Ing. Software Dlubal

Version
December 2011

Add-on Module

FRAME-JOINT Pro

Design of Bolted Frame Joints

Program Description

© Ing.-Software Dlubal Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0

Fax: +49 (0) 9673 9203-51

E-mail: info@dlubal.com

Web: www.dlubal.com

Contents

1. Introduction

1.1 Add-on Module FRAME-JOINT Pro

With the FRAME-JOINT Pro add-on module, typical steel construction bolted frame joints can be designed using the RSTAB or RFEM program system. FRAME-JOINT Pro is suitable both for design and analysis of existing I-section connections in steel constructions: the user can work optionally in *design mode* or in *check mode*.

The design and analyses do not only include ultimate limit state design but also a geometrical feasibility of connections with respect to the size of the wrench used for tightening screws.

DIN 18800 or EN 1993-1-8 codes (including national annexes) are implemented in FRAME-JOINT Pro and it is possible to classify the connections according to stiffness. More than 40 checks for various components such as cap plates, stiffeners or bolts, and all types of welds can be performed and described in detail. You will be guided through the program by comfortable input-data dialog boxes with graphical representation of the current connection.

Four basic types of joints are available: knee joint, T-joint, cross joint, and joint with continuous column. The four basic types can be varied in many ways with respect to beam inclination, taper arrangement optionally with cut section beams, stiffeners in columns and beams, backing plates, arrangement of bolts, and distances. Therefore, a large number of connection types can be designed. The following connections and variations are possible:

- Connections of rolled I-sections and/or simply symmetric welded sections
- Flush and extended end plates with checks of positive and negative moment loading
- Tapered beams with cut section beams or steel sheeting (three-flange beam)
- Stiffened and unstiffened connections with one or both side ribs on columns or beams as well as an arrangement of diagonal stiffeners in corners
- Connections with supplementary web plates in columns or beams
- Connections for duopitch or monopitch roof frames with or without cantilevers
- Arrangement of backing plates for bolts

The checking mode includes advanced possibilities to edit hole and edge distances, or thicknesses of welds. This is very suitable especially for quickly checking connections. Geometry and internal forces can be either exported from RSTAB/RFEM or directly entered without any RSTAB/RFEM model.

The results are then displayed in an editable interface in a table and graphically with the important structure dimensions.

We hope you will enjoy working with FRAME-JOINT Pro.

Your team from DLUBAL ENGINEERING SOFTWARE

1.2 FRAME-JOINT Pro-Team

The following people were involved in the development of FRAME-JOINT Pro:

Program coordination

Dipl.-Ing. Georg Dlubal
Dr.-Ing. Klaus-Dieter Klee
Dipl.-Ing. Jürgen Brockmann
M.Eng. Dipl.-Ing. (FH) Walter Rustler

Programming

Ing. Jan Miléř
Dipl.-Ing. Jürgen Brockmann

Cross-section and material database

Ing. Ph.D. Jan Rybín
Jan Brnušák

Program design, dialog box figures, and icons

Dipl.-Ing. Georg Dlubal
MgA. Robert Kolouch
Ing. Jan Miléř
M.Eng. Dipl.-Ing. (FH) Walter Rustler

Program supervision

Dr.-Ing. K.-D. Klee
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Ing. Jan Janovský
Dipl.-Ing. Jürgen Brockmann
Dipl.-Ing. (FH) Steffen Clauß

Manual, help system, and translation

Dipl.-Ing. (FH) Steffen Clauß
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Ing. Ladislav Kábrt
Mgr. Ing. Hana Macková
Bc. Chelsea Jennings
Dipl.-Ing. (FH) Robert Vogl

Technical support and quality management

Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. Moritz Bertram
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (FH) Matthias Entenmann
Dipl.-Ing. Frank Faulstich
Dipl.-Ing. (FH) René Flori
Dipl.-Ing. (FH) Stefan Frenzel
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Bastian Kuhn
M.Sc. Dipl.-Ing. (FH) Frank Lobisch
Dipl.-Ing. (BA) Sandy Matula
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
M.Eng. Dipl.-Ing. (FH) Walter Rustler
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Christian Stautner
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

Topics such as installation, graphical user interface, result evaluation, and printout reports are described in detail in the manuals for RSTAB/RFEM. This manual is focused only on typical features of the FRAME-JOINT Pro add-on module.

The manual describes the order and structure of the input and result tables. The described **buttons** (icons) are stated in the text in square brackets, for example [Apply]. At the same time, they are depicted in the left margin. **Expressions** used in program dialog boxes, tables, or menus are emphasized by *italics*, so the explanation is easy to follow.

At the end of the manual you can find an index. If you still cannot find what you are looking for, please check our website www.dlubal.com where you can go through the *FAQ* pages and find suitable suggestions.

1.4 Starting the FRAME-JOINT Pro Add-on Module

RSTAB/RFEM provides the following options to start the FRAME-JOINT Pro add-on module.

Menu

To start the program on the RSTAB/RFEM menu bar, select

Add-on Modules → Connections → (RF-)FRAME-JOINT Pro.

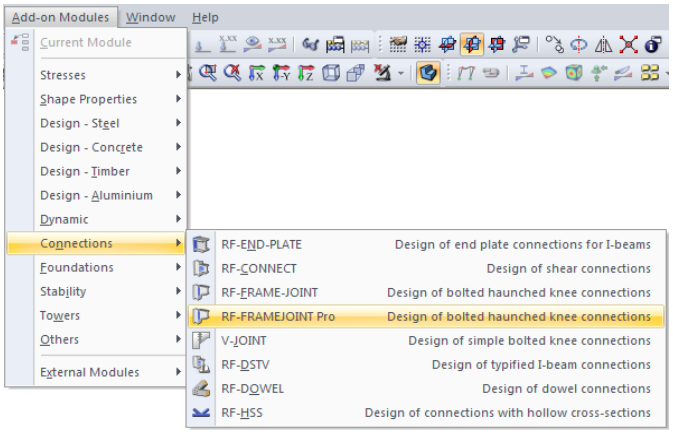


Figure 1.1: Menu *Add-on Modules → Connections → RF-FRAME-JOINT Pro*

Navigator

To start FRAME-JOINT Pro in the *Data* navigator, select

→**FRAME-JOINT Pro** in Add-on Modules.

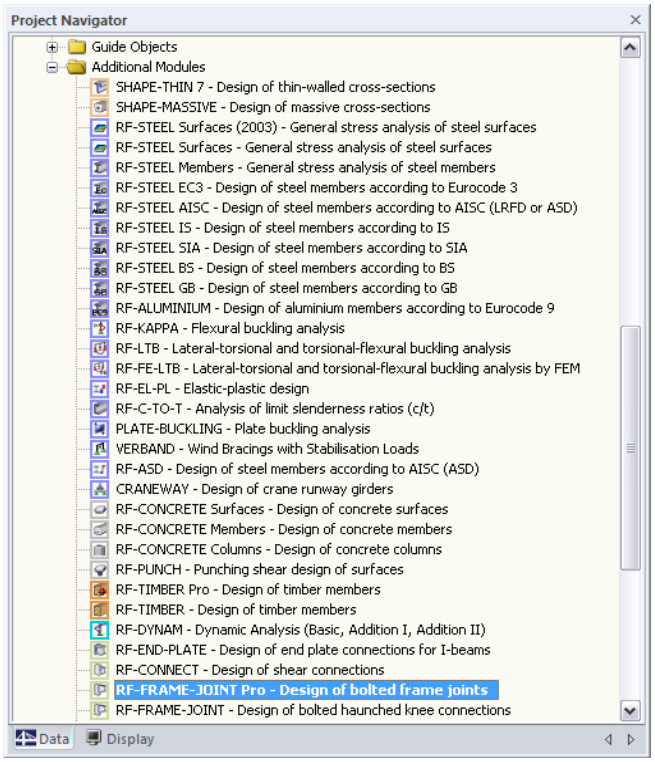


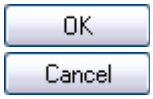
Figure 1.2: Data navigator: *Add-on Modules → RF-FRAME-JOINT Pro*

2. Input Data

The design cases must be defined in several input tables. The [Pick] function is supported for graphical selection of nodes.

After the FRAME-JOINT Pro module is opened, on the left side of the module window you can find a navigator with currently available tables. The pull-down list above the navigator contains the already available design cases (see Chapter 6.1, page 38).

To select a table, click the corresponding entry in the FRAME-JOINT Pro navigator or page through the tables by using the buttons shown in the left. You can also use the [F2] and [F3] function keys to select the previous or next table.



By clicking [OK] you save the entered data and close the FRAME-JOINT Pro module. By clicking [Cancel] you close the module without saving the data.

2.1 General Data

In the 1.1 *General Data* table you can define a node of a joint to design, the code used for the design, the type of frame joint, calculation mode, design type, and symmetry settings.

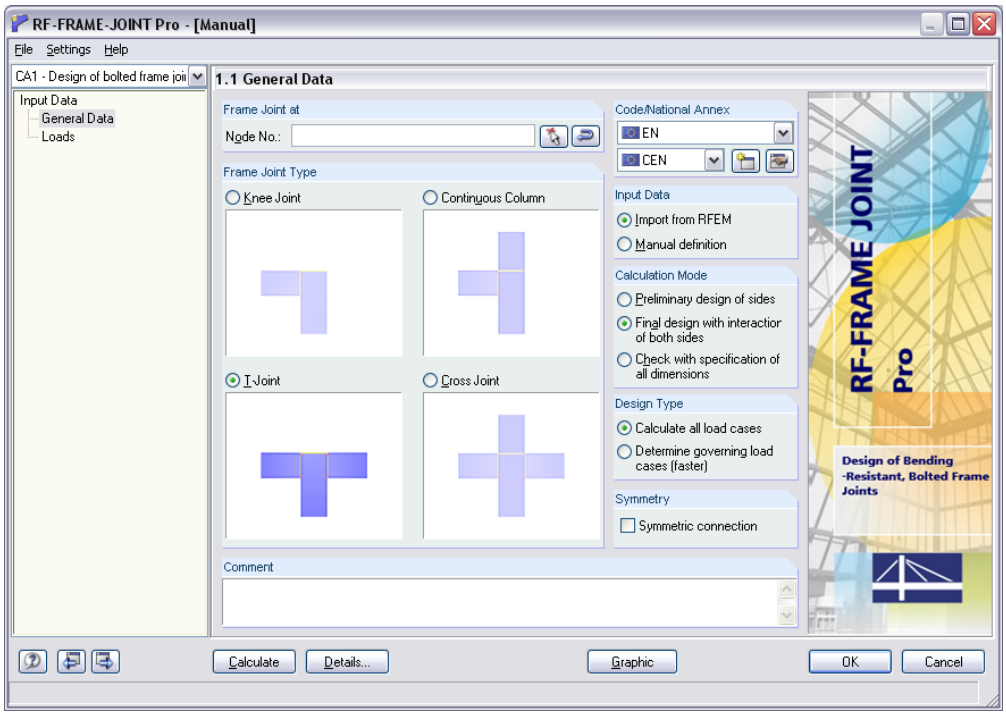


Figure 2.1: Table 1.1 *General Data*

Frame Joint at

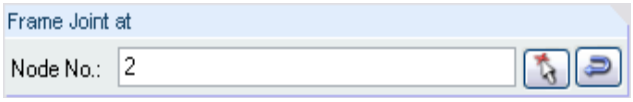
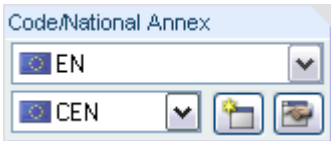


Figure 2.2: Table 1.1, Section *Frame Joint at*

When importing data from RSTAB/RFEM, you have to define the particular nodes of the joints. The program then distinguishes automatically the appropriate type of frame joint and selects it.

Only nodes with identical geometry can be selected. If the geometry is different, an error message appears during the input settings and the calculation is not possible. The program checks the connected members concerning material, cross-section, member inclination, rotation, and length. When this data does not correspond, the nodes cannot be calculated in the same case of FRAME-JOINT Pro. However, different frame joints can be calculated in different FRAME-JOINT Pro cases.

The [Update] button must be used when the model or the selected nodes are changed. Automatic update is not provided, because it could overwrite possible user-defined data.



Code/National Annex

Calculation according to DIN 18800 and EN 1993-1-8:2010 with related application documents is available.

For EN 1993-1-8:2010, you can set the national parameters in another dialog box using the [Edit National Annex] button. You can also create a user-defined "National Annex" via the [Create New ...] button.

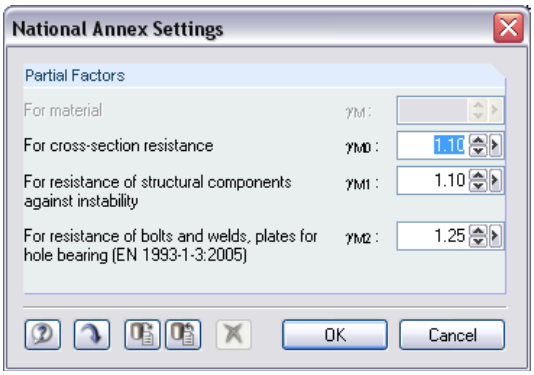


Figure 2.3: Dialog box *National Annex Settings*

There is the possibility to change the national parameters. You can revert all changes using the [Default] button.

If you want to use the selected parameters of the National Annex as default for next time, you can set them using the [Set as Default] button.

A user-defined National Annex can be deleted using the [Delete] button.

For design according to DIN 18800 you can set only the material partial factor in this dialog box.

Frame Joint Type

FRAME-JOINT Pro enables you to design four basic types of frame joints:

- Knee joint
- Continuous column
- T-joint
- Cross joint

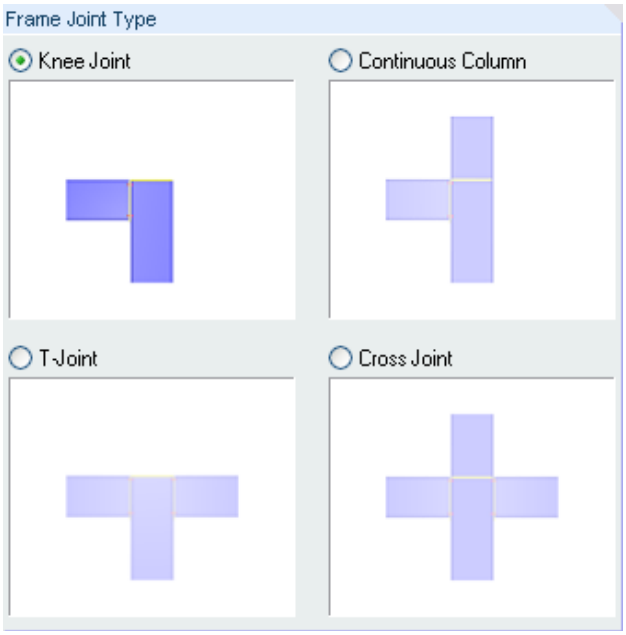


Figure 2.4: Table 1.1, Section *Frame Joint Type*

When you import data from RSTAB/RFEM, the type is automatically recognized and usually does not need to be changed. When you define the frame joint manually, the type must be selected.

In the following tables, you can define in detail stiffeners, various bolt layouts, tapers, etc. for all types of joints. Therefore, a large variety of different connections can be calculated.

Input Data

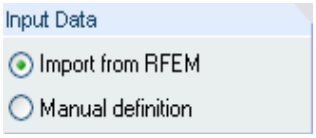


Figure 2.5: Table 1.1, Section *Input Data*

Two possibilities of input data definition are available.

When you use *Import from RSTAB/RFEM* you obtain cross-section geometry and also input data for load cases, load groups, or load combinations from the main program. It will not be possible to change the cross-sections of the connection in the following tables anymore.

When you use *Manual definition*, you can edit the cross-section afterward. Furthermore, you must define the internal forces in system node E. You can find more explanations in Table 1.6 *Loads* described in Chapter 2.7 from page 24.

Calculation Mode

FRAME-JOINT Pro provides you with three basic types of calculation modes:

- Preliminary design of sides
- Final design with interaction of both sides
- Check with specification of all dimensions

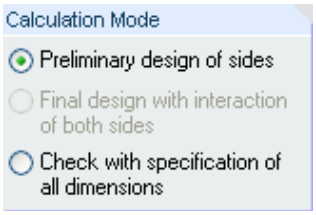


Figure 2.6: Table 1.1, Section *Calculation Mode*

When you select the **Preliminary design of sides** option, the connections of beams will be designed considering the number of required rows of bolts. The program provides for the right and left sides detailed results with different variants. You are then able to select from them the most suitable solution.

In the second step, the **Final design with interaction of both sides** option performs the remaining checks of columns and associated stiffeners and supplementary plates without changing the calculated number of bolt rows.

When using the **Check with specification of all dimensions** option, you must define explicitly all geometrical dimensions (distances, welds, etc.) This mode is intended mainly for checking the already existing frame joints; therefore it does not provide any design or optimization functions.

Recommendation:

For the first design of a frame joint, start with the *Preliminary design of sides* mode. You will obtain different solution suggestions to select. The program then automatically changes the mode to *Final design with interaction of both sides*.

For knee joints, the interaction calculation is not needed because only one side exists. The mode is then changed to the checking mode (Check with specification of all dimensions).

Design Type

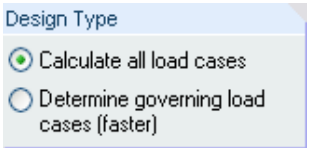


Figure 2.7: Table 1.1, Section *Design Type*

When you select the **Calculate all load cases** option, all load cases, load groups, and load combinations defined in Table 1.6 *Loads* are calculated, and variants and checks are displayed.

The **Determine governing load cases** option calculates internal forces of particular load cases and generates internally an envelope from the unfavorable internal force - combination.

Recommendation:

Use the *Calculate all load cases* option.

Symmetry

Figure 2.8: Table 1.1, Section *Symmetry*

Selecting this check box you define a symmetric connection. It means that the beam or column cross-sections are same on opposite sides. The data for beams can be then defined only once.

Comment

You can use this text box for your notes describing, for example, the current FRAME-JOINT Pro design case.

2.2 Cross-Sections

2.2.1 Case 1: *Import* and *Preliminary* or *Final design*

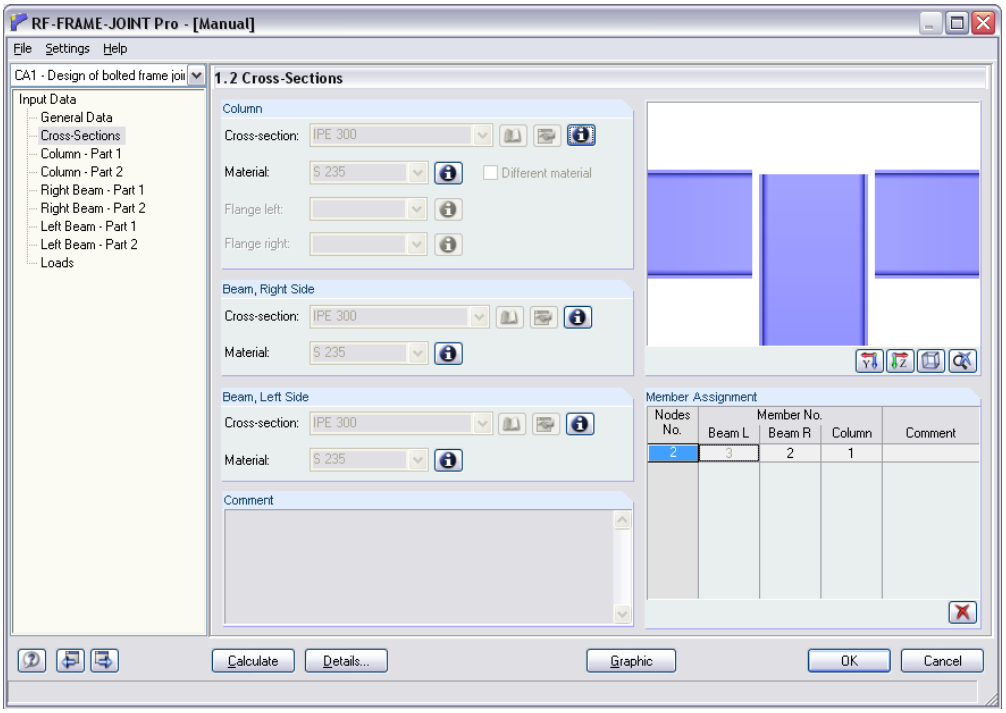
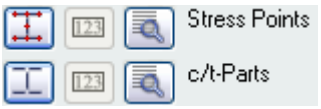


Figure 2.9: Table 1.2 *Cross-Sections* for data input option *Import from RSTAB/RFEM*

Column / Beam, Right Side / Beam, Left Side

The [Info] buttons are placed on the right of the input data text boxes for cross-section and material. Clicking the button on the right from the *Cross-section* you open the *Info About Cross-Section...* dialog box. There you can see all properties of the selected cross-section.



In the *Info About Cross-Section...* dialog box you can check the *Stress Points* and *(c/t)-Parts*: After selecting the *Stress Points* and *Numbering* options all created stress points are automatically displayed in the cross-section figure. Clicking the [Details] button shows the coordinates and static properties of the stress points in a new dialog box. After selecting the *(c/t)-Parts* and *Numbering* options all cross-sectional (c/t)-parts are automatically displayed in the cross-section figure. You can display the width c to thickness t ratio of all (c/t)-parts in a table by clicking the [Details] button.

The [Info] button next to *Material* shows an overview about the basic material properties and the properties concerning the design.

Member Assignment

In this table, members are to be assigned to the selected nodes. You can change the assignment using the [...] button. An automatic check follows to determine that the changes are acceptable for the calculation. If they are not, a comment number is displayed in the table and explained in the *Comment* section.

Graphic window

The graphic window in this table shows a rendered view of all cross-sections existing at the designed node. You can change the view by clicking the buttons under the graphic window. With the mouse pointer inside the graphic window, you can zoom or rotate the view. You can find more information in [3], Chapter 4 *User interface*.

2.2.2 Case 2: *Manual definition* and *Preliminary* or *Final design*

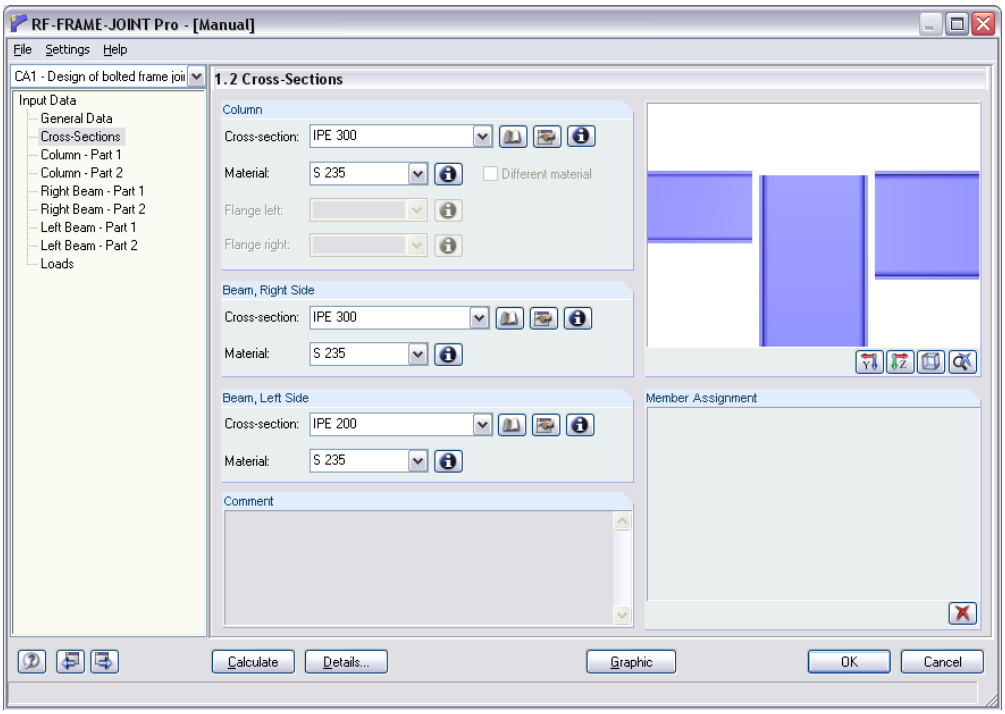


Figure 2.10: Table 1.2 *Cross-Sections* for data input option *Manual definition*

Column / Beam, Right Side / Beam, Left Side

Cross-section and *Material* must be defined manually. You can define this in the appropriate text boxes. There is the possibility to select the cross-sections defined in RSTAB/RFEM. The [Cross-Section Library] button opens the RSTAB/RFEM cross-section library and imports the settings of a selected cross-section.

You can edit the cross-sections by the [Edit Cross-Section] button.

The thicknesses of the welds from RSTAB/RFEM are used for the check of welds in welded sections. This data is already specified in the *Edit Cross-Section* dialog box for *top* and *bottom* parameters. The *bottom* weld from RSTAB/RFEM represents in FRAME-JOINT Pro the *right* side (resp. the *left* side for a member rotated about 180°).

You can use different materials for *left* and *right flanges* as well.

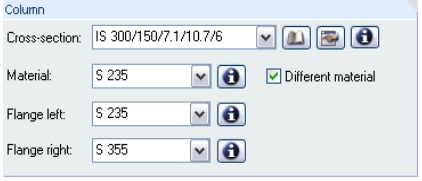


Figure 2.11: Use of different materials

2.3 Column - Part 1

The 1.3.1 *Column - Part 1* table deals with additional available column stiffeners such as end plates, web stiffeners, diagonal stiffeners, or supplementary web plates.

2.3.1 Case 1: *Import* and *Preliminary* or *Final design*

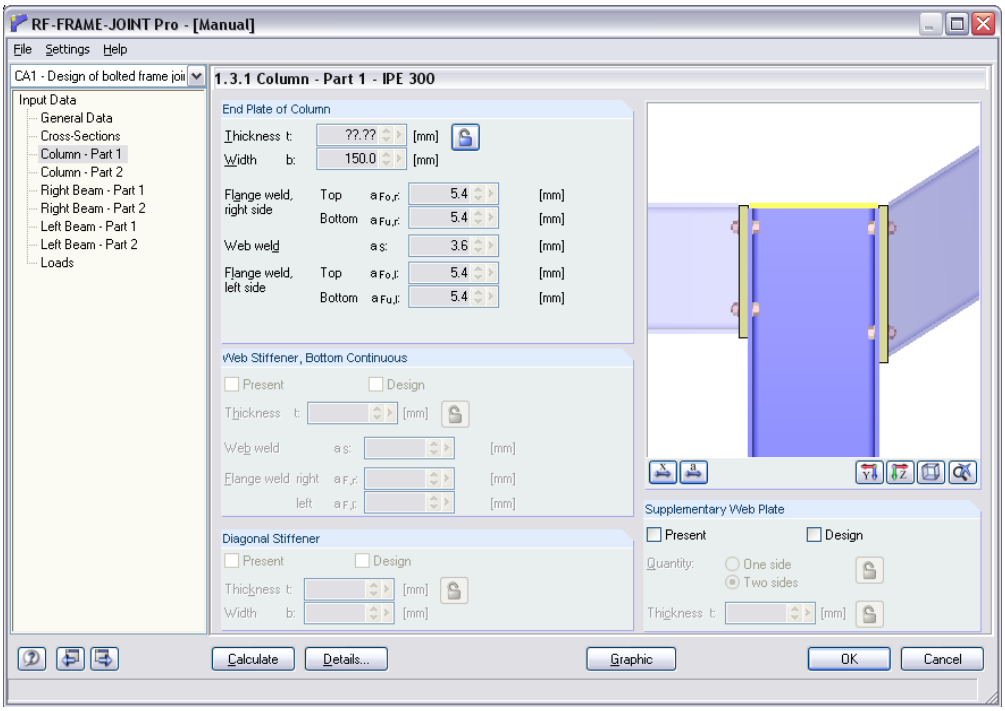


Figure 2.12: Table 1.3.1 *Column - Part 1* for data input option *Import* and *Preliminary* or *Final design*

End Plate of Column

Using the [Edit manually end plate thickness] button you can manually control the thickness of the end plate before the design. The width of the end plate is automatically set to the width of the connected cross-section.

Web Stiffener, Bottom Continuous

If the geometry enables a continuous web stiffener, you have the following possibilities:

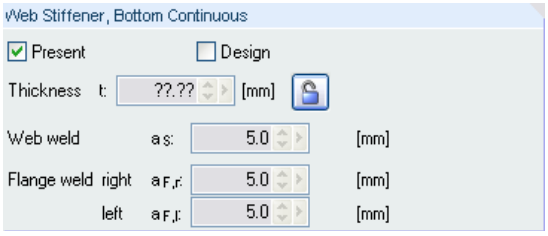


Figure 2.13: Options for continuous web stiffener

You can define the connection in the text boxes in this section. When the *Present* option is selected, you can define or limit manually using the [Edit manually thickness of web stiffener...] button the thickness of the web stiffener before the design. You define by this option, that the web stiffener is always present. When the *Design* option is used, the program analyzes the demand for the web stiffener.

Diagonal Stiffener

When you design a knee joint, you can define a diagonal stiffener in a similar way as described above.

Supplementary Web Plate

In the section under the graphic window you can define a supplementary web plate for the column. You have two possibilities:

When you select the *Present* option, a supplementary web plate is always considered. If there is nothing else defined, the program specifies the layout and thickness of the plate. However, you can define the *Quantity* and *Thickness t* manually, as well. The program then uses the defined parameters in case the plate is required.

The *Design* option in the design mode follows the same rules; however, the plate is considered only when it is required.

Graphic window

Together with the options described in Chapter 2.2 you can use the [Dimensioning] and [Dimensioning with symbols] buttons.

2.3.2 Case 2: *Manual definition* and *Preliminary* or *Final design*

There are no differences for Case 1 and Case 2 regarding Table 1.3.1 *Column - Part 1*.

2.3.3 Case 3: *Manual definition* and *Check*

End Plate of Column

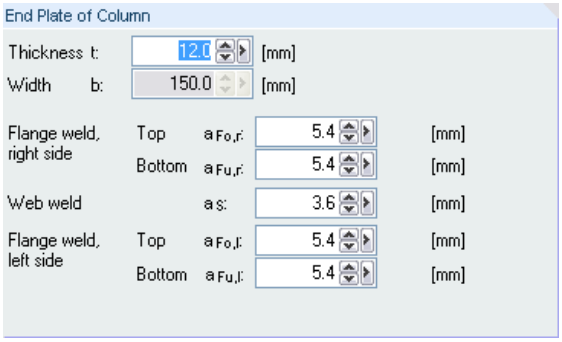


Figure 2.14: Table 1.3.1, Case 3: Section *End Plate of Column*

Define all input data for thickness of end plates as well as thicknesses of welds at particular places.

Web Stiffener, Bottom Continuous

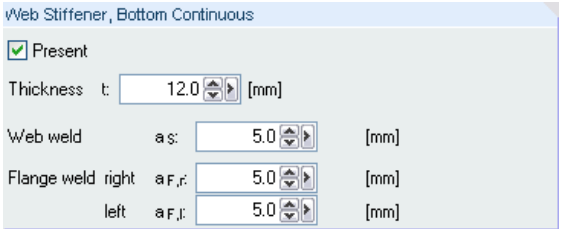


Figure 2.15: Table 1.3.1, Case 3: Section *Web Stiffener, Bottom Continuous*

By selecting the check box *Present* you can edit the text boxes underneath with thickness, web and flange welds.

Supplementary Web Plate

Figure 2.16: Table 1.3.1, Case 3: Section *Supplementary Web Plate*

When you select the *Present* option, you are enabled to fill in the thickness and number of the supplementary web plates.

2.4 Column - Part 2

The 1.3.2 *Column - Part 2* table deals with additional column stiffeners like backing plates or local web stiffeners.

2.4.1 Case 1: *Import* and *Preliminary* or *Final design*

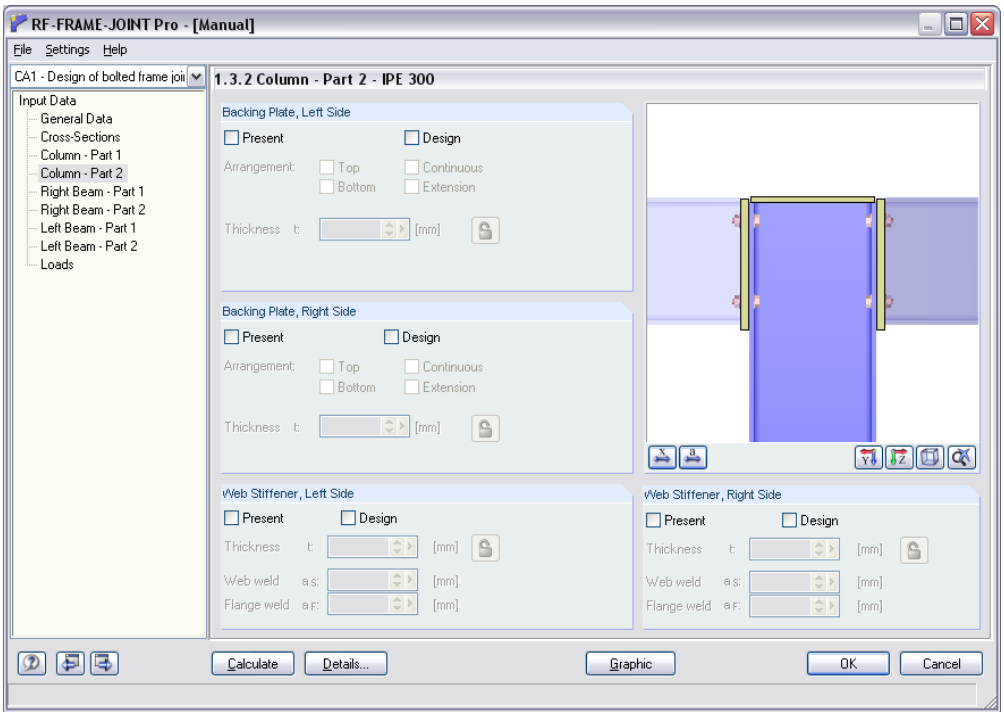


Figure 2.17: Table 1.3.2 *Column - Part 1* for data input option *Import* and *Preliminary* or *Final design*

Backing Plate Left / Right

The both sections control the arrangement and type of backing plates.

Under the *Present* option, you can edit or limit the thickness of the backing plate by the [Manually edit thickness ...] button before the design starts. You define by this option, that the backing plate is always present. On the contrary when the *Design* option is used, the program analyzes the demand for the backing plate.

Furthermore you can define the arrangement manually. For both sides you can select options *Top*, *Bottom*, *Continuous,* or *Extension*. All changes are immediately displayed in the graphic window.

Web Stiffener Bottom Left / Right

Use the same definition as described in Chapter 2.3 *Column - Part 1*.

Note:

If a web stiffener is required by the design of the connection on both sides, the program automatically arranges a continuous stiffener for the case of the stiffeners at the same height.

2.4.2 Case 2: *Manual definition* and *Preliminary* or *Final design*

There are no differences for Case 1 and Case 2 regarding Table 1.3.2 *Column - Part 2*.

2.4.3 Case 3: *Manual definition* and *Check*

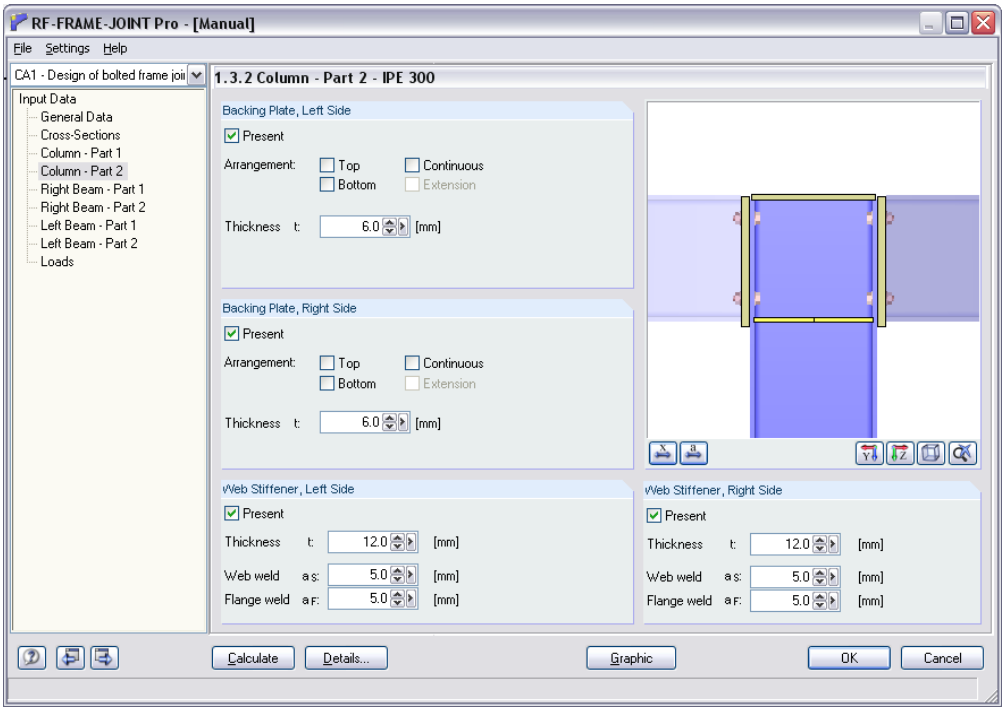


Figure 2.18: Table 1.3.2 *Column - Part 2* for data input option *Manual definition* and *Check*

Backing Plates Left / Right

You can edit the arrangement of the backing plates using the *Present* check box. You edit or limit the backing plate thickness manually. Define by this option, whether the backing plate is present. For both sides you can select the *Top*, *Bottom*, *Continuous,* or *Extension* options. All changes are immediately displayed in the graphic window.

Web Stiffener Bottom Left / Right

You can manually edit or limit the thickness of the web stiffener using the *Present* check box. By this option you define that the web stiffener is present.

By the other option you input the thicknesses of the welds.

2.5 Right Beam - Part 1 / Left Beam - Part 1

2.5.1 Case 1: *Import* and *Preliminary* or *Final design*

If the defined connection is asymmetric, you can see the 1.4.1 *Right Beam - Part 1* or *1.4.1 Left beam - Part 1* tables. Analogous to the above described tables for the columns, you can define here the following components of the beam.

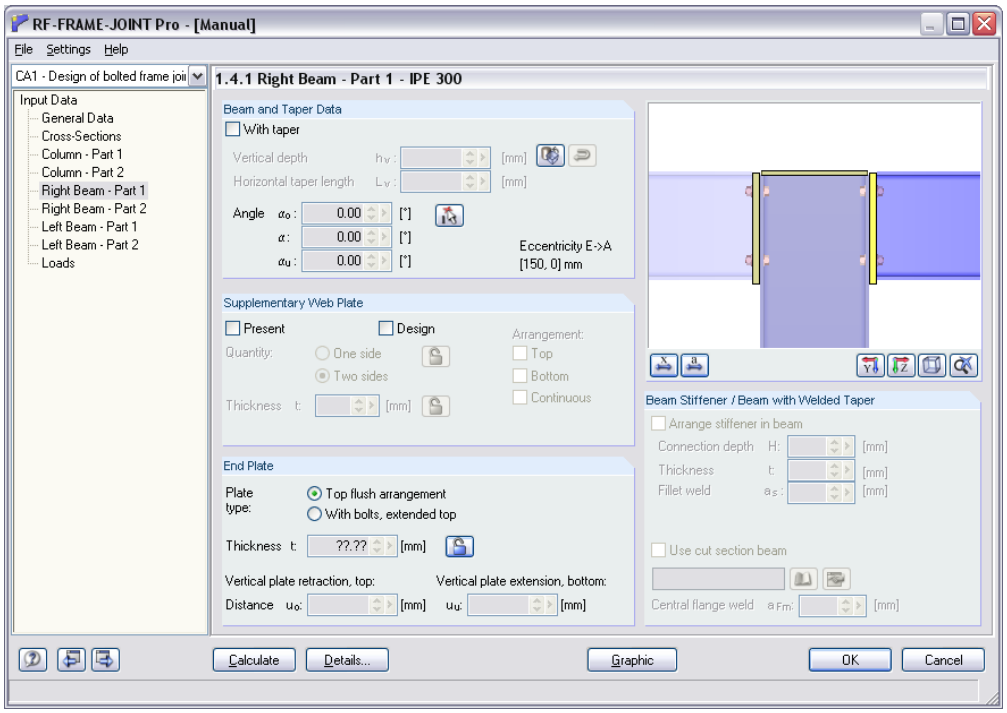


Figure 2.19: Table 1.4.1 *Right Beam - Part 1* for data input option *Import* and *Preliminary* or *Final design*

Beams and Taper Data

When a tapered beam is used in a RSTAB/RFEM model, the program recognizes that and automatically turns on the check box *With taper*. After that, you can type in the input text boxes.

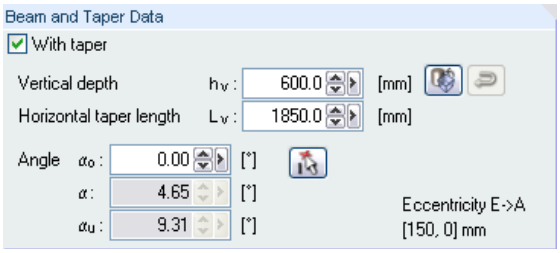


Figure 2.20: Section *With taper*

You can define the *Vertical depth* h_v manually or import it from RSTAB/RFEM using the [Reset hv from the model] button.

If you edit the *Horizontal taper length* L_v, the corresponding angles of the taper length and height are also changed. You can also select the *Angle* from the model by clicking the [Select Alpha...] button.

Supplementary Web Plate

In this section you can define a supplementary web plate for the beam. You have two possibilities:

When you select the *Present* option, the supplementary web plate is always considered. If there is nothing else defined, the program specifies the layout and thickness of the plate. However, you can define the *Quantity* and *Thickness t* manually, as well. The program then uses the defined parameters in case the plate is required.

The *Design* in the design mode follows the same rules; however the plate is considered only when it is required.

Furthermore you have three possibilities of *Arrangement* of the plate (*Top*, *Bottom,* or *Continuous*). All changes are immediately displayed in the graphic window.

End Plate

For the end plate on a beam, you can define a *Plate type*. You can select the *Top flush arrangement* and *With bolts, extended top* options. Using the [Edit manually end plate thickness] button you can control or limit manually the thickness of the end plate before the design.

Beam Stiffener / Beam with Welded Taper

You can use the options in this section when a tapered beam is defined (see previous).

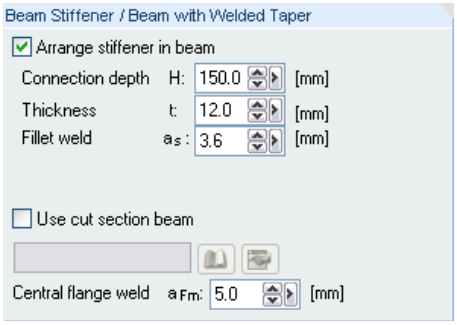


Figure 2.21: Section *Beam Stiffener / Beam with Welded Taper*

You can edit the following text boxes for position and thickness of the stiffener by selecting the *Arrange stiffener in beam* check box. The *Connection height* H is then the distance between the upper edge of the end plate and the upper edge of the stiffener. Furthermore, you can input the *Thickness* of the stiffener t and *Fillet weld thickness* a_s.

By selecting the *Use cut section beam* check box, you can use the [Cross-section library] button to select a cross-section, which should be connected to the existing section. The height of the cut section beam is adjusted then according to the vertical connection height h_v.

You can edit the cross-sections using the [Edit cross-section] button.

2.5.2 Case 2: *Manual definition* and *Preliminary* or *Final design*

There are no differences for Case 1 and Case 2 regarding Table 1.4.1 *Beam - Part 1*.

2.5.3 Case 3: *Manual definition* and *Check*

Supplementary Web Plate

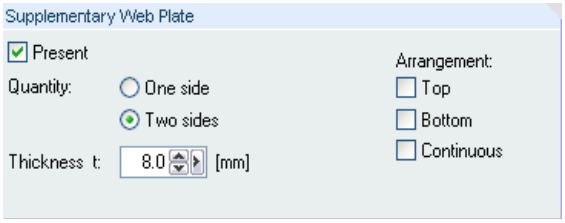


Figure 2.22: Table 1.4.1, Case 3: Section *Supplementary Web Plate*

You can edit properties of the supplementary web plate using the *Present* check box. Below the plate *Quantity* settings you can define the *Thickness*. You define by this option, whether the supplementary web plate is present.

Furthermore you have three possibilities of *Arrangement* of the plate (*Top*, *Bottom,* or *Continuous*). All changes are immediately displayed in the graphic window.

End Plate

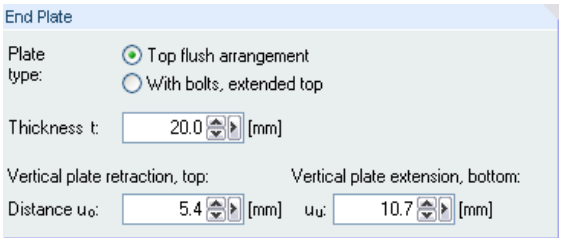


Figure 2.23: Table 1.4.1, Case 3: Section *End plate*

For the end plate on a beam, you can define a *Plate type*. You can select the *Top flush arrangement* or *With bolts, extended top* options.

After the end plate *Thickness* setting you can define the *Vertical plate retraction, top* and *Vertical plate extension, bottom*.

2.6 Right Beam - Part 2 / Left Beam - Part 2

2.6.1 Case 1: *Import* and *Preliminary* or *Final design*

Tables 1.4.2 *Right beam - Part 2* and 1.4.2 *Left beam - Part 2* describe the bolts and welds used for the beam.

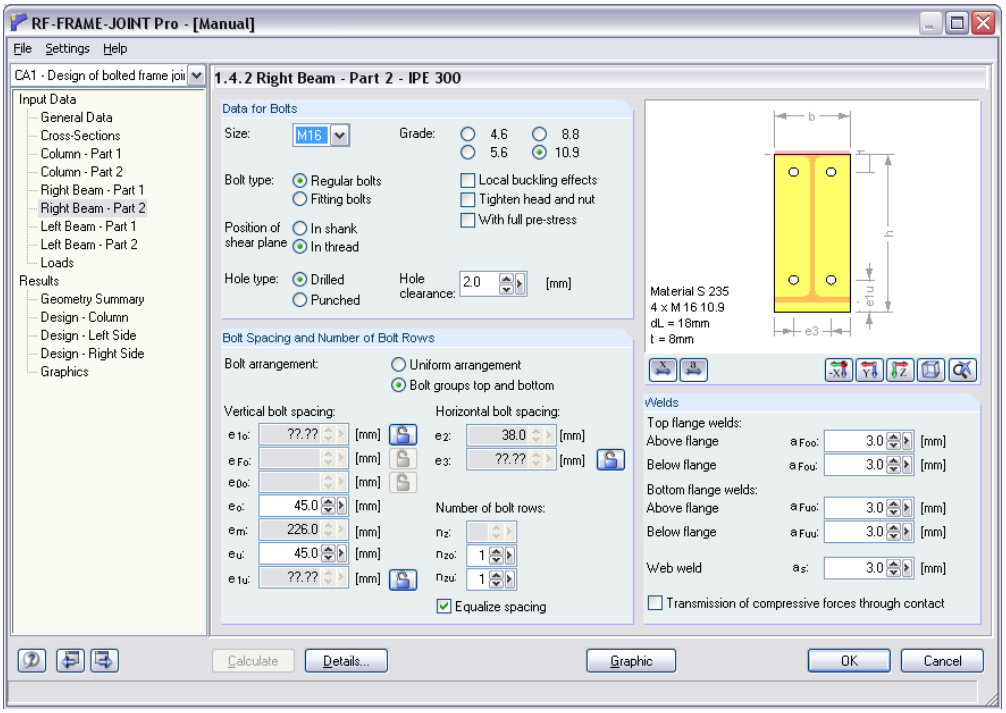


Figure 2.24: Table 1.4.2 *Left beam - Part 2* for data input option *Import* and *Preliminary* or *Final design*

Settings for Bolts

Using the first text box, *Size,* you can define the used bolt. Then set the *Grade* of the bolt. You can select Regular or Fitting bolts as the *Bolt type*. Furthermore you have to define whether the *Position of shear plane* is in the shank or the thread.

For the *Hole type* you can select between drilled and punched holes and if needed, you can define the *Hole clearance*. If applicable, the text boxes with other parameters for bolts or end plates are available (*Local buckling effects*, *Tighten head and nut* and *With full pre-stress*).

Bolt Spacing and Number of Bolt Rows

There are two check boxes for the *Bolt arrangement*. You can select the *Uniform arrangement* of the bolts over the connection height or arrangement in *Bolt groups top and bottom* (compression and tension zone).

The vertical and horizontal *Bolt spacing* can be defined either manually by the button displayed in the left margin or it can be determined by the program during the design.

The number of bolts is dependent on the selected bolt arrangement: when selecting a uniform arrangement, the n_z text box is available; when selecting bolt groups, the data input is divided for the top and bottom part.

Welds

The program analyzes the loading and determines the required dimensions of the welds between components. Finally it checks the welds.

Transmission of compression forces through contact

If the compressive forces are transferred by contact, the weld check is not preformed (Transmission of compressive forces through contact). According to EN 1993-1-8, Chapter 4.7.1 [2], if the ultimate limit state of butt welds is equal to the ultimate limit state of the weaker from the connected components, then this check can also be omitted.

2.6.2 Case 2: *Manual definition* and *Preliminary* or *Final design*

There are no differences for Case 1 and Case 2 regarding Table 1.4.2 *Beam - Part 2*.

2.6.3 Case 3: *Manual definition* and *Check*

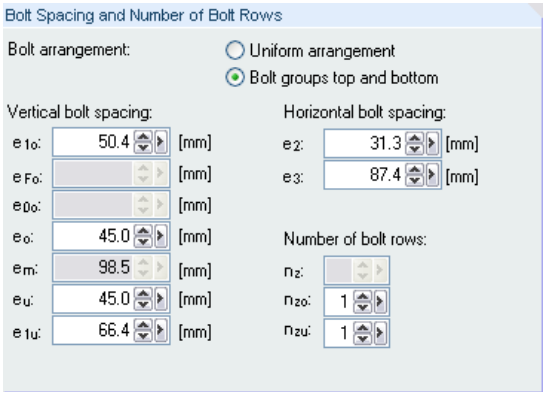


Figure 2.25: Table 1.4.2, Case 3: Bolt spacing and number of bolt rows

Define all text boxes for *Bolt spacing* and *Bolt rows* manually; the spacing restrictions between individual bolts must be respected. The number of bolts is dependent on the selected bolt arrangement. For the bolt group arrangement option, the n_{zo} and n_{zu} text boxes are available. The input option is separated for the top and bottom part.

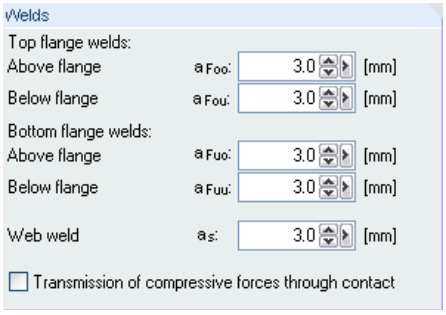


Figure 2.26: Table 1.4.2, Case 3: Welds

In this section, select *Welds* for design.

2.7 Loads

The appearance of Table 1.6 *Loads* depends on the input data from the previous tables.

2.7.1 Case 1: *Import* and *Preliminary* or *Final design*

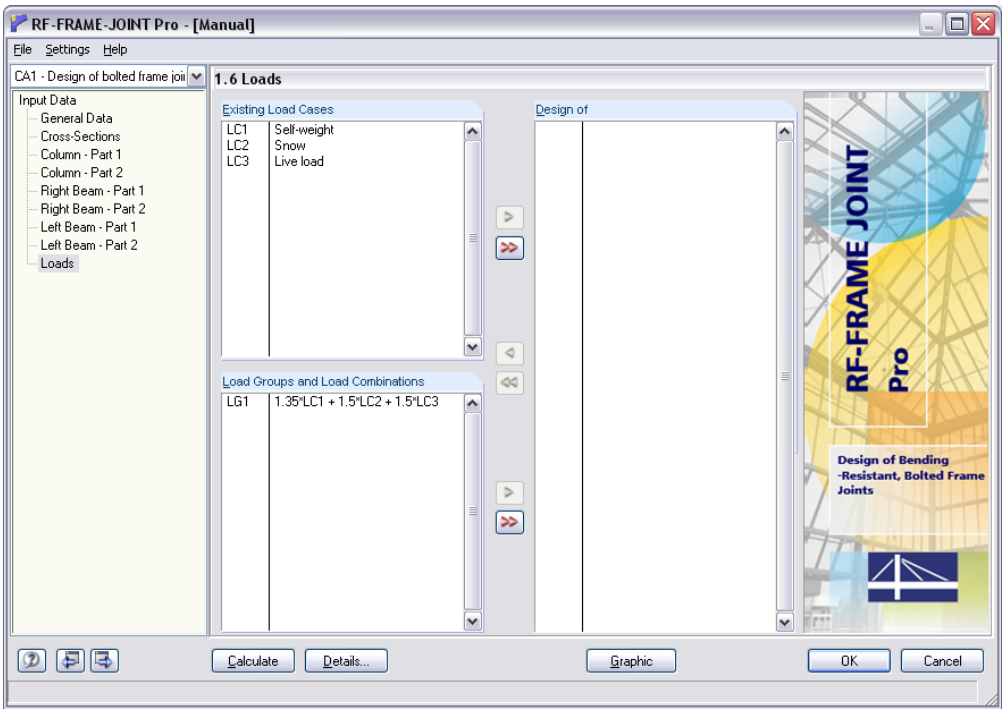


Figure 2.27: Table 1.6 *Loads* for data input option *Import* and *Preliminary* or *Final design*

Existing Load Cases / Load Groups and Load Combinations

In both sections there are listed all the load cases, groups, and combinations defined in RSTAB/RFEM, that can be used. Using the [▶] button, you can select load cases, groups, or combinations to the list *Design of* on the right side. The selection can be done also by double-clicking. The [▶▶] button moves the complete list to the right.

The load cases marked with an asterisk (*) cannot be used for design. Those are load cases without loads or imperfection load cases, for example.

Selected for Design

In the right column, there are the actions selected for design. Using the [◀] button, you can remove the load cases, groups, or combinations back to the left side. Again, the selection can be done by double-clicking. The [◀◀] button removes the complete list.

2.7.2 Case 3: *Manual definition* and *Check*

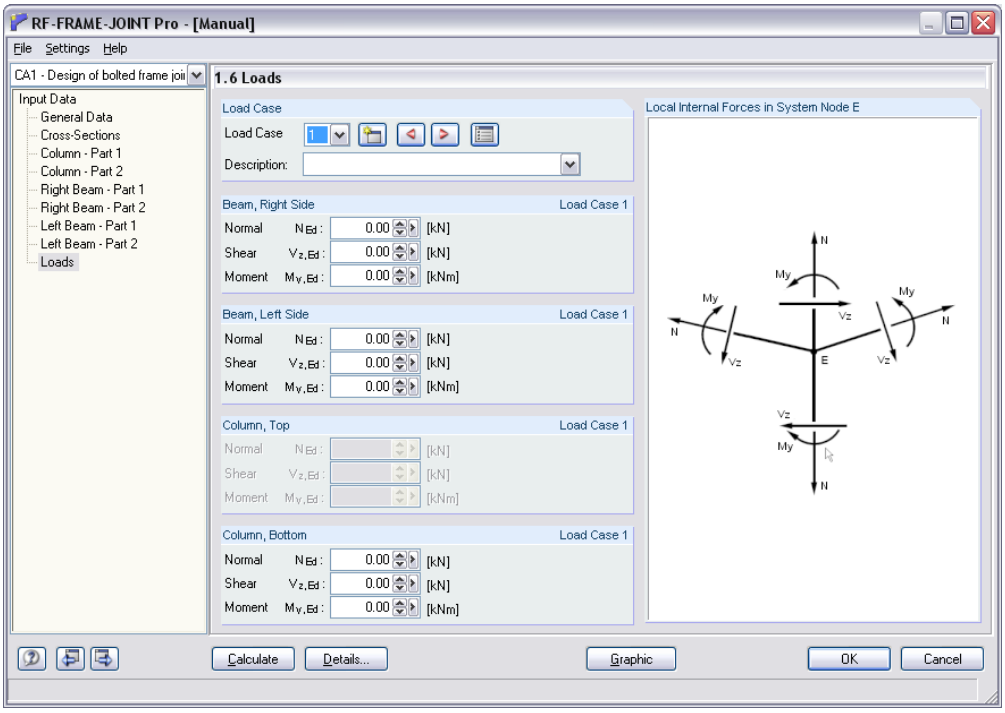


Figure 2.28: Table 1.6 *Loads* for data input option *Manual definition* and *Check*

For the *Manual definition* option, you can directly input the internal forces. You must consider that the internal forces are defined in the **System node E** with respect to the local coordinate system.

The available text boxes are also dependent on the *Frame joint type* selected in Table 1.1 *General Data*.

Note:

The internal forces must be defined in the system node E. It can happen that because of the connection geometry, the internal force at the beam edge is reduced when the shear force is positive and simultaneously the moment is negative.

Load Case

By using the [Create new load case] button you can define a new load case to design.

If you define more load cases, you can move between them using the [◀] and [▶] buttons.

The [List of load cases...] opens the *Overview of load cases* dialog box. There, you can check the already defined load cases. You can also [Delete] the no longer needed load cases.

2.8 Classification

2.8.1 Case 1: *Import* and *Preliminary* or *Final design*

Using the [Details] button, you can select to perform the classification of the connection (see Chapter 3.1, page 27). In that case Table 1.7 *Classification* appears and you can input the data there.

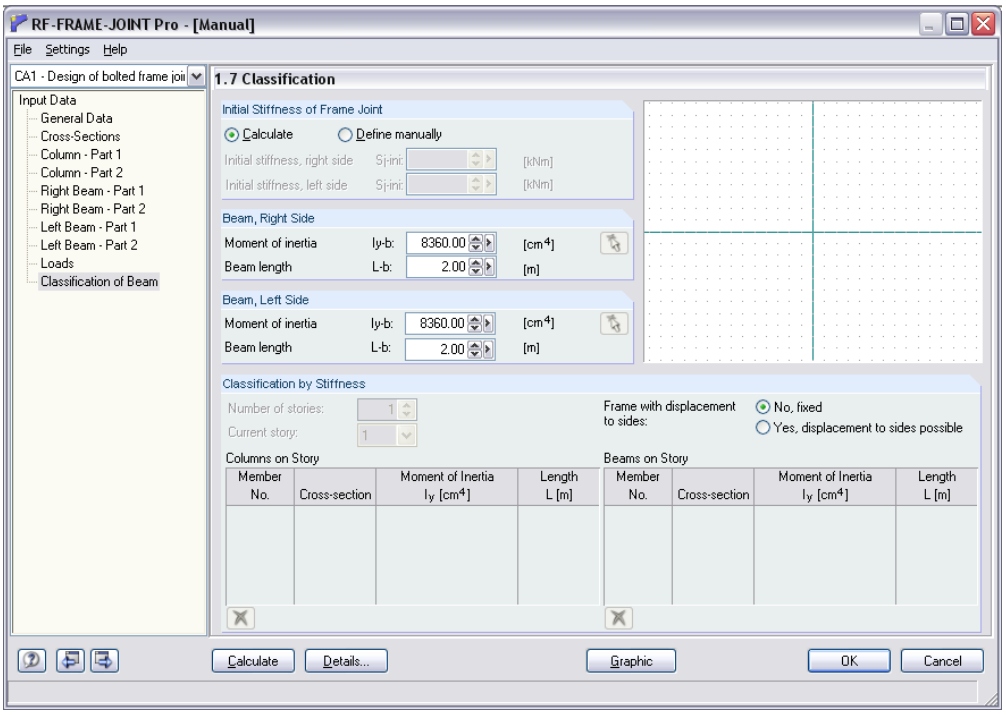


Figure 2.29: Table 1.7 *Classification*

Initial stiffness of frame joint

Initial stiffness *Sj-ini* can be either calculated by the program or defined manually in the text boxes.

Beam, Right Side / Beam, Left Side

You can define the beam moment of inertia *Iy,b* graphically by clicking the [Pick] button.

Classification by Stiffness

If a *Frame with displacement to sides* is defined, the *Column on story* or *Beams on story* tables are available. Then you can specify how many stories exist and in which story is the classified connection.

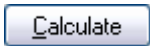
By clicking the [Select] button you can select in the graphic window the appropriate columns and beam on story.

2.8.2 Case 3: *Manual definition* and *Check*

Beam, Right Side / Beam, Left Side

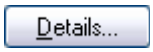
Moment of inertia *Iy,b* of the right and/or the left beam must be defined manually.

3. Calculation



The checks use the manually defined internal forces or the internal forces imported from RSTAB/RFEM. Before you start the calculation, first check the design details.

3.1 Details



The [Details] button is available in all input tables. Using this you can open the dialog box *Details*.

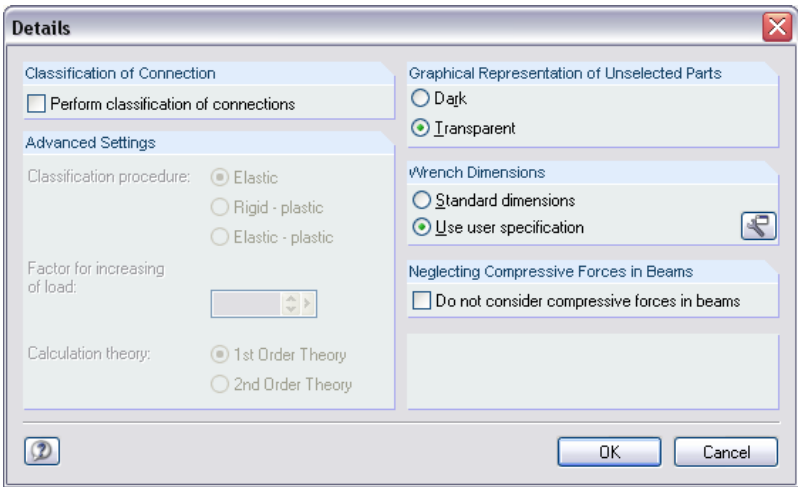


Figure 3.1: Dialog box *Details*

Classification of Connection

It is possible to *Perform classification of connections* (see Chapter 2.8, page 26). By selecting this check box, the ***Advanced Settings*** section below is available.

Graphical Representation of Unselected Parts

The objects, which are not selected in the graphic window, can be displayed as *Dark* or *Transparent*.

Wrench Dimensions

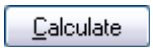
FRAME-JOINT Pro checks whether the dimensions of the selected wrench enable the assembly. You can select *Standard dimensions* or *Use user specification*.

Using the [Wrench dimensions] button you can open the *Wrench dimensions* table. The bolts selected in Table 1.4.2 are highlighted. To guarantee an easy assembly, the table also shows dimensions of a common socket wrench D · L and a torque wrench b · h. You can edit these values. The bolt spacing is determined by the boundary conditions.

Neglecting Compressive Forces in Beams

A compressive force in the beam affects the design of the end plate and bolts. You can reduce this force to zero by the *Do not consider compressive force in beams* option. The moment about the compression point is then not reduced.

3.2 Start Calculation



To start the calculation, click the [Calculate] button that is available in all input tables of the FRAME-JOINT Pro add-on module.

If the *Import from RSTAB/RFEM* option is selected in Table 1.1 *General Data*, the FRAME-JOINT Pro module searches for results of the selected load cases, combinations, and groups. If the results do not exist, the module starts calculation in the main program to determine the appropriate internal forces. In that case the calculation parameters defined in RSTAB/RFEM are used.

You can also start the calculation of the FRAME-JOINT Pro results from the main RSTAB/RFEM interface: The add-on modules are listed in the *To Calculate* dialog box as a load case or load group. To start this dialog box in RSTAB/RFEM, click in the main menu

Calculate → **To Calculate**.

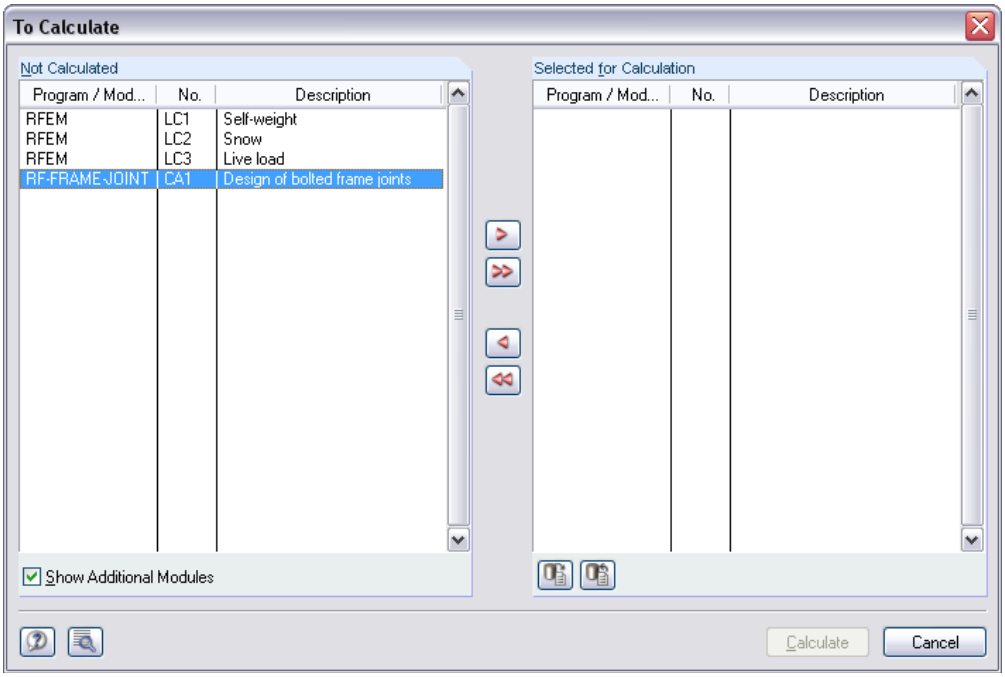


Figure 3.2: Dialog box *To Calculate*

If the FRAME-JOINT Pro design cases are missing in the *Not Calculated* list, select the check box *Show Additional Modules* below the list.

The [▶] button moves the list of the selected FRAME-JOINT Pro cases to the right. Then start the calculation by clicking the corresponding button.

You can also use the list on the toolbar above and directly specify a design case to calculate: select the FRAME-JOINT Pro case and click the [Results on/off] button.

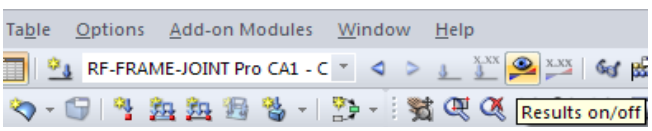


Figure 3.3: Direct calculation of a FRAME-JOINT Pro design case in RSTAB/RFEM

4. Results

Immediately after the calculation, the 2.1 *Geometry Summary* table is displayed. Table 2.2 to Table 2.4 shows checks together with explanations sorted according to components. In Table 2.5 you can view the checked connection.

You can select each result table directly from the FRAME-JOINT Pro navigator. You can also use the buttons displayed in the left margin or the [F2] and [F3] function keys to select the previous or next table.

Save the results by the [OK] button. The FRAME-JOINT Pro module will be closed and the RSTAB/RFEM work window will be available.

This chapter introduces the result tables one by one.

4.1 Geometry Summary

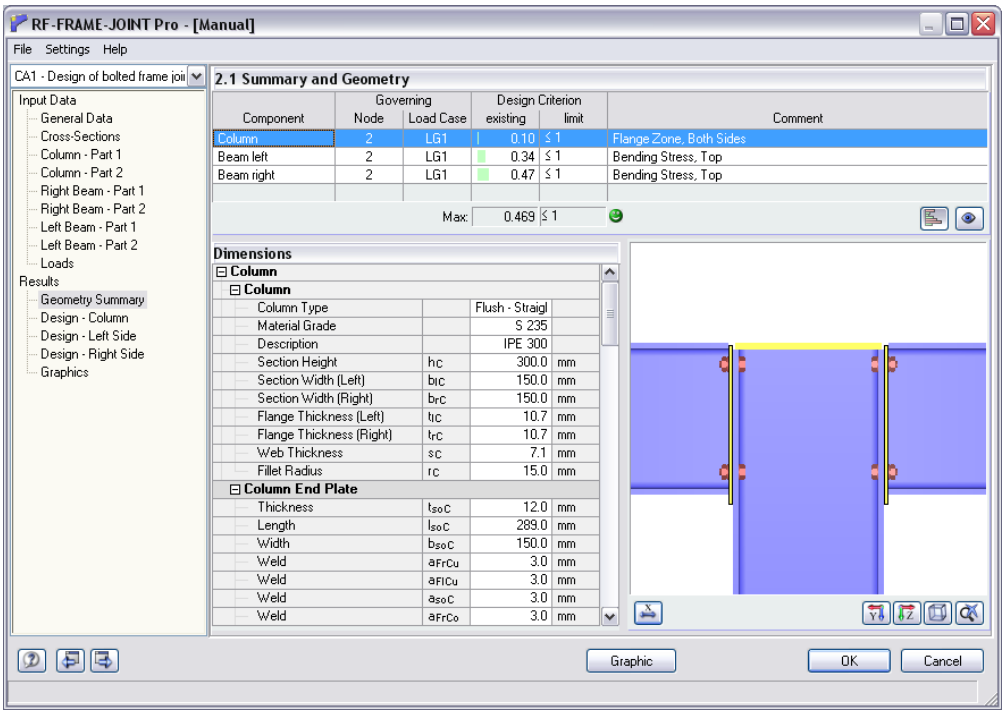


Figure 4.1: Table 2.1 *Geometry Summary*

Component

In this table, the results for all components of the checked connection are listed. For each component, the maximum design criteria calculated for the internal forces from the load cases, groups, and combinations, or those defined manually are displayed.

Nodes

For each checked component, the number of the designed node is stated.

Load Case

In this column, you will find the number of the load case, group, or combination, for which the internal forces cause the maximum design criterion.

Design Criterion - Existing / Limit

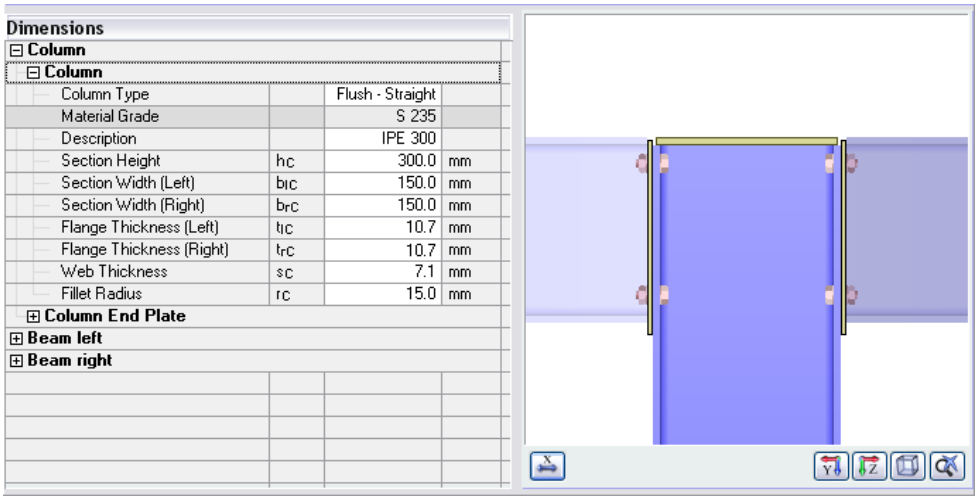
The result of the check is expressed in the form of the stress ratio. If the check is satisfied, the value of the result must be less than or equal to 1.00.

The values in the *Existing* column have a color bar in the background. The length of the bar represents the design ratio of the component. Furthermore, the green bar represents a satisfied check and the red bar shows an exceeded check. By clicking the button displayed in the left margin, the color bars can be turned on or off.

Dimensions

In the table below, you find the calculated components with their properties. It includes the dimensions of columns, beams, end plates, bolts, and welds for particular locations in the connection.

Furthermore, you find there the used materials and cross-sections.



Dimensions			
⊟ Column			
⊟ Column			
Column Type		Flush - Straight	
Material Grade		S 235	
Description		IPE 300	
Section Height	h_c	300.0	mm
Section Width (Left)	b_{lc}	150.0	mm
Section Width (Right)	b_{rc}	150.0	mm
Flange Thickness (Left)	t_{lc}	10.7	mm
Flange Thickness (Right)	t_{rc}	10.7	mm
Web Thickness	s_c	7.1	mm
Fillet Radius	r_c	15.0	mm
⊞ Column End Plate			
⊞ Beam left			
⊞ Beam right			

Figure 4.2: Section *Dimensions*

By clicking the [+] and [-] buttons you turn on and off the additional information for the particular component.

According to the cursor location in the table, the unselected components are displayed transparently or dark in the graphic window. For the settings, see the *Details* dialog box (Figure 3.1, page 27).

4.2 Design of Column

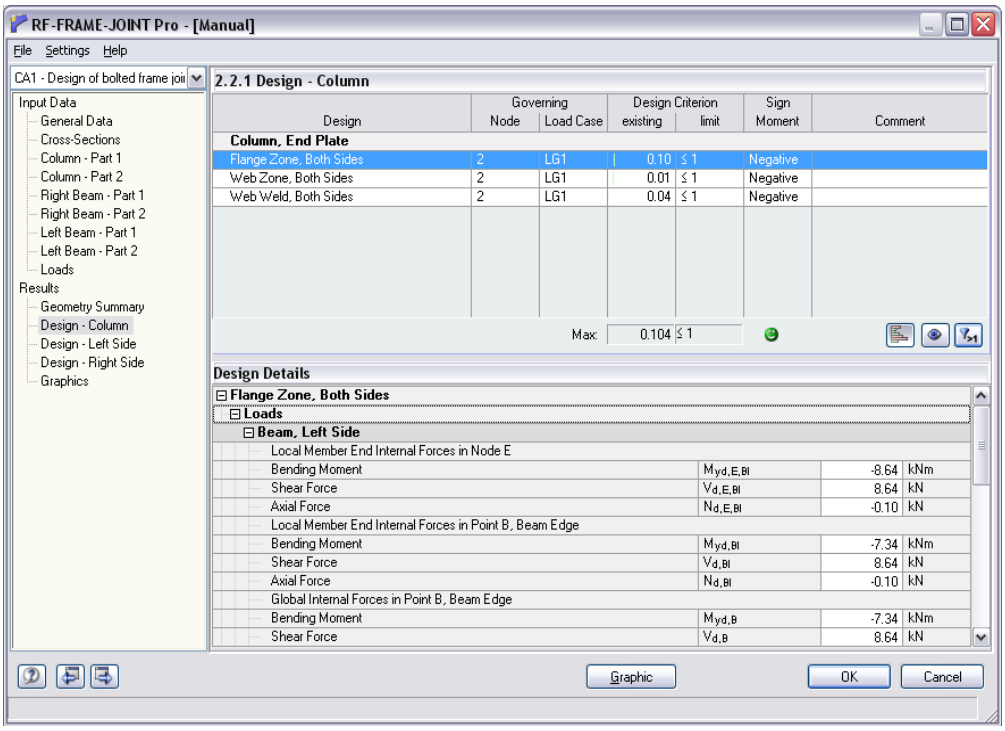


Figure 4.3: Table 2.2.1 *Design of Column*

This table presents the maximum design ratios of the column sorted by zones and design types.

The buttons in the upper section have the following functions:

Button	Description	Function
	View mode	Jumps to the RSTAB/RFEM work window to change the view
	Show Color Bars	Turns on and off the colored reference scales in the results tables
	Exceeding	Shows only the lines where design ratios are greater than 1 and therefore the check is not satisfied

Table 4.1: Buttons in result Table 2.2

Design Details

This table displays detailed results for the zone selected in the table above (i.e. the zone where the cursor is located). These are, for example, loads related to the system node and the member end node or internal forces defined in local or global coordinate system.

The internal forces imported from RSTAB/RFEM or defined manually are converted to the member end point of the beam using the existing angle and are presented as local and global member end internal forces at the beam interface.

The design details also give information about the existing stresses and related limit stresses.

4.3 Design Left Side

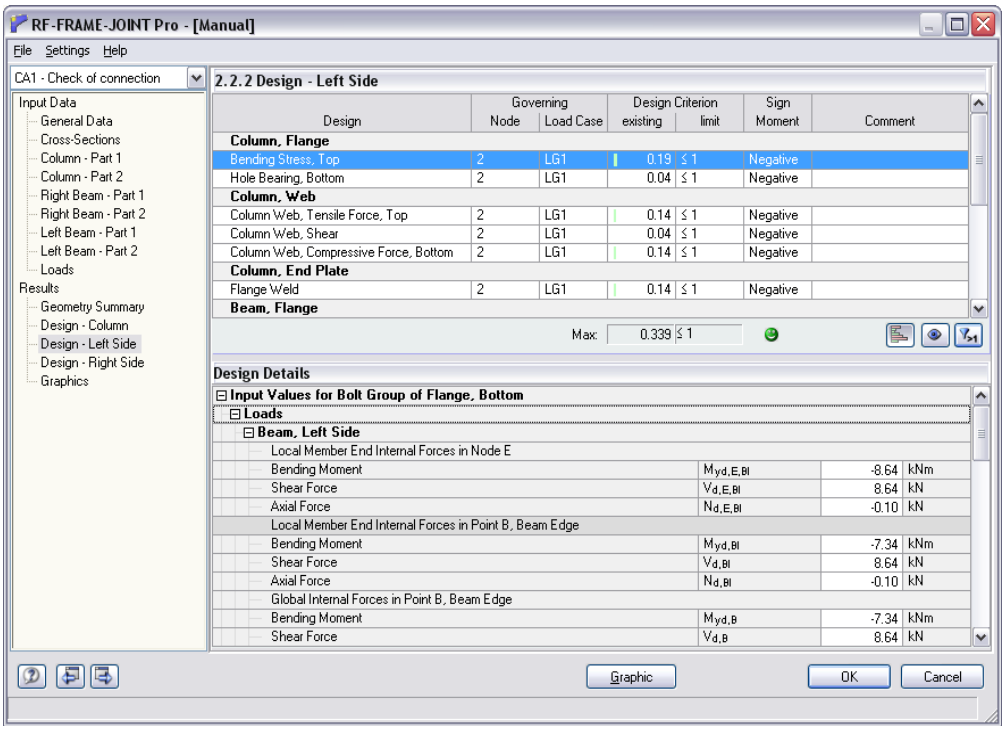


Figure 4.4: Table 2.2.2 *Design Left Side*

Table 2.2.2 *Design Left Side* presents various checks for the web and flange of the column and beam located on the left side of the connection. The individual zones are listed one by one.

The lower table displays *Design details* for the zone selected in the table above (i.e. the zone where the cursor is located). These are, for example, loads related to the system node E and internal forces related to the member end node, settings for components and bolt rows, as well as geometrical results relevant for the calculation.

4.4 Design Right Side

Table 2.2.3 *Design Right Side* is based on the same rules as the previous result table. It contains the results and *Design details* for the right side of the connection.

4.5 Graphics

Visualization of the component in Table 2.4 *Graphic* enables the graphical analysis of the calculated connection.

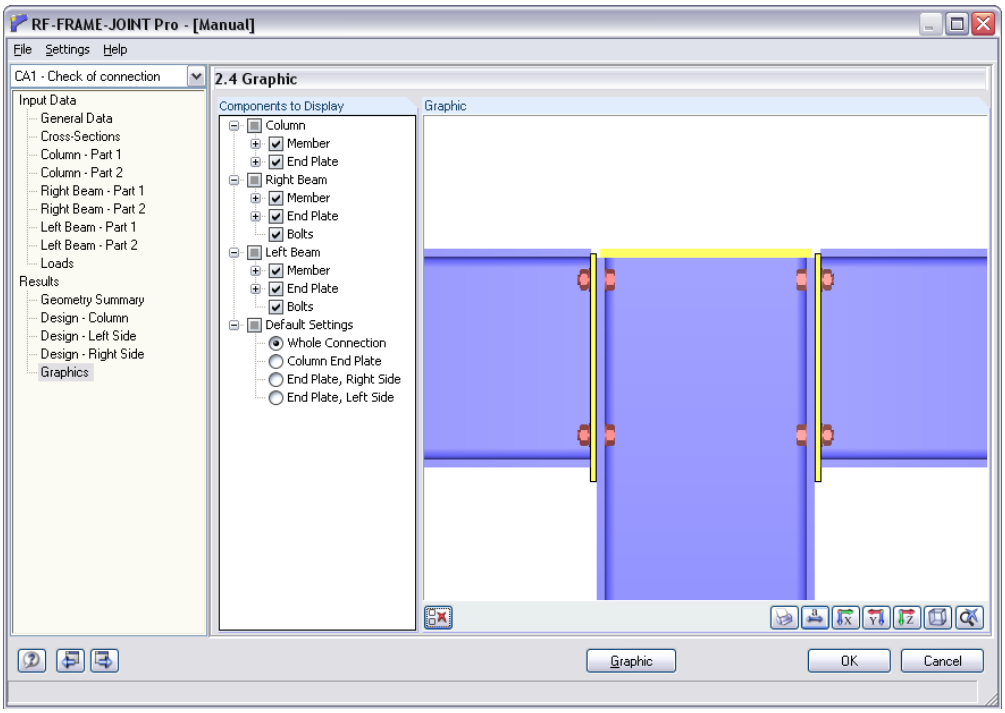


Figure 4.5: Table 2.4 *Graphic*

By clicking the [+] and [-] buttons you turn on and off the additional information for the particular component.

You can turn off all selected options by clicking the [Deselect All] button. A new selection can be done again.

Use the [Print] button to print the current graphic directly or to integrate it into the printout report.

Using the [Show All Graphic] button, you can set the current view to the center of the graphic window.

Standard settings

There are various default options for how to display the connection in the *Components to Display* section. This simplifies displaying particular connection components (e.g. only left end plate with end plate dimensions, bolt distances, and weld thicknesses).

5. Printout

5.1 Printout Report

You can generate a printout report for the FRAME-JOINT Pro data like in RSTAB/RFEM and add to it graphics and explanations. You can define which input and output tables will be displayed in the printout.

For detailed information about the printout report see the RSTAB/RFEM manual. Especially Chapters 10.1.3.4 and 11.1.3.4 respectively, *Selecting data of add-on modules,* deal with the selection of input and output data of the modules.

For complex structures with a high number of design cases, it is recommended to split the data into several small printout reports which allows a clearly-arranged printout and faster work.

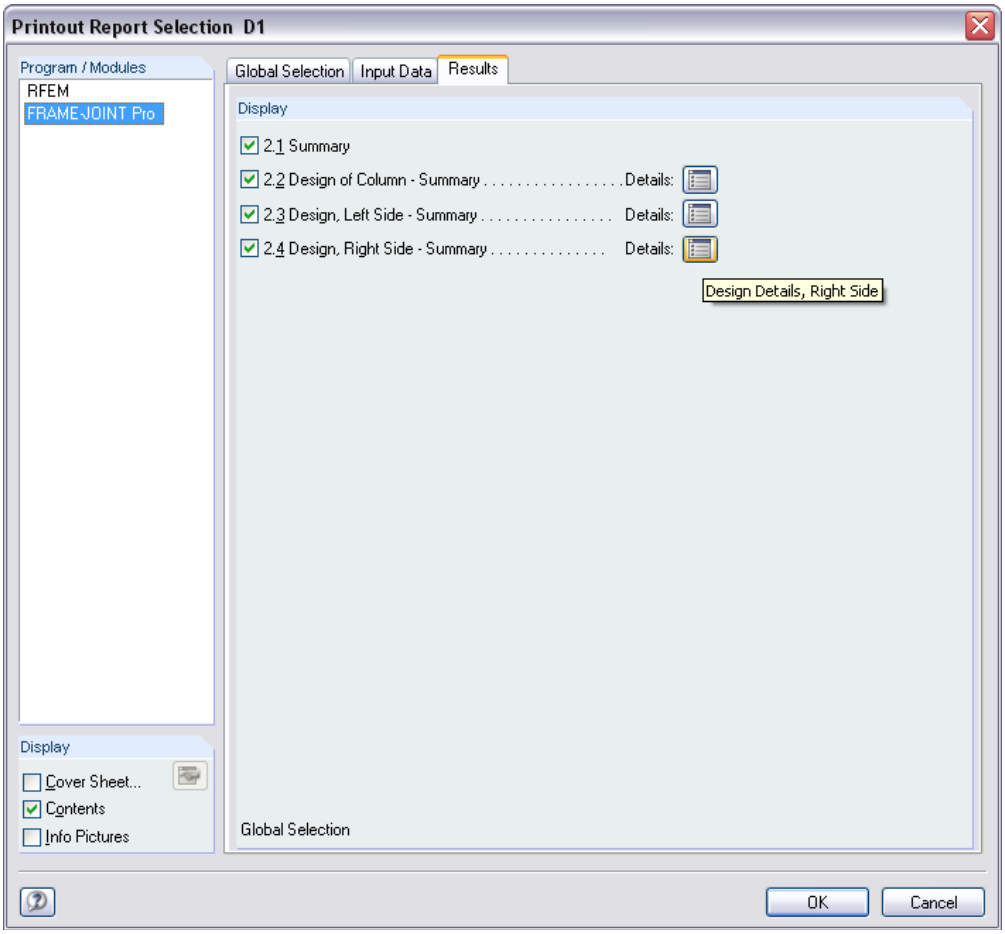


Figure 5.1: Printout report selection *Results*

In the *Result* tab of the printout report selection for FRAME-JOINT Pro, you can select in detail design types to display. Using the [Design details] button you can open a dialog box where you can specify detailed selections (see the following figure).

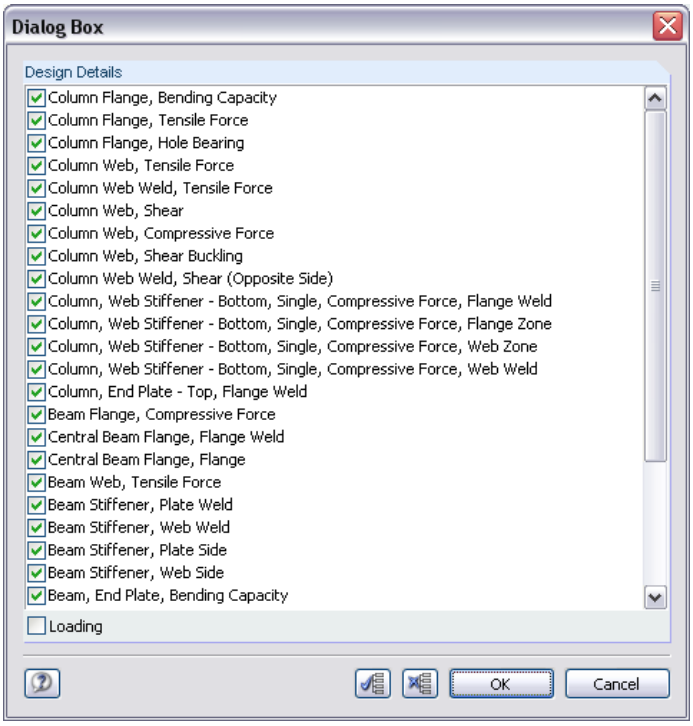


Figure 5.2: Selection of individual designs

5.2 Graphic Printout

You can either include the graphics of the frame joint (see Figure 4.5, page 33) in the printout report or you can send them directly to a printer. For a detailed explanation of the graphic printout see the RSTAB/RFEM manual, Chapter 10.2 and 11.2 respectively.

In the same way, you can export graphics from the work window of RSTAB or RFEM to the printout report. Using this feature, you can prepare a printout report containing the structure with displayed result diagrams.

To print the FRAME-JOINT Pro graphic that is currently displayed in the RSTAB/RFEM work window, select

File →Print

or use the toolbar button shown on the left.

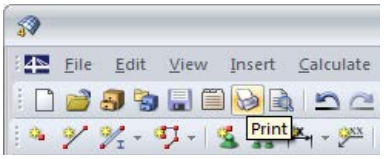


Figure 5.3: The button *Print* in the toolbar of the main window

The following dialog box opens:

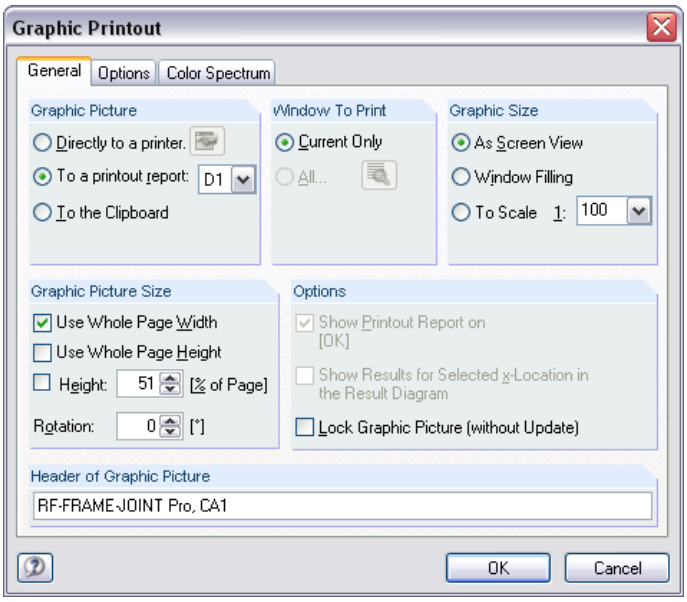
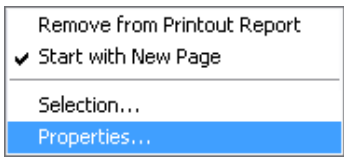


Figure 5.4: Dialog box *Graphic Printout*, tab *General*

Find further information in the RSTAB/RFEM manual, Chapter 10.2 and 11.2 respectively. The other tabs *Options* and *Color scale* are also described there.



You can move a FRAME-JOINT Pro graphic in the printout report as usual using the Drag & Drop function. You have also the possibility to edit the included figures additionally: right click the appropriate item in the printout report navigator and select the *Properties...* option. The refreshed *Graphic Printout* dialog box with various edit options appears.

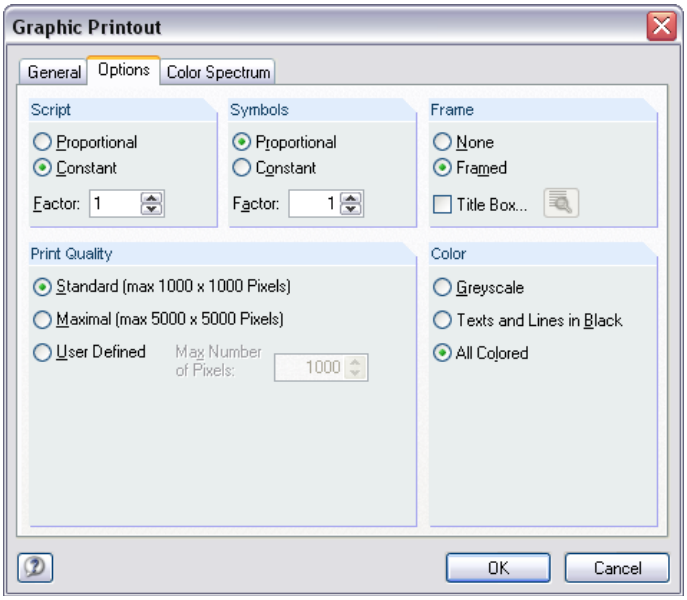


Figure 5.5: Dialog box *Graphic Printout*, tab *Options*

The following figure shows the graphic printout of an end plate. This graphic is displayed in Table 2.4 *Graphic* with the settings *End Plate, Right* (see Chapter 4.5, page 33).

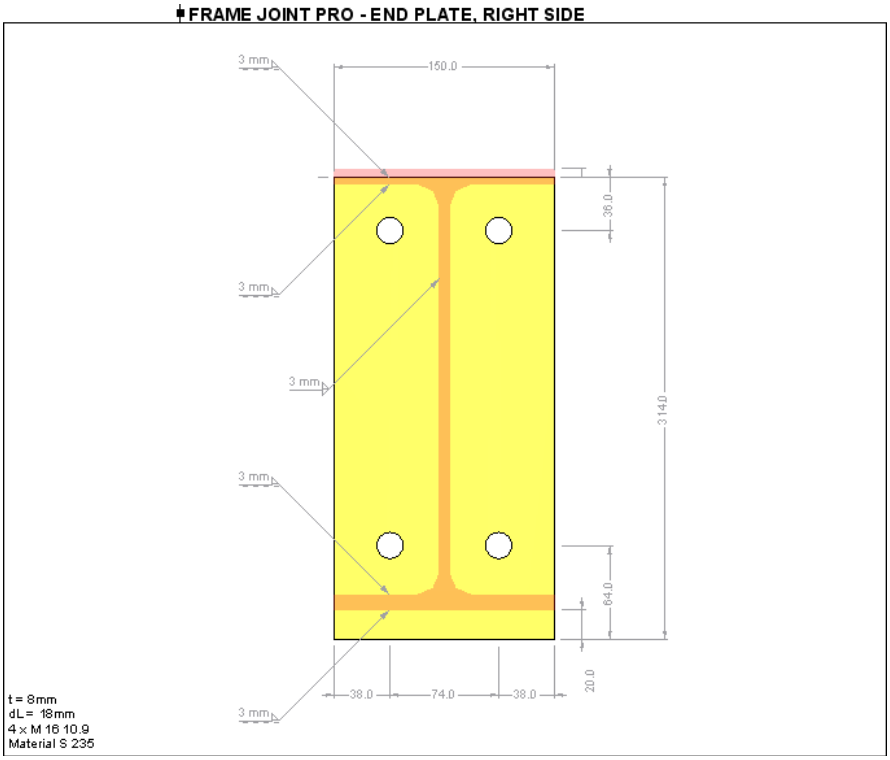


Figure 5.6: Graphic printout in the printout report

Using the [Dimensioning with symbols] button in Table 2.4, you can control whether the dimensions will be displayed as symbols or as numbers.

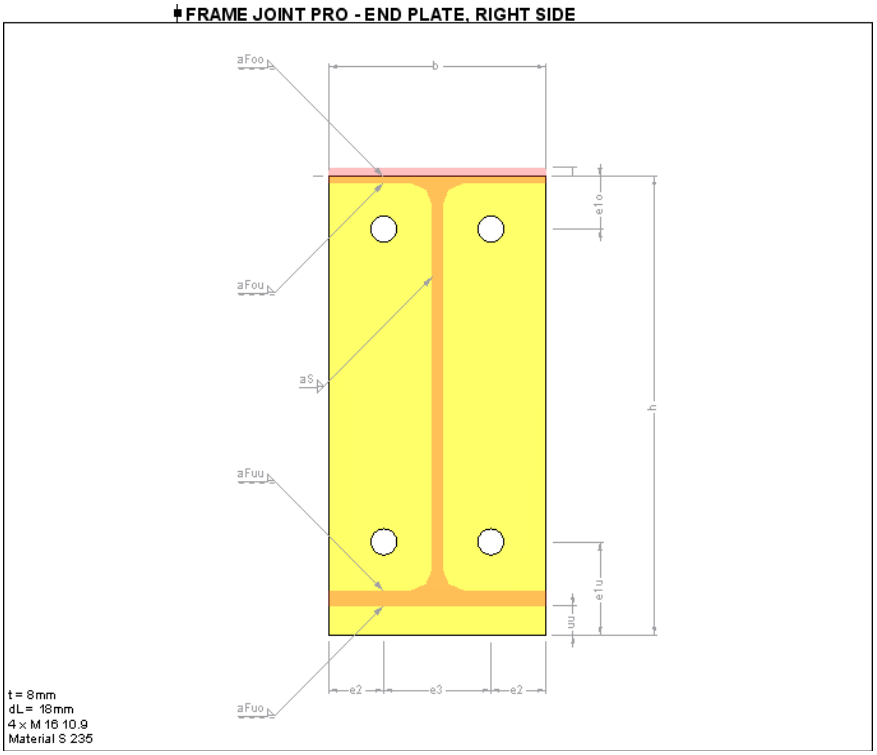
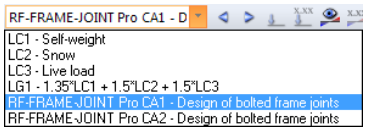


Figure 5.7: Graphic printout with symbols

6. General Functions

This chapter describes some menu functions as well as export options for the design results.

6.1 FRAME-JOINT Pro Design Cases



Nodes can be grouped to individual design cases. In this way, you can summarize the connection or divide the description according to specific frame joint types.

The FRAME-JOINT Pro design cases are available in the RSTAB/RFEM work window and can be displayed like a load case or load group by means of the toolbar list.

Create a new FRAME-JOINT Pro case

To create a new design case, select from the FRAME-JOINT Pro main menu

File →New Case.

The following dialog box appears.

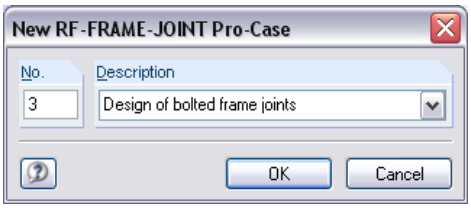


Figure 6.1: Dialog box *New FRAME-JOINT Pro-Case*

In this dialog box, you can define a (not yet used) *Number* and a *Description* for a new design case. Clicking [OK], the FRAME-JOINT Pro Table 1.1 *General Data* appears and you can define new design data.

Rename a FRAME-JOINT Pro case

To change the description of a design case subsequently, select from the FRAME-JOINT Pro main menu

File →Rename Case.

The dialog box *Rename FRAME-JOINT Pro-Case* appears.

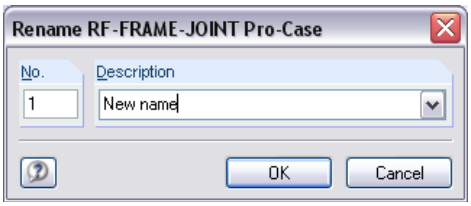


Figure 6.2: Dialog box *Rename a FRAME-JOINT Pro-Case*

Copy a FRAME-JOINT Pro case

To copy the input data of the current design case, select from the FRAME-JOINT Pro main menu

File →Copy Case.

The dialog box *Copy FRAME-JOINT Pro-Case* appears. You can specify the number and description of the new case.

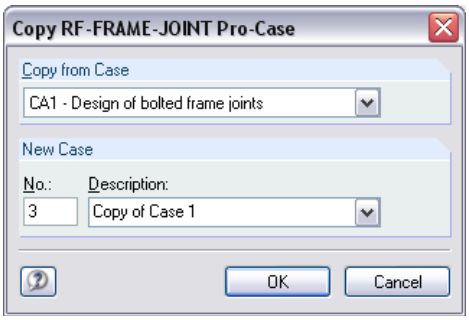


Figure 6.3: Dialog box *Copy FRAME-JOINT Pro-Case*

Delete a FRAME-JOINT Pro case

To delete design cases, select from the FRAME-JOINT Pro main menu

File →Delete Case.

In the dialog box *Delete Cases*, you can select the relevant design case in the *Available Cases* list to delete by clicking [OK].

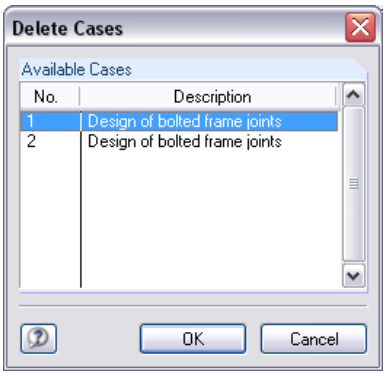


Figure 6.4: Dialog box *Delete Case*

6.2 Units and Decimal Places

You can manage the *Units and decimal places* centrally in RSTAB/RFEM for all add-on modules. In FRAME-JOINT Pro, you can open the dialog box for unit settings by selecting in the menu

Settings → Units and Decimal Places.

The dialog box already familiar from RSTAB/RFEM appears. The FRAME-JOINT Pro module is preset.

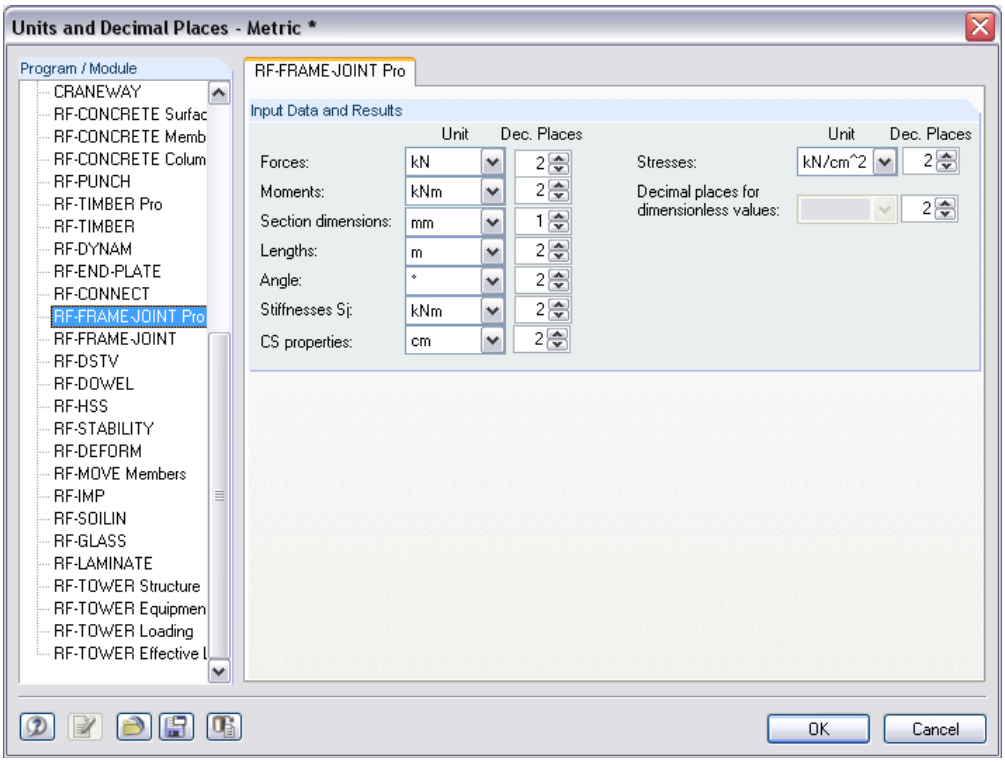


Figure 6.5: Dialog box *Units and Decimal Places*

You can save the settings using the [Save as Profile] button to use again for a different model. Find further information about this function in the RSTAB/RFEM manual, Chapter 11.6.2 and 12.6.2 respectively.

6.3 Exporting Results

The results from FRAME-JOINT Pro can also be used in other programs in many ways.

Clipboard

Selected cells from the FRAME-JOINT Pro result tables can be copied to the Clipboard using [Ctrl]+[C] and inserted by [Ctrl]+[V] to word-processing programs, for example. In this case, the headers of the table columns are not copied.

Printout Report

The data of the FRAME-JOINT Pro module can be printed into the global printout report (see Chapter 5.1, page 34) to export them subsequently. Then, in the printout report, select

File→ Export to RTF File or BauText.

This function is described in the RSTAB/RFEM manual, Chapter 10.1.11 and 11.1.11 respectively.

Excel / OpenOffice

FRAME-JOINT Pro allows a direct data export to MS Excel, OpenOffice.org Calc or CSV-format. To use this function, click in the main menu

File → Export tables....

The following export dialog box appears.

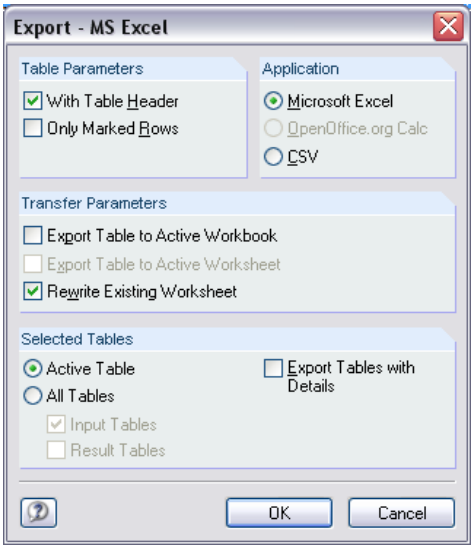


Figure 6.6: Dialog box *Export - MS Excel*

After you select the required parameters, you can run the export by clicking [OK]. Excel or OpenOffice starts automatically. The program does not need to be open in the background.

A3 | Column, End Plate

	A	B	C	D	E	F	G
1		Governing		Design Criterion		Sign	
2	Design	Node	Load Case	existing	limit	Moment	Comment
3	Column, End Plate						
4	Flange Zone, Both Sides	2	LG1	0.10	≤ 1	Negative	
5	Web Zone, Both Sides	2	LG1	0.01	≤ 1	Negative	
6	Web Weld, Both Sides	2	LG1	0.04	≤ 1	Negative	

Figure 6.7: Result in *Excel*

7. Examples

7.1 Comparative Calculation According to the DSTV Handbook

7.1.1 Case 1: Bending and Full Shear Force

To verify the results of the FRAME-JOINT Pro module, we will calculate the typified connection from the DSTV handbook: **IH 3.1, M16, HE 260 A, No. 443**

Frame joint type:	Continuous column
Input data:	Manual definition
Calculation mode:	Check with specification of all dimensions
Column cross-section:	HE360A, S 235
Beam cross-section:	HE260A, S 235
End plate and bolt dimension:	according to the DSTV handbook

For 100% utilization of the connection, the following internal forces are defined:

N_{Ed} = **0.00 kN**

$M_{y,1,Rd}$ = **-94.5 kNm**

$V_{z,Rd}$ = **195.10 kN**

Figure 7.1: Manual definition of the loads in Table 1.6

Results

As the governing check, the FRAME-JOINT Pro determines the **Design of bolts for shear**.

2.1 Summary and Geometry

Component	Governing Node	Governing Load Case	Design Criterion existing	Design Criterion limit	Comment
Column	1	LC 1	0.41	≤ 1	Flange Zone, Both Sides, Top
Beam left	1	LC 1	1.01	> 1	Bolt, Shear

Figure 7.2: Check of bending and full shear force

Comparison

DSTV:	**100 %**
FRAME-JOINT Pro:	**101 %**

7.1.2 Case 2: Pure Bending

Frame joint type:	Continuous column
Input data:	Manual definition
Calculation mode:	Check with specification of all dimensions
Column cross-section:	HE360A, S 235
Beam cross-section:	HE260A, S 235
End plate and bolt dimension:	according to the DSTV handbook

For 100% utilization of the connection, the following internal forces are defined:

N_{Ed} = **0.00 kN**

$M_{y,1,Rd}$ = **-94.5 kNm**

$V_{z,Rd}$ = **0.00 kN**

Figure 7.3: Manual definition of the loads in Table 1.6

Results

As the governing check, the FRAME-JOINT Pro determines the **Design of bending capacity.**

2.1 Summary and Geometry

Component	Governing Node	Governing Load Case	Design Criterion existing	Design Criterion limit	Comment
Column	1	LC 1	0.65	≤ 1	Flange Zone, Both Sides, Top
Beam left	1	LC 1	0.93	≤ 1	Bending Stress, Top

Figure 7.4: Check of pure bending

Comparison

DSTV:	**100 %**
FRAME-JOINT Pro:	**93 %**

7.2 Example of Classification

System figure

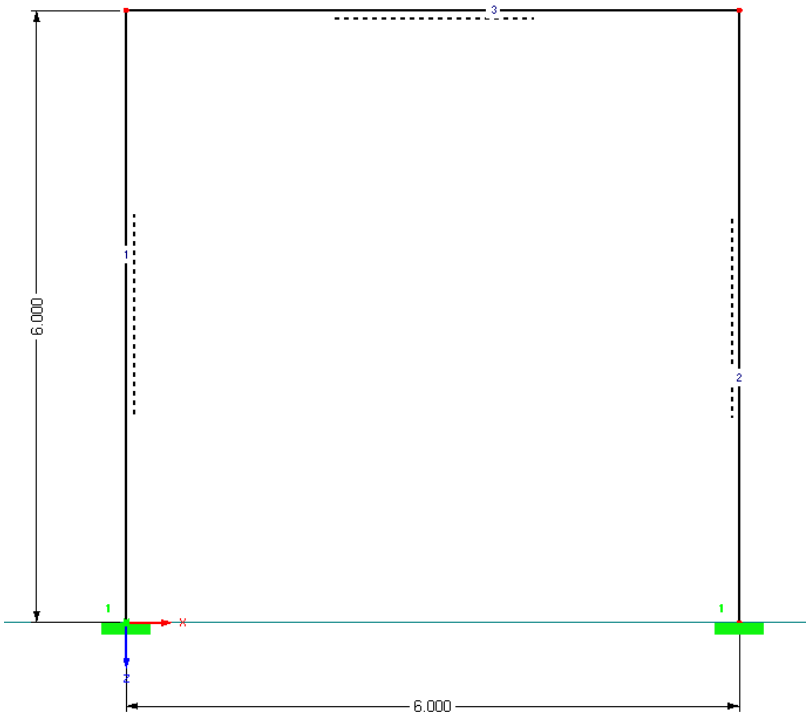


Figure 7.5: System figure: Example of classification

Cross-sections and materials

Beam: **IPE 300, S 235**

Columns: **IPE 300, S 235**

Frame spacing: **l = 5.00 m**

Actions

Load cases

Load Case 1: Self-weight $g_A = 0.40$ kN/m²

Load Case 2: Wind in +X $q_w = 0.65$ kN/m²

Load Case 3: Snow $q_s = 0.75$ kN/m²

Load groups

LG 1:1,35*LC1

LG 2:1,35*LC1+1,50*LC2

LG 3: 1,35*LC1+1,50*LC2+0,75*LC3

LG 4: 1,35*LC1+1,50*LC3

LG 5:1,35*LC1+0,9*LC2+1,50*LC3

Load combination

CO 1:LG1/s or LG2/s or LG3/s or LG4/s or LG5/s

End plate dimensions (according to design by FRAME-JOINT Pro)

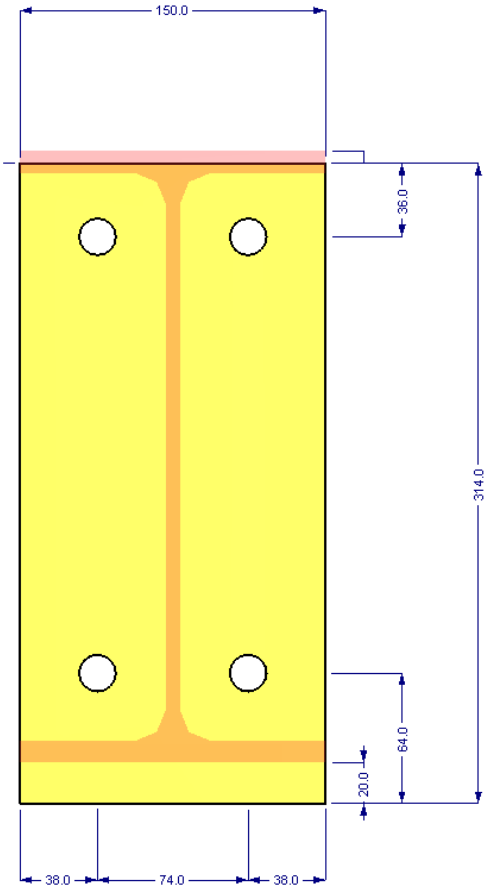


Figure 7.6: End plate dimensions

Initial stiffness $S_{j,ini}$

$$S_{j,ini} = \frac{E \cdot z^2}{\mu \cdot \sum \frac{1}{k_i}}$$ [2], Chap. 6.3, Eq. 6.27

1. Coefficient for column web plate, shear

$$k_1 = \frac{0.38 \cdot A_{v,c}}{z \cdot \beta} = \underline{\underline{0.386 \text{ cm}}}$$ [2], Chap. 6.3.2, Tab. 6.11

with $A_{v,c}$ acc. to [1], Chap. 6.2.6
z acc. to [2], Chap.6.2.7, Figure 6.15 a.)
β acc. to [2], Chap. 5.3, (7)

2. Coefficient for column web plate, compression

$$k_2 = \frac{0.7 \cdot b_{eff,c,wc} \cdot t_{w,c}}{d_c} = \underline{\underline{0.310 \text{ cm}}}$$ [2], Chap. 6.3.2, Tab. 6.11

with $b_{eff,c,wc}$ acc. to [2], Chap.6.2.6.2
d_c web depth between the fillets

3. Coefficient for column web plate, tension

$$k_3 = \frac{0.7 \cdot b_{eff,t,wc} \cdot t_{w,c}}{d_c} = \underline{\underline{0.266 \text{ cm}}}$$ [2], Chap. 6.3.2, Tab. 6.11

with $b_{eff,t,wc}$ acc. to [2], Chap.6.2.6.3
d_c web depth between the fillets

4. Coefficient for column flange, bending

$$k_4 = \frac{0.9 \cdot l_{eff} \cdot t_{fc}^{\,3}}{m^3} = \underline{\underline{1.506 \text{ cm}}}$$ [2], Chap. 6.3.2, Tab. 6.11

with l_{eff} acc. to [2], Tab.6.4
m acc. to [2], Figure 6.8
d_c web depth between the fillets

5. Coefficient for beam end plate, bending

$$k_5 = \frac{0.9 \cdot l_{eff} \cdot t_p^{\,3}}{m^3} = \underline{\underline{0.315 \text{ cm}}}$$ [2], Chap. 6.3.2, Tab. 6.11

with l_{eff} acc. to [2], Tab.6.6
m acc. to [2], Figure 6.11 and Figure 6.10

6. Coefficient for bolts, tension

$$k_{10} = 1.6 \cdot \frac{A_s}{L_b} = \underline{\underline{0.658 \text{ cm}}}$$ [2], Chap. 6.3.2, Tab. 6.11

with A_s shear surface of bolts
L_b linear strain of bolts acc. to [2], Tab. 6.11

7. Sum of the stiffness coefficients

$$\sum \frac{1}{k_i} = \frac{1}{k_1} + \frac{1}{k_2} + \frac{1}{k_3} + \frac{1}{k_4} + \frac{1}{k_5} + \frac{1}{k_{10}}$$

$$\sum \frac{1}{k_i} = \frac{1}{0.386} + \frac{1}{0.310} + \frac{1}{0.266} + \frac{1}{1.506} + \frac{1}{0.315} + \frac{1}{0.658} = \underline{\underline{14.84 \text{ } 1/\text{cm}}}$$

8. Distance between the compression point and bolt row under tension

$$z = h_p - e_{1o} - \left(u_u + \frac{t_{f,c}}{2} \right) =$$

$$z = 31.4 - 3.6 - \left(2.0 + \frac{10.7}{2} \right) = \underline{\underline{25.27 \text{ cm}}}$$ [2], Chap. 6.2.7, Figure 6.15 c.)

9. Initial stiffness Sj,ini

$$S_{j,ini} = \frac{E \cdot z^2}{\mu \cdot \sum \frac{1}{k_i}}$$ [2], Chap. 6.3., Eq. 6.27

$$S_{j,ini} = \frac{21000 \cdot 25.27^2}{1.0 \cdot 14.84} = \underline{\underline{903640 \text{ kNcm} \Rightarrow 9036.40 \text{ kNm}}}$$

Limit internal forces

Zone 1: Stiffness rigid [2], Chap. 5.2.2.5.

$$S_{j,limit,rigid} \le \frac{25 \cdot E \cdot I_b}{L_b} = \frac{25 \cdot 21000 \cdot 8360}{600} * 10^{-2} = \underline{\underline{73150\ kNm}}$$

Zone 3: Stiffness hinged [2], Chap. 5.2.2.5.

$$S_{j,limit,hinged} \le \frac{E \cdot I_b}{L_b} = \frac{21000 \cdot 8360}{2 \cdot 600} * 10^{-2} = \underline{\underline{1463\ kNm}}$$

Classification

$$S_{j,limit,hinged} \le S_{j,ini} \le S_{j,ini,rigid}$$

$$1463\ kNm \le \underline{\underline{9036.40\ kNm}} \le 73150\ kNm$$

$$\Rightarrow \text{ Zone 2 } \text{semi}-\text{rigid} \text{ connection}$$

Results in FRAME-JOINT Pro

The following figures show the FRAME-JOINT Pro results with the limit stiffnesses and individual stiffness coefficients.

Negative Ultimate Moment [about Bottom Compression Point]	M_{Rd}	-18.65	kNm
Limiting Stiffness - rigid	S_{j,Lim_rigid}	73150.00	kNm
Limiting Stiffness - pinned	S_{j,Lim_pinned}	1463.00	kNm
Initial Stiffness	$S_{j,ini}$	9033.51	kNm
Applicable Rotational Stiffness	S_j	4516.76	kNm
Coefficient Column Web Plate, Shear	k_1	3.9	mm
Coefficient Column Web Plate, Compression	k_2	3.1	mm
Coefficient Column Web, Tension	k_3 (1)	2.7	mm
Coefficient Column Flange, Bending	k_4 (1)	15.1	mm
Coefficient Beam End Plate, Bending	k_5 (1)	3.2	mm
Coefficient Bolts, Tension	k_{10} (1)	6.6	mm
Effective Stiffness Coefficient	k_{eff} (1)	1.1	mm
Equivalent Lever	z_{eq}	252.7	mm
Equivalent Stiffness	k_{eq}	1.1	mm

Figure 7.7: Calculation of the FRAME-JOINT Pro results

Stiffness Class		Semi-Rigid	
Lateral Bracing	unbraced	Yes	
min. ratio of kb/kc	k_b/k_c min	1.00	

Figure 7.8: Assignment to the stiffness class

A Literature

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

[2] EN 1993-1-8: 2010, Eurocode 3: Design of steel structures - Part 1-8: Design of joints

[3] Program description RSTAB/RFEM, DLUBAL ENGINEERING SOFTWARE, Version July 2009

B Index