Version
December 2018

Add-on Module

RF-/STEEL AISC

Design of Steel Members
According to ANSI/AISC 360

Program Description

The Graham Building
30 South 15th Street
15th Floor
Philadelphia, PA 19102
USA

Tel.: +1 267 702-2815
E-Mail: info-us@dlubal.com
Web: www.dlubal.com

Contents

1 Introduction

1.1 Add-on Module RF-/STEEL AISC

The American Institute of Steel Construction's *Specification for Structural Steel Buildings* [1] describes the design and construction of steel structures in the U.S. In this Specification, the design according to the provisions for load and resistance factor design (LRFD) and to the provisions for allowable strength design (ASD) is regulated. Both types of design can be performed with the powerful add-on modules RF-STEEL AISC (for RFEM) and STEEL AISC (for RSTAB).

In the following, the modules of the two main programs are described in one manual and are referred to as **RF-/STEEL AISC**.

RF-/STEEL AISC performs all typical designs of strength and stability, as well as for the deflections. The program takes into account various load combinations for the allowable strength design or for the load and resistance factor design. Classifying sections for local buckling into the appropriate types (slender, nonslender, compact, noncompact) is important for the design of members subject to axial compression and/or flexure. By this classification, the effects are accounted for where local buckling reduces the load capacity. RF-/STEEL AISC determines the limiting width-to-thickness ratios of compressed parts and carries out the classification automatically.

For the stability analysis in RF-/STEEL AISC, the user can determine for every single member or set of members whether buckling and torsional buckling is possible. Furthermore, the effective lengths can be modified when designing per the effective length method. Alternatively, the direct analysis method can be considered with member or set of member stiffness reductions in RFEM or RSTAB. Lateral restraints of members allow for a realistic representation of the structural model.

For models with slender cross-sections, the serviceability limit state has become an essential aspect of the design. The deflection limits are preset according to the Specification, but can be modified. If necessary, the reference lengths and precambers of members can be adjusted as well.

The program features an option to optimize sections and to export them to RFEM or RSTAB. By means of design cases, it is possible to design separate structural components of complex models or to analyze alternatives with different sections or materials.

Since RF-/STEEL AISC is integrated in the main program, the design relevant input data is preset when the module is opened. After the analysis, the design results can be evaluated graphically in the work window of RFEM or RSTAB. Last but not least, it is possible to keep records of the analysis in the global printout report which includes the internal forces and the design results.

We hope you will enjoy working with RF-/STEEL AISC.

Your DLUBAL team

1.2 Using the Manual

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

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

At the end of the manual, you can find the index. If you cannot find what you are looking for, go to the Knowledge Base where you can search for the solution of the problem. Or consult the FAQs on our website. Moreover, the design according to AISC is demonstrated in a recorded Webinar.

1.3 Starting RF-/STEEL AISC

RFEM and RSTAB provide the following options to open the RF-/STEEL AISC add-on module.

Menu

To start the program from the menu bar, select

Add-on Modules → **Design - Steel** → **RF-/STEEL AISC**.

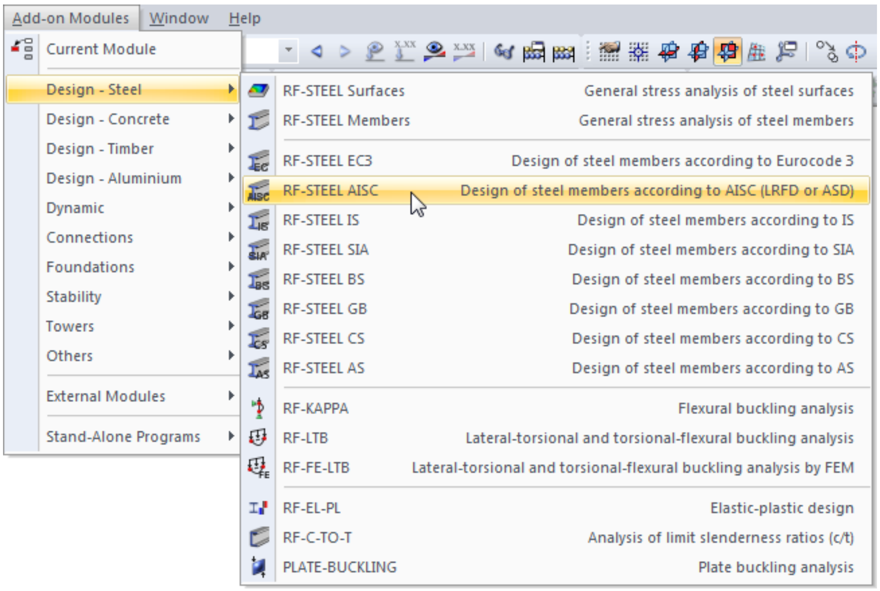


Figure 1.1: Menu *Add-on Modules* → *Design - Steel* → *RF-STEEL AISC*

Navigator

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

Add-on Modules → **RF-/STEEL AISC**.

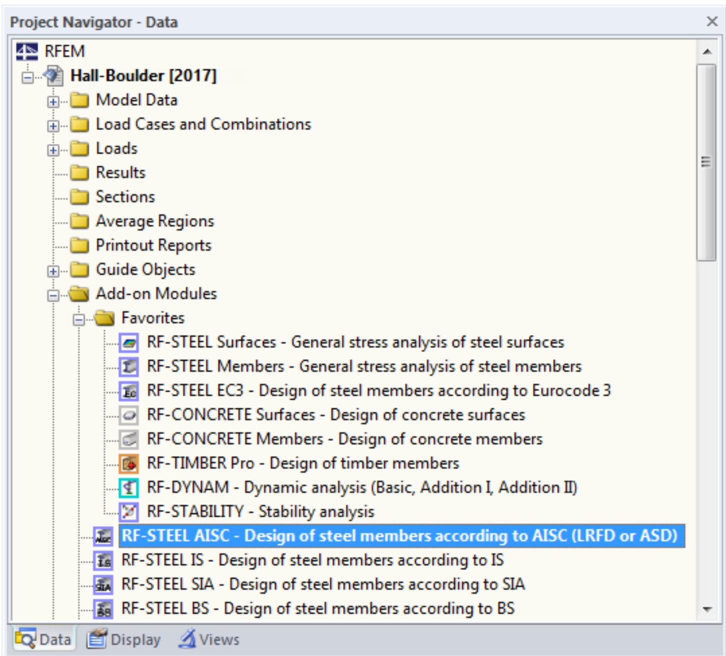


Figure 1.2: *Data* navigator *Add-on Modules* → *RF-STEEL AISC*

2 Input Data

When you have started the add-on module, a new window appears. In this window, a *Navigator* is displayed on the left. It manages the different window that can be currently selected. The drop-down list above the navigator contains the design cases (see Chapter 7.1, page 55).

The design-relevant data is to be defined in several input windows. When you open RF-/STEEL AISC for the first time, the following parameters are imported automatically:

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

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

[OK] saves the results. Thus, you quit RF-/STEEL AISC and return to RFEM or RSTAB. To exit the add-on module without saving any changes, click [Cancel].

2.1 General Data

In the *1.1 General Data* Window, you can select the objects and the loading relevant for the design. The two tabs manage the load cases, load combinations, and result combinations for the different types of design.

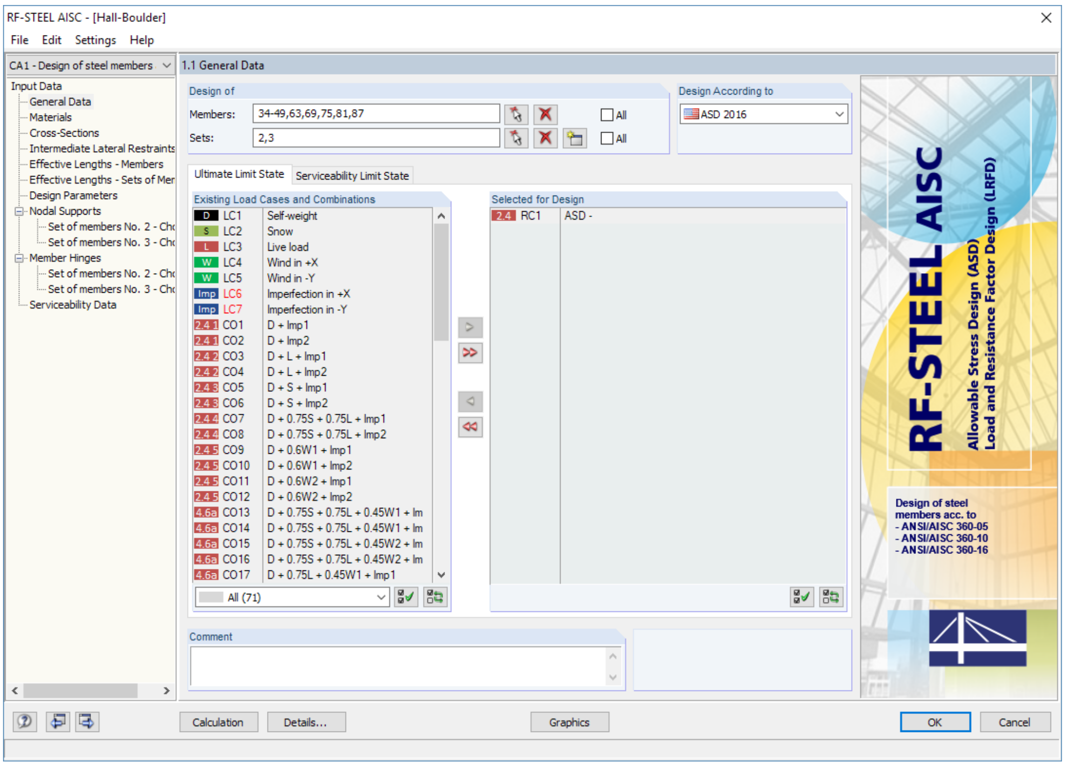


Figure 2.1: Window *1.1 General Data*

Design of

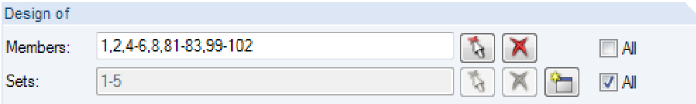


Figure 2.2: Design of members and sets of members

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 text boxes and enter the numbers of the relevant members or sets of members. The [Delete] button clears the list of preset numbers. The [Select] button enables you to define the objects graphically in the work window of RFEM or RSTAB.

When you design a set of members, the program determines the extreme values of the analyses 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 Windows *2.3 Designs 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 RFEM or RSTAB appears. There you can specify the parameters of the set of members.

Design According to

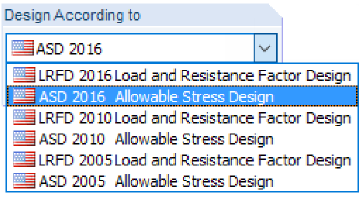


Figure 2.3: Design options

The list box options control whether the design is performed according to the provisions for load and resistance factor design (LRFD) or to the provisions for allowable strength design (ASD).

There are design methods available for the current version of ANSI/AISC 360-16 [1]. If required, one of the previous versions of this Specification dated from 2010 or 2005 can be selected for the design.

Comment

In this text box, you can enter notes, for example to describe the selected design method or the current design case.

2.1.1 Ultimate Limit State

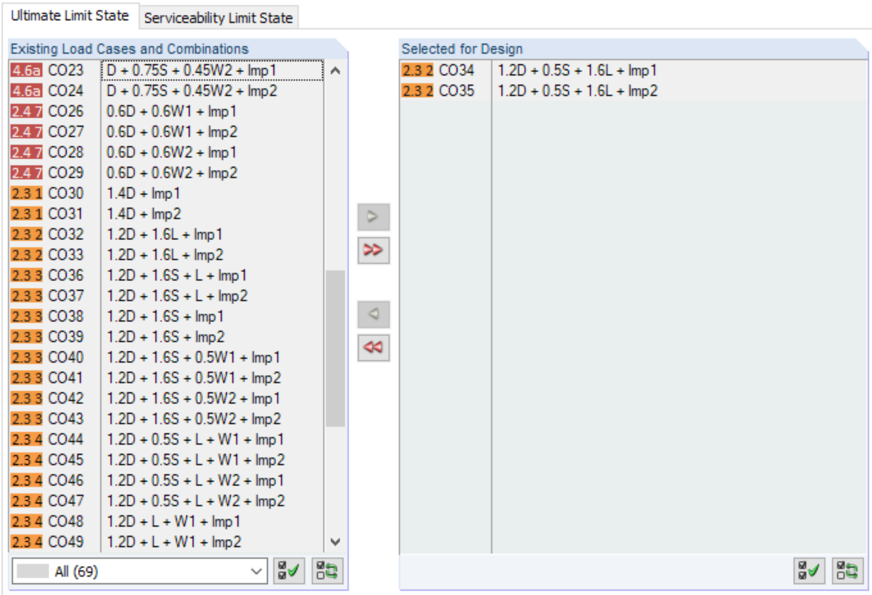


Figure 2.4: Window *1.1 General Data*, tab *Ultimate Limit State*

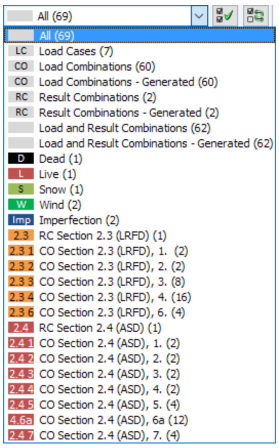
Existing Load Cases and Combinations

This column lists all load cases, load combinations, and result combinations that have been created in RFEM or RSTAB.

To transfer selected items to the *Selected for Design* list on the right, click [>]. Alternatively, you can double-click the items. To transfer the complete list to the right, click [>>].

To select several items at once, click them while pressing the [Ctrl] key – as common for Windows applications.

If a load case is highlighted in red, it cannot be designed. This happens when a load case has no loads or if it contains only imperfections (see Figure 2.5).



At the end of the list, several filter options are available. They help you to assign the items by load case, load combination, or action category (e.g. LRFD, ASD). The buttons next to the text box have the following functions:

[Select all icon]	Selects all load cases in the list
[Invert icon]	Inverts the selection of load cases

Table 2.1: Buttons in *Ultimate Limit State* tab

Selected for Design

The column on the right lists the load cases, load and result combinations selected for design. To remove an item from the list, select it and click [<]. You can also double-click the item. To transfer the entire list to the left, click [<<].

Result combination

The design of an enveloping max/min result combination, *RC*, is faster than the design of all contained load cases and load combinations. However, the influence of the contained actions is difficult to check afterwards.

2.1.2 Serviceability Limit State

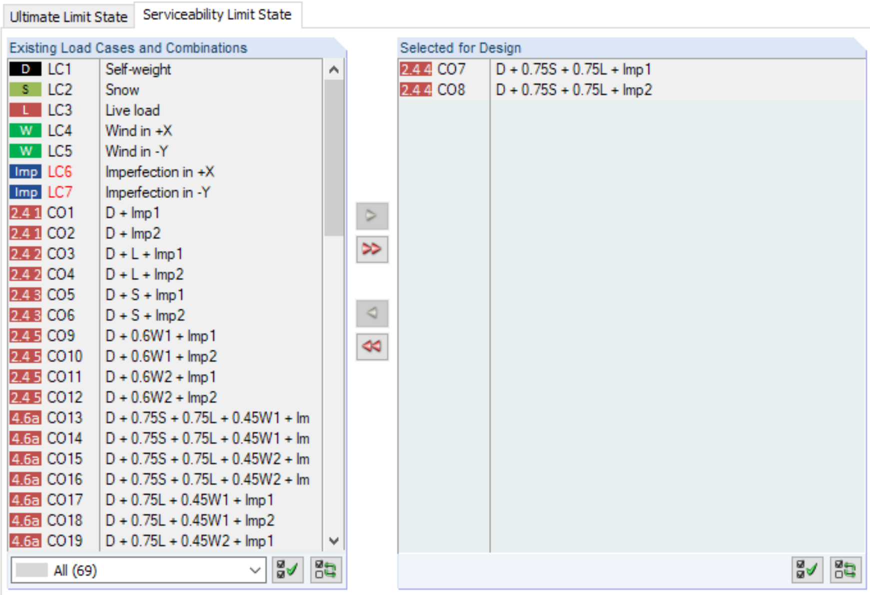


Figure 2.5: Window *1.1 General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

This section lists all load cases, load combinations, and result combinations that have been created in RFEM or RSTAB.

Selected for Design

You can add or remove load cases, load and result combinations as described in Chapter 2.1.1.

The limit values of the deflections are preset in the *Details* dialog box (see Figure 3.3, page 31). To adjust those values, click the [Details] button.

In the *1.11 Serviceability Data* Window, you can specify the reference lengths of the deflection analysis (see Chapter 2.11, page 27).

2.2 Materials

This window consists of two parts: The upper table lists all materials created in RFEM or RSTAB. The *Material Properties* section below shows the characteristics of the current material, i.e. the table row which is selected above.

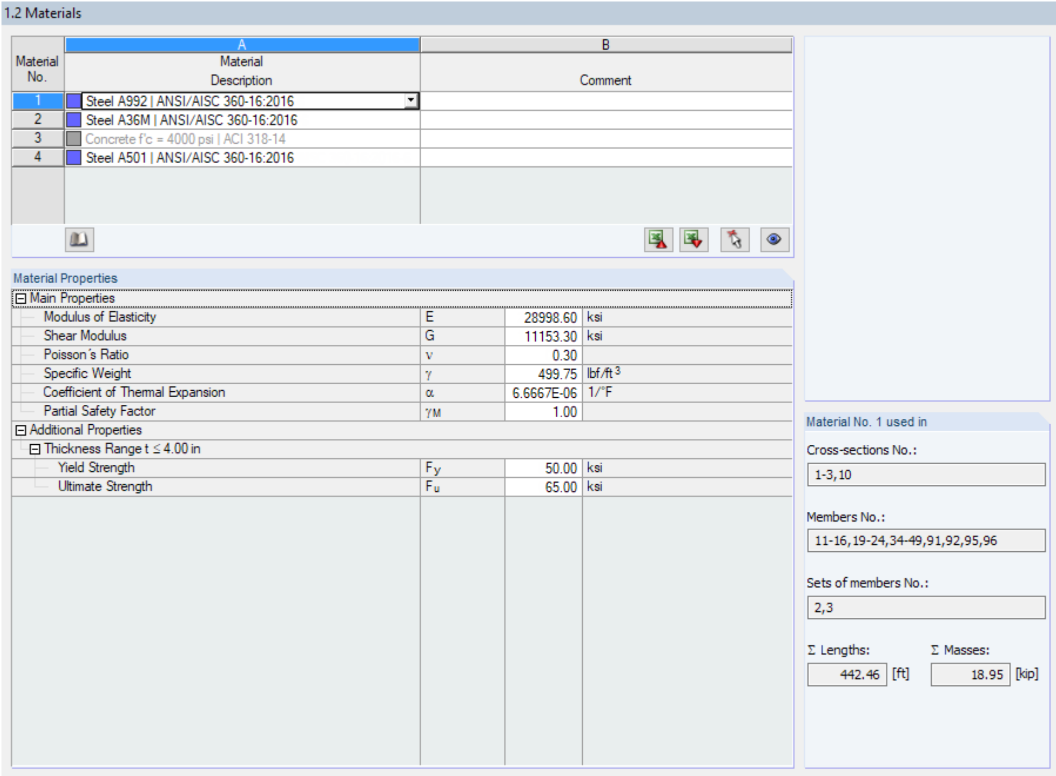


Figure 2.6: Window *1.2 Materials*

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

The material properties required to determine the internal forces (*Main Properties*) are described in Chapter 4.3 of the RFEM manual and Chapter 4.2 of the RFEM manual. The material properties required for design (*Additional Properties*) are stored in the global material library. These values are preset.

To adjust the units and decimal places of the material properties and stresses, select **Settings** → **Units and Decimal Places** from the menu bar of the module (see Chapter 7.3, page 59).

Material Description

The materials defined in RFEM or RSTAB are preset, but you can always modify them: Click the material in column A, and then click the button or press the function key [F7] to open the material list.

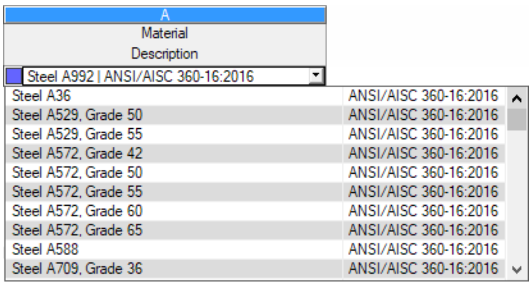


Figure 2.7: List of materials

According to the design concept of the Specification [1], only materials of the ASTM *Steel* category are available in the list.

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

As a matter of principle, the material properties cannot be edited in the RF-/STEEL AISC module.

Material Library

Alternatively, you can use the material library to change a material. To open the library, select

Edit → Material Library

or use the [Library] button.

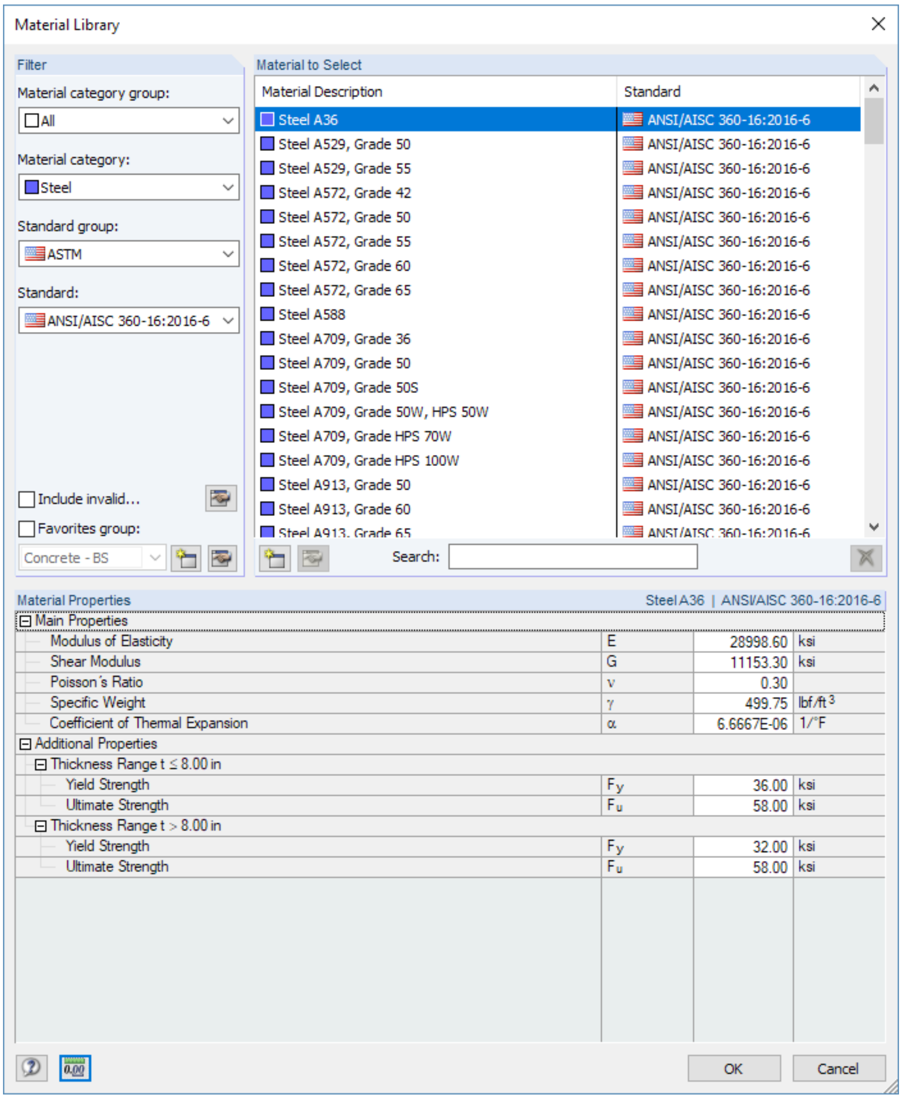


Figure 2.8: Dialog box *Material Library*

In the *Filter* section, *ANSI/AISC 360-16* is the default Standard. Select the material grade that you want to use for the design in the *Material to Select* list. You can check the corresponding properties in the dialog section below.

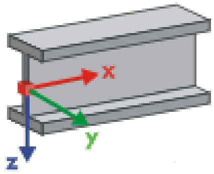
Click [OK] or press [↵] to transfer the selected material to Window 1.2 of RF-/STEEL AISC.

Chapter 4.3 of the RFEM manual and Chapter 4.2 of the RSTAB manual describe in detail how materials can be filtered, added, or rearranged.

In the library, you can also select materials of categories *Cast Iron* and *Stainless Steel*. Be aware that those materials are not covered by the design concept of the Specification [1].

2.3 Cross-Sections

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



Coordinate System

The sectional coordinate system **yz** of RF-/STEEL AISC corresponds to the one of RFEM or RSTAB: The **y**-axis represents the strong principal axis of the cross-section, the **z**-axis is its weak axis. This coordinate system is used for both the input data and the results.

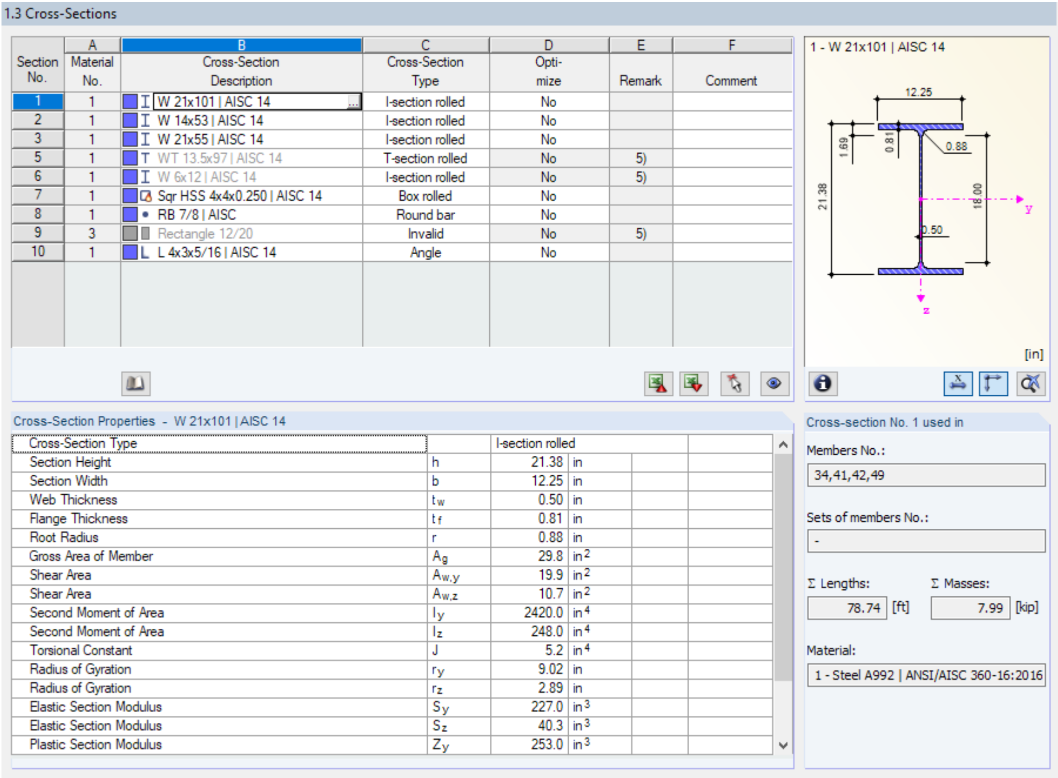
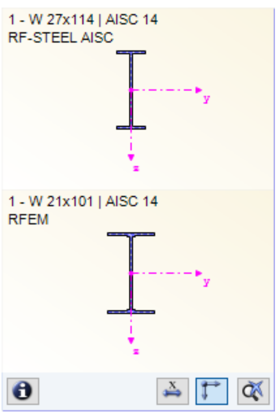


Figure 2.9: Window *1.3 Cross-Sections*

Cross-Section Description

The cross-sections defined in RFEM or RSTAB are preset with their material numbers.

If you want to modify a cross-section, select the entry in column B. Click the button or in the box, or press the function key [F7] to open the cross-section table of the current cross-section type (see Figure 2.10).



In this dialog box, you can select a different cross-section. To select a different section category, click to access the global library of cross-sections.

Chapter 4.13 of the RFEM manual and Chapter 4.3 of the RSTAB manual describe how sections can be selected from the library.

You can directly enter the new cross-section description in the text box. If the entry is listed in the database, RF-/STEEL AISC imports the cross-section parameters.

A modified cross-section will be highlighted in blue.

If the cross-section in RF-/STEEL AISC is different from the one of RFEM or RSTAB, both sections are displayed in the graphic area. The designs will then be performed with the internal forces of RFEM/RSTAB for the section defined in RF-/STEEL AISC.

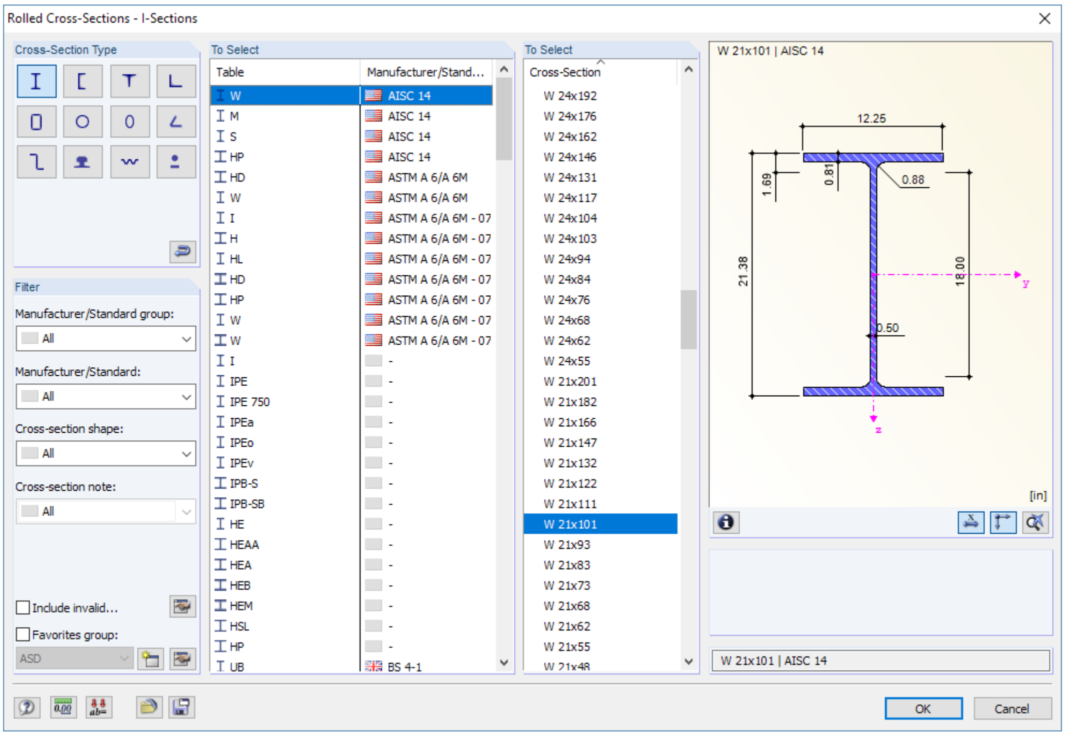


Figure 2.10: Rolled I-sections in cross-section library

Cross-Section Type
I-section rolled
I-section rolled
I-section rolled
I-section rolled
I-section rolled
Box rolled
Round bar
Invalid
Angle

Cross-Section Type

The program displays the type of cross-section that will be used for the classification according to [1] Section B4.

Max. Design Ratio

This column is shown only after the calculation. It is useful for the optimization: By means of the design ratios and colored relation scales, you can see which cross-sections are little utilized and thereby oversized, and overloaded and for this reason undersized.

Optimize

Details...

It is possible to optimize every cross-section from the library. The program searches the cross-section that comes as close as possible to a user-defined maximum utilization ratio. You can specify this maximum ratio in the *Details* dialog box (see Figure 3.5, page 34).

To optimize a cross-section, open the drop-down list in column D (resp. E) and select *From current row*. Recommendations on the optimization can be found in Chapter 7.2 on page 57.

Remark

This column shows remarks as footers. They are explained below the cross-section list.

Member with Tapered Cross-Section

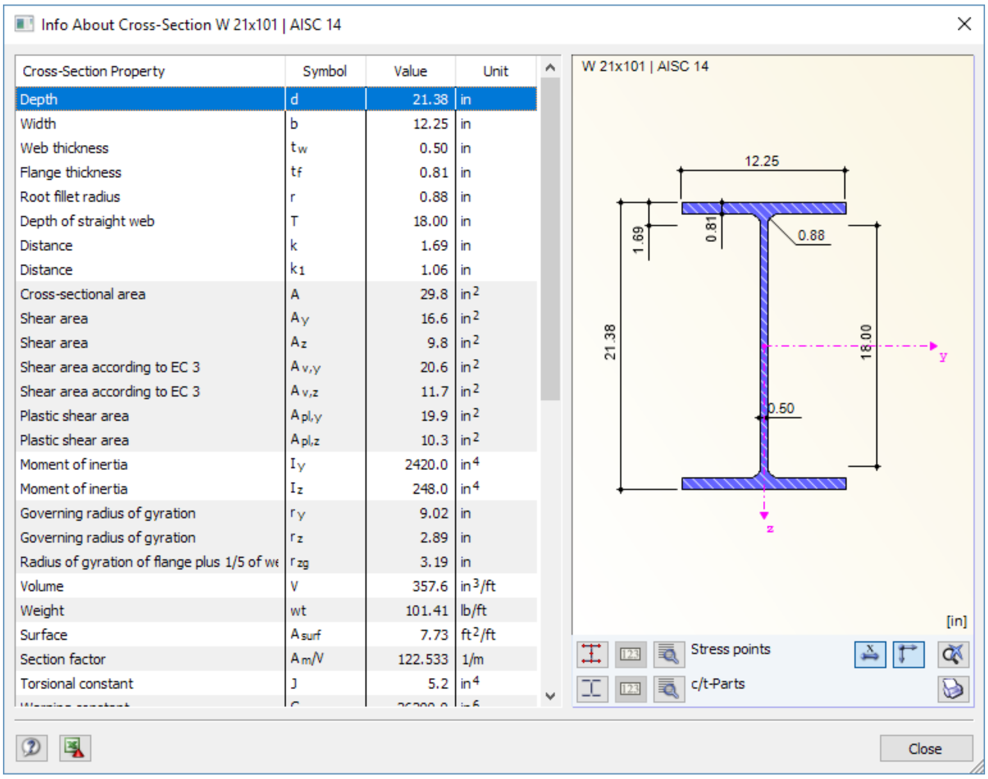
For tapered members with different cross-sections at the member start and member ends, the module displays both section numbers in two rows, according to the definition in RFEM or RSTAB.

RF-/STEEL AISC also designs tapered members, provided that the section at the member start has the same number of stress points as the section at the member end. If the two sections have different numbers of stress points, the intermediate values cannot be interpolated. In this case, the calculation is neither possible in RFEM/RSTAB nor in RF-/STEEL AISC.

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

Info About Cross-Section

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

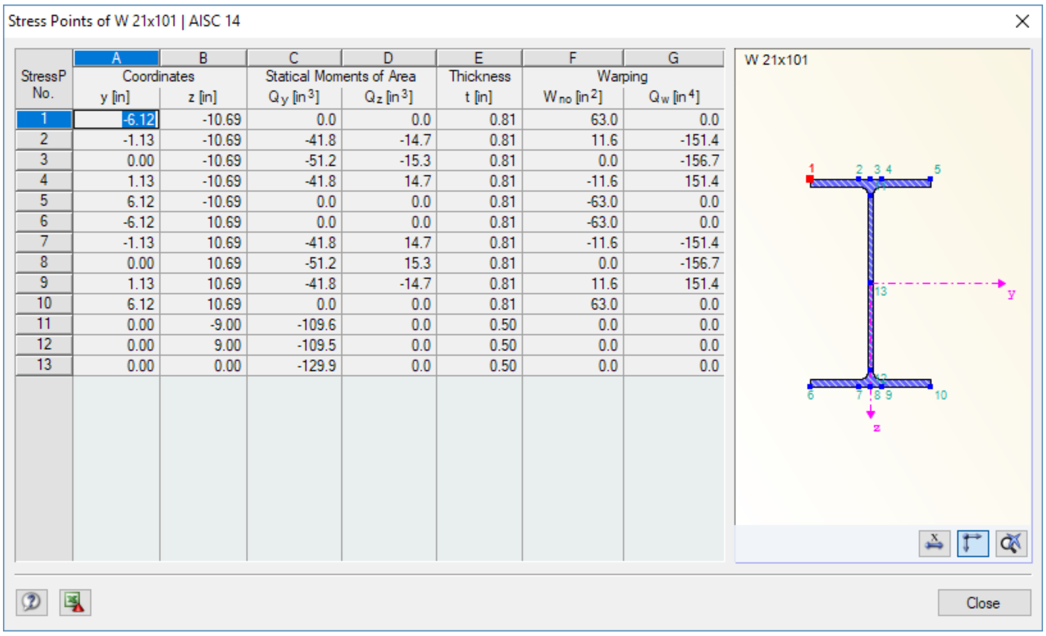


Cross-Section Property	Symbol	Value	Unit
Depth	d	21.38	in
Width	b	12.25	in
Web thickness	t_w	0.50	in
Flange thickness	t_f	0.81	in
Root fillet radius	r	0.88	in
Depth of straight web	T	18.00	in
Distance	k	1.69	in
Distance	k_1	1.06	in
Cross-sectional area	A	29.8	in^2
Shear area	A_y	16.6	in^2
Shear area	A_z	9.8	in^2
Shear area according to EC 3	$A_{v,y}$	20.6	in^2
Shear area according to EC 3	$A_{v,z}$	11.7	in^2
Plastic shear area	$A_{pl,y}$	19.9	in^2
Plastic shear area	$A_{pl,z}$	10.3	in^2
Moment of inertia	I_y	2420.0	in^4
Moment of inertia	I_z	248.0	in^4
Governing radius of gyration	r_y	9.02	in
Governing radius of gyration	r_z	2.89	in
Radius of gyration of flange plus 1/5 of we	r_{zg}	3.19	in
Volume	V	357.6	in^3/ft
Weight	wt	101.41	lb/ft
Surface	A_{surf}	7.73	ft^2/ft
Section factor	A_m/V	122.533	1/m
Torsional constant	J	5.2	in^4

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

The buttons below the cross-section graphic have the following functions:

Click the [button] buttons find detailed information on the *Stress points* (centroidal distances, statical moments of area, warping constants etc.) or *c/t-Parts*, respectively.



StressP No.	A Coordinates y [in]	B Coordinates z [in]	C Statical Moments of Area Q_y [in^3]	D Statical Moments of Area Q_z [in^3]	E Thickness t [in]	F Warping W_{no} [in^2]	G Warping Q_w [in^4]
1	-6.12	-10.69	0.0	0.0	0.81	63.0	0.0
2	-1.13	-10.69	-41.8	-14.7	0.81	11.6	-151.4
3	0.00	-10.69	-51.2	-15.3	0.81	0.0	-156.7
4	1.13	-10.69	-41.8	14.7	0.81	-11.6	151.4
5	6.12	-10.69	0.0	0.0	0.81	-63.0	0.0
6	-6.12	10.69	0.0	0.0	0.81	-63.0	0.0
7	-1.13	10.69	-41.8	14.7	0.81	-11.6	-151.4
8	0.00	10.69	-51.2	15.3	0.81	0.0	-156.7
9	1.13	10.69	-41.8	-14.7	0.81	11.6	151.4
10	6.12	10.69	0.0	0.0	0.81	63.0	0.0
11	0.00	-9.00	-109.6	0.0	0.50	0.0	0.0
12	0.00	9.00	-109.5	0.0	0.50	0.0	0.0
13	0.00	0.00	-129.9	0.0	0.50	0.0	0.0

Figure 2.12: Dialog box *Stress Points of W 21x101*

2.4 Intermediate Lateral Restraints

In Window 1.4, you can define lateral restraints for members. In RF-/STEEL AISC, this kind of support acts perpendicular to the z-axis of the cross-section (the minor axis, see Figure 2.11). Thus, you can manipulate the effective lengths of the members for the stability design concerning flexural buckling and lateral-torsional buckling.

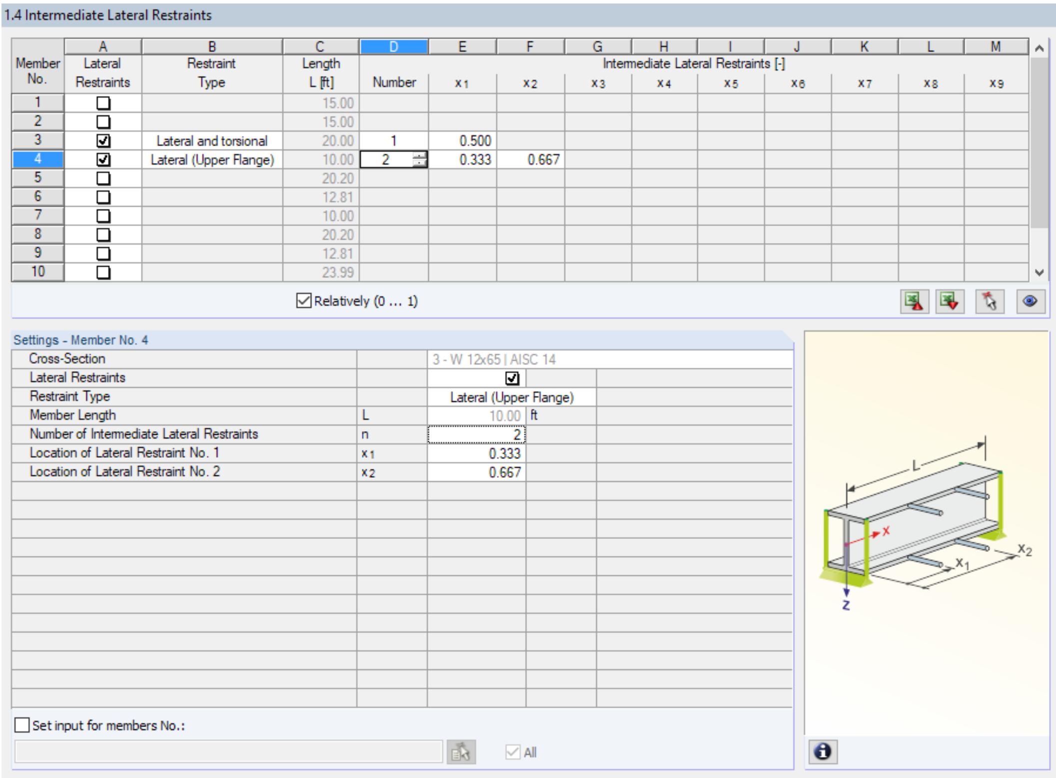
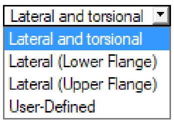


Figure 2.13: Window *1.4 Intermediate Lateral Restraints*

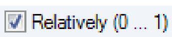
In the upper table, you can assign up to nine lateral restraints to each member. The *Settings* section below shows a column overview for the member selected above.

To define the restraints of a specific member, select the *Lateral Restraints* check box in column A. Then the other columns will be accessible where you can enter the parameters. To graphically select the member, click .



In column B, you can select the *Restraint Type* from the drop-down list. A lateral and torsional restraint is preset. It is also possible to place intermediate restraints at the lower or upper flanges. The *User-Defined* option allows you to specify the restraint parameters individually (support in direction of member axis y, restrained about longitudinal member axis x, eccentricity of support) in the *Settings* section.

In column D, you can specify the *Number* of the intermediate restraints. Depending on the specification, one or more of the following *Intermediate Lateral Restraints* columns will be available for the definition of the x-locations.



When the *Relatively (0 ... 1)* check box is activated, you can define the support points by their relative spacings. The positions of the intermediate restraints result from the member length and the relative distances from the member start. When you clear the *Relatively (0 ... 1)* check box, you can define the absolute distances.

2.5 Effective Lengths - Members

This window controls the effective lengths which are relevant when designing per the effective length method. This approach is described in [1] Appendix 7. Alternatively, the direct analysis method can be considered by reducing member stiffnesses in RFEM or RSTAB (see article in the Knowledge Base on our website).

Window 1.5 consists of two parts. The upper table presents a summary of all length factors of buckling, torsional buckling, and lateral-torsional buckling as well as the respective member lengths. The effective lengths defined in RFEM or RSTAB are preset. In the *Settings* section, additional information about the member selected in the upper table is given.

You can make any changes in the upper table as well as in the *Settings* tree.

Click to select a member graphically and show its row.

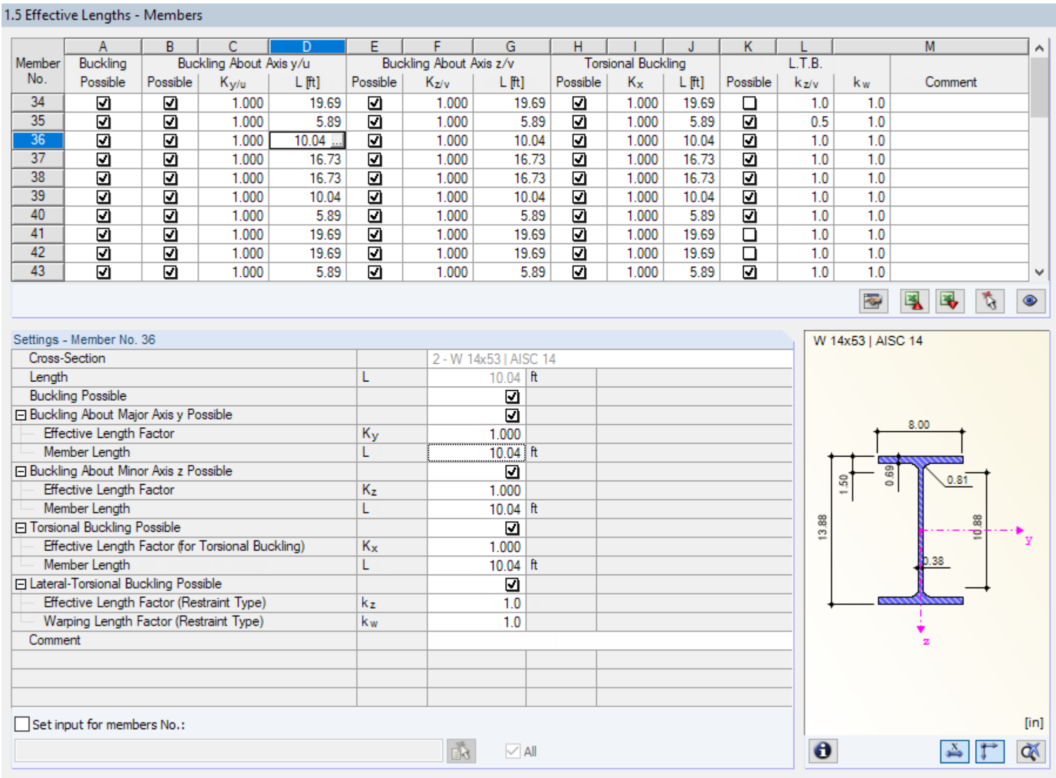


Figure 2.14: Window *1.5 Effective lengths - Members*

The effective lengths for buckling about the weak z-axis and torsional as well as lateral-torsional buckling are aligned automatically with the settings of Window *1.4 Intermediate Lateral Restraints* (see Chapter 2.4). If intermediate lateral restraints divide the member into segments of different lengths, no values are displayed in the table columns G, J, and L.

You can enter the effective lengths manually in the table and in the *Settings* tree, or define them graphically in the work window by clicking the button. The button is active when you place the cursor in the text box (see Figure 2.14).

The *Settings* tree includes the following parameters:

- *Cross-Section*
- *Length* of the member
- *Buckling possible* for the member (cf. column A)
- *Buckling about Major Axis y Possible* (cf. columns B to D)
- *Buckling about Minor Axis z Possible* (cf. columns E to G)

- *Torsional Buckling Possible* (cf. columns H to J)
- *Lateral-Torsional Buckling Possible* (cf. columns K and L)

The table controls for which members an analysis of buckling, torsional or lateral-torsional buckling is to be performed. In addition, the *Effective Length Factor* can be adjusted for the respective designs. If you modify the factor, the equivalent member length is adjusted automatically, and vice versa.

You can also define the effective length of a member in a separate dialog box. To open it, click the button below the upper table.

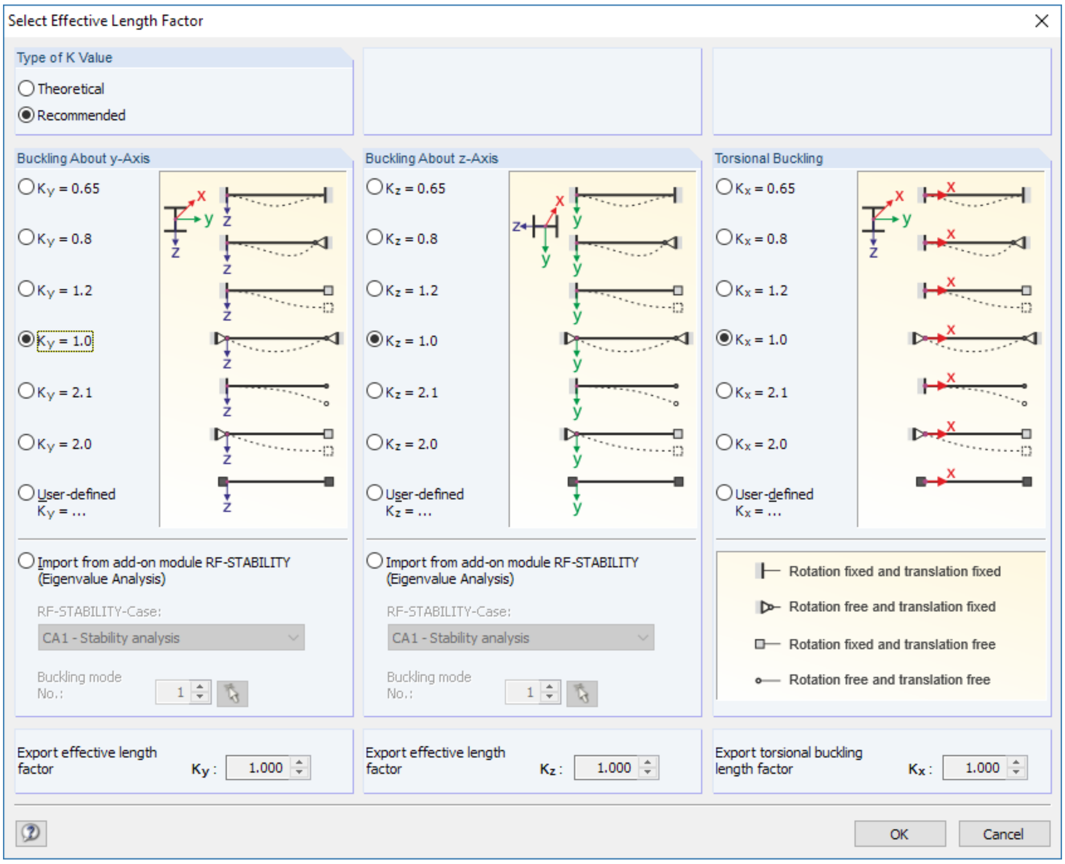


Figure 2.15: Dialog box *Select Effective Length Factor*

In this dialog box, the *Theoretical* or *Recommended* values of the factor, *K*, can be defined that are to be assigned to the selected member. The recommended values are described in [1] Appendix 7, Table C-A-7.1. For each direction, you can also select one of the theoretical Euler buckling modes or apply a *User-defined* factor.

If an eigenvalue analysis has been performed in the RF-STABILITY or RSBUCK add-on module, you can import the *Buckling mode* in order to determine the relevant factor.

Buckling Possible

The stability analysis for flexural and lateral-torsional buckling requires compressive forces to be included. Members for which this is not possible due to their member types (tension members, elastic foundations, rigid couplings) are disabled by default. The corresponding rows are dimmed, and a note appears in the *Comment* column.

The *Buckling possible* check boxes in table row A and in the *Settings* tree allow you to control the stability analyses: They determine whether the analyses for a member are to be performed or not.

Buckling About Axis y / Buckling about Axis z

The *Possible* columns control whether there is a buckling risk about the y-axis and/or z-axis. Those axes represent the local member axes, where the *y*-axis is the strong and the *z*-axis is the weak member axis. You can freely define the effective length factors, K_y, and K_z, for buckling about the strong or the weak axis.

You can check the position of the member axes in the cross-section graphic in the *1.3 Cross-Sections* Window (see Figure 2.9, page 11). To access the RFEM or RSTAB work window, click the [View Mode] button. There you can display the local member axes by using the shortcut menu of the member or the *Display* navigator.

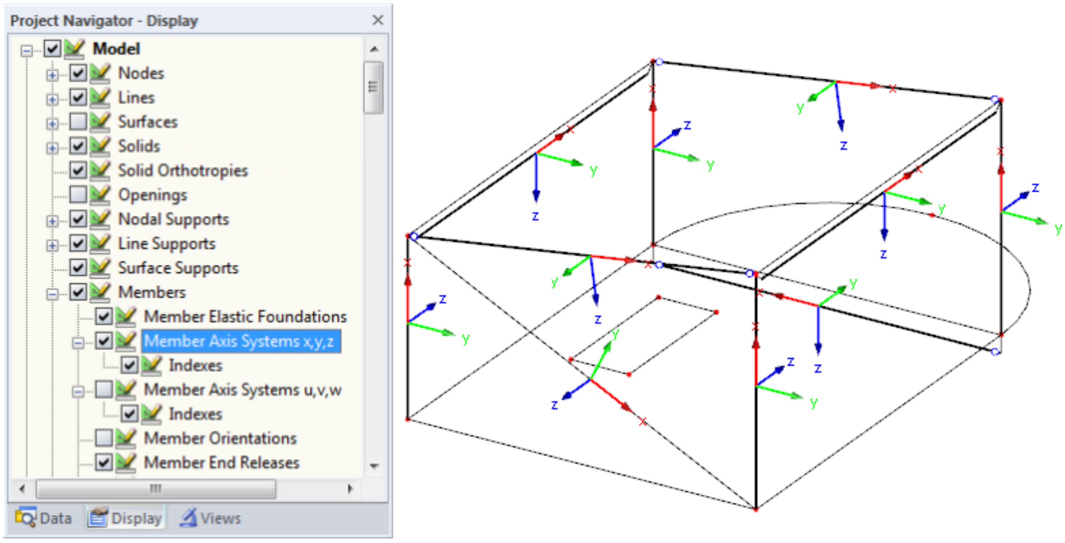
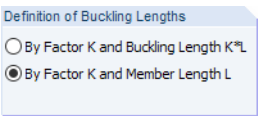


Figure 2.16: Displaying member axes in *Display* navigator of RFEM

If buckling is possible about one or both member axes, you can enter either the effective length factors, *K*, (columns C and F) or the effective lengths, *KL*, (columns D and G). The same is possible in the *Settings* tree.

When you specify the effective length factor, *K*, the program determines the effective length, *KL*, by multiplying the length with the effective length factor. The *K* and *KL* boxes are interactive.

To define the effective lengths graphically in the work window of RFEM or RSTAB, click [...]. This button becomes available when you place the cursor in a *KL* box (see Figure 2.14).

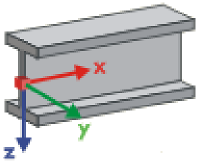


Alternatively, the parameters concerning buckling can be defined via the member lengths, *L*. This option is controlled by the settings in the *Details* dialog box, tab *Stability* (see Figure 3.2, page 30). When the option is set as seen in the image to the left, the titles of columns D and G will change to *L* each so that the real member lengths can be defined (instead of the buckling lengths, *K*L*).

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_x, can be defined in columns I and J. The *x*-axis represents the center line of a member.

L.T.B.

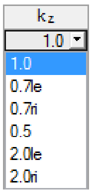


Member axes

Column K controls whether a lateral-torsional buckling analysis is to be carried out.

To determine M_{cr} by the eigenvalue calculation method, an internal member model with four degrees of freedom is created. These degrees must be defined by the factors k_z and k_w. By combining those two factors, you can define the support conditions for lateral-torsional buckling, e.g. lateral and torsional restraint.

Effective Length Factor k_z



The factor, k_z, controls the lateral displacements u_y and the rotations φ_z at both member ends.

$k_z = 1.0$ restrained against lateral displacement u_y on both member ends

$k_z = 0.7le$ restrained against displacement u_y on both ends and restraint about z left

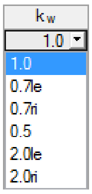
$k_z = 0.7ri$ restrained against displacement u_y on both ends and restraint about z right

$k_z = 0.5$ restrained against displacement u_y and restraint about z on both member ends

$k_z = 2.0le$ restrained against displacement u_y and restraint about z left; right end free

$k_z = 2.0ri$ restrained against displacement u_y and restraint about z right; left end free

Warping Length Factor k_w



The factor, k_w, controls the torsion about the longitudinal axis, φ_x, of the member as well as the warping, ω.

$k_w = 1.0$ restrained against rotation about x on both member ends; free to warp on both sides

$k_w = 0.7le$ restrained against rotation about x on both ends and warping restraint left

$k_w = 0.7ri$ restrained against rotation about x on both ends and warping restraint right

$k_w = 0.5$ torsion and warping restraint on both member ends

$k_w = 2.0le$ restrained against rotation about x and warping ω left; right end free

$k_w = 2.0ri$ restrained against rotation about x and warping ω right; left end free

The abbreviations *le* and *ri* refer to the left and right sides. The abbreviation *le* always describes the support conditions at the member start.

A lateral and torsional restraint is defined by the factors $k_z = 1.0$ (support in y with free rotation about z) and $k_w = 1.0$ (restrained against torsion about x with free warping). As the internal member model of the eigenvalue solver requires only four degrees of freedom, you need not define any extra boundary conditions.

Comment

In the last column, you can enter your own comments for each member, for example to describe the selected buckling lengths.

Set Input for Members No.

Below the *Settings* table, you find the *Set input for members No.* check box. If you select this check box, the subsequent settings will be applied to the selected members or *All* members (you can enter the member numbers manually or select them graphically with the button). This option is useful when you want to assign identical boundary conditions to several members (see https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000726).

With this function, you cannot change the settings you have already made.

2.6 Effective Lengths - Sets of Members

This window controls the effective lengths for sets of members. It is only displayed when you have selected one or more sets of members for design in the *1.1 General Data* Window (see Figure 2.2, page 6).

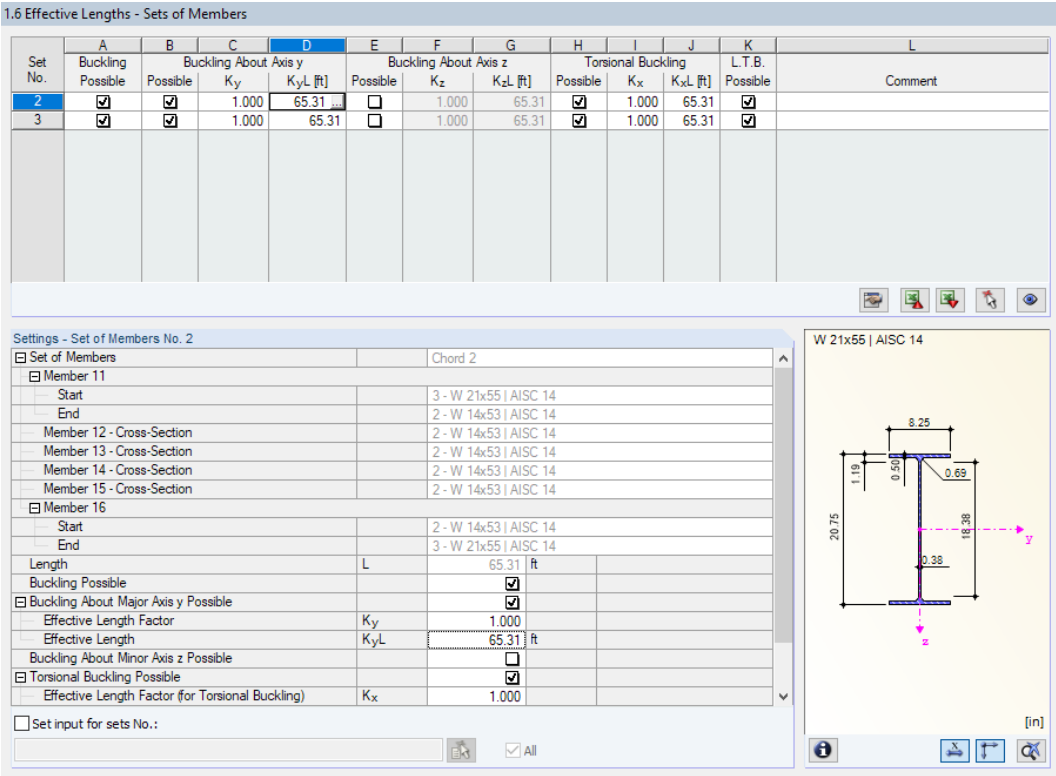
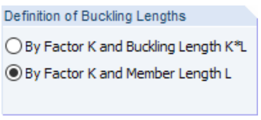


Figure 2.17: Window *1.6 Effective Lengths - Sets of Members*

The concept of the window is very similar to the previous *1.6 Effective Lengths - Members* Window. Here you can define the buckling lengths for sets of members, as described in Chapter 2.5.

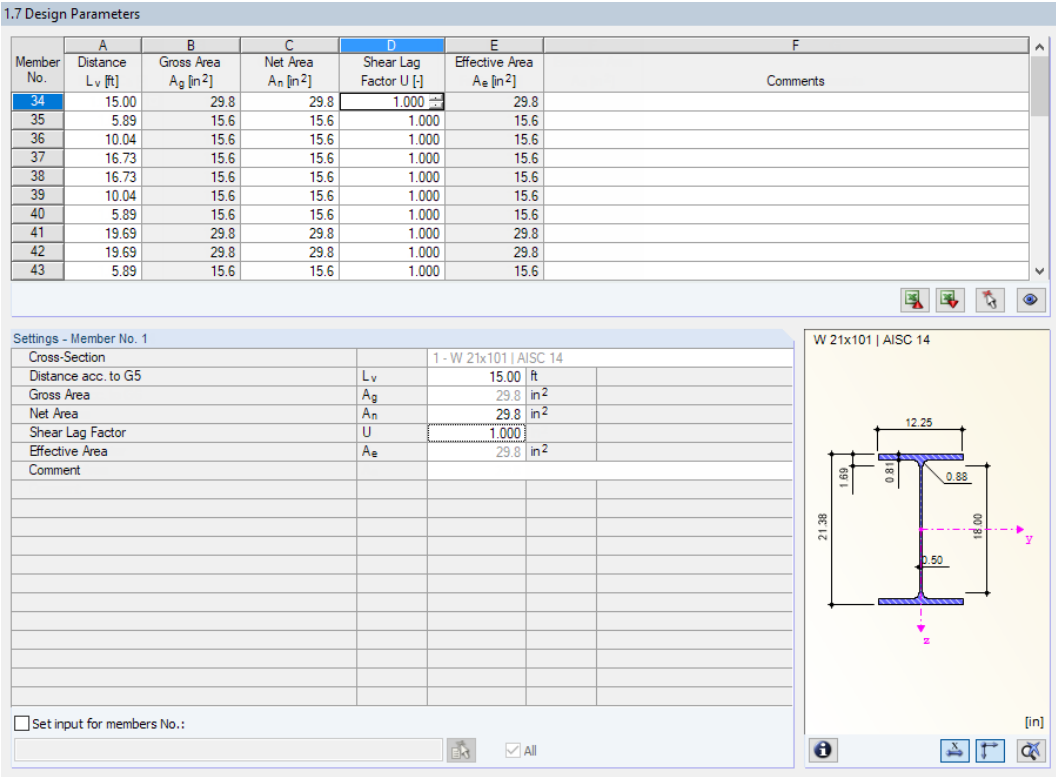
There are differences, however, with respect to the parameters for torsional and lateral-torsional buckling. Those are to be defined by means of boundary conditions in Windows *1.8 Nodal Supports* (see Chapter 2.8) and *1.9 Member Hinges* (see Chapter 2.9).



The parameters concerning buckling and torsional buckling can be defined by means of the buckling lengths, *KL*, or member lengths, *L*. This option is controlled by the settings in the *Details* dialog box, tab *Stability* (see Figure 3.2, page 30).

2.7 Design Parameters

In Window 1.7, several parameters can be defined that are required for design.



Member No.	A Distance L_v [ft]	B Gross Area A_g [in²]	C Net Area A_n [in²]	D Shear Lag Factor U [-]	E Effective Area A_e [in²]	F Comments
34	15.00	29.8	29.8	1.000	29.8	
35	5.89	15.6	15.6	1.000	15.6	
36	10.04	15.6	15.6	1.000	15.6	
37	16.73	15.6	15.6	1.000	15.6	
38	16.73	15.6	15.6	1.000	15.6	
39	10.04	15.6	15.6	1.000	15.6	
40	5.89	15.6	15.6	1.000	15.6	
41	19.69	29.8	29.8	1.000	29.8	
42	19.69	29.8	29.8	1.000	29.8	
43	5.89	15.6	15.6	1.000	15.6	

Figure 2.18: Window *1.7 Design Parameters*

Distance

The distance, L_v, is relevant for the shear design of round HSS sections according to [1] Section G5. It represents the distance between the points of maximum and zero shear force.

Gross Area / Net Area

In columns C and D, the gross area, A_g, and the net area, A_n, of each member is listed. If required, the net area can be modified in order to consider holes (see [1] Section B4.3b).

Shear Lag Factor

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

Effective Area

Column F lists the effective areas, A_e, which are determined according to [1] Equation D3-1 from the net area and the shear lag factor of each member.

2.8 Nodal Supports - Sets of Members

This window is displayed if at least one set of members has been selected for design in Window *1.1 General Data*.

The stability design of each set of members is based on the alternative methods defined in [1] Appendix 7. The respective loads and the boundary conditions are essential for the determination of the elastic critical moment which is important to design the entire set of members.

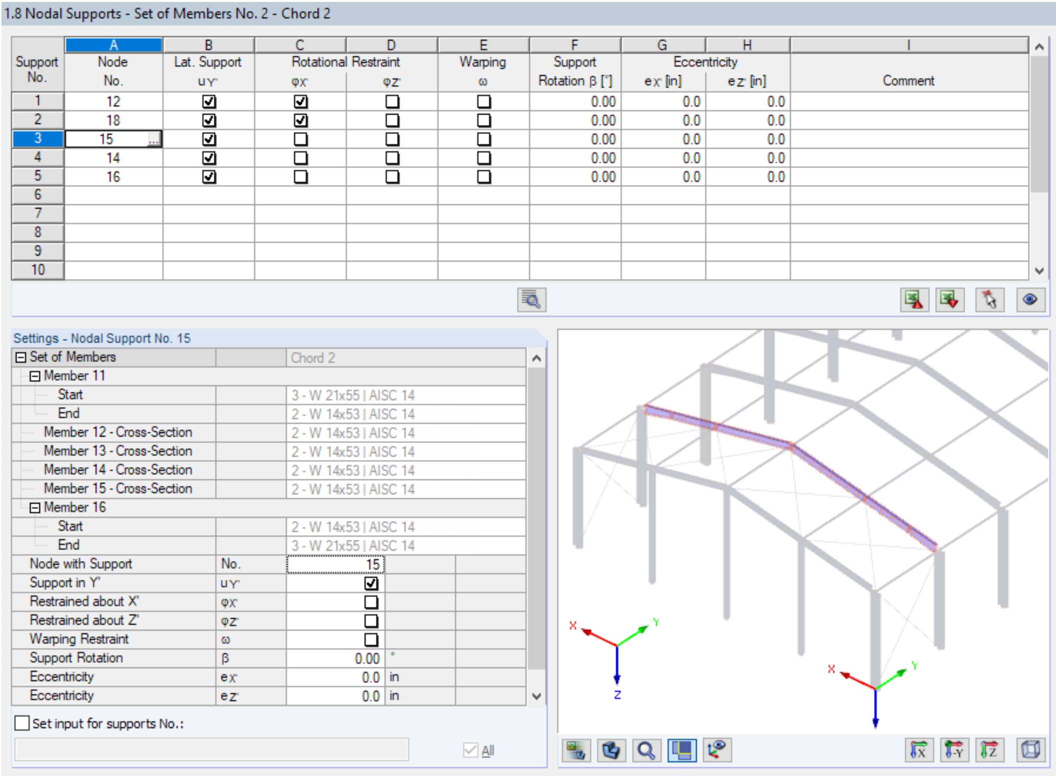


Figure 2.19: Window *1.8 Nodal Supports - Set of Members*

The current table manages the boundary conditions of the set of members that is selected in the navigator on the left!

The supports defined in RFEM or RSTAB (for example, in *Z* for a continuous beam) are not relevant in this table: The distributions of moments and shear forces for the determination of the elastic critical moment are automatically imported from RFEM/RSTAB. Here, you define the support conditions with respect to the failure modes of stability (buckling and lateral-torsional buckling).

Supports on the start and end nodes of each set of members are preset. Any other supports, for example due to connected members, must be added manually. Use the button to select nodes graphically in the RFEM/RSTAB work window.

In order to determine the elastic critical moment of the entire set of members, the program creates a planar framework model with four degrees of freedom for each node.

For the nodal supports, the orientation of axes within a set of members is important. The program checks the position of the nodes and internally defines the axes of the nodal supports for Window 1.8 according to Figure 2.20 through Figure 2.23. The [Local Coordinate System] button below the model graphic can help you with the orientation: Use it to display the set of members exclusively, with the axes being clearly visible (see also Knowledge Base article on our website).

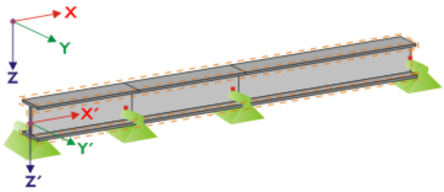


Figure 2.20: Auxiliary coordinate system for nodal supports – straight set of members

If all members of a set of members lie on a straight line, as shown in Figure 2.20, the local coordinate system of the first member within this set of members is applied to entire set of members.

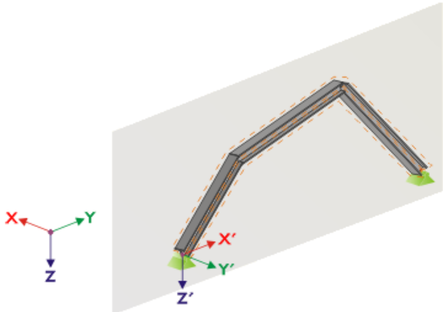


Figure 2.21: Auxiliary coordinate system for nodal supports – set of members in vertical plane

If the members do not lie on a straight line, they nevertheless must be located in one plane. We can see a vertical plane in Figure 2.21. In this case, the X′-axis is horizontal and oriented in the plane direction. The Y′-axis is horizontal as well, but perpendicular to the X′-axis. The Z′-axis points downwards.

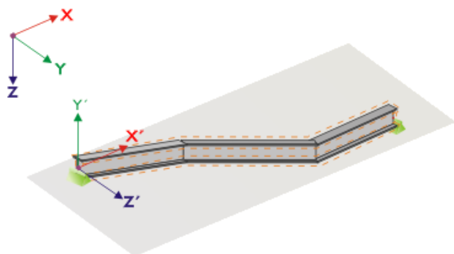


Figure 2.22: Auxiliary coordinate system for nodal supports – set of members in horizontal plane

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

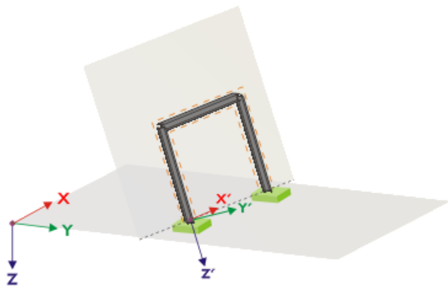


Figure 2.23: Auxiliary coordinate system for nodal supports – set of members in inclined plane

Figure 2.23 shows the general case: The members do not lie on a straight line, but are located in an inclined plane. The definition of the X′-axis results from the intersection line between the inclined and the horizontal plane. The Y′-axis is perpendicular to the X′-axis (and also perpendicular to the inclined plane). The Z′-axis is perpendicular to the X′- and Y′-axes.

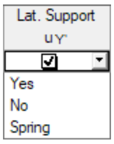
The buttons below the graphic have the following functions:

Button	Function
	Shows model or schematic sketch
	Shows members as 3D rendering or wire-frame model
	Shows current set of members or entire model
	Displays irrelevant members of model as transparent or opaque
	Shows set of members with local coordinate system or entire model
	Shows view in direction of X-axis
	Shows view in opposite direction of Y-axis
	Shows view in direction of Z-axis
	Shows isometric view

Table 2.2: Buttons for graphic window

If the set of members is laterally supported in specific locations, these nodes must be added manually in Window *1.8 Nodal Supports*. In this way, it is possible to consider, for example, the effect of a purlin which is connected in the spatial model of RFEM or RSTAB. If this support is neglected in the reduced set of members model, instabilities may occur.

The [...] button (see Figure 2.19) enables you to define the supported nodes graphically in the work window of RFEM or RSTAB.



In columns B to N of the table, you specify the support conditions of the selected nodes. To activate or deactivate the supports and restraints for the corresponding degrees of freedom, select the check boxes. Alternatively, you can enter the constants of a translational or rotational *Spring* manually.

The *Support rotation* and *Eccentricity* parameters allow for modeling support conditions close to reality.

With the [Edit warp stiffener] button (below the table) it is possible to make the program determine the constant of a warp spring.

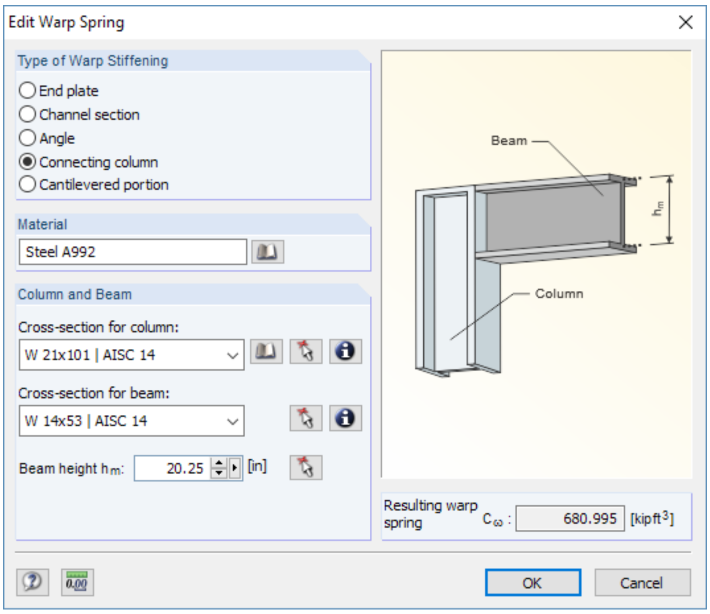


Figure 2.24: Dialog box *Edit Warp Spring*

The following warp stiffening types are available in the *Edit Warp Spring* dialog box:

- End plate
- Channel section
- Angle
- Connecting column
- Cantilevered portion

Materials and cross-sections can be selected via the lists and [Library] buttons. With the button, you can select them also graphically in the model of RFEM or RSTAB.

Based on the parameters, RF-/STEEL AISC determines the *Resulting warp spring*, C_ω, which can then be transferred to Window 1.8 with [OK].

Warping torsion with seven degrees of freedom

To analyze sets of members according to the second-order analysis for flexural-torsional buckling with warping torsion, select the corresponding check box in the *Warping Torsion* tab of the *Details* dialog box (see Figure 3.4, page 32). The table titles of Window 1.8 will be adjusted accordingly.

The warping analysis requires a license of the **RF-/STEEL Warping Torsion** module extension.

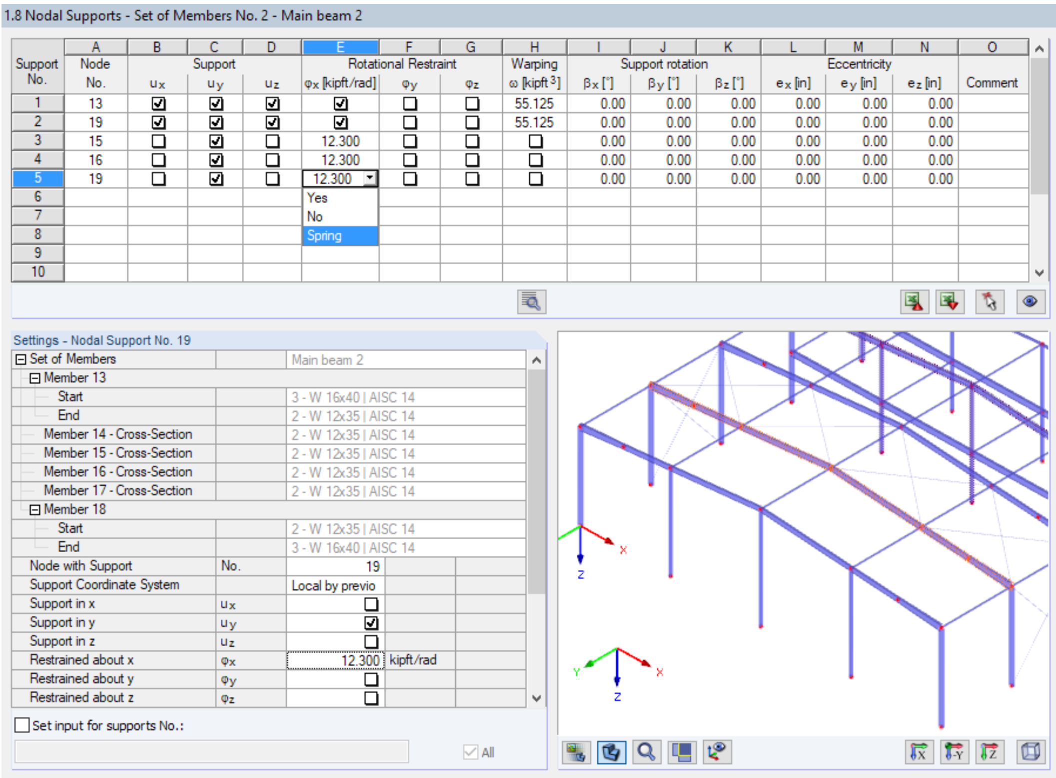


Figure 2.25: Window *1.8 Nodal Supports - Set of Members* for warping torsion with seven degrees of freedom

Define the support parameters of the set of members "extracted" from the structural model. Supports can be specified for all nodes of the respective members. The supports defined in RFEM or RSTAB are preset, as well as the supports on both ends of the set of members.

Lateral supports along the set of members have to be added as additional supports. In this way, e.g. purlins connected in the spatial model can be accounted for. If this specific support is missing in the model of the "extracted" set of members, stability problems may occur.

The supported nodes can be selected graphically with the button in the RFEM/RSTAB work window.

In columns B to N, the support conditions of the selected nodes can be defined. To activate or deactivate the supports and restraints for the corresponding degrees of freedom, select or deselect the check boxes. Alternatively, you can enter the constants of the translational and rotational springs manually.

The *Support rotation* and *Eccentricity* parameters allow for modeling support conditions close to reality.

2.9 Member Hinges - Sets of Members

This window is displayed if at least one set of members has been selected in the *1.1 General Data* Window. Here, you can define hinges for members within the set of members that, for structural reasons, do not transfer the degrees of freedom locked in Window 1.8 as internal forces. Make sure that no double hinges are generated in coaction with Window 1.8!

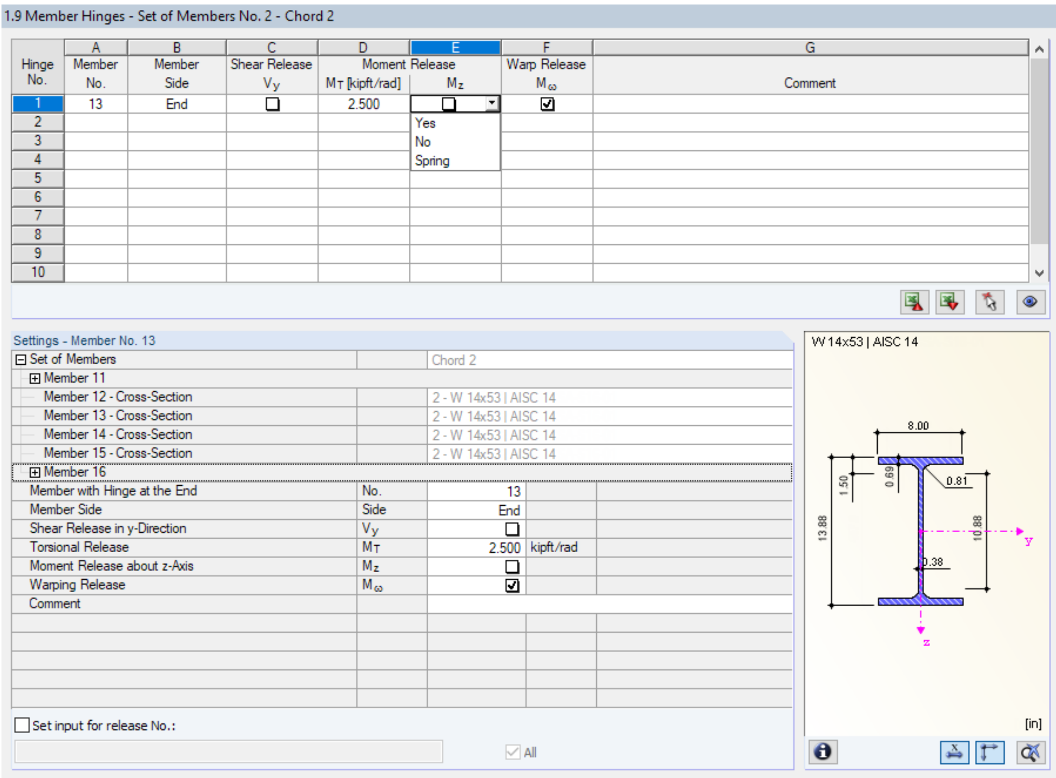
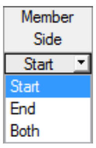


Figure 2.26: Window *1.9 Member Hinges*

The current table manages the hinges of the set of members that is selected in the navigator on the left.



In column B, you specify the *Member Side* where the hinge is located, or if there are hinges on both member sides.

In columns C to F, you can define the releases or spring constants in order to adjust the set of members model to the support conditions defined in Window 1.8.

Warping torsion with seven degrees of freedom

Details...

If the warping torsion with seven degrees of freedom has been selected in the *Warping Torsion* tab of the *Details* dialog box (requires license of **RF-/STEEL Warping Torsion** module extension), additional columns are available for the specific parameters.

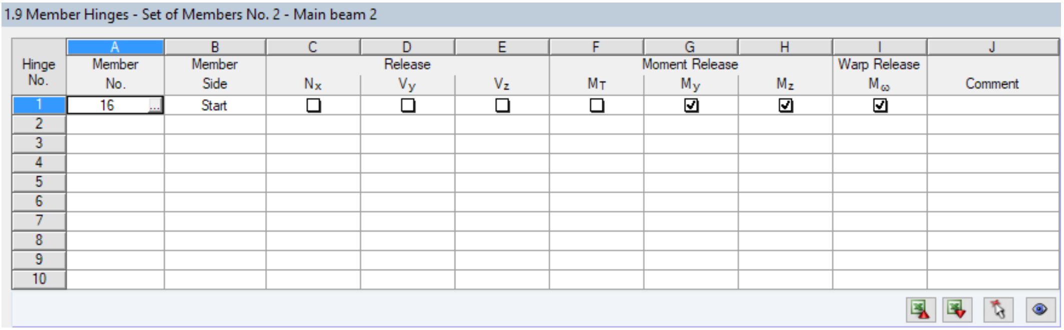


Figure 2.27: Window *1.9 Member Hinges - Set of Members* for warping torsion with seven degrees of freedom

2.10 Parameters - Sets of Members

Details...

This window is shown when the warping torsion with seven degrees of freedom has been selected in the *Warping Torsion* tab of the *Details* dialog box (see Figure 3.4, page 32). For this specific type of analysis, a license of **RF-/STEEL Warping Torsion** module extension is required.

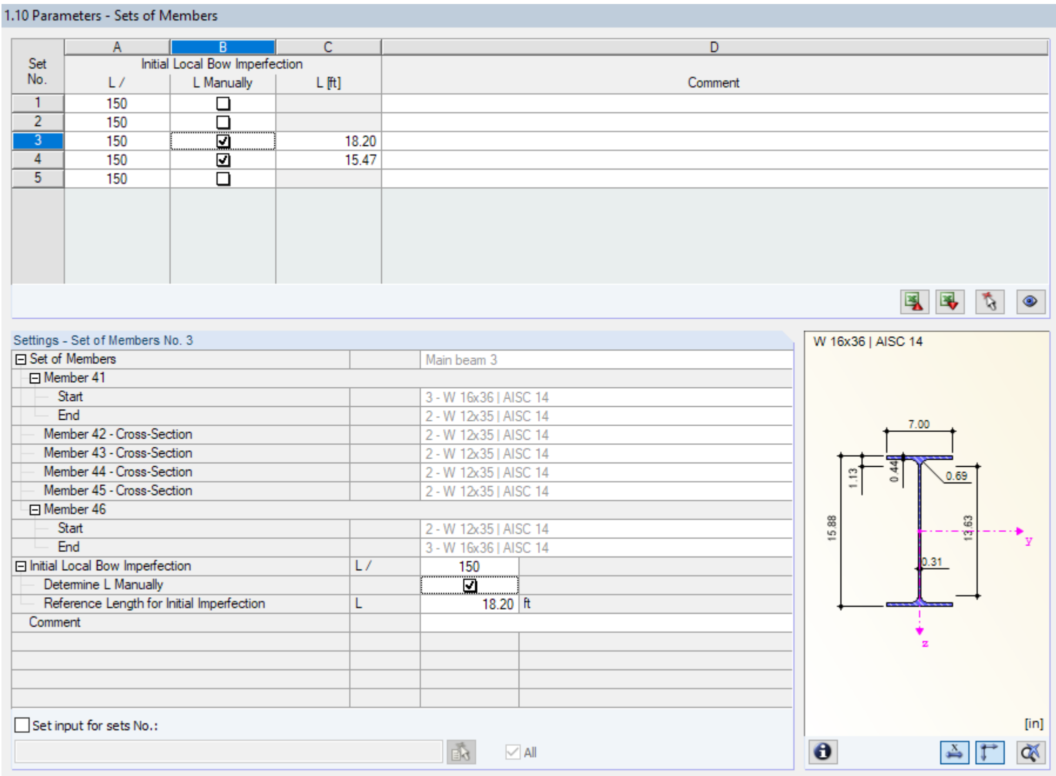


Figure 2.28: Window *1.10 Parameters - Sets of Members* for warping torsion with seven degrees of freedom

Taking into account the boundary conditions, such as nodal supports, hinges, or load application, RF-/STEEL AISC determines the mode shapes of each set of members before the actual design. They are then considered accordingly.

In column A, you can define an *Initial Local Bow Imperfection* that is related to the length of each set of members. The default value, L/150, can be adjusted to account for the imperfection characteristics of the model or cross-section. It will only be considered, however, if the length, *L*, defined in column C is greater than zero.

The reference length, *L*, can be adapted when as the *L Manually* check box is selected. In this way, it is possible to account for lateral supports along the set of members, for example.

As the Specification [1] does not provide any specific values for imperfections, the initial bow imperfection is disabled by means of the default length 0.00 ft.

After the design, the mode shapes can be checked graphically in the *Mode Shape View* Window (see Figure 5.8, page 52).

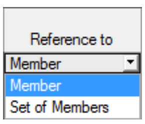
2.11 Serviceability Data

The last input window controls the settings for the serviceability limit state design of specific objects. It is available when you have selected one or more load cases or combinations in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 8).

1.11 Serviceability Data

	A	B	C	D	E	F	G	H
No.	Reference to	Set of Members No.	Reference Length Manually	L [ft]	Direction	Precamber $w_{c,z}$ [in]	Beam Type	Comment
1	Set of Members	2	☐	65.31	y, z	0.000	Beam	
2	Set of Members	5	☐	65.31	y, z	0.000	Beam	
3	Member	82	☐	16.40	y, z	0.000	Beam	
4	Member	81	☑	4.54	z	0.000	Cantilever End Free	
5	Member	83	☑	4.54	z	0.000	Cantilever End Free	
6	Member	15	☐	10.04	y, z	0.000	Beam	
7	Member	16	☐	5.89	y, z	0.000	Beam	
8	Member	25	☐	19.69	y, z	0.000	Beam	
9	Member	26	☐	19.69	y, z	0.000	Beam	
10	Member	92	☐	25.62	u, v	0.000	Beam	
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

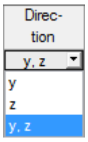
Figure 2.29: Window *1.11 Serviceability Data*



In Column A, you define whether the deformation refers to single members, lists of members, or sets of members.

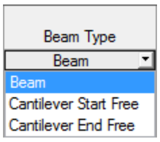
For a list or set of members, the orientation and rotation of all contained members must be identical. This will guarantee that the components of the deformation are taken into account correctly.

In column B, you can specify the numbers of the members or sets of members that are to be analyzed. The [...] button enables you to select the objects graphically in the work window. In column D, the *Reference Length* of each object is shown. The geometrical lengths of the members, lists or sets of members are set by default. If necessary, you can adjust those values after having selected the *Manually* check box in column C.

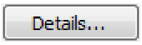


Column E controls the governing *Direction* for the deflection analysis. You can select the directions of the local member axes *y* and *z* (or *u* and *v* for unsymmetrical cross-sections).

You can consider a *Precamber*, w_c, in column F, if applicable. The reference to the axes is controlled by the specification in the *Details* dialog box (see Figure 3.3, page 31).



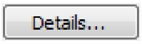
The *Beam Type* is important for the correct reference to the limit deformations. In column G, you can specify whether a beam or a cantilever is to be analyzed. For the latter, you can define which end has no support.



The *Details* dialog box controls whether the deformations are related to the undeformed system or the shifted ends of the members or sets of members (see Figure 3.3, page 31).

3 Calculation

3.1 Detail Settings



Before you start the calculation, it is recommended to check the design details. You can access the corresponding dialog box in all windows of the add-on module by clicking [Details].

The *Details* dialog box has the following tabs:

- Ultimate limit state
- Stability
- Serviceability
- Warping torsion
- General

3.1.1 Ultimate Limit State

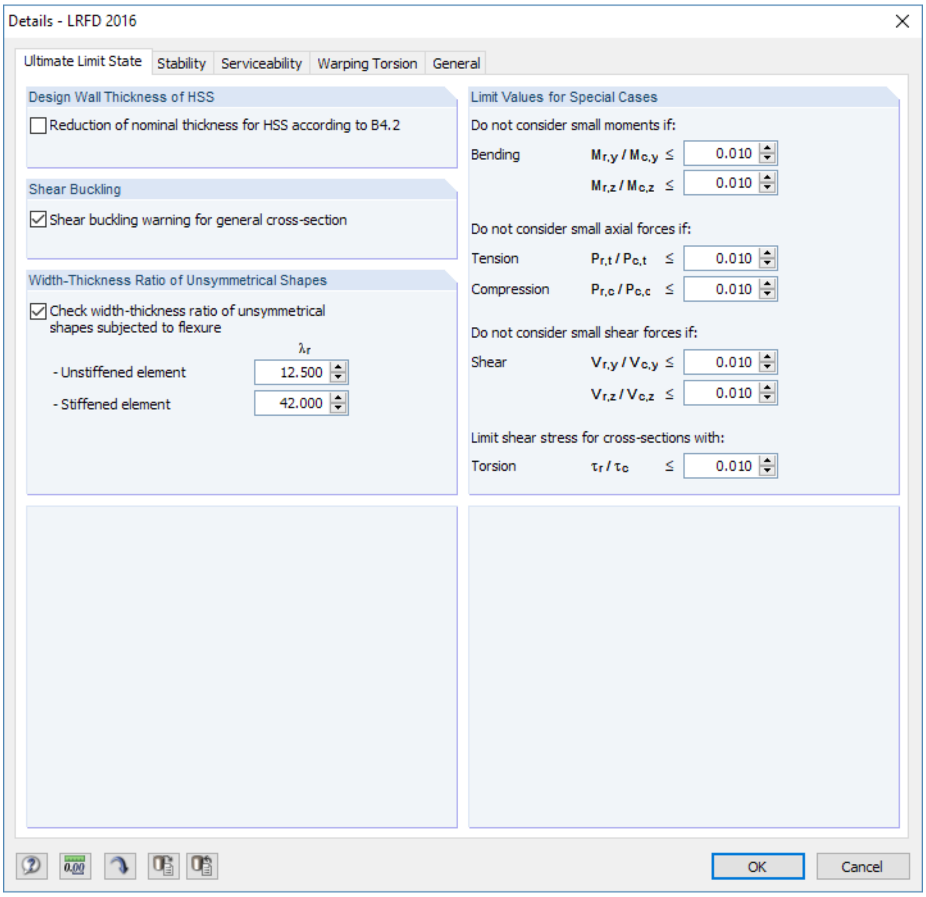


Figure 3.1: Dialog box *Details*, tab *Ultimate limit state*

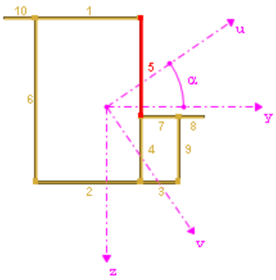
Design Wall Thickness of HSS

When this option is selected, the design wall thickness, *t*, of hollow structural sections will be taken equal to 0.93 times the nominal wall thickness. The reduction is recommended for HSS produced according to other standards than ASTM A1065/A1065M or ASTM A1085/A1085M (see [1] Section B4.2).

Shear Buckling

The Specification [1] covers the design of common cross-sections, such as I-shapes, tees, angles, channels, box sections, etc. User-defined sections or shapes not included in the Specification are classified as *General* types by the program. Nevertheless, RF-/STEEL AISC performs the design of those sections if specific conditions are fulfilled. The shear buckling design according to [1] Chapter G, however, represents a problem for general shapes. Due to their inexplicit geometrical proportions, they will be excluded from this type of design.

The option enables you to switch the warning *1009) Check of shear buckling for general cross-sections is not allowed* in the result tables on or off.



Width-Thickness Ratio of Unsymmetrical Shapes

For the classification according to [1] Table B4.1b, a distinction is made between unstiffened and unstiffened elements. *General* sections, such as Case 3 and Case 8 in Table B4.1a, are missing in Table B4.1b, however. For the design of the width-to-thickness ratios of user-defined sections, the check of unsymmetrical shapes can be activated. Then the limiting width-to-thickness ratios, λ_r, can be specified for unstiffened and stiffened elements.

Limit Values for Special Cases

For a simplified design, it is possible to neglect small bending moments, axial forces, or shear forces, as well as shear stresses due to torsion. The limit ratios of the moments, forces, or stresses can be defined separately in this dialog box section.

The preset limits are not part of the Specification [1]. Changing the values is in the responsibility of the user.

3.1.2 Stability

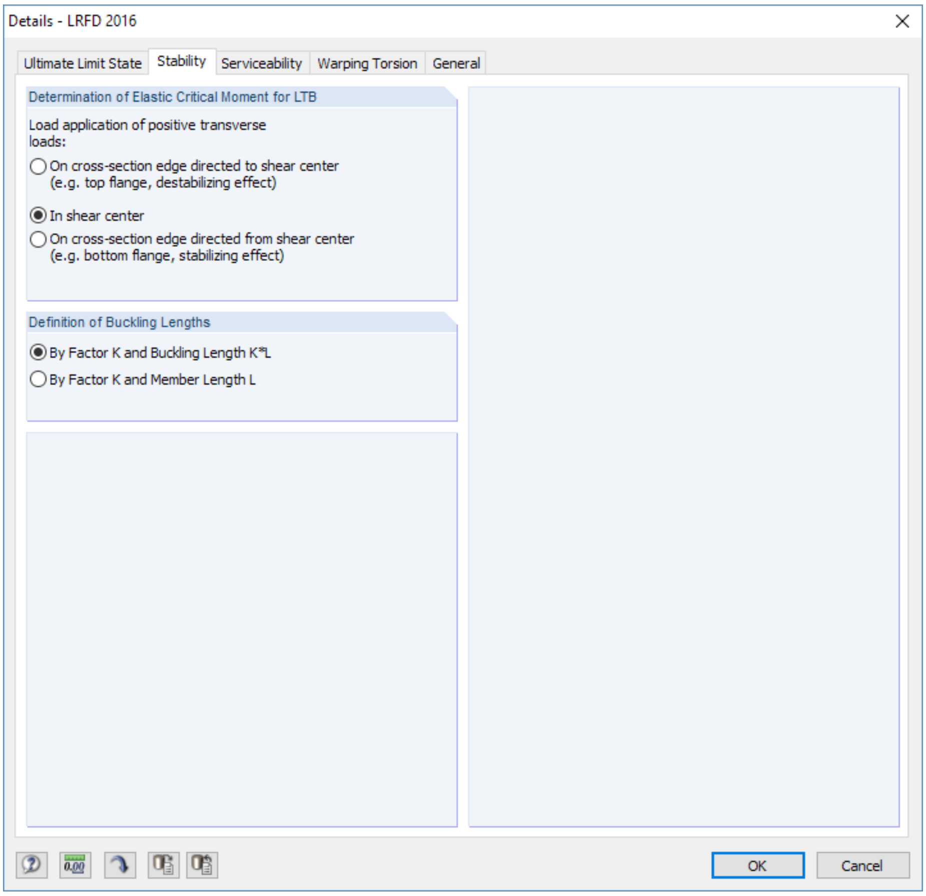


Figure 3.2: Dialog box *Details*, tab *Stability*

Determination of Elastic Critical Moment for LTB

The elastic critical moment is calculated automatically by the Eigenvalue solver.

If there are transverse loads, it is important to define the location where those forces are acting: Depending on the location of *Load application*, they can have stabilizing or destabilizing effects due to their eccentricities. Thus, they have a major impact on the elastic critical moment. This setting is globally applied to all loads and all sets of members of the design case.

The signs of the eccentricities are related to the shear center, *M*, of each cross-section. The following article on our website provides more information about the sign convention for transverse loads: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000880

Please note that the settings are globally applied to all members and sets of members. With loads acting in different locations, the objects can be designed in separate design cases with specific settings (see Chapter 7.1, page 55).

Definition of Buckling Lengths

The effective lengths which are relevant for buckling and torsional buckling are managed in Window 1.5 (for members) and Window 1.6 (for sets of members) – see Chapters 2.5 and 2.6 at pp. 15 - 19. There are two options how to define the parameters concerning buckling and torsional buckling: those can be referred either to the *Buckling Length K*L* or to the *Member Length L* of each member or set of members.

Depending on the setting in the *Details* dialog box, the titels of the respective columns in Windows 1.5 and 1.6 will be adapted.

3.1.3 Serviceability

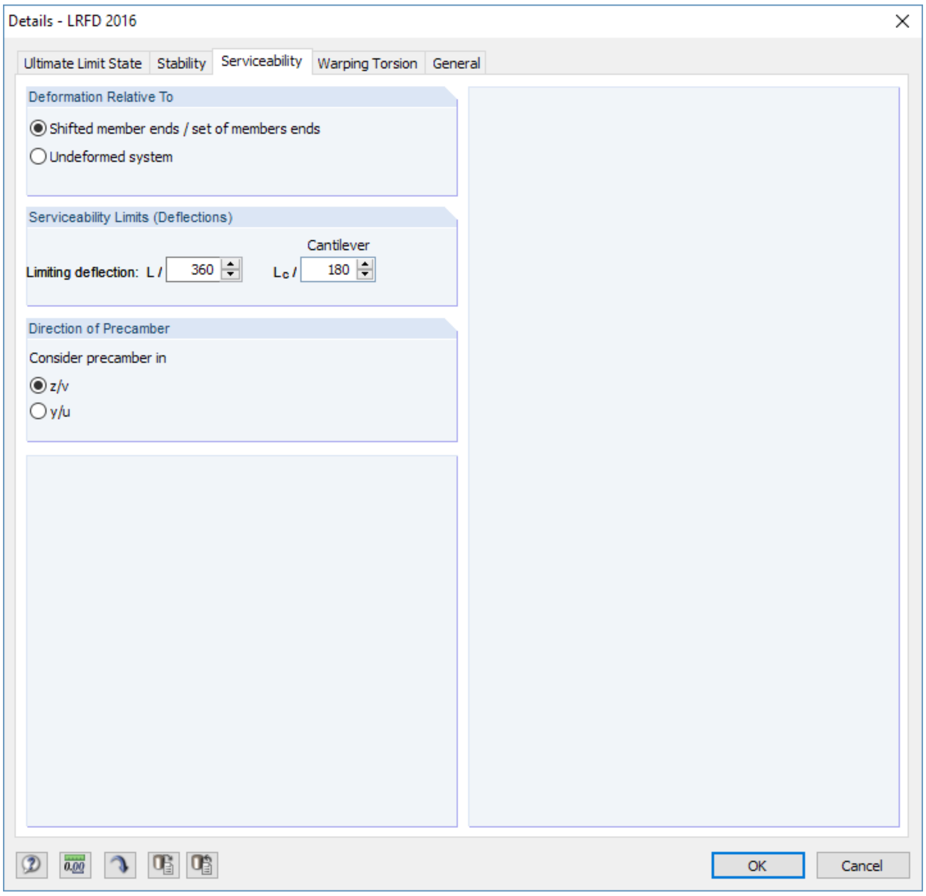


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Relative To

The options below specify whether the deformations are related to the shifted member ends or set of members ends (line between start and end nodes of deformed model) or to the undeformed original system. The difference is illustrated by an example which you can find on our website: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001081

Serviceability Limits (Deflections)

For the SLS design, the limiting deflections can be separately defined for beams (default: L/360) and cantilevers (default: L_c/180). Commentary Section L2 of the Specification [1] gives recommendations on the maximum values for deflections.

Direction of Precamber

When you specify two directions in column E of the *1.11 Serviceability Data* Window and apply a precamber in column F, you can determine for which direction the precamber is to be considered.

3.1.4 Warping Torsion

This tab allows for settings of the torsional analysis for sets of members. You can access the entries when the **RF-/STEEL Warping Torsion** module extension has been licensed.

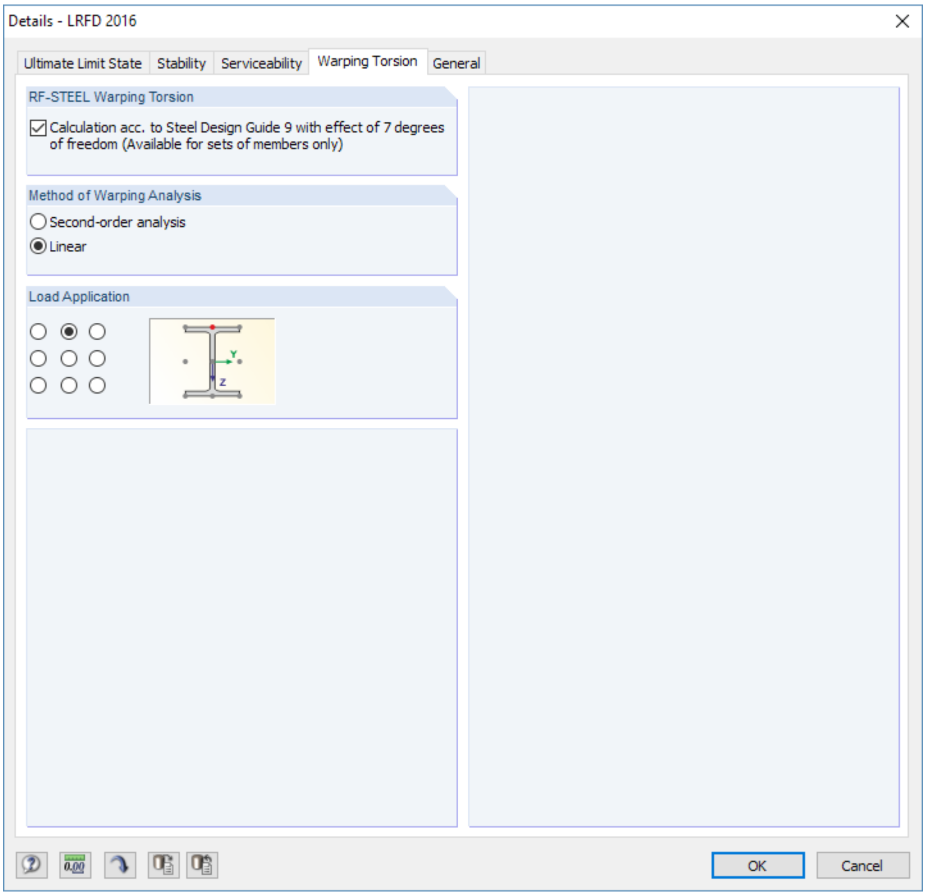


Figure 3.4: Dialog box *Details*, tab *Warping Torsion*

RF-/STEEL Warping Torsion

If you want RF-/STEEL AISC to perform a *Calculation according to Steel Design Guide 9* [2], select the check box.

The determination of torsional stresses and their combination with stresses due to bending and axial load is described in [2] Chapter 4. In the method with seven degrees of freedom, the stability calculation is carried out according to the second-order analysis for flexural-torsional buckling taking into account warping torsion and imperfections affine to mode shapes.

The degrees of freedom concerning the displacements and rotations in and about the three axes X', Y', Z' as well as warping can be defined in Window 1.8 (see Figure 2.25, page 24) and Window 1.9 (see Figure 2.27, 26).

In Window 1.10, an initial local bow imperfection can be specified (see Figure 2.28, page 26).

When the *Warping Torsion* option has been selected, Window *1.6 Effective Lengths - Sets of Members* is not shown. The parameters are to be defined in the specific Windows 1.8 through 1.10 instead.

The warping analysis is performed iteratively, with the stiffness matrix *K* changing due to already computed internal forces and deformations.

On our website, you can find an example illustrating the design with seven degrees of freedom: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001544

You can also watch a webinar which includes the warping torsion design of a steel frame: https://www.dlubal.com/en-US/support-and-learning/learning/webinars/000270

Method of Warping Analysis

Aside from a *Second-order analysis*, the warping analysis of the sets of members can be carried out by means of a *Linear* approach. This linear geometric analysis makes it is possible to mimimize the effects of the overall stability behavior and focus on the effects of warping instead.

Load Application

The point of *Load Application* is significant for the stability analysis with seven degrees of freedom. Depending on this location, the load has an either stabilizing or destabilizing effect on the stability characteristics.

There are nine options available to define the location where the load is applied on the cross-section. In the sketch of the section, the selected point is highlighted in red.

3.1.5 General

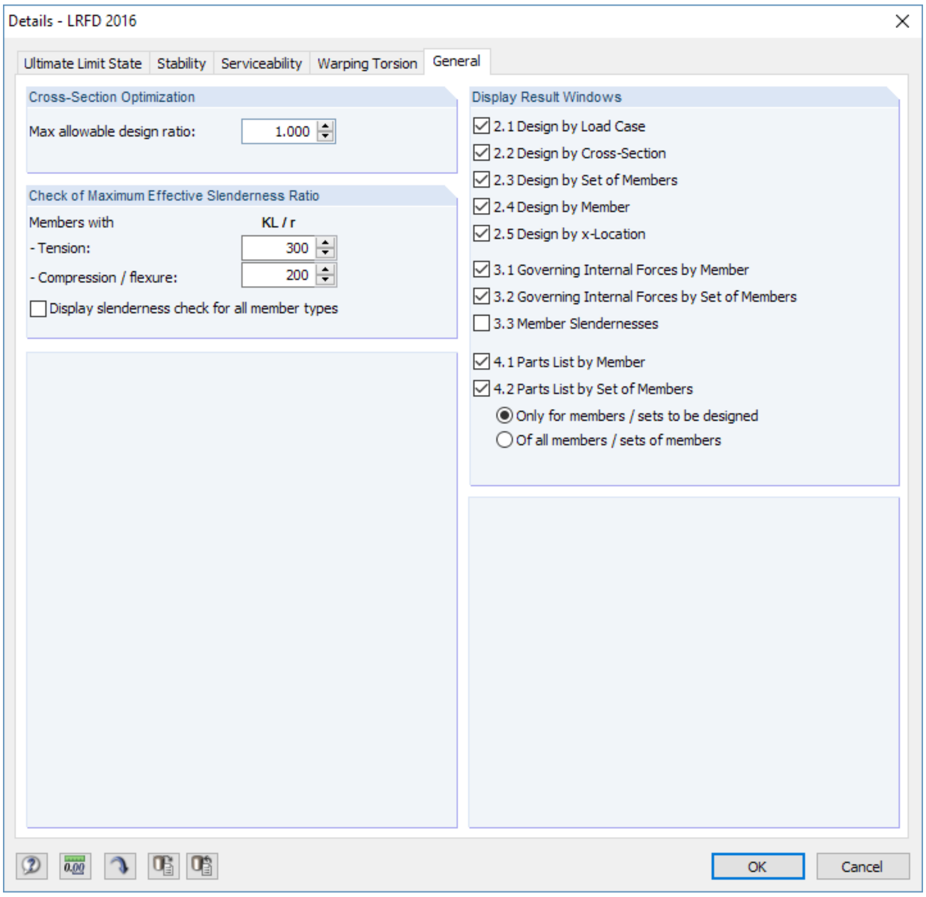


Figure 3.5: Dialog box *Details*, tab *General*

Cross-Section Optimization

By default, the optimization is targeted on the maximum design ratio of 100%. If required, you can change this limit value.

Check of Maximum Effective Slenderness Ratio

According to [1] User Note to Section D1, the slenderness ratio, *KL / r*, preferably should not exceed 300 for members in tension. For members in compression or flexure, the slenderness ratios should not exceed 200 (cf. [1] Section E2). If required, the limit ratios can be adjusted.

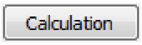
The limit ratios are compared to the real member slendernesses in Window 3.3. That window is available after the calculation (see Chapter 4.8, page 43) when the corresponding option has been checked in the *Display Result Windows* section of the *Details* dialog box.

Display Result Windows

In this dialog section, you can select which result windows including parts list are to be displayed in the output. Those windows are described in Chapter 4.

Window *3.3 Member Slendernesses* is deactivated by default.

3.2 Starting Calculation



In all input windows of RF-/STEEL AISC, you can start the design via the [Calculation] button.

The add-on module searches for the results of the load cases, load combinations, and result combinations that are to be designed. If they are not available yet, RF-/STEEL AISC starts the calculation in RFEM or RSTAB to determine the relevant internal forces.

You can also start the calculation in the RFEM or RSTAB user interface: The *To Calculate* dialog box (menu **Calculate** → **To Calculate**) lists the design cases of the add-on modules like load cases or load combinations.

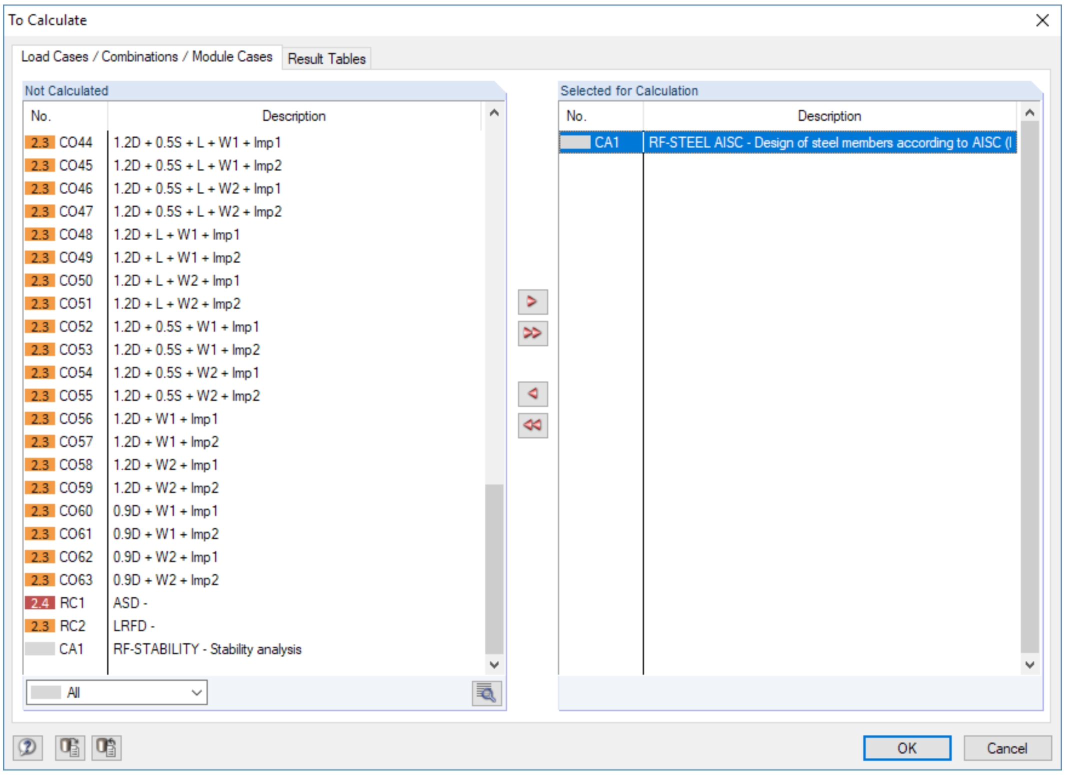
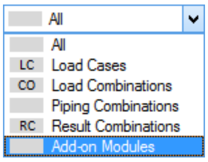


Figure 3.6: Dialog box *To Calculate*



If the RF-/STEEL AISC design cases are missing in the *Not Calculated* section, select *All* or *Add-on Modules* in the drop-down list below the list.

To transfer the selected RF-/STEEL AISC cases to the list on the right, use the [▷] button. Then, click [OK] to start the calculation.

You can also calculate a design case directly by using the list in the toolbar: Set the RF-/STEEL AISC case and click the [Show Results] button.

Figure 3.7: Direct calculation of an RF-STEEL AISC design case in RFEM

Subsequently, you can follow the calculation in a separate dialog box.

4 Results

Window *2.1 Design by Load Case* appears immediately after the calculation.

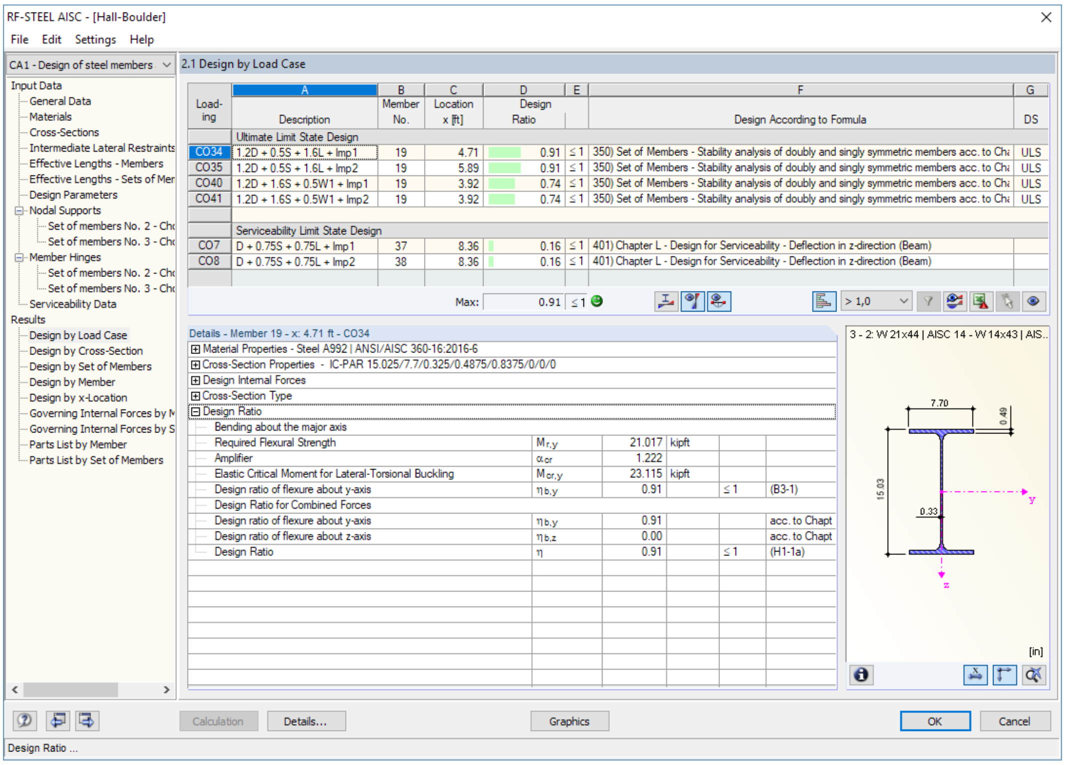
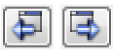


Figure 4.1: Result window with design results and details

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

Windows 3.1 and 3.2 list the governing internal forces, Window 3.3 gives information on the member slendernesses. The last two Windows 4.1 and 4.2 show the parts lists of members and sets of members.



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

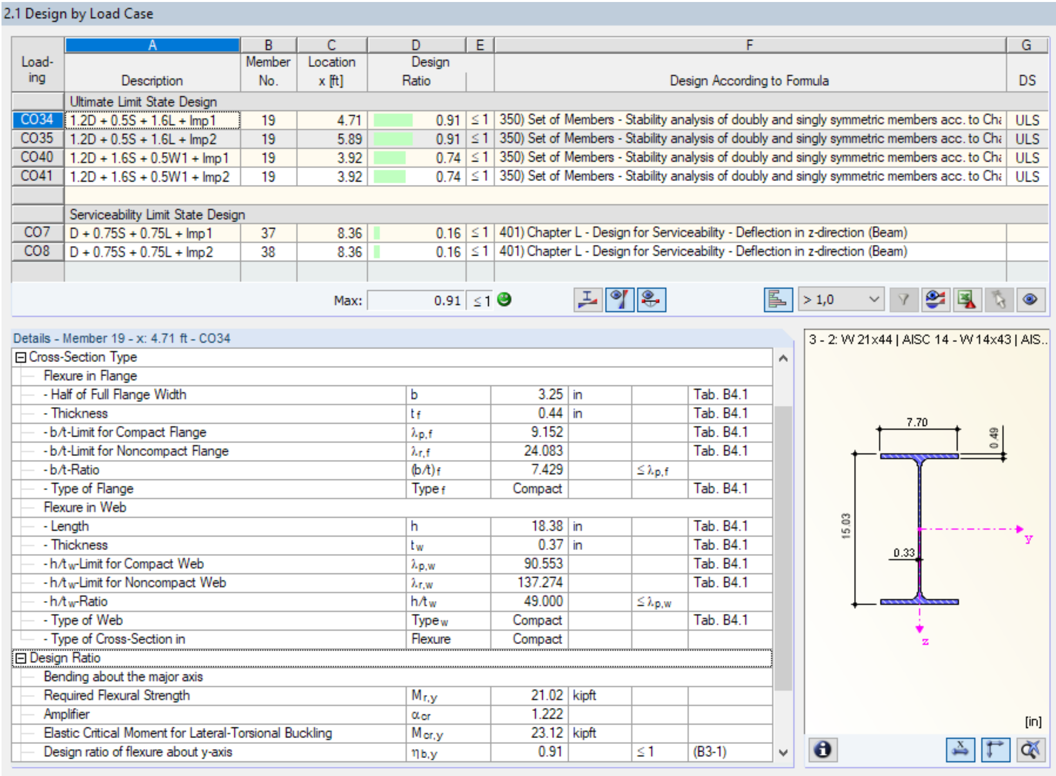
[OK] saves the results. RF-/STEEL AISC is closed and you return to the main program.

Chapter 4 describes the different result windows one by one. The evaluation and checking of the results is described in Chapter 5 starting on page 46.

4.1 Design by Load Case

The upper part provides a summary of the results, sorted by load case, load and result combinations of the governing designs. Furthermore, the list is split into *Ultimate Limit State Design* and *Serviceability Limit State Design* results.

The *Details* section below includes specific information on the cross-section properties, internal forces, and design parameters for the load case or combination selected in the upper table.


2.1 Design by Load Case

Loading	A Description	B Member No.	C Location x [ft]	D Design Ratio	E	F Design According to Formula	G DS
	Ultimate Limit State Design						
CO34	1.2D + 0.5S + 1.6L + Imp1	19	4.71	0.91	≤ 1	350) Set of Members - Stability analysis of doubly and singly symmetric members acc. to Chá	ULS
CO35	1.2D + 0.5S + 1.6L + Imp2	19	5.89	0.91	≤ 1	350) Set of Members - Stability analysis of doubly and singly symmetric members acc. to Chá	ULS
CO40	1.2D + 1.6S + 0.5W1 + Imp1	19	3.92	0.74	≤ 1	350) Set of Members - Stability analysis of doubly and singly symmetric members acc. to Chá	ULS
CO41	1.2D + 1.6S + 0.5W1 + Imp2	19	3.92	0.74	≤ 1	350) Set of Members - Stability analysis of doubly and singly symmetric members acc. to Chá	ULS
	Serviceability Limit State Design						
CO7	D + 0.75S + 0.75L + Imp1	37	8.36	0.16	≤ 1	401) Chapter L - Design for Serviceability - Deflection in z-direction (Beam)	
CO8	D + 0.75S + 0.75L + Imp2	38	8.36	0.16	≤ 1	401) Chapter L - Design for Serviceability - Deflection in z-direction (Beam)	
			Max:	0.91	≤ 1		

Details - Member 19 - x: 4.71 ft - CO34

Cross-Section Type					
Flexure in Flange					
- Half of Full Flange Width	b	3.25	in		Tab. B4.1
- Thickness	t_f	0.44	in		Tab. B4.1
- b/t-Limit for Compact Flange	$\lambda_{p,f}$	9.152			Tab. B4.1
- b/t-Limit for Noncompact Flange	$\lambda_{r,f}$	24.083			Tab. B4.1
- b/t-Ratio	$(b/t)_f$	7.429		$\leq \lambda_{p,f}$	
- Type of Flange	$Type_f$	Compact			Tab. B4.1
Flexure in Web					
- Length	h	18.38	in		Tab. B4.1
- Thickness	t_w	0.37	in		Tab. B4.1
- h/t_w-Limit for Compact Web	$\lambda_{p,w}$	90.553			Tab. B4.1
- h/t_w-Limit for Noncompact Web	$\lambda_{r,w}$	137.274			Tab. B4.1
- h/t_w-Ratio	h/t_w	49.000		$\leq \lambda_{p,w}$	
- Type of Web	$Type_w$	Compact			Tab. B4.1
- Type of Cross-Section in	Flexure	Compact			
Design Ratio					
Bending about the major axis					
Required Flexural Strength	$M_{r,y}$	21.02	kipft		
Amplifier	α_{cr}	1.222			
Elastic Critical Moment for Lateral-Torsional Buckling	$M_{cr,y}$	23.12	kipft		
Design ratio of flexure about y-axis	$\eta_{b,y}$	0.91		≤ 1	(B3-1)

Figure 4.2: Window *2.1 Design by Load Case*

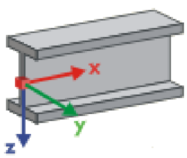
Description

This column shows the descriptions of each designed load case, load or result combination.

Member No.

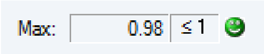
In this column, the number of each member is given that has the maximum design ratio of the respective loading.

Location x



The column shows the x-location of each member where the maximum design ratio occurs. For the tabular output, the program uses the following member locations *x*:

- Start and end nodes
- Division points according to optionally defined member divisions (see RFEM Table 1.16 or RSTAB Table 1.6)
- Member divisions according to specification for member results (see RFEM/RSTAB dialog box *Calculation Parameters*, tab *Global Calculation Parameters*)
- Extreme values of internal forces



Design Ratio

Columns D and E show the design conditions according to [1].

The lengths of colored bars represent the respective design ratios.

Design According to Formula

This column lists the references of the Standard [1] according to which the different types of design have been performed.

DS

The last column provides information on the respective design situations.

4.2 Design by Cross-Section

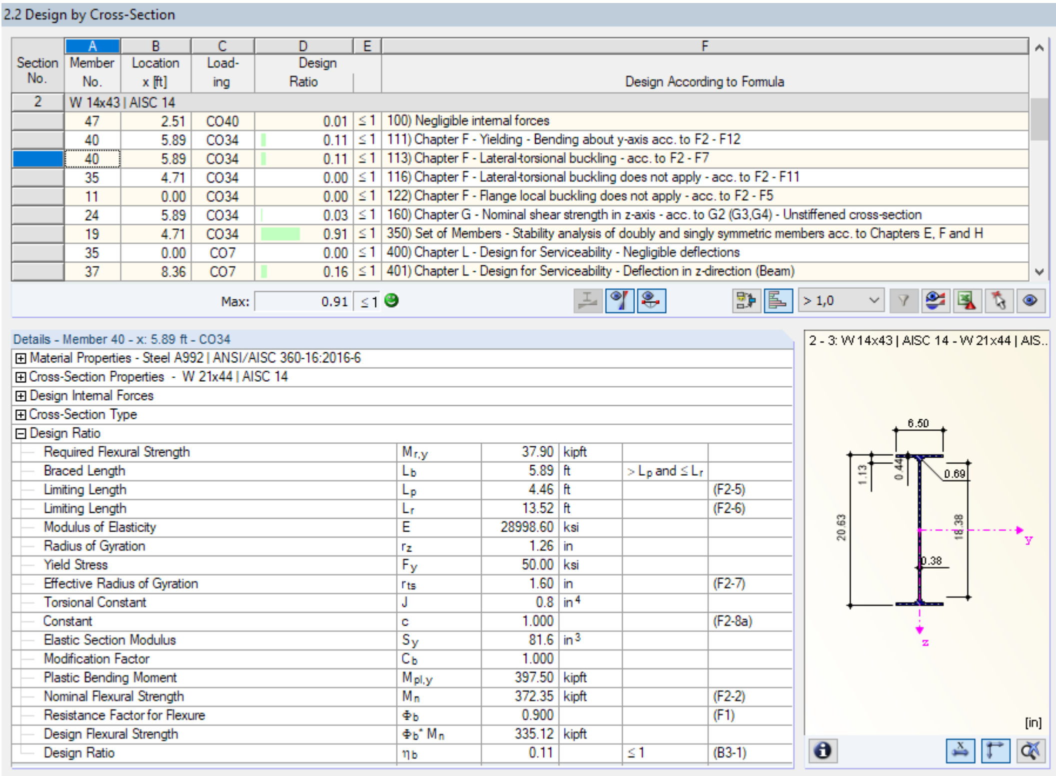


Figure 4.3: Window *2.2 Design by Cross-Section*

This window lists the maximum ratios of all members and loadings selected for design, sorted by cross-section. For each section, the results are given for cross-section design, stability analysis, and serviceability limit state design.

If there is a tapered member, the cross-sections of the member start and end are listed separately.

4.3 Design by Set of Members

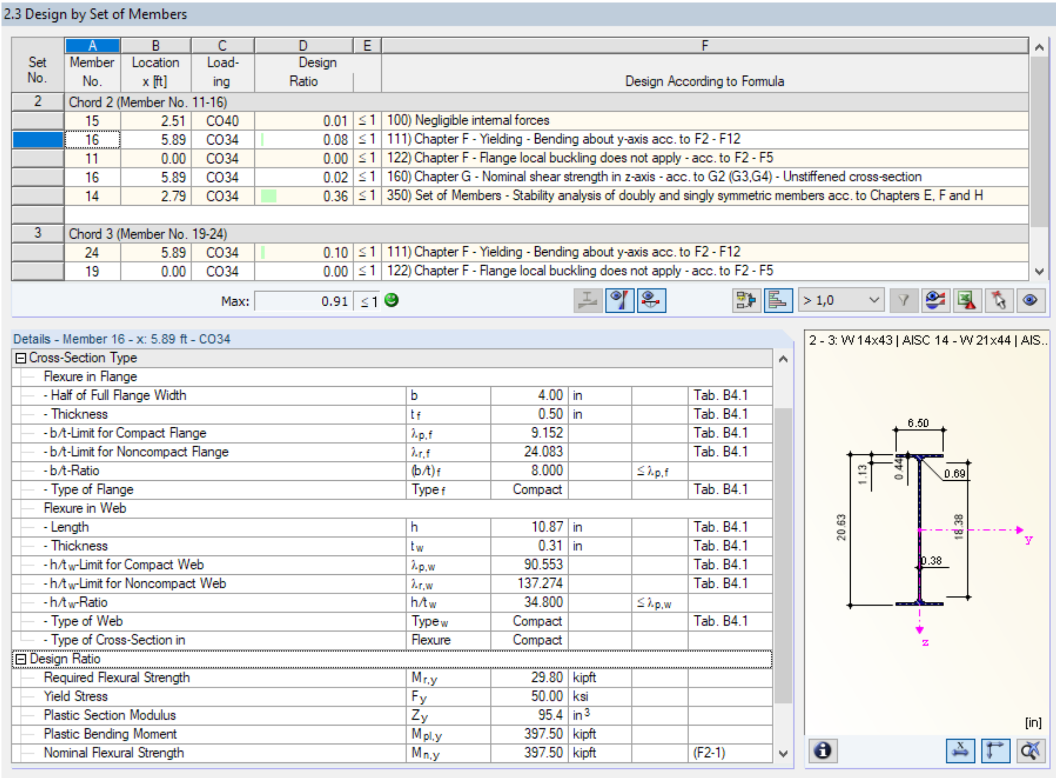


Figure 4.4: Window *2.3 Design by Set of Members*

This result window is displayed when you have selected at least one set of members for the design. It lists the maximum design ratios sorted by set of members.

The *Member No.* column shows the number of the member within the set which has the maximum ratio with respect to the specific design criterion.

The output by set of members clearly presents the design for an entire structural group, e.g. a frame.

4.4 Design by Member

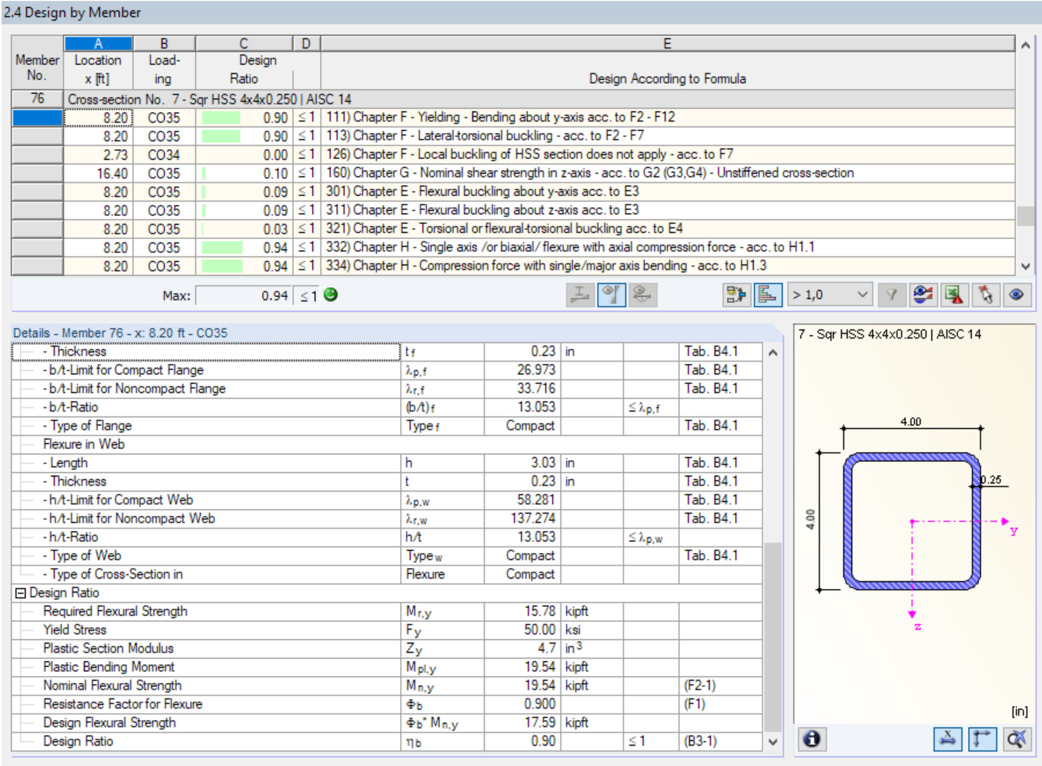


Figure 4.5: Window *2.4 Design by Member*

This result window lists the maximum ratios of the individual designs for each member. The columns are described Chapter 4.1 on page 37.

4.5 Design by x-Location

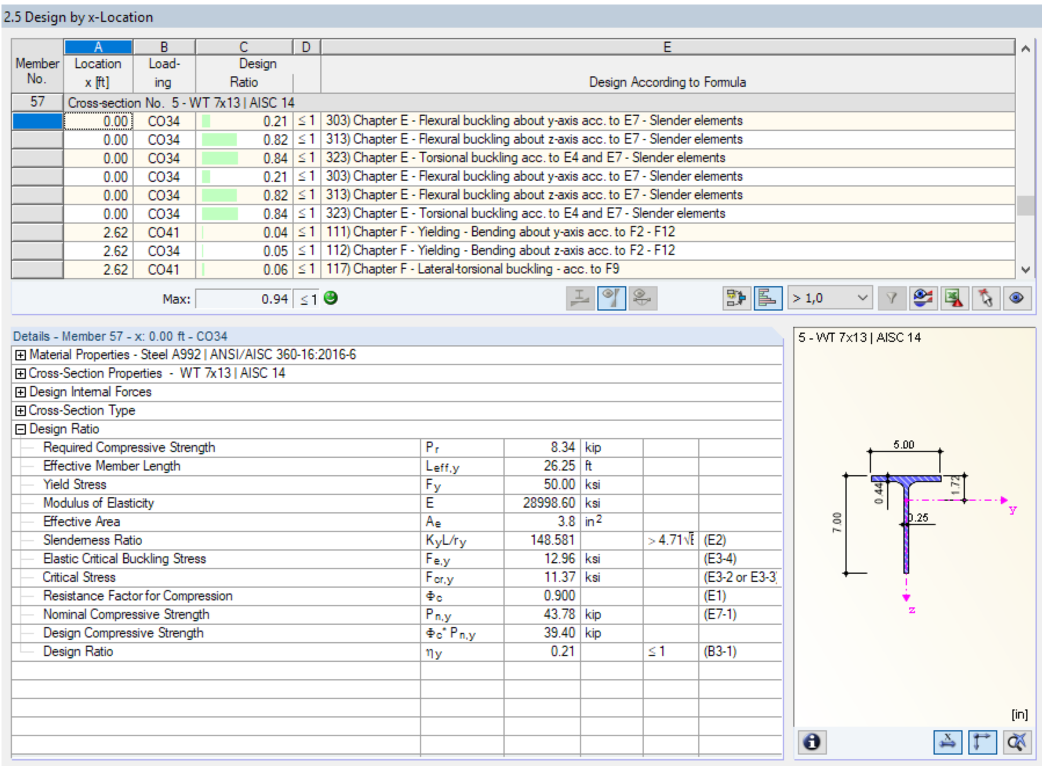


Figure 4.6: Window *2.5 Design by x-Location*

This result window lists the maxima for each member at all locations *x* resulting from the division points in RFEM or RSTAB:

- Start and end nodes
- Division points according to optionally defined member division (see RFEM Table 1.16 or RSTAB Table 1.6)
- Member divisions according to specification for member results (see RFEM/RSTAB dialog box *Calculation Parameters*, tab *Global Calculation 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 [ft]	Load-ing	Forces [kip] P	$V_{y/u}$	$V_{z/v}$	Moments [kipft] T_a	$M_{y/u}$	$M_{z/v}$	Design According to Formula
11	Cross-section No. 3 - W 21x44 | AISC 14 ... 2 - W 14x43 | AISC 14								
	0.00	CO35	-4.43	0.00	2.88	0.00	-25.55	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1.
	0.00	CO34	-4.42	0.00	2.87	0.00	-25.26	0.00	122) Chapter F - Flange local buckling does not apply - acc. to
	0.00	CO35	-4.43	0.00	2.88	0.00	-25.55	0.00	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (
	4.71	CO34	-4.33	0.00	2.23	0.00	-13.24	0.01	350) Set of Members - Stability analysis of doubly and singly syn
12	Cross-section No. 2 - W 14x43 | AISC 14								
	7.53	CO34	-4.16	0.00	1.27	0.00	2.30	0.03	100) Negligible internal forces
	0.00	CO35	-4.27	0.00	2.19	0.00	-10.95	0.01	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1.
	0.00	CO34	-4.26	0.00	2.18	0.00	-10.70	0.01	122) Chapter F - Flange local buckling does not apply - acc. to
	0.00	CO35	-4.27	0.00	2.19	0.00	-10.95	0.01	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (
	0.00	CO34	-4.26	0.00	2.18	0.00	-10.70	0.01	350) Set of Members - Stability analysis of doubly and singly syn
13	Cross-section No. 2 - W 14x43 | AISC 14								
	16.73	CO41	-4.73	0.00	-0.93	0.00	1.67	0.01	100) Negligible internal forces
	8.36	CO34	-6.09	0.00	-0.14	0.00	8.18	0.02	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1.
	0.00	CO34	-6.20	0.00	0.88	0.00	5.10	0.04	122) Chapter F - Flange local buckling does not apply - acc. to
	0.00	CO34	-6.20	0.00	0.88	0.00	5.10	0.04	350) Set of Members - Stability analysis of doubly and singly syn
14	Cross-section No. 2 - W 14x43 | AISC 14								
	0.00	CO41	-4.74	0.00	0.87	0.00	1.67	0.01	100) Negligible internal forces
	8.36	CO35	-6.10	0.00	0.13	0.00	8.07	0.02	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1.
	0.00	CO34	-5.98	0.00	1.14	0.00	2.74	0.01	122) Chapter F - Flange local buckling does not apply - acc. to
	16.73	CO34	-6.20	0.00	-0.90	0.00	4.75	0.03	350) Set of Members - Stability analysis of doubly and singly syn
15	Cross-section No. 2 - W 14x43 | AISC 14								
	5.02	CO40	-3.35	0.00	-1.22	0.00	-1.68	0.02	100) Negligible internal forces
	10.04	CO34	-4.26	0.00	-2.20	0.00	-11.26	0.01	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1.
	0.00	CO34	-4.13	0.00	-0.99	0.00	4.75	0.03	122) Chapter F - Flange local buckling does not apply - acc. to
	10.04	CO34	-4.26	0.00	-2.20	0.00	-11.26	0.01	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (
	0.00	CO34	-4.13	0.00	-0.99	0.00	4.75	0.03	350) Set of Members - Stability analysis of doubly and singly syn
16	Cross-section No. 2 - W 14x43 | AISC 14 ... 3 - W 21x44 | AISC 14								
	5.89	CO34	-4.42	0.00	-2.89	0.00	-25.93	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F1.

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

For all designed members, the internal forces are listed that effectuate the maximum ratios of each type of design.

Location x

This column shows the x-locations where the maximum design ratios occur.

Loading

This column displays the numbers of the load case, load or result combination whose internal forces result in the maximum design ratios.

Forces / Moments

For each member, these columns present the axial and shear forces as well as the torsional and bending moments which give the maximum ratios in the respective cross-section designs, stability analyses, and serviceability limit state designs.

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members

3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [ft]	B Load-ing	C Forces [kip] P	D V_y	E V_z	F Moments [kipft] T_a	G M_y	H M_z	I Design According to Formula
2	Chord 2 (Member No. 11-16)								
	7.53	CO34	-4.16	0.00	1.27	0.00	2.30	0.03	100) Negligible internal forces
	5.89	CO34	-4.42	0.00	-2.89	0.00	-25.93	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	0.00	CO34	-4.42	0.00	2.87	0.00	-25.26	0.00	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-4.42	0.00	-2.89	0.00	-25.93	0.00	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	16.73	CO34	-6.20	0.00	-0.90	0.00	4.75	0.03	350) Set of Members - Stability analysis of doubly and singly symme
3	Chord 3 (Member No. 19-24)								
	5.89	CO34	-2.91	0.02	-3.63	0.00	-37.05	-0.26	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	0.00	CO34	-2.90	0.00	3.61	0.00	-36.54	0.00	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-2.91	0.02	-3.63	0.00	-37.05	-0.26	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	0.00	CO34	-2.90	0.00	3.61	0.00	-36.54	0.00	350) Set of Members - Stability analysis of doubly and singly symme
4	Chord 4 (Member No. 27-32)								
	5.89	CO34	-2.95	0.00	-3.61	0.00	-37.87	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	0.00	CO34	-2.95	0.00	3.60	0.00	-37.48	0.00	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-2.95	0.00	-3.61	0.00	-37.87	0.00	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	1.96	CO35	-2.92	0.00	3.34	0.00	-30.86	-0.01	350) Set of Members - Stability analysis of doubly and singly symme
6	Chord 6 (Member No. 43-48)								
	5.89	CO34	-6.15	0.00	-4.33	-0.01	-51.26	-0.01	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	5.89	CO34	-6.15	0.00	-4.33	-0.01	-51.26	-0.01	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F7
	4.71	CO34	-6.03	0.00	3.67	0.00	-31.54	-0.03	116) Chapter F - Lateral-torsional buckling does not apply - acc. to
	0.00	CO34	-6.12	0.00	4.30	0.01	-50.33	-0.01	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-6.15	0.00	-4.33	-0.01	-51.26	-0.01	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	16.73	CO34	-7.88	0.01	-2.40	0.01	2.93	-0.04	301) Chapter E - Flexural buckling about y-axis acc. to E3
	5.89	CO35	-6.02	0.00	3.53	0.00	-27.73	-0.03	303) Chapter E - Flexural buckling about y-axis acc. to E7 - Slender
	16.73	CO34	-7.88	0.01	-2.40	0.01	2.93	-0.04	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.00	CO35	-6.13	0.00	4.32	0.01	-50.84	-0.01	313) Chapter E - Flexural buckling about z-axis acc. to E7 - Slender
	16.73	CO34	-7.88	0.01	-2.40	0.01	2.93	-0.04	321) Chapter E - Torsional or flexural-torsional buckling acc. to E4
	0.00	CO35	-6.13	0.00	4.32	0.01	-50.84	-0.01	323) Chapter E - Torsional buckling acc. to E4 and E7 - Slender ele
	5.89	CO34	-6.15	0.00	-4.33	-0.01	-51.26	-0.01	332) Chapter H - Single axis /or biaxial/ flexure with axial compress
	5.89	CO34	-6.15	0.00	-4.33	-0.01	-51.26	-0.01	334) Chapter H - Compression force with single/major axis bending
	5.89	CO35	-6.13	0.00	-4.32	-0.01	-50.84	-0.01	350) Set of Members - Stability analysis of doubly and singly symme

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

For each set of members, this window shows the internal forces that result in the maximum design ratios. The respective equations according to [1] are referred to in the last column.

4.8 Member Slendernesses

Details...

Window 3.3 is shown when you have selected the respective check box in the *Details* dialog box (see Figure 3.5, page 34).

3.3 Member Slendernesses

Member No.	A Under Stress	B Length L [ft]	C Major Axis y/u $k_{y/u}$ [-]	D $i_{y/u}$ [in]	E $\lambda_{y/u}$ [-]	F Minor Axis z/v $k_{z/v}$ [-]	G $i_{z/v}$ [in]	H $\lambda_{z/v}$ [-]	I
34	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868	
35	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979	
36	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595	
37	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991	
38	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991	
39	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595	
40	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979	
41	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868	
42	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868	
43	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979	
44	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595	
45	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991	
46	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991	
47	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595	
48	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979	
49	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868	
56	Compression / Flexure	22.97	1.000	2.04	134.866	1.000	1.54	178.507	
57	Compression / Flexure	26.25	1.000	2.04	154.133	1.000	1.54	204.008	
58	Compression / Flexure	22.97	1.000	2.04	134.866	1.000	1.54	178.507	
63	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391	
69	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391	
75	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391	
76	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391	
81	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391	
87	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391	

Members with compression / flexure:

Max $K_{y/u}L / r_{y/u}$: 154.133 ≤ 200

Max $K_{z/v}L / r_{z/v}$: 204.008 > 200

Figure 4.9: Window *3.3 Member Slendernesses*

The table lists the effective slenderness ratios of the designed members for both directions of the principal axes. They are determined in compliance with the load type.

Details...

Below the list, you find a comparison of the most unfavorable values with the limit values that have been defined in the *Details* dialog box (see Figure 3.5, page 34).

Members of the types 'tension' or 'cable' are not included in this table.

This window is only informative. It does not provide any stability analysis of slendernesses.

4.9 Parts List by Member

Finally, RF-/STEEL AISC provides a summary of all cross-sections contained in the design case.

4.1 Parts List by Member

Part No.	A Cross-Section Description	B Number of Members	C Length [ft]	D Total Length [ft]	E Surface Area [ft²]	F Volume [ft³]	G Unit Weight [lb/ft]	H Weight [lb]	I Total Weight [kip]
1	1 - W 16x67 | AISC 14	4	19.69	78.74	492.11	10.72	66.70	1313.02	5.25
2	2 - W 14x43 | AISC 14 ... 3 - W 21x44 | AISC	4	5.89	23.55	126.26	2.09	43.56	256.42	1.03
3	2 - W 14x43 | AISC 14	4	10.04	40.15	202.83	3.51	42.88	430.40	1.72
4	2 - W 14x43 | AISC 14	4	16.73	66.92	338.05	5.86	42.88	717.33	2.87
5	5 - WT 7x19 | AISC 14	2	22.97	45.93	102.80	1.78	18.99	436.11	0.87
6	5 - WT 7x19 | AISC 14	1	26.25	26.25	58.74	1.02	18.99	498.41	0.50
7	7 - Sqr HSS 4x4x0.250 | AISC 14	6	16.40	98.43	124.19	2.30	12.21	200.37	1.20
8	10 - L 4x3x5/16 | AISC 14	4	25.62	102.50	116.67	1.49	7.11	182.25	0.73
9	8 - RB 7/8 | AISC	8	23.43	187.44	42.94	0.78	2.05	47.95	0.38
Sum		37		669.89	1604.59	29.55			14.55

Figure 4.10: Window *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 (see Figure 3.5, page 34).

Part No.

The program automatically assigns item numbers to members with identical features.

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

Column B shows how many similar members are used for each part.

Length

This column shows the respective length of an individual member.

Total Length

In this column, the product determined from the two previous columns is given.

Surface Area

For each item, the program gives the surface area relative to the total length. This area is determined from the *Surface Area* of the cross-sections. It can be checked in Windows 1.3 and 2.1 to 2.5 in the cross-section properties (see Figure 2.11, page 13).

Volume

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

Unit Weight

The unit mass of a cross-section is related to the length of one meter. For tapered sections, the program averages both cross-section masses.

Weight

The values of this column represent the products of the entries in columns C and G.

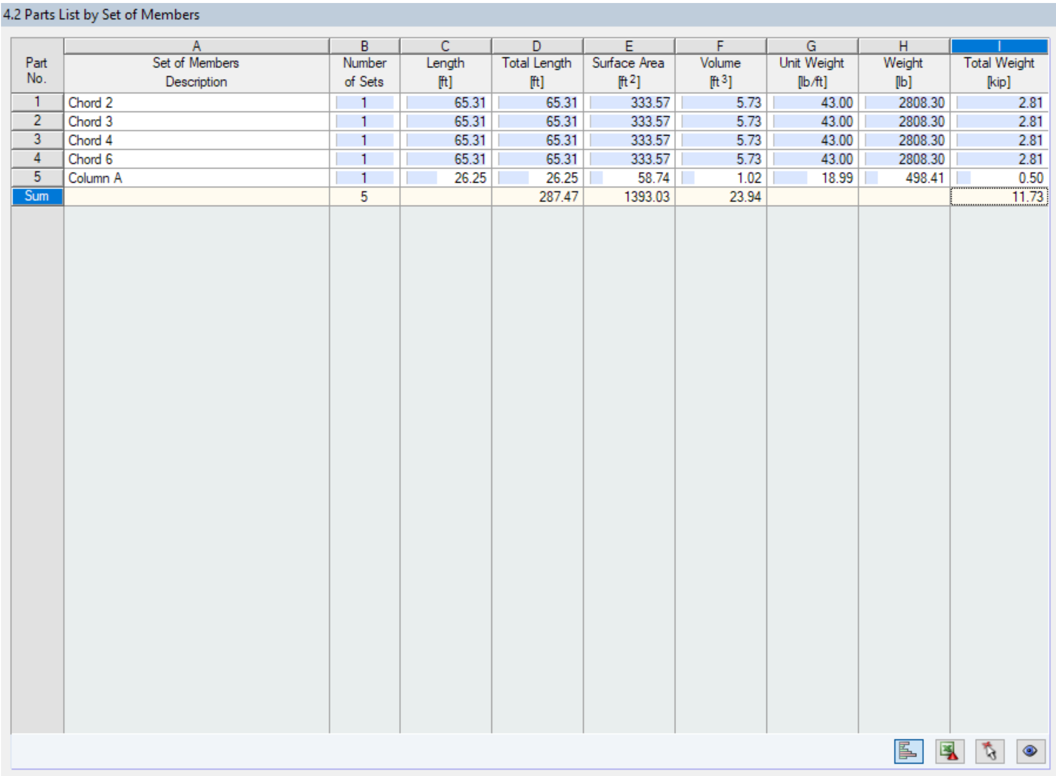
Total Weight

The final column gives the total mass of each sectional part.

Sum

At the end of the list, you find a summary of the values in the columns B, D, E, F, and I. The last row of the *Total Weight* column shows the total amount of required steel.

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 [ft]	D Total Length [ft]	E Surface Area [ft²]	F Volume [ft³]	G Unit Weight [lb/ft]	H Weight [lb]	I Total Weight [kip]
1	Chord 2	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.81
2	Chord 3	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.81
3	Chord 4	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.81
4	Chord 6	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.81
5	Column A	1	26.25	26.25	58.74	1.02	18.99	498.41	0.50
Sum		5		287.47	1393.03	23.94			11.73

Figure 4.11: Window *4.2 Parts List by Set of Members*

The last result window is displayed when you have selected at least one set of members for design. It represents the parts list of structural groups, for example horizontal beams.

Details on the various columns can be found in Chapter 4.9. If a set of members consists of different cross-sections, the program averages the surface area, volume, and cross-section weight.

5 Results Evaluation

You can evaluate the results in different ways. For this, the buttons below the tables are very useful.

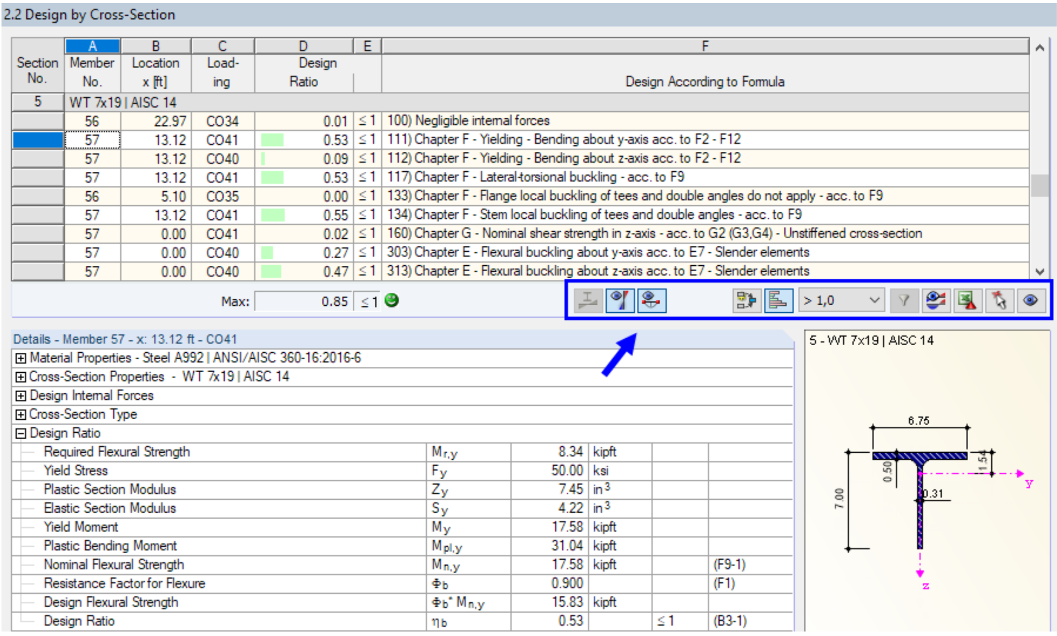


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Mode Shape	Opens *Mode Shape View* Window showing buckling shape → Chapter 5.4, page 52
	Ultimate Limit State Design	Shows or hides the results of the ULS design
	Serviceability Limit State Design	Shows or hides the results of the SLS design
	Result Combination	Creates a new result combination from the governing load cases and load combinations
	Color Bars	Shows or hides the colored relation scales in the tables
> 1,0 / > 1,0 / Max / Define	Filter Parameters	Describes the filter criterion for the output in the tables: Design ratios greater than 1, maximum value, or user-defined limit
	Apply Filter	Displays only rows where the filter parameters are valid (ratio > 1, maximum, user-defined limit)
	Result Diagrams	Opens the *Result Diagram on Member* Window → Chapter 5.2, page 50
	Excel Export	Exports the table to MS Excel → Chapter 7.4.3, page 60
	Member Selection	Option to select a member graphically to display its results in the table
	View Mode	Jumps to the work window of RFEM or RSTAB to view objects in model

Table 5.1: Buttons in Windows 2.1 to 2.5

5.1 Results in RFEM/RSTAB Model

You can also evaluate the design results in the work window of RFEM or RSTAB.

Background graphic and view mode

The work window of RFEM or RSTAB in the background is useful for you to find the location of a particular member in the model: There, the member selected in the RF-/STEEL AISC result window is highlighted. Furthermore, an arrow indicates the relevant location x on this member.

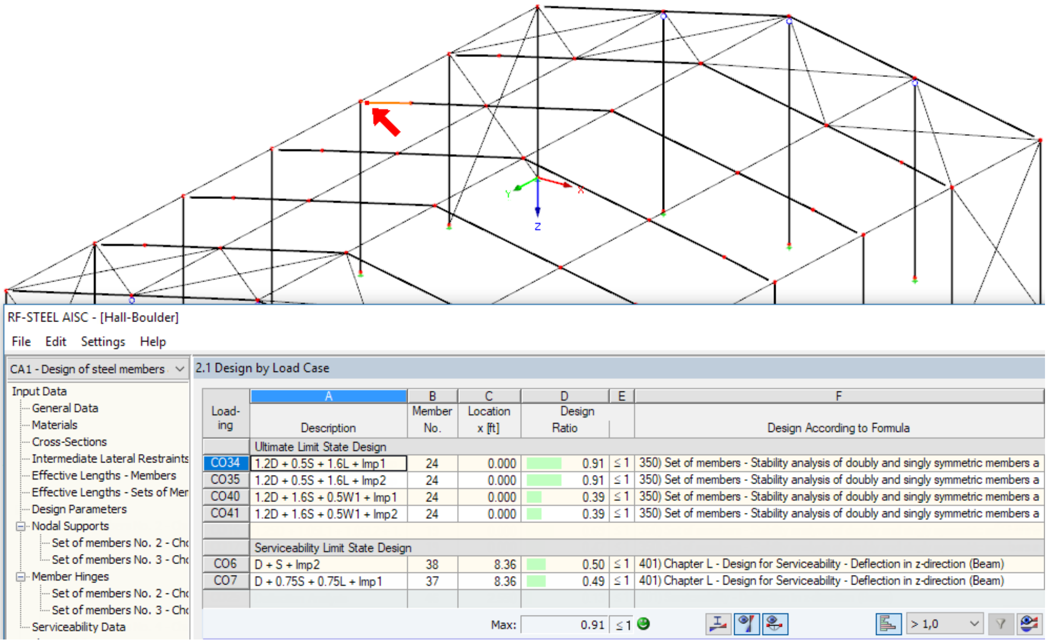
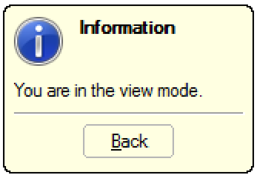


Figure 5.2: Indication of member and relevant *Location x* in RFEM model



If you cannot improve the display by moving the RF-/STEEL AISC module window, click the button to activate the *view mode*. Thus, you hide the module window so that you can change the view in the user interface of RFEM or RSTAB. In the view mode, you can use the functions of the *View* menu, e.g. zoom, move, or rotate the view. The arrow will remain visible when doing so.

Click [Back] to return to the RF-/STEEL AISC module.

RFEM/RSTAB work window

You can also check the design ratios graphically in the RFEM/RSTAB model: Click [Graphics] to quit the design module. In the work window of RFEM or RSTAB, the design ratios are now displayed like the internal forces of a load case.

In the *Results* navigator, you can select whether the ratios of the ULS and/or SLS designs are to be displayed.

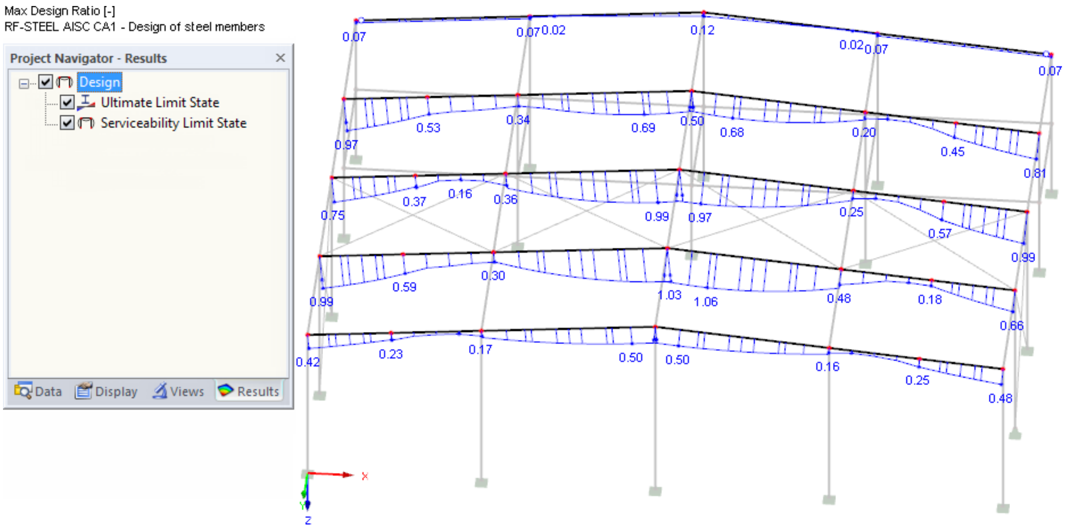
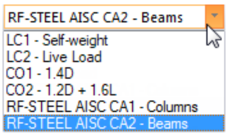


Figure 5.3: *Results* navigator for *Ultimate Limit State* and *Serviceability Limit State* designs

To turn the display of the design ratios on or off, use the [Show Results] button which is familiar from the display of internal forces. To switch the result values on or off, click the [Show Values] button next to it.

The tables of RFEM or RSTAB are of no relevance for the steel design results.



You can set the relevant RF-/STEEL AISC design case in the list of the toolbar.

The graphical representation of the design results can be controlled in the *Display* navigator, item **Results** → **Members**. The ratios are shown *Two-Colored* by default.

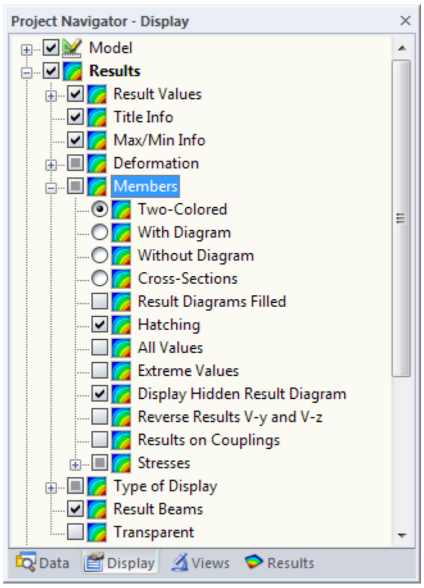


Figure 5.4: *Display* navigator: *Results* → *Members*

When you have selected a multicolor display (options *With/Without Diagram* or *Cross-Sections*), the color panel is available. It provides the common control functions which are described in the RFEM/RSTAB manual, Chapter 3.4.6.

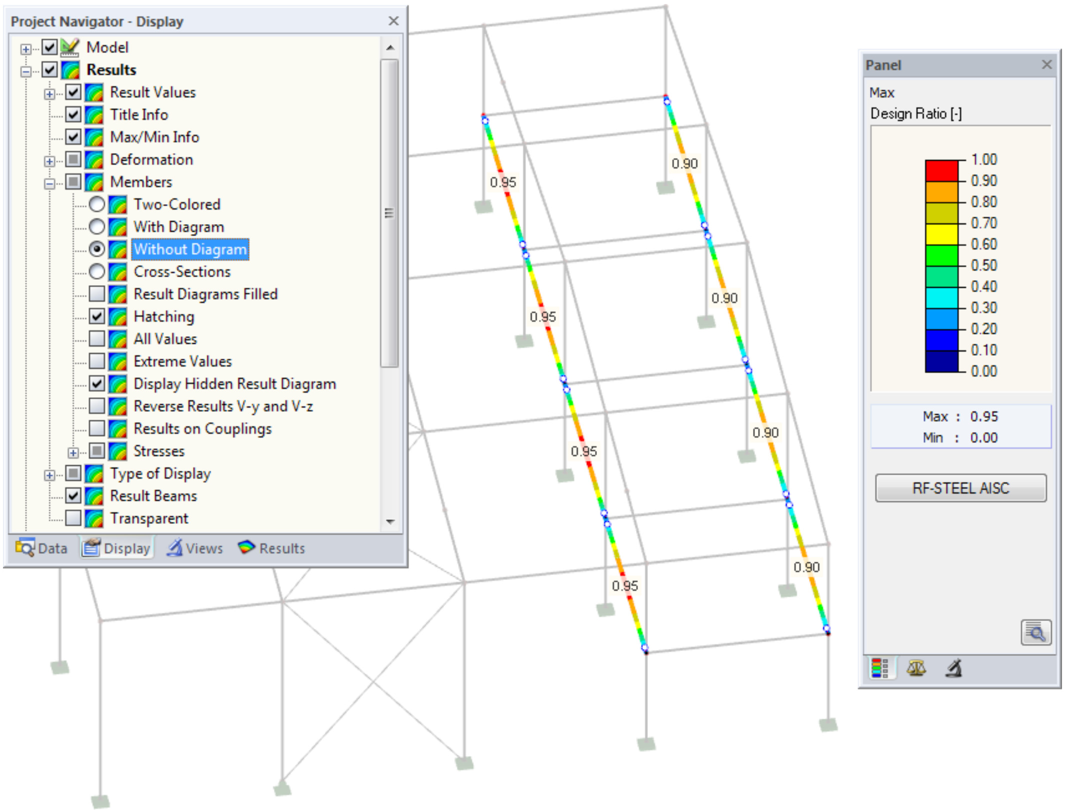


Figure 5.5: Design ratios with display option *Without Diagram*

The graphics of the design results can be transferred to the printout report (see Chapter 6.2, page 54).

RF-STEEL AISC

To return to the add-on module, use the [RF-/STEEL AISC] button in the panel.

5.2 Result Diagram

You can graphically evaluate the design ratios in a result diagram, without using the work window of RFEM or RSTAB.

Select the member (or set of members) in the RF-/STEEL AISC result window by clicking in the relevant table row. Then click the button to open the *Result Diagram on Member* dialog box. This button is located below the upper table (see Figure 5.1, page 46).

If you want to open the result diagram in the work window of RFEM or RSTAB, select the menu

Results → Result Diagrams for Selected Members

or the toolbar button shown on the left.

A new window opens. It presents the distribution of the maximum design values on the member or set of members.

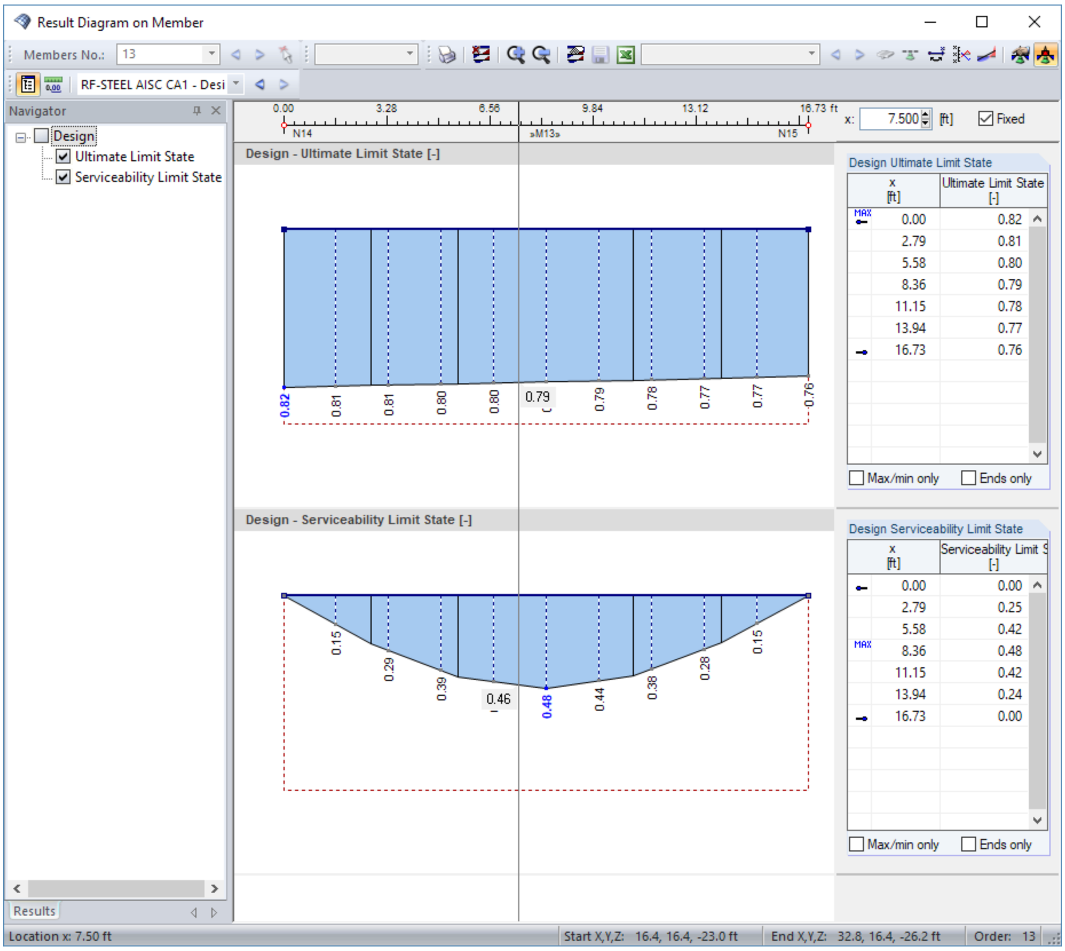
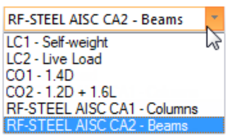


Figure 5.6: Dialog box *Result Diagram on Member*

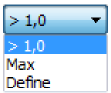
You can switch the ULS and SLS results on or off in the *Results* navigator.



Use the list in the toolbar to select the relevant RF-/STEEL AISC design case.

The *Result Diagram on Member* dialog box is described in the RFEM or RSTAB manual, Chapter 9.5.

5.3 Filter for Results



The RF-/STEEL AISC result windows allow you to sort the results by various criteria. In addition, you can use the filter options for the tables (see Figure 5.1, page 46) to reduce the numerical output according to specific ratios. This function is described in the *Knowledge Base* on our website: https://www.dlubal.com/en/support-and-learning/support/knowledge-base/000733

Furthermore, you can apply the filter options described in Chapter 9.9 of the RFEM manual or Chapter 9.7 of the RSTAB manual to evaluate the results graphically.

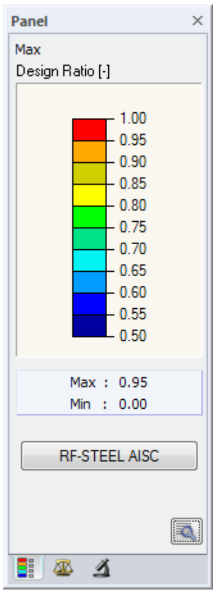
You can also use the *Visibility* options for RF-/STEEL AISC to filter the members and evaluate them (see RFEM manual, Chapter 9.9.1 or RSTAB manual, Chapter 9.7.1).

Filtering design ratios

The design ratios can be used as filter criteria in the RFEM/RSTAB work window which you access by clicking [Graphics]. To apply this filter function, the panel must be displayed. If it is not, select.

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

or use the toolbar button shown on the left.



The panel is described in the RFEM/RSTAB manual, Chapter 3.4.6. The filter settings for the results can be defined in the first tab (Color scale). As this tab is not available for the two-colored results display, you have to set the display option *Colored With/Without Diagram* or *Cross-Sections* in the *Display* navigator (see Figure 5.4, page 48).

As seen in the figure to the left, the color spectrum can be set in such a way that only ratios greater than 0.50 are shown in the color ranges between blue and red.

Filtering members

In the *Filter* tab of the control panel, you can specify the numbers of particular members to display their results exclusively, i.e. filtered. This function is described in the RFEM manual, Chapter 9.9.3 or RSTAB manual, Chapter 9.7.3.

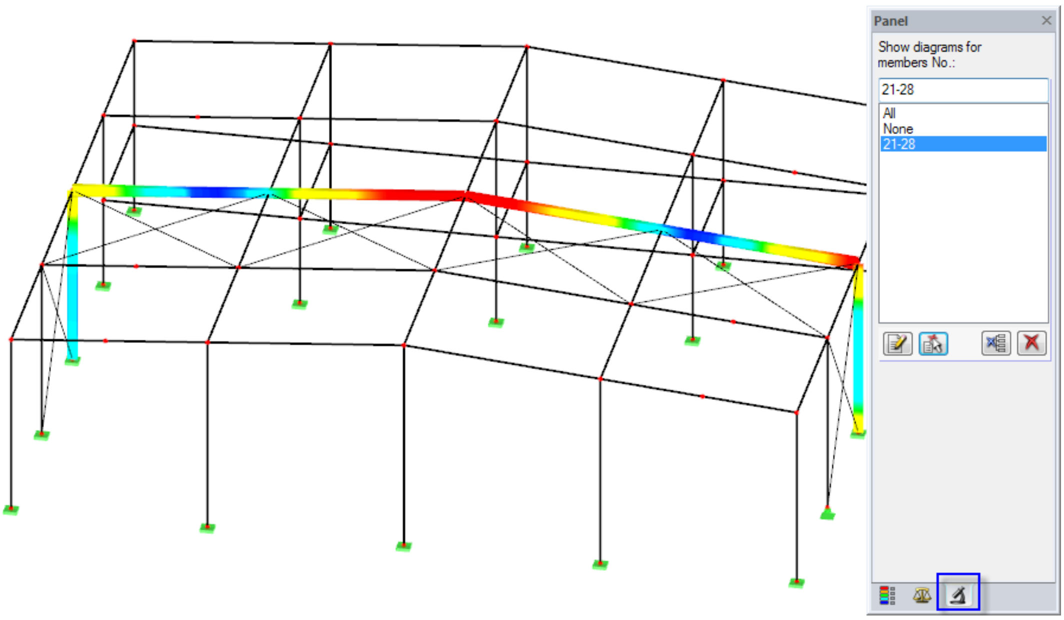


Figure 5.7: Filtering design ratios of one frame

Unlike the *Visibility* function, the entire model is displayed. Figure 5.7 shows the design ratios of one frame only. All other members are displayed in the model, but they have no design results.

5.4 Mode Shapes

The mode shapes of the members and sets of members can be checked graphically in a separate window: Go to the relevant object in the result window, and then select its results row of *Lateral torsion buckling* (for members) or *Set of members - Stability analysis* (for sets of members).

Then click the [Mode Shapes] button which is located below the upper table.

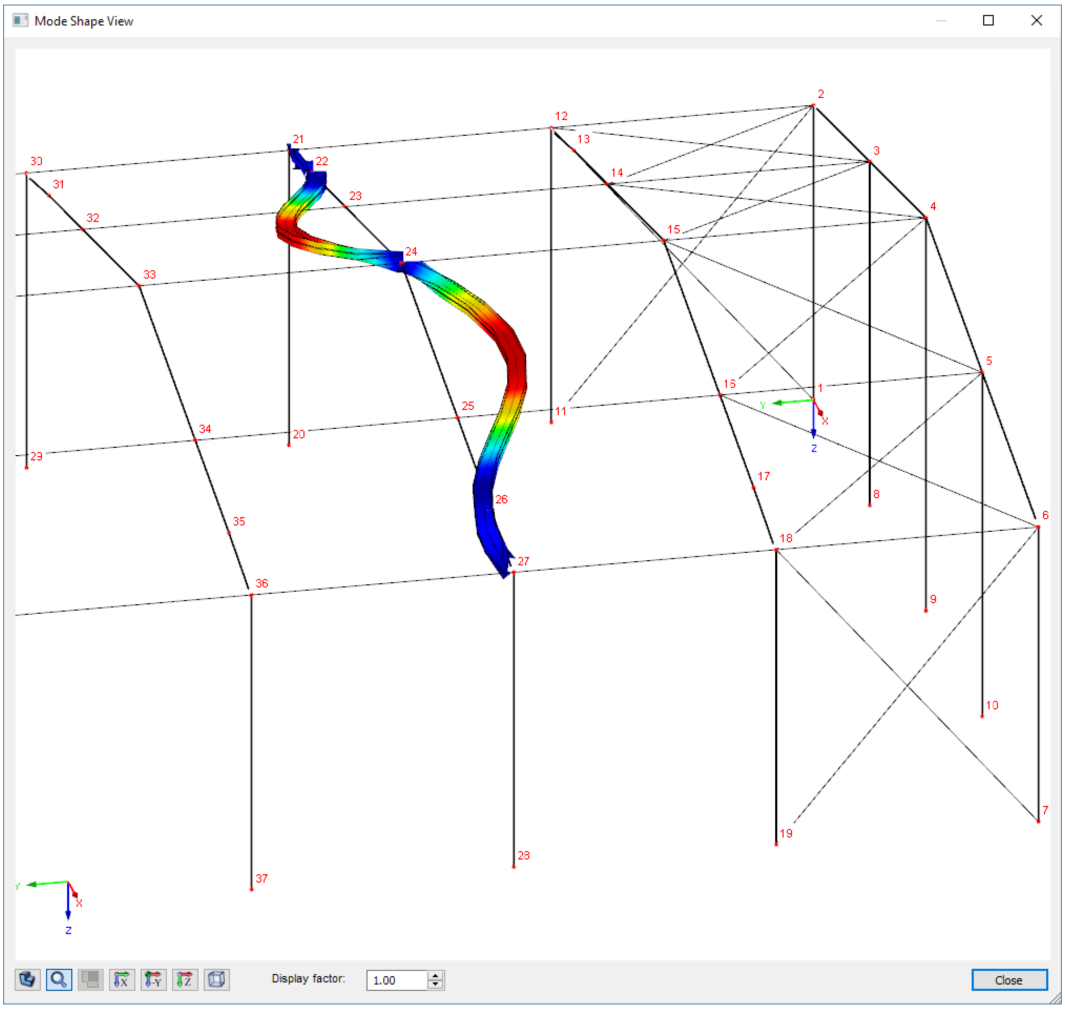


Figure 5.8: Mode shape of a set of members

The mode shapes are created automatically when the elastic critical moment is determined. They are not available in numerical form, however, only graphically.

The buttons below the graphic are described in Table 2.2 on page 23. Use the *Display factor* next to the buttons to scale the representation of the mode shape.

The following *Knowledge Base* article on our website illustrates how mode shapes can be used:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001483

6 Printout

6.1 Printout Report

Similarly to RFEM or RSTAB, the program generates a printout report for the RF-/STEEL AISC results. It can be supplemented by graphics and descriptions. The selection in the printout report controls which data of the design module are included in the final printout.

The printout report is described in the RFEM or RSTAB manual. In particular, Chapter 10.1.3.5 *Selecting Data of Add-on Modules* describes how the input and output data of add-on modules can be selected.

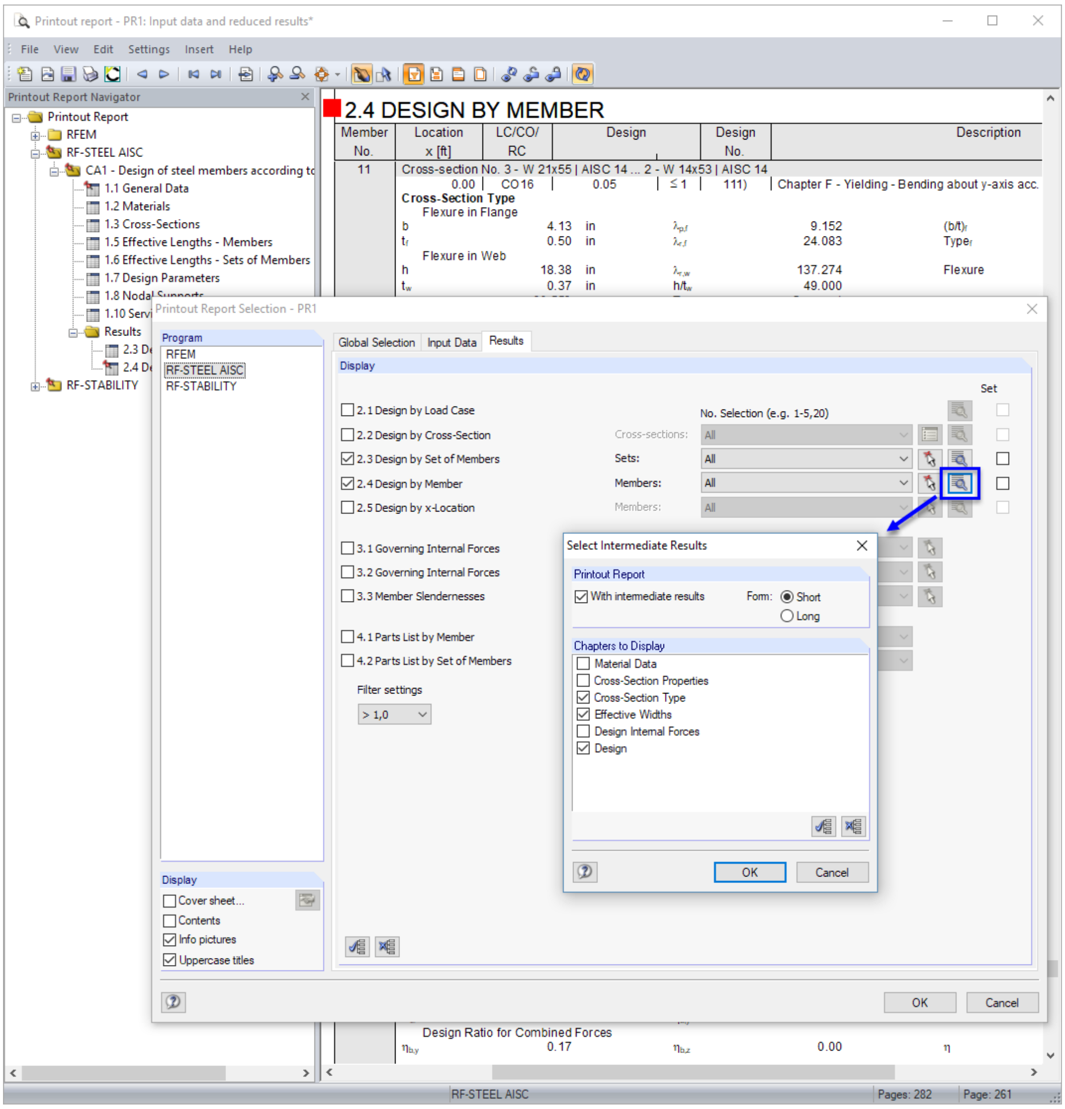


Figure 6.1: Selecting designs and intermediate results in printout report

Click the [Details] button when you want to include all or specific intermediate results in the printout report. They can be documented in a *Short* (compact list) or *Long* (descriptive list) form.

If you work on complex models featuring many design cases, you can split the data into several printout reports. This allows for a clearly arranged documentation.

6.2 Graphic Printout

In RFEM or RSTAB, you can add any picture of the work window to the printout report or send it directly to the printer. In this way, the design ratios shown on the RFEM/RSTAB model can be used for the documentation.

The printing of graphics is described in the RFEM or RSTAB manual, Chapter 10.2.

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

File → Print Graphic

or use the toolbar button shown on the left.

Figure 6.2: [Print Graphic] button in RFEM toolbar

The *Graphic Printout* dialog box appears.

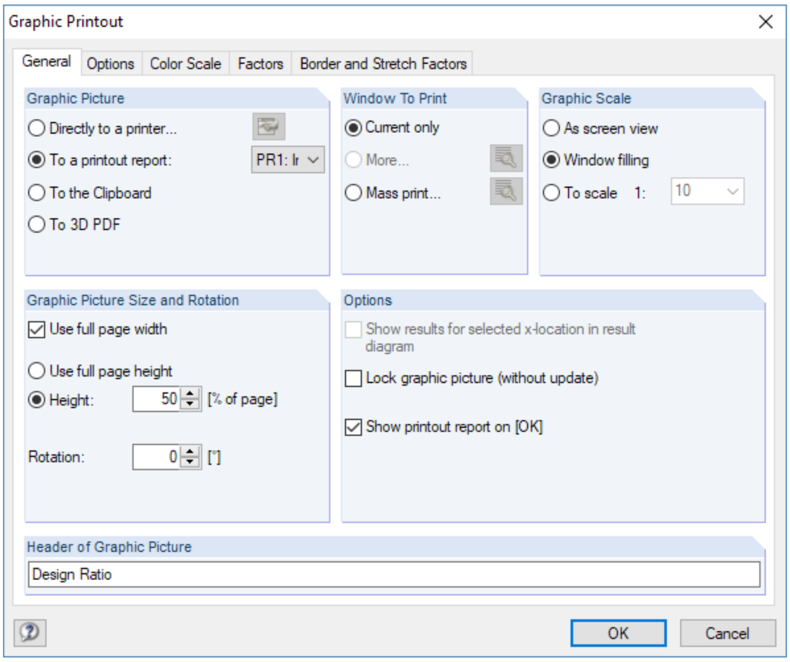
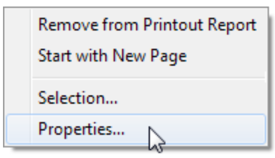


Figure 6.3: Dialog box *Graphic Printout*, tab *General*

The dialog box is described in the RFEM or RSTAB manual, Chapter 10.2. This chapter also describes the other tabs of the dialog box.

You can move a graphic anywhere within the printout report by using the drag-and-drop function.



If you want to modify an image in the printout report, right-click the relevant entry in the navigator of the printout report. The *Properties* option in the shortcut menu opens the *Graphic Printout* dialog box again. It offers you several options to adjust the image.

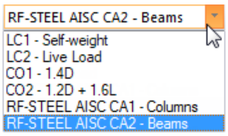
7 General Functions

This chapter describes the menu functions and export options for the design results.

7.1 Design Cases

Design cases allow you to arrange members for specific analyses. In this way, you can combine groups of structural components or analyze members with particular design specifications, e.g. modified materials, cross-sections, strength design, or optimization parameters.

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



To calculate a RF-/STEEL AISC design case, you can also use the load case list in the toolbar of RFEM or RSTAB.

Create design case

To create a new design case, use the RF-/STEEL AISC menu and select

File → **New Case**.

The following dialog box appears.

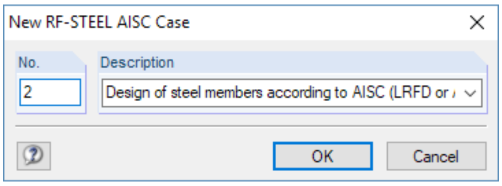


Figure 7.1: Dialog box *New RF-STEEL AISC Case*

Enter a *No.* (one that is still available) for the new design case and an optional *Description*. It facilitates the selection in the load case list.

Then click [OK] to open the *1.1 General Data* Window of RF-/STEEL AISC where you can enter the data of the new design case.

Rename design case

To change the description of a design case, use the RF-/STEEL AISC menu and select

File → **Rename Case**.

The following dialog box appears.

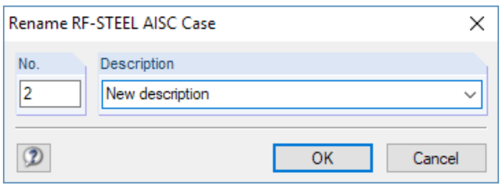


Figure 7.2: Dialog box *Rename RF-STEEL AISC Case*

You can specify a different *Description* as well as a different *No.* for the design case.

Copy design case

To copy the input data of the current design case, use the RF-/STEEL AISC menu and select

File → **Copy Case**.

The following dialog box appears.

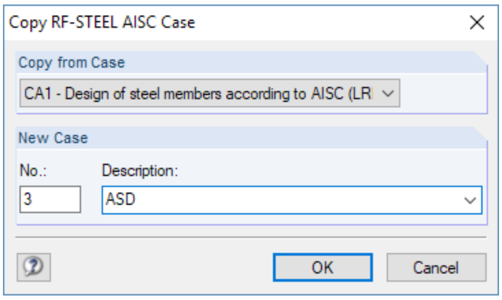


Figure 7.3: Dialog box *Copy RF-STEEL AISC Case*

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

Delete design case

To delete a design case, use the RF-/STEEL AISC menu and select

File → **Delete Case**.

The following dialog box appears.

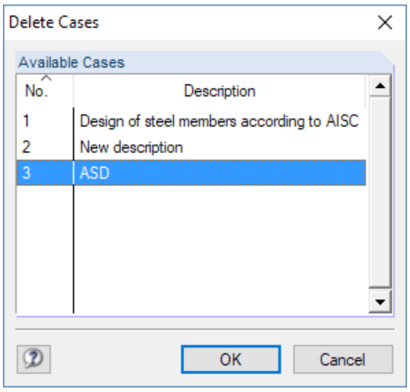
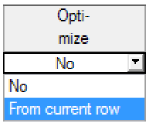


Figure 7.4: Dialog box *Delete Cases*

Select the design case in the list of *Available Cases*. To delete this case, click [OK].

7.2 Cross-Section Optimization



The design module offers you the option to optimize overstressed or little utilized cross-sections. Open the drop-down list in column D resp. E in Window *1.3 Cross-Sections* (see Figure 2.9, page 11) and select the optimization option *From current row*.

You can also start the optimization in the result windows via the shortcut menu.

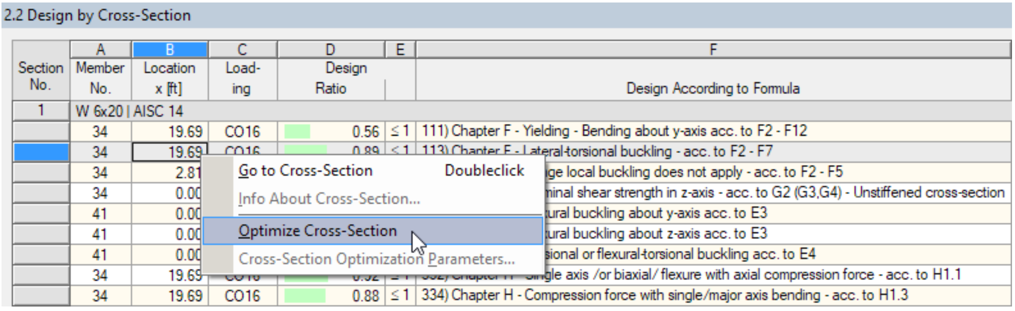


Figure 7.5: Shortcut menu to *Optimize Cross-Section*

During the optimization, the module determines the section that fulfills the analysis requirements in the "optimal" way, i.e. comes as close as possible to the maximum allowable design ratio specified in the *Details* dialog box (see Figure 3.5, page 34). The required cross-sectional properties are calculated with the internal forces of RFEM or RSTAB. If a different cross-section proves to be more favorable, it will be used for the design. In this case, the graphic in Window 1.3 shows two cross-sections – the original section from RFEM or RSTAB and the optimized one (see Figure 7.7).

When you optimize a parametric cross-section, the following dialog box appears:

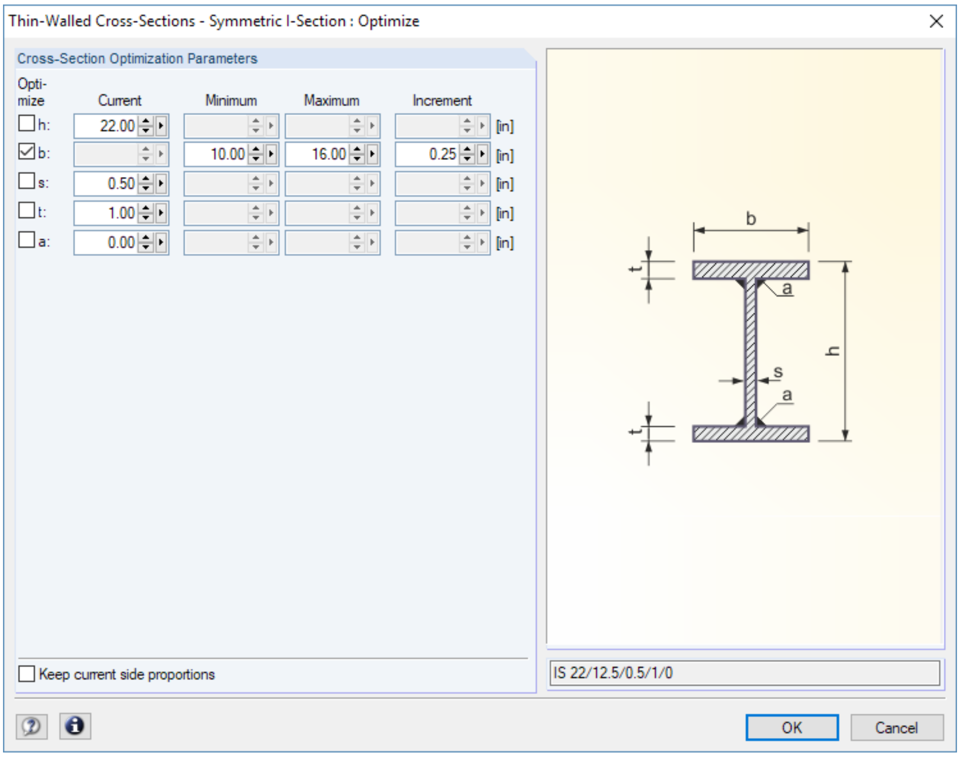


Figure 7.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section: Optimize*

By selecting the check box(es) in the *Optimize* column, you decide which parameter(s) you want to modify. They activate the *Minimum* and *Maximum* columns where you can specify the upper and lower limits of each parameter. The *Increment* column controls the interval in which the value of the parameter varies during the optimization.

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

Cross-sections composed of combined rolled cross-sections cannot be optimized.

Please note that the optimization does not recalculate the internal forces with the modified cross-sections: It is up to you to decide which sections should be transferred to RFEM or RSTAB for a new analysis. As a result of optimized cross-sections, the internal forces may vary considerably because of the changed stiffnesses of the model. Therefore, it is recommended to recalculate the internal forces resulting from the modified cross-sections after the first optimization, and then to optimize the sections once again.

To export the modified cross-section(s) to RFEM or RSTAB, go to Window *1.3 Cross-Sections* and select

Edit → Export All Cross-Sections to RFEM/RSTAB.

You can also use the shortcut menu in Window 1.3 to export one or all optimized cross-sections to RFEM or RSTAB.

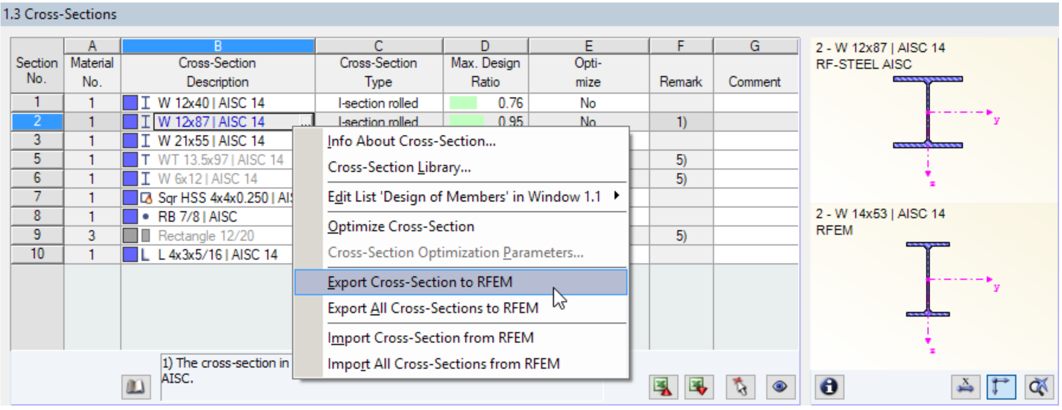


Figure 7.7: Shortcut menu in Window *1.3 Cross-Sections*

Before the modified cross-sections are transferred to RFEM or RSTAB, a confirmation is required as to whether the RFEM/RSTAB results should be deleted.

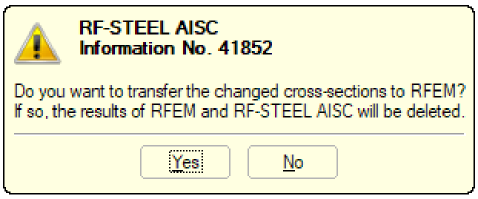
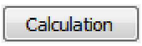


Figure 7.8: Confirmation when exporting cross-sections



By approving the confirmation and starting the [Calculation] in the RF-/STEEL AISC module, the internal forces of RFEM or RSTAB as well as the design ratios will be determined in one single calculation run.

If the modified cross-sections have not been exported to RFEM or RSTAB yet, you can reimport the original sections in the design module by using the last two menu options shown in Figure 7.7. Please note that this shortcut menu is only available in Window *1.3 Cross-sections*.

When optimizing a tapered member, the program modifies the cross-sections of the member start and member end. For the intermediate locations, the second moments of area are linearly interpolated. Since those values are considered with the fourth power, the designs may be inaccurate if the depths of the start and end cross-sections differ considerably. It is then recommended to divide the taper into several members, thus modeling the taper layout manually.

7.3 Units and Decimal Places

The units and decimal places of RFEM or RSTAB and of all add-on modules are managed in one dialog box. To define the units for RF-/STEEL AISC, select

Settings → **Units and Decimal Places.**

The dialog box which is familiar from RFEM or RSTAB appears. RF-/STEEL AISC is preset in the *Program / Module* list.

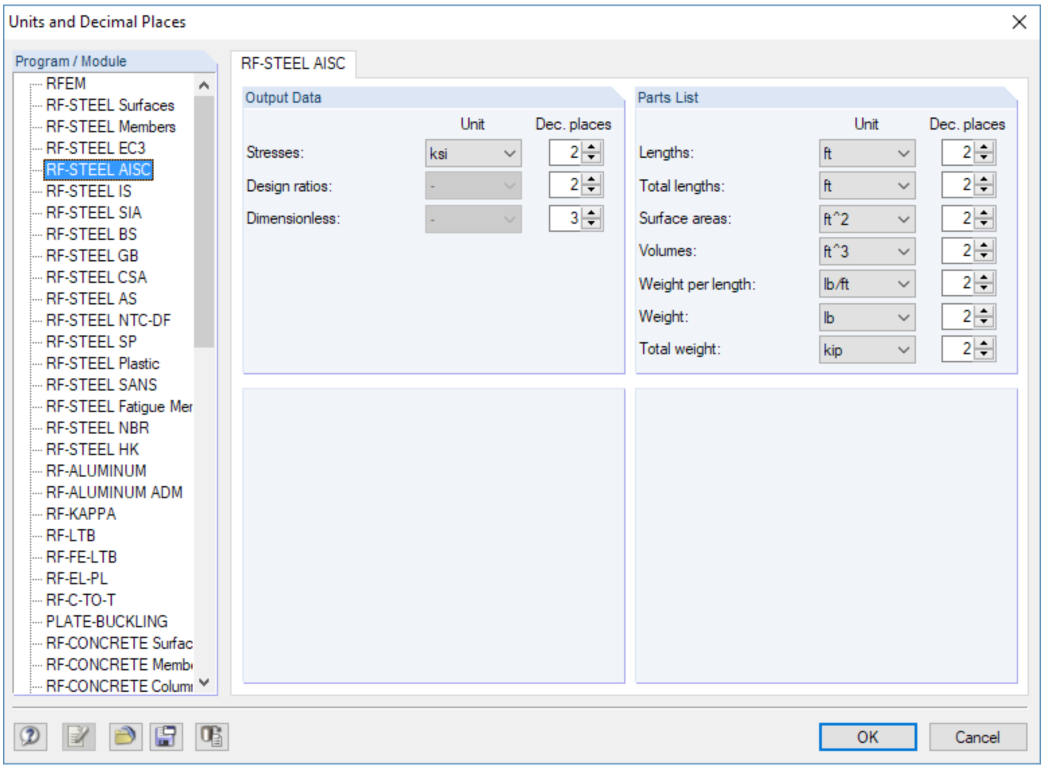


Figure 7.9: Dialog box *Units and Decimal Places*

You can save the settings as a user-defined profile to reuse them in other models. Those functions are described in the RFEM or RSTAB manual, Chapter 11.1.3.

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

If you have modified the materials in RF-/STEEL AISC for the design, you can export those materials to RFEM or RSTAB in a similar way as you export cross-sections: Open the *1.2 Materials* Window and then select

Edit → Export All Materials to RFEM/RSTAB.

You can also export the modified materials to RFEM or RSTAB by using the shortcut menu in Window 1.2.

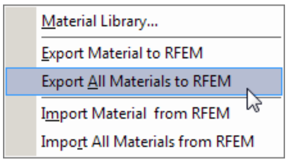
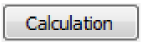


Figure 7.10: Shortcut menu of Window *1.2 Materials*



Before the modified materials are transferred to RFEM or RSTAB, a confirmation is required as to whether the results of the main program should be deleted. When you approve this confirmation and then start the [Calculation] in RF-/STEEL AISC, the new internal forces and design ratios will be determined in one single calculation run.

If the modified materials have not been exported to RFEM or RSTAB yet, you can transfer the original materials to the design module via the last two menu options shown in Figure 7.10. Please note that this shortcut menu is only available in Window *1.2 Materials*.

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

If you have adjusted the effective lengths in RF-/STEEL AISC for the design, you can export the modified values to RFEM or RSTAB in a similar way as you export cross-sections: Go to Window *1.5 Effective Lengths - Members* and then select

Edit → Export All Effective Lengths to RFEM/RSTAB.

You can also use the corresponding option on the shortcut menu of Window 1.5.

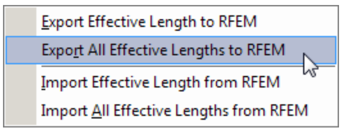


Figure 7.11: Shortcut menu of Window *1.5 Effective Lengths - Members*

Before the modified effective lengths are transferred to RFEM or RSTAB, a confirmation has to be approved as to whether the results of the main program should be deleted.

7.4.3 Exporting Results

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

Clipboard

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

Printout Report

You can print the data of RF-/STEEL AISC to the printout report (see Chapter 6.1, page 53). To export the tables and graphics, then select the printout report menu

File → Export to RTF.

The function is described in the RFEM or RSTAB manual, Chapter 10.1.11.

Excel

RF-/STEEL AISC provides a function for the direct data export to MS Excel or the CSV file format. To open the corresponding dialog box, select

File → Export Tables.

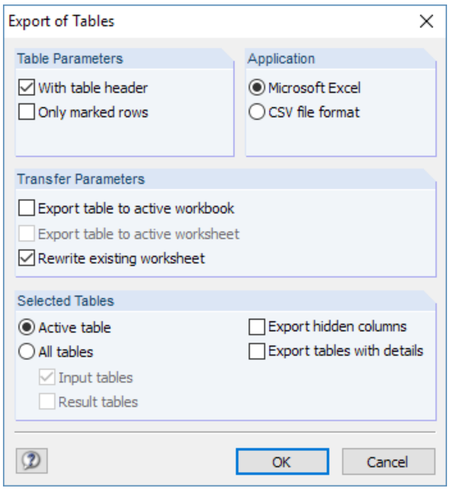
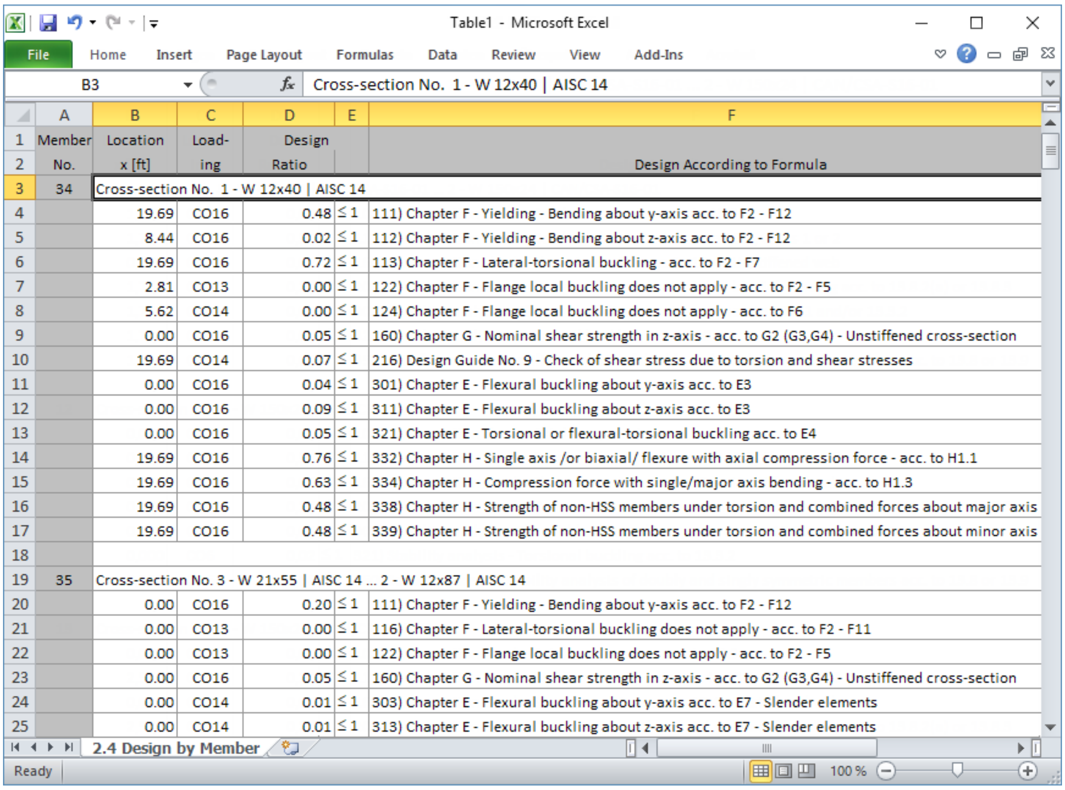


Figure 7.12: Dialog box *Export of Tables*

When you have selected the relevant options, you can start the export by clicking [OK]. Excel will be started automatically, i.e. you do not have to open the program before.



	A	B	C	D	E	F
1	Member	Location	Load-	Design		
2	No.	x [ft]	ing	Ratio		Design According to Formula
3	34	Cross-section No. 1 - W 12x40 | AISC 14				
4		19.69	CO16	0.48	≤ 1	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
5		8.44	CO16	0.02	≤ 1	112) Chapter F - Yielding - Bending about z-axis acc. to F2 - F12
6		19.69	CO16	0.72	≤ 1	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F7
7		2.81	CO13	0.00	≤ 1	122) Chapter F - Flange local buckling does not apply - acc. to F2 - F5
8		5.62	CO14	0.00	≤ 1	124) Chapter F - Flange local buckling does not apply - acc. to F6
9		0.00	CO16	0.05	≤ 1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,G4) - Unstiffened cross-section
10		19.69	CO14	0.07	≤ 1	216) Design Guide No. 9 - Check of shear stress due to torsion and shear stresses
11		0.00	CO16	0.04	≤ 1	301) Chapter E - Flexural buckling about y-axis acc. to E3
12		0.00	CO16	0.09	≤ 1	311) Chapter E - Flexural buckling about z-axis acc. to E3
13		0.00	CO16	0.05	≤ 1	321) Chapter E - Torsional or flexural-torsional buckling acc. to E4
14		19.69	CO16	0.76	≤ 1	332) Chapter H - Single axis /or biaxial/ flexure with axial compression force - acc. to H1.1
15		19.69	CO16	0.63	≤ 1	334) Chapter H - Compression force with single/major axis bending - acc. to H1.3
16		19.69	CO16	0.48	≤ 1	338) Chapter H - Strength of non-HSS members under torsion and combined forces about major axis
17		19.69	CO16	0.48	≤ 1	339) Chapter H - Strength of non-HSS members under torsion and combined forces about minor axis
18						
19	35	Cross-section No. 3 - W 21x55 | AISC 14 ... 2 - W 12x87 | AISC 14				
20		0.00	CO16	0.20	≤ 1	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
21		0.00	CO13	0.00	≤ 1	116) Chapter F - Lateral-torsional buckling does not apply - acc. to F2 - F11
22		0.00	CO13	0.00	≤ 1	122) Chapter F - Flange local buckling does not apply - acc. to F2 - F5
23		0.00	CO16	0.05	≤ 1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,G4) - Unstiffened cross-section
24		0.00	CO14	0.01	≤ 1	303) Chapter E - Flexural buckling about y-axis acc. to E7 - Slender elements
25		0.00	CO14	0.01	≤ 1	313) Chapter E - Flexural buckling about z-axis acc. to E7 - Slender elements

Figure 7.13: Results in Excel

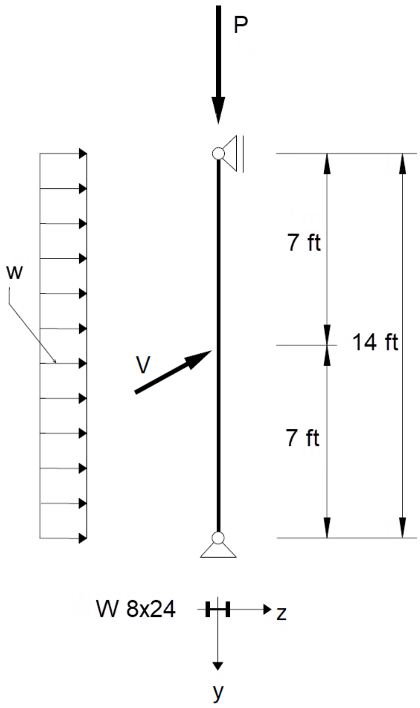
8 Example

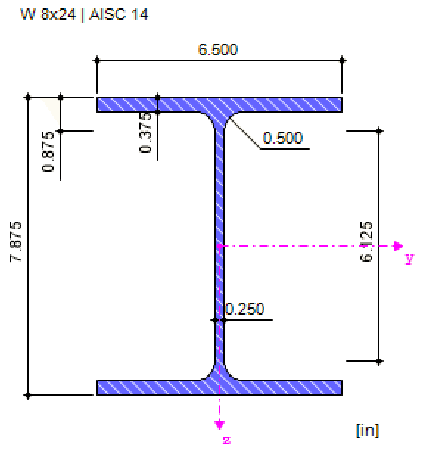
Column with Compression and Biaxial Bending

This example illustrates the stability design of a column with regard to flexural buckling and lateral torsional buckling. Furthermore, the interaction of flexure and compression is examined. The analysis follows the *Load Resistance Factor Design* provisions.

8.1 Design Values

Model and loads





Material: Steel A992

Design values of loads:
$P = 65.0$ kip
$w = 0.5$ kip/ft
$V = 2.0$ kip

Figure 8.1: Model and loads

Internal forces due to linear analysis

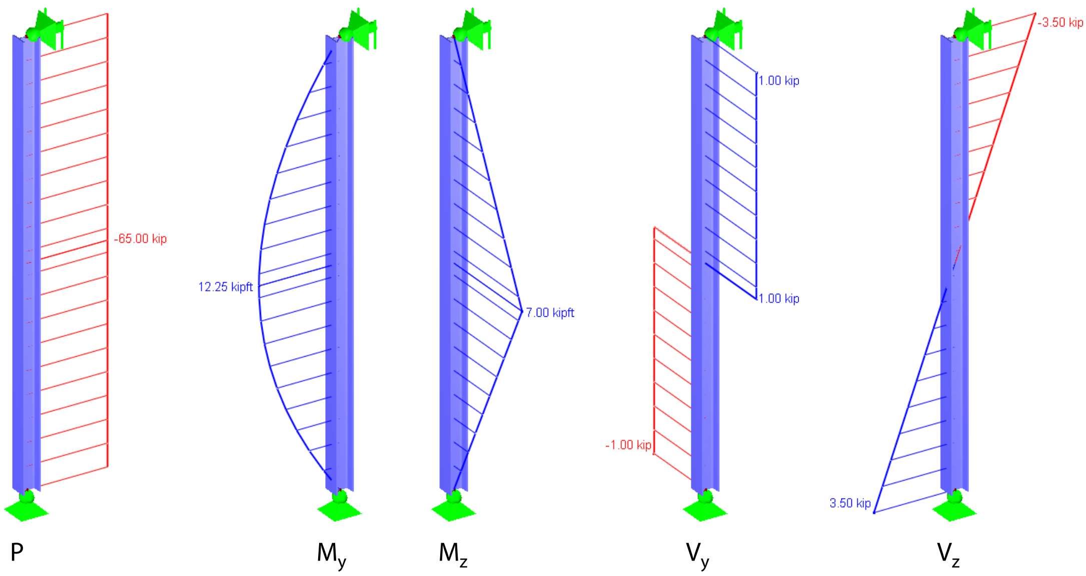
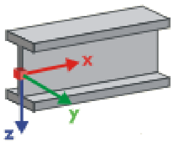


Figure 8.2: Internal forces

Design location



The design proceeds according to x-locations, i.e. for specific positions x of the member. The governing location is at $x = 7$ ft at the centerline of the member, with the following internal forces:

$P = -65.00$ kip $\quad M_y = 12.25$ kipft $\quad M_z = 7.00$ kipft $\quad V_y = 1.00$ kip $\quad V_z = 0.00$ kip

8.2 Cross-Section Properties - W 8x28

Cross-Section Property	**Symbol**	**Value**	**Unit**
Gross area	A_g	7.08	in^2
Moment of inertia about major principal axis	I_y	82.70	in^4
Moment of inertia about minor principal axis	I_z	18.30	in^4
Radius of gyration about major principal axis	r_y	3.42	in
Radius of gyration about minor principal axis	r_z	1.61	in
Cross-section weight	wt	24.11	lb/ft
Torsional constant	J	0.35	in^4
Warping constant	C_w	259.00	in^6
Elastic section modulus about major axis	S_y	20.90	in^3
Elastic section modulus about minor axis	S_z	5.63	in^3
Plastic section modulus about major axis	Z_y	23.10	in^3
Plastic section modulus about minor axis	Z_z	8.57	in^3

8.3 Material Properties - A992

Material Property	**Symbol**	**Value**	**Unit**
Modulus of elasticity	E	29,000	ksi
Shear modulus	G	11,200	ksi
Yield strength	F_y	50	ksi

8.4 Classification of Cross-Section

Compression – [1] Table B4.1a

Flange

Case 1: Flanges of rolled I-shaped sections

$b = 3.25$ in

$t = 0.375$ in

$$b/t = \frac{3.25}{0.375} = 8.67$$

$$\lambda_r = 0.56\sqrt{\frac{E}{F_y}} = 0.56\sqrt{\frac{29{,}000}{50}} = 13.49$$

$8.67 \leq 13.49 \quad \rightarrow$ nonslender

Web

Case 5: Webs of doubly symmetric rolled I-shaped sections

$b = 6.125$ in

$t = 0.25$ in

$$b/t = \frac{6.125}{0.25} = 24.50$$

$$\lambda_r = 1.49\sqrt{\frac{E}{F_y}} = 1.49\sqrt{\frac{29{,}000}{50}} = 35.88$$

$24.50 \leq 35.88 \quad \rightarrow$ nonslender

$\Rightarrow$ The section is a **nonslender**-element section.

Flexure – [1] Table B4.1b

Flange

Case 10: Flanges of rolled I-shaped sections

$b = 3.25$ in

$t = 0.375$ in

$$b/t = \frac{3.25}{0.375} = 8.67$$

$$\lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 0.38\sqrt{\frac{29{,}000}{50}} = 9.15$$

$$\lambda_r = 1.0\sqrt{\frac{E}{F_y}} = 1.0\sqrt{\frac{29{,}000}{50}} = 24.08$$

$8.67 \leq 9.15 \quad \rightarrow$ compact

Web

Case 15: Webs of doubly symmetric rolled I-shaped sections

$h = 6.125$ in

$t_w = 0.25$ in

$$h/t_w = \frac{6.125}{0.25} = 24.50$$

$$\lambda_p = 3.76\sqrt{\frac{E}{F_y}} = 3.76\sqrt{\frac{29{,}000}{50}} = 90.55$$

$$\lambda_r = 5.70\sqrt{\frac{E}{F_y}} = 5.70\sqrt{\frac{29{,}000}{50}} = 137.27$$

$24.50 \leq 90.55 \quad \rightarrow$ compact

$\Rightarrow$ The section is a **compact**-element section.

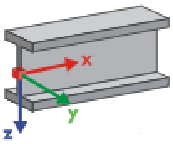
Classification in RF-/STEEL AISC

⊟ Cross-Section Type					
Uniform Compression in Flange					
- Half of Full Flange Width	b	3.250	in		Tab. B4.1
- Thickness	t_f	0.375	in		Tab. B4.1
- b/t-Limit for Compact Flange	$\lambda_{p,f}$	NA			Tab. B4.1
- b/t-Limit for Nonslender Flange	$\lambda_{r,f}$	13.487			Tab. B4.1
- b/t-Ratio	$(b/t)_f$	8.667		$\leq \lambda_{r,f}$	
- Type of Flange	$Type_f$	Nonslender			Tab. B4.1
Uniform Compression in Web					
- Length	h	6.125	in		Tab. B4.1
- Thickness	t_w	0.250	in		Tab. B4.1
- h/t_w-Limit for Compact Web	$\lambda_{p,w}$	NA			Tab. B4.1
- h/t_w-Limit for Nonslender Web	$\lambda_{r,w}$	35.884			Tab. B4.1
- h/t_w-Ratio	h/t_w	24.500		$\leq \lambda_{r,w}$	
- Type of Web	$Type_w$	Nonslender			Tab. B4.1
- Type of Cross-Section in	Compression	Nonslender			
Flexure in Flange					
- Half of Full Flange Width	b	3.250	in		Tab. B4.1
- Thickness	t_f	0.375	in		Tab. B4.1
- b/t-Limit for Compact Flange	$\lambda_{p,f}$	9.152			Tab. B4.1
- b/t-Limit for Noncompact Flange	$\lambda_{r,f}$	24.083			Tab. B4.1
- b/t-Ratio	$(b/t)_f$	8.667		$\leq \lambda_{p,f}$	
- Type of Flange	$Type_f$	Compact			Tab. B4.1
Flexure in Web					
- Length	h	6.125	in		Tab. B4.1
- Thickness	t_w	0.250	in		Tab. B4.1
- h/t_w-Limit for Compact Web	$\lambda_{p,w}$	90.553			Tab. B4.1
- h/t_w-Limit for Noncompact Web	$\lambda_{r,w}$	137.274			Tab. B4.1
- h/t_w-Ratio	h/t_w	24.500		$\leq \lambda_{p,w}$	
- Type of Web	$Type_w$	Compact			Tab. B4.1
- Type of Cross-Section in	Flexure	Compact			

Figure 8.3: Classification in RF-/STEEL AISC

8.5 Design for Compression

8.5.1 Flexural Buckling about y-Axis (Major Principal Axis)



Member slenderness:

$$\frac{K_y\,L}{r_y} = \frac{1.0 \cdot 168.00}{3.42} = 49.12$$

Limit of slenderness according to Section E3:

$$4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29{,}000}{50}} = 113.43$$

$$49.12 \leq 113.43$$

Elastic buckling stress according to Equation E3-4:

$$F_e = \frac{\pi^2\,E}{\left(\frac{K_y\,L}{r_y}\right)^2} = \frac{\pi^2 \cdot 29{,}000}{\left(\frac{1.0 \cdot 168.00}{3.42}\right)^2} = 118.61 \text{ ksi}$$

Critical stress according to Equation E3-2:

$$F_{cr} = \left(0.658^{\frac{F_y}{F_e}}\right) F_y = \left(0.658^{\frac{50}{118.61}}\right) \cdot 50 = 41.91 \text{ ksi}$$

Nominal compressive strength according to Equation E3-1:

$$P_n = F_{cr}\,A_g = 41.91 \cdot 7.08 = 296.72 \text{ kip}$$

Design compressive strength according to Section E1:

$$\phi\,P_n = 0.9 \cdot 296.72 = 267.05 \text{ kip}$$

Design ratio according to Equation B3-1:

$$\eta = \frac{P}{\phi\,P_n} = \frac{65.00}{267.05} = 0.24 \leq 1.00 \qquad \Rightarrow \textbf{OK}$$

Buckling design in RF-/STEEL AISC

Design Ratio					
Required Compressive Strength	P_r	65.00	kip		
Effective Member Length	$L_{eff,y}$	14.00	ft		
Yield Stress	F_y	50.00	ksi		
Modulus of Elasticity	E	28998.60	ksi		
Slenderness Ratio	K_yL/r_y	49.16		$\leq 4.71\sqrt{E/F_y}$	(E2)
Elastic Critical Buckling Stress	$F_{e,y}$	118.45	ksi		(E3-4)
Critical Stress	$F_{cr,y}$	41.90	ksi		(E3-2 or E3-3)
Resistance Factor for Compression	Φ_c	0.90			(E1)
Nominal Compressive Strength	$P_{n,y}$	296.67	kip		(E3-1)
Design Compressive Strength	$\Phi_c{}^* P_{n,y}$	267.00	kip		
Design Ratio	η_y	0.24		≤ 1	(B3-1)

Figure 8.4: Design details in RF-/STEEL AISC (check no. 301)

8.5.2 Flexural Buckling about z-Axis (Minor Principal Axis)

Member slenderness:

$$\frac{K_z L}{r_z} = \frac{1.0 \cdot 168.00}{1.61} = 104.35$$

Limit of slenderness according to Section E3:

$$4.71\sqrt{\frac{E}{F_y}} = 4.71\sqrt{\frac{29{,}000}{50}} = 113.43$$

$$104.35 \leq 113.43$$

Elastic buckling stress according to Equation E3-4:

$$F_e = \frac{\pi^2 E}{\left(\frac{K_z L}{r_z}\right)^2} = \frac{\pi^2 \cdot 29{,}000}{\left(\frac{1.0 \cdot 168.00}{1.61}\right)^2} = 26.29 \text{ ksi}$$

Critical stress according to Equation E3-2:

$$F_{cr} = \left(0.658^{\frac{F_y}{F_e}}\right) F_y = \left(0.658^{\frac{50}{26.29}}\right) \cdot 50 = 22.56 \text{ ksi}$$

Nominal compressive strength according to Equation E3-1:

$$P_n = F_{cr} A_g = 22.56 \cdot 7.08 = 159{,}72 \text{ kip}$$

Design compressive strength according to Section E1:

$$\phi P_n = 0.9 \cdot 159.72 = 143.75 \text{ kip}$$

Design ratio according to Equation B3-1:

$$\eta = \frac{P}{\phi P_n} = \frac{65.00}{143.75} = 0.45 \leq 1.00 \qquad \Rightarrow \textbf{OK}$$

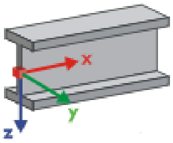
Buckling design in RF-/STEEL AISC

⊟ Design Ratio					
Required Compressive Strength	P_r	65.00	kip		
Effective Member Length	$L_{eff,z}$	14.00	ft		
Yield Stress	F_y	50.00	ksi		
Modulus of Elasticity	E	28998.60	ksi		
Slenderness Ratio	K_zL/r_z	104.50		$\leq 4.71\sqrt{E/F_y}$	(E2)
Elastic Critical Buckling Stress	$F_{e,z}$	26.21	ksi		(E3-4)
Critical Stress	$F_{cr,z}$	22.50	ksi		(E3-2 or E3-3)
Resistance Factor for Compression	Φ_c	0.90			(E1)
Nominal Compressive Strength	$P_{n,z}$	159.32	kip		(E3-1)
Design Compressive Strength	$\Phi_c^* P_{n,z}$	143.39	kip		
Design Ratio	η_z	0.45		≤ 1	(B3-1)

Figure 8.5: Design details in RF-/STEEL AISC (check no. 311)

8.6 Design for Flexure

According to the classification of the compact section (see Chapter 8.4), the limit states of yielding and lateral-torsionsal buckling are to be analyzed for flexure about the major axis (see [1], Table User Note F1.1). For flexure about the minor axis, the limit states of yielding and flange local buckling are relevant.



8.6.1 Flexure about y-Axis (Major Principal Axis)

8.6.1.1 Yielding

Nominal flexural strength according to Equation F2-1:

$$M_n = M_p = F_y\,Z_y = 50 \cdot 23.10 = 1{,}155 \text{ kipin} = 96.25 \text{ kipft}$$

Design flexural strength according to Section F1:

$$\phi_b\,M_n = 0.9 \cdot 96.25 = 86.63 \text{ kipft}$$

Design ratio according to Equation B3-1:

$$\eta = \frac{M_y}{\phi_b\,M_n} = \frac{12.25}{86.63} = 0.14 \leq 1.00 \qquad \Rightarrow \textbf{OK}$$

Yielding design in RF-/STEEL AISC

Design Ratio					
Required Flexural Strength	$M_{r,y}$	12.25	kipft		
Yield Stress	F_y	50.00	ksi		
Plastic Section Modulus	Z_y	23.10	in^3		
Plastic Bending Moment	$M_{pl,y}$	96.25	kipft		
Nominal Flexural Strength	$M_{n,y}$	96.25	kipft		(F2-1)
Resistance Factor for Flexure	Φ_b	0.90			(F1)
Design Flexural Strength	$\Phi_b{}^* M_{n,y}$	86.63	kipft		
Design Ratio	η_b	0.14		≤ 1	(B3-1)

Figure 8.6: Design details in RF-/STEEL AISC (check no. 111)

8.6.1.2 Lateral-Torsional Buckling

Lateral-torsional buckling modification factor according to Equation F1-1:

$$C_b = \frac{12.5\,M_{max}}{2.5\,M_{max} + 3\,M_A + 4\,M_B + 3\,M_C} = \frac{12.5 \cdot 12.25}{2.5 \cdot 12.25 + 3 \cdot 9.16 + 4 \cdot 12.25 + 3 \cdot 9.16} = 1.14$$

Limiting laterally unbraced length for limit state of yielding according to Equation F2-5:

$$L_p = 1.76\,r_z\sqrt{\frac{E}{F_y}} = 1.76 \cdot 1.61\sqrt{\frac{29{,}000}{50}} = 68.24 \text{ in} = 5.69 \text{ ft}$$

Limiting unbraced length for limit state of lateral-torsional buckling according to Equation F2-6:

$$L_r = 1.95\,r_{ts}\frac{E}{0.7\,F_y}\sqrt{\frac{J\,c}{S_y\,h_o} + \sqrt{\left(\frac{J\,c}{S_y\,h_o}\right)^2 + 6.76\left(\frac{0.7\,F_y}{E}\right)^2}}$$

$$= 1.95 \cdot 1.81 \cdot \frac{29{,}000}{0.7 \cdot 50}\sqrt{\frac{0.35 \cdot 1.0}{20.9 \cdot 7.5} + \sqrt{\left(\frac{0.35 \cdot 1.0}{20.9 \cdot 7.5}\right)^2 + 6.76\left(\frac{0.7 \cdot 50}{29{,}000}\right)^2}} = 228.11 \text{ in} = 19.00 \text{ ft}$$

$$\text{where} \quad r_{ts} = \sqrt{\frac{\sqrt{I_z\,C_w}}{S_y}} = \sqrt{\frac{\sqrt{18.30 \cdot 259.00}}{20.90}} = 1.81 \text{ in}$$

Limit state of lateral-torsional buckling according to Equation F2-2:

$L_b = 14\text{ ft}$

$L_p < L_b \leq L_r$

Nominal flexural strength according to Equation F2-2:

$$M_n = C_b \left[M_p - (M_p - 0.75\,F_y\,S_y)\left(\frac{L_b - L_p}{L_r - L_p}\right)\right] \leq M_p$$

$$M_n = 1.14\left[1{,}155 - (1{,}155 - 0.7 \cdot 50 \cdot 20.90)\left(\frac{14 \cdot 12 - 68.24}{228.11 - 68.24}\right)\right] \leq 1{,}155\text{ kipin}$$

$$M_n = 1{,}015.44\text{ kipin} = 84.62\text{ kipft} \leq M_p$$

Design flexural strength according to Section F1:

$$\phi_b\,M_n = 0.9 \cdot 84.62 = 76.16\text{ kipft}$$

Design ratio:

$$\eta = \frac{M_y}{M_r} = \frac{12.25}{76.16} = 0.16 \leq 1.00 \qquad \Rightarrow \textbf{OK}$$

Lateral-torsional buckling design in RF-/STEEL AISC

The lateral-torsional buckling design in RF-/STEEL AISC is based on an eigenvalue analysis. Hence, the analytical solution described above is not reflected in the *Details* table. As can be seen in Figure 8.7, the design ratio is in adequate accordance with the analytical solution. To determine the elastic critical moment, M_{cr}, a destabilizing effect of the transverse loads was assumed in the *Details* settings for stability.

Design Ratio					
Bending about the major axis					
Required Flexural Strength	$M_{r,y}$	12.25	kipft		
Amplifier	α_{cr}	6.26			
Elastic Critical Moment for Lateral-Torsional Buckling	M_{cr}	76.70	kipft		
Nominal Flexural Strength	$M_{n,y}$	76.70	kipft		
Resistance Factor for Flexure	Φ_b	0.90			(F1)
Design Flexural Strength	$\Phi_b{}^* M_n$	69.03	kipft		
Design Ratio	η_b	0.18		≤ 1	(B3-1)

Figure 8.7: Design details in RF-/STEEL AISC (check no. 115)

8.6.2 Flexure about z-Axis (Minor Principal Axis)

8.6.2.1 Yielding

Nominal flexural strength according to Equation F6-1:

$$M_n = M_p = F_y\,Z_z \leq 1.6\,F_y S_z$$

$$M_n = 50{\cdot}8.57 = 428.5\text{ kipin} = 35.71\text{ kipft} \leq 1.6\,F_y S_z = 1.6{\cdot}50{\cdot}5.63 = 450.4\text{ kipin} = 37.53\text{ kipft}$$

Design flexural strength according to Section F1:

$$\phi_b\,M_n = 0.9 \cdot 35.71 = 32.14\text{ kipft}$$

Design ratio according to Equation B3-1:

$$\eta = \frac{M_z}{\phi_b\,M_n} = \frac{7.00}{32.14} = 0.22 \leq 1.00 \qquad \Rightarrow \textbf{OK}$$

Yielding design in RF-/STEEL AISC

Design Ratio					
Required Flexural Strength	$M_{r,z}$	7.00	kipft		
Yield Stress	F_y	50.00	ksi		
Plastic Section Modulus	Z_z	8.57	in^3		
Plastic Bending Moment	$M_{pl,z}$	35.71	kipft		
Elastic Section Modulus	S_z	5.63	in^3		
Yield Moment	M_z	23.46	kipft		
Nominal Flexural Strength	$M_{n,z}$	35.71	kipft		(F6-1)
Resistance Factor for Flexure	Φ_b	0.90			(F1)
Design Flexural Strength	$\Phi_b{}^* M_{n,z}$	32.14	kipft		
Design Ratio	η_b	0.22		≤ 1	(B3-1)

Figure 8.8: Design details in RF-/STEEL AISC (check no. 112)

8.6.2.2 Flange Local Buckling

According to Section F6.2(a), the limit state of flange local buckling does not apply for sections with compact flanges.

Flange local buckling design in RF-/STEEL AISC

Design Ratio					
b/t-Limit for Compact Flange	$\lambda_{p,f}$	9.15			Tab. B4.1
b/t-Limit for Noncompact Flange	$\lambda_{r,f}$	24.08			Tab. B4.1
b/t-Ratio	$(b/t)_f$	8.67		$\leq \lambda_{p,f}$	
Design Ratio	η_b	0.00		≤ 1	(B3-2)

Figure 8.9: Design details in RF-/STEEL AISC (check no. 124)

8.7 Design for Combined Forces

Finally, the interaction of flexure and compression is examined according to Section H1.

Limit for required axial strength according to Equation H1-1a:

$$\frac{P_r}{P_c} = \frac{65.00}{143.75} = 0.45 \geq 0.2$$

Design for combined forces according to Equation H1-1a:

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

$$\frac{65.00}{143.75} + \frac{8}{9}\left(\frac{12.25}{76.16} + \frac{7.00}{32.14}\right) = 0.45 + \frac{8}{9}(0.16 + 0.22) = 0.79$$

Design ratio:

$0.79 \leq 1.00 \qquad \Rightarrow$ **OK**

Design for combined forces in RF-/STEEL AISC

Design Ratio					
Design ratio of axial compression	η_c	0.45			acc. to Chapter E
Design ratio of flexure about y-axis	$\eta_{b,y}$	0.16			acc. to Chapter F
Design ratio of flexure about z-axis	$\eta_{b,z}$	0.22			acc. to Chapter F
Design Ratio	η	0.79		≤ 1	(H1-1a)

Figure 8.10: Design details in RF-/STEEL AISC (check no. 332)

Literature

[1] *Specification for Structural Steel Buildings*. American Institute of Steel Construction, Chicago, IL, 2016.

[2] *AISC Steel Design Guide 9: Torsional Analysis of Structural Steel Members*. American Institute of Steel Construction, Chicago, IL, 1997.

Index