Version
March 2011

Add-on Module

STEEL

Stress Analysis
Cross-section Optimization

Program Description

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

Contents

1. Introduction

1.1 Add-on Module STEEL

STEEL is not a stand-alone program. STEEL is one of the add-on modules integrated in the graphical user interface of the main program RSTAB. Due to the integration, all structure-specific input data as well as the internal forces are automatically available in the STEEL module. In return, you can evaluate the design results in the RSTAB work window graphically and include them in the global printout report.

STEEL performs general stress designs by calculating existing stresses and comparing them with the limit stresses. The program provides a comprehensive cross-section library as well as an expandable material library with standard-specific limit stresses. For each cross-section, design relevant stress points that can also be used for graphical evaluations are already available.

During the stress analysis process, the program determines also the maximum stresses of sets of members and shows the governing internal forces for each member. In addition, STEEL offers you an automatic cross-section optimization including an export option for modified cross-sections to RSTAB.

Different STEEL design cases allow for a flexible analysis of stresses. The design is completed by a parts list with quantity surveying.

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

- 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 it is possible to select active objects in the background graphic
- View mode for modifying the view in the RSTAB work window
- Colored relation scales in results tables
- 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 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 DIN 18000 part 2 or EN 1993-1-1, cannot be performed. To analyze the stability of a structure, use the add-on modules KAPPA, LTB or STEEL EC3.

We hope you will enjoy working with the add-on module STEEL. Your comments and suggestions are always welcome.

Your team from ING.-SOFTWARE DLUBAL

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. Zdeněk Kosáček
Mgr. Petr Oulehle
David Schweiner
Ing. Roman Svoboda
Dis. Jiří Šmerák
Lukáš Tůma

Cross-section and material database

Ing. Ph.D. Jan Rybín
Jan Brnušák

Program design, dialog figures and icons

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

Program supervision

Ing. Martin Vasek
Petr Pražák
Michala Sobotková
Ing. František Knobloch

Manual, help system and translation

Dipl.-Ing. (FH) Robert Vogl
Ing. Dmitry Bystrov
Jan Jeřábek
Ing. Ladislav Kábrt
Mgr. Michaela Kryšková
Dipl.-Ü. Gundel Pietzcker
Mgr. Petra Pokorná

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (BA) Sandy Baumgärtel
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Andreas Hörold
Dipl.-Ing. (FH) Bastian Kuhn
M.Sc. Dipl.-Ing. Frank Lobisch
Dipl.-Ing. (FH) Alexander Meierhofer
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl
Dipl.-Ing. (FH) Andreas Wopperer

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 tables as well as their structure. The text of the manual shows the described **buttons** in square brackets, for example [Apply]. At the same time, they are pictured on the left. **Expressions** appearing in dialog boxes, tables 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 *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

To start the program in the menu bar,

point to **Design - Steel** on the **Additional Modules** menu, and then select **STEEL**.

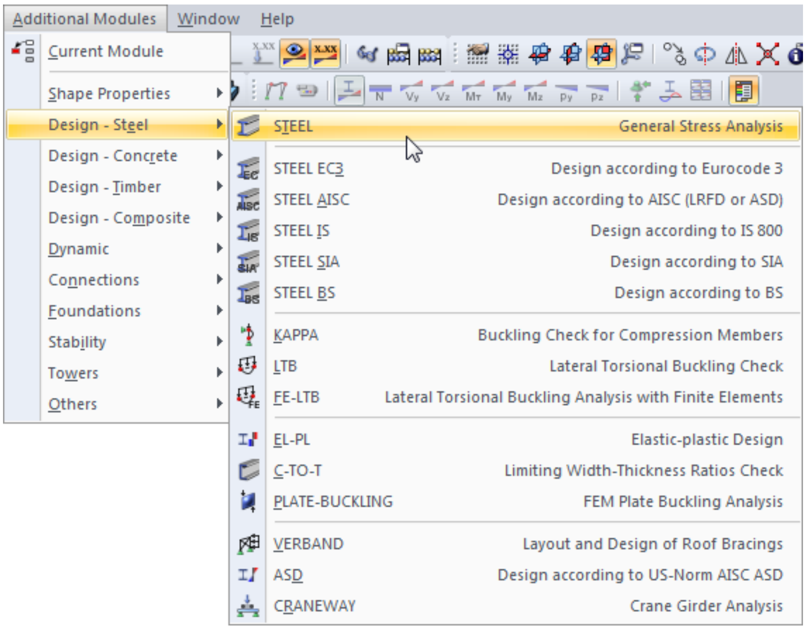


Figure 1.1: Menu: *Additional Modules* → *Design - Steel* → STEEL

Navigator

To start STEEL in the *Data* navigator,

select **STEEL** in the **Additional Modules** folder.

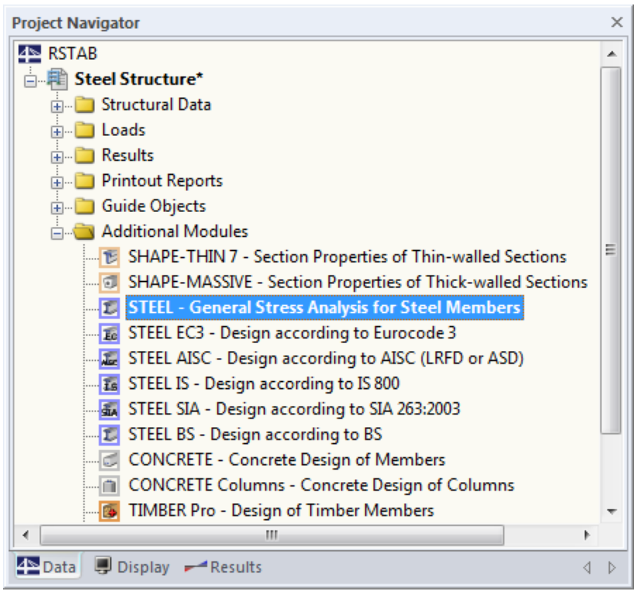
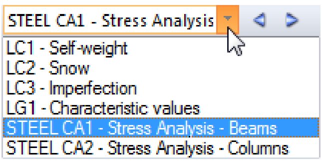


Figure 1.2: Data navigator: *Additional Modules* → STEEL

Panel



In case STEEL results are already available in the RSTAB structure, you can set the relevant STEEL design case in the load case list of the RSTAB toolbar. Maybe you have to activate the graphical results display first, showing the stresses and design ratios by using the button [Results on/off].

When the results display is activated, the panel is available, too. To access the add-on module quickly, use the panel button [STEEL].

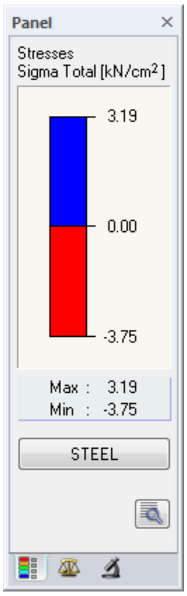


Figure 1.3: Panel button [STEEL]

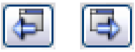
2. Input Data

All data required for the definition of design cases is entered in tables. The [Pick] button allows for a graphical selection of the members and sets of members that you want to design.

When you have started the add-on module, a new window opens where a navigator is displayed on the left, managing all available tables. The pull-down list above the navigator contains the design cases (see chapter 7.1, page 51).

If you open STEEL in an RSTAB structure for the first time, the module will import the following design relevant data automatically:

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



To select a table, click the corresponding entry in the STEEL navigator or page through the tables by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent table.

Click [OK] to save the entered data and quit the add-on module STEEL. When you click [Cancel], you quit the module but without saving the data.

2.1 General Data

In table 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 table 1.2 because the material properties are related to the standard.

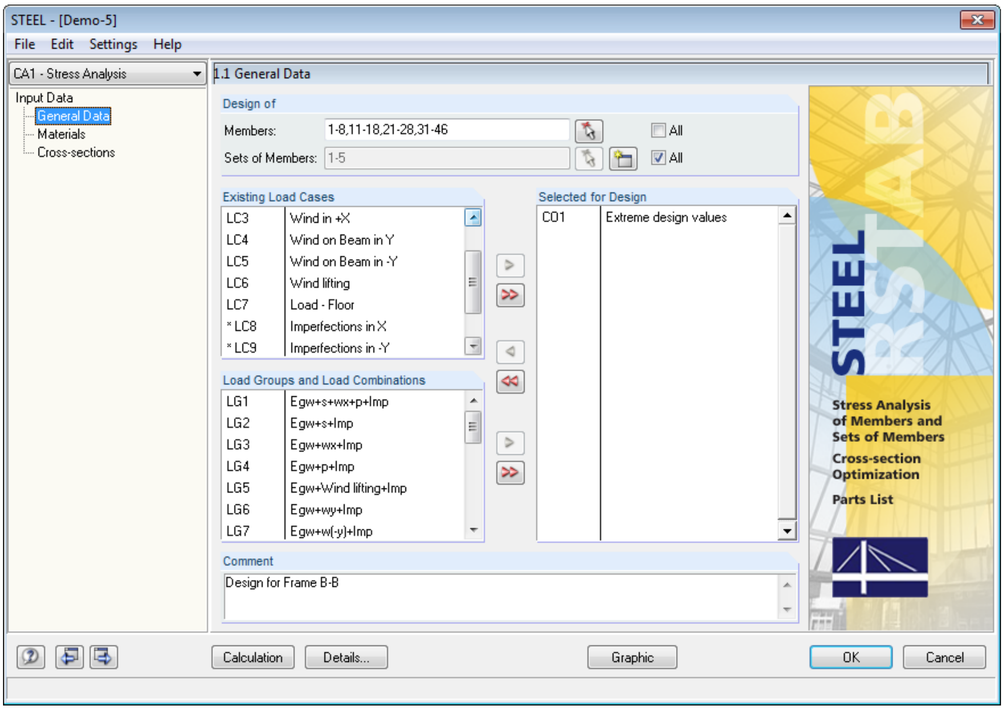


Figure 2.1: Table 1.1 *General Data*

Design of

The design can be carried out for *Members* as well as for *Sets of Members*. If you want to design only selected objects, clear the *All* check boxes. Then you can access the input fields to enter the numbers of the relevant members or sets of members. To select the objects graphically in the RSTAB work window, use the [Pick] button. The list of the object numbers preset in the field can be selected quickly by double click and overwritten by entering the data manually.

In case you have not yet defined any sets of members in RSTAB, you can create them in the STEEL module by clicking the [New] button. The dialog box that you already know from RSTAB appears where you can specify the data for a new set of members.

When you design sets of members, STEEL determines the maximum stress ratio of all members contained in the set of members. Subsequent to the analysis, the results are shown in table 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*.

Existing Load Cases / Load Groups and Load Combinations

The two dialog sections list all load cases, load groups and load combinations as well as super combinations defined in RSTATB that are relevant for the design. Dynamic load cases, if available, are listed, too. Use the button [▶] to transfer selected entries to the list *Selected for Design* on the right. You can also double-click the items. To transfer the complete list to the right, use the button [▶▶].

Load cases that are marked by an asterisk (*), like case 8 and 9 in Figure 2.1, cannot be designed. This happens when you haven't defined any loads or when the load case contains only imperfections as shown in the example.

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

Selected for Design

The column on the right lists the loads selected for the design. Use the button [◀] to remove selected items from the list. You can also double-click the entries. With the button [◀◀], you can transfer the entire list to the left.

Details...

The design of an enveloping "Or" load combination is often performed more quickly than the analysis of all actions that have been globally set. On the other hand, the influence of the actions contained in such a total CO are less transparent. Therefore, it is recommended to check the *Method of Stress Calculation* for load combinations in the *Details* dialog box (see chapter 3.1, page 18).

Comment

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

2.2 Materials

The table is subdivided into two parts. In the upper part, the materials defined in RSTAB are listed with their limit stresses. In the *Material Properties* section below, the properties of the current material, i.e. the table row currently selected in the upper section, are displayed.

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

The material properties required for the determination of internal forces in RSTAB are described in detail in chapter 5.2 of the RSTAB manual. The design relevant material properties are stored in the global material library. They are preset automatically but can be adjusted in the present table.

To adjust the units and decimal places of material properties and stresses, select **Units and Decimal Places** in the module's **Settings** menu (see Figure 7.10, page 56).

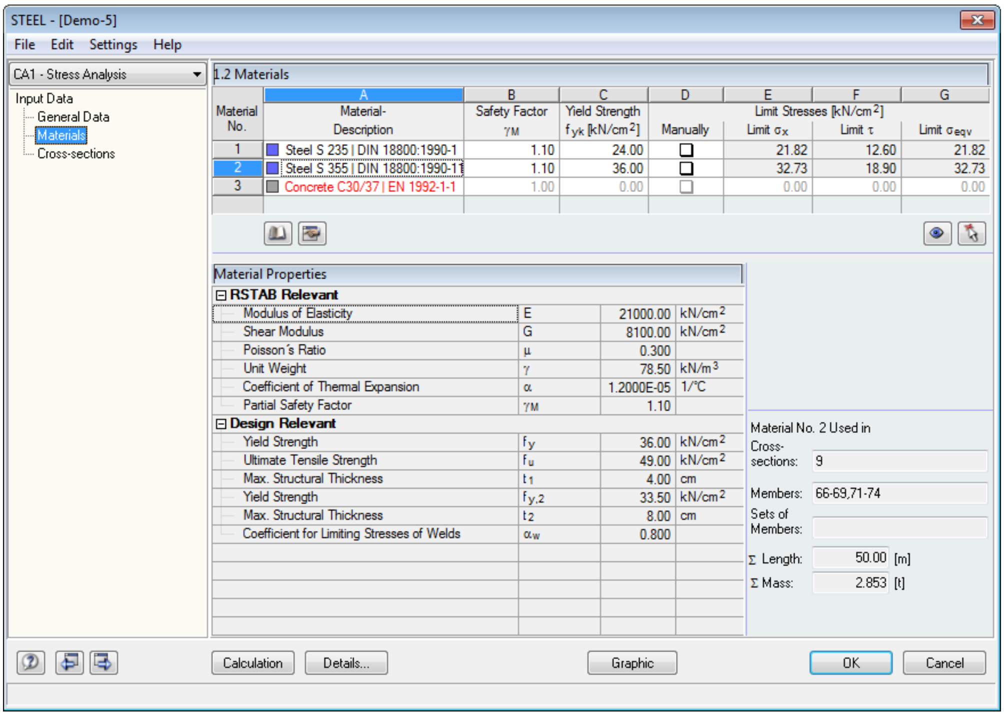
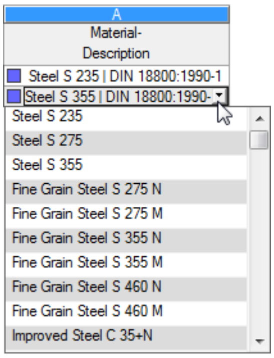


Figure 2.2: Table 1.2 *Materials*

Material Description



The materials defined in RSTAB are already preset. When you modify the material description and the manually entered material name corresponds to an entry of the material library, STEEL will import the material properties that are required for the design.

You can also modify the material by using the list: Place the pointer in a table row of column A, and then click the button [▼] or use the function key [F7] to open the list shown on the left. When you have selected a new material, the design relevant properties are entered into the remaining fields of the corresponding table row.

Only **Steel** materials are available in the list. You can choose among various steel grades of different standards and codes for steel construction. In principle, you can carry out the design with any material whose stress concept is based on the comparison of existing normal, shear and equivalent stresses with the respective allowable stresses (for example the design of cross-sections consisting of aluminum or stainless steel). Of course, you must take into account the respective standard specifications additionally.

If you have set a material whose limit stresses are not defined (for example timber), the entry is highlighted in red. It is possible, however, to define the limit stresses by ticking the check box **Manually** in column D and entering user-defined specifications. The red color disappears as soon as you have defined the allowable stresses in the columns E to G.

The import of materials from the library is described later.

Safety Factor γ_M

This factor describes the safety factor used to calculate 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 the second-order or the large deformation analysis: According to DIN 18800 part 2, el. (116), you must take into account 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 partial safety factor γ_M when you design the ultimate limit state.

Yield Strength f_{yk}

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 DIN 18800 part 1, section 4 and EN 1993-1-1, section 3.

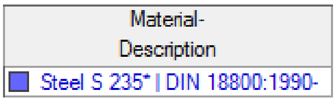
Limit Stresses

The limit stresses of materials that are stored in the general material library are preset automatically. These entries provide no access for modifications.

If you want to adjust the limit stresses, you can use the [Edit Material] button (see Figure 2.4, page 14). You can also use the check box *Manually* in column D.

Manually

If the check box is ticked, you can define the limit stresses in the 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. 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.

$$\tau_{R,d} = \frac{f_{yk}}{\gamma_M \cdot \sqrt{3}}$$

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.

Yield strength depending on thickness of structural components

For some materials, there is a relation between the characteristic yield strength $f_{y,k}$ and the thickness *t* of the relevant structural component. The *Max. Structural Thickness* of the individual ranges together with the corresponding yield strength is indicated in the dialog section *Material Properties* below.

The zones of the yield strength are specified in the standards, for example DIN 18800 part 1, table 1. Use the [Edit Material] button to check and adjust, if necessary, the structural thicknesses and the assigned stresses (see page 14).

Material Library

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

select **Material Library** on the **Edit** menu

or use the button shown on the left.

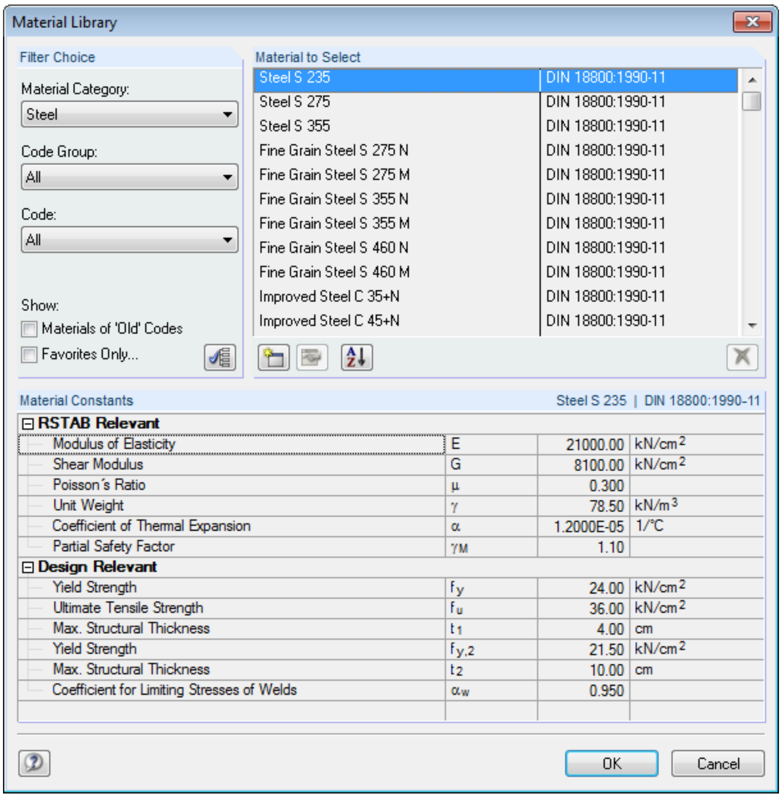


Figure 2.3: Dialog box *Material Library*

In the *Filter Choice* section, *Steel* is preset as material category. Select the steel grade 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 [↵] button to import the selected material to table 1.2 of the add-on module.

Chapter 5.2 of the RSTAB manual describes in detail how materials can be filtered, added or rearranged. By using the [Create New Material] button, you can create new types of steel with user-defined material properties and store them for later use.

When you select another material category than *Steel*, be aware that you can design only materials whose stress concept is based on the comparison of existing normal, shear and equivalent stresses with the respective allowable stresses. Therefore, it would be possible, with some restrictions, to design cross-sections consisting of aluminum or stainless steel.

If you have set a material whose limit stresses are not defined (for example timber), the entries of the corresponding table row in table 1.2 will be highlighted in red. However, you can define the limit stresses by ticking the check box **Manually** in column D and entering user-defined specifications. The red color disappears as soon as you have defined the allowable stresses in the columns E to G. Please note that designs, for example for timber cross-sections, won't be performed completely: Timber standards require further criteria implemented in the add-on module TIMBER Pro.

Edit Material

The yield strengths and limit stresses of the currently selected material can be adjusted by clicking the button shown on the left. The following dialog box appears:

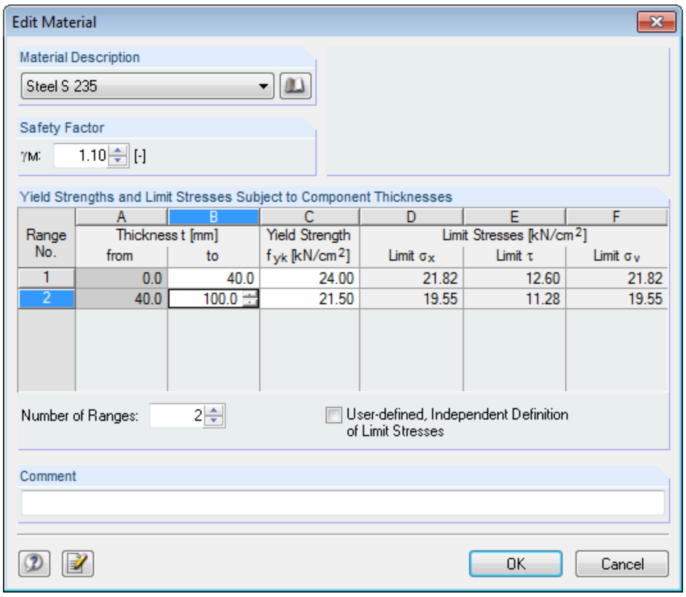
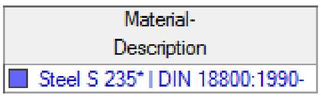


Figure 2.4: Dialog box *Edit Material*

By means of the factor γ_M in the dialog section *Safety Factor* you can reduce the characteristic values of the yield strength f_{yk} defined in column C in the dialog section below. The limit stresses determined according to Equation 2.1 and Equation 2.2 on page 12 are listed in the columns D to F.

In the dialog section *Yield Strengths and Limit Stresses Subject to Component Thicknesses*, you can change the ranges of the component *Thickness t*. The number of ranges is preset conforming to standards. You can shift the limits for the ranges by entering values manually in column B. Column A will be adjusted automatically. For each range, you can assign a specific *Yield Strength* f_{yk}.

To define the limit stresses individually, tick the check box *User-defined, Independent Definition of Limit Stresses*. Then you can access column D to F to enter user-defined data.



Modified material properties are marked by an asterisk in table 1.2.

2.3 Cross-sections

This table lists the cross-sections that are relevant for the design. In addition, the table allows for the definition of optimization parameters.

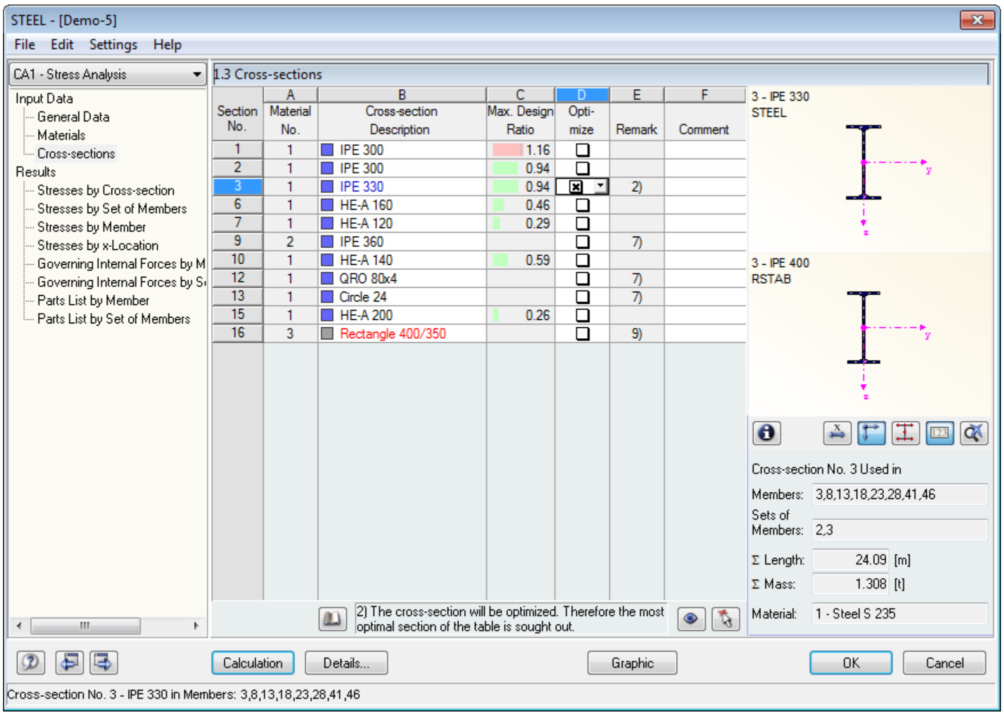


Figure 2.5: Table 1.3 *Cross-sections*

Cross-section Description

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

It is always possible to modify the cross-sections for the design. The description of a modified cross-section is highlighted in blue. A cross-section that is not allowed to be used (for example a solid cross-section) is highlighted by red letters.

To modify a cross-section, enter the new cross-section description directly into the corresponding table row. You can also select a new cross-section from the library. To open the library, use the button [Import Cross-section from Library] below the table. Alternatively, place the pointer in the respective table row and click the [...] button, or use the function key [F7]. The RSTAB cross-section library, or the cross-section table of the input field, appears.

The selection of cross-sections from the library is described in detail in chapter 5.3 of the RSTAB manual.

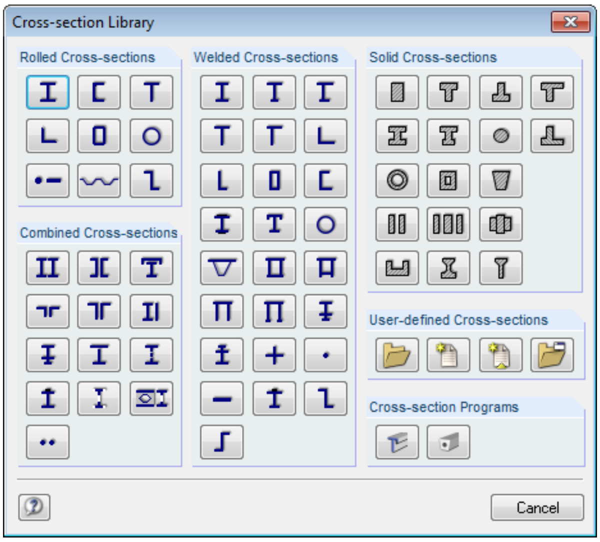
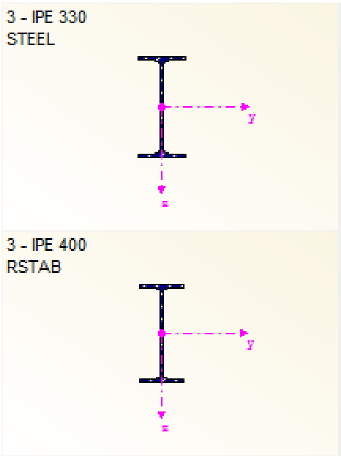


Figure 2.6: Cross-section Library



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 table. The stress analysis will be performed with the internal forces from RSTAB for the cross-section selected in STEEL.

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 second moments of area 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. Then, the rendering in RSTAB will display the cross-sections each to the middle of the member, but a calculation won't be possible, neither in RSTAB nor in STEEL.

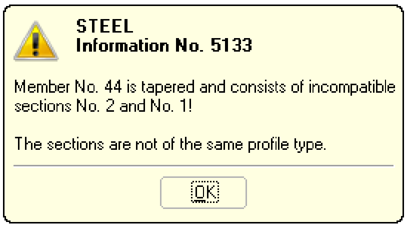
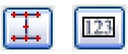


Figure 2.7: Warning in case of incompatible cross-sections



To check the data, you can display the cross-section's stress points including numbering in the cross-section graphic to the right. For more information on stress points, see chapter 4.1 on page 27.

It is important for the design to have the same number of stress points. To produce the same number, you can create the second cross-section by copying the start cross-section and adjusting the geometry parameters. If required, you define both cross-sections as parameterized ("Welded") cross-sections. With the cross-section group *I-Section Plus Lower Flange*, the library provides cross-sections especially used for tapers.

Max. Design Ratio

This column is intended to be a decision support for the optimization process. It will be displayed as soon as a design has been carried out. By means of the displayed design ratio and colored relation scales, you can see which cross-sections are hardly utilized and thus oversized, or extremely stressed and thus undersized.

Optimize

Each cross-section can be improved by an optimization process. By means of the internal forces of RSTAB, the program determines the cross-section of the cross-section table that comes as close as possible to the maximum stress ratio specified in the *Details* dialog box (see Figure 3.1, page 18).

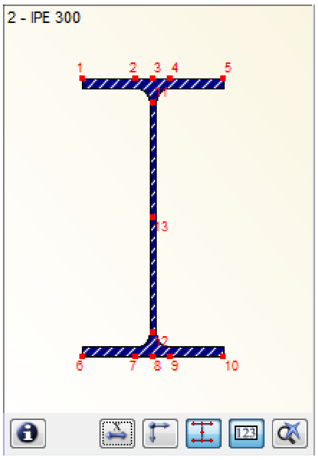
If you want to optimize a cross-section, tick the corresponding check box in column C or D. Recommendations for optimizing cross-sections can be found in chapter 7.2 on page 53.

Remark

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

Remark *6) Incorrect cross-section data! Members of this cross-section will not be designed.* indicates an unknown cross-section, which means a section that is not registered in the cross-section database. This may be a user-defined cross-section, or a SHAPE-THIN cross-section that has not yet been calculated. In such a case, switch to RSTAB and specify the settings for the **Stress Points** (see chapter 5.3 of the RSTAB manual, section *Create User-defined Cross-sections*).

Cross-section graphic



In the right part of table 1.3, a graphic of the currently selected cross-section is displayed. To adjust the size of the cross-section's graphical representation, use the zoom function (wheel button of the mouse device).

The buttons below the cross-section graphic are reserved for the following functions:

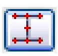
Button	Function
	Opens the dialog box *Info about cross-section* showing the cross-section details.
	Displays or hides the dimensions of the cross-section.
	Displays or hides the principal axes of the cross-section.
	Displays or hides the stress points.
	Displays or hides the numbering of stress points.
	Resets the full view of the cross-section graphic.

Table 2.1: Buttons of cross-section graphic

3. Calculation

To start the calculation, click the [Calculation] button that is available in all input tables. The stress analysis is carried out by using the internal forces determined in RSTAB. Before you start the calculation, it is recommended to check the calculation parameters.

3.1 Calculation Details

To check the calculation parameters, use the [Details] button to open the following dialog box that can be accessed from any table in the module.

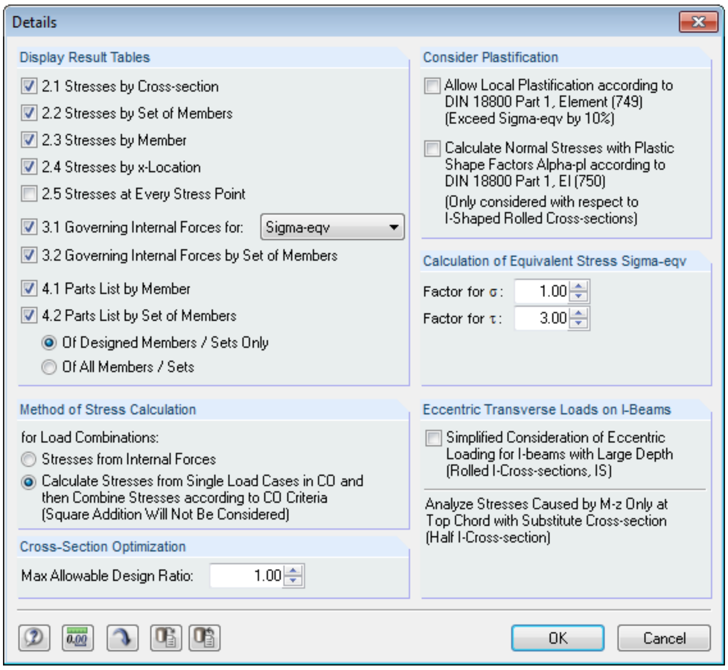
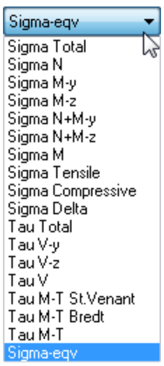


Figure 3.1: Dialog box *Details*

Display Result Tables

This dialog section controls the display of the results tables including parts list. The different results tables are described in chapter 4.

Table 2.5 *Stresses at Every Stress Point* is inactive by default because the stress graphic also can provide an appropriate evaluation of the results in the stress points. For a detailed table check of stresses, however, you may activate the display of this table.



The values displayed in table 3.1 *Governing Internal Forces* are normally related to the equivalent stress σ_{eqv}. If required, you can use the list to select another stress type.

Consider Plastification

Allow Local Plastification

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

STEEL checks if both conditions mentioned in el. (749) are fulfilled for the assumption of "small areas".

$$\left|\sigma_N + \sigma_{M_y}\right| \leq 0.8 \cdot \sigma_{R,d}$$

Equation 3.1

$$\left|\sigma_N + \sigma_{M_z}\right| \leq 0.8 \cdot \sigma_{R,d}$$

Equation 3.2

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

Plastic Shape Factors α_{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.

$$\sigma_M = \left| \pm \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z \pm \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y \right|$$

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 suggested in the standard. The allowance of local plastification is applied exclusively to rolled I-shaped cross-sections.

Calculation of Equivalent Stress Sigma-eqv

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

$$\sigma_{eqv} = \sqrt{f_1 \cdot \sigma_{total}^2 + f_2 \cdot \tau_{total}^2}$$

with f_1: factor for normal stresses

f_2: factor for shear stresses

Equation 3.4

The program presets the factors f_1 = 1.0 and f_2 = 3.0. which are mentioned in DIN 18800 part 1, el. (748).

Method of Stress Calculation for Load 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 both load cases will be superimposed with the "Variable" setting in a load combination, RSTAB will display the moment M_z as not related to the maximum moment M_y in table 3.1 *Members - Internal Forces*: The horizontal load does not contribute to the increase of the moment due to vertical load. If these CO internal forces are designed according to the maximum moments M_y and M_z separately, the program does not take into account the simultaneous influence of both internal forces for the combined analysis of the bending stresses.

Stresses from Internal Forces

The calculation method *Stresses from Internal Forces* uses the result rows of the RSTAB table 3.1 *Members - Internal Forces*. The max/min results are processed row by row. Thus, the program determines the stresses for each extreme value together with the corresponding internal forces.

The advantage of this type of calculation is that the results of the load 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 table 3.1 *Governing Internal Forces by Member* the module displays the result rows from the RSTAB results table 3.1 *Members - Internal Forces*.

Calculate Stresses from Single Load Cases and then Combine

This type of calculation is preset to calculate the stresses of load combinations. The program determines the normal and shear stresses of the contained load cases and superimposes them subsequently according to the specified CO combination criteria. In this way, the program ensures that the effects described above, in case of bending stresses that are each uniaxial, do not result in stress ratios that are too low.

The analysis is performed for each stress point: The compression, tension and shear stresses determined for the individual load cases are summed up according to the CO superposition criterion and displayed subsequently in the results tables.

The equivalent stress σ_{eqv} represents an exception because it is determined by the components of σ_{total} and τ_{total}. A superposition of equivalent stresses from the single load cases would not be correct and would result in stress ratios which are too high.

This type of calculation requires more computing time. Furthermore, the values displayed in table 3.1 *Governing Internal Forces* are more difficult to understand, in case they refer to the equivalent stresses.

As pure uniaxial bending usually does not occur in complex spatial systems, both calculation methods are normally supposed to show the same stress ratios in the output.

Cross-section Optimization

In case the optimization is not targeted on the maximum stress ratio of 100 %, you can specify another limit value in this input field.

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 the increasing cross-section depth. This is the reason why it is possible to consider eccentrically acting transverse loads by a simplified method for high I-sections: For each(!) rolled or welded symmetrical I-section that is set for design in the current STEEL case, the stress due to the bending moment M_z is calculated only on the upper flange. In such a case, the program uses an equivalent cross-section with half of the moment of inertia I_z.

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 the high I-beams in a separate STEEL case.

The button [Preset of Default Values] resets the program's presettings.

3.2 Stresses and Ratio

The normal stresses σ_{total}, τ_{total} and σ_{eqv} are displayed as presettings in the tables 2.1 to 2.5. To display the individual stress components, use the buttons [Select Stresses to Show] and [Extended Stress Diagram].

Normal stresses

According to the common conventions, tensile stresses are indicated by positive signs and compressive stresses by negative signs.

The analysis is carried out for each single stress point. Therefore, the components of the maximum stresses must not be summed up for a combined calculation (for example σ_{total}). Often, the maximum stresses occur at different stress points. You have to superimpose the stress components of the respective stress point.

The normal stresses σ have the following meanings:

σ_N	Stress due to axial force N $\sigma = \frac{N}{A}$ where A: cross-sectional area of cross-section
$\sigma_{M\text{-}y}$	Stress due to bending moment M_y $\sigma = \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z$ where $\alpha_{pl,y}$: plastic shape factor acc. to DIN 18800 part 1, el. (750) I_y: second moment of area related to principal axis y e_z: centroidal distance of stress point in direction z
$\sigma_{M\text{-}z}$	Stress due to bending moment M_z $\sigma = -\frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$ where $\alpha_{pl,z}$: plastic shape factor acc. to DIN 18800 part 1, el. (750) I_z: second moment of area related to principal axis z e_y: centroidal distance of stress point in direction y
σ_M	Stress due to bending moments M_y and M_z $\sigma = \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$
$\sigma_{tension}$	Tensile stress due to axial force N and bending moments M_y and M_z $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$
$\sigma_{compression}$	Compressive stress due to axial force N and bending moments M_y and M_z $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$
σ_{delta}	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 $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$

Table 3.1: Normal stresses σ

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

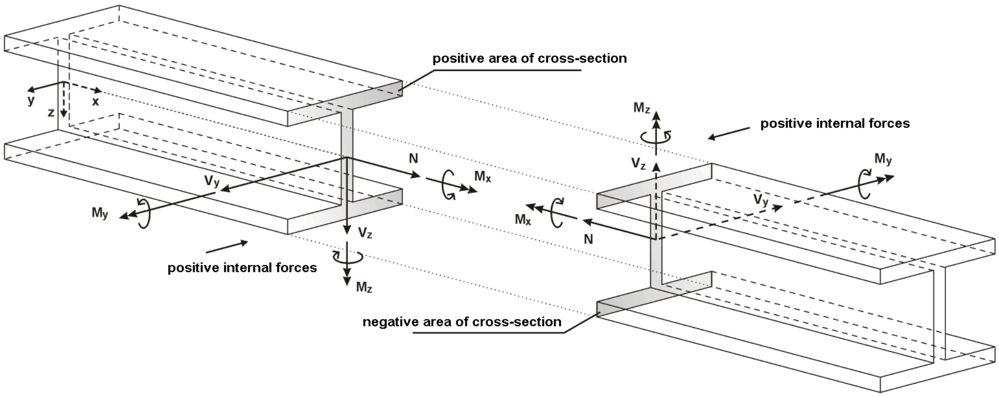


Figure 3.2: Positive definition of internal forces

The bending moment $\mathbf{M_y}$ is positive if tensile stresses are produced on the positive member side (in direction of the z-axis). $\mathbf{M_z}$ 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 usual conventions. These internal forces are positive if they act in a positive direction.

Shear stresses

The shear stresses τ have the following meanings:

$\tau_{V\text{-}y}$	Stress due to shear force V_y $\tau = -\dfrac{V_y \cdot Q_z}{I_z \cdot t}$ where Q_z: statical moment of area related to principal axis z I_z: second moment of area related to principal axis z t: governing thickness of cross-section
$\tau_{V\text{-}z}$	Stress due to shear force V_z $\tau = -\dfrac{V_z \cdot Q_y}{I_y \cdot t}$ where Q_y: statical moment of area related to principal axis y I_y: second moment of area related to principal axis y t: governing thickness of cross-section
τ_V	Stress due to shear forces V_y and V_z $\tau = -\dfrac{V_y \cdot Q_z}{I_z \cdot t} - \dfrac{V_z \cdot Q_y}{I_y \cdot t}$
$\tau_{M\text{-}T,\ St.Venant}$	Stress due to torsional moment M_T in case of open cross-section $\tau = \dfrac{M_T}{J_{St.V.}} \cdot t$ where $J_{St.V.}$: Saint Venant torsional constant t: governing thickness of cross-section

$\tau_{M\text{-}T,\ Bredt}$	Stress due to torsional moment M_T in case of closed cross-section $\tau = \frac{M_T}{2 \cdot A_m \cdot t}$ where A_m: area enclosed by the center lines of the cross-section t: governing thickness of cross-section
$\tau_{M\text{-}T}$	Stress due to torsional moment M_T $\tau = \frac{M_T}{J_{St.V.}} \cdot t$ or $\tau = \frac{M_T}{2 \cdot A_m \cdot t}$
τ_{total}	Shear stress due to shear forces V_y and V_z and torsional moment M_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 the following:

- 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*. In this case, 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 exclusively 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.

Equivalent stress

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

σ_{eqv}	Equivalent stress from normal stresses σ and shear stresses τ $\sigma_{eqv} = \sqrt{f_1 \cdot \sigma_{total}^2 + f_2 \cdot \tau_{total}^2}$ where f_1: factor for normal stresses f_2: factor for shear stresses

Table 3.3: Equivalent stress σ_{eqv}

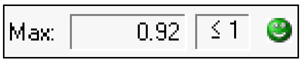
Details...

The factors f_1 and f_2 can be defined in the *Details* dialog box where 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.

For each internal force component, you can display the cross-section's stress ratio on the respective stress point (see chapter 5.1, page 37). 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 1.00 and the stress design was carried out successfully.

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

Equation 3.5: Design condition for normal stresses

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

Equation 3.6: Design condition for shear stresses

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

Equation 3.7: Design condition for equivalent stresses

3.3 Start Calculation

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

STEEL searches for the results of the load cases, load groups and load combinations that should be designed. If they cannot be found, the program starts the RSTAB calculation to determine the design relevant internal forces. In this determination process, the calculation parameters preset in RSTAB are applied.

If cross-sections should be optimized (see chapter 7.2, page 53), the program determines the required cross-sections and calculates the stresses.

It is also possible to start the calculation for STEEL results in the RSTAB user interface. The add-on modules are listed like load cases and load groups in the dialog box *To Calculate*. To open the dialog box in RSTAB,

select **To Calculate** on the **Calculate** menu.

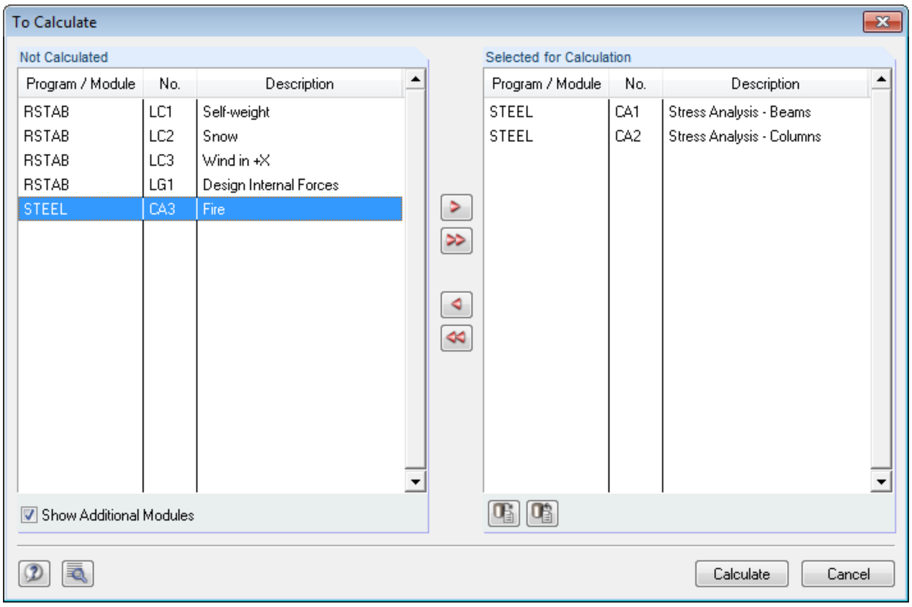


Figure 3.3: Dialog box *To Calculate*

If the STEEL design cases are missing in the *Not Calculated* list, tick the check box *Show Additional Modules* below the list.

To transfer the selected STEEL cases to the list on the right, use the button [▶]. Start the calculation by using the [Calculate] button.

You can also use the list in the RSTAB toolbar to calculate a design case directly: Select the STEEL case and click the button [Results on/off].

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

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

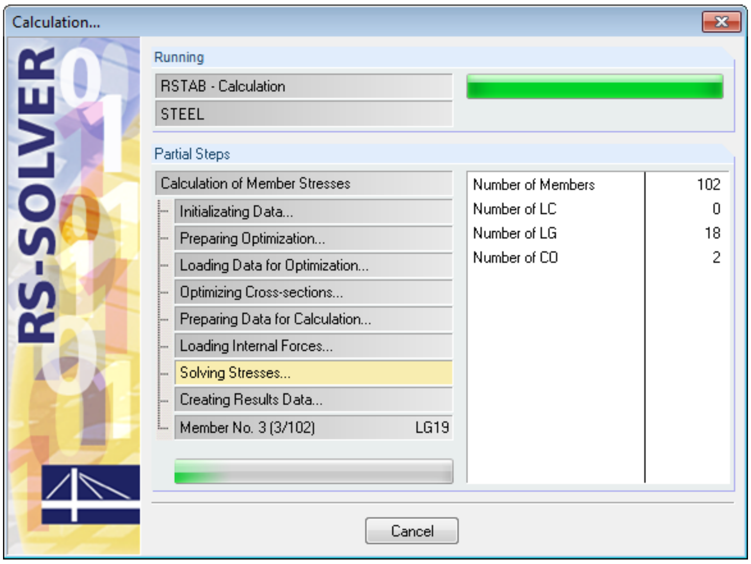


Figure 3.5: STEEL calculation

4. Results

Table 2.1 *Stresses by Cross-section* is displayed immediately after the calculation. The tables 2.1 to 2.5 show the stresses and ratios according to various criteria. The subsequent tables 3.1 to 4.2 display the governing internal forces as well as the parts lists in relation to members and sets of members. Use the STEEL navigator to access the relevant results table. You can also use the two buttons shown on the left or the function keys [F2] and [F3] to select the previous or subsequent table.

Click the [OK] button to save the results. The add-on module STEEL will be closed and you will return to the RSTAB work window.

Chapter 4 *Results* describes the different results tables one after the other. Evaluating and checking results is described in detail in chapter 5 *Results Evaluation* on page 36.

4.1 Stresses by Cross-section

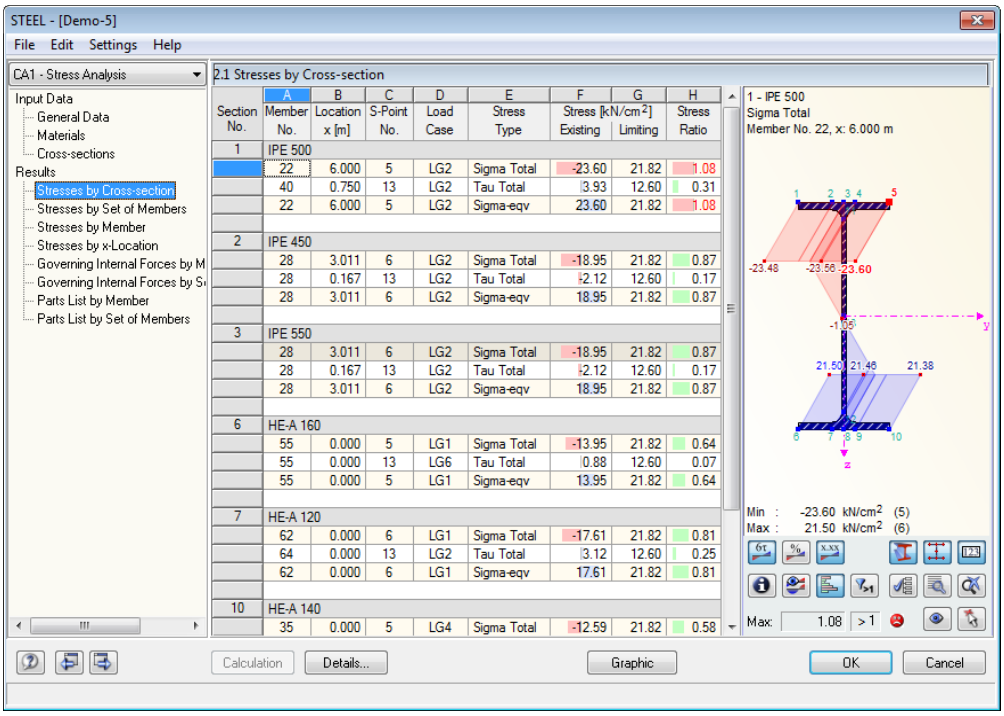


Figure 4.1: Table 2.1 *Stresses by Cross-section*

The table shows the maximum stress ratios of all members selected for the design, resulting from the internal forces of the governing loads or combinations. The results are listed by cross-sections.

The stress components displayed in column E *Stress Type* refer to the settings selected in the dialog box *Stresses - Filter* (see Figure 5.3, page 37). To open the dialog box, use the button shown on the left.

Member No.

For each cross-section and each stress type, the table shows the number of the member with the maximum stress ratio.

Location x

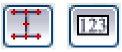
The column shows the respective x-location on the member for which the maximum value of stress has been determined. For the table output, the program uses the following RSTAB member locations *x*:

- Start and end node
- Partition points according to possibly defined member division
- Member division according to specification for member results (*Options* tab of RSTAB dialog box *Calculation Parameters*)
- Extreme values of internal forces

S-Point No.

The design is carried out on certain stress points of the cross-section. These points on the cross-section are defined by centroidal distances, static moments and cross-section thicknesses. Due to these cross-section properties, the design according to Table 3.1 and Table 3.2 is possible.

All default cross-sections of the library as well as the SHAPE-THIN and SHAPE-MASSIVE cross-sections are already provided with stress points on the design relevant cross-section locations. The parameters for user-defined cross-sections must be imported or defined manually.



The dialog graphic on the right shows the stress points including numbering. The active 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, use the button [Info about Current Cross-section]. First, the dialog box *Info about cross-section* opens showing the list of the section properties. At the bottom right below the dialog graphic, you find the button [Details of Stress Points] that provides access to further stress point information.

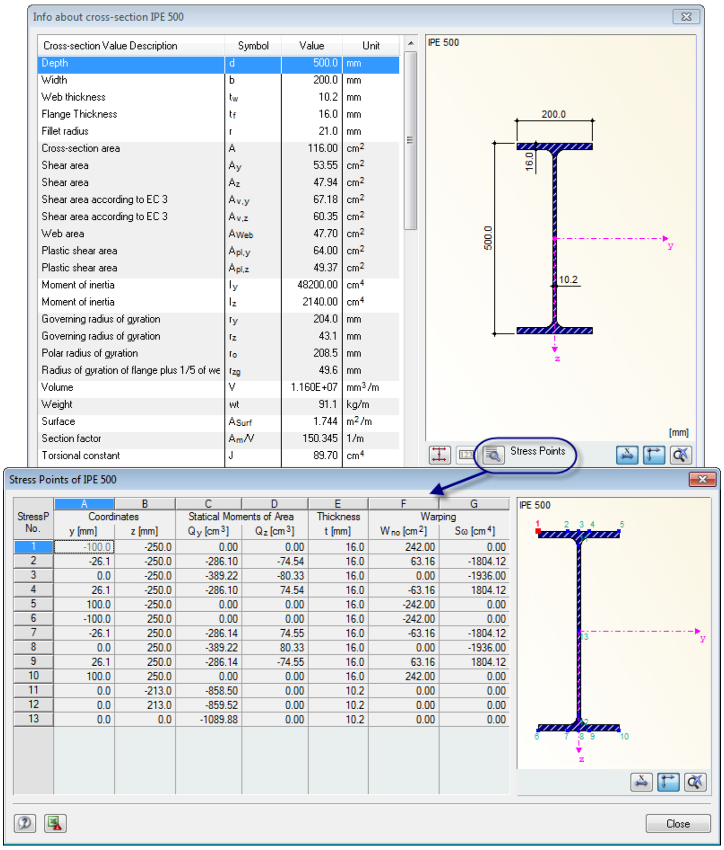


Figure 4.2: Info about cross-section: *Stress Points*

The columns *Coordinates y* and *z* show information about the centroidal distances e_y and e_z. The columns *Statical Moments of Area Q_y* and *Q_z* list the first moments of area in relation to the principal axis y or z. *Thickness t* represents the component thickness at the corresponding stress point. The values contained in the *Warping* columns are not relevant for the design carried out with STEEL.

The stress analysis is carried out for each single stress point so that normally the components of the maximum stresses must not be summed up for the combined calculation (for example σ_{eqv}): Often, the maximum stresses occur at different stress points. You have to superimpose the stress components that are available on the same stress points. Table 2.5 (see chapter 4.5, page 31), for example, or the window *Cross-section Values and Stress Diagram* (see Figure 5.5, page 39) allow you to evaluate the results by stress points.

Load Case

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

Stress Type

The normal stresses σ_{total}, the shear stresses τ_{total} and the equivalent stresses σ_{eqv} are listed by default. The determination of these stresses is shown in Table 3.1, Table 3.2 and Table 3.3 on page 21 to 23.

You can display all stress components affecting the total stresses in order to check the data (see Figure 4.3). The individual stress components can be selected in the dialog box *Stresses - Filter* (see Figure 5.3, page 37). To access the dialog box, use the button shown on the left.

Stress Existing

This column displays the extreme values of the existing stresses determined according to the equations of Table 3.1, Table 3.2 and Table 3.3 shown on page 21 to 23.

2D systems with unsymmetrical cross-sections

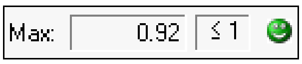
For unsymmetrical cross-sections such as L-sections in 2D systems, please note the following: As member rotation angles are only available for 0° and 180°, only the component in direction of the member axis y will be taken into account for the division of moments. The component about z is not applied in a 2D system. Therefore, STEEL designs only the moment about the y-axis. It is recommended to define cross-sections, that are stressed not only in direction of the principal axis y, as spatial 3D models. This is the only way how the stresses can be determined correctly.

Stress Limiting

This column shows the limit stresses of table 1.2 (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



For each stress component, STEEL determines the quotient from the existing and the limit stress. If the limit stress is not exceeded, the ratio is less than or equal to 1.00 and the stress design was carried out successfully.

The colored scales represent the stress ratios of each cross-section.

4.2 Stresses by Set of Members

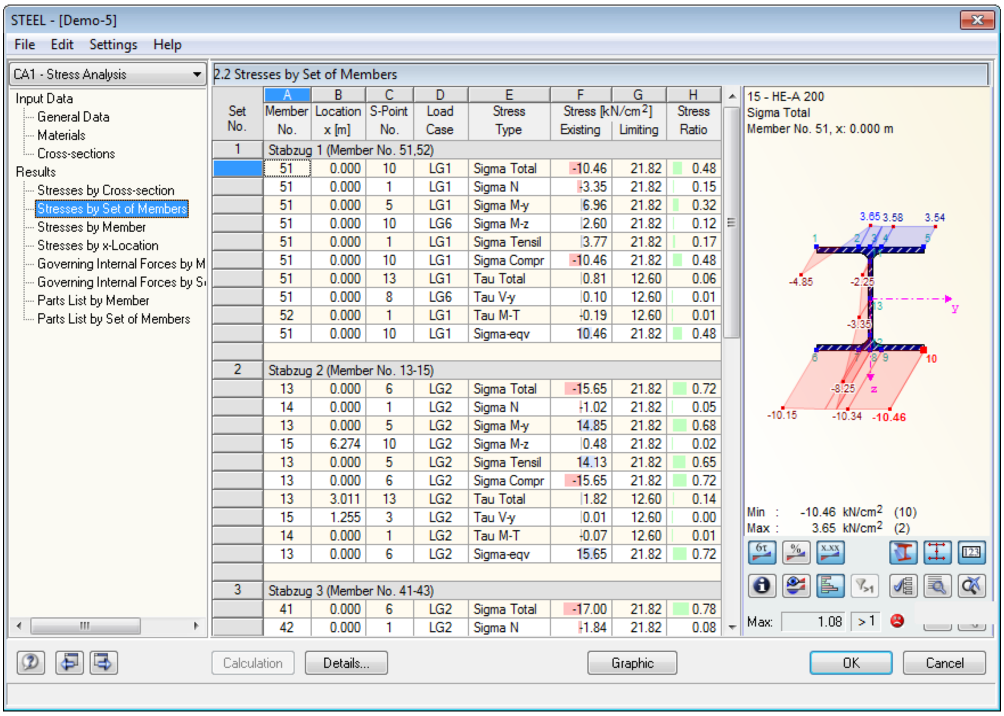


Figure 4.3: Table 2.2 *Stresses by Set of Members*

This table lists the maximum stress ratios sorted by sets of members. Details on the table columns can be found in the previous chapter 4.1. The table column *Member No.* shows the number of the member that bears the maximum stress ratio within the set of members.

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

4.3 Stresses by MemberStresses

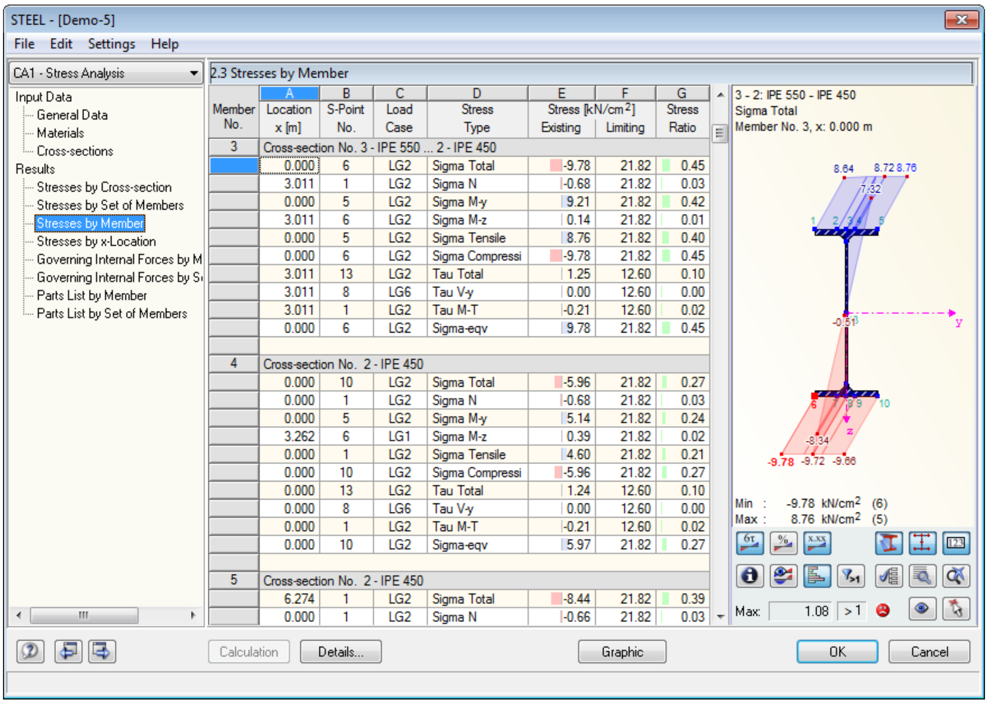


Figure 4.4: Table 2.3 *Stresses by Member*

This table lists the maximum stresses and stress ratios sorted by member numbers. The different columns are described in detail in chapter 4.1 on page 26.

If a taper is used, both cross-section descriptions are displayed.

4.4 Stresses by x-Location

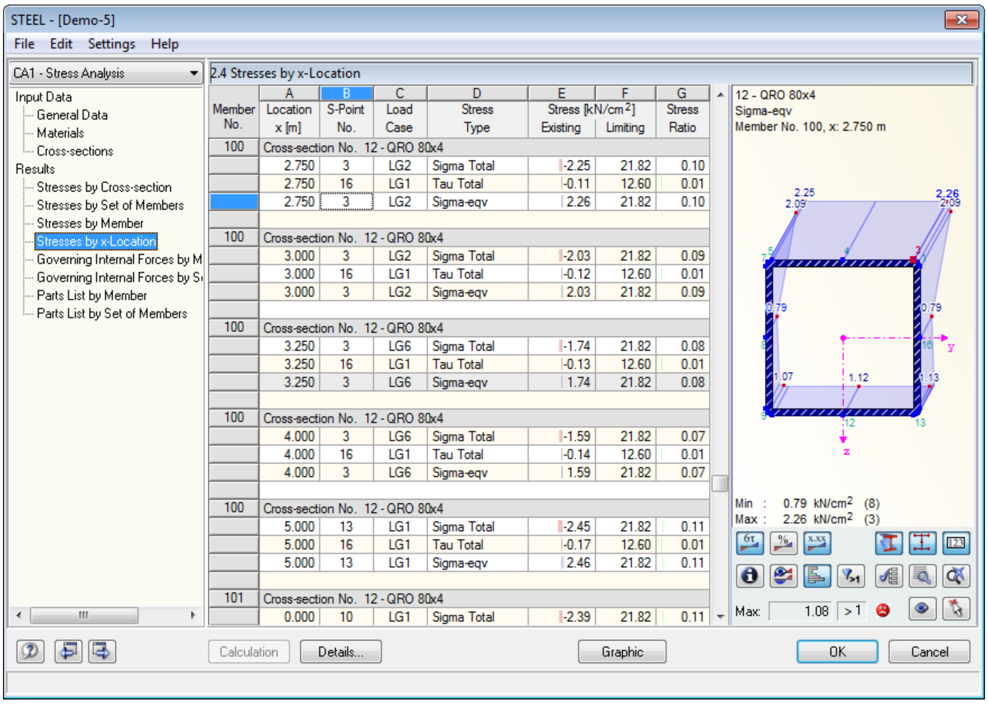


Figure 4.5: Table 2.4 *Stresses by x-Location*

This results table lists the maximum stresses of each member at the locations x resulting from the divisions defined in RSTAB:

- Start and end node
- Partition points according to possibly defined member division
- Member division according to specification for member results (*Options* tab of RSTAB dialog box *Calculation Parameters*)
- Extreme values of internal forces

4.5 Stresses at Every Stress Point

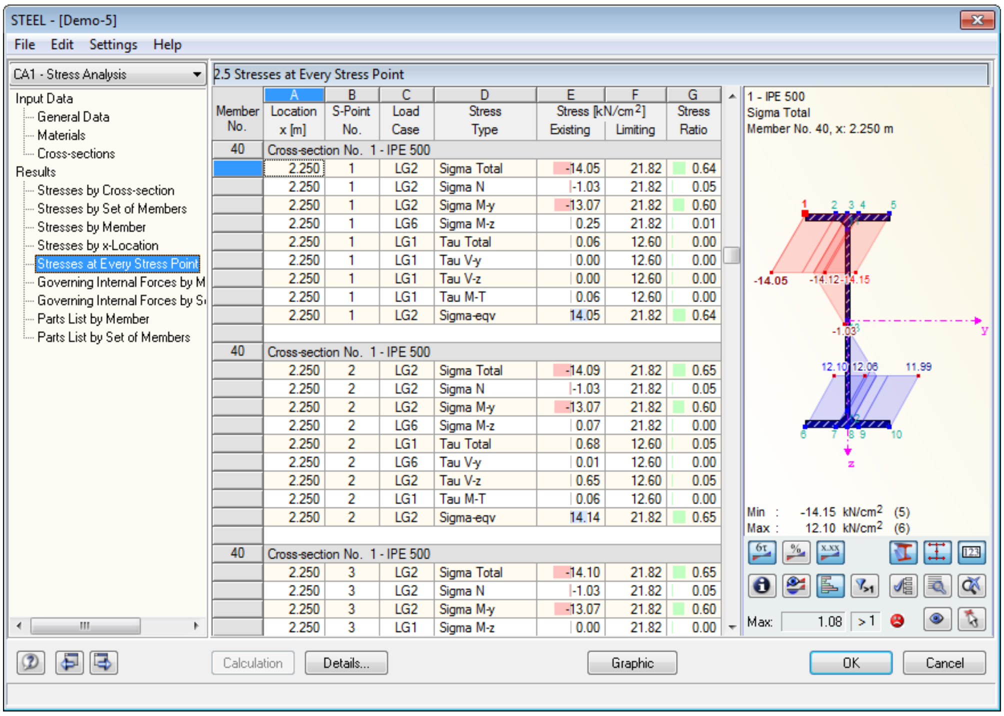


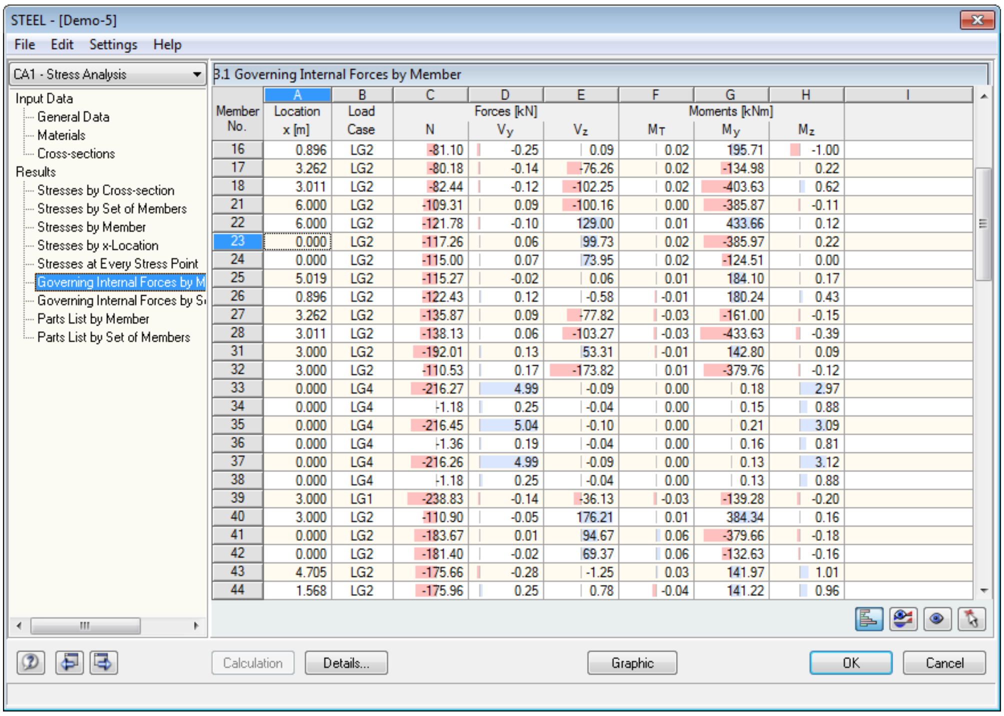
Figure 4.6: Table 2.5 *Stresses at Every Stress Point*

This results table is set inactive by default because a results evaluation by stress points is only required in exceptional cases. To display the table, tick the corresponding check box in the *Details* dialog box (see Figure 3.1, page 18). To open the dialog box, use the [Details] button shown on the left which is available in all tables.

The table manages a large amount of data. As STEEL determines the maximum stresses automatically, and thus the governing stress points, you usually do not need the display of table 2.5. In addition, the tables 2.1 to 2.4 provide appropriate evaluation options by clicking the button [Cross-section Values and Extended Stress Diagram]. The dialog box *Cross-section Values and Stress Diagram* opens (see Figure 5.5, page 39) where you can check the results of every single stress point graphically and numerically.

The stresses in table 2.5 are listed for each member according to *Location x* and *S-Point No.* (stress point). The different columns of this table are described in detail in chapter 4.1 on page 26.

4.6 Governing Internal Forces by Member



	A	B	C	D	E	F	G	H
Member No.	Location x [m]	Load Case	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z
16	0.896	LG2	-81.10	-0.25	0.09	0.02	195.71	-1.00
17	3.262	LG2	-80.18	-0.14	-76.26	0.02	-134.98	0.22
18	3.011	LG2	-82.44	-0.12	-102.25	0.02	-403.63	0.62
21	6.000	LG2	-109.31	0.09	-100.16	0.00	-385.87	-0.11
22	6.000	LG2	-121.78	-0.10	129.00	0.01	433.66	0.12
23	0.000	LG2	-117.26	0.06	99.73	0.02	-385.97	0.22
24	0.000	LG2	-115.00	0.07	73.95	0.02	-124.51	0.00
25	5.019	LG2	-115.27	-0.02	0.06	0.01	184.10	0.17
26	0.896	LG2	-122.43	0.12	-0.58	-0.01	180.24	0.43
27	3.262	LG2	-135.87	0.09	-77.82	-0.03	-161.00	-0.15
28	3.011	LG2	-138.13	0.06	-103.27	-0.03	-433.63	-0.39
31	3.000	LG2	-192.01	0.13	53.31	-0.01	142.80	0.09
32	3.000	LG2	-110.53	0.17	-173.82	0.01	-379.76	-0.12
33	0.000	LG4	-216.27	4.99	-0.09	0.00	0.18	2.97
34	0.000	LG4	-1.18	0.25	-0.04	0.00	0.15	0.88
35	0.000	LG4	-216.45	5.04	-0.10	0.00	0.21	3.09
36	0.000	LG4	-1.36	0.19	-0.04	0.00	0.16	0.81
37	0.000	LG4	-216.26	4.99	-0.09	0.00	0.13	3.12
38	0.000	LG4	-1.18	0.25	-0.04	0.00	0.13	0.88
39	3.000	LG1	-238.83	-0.14	-36.13	-0.03	-139.28	-0.20
40	3.000	LG2	-110.90	-0.05	176.21	0.01	384.34	0.16
41	0.000	LG2	-183.67	0.01	94.67	0.06	-379.66	-0.18
42	0.000	LG2	-181.40	-0.02	69.37	0.06	-132.63	-0.16
43	4.705	LG2	-175.66	-0.28	-1.25	0.03	141.97	1.01
44	1.568	LG2	-175.96	0.25	0.78	-0.04	141.22	0.96

Figure 4.7: Table 3.1 *Governing Internal Forces by Member*

Details...

This table displays for each member the governing internal forces resulting in maximum stress ratios. The output refers to the equivalent stress σ_{eqv} by default. In the *Details* dialog box (see Figure 3.1, page 18) that can be opened by using the [Details] button, you can change the reference for the output of the governing internal forces and set another stress component.

If you *Calculate Stresses from Single Load Cases in CO and then Combine Stresses according to CO Criteria*, you prevent to evaluate the result rows of the RSTAB table 3.1 *Members - Internal Forces* directly. Instead, the compressive, tensile and shear stresses available in the single load cases will be summed up respectively. The equivalent stress σ_{eqv} will be determined by the components of σ_{total} and τ_{total}. For this reason, in case of load combinations, you can retrace the internal forces displayed in table 3.1 only by retroactive calculation.

Location x

The column shows the respective x-location where the member's maximum stress ratio occur.

Load Case

This column indicates the numbers of the load case or load group, load combination or super combination whose internal forces result in the maximum stress ratio on the member.

Forces / Moments

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

4.7 Governing Internal Forces by Set of Members

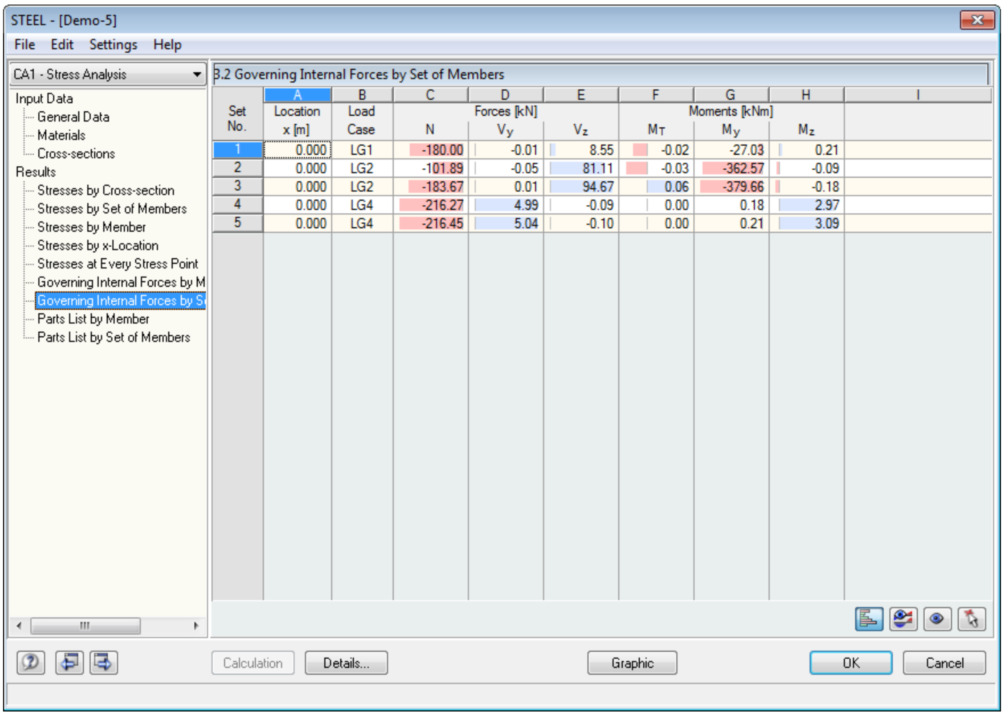
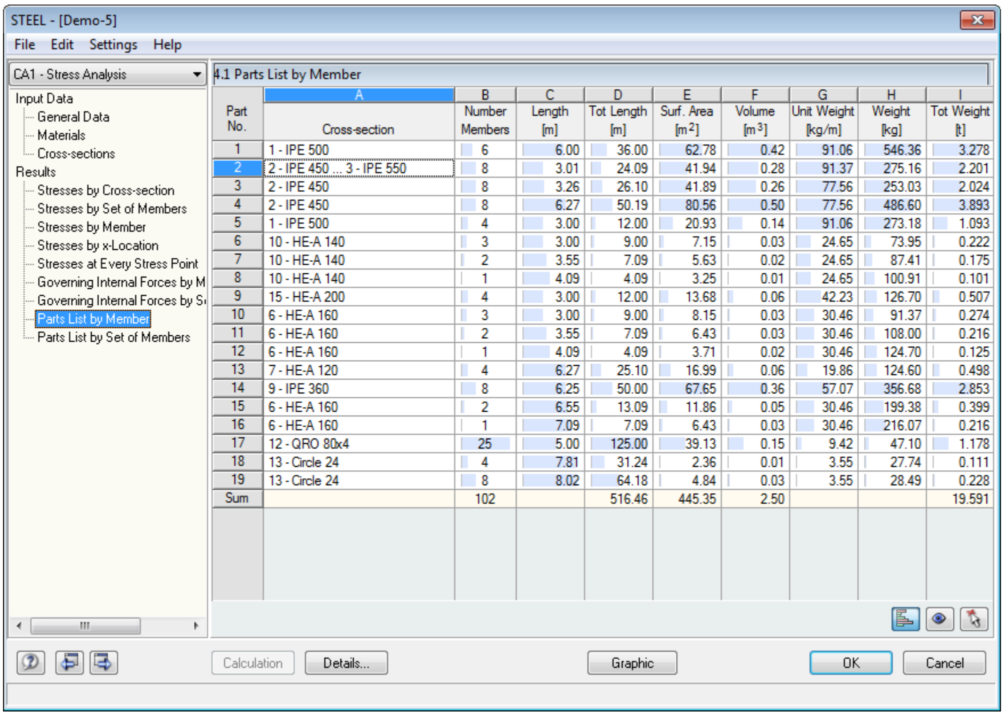


Figure 4.8: Table 3.2 *Governing Internal Forces by Set of Members*

This result table is displayed when sets of members have been selected for design. The internal forces that result in the maximum stress ratios are shown for each set of members.

4.8 Parts List by Member

The output is completed by a parts list.



	A	B	C	D	E	F	G	H	I
Part No.	Cross-section	Number Members	Length [m]	Tot Length [m]	Surf. Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Tot Weight [t]
1	1 - IPE 500	6	6.00	36.00	62.78	0.42	91.06	546.36	3.278
2	2 - IPE 450 ... 3 - IPE 550	8	3.01	24.09	41.94	0.28	91.37	275.16	2.201
3	2 - IPE 450	8	3.26	26.10	41.89	0.26	77.56	253.03	2.024
4	2 - IPE 450	8	6.27	50.19	80.56	0.50	77.56	486.60	3.893
5	1 - IPE 500	4	3.00	12.00	20.93	0.14	91.06	273.18	1.093
6	10 - HE-A 140	3	3.00	9.00	7.15	0.03	24.65	73.95	0.222
7	10 - HE-A 140	2	3.55	7.09	5.63	0.02	24.65	87.41	0.175
8	10 - HE-A 140	1	4.09	4.09	3.25	0.01	24.65	100.91	0.101
9	15 - HE-A 200	4	3.00	12.00	13.68	0.06	42.23	126.70	0.507
10	6 - HE-A 160	3	3.00	9.00	8.15	0.03	30.46	91.37	0.274
11	6 - HE-A 160	2	3.55	7.09	6.43	0.03	30.46	108.00	0.216
12	6 - HE-A 160	1	4.09	4.09	3.71	0.02	30.46	124.70	0.125
13	7 - HE-A 120	4	6.27	25.10	16.99	0.06	19.86	124.60	0.498
14	9 - IPE 360	8	6.25	50.00	67.65	0.36	57.07	356.68	2.853
15	6 - HE-A 160	2	6.55	13.09	11.86	0.05	30.46	199.38	0.399
16	6 - HE-A 160	1	7.09	7.09	6.43	0.03	30.46	216.07	0.216
17	12 - QRO 80x4	25	5.00	125.00	39.13	0.15	9.42	47.10	1.178
18	13 - Circle 24	4	7.81	31.24	2.36	0.01	3.55	27.74	0.111
19	13 - Circle 24	8	8.02	64.18	4.84	0.03	3.55	28.49	0.228
Sum		102		516.46	445.35	2.50			19.591

Figure 4.9: Table 4.1 *Parts List by Member*

By default, the list contains only the designed members. If you want to display a parts list with all members of the structure, select the corresponding option in the *Details* dialog box (see Figure 3.1, page 18). To open the dialog box, use the [Details] button shown on the left.

Part No.

The program assigns automatically part numbers for similar members.

Cross-section

This column lists the cross-section numbers and descriptions.

Number Members

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

Length

This column displays the length of an individual member.

Tot Length

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

Surf. Area

The program indicates the surface area of the respective parts in relation to the total length. The surface area is determined from the *Surface* of the cross-sections. You can find the relevant entry in the cross-section information, available in table 1.3 to 2.5 (see Figure 4.2, page 27).

Volume

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

Unit Weight

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

Weight

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

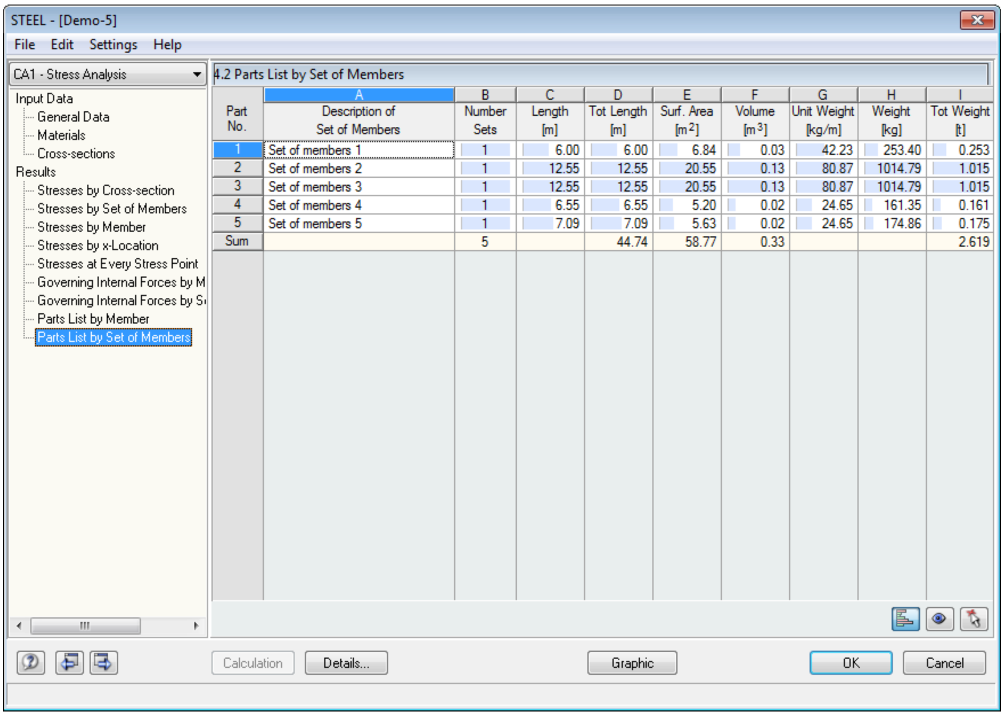
Tot Weight

The final column indicates the total weight of the respective part.

Sum

At the bottom of the list, you find a summary showing the sums of column B, D, E, F and I. The data field in the column *Tot Weight* shows the amount of steel that is required.

4.9 Parts List by Set of Members



Part No.	A Description of Set of Members	B Number Sets	C Length [m]	D Tot Length [m]	E Surf. Area [m2]	F Volume [m3]	G Unit Weight [kg/m]	H Weight [kg]	I Tot Weight [t]
1	Set of members 1	1	6.00	6.00	6.84	0.03	42.23	253.40	0.253
2	Set of members 2	1	12.55	12.55	20.55	0.13	80.87	1014.79	1.015
3	Set of members 3	1	12.55	12.55	20.55	0.13	80.87	1014.79	1.015
4	Set of members 4	1	6.55	6.55	5.20	0.02	24.65	161.35	0.161
5	Set of members 5	1	7.09	7.09	5.63	0.02	24.65	174.86	0.175
Sum		5		44.74	58.77	0.33			2.619

Figure 4.10: Table 4.2 *Parts List by Set of Members*

The final STEEL table is displayed when sets of members have been selected for design. The advantage of the output by sets of members is the display of a summarized parts list for an entire structural group (for example a frame).

Details on the table columns can be found in the previous chapter 4.8. When different cross-sections are used in the set of members, the program averages the surface area, the volume and the unit weight.

5. Results Evaluation

When the design is complete, several options are available for results evaluation. The buttons below the graphic can be helpful for the evaluation process.

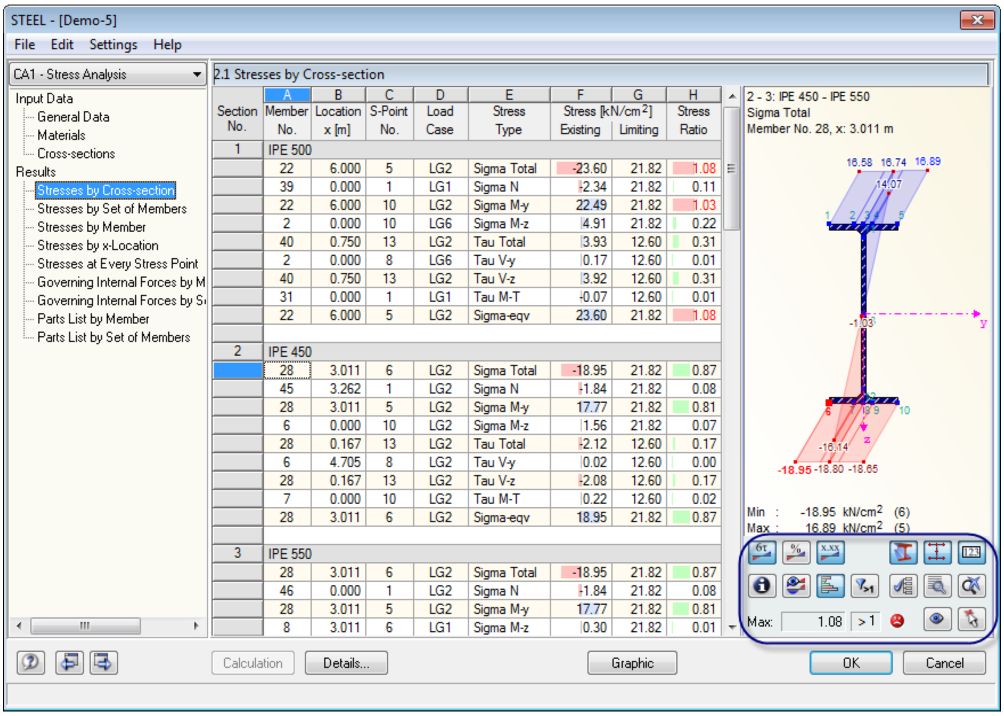
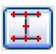


Figure 5.1: Buttons for results evaluation

The buttons are reserved for the following functions:

Button	Description	Function
	Stress diagram	Turns on and off the graphic display of stresses on the cross-section
	Ratio	Turns on and off the graphic display of stress ratios on the cross-section
	Values	Switches on and off the display of values in the stress or stress ratio graphic
	Cross-section outlines	Displays the cross-section's shape in the cross-section graphic
	Stress points	Turns on and off the display of stress points in the cross-section graphic
	Numbering	Switches on and off the numbering of stress points
	Cross-section info	Opens the dialog box *Info about cross-section* showing the properties of the currently selected cross-section → Figure 4.2, page 27
	Result diagrams	Opens the window *Result Diagram on Member* → Chapter 5.4, page 44

	Show Color Bars	Turns on and off the colored relation scales in the results tables
	Exceeding	Displays only the rows where the ratio is more than 1, thus the design is failed
	Stress selection	Opens the dialog box *Stresses - Filter* → Chapter 5.1, page 37
	Extended Stress Diagram	Opens the dialog box *Cross-section Values and Stress Diagram* → Chapter 5.2, page 38
	Full view	Resets the stress graphic in full view (zooming is possible by using the wheel button, moving by drag-and-drop)
	View mode	Jumps to the RSTAB work window to change the view → Chapter 5.3, page 40
	Member selection	Enables the selection of a member in the RSTAB window to display its stresses in the table

Table 5.1: Buttons of results tables 2.1 to 2.5

5.1 Selection of Stresses

Subsequent to the design, the following stress types are preset in the results tables:

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

Use the button [Select Stresses to Show] to activate further stress components. In this way, you can check the components affecting the total stress.

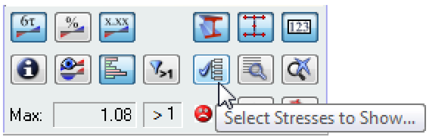


Figure 5.2: Button *Select Stresses to Show*

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

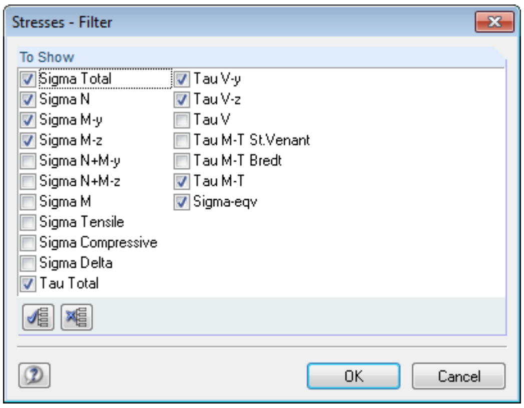


Figure 5.3: Dialog box *Stresses - Filter*

The different stresses are described in Table 3.1 and Table 3.2 on page 21 and 23.

The two dialog buttons facilitate the selection. They are reserved for the following functions:

Button	Description	Function
	Select all	All stress components are selected.
	Deselect all	All stresses are canceled.

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 must not be summed up for a combined calculation (for example σ_{total}): Often, the maximum stresses occur at different stress points. You have to superimpose the stress components that are available on the same stress points.

The selection made in the dialog box *Stresses - Filter* affects also the printout report. The printout includes the stress components that are active in the tables.

5.2 Results on Cross-section

In addition to the stresses listed in tables, a stress graphic is displayed to the right of the results tables. The graphic is dynamic, that means it shows the stress diagram of the current x-location or stress point that is determined by the pointer position in the table to the left. The currently selected stress point is highlighted in red in the graphic.

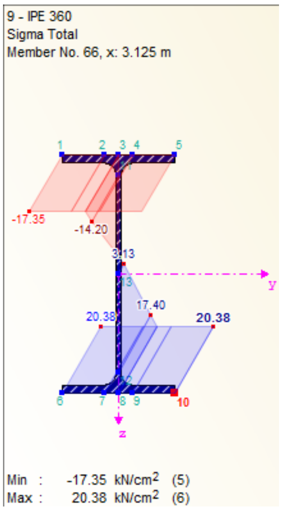


Figure 5.4: Diagram of normal stresses on cross-section

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.

The functions of the buttons below the graphic are described in Table 5.1 on page 37. The buttons control the graphic display with regard to

- the stress or stress ratio diagram including values
- the cross-section outlines
- the stress points and their numbering.

To evaluate the stresses for each stress point in detail, use the button [Show or Print Cross-section Values and Extended Stress Diagram]. The dialog box *Cross-section Values and Stress Diagram* opens.

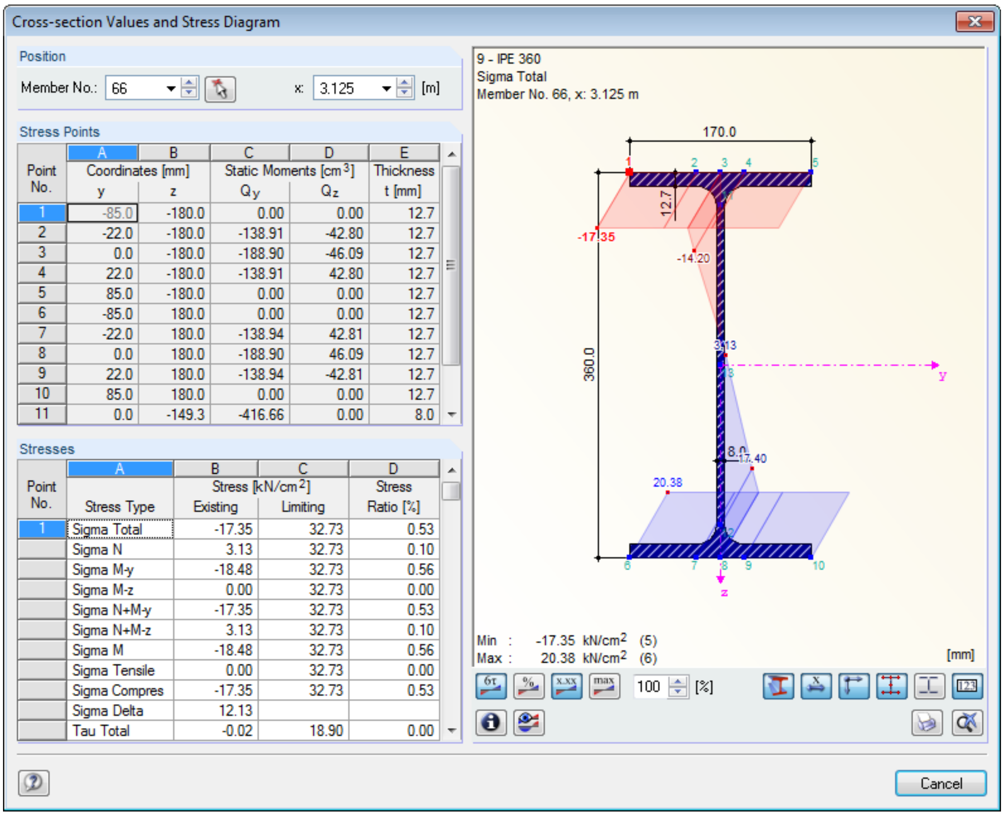


Figure 5.5: Dialog box *Cross-section Values and Stress Diagram*

The current *Member No.* and the location *x* 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 the stress points of the cross-section. The two *Coordinates* columns show the respective centroidal distances *y* and *z*. The *Static Moments* columns display the static moments 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, the stress components are displayed for the stress point that is currently selected in the dialog section above. To visualize a particular stress component in the dynamic graphic on the right, select it by clicking the relevant entry.

Most of the buttons below the graphic are identical with the buttons in the results tables (see Table 5.1, page 37). As usual, they are described by *ScreenTips*. The [Print] button enables the printout of the current stress graphic on the cross-section. For more information, see chapter 6.2.1 on page 47.

5.3 Results in the RSTAB Model

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

RSTAB background graphic and view mode

The RSTAB graphic in the background may be useful when you want to check the position of a particular member in the model. The member that is selected in the STEEL results table is highlighted in the selection color in the RSTAB background graphic. In addition, an arrow indicates the member's x-location that is displayed in the active STEEL table row.

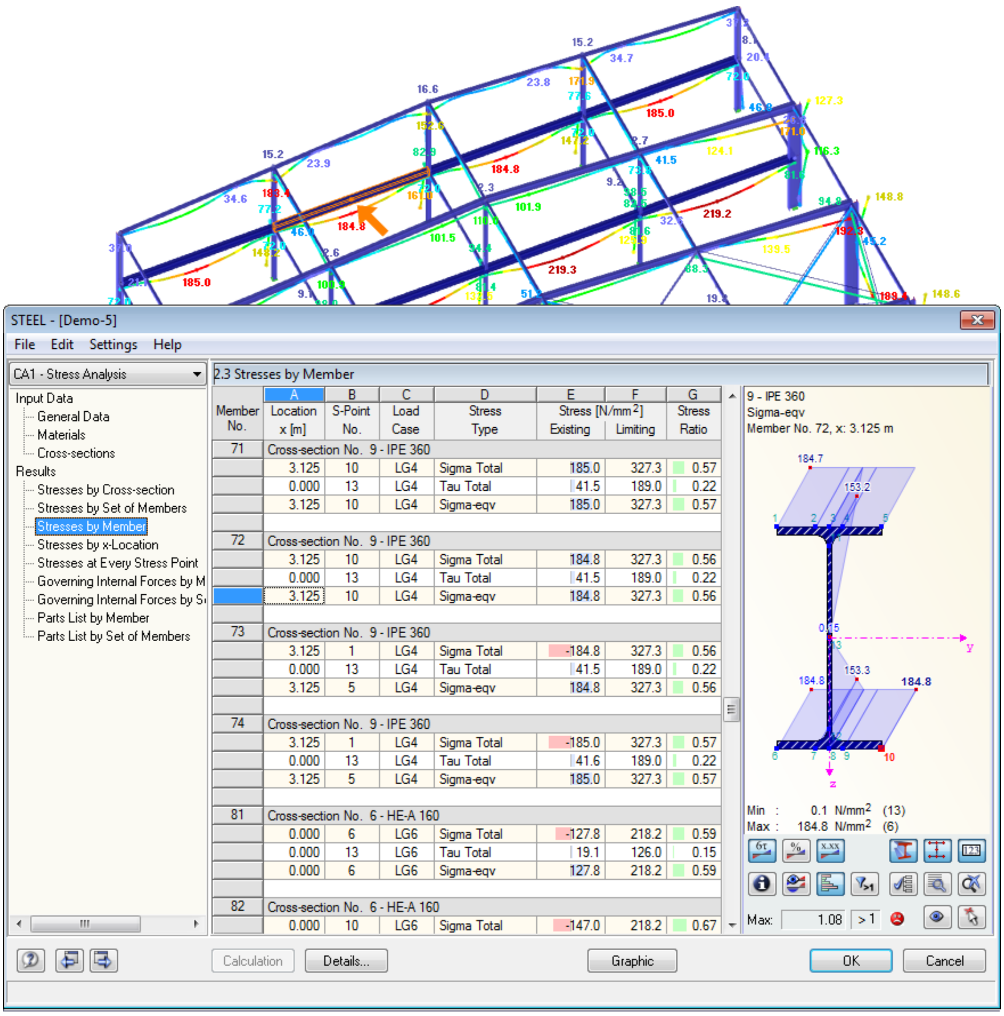
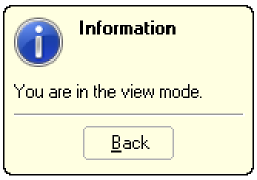


Figure 5.6: Indication of the member and the current *Location x* in the RSTAB model

If you move the STEEL window to another place in the display and you still cannot see the graphic clearly, use the button [Jump to Graphics] to activate the *view mode*: The STEEL window will be hidden so that you can adjust the view in the RSTAB user interface appropriately. The view mode provides the functions of the *View* menu, for example zooming, moving or rotating the display.



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

RSTAB work window

It is also possible to visualize the stresses and stress ratios directly in the structural model: Click the [Graphic] button to quit the STEEL module. Now, the individual design results like the internal forces or deformations of a RSTAB load case are displayed in the RSTAB work window.

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

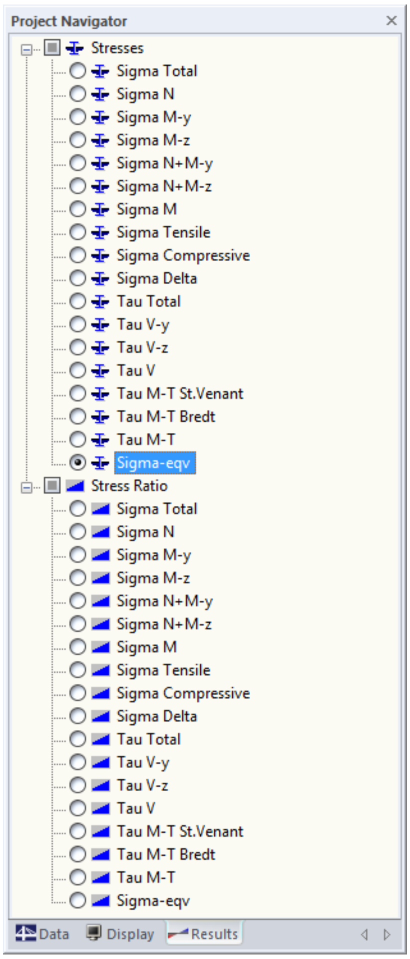
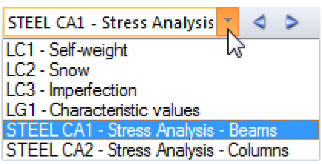


Figure 5.7: *Results* navigator

To turn the display of design results on or off, use the button [Results on/off] shown on the left. To display the result values in the graphic, use the toolbar button [Show Result Values] to the right.

As the RSTAB tables are of no relevance for the evaluation of STEEL results, you may deactivate them.



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

The graphical representation of results can be set in the *Display* navigator, by opening *Results* and selecting *Members*. Stresses and stress ratios are two-colored by default.

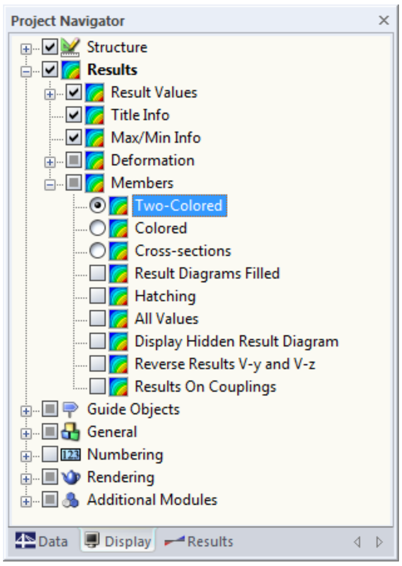


Figure 5.8: *Display* navigator: Results → Members

The representation of stresses is conforming to signs: Positive stresses are displayed in blue, in direction of the positive member axis z. Negative stresses are red, applied in opposite direction. Therefore, it is possible that the stress diagram on the member, in case of discontinuity for example due to concentrated loads, changes the sign and thus color and side.

In case of a multicolor representation (options *Cross-sections* or *Colored*), the color panel is available, providing common control functions. The panel functions are described in detail in the RSTAB manual, chapter 4.4.6, page 67.

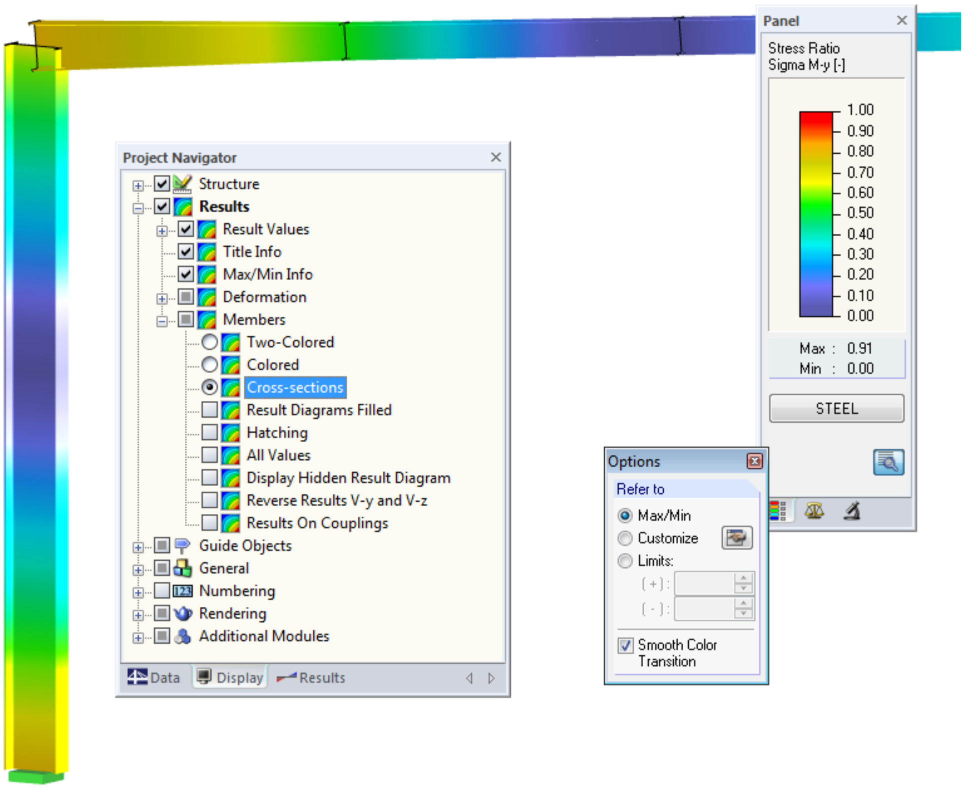


Figure 5.9: Stress ratios due to bending moments M_y with display option *Cross-sections*

In the *Factors* tab, you can scale the design results, as you know it from the member internal forces. If you enter the factor *0* in the input field *Member Diagrams*, the results will be represented without scaling but with an increased line thickness.

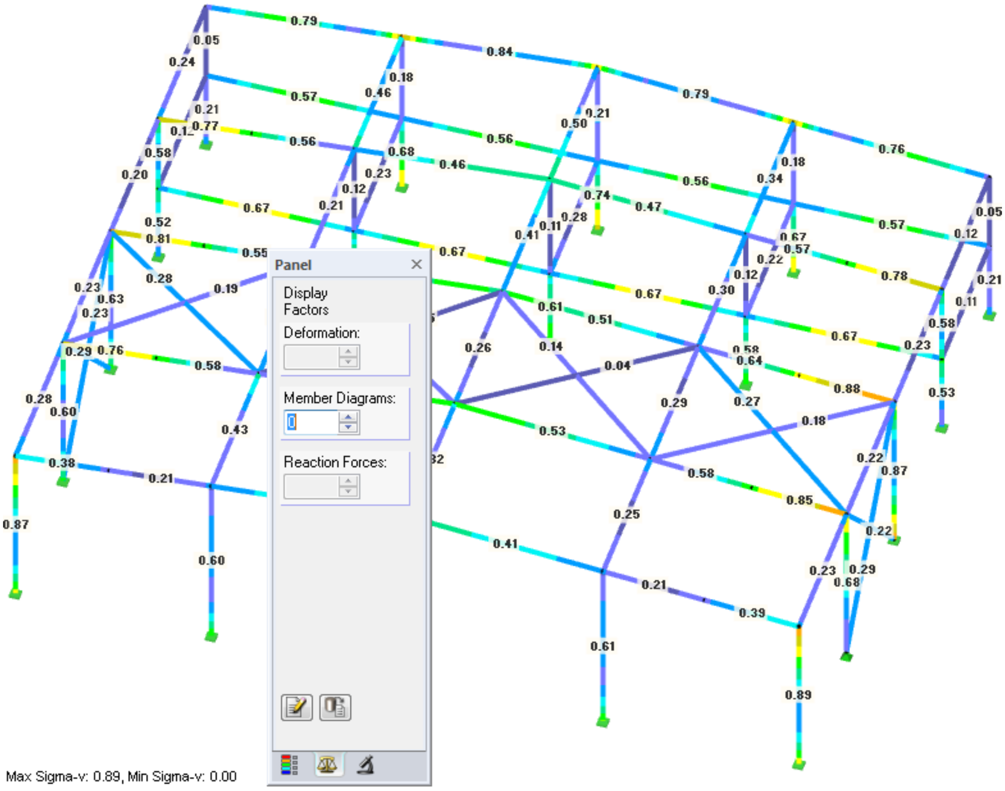
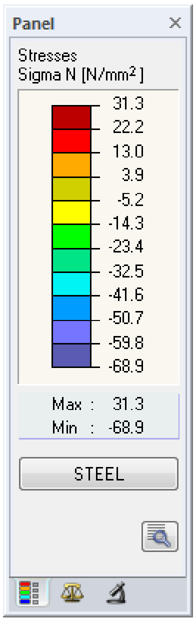


Figure 5.10: Panel tab *Factors*



The graphics can be transferred like RSTAB graphics to the printout report (see chapter 6.2.2, page 49).

To return to the STEEL module, use the [STEEL] button in the panel.

5.4 Result Diagrams

The result diagram can be useful if you want to see a member's result distribution displayed graphically. Select the member (or set of member) in the STEEL results table by placing the pointer in the corresponding table row and open the result diagram by clicking the button shown on the left. You find the button below the stress graphic (see Figure 5.1, page 36).

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

select **Result Diagrams on Selected Members** on the **Results** menu,

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

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

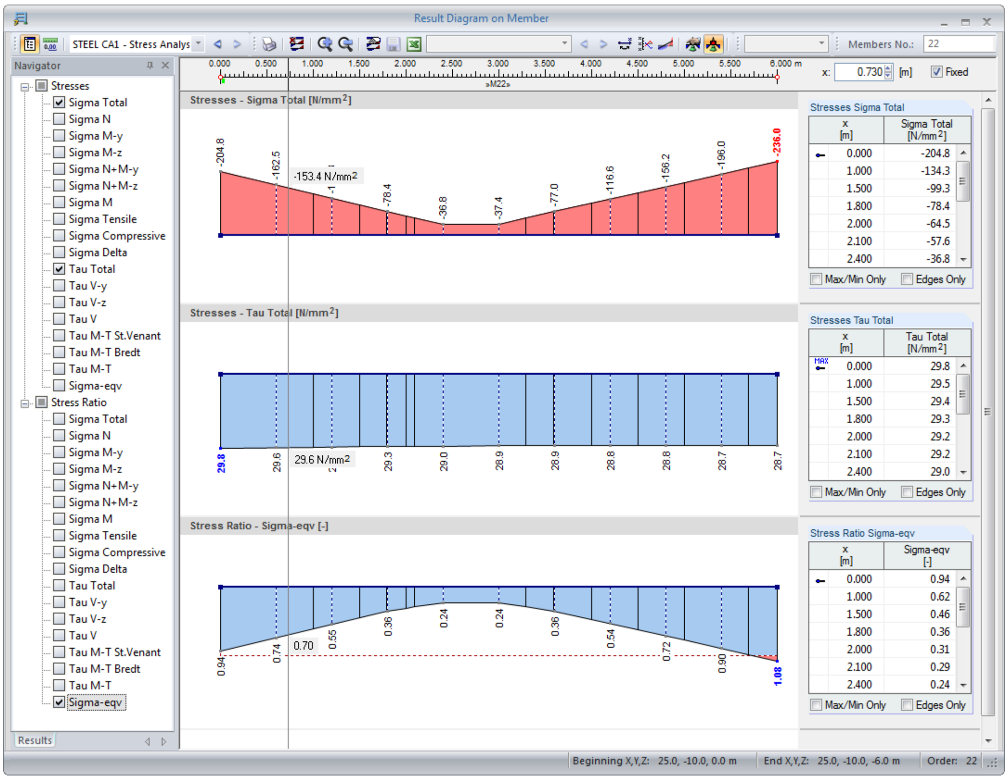
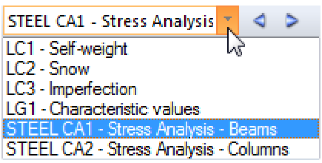


Figure 5.11: Dialog box *Result Diagram on Member*



In the navigator on the left, you can select the stresses and stress ratios that you want to be displayed in the result diagram. Use the list in the toolbar above to choose the relevant STEEL design case.

For more detailed information on the dialog box *Result Diagram on Member*, see the RSTAB manual, chapter 9.8.4, page 205.

5.5 Filter for Results

In addition to the results tables which already allow for a particular selection according to certain criteria because of their structure, you can use the filter options described in the RSTAB manual to evaluate the STEEL design results graphically.

Generally, you can take advantage of already defined partial views (see RSTAB manual, chapter 9.8.6, page 209) used to group objects appropriately.

Filtering designs

The stresses and stress ratios can be used easily as filter criteria in the RSTAB work window. To apply this filter function, the panel must be displayed. If the panel is not active,

select **Control Panel (Colour scale, Factors, Filter)** on the **View** menu

or use the toolbar button shown on the left.

The panel is described in the RSTAB manual, chapter 4.4.6, page 67. The filter settings for the results must be defined in the panel tab *Color spectrum*. As this tab is not available for the two-colored results display, you have to set the display option *Colored* or *Cross-sections* in the *Display* navigator.

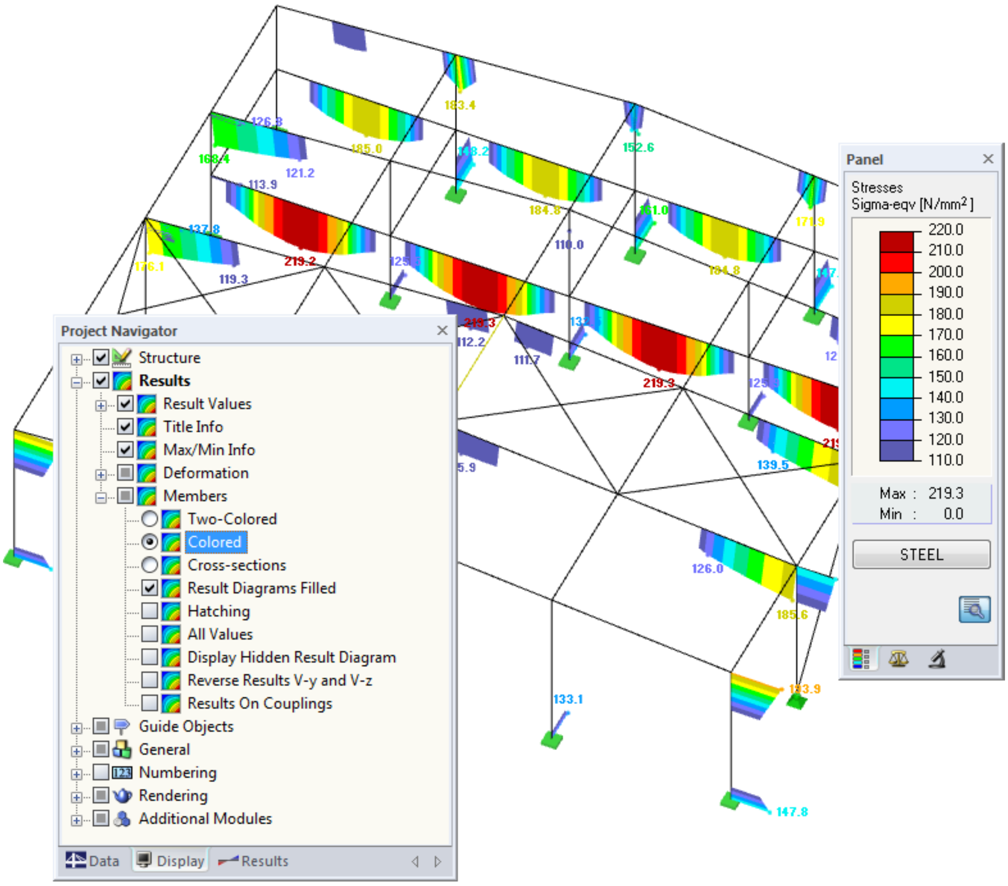


Figure 5.12: Filtering equivalent stresses with adjusted color spectrum

If you use a colored results display, you can use the panel to define that only equivalent stresses for example larger than +110 N/mm^2 are displayed. Furthermore, the color spectrum can be adjusted in such a way that a color range covers exactly 10 N/mm^2 as shown in the figure above.

When you select the option *Display Hidden Result Diagram* (under *Results* → *Members* in the *Display* navigator), you can even display all stress diagrams that do not fulfill the conditions. Those diagrams will be represented by dotted lines.

Filtering members

In the *Filter* tab of the control panel, you can define he numbers of the members whose results should be shown exclusively, which means filtered. The function is described in detail in the RSTAB manual, chapter 4.4.6, page 70.

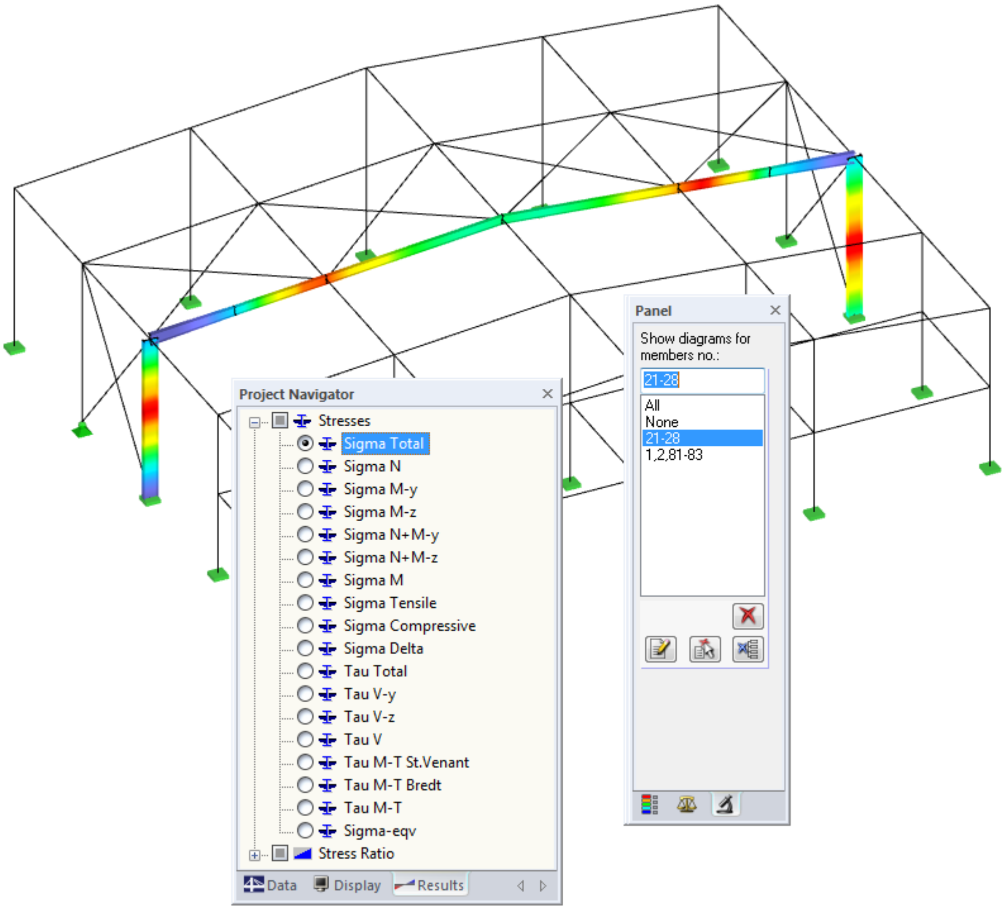


Figure 5.13: Filtering members: normal stresses of a frame

Unlike the partial view function, the structure is now displayed completely in the graphic. The figure above shows the normal stresses of a frame inside a hall. The remaining designed members are displayed in the model, but are shown without stresses.

6. Printout

6.1 Printout Report

Similar to RSTAB, the program generates a printout report for the STEEL results that can be added by graphics and descriptions. In the report you also decide which results will finally appear in the printout.

The printout report is described in detail in the RSTAB manual. In particular, chapter 10.1.3.4 *Selecting Data of Add-on Modules* on page 227 provides information concerning the selection of input and output data in add-on modules.

For complex structures with a high number of design cases, it is recommended to split the data into several small printout reports which allows for a clearly-arranged printout and a faster work.

The printout report only shows the stress types that are active in the STEEL design case for the results tables. For example, if you want to print the stresses due to axial force, activate the stresses σ_N in the STEEL add-on module (see chapter 5.1 *Selection of Stresses*, page 37).

6.2 Print STEEL Graphics

In addition to the graphical results evaluation, it is possible to provide the stresses and design ratios on the cross-section as well as in the RSTAB model for the printout. The graphics can be either integrated in the printout report or sent directly to the printer. Printing graphics is described in detail in the RSTAB manual, chapter 10.2.

6.2.1 Results on Cross-section

The access to the print function is not directly available. Instead, open the dialog box *Cross-section Values and Stress Diagram* (see Figure 5.5, page 39) by using the button [Show or Print Cross-section Values and Extended Stress Diagram] available in each results table.

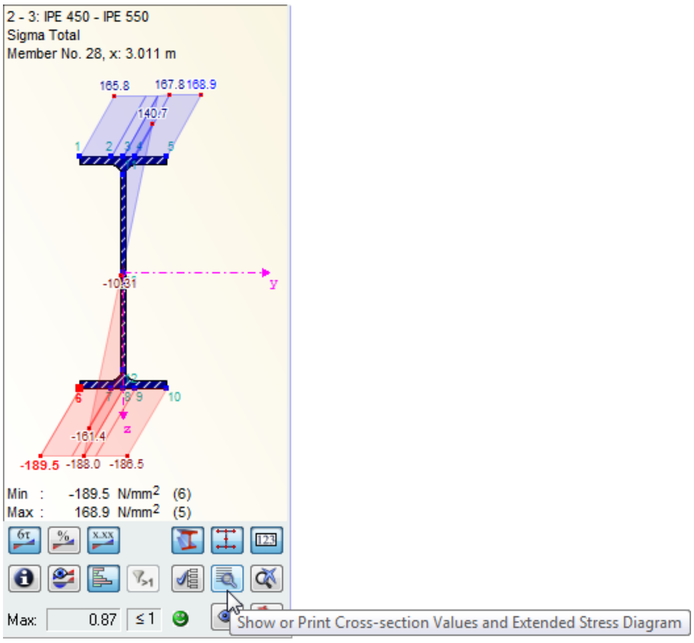


Figure 6.1: Button *Show or Print Cross-section Values and Extended Stress Diagram* in the graphic of the results tables

In the dialog box *Cross-section Values and Stress Diagram* (see Figure 5.5, page 39), specify the member, the relevant x-location and the type of stress for which you want to print the diagram. With the [Print] button in the bottom right corner of the dialog box, you open the following printing dialog box.

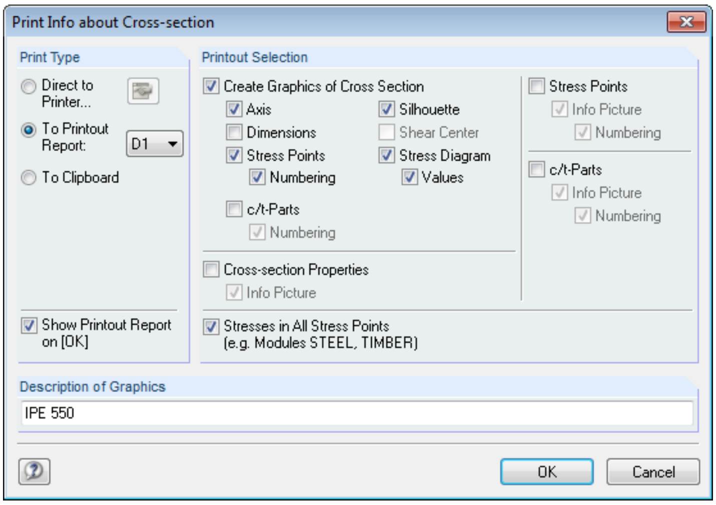


Figure 6.2: Dialog box *Print Info about Cross-section*

In the dialog section *Print Type*, the common options from RSTAB are available for selection:

- *Direct to Printer* sends the current graphic to the printer.
- *To Printout Report* inserts the graphic into the printout report.
- *To Clipboard* provides the graphic for other applications.

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

In the dialog section *Printout Selection*, you decide which elements appear in the print graphic and in the output table. The objects specified under *Create Graphics of Cross Section* do not require any further explanation. If you tick the check box for *Cross-section Properties*, the properties will be printed as a table, optionally added by a symbolic *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]*.

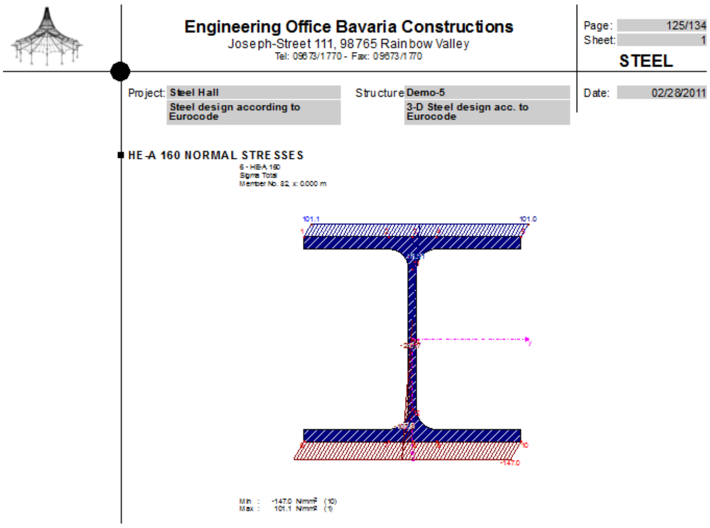


Figure 6.3: Stress graphic in the printout report

6.2.2 Results in the RSTAB Model

Every picture that is displayed in the graphic window of the main program RSTAB can be included in the printout report. Thus, the stresses and ratios displayed in the RSTAB model can be prepared for the printout, too.

Designs in the RSTAB model

To print the STEEL graphic currently displayed in the RSTAB work window,

select **Print** on the **File** menu

or use the toolbar button shown on the left.

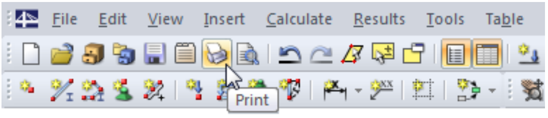


Figure 6.4: Button *Print* in the toolbar of the main window

Result diagrams

In the same way, you can integrate the result diagrams of members into the report by using the [Print] button. It is also possible, to print them directly.

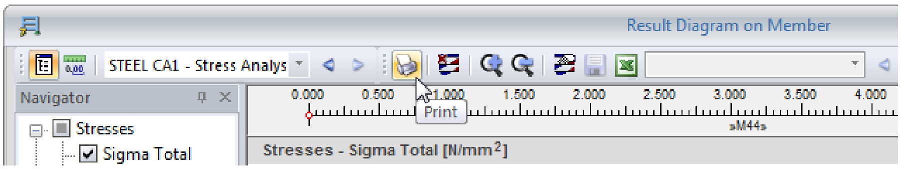


Figure 6.5: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog box opens:

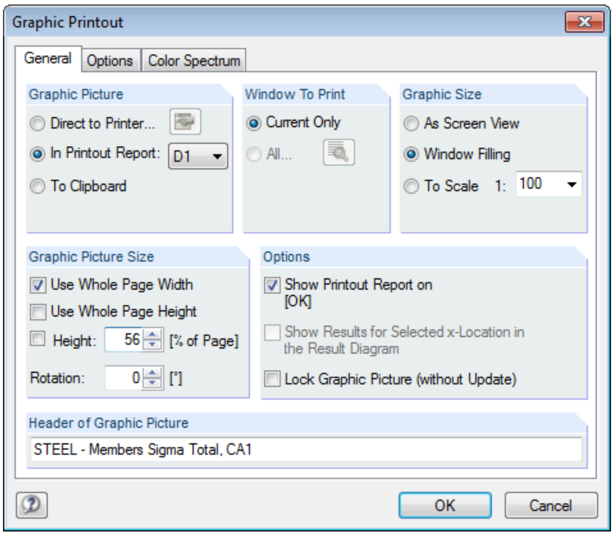
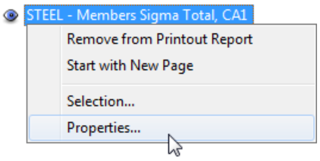


Figure 6.6: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in the RSTAB manual, chapter 10.2, page 243. 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. In addition, it is possible to adjust imported graphics subsequently: Right-click the relevant entry in the navigator of the printout report and select *Properties* in the context menu. The dialog box *Graphic Printout* appears again, offering various options for adjustment.

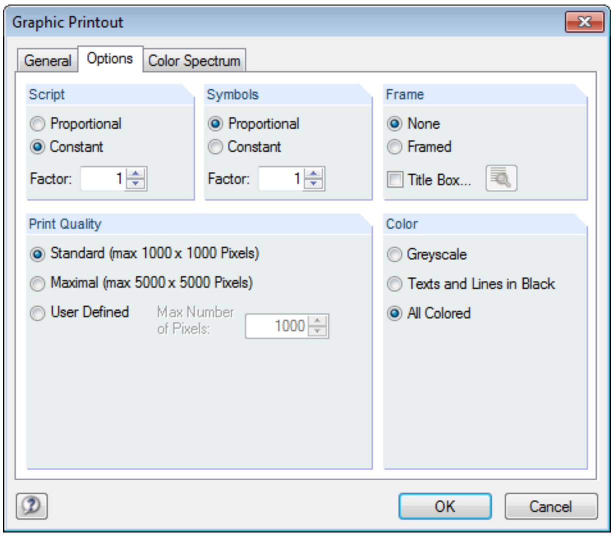


Figure 6.7: Dialog box *Graphic Printout*, tab *Options*

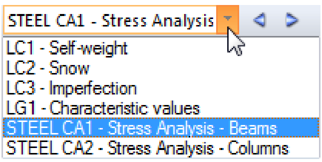
7. General Functions

The final chapter describes some menu functions as well as export options for the design results.

7.1 STEEL Design Cases

You can group members in separate design cases. In this way you can, for example, combine groups of structural components or define particular design specifications for them (limit stresses, partial safety factors, optimization etc.).

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



The STEEL design cases are available in the RSTAB work window and can be displayed like a load case or load group by means of the toolbar list.

Create a new STEEL case

To create a new design case,

select **New Case** on the **File** menu in the STEEL add-on module.

The following dialog box appears.

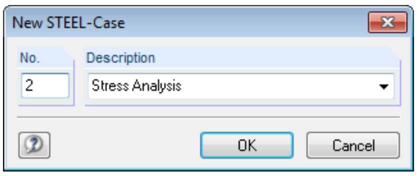


Figure 7.1: Dialog box *New STEEL-Case*

In this dialog box, enter a *No.* (which is not yet assigned) and a *Description* for the new design case. When you click [OK], table 1.1 *General Data* opens where you can enter the new design data.

Rename a STEEL case

To change the description of a design case subsequently,

select **Rename Case** on the **File** menu in the STEEL add-on module.

The dialog box *Rename STEEL-Case* appears.

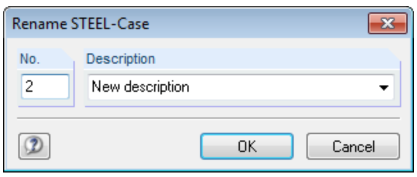


Figure 7.2: Dialog box *Rename STEEL-Case*

Copy a STEEL case

To copy the input data of the current design case,

select **Copy Case** on the **File** menu in the STEEL add-on module.

The dialog box *Copy STEEL-Case* appears where you can specify the number and description of the new case.

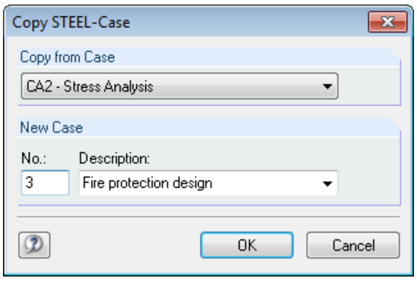


Figure 7.3: Dialog box *Copy STEEL-Case*

Delete a STEEL case

To delete design cases,

select **Delete Case** on the **File** menu in the STEEL add-on module.

In the dialog box *Delete Cases*, you can select the relevant design case in the *Available Cases* list to delete it by clicking [OK].

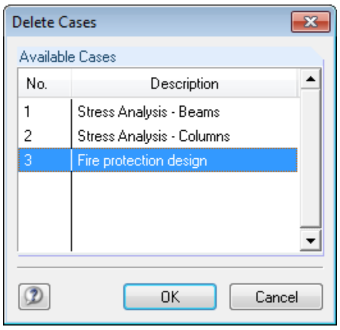
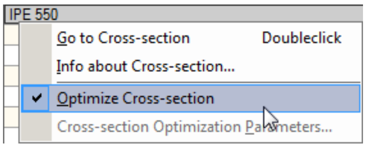


Figure 7.4: Dialog box *Delete Cases*

7.2 Cross-section Optimization

As mentioned in chapter 2.3, STEEL offers you the possibility to optimize cross-sections. Select the relevant cross-section by ticking its check box in column C or D in table 1.3 *Cross-sections* (see Figure 2.5, page 15).



You can also start the cross-section optimization out of the results tables by using the context menu.

During the optimization process, STEEL determines the cross-section within the same cross-section table that fulfills the analysis requirements in the most optimal way, that means comes as close as possible to the maximum possible ratio specified in the *Details* dialog box (see Figure 3.1, page 18). The required cross-section properties will be determined with the RSTAB internal forces. If a cross-section proves to be more favorable, two cross-sections will be displayed on the right of table 1.3 as shown in Figure 7.6, the original cross-section from RSTAB and the optimized one.

For parameterized cross-sections of the cross-section library, a dialog box with detailed specifications appears when you tick the check box for optimization.

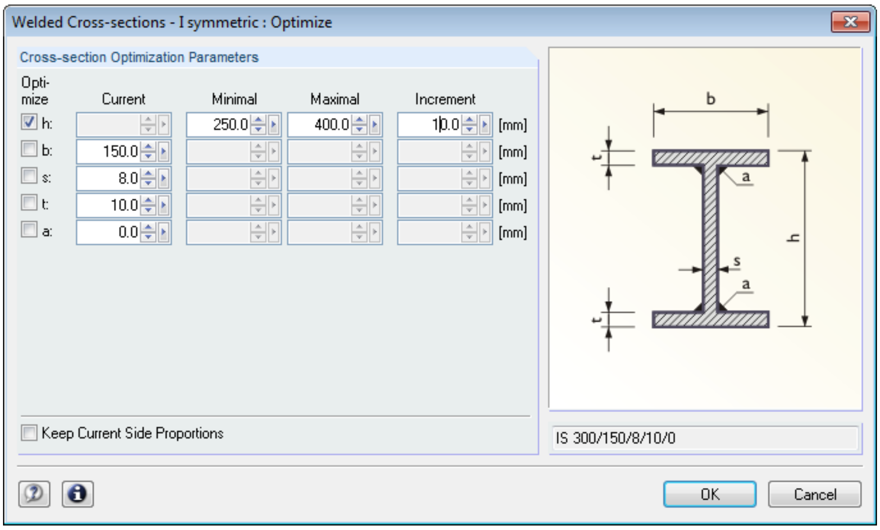


Figure 7.5: Dialog box *Welded Cross-sections - I symmetric : Optimize*

By ticking the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. The ticked check box enables the *Minimal* and *Maximal* columns where you specify the upper and lower limits of the parameter. The *Increment* column determines the interval in which the size of this parameter varies during the optimization process.

If you want to *Keep Current Side Proportions*, tick the corresponding check box. Additionally, you have to select at least two parameters for optimization to make the specification effective.

For cross-sections based on combined rolled cross-sections, no optimization options are available.

Please note for the optimization process that the internal forces won't be recalculated automatically with the changed cross-sections. It is up to you to decide which cross-sections should be transferred to RSTAB for a recalculation. As a result of optimized cross-sections, internal forces may vary considerably because of the changed stiffnesses in the structural system. It is recommended to recalculate the internal forces after the first optimization and then to optimize the cross-sections again.

You do not need to transfer the modified cross-sections to RSTAB manually: Set table 1.3 *Cross-sections*, and then

select **Export All Cross-sections to RSTAB** on the **Edit** menu.

Also the context menu in table 1.3 provides options to export optimized cross-sections to RSTAB.

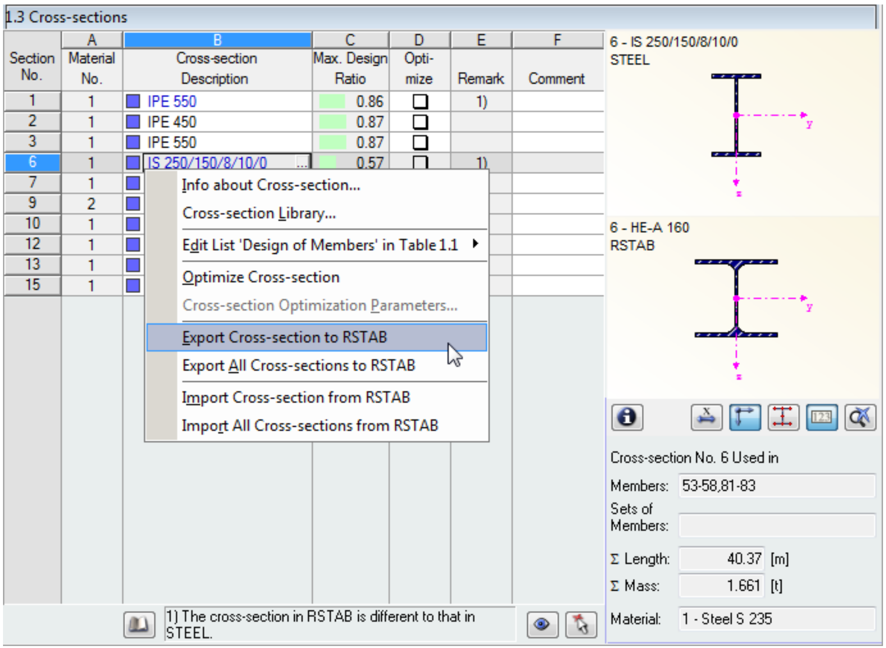


Figure 7.6: Context menu in table 1.3 *Cross-sections*

Before the changed cross-sections are transferred to RSTAB, a security query appears, because the transfer requires the deletion of results. When you confirm the query and start the calculation subsequently in the STEEL add-on module, the RSTAB internal forces as well as the stresses will be determined in one calculation run.

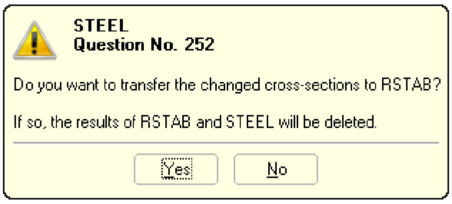


Figure 7.7: Query before transfer of modified cross-sections to RSTAB

By using the menu functions described above, you can also import the original RSTAB cross-sections to STEEL. Please note that this option is only available in table 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 stress analysis may be inaccurate if the depths of the start and end cross-section differ considerably. In this case, it is recommended to divide the tapers into several single members whose start and end cross-sections have minor cross-section differences.

7.3 Material Export to RSTAB

When you have changed materials in the STEEL table 1.2, you can export the modified materials to RSTAB, similar to the export of modified cross-sections. It is also possible to import the originally used materials from RSTAB. Materials modified in the add-on module are highlighted in blue.

You do not need to transfer the modified materials manually to RSTAB.
Set table 1.2 *Materials* and then

select **Export All Materials to RSTAB** on the **Edit** menu.

Also the context menu of table 1.2 provides options to transfer modified materials to RSTAB.

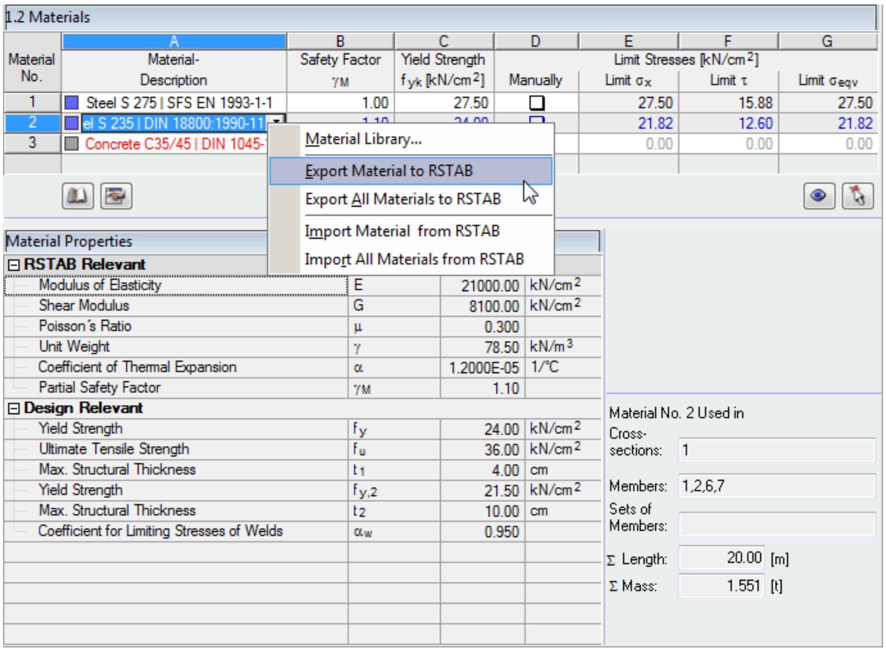


Figure 7.8: Context menu of table 1.2 *Materials*

Before the changed materials will be transferred to RSTAB, a security query appears, because the transfer requires the deletion of results. 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.

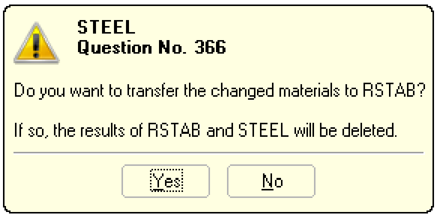


Figure 7.9: Query before transfer of modified materials to RSTAB

7.4 Units and Decimal Places

The units and decimal places for RSTAB and all add-on modules are managed in one global dialog box. In the add-on module STEEL, you can use the menu to define the units. To open the corresponding dialog box,

select **Units and Decimal Places** on the **Settings** menu.

The following dialog box opens, which you already know from RSTAB. The add-on module STEEL is preset.

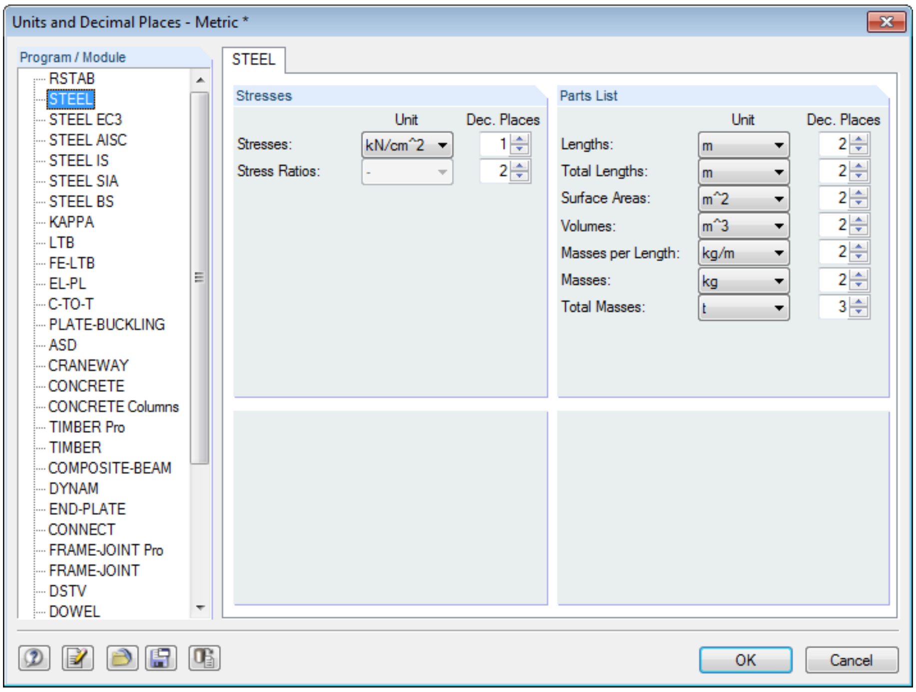


Figure 7.10: Dialog box *Units and Decimal Places*

The settings can be saved as user profile to reuse them in other structures. The corresponding functions are described in the RSTAB manual, chapter 11.6.2, page 336.

7.5 Export of Results

The results of the stress analysis can be provided for other programs in various ways.

Clipboard

To copy cells selected in the STEEL results tables to the clipboard, use the keyboard 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 47) to export them subsequently. Then, in the printout report,

select **Export to RTF File or BauText** on the **File** menu.

The function is described in detail in the RSTAB manual, chapter 10.1.11, page 239.

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,

select **Export Tables** on the **File** menu in the STEEL add-on module.

The following export dialog box appears.

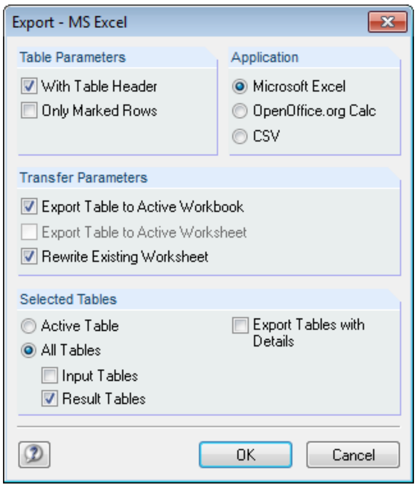


Figure 7.11: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, start the export by clicking [OK]. Excel or OpenOffice will be started automatically. It is not necessary to run the programs in the background.

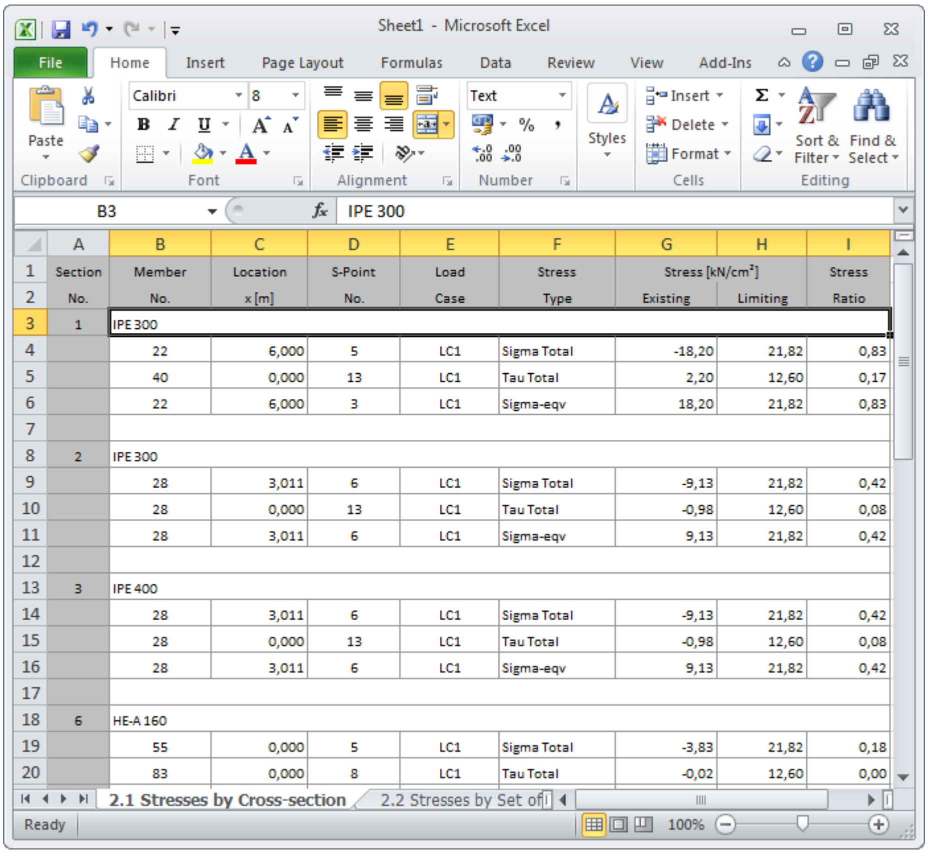


Figure 7.12: Result in MS Excel: Table 2.1 *Stresses by Cross-section*

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. Auflage1993

[4] SCHNEIDER Bautabellen, Werner Verlag, 17. Auflage 2006

[5] STAHLBAU Handbuch, Band 1, Stahlbau-Verlagsgesellschaft mbH, Köln 1993

B Index