Version
December 2009

Additional Module

STEEL IS

Ultimate Limit State and Serviceability Limit State Design according to IS 800 : 2007

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Additional Module STEEL IS

The Indian Standard IS 800 : 2007 determines rules for the design, analysis and construction of steel buildings in India. With the additional module STEEL IS from the company DLUBAL ENGINEERING SOFTWARE all users obtain a highly efficient and universal tool to design structures that have been analyzed in RSTAB according to this standard.

All typical designs of load capacity, stability and deformation are carried out in the module STEEL IS. Different actions are taken into account during the load capacity design, and the user has a choice of different interaction designs that are possible according to the Indian standard. The allocation of designed cross-sections to classes 1 to 3 makes an important part of the design according to IS 800 : 2007. The purpose of this classification is to determine the range in which the local buckling of cross-section parts limits the load capacity so that the rotational capacity of cross-sections can be verified. STEEL IS further automatically calculates the required ratios of compressed parts and carries out the classification automatically.

For the stability design, you can determine for every single member or set of members whether buckling is possible about the y-axis and/or z-axis. You can also define lateral supports. The non-dimensional slenderness ratio and elastic buckling stress are automatically determined in STEEL IS on the basis of the boundary conditions. For the design of lateral torsional buckling, the elastic critical moment that is necessary for the design can be either calculated automatically by STEEL IS or entered manually. The location where the loads are applied which influences the elastic critical moment can also be defined in the detailed settings.

The serviceability limit state represents an important element in static calculations in modern civil engineering. All relevant values of the maximum deflections can be defined in the detailed settings. In STEEL IS, the user can arrange load cases, load groups and combinations individually to different design situations.

Like other modules, STEEL IS is fully integrated into the RSTAB 7 program. However, it is not only an optical part of the program. The results of the module can be incorporated to the central printout report. Therefore, the entire design can be easily and especially uniformly organized and presented.

The program includes an automatic cross-section optimization and a possibility to export all modified sections to RSTAB.

Individual design cases make it possible to flexibly analyze separate parts of extensive structures.

We wish you much success and delight when working with our module STEEL IS.

Your DLUBAL ENGINEERING SOFTWARE company.

1.2 STEEL IS Team

The following people participated in the development of the STEEL IS module:

Program Coordinators

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem
Ing. Ph.D. Jaromír Křížek

Programmers

Ing. Zdeněk Kosáček
Ing. Ph.D. Jaromír Křížek
Dipl.-Ing. Georg Dlubal
Dr.-Ing. Jaroslav Lain
Ing. Martin Budáč
Mgr. Petr Oulehle
Ing. Roman Svoboda
David Schweiner
DiS. Jiří Šmerák

Library of Cross-Sections and Materials

Ing. Ph.D. Jan Rybín
Stanislav Krytinář
Jan Brnušák

Design of Program, Dialogs and Icons

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

Testing and Technical Support

Ing. Martin Vasek
Ing. Robert Michalovič
Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Alexandra Lazar
M. Sc. Dipl.-Ing. (FH) Frank Lobisch
Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Walter Rustler
M. Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl

Manuals, Documentation and Translations

Ing. Jaromír Křížek
Dipl.-Ing. (FH) Robert Vogl
Mgr. Michaela Kryšková
Ing. Ladislav Kábrt
Dipl.-Ü. Gundel Pietzcker
Ing. Dmitry Bystrov

1.3 Using the Manual

All general topics such as installation, user interface, results evaluation and printout report are described in detail in the manual for the main program RSTAB. Hence, we omit them in this manual and will focus on typical features of the additional module STEEL IS.

During the description of STEEL IS, we use the sequence and structure of the different input and output forms of this module. We feature the described **icons** (buttons) in square brackets, e.g. [Details]. The buttons are simultaneously displayed on the left margin. The **names** of dialogs, forms and particular menus are marked in *italics* in the text so that they can be easily found in the program.

We also included an index in this manual so that you can quickly look up important terms. If you should not find the requested ones, you can also use the search function on our website **www.dlubal.com** and browse the *FAQ* list there.

1.4 Starting STEEL IS

It is possible to initialize the add-on module STEEL IS in several ways.

Main Menu

You can call up STEEL IS by this command from the main menu of the RSTAB program:

Additional Modules → Design - Steel → STEEL IS.

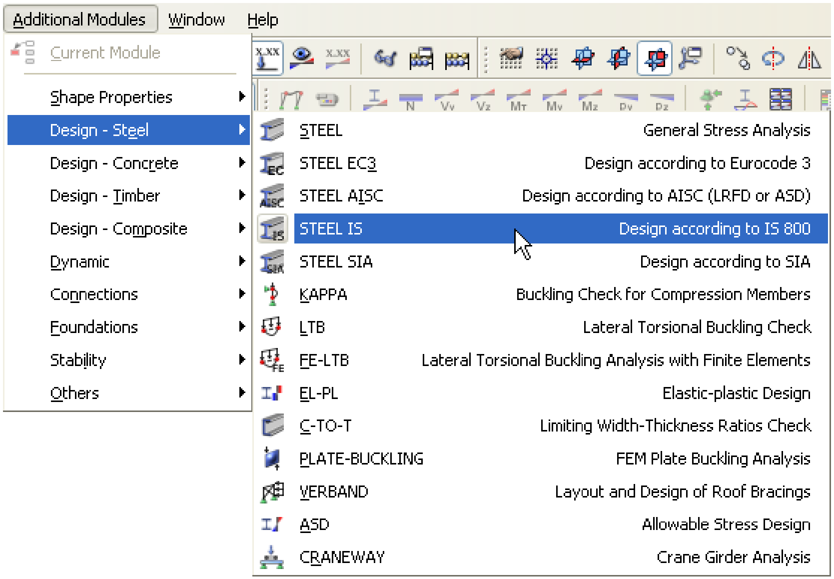


Figure 1.1: Main Menu: *Additional Modules → Design - Steel → STEEL IS*

Navigator

Further, it is possible to start STEEL IS from the *Data* navigator by clicking on the item

Additional Modules → STEEL IS.

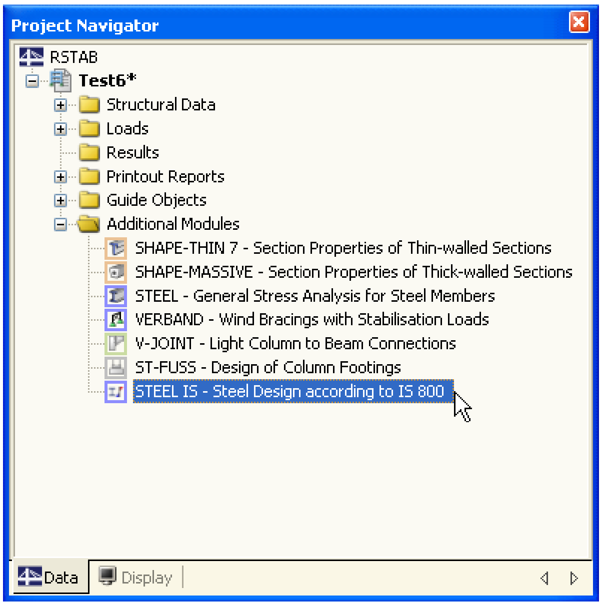
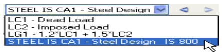


Figure 1.2: Data Navigator: *Additional Modules → STEEL IS*

Panel



If results of STEEL IS are already available in the RSTAB position, you can to set the relevant design case of this module in the list of load cases in the menu bar. The design criterion on members is displayed graphically in the work window of RSTAB using the [Results on/off] button.

The [STEEL IS] button which enables you to start STEEL IS is now available in the panel.

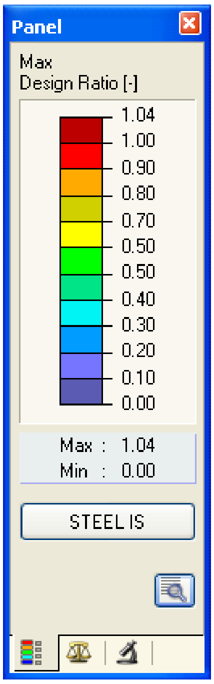


Figure 1.3: [STEEL IS] Button in Panel

2. Input Data

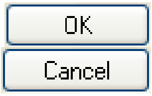
The data of the design cases is entered in different forms of this module. Furthermore, the graphic input using the function [Pick] is available for members and sets of members.

After the initialization of the STEEL IS module, a new window is displayed. In its left part, a navigator is shown that enables you to access all existing masks. The roll-out list of all possibly entered design cases is located above this navigator (see chapter 7.1).

If STEEL IS is called up for the first time in a position of RSTAB, the following important data is loaded automatically:

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

You can switch among the forms either by clicking on the individual navigator items of STEEL IS or by using the buttons visible on the left. The [F2] and [F3] function keys can also be used to browse the forms in both directions.



Save entered data by the [OK] button and close the module STEEL IS, while by the [Cancel] button you terminate the module without saving data.

2.1 General Data

In the form 1.1 *General Data*, members, sets of members and actions are selected for the design. You can specify load cases, load groups and combinations for the ultimate limit state and the serviceability limit state design separately in the corresponding registers.

2.1.1 Ultimate Limit State

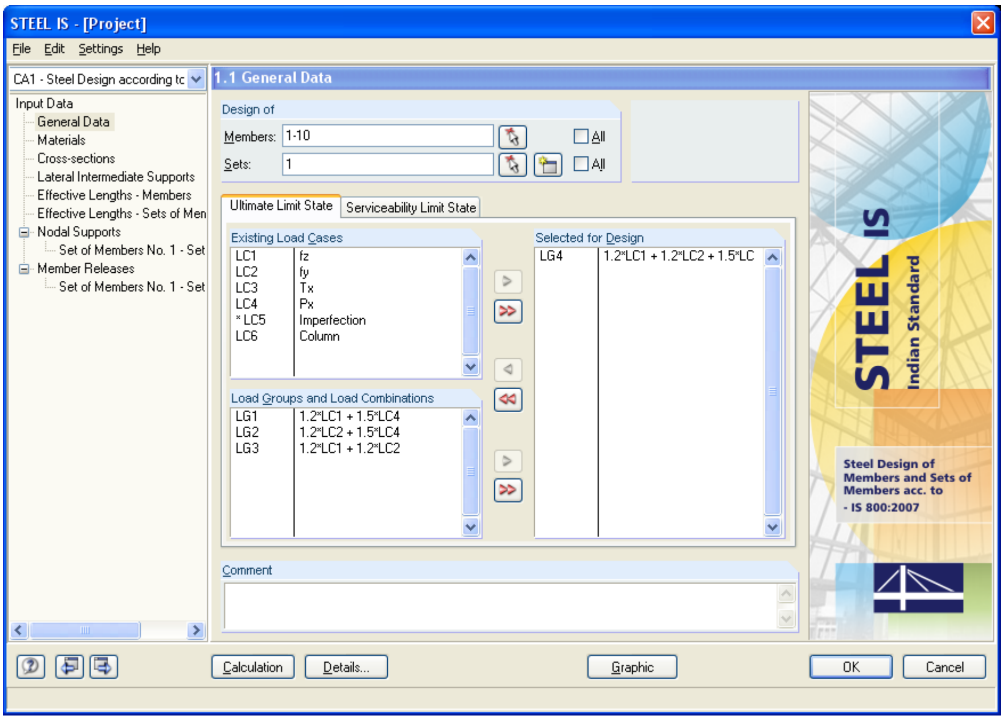


Figure 2.1: Form 1.1 *General Data - Ultimate Limit State*

Design of

You can select both *Members* and *Sets of Members* for the design. If only specific objects are to be designed, it is necessary to clear the check box *All*. By doing so, both input boxes become accessible and you can enter the numbers of the relevant members or sets of members there. With the [Pick] button, you can also select members or sets of members graphically in the RSTAB work window. To rewrite the list of default member numbers, select it by double-clicking it and then enter the relevant numbers.

If no sets of members have been defined in RSTAB yet, they can also be created in STEEL IS via the [New Set of Members…] button. The familiar RSTAB dialog to create a new set of members opens in which you enter the relevant data.

Designing sets of members has the advantage that selected members can be analyzed to determine the total maxima of the design ratios. In this case, the results forms 2.3 *Design by Sets of Members* and 4.2 *Parts List by Set of Members* are displayed additionally.

Existing Load Cases / Load Groups and Load Combinations

All design-relevant load cases, load groups and load combinations that were created in RSTAB are listed in these two sections. The [▶] button moves the selected load cases, load groups or combinations to the list *Selected for Design* on the right. Specific items can also be selected by double-clicks. The [▶▶] button transfers all items to the list on the right.

If an asterisk (*) is displayed at load cases or combinations, as you can see e.g. in Figure at load case 5, they are excluded from the design. It signifies that no loads were assigned to these load cases or that they contain only imperfections (as in our example). Furthermore, it is only possible to select load combinations for which the minimum and maximum values can be determined unambiguously. This restriction is necessary because the calculation of the elastic critical moment at lateral buckling requires the unambiguous assignment of moment diagrams. If an invalid load combination is selected, the following warning appears:

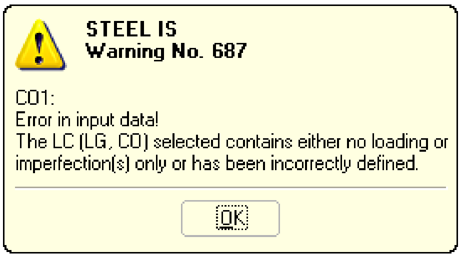


Figure 2.2: Warning when selecting an *invalid* LC, LG or CO

A multiple choice of load cases can be done by using the [Ctrl] key, as a routine procedure in Windows. Hence, you can select and transfer several load cases to the list on the right simultaneously.

Selected for Design

The loads selected for the design are listed in the right column. By the [◀] button you can remove the selected load cases, load groups or load combinations from the list. As before, the selection can be executed by double-clicks. The [◀◀] button removes all items from the list.

Generally, the calculation of an enveloping *Or* load combination is faster than the analysis of all contained load cases or groups. On the other hand, you must keep in mind the above-mentioned restriction: to determine the maximum or minimum values unambiguously, the *Or* load combination must only contain load cases, groups or combinations which enter the combination with the criterion *Constant*. Moreover, the design of an enveloping load combination makes it a bit difficult to retrace the influence of the contained actions.

2.1.2 Serviceability Limit State

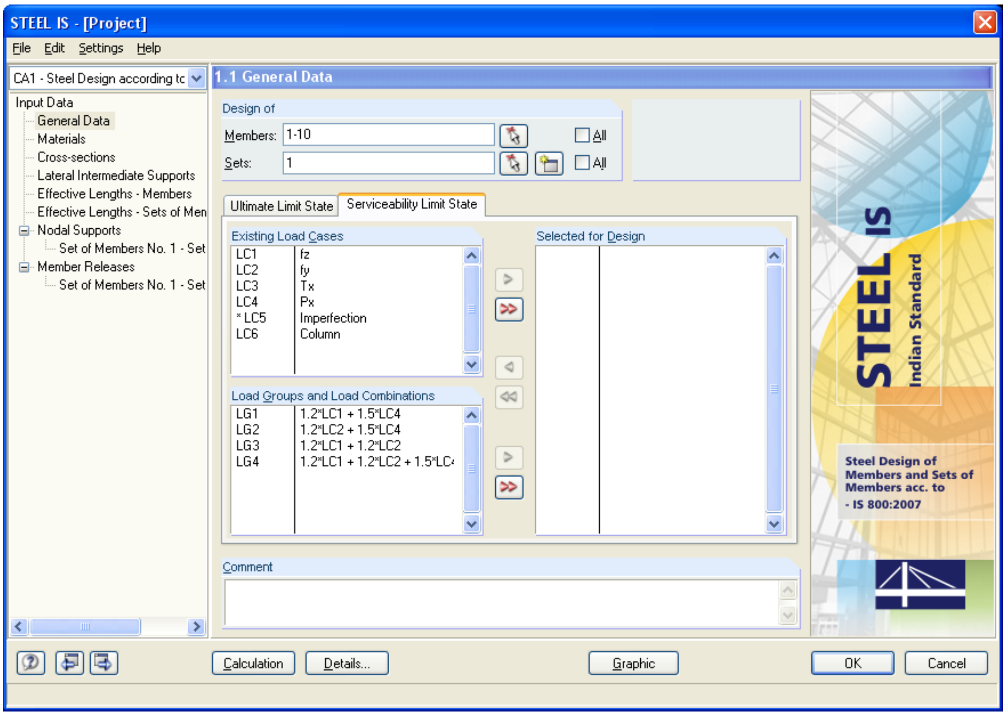


Figure 2.3: Form 1.1 *General Data - Serviceability Limit State*

Existing Load Cases / Load Groups and Load Combinations

All load cases, load groups and load combinations that were created in RSTAB are listed in these two sections.

Selected for Design

Adding load cases and their groups and combinations to the list for the design, resp. removing them from the list is done in the same way like in the previous register tab (see Chapter 2.1.1).

Comment

You can enter some additional notes here to describe the current design case.

2.2 Materials

This form is divided into two parts. The materials for the design are listed in the upper part. In the lower part, the *Material Properties* of the current material are displayed, i.e. the material whose line is selected in the upper form.

The material properties that are necessary to calculate the internal forces in RSTAB are described in detail in the RSTAB manual, Chapter 5.2. All design-relevant material characteristics are stored in the global material library. Those are automatically set as default.

The units and decimal places of the material properties and stresses can be edited from the main menu **Options** → **Units and Decimal Places...** (see Chapter 7.4).

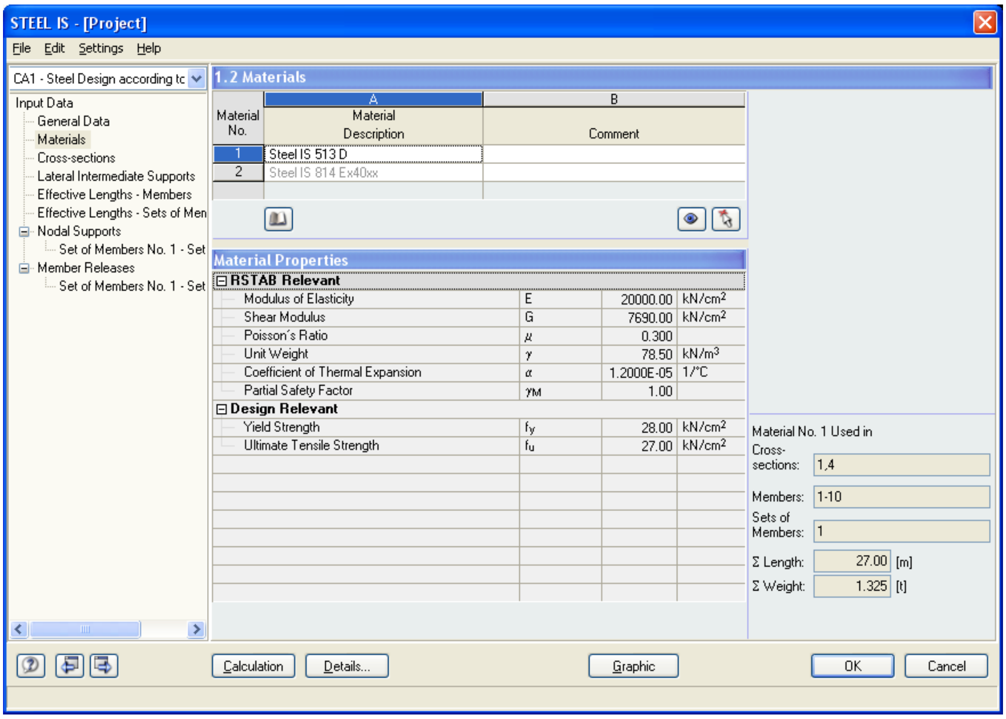
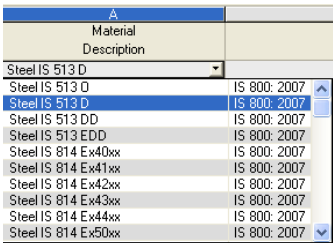


Figure 2.4: Form 1.2 *Materials*

Material Description



The materials that have been defined in RSTAB are set by default. You can also enter materials manually here. If the *Material Description* corresponds to an entry of the material library, STEEL IS automatically imports the relevant material properties.

To select a material from the list, place the cursor in column A and click on the [▼] button or press the [F7] function key. A list is opened that you can see on the left. As soon as you have chosen the appropriate material, the material properties are updated in the form below.

The list of materials includes only materials from the category **Steel**. How to import materials from the library is described below.

Import Material from Library

A considerable number of materials is already stored in a library. Open the library via menu

Edit → **Material Library...**

or by clicking on the button that is visible on the left.

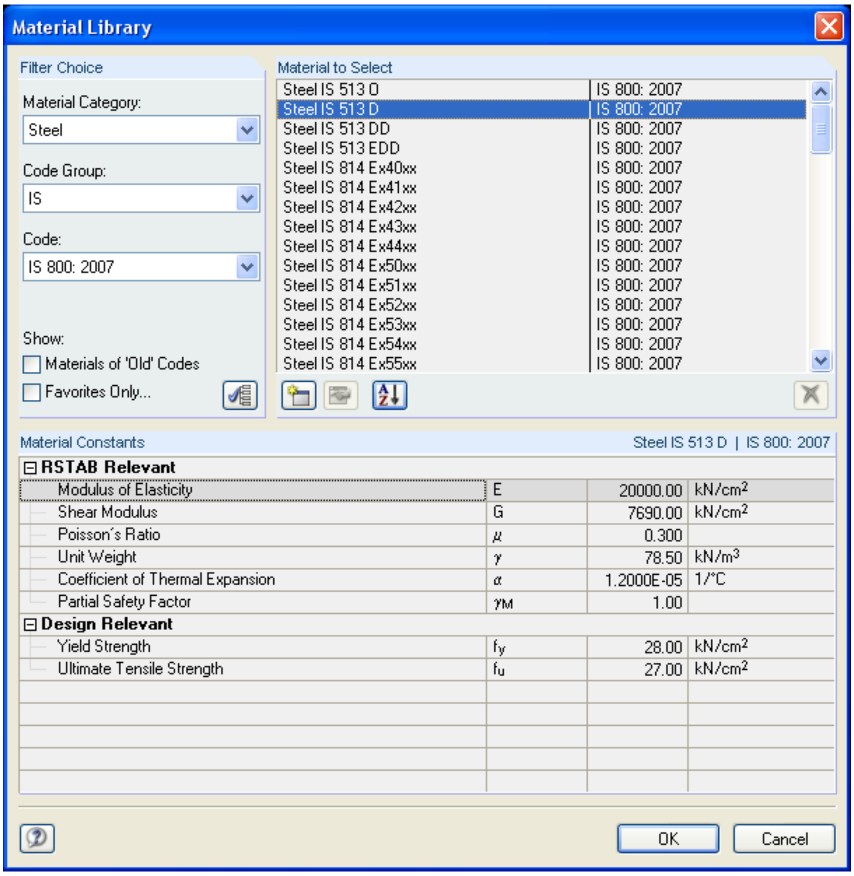


Figure 2.5: *Material Library* Dialog

In the section *Filter Choice,* the material category **Steel** is set by default. In the list *Material to Select* located on the right, you can select a particular material, and in the lower part of the dialog you can check its characteristic values. After clicking on [OK] or pressing the [↵] key, the material is taken over to the form 1.2 *Materials* of STEEL IS.

Chapter 5.2 of the RSTAB manual explains in detail how materials can be filtered, added to the library or newly classified.

Basically, you can also select materials of the categories *Cast Iron* and *Stainless Steel* of the library. However, you have to bear in mind that those materials are not covered by the Indian Standard IS 800 : 2007. For this reason, it is also impossible to significantly modify the material properties in STEEL IS.

2.3 Cross-Sections

This form controls the cross-sections that are to be designed. The parameters of the optimization can be defined here as well.

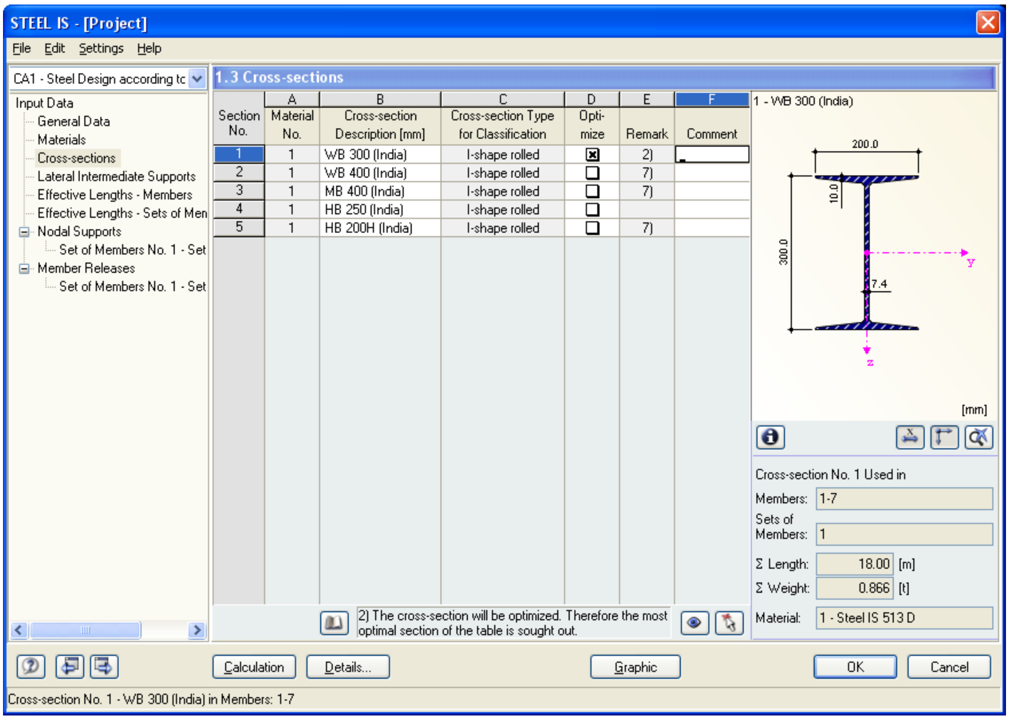


Figure 2.6: Form 1.3 *Cross-Sections*

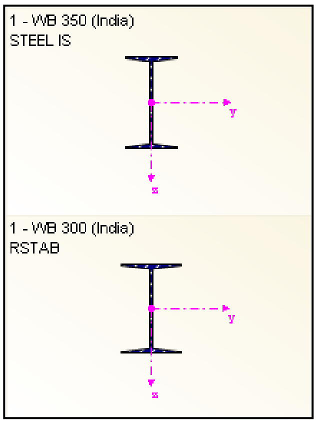
Cross-Section Description

When you open this form, the sections that were defined in RSTAB are set by default, including the assigned material numbers.

The cross-sections can be changed any time for the design. The description of a modified cross-section is highlighted in blue color.

In order to edit a cross-section, enter the new description in the corresponding line or select the new section from the library. Open the library by clicking on the [Import Cross-Section from Library…] button. Alternatively, place the cursor in the corresponding line and click on the […] button or press the [F7] key. The library opens which is already familiar from RSTAB, see Figure 2.7.

Chapter 5.3 of the RSTAB manual describes in detail how cross-sections can be selected from the library.



If the cross-sections are different in STEEL IS and RSTAB, both cross-sections are shown in the graphic window next to the table. The internal forces of RSTAB are then used for the stress design of the cross-section that is set in STEEL IS.

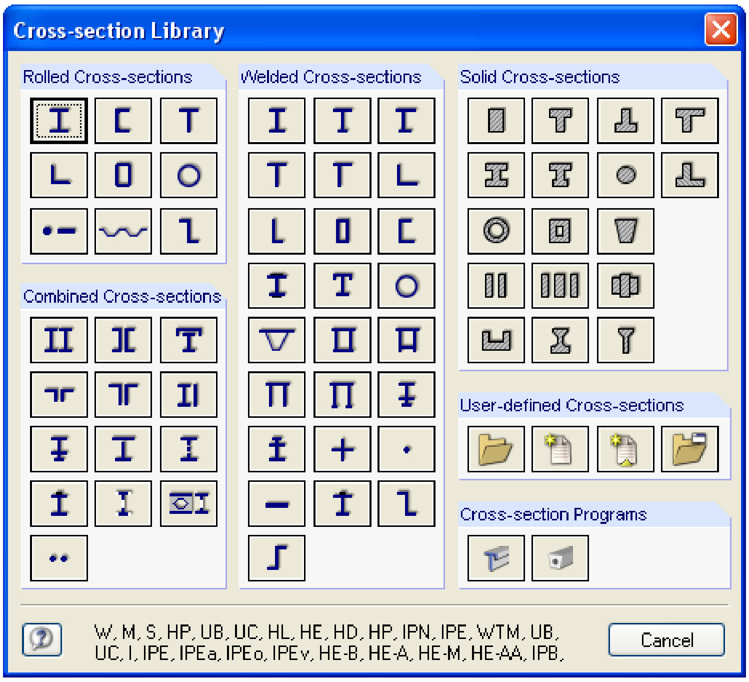
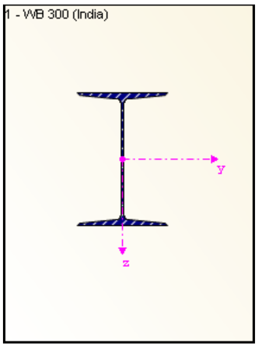


Figure 2.7: Cross-Section Library

Coordinate System



The y-z coordinate system is applied for cross-sections in accordance with RSTAB, see the figure on the left. The y-axis represents the major and the z-axis the minor axis of the cross-section. This coordinate system is used in all results forms as well (see chapter 4).

Tapered Member

In case of tapered members with different cross-sections at the member start and member end, both cross-section numbers are stated in two lines, following the definition in RSTAB. You can design tapered members in STEEL IS if the following condition is fulfilled: an equal number of stress points is required at both member ends.

For example, the normal stresses are calculated from the moments of inertia and from the centroidal distances of the stress points. If the start and end cross-sections of the tapered member have different numbers of stress points, STEEL IS cannot interpolate the intermediate values. An error message appears before the calculation:

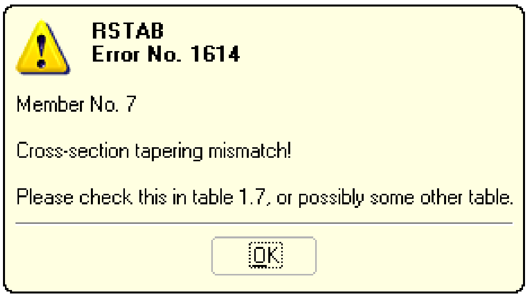


Figure 2.8: Warning in Case of Incompatible Cross-Sections

To check on the stress points of the cross-section, you can display them including their numbers: Select the cross-section in form 1.3 and then click on the [Info about Cross-Section…] button.

Info about Cross-Section

There are various display options for stress points and c/t cross-section parts in this dialog.

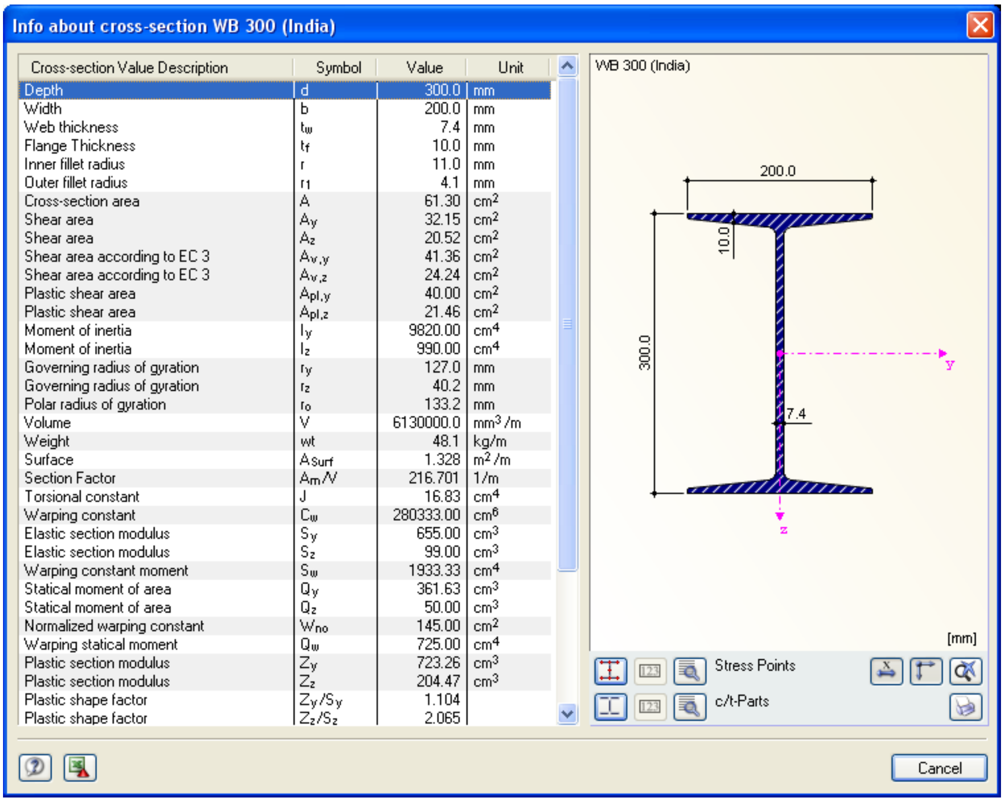


Figure 2.9: *Info about cross-section* Dialog

The currently selected cross-section is displayed in the right part of the dialog. The buttons below have the following functions:

Button	Function
	The stress points are switched on and off.
	The cross-section parts (c/t) are switched on and off.
	The numbering of stress points, resp. of cross-section parts (c/t) is switched on and off.
	The details of stress points, resp. of cross-section parts (c/t) are displayed.
	The dimensioning of the cross-section is switched on and off.
	The principal axes of the cross-section are switched on and off.

Table 2.1: Buttons for Cross-Section Graphics

Max. Design Ratio

Due to this column, you can decide whether to carry out an optimization. The column is only displayed when STEEL IS has designed the cross-sections. It becomes visible from the data and the colored relation scales in this column which cross-sections have a low design ratio and therefore are oversized, resp. which are overstrained and therefore are too weak.

Optimize

Every cross-section can be optimized. During the optimization process, the cross-section within the same group of cross-sections is determined on the basis of the internal forces from RSTAB which fulfills best the maximum design ratio. Figure 2.6 shows how the optimization of a particular cross-section is set by ticking the corresponding box in column D.

The maximum allowable design ratio for the optimization is controlled in the dialog *Details*, see chapter 3.1. Further information on the optimization of cross-sections can be found in chapter 7.2 of this manual.

Remark

In this column, the references to footnotes (below the list of cross-sections) are shown.

If the message *Inadmissible Cross-Section No. XX* appears before the design, then this due to a cross-section which is not contained in the cross-section library. It may be a user-defined cross-section or a cross-section that was not calculated in the module SHAPE-THIN. Via the [...] button in column B *Cross-section Description* you can set a cross-section that is suitable for the design (see Figure 2.7 with following remarks).

2.4 Lateral Intermediate Supports

In this form, lateral intermediate supports on members can be defined. The program always assumes these supports as perpendicular to the minor axis z (see Figure 2.9) of the cross-section. Hence, it is possible to change the effective lengths of the member that are important for the design of column buckling and lateral torsional buckling. It is also important to know that lateral intermediate supports are considered as a forked supports for the design.

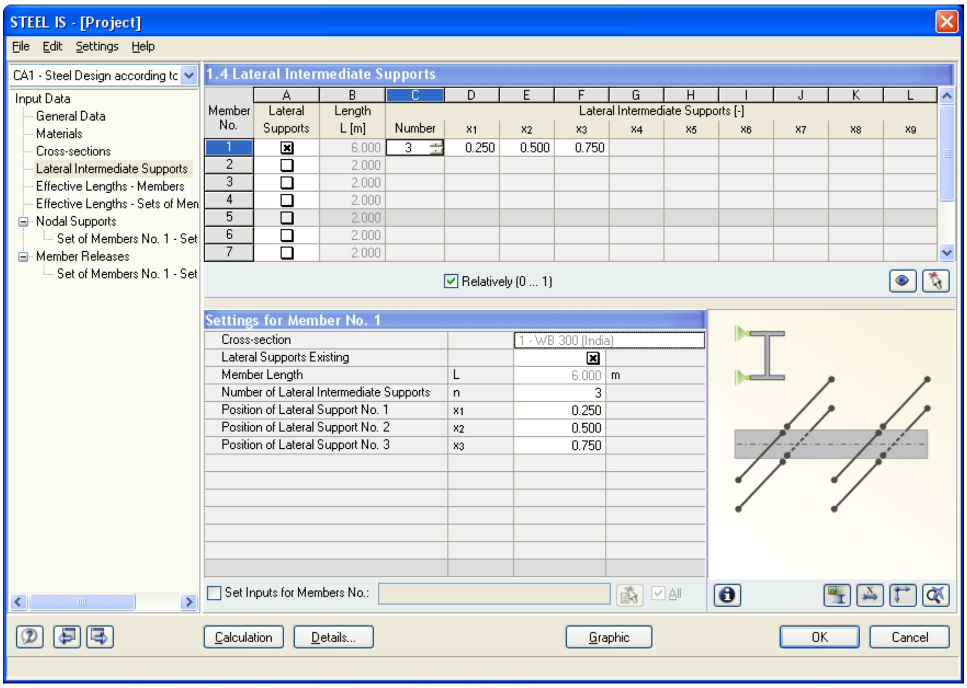


Figure 2.10: Form 1.4 *Lateral Intermediate Supports*

In the upper part of this form, up to nine lateral intermediate supports can be created per member. The lower part of the form displays the summary of the entered data for every single member.

Lateral intermediate supports can be defined either by directly entering the distances or by specifying the support locations *Relatively*. For the latter, it is necessary to tick the associated check box below the list. The relative distances of the supports are then calculated from the member lengths.

2.5 Effective Lengths - Members

The form 1.5 consists of two parts so that a good overview of the data is provided. In the upper table, the buckling length coefficients $k_{cr,y}$ and $k_{cr,z}$, the effective lengths KL_y and KL_z and the *Lateral-torsional Buckling* coefficients K and K_w are summarized for every member. In the lower part of this form, detailed information on the member that is selected in the upper table is displayed. The lower table contains all information about the relevant lengths of this member.

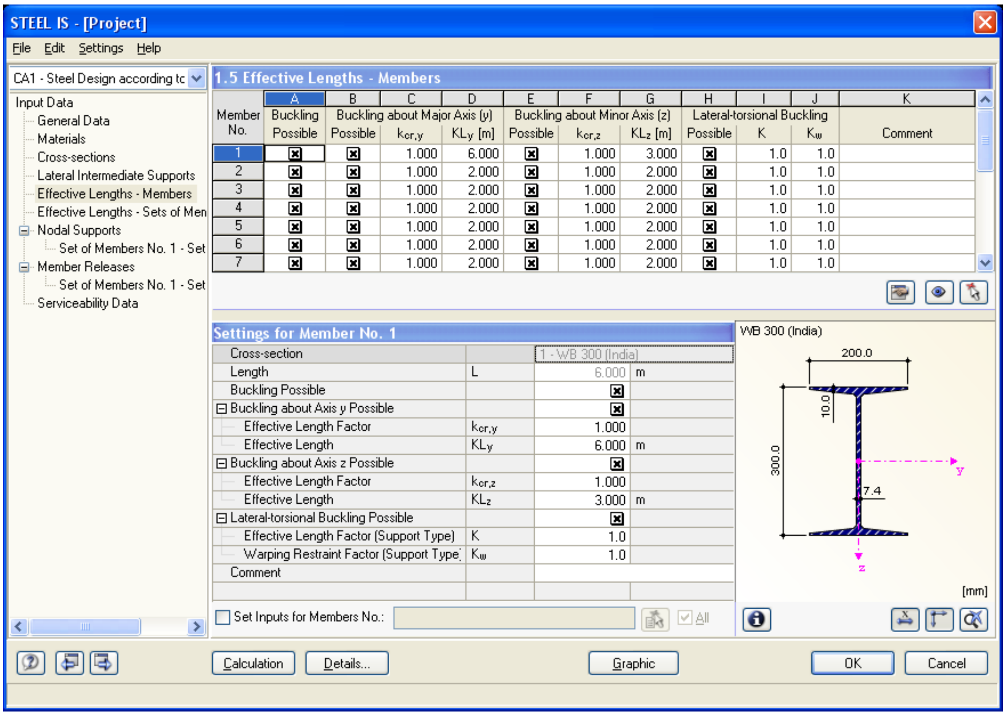


Figure 2.11: Form 1.5: *Effective Lengths - Members*

The effective lengths for the column buckling about the minor principal axis are automatically loaded from the previous form 1.4. If the member is divided into different lengths by lateral intermediate supports, then no values are displayed in the corresponding columns D and G of form 1.5. It is possible to change the buckling length coefficients both in the summary table in the upper part and in the detailed settings in the lower part. The data of the corresponding part of this form is then updated automatically. The buckling length of a member can also be defined graphically by using the function [Pick].

The tree structure in the table *Settings for Member No.* includes the following parameters:

- *Cross-section*
- *Length* (actual length of the member)
- *Buckling Possible* (cf column A)
- *Buckling about Axis y* (buckling lengths, cf columns B - D)
- *Buckling about Axis z* (buckling lengths, cf columns E - G)
- *Lateral-torsional Buckling* (buckling length coefficients, cf columns H - J)

It is also possible to modify the *Buckling Length Coefficients* in the relevant directions and decide whether the buckling design is to be executed. If a buckling length coefficient is changed, the respective effective member length is modified automatically.

Buckling Possible

For the buckling and the lateral torsional buckling design, it is necessary for the member to transfer compression forces. Members that cannot transfer compression forces due to their definition (e.g. tension members, elastic foundations, rigid couplings) are a priori excluded from the design in STEEL IS. In such a case, a corresponding comment is displayed in the column *Comment* for this member.

The column *Buckling Possible* enables you to classify specific members as compression ones or, alternatively, to exclude them from the design. Hence, the check boxes in column A and also in table *Settings for Member No.* control whether the input options for the buckling length parameters are accessible for a member.

Length

For your information, the actual length of the selected member is displayed in the lower table. It is not possible to modify this value.

Buckling about Axis y resp. Axis z

The columns *Buckling Possible* control whether members are prone to buckling about their axes y and/or axes z. The axis y represents the "major principal" member axis, the axis z the "minor principal" member axis. The buckling length factors k_{cr} can be freely chosen for the buckling about the major and minor axes.

The orientation of the member axes can be checked in the cross-section graphics of form 1.3 *Cross-Sections* (see Figure 2.6). In the RSTAB work window, which can be opened any time via the [Graphic] button, you can display the member axes in the *Display* navigator.

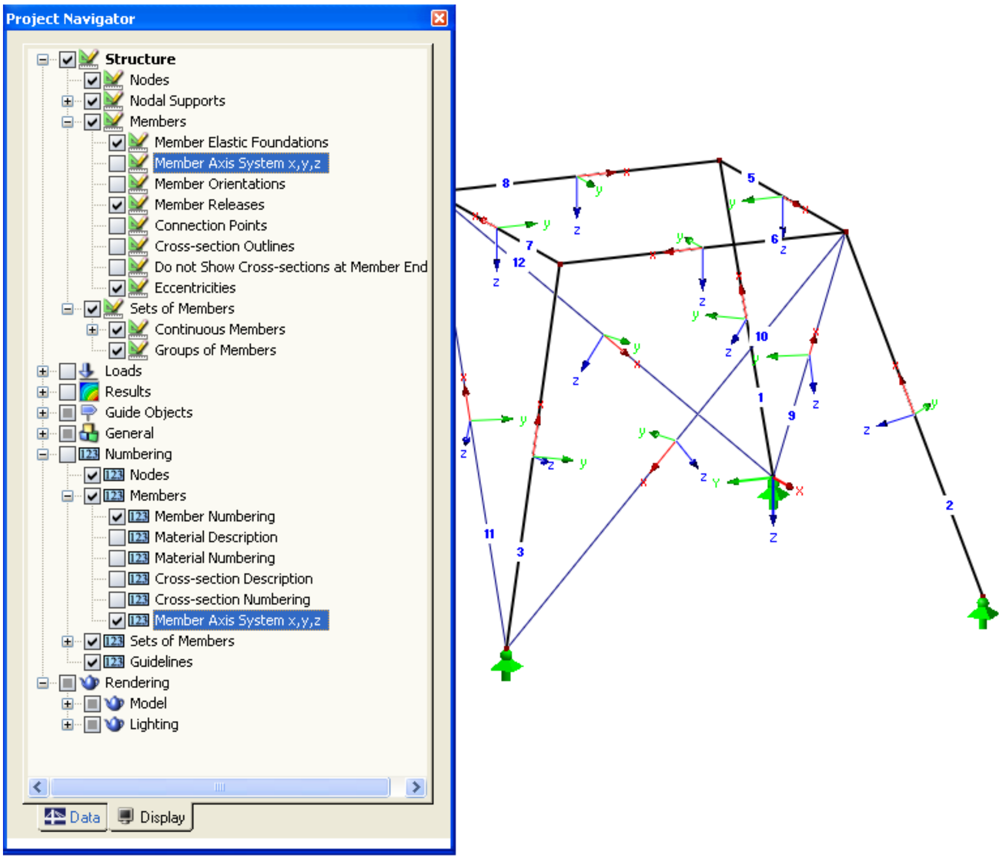


Figure 2.12: Displaying the Local Member Axes in the *Display* Navigator of RSTAB

If buckling is possible about one or both member axes, the precise values can be entered in columns C and D respectively F and G or in table *Settings for Member No.* below.

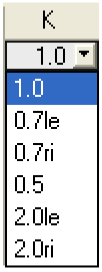
If you define the buckling length coefficient k_{cr}, the buckling length KL is determined by multiplying the member length L with this buckling length coefficient.

Via the [...] button at the end of the *KL* input field, you can select two nodes in RSTAB work window graphically. Their distance then defines the buckling length.

Lateral-Torsional Buckling

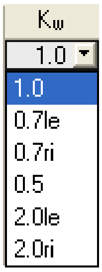
Column H controls whether a lateral-torsional buckling design is to be performed. In columns I and J, you can define the effective length factor *K* and warping restraint factor K_w. These two factors have an influence on the elastic critical moment at lateral-torsional buckling M_{cr}. To calculate M_{cr} via the eigenvalue method, an internal member model with four degrees of freedom has to be created. The following definitions of K and K_w are used to model the degrees of freedom at the supports of this internal member model.

The effective length factor **K** corresponds to the following definitions:



K = 1.0	forked support at both beam ends
K = 0.7le	restraint at left end and forked support at right end
K = 0.7ri	restraint at right end and forked support at left end
K = 0.5	restraint at both beam ends
K = 2.0le	restraint at left end and free right end
K = 2.0ri	restraint at right end and free left end

The warping restraint factor $\mathbf{K_w}$ corresponds to the following definitions:



K_w = 1.0	warp-free support at both beam ends
K_w = 0.7le	warping restraint at left end and forked support at right end
K_w = 0.7ri	warping restraint at right end and forked foundation at left end
K_w = 0.5	warping restraint at both beam ends
K_w = 2.0le	warping restraint at left end and free right end
K_w = 2.0ri	warping restraint at right end and free left end

A forked support with K = 1.0 represents a rigid support in the direction of the y-axis and a restriction of rotation around the x-axis (longitudinal axis) of the member. In the case of a restraint, the rotation about the z-axis is restricted additionally. The abbreviations "le" and "ri" characterize the location of the restraint. "Le" is related to the support at the member start, "ri" at its end. The orientation of the local member axes is shown in Figure 2.13.

By the factor K_w, the fourth degree of freedom on the support is defined. It has to be determined whether warping is possible for the cross-section or not. Regarding the fact that the internal member model uses only four degrees of freedom, no more degrees of freedom (displacements in directions x and z) have to be defined.

The coefficients K and K_w are related to the member start or member end. You have to be very careful if intermediate supports have been defined. Intermediate supports divide the member into several parts for the design. Thus, intermediate supports are to be avoided for cantilever beams because the result would be statically deficient pieces with fork-type supports on only one side each.

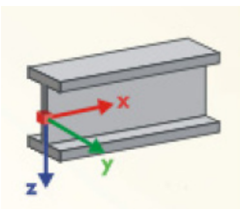


Figure 2.13: Member Axes for Factors K and K_w

The check box *Set Input for Members No.:* is located beneath the tree structure. If you tick this box, the data entered consequently will become valid for specific resp. *All* members. You can select the members graphically using the function [Pick] or enter their numbers manually. This option is useful when you want to assign the same boundary conditions to several members. Please notice that this function must be activated prior to data entering. If you define the data and choose this option later, the data will not be re-assigned.

Comment

You can insert your own remarks in the last column for every member, e.g. to explain the defined buckling lengths.

2.6 Effective Lengths - Sets of Members

The input form 1.6 controls the effective lengths for sets of members. It is only available if one or more sets of members have been selected in form 1.1 *General Data*.

This form is very similar to the previous form 1.5. With regard to the effective lengths for buckling about the major and minor axes of the cross-sections, it is identical to form 1.5. There are differences, however, as far as the parameters for lateral-torsional buckling are concerned. These are defined by means of specific boundary conditions in forms 1.7 and 1.8 (see Chapters 2.7 and 2.8).

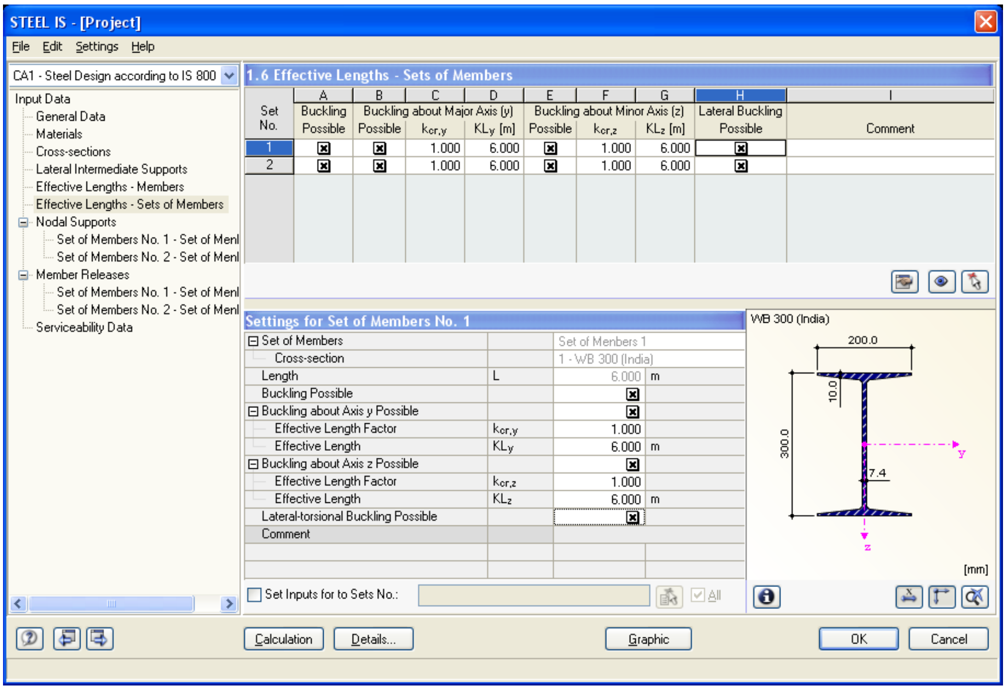


Figure 2.14: Form 1.6: *Effective Lengths - Sets of Members*

2.7 Nodal Supports - Sets of Members

The stability design is based on the loads and the boundary conditions of the selected sets of members. The value of the critical factor α_{cr} has to be determined for the entire set of members in order to obtain the critical moments M_{cr} which are necessary for the design. To determine this value, a planar member structure with four degrees of freedom per node is created. The specific support conditions are defined in form 1.7. This form is only available if you have selected one or more sets of members in form 1.1 *General Data*.

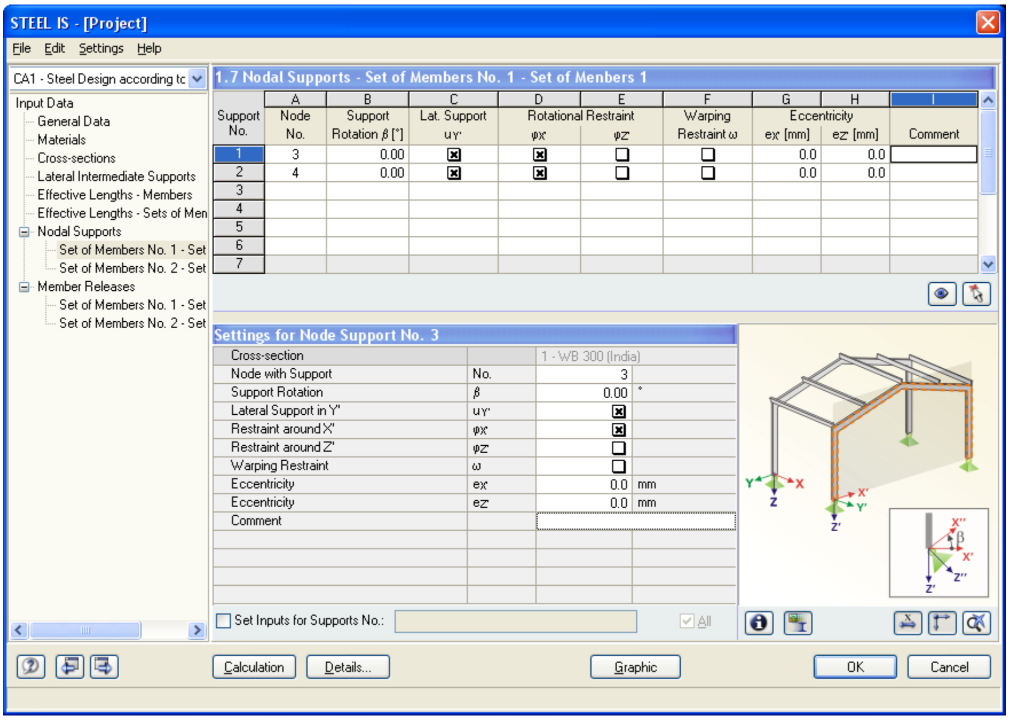


Figure 2.15: Form 1.7: *Nodal Supports - Set of Members*

To define the nodal supports, the orientation of the axes within a set of members is important. The program internally checks the location of the relevant nodes and then determines the axis system for the nodal supports that are to be defined in form 1.7 (see Figure 2.16 to Figure 2.19).

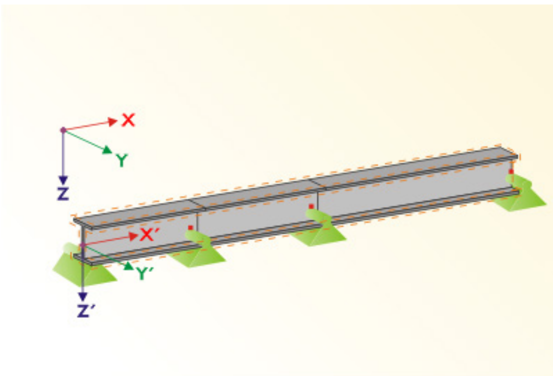


Figure 2.16: Auxiliary Coordinate System for Nodal Supports of Set of Members

If all members within the set of members lie on a straight line as we see in Figure 2.16, the local coordinate system of the first member in this set is applied as auxiliary coordinate system for the entire set of members.

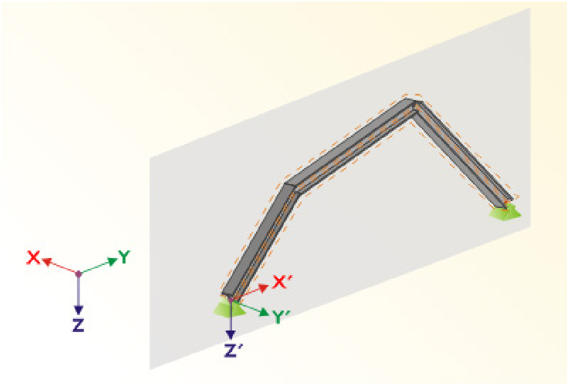


Figure 2.17: Auxiliary Coordinate System for Nodal Supports of Set of Members

Even if the members within a set do not lie on a straight line, they still must lie in a plane. We can see a vertical plane in Figure 2.17. In this case the axis X' is horizontal and in the plane direction. The axis Y' is also horizontal, but perpendicular to the axis X'. The axis Z' points vertically downwards.

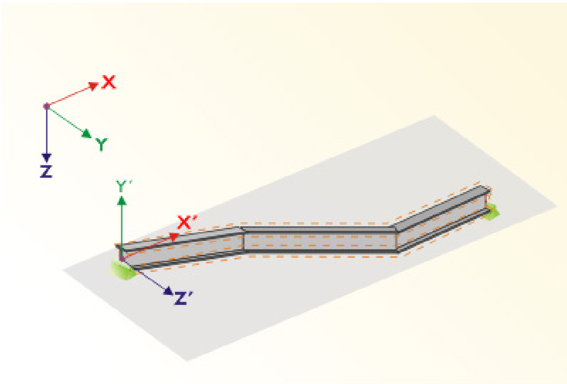


Figure 2.18: Auxiliary Coordinate System for Nodal Supports of Set of Members

If the members are located in a horizontal plane, the axis X' is parallel with the axis X of the global coordinate system. The axis Y' then points in the opposite direction of the global axis Z. The axis Z' is parallel with the axis Y of the global coordinate system.

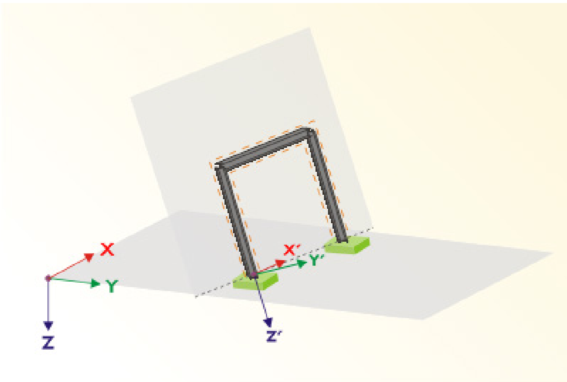


Figure 2.19: Auxiliary Coordinate System for Nodal Supports of Set of Members

Figure 2.19 shows the most general case. The members within a set of member do not lie on a straight line but are located in one oblique plane. The orientation of the axis X' is then determined by the intersection between the oblique and the horizontal plane. The axis Y' is perpendicular to the axis X' and is also perpendicular to the oblique plane. The axis Z' is perpendicular to the axes X' and Y'.

2.8 Member Releases - Sets of Members

This form is only available if one or more sets of members have been selected in form 1.1 *General Data*. If any member in a given set is not able to transfer internal forces corresponding to the degrees of freedom restricted in form 1.7, then nodal releases can be inserted to a set of members in form 1.8. There is also the possibility to exactly define on which side the release is to act or to place a release at both sides.

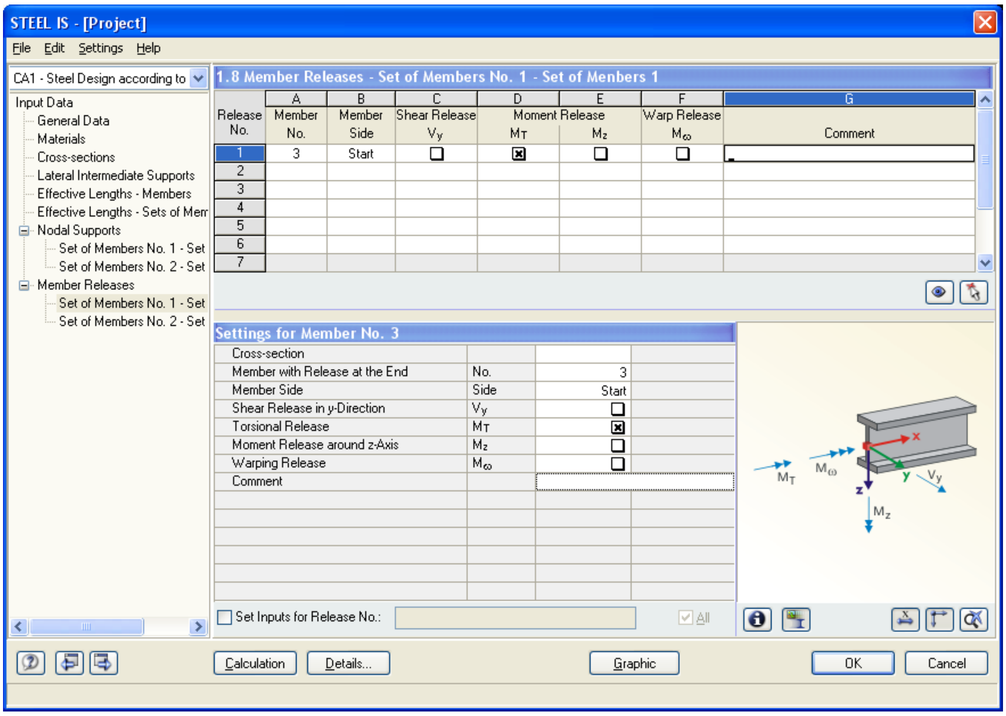


Figure 2.20: Form 1.8: *Member Releases - Set of Members*

2.9 Serviceability Data

The final input form includes different possibilities for the serviceability design. It is only displayed if the serviceability limit state design has been selected in form 1.1 *General Data* (cf Figure 2.3).

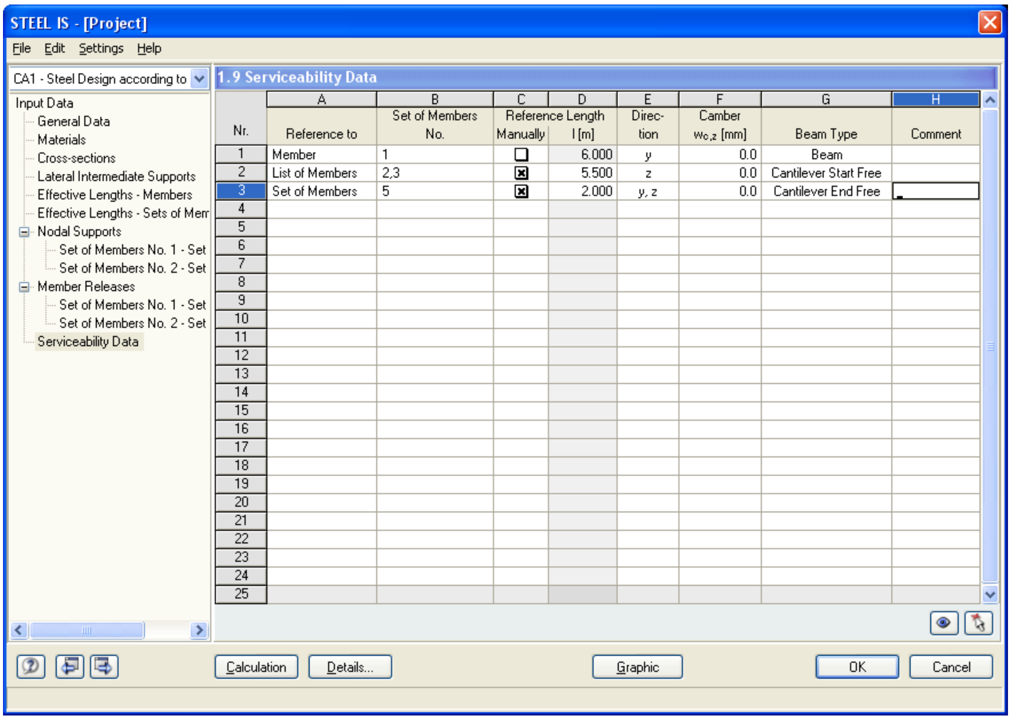


Figure 2.21: Form 1.9: *Serviceability Data*

In column A, you can refer the deformation to individual members, lists of members or sets of members. In column B, the relevant members or sets of members can be selected graphically by using the function [Pick]. The reference lengths *l* in column D are then filled automatically. The *Reference Length* is set as the length of the member or the entire length of the set of members resp. list of members. It can be changed *Manually* by using the corresponding checkbox in column C and setting the value in column D.

In column E, you specify the governing *Direction* for the serviceability design. Column F controls whether *Camber* is to be taken into account as well.

For a correct determination of the serviceability limit states, the *Beam Type* (beam or cantilever) is very important. It can be entered in column G.

3. Calculation

3.1 Details

A particular design is carried out with the internal forces calculated in the RSTAB program. Before the [Calculation] you should check the detailed setting for the design. Open the appropriate dialog from every input or output mask by clicking on the [Details…] button.

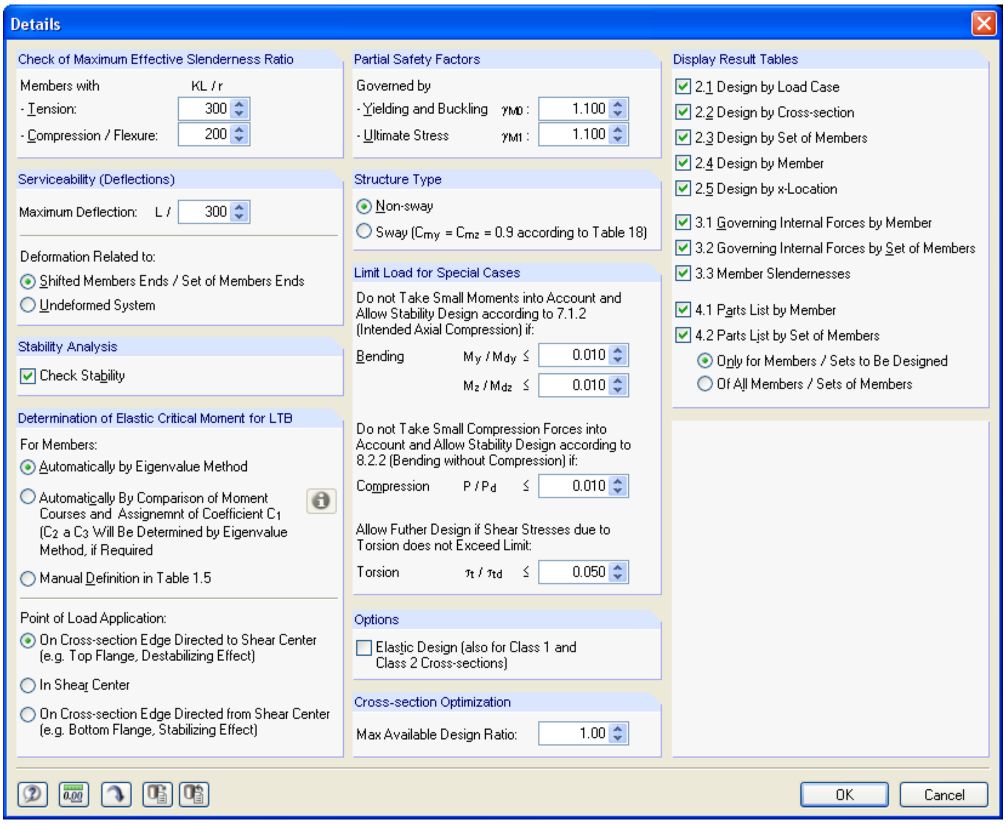


Figure 3.1: *Details* Dialog

Check of Maximum Effective Slenderness Ratio

It is possible to define the maximum effective slenderness ratio for members with tension and compression or flexure. These maximum values are compared with the actual member slendernesses in table 3.3 which is available after the calculation (see Chapter 4.8).

Serviceability Limits (Deflections)

In section *Serviceability Limits*, it is possible to set the maximum allowable deflection used in for the serviceability limit state design. The two selection fields below control whether the *Deformation* is to be related to the undeformed model or to an imaginary connecting line between the shifted start and end nodes of the member resp. set of members within the deformed structure.

Stability Analysis

This section controls globally whether an analysis of stability is to be carried out. If the box is unchecked, the input forms 1.4 to 1.8 are not active.

Determination of Elastic Critical Moment for LTB

The option to calculate M_{cr} *Automatically by Eigenvalue Method* is set by default. It starts from the internal model of finite elements, regarding the following items:

- Dimensions of gross cross-section
- Load type and location of load application point
- Real distribution of moments
- Lateral forced deformations (on basis of support conditions)
- Real boundary conditions

The degrees of freedom of the internal member model are defined by the coefficients K and K_w (see Chapter 2.5).

If the second method of the automatic calculation of the elastic critical moment is chosen, then the calculation of the coefficient C_1 can be influenced via the [Info…] button. The coefficients C_2 and C_3 are calculated automatically by the eigenvalue method again. If there are loads applied on a member, their application point has to be specified because they can have stabilizing or destabilizing effects, subject to the application point. The *Point of Load Application* can be set below.

Partial Safety Factors

It is possible to separately modify the partial safety factors of the material γ_{m0} and γ_{m1} for *Yielding and Buckling* or for *Ultimate Stress*. These factors are used in all cross-section and stability checks.

Structure Type

The structure type can be either *Non-sway* or *Sway*, which affects the calculation of C_{my} and C_{mz}. For a sway-type structure, the values of C_{my} and C_{mz} are assumed as 0.9.

Limit Load for Special Cases

It is possible to neglect small stresses due to bending, compressive forces and torsion and, thus, allow a simplified design which eliminates negligible internal forces. In this dialog section, the limits of these internal forces or stresses can be entered. Those are defined as the ratios between existing internal forces or stresses and the corresponding resistances of each cross-section.

Options

Cross-sections of the class 1 or 2 are not designed elastically in STEEL IS. If needed, the elastic design can be activated for those cross-section classes.

Cross-section Optimization

Cross-sections can be optimized if the *Optimize* option is chosen in form 1.3 *Cross-sections* (see Figure 2.6). The dialog *Details* enables you to set the maximum allowable design ratio as a limit for the optimization process.

Display Results Tables

In this section, the results tables can be specified which are to be displayed, inclusive of a parts list. The individual results tables are described in Chapter 4.

3.2 Start Calculation

In all input forms of STEEL IS, you can start the design by clicking the [Calculation] button.

At first, STEEL IS searches for the results of the selected load cases, groups and combinations of load cases. If they are not found, the calculation of the governing internal forces in RSTAB is started. The calculation parameters of RSTAB are used for this analysis.

If cross-sections are to be optimized (see chapter 7.2), the required sections are calculated and relevant designs are carried out.

The STEEL IS design can be also started from the RSTAB interface. All design cases of the add-on modules are displayed in the *To Calculate* dialog similar to load cases or groups. Open this dialog in RSTAB via the main menu

Calculate → **To Calculate**...

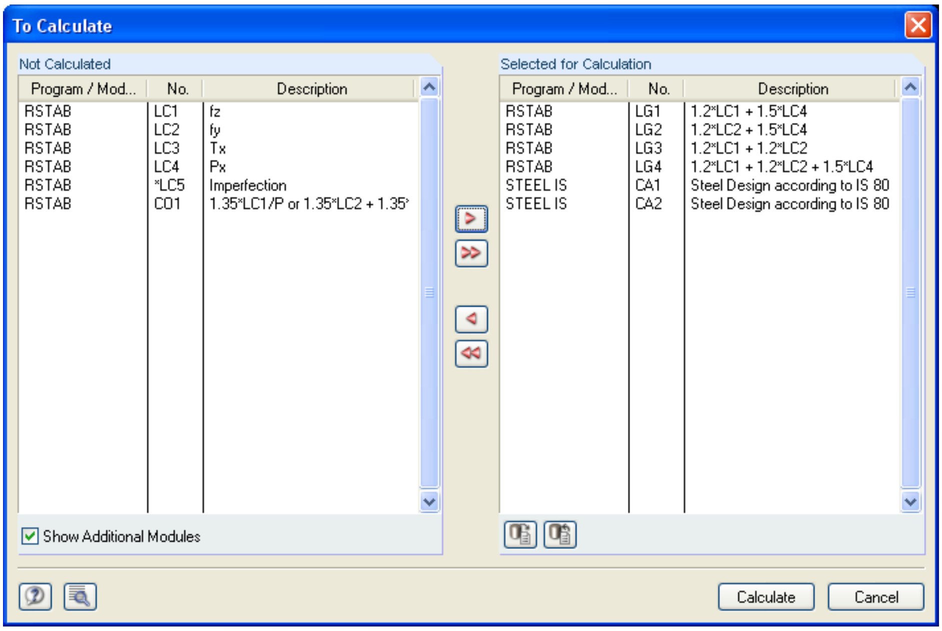


Figure 3.2: *To Calculate* Dialog

If the design cases of STEEL IS are missing in the list *Not Calculated*, it is necessary to tick the check box *Show Additional Modules*.

The [▶] button transfers selected design cases to the list on the right. You can then start the calculation by the [Calculate] button.

The calculation of a specific STEEL IS design case can also be directly started from the toolbar. Set the required design case in the list and then click on the [Results on/off] button.

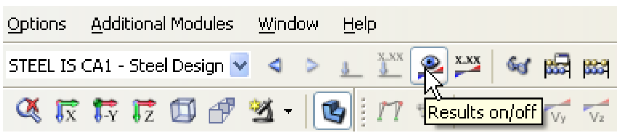


Figure 3.3: Direct Calculation of Design Case from STEEL IS in RSTAB

A dialog appears in which you can watch the design process.

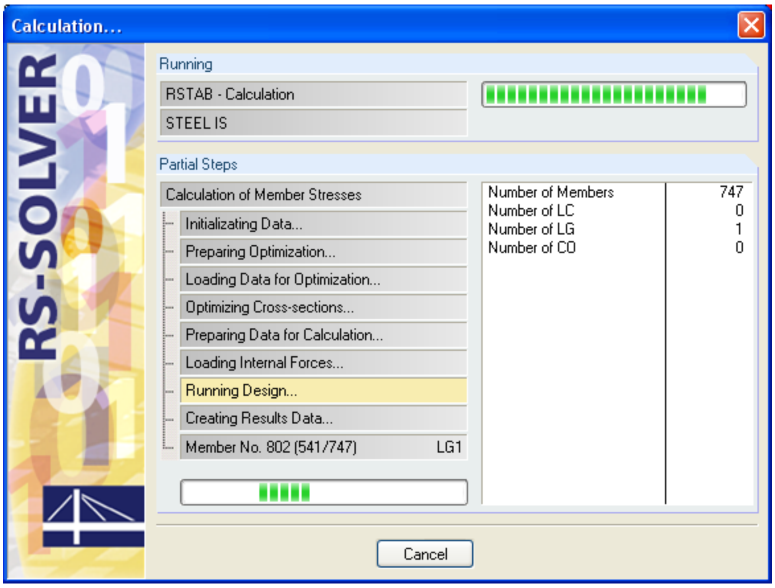


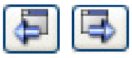
Figure 3.4: Calculation in STEEL IS

4. Results

4.1 Design by Load Case

Table 2.1 *Design by Load Case* is displayed immediately after the design. In the upper part of this table, a summary of all designs for every load case, load group and combination is displayed. The lower part includes all details of the material properties, design internal forces and design data of the load case which is selected in the upper part of this table.

The results tables 2.1 to 2.5 contain the detailed design summaries according to different selection criteria. Tables 3.1 and 3.2 include the governing internal forces. In table 3.3, the member slendernesses are compared with the maximum values as set in dialog *Details* (see Chapter 3.1). The parts lists are displayed in the last two tables 4.1 and 4.2.



The results tables are accessible from the navigator in STEEL IS. You can also switch among the tables by using the buttons visible on the left or the functional keys [F2] and [F3].

Save the results by the [OK] button and close STEEL IS.

In this chapter, we describe the particular tables in the given order. The following chapter 5 *Evaluation of Results* is devoted to the evaluation and checking of results.

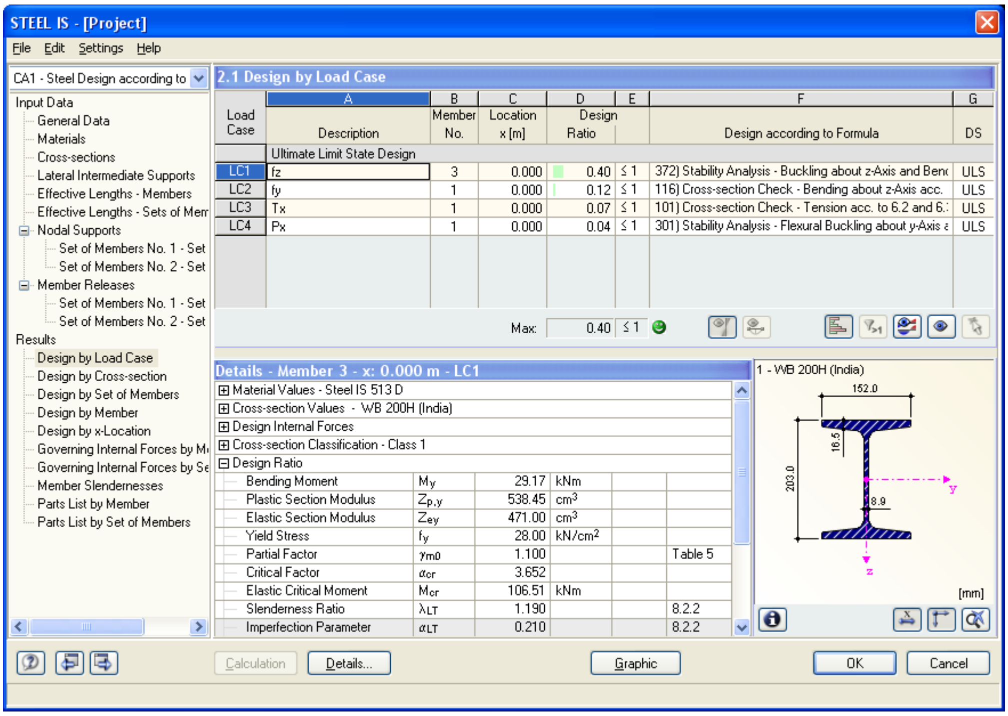


Figure 4.1: Table 2.1 *Design by Load Case*

Description

The descriptions of the load cases, load groups and combinations that are decisive for every relevant type of design are displayed in this column.

Member No.

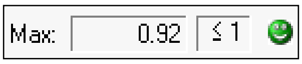
The number of the member with the highest design ratio is stated for every designed load case, load group or load combination.

Location x

The location x on the member where the maximum stress ratio occurs is displayed in this column. The following locations *x* on the member are taken into account:

- Start and end nodes
- Internal nodes according to a potential user-defined member division
- Extreme values of internal forces

Design



For every design type and for every load case, load group or load combination the design ratios according to the standard are displayed.

Design according to Formula

In this column, the equations that were followed in the design are displayed.

DS

The *Design Situation* which is relevant for the design is stated in this column. The first letter in the shortcuts *U* resp. *S* stands for "Ultimate Limit State" resp. "Serviceability Limit State", the second letter stands for the decisive combination of actions.

4.2 Design by Cross-Section

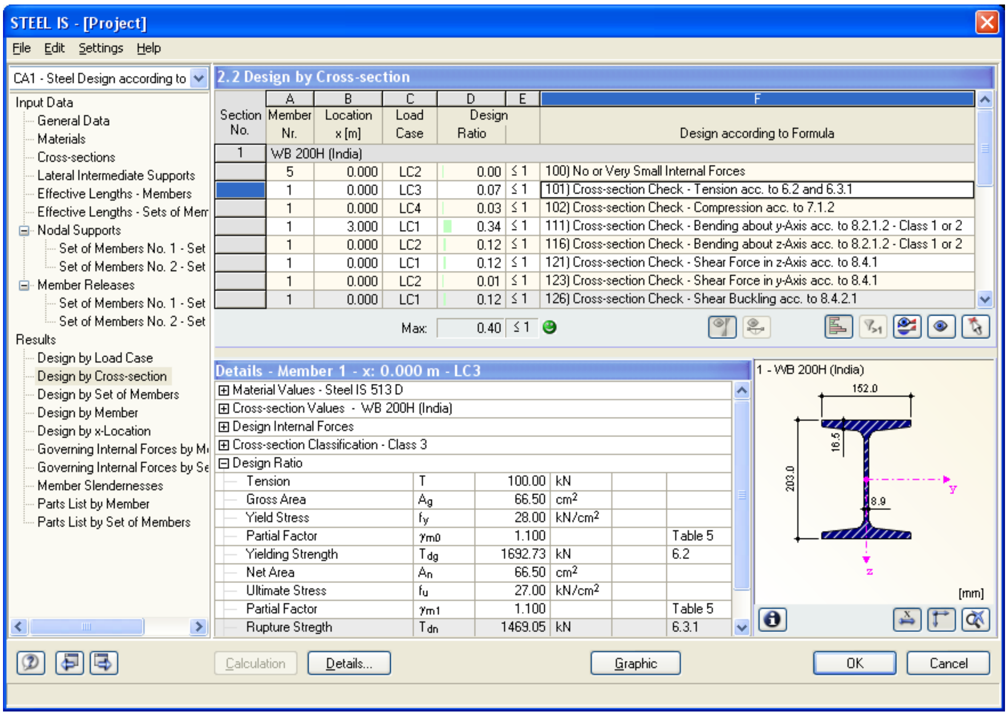


Figure 4.2: Table 2.2 *Design by Cross-section*

In this table, the maximum stress ratios are displayed for all designed members and for all designed load cases, load groups and combinations. The results are sorted according to cross-sections. For tapered members, both cross-section descriptions are shown in the line next to the cross-section number.

4.3 Design by Set of Members

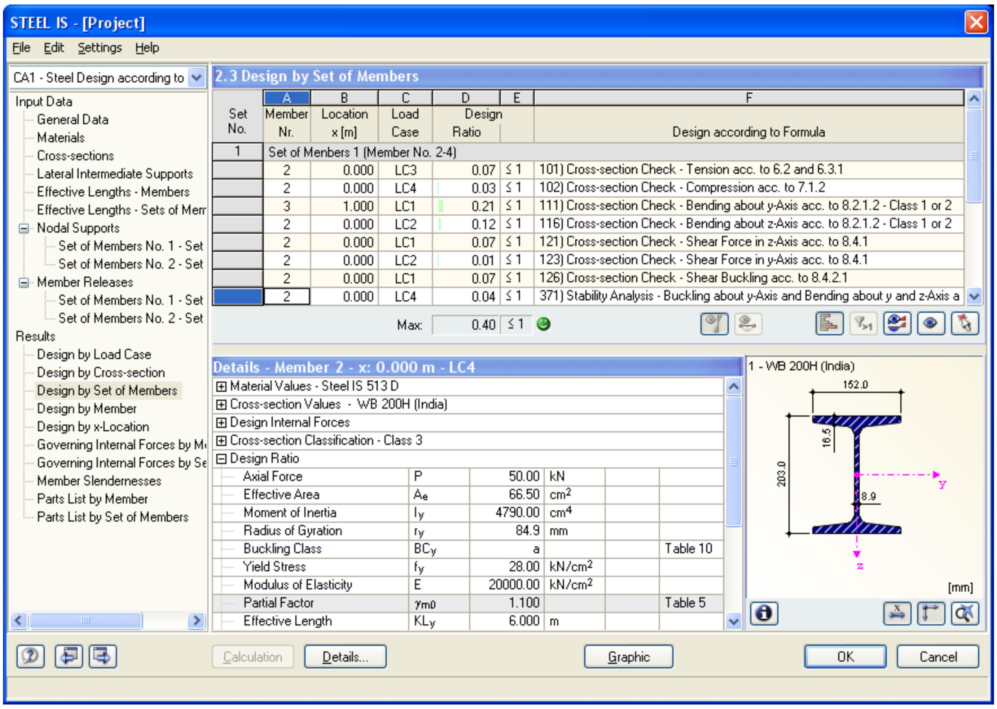


Figure 4.3: Table 2.3 *Design by Set of Members*

This table is displayed if at least one set of members was selected for design. The maximum design ratios are listed according to sets of members. The number of the member with the highest design ratio within each set of members is shown as well.

4.4 Design by Member

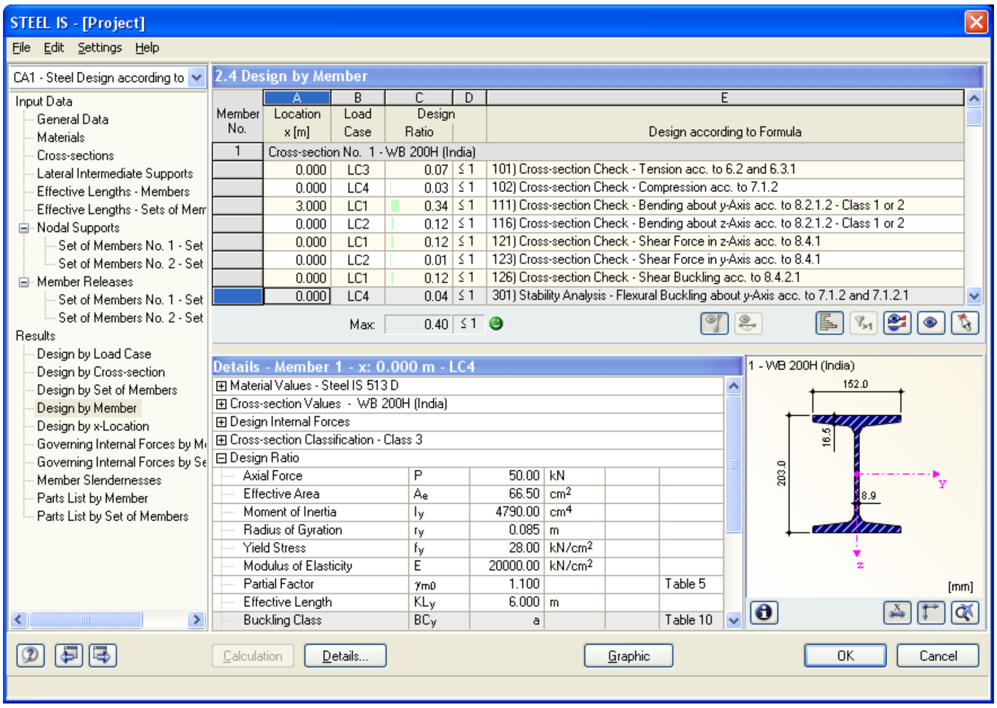


Figure 4.4: Table 2.4 *Design by Member*

In this table, the maximum design ratios are arranged according to member numbers. The *Location x* at which the maximum value occurs is stated for every member.

The description of the individual columns can be found in chapter 4.1.

4.5 Design by x-Location

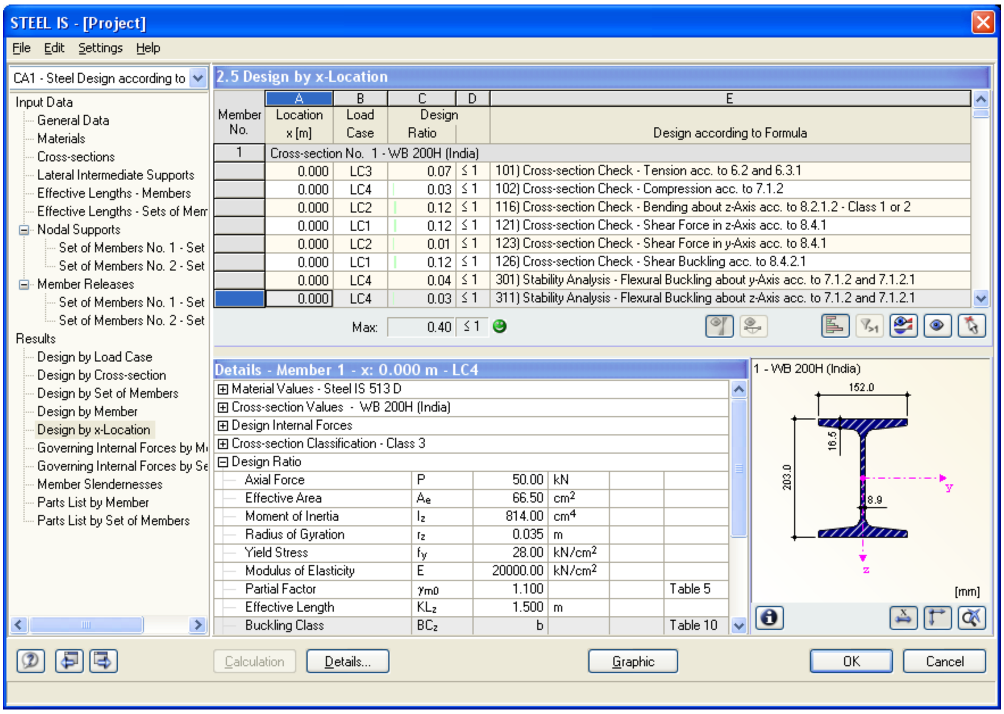


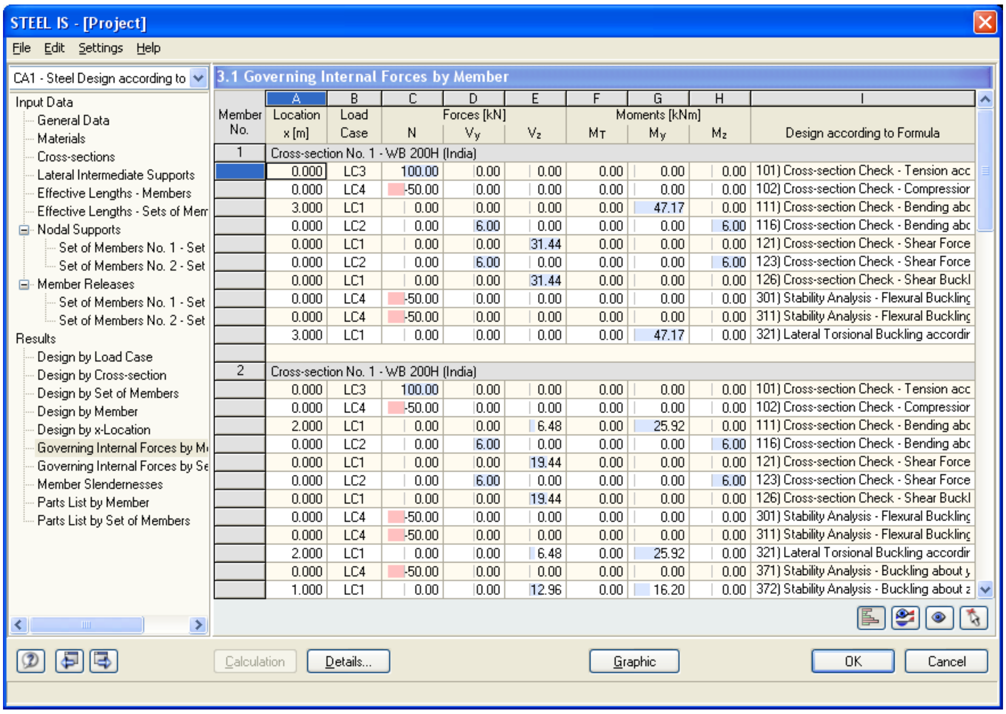
Figure 4.5: Table 2.5 *Design by x-Location*

This results table lists the maximum values of every member at the following locations x according to the member division points of RSTAB:

- Start and end nodes
- Internal nodes according to a potential user-defined member division
- Division points according to the number of member divisions that were set in the *Calculation Parameters* dialog in the *Options* register in RSTAB
- Extreme values of internal forces

4.6 Governing Internal Forces by Member

In this table, the governing internal forces are shown which lead to the maximum design ratios.



Member No.	A Location x [m]	B Load Case	C Forces [kN] N	D V_y	E V_z	F Moments [kNm] M_T	G M_y	H M_z	I Design according to Formula
1	Cross-section No. 1 - WB 200H (India)								
	0.000	LC3	100.00	0.00	0.00	0.00	0.00	0.00	101) Cross-section Check - Tension acc
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	102) Cross-section Check - Compression
	3.000	LC1	0.00	0.00	0.00	0.00	47.17	0.00	111) Cross-section Check - Bending abc
	0.000	LC2	0.00	6.00	0.00	0.00	0.00	6.00	116) Cross-section Check - Bending abc
	0.000	LC1	0.00	0.00	31.44	0.00	0.00	0.00	121) Cross-section Check - Shear Force
	0.000	LC2	0.00	6.00	0.00	0.00	0.00	6.00	123) Cross-section Check - Shear Force
	0.000	LC1	0.00	0.00	31.44	0.00	0.00	0.00	126) Cross-section Check - Shear Buckl
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	301) Stability Analysis - Flexural Buckling
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	311) Stability Analysis - Flexural Buckling
	3.000	LC1	0.00	0.00	0.00	0.00	47.17	0.00	321) Lateral Torsional Buckling accordir
2	Cross-section No. 1 - WB 200H (India)								
	0.000	LC3	100.00	0.00	0.00	0.00	0.00	0.00	101) Cross-section Check - Tension acc
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	102) Cross-section Check - Compression
	2.000	LC1	0.00	0.00	6.48	0.00	25.92	0.00	111) Cross-section Check - Bending abc
	0.000	LC2	0.00	6.00	0.00	0.00	0.00	6.00	116) Cross-section Check - Bending abc
	0.000	LC1	0.00	0.00	19.44	0.00	0.00	0.00	121) Cross-section Check - Shear Force
	0.000	LC2	0.00	6.00	0.00	0.00	0.00	6.00	123) Cross-section Check - Shear Force
	0.000	LC1	0.00	0.00	19.44	0.00	0.00	0.00	126) Cross-section Check - Shear Buckl
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	301) Stability Analysis - Flexural Buckling
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	311) Stability Analysis - Flexural Buckling
	2.000	LC1	0.00	0.00	6.48	0.00	25.92	0.00	321) Lateral Torsional Buckling accordir
	0.000	LC4	-50.00	0.00	0.00	0.00	0.00	0.00	371) Stability Analysis - Buckling about y
	1.000	LC1	0.00	0.00	12.96	0.00	16.20	0.00	372) Stability Analysis - Buckling about z

Figure 4.6: Table 3.1 *Governing Internal Forces by Member*

Location x

For every member, the location x on the member with the maximum design ratio is shown.

Load Case

In this column, the numbers of the load cases, load groups or combinations whose internal forces have the most unfavorable effect are displayed.

Forces / Moments

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

Design according to Formula

The last column includes the relevant equations that were followed in the design.

4.7 Governing Internal Forces by Set of Members

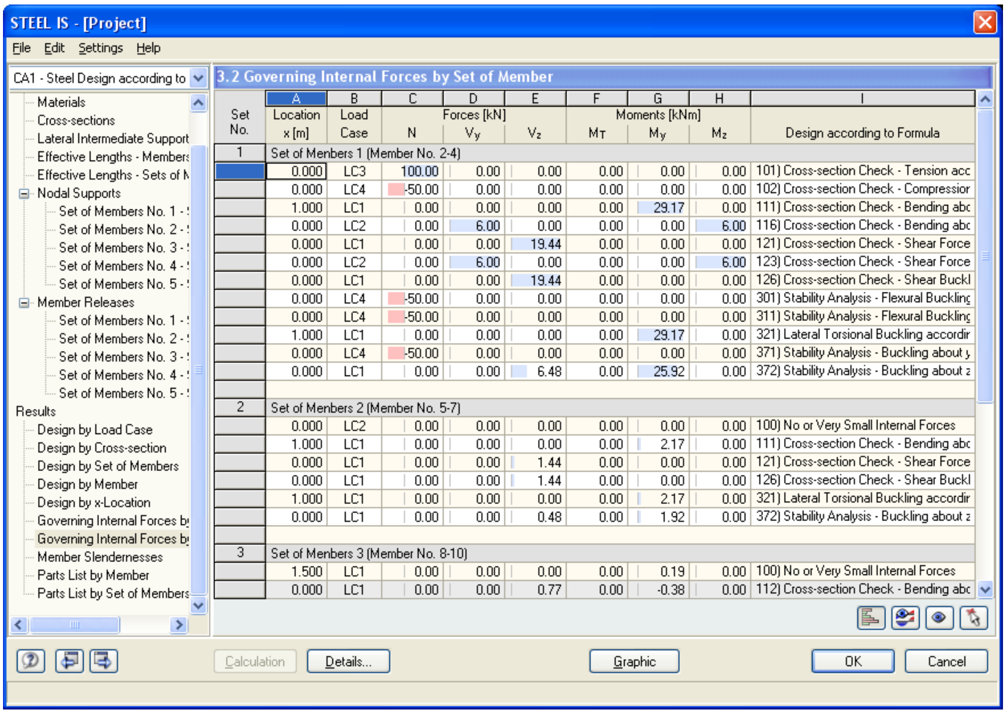


Figure 4.7: Table 3.2 *Governing Internal Forces by Set of Members*

In this results table, the governing internal forces that lead to the maximum design ratios of every set of members are shown.

4.8 Member Slendernesses

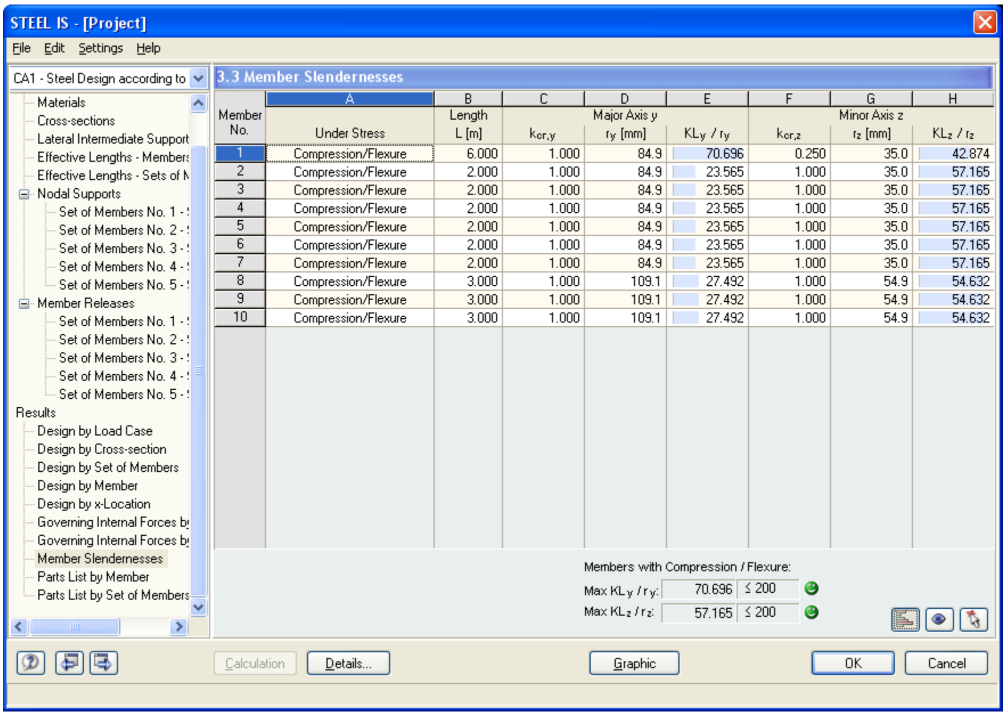
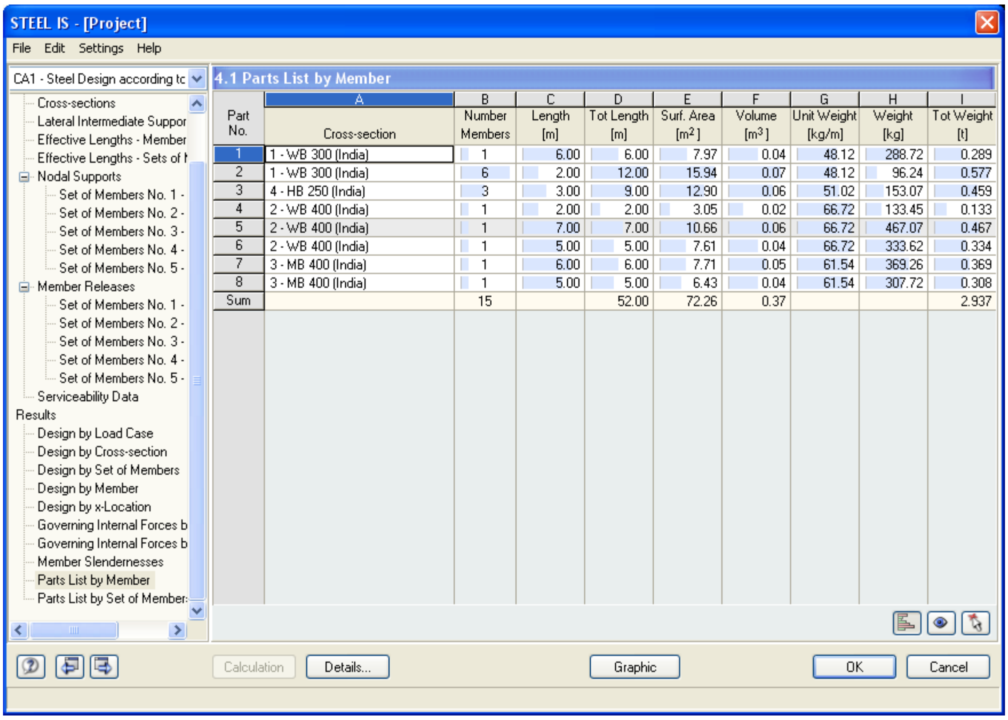


Figure 4.8: Table 3.3 *Member Slendernesses*

In table 3.3, the effective slenderness ratios of all designed members are compared with the maximum values that were set in the dialog *Details* (see Chapter 3.1). These ratios are listed with respect to the major and minor principal axes. This table provides information on the maximum effective slenderness ratios only. It does not give any design results. Members of the types *tension* or *cable* are excluded from this table.

4.9 Parts List by Member



Part No.	Cross-section	Number Members	Length [m]	Tot Length [m]	Surf. Area [m²]	Volume [m³]	Unit Weight [kg/m]	Weight [kg]	Tot Weight [t]
1	1 - WB 300 (India)	1	6.00	6.00	7.97	0.04	48.12	288.72	0.289
2	1 - WB 300 (India)	6	2.00	12.00	15.94	0.07	48.12	96.24	0.577
3	4 - HB 250 (India)	3	3.00	9.00	12.90	0.06	51.02	153.07	0.459
4	2 - WB 400 (India)	1	2.00	2.00	3.05	0.02	66.72	133.45	0.133
5	2 - WB 400 (India)	1	7.00	7.00	10.66	0.06	66.72	467.07	0.467
6	2 - WB 400 (India)	1	5.00	5.00	7.61	0.04	66.72	333.62	0.334
7	3 - MB 400 (India)	1	6.00	6.00	7.71	0.05	61.54	369.26	0.369
8	3 - MB 400 (India)	1	5.00	5.00	6.43	0.04	61.54	307.72	0.308
Sum		15		52.00	72.26	0.37			2.937

Figure 4.9: Table 4.1 *Parts List by Member*

Finally, the parts list of all cross-sections that are considered in the given design case is displayed. This list contains only designed members by default. If all members of the structure are to be included, you can modify the setting in the *Details* dialog (see Figure 3.1, page 26) that can be opened by the [Details] button.

Part No.

The same part number is automatically assigned to identical members.

Cross-section

In this column, the cross-section descriptions are displayed.

Number of Members

The number of identical members is given for each part.

Length

This column displays the unit lengths of every single member.

Total Length

This column represents the product of the values given in the two previous columns.

Surface Area

The surface area which is related to the total length of the relevant part is calculated on the basis of the value A_{Surf} of each cross-section. You can check on this value by clicking on the [Info about Current Cross-Section...] button in tables 1.3 or 2.1 to 2.5.

Volume

The volume of every part is calculated from the surface area and the total length.

Unit Weight

The *Unit Weight* of the cross-section represents the weight per length of 1 m. In case of tapered cross-sections, the unit weight is calculated as the mean value of both cross-sections.

Weight

The value in this column is calculated as the product of values in the columns C and G.

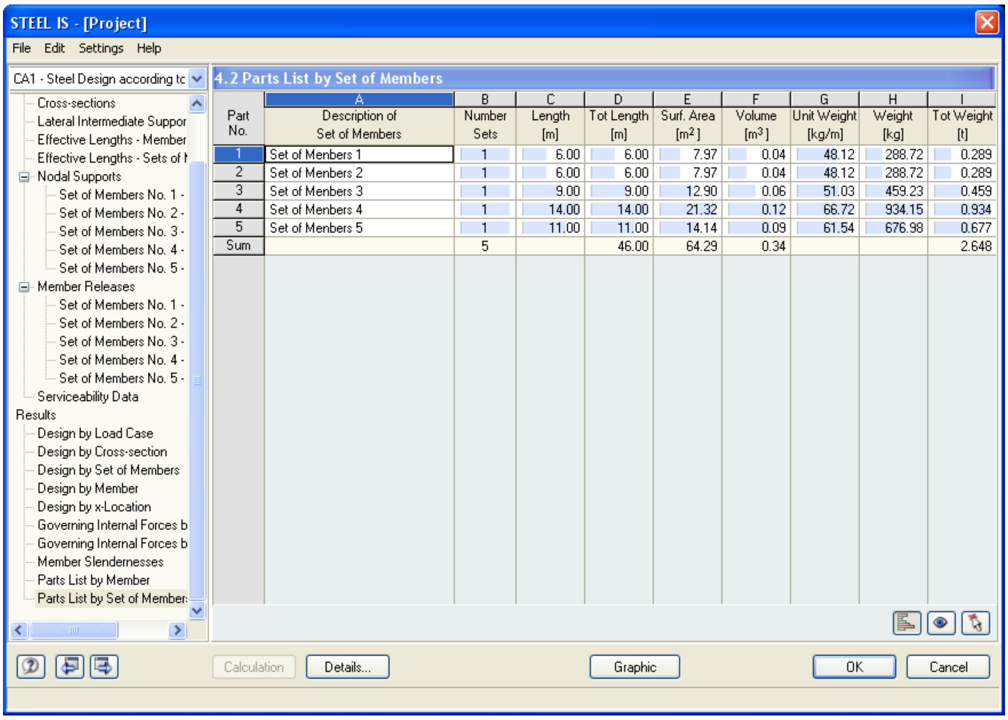
Total Weight

The total weight of each part is displayed in the last column.

Sum

The sums of the values in the individual columns are given in the final row of this list. The cell *Total Weight* shows the total required amount of steel.

4.10 Parts List by Set of Members



Part No.	A Description of Set of Members	B Number Sets	C Length [m]	D Tot Length [m]	E Surf. Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Tot Weight [t]
1	Set of Members 1	1	6.00	6.00	7.97	0.04	48.12	288.72	0.289
2	Set of Members 2	1	6.00	6.00	7.97	0.04	48.12	288.72	0.289
3	Set of Members 3	1	9.00	9.00	12.90	0.06	51.03	459.23	0.459
4	Set of Members 4	1	14.00	14.00	21.32	0.12	66.72	934.15	0.934
5	Set of Members 5	1	11.00	11.00	14.14	0.09	61.54	676.98	0.677
Sum		5		46.00	64.29	0.34			2.648

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

The last mask in STEEL IS is presented when at least one set of members was selected for the design. The advantage of this table is that a parts list is given for the various groups of elements (e.g. for a beam).

The table columns are described in Chapter 4.9. If there are different cross-sections within the set of members, the mean values of surface area, volume and unit weight are listed.

5. Evaluation of Results

The design values can be evaluated in different ways. For this, the buttons in the results tables are very useful which are located below the upper tables each.

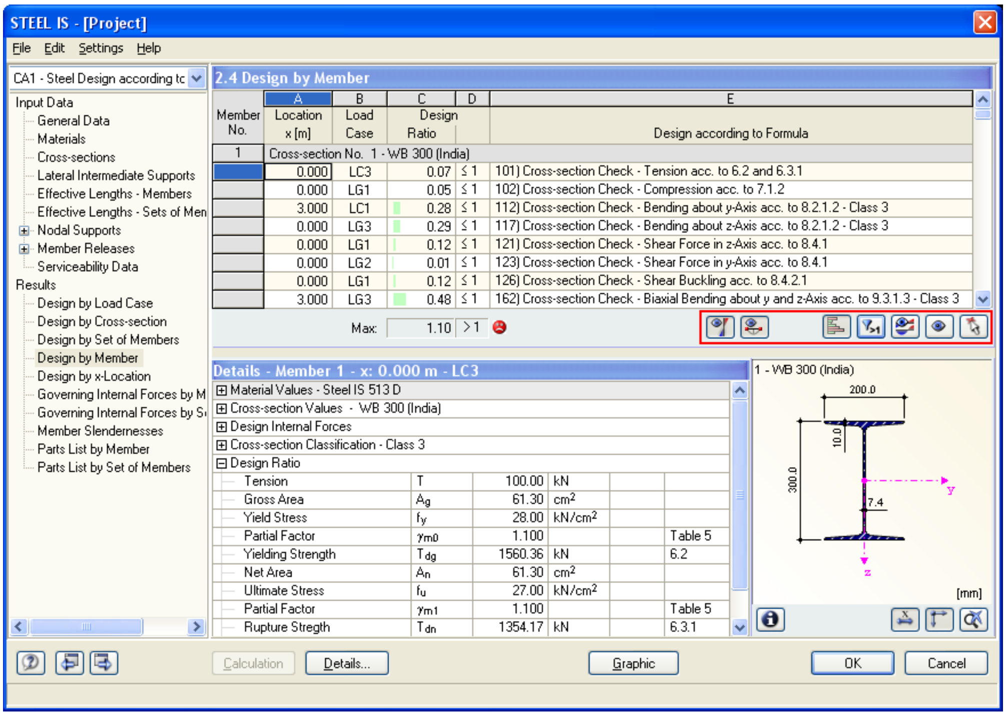


Figure 5.1: Buttons for Evaluation of Results

These buttons have the following functions:

Button	Name	Function
	Design of Ultimate Limit State	Switch on/off the design results of the ultimate limit state
	Design of Serviceability Limit State	Switch on/off the design results of the serviceability limit state
	Show Color Bars in Table	Switch on/off the color background in the results tables according to the reference scale
	Show Rows with Ratio > 1	Show only rows with stress ratios greater than 1 and, accordingly, the failed design
	Show Result Diagrams of Current Member	Open the diagram *Result Diagram on Member* → Chapter 5.2.
	Jump to Graphics to Change View	Go to the RSTAB work window in order to change the display settings
	Pick Member in Graphics and Go to This Member in Table	Click on a specific member in the RSTAB window whose results values are to be displayed in the table

Table. 5.1: Buttons in Results Tables 2.1 to 2.5

5.1 Results on RSTAB Model

You can use the RSTAB work window to evaluate the design results. The RSTAB graphics in the background can be useful if you want to check the location of a specific member in the model: the member that is selected in the STEEL IS results table is also highlighted in the selection color in the RSTAB background graphics. Additionally, an arrow marks the member location x which is stated as decisive in the selected line.

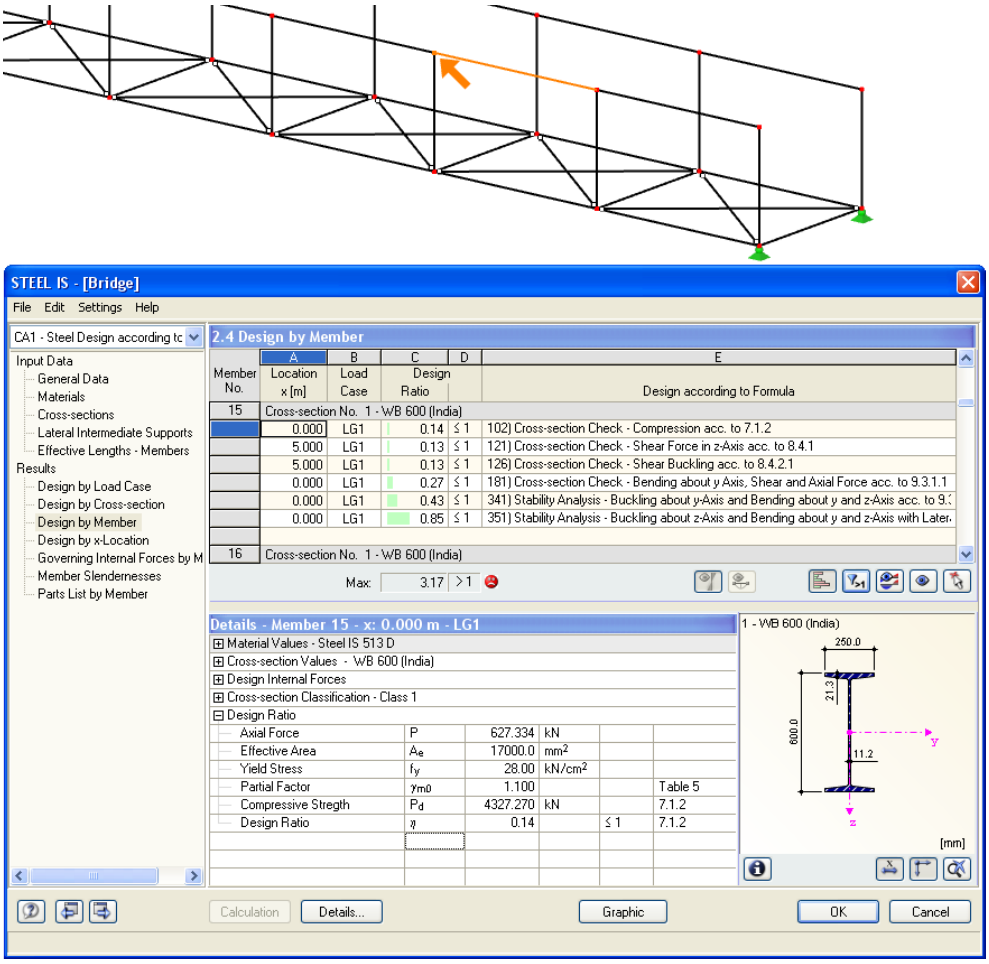
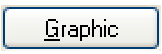


Figure 5.2: Selection of Member and Current *Location x* in RSTAB Model

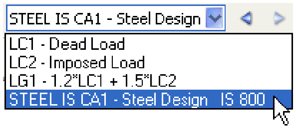
If you do not get a favorable view even by moving the STEEL IS window, you can apply the so-called *View Mode* by clicking on the [Jump to Graphics to Change View] button: the STEEL IS window is switched off and you can change the view of the RSTAB model. In this mode, the functions from the *View* menu are available, e.g. zoom, move or rotate the view.



The design ratios can also be displayed on the structural model. Close STEEL IS via the [Graphic] button. The design ratios are then shown graphically in the RSTAB work window.

Similar to the internal forces of RSTAB, you can activate or deactivate the design results by the [Results on/off] button. The [Show Result Values] button controls the display of the numerical values in the graphics.

Regarding the fact that the RSTAB tables are irrelevant to evaluate the STEEL IS results, you can deactivate them by using the button visible on the left.



A particular design case can be selected from the list of cases in the RSTAB toolbar.

The display of the results can be also controlled by the *Display* navigator, using the entry *Results* → *Members*. The design ratio is displayed *Two Colored* by default.

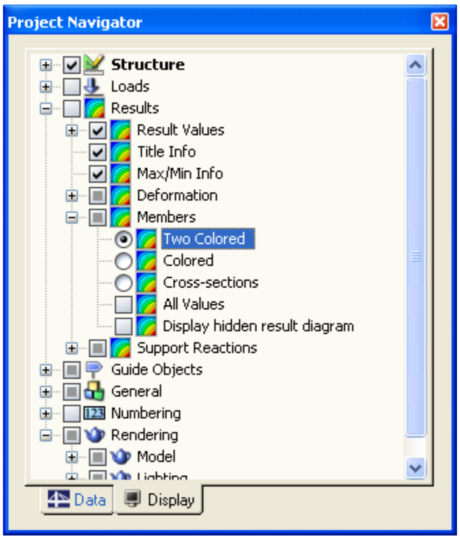
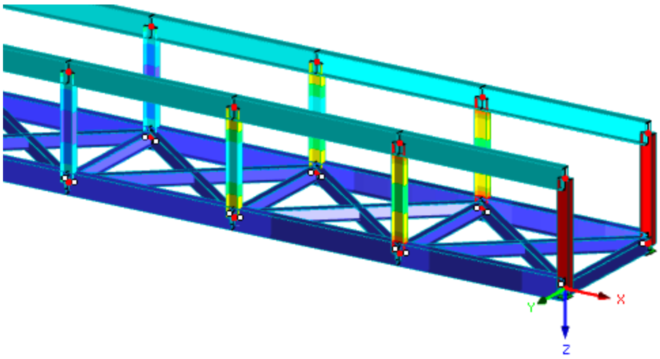


Figure 5.3: *Display* Navigator: Results → Members → **Two Colored**

If you select the *Colored* results display, the panel colors become available with various options for the multicolor display. Those are described in Chapter 4.4.6 of the RSTAB manual.



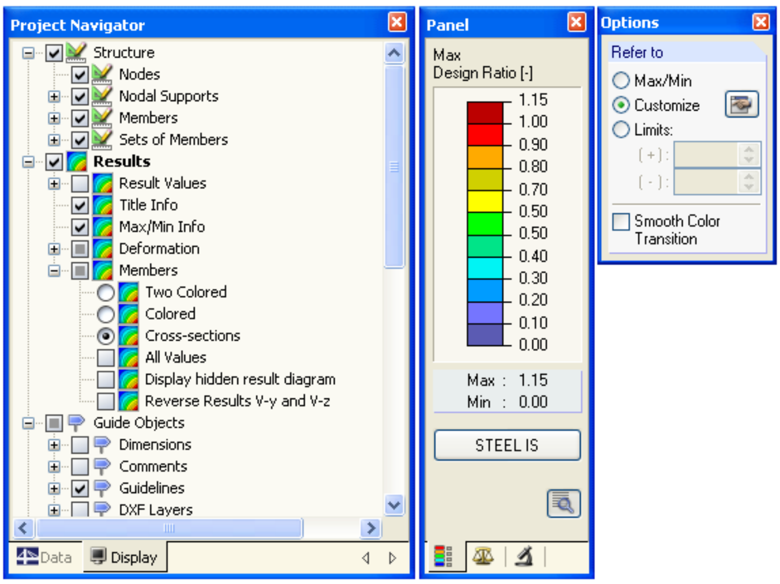


Figure 5.4: Design ratios for Option *Cross-sections* in the *Display* Navigator

As for the member internal forces, you can set the scale factor for the graphics of the design results in the *Factors* register. If you enter the factor *0* in the input field *Member Diagrams*, the design ratios will be shown with an increased line thickness.

This graphics can be incorporated to the global printout report (see chapter 0).

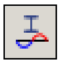
You can return to STEEL IS any time by clicking on the [STEEL IS] button in the panel.

5.2 Result Diagrams

In order to view the result diagram of a specific member, the graph of results can be used. Select the desired member or set of members in the results table of STEEL IS and then activate the diagram by the button as seen to the left. This button is located below the upper tables of results.

The result diagrams are also available in the RSTAB window via the main menu

Results → Member Results



or by using the corresponding button in the toolbar.

A new window is opened in which the result diagrams of the selected member or set of members are shown.

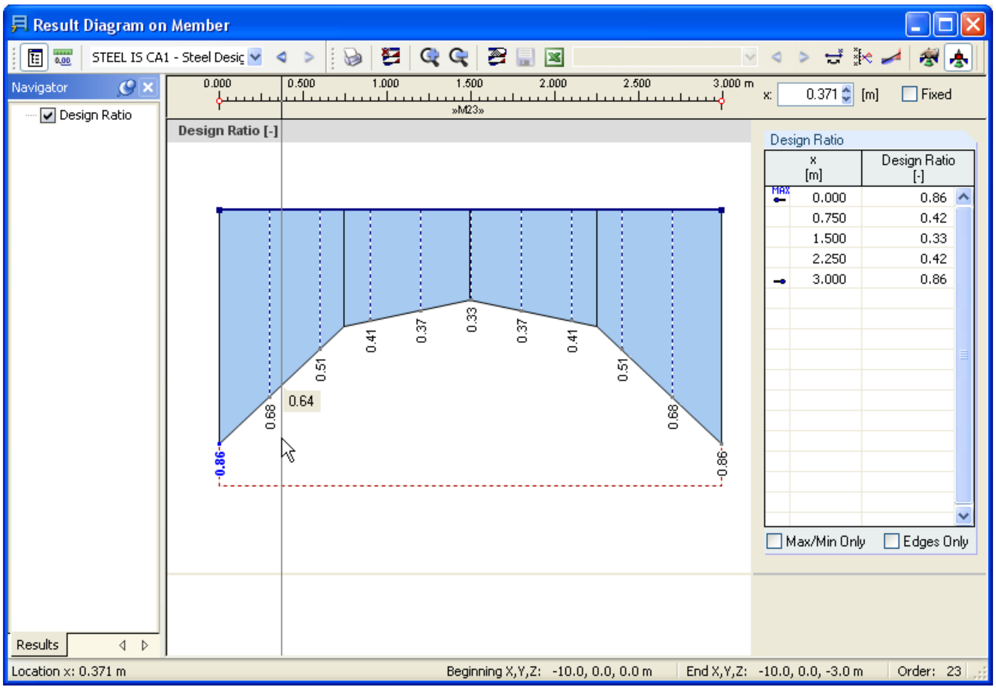
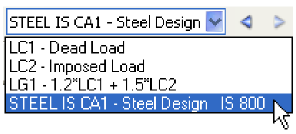


Figure 5.5: *Result Diagram on Member* Dialog



A particular design case can be selected from the list in the toolbar.

A detailed description of the *Result Diagram on Member* dialog can be found in Chapter 9.8.4 of the RSTAB manual.

5.3 Filtering Results

The structure of the STEEL IS masks makes it already possible to select the results according to certain criteria. Additionally, you can use the filter functions as described in the RSTAB manual to graphically evaluate the STEEL IS results.

Firstly, you can use already defined partial views (cf RSTAB manual, chapter 9.8.6) that group certain objects in a favorable way.

Secondly, you can set the stress ratios as a criterion to filter the results in the RSTAB work window. For this, the so-called control panel is to be displayed. If it is not visible, you can switch it on via the main menu

View → Control panel

or by clicking on the corresponding button in the *Results* toolbar.

This panel is described in chapter 4.4.6 of the RSTAB manual. The settings to filter the results are defined in the *Color Spectrum* register. As this register is not available in case of the two colored stress display, it can be switched on by selecting one of the display options *Colored* or *Cross-sections* in the *Display* navigator.

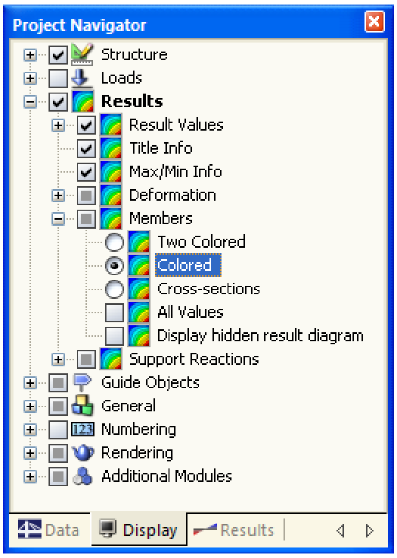


Figure 5.6: *Display* Navigator: Results → Members → **Colored**

For a colored view of the results, you can set in the panel that e.g. only stress ratios greater than 0.3 are to be displayed. Furthermore, you can adjust the color spectrum in a way that one single color range exactly covers the stress ratio 0.10 (see Figure 5.7).

By the option *Display hidden result diagram* (*Display* navigator, entry Results → Members) you can also display stress diagrams that do not satisfy the given condition. Those diagrams will then be drawn as dashed lines.

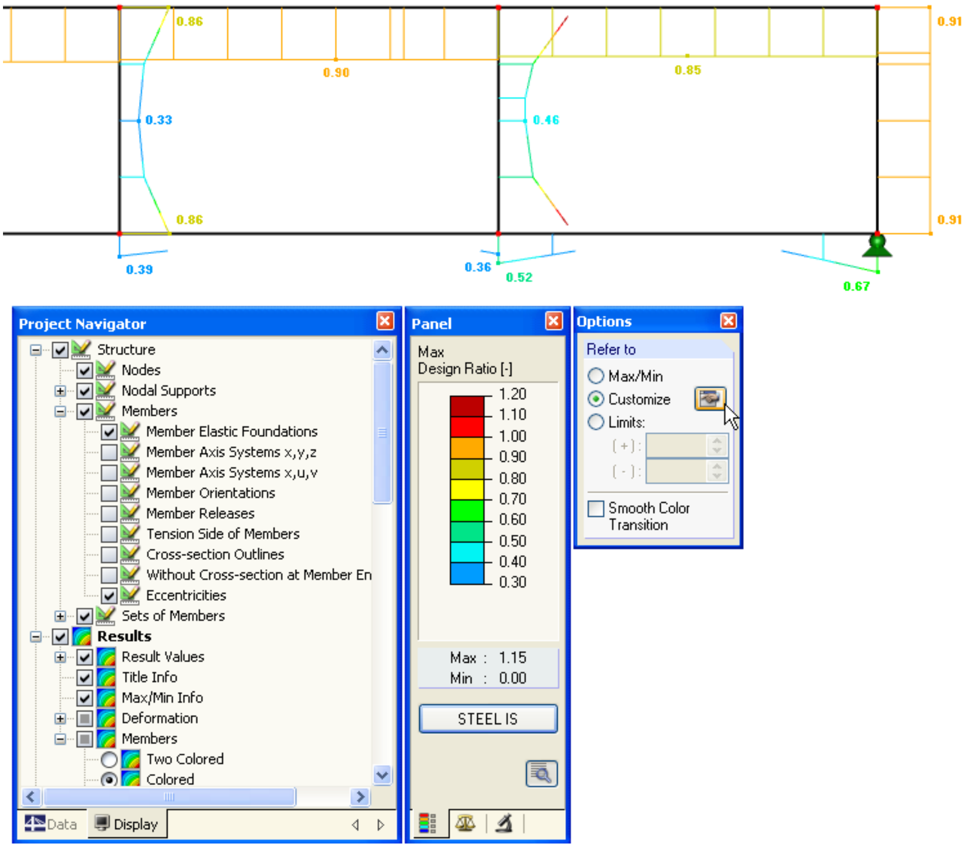


Figure 5.7: Filtering Stress Ratios with Customized Color Spectrum

Filtering Members

In the *Filter* register of the control panel, you can enter the numbers of the members whose result diagrams are to be shown in the graphics. This function is described in chapter 9.8.6 of the RSTAB manual.

Contrary to the partial view function, the entire structure is displayed here. The following figure shows the stress ratios in the compressed flange of a footbridge. The other members of this structure are also shown in the model but they are without any stress ratios.

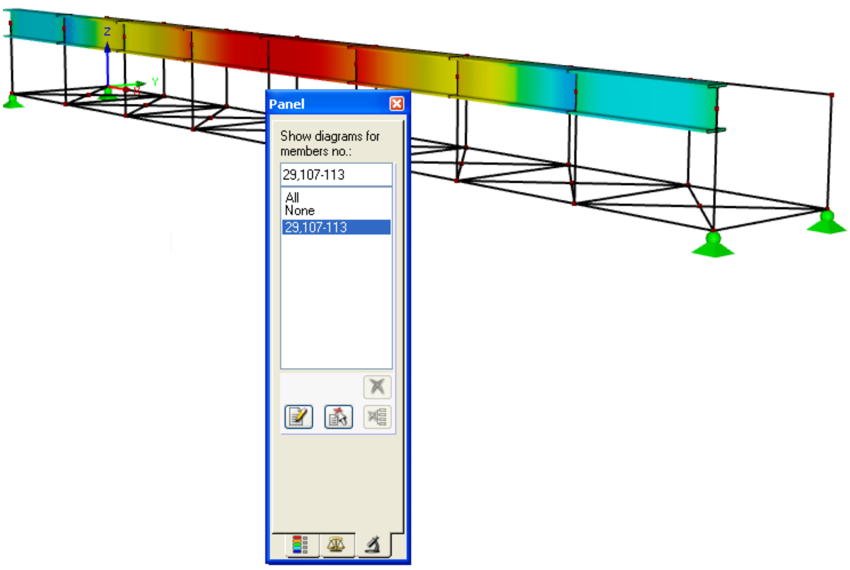


Figure 5.8: Filtering Members: Stress Ratios of Footbridge Flange

6. Printout

6.1 Printout Report

For the design results of STEEL IS, a printout report can be created to which you can add graphics and comments. In this printout report, it is also possible to select which results tables of STEEL IS are to be printed.

The printout report is described in detail in the RSTAB manual. In particular, Chapter 10.1.3.4 *Selection of Additional Modules Data* is important and deals with the selection of input and output data in all add-on modules.

You can create several printout reports for every design case. For very large structures, it is recommended to create several smaller reports instead of a single extensive one. If you create a specific report only for the data of the STEEL IS design case, the printout report will be processed fairly quickly.

6.2 Print STEEL IS Graphics

It is possible to print the stress ratios displayed on the RSTAB model. All graphics can be incorporated to the printout report or sent directly to the printer. Chapter 10.2 of the RSTAB manual describes in detail how to print graphic displays.

Results on RSTAB Model

Every image of the RSTAB work window can be included in the printout report. The current STEEL IS graphics is printed by using the main menu

File → Print...

or clicking on the corresponding button in the toolbar.

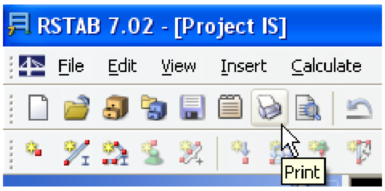


Figure 6.1: *Print* Button in Toolbar in Main Window

Result Diagram

You can also print the result diagrams of members by clicking on the [Print] button in the *Result Diagram on Member* dialog.

Figure 6.2: *Print* Button in Toolbar of *Result Diagram* Dialog

The following dialog opens.

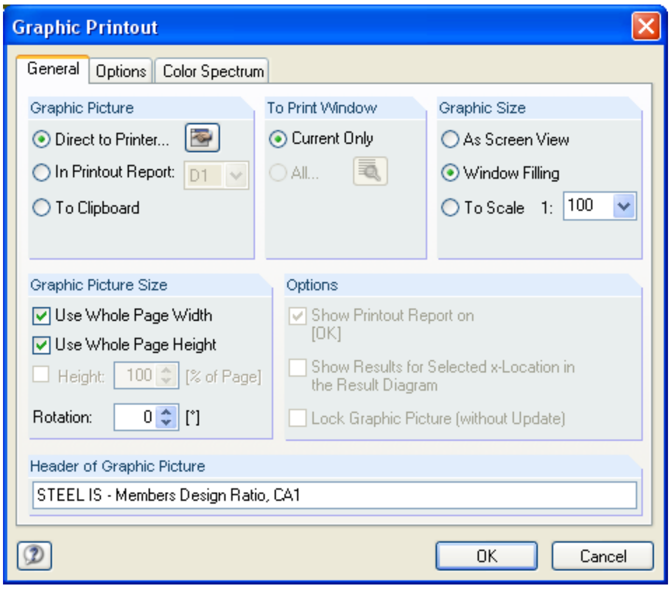


Figure 6.3: *Graphic Printout* Dialog, *General* Register

This dialog is described in detail in Chapter 10.2.1 of the RSTAB manual. The remaining two register tabs *Options* and *Color Spectrum* are also explained there.

In the printout report, any image of the STEEL IS results can be moved to a different location by the Drag&Drop function. It is also possible to adjust inserted images subsequently: right mouse click on the relevant entry in the report navigator, then select *Properties* in the context menu. The *Graphic Printout* dialog is displayed again where the possible changes can be set.

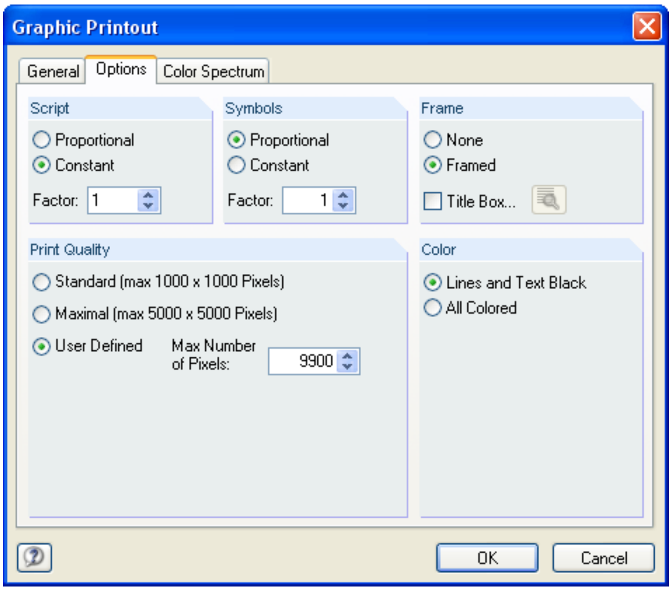


Figure 6.4: *Graphic Printout* Dialog, *Options* Register

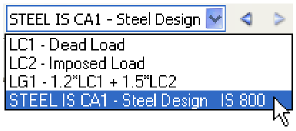
7. General Functions

This chapter describes the commonly used functions of the main menu as well as the export options of the design results.

7.1 STEEL IS Design Cases

The user has the option to group the members into separate design cases. In this way, it is possible e.g. to design separately certain structural parts or elements with specific parameters (limit stresses, partial safety factors, optimization etc.).

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



All design cases created in STEEL IS are contained in the list of load cases and load groups in the toolbar in the RSTAB work window.

Create New STEEL IS Case

A new design case can be created from the STEEL IS main menu

File → New Case…

The following dialog opens:

Figure 7.1: *New STEEL IS-Case* Dialog

In this dialog, you need to fill in the (not yet used) *Number* and *Description* of the new design case. After closing the dialog with [OK], the STEEL IS form 1.1 *General Data* is shown where you can define the new design data.

Rename STEEL IS Case

The description of a design case can be changed via the STEEL IS main menu

File → Rename Case…

The *Rename STEEL IS-Case* dialog is opened.

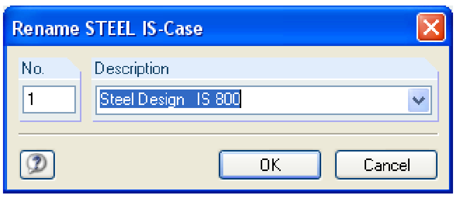


Figure 7.2: *Rename STEEL IS-Case* Dialog

Copy STEEL IS Case

The input data of the current design case can be copied via the STEEL IS main menu

File → Copy Case…

The *Copy STEEL IS-Case* dialog opens. Enter the number and description of the new design case into which the selected case is to be copied.

Figure 7.3: *Copy STEEL IS-Case* Dialog

Delete STEEL IS Case

Design cases can be deleted via the STEEL IS main menu

File → Delete Cases…

In the *Delete Cases* dialog, select a specific design case from the list of *Available Cases*. It will be deleted when clicking on [OK]

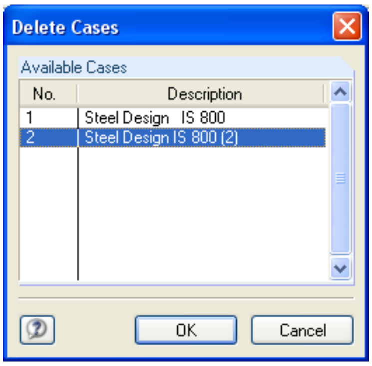
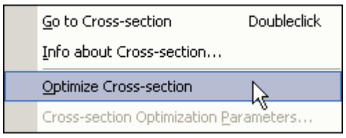


Figure 7.4: *Delete Cases* Dialog

7.2 Cross-Section Optimization

The module STEEL IS offers the possibility to optimize cross-sections. For this, select the cross-section that is to be optimized in column D resp. E of form 1.3 *Cross-sections* by ticking the appropriate box (cf Figure 2.6).



You can also start the optimization of a cross-section via the context menu in the results tables.

During the optimization, STEEL IS examines which cross-section within the same cross-sections series satisfies the design “optimally“, i.e. is the closest to the maximum allowable stress ratio which has been defined in the *Details* dialog (cf Figure 3.1). The required cross-section properties are calculated on the basis on the internal forces from RSTAB. Finally, the cross-section which satisfies the design with the highest possible stress ratio is chosen. For this reason, two cross-sections are shown graphically in form 1.3 on the right - the original cross-section from RSTAB and the optimized one (cf Figure 7.6).

When ticking the optimization box for parameterized cross-sections from the library, the following dialog appears for you to enter detailed data.

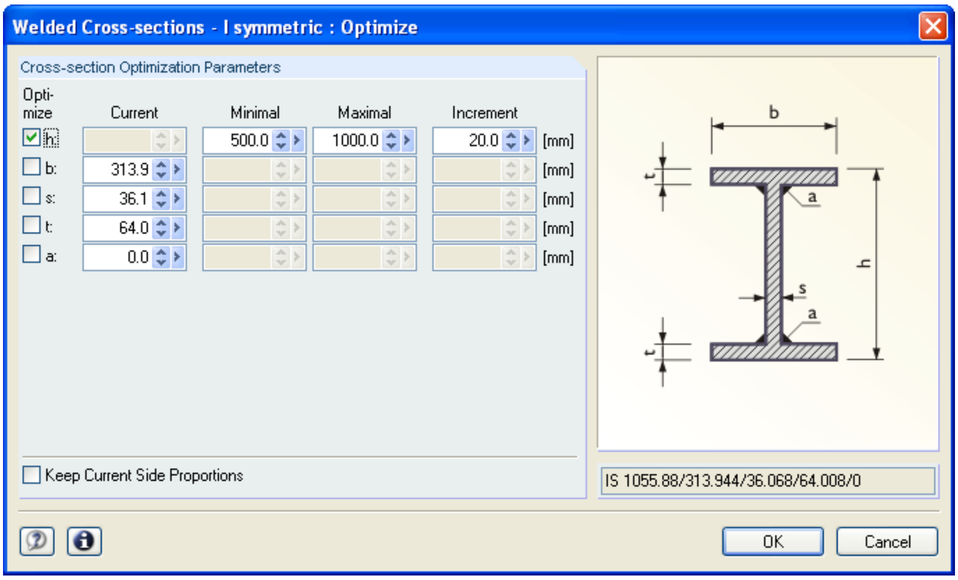


Figure 7.5: *Welded Cross-cections - I-Symmetric: Optimize* Dialog

At first, select the parameter(s) that you want to modify in column *Optimize*. Hence, the columns *Minimal* and *Maximal* become accessible where the upper and lower limits of the optimization parameter can be defined. The column *Increment* controls in which intervals the parameter dimensions vary during the optimization process.

If you want to *Keep Current Side Proportions*, tick the corresponding box in the lower part of this dialog. Additionally, it is necessary to tick all parameters for the optimization.

It is not possible to carry out the optimization for combined rolled cross-sections.

Please keep in mind that during the optimization the internal forces will not be recalculated automatically on the basis of the modified cross-sections. It depends on the user’s decision when and which cross-sections are to be adapted in RSTAB for a new analysis. The internal forces based on the optimized cross-sections may differ considerably due to the changed rigidities within the structural model. Thus, we recommend recalculating the internal forces after one optimization run and then optimizing the cross-sections once more.

It is not necessary to transfer the modified cross-sections to RSTAB manually. Open form 1.3 *Cross-sections* and select in the main menu

Edit → Export All Cross-Sections to RSTAB.

The option to export the modified cross-sections to RSTAB is also contained in the context menu of form 1.3.

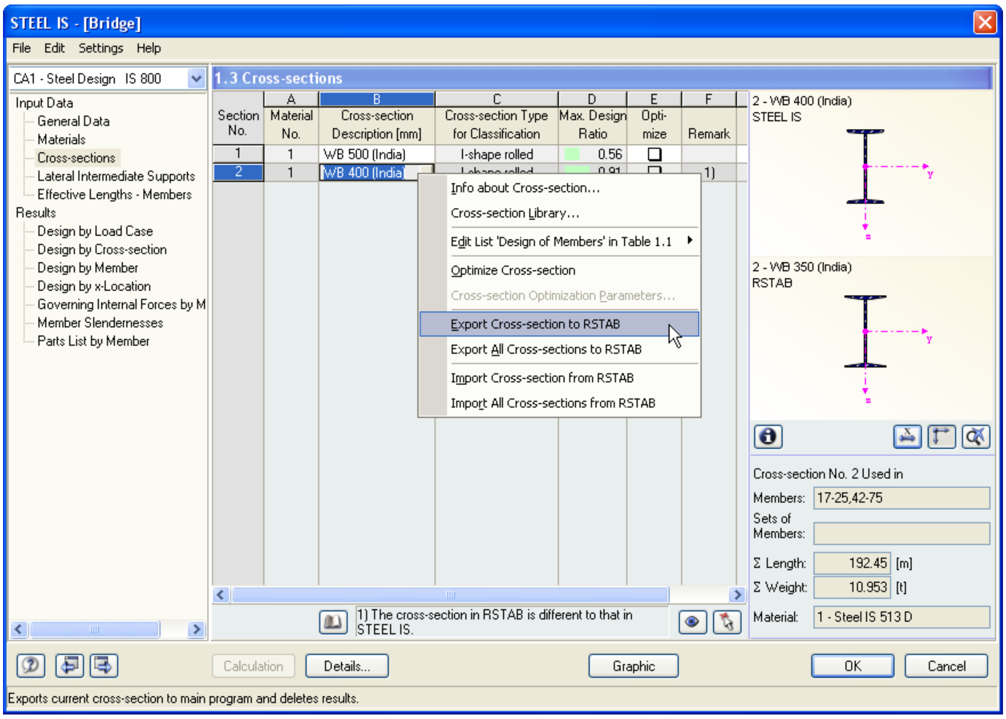


Figure 7.6: Context Menu in Form 1.3 *Cross-sections*

Before the cross-sections are transferred to RSTAB, a question appears because exporting also implies deleting the results. If you then start the [Calculation] in STEEL IS, the internal forces of RSTAB and the stresses of STEEL IS are calculated in one calculation run.

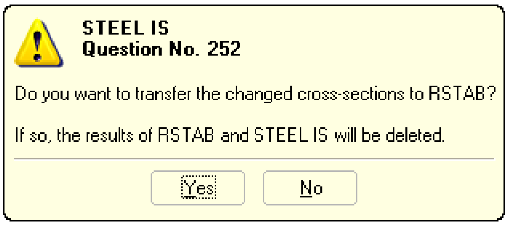


Figure 7.7: Question before Transferring Modified Cross-Sections to RSTAB

In a similar way, you can reload the original cross-sections from RSTAB to STEEL IS by using the appropriate functions in the main menu or context menu. Please note that this option is only available in form 1.3 *Cross-sections*.

If you want to optimize a tapered member, the cross-sections of the member start and of the member end are optimized. After that, the moments of inertia are linearly interpolated at the intermediate locations of the member. As those are considered by the fourth power, the stress design may be inaccurate if there are big differences in height of the start and end cross-sections. In such a case, we recommend dividing tapers into several members whose start and end cross-sections do not show such big differences.

7.3 Import / Export of Materials

If you change a material in form 1.2 of STEEL IS, you can export it to RSTAB like a cross-section or also reload the original material from RSTAB to the module. The materials that have been modified in the module are highlighted in blue color.

It is not necessary to transfer the modified materials to RSTAB manually. Instead, open form 1.2 *Materials* and choose in the main menu

Edit → Export all Materials to RSTAB.

The option to export modified materials to RSTAB is also included in the context menu of form 1.2.

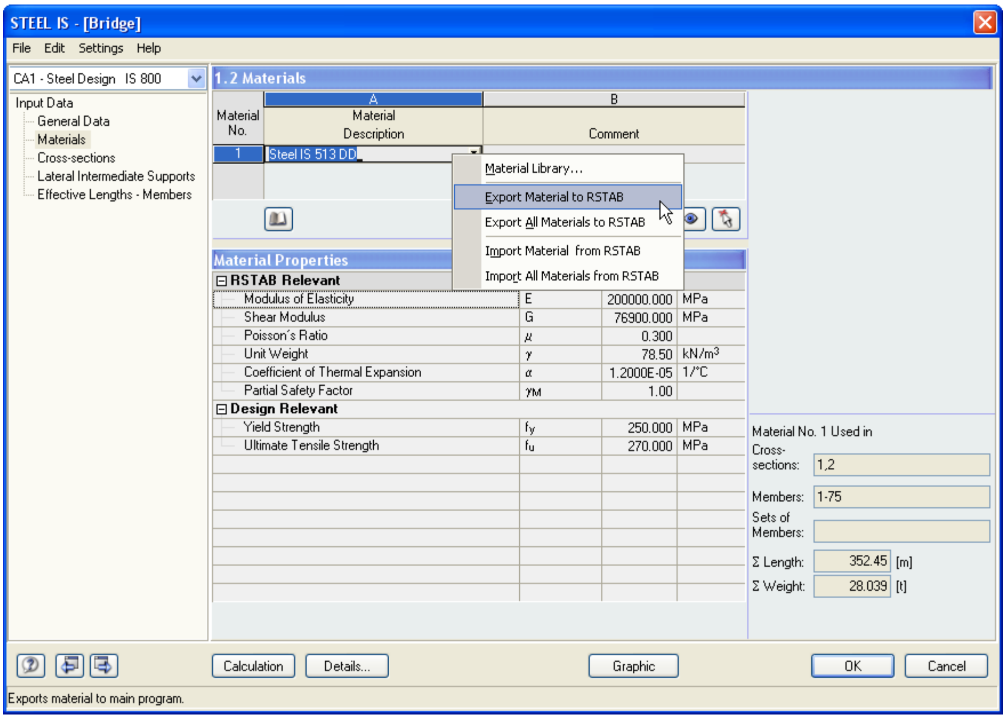


Figure 7.8: Context Menu in Form 1.2 *Materials*

Before the materials are transferred to RSTAB, a question is shown because exporting also implies deleting the results. If you then start the [Calculation] in STEEL IS, the internal forces of RSTAB and the stresses of STEEL IS are calculated in one calculation run.

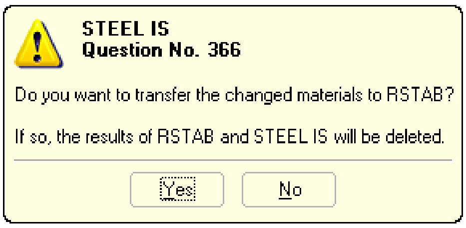


Figure 7.9: Question before Transferring Modified Materials to RSTAB

7.4 Units and Decimal Places

The units and decimal places are centrally managed for RSTAB and all its add-on modules. In STEEL IS, open the dialog to set the units via the main menu

Settings → Units and Decimal Places…

The familiar RSTAB dialog opens. The module STEEL IS is already set by default.

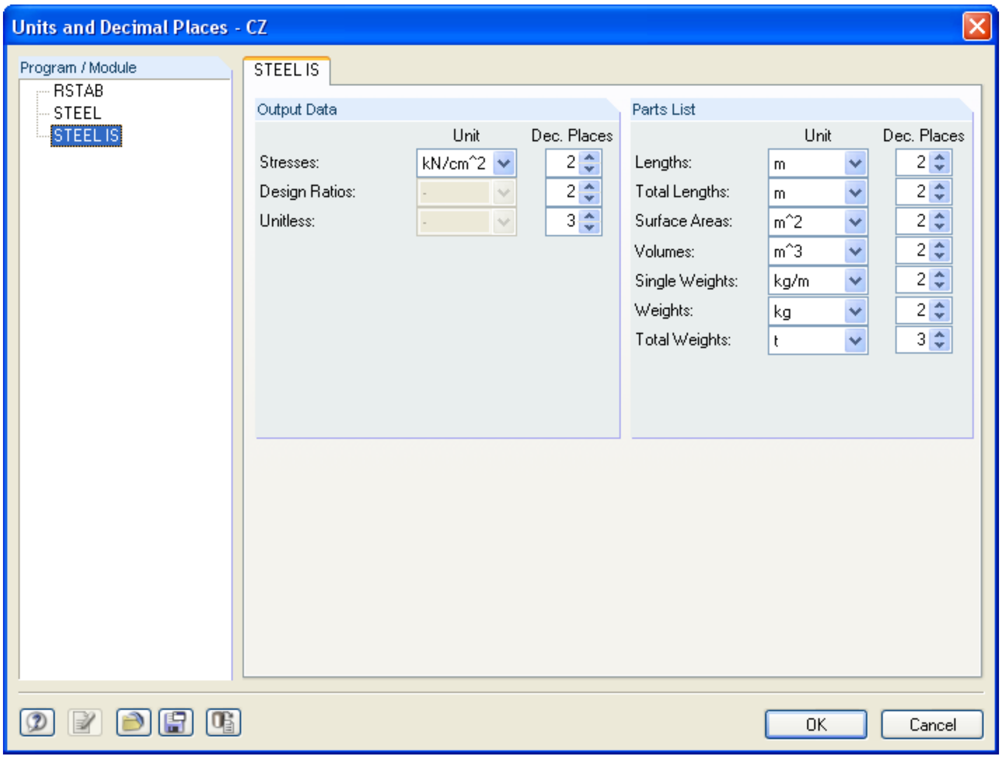


Figure 7.10: *Units and Decimal Places* Dialog

The settings can be stored as a user profile. They can also be applied later in different cases. This function is described in Chapter 11.6.2 of the RSTAB manual.

7.5 Export of Results

The results of the stress design can be transferred to other programs in different ways.

Clipboard

Select the relevant cells in the results tables of STEEL IS and copy them to the clipboard via [Ctrl]+[C]. The contents can then be inserted via [Ctrl]+[V] e.g. to some text processing program. The headers of the table columns are be exported.

Printout Report

The STEEL IS data can be sent to the printout report (cf Chapter 6.1) and then be exported via the main menu

File → Export to RTF File or BauText…

This function is described in Chapter 10.1.11 of the RSTAB manual.

Excel / OpenOffice

STEEL IS enables you to directly export data to MS Excel or OpenOffice.org Calc. Call up this function via the main menu

File → Export Tables…

The following dialog opens:

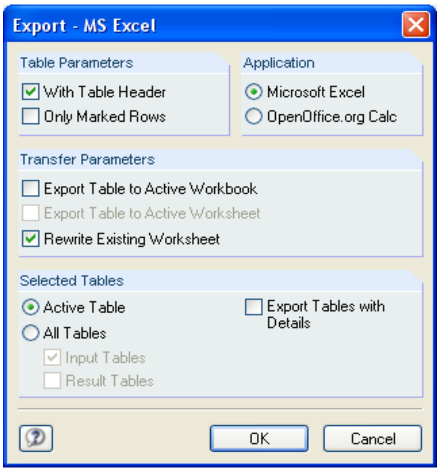


Figure 7.11: *Export - MS Excel* Dialog

As soon as you have chosen the relevant options, you can start the export by [OK]. Excel or OpenOffice do not need to run in the background, they will be started automatically before the export.

1	Section	Member	Location	Load	Design		
2	No.	Nr.	x [m]	Case	Ratio		Design according to Formula
3	1	WB 500 (India)					
4		4	0,000	LG1	0,18	≤ 1	101) Cross-section Check - Tension acc. to 6.2 and 6.3.1
5		12	0,000	LG1	0,21	≤ 1	102) Cross-section Check - Compression acc. to 7.1.2
6		1	0,000	LG1	0,15	≤ 1	121) Cross-section Check - Shear Force in z-Axis acc. to 8.4.1
7		1	0,000	LG1	0,15	≤ 1	126) Cross-section Check - Shear Buckling acc. to 8.4.2.1
8		1	0,000	LG1	0,19	≤ 1	181) Cross-section Check - Bending about y Axis, Shear and Axial Force acc.
9		8	5,000	LG1	0,44	≤ 1	331) Stability Analysis - Lateral Torsional Buckling and Axial Tension acc. to 9
10		11	0,000	LG1	0,30	≤ 1	341) Stability Analysis - Buckling about y-Axis and Bending about y and z-Axis
11		14	0,000	LG1	0,60	≤ 1	351) Stability Analysis - Buckling about z-Axis and Bending about y and z-Axis
12							
13	2	WB 400 (India)					
14		21	3,000	LC1	0,00	≤ 1	100) No or Very Small Internal Forces
15		62	0,000	LG1	0,07	≤ 1	101) Cross-section Check - Tension acc. to 6.2 and 6.3.1
16		61	0,000	LG1	0,09	≤ 1	102) Cross-section Check - Compression acc. to 7.1.2
17		18	0,000	LC1	0,38	≤ 1	111) Cross-section Check - Bending about y-Axis acc. to 8.2.1.2 - Class 1 or
18		18	3,000	LG1	0,46	≤ 1	121) Cross-section Check - Shear Force in z-Axis acc. to 8.4.1
19		18	3,000	LG1	0,46	≤ 1	126) Cross-section Check - Shear Buckling acc. to 8.4.2.1
20		18	3,000	LG1	1,08	> 1	181) Cross-section Check - Bending about y Axis, Shear and Axial Force acc.
21		61	0,000	LG1	0,09	≤ 1	301) Stability Analysis - Flexural Buckling about y-Axis acc. to 7.1.2 and 7.1.2
22		61	0,000	LG1	0,17	≤ 1	311) Stability Analysis - Flexural Buckling about z-Axis acc. to 7.1.2 and 7.1.2
23		18	0,000	LC1	0,40	≤ 1	321) Lateral Torsional Buckling according to 8.2.2 and 8.2.2.1
24		18	0,000	LG1	1,08	> 1	331) Stability Analysis - Lateral Torsional Buckling and Axial Tension acc. to 9
25		17	0,000	LG1	0,35	≤ 1	341) Stability Analysis - Buckling about y-Axis and Bending about y and z-Axis
26		17	0,000	LG1	0,86	≤ 1	351) Stability Analysis - Buckling about z-Axis and Bending about y and z-Axis

Figure 7.12: Results in Excel

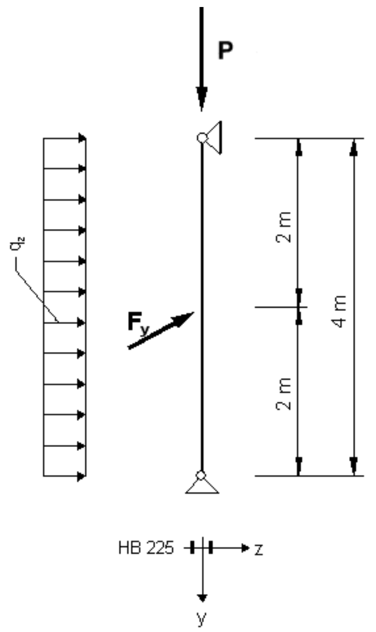
8. Example

Column with Biaxial Bending

In the following example, the decisive stability design of buckling and lateral buckling is carried out by analyzing the relevant interaction conditions.

Design Values

Structure and Loads



Design values of static loads:

P	= 300 kN
q_z	= 5.0 kN/m
F_y	= 7.5 kN

Cross-section:	HB 225
Material:	IS 513 DD

Figure 8.1: Structure and Design Loads (γ-fold)

Internal Forces according to Linear Static Analysis

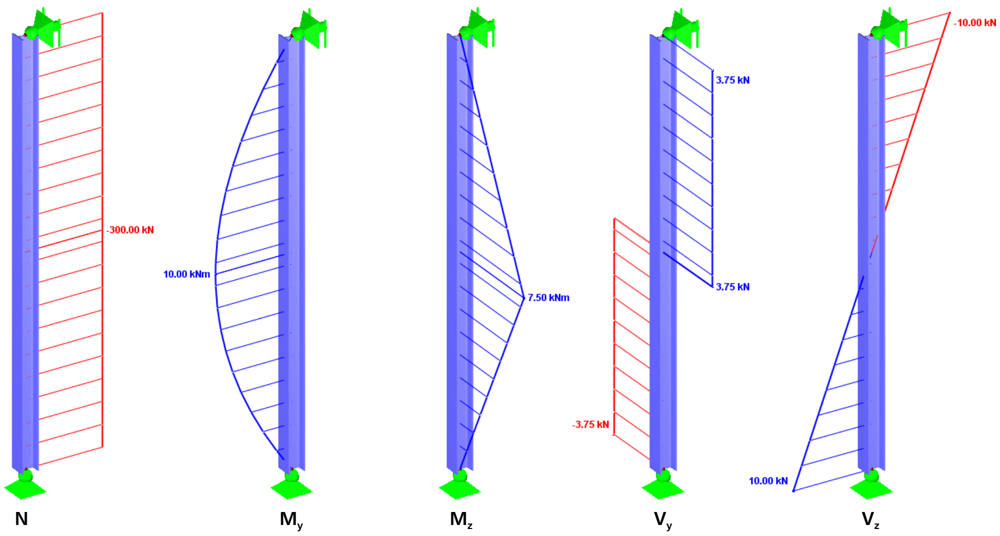


Figure 8.2: Internal Forces

Design Location (Decisive Location x)

The design proceeds according to locations x, i.e. on defined locations x of an equivalent member. The following internal forces act in the decisive location for x = 2.00:

N = –300.00 kN M_y = 10.00 kNm M_z = 7.50 kNm V_y = 3.75 kN V_z = 0.00 kN

Cross-Section Properties HB 225

Cross-Section Properties	Symbol	Value	Units
Cross-section area	A	54.90	cm^2
Moment of inertia about major axis	I_y	5300.00	cm^4
Moment of inertia about minor axis	I_z	1350.00	cm^4
Radius of inertia about major axis	r_y	9.83	cm
Radius of inertia about minor axis	r_z	4.96	cm
Cross-section weight	wt	43.1	kg/m
Torsion constant	I_T	13.28	cm^4
Warping constant	I_w	201318.00	cm^6
Elastic section modulus about major axis	Z_{ey}	469.00	cm^3
Elastic section modulus about minor axis	Z_{ez}	120.00	cm^3
Plastic section modulus about major axis	Z_{py}	514.78	cm^3
Plastic section modulus about minor axis	Z_{pz}	233.22	cm^3
Buckling class about major axis	BC_y	b	
Buckling class about minor axis	BC_z	c	

Material Properties IS 513 DD

Material Properties	Symbol	Value	Units
Modulus of elasticity	E	200000	N/mm^2
Modulus of rigidity	G	76900	N/mm^2
Characteristic yield stress	f_y	250	N/mm^2
Characteristic ultimate tensile stress	f_u	270	N/mm^2
Partial safety factor	γ_{m0}	1.1	

Classification of Cross-section

$$\varepsilon = \sqrt{250 / f_y} = \sqrt{250 / 250} = 1.0$$

Classification of the Flange

b = 112.5 mm

t_f = 9.1 mm

$$\lambda_{f,1} = 9.4\,\varepsilon = 9.4 \cdot 1.0 = 9.4$$

$$\lambda_{f,2} = 10.5\,\varepsilon = 10.5 \cdot 1.0 = 10.5$$

$$\lambda_{f,3} = 15.7\,\varepsilon = 15.7 \cdot 1.0 = 15.7$$

$$\frac{b}{t_f} = \frac{112.5}{9.1} = 12.363 \le 15.7 = \lambda_{f,3}$$

Class of the flange is 3.

Classification of the Web

$\sigma_{w,A} = -72.3\,N/mm^2$

$\sigma_{w,B} = -37.0\,N/mm^2$

d = 206.8 cm

t_w = 6.8 cm

$$r_1 = \frac{P}{d \cdot t_w} / (f_y / \gamma_{mo}) = \frac{300 \cdot 10^3}{206.8 \cdot 6.8} / (250/1.1) = 0.939$$

$$r_2 = \frac{P}{A} / (f_y / \gamma_{mo}) = \frac{300 \cdot 10^3}{54.9 \cdot 10^2} / (250/1.1) = 0.240$$

$$\lambda_{w,1} = \frac{84\,\varepsilon}{1+r_1} = \frac{84 \cdot 1.0}{1+0.939} = 43.329$$

$$\lambda_{w,2} = \frac{105\,\varepsilon}{1+1.5\,r_1} = \frac{105 \cdot 1.0}{1+1.5 \cdot 0.939} = 43.604$$

$$\lambda_{w,3} = \frac{126\,\varepsilon}{1+2\,r_2} = \frac{126 \cdot 1.0}{1+2 \cdot 0.240} = 85.085$$

$$\frac{d}{t_{fw}} = \frac{206.8}{6.8} = 30.412 \le 43.329 = \lambda_{w,1}$$

Class of the web is 1.

Class of the cross-section is **3**.

Classification in STEEL IS

Cross-section Classification - Class 3					
Flange					
- Width	b	11.25	cm		Table 2
- Thickness	t_f	0.91	cm		Table 2
- Yield Stress Ratio	ε_f	1.000			Table 2
- Maximal Ratio for Class 1	$\lambda_{f,1}$	9.400			Table 2
- Maximal Ratio for Class 2	$\lambda_{f,2}$	10.500			Table 2
- Maximal Ratio for Class 3	$\lambda_{f,3}$	15.700			Table 2
- Ratio	b/t_f	12.363		$\le \lambda_{f,3}$	
- Class of Flange		3			Table 2
Web					
- Stress at the End of Web	$\sigma_{w,A}$	-72.3	N/mm^2		
- Stress at the End of Web	$\sigma_{w,B}$	-37.0	N/mm^2		
- Depth	d	20.68	cm		Table 2
- Thickness	t_w	0.68	cm		Table 2
- Area of Cross-Section	A	54.90	cm^2		
- Design Yield Stress	$f_{yd,w}$	227.3	N/mm^2		
- Axial Compressive Force	P	-300.000	kN		
- Yield Stress Ratio	ε_w	1.000			Table 2
- Stress Ratio	r_1	0.939			Table 2
- Stress Ratio	r_2	0.240			Table 2
- Maximal Ratio for Class 1	$\lambda_{w,1}$	43.329			Table 2
- Maximal Ratio for Class 2	$\lambda_{w,2}$	43.604			Table 2
- Maximal Ratio for Class 3	$\lambda_{w,3}$	85.085			Table 2
- Ratio	d/t_w	30.412		$\le \lambda_{w,1}$	
- Class of Web		1			Table 2
Class of Cross-Section		**3**			Table 2

Buckling about y-Axis (Major Axis)

$KL_y = 4.0$ m

$\frac{h}{b_f} = \frac{225}{225} = 1.0 < 1.2$ $t_f = 9.1$ mm < 100 mm ⇒ Buckling Class b

$$f_{ccy} = \frac{\pi^2 E}{(KL_y / r_y)^2} = \frac{\pi^2 \cdot 200000}{(4 \cdot 10^3 / 98.25)^2} = 1191.0 \, N / mm^2$$

$$\lambda_y = \sqrt{\frac{f_y}{f_{ccy}}} = \sqrt{\frac{250}{1191.0}} = 0.458$$

$\alpha_y = 0.340$ (Table 7)

$\phi_y = 0.5 [1 + \alpha_y(\lambda_y - 0.2) + \lambda_y^2] = 0.5 [1 + 0.340 (0.458 - 0.2) + 0.458^2] = 0.649$

$$\chi_y = \frac{1}{[\phi_y + (\phi_y^2 - \lambda_y^2)^{0.5}]} = \frac{1}{[0.649 + (0.649^2 - 0.458^2)^{0.5}]} = 0.902$$

$$f_{cdy} = \chi_y \, f_y / \gamma_{m0} = 0.902 \cdot 250 / 1.1 = 205.1 \; N / mm^2$$

$$P_{dy} = A_e \, f_{cdy} = 54.9 \cdot 10^2 \cdot 205.1 = 1125.82 \, kN$$

$$n_y = P / P_{dy} = 300.0 / 1125.82 = 0.27$$

Design in STEEL IS

Effective Length	KL_y	4.000	m		
Buckling Class	BC_y	b			Table 10
Euler Buckling Stress	f_{ccy}	1191.0	N/mm²		7.1.2.1
Slenderness Ratio	λ_y	0.458			7.1.2.1
Imperfection Factor	α_y	0.340			Table 7
Auxiliary Factor	ϕ_y	0.649			7.1.2.1
Stress Reduction Factor	χ_y	0.902			7.1.2.1
Design Compressive Stress	f_{cdy}	205.1	N/mm²		7.1.2.1
Design Compressive Strength	P_{dy}	1125.82	kN		7.1.2
Axial Design Ratio	n_y	0.27			9.3.2.2

Buckling about z-Axis (Minor Axis)

$KL_z = 4.0$ m

Buckling Class c

$$f_{ccz} = \frac{\pi^2 E}{(KL_z / r_z)^2} = \frac{\pi^2 \cdot 200000}{(4 \cdot 10^3 / 49.6)^2} = 303.4 \, N / mm^2$$

$$\lambda_z = \sqrt{\frac{f_y}{f_{ccz}}} = \sqrt{\frac{250}{303.4}} = 0.908$$

$\alpha_z = 0.490$ (Table 7)

$\phi_z = 0.5 [1 + \alpha_z(\lambda_z - 0.2) + \lambda_z^2] = 0.5 [1 + 0.490 (0.908 - 0.2) + 0.908^2] = 1.085$

$$\chi_z = \frac{1}{[\phi_z + (\phi_z^2 - \lambda_z^2)^{0.5}]} = \frac{1}{[1.085 + (1.085^2 - 0.908^2)^{0.5}]} = 0.595$$

$$f_{cdz} = \chi_z \, f_y / \gamma_{m0} = 0.595 \cdot 250 / 1.1 = 135.2 \; N / mm^2$$

$$P_{dz} = A_e\, f_{cdz} = 54.9 \cdot 10^2 \cdot 135.2 = 742.5\,\text{kN}$$

$$n_z = P / P_{dz} = 300.0 / 742.5 = 0.40$$

Design in STEEL IS

Effective Length	KL_z	4.000	m		
Buckling Class	BC_z	c			Table 10
Euler Buckling Stress	f_{ccz}	303.4	N/mm²		7.1.2.1
Slenderness Ratio	λ_z	0.908			7.1.2.1
Imperfection Factor	α_z	0.490			Table 7
Auxiliary Factor	ϕ_z	1.085			7.1.2.1
Stress Reduction Factor	χ_z	0.595			7.1.2.1
Design Compressive Stress	f_{cdz}	135.2	N/mm²		7.1.2.1
Design Compressive Strength	P_{dz}	742.5	kN		7.1.2
Axial Design Ratio	n_z	0.40			9.3.2.2

Lateral Torsional Buckling

The check for lateral torsional buckling is based on the calculation of the critical moment M_{cr}. A forked support without any warping restraint is assumed for both member ends. The point of load application is set at the upper flange (this point can be modified in the detailed settings, see Chapter 3.1).

$K = 1.0$ (Free rotation about weak axis at both ends)

$K_w = 1.0$

$L_{LT} = 4.0$ m

$y_g = \frac{h}{2} = \frac{225}{2} = 112.5\,\text{mm}$ (Load applied at the upper flange)

$y_j = 0.0$ (Symmetrical cross-section)

$C_1 \,\acute{=}\, 1.132$

$C_2 \,\acute{=}\, 0.458$

$$M_{cr} = C_1 \frac{\pi^2 E I_z}{(L_{LT})^2} \left\{ \left[\left(\frac{K}{K_w}\right)^2 \frac{I_w}{I_y} + \frac{G I_t (L_{LT})^2}{\pi^2 E I_z} + (C_2\, y_g)^2 \right]^{0.5} - C_2\, y_g \right\} \acute{=}$$

$$= 1.132 \frac{\pi^2 \cdot 200000 \cdot 1350 \cdot 10^4}{(4000)^2}$$

$$\left\{ \left[\left(\frac{1.0}{1.0}\right)^2 \frac{201318 \cdot 10^6}{1350 \cdot 10^4} + \frac{76900 \cdot 13.28 \cdot 10^4 (4000)^2}{\pi^2\, 200000 \cdot 1350 \cdot 10^4} + \right.\right.$$

$$\left.\left. + \frac{76900 \cdot 13.28 \cdot 10^4 (4000)^2}{\pi^2\, 200000 \cdot 1350 \cdot 10^4} + (0.458 \cdot 112.5)^2 \right]^{0.5} - 0.458 \cdot 112.5 \right\} =$$

$$= 192.9\,\text{kNm}$$

$\beta_{by} = Z_{ey} / Z_{py} \,\acute{=}\, 469.0 / 514.78 = 0.911$ (Cross-section class 3)

$$\lambda_{LT} = \sqrt{\frac{\beta_{by} Z_{py} f_y}{M_{cr}}} = \sqrt{\frac{0.911 \cdot 514.78 \cdot 10^3 \cdot 250}{192.9 \cdot 10^6}} = 0.779$$

$\alpha_{LT} \,\acute{=}\, 0.21$ (Rolled cross-section)

$\phi_{LT} = 0.5\,[1 + \alpha_{LT}(\lambda_{LT} - 0.2) + \lambda_{LT}^2] = 0.5\,[1 + 0.210\,(0.779 - 0.2) + 0.779^2] = 0.865$

$$\chi_{LT} = \frac{1}{[\phi_{LT} + (\phi_{LT}^2 - \lambda_{LT}^2)^{0.5}]} = \frac{1}{[0.865 + (0.865^2 - 0.779^2)^{0.5}]} = 0.807$$

$$f_{bd} = \chi_{LT} f_y / \gamma_{m0} = 0.807 \cdot 250 / 1.1 = 183.5 \text{ N/mm}^2$$

$$M_{dy} = \beta_{by} Z_{py} f_{bd} \acute{=} 0.911 \cdot 514.78 \cdot 183.5 = 86.044 \text{ kNm}$$

Design in STEEL IS

Effective Length Factor	K	1.000			Annex E
Warping Restrained Factor	K_w	1.000			Annex E
Length of the Segment	L	4.000	m		
Effective Length	L_{LT}	4.000	m		
Load Application Position	y_g	11.25	cm		Annex E
Cross-section Factor	y_j	0.00	cm		Annex E
Moment Factor	C_1	1.132			Eigenvalue
Moment Factor	C_2	0.458			Eigenvalue
Elastic Critical Moment	M_{cr}	192.989	kNm		Annex E
Slenderness Ratio	λ_{LT}	0.779			8.2.2
Imperfection Parameter	α_{LT}	0.210			8.2.2
Auxiliary Factor	(LT	0.865			8.2.2
Stress Reduction Factor	χLT	0.807			8.2.2
Design Compressive Stress	fbd	183.5	N/mm2		8.2.2
Bending Factor	β_{by}	0.911			8.2.2
Bending Strength	M_{dy}	86.044	kNm		8.2.2

Bending

$$\beta_{by} = Z_{ey} / Z_{py} \acute{=} 469.0 / 514.78 = 0.911$$

$$\beta_{bz} = Z_{ez} / Z_{pz} \acute{=} 120.0 / 233.22 = 0.515$$

$$M_{dy} = \beta_{by} Z_{py} f_y / \gamma_{m0} = 0.911 \cdot 514.78 \cdot 10^3 \cdot 250 / 1.1 = 106.591 \text{kNm}$$

$$M_{dz} = \beta_{bz} Z_{pz} f_y / \gamma_{m0} = 0.515 \cdot 233.22 \cdot 10^3 \cdot 250 / 1.1 = 27.273 \text{kNm}$$

Design in STEEL IS

Bending Factor	β_{by}	0.911			8.2.2
Bending Factor	β_{bz}	0.515			8.2.2
Bending Strength	M_{dy}	106.6	kNm		8.2.1.2
Bending Strength	M_{dz}	27.3	kNm		8.2.1.2

Equivalent Uniform Moment Factors and Interaction Factors

Bending about y-Axis (Major Axis)

The equivalent uniform moment factor C_{my} is set according to table 18. Uniform loading (column 3) and maximum bending moment in the field (diagram 3) are assumed.

$$\alpha_{hy} = M_{hy} / M_{sy} = 0.0 / 10.0 = 0.0$$

$$C_{my} = 0.95 + 0.05\,\alpha_{hy} = 0.95 + 0.05 \cdot 0.0 = 0.95$$

$$K_y = 1 + (\lambda_y - 0.2)\,n_y = 1 + (0.458 - 0.2) \cdot 0.27 = 1.069$$

Bending about z-Axis (Minor Axis)

The equivalent uniform moment factor C_{mz} is set according to table 18. Concentrated load (column 4) and maximum bending moment in the field (diagram 3) are assumed.

$$\alpha_{hz} = M_{hz} / M_{sz} = 0.0 / 7.5 = 0.0$$

$$C_{mz} = 0.90 + 0.10\,\alpha_{hz} = 0.95 + 0.10 \cdot 0.0 = 0.90$$

$$K_z = 1 + (\lambda_z - 0.2)\,n_z = 1 + (0.908 - 0.2) \cdot 0.40 = 1.286$$

Lateral Torsional Buckling

$$C_{myLT} = C_{my} = 0.95$$

$$K_{LT} = 1 - \frac{0.1 \lambda_{LT}\,n_y}{(C_{mLT} - 0.25)} = 1 - \frac{0.1 \cdot 0.779 \cdot 0.27}{(0.95 - 0.25)} = 0.97$$

Design in STEEL IS

Bending Moment Diagram	Diagr M_y	3) Max in Field			Table 18
Bending Moment Diagram	Diagr M_z	3) Max in Field			Table 18
Hogging Moments Ratio	ψ_y	0.0			Table 18
Hogging Moments Ratio	ψ_z	0.0			Table 18
Hogging Moment	M_{hy}	0.0	kNm		Table 18
Hogging Moment	M_{hz}	0.0	kNm		Table 18
Sagging Moment	M_{sy}	10.0	kNm		Table 18
Sagging Moment	M_{sz}	7.5	kNm		Table 18
Ratio M_{hy} / M_{sy}	α_{hy}	0.0			Table 18
Ratio M_{hz} / M_{sz}	α_{hz}	0.0			Table 18
Load Type	Load y	Unif. Loading			Table 18
Load Type	Load z	Conc. Load			Table 18
Moment Factor	C_{my}	0.950			Table 18
Moment Factor	C_{mz}	0.900			Table 18
Moment Factor	C_{myLT}	0.950			Table 18
Interaction Factor	K_y	1.069			9.3.2.2
Interaction Factor	K_z	1.286			9.3.2.2
Interaction Factor	K_{LT}	0.970			9.3.2.2

Buckling about y-axis (Major Axis) and Biaxial Bending

$\eta_{ny} = P / P_{dy} = 300.0 / 1125.82 = 0.27$

$\eta_{my} = M_y / M_{dy} = 10.0 / 106.591 = 0.09$

$\eta_{mz} = M_z / M_{dz} = 7.5 / 27.273 = 0.28$

$$\eta = \frac{P}{P_{dy}} + K_y \frac{C_{my} M_y}{M_{dy}} + 0.6 K_z \frac{C_{mz} M_z}{M_{dz}}$$

$$= \eta_{ny} + K_y C_{my} \eta_{my} + 0.6 K_z C_{mz} \eta_{mz}$$

$$= 0.27 + 1.068 \cdot 0.95 \cdot 0.09 + 0.6 \cdot 1.286 \cdot 0.90 \cdot 0.28 = 0.55 \leq 1.0 \Rightarrow \text{Proven}$$

Design in STEEL IS

Compressive Design Ratio	η_{ny}	0.27			9.3.2.2
Bending Design Ratio	η_{my}	0.09			9.3.2.2
Bending Design Ratio	η_{mz}	0.28			9.3.2.2
Design Ratio	η	0.55		≤ 1	9.3.2.2

Buckling about z-axis (Minor Axis) and Biaxial Bending

$\eta_{nz} = P / P_{dz} = 300.0 / 742.5 = 0.40$

$\eta_{my} = M_y / M_{dy} = 10.0 / 86.044 = 0.12$

$\eta_{mz} = M_z / M_{dz} = 7.5 / 27.273 = 0.28$

$$\eta = \frac{P}{P_{dy}} + K_{LT} \frac{M_y}{M_{dy}} + K_z \frac{C_{mz} M_z}{M_{dz}}$$

$$= \eta_{ny} + K_{LT} \eta_{my} + K_z C_{mz} \eta_{mz}$$

$$= 0.40 + 0.97 \cdot 0.12 + 1.286 \cdot 0.90 \cdot 0.28 = 0.84 \leq 1.0 \Rightarrow \text{Provem}$$

Design in STEEL IS

Compressive Design Ratio	η_{nz}	0.40			9.3.2.2
Bending Design Ratio	η_{my}	0.12			9.3.2.2
Bending Design Ratio	η_{mz}	0.28			9.3.2.2
Design Ratio	η	0.84		≤ 1	9.3.2.2

A Literature

[1] IS 800 : 2007 General Construction in Steel – Code of Practice, 2007

[2] Comprehensive Design of Steel Structures, B.C Punmia, Ashok K. Jain, Arun K. Jain, Laxmi Publications Ltd 2008.

[3] The Behaviour and Design of Steel Structures to IS , N.S. Trahair, M.A. Bradford, D.A. Nethercot, L. Gardner, Taylor & Francis Ltd 2007

[4] Rules for Member Stability in EN 1993-1-1, ECCS Technical Committee 8 - Stability

[5] Die neuen Stabilitätsnachweise im Stahlbau nach Eurocode 3, Naumes J., Strohmann I., Ungermann D., Sedlacek G., Stahlbau 77 (2008) Heft 10, Ernst & Sohn Verlag

B Index