

StruSoft

Structural Design Software

FEM-Design Steel Joint User Manual

StruSoft AB

Visit the StruSoft website for company and FEM-Design information at
<http://www.strusoft.com>

User Manual for Steel Joint module

Copyright © 2016 by StruSoft, all rights reserved.

Trademarks

FEM-Design is a registered trademark of StruSoft.

All other trademarks are the property of their respective holders.

Legend



Pay attention / Note



Useful hint



Example



Clicking left mouse button



Clicking right mouse button



Clicking middle mouse button

Steel Joint module is a part of the FEM-Design software package and this user manual covers only the specific features of this module.

All the additional information is available in the *FEM-Design User Manual* that is accessible from the Steel Joint *Help* menu. We highly recommend reading the following chapters of *FEM-Design User Manual*:

BASIC CONCEPTS

- User interface

- Program settings

- Data Safety

- Element Types (Drawing elements)

- Coordinate Systems

- Point Definition with Co-ordinates

- Selections

- Navigation

- Views

- Display modes

- Hiding elements

- Transparency

SECTION EDITOR

DOCUMENTATION

- Documentation module

DRAWING AND EDITING TOOLS

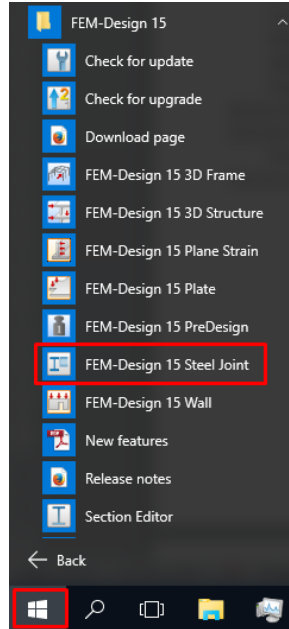
Table of contents

1.	Starting the program	6
2.	Introduction	8
3.	Steel joint	13
3.1.	User interface	13
	Navigation panel	13
	3D model of the joint	13
	Calculation window	14
	Collection	14
	Entering Documentation module	14
3.2.	Joint	14
3.3.	Data	15
	Basic data	15
	Load combinations	18
3.4.	Design	20
	Modify, add, remove components	20
	Bolts	20
	Change solution	23
3.5.	Calculation window	25
	Content	25
	Moving the calculation window	25
3.6.	Collection	26
3.7.	Documentation	28
	General	28
	Printing	28
	Export to .docx format	30
	Advanced documentation tools	31

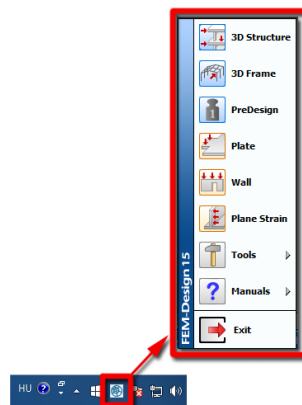
1. Starting the program

The Steel Joint module can be started in one of the following ways:

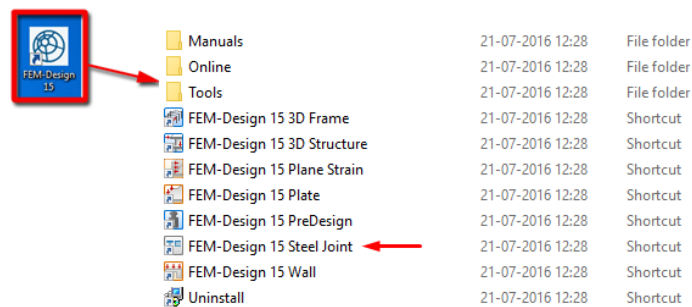
- Click *Start* button of Windows status bar and select *All programs* -> *FEM-Design...*



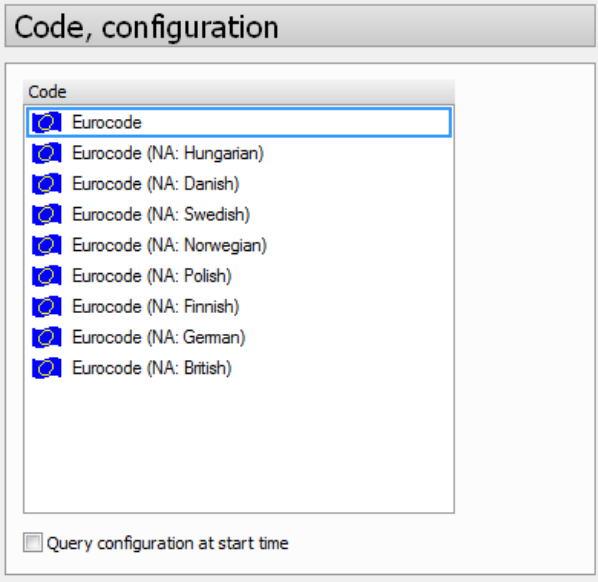
- Click *Windows Taskbar* -> *FEM-Design System tray* , if it is installed by default.



- Double click on the FEM-Design 15 shortcut, if it is placed on the Desktop.



When starting the Steel Joint module a *Code, configuration* dialog pops out. Here the user has to select a code for the calculation. The code selection influences the values of safety factors, as well as the range of available modelling materials.







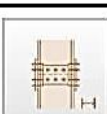
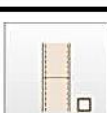

2. Introduction









The Steel Joint module covers 51 joints that are classified into 7 types:







- Column splice
- Beam splice
- Column base
- Beam to beam
- Beam to column
- Knee
- Bracing




Each type includes several solutions, where different bar cross-section or different components are used, (end plate, splice plate, weld, bolt...). Solutions may include additional options that can be chosen for different number of joined bars or different bar topology (e.g. Y, X, K bracing joints).

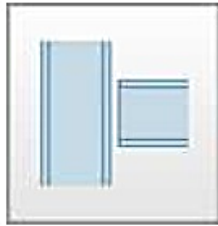










The following table shows all the possible joints.
















Type	Solution	Options
 <p style="text-align: center;">Column splice</p>	 <p style="text-align: center;">CS1</p>	
	 <p style="text-align: center;">CS2</p>	
	 <p style="text-align: center;">CS3</p>	
	 <p style="text-align: center;">CS4</p>	
	 <p style="text-align: center;">CS5</p>	
	 <p style="text-align: center;">CS6</p>	

















Type	Solution	Options
 <p style="text-align: center;">Beam splice</p>	 <p style="text-align: center;">BS1</p>	
	 <p style="text-align: center;">BS2</p>	
	 <p style="text-align: center;">BS3</p>	
	 <p style="text-align: center;">BS4</p>	
	 <p style="text-align: center;">BS5</p>	
	 <p style="text-align: center;">BS6</p>	
	 <p style="text-align: center;">BS7</p>	

Type	Solution	Options
 <p style="text-align: center;">Beam to beam</p>	 <p style="text-align: center;">BB1</p>	
	 <p style="text-align: center;">BB2</p>	
	 <p style="text-align: center;">BB3</p>	
	 <p style="text-align: center;">BB4</p>	
	 <p style="text-align: center;">BB5</p>	

Type	Solution	Options
 <p style="text-align: center;">Column base</p>	 <p style="text-align: center;">CB1</p>	
	 <p style="text-align: center;">CB2</p>	

Type	Solution	Options
 <p style="text-align: center;">Beam to column</p>	 <p style="text-align: center;">BC1</p>	
	 <p style="text-align: center;">BC2</p>	
	 <p style="text-align: center;">BC3</p>	
	 <p style="text-align: center;">BC4</p>	
	 <p style="text-align: center;">BC5</p>	 <p style="text-align: center;">A</p>
		 <p style="text-align: center;">B</p>
	 <p style="text-align: center;">BC6</p>	
	 <p style="text-align: center;">BC7</p>	
 <p style="text-align: center;">BC8</p>		

Type	Solution	Options	
		KN1	
		KN2	
		KN3	
		KN4	
		KN5	
		KN5	
		KN6	
		KN6	
	KN7		
	KN8		

Type	Solution	Options
		 Y
		 X
		 K
		 Y
		 X
		 K
		 Y
		 X
		 K
		
		
		

3. Steel joint

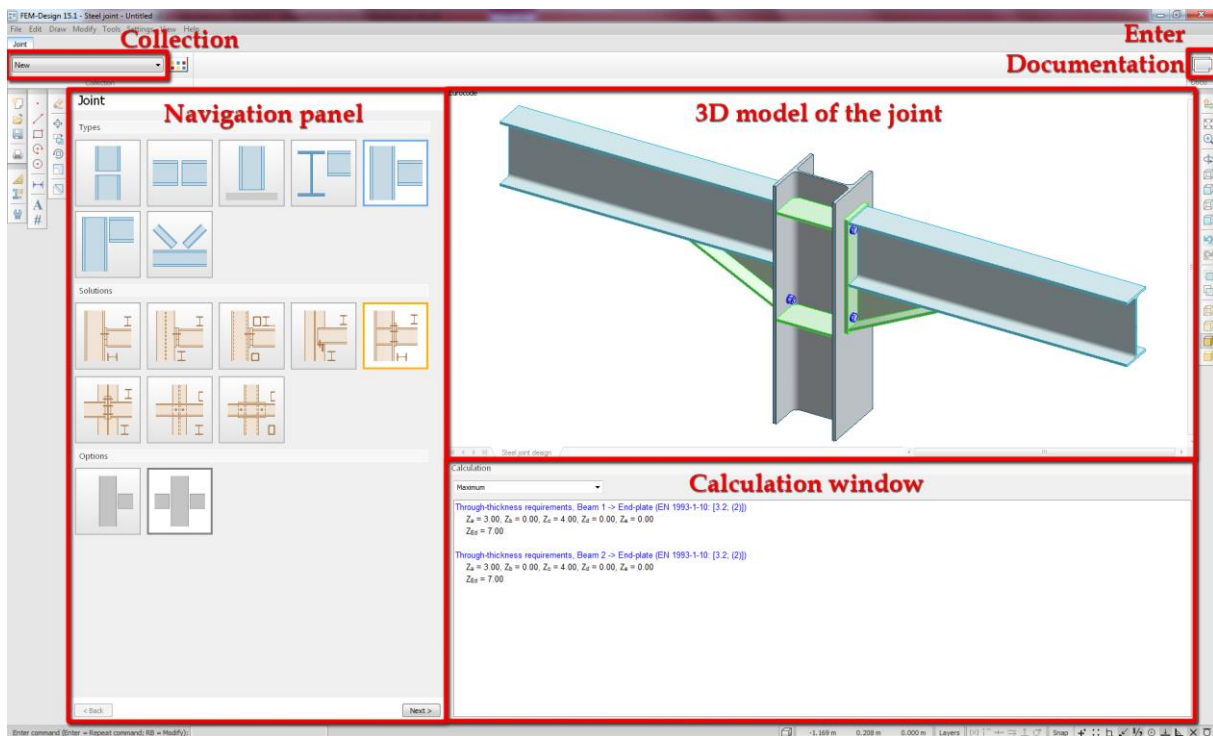
3.1. User interface

The user interface of the standalone module contains the following parts:

- Navigation panel
- 3D model of the joint
- Calculation window
- Collection
- Enter Documentation module



Toolbars, Menu bar and Status bar are the standard FEM-Design components that are located in all modules' interface (see FEM-Design User Manual, *Basic concept* -> *User interface* chapter).



Navigation panel

Navigation panel is located on the left side. All user input takes place on this panel.

3D model of the joint

The 3D model of the joint is located on the right side. It is updated in real time. When any joint's parameter is modified, the 3D model will be updated immediately.

Calculation window

Calculation window by default is placed in Navigation panel, in the Design panel. The user can however, change its position. This window displays calculations, errors and warnings.

Collection

The collection of the designed joints is located at the up-left corner. In this part of the interface the designed joints can be selected for modification.

Entering Documentation module

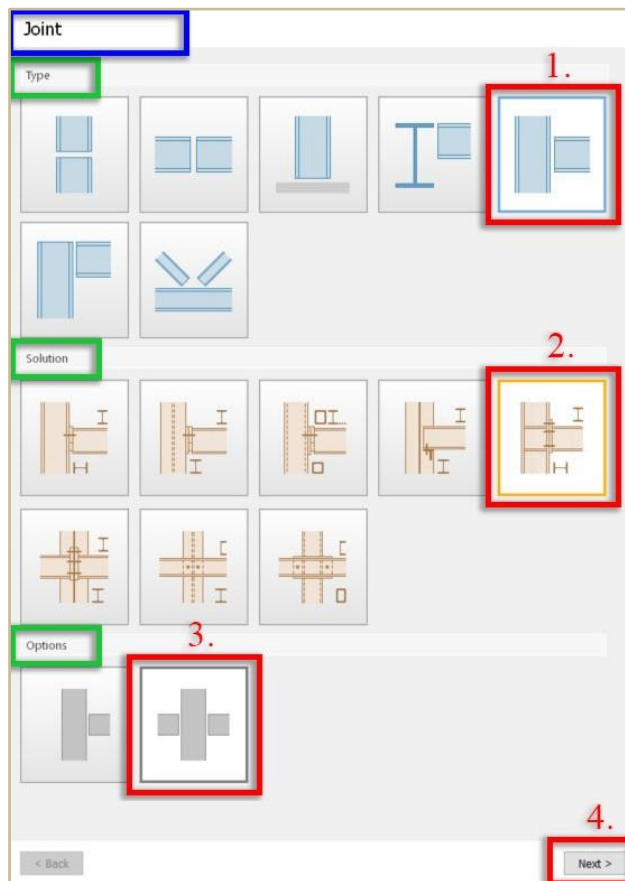
The button to enter the Documentation module is located on the top-right corner. For more information see Chapter 2.7.

3.2. Joint

Designing of a joint in the standalone module is performed in 3 main steps.

- Selecting a joint
- Setting data
- Designing the joint

First, the user has to choose a joint type (1), then a solution (2), and an option (3), if available. User can step forward to the *Data* panel by clicking on the *Next* button (4).

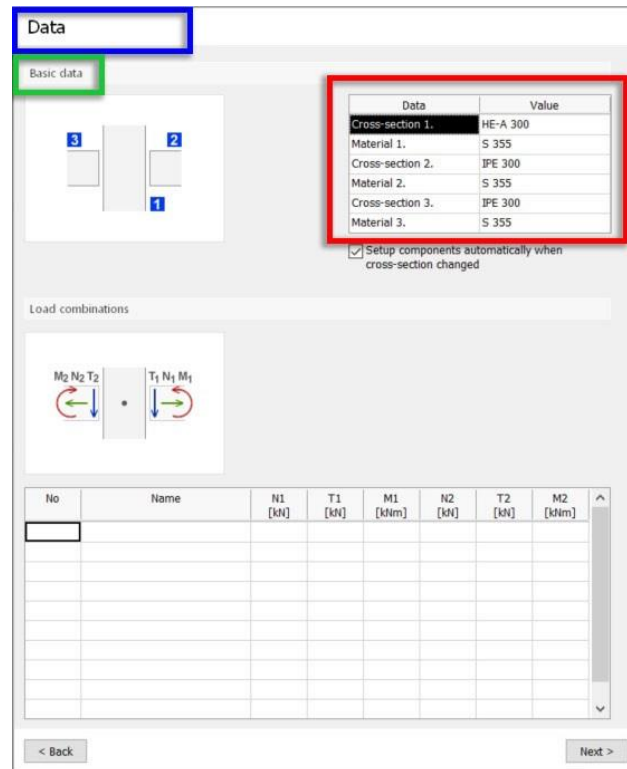


3.3. Data

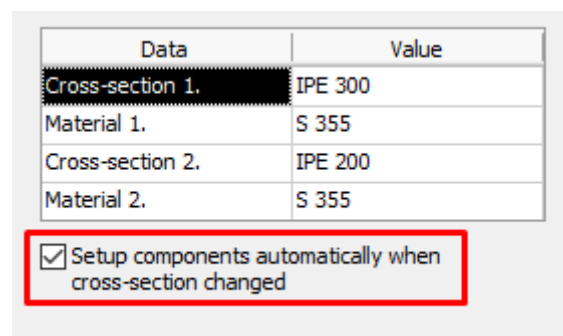
Basic data

In the up-right of the *Data* panel the following data of bars connected to a joint can be set:

- Cross-section(s)
- Material(s)
- Geometry (in some solutions)
- Buckling data (in case of column splices and column bases, more details come later)

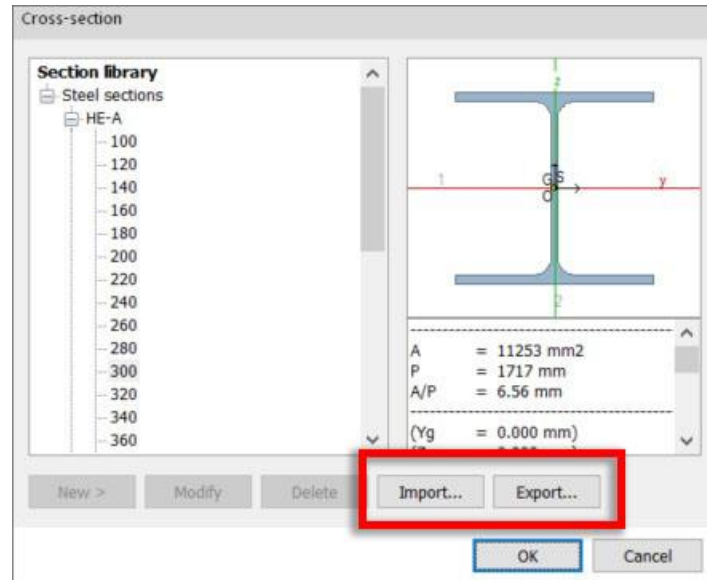


The program will automatically set the geometry of the joint components if the *Setup components automatically when cross-section changed* option is enabled. Otherwise, the components will not be modified after the cross-section has changed.



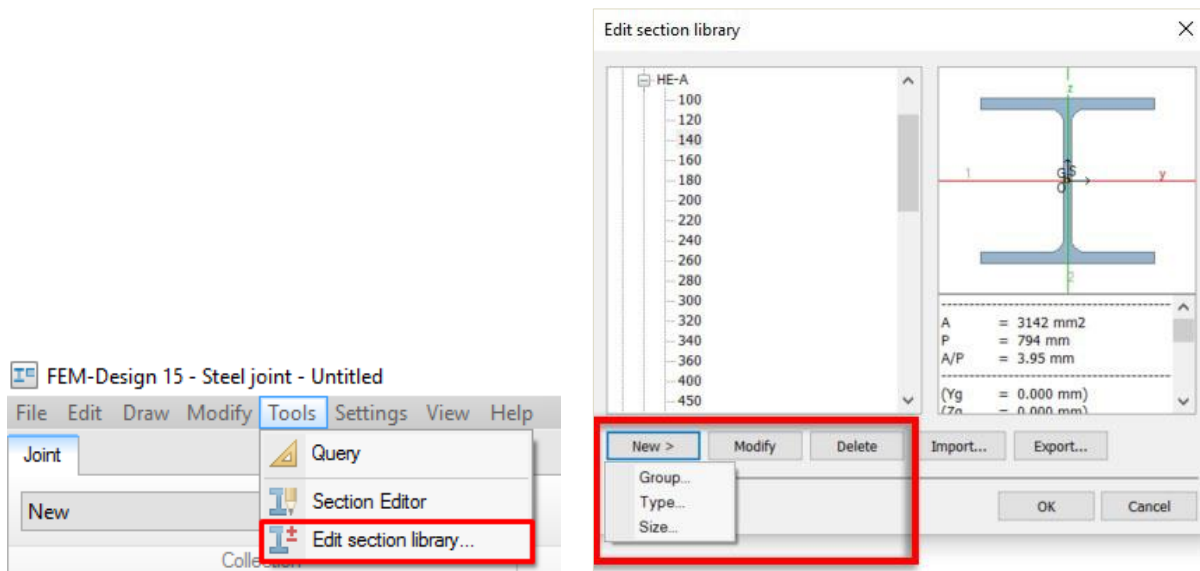
a. Cross section

The section library displays only those sections that are relevant for selected joint type. The library can be exported to a file, or imported from a file.



In the cross-section library only the same type of sections can be imported.

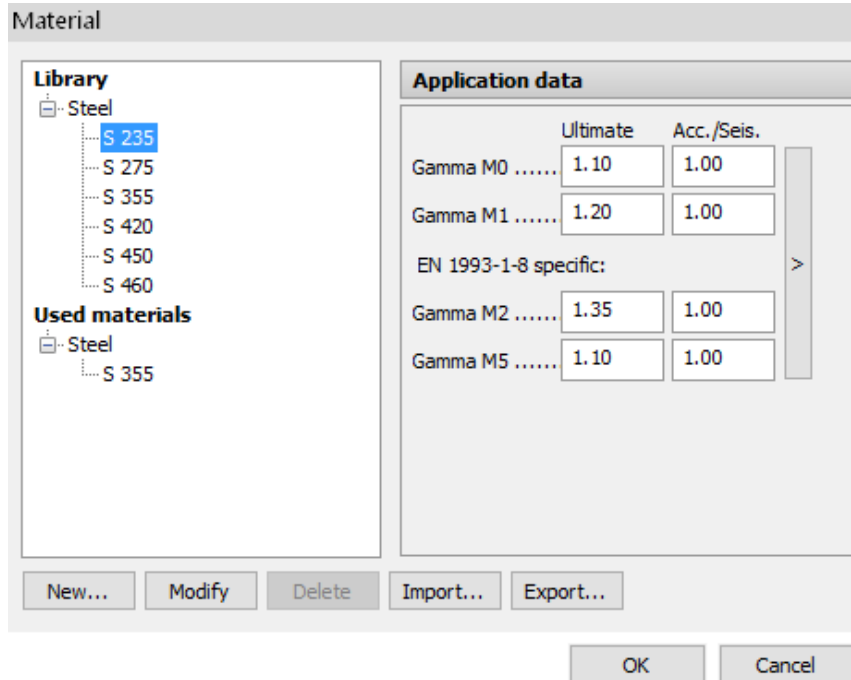
If a new section needs to be created, or the user wants to modify or delete the existing sections, it can be done in the *Edit section library* dialog, or by launching the *Section Editor* from the Tools menu (or the Toolbar).



For more information about section editing, see FEM-Design User Manual, *Structure Definition* -> *Properties* -> *Cross-section* chapter.

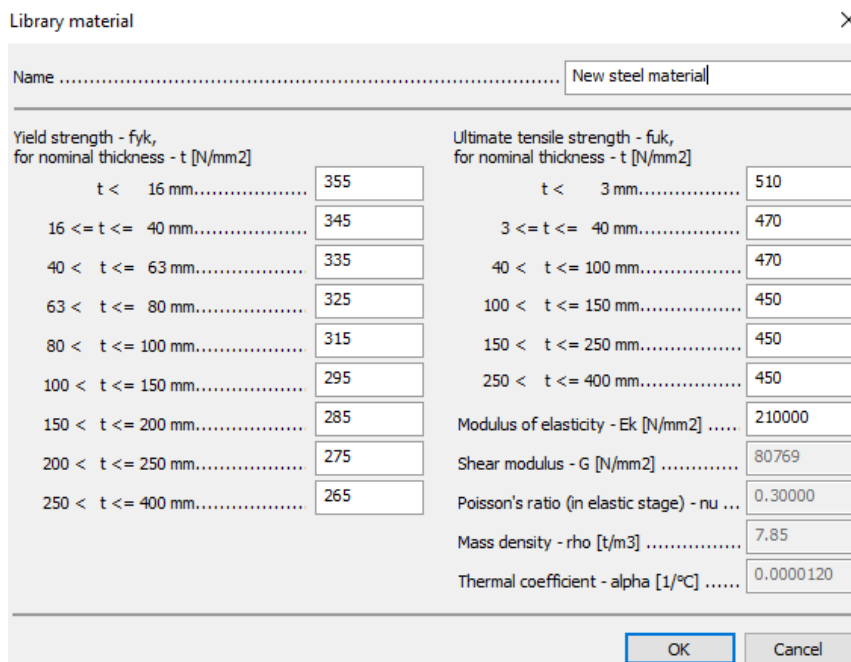
b. Material

The material library contains some standard steel grades. The library can be exported to a file, or imported from a file.



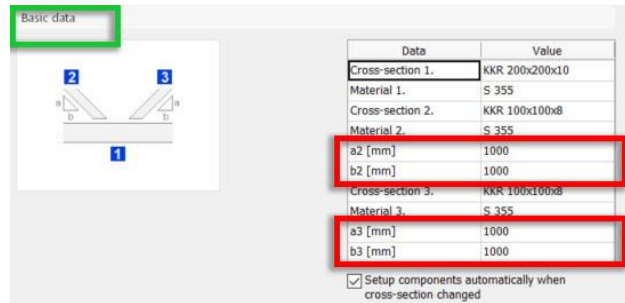
In the material library there is a new partial factor called *Gamma M5*, which refers to the resistance of hollow and pipe sections (see Eurocode 1993-1-8, 2.2).

The user can create a new material by pressing the *New* button.



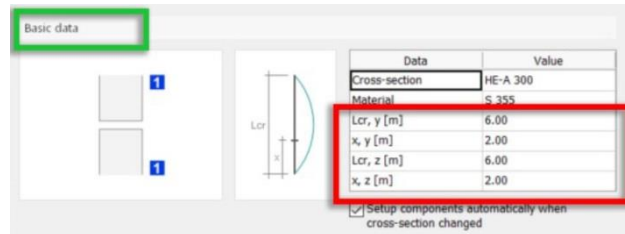
c. Geometry

In case of some joints it is possible to define the inclination of bars.



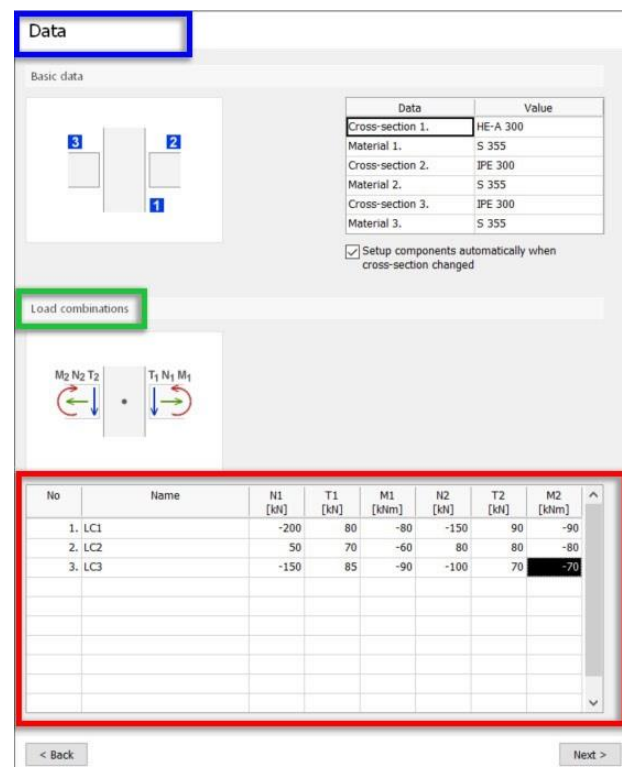
d. Buckling data

In some joints buckling data are present. They are considered in automatic calculation of 2nd order effects, if necessary.



Load combinations

Load combinations can be defined at the bottom of the *Data* panel.





A selected load combination can be deleted by pressing *Delete*.

The 2nd order effect is considered in column splice and column base joints. If 2nd order calculation is marked by X in the *Load combinations* table, it is supposed that the internal forces come from 2nd order calculation.

Otherwise, internal forces are supposed to come from 1st order calculations and 2nd order effect is calculated automatically, based on the buckling data defined by the user in the *Basic data* table.

Data

Basic data

Data	Value
Cross-section	HE-A 300
Material	S 355
Lcr, y [m]	6.00
x, y [m]	2.00
Lcr, z [m]	6.00
x, z [m]	2.00

Setup components automatically when cross-section changed

Load combinations

2nd order effects

No	Name	2nd	N [kN]	Ty [kN]	Tz [kN]	My [kNm]	Mz [kNm]
1.	LC1	X	0.00	0.00	0.00	0.00	0.00

< Back Next >

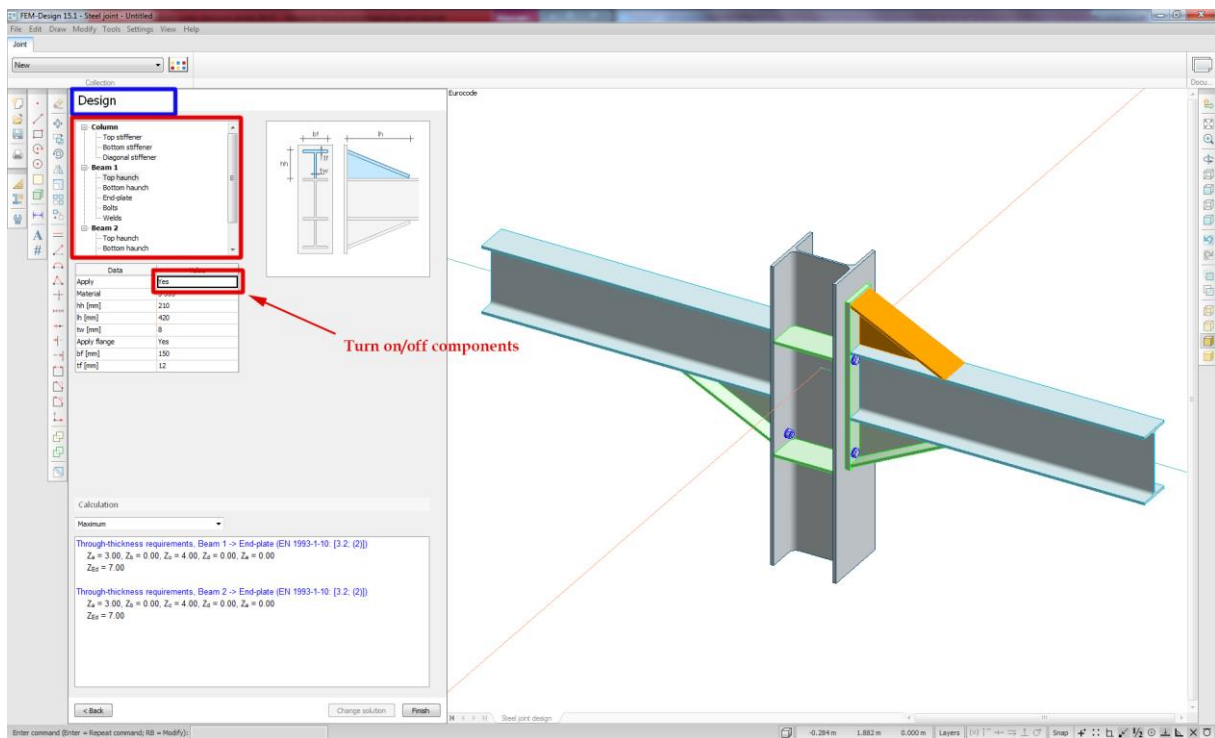
The user can step forward to the *Design* panel by clicking on the *Next* button.

3.4. Design

Modify, add and remove components

In the *Design* panel, components of the joint (e.g. end plate, splice plate, bolts, weld, etc.) can be modified. Additionally, some components such as stiffeners, haunches, etc. can be added or removed by setting *Yes/No* option in *Apply*.

A selected component is always highlighted in orange color in the 3D model.



Bolts

By selecting *Bolts* the user can define the bolt's size, quality and other parameters.

The *Bolt library* provides a possibility to create, modify, and delete the bolts, as well as the import or export from / to a file.

a. Bolt size

The Design software interface shows a bolted joint model. The left sidebar lists components: Column (Top stiffener, Bottom stiffener, Diagonal stiffener), Beam 1 (Top haunch, Bottom haunch, End-plate, Bolts, Welds), and Beam 2 (Top haunch, Bottom haunch, End-plate). The main area displays a 3D model of the joint and a 2D technical drawing. A table below the drawing shows bolt properties:

Data	Value
Size	HR-M20
Quality	8.8
Prestressed	No
Flip	No
Washer at nut	Yes
Washer at bolt head	Yes
c1 [mm]	98
c2 [mm]	55

Below this table is a table for flange placement:

Placed relative to...	n	e
[-]	[mm]	[mm]
Bottom haunch	2	50
Top flange	2	40
Top flange	2	-30

At the bottom, the Calculation section shows joint utilization: 92 % (LC: LC1 - Beam 2. flange weld). It also lists stress values for Beam 1 flange weld and Beam 1 web weld.

Click on bolt "Value"

Click on bolt "Edit library"

The Bolt library dialog box shows a list of bolt sizes: M12, M16, M20, M22, M24, M27, M30, M36. Under 'Used bolts', HR and M20 are listed. An 'Edit library' button is highlighted with a red box. 'OK' and 'Cancel' buttons are at the bottom.

Set bolt sizes

The Bolt library configuration dialog box shows the 'HR' family selected. The 'Name' field is set to 'M12'. A list of sizes (M12, M16, M20, M22, M24, M27, M30, M36) is shown, with 'M12' selected. The 'Length' field is set to '35'. Technical drawings of the bolt are shown. Dimensions are listed in a grid:

d	12.00	da	15.20	k	7.05	m	10.37
P	1.75	ds	11.30	r	1.20	en	23.91
b	30.00	dw	20.10	s	21.16	sn	21.16
c	0.40	e	23.91	As	84.30	tw	3.00

Additional dimensions: ls = 6.00, lg = 11.25. At the bottom, buttons for 'New', 'Modify', 'Delete', 'Import...', and 'Export...' are highlighted with a red box. 'OK' and 'Cancel' buttons are at the bottom right.

b. Bolt position in general case

In general cases (like splice plates, fin plates, etc.) the number of the bolt rows, columns, distance between the columns (c), distance between the rows (e) and the distance of the first row from the plate's edge (e_0) can be set by the user. The bolt columns are automatically positioned symmetrically in the plate.

Bolt position parameters

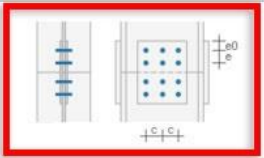
Design

- Column
- Web plate
- Web bolts
- Flange plate
- Flange bolts

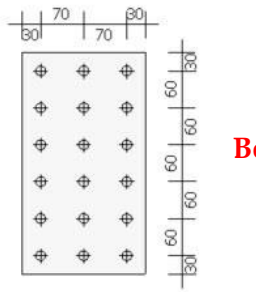
Data	Value
Size	HR-M20
Quality	8.8
Prestressed	No
Flip	No
Washer at nut	Yes
Washer at bolt head	Yes
n column	3
n row	3
c [mm]	70
e0 [mm]	30
e [mm]	60

< Back
Change solution
Finish

Explanation figure



Bolt arrangement figure





The bolts arrangement figure is automatically refreshed when changes are applied.

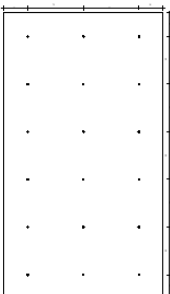
If in some cases, the bolts arrangement drawing and the dimensions are too small or too big, it is because the figure is not scaled properly. The user has the opportunity to change the scale of the drawing in order to see the figure clearly.

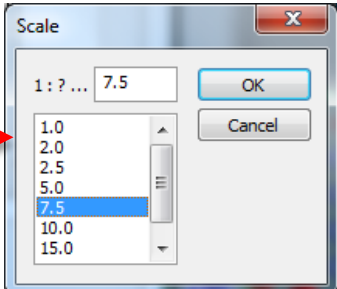
Bad scaling

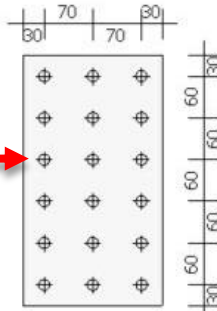
1. Right click in the figure

2. Set the scale by writing into the textbox or selecting the proper value

Proper scaling





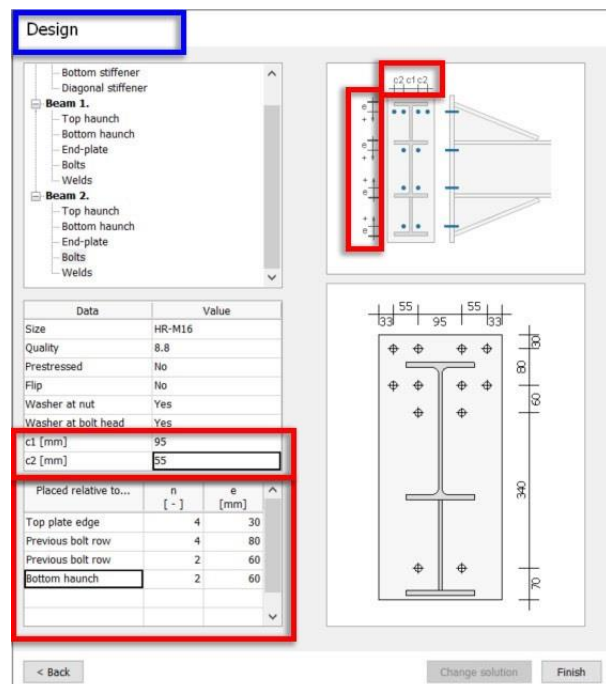
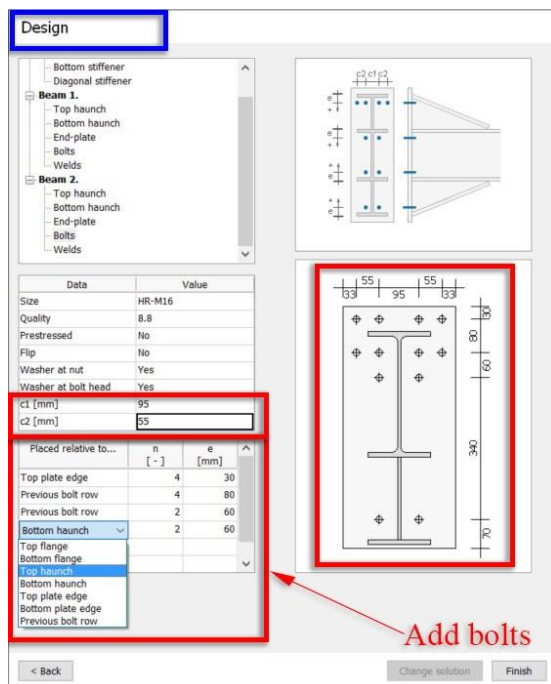


c. Bolt position in end plate for moment-resistant joints

In case of end plates for moment-resistant joints the user has more possibilities to define the bolts position.

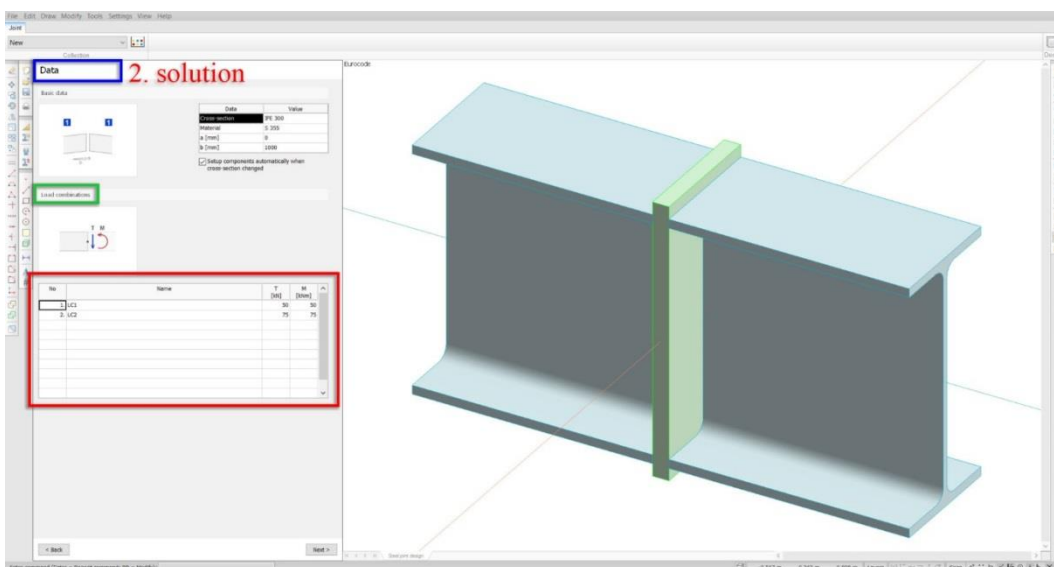
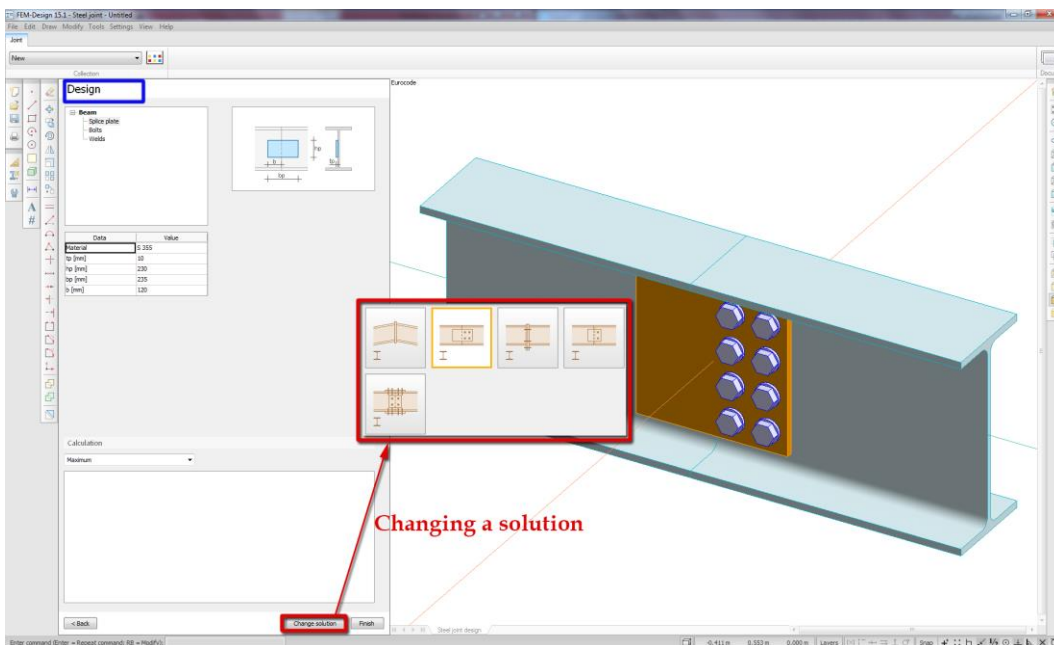
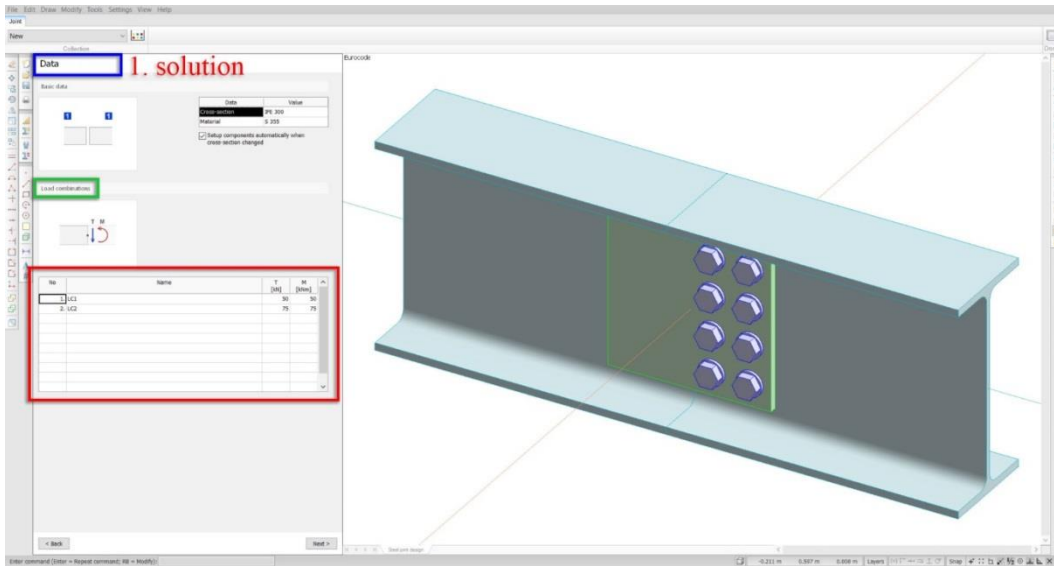
There may be either 2 or 4 bolts in a row and the distance between the inner bolts ($c1$) and the outer bolts ($c2$) have to be set. Then for each row the user can define the:

- Number of the bolts
- Distance of the row relative to one of the following positions:
 - Top flange
 - Bottom flange
 - Top haunch (if available)
 - Bottom haunch (if available)
 - Bottom plate edge
 - Top plate edge
 - Previous bolt row



Change solution

Change solution button is located in the bottom-right of the *Design* panel. A joint solution can be changed only if there is/are another solution(s) available (if the button is active). When the solution is changed, internal forces will remain the same.

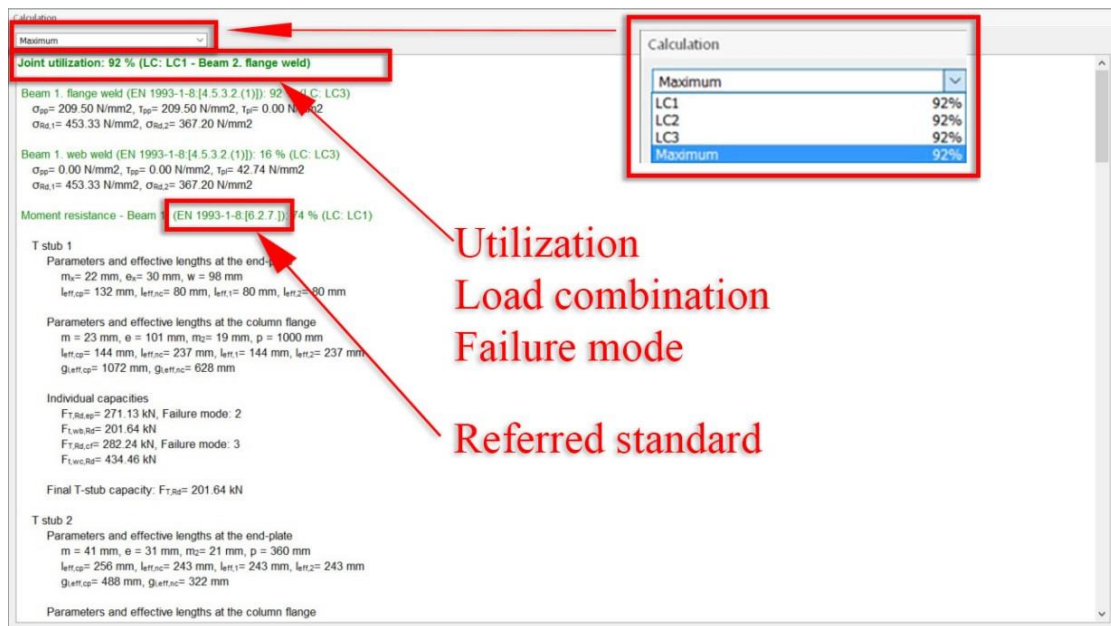


3.5. Calculation window

Content

Utilizations and a failure mode of the joint are displayed in the *Calculation* dialog. Each calculation has a reference to the specific standard.

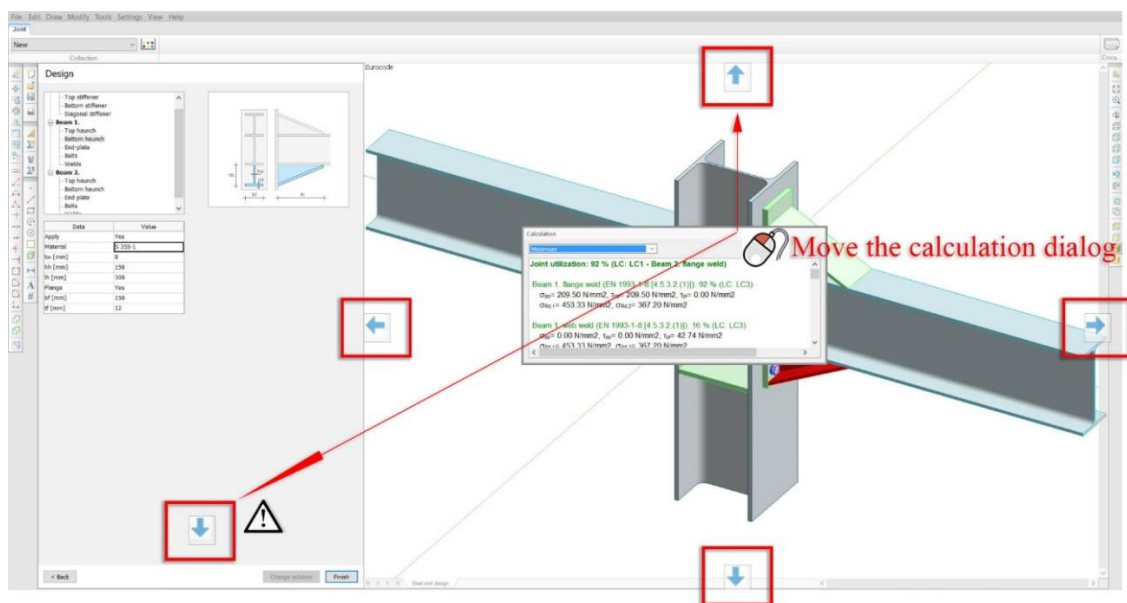
The calculation results can be displayed either for the maximum, or for selected load combination.



Utilization
Load combination
Failure mode
Referred standard

Moving the calculation window

The *Calculation* window is located at the bottom left corner as default, but the user can relocate it by docking the window to the blue arrows shown below.





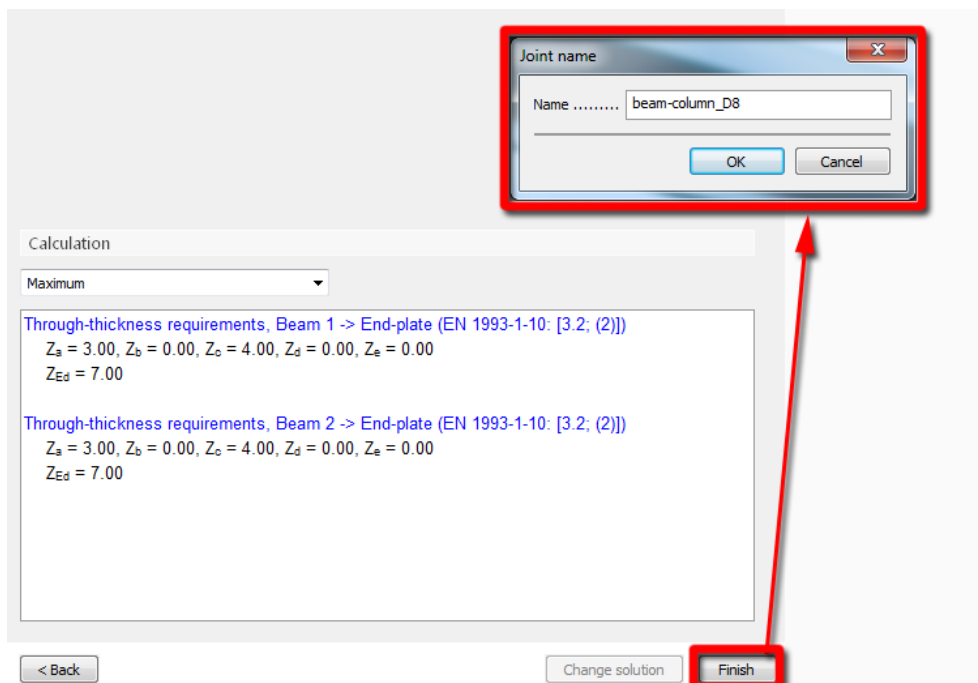
When the *Calculation* window is docked on the bottom-left (default solution) it is not visible in the interface in *Joint* and *Data* panels – only in *Design* panel.

If the user wants to see the window in all interfaces, then it should be relocated to one of the remaining four docking positions.

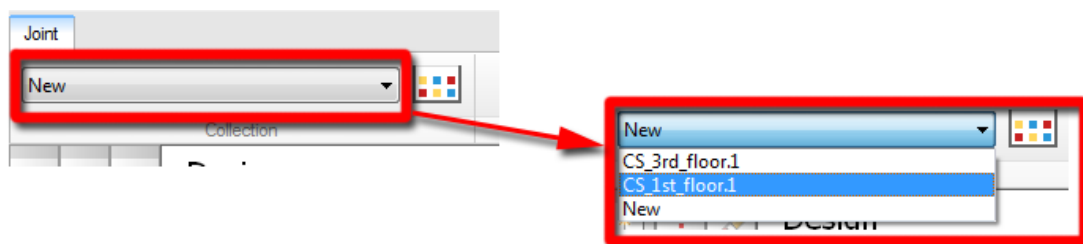
3.6. Collection

The purpose of *Collection* is to collect and document the joints that belong to one model.

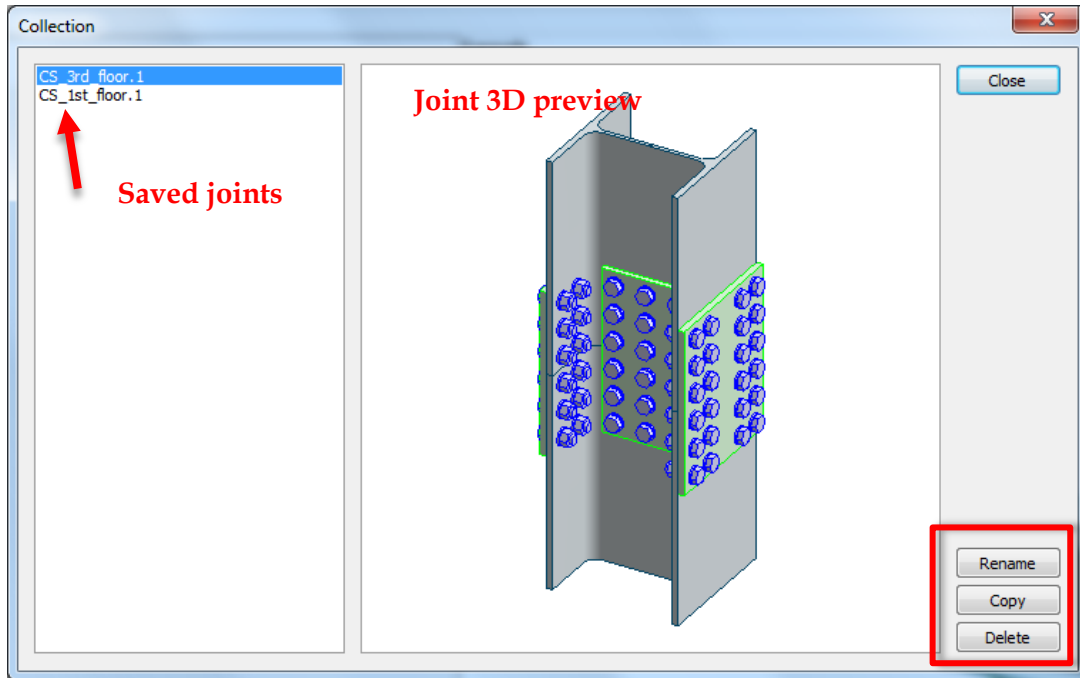
When a joint is designed, it can be added to the collection by clicking on *Finish* button and assigning a name to it.



After saving the designed joint, a new joint can be started. Designed joint can be reloaded from the list for modification.

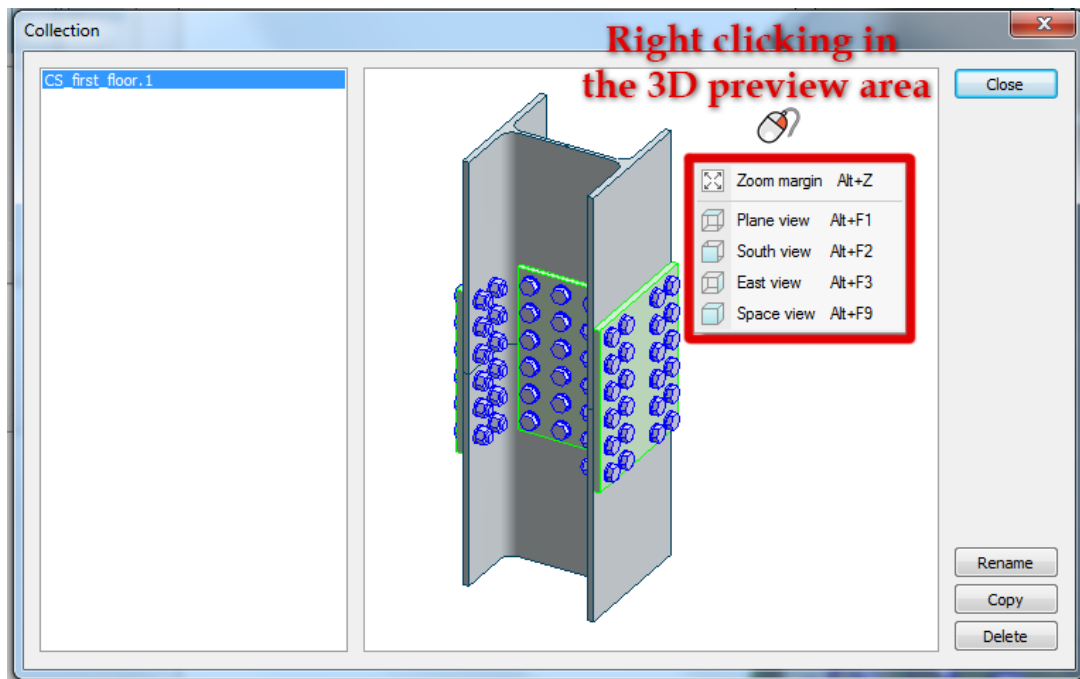


Any joint in *Collection* can be displayed in a separate dialog along with a 3D model of it. In this dialog the user can rename, copy or delete the joints.



The 3D preview area behaves like the model space in other modules, therefore:

- right clicking in the model space will pop up the view commands,
- dragging the view by holding the middle mouse button will animate pan the view,
- using the wheel on the mouse will zoom in and out the model view,
- using CTRL + middle mouse button will animate rotate the model view.



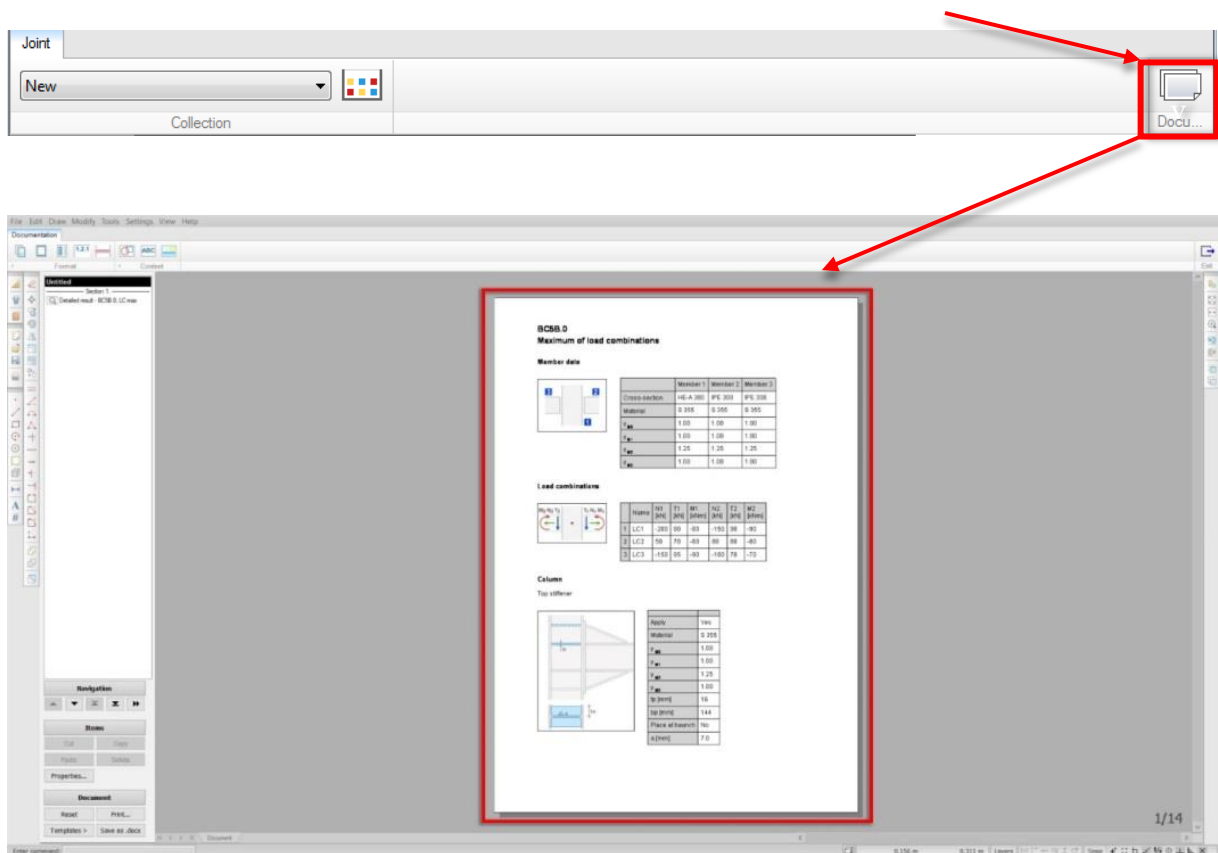
3.7. Documentation

General

FEM-Design automatically generates documentation for each joint added to the *Collection*. The documentation is located in *Documentation* module.

The documentation contains the input data of the joint (Member data, Load combinations, data of all components) and detailed calculations, so called detailed result.

**Click on the Documentation icon
to enter Documentation module**

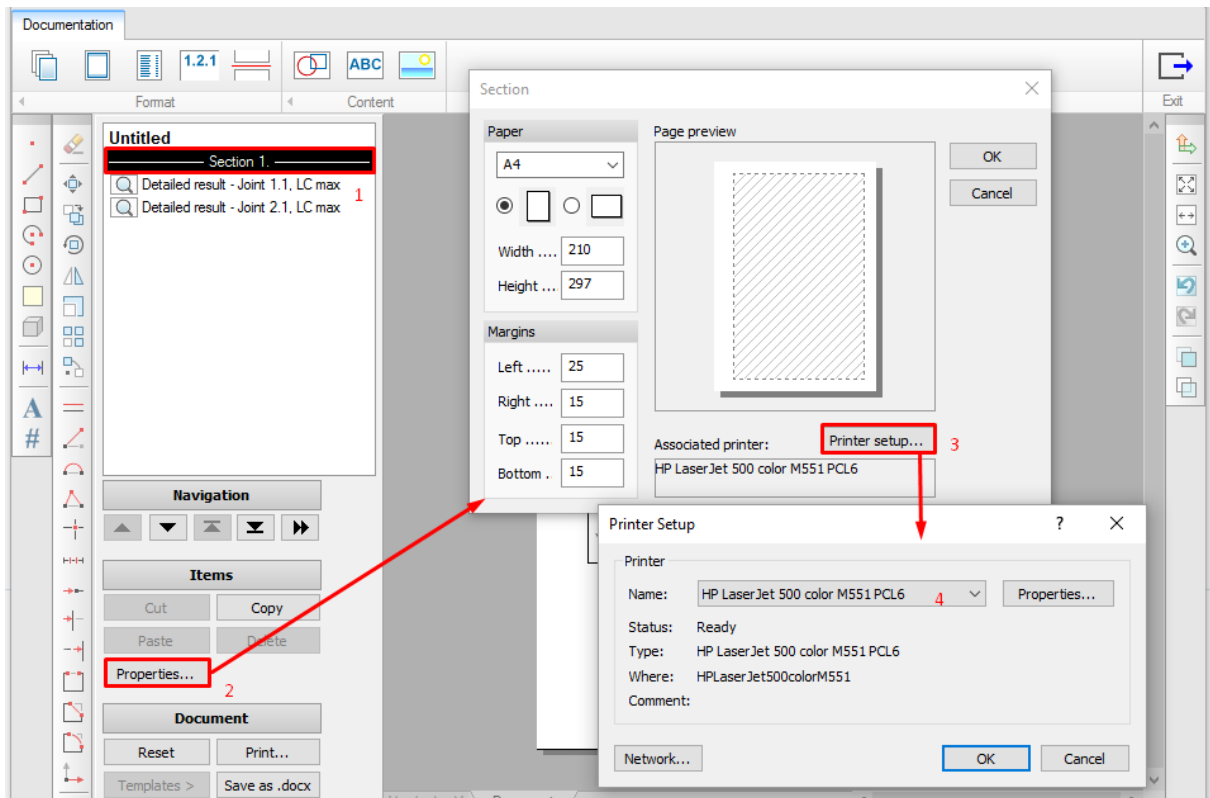



Printing

a. Setup a printer

The printing process requires a printer(s) associated with the section(s) of the current documentation. If a section does not have a defined printer (*Undefined*), you can add one by:

- selecting the section name in the document list (1),
- pressing *Properties* button in the Control panel (2),
- pressing *Printer setup* button (3),
- Selecting a printer from the dropdown list (4).



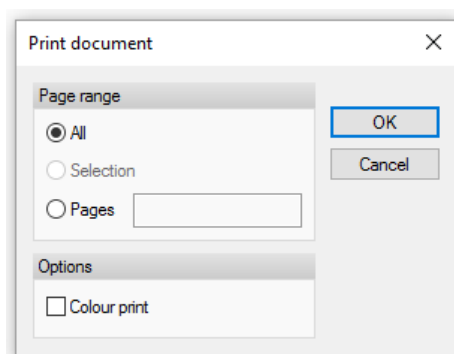
 If you assign a PDF printer all sections, you can save your documentation as a PDF file.

b. Print range

The completed document, or only some parts of it, can be printed out with Control Panel's *Print* tool or *File -> Print document*.

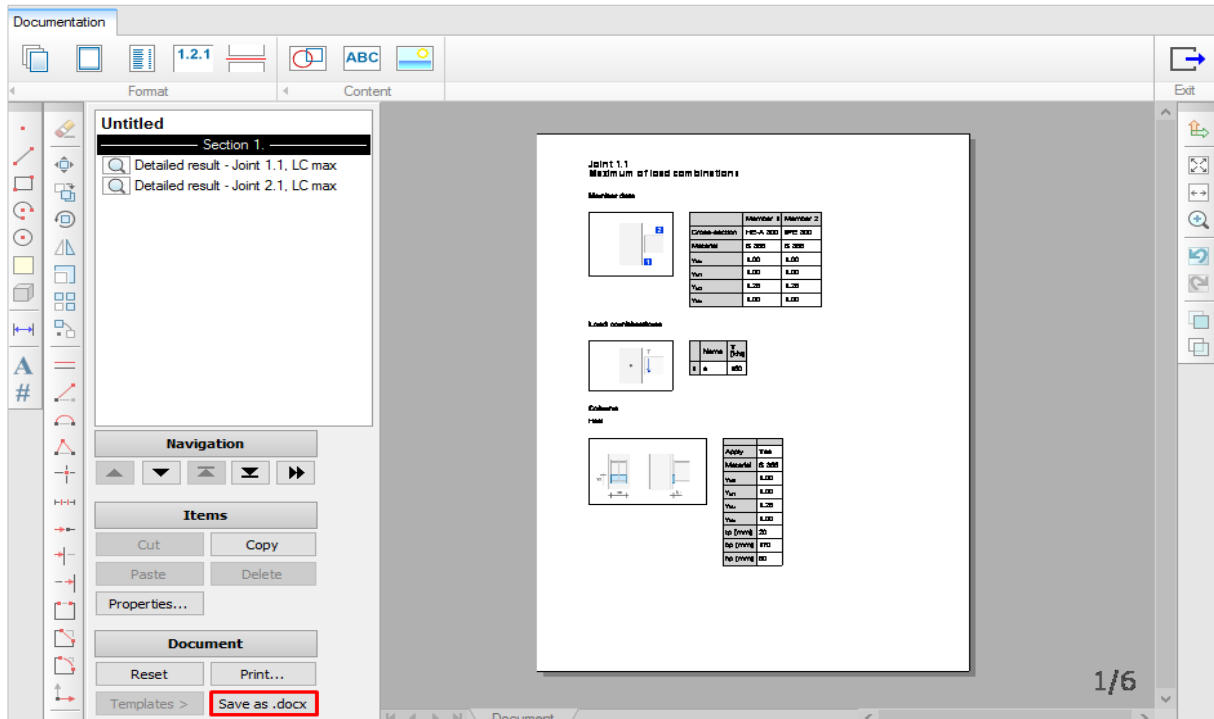
The user can set the *Page range* as:

- *All* -> The whole document (all pages) will be printed out.
- *Selection* -> The objects selected in the *Control panel* (multi-selection also works) and their pages will be printed out.
- *Pages* -> Only selected pages will be printed. Enter page number and/or page ranges by separating with commas and by using "-" (an example: 1,10,15-20,25, etc.).



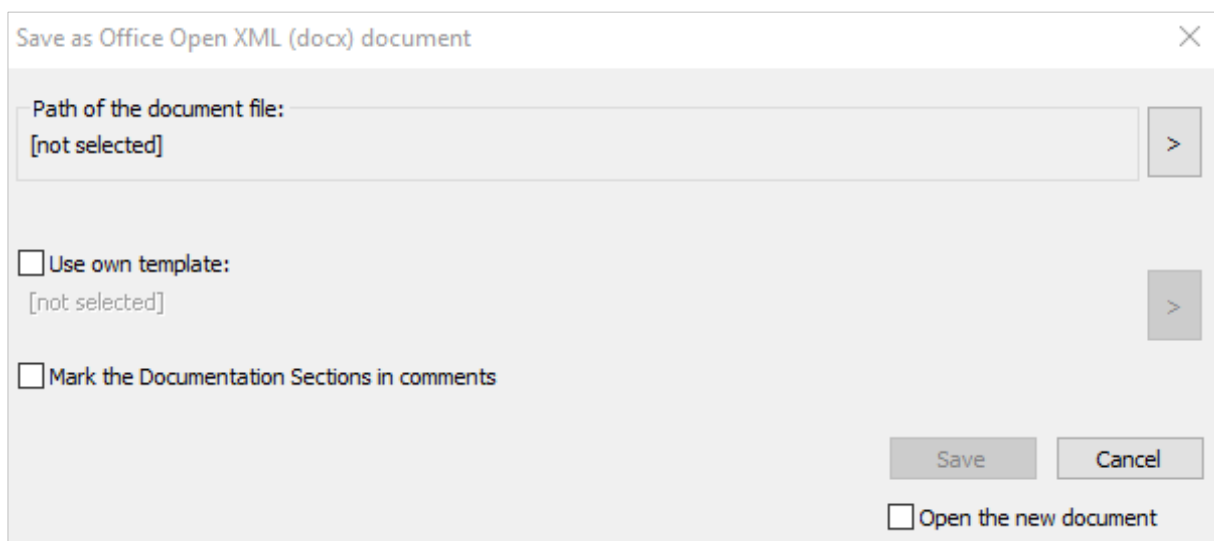
Export to .docx format

It is possible to export the documentation into the Office OpenXML (.docx) file. In the *Documentation* module press the *Save as .docx* button at the bottom right corner of the Control panel.



Save as Office open XML (docx) document dialog will appear. Here the user can decide the followings:

- here to save the result file,
- which MS Office Word template file to use (optional),
- whether to mark the Documentation Sections in comments or not (only in Documentation module).





The own template must be based on the original template file called *Template for fem-design document templates.dotx* located in your local user folder at your PC in *Documents\FEM-Design document templates* folder, as this file contains the style required for the correct export.



MS Office Word 2007 or newer is required for opening files with .docx distribution.

It is highly recommended to mark the documentation sections in the exported file, since the page rendering in MS Word and in FEM-Design are different. Furthermore, the exported document has A4 format pages. Therefore, if there are pages in the FD Documentation with different size, it is easier to find them and change their size in the result .docx file with section marks.

Using own Word templates is useful when the FEM-Design documentation contains Title block, since it is not possible to export it to .docx. As an alternative, the user can create a Word template with custom headers and footers, and use it for the export.



Since MS Word has more sophisticated solutions for creating the table of contents, FEM-Design does not export it to the .docx document. However, the chapters are exported therefore, the table of contents can be created in MS Word easily.



More information and useful hint can be found in the *Save as .docx FAQ* document accessible from the Help menu.

Advanced documentation tools

Documentation module is a powerful tool that allows you to create multi-page documentation with titles, figures, texts, tables, images, headers, footers, table of content etc.

In order to find out more about all the possibilities that the documentation module can offer, please read the *Documentation -> Documentation module* chapter in *FEM-Design User Manual*.