Ingenieur-Software Dlubal

Version
October 2007

Additional Module

STEEL

General Stress Analysis
Cross-Section Optimization

Program Description

© Dlubal Engineering Software
Am Zellweg 2 D-93464 Tiefenbach

Tel: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 1770
E-mail: info@dlubal.com
Web: www.dlubal.com

Table of Contents

1. Introduction

1.1 Additional Module STEEL

STEEL is not an independently running program. It is closely integrated in the user interface of the main program RSTAB. The structure specific input data as well as the internal forces are thus available for the module. On the other hand, the STEEL results can also be evaluated in the RSTAB work window and you can include them in the global printout report.

STEEL delivers general stress analyses by calculating the existing stresses and then by comparing these with the limit stresses. For this, an extensive cross-section library and an extendable material library with the standard specific limit stresses are available. Every cross-section has design-related stress points. These are also available for the graphical evaluation.

During the stress analysis, also the maximum stresses for sets of members are determined and the governing internal forces will be documented. In addition, STEEL also offers an automatic cross-section optimization including an option to export the optimized cross-sections to RSTAB.

Separate STEEL design cases allow for flexible examination of stresses. The design is completed by a parts list with quantity survey.

Essential innovations in STEEL are:

- Display of the maximum stress ratios in the cross-section mask to facilitate the cross-section optimization
- STEEL masks linked to the RSTAB work window so that the current objects are highlighted in the background graphics
- View mode to change the graphical display in the RSTAB work window
- Colored relation bars in the results masks
- Quick info of fulfilled or failed stress design
- Display of the STEEL stresses and stress ratios as result diagrams
- Filter option for the stress display in the RSTAB graphics
- Display of the STEEL stresses and stress ratios on the rendered model
- Direct data export to MS Excel

We wish you much success using STEEL.

DLUBAL ENGINEERING SOFTWARE

1.2 STEEL Team

The following people where involved in the development of the STEEL program:

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 Images and Icons

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

Program Testing:

Ing. Robert Michalovič
Petr Pražák
Michala Sobotková
Ing. Martin Vasek

Manual, Support and Translations

Dipl.-Ing. (FH) Robert Vogl
Ing. Dmitry Bystrov
Jan Jeřábek
Ing. Ladislav Kábrt
Ing. Petr Míchal
Ing. Robert Michalovič
Mgr. Florian Nadge
Mgr. Petra Pokorná

Technical Support and Quality Control

Dipl.-Ing. (FH) Michael Bausch
Dipl.-Ing. Rafael Ceglarek
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. David Röseler
Dipl.-Ing. (FH) Walter Rustler
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Paul Stolbunski
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

Details to installation, user interface, results evaluation and printout can be found in the RSTAB manual. This manual focuses on the particularities that occur when working with the additional module STEEL.

The STEEL manual follows the sequence and the structure of the input and results masks. The text presents the described **buttons** in square brackets, e.g. [OK]. They are also shown on the left. The used **expressions** for dialogs, tables and menus are set in *italics* to clarify the explanation.

This manual features also an index to find certain expressions quickly. If you should not find what you are looking for, please check our website at **www.dlubal.com** where you can go through our well-organized FAQ pages.

1.4 Starting STEEL

You have the following options to start the add-on module STEEL.

Menu

STEEL can be called via menu

Additional Modules → Design - Steel → STEEL.

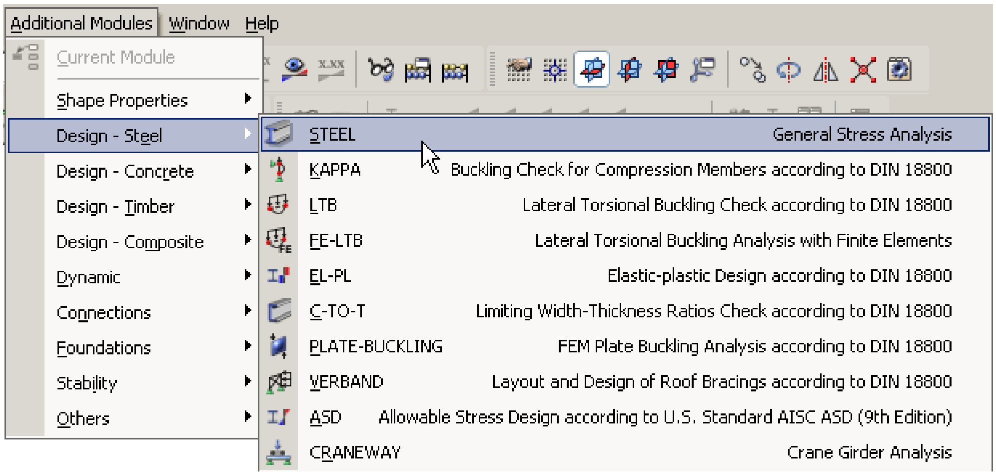


Figure 1.1: Menu: *Additional Modules → Design – Steel → STEEL*

Navigator

In the *Data* navigator, you can open STEEL via the item

Additional Modules → STEEL.

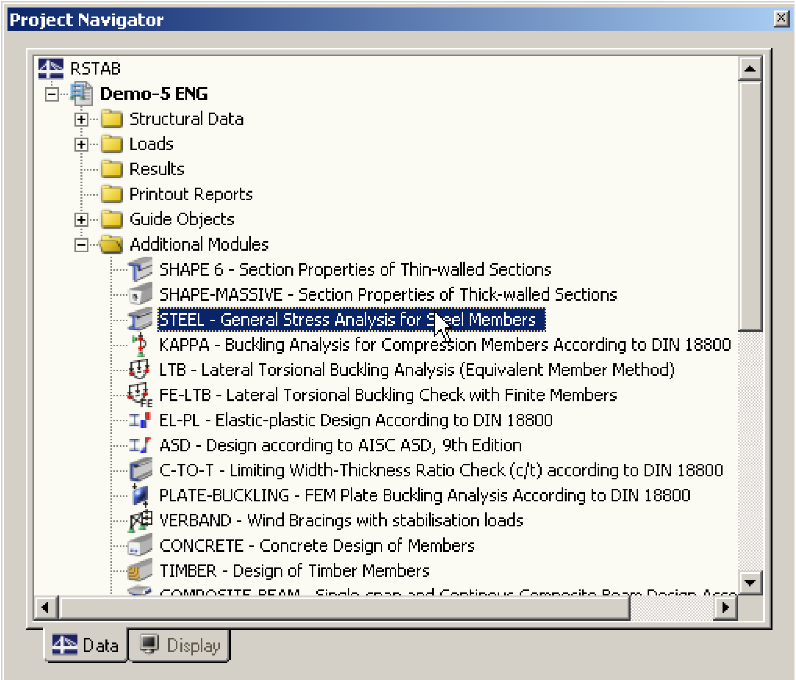
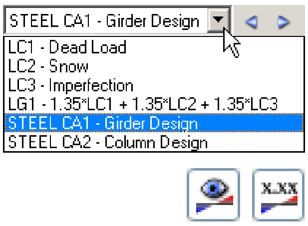


Figure 1.2: Data-Navigator: *Additional Modules → STEEL*

Panel



If the RSTAB structure already contains STEEL results, you can select the STEEL case in the load case list. The stresses and stress ratios can be displayed in the graphics via the button [Results on/off].

Now, the button [STEEL] becomes available in the panel to call up the module STEEL.

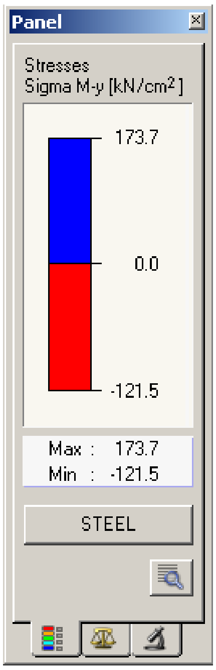


Figure 1.3: Panel: Button [STEEL]

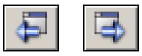
2. Input Data

The input for the definition of the design cases is carried out in masks. The module offers also the [Pick] function to select members and sets of members in the graphics.

After starting STEEL, a navigator appears in a new window that lists all available masks. A pull-down list at the top offers all available *STEEL Cases* (see chapter 7.1, page 47).

When STEEL is opened for the first time in a RSTAB structure, then the module imports the following design-related 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 the background - if calculated)



The masks can be scrolled through either by clicking on a selected item in the STEEL navigator or by clicking on the buttons shown on the left. The buttons [F2] and [F3] can also be used to scroll one mask forward or backward.

[OK] stores the input and closes the STEEL module. [Cancel] closes the module without storing any data.

2.1 General Data

In mask 1.1 *General Data*, you have to select the members and actions for the design. The design standard will be defined in mask 1.2 because it is linked to the material properties.

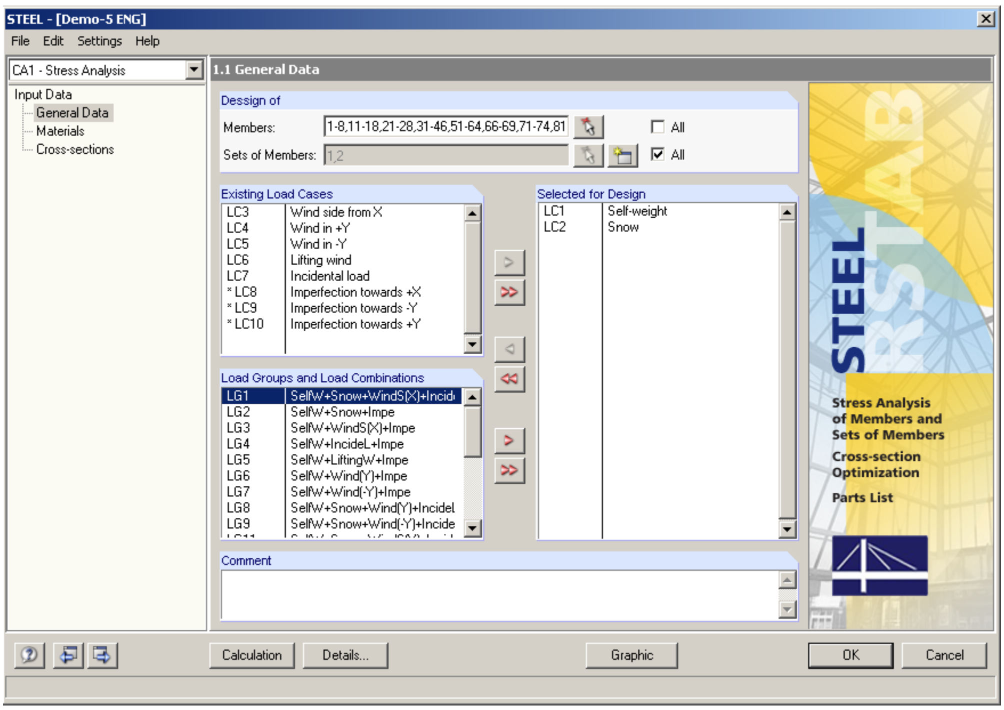


Figure 2.1: Mask 1.1 *General Data*

Design

The design can be carried out for *Members* as well as for *Sets of Members*. If only selected objects are to be designed, the checkbox *All* must have been deactivated. This makes the input boxes accessible and the numbers of the selected members resp. sets of members can be entered. With [Pick], you can also select the relevant objects graphically in the RSRTAB work window. The list of the preset member numbers can be selected quickly by double clicking it. Then, it is possible to overwrite the list.

If no sets of members have been defined in RSTAB, you can create them also in the STEEL module via the button [New]. The dialog for creating a new set of members appears which is already familiar from RSTAB. There, the specifications can be made.

The advantage of the sets of members design is that selected members can be designed to determine the total maxima of their stresses and stress ratios. In this case, the results masks 2.2 *Stresses by Set of Members* and 3.3 *Parts List by Set of Members* will be displayed.

Existing Load Cases / Load Groups and Combinations

These two sections contain all design-relevant load cases, load groups, load combinations and super combinations that have been defined in RSTAB. Button [▶] moves the selected load cases resp. combinations to the list *Selected for Design*. This action can also be carried out by double clicking the relevant items. Button [▶▶] transfers the entire list to the right.

Load cases that are marked with an asterisk (*), e.g. load cases 8 to 10 in Figure 2.1, are excluded from the design. This is the case when no load has been defined or when those are imperfection-type load cases as in the example above.

Selected for Design

The selected actions for the design are listed in the right column. Button [◀] removes only the selected item from the list. As before, you can also select by double clicking on the item. Button [◀◀] removes all items from the list.

The design of an enveloping *Or* load combination is faster than analyzing all contained load cases or load groups. On the other hand, the governing actions for the design of a complete combination will not be too transparent. The max/min stresses of each load case or load group are used for the design that are then superimposed according to the relevant code. For load combinations, it is recommended to check the *Method of Stress Calculation* in dialog *Details* (cf. chapter 3.2, page 20).

Details...

Comment

This input box can be used for user defined comments, e.g. to describe the design case.

2.2 Materials

This mask has two parts. The upper section lists materials including the limit stresses that have been selected for the design. The section *Material Properties* lists the constants of the current material, i.e. the material that has been selected in the upper section.

Details to the material properties for the determination of the internal forces can be found in chapter 5.2 of the RSTAB manual. The design-relevant material properties are stored in the global material library and are set automatically. These can be adjusted in this mask.

The units and decimal places of the material constants and stresses can be edited via menu **Settings** → **Units and Decimal Places**.

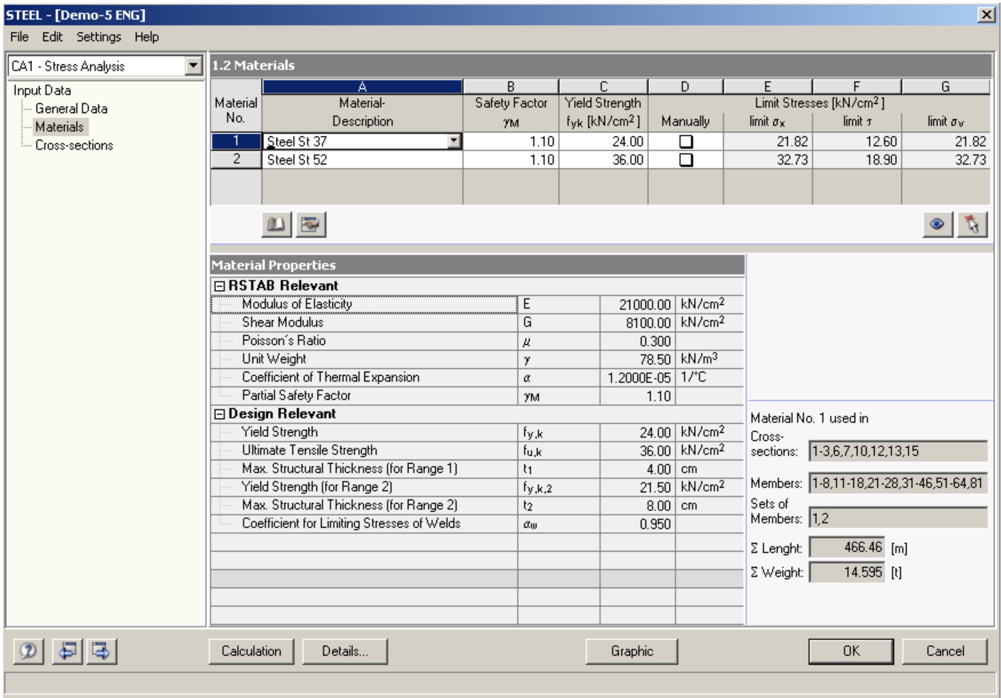
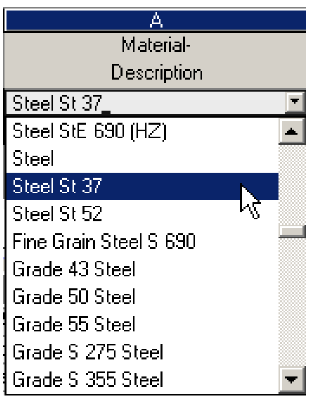


Figure 2.2: Mask 1.2 *Materials*

Material Description

The materials that have been defined in RSTAB are set by default. If the *Material Description* corresponds to an item of the material library, STEEL automatically imports the design-relevant material constants.



The material selection can be carried out via the list: Place the cursor in column A and then click the button [▼] or press the function key [F7]. The list shown on the left appears. After the import, the design relevant parameters are entered in the remaining fields of the row.

The list contains only STEEL materials. In general, any material can be designed with a stress concept based on the comparison of the existing axial, shear and equivalente stresses with the limit stresses. In this way, also the design of aluminum or high grade steel cross-sections can be carried out. Of course, the appropriate standards have to be considered as well.

If there is a material name with non-defined limit stresses (e. g. timber), then the entries of this row are displayed in red. User-defined limit stresses can also be entered by activating the function **Manually** in column D. The red entries disappears as soon as the permitted stresses have been defined in columns E to G.

The import of materials from the library is described further below.

Safety Factor γM

This factor describes the safety factor for calculating the design values of the material stiffness. Therefore, the index *M* is used. The characteristic value of the yield strength f_{yk} for the determination of the limit axial and bending stress $\sigma_{R,d}$ (see Equation 2.1) and the limit shear stress $\tau_{R,d}$ (Equation 2.1) is reduced by this factor γ_M.

When therefore calculating according to 2nd or 3rd order analysis, the factor γ_M enters the design twice: First, a stiffness reduced by 10 % has to be considered for the influence of the deformations when determining the internal forces according to DIN 18800 T2, El. (116), and then the design values of the stiffnesses for the limit state are to be reduced by the partial safety factor γ_M.

Yield Strength f_{yk}

The yield strength describes the limit to which the material can be strained without permanent deformation. The characteristic values of different steel grades can be taken from e.g. DIN 18800 T1, section 4 und EC 3, section 3.

Limit Stresses

For materials that are contained in the general material library, the limit stresses are set automatically and can not be modified here.

If you want to modify the limit stresses, then you have to change the material properties via the button [Edit Material] (see page 13) or the checkbox *Manual*.

Manually

If this checkbox has been activated then you can define the following columns manually.

Modified materials are marked in the column *Material Description* by an asteriskr.

Limit σ_x

The limit normal stress represents the allowable stress for the loading due to bending and axial force. It is defined according to DIN 18800 T1, El. (746) by the characteristic value of the yield strength that is 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 specifies the allowable shear stress due to shear force and torsion. According to DIN 18800 T1, El. (746), the partial safety factor γ_M enters the equation for the determination of the limit shear stress, too.

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

Equation 2.2

Limit σ_v

The limit equivalent stress represents the allowable comparative stress for the simultaneous action of several stresses. It is also defined e.g. by Equation 2.1 according to DIN 18800 T1, El. (746).

Yield Strengths Subject to Component Thickness

For some materials, the characteristic yield strength $f_{y,k}$ is related to the thickness of the member components *t*. The *Maximum Structural Thickness* of the selected ranges including the corresponding yield strength is displayed in the lower section *Material Properties*.

The assignment of the yield strengths is managed in the codes, e. g. table 1 of DIN 18800 T1. Via the button [Edit Material], you can control and, if needed, edit the component thicknesses and the assigned stresses (cf. page 13).

Material Library

Many materials are already stored in a library. It can be called up via menu

Edit → Material Library

or the button shown on the left.

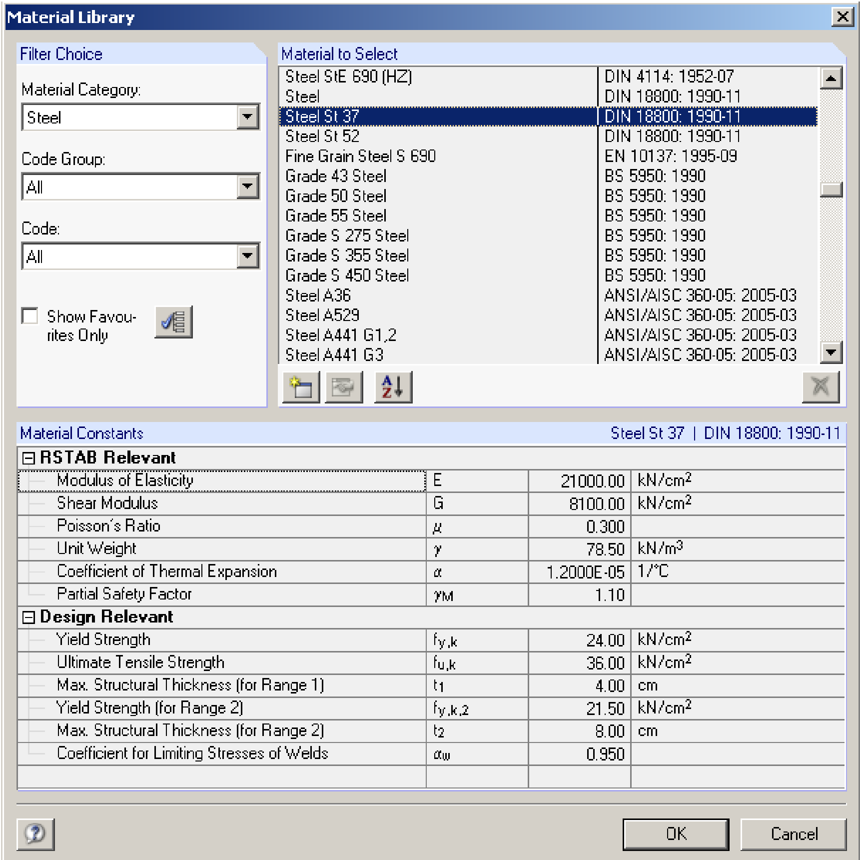


Figure 2.3: Dialog *Material Library*

In section *Filter Choice*, the material category *Steel* is preset. From the list *Material to Select*, you can select a material and in the lower part of the dialog you can check its parameters. With [OK] or [↵] you can set it in the STEEL mask 1.2.

Details on how to filter, add or reorganize materials can be found in chapter 5.2 of the RSTAB manual.

When selecting another material category than *Steel*, you have to consider that only materials can be designed with a stress concept that is based on the comparison of the existing axial, shear and comparative stresses with the allowable stresses. In this way, also the design of aluminum or high-rade steel cross-sections can be carried out in principle.

If a material with non-defined limit stresses (e.g. timber) is imported, then the entries of this row in mask 1.2 is displayed in red. User-defined limit stresses can also be defined by activating the function **Manually** in column D. The red entries disappear as soon as the limit stresses have been entered in columns E to G . The stress design for e.g. timber cross-sections can only be carried out incompletely because the relevant codes require differentiated criteria. For this, the add-on module TIMBER is recommended.

Edit Material

The yield strengths and limit stresses of the current material can be adjusted via the button shown on the left. The following dialog appears.

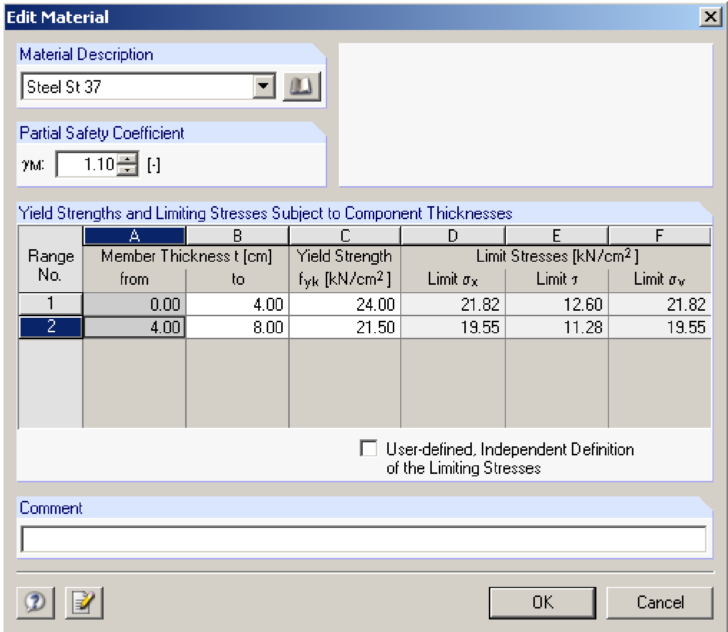


Figure 2.4: Dialog *Edit Material*

The defined characteristic values of the yield strength f_{yk} are reduced by the factor γ_M in section *Partial Safety Factor*. The resulting limit stresses according to Equation 2.1 and Equation 2.2 (page 11) are listed in columns D to E.

In section *Yield Strengths and Limit Stresses Subject to Component Thicknesses*, the ranges of the *Member Thickness t* can be modified. The number of the ranges is set by the codes. The range limits can be modified by defining different values in column B manually. Column A will be automatically adjusted. A specific *Yield strength* f_{yk} can assigned to every range.

If the limit stresses are to be defined freely, then the checkbox *User-defined, Independent Definition of the Limit Stresses* is to be activated. Columns D to E then become accessible for user-defined entries.

Modified materials are marked by an asterisk in mask 1.2.

2.3 Cross-Sections

This mask controls the cross-sections for the design and the optimization parameters.

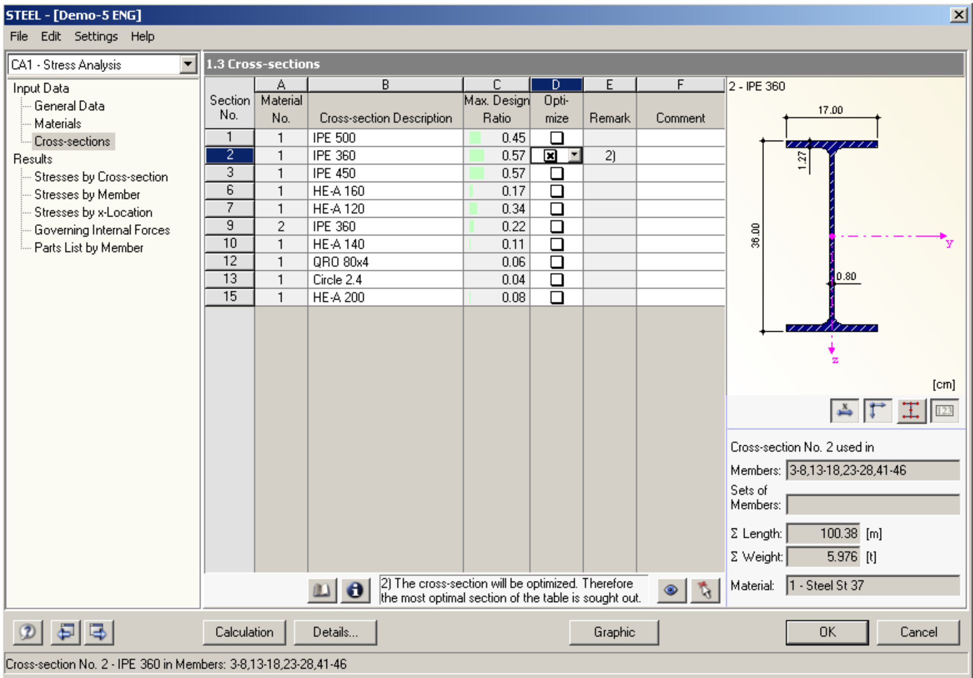


Figure 2.5: Mask 1.3 *Cross-Sections*

Cross-Section Description

When calling up this mask, the cross-sections used in RSTAB are preset including the corres-ponding material numbers.

These cross-sections can be edited for the design. The description of a modified cross-section will be marked in blue in column B.

In order to edit a cross-section, the new description can be entered in the corresponding row, or the new cross-section is selected from the library. It can be called up as usual via the [Cross-Section Library] button. Alternatively, place the cursor in the selected row and then press [...] or the function key [F7]. The library which is already familiar from RSTAB appears.

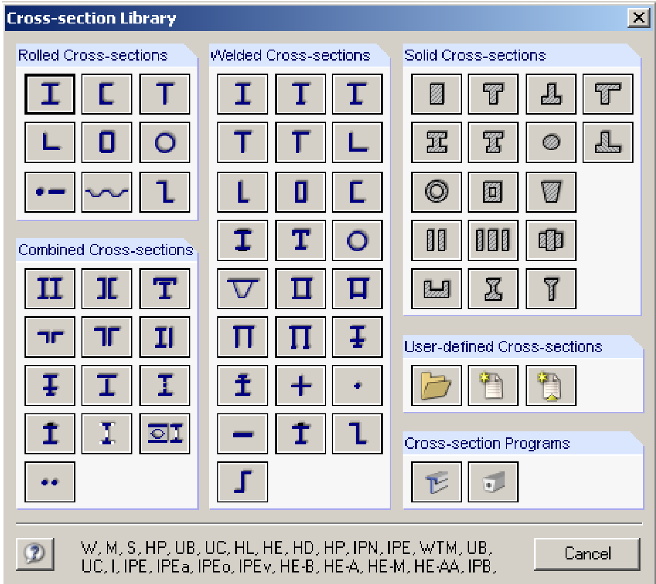
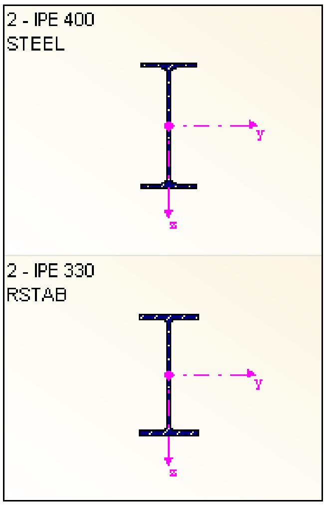


Figure 2.6: Cross-Section Library



Details to the selection of cross-sections from the library can be found in chapter 5.3 of the RSTAB manual.

If there are different cross-sections in STEEL and RSTAB, then the figure on the right of the mask shows both cross-sections. The stress design is then carried out with the internal forces from RSTAB for the cross-section that was defined in STEEL.

Member with Tapered Cross-Section

For tapered members with different cross-sections at member start and member end, the two cross-section numbers are displayed in two lines following the definition in RSTAB. STEEL carries out the design of tapered members as long as the following condition is fulfilled: An equal number of stress points is required.

The axial stresses are for example determined from the moments of inertia as well as from the centroid distances of the stress points. STEEL can not interpolate the intermediate values if the number of stress points for the start and end cross-section of a tapered member is different. In that case, RSTAB assumes for the calculation and for the 3D rendering the two cross-section properties for each member half. In STEEL, however, no design is possible. A warning will appear before the calculation.

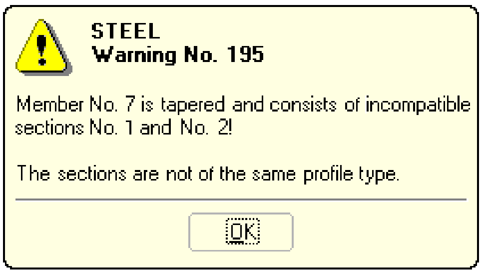
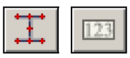
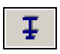


Figure 2.7: Figure 2.8: Warning for incompatible cross-sections





You can check the stress points of a cross-section and their numbering in the cross-section graphics on the right. Details to the stress points can be found in chapter 4.1, page 24.

Thus, for a successful design an equal number of stress points has to be generated. This can be achieved, for example, by modeling the taper as a copy of the start cross-section and then modifying the geometrical parameters. If required, the two cross-sections should be defined as parametrized, i.e. welded, cross-sections. Especially for tapers, the **ICU** tpye *Combined Sections I-Section Plus Cut Section Below* is recommended.

Maximum Design Ratios

This column assists with the optimization process. It is displayed when a design was already carried out. The ratio of design and the color relation scales show which cross-sections have a low design ratio and therefore will be oversized resp. are overstrained and therefore will be too weak.

Optimizing

Every cross-section can be subjected to an optimization process. Here, the internal forces of RSTAB are used to determine the cross-section in the same cross-section group that is closest to the maximum stress ratio of 1.0.

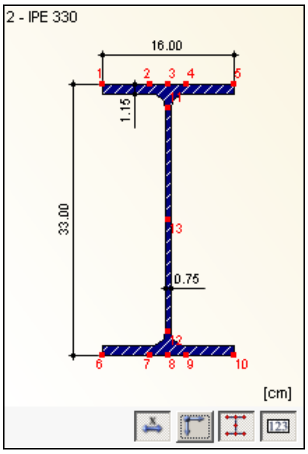
If a selected cross-section is to be optimized, the checkbox in column D has to be activated. Recommendations for the cross-section optimizations can be found in chapter 7.2, page 49.

Comment

In this column, comments are shown as footnotes and are explained in the lower part of the cross-section list.

If comment *1) The cross-section will not be designed because the cross-section data are not defined!* appears, then you deal with an unknown cross-section that is not registered in the cross-section library. This might for example be a user-defined cross-section or a section that was not calculated in SHAPE-THIN. In this case, you should return to RSTAB and complete the required entries for e.g. the **stress points** (see chapter 5.3 of the RSTAB manual, section *Create User-Defined Cross-Section*).

Cross-Section Graphics



The current cross-section is displayed in the right part of mask 1.3. The buttons shown below have the following functions:

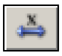
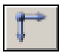
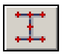
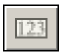
Button	Function
	The dimensioning of the cross-section is switched on and off.
	The principal axes of the cross-section are switched on and off.
	The stress points are displayed or switched off.
	The numbering of the stress points is switched on and off.

Table 2.1: Buttons in the graphics

3. Calculation

The stress analysis is carried out with the internal forces that have been determined in RSTAB. Before starting the [Calculation], the design details should be checked. This dialog is called up via [Details]. Details can be found in chapter 3.2, page 20.

3.1 Stresses and Stress Ratio

The axial stresses σ_{total}, τ_{total} and σ_{eqv} are shown in masks 2.1 to 2.5. The individual stress components can be switched on via the buttons [Select Stresses to Show] and [Show or Print Cross-section Values and Extended Stress Diagram].

Axial Stresses

According to the typical conventions, tensile stresses are issued with positive signs and compressive stresses with negative signs.

The analysis is carried out for every individual stress point so that usually not all components of the maximum stresses must be added for the combined evaluation (z. B. σ_{total}): In most cases, these are available at different stress points. The relevant stress components of the selected stress point have to be superimposed.

Their meaning of the axial stresses σ is as follows:

σ_N	Stress due to axial force N $\sigma = \frac{N}{A}$ with A: Cross-section surface of the cross-section
$\sigma_{M\text{-}y}$	Stress due to bending moment M_y $\sigma = \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z$ with $\alpha_{pl,y}$: plastic form coefficient acc. to DIN 18800 T1, El. (750) I_y: moment of inertia with reference to principal axis y e_z: Centroid distance of the stress point in direction z
$\sigma_{M\text{-}z}$	Stress due to bending moment M_z $\sigma = -\frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$ with $\alpha_{pl,z}$: plastic form coefficient acc. to DIN 18800 T1, El. (750) I_z: moment of inertia with reference to principal axis z e_y: Centroid distance of the 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 My 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 the axial forces of the individual load cases, e.g. for the fatigue design
σ_{total}	Total axial stress due to axial force N and bending moments My 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: Axial stresses σ

The local member axis system influences also the signs of the internal forces.

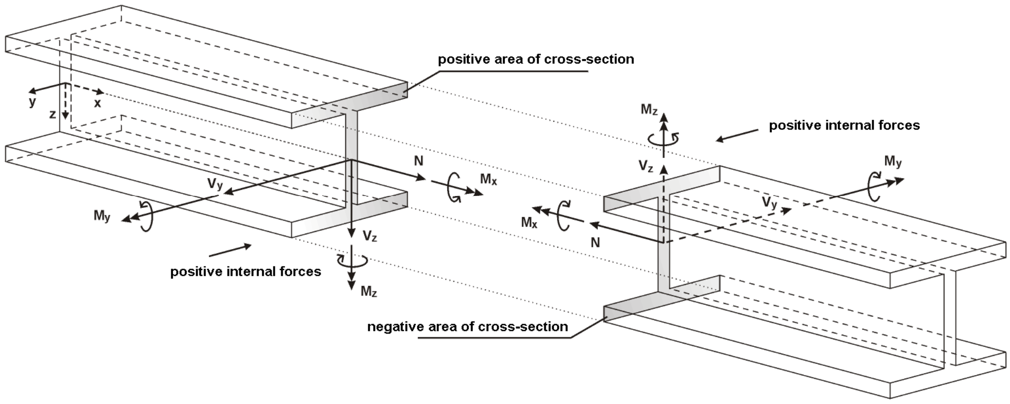


Figure 3.1: Positive definition of the internal forces

The bending moment $\mathbf{M_y}$ is positive when tensile stresses occur on the positive member side (in direction of axis z). $\mathbf{M_z}$ is positive when compressive stresses occur on the positive member side (in direction of axis y). The sign definition for torsional moments, normal forces and shear forces conforms to the usual conventions: The internal forces are positive if they act in a positive direction.

Shear Stresses

Their meaning of the shear stresses τ is as follows:

$\tau_{V\text{-}y}$	Stress due to axial force V_y $\tau = -\frac{V_y \cdot Q_z}{I_z \cdot t}$ with Q_z: statical moment of area with reference to principal axis z; I_z: moment of inertia with reference to principal axis z; t: Governing thickness of the cross-section
$\tau_{V\text{-}z}$	Stress due to shear force V_z $\tau = -\frac{V_z \cdot Q_y}{I_y \cdot t}$ with Q_y: statical moment of area with reference to principal axis y; I_y: moment of inertia with reference to principal axis y; t: Governing thickness of the cross-section
τ_V	Stress due to shear forces V_y and V_z $\tau = -\frac{V_y \cdot Q_z}{I_z \cdot t} - \frac{V_z \cdot Q_y}{I_y \cdot t}$

$\tau_{\text{M-T, St.Venant}}$	Stress due to torsional moment M_T for open cross-section $\tau = \frac{M_T}{I_{T,St.V.}} \cdot t$ with $I_{T,St.V.}$: Torsional moment according to Saint Venant t: Governing thickness of the cross-section
$\tau_{\text{M-T, Bredt}}$	Stress due to torsional moment M_T for hollow cross-section $\tau = \frac{M_T}{2 \cdot A_m \cdot t}$ with A_m: surface that is enclosed by the cross-section center lines t: Governing thickness of the cross-section
$\tau_{\text{M-T}}$	Stress due to torsional moment M_T $\tau = \frac{M_T}{I_{T,St.V.}} \cdot t$ or $\tau = \frac{M_T}{2 \cdot A_m \cdot t}$
τ_{total}	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

For the determination of the shear stresses due to shear forces, the statical moments are used, not the shear surfaces of the cross-section.

For the shear stresses due to torsion, the following must be observed:

- If the current cross-section is only partially open and contains a hollow cell, then the complete cross-section will be classified as *hollow*. In this case, the shear stress is determined exclusively according to Bredt's formula. Unlike the cross-section program SHAPE, no analysis will be carried out by its components of $M_{T,St.Venant}$ and $M_{T,Bredt}$.
- The influence of the torsional buckling can not be determined in STEEL. The design is – like the internal force determination in RSTAB – limited only to the primary torsional moments. If the warping stresses due to secondary torsional moments or bimoments can not be neglected, then the additional module FE-LTB is recommended.

Equivalent Stress

The equivalent stress σ_{eqv} is determined e.g. according to DIN 18800 T1, El. (748) as follows:

σ_{eqv}	Equivalent stress resulting from the axial stresses σ and shear stresses τ $\sigma_{eqv} = \sqrt{f_1 \cdot \sigma_{total}^2 + f_2 \cdot \tau_{total}^2}$ with f_1: Factor for axial stresses f_2: Factor for shear stresses

Table 3.3: Equivalent Stress σ_v

Details...

The factors f_1 and f_2 can be defined in dialog *Details*. The factors $f_1 = 1.0$ and $f_2 = 3.0$ according to DIN 18800 T1, El. (748) are set by default.

Stress Ratio

The quotient of the existing stress and the limit stress is to be determined for the stress design as shown e.g. in DIN 18800 T1, El. (747).

The stress ratio of the cross-section at the selected stress point can be read for every type of internal force (cf. chapter 5.1, page 34). The stress ratios due to axial, shear and equivalent stresses are set by default for the tabular output data. If the limit stress is not exceeded, the stress ratio is smaller than or equal to 1.0 and the stress design is considered to be fulfilled.

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

Equation 3.1: Design requirement for axial stresses

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

Equation 3.2: Design requirement for shear stresses

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

Equation 3.3: Design requirement for equivalent stresses

3.2 Details

The dialog to check the calculation parameters can be called up via the [Details] button in every STEEL mask.

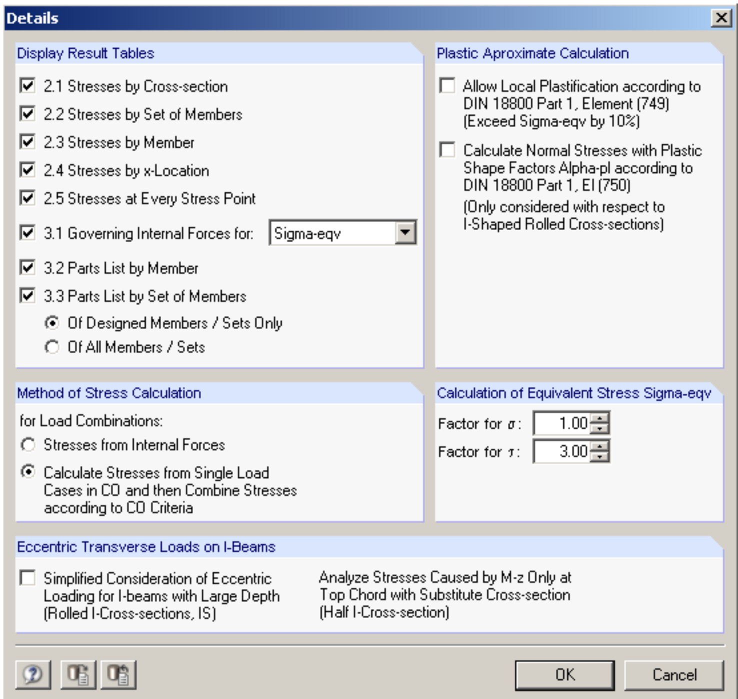


Figure 3.2: Dialog *Details*

Display Result Tables

This section controls the display of the result masks.

The mask 2.5 *Stresses at Every Stress Point* is by default deactivated because the stress graphics offer also access to the stress points. When checking the stresses in the tables, it might be advisable to display this mask.

The displayed values in mask 3.1 *Governing Internal Forces* usually refer to the equivalent stress σ_{eqv}. It is possible to select a different stress type from the list.

Plastic Approximate Calculation

Allow Local Plastification

Also the *Local Plastification* according to DIN 18800 T1, El. (749) can be considered for the design. Here, the equivalent stress σ_v is permitted to exceed the permitted limit stress in small areas by 10 %. STEEL determines whether the two conditions described in El. (749) for accepting 'small areas' are fulfilled.

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

Equation 3.4

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

Equation 3.5

If this is the case, the limit stress for the design of σ_v will be increased.

Plastic Shape Factor α_{pl}

The axial stresses can optionally be reduced by the *Plastic Shape Factor* α_{pl} as described in DIN 18800 T1, El. (750). This option refers to the 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 3.6

If this option is to be applied, STEEL assumes the shape factors $\alpha_{pl,y}$ = 1.14 and $\alpha_{pl,z}$ = 1.25. The allowance for local plastification extends exclusively to rolled I-shaped cross-sections.

Method of Stress Calculation

When acting on two axes, the member internal forces of load combinations may under certain circumstances not lead to the maximum stresses. For example, in one load case with vertical loads only moments M_y (and no moments M_z) occur and in a different load case with horizontal loads only moments M_z (and no moments M_y) occur. The two load cases are superimposed with as 'variable' load cases within a load combination. In the RSTAB table 3.1 *Internal Forces*, the moment M_z is classified as not corresponding to the maximum moment M_y because the horizontal loads do not increase the moment due to vertical loads. If the design of the *Stresses from Internal Forces* was carried out, the simultaneous influence of the two different types of internal forces would not be taken into account for the combined consideration of the bending stresses.

Stresses from Internal Forces

According to this method, the results of the RSTAB table 3.1 *Internal Forces* are used for the design. The max/min results are processed row by row for every extreme value including the corresponding internal forces.

The advantage of this design option is that the results of the load combinations can be used directly. This speeds up the processing time. Additionally, the designed internal forces are transparent because in the STEEL mask 3.1 *Governing Internal Forces* the relevant rows of the RSTAB results table 3.1 *Internal Forces* reappear.

Calculate Stresses from Load Cases and then Combine Stresses

This method of calculation is set by default for load combinations. Initially, the axial and shear stresses of the contained load cases are determined. They are then superimposed according to the CO combination criterion. This guarantees that the above-described effects in the case of exclusively uniaxial bending loads do not result in too small stress ratios.

The design is carried out by stress points. The compressive, tensile and shear stresses are added accordingly and then presented in the results masks. The equivalent stress σ_{eqv} makes an exception, however: It is determined with the components from σ_{total} und τ_{total}. Superimposing the equivalent stresses from the single load cases would not be correct as this would lead to oversized stress ratios.

This method of calculation is slightly more time-consuming. Additionally, the values that are represented in mask 3.1 *Governing Internal Forces* are more difficult to reconstruct if they are related to the equivalent stresses.

In more complex 3D models of RSTAB, hardly any exclusively uniaxial bending moments occur. Thus, both methods of stress calculation should lead to the same stress ratios.

Calculation of Equivalent Stress Sigma-eqv

In this dialog section, the factors for the determination of the equivalent stress can be adjusted.

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

with f_1: Factor for axial stresses

f_2: Factor for shear stresses

Equation 3.7

The factors $f_1 = 1.0$ and $f_2 = 3.0$ as specified in DIN 18800 T1, El. (748) are set by default.

Eccentric Loading for I-Beams with Large Depths

If transverse loads are introduced to the top flange of a girder, their influence on the bending stress of the bottom flange will decrease with growing cross-section depth. For this reason, eccentric transverse loads can be considered for high I-shape cross-sections with a simplified approach: The stress due to the bending moment M_z is calculated only on the top flange for every (!) rolled or welded symmetric I-shape cross-section that is to be designed in the current STEEL case. For this, an equivalent cross-section with half of the moment of inertia I_z is applied.

With this option, the loads can be entered in the RSTAB model with respect to the centroid axis to avoid torsion. Because this checkbox affects all rolled and symmetric I-shape cross-sections of the design case, it is recommended to design all I-beams with large depths in a separate STEEL case.

3.3 Starting the Calculation

You can [Calculate] the current STEEL case via the corresponding button in all three input masks of the STEEL module.

STEEL searches for the results of the load cases, load groups, combinations and super combinations that are to be analyzed. If these are not found, then STEEL first starts the RSTAB calculation of the internal forces that are necessary for the design. For this, the current RSTAB calculation parameters are used.

If an optimization of the cross-sections (c.f. chapter 7.2, page 49) is to be carried out, then first the required cross-sections will be defined and after that the stresses will be calculated.

You can also initiate the calculation of the STEEL results from the RSTAB interface. The additional modules are listed in the dialog *To Calculate* like load cases or load groups. This dialog can be called up in RSTAB via

Calculation → To Calculate.

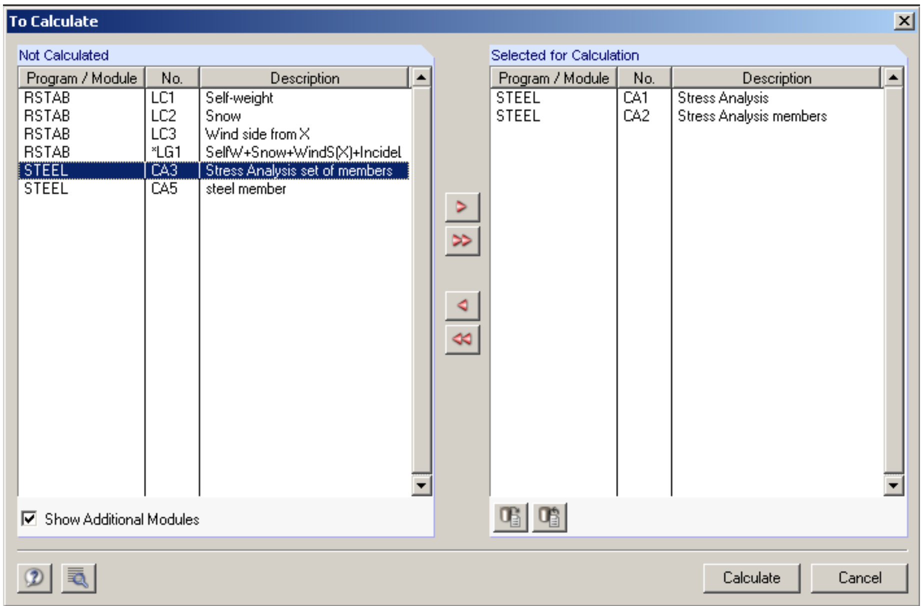


Figure 3.3: Dialog *To Calculate*

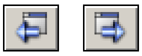
If the STEEL design cases are missing in the list *Not Calculated*, then the checkbox *Show Additional Modules* has to be activated.

The selected STEEL cases can be moved to the right via the [▶] button. The calculation can the be started by pressing the corresponding button.

A selected STEEL case can also be started via the tool bar. Set the selected design case and click on the button [Results on/off].

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

4. Results



The mask 2.1 *Stresses by Cross-Section* appears right after the calculation. The stresses and stress ratios are listed in detail in the result masks 2.1 to 2.5. The following masks 3.1 to 3.3 contain the governing internal forces. The result masks are accessible via the STEEL navigator. Alternatively, the two buttons shown on the left or the keys [F2] and [F3] can be used to scroll from one mask to the next.

With [OK], you can store all the results and close the STEEL module.

This chapter presents a description of the individual masks. Details to evaluating and checking the results can be found in the next chapter 5.

4.1 Stresses by Cross-Section

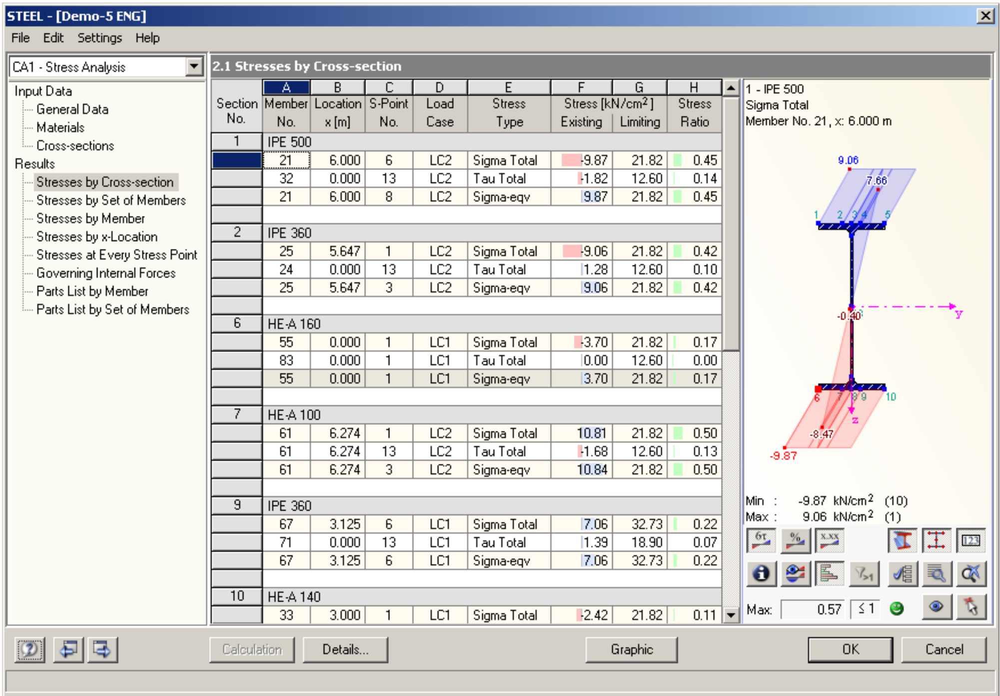


Figure 4.1: Mask 2.1 *Stresses by Cross-Section*

This mask issues the maximum stresses for all members selected for the design resulting from the relevant load cases, load groups or combinations. The listing is sorted by cross-sections. For a tapered member, both cross-section descriptions are shown in the row next to the cross-section number.

The stress components displayed in column E *Stress Type* depend on the settings in dialog *Stresses - Filter* (see Figure 5.3, page 34). This dialog is accessible via the button shown on the left.

Member No.

For every cross-section and stress type, the member number is issued that presents the highest stress ratio.

Location x

The location x in the member is presented for which the maximum value of the stress was determined. For the tabular output, these RSTAB member locations *x* are used:

- Start and end nodes
- Division points according to possible user-defined member division
- Extreme values of internal forces

S-Point No.

The design is carried out at so-called *Stress Points* of the cross-sections. These are locations of the cross-sections that are defined by centroid distances, statical moments and cross-section thicknesses. The design according to the equations of Table 3.1 and Table 3.2 can be carried out with the help of the cross-section properties.

All standard cross-sections of the library as well as the Shape-Thin and Shape-Massive cross-sections already contain stress points at the design-relevant locations of the cross-sections. The parameters for user-defined cross-sections have to be defined manually or imported.

You can view the stress points and their numbering in the graphics on the right. The current stress point (i.e. the stress point in the row of the cursor position) is highlighted in red.

The [Info] button can be used to check the relevant parameters of the stress points. First, the dialog *Info about Cross-Section* with the listing of all cross-section parameters is called up. In this dialog below the picture of the section, the button [Details] allows to access the details for the stress points.

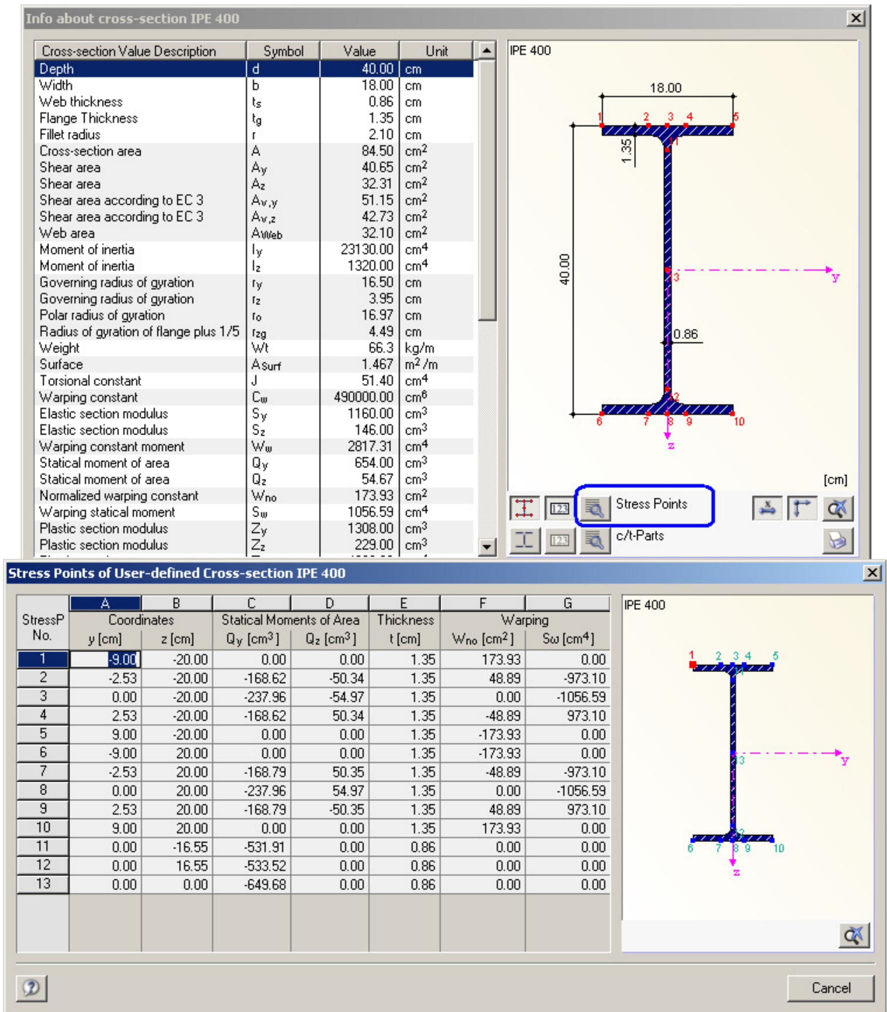


Figure 4.2: Info about cross-section: *Stress Points*

The columns *Coordinates y* and *z* list the centroid distances e_y resp. e_z, the columns *Statical Moments of Area* the values of Q_y and Q_z with respect to the governing axis y resp. z. The *Thickness t* represents the component thickness at the selected stress point. The values in the columns *Warping* are not relevant for the STEEL design.

The stress analysis is carried out for every individual stress point. Therefore, usually not all components of the maximum stresses may be added for the combined display (e. g. σ_{eqv}) as those are available at different stress points in most cases. Instead, the stress components that exist at the same stress points must be combined. The evaluation of the results by stress points can be carried out e.g. in mask 2.5 (see chapter 4.5, page 29) or in window *Cross-Section Values and Stress Diagram* (see Figure 5.5, page 36).

Load Case

Column C presents the load case, load group, load combination or super combination whose internal forces produce the maximum stresses.

Stress Type

The axial stresses σ_{total}, the shear stresses τ_{total} and the equivalent stresses σ_{eqv} are listed by default. The determination of these stresses is presented in Table 3.1, Table 3.2 and Table 3.3 on pages 18 to 19.

The stress components that are contained in the total stresses can be viewed as shown in Figure 4.3. The individual stress components can be selected in dialog *Stresses - Filter* (c.f. Figure 5.3, page 34). This dialog is accessible via the button shown on the left.

Stress Existing

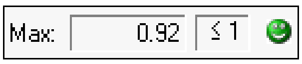
This column presents the extreme values of the existing stresses that are determined according to the equations of Table 3.1, Table 3.2 and Table 3.3 (see pages 18 to 19).

Stress Limiting

Here you can find the limit stresses of mask 1.2 (see chapter 2.2, page 10). In detail, these are the following stresses:

- The limit axial stress σ_x represents the allowable stress for the loading due to bending and axial force.
- The limit shear stress τ is the allowable shear stress due to shear force and torsion.
- The limit equivalent stress σ_{eqv} represents the allowable equivalent stress for the simultaneous action of axial and shear stresses.

Stress Ratio



For every stress component, the quotient of the *Stress Existing* and the *Stress Limiting* is determined. If the limit stress is not exceeded, then the stress ratio is smaller than or equal to 1.00 and the stress design is considered to be fulfilled.

4.2 Stresses by Set of Members

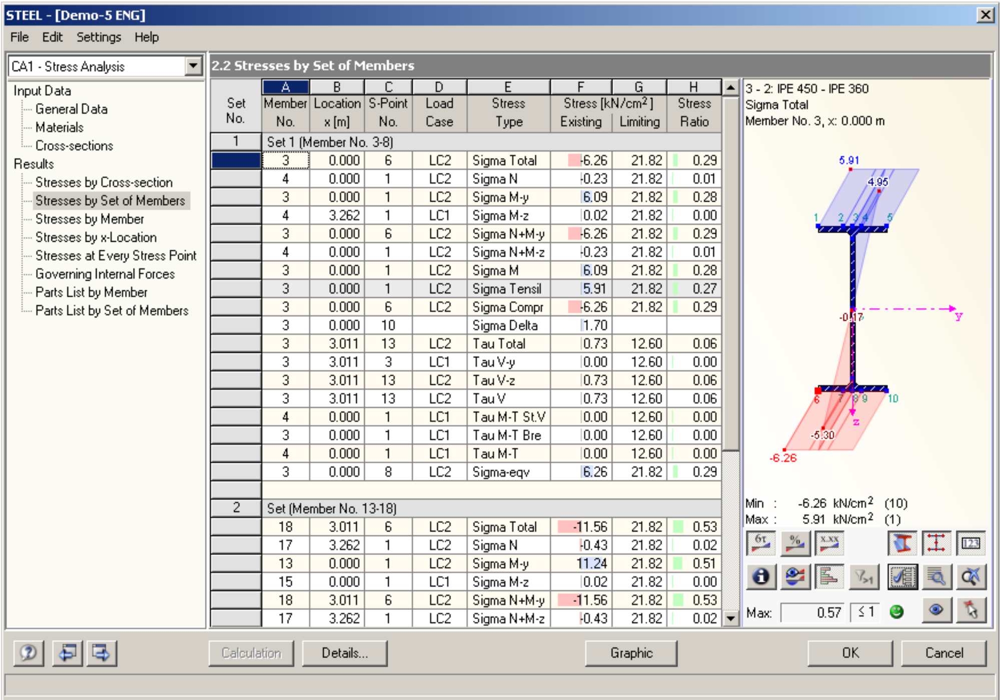


Figure 4.3: Mask 2.2 *Stresses by Set of Members*

This result mask is displayed when one or more sets of members have been selected for design. The listing of the maximum stresses is sorted by sets of members.

The numerical output by sets of members has the advantage that the stress design of a complete structural group (e.g. a frame) is presented in a single results mask.

The individual columns are presented in chapter 4.1, page 24. Additionally, in column A *Member No.* the number of the member is presented that has the highest stress ratio within the set of members.

4.3 Stresses by Member

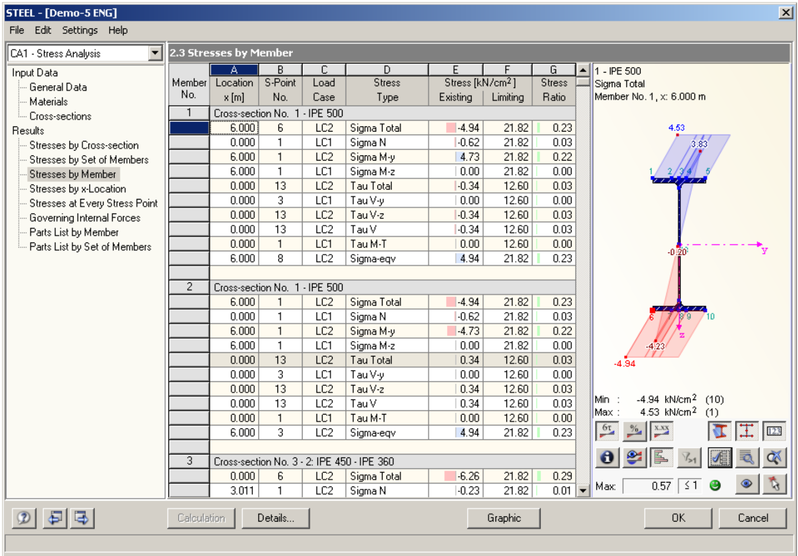


Figure 4.4: Mask 2.3 *Stresses by Member*

In this results mask, the maximum stresses are listed by member numbers. For every member, the *Location x* is specified where the maximum occurs.

The individual columns are presented in the chapter chapter 4.1, page 24.

4.4 Stresses by x-Location

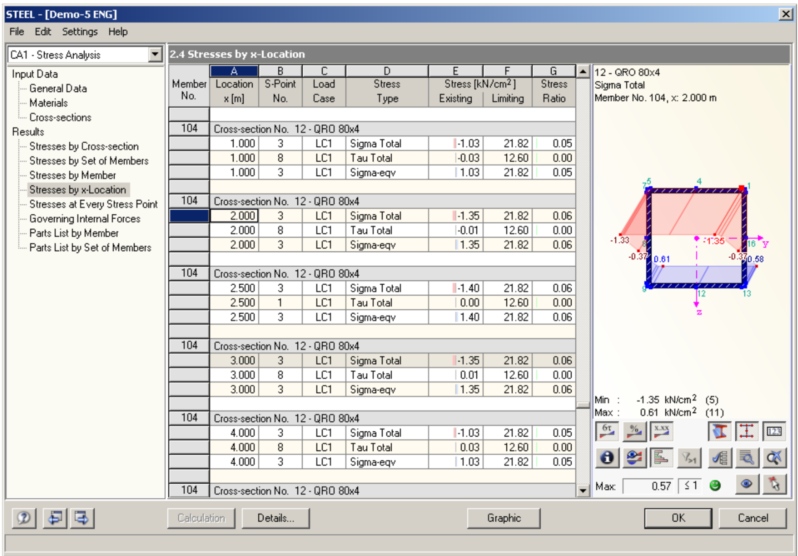


Figure 4.5: Mask 2.4 *Stresses by x-Location*

This result mask lists the maximum stresses for every member at the locations ***x*** resulting from the RSTAB member divisions:

- Start and end nodes
- Division points according possible user-defined member division
- Number of divisions for the member results as defined in register *Options* of the RSTAB dialog *Calculation Parameters*
- Extreme values of the internal forces

4.5 Stresses at Every Stress Point

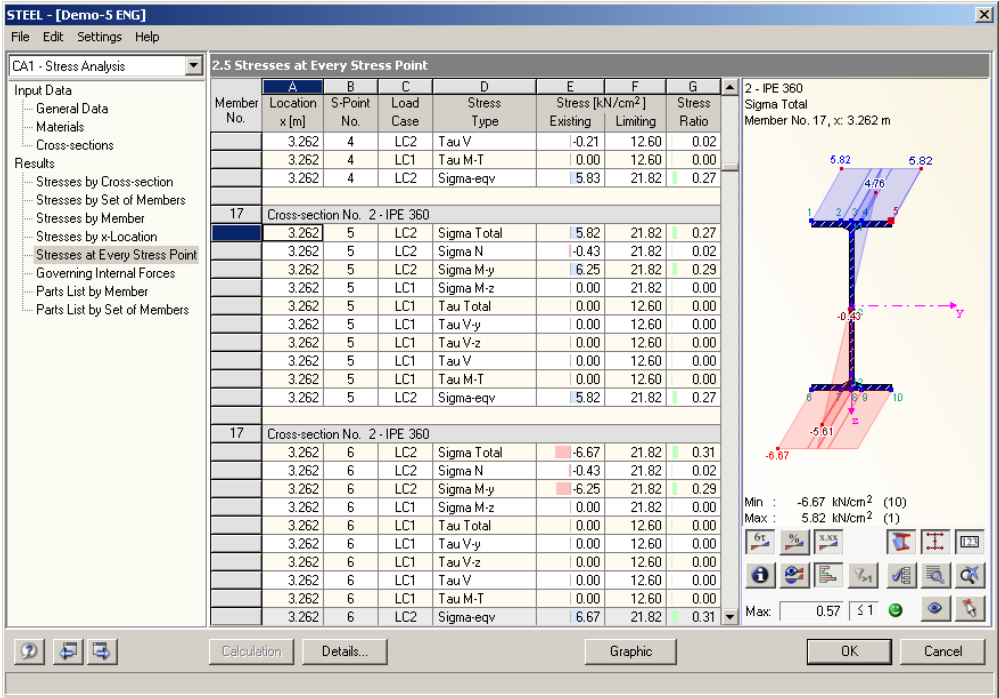


Figure 4.6: Mask 2.5 *Stresses at Every Stress Point*

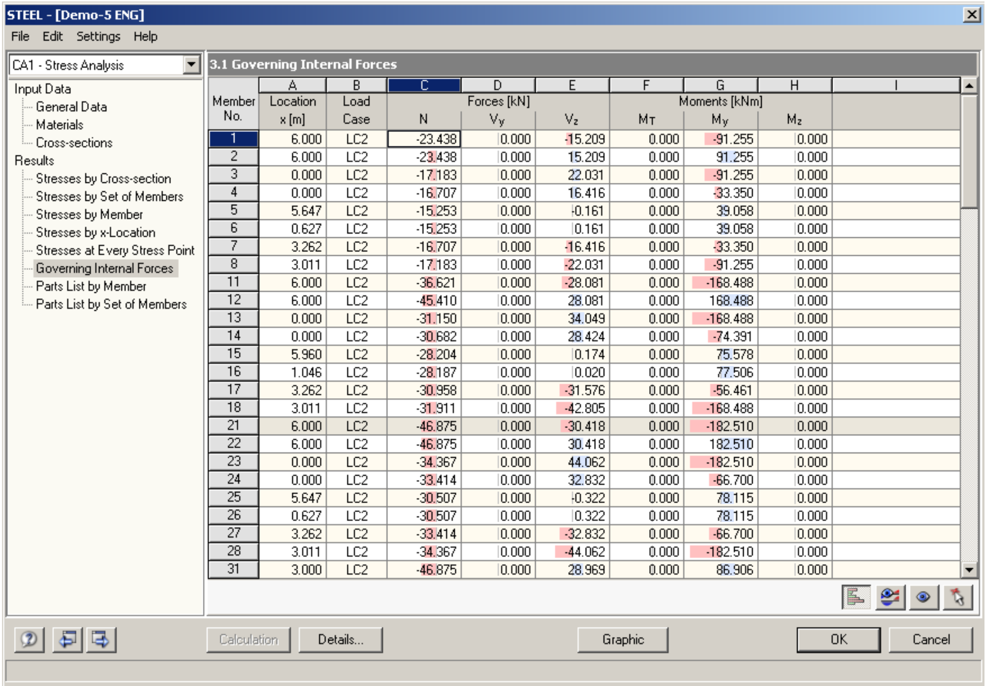
Details...

Because the evaluation by stress points is usually not required, this results mask is disabled by default. However, you can activate it the *Details* dialog (see Figure 3.2, page 20) that can be called up via the [Details] button from every mask.

The presentation of this mask results in a large amount of data. Because the maximum stresses and, along with them, the governing stress points are determined automatically, it is usually not necessary to display the mask 2.5. Additionally, specific means of evaluation are available in the previous masks via the [Cross-Section and Stress Diagram] button. In the *Cross-Section Values and Stress Diagram* dialog (cf. Figure 5.5, page 36), the design results can be evaluated graphically and numerically at every stress point.

The listing of the stresses is sorted for every member by *Location x* and by *Stress Point*. Details to the individual columns of this mask can be found in chapter 4.1, page 24.

4.6 Governing Internal Forces



Member No.	A Location x [m]	B Load Case	C Forces [kN] N	D Vy	E Vz	F Moments [kNm] MT	G My	H Mz
1	6.000	LC2	-23.438	0.000	-15.209	0.000	-91.255	0.000
2	6.000	LC2	-23.438	0.000	15.209	0.000	91.255	0.000
3	0.000	LC2	-17.183	0.000	22.031	0.000	-91.255	0.000
4	0.000	LC2	-16.707	0.000	16.416	0.000	-33.350	0.000
5	5.647	LC2	-15.253	0.000	-0.161	0.000	39.058	0.000
6	0.627	LC2	-15.253	0.000	0.161	0.000	39.058	0.000
7	3.262	LC2	-16.707	0.000	-16.416	0.000	-33.350	0.000
8	3.011	LC2	-17.183	0.000	-22.031	0.000	-91.255	0.000
11	6.000	LC2	-36.621	0.000	-28.081	0.000	-168.488	0.000
12	6.000	LC2	-45.410	0.000	28.081	0.000	168.488	0.000
13	0.000	LC2	-31.150	0.000	34.049	0.000	-168.488	0.000
14	0.000	LC2	-30.682	0.000	28.424	0.000	-74.391	0.000
15	5.960	LC2	-28.204	0.000	0.174	0.000	75.578	0.000
16	1.046	LC2	-28.187	0.000	0.020	0.000	77.506	0.000
17	3.262	LC2	-30.958	0.000	-31.576	0.000	-56.461	0.000
18	3.011	LC2	-31.911	0.000	-42.805	0.000	-168.488	0.000
21	6.000	LC2	-46.875	0.000	-30.418	0.000	-182.510	0.000
22	6.000	LC2	-46.875	0.000	30.418	0.000	182.510	0.000
23	0.000	LC2	-34.367	0.000	44.062	0.000	-182.510	0.000
24	0.000	LC2	-33.414	0.000	32.832	0.000	-66.700	0.000
25	5.647	LC2	-30.507	0.000	-0.322	0.000	78.115	0.000
26	0.627	LC2	-30.507	0.000	0.322	0.000	78.115	0.000
27	3.262	LC2	-33.414	0.000	-32.832	0.000	-66.700	0.000
28	3.011	LC2	-34.367	0.000	-44.062	0.000	-182.510	0.000
31	3.000	LC2	-46.875	0.000	28.969	0.000	86.906	0.000

Figure 4.7: Mask 3.1*Governing Internal Forces*

In this mask, the governing internal forces are displayed that entail the maximum stress ratios of every member. The reference to the equivalent stress σ_{eqv} is set by default. It is possible to define the reference to a different stress type in the *Details* dialog (see Figure 3.2, page 20). This dialog can be called up via button [Details].

The design option *Calculate Stresses from Single Load Cases in CO and then Combine Stresses according to CO Criteria* makes it impossible to directly use the table rows of the RSTAB results table 3.1 *Internal Forces*. The compressive, tensile and shear stresses in the single load cases are added accordingly and then presented in the STEEL results masks. The equivalent stresses σ_{eqv}, however, are calculated with the components of σ_{total} and τ_{total} so that the values of this mask 3.1 may not be transparent immediately.

Location x

For every member, the location x is specified where the maximum stress ratio occurs.

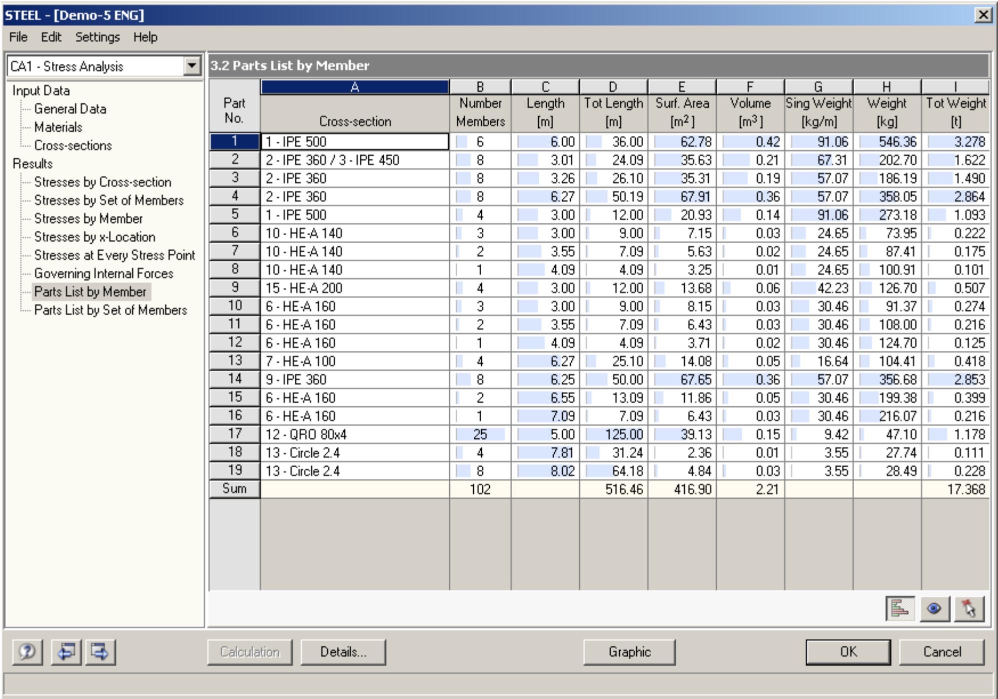
Load Case

This column displays the numbers of the load case, group, combination or super combination whose internal forces cause the maximum stresses on the member.

Forces / Moments

The governing axial and shear forces as well as the torsional and bending moments are issued for every member.

4.7 Parts List by Members



Part No.	A Cross-section	B Number Members	C Length [m]	D Tot Length [m]	E Surf. Area [m²]	F Volume [m³]	G Sing Weight [kg/m]	H Weight [kg]	I 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 360 / 3 - IPE 450	8	3.01	24.09	35.63	0.21	67.31	202.70	1.622
3	2 - IPE 360	8	3.26	26.10	35.31	0.19	57.07	186.19	1.490
4	2 - IPE 360	8	6.27	50.19	67.91	0.36	57.07	358.05	2.864
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 100	4	6.27	25.10	14.08	0.05	16.64	104.41	0.418
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 2.4	4	7.81	31.24	2.36	0.01	3.55	27.74	0.111
19	13 - Circle 2.4	8	8.02	64.18	4.84	0.03	3.55	28.49	0.228
Sum		102		516.46	416.90	2.21			17.368

Figure 4.8: Mask 3.2 *Parts List by Member*

Finally, a summary of the cross-sections from the design case is displayed. By default, only the designed members are contained in this list . The *Details* dialog can be used to select a parts list of all members of the structure (c.f. Figure 3.2, page 20). This dialog can be called via the button shown on the left.

Part No.

Part numbers are issued automatically for members of the same type.

Cross-Section

This column lists the cross-section descriptions.

Number Members

This column shows how many members of the same type exist.

Length

In this column, the lengths of a single member are listed.

Total Length

This column contains the product of the two previous columns.

Surface Area

This column shows the surface areas of every part. They refer to the total lengths and are determined from the value A_{Surf} of every cross-section. This value can be checked in masks 1.3 to 2.4 via the [Info about Cross-Section] button.

Volume

The volume of every part is determined from the cross-section area and the total length.

Single Weight

The *Single Weight* of the cross-section represents the mass with reference to a standard length. For tapered cross-sections, the average is determined of both cross-sections.

Weight

The values of this column represent the product of columns C and G.

Total Weight

The last column displays the total weight of every part.

Sum

The final row gives the sums of every column. The required amount of steel is displayed in the final cell *Total Weight*.

4.8 Parts List by Set of Members

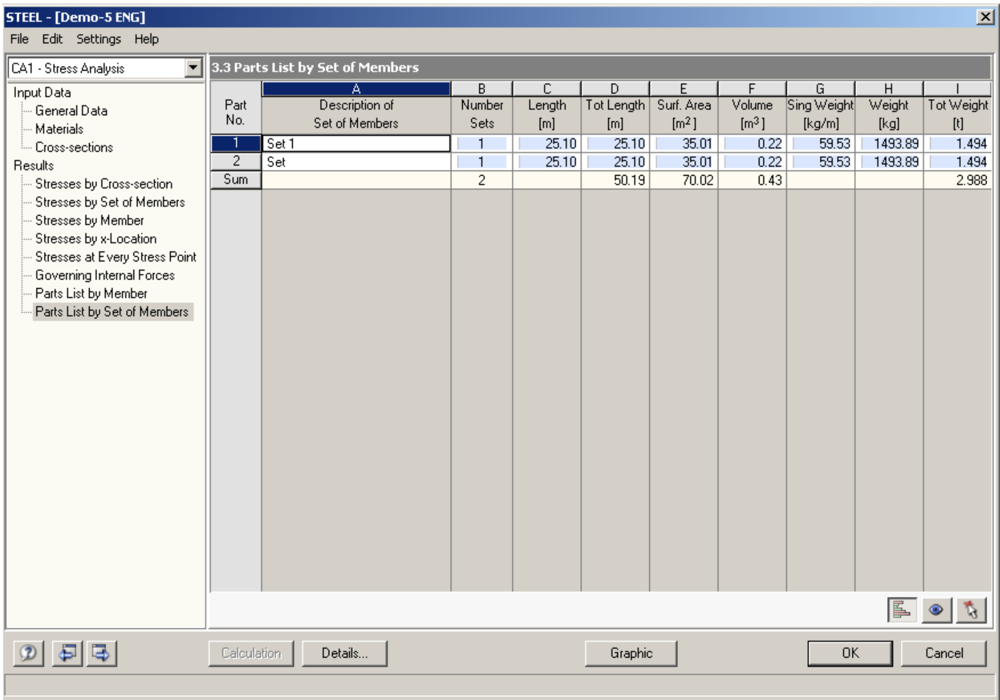


Figure 4.9: Mask 3.3 *Parts List by Set of Members*

The last STEEL mask is only available if one or more sets of members have been selected for the design. The output by sets of members allows for a specific parts list of a complete structural group (e.g. a frame).

The individual columns are presented in the pervious chapter 4.7. If there are different cross-sections within the set of members, the average values of surface area, volume and single weight will be determined.

5. Result Evaluation

After the design, the results can be evaluated in different ways. The buttons below the graphics on the right are used for the different evaluation types.

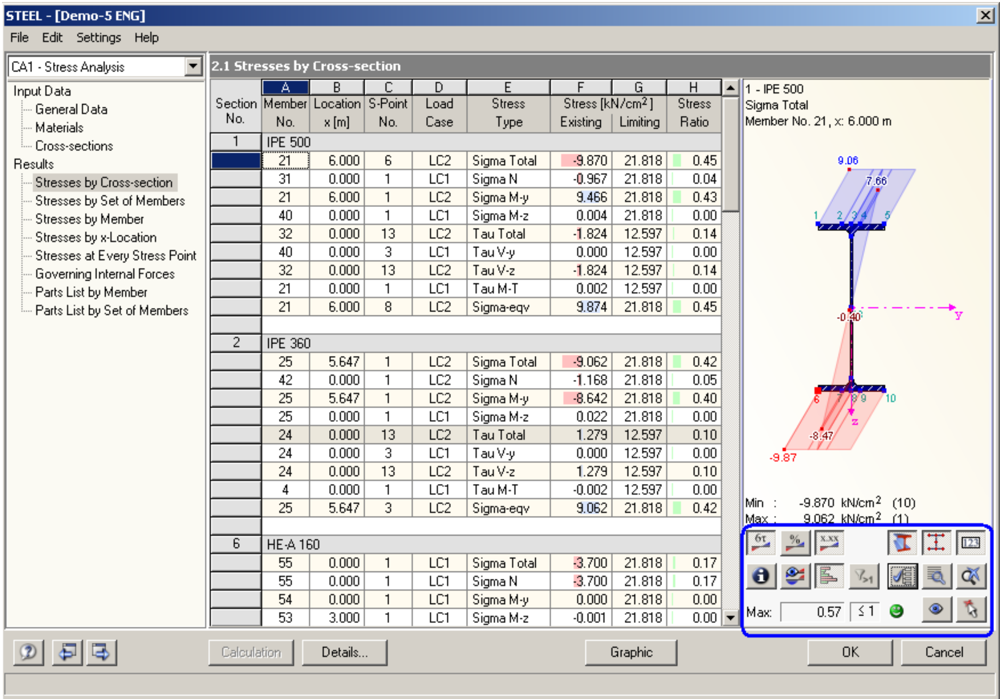
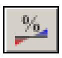
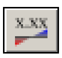
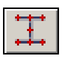
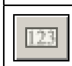
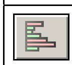


Figure 5.1: Buttons for result evaluation

The buttons in this dialog have the following functions:

Button	Description	Function
	Stress Diagram	Switches the graphic display of the stresses on the cross-section on and off
	Stress Ratio	Switches the graphic display of the stress ratio on and off
	Values	Switches the values in the graphics of the stresses or stress ratio on and off
	Cross-Section Outlines	Controls the display of the outlines in the cross-section graphics
	Stress Points	Switches the stress points in the cross-section graphics on and off
	Numbering	Switches the numbering of the stress points on and off
	Cross-Section Info	Calls up the *Info about Cross-Section* dialog with the parameters of the current cross-section
	Result Diagram	Opens the *Result Diagram on Member* → chapter 5.4, page 40
	Relation Scales	Switches the color bars in the results masks on and off

	Exceeded Ratio > 1	Shows only rows with stress ratios larger than 1 and, accordingly, failed design
	Select Stresses	Opens the *Stresses - Filter* dialog → chapter 5.1, page 34
	Extended Stress Diagram	Opens the *Cross-Section Values - Stress Diagram* dialog → chapter 5.2 page 35
	General View	Displays the stress graphics again completely (Zoom with scroll wheel, move via Drag & drop)
	View Mode	Allows jumping to the RSTAB work window in order to arrange a different view
	Pick Member	A member can be selected in the RSTAB window whose stresses will then be set in the table.

Table 5.1: Buttons in the Results Masks 2.1 to 2.5

5.1 Selection of the Stresses

After the design, the following stress types are presented by default:

- Axial stresses σ_{total}
- Shear stresses τ_{total}
- Equivalent stress σ_{eqv}

Via the button [Select Stresses to Show], additional stress types can be activated. They can be used to check the individual stress components that are contained in the total stresses.

Figure 5.2: Button [Select Stresses to Show]

The following dialog appears:

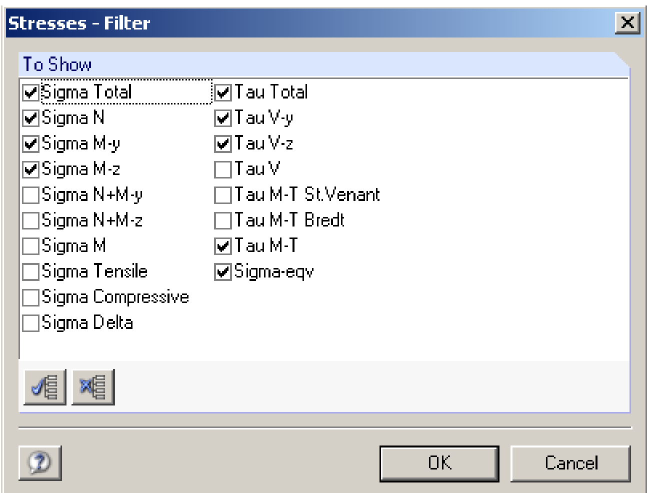


Figure 5.3: Dialog *Stresses - Filter*

In this dialog, the relevant stress types can be selected. Details to the individual stresses can be found in Table 3.1 and Table 3.2 on page 18 ff.

Two buttons assist with the selection. These have the following functions:

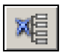
Button	Description	Function
	Select all	All stress components are selected.
	Deselect all	All stresses types are deselected.

Table 5.2: Buttons in the *Stresses - Filter* dialog

The design is carried out at every single stress point. Thus, usually not all components of the maximum stresses must be added for the combined evaluation (e.g. σ_{total}) as those appear at different stress points. For the correct evaluation, only the stress components of the selected stress point are to be superimposed.

5.2 Results on Cross-Section

The graphics of the stresses are displayed to the right of the tabular listing of the stresses. These are dynmacial graphics because they show the stresses at the current location x resp. stress point of the cursor position in the table. The current stress point is highlighted in red.

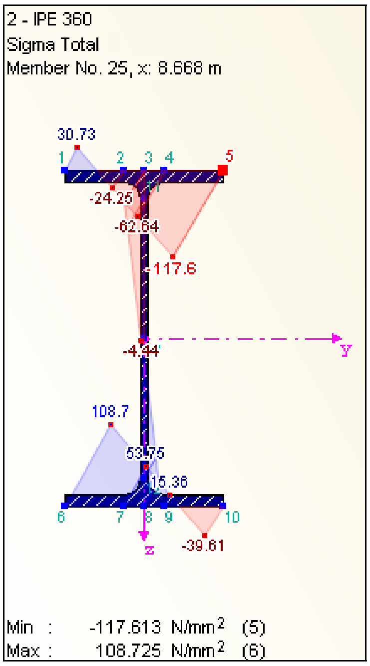
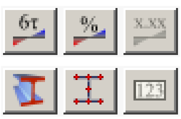


Figure 5.4: Distribution of the axial and bending stresses on the cross-section

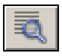
With the scroll wheel, you can zoom the display in and out, via drag & drop you can move the stress graphics. The button shown on the left restores the general view.



Details to the buttons below can be found in Table 5.1 on page 34. With these you can control whether

- the stresses or stress ratio including their values,
- the cross-section outlines,
- the stress points and their numbering

are to be displayed.



Via the button [Cross-Section Values and Extended Stress Diagram], a specific evaluation of the stresses is possible for every stress point. It calls up the following dialog.

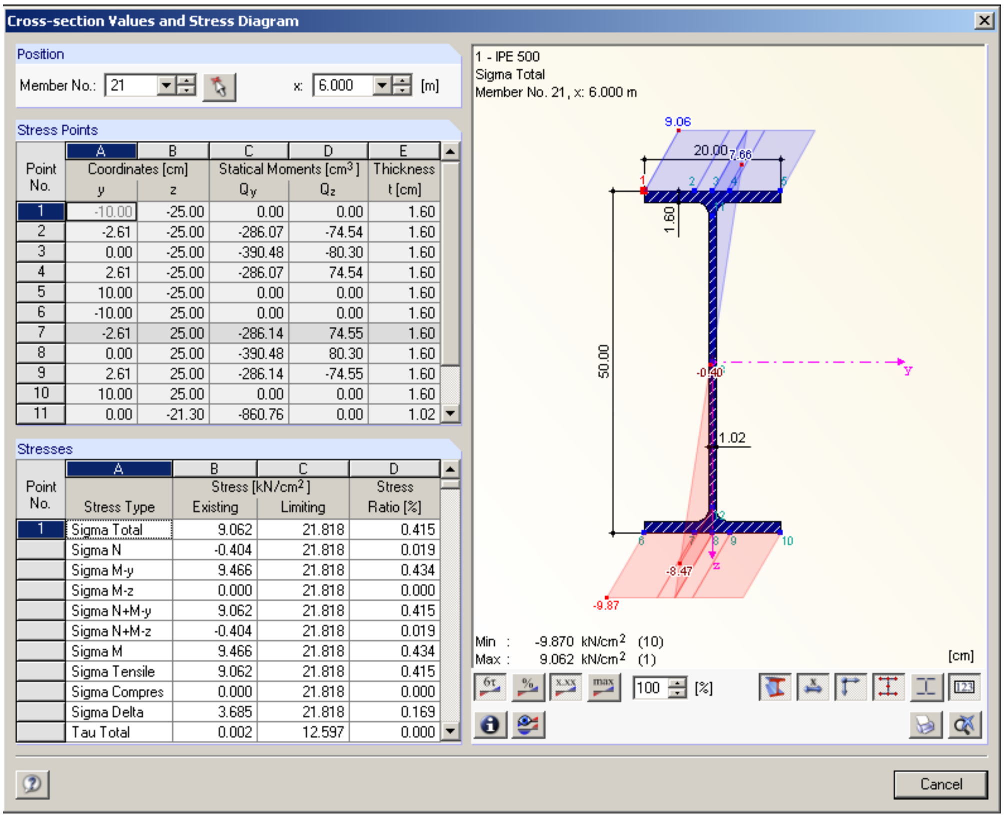


Figure 5.5: Dialog *Cross-Section Values and Stress Diagram*

In section *Position*, the current *Member No.* and the location *x* are preset. You can also select different members or x-locations from the list.

The section *Stress Points* lists all stress points of the cross-section. The columns *Coordinates* show the centroid distances e_y and e_z, the column *Statical Moments* present the statical moments of area Q_y and Q_z. The last column shows the *Thickness t* for the component that is relevant for the determination of the shear stresses.

Section *Stresses* below lists the individual stress components at the current stress point, i.e. the *Point* that is selected in the section above. Here too, you can select stress components that are then displayed dynamically in the graphics to the right.

The buttons below the graphics are largely similar to those presented in Table 5.1, page 34. They are also explained by *Quick Infos*. A special function is assigned to the [Print] button that allows you to print the current stress graphics. For more details, see chapter 6.2.1 on page 44.

5.3 Results on the RSTAB Model

You can also use the RSTAB work window for the evaluation of the design results. On the one hand, the RSTAB graphics in the background are of assistance when you want to check the location of a specific member in the model: The member that is selected in the STEEL results mask is highlighted in the RSTAB background graphics. Additionally, an arrow marks the location x on the member that is marked as governing in the current STEEL row.

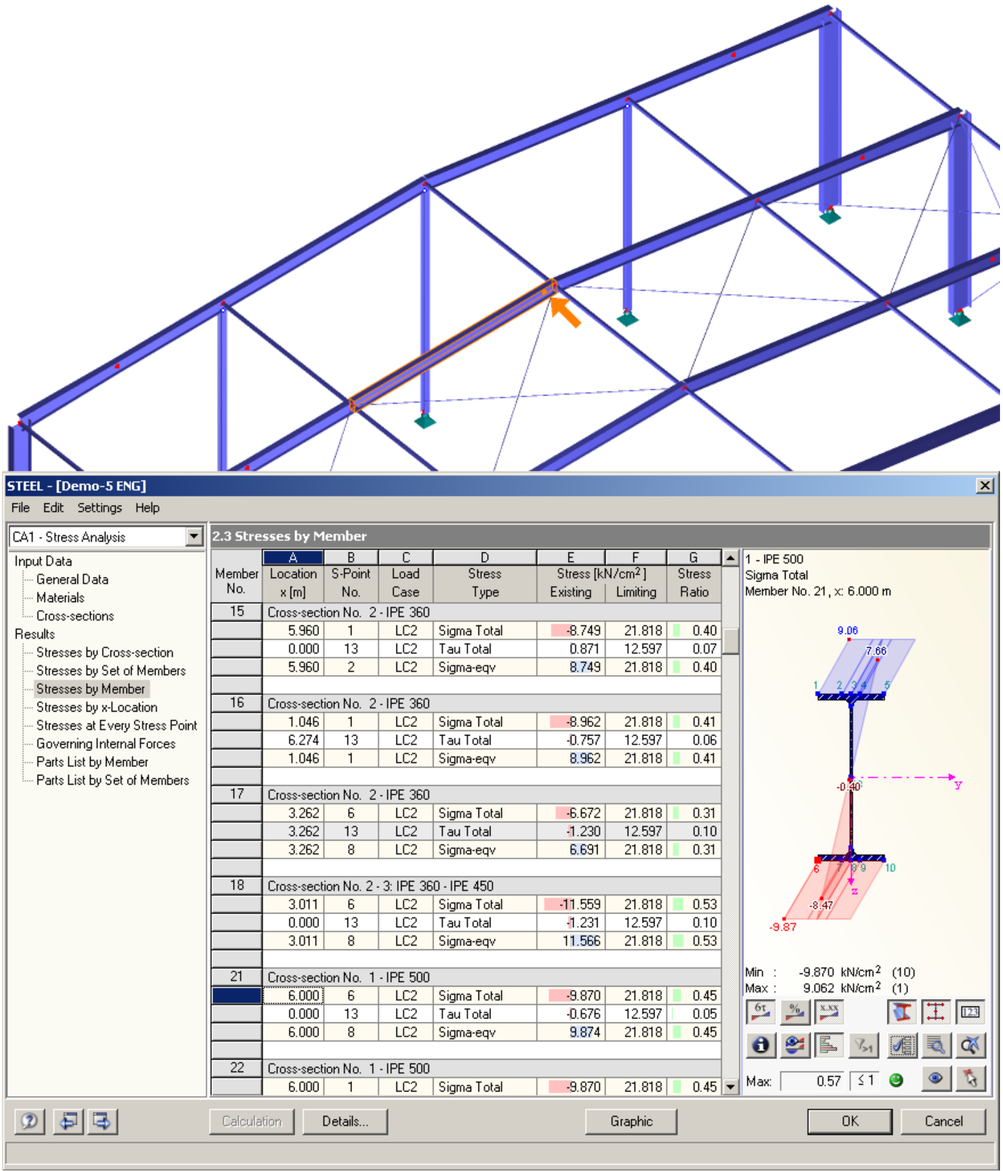
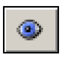
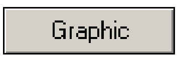


Figure 5.6: Identifying the *Member* and the current *Location x* in the RSTAB model



If you cannot correct a disadvantageous view by moving the STEEL window, then it is possible to use the so-called *View Mode* via the button [Change View] to adjust the view: The STEEL window will be hidden, and the view can be modified in the RSTAB interface. In this mode, you can access the *View* menu, e.g. to zoom, move or rotate the view.



On the other hand, the stresses as well as ratios can be visualized directly on the model of structure. The [Graphic] button is used to quit the STEEL module. The design results will be presented in the RSTAB work window graphically, comparable to the internal forces of a load case.

The items of the *Results* navigator are adjusted to the STEEL design results. Here, all stress components as well as the ratios referring to the single stress components are available.

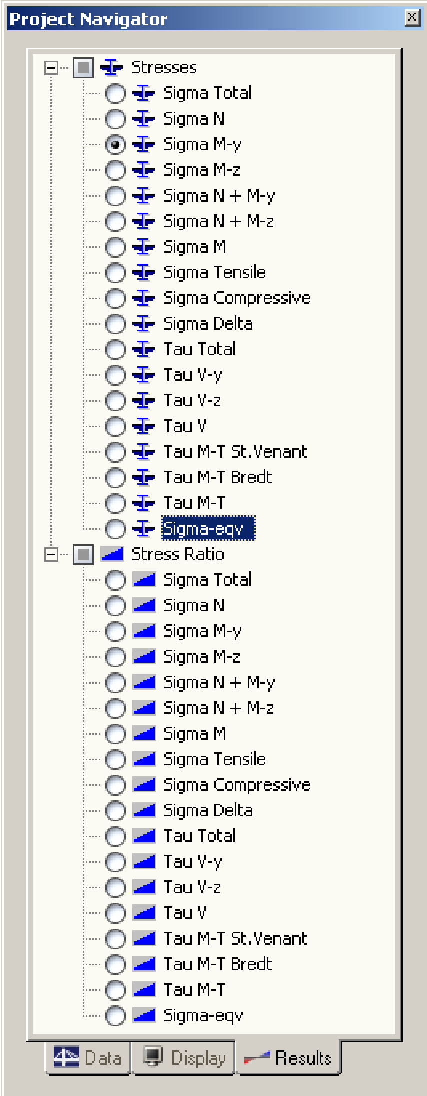
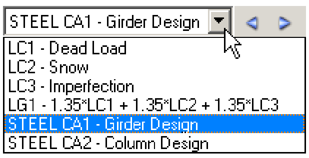


Figure 5.7: *Results* Navigator

As for the RSTAB internal forces, the button [Results on/off] switches the display of the design results on or off. The button [Show Result Values] controls the display of the result values in the graphics.

Because the RSTAB tables are irrelevant for evaluating the STEEL results, you can deactivate the table display.



The selection of the design cases is carried out as usual via the list in the RSTAB menu bar.

The presentation of the results can be controlled in the *Display* navigator among the item *Results* → *Members*. The stresses and stress ratios are displayed in two colors by default.

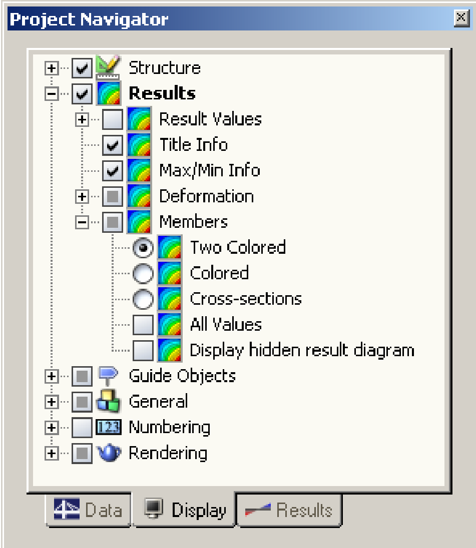
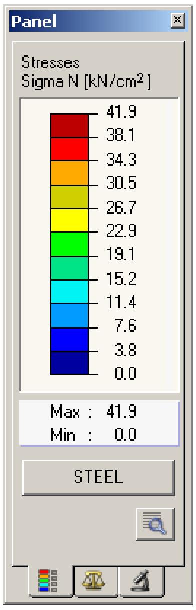


Figure 5.8: *Display* Navigator: Results → Members

The stresses are also in two colored view shown with their signs: Positive stresses are displayed in blue in the direction of the positive member axis *z*, negative stresses are drawn in red at the opposite side of the member. Therefore, it is possible that the sign and therefore the stress diagram swaps color and side if there are discontinuities on certain members, e.g. due to single loads.

For a multicolor view, the control panel is available. The description of this *Panel* can be found in the RSTAB manual (chapter 4.4.6, page 71 ff). As for the internal forces, you can scale the design results in register *Filter*. If you enter the factor *0* in the input field *Member Diagrams*, the stresses and ratios are automatically shown with an increased line thickness.



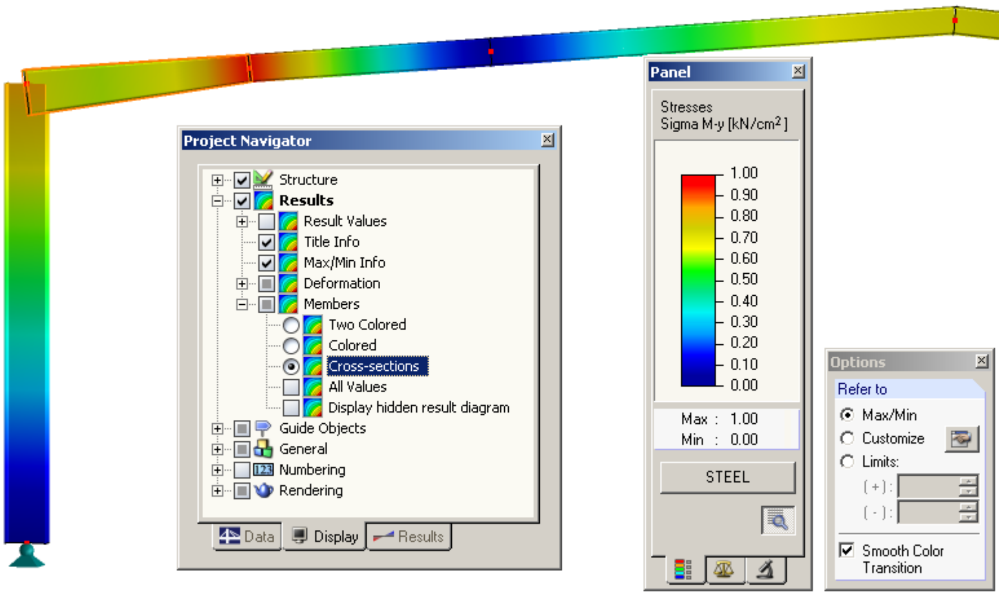


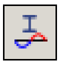
Figure 5.9: Stress ratios due to bending moment M_y with display type *Cross-Sections*

These graphics can be also printed to the RSTAB printout report (cf. chapter 6.2.2, page 45).

You can always return to the STEEL module via the [STEEL] button in the panel as shown on the left.

5.4 Result Diagrams

If you want to view the results of one or more selected members, then also the result diagrams can be used. Select the member(s) or set of members in the STEEL results mask and activate the result diagram via the button shown on the left.



Alternatively, the result diagrams are available in the RSTAB graphics via menu

Results → Result Diagrams on Selected Members

or the corresponding button in the RSTAB toolbar.

A new window appears that displays the result diagrams of the member(s) or set of members.

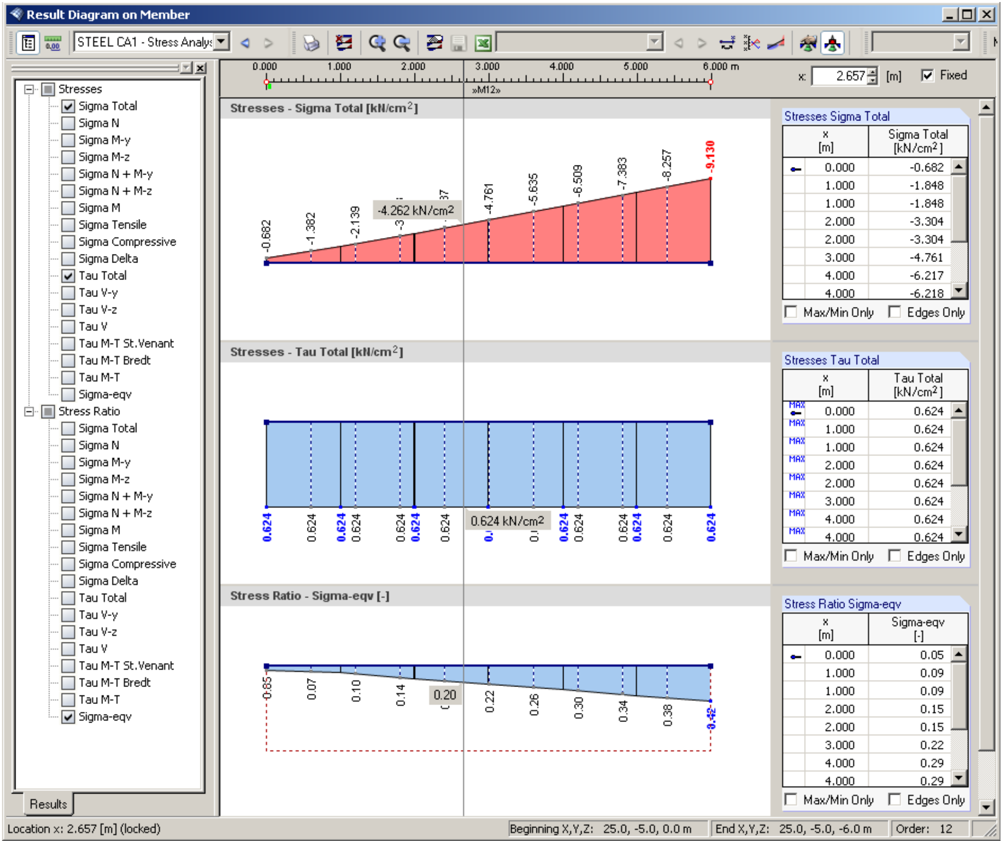


Figure 5.10: Dialog *Result Diagrams on Member*

In the navigator on the left, the stresses and stress ratios can be selected that are to be displayed in the diagram. Via the list of the tool bar, you can choose among the different STEEL design cases.

The detailed description of the *Result Diagram on Member* dialog can be found in chapter 9.8.4 of the RSTAB manual, page 221 ff.

5.5 Filtering Results

Apart from the STEEL results masks that allow for a selection according to specific criteria due to their structuring, the filter options as described in the RSTAB manual are available to graphically evaluate the STEEL design results.

You can use already-defined partial views (cf. RSTAB manual, chapter 9.8.6, page 224 ff) which group objects in a suitable manner.

It is also possible to use the stresses and stress ratios as filter criteria within the RSTAB window. For this, the panel is to be displayed. If it is not visible, you can activate it via menu

View → Panel

or the corresponding button in the *Results* toolbar.

Details to the control panel can be found in chapter 4.4.6 of the RSTAB manual, page 71 ff. The filter settings for all results are carried out in the register *Color Spectrum*. As this register is not presented in two-colored display, it can be switched on by selecting the display types *Colored* or *Cross-Sections* in the *Display* Navigator.

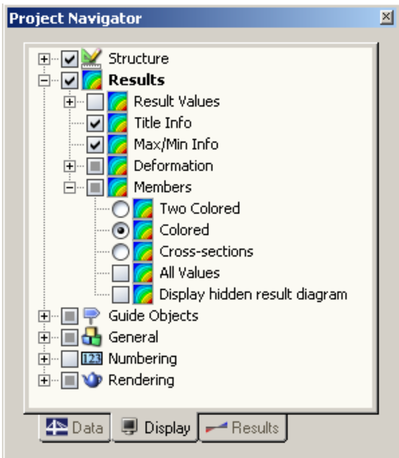


Figure 5.11: *Display* Navigator: Results → Members → **Colored**

For a colored view of the stresses, you can define in the panel that e.g. only the equivalent stresses greater than 100 N/mm´ are to be displayed. Additionally, the color spectrum can be adjusted so that each color range covers 10 N/mm´ (see Figure 5.12).

Via the option *Display hidden result diagram* (in the *Display* navigator among the item *Results → Members*), also all stresses can be displayed that do not fulfill this requirement. These would be shown as dotted lines.

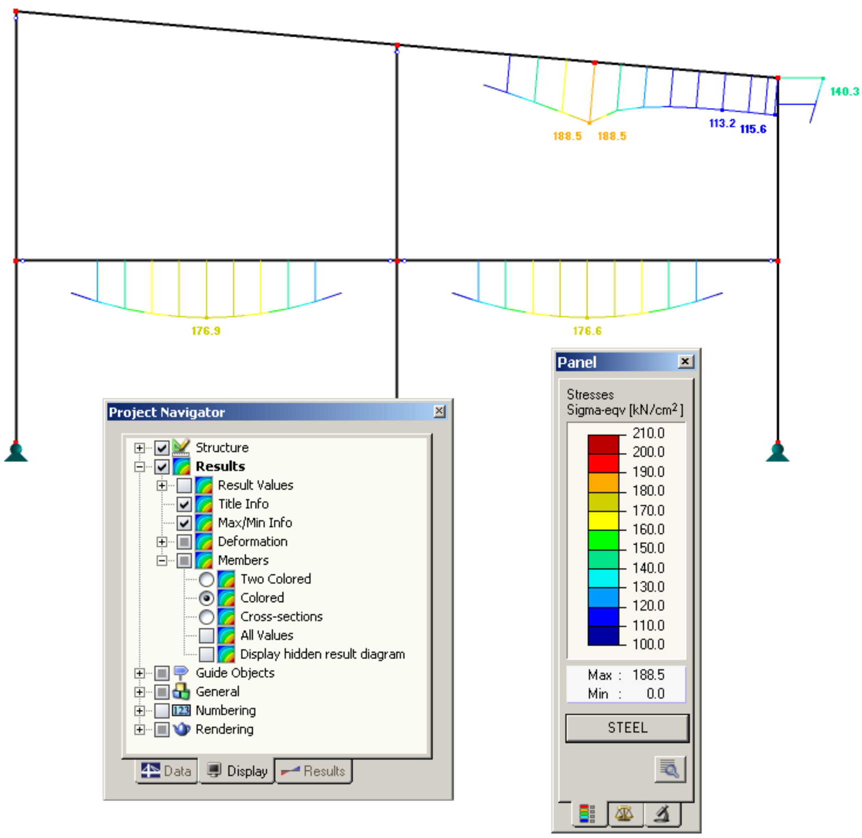


Figure 5.12: Filtering stresses with adjusted color spectrum

Filtering Members

In the register *Filter* of the panel, you can specify the numbers of the members whose stresses are to be shown exclusively in the graphics. Details to this function can be found in chapter 9.8.6 of the RSTAB manual on page 224.

Unlike the partial view, here the structure is shown completely. The following figure shows the axial stresses in the inner frame of a structure. The remaining members are displayed in the model but are without internal forces.

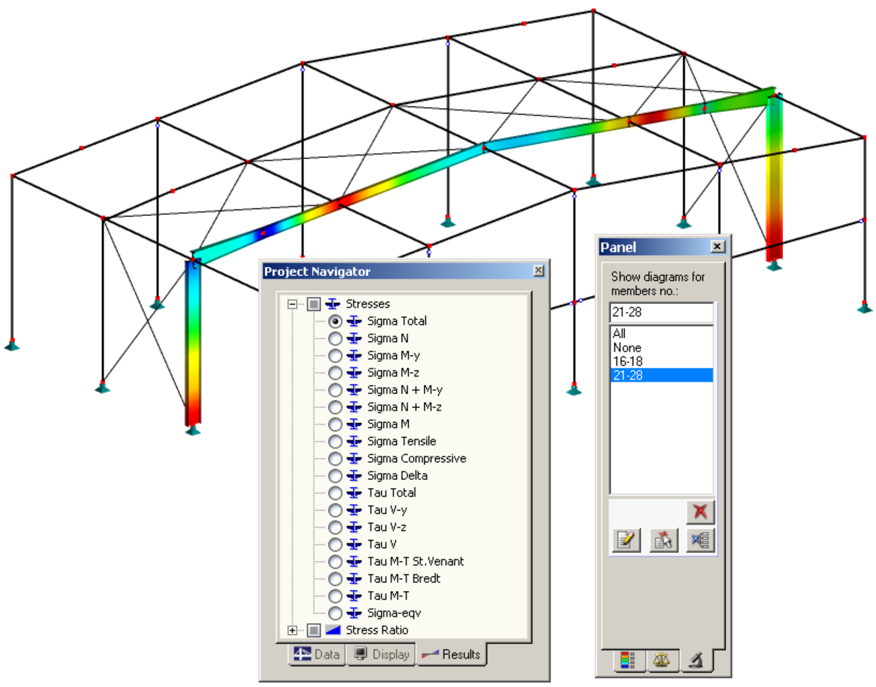


Figure 5.13: Filtering members: Axial stresses of a selected frame

6. Printout

6.1 Printout Report

Also for the STEEL design results, a printout report is generated that can be amended by graphics and descriptions. There, you can also define which STEEL result masks are to be included in the printout report.

Details to the printout report can be found in the RSTAB manual. Especially chapter 10.1.3.4 *Selection of Add-On Modules Data* (page 240) deals with the selection of the input and output data for all add-on modules.

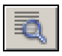
For very large structures, it is advisable to split the data into several reports. If you create an extra report with only the STEEL data of a design case, then this printout report can be easily created.

In the printout report, those stress components are listed that have been set in the selected STEEL design case for the result masks. If e.g. the stresses due to axial forces are to be included, the stresses σ_N are to be activated in the STEEL module. Details to this function can be found in chapter 5.1, page 34.

6.2 Printing STEEL Graphics

Similar to the results evaluation, the stresses and stress ratios can be used for the printout both on the cross-section as well as on the RSTAB model. It is possible to include the images in the printout report or to directly print them. Details on printing graphics can found in chapter 10.2 of the RSTAB manual.

6.2.1 Results on Cross-Section



The print function is indirectly accessible via the *Cross-Section Values and Stress Diagram* dialog (see Figure 5.5, page 36). This dialog can be called up in the results masks 2.1 to 2.5 via the button [Cross-Section Values and Stresses].

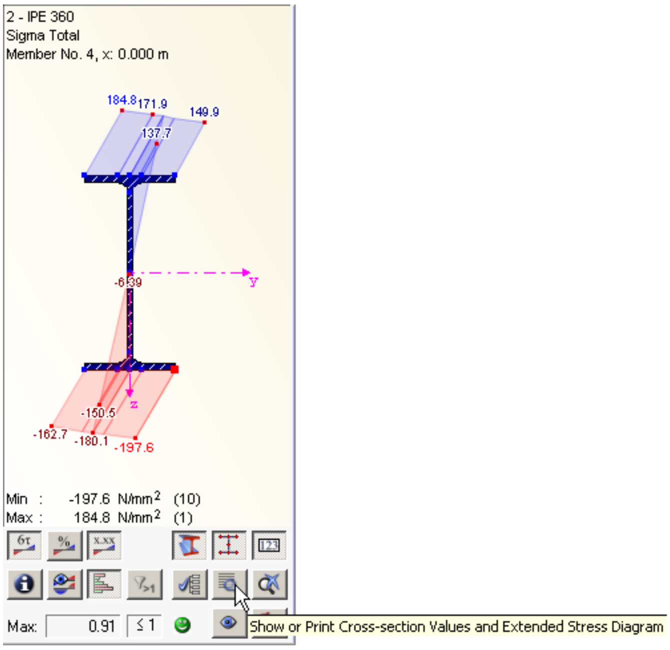


Figure 6.1: Button *Cross-Section Values and Stresses* in the graphics of the result masks

In the *Cross-Section Values and Stress Diagram* dialog, you can select the relevant member, location x and stress type whose diagram is to be printed. Via the [Print] button in this dialog (at the bottom to the right), the following dialog is called up.

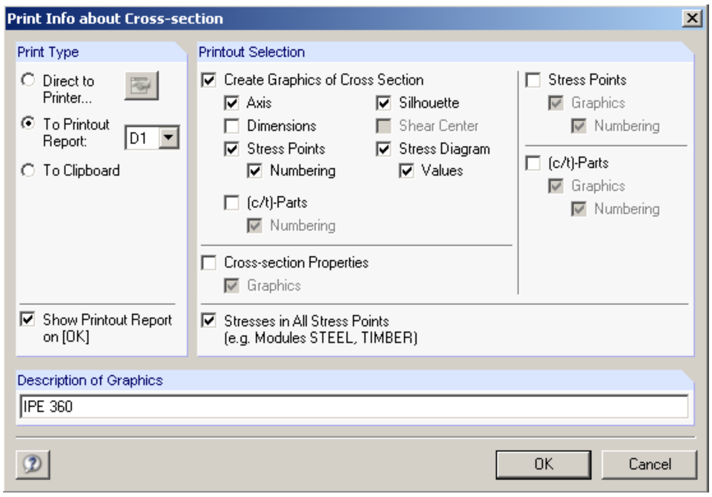


Figure 6.2: Dialog *Print Info about Cross-section*

The familiar options are available in section *Print Type*:

- *Direct to Printer* sends the current graphics to the printer.
- *To Printout Report* inserts the graphics in a printout report.
- *To Clipboard* makes the graphics available for other applications.

If several printout reports exist, the number of a specific report can be selected from the list.

The section *Printout Selection* controls which elements will appear in the printed image and in a numerical table. The objects listed under *Create Graphics of Cross-Section* do not require any additional explanation. If the option *Cross-Section Properties* is active, then the cross-section parameters will be printed tabularly. Symbolizing graphics can optionally be added at the margin. In the same way, the properties of the *Stress Points* and *(c/t) Parts* as well as the *Stresses in All Stress Points* can be included in the printout report.

When you close the dialog with [OK], the printout report will be opened by default. If you want print several graphics successively, it is recommended to deactivate the checkbox *Show Printout Report on [OK]*.

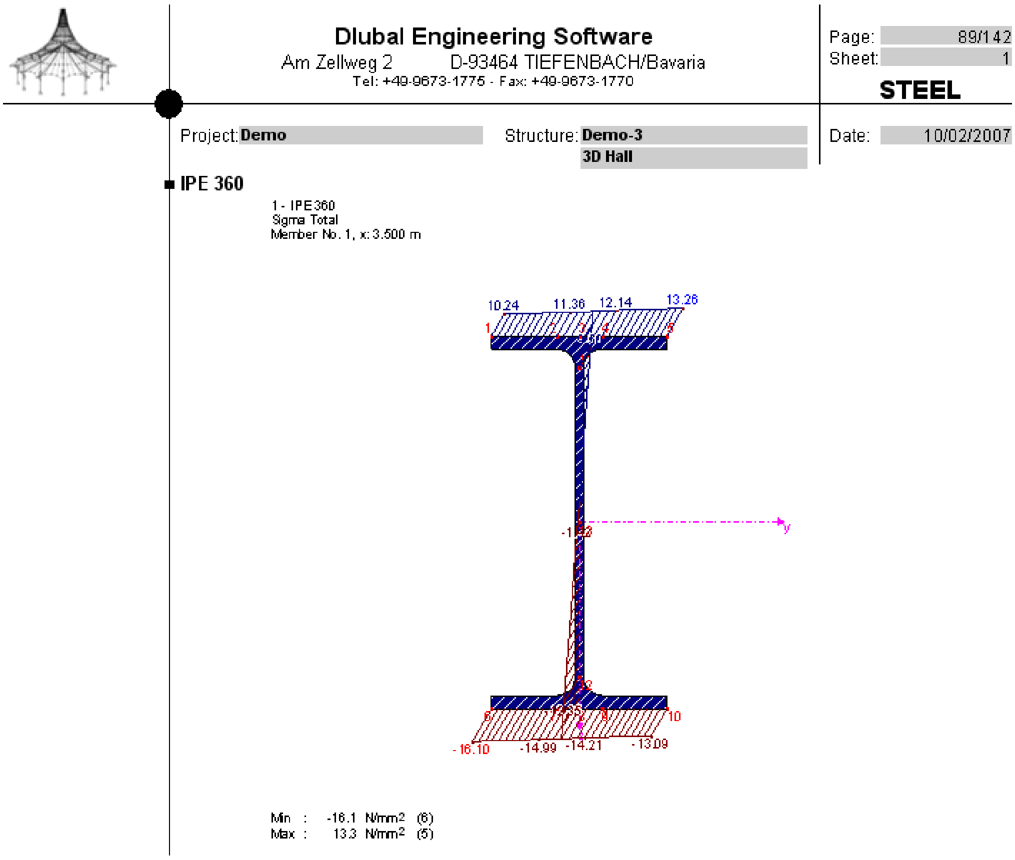


Figure 6.3: Stress graphics in the Printout Report

6.2.2 Results on the RSTAB Model

Like in RSTAB, every image that is shown in the graphics of the main program can be included in the printout report. In the same way, the result diagrams can be added to the report via [Print].

The current STEEL graphics can be printed via menu

File → Print

or via the appropriate button in the toolbar.

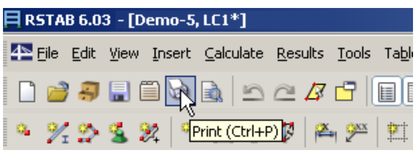


Figure 6.4: Button *Print* in the RSTAB toolbar

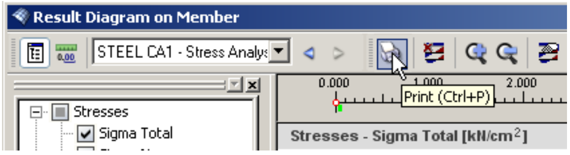


Figure 6.5: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog appears.

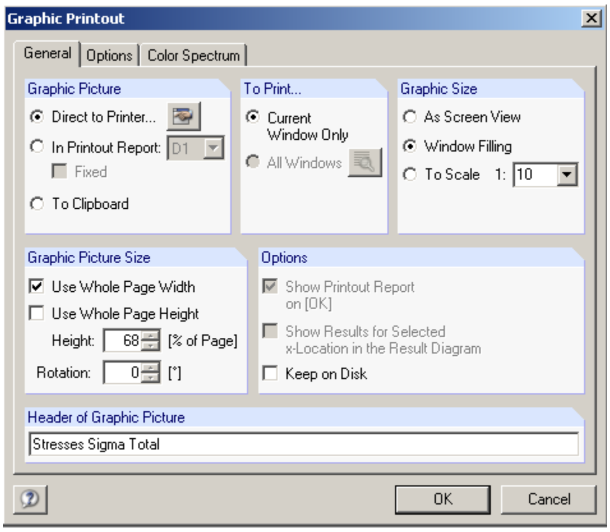
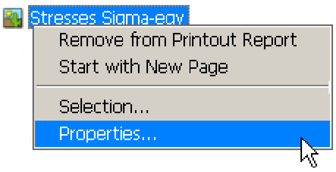


Figure 6.6: Dialog *Graphic Printout*, register *General*

Details to this dialog can be found in chapter 10.2 of the RSTAB manual, page 259 ff. There, also the detailed description of the registers *Options* and *Color Spectrum* can found.



In the printout report, every STEEL image can be moved by drag & drop to a different location. It is also possible to modify an image: Click on the selected item in the report navigator with the right mouse button and select its *Properties* from the context menu. Again the dialog *Graphics Printout* with several options appears to edit the image.

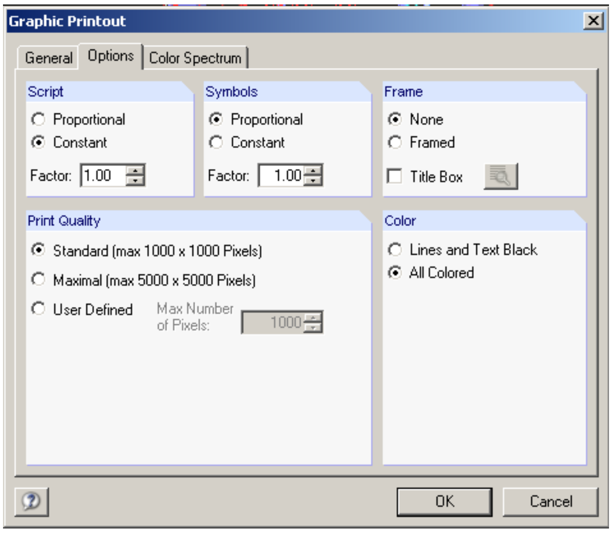


Figure 6.7: Dialog *Graphic Printout*, register *Options*

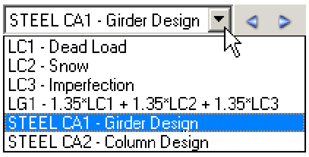
7. General Functions

The last chapter explains some menu functions as well as export options of the design results.

7.1 STEEL Design Cases

You can organize the members in separate design cases. With this, you can combine certain components of the structure or to assigne specific design parameters (e.g. limit stresses, partial safety factors, optimization etc.).

A member or a set of members can be examined without any problem in different design cases.



The STEEL cases are available in the RSTAB work area like any load case or load group in the list of the menu bar.

Create a New STEEL Case

A new design case can be created via the STEEL menu

File → **New Case**.

The following dialog appears.

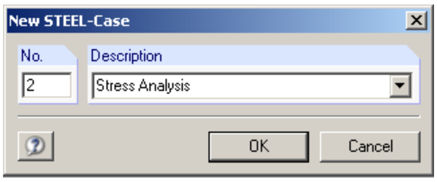


Figure 7.1: Dialog *New STEEL Case*

In this dialog, you can enter a *Number* that has been not used yet and a *Description* for the new design case. After [OK], the STEEL mask 1.1 *General Data* for the input of the new data appears.

Rename STEEL Case

The description of a design case can be modified via the STEEL menu

File → **Rename Case**.

The dialog *Rename STEEL Case* appears.

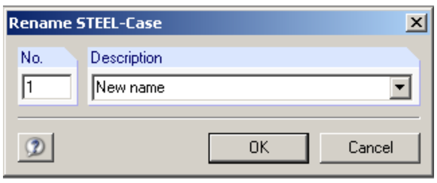


Figure 7.2: Dialog *Rename STEEL Case*

Copy STEEL Case

The input data of the current design case are copied via the STEEL menu

File → Copy Case.

The dialog *Copy STEEL Case* appears. Here you have to enter the number and a description for the new design case.

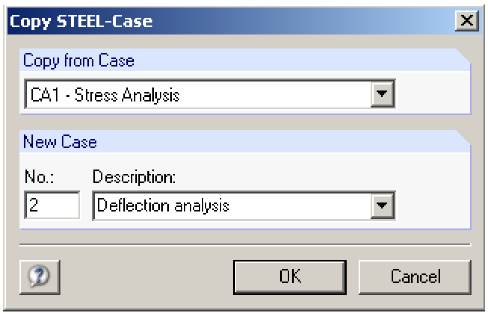


Figure 7.3: Dialog *Copy STEEL Case*

Delete STEEL Case

Design cases can be deleted via the STEEL menu

File → Delete Case.

In dialog *Delete STEEL Case*, you can select a specific STEEL case that is to be deleted.

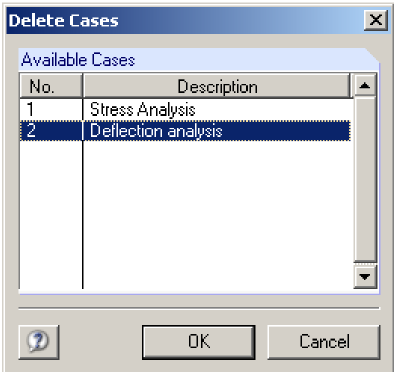


Figure 7.4: Dialog *Delete Cases*

7.2 Cross-Section Optimization

STEEL is able to optimize cross-sections during the design. For this, you have to select the cross-section by activating the checkbox in column D of mask 1.3 (see Figure 2.5, page 14).

During the optimization, STEEL examines which cross-section within the same cross-section series fulfills the design "best", i.e. closest to the maximum stress ratio of 1.0. With the internal forces from RSTAB, the required moment of inertia is determined. After this, the cross-section is chosen from the relevant cross-section group that fulfills the design with the highest possible ratio. As a result, two cross-sections are shown graphically in mask 1.3 (cf. Figure 2.5, page14): the original cross-section from RSTAB and the optimized cross-section.

You can enter detailed settings for the parametrized cross-sections. After activating the optimization box, a dialog appears.

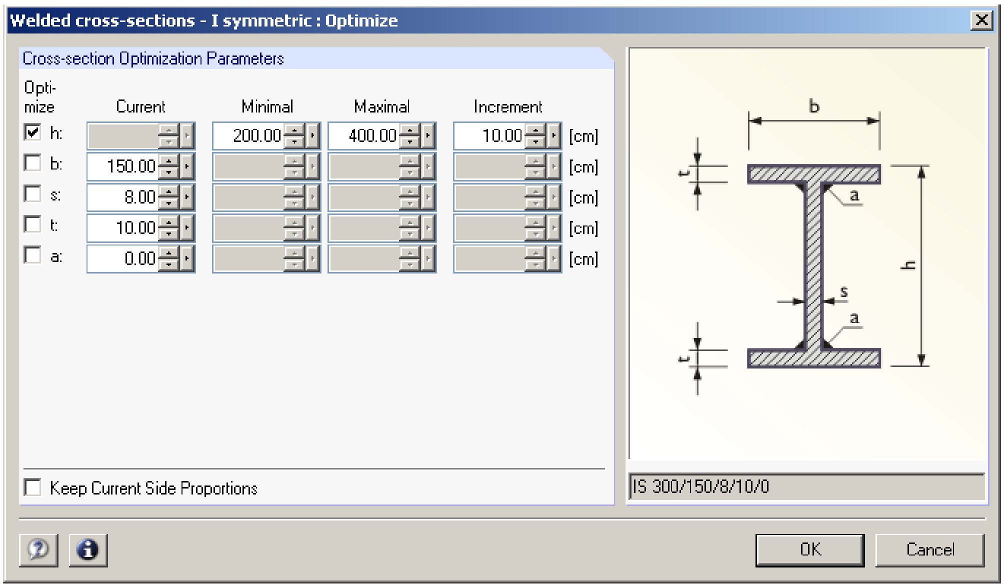


Figure 7.5: Dialog *Welded Section - I Symmetric: Optimize*

In column *Optimize*, you can select the parameter that is to be modified by activating the checkbox. This makes the columns *Minimal* and *Maximal* accessible. These define the lower and upper limits of the parameter for the optimization. The column *Increment* controls in which intervals the dimensions of this parameter vary during the optimization process.

To *Keep the Current Side Proportions*, you have to activate the checkbox below the parameter settings. In addition, all parameters have to be activated for the optimization.

No optimization can be carried out for combined rolled cross-sections.

When optimizing, keep in mind that the internal forces are automatically recalculated together with the modified cross-sections. The user can decide when and which cross-sections are to be recalculated in RSTAB. Due to the changed stiffnesses within the model, significant differences can occur in the internal forces that result from optimized cross-sections. It is therefore advisable to recalculate the internal forces after a first optimization run and then optimize the cross-sections once more.

The export of the modified cross-sections to RSTAB does not need to be carried out by hand. Open mask 1.3 *Cross-Sections* and select menu

Edit → Export All Cross-Sections to RSTAB.

Also the context menu of the cross-sections in mask 1.3 can be used to export modified cross-sections to RSTAB.

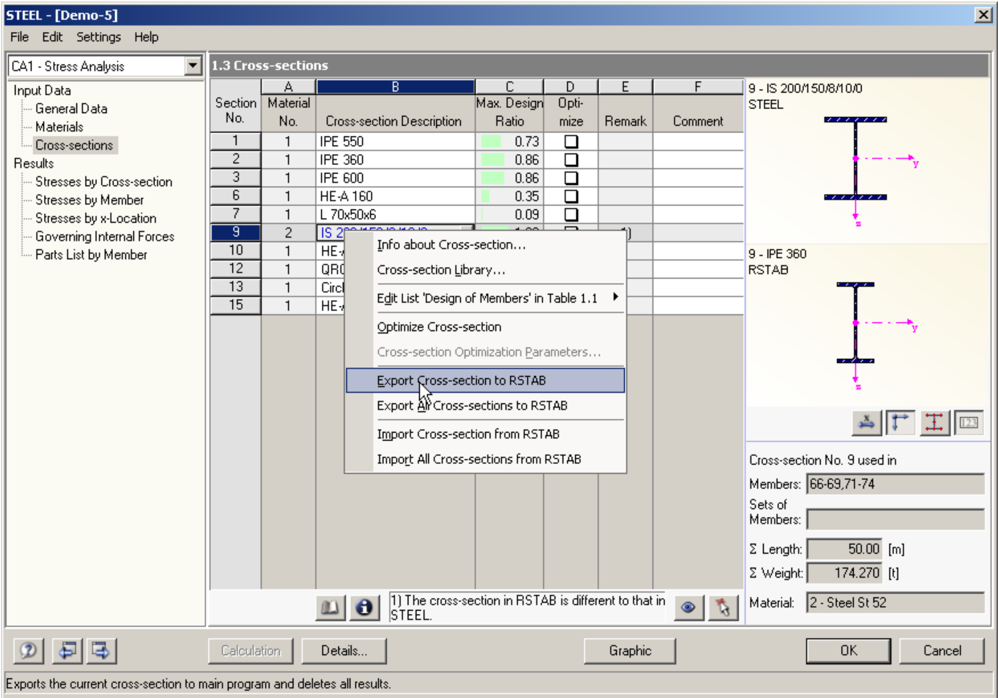


Figure 7.6: Context menu of Mask 1.3 *Cross-Sections*

Because this action involves the deleting of results, a query appears before exporting. If the [Calculation] is then started in STEEL, then the calculation of both the internal forces in RSTAB and the stresses in STEEL is carried out in one calculation run.

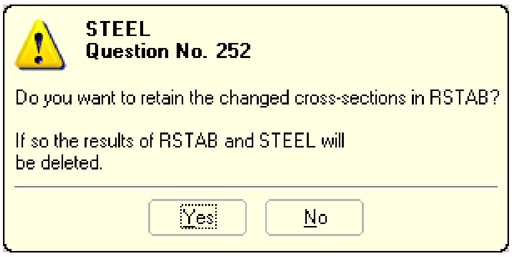


Figure 7.7: Query before exporting modified cross-section to RSTAB

You can also reimport the original RSTAB cross-sections via the above described menu functions. This option is only available in mask 1.3 *Cross-Sections*.

If a tapered member is to be optimized, then the start and end locations are optimized. After this, the moments of inertia can be linearly interpolated on the intermediate locations. The stress analysis becomes inaccurate when encountering extremely different start and end cross-sections as these are taken into account by the power of four. In this case, the tapered members should be divided in single members whose start and end cross-sections have smaller differences.

7.3 Units and Decimal Places

The units and decimal places are managed globally for RSTAB and for all add-on modules. In STEEL, the dialog for setting the units is available via menu

Settings → Units and Decimal Places.

The dialog that is already known from RSTAB is called up, and the module STEEL is preset.

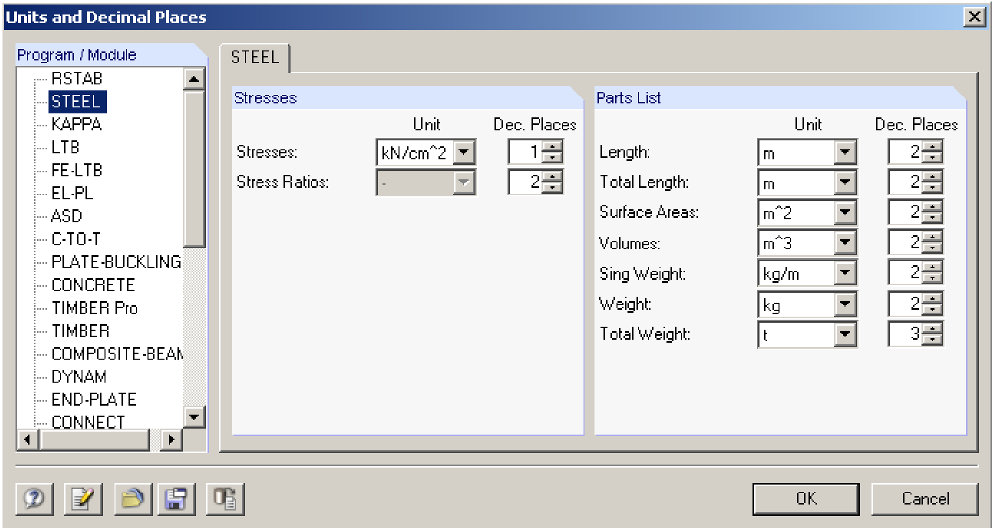
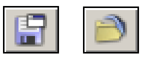


Figure 7.8: Dialog *Units and Decimal Places*



The settings can be saved as user profile and applied later in different structures. Details to this function can be found in chapter 11.6.2 of the RSTAB manual, page 344.

7.4 Exporting Results

The results of the stress analysis can be made available for other programs in different ways.

Clipboard

Marked cells of the STEEL results masks can be copied via [Ctrl]+[C] in the clipboard and then be inserted via [Ctrl]+[V] e.g. in a text processing program. The headers of the table columns are not considered for this.

Printout Report

The STEEL data can be printed to the printout report (cf. chapter 6.1, page 43) and then be exported via menu

File → Export to RTF file or BauText.

Details to this function can be found in chapter 10.1.11, page 254 of the RSTAB manual.

MS Excel Format *.xls

STEEL allows to directly export data to MS Excel. This function can be activated via

File → Export in MS Excel.

The following export dialog is opened.

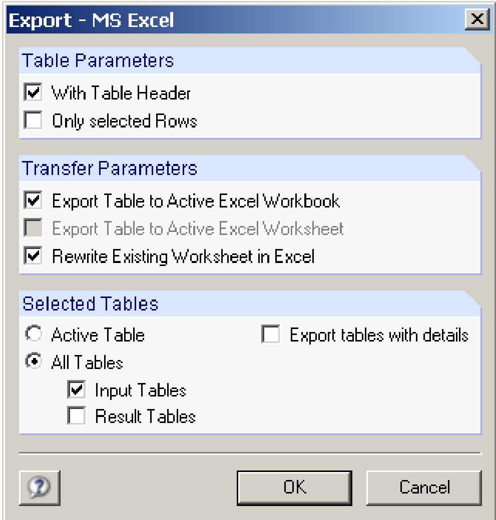


Figure 7.9: Dialog *Export - MS Excel*

The export can be started via [OK] as soon as the parameters have been selected. Excel is automatically called up, it does not need to be opened in the background.

	A	B	C	D	E	F	G	H	I
1	Section	Member	Location	S-Point	Load	Stress	Stress [MPa]		Stress
2	No.	No.	x [m]	No.	Case	Type	Existing	Limiting	Ratio
3	1	IPE 550							
4		12	1,286	10	LC1	Sigma Total	-158,6	218,2	0,73
5		40	0,000	13	LC1	Tau Total	26,0	126,0	0,21
6		12	1,286	10	LC1	Sigma-eqv	158,6	218,2	0,73
7									
8	2	IPE 360							
9		45	3,262	6	LC1	Sigma Total	-188,5	218,2	0,86
10		17	3,262	13	LC1	Tau Total	-21,6	126,0	0,17
11		45	3,262	8	LC1	Sigma-eqv	188,5	218,2	0,86
12									
13	6	HE-A 160							
14		53	3,000	6	LC1	Sigma Total	-76,2	218,2	0,35
15		51	0,000	13	LC1	Tau Total	6,8	126,0	0,05
16		53	3,000	6	LC1	Sigma-eqv	76,2	218,2	0,35
17									
18	7	L 70x50x6							
19		122	0,000	1	LC1	Sigma Total	19,5	218,2	0,09
20		121	0,000	1	LC1	Tau Total	0,0	126,0	0,00
21		122	0,000	1	LC1	Sigma-eqv	19,5	218,2	0,09

Figure 7.10: Results in *Excel*

A Literature

[1] DIN 18 800 Teil 1: Stahlbauten, Bemessung und Konstruktion, 1990

[2] Erläuterungen zu DIN 18 800 Teil 1 bis 4, Beuth-Kommentar, Beuth Verlag, 2. Auflage 1994

[3] PETERSEN, Chr.: Stahlbau, Vieweg und Sohn, Braunschweig/Wiesbaden, 3. Auflage 1993

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

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

B Index