

Version December 2018

Add-on Module

RF-/STEEL AISC

Design of Steel Members According to ANSI/AISC 360

Program Description

All rights, including those of translations, are reserved. No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of DLUBAL SOFTWARE, INC.



© Dlubal Software, Inc. 2018 The Graham Building 30 South 15th Street 15th Floor Philadelphia, PA 19102 USA

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

Contents

Contents

1.	Introduction
1.1	Add-on Module RF-/STEEL AISC3
1.2	Using the Manual
1.3	Starting RF-/STEEL AISC
2.	Input Data
2.1	General Data
2.1.1	Ultimate Limit State
2.1.2	Serviceability Limit State
2.2	Materials
2.3	Cross-Sections
2.4	Intermediate Lateral Restraints
2.5	Effective Lengths - Members
2.6	Effective Lengths - Sets of Members
2.7	Design Parameters
2.8	Nodal Supports - Sets of Members
2.9	Member Hinges - Sets of Members
2.10	Parameters - Sets of Members
2.11	Serviceability Data
3.	Calculation
3.1	Detail Settings
3.1.1	Ultimate Limit State
3.1.2	Stability
3.1.3	Serviceability
3.1.4	Warping Torsion
3.1.5	General
3.2	Starting Calculation
4.	Results
4.1	Design by Load Case
4.2	Design by Cross-Section
4.3	Design by Set of Members
4.4	Design by Member
4.5	Design by x-Location
4.6	Governing Internal Forces by Member
4.7	Governing Internal Forces by Set of Members
4.8	Member Slendernesses
4.9	Parts List by Member
4.10	Parts List by Set of Members
5.	Results Evaluation
5.1	Results in RFEM/RSTAB Model
5.2	Result Diagram
5.3	Filter for Results
5.4	Mode Shapes
6.	Printout
6.1	Printout Report
6.2	Graphic Printout
7.	General Functions

⊿ Dlubal

Contents



7.1	Design Cases	. 55
7.2	Cross-Section Optimization	57
7.3	Units and Decimal Places	59
7.4	Data Transfer	. 60
7.4.1	Exporting Materials to RFEM/RSTAB	. 60
7.4.2	Exporting Effective Lengths to RFEM/RSTAB	. 60
7.4.3	Exporting Results	. 60
8.	Example	62
8.1	Design Values	. 62
8.2	Cross-Section Properties - W 8x28	63
8.3	Material Properties - A992	63
8.4	Classification of Cross-Section	. 64
8.5	Design for Compression	66
8.5.1	Flexural Buckling about y-Axis (Major Principal Axis)	66
8.5.2	Flexural Buckling about z-Axis (Minor Principal Axis)	67
8.6	Design for Flexure	68
8.6.1	Flexure about y-Axis (Major Principal Axis)	. 68
8.6.1.1	Yielding	68
8.6.1.2	Lateral-Torsional Buckling	68
8.6.2	Flexure about z-Axis (Minor Principal Axis)	. 69
8.6.2.1	Yielding	. 69
8.6.2.2	Flange Local Buckling	70
8.7	Design for Combined Forces	70
Α.	Literature	71
В.	Index	72



1.1 Add-on Module RF-/STEEL AISC

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



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

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

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

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

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

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

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

Your DLUBAL team

1.2 Using the Manual

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



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



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



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

Menu

To start the program from the menu bar, select

Add-on Modules \rightarrow Design - Steel \rightarrow RF-/STEEL AISC.

Add	d-on Modules Window	H	elp		
-	Current Module			- d > <u>@ 1</u>	🖺 🔎 💯 🖬 📾 🔡 🎬 🎬 🍄 🥵 🎘 🕼 🖒 🔿
	Design - Steel	•	<i>_</i>	RF-STEEL Surfaces	General stress analysis of steel surfaces
	Design - Concrete	×	Ľ	RF-STEEL Member	General stress analysis of steel members
	Design - Timber	×	Lec	RF-STEEL EC3	Design of steel members according to Eurocode 3
	Design - Aluminium	ł	Aisc	RF-STEEL AISC	Design of steel members according to AISC (LRFD or ASD)
	Dynamic	+	II.	RF-STEEL IS	Design of steel members according to IS
	Connections		SIA	RF-STEEL SIA	Design of steel members according to SIA
	Stability		IBS	RF-STEEL BS	Design of steel members according to BS
			1 _{G8}	RF-STEEL GB	Design of steel members according to GB
	Others		Les	RF-STEEL CS	Design of steel members according to CS
		·	TAS	RF-STEEL AS	Design of steel members according to AS
	External Modules	+	*	RF-KAPPA	Flexural buckling analysis
	Stand-Alone Programs	×	₽	RF-LTB	Lateral-torsional and torsional-flexural buckling analysis
			₽.	RF-FE-LTB	Lateral-torsional and torsional-flexural buckling analysis by FEM
			12	RF-EL-PL	Elastic-plastic design
				RF-C-TO-T	Analysis of limit slenderness ratios (c/t)
			1	PLATE-BUCKLING	Plate buckling analysis

Figure 1.1: Menu Add-on Modules \rightarrow Design - Steel \rightarrow RF-STEEL AISC

Navigator

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

```
\textbf{Add-on Modules} \rightarrow \textbf{RF-/STEEL AISC}.
```

Project Navigator - Data	×						
RFEM							
🗄 📲 Hall-Boulder [2017]							
🚋 🛅 Model Data							
🗄 💼 Load Cases and Combinations							
E Loads	-						
Results	-						
Sections							
🛅 Average Regions							
Drintout Reports							
🗄 🛅 Guide Objects							
🖕 🚞 Add-on Modules							
🚍 🖮 Favorites							
RF-STEEL Surfaces - General stress analysis of steel surfaces							
E RF-STEEL Members - General stress analysis of steel members							
🔤 🚮 RF-DYNAM - Dynamic analysis (Basic, Addition I, Addition II)							
RF-STABILITY - Stability analysis							
🜆 RF-STEEL AISC - Design of steel members according to AISC (LRFD or ASD)							
	Ŧ						
🔽 Data 🖀 Display 🔏 Views							

Figure 1.2: Data navigator Add-on Modules \rightarrow RF-STEEL AISC

2 Input Data

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

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

- Members and sets of members
- Load cases, load and result combinations
- Materials

Cancel

OK

- Cross-sections
- Effective lengths
- Internal forces (in background, if calculated)

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

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

2.1 General Data

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



Figure 2.1: Window 1.1 General Data

2 Input Data

Design of

Design of			
Members:	1,2,4-6,8,81-83,99-102	X	AI
Sets:	1-5	🚯 🗙 🎦	V AI

Figure 2.2: Design of members and sets of members

ſ	X
í	*
l	3

The design can be carried out for *Members* as well as for *Sets of Members*. If you want to design only selected objects, clear the *All* check box. Then you can access the text boxes and enter the numbers of the relevant members or sets of members. The [Delete] button clears the list of preset numbers. The [Select] button enables you to define the objects graphically in the work window of RFEM or RSTAB.

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

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

Design According to



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

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

Comment

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

2.1.1 Ultimate Limit State

Ultimate Limit 9	State	Serviceability Limit State					
Existing Load (Existing Load Cases and Combinations Selected for Design						
4.6a CO23	D + 0	.75S + 0.45W2 + Imp1	1		2.3 2 CO34	1.2D + 0.5S + 1.6L + Imp1	
4.6a CO24	D + 0	.75S + 0.45W2 + Imp2			2.3 2 CO35	1.2D + 0.5S + 1.6L + Imp2	
2.4 7 CO26	0.6D	+ 0.6W1 + Imp1					
2.4 7 CO27	0.6D	+ 0.6W1 + Imp2					
2.4 7 CO28	0.6D	+ 0.6W2 + Imp1					
2.4 7 CO29	0.6D	+ 0.6W2 + Imp2					
2.3 1 CO30	1.4D	+lmp1					
2.3 1 CO31	1.4D	+ Imp2		>			
2.3 2 CO32	1.2D	+ 1.6L + Imp1					
2.3 2 CO33	1.2D	+ 1.6L + Imp2		>>			
2.3 3 CO36	1.2D	+ 1.6S + L + Imp1					
2.3 3 CO37	1.2D	+ 1.6S + L + Imp2		_			
2.3 3 CO38	1.2D	+ 1.6S + Imp1		\triangleleft			
2.3 3 CO39	1.2D	+ 1.6S + Imp2					
2.3 3 CO40	1.2D	+ 1.6S + 0.5W1 + Imp1		~			
2.3 3 CO41	1.2D	+ 1.6S + 0.5W1 + Imp2					
2.3 3 CO42	1.2D	+ 1.6S + 0.5W2 + Imp1					
2.3 3 CO43	1.2D	+ 1.6S + 0.5W2 + Imp2					
2.3 4 CO44	1.2D	+ 0.5S + L + W1 + Imp1					
2.3 4 CO45	1.2D	+ 0.5S + L + W1 + Imp2					
2.3 4 CO46	1.2D	+ 0.5S + L + W2 + Imp1					
2.3 4 CO47	1.2D	+ 0.5S + L + W2 + Imp2					
2.3 4 CO48	1.2D	+ L + W1 + Imp1					
2.3 4 CO49	1.2D	+ L + W1 + Imp2	4				
All (69)		~ 24	33				1 SC

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

Existing Load Cases and Combinations

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

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

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

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

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

21	Selects all load cases in the list			
82	Inverts the selection of load cases			

Table 2.1: Buttons in Ultimate Limit State tab

Selected for Design

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



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



2.1.2 Serviceability Limit State



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

Existing Load Cases and Combinations

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

Selected for Design



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

Details...

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

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

2.2 Materials

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

1.2 Materi	als						
	А			B			
Material	Material						
No.	Description	Description Comment					
1	Steel A992 ANSI/AISC 360-16:2016	-					
2	Steel A36M LANSI/AISC 360-16:2016						
3	Concrete f'c = 4000 psi ACL 318-14						
4	Steel A501 ANSI/AISC 360-16:2016						
				× ×	۲		
Material P	roperties						
🖂 Main F	roperties						
Mod	ulus of Elasticity	E	28998.60	ksi			
She	ar Modulus	G	11153.30	ksi			
Pois	son´s Ratio	v	0.30				
Spe	cific Weight	γ	499.75	lbf/ft3			
Coe	fficient of Thermal Expansion	α	6.6667E-06	1/°F			
- Part	ial Safety Factor	ΥM	1.00		Material No. 1 used in		
Additio	nal Properties						
🗆 Thio	kness Range t ≤ 4.00 in	-			Cross-sections No.:		
— Y	ield Strength	Fy	50.00	ksi	1-3,10		
	ltimate Strength	Fu	65.00	ksi			
					Members No.:		
					11-16, 19-24, 34-49, 91, 92, 95, 96		
					Sets of members No.:		
					2,3		
					T Loopher T Manage		
					2. Lengths: 2. Masses:		
					442.46 [ft] 18.95 [kip]		

Figure 2.6: Window 1.2 Materials

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

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

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

Material Description

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

A		
Material		
Description		
Steel A992 ANSI/AISC 360-16:2016		
Steel A36	ANSI/AISC 360-16:2016	~
Steel A529, Grade 50	ANSI/AISC 360-16:2016	
Steel A529, Grade 55	ANSI/AISC 360-16:2016	
Steel A572, Grade 42	ANSI/AISC 360-16:2016	
Steel A572, Grade 50	ANSI/AISC 360-16:2016	
Steel A572, Grade 55	ANSI/AISC 360-16:2016	
Steel A572, Grade 60	ANSI/AISC 360-16:2016	
Steel A572, Grade 65	ANSI/AISC 360-16:2016	
Steel A588	ANSI/AISC 360-16:2016	
Steel A709, Grade 36	ANSI/AISC 360-16:2016	¥

Figure 2.7: List of materials

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

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

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

Material Library

or use the [Library] button.

Alternatively, you can use the material library to change a material. To open the library, select **Edit** \rightarrow **Material Library**

Material Library				×	
Filter	Material to Select				
Material category group:	Material Description	Standard		^	
	Steel A36	📰 ANSI/A	ISC 360-16:201	6-6	
	Steel A529, Grade 50	ANSI/A	ISC 360-16:201	6-6	
Material category:	Steel A529, Grade 55	ANSI/A	ISC 360-16:201	6-6	
Steel V	Steel A572, Grade 42	ANSI/A	ISC 360-16:201	6-6	
	Steel A572, Grade 50	ANST/A	JSC 360-16:201	5-6	
Standard group:	Steel 4572 Grade 55		USC 360-16:201	5-6	
ASTM ~	Steel A572, Grade 60		USC 360-16:201	5-6	
Standard	Steel 4572, Grade 65		USC 360-16:201	5-6	
Standard:	Steel A588		USC 360-16:201	5-6	
ANSI/AISC 360-16:2016-6 V	Steel A 200, Crade 36		ISC 360-16:201		
	Steel A709, Grade 50		15C 260-16:201	5-0 5-6	
	Steel A709, Grade 50		150 260 16:201	5-0 5-0	
	Steel A709, Grade 505	AINS1/A	150 360-16:201	0-0	
	Steel A 709, Grade S0W, HPS S0W	ANSI/A	ISC 360-16:201	5-6	
	Steel A709, Grade HPS 70W	ANSI/A	ISC 360-16:201	5-6	
	Steel A709, Grade HPS 100W	W ANSI/AISC 360-16:2010			
🗌 Include invalid 🔯	Steel A913, Grade 50	ANSI/A	ISC 360-16:201	5-6	
	Steel A913, Grade 60	ANSI/A	ANSI/AISC 360-16:2016-6		
Favorites group:	Steel A913. Grade 65	ANST/A	JSC 360-16:201	6-6	
Concrete - BS 🗸 🗠 🎦 쨜	Search:			×	
Material Properties		Steel A	36 ANSVAISC	360-16:2016-6	
Main Properties					
Modulus of Elasticity		E	28998.60	ksi	
- Shear Modulus		G	11153.30	ksi	
Poisson's Ratio		V	0.30	11.0.0.3	
Coefficient of Thermal Expansion		7	499.75	IDT/πt ⁻⁹	
E Additional Properties		u.	0.0007E-00	17.1	
□ Thickness Range t ≤ 8.00 in					
Yield Strength		Fy	36.00	ksi	
Ultimate Strength		Fu	58.00	ksi	
Thickness Range t > 8.00 in					
Yield Strength		Fy	32.00	ksi	
Ultimate Strength		Fu	58.00	ksi	
2			ОК	Cancel	

Figure 2.8: Dialog box *Material Library*

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

OK

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

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

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

2.3 Cross-Sections

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



Coordinate System

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



Figure 2.9: Window 1.3 Cross-Sections

Cross-Section Description

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

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

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

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

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

A modified cross-section will be highlighted in blue.

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



Rolled Cross-Sections - I-Sections					×
Cross-Section Type	To Select		To Select		W 21x101 AISC 14
тгті	Table	Manufacturer/Stand ^	Cross-Section	^	
	IW	AISC 14	W 24x192		
	IM	AISC 14	W 24x176		12.25
	IS	AISC 14	W 24x162		
1 🔹 🛶 📩	ΙHP	AISC 14	W 24x146		
	ТHD	ASTM A 6/A 6M	W 24x131		
	IW	ASTM A 6/A 6M	W 24x117		
	II	ASTM A 6/A 6M - 07	W 24x104		
	IH	ASTM A 6/A 6M - 07	W 24x103		
2	IHL	ASTM A 6/A 6M - 07	W 24x94		8
Filter	ТHD	ASTM A 6/A 6M - 07	W 24x84		71 Jan 19
	ΙHP	ASTM A 6/A 6M - 07	W 24x76		
Manufacturer/Standard group:	IW	ASTM A 6/A 6M - 07	W 24x68		0.50
All ~	Ιw	ASTM A 6/A 6M - 07	W 24x62		• • • • • • • • • • • • • • • • • • •
Manu Garbara (Chandrad	II	-	W 24x55		
Mariuracturer/standard:	I IPE	-	W 21x201		
All	I IPE 750	-	W 21x182		• • • • • • • • • • • • • • • • • • •
Cross-section shape:	I IPEa	-	W 21x166		*
	I IPEo	•	W 21x147		
All	I IPEv	-	W 21x132		
Cross-section note:	I IPB-S	-	W 21x122		
	I IPB-SB	•	W 21x111		
	IHE	•	W 21x101		ð 🕇 🕰
	I HEAA	-	W 21x93		
	THEA	•	W 21x83		
	I HEB	•	W 21x73		
Include invalid 🔗	I HEM	-	W 21x68		
Eavorites group:	I HSL	•	W 21x62		
	ΙHP	-	W 21x55		
	T UB	🚟 BS 4-1 💙	W 21x48	~	W 21X101 AISC 14
⑦ 註 🔊 😱					

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

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

Details...

Cross-Section Type

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

Max. Design Ratio

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

Optimize

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

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

Remark

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

Member with Tapered Cross-Section

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

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

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

Info About Cross-Section



0

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

Info About Cross-Section W 21x101	AISC 14				×
Cross-Section Property	Symbol	Value	Unit	^	W 21x101 AISC 14
Depth	d	21.38	in		
Width	b	12.25	in		
Web thickness	tw	0.50	in		
Flange thickness	tf	0.81	in		12.25
Root fillet radius	r	0.88	in		
Depth of straight web	т	18.00	in		
Distance	k	1.69	in		
Distance	k1	1.06	in		
Cross-sectional area	Α	29.8	in ²		
Shear area	Ay	16.6	in ²		
Shear area	Az	9.8	in ²		2 2 2 2 2 2 2
Shear area according to EC 3	A _{v,y}	20.6	in ²		0 - Y
Shear area according to EC 3	A _{v,z}	11.7	in ²		
Plastic shear area	A pl,y	19.9	in ²		0.50
Plastic shear area	A pl,z	10.3	in ²		
Moment of inertia	Iy	2420.0	in ⁴		
Moment of inertia	Iz	248.0	in ⁴		
Governing radius of gyration	ry	9.02	in		÷
Governing radius of gyration	rz	2.89	in		Z
Radius of gyration of flange plus 1/5 of we	r zg	3, 19	in		
Volume	V	357.6	in ³ /ft		
Weight	wt	101.41	lb/ft		lini
Surface	Asurf	7.73	ft²/ft		
Section factor	Am/V	122.533	1/m		🕂 🔝 🛋 Stress points
Torsional constant	J	5.2	in ⁴		📜 🔝 👼 c/t-Parts
14/		20200.0		~	
۵					Close

Figure 2.11: Dialog box Info About Cross-Section

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

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

Stress Poi	nts of W 21x1	01 AISC 14						×			
	A	B	С	D	E	F	G	W 21x101			
StressP	Coordi	nates	Statical Mom	ents of Area	Thickness	Wan	bing				
No.	y [in]	z [in]	Qy [in ³]	Q _z [in ³]	t [in]	W _{no} [in ²]	Qw [in 4]				
1	-6.12	-10.69	0.0	0.0	0.81	63.0	0.0				
2	-1.13	-10.69	-41.8	-14.7	0.81	11.6	-151.4				
3	0.00	-10.69	-51.2	-15.3	0.81	0.0	-156.7	1 2 3 4 5			
4	1.13	-10.69	-41.8	14.7	0.81	-11.6	151.4				
5	6.12	-10.69	0.0	0.0	0.81	-63.0	0.0				
6	-6.12	10.69	0.0	0.0	0.81	-63.0	0.0				
7	-1.13	10.69	-41.8	14.7	0.81	-11.6	-151.4				
8	0.00	10.69	-51.2	15.3	0.81	0.0	-156.7				
9	1.13	10.69	-41.8	-14.7	0.81	11.6	151.4	12			
10	6.12	10.69	0.0	0.0	0.81	63.0	0.0	i y			
11	0.00	-9.00	-109.6	0.0	0.50	0.0	0.0				
12	0.00	9.00	-109.5	0.0	0.50	0.0	0.0				
13	0.00	0.00	-129.9	0.0	0.50	0.0	0.0				
								in the second			
								0 / 0 0 10			
								z			
2	D 🛐										

Figure 2.12: Dialog box Stress Points of W 21x101

2.4 Intermediate Lateral Restraints

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



Figure 2.13: Window 1.4 Intermediate Lateral Restraints

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

To define the restraints of a specific member, select the *Lateral Restraints* check box in column A. Then the other columns will be accessible where you can enter the parameters. To graphically select the member, click 3.



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

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

Relatively (0 ... 1)

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

2.5 Effective Lengths - Members

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

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

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

Click 🚯 to select a member graphically and show its row.

	-														
	Α	B	С	D	E	F	G	H		J	K	L		М	/
Member	Buckling	Buc	kling About /	Axis y/u	Buc	kling About /	Axis z/v	To	rsional Bucl	kling		L.T.B.			
No.	Possible	Possible	Ky/u	L [ft]	Possible	K _{z/v}	L [ft]	Possible	K _x	L [ft]	Possible	k _{z/v}	kw	Comment	
34	V	V	1.000	19.69	V	1.000	19.	59 🔽	1.000	19.69		1.0	1.0		
35		V	1.000	5.89	V	1.000	5.	39 🔽	1.000	5.89	V	0.5	1.0		
36	√	V	1.000	10.04	V	1.000	10.)4 🔽	1.000	10.04	V	1.0	1.0		
37	V	V	1.000	16.73	V	1.000	16.	73 🔽	1.000	16.73	V	1.0	1.0		
38	V	V	1.000	16.73	V	1.000	16.	73 🗹	1.000	16.73	V	1.0	1.0		
39	2	V	1.000	10.04	✓	1.000	10.)4 🔽	1.000	10.04	V	1.0	1.0		
40	✓	V	1.000	5.89	V	1.000	5.	39 🔽	1.000	5.89	☑	1.0	1.0		
41		V	1.000	19.69	V	1.000	19.	i9 🔽	1.000	19.69		1.0	1.0		
42		✓	1.000	19.69	V	1.000	19.	i9 🔽	1.000	19.69		1.0	1.0		
43	✓	✓	1.000	5.89	✓	1.000	5.	39 🔽	1.000	5.89	✓	1.0	1.0		
Settings - Member No. 36												W 1	4x53 Als	SC 14	3 9
Cross-S	Section					2 - W 1	4x53 AIS	: 14							
Length					L	10.04 ft									
Bucklin	ng Possible						2								
🖃 Bucklin	ng About Majo	or Axis y Po	ssible				2								
Effe	ctive Length	Factor			Ky		1.000						+	8.00	
Men	ber Length				L		10.04	t				+	_+_'_		
🖃 Bucklin	ng About Mina	or Axis z Po	ssible				V						- °		-
- Effe	ctive Length	Factor			Kz		1.000						51	· · · · · ·	
- Men	ber Length				L		10.04	t							
Torsion	nal Buckling F	Possible					V					3.8			
Effe	ctive Length	Factor (for	Torsional Bu	ckling)	Kx		1.000					_			¥
Men	ber Length				L		10.04	t						0.38	
Lateral	-Torsional Bu	ckling Pos	sible				V								-
Effe	ctive Length	Factor (Re	straint Type)		kz		1.0					_ + +	N N	undhun	
- War	ping Length F	Factor (Res	traint Type)		kw		1.0							÷ .	
Comme	ent											_		z	
												_			
Set in	Set input for members No.:														
										0		×	- Q		

1.5 Effective Lengths - Members

Figure 2.14: Window 1.5 Effective lengths - Members

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

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

The Settings tree includes the following parameters:

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

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

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

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

Select Effective Length	Factor				×
Type of K Value					
O Theoretical					
Recommended					
Buckling About y-Axis		Buckling About z-Axis		Torsional Buckling	
⊖K _y = 0.65	X	⊖K _z = 0.65	× [⊖K _x = 0.65	X X
⊖K _Y = 0.8		⊖K _z = 0.8		○K _x = 0.8	
⊖K _y = 1.2	2 2	⊖K _z = 1.2		⊖K _x = 1.2	×
• K _y = 1.0	Z			● K _x = 1.0	X
○K _Y = 2.1	Z	○K _z = 2.1	y y	○K _x = 2.1	×
Ок _у = 2.0	ž	⊖K _z = 2.0	₽ <mark></mark>	⊖K _x = 2.0	×
$\bigcirc \underbrace{U}_{\text{Ser-defined}}_{K_{\text{Y}}} = \dots$	z	$\bigcirc U_{\underline{s}er} \text{-defined} \\ K_{z} = \dots$	ÿ	$\bigcirc User - \underline{d}efined \\ K_X = \dots$	
O Import from add-on (Eigenvalue Analysis	module RF-STABILITY ;)	O Import from add-on (Eigenvalue Analysis	module RF-STABILITY ;)	- Rotation fix	ked and translation fixed
RF-STABILITY-Case		RF-STABILITY-Case	:	D→ Rotation free	ee and translation fixed
CA1 - Stability anal	ysis 🗸 🗸	CA1 - Stability anal	ysis 🗸 🗸	D— Rotation for	ted and translation free
Buckling mode No.:	1 🔹 🏷	Buckling mode No.:	1 🔹 🏷	Rotation free	ee and translation free
Export effective length factor	Ky: 1.000 🗘	Export effective length factor	Kz: 1.000 🗘	Export torsional bucklin length factor	g K _x : 1.000 ‡
D					OK Cancel

Figure 2.15: Dialog box Select Effective Length Factor

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

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

Buckling Possible

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

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

۲

Buckling About Axis y / Buckling about Axis z

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

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



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

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

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

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

Torsional Buckling

Column H controls whether a torsional buckling design is to be performed. The effective length factors, K_x , and the torsional buckling lengths, $K_x L_x$, can be defined in columns I and J. The *x*-axis represents the center line of a member.

L.T.B.



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

Member axes

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

Effective Length Factor k_z



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

- $k_z = 1.0$ restrained against lateral displacement u_v on both member ends
- $k_z=0.7le\ \ restrained against displacement <math display="inline">u_y$ on both ends and restraint about z left
- $k_z = 0.7ri$ restrained against displacement u_y on both ends and restraint about z right
- $k_z=0.5~$ restrained against displacement u_y and restraint about z on both member ends
- $k_z = 2.0$ le restrained against displacement u_y and restraint about z left; right end free
- $k_z = 2.0ri$ restrained against displacement u_y and restraint about z right; left end free

Warping Length Factor $\mathbf{k}_{\mathbf{w}}$



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

- $k_w = 1.0$ restrained against rotation about x on both member ends; free to warp on both sides
- $k_{\rm w}=0.7le~$ restrained against rotation about x on both ends and warping restraint left
- $k_w^{}=0.7ri~$ restrained against rotation about x on both ends and warping restraint right
- ${f k}_w=0.5$ torsion and warping restraint on both member ends
- $k_w = 2.0le$ restrained against rotation about x and warping ω left; right end free
- $k_w = 2.0$ ri restrained against rotation about x and warping ω right; left end free



The abbreviations *le* and *ri* refer to the <u>le</u>ft and <u>right</u> sides. The abbreviation *le* always describes the support conditions at the member start.



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

Comment

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

Set Input for Members No.

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



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

© DLUBAL SOFTWARE 2018



2.6 Effective Lengths - Sets of Members

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

.6 Effectiv	ve Lengths -	Sets of M	lembers													
	Α	B	C	D	F	F	G	н		J	K			1		
Set	Buckling	Bu	ckling About	Axis y	Bu	ckling About	Axis z	Tors	ional Buck	ding	L.T.B.					
No.	Possible	Possible	Ky	KyL [ft]	Possible	Kz	KzL [ft]	Possible	Kx	K _x L [ft]	Possible			Comment		
2	2	V	1.000	65.31		1.000	65.31	V	1.000	65.31	Ø					
3		V	1.000	65.31		1.000	65.31	V	1.000	65.31	V					
													2	× ×	\$	۲
Settings -	Set of Memb	ers No. 2											W 21x55 Al	SC 14		
⊡ Set of	Members					Chord 2)					<u> </u>				
🖃 Men	nber 11															
- S	tart					3 - W 2	1x55 AISC 1	4								
- E	nd					2 - W 1	4x53 AISC 1	4						8.25		
Men	nber 12 - Cros	s-Section				2 - W 1	4x53 AISC 1	4						t		
- Men	nber 13 - Cros	s-Section				2 - W 1	4x53 AISC 1	4					+=		-	
Men	ber 14 - Cros	s-Section				2 - W 1	4x53 AISC 1	4					1.19	50 0.0	59	
Men	ber 15 - Cros	s-Section				2 - W 1	2 - W 14x53 AISC 14							·		
Men	nber 16												22			
- 9	tart					2 - W 1	4x53 AISC 1	4					20.3		œ	•
- E	nd					3 - W 2	1x55 AISC 1	4						0.20		-
Length	Des 11				L		65.31 It							0.00		
BUCKII	ng Possible	n Auis v Da	a sila la				<u> </u>								_	
	ng About Majo	n Axisy Fo	Issible		V		1.000						+			
Cite Cite	ctive Length I	actor			Ny I		1.000							÷		
Bucklin	a About Mine	v Avia z Pa	eeible		RyL .	L	03.31 1							2		
	al Buckling P	n nus z Fu loeeible	aaibic		_											
Effe	ctive Length I	Factor (for	Torsional Bu	ckling)	K _v		1 000					J				
Set input for sets No.:							All						0	2	• [*	[in]

Figure 2.17: Window 1.6 Effective Lengths - Sets of Members

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

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

Definition of Buckling Lengths

By Factor K and Buckling Length K*L
 By Factor K and Member Length L

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

2.7 Design Parameters

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

.7 Design	Parameters							
	A	В	C	D	E		F	
Member	Distance	Gross Area	Net Area	Shear Lag	Effective Are	ea		
No.	Lv [ft]	Ag [in ²]	An [in ²]	Factor U [-]	A _e [in ²]	A. [n ²]	Comm	ents
34	15.00	29.8	29.8	1.000 \pm	29	9.8		
35	5.89	15.6	15.6	1.000	15	5.6		
36	10.04	15.6	15.6	1.000	15	5.6		
37	16.73	15.6	15.6	1.000	15	5.6		
38	16.73	15.6	15.6	1.000	15	5.6		
39	10.04	15.6	15.6	1.000	18	5.6		
40	5.89	15.6	15.6	1.000	10	5.6		
41	19.65	29.8	29.8	1.000	23	9.8		
42	5.99	23.0	25.0	1.000	23	5.6		
Settings -	Member No.	1						W 21x101 AISC 14
Cross-	Section			1	- W 21x101 A	AISC 14		
Distan	ce acc. to G5			Lv	15.00	nt In 2		
Not An	Aled .			Ag	29.8	in ²		
Shear	Lag Eactor			U	1 000			12.25
Effecti	ve Area			Ae	29.8	in ²		
	ent	are No. 1						
set in	put for membe	EI 5 140.1			N V	All		0 1 4 C

Figure 2.18: Window 1.7 Design Parameters

Distance

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

Gross Area / Net Area

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

Shear Lag Factor

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

Effective Area

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

2.8 Nodal Supports - Sets of Members

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

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



Figure 2.19: Window 1.8 Nodal Supports - Set of Members



2

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

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

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

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

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



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

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



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

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



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

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



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

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

The buttons below the graphic have the following functions:

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

Table 2.2: Buttons for graphic window



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

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



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

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



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

Edit Warp Spring	×
Type of Warp Stiffening O End plate O Channel section Angle Image: Connecting column O Cantilevered portion Material Steel A992 Image: Column and Beam Cross-section for column: W 21x101 AISC 14 W 21x101 AISC 14 Image: Cross-section for beam: W 14x53 AISC 14 Beam height hm: 20.25 Image: [n]	Beam Column
	Resulting warp C_{ω} : 680.995 [kipft ³]
	OK Cancel

Figure 2.24: Dialog box Edit Warp Spring

2 Input Data

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

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

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

2

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

Warping torsion with seven degrees of freedom



1

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



[->)

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



1.8 Nodal Supports - Set of Members No. 2 - Main beam 2

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

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

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

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

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

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

2.9 Member Hinges - Sets of Members

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

.9 Memb	er Hinges	- Set of Memb	ers No. 2 - Chor	d 2							
	A	B	C	D	F	F		G			ï
Hinge	Member	Member	Shear Release	Moment	Release	Warp Release		<u> </u>		-11	ł
No.	No.	Side	Vv	M T [kipft/rad]	M ₇	Ma	Co	mment			l
1	13	End		2 500		20				-	l
2	15	Lind		2.000	Yee					-	l
3					No					-	l
4					Spring					-	l
5					oping					-	l
6										-	l
7										-	l
8										-	l
9											1
10											I
									× ×	۲	
Settings	- Member N	o. 13						W14x531.	AISC 14	_	1
⊟ Set of	Members				Chord 2						
⊞ Mer	mber 11										
- Mer	mber 12 - Cr	oss-Section			2 - W 14x53	AISC 14					
- Mer	mber 13 - Cr	oss-Section			2 - W 14x53	AISC 14					
- Mer	mber 14 - Cr	oss-Section			2 - W 14x53	AISC 14		· ·	* ^{8.00}		
- Mer	mber 15 - Cr	oss-Section			2 - W 14x53	AISC 14		→ →			
🛨 Mer	mber 16	******		******				∣∣₄⊢			
Memb	er with Hing	e at the End		No.		13		1.5	0 0.81		
Memb	er Side			Side		End					
Shear	Release in	y-Direction		Vy				33.88			
Torsio	nal Release	•		MT	2.	500 kipft/rad		-	-	У	
Mome	nt Release	about z-Axis		Mz					0.38		
Warpi	ng Release			Μω		V					
Comm	ent							∔	unnunnn		
									z		
Set in	out for rele	ase No.:								(in)	
						III 🗹		0	⇒ 🕂	X	
											н,

Figure 2.26: Window 1.9 Member Hinges



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



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

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



Details...

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

1.9 Member Hinges - Set of Members No. 2 - Main beam 2

	-									
	A	В	C	D	E	F	G	Н		J
Hinge	Member	Member		Release			Moment Release		Warp Release	
No.	No.	Side	Nx	Vy	Vz	Мт	My	Mz	Mω	Comment
1	16	Start					V	J	V	
2										
3										
4										
5										
6										
7										
8										
9										
10										
									4	. 🕸 🏷 👁

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

2.10 Parameters - Sets of Members

Details...

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

10 Parameters - Sets of Members												
	А	В	C		D							
Set	Initial	Local Bow Imperfe	ction									
No.	L/	L Manually	L [ft]		Comment							
1	150											
2	150											
3	150	<u> </u>	18.20									
4	150		15.47									
5	150											
					A							
Settings -	Set of Members N	lo. 3			W 16x36 AISC 14							
Set of	Members			Main beam 3								
Men	nber 41											
S	tart			3 - W 16x36 AISC 14								
E	nd			2 - W 12x35 AISC 14	. 7.00							
Men	nber 42 - Cross-Sec	tion		2 - W 12x35 AISC 14								
Men	nber 43 - Cross-Sec	tion		2 - W 12x35 AISC 14								
Men	nber 44 - Cross-Sec	tion		2 - W 12x35 AISC 14	<u> </u>							
Men	nber 45 - Cross-Sec	tion		2 - W 12x35 AISC 14								
- Men	nber 46											
- 5	tart			2 - W 12x35 AISC 14								
E lawal I	na 		1.7	3 - W 16x36 AISC 14	0.31							
	ocal Bow imperfec	tion	L/	150								
Det	ermine L manually	atial Importantian	1	10.20 0								
Comm	erence Lengin for i	niuai imperiection	L	10.20 IL								
	51 K											
Set in	put for sets No.:				[in]							
				All 🗸	0 🕹 🏹 🕰							

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

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





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

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

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



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

2.11 Serviceability Data

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

	A	В	C	D	E	F	G	Н
		Set of Members	Referen	ce Length	Direc-	Precamber		
0.	Reference to	No.	Manually	L [ft]	tion	w _{o,z} [in]	Beam Type	Comment
1	Set of Members	2		65.31	y, z	0.000	Beam	
2	Set of Members	5		65.31	y, z	0.000	Beam	
3	Member	82		16.40	y, z	0.000	Beam	
4	Member	81	V	4.54	z	0.000	Cantilever End Free	
5	Member	83	V	4.54	z	0.000	Cantilever End Free	
6	Member	15		10.04	y, z	0.000	Beam	
7	Member	16		5.89	y, z	0.000	Beam	
8	Member	25		19.69	y, z	0.000	Beam	
9	Member	26		19.69	y, z	0.000	Beam	
0	Member	92		25.62	u, v	0.000	Beam	
1								
2								
13								
14								
15								
16								
17								
18								
19								
20								
21								
22								
23								
24								
25								
26								
27								
28								
29								
30								
31								
32								
								3

Figure 2.29: Window 1.11 Serviceability Data



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

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

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

© DLUBAL SOFTWARE 2018

2 Input Data



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

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



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

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

3 Calculation

3.1 Detail Settings

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

The Details dialog box has the following tabs:

- Ultimate limit state
- Stability

Details...

- Serviceability
- Warping torsion
- General

3.1.1 Ultimate Limit State

Details - LRFD 2016	×
Ultimate Limit State Stability Serviceability Warping Torsion Gen	eral
Design Wall Thickness of HSS	Limit Values for Special Cases
Reduction of nominal thickness for HSS according to B4.2	Do not consider small moments if:
	Bending Mr,y / Me,y ≤ 0.010 荣
Shear Buckling	Mr,z/Mc,z ≤ 0.010 荣
Shear buckling warning for general cross-section	Do not consider small axial forces if:
Width-Thickness Ratio of Unsymmetrical Shapes	
Check width-thickness ratio of unsymmetrical	
shapes subjected to flexure	Do not consider small shear forces if:
- Unstiffened element 12.500 🖨	Shear V _{r,y} / V _{c,y} ≤ 0.010 €
- Stiffened element 42.000 🖨	V _{r,z} /V _{c,z} ≤ 0.010 🖨
	Limit shear stress for cross-sections with:
	Torsion $\tau_r / \tau_c \leq 0.010$
	OK Cancel
	Caller

Figure 3.1: Dialog box Details, tab Ultimate limit state

Design Wall Thickness of HSS

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

Shear Buckling

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

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

Width-Thickness Ratio of Unsymmetrical Shapes

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

Limit Values for Special Cases

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



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

3.1.2 Stability

De	tails - LRFD 2016				×	
	Ultimate Limit State	Stability	Serviceability	Warping Torsion	General	
	Determination of Ela	istic Critica	I Moment for LTE	3		
	Load application of loads:	positive tra	ansverse			
	On cross-section (e.g. top flange)	n edge dire , destabiliz	cted to shear ce ing effect)	enter		
	• In shear center					
	On cross-section (e.g. bottom flam	n edge dire nge, stabili	cted from shear zing effect)	center		
	Definition of Bucklin	g Lengths				
	By Factor K and	Buckling Le	ength K*L			
	O By Factor K and	Member Le	ength L			
(2 0.00]		OK Cancel	
_			_			

Figure 3.2: Dialog box Details, tab Stability



Determination of Elastic Critical Moment for LTB

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

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

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

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

Definition of Buckling Lengths

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

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

Details - LRFD 2016	×
Ultimate Limit State Stability Serviceability Warping Torsion General	
Deformation Relative To	
● Shifted member ends / set of members ends	
O Undeformed system	
Serviceability Limits (Deflections)	
Cantilever	
Direction of Precamber	
Consider precamber in	
0 y/u	
	OK Cancel

3.1.3 Serviceability

Figure 3.3: Dialog box Details, tab Serviceability

Deformation Relative To

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

Serviceability Limits (Deflections)

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

Direction of Precamber

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

3.1.4 Warping Torsion

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

Details - LRFD 2016	×
Ultimate Limit State Stability Serviceability Warping Torsion General	
RF-STEEL Warping Torsion	
of freedom (Available for sets of members only)	
Method of Warping Analysis	
O Second-order analysis	
Load Application	
	OK Cancel

Figure 3.4: Dialog box Details, tab Warping Torsion

RF-/STEEL Warping Torsion

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

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

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

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



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

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



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

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

Method of Warping Analysis

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

Load Application

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

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

3.1.5 General

letails - LRFD 2016	×
Ultimate Limit State Stability Serviceability Warping Torsion Gen	eral
Cross-Section Optimization	Display Result Windows
Max allowable design ratio: 1.000 🖨	2.1 Design by Load Case 2.2 Design by Cross-Section 2.3 Design by Set of Members
Check of Maximum Effective Slenderness Ratio	2.4 Design by Member
Members with KL / r	✓ 2.5 Design by x-Location
Compression / flexure: 200 Display slenderness check for all member types	 ✓ 3.1 Governing Internal Forces by Member ✓ 3.2 Governing Internal Forces by Set of Members ☐ 3.3 Member Slendernesses
	 S. Member Sendernesses I. Parts List by Member I. Parts List by Set of Members Only for members / sets to be designed Of all members / sets of members
	OK Cancel

Figure 3.5: Dialog box Details, tab General

Cross-Section Optimization

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

Check of Maximum Effective Slenderness Ratio

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

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

Display Result Windows

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

Window 3.3 Member Slendernesses is deactivated by default.

3.2 Starting Calculation



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

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

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

To Calculate						×
Load Cases /	Combinations / Module Cases Result Tables					
Not Calculate	culated			Selected for 0	Calculation	
No.	Description	^		No.	Description	^
2.3 CO44	1.2D + 0.5S + L + W1 + Imp1			CA1	RF-STEEL AISC - Design of steel members according to AISC (I	
2.3 CO45	1.2D + 0.5S + L + W1 + Imp2					
2.3 CO46	1.2D + 0.5S + L + W2 + Imp1					
2.3 CO47	1.2D + 0.5S + L + W2 + Imp2					
2.3 CO48	1.2D + L + W1 + Imp1					
2.3 CO49	1.2D + L + W1 + Imp2					
2.3 CO50	1.2D + L + W2 + Imp1					
2.3 CO51	1.2D + L + W2 + Imp2		>			
2.3 CO52	1.2D + 0.5S + W1 + Imp1		>>			
2.3 CO53	1.2D + 0.5S + W1 + Imp2					
2.3 CO54	1.2D + 0.5S + W2 + Imp1					
2.3 CO55	1.2D + 0.5S + W2 + Imp2		4			
2.3 CO56	1.2D + W1 + Imp1		44			
2.3 CO57	1.2D + W1 + Imp2					
2.3 CO58	1.2D + W2 + Imp1					
2.3 CO59	1.2D + W2 + Imp2	- 10				
2.3 CO60	0.9D + W1 + Imp1					
2.3 CO61	0.9D + W1 + Imp2					
2.3 CO62	0.9D + W2 + Imp1					
2.3 CO63	0.9D + W2 + Imp2					
2.4 RC1	ASD -					
2.3 RC2	LRFD -					
CA1	RF-STABILITY - Stability analysis	~				v
All	~	2				
9 P					OK Cance	el l

Figure 3.6: Dialog box To Calculate



۲

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

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

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

Ta <u>b</u> le	<u>O</u> ptions	<u>A</u> dd-on Modules <u>W</u> indow <u>H</u> elp
-	II 🗳	RF-STEEL AISC CA1 - Design of steel members acc 🝸 < > 🔗 🎬 👰 💯 🔤 ன 📓 🗱 👹
D - 1	📬 - 🗊	🔐 - 🗊 - 🌯 🏂 🎘 🖏 🦓 - 🤅 🧙 🔍 🍳 🕼 🗗 🕅 🖏 🐜 🗤 👘 /// 🖘

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

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

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

RF-STEEL AISC - [Hall-Boulder]														×
File Edit Settings Help														
CA1 - Design of steel members - ~	2.1 Desigr	n by Load Case												
Input Data		A	В	C	D	E	1			F				G
- General Data	Load-	Î.	Member	Location	Design									
Materials	ing	Description	No.	x [ft]	Ratio				Desi	gn According to Fo	omula			DS
- Cross-Sections		Ultimate Limit State Design												
 Intermediate Lateral Restraints 	CO34	1.2D + 0.5S + 1.6L + Imp1	19	4.71	0.91	≤ 1	350) Set of Men	nbers - !	Stability anal	ysis of doubly and	singly sym	metric membe	ers acc. to Ch	i ULS
 Effective Lengths - Members 	CO35	1.2D + 0.5S + 1.6L + Imp2	19	5.89	0.91	≤1	350) Set of Men	nbers - S	Stability anal	ysis of doubly and	singly sym	metric membe	ers acc. to Ch	ULS
 Effective Lengths - Sets of Mer 	CO40	1.2D + 1.6S + 0.5W1 + Imp1	19	3.92	0.74	≤1	350) Set of Men	nbers - !	Stability anal	ysis of doubly and	singly sym	metric membe	ers acc. to Ch	ULS
— Design Parameters	CO41	1.2D + 1.6S + 0.5W1 + Imp2	19	3.92	0.74	≤1	350) Set of Men	nbers - !	Stability anal	ysis of doubly and	singly sym	metric membe	ers acc. to Ch	i ULS
Nodal Supports														
···· Set of members No. 2 - Cho		Serviceability Limit State Desig	n											
Set of members No. 3 - Cho	C07	D + 0.75S + 0.75L + Imp1	37	8.36	0.16	≤1	401) Chapter L	- Desigr	for Service	ability - Deflection i	in z-directio	on (Beam)		
Member Hinges	CO8	D + 0.75S + 0.75L + Imp2	38	8.36	0.16	≤1	401) Chapter L	- Design	for Service	ability - Deflection i	in z-directio	on (Beam)		
Set of members No. 2 - Cho														
Set of members No. 3 - Cho				_										
 Serviceability Data 				Max:	0.91	≤ 1	Θ 🔒	- 7	*	<u>i</u>	> 1,0	$\sim \gamma$	😂 📲 🚺	3 📀
Results														
 Design by Load Case 	Details -	Member 19 - x: 4.71 ft - CO34									3 - 2: W	21×44 AIS0	C 14 - W 14×	43 AIS
 Design by Cross-Section 		al Properties - Steel A992 ANS	I/AISC 36	0-16:2016-6										
 Design by Set of Members 	Cross-	Section Properties - IC-PAR 15	5.025/7.7/	0.325/0.4875/	0.8375/0/0/0									
 Design by Member 	Design	n Internal Forces												
 Design by x-Location 	E Cross-	Section Type										7.70		
— Governing Internal Forces by №	Design	n Ratio											49	
- Governing Internal Forces by S	Ben	iding about the major axis										t-'	0	ŧ
— Parts List by Member	Rec	quired Flexural Strength			Mr.y		21.017	kipft				Y		
Parts List by Set of Members	- Amp	olifier			0.or		1.222							
	- Elas	stic Critical Moment for Lateral-T	orsional Bu	uckling	Mcr.y		23.115	kipft						
	Des	ign ratio of flexure about y-axis			ηь.у		0.91		≤1	(B3-1)	15.0			
	Des	ign Ratio for Combined Forces										0.33		
	Des	ign ratio of flexure about y-axis			ηb.y		0.91			acc. to Chapt				
	Des	ign ratio of flexure about z-axis			ηb,z		0.00			acc. to Chapt				
	- Des	ign Ratio			η		0.91		≤1	(H1-1a)	· ·			
												- ÷		
													5	
														[in]
< >>											0		÷⇒ ↓	C(
0 5 5	Calculati	on Details			Gr	aphic						OK		ancel
Design Ratio												- OK		

Figure 4.1: Result window with design results and details

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

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



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



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

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





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

Δ

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



Figure 4.2: Window 2.1 Design by Load Case

Description

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

Member No.

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

Location **x**



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

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



Design Ratio



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

The lengths of colored bars represent the respective design ratios.

Design According to Formula

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

DS

The last column provides information on the respective design situations.

4.2 Design by Cross-Section

2.2 Desigr	n by Cross	-Section													
	A	В	С	D	E				F	:					
Section	Member	Location	Load-	Design											
No.	No.	x [ft]	ing	Ratio					Design Accord	ling to Formula					
2	W 14x43	AISC 14													
	47	2.51	CO40	0.01	≤1	100) Negligible	internal forces								
	40	5.89	CO34	0.11	≤1	111) Chapter F	- Yielding - Ben	ding about	y-axis acc. to F2	2 - F12					
	40	5.89	CO34	0.11	≤1	113) Chapter F	 Lateral-torsion 	al buckling	- acc. to F2 - F7	7					
	35	4.71	CO34	0.00	≤1	116) Chapter F	 Lateral-torsion 	al buckling	does not apply	-acc. to F2 - F1	1				
	11	0.00	CO34	0.00	≤1	122) Chapter F	 Flange local b 	uckling doe	es not apply - ac	c. to F2 - F5					
	24	5.89	CO34	0.03	≤1	160) Chapter G	- Nominal shea	r strength ir	n z-axis - acc. to	G2 (G3,G4) - Ur	nstiffened	cross-s	section		
	19	4.71	CO34	0.91	≤1	350) Set of Mer	mbers - Stability	analysis of	doubly and sing	ly symmetric mer	nbers acc	c. to Ch	apters E, F a	and H	
	35	0.00	C07	0.00	≤1	400) Chapter L	 Design for Ser 	viceability	 Negligible defle 	ctions					
	37	8.36	C07	0.16	≤1	401) Chapter L	 Design for Ser 	viceability	 Deflection in z- 	direction (Beam)					v
			Max:	0.91	≤ 1	•		I.	18	*	> 1,0	~	7 🗳	🐴 🏷	۲
Details -	Member 40 al Propertie Section Pr) - x: 5.89 ft es - Steel A9 operties - W	- CO34 92 ANSI/A / 21x44 Al	NSC 360-16:2016- SC 14	-6						2 - 3: \	/V14×4	3 AISC 14	- VV 21×44	I AIS
Design	n Internal F	orces													
E Cross-	Section Ty	pe											6.50		
Design	n Ratio														
Rec	quired Flexi	ural Strength				Mr.y	37.90	kipft				t.±	-	-	
Bra	ced Length	1				Lb	5.89	h	$> L_p and \le L_r$			1.13	8 V	.69	
Lim	iting Lengt	1				Lp	4.46	h		(F2-5)		1 '			
- Lim	ting Lengt	1				Lr	13.52	h		(F2-6)					
Mo	dulus of Ela	asticity				E	28998.60	ksi			20.6			80 (r)	•
Rad	dius of Gyra	ation				rz	1.26	in							1
Yie	d Stress					Fy	50.00	ksi					0.38		
Effe	ective Radi	us of Gyratio	n			۲ts	1.60	in		(F2-7)					
- Tor	sional Con	stant				J	0.8	in ⁴				+		-	
- Cor	nstant					c	1.000			(F2-8a)			÷ .		
Ela:	stic Sectior	n Modulus				Sy	81.6	in ³					z		
Mo	dification F	actor				Сь	1.000								
- Pla	stic Bendin	g Moment				M _{pl,y}	397.50	kipft							
- Nor	minal Flexu	ral Strength				Mn	372.35	kipft		(F2-2)					
- Res	sistance Fa	ctor for Flexu	ire			Φb	0.900			(F1)					fin1
- Des	sign Flexura	al Strength				Фь*М n	335.12	kipft					Г	X at	
- Des	sign Ratio					ηь	0.11		≤1	(B3-1)	0			≏ ,	Q

Figure 4.3: Window 2.2 Design by Cross-Section

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

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

4.3 Design by Set of Members

2.3 Desig	n by Set o	f Members											
	A	B	С	D	F					F			
Set	Member	Location	Load-	Design									
No.	No.	x [ft]	ing	Ratio					Design Acc	cording to Formu	ıla		
2	Chord 2 (Member No.	11-16)										
	15	2.51	CO40	0.01	≤1	100) Negligible int	ternal forces						
	16	5.89	CO34	0.08	≤1	111) Chapter F - 1	rielding - Bendi	ng about	y-axis acc. t	o F2 - F12			
	11	0.00	CO34	0.00	≤1	122) Chapter F - F	lange local bu	ckling do	es not apply	- acc. to F2 - F5			
	16	5.89	CO34	0.02	≤1	160) Chapter G - I	Nominal shear	strength i	n z-axis - acc	c. to G2 (G3,G4)	- Uns	tiffened	cross-section
	14	2.79	CO34	0.36	≤1	350) Set of Memb	ers - Stability a	nalysis of	doubly and	singly symmetric	memt	bers acc	c. to Chapters E, F and H
3	Chord 3 (Member No.	19-24)										
	24	5.89	CO34	0.10	≤1	111) Chapter F - 1	rielding - Bendi	ng about	y-axis acc. t	o F2 - F12			
	19	0.00	CO34	0.00	≤1	122) Chapter F - F	Plange local bu	ckling do	es not apply	- acc. to F2 - F5			~ ~
			Max:	0.91	≤1	•		H	1 😜	9	<u></u>	> 1,0	~ 7 🗳 🛃 🇞 📀
Details -	Member 10	6 - x: 5.89 ft	- CO34									2 - 3: V	W 14x43 AISC 14 - W 21x44 AIS
⊡ Cross	-Section Ty	/pe									~		
- Fle	xure in Flar	ige											
H	alf of Full F	lange Width				b	4.00	in		Tab. B4.1			
T	hickness					tr	0.50	in		Tab. B4.1			e 50
- b	/t-Limit for (Compact Flan	ige			λp,f	9.152			Tab. B4.1			+ 0.50
- b.	/t-Limit for N	Voncompact	Flange			λr,f	24.083			Tab. B4.1			+ +
- b	/t-Ratio					(b/t) f	8.000		≦λp,f				F 8 0.69
T	ype of Flan	ge				Type f	Compact			Tab. B4.1			
- Fle	xure in We	b											
L	ength					h	10.87	in		Tab. B4.1		50.6	**
T	hickness					tw	0.31	in		Tab. B4.1			Y Y
h	/tw-Limit for	Compact W	eb			λ _{p,w}	90.553			Tab. B4.1			0.38
h	/tw-Limit for	Noncompac	t Web			λr,w	137.274			Tab. B4.1			
h	/t _w -Ratio					h/t _w	34.800		≤λ _{p,w}			· ·	
T	ype of Web)				Typew	Compact			Tab. B4.1			
T	ype of Cros	s-Section in				Flexure	Compact						z
Desig	in Ratio												
Re	quired Flex	ural Strength				M _{r,y}	29.80	kipft					
- Yie	eld Stress					Fy	50.00	ksi					
- Pla	Plastic Section Modulus					Zy	95.4	in ³					[in]
- Pla	astic Bendin	g Moment				M _{pl,y}	397.50	kipft					X + X
- No	minal Flexu	ral Strength				Mn,y	397.50	kipft		(F2-1)	¥	0	🛥 🖵 🕰

Figure 4.4: Window 2.3 Design by Set of Members

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

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

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

4

-	-,																
	A	B	С	D					E								
Member	Location	Load-	Design														
No.	x [ft]	ing	Ratio					Desig	gn Accordi	ng to Formula							
76	Cross-section	n No. 7 - S	qr HSS 4x4x0.25	O I AI	SC 14												
	8.20	CO35	0.90	≤1	111) Chapter	F - Yielding - B	ending about y	-axis acc.	to F2 - F12	2							
	8.20	CO35	0.90	≤1	113) Chapter	F - Lateral-torsi	onal buckling -	acc. to F2	2 - F7								
	2.73	CO34	0.00	≤1	126) Chapter	F - Local buck	ling of HSS sec	tion does	not apply	-acc.to F7							
	16.40	CO35	0.10	≤1	160) Chapter	G - Nominal sh	ear strength in	z-axis - ac	c. to G2 (0	33,G4) - Unstiff	ened c	ross-sec	tion				
	8.20	CO35	0.09	≤1	301) Chapter	E - Flexural bu	ckling about y-i	axis acc. to	o E3								
	8.20	CO35	0.09	≤1	311) Chapter	E - Flexural bu	ckling about z-i	axis acc. to	o E3								
	8.20	CO35	0.03	≤1	321) Chapter	E - Torsional o	r flexural-torsior	nal bucklin	g acc. to E	4							
	8.20	CO35	0.94	≤1	332) Chapter	·H - Single axis	/or biaxial/ flex	ure with a	xial compr	ession force - a	cc.to	H1.1					
	8.20	CO35	0.94	≤1	334) Chapter	H - Compressio	on force with sir	ngle/major	axis bend	ing - acc. to H	1.3						
		Max:	0.94	9			I.	₽÷	2	E.	> 1,0	\sim	7	¥ 🐴	\$	0	
etails - I	Member 76 -	x: 8.20 ft - (CO35								_	7 - Sqr	HSS 4:	×4×0.25	0 AISC 1	14	
— - Th	Thickness					tr	0.23	in		Tab. B4.1	^						
b/t	-Limit for Con	pact Flang	e			λp,f	26.973			Tab. B4.1	_						
b/t	-Limit for Nor	compact H	ange			λr,f	33./16			Tab. B4.1							
b/t	-Ratio					(b/t)f	13.053		≦λp,f					4.00			
- • Ty	pe of Flange					Type f	Compact			Tab. B4.1			<u>†</u>	4.00		+	
- Hex	ure in Web					1.						+-	-				
- Le	ngth					h	3.03	in		Tab. B4.1	_					1	
- • Ih	ickness					t	0.23	in		Tab. B4.1						0.25	-
h/t	-Limit for Con	pact Web				λp,w	58.281			Tab. B4.1	_						
h/t	-Limit for Nor	compact W	/eb			λ _{r,w}	137.274			Tab. B4.1		4.0		+		- •	ŀ,
h/t	-Ratio					h/t	13.053		≤λp,w		-			- E			1
Ty	pe of Web					Туре	Compact			Tab. B4.1	_			1.1			
- Ty	pe of Cross-S	ection in				Flexure	Compact									9	
] Design	n Ratio											-					
- Req	Required Flexural Strength					M _{r.y}	15.78	kipft						÷			
 Yield 	Yield Stress					Fy	50.00	ksi						z			
Plas	Plastic Section Modulus					Zy	4.7	in ³									
Plas	Plastic Bending Moment					M _{pl,y}	19.54	kipft									
Non	Nominal Flexural Strength					M _{n.y}	19.54	kipft		(F2-1)							
Res	Resistance Factor for Flexure					Φb	0.900			(F1)							
Des	Design Flexural Strength					Φb [*] Mn,y	17.59	kipft									1
Des	ign Ratio					η 6	0.90		≤1	(B3-1)	v	0			ŝ	5	¢

Figure 4.5: Window 2.4 Design by Member

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

4.5 Design by x-Location

.5 Desigr	n by x-Locati	ion										
	A	В	С	D					E			<u>^</u>
Member	Location	Load-	Design									
No.	x [ft]	ing	Ratio					Design A	ccording to	Formula		
57	Cross-section	n No. 5 - V	VT 7x13 AISC 14	1								
	0.00	CO34	0.21	≤1	303) Chapter E -	Flexural bucklin	g about y-axis	acc. to E7	7 - Slender e	elements		
	0.00	CO34	0.82	≤1	313) Chapter E	Flexural bucklin	g about z-axis	acc. to E7	7 - Slender e	elements		
	0.00	CO34	0.84	≤1	323) Chapter E ·	 Torsional buckl 	ing acc. to E4	and E7 - S	ölender elen	nents		
	0.00	CO34	0.21	≤1	303) Chapter E ·	Flexural bucklin	g about y-axis	acc. to E7	7 - Slender e	elements		
	0.00	CO34	0.82	≤1	313) Chapter E ·	Flexural bucklin	g about z-axis	acc. to E7	7 - Slender e	elements		
	0.00	CO34	0.84	≤1	323) Chapter E -	Torsional buckl	ing acc. to E4	and E7 - S	blender elen	nents		
	2.62	CO41	0.04	≤1	111) Chapter F -	Yielding - Bendi	ng about y-axi	s acc. to F.	2 - F12			
	2.62	CO34	0.05	≤1	112) Chapter F -	Yielding - Bendi	ng about z-axi	s acc. to F.	2 - F12			
	2.62	CO41	0.06	≤1	11/) Chapter F -	Lateral-torsional	buckling - ac	c. to F9				¥
		Max:	0.94	≤1	۲			9) U	3	> 1,0	~ 7 👺 💐 🗞 👁
El Halden El Cross- El Cross- El Desigr Rec Effe Yiel Moo Effe Sler Elas Corti Res Non Des	Section Proportional Section Type In Ratio Julied Compre- citive Member d Stress dulus of Elasti citive Area Indemess Rati stic Ontical Bu cal Stress ign Compress ign Compress ign Ratio	extended and a series of the s	rssion th			Pr Leff.y Fy E KyL/fy Fe.y For.y Φo Pn.y Φo* Pn.y	8.34 26.25 50.00 28998.60 3.8 1148.561 11.296 11.37 0.900 43.78 39.40 0.21	kip ft ksi in 2 ksi ksi kip kip	> 4.71\Ē ≤ 1	(E2) (E3-4) (E3-2 or E3-3) (E1) (E7-1) (E7-1)	1 200	<u>5.00</u> <u>7</u> <u>7</u> <u>7</u> <u>7</u> <u>7</u> <u>7</u> <u>7</u> <u>7</u>
											0	(in)

Figure 4.6: Window 2.5 Design by x-Location

4



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

4.6 Governing Internal Forces by Member

	A	В	C	D	E	F	G	н	
lember	Location	Load-		Forces [kip]		N	loments [kipft]		
No.	x [ft]	ing	P	V _{y/u}	V _{z/v}	Ta	M _{y/u}	M _{z/v}	Design According to Formula
11	Cross-section	No. 3 - W	21x44 AISC	14 2 - W 14	43 AISC 14				
	0.00	CO35	-4.43	0.00	2.88	0.00	-25.55	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F
	0.00	CO34	-4.42	0.00	2.87	0.00	+25.26	0.00	122) Chapter F - Flange local buckling does not apply - acc. to
	0.00	CO35	-4.43	0.00	2.88	0.00	-25.55	0.00	160) Chapter G - Nominal shear strength in z-axis - acc. to G2
	4.71	CO34	-4.33	0.00	2.23	0.00	+13.24	0.01	350) Set of Members - Stability analysis of doubly and singly sy
12	Cross-section	No 2-W	14x431AISC	14					
	7.53	CO34	-4.16	0.00	1.27	0.00	2.30	0.03	100) Negligible internal forces
	0.00	CO35	-4.27	0.00	2.19	0.00	10.95	0.01	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F
	0.00	CO34	-4.26	0.00	2.18	0.00	+10.70	0.01	122) Chapter F - Flange local buckling does not apply - acc. to
_	0.00	CO35	-4.27	0.00	2.19	0.00	+10.95	0.01	160) Chapter G - Nominal shear strength in z-axis - acc. to G2
	0.00	CO34	-4.26	0.00	2.18	0.00	10.70	0.01	350) Set of Members - Stability analysis of doubly and singly sy
13	Cross-section	No. 2 . W	1/4/21 4190	14					
	16 73	CO41	-4 73	0.00	-0.93	0.00	1.67	0.01	100) Negligible internal forces
	8.36	CO34	-6.09	0.00	-0.14	0.00	8.18	0.02	111) Chapter F - Yielding - Bending about v-axis acc. to F2 - F
	0.00	CO34	-6.20	0.00	0.88	0.00	5.10	0.04	122) Chapter F - Flange local buckling does not apply - acc. to
	0.00	CO34	-6.20	0.00	0.88	0.00	5.10	0.04	350) Set of Members - Stability analysis of doubly and singly sy
14	Cross-section	No 2.W	14×4314150	14					
	0.00	CO41	-4 74	0.00	0.87	0.00	1.67	0.01	100) Negligible internal forces
	8.36	C035	-6.10	0.00	0.13	0.00	8.07	0.02	111) Chapter F - Yielding - Bending about v-axis acc. to F2 - F
	0.00	CO34	-5.98	0.00	1 14	0.00	2.74	0.02	122) Chapter F - Flange local buckling does not apply - acc. to
	16.73	CO34	-6.20	0.00	-0.90	0.00	4.75	0.03	350) Set of Members - Stability analysis of doubly and singly sy
15	.								
15	Cross-section	No. 2 - W	14x43 AISC	14	1.00	0.00	0.1	0.02	100) No -li -ible internel ferrere
	5.02	C040	-3.30	0.00	-1.22	0.00	-1.68	0.02	100) Negligible Internal forces
	10.04	CO34	-4.26	0.00	-2.20	0.00	11.26	0.01	12) Chapter F - Heiging - beriding about y-axis acc. to F2 - F
_	0.00	C034	-4.13	0.00	-0.99	0.00	4./5	0.03	122) Chapter C. Newing chapter that the action of apply - acc. to
	10.04	0034	-4.26	0.00	-2.20	0.00	11.26	0.01	250) Cet of Members - Stability analysis of doubly and singly an
	0.00	0034	-4.13	0.00	-0.99	0.00	4.75	0.03	500/ Set of members - Stability analysis of doubly and singly sy
16	Cross-section	No. 2 - W	14x43 AISC	14 3 - W 21:	44 AISC 14				
	5.89	CO34	-4.42	0.00	-2.89	0.00	+25.93	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F

Figure 4.7: Window 3.1 Governing Internal Forces by Member

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

Location **x**

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

Loading

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

Forces / Moments

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

Design According to Formula

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

4.7 Governing Internal Forces by Set of Members

				(2		-			
C-1	A	B	C	D D	E	- F	G	н	
No	Location	Load-		Forces [kip]		- '	vioments [kipit]		Desire Assertion to French
	x [tt]	ing	P	Vy	٧z	la	IMA	MZ	Design According to Formula
	Chord 2 (Men	iber No. 11	-16)		4.07	0.00	0.00	0.00	100 N P 11 1 1
	7.53	CO34	-4.1	i 0.00	1.27	0.00	2.30	0.03	100) Negligible internal forces
	5.89	CO34	-4.4	2 0.00	-2.89	0.00	-25.93	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	0.00	CO34	-4.4	2 0.00	2.87	0.00	-25.26	0.00	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-4.4	2 0.00	-2.89	0.00	-25.93	0.00	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	16.73	CO34	-6.2	0.00	-0.90	0.00	4.75	0.03	350) Set of Members - Stability analysis of doubly and singly symme
3	Chord 3 (Mem	iber No. 19)-24)						
	5.89	CO34	-2.9	0.02	-3.63	0.00	-37.05	-0.26	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	0.00	CO34	-2.9	0.00	3.61	0.00	-36.54	0.00	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-2.9	0.02	-3.63	0.00	-37.05	-0.26	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	0.00	CO34	-2.9	0.00	3.61	0.00	-36.54	0.00	350) Set of Members - Stability analysis of doubly and singly symme
4	Chord 4 (Mem	nber No. 27	7-32)						
	5.89	CO34	-2.9	5 0.00	-3.61	0.00	-37.87	0.00	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	0.00	CO34	-2.9	0.00	3.60	0.00	-37.48	0.00	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-2.9	0.00	-3.61	0.00	-37.87	0.00	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,(
	1.96	CO35	-2.9	2 0.00	3.34	0.00	-30.86	-0.01	350) Set of Members - Stability analysis of doubly and singly symme
6	Chord 6 (Mem	iber No. 43	3-48)						
	5.89	CO34	-6.1	i 0.00	-4.33	-0.01	-51.26	-0.01	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12
	5.89	CO34	-6.1	0.00	-4.33	-0.01	-51.26	-0.01	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F7
	4.71	CO34	-6.0	0.00	3.67	0.00	-31.54	-0.03	116) Chapter F - Lateral-torsional buckling does not apply - acc. to
	0.00	CO34	-6.1	2 0.00	4.30	0.01	-50.33	-0.01	122) Chapter F - Flange local buckling does not apply - acc. to F2 -
	5.89	CO34	-6.1	i 0.00	-4.33	-0.01	-51.26	-0.01	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,0
	16.73	CO34	-7.8	0.01	-2.40	0.01	2.93	-0.04	301) Chapter E - Flexural buckling about y-axis acc. to E3
	5.89	CO35	-6.0	2 0.00	3.53	0.00	-27.73	-0.03	303) Chapter E - Flexural buckling about y-axis acc. to E7 - Slender
	16.73	CO34	-7.8	0.01	-2.40	0.01	2.93	-0.04	311) Chapter E - Flexural buckling about z-axis acc. to E3
	0.00	CO35	-6.1	0.00	4.32	0.01	-50.84	-0.01	313) Chapter E - Flexural buckling about z-axis acc. to E7 - Slender
	16.73	CO34	-7.8	3 0.01	-2.40	0.01	2.93	-0.04	321) Chapter E - Torsional or flexural-torsional buckling acc. to E4
	0.00	CO35	-6.1	0.00	4.32	0.01	-50.84	-0.01	323) Chapter E - Torsional buckling acc. to E4 and E7 - Slender ele
	5.89	CO34	-6.1	0.00	-4.33	-0.01	-51.26	-0.01	332) Chapter H - Single axis /or biaxial/ flexure with axial compress
	5.89	CO34	-6.1	0.00	-4.33	-0.01	-51.26	-0.01	334) Chapter H - Compression force with single/major axis bending
	5.89	CO35	-6.1	0.00	-4.32	-0.01	-50.84	-0.01	350) Set of Members - Stability analysis of doubly and singly symme
									E

3.2 Governing Internal Forces by Set of Members

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

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





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

	А	B	С	D	E	F	G	H				
Member		Length		Major Axis y/u			Minor Axis z/v					
No.	Under Stress	L [ft]	k y/u [-]	i _{y/u} [in]	λ _{y/u} [-]	k _{z/v} [-]	i _{z/v} [in]	λ.z/v [-]				
34	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868				
35	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979				
36	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595				
37	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991				
38	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991				
39	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595				
40	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979				
41	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868				
42	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868				
43	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979				
44	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595				
45	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991				
46	Compression / Flexure	16.73	1.000	5.83	34.444	1.000	1.89	105.991				
47	Compression / Flexure	10.04	1.000	5.83	20.667	1.000	1.89	63.595				
48	Compression / Flexure	5.89	1.000	5.83	12.120	1.000	1.26	55.979				
49	Compression / Flexure	19.69	1.000	6.98	33.859	1.000	2.46	95.868				
56	Compression / Flexure	22.97	1.000	2.04	134.866	1.000	1.54	178.507				
57	Compression / Flexure	26.25	1.000	2.04	154.133	1.000	1.54	204.008				
58	Compression / Flexure	22.97	1.000	2.04	134.866	1.000	1.54	178.507				
63	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391				
69	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391				
75	75 Compression / Flexure 16.40 1.000 1.52 129.391 1.000 1.52 129.391											
76	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391				
81	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391				
87	Compression / Flexure	16.40	1.000	1.52	129.391	1.000	1.52	129.391				
				Members	s with compress	sion / flexure:						
				Max Ky/ul / radio	154,133	< 200						

Figure 4.9: Window 3.3 Member Slendernesses

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

Details...

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

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



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

4.9 Parts List by Member

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

1 Parts l	.ist by Member								
	A	B	С	D	E	F	G	Н	
Part	Cross-Section	Number of	Length	Total Length	Surface Area	Volume	Unit Weight	Weight	Total Weight
No.	Description	Members	[ft]	[ft]	[ft 2]	[ft 3]	[b/ft]	[b]	[kip]
1	1 - W 16x67 AISC 14	4	19.69	78,74	492.11	10.72	66.70	1313.02	5.25
2	2 - W 14x43 AISC 14 3 - W 21x44 AISC	4	5.89	23.55	126.26	2.09	43.56	256.42	1.03
3	2 - W 14x43 AISC 14	4	10.04	40.15	202.83	3.51	42.88	430.40	1.72
4	2 - W 14x43 AISC 14	4	16.73	66.92	338.05	5.86	42.88	717.33	2.87
5	5 - WT 7x19 AISC 14	2	22.97	45.93	102.80	1.78	18.99	436.11	0.87
6	5 - WT 7x19 AISC 14	1	26.25	26.25	58.74	1.02	18.99	498.41	0.50
7	7 - Sgr HSS 4x4x0.250 AISC 14	6	16.40	98.43	124.19	2.30	12.21	200.37	1.20
8	10 - L 4x3x5/16 AISC 14	4	25.62	102.50	116.67	1.49	7.11	182.25	0.73
9	8 - RB 7/8 AISC	8	23.43	187.44	42.94	0.78	2.05	47.95	0.38
Sum		37		669.89	1604.59	29.55		-	14.55
									 3

Figure 4.10: Window 4.1 Parts List by Member

By default, this list contains only the designed members. If you need a parts list for all members of the model, select the corresponding option in the *Details* dialog box (see Figure 3.5, page 34).

Part No.

Details...

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

Cross-Section Description

This column lists the cross-section numbers and descriptions.

Number of Members

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

Length

This column shows the respective length of an individual member.

Total Length

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

Surface Area



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



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

Unit Weight

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

Weight

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

Total Weight

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

Sum

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

4.10 Parts List by Set of Members

Set of Members Description Number of Sets Length [R] Total Length [R] Surface Area [R3] Volume [R43] Unit Weight [B,h] Weight [B,h] Total Weigh [B,h] Total Weigh [B,h] Weight [B,h] Total Weigh [B,h] Weight [B,h] Total Weight [B,h] Total Weight [B,h] Weight [B,h] Weight [B,h] Total Weight [B,h] Weight [B,h] Total Weight [B,h] Weight [B,h] Weight [B,h] Weight [B,h] Weight [B,h] Weight [B,h] Weight [B,h] Total Weight [B,h] Weight [B,h] Total Weight [B,h] Weight [B,h] Weight [B,h] Total Weight [B,h] Weight [B,h]	Set of Members	Number	-						
O. Description of Sets [ft] [ft] [ft]2] [ft]3] [b.ft] [lb] [lb] <th>Description</th> <th>number</th> <th>Length</th> <th>Total Length</th> <th>Surface Area</th> <th>Volume</th> <th>Unit Weight</th> <th>Weight</th> <th>Total Weight</th>	Description	number	Length	Total Length	Surface Area	Volume	Unit Weight	Weight	Total Weight
1 Chord 2 1 66.31 66.31 333.57 5.73 43.00 2008.30 2. 2 Chord 3 1 66.31 65.31 333.57 5.73 43.00 2008.30 2. 3 Chord 4 1 66.31 65.31 333.57 5.73 43.00 2008.30 2. 4 Onord 4 1 66.31 65.31 333.57 5.73 43.00 2008.30 2. 5 Colum A 1 65.31 65.31 333.57 5.73 43.00 2008.30 2. 5 Colum A 1 26.25 26.25 58.74 1.02 18.99 498.41 0. m 5 287.47 1393.03 23.94 11. 1	Description	of Sets	[ft]	[ft]	[ft 2]	[ft 3]	[lb/ft]	[b]	[kip]
2 Chord 3 1 65.31 333.57 5.73 43.00 2808.30 2. 3 Chord 4 1 65.31 65.31 333.57 5.73 43.00 2808.30 2. 4 Chord 6 1 65.31 63.31 333.57 5.73 43.00 2808.30 2. 4 Chord 6 1 65.31 65.31 333.57 5.73 43.00 2808.30 2. 5 Column A 1 26.25 26.25 58.74 1.02 18.99 498.41 0. m 5 287.47 1393.03 23.94 11 11	Chord 2	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.8
B Chord 4 1 65.31 65.31 333.57 5.73 43.00 2808.30 2. Chord 6 1 65.31 65.31 333.57 5.73 43.00 2808.30 2. Column A 1 0.531 65.31 333.57 5.73 43.00 2808.30 2. Column A 1 0.521 65.25 5.574 1.02 18.99 498.41 0. m 5 287.47 1393.03 23.94 11. 11.	Chord 3	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.8
Cord 6 1 65.31 63.31 333.57 5.73 43.00 2808.30 2.2 Column A 1 26.25 26.25 58.74 1.02 18.99 498.41 0. D 5 287.47 1393.03 23.94 11. 11.	Chord 4	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.
Column A 1 26.25 26.25 58.74 1.02 18.99 498.41 0. n 5 287.47 1393.03 23.94 11.	Chord 6	1	65.31	65.31	333.57	5.73	43.00	2808.30	2.
0 5 28747 1393.03 23.94 11.	Column A	1	26.25	26.25	58.74	1.02	18.99	498.41	0.
		Chord 4 Chord 6 Column A	Chord 4 1 Chord 6 1 Column A 1 5	Chord 4 1 65.31 Chord 6 1 65.31 Column A 1 26.25 5 5	Chord 4 1 65.31 66.31 Chord 6 1 1 65.31 65.31 Column A 1 1 26.25 287.47	Chord 4 1 65.31 63.31 333.57 Chord 6 1 65.31 65.31 333.57 Column A 1 26.25 282.5 58.74 5 287.47 1393.03	Chord 4 1 65.31 65.31 333.57 5.73 Chord 6 1 65.31 65.31 65.31 333.57 5.73 Column A 1 26.25 58.74 1.02 5 287.47 1393.03 23.94	Chod 4 1 65.31 333.57 5.73 43.00 Chord 6 1 65.31 65.31 333.57 5.73 43.00 Column A 1 26.25 58.74 1.02 18.99 5 287.47 1393.03 23.94 1 9	Chord 4 1 65.31 63.31 333.57 5.73 43.00 2008.30 Chord 6 1 65.31 63.31 333.57 5.73 43.00 2008.30 Column A 1 26.25 26.25 58.74 1.02 18.99 498.41 S 287.47 1393.03 23.94 33.97 5.73 49.00 2008.30 Column A 1 26.25 287.47 1393.03 23.94 498.41

Figure 4.11: Window 4.2 Parts List by Set of Members

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

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

5 Results Evaluation

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

2.2 Desigr	n by Cross	s-Section											
	A	B	C	D	Е					F			
Section	Member	Location	Load-	Design									
No.	No.	x [ft]	ing	Ratio				Des	sign Accor	ding to Formula			
5	WT 7x19	AISC 14											
	56	22.97	CO34	0.01	≤1	100) Negligible interr	nal forces						
	57	13.12	CO41	0.53	≤1	111) Chapter F - Yiel	ding - Bending (about y-axi	s acc. to F	2 - F12			
	57	13.12	CO40	0.09	≤1	112) Chapter F - Yiel	ding - Bending (about z-axi	s acc. to F	2 - F12			
	57	13.12	CO41	0.53	≤1	117) Chapter F - Late	eral-torsional bu	ckling - aco	c. to F9				
	56	5.10	CO35	0.00	≤1	133) Chapter F - Flar	nge local buckli	ng of tees a	and double	e angles do not ap	ply - acc.	to F9	
	57	13.12	CO41	0.55	≤1	134) Chapter F - Ster	m local buckling) of tees an	d double	angles - acc. to F)		
	57	0.00	CO41	0.02	≤1	160) Chapter G - Nor	minal shear stre	ngth in z-a	kis - acc. t	o G2 (G3,G4) - Ur	stiffened o	cross-section	
	57	0.00	CO40	0.27	≤1	303) Chapter E - Flex	ural buckling a	bout y-axis	acc. to E	7 - Slender elemer	its		
	57	0.00	CO40	0.47	≤1	313) Chapter E - Flex	ural buckling a	bout z-axis	acc. to E	7 - Slender elemer	its		×
			Max:	0.85	≤ 1	0		L 91 :	e	3	> 1,0	~ 7 🗳 🛃 1	8
Details -	Member 57	7 - x: 13.12 ft	- CO41					-			5 - WT	7x19 AISC 14	
⊞ Materi	al Propertie	es - Steel A99	02 ANSI/A	ISC 360-16:2016-	6								
⊕ Cross-	Section Pr	operties - W	T 7x19 AI	SC 14				•					
Design	n Internal F	orces											
	Section Ty	/pe										6.75	
Design	n Ratio												
Rec	quired Flexi	ural Strength				Mr,y	8.34	kipft			1 1		
Yiel	d Stress					Fy	50.00	ksi				1	
Pla	stic Sectior	n Modulus				Zy	/.45	in ³			00	0.31	1
Ela	Elastic Section Modulus					Sy	4.22	in ³			-		
Yiel	Yield Moment Plastia Randina Manaat					My	17.58	kipft					
Pla	stic Bendin	g Moment				M _{pl,y}	31.04	kipft			+	- ^N	
Nor	ninal Hexu	ral Strength				Mn,y	17.58	kipft		(F9-1)		÷	
Res	sistance Fa	actor for Flexu	re			Фb	0.900	1.0		(F1)		z	
Des	sign Hexura	ai Strength				⊕b [™] Mn,y	15.83	kipft		00.4			
Design Ratio η _b 0.53 ≤ 1 (B3-1)									≤1				

Figure 5.1: Buttons for results evaluation

The buttons have the following functions:

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

Table 5.1: Buttons in Windows 2.1 to 2.5



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

Background graphic and view mode

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



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



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

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

RFEM/RSTAB work window

Graphics

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

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



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

X_XX

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

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



RF-STEEL AISC CA1 - Columns

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



Figure 5.4: *Display* navigator: *Results* \rightarrow *Members*

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

5



Figure 5.5: Design ratios with display option Without Diagram

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

RF-STEEL AISC

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

5.2 Result Diagram

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

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

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

Results ightarrow Result Diagrams for Selected Members



or the toolbar button shown on the left.

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



Figure 5.6: Dialog box Result Diagram on Member

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

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









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

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



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

Filtering design ratios

Graphics

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

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



or use the toolbar button shown on the left.

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

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

Filtering members

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



Figure 5.7: Filtering design ratios of one frame

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

5.4 Mode Shapes

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



I

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

Figure 5.8: Mode shape of a set of members

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

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



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

6.1 Printout Report

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



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

A Printout report - PR1: I	nput data and reduced results*				– 🗆 X
File View Edit Setti	ngs insert Help				
1 🔁 🖬 🕞 💟 <	S & & B & A & A	- 🖎 🔂 🗈 🗅 🔗 🖉			
Printout Report Navigator	×	2.4 DESIGN BY MEN	1BER		^
RFEM		Member Location LC/CO/	Design	Design	Description
RF-STEEL AISC	of steel members according to	No. x [ft] RC 11 Cross-section No. 3 - W 21	x55 AISC 14 2 - W 14x	No. 53 AISC 14	
1 1 1.1 Gener	ral Data	0.00 CO16 Cross-Section Type	0.05 ≤ 1	111) Chapter F - Yieldi	ng - Bending about y-axis acc.
1.2 Mater 1.3 Cross	rials -Sections	Flexure in Flange	.13 in λ _α ζ	9 152	(bf)r
1.5 Effect	ive Lengths - Members	tr C	.50 in λ_{ef}	24.083	Typer
1.6 Effect	n Parameters	h 18	.38 in λ _{r,w}	137.274	Flexure
1.8 Noda	Printout Report Selection - PR1	tw C	.37 in n/w	49.000	×
Results	Program				~
- 🛅 2.3 De	RFEM	Display			
	RF-STABILITY	Display			Set
		2.1 Design by Load Case		No. Selection (e.g. 1-5,20)	
		2.2 Design by Cross-Section	Cross-sections:	All	
		2.3 Design by Set of Members	Sets:	All	
		2.4 Design by Member	Members:	All	
		2.5 Design by x-cocation	monbors.	All	
		3.1 Governing Internal Forces	Select Intermediate Result	ts X	~ 3
		3.2 Governing Internal Forces	Printout Report		- B
		3.3 Member Slendernesses	With intermediate result	s Form: () Short O Long	~ 3
		4.1 Parts List by Member	Chapters to Display		~
		4.2 Parts List by Set of Members	Material Data		~
		Filter settings	Cross-Section Propertie	es	
		> 1,0 ~	Effective Widths		
			Design		
				3	
			2	OK Cancel	
	Cover sheet				
	Contents				
	Info pictures				
	D				OK Cancel
		Design Ratio for Combi	ned Forces 17 ŋ _{b.z}	0.00	η
<	>	<			>
		RF-STEEL AISC			Pages: 282 Page: 261 .:

Figure 6.1: Selecting designs and intermediate results in printout report



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

R

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

6.2 Graphic Printout

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



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

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

File \rightarrow Print Graphic



or use the toolbar button shown on the left.

4⊵	<u>F</u> ile	<u>E</u> dit	<u>V</u> iew	Insert	<u>C</u> alculate	<u>R</u> esults	<u>T</u> ools	Ta <u>b</u> le	<u>O</u> ptions
	2	33			50	🖉 🧐	Q 🛃		<u> </u>
۹.	- 🦻	/ ½ -	• 1 -	Print	Graphic	- <u>9×x</u> 🖭	1 🐴	- 🛅	📬 = 🗊

Figure 6.2: [Print Graphic] button in RFEM toolbar

The Graphic Printout dialog box appears.

Graphic Printout		×			
General Options Color Scale Factors Bord	der and Stretch Factors				
Graphic Picture	Window To Print Graphic Scale				
O Directly to a printer	Current only As screen view				
To a printout report: PR1: Ir	O More 🔍 🖲 Window filling				
◯ To the Clipboard	O Mass print ◯ To scale 1: 10 ∨				
⊖ To 3D PDF					
Graphic Picture Size and Rotation	Options				
☑ Use full page width	Show results for selected x-location in result diagram				
O Use full page height	Lock graphic picture (without update)				
● Height: 50 [% of page]					
	Show printout report on [OK]				
Rotation: 0 🐳 [°]					
Header of Graphic Picture					
Design Ratio					
\mathfrak{D}	OK Cancel				

Figure 6.3: Dialog box Graphic Printout, tab General

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

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

Remove from	Printout Report					
Start with New	Start with New Page					
Selection						
Properties						

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

7 General Functions

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

7.1 Design Cases

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

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

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

Create design case

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

File ightarrow New Case.

The following dialog box appears.

New RF-STEEL AISC Case							
No.	Description						
		1					
\mathfrak{D}	OK Cancel						

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

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

Rename design case

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

$\textbf{File} \rightarrow \textbf{Rename Case}.$

The following dialog box appears.

Rename RF-STEEL AISC	Case	×
No. Description 2 New description	tion	~
D	ОК	Cancel

Figure 7.2: Dialog box Rename RF-STEEL AISC Case

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



Figure 7.1: Dialog box New RF-STEEL AISC Case

Copy design case

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

File ightarrow Copy Case.

The following dialog box appears.

Copy RF-	-STEEL AISC Case	×
Copy fro	om Case	
CA1 - D	Design of steel members according to AISC (LRI $ \sim $	
New Cas	se	
No.:	Description:	
3	ASD	~
\mathfrak{D}	OK Ca	ncel

Figure 7.3: Dialog box Copy RF-STEEL AISC Case

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

Delete design case

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

 $\textbf{File} \rightarrow \textbf{Delete Case}.$

The following dialog box appears.

Delete Cases							
Availab	le Cases						
No.	Description						
1	Design of steel members according to AISC						
2	New description						
3	ASD						
	-						
	I <u> </u>						
٢	OK Cancel						

Figure 7.4: Dialog box Delete Cases

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



7.2 Cross-Section Optimization

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

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

.2 Design	h by Cross	-Section					
	A	В	С	D	F		F
Section	Member	Location	Load-	Design			
No.	No.	x [ft]	ing	Ratio			Design According to Formula
1	W 6x20	AISC 14					
	34	19.69	CO16	0.56	≤1	111) Chapter F - Yie	lding - Bending about y-axis acc. to F2 - F12
	34	19.69	CO16	0.89	<1	113) Chanter F - Lat	eral-torsional buckling - acc. to F2 - F7
	34	2.81	<u>G</u> o to	Cross-Section		Doubleclick	ge local buckling does not apply - acc. to F2 - F5
	34	0.00	Info A	bout Cross-Sec	tion.		ninal shear strength in z-axis - acc. to G2 (G3,G4) - Unstiffened cross-section
	41	0.00		ibout cross see			ural buckling about y-axis acc. to E3
	41	0.00	<u>O</u> ptin	nize Cross-Sectio	n n		ural buckling about z-axis acc. to E3
	41	0.00	Cross	-Section Optimi	zatio	n Parameters	sional or flexural-torsional buckling acc. to E4
	34	19.69		0.32		Jozy Chapter 11 Jan	gle axis /or biaxial/ flexure with axial compression force - acc. to H1.1
	34	19.69	CO16	0.88	≤1	334) Chapter H - Co	mpression force with single/major axis bending - acc. to H1.3

Figure 7.5: Shortcut menu to Optimize Cross-Section

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

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



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

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

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

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



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

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

$\textbf{Edit} \rightarrow \textbf{Export All Cross-Sections to RFEM/RSTAB}.$

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



Figure 7.7: Shortcut menu in Window 1.3 Cross-Sections

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

RF-STEEL AISC Information No. 41852							
Do you want to transfer the changed cross-sections to RFEM? If so, the results of RFEM and RF-STEEL AISC will be deleted.							
	ļ						

Figure 7.8: Confirmation when exporting cross-sections

Calculation

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

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



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

7.3 Units and Decimal Places

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

Settings \rightarrow Units and Decimal Places.

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

Units and Decimal Places				×
Program / Module	RF-STEEL AISC			
	Output Data		Parts List	
RF-STEEL Surfaces		Linit Dec places		Linit Dec places
DE STEEL CC2		onit Dec. places		Unit Dec. pideea
	Stresses:	ksi 🗸 2 🚽	Lengths:	ft 🗸 2 🛨
	Design ratios:	- ~ 2	Total lengths:	ft ~ 2
RF-STEEL SIA	Dimensionless:		Surface areas:	₽^2 v 2≜
RF-STEEL BS	Dimension need.		candoo aroao.	
RF-STEEL GB			Volumes:	ft^3 🗸 2 🖨
			Weight per length:	lb/ft ∨ 2≑
RF-STEEL AS			Maisht.	
RF-STEEL NTC-DF			vveignt:	
RF-STEEL SP			Total weight:	kip 🗸 2 🜩
···· RF-STEEL Plastic				
···· RF-STEEL SANS				
···· RF-STEEL Fatigue Mer				
···· RF-STEEL NBR				
RF-STEEL HK				
···· RF-ALUMINUM				
···· RF-ALUMINUM ADM				
···· RF-KAPPA				
···· RF-LTB				
···· RF-FE-LTB				
···· RF-EL-PL				
RF-C-TO-T				
···· PLATE-BUCKLING				
RF-CONCRETE Columi				
2 2 🖻 F 🖷				OK Cancel

Figure 7.9: Dialog box Units and Decimal Places

۱

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

7.4 Data Transfer

7.4.1 Exporting Materials to RFEM/RSTAB

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

```
Edit \rightarrow Export All Materials to RFEM/RSTAB.
```

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



Figure 7.10: Shortcut menu of Window 1.2 Materials

Calculation

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

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

7.4.2 Exporting Effective Lengths to RFEM/RSTAB

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

```
Edit \rightarrow Export All Effective Lengths to RFEM/RSTAB.
```

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

```
Export Effective Length to RFEM
Export All Effective Lengths to RFEM
Import Effective Length from RFEM
Import All Effective Lengths from RFEM
```

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

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

7.4.3 Exporting Results

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

Clipboard

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

Printout Report

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

```
\textbf{File} \rightarrow \textbf{Export to RTF}.
```

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

Excel

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

File \rightarrow Export Tables.

Export of Tables	×					
Table Parameters	Application					
With table header	Microsoft Excel					
Only marked rows	○ CSV file format					
Transfer Parameters						
Export table to active workbook						
Export table to active workshee	t					
Selected Tables						
Active table	Export hidden columns					
○ All tables	Export tables with details					
✓ Input tables						
Result tables						
(?)	OK Cancel					
	Calicer					

Figure 7.12: Dialog box *Export of Tables*

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

	🔒 🤊	• (°" ~ -				Table1 - Microsoft Excel — 🗆 🗙						
F	ile	Home Ins	ert Pa	ige Layout	Form	mulas Data Review View Add-Ins 🛛 🛇 🕜 🗆 📾	23					
	B	3	+ (<i>f</i> ∗ Cro	oss-s	section No. 1 - W 12x40 AISC 14	~					
	Α	В	С	D	Е	F	E					
1	Member	Location	Load-	Design								
2	No.	x [ft]	ing	Ratio		Design According to Formula						
3	34 Cross-section No. 1 - W 12x40 AISC 14											
4		19.69	CO16	0.48	≤1	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12	1					
5		8.44	CO16	0.02	≤1	112) Chapter F - Yielding - Bending about z-axis acc. to F2 - F12						
6		19.69	CO16	0.72	≤1	113) Chapter F - Lateral-torsional buckling - acc. to F2 - F7						
7		2.81	CO13	0.00	≤1	122) Chapter F - Flange local buckling does not apply - acc. to F2 - F5						
8		5.62	CO14	0.00	≤1	124) Chapter F - Flange local buckling does not apply - acc. to F6						
9		0.00	CO16	0.05	≤1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,G4) - Unstiffened cross-section						
10		19.69	CO14	0.07	≤1	216) Design Guide No. 9 - Check of shear stress due to torsion and shear stresses						
11		0.00	CO16	0.04	≤1	301) Chapter E - Flexural buckling about y-axis acc. to E3						
12		0.00	CO16	0.09	≤1	311) Chapter E - Flexural buckling about z-axis acc. to E3						
13		0.00	CO16	0.05	≤1	321) Chapter E - Torsional or flexural-torsional buckling acc. to E4						
14		19.69	CO16	0.76	≤1	332) Chapter H - Single axis /or biaxial/ flexure with axial compression force - acc. to H1.1						
15		19.69	CO16	0.63	≤1	334) Chapter H - Compression force with single/major axis bending - acc. to H1.3						
16		19.69	CO16	0.48	≤1	338) Chapter H - Strength of non-HSS members under torsion and combined forces about major axis						
17		19.69	CO16	0.48	≤1	339) Chapter H - Strength of non-HSS members under torsion and combined forces about minor axis						
18												
19	35	Cross-sectio	n No. 3 - 1	W 21x55 AIS	C 14	2 - W 12x87 AISC 14						
20		0.00	CO16	0.20	≤1	111) Chapter F - Yielding - Bending about y-axis acc. to F2 - F12						
21		0.00	CO13	0.00	≤1	116) Chapter F - Lateral-torsional buckling does not apply - acc. to F2 - F11						
22		0.00	CO13	0.00	≤1	122) Chapter F - Flange local buckling does not apply - acc. to F2 - F5						
23		0.00	CO16	0.05	≤1	160) Chapter G - Nominal shear strength in z-axis - acc. to G2 (G3,G4) - Unstiffened cross-section						
24		0.00	CO14	0.01	≤1	303) Chapter E - Flexural buckling about y-axis acc. to E7 - Slender elements						
25		0.00	CO14	0.01	≤1	313) Chapter E - Flexural buckling about z-axis acc. to E7 - Slender elements	Ŧ					
	()	2.4 Design	by Memb	per 🖉								
Rea	dv											

Figure 7.13: Results in Excel

8 Example

Column with Compression and Biaxial Bending

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

8.1 Design Values

Model and loads





8

Design values of loads: P = 65.0 kip w = 0.5 kip/ftV = 2.0 kip

Figure 3.2: Internal forces

Internal forces due to linear analysis

o Example

Design location



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

8

 $P = -65.00 \text{ kip} \qquad M_{v} = 12.25 \text{ kipft} \qquad M_{z} = 7.00 \text{ kipft} \qquad V_{v} = 1.00 \text{ kip} \qquad V_{z} = 0.00 \text{ kip}$

8.2 Cross-Section Properties - W 8x28

Cross-Section Property	Symbol	Value	Unit
Gross area	A _g	7.08	in ²
Moment of inertia about major principal axis	l _y	82.70	in ⁴
Moment of inertia about minor principal axis	l _z	18.30	in ⁴
Radius of gyration about major principal axis	r _y	3.42	in
Radius of gyration about minor principal axis	r _z	1.61	in
Cross-section weight	wt	24.11	lb/ft
Torsional constant	J	0.35	in ⁴
Warping constant	C _w	259.00	in ⁶
Elastic section modulus about major axis	Sy	20.90	in ³
Elastic section modulus about minor axis	Sz	5.63	in ³
Plastic section modulus about major axis	Zy	23.10	in ³
Plastic section modulus about minor axis	Zz	8.57	in ³

8.3 Material Properties - A992

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

8.4 Classification of Cross-Section

Compression – [1] Table B4.1a

Flange

Case 1: Flanges of rolled I-shaped sections

$$b = 3.25 \text{ in}$$

$$t = 0.375 \text{ in}$$

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

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

 $8.67 \leq 13.49 \quad \rightarrow \text{nonslender}$

Web

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

$$b = 6.125 \text{ in}$$

$$t = 0.25 \text{ in}$$

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

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

 $24.50 \leq 35.88 \quad \rightarrow \text{nonslender}$

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

Flexure – [1] Table B4.1b

Flange

Case 10: Flanges of rolled I-shaped sections

$$b = 3.25 \text{ in}$$

$$t = 0.375 \text{ in}$$

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

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

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

8.67
$$\leq$$
 9.15 \rightarrow compact

8

Web

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

$$h = 6.125 \text{ in}$$

$$t_w = 0.25 \text{ in}$$

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

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

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

 $24.50 \leq 90.55 \quad \rightarrow \text{compact}$

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

Classification in RF-/STEEL AISC

Cross-Section Type								
Uniform Compression in Flange								
- Half of Full Flange Width	b	3.250	in		Tab. B4.1			
Thickness	tr	0.375	in		Tab. B4.1			
 b/t-Limit for Compact Flange 	λp,f	NA			Tab. B4.1			
- b/t-Limit for Nonslender Flange	λr.f	13.487			Tab. B4.1			
- b/t-Ratio	(b/t) f	8.667		≦λr,f				
 Type of Flange 	Type f	Nonslender			Tab. B4.1			
 Uniform Compression in Web 								
- Length	h	6.125	in		Tab. B4.1			
Thickness	tw	0.250	in		Tab. B4.1			
 - h/t_w-Limit for Compact Web 	λ _{p,w}	NA			Tab. B4.1			
 - h/t_w-Limit for Nonslender Web 	λr,w	35.884			Tab. B4.1			
— - h/t _w -Ratio	h/tw	24.500		$\leq \lambda_{r,w}$				
 Type of Web 	Typew	Nonslender			Tab. B4.1			
 Type of Cross-Section in 	Compression	Nonslender						
Flexure in Flange								
- Half of Full Flange Width	b	3.250	in		Tab. B4.1			
— - Thickness	tr	0.375	in		Tab. B4.1			
 b/t-Limit for Compact Flange 	λp,f	9.152			Tab. B4.1			
 b/t-Limit for Noncompact Flange 	λr,f	24.083			Tab. B4.1			
b/t-Ratio	(b/t) f	8.667		≦λp,f				
- Type of Flange	Type f	Compact			Tab. B4.1			
- Flexure in Web								
- Length	h	6.125	in		Tab. B4.1			
Thickness	tw	0.250	in		Tab. B4.1			
 - h/t_w-Limit for Compact Web 	λ _{p,w}	90.553			Tab. B4.1			
 - h/t_w-Limit for Noncompact Web 	λr,w	137.274			Tab. B4.1			
- h/t _w -Ratio	h/t _w	24.500		$\leq \lambda_{p,W}$				
- Type of Web	Туре и	Compact			Tab. B4.1			
- Type of Cross-Section in	Flexure	Compact						

Figure 8.3: Classification in RF-/STEEL AISC

8

8.5 Design for Compression



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

Member slenderness:

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

Limit of slenderness according to Section E3:

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

 $49.12 \leq 113.43$

Elastic buckling stress according to Equation E3-4:

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

Critical stress according to Equation E3-2:

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

Nominal compressive strength according to Equation E3-1:

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

Design compressive strength according to Section E1:

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

Design ratio according to Equation B3-1:

$$\eta = \frac{P}{\phi P_n} = \frac{65.00}{267.05} = 0.24 \le 1.00 \Rightarrow \mathbf{OK}$$

Buckling design in RF-/STEEL AISC

🗇 Design Ratio							
 Required Compressive Strength	Pr	65.00	kip				
 Effective Member Length	Leff,y	14.00	ft				
 Yield Stress	Fy	50.00	ksi				
 Modulus of Elasticity	E	28998.60	ksi				
 Slendemess Ratio	KyL/ry	49.16		≤ 4.71√E/Fy	(E2)		
 Elastic Critical Buckling Stress	F _{e,y}	118.45	ksi		(E3-4)		
 Critical Stress	F _{cr,y}	41.90	ksi		(E3-2 or E3-3)		
 Resistance Factor for Compression	Φc	0.90			(E1)		
 Nominal Compressive Strength	P _{n,y}	296.67	kip		(E3-1)		
 Design Compressive Strength	Φc*Pn,y	267.00	kip				
Design Ratio	ηv	0.24		≤1	(B3-1)		

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

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

Member slenderness:

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

Limit of slenderness according to Section E3:

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

 $104.35 \leq 113.43$

Elastic buckling stress according to Equation E3-4:

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

Critical stress according to Equation E3-2:

-

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

Nominal compressive strength according to Equation E3-1:

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

Design compressive strength according to Section E1:

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

Design ratio according to Equation B3-1:

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

Buckling design in RF-/STEEL AISC

🗇 Design Ratio							
 Required Compressive Strength 	Pr	65.00	kip				
Effective Member Length	Leff,z	14.00	ft				
 Yield Stress 	Fy	50.00	ksi				
 Modulus of Elasticity 	E	28998.60	ksi				
Slenderness Ratio	KzL/rz	104.50		≤ 4.71√E/Fy	(E2)		
 Elastic Critical Buckling Stress 	F _{e,z}	26.21	ksi		(E3-4)		
 Critical Stress 	F _{or,z}	22.50	ksi		(E3-2 or E3-3)		
 Resistance Factor for Compression 	Φc	0.90			(E1)		
 Nominal Compressive Strength 	Pn,z	159.32	kip		(E3-1)		
 Design Compressive Strength 	Φc*Pn,z	143.39	kip				
Design Ratio	ηz	0.45		≤1	(B3-1)		

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





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

8.6.1 Flexure about y-Axis (Major Principal Axis)

8.6.1.1 Yielding

Nominal flexural strength according to Equation F2-1:

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

Design flexural strength according to Section F1:

 $\phi_b M_n = 0.9 \cdot 96.25 = 86.63$ kipft

Design ratio according to Equation B3-1:

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

Yielding design in RF-/STEEL AISC

🛱 Design Ratio					
 Required Flexural Strength 	M _{r,y}	12.25	kipft		
 Yield Stress 	Fy	50.00	ksi		
 Plastic Section Modulus 	Zy	23.10	in ³		
 Plastic Bending Moment 	M _{pl,y}	96.25	kipft		
 Nominal Flexural Strength 	Mn,y	96.25	kipft		(F2-1)
 Resistance Factor for Flexure 	Φb	0.90			(F1)
 Design Flexural Strength 	Φb [*] Mn,y	86.63	kipft		
Design Ratio	ηь	0.14		≤1	(B3-1)

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

8.6.1.2 Lateral-Torsional Buckling

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

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

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

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

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

$$L_{r} = 1.95 r_{ts} \frac{E}{0.7 F_{y}} \sqrt{\frac{Jc}{S_{y} h_{o}} + \sqrt{\left(\frac{Jc}{S_{y} h_{o}}\right)^{2} + 6.76 \left(\frac{0.7 F_{y}}{E}\right)^{2}}}$$

= 1.95 \cdot 1.81 \cdot \frac{29,000}{0.7 \cdot 50} \sqrt{\frac{0.35 \cdot 1.0}{20.9 \cdot 7.5} + \sqrt{\left(\frac{0.35 \cdot 1.0}{20.9 \cdot 7.5}\right)^{2} + 6.76 \left(\frac{0.7 \cdot 50}{29,000}\right)^{2}}} = 228.11 \text{ in } = 19.00 \text{ ft}}
where $r_{ts} = \sqrt{\frac{\sqrt{I_{z} C_{w}}}{S_{y}}} = \sqrt{\frac{\sqrt{18.30 \cdot 259.00}}{20.90}} = 1.81 ext{ in}}$

© DLUBAL SOFTWARE 2018

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

$$L_b = 14 \, \text{ft}$$

 $L_p < L_b \leq L_r$

Nominal flexural strength according to Equation F2-2:

$$\begin{split} M_n &= C_b \left[M_p - \left(M_p - 0.75 \, F_y \, S_y \right) \left(\frac{L_b - L_p}{L_r - L_p} \right) \right] \le M_p \\ M_n &= 1.14 \left[1,155 - \left(1,155 - 0.7 \cdot 50 \cdot 20.90 \right) \left(\frac{14 \cdot 12 - 68.24}{228.11 - 68.24} \right) \right] \le 1,155 \text{ kipin} \\ M_n &= 1,015.44 \text{ kipin} = 84.62 \text{ kipft} \le M_p \end{split}$$

Design flexural strength according to Section F1:

$$\phi_b M_n = 0.9 \cdot 84.62 = 76.16$$
 kipft

Design ratio:

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

Lateral-torsional buckling design in RF-/STEEL AISC

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

🗇 Design Ratio							
Bending about the major axis							
 Required Flexural Strength 	M _{r,y}	12.25	kipft				
- Amplifier	αcr	6.26					
 Elastic Critical Moment for Lateral-Torsional Buckling 	Mor	76.70	kipft				
 Nominal Flexural Strength 	M _{n,y}	76.70	kipft				
 Resistance Factor for Flexure 	Φb	0.90			(F1)		
 Design Flexural Strength 	⊕ b [*] Mn	69.03	kipft				
Design Ratio	ηь	0.18		≤1	(B3-1)		

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

8.6.2 Flexure about z-Axis (Minor Principal Axis)

8.6.2.1 Yielding

Nominal flexural strength according to Equation F6-1:

$$M_n = M_p = F_y Z_z \le 1.6 F_y S_z$$

$$M_n = 50.8.57 = 428.5 \text{ kipin} = 35.71 \text{ kipft} \le 1.6 F_y S_z = 1.6.50.5.63 = 450.4 \text{ kipin} = 37.53 \text{ kipft}$$

Design flexural strength according to Section F1:

$$\phi_b M_n = 0.9 \cdot 35.71 = 32.14$$
 kipft

Design ratio according to Equation B3-1:

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

Yielding design in RF-/STEEL AISC

🗇 Design Ratio					
 Required Flexural Strength 	Mr,z	7.00	kipft		
 Yield Stress 	Fy	50.00	ksi		
 Plastic Section Modulus 	Zz	8.57	in ³		
 Plastic Bending Moment 	M _{pl,z}	35.71	kipft		
 Elastic Section Modulus 	Sz	5.63	in ³		
 Yield Moment 	Mz	23.46	kipft		
 Nominal Flexural Strength 	Mn,z	35.71	kipft		(F6-1)
 Resistance Factor for Flexure 	Φb	0.90			(F1)
 Design Flexural Strength 	Φb [*] Mn,z	32.14	kipft		
Design Ratio	ηь	0.22		≤1	(B3-1)

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

8.6.2.2 Flange Local Buckling

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

Flange local buckling design in RF-/STEEL AISC

🗇 Design Ratio						
 b/t-Limit for Compact Flange 	λp,f	9.15			Tab. B4.1	
 b/t-Limit for Noncompact Flange 	λr,f	24.08			Tab. B4.1	
— b/t-Ratio	(b/t)f	8.67		≦λp,f		
Design Ratio	ηь	0.00		≤1	(B3-2)	

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

8.7 Design for Combined Forces

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

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

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

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

$$\frac{P_r}{P_c} + \frac{8}{9} \left(\frac{M_y}{M_{c,y}} + \frac{M_z}{M_{c,z}} \right) \le 1.0$$
$$\frac{65.00}{143.75} + \frac{8}{9} \left(\frac{12.25}{76.16} + \frac{7.00}{32.14} \right) = 0.45 + \frac{8}{9} \left(0.16 + 0.22 \right) = 0.79$$

Design ratio:

 $0.79 \leq 1.00 \qquad \Rightarrow ~\textbf{OK}$

Design for combined forces in RF-/STEEL AISC

🖂 Design Ratio							
 Design ratio of axial compression 	ηο	0.45			acc. to Chapter E		
 Design ratio of flexure about y-axis 	ηь,у	0.16			acc. to Chapter F		
 Design ratio of flexure about z-axis 	η _{b,z}	0.22			acc. to Chapter F		
Design Ratio	η	0.79		≤1	(H1-1a)		

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



- [1] *Specification for Structural Steel Buildings*. American Institute of Steel Construction, Chicago, IL, 2016.
- [2] AISC Steel Design Guide 9: Torsional Analysis of Structural Steel Members. American Institute of Steel Construction, Chicago, IL, 1997.
⊿ ■ Dlubal

Index

A

ASD	 	 	 6
Axis	 ••••	 	 17

В

Background graphic	47
Beam type	28
Buckling	17
Buckling length	31
Buttons	46

С

Calculation	29
Cantilever	28, 32
Clipboard	60
Color bars	46
Color scale	51
Comment	. 6, 18
Control Panel	51
Coordinate system	11
Cross-section	44, 57
Cross-section design	38
Cross-section info	13
Cross-section library	11
Cross-section optimization	34, 57
Cross-section type	12

D

Decimal places	
Deflection	
Deflection analysis	
Degree of freedom	
Design	7, 36, 37, 39, 40, 42
Design case	
Design colored	
Design parameters	
Design ratio	
Detail settings	
Display navigator	

Ε

Effective area	20
Effective length	9
Effective length factor 1	8
Elastic critical moment 3	1
Excel	1
Exit RF-/STEEL AISC	5
Export	0
Export cross-section	8

Export effective length	. 60
Export material	. 60

F

Filter	51
Filtering members	51
Flexural buckling	16

G

General data	
Graphic printout	
Graphics	
Gross area	

Н

Hinge																		 .2	25
HSS	•	• •	•	•	•	 •	•	 •	•	•	•	•	•	•	•	•	•	 .2	29

I

Initial bow imperfection	7
Installation	3
Intermediate lateral restraints	ł
Internal forces	3

L

Lateral and torsional restraint
Lateral restraints
Lateral-torsional buckling 14, 18, 31
Length
Limit values
List of member
Load application
Load case
Load combination7
Loading
Location x
LRFD

М

. 9, 60
9
10
9,10
6
43
46 , 52

Ν

Navigator .																														5	
-------------	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	--	---	--

Net area)
Nodal support	1

0

Optimization 12, 34, 57, 58

Ρ

Panel	49, 51
Parameter	26
Parametric cross-section	57
Part	44
Parts list	44, 45
Precamber	28, 32
Print graphic	54
Printout report	53, 54
Program start	4

R

Reference length	8
Relatively	14
Remark	12
Rendering	51
Result combination	7, 8, 46
Result diagram	50
Result values	48
Result window	36
Results display	
Results evaluation	46
RF-/STEEL Warping Torsion	32
RF-STABILITY	16
RFEM/RSTAB graphic	54
RSBUCK	16

S

Second-order analysis	3
Selecting windows	5
Serviceability	1
Serviceability limit state	6
Set of members . 6, 19, 21, 25, 26, 27, 39, 42, 4	5
Shear buckling	0
Shear lag factor	0
Shifted members ends	2

Slenderness 43	3
Slenderness ratio	4
Special cases	0
Stability	0
Stability analysis	8
Stainless steel10	0
Start calculation	5
Start RF-/STEEL AISC	4
Stress point	3
Sum	5
Surface area	4

B

т

Taper	8
Torsional buckling 1	7
Transverse load	1

U

29, 46
32
. 9, 59
30
59

V

View mode	46, 47
Visibility	51
Volume	45

X

x-Location	• •	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	•	3	7,	4	1	
------------	-----	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	---	----	---	---	--

w

23
32
18
32, 33
45
30
5
48, 51