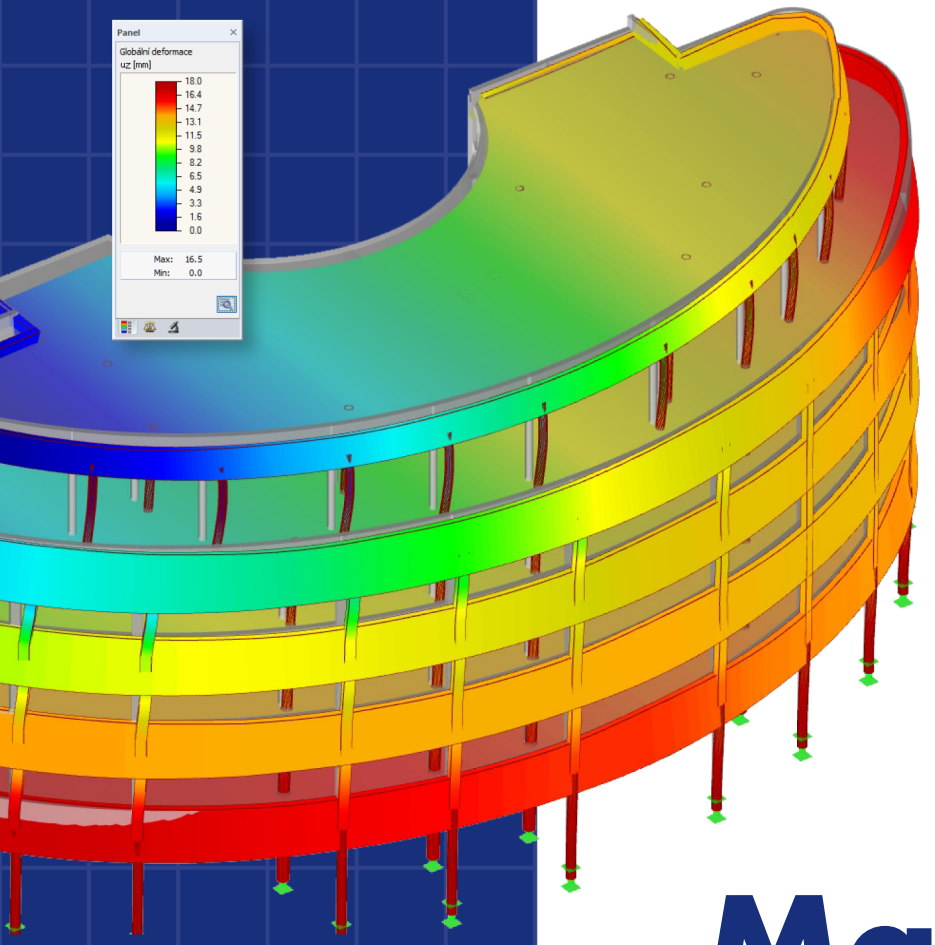
RF-/JOINTS

Design of Steel and Timber Connections



Manual

Version

October 2020

Dlubal

Dlubal Software

Short Overview

Dlubal Software GmbH
Am Zellweg 2
93464 Tiefenbach
Germany

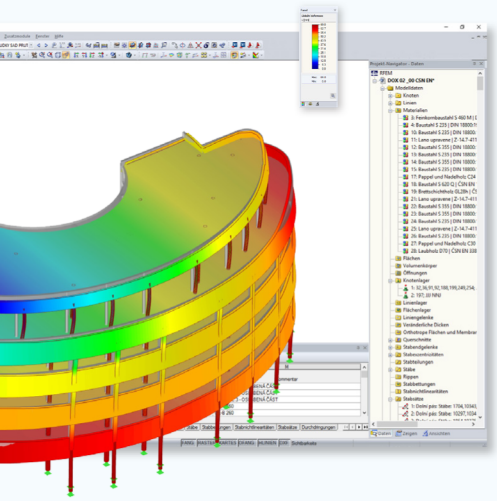
Telephone: +49 9673 9203-0
Fax: +49 9673 9203-51
E-mail: info@dlubal.com

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

Phone: +1 267 702-2815
E-mail: info@dlubal.com

Using the Manual

The program description is divided into chapters that describe the modules of the RF-/JOINTS connections. The explanations are based on the order and structure of the input windows with their specific parameters. General functions are described in the manuals of the main program RFEM or RSTAB.



Hint

The text of the manual shows the described buttons in square brackets, for example [OK]. They are also depicted on the left. Expressions that appear in dialog boxes, tables, and menus are set in *italics* to clarify the explanations. You can also use the search function for the Knowledge Base and FAQs on our website to find a solution in the posts about the connection modules.

Topicality

The high quality standards placed on the software are guaranteed by a continuous development of the program versions. This may result in differences between program description and the current software version you are using. Thank you for your understanding that no claims can be derived from the figures and descriptions. We always try to adapt the documentation to the current state of the software.

Table of Contents

1 Introduction

1.1 RF-/JOINTS Add-on Module

With the introduction of Eurocodes, the design of joints becomes more and more important. The European standard EN 1993-1-8 [1] describes the designs of steel connections. The design of timber connections is covered in EN 1995-1-1 [2]. The add-on modules RF-JOINTS (for RFEM) and JOINTS (for RSTAB) combine the connection modules for member elements in a single user interface. The scope of functions of the modules has also been extended and adapted to current requirements.

This manual describes the add-on modules of both main programs together under the name **RF-/JOINTS**.

At present, RF-/JOINTS covers the steel categories *Column Base*, *Pinned*, *Rigid*, *Tower*, *DSTV*, and *SIKLA*, as well as the timber categories *Steel to Timber* and *Timber to Timber*.

RF-/JOINTS Steel - Column Base designs footings of hinged and restrained column bases.

The Hinged Column Footing category includes the following base plate connections:

- Simple column base
- Tapered column base
- Column base plate for rectangular hollow sections
- Column base plate for circular hollow sections

For restrained column footings, the following design variants for I-sections are available:

- Base plate without stiffening
- Base plate with stiffeners in center of flanges
- Base plate with stiffeners on both sides of column
- Base plate with channel sections
- Bucket footing

RF-/JOINTS Steel - Pinned performs designs for the following shear connections of I-beams:

- Web cleat connection
- Fin plate connection
- End plate connection
- End plate connection with cleat

RF-JOINTS Steel - Rigid designs the following moment-resisting connections of I-beams:

- Beam connection to column with end plate
- Beam joint with end plate
- Beam joint with splices

RF-/JOINTS Steel - Tower designs nominally pinned bolted connections of lattice tower members for the following cases:

- Connection of diagonals without gusset plate 2D
- Connection of diagonals without gusset plate 3D
- Column joint with splices

RF-/JOINTS Steel - DSTV designs moment-resisting and pinned I-beam connections in compliance with the German guideline "Standardised Joints in Steel Structures" [3] [4]. The resistances are determined according to DIN EN 1993-1-8 [1].

For moment-resisting connections, the following design variants are available:

- End plate without column (type IH/IM)
- Beam joint with end plate (type IH/IM)
- One-sided beam with end plate (type IH/IM)
- Double-sided beams with end plates (type IH/IM)
- Purlin joint with splices (type PM)

For pinned connections, you can choose between the following designs:

- End plate (type IS), where applicable with notches (type IK)
- Angle (type IW), where applicable with notches (type IK)
- Long angles (type IG)
- Purlin joint with splices (type PM)

RF-/JOINTS Steel - Sikla analyzes the connections for sections of the fastening system manufacturer Sikla.

RF-/JOINTS Timber - Steel to Timber designs dowel, bolt, nail, and screw connections of timber members that are indirectly connected by steel plates.

Hinged, semi-rigid, and bending-resistant dowel connections are available for the following cases:

- Single member
- Continuous member with connected members
- Connection of several single members

RF-/JOINTS Timber - Timber to Timber performs the designs for screw connections of timber members.

The following joint types are possible:

- Single member or connection of several single members
- Front to side
- Front to front

The results including dimensions are shown in tables and graphics. Using the design cases, you can analyze different design variants.

Since RF-/JOINTS is integrated into the graphical user interface of the main program, you can use all the input data of the model as well as the internal forces for the design. Some modules also allow you to consider connection stiffnesses and eccentricities when determining internal forces. You can visualize the results in RFEM's or RSTAB's work window and include them in the global printout report.

We hope you will enjoy working with the RF-/JOINTS add-on modules.

Your Dlubal team

1.2 Using the Manual

Topics such as installation, graphical user interface, results evaluation, and printout are described in detail in the manuals of the main programs RFEM and RSTAB. This manual focuses on the typical features of the RF-/JOINTS add-on modules.

The descriptions in this manual follow the order and structure of the module's input and result windows. Chapter 2 describes the input parameters that apply to all steel and timber connections. The subsequent chapters describe specific input parameters of the individual connections modules. What follows is an explanation of the calculation as well as the evaluation and documentation of the results. The manual concludes with a description of general program functions and information on the export of data.

In the text, the described **buttons** are given in square brackets, for example [New]. They are also pictured on the left. Expressions that appear in dialog boxes, tables, and menus are highlighted in *italics* to clarify explanations.

As usual, you can use [Ctrl]+F to perform a full-text search in the PDF manual. If you cannot find what you are looking for, you can use the search function for the Knowledge Base on our website to find a solution among the articles about the connection add-on modules. You can also consult the FAQs on our website.

1.3 Starting the RF-/JOINTS Module

RFEM and RSTAB provide the following options to start the RF-/JOINTS add-on module.

Menu

To open the add-on module, go to the RFEM or RSTAB menu and select

Add-on Modules → Connections → RF-/JOINTS.

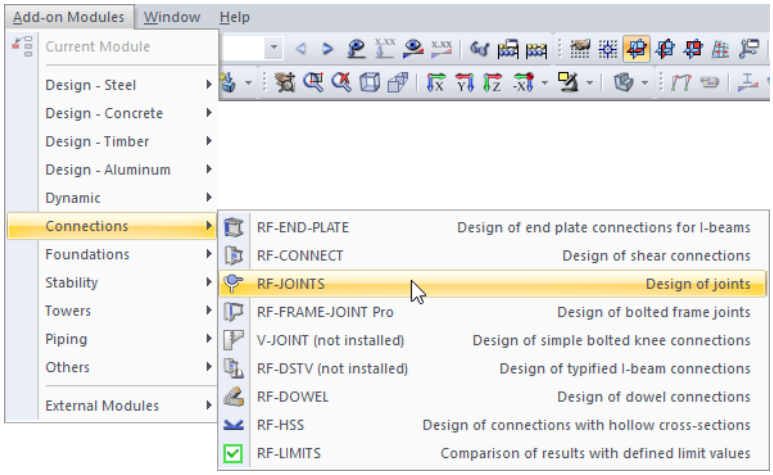


Image 1.1 Menu - *Add-on Modules → Connections → RF-JOINTS*

Navigator

Alternatively, you can start the add-on module in the *Data* navigator by selecting

Add-on Modules → RF-/JOINTS.

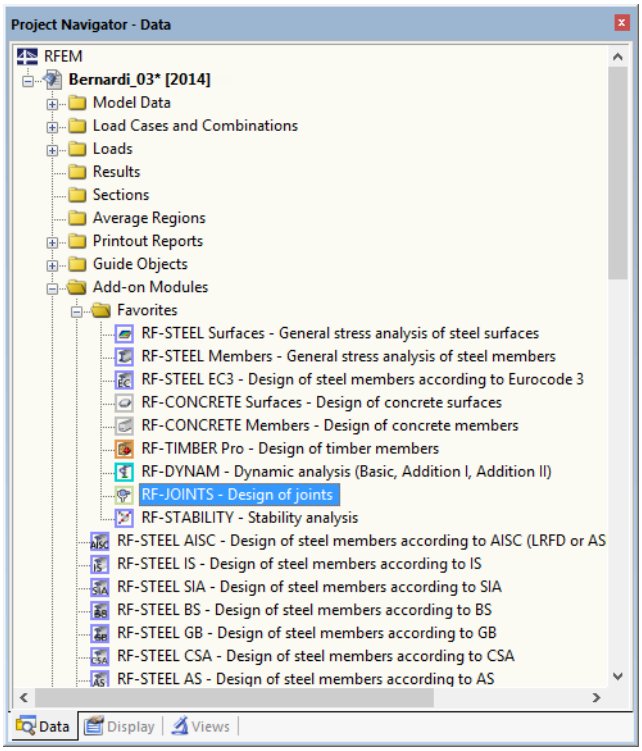
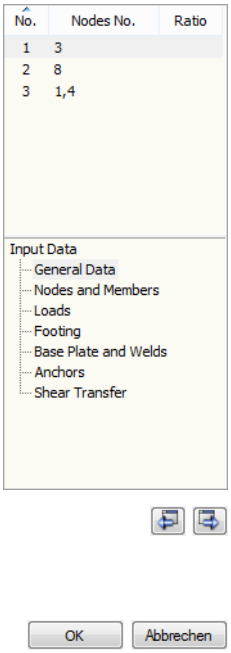


Image 1.2 Data navigator: *Add-on Modules* → *RF-JOINTS*

2 General Input Data



This chapter describes the input parameters that apply to all categories of steel and timber connections. The following chapters describe specific input parameters of the individual connection modules.

When you open RF-/JOINTS, a new window appears. In this window, a navigator is displayed on the left. The upper part of the navigator lists the design cases (see Chapter 15.1) with the selected nodes, the lower part manages the windows of specific modules.

The input data is defined in several module windows. When you open RF-/JOINTS for the first time, the following parameters are imported automatically:

- Load cases, load and result combinations, and dynamic combinations
- Materials
- Cross-sections
- Internal forces (in background, if calculated)

To open a window, click the corresponding entry in the navigator. Use the buttons shown on the left to set the previous or next window. You can also use the function keys [F2] (forwards) and [F3] (backwards) to go through the windows.

To save the entered data, click [OK]. RF-/JOINTS closes and you return to the main program. To exit the add-on module without saving the data, click [Cancel].

2.1 General Data

In Window *1.1 General Data*, you can define the basic settings necessary for the design of the connection. You can use the filter functions to select the type of joint.

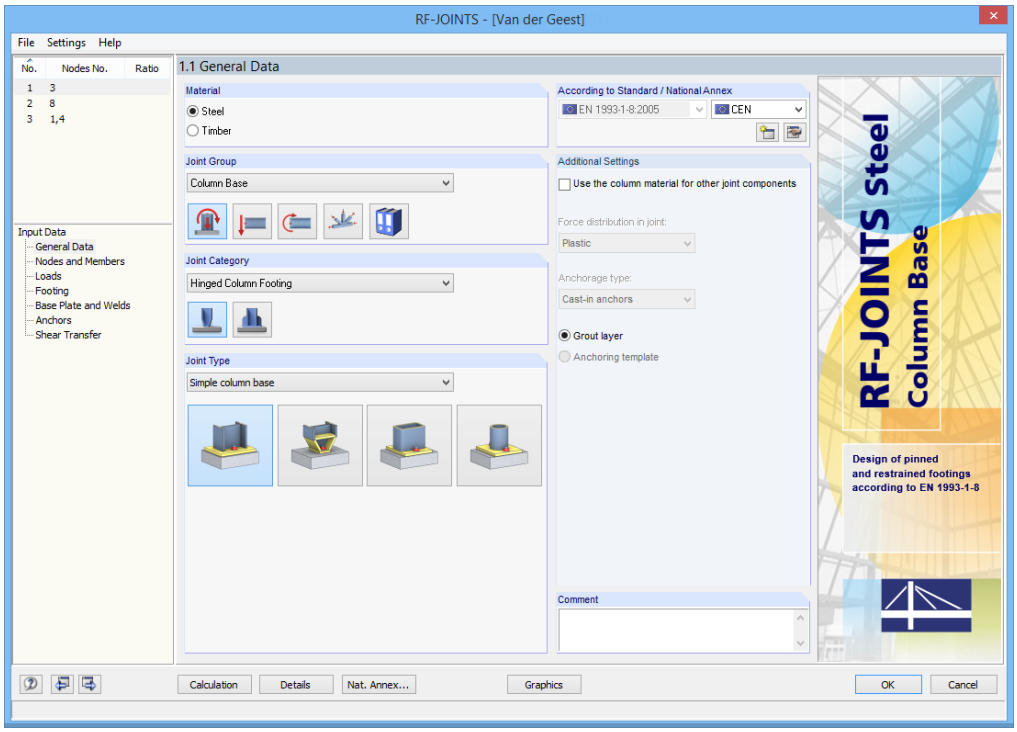


Image 2.1 Window *1.1 General Data*

Material

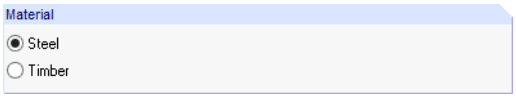


Image 2.2 Filter for steel and timber connections

In the *Material* section, you can define whether you want to design a steel or timber connection. This setting controls the available options in the other window sections.

Joint Group

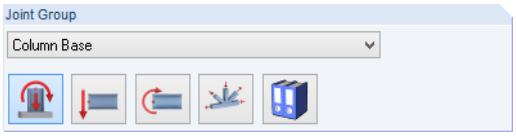


Image 2.3 Filter for joint groups for steel structures

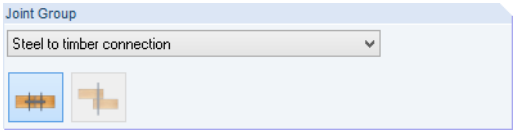


Image 2.4 Filter for joint groups for timber structures

This section manages the type of connection. You can select the desired joint group by using the drop-down list or clicking the buttons with the connection icons.

The green dots in the icons illustrate which connection modules are in your license.

Joint Category

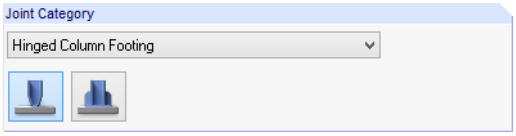


Image 2.5 Filter for joint categories of column footings

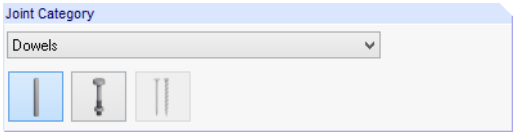


Image 2.6 Filter for joint categories of steel-to-timber connections

You can use the drop-down list or the buttons to select the relevant joint category.

Joint Type

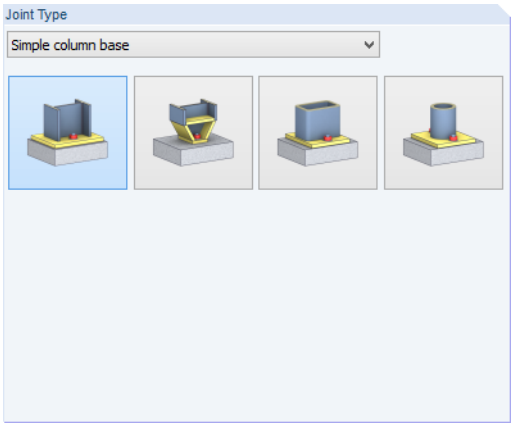


Image 2.7 Filter for pinned column base joint types

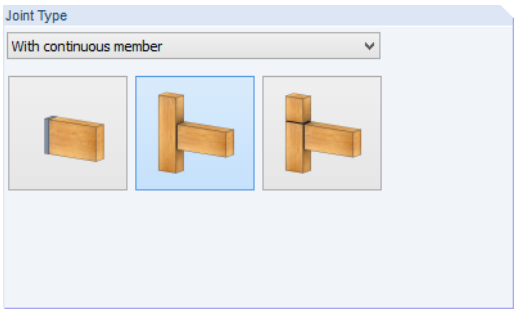


Image 2.8 Filter for dowel joint types

In this window section, you can specify the exact type of the joint. You can select it by using the drop-down list or the connection icon buttons.

According to Standard / National Annex

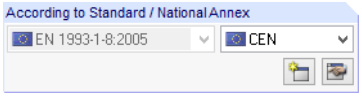


Image 2.9 Standard and National Annex

Steel connections are automatically designed according to EN 1993-1-8 [1] and timber connections according to EN 1995-1-1 [2]. Steel-to-timber connections can also be designed according to ANSI/AWC NDS-2018[5]. In the list to the right of the standard, you can select the National Annex whose parameters you want to apply to the designs.

The button opens a dialog box where you can check the parameters of the selected National Annex. This dialog box is described in Chapter 2.4.

Comment

In this text box, you can enter a user-defined note to describe the current joint, for example.

2.2 Nodes and Members

In the second window, you can select the nodes you want to analyze. In addition, you can define the parameters of the members connected to the nodes.

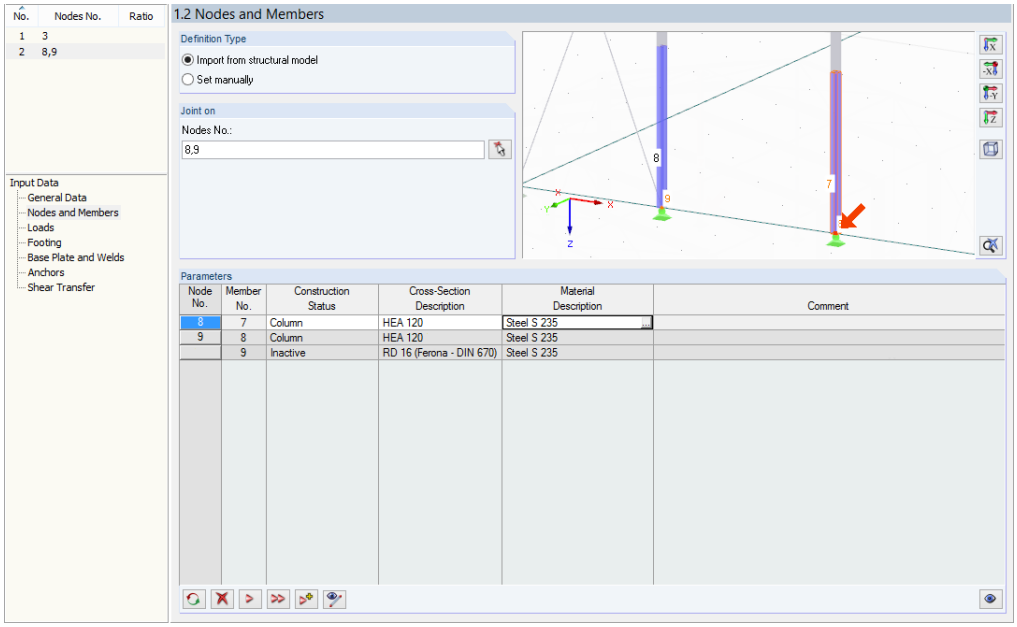


Image 2.10 Window 1.2 Nodes and Members

Definition Type

You can import nodes using the *Import from structural model* option. The geometric parameters such as the number and properties of the connected members are automatically imported from RFEM/RSTAB. As an alternative, the joint can be *Set manually*.

Joint on Nodes No.

In this text box, you can directly enter the numbers of the nodes to be analyzed. You can use the [Select] button to define the nodes graphically in the RFEM/RSTAB work window.

Multiple selection of nodes is only useful if they have identical input parameters. If there are different design conditions, it is not possible to combine the nodes in this window. In such a case, you have to create a new design case: To do so, click **File** ⟶ **New Case** in the menu or use the > and >> buttons in the *Parameters* section below (see description for the following section).

If you select *Set manually*, the import of data from RFEM or RSTAB is disabled. A note that says *User-defined* appears in the text box. Then you can enter the *Parameters* independently of the model.

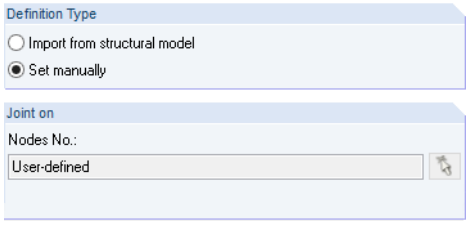


Image 2.11 Defining a joint manually

Parameters

In this window section, you can manage the properties of the structural components connected to the nodes specified above.

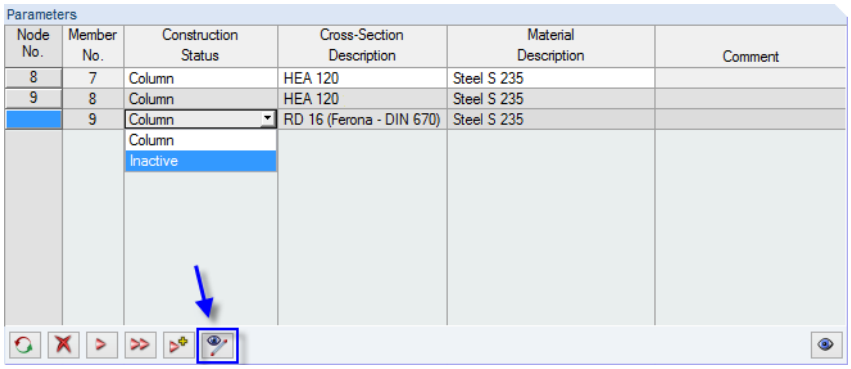
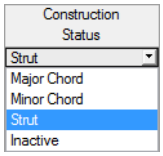


Image 2.12 *Parameters* dialog section with [Show or Hide Inactive Members] button

Construction Status



In this column, you can classify members for the design. The available options depend on the selected joint group and joint category.

Cross-Section Description

When importing nodes from RFEM/RSTAB, the member cross-sections are preset.

To change a cross-section, click its entry in this column to activate the text box. To open the cross-section library, use the ... button in the text box (see Figure 2.10) or press [F7].

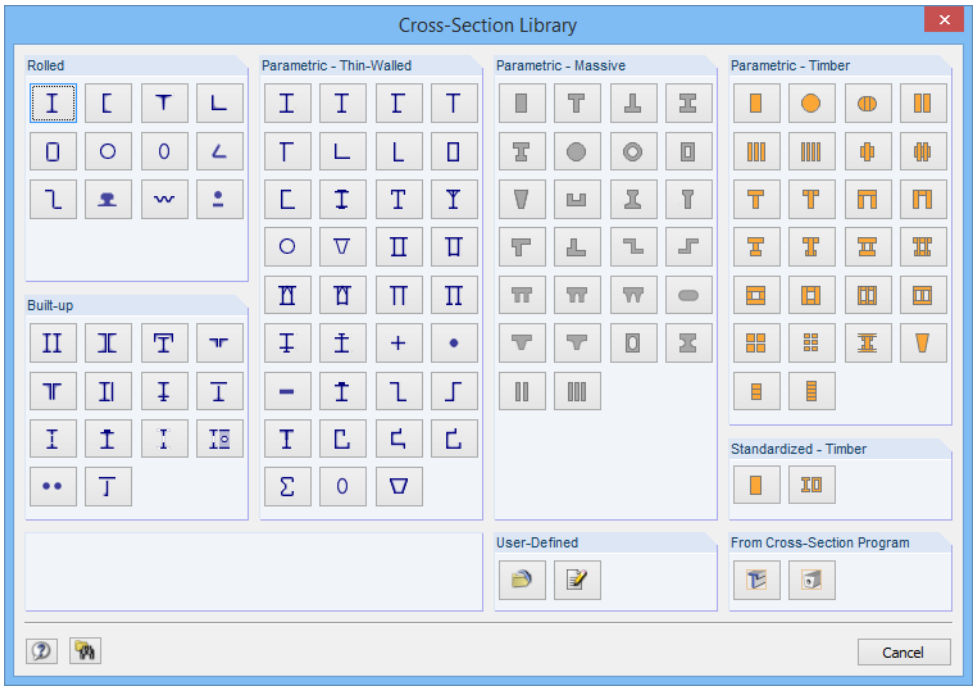


Image 2.13 Cross-section library

Chapter 4.13 of the RFEM manual, or Chapter 4.3 of the RSTAB manual, describes how to select cross-sections from the library.

Modified cross-sections are highlighted in blue.

Material Description

When importing nodes from RFEM or RSTAB, the cross-section materials are preset as well.

To change materials, click the entry in this column to activate the text box. To open the material library, use the ... button in the text box (see Figure 2.10) or press [F7].

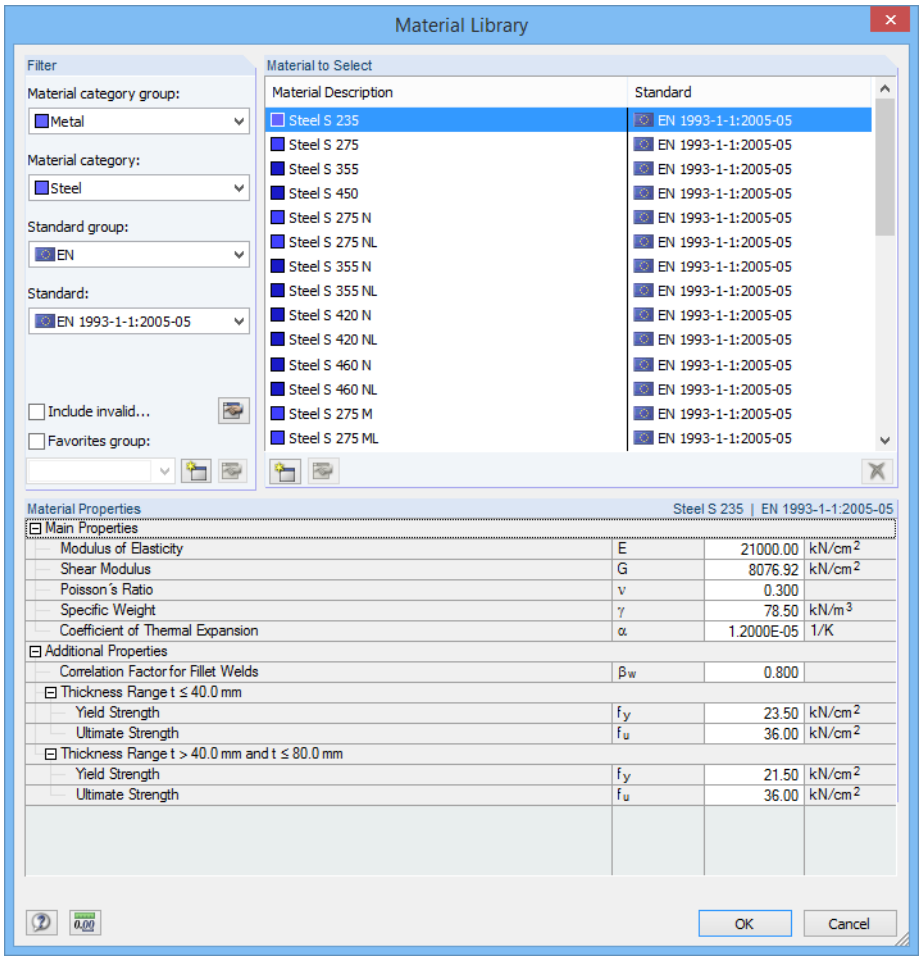


Image 2.14 Material library

Chapter 4.3 of the RFEM manual, or Chapter 4.2 of the RSTAB manual, describes how to select materials from the library.

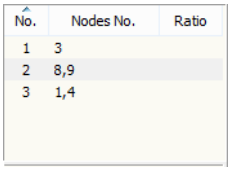
The buttons in the *Parameters* section have the following functions:

Button	Function
	Sets the default values for the connection
	Deletes the node selected in the table above
	Transfers the selected node to a new design case
	Transfers all unsuitable nodes to a new design case
	Sets the connection type for all nodes of the design case

	Shows or hides inactive members in the table
	Shows the RFEM/RSTAB work window for changing the view

Table 2.1 Buttons in the *Parameters* section

The buttons > and >> have an important function: If you want to design several nodes with different boundary conditions, such as the number of connected members or the member cross-sections, these buttons allow for a manual or automatic assignment to new design cases. You can then define the parameters for specific nodes in the different design cases. Alternatively, you can also click **File** ⟶ **New Case** in the menu.



The design cases including the data related to the nodes are displayed at the top of the navigator. Window *1.2 Nodes and Members* always displays the parameters of the nodes selected in the navigator list. To switch the design case, simply click the relevant entry in the list.

Graphic window

The graphic window presents an overview of the connections to be designed. It shows a dynamic partial view of the model. The node selected in the *Parameters* section is indicated by an arrow; the connected members are highlighted.

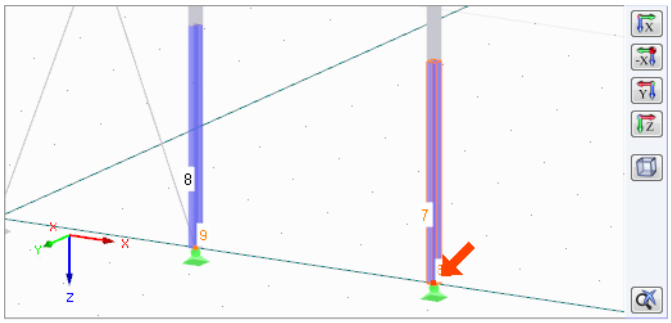


Image 2.15 Graphic with selection arrow

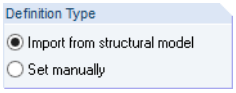
You can control the graphic by using the same mouse functions as in RFEM or RSTAB in order to zoom, move, or rotate the view.

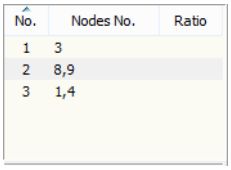
The buttons next to the graphic have the following functions:

Button	Function
	Shows the view in the direction of the X-axis
	Shows the view in the opposite direction of the X-axis
	Shows the view in the direction of the Y-axis
	Shows the view in the direction of the Z-axis
	Displays the isometric view
	Resets to full view of the graphic

Table 2.2 Buttons in graphic window

2.3 Loads and Internal Forces





The structure of Window 1.3 depends on the *Definition Type* selected in Window 1.2 (see Figure 2.10).

The input in this window always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

2.3.1 Loads

If you select the *Import from structural model* definition type in Window 1.2, you have to define the load cases or load combinations including internal forces to be applied for the design in Window *1.3 Loads*.

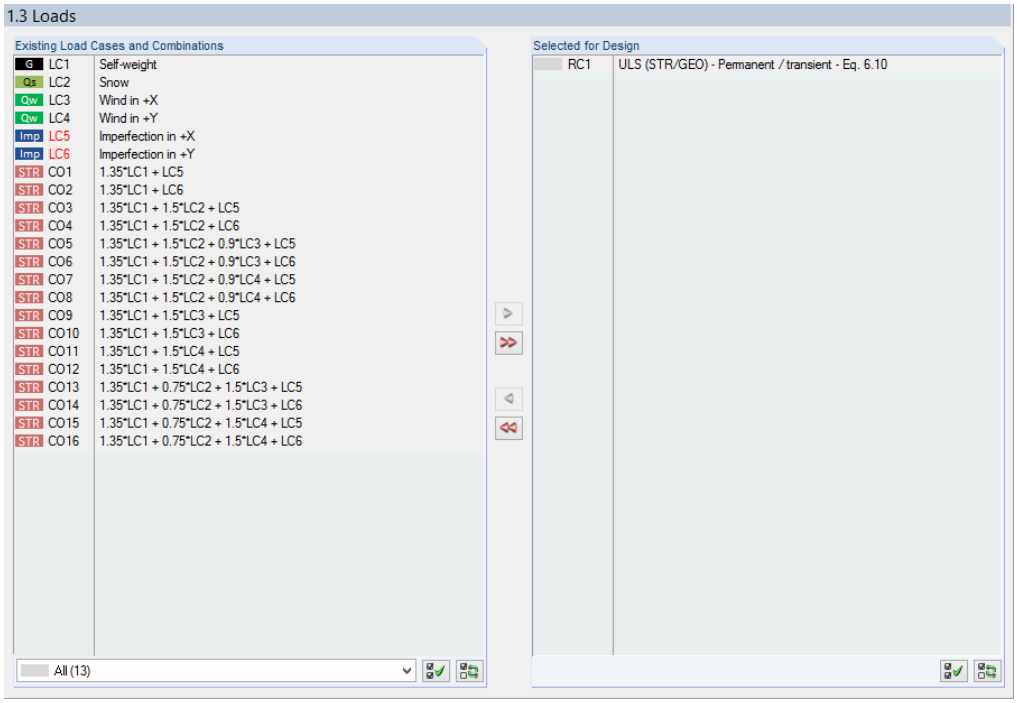
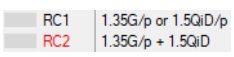


Image 2.16 Window *1.3 Loads*

Existing Load Cases and Combinations

This column lists all the load cases, load combinations, and result combinations that have been created in RFEM or RSTAB. RF-/DYNAM Pro combinations can also be analyzed.

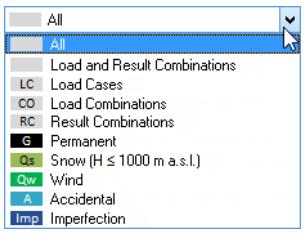
The design of a joint requires a clear constellation of internal forces. For the max and min values of a result combination, this is the case if the internal forces are superimposed with the criteria 'Permanent' and 'Or', if necessary. It is therefore not possible to design RCs in which one or more actions are classified as 'Variable'.



To transfer selected entries to the *Selected for Design* list on the right, click the > button. You can also double-click the items to transfer them. To transfer the entire list to the right, use the >> button.

To select multiple load cases, click them while holding down the [Ctrl] key, as is usual in Windows applications. This allows you to transfer several load cases at once.

Load cases marked in red, such as LC 5 or LC 6 in Figure 2.16, cannot be designed: This is the case for load cases without loads, imperfection load cases, or invalid RCs (see above). A corresponding warning appears when you transfer them.



Several filter options are available below the list. They make it easier to assign the entries sorted by load cases, combinations, or action categories. The buttons have the following functions:

	Selects all load cases in the list.
	Inverts the selection of load cases.

Table 2.3 Buttons in the *Existing Load Cases and Combinations* section

Selected for Design

The column on the right lists the load cases, load combinations, and result combinations selected for the design. To remove selected items from the list, use or double-click them. To empty the entire list, click .

You can select several items at once by holding down [Ctrl] and clicking them.

2.3.2 Internal Forces

If you have selected the *Set manually* definition type (see Chapter 2.2), you have to specify the internal forces you want to use for the design in Window *1.3 Internal Forces*.

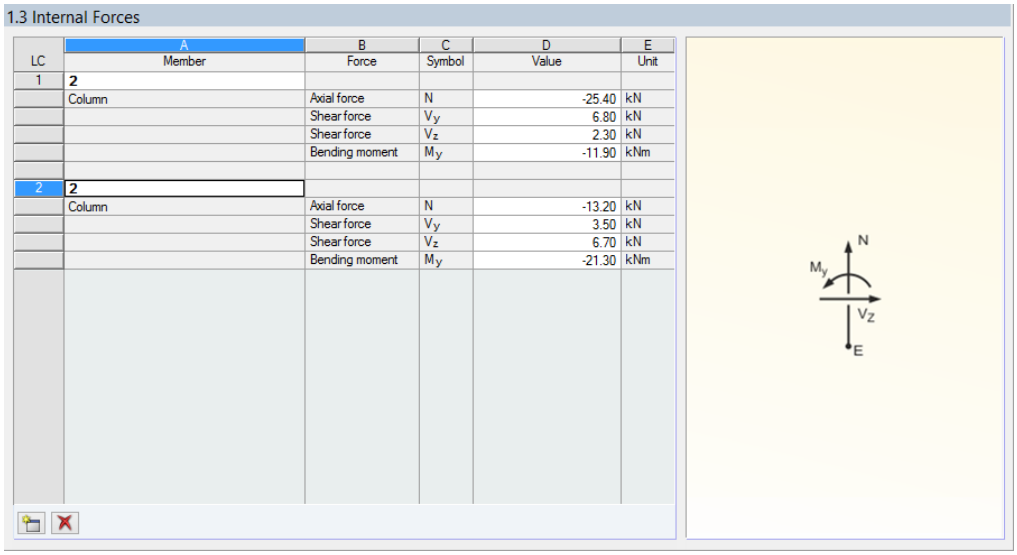


Image 2.17 Window *1.3 Internal Forces*

LC

The internal forces are managed in a load case. When you open the window for the first time, LC No. 1 is preset.

You can use the button to create a new load case. With the help of load cases, you can specify different constellations of internal forces for the design.

Member

In this column, you can enter a member number that facilitates the assignment of internal forces.

Force / Symbol

These two columns describe which type of internal force is respectively applied.

Value / Unit

Here you can enter the values of the internal forces.

You can adjust the units and decimal places of the internal forces by using **Settings → Units and Decimal Places** in the menu (see Chapter 15.2).

The buttons below the table have the following functions:

	Creates a new load case for additional internal forces
	Deletes the load case selected above

Table 2.4 Buttons in Window *1.3 Internal Forces*

2.4 National Annex

In the upper-right part of Window *1.1 General Data*, you can select the National Annex whose parameters you want to use for the design (see Figure 2.9). Click [Edit] to check the preset parameters. The content of the dialog box depends on the specified material (steel or timber connections).

Steel connections

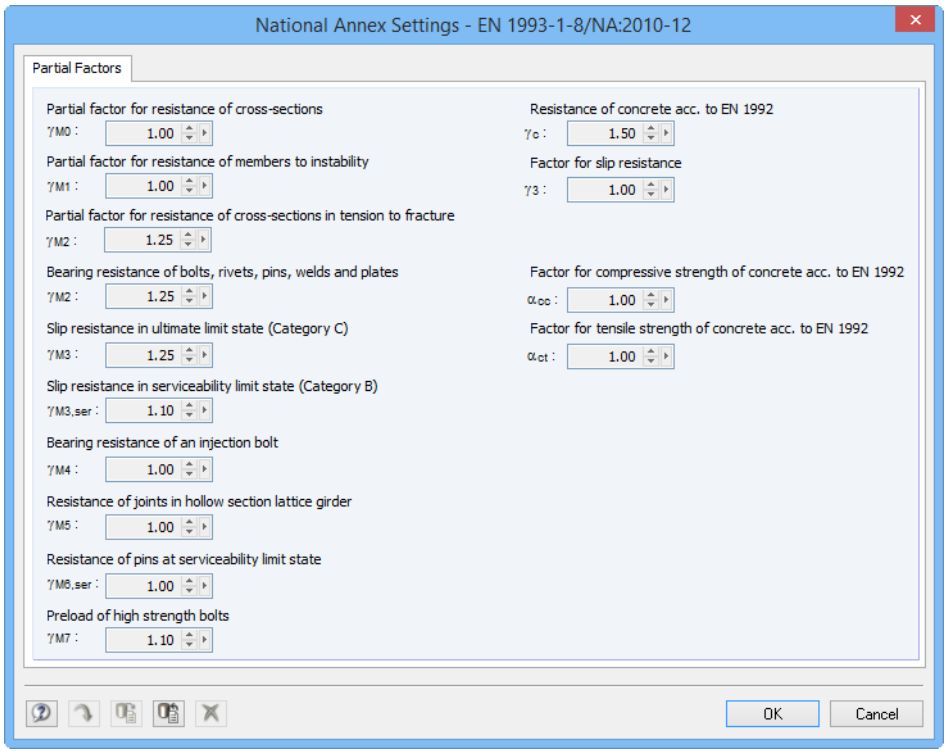


Image 2.18 *National Annex Settings - EN 1993-1-8/NA:2010-12* dialog box

Timber connections

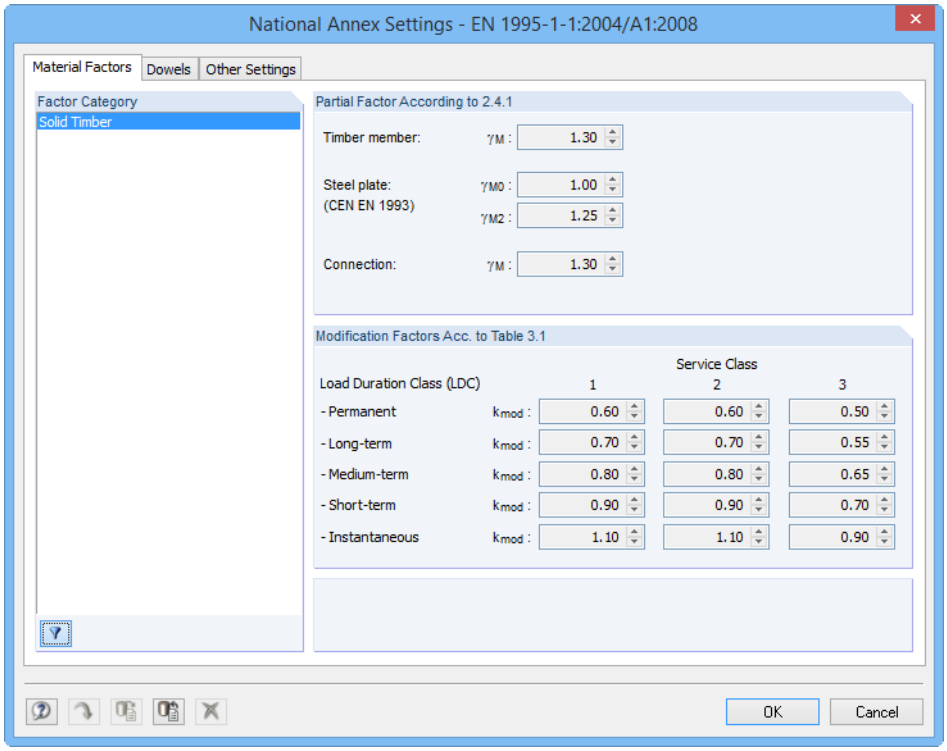


Image 2.19 *National Annex Settings - EN 1995-1-1:2004/A1:2008* dialog box

These dialog boxes display various factors of the National Annex such as the material factor, modification factor, and shear correction factor. Generally, you cannot change the values as they are codified in the National Annexes.

You can click [New] to create a user-defined National Annex where you can freely define the factors.

The [Nat. Annex] button is available in every input window. You can use it to open the *National Annex Settings* dialog box as well.

The buttons in the *National Annex Settings* dialog box have the following functions:

Button	Function
	Restores the program's default settings
	Imports user-defined default settings
	Saves modified settings as default
	Deletes user-defined National Annex

Table 2.5 Buttons in the *National Annex Settings* dialog box

3 Steel - Column Base

This chapter describes the windows that are particularly relevant for the **RF-JOINTS Steel - Column Base** module. The general input parameters are described in Chapter 2.

You can find two design examples for RF-/JOINTS Steel - Column Base in our Knowledge Base:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001430
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001468

The input windows of the add-on module are accessible once you select the material *Steel* and the joint group *Column Base*.

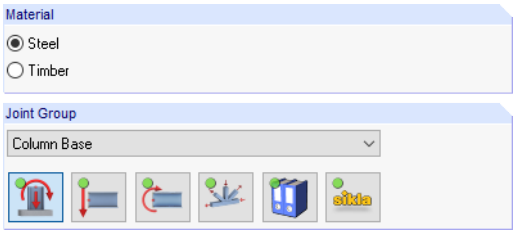
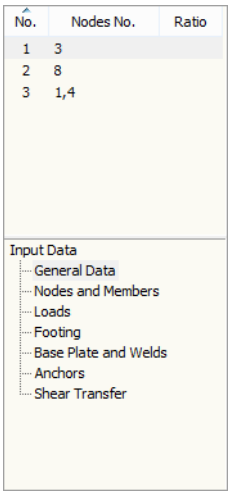


Image 3.1 *RF-JOINTS Steel - Column Base add-on module*



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the entries *Footing*, *Base Plate*, *Anchors*, etc. are missing in the navigator, go to Window *1.2 Nodes and Members* and check if the boundary conditions required to input the column base are correct. For example, it may be necessary to deactivate connected members for the design (see Figure 3.8).

The input windows of the *RF-JOINTS Steel - Column Base* module are divided into two parts: On the left you can find the input parameters of the footing; they are illustrated by graphics on the right (see Figure 3.9). The upper graphic shows a system sketch of the current parameter, the lower graphic shows a 3D visualization of the column base model.

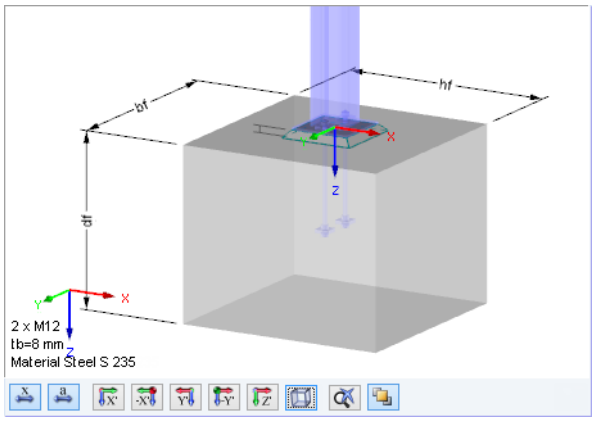


Image 3.2 3D visualization of column base

The buttons below the 3D graphic are described in the following table.

Button	Function
	Shows or hides the dimensioning
	Displays the values or symbols of the dimensioning
	Shows the view in the direction of the X-axis
	Shows the view in the opposite direction of the X-axis
	Shows the view in the direction of the Y-axis
	Shows the view in the direction of the Z-axis
	Sets the isometric view
	Resets to full view of the graphic
	Shows or hides irrelevant parts of the window

Table 3.1 Buttons of 3D graphic

3.1 General Data

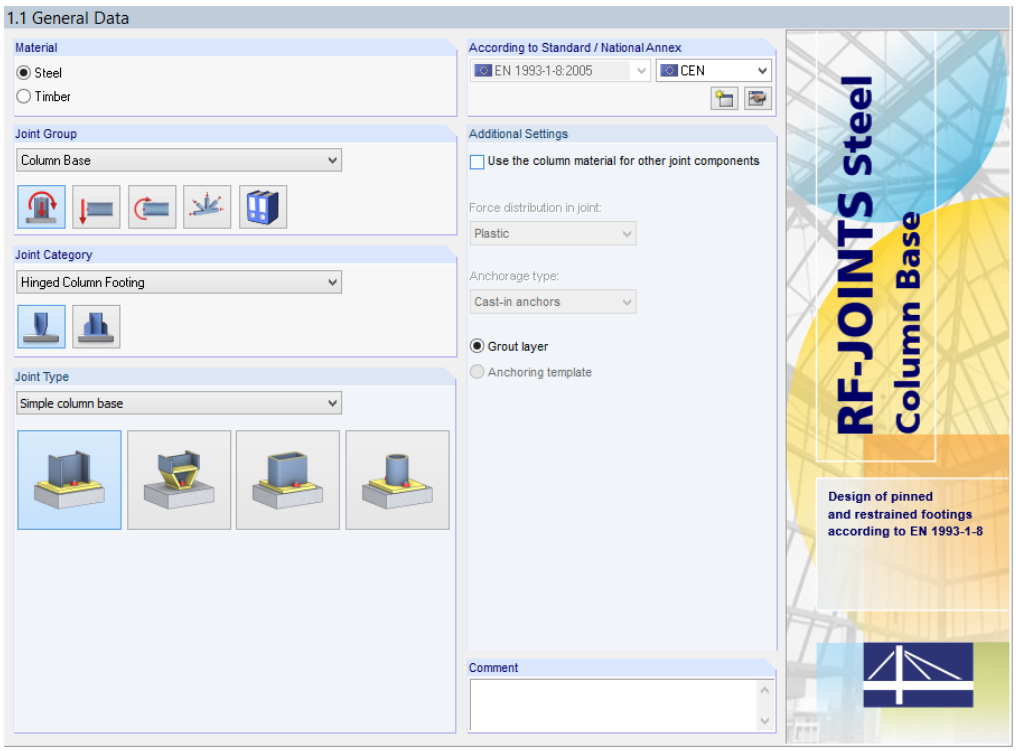
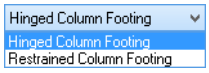


Image 3.3 Window 1.1 General Data

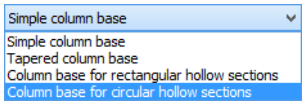
Joint Category

Image 3.4 Joint category



You have to specify whether a *Hinged* or *Restrained* column base is present. You can select the category by using the drop-down list or clicking the buttons with the connection icons.

Joint Type



The available options depend on the joint category.

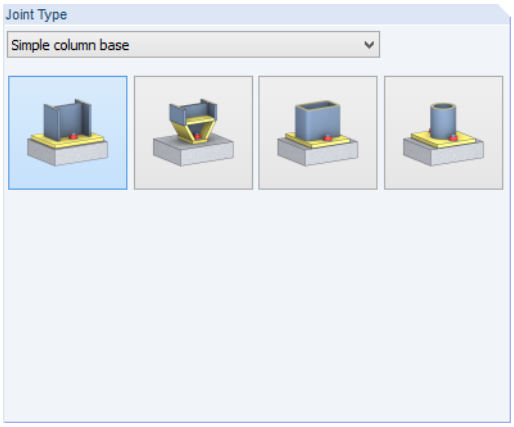


Image 3.5 *Joint Type* section for *Hinged Column Footing* category

The *Hinged Column Footing* category provides the following design variants:

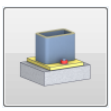
	Column base plate without stiffening
	Tapered column base
	Column base plate for rectangular hollow sections
	Column base plate for circular hollow sections

Table 3.2 Joint types for hinged column footings

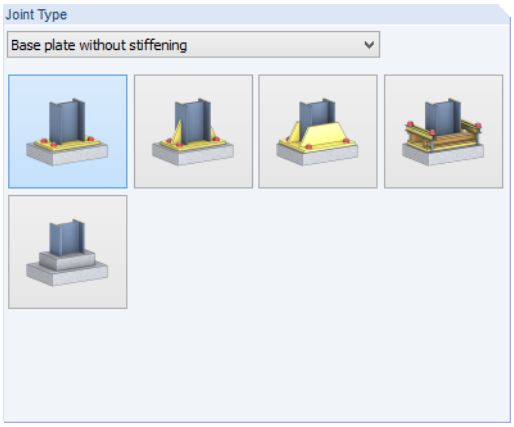


Image 3.6 *Joint Type* section for *Restrained Column Footing* category

The *Restrained Column Footing* category provides the following design variants for I-sections:

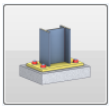
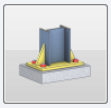
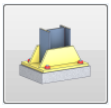
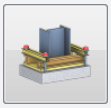
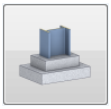
	Base plate without stiffening
	Base plate with stiffeners in center of flanges
	Base plate with stiffeners on both sides of column
	Base plate with channel sections on both sides of column and web members
	Bucket footing

Table 3.3 Joint types for restrained column footings

Additional Settings

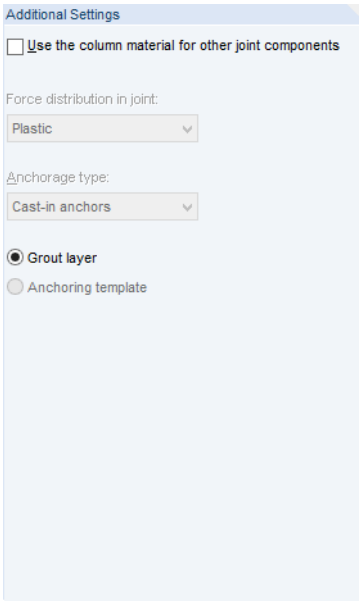


Image 3.7 *Additional Settings section*

When you select the *Use the column material for other joint components* check box, the material of the column cross-section is automatically used for the base plate, anchors, and shear keys. In this case, you cannot define the materials separately.

A plastic *Force distribution in joint* is assumed. Cast-in anchors are preset as the *Anchorage type*.

The connection between the steel base plate and foundation can be established by means of a *Grout Layer* or a cast-in *Anchoring Template* (in preparation).

3.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2.

In the *Parameters* section, you can check the boundary conditions of the connected components. If there is an *Invalid cross-section*, you should adjust the cross-section series to match the joint type set in Window 1.1.

If several members such as diagonals connect to a column base node, the redundant members can be set to be *Inactive*.

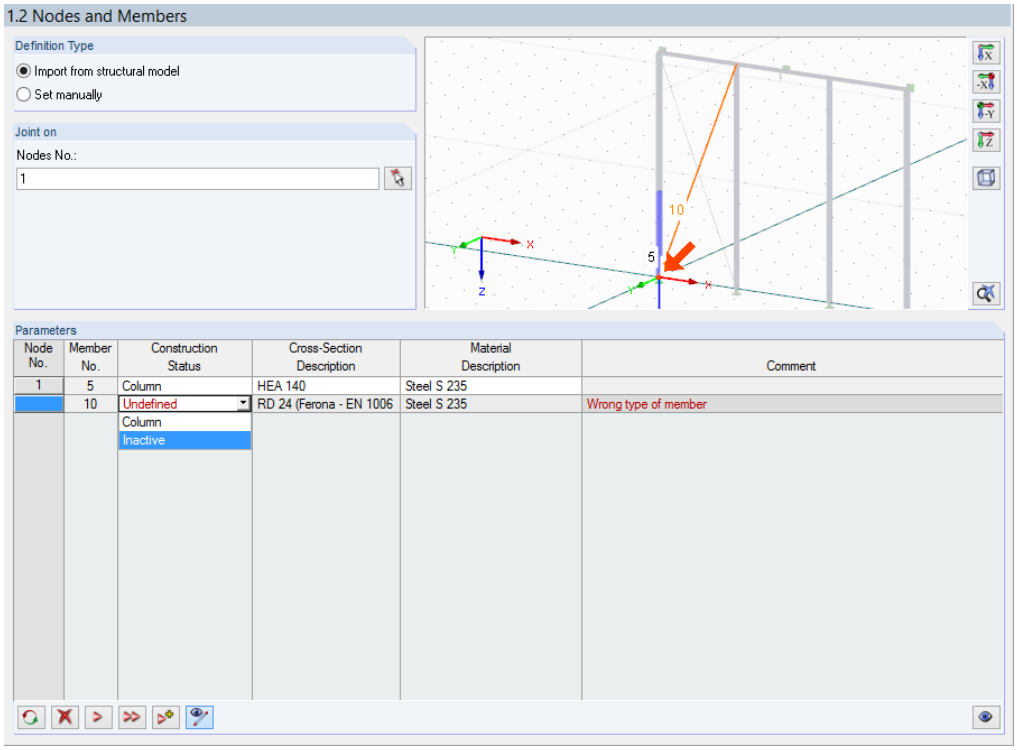


Image 3.8 Setting diagonal member *Inactive*

3.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

3.4 Footing

In Window *1.4 Footing*, you can specify the foundation parameters and define the position of the base plate.

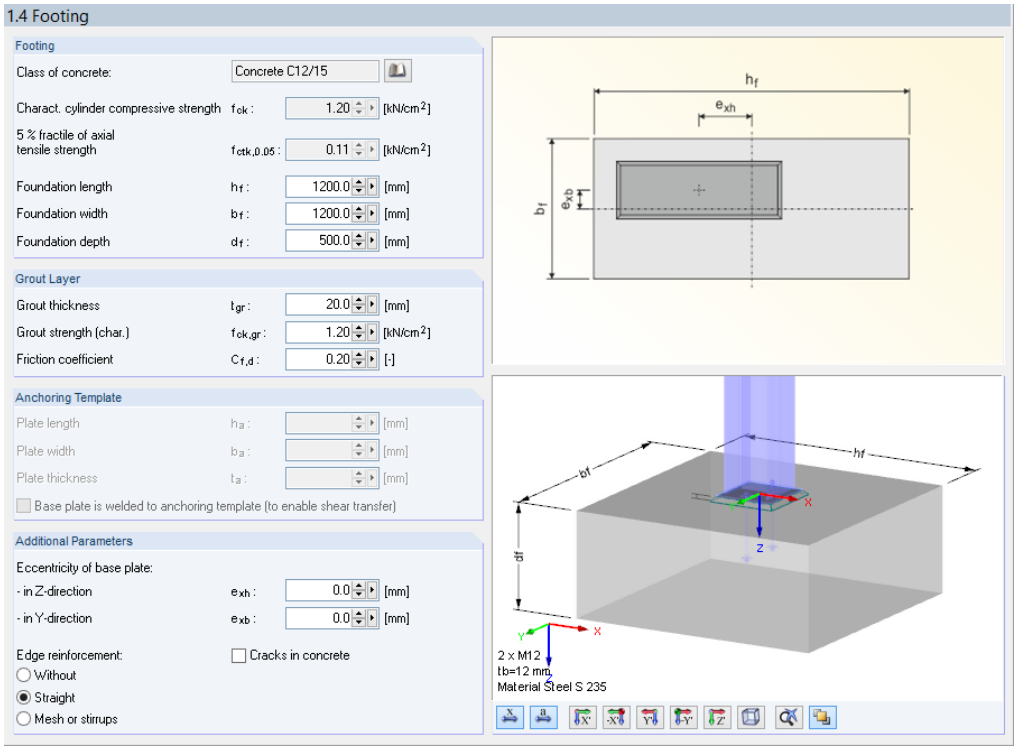


Image 3.9 Window *1.4 Footing*

Footing

In this window section, you can define the material and dimensions of the foundation. You can use the button to open the material library where you can find the properties of various types of concrete.

Grout Layer

Here you can define the grout thickness, the characteristic grout strength, and the friction coefficient of the grout layer between the base plate and the foundation.

Anchoring Template (in preparation)

If you have selected the cast-in anchors option in Window *1.1 General Data*, you can define the dimensions of the anchor plate in this section. You can also set a welded connection of the anchoring template and base plate.

Additional Parameters

Using the parameters of the *Eccentricity of base plate*, you can arrange the base plate eccentrically on the foundation.

In this section, you can also specify whether there is an *Edge reinforcement* and what its form is.

The *Cracks in concrete* check box controls whether the calculation is performed with the concrete in the cracked state.

Restrained Column Footing - Bucket Footing

If you have selected the *Bucket footing* joint type in Window *1.1 General Data* (see Figure 3.6), Window *1.4 Footing* looks as follows:

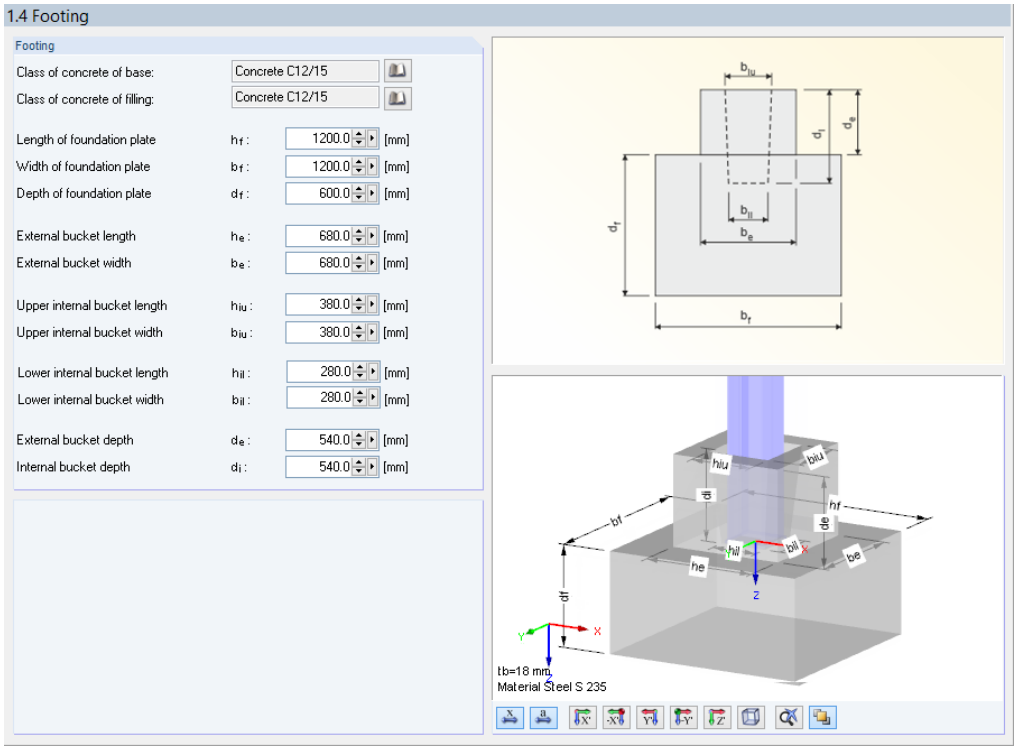


Image 3.10 Window *1.4 Footing* for the *Bucket footing* joint type

Click to open the material library where you can select the materials of the foundation.

You can enter the dimensions of the foundation plate and the bucket in the individual text boxes.

3.5 Base Plate and Welds

In Window *1.5 Base Plate and Welds*, you can define the base plate parameters.

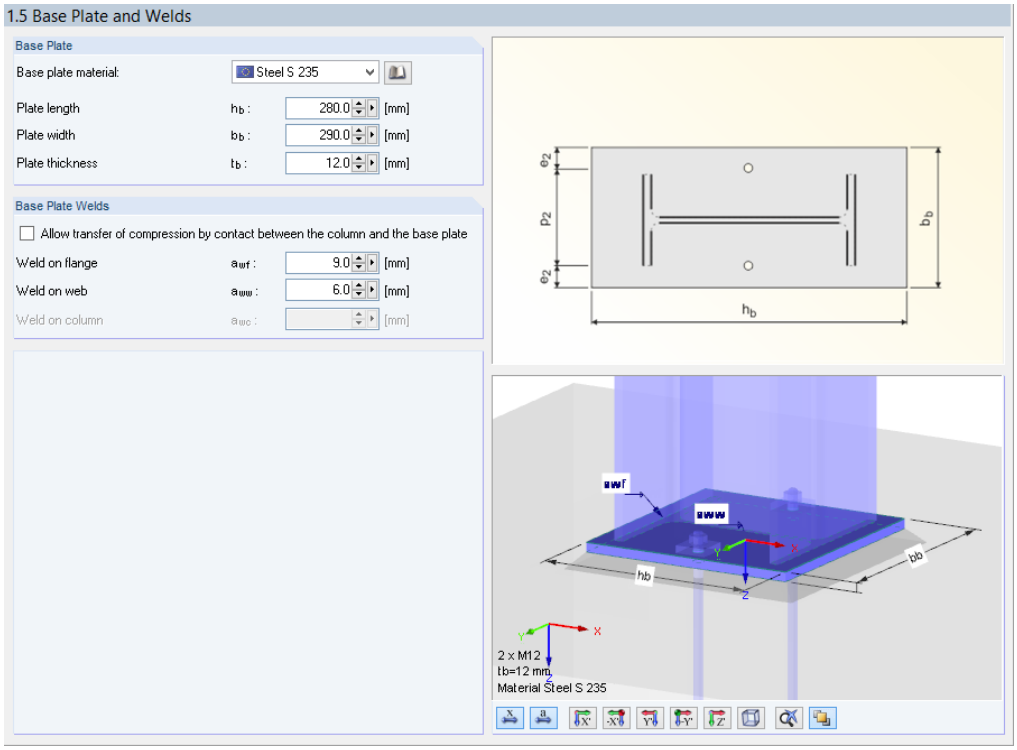


Image 3.11 Window *1.5 Base Plate and Welds*

Base Plate

In this window section, you can define the material and dimensions of the base plate. You can use the button to open the material library where you can find the properties of various steel grades.

Base Plate Welds

The *Allow transfer of compression by contact between the column and the base plate* check box allows you to reduce the weld loading; the design force must be a compression force in this case. Tension forces are transferred by the welds without exception.

The welds of the column cross-section to the base plate are applied as shown in the sketch at the top right of the window. You can enter the parameters for the *Weld on flange* and *Weld on web*. For rectangular and circular hollow sections, you can define the *Weld on column*.

During the calculation, the module also checks structural details. If, for example, the selected weld thicknesses are too large for the base plate dimensions, a corresponding message appears.

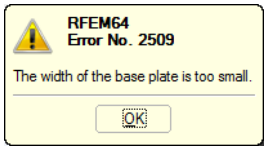


Image 3.12 Conflict in structural details

Restrained Column Footing - Bucket Footing

If you have selected the *Bucket footing* joint type in Window *1.1 General Data* (see Figure 3.6), the name of Window 1.5 is *Column* and it looks as follows.

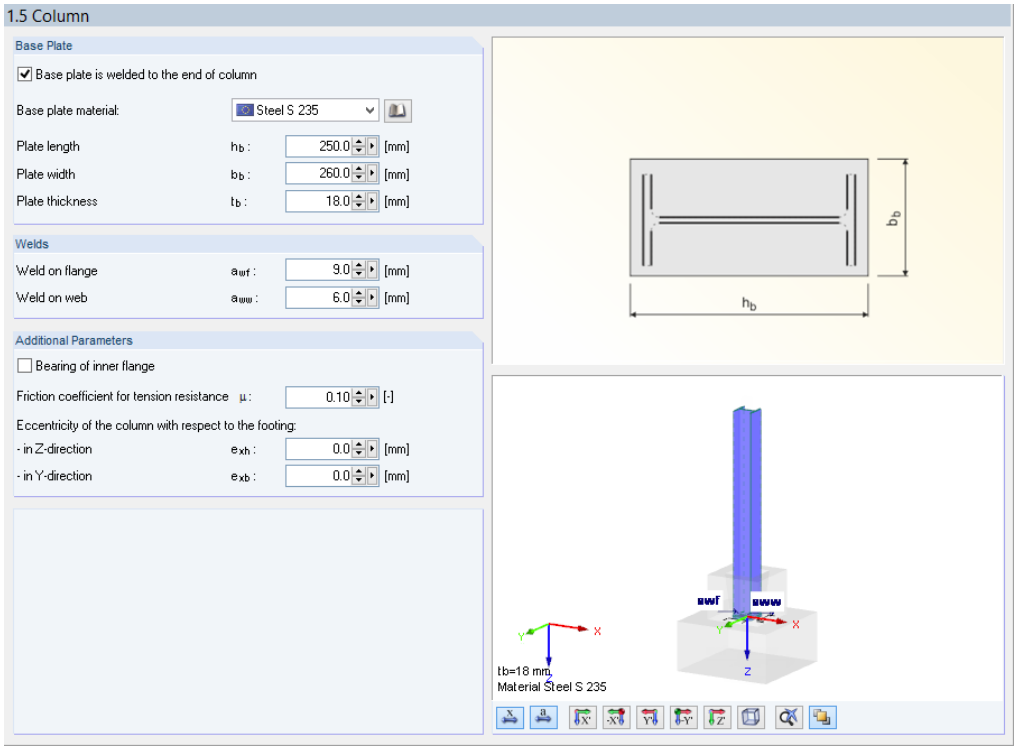


Image 3.13 Window *1.5 Column* for the *Bucket footing* joint type

The *Base plate is welded to the end of column* check box determines if there is a column base plate. If that is the case, you can enter the parameters of the base plate and weld as described above.

You can impact the calculation with the *Bearing of inner flange* check box. By selecting it, the program takes the elastic compression of the concrete into account. This affects the load introduction length.

If necessary, you can adjust the preset value of the *Friction coefficient* μ for the tension resistance.

The parameters of an *Eccentricity of the column* allow you to determine the eccentric arrangement of the column in the foundation.

3.6 Anchors

Window *1.6 Anchors* manages the parameters of column base anchors.

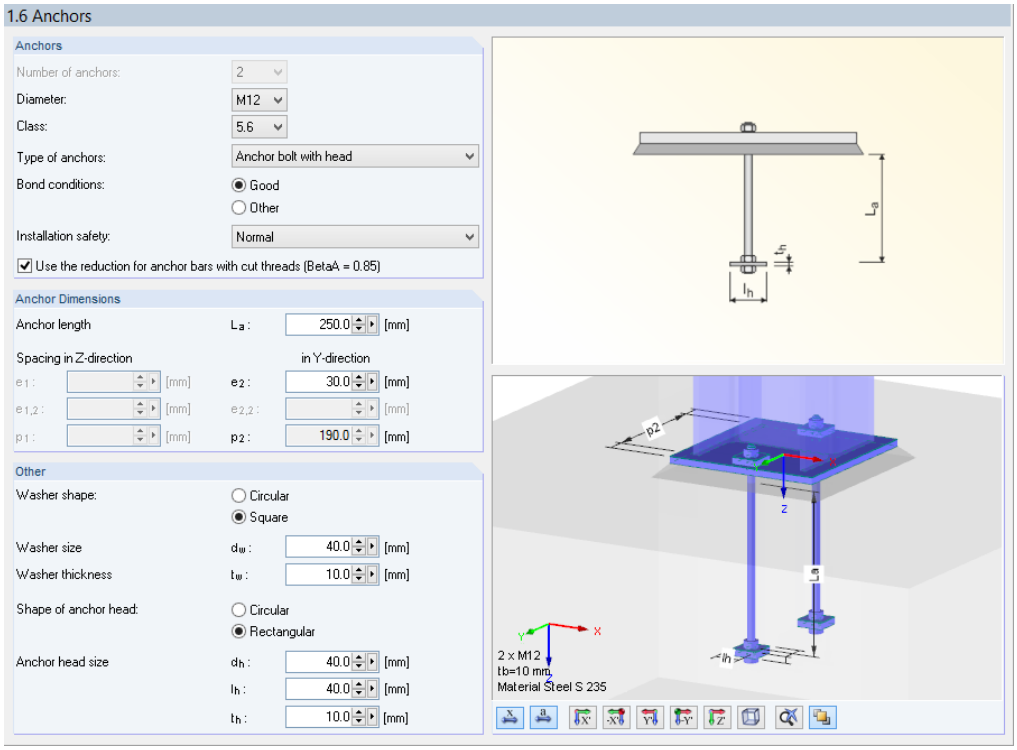
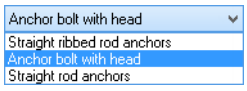


Image 3.14 Window *1.6 Anchors*

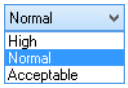
Anchors

The *Number of anchors* is currently fixed with two anchors (hinged column footings) or four anchors (restrained column footings).

You can specify the *Diameter* and the strength *Class* of the anchors by using the drop-down lists.



There are three variants for the *Type of anchors* available. They affect the design resistance when anchors are pulled out under tensile stress.



The partial safety factors for the design are controlled with the settings for the *Bond conditions* and *Installation safety*.

Depending on the design of the anchor bolts, a *reduction* of the determined design resistance by the factor $\beta_A = 0.85$ can be considered according to [1], Table 3.4 (see [1], 3.6.1(3)).

Anchor Dimensions

You can adjust the arrangement of the anchors on the base plate by using the *Anchor length* parameter and entering the *Spacing* from the edges of the base plate.

Other

In this section, you can specify the shape, size, and thickness of the *Washers* and *Anchor heads*. The graphic on the right provides a dynamic visualization of the input parameters.

3.7 Shear Transfer

In Window *1.7 Shear Transfer*, you can enter the shear key parameters.

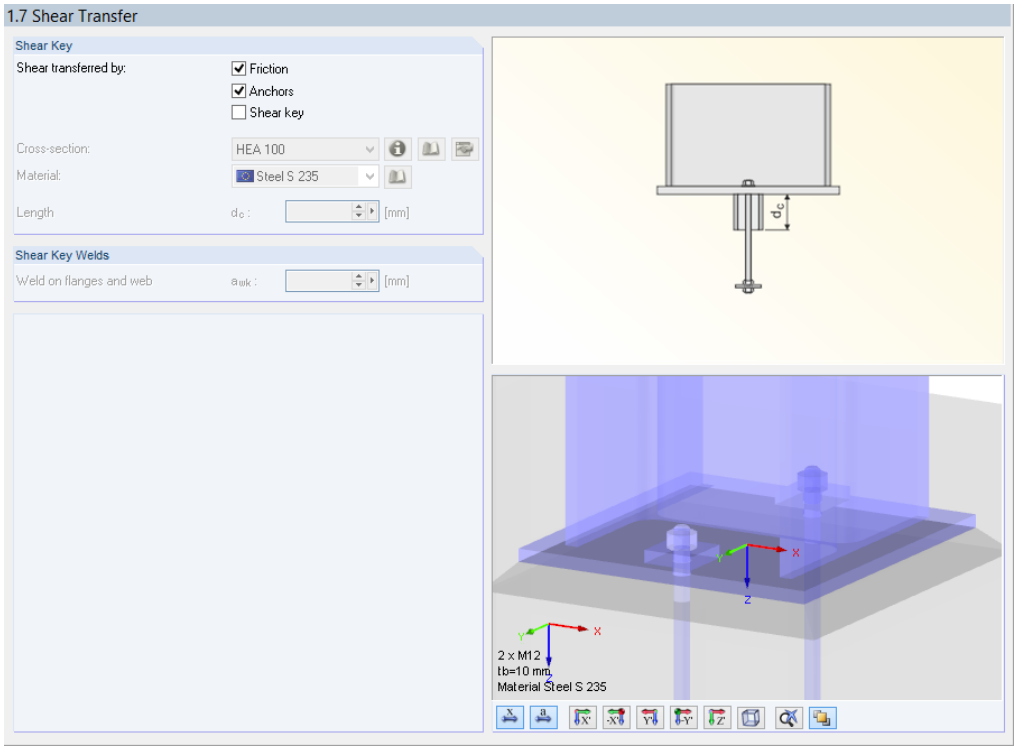


Image 3.15 Window *1.7 Shear Transfer*

Shear Key

There are three options available for *Shear transfer*, but they cannot be combined at will: Friction, Anchors, and Shear key. Select the corresponding check box to increase the shear failure resistance. The friction resistance is considered if there is a compressive force. It reduces the existing shear force so that anchors or shear keys have to absorb less shear.

When using a shear key, you can select a cross-section from the list that is available in the RFEM/RSTAB model. You can use the button to define a different cross-section. shows the properties of the cross-section. You can change the cross-section retroactively using .

You can define a separate material for the shear key using the button (provided that the *Use the column material for other joint components* check box has not been activated in Window *1.1 General Data*). Thus, the materials of the column, base plate, and shear key do not have to be identical.

You can specify the length of the shear key in the corresponding text box.

Shear Key Welds

In this window section, you can enter the thickness of the circumferential weld that connects the shear key with the base plate.

3.8 Stiffeners

Window *1.8 Stiffeners* is available for the following joint types:

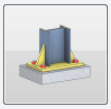
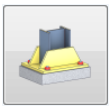
	Tapered column base
	Restrained column base plate with stiffeners in center of flanges
	Restrained column base plate with stiffeners on both sides of column

Table 3.4 Joint types with stiffeners

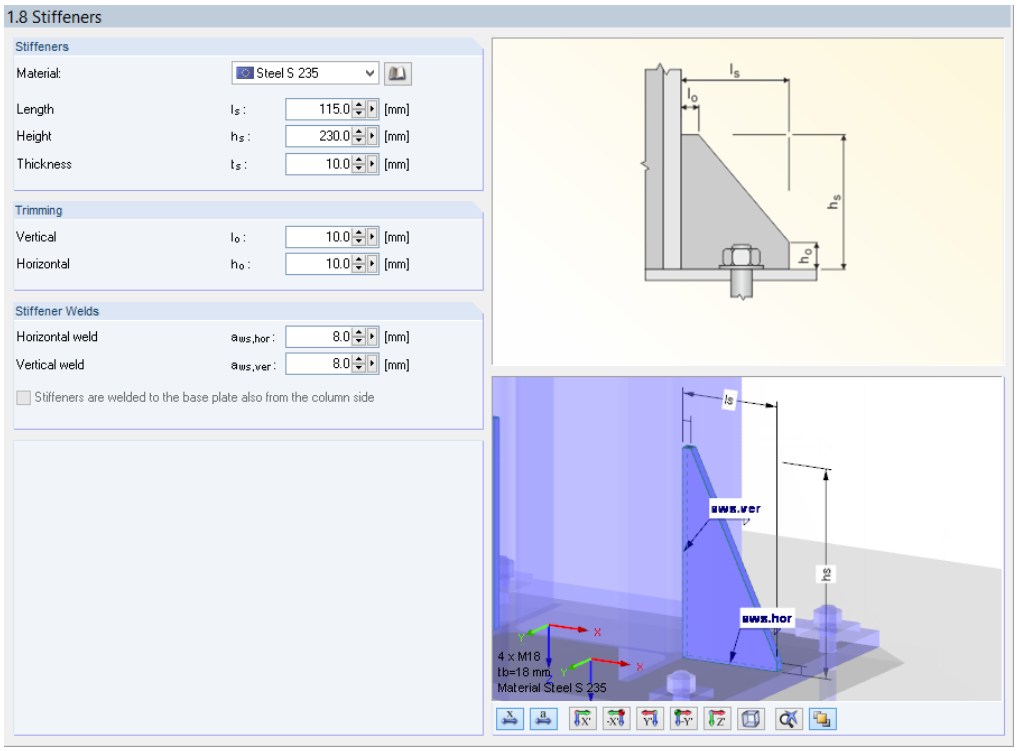
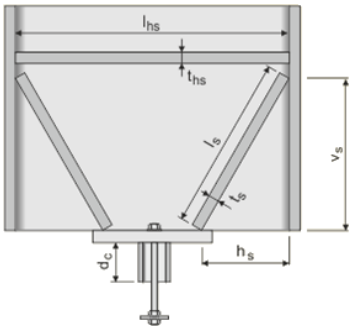


Image 3.16 Window *1.8 Stiffeners*

Stiffeners

In this window section, you can define the geometric parameters of the stiffeners. You can use the button to define the material separately, unless the *Use the column material for other joint components* check box in Window *1.1 General Data* has been selected.



Trimming

For restrained column base plates with stiffeners, you can enter the length of the vertical and horizontal trimmings.

Stiffener Welds

In this section, you can define the thicknesses of the stiffener welds depending on the joint type. The individual parameters are illustrated in the 3D graphic on the right.

Horizontal Stiffener Welds

For hinged column footings with stiffeners, you can define a horizontal stiffener in this section. Select the check box to activate the text boxes where you can define the thickness and the weld of the stiffener in accordance with the sketch.

Restrained Column Footing - Base Plate with Channel Sections (Stiffeners/Crossbars)

If you have selected the *Base plate with channel sections* joint type in Window *1.1 General Data* (see Figure 3.6), the name of Window 1.8 is *Stiffeners and Crossbars* and it looks as follows.

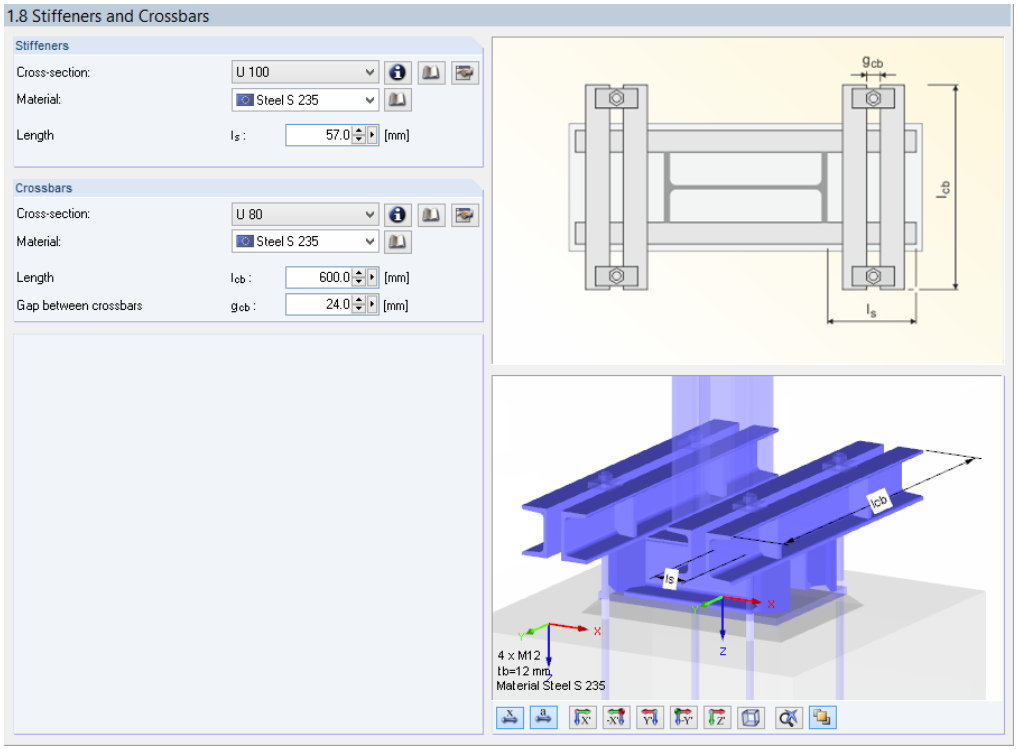


Image 3.17 Window *1.8 Stiffeners and Crossbars* for *Base plate with channel sections* joint type

In the two lists, you can select the *Cross-section* of the reinforcing channel sections. You can use the button to define a different cross-section. To subsequently modify the section, click .

The *Material* of the cross-sections can be selected in a library by using the buttons.

The *Length* and *Gap between crossbars* text boxes specify the geometric parameters.

4 Steel - Pinned

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Steel - Pinned** module for designing shear connections of I and H beams. The general input parameters are described in Chapter 2.

The functionality of this add-on module is presented in a Dlubal webinar:
https://www.dlubal.com/en-US/support-and-learning/learning/videos/000744

The input windows of the add-on module are accessible once you select the material *Steel* and the joint group *Pinned Joints*.

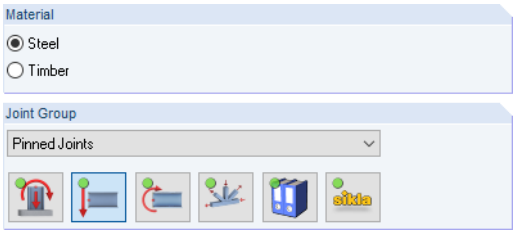
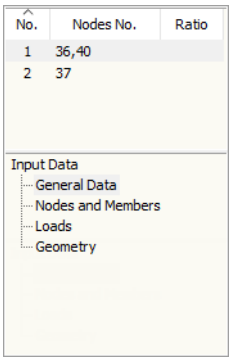


Image 4.1 *RF-/JOINTS Steel - Pinned add-on module*



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the *Geometry* entry is missing in the navigator, check whether the boundary conditions of the joint are correct in Window *1.2 Nodes and Members*. For example, it may be necessary to deactivate connected members for the design (see Figure 4.6).

4.1 General Data

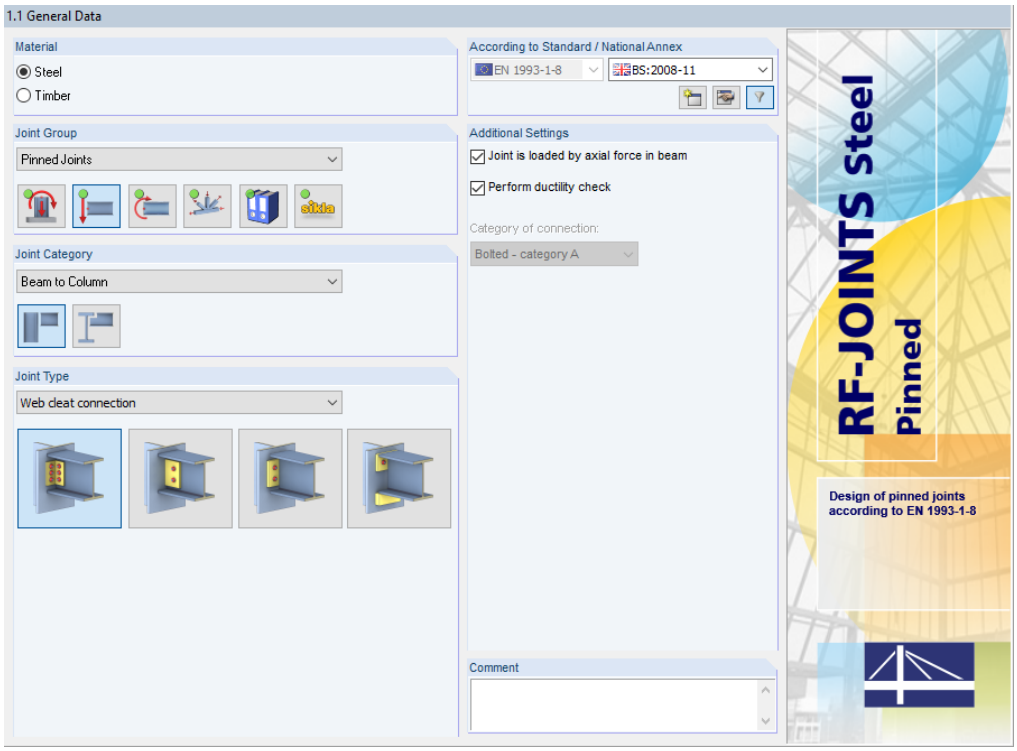
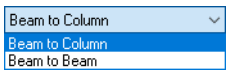


Image 4.2 Window 1.1 General Data

Joint Category



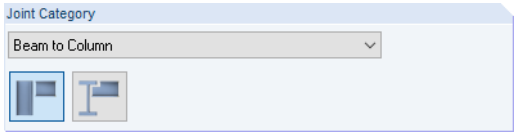


Image 4.3 Joint category

The joint categories *Beam to Column* and *Beam to Beam* are available for RF-/JOINTS Steel - Pinned. You can select the category by using the drop-down list or clicking the buttons with the connection icons.

Joint Type

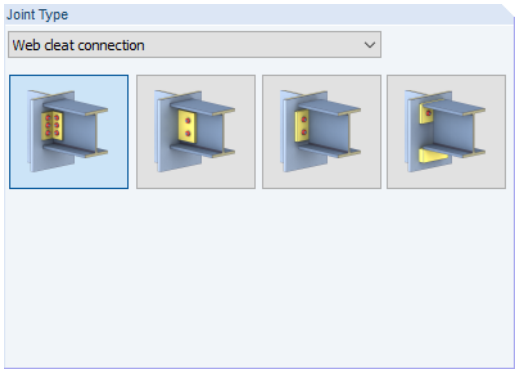
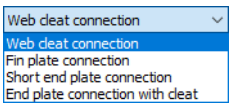


Image 4.4 Joint type



The following joint types are available:

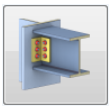
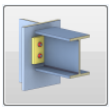
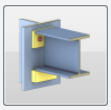
	Web cleat connection - angle sections arranged on both sides of the beam
	Fin plate connection - splice welded to column, arranged on one side
	Short end plate connection - bolted connection via welded end plate
	End plate connection with cleat for fixation (for *Beam to Column* category)

Table 4.1 Joint types

Additional Settings

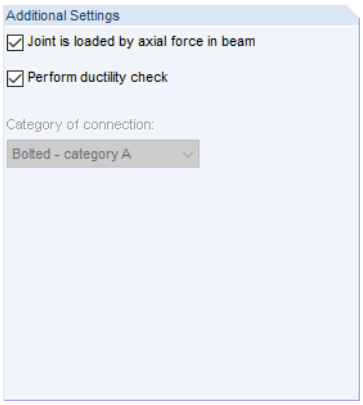


Image 4.5 *Additional Settings* section

With the *Joint is loaded by axial force in beam* check box, you can control additional designs for tension and compression loads in the joint. This setting is activated by default.

Pinned connections are somewhat special with regard to the *ductility check*: Several connections of this joint group would have to be classified as Zone 2 (semi-rigid) when classified according to their rotational stiffness as per [1]. Thus, they would have a certain moment resistance and would have to be taken into account as springs in the structural system. In actuality, however, a hinge-like situation occurs in the ultimate limit state because parts of the joint deform plastically. According to [6], a stiffness classification can therefore be dispensed with if the following criteria are met:

- Sufficient rotation capacity: Ensuring geometric boundary conditions in order not to impede rotations.
- Sufficient ductility: Ensuring that the joint can deform plastically and that brittle components such as screws or welds do not fail beforehand.

The two criteria are not included in [1] and are therefore not normative. It is therefore possible to deactivate the ductility check. However, the rotation capacity is always checked.

You can find more information on ductility checks in the following technical article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001128

The *Category of connection* is preset according to the standard [1]. It cannot be changed.

4.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2.

Valid cross-sections

In the *Parameters* section, you can check the boundary conditions of the connected structural components. A section without **I**- or **H**-shaped cross section is marked as an *Invalid cross-section*.

If other members such as transverse beams or diagonals connect to the node, the redundant members can be set *Inactive*, as shown in the following image.

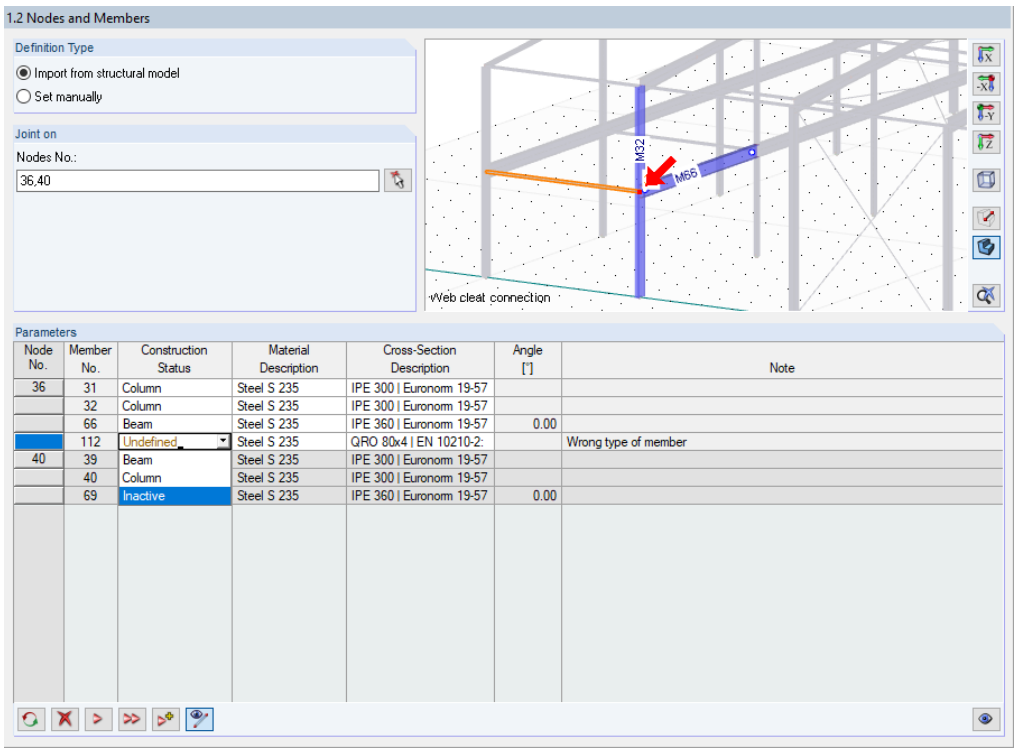


Image 4.6 Window *1.2 Nodes and Members*: Setting post *Inactive*

Since the joint does not transfer any moments, there must be a moment hinge for the beam in RFEM or RSTAB.

The column members can also be rotated by 90°.

If all members except the connecting beam are set *Inactive*, a connection is made to a so-called "anchor plate". This allows for connections to invalid column cross-sections (concrete columns), for example.

The modeling of an anchor plate is described in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001034

4.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

4.4 Geometry

In Window *1.4 Geometry*, you can define the geometric parameters of the shear connection. Default values are already preset.

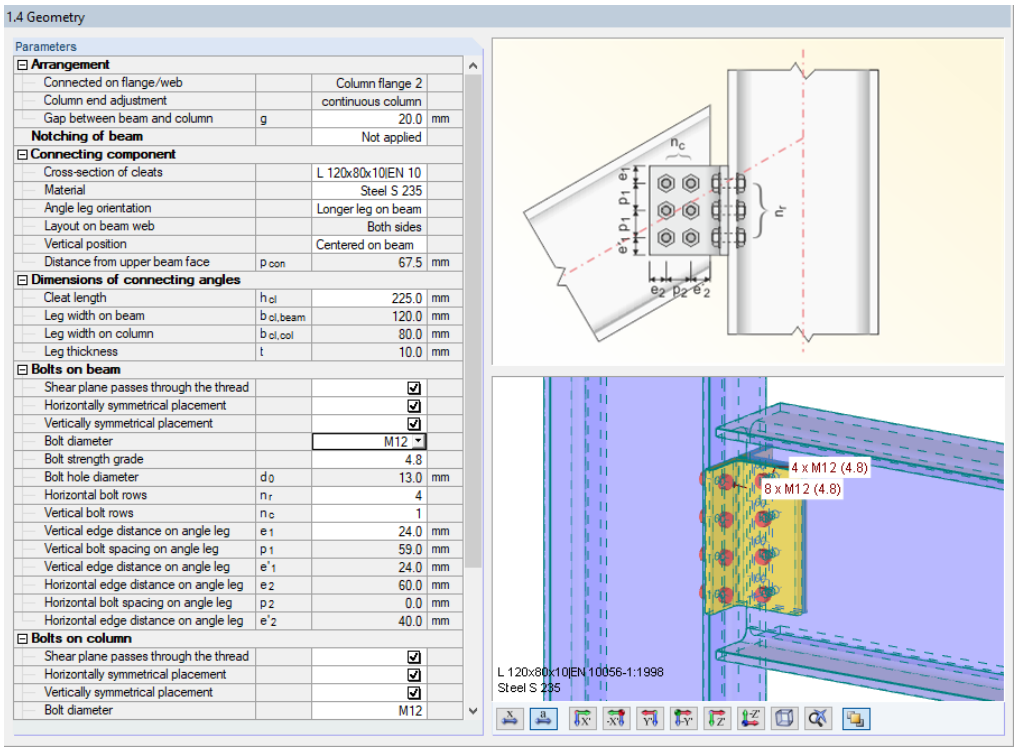
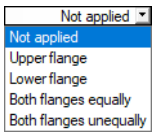


Image 4.7 Window *1.4 Geometry*

Arrangement

This section manages general information about the connection. For inclined connections, the column end can be adjusted. Furthermore, it is possible to adjust the *Gap* between beam and column and to arrange the *Notching*.

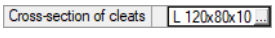
Notching of beam



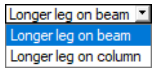
The position of the beam notching can be selected in the list. The other geometry specifications (length, height, radius) can then be defined in detail.

Connecting component

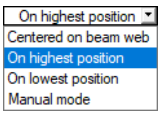
The parameters of the connecting component depend on the selected joint type.



The preset web cleat connection can be changed using the cross-section library: When you click the *Cross-section of cleats* text box, the ... button appears (see figure on the left). It provides access to the cross-section library of RFEM/RSTAB where you can select a different cleat section (see Figure 4.8).



The *Angle leg orientation* can be adjusted using the corresponding list. Only a layout of the web angles on both sides is possible.



The *Vertical position* describes how the angle is positioned on the beam. You can select the position using the list, which also allows for manual input.

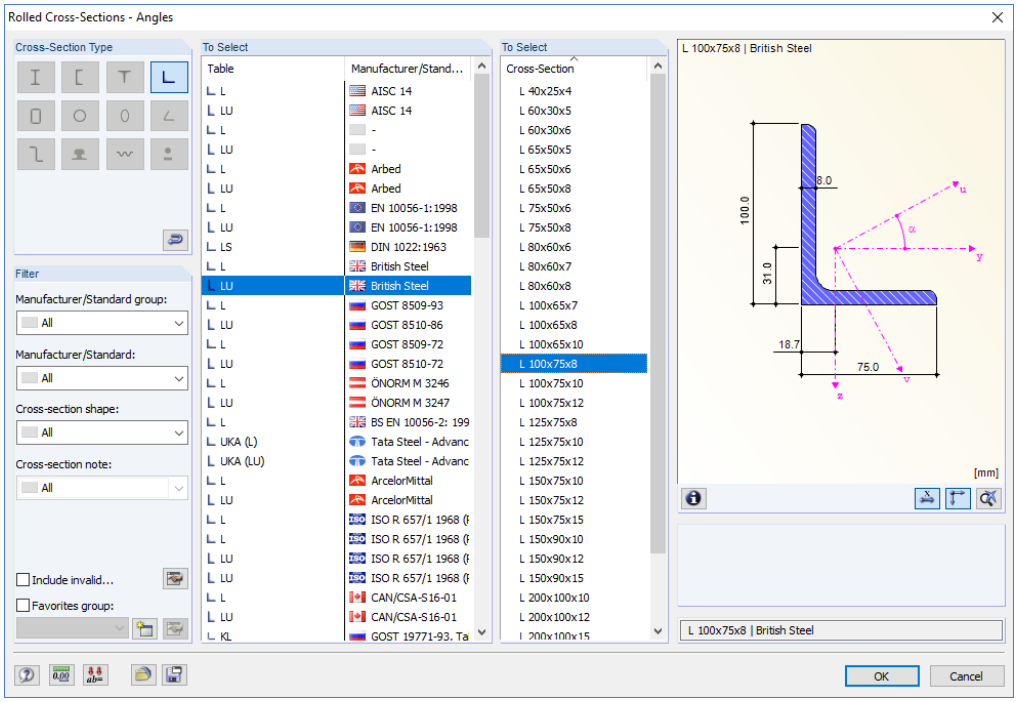
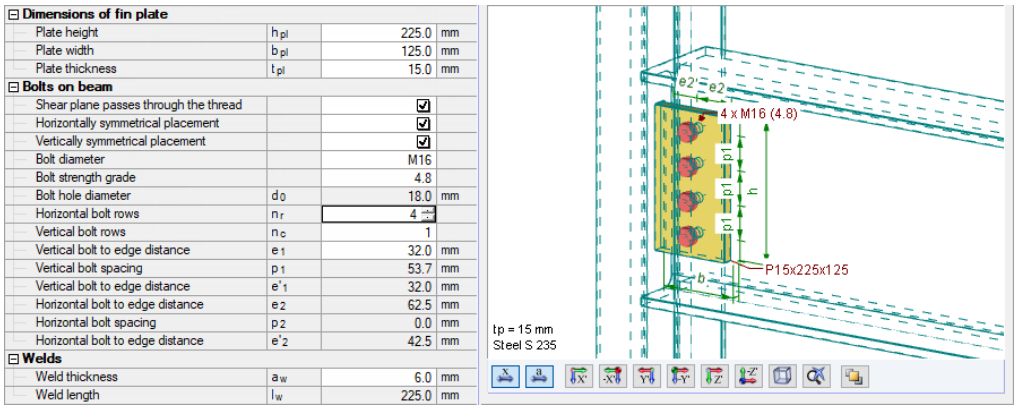


Image 4.8 Selecting cleat section in library

Dimensions

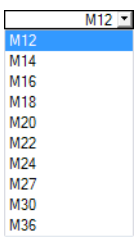
For each joint type, you can define the specific geometry parameters (height, width, thickness) of the angles, splices, end plates, and cleats.



Dimensions of fin plate			
Plate height	h_{pl}	225.0	mm
Plate width	b_{pl}	125.0	mm
Plate thickness	t_{pl}	15.0	mm
Bolts on beam			
Shear plane passes through the thread		☑	
Horizontally symmetrical placement		☑	
Vertically symmetrical placement		☑	
Bolt diameter		M16	
Bolt strength grade		4.8	
Bolt hole diameter	d_0	18.0	mm
Horizontal bolt rows	n_r	4	
Vertical bolt rows	n_c	1	
Vertical bolt to edge distance	e_1	32.0	mm
Vertical bolt spacing	p_1	53.7	mm
Vertical bolt to edge distance	e'_1	32.0	mm
Horizontal bolt to edge distance	e_2	62.5	mm
Horizontal bolt spacing	p_2	0.0	mm
Horizontal bolt to edge distance	e'_2	42.5	mm
Welds			
Weld thickness	a_w	6.0	mm
Weld length	l_w	225.0	mm

Image 4.9 Defining dimensions of fin plate, bolt and weld parameters

Bolts on beam/column



The bolt parameters (diameter, strength grade, number of horizontal and vertical bolt rows, edge distances, etc.) can be defined in the corresponding text boxes or using the lists.

For a simplified input, you can also specify symmetry conditions.

Welds

For the joint types *Fin plate connection*, *Short end plate connection*, and *End plate connection with cleat*, you can specify the weld thicknesses and lengths.

The end plate required for the static equilibrium in the case of a cleat connection must be welded to the web **and** to the flange of the beam.

Joint category Beam to Beam

For the *Beam to Beam* joint category, you can define the geometry parameters in the same way.

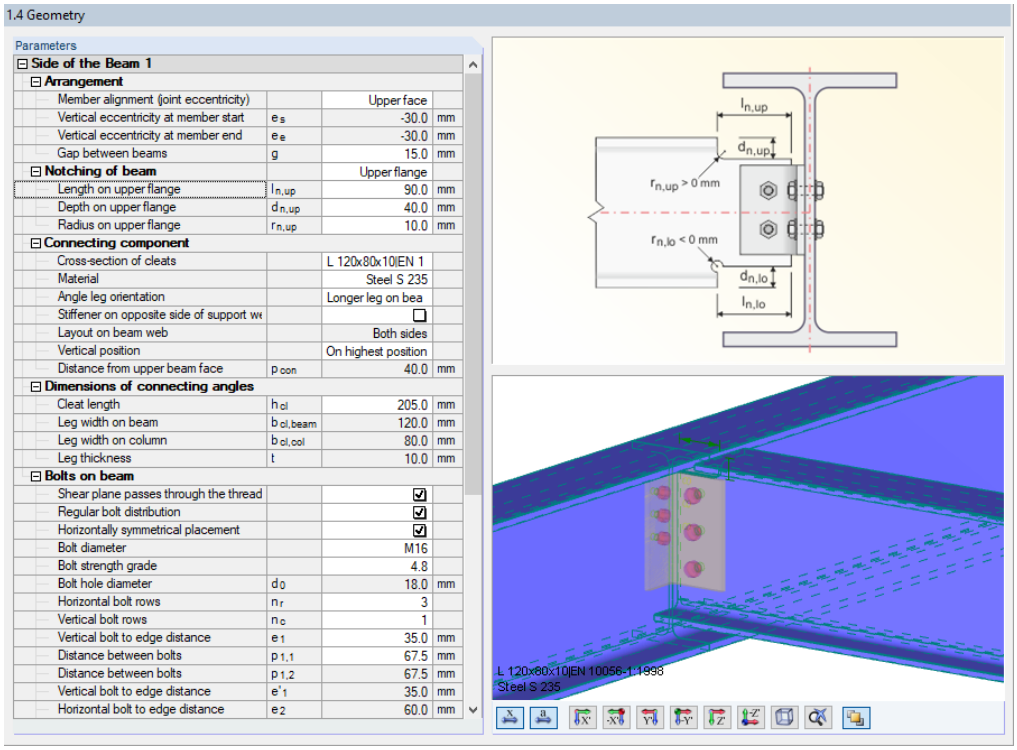


Image 4.10 Window *1.4 Geometry* for *Beam to Beam* joint category with web cleat connection

On our website, you can find a technical article in which various forms of pinned connections of notched secondary beams with fin plates are discussed:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001532

5 Steel - Rigid

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Steel - Rigid** module for designing moment resisting joints of I and H beams. The general input parameters are described in Chapter 2.

The input windows of the add-on module are accessible once you select the material *Steel* and the joint group *Rigid Joints*.

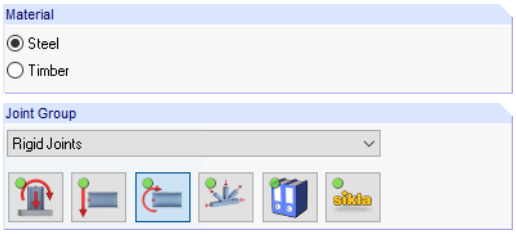
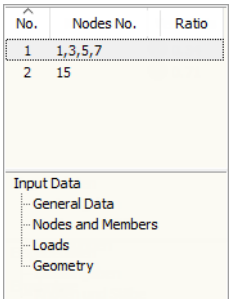


Image 5.1 *RF-/JOINTS Steel - Rigid add-on module*



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the *Geometry* entry is missing in the navigator, check whether the boundary conditions of the joint are correct in Window *1.2 Nodes and Members*. For example, it may be necessary to deactivate connected members for the design (see Figure 5.6).

5.1 General Data

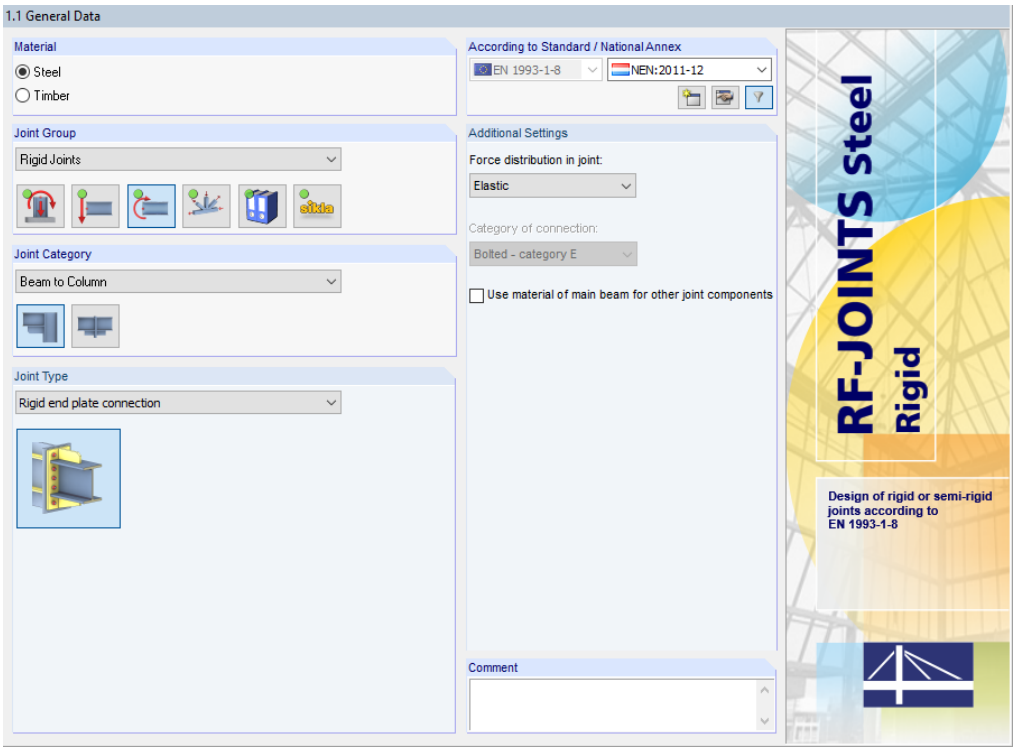
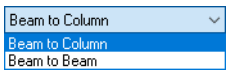


Image 5.2 Window *1.1 General Data*

Joint Category



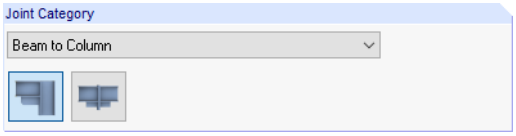


Image 5.3 Joint Category

The joint categories *Beam to Column* and *Beam to Beam* are available for RF-/JOINTS Steel - Rigid. You can select the category by using the drop-down list or clicking the buttons with the connection icons.

Joint Type

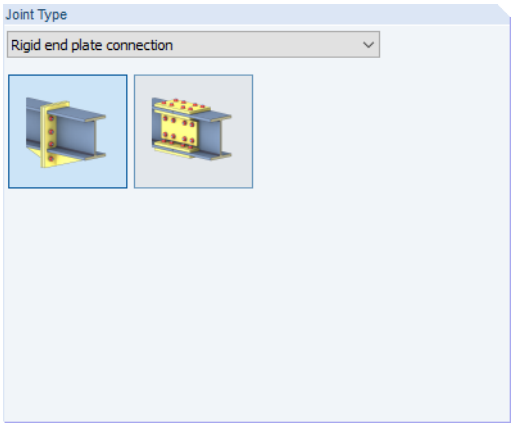
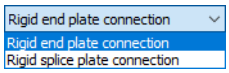


Image 5.4 Joint type



The following joint types are available:

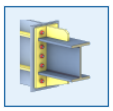
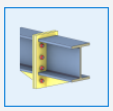
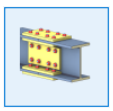
	End plate connection - bolted beam-to-column connection via welded end plate
	End plate joint - bolted beam joint via welded end plates
	Splice plate - bolted beam joint via splice plates

Table 5.1 Joint types

Additional Settings

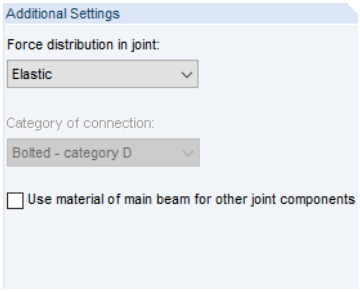


Image 5.5 *Additional Settings* section

You can use the list for the *Force distribution in joint* to control whether an elastic or plastic distribution of the bolt forces should be assumed in the joint. The default setting is *Elastic*.

The *Use material of main beam for other joint components* check box allows you to use a material globally for all components. Excepted from this are screws, for which the strength grade must always be specified. If the check box is cleared (default setting), you can define the materials for each component separately in Window *1.2 Nodes and Members*.

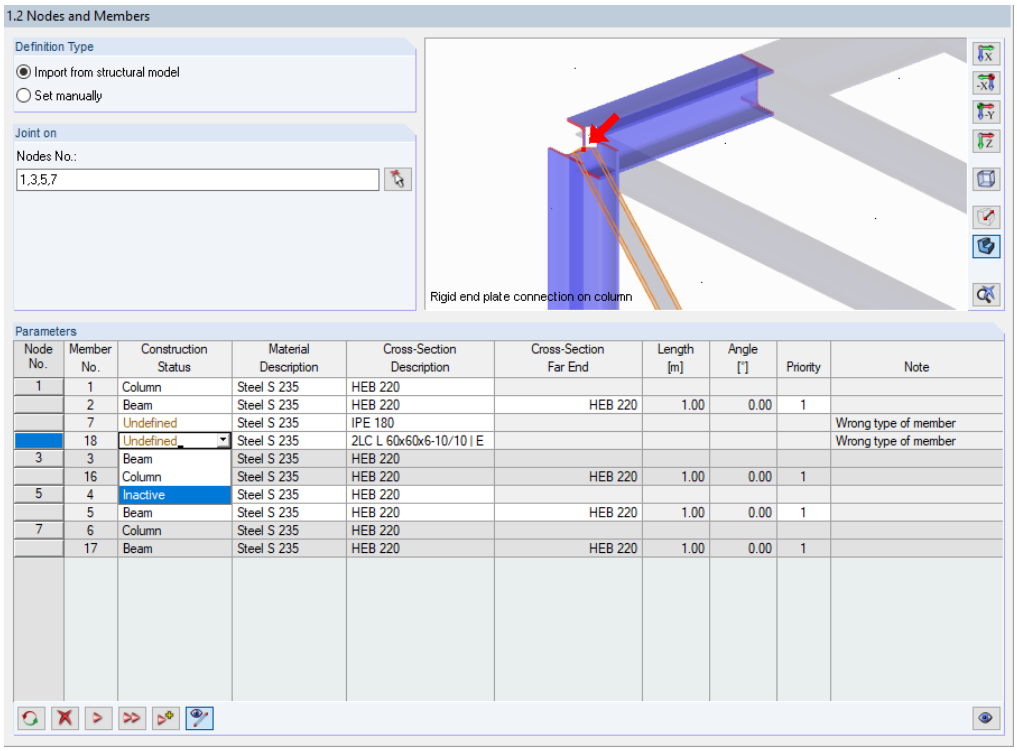
5.2 Nodes and Members

Valid cross-sections

The selection of nodes and members is described in Chapter 2.2.

In the *Parameters* section, you can check the boundary conditions of the connected structural components. A section without **I**- or **H**-shaped cross-section is marked as an *Invalid cross-section*.

If other members such as transverse beams or diagonals connect to the node, the redundant members can be set *Inactive*, as shown in the following image.



Node No.	Member No.	Construction Status	Material Description	Cross-Section Description	Cross-Section Far End	Length [m]	Angle [°]	Priority	Note
1	1	Column	Steel S 235	HEB 220					
	2	Beam	Steel S 235	HEB 220	HEB 220	1.00	0.00	1	
	7	Undefined	Steel S 235	IPE 180					Wrong type of member
	18	Undefined	Steel S 235	2LC L 60x60x6-10/10 | E					Wrong type of member
3	3	Beam	Steel S 235	HEB 220					
	16	Column	Steel S 235	HEB 220	HEB 220	1.00	0.00	1	
5	4	Inactive	Steel S 235	HEB 220					
	5	Beam	Steel S 235	HEB 220	HEB 220	1.00	0.00	1	
7	6	Column	Steel S 235	HEB 220					
	17	Beam	Steel S 235	HEB 220	HEB 220	1.00	0.00	1	

Image 5.6 Window *1.2 Nodes and Members*: setting platform beam and bracing member *Inactive*

In contrast to the *Pinned Joints* joint group, it is not possible to rotate the column member by 90° for *Rigid Joints*. The beams or columns connected in a node must always be connected in their main plane. For a bolted beam-to-column joint, for example, this means that a beam perpendicular to the web must be connected to the column flange.

Although the joint transfers bending moments in the plane, it is generally not a problem if the beam has a moment hinge in RFEM/RSTAB on the node to be designed.

If all members except the connecting beam are set *Inactive* for the *Rigid end plate connection* joint type, a connection is made to a so-called "anchor plate". This allows for connections to invalid column cross-sections (concrete columns), for example. In this case, all design components that refer to the "inactive part" of the connection are not taken into account (the anchorage in concrete, for example).

The modeling of an anchor plate is described in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001034

5.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

5.4 Geometry

In Window *1.4 Geometry*, you can define the geometric parameters of the moment-resisting joint. Default values are already preset.

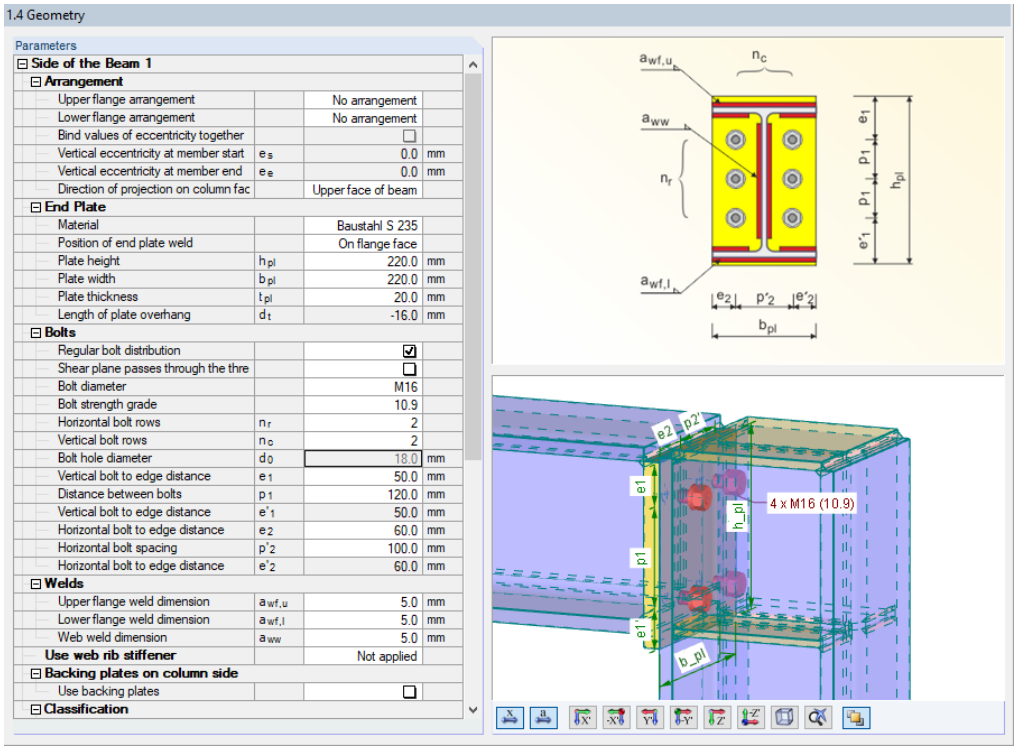
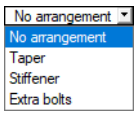


Image 5.7 Window *1.4 Geometry* for bolted beam-to-column joint

Arrangement



This section manages general information about the connection. For example, you can define buckling stiffeners or tapers on the top and bottom side of the beam at the column joint.

In the case of a splice plate connection, you can define the distance (gap) between the beams as well as the vertical position of the web joint in this section.

End Plate

This section is available if the joint configuration has an end plate:

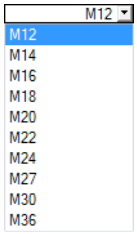
- *Rigid end plate connection* for beam to column
- *Rigid end plate connection* for beam to beam

Here you can define the basic properties of the end plate: Plate height, width, and thickness. Furthermore, you have to specify if the end plate is "retracted" at the top or if there is a *plate overhang*.

If the existing joint configuration (also) provides a "retracted" end plate at the bottom of the beam, this has to be controlled by the plate height.

If the *Use material of main beam for other joint components* option is deactivated in Window *1.1 General Data*, you can also define the material of the end plate in this section.

Bolts



This section is also only available for bolted end plate connections. Here you have to enter all the important data concerning the bolt layout. The interactive graphic supports most of the input data, so that changes in the geometry can be immediately understood. In this section, you can also define the bolt strength grade.

Two or four-row joint configurations are possible.

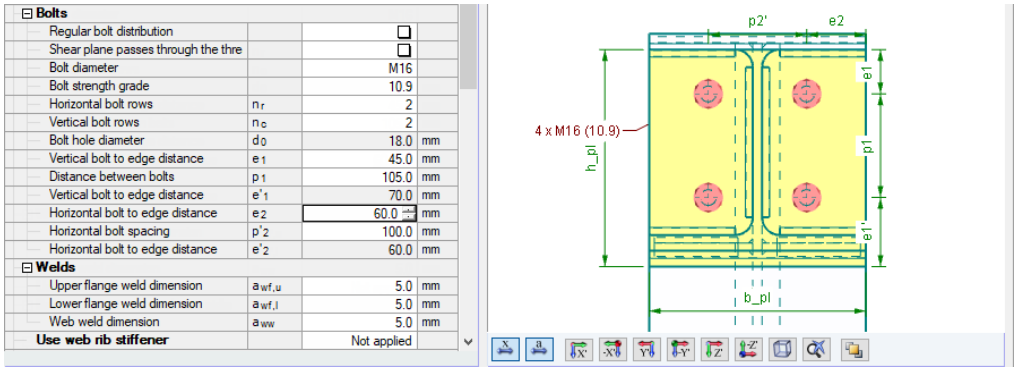


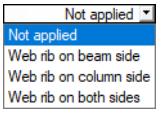
Image 5.8 Parameters for bolts and bolt layout

Welds

This section is only relevant for end plate connections as well. You can define the fillet weld thicknesses separately for the connection of the upper and lower flange to the end plate as well as for the connection of the beam web to the end plate.

Fillet welds are always assumed; you cannot enter butt welds.

Use web rib stiffener



This section is only available for rigid beam-to-column joints. In it, you can specify whether there are additional horizontal web ribs and where they are arranged. The *Use web rib stiffener* list allows you to arrange one or more web ribs on the beam, in the column, or in both structural components.

In the other rows, you can define the geometric parameters that affect the dimensions and position of the stiffeners. Furthermore, you have to specify the fillet weld thickness for the connection of the stiffeners.

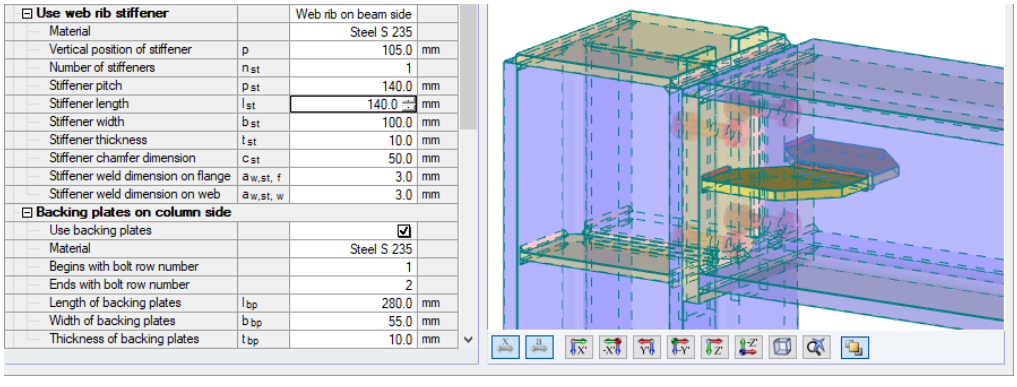


Image 5.9 Parameters for web rib and backing plates

Backing plates on column side

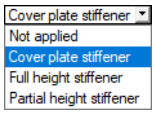
For bolted, rigid beam-to-column joints, you can specify whether you want to reinforce the column flange by backing plates in this section. Backing plates are described in detail in [1] Section 6.2.4.3.

After selecting the *Use backing plates* option, you can define the parameters that affect the backing plates at the column flange in the rows below (see Figure 5.9).

Backing plates are inserted loosely. It is not possible to weld these plates firmly into the column cross-section.

Column part

The *Column part* section is only relevant for bolted, rigid beam-to-column joints and is therefore only available for this joint configuration.



In this section, you can enter all the information that is important for the design of the column components. This includes the formation of the upper part of the column, the arrangement of stiffeners in the plane of the upper and lower flange of the connected beam, as well as the option to reinforce the column web with a *web stiffener* for shear loading (plate reinforcement or diagonal stiffening of the column web).

1.4 Geometry

Parameters			
Side of the Beam 1			
Arrangement			
End Plate			
Bolts			
Welds			
Use web rib stiffener		Not applied	
Backing plates on column side			
Classification			
Column part			
Column end adjustment		Perpendicular	
Extension length	$l_{c,ext}$	0.0	mm
Column upper stiffener		Cover plate stiffener	
Material		Steel S 235	
Stiffener length	l_{st}	190.0	mm
Stiffener width	b_{st}	220.0	mm
Stiffener thickness	t_{st}	10.0	mm
Stiffener weld dimension on flange	$a_{w,st,f}$	5.0	mm
Stiffener weld dimension on web	$a_{w,st,w}$	3.0	mm
Column lower stiffener		Full height stiffener	
Material		Steel S 235	
Stiffener length	l_{st}	188.0	mm
Stiffener width	b_{st}	100.0	mm
Stiffener thickness	t_{st}	8.0	mm
Chamfer on stiffener length	$c_{0,1}$	15.0	mm
Chamfer on stiffener width	$c_{0,2}$	15.0	mm
Stiffener weld dimension on flange	$a_{w,st,f}$	4.0	mm
Stiffener weld dimension on web	$a_{w,st,w}$	3.0	mm
Column web stiffener		Supplementary plate	
Both sides		☑	
Vertical position of stiffener	p	-90.0	mm
Material		Steel S 235	
Stiffener length	l_{st}	180.0	mm
Stiffener width	b_{st}	150.0	mm
Stiffener thickness	t_{st}	8.0	mm
Web stiffener weld dimension	$a_{w,st}$	5.0	mm

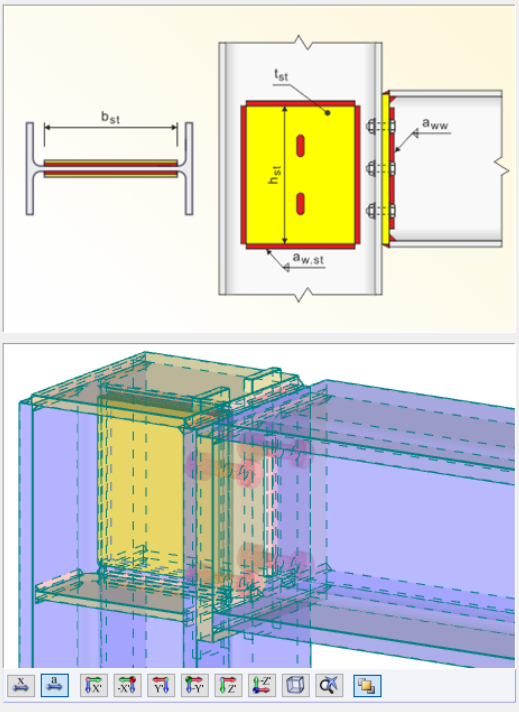


Image 5.10 Window *1.4 Geometry* with specifications for *Column part*

Splice plates on flanges

The *Splice plates on flanges* section is only available for the *Rigid splice plate connection* joint type. In this section, you can enter all geometry specifications for the external and internal splice plates.

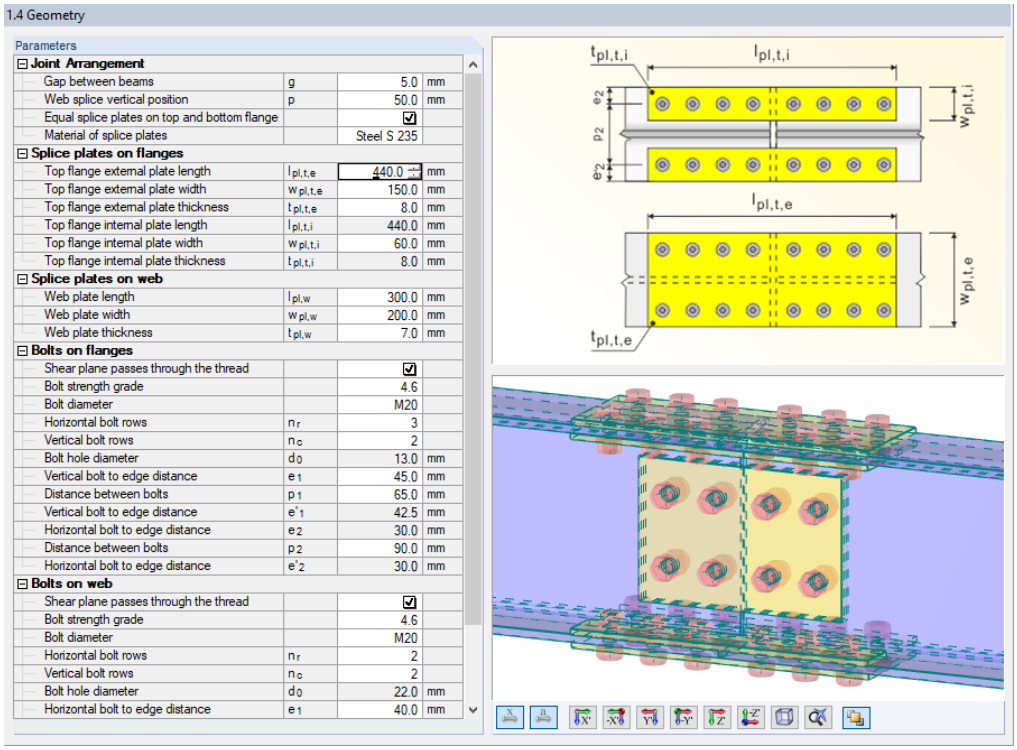


Image 5.11 Window *1.4 Geometry* for rigid splice plate connection

Currently, only splice plate connections that have external and internal splice plates on the flanges are supported. Configurations that only have external splice plates are not possible.

Splice plates on web

This section is only available for the *Rigid splice plate connection* joint type as well. This is where you define the geometric specifications for the web splices.

Bolts on flanges

This section manages the specifications concerning the bolt layout of a rigid splice plate connection. Furthermore, you can specify the bolt strength grade and bolt diameters.

Currently, only connections with a uniform bolt diameter and with one bolt row per flange side are supported. It is therefore not possible to enter four-row configurations.

Bolts on web

In this section, you can specify the bolt parameters to be applied to the web for a rigid splice plate connection: Bolt layout, bolt diameter, and bolt strength grade.

6 Steel - Tower

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Steel - Tower** module. The general input parameters are described in Chapter 2.

You can access the input windows of the add-on module once you select the material *Steel* and the joint group *Tower*.

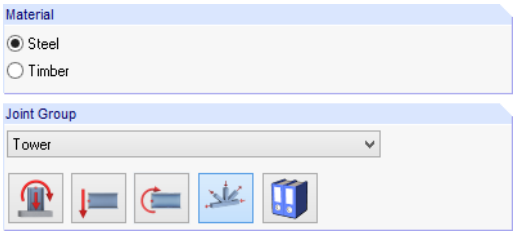
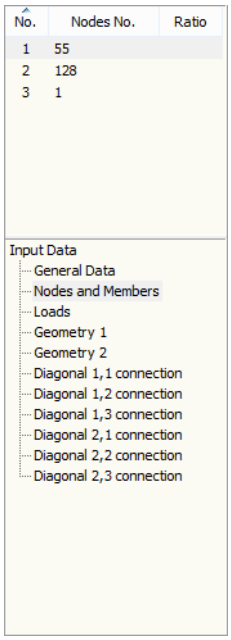


Image 6.1 *RF-JOINTS Steel - Tower add-on module*



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the entries *Geometry 1*, *Diagonal 1,1 connection*, etc. are missing in the navigator, go to Window *1.2 Nodes and Members* and check if the boundary conditions of the node are correct. For example, it might be necessary to adjust the construction status of the connected members (see Figure 6.9).

The input windows of the *RF-JOINTS Steel - Tower* module are divided into two parts: You can find the input parameters of the tower component on the left; they are illustrated by graphics on the right (see Figure 6.10). The upper graphic shows a system sketch of the current parameter, the lower graphic shows a 3D visualization of the node.

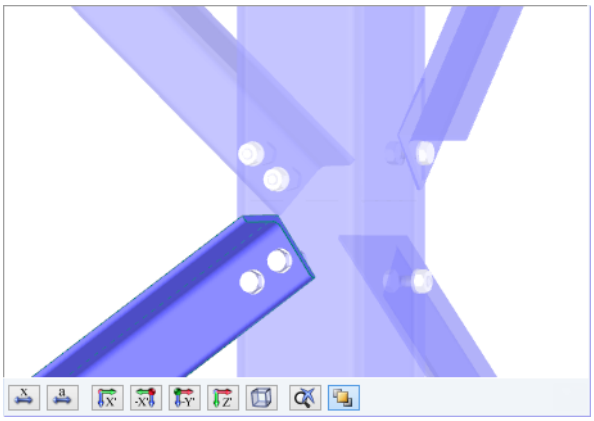


Image 6.2 3D visualization of node

The buttons below the 3D graphic are described in Table 3.1.

6.1 General Data

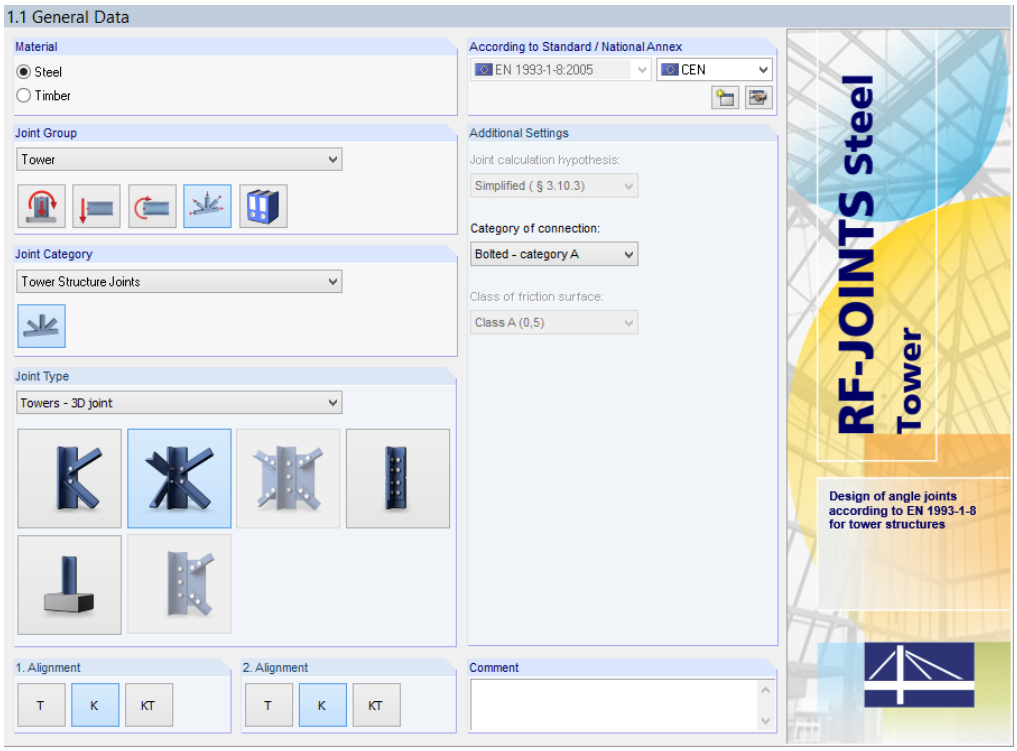


Image 6.3 Window *1.1 General Data*

Joint Category

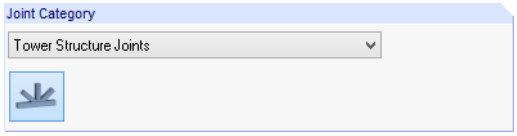


Image 6.4 Joint category

Only the *Tower Structure Joints* category is available for RF-JOINTS Steel - Tower.

Joint Type

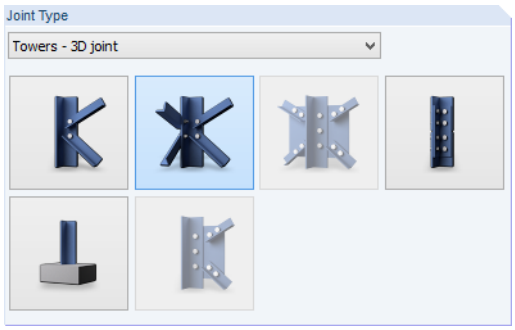
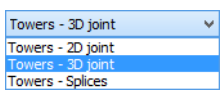


Image 6.5 Joint type

The following joint types are available:



	Uniplanar joint with diagonals on one column leg
	Spatial joint with diagonals on both column legs
	Splice plate connection

Table 6.1 Tower - joint types

1. Alignment / 2. Alignment

Image 6.6 1. and 2. alignment

In this section, you can define the basic geometric shape of the joint for planes *1* and, if necessary, *2* (for 3D connections).

The shape of the letters indicates the number of members on the connection node and their function:

	Major chord, minor chord, one strut
	Major chord, minor chord, two struts
	Major chord, minor chord, three struts

Table 6.2 Alignment

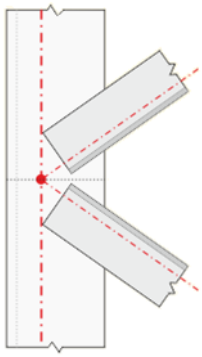


Image 6.7 K alignment

Additional Settings

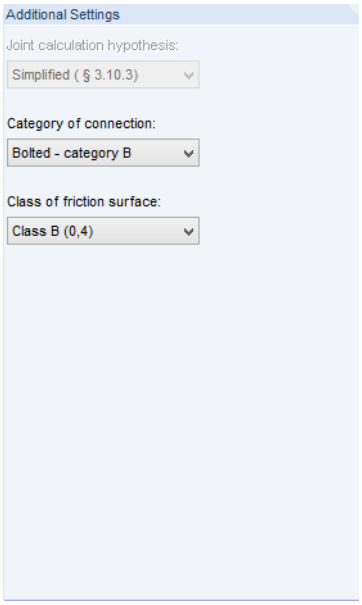
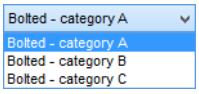


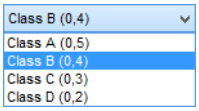
Image 6.8 *Additional Settings section*

The program uses the simplified *Joint calculation hypothesis* according to [1] Section 3.10.3 that is applicable for single-leg bolted connections of angles subjected to tensile stress (see [1] Section 2.7 (2)).



You can select the *Category of connection* according to [1] Section 3.4 using the drop-down list. The categories A to C are available for bolt connections with shear stress.

- Category A: Shear/hole bearing connections
- Category B: Slip-resistant connections in the serviceability limit state
- Category C: Slip-resistant connections in the ultimate limit state



For high-strength prestressed bolt connections (Category B or C), you can select the *Class of friction surface* from the corresponding list. The classes with the corresponding friction coefficients μ are regulated in [1] Table 3.7.

6.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2.

In the *Parameters* section, you can check the boundary conditions of the connected structural components. If there is an *Invalid cross-section*, you should adjust the cross-section series to match the joint type and alignment set in Window 1.1.

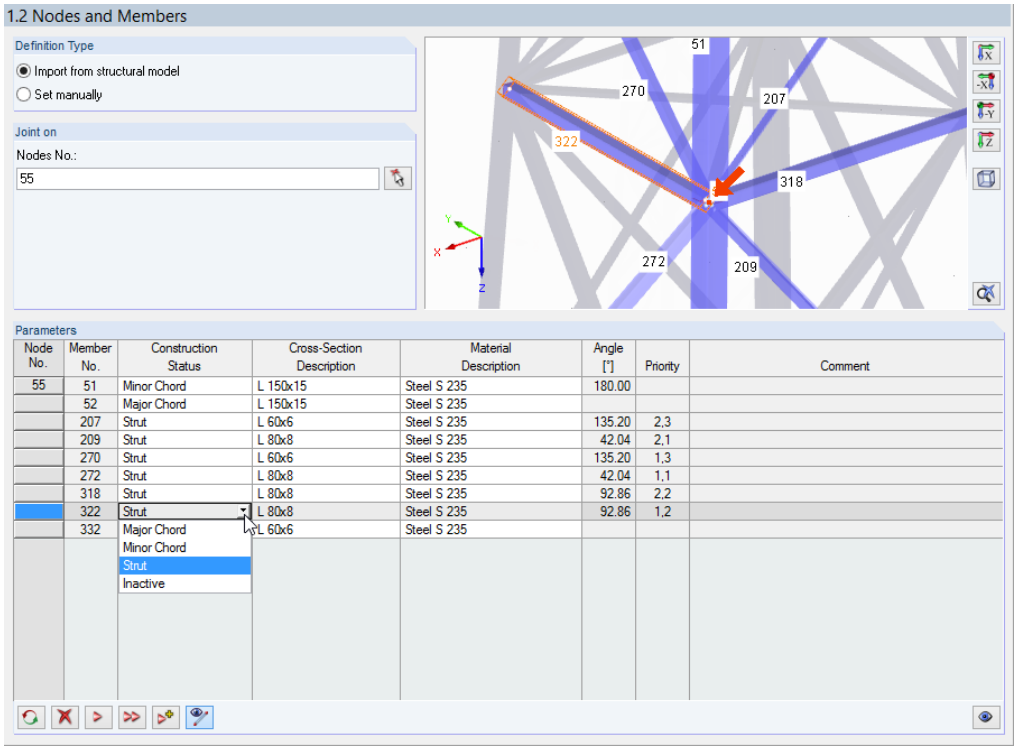


Image 6.9 Adjusting the construction status of members

The *Angle* and *Priority* columns provide information about the connected members.

The angles are based on the geometric conditions of the RFEM/RSTAB model. If you select the *Set manually* definition type in Window 1.2, you can enter user-defined angles of the connected members.

The priority of the struts controls the assignment for the input windows *Diagonal 1,1 connection*, *Diagonal 1,2 connection*, etc. The digit **before** the comma indicates struts of plane 1 ("1. Alignment") or plane 2 ("2. Alignment" - only for 3D connections). The digit **after** the comma numbers the members within the respective plane. The order is not relevant for the design.

Example: Priority *1,2* means "plane 1, strut 2". You can define the bolt parameters of this strut in the *Diagonal 1,2 connection* window.

When you click on a row of the table, the selected member is highlighted in the graphic.

6.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

6.4 Geometry 1 / Geometry 2

The *Geometry* window manages the geometric boundary conditions of the members on the connection node. There are two separate windows for both planes: The *Geometry 1* window is responsible for plane 1 ("1. Alignment"), the *Geometry 2* window for plane 2 ("2. Alignment" - only for 3D connections).

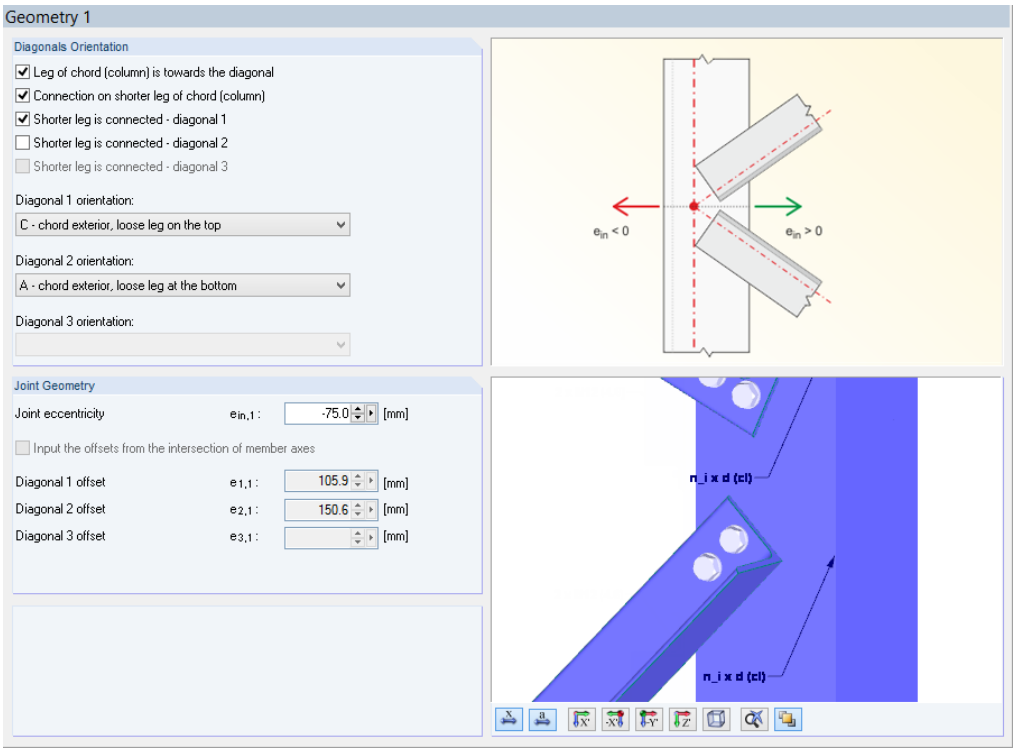
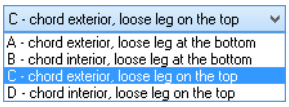


Image 6.10 *Geometry 1* window

Diagonals Orientation

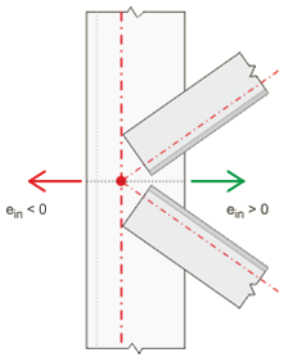
The arrangement of diagonals on a node is preset during the data transfer from the RFEM/RSTAB model (see Figure 6.9). The text boxes in this window section are locked.



If you define the geometry and internal forces manually, you can enter a user-defined arrangement and orientation of diagonals (see figure above). The check boxes and list entries describe the connected leg and its position.

Joint Geometry

The *Joint eccentricity* describes the eccentricity of the diagonals. It is related to the intersection of the cross-section central axes. As shown in the system sketch, positive values move the joint in the direction of the loose column leg, negative values in the direction of the angle corner.



In the fields below, the local *offset* of the diagonals in the longitudinal direction of the member is updated automatically.

The graphic on the right provides a dynamic visualization of the geometric parameters.

6.5 Geometry Plates

If you select the *Towers - Splices* joint type in Window *1.1 General Data* (see Figure 6.5), the *Geometry, Plates* window appears.

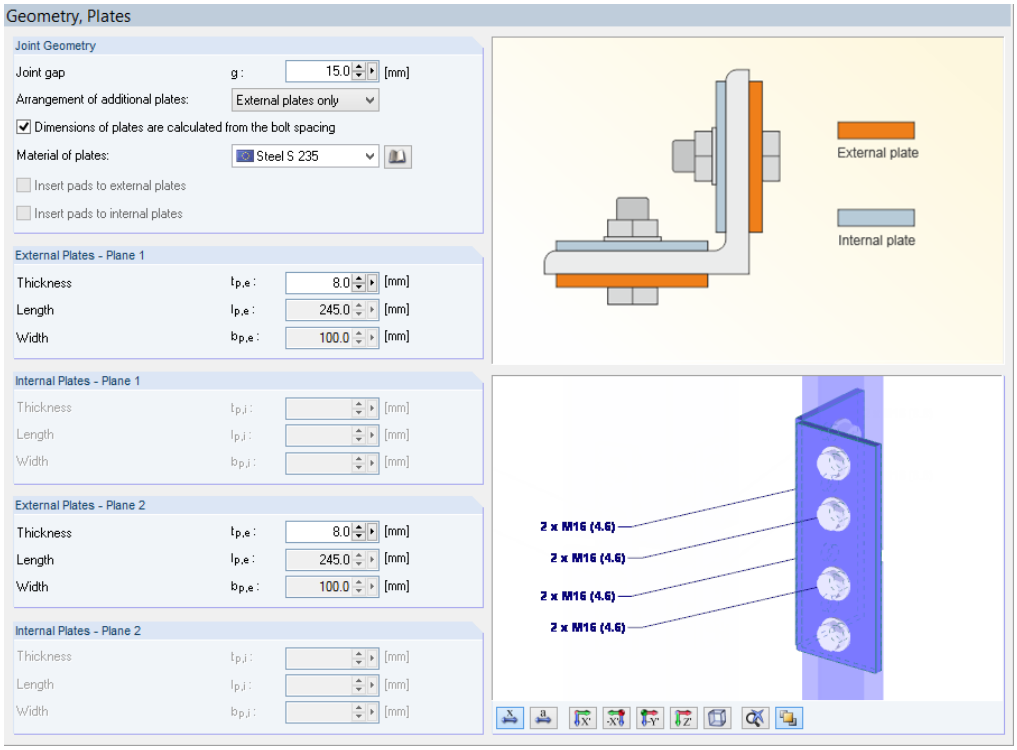
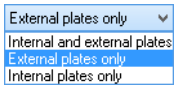


Image 6.11 *Geometry, Plates* window for *Towers - Splices* joint type

Joint Geometry

In this section, you can specify the splice plate connection's *gap* size between the cross-sections.



Concerning the arrangement of splices, you can select *External plates* and/or *Internal plates*. The system sketch on the right illustrates the position of the plates.

If you select the *Dimensions of plates are calculated from the bolt spacing* check box, the text boxes for entering length and width of plates below the check box are locked.

The *Material of plates* can be selected from the list or in a library by using the button.

If there are different cross-sections, it is possible to *Insert pads* for external or internal plates in order to compensate for the differences in thickness.

External/Internal Plates - Plane 1 / Plane 2

In these text boxes, you can define the *Thickness*, *Length*, and *Width* of external or internal plates. Planes 1 and 2 are related to the two legs of the angle.

If you select the *Dimensions of plates are calculated from the bolt spacing* check box in the section above, you can only enter the thicknesses.

During the calculation, the module also checks structural details. For example, if the splice dimensions are too small for the bolts, a corresponding error message appears.

6.6 Diagonal 1,1 connection / Diagonal 1,2 connection

In the *Diagonal X,Y connection* windows, you can define the parameters of the diagonals.

There is a separate window for each plane and member. The *Diagonal 1,1 connection* window, for example, is responsible for plane 1 ("1. Alignment") and strut 1, and the *Diagonal 1,2 connection* window for plane 1 and strut 2. The *Diagonal 2,1 connection* window manages the parameters for strut 1 in plane 2.

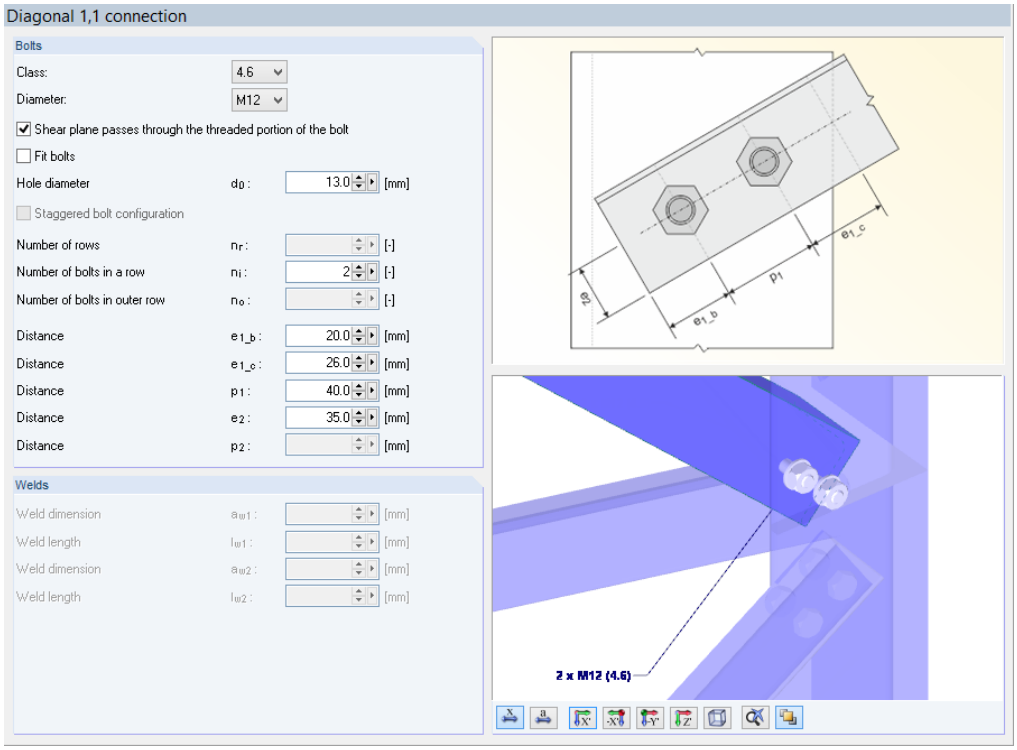
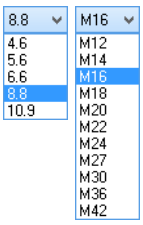


Image 6.12 *Diagonal 1,1 connection window*

Bolts



You can select the *Class* and *Diameter* of the bolts in the respective lists. Only bolts of the same type are permitted for each connection.

You can use the text boxes to define the *Hole diameter*, *Number of rows* (currently only one row), *Number of bolts in a row*, as well as the *Distance* from the edges and between the bolts. The system sketch on the right illustrates the individual parameters.

Welds

This section is intended for the weld parameters of gusset plates. Since these joint types are not implemented yet, the text boxes are locked.

6.7 Definition of Fasteners

If you have selected the *Towers - Splices* joint type in Window *1.1 General Data* (see Figure 6.5), you can define the bolt parameters in the *Definition of Fasteners* window.

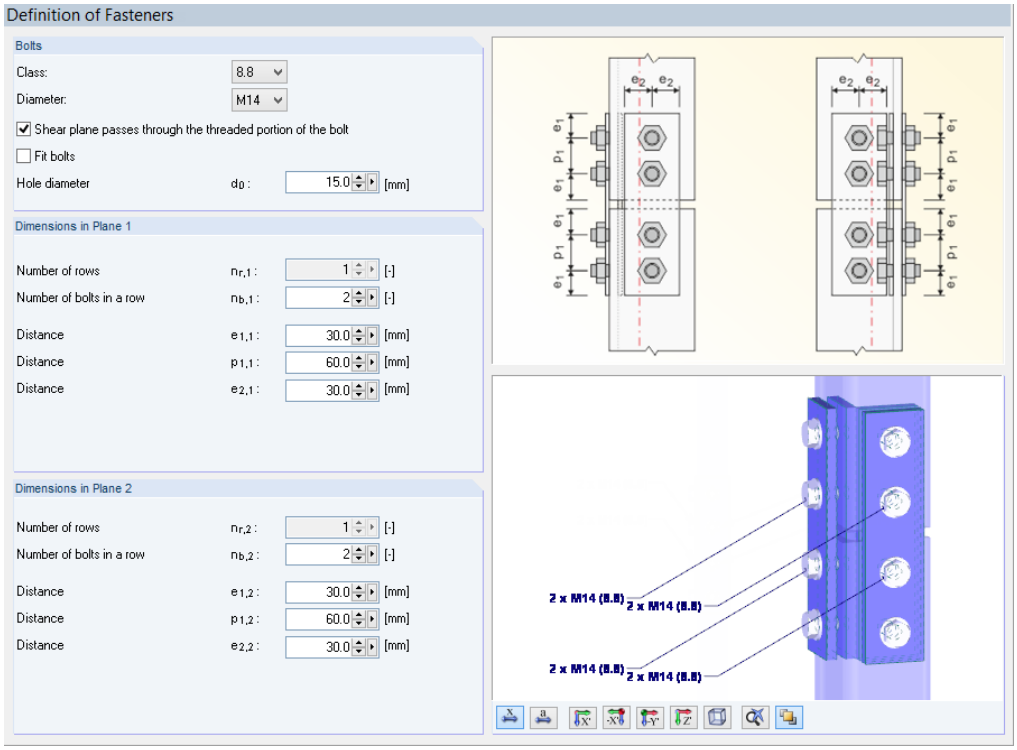
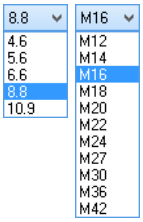


Image 6.13 *Definition of Fasteners window*

Bolts



You can select the *Class* and *Diameter* of the bolts in the respective lists. Only bolts of the same type are permitted for the connection.

The check boxes allow you to specify the position of the *Shear plane* and the bolt type (black bolts or *Fit bolts*). You can enter the *Hole diameter* separately.

Dimensions in Plane 1 / Plane 2

There is an individual section for each angle plane.

You can use the text boxes to define the *Number of rows* (currently only one row), the *Number of bolts in a row*, as well as the respective *Distance* from the edges and between the bolts. The system sketch on the right illustrates the individual parameters.

7 Steel - DSTV

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Steel - DSTV** module. The general input parameters are described in Chapter 2.

The functionality of this add-on module is presented in a Dlubal webinar:
https://www.dlubal.com/en-US/support-and-learning/learning/videos/000792

The input windows of the add-on module are accessible once you select the material *Steel* and the joint group *Typified joints - DSTV*.

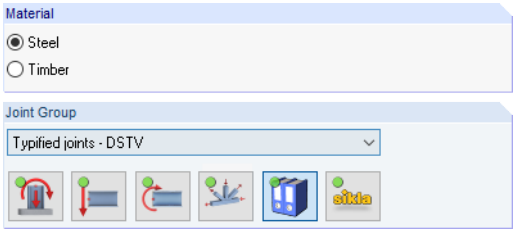
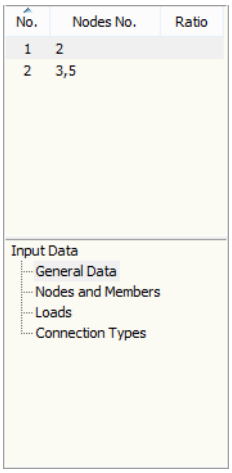


Image 7.1 *RF-/JOINTS Steel - DSTV add-on module*



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the *Connection Types* entry is missing in the navigator, check whether the boundary conditions for entering the connection are correct in Window *1.2 Nodes and Members*. For example, it may be necessary to deactivate connected members for the design (see Figure 7.7).

7.1 General Data

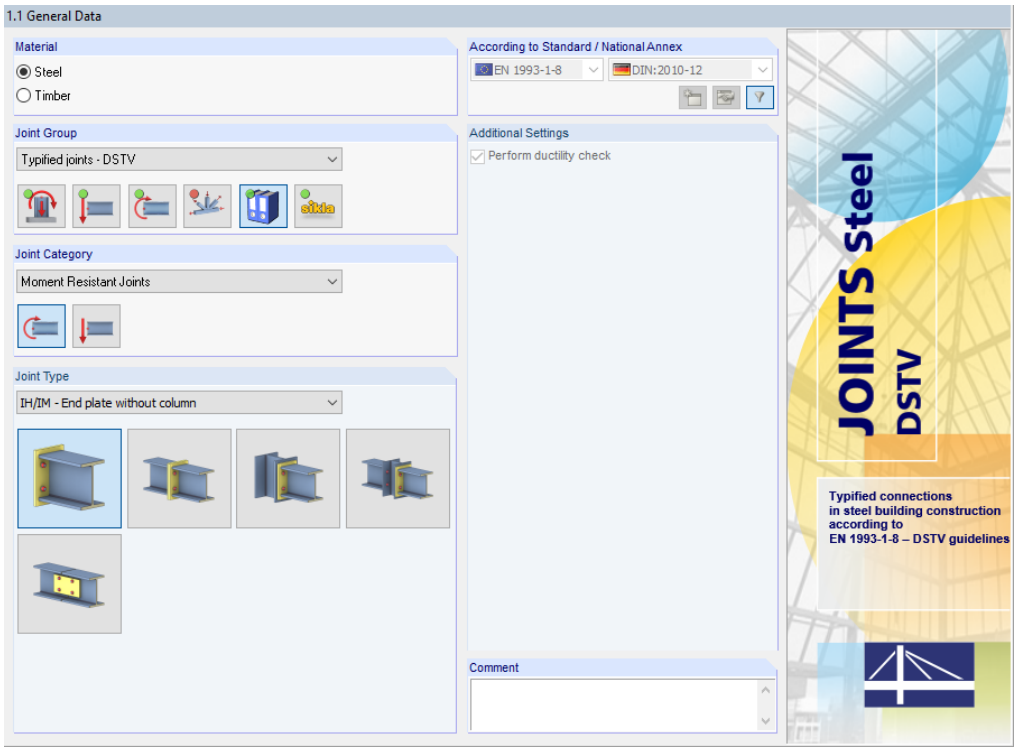


Image 7.2 Window *1.1 General Data*

Joint Category

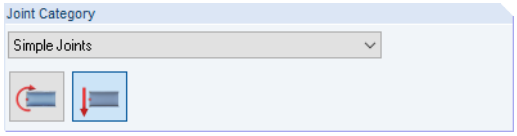
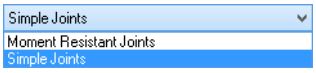
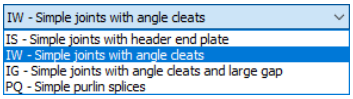


Image 7.3 Joint category



You must specify if the connection is a *Simple Joint* or a *Moment Resistant Joint*. You can select the category by using the drop-down list or clicking the buttons with the connection icons.

Joint Type



The available options depend on the joint category.

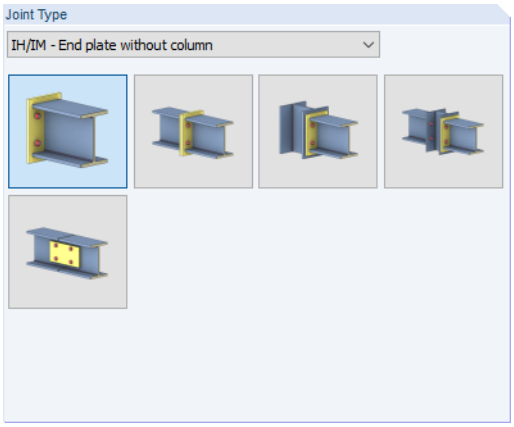


Image 7.4 *Joint Type* section for *Moment Resistant Joints* category

The *Moment Resistant Joints* category provides the following design variants:

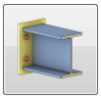
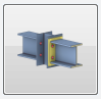
	End plate without column
	Beam joint
	Single-sided beam on column
	Double-sided beam on column
	Purlin joint

Table 7.1 Joint types for moment resistant joints

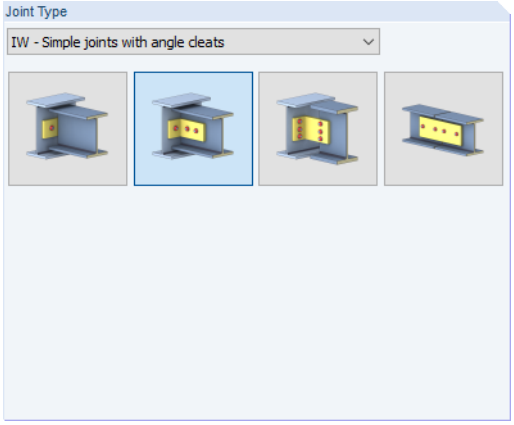


Image 7.5 *Joint Type* section for *Simple Joints* category

The *Simple Joints* category provides the following design variants:

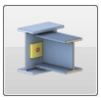
	End plate
	Angle cleats
	Angle cleats and large gap
	Purlin splices

Table 7.2 Joint types for simple joints

Additional Settings

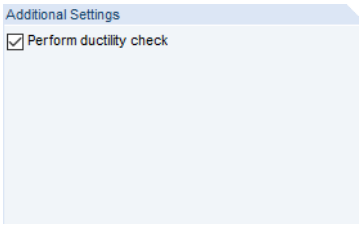


Image 7.6 *Additional Settings* section

The *Perform ductility check* check box is available for simple joints. You can use it to check whether the connection to the load-bearing components fulfills the ductility criterion according to the European Recommendations for the Design of Simple Joints in Steel Structures [6]. This aims to prevent a premature and brittle failure before complete formation of the hinge in the connection.

It is only checked whether the design criterion for the bolts to the plate of the load-bearing structural component is fulfilled. The remaining components of the joint — angles, bolts on the connected structural component, and the connected component itself — are not checked, although they greatly influence the ductility as well!

7.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2.

In the *Parameters* section, you can check the boundary conditions of the connected structural components. If there is an *Invalid cross-section*, you should adjust the cross-section series to match the joint type set in Window 1.1.

If several members such as beams, purlins, and diagonals connect to the node, the redundant members must be set to be *Inactive*.

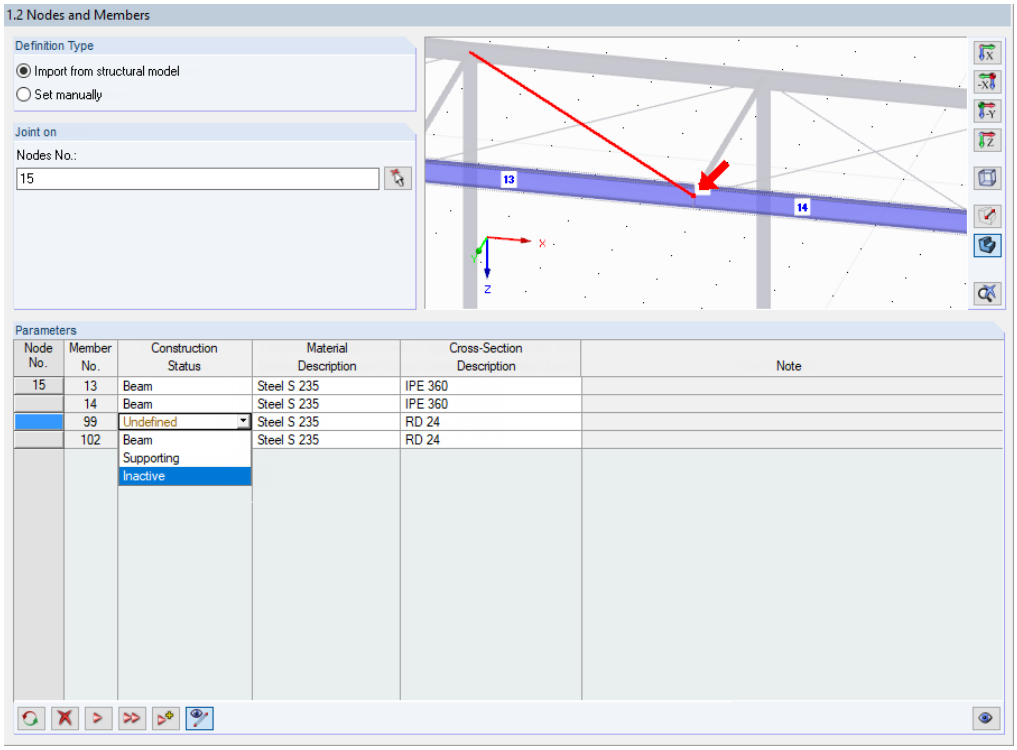


Image 7.7 Setting diagonal member *Inactive*

When the comment "Wrong geometry" is displayed, you have to check whether the joint geometry meets the requirements of the typified connections.

7.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

7.4 Connection Types

In Window *1.4 Connection Types*, you can define the specific input parameters of the connection according to the DSTV (German Steel Construction Association) type catalog.

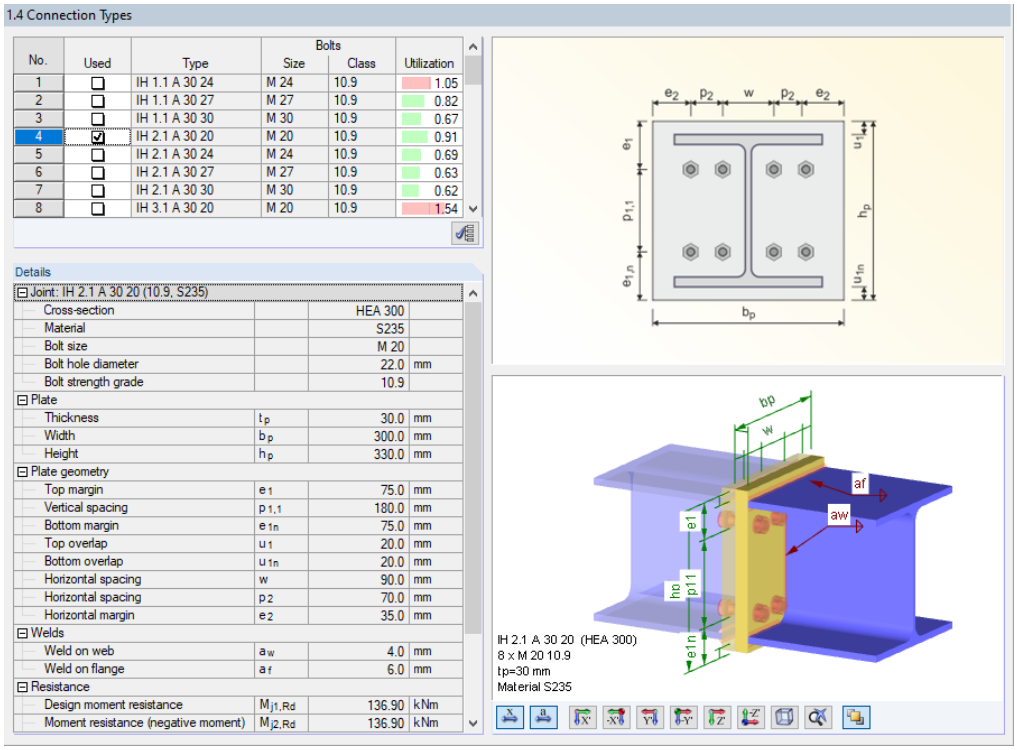


Image 7.8 Window *1.4 Connection Types*

The window is divided into two parts: On the left, the connection parameters are displayed; on the right, they are illustrated by graphics. The upper graphic shows a system sketch of the current parameter, the lower graphic shows a 3D visualization of the connection.

The buttons below the 3D graphic are described in Table 3.1.

In the upper left section, you can see the connection types that are possible according to the DSTV guideline [3] [4]. Each *Type* is characterized by its label and the used *Size* and *Class* of *Bolts*.

The connection type can be specified by using the check mark in the *Used* column. The *Details* section below shows the parameters of this connection. The 3D graphic displays the connection geometry dynamically.

If you click the [Suggest the Best Type] button, RF-/JOINTS performs a quick design of the connection. In the last column, the *Utilization* of each variant is displayed (see figure above). It makes it easier to select the appropriate connection for the analysis.

Details

This section contains all the information about the selected joint, such as the cross-section and plate geometry, bolts, welds, as well as resistances and stiffnesses.

There are additional modification options for some categories.

Connection types IH 3/IH 4

The extension of the end plate can be arranged at the top or bottom by using the *Mirrored plate* option. This specification affects the results because the reverse moment may become governing.

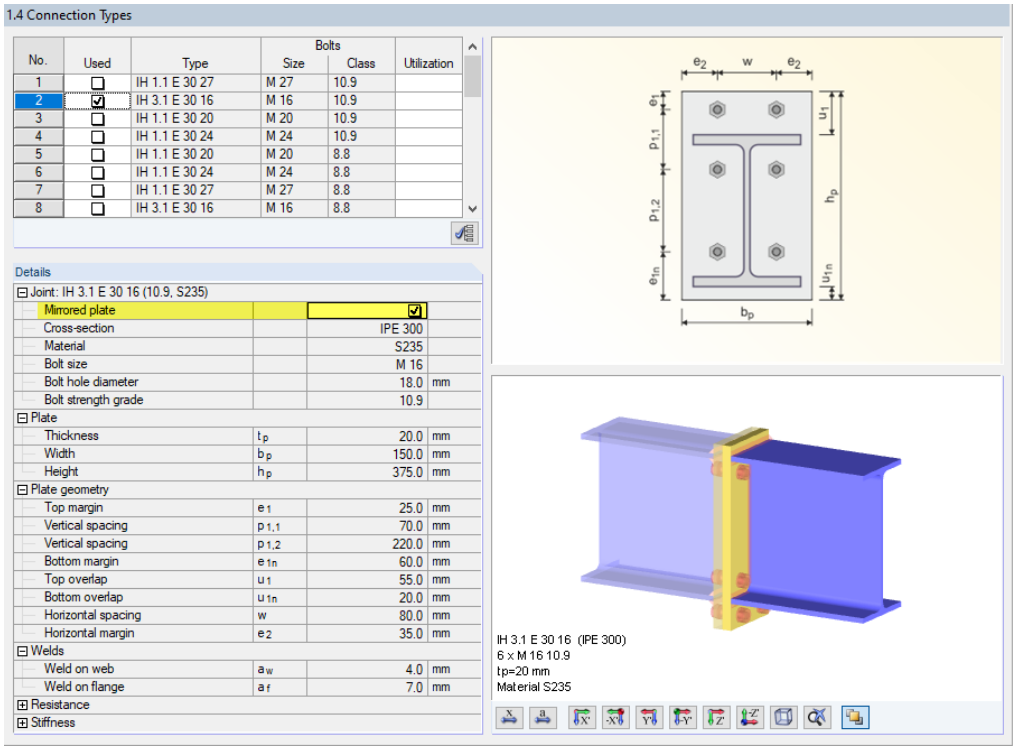
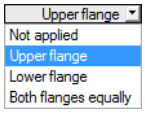


Image 7.9 IH 3 connection with *Mirrored plate* option

Connection types IS/IW



Pinned connections of the types IS (connections with end plate) and IW (connections with angles) can also be combined with the type **IK** (notches).

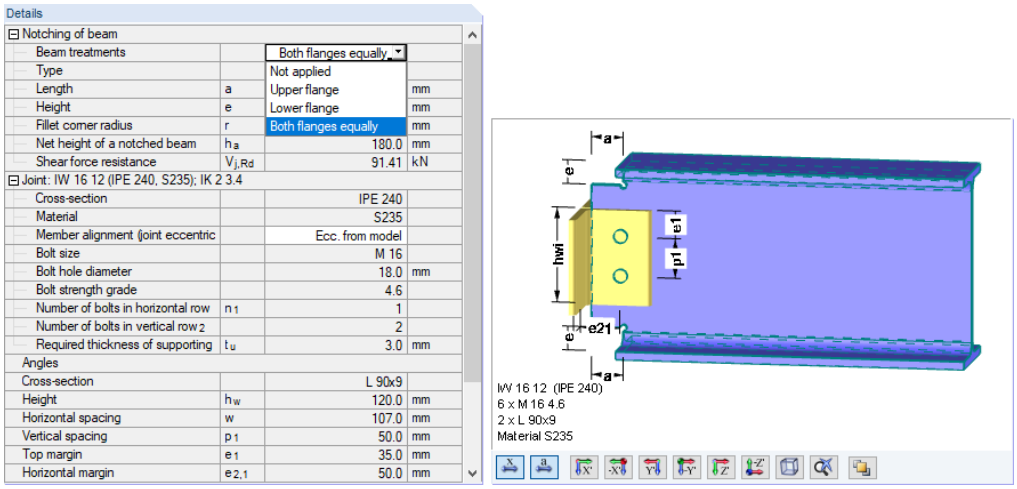


Image 7.10 IW connection with *Beam treatments* for notching (type IK)

In the other boxes you can define the geometry parameters of the notching: the *Length*, *Height*, and *Fillet corner radius*.

Details

⊟ Notching of beam			
Beam treatments		Both flanges equally	
Type		IK 2 3.4	
Length	a	40.0	mm
Height	e	40.0	mm
Fillet corner radius	r	60.0	mm
Net height of a notched beam	h_a	80.0	mm
Shear force resistance	$V_{j,Rd}$	100.0	kN
⊞ Joint: IW 16 12 (IPE 240, S235); IK 2 3.4		120.0	
Angles		150.0	
Cross-section		L 90x9	

Image 7.11 Selection of geometry parameters for notching

The resistance of the end plate or angle connection (IS/IW) and the resistance of the notching (IK) are analyzed for the design. The smaller of the two values is governing.

8 Steel - Sikla

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Steel - Sikla** module. The general input parameters are described in Chapter 2.

The module's joints are matched to the sections of the support system manufacturer Sikla. These sections are available for selection in the cross-section library of RFEM or RSTAB among the rolled square and rectangular hollow sections (see also Figure 8.9).

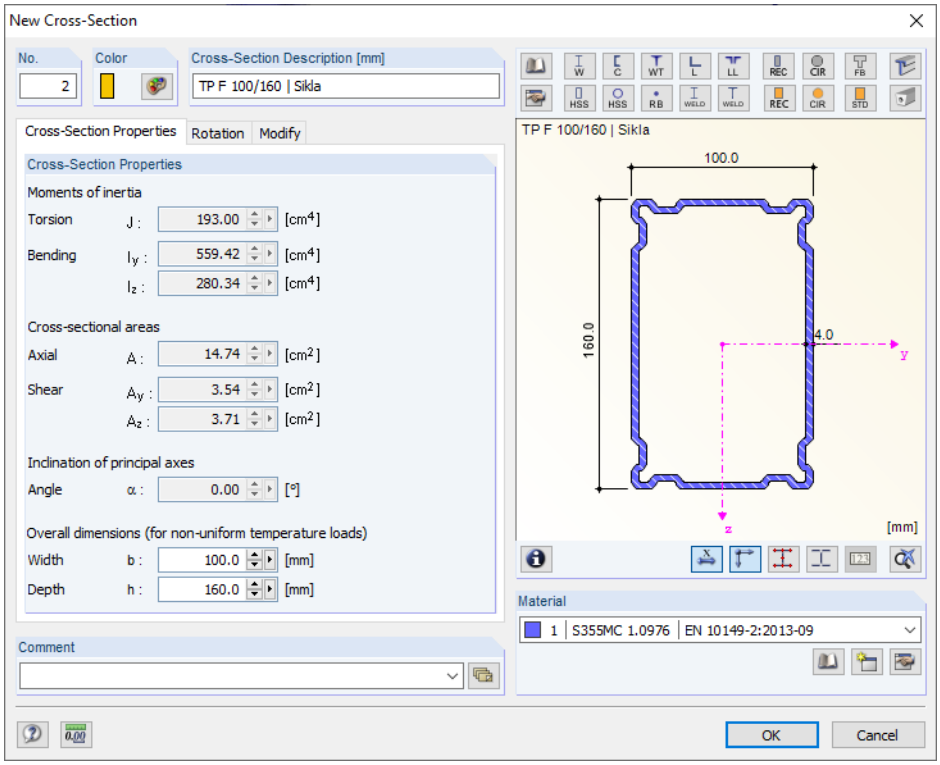


Image 8.1 Sikla cross-section in RFEM/RSTAB

The input windows of the add-on module are accessible once you select the material *Steel* and the joint group *Typified joints - Sikla*.

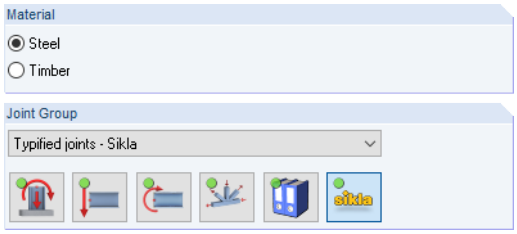
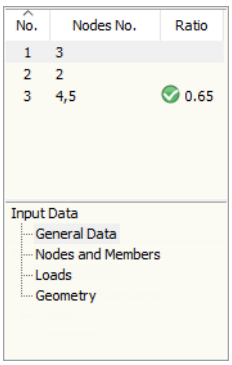


Image 8.2 *RF-/JOINTS Steel - Sikla* add-on module



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the *Geometry* entry is missing in the navigator, check whether the boundary conditions for entering the connection are correct in Window *1.2 Nodes and Members*. For example, it may be necessary to deactivate connected members for the design (see Figure 7.7).

8.1 General Data

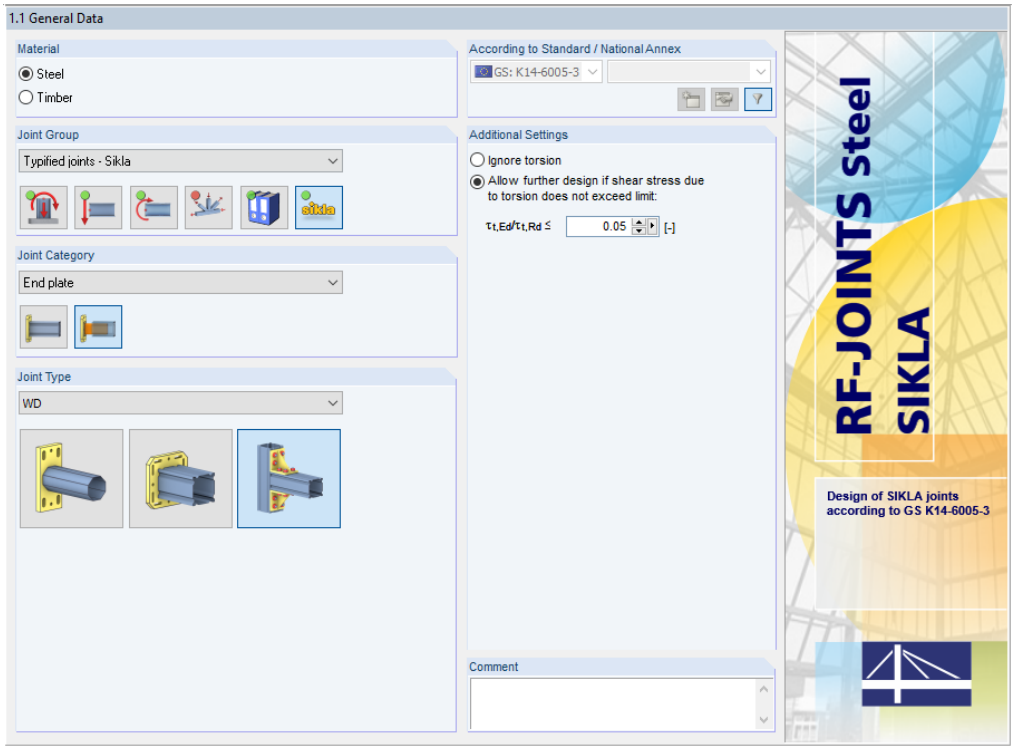


Image 8.3 Window *1.1 General Data*

Joint Category

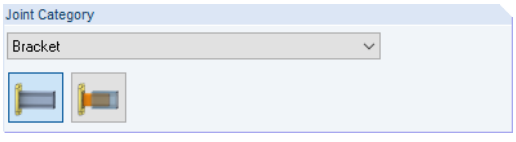


Image 8.4 Joint category

You have to specify whether the joint is a *Bracket* or an *End plate*. Sikla refers to the latter as an "End Support". You can select the category by using the drop-down list or clicking the buttons with the connection icons.

Joint Type

The available options depend on the joint category.

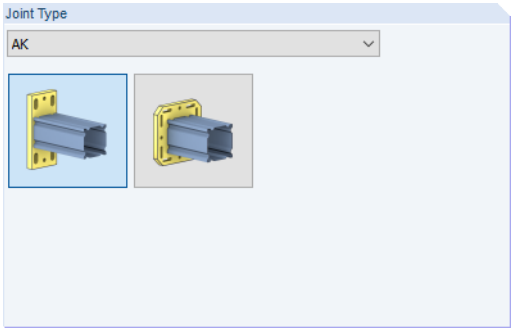


Image 8.5 *Joint Type* section for *Bracket* category

The *Bracket* category provides the following design variants:

	Cantilever bracket — type *AK*
	Beam bracket — type *TKO*

Table 8.1 Joint types for brackets

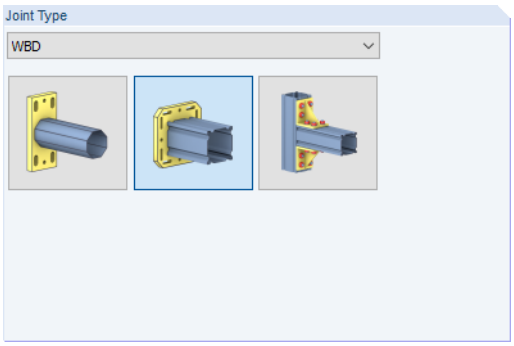
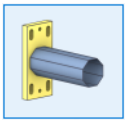
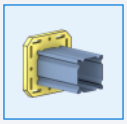


Image 8.6 *Joint Type* section for *End plate* category

The *End plate* category provides the following design variants:

	End support — type *STA*
	End support — type *WBD*

	Corner bracket — type *WD*

Table 8.2 Joint types for end plates

According to Standard

There are no setting options in this section. The connections are designed according to German Expert Opinion (*GS*) with Test Report No. K14-6005-3.

Sikla's installation guidelines for connections contain information about the permissable load capacity of typical Sikla structures for building equipment and industrial and plant construction. They are available for download on the manufacturer's website.

Additional Settings

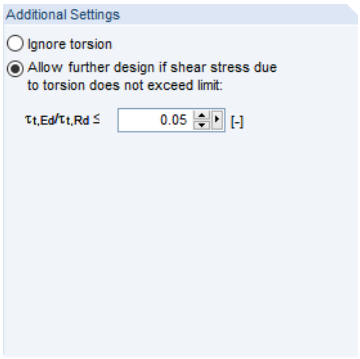


Image 8.7 *Additional Settings* section

EN 1993 does not give any clear recommendations for scheduled torsion. The two check boxes allow you to completely *Ignore* the torsional stresses or neglect them up to a user-defined *limit*. 5% is preset as the maximum ratio of the existing torsional stress $\tau_{t,Ed}$ to the torsional shear resistance $\tau_{t,Rd}$. If the value is exceeded during the design, a corresponding error message appears.

8.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2.

In the *Parameters* section, you can check the boundary conditions of the connected structural components. If there is an *Invalid cross-section*, you should adjust the cross-section series to match the joint type set in Window 1.1.

If several members such as brackets and lifting beams connect to the node, the redundant members must be set to be *Inactive*.

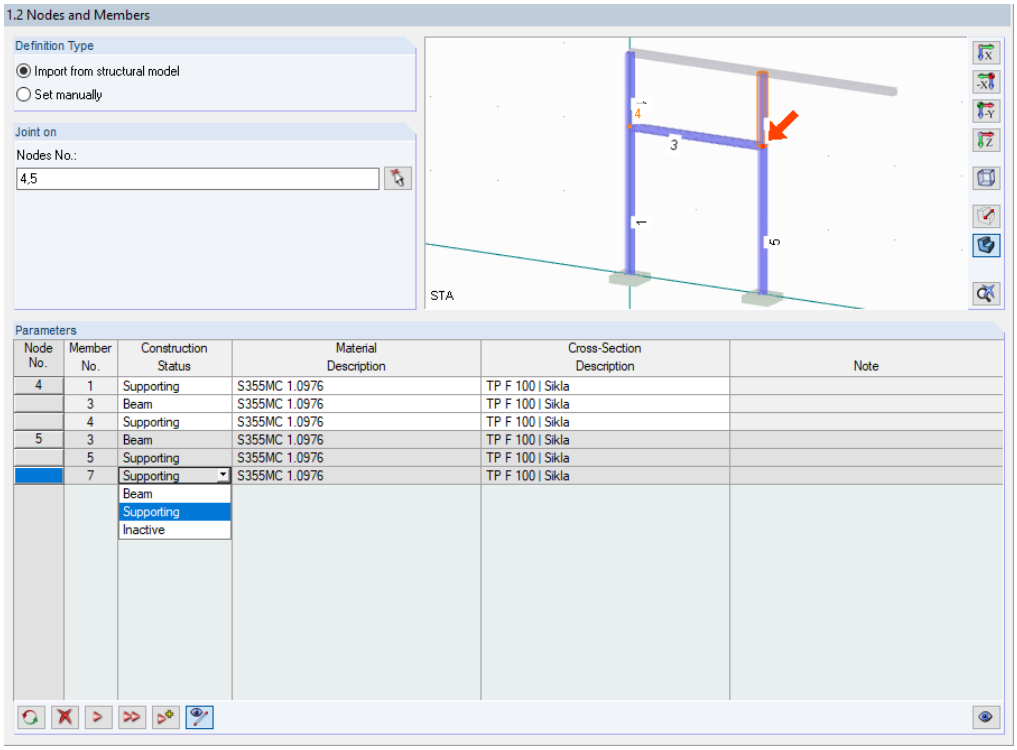


Image 8.8 Adjusting the *Construction Status*

Material Description
S355MC 1.0976

The only possible material is **S355MC 1.0976** according to EN 10149-2, since Sikla products are exclusively manufactured in this steel grade. Therefore, the model should be created with the appropriate material in RFEM or RSTAB. The analysis in the RF-/JOINTS Steel - Sikla module does not allow for any variants.

The Sikla cross-sections can be selected in the section library among the rolled square and rectangular hollow sections.

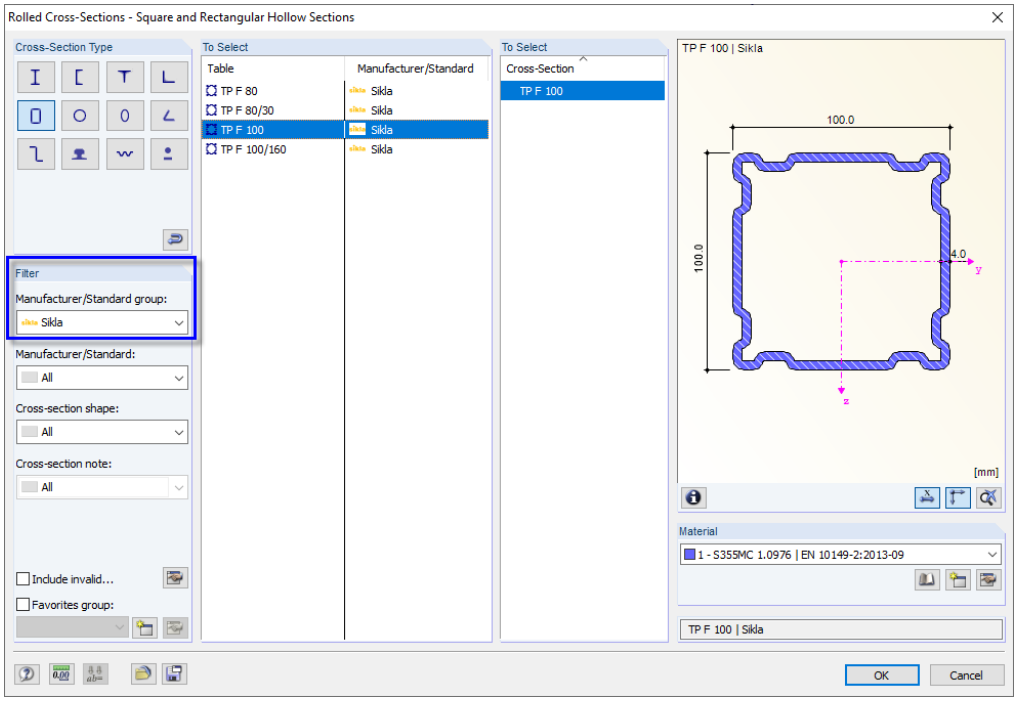


Image 8.9 Cross-section database for Sikla sections

When the comment "Wrong geometry" is displayed, you have to check whether the joint geometry meets the requirements of Sikla systems.

8.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

8.4 Geometry

In Window *1.4 Geometry,* you can define the input parameters of the connection according to the Sikla catalog.

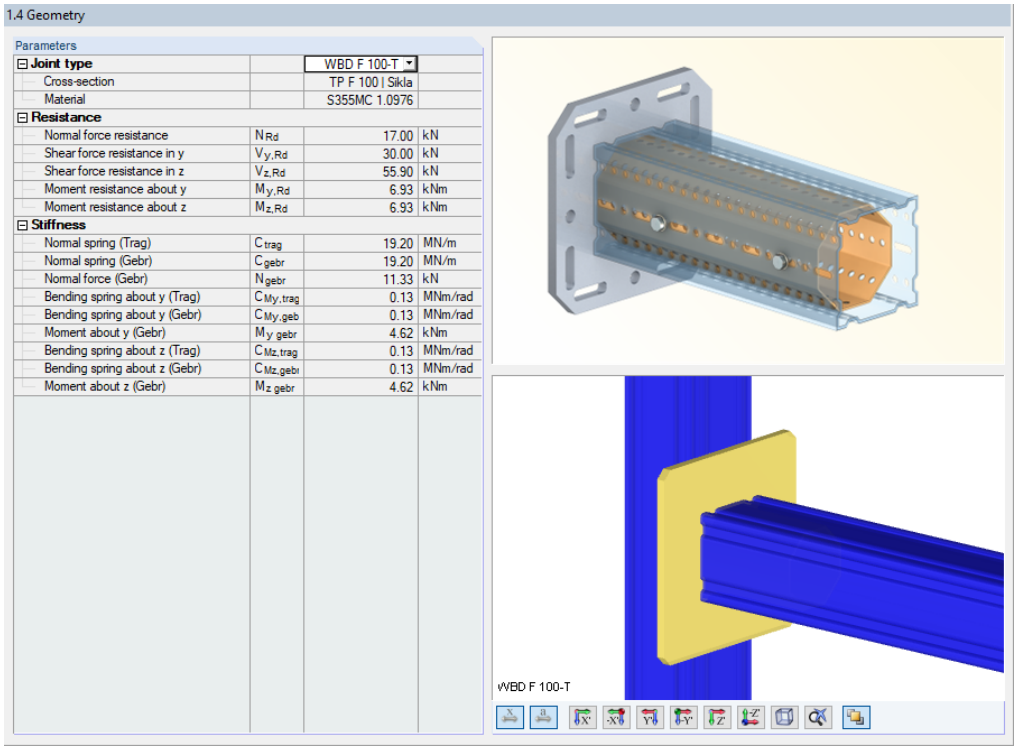
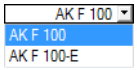


Image 8.10 Window *1.4 Geometry*

The window is divided into two parts: On the left, the parameters of the connection are displayed; on the right, they are illustrated by graphics. The upper graphic shows a system sketch of the current joint type, the lower graphic shows a 3D visualization of the connection.

The buttons below the 3D graphic are described in Table 3.1.



The *Joint type* section shows the connection types that are possible according to Sikla's type catalog. Each connection is characterized by its type code.

For the selected joint type, the *Resistance* and *Stiffness* are specified according to the technical approval.

8.5 Details

In the *Details* dialog box, you can specify additional settings for the design (see Figure 9.35). This dialog box is available in every input window using the [Details] button.

The *Details* dialog box is described in Chapter 9.6.

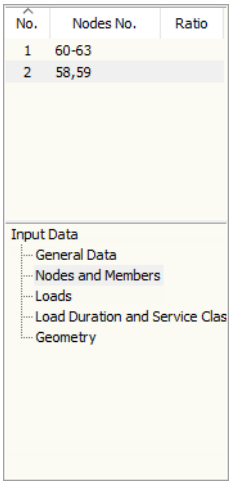
9 Timber - Steel to Timber

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Timber - Steel to Timber** module. The general input parameters are described in Chapter 2.

The functionality of this add-on module is presented in a Dlubal webinar:
https://www.dlubal.com/en-US/support-and-learning/learning/videos/000793

The input windows of the add-on module are accessible once you select the material *Timber* and the joint group *Steel to timber connection*.

Image 9.1 *RF-/JOINTS Timber - Steel to Timber add-on module*



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the entries *Load Duration and Service Class* and *Geometry* are missing in the navigator, go to Window *1.2 Nodes and Members* and check if the boundary conditions of the node are correct and if load cases are available for the design. For example, it might be necessary to adjust the status of the connected members (see Figure 9.16).

9.1 General Data

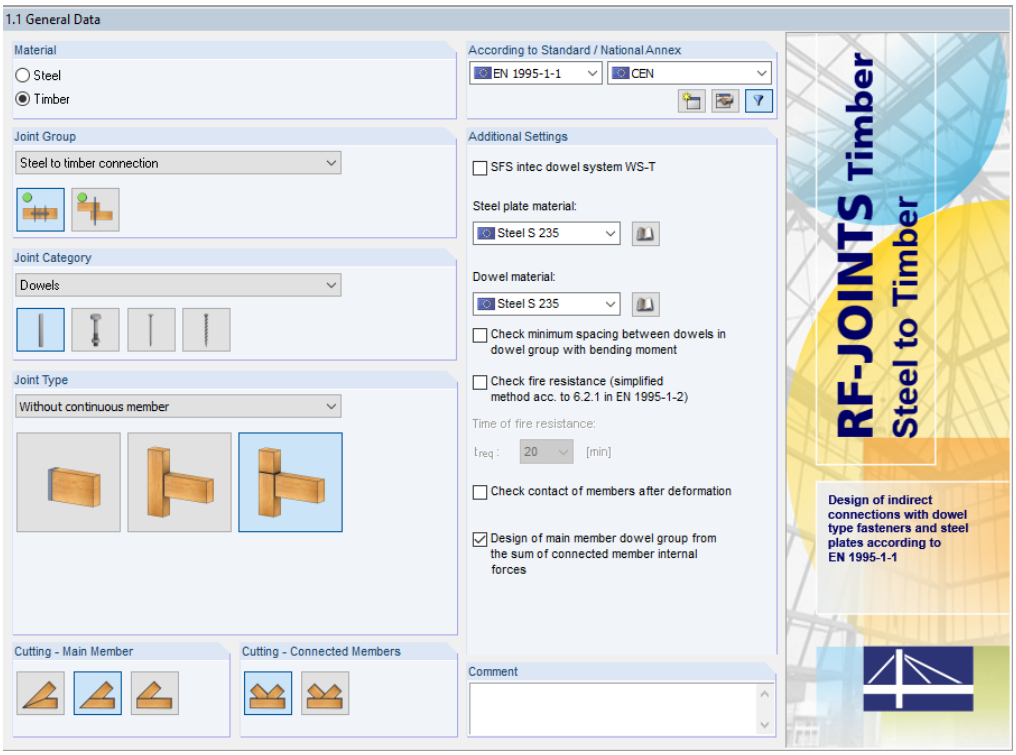
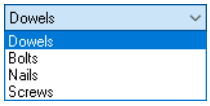


Image 9.2 Window *1.1 General Data*

Joint Category



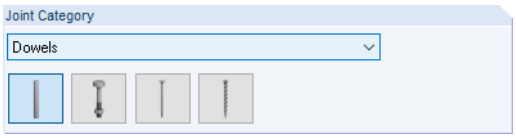


Image 9.3 Joint category

In the list of steel-to-timber joint categories, you can select the following fasteners: *Dowels*, *Bolts*, *Nails*, and *Screws*.

Joint Type

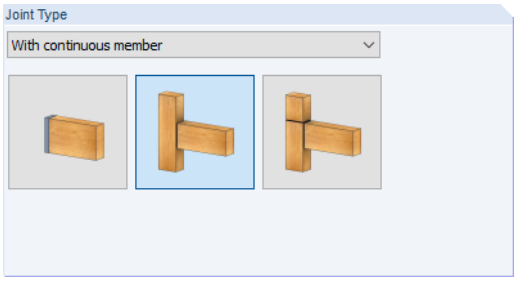
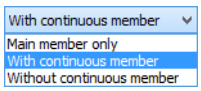


Image 9.4 Joint type



The following joint types are available:

	Main member only	Fastening of a member to an existing structural component (or any number of members) at any angle
	With continuous member	Connection of a maximum of six diagonals to a continuous beam (*Main member* and *Continuous member*)
	Without continuous member	Free definition of a node with up to eight connected members

Table 9.1 Dowel joint types

Cutting - Main Member

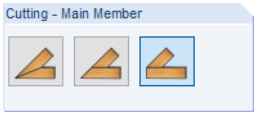


Image 9.5 Cutting - Main Member

The secondary member can be connected symmetrically or asymmetrically to the main member.

	Symmetrical connection at an angle of 45°
	Main member cut by secondary member
	Main member cut perpendicularly to member axis

Table 9.2 Connection options for main member

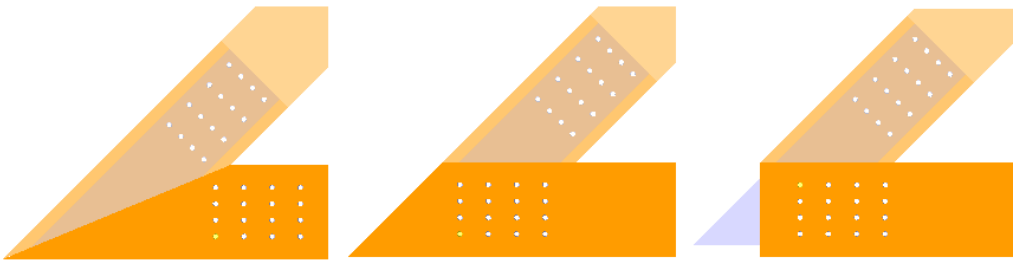


Image 9.6 Symmetrical connection (left), cut by secondary member (center) and cut perpendicularly (right)

Cutting - Connected Members

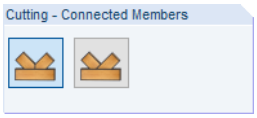


Image 9.7 Cutting - Connected Members

When connecting two web members, the cut can be symmetrical or continuous.

	Symmetrical connection
	Continuous connection

Table 9.3 Connection options for secondary member

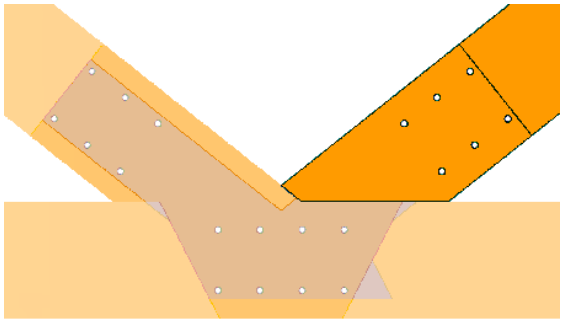


Image 9.8 Continuous connected member

According to Standard / National Annex

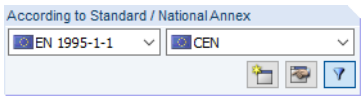
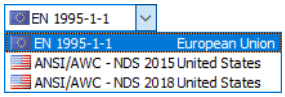


Image 9.9 *According to Standard / National Annex section*



Selecting the standard

The design-relevant coefficients are specified according to the standard (EN 1995-1-1 [2] or ANSI/AWC [5]) and, if applicable, the National Annex (see Figure 2.19). If you want to apply user-defined factors for the designs, you have to create a new standard or National Annex first by clicking the button. Afterwards, the coefficients can be customized in the *Standard Settings* or *National Annex Settings* dialog box.

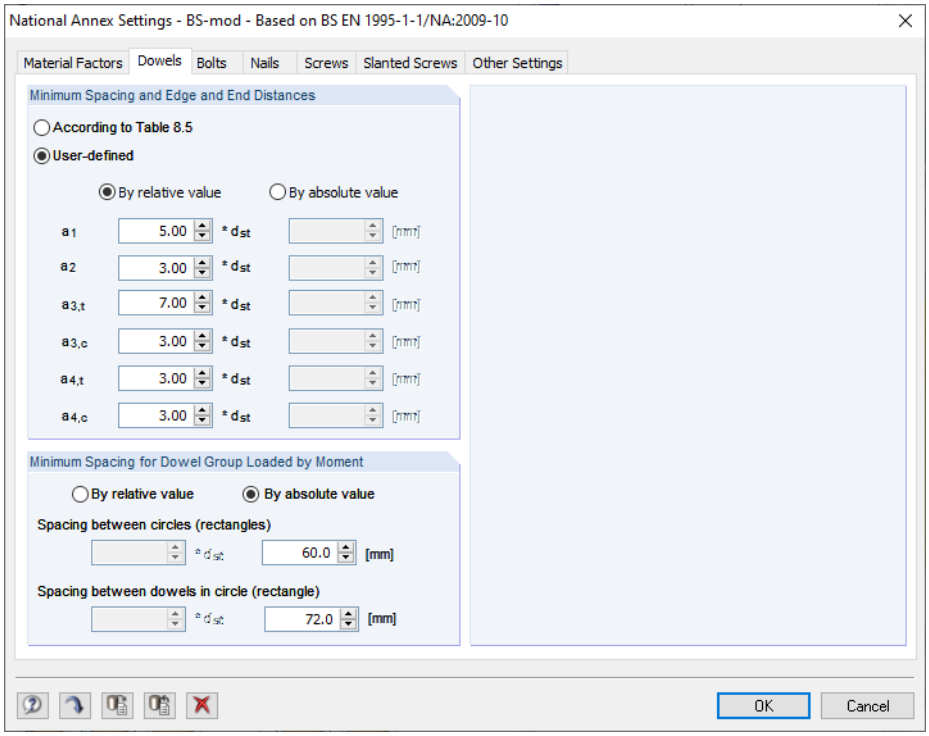


Image 9.10 *National Annex Settings* dialog box, *Dowels* tab

In the *Dowels*, *Bolts*, *Nails*, and *Screws* tabs you can define user-defined minimum distances between the dowels, bolts, and nails, as well as the moment-stressed dowel, bolt, or nail groups. Adjustments are necessary, for example, for the connection system of the company BSB, whose technical approval is based on other values. User-defined minimum distances between dowel groups are useful for dowel circles of a frame joint, for example. They are not clearly defined in the design standard.

In the *Other Settings* tab, you can adjust the shear correction factors k_{cr}, if necessary.

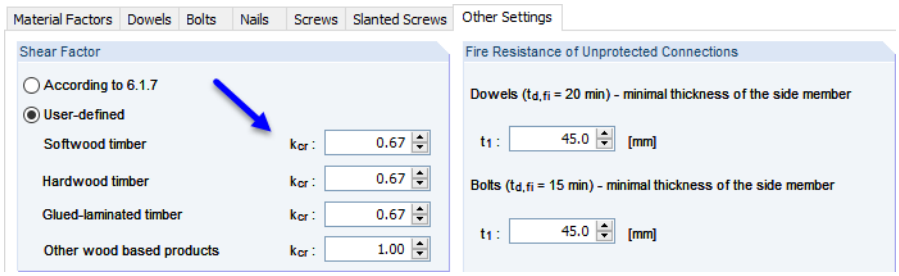


Image 9.11 *Other Settings* tab

Additional Settings

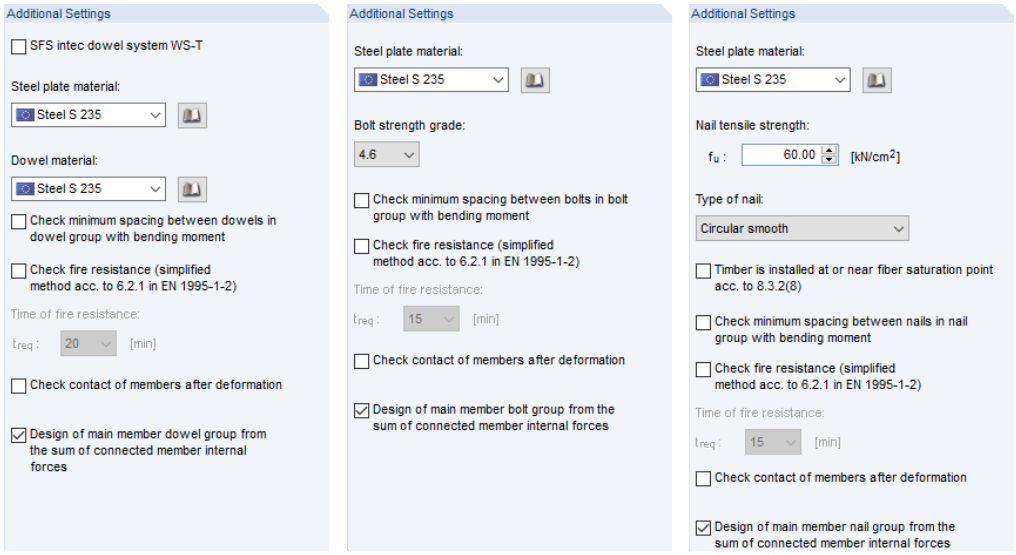
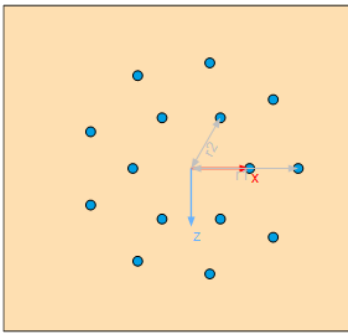


Image 9.12 *Additional Settings* sections for *Dowels, Bolts/Screws,* and *Nails*

You can use the lists and buttons to define the material grades of the slotted steel plate and the dowels, screws, or nails.

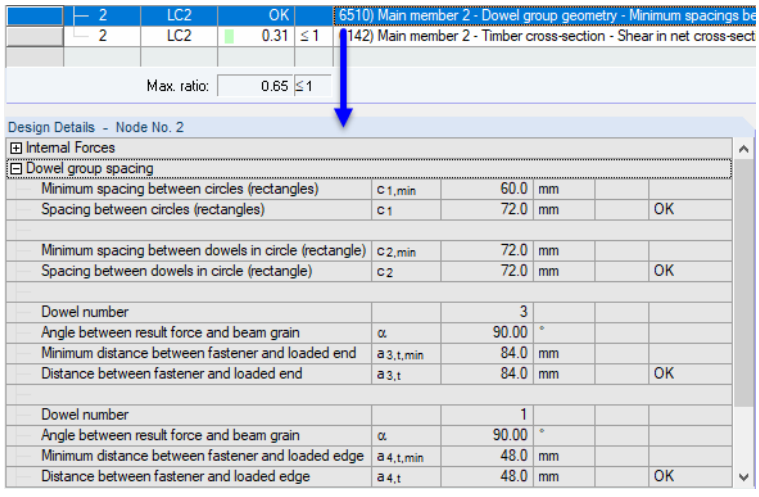
If the *SFS intec dowel system WS-T* is used, the material grades are unmodifiably preset according to the manufacturer's technical approval.



Distance between dowel circles

When you select the *Check minimum spacing between dowels/bolts/nails in dowel/bolt/nail group with bending moment* option, the minimum distances between individual fastener groups are also checked during the calculation. This applies to both circular and rectangular joints. The calculation of the minimum distances between the individual groups is not regulated by standards. Therefore, the minimum distance is checked according to [7] in this case. This value can be adjusted in the *National Annex Settings* dialog box (see Figure 9.10).

After the calculation, the design of all minimum distances is shown in the details.



2	LC2	OK		(6510) Main member 2 - Dowel group geometry - Minimum spacings be		
2	LC2	0.31	≤ 1	(142) Main member 2 - Timber cross-section - Shear in net cross-sect		
Max. ratio:		0.65	≤ 1			

Design Details - Node No. 2

⊞ Internal Forces					
⊟ Dowel group spacing					
Minimum spacing between circles (rectangles)	$c_{1,min}$	60.0	mm		
Spacing between circles (rectangles)	c_1	72.0	mm		OK
Minimum spacing between dowels in circle (rectangle)	$c_{2,min}$	72.0	mm		
Spacing between dowels in circle (rectangle)	c_2	72.0	mm		OK
Dowel number		3			
Angle between result force and beam grain	α	90.00	°		
Minimum distance between fastener and loaded end	$a_{3,t,min}$	84.0	mm		
Distance between fastener and loaded end	$a_{3,t}$	84.0	mm		OK
Dowel number		1			
Angle between result force and beam grain	α	90.00	°		
Minimum distance between fastener and loaded edge	$a_{4,t,min}$	48.0	mm		
Distance between fastener and loaded edge	$a_{4,t}$	48.0	mm		OK

Image 9.13 Details for design no. *6510*: Checking minimum spacings

With the *Check contact of members after deformation* option, you can check the deformation of the entire connection. To do this, specify the distance between the timbers in Window *1.5 Geometry* using the parameter o_g.

Dowel group settings			
Pattern		Rectangle	
Diameter	d_{st}	10.0	mm
Dowel length	l_{st}	160.0	mm
Plug length	l_{plug}	0.0	mm
Number of dowel columns (x-direction)	n_{dx}	4	
Number of dowel rows (z-direction)	n_{dz}	3	
Staggered rows		☐	
Method of dowel group placement		Minimum edge distance	
Orientation of rows and columns		Basic	
Dowel group reinforced by screws (n_{ef} = n)		☐	
Member settings			
Section cut offset	o_g	10.0	mm
Member eccentricity in X-direction	X	70.0	mm
Member eccentricity in Z-direction	Z	130.0	mm

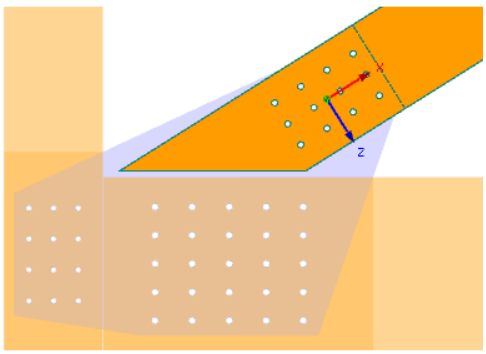


Image 9.14 Specification for *Check contact of members after deformation*

Design no. 6530 checks whether the total deformation of the connection is greater than the specified distance o_g. If the design is not fulfilled, the members are in contact.

These contact properties are not automatically considered by the RF-/JOINTS module! Therefore, appropriate measures must be taken in case of a design failure.

The *Design of main member dowel/bolt/nail group from the sum of connected member internal forces* check box controls whether the design is performed using the individual internal forces of the member ends or whether the resulting internal force is used that consists of two or more members connected to the node. For the *Main member* joint type, this option is inevitably not available.

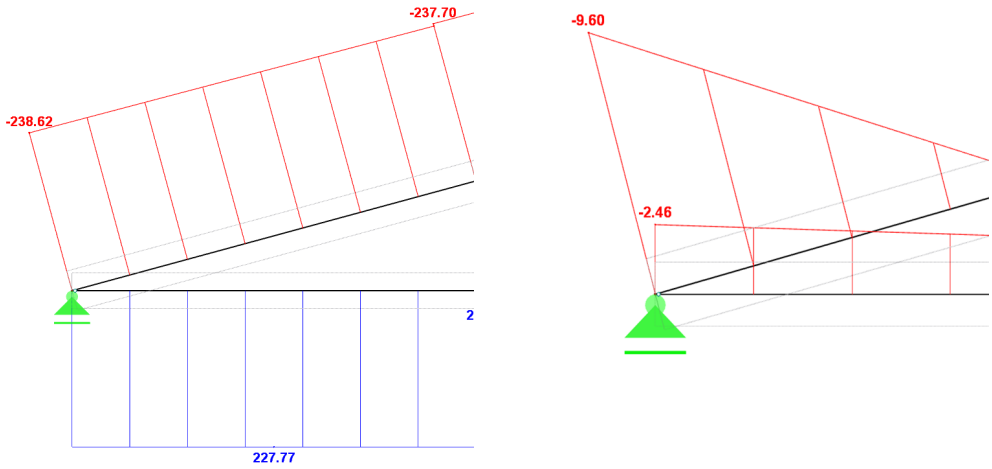


Image 9.15 Internal forces on member

In the constellation of internal forces in the figure above, the axial force -238.62 kN and the shear force -9.60 kN are used for the design in the top flange **without** the *sum of connected member internal forces* - for example, if the axial force of the connection is introduced directly into the support.

With the *sum of connected member internal forces* option, however, the resulting axial and shear force are used for the design of the eaves node.

9.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2.

In the *Parameters* section, you can check the boundary conditions of the connected structural components. You can also adjust the *Construction Status*, if necessary.

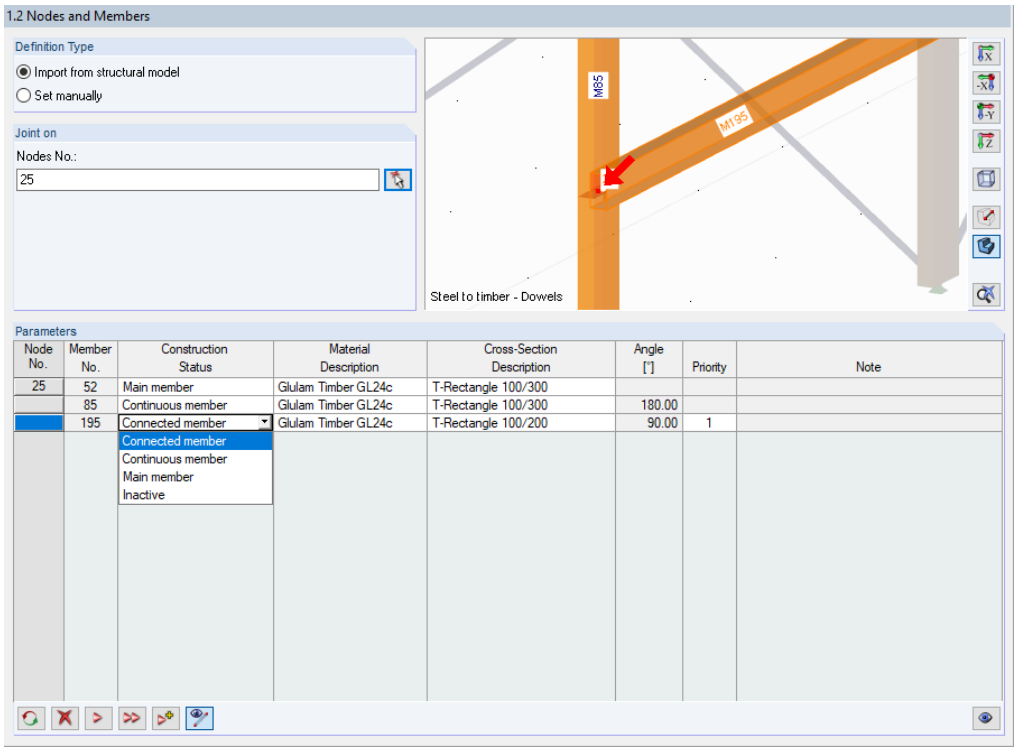


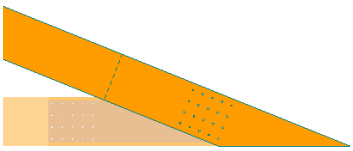
Image 9.16 Window *1.2 Nodes and Members* - adjusting the status of members

The *Angle* and *Priority* columns provide information about the connected members.

The angles are based on the geometric conditions of the RFEM/RSTAB model. If you select the *Set manually* definition type in Window 1.2, you can enter user-defined angles of the connected members.

When you click on a row of the table, the selected member is highlighted in the graphic.

This window is important for defining the priority of a connection. If you want the top flange of the eaves node to be continuous, as shown in the figure on the left, you have to define it as the main member (the *Without continuous member* joint type must have been set in Window 1.1).



Eaves node joint

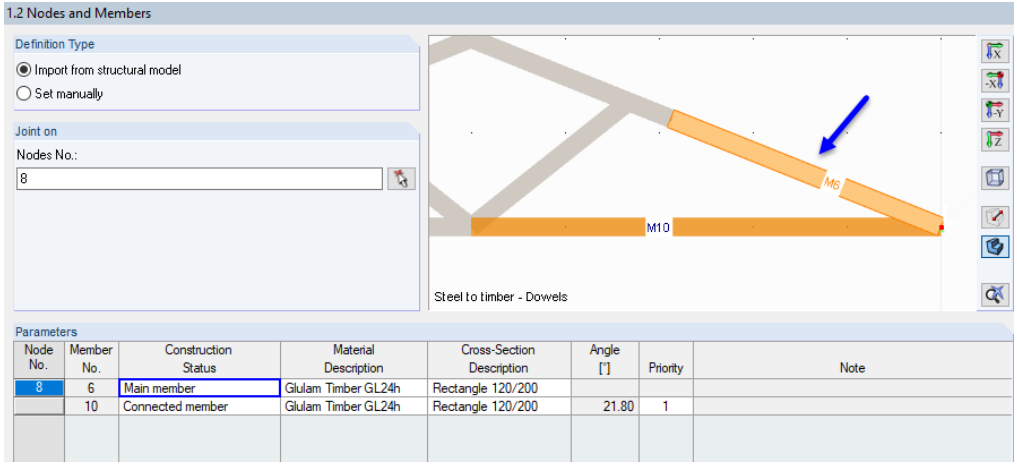
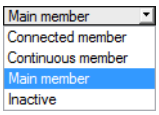


Image 9.17 Eaves node with continuous top flange



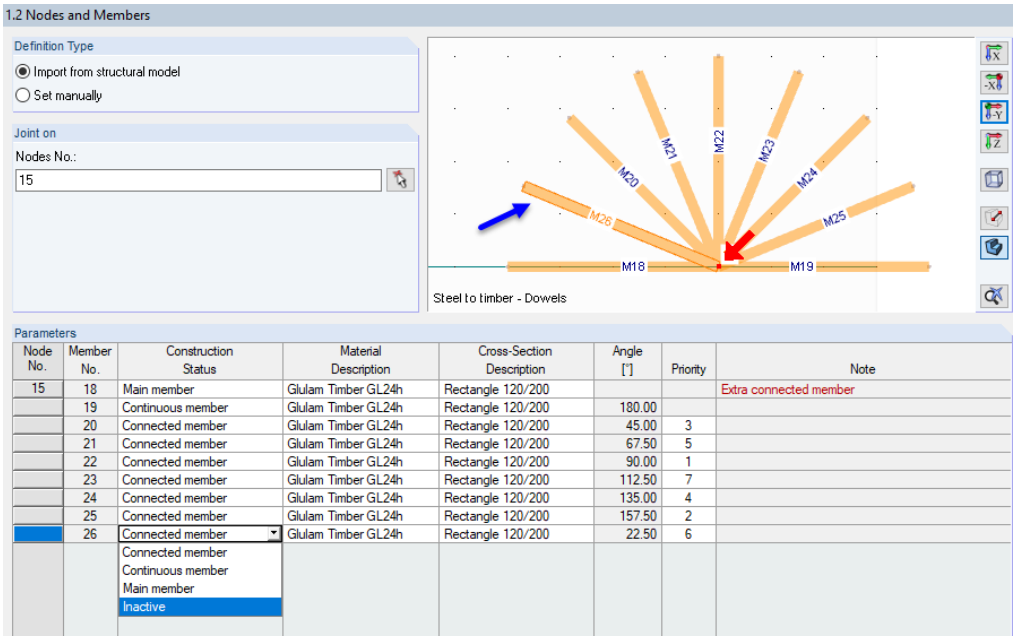
If more than two members are connected to a node, the following definition options are available for the *Construction Status* of the members:

- Main member - controls all other members as well as the cutting and its priority
- Continuous member - only available for the *With continuous member* joint type
- Connected member - assigns a lower priority to the member (e.g. strut, post)
- Inactive - excludes the member from the design

When defining main and connecting members, it is necessary to observe certain geometric conditions:

- Maximum number of connected members: 8
- Minimum length of a member: 42 cm
- Minimum angle between members: 15°

In Figure 9.18, more than eight members are connected to the node. Furthermore, the angle of member 26 is too small. The geometry of the connection is only functional if member 26 is set to be *Inactive*.



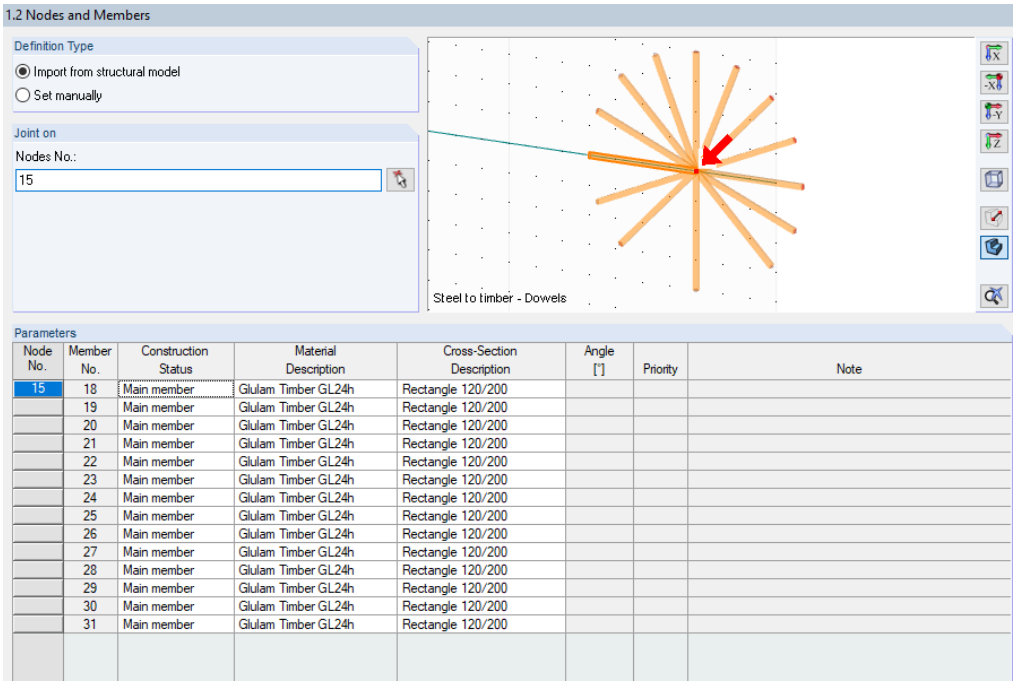
Node No.	Member No.	Construction Status	Material Description	Cross-Section Description	Angle [°]	Priority	Note
15	18	Main member	Glulam Timber GL24h	Rectangle 120/200			Extra connected member
	19	Continuous member	Glulam Timber GL24h	Rectangle 120/200	180.00		
	20	Connected member	Glulam Timber GL24h	Rectangle 120/200	45.00	3	
	21	Connected member	Glulam Timber GL24h	Rectangle 120/200	67.50	5	
	22	Connected member	Glulam Timber GL24h	Rectangle 120/200	90.00	1	
	23	Connected member	Glulam Timber GL24h	Rectangle 120/200	112.50	7	
	24	Connected member	Glulam Timber GL24h	Rectangle 120/200	135.00	4	
	25	Connected member	Glulam Timber GL24h	Rectangle 120/200	157.50	2	
	26	Connected member	Glulam Timber GL24h	Rectangle 120/200	22.50	6	

Connected member
Continuous member
Main member
Inactive

Image 9.18 Functional connection geometry via inactive member

With the status *Main member* for all members, you can even connect more than eight members to the node (see Figure 9.19). This is advantageous if you only want to design the connection in a complex joint geometry.

For the joint type *Main member,* you can freely select the inclination, cutting, and eccentricity.



Node No.	Member No.	Construction Status	Material Description	Cross-Section Description	Angle [°]	Priority	Note
15	18	Main member	Glulam Timber GL24h	Rectangle 120/200			
	19	Main member	Glulam Timber GL24h	Rectangle 120/200			
	20	Main member	Glulam Timber GL24h	Rectangle 120/200			
	21	Main member	Glulam Timber GL24h	Rectangle 120/200			
	22	Main member	Glulam Timber GL24h	Rectangle 120/200			
	23	Main member	Glulam Timber GL24h	Rectangle 120/200			
	24	Main member	Glulam Timber GL24h	Rectangle 120/200			
	25	Main member	Glulam Timber GL24h	Rectangle 120/200			
	26	Main member	Glulam Timber GL24h	Rectangle 120/200			
	27	Main member	Glulam Timber GL24h	Rectangle 120/200			
	28	Main member	Glulam Timber GL24h	Rectangle 120/200			
	29	Main member	Glulam Timber GL24h	Rectangle 120/200			
	30	Main member	Glulam Timber GL24h	Rectangle 120/200			
	31	Main member	Glulam Timber GL24h	Rectangle 120/200			

Image 9.19 Connection of many members with status *Main member*

9.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

9.4 Load Duration and Service Class

The moisture-dependent change in strength of the anisotropic material timber is determined by means of the load duration class (LDC) and the service class (SECL).

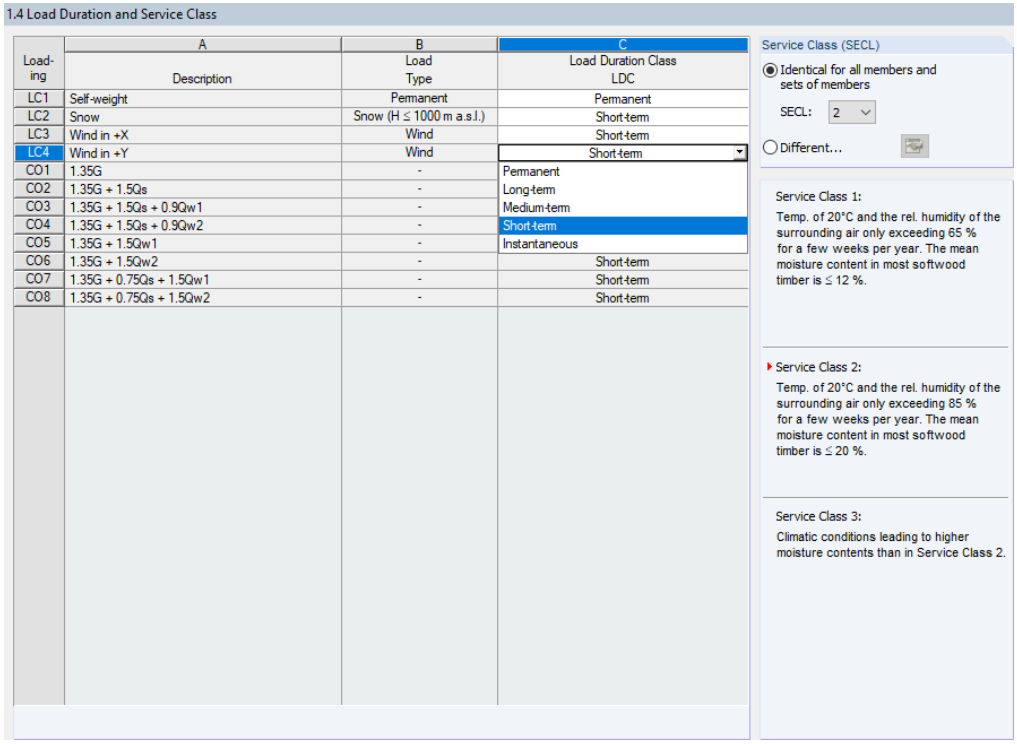


Image 9.20 Window *1.4 Load Duration and Service Class*

Loading

All actions that were selected for design in Window *1.3 Loads* are listed. For combinations, the contained load cases are specified as well.

Description

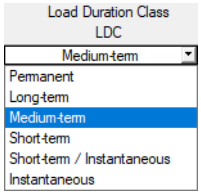
The load case descriptions make the classification easier.

Load Type

This column shows the action types of the load cases as they were defined in RFEM or RSTAB during their creation. They form the basis for the default settings in the next column.

Load Duration Class LDC

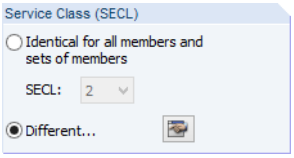
The designs require you to assign the loads and their superpositions to particular load duration classes. The classification of actions is described in [2] Table 2.1.



For load cases and result combinations, you can change the load duration using the list. For load combinations and *Or* result combinations, RF-/JOINTS performs the classification automatically, taking into account the respective leading action or the contained load cases.

The load duration class is required to determine the modification factor k_{mod} which affects the strength properties of the material (see [2] Table 3.1). The coefficients k_{mod} can be checked in the *National Annex Settings* dialog box and adjusted if necessary (see Figure 2.19).

Service Class (SECL)



The classification into service classes makes it possible to assign strength parameters in consideration of environmental conditions. The service classes are specified in [2] Section 2.3.1.3, for example.

By default, all members are assigned to the same service class. To classify objects into different service classes, select the *Different* check box. You can then use the button to open the following dialog box.

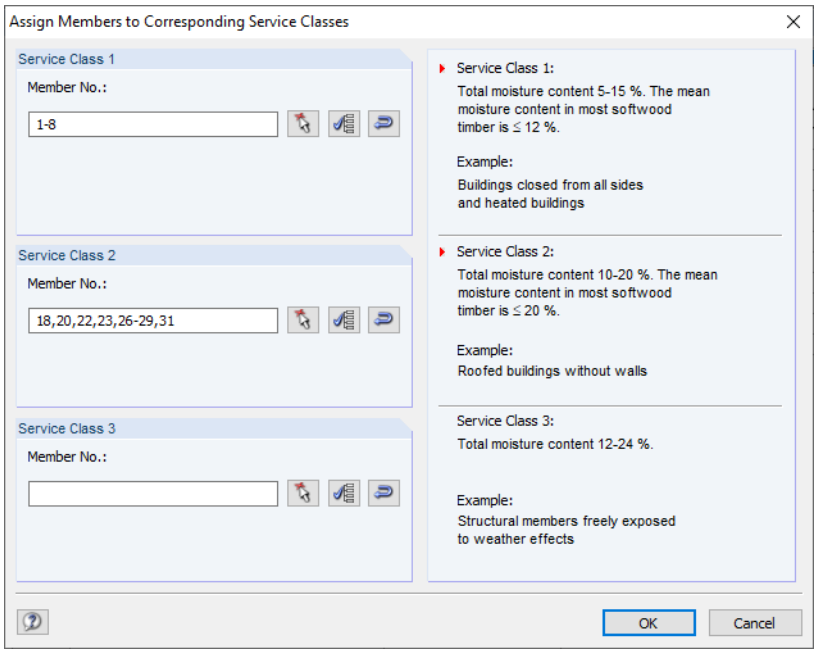


Image 9.21 *Assign Members to Corresponding Service Classes dialog box*

It allows you to individually classify the members into service classes. The buttons next to the text boxes facilitate the assignment. They have the following functions:

Button	Function
	Allows for graphical selection of members in RFEM/RSTAB work window
	Assigns all members to this service class
	Assigns all members not yet assigned to this service class

Table 9.4 *Buttons in the Assign Members to Corresponding Service Classes dialog box*

9.5 Geometry

In Window *1.5 Geometry*, you can define the steel plate and fastener parameters.

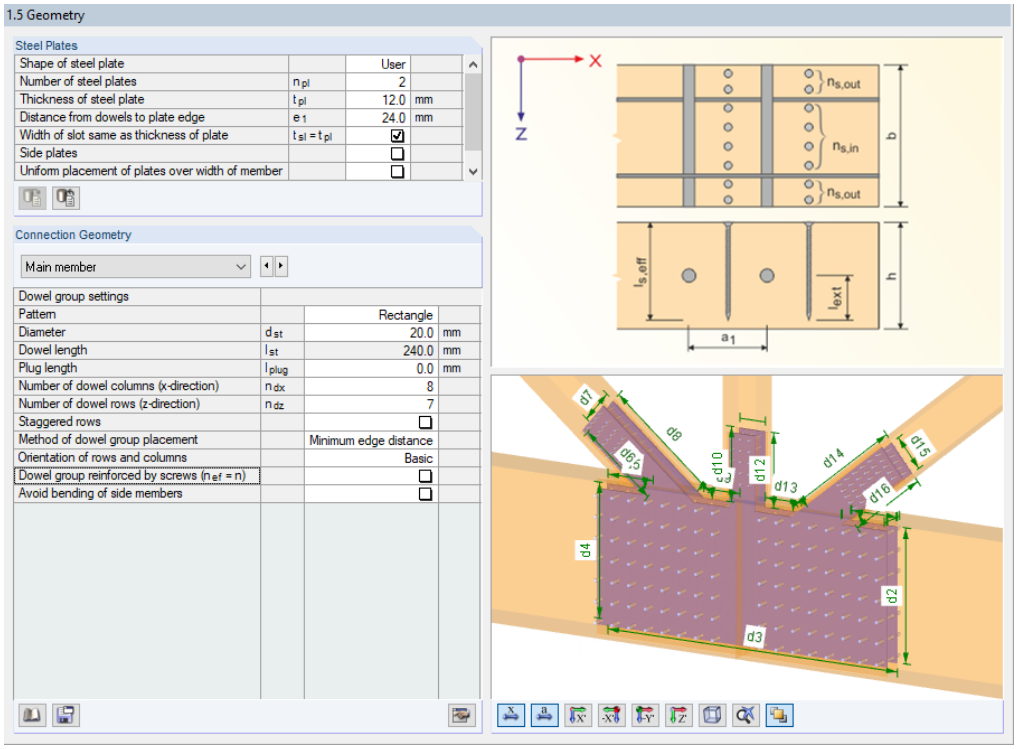


Image 9.22 Window *1.5 Geometry*

This window is divided into two parts: On the left, the input parameters of the connection node are displayed; on the right, they are illustrated by graphics. The upper graphic shows a system sketch of the current parameter, the lower graphic shows a 3D visualization of the node.

The graphic buttons are explained in Table 3.1.

Steel Plates

In this section, you can define the properties of the steel plates. Please note the following.

- A maximum *Number* of five slotted plates is possible.
- The *Thickness of steel plate* must be between 1 mm (for nails) and 40 mm (for SFS: 3 mm).
- The *Distance from fasteners to plate edge* must meet the standard requirements so that the hole bearing designs can be fulfilled (see [1] Table 3.3 and 3.4).
- The *Width of slot* is usually the same as the plate thickness. If the connection is made with tolerances, the slot width can be increased by a maximum of 1 mm. When using the SFS intec system, the limit value of 2 mm must be observed. However, this setting makes no difference for the calculation, as only geometry constraints are queried here.
- The plates can also be designed as *Side plates*. For this purpose, at least two slotted plates must be provided.

Modified slotted plate designs are dynamically visualized in the graphic.

If more than one slotted steel sheet is used, there may be a problem in the generation of the predominant failure modes according to [2] Section 8.2.3, Figure 8.3. The predominant (governing) failure mode of the fasteners in the corresponding joint must be compatible with every other one. The combination of failure modes (c), (f), and (j/l) with other failure modes is thus not allowed.

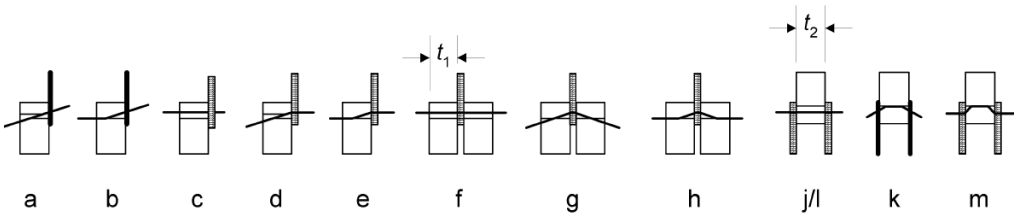


Image 9.23 Failure modes according to [2] Figure 8.3

RF-/JOINTS always checks the hole bearing in the inner and outer shear of a multishear connection. For the cuts at the outer edge of the plates, the failure modes (f), (g), and (h) are checked - both for thick and thin steel plates. The modes according to [2] Equation (8.9) and (8.10) are identical to them.

In the middle part, the failure cases are analyzed according to [2] Equations (8.12) and (8.13). Here too, a distinction is made between thick steel plates with the cases (l), (m) and thin steel plates with the cases (j), (k).

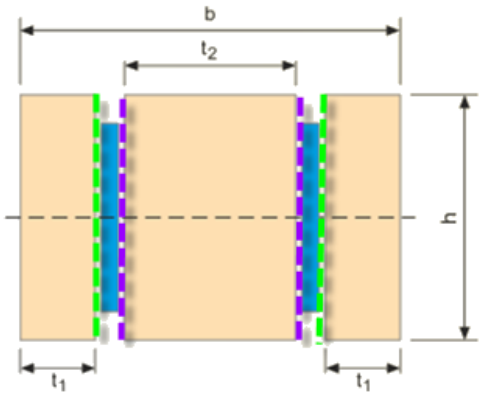
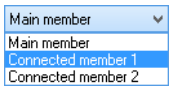


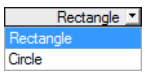
Image 9.24 Failure modes: *dashed green* (f), (g), and (h); *dashed purple* (l), (m) or (j), (k)

RF-/JOINTS always determines the governing failure mode in the respective joint. If, for a thin sheet, the failure mode (j) is governing at the inner (purple) shear planes and the mode (g) in the outer (green) shear planes, the calculation is not possible. However, if the mode (f) were governing, the calculation could be performed.

Connection Geometry



This section describes the fastener layout using parameters. The specifications must be made separately for each member. You can use the list or the buttons to switch between the individual members. Different diameters and distances are possible for the respective categories (dowels, bolts, nails, screws).



- The *Pattern* of the fastener group can be defined as a rectangle or a circle.
- The *Diameter* of the fasteners can be selected within the respective allowable limits. For dowels, the minimum diameter is 6 mm, for screws it is 1.8 mm. If the SFS intec fastening system has been specified in Window 1.1, 7 mm is set. Combinations with different diameters are also possible.
- If you want the length of the dowel to be shorter than the cross-section width (e.g. for fire protection), you have to enter the *Plug length*. This automatically reduces the length of the dowel. For nails and screws, the nail or screw length is shortened on one side.

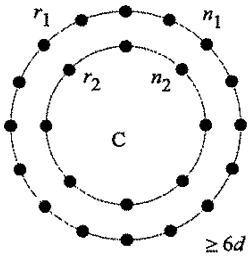
Pattern of the connection

The design of dowel, bolt, screw, and nail connections can be circular or rectangular.

In the case of a circular arrangement, the *Number of circles* is limited by the cross-section height. In the input lines, you can specify the *Number of fasteners in circle*.

Dowel group settings			
Pattern		Circle	
Diameter	d_{st}	20.0	mm
Dowel length	l_{st}	240.0	mm
Plug length	l_{plug}	0.0	mm
Number of circles	n_{cr}	2	
Number of dowels in circle 1	$n_{dw,1}$	6	
Number of dowels in circle 2	$n_{dw,2}$	10	
Method of dowel group placement		Minimum edge distance	
Avoid bending of side members		☐	

Image 9.25 Defining the number of dowel circles



Minimum radius of dowel circle

For circular arrangements, the condition according to [7] that the radius of the circle must be six times larger than the fastener diameter also applies. In the program, this criterion is checked using the height of the fastener that is furthest from the center.

$$d_{core,max} = \frac{\frac{h}{6} \cdot sin60}{1 + sin60}$$

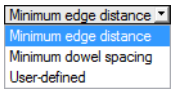
Equation 9.1

For a rectangular arrangement of the dowels, specify the *Number* of fasteners in x-direction and in z-direction.

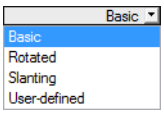
Dowel group settings			
Pattern		Rectangle	
Diameter	d_{st}	12.0	mm
Dowel length	l_{st}	100.0	mm
Plug length	l_{plug}	0.0	mm
Number of dowel columns (x-direction)	n_{dx}	4	
Number of dowel rows (z-direction)	n_{dz}	2	
Staggered rows		☑	
Method of dowel group placement		Minimum edge distance	
Orientation of rows and columns		Basic	
Dowel group reinforced by screws (n_{ef} = n)		☐	

Image 9.26 Defining the number of dowel columns

It is also possible to arrange *Staggered rows* in order to improve the crack behavior of the joint.



The *Method of placement* can aim for the smallest possible distance of the fasteners to each other or the minimum edge distance. User-defined distances are also possible.



The following options are available for the *Orientation* of fastener columns and rows:

- Basic - orientation on local member coordinate system
- Rotated - orientation on global coordinate system
- Slanting - orientation on edges with staggered rows
- User-defined - free definition of inclination and rotation

If the *Dowel group* is *reinforced by screws* to prevent cracking, the effective number of fasteners does not need to be reduced. The parameters of the reinforcement must then be defined separately (see Figure 9.30).

For the joint type *Main member only*, you can cut the member at any angle. This allows for the connection to a reinforced concrete wall, for example.

Dowel group settings			
Pattern		Rectangle	
Diameter	d_{st}	12.0	mm
Dowel length	l_{st}	120.0	mm
Plug length	l_{plug}	0.0	mm
Number of dowel columns (x-direction)	n_{dx}	5	
Number of dowel rows (z-direction)	n_{dz}	3	
Staggered rows		☐	
Method of dowel group placement		Minimum edge distance	
Orientation of rows and columns		Basic	
Dowel group reinforced by screws ($n_{ef} = n$)		☐	
Member settings			
Cutting angle	δ	111.80	°
Avoid bending of side members		☐	

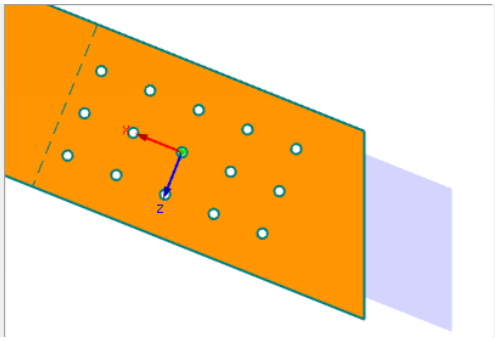


Image 9.27 Defining the cutting angle

For the connected web members of a beam, you can define a *Member eccentricity* that geometrically determines the outlines of the members. The local member coordinate system is shown in the graphic.

Dowel group settings			
Pattern		Rectangle	
Diameter	d_{st}	12.0	mm
Dowel length	l_{st}	120.0	mm
Plug length	l_{plug}	0.0	mm
Number of dowel columns (x-direction)	n_{dx}	5	
Number of dowel rows (z-direction)	n_{dz}	3	
Staggered rows		☐	
Method of dowel group placement		Minimum edge distance	
Orientation of rows and columns		Basic	
Dowel group reinforced by screws ($n_{ef} = n$)		☐	
Member settings			
Member eccentricity in X-direction	X	0.0	mm
Member eccentricity in Z-direction	Z	-100.0	mm
Avoid bending of side members		☑	

Details - List of Dowels

Image 9.28 Defining the member eccentricity

With the *Avoid bending of side members* option (see Figure 9.28), you can prevent additional bending moments due to eccentric load introduction. For this purpose, the program applies a reduced tension resistance for the connecting members. You can find more information about connections subjected to tension in the following article:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001299

Details...

The general parameters for reducing the tensile strength can be found in the *Timber* tab of the *Details* dialog box (see Figure 9.38).

Details

The [Details] button below the section (see image 9.28) opens the *Details* dialog box. In this window, you can deactivate fasteners and adjust the diameters individually.

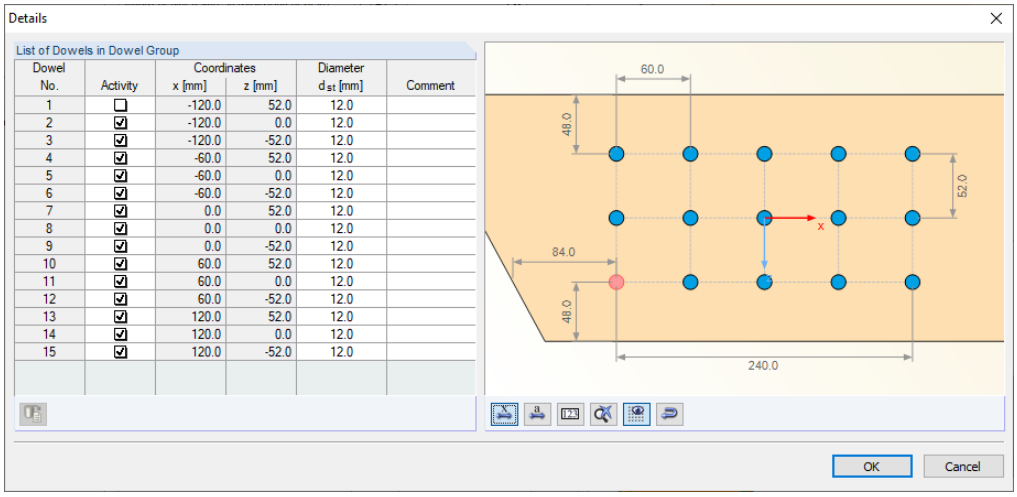


Image 9.29 *Details* dialog box

In the figure above, the *Activity* of dowel No. 1 is suspended.

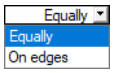
Dowel group reinforced by screws

In order to calculate with $n_{ef} = n$, you can define user-defined reinforcements with screws. The screw reinforcement is identical for all dowel-type fasteners.

Dowel group settings			
Pattern		Rectangle	
Diameter	d_{st}	12.0	mm
Dowel length	l_{st}	120.0	mm
Plug length	l_{plug}	0.0	mm
Number of dowel columns (x-direction)	n_{dx}	5	
Number of dowel rows (z-direction)	n_{dz}	3	
Staggered rows		☐	
Method of dowel group placement		Minimum edge dis	
Orientation of rows and columns		Basic	
Dowel group reinforced by screws ($n_{ef} = n$)		☑	
Specify parameters of the reinforcement		Automatically	
Specify placement of the reinforcement		Automatically	
Number of reinforcing screws in the outer parts	$n_{s,out}$	Manually	
Screw extension	l_{ext}	10.0	mm
Screw length	l_{ef}	162.0	mm
Nominal diameter	d	6.0	mm
Manual definition of core diameter		☐	
Core diameter	d_{core}	3.6	mm
Withdrawal strength determination		acc. to 8.7.2(4)	
Ultimate strength of fastener	$f_{u,b}$	700.00	N/mm²

Image 9.30 Dowel group reinforced by screws

When defining *Automatically*, you have to specify the ultimate tensile strength of the screw.



For the design of the reinforcement, you can arrange the screws between each dowel-type fastener *Equally* or only on the edges of the fastener group.

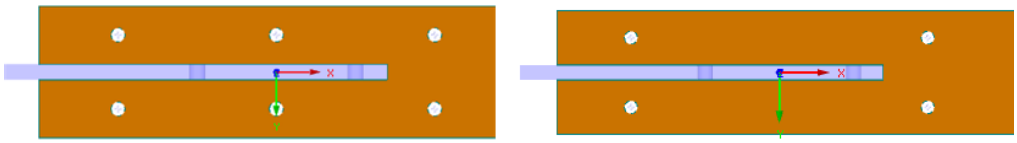


Image 9.31 Screws between each dowel *Equally* (left) or only for edges (right)

The *Number of reinforcing screws* per dowel column is defined in pairs by default. This corresponds to two screws for a steel plate, three screws for two plates, and so on.

The *Screw length* is specified up to the axis of the fastener that is furthest from the screw-in point. A *Screw extension* to the edge of the cross-section with the value l_{ext} is also possible. The screw length will

be calculated automatically.

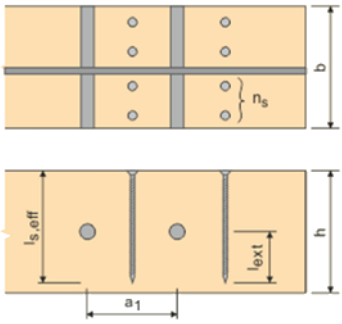
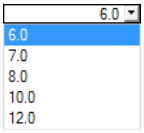


Image 9.32 Definition of screw length



The *Nominal diameter* of the screw can be selected from the list or entered directly.

The design of the screws is performed in the direction of the screw axis according to [2] Section 8.7.2.

When automatically defining the screw reinforcement, you have to specify whether the *Withdrawal strength determination* is carried out according to [2] Section 8.7.2(4) or 8.7.2(5).

The calculation of the pull-out resistance is thus performed either according to

Equation (8.38)

$$F_{ax,\alpha,Rk} = \frac{n_{ef} f_{ax,k} d \ell_{ef} k_d}{1.2 \cos^2 \alpha + \sin^2 \alpha}$$

Equation 9.2

or Equation (8.40a)

$$F_{ax,\alpha,Rk} = \frac{n_{ef} f_{ax,k} d \ell_{ef}}{1.2 \cos^2 \alpha + \sin^2 \alpha} \left(\frac{\rho_k}{\rho_a}\right)^{0.8}$$

Equation 9.3

Since there is no information about the *Ultimate strength of fastener* in [2], the value $f_{u,b}$ must be user-defined. The screw's tensile strength is calculated with the *Core diameter* of the screw.

$$f_{tens,k} = f_{u,k} \left(\frac{d_{core}}{2}\right)^2 \pi$$

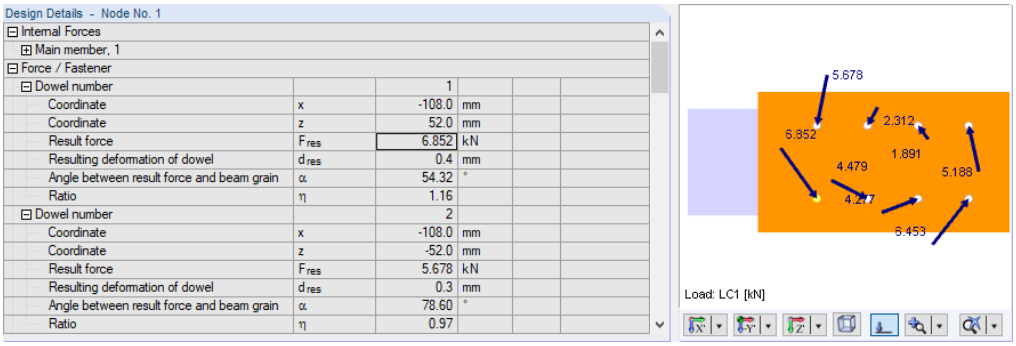
Equation 9.4

The screw action is assumed to be acting separately. Therefore, $n = n_{ef}$ and thus $F_{t,Rk} = n_{ef} f_{tens,k}$. This design of the tensile strength is performed with the number *6201* in the module.

The screw's pull-out resistance from the wood is determined in design *6200* when automatically defining with Equation (8.38) or (8.40) (see above).

In the manual screw design, you can freely specify the withdrawal capacity and the tensile strength of the screw.

When **determining the screw load**, the resulting force F_{res} in each screw is calculated using the force F_{res} in each fastener. The force is displayed in design *6010* (see Figure 9.33). The screw design uses the maximum force in vertical direction $F_{res,z}$ of the member.



Design Details - Node No. 1

Internal Forces					
Main member, 1					
Force / Fastener					
Dowel number		1			
Coordinate	x	-108.0	mm		
Coordinate	z	52.0	mm		
Result force	F_{res}	6.852	kN		
Resulting deformation of dowel	d_{res}	0.4	mm		
Angle between result force and beam grain	α	54.32	°		
Ratio	η	1.16			
Dowel number		2			
Coordinate	x	-108.0	mm		
Coordinate	z	-52.0	mm		
Result force	F_{res}	5.678	kN		
Resulting deformation of dowel	d_{res}	0.3	mm		
Angle between result force and beam grain	α	78.60	°		
Ratio	η	0.97			

Image 9.33 Maximum force of each dowel

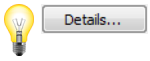
For a slotted steel plate, the force of each dowel is respectively absorbed by one screw on the left and right of the plate. The force is therefore divided by two and multiplied by 0.3 according to [8]. This gives $F_{res,0.3}$.

F_{res} (already divided by two) is again divided by 4 for the force F_{notch} (example of a slotted plate with two screws per dowel), which is why this division only works for the *Equally* arrangement, as shown (see Figure 9.31).

The force used to design the screws is therefore:

$$F_{k,split} = F_{res,0.3} + F_{notch}$$

Equation 9.5



When determining the forces in a screw, it should be noted that only resulting forces acting at an angle of less than 30° are used for the design. If loads that are applied at a steeper angle are to be taken into account as well, the limit angle in the *Timber* tab of the *Details* dialog box can be adjusted (see Figure 9.38).

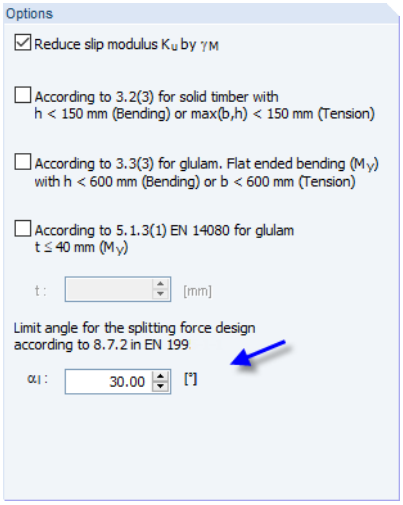


Image 9.34 Adjusting the limit angle

9.6 Details

Details...

In the *Details* dialog box, you can give additional specifications for the design. This dialog box is available in every input window using the [Details] button.

General

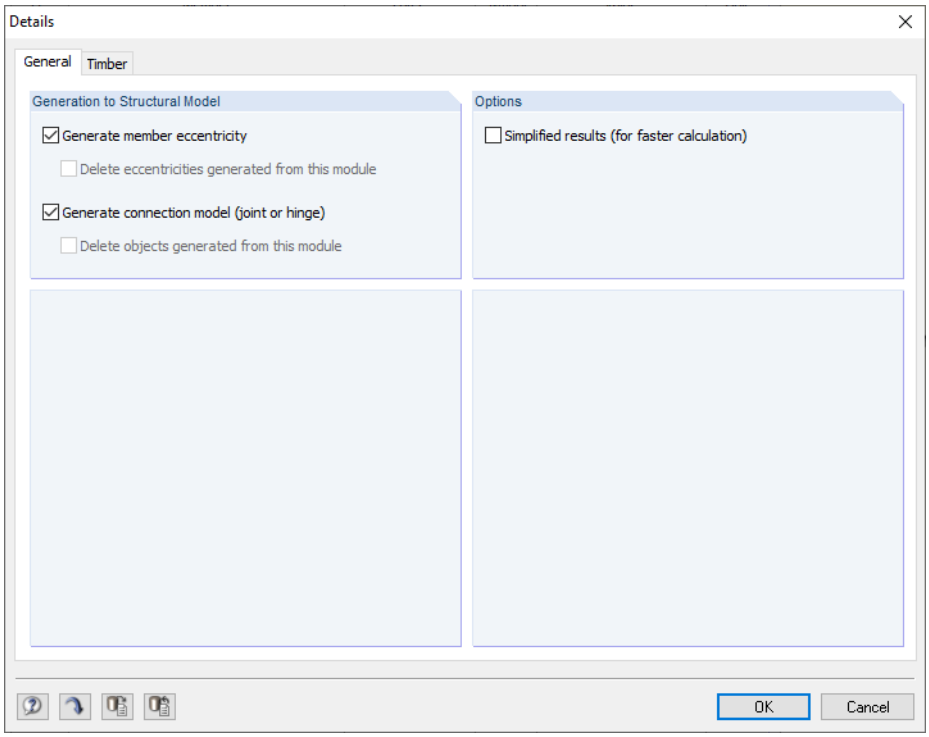


Image 9.35 *Details* dialog box, *General* tab

Generation to Structural Model

The eccentricities and joints that are available due to the geometry parameters of RF-/JOINTS can also be used for the modeling. You can use the *Generate member eccentricity* and *Generate connection model* check boxes to export this specific member information to RFEM or RSTAB. However, no additional structural model is created there. Instead, when you start the RF-/JOINTS calculation, the eccentricity and connection are transferred to RFEM/RSTAB as a member property and nodal releases are generated in RFEM. This information can be found in the RSTAB Tables *1.4 Member Hinges* and *1.5 Member Eccentricities* or the RFEM Tables *1.14 Member Hinges*, *1.15 Member Eccentricities*, *1.24 Nodal Releases*, and *1.30 Joints*. The internal forces for the designs are then determined with the modified model.

There are export options for all members you can define eccentricities and hinges for. If there are already members with hinges or if there are trusses in the model, the additional connection hinges would lead to instabilities in the calculation. Therefore, a corresponding message appears before the dialog box is closed.

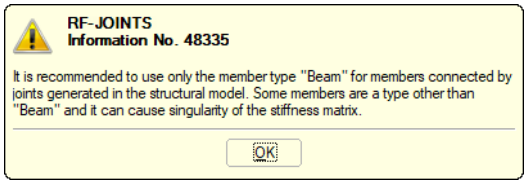


Image 9.36 Note for modeling

The generated eccentricities, for example, can be checked in the *Edit Member Eccentricity* dialog box in RFEM/RSTAB. However, it is not possible to change the values.

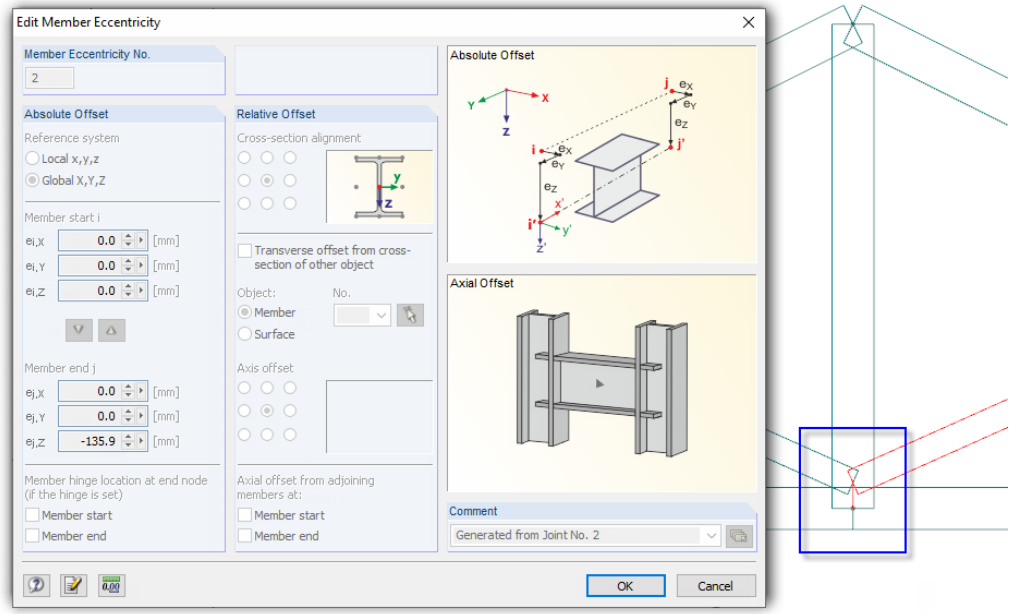


Image 9.37 Generated member eccentricity in RFEM or RSTAB

Options

The *Simplified results* option is recommended if you want to analyze a large number of load combinations. Only a summary of the governing results is then displayed in the result windows. This not only speeds up the calculation, but also the evaluation of the results.

Timber

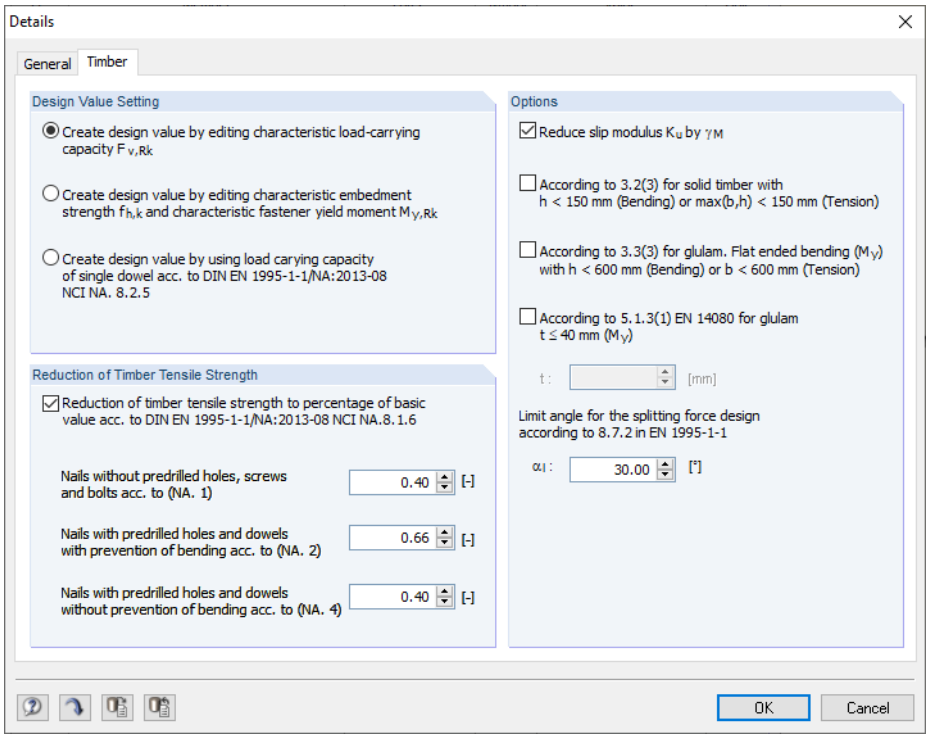


Image 9.38 *Details* dialog box, *Timber* tab

Design Value Setting

If the design value is generated by *editing* the *characteristic load-carrying capacity* $F_{v,Rk}$, the load-bearing capacity is adjusted to the semi-probabilistic safety concept with the factors k_{mod} and γ_M.

The design value of the load-bearing capacity for each dowel and slotted plate is then:

$$F_{v,Rd} = k_{mod} \cdot \frac{F_{v,Rk}}{\gamma_M}$$

Equation 9.6

Alternatively, the design value can be formed by *editing* the *characteristic embedment strength* $f_{h,k}$ *and fastener yield moment* $M_{y,Rk}$. In this case, the hole bearing strength and the yield moment are adjusted with the corresponding partial safety factors.

Hole bearing strength of timber:

$$F_{h\alpha,d} = k_{mod} \cdot \frac{f_{h\alpha,k}}{\gamma_M}$$

Equation 9.7

Yield moment of the dowel:

$$M_{y,Rd} = \frac{M_{y,Rk}}{\gamma_{M0}}$$

Equation 9.8

In the third option, the design value is determined using the *load carrying capacity of single dowel*, taking into account the minimum timber thickness. This method is only specified in the German Annex to [2]. The minimum timber thickness is checked according to equation (NA.116) and then the ultimate limit state design is performed according to equation (NA.115). This procedure corresponds to a rather simplified design. If the failure criteria according to Johansen [9] are checked, the design is not necessary.

Reduction of Timber Tensile Strength

If the *Reduction of timber tensile strength to percentage of basic value* option is activated, the tensile strength of the timber is reduced in the design for bending and compression according to [2] Section 6.2.3. This reduction can be omitted if the warping of the connection is prevented by a guide pin, for example.

According to the German Annex to [2] mentioned below, separate reductions are required for nails and screws. The coefficients can be defined separately here.

For each connected member, you can separately specify whether the reduction should be taken into account (see Figure 9.28).

Options

The *Reduce slip modulus* option reduces the stiffness of the connection determined by the modulus by the material's partial safety factor.

With the *According to 3.2(3) for solid timber*, *According to 3.3(3) for glulam*, and *According to 5.1.3(1) for glulam* check boxes, you can increase the bending and tensile strengths for the designs. The conditions and factors k_h are given in the corresponding sections of the standard [2].

The *Limit angle* text box controls which force is assigned to an optional screw reinforcement (see Equation 9.5). With the default setting, only forces acting at a flatter angle than 30° in the respective fastener are considered.

10 Timber - Timber to Timber

This chapter describes the windows that are especially relevant for the **RF-/JOINTS Timber - Timber to Timber** module. The general input parameters are described in Chapter 2.

The input windows of the add-on module are accessible once you select the material *Timber* and the joint group *Timber to timber connection*.

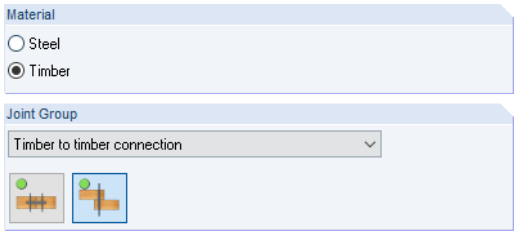
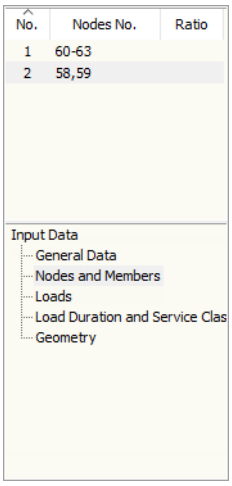


Image 10.1 *RF-/JOINTS Timber - Timber to Timber add-on module*

The module generally requires the connected members to have a pinned joint.



The input always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

If the entries *Load Duration and Service Class* and *Geometry* are missing in the navigator, go to Window *1.2 Nodes and Members* and check if the boundary conditions of the node are correct and if load cases are available for the design. For example, it might be necessary to adjust the status of the connected members (see Figure 10.8).

10.1 General Data

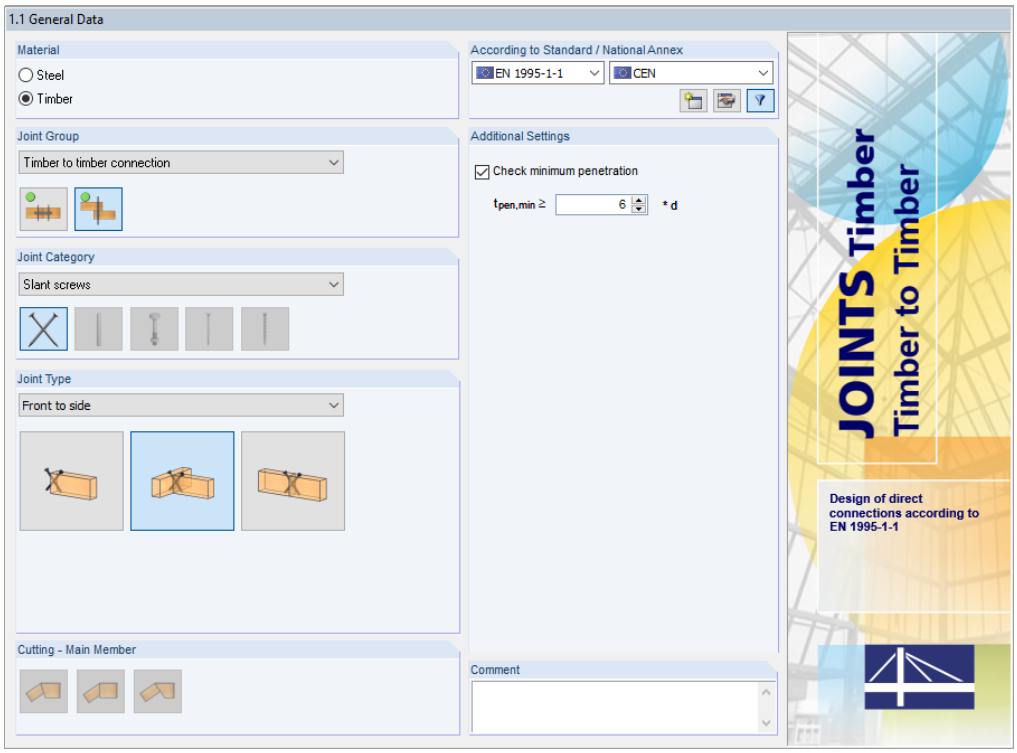


Image 10.2 Window *1.1 General Data*

Joint Category

Image 10.3 Joint category

The *Slant screws* fastener is currently the only available timber-to-timber joint category.

Joint Type

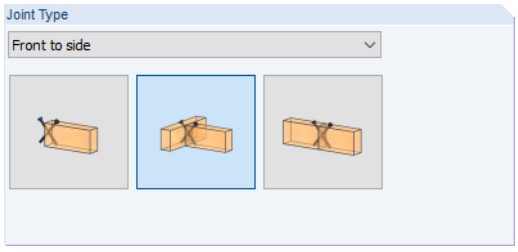
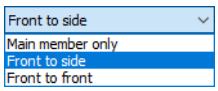


Image 10.4 Joint type



The following joint types are available:

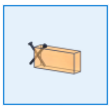
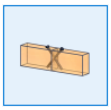
	Main member only	Fastening of a member to an existing component (or any number of members) at any angle
	Front to side	Connection of a diagonal member to a continuous beam (*Main member* and *Continuous member*)
	Front to front	Free definition of a node with a connected member

Table 10.1 Joint types

Cutting - Main Member

Image 10.5 Cutting - Main Member

This section is only accessible for the *Front to front* joint type. With an inclined secondary member, the connection can be symmetrical or asymmetrical (cf. Figure 9.6).

	Symmetrical connection at an angle of 45°
	Main member cut perpendicularly to member axis
	Secondary member cut perpendicularly to member axis

Table 10.2 Connection options for main member

According to Standard / National Annex

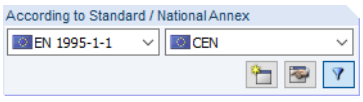


Image 10.6 *According to Standard / National Annex section*

The design-relevant coefficients are specified according to the standard and the National Annex (see Figure 2.19).

If you want to apply user-defined factors for the designs, you have to create a new National Annex first by clicking the button. Afterwards, the coefficients can be customized in the *National Annex Settings* dialog box (cf. Figure 9.10).

Additional Settings

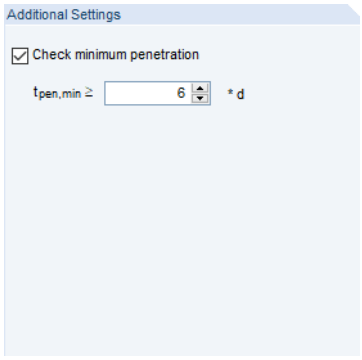


Image 10.7 *Additional Settings section*

According to [2] 8.7.2(3), the minimum embedment depth of the threaded part on the side of the screw head should be six times the screw diameter *d*. If necessary, the factor for determining the *minimum penetration* $t_{pen,min}$ can be adjusted in this section.

10.2 Nodes and Members

The selection of nodes and members is described in Chapter 2.2 and Chapter 9.2.

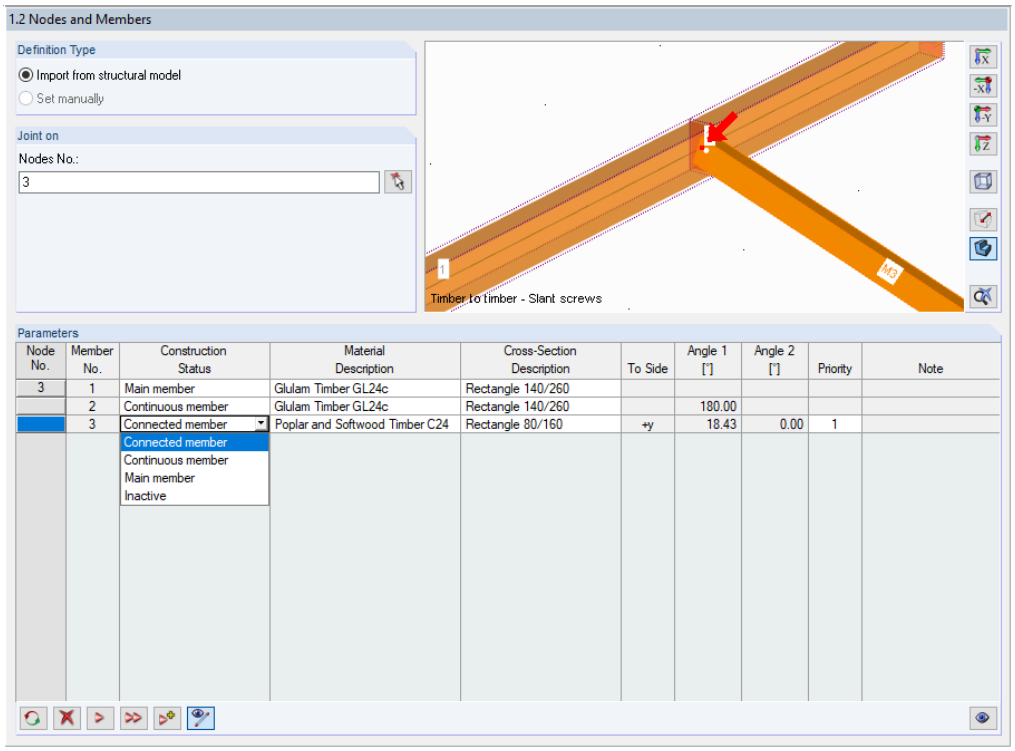
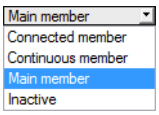


Image 10.8 *Window 1.2 Nodes and Members - adjusting construction status of members*

In the *Parameters* section, you can check the boundary conditions of the connected structural components. If necessary, you can adjust the *Construction Status* in this section. The current member is highlighted in color in the graphic window.

The *Angles* of the members are based on the geometric conditions of the model defined in RFEM or RSTAB.



If more than two members are connected to a node, the following definition options are available for the *Construction Status* of the members:

- Main member - controls all other members as well as the cutting and its priority
- Continuous member - only available for the *With continuous member* joint type
- Connected member
- Inactive - excludes the member from the design

When defining main and connecting members, it is necessary to observe certain geometric conditions:

- A maximum of one member can be connected.
- For the minimum length of the connected member, the condition applies that the screwed-in screw must not protrude from the member.
- There must be a minimum angle of 30° between the members in the xy plane. In the xz plane, any angle is possible. So-called jack rafter cuts between members are therefore possible as well (see Figure 10.9).

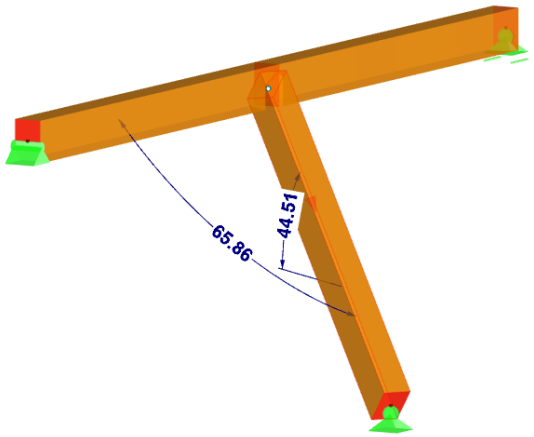


Image 10.9 Jack rafter joint

- For the *Main member* joint type, the cuts are limited to angles between 75° and 105°.
- For the *Front to front* joint type, the angle of the connected member must not exceed 60°.

With the status *Main member* for all members, you can connect more than one member to the node, like in the Timber - Steel to Timber module (see Figure 9.19). This is advantageous if you only want to design the connection in a complex joint geometry.

For the joint type *Main member,* you can freely select the inclination, cutting, and eccentricity within the described limits.

10.3 Loads

Entering loads and internal forces is described in Chapter 2.3.

10.4 Load Duration and Service Class

Window *1.4 Load Duration and Service Class* is described in Chapter 9.4.

10.5 Geometry

In Window *1.5 Geometry*, you can specify the details for the connection of the selected members.

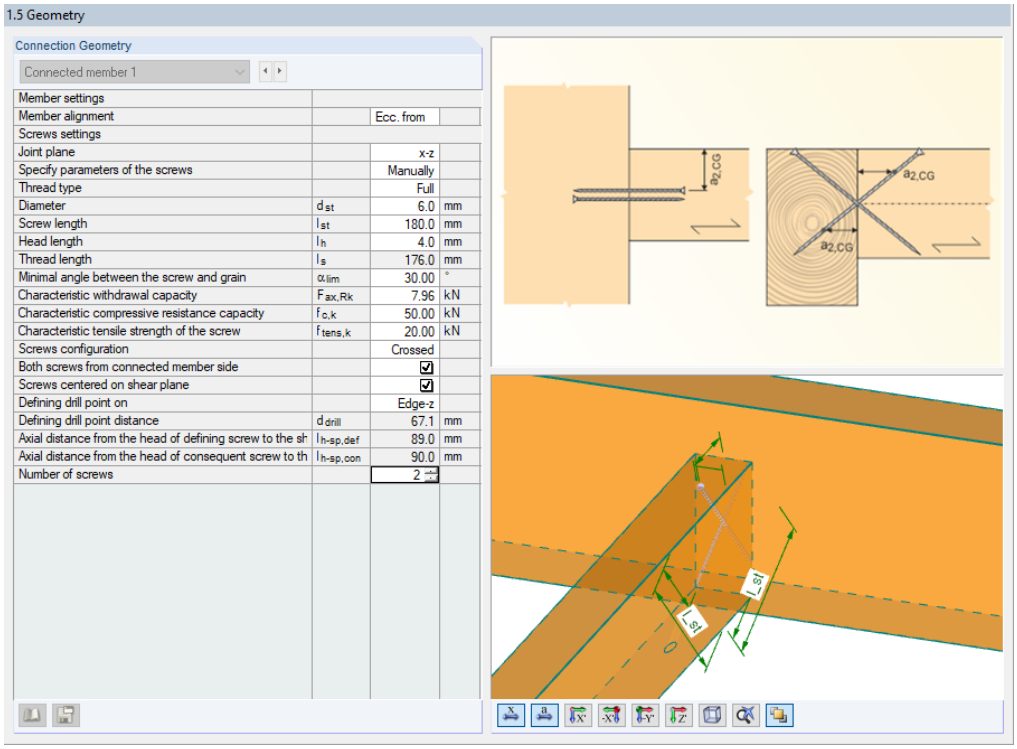
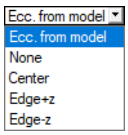


Image 10.10 Window *1.5 Geometry*

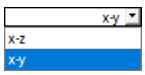
This window is divided into two parts: On the left, the input parameters of the connection node are displayed; on the right, they are illustrated by graphics. The upper graphic shows a system sketch of the current parameter, the lower graphic shows a 3D visualization of the node.

The graphic buttons are explained in Table 3.1.

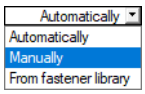
The window is adapted to the joint type. In the following, the entries for the *Front to side* type are presented. This connection is the most common type of design for slant screws.



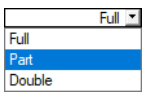
If there is an eccentricity for the *Member alignment* in the model, it can be imported from RFEM or RSTAB. You can use the *Edge* options to specify a user-defined eccentricity. However, this eccentricity is not transferred to the main program. Thus, the eccentricity does not influence the internal forces, just the geometric boundary conditions of the connection.



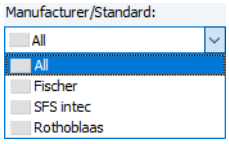
For the connected member, the plane x-y or x-z can be selected as the *Joint plane*. Depending on the specification, the pair of screws is rotated accordingly. The graphic window shows the arrangement of the screws dynamically.



The *Define parameters of screws* option allows you to define the screws automatically, manually, or in a library. The library of fasteners (see Figure 10.11) can be opened in the text box below by using the ... button.



The *Thread type* can be selected from the list.



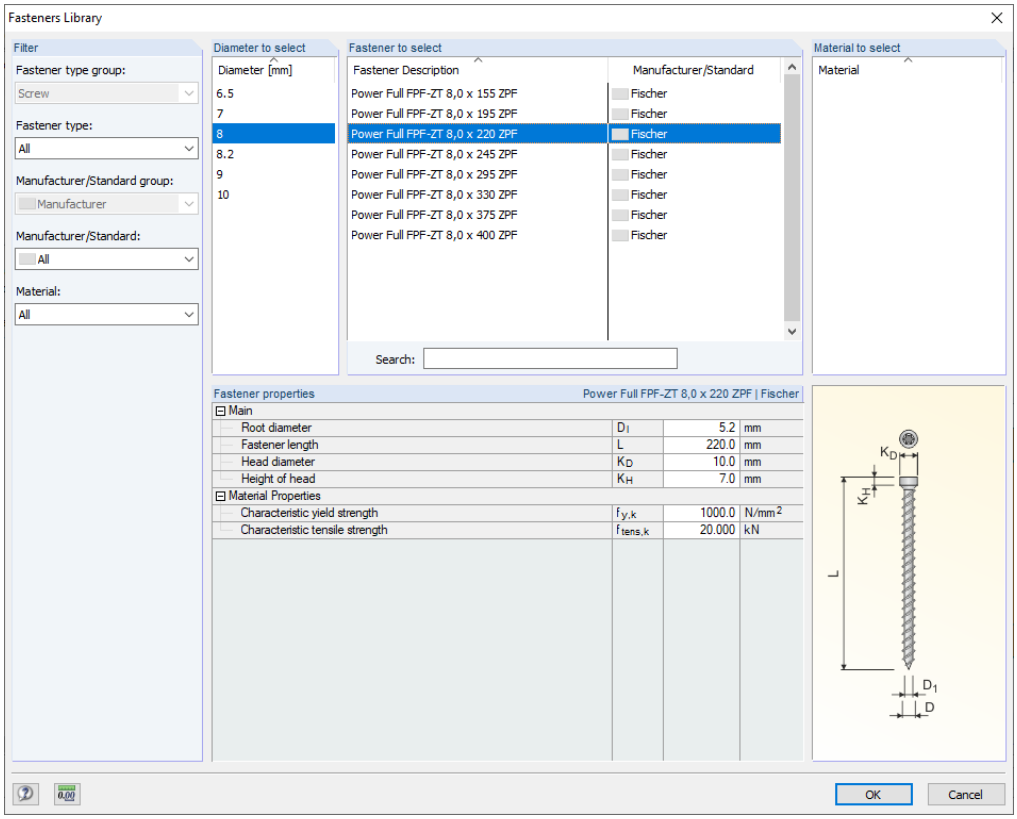
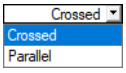


Image 10.11 Fastener library

When you define the screws manually, note the following:

- The *Diameter* d_{st} of the screw must have at least 6 mm and a maximum of 12 mm.
- The *Screw length* l_{st} is freely definable. However, the screw must not protrude from the wood.
- The *Head length* l_h is limited to a maximum of 50 mm.
- The *Minimal angle between the screw and grain* α_{lim} must be between 0° and 90°. According to [2] 8.7.2, this angle may not be smaller than 30° (default setting). However, some manufacturers also offer the option to apply flatter angles according to the technical approval.
- The *Characteristic withdrawal capacity* $F_{ax,Rk}$ can be calculated according to [2] Equations (8.38), (8.40a), or (8.40b). Alternatively, this value can be taken from the technical approval of a screw manufacturer.
- The *Characteristic compressive resistance capacity* $f_{c,k}$ in the buckling design of the bolt is assumed to be 50 kN.
- According to the technical approval, the *Characteristic tensile strength of the screw* $f_{tens,k}$ is assumed to be 20 kN.



- The *Screws configuration* can be selected from the list.
- With the *Both screws from connected member side* option, you can control the screw-in direction of the screws (see Figure 10.12).
- The *Screws centered on shear plane* option allows you to create an offset of height from the connected member to the main beam.

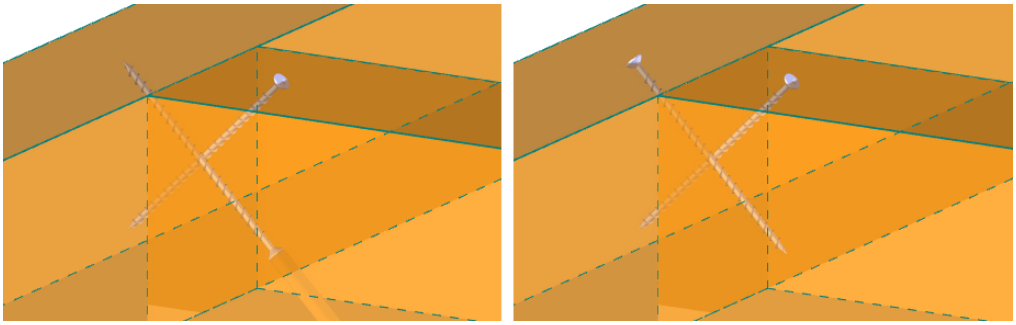
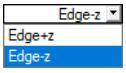
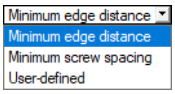


Image 10.12 Screw-in direction: *Both screws from connected member side* (left)



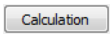
- *Defining drill point* determines whether the screw is connected from the bottom or top edge.
- If the screw is not placed in the center of the shear plane, you can enter the *Drill point distance*, the *distance from the head of defining screw to the shear plane*, and the *distance from head of following screw to the shear plane* in the text boxes.
- After that, define the *Number of screws*. There can be at most 20 pairs of screws if the alignment is crosswise or 20 screws if the alignment is parallel.



- If more than one screw pair or, in case of a parallel alignment, more than one screw is defined, information about the screw spacings is required. The *Method of screws placement* can take the minimum or maximum edge distances or user-defined specifications into account.

On our website, you can find a technical article that explains how to determine the screw forces for a secondary beam connected to a torsionally rigid main beam:
https://www.dlubal.com/en-US/support-and-learning/support/knowledge-base/001502

11 Calculation



You can start the [Calculation] in every RF-/JOINTS window by using the corresponding button.

RF-/JOINTS searches for the results of the load cases, load combinations, and result combinations to be designed. If they cannot be found, the RFEM or RSTAB calculation for determining the design-relevant internal forces starts first. If you have selected direct input of internal forces (see Chapter 2.3.2), the RFEM/RSTAB results are not relevant for the design.

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

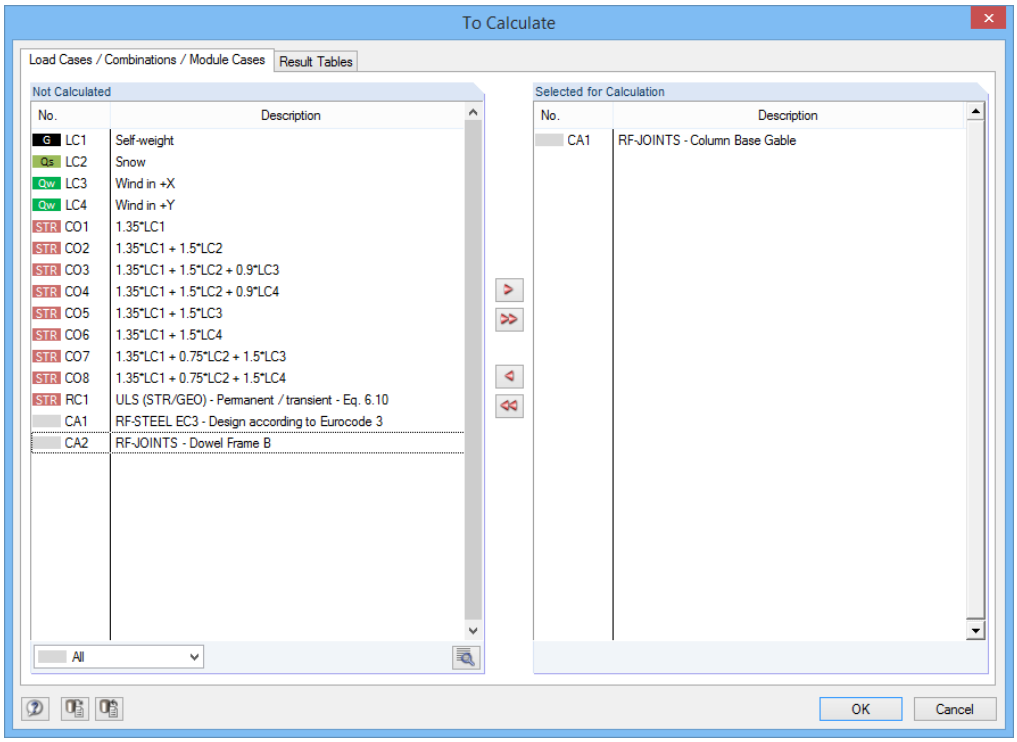
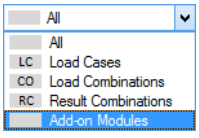


Image 11.1 *To Calculate* dialog box



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

You can use the button to transfer the selected RF-/JOINTS cases to the list on the right. Click [OK] to start the calculation.

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

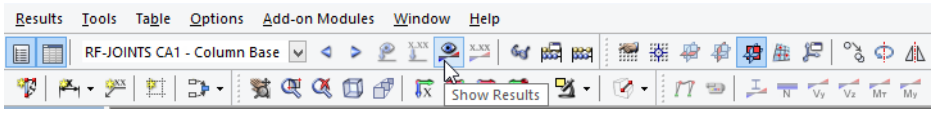


Image 11.2 Direct calculation of an RF-JOINTS case in RFEM

You can subsequently observe the calculation process in a dialog box.

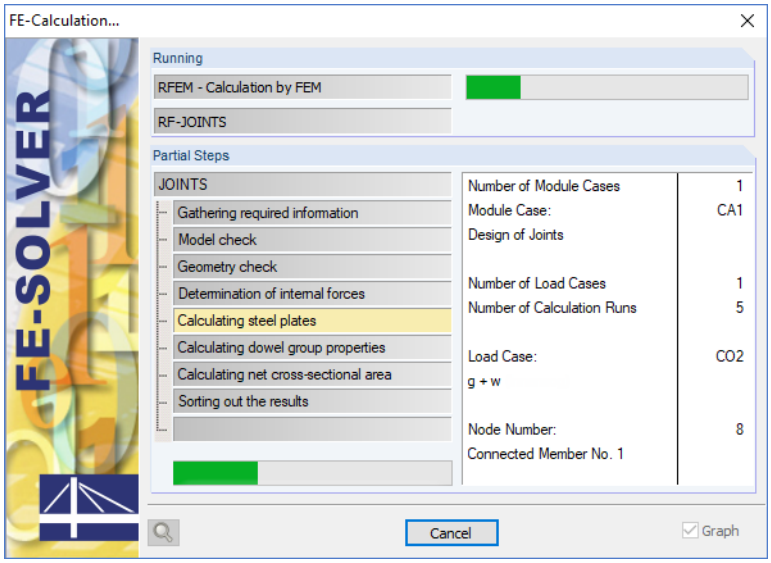


Image 11.3 *FE-Calculation* dialog box

12 Results

Window 3.1 *Design - Summary* appears immediately after the calculation.

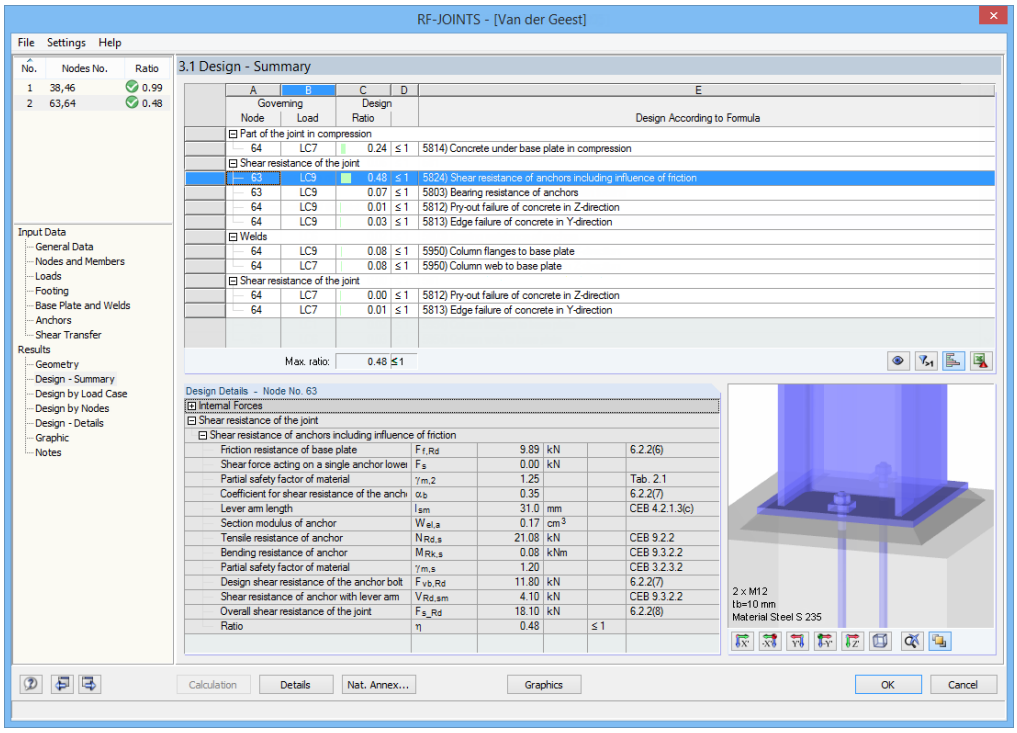
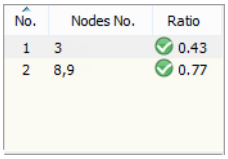


Image 12.1 Result window with designs, design ratios, and design details



The output always refers to the design case selected in the upper part of the navigator. To change the design case, simply click the relevant entry in the list.

The content of the result windows is adapted to the results of the different RF-/JOINTS modules. The sequence and the concept of the windows are the same.

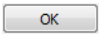
Window 2.1 shows an overview graphic including the geometric parameters of the connection components.

Result windows 3.1 to 3.3 show the designs, sorted by different criteria. Window 3.4 lists the intermediate values of the individual designs.

Window 4.1 shows a graphic of the joint with all geometric details, which can also be printed.

Window 5.1 lists notes that are relevant for the design.

You can directly select a window by clicking its entry in the navigator. Use the buttons shown on the left to set the previous or next window. You can also use the function keys [F2] and [F3] to go through the windows.



Click [OK] to save the results. The RF-/JOINTS module closes and you return to the main program.

Chapter 12 presents the result windows one by one. Evaluating and checking the results is described in Chapter 13.

12.1 Geometry

This window lists all the geometric parameters of the connection.

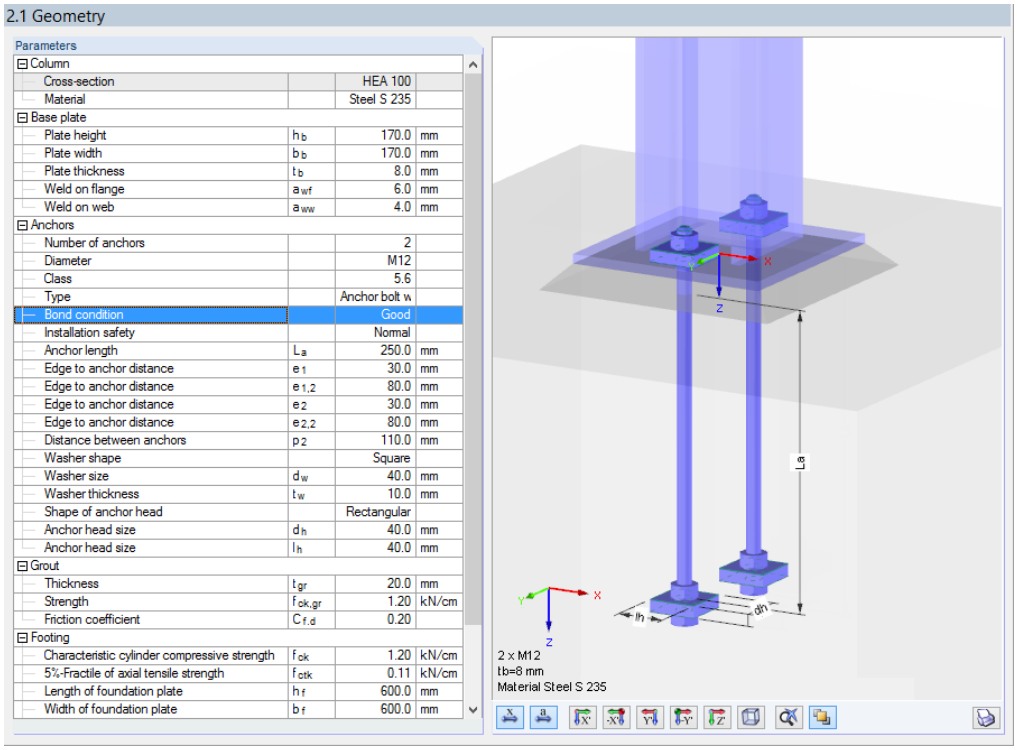


Image 12.2 Window 2.1 Geometry

As is usual in Windows applications, the list entries can be expanded with ⊞ and reduced with ⊟.

The graphic shows the component selected in the list including the parameters.

The buttons are described in Table 13.1.

12.2 Design - Summary

This window lists the maximum design ratios of all designed nodes. The design ratios for the internal forces of the governing load cases and combinations are sorted by design type.

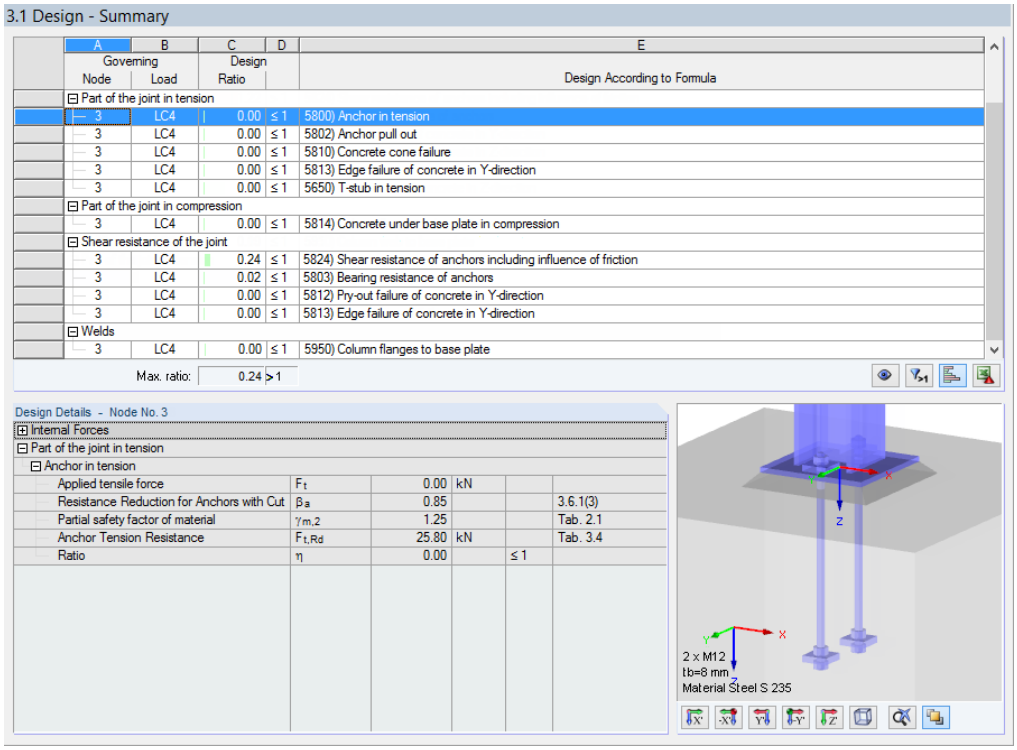


Image 12.3 Window 3.1 Design - Summary

Governing Node

This column displays the number of the node with the highest design ratio for the design type indicated in column E.

Governing Load

In this column, you can find the numbers of the load cases or load combinations whose internal forces result in the maximum design ratios.

Design Ratio

Max. ratio: 0.84 ≤ 1

Columns C and D show the design conditions according to EN 1993-1-8 [1], EN 1995-1-1 [2], or ANSI/AWC NDS 2018 [5].

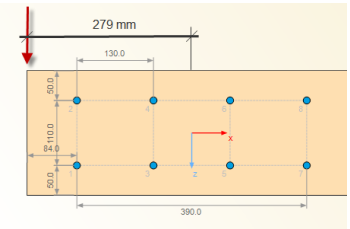
The length of the colored bars represents the respective design ratio graphically.

Design According to Formula

This column names the individual designs according to [1], [2], or [5]. The design numbers according to the Eurocode are explained in Table 12.1 for the RF-/JOINTS Timber - Steel to Timber module.

Number	Design	Section in standard
5005	Steel plate for bending, shear and axial force	[14] 6.2.10
5009	Hole bearing	[1] Table 3.4
5010	Steel plate for bending and shear without axial force	[14] 6.2.8
6010	Load-carrying capacity per dowel	[2] 8.2.3 and 8.6
6030	Block shear failure	[2] Annex A
6031	Effective number of fasteners (n_{ef})	[2] 8.1.2; 8.5.1.1(4)
6032	Dowel group	[2] 8.1.4
6033	Dowel group due to moment	[2] 8.1.4
6140	Tensile stress	[2] 6.1.2
6142	Shear stress	[2] 6.1.7
6143	Bending stress	[2] 6.1.6
6144	Tensile and bending stress	[2] 6.2.3
6146	Shear stress due to eccentricity (see figure on the left)	[2] 6.1.7
6200	Reinforcement with screws	[2] 8.7.2(4/5)
6201	Reinforcement with screws	[2] 8.7.2(7)
6500	Minimum spacings	[2] 8.6, Table 8.5
6530	Check of contact	

Table 12.1 Design numbers and designs



Eccentricity of shear force to center of dowel

Design Details

In this window section, you can find detailed information about the design parameters of the entry selected above.

12.3 Design by Load Case

The upper part of the window lists the governing designs sorted by load cases and load combinations. The lower part includes detailed information on the internal forces and design parameters of the load case selected above.

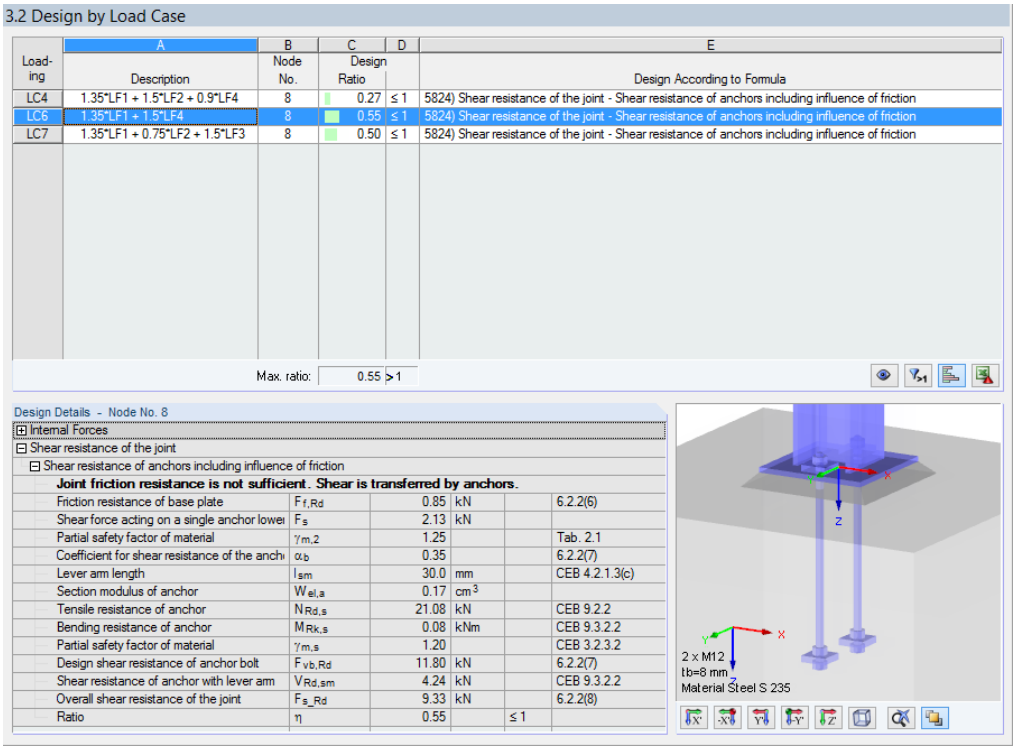


Image 12.4 Window 3.2 Design by Load Case

Description

This column provides the descriptions of the load cases and load combinations for which the designs were performed.

Nodes No.

This column displays the number of the node with the maximum design ratio for the designed action.

Design Ratio

Max. ratio: 0.84 ≤ 1

Columns C and D show the design conditions according to [1], [2], or [5]. The length of the colored bar represents the respective design ratio graphically.

Design According to Formula

This column lists the equations of the standard that were used for the designs.

12.4 Design by Nodes

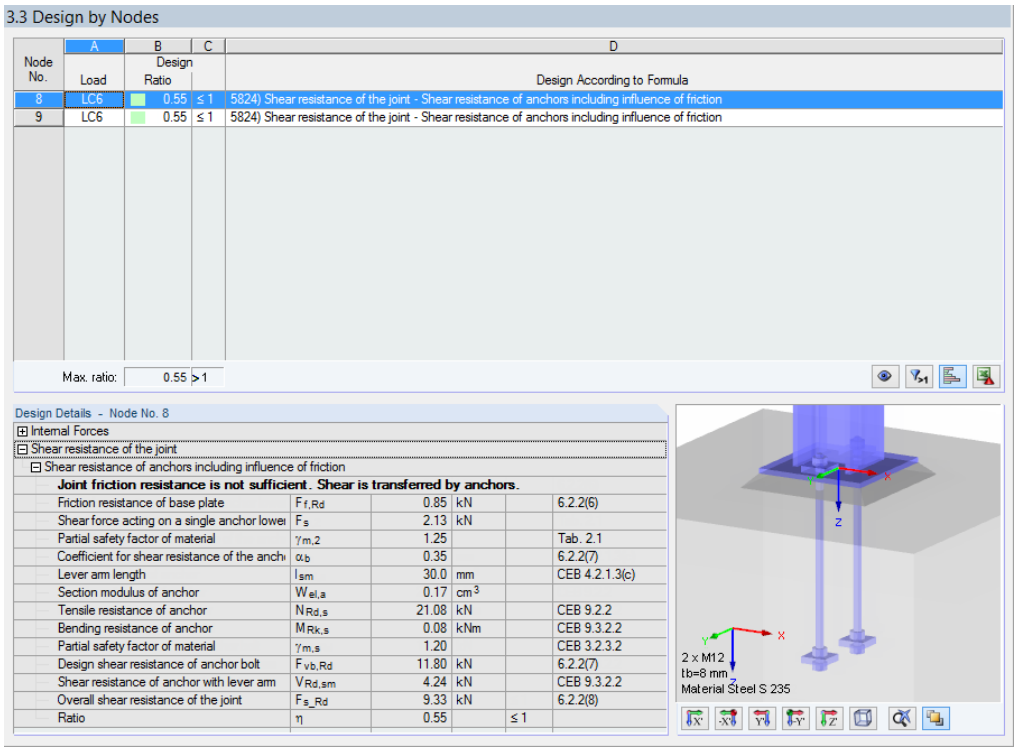


Image 12.5 Window 3.3 Design by Nodes

This window lists the maximum design ratios of the designed nodes.

12.5 Design - Details

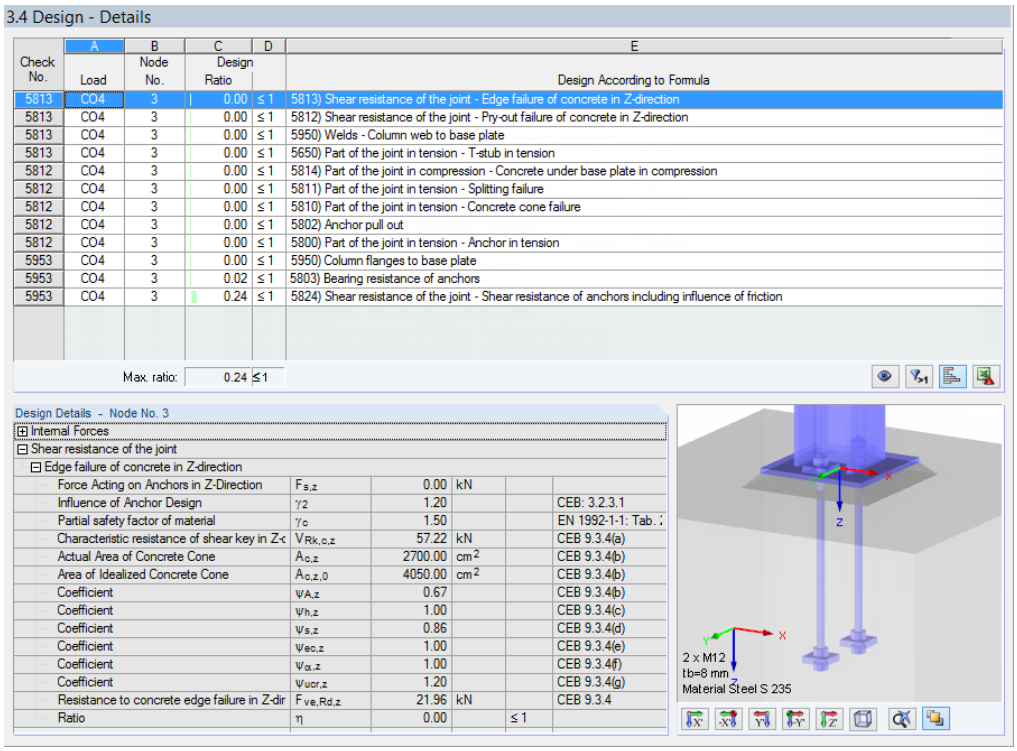


Image 12.6 Window 3.4 Design - Details

This result window lists all individual designs that were carried out for the connection, including design ratios.

The lower section of the window provides detailed information about the parameters of the design selected above.

12.6 Graphic

This window shows a graphic of the connection including all of the components.

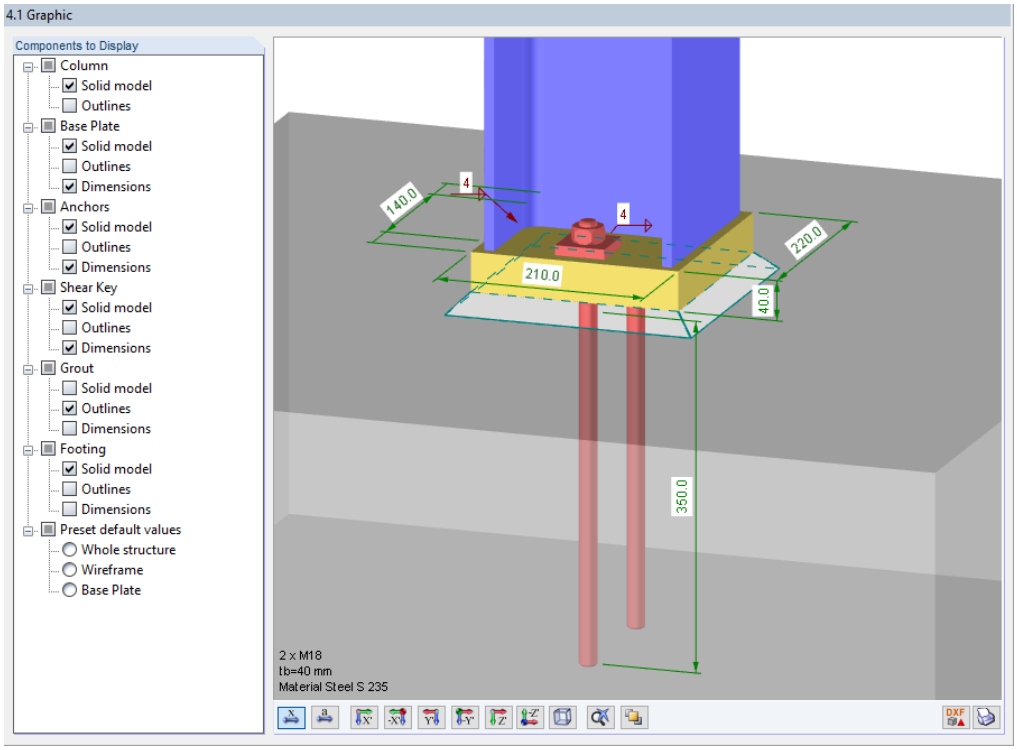


Image 12.7 Window *4.1 Graphic*

The *Components to Display* section lists the structural components of the connection. You can display or hide individual components in the graphic using the corresponding check boxes.

Use the ⊞ button to access additional subentries. This allows you to display the *Dimensions* and *Outlines* of certain components.

The graphical display is dynamic. You can use the mouse functions known from RFEM or RSTAB to zoom, move, or rotate the view.

The buttons are described in Table 13.1.

12.7 Notes

5.1 Notes

Note No.	Description
100	The bending moment M_y is not considered in the calculation.
101	The bending moment M_z is not considered in the calculation.

Image 12.8 Window 5.1 Notes

The last result window contains numbered notes with explanations that are relevant for the design of the connection.

13 Result Evaluation

Windows 3.1 to 3.4 display the results sorted by various criteria. The buttons below the tables facilitate the evaluation.

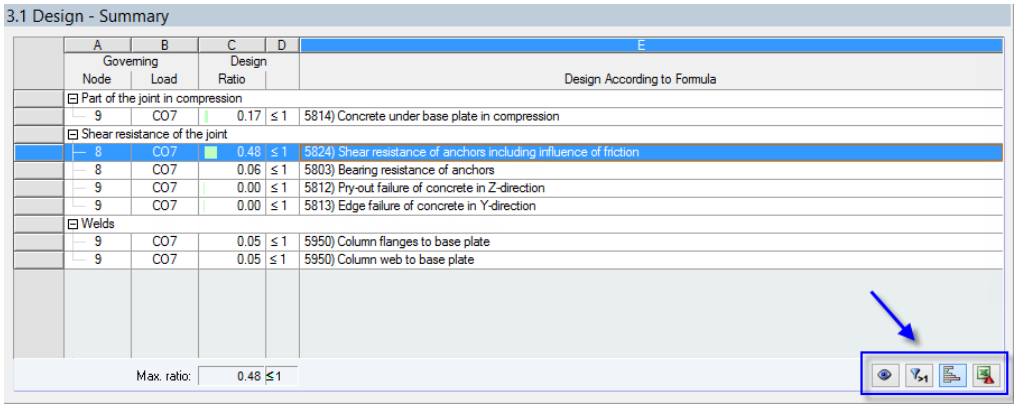


Image 13.1 Buttons for result evaluation in the tables

The buttons have the following functions:

Button	Description	Function
	View mode	Jumps to the RFEM or RSTAB work window to change the view
	Exceeding	Only displays rows with a ratio greater than 1 (design not fulfilled)
	Relation scales	Displays or hides the colored relation scales in the result windows
	Export to Excel	Exports the table to MS Excel or as a CSV file → Chapter 15.3

Table 13.1 Buttons in result windows 3.1 to 3.4

13.1 Graphic of Connection in RF-/JOINTS

Each result window shows a dynamic graphic of the joint. It provides an overview and illustrates the parameters.

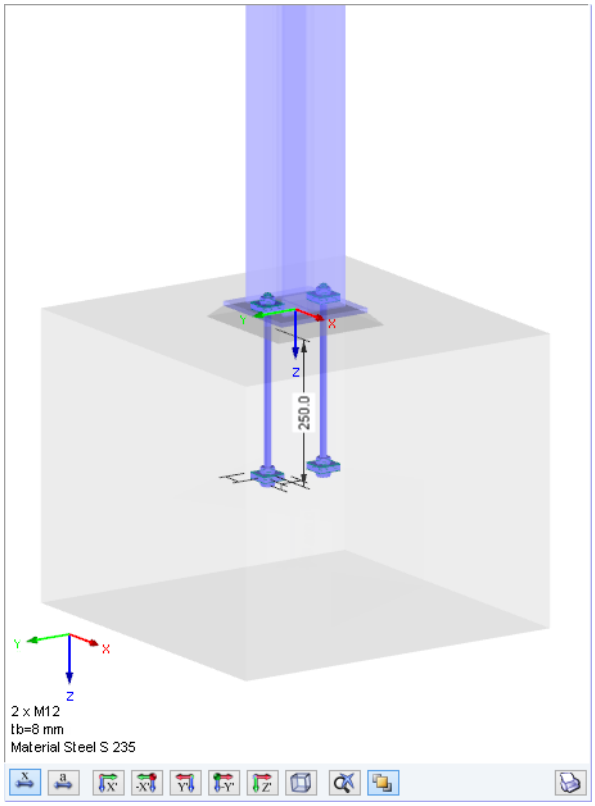


Image 13.2 Graphic of connection

The buttons below the graphic have the following functions:

Button	Function
	Shows or hides the dimensioning
	Displays the values or symbols of the dimensioning
	Shows the view in the direction of the X-axis
	Shows the view in the opposite direction of the X-axis
	Shows the view in the direction of the Y-axis
	Shows the view in the direction of the Z-axis
	Displays the isometric view
	Resets the full view of the graphic

	Shows or hides irrelevant components
	Prints the current 3D graphic

Table 13.2 Graphic buttons in result windows

You can use the mouse to zoom, move, or rotate the view. These functions are described in Chapter 3.4.9 of the RFEM or RSTAB manual.

You can use the [Show Unselected Parts Transparently] button to display only selected components such as anchors with dimensions, for example.

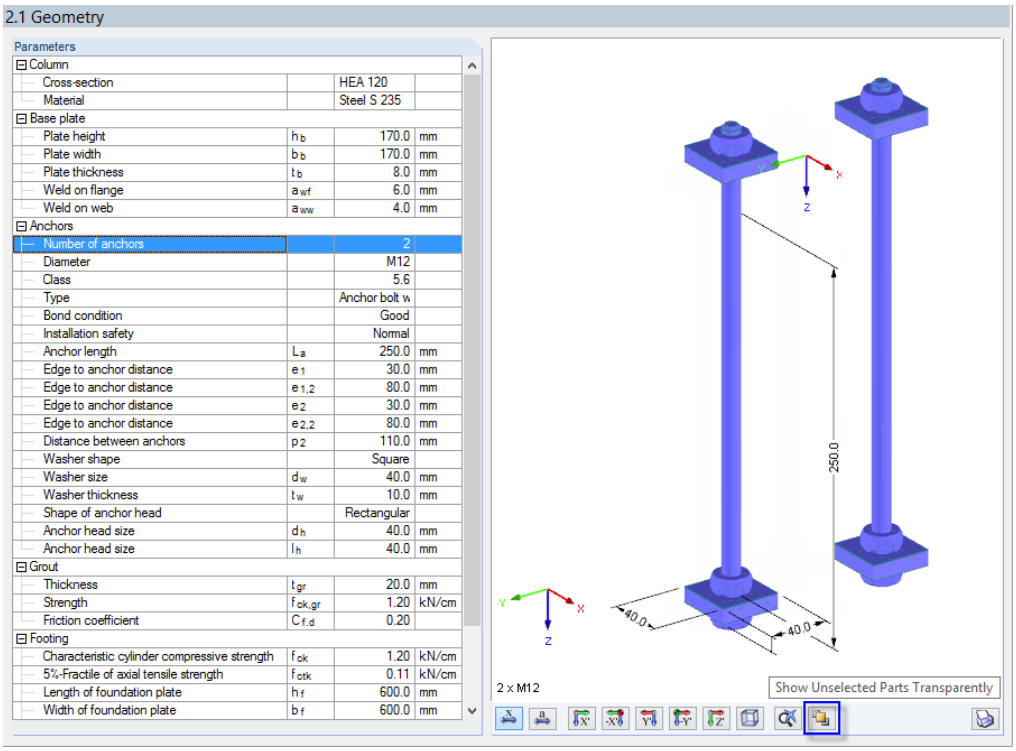
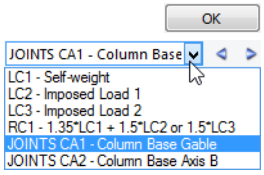


Image 13.3 *Show Unselected Parts Transparently* button

Click to print the graphic. This option is described in Chapter 14.2.1.

13.2

Graphic of Connection in RFEM/RSTAB Model



The graphic of the connection can also be displayed graphically on the RFEM or RSTAB model: Click [OK] to close the RF-/JOINTS module. Then, select the RF-/JOINTS design case in the RFEM/RSTAB menu bar.

The connections are now visualized in 3D rendering in the work window of RFEM or RSTAB.

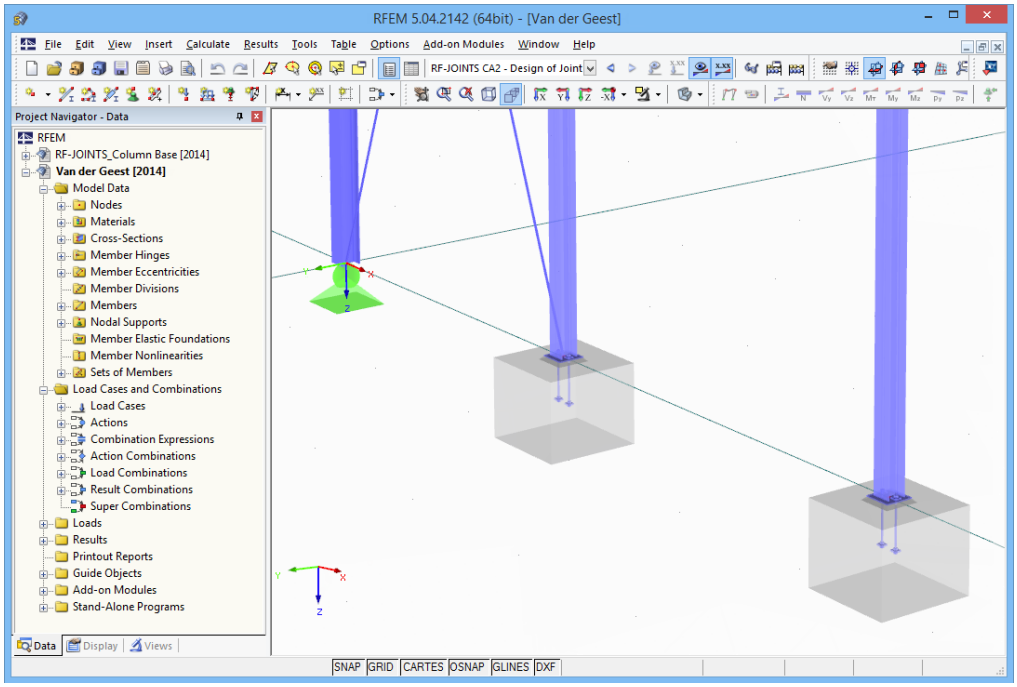


Image 13.4 Display of connections in RFEM work window

Similar to the display of internal forces, the [Show Results] button displays or hides the module results.

RFEM/RSTAB do not provide a separate *Results* navigator for the design cases of RF-/JOINTS. The RFEM/RSTAB tables are not relevant for RF-/JOINTS as well.

You can also use the *Visibility* options for the display in RFEM or RSTAB (see Chapter 9.9.1.2 of the RFEM manual or Chapter 9.7.1.2 of the RSTAB manual) in order to filter the nodes of the connections.

The graphic of the connection can be printed directly or transferred to the printout report (see Chapter 14.2.2).

14 Printout

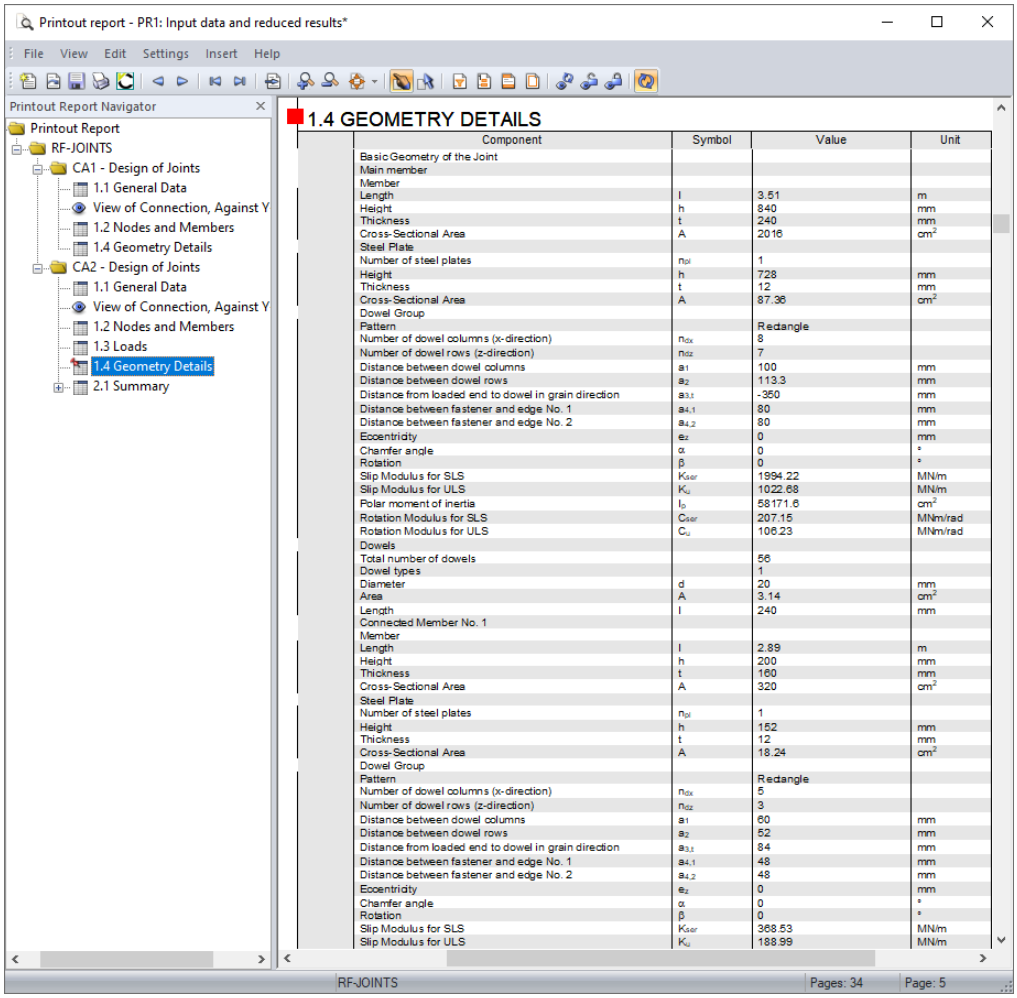
14.1 Printout Report

Like in RFEM or RSTAB, you can generate a printout report for the data of the RF-/JOINTS module where you can add graphics and descriptions. The selection in the printout report determines which data from the connection module is eventually included in the printout.

The printout report is described in the RFEM and RSTAB manuals. Chapter 10.1.3.5 'Selecting data of add-on modules' explains how the input and output data of add-on modules can be prepared for the printout.

For complex structural systems with design cases from different connection modules, dividing the data up into several printout reports enhances the clarity of the printout.

The parameters of input windows 1.4 to 1.8. are summarized in Table *1.4 Geometry Details* of the printout report.



Component	Symbol	Value	Unit
Basic Geometry of the Joint			
Main member			
Member			
Length	l	3.51	m
Height	h	840	mm
Thickness	t	240	mm
Cross-Sectional Area	A	2016	cm²
Steel Plate			
Number of steel plates	n_{pl}	1	
Height	h	728	mm
Thickness	t	12	mm
Cross-Sectional Area	A	87.36	cm²
Dowel Group			
Pattern		Rectangle	
Number of dowel columns (x-direction)	n_{dx}	8	
Number of dowel rows (z-direction)	n_{dz}	7	
Distance between dowel columns	a_1	100	mm
Distance between dowel rows	a_2	113.3	mm
Distance from loaded end to dowel in grain direction	$a_{3,t}$	-350	mm
Distance between fastener and edge No. 1	$a_{4,1}$	80	mm
Distance between fastener and edge No. 2	$a_{4,2}$	80	mm
Eccentricity	e_z	0	mm
Chamfer angle	α	0	°
Rotation	β	0	°
Slip Modulus for SLS	K_{ser}	1994.22	MN/m
Slip Modulus for ULS	K_u	1022.68	MN/m
Polar moment of inertia	I_p	58171.6	cm²
Rotation Modulus for SLS	C_{ser}	207.15	MNm/rad
Rotation Modulus for ULS	C_u	106.23	MNm/rad
Dowels			
Total number of dowels		56	
Dowel types		1	
Diameter	d	20	mm
Area	A	3.14	cm²
Length	l	240	mm
Connected Member No. 1			
Member			
Length	l	2.89	m
Height	h	200	mm
Thickness	t	160	mm
Cross-Sectional Area	A	320	cm²
Steel Plate			
Number of steel plates	n_{pl}	1	
Height	h	152	mm
Thickness	t	12	mm
Cross-Sectional Area	A	18.24	cm²
Dowel Group			
Pattern		Rectangle	
Number of dowel columns (x-direction)	n_{dx}	5	
Number of dowel rows (z-direction)	n_{dz}	3	
Distance between dowel columns	a_1	60	mm
Distance between dowel rows	a_2	52	mm
Distance from loaded end to dowel in grain direction	$a_{3,t}$	84	mm
Distance between fastener and edge No. 1	$a_{4,1}$	48	mm
Distance between fastener and edge No. 2	$a_{4,2}$	48	mm
Eccentricity	e_z	0	mm
Chamfer angle	α	0	°
Rotation	β	0	°
Slip Modulus for SLS	K_{ser}	368.53	MN/m
Slip Modulus for ULS	K_u	188.99	MN/m

Image 14.1 Printout report table *1.4 Geometry Details*

14.2 Graphic Printout

You can print graphics of the RF-/JOINTS module as well as of the RFEM work window.

14.2.1 RF-/JOINTS Graphic

The print function can be accessed by using the [Print] button that is available in Window *2.1 Geometry* and *4.1 Graphic* below the graphic (see Figure 13.2).

Clicking this button opens the print dialog box shown in Figure 14.2.

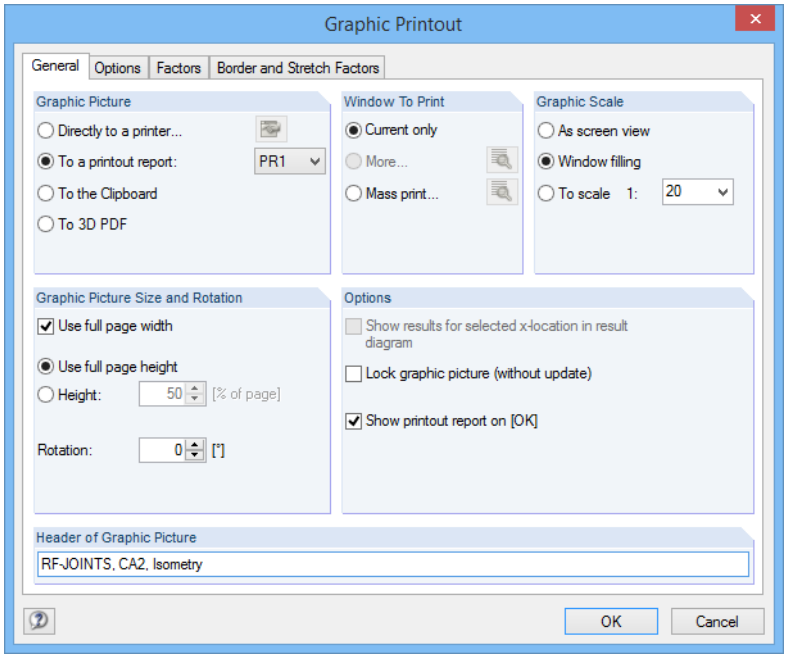


Image 14.2 *Graphic Printout* dialog box, *General* tab

The *Graphic Printout* dialog box is described in Chapter 10.2 of the RFEM or RSTAB manual. The *Options* tab is described in that chapter as well.

If several printout reports are available, you can select the number of the target report in the list.

In order to add several graphics to the printout report in succession, deselect the *Show printout report on [OK]* check box.

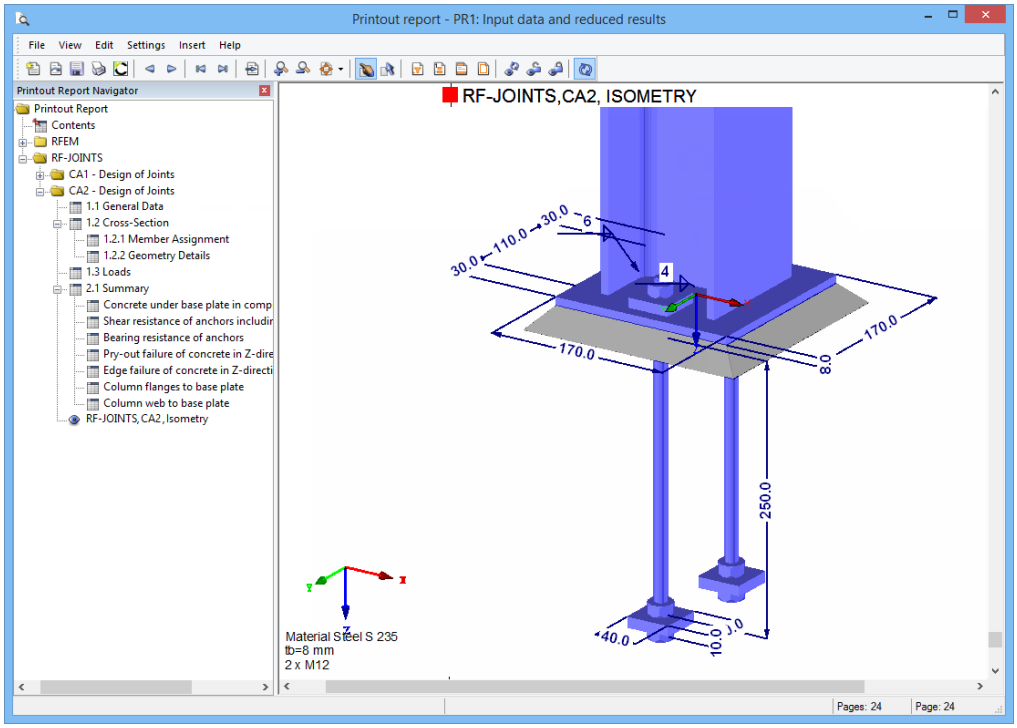


Image 14.3 Connection in printout report

14.2.2 RFEM/RSTAB Graphic

In RFEM or RSTAB, you can transfer every image displayed in the work window to the printout report or send it directly to a printer. In this way, you can prepare the connections displayed on the RFEM or RSTAB model for the printout.

To print the currently displayed graphic of the connection, select

File ⟶ Print Graphic

in the menu or use the corresponding toolbar button.

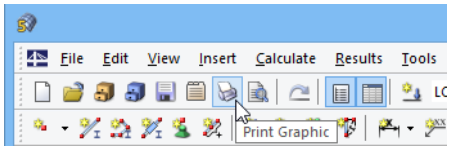
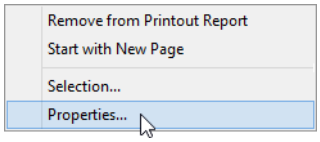


Image 14.4 [Print Graphic] button in RFEM toolbar

The *Graphic Printout* dialog box shown in Figure 14.2 appears.

You can move a graphic to another position in the printout report as usual by using drag-and-drop.



To retroactively adjust a graphic in the printout report, right-click the corresponding entry in the report navigator. The *Properties* option in the shortcut menu opens the *Graphic Printout* dialog box where you can make various adjustments.

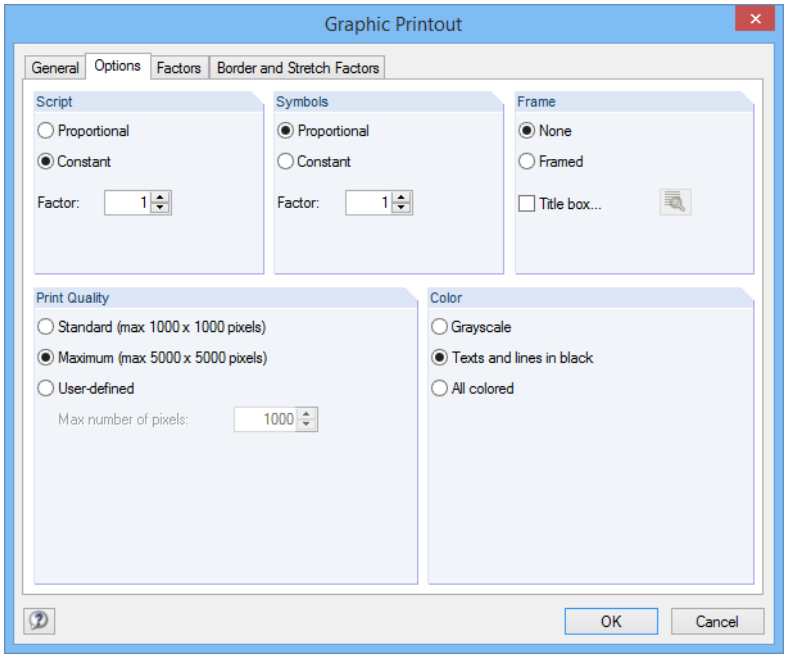


Image 14.5 *Graphic Printout* dialog box, *Options* tab

15 General Functions

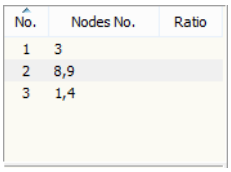
This chapter describes useful menu functions as well as export options for the designs.

15.1 Design Cases

Design cases allow you to group nodes or members for the designs. In this way, you can group connection nodes with the same parameters or analyze members with certain design specifications (e.g. materials, internal forces).

If the conditions of designed nodes are different in terms of the cross-sections, dimensions, number of bolts, etc., you **have to** create a new design case. You cannot manage the design specifications in a single case.

In Window *1.2 Nodes and Members,* you can quickly create new design cases by using the [Transfer the Node to a New Case] or [Transfer Unsuitable Nodes to New Case] button (see Figure 2.10).
Then you can define the node-specific parameters.



The design cases are displayed in the navigator on the top left. To change the design case, click the relevant entry in the list. You can also access the RF-/JOINTS design cases in RFEM or RSTAB by using the load case list in the toolbar.

Creating a new design case

To create a new design case, go to the RF-/JOINTS menu and select

File → New Case.

The following dialog box appears.

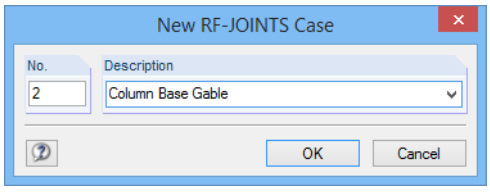


Image 15.1 *New RF-JOINTS Case dialog box*

Enter a *Number* (that is still available) for the new design case in this dialog box. The description facilitates the selection from the load case list.

Click [OK] to open Window *1.1 General Data* in RF-/JOINTS where you can enter the design data.

Renaming a design case

To change the description of a design case, go to the RF-/JOINTS menu and select

File → Rename Case.

The following dialog box appears.

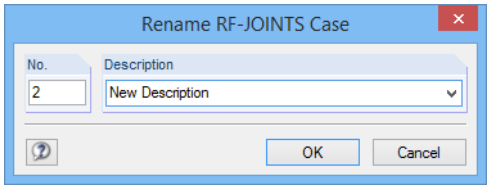


Image 15.2 *Rename RF-JOINTS Case dialog box*

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

Copying a design case

To copy the input data of the current design case, go to the RF-JOINTS menu and select

File ⟶ Copy Case.

The following dialog box appears.

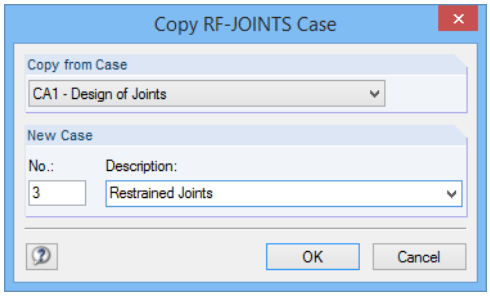


Image 15.3 *Copy RF-JOINTS Case dialog box*

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

Deleting a design case

To delete a design case, go to the RF-JOINTS menu and select

File ⟶ Delete Case.

The following dialog box appears.

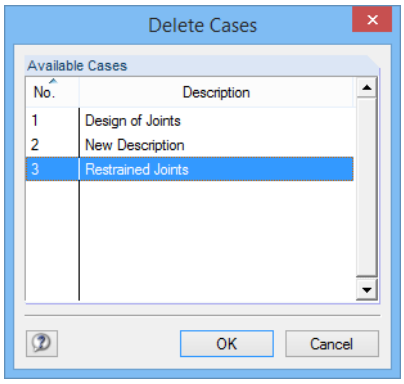


Image 15.4 *Delete Cases dialog box*

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

15.2 Units and Decimal Places

The units and decimal places of RFEM/RSTAB and their add-on modules are managed in one dialog box. To open it, go to the RF-JOINTS menu and select

Settings ⟶ Units and Decimal Places.

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

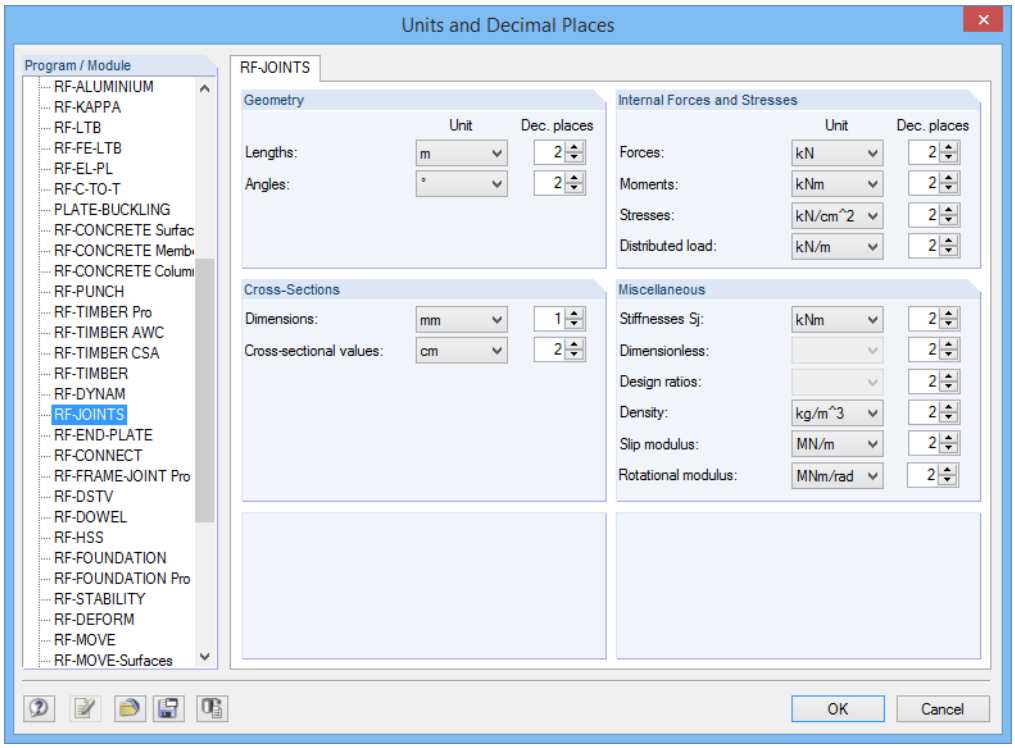


Image 15.5 *Units and Decimal Places dialog box*

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

15.3 Data Export

You can use the input and output data of RF-/JOINTS in other programs as well.

Clipboard

You can copy selected cells in the result windows to the clipboard with [Ctrl]+[C] and paste them, for example, in a word processing program using [Ctrl]+[V]. The headers of the table columns are not transferred.

Printout report

You can print the RF-/JOINTS data into the printout report (see Chapter 14.1) where you can export them using

File ⟶ Export to RTF.

This function is described in Chapter 10.1.11 of the RFEM and RSTAB manual.

Excel

RF-/JOINTS allows you to directly export data to MS Excel or into the CSV format. To access this function, go to the menu and select

File → Export Tables.

The following export dialog box appears.

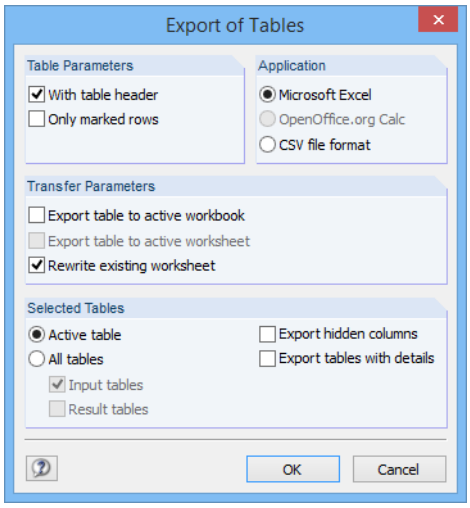


Image 15.6 *Export of Tables* dialog box

When you have selected the relevant options, you can start the export by clicking [OK]. Excel is started automatically, that is, you do not need to open the program beforehand.

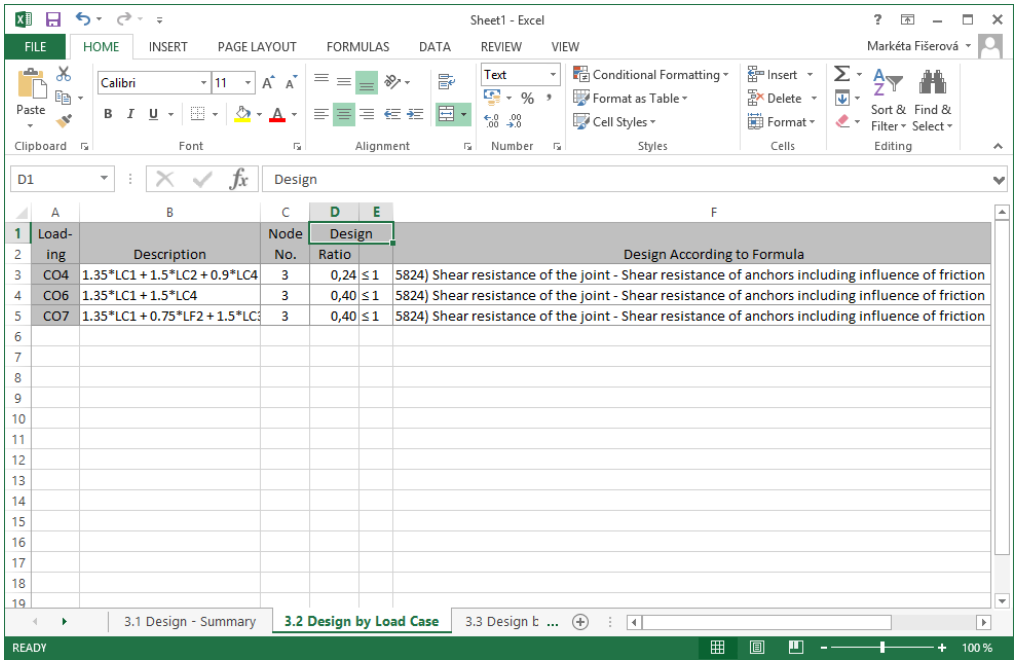


Image 15.7 Results in *Excel*

DXF export

The graphic of the steel or timber connection can be exported as a DXF file. This function is available in Window *4.1 Graphic* by using the [DXF] button.

For the export, the display as a *Wireframe Model* is recommended. Make sure that the relevant

components are displayed in the graphic window.

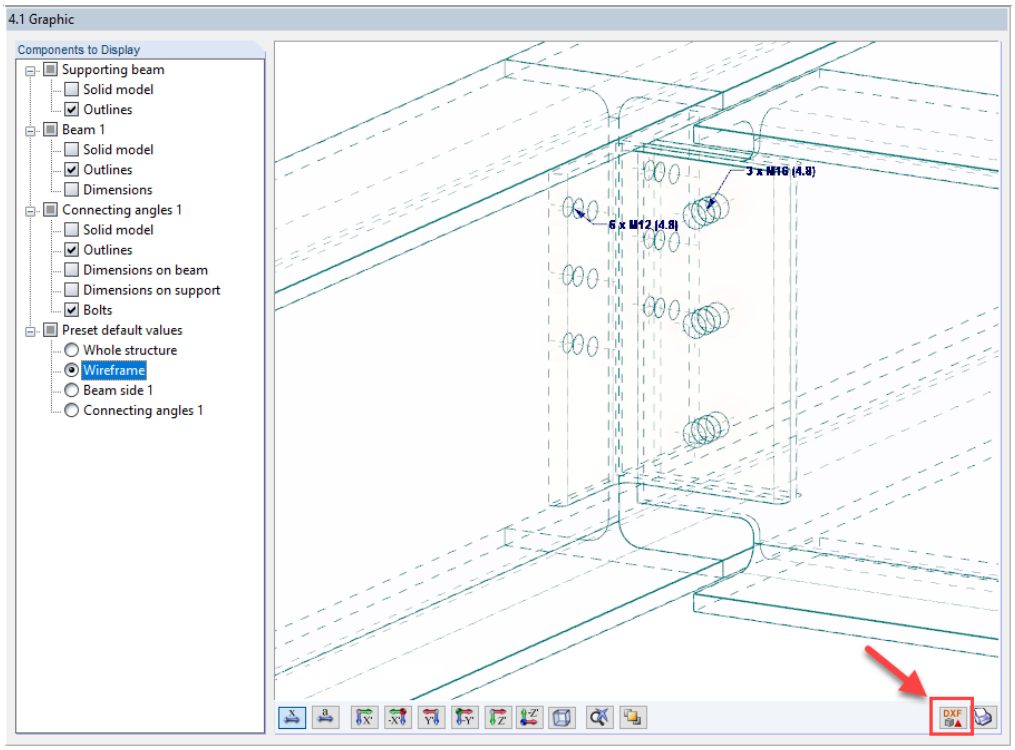


Image 15.8 [Export DXF] button in Window *4.1 Graphic*

After you click the button, the Windows dialog box "Save as" appears where you enter the name and the file path of the DXF file.

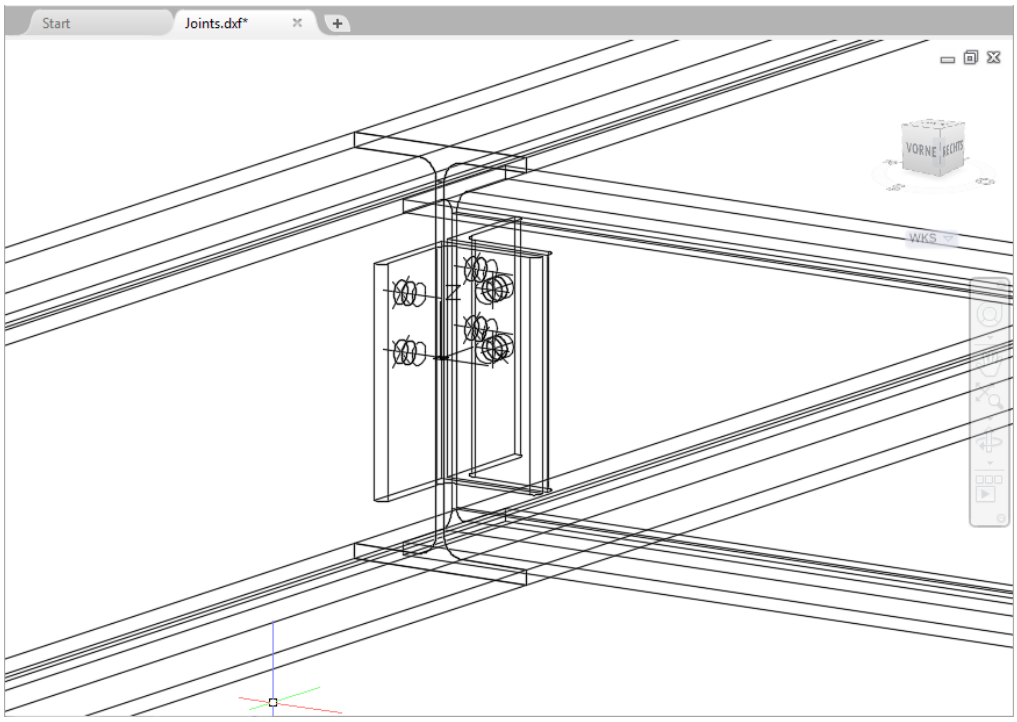


Image 15.9 Result in *AutoCAD*

16 Literature

[1] Eurocode 3: Design of steel structures - Part 1-8: Design of joints; EN 1993–1–8:2005 + AC:2009

[2] Eurocode 5: Design of timber structures - Part 1-1: General - Common rules and rules for buildings; EN 1995-1-1:2010-12

[3] Weynand, K.; Oerder, R.: Typisierte Anschlüsse im Stahlhochbau nach DIN EN 1993-1-8. Düsseldorf: Stahlbau, 2013

[4] Weynand, K.; Oerder, R.: Typisierte Anschlüsse im Stahlhochbau nach DIN EN 1993-1-8 — Ergänzungsband 2018. Düsseldorf: Stahlbau, 2018

[5] National Design Specification (NDS) for Wood Construction 2018 Edition

[6] European Recommendations for the Design of Simple Joints in Steel Structures. ECCS - European Convention for Constructional Steelwork, Mem Martins, edition = 1. 2009.

[7] Informationsdienst Holz Bemessung und Baustoffe STEP1. Informationsdienst Holz, edition = 9.

[8] Blass, H., Ehlbeck, J., Kreuzinger, H., & Steck, G. (2005). Erläuterungen zu DIN 1052: Entwurf, Berechnung und Bemessung von Holzbauwerken (2nd ed.). Karlsruhe: Bruderverlag.

[9] K. W. Johansen. Theory of timber connections. Bern, edition = 9, 1949.

[10] Eurocode 2: Design of concrete structures - Part 1-1: General rules and rules for buildings; EN 1992-1-1:2011-01

[11] Eduard Kahlmeyer and Karin Hebestreit and Werner Vogt. Stahlbau nach EC 3. Werner Verlag, Köln, edition = 6. 2012.

[12] Eurocode 3: Bemessung und Konstruktion von Stahlbauten, Band 2: Anschlüsse, DIN EN 19931-1-8 mit Nationalem Anhang, Kommentar und Beispiele. Beuth Verlag GmbH, Berlin, 2015.

[13] Rolf Kindmann and Michael Stracke. Verbindungen im Stahl- und Verbundbau. Ernst & Sohn, Berlin, edition = 3. 2012.

[14] Eurocode 3: Design of steel structures - Part 1–1: General rules and rules for buildings; EN 1993–1–1:2010–12

[15] Eurocode 3: Design of steel structures - Part 1-5: General rules - Plated structural elements; EN 1993-1-5:2006 + AC:2009

[16] Albert, A. (2018). Schneider - Bautabellen für Ingenieure mit Berechnungshinweisen und Beispielen (23rd ed.). Cologne: Bundesanzeiger.