
 3.1 Governing Internal Forces for: Sigma-eqv 3.2 Governing Internal Forces by Set of Members 4.1 Parts List by Member 4.2 Parts List by Set of Members Of designed members / sets of members only 	Eccentric Transverse Loads on I-Beams Simplified consideration of eccentric loading for I-beams with large depth (Rolled I-sections, IS) Analyze stresses caused by M-z only at top chord with substitute cross-section (Half of an I-section) Calculation of Equivalent Stress Sigma-eqv				
Of all members / sets of members					
Method of Stress Calculation For result combinations: Stresses from internal forces	Factor for σ: 1.00 * Factor for τ: 3.00 *				
Collection internal notices Calculate stresses from single load cases in RC and then combine stresses according to RC criteria Combine stresses from individual extreme values	Cross-Section Optimization Maximum allowable design ratio:				
	OK				

Figure 3.1: Dialog Details

Display Result Tables

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

Window 2.5 *Stresses by Stress Point* is inactive by default because the stress graphic can also provide an evaluation of the results in the stress points. For a check of stress points listed in tables, activate the display of this window.

The values displayed in window 3.1 *Governing Internal Forces* are related to the maximum equivalent stress σ_{eqv} (default setting). If you want to evaluate the results specifically, you can select a different stress type in the list.

Method of Stress Calculation for Result Combinations

A biaxial loading in different load cases may result in the fact that the combined member internal forces do not produce the maximum stresses. This is the case, for example, when the first load case with vertical load has only M_y moments but no M_z moments, and the second load case with horizontal load has only M_z moments but no M_y moments. If you superimpose the load cases with the value "permanent" in a result combination, RSTAB will display in the table 4.1 *Members - Internal Forces* that the moment M_z is not classified as belonging to the maximum moment M_y: The horizontal load does not contribute to the increase of the moment due to vertical load. The RC design is then carried out separately for the maximum moments M_y and M_z so that the simultaneous effect of both moments for the combined analysis of the bending stresses is not taken into account.

Sigma-eqv Sigma Total Sigma N-y Sigma M-y Sigma M-y Sigma N+M-2 Sigma Compressive Sigma Compressive Sigma Delta Tau Total Tau V-y Tau V-y Tau V-2 Tau V-2 Tau M-T St.Venant Tau M-T St.Venant Tau M-T Bredt



Stresses from Internal Forces

This type of calculation makes direct use of the RSTAB table 4.1 *Members - Internal Forces*. By processing the maximum or minimum internal forces row by row, the stresses for each extreme value together with the corresponding internal forces are determined.

The advantage is that the values of the result combinations can be used directly. This has a positive impact on the speed of the calculation. In addition, the designed internal forces become transparent because in the STEEL window *3.1 Governing Internal Forces by Member* the module displays the result rows from the RSTAB results table *4.1 Members - Internal Forces*.

Calculate Stresses from Single Load Cases and then Combine

This type of calculation is preset to calculate the stresses of result combinations. First, the program determines the normal and shear stresses of the load cases contained in the RC. Then, these stress components are superimposed according to the conditions specified in the combination criterion. Thus, the program ensures that a biaxial bending stress from different load cases results in correct stresses in a biaxial load state.

The calculation is performed for each stress point. The compression, tension and shear stresses determined for each load case are summed up according to the RC superposition criterion and displayed subsequently as stresses of the result combination. The equivalent stress σ_{eqv} represents an exception because it is determined by the RC components of σ_{total} and τ_{total} . A superposition of equivalent stresses from the load cases would not be correct as it would lead to too high stress ratios.

This type of calculation requires more computing time than the direct use of RC internal forces. Furthermore, the values displayed in window 3.1 *Governing Internal Forces by Member* for the equivalent stresses are more difficult to understand.

As pure uniaxial bending usually does not occur in complex 3D models, both calculation methods should show the same ratios in the output.

Combine stresses from individual extreme values

The check box is deactivated by default so that the stresses are considered row by row (table of internal forces from RSTAB or stresses from load case internal forces).

If you select this check box, STEEL calculates the stresses not from the according internal forces or stresses but from the extreme values of the internal forces or stress components at each x-location (for example max/min values of N, M_y, and M_z for normal stress). These "extreme value diagonals" guarantee that the most unfavorable constellation is considered (see the text above introducing the section "Method of Stress Calculation").

Consider Plastification

Allow Local Plastification

If a local plastification according to DIN 18800 part 1, el. (749) is allowed, the equivalent stress σ_{eqv} may exceed the allowed limit stress by 10 % in "small areas".

STEEL analyzes the conditions mentioned in el. (749) for the small areas.

$$\left|\sigma_{\mathsf{N}} + \sigma_{\mathsf{M}_{\mathsf{y}}}\right| \leq 0.8 \cdot \sigma_{\mathsf{R},\mathsf{d}}$$

Equation 3.1

$$\left|\sigma_{\mathsf{N}} + \sigma_{\mathsf{M}_{\mathsf{Z}}}\right| \leq 0.8 \cdot \sigma_{\mathsf{R},\mathsf{d}}$$

Equation 3.2

If the conditions are fulfilled, the limit stress for the design of σ_{eqv} will be increased.



Calculate normal stresses with plastic shape factors $\,\alpha_{\mbox{\tiny pl}}$

According to DIN 18800 part 1, el. (750), stresses can be reduced by plastic shape factors α_{pl} . This reduction refers to the normal stresses σ_M due to the bending moments M_y and M_z .

$$\sigma_{M} = \pm \frac{M_{y}}{\alpha_{pl,y} \cdot l_{y}} \cdot e_{z} \pm \frac{M_{z}}{\alpha_{pl,z} \cdot l_{z}} \cdot e_{y}$$

Equation 3.3

If you use this plastification option, STEEL applies the plastic shape factors $\alpha_{pl,y} = 1.14$ and $\alpha_{pl,z} = 1.25$ which are mentioned in the standard.

The reduction of normal stresses by plastic shape factors according to DIN 18800 part 1, el. (750) is allowed only for rolled I-shaped cross-sections.

Eccentric Transverse Loads on I-Beams

If transverse loads are introduced on the upper flange of beams, their influence on the bending stress in the lower flange decreases with increasing cross-section depth. For this reason, it is possible to consider eccentrically acting transverse loads by a simplified method for high I-sections. For each(!) rolled or parametrical symmetrical I-section that is set for design, the bending moment M_z is calculated only on the upper flange. STEEL uses a substitute crosssection with half of the moment of inertia I_z for stress determination.

The advantage of this option is that you can enter the loads in the RSTAB model in relation to the centroidal axes in order to avoid torsion.

As the ticked check box affects all symmetrical I-sections of the design case, it is recommended to design all high I-beams in a separate STEEL case (see chapter 7.1, page 53).

Equivalent stress σ_{eqv}

In this dialog section, you can adjust the factors for the determination of the equivalent stress.

$$\sigma_{eqv} = \sqrt{f_1 \cdot \sigma_{total}^2 + f_2 \cdot \tau_{total}^2}$$
where f_1 : factor for normal stresses f_2 : factor for shear stresses

Equation 3.4

The factors mentioned in DIN 18800 part 1, el. (748) are preset with $f_1 = 1.0$ and $f_2 = 3.0$.

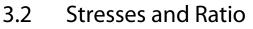
Cross-Section Optimization

The optimization is targeted on the maximum stress ratio of 100 %. If necessary, you can specify a different limit value in this input field.









The normal stresses σ_{total} , τ_{total} , and σ_{eqv} are displayed as presettings in the windows 2.1 to 2.5. To display further stress components, click [Select Stresses to Show] and [Extended Stress Diagram] (see Figure 5.1, page 39).

Normal stresses

For signs, the following rule applies in STEEL: Tensile stresses are indicated by positive signs and compressive stresses by negative signs (see Figure 3.2).

The analysis is carried out for each single stress point. Therefore, the components of the maximum stresses for a combined calculation (for example σ_{total}) must <u>not</u> be summed up: Often, the maximum stresses occur at different stress points. You have to superimpose the stress components that are available in the same stress point.

The normal stresses σ have the following meanings:

	Stress due to axial force N
σ_{N}	$\sigma = \frac{N}{A}$
	where A: cross-sectional area of cross-section
бм-у	Stress due to bending moment M _y $\sigma = \frac{M_y}{\alpha_{pl,y} \cdot l_y} \cdot e_z$
,	$ \begin{array}{ll} \mbox{where} & \alpha_{pl,y} \mbox{:} & \mbox{plastic shape factor according to DIN 18800 part 1, el. (750)} \\ & & \mbox{l}_{y} \mbox{:} & \mbox{second moment of area related to principal axis y} \\ & & \mbox{e}_z \mbox{:} & \mbox{centroidal distance of the stress point in direction z} \end{array} $
σ _{M-z}	Stress due to bending moment M _z $\sigma = -\frac{M_z}{\alpha_{pl,z} \cdot l_z} \cdot e_y$
	$ \begin{array}{ll} \mbox{where} & \alpha_{pl,z} \mbox{:} & \mbox{plastic shape factor according to DIN 18800 part 1, el. (750)} \\ & \mbox{l}_z \mbox{:} & \mbox{second moment of area related to principal axis z} \\ & \mbox{e}_y \mbox{:} & \mbox{centroidal distance of stress point in direction y} \end{array} $
σ_{M}	Stress due to bending moments M _y and M _z $\sigma = \frac{M_y}{\alpha_{pl,y} \cdot l_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot l_z} \cdot e_y$
$\sigma_{tension}$	Tensile stress due to axial force N and bending moments M _y and M _z $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot l_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot l_z} \cdot e_y$
$\sigma_{\text{compression}}$	Compressive stress due to axial force N and bending moments M _y and M _z $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot l_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot l_z} \cdot e_y$
σ_{range}	Maximum difference between normal stresses of different load cases that are required, for example, for the fatigue design
σ_{total}	Normal stress due to axial force N and bending moments M _y and M _z $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot l_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot l_z} \cdot e_y$

Table 3.1: Normal stresses $\boldsymbol{\sigma}$



The local member axis system has a certain impact on the signs of the internal forces and stresses.

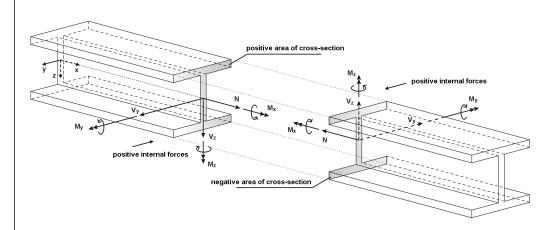


Figure 3.2: Positive definition of internal forces

The bending moment $\mathbf{M}_{\mathbf{y}}$ is positive if tensile stresses are generated on the positive member side (in direction of the z-axis). $\mathbf{M}_{\mathbf{z}}$ is positive if compressive stresses occur on the positive member side (in direction of the y-axis). The sign definition for torsional moments, axial forces, and shear forces conforms to the following convention. These internal forces are positive if they act in a positive direction.



If the *Type of Model* was defined as **2D** in the *General Data* of RSTAB, the design of unsymmetrical cross-sections like angles or C-sections needs to be handled with care. In a plane model, only moments about the member axis *y* are possible so that RSTAB performs a division of the moment in the direction of the principal axes *u* and *v*. STEEL designs only the part of the moment about the y-axis. Therefore, systems with unsymmetrical cross-sections should be checked by means of a spatial model (model type *3D*).



Shear stresses

The shear stresses $\boldsymbol{\tau}$ have the following meanings:

	Т						
	Stress due to shear force V_y						
	$\tau = -\frac{V_y \cdot Q_z}{I_z \cdot t}$						
τı	2						
τ _{V-y}	where Q_{z} : statical moment related to principal axis z						
	I _z : second moment of area related to principal axis z						
	t: governing thickness of cross-section						
	Stress due to shear force V _z						
	$\tau = -\frac{V_z \cdot Q_y}{I_y \cdot t}$						
τ _{V-z}	l _y ·t						
€V-Z	where Q _y : statical moment related to principal axis y						
	l _y : second moment of area related to principal axis y						
	t: governing thickness of cross-section						
	Stress due to shear forces V_{y} and V_{z}						
τ_{v}	$\tau = -\frac{V_y \cdot Q_z}{I_z \cdot t} - \frac{V_z \cdot Q_y}{I_z \cdot t}$						
	$l_z \cdot t = l_y \cdot t$						
	Stress due to torsional moment M _T in case of open cross-section						
	$\tau = \frac{M_T}{J} \cdot t$						
$ au_{M-T, St.Venant}$							
	where J: Saint Venant torsional constant						
	t: governing thickness of cross-section						
	Stress due to torsional moment M_T in case of closed cross-section						
	$\tau = \frac{M_{T}}{2 \cdot A_{m} \cdot t}$						
$ au_{M-T, Bredt}$							
	where A_m : area enclosed by the center lines of the cross-section						
	t: governing thickness of cross-section						
	Stress due to torsional moment M ⁺						
τ_{M-T}	$\tau = \frac{M_T}{J} \cdot t \text{ or } \tau = \frac{M_T}{2 \cdot A_m \cdot t}$						
$ au_{total}$	Shear stress due to shear forces V_y and V_z and torsional moment $M_{\rm T}$						
	$\tau = \tau_V + \tau_{M_T}$						

Table 3.2: Shear stresses τ

As the equations show, the program uses the static moments instead of the shear areas of the cross-section to determine the shear stresses due to shear force.



For shear stresses due to torsion, please note:

- If you have a cross-section that is partially open but has one closed cell, the program will classify the entire cross-section as *closed*. The shear stress will be determined exclusively according to the Bredt formula. Thus, STEEL won't perform any pro-rata analysis for M_{T,St.Venant} and M_{T,Bredt} like it is done in the cross-section module SHAPE-THIN.
- The influence of the warping torsion is not considered in STEEL. The design, like the determination of internal forces in RSTAB, is limited to the primary torsional moment. However, if you have to consider warping stresses due to the secondary torsional moment or the warping bimoment, it is recommended to use the add-on module FE-LTB for the analysis.

For thin-walled cross-sections, we can assume as a simplification that the shear stress runs parallel to the wall of the cross-section. Therefore, the parts of the shear stresses resulting from both the components of the shear forces are added. The sign of the statical moments defines here which parts are applied positively and which negatively.

The shear stress due to the torsional moment is to be considered differently for the total shear stress, depending on whether it is an open or a closed cross-section. For an open cross-section, the torsion shear stress is added with the sign to that sum from the individual shear stresses that results in the greatest absolute value of the sum.

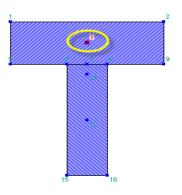
For a closed cross-section, on the other hand, the torsional shear stress is simply added to the sum from the individual shear stresses. Here, the signs for core area and statical moments are set in such a way that they correspond to the program-specific sign conventions of the shear stress that is dependent on the loading.

Shear stresses within the cross-section

Stress points lying within the cross-section do not permit the assumption mentioned above that the shear stress runs parallel to the wall of the cross-section. Here, a special method with twin stress points is used that creates two stress points with identical coordinates in the cross-section.

The one stress point considers the statical moment about the **y**-axis (parameter for the shear stress due to vertical shear force), the other considers the statical moment about the **z**-axis (parameter for shear stress due to horizontal shear force). For these stress points, the complementary statical moment is zero, respectively. It is possible to assign different thicknesses to the twin stress points that have an influence on the calculation of the shear stress. The shear stresses are considered as interdependent components acting perpendicularly to each other: they are components of one stress state. For the determination of the total shear stress, both parts are quadratically added. The shear stress due to the torsional moment is not considered in these points.

The shear stresses of result combinations in the twin stress points may <u>not</u> be combined linearly. Therefore, the extreme values of both components are evaluated with the corresponding complementary shear stresses in order to determine the greatest total shear stress.



Twin stress points



Equivalent stress

The equivalent stress σ_{eqv} , for example according to DIN 18800 part 1, el. (748), is determined as follows:

	Equivale	ent stress	from normal stresses σ and shear stresses $ au$
σ_{eqv}	$\sigma_{eqv} = \gamma$	$f_1 \cdot \sigma_{total}^2$	$+ f_2 \cdot \tau_{total}^2$
- cqv	where	f ₁ :	factor for normal stresses
		f ₂ :	factor for shear stresses

Table 3.3: Equivalent stress σ_{eqv}

Details...

Factors f_1 and f_2 can be defined in the dialog box *Details* (see Figure 3.1, page 19). The factors $f_1 = 1.0$ and $f_2 = 3.0$ are preset according to DIN 18800 part 1, el. (748).

Stress ratio

For stress designs, as described for example in DIN 18800 part 1, el. (747), STEEL determines the quotient from the existing and the limit stress.

$$\frac{\sigma}{\sigma_{R,d}} \le 1$$

Equation 3.5: Design condition for normal stresses

$$\frac{\tau}{\tau_{R,d}} \le 1$$

Equation 3.6: Design condition for shear stresses

$$\frac{\sigma_{eqv}}{\sigma_{R,d}} \leq 1$$

carried out successfully.

Equation 3.7: Design condition for equivalent stresses

For each internal force component, you can display the cross-section's stress ratio on the respective stress point (see chapter 5.1, page 40). The ratios due to normal, shear, and equivalent stress are displayed in the table output by default.

If the limit stress is not exceeded, the ratio is less than or equal to 1 and the stress design was

Max: 0.96 ≤1 ❷



3.3 Start Calculation

Calculation

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

STEEL searches for the results of the load cases, load combinations, and result combinations you want to design. If they cannot be found, the program starts the RSTAB calculation to determine the design relevant internal forces.

You can also start the calculation in the RSTAB user interface: In the dialog box *To Calculate* (menu *Calculate* \rightarrow *To Calculate*) design cases of the add-on modules like load cases and load combinations are listed.

Not Calculate	d			Selected for	Calculation	
No.	Description	_		No.	Description	_
G LC1	Self-weight			ACC CA1	STEEL - General Stress Analysis of Steel Members	
Qi A LC3	Live load					
Qs LC2	Snow	E				
Qw LC4	Wind in X					
Qw LC5	Wind in Y					
A LC8	Shock load					
Imp LC6	Imperfections in X					
Imp LC7	Imperfections in Y					
STR CO1	1.35*LC1		>>			
STR CO2	1.35*LC1 + LC6					
STR CO3	1.35*LC1 + LC7					
STR CO4	1.35*LC1 + 1.5*LC2		4			
STR CO5	1.35*LC1 + 1.5*LC2 + LC6		44			
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 + LC6					
STR CO9	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + LC7					
STR CO10	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + 0.9*LC4					
STR CO11	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + 0.9*LC4 + LC6					
STR CO12	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + 0.9*LC5					
STR CO13	1.35*LC1 + 1.5*LC2 + 1.05*LC3 + 0.9*LC5 + LC7					
STR CO14	1.35*LC1 + 1.5*LC2 + 0.9*LC4					
STR CO15	1.35*LC1 + 1.5*LC2 + 0.9*LC4 + LC6					
STR CO16	1 351 01 + 1 51 02 + 0 91 05	*				
All	▼	2				

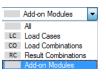




Figure 3.3: Dialog box To Calculate

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

To transfer the selected STEEL cases to the list on the right, click [▶]. Click [OK] to start the calculation.

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

Help	
A > 2 2 2 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4 4	i 🎾 🖄 💠 🛝 🗙 🗗 🕅
🗱 🕶 💆 🕶 🛛 🎯 👻 Show Results 📉 👽 👽 🗤 🕅 🧰 py	77 🛟 🌫 🔡 🗊

Figure 3.4: Direct calculation of a STEEL design case in RSTAB

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



4. Results

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

	2.1 Stresse	es by Cros	s-Section							
t Data		A	B	C	D	E	F	G	Н	1 - HE A 400 DIN 1025-3:1994
General Data	Section	Member	Location	S-Point	Load-	Stress	Stress []		Stress	Sigma Total
Materials	No.	No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio	Member No. 22, x: 0.000 m
Cross-sections	1	HE A 400	DIN 1025-3	:1994						
ılts		22	0.000	6	RC1	Sigma Total	-188.2	218.2	0.86	
tresses by Cross-Section		39	0.000	1	RC1	Sigma N	-16.1	218.2	0.07	
tresses by Set of Members		22	6.000	10	RC1	Sigma M-y	174.9	218.2	0.80	E
tresses by Member		2	0.000	10	RC1	Sigma M-z	30.5	218.2	0.14	
tresses by x-Location		40	0.000	13	RC1	Tau Total	47.4	126.0	0.38	
tresses at Every Stress Point		40	0.000	13	RC1	Tau V	47.0	126.0	0.37	160.3 162.5 163.7 165.8
overning Internal Forces by M		22	6.000	1	RC1	Tau M-T	-4.2	126.0	0.03	
ioverning Internal Forces by S		22	0.000	6	RC1	Sigma-eqv	188.2	218.2	0.86	121.9
arts List by Member										1 2 3 5
arts List by Set of Members	2	IS 360/1	70/8/14/0							
		28	3.011	6	RC1	Sigma Total	-312.7	327.3	0.96	Y'
		46	0.000	1	RC1	Sigma N	-24.7	327.3	0.08	
		28	3.011	5	RC1	Sigma M-y	287.7	327.3	0.88	
		28	3.011	10	RC1	Sigma M-z	8.8	327.3	0.03	
		18	3.011	13	RC1	Tau Total	-32.0	189.0	0.17	
		28	0.000	13	RC1	Tau V	-27.5	189.0	0.15	-12/30 3
		18	3.011	1	RC1	Tau M-T	-11.6	189.0	0.06	
		28	3.011	6	RC1	Sigma-eqv	312.7	327.3	0.96	
	3	IS 500/1	70/8/14/0							
		28	3.011	6	RC1	Sigma Total	-312.7	327.3	0.96	8 8 9 10
		46	0.000	1	RC1	Sigma N	-24.7	327.3	0.08	.144 6 👻
		28	3.011	5	RC1	Sigma M-y	287.7	327.3	0.88	z
		28	3.011	10	RC1	Sigma M-z	8.8	327.3	0.03	-188.2 -188.1 -184.9 -182.7
		18	3.011	13	RC1	Tau Total	-32.0	189.0	0.17	-100.2 -100.1 -104.0 -102.7
		28	0.000	13	RC1	Tau V	-27.5	189.0	0.15	
		18	3.011	1	RC1	Tau M-T	-11.6	189.0	0.06	
		28	3.011	6	RC1	Sigma-eqv	312.7	327.3	0.96	
	6	115 4 400		4004						
	0	82 HE A 16L	0.000 DIN 1025-3	1994	RC1	Sigma Total	-148.6	218.2	0.68	
		82	0.000	1	RC1	Sigma N	-23.0	218.2	0.00	
		82	0.000	5	RC1	Sigma M-y	125.6	218.2	0.58	Min : -188.2 N/mm ² (6) Max : 165.8 N/mm ² (5)
		Max	0.96	≤1 🕲		1		54 😂 🖪	3	

Figure 4.1: Results window with stresses and stress ratios

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

The windows 3.1 and 3.2 list the governing internal forces. In the last two results windows 4.1 and 4.2, parts lists are displayed by members and set of members.

OK

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

Click [OK] to save the results. You exit STEEL and return to the main program.

Chapter 4 *Results* describes the different results windows one after the other. Evaluating and checking results is described in chapter 5 *Results Evaluation*, page 39 ff.





4.1 Stresses by Cross-Section

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



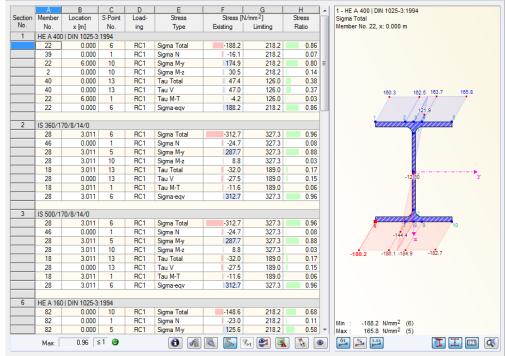


Figure 4.2: Window 2.1 Stresses by Cross-Section

Section No.

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

Member No.

It shows the number of the member that bears the maximum stress ratio within the set of members indicated in column E.

Location **x**

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

- Start and end node
- Dividing points according to possibly defined member division (see RSTAB table 1.6)
- Member division according to specification for member results (RSTAB dialog box *Calculation Parameters*, tab *Global Register Parameters*)
- Extreme values of internal forces

S-Point No.

The design is carried out on certain stress points of the cross-section. These points are defined by centroidal distances, statical moments, and cross-section thicknesses, which allow for a design according to Table 3.1 and Table 3.2 (see page 22).



All default cross-sections of the library as well as the SHAPE cross-sections are provided with stress points on the design relevant cross-section locations. For user-defined cross-sections, the parameters of the stress points must be defined manually or else a design in STEEL will not be possible.



a

The cross-section dialog graphic on the right shows the stress points, including numbering. The currently selected stress point (that means the stress point of the table row where the pointer is placed) is highlighted in red.

To check the stress point's properties, click [Extended Stress Diagram] (see chapter 5.2, page 42).

Loading

Column D displays the numbers of the load cases, load combinations, and result combinations whose internal forces produce the respective maximum stresses.

Stress Type

Ratios due to normal stress σ_{total} , shear stress τ_{total} , and equivalent stress σ_{eqv} are set by default. The determination of these stresses is described in Table 3.1, Table 3.2, and Table 3.3 on page 22 to 26.

You can display the components of the total stresses in order to check the data (see Figure 4.3). The stress components can be selected in the dialog box *Stresses - Filter* (see Figure 5.2, page 40).

Stress Existing

This column displays the extreme values of the existing stresses determined according to the equations in Table 3.1, Table 3.2, and Table 3.3 (see page 22 to 26).

Stress Limiting

This column shows the limit stresses of window 1.2, column E to G (see chapter 2.2, page 10). In particular, they are the following:

- Limit normal stress σ_x as the allowable stress for actions due to bending and axial force
- Limit shear stress τ as the allowable shear stress due to shear and torsion
- Limit equivalent stress σ_{eqv} as the allowable equivalent stress for the simultaneous effect of normal and shear stresses

Stress ratio

The last column shows the quotient from the existing and the limiting stress. If the limiting stress is kept, the ratio is less than or equal to 1 and the stress design was carried out successfully.

The length of the colored scales represents the respective stress ratios.





4.2 Stresses by Set of Members

	A	B	С	D	E	F	G	H		9 - IPE 360 DIN 1025-5:1994
Set	Member	Location	S-Point	Load-	Stress	Stress [N	/mm ²]	Stress		Sigma Total
No.	No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio		Member No. 68, x: 3.125 m
1	Floor bear	m B-B (Membe	er No. 66-6	(9)						
	68	3.125	6	RC1	Sigma Total	206.4	218.2	0.95		
	66	0.000	13	RC1	Sigma N	34.7	218.2	0.16		
	69	3.125	10	RC1	Sigma M-y	182.6	218.2	0.84		
	68	3.125	6	RC1	Sigma M-z	3.4	218.2	0.02		
	66	6.250	13	RC1	Tau Total	-41.5	126.0	0.33	=	
	69	0.000	13	RC1	Tau V	41.5	126.0	0.33		
	66	3.125	1	RC1	Tau M-T	-4.2	126.0	0.03		
	68	3.125	6	RC1	Sigma-eqv	206.4	218.2	0.95		1 2 3 4 5
2	Floor hear	n A-A (Membe	ar No. 71-7	740						the second s
-	73	3.125	6	RC1	Sigma Total	187.6	218.2	0.86		
	71	3.125	1	RC1	Sigma N	-0.6	218.2	0.00		-170.8 -170.74-172.5
	71	3.125	10	RC1	Sigma M-y	184.8	218.2	0.85	11	-110.8 -110.1 -112.5
	73	2.813	6	RC1	Sigma M-z	4.0	218.2	0.02		34.42
	71	0.000	13	RC1	Tau Total	42.0	126.0	0.33		13
	71	6.250	13	RC1	Tau V	-41.9	126.0	0.33		
	71	3.125	1	RC1	Tau M-T	-4.3	126.0	0.03		206.4 204.8 204.8
	73	3.125	6	RC1	Sigma-eqv	187.6	218.2	0.86		
3		am E-E (Memb								5 7 18 9 10
	8	3.011	10	RC1	Sigma Total	-162.3	327.3	0.50		
	7	3.262	1	RC1	Sigma N	-8.9	218.2	0.04		z
	8	3.011	5	RC1	Sigma M-y	147.9	327.3	0.45		
	2	0.000	10	RC1	Sigma M-z	30.5	218.2	0.14		
	8	3.011	13	RC1	Tau Total	-17.7	189.0	0.09		
	2	0.000	13	RC1	Tau V	17.0	126.0	0.13		
	6	0.000	10	RC1	Tau M-T	7.0	126.0	0.06		
_	8	3.011	10	RC1	Sigma-eqv	162.3	327.3	0.50		
4	Column 1	1 (Member No	o. 31,32)							
	32	3.000	10	RC1	Sigma Total	-155.3	218.2	0.71		
	31	0.000	1	RC1	Sigma N	-16.1	218.2	0.07		Min : -172.5 N/mm ² (5)
	32	3.000	5	RC1	Sigma M-y	148.6	218.2	0.68	-	Max: 206.4 N/mm ² (6)

Figure 4.3: Window 2.2 Stresses by Set of Members

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

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 type.

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



		B	С	D	E	F	G	1 - HE A 400 DIN 1025-3:1994
Member	Location	S-Point	Load-	Stress	Stress [N/mm ²]	Stress	E Sigma Total
No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio	Member No. 1, x: 6.000 m
1	Cross-section	No. 1 - H	E A 400 I	DIN 1025-3:1994				
	6.000	6	RC1	Sigma Total	-92.7	218.2	0.42	
	0.000	1	RC1	Sigma N	-8.2	218.2	0.04	
	6.000	5	RC1	Sigma M-y	88.5	218.2	0.41	
	0.000	10	RC1	Sigma M-z	26.9	218.2	0.12	
	6.000	13	RC1	Tau Total	-16.1	126.0	0.13	
	2.100	13	RC1	Tau V	-15.7	126.0	0.12	84.43 84.88 85.21 85.81
	6.000	10	RC1	Tau M-T	1.3	126.0	0.01	
	6.000	6	RC1	Sigma-eqv	92.7	218.2	0.42	64.17
								1 2/3 4 5
2				DIN 1025-3:1994				
	0.000	6	RC1	Sigma Total	-99.3	218.2	0.46	
	0.000	1	RC1	Sigma N	-8.2	218.2	0.04	
	6.000	10	RC1	Sigma M-y	90.0	218.2	0.41	
	0.000	10	RC1	Sigma M-z	30.5	218.2	0.14	
	0.000	13	RC1	Tau Total	17.0	126.0	0.14	-344
	0.000	13	RC1	Tau V	17.0	126.0	0.13	-3144 Y
	6.000	1	RC1	Tau M-T	-2.1	126.0	0.02	
	0.000	6	RC1	Sigma-eqv	99.3	218.2	0.46	
	_							
3				/8/14/0 2 - IS 360/170/8/1	North Contraction of the Contrac			
	0.000	6	RC1	Sigma Total	-154.9	327.3	0.47	6 / /8/9 10
	3.011	1	RC1	Sigma N	-8.6	327.3	0.03	-71.05 🔻
	0.000	5	RC1	Sigma M-y	145.5	327.3	0.44	2
	0.000	10	RC1	Sigma M-z	3.5	327.3	0.01	92.69 92.09 91.76 91.28
	0.000	13	RC1	Tau Total	16.1	189.0	0.09	
	3.011	13	RC1	Tau V	14.2	189.0	0.08	
	0.000	10	RC1	Tau M-T	4.8	189.0	0.03	
	0.000	6	RC1	Sigma-eqv	155.1	327.3	0.47	
4	Cross-section	No. 17-	PE 360 L	DIN 1025-5:1994				
	0.000	6	RC1	Sigma Total	-95.2	218.2	0.44	
	0.000	1	RC1	Sigma N	-8.8	218.2	0.04	Min : -92.7 N/mm ² (6)
	0.000	5	RC1	Sigma M-y	84.1	218.2	0.39	

4.3 Stresses by Member

Figure 4.4: Window 2.3 Stresses by Member

This results window presents the maximum ratios for different stress types sorted by member numbers. The columns are described in detail in chapter 4.1 on page 29.

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



4.4 Stresses by x-Location

	A	B	С	D	E	F	G	1 - HE A 400 DIN 1025-3:1994
Member		S-Point	Load-	Stress	Stress [N/	/mm ²]	Stress	Sigma Total
No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio	Member No. 1, x: 0.000 m
1	Cross-section	No. 1 - H	E A 400 0	DIN 1025-3:1994				
	0.000	1	RC1	Sigma Total	-91.5	218.2	0.42	
	0.000	1	RC1	Sigma N	-8.2	218.2	0.04	
	0.000	10	RC1	Sigma M-y	69.5	218.2	0.32	
	0.000	10	RC1	Sigma M-z	26.9	218.2	0.12	
	0.000	13	RC1	Tau Total	-15.7	126.0	0.12	
	0.000	13	RC1	Tau V	-15.6	126.0	0.12	
	0.000	10	RC1	Tau M-T	0.3	126.0	0.00	
	0.000	1	RC1	Sigma-eqv	91.5	218.2	0.42	1 2 3 4 5
1			E 4 400 L					ann the former
	Cross-section 1.000		E A 400 [L RC1	DIN 1025-3:1994	-57.4	218.2	0.26	
	1.000	1	RC1	Sigma Total	-57.4	218.2	0.03	
	1.000	1	RC1	Sigma N	43.2	218.2	0.03	-91.50 -80.42 -78.69 -82.16 -81.33
	1.000	10	RC1	Sigma M-y	43.2	218.2	0.20	01.55
	1.000	13	RC1	Sigma M-z Tau Total	-15.7	126.0	0.07	
								-8 20
	1.000	13	RC1	Tau V	-15.7	126.0	0.12	A 92
	1.000	1	RC1	Tau M-T	-0.1	126.0	0.00	65.76 62.29 64.07 75.14
	1.000	1	RC1	Sigma-eqv	57.4	218.2	0.26	
1	Cross-section	No. 1 - H	E A 400 I D	DIN 1025-3:1994				A Company and A
	1.200	1	RC1	Sigma Total	-50.8	218.2	0.23	6 7 8 9 10
	1.200	1	RC1	Sigma N	-7.2	218.2	0.03	The second se
	1.200	10	RC1	Sigma M-y	37.9	218.2	0.17	
	1.200	10	RC1	Sigma M-z	12.3	218.2	0.06	
	1.200	13	RC1	Tau Total	-15.8	126.0	0.13	
	1.200	13	RC1	Tau V	-15.7	126.0	0.12	
	1.200	1	RC1	Tau M-T	-0.1	126.0	0.00	
	1.200	1	RC1	Sigma-eqv	50.8	218.2	0.23	
1								
- (DIN 1025-3:1994	41.0	210.2	0.10	
	1.500	1	RC1	Sigma Total	-41.0	218.2	0.19	
	1.500	1	RC1	Sigma N	-7.0	218.2	0.03	Min : -91.5 N/mm ² (1)
	1.500	10	RC1	Sigma M-y	30.0	218.2	0.14	 Max : 75.1 N/mm² (10)

Figure 4.5: Window 2.4 Stresses by x-Location

This results window lists the maximum stresses for each member at all locations **x** resulting from the division points defined in RSTAB:

- Start and end node
- Dividing points according to possibly defined member division (see RSTAB table 1.6)
- Member division according to specification for member results (RSTAB dialog box *Calculation Parameters*, tab *Global Register Parameters*)
- Extreme values of internal forces



4.5 Stresses at Every Stress Point

	A	В	С	D	E	F	G	2 - 3: IS 360/170/8/14/0 - IS 500/170/8/14/0
Member	Location	S-Point	Load-	Stress	Stress []	1/mm ²]	Stress	Sigma Total
No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio	Member No. 18, x: 1.004 m
	1.004	1	RC1	Sigma-eqv	193.6	327.3	0.59	
18	Cross-section	No. 2 - 19	360/170/	/8/14/0 3 - IS 500/170/	/8/14/0			
	1.004	2	RC1	Sigma Total	190.3	327.3	0.58	
	1.004	2	RC1	Sigma N	-11.5	327.3	0.04	
	1.004	2	RC1	Sigma M-y	199.5	327.3	0.61	
	1.004	2	RC1	Sigma M-z	0.2	327.3	0.00	192.6 400.0 400.0
	1.004	2	RC1	Tau Total	-9.1	189.0	0.05	193.6 190.3 188.6 178.4
	1.004	2	RC1	Tau V	-5.5	189.0	0.03	
	1.004	2	RC1	Tau M-T	-5.6	189.0	0.03	1 22 =
	1.004	2	RC1	Sigma-eqv	190.8	327.3	0.58	the second s
18				/8/14/0 3 - IS 500/170/				
	1.004	3	RC1	Sigma Total	190.1	327.3	0.58	
	1.004	3	RC1	Sigma N	-11.5	327.3	0.04	
	1.004	3	RC1	Sigma M-y	199.5	327.3	0.61	
	1.004	3	RC1	Sigma M-z	0.0	327.3	0.00	-1164
	1.004	3	RC1	Tau Total	-9.4	189.0	0.05	
	1.004	3	RC1	Tau V	-5.8	189.0	0.03	
	1.004	3	RC1	Tau M-T	-5.6	189.0	0.03	
	1.004	3	RC1	Sigma-eqv	190.7	327.3	0.58	
18	Cross-section	No 2-19	360/170/	/8/14/0 3 - IS 500/170/	/8/14/0			6 78 10
	1.004	4	RC1	Sigma Total	189.9	327.3	0.58	z
	1.004	4	RC1	Sigma N	-11.5	327.3	0.04	-195.1 -205.4 -209.0 -212.3
	1.004	4	RC1	Sigma M-y	199.5	327.3	0.61	-205.4 -209.0 -212.3
	1.004	4	RC1	Sigma M-z	-0.2	327.3	0.00	
	1.004	4	RC1	Tau Total	-9.1	189.0	0.05	
	1.004	4	RC1	Tau V	-5.4	189.0	0.03	
	1.004	4	RC1	Tau M-T	-5.6	189.0	0.03	
	1.004	4	RC1	Sigma-eqv	190.5	327.3	0.58	
18				(8/14/0 3 - IS 500/170)		227.2	0.57	Min : -212.3 N/mm ² (10)
	1.004	5	RC1	Sigma Total	186.6	327.3	0.57	 Max: 193.6 N/mm² (1)

Figure 4.6: Window 2.5 Stresses at Every Stress Point



Details...

This results window is inactive by default because STEEL determines the maximum stresses, and thus the governing stress points, automatically. In addition, the windows 2.1 to 2.4 allow you to check the results of each stress point graphically and numerically by clicking the button [Extended Stress Diagram] (see chapter 5.2, page 42).

If a results evaluation by stress points is required, you can display the window by opening the dialog box Details (see Figure 3.1, page 19). The dialog box can be opened in every window by clicking [Details].

The stresses in window 2.5 are listed for each member according to *Location x* and *S*-Point No. The different columns of this window are described in detail in chapter 4.1 on page 29.



4.6 Governing Internal Forces by Member

	A	B	C	D	E	F	G	H	l.
Member	Location	Load-		Forces [kN]			oments [kNm]		
No.	x [m]	ing	N	Vy	Vz	MT	My	Mz	
1	6.000	RC1	-54.69	0.49	-60.66	-0.08	-204.52	-0.44	
2	0.000	RC1	-130.16	-5.20	61.36	-0.03	-164.31	-11.42	
3	0.000	RC1	-64.92	-0.19	48.68	0.10	-203.37	-0.35	
4	0.000	RC1	-63.77	-0.35	36.39	-0.08	-76.05	-0.28	
5	4.705	RC1	-60.93	0.33	1.79	-0.10	77.49	0.34	
6	0.000	RC1	-61.06	-0.59	4.76	0.21	75.45	-0.96	
7	3.262	RC1	-64.72	0.12	-36.68	0.10	-78.63	-0.16	
8	3.011	RC1	-66.13	-0.10	-49.13	-0.05	-207.85	0.90	
11	6.000	RC1	-89.74	0.46	-91.77	-0.14	-336.66	-0.29	
12	6.000	RC1	-97.41	-0.04	59.38	-0.08	382.86	0.27	
13	0.000	RC1	-98.28	0.00	70.66	-0.08	-336.72	0.13	
14	0.000	RC1	-96.38	-0.16	59.05	-0.06	-141.34	-0.27	
15	5.378	RC1	-84.16	0.12	1.31	-0.06	156.32	0.56	
16	0.941	RC1	-75.95	-0.26	-0.15	0.06	157.30	0.16	
17	3.262	RC1	-72.18	-0.10	-68.52	0.08	143.04	0.01	
18	3.011	RC1	-75.17	-0.15	-90.81	-0.10	-383.03	0.78	
21	6.000	RC1	-111.17	-0.59	+102.27	0.20	372.57	0.49	
22	0.000	RC1	-178.15	-0.25	136.96	0.02	-402.74	1.58	
23	0.000	RC1	-110.48	-0.07	89.50	0.26	-372.70	-0.24	
24	0.000	RC1	-107.56	0.04	67.69	0.11	135.84	0.18	
25	5.333	RC1	-95.05	-0.03	-1.18	0.02	154.34	-0.35	
26	0.896	RC1	-107.66	0.14	-0.88	-0.05	152.18	-0.02	
27	3.262	RC1	-134.86	0.02	-69.54	0.02	-161.17	0.10	
28	3.011	RC1	-138.14	0.24	-91.39	0.00	-404.22	-1.19	
31	3.000	RC1	-222.37	-2.10	72.81	-0.09	187.71	0.76	
32	3.000	RC1	-98.36	0.18	+176.74	0.14	-343.44	0.30	
33	0.000	RC1	-216.25	5.02	-0.09	0.00	0.22	3.17	
34	0.000	RC1	-1.18	0.23	-0.04	0.00	0.15	0.83	
35	0.000	RC1	-216.46	5.05	-0.10	0.00	0.35	3.30	
36	0.000	RC1	-1.36	0.20	-0.06	0.00	0.23	0.81	
37	0.000	RC1	-216.24	4.93	-0.09	0.00	0.20	3.32	
38	0.000	RC1	-1.18	0.28	-0.04	0.00	0.15	1.02	
39	3.000	RC1	-222.76	-0.32	-63.94	0.04	-194.07	-0.75	
40	3.000	RC1	-98.56	-0.08	178.80	-0.05	348.43	-0.13	

Figure 4.7: Window 3.1 Governing Internal Forces by Member

Details...

This window displays for each member the governing internal forces, whose stresses result in maximum stress ratios. They refer to the maximum equivalent stress σ_{eqv} by default. In the dialog box *Details* (see Figure 3.1, page 19), which can be accessed by using the [Details] button, you can set another stress type.

If you Calculate stresses from single load cases in RC and then combine stresses according to RC criteria (see Figure 3.1, page 19), you disable the direct evaluation of the RC result rows of RSTAB table 4.1 *Members - Internal Forces*. Instead, compression, tension, and shear stresses determined for each load case are summed up according to the RC superposition criterion. The equivalent stress σ_{eqv} represents an exception because it is determined by the RC components of σ_{total} and τ_{total} . Therefore, the governing internal forces for result combinations are not immediately transparent.

Location **x**

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

Loading

Column B displays the numbers of the load case or the load and result combinations whose internal forces produce maximum stresses.

Forces / Moments

For each member, the normal and shear forces as well as the torsional and bending moments producing maximum stresses are displayed.



4.7 Governing Internal Forces by Set of Members

3.2 Governing	Internal Forces	s by Set of Members
Siz Governing	ancennari oree.	by bee of members

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

This window shows the internal forces that result in the maximum stress ratios for each set of members.



4.8 Parts List by Member

Finally, STEEL provides a summary of all cross-sections that are included in the design case.

Cross-Section Description	Number Members	Length	Total Length	Surface Area	Volume	Cross-Section	Mass	Total Mass
Description	Members							rotar made
	Monibora	[m]	[m]	[m ²]	[m ³]	[kg/m]	[kg]	[t]
1 - HE A 400 DIN 1025-3:1994	6	6.00	36.00	68.76	0.57	124.82	748.89	4.493
2 - IS 360/170/8/14/0 3 - IS 500/170/8/14/0	8	3.01	24.09		0.19	62.61		1.508
17 - IPE 360 DIN 1025-5:1994	8				0.19			1.490
17 - IPE 360 DIN 1025-5:1994	8	6.27	50.19	67.91	0.36	57.07	358.05	2.864
1 - HE A 400 DIN 1025-3:1994	4	3.00	12.00	22.92	0.19	124.82	374.45	1.498
10 - HE A 140 DIN 1025-3:1994	3	3.00	9.00	7.15	0.03	24.65	73.95	0.222
10 - HE A 140 DIN 1025-3:1994	2	3.55	7.09	5.63	0.02	24.65	87.41	0.175
10 - HE A 140 DIN 1025-3:1994	1	4.09	4.09	3.25	0.01	24.65	100.91	0.101
15 - HE A 200 DIN 1025-3:1994	4	3.00	12.00	13.68	0.06	42.23	126.70	0.507
16 - Rectangle 200/200	3	3.00	9.00	7.20	0.36	100.00	300.00	0.900
6 - HE A 160 DIN 1025-3:1994	2	3.55	7.09	6.43	0.03	30.46	108.00	0.216
6 - HE A 160 DIN 1025-3:1994	1	4.09	4.09	3.71	0.02	30.46	124.70	0.125
7 - HE A 120 DIN 1025-3:1994	4	6.27	25.10	16.99	0.06	19.86	124.60	0.498
9 - IPE 360 DIN 1025-5:1994	8	6.25	50.00	67.65	0.36	57.07	356.68	2.853
6 - HE A 160 DIN 1025-3:1994	2	6.55	13.09	11.86	0.05	30.46	199.38	0.399
6 - HE A 160 DIN 1025-3:1994	1	7.09	7.09	6.43	0.03	30.46	216.07	0.216
12 - QRO 80x4 DIN 59410:1974	25	5.00	125.00	39.13	0.15	9.42	47.10	1.178
13 - RD 24 DIN 1013-1	4				0.01	3.55		0.111
13 - RD 24 DIN 1013-1	8				0.03	3.55		0.228
								19.581
	17 - IPE 360 DIN 1025-5:1994 1 - HE A 400 DIN 1025-3:1994 10 - HE A 140 DIN 1025-3:1994 10 - HE A 140 DIN 1025-3:1994 10 - HE A 140 DIN 1025-3:1994 15 - HE A 1201 DIN 1025-3:1994 16 - Rectangle 200/200 6 - HE A 160 DIN 1025-3:1994 6 - HE A 160 DIN 1025-3:1994 9 - IPE 360 DIN 1025-3:1994 6 - HE A 160 DIN 1025-3:1994 12 - QRO 804 DIN 5941(0:1974)	17 - IPE 360 I DIN 1025-5:1994 8 17 - IPE 360 I DIN 1025-5:1994 8 17 - IPE 360 I DIN 1025-3:1994 4 10 - HE A 140 I DIN 1025-3:1994 3 10 - HE A 140 I DIN 1025-3:1994 1 10 - HE A 140 I DIN 1025-3:1994 1 11 - HE A 400 I DIN 1025-3:1994 1 15 - HE A 140 I DIN 1025-3:1994 4 16 - Rectangle 200/200 3 6 - HE A 160 I DIN 1025-3:1994 1 7 - HE A 120 I DIN 1025-3:1994 1 7 - HE A 120 I DIN 1025-3:1994 4 6 - HE A 160 I DIN 1025-3:1994 8 6 - HE A 160 I DIN 1025-3:1994 2 6 - HE A 160 I DIN 1025-3:1994 1 12 - QRO 80x4 DIN 50410:1974 25 13 - RD 24 I DIN 103-1 4	17 - IPE 360 DIN 1025-5:1994 8 3.26 17 - IPE 360 DIN 1025-5:1994 8 6.27 1 - HE A400 [DIN 1025-3:1994 4 3.00 10 - HE A 140 DIN 1025-3:1994 3 3.00 10 - HE A 140 DIN 1025-3:1994 2 3.55 10 - HE A 140 DIN 1025-3:1994 1 4.09 15 - HE A 200 DIN 1025-3:1994 4 3.00 16 - Rectangle 200/200 3 3.00 6 - HE A 160 DIN 1025-3:1994 2 3.55 6 - HE A 160 DIN 1025-3:1994 2 3.55 7 - HE A 120 DIN 1025-3:1994 4 6.27 9 - IPE 360 DIN 1025-3:1994 4 6.27 9 - IPE 360 DIN 1025-3:1994 2 6.55 6 - HE A 160 DIN 1025-3:1994 1 7.09 12 - CRO 80x4 DIN 5340:1974 25 5.00 13 - RO 24 DIN 1013-1 4 7.81	17 - IPE 360 DIN 1025-5:1994 8 3.26 26 10 17 - IPE 360 DIN 1025-5:1994 8 6.27 50.19 17 - IPE 360 DIN 1025-5:1994 8 6.27 50.19 10 - HE A 140 DIN 1025-3:1994 4 3.00 12.00 10 - HE A 140 DIN 1025-3:1994 2 3.55 7.09 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 15 - HE A 200 DIN 1025-3:1994 4 3.00 12.00 16 - Rectangle 200/200 3 3.00 9.00 6 - HE A 160 DIN 1025-3:1994 2 3.55 7.09 6 - HE A 160 DIN 1025-3:1994 4 6.27 25.10 9 - IPE 360 DIN 1025-3:1994 4 6.27 25.10 9 - IPE 360 DIN 1025-3:1994 4 6.25 50.00 6 - HE A 160 DIN 1025-3:1994 2 6.55 13.09 9 - IPE 360 DIN 1025-3:1994 1 7.09 7.09 12 - GN 80x4 DIN 53410:1974 25 5.00 125.00 13 - RD 24 DIN 1013-1 4 7.81	17 - IPE 360 DIN 1025-5:1994 8 3.26 26.10 35.31 17 - IPE 360 DIN 1025-5:1994 8 6.27 50.19 67.91 1 - HE A400 DIN 1025-3:1994 4 3.00 12.00 22.92 10 - HE A 140 DIN 1025-3:1994 3 3.00 9.00 7.15 10 - HE A 140 DIN 1025-3:1994 2 3.55 7.09 5.63 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 3.26 15 - HE A 200 DIN 1025-3:1994 4 3.00 9.00 7.20 16 - Rectangle 200/200 3 3.00 9.00 7.20 6 - HE A 160 DIN 1025-3:1994 2 3.55 7.09 6.43 6 - HE A 160 DIN 1025-3:1994 4 6.27 25.10 16.99 9 - IPE 360 DIN 1025-3:1994 4 6.25 50.00 67.65 6 - HE A 160 DIN 1025-3:1994 2 6.55 13.09 11.86 6 - HE A 160 DIN 1025-3:1994 1 7.09 6.43 12 - ORO 80x4 DIN 5940:1974 25	17 - IPE 360 DIN 1025-5:1994 8 3.26 26.10 35.31 0.19 17 - IPE 360 DIN 1025-5:1994 8 6.27 50.19 67.91 0.36 1 - HE A400 DIN 1025-3:1994 4 3.00 12.00 22.92 0.19 10 - HE A 140 DIN 1025-3:1994 4 3.00 9.00 7.15 0.03 10 - HE A 140 DIN 1025-3:1994 2 3.55 7.09 5.63 0.02 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 3.25 0.01 15 - HE A 200 DIN 1025-3:1994 1 4.09 4.09 3.26 0.01 16 - Rectangle 200/200 3 3.00 9.00 7.20 0.36 6 - HE A 160 DIN 1025-3:1994 2 3.55 7.09 6.43 0.03 6 - HE A 160 DIN 1025-3:1994 4 6.27 25.10 16.99 0.06 9 - IPE 360 DIN 1025-3:1994 4 6.27 25.10 16.99 0.06 9 - IPE 360 DIN 1025-3:1994 2 6.55 13.09 1.86 0.05 6 - HE A 160 DIN 1025-3:1994 1 7.09 6.43 0.03 7 - HE A 120 DIN 1025-3:1994 1 7.09 6.43 0.03 6 - HE A 160 DI	17 - IPE 360 DIN 1025-5:1994 8 3.26 26.10 35.31 0.19 57.07 17 - IPE 360 DIN 1025-5:1994 8 6.27 50.19 67.91 0.36 57.07 1 - HE A400 DIN 1025-3:1994 4 3.00 12.00 22.92 0.19 124.82 10 - HE A 140 DIN 1025-3:1994 4 3.00 9.00 7.15 0.03 24.65 10 - HE A 140 DIN 1025-3:1994 2 3.55 7.09 5.63 0.02 24.65 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 3.25 0.01 24.65 15 - HE A 200 DIN 1025-3:1994 4 3.00 12.00 13.68 0.06 42.23 16 - Rectangle 200/200 3 3.00 9.00 7.20 0.36 10000 6 - HE A 160 DIN 1025-3:1994 2 3.55 7.09 6.43 0.03 3.046 6 - HE A 160 DIN 1025-3:1994 4 6.27 25.10 15.99 0.06 19.86 9 - IPE 360 DIN 1025-3:1994	17 - IPE 360 DIN 1025-5:1994 8 3.26 26.10 35.31 0.19 57.07 186.19 17 - IPE 360 DIN 1025-5:1994 8 6.27 50.19 67.91 0.36 57.07 388.05 11 - HE A400 DIN 1025-3:1994 4 3.00 12.00 22.23 0.19 124.82 374.45 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 3.00 12.00 22.465 173.95 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 3.25 0.01 2.465 170.95 10 - HE A 140 DIN 1025-3:1994 1 4.09 4.09 3.25 0.01 2.465 10.91 15 - HE A 200 DIN 1025-3:1994 1 4.09 4.09 3.20 3.06 9.06 42.23 126.70 16 - Rectangle 200/200 3 3.00 9.00 7.20 0.36 100.00 30.00 9.006 142.60 3.04.6 108.00 6.43 0.03 30.46 128.70 144.70 7.46 16.99



By default, this list contains only the designed members. If you need a parts list with all members of the model, select the corresponding option in the *Details* dialog box (see Figure 3.1, page 19).

Part No.

The program assigns automatically item numbers for similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number Members

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

Length

This column displays the respective length of an individual member.

Total Length

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

Surface Area



Details..

For each part, the program indicates the surface area related to the total length. The surface area is determined from the *Surface* of the cross-sections that can be seen in windows 1.3 and 2.1 to 2.5 in the cross-section information (see Figure 2.10, page 18).



Volume

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

Cross-Section Mass

The cross-section mass represents the unit mass of the cross-section in relation to the length of one meter. For tapered cross-sections, the program averages both cross-section properties.

Mass

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

Total Mass

The final column indicates the total mass of each part.

Sum

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

4.9 Parts List by Set of Members

	A	B	С	D	E	F	G	H	
Part	Set of Members	Number	Length	Total Length	Surface Area	Volume	Cross-Section	Mass	Total Mass
No.	Description	Set	[m]	[m]	[m ²]	[m ³]	[kg/m]	[kg]	[t]
1	Floor beam B-B	1	25.00	25.00	33.83	0.18	57.07	1426.74	1.42
2	Floor beam A-A	1	25.00			0.18	57.07		1.42
3	Ridge beam E-E	1	37.10		57.90	0.38	79.88	2963.35	2.90
4	Column 1-1	1	6.00	6.00	11.46	0.10	124.82	748.89	0.7
5	Column 2-2	1	6.55			0.02		161.35	0.1
6	Column 3-3	1	7.09	7.09	5.63	0.02	24.65	174.86	0.1
7	Column 4-4	1	6.55	6.55	5.20	0.02	24.65	161.35	0.1
8	Column 5-5	1	6.00	6.00	11.46	0.10	124.82	748.89	0.7
Sum		8		119.28	164.50	1.00			7.8

Figure 4.10: Window 4.2 Parts List by Set of Members

The last results window is displayed when you have selected at least one set of members for the 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. When different crosssections are used in the set of members, the program averages the surface area, the volume, and the cross-section mass.



5. **Results Evaluation**

To evaluate the results, use the buttons at the end of the table.

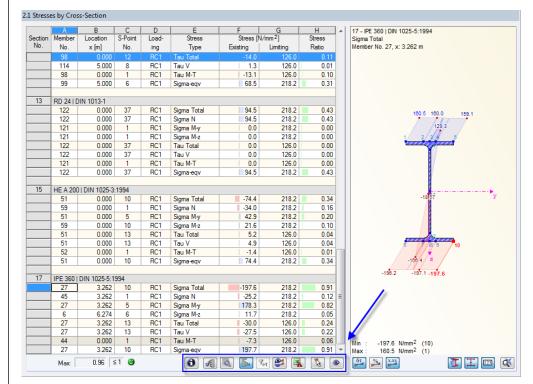


Figure 5.1: Buttons for results evaluation in the windows

The buttons are reserved for the following functions:

Button	Description	Function
•	Cross-section info	Opens the dialog box <i>Info about Cross-section</i> → Figure 2.10, page 18
	Stress selection	Opens the dialog box <i>Stresses</i> - <i>Filter</i> → Chapter 5.1, page 40
	Extended stress diagram	Opens the dialog box Cross-Section Values and Stress Diagram \rightarrow Chapter 5.2, page 42
	Show color bars	Turns on and off the colored relation scales in the results tables
% >1	Exceeding	Displays only the rows where the ratio is more than 1, and thus the design is failed
2	Result diagrams	Opens the window <i>Result Diagram on Member</i> → Chapter 5.4, page 46
	Excel export	Opens the dialog box <i>Export - MS Excel</i> → Chapter 7.4.2, page 59
₹ 3	Member selection	Enables the graphical selection of a member to display its results in the module window
۲	View mode	Jumps to the RSTAB work window to change the view

Table 5.1: Buttons of results windows 2.1 to 2.5



5.1 Selection of Stresses

The following stress types are displayed by default in the results windows:

- Normal stress σ_{total}
- Shear stress τ_{total}
- Equivalent stress σ_{eqv}



Click [Select Stresses to Show] to activate further stress components. In this way, you can check the components affecting the total stress. The button is at the bottom of the table (see Figure 5.1).

The dialog box Stresses - Filter appears where you can select the stress types.

Stresses - Filter	×
To Show Sigma Total Sigma N-y Sigma N-y Sigma N-M-y Sigma N+M-y Sigma N+M-z Sigma N+M-z Sigma N+M-z Sigma Central Sigma Central Sigma Central	Tau V-y Tau V-z ▼Tau V Tau M-T St.Venant Tau M-T Bredt ▼Tau M-T V Sigma-eqv
	OK Cancel

Figure 5.2: Dialog box Stresses - Filter

The stress types are described in Table 3.1 and Table 3.2 on page 22 to 24.

The buttons in the dialog section To Show facilitate the selection of the stress types.

Button	Description	Function
	Select All	Ticks all stress type check boxes
	Deselect All	Deactivates all check boxes

Table 5.2: Buttons in the dialog box Stresses - Filter



The analysis is carried out for each single stress point. Therefore, the components of the maximum stresses for a combined calculation (for example σ_{total}) must <u>not</u> be summed up: Often, the maximum stresses occur at different stress points. You have to superimpose the stress components that are available in the same stress point.



The specifications in the dialog box *Stresses - Filter* affect also the printout report: The report displays only stress types shown in the results windows.



5.2 Results on Cross-section

The results from the table are visualized by a dynamic stress graphic. This graphic shows the stress distribution on the cross-section for the current x-location of the selected stress type. If a different x-location or stress type is selected by mouse-click, this is shown in the graphic. The governing stress point is highlighted in red.



The graphic can display stresses as well as stress ratios.

7 - HE A 120 D	IN 1025-3:	1994			
Sigma Total Member No. 61,	x: 6 274 n				
	165.9	154.4	148.4	141.	0
	/		1/		
	· .		15		
-		1111	7		
		~			
		0 65			
		3		У	
		/ 💈			
				~	
		7.15/8	9	10	
	Ĩ	//↓/			
		/z			
-140.0	-147.4	153.3	-164.8		
	N/mm ² N/mm ²				
	_	0		-	
6T % XX	2			123	9

Figure 5.3: Diagram of normal stresses on cross-section

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

Button	Description	Function
6τ	Stress diagram	Displays or hides the stress points
%	Stress ratio	Displays or hides the ratios
X.XX	Values	Switches on and off the result values
	Cross-section outlines	Displays or hides the cross-section outlines
I	Stress points	Displays or hides the stress points
123	Numbering	Switches on and off the numbering of stress points
X	Show all graphic	Resets the full view of the results graphic

Table 5.3: Buttons of graphic in results windows 2.1 to 2.5



Q)

9

Use the wheel button to maximize or minimize the display. Use the drag-and-drop function to move the stress graphic. The button [Show All Graphic] resets the graphic's full view.

Extended Diagram of stresses and ratios

To evaluate every stress point specifically, click [Extended Stress Diagram]. The dialog box *Cross-Section Values and Stress Diagram* opens.

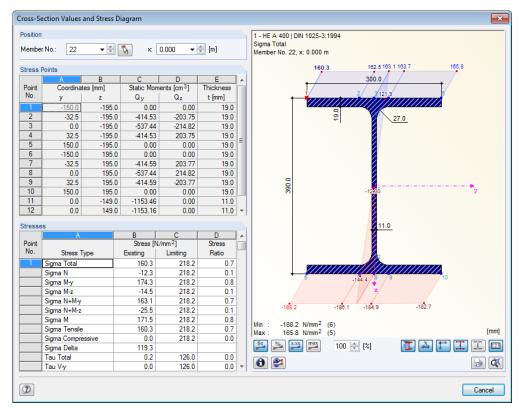


Figure 5.4: Dialog box Cross-Section Values and Stress Diagram

The current *Member No*. and the *x*-location on the member are already preset in the dialog section *Position*. To select another member or *x*-location, use the list.

The dialog section *Stress Points* lists all stress points of the cross-section. The two *Coordinates* columns show the respective centroidal distances e_y and e_z . The *Static Moments* columns display the corresponding first moments of area Q_y and Q_z . The final column indicates the *Thickness t* of the cross-section part which is required to determine the shear stresses.

In the *Stresses* dialog section, all stresses are displayed for the stress point that is currently selected in the dialog section above. In this dialog, you can also select a stress type by mouseclick to display its diagrams in the graphic.

Most of the buttons below the graphic are identical with the buttons in the results windows (see Table 5.3, page 41). As usual, they are described by *ScreenTips*. To print the current stress graphic, click [Print] (see chapter 6.2.1, page 49).





5.3 Results in RSTAB Model

To evaluate the design results, you can also use the RSTAB work window.

RSTAB background graphic and view mode

The RSTAB work window in the background is useful when you want to find the position of a particular member in the model: The member selected in the STEEL results window is high-lighted in the selection color in the background graphic. Furthermore, an arrow indicates the member's x-location that is displayed in the active table row.

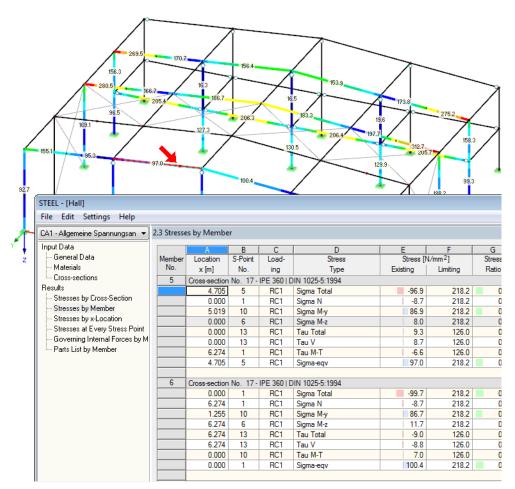


Figure 5.5: Indication of member and current *Location x* in RSTAB model

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

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





RSTAB work window

Graphics

The stresses and stress ratios can also be checked graphically in the RSTAB model. Click [Graphics] to exit the design module. Now, the design results are displayed in the RSTAB work window like the internal forces of a load case.

The *Results* navigator is aligned with the results from the add-on module STEEL. You can choose various stress types as well as the stress ratios in relation to the individual stress components.

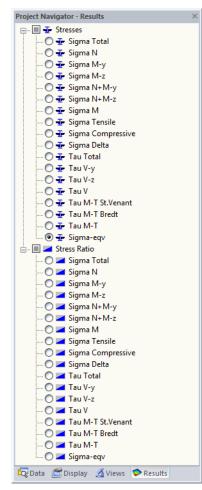
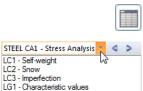


Figure 5.6: Results Navigator for STEEL

To turn the display of design results on or off, use the button [Show Results], as you know it from the display of internal forces in RSTAB. To display the result values, use the toolbar button [Show Result Values] to the right.

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

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



X.XX

٩

STEEL CA1 - Stress Analysis - Beams STEEL CA2 - Stress Analysis - Columns



The graphical representation of the results can be set in the *Display* navigator by clicking *Results* \rightarrow *Members*. Stresses and stress ratios are displayed *Two-Colored* by default.

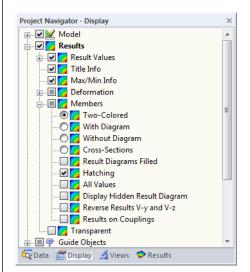


Figure 5.7: *Display* navigator: Results → Members

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

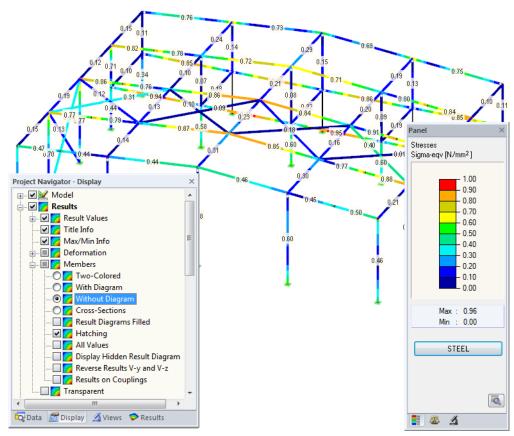


Figure 5.8: Stress ratios with display type Without Diagram

The graphics of stresses and stress ratios can be transferred to the printout report (see chapter 6.2, page 49).

STEEL

You can return to the add-on module by using the panel button [STEEL].

2

Ł



5.4 Result Diagrams

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

To do this, select the member (or set of members) in the STEEL results window by clicking in the table row of the member. Then open the dialog box *Result Diagram on Member* by clicking on the button shown on the left. You find the button below the table (see Figure 5.1, page 39).

The result diagrams are available in the RSTAB graphic. To display the diagrams, click

$\textbf{Results} \rightarrow \textbf{Result Diagrams for Selected Members}$

or use the button in the RSTAB toolbar shown on the left.

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

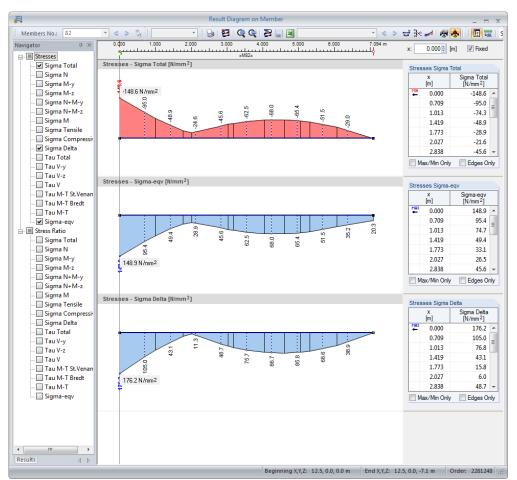
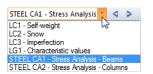


Figure 5.9: Dialog box Result Diagram on Member

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

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





5.5 Filter for Results

The STEEL results windows allow you to sort the results by various criteria. In addition, you can use the filter options described in chapter 9.7 of the RSTAB manual to evaluate the design results graphically.

You can use the option *Visibility* also for STEEL (see RSTAB manual, chapter 9.7.1), to filter the members for evaluation.

Filtering designs

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

View ightarrow Control Panel (Color Scale, Factors, Filter)

or use the toolbar button shown on the left.

The panel is described in the RSTAB manual, chapter 3.4.6. The filter settings for the results must be defined in the first panel tab (Color spectrum). As this register is not available for the two-colored results display, you have to set the display option *Colored* or *Cross-Sections* in the *Display* navigator first.

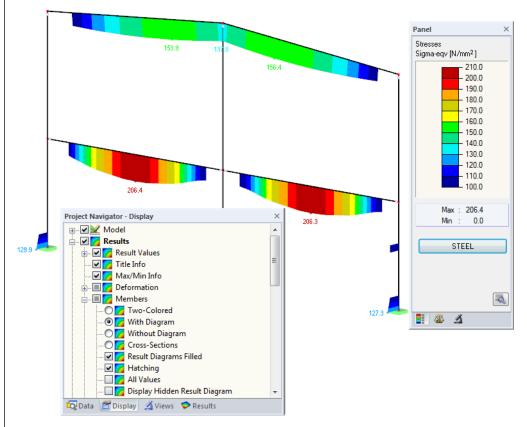


Figure 5.10: Filtering equivalent stresses with adjusted color spectrum

As the figure above shows, the color spectrum can be set in such a way that the equivalent stresses greater than 100 N/mm² are shown in a color range between blue and red. Furthermore, the color spectrum can also be adjusted in such a way that a color range covers exactly 10 N/mm², for example.



× 12





With the option *Display Hidden Result Diagram* in the *Display* navigator (under *Results* \rightarrow *Members*) you can display all stress diagrams that are not within this value spectrum. Those diagrams will be represented by dotted lines.

Filtering members

In the *Filter* tab of the control panel, we can specify the numbers of particular members to display their results exclusively, that means filtered. That function is described in detail in the RSTAB manual, chapter 9.7.3.

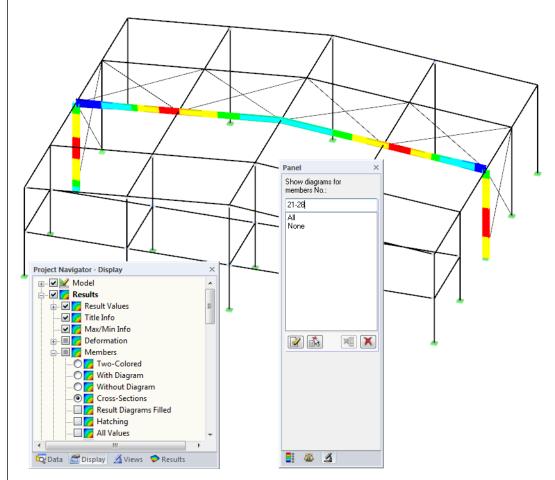


Figure 5.11: Member filter for normal stresses of a hall frame

In contrast to the visibility function, the model will be displayed completely in the graphic. The figure above shows the normal stresses of a hall frame. The remaining designed members are displayed in the model but are shown without stresses.

1



6. Printout

6.1 Printout Report

Similar to RSTAB, the program generates a printout report for the STEEL results to which graphics and descriptions can be added. In this printout report, you decide which data from the design module should appear in the final printout.

The printout report is described in the RSTAB manual. In particular, chapter 10.1.3.4 *Selecting Data of Add-on Modules* provides information concerning the selection of input and output data in add-on modules.

The printout report shows the stress types that are displayed in the STEEL results windows. Therefore, if you want to print, for example, the stresses due to axial force, activate the stresses σ_N for display. The selection of the stresses is described in chapter 5.1 on page 40.

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

6.2 Printing STEEL Graphics

6.2.1 Results on Cross-Section

To access the print function, use the dialog box *Cross-Section Values and Stress Diagram*. To open this dialog box, click [Extended Stress Diagram] at the end of the tables in the results window (see Figure 5.1, page 39).

In the dialog box *Cross-Section Values and Stress Diagram* (see Figure 5.4, page 42), you have to specify the member, the x-location, and the stress type whose diagram you want to print. Click the [Print] button in the bottom right corner of the dialog box to open the following printing dialog box.

Print Info About Cross-Section			×
Print Type Directly to a printer To a printout report: PR1 To the Clipboard 	Printout Selection Image: Graphics of cross-section Image: Axis Image: Dimensions Image: Stress points Image: Numbering Image: C/t-parts Image: Numbering	 ✓ Silhouette Shear center ✓ Stress diagram ✓ Values 	Stress points Info picture C/-parts Info picture Numbering
	Cross-section properties Cross-section properties Stresses in all stress points (e.g. Modules STEEL, TIM	BER)	
Show printout report on [OK]	Description of Graphics HE A 400 DIN 1025-3:1994		
			OK Cancel

Figure 6.1: Dialog box Print Info About Cross-Section

Print Type

In this dialog section, the common options from RSTAB are available for selection:

- Directly to a printer sends the current graphic to the printer.
- To a printout report inserts the graphic into the printout report.
- To the Clipboard provides the graphic for other applications.



1

Program STEEL © 2013 Dlubal Engineering Software



If several printout reports are available, you can select the number of the target report in the selection field.

Printout Selection

In this dialog section, you decide which details should appear in the print graphic and in the output table. The check boxes for *Graphics of cross-section* are self-explanatory. If you tick *Cross-section properties*, the properties will be printed as a table, to which you can add an *Info picture* in the margin. In the same way, you can integrate the properties of the *Stress points* and *c/t-parts* as well as the *Stresses in all stress points* in the printout.

If you want to import several graphics consecutively into the printout report, clear the check box *Show printout report on [OK*].

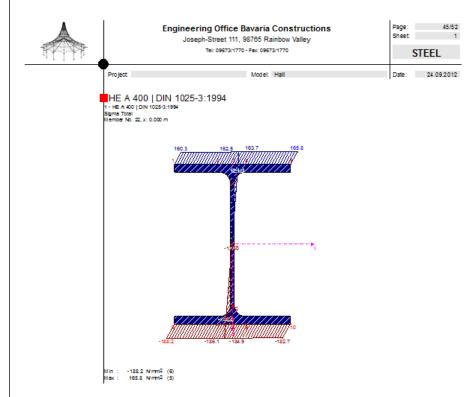


Figure 6.2: Stress graphic in printout report

6.2.2 Results in the RSTAB Model

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



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

To print the currently displayed graphic of the STEEL results directly, click the menu

$\textbf{File} \rightarrow \textbf{Print Graphic}$

or use the toolbar button shown on the left.

•	File Fi	dit Vie	w Inse	rt Calcu	late <u>R</u> esu	Its Tools
					<u>a</u> <u>B</u> (
					1 - 1 - 4 1 - 1 - 4 1 - 1 - 4 1 - 4 1 - - - - - - - - - - - - - - - - -	-3 -3 -4-

Figure 6.3: Button Print Graphic in RSTAB toolbar



The following dialog box opens:

Graphic Printout		×
General Options Color Spectrum		
Graphic Picture O Directly to a printer Image: Second secon	Window To Print © Current only More Mass print	Graphic Size As screen view Window filling To scale 1: 100 ▼
Graphic Picture Size and Rotation Image: Use whole page width Image: Use whole page height	Options Show results for selected a diagram Lock graphic picture (with	
Header of Graphic Picture STEEL - Members Sigma-eqv, CA1, Isometric		
		OK 🕞 Cancel

Figure 6.4: Dialog box Graphic Printout, tab General

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

A STEEL graphic that has been integrated in the printout report can be moved anywhere within the report by using the drag-and-drop function.

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

Graphic Printout General Options Color Spectrum				X
Coor spectrum Script Proportional Constant Factor: 1	Symbols Proportional Constant Factor: 1 ÷		Frame None Framed Title box	
Print Quality Standard (max 1000 x 1000 Pixels) Maximum (max 5000 x 5000 Pixels) User-defined Max number of pixels:		Color Carayscal Texts an All colore	d lines in black	
\mathfrak{D}			ОК	Cancel

Figure 6.5: Dialog box Graphic Printout, tab Options

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



6.2.3 Result Diagrams

In the dialog box *Result Diagram on Member* (see Figure 5.9, page 46), you can also transfer the design values by clicking [Print] or print them directly.

a	Result Diagram on Member	
Members No.: 22	- < > % · · · · · · · · · · · · · · · · · ·	
Navigator 4 ×	0.dpo 1.000 2.000 Print 0 4.000 5.000 6.000 m 	X:
Sigma Total	Stresses - Sigma N [N/mm ²]	St

Figure 6.6: Button Print in the dialog box Result Diagram on Member

The dialog box *Graphic Printout* opens (see Figure 6.4 and Figure 6.5), where you can adjust the settings before printing the result diagrams.



7. General Functions

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

7.1 Design Cases

Design cases allow you to group members for design: In this way you can, for example, combine groups of structural components or check members with particular design specifications (for example modified materials, calculation of result combinations, optimization).

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

To calculate a design case, you can also use the load case list in the toolbar.

Create a new design case

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

File ightarrow New Case.

The following dialog box appears:

	EL-Case
No.	Description
þ	General stress analysis of steel members
	OK Cancel

In this dialog box, enter a *No*. (that is still available) for the new design case. The corresponding *Description* will make the selection in the load case list easier.

When you click [OK], window 1.1 General Data opens where you can enter the design data.

Rename a design case

To change the description of a design case subsequently, use the STEEL menu and click

File \rightarrow Rename Case.

The following dialog box appears:

Rename S	TEEL-Case	X
No.	Description New Description	-
D		OK Cancel

Figure 7.2: Dialog box Rename STEEL-Case

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

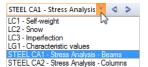


Figure 7.1: Dialog box New STEEL-Case



Copy a design case

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

```
File \rightarrow Copy Case.
```

The following dialog box appears:

	EEL-Case							
Copy from Case								
CA1 - G	ieneral stress analysis of steel members 🔹 🔻							
New Ca	se							
No.:	Description:							
3	Reduced Limit Stresses							
()	OK Cancel							
1	UK							

Figure 7.3: Dialog box Copy STEEL-Case

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

Delete a design case

To delete design cases, use the STEEL menu and click

```
File \rightarrow Delete Case.
```

The following dialog box appears:

No. Description 1 General stress analysis of steel members 2 New Description 3 Reduced Limit Stresses	Availab	le Cases	
2 New Description	No.	Description	-
· · · · · · · · · · · · · · · · · · ·	1	General stress analysis of steel members	
3 Reduced Limit Stresses	2	New Description	
		Reduced Limit Stresses	

Figure 7.4: Dialog box Delete Case

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





7.2 Cross-Section Optimization

The design module offers you the possibility to optimize overloaded or little utilized crosssections. To do this, open the drop-down lists in columns D or E of the corresponding crosssections in the window 1.3 *Cross-Sections* and select whether the cross-sections should be determined *From the current row* or from user-defined *Favorites* (see Figure 2.8, page 15). You can also start the cross-section optimization out of the results windows by using the context menu.

	A	В	С	D	E	F	G
Member	Location	S-Point	Load-	Stress	Stress [N	1/mm ²]	Stress
No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio
6	Cross-section	No. 17-	IPE 360 D	DIN 1025-5:1994			
	0	<u> </u>		·	-99.7	218.2	0.4
	6.	<u>6</u> 0 to C	ross-Sectio	on Doubleclick	-8.7	218.2	0.0
	1.	Info Ab	out Cross-	Section	86.7	218.2	0.4
	6.	0			11.7	218.2	0.0
	6.		e Cross-Se		-9.0	126.0	0.0
	6.	Cross-S	ection Opt	timization Palameters	-8.8	126.0	0.0
	0.000	10	RC1	Tau M-T	7.0	126.0	0.0
	0.000	1	RC1	Sigma-eqv	100.4	218.2	0.4

Figure 7.5: Context menu for cross-section optimization

During the optimization process, STEEL determines the cross-section that fulfills the analysis requirements in the most optimal way, that is, comes as close as possible to the maximum allowable ratio specified in the *Details* dialog box (see Figure 3.1, page 19). The required cross-section properties will be determined with the internal forces from RSTAB. If another cross-section proves to be more favorable, this cross-sections: the original cross-section from RSTAB and the optimized one (see Figure 7.7).

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

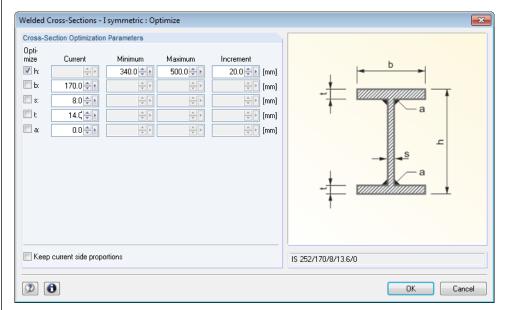


Figure 7.6: Dialog box Welded Cross-Sections - I symmetric: Optimize

By ticking the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. The ticked check box activates 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*, tick the corresponding check box. In addition, you have to select at least two parameters for optimization.



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

5

Please note that the internal forces won't be recalculated automatically with the modified cross-sections: It is up to you to decide which cross-sections should be transferred to RSTAB for a new calculation. As a result of optimized cross-sections, the internal forces may differ significantly because of the changed stiffnesses in the structural system. Therefore, it is recommended to recalculate the internal forces of the modified cross-sections after the first optimization and then to optimize the cross-sections once again.

The modified cross-sections can be exported to RSTAB: Set window 1.3 *Cross-Sections* and use the menu

$\textbf{Edit} \rightarrow \textbf{Export All Cross-Sections to RSTAB}.$

The context menu in window 1.3 provides options to export optimized cross-sections to RSTAB.

	A	В	С	D	E	F	6 - HE A 140 DIN 1025-3:1994
	Material	Cross-Section	Max. Design	Opti-			STEEL
No.	No.	Description	Ratio	mize	Remark	Comment	
1	1	T HE A 400 DIN 1025-3:1994	0.86				
2	2	T IS 360/170/8/14/0	0.96				У
3	2	T IS 500/170/8/14/0	0.96				
6	1	T HE A 140 DIN 1025-3:1994	Info About Cros	c Section	43	-	
7	1	T HE A 120 DIN 1025-3:1994					
9	1	I IPE 360 DIN 1025-5:1994	Cross-Section Li	ibrary			6 - HE A 160 DIN 1025-3:1994
10 12	1	T HE A 140 DIN 1025-3:1994	Edit List 'Design	of Memb	ers' in Tabl	e1.1 +	RSTAB
12	1	QRO 80x4 DIN 59410:1974					
13	1	RD 24 DIN 1013-1 HE A 200 DIN 1025-3:1994	Optimize Cross-	Section			
16		Rectangle 200/200	Cross-section O	ptimizatio	n <u>P</u> aramet	ers	*y
17		I IPE 360 DIN 1025-5:1994	Export Cross-Sec		TAD		
		1 1 2 300 Ditt 1023-3.1334	Export Cross-Sec	ction to K		2	
			Export <u>A</u> ll Cross-	Sections	o RSTAB	°	1
			Import Cross-Se	ction fron	n RSTAB		
			Import All Cross			D I	🔒 🚰 🎞 🖾 à
			Impolit All Cross	-sections	ITOM KSTA	·•	
							Cross-section No. 6 used in
							Members No.:
							54,56,58,81-83
							01,00,00,01 00
							Sets of members No :
							Σ Lengths: Σ Masses:
							31.37 [m] 0.956 [t]
							31.37 [m] 0.356 [t]
							14 A 14
							Material:
							1 - Steel S 235
		1) The cross-section in RSTAB is	different to that in STEE	EL.			
							•

Figure 7.7: Context menu in window 1.3 Cross-Sections

When data has been modified, a security query appears as to whether the results of RSTAB should be deleted.

STEEL Question No. 252	
Do you want to transfer the changed cross-sections to RS	TAB?
If so, the results of RSTAB and STEEL will be deleted.	
Yes No	

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

Calculation

When you confirm the query and start the [Calculation] subsequently in the STEEL add-on module, the RSTAB internal forces as well as the design will be determined and performed in one single calculation run.

7 General Functions



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



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

7.3 Units and Decimal Places

Units and decimal places for RSTAB and the add-on modules are managed in one dialog box. In the add-on module STEEL, 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 RSTAB. STEEL is preset in the list *Program / Module*.

Units and Decimal Places						×
Program / Module		STEEL				
STEEL	^	Stresses		Parts List		
STEEL EC3		0	Unit Dec. places		Unit	Dec. places
STEEL AISC						
STEEL IS		Stresses:	N/mm^2 ▼ 1 🚔	Lengths:	m 🔻	2 🌩
STEEL SIA		Stress ratios:	2 🖨	Total lengths:	m 🔻	2 🌲
STEEL BS				- Surface areas:	m^2 ▼	
STEEL GB				Surface areas:	m^2 ▼	2 🜩
STEEL CS				Volumes:	[m^3 ▼]	2 🌲
ALUMINIUM	=			Masses per length:	kg/m ▼	2 🌩
NAFFA	=			· -		
LTB				Masses:	kg 🔻	2 🌩
FE-LTB C-TO-T				Total masses:	t 🗸	3 🜩
PLATE-BUCKLING						
ASD						
CRANEWAY						
CONCRETE						
CONCRETE Columns						
- TIMBER Pro						
- TIMBER						
COMPOSITE-BEAM						
DYNAM						
Joints						
END-PLATE						
CONNECT						
FRAME-JOINT Pro						
FRAME-JOINT DSTV	-					
: U31V						
(2)	Œ				ОК	Cancel
	رتعت					

Figure 7.9: Dialog box Units and Decimal Places



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



7.4 Data Transfer

7.4.1 Material Export to RSTAB

When you have adjusted the materials in STEEL for the design, you can export the modified materials to RSTAB, similar to the export of cross-sections: Set window 1.2 *Materials* and then click the menu

Edit \rightarrow Export All Materials to RSTAB.

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

Material Library
Export Material to RSTAB
Export <u>A</u> ll Materials to RSTAB
Import Material from RSTAB
Import All Materials from RSTAB

Figure 7.10: Context menu of window 1.2 Materials

Calculation

Before the modified materials are transferred to RSTAB, a security query appears as to whether the results of RSTAB should be deleted. When you confirm the query and start the calculation subsequently in the STEEL add-on module, the RSTAB internal forces as well as the design will be determined in one single calculation run.

If the modified materials were not yet exported to RSTAB, you can transfer the original materials to the design module, using the options shown in Figure 7.10. Please note that this option is only available in window 1.2 *Materials*.

7.4.2 Export of Results

The STEEL results can also be used by other programs.

Clipboard

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

Printout report

The data of the STEEL add-on module can be printed into the global printout report (see chapter 6.1, page 49) to export them subsequently. Then, in the printout report, click

File \rightarrow Export to RTF.

The function is described in the RSTAB manual, chapter 10.1.11.

Excel / OpenOffice

STEEL provides a function for the direct data export to MS Excel, OpenOffice.org Calc, or the file format CSV. To open the corresponding dialog box, click

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

The following export dialog box appears.

7 General Functions



Table Parameters	Application
📝 With table header	Microsoft Excel
📃 Only marked rows	OpenOffice.org Calc
	CSV file format
Transfer Parameters	
Export table to active work!	book
Export table to active work:	shoot
	sneet
Rewrite existing worksheet	
Rewrite existing worksheet	
Selected Tables	
Selected Tables	
Selected Tables Active table All tables	

Figure 7.11: Dialog Export - MS Excel

When you have selected the relevant parameters, start the export by clicking [OK]. Excel or OpenOffice will be started automatically, that means the programs do not have to be opened first.

	9	• (°" • -		Table	e1.xlsx - Micro	soft Excel			• X	3
Fi	ile	Home Ins	sert Pagel	ayout Fo.	rmulas Da	ta Review	View Add	i-Ins 🛆 🚱	- 6	23
	ي 📕	Calibri	· 11 ·	= = =	E Text	• • A	¦ate Insert →	Σ - Α	7 A	
	- ₋	BI	J - A A		i 🔤 - 🕎	- % ,	Pelete 🎬			
Pas +	te 🝼		- <u>A</u> -		\$¢,- €.0	.00 ⇒.0 Style	s 🔡 Format		rt& Find & ter ∗ Select	
Clip	board r	Ξ Fo	nt G	Alignmer	nt 🖬 Nu	imber 🕞	Cells	Ec	diting	
B4 • <i>f</i> x 68								1		
	А	В	С	D	E	F	G	Н	I.	E
1	Set	Member	Location	S-Point	Load-	Stress	Stress [N	l/mm²]	Stress	I
2	No.	No.	x [m]	No.	ing	Туре	Existing	Limiting	Ratio	
3	1	Floor beam	B-B (Memb	er No. 66-69)					
4		68	3,125	6	RC1	Sigma Total	206,4	218,2	0,95	
5		66	0,000	13	RC1	Sigma N	34,7	218,2	0,16	
6		69	3,125	10	RC1	Sigma M-y	182,6	218,2	0,84	
7		68	3,125	6	RC1	Sigma M-z	3,4	218,2	0,02	
8		66	6,250	13	RC1	Tau Total	-41,5	126,0	0,33	
9		69	0,000	13	RC1	Tau V	41,5	126,0	0,33	
10		66	3,125	1	RC1	Tau M-T	-4,2	126,0	0,03	
11		68	3,125	6	RC1	Sigma-eqv	206,4	218,2	0,95	
12										
13	2		A-A (Memb							
14		73	3,125	6	RC1	Sigma Total	187,6	218,2	0,86	
15		71	3,125	1	RC1	Sigma N	-0,6	218,2	0,00	
16		71	3,125	10	RC1	Sigma M-y	184,8	218,2	0,85	
17		73	2,813	6	RC1	Sigma M-z	4,0	218,2	0,02	
18		71	0,000	13	RC1	Tau Total	42,0	126,0	0,33	
19		71	6,250	13	RC1	Tau V	-41,9	126,0	0,33	
20		71	3,125	1	RC1	Tau M-T	-4,3	126,0	0,03	
21		73	3,125	6	RC1	Sigma-eqv	187,6	218,2	0,86	1
H 4		Tabelle3	2.2 Stresse	s by Set of	Members /		III 1000((1
Rea	ay						100% (-		+	

Figure 7.12: Result in Excel



A Literature

- [1] DIN 18800 Teil 1: Stahlbauten, Bemessung und Konstruktion, 1990
- [2] Erläuterungen zu DIN 18800 Teil 1 bis 4, Beuth-Kommentar, Beuth Verlag, 2. Auflage 1994
- [3] PETERSEN, Chr.: Stahlbau, Vieweg und Sohn, Braunschweig/Wiesbaden, 3. Auflage 1993
- [4] SCHNEIDER Bautabellen, Werner Verlag, 20. Auflage 2012
- [5] STAHLBAU Handbuch, Band 1, Stahlbau-Verlagsgesellschaft mbH, Köln 1993



B Index

2
2D model type23
А
Aluminum
В
- Background graphic43
Buttons
с
c/t-parts
Calculation
Clipboard
Closed cross-section
Color bar
Color spectrum
Comment
Component thickness
Control panel
Coordinates stress point
Cross-section
Cross-section graphic17
Cross-section library15
Cross-section optimization55
Cross-section stresses
D
Decimal places 10, 57
Design
Design case
Design colored47
Design of9
Design standard8
Detail settings19
Display navigator45, 47
DYNAM9
E
Eccentric transverse load21
Equivalent stress
Excel
Exit STEEL
Export
Export cross-sections

Extreme value 20
F
Fatigue design 22
Favorite
FE-LTB
Filter 40, 47
Filtering members
G
General Data
Governing internal forces 19, 35
Graphics
н
Hidden result diagram 48
1
Info about cross-section
Installation
Internal forces
L
Length
Limit τ
Limit σ _{eqv}
Limit σ _x
Limit stress
Load case
Load combination
Loading
Location x 29, 33
Μ
Manually defined limit stresses
Mass
Material10, 14, 58
Material description 11
Material library13
Material properties10
Member9, 32
Method of stress calculation 19
Ν
Navigator
Normal stresses
0
Open cross-section

Export material......58

OpenOffice58
Optimization
Р
Panel7, 45, 47
Parametric cross-section55
Part No
Partial safety factor γ _M 11, 14
Parts list
Plastic shape factor α_{pl}
Plastification
Print
Print graphic49
Printout report
Printout selection
В
Remark
Rendering47
Result combination
Result diagrams
Result graphic45
Result values
Results evaluation
Results navigator44
Results tables
RSTAB graphic50
RSTAB work window43
s
Selecting windows
Selection of stresses
Set of members
SHAPE-MASSIVE
SHAPE-THIN
Shear stresses
Signs
Stainless steel
Start calculation
Start program



Start STEEL6
Statical moment
STEEL case
Stress components 40, 44
Stress design 26, 30
Stress diagram
Stress graphic
Stress point 18, 19, 22, 25, 29, 34, 40, 41, 42, 50
Stress ratio26, 30, 41
Stress type
Stresses
Sum
Super combination9
Surface area
т
Taper
Thickness of structural components
Torsion21, 24, 25
Transverse load21
Twin stress point
U
Units 10, 57
Unsymmetrical cross-section
User profile
User-defined cross-section
v
View mode
Visibility
Volume
w
Warping torsion
Windows
X
x-Location
Y
Yield strength f _{y,k} 11, 14