Version
March 2010

Add-on Module

RF-STEEL

Stress Analysis for Surfaces and Members

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

Contents

1. Introduction

1.1 RF-STEEL Add-on Modules

Both RF-STEEL programs are integrated as additional modules in the graphical user interface of the main program RFEM. They are no stand-alone modules, which means that the main program is always required. **RF-STEEL Surfaces** is used for the stress design of surface and shell elements. Member elements are designed by **RF-STEEL Members**. Both modules are described in this manual.

The design relevant input data as well as the internal forces are imported automatically when you open the modules. Of course, the design results from RF-STEEL Surfaces and RF-STEEL Members are also available in the RFEM work window to evaluate them graphically and to integrate them into the global printout report.

RF-STEEL carries out general stress designs according to the elastic-elastic method by calculating existing stresses and comparing them with the limit stresses. The programs provide an expandable material library with the standard-specific limit stresses as well as a comprehensive cross-section library for member elements. For each member cross-section, design relevant stress points that can also be used for graphical evaluations are already available.

During the stress analysis process, the maximum stress ratios of surfaces, members and sets of members are determined. The governing internal forces of each member and set of members are documented additionally. Furthermore, the modules provide optimization options for surfaces and cross-sections including a transfer function to transfer data to RFEM.

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

Some of the new features in both RF-STEEL modules are the following:

- Determination of equivalent stresses according to different approaches:
  VON MISES, TRESCA, RANKINE, BACH
- Selection of calculation method for load combinations
- Serviceability limit state design by checking surface displacements
- Output of stress ranges Δ for fatigue designs
- Optimization of surface thicknesses and option to transfer data to RFEM
- Selection of a maximum stress ratio for optimization process
- Export option for modified materials to RFEM
- Detailed output of different stress components and ratios in tables and graphic
- Filter function for surface, line and node numbers in tables
- Output of governing internal forces for sets of members
- Parts list of designed surfaces
- Direct data export to MS Excel and OpenOffice.org Calc

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

Your team from DLUBAL ENGINEERING SOFTWARE

1.2 RF-STEEL Team

The following people were involved in the development of RF-STEEL:

Program coordination

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem

Programming

Ing. Zdeněk Kosáček
Mgr. Petr Oulehle
Ing. Tomáš Pecholt
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. Hana Robovská
Ing. Martin Vasek

Manual, help system and translation

Dipl.-Ing. (FH) Robert Vogl
Jan Jeřábek
Dipl.-Ü. (Uni) Gundel Pietzcker

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (BA) Sandy Baumgärtel
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Alexandra Lazar
M.Sc. Dipl.-Ing. (FH) Frank Lobisch
Dipl.-Ing. (FH) Alexander Meierhofer
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

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 RFEM. The present manual focuses on typical features of the RF-STEEL add-on modules.

The description of the modules follows the sequence of their input and results tables as well as their structure. **RF-STEEL Surfaces** is described in chapter 2, **RF-STEEL Members** in chapter 3. The subsequent chapters describe common functions.

The text of the manual shows the described **buttons** in square brackets, for example [New]. At the same time, they are pictured on the left. In addition, **expressions** used in dialog boxes, tables and menus are set in *italics* to clarify the explanations.

Finally, you find an index at the end of the manual. 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 Start RF-STEEL Modules

RFEM provides the following options to start the RF-STEEL add-on modules

Menu

To start the programs,

> point to **Design - Steel** in the **Additional Modules** menu, and then select **RF-STEEL Surfaces** or **RF-STEEL Members**.

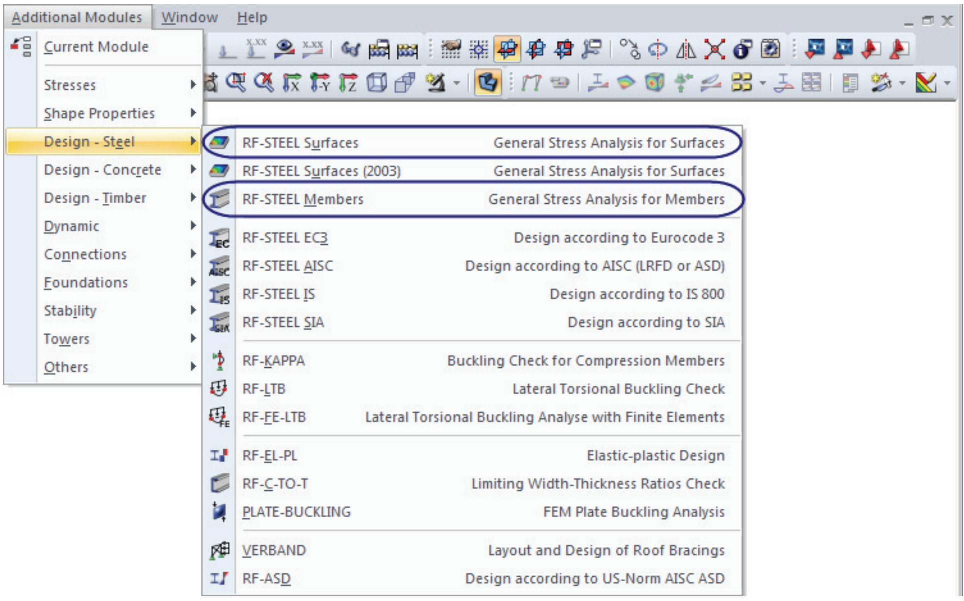


Figure 1.1: Menu: *Additional Modules → Design - Steel → RF-STEEL Surfaces* or *RF-STEEL Members*

The add-on module **RF-STEEL Surfaces (2003)** is a previous version. The present manual does not go into detail regarding its description.

Navigator

To start the RF-STEEL modules in the *Data* navigator,

open the **Additional Modules** folder and select **RF-STEEL Surfaces** or **RF-STEEL Members**.

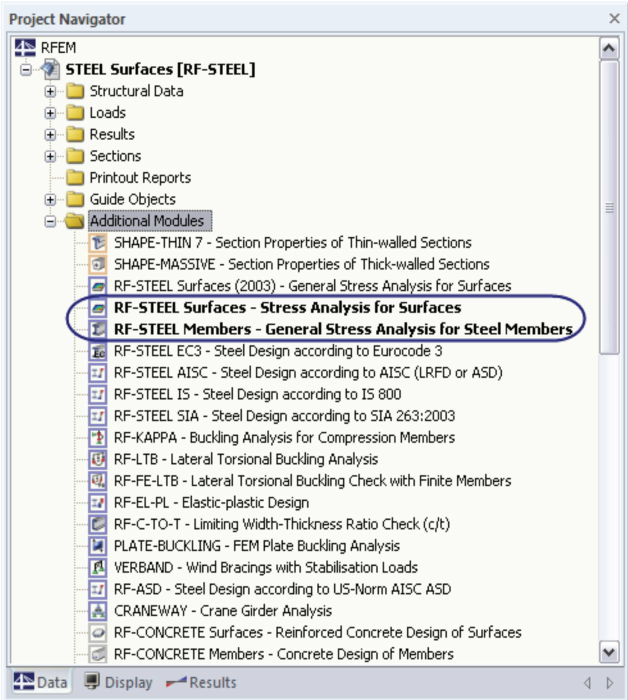
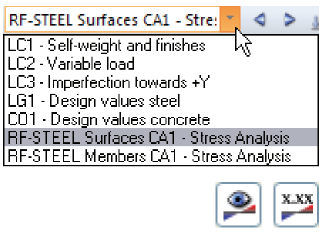


Figure 1.2: Data navigator: *Additional Modules* → *RF-STEEL Surfaces* or *RF-STEEL Members*

Panel



In case RF-STEEL results are already available in the RFEM structure, you can set the corresponding RF-STEEL case in the load case list of the RFEM toolbar. By using the button [Results on/off], you can display the stresses or stress ratios in the graphic.

If the results display is activated, the panel appears showing the button [RF-STEEL Surfaces] or [RF-STEEL Members] which you can use to open the RF-STEEL module.

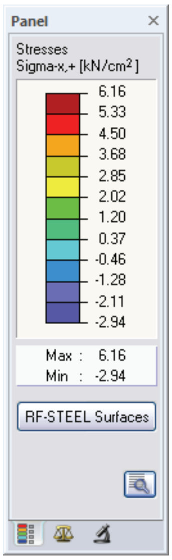


Figure 1.3: Panel button [RF-STEEL Surfaces]

2. RF-STEEL Surfaces

2.1 Input Data

The data for the definition of design cases is entered in tables.

When you have started RF-STEEL Surfaces, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently. The pull-down list above the navigator contains the design cases that are already available (see chapter 6.1, page 82).

If you open RF-STEEL Surfaces in an RFEM structure for the first time, the module imports the following design relevant data automatically:

- Surfaces
- Load cases, groups and combinations
- Materials
- Surface thicknesses
- Internal forces (in background, if calculated)

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

To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the module but without saving the data.

2.1.1 General Data

In table 1.1 *General Data*, you define the surfaces that you want to design. The two tabs below contain the actions that you can select for the ultimate and the serviceability limit state design.

RF-STEEL Surfaces is able to design both plane and curved surfaces.

The design standard is specified in table 1.2, even though indirectly, because the material properties are related to the standard.

Design of

The design is carried out for *Surfaces*, but the surface thickness in the add-on module is restricted to the thickness types 'Constant" and 'Variable'. If you want to design only selected surfaces, clear the *All* check box. Then you can access the input field to enter the numbers of the relevant surfaces. The list of the preset numbers can be selected quickly by double click and overwritten by entering the data manually.

To select the surfaces graphically in the RFEM work window, use the [Pick] button.

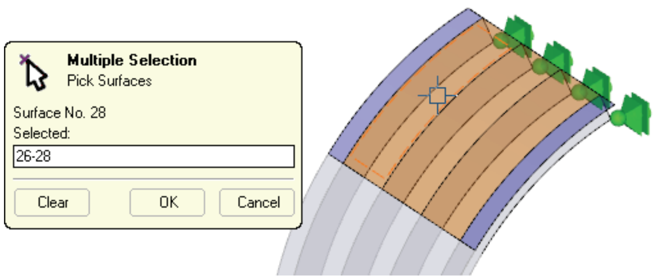


Figure 2.1: Graphical selection in the model

2.1.1.1 Ultimate Limit State

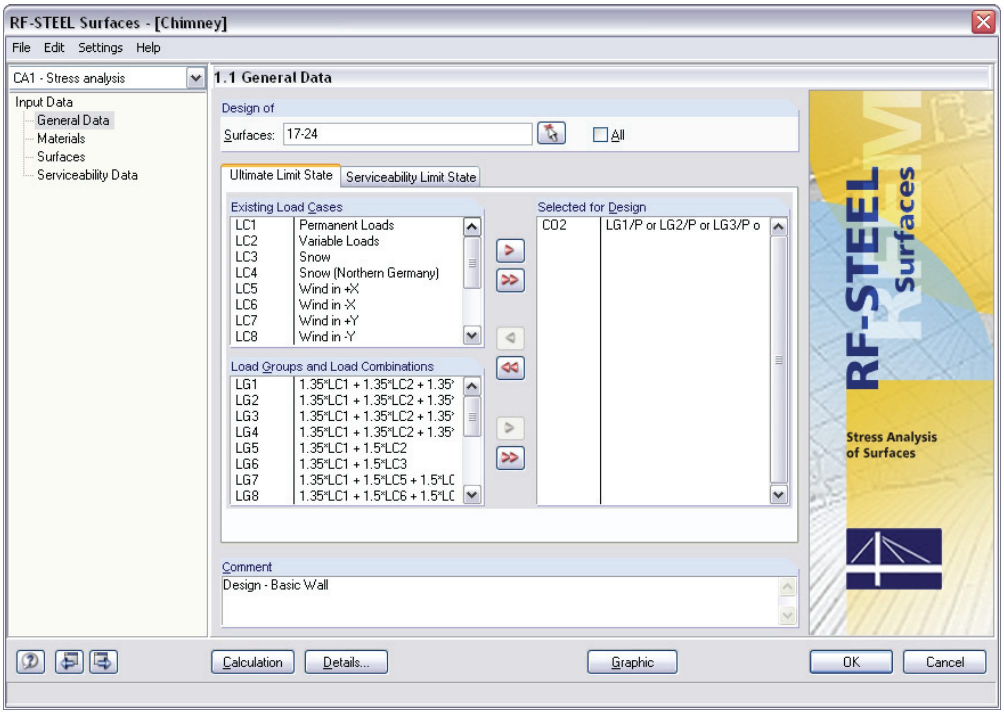


Figure 2.2: Table 1.1 *General Data*, tab *Ultimate Limit State*

Existing Load Cases / Load Groups and Load Combinations

These two sections list all load cases, load groups and load combinations defined in RFEM that are relevant for the ultimate limit state design. Use the button [▶] to transfer selected load cases or combinations to the list *Selected for Design* on the right. You can also double-click the entries. To transfer the complete list to the right, use the button [▶▶].

Selected for Design

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

The stress analysis of an enveloping *Or* load combination is often carried out more quickly than the global design of all created load cases and load groups. When you select load combinations, it is recommended to check the settings for *Analysis Method for Load Combinations* in the *Options* tab of the *Details* dialog box additionally (see chapter 2.2.1.3, page 26).

Comment

This input field allows for user-defined remarks, for example to describe the current design case.

2.1.1.2 Serviceability Limit State

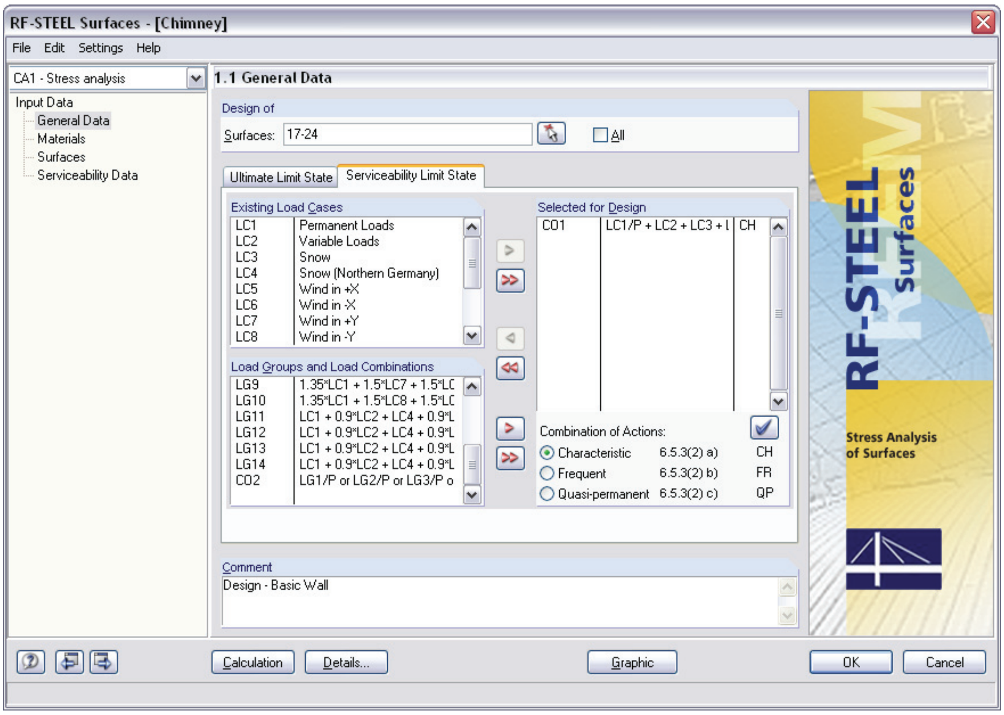
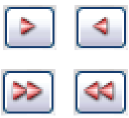


Figure 2.3: Table 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases / Load Groups and Load Combinations

In these two sections, all load cases, load groups and load combinations are listed that have been created in RFEM.

Selected for Design



Adding and removing load cases, load groups or load combinations is described in the previous chapter. The serviceability limit state design requires particular partial safety factors that can be considered in corresponding combinations of load cases and actions.

Combination of Actions

In the lower part of the *Selected for Design* section, you can assign limit values to the selected load cases, load groups and load combinations with regard to deflection: First, select an entry in the *Selected for Design* list, and then choose one of the three action combinations. To assign the action combination to the selected load case, click the button [☑].

The following action combinations are available:

- Characteristic (CH)
- Frequent (FR)
- Quasi-permanent (QP)

The deformation limit values are defined in the standards and can be modified for the different action combinations in the *Serviceability* tab of the *Details* dialog box (see chapter 2.2.1.2, page 25).

The reference lengths that are decisive for the serviceability limit state design are determined in table 1.4 *Serviceability Data* (see chapter 2.1.4, page 18).

2.1.2 Materials

The table is subdivided into two parts. In the upper part, the materials used for the design are listed with the 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.

The material properties required for the determination of internal forces in RFEM are described in detail in chapter 5.3 of the RFEM manual. The design relevant material properties are stored in the global material library and 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.

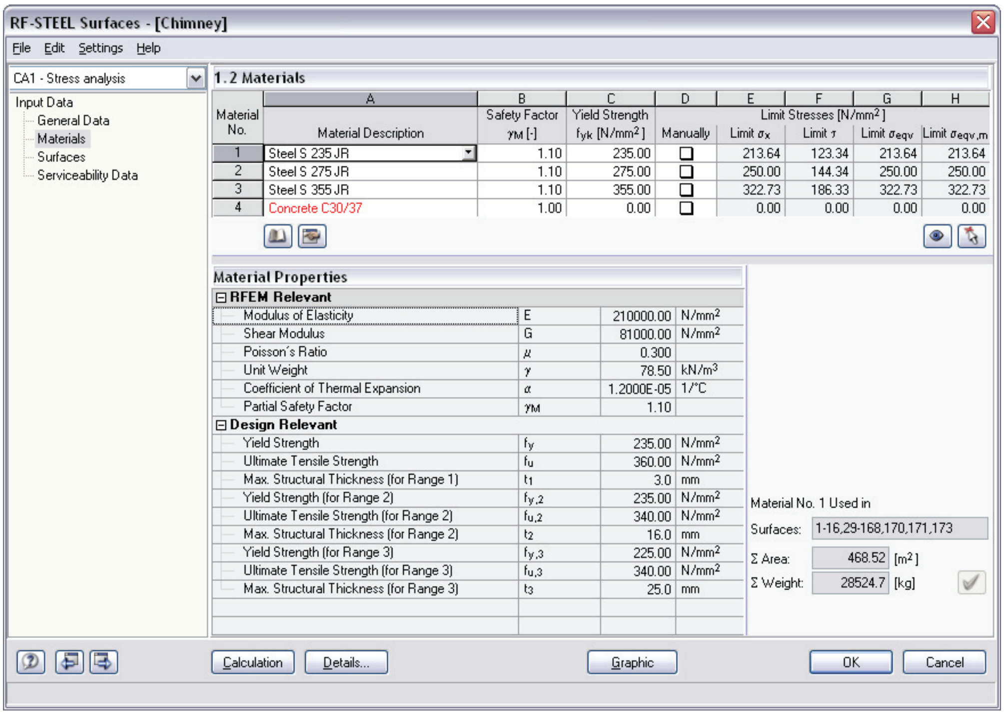
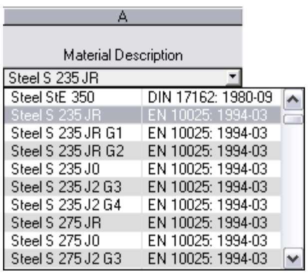


Figure 2.4: Table 1.2 *Materials*

Material Description



The materials defined in RFEM are already preset. But it is possible to select another 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]. The list shown on the left opens. When you have selected a new material, the design relevant properties are entered into the corresponding table row and are highlighted in blue.

Only steel materials are available in the list. 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. Therefore, it would be possible to design structural components made of aluminum or stainless steel. However, you must consider the corresponding standard specifications additionally.

If you have set a material whose limit stresses are not defined (for example glass), 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. When you have defined the allowable stresses in the columns E to G, the red color of the table row will disappear.

The import of materials from the library is described on page 14.

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 using the factor γ_M, the characteristic value of the yield strength f_{yk} is reduced for the determination of the limit stresses according to Equation 2.1 or Equation 2.2.

Thus, 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: On the one hand, for example according to DIN 18800 part 2, el. (116), you must consider the influence of deformations by a stiffness that is reduced about 10 % when determining internal forces. On the other hand, additionally, you must reduce the design values of 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 or EC 3, section 3.

Limit Stresses

The limit stresses for materials from the general material library are preset automatically. Those entries provide no access for modifications.

In case you want to modify the limit stresses, you can use the [Edit Material] button to open the *Edit Material* dialog box where you can change the material properties (see Figure 2.6, page 15). 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 subsequent columns manually. Materials that have been modified are highlighted in blue and marked by an asterisk in the column *Material Description*.

Limit σ_x

The limit normal stress as the allowable stress for stresses due to bending moments and membrane forces is determined, for example according to DIN 18800 part 1, el. (746), 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. For example according to DIN 18800 part 1, el. (746), the partial safety factor γ_M is also considered in the equation for the determination of 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.

Limit $\sigma_{eqv,m}$

The limit membrane equivalent stress describes the allowable equivalent stress due to membrane stresses. It is determined according to Equation 2.1.

Yield strength depending on thickness of structural components

For some materials, there is a relation between the characteristic yield strength f_{yk} and the thickness t of the relevant structural component. The *Max. Structural Thickness* of the respective ranges with the corresponding yield strength is indicated in the *Material Properties* section of table 1.2.

The yield strengths of structural thicknesses are defined in the standards, for example DIN 18800 part 1, table 1. To check and adjust, if necessary, the structural thicknesses and assigned stresses, use the [Edit Material] button (see Figure 2.6, page 15).

Material Library

A number of materials is stored in the library that you can access by using the button below column A. The following dialog box opens which you already know from RFEM.

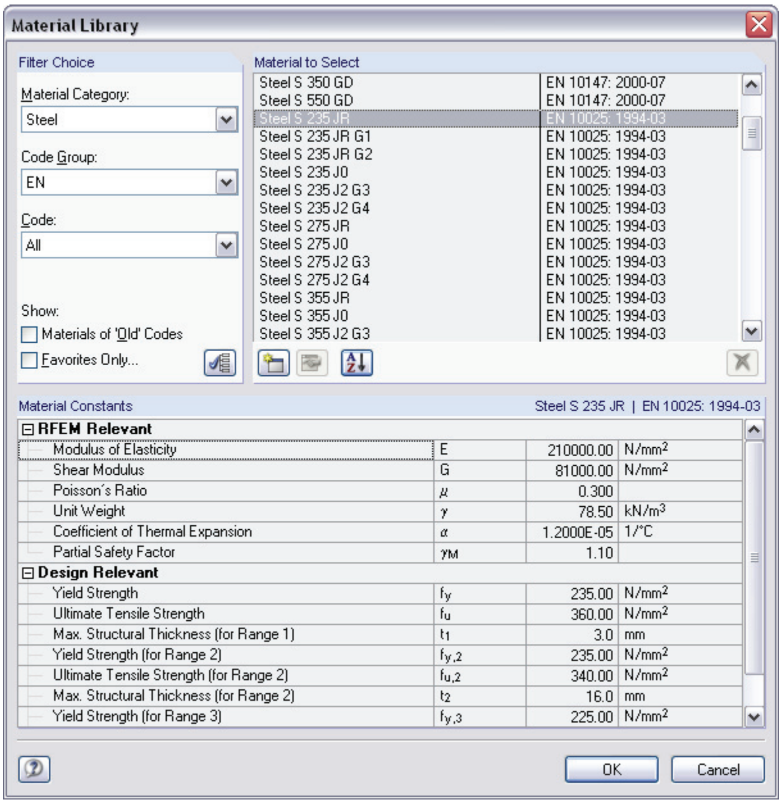


Figure 2.5: Dialog box *Material Library*

In the *Filter Choice* section, *Steel* is preset as material category. You can select a material from the list *Material to Select* on the right and check the corresponding parameters in the lower part of the dialog box. To transfer the material to table 1.2, click [OK], [↵] or double-click the material itself.

Chapter 5.3 in the RFEM manual describes in detail how materials can be filtered, added or rearranged.

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. Under these conditions, it would be possible to design, for example, structural components made of aluminum or stainless steel.

When you import a material whose limit stresses (for example glass) are not defined, the entries of the corresponding table row in table 1.2 are highlighted in red. It is possible, however, to define the limit stresses of this material by ticking the check box **Manually** in column D and entering user-defined specifications. The red color of the entries disappears as soon as you have entered the allowable stresses into column E to H. Please note that some stress designs, for example for glass surfaces, can only be carried out to some extent. In such a case, the use of the add-on module RF-GLASS is recommended.

Edit Material

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

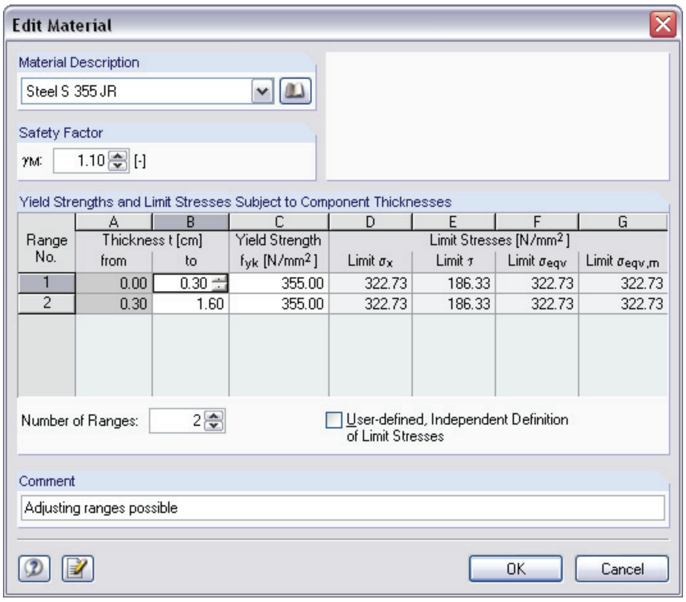
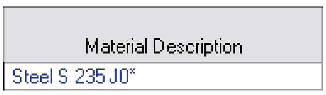


Figure 2.6: Dialog box *Edit Material*

The *Safety Factor* γ_M determines how the characteristic values of the yield strength f_{yk} indicated in column C are reduced. The determined limit stresses (cf. Equation 2.1 and Equation 2.2, page 13) are listed in column D to G.

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 in accordance with the standard, but it can be adjusted in the input field *Number of Ranges* below the table. In addition, you can modify the range limits by manual data input in column B. Column A will be adjusted automatically. For each range, you must assign a particular *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 G to enter user-defined data.



Materials that have been modified are marked by an asterisk in table 1.2.

2.1.3 Surfaces

In this table, the surfaces that should be designed are listed. The material numbers already assigned in RFEM are preset (see description of previous table 1.2 *Materials*). Additionally, it is possible to define optimization parameters.

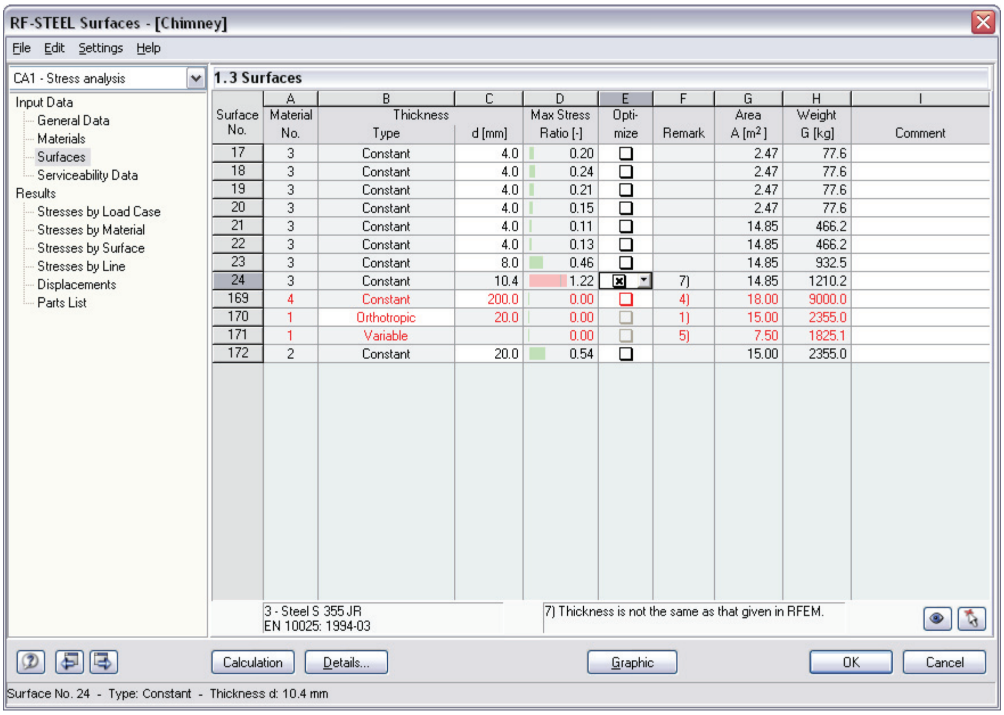


Figure 2.7: Table 1.3 *Surfaces*

Thickness

Type

When you open the table, the thickness types of the surfaces used in RFEM are preset together with the assigned material numbers.

The surface thickness in the add-on module is restricted to the RFEM thickness types *Constant*, *Variable* and *Membrane isotropic*. The design of other thickness types like orthotropic surfaces is currently not possible. But if you want to design such a surface, you can use the table 1.3 to change the thickness type to *Constant*: Place the pointer into the corresponding table row and click the button [▼] or use the function key [F7] to access the selection list. Then the design will be carried out with the RFEM internal forces as a surface with constant thickness and isotropic properties.

d

The thicknesses already defined in RFEM are preset but can be modified in column C. If you click in an input field, you can enter the new thickness manually or adjust it by using the spin buttons.

Please note that the internal forces won't be redetermined automatically after optimizing parameters or modifying surface thicknesses. As a result of modified thicknesses, internal forces may vary considerably because of the changed stiffnesses in the structural system. Therefore, after the first design, it is recommended to adjust the thicknesses also in RFEM. Then you can design the surfaces again with the RF-STEEL Surfaces add-on module.

Max Stress Ratio

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

Optimize

For each surface, you can carry out an optimization analysis. By using the RFEM internal forces, the program determines the surface thickness that comes as close as possible to the maximum stress ratio.

To optimize a particular surface, tick its corresponding check box in column D. The following dialog box appears.

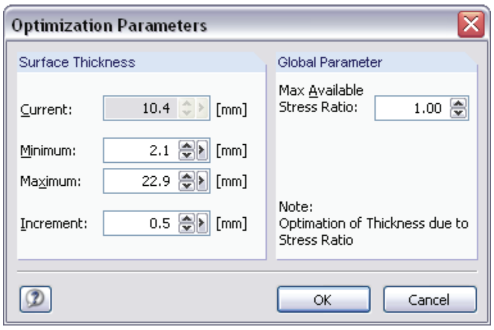


Figure 2.8: Dialog box *Optimization Parameters*

This dialog box is described in chapter 6.2.1 on page 84. There, you find further recommendations concerning the optimization process.

Remark

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

If the display indicates remark *1) Surface cannot be designed because it is of type* '*Orthotropic*', you can set the thickness type to 'Constant' to experiment, if required. Click in the corresponding input field of column B and use the button [▼] to open the selection list.

Area

This column gives information about the area of each surface.

Weight

This column displays the masses of the surfaces that you want to design.

Comment

The final column allows for user-defined comments, for example to document surface modifications.

2.1.4 Serviceability Data

The last input table provides setting options for the serviceability limit state design. It is only available if you have set the relevant entries in the *Serviceability Limit State* tab of table 1.1 (see chapter 2.1.1.2, page 11).

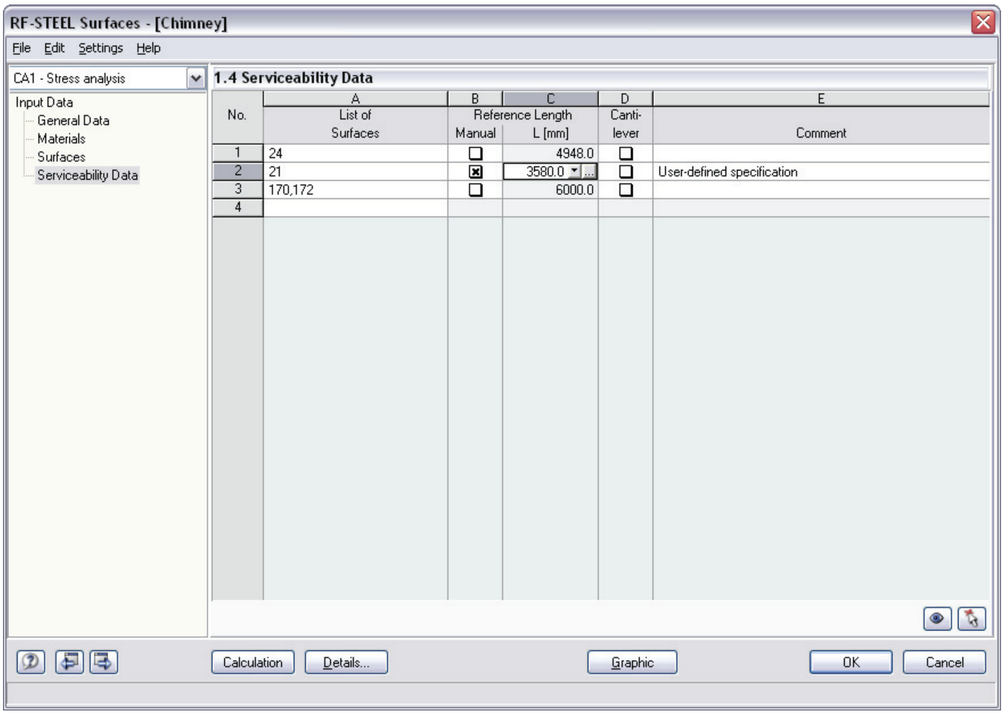


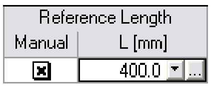
Figure 2.9: Table 1.4 *Serviceability Data*

List of Surfaces

In this column, you enter the numbers of the surfaces that you want to design. You can also use the [Pick] function to select the surfaces graphically in the RFEM work window. The respective reference lengths will be entered automatically in column C.

In case several surfaces are entered in one input field, they should have the same geometrical conditions. The reference length *L* is preset as the maximum length of all boundary lines included in these surfaces.

Reference Length



Column C contains the values of the longest boundary lines in the respective surfaces. Here, the program refers to the lengths of single lines, continuous lines are not considered.

To adjust a reference length, tick the *Manual* check box. Then you can access the input field for *L* where you can enter the value manually. You can also click the button [▼] to select it from the list or use the [...] function to determine it graphically in the RFEM work window. Manual corrections may be required, for example for surfaces that are placed within other surfaces.

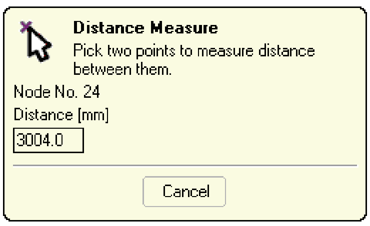


Figure 2.10: Dialog box to determine boundary points graphically

Cantilever

To apply limit deformations correctly, it is also important to know if the surface is supported on all sides or if it is a cantilevered surface.

Details...

In column D, you can determine the surface to be a cantilever in order to apply other serviceability limit values. They can be checked and, if necessary, also adjusted in the *Serviceability* tab of the *Details* dialog box (see Figure 2.12, page 25).

2.2 Calculation

Calculation

Details...

The stress analysis is carried out by using the internal forces determined in RFEM. Before you start the calculation by clicking the [Calculation] button, it is recommended to check the design details. To open the corresponding dialog box, use the [Details] button. Details on this dialog box can be found in chapter 2.2.1.2 on page 25.

2.2.1 Calculation Details

2.2.1.1 Stresses

Tables 2.1 to 2.5 display the following stresses by default:

- Shear stresses τ_{max}
- Principal stresses σ in direction of principal axes on the surface's top and bottom side
- Membrane stresses σ_m in direction of the principal axes
- Equivalent stress σ_{eqv}
- Membrane equivalent stress $\sigma_{eqv,m}$

Details...

The display of stresses and stress components can be adjusted in the *Stresses* tab of the *Details* dialog box.

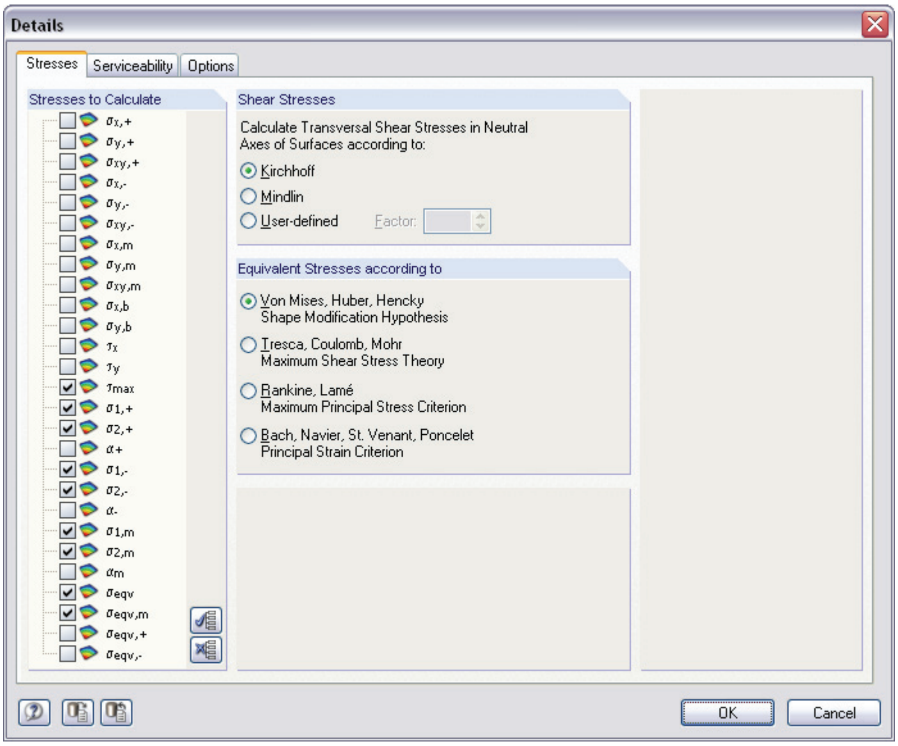


Figure 2.11: Dialog box *Details*, tab *Stresses*

The stresses listed in the dialog section *Stresses to Calculate* are described in detail in the following table.

$\sigma_{x,+}$	Stress in direction of the local x-axis on the positive side of the surface (that means the side in direction of the positive surface axis z) $\sigma_{x,+} = \frac{n_x}{d} + \frac{6 \cdot m_x}{d^2}$ with d: thickness of surface
$\sigma_{y,+}$	Stress in direction of the local y-axis on the positive side of the surface $\sigma_{y,+} = \frac{n_y}{d} + \frac{6 \cdot m_y}{d^2}$
$\sigma_{xy,+}$	Torsional stress on the positive side of the surface $\sigma_{xy,+} = \frac{n_{xy}}{d} + \frac{6 \cdot m_{xy}}{d^2}$
$\sigma_{x,-}$	Stress in direction of the x-axis on the negative side of the surface $\sigma_{x,-} = \frac{n_x}{d} - \frac{6 \cdot m_x}{d^2}$
$\sigma_{y,-}$	Stress in direction of the y-axis on the negative side of the surface $\sigma_{y,-} = \frac{n_y}{d} - \frac{6 \cdot m_y}{d^2}$
$\sigma_{xy,-}$	Torsional stress on the negative side of the surface $\sigma_{xy,-} = \frac{n_{xy}}{d} - \frac{6 \cdot m_{xy}}{d^2}$
$\sigma_{x,m}$	Membrane stress due to axial force n_x $\sigma_{x,m} = \frac{n_x}{d}$
$\sigma_{y,m}$	Membrane stress due to axial force n_y $\sigma_{y,m} = \frac{n_y}{d}$
$\sigma_{xy,m}$	Membrane stress due to shear flow n_{xy} $\sigma_{xy,m} = \frac{n_{xy}}{d}$
$\sigma_{x,b}$	Stress due to bending moment m_x $\sigma_{x,b} = \frac{6 \cdot m_x}{d^2}$
$\sigma_{y,b}$	Stress due to bending moment m_y $\sigma_{y,b} = \frac{6 \cdot m_y}{d^2}$
τ_x	Shear stress orthogonal to the surface in direction of the x-axis $\frac{3 \cdot v_x}{2 \cdot d}$
τ_y	Shear stress orthogonal to the surface in direction of the y-axis $\frac{3 \cdot v_y}{2 \cdot d}$
τ_{max}	Maximum shear stress perpendicular to the surface $\tau_{max} = \sqrt{\tau_x^2 + \tau_y^2}$

$\sigma_{1,+}$	Stress in direction of the principal axis 1 on the positive side of the surface (that means the side in direction of the positive surface axis z) $\sigma_{1,+} = \frac{1}{2}\left(\sigma_{x,+} + \sigma_{y,+} + \sqrt{(\sigma_{x,+} - \sigma_{y,+})^2 + 4 \cdot \sigma_{xy,+}{}^2}\right)$
$\sigma_{2,+}$	Stress in direction of the principal axis 2 on the positive side of the surface $\sigma_{2,+} = \frac{1}{2}\left(\sigma_{x,+} + \sigma_{y,+} - \sqrt{(\sigma_{x,+} - \sigma_{y,+})^2 + 4 \cdot \sigma_{xy,+}{}^2}\right)$
α_{+}	Angle between local axis x (or y) and principal axis 1 (or 2) for stresses on the positive side of the surface $\alpha_{+} = \frac{1}{2}\left(\arctan\left(\frac{2 \cdot \sigma_{xy,+}}{\sigma_{x,+} - \sigma_{y,+}}\right)\right)$
$\sigma_{1,-}$	Stress in direction of the principal axis 1 on the negative side of the surface $\sigma_{1,-} = \frac{1}{2}\left(\sigma_{x,-} + \sigma_{y,-} + \sqrt{(\sigma_{x,-} - \sigma_{y,-})^2 + 4 \cdot \sigma_{xy,-}{}^2}\right)$
$\sigma_{2,-}$	Stress in direction of the principal axis 2 on the negative side of the surface $\sigma_{2,-} = \frac{1}{2}\left(\sigma_{x,-} + \sigma_{y,-} - \sqrt{(\sigma_{x,-} - \sigma_{y,-})^2 + 4 \cdot \sigma_{xy,-}{}^2}\right)$
α_{-}	Angle between local axis x (or y) and principal axis 1 (or 2) for stresses on the negative side of the surface $\alpha_{-} = \frac{1}{2}\left(\arctan\left(\frac{2 \cdot \sigma_{xy,+}}{\sigma_{x,-} - \sigma_{y,-}}\right)\right)$
$\sigma_{1,m}$	Membrane stress due to axial force n_1 $\sigma_{1,m} = \frac{n_1}{d}$ with d: thickness of surface
$\sigma_{2,m}$	Membrane stress due to axial force n_2 $\sigma_{2,m} = \frac{n_2}{d}$
α_{m}	Angle between axis x and principal axis 1 (for axial force n_1) $\frac{1}{2}\left[\arctan\left(\frac{2 \cdot n_{xy}}{n_x - n_y}\right)\right]$
σ_{eqv}	Maximum equivalent stress as maximum of $\sigma_{eqv,+}$ and $\sigma_{eqv,-}$ (see below)
$\sigma_{eqv,+}$	Equivalent stress on the positive side of the surface (that means the side in direction of the positive surface axis z) according to selected stress hypothesis (see Table 2.2 to Table 2.5)
$\sigma_{eqv,-}$	Equivalent stress on the positive or negative side of the surface according to selected stress hypothesis (see Table 2.2 to Table 2.5)
$\sigma_{eqv,m}$	Membrane equivalent stress according to selected stress hypothesis (see Table 2.2 to Table 2.5)

Table 2.1: Stresses

Shear Stresses

To determine the *Transversal Shear Stresses* in the surfaces' neutral axes, select one of the following three methods (see Figure 2.11, page 19).

Kirchhoff

$$\tau_x = 1{,}5 \cdot \frac{v_x}{d}$$

$$\tau_y = 1{,}5 \cdot \frac{v_y}{d}$$

Equation 2.3

Mindlin

$$\tau_x = 1{,}0 \cdot \frac{v_x}{d}$$

$$\tau_y = 1{,}0 \cdot \frac{v_y}{d}$$

Equation 2.4

User-defined

You can specify a *Factor* that is used by the program to determine the shear stresses τ_x and τ_y in terms of Equation 2.3 or Equation 2.4.

Equivalent Stresses

The equivalent stresses from the individual stress components can be determined according to four different approaches (see Figure 2.11, page 19).

Von Mises, Huber, Hencky

This stress hypothesis is also known as *Shape Modification Hypothesis* or "equivalent stress according to VON MISES". It is assumed that the material fails as soon as the shape modifying energy exceeds a certain limit. This shape modification energy represents the kind of energy that causes distortion or deformation of the element.

The equivalent stress according to the shape modification hypothesis is the most well-known and frequently used equivalent stress hypothesis. It is the appropriate method for all materials that are not brittle. Therefore, it is widely used in steel building construction.

However, the hypothesis is not applicable for hydrostatic stress conditions with similar principal stresses in all directions, as the equivalent stress is zero in such cases.

The equivalent stresses according to VON MISES for the plane stress conditions have the following meanings:

$\sigma_{eqv,+}$	Equivalent stress on the positive side of the surface (that means the side in direction of the positive surface axis z) $\sigma_{eqv,+} = \sqrt{\sigma_{x,+}^2 + \sigma_{y,+}^2 - \sigma_{x,+} \cdot \sigma_{y,+} + 3 \cdot \sigma_{xy,+}^2}$
$\sigma_{eqv,-}$	Equivalent stress on the negative side of the surface $\sigma_{eqv,-} = \sqrt{\sigma_{x,-}^2 + \sigma_{y,-}^2 - \sigma_{x,-} \cdot \sigma_{y,-} + 3 \cdot \sigma_{xy,-}^2}$

$\sigma_{eqv,m}$	Membrane equivalent stress as the maximum absolute value of $\sigma_{eqv,m} = \frac{\sigma_{x,m} + \sigma_{y,m}}{2} + \sqrt{\left(\frac{\sigma_{x,m} - \sigma_{y,m}}{2}\right)^2 + \sigma_{xy,m}{}^2}$ or $\sigma_{eqv,m} = \frac{\sigma_{x,m} + \sigma_{y,m}}{2} - \sqrt{\left(\frac{\sigma_{x,m} - \sigma_{y,m}}{2}\right)^2 + \sigma_{xy,m}{}^2}$ or $\sigma_{eqv,m} = \sqrt{(\sigma_{x,m} - \sigma_{y,m})^2 + 4 \cdot \sigma_{xy,m}{}^2}$ where $\sigma_{x,m} = \frac{n_x}{d}$ $\sigma_{y,m} = \frac{n_y}{d}$ with d: thickness of surface $\sigma_{xy,m} = \frac{n_{xy}}{d}$

Table 2.2: Equivalent stresses according to VON MISES, HUBER, HENCKY

Tresca, Coulomb, Mohr

In the *Maximum Shear Stress Theory*, which is also known as "equivalent stress according to TRESCA", it is assumed that failure is caused by the maximum shear stress.

As this hypothesis is especially applicable for brittle materials, it is frequently used in mechanical engineering.

These equivalent stresses are determined according to the following equations:

$\sigma_{eqv,+}$	Equivalent stress on the positive side of the surface (that means the side in direction of the positive surface axis z) $\sigma_{eqv,+} = \sqrt{(\sigma_{x,+} - \sigma_{y,+})^2 + 4 \cdot \sigma_{xy,+}{}^2}$
$\sigma_{eqv,-}$	Equivalent stress on the negative side of the surface $\sigma_{eqv,-} = \sqrt{(\sigma_{x,-} - \sigma_{y,-})^2 + 4 \cdot \sigma_{xy,-}{}^2}$
$\sigma_{eqv,m}$	Membrane equivalent stress $\sigma_{eqv,m} = \sqrt{(\sigma_{x,m} - \sigma_{y,m})^2 + 4 \cdot \sigma_{xy,m}{}^2}$

Table 2.3: Equivalent stresses according to TRESCA, COULOMB, MOHR

Rankine, Lamé

This equivalent stress hypothesis is also known as *Maximum Principal Stress Criterion* or as "equivalent stress according to RANKINE". It is assumed that failure is caused by the maximum principal stress.

The equivalent stresses are determined according to the following equations:

$\sigma_{eqv,+}$	Maximum absolute value of equivalent stress on the positive side of the surface $\sigma_{eqv,+} = \frac{1}{2}\left(\sigma_{x,+} + \sigma_{y,+}\right) \pm \frac{1}{2}\sqrt{\left(\sigma_{x,+} - \sigma_{y,+}\right)^2 + 4 \cdot \sigma_{xy,+}{}^2}$
$\sigma_{eqv,-}$	Maximum absolute value of equivalent stress on the negative side of the surface $\sigma_{eqv,-} = \frac{1}{2}\left(\sigma_{x,-} + \sigma_{y,-}\right) \pm \frac{1}{2}\sqrt{\left(\sigma_{x,-} - \sigma_{y,-}\right)^2 + 4 \cdot \sigma_{xy,-}{}^2}$
$\sigma_{eqv,m}$	Maximum absolute value of membrane equivalent stress $\sigma_{eqv,m} = \frac{1}{2}\left(\sigma_{x,m} + \sigma_{y,m}\right) \pm \frac{1}{2}\sqrt{\left(\sigma_{x,m} - \sigma_{y,m}\right)^2 + 4 \cdot \sigma_{xy,m}{}^2}$

Table 2.4: Equivalent stresses according to RANKINE, LAMÉ

Bach, Navier, St. Venant, Pocelet

For the *Principal Strain Criterion*, also known as the "equivalent stress according to BACH", it is assumed that the failure occurs in direction of the maximum strain. This approach is similar to the stress determination according to RANKINE described above. Here, the principal strain is used instead of the principal stress.

These equivalent stresses are determined as follows:

$\sigma_{eqv,+}$	Maximum absolute value of equivalent stress on the positive side of the surface $\sigma_{eqv,+} = \frac{1-\mu}{2}\left(\sigma_{x,+} + \sigma_{y,+}\right) \pm \frac{1+\mu}{2}\sqrt{\left(\sigma_{x,+} - \sigma_{y,+}\right)^2 + 4 \cdot \sigma_{xy,+}{}^2}$ with μ : Poisson's ratio of material
$\sigma_{eqv,-}$	Maximum absolute value of equivalent stress on the negative side of the surface $\sigma_{eqv,-} = \frac{1-\mu}{2}\left(\sigma_{x,-} + \sigma_{y,-}\right) \pm \frac{1+\mu}{2}\sqrt{\left(\sigma_{x,-} - \sigma_{y,-}\right)^2 + 4 \cdot \sigma_{xy,-}{}^2}$
$\sigma_{eqv,m}$	Maximum absolute value of membrane equivalent stress $\sigma_{eqv,m} = \frac{1-\mu}{2}\left(\sigma_{x,m} + \sigma_{y,m}\right) \pm \frac{1+\mu}{2}\sqrt{\left(\sigma_{x,m} - \sigma_{y,m}\right)^2 + 4 \cdot \sigma_{xy,m}{}^2}$

Table 2.5: Equivalent stresses according to BACH, NAVIER, ST. VENANT, POCELET

2.2.1.2 Serviceability

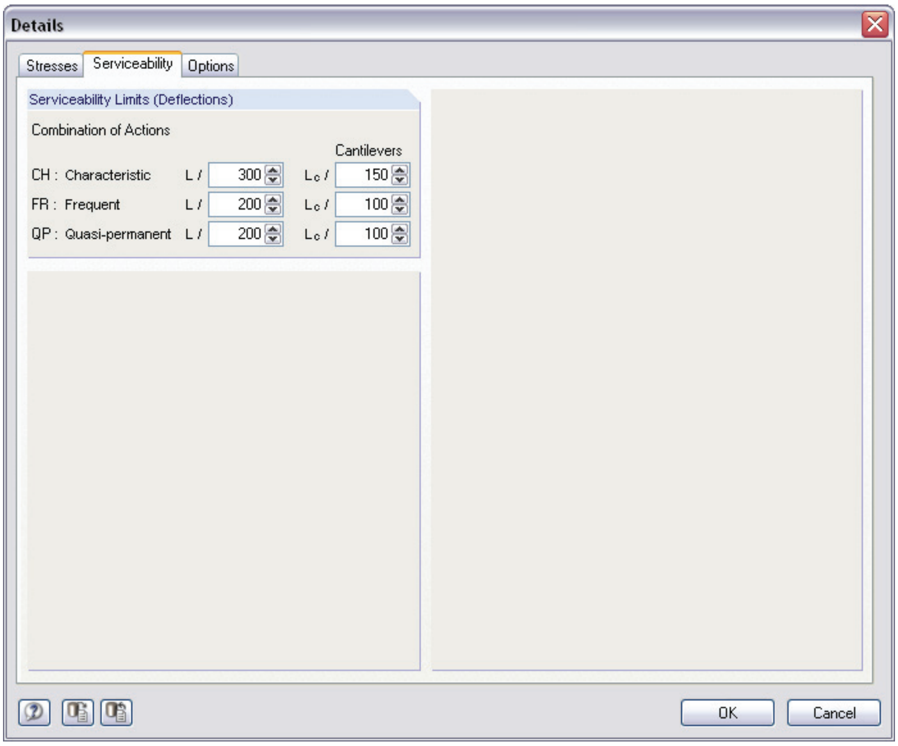


Figure 2.12: Dialog box *Details*, tab *Serviceability*

The six input fields are used to manage the *Serviceability Limits* of the allowable deflections. It is possible to enter specific settings for the different action combinations (Characteristic, Frequent, Quasi-permanent) as well as for both- and one-sided supported surfaces.

The classification of action combinations is determined in the *Serviceability Limit State* tab of table 1.1 *General Data* (see chapter 2.1.1.2, page 11). The reference lengths *L* are defined for each surface in table 1.4 *Serviceability Data* (see chapter 2.1.4, page 18).

2.2.1.3 Options

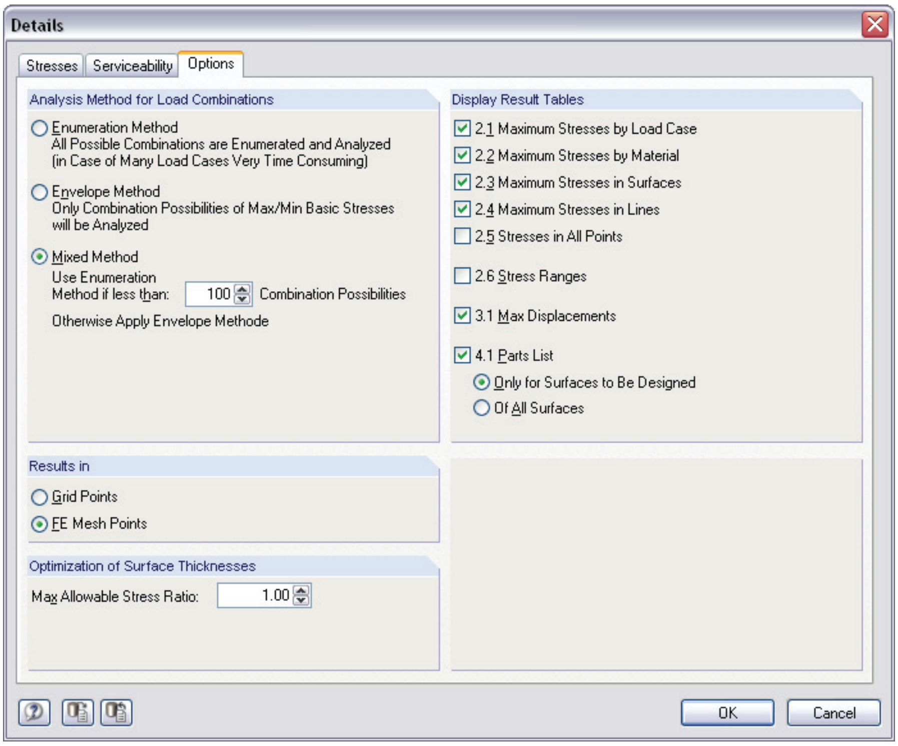


Figure 2.13: Dialog box *Details*, tab *Options*

Analysis Method for Load Combinations

To some extent it is possible to influence the method of stress determination for the design of load combinations. The *Mixed Method* is preset by default: Before the design is carried out, the program checks if the *Enumeration Method* is more effective than the *Envelope Method* or vice versa.

Enumeration Method

If this method is selected, the module evaluates the internal forces from RFEM line by line to superimpose them appropriately. This is the approach that comprehends all combination possibilities accurately.

The disadvantage of the enumeration method is that the number of combinations for the analysis is growing exponentially with the number of load cases when processing the data line by line. The relation is the following:

$$\text{Number of combination possibilities} = 2^n$$

with n = number of load cases

The more load cases the combination contains, the more time is required for the stress analysis. In the results, however, all possible combinations are included.

Envelope Method

If this method is selected, the module considers only the extreme values of the basic stresses of each load case to combine them subsequently. In this way, this approach may possibly not comprehend the most unfavorable combinations that would be included in a line by line process. However, the computing time for a relatively large number of load cases in a load combination is quite acceptable when this kind of method is used.

As only the maximum values are analyzed, the stresses designed according to this method may be incorrect. In particular structures with load cases whose effective directions tend to be orthogonal must be handled carefully. In such cases, a calculation according to the enumeration method is recommended to check the results.

Mixed Method

If this method is selected, the module checks how many combination possibilities exist on the basis of the load cases contained within (cf. enumeration method) before the actual design will be carried out. For example, if a load combination contains seven load cases,

$$2^7 = 128$$

combination possibilities must be analyzed. As this number is higher than the preset number of 100 possibilities, the subsequent design will be carried out according to the envelope method.

The choice of the analysis method can be influenced by means of the input field. This field defines the upper limit of the combination possibilities for the design according to the accurate enumeration method.

Thus, the *Mixed Method* represents a compromise between result accuracy and design velocity, and therefore it becomes the appropriate method for most of the application cases.

Results

By default, stresses and displacements are displayed in all *FE Mesh Points*. As an alternative, the results output is available in the user-defined *Grid Points* managed as surface properties in RFEM (see RFEM manual, chapter 9.9).

Especially for small surfaces, the default mesh size of 500 mm for the results grid may produce only few grid points or even only one grid point in the grid origin. In these cases, the maximum values are often not considered in the output tables because the results grid is too coarse. The spacing of grid points in RFEM should be adjusted to the surfaces' dimensions in order to generate more grid points.

Optimization of Surface Thicknesses

If the optimization process does not gear towards the maximum possible stress ratio of 100 %, you can specify a different limit value in this input field.

Display Result Tables

This dialog section controls the display of the results tables. You can select several output tables for stresses, displacements and parts lists which can be switched on and off by ticking the corresponding check boxes.

The results tables are described in detail in chapter 2.3.

2.2.2 Start Calculation

You can start the calculation out of each of the three or four input tables by clicking the [Calculation] button.

RF-STEEL Surfaces 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 RFEM calculation to determine the design relevant internal forces. In this determination process, the calculation parameters preset in RFEM are applied.

You can also start the calculation for RF-STEEL Surfaces out of the RFEM user interface. The add-on modules are listed in the dialog box *To Calculate* like load cases or load groups. To open the dialog box in RFEM,

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

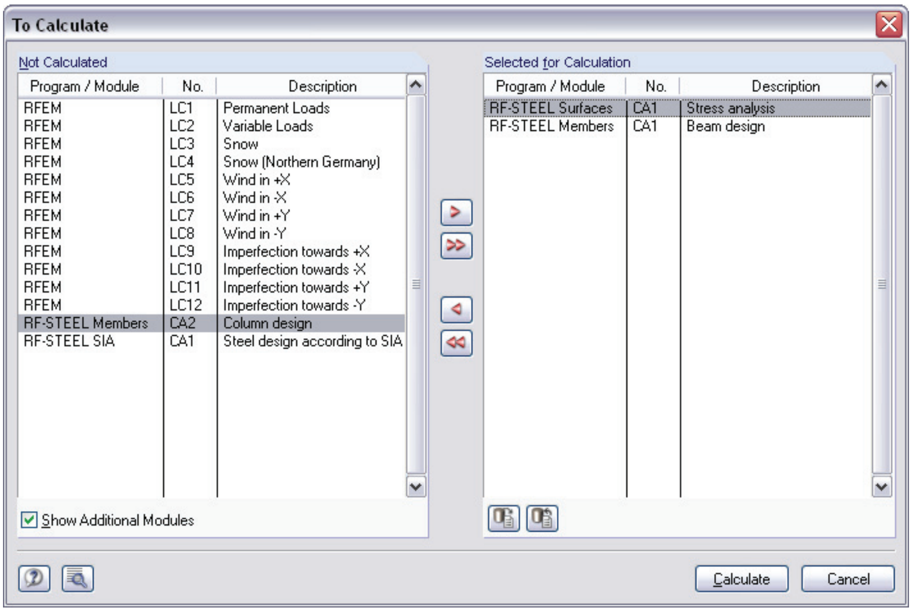
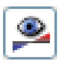


Figure 2.14: RFEM dialog box *To Calculate*

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



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

To calculate an RF-STEEL case directly, use the list in the RFEM toolbar. Select the relevant design case in the toolbar list and click the button [Results on/off].

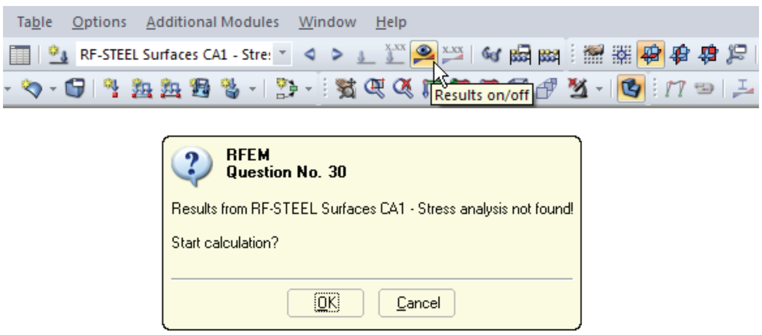


Figure 2.15: Direct calculation of an RF-STEEL design case in RFEM

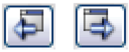
2.3 Results

Table 2.1 *Stresses by Load Case* is displayed immediately after the calculation. All stresses are shown in the results tables 2.1 to 2.5, sorted by different criteria.

Table 2.6 *Stress Ranges* lists the maximum differences between the stresses (for example for fatigue designs). Table 3.1 *Displacements* contains the maximum deformations in relation to the allowable limit values.

The final table 4.1 *Parts List* offers an overview about the required amount of steel.

In the *Options* tab of the *Details* dialog box, you can select the results tables that you want to display in particular (see Figure 2.13, page 26).



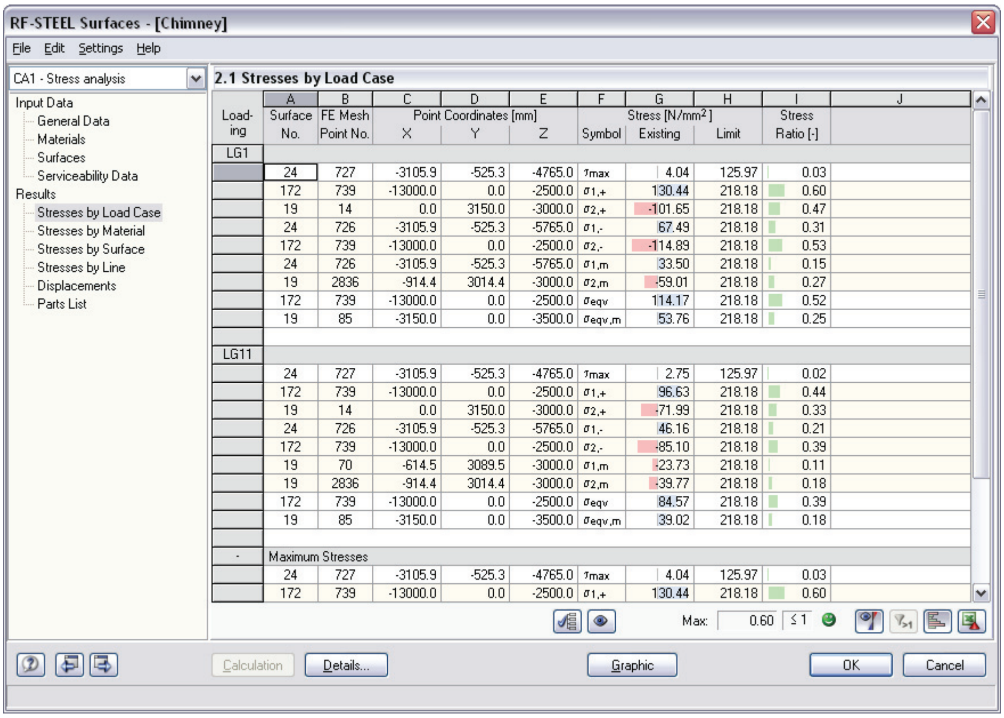
To access the results tables, use the RF-STEEL navigator. You can also use the two buttons shown on the left or the function keys [F2] and [F3] to select the previous or subsequent table.

By default, the result values are displayed in the FE mesh points. As an alternative, the results output is available in the user-defined grid points. The corresponding setting can be defined in the *Options* tab of the *Details* dialog box.

Click [OK] to save the results and quit the add-on module RF-STEEL Surfaces.

In the following, the different results tables are described in sequence. Evaluating and checking results is described in chapter 4 *Results Evaluation*.

2.3.1 Stresses by Load Case



Loading	A Surface No.	B FE Mesh Point No.	C Point Coordinates [mm] X	D Y	E Z	F Symbol	G Stress [N/mm²] Existing	H Limit	I Stress Ratio [-]
LG1									
	24	727	-3105.9	-525.3	-4765.0	τ_{max}	4.04	125.97	0.03
	172	739	-13000.0	0.0	-2500.0	$\sigma_{1,+}$	130.44	218.18	0.60
	19	14	0.0	3150.0	-3000.0	$\sigma_{2,+}$	-101.65	218.18	0.47
	24	726	-3105.9	-525.3	-5765.0	$\sigma_{1,-}$	67.49	218.18	0.31
	172	739	-13000.0	0.0	-2500.0	$\sigma_{2,-}$	-114.89	218.18	0.53
	24	726	-3105.9	-525.3	-5765.0	$\sigma_{1,m}$	33.50	218.18	0.15
	19	2836	-914.4	3014.4	-3000.0	$\sigma_{2,m}$	-59.01	218.18	0.27
	172	739	-13000.0	0.0	-2500.0	σ_{eqv}	114.17	218.18	0.52
	19	85	-3150.0	0.0	-3500.0	$\sigma_{eqv,m}$	53.76	218.18	0.25
LG11									
	24	727	-3105.9	-525.3	-4765.0	τ_{max}	2.75	125.97	0.02
	172	739	-13000.0	0.0	-2500.0	$\sigma_{1,+}$	96.63	218.18	0.44
	19	14	0.0	3150.0	-3000.0	$\sigma_{2,+}$	-71.99	218.18	0.33
	24	726	-3105.9	-525.3	-5765.0	$\sigma_{1,-}$	46.16	218.18	0.21
	172	739	-13000.0	0.0	-2500.0	$\sigma_{2,-}$	-85.10	218.18	0.39
	19	70	-614.5	3089.5	-3000.0	$\sigma_{1,m}$	-23.73	218.18	0.11
	19	2836	-914.4	3014.4	-3000.0	$\sigma_{2,m}$	-39.77	218.18	0.18
	172	739	-13000.0	0.0	-2500.0	σ_{eqv}	84.57	218.18	0.39
	19	85	-3150.0	0.0	-3500.0	$\sigma_{eqv,m}$	39.02	218.18	0.18
-	Maximum Stresses								
	24	727	-3105.9	-525.3	-4765.0	τ_{max}	4.04	125.97	0.03
	172	739	-13000.0	0.0	-2500.0	$\sigma_{1,+}$	130.44	218.18	0.60

Figure 2.16: Table 2.1 *Stresses by Load Case*

The table shows a summary of results for each load case, load group and/or load combination that has been selected for design in the *Ultimate Limit State* tab of table 1.1 *General Data*. The relevant load cases, load groups and/or load combinations are indicated in the first column, specifying the arrangement of the results output.

Surface No.

The column lists the numbers of surfaces for which maximum stress components or ratios have been determined. The output is displayed by load cases.

FE Mesh Point No. / Grid Point No.

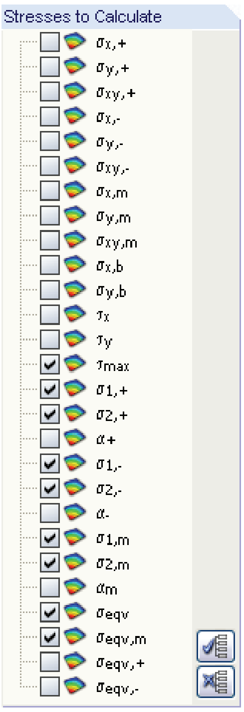
The column displays the numbers of grid or FE mesh points in which the ratios' maximum values occur. The grid points represent an option to display the results in user-defined, regular spacings, completely independent from the FE mesh. The number of grid points and their arrangement is managed in the *Grid* tab of the RFEM dialog box *Edit Surface*.

By selection in the *Options* tab of the *Details* dialog box, you can decide if you want to consider either FE mesh nodes or user-defined grid points for the evaluation (cf. Figure 2.13, page 26). When data has been modified, a query appears before the results will be recalculated.

Point Coordinates X/Y/Z

The three columns contain the coordinates of the governing FE mesh or grid points. They refer to the global coordinate system XYZ.

Stress - Symbol



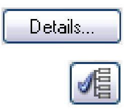
The following stresses are displayed by default:

- shear stresses τ_{max}
- principal stresses σ in direction of the principal axes on the surface's top and bottom side
- membrane stresses σ_m in direction of the principal axes
- equivalent stress σ_{eqv}
- membrane equivalent stress $\sigma_{eqv,m}$

Each stress type or component has its own symbol. The indices of the normal stresses σ, the shear stresses τ and the equivalent stresses σ_{eqv} have the following meanings:

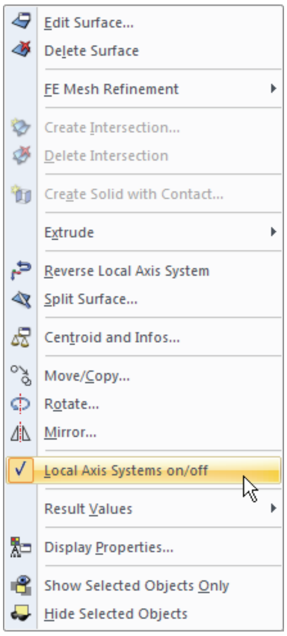
Symbol	Meaning
x	Direction of the local surface axis x
y	Direction of the local surface axis y
1	Direction of the principal axis 1
2	Direction of the principal axis 2
+	Positive surface side (side in direction of the positive local surface axis z)
–	Negative surface side (side in the opposite direction of the positive surface axis z)
m	Stress due to membrane force (axial force)
b	Stress due to bending moment

Table 2.6: Symbols of stresses



The display of stresses and stress components can be adjusted in the *Stresses* tab of the *Details* dialog box (see Figure 2.11, page 19). To open the dialog box, use the button shown on the left (at the bottom of the list below column E).

The definition of positive and negative surface sides corresponds to the conventions given in RFEM: The positive surface side is always defined in direction of the positive local axis z of each surface, regardless of the orientation of the global axis Z.



Context menu of surface

To display the local surface axes, use the *Display* navigator in RFEM or the context menu of the corresponding surface.

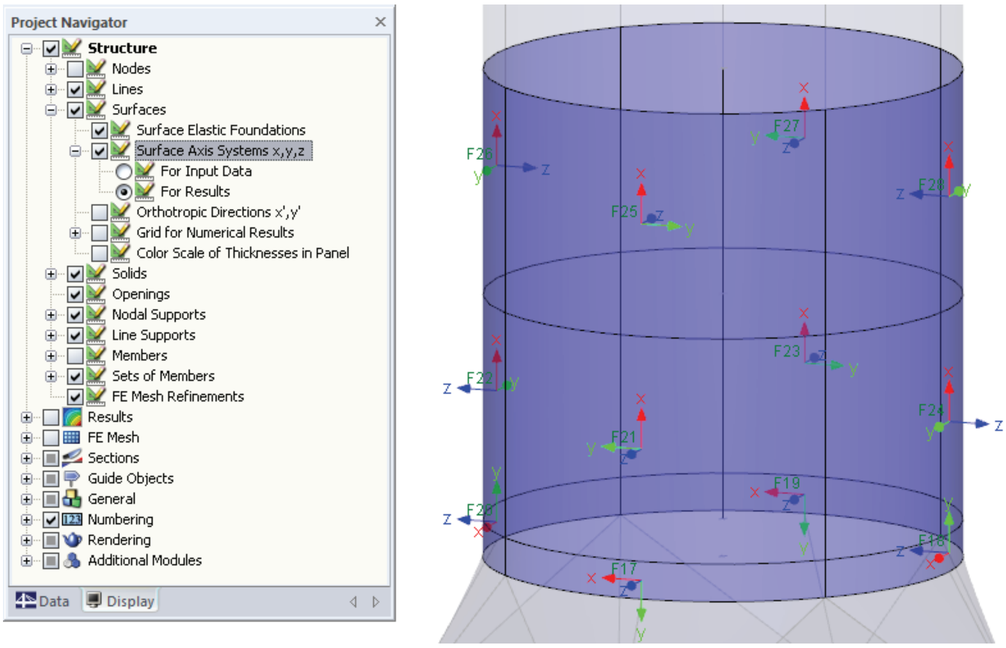
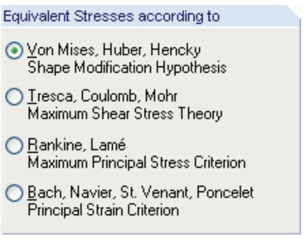


Figure 2.17: Activating the surface axis systems in the *Display* navigator of RFEM

Stress - Existing

This column displays the extreme values of the existing stresses analyzed according to the equations described in Table 2.1 (see page 20).

For each stress type, the program calculates the maximum (positive) and minimum (negative) stress values to compare their results subsequently. The value that is higher than the other one will appear in column G.



The equivalent stresses σ_{eqv} are determined according to the equivalent stress hypothesis specified in the *Stresses* tab of the *Details* dialog box (see Figure 2.11, page 19). Four approaches, that are described in detail in chapter 2.2.1.1 page 22, are available for selection:

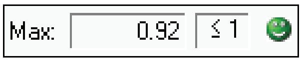
- shape modification hypothesis according to VON MISES, HUBER, HENCKY
- maximum shear stress theory according to TRESCA, COULOMB, MOHR
- maximum principal stress criterion according to RANKINE, LAMÉ
- principal strain criterion according to BACH, NAVIER, ST. VENANT, PONCELET

Stress - Limit

This column shows the limit stresses of table 1.2 (see chapter 2.1.2, page 13). In particular, they are the following:

- Limit normal stress σ as the allowable stress for actions due to bending moments and membrane forces
- 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 several stresses
- Limit membrane equivalent stress $\sigma_{eqv,m}$ as the allowable equivalent stress due to membrane stresses

Stress Ratio



For each stress component, RF-STEEL Surfaces determines the quotient from the existing and the limit stress, as shown for example in DIN 18800 part 1, el. (747). The stress ratio of the surface on the respective FE mesh or grid point is displayed for every selected stress type. If the relevant limit stress is not exceeded, the ratio is less than or equal to 1 and the stress design was carried out successfully.

Thus, column I allows for a quick evaluation of efficiency.

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

Equation 2.5: Design condition for normal stresses

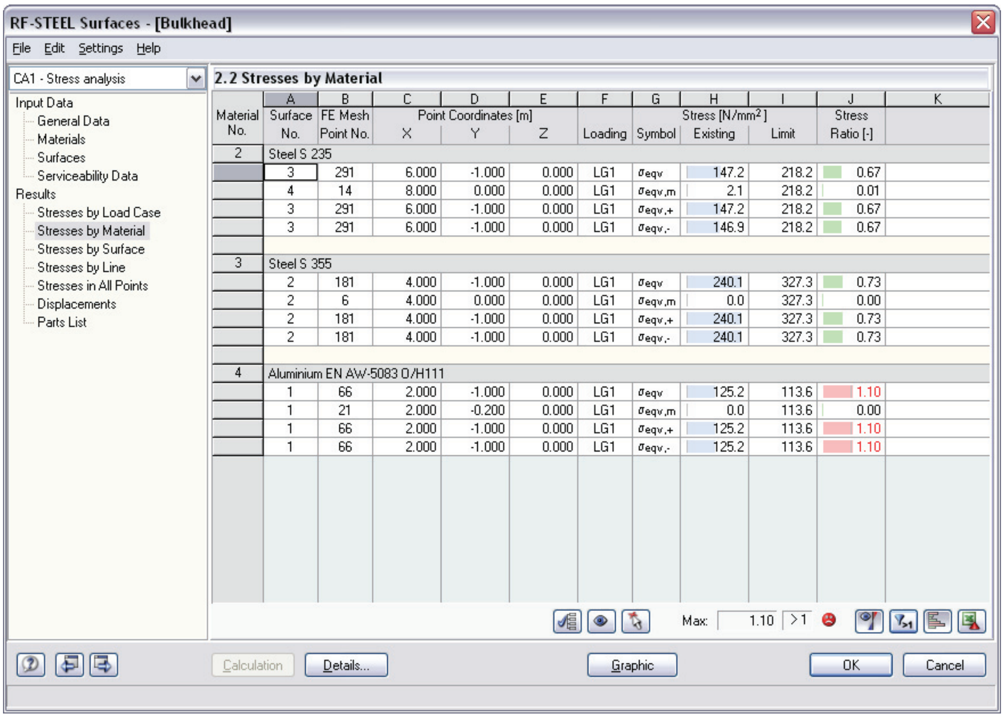
$$\frac{\tau}{\tau_{R,d}} \leq 1$$

Equation 2.6: Design condition for shear stresses

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

Equation 2.7: Design condition for equivalent and membrane equivalent stresses

2.3.2 Stresses by Material



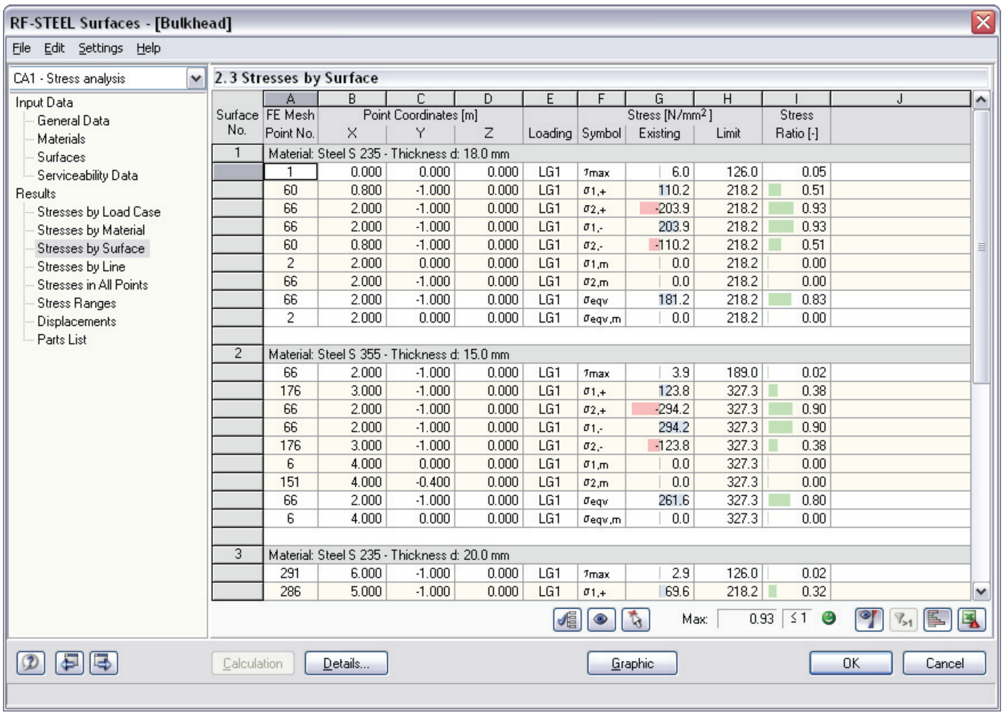
Material No.	A Surface No.	B FE Mesh Point No.	C Point Coordinates [m] X	D Y	E Z	F Loading	G Symbol	H Stress [N/mm²] Existing	I Limit	J Stress Ratio [-]	K
2	Steel S 235										
	3	291	6.000	-1.000	0.000	LG1	σ_{eqv}	147.2	218.2	0.67	
	4	14	8.000	0.000	0.000	LG1	$\sigma_{eqv,m}$	2.1	218.2	0.01	
	3	291	6.000	-1.000	0.000	LG1	$\sigma_{eqv,+}$	147.2	218.2	0.67	
	3	291	6.000	-1.000	0.000	LG1	$\sigma_{eqv,-}$	146.9	218.2	0.67	
3	Steel S 355										
	2	181	4.000	-1.000	0.000	LG1	σ_{eqv}	240.1	327.3	0.73	
	2	6	4.000	0.000	0.000	LG1	$\sigma_{eqv,m}$	0.0	327.3	0.00	
	2	181	4.000	-1.000	0.000	LG1	$\sigma_{eqv,+}$	240.1	327.3	0.73	
	2	181	4.000	-1.000	0.000	LG1	$\sigma_{eqv,-}$	240.1	327.3	0.73	
4	Aluminium EN AW-5083 O/H111										
	1	66	2.000	-1.000	0.000	LG1	σ_{eqv}	125.2	113.6	1.10	
	1	21	2.000	-0.200	0.000	LG1	$\sigma_{eqv,m}$	0.0	113.6	0.00	
	1	66	2.000	-1.000	0.000	LG1	$\sigma_{eqv,+}$	125.2	113.6	1.10	
	1	66	2.000	-1.000	0.000	LG1	$\sigma_{eqv,-}$	125.2	113.6	1.10	

Figure 2.18: Table 2.2 *Stresses by Material*

The maximum stress ratios listed in this table are sorted by materials. The materials are indicated in the first column, specifying the arrangement of the results output.

The different columns are described in detail in the previous chapter 2.3.1.

2.3.3 Stresses by Surface

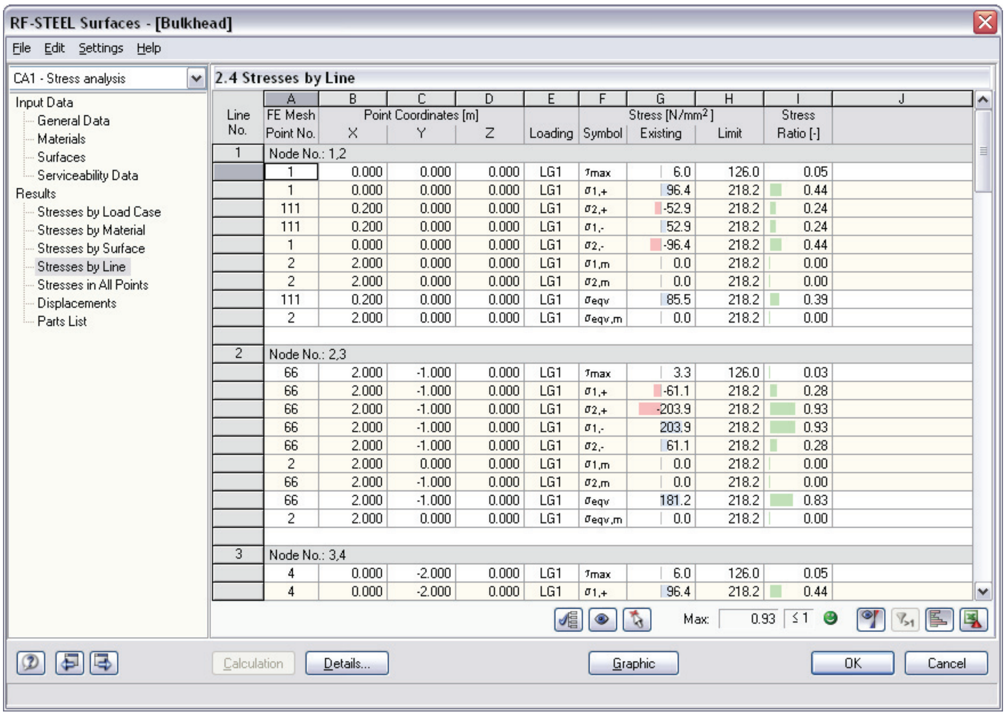


Surface No.	FE Mesh Point No.	Point Coordinates [m] X	Y	Z	Loading	Symbol	Stress [N/mm²] Existing	Limit	Stress Ratio [-]
1	Material: Steel S 235 - Thickness d: 18.0 mm								
	1	0.000	0.000	0.000	LG1	τ_{max}	6.0	126.0	0.05
	60	0.800	-1.000	0.000	LG1	$\sigma_{1,+}$	110.2	218.2	0.51
	66	2.000	-1.000	0.000	LG1	$\sigma_{2,+}$	-203.9	218.2	0.93
	66	2.000	-1.000	0.000	LG1	$\sigma_{1,-}$	203.9	218.2	0.93
	60	0.800	-1.000	0.000	LG1	$\sigma_{2,-}$	-110.2	218.2	0.51
	2	2.000	0.000	0.000	LG1	$\sigma_{1,m}$	0.0	218.2	0.00
	66	2.000	-1.000	0.000	LG1	$\sigma_{2,m}$	0.0	218.2	0.00
	66	2.000	-1.000	0.000	LG1	σ_{eqv}	181.2	218.2	0.83
	2	2.000	0.000	0.000	LG1	$\sigma_{eqv,m}$	0.0	218.2	0.00
2	Material: Steel S 355 - Thickness d: 15.0 mm								
	66	2.000	-1.000	0.000	LG1	τ_{max}	3.9	189.0	0.02
	176	3.000	-1.000	0.000	LG1	$\sigma_{1,+}$	123.8	327.3	0.38
	66	2.000	-1.000	0.000	LG1	$\sigma_{2,+}$	-294.2	327.3	0.90
	66	2.000	-1.000	0.000	LG1	$\sigma_{1,-}$	294.2	327.3	0.90
	176	3.000	-1.000	0.000	LG1	$\sigma_{2,-}$	-123.8	327.3	0.38
	6	4.000	0.000	0.000	LG1	$\sigma_{1,m}$	0.0	327.3	0.00
	151	4.000	-0.400	0.000	LG1	$\sigma_{2,m}$	0.0	327.3	0.00
	66	2.000	-1.000	0.000	LG1	σ_{eqv}	261.6	327.3	0.80
	6	4.000	0.000	0.000	LG1	$\sigma_{eqv,m}$	0.0	327.3	0.00
3	Material: Steel S 235 - Thickness d: 20.0 mm								
	291	6.000	-1.000	0.000	LG1	τ_{max}	2.9	126.0	0.02
	286	5.000	-1.000	0.000	LG1	$\sigma_{1,+}$	69.6	218.2	0.32

Figure 2.19: Table 2.3 *Stresses by Surface*

This results table contains the maximum stresses and stress ratios that are available for each designed surface. The results are listed by surfaces. In addition, information concerning material and thickness is displayed.

2.3.4 Stresses by Line



Line No.	FE Mesh Point No.	Point Coordinates [m] X	Y	Z	Loading	Symbol	Stress [N/mm²] Existing	Limit	Stress Ratio [-]
1	Node No.: 1,2								
	1	0.000	0.000	0.000	LG1	τ_{max}	6.0	126.0	0.05
	1	0.000	0.000	0.000	LG1	$\sigma_{1,+}$	96.4	218.2	0.44
	111	0.200	0.000	0.000	LG1	$\sigma_{2,+}$	-52.9	218.2	0.24
	111	0.200	0.000	0.000	LG1	$\sigma_{1,-}$	52.9	218.2	0.24
	1	0.000	0.000	0.000	LG1	$\sigma_{2,-}$	-96.4	218.2	0.44
	2	2.000	0.000	0.000	LG1	$\sigma_{1,m}$	0.0	218.2	0.00
	2	2.000	0.000	0.000	LG1	$\sigma_{2,m}$	0.0	218.2	0.00
	111	0.200	0.000	0.000	LG1	σ_{eqv}	85.5	218.2	0.39
	2	2.000	0.000	0.000	LG1	$\sigma_{eqv,m}$	0.0	218.2	0.00
2	Node No.: 2,3								
	66	2.000	-1.000	0.000	LG1	τ_{max}	3.3	126.0	0.03
	66	2.000	-1.000	0.000	LG1	$\sigma_{1,+}$	-61.1	218.2	0.28
	66	2.000	-1.000	0.000	LG1	$\sigma_{2,+}$	-203.9	218.2	0.93
	66	2.000	-1.000	0.000	LG1	$\sigma_{1,-}$	203.9	218.2	0.93
	66	2.000	-1.000	0.000	LG1	$\sigma_{2,-}$	61.1	218.2	0.28
	2	2.000	0.000	0.000	LG1	$\sigma_{1,m}$	0.0	218.2	0.00
	66	2.000	-1.000	0.000	LG1	$\sigma_{2,m}$	0.0	218.2	0.00
	66	2.000	-1.000	0.000	LG1	σ_{eqv}	181.2	218.2	0.83
	2	2.000	0.000	0.000	LG1	$\sigma_{eqv,m}$	0.0	218.2	0.00
3	Node No.: 3,4								
	4	0.000	-2.000	0.000	LG1	τ_{max}	6.0	126.0	0.05
	4	0.000	-2.000	0.000	LG1	$\sigma_{1,+}$	96.4	218.2	0.44

Figure 2.20: Table 2.4 *Stresses by Line*

The maximum stresses are displayed for all lines that are available in the designed surfaces. The results are listed by line numbers.

2.3.5 Stresses in All Points

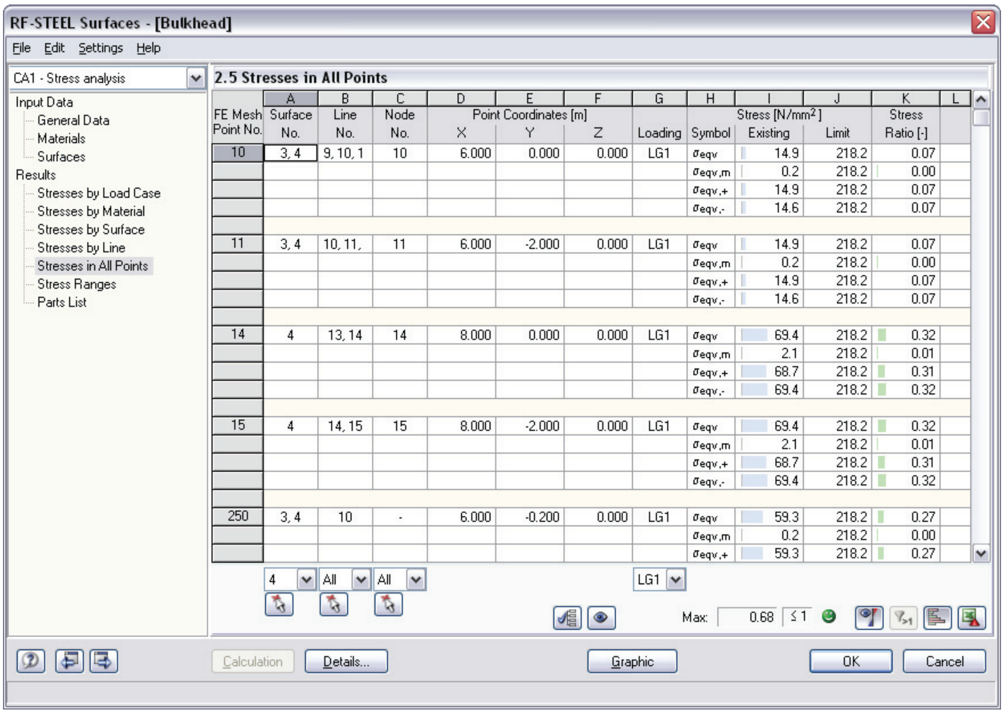


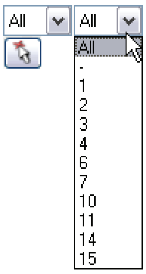
Figure 2.21: Table 2.5 *Stresses in All Points*

This results table is inactive by default because the program must manage a considerable amount of data when complex structures are analyzed. However, for a specific numerical evaluation, it is possible to activate the table by selecting the corresponding option in the *Options* tab of the *Details* dialog box (see Figure 2.13, page 26).

The table shows the stresses and stress ratios for each FE mesh or grid point that is available in the designed surfaces. By selection in the *Options* tab of the *Details* dialog box, you can decide if you want to consider either FE mesh nodes or user-defined grid points for the evaluation (cf. Figure 2.13, page 26). When data has been modified, a query appears before the results will be recalculated.

The display of stresses and stress components can be adjusted in the *Stresses* tab of the *Details* dialog box (see Figure 2.11, page 19). To open the dialog box, use the button shown on the left at the bottom of the list.

The different columns are described in detail in chapter 2.3.1, page 30.



Filter function

The table results can be sorted according to surface, line and node numbers as well as load cases if you want to evaluate the results specifically. Use the lists below the corresponding columns to select the data. You can also use the [Pick] function to select structural objects graphically.

2.3.6 Stress Ranges

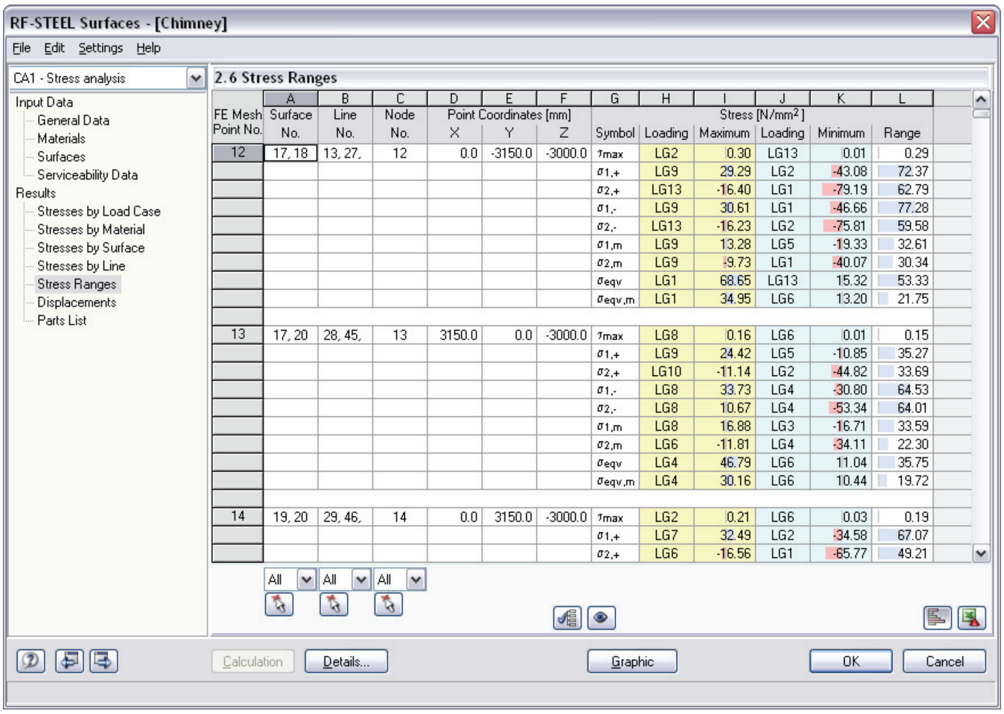
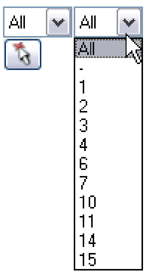


Figure 2.22: Table 2.6 *Stress Ranges*

This results table is inactive by default. However, to activate the table, tick the corresponding check box in the *Options* tab of the *Details* dialog box (see Figure 2.13, page 26).

The stress ranges of the stress intensities are required for fatigue designs when analyzing the fatigue behavior. The table displays the stress differences for each FE mesh or grid point of the designed surfaces. By selection in the *Options* tab of the *Details* dialog box, you can decide if you want to consider either FE mesh nodes or user-defined grid points for the evaluation (cf. Figure 2.13, page 26).

The display of stress types can be defined in the *Stresses* tab of the *Details* dialog box (see Figure 2.11, page 19). To open the dialog box, use the button shown on the left at the bottom of the list.



The table results can be filtered according to surface, line and node numbers. Use the lists below the corresponding columns to select the data. You can also use the [Pick] function for the graphical selection.

Details on columns A to G can be found in chapter 2.3.1 on page 30.

Loading

Columns H and J show the relevant load cases, load groups or load combinations that bear the maximum and minimum stress components. In these two columns, all actions that have been set for the ultimate limit state design are considered.

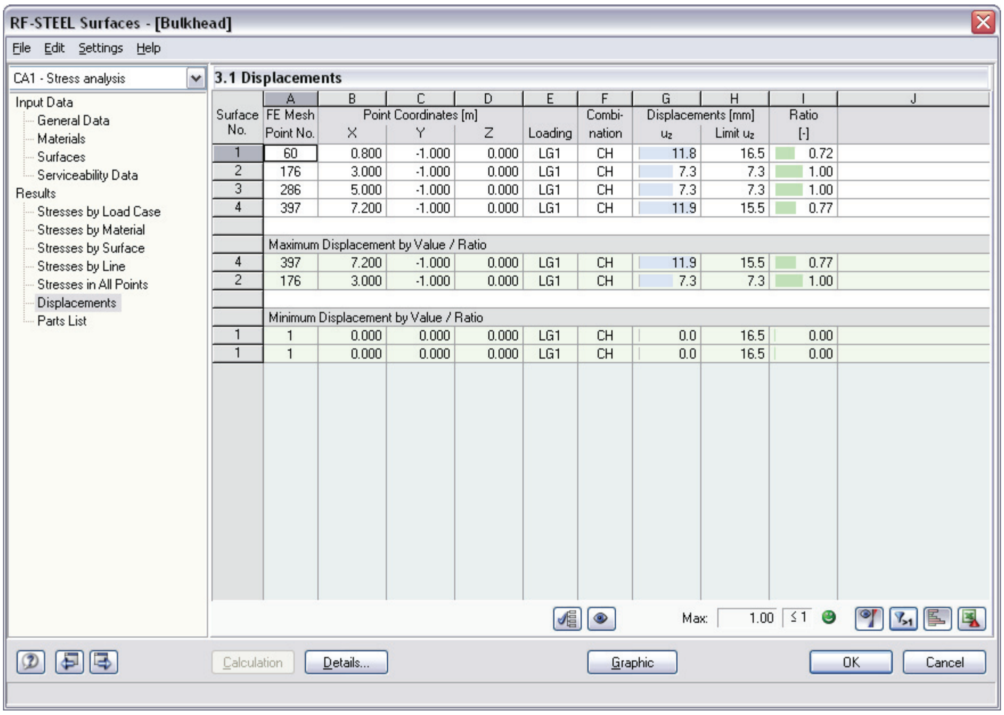
Maximum / Minimum

Column I lists the maximum stress values that are available for each stress type. Column K lists the minimum stress values. The *Maximum* is represented by the positive extreme values, the *Minimum* is represented by the negative ones.

Range

The values displayed in this column represent the range of stresses that results from the relevant extreme values in column I and K.

2.3.7 Displacements



Surface No.	A FE Mesh Point No.	B Point Coordinates [m] X	C Y	D Z	E Loading	F Combi-nation	G Displacements [mm] u_z	H Limit u_z	I Ratio [-]	J
1	60	0.800	-1.000	0.000	LG1	CH	11.8	16.5	0.72	
2	176	3.000	-1.000	0.000	LG1	CH	7.3	7.3	1.00	
3	286	5.000	-1.000	0.000	LG1	CH	7.3	7.3	1.00	
4	397	7.200	-1.000	0.000	LG1	CH	11.9	15.5	0.77	
	Maximum Displacement by Value / Ratio									
4	397	7.200	-1.000	0.000	LG1	CH	11.9	15.5	0.77	
2	176	3.000	-1.000	0.000	LG1	CH	7.3	7.3	1.00	
	Minimum Displacement by Value / Ratio									
1	1	0.000	0.000	0.000	LG1	CH	0.0	16.5	0.00	
1	1	0.000	0.000	0.000	LG1	CH	0.0	16.5	0.00	

Figure 2.23: Table 3.1 *Displacements*

The deformation analyses are only displayed when you have selected at least one action for the design in the *Serviceability Limit State* tab of table 1.1 *General Data* (see chapter 2.1.1.2, page 11). Furthermore, it is necessary to have defined surface reference lengths for the limit deformations in table 1.4 *Serviceability Data* (see chapter 2.1.4, page 18).

The results table displays the maximum displacements due to the serviceability load cases or combinations. The results are listed by surfaces.

In addition, the table shows the deformations and stress ratios for each FE mesh or grid point that is available in the designed surfaces. The settings in the *Options* tab of the *Details* dialog box (see Figure 2.13, page 26) determine if the table displays nodes of the FE mesh or user-defined grid points.

Details on columns A to D can be found in chapter 2.3.1 on page 30.

Loading

All actions that have been selected for the deformation analysis are listed by surfaces in column E. In this way, you can evaluate specifically the maximum deformation for each single load case or load combination.

Combination

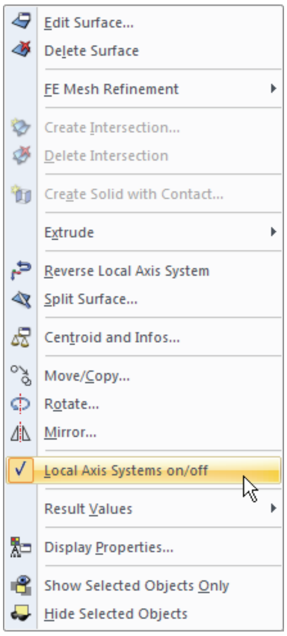
This column shows the action combinations that have been assigned in the *Serviceability Limit State* tab of table 1.1 *General Data* (see chapter 2.1.1.2, page 11).

- Characteristic (CH)
- Frequent (FR)
- Quasi-permanent (QP)

Displacements - u_z

This column displays the governing deformations in direction of the local surface axes z. The maximum deformations u_z refer to the non-deformed original structural system.

To display the local surface axes, use the *Display* navigator in RFEM or the context menu of the corresponding surface (see Figure 2.17, page 31).



Context menu of surface

Displacements - Limit u_z

Column H contains the limit deformations in direction of the z-axis of each surface. These deformations are determined by the reference lengths L of the boundary lines that have been defined in table 1.4 (see chapter 2.1.4, page 18) and the general serviceability limits specified in the *Serviceability* tab of the *Details* dialog box (see chapter 2.2.1.2, page 25).

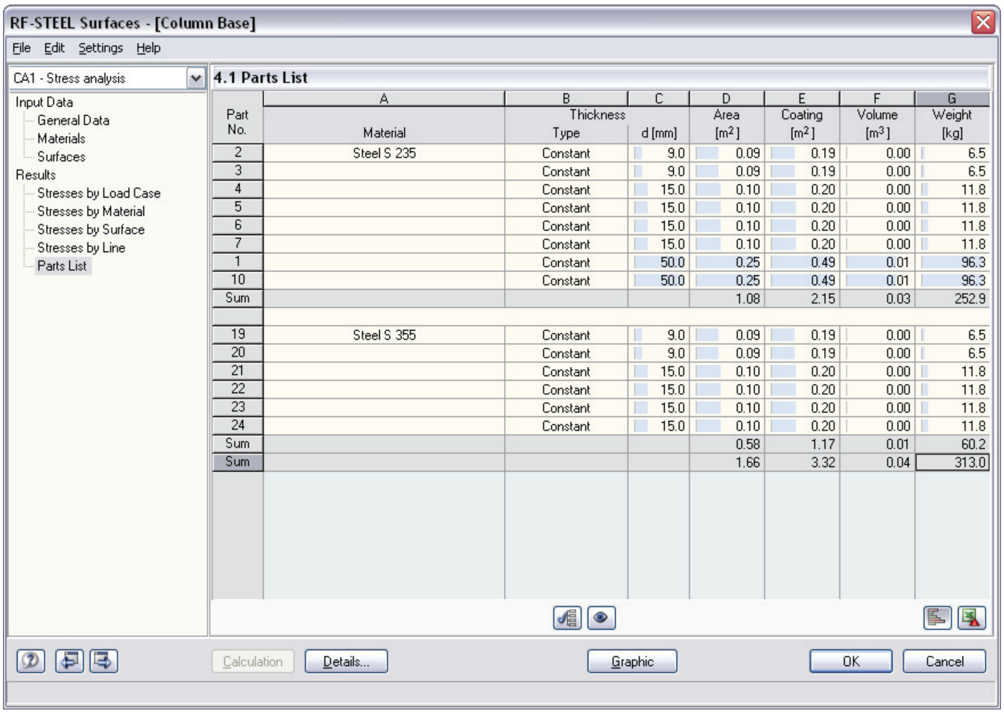
Ratio

The final column shows the quotients determined from the existing displacement u_z (column G) and the allowable displacement *Limit* u_z (column H). If the limit deformations are not exceeded, the ratio is less than or equal to 1.00 and the serviceability limit state design was carried out successfully.

Maximum and Minimum Displacement by Value / Ratio

At the bottom of the table, a summary is displayed showing the extreme displacement values and stress ratios of all designed surfaces. The output is divided in maximum (positive) and minimum (negative) results.

2.3.8 Parts List



Part No.	A Material	B Thickness Type	C d [mm]	D Area [m²]	E Coating [m²]	F Volume [m³]	G Weight [kg]
2	Steel S 235	Constant	9.0	0.09	0.19	0.00	6.5
3		Constant	9.0	0.09	0.19	0.00	6.5
4		Constant	15.0	0.10	0.20	0.00	11.8
5		Constant	15.0	0.10	0.20	0.00	11.8
6		Constant	15.0	0.10	0.20	0.00	11.8
7		Constant	15.0	0.10	0.20	0.00	11.8
1		Constant	50.0	0.25	0.49	0.01	96.3
10		Constant	50.0	0.25	0.49	0.01	96.3
Sum				1.08	2.15	0.03	252.9
19	Steel S 355	Constant	9.0	0.09	0.19	0.00	6.5
20		Constant	9.0	0.09	0.19	0.00	6.5
21		Constant	15.0	0.10	0.20	0.00	11.8
22		Constant	15.0	0.10	0.20	0.00	11.8
23		Constant	15.0	0.10	0.20	0.00	11.8
24		Constant	15.0	0.10	0.20	0.00	11.8
Sum				0.58	1.17	0.01	60.2
Sum				1.66	3.32	0.04	313.0

Figure 2.24: Table 4.1 *Parts List*

Finally, RF-STEEL Surfaces provides a summary of all surfaces that are included in the design case. By default, the list contains only the designed surfaces. If you want to display a parts list with all surfaces of the structure, select the corresponding option in the *Options* tab of the *Details* dialog box (see Figure 2.13, page 26).

Part No.

Referring to the surface numbers, the program assigns part numbers to the surfaces.

Material

The surfaces are listed by materials. Each material is closed by the *Sum* of the values that are displayed in column D to G.

Thickness

Columns B and C show the *Type* of thickness as well as the thickness *d*. The entries refer to the input data in table 1.3 *Surfaces*.

Area

This column gives information about the area of each surface.

Coating

The surface area of a surface is determined from the top and bottom side of the surface. As steel surfaces are usually thin-walled, the lateral surfaces are not considered.

Volume

The volume of a surface is determined by the product of its thickness and area. If the surface has a variable thickness, the program considers this property accordingly.

Weight

The final column displays the masses of each surface. The value is determined from the relevant volume of the surface and the weight density of the material that is used.

Sum

At the bottom of the list, you find a summary showing the sums of the columns D to G. The results output in the *Weight* column gives information about the overall steel mass that is required.

3. RF-STEEL Members

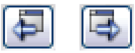
3.1 Input Data

The data for the definition of design cases is entered in tables.

When you have started RF-STEEL Members, a new window opens where a navigator is displayed on the left, managing all tables that can be selected currently. The pull-down list above the navigator contains the design cases already available (see chapter 6.1, page 82).

If you open RF-STEEL Members in an RFEM structure for the first time, the module imports the following design relevant data automatically:

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



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

To save the defined settings and quit the module, click [OK]. When you click [Cancel], you quit the module but without saving the data.

3.1.1 General Data

In table 1.1 *General Data*, you define the loads and members that you want to design. The design standard is specified in table 1.2, even though indirectly, because the material properties are related to the standard.

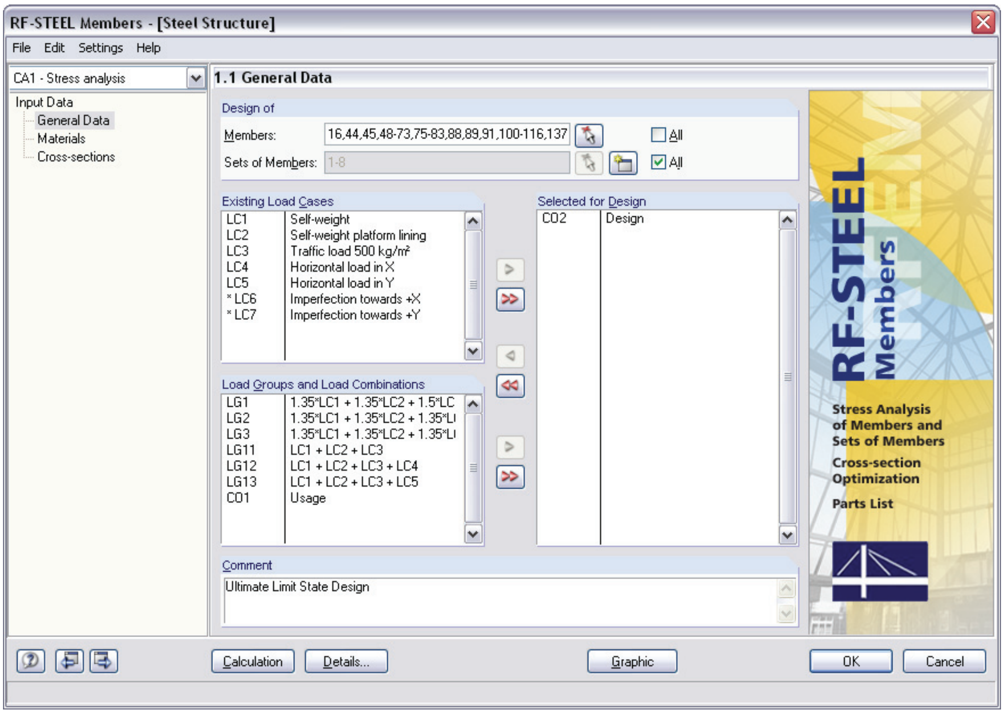


Figure 3.1: Table 1.1 *General Data*

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 two input fields to enter the numbers of the relevant members or sets of members. The list of the pre-set member numbers can be selected quickly by double click and overwritten by entering the data manually.

To select the objects graphically in the RFEM work window, use the [Pick] button.

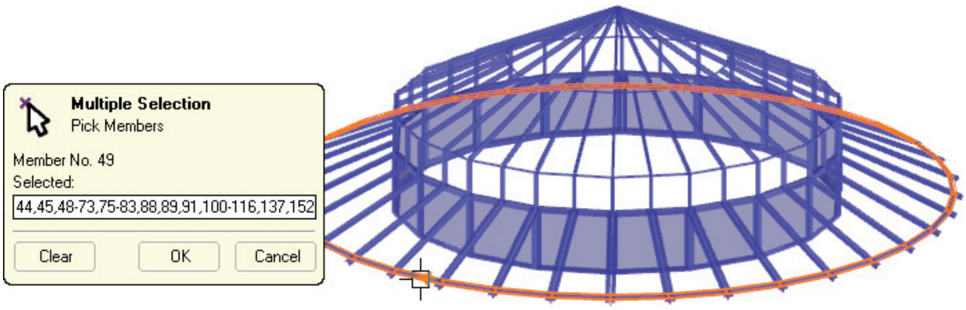


Figure 3.2: Graphical selection in the model

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

By means of a set of members design, it is possible to design several selected members and determine the total maxima of the stresses and stress ratios of all members contained in the set of members. In such a case, table 2.2 *Stresses by Set of Members* and table 4.2 *Parts List by Set of Members* are displayed.

Existing Load Cases / Load Groups and Load Combinations

These two sections list all load cases, load groups and load combinations defined in RFEM that are relevant for the design. Use the button [▶] to transfer selected load cases or combinations to the list *Selected for Design* on the right. You can also double-click the entries. To transfer the complete list to the right, use the button [▶▶].

When load cases are marked by an asterisk (*) like load cases 6 and 7 in Figure 3.1, they cannot be calculated. This may be the case when no loads are defined or when the load case contains only imperfections.

Selected for Design

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

The analysis of an enveloping *Or* load combination is often carried out more quickly than the global design of all created load cases and load groups. On the other hand, when analyzing the entire load combination, the influence of the contained loads is hardly transparent: RF-STEEL Members uses the max/min results of the RFEM table 3.5 *Members - Internal Forces* where the internal forces of the different load cases are superimposed accordingly. Therefore, when selecting load combinations, it is recommended to check the settings for the *Method of Stress Calculation* in the *Details* dialog box additionally (see chapter 3.2.2, page 50).

Comment

This input field allows for user-defined remarks, for example to describe the current design case.

3.1.2 Materials

The table is subdivided into two parts. In the upper part, the materials used for the design are listed with the relevant 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.

The material properties required for the determination of internal forces in RFEM are described in detail in chapter 5.3 of the RFEM manual. All design relevant material properties are stored in the global material library and 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.

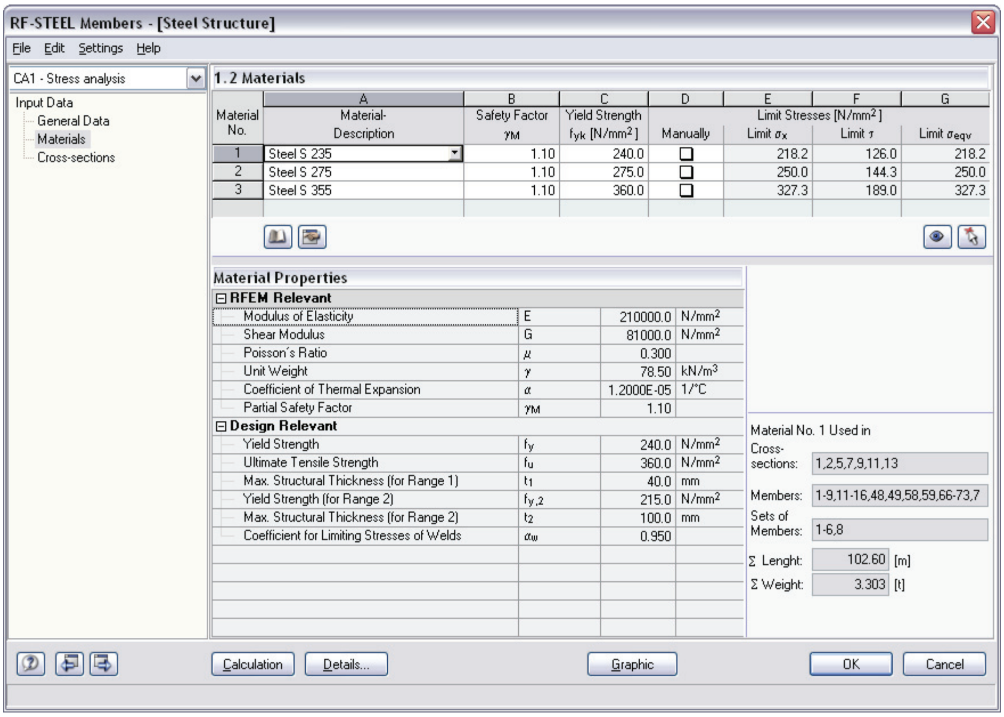
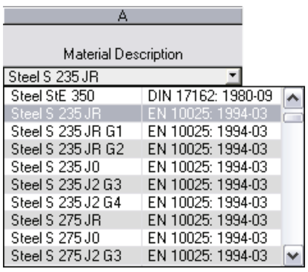


Figure 3.3: Table 1.2 *Materials*

Material Description



The materials defined in RFEM are already preset. But it is possible to select another 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]. The list shown on the left opens. When you have selected a new material, the design relevant properties are entered into the corresponding table row and are highlighted in blue.

Only steel materials are available in the list. 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. Therefore, it would be possible to design cross-sections made of aluminum or stainless steel. However, you must consider the corresponding 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 specified data. When you have defined the allowable stresses in the columns E to G, the red color of the table row will disappear.

The import of materials from the library is described on page 14.

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 using the factor γ_M, the characteristic value of the yield strength f_{yk} is reduced for the determination of the limit normal stress $\sigma_{R,d}$ (see Equation 2.1) and the limit shear stress $\tau_{R,d}$ (Equation 2.2).

Thus, 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: On the one hand, you must consider, for example according to DIN 18800 part 2, el. (116), the influence of deformations by a stiffness that is reduced about 10 % when determining internal forces. On the other hand, additionally, you must reduce the design values of 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 or EC 3, section 3.

Limit Stresses

The limit stresses for materials from the general material library are preset automatically. Those entries provide no access for modifications.

In case you want to modify the limit stresses, you can use the [Edit Material] button to open the *Edit Material* dialog box where you can change the material properties (see Figure 2.6, page 15). 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 subsequent columns manually. Materials that have been modified are highlighted in blue and 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 membrane force. It is determined, for example according to DIN 18800 part 1, 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 3.1

limit τ

The limit shear stress indicates the allowable shear stress due to shear and torsion. For example according to DIN 18800 part 1, el. (746), the partial safety factor γ_M is also considered in the equation for the determination of limit shear stress.

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

Equation 3.2

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 3.1.

Yield strength depending on thickness of structural components

For some materials, there is a relation between the characteristic yield strength f_{yk} and the thickness *t* of the relevant structural component. The *Max. Structural Thickness* of the respective ranges with the corresponding yield strength is indicated in the *Material Properties* section of table 1.2.

The yield strengths of structural thicknesses are defined in the standards, for example DIN 18800 part 1, table 1. To check and adjust, if necessary, the structural thicknesses and assigned stresses, use the [Edit Material] button (see Figure 2.6, page 15).

Material Library

A number of materials is stored in the library that you can access by using the button below column A. The dialog box which you already know from RFEM opens (see Figure 2.5, page 14).

The import of materials from the library is described in chapter 2.1.2 on page 14. Chapter 5.3 of the RFEM manual describes in detail how materials in the library can be filtered, added or rearranged.

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. Under these conditions, it would be possible to design, for example, structural components made of aluminum or stainless steel.

When you import a material whose limit stresses (for example timber) are not defined, the entries of the corresponding table row in table 1.2 are highlighted in red. It is possible, however, to define the limit stresses of this material by ticking the check box **Manually** in column D and entering user-defined specifications. As soon as you have defined the allowable stresses in the columns E to G, the red color of the table row disappears. Please note that some stress designs, for example for timber cross-sections, can only be carried out to some extent. The corresponding standard criteria are 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 below column A. It is also shown on the left.

The *Edit Material* dialog box shown in Figure 2.6 and described on page 15 opens. The manual text describes how to adjust the partial safety factor, the characteristic values of the yield strength f_{yk} or the yield strengths and limit stresses in relation to the component thickness as well as the ranges of the component thicknesses.

3.1.3 Cross-sections

In this table, the cross-sections that should be designed are listed. The material numbers already assigned in RFEM are preset (see description of previous table 1.2 *Materials*). Additionally, it is possible to define optimization parameters.

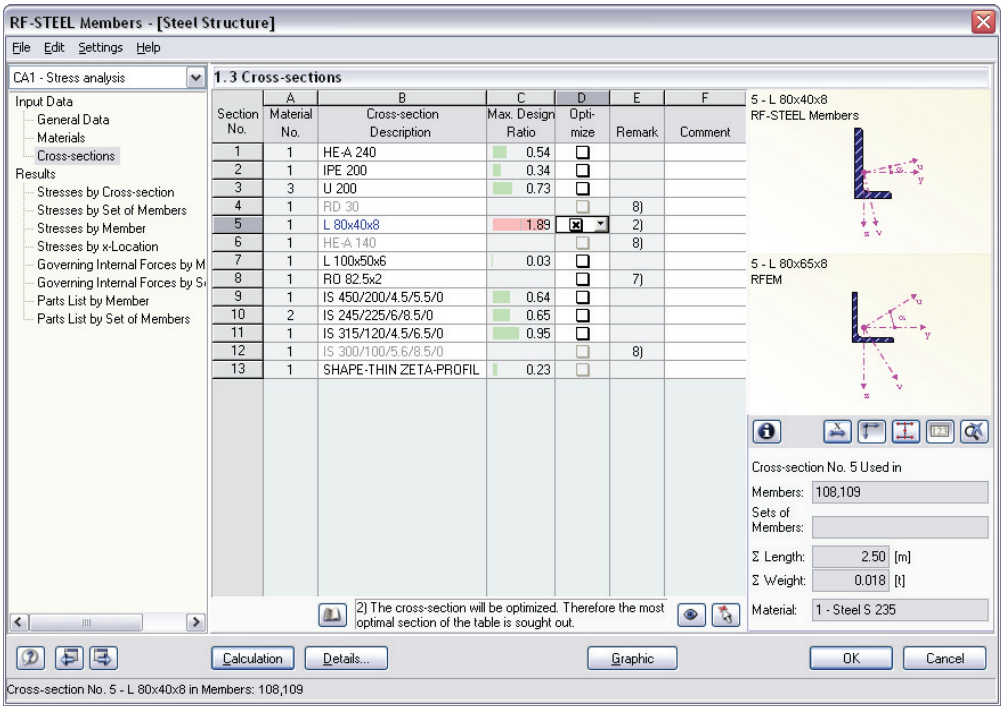


Figure 3.4: Table 1.3 *Cross-sections*

Cross-section Description

When you open the table, the thickness types of the cross-sections used in RFEM are preset together with the assigned material numbers.

It is always possible to modify the preset cross-sections for the design. The description of a modified cross-section is highlighted in blue.

If you want to modify a cross-section, you can enter the new cross-section description directly into the corresponding table row. As an alternative, you can select the relevant cross-section from the library. The library can be opened by clicking the button [Import Cross-section from Library], by placing the pointer into the corresponding table row to enable the [...] button or by using the function key [F7]. The library dialog box from RFEM appears with the selected cross-section table.

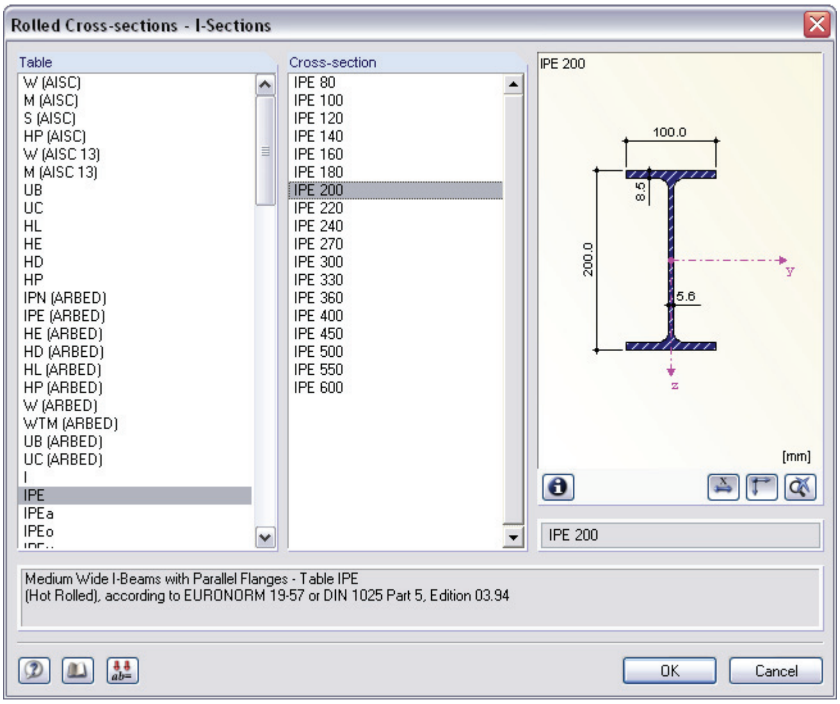
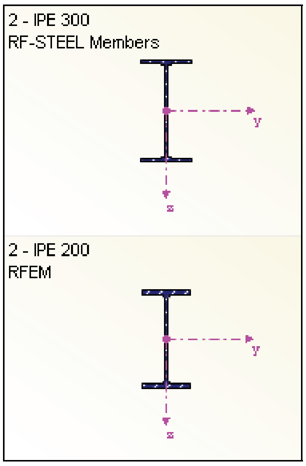


Figure 3.5: Dialog box of cross-section library *Rolled cross-sections - I-Sections*



The selection of cross-sections from the library is described in details in chapter 5.13 of the RFEM manual.

If the cross-sections in RF-STEEL are different from the ones used in RFEM, both cross-sections are displayed in the graphic in the right part of the table. In such a case, the program will carry out the stress designs for the cross-section selected in RF-STEEL with the internal forces from RFEM.

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 RFEM definition, in two table rows. RF-STEEL Members can design tapered members when both cross-sections have the same number of stress points.

The normal stresses are determined from the moments of inertia and the centroidal distances of the stress points. If the start and the end cross-section of a tapered member have not the same number of stress points, the intermediate values cannot be interpolated. In such a case, RFEM applies the corresponding cross-section properties up to the middle of the member to facilitate the calculation and the rendering. In RF-STEEL, however, a design is not possible. Therefore, a warning appears before the calculation will be carried out.

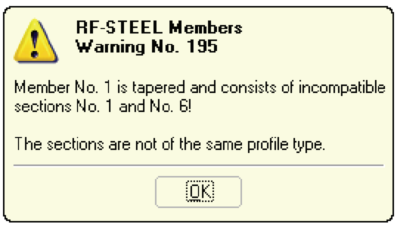
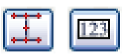
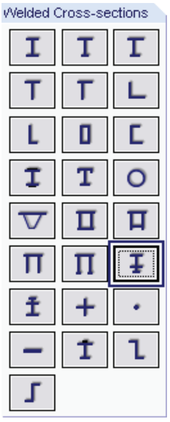


Figure 3.6: Warning in case of incompatible cross-sections



To check the data, you can display the cross-section's stress points including numbering in the cross-section graphic (in the right part of the table). For more information on stress points, see chapter 3.3.1 on page 55.



IVU-sections

To produce the same number of stress points for tapered members, you can, for example, model the taper's end cross-section as a copy of the cross-section start adjusting only 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 for tapers.

Max. Design Ratio

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

Optimize

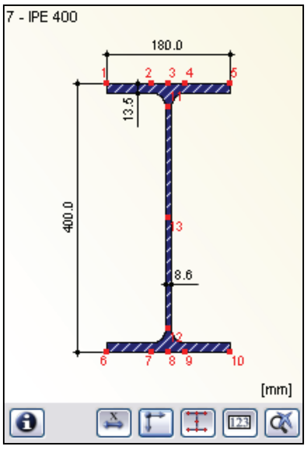
For each cross-section, you can carry out an optimization process. By using the RFEM internal forces, the module determines the cross-section within the respective cross-section table that comes as close as possible to a user-defined maximum ratio. This ratio is defined in the *Details* dialog box (see Figure 3.8, page 50).

To optimize a particular cross-section, tick its corresponding check box in column C or D. Recommendations for the cross-section optimization can be found in chapter 6.2.2 on page 85.

Remark

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

If the display indicates remark *6) Incorrect cross-section data! Members of this cross-section will not be designed.*, you have defined an unknown cross-section, i.e. it is not available in the cross-section database. This may be the case for a user-defined or not yet calculated SHAPE cross-section. In such a case, define the required settings in RFEM, for example the **Stress Points** (see chapter 5.13 of the RFEM manual, section *Create User-defined Cross-sections*).



Cross-section graphic

In the right part of table 1.3, the currently selected cross-section is displayed graphically. The buttons are reserved for the following functions:

Button	Function
	Opens the dialog box *Info about cross-section* with the cross-section properties.
	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 cross-section graphic.

Table 3.1: Buttons of cross-section graphic

3.2 Calculation

Calculation

Details...

The stress analysis is carried out by using the internal forces determined in RFEM. Before you start the calculation by clicking the [Calculation] button, it is recommended to check the design details. To open the corresponding dialog box, use the [Details] button. Details on this dialog box can be found in chapter 3.2.2 on page 50.

3.2.1 Stresses and Ratio

Tables 2.1 to 2.5 show the normal stresses σ_{total}, τ_{total} and σ_{eqv} by default. To display the individual stress components, use the buttons [Select Stresses to Show] and [Extended Stress Diagram].

Normal stresses

The normal stresses σ have the following meanings:

σ_N	Stress due to axial force N $\sigma = \frac{N}{A}$ with A: cross-sectional area of cross-section
σ_{M-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 factor according to DIN 18800 part 1, el. (750) I_y: moment of inertia related to principal axis y e_z: centroidal distance of the stress point in direction z
σ_{M-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 factor according to DIN 18800 part 1, el. (750) I_z: moment of inertia related to principal axis z e_y: centroidal distance of the stress point in direction y
σ_M	Stress due to bending moments M_y and M_z $\sigma = \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$
$\sigma_{tensile}$	Tensile stress due to axial force N and bending moments M_y and M_z $\sigma = \frac{N}{A} + \frac{M_y}{\alpha_{pl,y} \cdot I_y} \cdot e_z - \frac{M_z}{\alpha_{pl,z} \cdot I_z} \cdot e_y$
$\sigma_{compressive}$	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 the normal stresses of the 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 3.2: Normal stresses σ

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

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

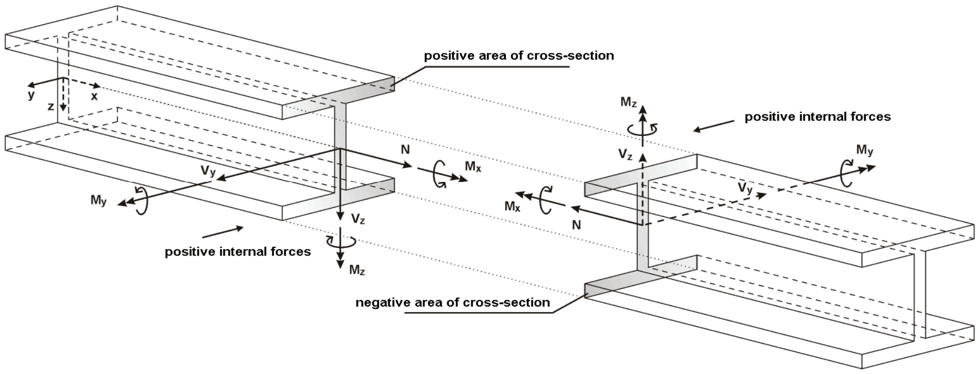


Figure 3.7: Positive definition of internal forces

The bending moment $\mathbf{M_y}$ is positive if tensile stresses are generated 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.

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

Shear stresses

The shear stresses τ have the following meanings:

$\tau_{V\text{-}y}$	Stress due to shear force V_y $\tau = -\dfrac{V_y \cdot Q_z}{I_z \cdot t}$ with Q_z: static moment related to principal axis z I_z: moment of inertia related to principal axis z t: governing thickness of cross-section
$\tau_{V\text{-}z}$	Stress due to shear force V_z $\tau = -\dfrac{V_z \cdot Q_y}{I_y \cdot t}$ with Q_y: static moment related to principal axis y I_y: moment of inertia related to principal axis y t: governing thickness of cross-section
τ_V	Stress due to shear forces V_y and V_z $\tau = -\dfrac{V_y \cdot Q_z}{I_z \cdot t} - \dfrac{V_z \cdot Q_y}{I_y \cdot t}$

$\tau_{\text{M-T, St.Venant}}$	Stress due to torsional moment M_T in case of open cross-section $\tau = \frac{M_T}{J_{St.V.}} \cdot t$ with $J_{St.V.}$: Saint Venant torsional constant t: governing thickness of cross-section
$\tau_{\text{M-T, Bredt}}$	Stress due to torsional moment M_T in case of closed cross-section $\tau = \frac{M_T}{2 \cdot A_m \cdot t}$ with A_m: area enclosed by the center lines of the cross-section t: governing thickness of cross-section
$\tau_{\text{M-T}}$	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 3.3: 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.

Concerning the 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. RF-STEEL Members won't carry out any pro-rata analysis for $M_{T,St.Venant}$ and $M_{T,Bredt}$ like it is done in the cross-section module SHAPE.
- The influence of the warping torsion is not considered in RF-STEEL. The design, like the determination of internal forces in RFEM, 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 RF-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:

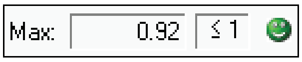
σ_{eqv}	Equivalent stress from normal stresses σ and shear stresses τ $\sigma_{eqv} = \sqrt{f_1 \cdot \sigma_{total}^2 + f_2 \cdot \tau_{total}^2}$ with f_1: factor for normal stresses f_2: factor for shear stresses

Table 3.4: Equivalent stress σ_{eqv}

Factors f_1 and f_2 can be defined in the *Details* dialog box (see Figure 3.8, page 50). According to DIN 18800 part 1, el. (748), the factors are preset with f_1 = 1.0 and f_2 = 3.0.

Stress Ratio

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



For each internal force component, you can display the cross-section's stress ratio on the governing stress point (cf. chapter 4.2.2, page 70). The ratios due to normal, shear and equivalent stress are displayed in the results table by default. If the limit stress is not exceeded, the ratio is less than or equal to 1 and the stress design was carried out successfully.

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

Equation 3.3: Design condition for normal stresses

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

Equation 3.4: Design condition for shear stresses

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

Equation 3.5: Design condition for equivalent stresses

3.2.2 Calculation Details

To check the different calculation parameters, open the corresponding dialog box by using the [Details] button. The dialog box can be accessed from any results table in the module.

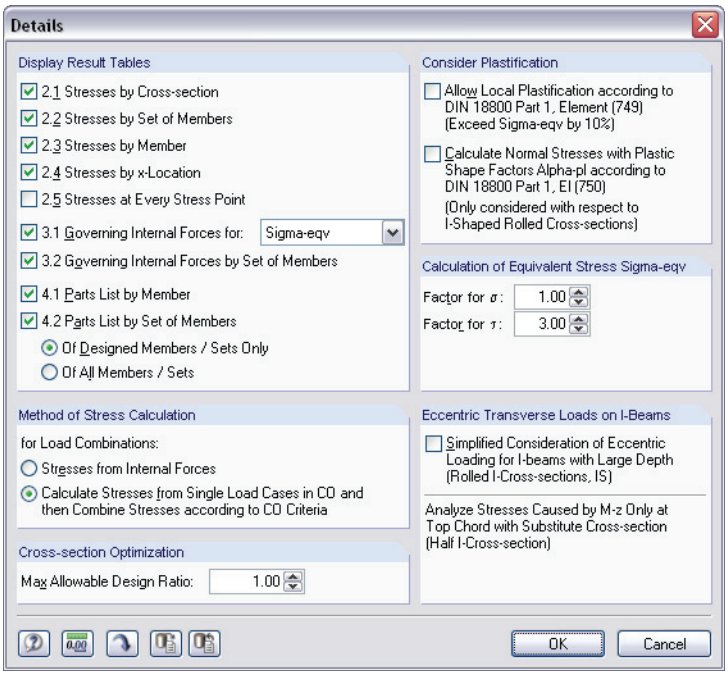
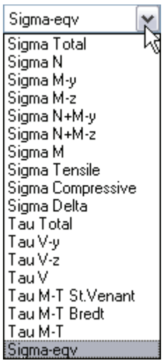


Figure 3.8: Dialog box *Details*

Display Result Tables

This dialog section controls the display of the results tables.

Table 2.5 *Stresses at Every Stress Point* is inactive by default as the stress graphic also provides access to the stress point results. However, for a detailed table check of the stresses, it may be useful to activate the table's display.



The values displayed in table 3.1 *Governing Internal Forces* are generally related to the equivalent stress σ_{eqv}. Use the list to select another stress type.

Method of Stress Calculation

In case of a biaxial loading, it may occur that the combined member internal forces do not produce the maximum stresses. This is the case, for example, when a load case with vertical load has M_y moments but no M_z moments, and another load case with horizontal load has M_z moments but no M_y moments. Both load cases are superimposed with the setting "Variable" in a load combination. In the RFEM table 3.5 *Members - Internal Forces*, the program will display the moment M_z as not corresponding to the maximum moment M_y because the horizontal load does not contribute to the increase of the moment due to vertical load. Therefore, in a separate design carried out only for the maximum moments M_y and M_z, a simultaneous influence of both internal forces on the combined bending stresses would not be considered.

Stresses from Internal Forces

The calculation method *Stresses from Internal Forces* uses the result rows of the RFEM table 3.5 *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 are transparent: In table 3.1 *Governing Internal Forces*, the module displays the result rows from the RFEM table 3.5 *Members - Internal Forces*.

Calculate Stresses from Single Load Cases and then Combine

This type of calculation is preset for the stress analysis 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 too low stress ratios.

The analysis is carried out stress point by stress point. The compressive, tensile and shear stresses available in the single load cases are summed up respectively and then displayed in the results tables. The equivalent stress σ_{eqv} is an exception: 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 thus would result in too high stress ratios. However, this calculation method may produce problems in "Or" superpositions because the equivalent stresses determined according to this approach possibly do not occur in such a way.

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 supposed to show the same stress ratios in the output.

Cross-section Optimization

If the optimization process does not gear towards the maximum stress ratio of 100 %, you can specify a different limit value in this input field.

Consider Plastification

Allow Local Plastification

Optionally, you can allow a *Local Plastification* for the design according to DIN 18800 part 1, el. (749). This means that the equivalent stress σ_{eqv} may exceed the allowable limit stress by 10 % in small cross-section areas. RF-STEEL checks if both conditions mentioned in el. (749) are fulfilled for the assumption "small areas".

$$\left|\sigma_N + \sigma_{M_y}\right| \leq 0{,}8 \cdot \sigma_{R,d}$$

Equation 3.6

$$\left|\sigma_N + \sigma_{M_z}\right| \leq 0{,}8 \cdot \sigma_{R,d}$$

Equation 3.7

If this is the case, the limit stress for the design of σ_{eqv} will be increased appropriately.

Plastic shape factors α_{pl}

You can reduce the stresses by the *Plastic Shape Factors* α_{pl} mentioned in DIN 18800 part 1, el. (750). This option 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 3.8

If you use this plastification option, RF-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 3.9

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

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 RF-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 RFEM 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 create a separate RF-STEEL case for such high I-beams.

3.2.3 Start Calculation

You can start the calculation out of each of the three input tables by clicking the [Calculation] button.

RF-STEEL Members 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 RFEM calculation to determine the design relevant internal forces. In this determination process, the calculation parameters preset in RFEM are applied.

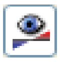
If you have specified an optimization of cross-sections (see chapter 6.2, page 83), the program determines the required cross-sections first and calculates their stresses subsequently.

You can also start the calculation for RF-STEEL Members out of the RFEM user interface. The add-on modules are listed in the dialog box *To Calculate* like load cases or load groups. To open the dialog box in RFEM,

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

The dialog box is shown in chapter Figure 2.14 on page 28.

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



To calculate an RF-STEEL case directly, use the list in the RFEM toolbar. Select the relevant design case in the toolbar list and click the button [Results on/off].

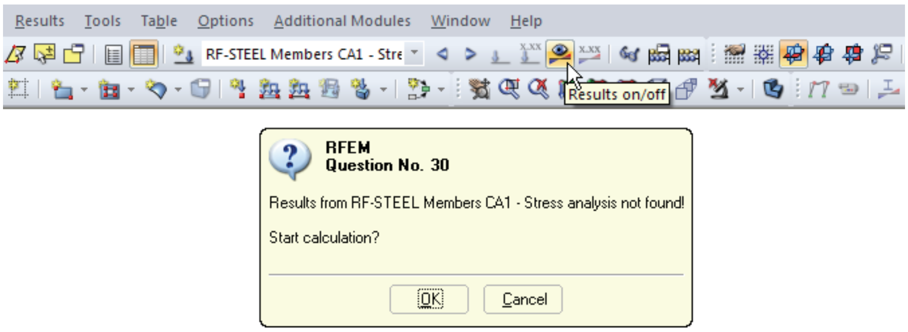
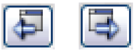


Figure 3.9: Direct calculation of an RF-STEEL design case in RFEM

3.3 Results

Table 2.1 *Stresses by Cross-section* is displayed immediately after the calculation. All stresses are shown in the results tables 2.1 to 2.5, sorted by different criteria. Tables 3.1 and 3.2 contain the governing internal forces, tables 4.1 and 4.2 show the parts lists for the members and sets of members.

In the *Details* dialog box, you can select the results tables that you want to display in particular (see Figure 3.8, page 50).



To access the results tables, use the RF-STEEL navigator. 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 [OK] to save the results and quit the add-on module RF-STEEL Members.

In the following, the different results tables are described in sequence. Evaluating and checking results is described in chapter 4 *Results Evaluation*.

3.3.1 Stresses by Cross-section

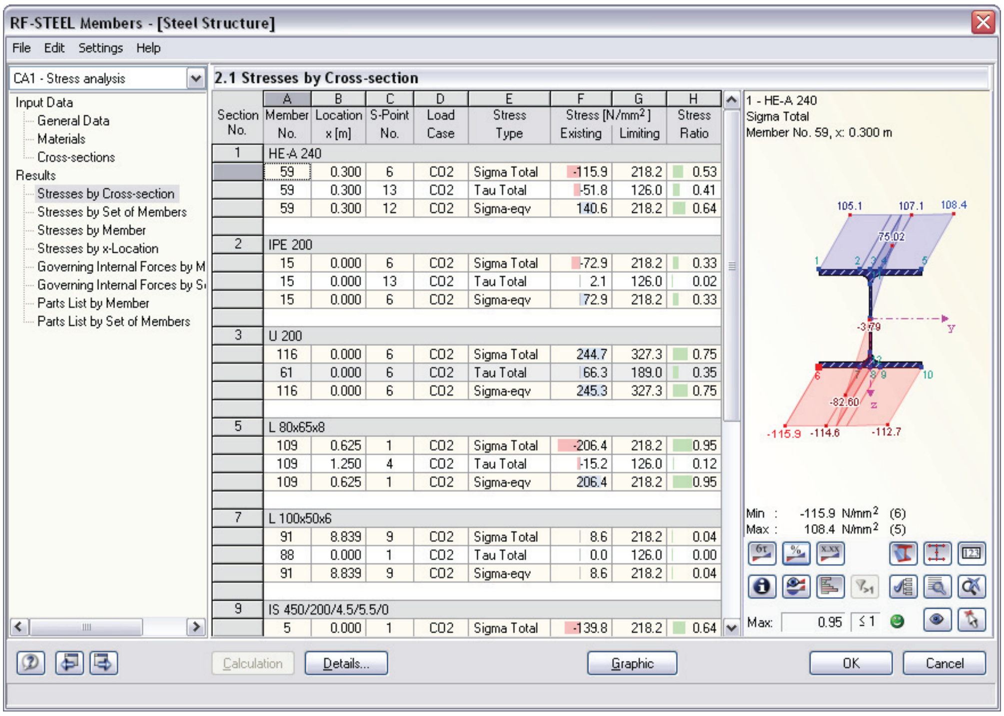


Figure 3.10: Table 2.1 *Stresses by Cross-section*

The table shows the maximum stress ratios of all designed members resulting from the selected load cases, load groups and load combinations. The results are listed by cross-sections. If you have a tapered member, both cross-section descriptions are displayed in the table row next to the section number.

The stress components displayed in the column *Stress Type* are based on the settings selected in the dialog box *Stresses - Filter* (see Figure 4.7, page 69). 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 where the member's maximum stress ratio occur. For the table output, the program uses the following RFEM member locations *x*:

- Start and end node
- Partition points according to possibly defined member division
- Extreme values of internal forces

S-Point No.

The design is carried out on certain stress points of the cross-section. These points are cross-section locations that are defined by centroidal distances, static moments and cross-section thicknesses. Due to these cross-section properties, the design according to equation Table 3.2 and Table 3.3 is possible.

All default cross-sections of the library as well as the SHAPE 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.

In the dialog graphic on the right, you can display the stress points together with their numbering. The currently selected stress point (that means the stress point of the table row where the pointer is placed) is highlighted in red.

Use the button [Info about Current Cross-section] to check the stress point's properties. First, the dialog box *Info about cross-section* opens showing a list of all section properties. Below the dialog graphic, you find the button [Details of Stress Points] that provides access to the stress point information.

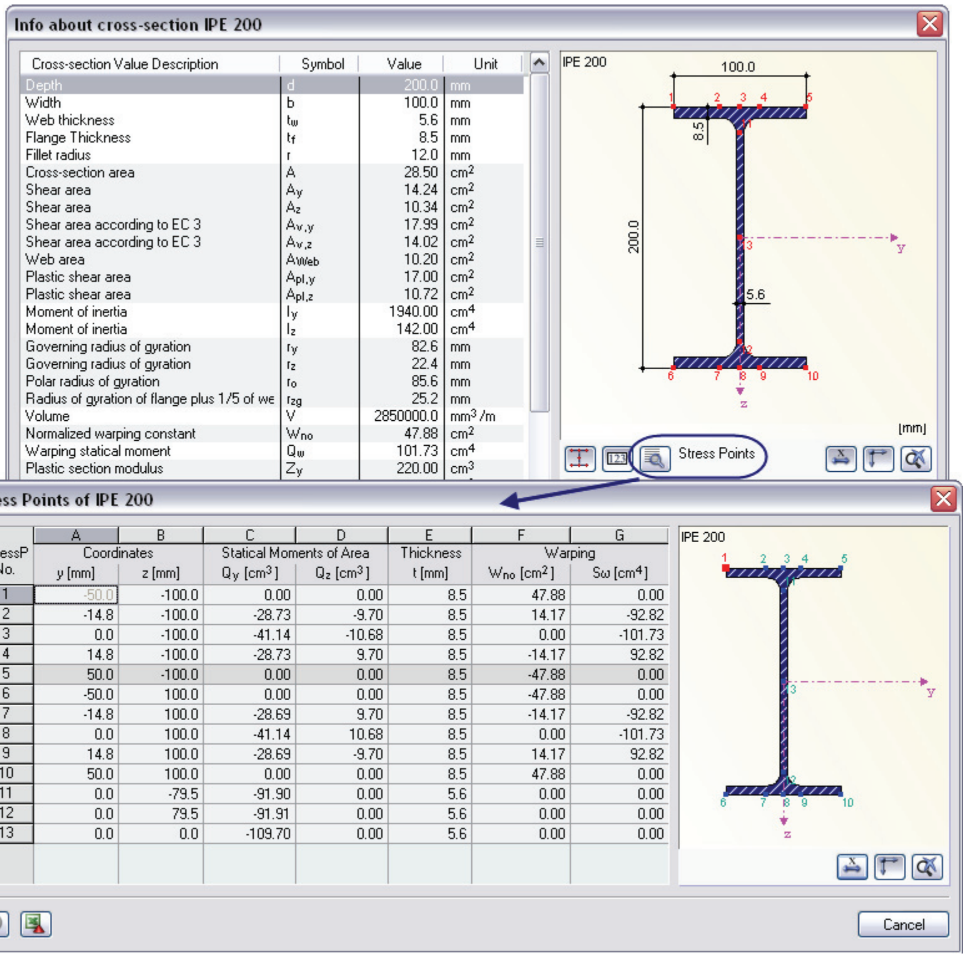


Figure 3.11: Dialog box *Info about cross-section* and *Stress Points*

In the dialog box *Stress Points* (Figure 3.11), the relevant centroidal distances e_y and e_z are listed below *y* and *z* of the *Coordinates* column. Columns Q_y and Q_z of the *Statical Moments of Area* show the static moments 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* column are not relevant for the design carried out with the add-on module RF-STEEL Members.

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 a combined calculation (for example σ_{eqv}): Often, the maximum stresses occur at different stress points. You have to superimpose the stress components of the respective stress point! To evaluate the stress point results, use table 2.5 (see chapter 3.3.5, page 59) or the dialog box *Cross-section Values and Stress Diagram* (see Figure 4.9, page 71).

Load Case

Column D shows the relevant load cases, load groups or load combinations that produce the maximum stress ratios.

Stress Type

The normal stresses σ_{total}, the shear stresses τ_{total} and the equivalent stresses σ_{eqv} are displayed by default. The determination of these stresses is documented in Table 3.2, Table 3.3 and Table 3.4 on page 47 to 49.

The stress components considered in the total stresses can be displayed as shown in Figure 3.12 below. You can select the different stress components in the dialog box *Stresses - Filter* (see Figure 4.7, page 69). To access this dialog box, use the button shown on the left.

Stress Existing

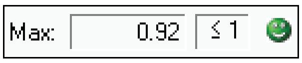
This column displays the extreme values of the existing stresses determined according to the equations of Table 3.2, Table 3.3 and Table 3.4 shown on page 47 to 49.

Stress Limiting

This column shows the limit stresses of table 1.2 (see chapter 3.1.2, page 41). In particular, they are the following:

- Limit normal stress σ_x as the allowable stress for actions due to bending moment and membrane 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, RF-STEEL Members 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 and the stress design was carried out successfully.

3.3.2 Stresses by Set of Members

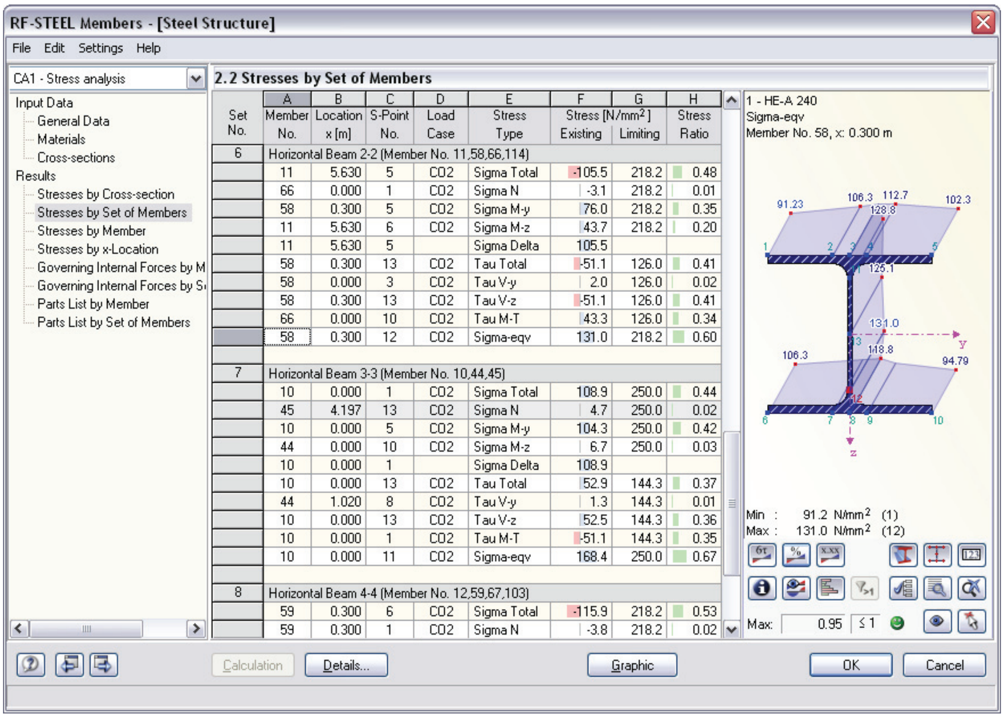


Figure 3.12: Table 2.2 *Stresses by Set of Members*

This results table is only displayed when you have selected at least one set of members for the design in table 1.1 *General Data* (see chapter 3.1.1, page 39). The maximum stress ratios listed in this table are sorted by sets of members.

The advantage of this results output is that the design results of a complete structural group, for example a frame beam, are clearly arranged in one results table, which facilitates the results evaluation.

Details on the columns can be found in the previous chapter 3.3.1. In addition, the column *Member No.* is displayed. It shows the number of the member that bears the maximum stress ratio within the set of members.

3.3.3 Stresses by Member

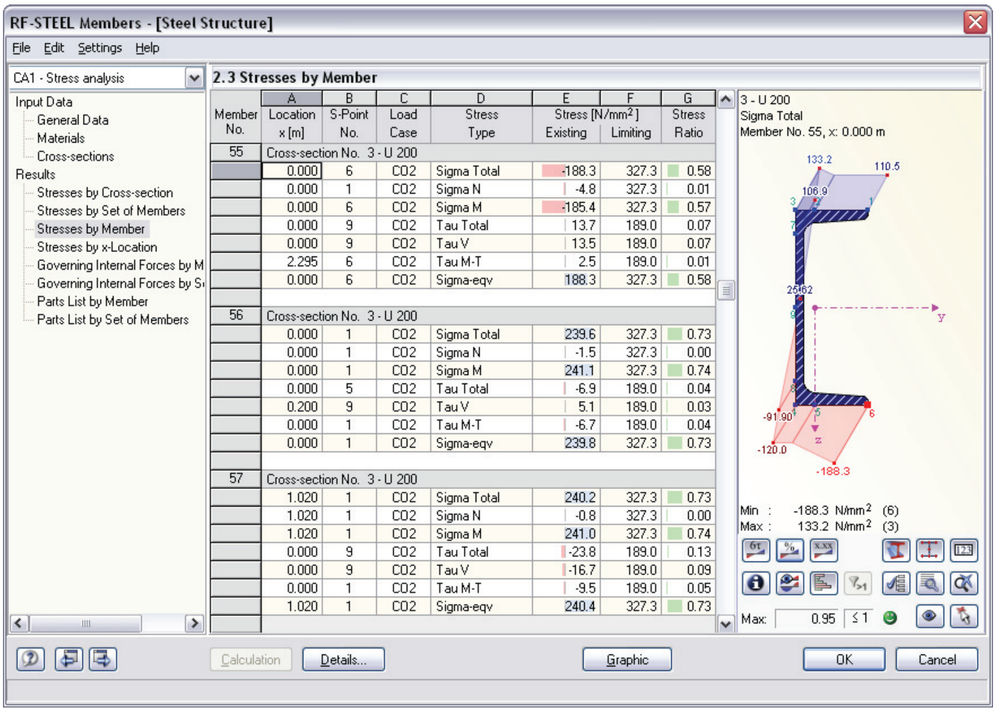


Figure 3.13: Table 2.3 *Stresses by Member*

This results table shows the maximum stress ratios sorted by member numbers. For each member, the program indicates the *Location x* on which the maximum ratio occurs.

The different columns are described in detail in chapter 3.3.1 on page 54.

3.3.4 Stresses by x-Location

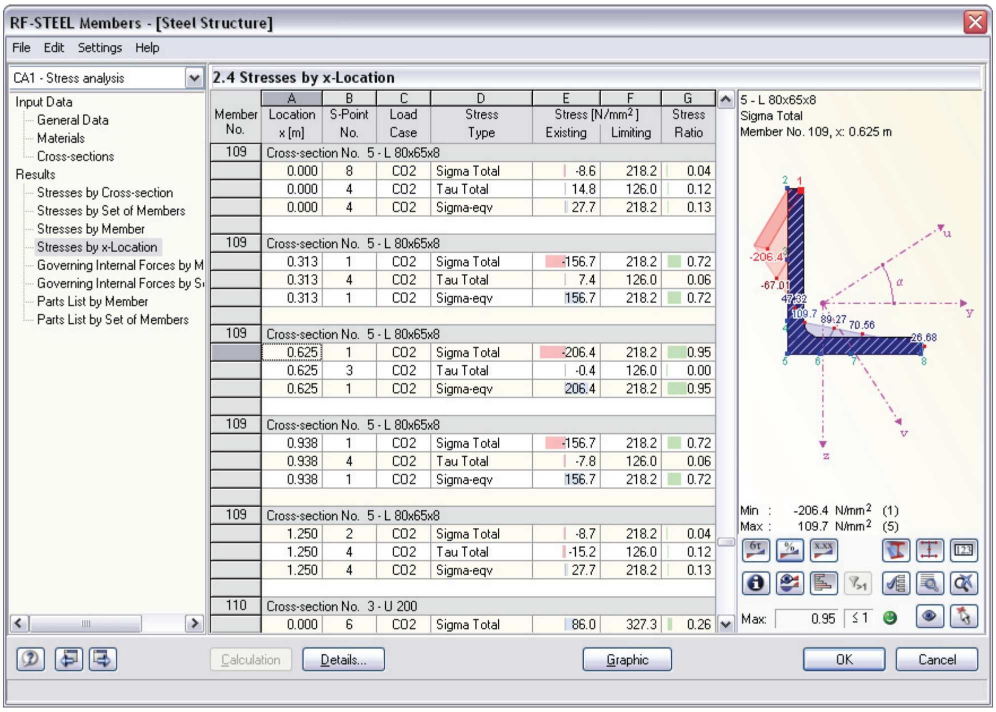


Figure 3.14: Table 2.4 *Stresses by x-Location*

For each member, the table shows the maximum stress ratios on each x-location that is determined by the following types of RFEM divisions:

- Start and end node
- Partition points according to possibly defined member division
- Specification of member division for member results in the *Options* tab of the RFEM dialog box *Calculation Parameters*
- Extreme values of internal forces

3.3.5 Stresses at Every Stress Point

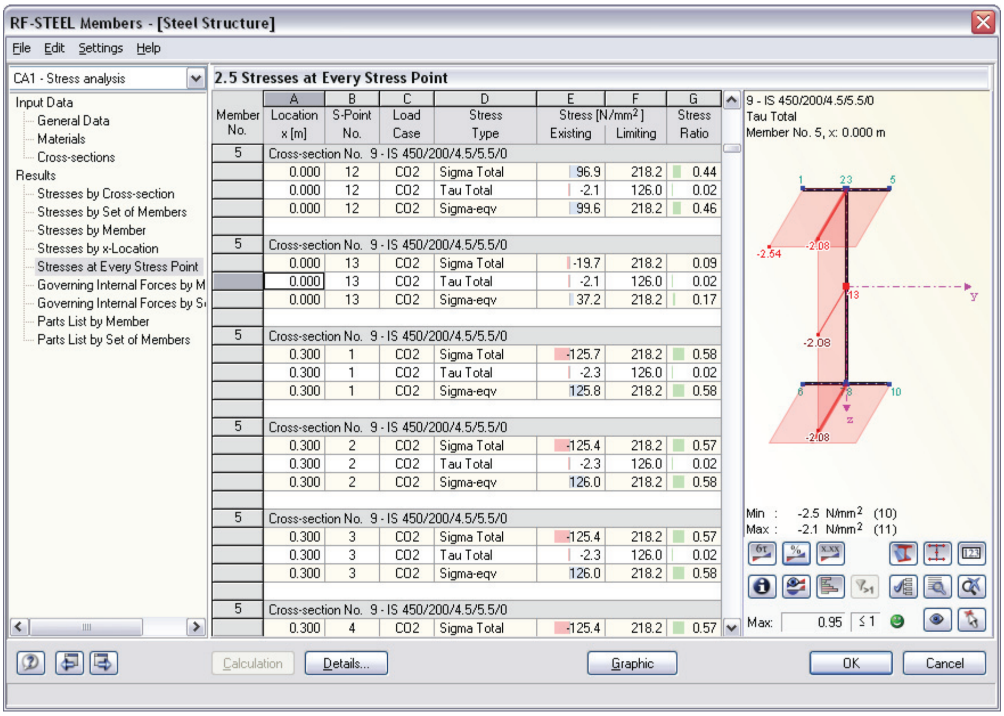


Figure 3.15: Table 2.5 *Stresses at Every Stress Point*

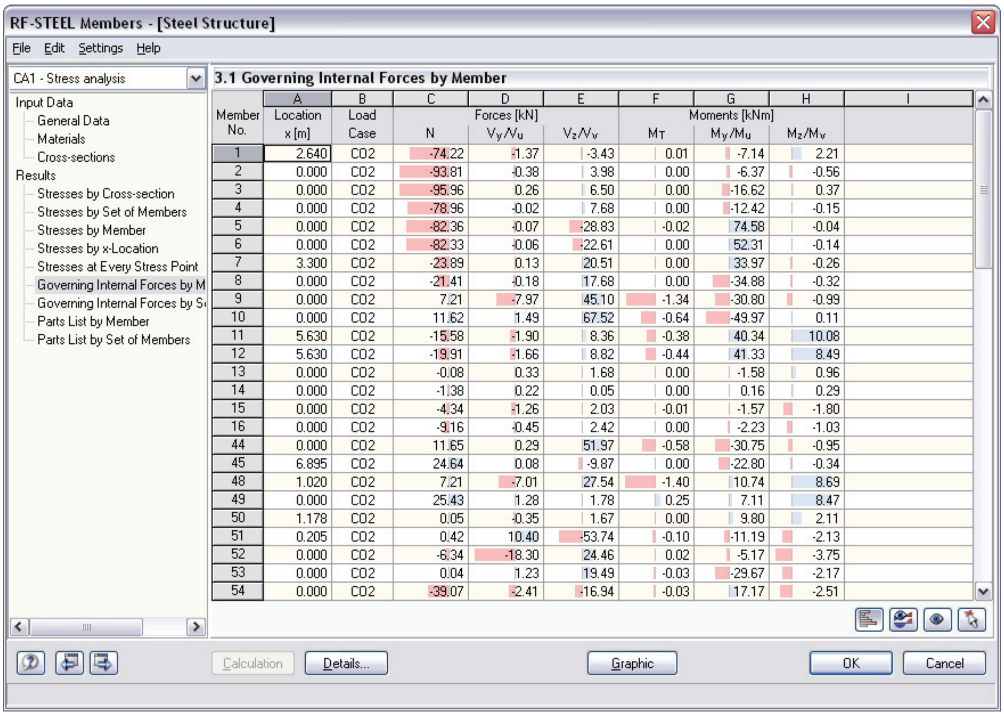
As a results evaluation by stress points is not required in most cases, the table is not displayed by default. However, you can display it by using the *Details* dialog box (see Figure 3.8, page 50). To open the dialog box, use the corresponding button shown on the left.

The display of this results table includes a considerable amount of data. As the maximum stresses, and thus the governing stress points of RF-STEEL Members, are determined automatically, you usually do not need the display of this table.

For each member, the stresses are listed by *Locations x* and stress points. The different columns of the table are described in detail in chapter 3.3.1 on page 54.

As an alternative to this table, specific evaluation options are available for the previous tables, which can be accessed by clicking the button [Cross-section Values and Stress Diagram]. The dialog box *Cross-section Values and Stress Diagram* opens (see Figure 4.9, page 71) where you can evaluate the design results of every single stress point graphically and numerically.

3.3.6 Governing Internal Forces by Member



	A	B	C	D	E	F	G	H	I
Member No.	Location x [m]	Load Case	Forces [kN] N	V_y/V_u	V_z/V_v	Moments [kNm] M_T	M_y/M_u	M_z/M_v	
1	2.640	CO2	-74.22	-1.37	-3.43	0.01	-7.14	2.21	
2	0.000	CO2	-93.81	-0.38	3.98	0.00	-6.37	-0.56	
3	0.000	CO2	-95.96	0.26	6.50	0.00	-16.62	0.37	
4	0.000	CO2	-78.96	-0.02	7.68	0.00	-12.42	-0.15	
5	0.000	CO2	-82.36	-0.07	-28.83	-0.02	74.58	-0.04	
6	0.000	CO2	-82.33	-0.06	-22.61	0.00	52.31	-0.14	
7	3.300	CO2	-23.89	0.13	20.51	0.00	33.97	-0.26	
8	0.000	CO2	-21.41	-0.18	17.68	0.00	-34.88	-0.32	
9	0.000	CO2	7.21	-7.97	45.10	-1.34	-30.80	-0.99	
10	0.000	CO2	11.62	1.49	67.52	-0.64	-49.97	0.11	
11	5.630	CO2	-15.58	-1.90	8.36	-0.38	40.34	10.08	
12	5.630	CO2	-19.91	-1.66	8.82	-0.44	41.33	8.49	
13	0.000	CO2	-0.08	0.33	1.68	0.00	-1.58	0.96	
14	0.000	CO2	-1.38	0.22	0.05	0.00	0.16	0.29	
15	0.000	CO2	-4.34	-1.26	2.03	-0.01	-1.57	-1.80	
16	0.000	CO2	-9.16	-0.45	2.42	0.00	-2.23	-1.03	
44	0.000	CO2	11.65	0.29	51.97	-0.58	-30.75	-0.95	
45	6.895	CO2	24.64	0.08	-9.87	0.00	-22.80	-0.34	
48	1.020	CO2	7.21	-7.01	27.54	-1.40	10.74	8.69	
49	0.000	CO2	25.43	1.28	1.78	0.25	7.11	8.47	
50	1.178	CO2	0.05	-0.35	1.67	0.00	9.80	2.11	
51	0.205	CO2	0.42	10.40	-53.74	-0.10	-11.19	-2.13	
52	0.000	CO2	-6.34	-18.30	24.46	0.02	-5.17	-3.75	
53	0.000	CO2	0.04	1.23	19.49	-0.03	-29.67	-2.17	
54	0.000	CO2	-39.07	-2.41	-16.94	-0.03	17.17	-2.51	

Figure 3.16: Table 3.1 *Governing Internal Forces by Member*

For each member, the table shows the governing constellation of internal forces that leads to the respective maximum stress ratio. By default, the output refers to the equivalent stress σ_{eqv}. In the *Details* dialog box (see Figure 3.8, page 50) that can be accessed by using the [Details] button, you can change the reference for another stress component.

When analyzing load combinations, please note the following concerning the results output: If you select *Calculate Stresses from Single Load Cases in CO and then Combine Stresses according to CO Criteria* (see *Details* dialog box, Figure 3.8, page 50), it is not possible to evaluate the result rows of the RFEM results table 3.5 *Members - Internal Forces* directly. The compressive, tensile and shear stresses resulting from the single load cases are summed up according to the superposition rules and then displayed as total stresses in the results tables. The equivalent stress σ_{eqv}, however, is determined with the components of σ_{total} and τ_{total} according to Table 3.4 on page 49. Thus, the internal forces displayed in table 3.1 are not directly transparent, if they are related to the equivalent stresses.

Location x

The column shows the respective x-location for which the program has determined the member's maximum stress ratio.

Load Case

This column indicates the numbers of the load case, load group or load 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.

3.3.7 Governing Internal Forces by Set of Member

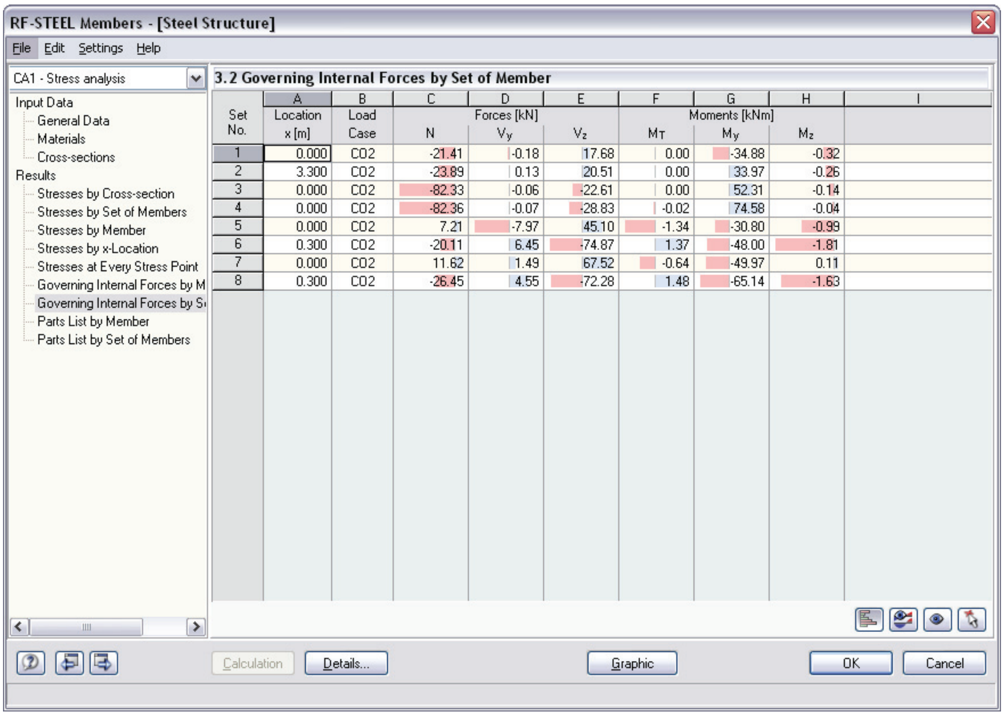
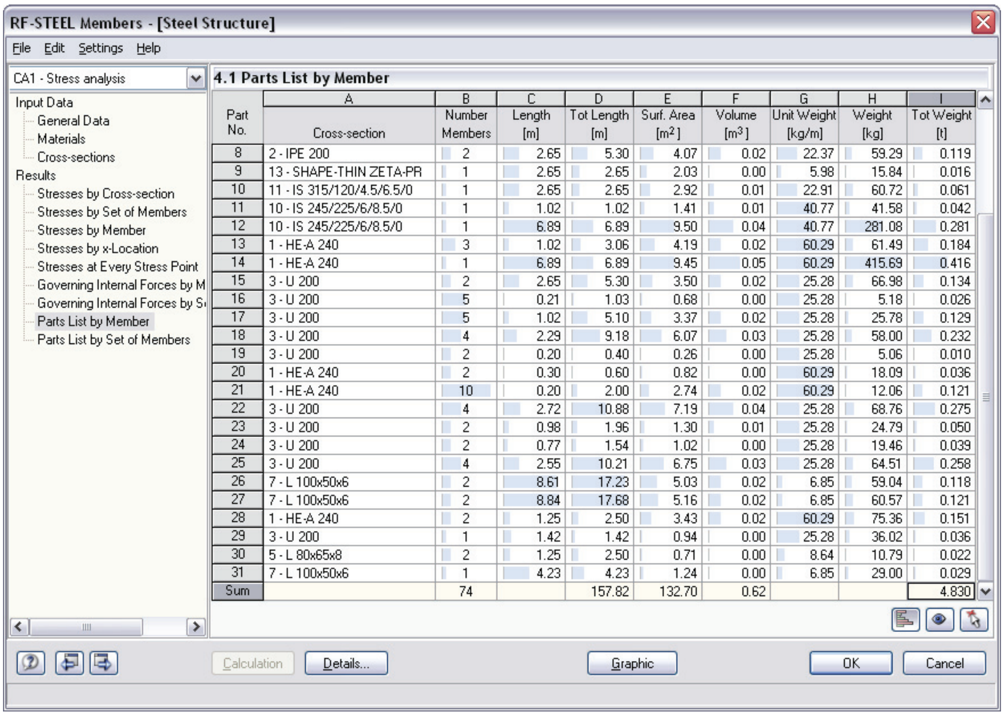


Figure 3.17: Table 3.2 *Governing Internal Forces by Set of Member*

This results table is only displayed when you have selected at least one set of members for the design in table 1.1 *General Data* (see chapter 3.1.1, page 39). The governing internal forces are sorted by sets of members.

3.3.8 Parts List by Member



Part No.	A Cross-section	B Number Members	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]
8	2 - IPE 200	2	2.65	5.30	4.07	0.02	22.37	59.29	0.119
9	13 - SHAPE-THIN ZETA-PR	1	2.65	2.65	2.03	0.00	5.98	15.84	0.016
10	11 - IS 315/120/4.5/6.5/0	1	2.65	2.65	2.92	0.01	22.91	60.72	0.061
11	10 - IS 245/225/6/8.5/0	1	1.02	1.02	1.41	0.01	40.77	41.58	0.042
12	10 - IS 245/225/6/8.5/0	1	6.89	6.89	9.50	0.04	40.77	281.08	0.281
13	1 - HE-A 240	3	1.02	3.06	4.19	0.02	60.29	61.49	0.184
14	1 - HE-A 240	1	6.89	6.89	9.45	0.05	60.29	415.69	0.416
15	3 - U 200	2	2.65	5.30	3.50	0.02	25.28	66.98	0.134
16	3 - U 200	5	0.21	1.03	0.68	0.00	25.28	5.18	0.026
17	3 - U 200	5	1.02	5.10	3.37	0.02	25.28	25.78	0.129
18	3 - U 200	4	2.29	9.18	6.07	0.03	25.28	58.00	0.232
19	3 - U 200	2	0.20	0.40	0.26	0.00	25.28	5.06	0.010
20	1 - HE-A 240	2	0.30	0.60	0.82	0.00	60.29	18.09	0.036
21	1 - HE-A 240	10	0.20	2.00	2.74	0.02	60.29	12.06	0.121
22	3 - U 200	4	2.72	10.88	7.19	0.04	25.28	68.76	0.275
23	3 - U 200	2	0.98	1.96	1.30	0.01	25.28	24.79	0.050
24	3 - U 200	2	0.77	1.54	1.02	0.00	25.28	19.46	0.039
25	3 - U 200	4	2.55	10.21	6.75	0.03	25.28	64.51	0.258
26	7 - L 100x50x6	2	8.61	17.23	5.03	0.02	6.85	59.04	0.118
27	7 - L 100x50x6	2	8.84	17.68	5.16	0.02	6.85	60.57	0.121
28	1 - HE-A 240	2	1.25	2.50	3.43	0.02	60.29	75.36	0.151
29	3 - U 200	1	1.42	1.42	0.94	0.00	25.28	36.02	0.036
30	5 - L 80x65x8	2	1.25	2.50	0.71	0.00	8.64	10.79	0.022
31	7 - L 100x50x6	1	4.23	4.23	1.24	0.00	6.85	29.00	0.029
Sum		74		157.82	132.70	0.62			4.830

Figure 3.18: Table 4.1 *Parts List by Member*

Finally, RF-STEEL Members provides a summary of all cross-sections that are included in the design case. 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 3.8, page 50).

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 respective length of an individual member.

Tot Length

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

Surf. Area

For each part, the program indicates the surface area related to the total length. The surface area is determined from the *Surface* of the cross-sections. You can find the relevant entry in the cross-section information, available in table 1.3 to 2.5 (see Figure 3.11, page 55).

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.

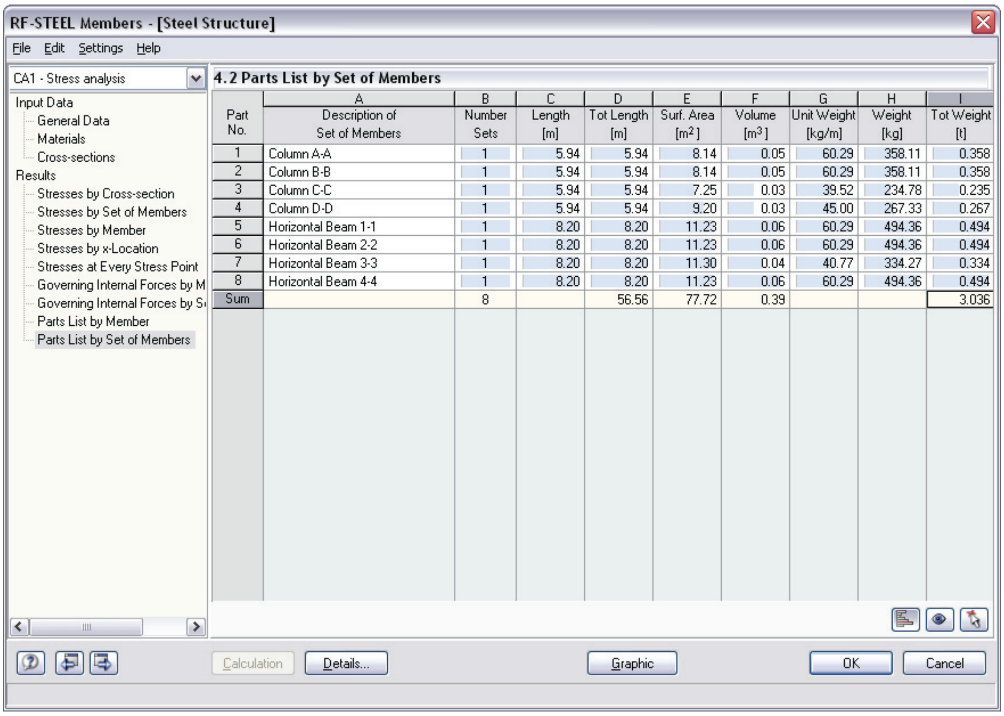
Tot Weight

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

Sum

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

3.3.9 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	Column A-A	1	5.94	5.94	8.14	0.05	60.29	358.11	0.358
2	Column B-B	1	5.94	5.94	8.14	0.05	60.29	358.11	0.358
3	Column C-C	1	5.94	5.94	7.25	0.03	39.52	234.78	0.235
4	Column D-D	1	5.94	5.94	9.20	0.03	45.00	267.33	0.267
5	Horizontal Beam 1-1	1	8.20	8.20	11.23	0.06	60.29	494.36	0.494
6	Horizontal Beam 2-2	1	8.20	8.20	11.23	0.06	60.29	494.36	0.494
7	Horizontal Beam 3-3	1	8.20	8.20	11.30	0.04	40.77	334.27	0.334
8	Horizontal Beam 4-4	1	8.20	8.20	11.23	0.06	60.29	494.36	0.494
Sum		8		56.56	77.72	0.39			3.036

Figure 3.19: Table 4.2 *Parts List by Set of Members*

The final results table is only displayed when you have selected at least one set of members for the design in table 1.1 *General Data* (see chapter 3.1.1, page 39). The advantage of the output by sets of members is the display of a summarized parts list for particular structural groups (for example a frame beam).

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

4. Results Evaluation

The results of the stress analysis can be evaluated in numerous ways. A specific evaluation is already available due to the data arrangement in the tables. They present the results sorted according to particular criteria. In addition, the tables provide useful **buttons**.

RF-STEEL Surfaces

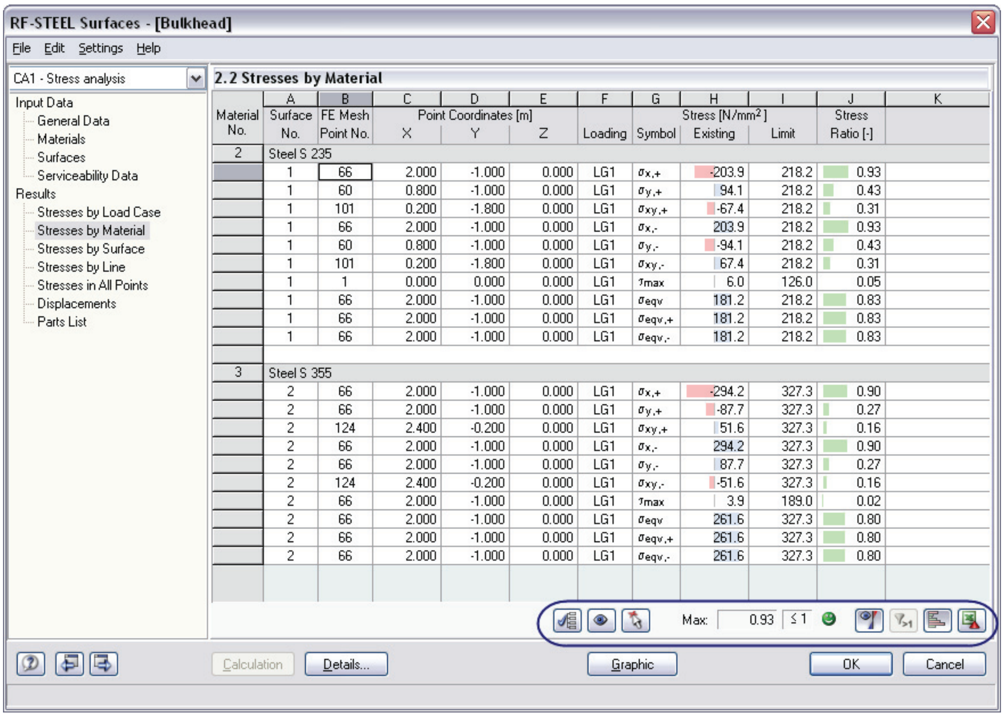


Figure 4.1: Buttons for results evaluation in the add-on module **RF-STEEL Surfaces**

The buttons are reserved for the following functions:

Button	Description	Function
	Details	Opens the *Details* dialog box to select the displayed stresses ➝ chapter 4.1.1, page 66
	View mode	Opens the RFEM work window for a graphical check without closing RF-STEEL ➝ chapter 4.1.2, page 67
	Object selection	Opens the RFEM work window to select a surface or line graphically
	Result distribution	Displays the results of the currently selected table row in the RFEM background graphic
	Exceeding	Displays only the table rows where the ratio is more than 1
	Color bars	Turns on and off the display of the colored relation scales in the results tables
	Excel	Exports the content of the current table to MS Excel or OpenOffice.org Calc ➝ chapter 6.4, page 87

Table 4.1: Buttons of results tables in RF-STEEL Surfaces

RF-STEEL Members

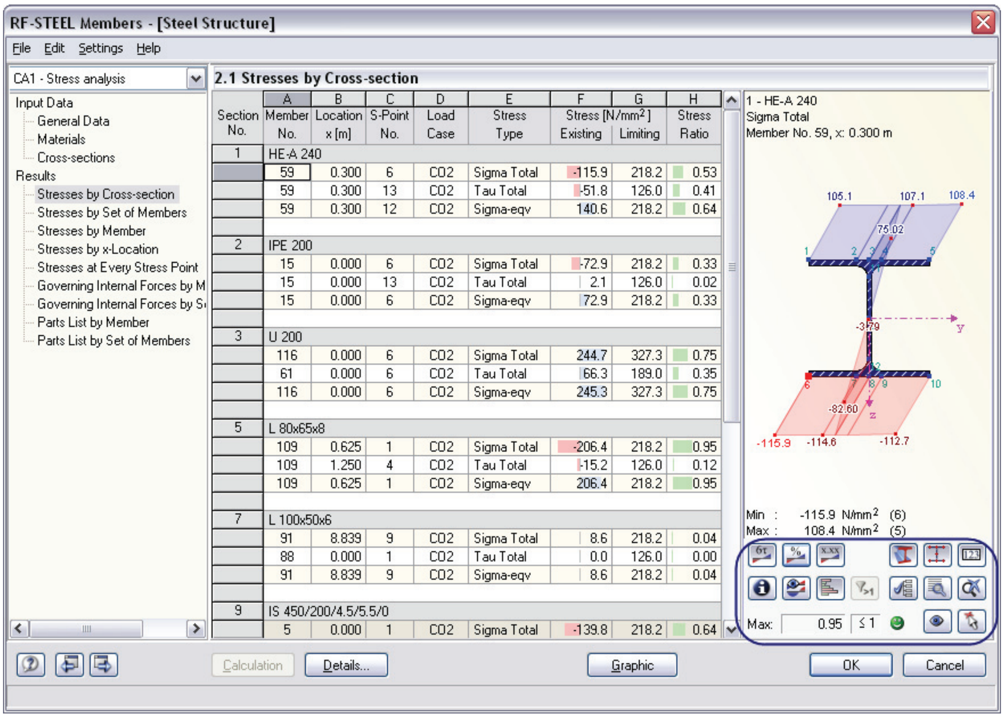
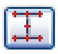


Figure 4.2: Buttons for results evaluation in the add-on module **RF-STEEL Members**

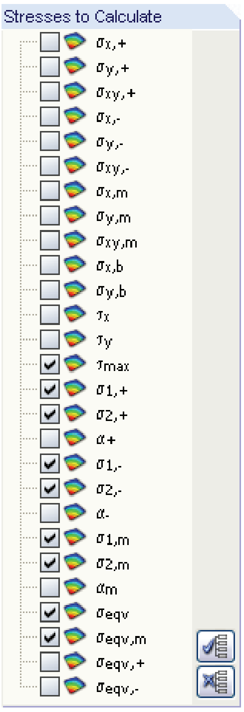
Button	Description	Function
	Stress diagram	Displays the stress diagram in the cross-section graphic above
	Stress ratio	Turns on and off the graphic display of the stress ratios, as an alternative to the stress diagram
	Values	Displays the values in the cross-section graphic
	Cross-section outlines	Displays the cross-section's shape in the cross-section graphic
	Stress points	Displays the stress points in the cross-section graphic
	Numbering	Turns on and off the numbering of stress points in the cross-section graphic
	Cross-section info	Opens the dialog box *Info about cross-section* showing the properties of the currently selected cross-section → chapter 4.2.2, page 71
	Result diagrams	Opens the window *Result Diagram on Member* → chapter 4.2.4, page 75
	Color bars	Turns on and off the display of the colored scales in the results tables
	Exceeding	Displays only the rows where the ratio is more than 1 and the design is failed
	Stress selection	Opens the dialog box *Stresses - Filter* → chapter 4.2.1, page 69

	Detail display	Opens the dialog box *Cross-section Values and Stress Diagram* ➙ chapter 4.2.2, page 71
	Full view	Resets the stress graphic in full view (zooming by wheel button, moving by drag-and-drop)
	View mode	Opens the RFEM work window for a graphical check without closing RF-STEEL ➙ chapter 4.2.3, page 72
	Object selection	Opens the RFEM work window to select a member graphically

Table 4.2: Buttons of results tables in RF-STEEL Members

4.1 RF-STEEL Surfaces

4.1.1 Selection of Stresses



The following stress types are displayed by default when the design has been carried out:

- Shear stresses τ_{max}
- Principal stresses σ in direction of the principal axes on the surface's top and bottom side
- Membrane stresses σ_m in direction of the principal axes
- Equivalent stress σ_{eqv}
- Membrane equivalent stress $\sigma_{eqv,m}$

Use the [Details] button to select the stress types that you want to be displayed.

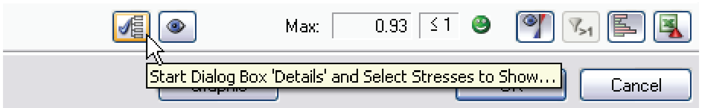


Figure 4.3: Button [Start Dialog Box 'Details' and Select Stresses to Show]

The *Details* dialog box appears (see Figure 2.11, page 19). In the *Stresses* tab, you can tick or clear the check boxes of the corresponding stress components that you want to display in the results table. All available stress types are shown on the left.

The different stress components and their formulas are described in detail in chapter 2.2.1.1, page 20.

The settings in the *Details* dialog box influence the results tables as well as the printout report. The stress components displayed in the tables can also be found in the printout. As you cannot select the stress types in the printout report (see chapter 5.1, page 78), you must specify the corresponding settings in the *Details* dialog box.

4.1.2 Results in the RFEM Model

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

Background graphic and view mode

The RFEM graphic in the background may be useful when you want to check the position of a particular surface in the model. An arrow in the RFEM background graphic indicates the location of the FE mesh or grid point that you have selected in the results table. When the button [Show Current Results] is set active, the current stresses are displayed additionally.

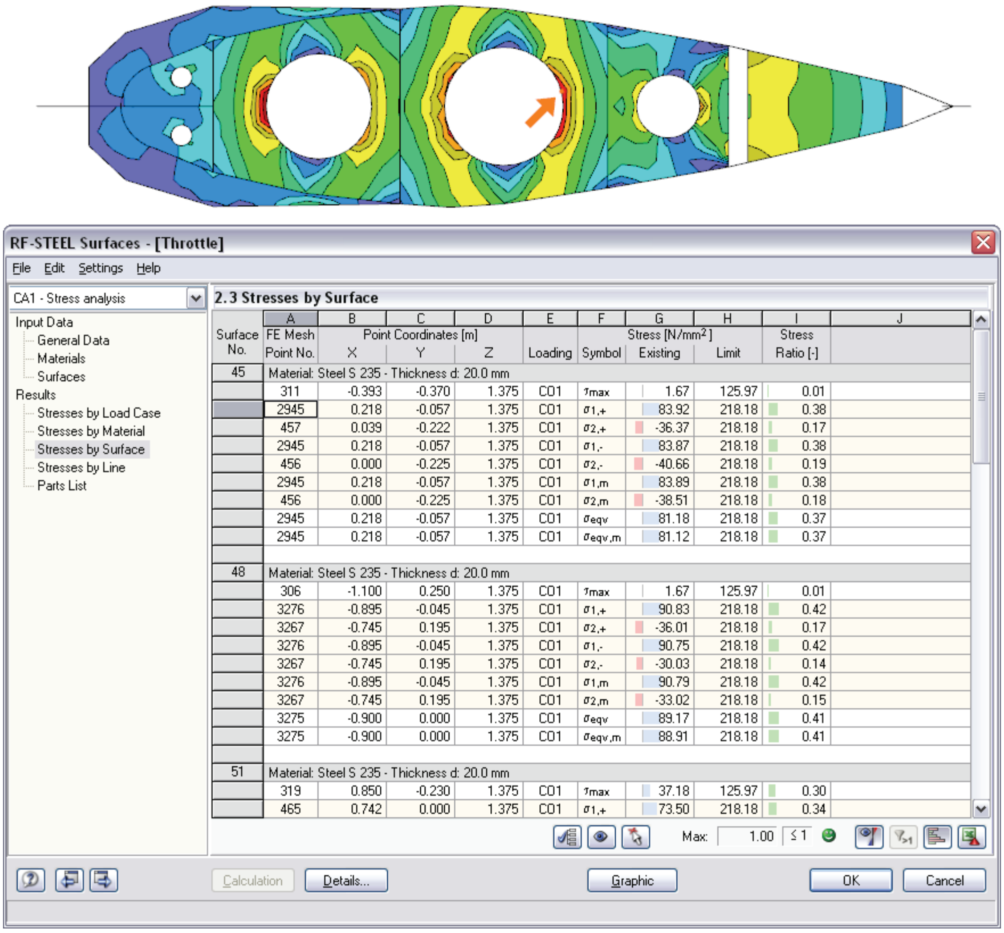
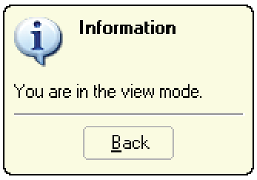


Figure 4.4: Indication of the current FE mesh point in the RFEM model



In case the graphic view cannot be displayed appropriately by moving the RF-STEEL module window, use the button [Jump to Graphics] to change to the *view mode*. The program hides the RF-STEEL window so that you can modify the display in the RFEM user interface. The view mode only provides the functions of the *View* menu, for example zooming, moving or rotating the display, as well as the partial view options.

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

RFEM work window

All stresses and stress ratios can be visualized directly in the structural model. Use the [Graphic] button to quit the RF-STEEL module. Now, you can evaluate the results, such as the internal forces of a load case or a load group, in the RFEM work window graphically.

The *Results* navigator is adjusted to the design results from RF-STEEL. In the graphic, you can display and check the different stress types as well as the corresponding stress ratios for each of the designed actions.

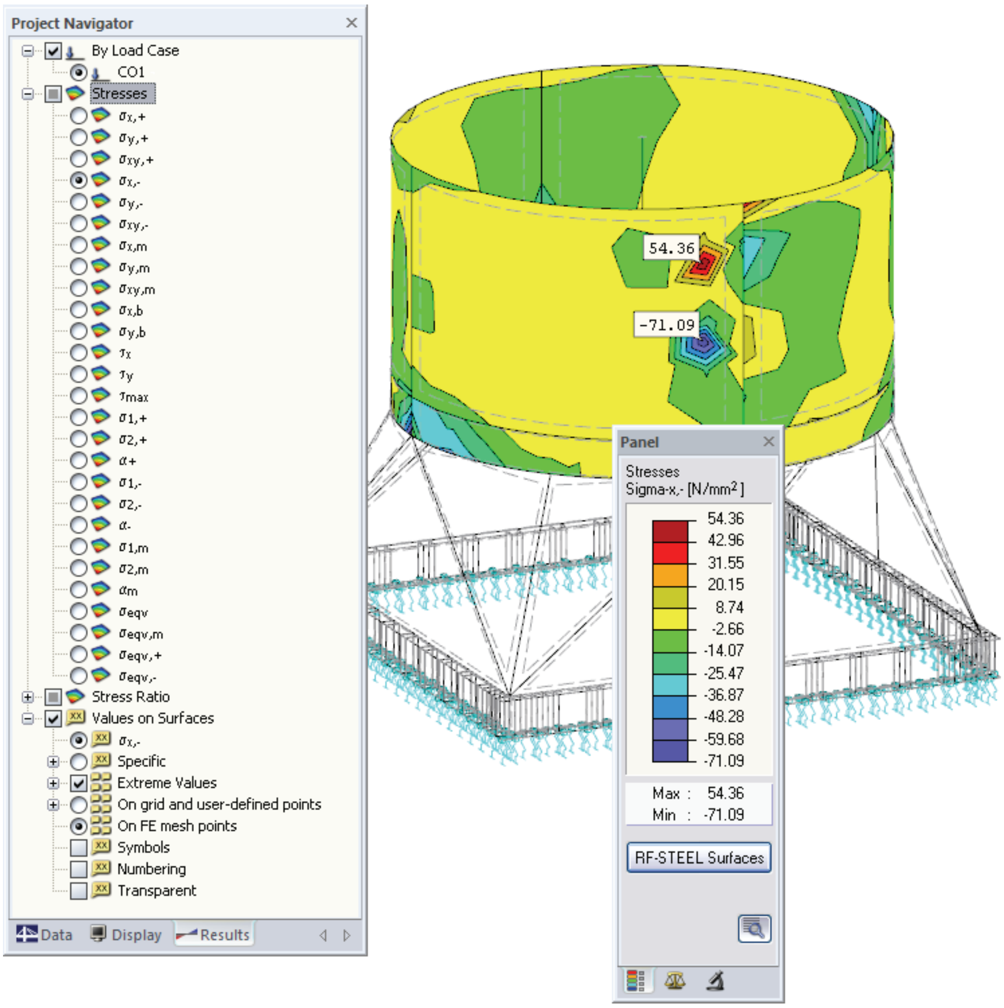
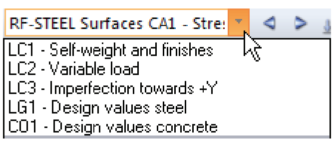


Figure 4.5: *Results* navigator to select stresses and stress ratios from designed actions

To turn the display of design results on or off, use the button [Results on/off], as you know it from the RFEM internal forces.

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



The design cases are selected as usual by means of the list in the RFEM menu bar.

To display the surface results and the result values, you can use all options that are available in RFEM. The corresponding display functions are described in detail in chapter 10.4 of the RFEM manual.

It is always possible to return to the RF-STEEL add-on module by clicking the button [RF-STEEL Surfaces] in the panel.

4.2 RF-STEEL Members

4.2.1 Selection of Stresses

The following stresses are displayed by default in the table display of the add-on module RF-STEEL Members:

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

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

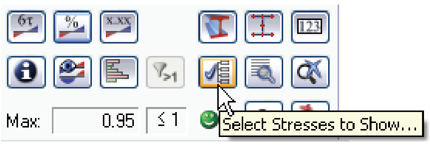


Figure 4.6: Button [Select Stresses to Show]

The following dialog box appears:

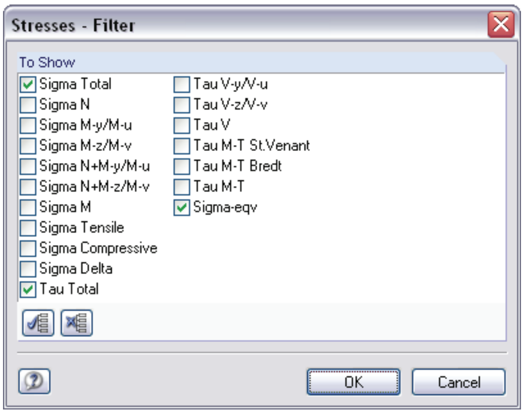


Figure 4.7: Dialog box *Stresses - Filter*

In this dialog box, you can define the relevant stress components. The different stresses are described in Table 3.2 and Table 3.3 on page 47 and 49.

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

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

Table 4.3: Buttons in the dialog box *Stresses - Filter*

The analysis is carried out for each single stress point so that normally the components of the maximum stresses must <u>not</u> 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 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. However, it is not possible to select stress types in the printout report (see chapter 5.1, page 78).

4.2.2 Results on the Cross-section

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

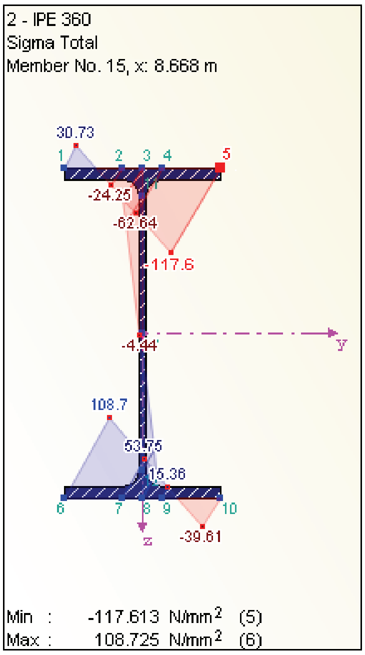
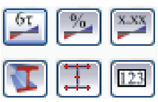


Figure 4.8: Diagram of normal stresses on cross-section

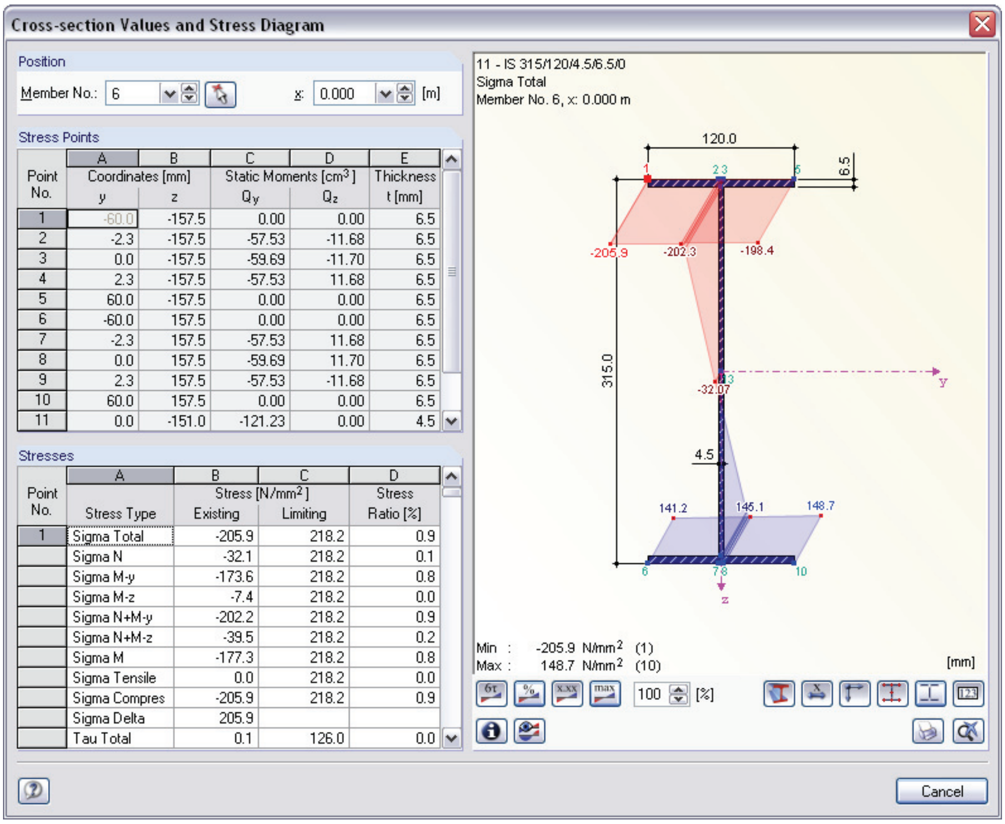
Use the wheel button to maximize or minimize the display. Use the drag-and-drop function to move the stress graphic. The button shown on the left resets the graphic's full view.

The functions of the buttons below the graphic are described in Table 4.2 on page 66. Use these buttons to display



- 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 specifically, use the button [Show or Print Cross-section Values and Extended Stress Diagram]. The dialog box *Cross-section Values and Stress Diagram* opens (see Figure 4.9).



Point No.	A: Coordinates [mm] y	B: z	C: Static Moments [cm³] Q_y	D: Q_z	E: Thickness t [mm]
1	-60.0	-157.5	0.00	0.00	6.5
2	-2.3	-157.5	-57.53	-11.68	6.5
3	0.0	-157.5	-59.69	-11.70	6.5
4	2.3	-157.5	-57.53	11.68	6.5
5	60.0	-157.5	0.00	0.00	6.5
6	-60.0	157.5	0.00	0.00	6.5
7	-2.3	157.5	-57.53	11.68	6.5
8	0.0	157.5	-59.69	11.70	6.5
9	2.3	157.5	-57.53	-11.68	6.5
10	60.0	157.5	0.00	0.00	6.5
11	0.0	-151.0	-121.23	0.00	4.5

Point No.	A: Stress Type	B: Stress [N/mm²] Existing	C: Limiting	D: Stress Ratio [%]
1	Sigma Total	-205.9	218.2	0.9
	Sigma N	-32.1	218.2	0.1
	Sigma M-y	-173.6	218.2	0.8
	Sigma M-z	-7.4	218.2	0.0
	Sigma N+M-y	-202.2	218.2	0.9
	Sigma N+M-z	-39.5	218.2	0.2
	Sigma M	-177.3	218.2	0.8
	Sigma Tensile	0.0	218.2	0.0
	Sigma Compres	-205.9	218.2	0.9
	Sigma Delta	205.9		
	Tau Total	0.1	126.0	0.0

Figure 4.9: 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 all stress points of the cross-section. The two *Coordinates* columns show the respective centroidal distances *y* and *z*. The *Static Moments* columns display the corresponding static moments Q_y and Q_z. The final column contains the values of the *Thickness t* of the respective cross-section element. This thickness is relevant for the determination of shear stress.

In the *Stresses* dialog section, the stress components are displayed in detail for the stress point that is currently selected in the dialog section above. To visualize a particular stress type 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 4.2, page 66). As usual, they are described by *ScreenTips*. A special feature is provided by the [Print] button that enables the printout of the current stress graphic on the cross-section. For more information, see chapter 5.2.2, on page 80.

4.2.3 Results in the RFEM Model

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

Background graphic and view mode

The RFEM 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 results table is highlighted in the RFEM background graphic in the selection color. In addition, an arrow indicates the current x-location on the member.

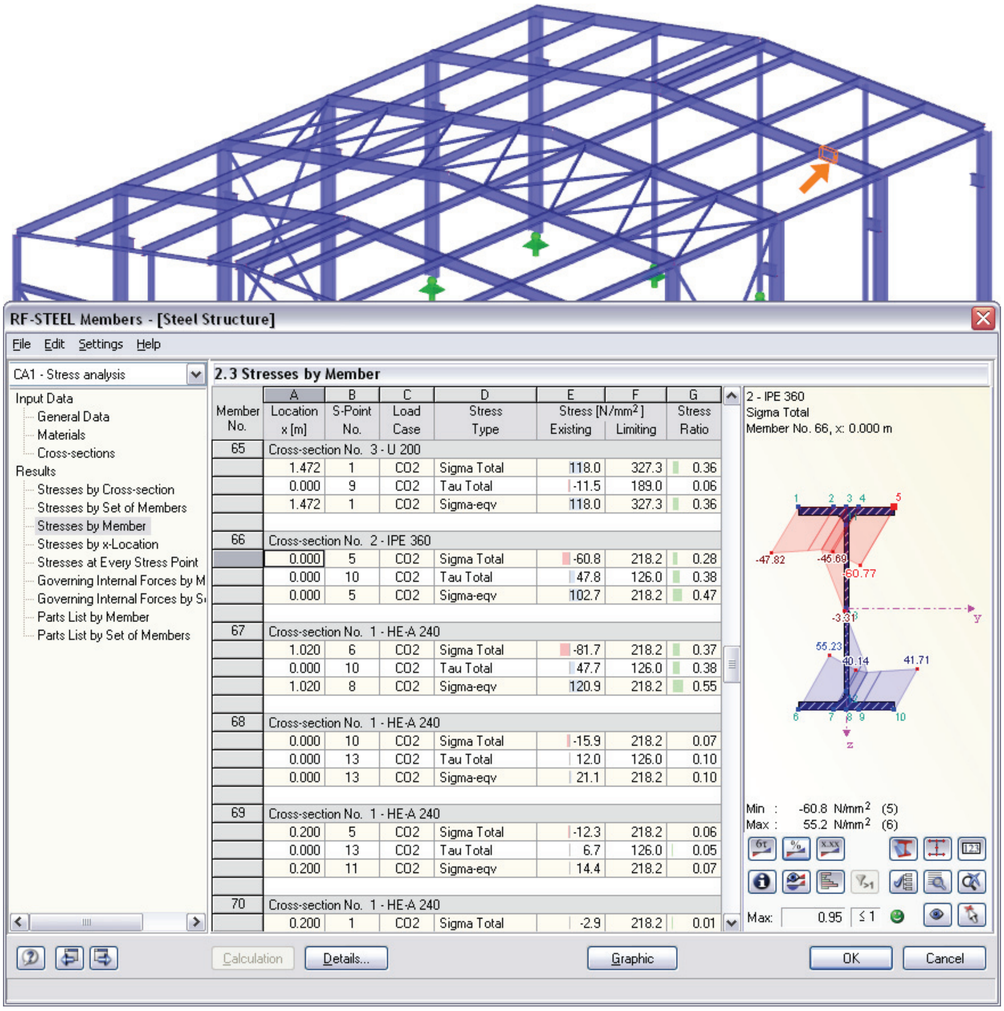
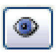
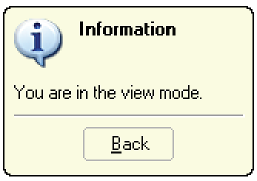


Figure 4.10: Indication of the member and the current *Location x* in the RFEM model



In case the graphic view cannot be displayed appropriately by moving the RF-STEEL module window, use the button [Jump to Graphics] to change to the *view mode*. The program hides the RF-STEEL window so that you can modify the display in the RFEM user interface. The view mode only provides the functions of the *View* menu, for example zooming, moving or rotating the display, as well as the partial view options.



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

RFEM work window

All stresses and stress ratios can be visualized directly in the structural model. Use the [Graphic] button to quit the RF-STEEL module. Now, you can evaluate the results, such as the internal forces of a load case or a load group, in the RFEM work window graphically.

The *Results* navigator is adjusted to the design results from RF-STEEL Members. You can select the individual stress components as well as the stress ratios in relation to the respective stress components.

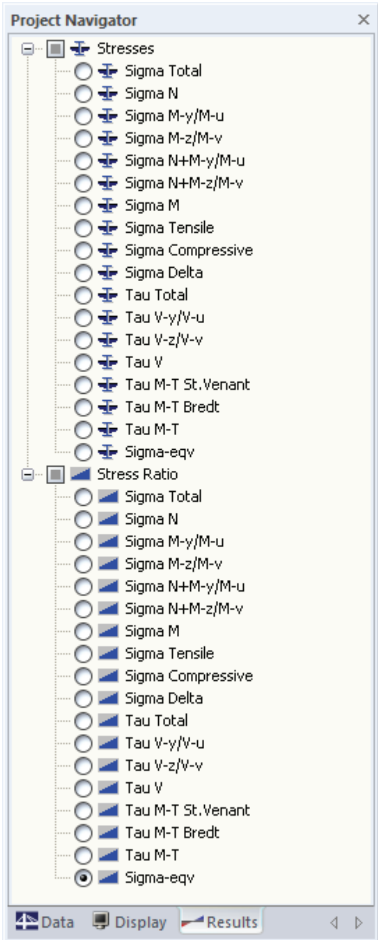
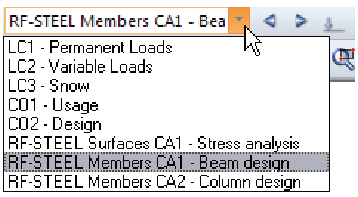


Figure 4.11: *Results* navigator of RF-STEEL Members

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

As the RFEM tables are of no relevance for the evaluation of stresses and stress ratios, you may deactivate them.



The design cases are selected as usual by means of the list in the RFEM menu bar.

Stresses and stress ratios are two colored by default. To adjust the results display, use the *Display* navigator, select *Results* and then *Members*.

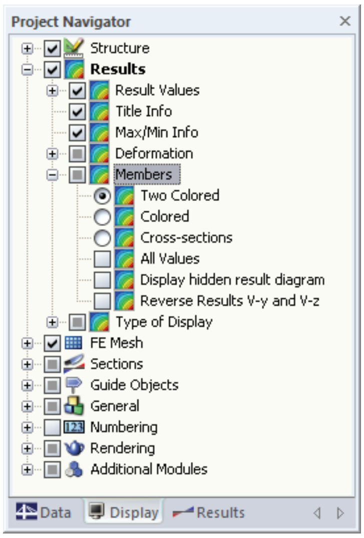
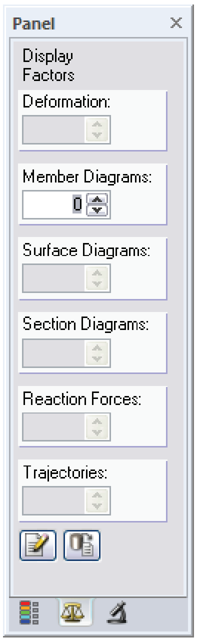


Figure 4.12: *Display* navigator: Results → Members

The stresses are represented with two colors, conforming to the signs. Positive stresses are displayed in blue, in direction of the positive member axis *z*. Negative stresses are red, applied in opposite direction. Thus, it is possible that the stress diagram on the member, in case of discontinuity, changes the sign and therefore the color and member side.



In case of a multicolor display representation, a panel is available, providing the common control functions. The panel functions are described in detail in the RFEM manual, chapter 4.4.6, page 77. In the *Filter* tab shown on the left, 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 stresses and stress ratios will be represented directly on the member axes with an increased line thickness.

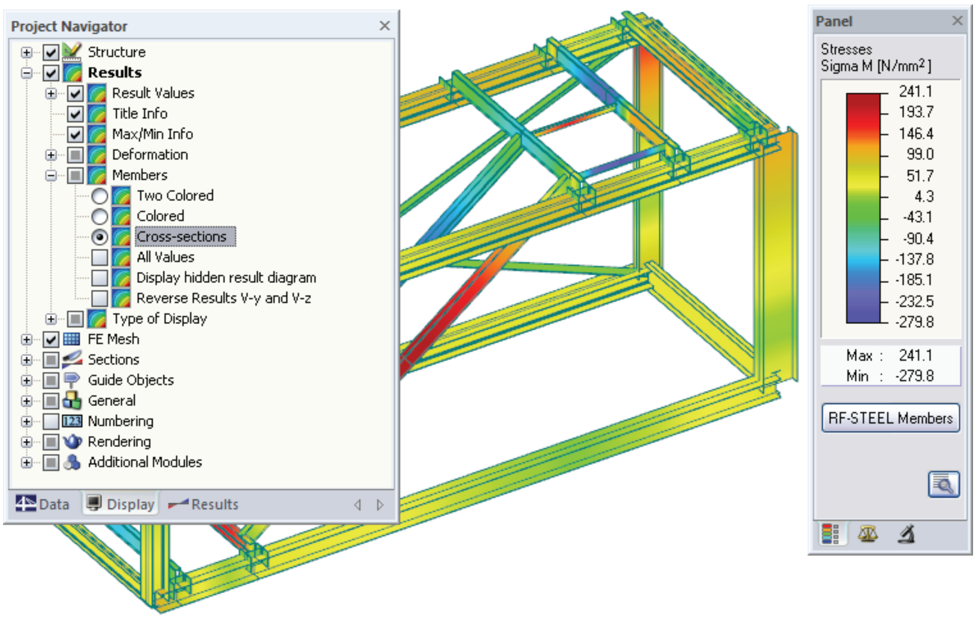


Figure 4.13: Stresses due to bending moments M with display option *Cross-sections*

It is always possible to return to the RF-STEEL add-on module by clicking the button [RF-STEEL Members] in the panel.

4.2.4 Result Diagrams

To display the distribution of stresses and stress ratios for a particular member, you can also use the result diagram you already know from RFEM. First, select the member or set of member in the results table of RF-STEEL Members, and then click the button below the stress graphic shown on the left.

Alternatively, the result diagrams are available in the RFEM graphic. To display the diagrams,

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

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

A window opens that shows the stress and stress ratio diagrams on the selected member or set of members.

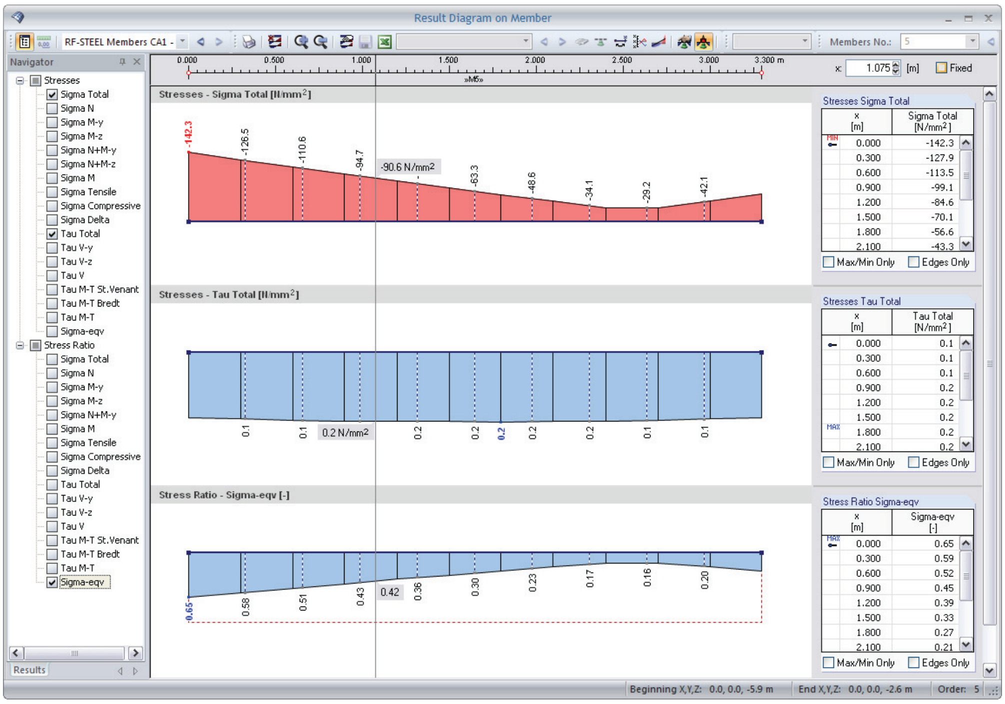


Figure 4.14: Dialog box *Result Diagram on Member*

In the navigator on the left, define the types of stresses and stress ratios that you want to be displayed in the result diagram. Use the lists in the toolbar above to choose a particular design case or member.

For more detailed information on the dialog box *Result Diagram on Member*, see the RFEM manual, chapter 10.5, page 311.

4.3 Filter for Results

Due to their data arrangement, the RF-STEEL results tables already provide a specific selection according to certain criteria. In some detail tables, you find additional filter functions for structural objects and actions (see Figure 2.21, page 34).

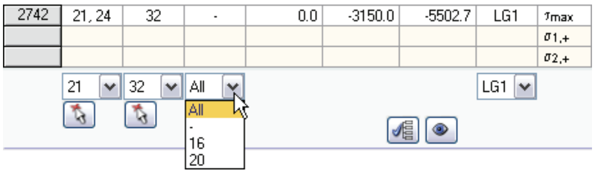


Figure 4.15: Filtering structural objects and actions in table 2.5

In principle, you can use all filter functions described in the RFEM manual for the graphical evaluation of RF-STEEL design results. For example, you can activate or recreate partial views already defined (cf. RFEM manual, chapter 10.9, page 321) containing selected objects that are arranged appropriately. You may also use sections (cf. RFEM manual, chapter 10.6, page 313).

In the RFEM work window, you can also use the stresses and stress ratios as filter criteria. The required specifications are defined in the panel. If the panel is not displayed, select *Control Panel* on the *View* menu in RFEM, or use the corresponding button in the *Results* toolbar.

The panel is described in the RFEM manual, chapter 4.4.6, page 77. The filter settings for the results are defined by the color spectrum in the first panel tab. As this tab is not available for the two colored results display of members, you have to set the display option *Colored* or *Cross-sections* in the *Display* navigator (see Figure 4.13, page 74).

You can use the panel, for example, to specify that only stresses higher than 100 N/mm^2 are displayed. Furthermore, the color spectrum can be modified in such a way that a color range covers exactly 10 N/mm^2. In this way, you may handle singularity effects for the documentation.

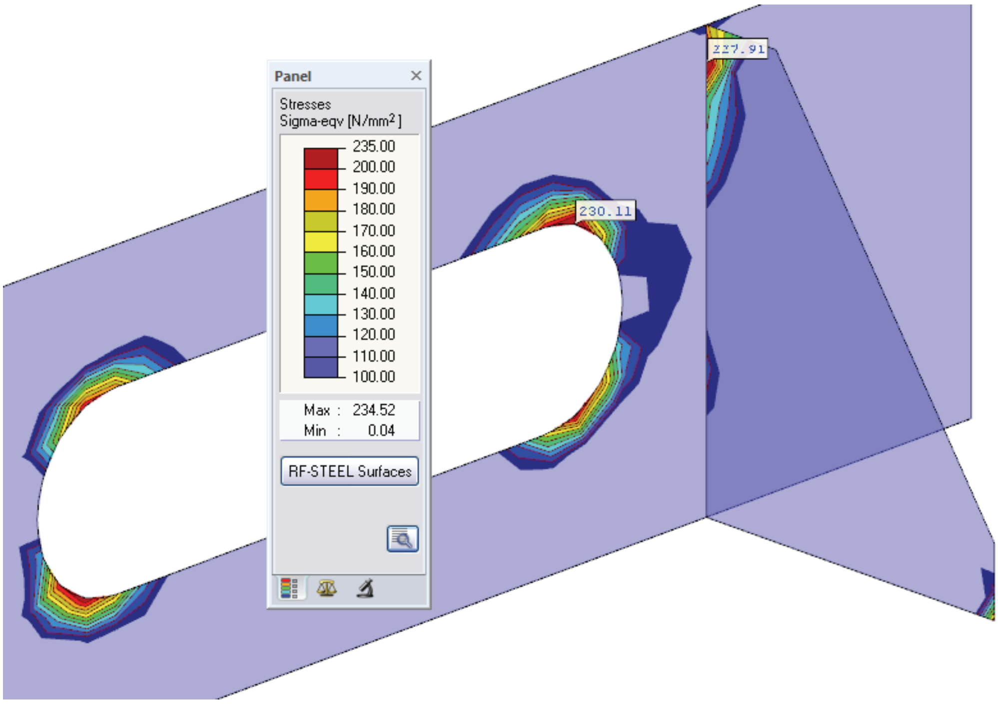


Figure 4.16: Filtering equivalent stresses with adjusted color spectrum

Filtering surfaces and members

In the *Filter* tab of the control panel, you can define the numbers of the surfaces or members whose results diagrams should be displayed exclusively in the graphic. For more detailed information, see the RFEM manual, chapter 10.9, page 321.

In contrast to the partial view function, the structural model is now displayed completely in the graphic. The following figure shows the equivalent stresses of an internal hall frame. All the other designed members are displayed in the model, but are shown without stresses.

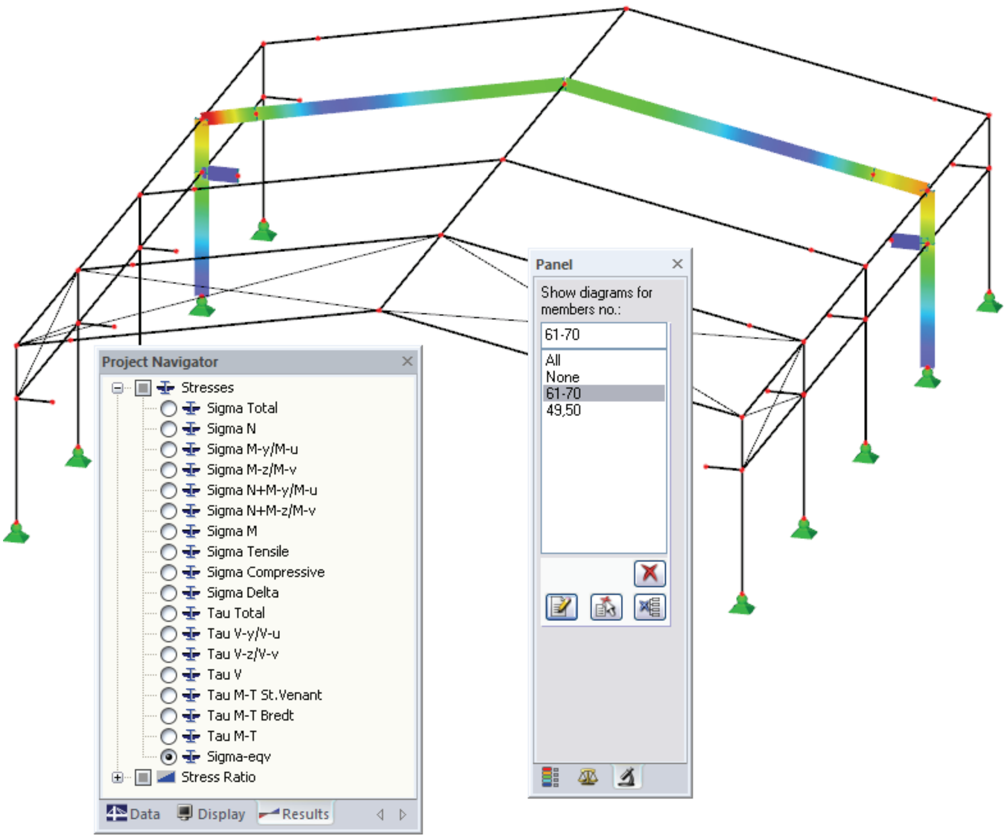


Figure 4.17: Filtering members: equivalent stresses of a frame

5. Printout

5.1 Printout Report

As you know it from RFEM, the program generates a printout report for the RF-STEEL design results that can be added by graphics and descriptions. In this report, you also decide which results tables from RF-STEEL appear in the printout.

The printout report is described in detail in the RFEM manual. In particular, chapter 11.1.3.4 *Selecting Data of Add-on Modules* on page 338 provides information concerning the selection of input and output data in add-on modules.

When your structure is quite extensive, it is advisable to split the data into several small reports. For example, if you create a new printout report only for the data of a single design case from RF-STEEL Surfaces, this printout report will be generated relatively quickly.

The printout report shows the stress components that are active in the results tables of the corresponding RF-STEEL design case. For example, if you want to show the stresses due to axial force in the printout, you have to activate the stresses $\sigma_{1,m}$ and $\sigma_{2,m}$ in RF-STEEL Surfaces. In RF-STEEL Members, you would have to select the stresses σ_N. The corresponding functions are described in chapter 4.1 *RF-STEEL Surfaces* on page 66 and 4.2 *RF-STEEL Members* on page 69.

5.2 Print RF-STEEL Graphics

As described in chapter 4 *Results Evaluation*, all stresses and stress ratios can be evaluated graphically. It is also possible to document each graphic by integrating it in the printout report or sending it directly to the printer.

Printing graphics is described in detail in the RFEM manual, chapter 11.2, page 354.

5.2.1 Results in the RFEM Model

Every picture that is displayed in the graphic window of the main program RFEM can be integrated in the printout report. The result diagrams of sections and members can be imported to the report as well by using the [Print] button.

To print the current RF-STEEL graphic in the RFEM work window,

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

or use the toolbar button shown on the left.

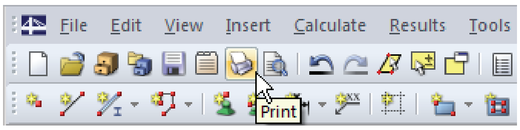


Figure 5.1: Button *Print* in the toolbar of the main window

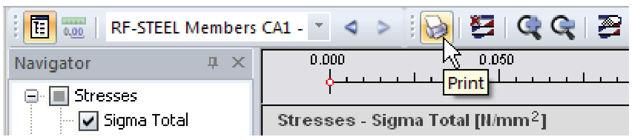


Figure 5.2: Button *Print* in the toolbar of the *Result Diagram* window

The following dialog box opens:

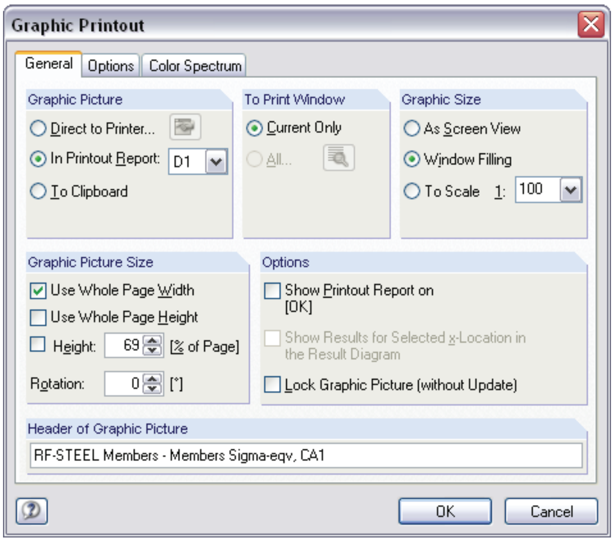
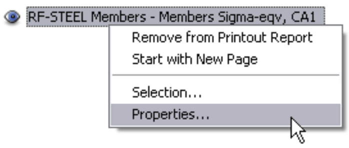


Figure 5.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in detail in the RFEM manual, chapter 11.2, page 354. The RFEM manual also describes the *Options* and *Color Spectrum* tab.

A graphic from RF-STEEL, 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* opens again, offering different options for modification.

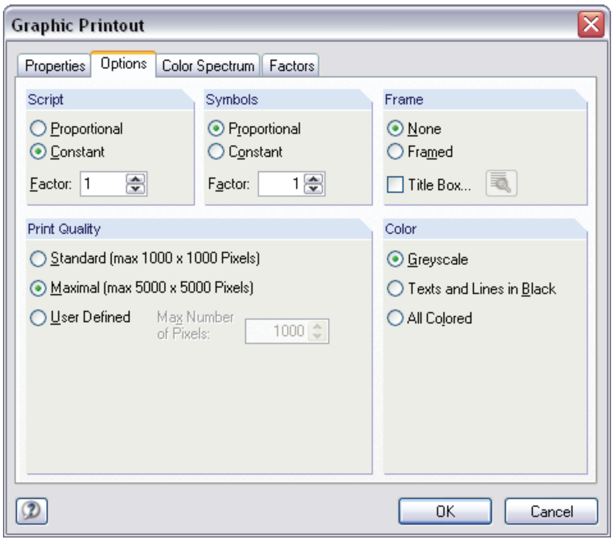


Figure 5.4: Dialog box *Graphic Printout*, tab *Options*

5.2.2 Results on the Cross-section

In RF-STEEL Members, the access to the print function is not directly available. Instead, the corresponding button can be found in the dialog box *Cross-section Values and Stress Diagram*. To open the dialog box, use the button [Print Cross-section Values and Extended Stress Diagram].

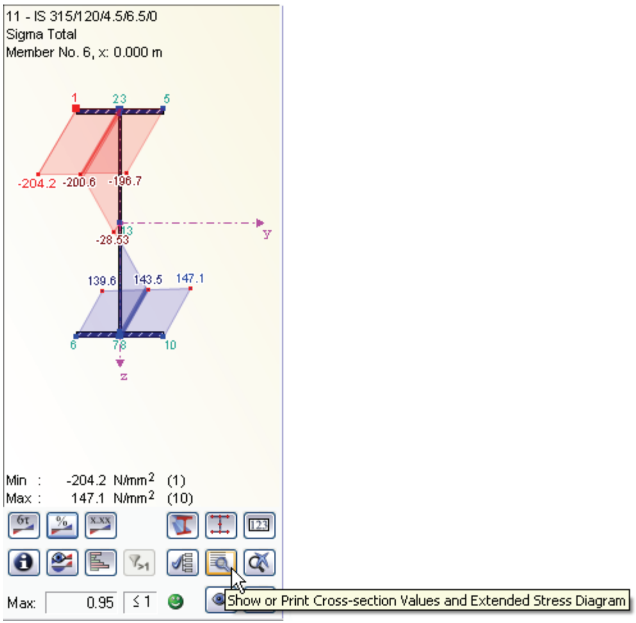


Figure 5.5: Button *Show or Print Cross-section Values and Extended Stress Diagram* in the graphic of the results tables

In the dialog box *Cross-section Values and Stress Diagram* (see Figure 4.9, page 71), you specify the member, the relevant x-location and the stress type. To open the following printing dialog box, use the [Print] button at the bottom right in the dialog box.

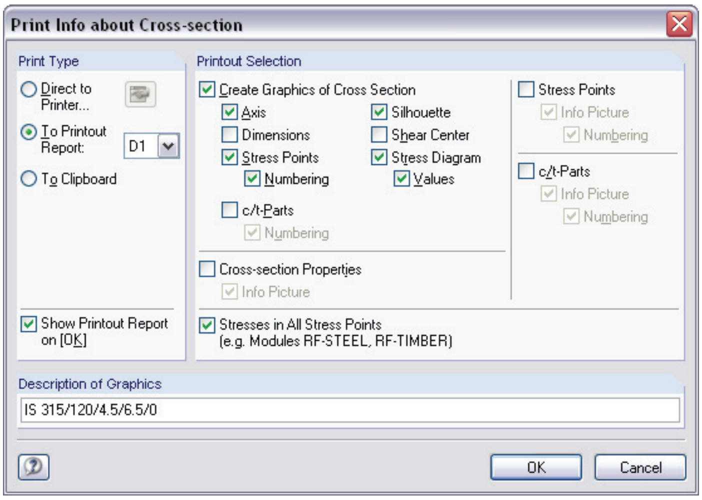


Figure 5.6: Dialog box *Print Info about Cross-section*

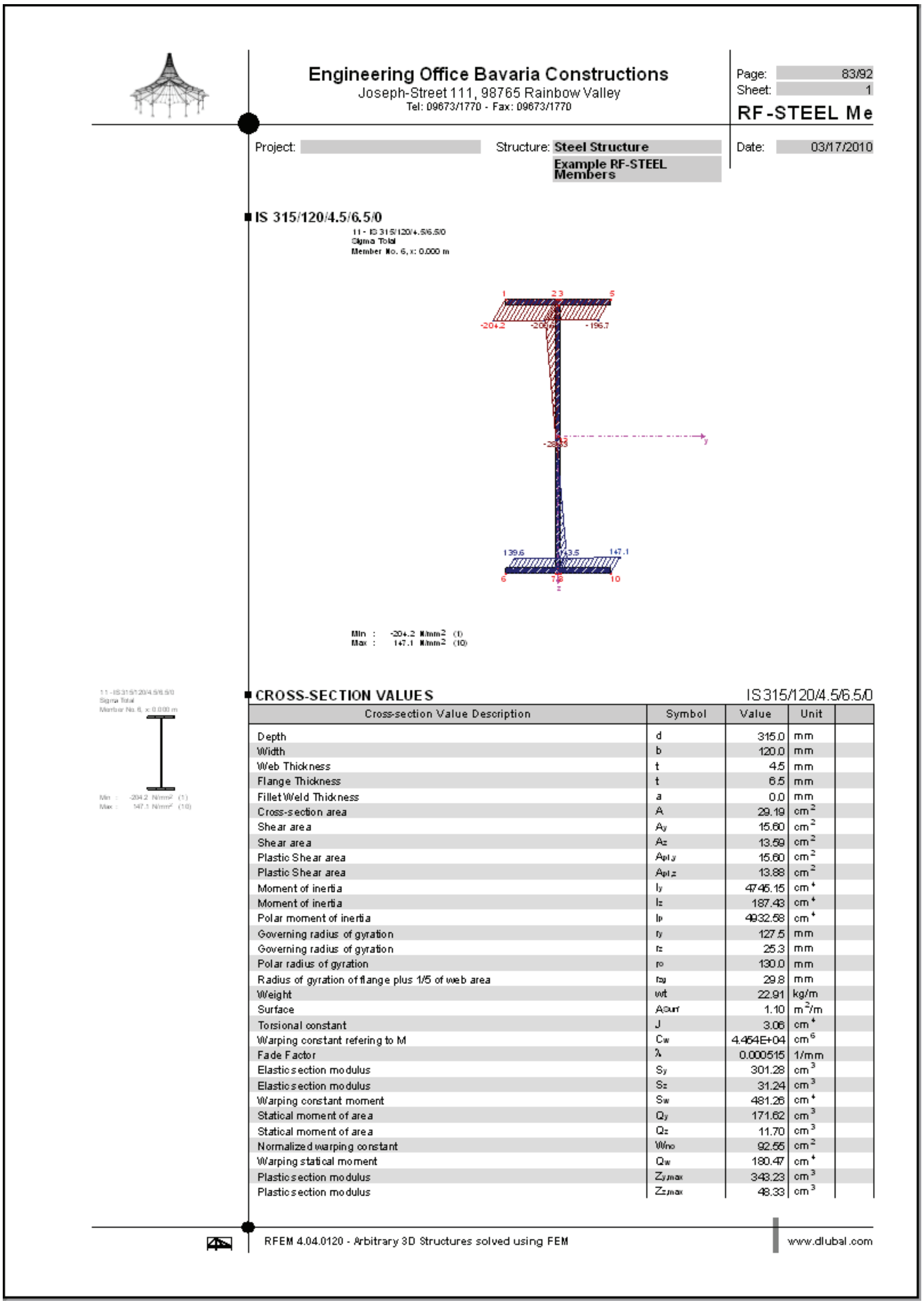
In the dialog section *Print Type*, the common options are available for selection:

- *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 listed under *Create Graphics of Cross Section* can each be displayed or hidden for the stress graphic. If you tick the check box for *Cross-section Properties*, the cross-section 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.

When you click [OK], the printout report usually opens. If you want to import several graphics consecutively into the printout report, clear the check box *Show Printout Report on [OK]*.



Engineering Office Bavaria Constructions
Joseph-Street 111, 98765 Rainbow Valley
Tel: 09673/1770 - Fax: 09673/1770

Page: 83/92
Sheet: 1
RF-STEEL Me

Project: Structure: Steel Structure
Example RF-STEEL Members
Date: 03/17/2010

IS 315/120/4.5/6.5/0

11 - IS 315/120/4.5/6.5/0
Sigma Total
Member No. 6, x: 0.000 m

Min : -204.2 N/mm² (1)
Max : 147.1 N/mm² (10)

CROSS-SECTION VALUES IS 315/120/4.5/6.5/0

Cross-section Value Description	Symbol	Value	Unit
Depth	d	315.0	mm
Width	b	120.0	mm
Web Thickness	t	4.5	mm
Flange Thickness	t	6.5	mm
Fillet Weld Thickness	a	0.0	mm
Cross-section area	A	29.19	cm^2
Shear area	A_y	15.60	cm^2
Shear area	A_z	13.59	cm^2
Plastic Shear area	$A_{pl,y}$	15.60	cm^2
Plastic Shear area	$A_{pl,z}$	13.88	cm^2
Moment of inertia	I_y	4745.15	cm^4
Moment of inertia	I_z	187.43	cm^4
Polar moment of inertia	I_p	4932.58	cm^4
Governing radius of gyration	r_y	127.5	mm
Governing radius of gyration	r_z	25.3	mm
Polar radius of gyration	r_o	130.0	mm
Radius of gyration of flange plus 1/5 of web area	r_{zg}	29.8	mm
Weight	wt	22.91	kg/m
Surface	A_{Surf}	1.10	m^2/m
Torsional constant	J	3.06	cm^4
Warping constant refering to M	C_w	4.454E+04	cm^6
Fade Factor	λ	0.000515	1/mm
Elastic section modulus	S_y	301.28	cm^3
Elastic section modulus	S_z	31.24	cm^3
Warping constant moment	S_w	481.26	cm^4
Statical moment of area	Q_y	171.62	cm^3
Statical moment of area	Q_z	11.70	cm^3
Normalized warping constant	W_{no}	92.55	cm^2
Warping statical moment	Q_w	180.47	cm^4
Plastic section modulus	$Z_{y,max}$	343.23	cm^3
Plastic section modulus	$Z_{z,max}$	48.33	cm^3

RFEM 4.04.0120 - Arbitrary 3D Structures solved using FEM
www.dlubal.com

Figure 5.7: Stress graphic in the printout report

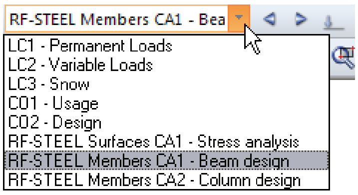
6. General Functions

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

6.1 RF-STEEL Design Cases

Surfaces or members can be managed specifically 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 surface or member/set of member in different design cases.



The RF-STEEL cases are available in the RFEM workspace and can be displayed like a load case or load group by means of the toolbar list.

Create a new RF-STEEL case

To create a new design case,

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

The following dialog box appears.

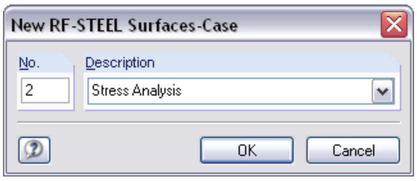


Figure 6.1: Dialog box *New RF-STEEL Surfaces-Case*

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* of the respective RF-STEEL add-on module opens where you can enter the new design data.

Rename a RF-STEEL case

To change the description of a design case subsequently,

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

The dialog box *Rename RF-STEEL Surfaces-/Members-Case* appears.

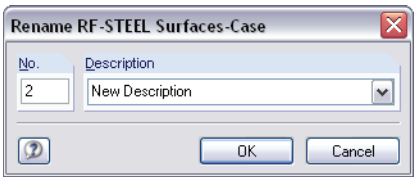


Figure 6.2: Dialog box *Rename RF-STEEL Surfaces-Case*

Copy a RF-STEEL case

To copy the input data of the current design case,

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

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

Figure 6.3: Dialog box *Copy RF-STEEL Surfaces-Case*

Delete a RF-STEEL case

To delete design cases,

select **Delete Case** on the **File** menu in the RF-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].

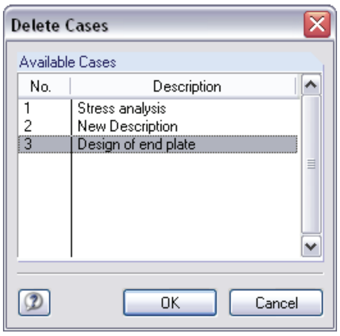


Figure 6.4: Dialog box *Delete Cases*

6.2 Optimization

In both RF-STEEL modules, it is possible to optimize the surface thicknesses or cross-sections. During the optimization process, the program determines the surface thickness or cross-section within the respective cross-section table that fulfills the analysis requirements for the existing RFEM internal forces in the most optimal way, that means comes as close as possible to a user-defined maximum ratio.

The maximum allowable stress ratio is defined in the *Details* dialog box (see Figure 2.13, page 26 or Figure 3.8, page 50).

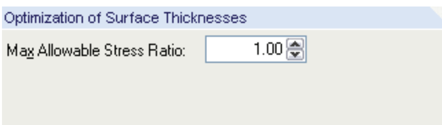
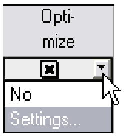


Figure 6.5: Specification of the maximum stress ratio

Please note that the internal forces won't be recalculated automatically with the changed surface thicknesses or cross-sections! It is up to you to decide when to use the initial or optimized surface thicknesses or cross-sections for a new calculation run in RFEM. As a result of optimized objects, internal forces may vary considerably because of the changed stiffnesses in the structural system. Therefore, it is recommended to recalculate the internal forces after the first optimization and then to modify the surface thicknesses or cross-sections once again.

6.2.1 RF-STEEL Surfaces

During the optimization process, the program determines the surface thickness that comes as close as possible to the maximum stress ratio for the existing RFEM internal forces.



The optimization options are only available in table 1.3 *Surfaces*. To optimize a particular surface, tick its corresponding check box in column D or E (see Figure 2.7, page 16). The following dialog box opens.

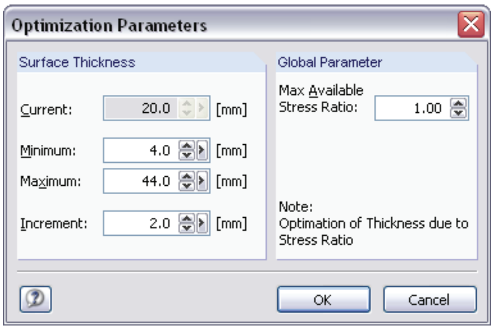


Figure 6.6: Dialog box *Optimization Parameters*

In the input fields *Minimum* and *Maximum*, you define the upper and lower limits of the allowable surface thicknesses. The current thickness is also displayed so that you can compare the values. The *Increment* determines the interval in which the thickness may vary during the optimization process.

When you have recalculated the data, table 1.3 *Surfaces* shows the optimized thicknesses. You do not need to transfer the new surface thicknesses to RFEM manually: Set table 1.3 *Surfaces*, and then

select **Export All Surfaces to RFEM** on the **Edit** menu.

The context menu in table 1.3 also provides options to export modified surfaces to RFEM.

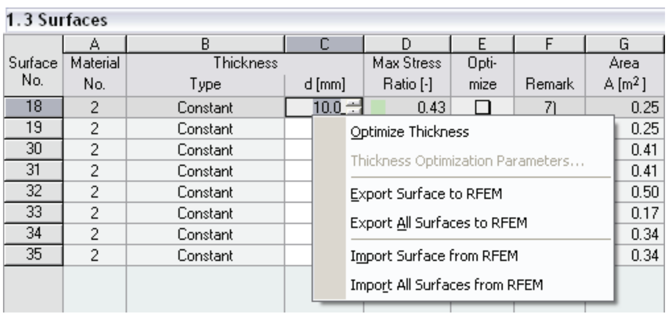
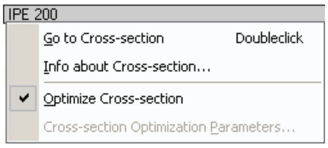


Figure 6.7: Context menu in table 1.3 *Surfaces*

By using the menu functions described above, you can also import the original RFEM surfaces to RF-STEEL Surfaces. Please note that this option is only available in table 1.3 *Surfaces*.

6.2.2 RF-STEEL Members



Context menu in results tables

To optimize a particular cross-section, tick its corresponding check box in column C or D of table 1.3 *Cross-sections* (cf. Figure 3.4, page 44). You can also start the cross-section optimization out of the results tables by using the context menu.

During the optimization process, RF-STEEL determines the cross-section within the specified cross-section table that fulfills the analysis requirements in the most optimal way, that means comes as close as possible to a user-defined maximum ratio. The program determines the second moment of area required for the RFEM internal forces and uses the cross-section of the given cross-section table that completes the design with a stress ratio that is as high as possible. After the calculation, the graphic in table 1.3 *Cross-sections* shows two cross-sections: the original from RFEM and the optimized (cf. Figure 3.4, page 44).

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

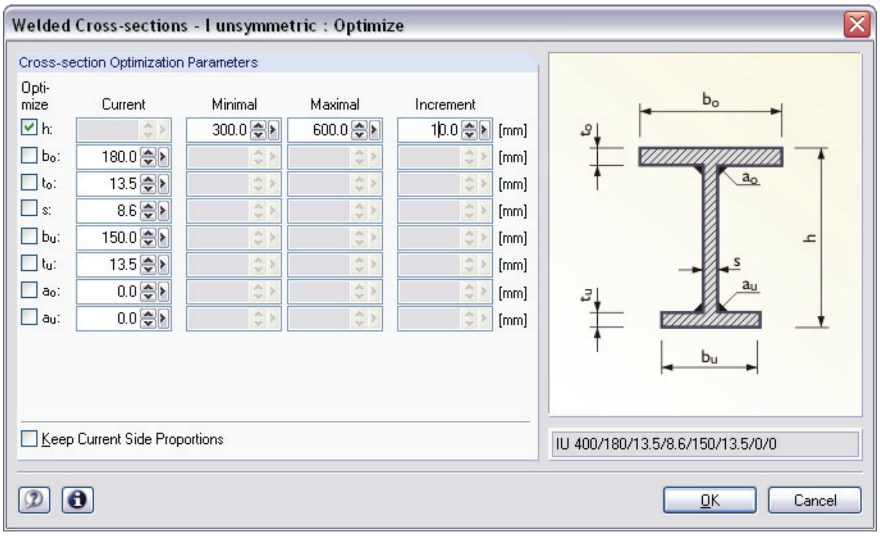


Figure 6.8: Dialog box *Welded Cross-sections - I unsymmetric : Optimize*

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 that define the upper and lower limits of the corresponding parameter for the optimization. The *Increment* column determines the interval in which the parameter varies during the optimization process.

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

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

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

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

Also the context menu in table 1.3 provides options to export modified cross-sections to RFEM.

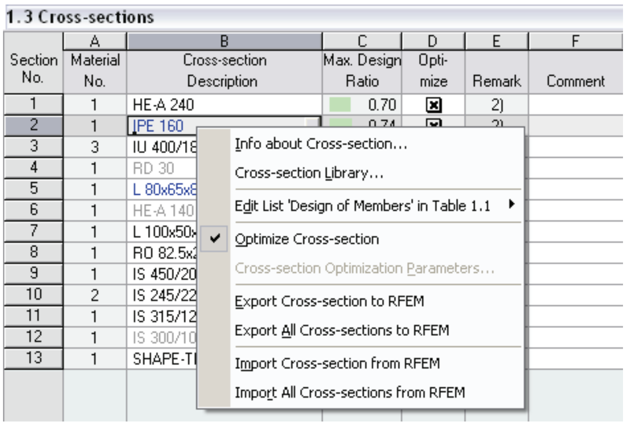


Figure 6.9: Context menu in table 1.3 *Cross-sections*

Before the changed cross-sections are transferred to RFEM, a security query appears, because the transfer requires the deletion of results. When you confirm the query and start the calculation subsequently in RF-STEEL Members, the RFEM internal forces as well as the RF-STEEL stresses are determined in one continuous calculation run.

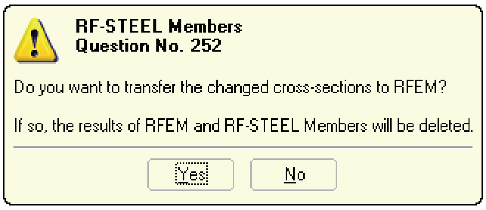


Figure 6.10: Query before transfer of modified cross-sections to RFEM

By using the menu functions described above, you can also import the original RFEM cross-sections to RF-STEEL Members. 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. As these moments are considered with the fourth power, the stress determination may be inaccurate if the depths of the start and end cross-section differ considerably. In this case, it is recommended to divide the taper into several single members whose start and end cross-sections have minor cross-section differences.

6.3 Units and Decimal Places

The units and decimal places for RFEM and all add-on modules are managed in one global dialog box. In both RF-STEEL modules, you can use the menu to adjust 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 RFEM. In the *Program / Module* list on the left, the relevant RF-STEEL Surfaces or RF-STEEL Members add-on module is already preset.

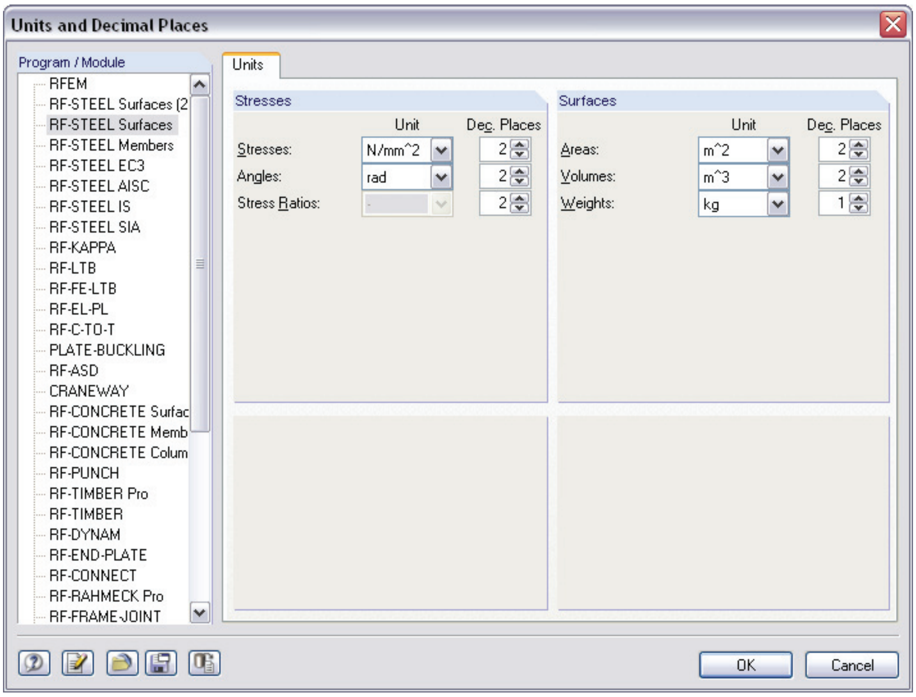


Figure 6.11: Dialog box *Units and Decimal Places*

The settings can be saved as user profile to reuse them in other structures. The functions are described in the RFEM manual, chapter 12.6.2, page 453.

6.4 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 RF-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 RF-STEEL add-on modules can be printed into the global printout report (cf. chapter 5.1, page 78) to export them subsequently. In the printout report,

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

The function is described in detail in the RFEM manual, chapter 11.1.11, page 350.

Excel / OpenOffice

RF-STEEL provides a function for the direct data export to MS Excel or OpenOffice.org Calc. In the RF-STEEL add-on module,

select **Export Tables** on the **File** menu.

The following export dialog box appears.

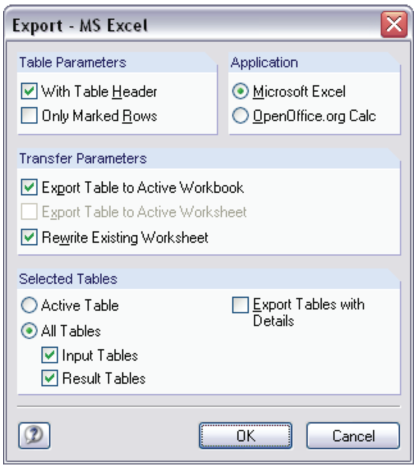
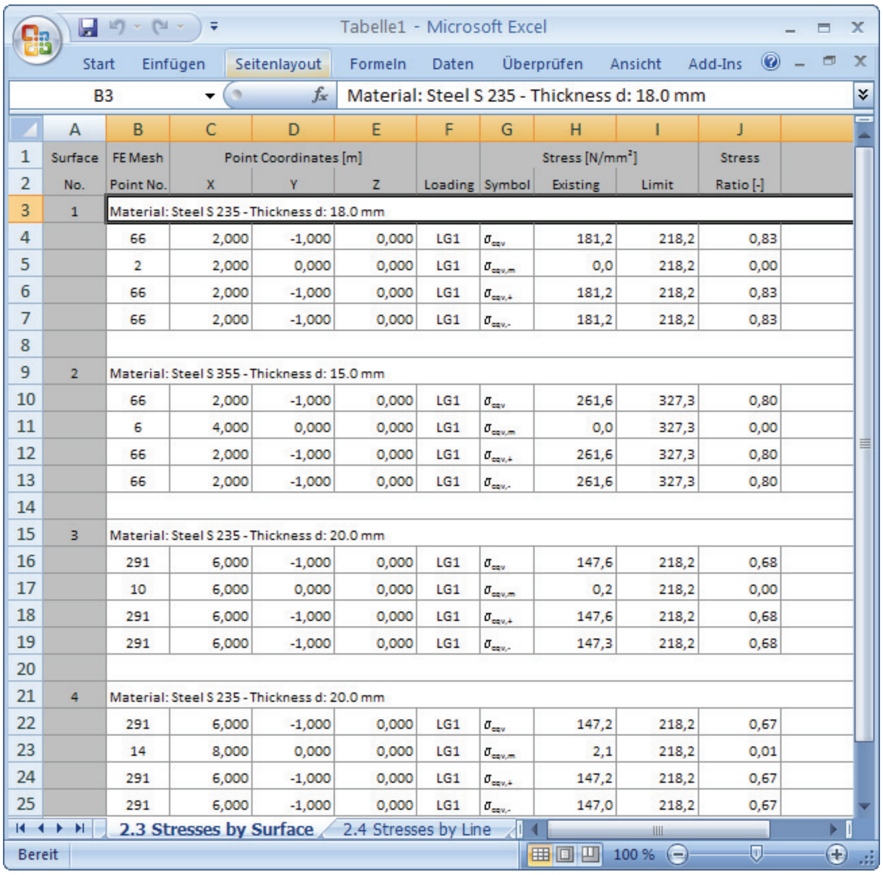


Figure 6.12: Dialog box *Export - MS Excel*

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.



	A	B	C	D	E	F	G	H	I	J
1	Surface	FE Mesh	Point Coordinates [m]					Stress [N/mm²]		Stress
2	No.	Point No.	X	Y	Z	Loading	Symbol	Existing	Limit	Ratio [-]
3	1	Material: Steel S 235 - Thickness d: 18.0 mm								
4		66	2,000	-1,000	0,000	LG1	σ_{eqv}	181,2	218,2	0,83
5		2	2,000	0,000	0,000	LG1	$\sigma_{eqv,m}$	0,0	218,2	0,00
6		66	2,000	-1,000	0,000	LG1	$\sigma_{eqv,+}$	181,2	218,2	0,83
7		66	2,000	-1,000	0,000	LG1	$\sigma_{eqv,-}$	181,2	218,2	0,83
8										
9	2	Material: Steel S 355 - Thickness d: 15.0 mm								
10		66	2,000	-1,000	0,000	LG1	σ_{eqv}	261,6	327,3	0,80
11		6	4,000	0,000	0,000	LG1	$\sigma_{eqv,m}$	0,0	327,3	0,00
12		66	2,000	-1,000	0,000	LG1	$\sigma_{eqv,+}$	261,6	327,3	0,80
13		66	2,000	-1,000	0,000	LG1	$\sigma_{eqv,-}$	261,6	327,3	0,80
14										
15	3	Material: Steel S 235 - Thickness d: 20.0 mm								
16		291	6,000	-1,000	0,000	LG1	σ_{eqv}	147,6	218,2	0,68
17		10	6,000	0,000	0,000	LG1	$\sigma_{eqv,m}$	0,2	218,2	0,00
18		291	6,000	-1,000	0,000	LG1	$\sigma_{eqv,+}$	147,6	218,2	0,68
19		291	6,000	-1,000	0,000	LG1	$\sigma_{eqv,-}$	147,3	218,2	0,68
20										
21	4	Material: Steel S 235 - Thickness d: 20.0 mm								
22		291	6,000	-1,000	0,000	LG1	σ_{eqv}	147,2	218,2	0,67
23		14	8,000	0,000	0,000	LG1	$\sigma_{eqv,m}$	2,1	218,2	0,01
24		291	6,000	-1,000	0,000	LG1	$\sigma_{eqv,+}$	147,2	218,2	0,67
25		291	6,000	-1,000	0,000	LG1	$\sigma_{eqv,-}$	147,0	218,2	0,67

Figure 6.13: Result in MS Excel: Table 2.3 *Stresses by Surface*

A Literature

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

[2] DIN 18 800 Teil 2: Stahlbauten, Stabilitätsfälle, Knicken von Stäben und Stabwerken, 1990

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

[4] Eurocode 3 Teil 1-1: Bemessung und Konstruktion von Stahlbauten, 1993

[5] PETERSEN, Chr.: Stahlbau, Vieweg und Sohn, Braunschweig/Wiesbaden, 3. Auflage1993

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

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

[8] ZIENKIEWICZ, O. C., CHEUNG, Y.K.: The Finite Element Method in Structural and Continuum Mechanics, McGraw-Hill, New York/London, 1967

[9] KOLÁR, V. et al.: Berechnung von Flächen- und Raumtragwerken nach der Methode der finiten Elemente, Springer Verlag, Wien/New York, 1975

[10] TIMOSHENKO, S.P., WOINOWSKI-KRIEGER, S.: Theory of Plates and Shells, 2. Auflage, McGraw-Hill, New York, 1959

[11] KOLÁR, V., NEMEC, I.: Finite Element Analysis of Structures. United Nations Development Program, Economic Com. for Europe, Workshop on CAD Techniques, June 1984, Prague/Geneva, Vol. 1, 248 pp.

[12] BERGAN, P.G. - FELIPPA, C. A.: A Triangular Membrane Element With Rotational Degrees of Freedom. Computer Methods in Applied Mechanics and Engineering, 50 (1985), 25 - 69

[13] ZIENKIEWIC, O.C.: The Finite Element Method in Engineering Science, Mc Graw - Hill, London 3rd Ed., repr. 1979, 787 pp., Chapter 18 - 19 (Nonlinear Problems)

[14] ŠEVČÍK, I., 3D Finite Element with Rotational Degrees of Freedom, FEM-Consulting s.r.o., Brno

[15] MANG, H., HOFSTETTER, G.: Festigkeitslehre, Springer Verlag, Wien/New York, 2000

B Index