Engineering Software
Dlubal

Version
March 2010

Add-on Module

STEEL AISC

Allowable Stress Design (ASD),
Load and Resistance Factor Design (LRFD),
Serviceability Limit State Design
according to ANSI/AISC 360-05

Program Description

Am Zellweg 2 D-93464 Tiefenbach

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

Contents

1. Introduction

1.1 Additional Module STEEL AISC

The U.S. *Specification for Structural Steel Buildings* (ANSI/AISC 360-05) determines rules for the design, analysis and construction of steel buildings in the United States of America. With the add-on module STEEL AISC from the company DLUBAL ENGINEERING SOFTWARE all users obtain a highly efficient and universal tool to design steel structures according to this standard.

All typical designs of load capacity, stability and deformation are carried out in the module STEEL AISC. Different actions are taken into account during the load capacity design. The allocation of designed cross-sections into three types (compact, noncompact and slender) makes an important part of the design according to the Specification mentioned above. The purpose of this classification is to determine the range in which the local buckling in cross-section parts limits the load capacity so that the rotational capacity of cross-sections can be verified. Further, STEEL AISC automatically calculates the limiting width-to-thickness 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 in the direction of y-axis and/or z-axis. Lateral supports can be added for a realistic representation of the structural model. All comparative slendernesses and critical stresses are automatically determined by STEEL AISC 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 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 has become important for the static calculations of modern civil engineering as more and more slender cross-sections are being used. In STEEL AISC, load cases and groups and combinations of load cases can be arranged individually to cover the various design situations.

Like other modules, STEEL AISC 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 profiles 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 AISC.

Your DLUBAL ENGINEERING SOFTWARE company.

1.2 STEEL AISC Team

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

Program Coordinators

Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem
Ing. Ph.D. Peter Chromiak

Programmers

Ing. Zdeněk Kosáček
Ing. Ph.D. Peter Chromiak
Dipl.-Ing. Georg Dlubal
Dr.-Ing. Jaroslav Lain
Ing. Martin Budáč
Mgr. Petr Oulehle
Ing. Roman Svoboda
David Schweiner
Ing. Tomáš Jelínek
Ing. Zbyněk Zámečník
DiS. Jiří Šmerák

Library of Cross-Sections and Materials

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

Design of Program, Dialog Boxes and Icons

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

Testing and Technical Support

Ing. Martin Vasek
Ing. Ph.D. Peter Chromiak
Ing. Ph.D. Jaromír Křížek
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
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. Ph.D. Peter Chromiak
Dipl.-Ing. (FH) Robert Vogl
Mgr. Petra Pokorná
Ing. Petr Míchal
Ing. Ladislav Kábrt
Dipl.-Ü. Gundel Pietzcker
Ing. Dmitry Bystrov
Mgr. Michaela Kryšková

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 add-on module STEEL AISC.

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

The index at the end of this manual enables you to quickly look up specific terms.

1.4 Starting STEEL AISC

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

Main Menu

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

Add-on Modules → Design - Steel → STEEL AISC.

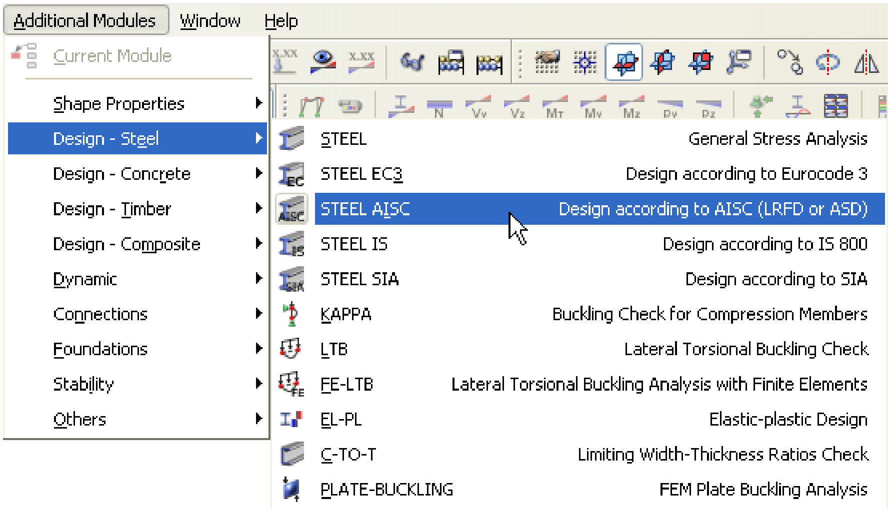


Figure 1.1: Main Menu: *Additional Modules → Design - Steel → STEEL AISC*

Navigator

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

Add-on Modules → STEEL AISC.

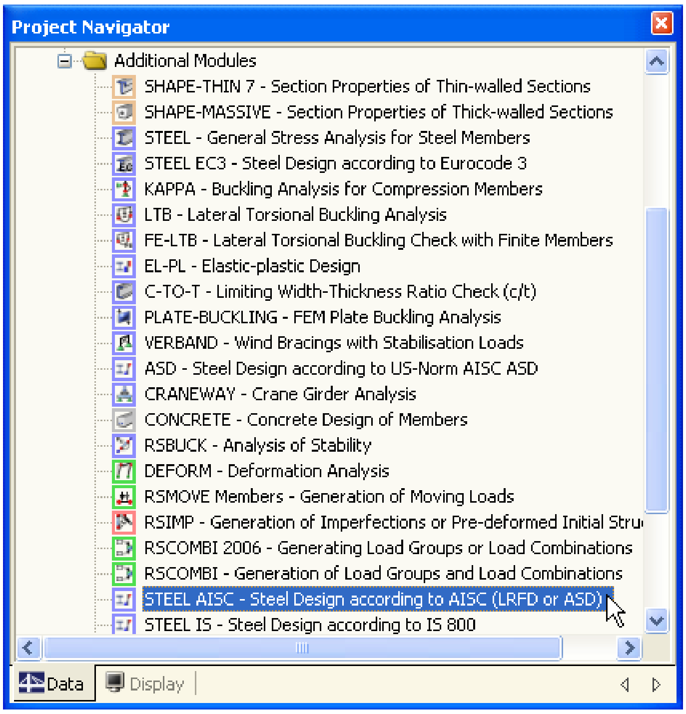
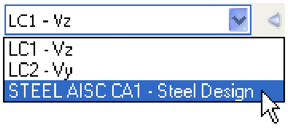


Figure 1.2: Data Navigator: *Additional Modules → STEEL AISC*

Panel



If results of STEEL AISC are already available in the RSTAB position, you can 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 by using the [Results on/off] button.

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

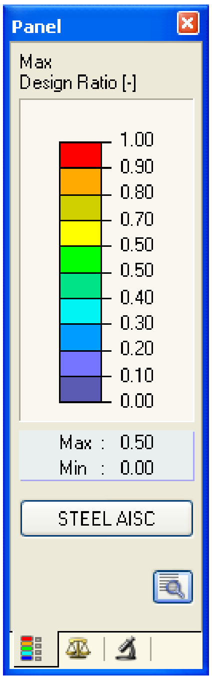


Figure 1.3: [STEEL AISC] Button in Panel

2. Input Data

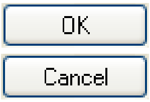
The data of the design cases is entered in different tables of this module. Furthermore, the graphic input using the function [Pick] is available for members and sets of members.

After the initialization of the STEEL AISC module, a new window is displayed. In its left part, a navigator is shown that enables you to access all existing tables. The roll-out list of all possibly entered design cases is located above this navigator (see chapter 7.1, page 46).

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



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

2.1 General Data

In the table 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 tabs.

2.1.1 Ultimate Limit State

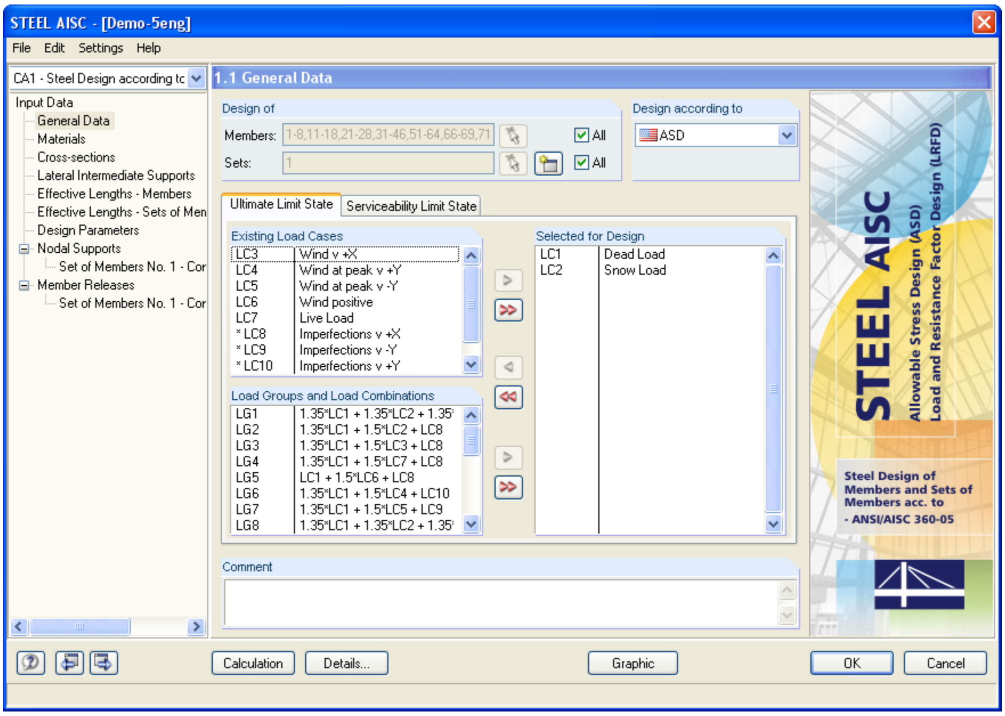


Figure 2.1: Table 1.1 *General Data*, *Ultimate Limit State* tab

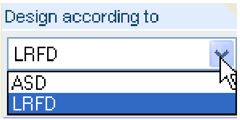
Design of

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 be created in STEEL AISC via the [New Set of Members…] button. The familiar RSTAB dialog box 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 tables 2.3 *Design by Set of Members* and 4.2 *Parts List by Set of Members* are displayed additionally.

Design according to



The list box controls whether the analysis is carried out according to the provisions of the Allowable Strength Design (ASD) or the Load and Resistance Factor Design (LRFD).

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 2.1 at load cases 8 to 10, 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:

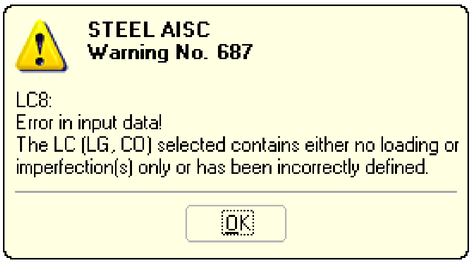


Figure 2.2: Warning when Selecting Invalid LC, LG or CO

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

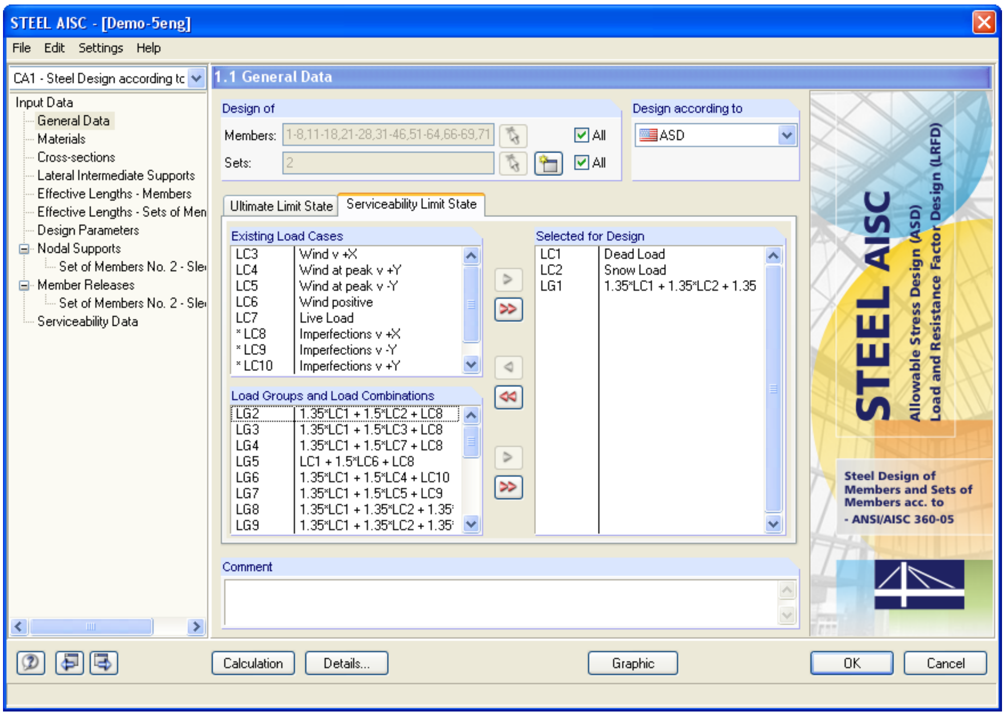


Figure 2.3: Table 1.1 *General Data*, *Serviceability Limit State* tab

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).

2.2 Materials

This table 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 table.

The material properties that are necessary to calculate the internal forces in RSTAB are described in detail in the RSTAB manual, Chapter 5.2. The 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, page 51).

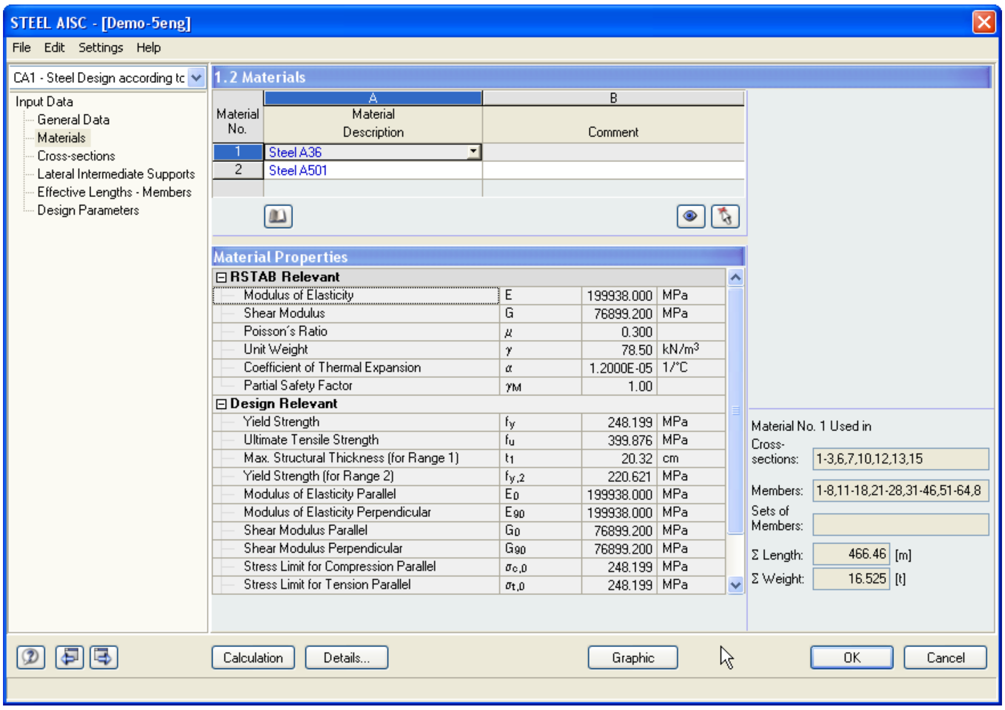
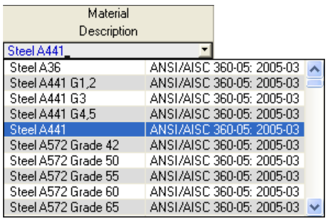


Figure 2.4: Table 1.2 *Materials*

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 in the material library, STEEL AISC 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 characteristics are updated in the table 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 amount of materials is stored in the library. Open the library via menu

Edit → **Material Library...**

or by clicking on the button visible on the left.

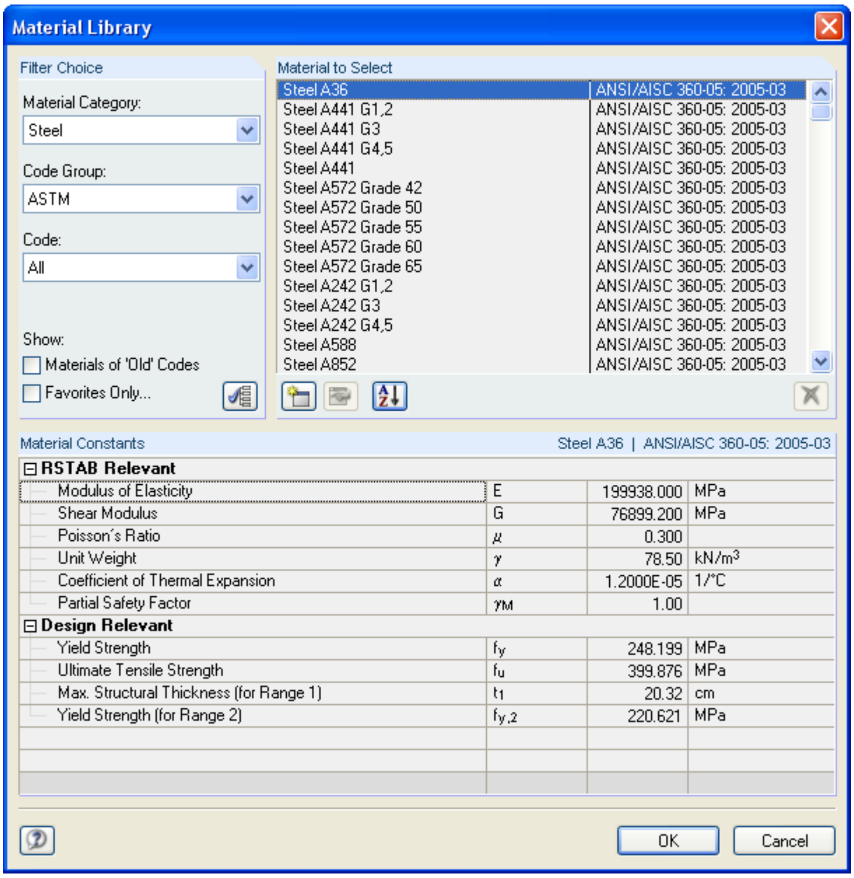


Figure 2.5: *Material Library* Dialog Box

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

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 thoroughly covered by the ANSI/AISC 360-05 standard. For this reason, it is not possible to significantly modify the material properties in STEEL AISC.

2.3 Cross-Sections

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

Coordinate System

The sectional coordinate system **yz** of STEEL AISC corresponds to the one of RSTAB (see image in Figure 2.6). The **y**-axis is the major principal axis of the cross-section, the **z**-axis the minor axis. This coordinate system is used for both the input data and the results.

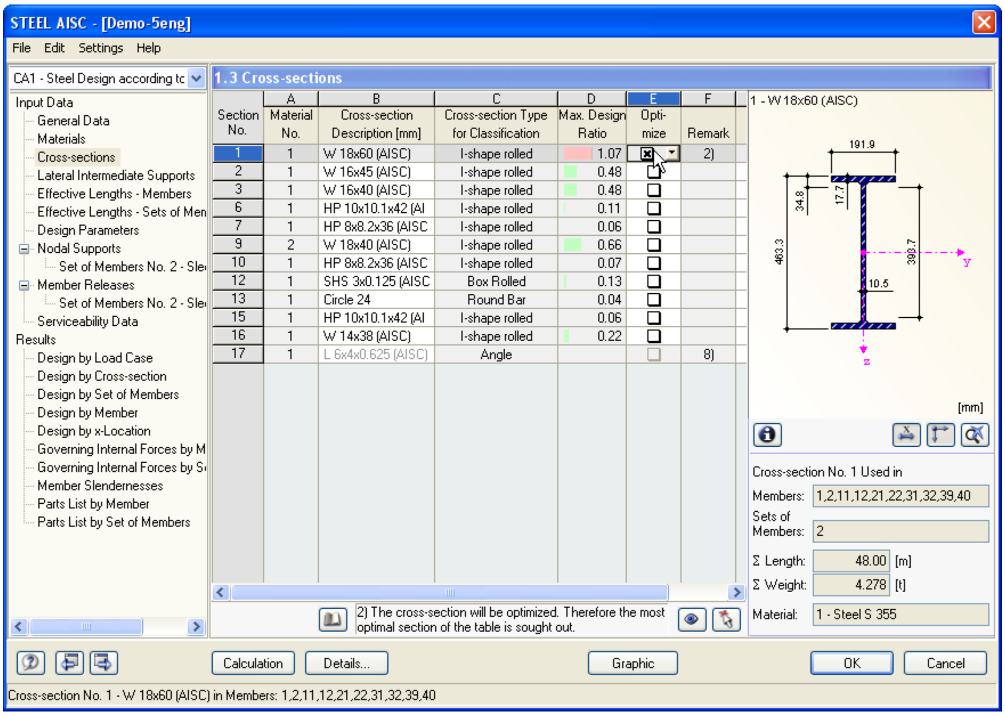
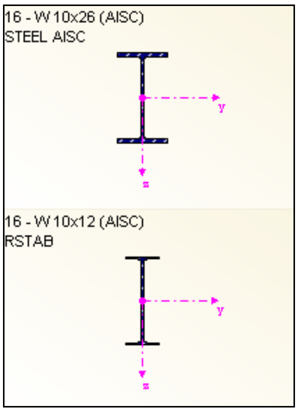


Figure 2.6: Table 1.3 *Cross-Sections*

Cross-Section Description

When you open this table, 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 AISC and RSTAB, both cross-sections are shown in the graphic window next to the table. The internal forces from RSTAB are then used for the stress design of the cross-section that is set in STEEL AISC.

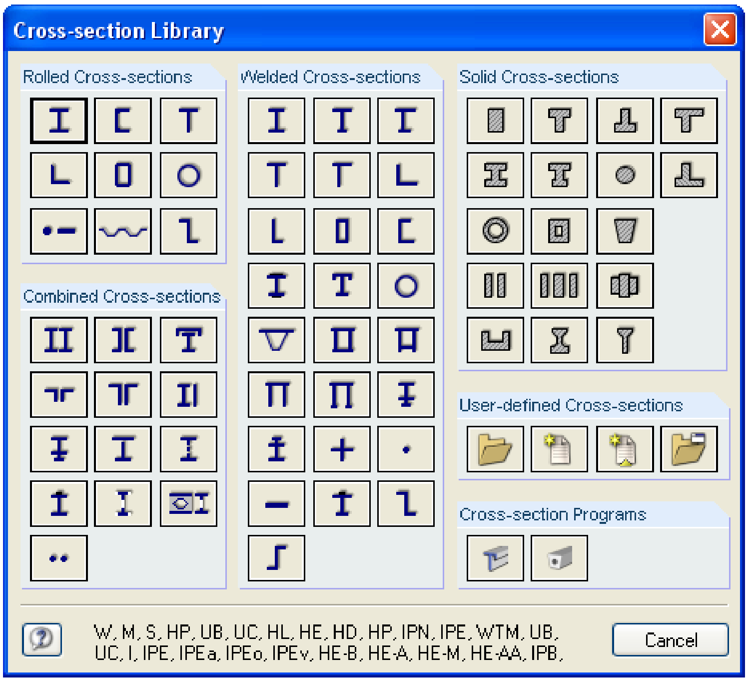


Figure 2.7: Cross-Section Library

HSS cross-sections can only be designed if the thicknesses of web and flanges are the same. If they are different, footnote *1004) Non-permissible cross-section type of 'Rectangular HSS'* is shown.

Tapered Member

In case of a tapered member 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 AISC 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 AISC cannot interpolate the intermediate values. An error message appears before the calculation:

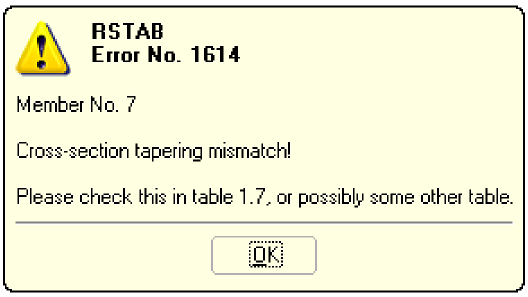


Figure 2.8: Warning in Case of Incompatible Cross-Sections

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

Info about Cross-Section

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

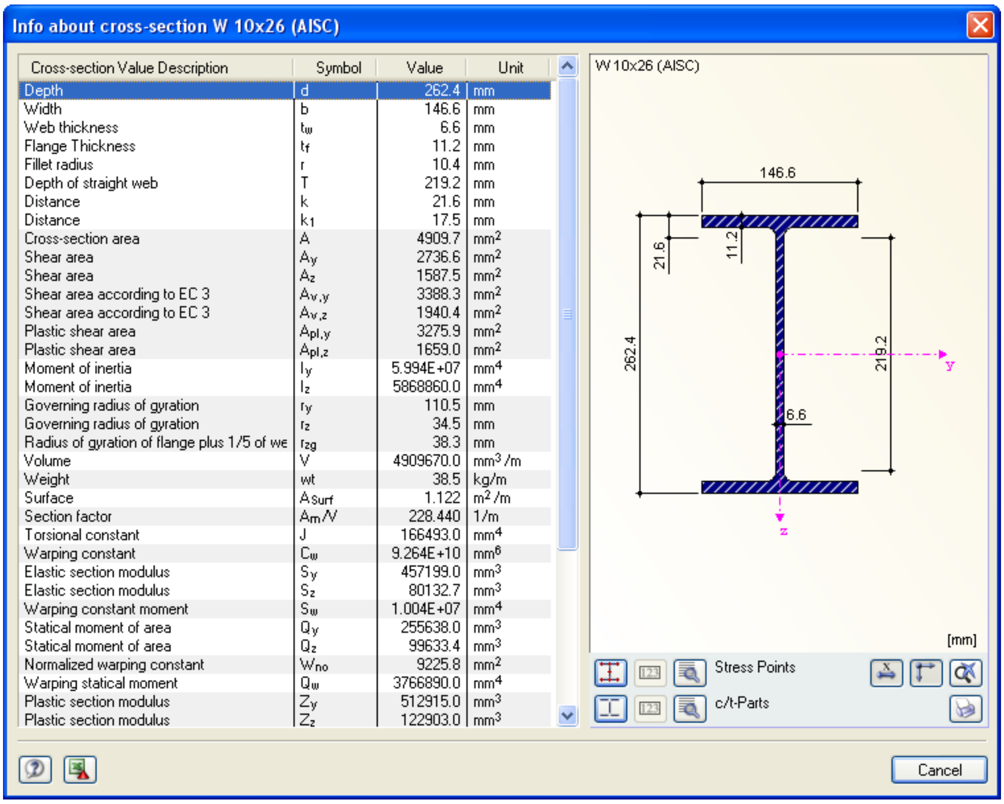


Figure 2.9: *Info about Cross-Section* Dialog Box

The currently selected cross-section is displayed in the right part of the dialog box. The various 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

Cross-Section Type for Classification

In this column, the various cross-section types are listed which are applied for the design (e.g. I-shape rolled or welded, box, round bar etc.)

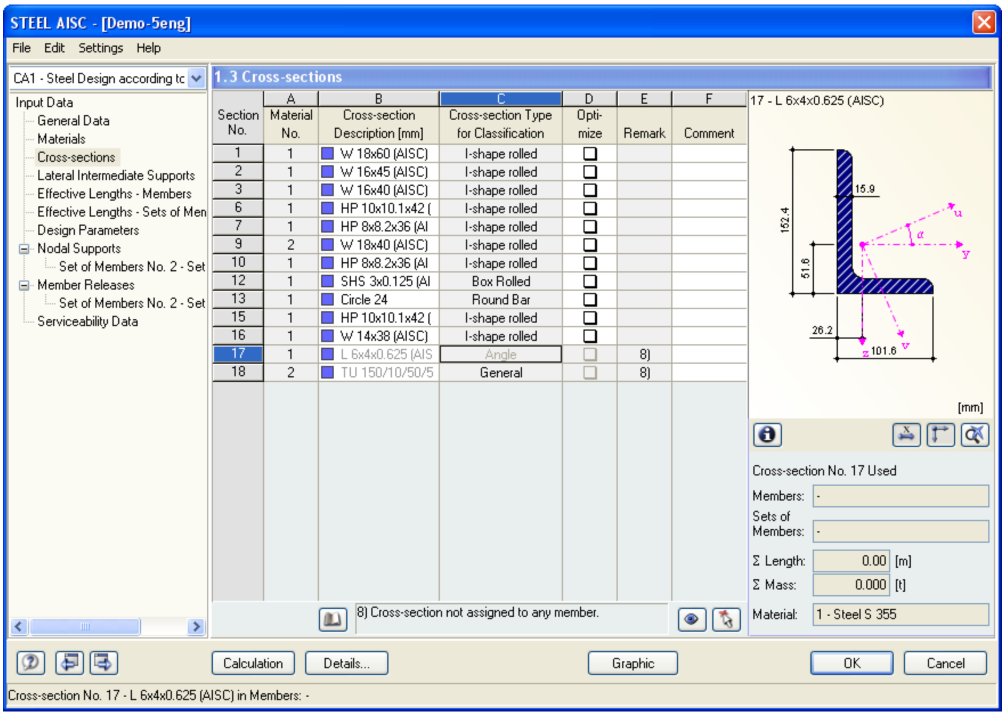


Figure 2.10: Cross-section Types for Classification

Max. Design Ratio

Due to this column, you can decide whether to carry out an optimization. This column is only displayed after STEEL AISC 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 *Details* dialog box, see chapter 3.1. Further information on the optimization of cross-sections can be found in chapter 7.2 on page 48.

Remark

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

If the message *Non-permissible Cross-Section No. XX* appears before the design, then this is 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 table, lateral intermediate supports on members can be defined. The program always assumes these supports as perpendicular to the minor axis z (see Figure 2.6) 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 forked supports for the design.

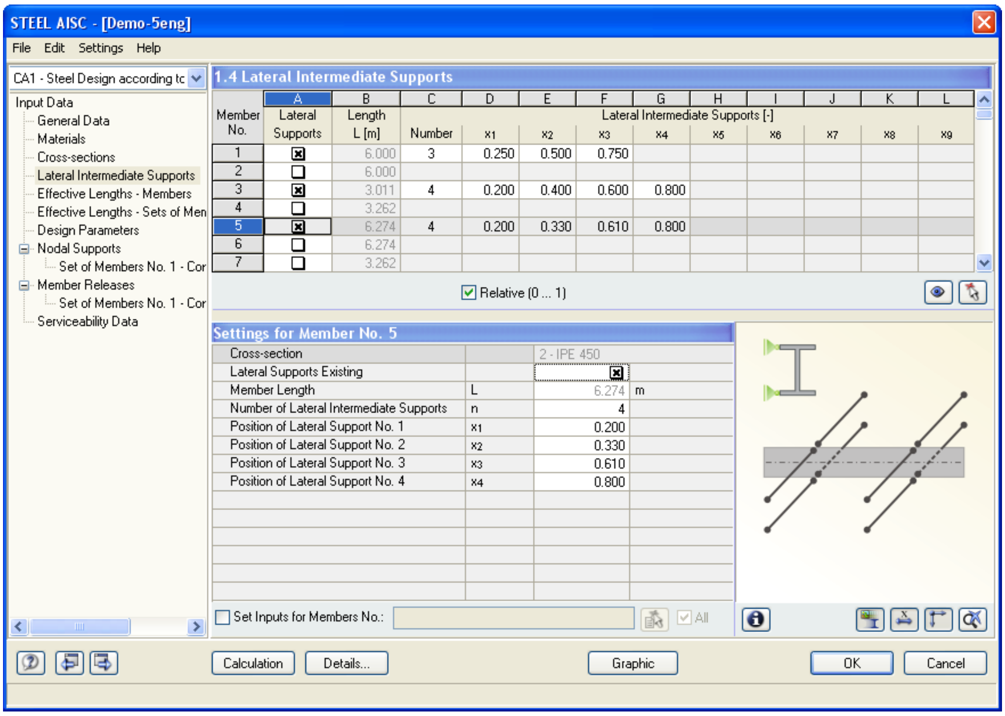


Figure 2.11: Table 1.4 *Lateral Intermediate Supports*

In the upper part of this table, up to nine lateral intermediate supports can be created per member. The lower part of the table 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.

You have to be very careful if the model contains cantilever beams. Intermediate supports divide the member into several parts for the design. Therefore, intermediate supports are to be avoided for cantilever beams because they would imply statically underdetermined pieces with fork-type supports on only one side each.

2.5 Effective Lengths - Members

The table 1.5 consists of two parts so that a good overview of the data is provided. In the upper table, the effective length factors K_y and K_z, the effective lengths $K_y L$ and $K_z L$, the *Torsional Buckling* effective length factors K_x and the effective lengths $K_x L$ for torsional buckling are summarized for every member. In the lower part of this table, 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.

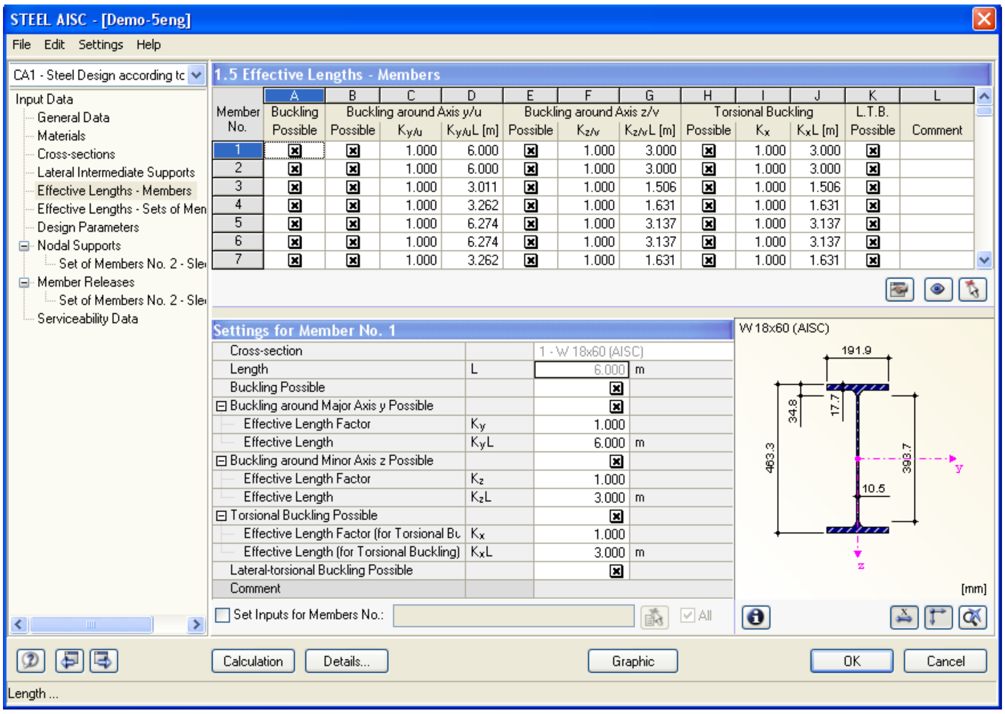


Figure 2.12: Table 1.5 *Effective Lengths - Members*

The effective lengths for the column buckling about the minor principal axis and the effective lengths for torsional buckling are automatically loaded from the previous table 1.4. If a member is divided into different lengths by lateral intermediate supports, then no values are displayed in the corresponding columns G and J of table 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 table 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 lower part of the *Settings for Member* table contains the following parameters:

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

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.

The effective length factors of the members can also be defined in a special dialog box which is called by the button [Select Effective Length Factor...] below the upper table.

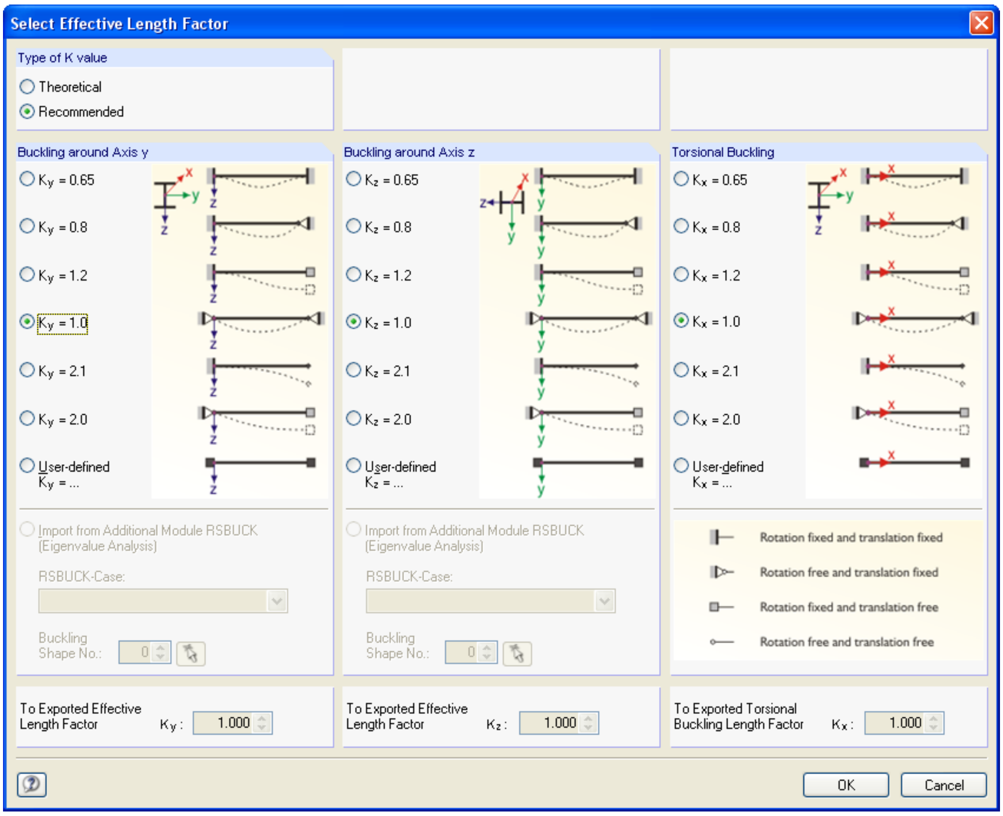


Figure 2.13: Dialog box: *Select Effective Length Factor*

The *Theoretical* or *Recommended* values of the K factor that are to be assigned to the selected member or group of members can be defined in this dialog box. The theoretical and the recommended values are described in [2] on page 240. Generally, it is possible to select predefined K factors or to enter *User-defined* values.

The effective lengths for buckling can also be imported from the add-on module RS-BUCK.

Buckling Possible

For the stability design of the buckling and lateral buckling, 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 stability design in STEEL AISC. 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 according to Chapter E. 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 around Axis y resp. Axis z

The columns *Buckling Possible* control members are prone to buckling around 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_y and K_z can be freely chosen for the buckling around the major and minor axes.

The orientation of the member axes can be checked in the cross-section graphics of this table. In the RSTAB work window which can be opened any time via the [Graphic] button, you can display the local member axes from the *Display* navigator.

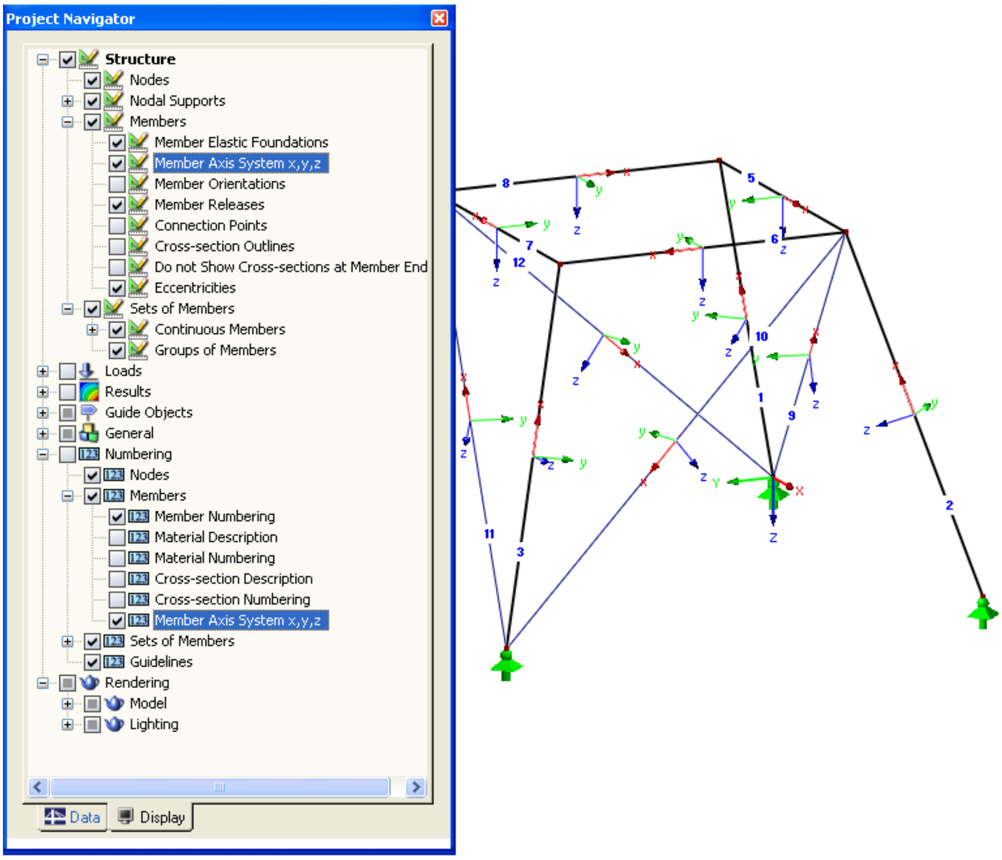


Figure 2.14: Displaying the Local Member Axes in the *Display* Navigator of RSTAB

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

If you define the buckling length coefficient K, 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 fields, you can select two nodes in the RSTAB work window graphically. Their distance then defines the buckling length.

Torsional Buckling

Column H controls whether a torsional buckling design is to be performed. The effective length factors K_x and the torsional buckling lengths K_{xL} can be defined in columns I and J. The axis x represents the centre line of a member.

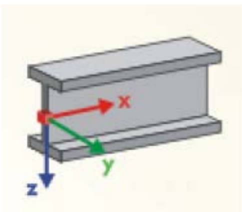


Figure 2.15: Member Axes

Lateral-Torsional Buckling L.T.B.

Column K controls whether a lateral torsional buckling design is to be carried out. The lateral-torsional buckling lengths depend on the settings of table 1.4 *Lateral Intermediate Supports*. There is no possibility to insert a user-defined value here.

The modification factor C_b for lateral-torsional buckling can be defined in table 1.7 *Design Parameters* (see Chapter 2.7).

Comment

In the last column the user can write down his own remarks at every member, e.g. to explain more closely the specific lengths of a member.

The check box *Set Inputs for Members No.* is located beneath the tree-structure lower table. If you tick this box, the data entered consequently will become valid for specific resp. *All* members. You can select the members graphically by 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 is not re-assigned.

2.6 Effective Lengths - Sets of Members

The input table 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 table 1.1 *General Data.*

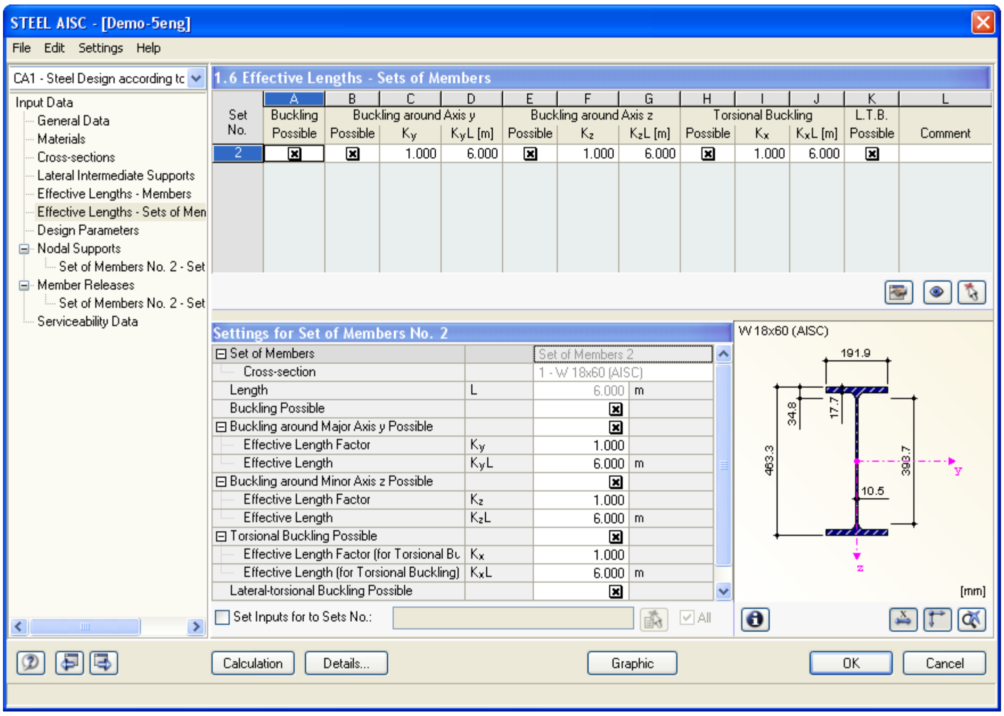


Figure 2.16: Table 1.6 *Effective Lengths - Set of Members*

This table is very similar to the previous table 1.5. With regard to the effective lengths for buckling around the major and minor axes of the cross-sections, it is identical to table 1.5.

There are differences, however, as far as the parameters for torsional and lateral-torsional buckling are concerned. These are defined by means of specific boundary conditions in table 1.8 and 1.9 (see Chapters 2.8 and 2.9).

2.7 Design Parameters

In this table, several parameters can be defined that are necessary for design.

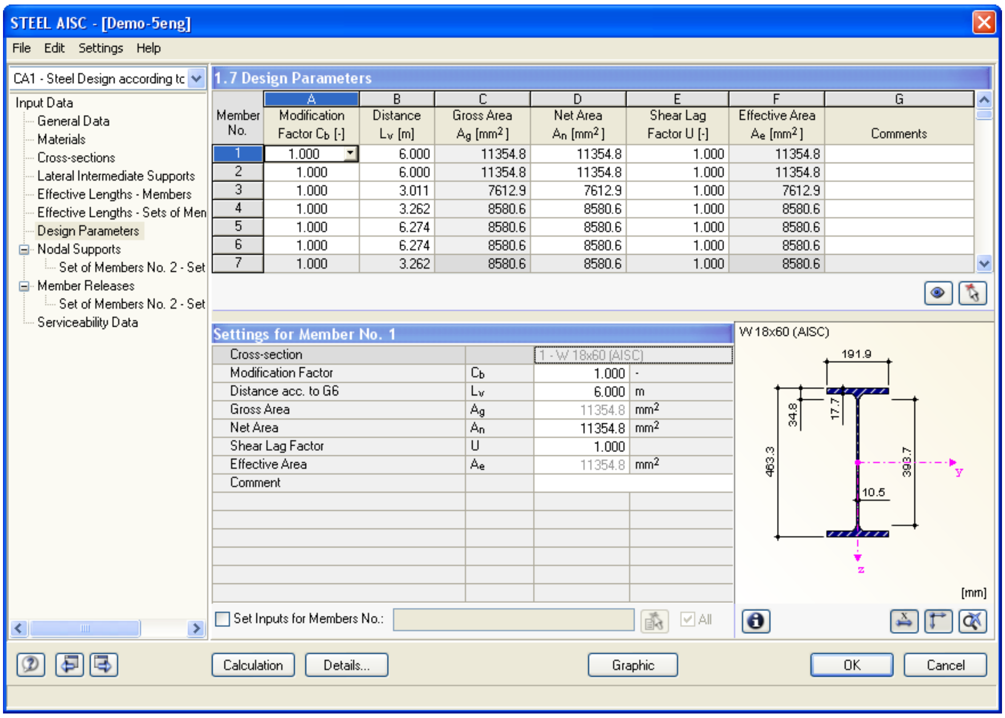
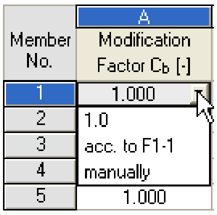


Figure 2.17: Table 1.7 *Design Parameters*

Modification Factor



In column A, there is the possibility to choose among the three options of the lateral-torsional buckling modification factor C_b. The default value is set to 1.0. The C_b factor can also be entered manually or determined by the program according to Equation F1-1 [1].

Distance L_v

The distance L_v specifies the distance between the points of maximum and zero shear force according to the Section G6 [1]. This applies to round HSS members.

Gross Area A_g / Net Area A_n

In the columns C and D, the gross and net areas of all members are listed. If required, the net area can be modified in order to consider holes.

Shear Lag Factor U

For every member, the shear lag factor for tension design can be defined according to Table D3.1 [1].

Effective Area A_e

Column F lists the effective areas of the members which are determined according to Equation D3-1 [1] from the net areas and the shear lag factors.

2.8 Nodal Supports - Sets of Members

The stability design of sets of members is carried out according to Chapter C of the ANSI/ AISC Specification [1]. According to Chapter H of this standard, you can design doubly symmetrical, singly symmetrical and unsymmetrical cross-sections stressed by compression and/or bending in a plane. The value of the multiplier α_{cr} has to be determined for the entire set of members in order to obtain the critical moment M_{cr} which is necessary for the design. The calculation of α_{cr}, the bifurcation factor, also depends on the setting in the *Details* dialog box (see Chapter 3.1, page 27).

To determine α_{cr}, a planar member structure with four degrees of freedom per node is created. The specific support conditions are defined in table 1.8. This table is only available if you have selected one or more sets of member in table 1.1 *General Data.*

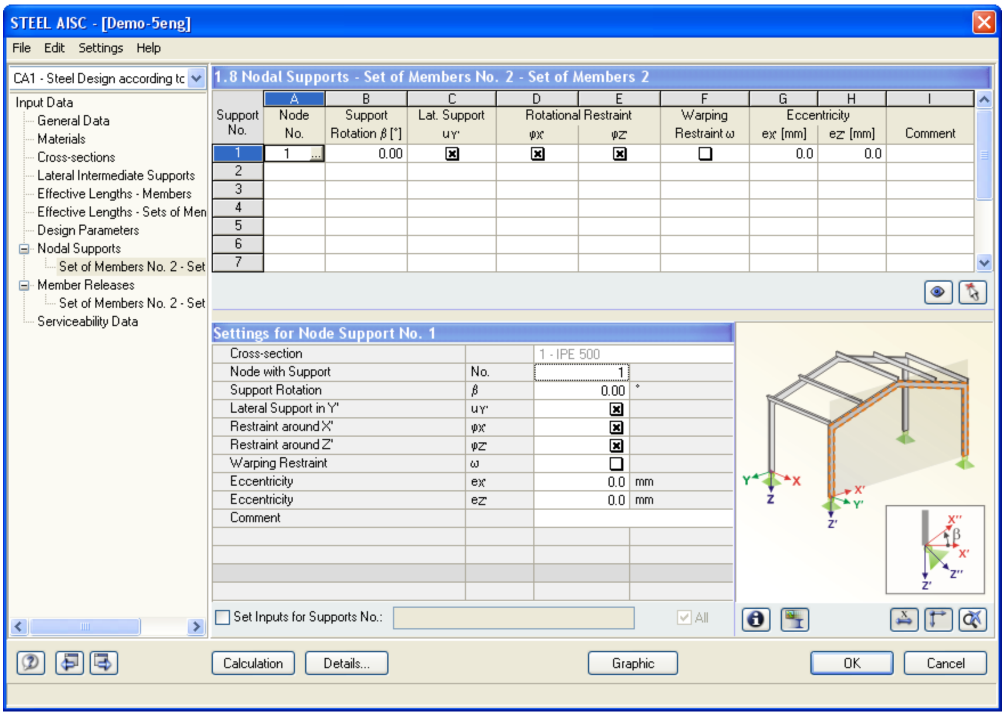


Figure 2.18: Table 1.8 *Nodal Supports - Set of Members*

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 table 1.8 (see Figure 2.19 to Figure 2.22).

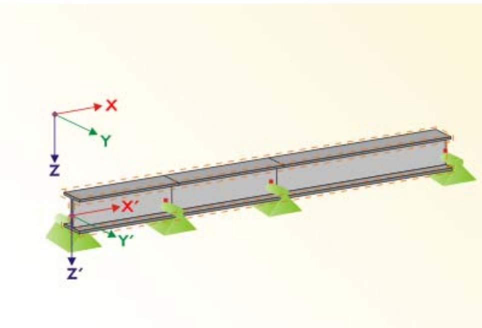


Figure 2.19: Auxiliary Coordinate System for Nodal Supports of Sets of Members

If all members within the set of members lie on a straight line, the local coordinate system of the first member within this set is applied for the entire set of members.

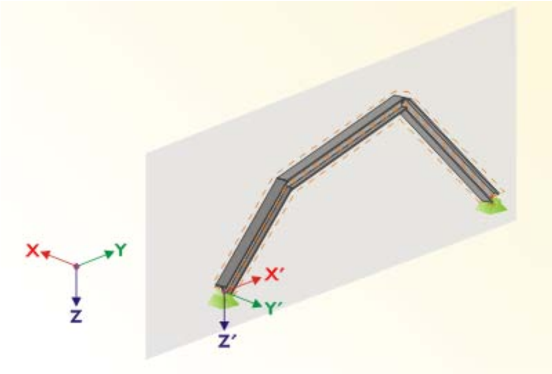


Figure 2.20: Auxiliary Coordinate System for Nodal Supports of Set of Members

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 2.20. 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.

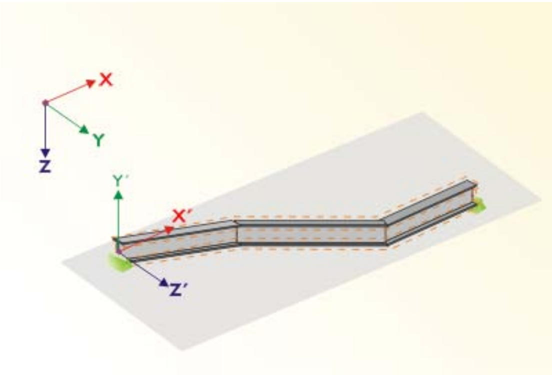


Figure 2.21: Auxiliary Coordinate System for Nodal Supports of Set of Members

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.

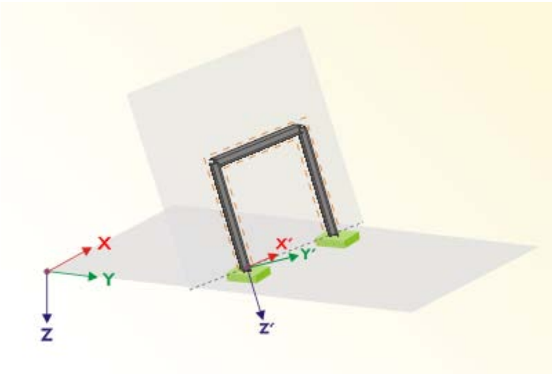


Figure 2.22: Auxiliary Coordinate System for Nodal Supports of Set of Members

Figure 2.22 shows the most general case. The members within a set of members 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'.

2.9 Member Releases - Sets of Members

This table is only available if one or more sets of members have been selected in table 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 table 1.8, then nodal releases can be inserted to a set of members in table 1.9. There is also the possibility to exactly define on which side the release is to act or to place a release at both sides.

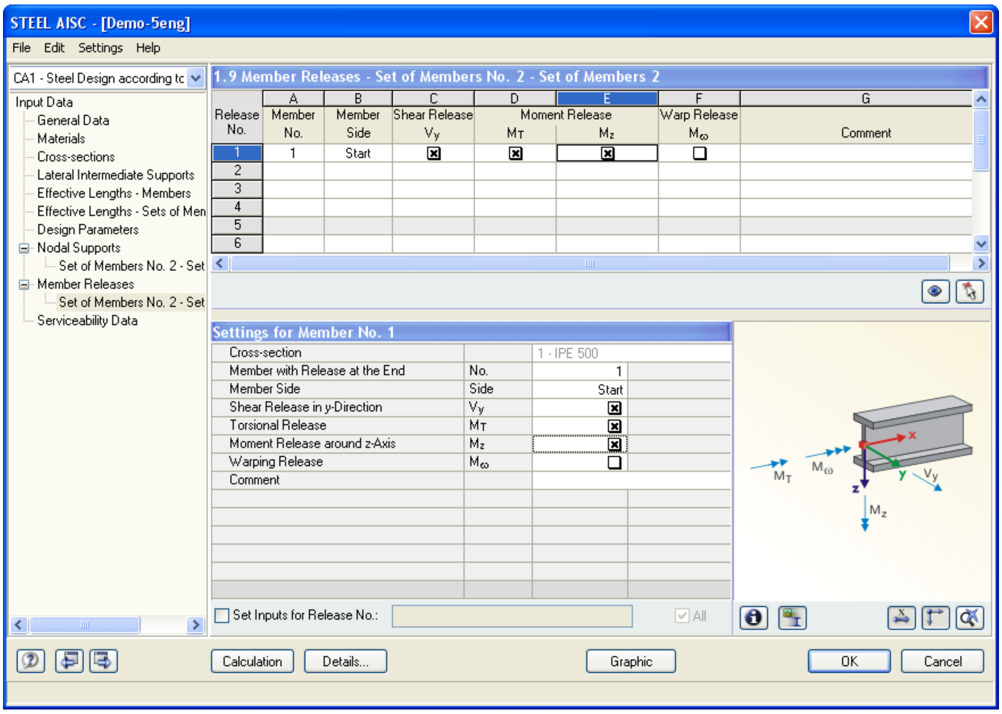
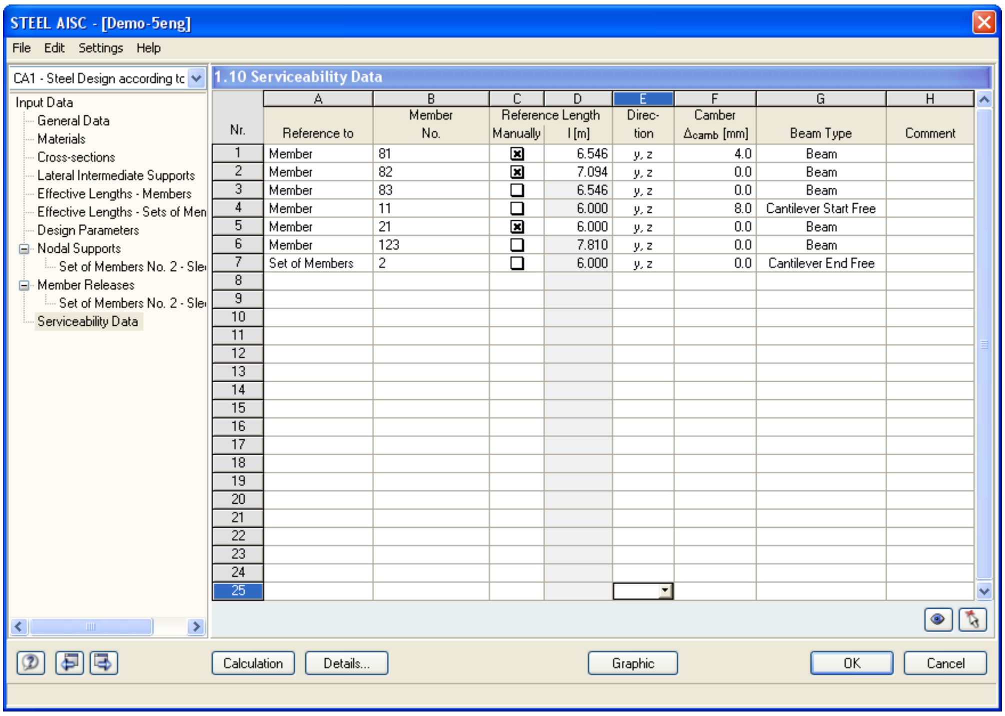


Figure 2.23: Table 1.9 *Member Releases - Set of Members*

2.10 Serviceability Data

The final input table includes different possibilities for the serviceability design. It is only displayed if the serviceability limit state design has been selected in table 1.1 *General Data* (cf Figure 2.3, page 10).



Nr.	A Reference to	B Member No.	C Reference Length Manually	D Reference Length l [m]	E Direc-tion	F Camber Δcamb [mm]	G Beam Type	H Comment
1	Member	81	☒	6.546	y, z	4.0	Beam	
2	Member	82	☒	7.094	y, z	0.0	Beam	
3	Member	83	☐	6.546	y, z	0.0	Beam	
4	Member	11	☐	6.000	y, z	8.0	Cantilever Start Free	
5	Member	21	☒	6.000	y, z	0.0	Beam	
6	Member	123	☐	7.810	y, z	0.0	Beam	
7	Set of Members	2	☐	6.000	y, z	0.0	Cantilever End Free	

Figure 2.24: Table 1.10 *Serviceability Data*

In column A, you can refer the deformation to individual members, list 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 check box 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* Δ_{camb} 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 box from every input or output table by clicking on the [Details...] button.

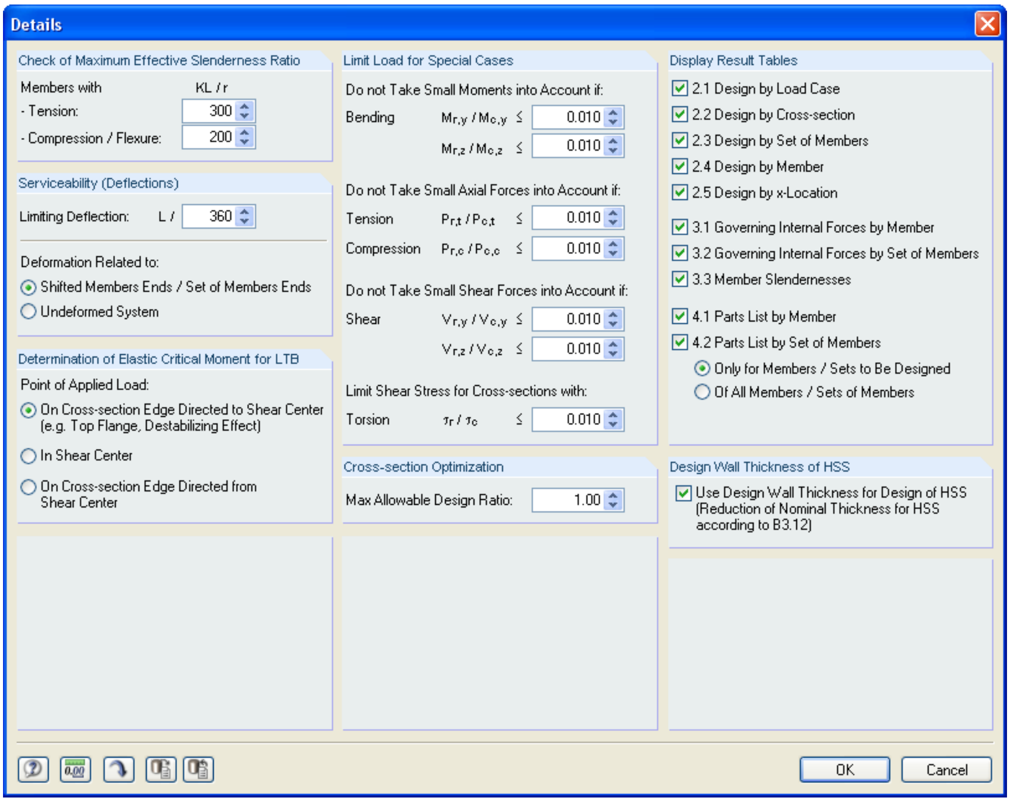


Figure 3.1: *Details* Dialog box

Check of Maximum Effective Slenderness Ratio

For members designed according to Chapter D [1], the slenderness ratio preferably should not exceed 300. This does not apply to the member types "Tension" and "Cable" because they are excluded from this check. For members designed according to Chapter E [1], the slenderness ratio preferably should not exceed 200. This value is applicable to all members with compression or flexure.

It is possible to set user-defined slenderness ratios for members with tension resp. 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 (Deflections)

In this section, it is possible to change set the allowable deflection for the serviceability limit state design if the default value L/360 is not appropriate.

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.

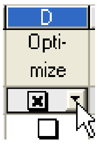
Determination of Elastic Critical Moment for LTB

The elastic critical moment M_{cr} for set of members is calculated automatically.

Usually, loads act on members. Then their application point has to be specified because this can have stabilizing or destabilizing effects, subject to the eccentricity. The *Point of Applied Load* can be set globally for all loads.

Limit Load for Special Cases

It is possible to neglect small stresses due to bending, tension or compression, shear and torsion and, thus, allow a simplified design which eliminates negligible internal forces. In this 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.



Cross-section Optimization

Cross-sections can be optimized if the Optimize option is chosen in table 1.3 *Cross-Sections*. (see Figure 2.6, page 13). The dialog box *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 results tables are described individually in Chapter 4.

Design Wall Thickness of HSS

The design wall thickness shall be taken equal to 0.93 times the nominal wall thickness for Hollow Structural Sections if the check is set here. This reduction is recommended for electric-resistance-welded HSS, not for submerged-arc-welded HSS.

3.2 Start Calculation

In all input tables of STEEL AISC, you can start the design via the [Calculation] button.

At first, STEEL AISC 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 should be optimized (see Chapter 7.2, page 48), the required sections are calculated and relevant designs are carried out.

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

Calculate → To Calculate…

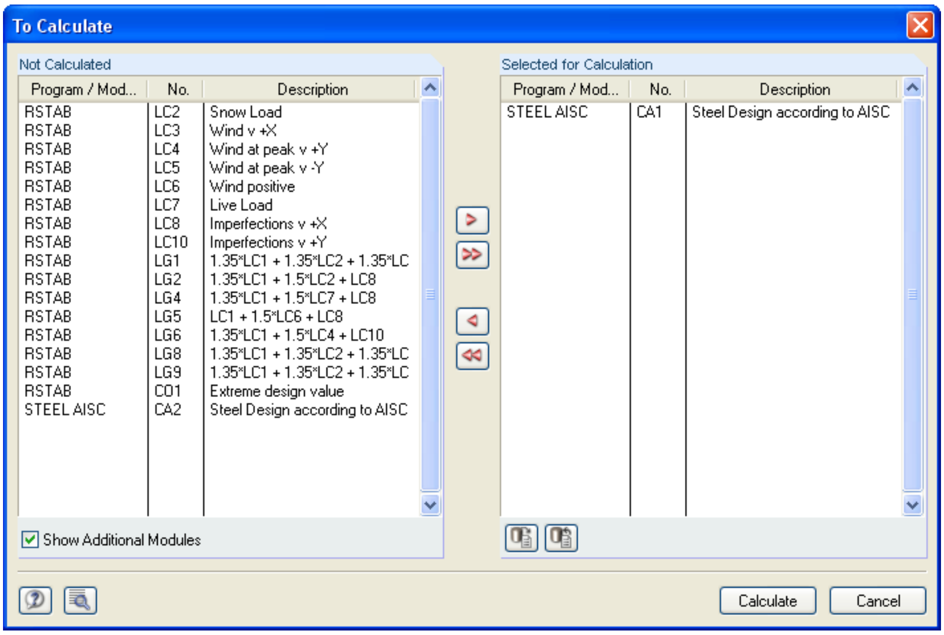
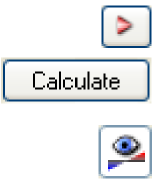


Figure 3.2: *To Calculate* Dialog box

If the design cases of STEEL AISC 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 AISC 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.

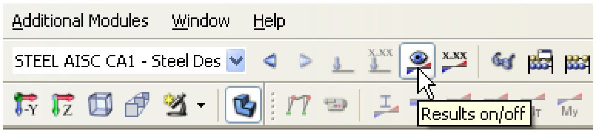


Figure 3.3: Direct Calculation of Design Case from STEEL AISC in RSTAB

A dialog box appears in which you can watch the progress of the design.

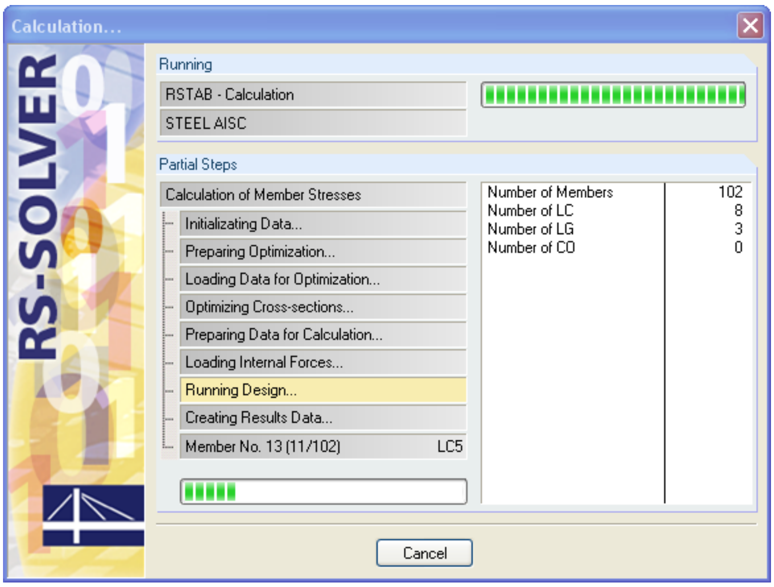


Figure 3.4: Calculation in STEEL AISC

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 the 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 the *Details* dialog box (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 AISC. You can also switch among the tables via the buttons as seen to the left or the functional keys [F2] and [F3].

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

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.

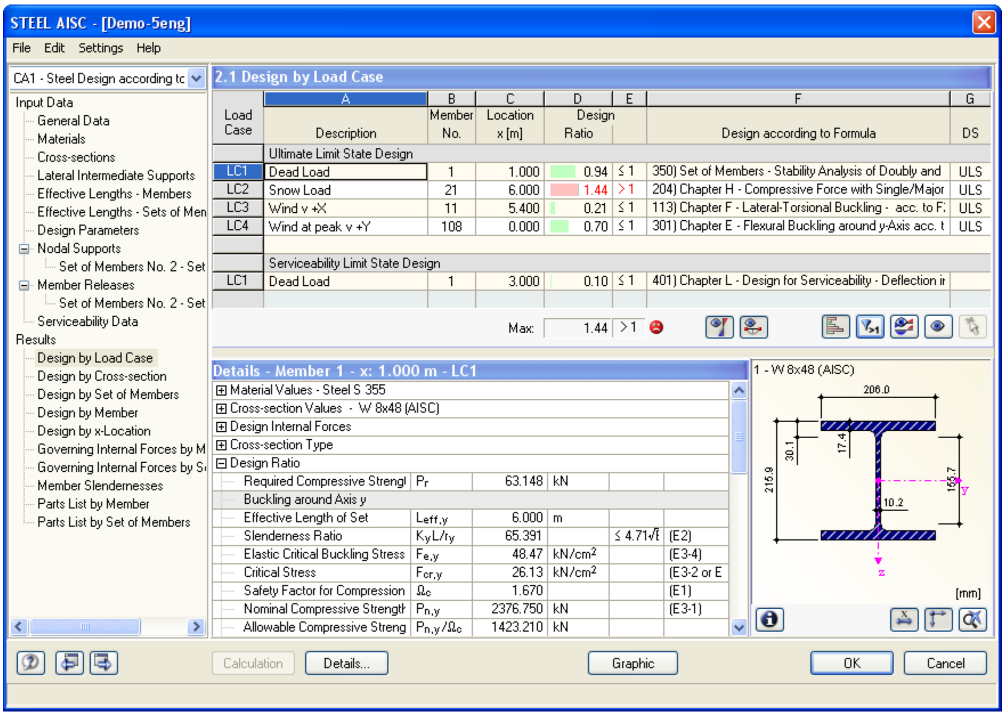


Figure 4.1: Table 2.1 *Design by Load Case*

Description

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

Member No.

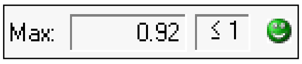
The number of the member with the highest design ratio is stated for every designed load case, load group or load combination.

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 Ratio



For every design type and for every load case, load group or load combination, the design quotients according to the standard are displayed in this column.

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: *ULS* (Ultimate Limit State) or *SLS* (Serviceability Limit State).

4.2 Design by Cross-Section

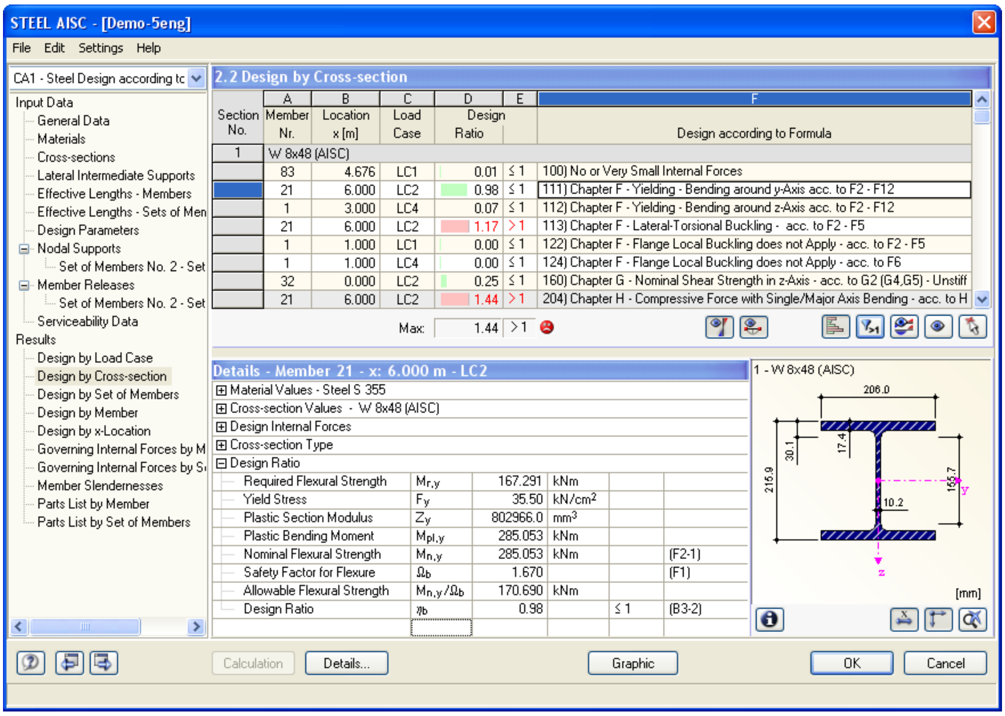


Figure 4.2: Table 2.2 *Design by Cross-Section*

In this table, the maximum design 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

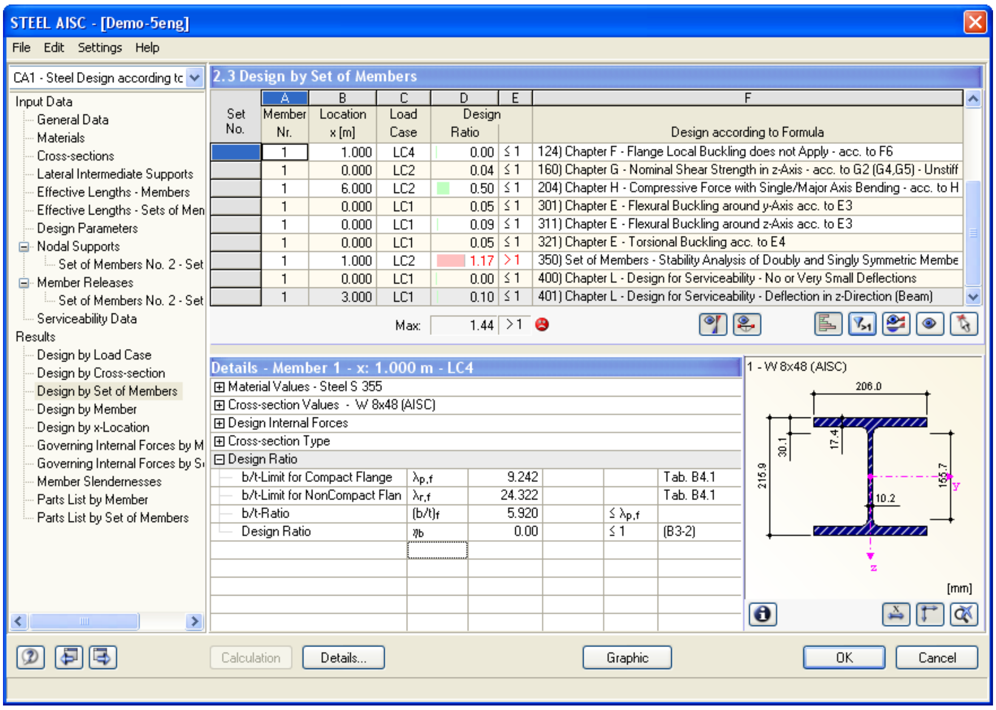


Figure 4.3: Table 2.3 *Design by Set of Members*

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

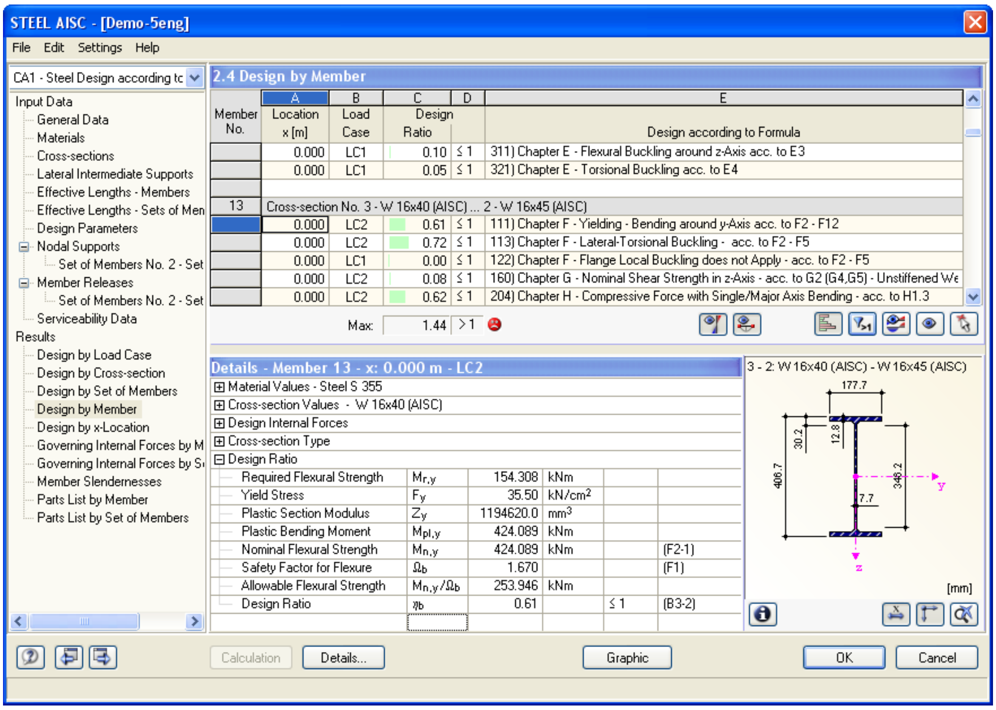


Figure 4.4: Table 2.4 *Design by Member*

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 on page 30.

4.5 Design by x-Location

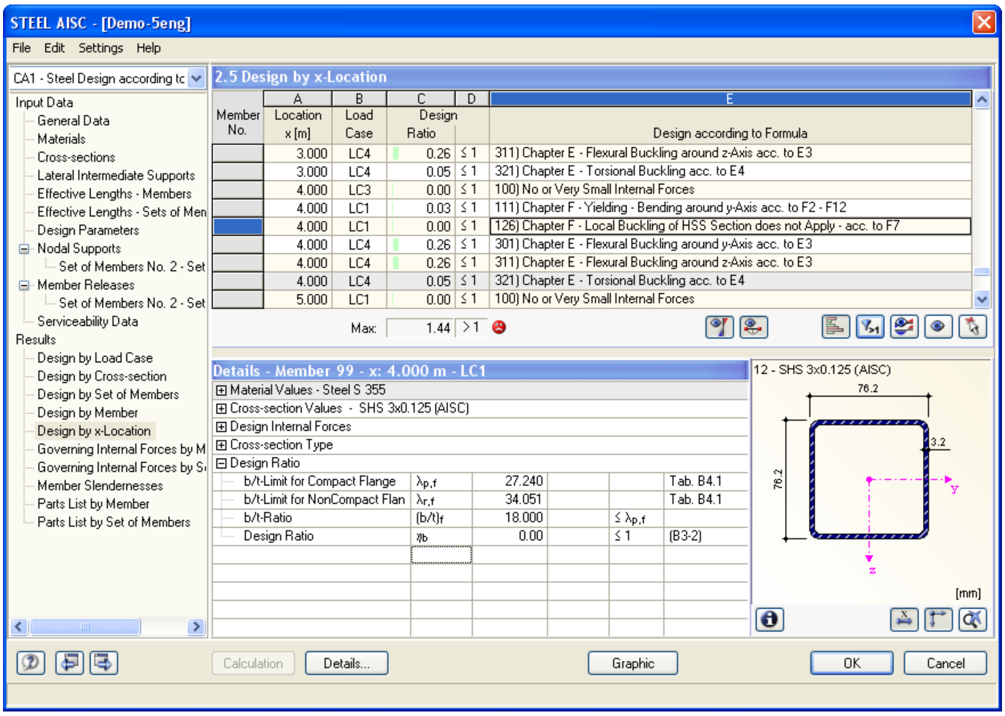


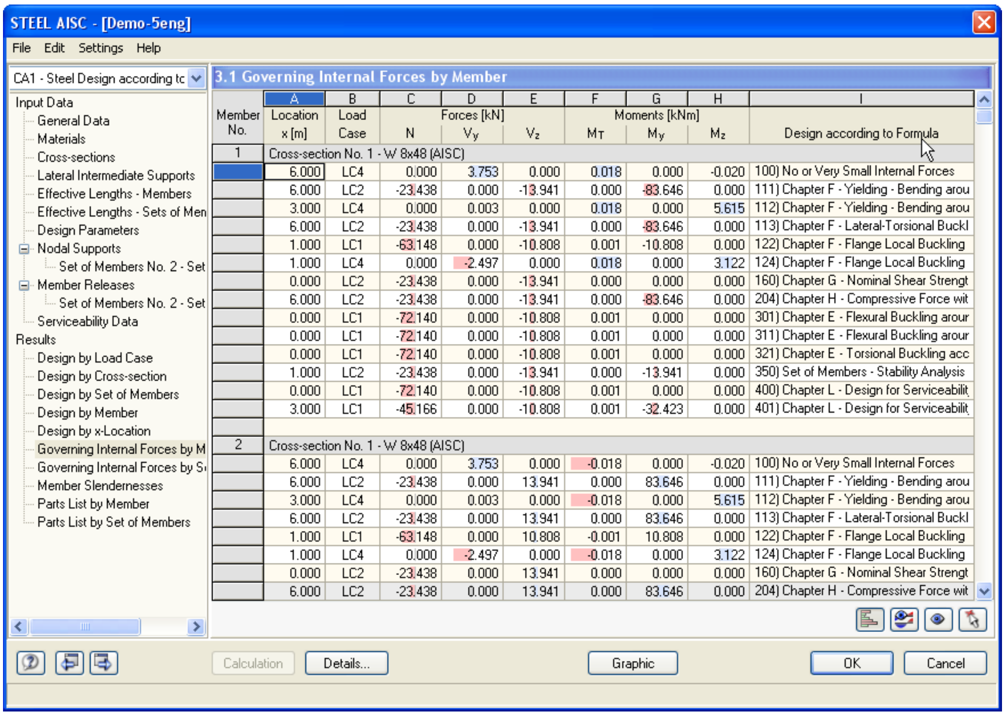
Figure 4.5: Table 2.5 *Design by x-Location*

This results table lists the maximum values of every member at the following locations x according to the 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 RSTAB *Calculation Parameters* dialog box in the *Options* tab
- 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 - W 8x48 (AISC)								
	6.000	LC4	0.000	3.753	0.000	0.018	0.000	-0.020	100) No or Very Small Internal Forces
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	111) Chapter F - Yielding - Bending arou
	3.000	LC4	0.000	0.003	0.000	0.018	0.000	5.615	112) Chapter F - Yielding - Bending arou
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	113) Chapter F - Lateral-Torsional Buckl
	1.000	LC1	-63.148	0.000	-10.808	0.001	-10.808	0.000	122) Chapter F - Flange Local Buckling
	1.000	LC4	0.000	-2.497	0.000	0.018	0.000	3.122	124) Chapter F - Flange Local Buckling
	0.000	LC2	-23.438	0.000	-13.941	0.000	0.000	0.000	160) Chapter G - Nominal Shear Strengt
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	204) Chapter H - Compressive Force wit
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	301) Chapter E - Flexural Buckling arour
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	311) Chapter E - Flexural Buckling arour
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	321) Chapter E - Torsional Buckling acc
	1.000	LC2	-23.438	0.000	-13.941	0.000	-13.941	0.000	350) Set of Members - Stability Analysis
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	400) Chapter L - Design for Serviceabilit
	3.000	LC1	-45.166	0.000	-10.808	0.001	-32.423	0.000	401) Chapter L - Design for Serviceabilit
2	Cross-section No. 1 - W 8x48 (AISC)								
	6.000	LC4	0.000	3.753	0.000	-0.018	0.000	-0.020	100) No or Very Small Internal Forces
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	111) Chapter F - Yielding - Bending arou
	3.000	LC4	0.000	0.003	0.000	-0.018	0.000	5.615	112) Chapter F - Yielding - Bending arou
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	113) Chapter F - Lateral-Torsional Buckl
	1.000	LC1	-63.148	0.000	10.808	-0.001	10.808	0.000	122) Chapter F - Flange Local Buckling
	1.000	LC4	0.000	-2.497	0.000	-0.018	0.000	3.122	124) Chapter F - Flange Local Buckling
	0.000	LC2	-23.438	0.000	13.941	0.000	0.000	0.000	160) Chapter G - Nominal Shear Strengt
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	204) Chapter H - Compressive Force wit

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 combination whose internal forces have the most unfavorable effects are displayed.

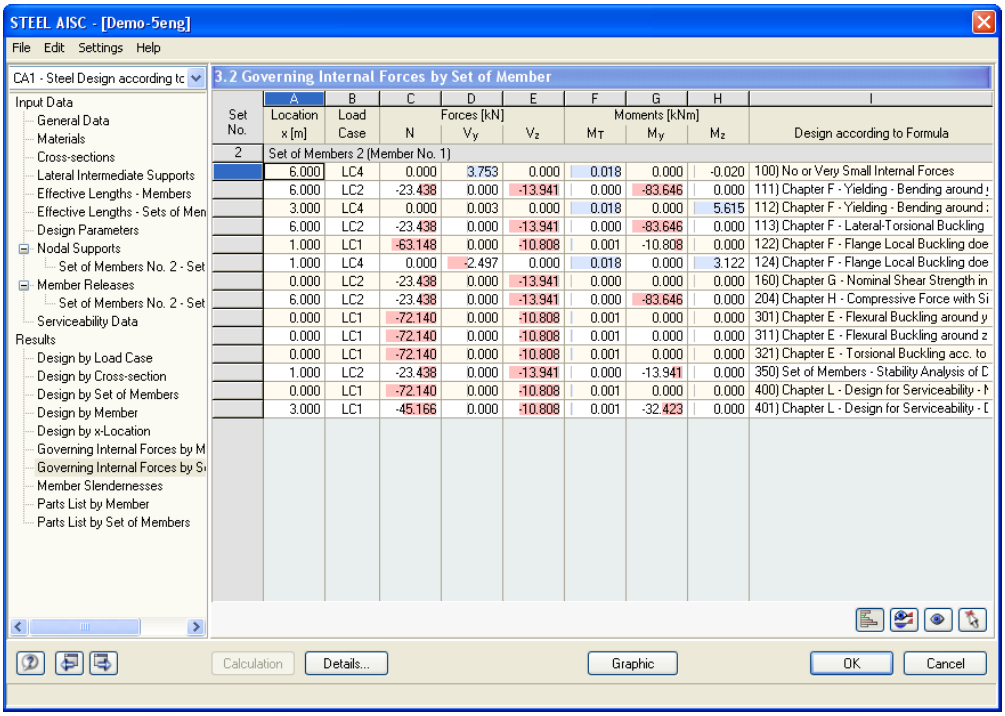
Forces / Moments

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

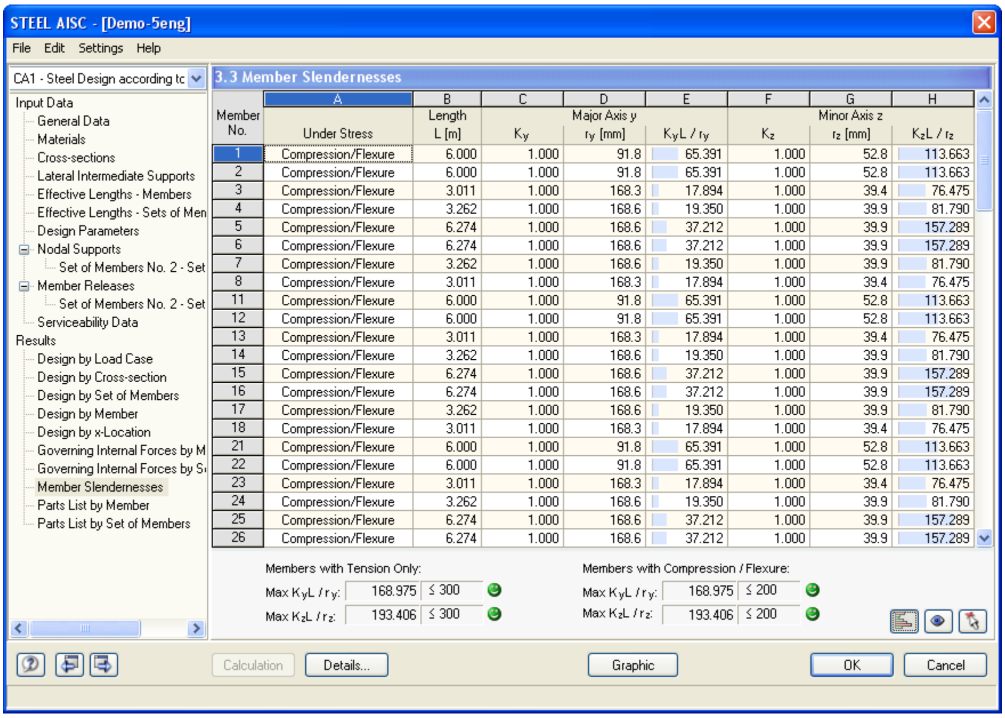


Set No.	A Location x [m]	B Load Case	C Forces [kN] N	D Forces [kN] V_y	E Forces [kN] V_z	F Moments [kNm] M_T	G Moments [kNm] M_y	H Moments [kNm] M_z	I Design according to Formula
2	Set of Members 2 (Member No. 1)								
	6.000	LC4	0.000	3.753	0.000	0.018	0.000	-0.020	100) No or Very Small Internal Forces
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	111) Chapter F - Yielding - Bending around y
	3.000	LC4	0.000	0.003	0.000	0.018	0.000	5.615	112) Chapter F - Yielding - Bending around z
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	113) Chapter F - Lateral-Torsional Buckling
	1.000	LC1	-63.148	0.000	-10.808	0.001	-10.808	0.000	122) Chapter F - Flange Local Buckling doe
	1.000	LC4	0.000	-2.497	0.000	0.018	0.000	3.122	124) Chapter F - Flange Local Buckling doe
	0.000	LC2	-23.438	0.000	-13.941	0.000	0.000	0.000	160) Chapter G - Nominal Shear Strength in
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	204) Chapter H - Compressive Force with Si
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	301) Chapter E - Flexural Buckling around y
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	311) Chapter E - Flexural Buckling around z
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	321) Chapter E - Torsional Buckling acc. to
	1.000	LC2	-23.438	0.000	-13.941	0.000	-13.941	0.000	350) Set of Members - Stability Analysis of D
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	400) Chapter L - Design for Serviceability - M
	3.000	LC1	-45.166	0.000	-10.808	0.001	-32.423	0.000	401) Chapter L - Design for Serviceability - D

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



Member No.	A Under Stress	B Length L [m]	C Major Axis y K_y	D Major Axis y r_y [mm]	E Major Axis y K_yL / r_y	F Minor Axis z K_z	G Minor Axis z r_z [mm]	H Minor Axis z K_zL / r_z
1	Compression/Flexure	6.000	1.000	91.8	65.391	1.000	52.8	113.663
2	Compression/Flexure	6.000	1.000	91.8	65.391	1.000	52.8	113.663
3	Compression/Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.475
4	Compression/Flexure	3.262	1.000	168.6	19.350	1.000	39.9	81.790
5	Compression/Flexure	6.274	1.000	168.6	37.212	1.000	39.9	157.289
6	Compression/Flexure	6.274	1.000	168.6	37.212	1.000	39.9	157.289
7	Compression/Flexure	3.262	1.000	168.6	19.350	1.000	39.9	81.790
8	Compression/Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.475
11	Compression/Flexure	6.000	1.000	91.8	65.391	1.000	52.8	113.663
12	Compression/Flexure	6.000	1.000	91.8	65.391	1.000	52.8	113.663
13	Compression/Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.475
14	Compression/Flexure	3.262	1.000	168.6	19.350	1.000	39.9	81.790
15	Compression/Flexure	6.274	1.000	168.6	37.212	1.000	39.9	157.289
16	Compression/Flexure	6.274	1.000	168.6	37.212	1.000	39.9	157.289
17	Compression/Flexure	3.262	1.000	168.6	19.350	1.000	39.9	81.790
18	Compression/Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.475
21	Compression/Flexure	6.000	1.000	91.8	65.391	1.000	52.8	113.663
22	Compression/Flexure	6.000	1.000	91.8	65.391	1.000	52.8	113.663
23	Compression/Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.475
24	Compression/Flexure	3.262	1.000	168.6	19.350	1.000	39.9	81.790
25	Compression/Flexure	6.274	1.000	168.6	37.212	1.000	39.9	157.289
26	Compression/Flexure	6.274	1.000	168.6	37.212	1.000	39.9	157.289

Members with Tension Only:
Max K_yL / r_y: 168.975 ≤ 300
Max K_zL / r_z: 193.406 ≤ 300

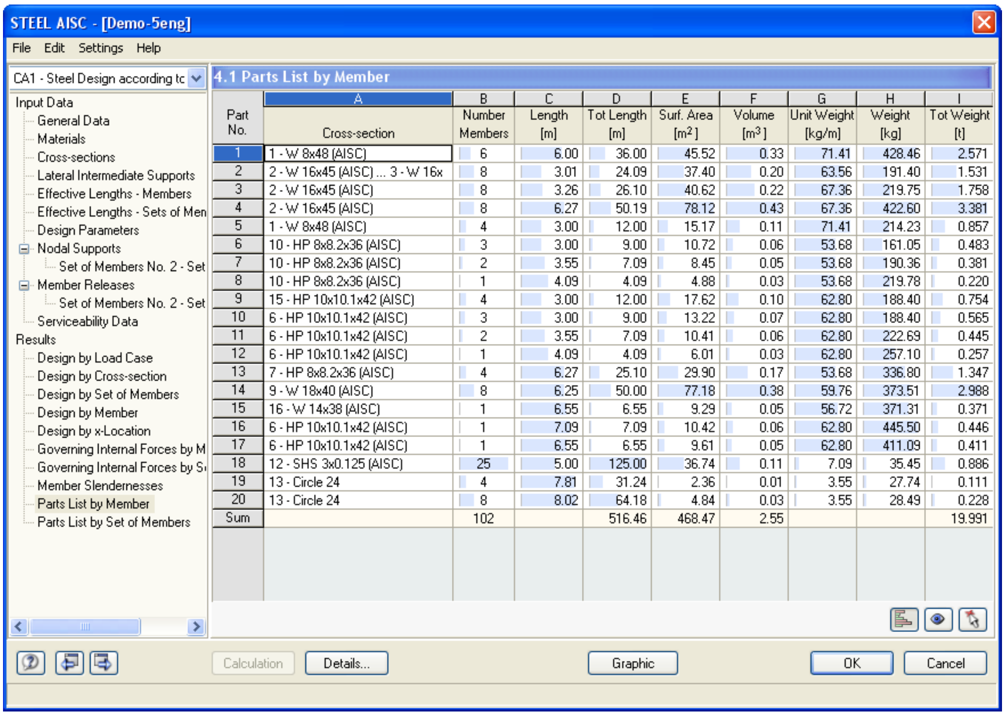
Members with Compression / Flexure:
Max K_yL / r_y: 168.975 ≤ 200
Max K_zL / r_z: 193.406 ≤ 200

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 *Details* dialog box (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.	A Cross-section	B Number Members	C Length [m]	D Tot Length [m]	E Surf. Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Tot Weight [t]
1	1 - W 8x48 (AISC)	6	6.00	36.00	45.52	0.33	71.41	428.46	2.571
2	2 - W 16x45 (AISC) ... 3 - W 16x	8	3.01	24.09	37.40	0.20	63.56	191.40	1.531
3	2 - W 16x45 (AISC)	8	3.26	26.10	40.62	0.22	67.36	219.75	1.758
4	2 - W 16x45 (AISC)	8	6.27	50.19	78.12	0.43	67.36	422.60	3.381
5	1 - W 8x48 (AISC)	4	3.00	12.00	15.17	0.11	71.41	214.23	0.857
6	10 - HP 8x8.2x36 (AISC)	3	3.00	9.00	10.72	0.06	53.68	161.05	0.483
7	10 - HP 8x8.2x36 (AISC)	2	3.55	7.09	8.45	0.05	53.68	190.36	0.381
8	10 - HP 8x8.2x36 (AISC)	1	4.09	4.09	4.88	0.03	53.68	219.78	0.220
9	15 - HP 10x10.1x42 (AISC)	4	3.00	12.00	17.62	0.10	62.80	188.40	0.754
10	6 - HP 10x10.1x42 (AISC)	3	3.00	9.00	13.22	0.07	62.80	188.40	0.565
11	6 - HP 10x10.1x42 (AISC)	2	3.55	7.09	10.41	0.06	62.80	222.69	0.445
12	6 - HP 10x10.1x42 (AISC)	1	4.09	4.09	6.01	0.03	62.80	257.10	0.257
13	7 - HP 8x8.2x36 (AISC)	4	6.27	25.10	29.90	0.17	53.68	336.80	1.347
14	9 - W 18x40 (AISC)	8	6.25	50.00	77.18	0.38	59.76	373.51	2.988
15	16 - W 14x38 (AISC)	1	6.55	6.55	9.29	0.05	56.72	371.31	0.371
16	6 - HP 10x10.1x42 (AISC)	1	7.09	7.09	10.42	0.06	62.80	445.50	0.446
17	6 - HP 10x10.1x42 (AISC)	1	6.55	6.55	9.61	0.05	62.80	411.09	0.411
18	12 - SHS 3x0.125 (AISC)	25	5.00	125.00	36.74	0.11	7.09	35.45	0.886
19	13 - Circle 24	4	7.81	31.24	2.36	0.01	3.55	27.74	0.111
20	13 - Circle 24	8	8.02	64.18	4.84	0.03	3.55	28.49	0.228
Sum		102		516.46	468.47	2.55			19.991

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 box (see Figure 3.1 on page 27) that can be opened via the [Details] button.

Part No.

The same part number is automatically assigned to identical members.

Cross-section

In this column, the cross-section description is 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 the list. The cell *Total Weight* shows the total required amount of steel.

4.10 Parts List by Set of Members

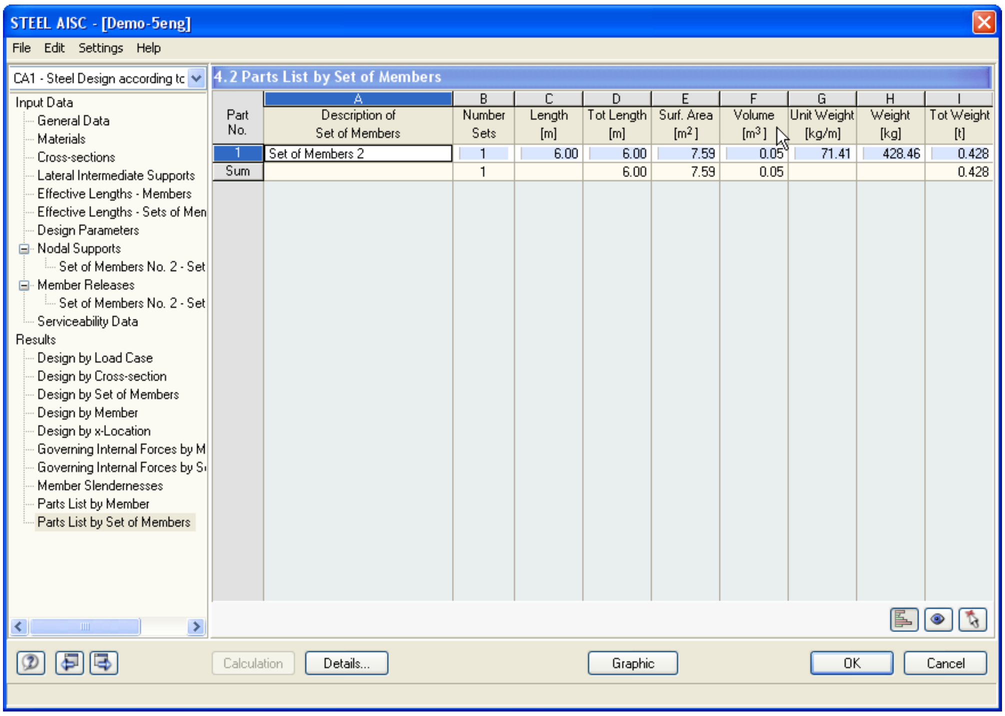


Figure 4.10: Table 4.2 *Parts List by Set of Members*

The last table in STEEL AISC 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 results can be evaluated in different ways. For this, the buttons in the results tables are very useful which are located below the upper tables.

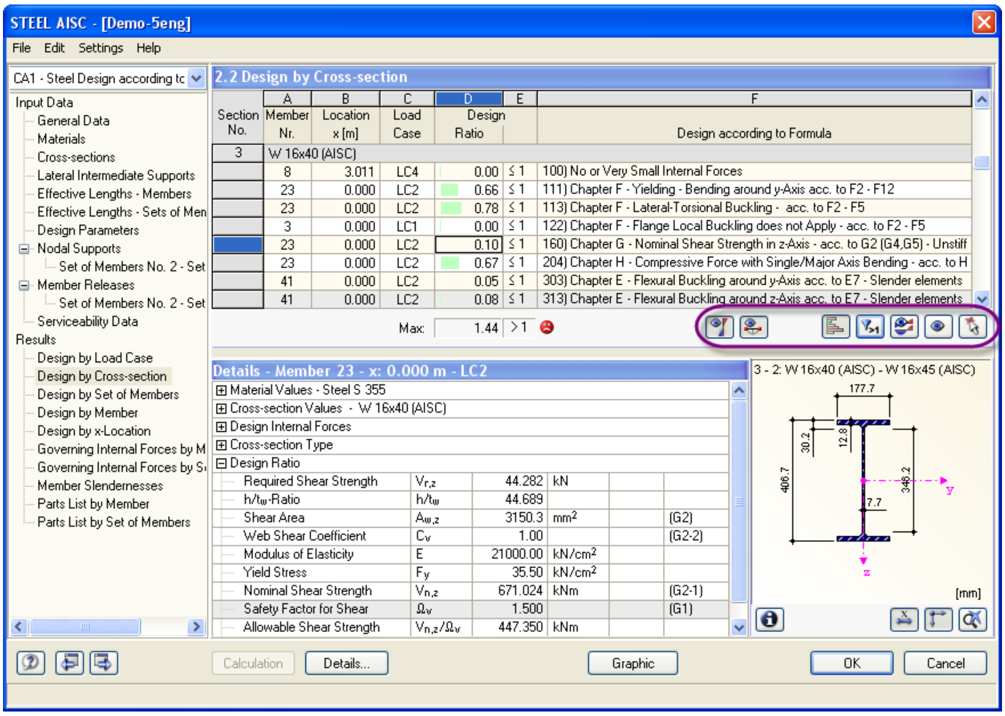


Figure 5.1: Buttons for Evaluation of Results

These buttons have the following functions:

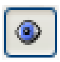
Button	Name	Function
	Design of Ultimate Limit State	Switch on/off the design results of the ultimate limit state
	Design of Serviceability Limit State	Switch on/off the design results of the serviceability limit state
	Show Color Bars in Table	Switch on/off the color background in the results tables according to the reference scale
	Show Rows with Ratio > 1	Show only rows with the 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, page 41
	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 result 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 AISC 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.

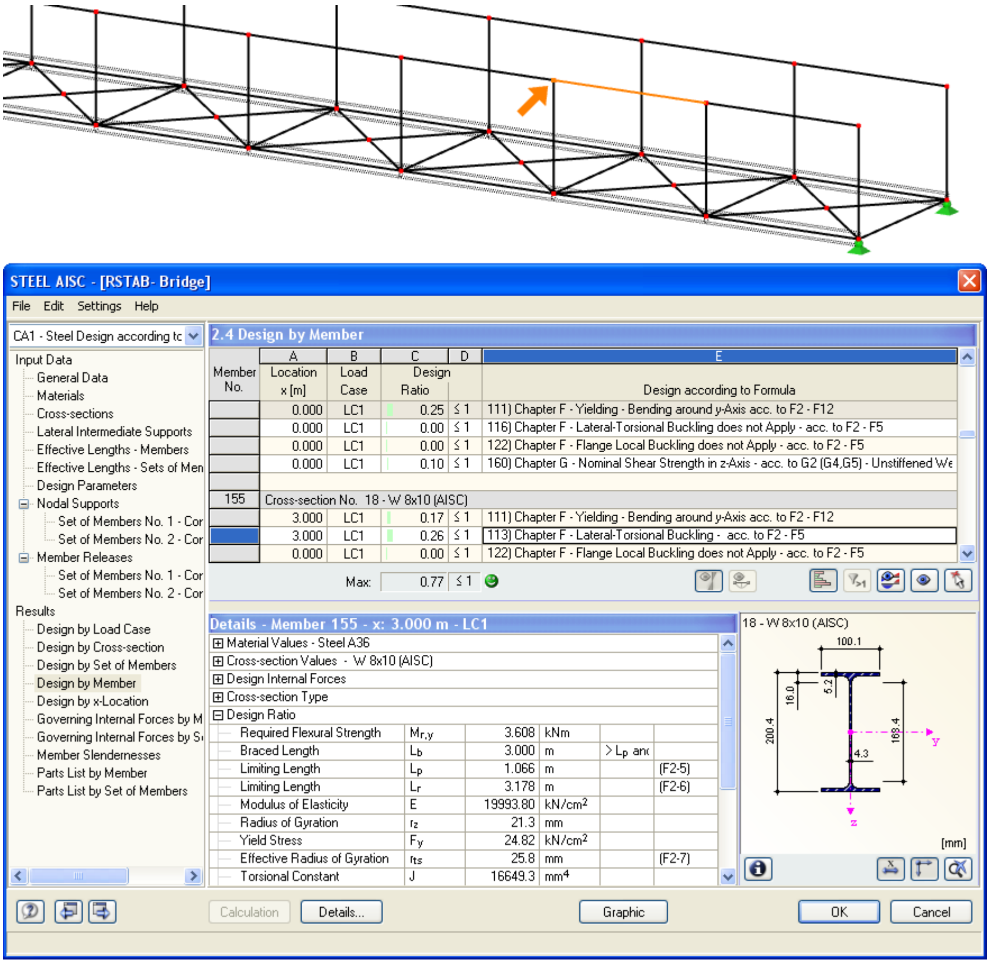
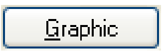


Figure 5.2: Selection of Member and Current *Location x* in RSTAB Model

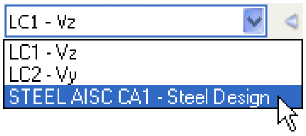
If you do not get a favorable view even by moving the STEEL AISC window, you can apply the so-called *View Mode* by clicking on the [Change View] button: the STEEL AISC window is switched off and you can change the view on the RSTAB model. In this mode, the functions from the *View* menu are available,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



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

Similarly 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 AISC 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.

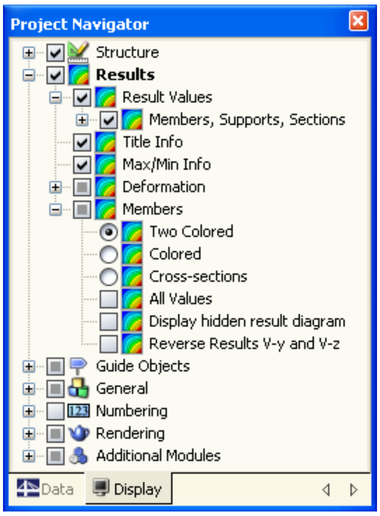
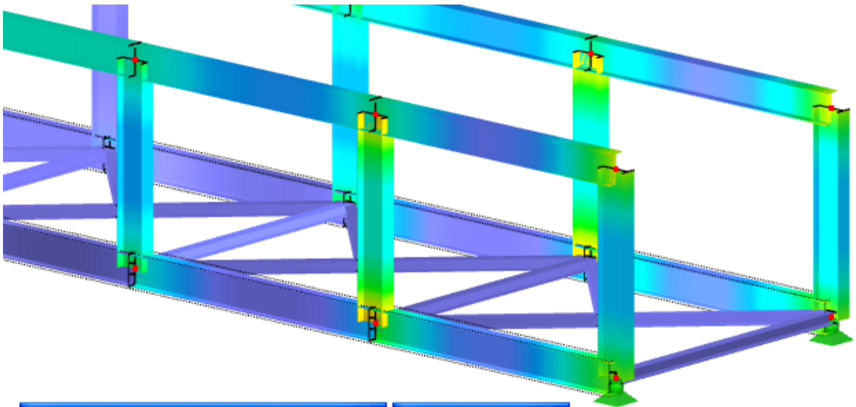


Figure 5.3: *Display* Navigator: Results → Members → **Two Colored**

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



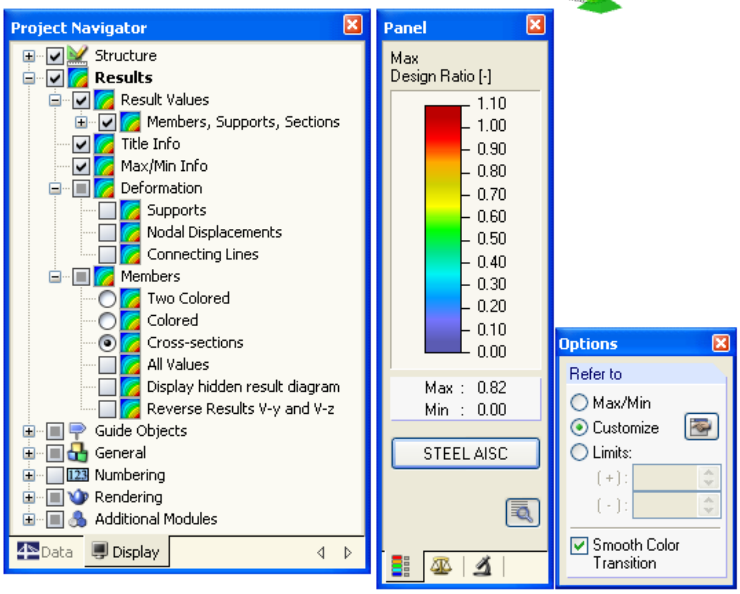


Figure 5.4: Design ratios for Option *Cross-sections* in the *Display* Navigator

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

This graphic view can be incorporated to the global printout report (see Chapter 6.2 on page 44).

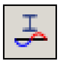
You can return to STEEL AISC any time by clicking on the [STEEL AISC] button in the panel.

5.2 Result Diagrams

In order to view the detailed distribution of results of a specific member, the graph of results can be used. Select the relevant member or set of members in the results table of STEEL AISC 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 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.

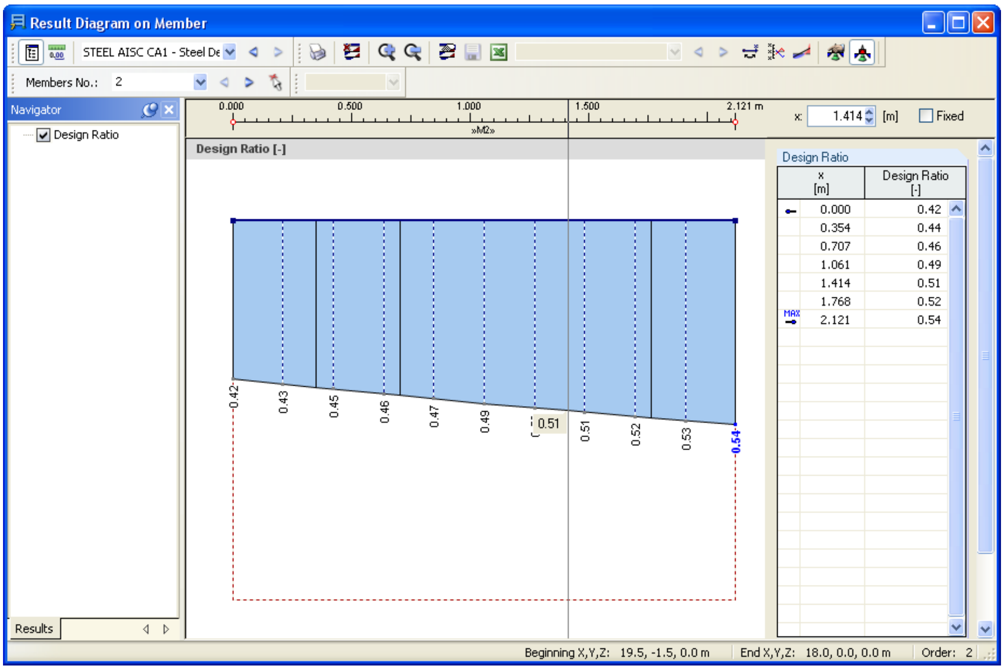
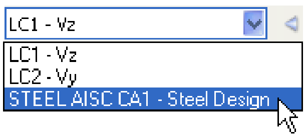


Figure 5.5: *Result Diagram on Member* Dialog Box



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

The *Result Diagram on Member* dialog box is described in detail in Chapter 9.8.4 of the RSTAB manual.

5.3 Filtering Results

The structure of the STEEL AISC tables 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 AISC 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 criteria for filtering 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 in 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* tab of the panel. 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.

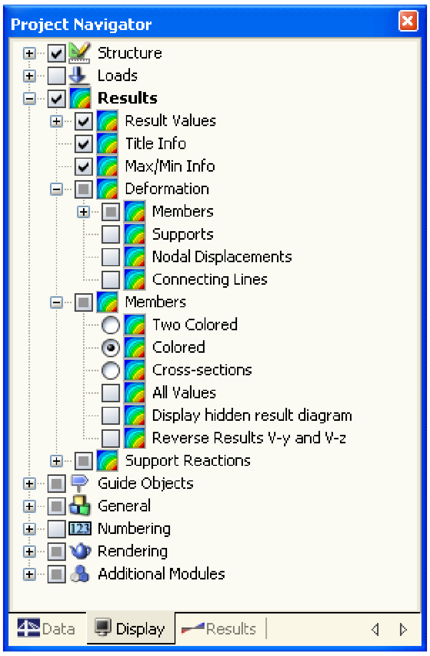


Figure 5.6: *Display* Navigator: Results → Members → **Colored**

For a colored view of the results, you can set in the panel that e.g. only design ratios greater than 0.30 are to be displayed. Furthermore, you can adjust the color spectrum in a way that one single color range exactly covers the design ratio 0.05 (see Figure 5.7).

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

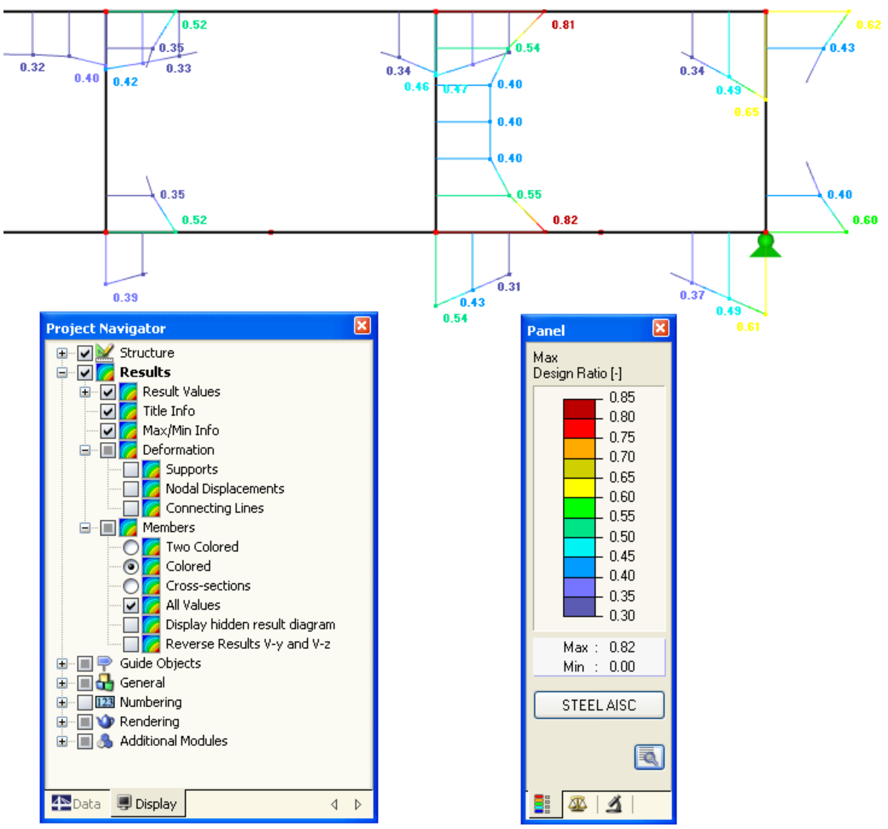


Figure 5.7: Filtering of Stress Ratios with Adjusted Color Spectrum

Filtering Members

In the *Filter* tab of the control panel, you can enter the numbers of the members whose design ratios 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 design 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 design ratios.

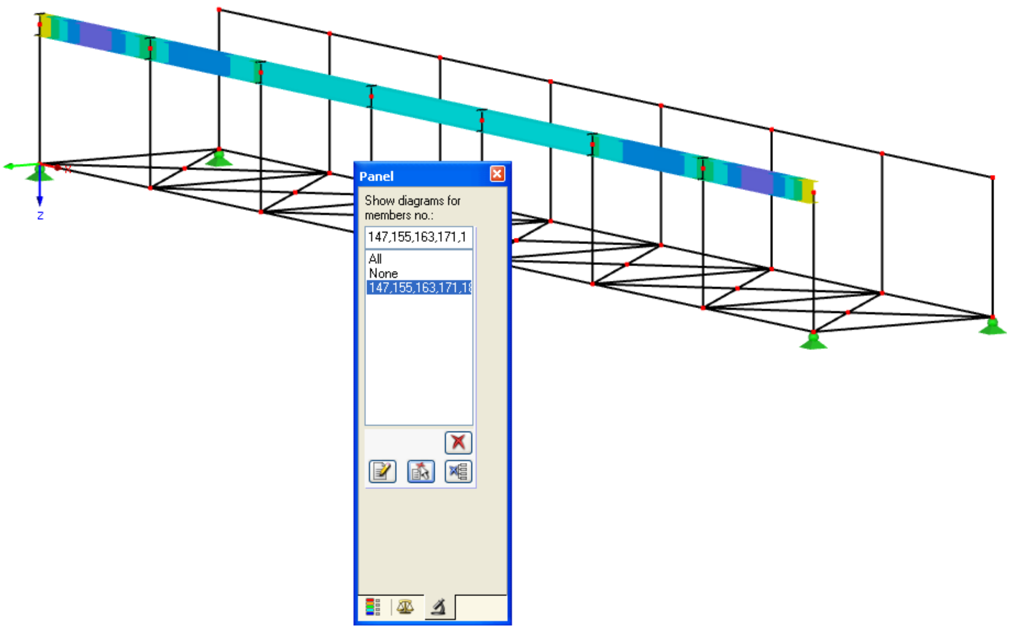


Figure 5.8: Filtering Members: Design Ratios of Footbridge Flange

6. Printout

6.1 Printout Report

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

The printout report is described in detail in the manual of the RSTAB program. In particular, Chapter 10.1.3.4 *Selecting Data of Add-on Modules* on page 227 is important. It 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 data of the STEEL AISC design case, the printout report will be processed fairly quickly.

6.2 Print STEEL AISC 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 AISC graphics is printed by using the main menu

File → Print...

or by 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* window.

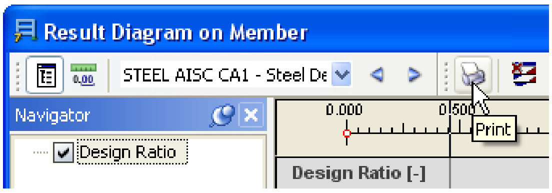


Figure 6.2: *Print* Button in Toolbar of *Result Diagram* Window

The following dialog box opens.

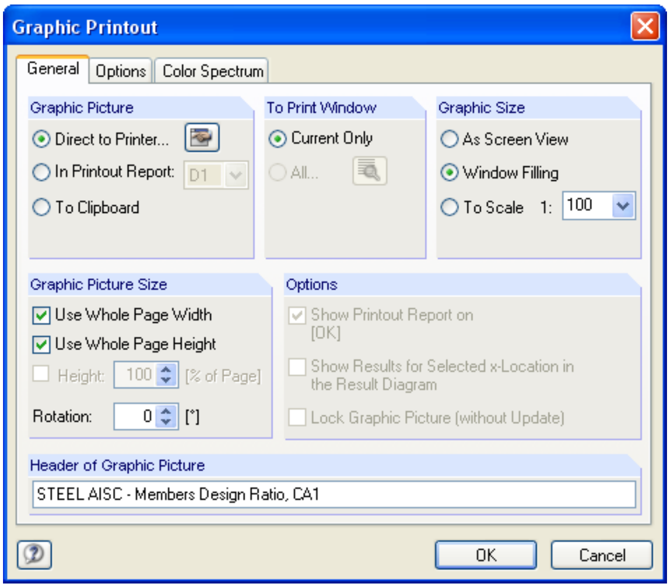


Figure 6.3: *Graphic Printout* Dialog Box, *General* Tab

This dialog box is described in detail in chapter 10.2 on page 243 in the RSTAB manual. The remaining two tabs *Options* and *Color Spectrum* are also explained there.

In the printout report, any image of the STEEL AISC 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 box is displayed again in which the possible changes can be set.

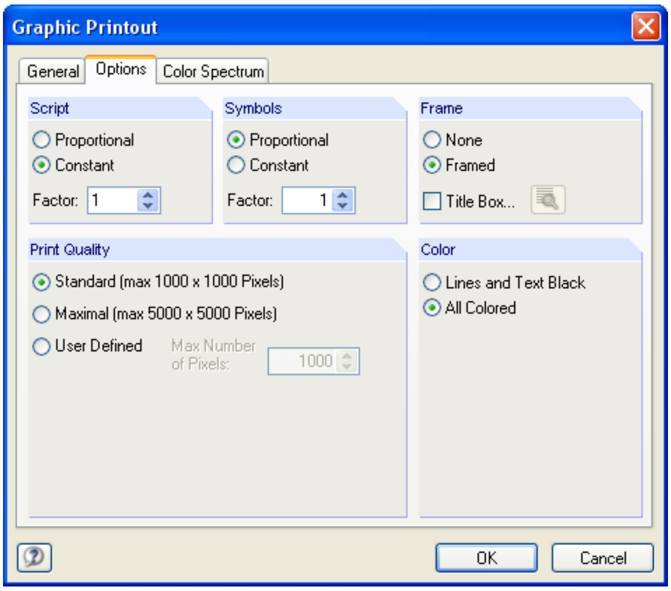


Figure 6.4: *Graphic Printout* Dialog Box, *Options* Tab

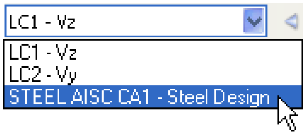
7. General Functions

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

7.1 STEEL AISC Design Cases

There is an option to group members into separate design cases. In this way, it is possible to design separately certain structural parts with specific parameters, for example.

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



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

Create New STEEL AISC Case

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

File → New Case…

The following dialog box opens:

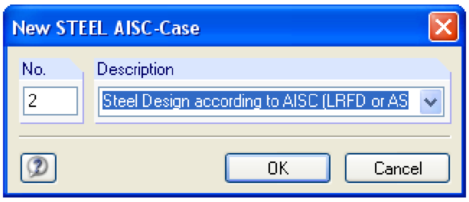


Figure 7.1: *New STEEL AISC-Case* Dialog Box

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

Rename STEEL AISC Case

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

File → Rename Case…

The *Rename STEEL AISC-Case* dialog box is opened.

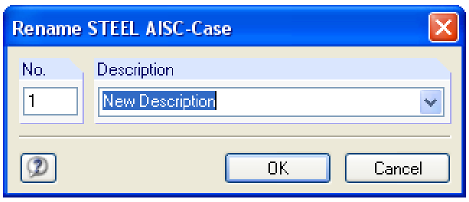


Figure 7.2: *Rename STEEL AISC-Case* Dialog Box

Copy STEEL AISC Case

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

File → Copy Case…

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

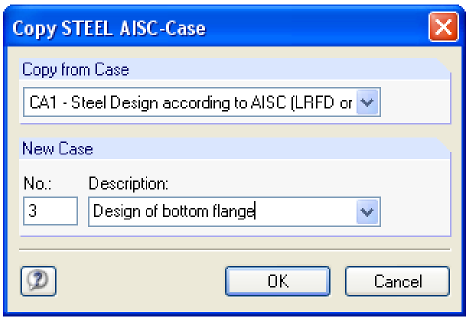


Figure 7.3: *Copy STEEL AISC-Case* Dialog Box

Delete STEEL AISC Case

Design cases can be deleted via the STEEL AISC main menu

File → Delete Cases…

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

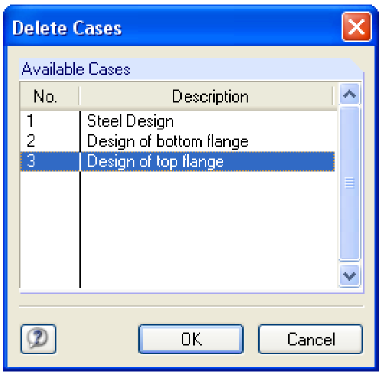
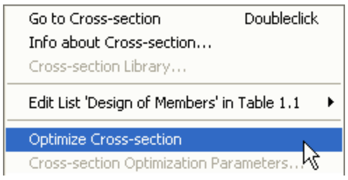


Figure 7.4: *Delete Cases* Dialog Box

7.2 Cross-Section Optimization

The module STEEL AISC offers the possibility to optimize cross-sections. For this, select the cross-section that is to be optimized in column D resp. E of table 1.3 *Cross-sections* by ticking the appropriate box (see Figure 2.6 on page 13).



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

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

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

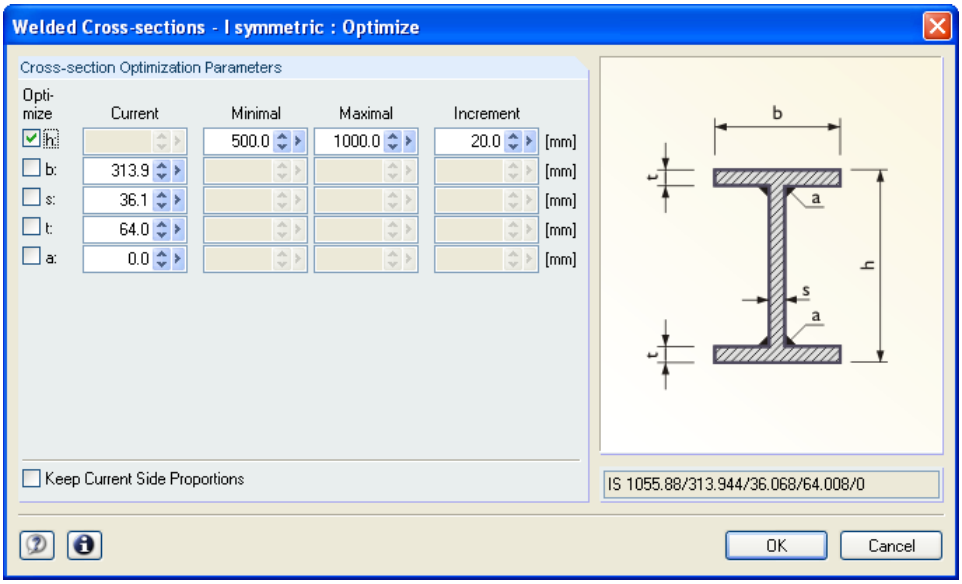


Figure 7.5: *Welded Cross-Sections - I-Symmetric: Optimize* Dialog Box

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 each 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 box. 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 table 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 table 1.3.

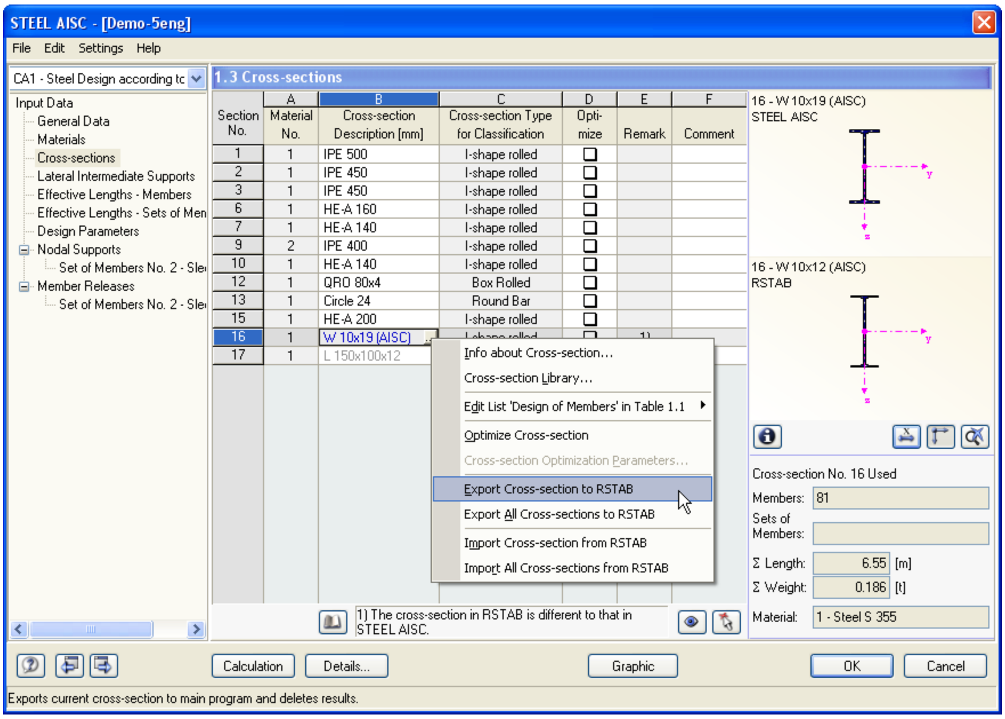


Figure 7.6: Context Menu in Table 1.3 *Cross-Sections*

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 AISC, the internal forces of RSTAB and the design ratios of STEEL AISC are calculated in one calculation run.

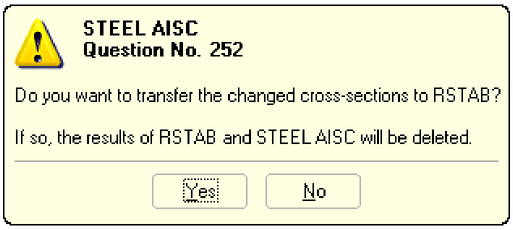


Figure 7.7: Question before Transferring Modified Cross-Sections to RSTAB

In a similar way, you can reload the original cross-sections from RSTAB to STEEL AISC by using the appropriate functions in the main menu or context menu. Please note that this option is only available in table 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 this, 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 table 1.2 of STEEL AISC, you can export it to RSTAB like cross-sections or also reload the original material from RSTAB to the module. The materials that have been modified in the module are marked in blue color.

It is not necessary to transfer the modified materials to RSTAB manually. Instead, open table 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 table 1.2.

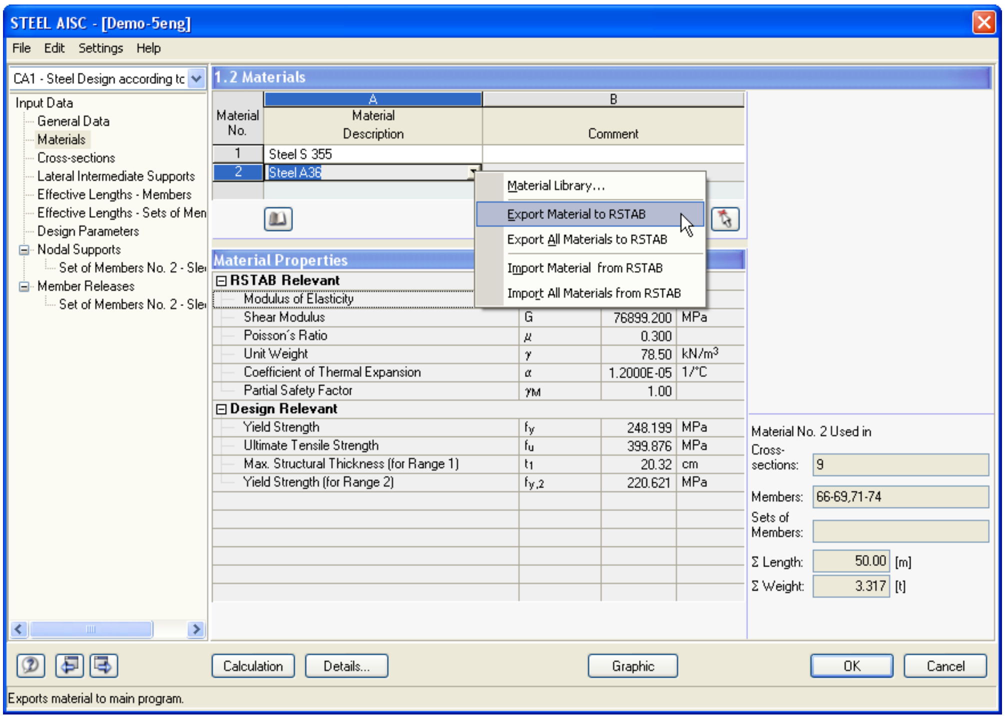


Figure 7.8: Context Menu in Table 1.2 *Materials*

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 AISC, the internal forces of RSTAB and the design ratios of STEEL AISC are calculated in one calculation run.

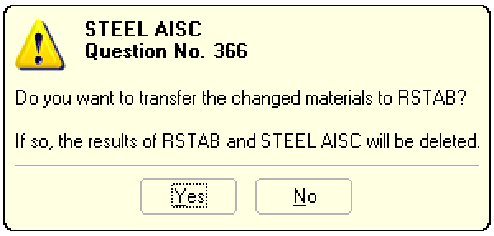


Figure 7.9: Question before Transferring Modified Materials to RSTAB

7.4 Units and Decimal Places

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

Settings → Units and Decimal Places…

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

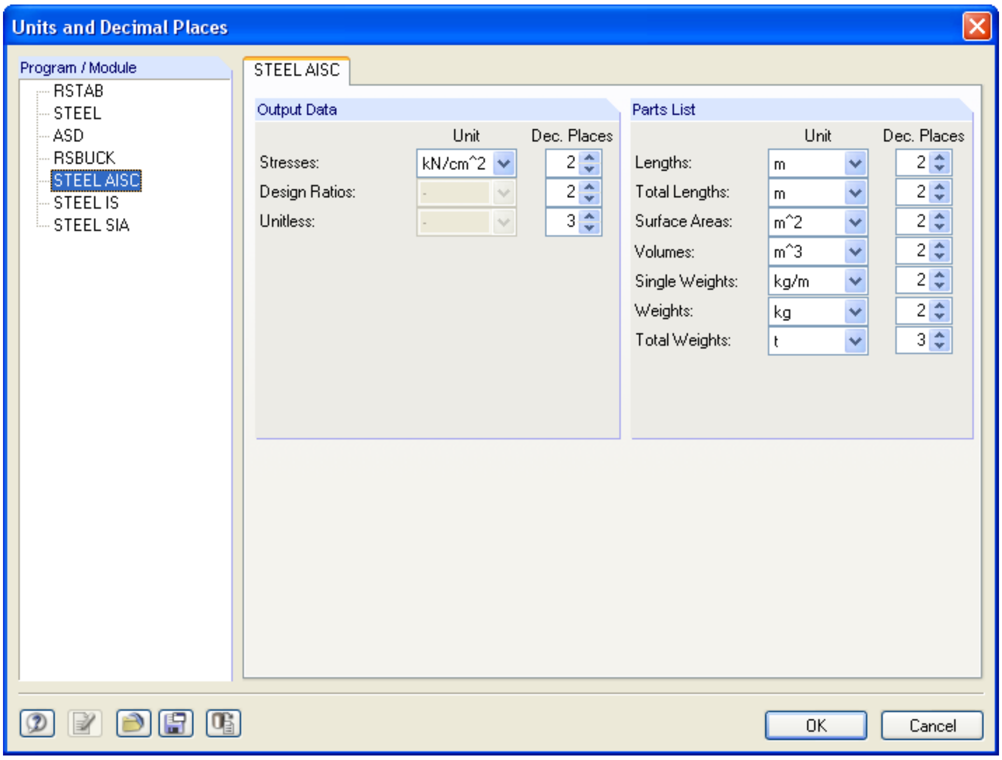


Figure 7.10: *Units and Decimal Places* Dialog Box

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 Exporting Results

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

Clipboard

Select the relevant cells in the results table of STEEL AISC and copy them to the clipboard via [Ctrl]+[C]. The contents can then be inserted via [Ctrl]+[V] to e.g. some text processing program. The headers of the table columns are not exported.

Printout Report

The STEEL AISC data can be sent to the printout report (see Chapter 6.1, page 44) 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 on page 239.

Excel / OpenOffice

STEEL AISC 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 box opens:

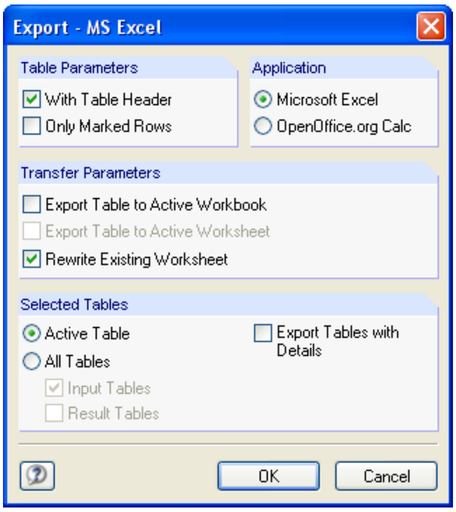


Figure 7.11: *Export - MS Excel* Dialog Box

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.

	A	B	C	D	E	F	G
1	Section	Member	Location	Load	Design		
2	No.	Nr.	x [m]	Case	Ratio		Design according to Formula
3	1	IPE 500					
4		21	6,000	LC2	0,36	≤ 1	111) Chapter F - Yielding - Bending around y-Axis acc. to F2 - F12
5		21	6,000	LC2	0,66	≤ 1	113) Chapter F - Lateral-Torsional Buckling - acc. to F2 - F5
6		1	1,000	LC2	0,00	≤ 1	122) Chapter F - Flange Local Buckling does not Apply - acc. to F2 - F5
7		32	0,000	LC2	0,10	≤ 1	160) Chapter G - Nominal Shear Strength in z-Axis - acc. to G2 (G4,G5) - Uns
8		21	6,000	LC2	0,51	≤ 1	204) Chapter H - Compressive Force with Single/Major Axis Bending - acc. to
9		21	0,000	LC2	0,02	≤ 1	303) Chapter E - Flexural Buckling around y-Axis acc. to E7 - Slender element
10		21	0,000	LC2	0,07	≤ 1	313) Chapter E - Flexural Buckling around z-Axis acc. to E7 - Slender element
11		21	0,000	LC2	0,03	≤ 1	323) Chapter E - Torsional Buckling acc. to E4 and E7 - Slender elements
12		1	1,000	LC2	1,33	> 1	350) Set of Members - Stability Analysis of Doubly and Singly Symmetric Mem
13							
14	2	IPE 450					
15		23	0,000	LC2	0,46	≤ 1	111) Chapter F - Yielding - Bending around y-Axis acc. to F2 - F12
16		25	5,647	LC2	0,56	≤ 1	113) Chapter F - Lateral-Torsional Buckling - acc. to F2 - F5
17		3	0,000	LC2	0,00	≤ 1	122) Chapter F - Flange Local Buckling does not Apply - acc. to F2 - F5
18		23	0,000	LC2	0,07	≤ 1	160) Chapter G - Nominal Shear Strength in z-Axis - acc. to G2 (G4,G5) - Uns
19		23	0,000	LC2	0,47	≤ 1	204) Chapter H - Compressive Force with Single/Major Axis Bending - acc. to
20		43	0,000	LC2	0,04	≤ 1	303) Chapter E - Flexural Buckling around y-Axis acc. to E7 - Slender element
21		43	0,000	LC2	0,16	≤ 1	313) Chapter E - Flexural Buckling around z-Axis acc. to E7 - Slender element
22		43	0,000	LC2	0,06	≤ 1	323) Chapter E - Torsional Buckling acc. to E4 and E7 - Slender elements
23							
24	3	IPE 450					
25		23	0,000	LC2	0,46	≤ 1	111) Chapter F - Yielding - Bending around y-Axis acc. to F2 - F12
26		23	0,000	LC2	0,53	≤ 1	113) Chapter F - Lateral-Torsional Buckling - acc. to F2 - F5

Figure 7.12: Results in Excel

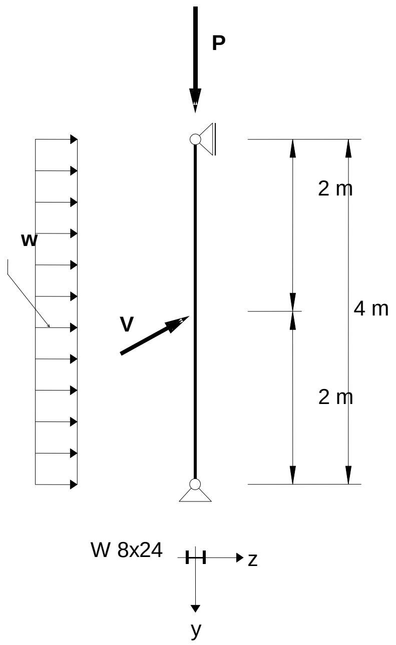
8. Example

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. The calculation described below follows the *Load and Resistance Factor Design* provisions.

Design Values

Structure and Loads



Design values of static loads:

P = 300 kN
w = 5.0 kN/m
V = 7.5 kN

Cross-section: W 8x24

Material: ASTM A36

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

Internal Forces according to Linear Static Analysis

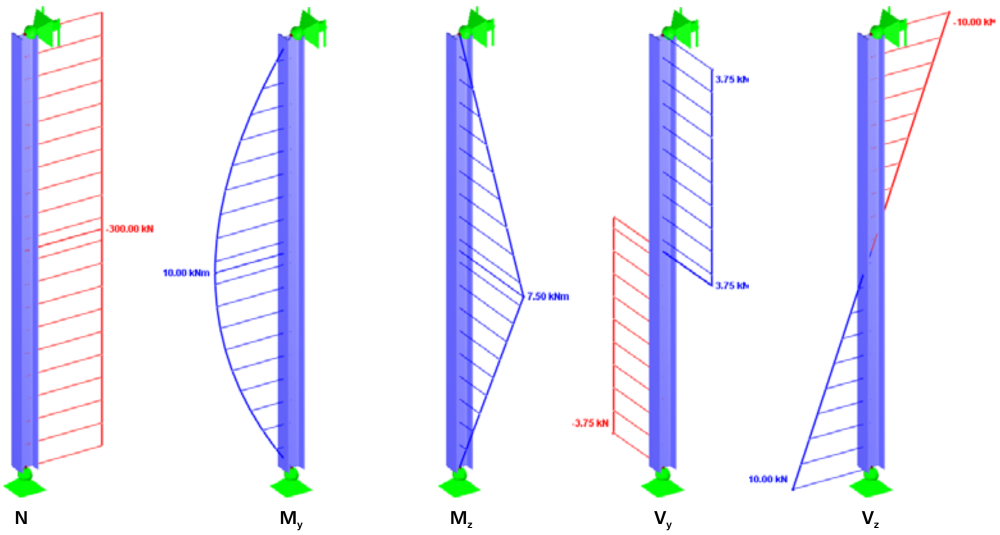


Figure 8.2: Internal Forces

Design Location (Decisive Location x)

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

P_r = –300.00 kN M_{ry} = 10.00 kNm M_{rz} = 7.50 kNm V_{ry} = 3.75 kN V_{rz} = 0.00 kN

Cross-Section Properties W 8x24, A36

Cross-Section Property	Symbol	Value	Units
Gross area of cross-section	A_g	45.677	cm^2
Moment of inertia	I_y	3446.40	cm^4
Moment of inertia	I_z	761.70	cm^4
Radius of inertia	r_y	8.687	cm
Radius of inertia	r_z	4.089	cm
Cross-section weight	G	35.9	kg/m
Moment of torsional rigidity	J	14.57	cm^4
Warping moment of inertia	C_w	69550.80	cm^6
Elastic cross-section modulus	S_y	342.49	cm^3
Elastic cross-section modulus	S_z	92.26	cm^3
Plastic cross-section modulus	Z_y	380.18	cm^3
Plastic cross-section modulus	Z_z	140.44	cm^3

Classification of cross-section – Table B4.1

Compression

Flange

Case 3 – Uniform compression in flanges of rolled I-shaped section:

$$b / t = 82.49 / 10.16 = 8.119$$

$$\lambda_p = N / A$$

$$\lambda_r = 0.56\sqrt{E / F_y} = 0.56\sqrt{199938 / 248.2} = 15.894$$

Flange is *NonCompact* in compression.

Web

Case 10 – Uniform compression in web of doubly symmetric rolled I-shaped section:

$$h / t_w = 155.7 / 6.22 = 25.02$$

$$\lambda_p = N / A$$

$$\lambda_r = 1.49\sqrt{E / F_y} = 1.49\sqrt{199938 / 248.2} = 42.29$$

Web is *NonCompact* in compression.

The section is NonCompact in compression.

Flexure

Case 1 – Flexure in flanges of rolled I-shaped section:

$$b / t = 82.49 / 10.16 = 8.119$$

$$\lambda_p = 0.38\sqrt{E / F_y} = 0.38\sqrt{199938 / 248.2} = 10.785$$

$$\lambda_r = 1.0\sqrt{E / F_y} = 1.0\sqrt{199938 / 248.2} = 28.382$$

Flange is *Compact* in flexure.

Case 9 – Flexure in web of doubly symmetric rolled I-shaped section:

$h / t_w = 155.7 / 6.22 = 25.02$

$\lambda_p = 3.76\sqrt{E / F_y} = 3.76\sqrt{199938 / 248.2} = 106.717$

$\lambda_r = 5.7\sqrt{E / F_y} = 5.7\sqrt{199938 / 248.2} = 161.779$

Web is *Compact* in flexure.

The section is Compact in flexure.

Chapter E

Buckling about Minor Axis (⊥ to z-z Axis)

Check slenderness ratio:

$$\frac{K_z L}{r_z} = \frac{1.0 \cdot 4000}{40.89} = 97.953$$

Check limit:

$$4.71\sqrt{\frac{E}{F_y}} = 4.71 \cdot \sqrt{\frac{199938.0}{248.2}} = 133.73$$

Calculate the elastic critical buckling stress F_e:

$$Fe = \frac{\pi^2 E}{\left(\frac{K_z L}{r_z}\right)^2} = \frac{\pi^2 \cdot 199938}{\left(\frac{1.0 \cdot 4000}{40.89}\right)^2} = 205.665 \text{ MPa}$$

Calculate flexural buckling stress $F_{cr,z}$:

Because $\frac{K_z L}{r_z} \leq 4.71\sqrt{\frac{E}{F_y}}$

$$F_{cr,z} = \left[0.658^{\frac{F_y}{F_e}}\right] \cdot F_y = \left[0.658^{\frac{248.2}{205.665}}\right] \cdot 248.2 = 149.773 \text{ MPa}$$

Nominal compressive strength $P_{n,z}$:

$$P_{n,z} = F_{cr,z} \cdot Ag = 149.773 \cdot 4567.7 = 684\,120 \text{ N} = 684.12 \text{ kN}$$

Design compressive strength

$$\phi_c \cdot P_{n,z} = 0.9 \cdot 684.12 = 615.71 \text{ kN}$$

Design ratio

$$\eta_z = \frac{P_r}{\phi_c \cdot P_{n,z}} = \frac{300}{0.9 \cdot 684.12} = 0.487$$ **- O.K, Decisive**

Chapter E

Buckling about Major Axis (⊥ to y-y Axis)

Check slenderness ratio:

$$\frac{K_y L}{r_y} = \frac{1.0 \cdot 4000}{86.87} = 46.05$$

Check limit:

$$4.71\sqrt{\frac{E}{F_y}} = 4.71 \cdot \sqrt{\frac{199938.0}{248.2}} = 133.73$$

Calculate the elastic critical buckling stress F_e:

$$Fe = \frac{\pi^2 E}{\left(\frac{K_y L}{r_y}\right)^2} = \frac{\pi^2 \cdot 199938}{\left(\frac{1.0 \cdot 4000}{86.87}\right)^2} = 930.55\,\text{MPa}$$

Calculate flexural buckling stress $F_{cr,y}$:

Because $\frac{K_y L}{r_y} \leq 4.71\sqrt{\frac{E}{F_y}}$

$$F_{cr,y} = \left[0.658^{\frac{F_y}{F_e}}\right] \cdot F_y = \left[0.658^{\frac{248.2}{930.55}}\right] \cdot 248.2 = 221.981\,\text{MPa}$$

Nominal compressive strength $P_{n,y}$:

$$P_{n,y} = F_{cr,y} \cdot Ag = 221.981 \cdot 4567.7 = 1\,013\,950\ \text{N} = 1013.95\ \text{kN}$$

Design compressive strength

$$\phi_c \cdot P_{n,y} = 0.9 \cdot 1013.95 = 912.56\ \text{kN}$$

Design ratio

$$\eta_y = \frac{P_r}{\phi_c \cdot P_{n,y}} = \frac{300}{0.9 \cdot 1013.95} = 0.328$$ **- O.K**

Chapter F

I-Shaped Member Bent about Major Axis

Note: The nominal flexural strength $M_{n,y}$, shall be the lower value obtained according to the *limit states* of *yielding* (*plastic moment*) and *lateral-torsional buckling*.

1.Yielding

Calculate the nominal flexural strength $M_{n,y}$:

$$M_{n,y} = M_{p,y} = F_y Z_y = 248.2 \cdot 380180 = 94\,360\,676\ \text{Nmm} = 94.36\ \text{kNm}$$

Design flexural strength

$$\phi_b \cdot M_{n,y} = 0.9 \cdot 94.36 = 84.92\ \text{kNm}$$

Design ratio

$$\eta_{b,y} = \frac{M_{r,y}}{\phi_b \cdot M_{n,y}} = \frac{10}{84.92} = 0.117$$ **- O.K**

2.Lateral-Torsional Buckling

Note: The calculation of lateral-torsional buckling modification factor C_b is based on formula F1-1.

Calculate C_b:

$$C_b = \frac{12.5 M_{max}}{2.5 M_{max} + 3M_A + 4M_B + 3M_C} R_m \le 3.0$$

$$C_b = \frac{12.5 \cdot 10}{2.5 \cdot 10 + 3 \cdot 7.5 + 4 \cdot 10 + 3 \cdot 7.5} \cdot 1.0 \le 3.0$$

$$C_b = 1.136 \le 3.0$$

Limiting lengths L_p and L_r:

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} = 1.76 \cdot 40.89 \cdot \sqrt{\frac{199938.0}{248.2}} = 2039.9\ \text{mm}$$

$$L_r = 1.95 r_{ts} \frac{E}{0.7F_y} \sqrt{\frac{Jc}{S_y h_0}} \sqrt{1.0 + \sqrt{1.0 + 6.76\left(\frac{0.7F_y}{E}\frac{S_y h_0}{Jc}\right)^2}} =$$

$$= 1.95 \sqrt{\frac{\sqrt{I_z.C_w}}{S_y}} \frac{E}{0.7F_y} \sqrt{\frac{J \cdot 1.0}{S_y h_0}} \sqrt{1.0 + \sqrt{1.0 + 6.76\left(\frac{0.7F_y}{E}\frac{S_y h_0}{J \cdot 1.0}\right)^2}} =$$

$$= 1.95 \sqrt{\frac{\sqrt{7617000 \cdot 69550.8 \cdot 10^6}}{342490}} \frac{199938}{0.7 \cdot 248.2} \sqrt{\frac{145700}{342490 \cdot 191.2}} \cdot$$

$$\cdot \sqrt{1.0 + \sqrt{1.0 + 6.76 \cdot \left(\frac{0.7 \cdot 248.2}{199938}\frac{342490 \cdot 191.2}{145700}\right)^2}} = 7597.9\ \text{mm}$$

Check limit:

$L_p < L_b \le Lr$ **- formula (F2-2)**

Calculate the nominal flexural strength $M_{n,y}$:

$$M_{n,y} = C_b \left[M_{p,y} - \left(M_{p,y} - 0.7F_y S_y\right)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \le M_{p,y}$$

$$M_{n,y} = 1.136 \cdot \left[94.36 \cdot 10^6 - \left(94.36 \cdot 10^6 - 0.7 \cdot 248.2 \cdot 342490\right)\left(\frac{4000 - 2039.9}{7597.9 - 2039.9}\right)\right] \le 94.36 \cdot 10^6\ \text{Nmm}$$

$$M_{n,y} = 93.26 \cdot 10^6\ \text{Nmm} = 93.26\ \text{kNm}$$

Design flexural strength

$$\phi_b \cdot M_{n,y} = 0.9 \cdot 93.26 = 83.93\ \text{kNm}$$

Design ratio

$\eta_{b,y} = \frac{M_{r,y}}{\phi_b \cdot M_{n,y}} = \frac{10}{83.93} = 0.119$ **- O.K, Decisive**

I-Shaped Member Bent about Minor Axis

Note: The nominal flexural strength $M_{n,z}$, shall be the lower value obtained according to the *limit states* of *yielding* (*plastic moment*) and *flange local buckling*.

1.Yielding

Calculate the nominal flexural strength $M_{n,z}$:

$$M_{n,z} = M_{p,z} = F_y Z_z \leq 1.6 \cdot F_y S_z$$

$$F_y Z_z = 248.2 \cdot 140440 = 34\,857\,208 \text{ Nmm} = 34.86 \text{ kNm}$$

$$1.6 \cdot F_y S_z = 1.6 \cdot 248.2 \cdot 92260 = 36.64 \text{ kNm}$$

$$M_{p,z} = 34.86 \text{ kNm}$$

Design flexural strength

$$\phi_b \cdot M_{n,z} = 0.9 \cdot 34.86 = 31.37 \text{ kNm}$$

Design ratio

$$\eta_{b,z} = \frac{M_{r,z}}{\phi_b \cdot M_{n,z}} = \frac{7.5}{31.37} = 0.239$$ **- O.K**

2.Flange Local Buckling

For section with compact flanges, the limit state of yielding shall apply.

Chapter G

Shear in the Major axis

Note: The nominal shear strength $V_{n,y}$ of unstiffened or stiffened webs is calculated according to the limit states of shear yielding and shear buckling

Calculate the nominal shear strength $V_{n,y}$:

$$V_{n,y} = 0.6 \cdot F_y C_v A_{w,y}$$

$$k_v = 1.2$$

Check limit:

$$b / t_f \leq 1.10 \sqrt{k_v \cdot E / F_y}$$

$$8.119 \leq 34.2$$

if true $C_v = 1.0$

Shear area $A_{w,y}$:

$$A_{w,y} = 2 \cdot b_f t_f = 2 \cdot 164.97 \cdot 10.16 = 3352.25 \text{ mm}^2$$

after

$$V_{n,y} = 0.6 \cdot 248.2 \cdot 1.0 \cdot 3352.25 = 499\,220 \text{ N} = 499.22 \text{ kN}$$

Design shear strength

$$\phi_v \cdot V_{n,y} = 1.0 \cdot 499.22 = 499.22 \text{ kN/m}$$

Design ratio

$$\eta_{v,y} = \frac{V_{r,y}}{\phi_b \cdot V_{n,y}} = \frac{3.75}{499.22} = 0.007$$ **- O.K**

Chapter H

H1. Interaction of Flexure and Compression in Doubly Symmetric Members

Check limit:

$$\frac{P_r}{P_c} = \frac{P_r}{\phi_c \cdot P_{n,y}} \geq 0.2$$

$0.487 \geq 0.2$ **- true, then formula (H1-1a)**

Interaction formula:

$$\frac{P_r}{P_c} + \frac{8}{9}\left(\frac{M_{r,y}}{M_{c,y}} + \frac{M_{r,z}}{M_{c,z}}\right) \leq 1.0$$

$$\frac{P_r}{\phi_c \cdot P_{n,y}} + \frac{8}{9}\left(\frac{M_{r,y}}{\phi_b \cdot M_{n,y}} + \frac{M_{r,z}}{\phi_b \cdot M_{n,z}}\right) \leq 1.0$$

$$\frac{300}{0.9 \cdot 684.12} + \frac{8}{9}\left(\frac{10}{0.9 \cdot 93.26} + \frac{7.5}{0.9 \cdot 34.86}\right) \leq 1.0$$

$$0.487 + \frac{8}{9}(0.119 + 0.239) \leq 1.0$$

$0.805 \leq 1.0$ **- O.K**

A Literature

[1] Specification for Structural Steel Buildings ANSI/AISC 360-05, U.S. Standard, March 9, 2005

[2] Commentary on the Specification for Structural Steel Buildings, March 9, 2005

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

B Index