Version
June 2013

Add-on Module

STEEL AISC

**Allowable Stress Design (ASD),
Load and Resistance Factor Design (LRFD),
Serviceability Limit State Design acc. to
ANSI/AISC 360-05 and ANSI/AISC 360-10**

Program Description

Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 9673 9203-0
Fax: +49 9673 9203-51
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 and ANSI/AISC 360-10) 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 DLUBAL ENGINEERING SOFTWARE company 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, load combinations and result combinations can be arranged individually to cover the various design situations.

If required, you can use the add-on module to optimize cross-sections and export the modified cross-sections to RSTAB. Using the design cases, you can design separate structural components in complex structures or analyze variants.

STEEL AISC is an add-on module integrated in RSTAB. For this reason, the design relevant input data is already preset when you have started the module. Subsequent to the design, you can use the graphical RSTAB user interface to evaluate the results. Finally, the design process can be documented in the global printout report, from determination of internal forces to design.

We hope you will enjoy working with STEEL AISC.

Your DLUBAL Team

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
Ing. Martin Budáč
Mgr. Petr Oulehle
Dipl.-Ing. Georg Dlubal
Dr.-Ing. Jaroslav Lain
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
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) Walter Rustler
M. Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Robert Vogl

Manuals, Documentation and Translations

Ing. Ph.D. Peter Chromiak
Dipl.-Ing. (FH) Robert Vogl
Ing. Ladislav Kábrt
Dipl.-Ing. Frank Faulstich
Dipl.-Ü. Gundel Pietzcker

1.3 Using the Manual

Topics like installation, graphical user interface, results evaluation, and printout are described in detail in the manual of the main program RSTAB. The present manual focuses on typical features of the add-on module STEEL AISC.

The descriptions in this manual follow the sequence of the module's input and results tables as well as their structure. The text of the manual shows the described **buttons** in square brackets, for example [View mode]. At the same time, they are pictured on the left. **Expressions** appearing in dialog boxes, tables, and menus are set in *italics* to clarify the explanations.

At the end of the manual, you find the index. However, if you don't find what you are looking for, please check our website **www.dlubal.com** where you can go through our comprehensive *FAQ* pages by selecting particular criteria.

1.4 Starting STEEL AISC

RSTAB provides the following options to start the add-on module STEEL AISC.

Menu

To start the program in the RSTAB menu bar, click

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

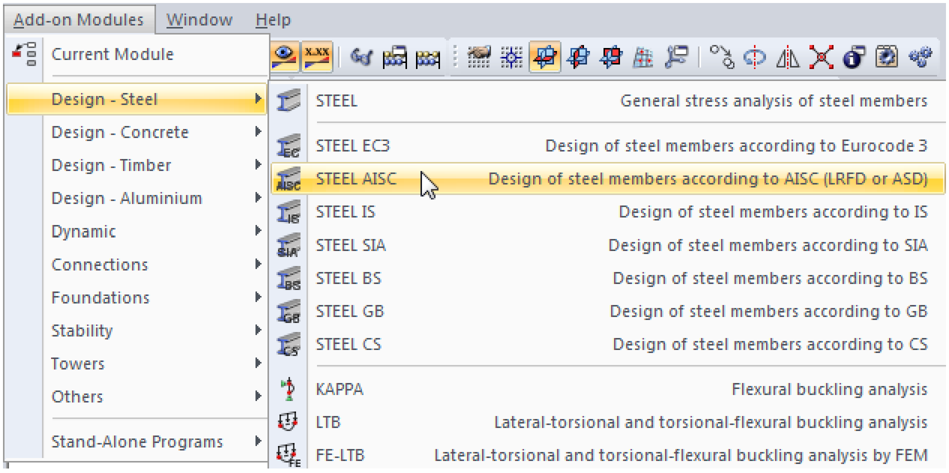


Figure 1.1: Main Menu: *Additional Modules → Design - Steel → STEEL AISC*

Navigator

You can also start the add-on module in the *Data* navigator by clicking

Add-on Modules → STEEL AISC.

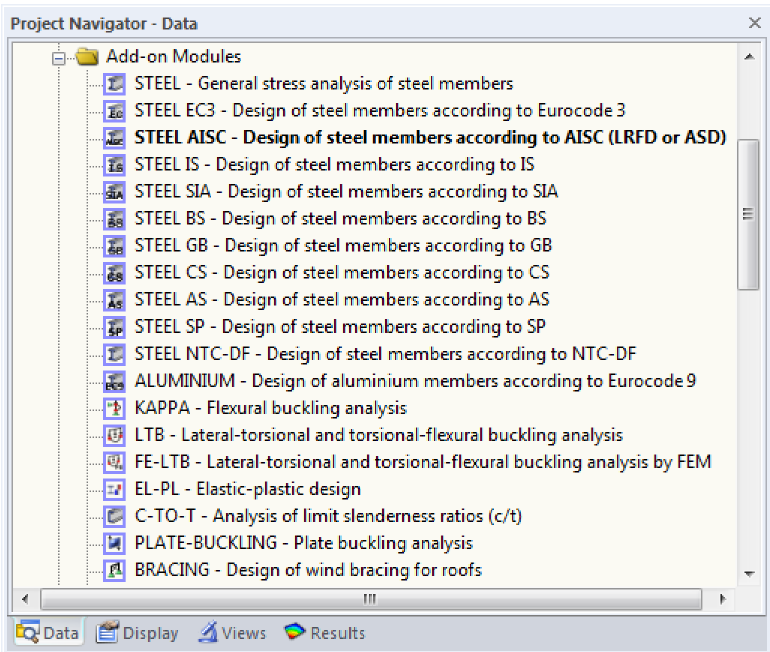
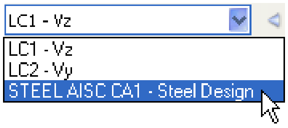


Figure 1.2: Data Navigator: *Add-on Modules → STEEL AISC*

Panel



If results from STEEL AISC are already available in the RSTAB model, you can also open the design module in the panel:

Set the relevant STEEL AISC design case in the load case list of the RSTAB toolbar. Use the button [Show Results] to display the design criterion on the members graphically.

When the results display is activated, the panel is available, too. Now you can click [STEEL AISC] in the panel to open the module.

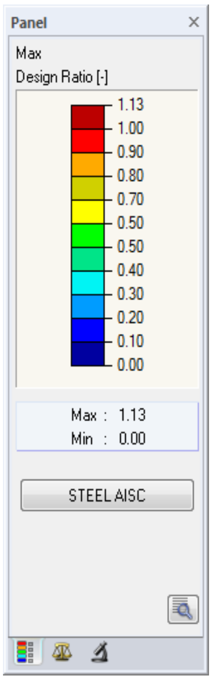


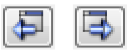
Figure 1.3: Panel button [STEEL AISC]

2. Input Data

When you have started the add-on module, a new window opens. In this window, a Navigator is displayed on the left, managing the tables that can be currently selected. The drop-down list above the navigator contains the design cases (see chapter 7.1, page 49).

The design relevant data is defined in several input tables. When you open STEEL AISC for the first time, the following parameters are imported automatically:

- Members and sets of members
- Load cases, load combinations, result combinations, and super combinations
- Materials
- Cross-sections
- Effective lengths
- Internal forces (in background, if calculated)



To select a table, click the corresponding entry in the navigator. To set the previous or next input table, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] table.

Click [OK] to save the results. Thus you exit STEEL AISC and return to the main program. If you click [Cancel], you exit the module but without saving the data.

2.1 General Data

In table 1.1 *General Data*, you select the members, sets of members, and actions that you want to design. The tabs are managing the load cases, load combinations, result combinations and super combinations for the different designs.

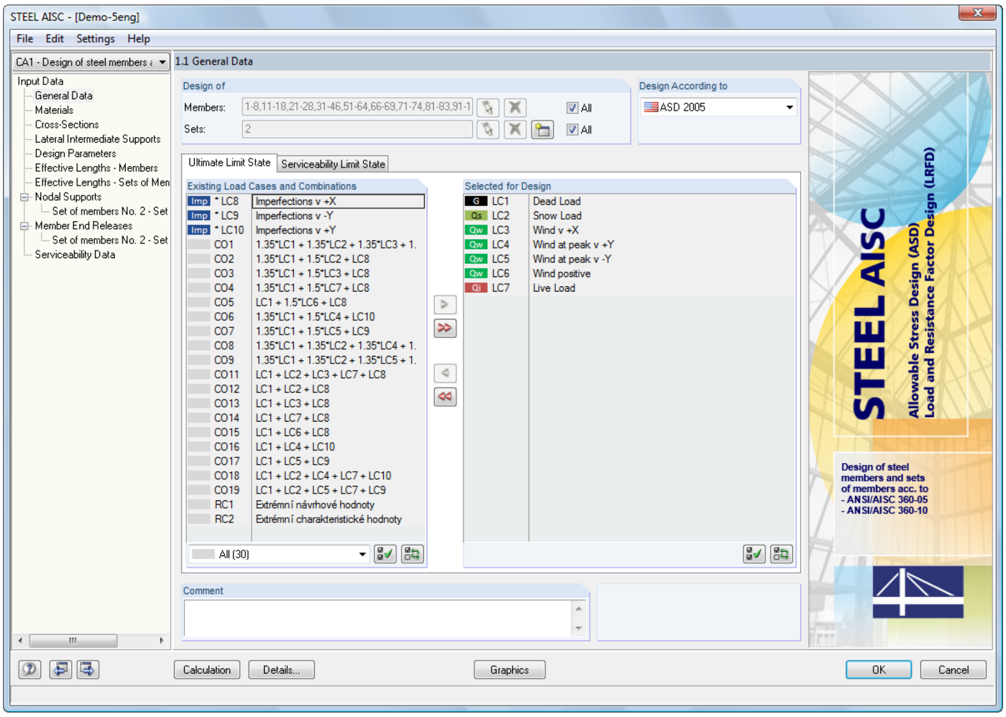


Figure 2.1: Table 1.1 *General Data*

Design of

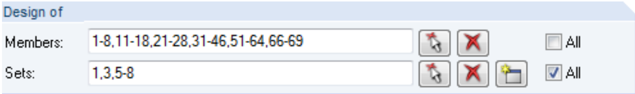


Figure 2.2: Design of members and sets of members

The design can be carried out for *Members* as well as for *Sets of Members*. If you want to design only selected objects, clear the *All* check box: Then you can access the input fields to enter the numbers of the relevant members or sets of members. The list of the numbers preset in the field can be selected by double-clicking and overwritten by entering the data manually. Click [↖] to select the objects graphically in the RSTAB work window.

When you design a set of members, the program determines the extreme values of the designs of all members contained in the set of members and takes into account the boundary conditions of connected members for the stability analysis. The results are shown in the result table 2.3 *Design by Set of Members*, 3.2 *Governing Internal Forces by Set of Members*, and 4.2 *Parts List by Set of Members*.

Click [New] to create a new set of members. The dialog box that you already know from RSTAB appears where you can specify the parameters for a set of members.

Design according to

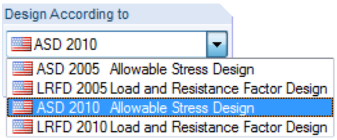


Figure 2.3: Design according to standard

The options of the list box control whether the analysis is carried out according to the provisions of the Allowable Strength Design (ASD) or the Load and Resistance Factor Design (LRFD). There are design methods available for both ANSI/AISC 360-05 and ANSI/AISC 360-10.

2.1.1 Ultimate Limit State

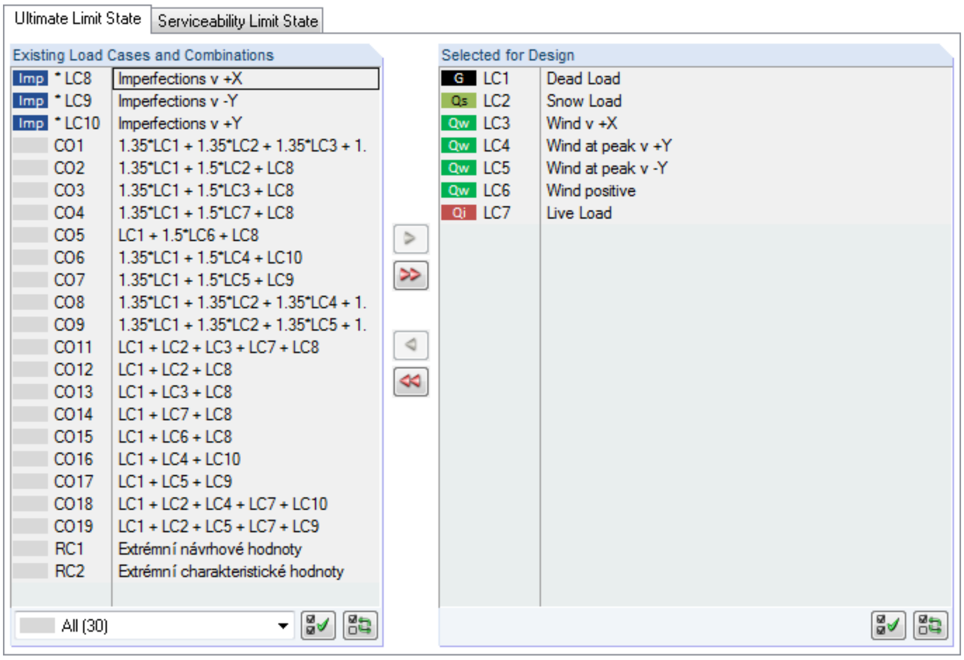


Figure 2.4: Table 1.1 *General Data*, *Ultimate Limit State* tab

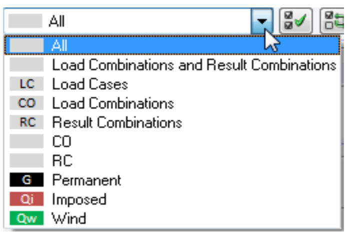
Existing Load Cases and Combinations

In this column, all load cases, load combinations, result combinations and super combinations created in RSTAB are listed.

Click [▶] to transfer selected entries to the list *Selected for Design* on the right side. You can also double-click the items. To transfer the complete list to the right, click [▶▶].

To transfer multiple entries of load cases, click the entries while pressing the [Ctrl] key, as common for Windows applications. Thus, you can transfer several load cases at the same time.

Load cases marked by an asterisk (*), like load case 8 in Figure 2.4, cannot be designed: This happens when the load cases are defined without any load data or the load cases contain only imperfections. When you transfer the load cases, a corresponding warning appears.



At the end of the list, several filter options are available. They will help you assign the entries sorted according to load cases, load combinations, or action categories. The buttons are reserved for the following functions:

	Select all cases in the list.
	Invert selection of load cases.

Table 2.1: Buttons in the tab *Ultimate Limit State*

Selected for Design

The column on the right lists the load cases as well as the load and result combinations selected for design. To remove selected items from the list, click [◀] or double-click the entries. To transfer the entire list to the left, click [◀◀].

Result combination

The design of an enveloping max/min result combination is performed faster than the design of all contained load cases and load combinations. However, the analysis of a result combination has also disadvantages: First, the influence of the contained loads is difficult to discern.

Second, for the determination of the elastic critical moment for lateral-torsional buckling, the envelope of the moment distributions is analyzed, from which the most unfavorable distribution (max or min) is taken. However, this distribution only rarely reflects the moment distribution in the individual load combinations. Thus, in the case of a RC design, more unfavorable values for the elastic critical moment are to be expected, leading to higher ratios.

Result combinations should be selected for design only for dynamic combinations. For "usual" combinations, load combinations are recommended.

2.1.2 Serviceability Limit State

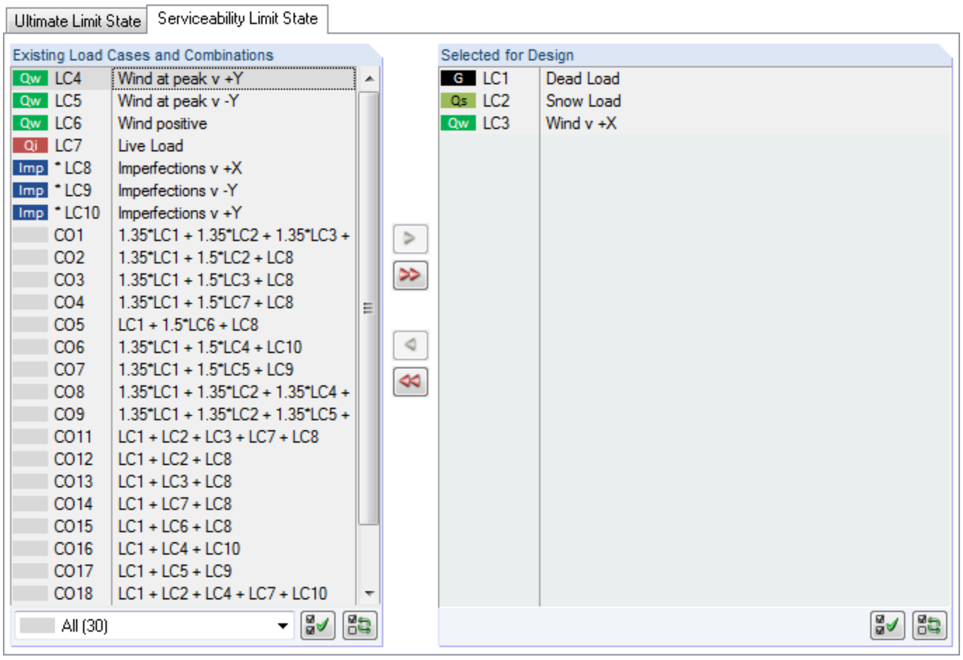


Figure 2.5: Table 1.1 *General Data*, *Serviceability Limit State* tab

Existing Load Cases and Combinations

This section lists all load cases, load combinations, and result combinations created in RSTAB.

Selected for Design

Load cases, load combinations, and result combinations can be added or removed, as described in chapter 2.1.1.

The limit values of the deformations are controlled by the settings in the *Details* dialog box (see Figure 3.1, page 28) which you can call up by clicking the [Details] button.

In the table 1.10 *Serviceability Data*, the reference lengths decisive for the deformation check are managed (see chapter 2.10, page 27).

2.2 Materials

The table consists of two parts. In the upper part, all materials created in RSTAB are listed. In the *Material Properties* section, the properties of the current material, that is, the table row currently selected in the upper section, are displayed.

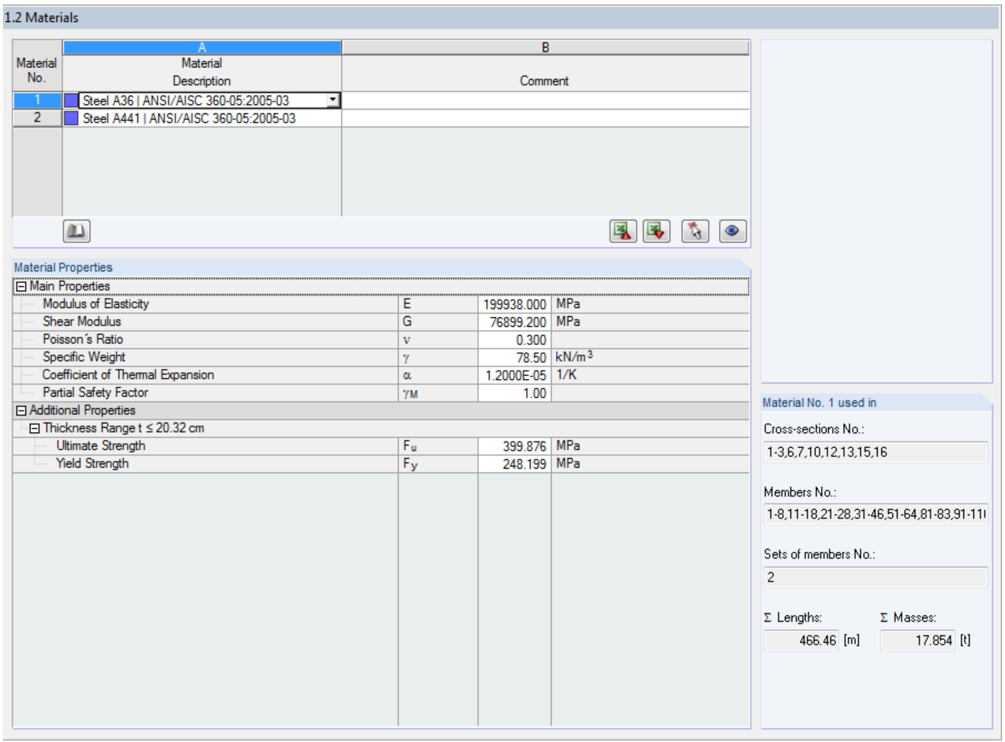


Figure 2.6: Table 1.2 *Materials*

Materials that will not be used in the design are dimmed. Materials that are not allowed are highlighted in red. Modified materials are displayed in blue.

The material properties required for the determination of internal forces are described in chapter 4.2 of the RSTAB manual (*Main Properties*). The material properties required for design are stored in the global material library. The values are preset (*Additional Properties*).

To adjust the units and decimal places of material properties and stresses, select in the module's menu **Settings** → **Units and Decimal Places** (see chapter 7.3, page 53).

Material Description

The materials defined in RSTAB are already preset, but it is always possible to modify them: To select the field, click the material in column A. Then click [▼] or press function key [F7] to open the material list.

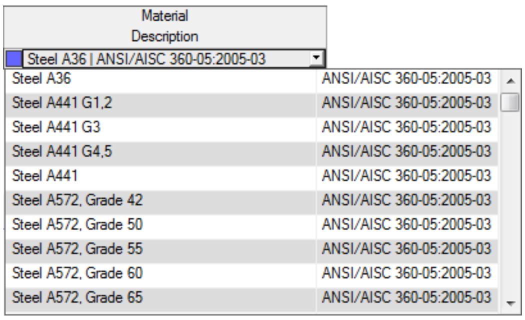


Figure 2.7: List of materials

According to the design concept of the ANSI/AISC 360-05 and ANSI/AISC 360-10 standards, you can select only materials of the "Steel" category.

When you have imported a material, the design relevant *Material Properties* are updated.

If you change the material description manually and the entry is stored in the material library, STEEL AISC will import the material properties, too.

Principally, it is not possible to edit the material properties in the add-on module STEEL AISC.

Material Library

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

Edit → Material Library

or use the button shown on the left.

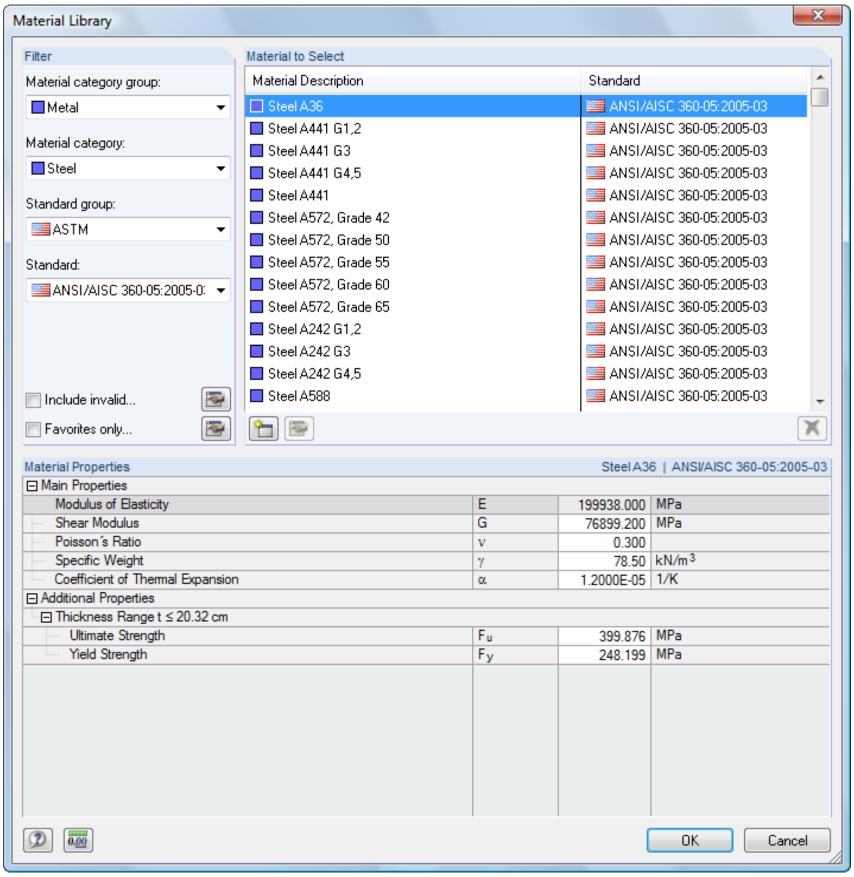


Figure 2.8: Dialog box *Material Library*

In the *Filter* section, *Steel* is preset as material category. Select the material quality that you want to use for the design in the list *Material to Select*. The corresponding properties can be checked in the dialog section below.

Click [OK] or [↵] to transfer the selected material to table 1.2 of the add-on module STEEL AISC.

Chapter 4.2 in the RSTAB manual describes in detail how materials can be filtered, added, or rearranged.

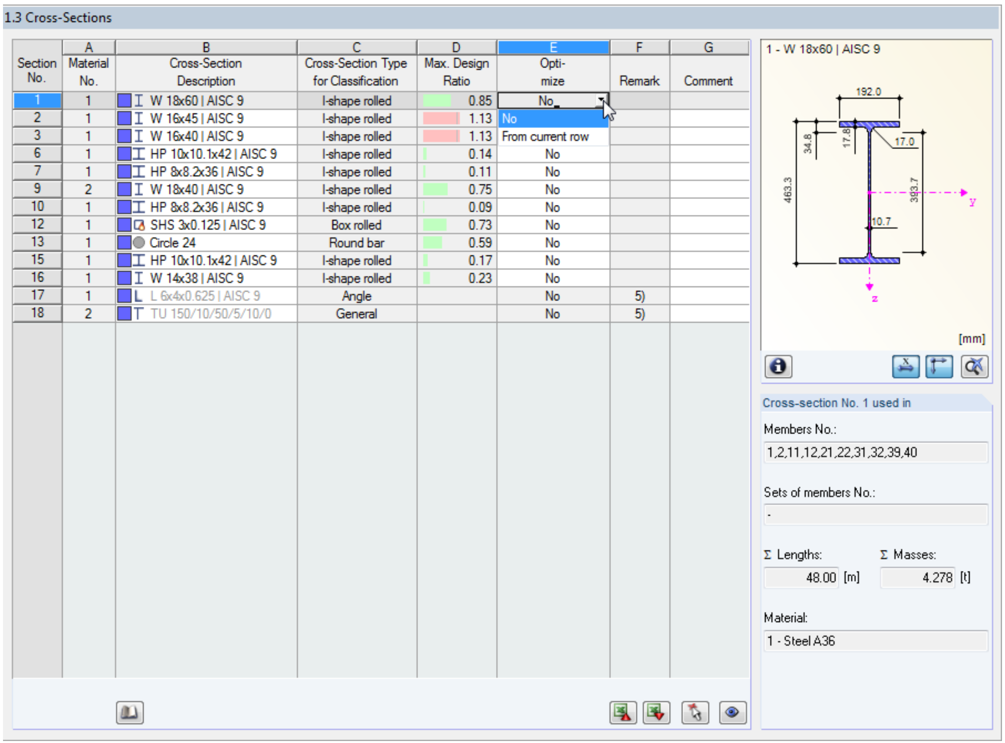
You can also select material categories like *Cast Iron* or *Stainless Steel*. Please check, however, whether these materials are covered by the design concept of the ANSI/AISC 360-05 or 360-10 standards.

2.3 Cross-Sections

This table manages the cross-sections used for design. In addition, the table allows you to specify optimization parameters.

Coordinate System

The sectional coordinate system **yz** of STEEL AISC corresponds to the one of RSTAB (see image in Figure 2.9). 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.


1.3 Cross-Sections

Section No.	A Material No.	B Cross-Section Description	C Cross-Section Type for Classification	D Max. Design Ratio	E Opti-mize	F Remark	G Comment
1	1	W 18x60 | AISC 9	I-shape rolled	0.85	No		
2	1	W 16x45 | AISC 9	I-shape rolled	1.13	No		
3	1	W 16x40 | AISC 9	I-shape rolled	1.13	From current row		
6	1	HP 10x10.1x42 | AISC 9	I-shape rolled	0.14	No		
7	1	HP 8x8.2x36 | AISC 9	I-shape rolled	0.11	No		
9	2	W 18x40 | AISC 9	I-shape rolled	0.75	No		
10	1	HP 8x8.2x36 | AISC 9	I-shape rolled	0.09	No		
12	1	SHS 3x0.125 | AISC 9	Box rolled	0.73	No		
13	1	Circle 24	Round bar	0.59	No		
15	1	HP 10x10.1x42 | AISC 9	I-shape rolled	0.17	No		
16	1	W 14x38 | AISC 9	I-shape rolled	0.23	No		
17	1	L 6x4x0.625 | AISC 9	Angle		No	5)	
18	2	TU 150/10/50/5/10/0	General		No	5)	

Figure 2.9: Table 1.3 *Cross-Sections*

Cross-Section Description

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

If you want to modify a cross-section, click the entry in column B to select this field. Click [Cross-section Library] or [...] in the field or press function key [F7] to open the cross-section table of the current input field (see the following figure).

In this dialog box, you can select a different cross-section or a different cross-section table. To select a different cross-section category, click [Back to cross-section library] to access the general cross-section library.

Chapter 4.3 of the RSTAB manual describes how cross-sections can be selected from the library.

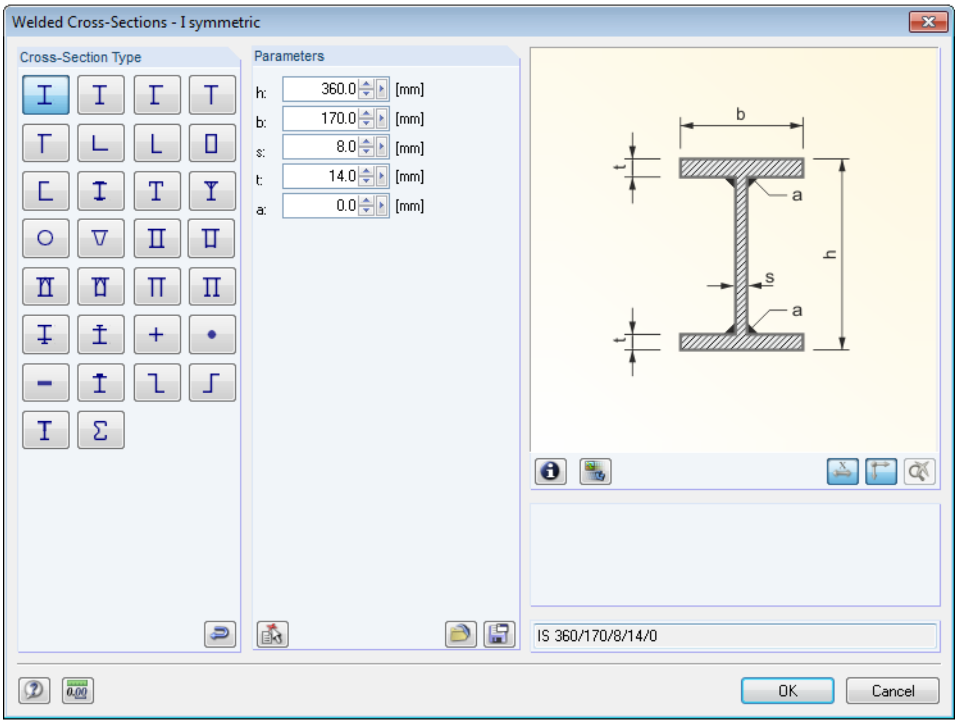
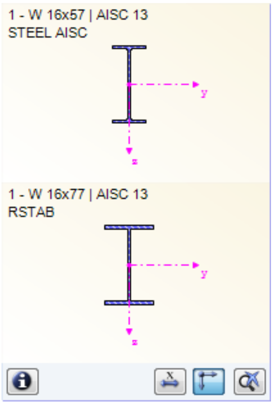


Figure 2.10: IS cross-sections in the cross-section library



The new cross-section description can be entered in the input field directly. If the data base contains an entry, STEEL AISC imports these cross-section parameters, too.

A modified cross-section will be highlighted in blue.

If cross-sections set in STEEL AISC are different from the ones used in RSTAB, both cross-sections are displayed in the graphic in the right part of the table. The designs will be performed with the internal forces from RSTAB for the cross-section selected in STEEL AISC.

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.

Cross-Section Type for Classification

The cross-section type used for the classification is displayed (e.g. I-shape rolled, welded, box, round bar etc.) Cross-sections that are not covered by this table are classified as *General*.

Max. Design Ratio

This table column is displayed only after the calculation. It is a decision support for the optimization. By means of the displayed design ratio and colored relation scales, you can see which cross-sections are little utilized and thus oversized, or overloaded and thus undersized.

Optimize

You can optimize every cross-section from the library: For the RSTAB internal forces, the program searches the cross-section in the same table that comes as close as possible to a user-defined maximum ratio. The maximum ratio can be defined in the *Details* dialog box (see Figure 3.1, page 28).

If you want to optimize a cross-section, open the drop-down list in column D or E and select the desired entry: *From Current Row* or, if available, *From favorites 'Description'*. Recommendations for the cross-section optimization can be found in chapter 7.2 on page 51.

Remark

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

A warning might appear before the calculation: *Incorrect type of cross-section!* This means that there is a cross-section that is not registered in the data base. This may be a user-defined cross-section, or a SHAPE-THIN cross-section that has not been calculated yet. To select an appropriate cross-section for design, click [Library] (see description after Figure 2.9).

Member with tapered cross-section

For tapered members with different cross-sections at the member start and member end, the module displays both cross-section numbers in two tables, in accordance with the definition in RSTAB.

STEEL AISC also designs tapered members, provided that the cross-section at the member's start has the same number of stress points as the cross-section at the member end. The normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the start and the end cross-section of a tapered member have not the same number of stress points, the intermediate values cannot be interpolated. The calculation is neither possible in RSTAB nor in STEEL AISC.

The cross-section's stress points including numbering can also be checked graphically: Select the cross-section in table 1.3 and click [Info]. The dialog box shown in Figure 2.11 appears.

Info About Cross-Section

In the dialog box *Info About Cross-Section*, you can view the cross-section properties, stress points and c/t-parts.

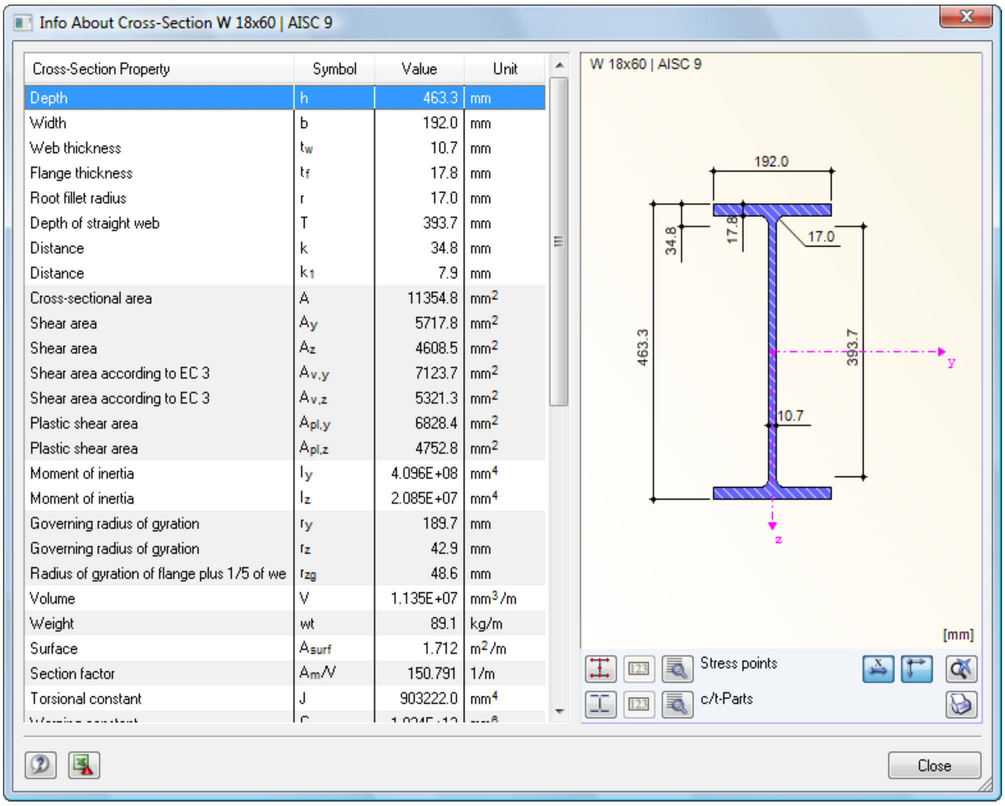


Figure 2.11: Dialog box *Info about Cross-Section*

In the right part of the dialog box, the currently selected cross-section is displayed.

The buttons below the graphic are reserved for the following functions:

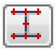
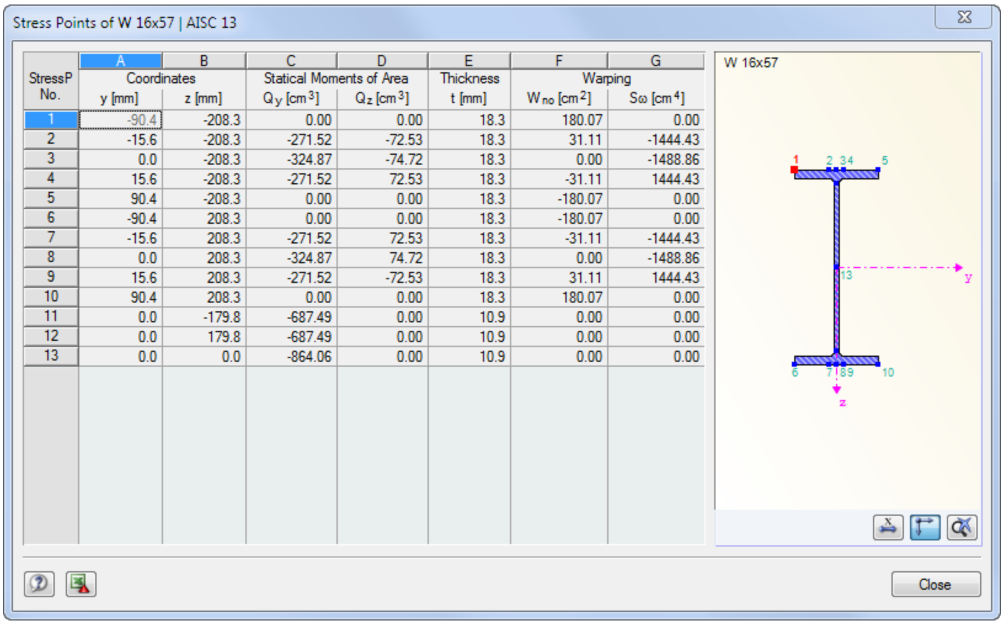
Button	**Function**
	Displays or hides the stress points
	Displays or hides the c/t-parts
	Displays or hides the numbering of stress points or c/t-parts
	Displays or hides the details of the stress points or c/t-parts (see Figure 2.12)
	Displays or hides the dimensions of the cross-section
	Displays or hides the principal axes of the cross-section
	Resets the full view of the cross-section graphic

Table 2.2: Buttons of cross-section graphic

Click [Details] to call up detailed information on stress points (distance to center of gravity, statical moments of area, normalized warping constants etc.) and c/t-parts.



StressP No.	A Coordinates y [mm]	B z [mm]	C Statical Moments of Area Q_y [cm^3]	D Q_z [cm^3]	E Thickness t [mm]	F Warping W_{no} [cm^2]	G S_ω [cm^4]
1	-90.4	-208.3	0.00	0.00	18.3	180.07	0.00
2	-15.6	-208.3	-271.52	-72.53	18.3	31.11	-1444.43
3	0.0	-208.3	-324.87	-74.72	18.3	0.00	-1488.86
4	15.6	-208.3	-271.52	72.53	18.3	-31.11	1444.43
5	90.4	-208.3	0.00	0.00	18.3	-180.07	0.00
6	-90.4	208.3	0.00	0.00	18.3	-180.07	0.00
7	-15.6	208.3	-271.52	72.53	18.3	-31.11	-1444.43
8	0.0	208.3	-324.87	74.72	18.3	0.00	-1488.86
9	15.6	208.3	-271.52	-72.53	18.3	31.11	1444.43
10	90.4	208.3	0.00	0.00	18.3	180.07	0.00
11	0.0	-179.8	-687.49	0.00	10.9	0.00	0.00
12	0.0	179.8	-687.49	0.00	10.9	0.00	0.00
13	0.0	0.0	-864.06	0.00	10.9	0.00	0.00

Figure 2.12: Dialog box *Stress Points of W 16x57*

2.4 Lateral Intermediate Supports

In table 1.4, you can define lateral intermediate supports for members. STEEL AISC always assumes this kind of support to be perpendicular to the cross-section's minor axis z (see (see Figure 2.9). Thus, it is possible to influence the members' effective lengths which are important for the design of column buckling and lateral torsional buckling.

For the calculation, all lateral intermediate supports are considered as forked supports.

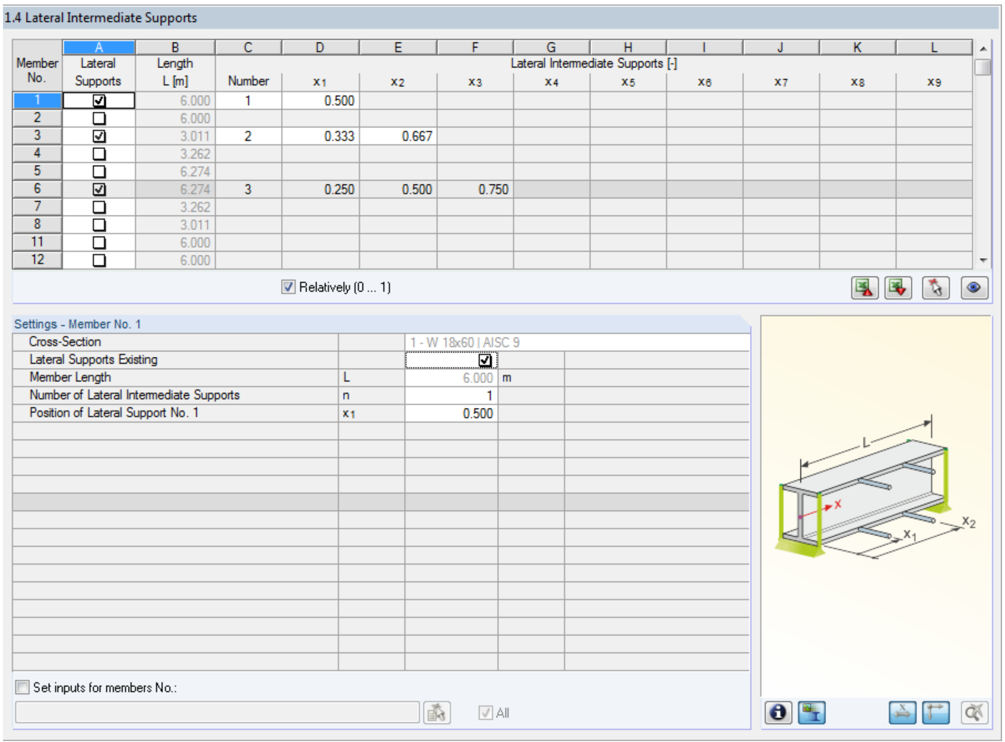


Figure 2.13: Table 1.4 *Lateral Intermediate Supports*

In the upper part of the table, you can assign up to nine lateral supports for each member. The lower table part shows you a summary of the data entered for the member selected above.

Relatively (0 ... 1)

If the check box *Relatively (0 … 1)* is selected, the support points can be defined by relative-input. The positions of the intermediate supports are determined from the member length and the relative distances from the member start. When the check box *Relatively (0 ... 1)* is cleared, you can define the distances manually in the upper table.

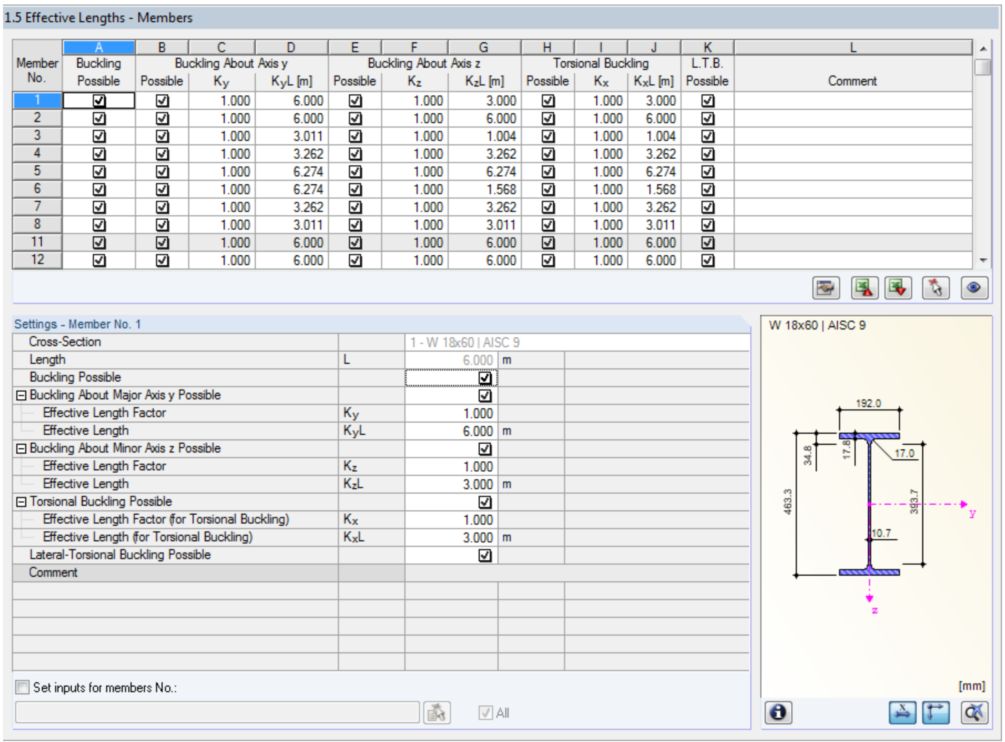
In case of cantilevers, avoid intermediate supports, as such supports divide the member in segments. Thus for cantilevered beams, this would results in segments that are forked supported on one side and thus statically underdetermined (forked support respectively on one end only).

2.5 Effective Lengths - Members

The table 1.5 consists of two parts. 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 listed for every member. The effective lengths defined in RSTAB are already preset. In the section *Settings*, further information is shown about the member whose row is selected in the upper section.

Click the button [↖] to select a member graphically and to show its row.

Changes can be made in the table as well as in the *Settings* tree.



1.5 Effective Lengths - Members

	A	B	C	D	E	F	G	H	I	J	K	L
Member No.	Buckling Possible	Buckling About Axis y Possible	K_y	K_yL [m]	Buckling About Axis z Possible	K_z	K_zL [m]	Torsional Buckling Possible	K_x	K_xL [m]	L.T.B. Possible	Comment
1	☑	☑	1.000	6.000	☑	1.000	3.000	☑	1.000	3.000	☑	
2	☑	☑	1.000	6.000	☑	1.000	6.000	☑	1.000	6.000	☑	
3	☑	☑	1.000	3.011	☑	1.000	1.004	☑	1.000	1.004	☑	
4	☑	☑	1.000	3.262	☑	1.000	3.262	☑	1.000	3.262	☑	
5	☑	☑	1.000	6.274	☑	1.000	6.274	☑	1.000	6.274	☑	
6	☑	☑	1.000	6.274	☑	1.000	1.568	☑	1.000	1.568	☑	
7	☑	☑	1.000	3.262	☑	1.000	3.262	☑	1.000	3.262	☑	
8	☑	☑	1.000	3.011	☑	1.000	3.011	☑	1.000	3.011	☑	
11	☑	☑	1.000	6.000	☑	1.000	6.000	☑	1.000	6.000	☑	
12	☑	☑	1.000	6.000	☑	1.000	6.000	☑	1.000	6.000	☑	

Settings - Member No. 1

Cross-Section		1 - W 18x60 | AISC 9	
Length	L	6.000	m
Buckling Possible		☑	
⊟ Buckling About Major Axis y Possible		☑	
Effective Length Factor	K_y	1.000	
Effective Length	K_yL	6.000	m
⊟ Buckling About Minor Axis z Possible		☑	
Effective Length Factor	K_z	1.000	
Effective Length	K_zL	3.000	m
⊟ Torsional Buckling Possible		☑	
Effective Length Factor (for Torsional Buckling)	K_x	1.000	
Effective Length (for Torsional Buckling)	K_xL	3.000	m
Lateral-Torsional Buckling Possible		☑	
Comment			

Figure 2.14: 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 aligned automatically with the entries of table 1.4 *Lateral Intermediate Supports*. If lateral intermediate supports are dividing the member into member segments of different lengths, the program displays no value in the table columns G and J of table 1.5.

The effective lengths can be entered manually in the table and in the *Settings* tree, or defined graphically by clicking the button [...] in the work window. This button is enabled when you click in the input field.

The *Settings* tree manages the following parameters:

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

It is also possible to modify the *Effective Length Factors* in the relevant directions and decide whether the buckling design is to be executed. When a buckling length factor is modified, the equivalent member length will be adjusted automatically, and vice versa.

The buckling length of a member can also be defined in a dialog box that can be accessed by clicking the button shown on the left. The button is located below the table.

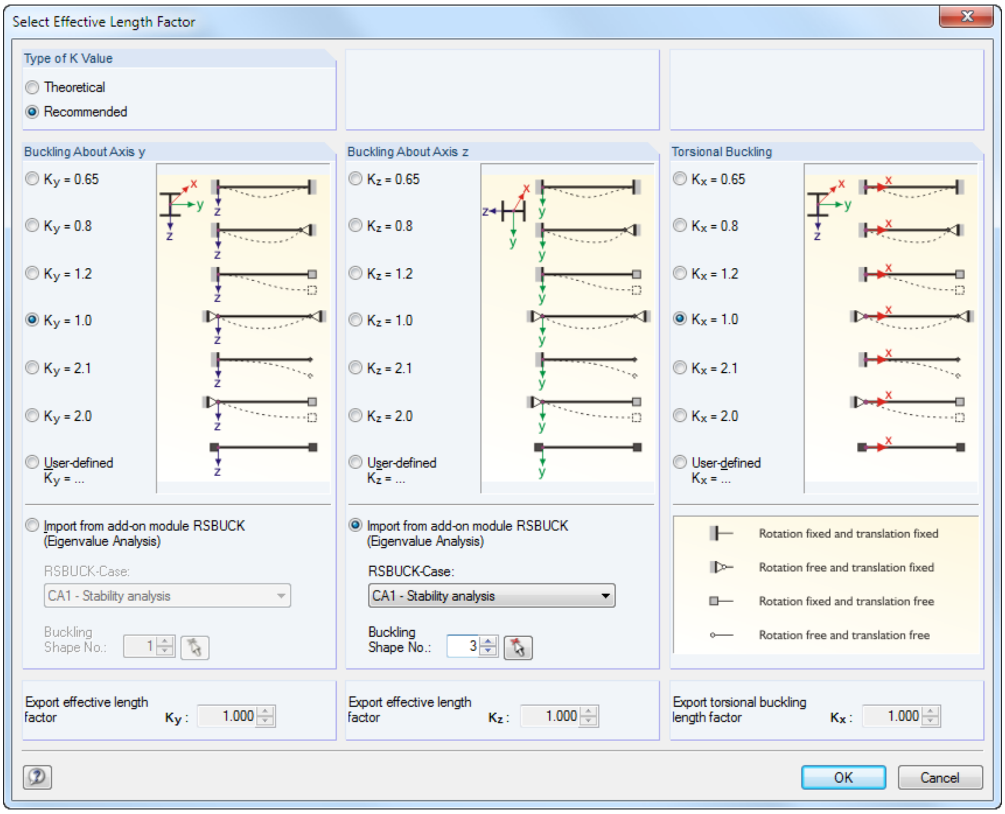


Figure 2.15: Dialog box *Select Effective Length Factor*

In this dialog box, the *Theoretical* or *Recommended* values of the K factor can be defined that are to be assigned to the selected member or group of members. The theoretical and the recommended values are described in [3] on page 240. Generally, it is possible to select predefined K factors or to enter *User-defined* values.

If a RSBUCK case calculated according to the eigenvalue analysis is already available, you can also select a *Buckling Shape* to determine the factor.

Buckling Possible

A stability analysis for buckling and lateral buckling requires the ability of members to absorb compressive forces. Therefore, members for which such absorption is not possible because of the member type (for example tension members, elastic foundations, rigid connections) are excluded from design in the first place. The corresponding rows appear dimmed and a note is indicated in the *Comment* column.

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.

Buckling About Axis y resp. Axis z

With the check boxes in the *Possible* table columns, you decide whether a member has a risk of buckling about the axis y and/or z. The axis y represents the "major" principal member axis, the axis z the "minor" principal member axis. The effective length factors K_y and K_z for buckling about the major or the minor axis can be selected freely The effective length factors K_y and K_z can be freely chosen for the buckling around the major and minor axes.

The position of the member axes can be checked in the cross-section graphic in table 1.3 *Cross-Sections* (see Figure 2.9, page 14). To access the RSTAB work window, click [View mode]. To show the local member axes in the RSTAB work window, you can use the member's context menu or the *Display* navigator.

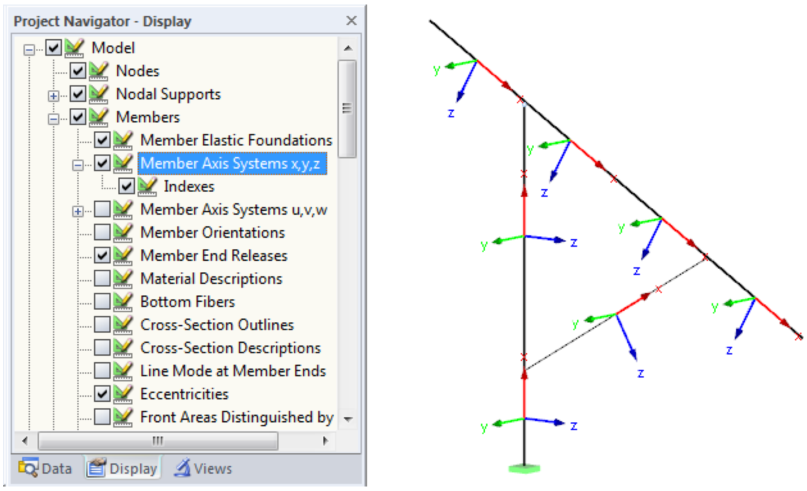


Figure 2.16: Displaying the member axes in the *Display* navigator of RSTAB

If buckling is possible about one or even both member axes, you can enter the buckling length coefficients as well as the buckling lengths in the columns D and E respectively F and G. The same is possible in the *Settings* tree.

To specify the buckling lengths in the work window graphically, click [...]. This button becomes available when you click in a *KL* input field.

If you define the effective length factor *K*, the program determines the effective length *KL* by multiplying the member length *L* by this buckling length coefficient.

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 center line of a member.

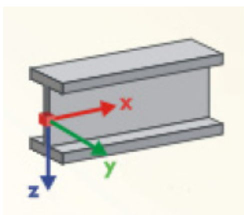


Figure 2.17: Member Axes

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 table column, you can enter user-defined comments for each member to describe, for example, the effective member lengths.

Below the *Settings* table you find the check box *Set inputs for members No.* If selected, the settings entered afterwards will be applied to the selected or even to *All* members. Members can be selected by typing the member number or by selecting them graphically using the [↖] button. This option is useful when you want to assign the same boundary conditions to several members. Please note that settings that have been already defined cannot be changed subsequently with this function.

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*.

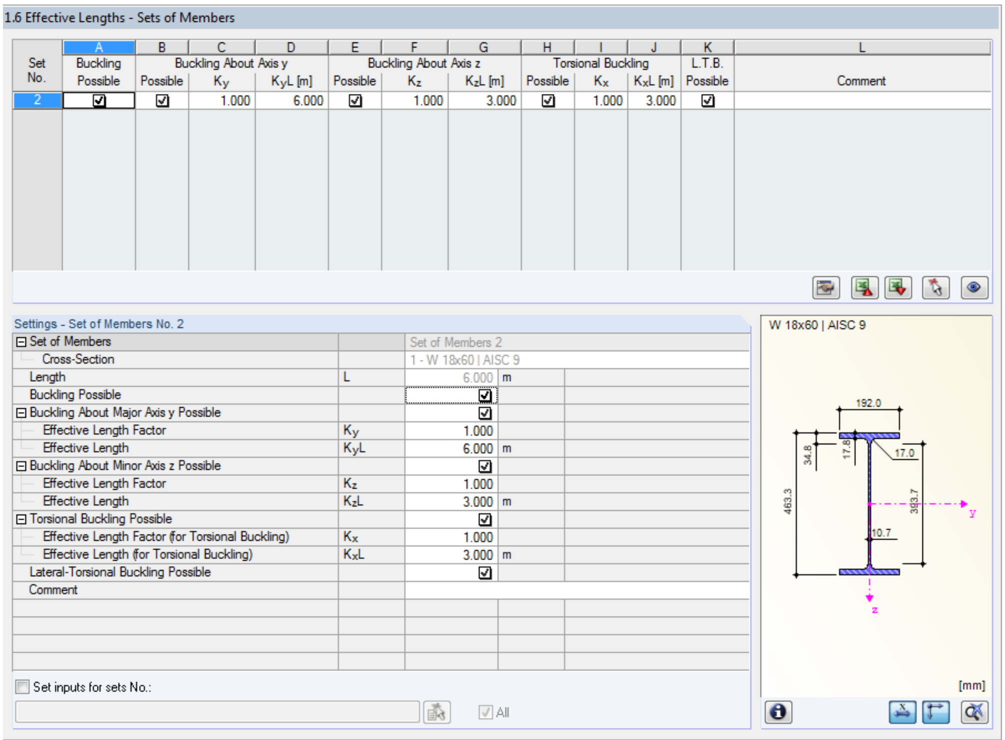


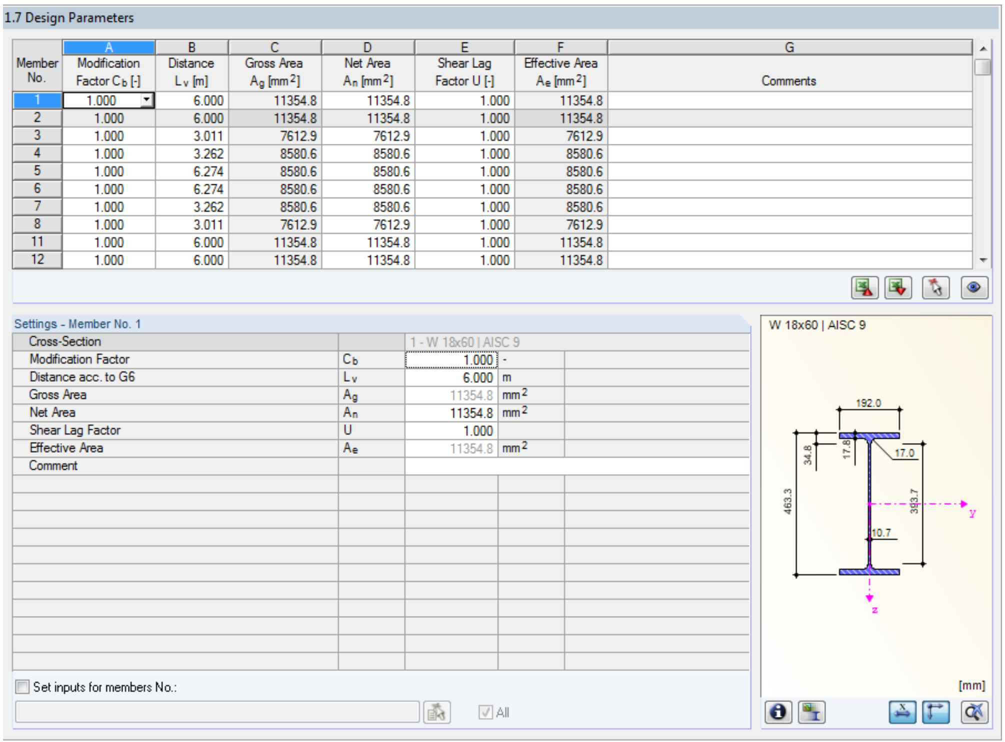
Figure 2.18: Table 1.6 *Effective Lengths - Set of Members*

This table's concept is similar to the one in the previous table 1.5 *Effective Lengths - Members*. In this table, you can enter the effective lengths for the buckling about the two principal axes of the set of members as described in chapter 2.5.

There are differences, however, as far as the parameters for torsional and lateral-torsional buckling are concerned. Those 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

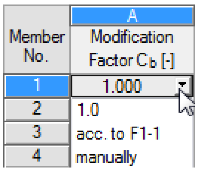
In this table, several parameters can be defined that are necessary for design.



Member No.	A Modification Factor Cb [-]	B Distance Lv [m]	C Gross Area Ag [mm2]	D Net Area An [mm2]	E Shear Lag Factor U [-]	F Effective Area Ae [mm2]	G Comments
1	1.000	6.000	11354.8	11354.8	1.000	11354.8	
2	1.000	6.000	11354.8	11354.8	1.000	11354.8	
3	1.000	3.011	7612.9	7612.9	1.000	7612.9	
4	1.000	3.262	8580.6	8580.6	1.000	8580.6	
5	1.000	6.274	8580.6	8580.6	1.000	8580.6	
6	1.000	6.274	8580.6	8580.6	1.000	8580.6	
7	1.000	3.262	8580.6	8580.6	1.000	8580.6	
8	1.000	3.011	7612.9	7612.9	1.000	7612.9	
11	1.000	6.000	11354.8	11354.8	1.000	11354.8	
12	1.000	6.000	11354.8	11354.8	1.000	11354.8	

Figure 2.19: 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], [2].

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], [2]. 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], [2].

Effective Area A_e

Column F lists the effective areas of the members which are determined according to Equation D3-1 [1], [2] from the net areas and the shear lag factors.

2.8 Nodal Supports - Sets of Members

This table is only available if you have selected one or more sets of member in table 1.1 *General Data*.

The stability design of sets of members is carried out according to Chapter C of the ANSI/AISC Specification. 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. For this analysis method, it is necessary to know the amplification factor α_{cr}, of the entire set of members to obtain the critical moment. To determine this factor, a planar framework is created with four degrees of freedom for each node which is to be defined in table 1.8. This table refers to the current set of members (selected in the add-on module's navigator on the left). The calculation of α_{cr}, the bifurcation factor, also depends on the setting in the *Details* dialog box (see chapter 3.1, page 28).

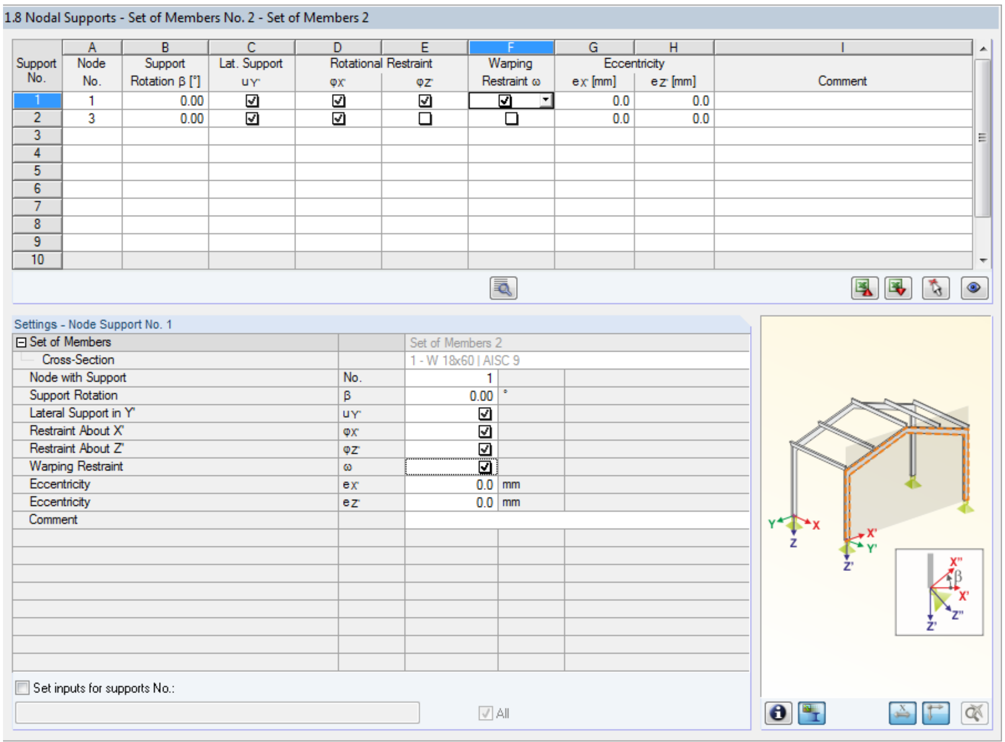


Figure 2.20: Table 1.8 *Nodal Supports - Set of Members*

The orientation of axes in the set of members is important for the definition of nodal supports. The program checks the position of the nodes and internally defines, according to Figure 2.21 to Figure 2.24, the axes of the nodal supports for table 1.8.

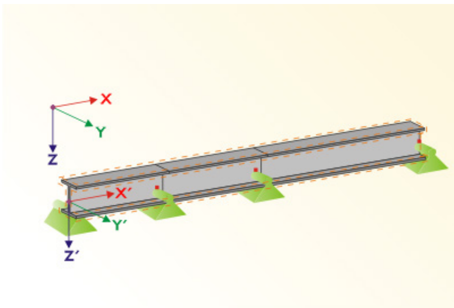


Figure 2.21: Auxiliary coordinate system for nodal supports - straight set of members

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.

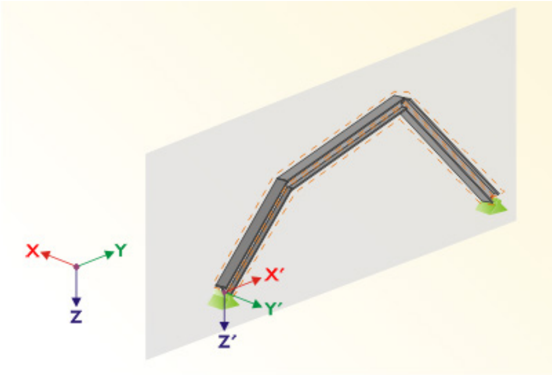


Figure 2.22: Auxiliary coordinate system for nodal supports - set of members in vertical plane

If members of a set of members are not lying in a straight line, they must at least lie in the same plane. In Figure 2.22, they are lying in a vertical plane. In this case, the axis X' is horizontal and aligned in direction of the plane. The axis Y' is horizontal as well and defined perpendicular to the axis X'. The axis Z' is directed perpendicularly downwards.

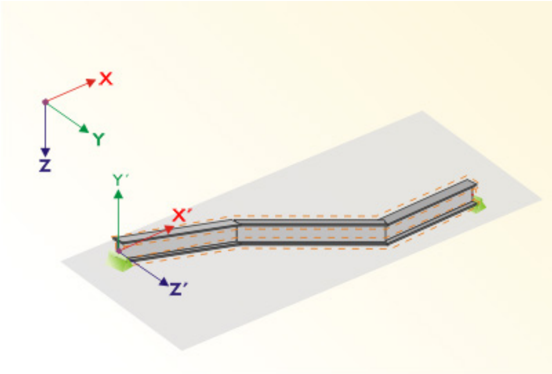


Figure 2.23: Auxiliary coordinate system for nodal supports - set of members in horizontal plane

If the members of a buckled set of members are lying in a horizontal plane, the axis X' is defined parallel to the X-axis of the global coordinate system. Thus, the axis Y' is set in opposite direction to the global Z-axis and the axis Z' is directed parallel to the global Y-axis.

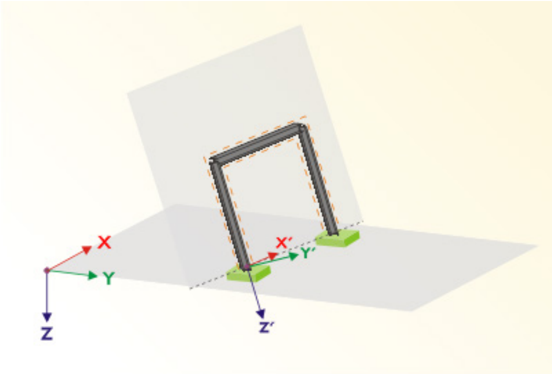


Figure 2.24: Auxiliary coordinate system for nodal supports - set of members in inclined plane

Figure 2.24 shows the general case of a buckled set of members: The members are not lying in one straight line but in an inclined plane. The definition of the axis X' arises out of the intersection line of the inclined plane with the horizontal plane. Thus, the axis Y' is defined right-angled to the axis X' and directed perpendicular to the inclined plane. The axis Z' is defined perpendicular to the axis X' and Y'.

2.9 Member End Releases - Sets of Members

This table is displayed only if you have selected at least one set of members for the design in table 1.1 *General Data*. Here, you can define releases for members and sets of members that, due to structural reasons, do not pass the locked degrees of freedom specified in table 1.8 as internal forces. There is also the possibility to exactly define on which side the release is to act or to place a release at both sides.

This table refers to the current set of members (selected in the add-on module's navigator on the left).

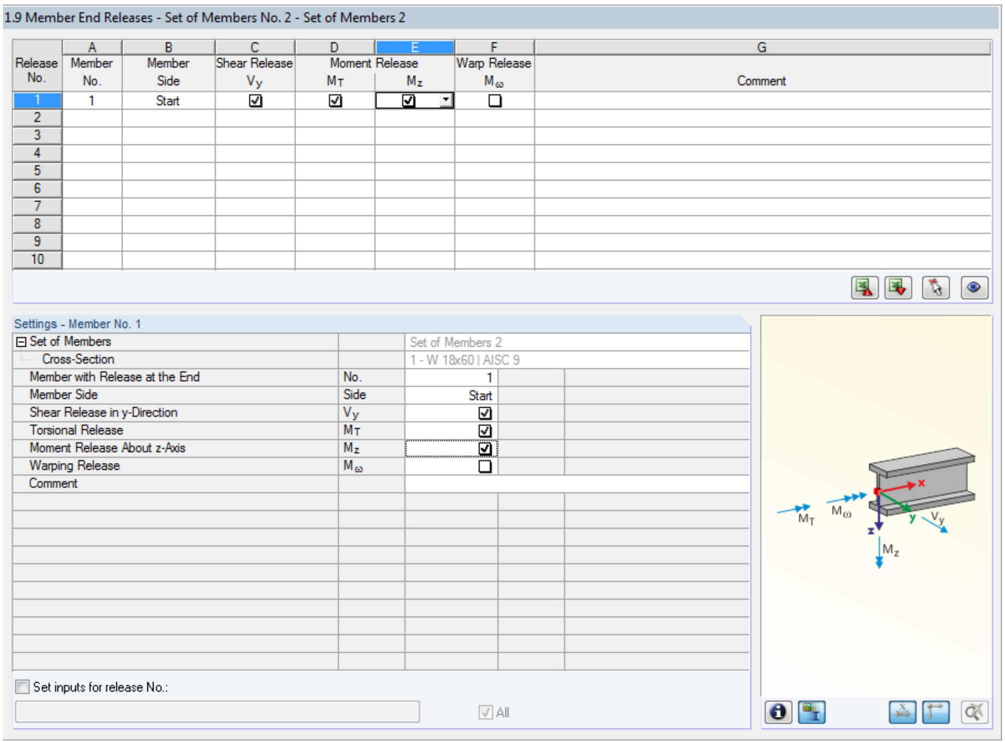
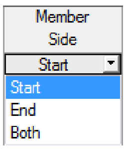


Figure 2.25: Table 1.9 *Member Releases - Set of Members*



In table column B, you define the *Member Side* to which the release should be assigned. You can also connect the releases to both member sides.

In the columns C to F, you can define releases or spring constants to align the set of members model with the support conditions in table 1.8.

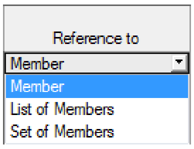
2.10 Serviceability Data

The last input table controls several settings for the serviceability design. It is only available if you have set relevant entries in the *Serviceability Limit State* tab of table 1.1 *General Data* (see Figure 2.5, page 11).

1.10 Serviceability Data

No.	A Reference to	B Member No.	C Reference Length Manually	D Reference Length L [m]	E Direc-tion	F Precamber w_c [mm]	G Beam Type	H Comment
1	Member	2	☐	6.000	y, z	0.0	Beam	
2	List of Members	1,32	☐	9.000	y, z	0.0	Beam	
3	Member	7	☐	3.262	y, z	0.0	Beam	
4	Member	1	☐	6.000	y, z	0.0	Beam	
5	Member	16	☐	6.274	y, z	0.0	Beam	
6	Set of Members	2	☐	6.000	y, z	0.0	Beam	
7								
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

Figure 2.26: Table 1.10 *Serviceability Data*



In column A, you decide whether you want to apply the deformation to single members, lists of members, or sets of members.

In table column B, you enter the numbers of the members or sets of members that you want to design. You can also click [...] to select them graphically in the RSTAB work window. Then, the *Reference Length* appears in column D automatically. This column presets the lengths of the members, sets of members, or member lists. If required, you can adjust these values after selecting the *Manually* check box in column C.

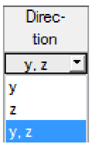
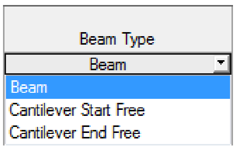


Table column E defines the governing *Direction* for the deformation analysis. You can select the directions of the local member axes y and z (or u and v for unsymmetrical cross-sections).

A *precamber* w_c can be taken into account by using entries specified in column F.

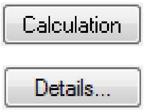


The *Beam Type* is of vital importance for the correct application of limit deformations. In table column G, you can select the girder to be a beam or a cantilever and decide which end should have no support.

The settings of the *Details* dialog box determine whether the deformations are related to the undeformed initial model or to the shifted ends of members or sets of members (see Figure 3.1, page 28).

3. Calculation

3.1 Details



Before you start the calculation by clicking [Calculation], it is recommended to check the design details. The corresponding dialog box can be accessed in all tables of the add-on module by clicking [Details].

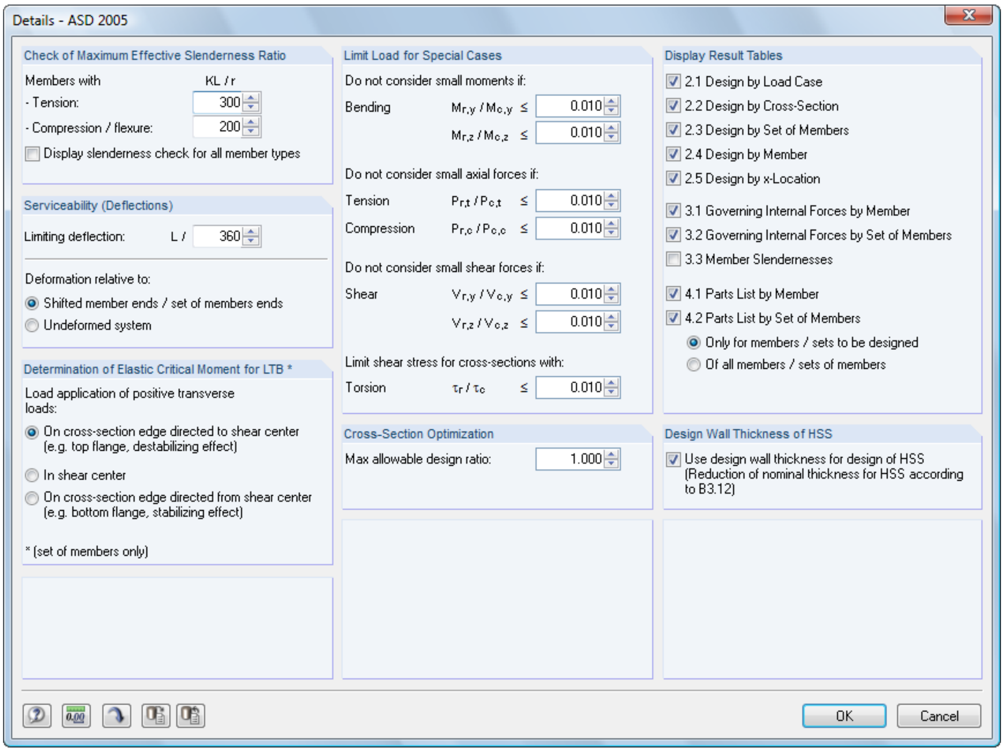


Figure 3.1: Dialog box *Details*

Check of Maximum Effective Slenderness Ratio

For members designed according to Chapter D [1], [2], 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], [2], 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) if the corresponding check box is selected in the dialog section *Display Result Tables*.

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 option fields below control whether the *Deformation* is to be related to the shifted ends of members or sets of members (connection line between start and end nodes of the deformed system) or to the undeformed initial system. As a rule, the deformations have to be checked relative to the displacements in the entire structural system.

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 (transverse loads). It is important to define where these forces are acting on the cross-section: Depending on the *Load Application* point, transverse loads can be stabilizing or destabilizing, and in this way they can critically influence the ideal critical moment.

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 dialog section, the limits of these internal forces or stresses can be entered. Those are defined as the ratios between existing internal forces or stresses and the corresponding resistances of each cross-section.

These limit settings are not part of the ANSI/AISC Specification. Changing the limits is in the responsibility of the program user.

Cross-Section Optimization

The optimization is targeted on the maximum design ratio of 100 %. If necessary, you can specify a different limit value in this input field.

Display Results Tables

In this dialog section, you can select the results tables including parts list that you want to be displayed. The tables are described in chapter 4 *Results*.

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.

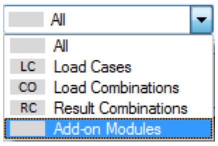
3.2 Start Calculation

Calculation

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

STEEL AISC searches for the results of the load cases, load and result combinations to be designed. If these cannot be found, the program starts the RSTAB calculation to determine the design relevant internal forces.

You can also start the calculation in the RSTAB user interface: In the dialog box *To Calculate* (menu *Calculate → To Calculate*), design cases of the add-on modules like load cases and load combinations are listed.



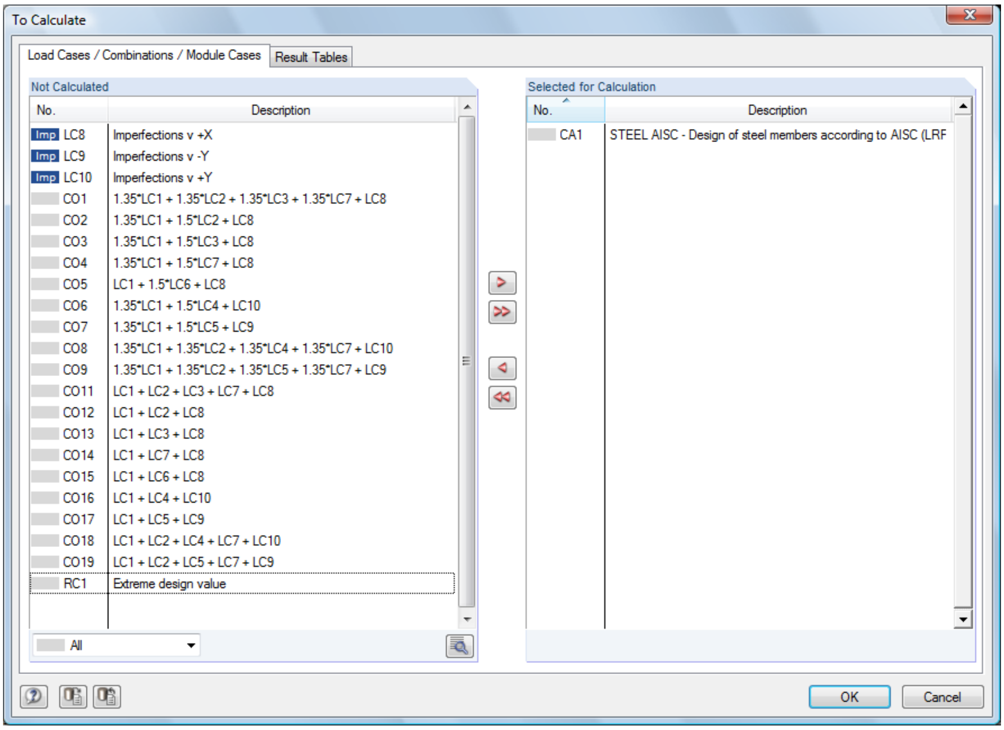


Figure 3.2: *To Calculate* Dialog box

If the STEEL AISC design cases are missing in the *Not Calculated* list, select *All* or *Add-on Modules* in the drop-down list at the end of the list.

To transfer the selected STEEL AISC cases to the list on the right, use the button [▶]. Click [OK] to start the calculation.

To calculate a design case directly, use the list in the toolbar. Select the STEEL AISC design case in the toolbar list, and then click [Show Results].

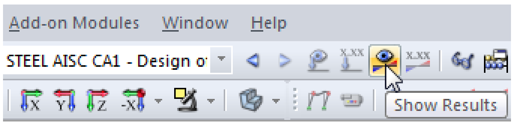


Figure 3.3: Direct calculation of a STEEL AISC design case in RSTAB

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

4. Results

Table 2.1 *Design by Load Case* is displayed immediately after the calculation.

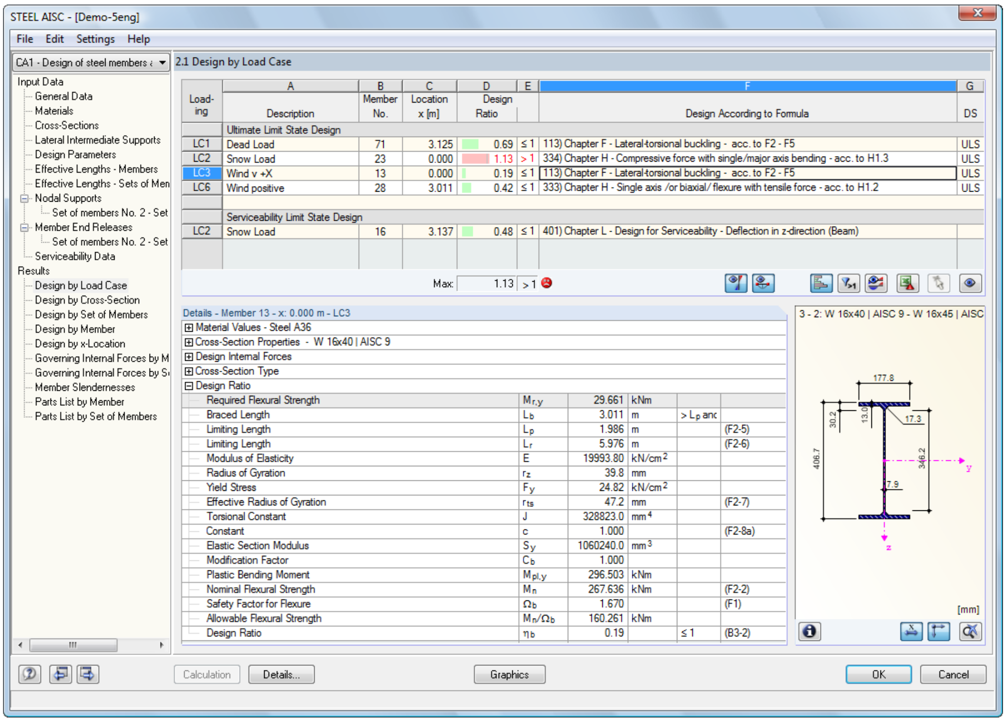
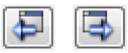


Figure 4.1: Results table with designs and intermediate values

The designs are shown in the results tables 2.1 to 2.5, sorted by different criteria.

The tables 3.1 and 3.2 list the governing internal forces. Table 3.3 informs you about the member slendernesses. In the last two results tables, 4.1 and 4.2, parts lists are displayed by member and set of members.



Every table can be selected by clicking the corresponding entry in the navigator. To set the previous or next input table, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] table.

Click [OK] to save the results. Thus you exit STEEL AISC and return to the main program.

Chapter 4 *Results* describes the different results tables one by one. Evaluating and checking results is described in chapter 5 *Results Evaluation*, page 41 ff..

4.1 Design by Load Case

The upper part of this table offers a summery, sorted by load cases, load combinations and result combinations of the governing designs. Furthermore, this table is divided into *Ultimate Limit State Design* and *Serviceability Limit State Design* results.

The lower part of the table contains detailed information on the cross-section characteristics, analyzed internal forces and design parameters for the load case selected above.

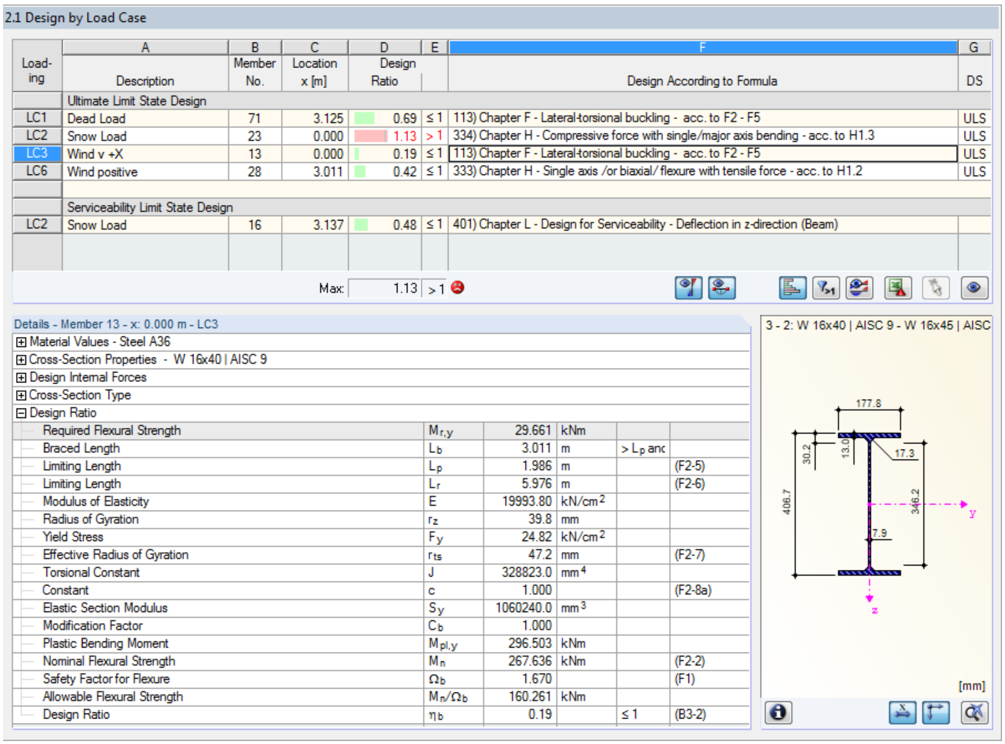


Figure 4.2: Table 2.1 *Design by Load Case*

Description

This column shows the descriptions of the load cases, load and result combinations used for the design.

Member No.

This column shows the number of the member that bears the maximum stress ratio of the designed loading.

Location x

This column shows the respective x-location where the member's maximum stress ratio occurs. For the table output, the program uses the following member locations *x*:

- Start and end node
- Division points according to possibly defined member division (see RSTAB table 1.6)
- Member division according to specification for member results (RSTAB dialog box *Calculation Parameters*, tab *Global Register Parameters*)
- Extreme values of internal forces

Design Ratio

Columns D and E display the design conditions according to the ANSI/AISC Specification [1], [2]. The lengths of the colored scale represent the respective utilizations.

Design according to Formula

This column lists the code's equations by which the designs have been performed.

DS

The last column contains information on the respective design-relevant design situation *(DS)*: *ULS* (Ultimate Limit State) or *SLS* (Serviceability Limit State).

4.2 Design by Cross-Section

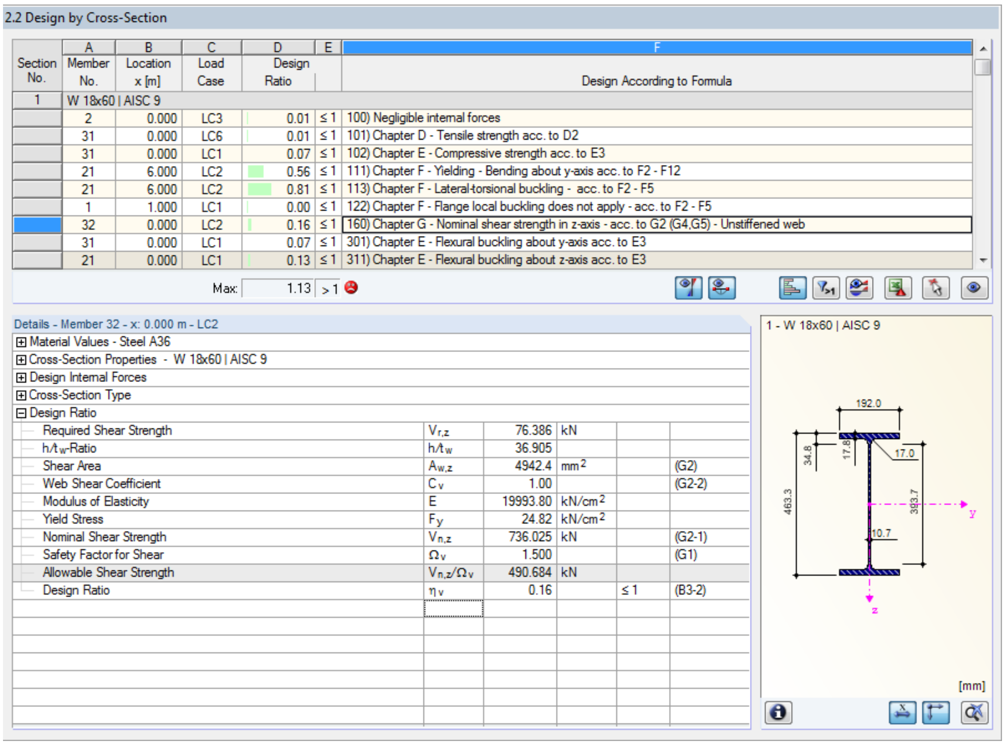
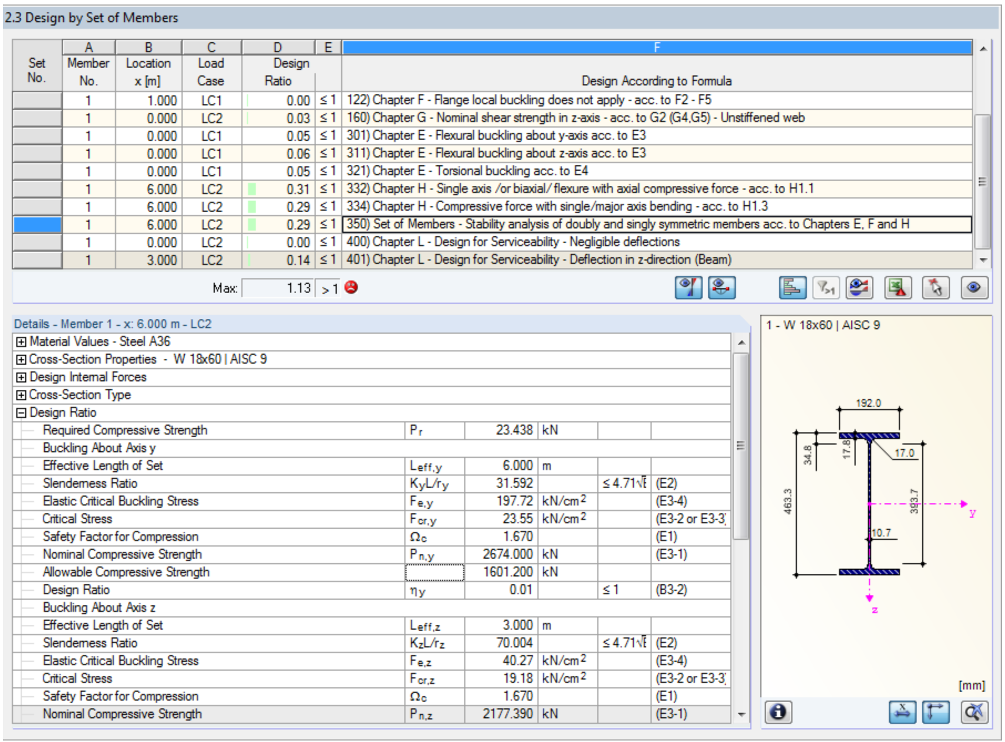


Figure 4.3: Table 2.2 *Design by Cross-Section*

This table lists the maximum ratios of all members and actions selected for design, sorted by cross-section. The results are sorted by cross-section design and serviceability limit state design.

If you have a tapered member, both cross-section descriptions are displayed in the table row next to the section number.

4.3 Design by Set of Members


2.3 Design by Set of Members

Set No.	A Member No.	B Location x [m]	C Load Case	D Design Ratio	E	F Design According to Formula
	1	1.000	LC1	0.00	≤ 1	122) Chapter F - Flange local buckling does not apply - acc. to F2 - F5
	1	0.000	LC2	0.03	≤ 1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G4,G5) - Unstiffened web
	1	0.000	LC1	0.05	≤ 1	301) Chapter E - Flexural buckling about y-axis acc. to E3
	1	0.000	LC1	0.06	≤ 1	311) Chapter E - Flexural buckling about z-axis acc. to E3
	1	0.000	LC1	0.05	≤ 1	321) Chapter E - Torsional buckling acc. to E4
	1	6.000	LC2	0.31	≤ 1	332) Chapter H - Single axis /or biaxial/ flexure with axial compressive force - acc. to H1.1
	1	6.000	LC2	0.29	≤ 1	334) Chapter H - Compressive force with single/major axis bending - acc. to H1.3
	1	6.000	LC2	0.29	≤ 1	350) Set of Members - Stability analysis of doubly and singly symmetric members acc. to Chapters E, F and H
	1	0.000	LC2	0.00	≤ 1	400) Chapter L - Design for Serviceability - Negligible deflections
	1	3.000	LC2	0.14	≤ 1	401) Chapter L - Design for Serviceability - Deflection in z-direction (Beam)
			Max:	1.13	> 1	

Details - Member 1 - x: 6.000 m - LC2

⊞ Material Values - Steel A36					
⊞ Cross-Section Properties - W 18x60 | AISC 9					
⊞ Design Internal Forces					
⊞ Cross-Section Type					
⊟ Design Ratio					
Required Compressive Strength	P_r	23.438	kN		
Buckling About Axis y					
Effective Length of Set	$L_{eff,y}$	6.000	m		
Slenderness Ratio	K_yL/r_y	31.592		≤ 4.71√E	(E2)
Elastic Critical Buckling Stress	$F_{e,y}$	197.72	kN/cm²		(E3-4)
Critical Stress	$F_{cr,y}$	23.55	kN/cm²		(E3-2 or E3-3)
Safety Factor for Compression	Ω_c	1.670			(E1)
Nominal Compressive Strength	$P_{n,y}$	2674.000	kN		(E3-1)
Allowable Compressive Strength		1601.200	kN		
Design Ratio	η_y	0.01		≤ 1	(B3-2)
Buckling About Axis z					
Effective Length of Set	$L_{eff,z}$	3.000	m		
Slenderness Ratio	K_zL/r_z	70.004		≤ 4.71√E	(E2)
Elastic Critical Buckling Stress	$F_{e,z}$	40.27	kN/cm²		(E3-4)
Critical Stress	$F_{cr,z}$	19.18	kN/cm²		(E3-2 or E3-3)
Safety Factor for Compression	Ω_c	1.670			(E1)
Nominal Compressive Strength	$P_{n,z}$	2177.390	kN		(E3-1)

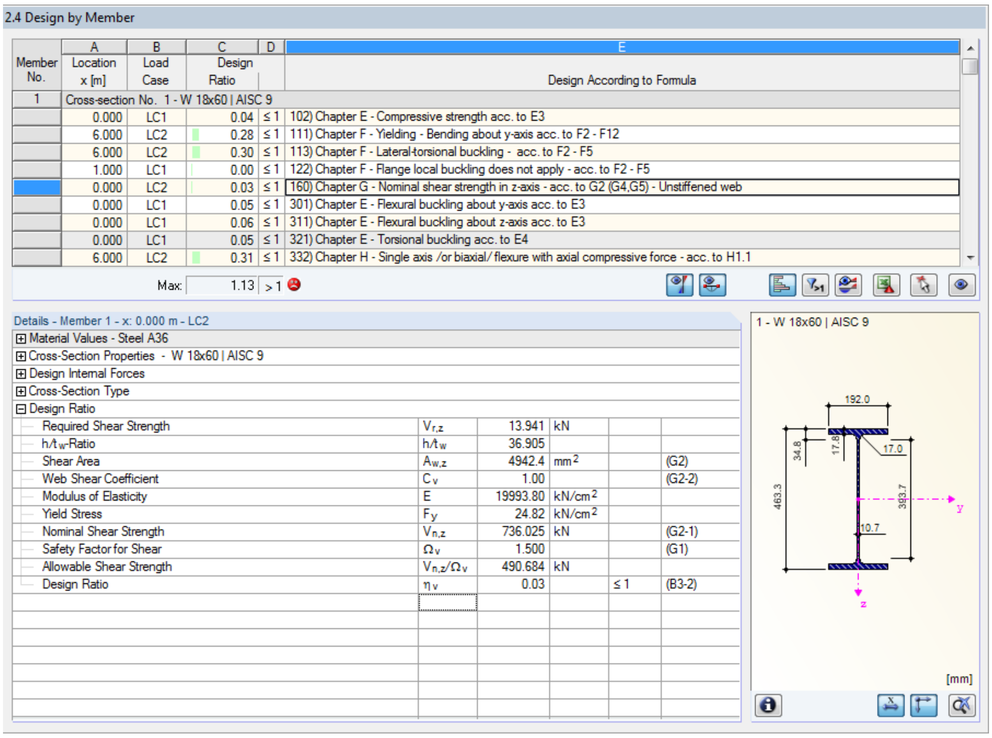
Figure 4.4: Table 2.3 *Design by Set of Members*

This results table is displayed if you have selected at least one set of members for design. The table lists the maximum ratios sorted by set of members.

The Column *Member No.* shows the number of the one member within the set of members that bears the maximum ratio for the individual design criteria.

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

4.4 Design by Member



2.4 Design by Member

Member No.	A Location x [m]	B Load Case	C Design Ratio	D	E Design According to Formula
1	Cross-section No. 1 - W 18x60 | AISC 9				
	0.000	LC1	0.04	≤ 1	102) Chapter E - Compressive strength acc. to E3
	6.000	LC2	0.28	≤ 1	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	6.000	LC2	0.30	≤ 1	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F5
	1.000	LC1	0.00	≤ 1	122) Chapter F - Flange local buckling does not apply - acc. to F2 - F5
	0.000	LC2	0.03	≤ 1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G4,G5) - Unstiffened web
	0.000	LC1	0.05	≤ 1	301) Chapter E - Flexural buckling about y-axis acc. to E3
	0.000	LC1	0.06	≤ 1	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.000	LC1	0.05	≤ 1	321) Chapter E - Torsional buckling acc. to E4
	6.000	LC2	0.31	≤ 1	332) Chapter H - Single axis /or biaxial/ flexure with axial compressive force - acc. to H1.1
	Max:		1.13	> 1	

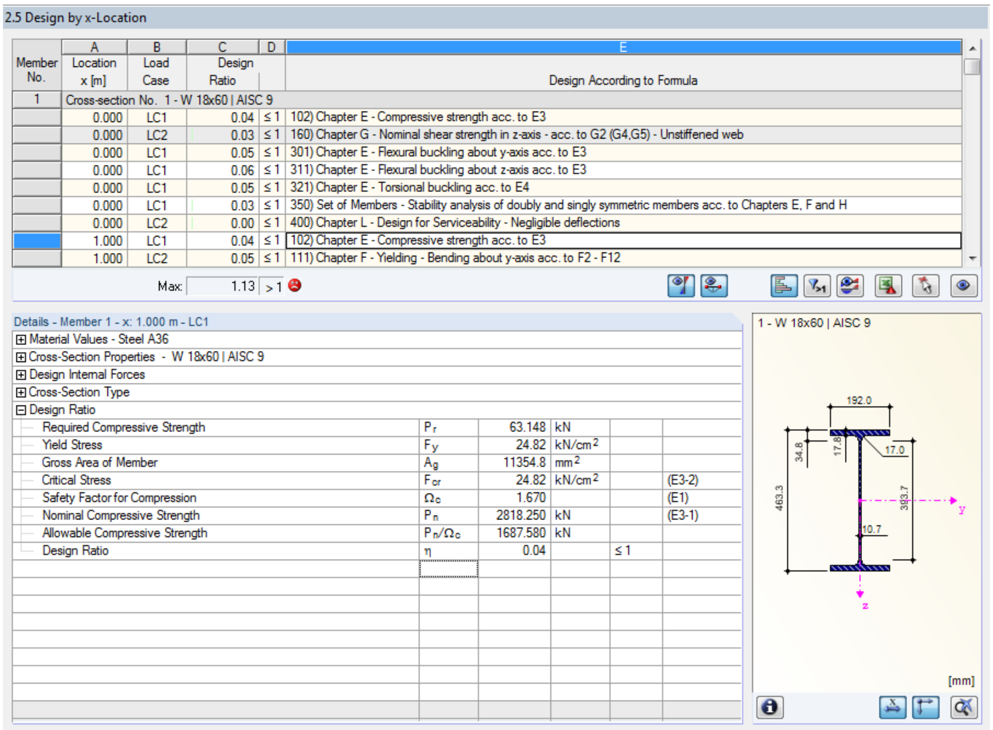
Details - Member 1 - x: 0.000 m - LC2

⊞ Material Values - Steel A36					
⊞ Cross-Section Properties - W 18x60 | AISC 9					
⊞ Design Internal Forces					
⊞ Cross-Section Type					
⊟ Design Ratio					
Required Shear Strength	$V_{r,z}$	13.941	kN		
h/t_w-Ratio	h/t_w	36.905			
Shear Area	$A_{w,z}$	4942.4	mm^2		(G2)
Web Shear Coefficient	C_v	1.00			(G2-2)
Modulus of Elasticity	E	19993.80	kN/cm^2		
Yield Stress	F_y	24.82	kN/cm^2		
Nominal Shear Strength	$V_{n,z}$	736.025	kN		(G2-1)
Safety Factor for Shear	Ω_v	1.500			(G1)
Allowable Shear Strength	$V_{n,z}/\Omega_v$	490.684	kN		
Design Ratio	η_v	0.03		≤ 1	(B3-2)

Figure 4.5: Table 2.4 *Design by Member*

This results table presents the maximum ratios for the individual designs sorted by member number. The columns are described in detail in chapter 4.1 on page 32.

4.5 Design by x-Location



2.5 Design by x-Location

Member No.	A Location x [m]	B Load Case	C Design Ratio	D	E Design According to Formula
1	Cross-section No. 1 - W 18x60 | AISC 9				
	0.000	LC1	0.04	≤ 1	102) Chapter E - Compressive strength acc. to E3
	0.000	LC2	0.03	≤ 1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G4,G5) - Unstiffened web
	0.000	LC1	0.05	≤ 1	301) Chapter E - Flexural buckling about y-axis acc. to E3
	0.000	LC1	0.06	≤ 1	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.000	LC1	0.05	≤ 1	321) Chapter E - Torsional buckling acc. to E4
	0.000	LC1	0.03	≤ 1	350) Set of Members - Stability analysis of doubly and singly symmetric members acc. to Chapters E, F and H
	0.000	LC2	0.00	≤ 1	400) Chapter L - Design for Serviceability - Negligible deflections
	1.000	LC1	0.04	≤ 1	102) Chapter E - Compressive strength acc. to E3
	1.000	LC2	0.05	≤ 1	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	Max:		1.13	> 1	

Details - Member 1 - x: 1.000 m - LC1

⊞ Material Values - Steel A36					
⊞ Cross-Section Properties - W 18x60 | AISC 9					
⊞ Design Internal Forces					
⊞ Cross-Section Type					
⊟ Design Ratio					
Required Compressive Strength	P_r	63.148	kN		
Yield Stress	F_y	24.82	kN/cm^2		
Gross Area of Member	A_g	11354.8	mm^2		
Critical Stress	F_{cr}	24.82	kN/cm^2		(E3-2)
Safety Factor for Compression	Ω_c	1.670			(E1)
Nominal Compressive Strength	P_n	2818.250	kN		(E3-1)
Allowable Compressive Strength	P_n/Ω_c	1687.580	kN		
Design Ratio	η	0.04		≤ 1	

Figure 4.6: Table 2.5 *Design by x-Location*

This results table lists the maxima for each member at the locations ***x*** resulting from the division points defined in RSTAB:

- Start and end node
- Division points according to possibly defined member division (see RSTAB table 1.6)
- Member division according to specification for member results (RSTAB dialog box *Calculation Parameters*, tab *Global Register Parameters)*
- Extreme values of internal forces

4.6 Governing Internal Forces by Member

3.1 Governing Internal Forces by Member

	A	B	C	D	E	F	G	H	I
Member No.	Location x [m]	Load-ing	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
1	Cross-section No. 1 - W 18x60 | AISC 9								
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	102) Chapter E - Compressive strength acc. to E3
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1:
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F5
	1.000	LC1	-63.148	0.000	-10.808	0.001	-10.808	0.000	122) Chapter F - Flange local buckling does not apply - acc. to
	0.000	LC2	-23.438	0.000	-13.941	0.000	0.000	0.000	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	301) Chapter E - Flexural buckling about y-axis acc. to E3
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	321) Chapter E - Torsional buckling acc. to E4
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	332) Chapter H - Single axis /or biaxial/ flexure with axial compr
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	334) Chapter H - Compressive force with single/major axis benc
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	350) Set of Members - Stability analysis of doubly and singly syn
	0.000	LC2	0.000	0.000	0.000	0.000	0.000	0.000	400) Chapter L - Design for Serviceability - Negligible deflection
	3.000	LC2	0.000	0.000	0.000	0.000	0.000	0.000	401) Chapter L - Design for Serviceability - Deflection in z-direct
2	Cross-section No. 1 - W 18x60 | AISC 9								
	0.000	LC3	-1.170	0.000	4.278	0.000	0.000	0.000	100) Negligible internal forces
	0.000	LC1	-72.140	0.000	10.808	-0.001	0.000	0.000	102) Chapter E - Compressive strength acc. to E3
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1:
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F5
	1.000	LC1	-63.148	0.000	10.808	-0.001	10.808	0.000	122) Chapter F - Flange local buckling does not apply - acc. to
	0.000	LC2	-23.438	0.000	13.941	0.000	0.000	0.000	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (
	0.000	LC1	-72.140	0.000	10.808	-0.001	0.000	0.000	301) Chapter E - Flexural buckling about y-axis acc. to E3
	0.000	LC1	-72.140	0.000	10.808	-0.001	0.000	0.000	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.000	LC1	-72.140	0.000	10.808	-0.001	0.000	0.000	321) Chapter E - Torsional buckling acc. to E4
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	332) Chapter H - Single axis /or biaxial/ flexure with axial compr
	6.000	LC2	-23.438	0.000	13.941	0.000	83.646	0.000	334) Chapter H - Compressive force with single/major axis benc
	0.000	LC2	0.000	0.000	0.000	0.000	0.000	0.000	400) Chapter L - Design for Serviceability - Negligible deflection
	3.000	LC2	0.000	0.000	0.000	0.000	0.000	0.000	401) Chapter L - Design for Serviceability - Deflection in z-direct
3	Cross-section No. 3 - W 16x40 | AISC 9 ... 2 - W 16x45 | AISC 9								
	0.000	LC2	-15.920	0.000	22.141	0.000	-83.646	0.000	102) Chapter E - Compressive strength acc. to E3
	0.000	LC2	-15.920	0.000	22.141	0.000	-83.646	0.000	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1:
	0.000	LC1	-12.328	-0.005	17.010	-0.001	-64.847	-0.001	116) Chapter F - Lateral-torsional buckling does not Apply - acc

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

This table displays for each member the governing internal forces that result in maximum stress ratios in each design.

Location x

At this x location of the member, the respective maximum design ratio occurs.

Load Case

This column displays the number of the load case, the load combination or result combination whose internal forces produce maximum stresses.

Forces / Moments

For each member, this column displays the axial and shear forces as well as the torsional and bending moments producing maximum design ratios in the respective ultimate limit state and serviceability limit state designs.

Design According to Formula

The final column informs you about the design types and the equations by which the designs according to [1] or [2] have been performed.

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

	A	B	C	D	E	F	G	H	I
Set No.	Location x [m]	Load-ing	Forces [kN] N	V_y	V_z	Moments [kNm] M_T	M_y	M_z	Design According to Formula
2	Set of Members 2 (Member No. 1)								
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	102) Chapter E - Compressive strength acc. to E3
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F5
	1.000	LC1	-63.148	0.000	-10.808	0.001	-10.808	0.000	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	0.000	LC2	-23.438	0.000	-13.941	0.000	0.000	0.000	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G4,(
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	301) Chapter E - Flexural buckling about y-axis acc. to E3
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.000	LC1	-72.140	0.000	-10.808	0.001	0.000	0.000	321) Chapter E - Torsional buckling acc. to E4
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	332) Chapter H - Single axis /or biaxial/ flexure with axial compress
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	334) Chapter H - Compressive force with single/major axis bending
	6.000	LC2	-23.438	0.000	-13.941	0.000	-83.646	0.000	350) Set of Members - Stability analysis of doubly and singly symme
	0.000	LC2	0.000	0.000	0.000	0.000	0.000	0.000	400) Chapter L - Design for Serviceability - Negligible deflections
	3.000	LC2	0.000	0.000	0.000	0.000	0.000	0.000	401) Chapter L - Design for Serviceability - Deflection in z-direction

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

This table contains the internal forces that result in the maximum ratios within the design of each set of members.

4.8 Member Slendernesses

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [m]	C Major Axis y k_y [-]	D i_y [mm]	E λ_y [-]	F Minor Axis z k_z [-]	G i_z [mm]	H λ_z [-]	I
1	Compression / Flexure	6.000	1.000	189.9	31.592	0.500	42.9	70.004	
2	Compression / Flexure	6.000	1.000	189.9	31.592	1.000	42.9	140.009	
3	Compression / Flexure	3.011	1.000	168.3	17.894	0.333	39.4	25.503	
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	0.250	39.9	39.322	
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	76.509	
11	Compression / Flexure	6.000	1.000	189.9	31.592	1.000	42.9	140.009	
12	Compression / Flexure	6.000	1.000	189.9	31.592	1.000	42.9	140.009	
13	Compression / Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.509	
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	76.509	
21	Compression / Flexure	6.000	1.000	189.9	31.592	1.000	42.9	140.009	
22	Compression / Flexure	6.000	1.000	189.9	31.592	1.000	42.9	140.009	
23	Compression / Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.509	
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	
27	Compression / Flexure	3.262	1.000	168.6	19.350	1.000	39.9	81.790	
28	Compression / Flexure	3.011	1.000	168.3	17.894	1.000	39.4	76.509	
31	Compression / Flexure	3.000	1.000	189.9	15.796	1.000	42.9	70.004	
32	Compression / Flexure	3.000	1.000	189.9	15.796	1.000	42.9	70.004	
33	Compression / Flexure	3.000	1.000	85.1	35.251	1.000	49.5	60.574	
34	Compression / Flexure	3.546	1.000	85.1	41.666	1.000	49.5	71.599	
35	Compression / Flexure	3.000	1.000	85.1	35.251	1.000	49.5	60.574	
36	Compression / Flexure	4.094	1.000	85.1	48.105	1.000	49.5	82.664	

Members with compression / flexure:

Max $K_y L / r_y$: 168.975 ≤ 200

Max $K_z L / r_z$: 193.406 ≤ 200

Figure 4.8: Table 3.3 *Member Slendernesses*

Details...

This results table is shown only if you have selected the respective check box in the *Details* dialog box (see Figure 3.1, page 28).

Details...

The table lists the effective slendernesses of the designed members for both directions of the principal axes. They were determined depending on the type of load. At the end of the list, you find a comparison with the limit values that have been defined in the *Details* dialog box.

Members of the member type "Tension" or "Cable" are not included in this table.

This table is displayed only for information. No design of the slendernesses is intended.

4.9 Parts List by Member

Finally, STEEL AISC provides a summary of all members that are included in the design case.

4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	1 - W 18x60 | AISC 9	6	6.00	36.00	61.64	0.41	89.14	534.81	3.209
2	2 - W 16x45 | AISC 9 ... 3 - W 16x40 | AISC 9	8	3.01	24.09	37.39	0.20	63.56	191.40	1.531
3	2 - W 16x45 | AISC 9	8	3.26	26.10	40.62	0.22	67.36	219.75	1.758
4	2 - W 16x45 | AISC 9	8	6.27	50.19	78.11	0.43	67.36	422.60	3.381
5	1 - W 18x60 | AISC 9	4	3.00	12.00	20.55	0.14	89.14	267.41	1.070
6	10 - HP 8x8.2x36 | AISC 9	3	3.00	9.00	10.72	0.06	53.68	161.05	0.483
7	10 - HP 8x8.2x36 | AISC 9	2	3.55	7.09	8.45	0.05	53.68	190.36	0.381
8	10 - HP 8x8.2x36 | AISC 9	1	4.09	4.09	4.88	0.03	53.68	219.78	0.220
9	15 - HP 10x10.1x42 | AISC 9	4	3.00	12.00	17.62	0.10	62.80	188.40	0.754
10	6 - HP 10x10.1x42 | AISC 9	3	3.00	9.00	13.22	0.07	62.80	188.40	0.565
11	6 - HP 10x10.1x42 | AISC 9	2	3.55	7.09	10.41	0.06	62.80	222.69	0.445
12	6 - HP 10x10.1x42 | AISC 9	1	4.09	4.09	6.01	0.03	62.80	257.10	0.257
13	7 - HP 8x8.2x36 | AISC 9	4	6.27	25.10	29.90	0.17	53.68	336.80	1.347
14	9 - W 18x40 | AISC 9	8	6.25	50.00	77.18	0.38	59.76	373.51	2.988
15	16 - W 14x38 | AISC 9	1	6.55	6.55	9.29	0.05	56.72	371.31	0.371
16	6 - HP 10x10.1x42 | AISC 9	1	7.09	7.09	10.42	0.06	62.80	445.50	0.446
17	6 - HP 10x10.1x42 | AISC 9	1	6.55	6.55	9.61	0.05	62.80	411.09	0.411
18	12 - SHS 3x0.125 | AISC 9	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	489.95	2.66			20.842

Figure 4.9: Table 4.1 *Parts List by Member*

By default, this list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box, tab *Other* (see Figure 3.1 on page 28).

Part No.

The program automatically assigns item numbers to similar members.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column displays the respective length of an individual member.

Total Length

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

Surface Area

For each part, the program indicates the surface area related to the total length. The surface area is determined from the *Surface Area* of the cross-sections that can be seen in tables 1.3 and 2.1 to 2.5 in the cross-section information (see Figure 2.11, page 16).

Volume

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

Cross-Section Mass

The *Cross-Section Mass* is related to the length of one meter. For tapered cross-sections, the program averages both cross-section properties.

Mass

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

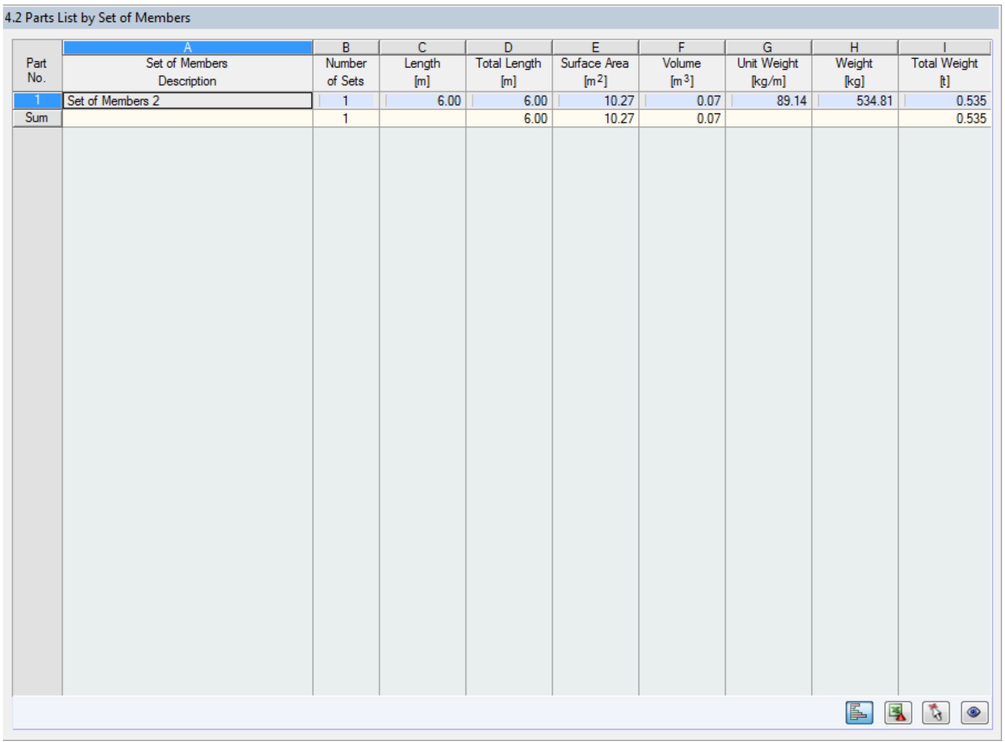
Total Mass

The final column indicates the total mass of each part.

Sum

At the bottom of the list, you find a summary of the summed up values of column B, D, E, F, and I. The last data field of the column *Total Mass* gives information about the total amount of steel required.

4.10 Parts List by Set of Members



4.2 Parts List by Set of Members

Part No.	A Set of Members Description	B Number of Sets	C Length [m]	D Total Length [m]	E Surface Area [m²]	F Volume [m³]	G Unit Weight [kg/m]	H Weight [kg]	I Total Weight [t]
1	Set of Members 2	1	6.00	6.00	10.27	0.07	89.14	534.81	0.535
Sum		1		6.00	10.27	0.07			0.535

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

The last results table is displayed if you have selected at least one set of members for design. The table summarizes an entire structural group (for example a horizontal beam) in a parts list.

Details on the various columns can be found in the previous chapter. If different cross-sections are used in the set of members, the program averages the surface area, the volume, and the cross-section mass.

5. Results Evaluation

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.

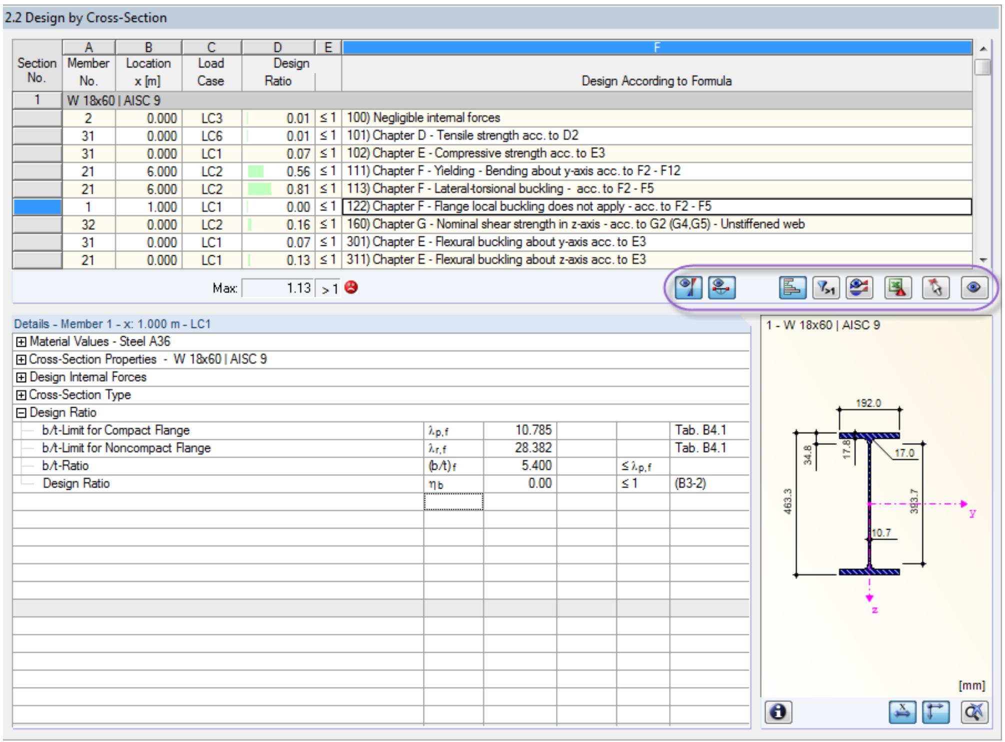


Figure 5.1: Buttons for results evaluation

These buttons have the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Turns on and off the results of the ultimate limit state design
	Serviceability Limit State Designs	Turns on and off the results of the serviceability limit state design
	Show Color Bars	Turns on and off the colored reference scales in the results tables
	Show Rows with Ratio > 1	Displays only the rows where the ratio is more than 1, and thus the design is failed
	Result Diagrams	Opens the window *Result Diagram on Member* → chapter 5.2, page 44
	Excel Export	Exports the table to MS Excel / OpenOffice → chapter 7.4.3, page 54
	Member Selection	Allows you to graphically select a member to display its results in the table
	View Mode	Jumps to the RSTAB work window to change the view

Table 5.1: Buttons in results tables 2.1 to 2.5

5.1 Results in the RSTAB Model

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

RSTAB background graphic and view mode

The RSTAB work window in the background is useful for finding the position of a particular member in the model: The member selected in the STEEL AISC results table is highlighted in the selection color in the background graphic. Furthermore, an arrow indicates the member's x-location that is displayed in the selected table row.

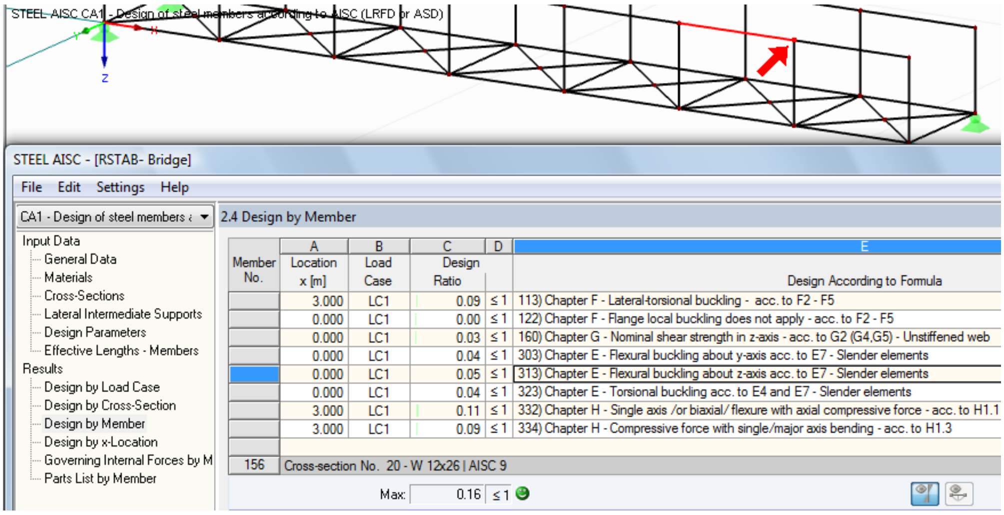


Figure 5.2: Indication of the member and the current *Location x* in the RSTAB model

In case you cannot improve the display by moving the STEEL AISC module window, click [Jump to Graphic] to activate the *View Mode*: The program hides the module window so that you can modify the display in the RSTAB user interface. The view mode provides the functions of the *View* menu, for example zooming, moving, or rotating the display. The pointer remains visible.

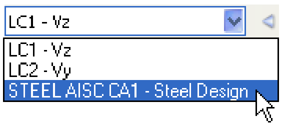
Click [Back] to return to the add-on module STEEL AISC.

RSTAB work window

The ratios can also be checked graphically in the RSTAB model. Click [Graphics] to exit the design module. The ratios are displayed in the RSTAB work window like internal forces of a load case.

To turn the display of design results on or off, use the [Show Results] button known from the display of internal forces in RSTAB. To display the result values, use the toolbar button [Show Values] to the right.

As the RSTAB tables are of no relevance for the evaluation of design results, you can hide them.



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

The graphical representation of the results can be controlled in the *Display* navigator by selecting *Results* → *Members*. The display of ratios is *Two-Colored* by default.

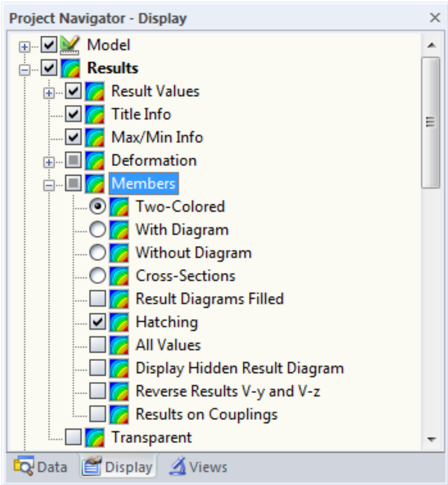


Figure 5.3: *Display* navigator: Results → Members

In case of a multicolor representation (options *With/Without Diagram* or *Cross-Sections*), the color panel is available, providing common control functions. The functions are described in detail in the RSTAB manual, chapter 3.4.6.

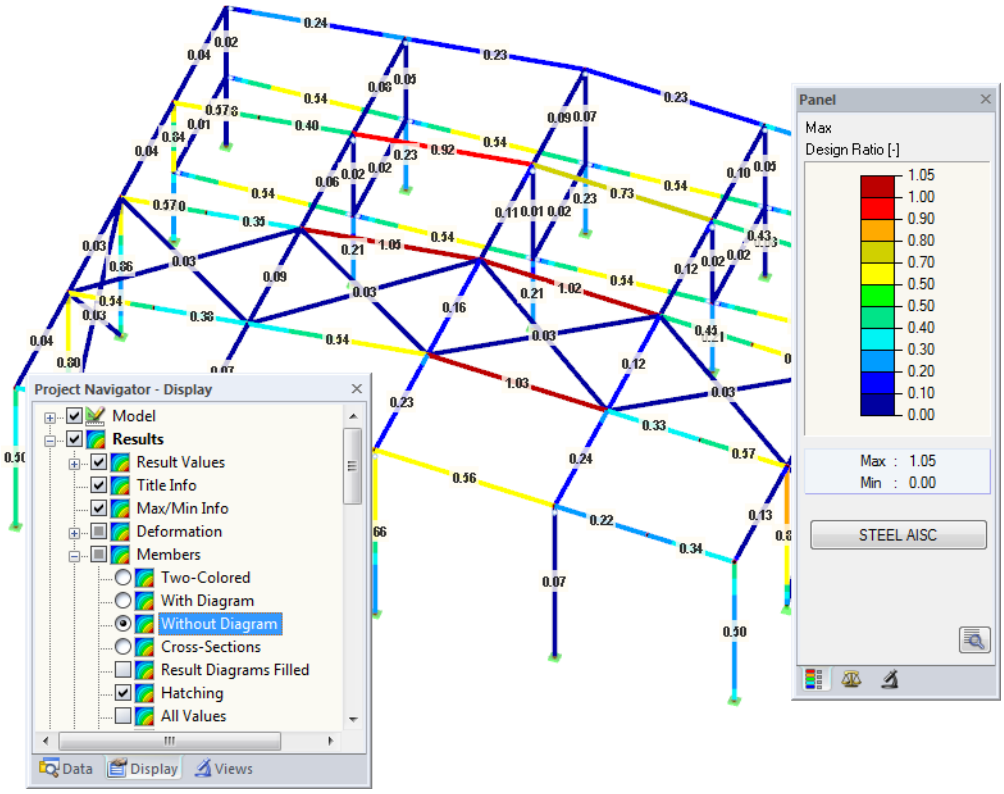


Figure 5.4: Design ratios with display option *Without Diagram*

This graphics of the design results can be transferred to the printout report (see chapter 6.2, page 47).

You can return to the STEEL AISC module by clicking the panel button [STEEL AISC].

5.2 Result Diagrams

You can also evaluate a member's result distributions in the result diagram graphically.

To do this, select the member (or set of members) in the STEEL AISC results table by clicking in the table row of the member. Then open the dialog box *Result Diagram on Member* by clicking the button shown on the left. The button is at the bottom of the upper results table (see Figure 5.1, page 41).

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

Results → Result Diagrams for Selected Members

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

A window opens, graphically showing the distribution of the maximum design values on the member or set of members.

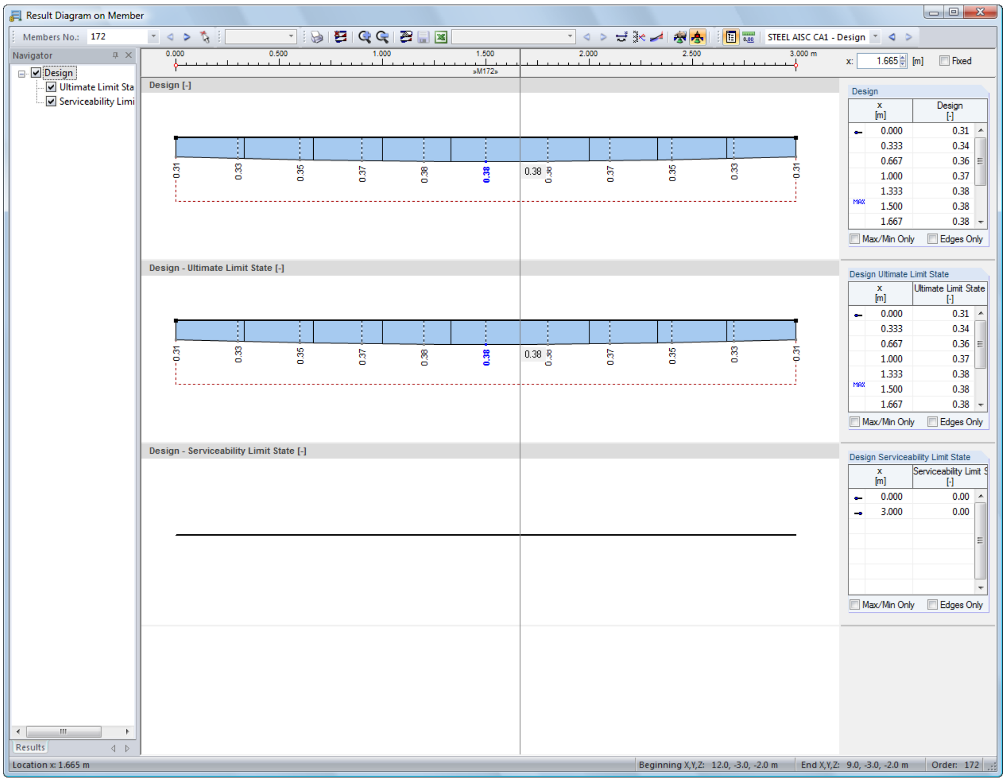
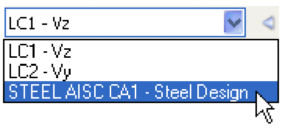


Figure 5.5: Dialog box *Result Diagram on Member*



Use the list in the toolbar above to choose the relevant STEEL AISC design case.

The dialog box *Result Diagram on Member* is described in detail in the RSTAB manual, chapter 9.5.

5.3 Filter for Results

The STEEL AISC results tables allow you to sort the results by various criteria. In addition, you can use the filter options described in chapter 9.7 of the RSTAB manual to evaluate the design results graphically.

You can use the option *Visibility* also for STEEL AISC (see RSTAB manual, chapter 9.7.1) to filter the members in order to evaluate them.

Filtering designs

The design ratios can easily be used as filter criteria in the RSTAB work window, which can be accessed by clicking [Graphics]. To apply this filter function, the panel must be displayed. If the panel is not active, click

View → Control Panel (Color Scale, Factors, Filter)

or use the toolbar button shown on the left.

The panel is described in the RSTAB manual, chapter 3.4.6. The filter settings for the results must be defined in the first panel tab (Color spectrum). As this register is not available for the two-colored results display, you have to use the *Display* navigator and set the display options *Colored With/Without Diagram* or *Cross-Sections* first.

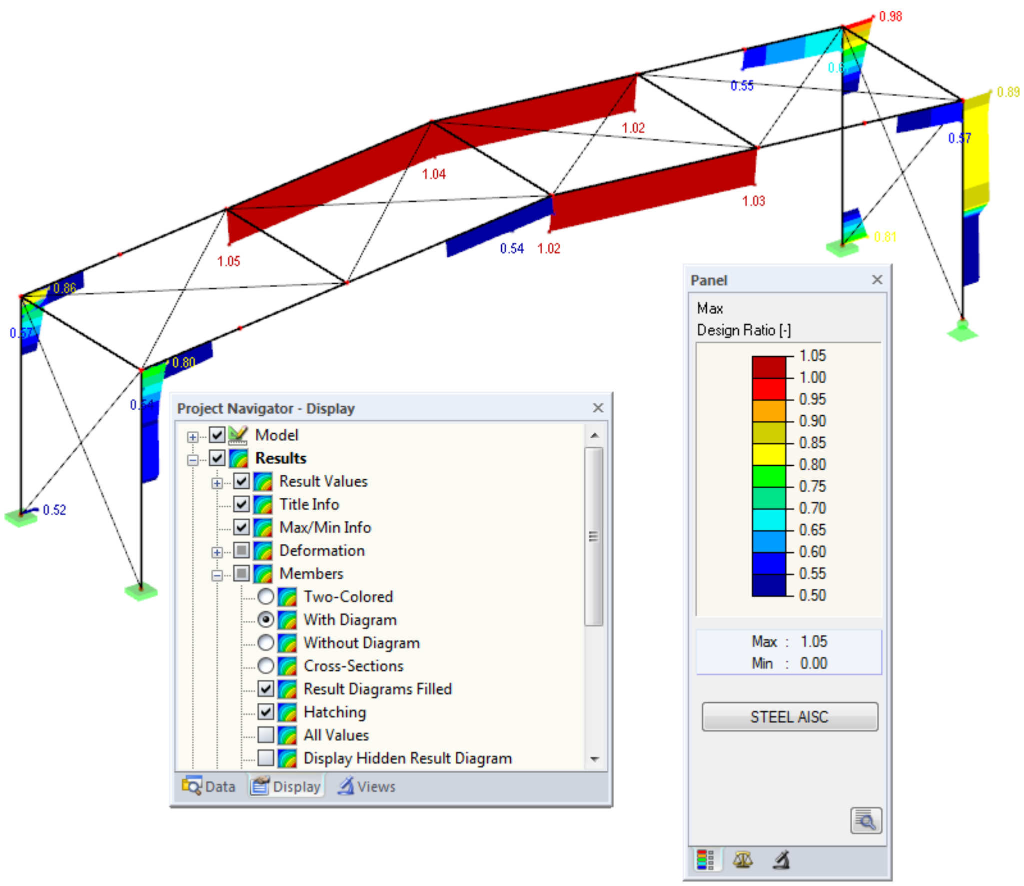


Figure 5.6: Filtering design ratios with adjusted color spectrum

As the figure above shows, the color spectrum can be set in such a way that only ratios higher than 0.50 are shown in a color range between blue and red.

If you select the option *Display Hidden Result Diagram* in the *Display* navigator (*Results → Members*), you can display all stress ratio diagrams that are not covered by the color spectrum. Those diagrams will be represented by dotted lines.

Filtering members

In the *Filter* tab of the control panel, you can specify the numbers of particular members to display their results exclusively, that is, filtered. This function is described in detail in the RSTAB manual, chapter 9.7.3.

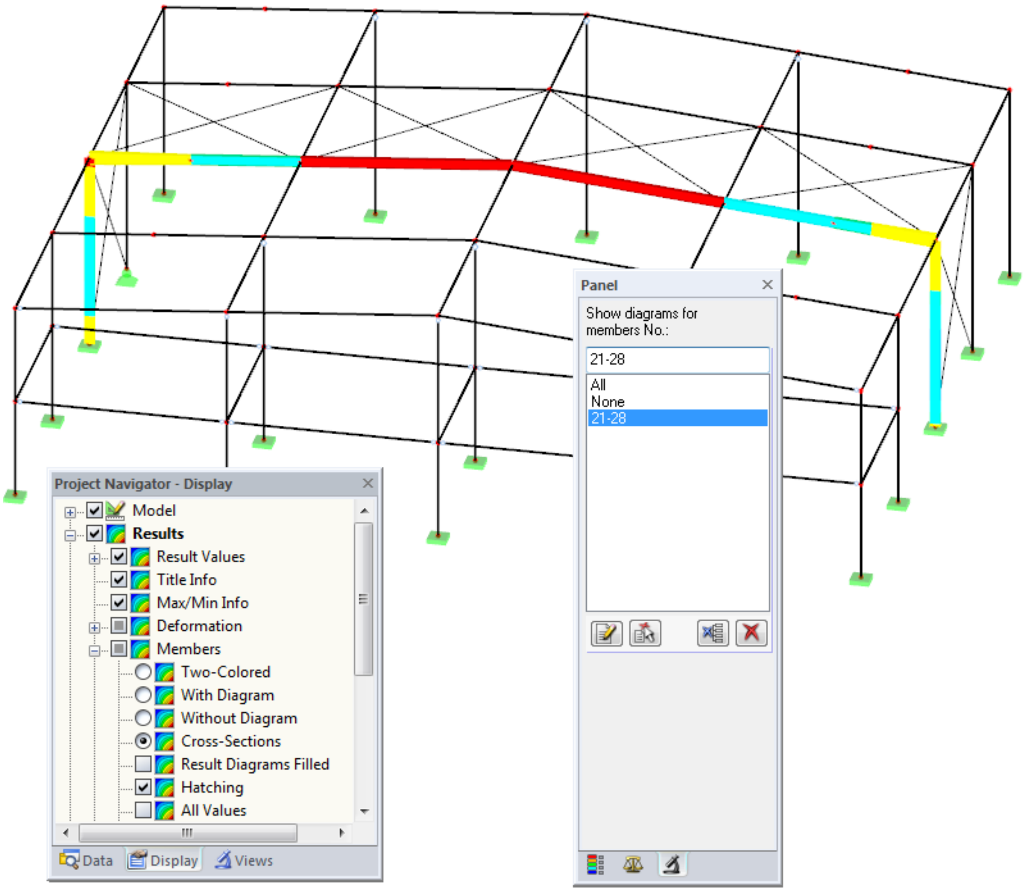


Figure 5.7: Member filter for the stress ratios of a hall frame

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

6. Printout

6.1 Printout Report

Similar to RSTAB, the program generates a printout report for the STEEL AISC results, to which graphics and descriptions can be added. In the printout report, you can select the data from the design module to be included in the printout.

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

For large structural systems with many design cases, it is recommended to split the data into several printout reports, thus allowing for a clearly-arranged printout.

6.2 STEEL AISC Graphic Printout

In RSTAB, every picture that is displayed in the work window can be included in the printout report or send directly to a printer. Thus, the design ratios displayed in the RSTAB model can be prepared for the printout, too.

The printing of graphics is described in the RSTAB manual, chapter 10.2.

Designs in the RSTAB model

To print the currently displayed graphic of the design ratios, click

File → Print Graphic

or use the toolbar button shown on the left.

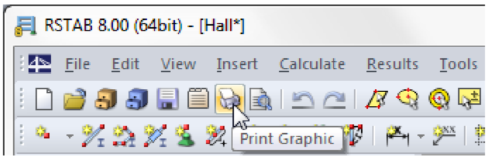


Figure 6.1: Button *Print Graphic* in RSTAB toolbar

Result Diagrams

You can also transfer the *Result Diagram on Member* to the report by using the [Print] button. It is also possible to print it directly.

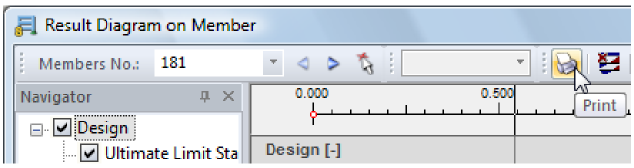


Figure 6.2: Button *Print Graphic* in the dialog box *Result Diagram on Member*

The dialog box *Graphic Printout* opens (see following page).

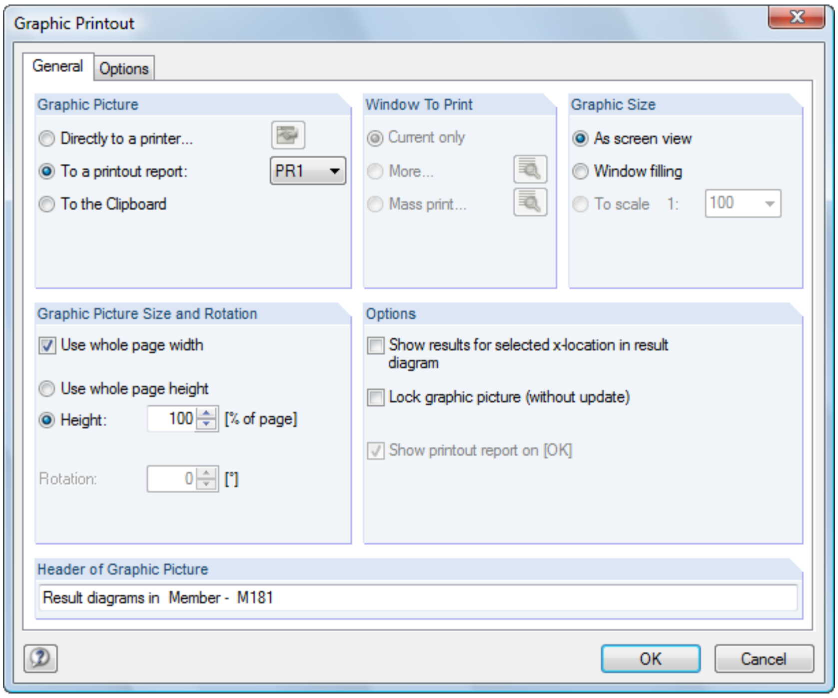
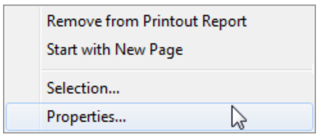


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

This dialog box is described in the RSTAB manual, chapter 10.2. The RSTAB manual also describes the *Options* and *Color Spectrum* tab.

A graphic can be moved anywhere within the printout report by using the drag-and-drop function.



To adjust a graphic subsequently in the printout report, right-click the relevant entry in the navigator of the printout report. The option *Properties* in the context menu opens the dialog box *Graphic Printout*, offering various options for adjustment.

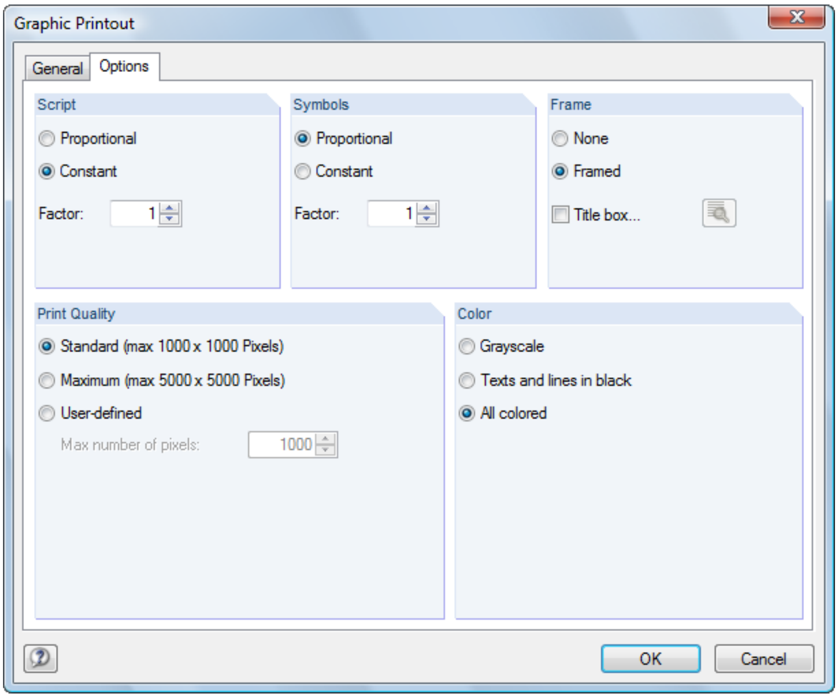


Figure 6.4: Dialog box *Graphic Printout*, tab *Options*

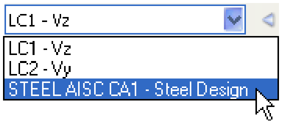
7. General Functions

This chapter describes useful menu functions as well as export options for the designs.

7.1 Design Cases

Design cases allow you to group members for the design: In this way, you can combine groups of structural components or analyze members with particular design specifications (for example changed materials, partial safety factors, optimization).

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



To calculate a STEEL AISC design case, you can also use the load case list in the RSTAB toolbar.

Create New Design Case

To create a new design case, use the STEEL AISC menu and click

File → New Case.

The following dialog box appears:

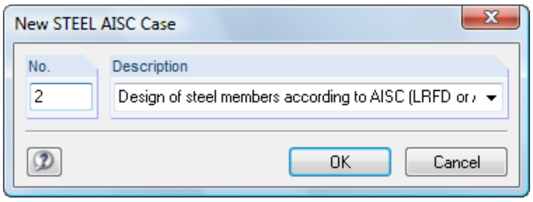


Figure 7.1: Dialog box *New STEEL AISC Case*

In this dialog box, enter a *No.* (one that is still available) for the new design case. The corresponding *Description* will make the selection in the load case list easier.

Click [OK] to open the STEEL AISC table 1.1 *General Data* where you can enter the design data.

Rename a Design Case

To change the description of a design case, use the STEEL AISC menu and click

File → Rename Case.

The following dialog box appears:

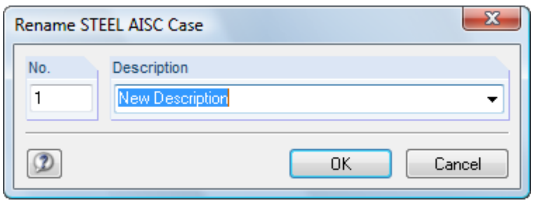


Figure 7.2: Dialog box *Rename STEEL AISC Case*

In this dialog box, you can define a different *Description* as well as a different *No.* for the design case.

Copy a Design Case

To copy the input data of the current design case, use the STEEL AISC menu and click

File → Copy Case.

The following dialog box appears:

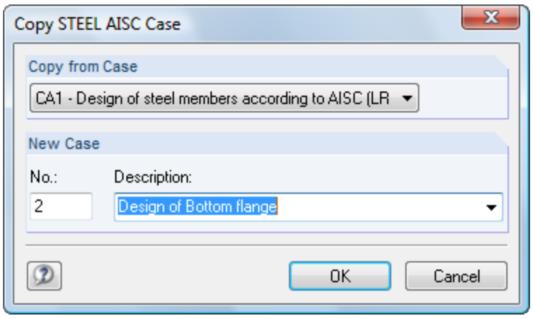


Figure 7.3: Dialog box *Copy STEEL AISC Case*

Define the *No.* and, if necessary, a *Description* for the new case.

Delete a Design Case

To delete design cases, use the STEEL AISC menu and click

File → Delete Case.

The following dialog box appears:

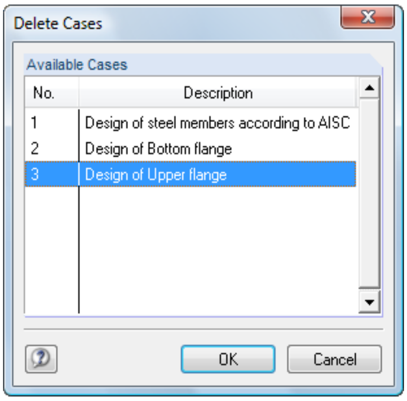


Figure 7.4: Dialog box *Delete Cases*

The design case can be selected in the list *Available Cases*. To delete the selected case, click [OK].

7.2 Cross-Section Optimization

The design module offers you the option to optimize overloaded or little utilized cross-sections. To do this, select in column D of the relevant cross-sections in table 1.3 *Cross-Sections* whether to determine the cross-section *From the current row* or the user-defined *Favorites* (see Figure 2.9, page 14). You can also start the cross-section optimization in the results tables by using the context menu

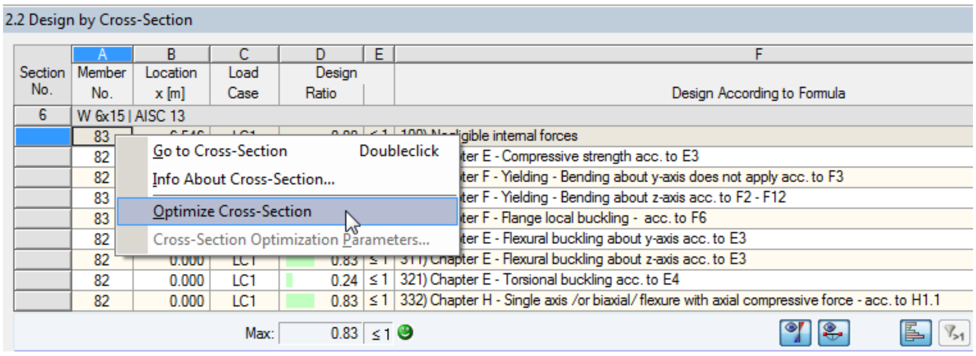


Figure 7.5: Context menu for cross-section optimization

During the optimization process, the module determines the cross-section within the same cross-section table that fulfills the analysis requirements in the most optimal way, that is, comes as close as possible to the maximum allowable stress ratio specified in the *Details* dialog box (see Figure 3.1). The required cross-section properties will be determined with the internal forces from RSTAB. If another cross-section proves to be more favorable, this cross-section will be used for the design. Then, the graphic in table 1.3 will show two cross-sections: the original cross-section from RSTAB and the optimized one (see Figure 7.7).

For a parameterized cross-section, the dialog box *Optimization* appears if you select the corresponding check box.

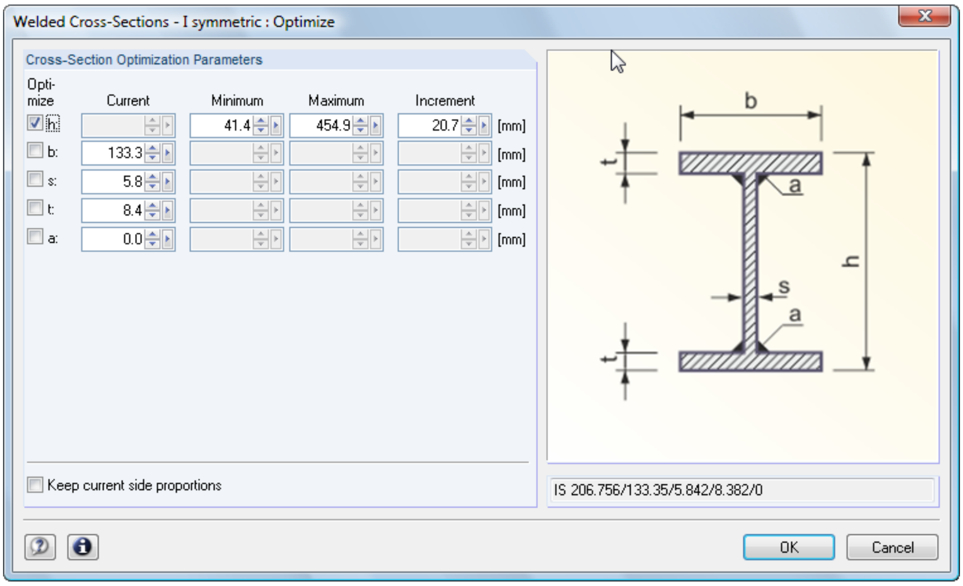


Figure 7.6: Dialog box *Welded Cross-Sections - I symmetric : Optimize*

By selecting the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. This enables the *Minimum* and *Maximum* columns, where you can specify the upper and lower limits of the parameter. The *Increment* column determines the interval in which the size of the parameter varies during the optimization process.

If you want to *Keep current side proportions*, select the corresponding check box. In addition, you have to select at least two parameters for optimization.

Cross-sections based on combined rolled cross-sections cannot be optimized.

Please note for the optimization process that the internal forces will not be recalculated automatically with the changed cross-sections: It is up to you to decide which thicknesses or cross-sections should be transferred to RSTAB for recalculation. As a result of optimized cross-sections, internal forces may vary significantly because of the changed stiffnesses in the structural system. Therefore, it is recommended to recalculate the internal forces of the modified cross-section data after the first optimization, and then to optimize the cross-sections once again.

The modified cross-sections can be exported to RSTAB: Set table 1.3 *Cross-Sections*, and then click

Edit → Export All Cross-Sections to RSTAB.

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

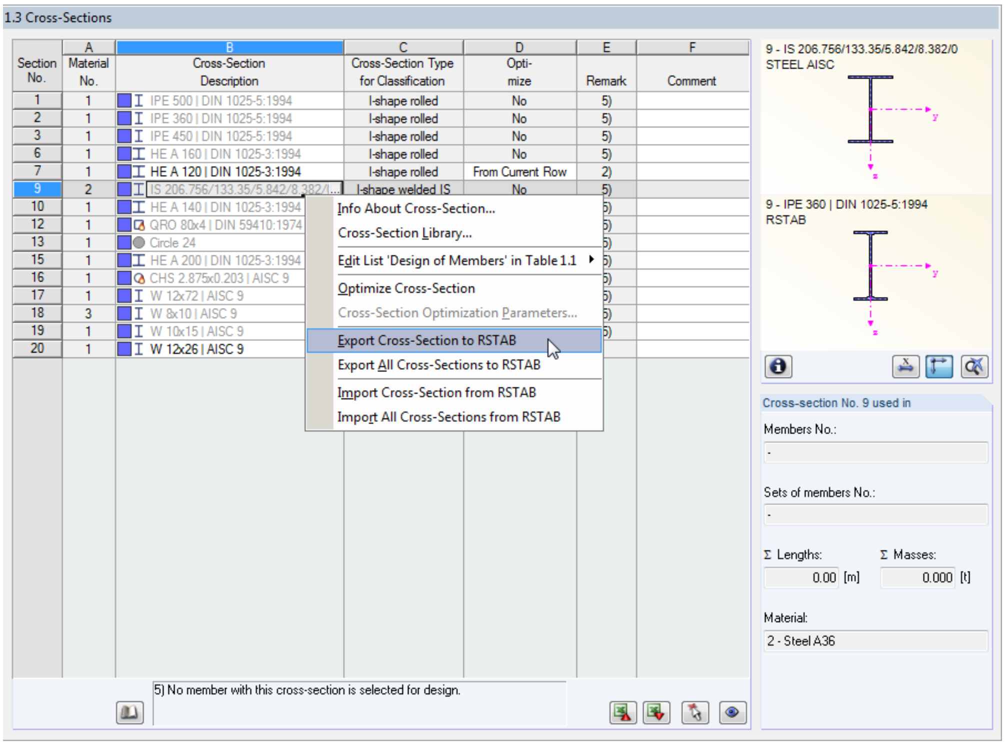


Figure 7.7: Context menu in table 1.3 *Cross-Sections*

Before the modified cross-sections are transferred to RSTAB, a security query appears as to whether the results of RSTAB should be deleted.

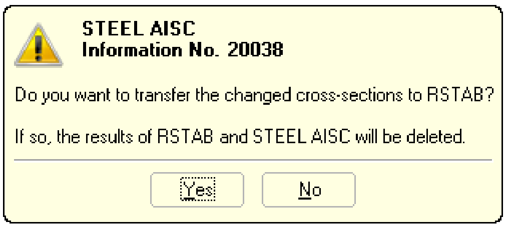


Figure 7.8: Query before transfer of modified cross-sections to RSTAB

By confirming the query and starting the [Calculation] subsequently in the STEEL AISC add-on module, the RSTAB internal forces as well as the designs will be determined in one single calculation run.

If the modified cross-sections have not been exported to RSTAB yet, you can import the original cross-sections in the design module by using the options shown in Figure 7.7. Please note that this option is only available in table 1.3 *Cross-sections*.

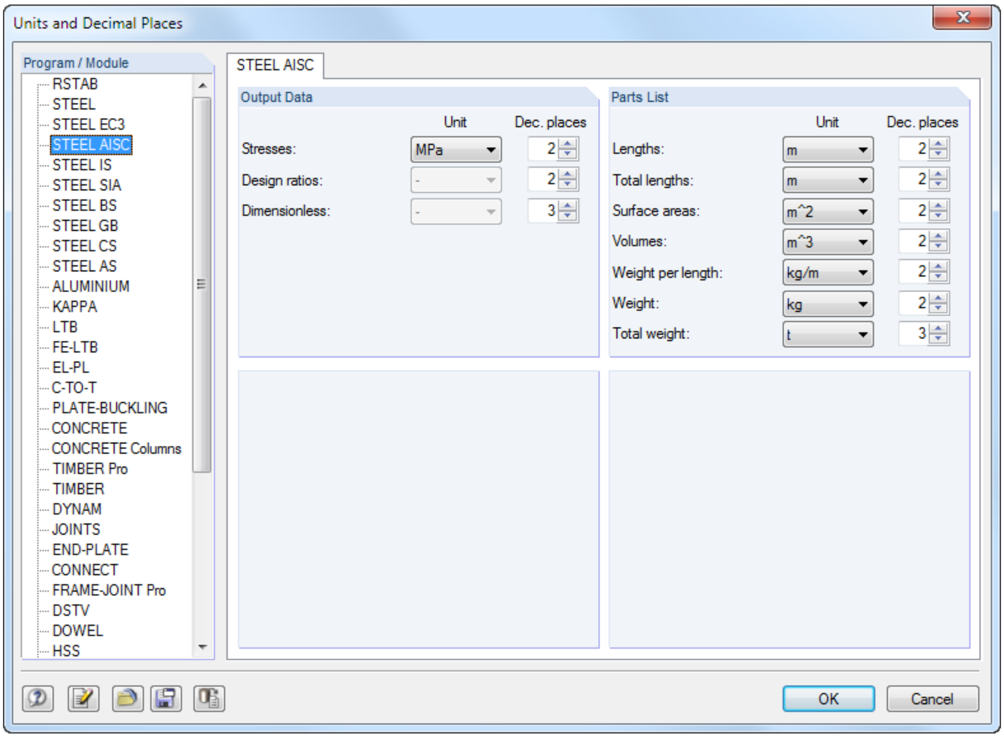
If you optimize a tapered member, the program modifies the member start and end. Then it linearly interpolates the second moments of area for the intermediate locations. As these moments are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-section differ considerably. In such a case, it is recommended to divide the taper into several members, thus manually modeling the taper layout.

7.3 Units and Decimal Places

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

Settings → **Units and Decimal Places**.

The program opens the following dialog box that you already know from RSTAB. STEEL AISC will be preset in the list *Program / Module*.



Dialog box *Units and Decimal Places*

The settings can be saved as user profile to reuse them in other models. These functions are described in the RSTAB manual, chapter 11.1.3.

7.4 Data Transfer

7.4.1 Export Material to RSTAB

If you have adjusted the materials in STEEL AISC for design, you can export the modified materials to RSTAB in a similar manner as you export members and cross-sections: Open table 1.2 *Materials*, and then click

Edit → Export All Materials to RSTAB.

The modified materials can also be exported to RSTAB by using the context menu of table 1.2.

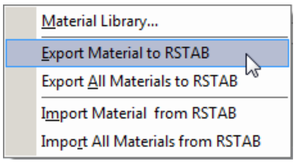


Figure 7.9: Context menu of table 1.2 *Materials*

Before the modified materials are transferred to RSTAB, a security query appears as to whether the results of RSTAB should be deleted. After you confirm the query and start the [Calculation] subsequently in STEEL AISC, the RSTAB internal forces and designs will be determined in one single calculation run.

If the modified materials have not been exported to RSTAB yet, you can transfer the original materials to the design module, using the options shown in Figure 7.9. Please note, however, that this option is only available in table 1.2 *Materials*.

7.4.2 Export Effective Lengths to RSTAB

If you have adjusted the materials in STEEL AISC for design, you can export the modified materials to RSTAB in a similar manner as you export cross-sections: Open table 1.5 *Effective Lengths - Members*, and then click

Edit → Export All Effective Lengths to RSTAB.

The modified effective lengths can also be exported to RSTAB via the context menu of table 1.5.

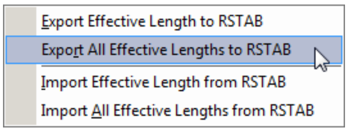


Figure 7.10: Context menu of table 1.5 *Effective Lengths - Members*

Before the modified materials are transferred to RSTAB, a security query appears as to whether the results of RSTAB should be deleted.

If the modified effective lengths have not been exported to RSTAB yet, you can retransfer the original effective lengths to the design module, using the options shown in Figure 7.10. Please note, however, that this option is only available in table 1.5 *Effective Lengths - Members* and 1.6 *Effective Lengths - Sets of Members*.

7.4.3 Export Results

The STEEL AISC results can also be used by other programs.

Clipboard

To copy cells selected in the results tables to the Clipboard, press the keys [Ctrl]+[C]. To insert the cells, for example in a word-processing program, press [Ctrl]+[V]. The headers of the table columns will not be transferred.

Printout report

The data of the STEEL AISC add-on module can be printed into the global printout report (see chapter 6.1, page 47) for export. Then, in the printout report, click

File → Export to RTF.

The function is described in the RSTAB manual, 10.1.11.

Excel / OpenOffice

STEEL AISC provides a function for the direct data export to MS Excel, OpenOffice.org Calc or the file format CSV. To open the corresponding dialog box, click

File → Export Tables.

The following export dialog box appears

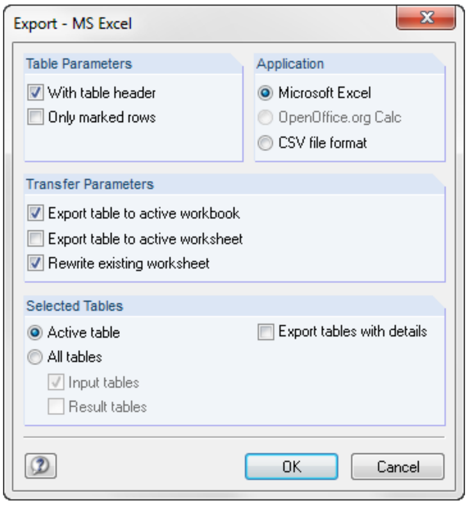


Figure 7.11: Dialog box *Export - MS Excel*

When you have selected the relevant parameters, you can start the export by clicking [OK]. Excel or OpenOffice will be started automatically, that is, the programs do not have to be opened first.

	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

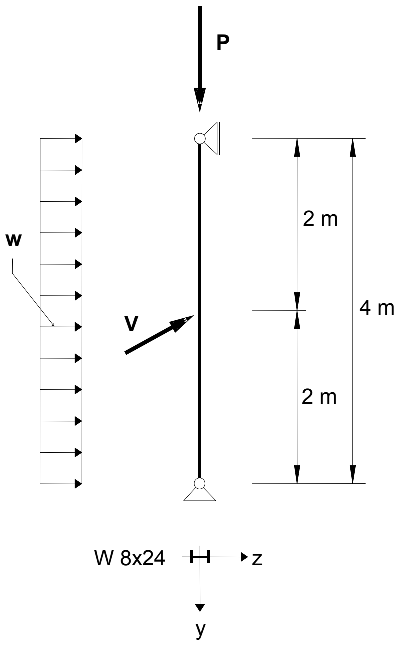
8. Example

Column with Biaxial Bending

In this example, the 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

System 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: System and design loads (γ-fold)

Internal forces according to linear static analysis

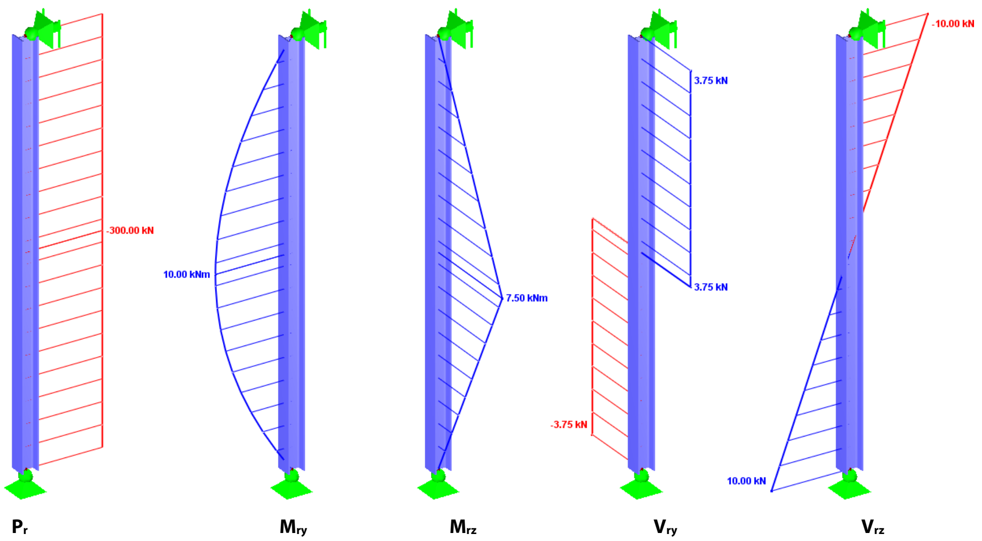


Figure 8.2: Internal Forces

Design location (decisive location x)

The design is performed for all locations x (see chapter 4.5) of the 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

Property	Symbol	Value	Unit
Gross area of cross-section	A_g	45.677	cm²
Moment of inertia	I_y	3446.40	cm⁴
Moment of inertia	I_z	761.70	cm⁴
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	cm⁴
Warping moment of inertia	C_w	69550.80	cm⁶
Elastic cross-section modulus	S_y	342.49	cm³
Elastic cross-section modulus	S_z	92.26	cm³
Plastic cross-section modulus	Z_y	380.18	cm³
Plastic cross-section modulus	Z_z	140.44	cm³

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

$$\eta_z = \frac{P_r}{\phi_c \cdot P_{n,z}} = \frac{300}{0.9 \cdot 684.12} = 0.487$$ **- OK, 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 $\frac{K_y L}{r_y} \le 4.71\sqrt{\frac{E}{F_y}}$

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

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

$$P_{n,y} = F_{cr,y} \cdot A_g = 221.981 \cdot 4567.7 = 1013\,950\,\text{N} = 1013.95\,\text{kN}$$

Design compressive strength

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

Design ratio

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

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

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

2. Lateral-torsional buckling

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

Calculate C_b:

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

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

$$C_b = 1.136 \le 3.0$$

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 $M_{n,y}$:

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

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

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

Design flexural strength

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

Design ratio

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

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

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

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

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:

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

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

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

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

$0.805 \leq 1.0$ **- OK**

A Literature

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

[2] Specification for Structural Steel Buildings ANSI/AISC 360-10, U.S. Standard, June 22, 2010

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

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

B Index