Version
January 2021

Add-on Module

RF-/ALUMINUM ADM

Ultimate and Serviceability Limit State
Design According to ADM

Program Description

Am Zellweg 2
93464 Tiefenbach
Germany

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

Contents

1 Introduction

1.1 Add-on Module RF-/ALUMINUM ADM

The *Aluminum Design Manual* [1] provides design rules for aluminum structures in the United States of America. With the add-on modules RF-ALUMINUM ADM (for RFEM) and ALUMINUM ADM (for RSTAB), users obtain highly efficient and universal tools to design aluminum structures modeled with member elements according to this Specification.

This manual describes the add-on modules of both main programs conjointly under the designation **RF-/ALUMINUM ADM**.

RF-/ALUMINUM ADM carries out all typical ultimate limit state designs and stability analyses using either the allowable strength or load and resistance factor design method. They include the calculation of the allowable stresses for sections subject to tension, compression, flexure, shear, torsional or flexural-torsional buckling, as well as the design for combined forces.

For the stability analysis, you can specify for each member or set of members whether flexural buckling in the y- and/or z-direction is possible. You can also define additional lateral restraints in order to represent the model close to reality. From the boundary conditions, RF-/ALUMINUM ADM determines the slenderness ratios and the elastic critical buckling loads. The point of load application is considered as well, which has an effect on the torsional resistance.

For structures with slender cross-sections, the serviceability limit state represents an important design. The load cases, load or result combinations – defined with the relevant safety factors – can be adequately selected for the deflection analysis. If necessary, you can adjust the limit deformations to your specific demands.

The program allows you to optimize cross-sections and export the modified cross-sections to RFEM or RSTAB. Using the design cases, you can design separate structural components in complex structures or analyze variants.

Since RF-/ALUMINUM ADM is integrated in the main programs RFEM and RSTAB, the design relevant input data is already available when you open the module. After the calculation, you can evaluate the designs graphically in the RFEM or RSTAB work window.

Last but not least, it is possible to cover the analysis progress in the global printout report, from the determination of internal forces up to the results of the design.

We hope you enjoy working with RF-/ALUMINUM ADM.

Your DLUBAL team

1.2 Using the Manual

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

The explanations in this manual follow the sequence of the 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 or windows are set in *italics*.

At the end of the manual, you find the index. However, if you still cannot find what you are looking for, please check our website www.dlubal.com where you can go through the FAQ pages by selecting specific criteria.

1.3 Starting RF-/ALUMINUM ADM

In RFEM/RSTAB, you have two options to start the add-on module RF-/ALUMINUM ADM.

Menu

To start the program from the RFEM or RSTAB menu bar, click

Add-on Modules → **Design - Aluminum** → **RF-/ALUMINUM ADM**.

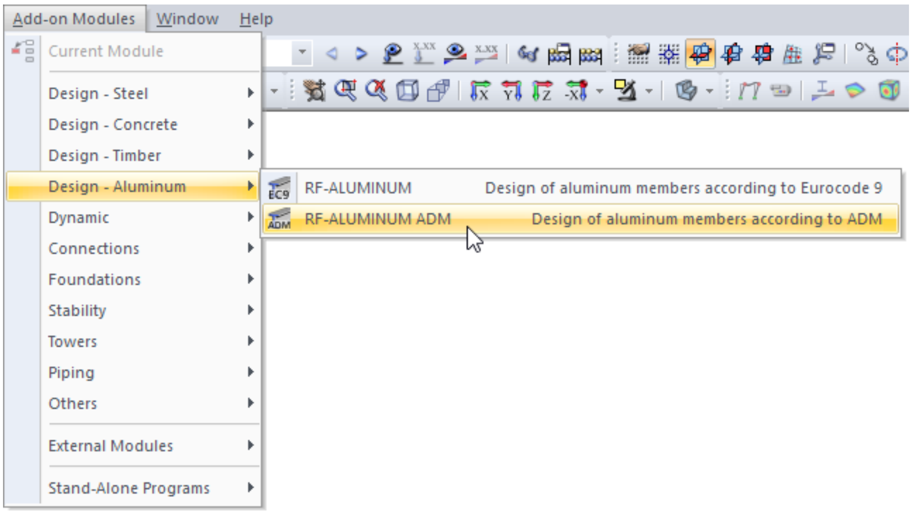


Figure 1.1: Menu *Add-on Modules → Design - Aluminum → RF-ALUMINUM ADM*

Navigator

As an alternative, you can start the add-on module in the *Data* navigator by clicking

Add-on Modules → **RF-/ALUMINUM ADM**.

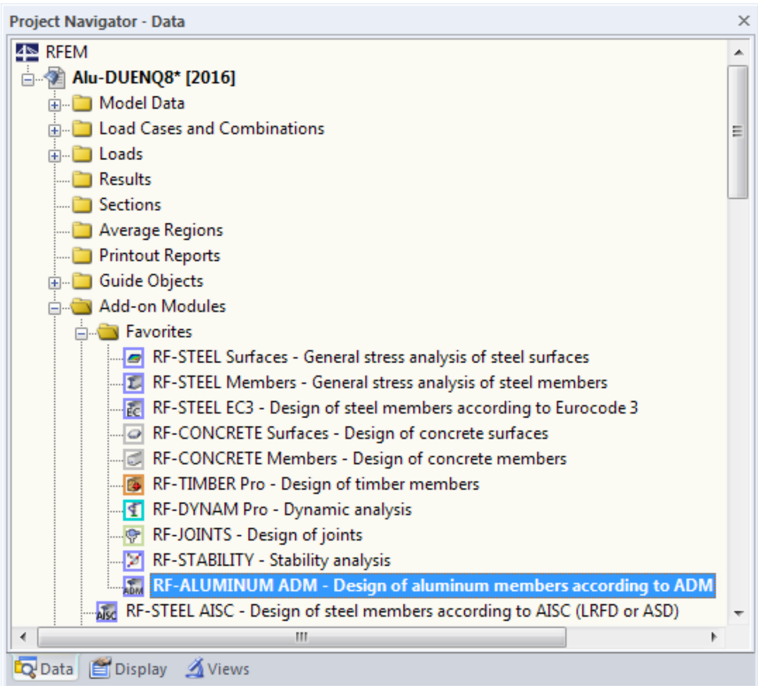


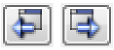
Figure 1.2: *Data* navigator *Add-on Modules → RF-ALUMINUM ADM*

2 Input Data

When you start the add-on module RF-/ALUMINUM ADM, a new window appears. In this window, a navigator is displayed on the left, managing all available tables. The drop-down list above the navigator contains the design cases (see Chapter 7.1, page 54).

The design-relevant data is defined in several input windows. When you open the add-on module 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 exit RF-/ALUMINUM ADM and return to the main program RFEM or RSTAB. To quit the add-on module without saving the new data, click [Cancel].

2.1 General Data

In the *1.1 General Data* Window, you can select the members, sets of members and actions you want to design. The two tabs manage the load cases, load and result combinations for the ULS and SLS analyses.

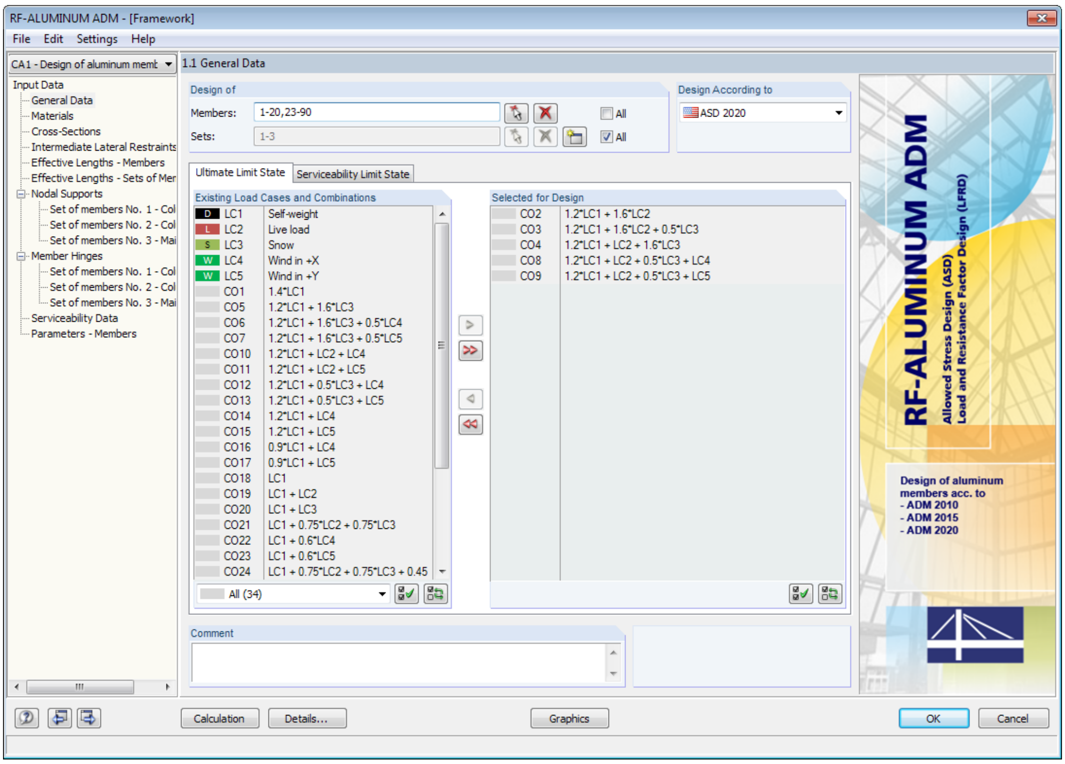


Figure 2.1: Window *1.1 General Data*

Design of

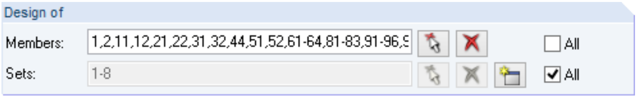


Figure 2.2: Design of members and sets of members

The design can be carried out for *Members* and *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. You can remove the list of numbers in the text boxes with the [Delete] button. Use the [Select] button to choose the objects graphically in the RFEM or RSTAB work window.

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 the result Windows *2.3 Design by Set of Members*, *3.2 Governing Internal Forces by Set of Members* and *4.2 Parts List by Set of Members*.

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

Design According to

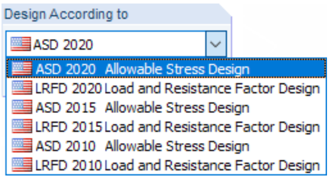


Figure 2.3: Design according to ASD or LRFD

In the drop-down list box in the upper-right corner of the window, you can select the design method for strength. The design for strength using the provisions for Allowable Strength Design *ASD 2020* is based on [1] Equation B.3-2:

$$R_a \leq \frac{R_n}{\Omega} \tag{2.1}$$

The design for strength using the provisions for Load and Resistance Factor Design *LRFD 2020* is covered by [1] Equation B.3-1:

$$R_u \leq \phi R_n \tag{2.2}$$

If a model has to be recalculated according to the revised Specifications ADM 2015 [2] or ADM 2010 [3], the respective items can be selected from the list.

Comment

Figure 2.4: User-defined comment

In this text box, you can enter user-defined notes, for example to describe the current design case.

2.1.1 Ultimate Limit State

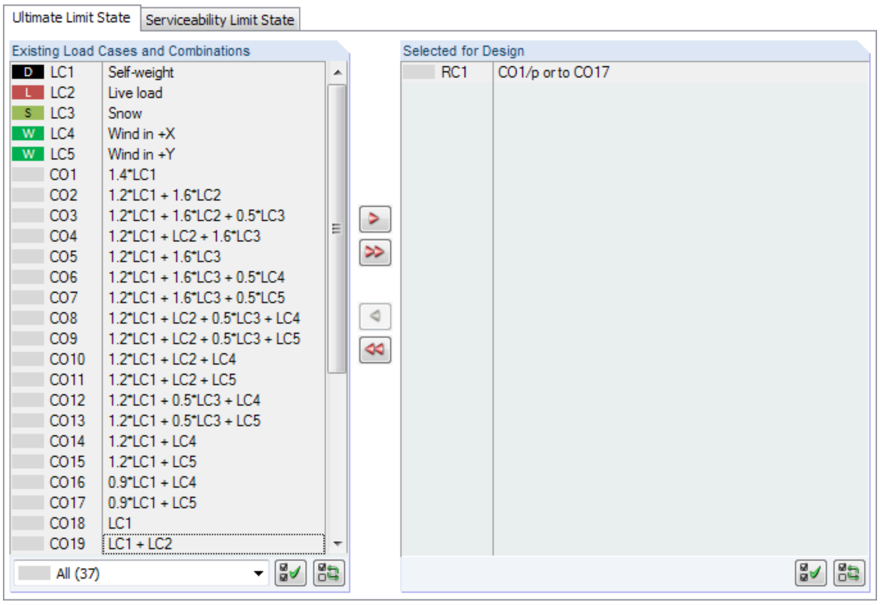


Figure 2.5: Window *1.1 General Data*, tab *Ultimate Limit State*

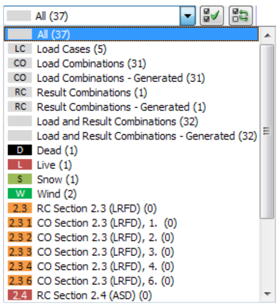
Existing Load Cases and Combinations

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

Use the [>] button to transfer the selected entries to the *Selected for Design* table on the right. Alternatively, you can double-click the entries. To transfer the entire list to the right, use the [>>] button.

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

Load cases marked in red cannot be designed: This happens when the load cases are defined without any load data or contain only imperfections.



At the end of the list, several filter options are available. They will help you to assign the entries sorted by load case, load combination, or action category. The buttons have the following functions:

	Select all cases in the list
	Invert selection of load cases

Table 2.1: Buttons in tab *Ultimate Limit State*

Selected for Design

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

Result combination

The design of an enveloping max/min result combination is performed faster than the design of all contained load cases and load combinations. However, the analysis of a result combination has also disadvantages: First, the influence of the contained actions is difficult to discern. Second, for the determination of the elastic critical moment M_e for lateral-torsional buckling, the envelope of the moment distributions is analyzed, from which the most unfavorable distribution (max or min) is taken. However, this distribution only rarely reflects the moment distribution in the individual load combinations. Therefore, unfavorable values for M_e may be applied for result combinations.

In the dialog box *Details*, tab *General* you can specify how result combinations of the 'or' type are to be handled for the design (see Chapter 3.1.4, page 31).

Result combinations should be selected for design only in case of dynamic combinations. For "ordinary" combinations, load combinations are recommended, since there are the actual moment distributions taken for the determination of M_e.

2.1.2 Serviceability Limit State

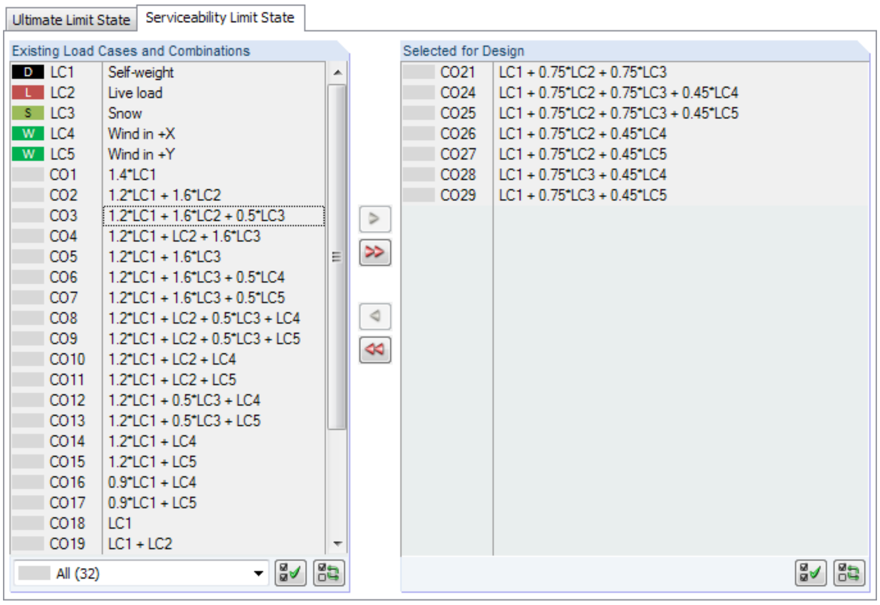


Figure 2.6: Window 1.1 *General Data*, tab *Serviceability Limit State*

Existing Load Cases and Combinations

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

Selected for Design

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

Chapter L of the Aluminum Design Manual [2] provides recommendations concerning the deflections for serviceability. To adjust the preset values, click the [Details] button. The *Details* dialog box appears (see Figure 3.3, page 30) where you can change the limits in the *Serviceability* tab.

In the *1.9 Serviceability Data* Window, you can specify the reference lengths that are governing for the deformation analysis (see Chapter 2.9, page 25).

2.2 Materials

This window is subdivided into two parts. The upper part lists all materials created in RFEM or RSTAB. The *Material Properties* section shows the properties of the current material, that is, the table row currently selected in the upper section.

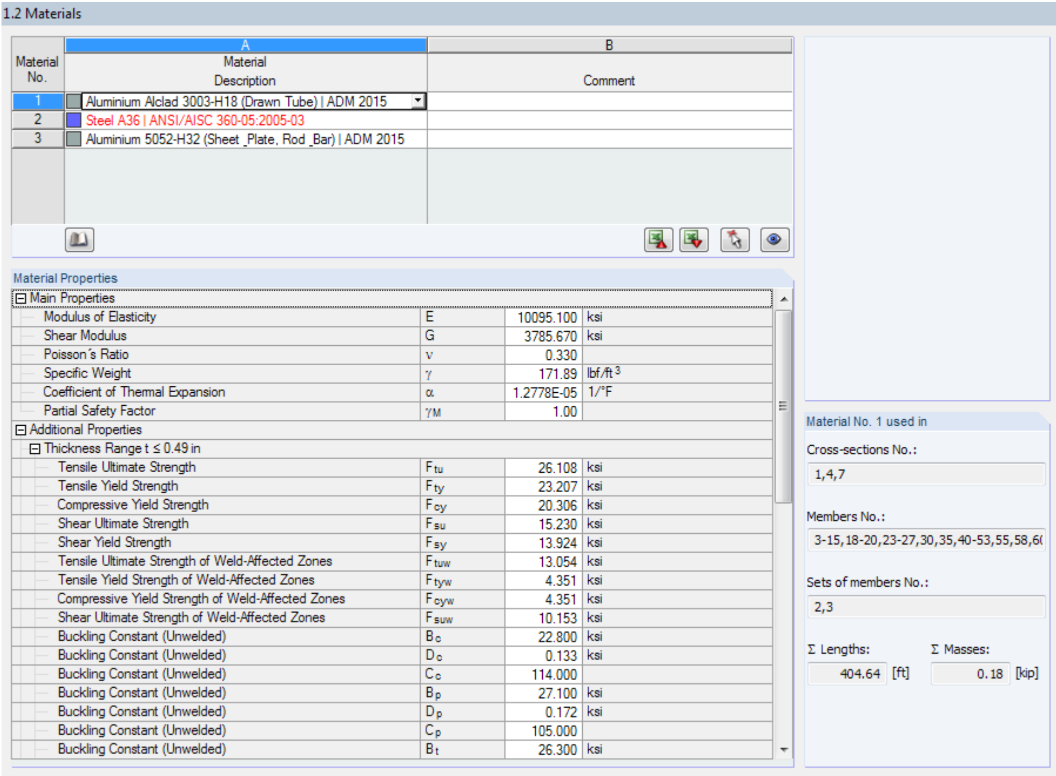


Figure 2.7: Window *1.2 Materials*

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

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

You can adjust the units and decimal places of the material properties and stresses using the menu **Settings** → **Units and Decimal Places** (see Chapter 7.3, page 58).

Material Description

The materials defined in RFEM or RSTAB are preset, but you can always modify them: To do this, click the material in column A. Then click [▾] or press function key [F7] to open the material list.

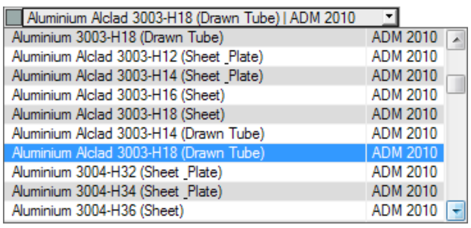


Figure 2.8: List of Materials

According to the design concept of the Specification [2], only materials of the *Aluminium* category are available in the list.

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

If you change the material description manually and the entry is stored in the material library, RF-/ALUMINUM ADM will import the material properties, too.

Generally, the material properties are not editable in the RF-/ALUMINUM ADM add-on module.

Material Library

Many materials are available in a library. To open the material library, click

Edit → Material Library

or use the button shown on the left.

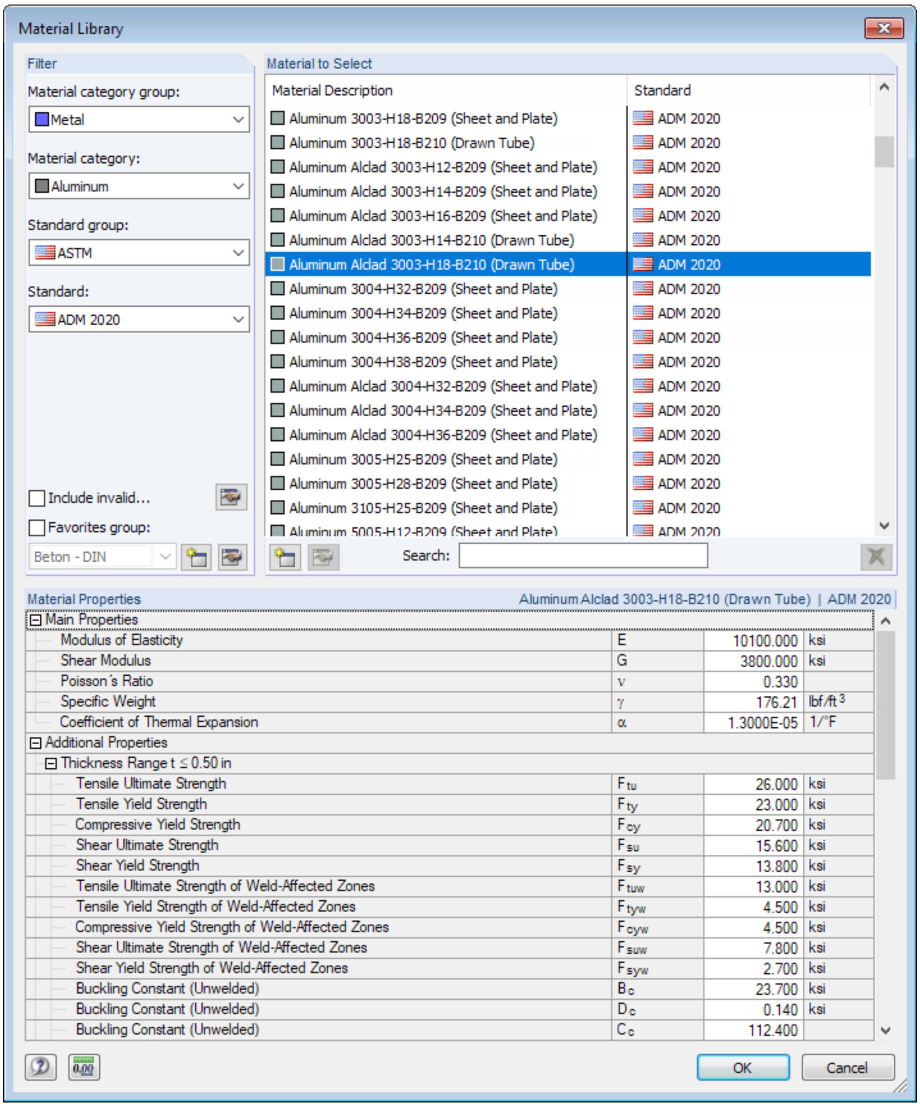


Figure 2.9: Dialog box *Material Library*

In the *Filter* section, the current *Aluminium* material category is preset. Select the specific material in the *Material to Select* list. You can check the corresponding properties in the dialog section below.

OK

Click [OK] or use the [↵] button to transfer the selected material to Window 1.2.

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

2.3 Cross-Sections

This window lists the cross-sections used for design. In addition, you can specify optimization parameters here.

Coordinate System

The sectional coordinate system **yz** of RF-/ALUMINUM ADM corresponds to the one of RFEM or RSTAB (see image in Figure 2.10). The **y**-axis represents the major principal axis of the cross-section, the **z**-axis the minor axis. This coordinate system is used for both the input data and the results.

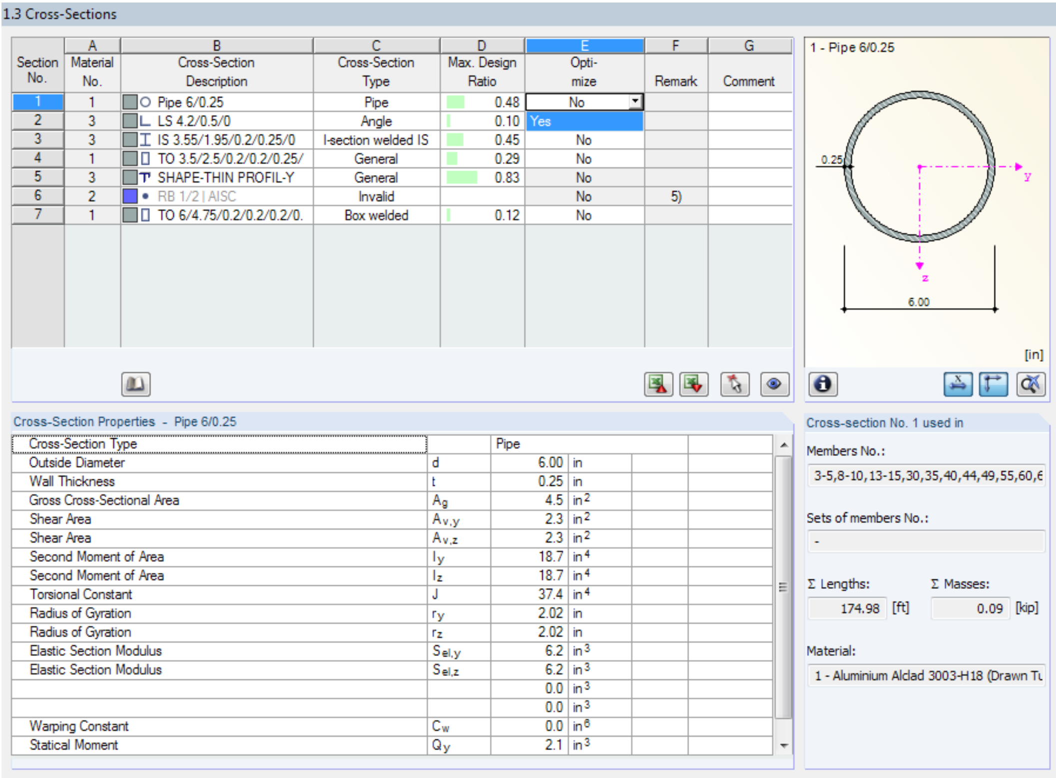


Figure 2.10: Window 1.3 *Cross-Sections*

Cross-Section Description

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

If you want to modify a cross-section, click the entry in column B. Then click [Cross-Section Library] or [...] in the box, or press the function key [F7] to open the cross-section table of the current cross-section (see Figure 2.11).

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

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

RF-/ALUMINUM ADM designs many types of cross-sections from the library or SHAPE-THIN sections. If the type of a section is *General* (see Figure 2.12), not all design options are available, however.

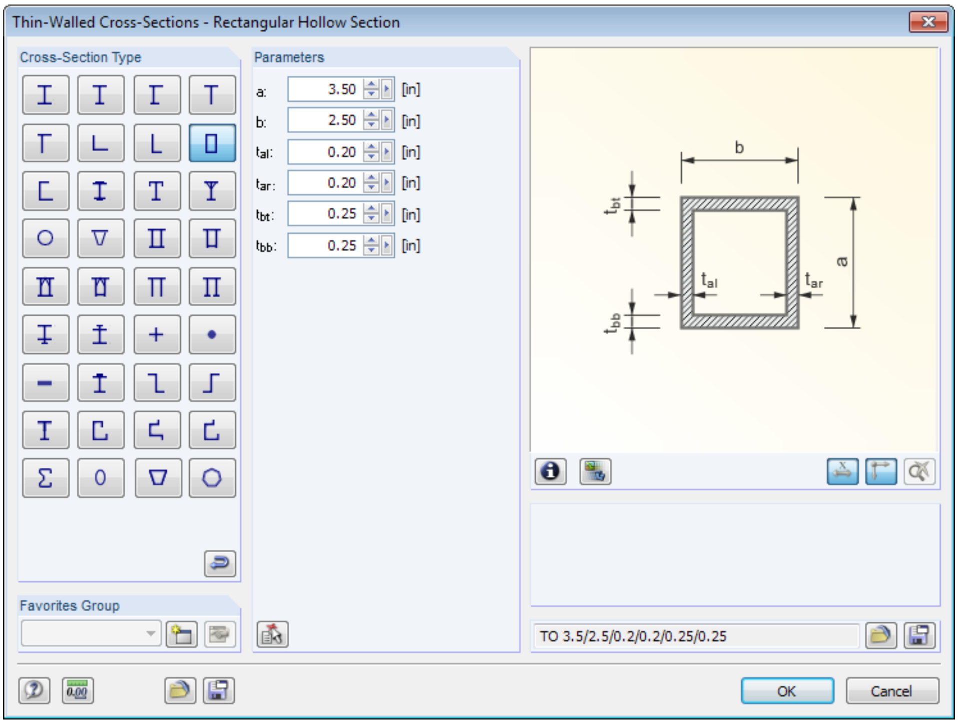


Figure 2.11: Parametric rectangular hollow section

RF-/ALUMINUM ADM performs all relevant designs for the following cross-section types:

- I-sections: rolled/welded, doubly symmetrical or mono-symmetric about the z-axis
- Hollow and box sections: rolled/welded, square-edged/rectangular, doubly-symmetrical
- Massive cross-sections: circular/rectangular
- Pipes
- Angles: rolled/welded simple sections with equal or unequal legs
- T-sections: rolled/welded, symmetrical about z-axis
- Channel-sections: rolled/welded, symmetrical about y-axis

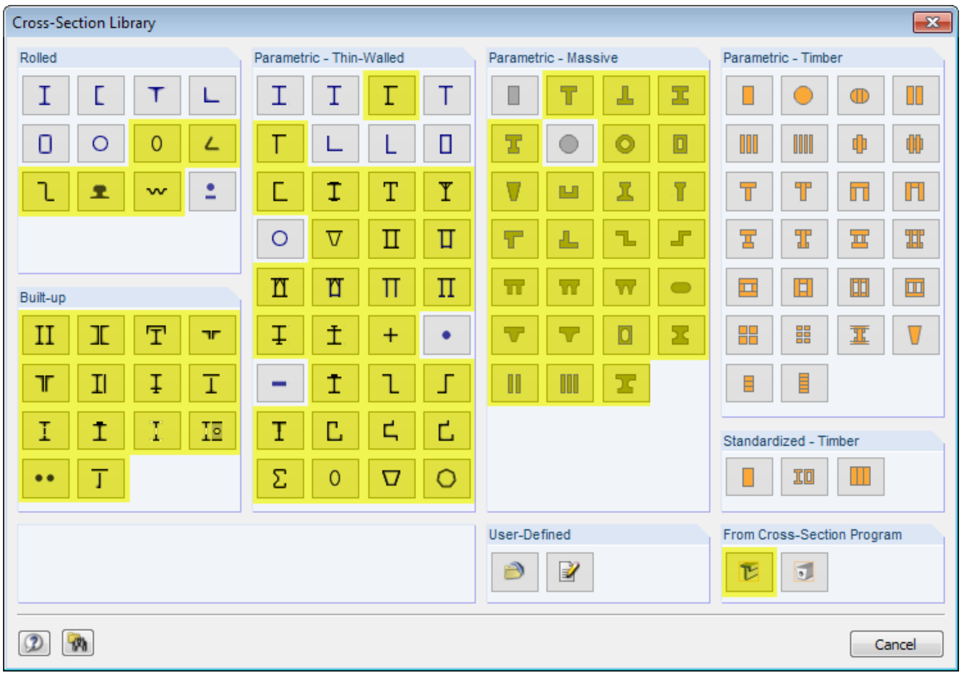
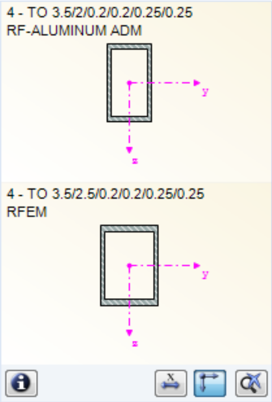


Figure 2.12: Cross-sections of type *General* (highlighted)



You can directly enter the new cross-section description in column B. If the entry is listed in the database, RF-/ALUMINUM ADM imports the cross-section parameters.

A modified cross-section will be highlighted in blue.

If the cross-sections in RF-/ALUMINUM ADM are different from the ones used in RFEM or RSTAB, both cross-sections are displayed in the graphic on the right. The designs will be performed for the cross-section selected in RF-/ALUMINUM ADM, using the internal forces from RFEM or RSTAB.

Cross-Section Type

The program displays the cross-section type that will be used for the design. The nominal strengths and stability design depend on the type of cross-section. Cross-sections that are classified as *General* (see Figure 2.12) can only be designed for specific criteria.

Max. Design Ratio

This column is displayed after the calculation. It is useful to decide on the optimization of sections. The design ratios and colored relation scales show you which cross-sections have low utilizations and, thus, are oversized, or which ones are overloaded and undersized.

Optimize

You can optimize cross-sections from the library: For the RFEM/RSTAB internal forces, the program searches the section that comes as close as possible to a user-defined maximum ratio. You can define that maximum ratio in the *Details* dialog box, tab *General* (see Figure 3.4, page 31).

To optimize a cross-section, open the corresponding drop-down list in column D or E and select *From current row*. Recommendations on the optimization can be found in Chapter 7.2 on page 56.

Remark

This column shows remarks in the form of footers that 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 end, the module displays both section numbers in two rows, in accordance with the definition in RFEM or RSTAB.

RF-/ALUMINUM ADM also designs tapered members, provided that the cross-section at the member start has the same number of stress points as the cross-section at the member end. The normal stresses, for example, are determined from the moments of inertia and the centroidal distances of the stress points. If the cross-section at the start and the end of a tapered member have a different number of stress points, the intermediate values cannot be interpolated.

The stress points including their numbering can also be checked graphically: Select the cross-section in Window 1.3 and click the [Info] button. The dialog box shown in Figure 2.13 appears.

Info About Cross-Section

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

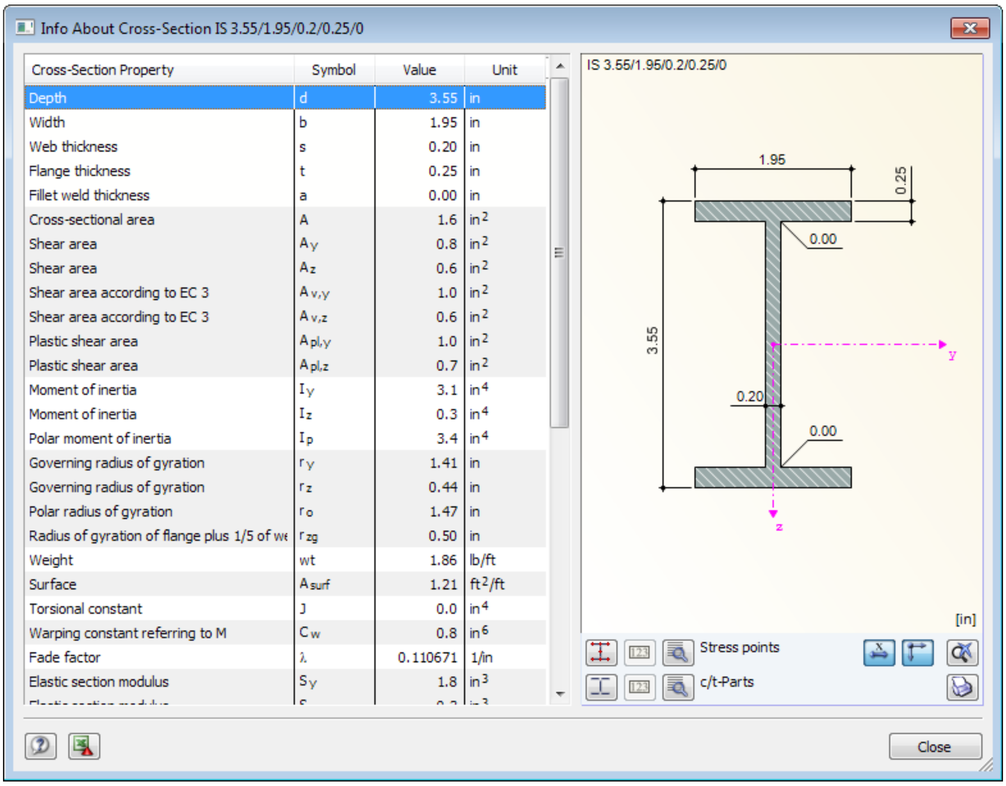


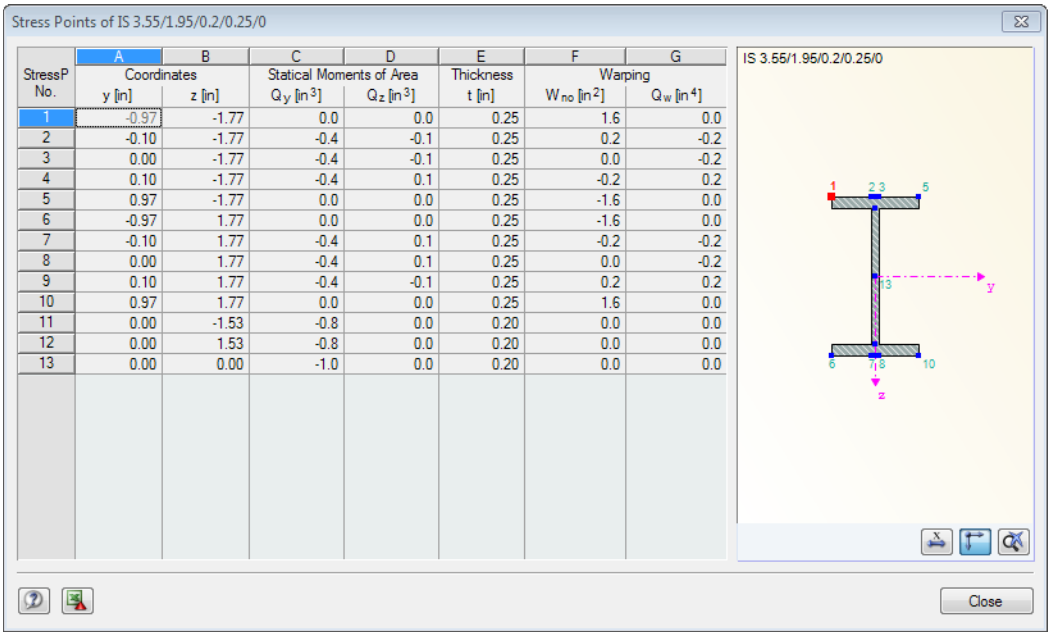
Figure 2.13: Dialog box *Info About Cross-Section*

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

Button	Function
	Display or hide the stress points
	Display or hide the cross-section c/t-parts.
	Display or hide the numbering of stress points or c/t-parts
	Display the details of stress points or c/t-parts (see Figure 2.14)
	Display or hide the cross-section dimensions
	Display or hide the principal axes of the cross-section
	Reset the full view of the cross-section graphic

Table 2.2: Buttons of cross-section graphic

Click [Details] to call up detailed information on stress points (centroidal distances, statical moments of area, warping constants etc.) and c/t-parts (see Figure 2.14).

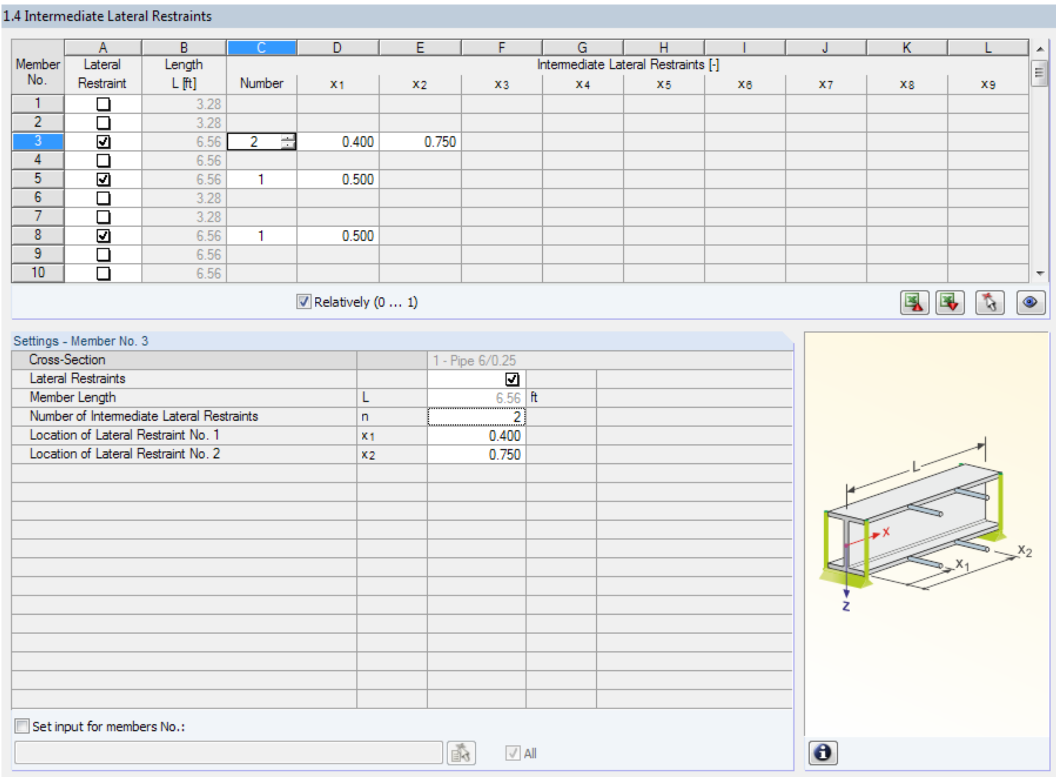


StressP No.	A Coordinates y [in]	B z [in]	C Statical Moments of Area Q_y [in^3]	D Q_z [in^3]	E Thickness t [in]	F Warping W_{no} [in^2]	G Q_w [in^4]
1	-0.97	-1.77	0.0	0.0	0.25	1.6	0.0
2	-0.10	-1.77	-0.4	-0.1	0.25	0.2	-0.2
3	0.00	-1.77	-0.4	-0.1	0.25	0.0	-0.2
4	0.10	-1.77	-0.4	0.1	0.25	-0.2	0.2
5	0.97	-1.77	0.0	0.0	0.25	-1.6	0.0
6	-0.97	1.77	0.0	0.0	0.25	-1.6	0.0
7	-0.10	1.77	-0.4	0.1	0.25	-0.2	-0.2
8	0.00	1.77	-0.4	0.1	0.25	0.0	-0.2
9	0.10	1.77	-0.4	-0.1	0.25	0.2	0.2
10	0.97	1.77	0.0	0.0	0.25	1.6	0.0
11	0.00	-1.53	-0.8	0.0	0.20	0.0	0.0
12	0.00	1.53	-0.8	0.0	0.20	0.0	0.0
13	0.00	0.00	-1.0	0.0	0.20	0.0	0.0

Figure 2.14: Dialog box *Stress Points* of symmetric I-section

2.4 Intermediate Lateral Restraints

In Window 1.4, you can define lateral restraints along each member. In RF-/ALUMINUM ADM, this kind of support is always perpendicular to the minor z-axis of the cross-section (see Figure 2.13). Thus, you can manipulate the effective lengths which are important for the stability analyses.



Member No.	A Lateral Restraint	B Length L [ft]	C Number	D x_1	E x_2	F x_3	G x_4	H x_5	I x_6	J x_7	K x_8	L x_9
1	☐	3.28										
2	☐	3.28										
3	☑	6.56	2	0.400	0.750							
4	☐	6.56										
5	☑	6.56	1	0.500								
6	☐	3.28										
7	☐	3.28										
8	☑	6.56	1	0.500								
9	☐	6.56										
10	☐	6.56										

Settings - Member No. 3			
Cross-Section		1 - Pipe 6/0.25	
Lateral Restraints		☑	
Member Length	L	6.56	ft
Number of Intermediate Lateral Restraints	n	2	
Location of Lateral Restraint No. 1	x_1	0.400	
Location of Lateral Restraint No. 2	x_2	0.750	

Figure 2.15: Window *1.4 Intermediate Lateral Restraints*

In the upper table of the window, you can assign up to nine lateral restraints to every member. The *Settings* section shows the entry in a column overview for the member selected above.

To define the lateral restraints of a member, select the *Lateral Restraints* check box in column A. To graphically select the member, click [button]. If you select the check box, the other columns become accessible and you can enter the parameters.

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

☑ Relatively (0 ... 1)

If you select the *Relatively (0 ... 1)* check box, you can define the support points by relative input. The positions of the intermediate restraints are determined by the member length and the relative distances of the member start. If you clear the *Relatively (0 ... 1)* check box, it is possible to define the distances manually in the upper table.

2.5 Effective Lengths - Members

The window is subdivided into two parts. The upper table includes summarized information about the length factors of buckling, lateral-torsional and flexural-torsional buckling as well as the equivalent lengths of the members. The lengths of the members are preset. In the *Settings* section, you can see additional information about the member selected in the upper table.

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

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

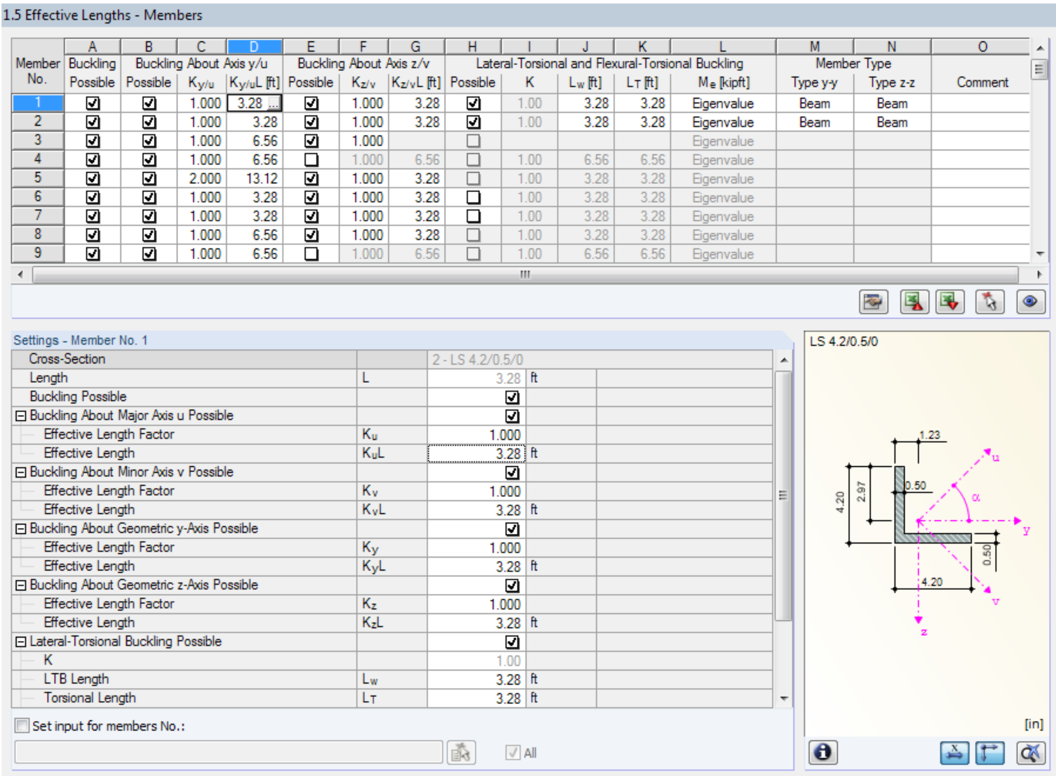


Figure 2.16: Window *1.5 Effective lengths - Members*

The effective lengths for buckling about the minor z-axis are aligned automatically with the entries of the *1.4 Intermediate Lateral Restraints* Window. If a member is divided into segments of different lengths, no values are displayed in the table columns G, J, and K of Window 1.5.

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 2.16).

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)
- *Lateral-Torsional Buckling Possible* (cf. columns H to L)
- *Member Type* (cf. columns M and N)

In the table, you can determine a buckling analysis or lateral-torsional and flexural-torsional analysis to be performed for the selected member. In addition, you can adjust the *Effective Length Factor* for the respective lengths. If you modify any 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 shown on the left (it is located below the upper table).

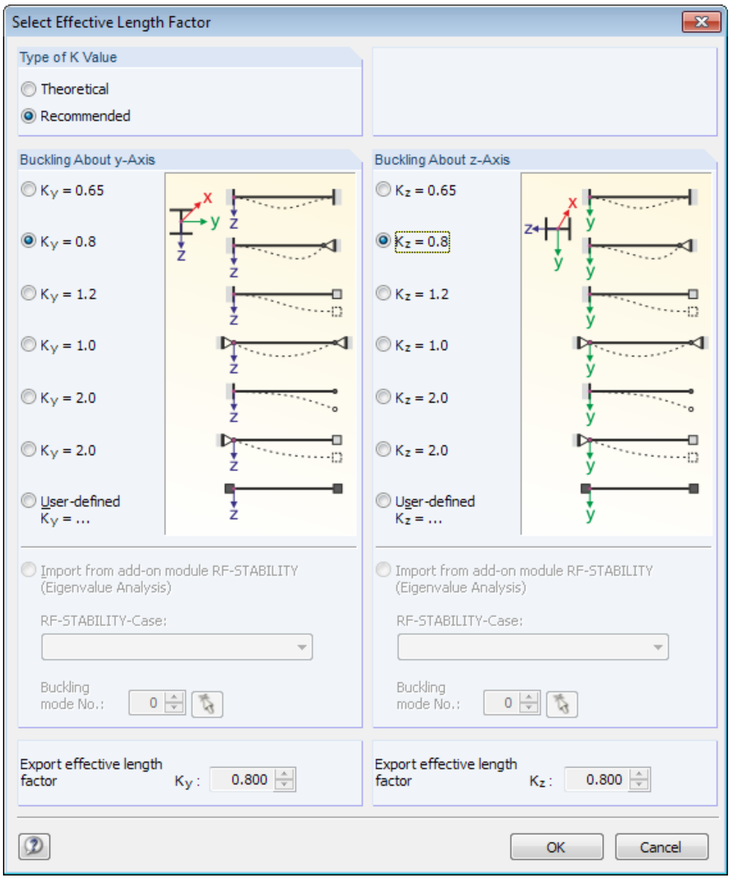


Figure 2.17: Dialog box *Select Effective Length Factor*

In this dialog box, the *Theoretical* or *Recommended* values of the K factor can be defined that are to be assigned to the selected member. The theoretical and the recommended values are described in the *Commentary on the Specification for Structural Steel Buildings* [4] Table C-C2.2. Generally, it is possible to select predefined *K* factors or to enter *User-defined* values.

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

Buckling Possible

The stability analysis for buckling, lateral-torsional and flexural-torsional buckling requires the ability of members to absorb compressive forces. Members for which this is not possible due to their types (for example tension members, elastic foundations, rigid couplings) are excluded from the analysis a priori. 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 enables you to control the stability design according to [2] Chapter C. They determine whether the analyses are to be performed for the members.

Buckling About Axis y / z

The *Possible* columns control if there is a buckling risk about the y-axis and/or z-axes. The y-axis represents the "major" principal member axis, the z-axis the "minor" axis. You can freely select the effective length factors K_y and K_z for buckling about the major and minor axes.

You can check the member axes in the cross-section graphic in the *1.3 Cross-Sections* Window (see Figure 2.10, page 11). To access the RFEM or RSTAB work window, click the [View Mode] button. There you can display the local member axes via the shortcut menu of a member or the *Display* navigator.

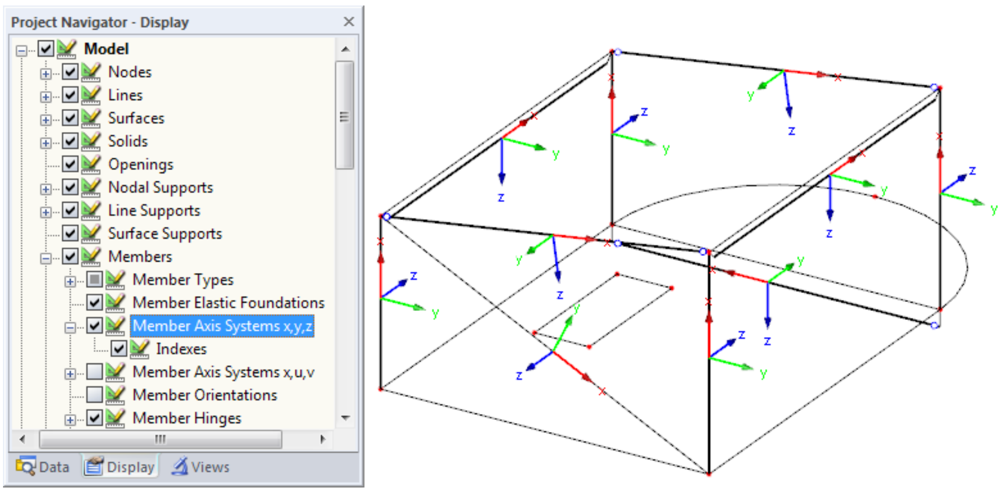


Figure 2.18: Selecting the member axis system in the *Display* navigator of RFEM

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

To specify the effective lengths in the work window graphically, click [...]. You can click the button only if the cursor is placed in a K_yL or K_zL text box (see Figure 2.16).

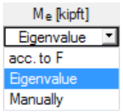
When you specify the effective length factor K, the program determines the effective length KL by multiplying the member length L by the effective length factor. The text boxes K and KL are interactive.

Lateral-Torsional and Flexural-Torsional Buckling

Column H controls which members are included in the analysis of lateral-torsional and flexural-torsional buckling.

The effective lengths L_w and L_T can be defined in columns J and K. L_w represents the LTB unbraced length L_b as referred to in [2] Chapter F (Flexure), whereas L_T corresponds to the torsional and flexural-torsional unbraced length L_z of [2] Chapter E (Compression).

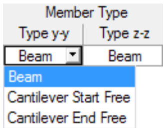
If lateral restraints have been defined in Window *1.4 Intermediate Lateral Restraints*, there is no possibility to apply any user-defined values.



To determine M_e by the *Eigenvalue* method, a member with four degrees of freedom at either side is created internally for the design. For *Beam* member types (see columns M and N), the lateral and torsional restraints are assumed to be fixed while the warping restraint is free. For *Cantilever* member types, however, lateral, torsional, and warping restraints are all assumed to be fixed while the free end is of course assumed to be free.

Alternatively, you can apply the values *according to* [2] *Chapter F* or define them *Manually* in the *Settings* table.

Member Type



For the design, the member lengths are considered variably for the brace points of a *Beam* or *Cantilever* with free ends. The relevant boundary settings can be selected in the list box for each member axis.

Comment

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

Set Input for Members No.

Below the *Settings* table, you find the *Set input for members No.* check box. When you select the 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 using the button). This option is useful when you want to assign the same boundary conditions to several members (see also https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000726).

2.6 Effective Lengths - Sets of Members

This window is available when you have selected one or more sets of members for design in the *1.1 General Data* Window (see Figure 2.1, page 5).

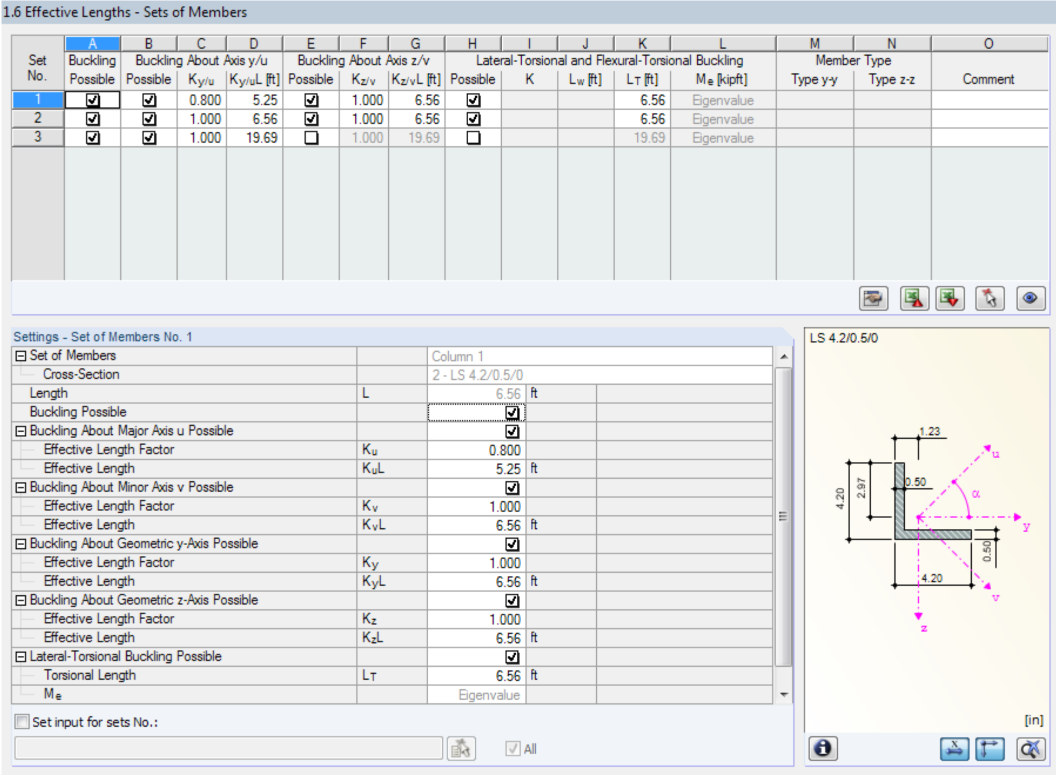


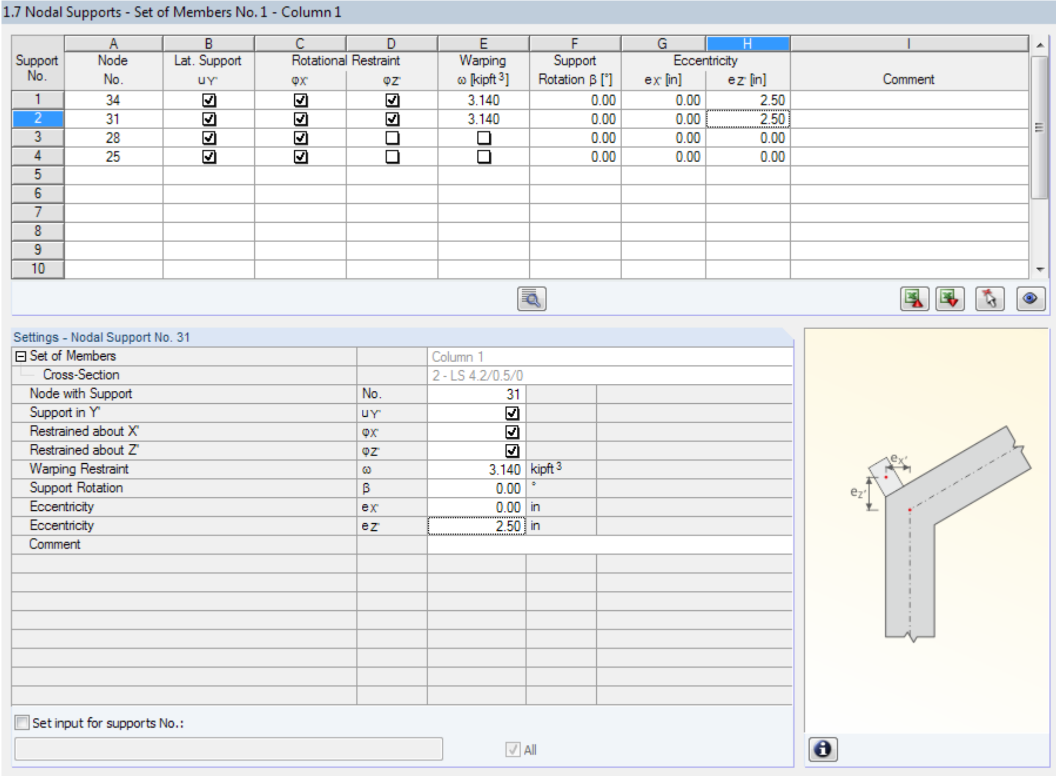
Figure 2.19: Window *1.6 Effective Lengths - Sets of Members*

The concept is similar to the previous *1.5 Effective Lengths - Members* Window. Here you can enter the effective lengths for buckling about the two principal axes of the set of members, as described in Chapter 2.5.

2.7 Nodal Supports - Set of Members

This window appears when at least one set of members has been selected in the *1.1 General Data* Window for design and when the option *Do not use member-like input* is set in the *Details* dialog box (see Figure 3.2, page 28).

The stability design of sets of members is carried out according to Chapter C of the Aluminum Design Manual [2]. According to Chapter E, you can design doubly symmetric, singly symmetric and unsymmetric cross-sections stressed by compression and/or bending in a plane. To obtain the critical moment M_e, a planar framework is created with four degrees of freedom whose parameters are to be defined in Window 1.7.



Support No.	A Node No.	B Lat. Support uY'	C Rotational Restraint φX'	D φZ'	E Warping ω [kipft³]	F Support Rotation β [°]	G Eccentricity eX' [in]	H eZ' [in]	I Comment
1	34	☑	☑	☑	3.140	0.00	0.00	2.50	
2	31	☑	☑	☑	3.140	0.00	0.00	2.50	
3	28	☑	☑	☐	☐	0.00	0.00	0.00	
4	25	☑	☑	☐	☐	0.00	0.00	0.00	
5									
6									
7									
8									
9									
10									

Figure 2.20: Window *1.7 Nodal Supports - Set of Members*

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

The supports defined in RFEM or RSTAB, such as the supports in Z of a continuous beam, are not relevant in this window. The program imports the distributions of moments and shear forces automatically from RFEM/RSTAB. You can define several support conditions here that have an impact on stability failure (buckling, lateral-torsional buckling).

Supports on the start and end nodes of a set of members are preset. Any other supports, for example as a result of additional members, has to be added manually. You can select nodes graphically in RFEM/RSTAB work window using the button.

The orientation of axes in a set of members is important for the definition of nodal supports. The program checks the position of nodes and internally defines the axes of nodal supports for Window 1.7, according to Figure 2.21 to Figure 2.24. The [Local Coordinate System] button below the model graphic can help you with the orientation. With it, you can display the set of members in a partial view where the axes are shown (see also Knowledge Base article https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000961).

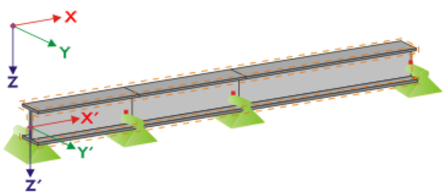


Figure 2.21: Auxiliary coordinate system of nodal supports - straight set of members

If all members of a set of members are on a straight line (see Figure 2.21), the local coordinate system of the first member in this set of members is applied for the entire set.

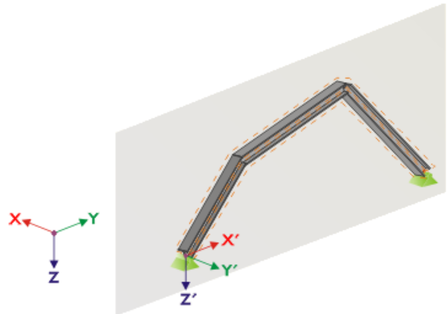


Figure 2.22: Auxiliary coordinate system of nodal supports - set of members in vertical plane

If the members of a set of members are not on a straight line, they still must be located in one plane. In Figure 2.22, the members are in a vertical plane. In this case, the X′-axis is horizontal and it is oriented in the direction of the plane. The Y'-axis is horizontal as well and defined as perpendicular to the X′-axis. The Z′-axis is oriented perpendicularly downwards.

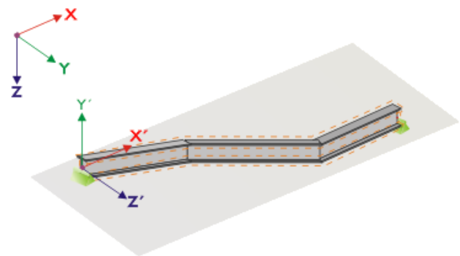


Figure 2.23: Auxiliary coordinate of nodal supports - set of members in horizontal plane

If the non-straight members are in a horizontal plane, the X′-axis is defined parallel to the X-axis of the global coordinate system. Thus, the Y′-axis is oriented in the opposite direction to the global Z-axis, and the Z′-axis is directed parallel to the global Y-axis.

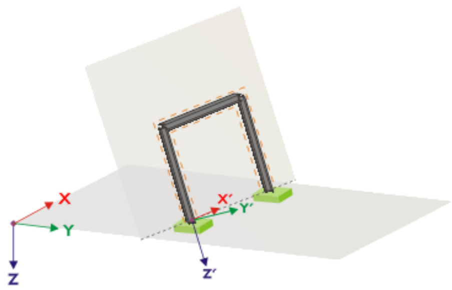


Figure 2.24: Auxiliary coordinate system of nodal supports - set of members in inclined plane

Figure 2.24 shows the general case of a non-straight set of members where the members are in an inclined plane. The definition of the X′-axis results from the intersection line between the inclined and horizontal planes. Thus, the Y′-axis is perpendicular to the X′-axis and directed perpendicularly to the inclined plane. The Z′-axis is defined perpendicular to the X′-axis and Y′-axis.

The buttons below the graphic have the following functions:

Button	Function
	Display the model or the system sketch
	Display the members as a 3D rendering or a wireframe model
	Display the current set of members or the entire model
	Display the irrelevant members of the model transparent or opaque
	Display the set of members in the local coordinate system or the entire model
	Show the view in the direction of the X-axis
	Show the view against the Y-axis
	Show the view in the direction of the Z-axis
	Display the isometric view

Table 2.3: Buttons of cross-section graphic

Click the button to determine the constant of a warp spring at the supported nodes.

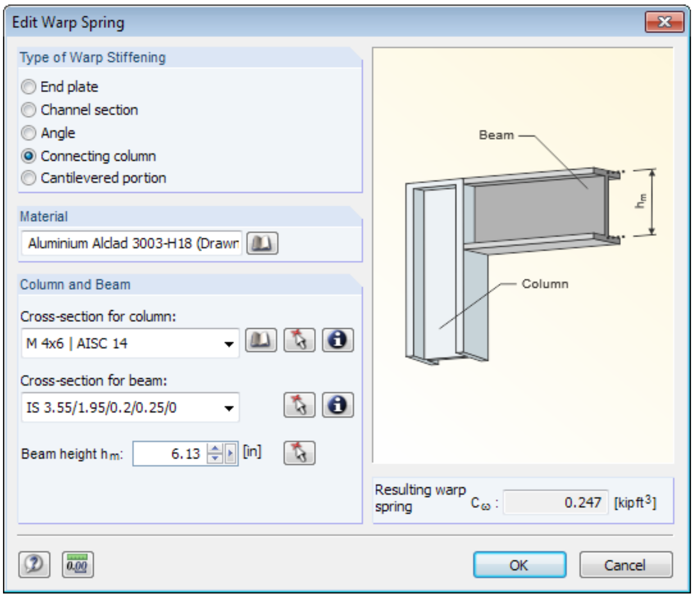


Figure 2.25: Dialog box *Edit Warp Spring*

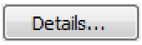
In the *Edit Warp Spring* dialog box, you can select one of the following types of warp springs:

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

You can select the materials and cross-sections using the lists and [Library] buttons. Click the button for graphical selections in the RFEM or RSTAB model.

RF-/ALUMINUM ADM determines the *Resulting warp spring* C_ω from the parameters. By clicking [OK], you import the result in Window 1.7.

2.8 Member Hinges - Set of Members



This window appears when at least one set of members has been selected in the *1.1 General Data* Window and when the option *Do not use member-like input* is set in the *Details* dialog box (see Figure 3.2, page 28). In this window, you can define releases for members within sets of members that do not transfer specific internal forces for structural reasons. Make sure that no double hinges are created in interaction with Window 1.7.

The current table manages the parameters of the set of members selected in the navigator on the left.

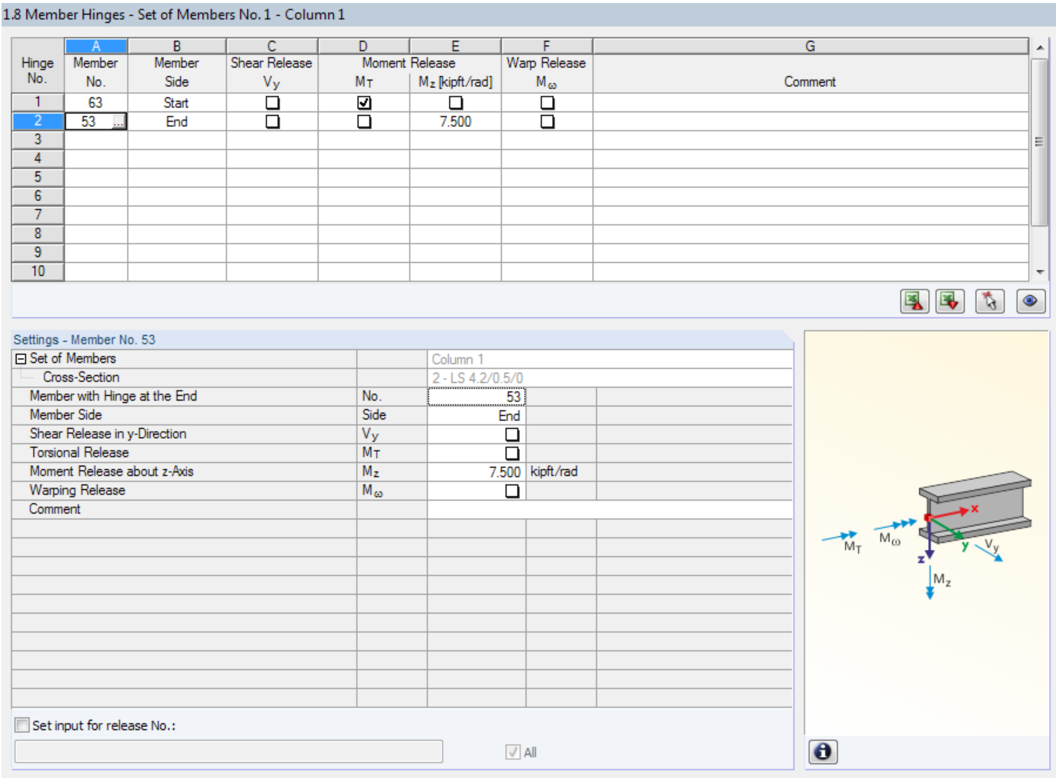
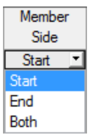


Figure 2.26: Window *1.8 Member Hinges - Set of Members*

The *Member No.* is to be entered in column A. Alternatively, use the button [...] of the text box to select the member graphically in the work window.

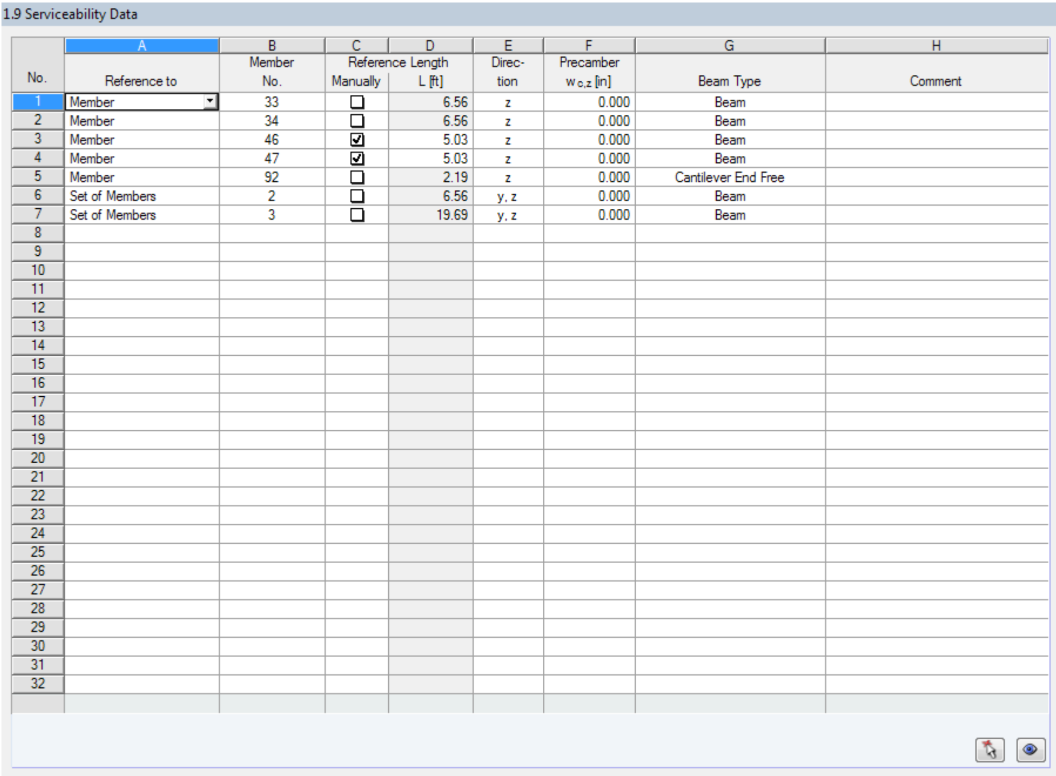


In column B, you define the *Member Side* on which the hinge is located. You can also allocate the hinge to both member ends.

The table columns C to F control the hinges – or spring constants – of each release. By these hinges, you can adjust the set of members model to the real boundary conditions.

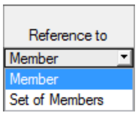
2.9 Serviceability Data

This window controls important settings of the serviceability limit state design. It is displayed when you have entered the corresponding data in the *Serviceability Limit State* tab of Window 1.1 (see Chapter 2.1.2, page 8).



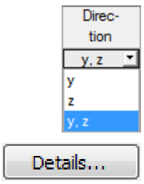
No.	A Reference to	B Member No.	C Reference Length Manually	D Reference Length L [ft]	E Direc-tion	F Precamber $w_{c,z}$ [in]	G Beam Type	H Comment
1	Member	33	☐	6.56	z	0.000	Beam	
2	Member	34	☐	6.56	z	0.000	Beam	
3	Member	46	☑	5.03	z	0.000	Beam	
4	Member	47	☑	5.03	z	0.000	Beam	
5	Member	92	☐	2.19	z	0.000	Cantilever End Free	
6	Set of Members	2	☐	6.56	y, z	0.000	Beam	
7	Set of Members	3	☐	19.69	y, z	0.000	Beam	
8								
9								
10								
11								
12								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								

Figure 2.27: Window *1.9 Serviceability Data*



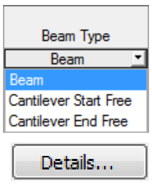
In column A, you decide whether the analysis of the deflections refers to single members or sets of members.

Column B controls the numbers of members or sets of members that are to be designed. You can select the objects graphically in the RFEM or RSTAB work window using the [...] button. The *Reference Length* is then shown in column D. This value represents the length of the member or set of members. If required, you can adjust it *Manually* when you have selected the check box in column C.



In column E, you can define 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).

Table column F enables you to take into account a possible *Precamber* w_c. By default, the precamber refers to the major axes z and v. To change this, click the [Details] button. The *Details* dialog box appears (see Figure 3.3, page 30) where you can set the minor axes y and u in the *Serviceability* tab.



For the correct application of limit deformations, the *Beam Type* is essential. In column G, you can specify whether a beam or a cantilever is to be designed and which end is without support.

The setting in the *Serviceability* tab of the *Details* dialog box controls whether the deflections are related to the undeformed model or to the shifted ends of members or sets of members (see Figure 3.3, page 30).

2.10 Parameters - Members

In the last window, you can define specific parameters for members subjected to axial tension.

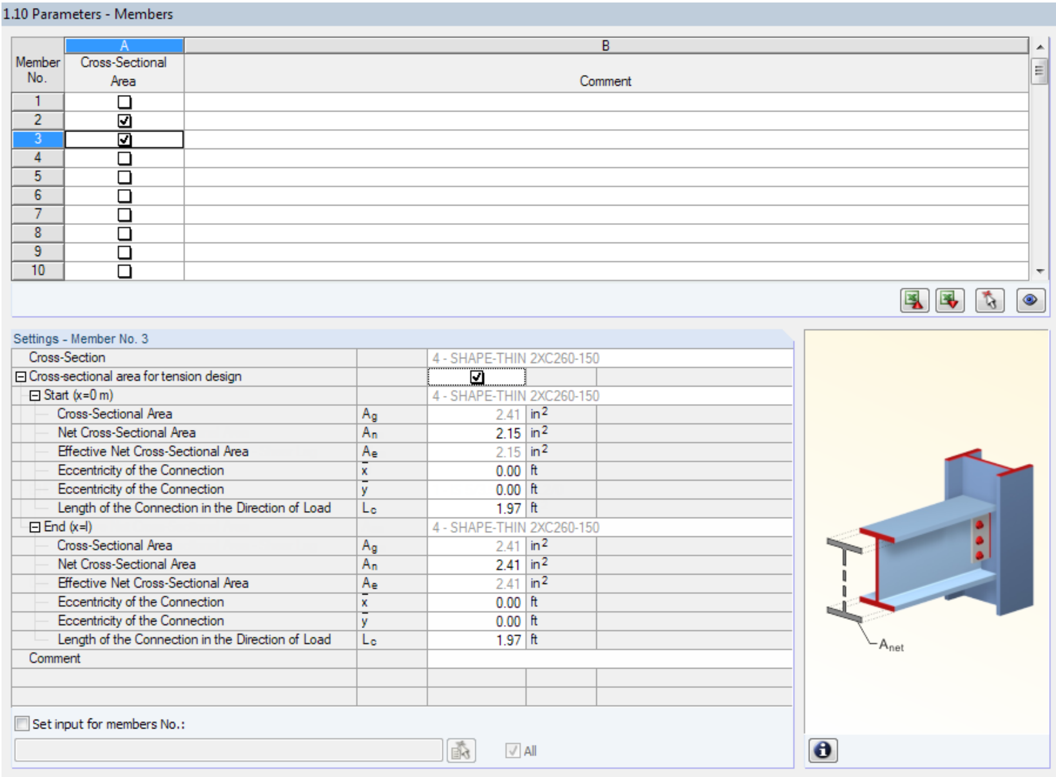


Figure 2.28: Window *1.10 Parameters - Members*

Cross-Sectional Area

When you select the check box of a member, the *Settings* table below becomes accessible where you can modify the cross-sectional area for tension design. By defining the net cross-sectional area A_n, you can control the effective net area of the section A_e to account for the effect of shear lag. Furthermore, eccentricities of the connection can be definied which refer to the local axes $\bar{x}$ and $\bar{y}$ of the member. Those have an influence on the effective net cross-sectional area, too.

The above-mentioned parameters can be specified for the *Start* and the *End* of each member.

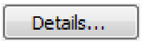
Comment

In this column, you can enter user-defined comments for each member.

Below the *Settings* table, you find the *Set input for members No.* check box. When you select the 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 using the button).

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 the [Details] button.

The *Details* dialog box has the following tabs:

- Ultimate Limit State
- Stability
- Serviceability
- General

3.1.1 Ultimate Limit State

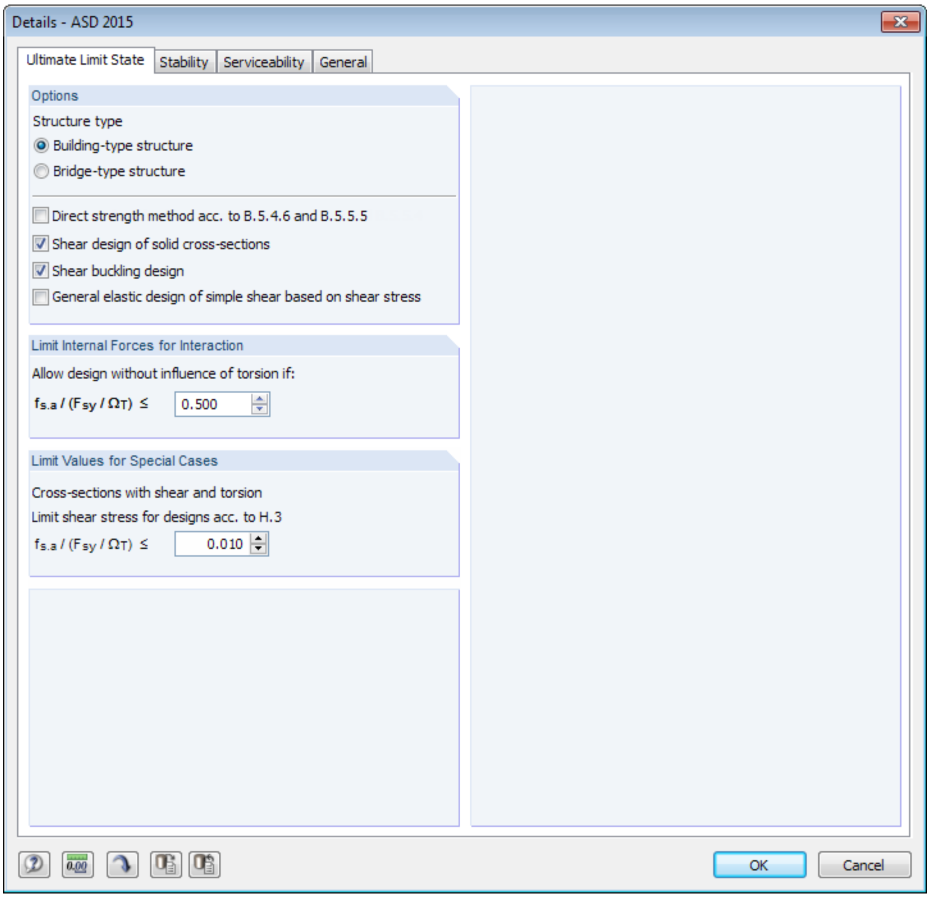


Figure 3.1: Dialog box *Details*, tab *Ultimate Limit State*

The buttons in the *Details* dialog box have the following functions:

Button	Function
	Reset default settings of the program
	Load user-defined default settings
	Save modified settings as default

Table 3.1: Buttons in *Details* dialog box

Options

The *Structure type* options control whether the model is designed for the nominal loads and load combinations as appropriate for building-type structures (cf. [2] Section B.2.1) or for bridge-type structures (cf. [2] Section B.2.2). The Load and Resistance Factor Design (LRFD) is limited to building-type structures.

Solid cross-sections

For flat elements in uniform compression or flexure, the *Direct strength method* can alternatively be applied according to [2] Sections B.5.4.6 and B.5.5.5.

By default, the *Shear design of solid cross-sections* is enabled. The figure to the left shows all types of solid cross-sections which can be defined by their parameters in RFEM or RSTAB.

The *Shear buckling design* of the members can be disabled if it is not revelant for the design case.

Optionally, a *General elastic design of simple shear based on shear stress* can be performed. In that case, the limit state stresses based on elastic parameters are applied for the elements.

Limit Internal Forces for Interaction / Special Cases

For the design of members for combined forces and torsion, it is possible to ignore the torsional components of the stresses. If the shear ratios $\frac{f_s}{F_{sy}/\Omega_T}$ (no square) do not exceed the default of 50% for interaction, torsion is not considered in the design.

In the same way, you can ignore effects due to *shear and torsion* according to [2] Section H.3. If the shear ratios do not exceed the default of 1%, they are not considered.

These values are not part of the Specification [2]. They should be adjusted by the user to the respective situations.

3.1.2 Stability

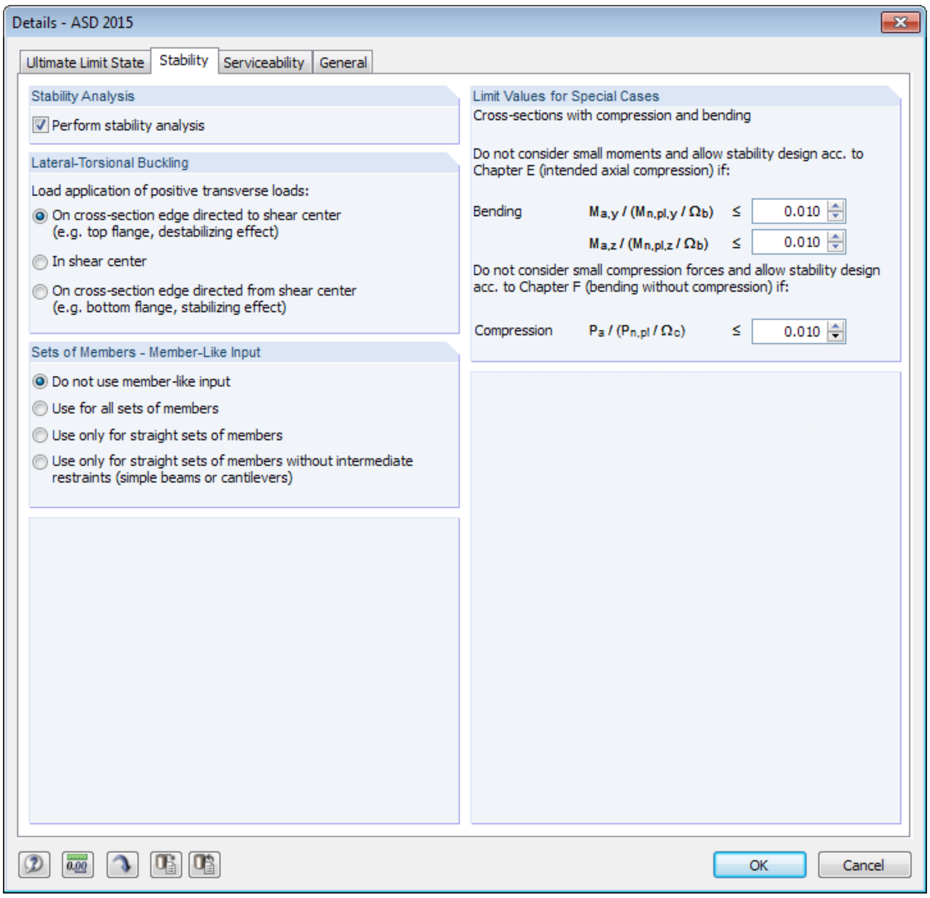


Figure 3.2: Dialog box *Details*, tab *Stability*

Stability Analysis

The *Perform stability analysis* check box controls whether to run a stability analysis in addition to the cross-section designs. If you clear the check box, the Windows 1.4 through 1.8 are not shown.

Lateral-Torsional Buckling

When there are *transverse loads*, the location must be specified at which those forces are acting: Depending on the load application point, the transverse loads can be stabilizing or destabilizing, which has an influence on the elastic critical moment.

Set of Members - Member-Like Input

By default, the *member-like input* is not applied for sets of members. That means that the boundary conditions have to be defined in Windows *1.7 Nodal Supports* and *1.8 Member Hinges* for each set of members.

Alternatively, it is possible to treat sets of members as "one long member". For that, you can define the factors K_z and K_w in Window *1.6 Effective Lengths - Sets of Members*. Those factors are used to determine the support conditions β, u_y, φ_x, φ_z, and ω. In that case, the Windows 1.7 and 1.8 are not displayed.

If at all, it is recommended to use the member-like input method for straight sets of members only.

Limit Values for Special Cases

To design cross-sections with compression according to [2] Section E, it is possible to neglect *small moments* about the major and/or minor axes. The default values in the two text boxes are 1%, which represents the ratio of design moment and strength.

For the straight check of bending according to [2] Section F, you can similarly neglect *small compression forces* by a user-defined limit ratio.

These limit settings are not part of the Specification [2]. It is the responsibility of the user to apply or modify them.

3.1.3 Serviceability

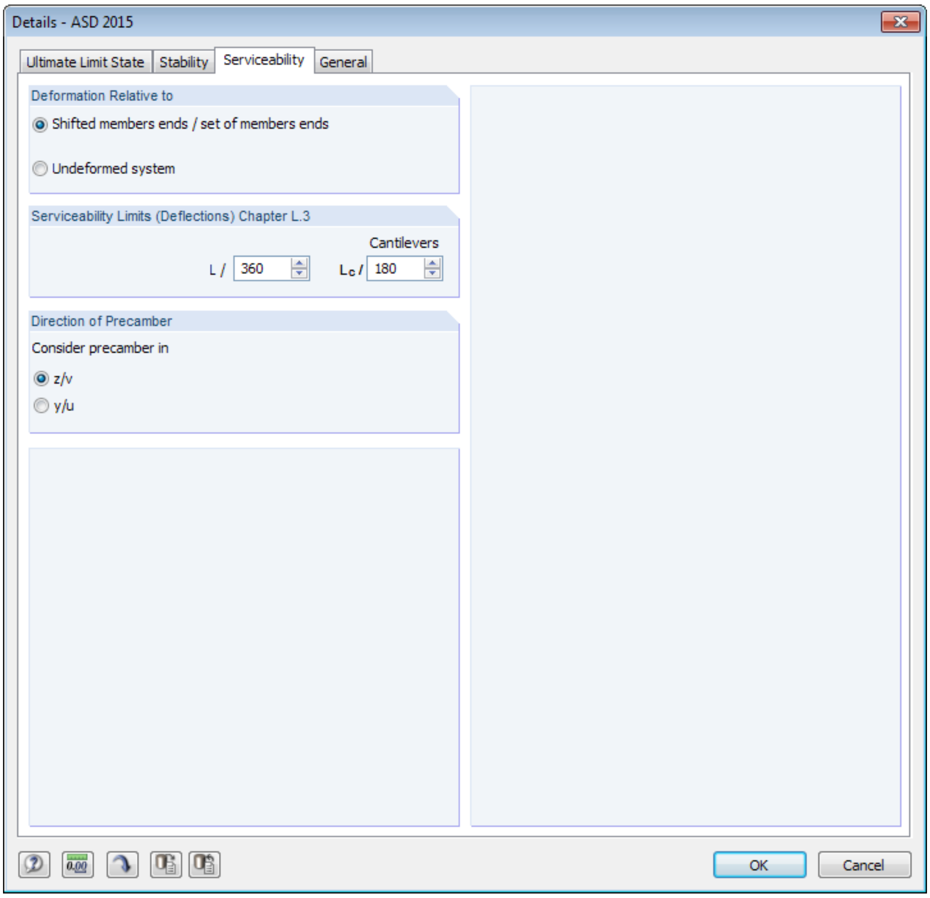


Figure 3.3: Dialog box *Details*, tab *Serviceability*

Deformation Relative To

The two options specify whether the maximum deformations are to be referred to the shifted ends of the members / sets of members (i.e. line between first and last shifted nodes of deformed model) or to the undeformed initial system.

In most cases, the deformations are designed as relative to the displacements of the entire model.

The following Knowledge Base article illustrates how to refer deflections:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001081

Serviceability Limits (Deflections) Chapter L.3

In the two text boxes, you can check and, if necessary, adjust the limit deformations for beams and cantilevers.

Chapter L of the Aluminum Design Manual [2] provides recommendations concerning the deflections for serviceability. In RF-/ALUMINUM ADM, the existing deflections are compared to the limit values which refer to the lengths of the members or sets of members. Recommendations on the limit values can be found in the *Commentary on the Specification for Structural Steel Buildings* [4] Section L3, for example.

Direction of Precamber

By default, the precamber as specified in Window *1.9 Serviceability* (see Figure 2.27, page 25) refers to the major axes z and v. If appropriate, the minor axes y and u can be set.

3.1.4 General

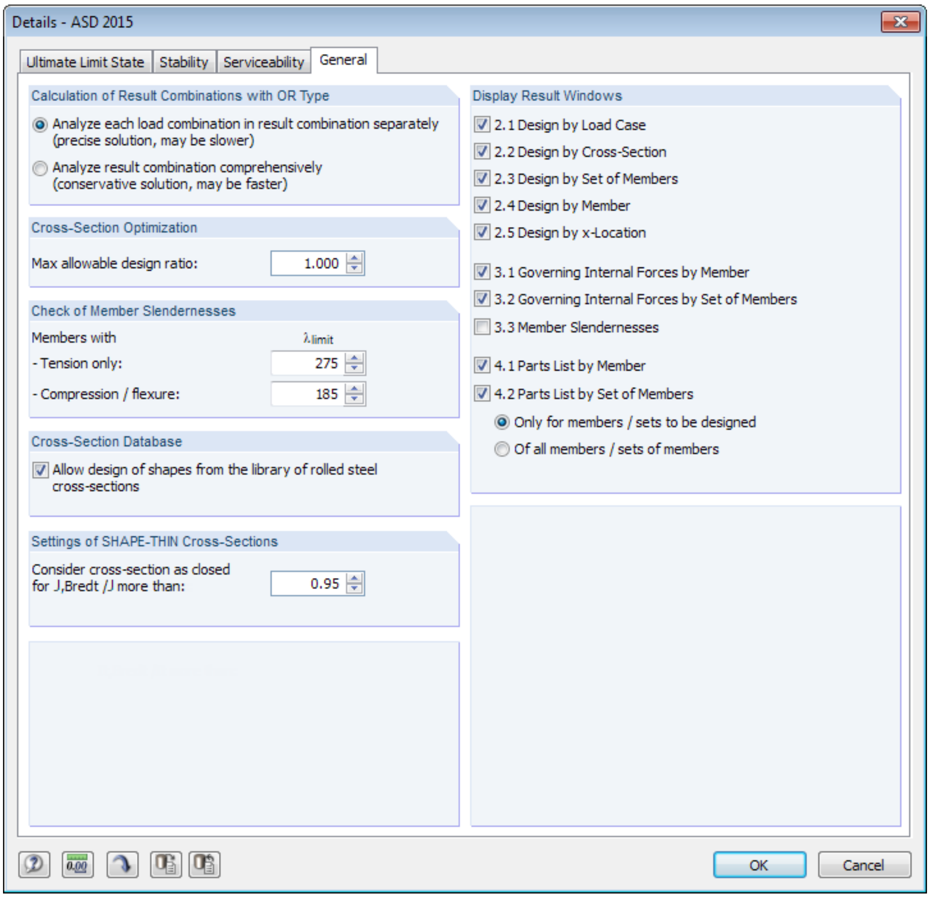


Figure 3.4: Dialog box *Details*, tab *General*

Calculation of Result Combinations with OR Type

When combinations are created automatically, often many load combinations (LCs) are produced. Usually, they are summarized in a result combination (RC) as alternatively effective ("OR combination") in order to determine the enveloping results: LC1/p or LC2/p or LC3/p or LC4/p etc. RF-/ALUMINUM ADM provides two options to design this result combination:

The components of each load combination can be analyzed *separately*. By this, the elastic critical moments are specifically determined for each load arrangement and then used for the design. This approach gives the exact results but implies a rather high demand of computing time as well.

Alternatively, the result combination can be analyzed *comprehensively*. The calculation is faster because only the extreme values with the corresponding internal forces are used for the design. The results, however, may be not be on the safe side if there is a RC with one combination in which several internal forces (e.g. N and M_y) are all slightly below the overall extremes.

Cross-Section Optimization

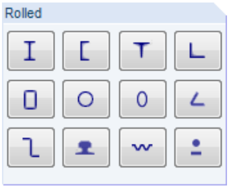
By default, the optimization is targeted on the maximum allowable design ratio of 100%. You can set a lower (or higher) limit in the text box.

Check of Member Slendernesses

In the text boxes, you can specify the limit values λ_{limit} of the member slendernesses. It is possible to enter the specifications separately for members with tension forces only and for members with compression and flexure. The slenderness ratios are given in the tables of [2] Part VI.

The limit values are compared to the actual member slendernesses. These results are listed in the *3.3 Member Slendernesses* Window (see Chapter 4.8, page 41) when the corresponding check box has been selected in the *Display Result Windows* dialog box section (see below).

Cross-Section Database



Rolled cross-sections

In general, aluminum cross-sections are fabricated by extrusion molding. When the check box is selected, the design covers *rolled steel cross-sections* – possibly with aluminum material properties – of the library, too.

The figure to the left shows the rolled types of library cross-sections.

Settings of SHAPE-THIN Cross-Sections

For closed shapes of cross-sections, the nominal flexural strengths that are to be applied are different from those for open sections (see [2] Sections F.4.2.1 and F.4.2.3 for lateral-torsional buckling). Closed shapes resist lateral-torsional buckling primarily by torsional resistance.

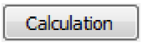
RF-/ALUMINUM ADM classifies SHAPE-THIN sections as *closed* when the share of the Bredt torsional resistance J_{Bredt} of the overall torsional resistance J is greater than the value set in the text box.

Display Result Windows

In this dialog section, you can select the result windows, including parts list, that are to be displayed. The windows are described in Chapter 4 - Results.

The *3.3 Member Slendernesses* Window is deactivated by default.

3.2 Start Calculation



In all input windows of the RF-/ALUMINUM ADM module, you can start the calculation by clicking the [Calculation] button.

RF-/ALUMINUM ADM searches for the results of the relevant load cases, load and result combinations. If they are not available yet, the program starts the calculation in RFEM or RSTAB to determine the respective internal forces.

You can also start the calculation in the RFEM/RSTAB user interface: The *To Calculate* dialog box (menu **Calculate** → **To Calculate**) includes the design cases of add-on modules, similar to load cases and load combinations.

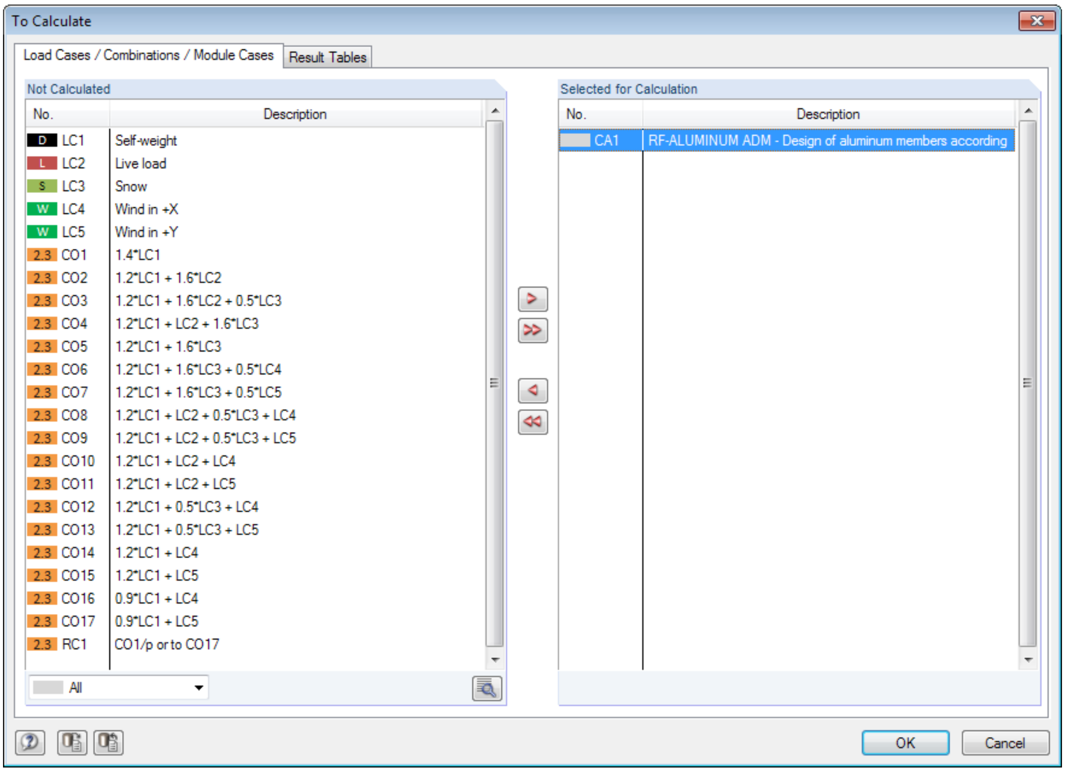
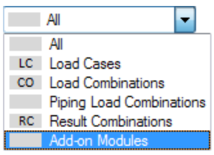


Figure 3.5: Dialog box *To Calculate*



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

Select the RF-/ALUMINUM ADM case and transfer it to the list on the right with the [▶] button. Then click [OK] to start the calculation.

To calculate a design case directly, use the list in the toolbar. Select the RF-/ALUMINUM ADM case in the toolbar list and click [Show Results].

Figure 3.6: Direct calculation of a RF-ALUMINUM ADM case in RFEM

You can then follow the design process in a separate dialog box.

4 Results

This chapter describes the result windows one by one. The evaluation of the results is described in Chapter 5.

After the calculation, the *2.1 Design by Load Case* Window is displayed.

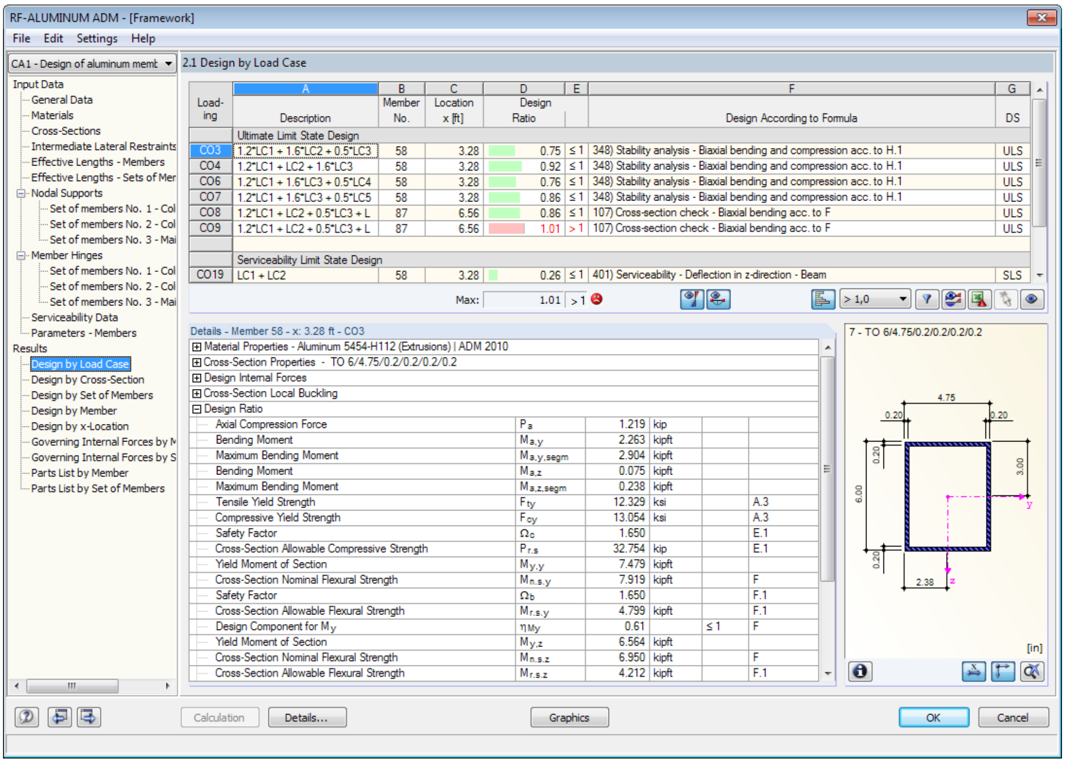
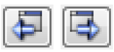


Figure 4.1: Result window with designs and details of results

The designs are presented in the result Windows 2.1 to 2.5. They are 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 for members and sets of members.



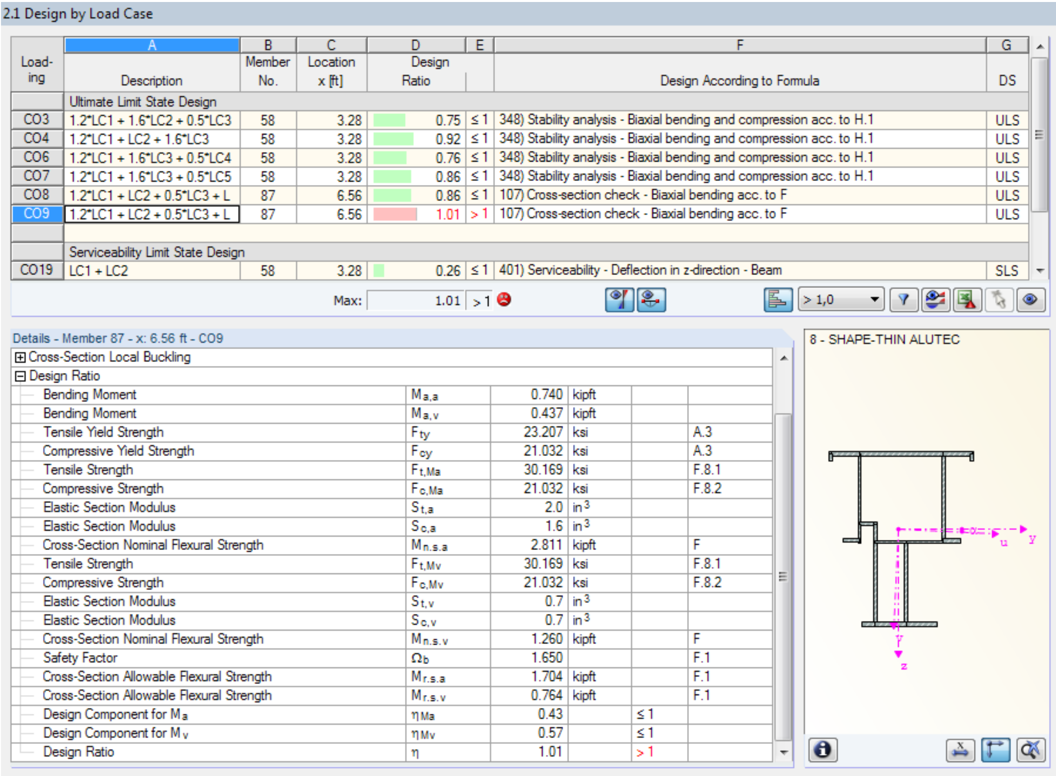
Every window can be selected by clicking the corresponding entry in the navigator. To set the previous or next window, use the buttons shown on the left. You can also use the function keys to select the next [F2] or previous [F3] window.

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

4.1 Design by Load Case

The upper part of the window shows a summary of the governing designs. They are sorted by load case, load combination, and result combination. Furthermore, the table is divided in *Ultimate Limit State Design* and *Serviceability Limit State Design* results.

The lower part of the window contains the *Details* of the cross-section properties, internal forces, and design parameters for the load case that is 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						
CO3	1.2*LC1 + 1.6*LC2 + 0.5*LC3	58	3.28	0.75	≤ 1	348) Stability analysis - Biaxial bending and compression acc. to H.1	ULS
CO4	1.2*LC1 + LC2 + 1.6*LC3	58	3.28	0.92	≤ 1	348) Stability analysis - Biaxial bending and compression acc. to H.1	ULS
CO6	1.2*LC1 + 1.6*LC3 + 0.5*LC4	58	3.28	0.76	≤ 1	348) Stability analysis - Biaxial bending and compression acc. to H.1	ULS
CO7	1.2*LC1 + 1.6*LC3 + 0.5*LC5	58	3.28	0.86	≤ 1	348) Stability analysis - Biaxial bending and compression acc. to H.1	ULS
CO8	1.2*LC1 + LC2 + 0.5*LC3 + L	87	6.56	0.86	≤ 1	107) Cross-section check - Biaxial bending acc. to F	ULS
CO9	1.2*LC1 + LC2 + 0.5*LC3 + L	87	6.56	1.01	> 1	107) Cross-section check - Biaxial bending acc. to F	ULS
	Serviceability Limit State Design						
CO19	LC1 + LC2	58	3.28	0.26	≤ 1	401) Serviceability - Deflection in z-direction - Beam	SLS
			Max:	1.01	> 1		

Details - Member 87 - x: 6.56 ft - CO9

⊞ Cross-Section Local Buckling					
⊟ Design Ratio					
Bending Moment	$M_{a,a}$	0.740	kipft		
Bending Moment	$M_{a,v}$	0.437	kipft		
Tensile Yield Strength	F_{ty}	23.207	ksi		A.3
Compressive Yield Strength	F_{cy}	21.032	ksi		A.3
Tensile Strength	$F_{t,Ma}$	30.169	ksi		F.8.1
Compressive Strength	$F_{c,Ma}$	21.032	ksi		F.8.2
Elastic Section Modulus	$S_{t,a}$	2.0	in³		
Elastic Section Modulus	$S_{c,a}$	1.6	in³		
Cross-Section Nominal Flexural Strength	$M_{n.s.a}$	2.811	kipft		F
Tensile Strength	$F_{t,Mv}$	30.169	ksi		F.8.1
Compressive Strength	$F_{c,Mv}$	21.032	ksi		F.8.2
Elastic Section Modulus	$S_{t,v}$	0.7	in³		
Elastic Section Modulus	$S_{c,v}$	0.7	in³		
Cross-Section Nominal Flexural Strength	$M_{n.s.v}$	1.260	kipft		F
Safety Factor	Ω_b	1.650			F.1
Cross-Section Allowable Flexural Strength	$M_{r.s.a}$	1.704	kipft		F.1
Cross-Section Allowable Flexural Strength	$M_{r.s.v}$	0.764	kipft		F.1
Design Component for M_a	η_{Ma}	0.43		≤ 1	
Design Component for M_v	η_{Mv}	0.57		≤ 1	
Design Ratio	η	1.01		> 1	

Figure 4.2: Window *2.1 Design by Load Case*

Description

In the first column, the descriptions of the load cases, load and result combinations used for the designs are displayed.

Member No.

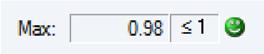
This column contains the numbers of the members that have the highest design ratios of each designed load case or combination.

Location x

This column shows the x-locations of each member where the highest design ratios occur. For the table output, the program uses the following member x-locations:

- Start and end node
- Division points according to member division as defined in RFEM Table 1.16 or RSTAB Table 1.6
- Division of members according to specification for result diagrams (*Calculation Parameters* dialog box, *Global Calculation Parameters* tab in RFEM or RSTAB)
- Extreme values of internal forces

Design Ratio



Columns D and E show the terms of design according to [2]. The lengths of the colored bars represent the respective design ratios.

Design According to Formula

This column lists the Sections of the Specification [2] by which the designs have been performed.

DS

The last column provides information on the design situations: *ULS* or *SLS* for the ultimate limit state or serviceability limit state design.

4.2 Design by Cross-Section

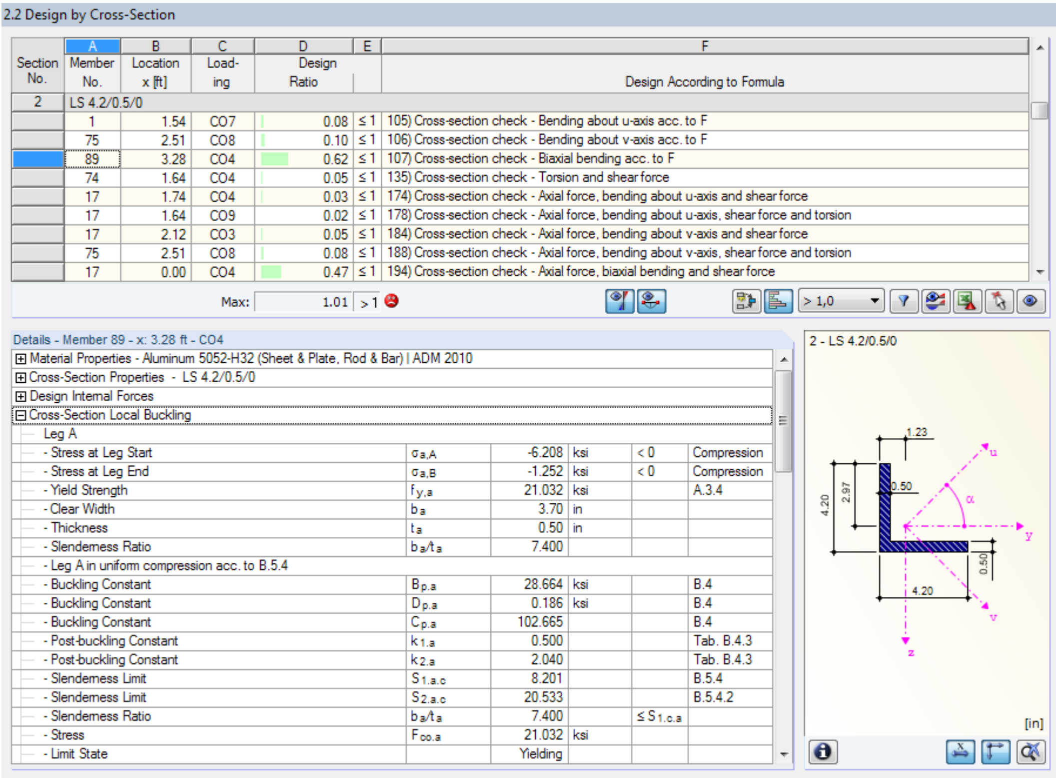


Figure 4.3: Window *2.2 Design by Cross-Section*

In the upper table, the highest ratios of all designed members and load cases or combinations are listed by cross-section. The results are sorted by cross-section design and stability analysis, as well as serviceability limit state design.

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

4.3 Design by Set of Members

This result window is displayed when you have selected at least one set of members for design. The table lists the highest design ratios of each set of members.

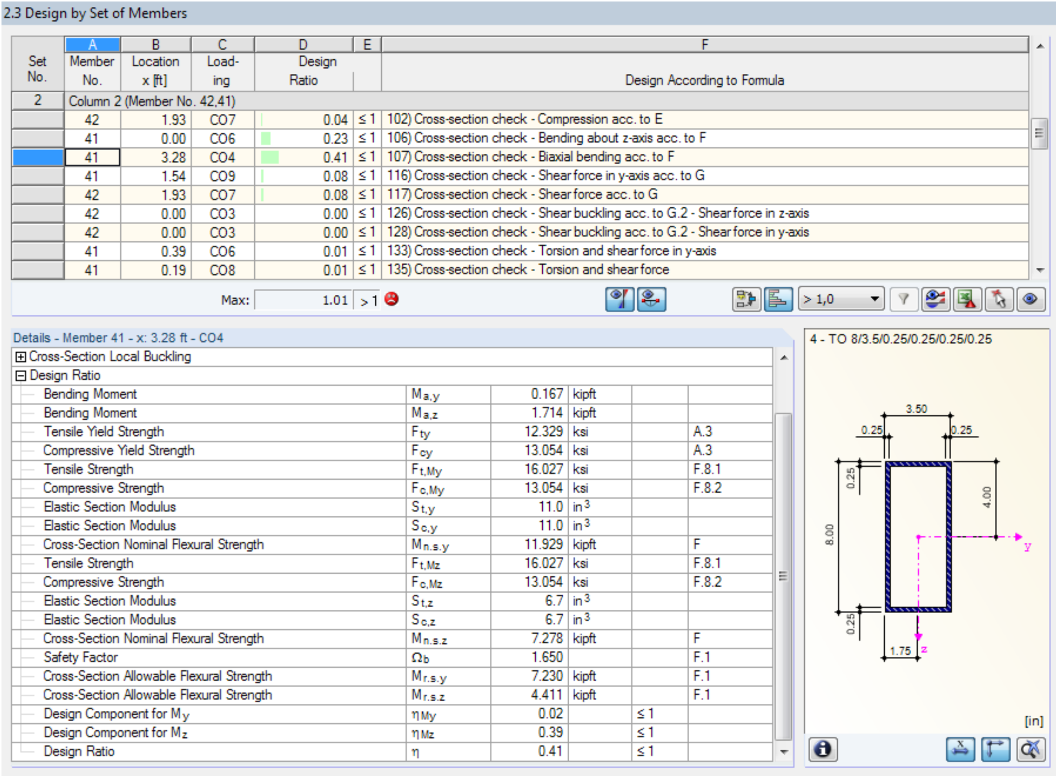


Figure 4.4: Window *2.3 Design by Set of Members*

The column *Member No.* shows the number of the member within the set that has the highest stress ratio for the respective design criteria.

The output by set of members presents the maximum design results of structural groups, such as frames or multi-part columns.

4.4 Design by Member

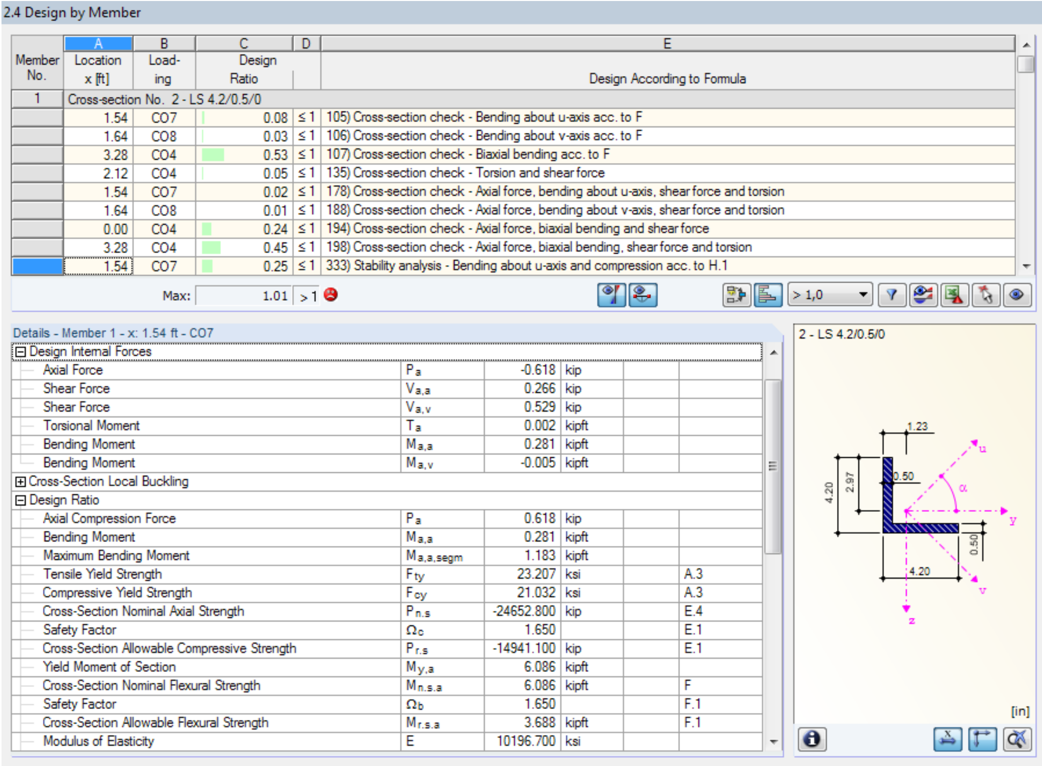


Figure 4.5: Window *2.4 Design by Member*

In this window, the highest ratios of the cross-section checks, stability analyses, and serviceability designs are shown. The results are sorted by member number.

4.5 Design by x-Location

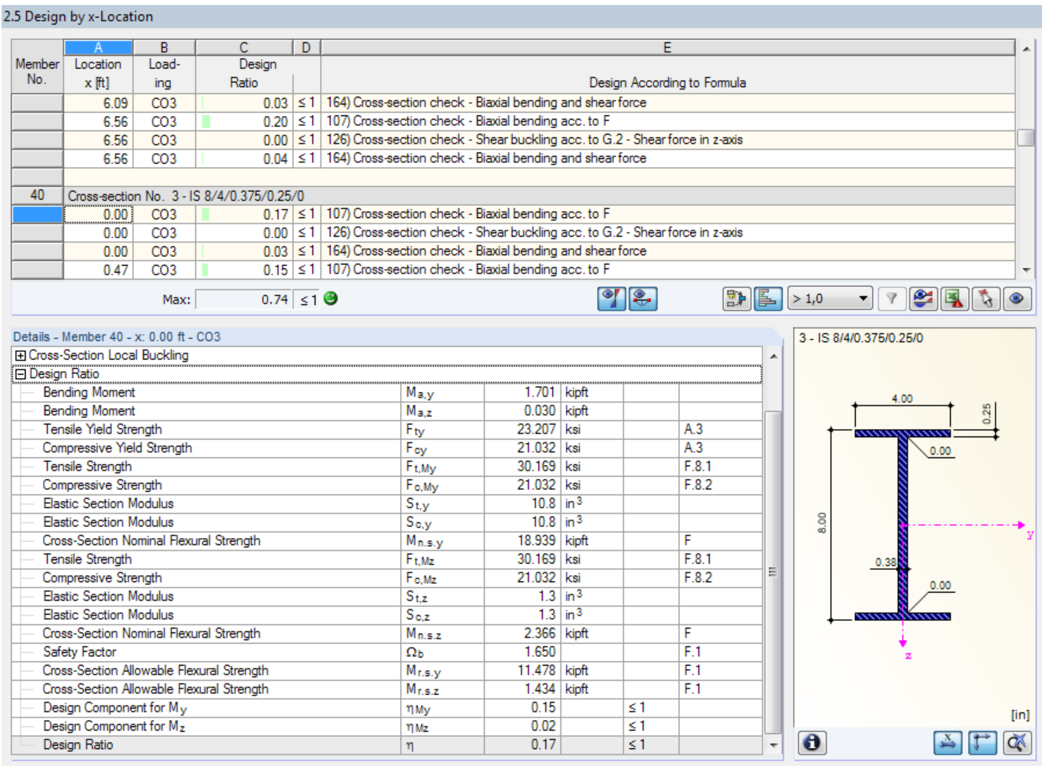


Figure 4.6: Window *2.5 Design by x-Location*

This window lists the maximum values of each member at every location **x**:

- Start and end node
- Division points according to member division as defined in RFEM Table 1.16 or RSTAB Table 1.6
- Division of members according to specification for result diagrams (*Calculation Parameters* dialog box, *Global Calculation Parameters* tab in RFEM or RSTAB)
- 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] N	V_y/V_u	V_z/V_v	Moments [kipft] M_T	M_y/M_u	M_z/M_v	Design According to Formula
1	Cross-section No. 2 - LS 4.2/0.5/0								
	1.54	CO7	-0.618	0.266	0.529	0.002	0.282	-0.005	105) Cross-section check - Bending about u-axis acc. to F
	1.64	CO8	-0.396	0.272	0.420	-0.008	0.000	-0.117	106) Cross-section check - Bending about v-axis acc. to F
	3.28	CO4	-0.689	0.340	0.659	-0.002	1.342	-0.617	107) Cross-section check - Biaxial bending acc. to F
	2.12	CO4	-0.694	0.342	0.660	-0.001	0.578	-0.223	135) Cross-section check - Torsion and shear force
	1.54	CO7	-0.618	0.266	0.529	0.002	0.282	-0.005	178) Cross-section check - Axial force, bending about u-axis, sh
	1.64	CO8	-0.396	0.272	0.420	-0.008	0.000	-0.117	188) Cross-section check - Axial force, bending about v-axis, sh
	0.00	CO4	-0.706	0.341	0.660	0.000	-0.823	0.503	194) Cross-section check - Axial force, biaxial bending and shea
	3.28	CO4	-0.689	0.340	0.659	-0.002	1.342	-0.617	198) Cross-section check - Axial force, biaxial bending, shear fo
	1.54	CO7	-0.618	0.266	0.529	0.002	0.281	-0.005	333) Stability analysis - Bending about u-axis and compression a
	1.54	CO7	-0.618	0.266	0.529	0.002	0.282	-0.005	335) Stability analysis - Bending about u-axis, compression and
	1.64	CO8	-0.396	0.272	0.420	-0.008	0.000	-0.117	339) Stability analysis - Bending about v-axis and compression a
	1.16	CO6	-0.568	0.276	0.447	-0.004	-0.011	0.062	341) Stability analysis - Bending about v-axis, compression and
	3.28	CO4	-0.689	0.340	0.659	-0.002	1.342	-0.617	342) Stability analysis - Biaxial bending and compression acc. to
	3.28	CO4	-0.689	0.340	0.659	-0.002	1.342	-0.617	344) Stability analysis - Biaxial bending, compression and shear
2	Cross-section No. 2 - LS 4.2/0.5/0								
	2.12	CO6	-0.941	0.300	0.328	0.008	0.005	-0.105	106) Cross-section check - Bending about v-axis acc. to F
	0.00	CO4	-1.620	0.370	0.572	0.001	-1.104	0.679	107) Cross-section check - Biaxial bending acc. to F
	0.39	CO9	-1.383	0.389	0.612	-0.008	-0.827	0.552	135) Cross-section check - Torsion and shear force
	1.93	CO4	-1.607	0.376	0.574	0.000	0.003	-0.042	184) Cross-section check - Axial force, bending about v-axis an
	1.74	CO8	-1.131	0.316	0.330	0.015	-0.010	0.021	188) Cross-section check - Axial force, bending about v-axis, sh
	0.19	CO4	-1.619	0.371	0.573	0.001	-0.994	0.608	194) Cross-section check - Axial force, biaxial bending and shea
	0.00	CO4	-1.620	0.370	0.572	0.001	-1.104	0.679	198) Cross-section check - Axial force, biaxial bending, shear fo
	1.93	CO4	-1.607	0.376	0.574	0.000	0.003	-0.042	339) Stability analysis - Bending about v-axis and compression a
	1.93	CO4	-1.607	0.376	0.574	0.000	0.003	-0.042	341) Stability analysis - Bending about v-axis, compression and
	3.28	CO4	-1.601	0.372	0.573	-0.001	0.779	-0.548	342) Stability analysis - Biaxial bending and compression acc. to
	0.00	CO4	-1.620	0.370	0.572	0.001	-1.104	0.679	344) Stability analysis - Biaxial bending, compression and shear
3	Cross-section No. 1 - TO 8/4/0.375/0.375/0.375/0.375								
	6.09	CO3	-0.612	-0.009	-0.124	-0.040	-0.683	0.014	105) Cross-section check - Bending about y-axis acc. to F
	2.34	CO8	-0.505	0.315	-0.211	-0.022	-0.005	0.218	106) Cross-section check - Bending about z-axis acc. to F
	0.00	CO8	-0.506	0.356	-0.277	-0.022	0.572	1.019	107) Cross-section check - Biaxial bending acc. to F
	0.00	CO3	-0.612	-0.009	-0.348	-0.040	0.833	-0.043	126) Cross-section check - Shear buckling acc. to G.2 - Shear

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

For every member, this window displays the governing internal forces. Those are the internal forces that result in the maximum design ratios of each design.

Location x

This column informs you about the x-locations of each member where the highest ratios occur.

Loading

This column shows numbers of the load cases and combinations whose internal forces result in the maximum design ratios for each type of design.

Forces / Moments

For every member, this column displays the axial and shear forces as well as the torsional and bending moments that give the maximum ratios each design.

Design According to Formula

The last column shows the types of design and gives the Sections of the Specification [2] that were used for the analysis.

4.7 Governing Internal Forces by Set of Members

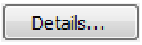
3.2 Governing Internal Forces by Set of Members

Set No.	A Location x [ft]	B Load-ing	C Forces [kip] N	D V_y/V_u	E V_z/V_v	F Moments [kipft] M_T	G M_y/M_u	H M_z/M_v	I Design According to Formula
1	Column 1 (Member No. 17,16)								
	1.64	CO9	-0.989	-0.168	-0.190	0.013	0.222	0.002	105) Cross-section check - Bending about y-axis acc. to F
	0.77	CO8	-0.383	-0.323	-0.379	0.008	0.005	-0.292	106) Cross-section check - Bending about z-axis acc. to F
	3.28	CO4	-0.629	-0.507	-0.438	-0.001	-0.907	0.913	107) Cross-section check - Biaxial bending acc. to F
	2.32	CO4	-0.632	-0.509	-0.439	-0.001	-0.484	0.423	135) Cross-section check - Torsion and shear force
	1.74	CO4	-1.536	-0.442	-0.483	0.000	0.164	-0.001	174) Cross-section check - Axial force, bending about y-axis an
	1.64	CO9	-0.989	-0.168	-0.190	0.013	0.222	0.002	178) Cross-section check - Axial force, bending about y-axis, sh
	2.12	CO3	-1.520	-0.421	-0.405	0.000	0.009	0.195	184) Cross-section check - Axial force, bending about z-axis an
	0.77	CO8	-0.383	-0.323	-0.379	0.008	0.005	-0.292	188) Cross-section check - Axial force, bending about z-axis, sh
	0.00	CO4	-1.548	-0.437	-0.481	0.001	1.001	-0.766	194) Cross-section check - Axial force, biaxial bending and she
	3.28	CO4	-0.629	-0.507	-0.438	-0.001	-0.907	0.913	198) Cross-section check - Axial force, biaxial bending, shear fo
	1.74	CO4	-1.536	-0.442	-0.483	0.000	0.164	-0.001	333) Stability analysis - Bending about y-axis and compression a
	1.74	CO4	-1.536	-0.442	-0.483	0.000	0.164	-0.001	335) Stability analysis - Bending about y-axis, compression and
	2.12	CO3	-1.520	-0.421	-0.405	0.000	0.009	0.195	339) Stability analysis - Bending about z-axis and compression a
	2.12	CO3	-1.520	-0.421	-0.405	0.000	0.009	0.195	341) Stability analysis - Bending about z-axis, compression and
	3.28	CO4	-0.629	-0.507	-0.438	-0.001	-0.907	0.913	342) Stability analysis - Biaxial bending and compression acc. to
	3.28	CO4	-0.629	-0.507	-0.438	-0.001	-0.907	0.913	344) Stability analysis - Biaxial bending, compression and shear
2	Column 2 (Member No. 42,41)								
	1.93	CO7	-1.935	-0.622	0.040	-0.003	0.002	-0.007	102) Cross-section check - Compression acc. to E
	0.00	CO6	-1.280	-0.754	-0.014	0.011	-0.003	-1.004	106) Cross-section check - Bending about z-axis acc. to F
	3.28	CO4	-1.375	-0.894	0.094	0.004	0.167	1.714	107) Cross-section check - Biaxial bending acc. to F
	1.54	CO9	-0.849	-0.606	0.030	-0.006	-0.008	0.007	116) Cross-section check - Shear force in y-axis acc. to G
	1.93	CO7	-1.935	-0.622	0.040	-0.003	0.002	-0.007	117) Cross-section check - Shear force acc. to G
	0.00	CO3	-2.568	-0.703	0.067	-0.003	-0.128	-1.307	126) Cross-section check - Shear buckling acc. to G.2 - Shear
	0.00	CO3	-2.568	-0.703	0.067	-0.003	-0.128	-1.307	128) Cross-section check - Shear buckling acc. to G.2 - Shear
	0.39	CO6	-1.277	-0.756	-0.014	0.011	-0.008	-0.712	133) Cross-section check - Torsion and shear force in y-axis
	0.19	CO8	-0.857	-0.656	-0.112	0.023	0.116	-0.764	135) Cross-section check - Torsion and shear force
	1.64	CO4	-2.829	-0.881	0.087	-0.002	-0.017	-0.194	181) Cross-section check - Axial force, bending about z-axis an
	0.00	CO4	-2.843	-0.854	0.085	-0.003	-0.158	-1.624	191) Cross-section check - Axial force, biaxial bending and she
	1.93	CO7	-1.935	-0.622	0.040	-0.003	0.002	-0.007	302) Stability analysis - Flexural buckling about y-axis acc. to E
	1.93	CO7	-1.935	-0.622	0.040	-0.003	0.002	-0.007	306) Stability analysis - Flexural buckling about z-axis acc. to E
	1.64	CO4	-2.829	-0.881	0.087	-0.002	-0.017	-0.194	339) Stability analysis - Bending about z-axis and compression a
	1.64	CO4	-2.829	-0.881	0.087	-0.002	-0.017	-0.194	340) Stability analysis - Bending about z-axis, compression and

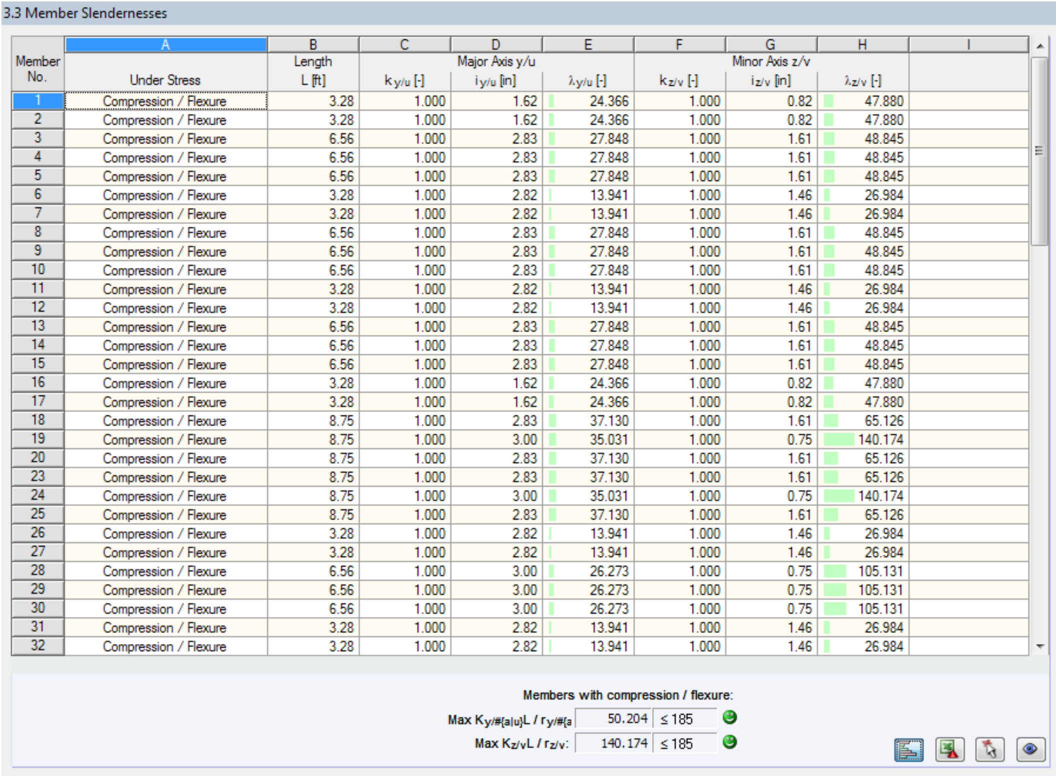
Figure 4.8: Window *3.2 Governing Internal Forces by Set of Members*

This result window is displayed when you have selected at least one set of members for design. The table lists the internal forces that result in the highest ratios for each set of members.

4.8 Members Slendernesses



This window is shown when you have selected the respective check box in the *Details* dialog box, tab *General* (see Figure 3.4, page 31).

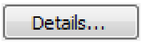

3.3 Member Slendernesses

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
1	Compression / Flexure	3.28	1.000	1.62	24.366	1.000	0.82	47.880	
2	Compression / Flexure	3.28	1.000	1.62	24.366	1.000	0.82	47.880	
3	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
4	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
5	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
6	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
7	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
8	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
9	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
10	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
11	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
12	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
13	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
14	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
15	Compression / Flexure	6.56	1.000	2.83	27.848	1.000	1.61	48.845	
16	Compression / Flexure	3.28	1.000	1.62	24.366	1.000	0.82	47.880	
17	Compression / Flexure	3.28	1.000	1.62	24.366	1.000	0.82	47.880	
18	Compression / Flexure	8.75	1.000	2.83	37.130	1.000	1.61	65.126	
19	Compression / Flexure	8.75	1.000	3.00	35.031	1.000	0.75	140.174	
20	Compression / Flexure	8.75	1.000	2.83	37.130	1.000	1.61	65.126	
23	Compression / Flexure	8.75	1.000	2.83	37.130	1.000	1.61	65.126	
24	Compression / Flexure	8.75	1.000	3.00	35.031	1.000	0.75	140.174	
25	Compression / Flexure	8.75	1.000	2.83	37.130	1.000	1.61	65.126	
26	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
27	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
28	Compression / Flexure	6.56	1.000	3.00	26.273	1.000	0.75	105.131	
29	Compression / Flexure	6.56	1.000	3.00	26.273	1.000	0.75	105.131	
30	Compression / Flexure	6.56	1.000	3.00	26.273	1.000	0.75	105.131	
31	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	
32	Compression / Flexure	3.28	1.000	2.82	13.941	1.000	1.46	26.984	

Members with compression / flexure:

Max $K_{y/\#\{a\|u\}}L$ / $r_{y/\#\{a}$	50.204	≤ 185
Max $K_{z/v}L$ / $r_{z/v}$:	140.174	≤ 185

Figure 4.9: Window *3.3 Member slendernesses*



The table lists the effective slendernesses of the designed members for both directions of the principal axes. They are determined in compliance with the load type. At the end of the list, there is a comparison with the limit values that have been defined in the *Details* dialog box, tab *General* (see Figure 3.4, page 31).

Members of the member types “Tension” or “Cable” are not included in the table.

This window is displayed only for information. No stability analysis of the slendernesses is provided.

4.9 Parts List by Member

Finally, RF-/ALUMINUM ADM gives a summary of all designed cross-sections.

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	2 - LS 4.2/0.5/0	8	3.28	26.25	36.75	0.72	4.62	15.17	0.01
2	1 - TO 8/4/0.375/0.375/0.375/0.375	18	6.56	118.11	236.22	6.92	9.88	64.80	0.12
3	4 - TO 8/3.5/0.25/0.25/0.25/0.25	24	3.28	78.74	150.92	3.01	6.44	21.12	0.05
4	1 - TO 8/4/0.375/0.375/0.375/0.375	18	8.75	157.48	314.96	9.23	9.88	86.41	0.16
5	3 - IS 10/6/0.5/1/0	2	8.75	17.50	62.70	1.94	18.73	163.85	0.03
6	3 - IS 10/6/0.5/1/0	12	6.56	78.74	282.15	8.75	18.73	122.89	0.15
7	7 - TO 6/4.75/0.2/0.2/0.2/0.2	3	6.56	19.69	35.27	0.57	4.85	31.80	0.01
8	1 - TO 8/4/0.375/0.375/0.375/0.375	4	3.28	13.12	26.25	0.77	9.88	32.40	0.01
9	8 - SHAPE-THIN ALUTEC	1	6.56	6.56	12.52	0.08	2.00	13.10	0.00
10	5 - SHAPE-THIN PROFIL-Y	1	3.28	3.28	6.64	0.02	0.92	3.01	0.00
Sum		91		519.47	1164.37	32.00			0.55

Figure 4.10: Window *4.1 Parts List by Member*

Details...

By default, the table contains only the designed members. If you need a parts list of all members, select the corresponding option in the *Details* dialog box, tab *General* (see Figure 3.4, page 31).

Part No.

The program automatically assigns item numbers to identical members.

Cross-Section Description

In this column, the numbers and descriptions of the cross-sections are listed.

Number of Members

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

Length

In column C, the lengths of the representative members are given.

Total Length

The values in this column represent the product of the previous two columns.

Surface Area

The area, which is related to the total length, is determined from the *Surface* of the cross-sections. It can be checked in Windows 1.3 and 2.1 through 2.5 in the *Info* dialog box (see Figure 2.13, page 14).

Volume

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

Unit Weight

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

Weight

The values of this column represent the product of columns C and G.

Total Weight

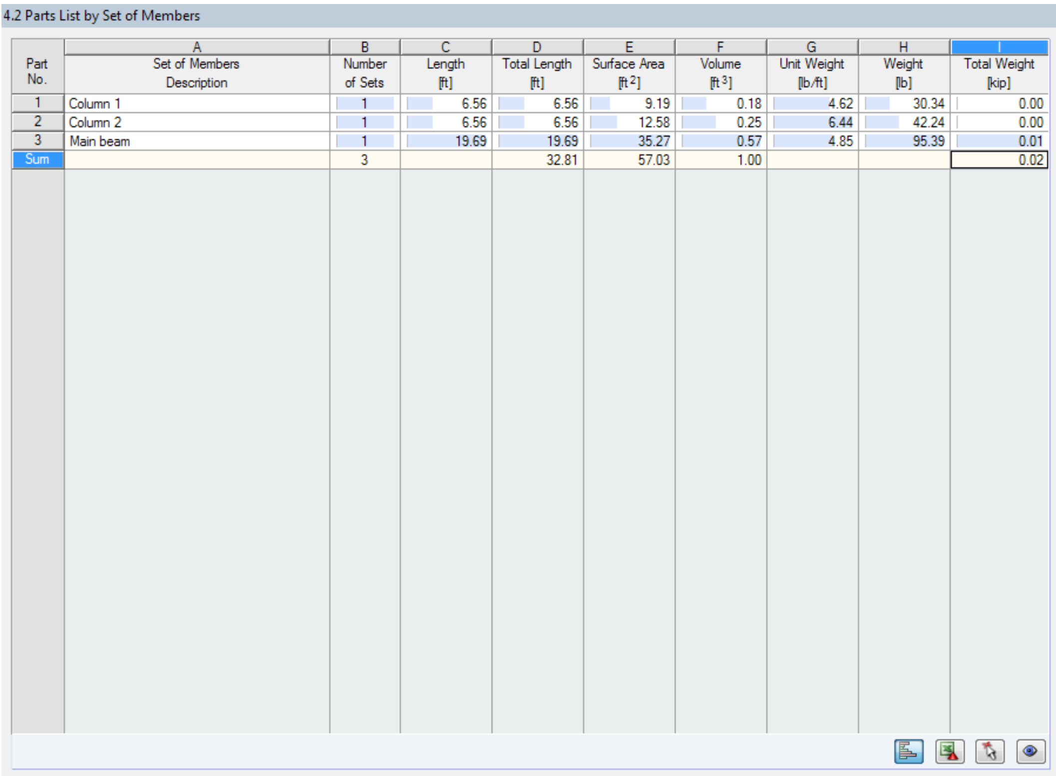
The final column gives the total mass of each part.

Sum

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

4.10 Parts List by Set of Members

The last result window is displayed when you have selected at least one set of members for design. The table summarizes structural groups in a parts list, for example horizontal beams.



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	Column 1	1	6.56	6.56	9.19	0.18	4.62	30.34	0.00
2	Column 2	1	6.56	6.56	12.58	0.25	6.44	42.24	0.00
3	Main beam	1	19.69	19.69	35.27	0.57	4.85	95.39	0.01
Sum		3		32.81	57.03	1.00			0.02

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

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

5 Results Evaluation

You can evaluate the results in many ways. For this, the buttons below the upper table are useful.

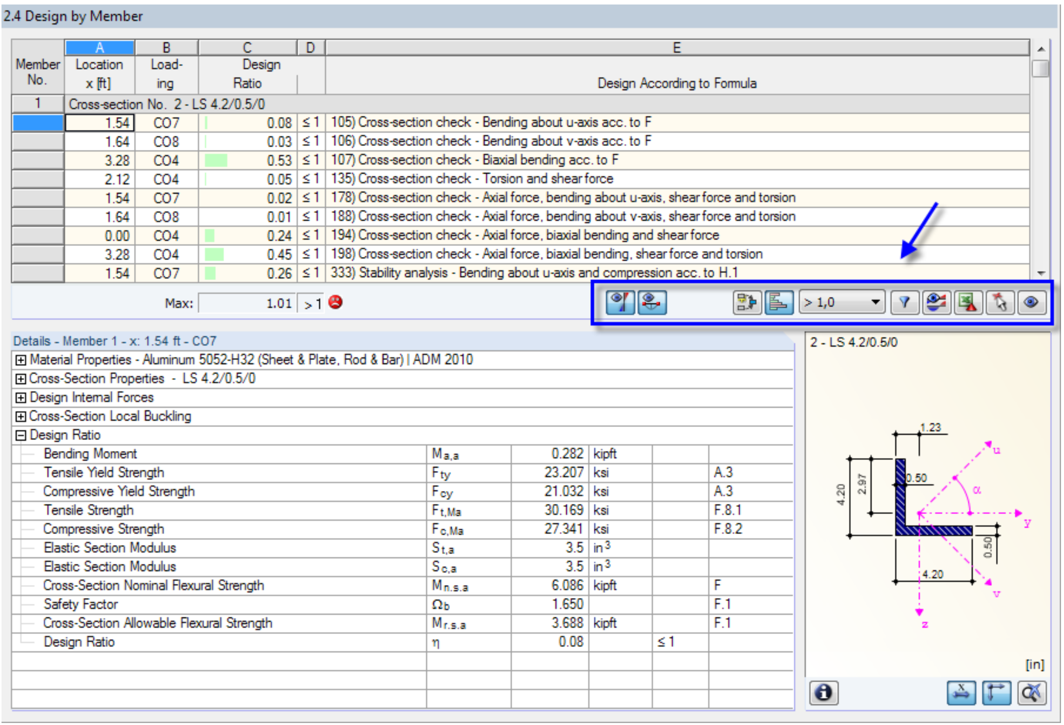


Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

Button	Description	Function
	Ultimate Limit State Designs	Display or hide the results of ULS design
	Serviceability Limit State Designs	Display or hide the results of SLS design
	Result Combination	Create a new result combination from the governing load cases and load combinations
	Color Bars	Display or hide the colored relation scales in the result tables
> 1,0 > 1,0 Max Define	Filter Parameters	Define the criterion to filter results in tables: ratios greater than 1, maximum, or user-defined limit
	Apply Filter	Display only the rows with applied filter parameter (ratios greater than 1, maximum, user-defined limit)
	Result Diagrams	Open the *Result Diagram on Member* Window → Chapter 5.2, page 48
	Excel Export	Export the table to MS Excel or OpenOffice → Chapter 7.4.3, page 59
	Member Selection	Select a member graphically to display its results in the table
	View Mode	Jump to the RFEM/RSTAB work window to change the view

Table 5.1: Buttons in the result windows 2.1 to 2.5

5.1 Results in RFEM/RSTAB Model

You can evaluate the results graphically in the RFEM or RSTAB work window.

RFEM/RSTAB background graphic and view mode

The RFEM or RSTAB work window in the background is useful when you want to find a specific member in the model: The member selected in the result window of RF-/ALUMINUM ADM is highlighted in the selection color in the background graphic. An arrow indicates the relevant x-location of the member that is selected in the table.

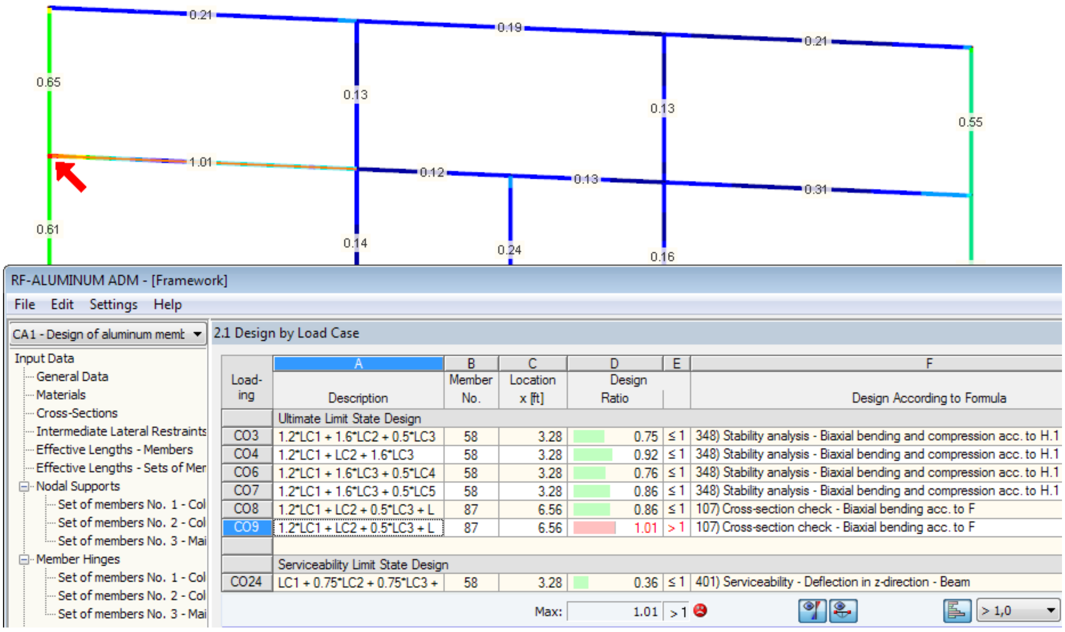
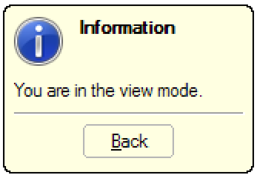


Figure 5.2: Indication of member and relevant *Location x* in the RFEM model



If you cannot improve the view by moving the RF-/ALUMINUM ADM window, click the [button] button to activate the *View mode*: The program hides the window so that you can modify the display in the RFEM/RSTAB user interface. The view mode provides the functions of the *View* menu, for example zooming, moving, or rotating the view. The pointer remains visible.

Click [Back] to return to the RF-/ALUMINUM ADM add-on module.

RFEM/RSTAB work window

You can graphically check the design ratios in the RFEM or RSTAB model: Click [Graphics] to exit the add-on module. Then the design ratios are displayed in the work window of RFEM or RSTAB – like internal forces of a load case.

In the *Results* navigator, you can specify which design ratios of the ultimate or serviceability limit state designs are to be displayed.

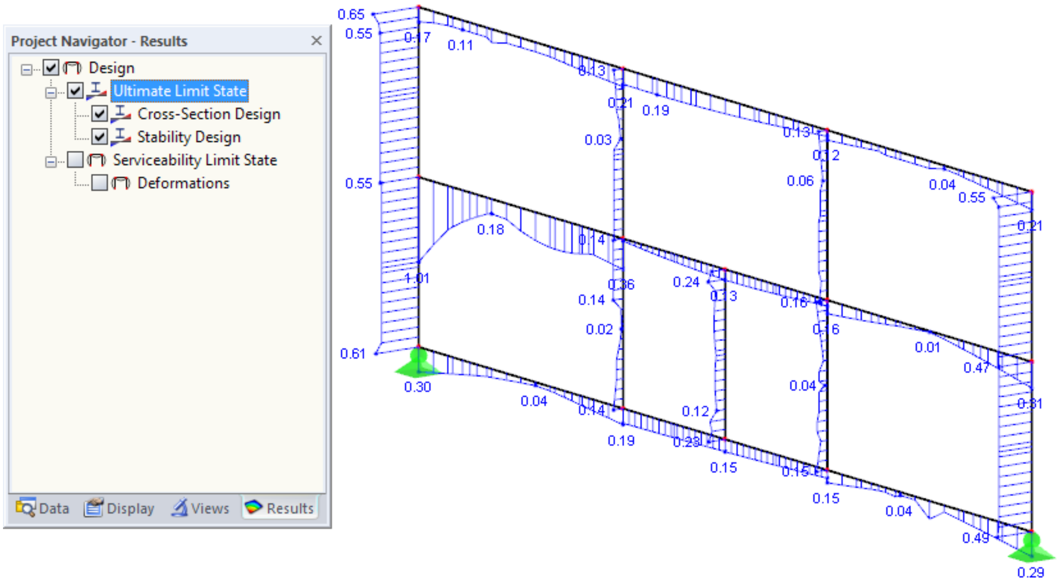
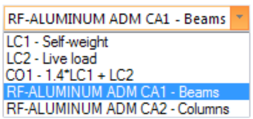


Figure 5.3: *Results* navigator for *Ultimate Limit State* and *Serviceability Limit State* designs

To turn the display of results on or off, click the [Show Results] button (likewise to turning internal forces on or off). To display the result values, click the [Show Values] button to the right of it.

The RFEM/RSTAB tables are not relevant for the evaluation of the RF-/ALUMINUM ADM results.



You can set the design cases (see Chapter 7.1) by means of the list in the RFEM or RSTAB menu bar.

To adjust the graphical representation of the results, you can select **Results** → **Members** in the *Display* navigator. The display of the design ratios is *Two-Colored* by default.

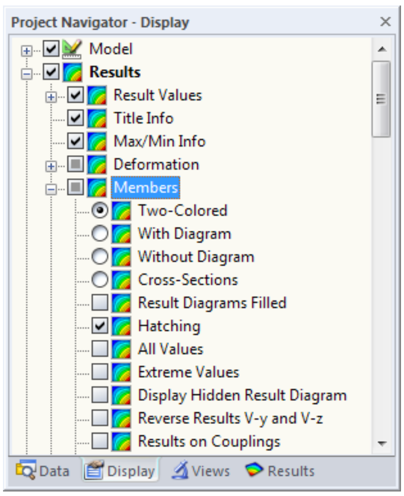


Figure 5.4: *Display* navigator: *Results* → *Members*

When you have chosen one of the multicolor representations (options *With/Without Diagram*, *Cross-Sections*), the color panel is displayed, too. It provides common control functions which are described in the RFEM or RSTAB manual, Chapter 3.4.6.

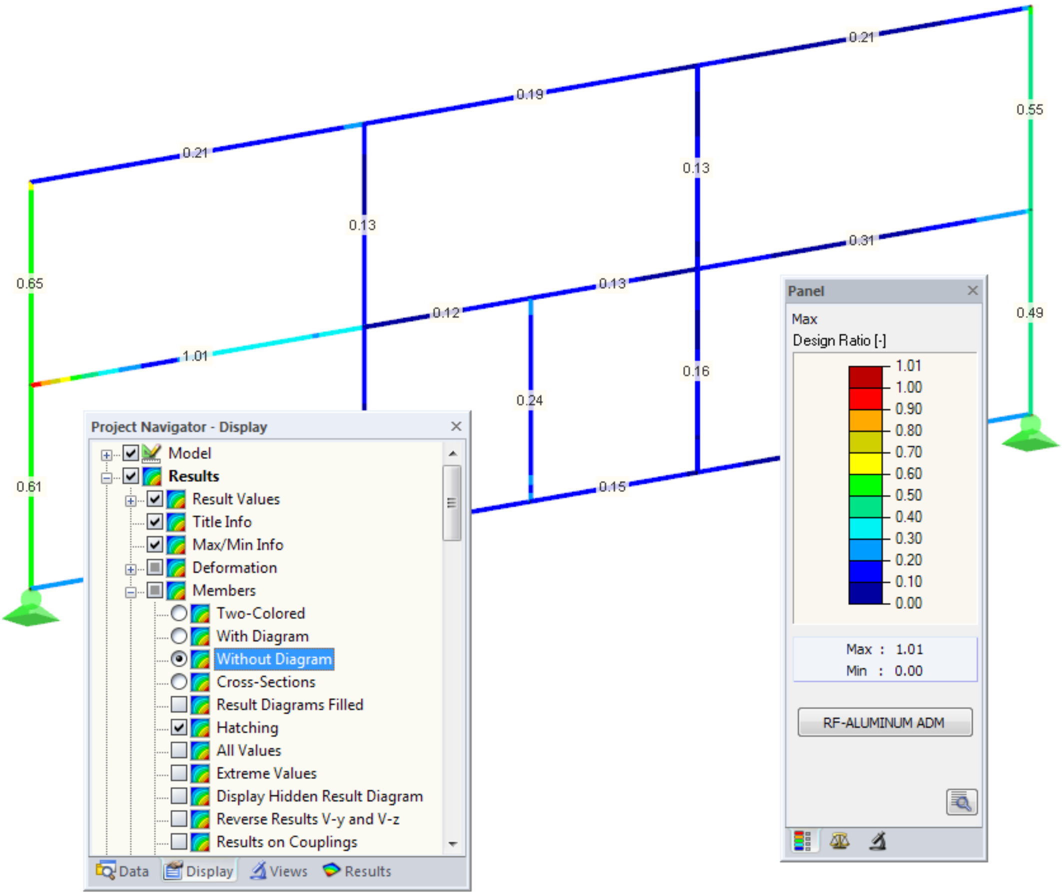


Figure 5.5: Design ratios with display option *Without Diagram*

You can transfer the graphics of design results to the printout report (see Chapter 6.2, page 52).

To return to the add-on module, click [RF-/ALUMINUM ADM] in the panel.

5.2 Result Diagrams

You can also graphically evaluate the member results in the result diagrams.

Select the member (or set of members) in the RF-/ALUMINUM ADM result window by clicking in the table row of the member. Then, open the *Result Diagram on Member* dialog box by clicking the button shown on the left. This button is located below the upper result table (see Figure 5.1, page 44).

The result diagrams can also be accessed in the work window of RFEM or RSTAB via the menu

Results → Result Diagrams for Selected Members

or the corresponding button in the RFEM/RSTAB toolbar.

A new window appears. It shows the distribution of the design ratios along the selected member or set of members.

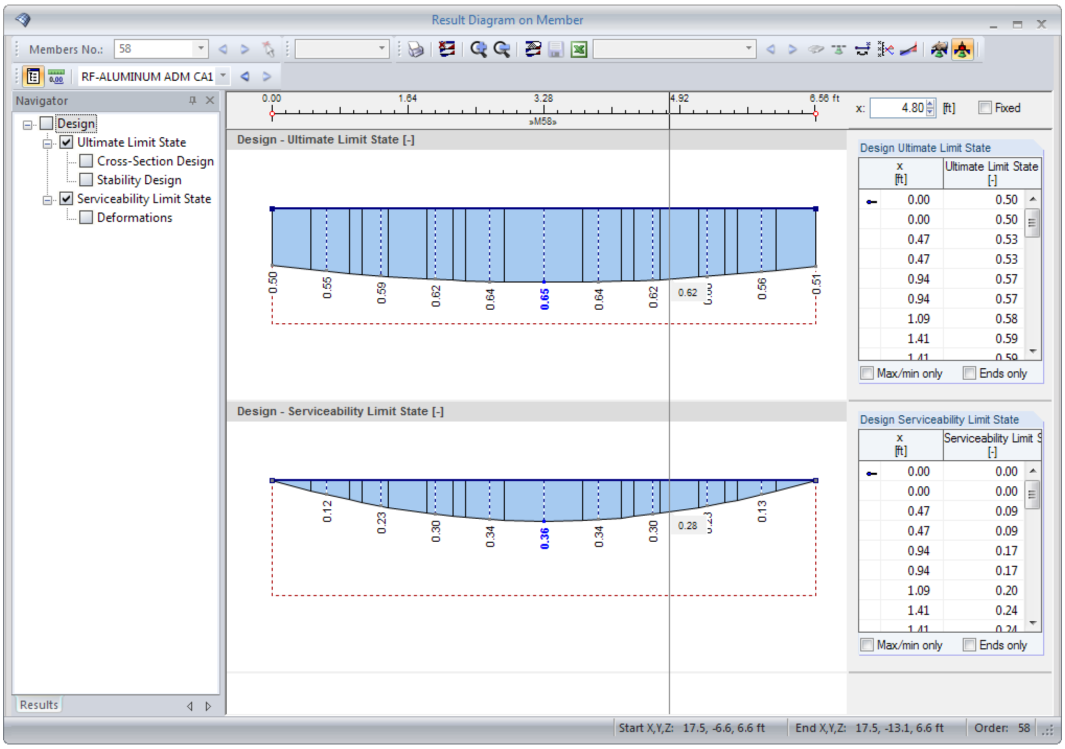
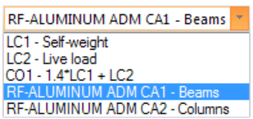


Figure 5.6: Dialog box *Result Diagram on Member*

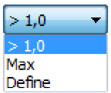
You can set the results of the ultimate or serviceability limit state designs in the navigator.



Use the list in the toolbar above to select the relevant RF-/ALUMINUM ADM 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



Basically, the RF-/ALUMINUM ADM result windows allow you to sort the results by various criteria. You can also use the filter options (see Figure 5.1, page 44) to reduce the tabular output with respect to design ratios. The function is described in this Knowledge Base article: https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/000733

Furthermore, you can use the filter options described in Chapter 9.9 of the RFEM manual to evaluate the results graphically.

For the RF-/ALUMINUM ADM graphics, you can also use the *Visibility* options to filter the members and evaluate them specifically (see RFEM manual, Chapter 9.9.1 or RSTAB manual, Chapter 9.7.1).

Filtering designs

You can use the design ratios as filter criteria in the work window of RFEM or RSTAB (accessible by clicking [Graphics]). To apply this function, the panel must be displayed. If it is disabled, click the RFEM/RSTAB menu

View → Control panel

or use the toolbar button shown on the left.

The panel is described in the RFEM or RSTAB manual, Chapter 3.4.6. The settings for filtering the ratios can be defined in the first tab (Color spectrum). As this tab is not available for the two-colored results, the options *Colored With/Without Diagram* or *Cross-Sections* have to be set in the *Display* navigator.

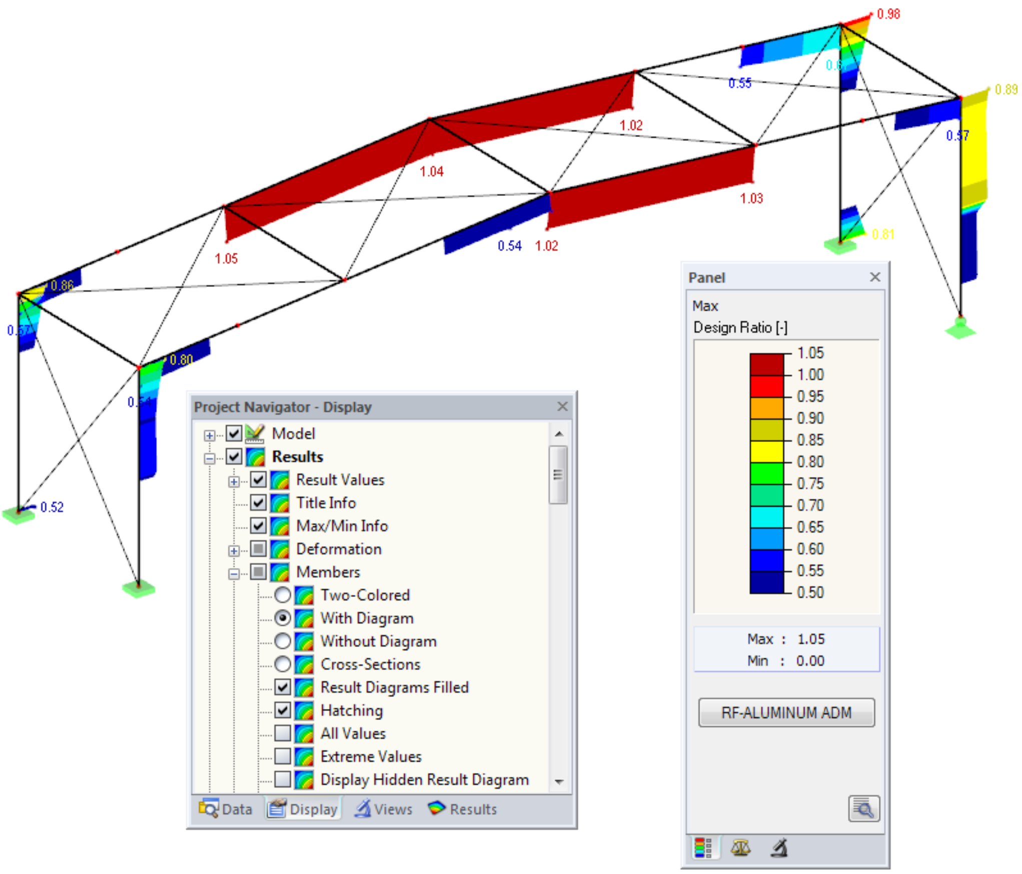


Figure 5.7: Filtering design ratios with adjusted color scale

Figure 5.7 presents a modified color spectrum: Only ratios greater than 0.50 are displayed in the color ranges between blue and red.

The function *Display hidden result diagram* in the *Display* navigator (**Results** → **Members**) displays all ratios beyond the values of the color spectrum. Those diagrams are drawn as dotted lines.

Filtering members

In the *Filter* tab of the control panel, you can enter the numbers of members whose results you want to see exclusively. This function is described in Chapter 9.9.3 of the RFEM manual or Chapter 9.7.3 of the RSTAB manual.

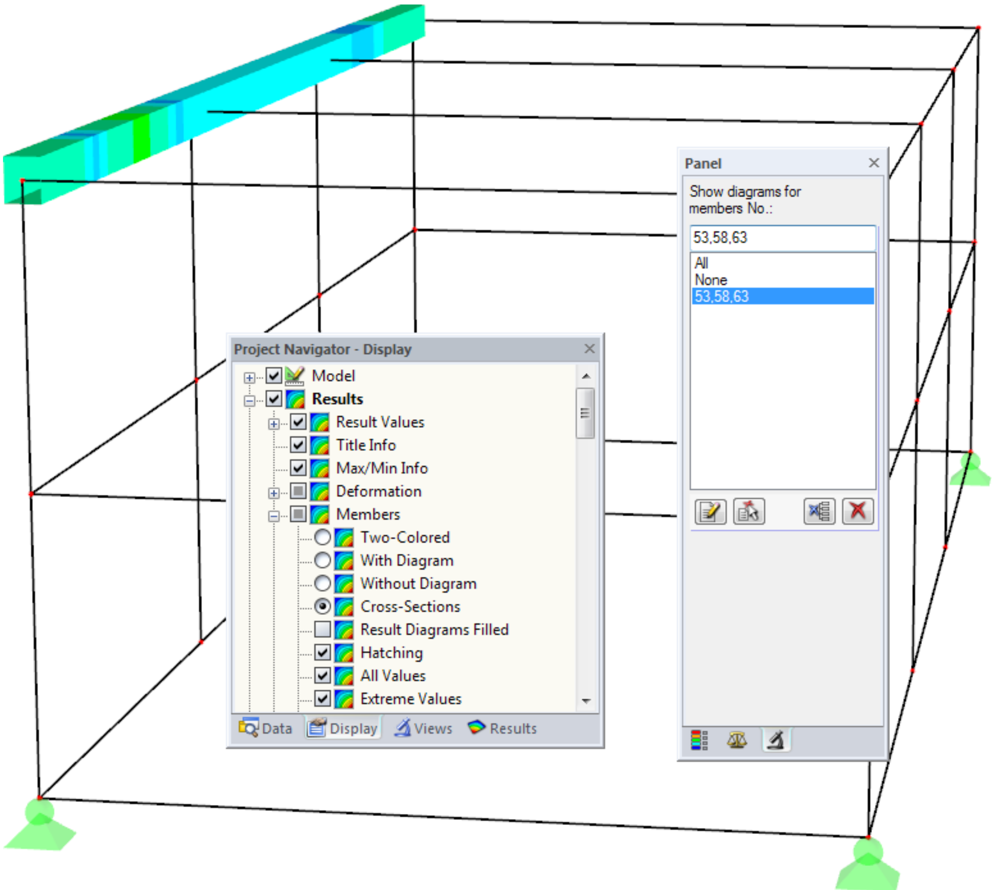


Figure 5.8: Member filter for the design ratios of a box girder

In contrast to the visibility mode, the entire model is displayed. Figure 5.8 shows the design ratios of one specific box girder. All other members of the model are displayed as well, but they have no design ratios.

6 Printout

6.1 Printout Report

In the same way as for RFEM or RSTAB, a printout report can be created for the RF-/ALUMINUM ADM data. You can illustrate the documentation by individual images and comments. The selection in the printout report controls which data of the add-on module is included in the final printout.

The printout report is described in the manuals of RFEM and RSTAB. 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 arranged.

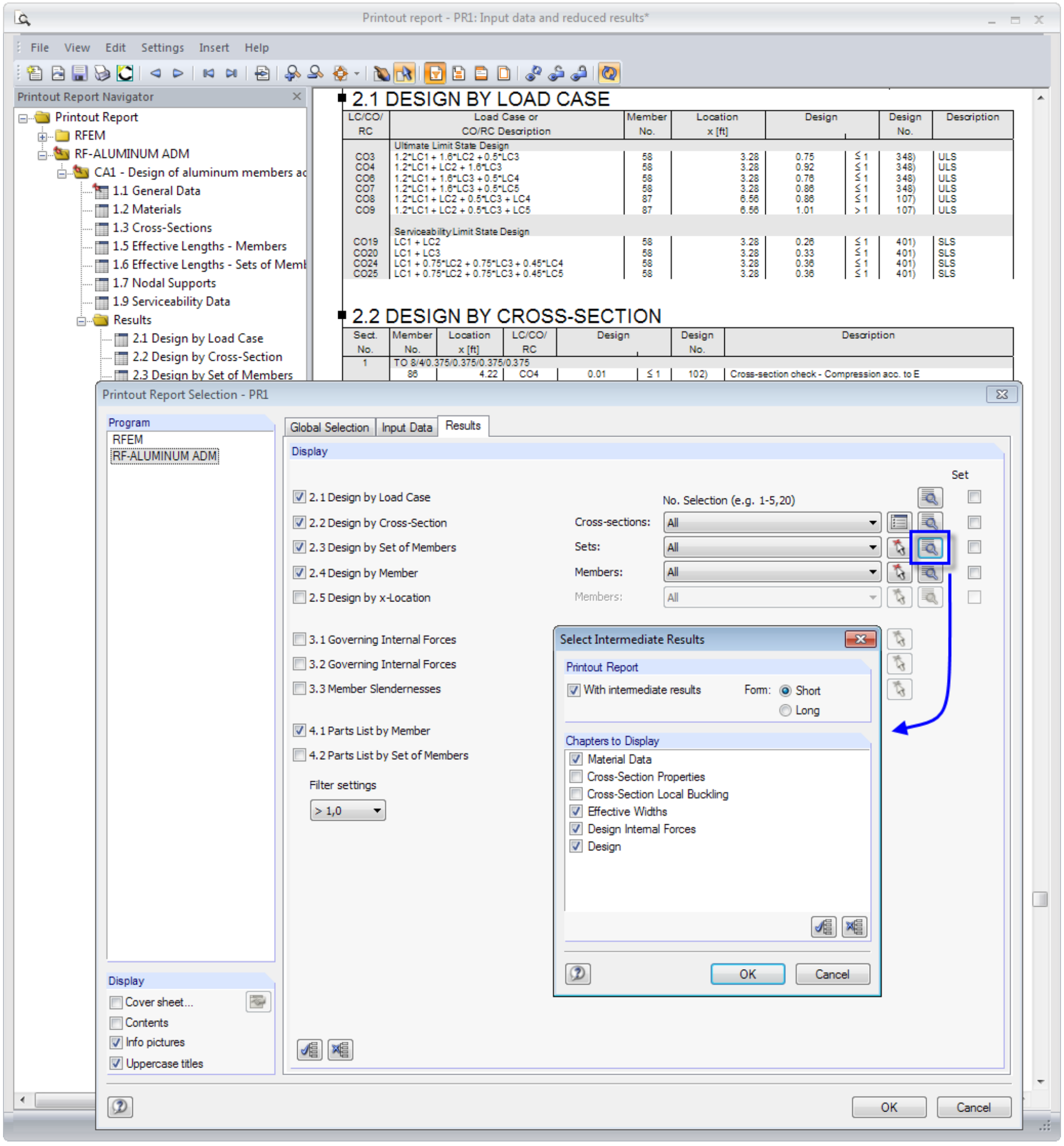


Figure 6.1: Selecting designs and intermediate results in the printout report

Click the [Details] button to specify if you want to include the intermediate results in the printout report as well. You can then select the *Chapters to Display* as seen in Figure 6.1. The detailed results can be recorded in a *Short* (compact compilation) or *Long* (list representation) form.

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

6.2 Graphic Printout

In RFEM or RSTAB, you can add every view of the work window to the printout report or send the image directly to your printer. In this way, you can prepare the design ratios displayed in the RFEM or RSTAB model for the printout.

The printing of graphics is described in the manuals of RFEM and RSTAB, Chapter 10.2.

Designs in RFEM/RSTAB model

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

File → Print Graphic

or use the toolbar button shown on the left.

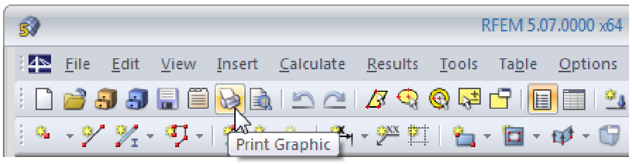


Figure 6.2: *Print Graphic* button in RFEM toolbar

Result Diagrams

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

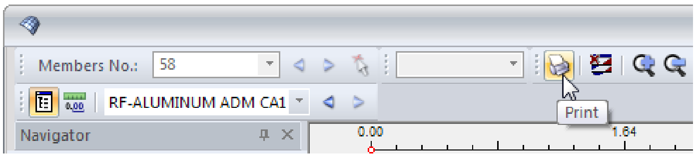


Figure 6.3: *Print* button in the dialog box *Result Diagram on Member*

The *Graphic Printout* dialog box appears.

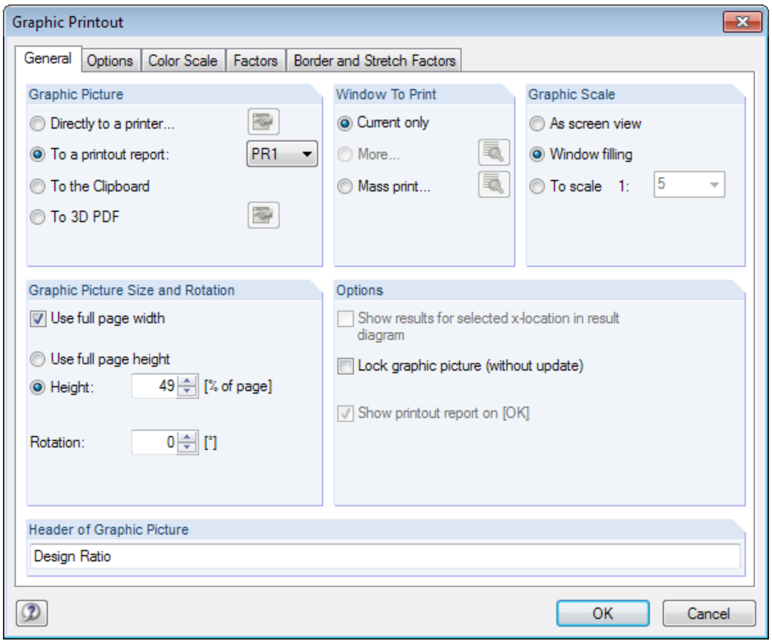
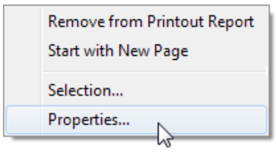


Figure 6.4: Dialog box *Graphic Printout*, tab *General*



This dialog box is described in detail in the manuals of RFEM and RSTAB, Chapter 10.2, inclusive of the other tabs of the dialog box.

When you have printed the image to the printout report, you can move it anywhere within the report by drag-and-drop.

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

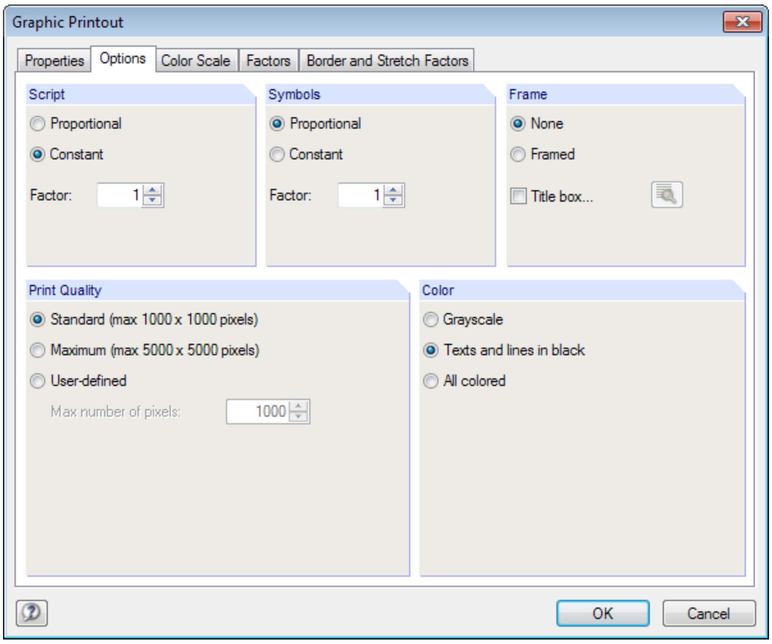


Figure 6.5: Dialog box *Graphic Printout*, tab *Options*

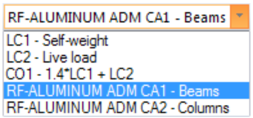
7 General Functions

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

7.1 Design Cases

Design cases allow you to group members for the design. In this way, you can combine groups of structural elements or analyze members with particular design specifications (for example modified materials, strengths, restraints, etc.).

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



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

Create New Design Case

To create a new design case, use the RF-/ALUMINUM ADM menu and click

File → **New Case**.

The following dialog box appears:

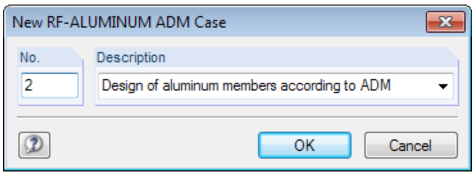


Figure 7.1: Dialog box *New RF-ALUMINUM ADM Case*

In this dialog box, enter the (not yet used) *Number* of the new design case. The *Description* of the design case makes the selection in the load case list easier.

Click [OK] to open the *1.1 General Data* Window of RF-/ALUMINUM ADM and enter the new data.

Rename Design Case

To change the description of a design case, use the RF-/ALUMINUM ADM menu and click

File → **Rename Case**.

The following dialog box appears:

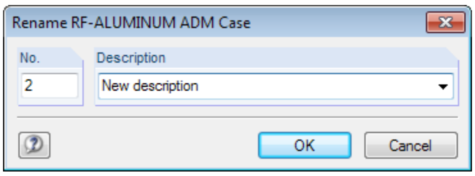


Figure 7.2: Dialog box *Rename RF-ALUMINUM ADM Case*

In this dialog box, you can specify a different *Description* as well as a different *Number* of the design case.

Copy Design Case

To copy the input data of the current design case, use the RF-/ALUMINUM ADM menu and click

File → **Copy Case**.

The following dialog box appears:

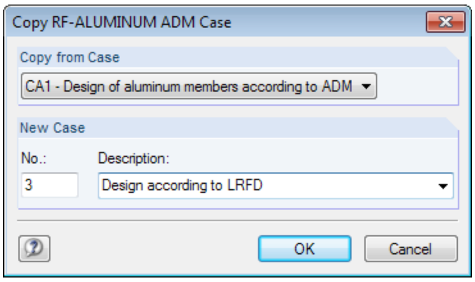


Figure 7.3: Dialog box *Copy RF-ALUMINUM ADM Case*

Define the *Number* and, if necessary, a *Description* for the new case.

Delete Design Case

To delete design cases, use the RF-/ALUMINUM ADM menu and click

File → **Delete Case**.

The following dialog box appears:

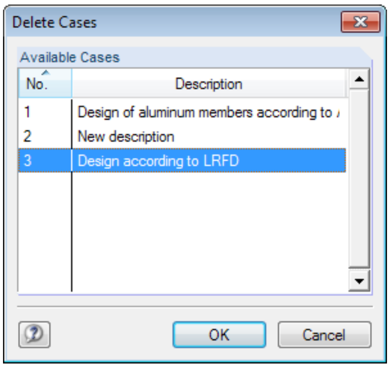
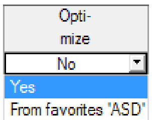


Figure 7.4: Dialog box *Delete Case*

You can select the design case in the list of *Available Cases*. To delete the selected case, click [OK].

7.2 Cross-Section Optimization



RF-/ALUMINUM ADM offers you the option to optimize overloaded or low-utilized cross-sections. To do this, open the drop-down list in column D or E in the *1.3 Cross-Sections* Window and select *Yes* (for parametric sections) or *From current row* (for rolled sections).

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

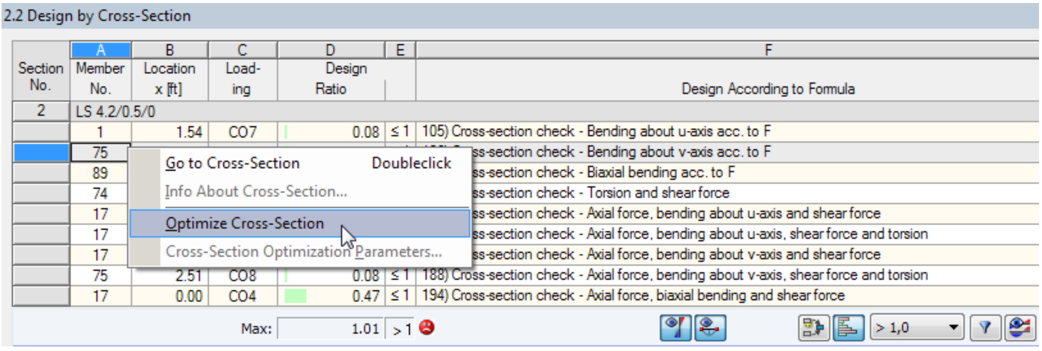


Figure 7.5: Shortcut menu in tables

During the optimization, RF-/ALUMINUM ADM determines the section that fulfills the resistance analysis requirements in the most "optimal" way, that is, comes as close as possible to the maximum allowable design ratio as specified in the *Details* dialog box (see Figure 3.4, page 31). The required cross-section properties are determined with the internal forces from RFEM or RSTAB. If a different section proves to be more favorable, that one is used for the design. Then, the graphic in Window 1.3 shows two cross-sections: the original section from RFEM/RSTAB and the optimized one (see Figure 7.7).

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

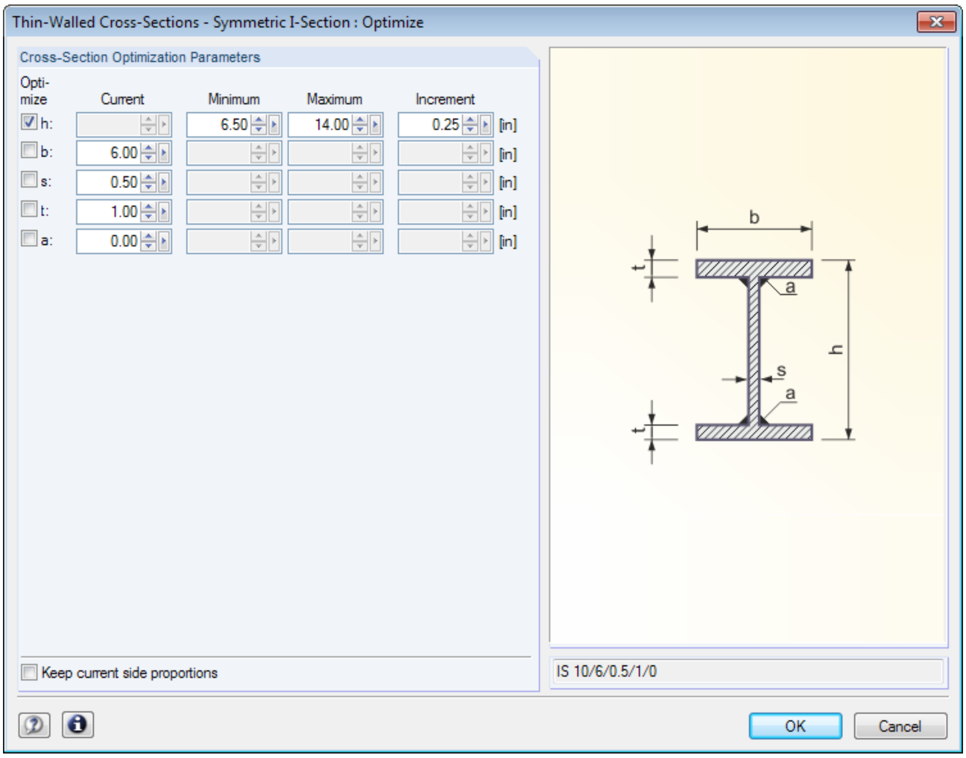


Figure 7.6: Dialog box *Thin-Walled Cross-Sections - Symmetric I-Section: Optimize*

By selecting the check boxes in the *Optimize* column, you decide which parameter(s) you want to modify. You can then specify the upper and lower limits of the parameter(s) in the *Minimum* and

Maximum columns. The *Increment* column controls the interval in which the size of the parameter varies during the optimization.

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

During the optimization, the internal forces are not automatically recalculated with the modified cross-sections: It is up to you to decide which sections are to be transferred to RFEM or RSTAB for a new calculation. Due to the optimized cross-sections and modified stiffnesses, the internal forces of the model may differ considerably. Therefore, we recommend recalculating the internal forces of the modified model after the first optimization. After that, you can optimize the cross-sections once again.

To export the modified cross-sections to RFEM or RSTAB, go to the *1.3 Cross-Sections* Window and click

Edit → **Export All Cross-Sections to RFEM**.

Alternatively, you can use the shortcut menu in Window 1.3.

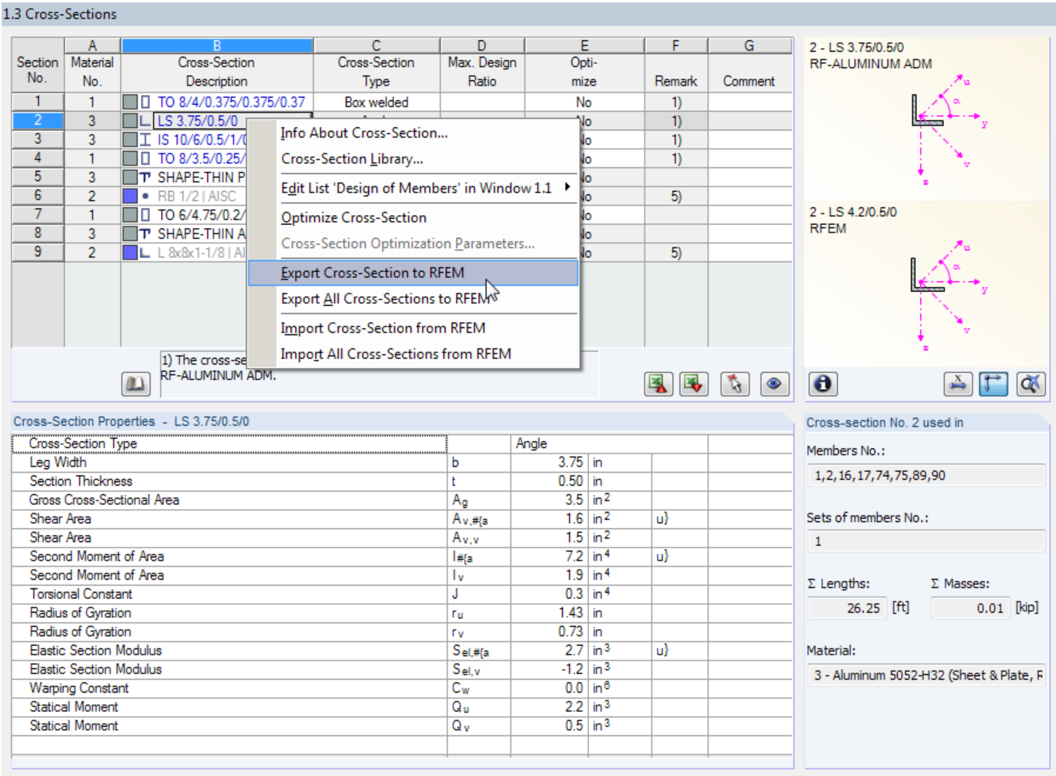


Figure 7.7: Shortcut menu in Window *1.3 Cross-Sections*

Before the modified cross-sections are transferred to RFEM or RSTAB, the program asks you whether the results of RFEM/RSTAB should be deleted.

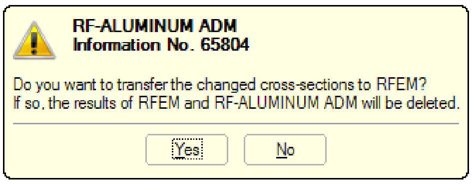
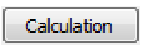


Figure 7.8: Query when exporting sections



When you confirm this query and start the [Calculation] in the RF-/ALUMINUM ADM module, the new internal forces and design ratios will be determined in a 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 options shown in Figure 7.7. Please note that this option is only available in the *1.3 Cross-Sections* Window.

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

7.3 Units and Decimal Places

The units and decimal places are managed in one dialog box for RFEM/RSTAB and for all add-on modules. In RF-/ALUMINUM ADM, you can use the menu to adjust the units by clicking

Settings → Units and Decimal Places.

The *Units and Decimal Places* dialog box appears which you already know from RFEM or RSTAB. RF-/ALUMINUM ADM is preset in the *Program / Module* list.

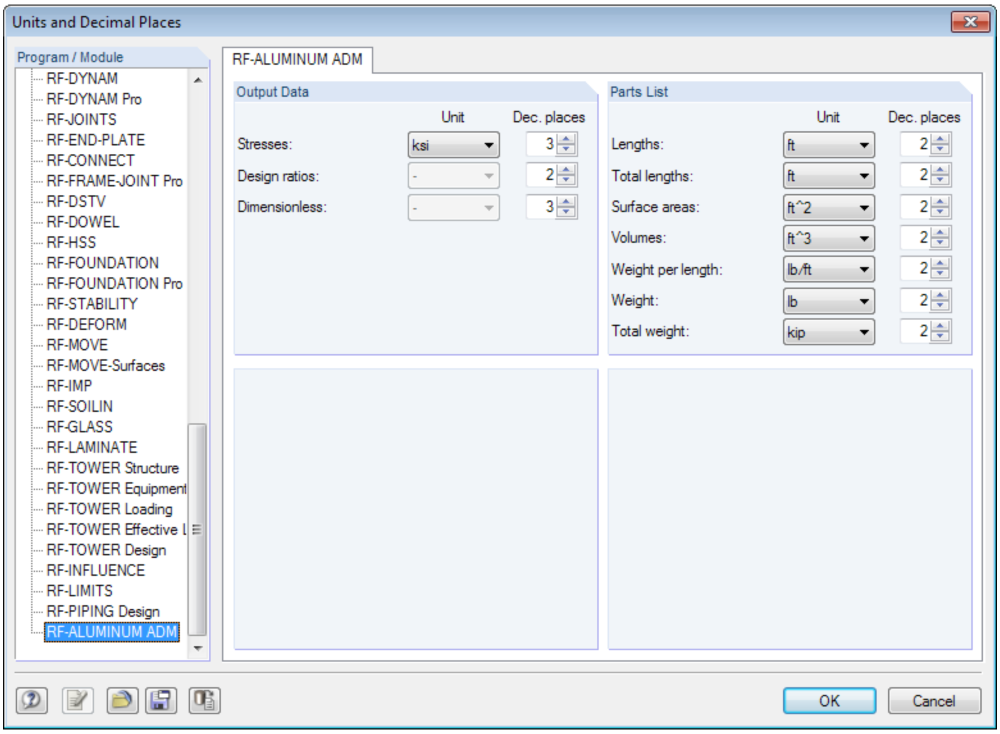


Figure 7.9: Dialog box *Units and Decimal Places*

You can save the settings as a user profile to apply them in other models. These functions are described in Chapter 11.1.3 of the manual of RFEM or RSTAB.

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RFEM

When you have adjusted the materials in RF-/ALUMINUM ADM for design, you can export the modified materials to RFEM or RSTAB in a similar way as you export cross-sections: Open the *1.2 Materials* Window and then use the menu

Edit → Export All Materials to RFEM/RSTAB.

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

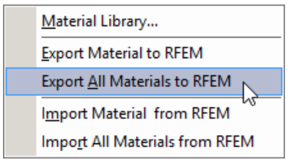
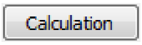


Figure 7.10: Shortcut menu in Window *1.2 Materials*



Before the materials are transferred to RFEM or RSTAB, the program asks you whether the results should be deleted. When you confirm the query and start the [Calculation] in RF-/ALUMINUM ADM, the new internal forces and design ratios will be determined in a single calculation run.

If the modified materials have not been exported to RFEM/RSTAB yet, you can reimport the original materials in the design module by using the options shown in Figure 7.10. Please note that this option is only available in the *1.2 Materials* Window.

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

When you have adjusted the effective lengths in RF-/ALUMINUM ADM for design, you can export the modified lengths to RFEM or RSTAB in a similar way as you export cross-sections: Go to the *1.5 Effective Lengths - Members* Window and click

Edit → Export All Effective Lengths to RFEM/RSTAB

or use the corresponding option in the shortcut menu of Window 1.5.

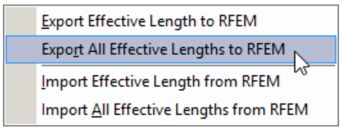


Figure 7.11: Shortcut menu in Window *1.5 Effective Lengths - Members*

Before the effective lengths are transferred to RFEM or RSTAB, the program asks you whether the results should be deleted.

If the modified effective lengths have not been exported to RFEM/RSTAB yet, you can reimport the original lengths in Window *1.5 Effective Lengths - Members* via the options shown in Figure 7.11.

7.4.3 Exporting Results

The RF-/ALUMINUM ADM results can also be used by other programs.

Clipboard

To copy cells selected in the result tables 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-/ALUMINUM ADM into the global printout report (see Chapter 6.1, page 51) and export them subsequently by clicking

File → Export to RTF.

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

Excel / OpenOffice

RF-/ALUMINUM ADM provides a function for directly exporting data to MS Excel, OpenOffice Calc, or the CSV file format. To open the corresponding dialog box, click

File → Export Tables.

The following export dialog box appears:

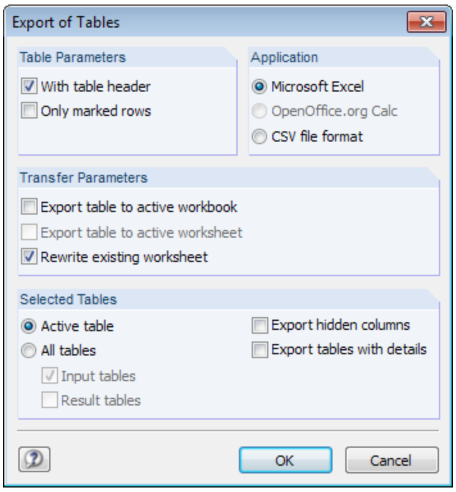


Figure 7.12: Dialog box *Export of Tables*

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

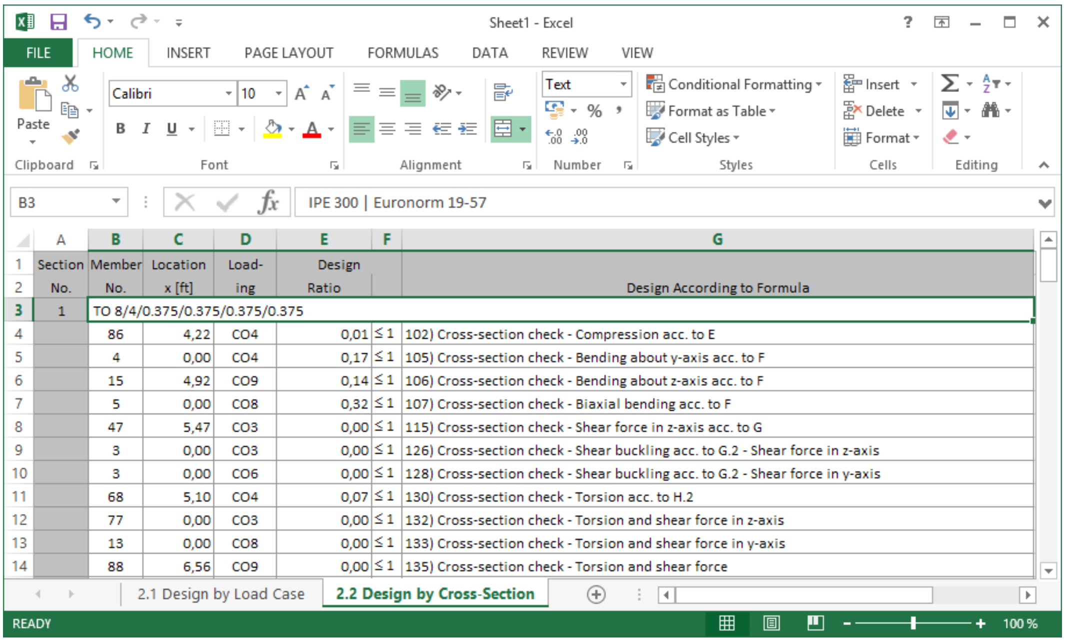


Figure 7.13: Results in *Excel*

Literature

[1] *Aluminum Design Manual*. The Aluminum Association, Inc., Arlington, VA, 2020.
[2] *Aluminum Design Manual*. The Aluminum Association, Inc., Arlington, VA, 2015.
[3] *Aluminum Design Manual*. The Aluminum Association, Inc., Arlington, VA, 2010.
[4] *Commentary on the Specification for Structural Steel Buildings*. American Institute of Steel Construction, Inc., Chicago, IL, 2005.

Index