csJoint 9.0 USER MANUAL

EuroCode version

Version 9.0

29.05.2015.

www.consteelsoftware.com

CONTENT

1 GENERAL DESCRIPTION

1.1 INSTALLING THE SOFTWARE

csJoint runs successfully on the following systems. If your computer does not meet these requirements, you may not able to use *csJoint*, or their lack may result a slow operation:

Minimum hardware requirements:

- Processor: Intel Pentium IV or equivalent
- Memory: 2 GB
- HDD: 100 MB
- Video-card: 128 MB non-integrated
- Operating-system: MS Windows XP, 7 or 8

Recommended hardware:

- Processor: Intel Core 2 Duo or equivalent
- Memory: 4 GB
- Video-card: 512 MB non-integrated

You will need to have administrative or power user rights for the installation. Without administrative rights, the hard lock driver and those .**dll** files, which are indispensable for the system is cannot be installed.

The *csJoint* software can be installed from CD or over the internet. Install package can be downloaded from the *Downloads/Install packages* section of the ConSteel website (www.consteelsoftware.com/en/downloads/install-packages).

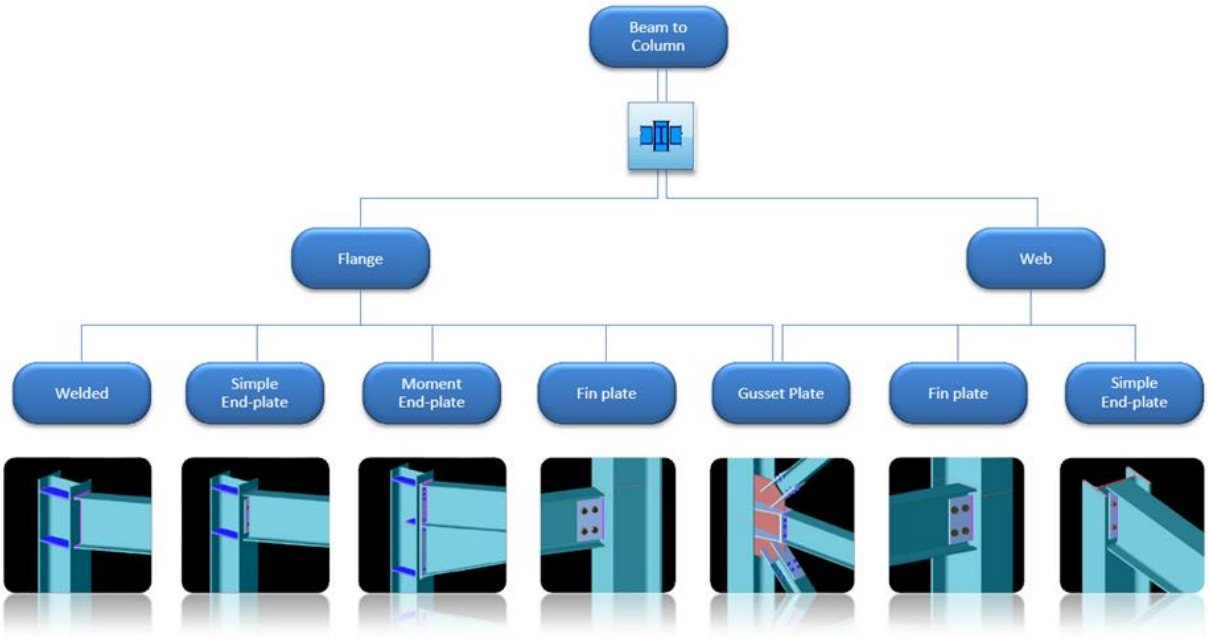
Open the *csJoint* install file (csJoint_9_0_install_xxxx_xx_xx.exe) and follow the instructions. As a first step the language of the setup has to be set. The *csJoint* will use this language for the first time you start, but you may switch to another language later. The installing program will copy the elements of the software into the directory you have specified (default directory: C:\Program Files\csJoint 9.0), then it will position the Menu of the program onto the selected place of the „START”-menu. Finally the install shield will place the starting-icon onto the desktop. The last step of the setup is the hard lock driver installation. This has no progress signal, so you will have to wait while it finishes.

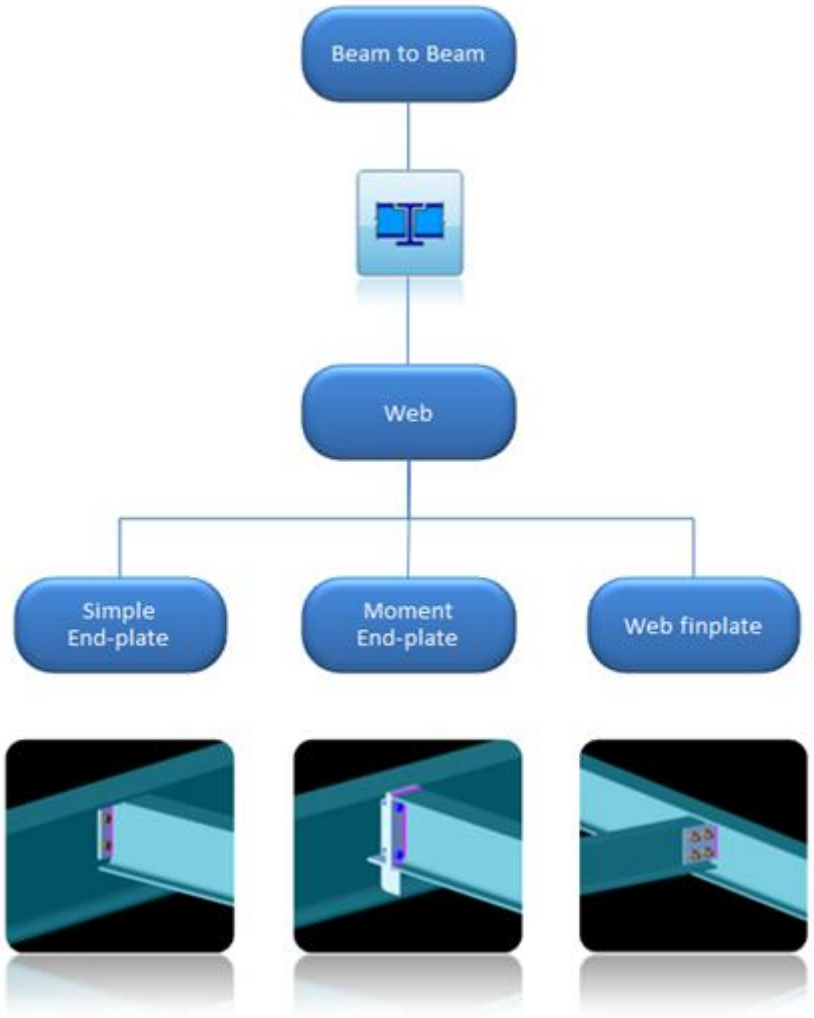
1.2 DESIGN STANDARD

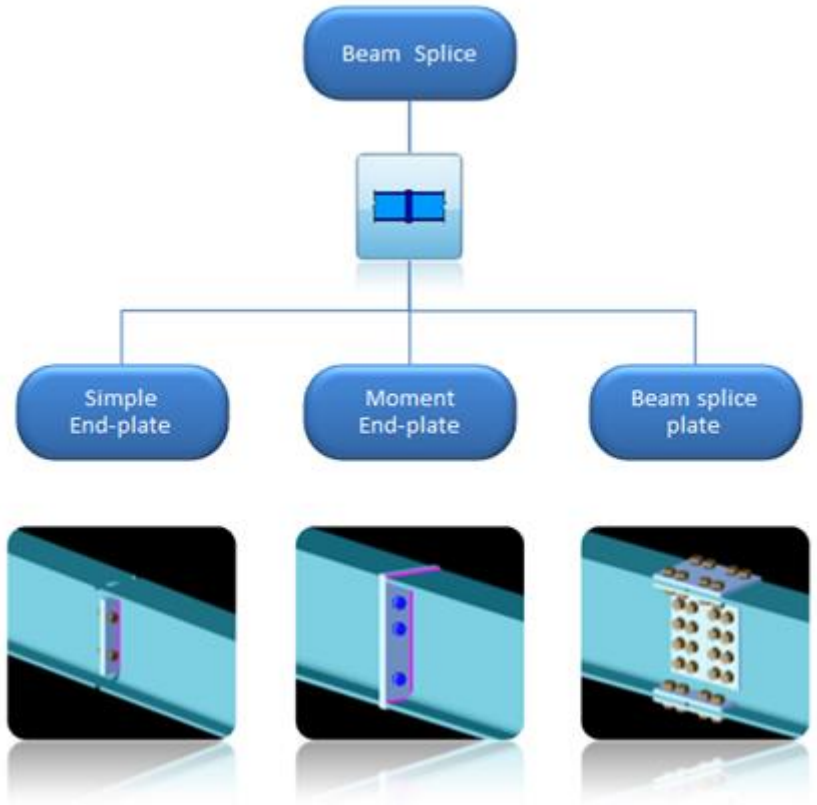
The design tools cover practically the whole joint standard EuroCode 3 Part 1-8. In some cases csJoint supports wider range of connection types which are not specified in the EuroCode 3 Part 1-8 design standard. In these cases the source of the calculation methods are specified in the detailed results of the calculations.

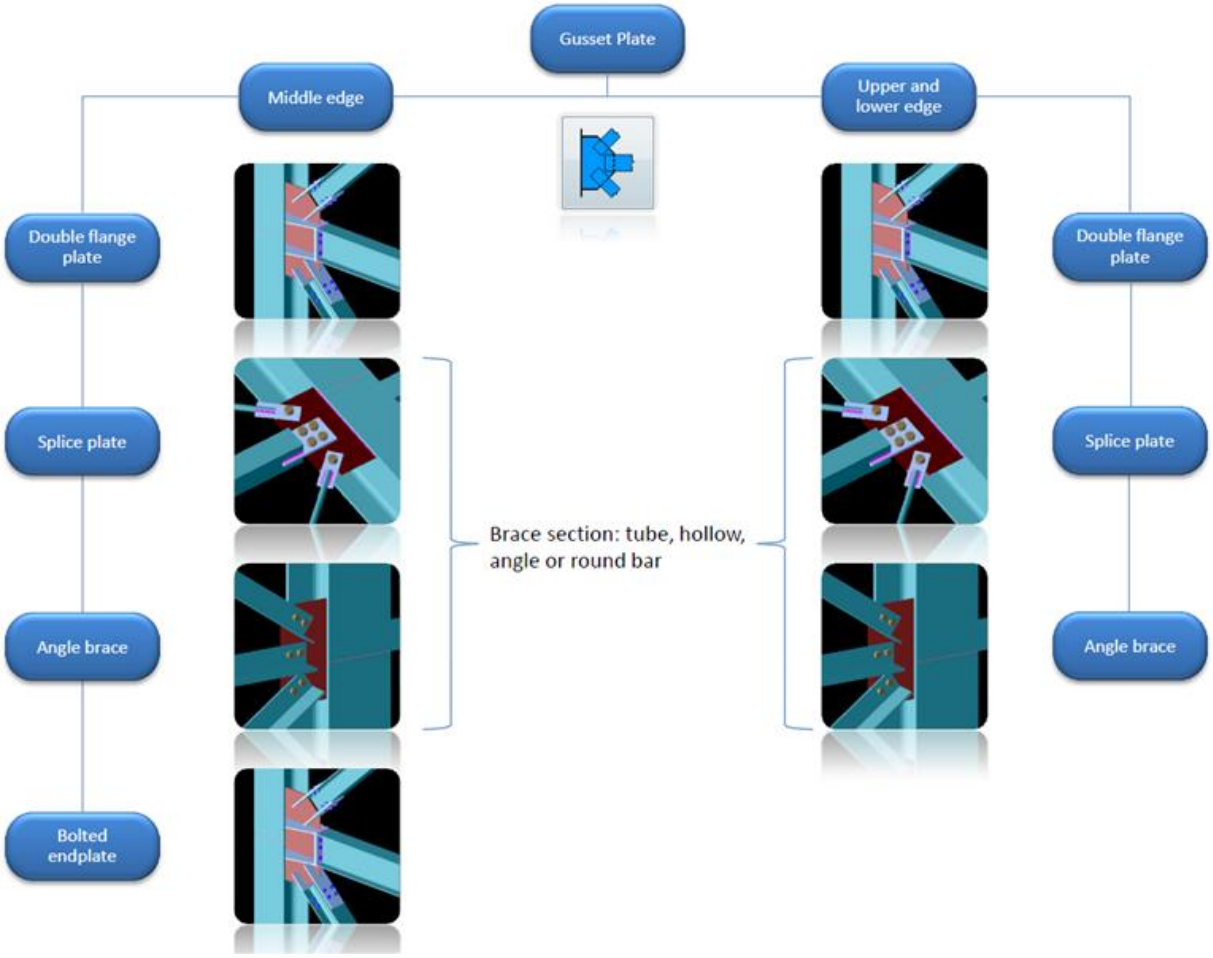
1.3 JOINT TYPES

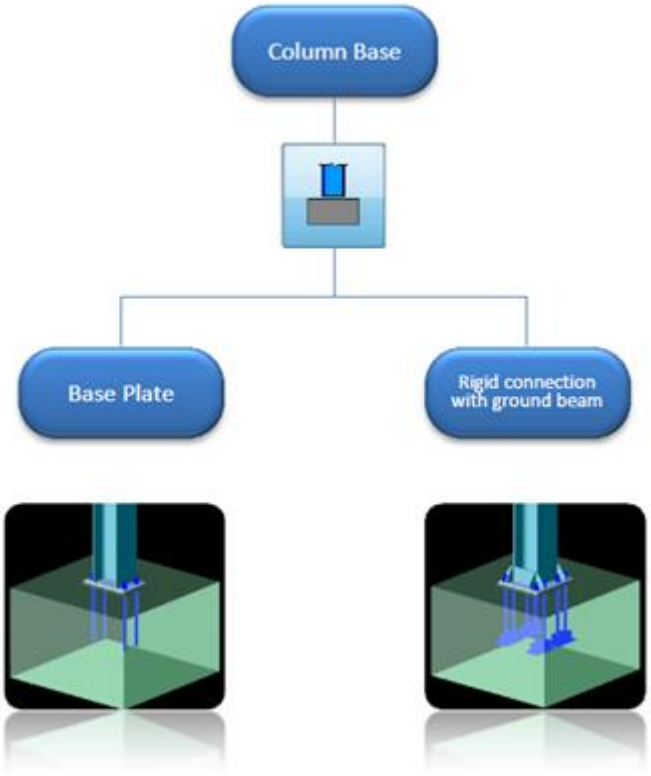
The number of the joint types is one of the most dynamically developing parts of the software, usually considering the claims of the users. At the moment the *csJoint* has the following joint and connection types:

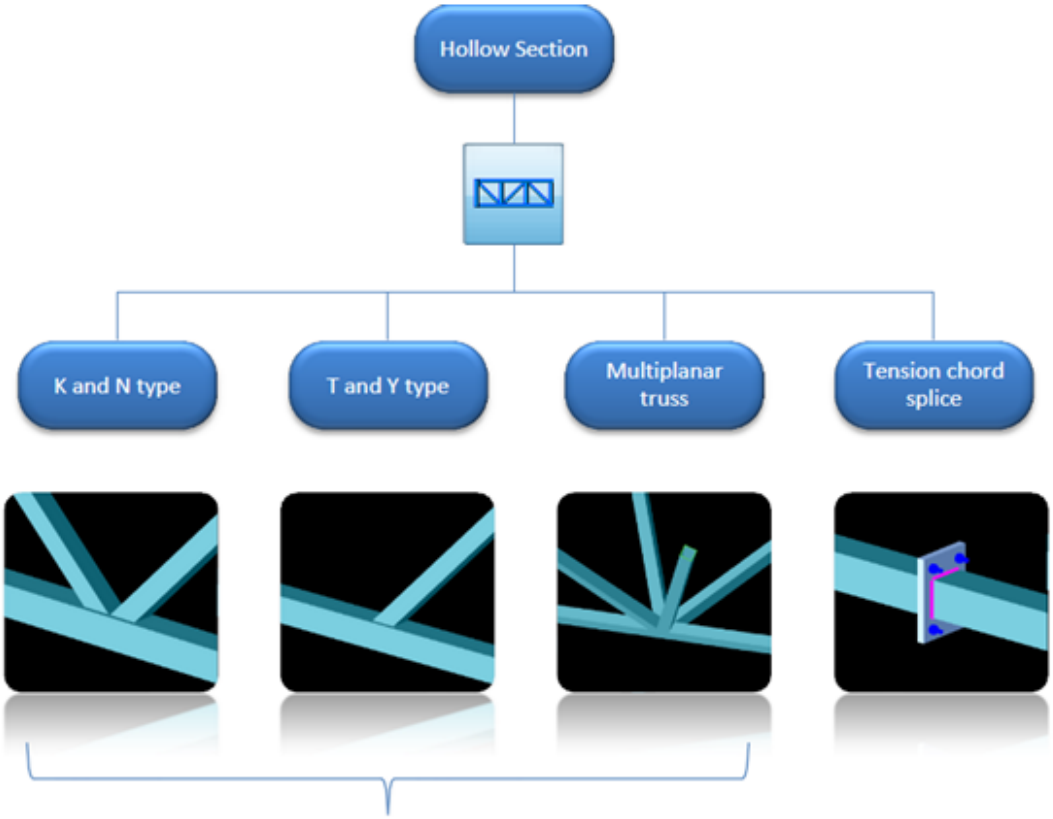



Beam to Beam
Web
Simple
End-plate
Moment
End-plate
Web finplate

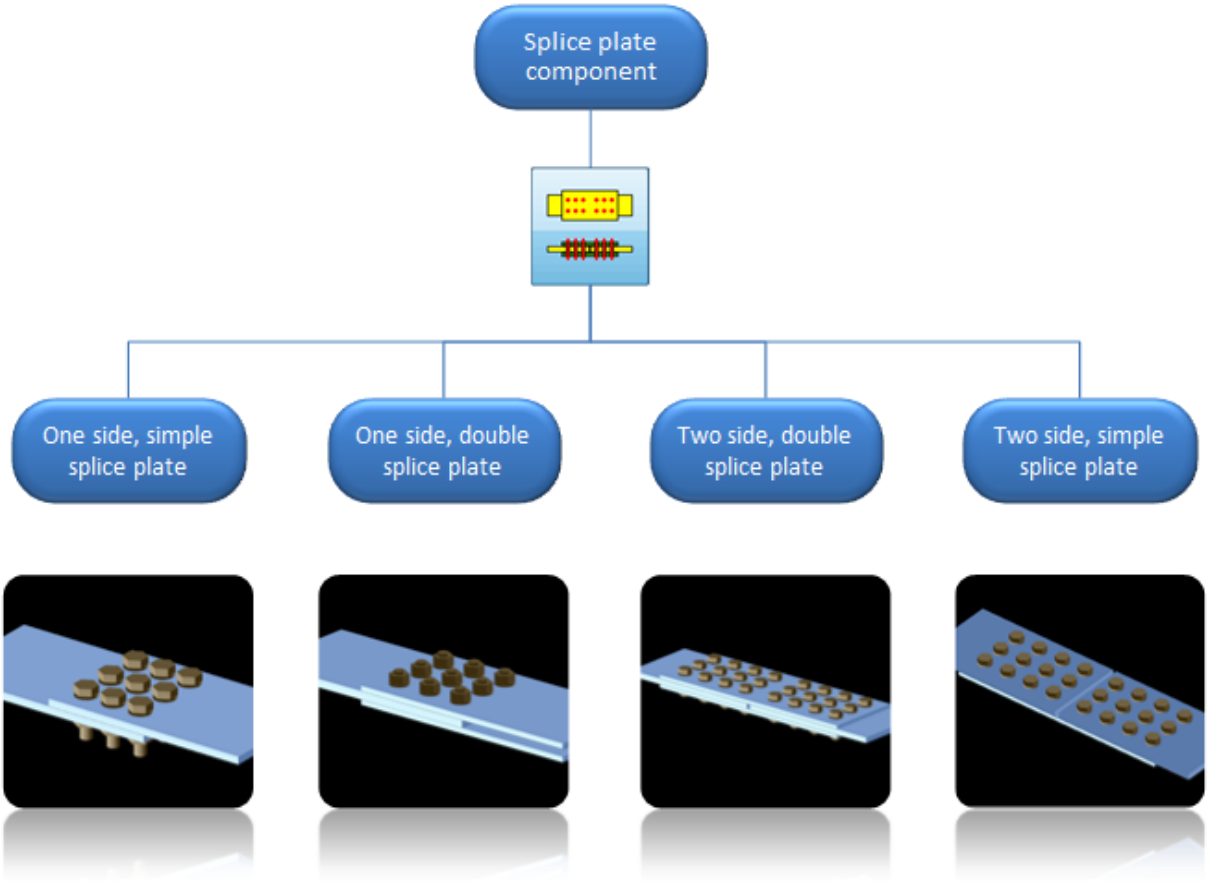

Beam Splice
Simple
End-plate
Moment
End-plate
Beam splice
plate


Gusset Plate
Middle edge
Upper and lower edge
Double flange plate
Splice plate
Angle brace
Bolted endplate
Brace section: tube, hollow, angle or round bar
Double flange plate
Splice plate
Angle brace


Column Base
Base Plate
Rigid connection with ground beam



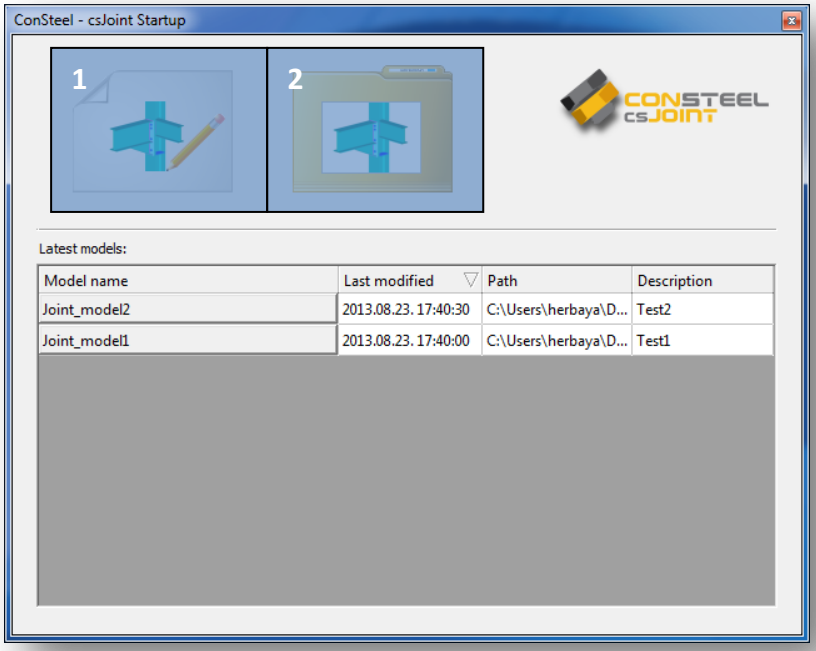
Chord section: tube, hollow, H or I
Brace section: tube or hollow section


Splice plate component
One side, simple splice plate
One side, double splice plate
Two side, double splice plate
Two side, simple splice plate

2 CREATE JOINT

2.1 THE STARTUP WINDOW

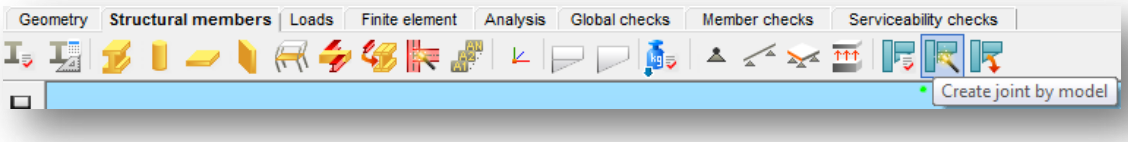
After starting *csJoint* there is a startup dialog window which allows creating and opening models easily. The latest models can be opened without browsing folders. They are sorted by the last modification date as a default setting but it can also be sorted by name or by model path.



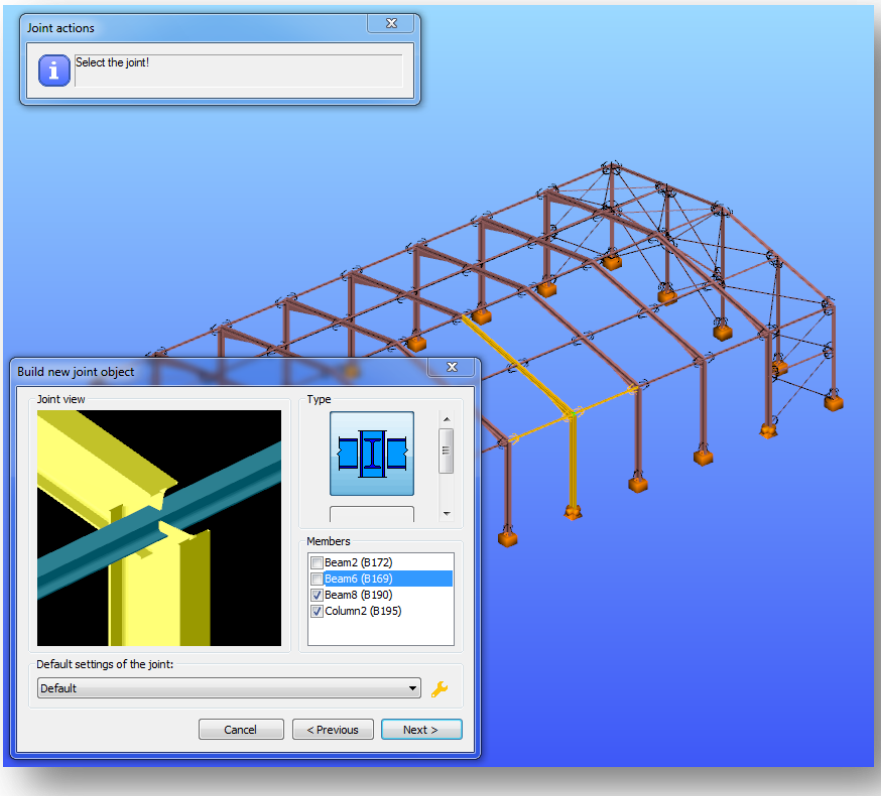
The first big icon (#1) is for creating a new model, the second (#2) is for open model from folder. Models can contain any number of joints.

2.2 CREATE JOINT BY MODEL

Creating joint by the model is simple and easy. Just click on the ***CREATE JOINT BY MODEL*** button and select the joint.



The members will be automatically identified and the possible connection types are offered. Here is possibility to remove some some members from the connection by removing the appropriate tick.



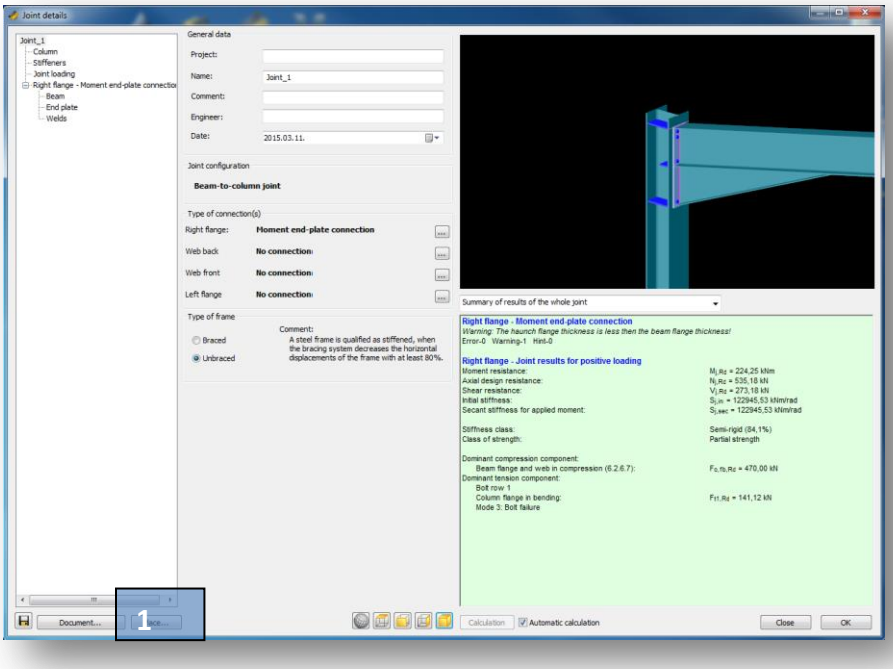
2.2.1 PLACE A JOINT

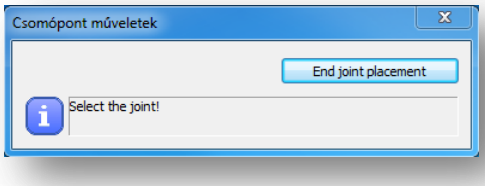
In *ConSteel*, after creating the joint it is advisable to place it in the global model. There are two options to do this.

2.2.1.1 PLACE JOINT DIRECTLY FROM JOINT DETAILS DIALOG

The fastest way of placing a joint, is to place it from the ***JOINT DETAILS*** dialog directly, right after when it is been created. ***PLACE…*** button can be found at the bottom left corner (#1) of Joint details dialog (see picture below). By choosing it, it is possible to place the joint to several places by single clicks, where same geometry exists. By clicking ***END JOINT***

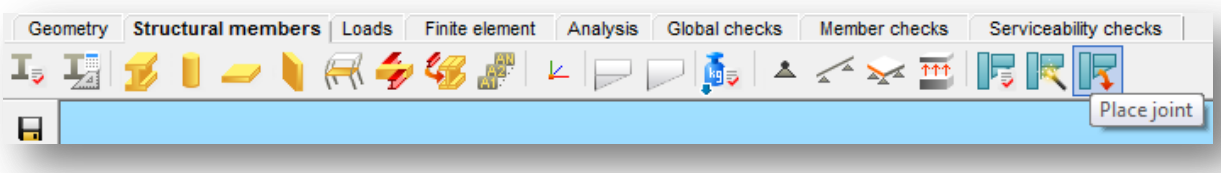
PLACEMENT (see picture below), loads will automatically imported from the model if analysis results are available.





2.2.1.2 PLACE JOINTS FROM STRUCTURAL MEMBERS TAB

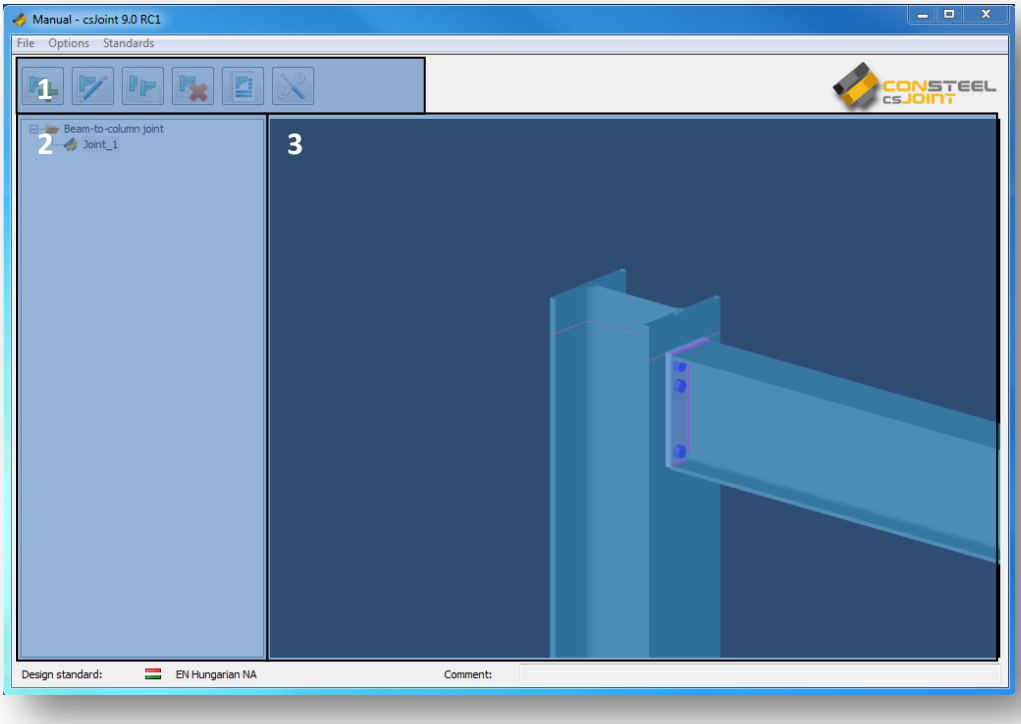
After a joint is created, it can be placed from the status bar too, by clicking on the ***PLACE JOINT*** icon. One joint can be placed to several places in the model where the same geometry exists. After the joint is placed, *csJoint* automatically imports the loads from the global model.



IT SHOULD BE BEAR IN MIND THAT THE SETTINGS IN THE CSJOINT MODULE (PROFILES SIZES, MATERIAL GRADES, ETC.) WILL NOT EFFECTS THE 3D MODEL. THE CHANGED VALUES WILL BE CONSIDERED IN THE JOINT CALCULATION ONLY! THE USER HAVE TO WAY TO KEEP THE MODEL CONSISTENT: 1. AFTER MODIFYING THE CONNECTION THE 3D MODEL HAS TO BE CHANGED ACCORDINGLY, 2. CHANGE THE 3D MODEL FIRST RUN THE ANALYSIS, REDEFINE THE CONNECTION.

2.3 CREATE JOINT WITHOUT MODEL

2.3.1 JOINT MANAGEMENT



The joint management window consists three separate parts containing different functionalities.

The #1 part, is the ***ICON BAR***, it contains the basic model management functions:

- *Create ():* create a new Joint
- *Details ():* details of a created joint, manipulate a created joint

- *Copy ():* copy the selected joint
- *Delete ():* delete the selected joint
- *Document ():* start the generation documentation of the selected joint
- *Default Joint Settings ():* create user defined default joint settings

The #2 part of the picture above is the tree structure of the previously created joints. Double click on a joint open the ***JOINT DETAILS*** dialog.

The #3 part of the picture above is the graphical window. 3D graphical model of the selected joint is shown here.

2.3.1.1 THE MENU

Three important function groups appear in the menu: the ***FILE*** handling, ***OPTIONS*** for settings (for saving, updating, selecting language) and ***STANDARDS*** for reviewing and defining standard parameters for design.

FILE menu:

New model

New model can be created.

Model attribute

Model description and the used EN National Annex can be changed. (These parameters can be set at first by creating a new model.)

Open, Save, Save As

The functions are according to their names.

OPTIONS menu:

The following settings can be found: ***SAVE***, ***UPDATE***, and ***LANGUAGE***.

Save

If **Autosave** is clicked, the program automatically performs a save periodically in accordance with the adjusted number of minutes. If **Backup** is clicked, *csJoint* creates a

backup file *after manual save is performed*. The backup model file can be used as a normal model by removing the ***.bak*** extension.

Update

csJoint looks for an available new version on the web at every startup. It can be turned off. The check for a new version can be performed manually by clicking on the **Search for updates now** () button.

Customer experience improvement

In order to make our products as much as possible to meet the expectations of our customers, our softwares collect and send us usage statistics. KÉSZ Holding cPlc uses this information to improve the products and features customers use most often and to help solve problems.

Our products collects and sends the following information: software version, used functionalities, running time, location

Sent information does not contain any personal information (such as your name, address, or phone number) and is not used to identify you.

In case of commercial license, collection of usage statistics can be turn off by clicking off the checkbox. In case of trial and educational licenses, collection of usage statistics is continuous.

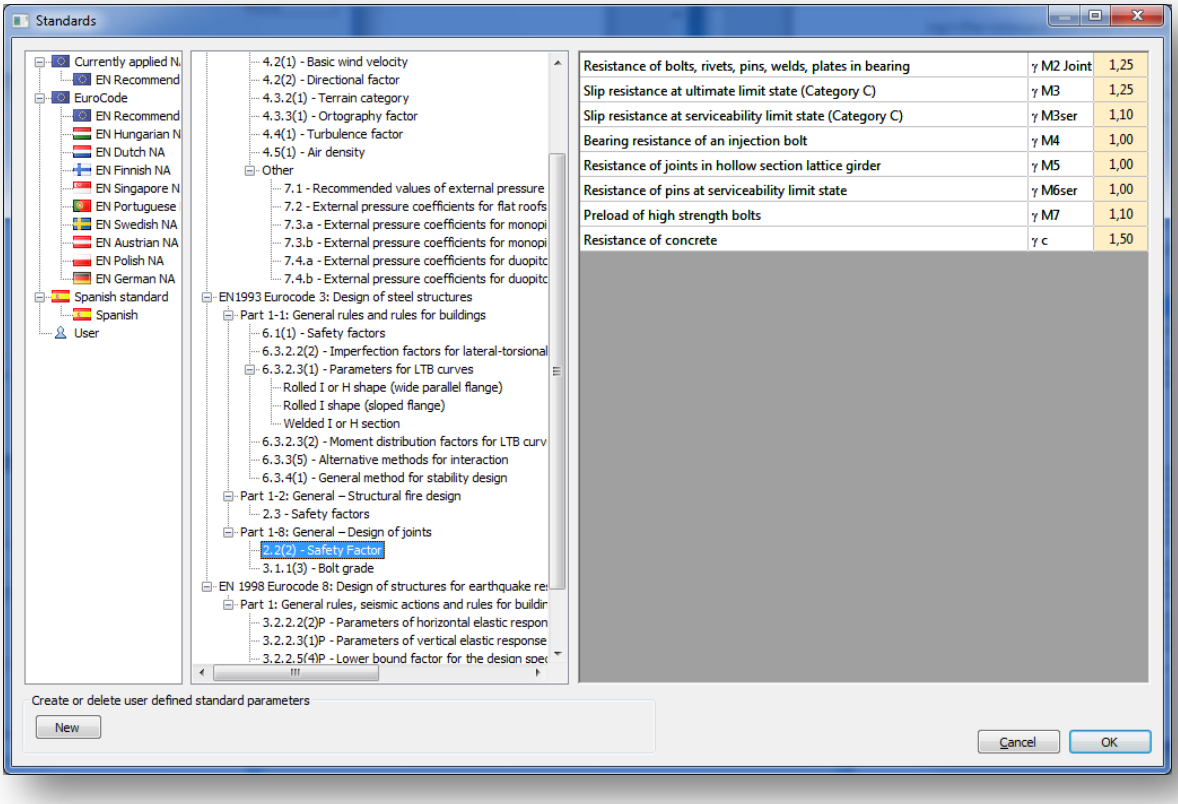
Language

The language of the user interface can be set here from the following languages: English, German, Spanish, Italian, Greek, Portuguese, Russian, Chinese, Turkish, Hungarian, Polish, Slovak, Slovenian, Rumanian, Bulgarian, and Serbian.

csJoint has to be restarted after a change.

STANDARDS menu:

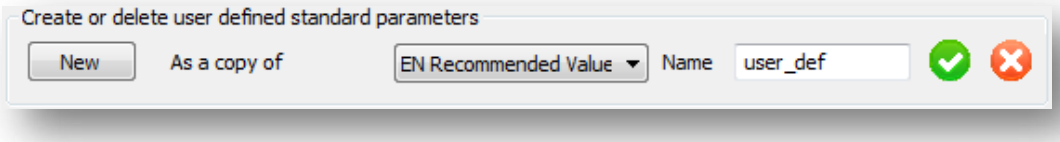
The menu provides a great opportunity to view existing and to create new standard parameters in an easy way. The used standard can be set in the ***FILE*** menu.



The first row of the *Standard tree* shows the applied standard which is stored in the model.

For creating a user defined standard:

- press ***NEW*** button
- select one of the existing standard from the list



- name the new defined standard
- parameters for the new defined standard can be changed in the chapters

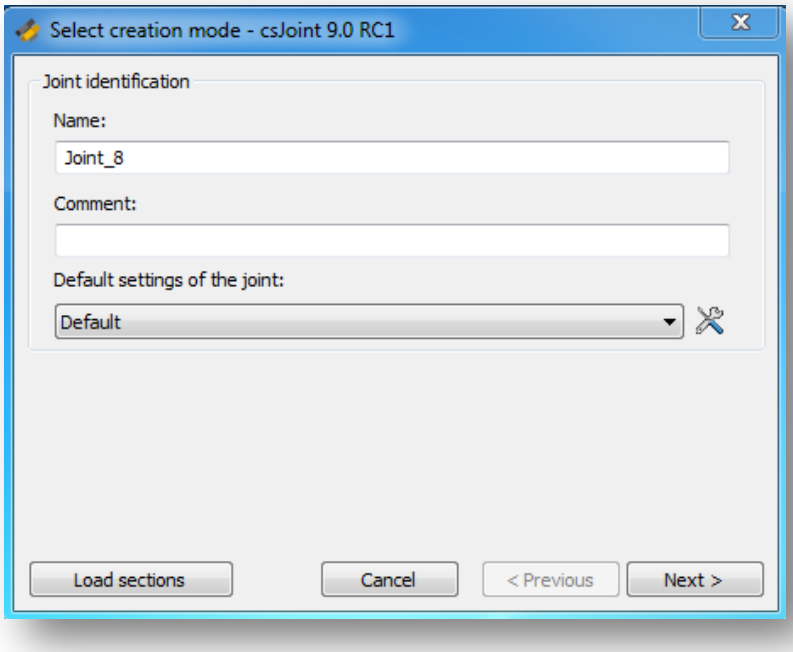
User defined National Annex saved to the following file: *\Documents\ConSteel\UserStandard.xml*

2.3.2 JOINT WIZARD

Creating a joint is an easy, simple, guided process by the ***JOINT WIZARD***. In 3 single steps, the joint can be created, and then it can be modified to fulfill personal demands.

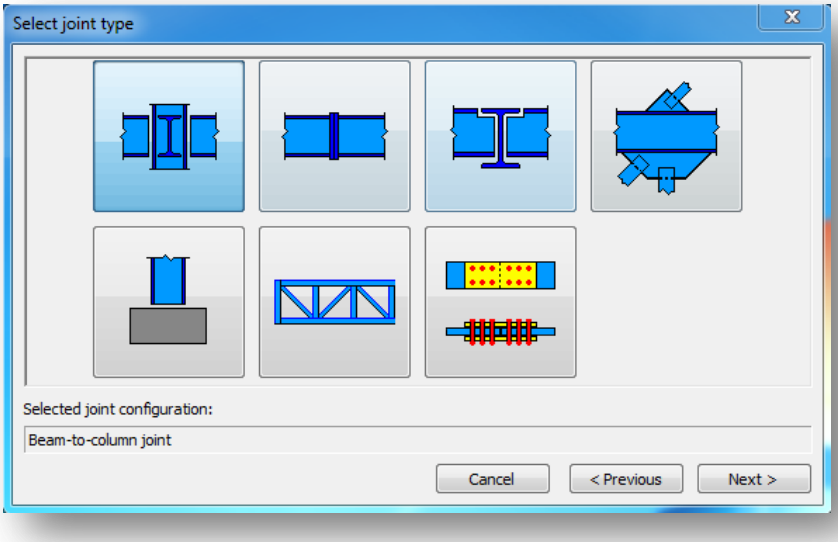
- *Step 1* – Joint identification

After clicking on the ***CREATE*** button () in the ***JOINT MANAGEMENT*** dialog, identification of the joint has to be made. It is possible to leave a comment. Although it is optional, it can be very useful for a better identification when several joints are created. Default joint settings can be selected for the joint creation. If it is necessary sections can be loaded, by clicking the ***LOAD SECTIONS*** button.



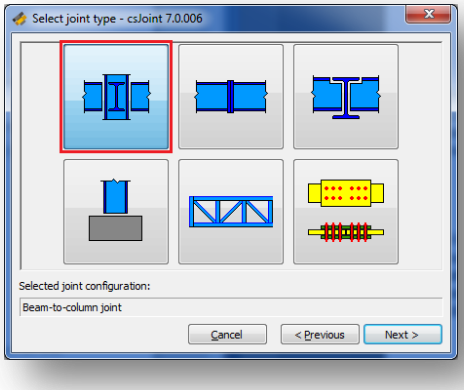
- *Step 2* – Select Joint type

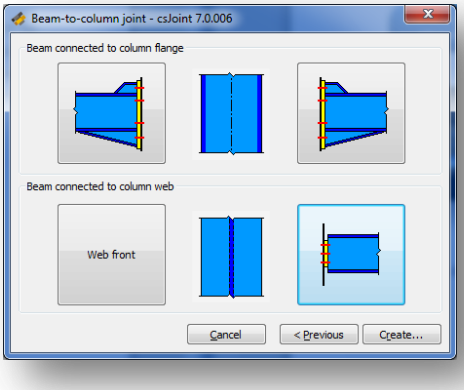
On the next dialog, joint configuration has to be set. Each joint type has an illustrated icon (picture below). See chapter 1.3 for a complete list of ***JOINT TYPES.***



- *Step 3* – Select connection type

After the selection of the joint type the connection type needs to be selected. In case of beam-to-column and beam-to-beam joint more than one connection type could be chosen for the joint, because different connection could be selected for example for the left and the right web of the main girder if it is necessary. By clicking on the ***CREATE…*** button the selected joint will be created according to the ***DEFAULT JOINT SETTINGS*** or the selected parameter file.



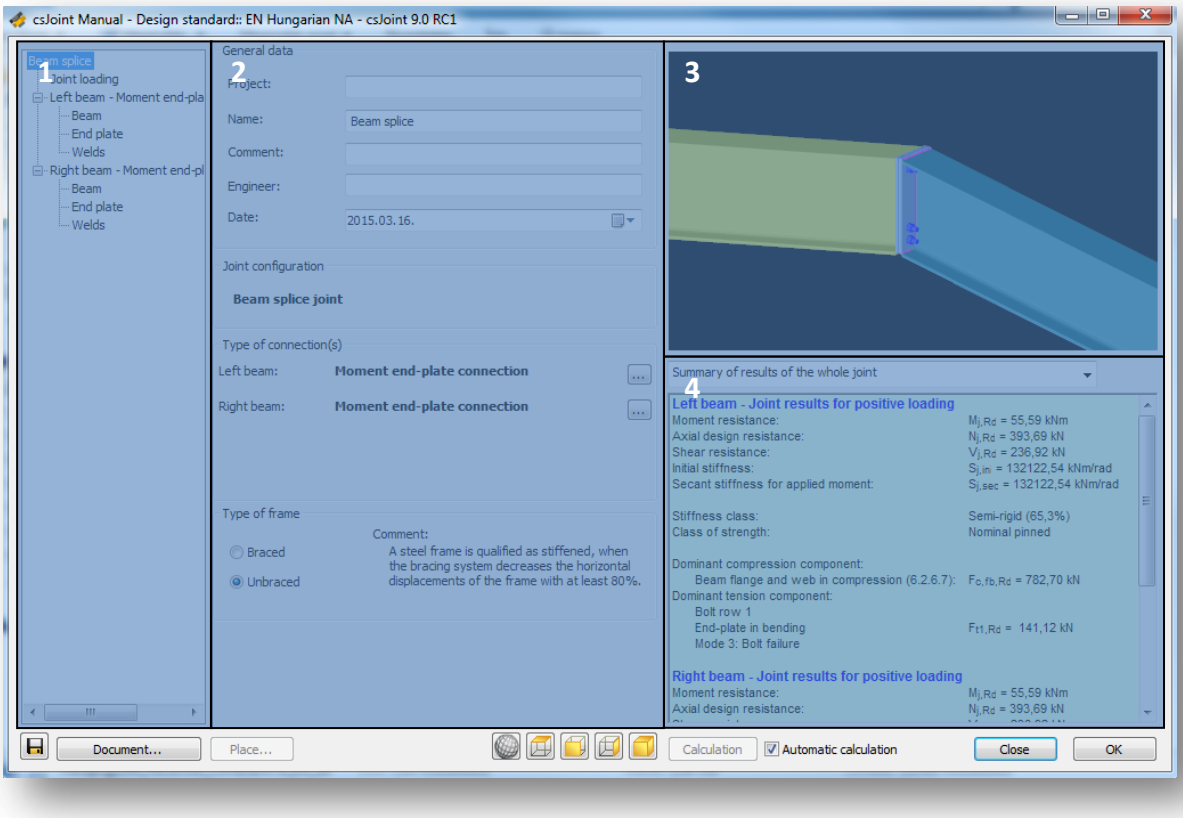


3. MANIPULATE DIFFERENT JOINT TYPES

3.1 MAIN WINDOW

No matter if a joint is created with our without model, manipulating the joint is the same. Joint details appear in a new window where numerous settings can be made. In the following pages we would like to give you an overview of the cs*Joint*, with showing all the possible adjustable parameters.

The *main window* consists of four separate parts.



- The #1 part shows the tree structure about the joint: all components of the connection can be seen and selected.

- On the #2 part, the selected component related information / properties can be seen and set: Joint name, project, engineer, date, etc. These properties will appear in the joint documentation and also useful to distinguish joints from each other.
- The #3 part is the graphical window
- The #4 part shows the results of the joint calculation. Calculation is automatically run when any changes take place and the ***AUTOMATIC CALCULATION*** checkbox is clicked. If the checkbox is unchecked the calculation can be run with the ***CALCULATION*** button. In the pull down menu the type of the result visualisation can be set to three different types: Summary for the whole joint, Summary for the selected connection, Detailed results of the selected connection. In the last two cases a connection must be chosen in the connection tree.

DOCUMENTATION can be generated by clicking the ***DOCUMENT…*** button on the bottom left corner of the dialog.

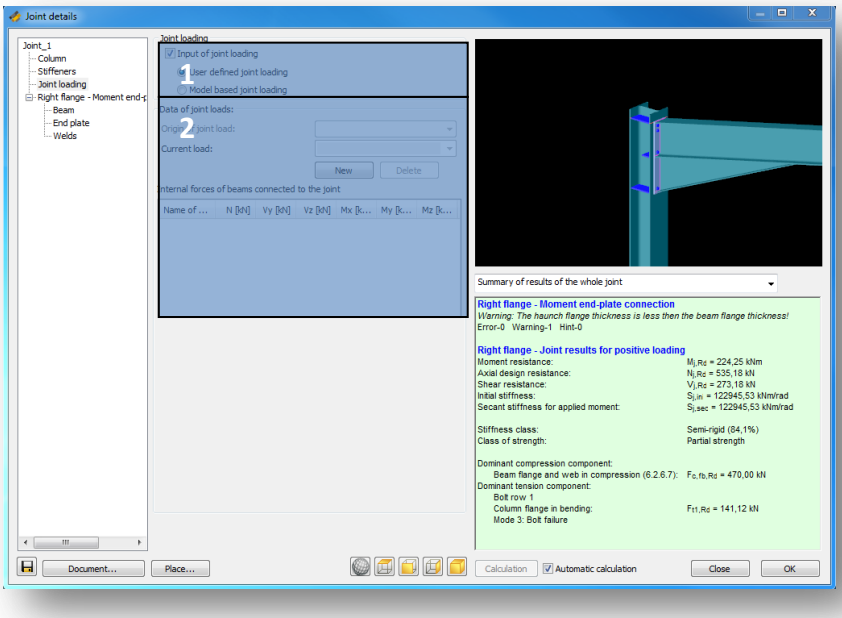
Furthermore, views of the joint can be changed, using the ***VIEW TOOLS,*** which can be find at the bottom of the dialog ().

3.2 PARTS OF THE TREE STRUCTURE

According to the chosen Joint configuration (moment end-plate beam to column, welded beam to column, etc.) certain elements of the tree structure will be different. All of the connection types can be modified separately, with the entire element parameters related to a connection (welds, end plate, etc.). In the following, all possible elements will be presented.

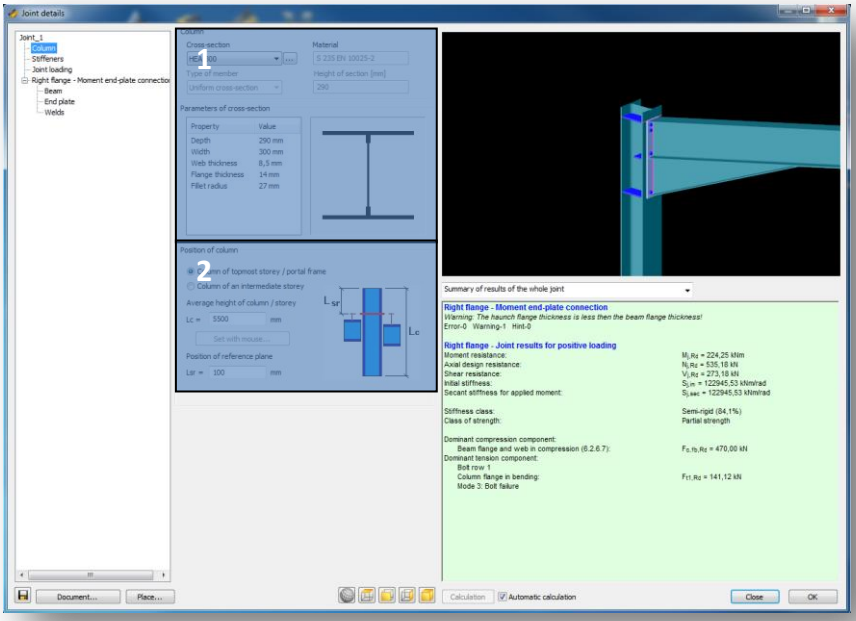
3.2.1 JOINT LOADING

If ***JOINT LOADING*** is selected, it is possible to choose from two options (#1). When *User defined joint loading* is selected, loads to the elements have to be given manually by defining load cases, and internal forces (#2) individually. If *Model based joint loading* is selected, *csJoint* automatically import the internal forces from all of the combinations after the joint is placed in the modelling area.



3.2.2 COLUMN

If the ***COLUMN*** is selected in the connection tree, then column related properties of the connection can be set like the section size and material grade (#1). Setting the position of the column is also very important (#2). Furthermore the position of the reference plane is essential for the joint calculation. The image on the right gives a visual help about the meaning of 'Lsr' and 'Lc' variables. Position of the reference plane can be changed.



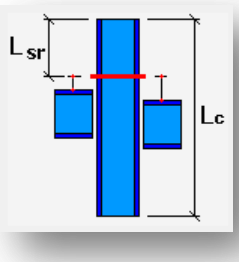
- Parameters of cross section (#1)

  Cross section can be changed by clicking to the dropdown menu. More cross sections can be loaded by clicking to the three dots button beside of the dropdown menu. Material can be changed by loading a new section with proper material. Parameters and a graphic display of the cross section been shown here.

- Position of column and reference plane (#2)

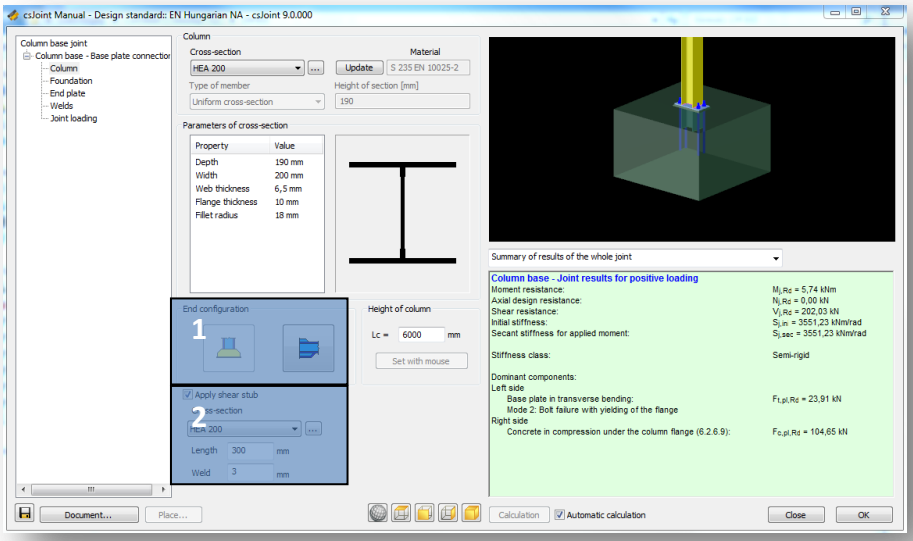
  It can be decided if the column is a topmost storey column/portal frame, or it is an intermediate storey column. The average height of the column/storey (Lc) can be set, by default it is 6000 mm.

  Position of the reference plane can be set, by default it is 100mm (Lsr). Reference plane is displayed in the graphical windows too,



with a transverse red line on the column. The value given to Lsr means the edge of the column's distance from the reference line.

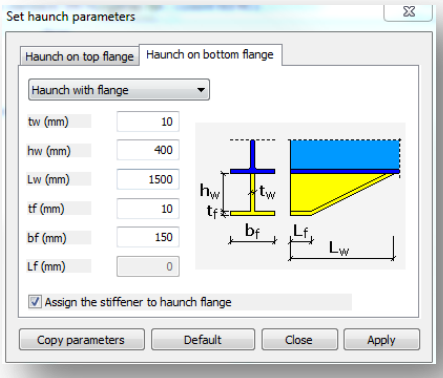
*In case of Column base joint, (see chapter ***3.3.4 CREATE COLUMN BASE JOINT***) additional Haunch, and rib can be added to the end of the column. Parameters of the wing plate can also be modified (#1). At #2 on the picture below additional shear stub can be applied to the column, and cross section, length end weld size can be configured.



- Parameters of wing plate and rib (#1)

As the illustration on the right side of the dialog shows, the following parameters can be set for wing plates and rib. The Height (hs), thickness (ts, tb), and the gap between the edges of the wing, and base plate (ds).

- Haunch (#1)

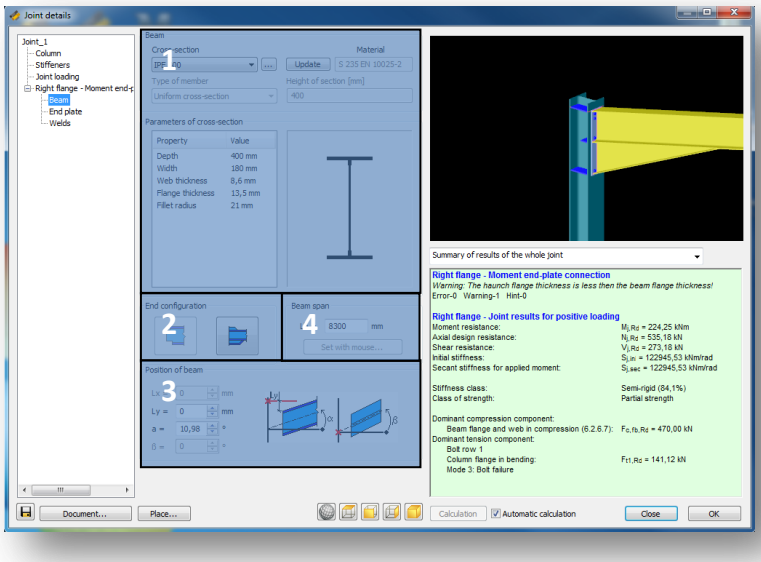


It is possible to define Haunches, both on the top and on the bottom flange. For better understanding, an illustration of the haunch is attached on the right side of the dialog. With ***COPY PARAMETERS***, defined settings can be applied to a haunch on the top/bottom flange.

- Shear stub (#2)
  In case of a column base joint with base plate, additional shear stub can be added, to increase the connections shear resistance. After turning on the tickbox, cross section, length and weld size of the stub can be defined.

3.2.3 BEAM

When ***BEAM*** is selected in the connection tree, the beam related properties of the connection can be set, which are the following.



- Parameters of cross section #1

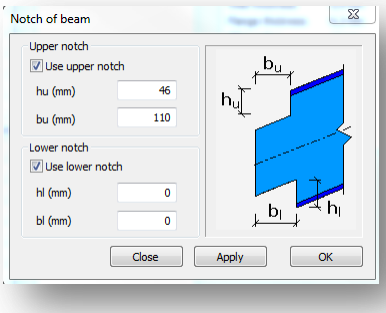
  Cross section can be changed by clicking to the dropdown menu. More cross sections can be loaded by clicking to the ***THREE DOTS BUTTON*** beside of the dropdown menu. ***MATERIAL*** button is only for end plates, finplates, splice plates. For a beams it can only be changed by loading a new section with proper material, by clicking the **…** button. Parameters and a graphic display of the cross section been shown here.

- *Set haunch parameters #2*:

  It works the same as it is been shown in ***3.2.2 COLUMN.*** If a lower transverse plate stiffener is applied to the columns bottom flange, it can be assigned it to the haunch flange by clicking the checkbox at the bottom of the dialog.

- *Notch of beam #2*:

  Upper and lower notches can be used simultaneously, or individually. Illustration for setting parameters can be found on the right side of the dialog.



- Position of beam #3

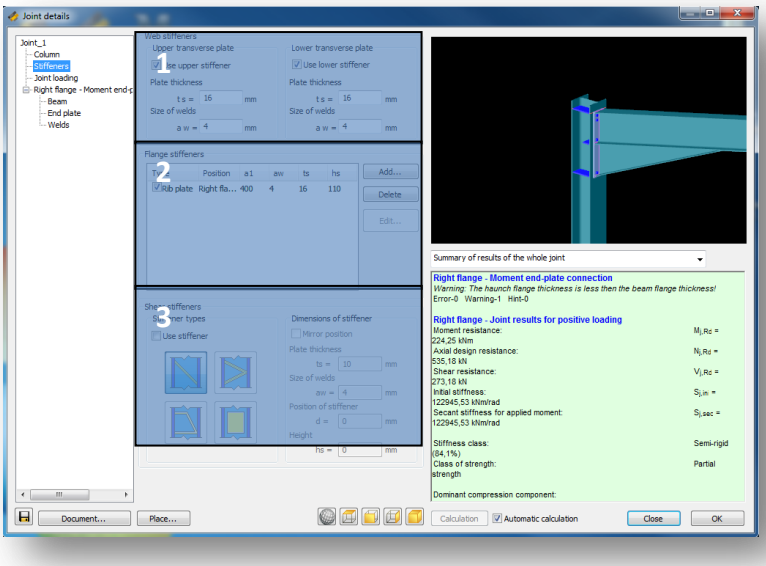
  Pitch of the beam can be set here both vertically, and horizontally, as the illustration shows.

- Beam span #4

  Span of the beam can be set. The default value is 6000mm. In ***CONSTEEL***, it can be set with mouse on the 3d model.

3.2.4 STIFFENERS

If the stiffeners is selected in the joint tree structure, different type of stiffeners can be added to the joint. It is possible to use:



- web stiffeners (#1)

  Upper or lower transverse plate can be added to the joint by clicking the proper checkbox. Plate thickness, and the size of the welds can be set for each stiffener individually.

- Flange stiffeners (#2)

  Flange stiffeners can be added by the ***ADD…*** button. By clicking it, the Stiffener of column flange dialog will pop up. Here it is possible to set the flange stiffener type, the place of the stiffener (right or left flange), plate thickness, length, and weld size. Distance from the reference plane can also be set here. Reference plane is displayed on the graphical window with a transverse red line on the column. The reference plane of the column can be set, if the ***COLUMN*** (***3.2.2***) is clicked in the joint tree. If flange stiffeners are given, they can easily removed with the checkboxes in the table, or they can be deleted with the ***DELETE*** button.

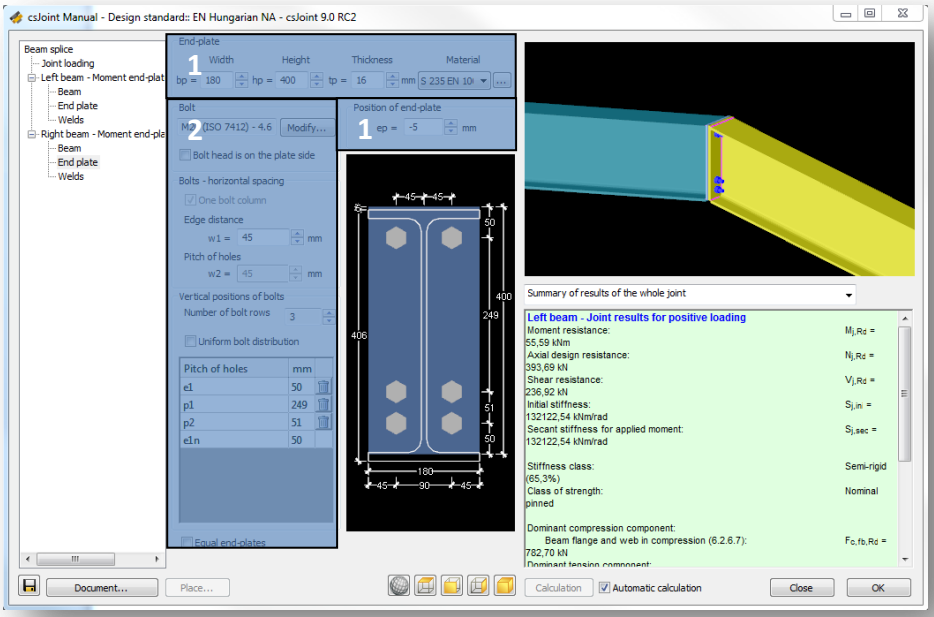
- Shear stiffeners (#3)

  Shear stiffeners can be used by ticking the Use stiffener checkbox. Four different shear stiffeners are available in csJoint. Beside the option of setting the plate thickness and

weld size, it is possible to mirror the position of the stiffener by ticking ***MIRROR POSITION*** checkbox.

3.2.5 END PLATE/BASE PLATE

When ***END PLATE*** is selected in the connection tree, the end plate related properties of the connection can be set, which are the following.

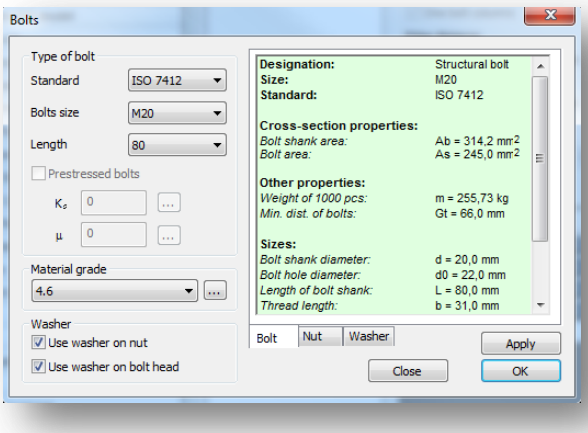


- End plate properties (#1)

  At #1 on the picture above, end plate geometry can be set. Predefined materials can be chosen in the dropdown menu, or user defined materials can be defined on the dialog appears if clicking the **…** button. Position of end plate can be set here, the default value is -5mm because of the default size of flange weld.

- Bolt properties (#2)

  By clicking ***MODIFY…*** the parameters can be changed for the bolts on the dialog. Beside the size, length and standard of the bolt material can be changed too. Prestressed bolts can be used in certain joint types. Details of



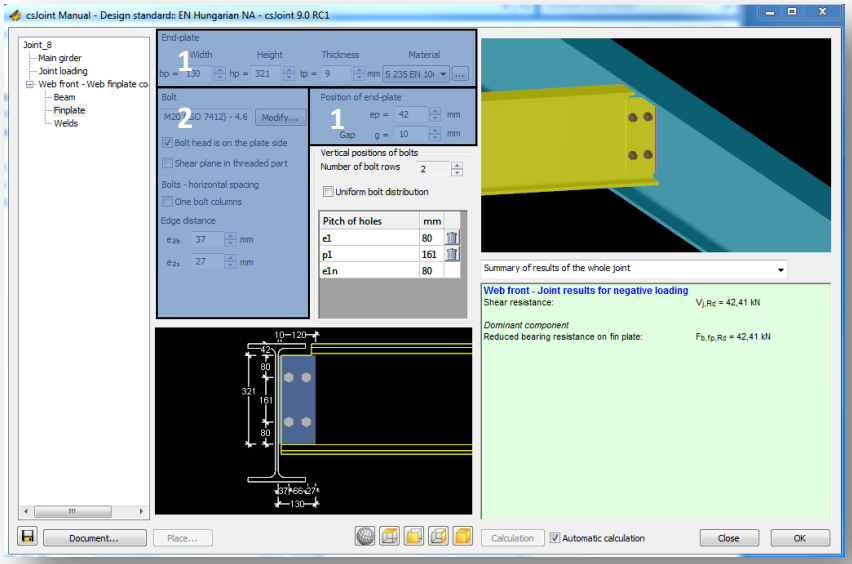
the chosen bolt configuration can be checked for the bolt, nut, and the washer too.

The spacing of the bolts can be set easily by giving the desired parameters (edge distance, pitch distance). Additional bolt rows can be added or removed by setting the number of the bolt rows. Pitch of holes can be given manually in table form, or Uniform bolt distribution can be chosen.

*In case of beam splice connection (with end plates on both beams), there is a checkbox under the patch of holes table for ***EQUAL END PLATES***. It is useful when both end plates have similar geometric parameters.

3.2.6 FINPLATE

When ***FINPLATE*** is selected in the connection tree, the finplate related properties of the connection can be set, which are the following.

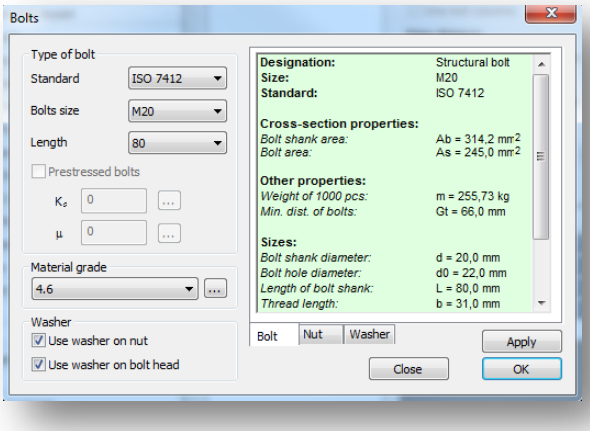


- Finplate properties (#1)

  At #1 on the picture above, end plate geometry can be set. Predefined materials can be choosen in the dropdown menu, or user defined materials can be defined on the dialog appears if clicking the ***THREE DOTS BUTTON***. The position of the finplate and the gap (between the flanges in a beam-to-beam joint) can also be set here.

- Bolt properties and geometry (#2)

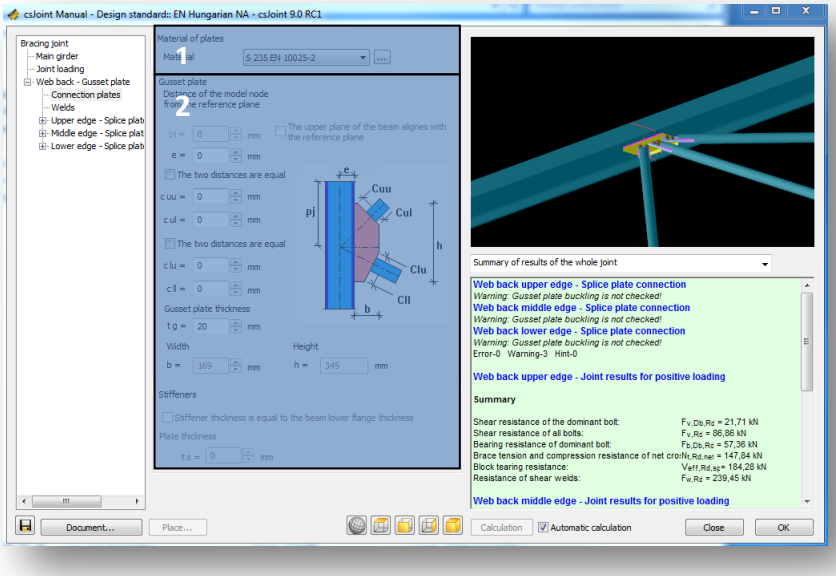
  By clicking ***MODIFY…*** the parameters can be changed for the bolts on the dialog. Beside the size, length and standard of the bolt material can be changed too. Prestressed bolts can be used in certain joint types. Details of the chosen bolt configuration can be checked for the bolt, nut, and the washer too.



The spacing of the bolts can be set easily by giving the desired parameters (edge distance, pitch distance). Additional bolt rows can be added (maximum of 10) or removed by setting the number of the bolt rows. Pitch of holes can be given manually in table form, or Uniform bolt distribution can be chosen. Number of bolt columns can be set to 1 by clicking the tickbox.

3.2.7 CONNECTION PLATES

When ***CONNECTION PLATES*** is selected in the connection tree, the connection plates related properties can be set, which are the following.

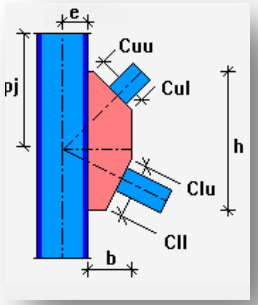


- Gusset plate Material (#1)

  Predefined materials can be chosen in the dropdown menu, or user defined materials can be defined on the dialog appears if clicking the ***THREE DOTS*** button.

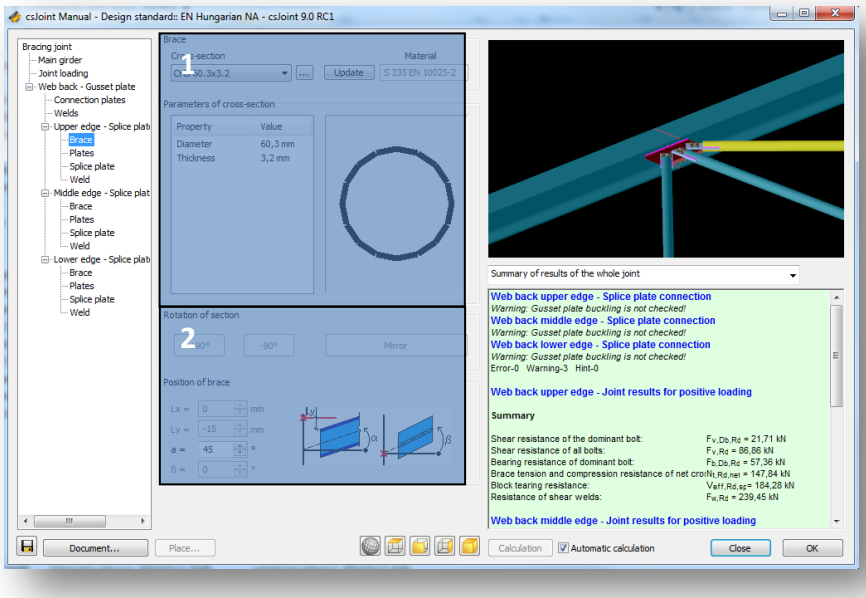
- Gusset plate geometric parameters (#2)

  Reference plane is a transverse plane on the main girder, in the node of the main girder's and the brace's axis. Value "e" is measured from here. Values Cu, Cuu, Cl, Cll are measured from the side of the brace section. The width and height of the gusset plate is calculated automatically with the given parameters. Illustration helps the better understanding of the values.



3.2.8 BRACE

When ***BRACE*** is selected in the connection tree, the brace related properties of the connection can be set, which are the following.



- Parameters of cross section #1

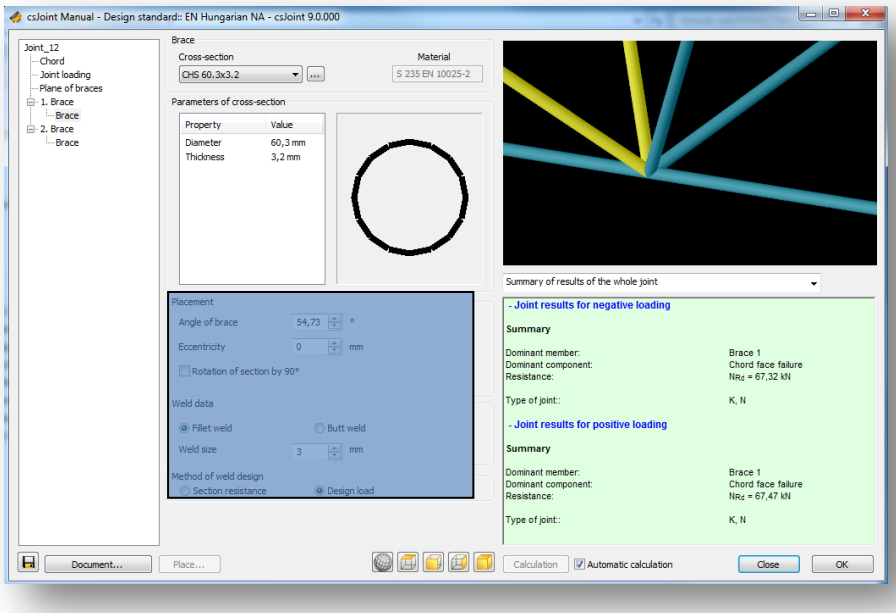
  Cross section can be changed by clicking to the dropdown menu. More cross sections can be loaded by clicking to the ***THREE DOTS BUTTON*** beside of the dropdown menu. ***MATERIAL*** button is only for end plates, finplates, and splice plates. For a beam it can only be changed by loading a new section with proper material, by clicking the **…** button. Parameters and a graphic display of the cross section been shown here.

  As a brace element, Hollow, Tube, L and round bar sections can be used.

- Rotate and Position of brace #2

  Pitch of the selected brace can be set here by setting the α value. Sections can also be rotated with +-90 degrees, using the rotation function.

In case of hollow section joints, the following dialog can be seen:

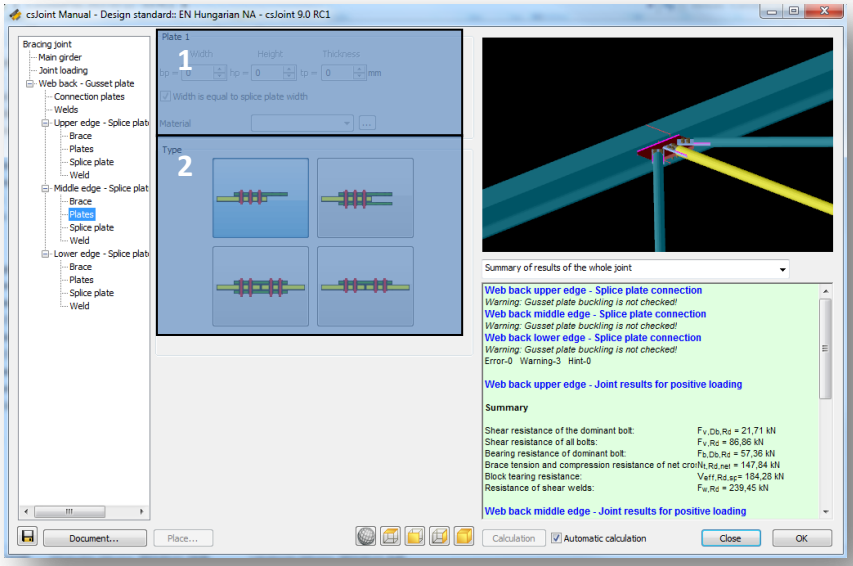


Placement of the brace members covers the following adjustable parameters (picture above):

- Angle of brace: angle between the chord and the brace
- Eccentricity of the braces
- Weld data: Fillet or butt weld and method of weld design (section resistance or design load) can be set.

3.2.9 PLATES

When ***PLATES*** is selected in the connection tree, the plates related properties of the connection can be set, which are the following.



- Parameters of the plate (#1)

  Geometric and material parameters can be set for the plate. Predefined materials can be chosen in the dropdown menu, or user defined materials can be defined on the dialog appears if clicking the ***THREE DOTS BUTTON***. Width of the plate can be equated with the splice plate's width.

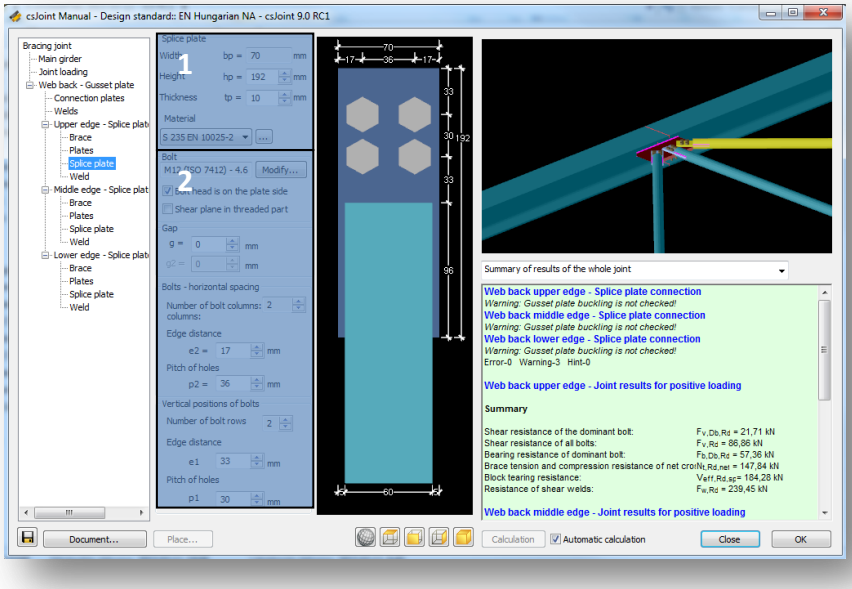
- Type of the connection (#2)

  At #2 part on the picture, it is possible to choose the type of the plate component. The following types can be chosen:

  - One side, simple splice plate
  - One side, double splice plate
  - Two sides, double splice plate
  - Two sides, simple splice plate

3.2.10 SPLICE PLATE

When ***SPLICE PLATES*** is selected in the connection tree, the splice plates related properties of the connection can be set, which are the following.

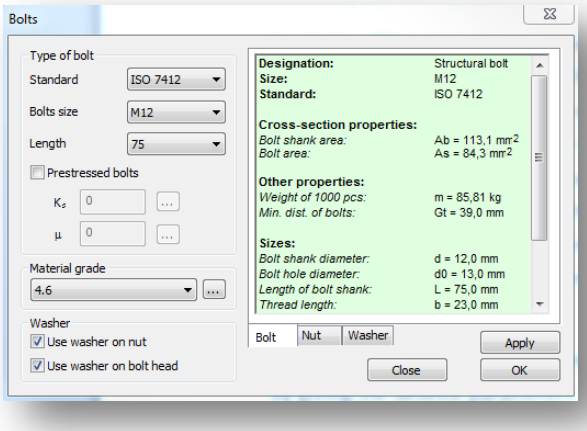


- Splice plate parameters (#1)

  At #1 on the picture above, splice plate geometry can be set. Predefined materials can be chosen in the dropdown menu, or user defined materials can be defined on the dialog appears if clicking the ***THREE DOTS*** button. Position of end plate can be set here; the default value is -5mm because of the default size of flange weld.

- Bolt properties (#2)

  By clicking ***MODIFY…*** the parameters can be changed for the bolts on the dialog. Beside the size, length and standard of the bolt material can be changed too. Prestressed bolts can be used in certain joint types. Details of the chosen bolt configuration can be checked for the bolt, nut, and the washer too.



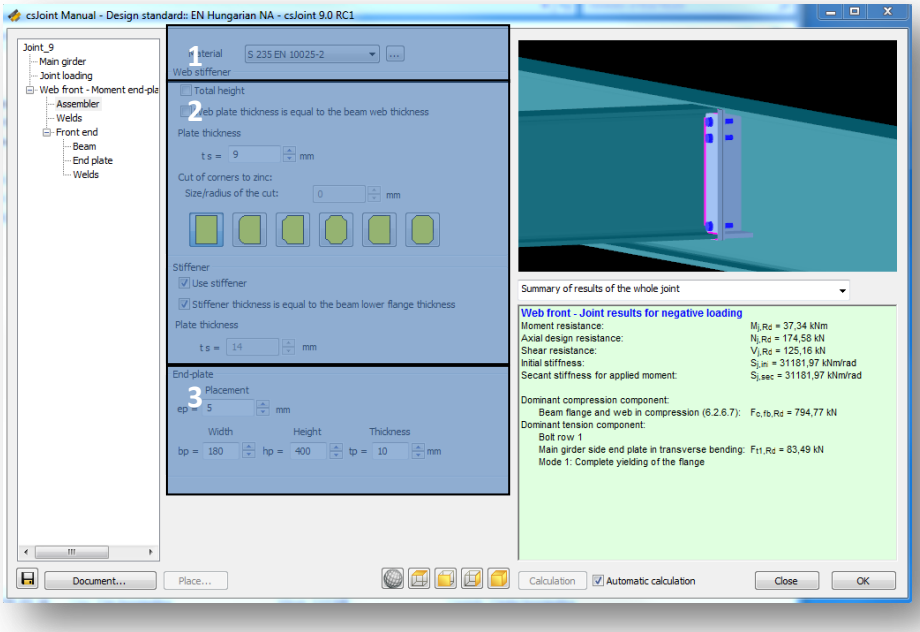
  In case of splice plate connections,

prestressed bolts can be used by ticking the checkbox, and setting the Ks factor and the μ slip factor. Standard values of the factors can be chosen in the pop-up table by clicking the ***THREE DOTS BUTTON.***

The spacing of the bolts can be set easily by giving the desired parameters (edge distance, pitch distance). Additional bolt rows and columns can be added or removed by setting the number of the bolt rows. Pitch of holes can be given manually in table form, or Uniform bolt distribution can be chosen.

3.2.11 ASSEMBLER

When ***ASSEMBLER*** is selected in the connection tree, the Assembler plate related properties of the connection can be set, which are the following.



- Material set (#1)

  Predefined materials can be chosen in the dropdown menu, or user defined materials can be defined on the dialog appears if clicking the ***THREE DOTS*** button.

- Stiffeners (#2)

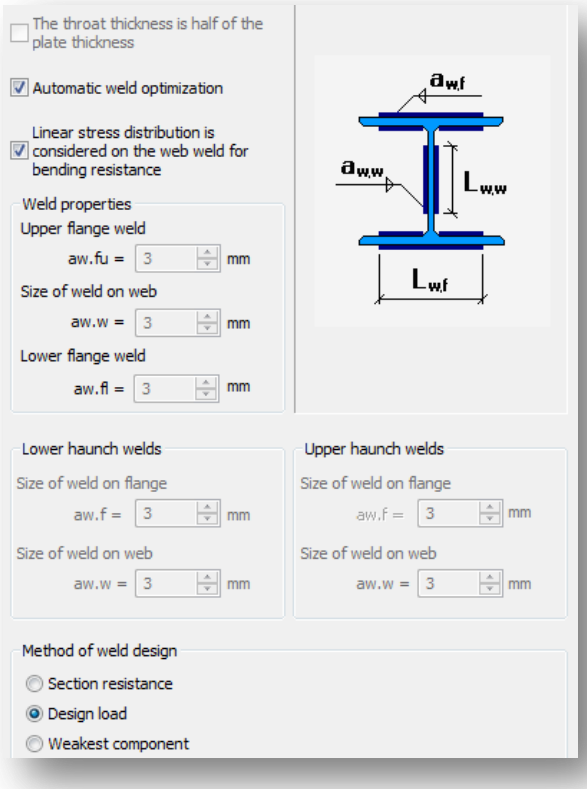
  Stiffeners can be placed here. Web stiffener can be applied in total height, or beam height, stiffener thicknesses can be also adjusted.

- End-plate properties (#3)

beyond the geometric parameters of the end plate (width height and thickness) placement can be adjusted.

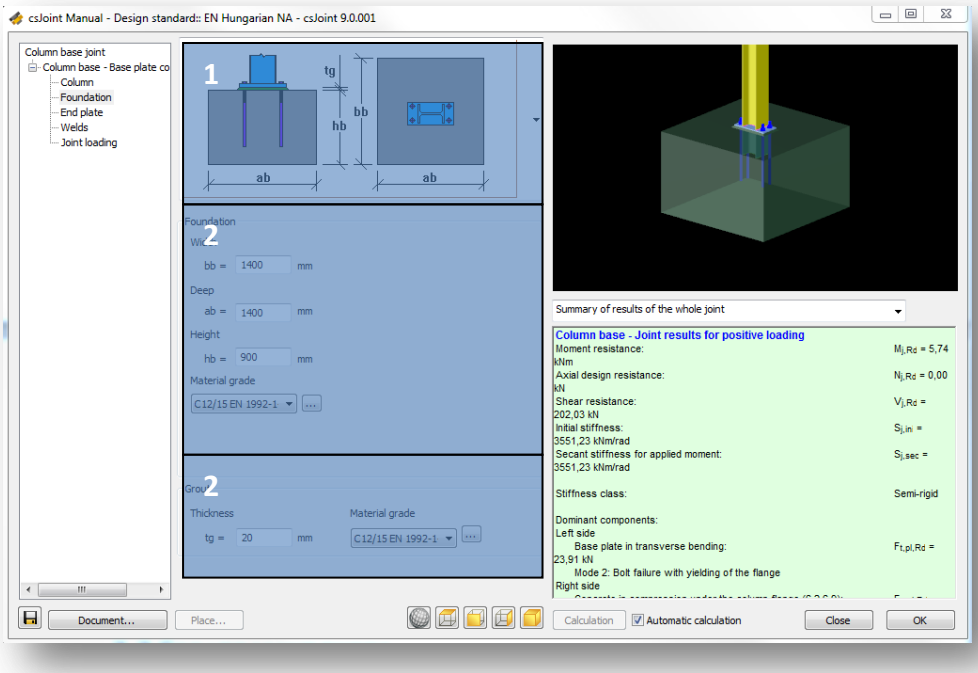
3.2.12 WELDS

When ***WELDS*** is selected in the connection tree, the welds related properties can be set. Design resistance is determined using the Directional method given in EN 1993-1-8.



- If automatic weld optimization is checked in, weld sizes are geared to the stresses. If not, weld size values can be set manually.
- At the bottom, method can be chosen, if the welds should be designed for section resistance, design load, or for the weakest component.

3.2.13 FOUNDATION

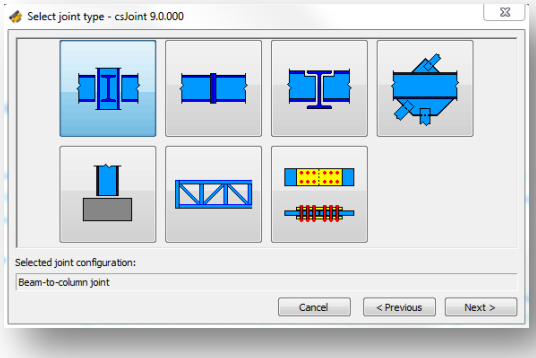


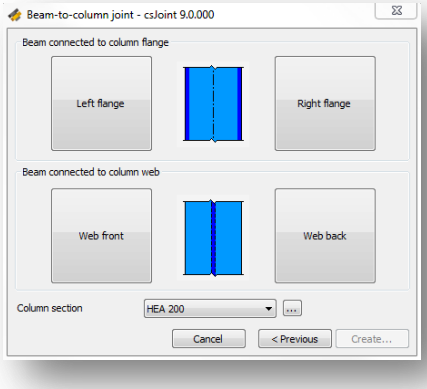
- At the #1 part of the picture above, illustration about the editable variables can be seen
- At part #2 dimensions of the foundation can be modified. Concrete materials can be chosen from the dropdown menu. By clicking the ***THREE DOTS BUTTON***, additional materials can set.
- At part #3, parameters of the grouting can be set, like dimensions, and material.

3.3 CREATE JOINT TYPES

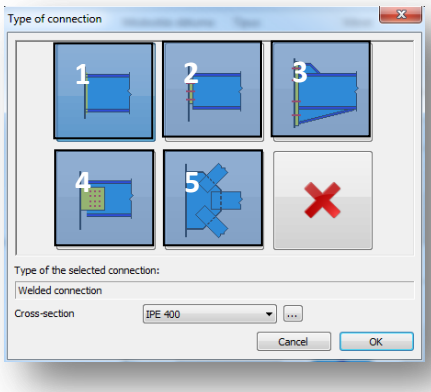
3.3.1 CREATE BEAM-TO-COLUMN JOINT

To create a beam-to-column joint at first the ***BEAM-TO-COLUMN JOINT*** type has to be chosen on the ***SELECT JOINT TYPE*** dialog. Next step is to choose the necessary connection type for the proper place. Connection can be placed on the left and right flange and the sides of the web of the column. Maximum of four connections can be placed on a beam-to-column joint. Section of the column has to be set on the ***BEAM-TO-COLUMN JOINT*** dialog (right picture below).





The types of the possible connections on the web and the flanges are:



- Welded connection on flanges (#1)

  Welded connections can be placed on the flanges. Section of the beam can be defined in ***TYPE OF CONNECTION*** dialog. After accepting the welded connection on flanges, and

creation of the Joint, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Column (see chapter ***3.2.2 COLUMN***)
- Stiffeners (see chapter ***3.2.4 STIFFENERS***)
- Beam (see chapter ***3.2.3 BEAM***)
- Welds (see chapter ***3.2.12 WELDS***)

- Simple plate connection (#2)

Simple plate connections can be placed both on the flanges, and on the web. Section of the beam can be defined in ***TYPE OF CONNECTION*** dialog. After accepting the simple plate connection, and creation of the Joint, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Column (see chapter ***3.2.2 COLUMN***)
- Stiffeners (see chapter ***3.2.4 STIFFENERS***)
- Beam (see chapter ***3.2.3 BEAM***)
- End plate (see chapter ***3.2.5 END PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

- Moment end-plate connection on flange (#3)

Moment end-plate connections can be placed on the flanges. Section of the beam can be defined in ***TYPE OF CONNECTION*** dialog. After accepting the moment end-plate connection on flanges, and creation of the Joint, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Column (see chapter ***3.2.2 COLUMN***)
- Stiffeners (see chapter ***3.2.4 STIFFENERS***)
- Beam (see chapter ***3.2.3 BEAM***)
- End plate (see chapter ***3.2.5 END PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

- Finplate connection (#4)

Finplate connection can be placed both on the flanges, and the web. Section of the beam can be defined in ***TYPE OF CONNECTION*** dialog. After finplate is selected, beam section

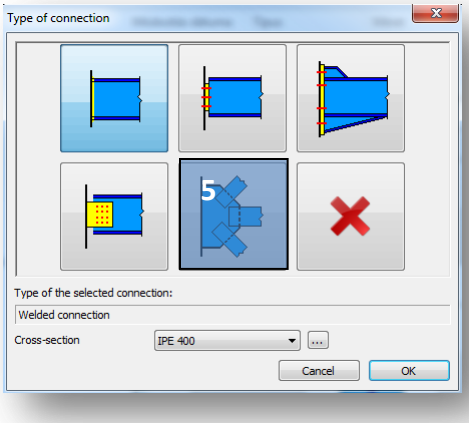
is set, and joint is created, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

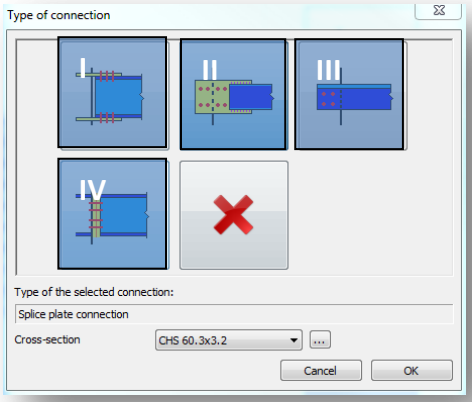
- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Column (see chapter ***3.2.2 COLUMN***)
- Stiffeners (see chapter ***3.2.4 STIFFENERS***)
- Beam (see chapter ***3.2.3 BEAM***)
- Finplate (see chapter ***3.2.6 FINPLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

- Gusset plate connection (#5)

Gusset plate connection can be placed both on the flanges, and the web. Section of the beam can be defined in ***TYPE OF CONNECTION*** dialog.

In case of beam to column gusset plate connection, more types of connections can be set for the gusset plate, which are the following:





- Double plate flange connection (#I)
- Splice plate connection (#II)

Tube, hollow and round bar sections can be used.

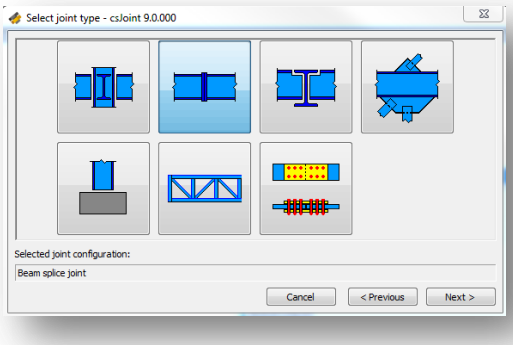
- Angle Brace connection (#III)
- Bolted end-plate connection (#IV)

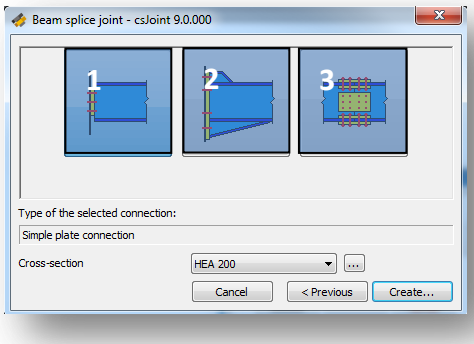
After clicking the Create button, joint will be generated according to the default joint settings, manipulation of the joint can be started to satisfy personal demands. The following connections can be selected for configuration (according to the chosen connections on the gusset plate):

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Column (see chapter ***3.2.2 COLUMN***)
- Stiffeners (see chapter ***3.2.4 STIFFENERS***)
- Beam (see chapter ***3.2.3 BEAM***)
- Splice plate (see chapter ***3.2.10 SPLICE PLATE***)
- Brace (see chapter ***3.2.8 BRACE***)
- Connection plates (see chapter ***3.2.7 CONNECTION PLATES***)
- Welds (see chapter ***3.2.12 WELDS***)

3.3.2 Create Beam splice Joint

To create a beam splice joint at first the ***BEAM SPLICE JOINT*** type has to be chosen on the ***SELECT JOINT TYPE*** dialog. Next step is to choose the necessary connection type.





- Simple plate beam splice connection (#1)

  Section of the beam's cross section can be set on the ***BEAM SPLICE JOINT*** dialog. If the proper section cannot be found in the dropdown menu, more sections can be loaded by clicking the ***THREE DOTS*** button (section can be modified later too). After simple plate connection is selected, beam section is set, and joint is created, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

  - Joint loading (see chapter ***3.2.1 JOINT LOADING***)
  - Beam (see chapter ***3.2.3 BEAM***)
  - End plate (see chapter ***3.25 END PLATE***)
  - Welds (see chapter ***3.2.12 WELDS***)

- Moment end-plate connection (#2)

Section of the beam's cross section can be set on the ***BEAM SPLICE JOINT*** dialog. If the proper section cannot be found in the dropdown menu, more sections can be loaded by clicking the ***THREE DOTS BUTTON*** (section can be modified later too). After moment end-plate connection is selected, beam section is set, and joint is created, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Beam (see chapter ***3.2.3 BEAM***)
- End plate (see chapter ***3.25 END PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

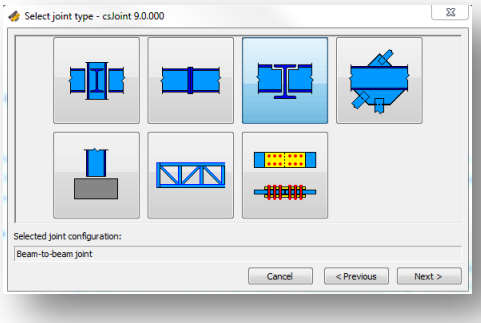
- Beam splice plate connection (#3)

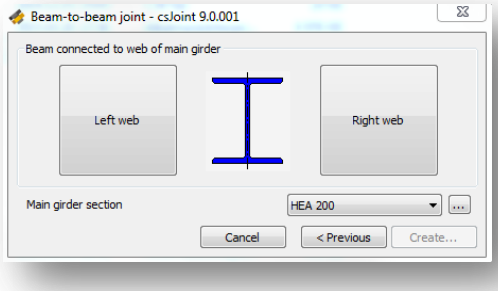
Section of the beam's cross section can be set on the ***BEAM SPLICE JOINT*** dialog. If the proper section cannot be found in the dropdown menu, more sections can be loaded by clicking the ***THREE DOTS BUTTON*** (section can be modified later too). After splice plate connection is selected, beam section is set, and joint is created, the following connections can be selected on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Beam (see chapter ***3.2.3 BEAM***)
- Splice plate (see chapter ***3.25 END PLATE***)

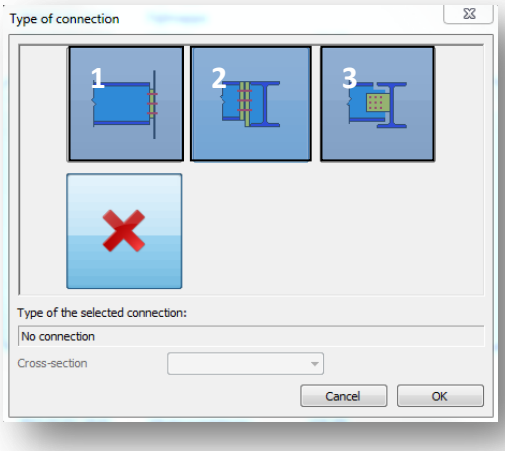
3.3.3 CREATE BEAM-TO-BEAM JOINT

To create a beam-to-beam joint at first the ***BEAM-TO-BEAM*** type has to be chosen on the ***SELECT JOINT TYPE*** dialog. Next step is to choose the necessary connection type for the proper place. Connection can be placed on the left and right flange. Maximum of two connections can be placed on a beam-to-beam joint. Section of the main girder can be chosen here, from the dropdown menu, and more sections can be loaded from the section administrator with the ***THREE DOTS BUTTON***.





After clicking the left and/or tight web (right picture above), three connection types can be chosen for a beam-to-beam joint, which are the following:



- Simple plate beam-to-beam connection (#1)

  Section of the beam can be set on the ***TYPE OF CONNECTION*** dialog. If the proper section cannot be found in the dropdown menu, more sections can be loaded by clicking the ***THREE DOTS BUTTON*** (section can be modified later too). After simple plate connection is selected, beam section is set, and joint is created, the following

connections can be selected and manipulated on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Beam (see chapter ***3.2.3 BEAM***)
- End plate (see chapter ***3.25 END PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)
- Assembler (see chapter ***3.2.11 ASSEMBLER***)

- Moment end-plate connection (#2)

Section of the beam can be set on the ***TYPE OF CONNECTION*** dialog. If the proper section cannot be find in the dropdown menu, more sections can be loaded by clicking the ***THREE DOTS BUTTON*** (section can be modified later too). After simple plate connection is selected, beam section is set, and joint is created, the following connections can be selected and manipulated on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Beam (see chapter ***3.2.3 BEAM***)
- End plate (see chapter ***3.25 END PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

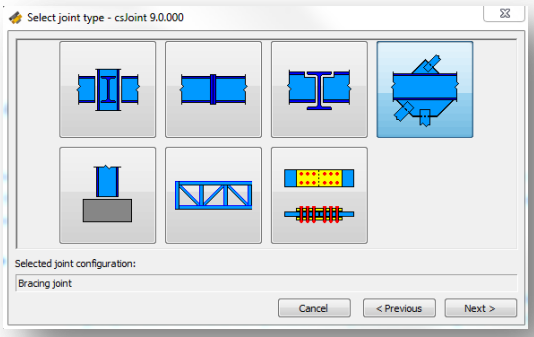
- Web finplate connection (#3)

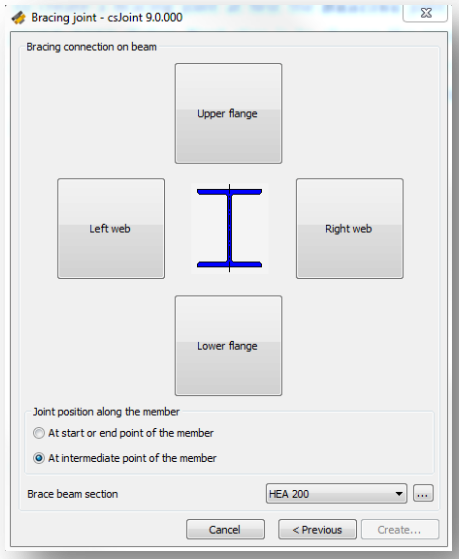
Section of the beam can be set on the ***TYPE OF CONNECTION*** dialog. If the proper section cannot be find in the dropdown menu, more sections can be loaded by clicking the ***THREE DOTS BUTTON*** (section can be modified later too). After simple plate connection is selected, beam section is set, and joint is created, the following connections can be selected and manipulated on the ***MAIN WINDOW***'s ***(3.1)*** connection tree:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Beam (see chapter ***3.2.3 BEAM***)
- Finplate (see chapter ***3.26 END PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

3.3.4 CREATE BRACING JOINT

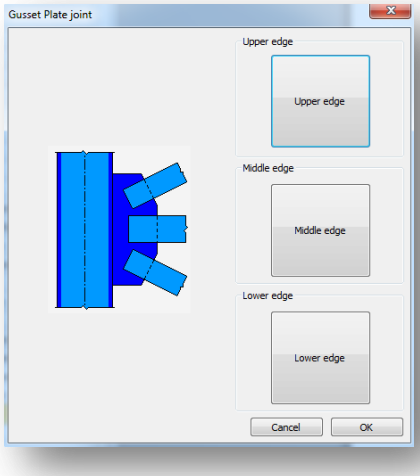
To create a Bracing joint at first the ***BRACING*** joint type has to be chosen on the ***SELECT JOINT TYPE*** dialog. Next step is to choose the necessary connection type for the proper place. Gusset plate connections can be placed on the upper and lower flange, and on the left and right side of the web. Maximum of four connections can be placed on a Bracing joint.

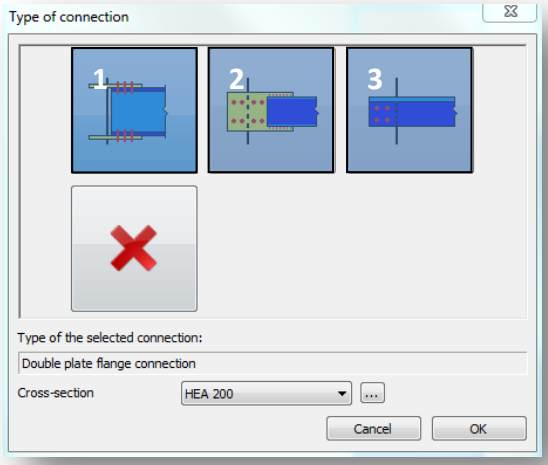




On the ***BRACING JOINT DIALOG*** (right picture above) it is have to be selected if the joint is positioned at the start/end point, or at an intermediate point of the member. The main girder's cross section can also be defined here. (it can be changed later, during the joint manipulation)

On a gusset plate, three connections can be set. The middle, which is always perpendicular to the main girder, and two more on the upper and lower edge which can settled in angle (angle can be set later if ***BRACE*** is selected in the connection tree)





- Double plate flange connection (#1)
- Splice plate connection (#2)
- Angle Brace connection (#3)

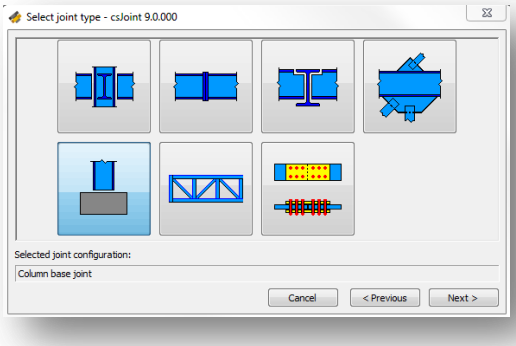
After clicking the Create button, joint will be generated according to the default joint settings, manipulation of the joint can be started to satisfy personal demands. The following connections can be selected for configuration (according to the chosen connections on the gusset plate):

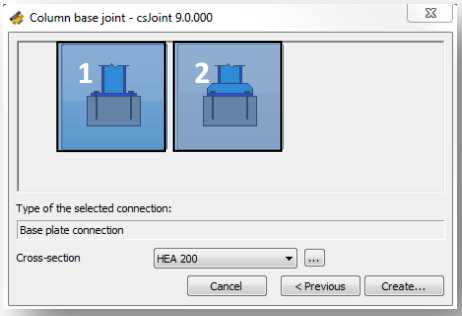
- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Splice plate (see chapter ***3.2.10 SPLICE PLATE***)

- Plates (see chapter ***3.2.9 PLATES***)
- Brace (see chapter ***3.2.8 BRACE***)
- Connection plates (see chapter ***3.2.7 CONNECTION PLATES***)
- Welds (see chapter ***3.2.12 WELDS***)

3.3.5 CREATE COLUMN BASE JOINT

To create a column base joint at first the ***COLUMN BASE*** joint type has to be chosen on the ***SELECT JOINT TYPE*** dialog (left picture below). Next step is to choose the type of the connection (right picture below). Column base joint can be created using Base plate connection (#1), or Rigid connection with ground-beam (#2). Cross section of the column can also be defined here by choosing it from the dropdown menu. (it can be changed later, during the joint manipulation). If the needed cross section cannot be find in the dropdown menu, more sections can be loaded from the section administrator, by clicking the ***THREE DOTS BUTTON***.





After clicking the Create button, joint will be generated automatically, according to the default joint settings, manipulation of the joint can be started to satisfy personal demands. The following connections can be selected for configuration (according to the selected connection type):

- Column (see chapter ***3.2.2 COLUMN***)

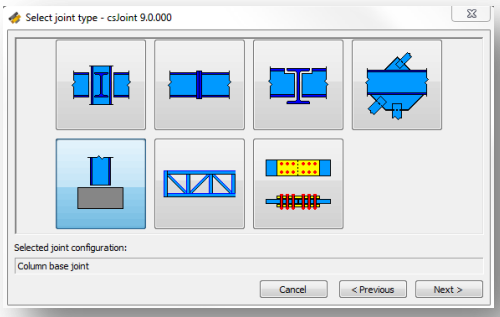
End configuration of the column can be modified with additional Haunch, and rib. See more in the related Column chapter.

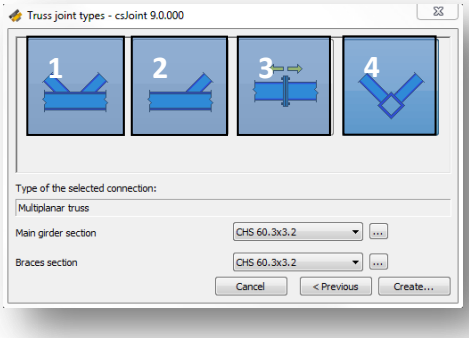
- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Foundation (see chapter ***3.2.4 FOUNDATION***)
- Base plate (see chapter ***3.2.5 END PLATE/ BASE PLATE***)
- Welds (see chapter ***3.2.12 WELDS***)

3.3.6 CREATE HOLLOW SECTION JOINT

To create a hollow section joint at first the ***HOLLOW SECTION*** joint type has to be chosen on the ***SELECT JOINT TYPE*** dialog. The following truss joint types are available in *csJoint*:

- Hollow section K and N joint (#1)
- Hollow section T and Y joint (#2)
- Tension chord splice (#3)
- Multiplanar truss (#4)





- T, Y, N, K and multiplanar truss connection (#1,#2 and #4)

  In the ***TRUSS JOINT TYPES DIALOG***, section of the main girder and the braces has to be defined. Sections can be loaded from the section administrator, by clicking the ***THREE DOTS BUTTON***. Loaded Sections can be chosen from the dropdown menu.

  In case of ***T, Y, N, K JOINTS***, for braces, hollow and tube sections can be used. For the main girder, I and H profiles are also allowed.

  For multiplanar truss joints, only hollow, and tube sections are allowed to choose.

  Sections can be modified later too, during the Joint manipulation.

  After creating the joint, the following connections can be selected in the structure tree:

  - Joint loading (see chapter ***3.2.1 JOINT LOADING***)
  - Brace (see chapter ***3.2.8 BRACE***)

- Tension chord splice connection (#3)

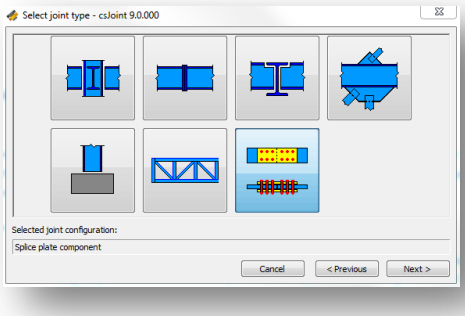
  At the ***TRUSS JOINT TYPES*** dialog, as for the main girder's section, hollow sections are allowed to choose. Loaded sections from the section administrator (***THREE DOTS BUTTON***) will appear in the dropdown menu. Sections can be modified during the joint manipulation too.

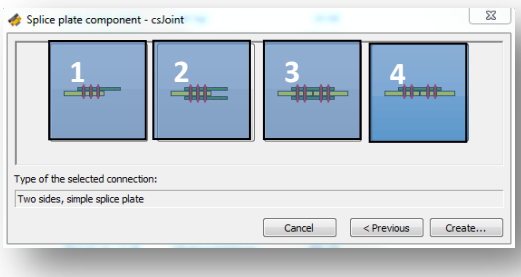
  After creating the joint, the following connections can be selected in the structure tree:

  - Joint loading (see chapter ***3.2.1 JOINT LOADING***)
  - Brace (see chapter ***3.2.8 BRACE***)
  - End plate (see chapter ***3.2.5 END PLATE***)

3.3.7 CREATE SPLICE PLATE COMPONENT

Single splice plate components can be designed using the ***SPLICE PLATE COMPONENT*** button at ***SELECT JOINT TYPE*** dialog (left picture below). After clicking the next button, type of the selected connection has to be chosen. Four types of splice plate connections can be picked, which are the following (right picture below):





- one side, simple splice plate
- one side, double splice plate
- two sides, double splice plate
- two sides, simple splice plate

After the creation of a splice plate component, the parts of the tree structure on the ***MAIN WINDOW*** (***3.1***) are the following:

- Joint loading (see chapter ***3.2.1 JOINT LOADING***)
- Plates (see chapter ***3.2.9 PLATES***)
- Splice plate (see chapter ***3.2.10 SPLICE PLATE***)

In the pull down menu the type of the result visualisation can be set to three different types: Summary for the whole joint, Summary for the selected connection, Detailed results of the selected connection. In the last two cases a connection must be chosen in the connection tree.

The result of the joint calculation can be seen in the highlighted part of the window. If the detailed result is selected then all the results of the necessary calculations can be seen according to the selected load combination or according to the dominant load case.

3.3.8 DESIGN OF PAD FOUNDATION

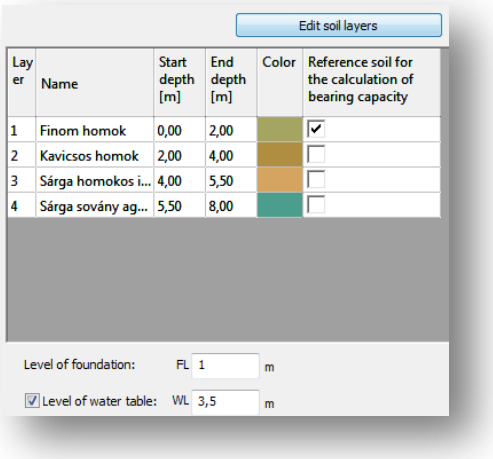
From the version of ConSteel 9, and csJoint 9, it is possible to design and document pad foundations, giving a new tool to the hands of the engineers. Design of pad foundations was implemented to the ***COLUMN BASE JOINT (3.3.4)*** type.

Spread foundation design covers the following examinations:

- Bearing resistance
- Sliding resistance
- Total settlement

Elements of the Design of spread foundations tree structure:

- Soil layers

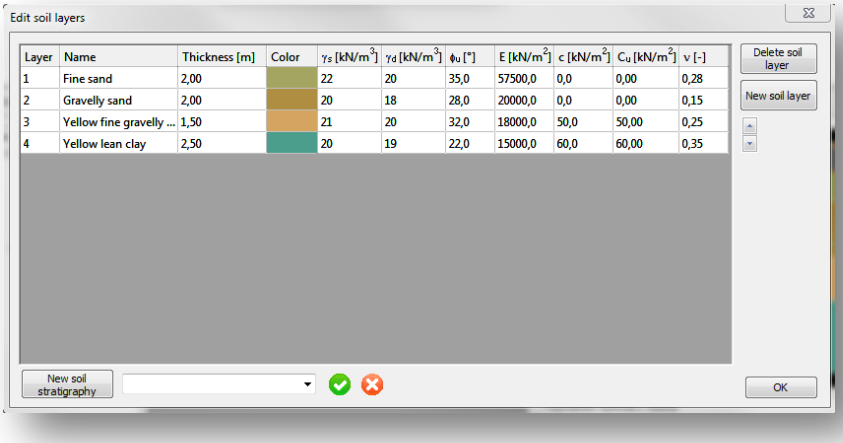


The following settings can be made on this dialog:

It is possible to choose with the checkbox, which soil layer should be used as a reference soil for the calculation of bearing capacity (picture above).

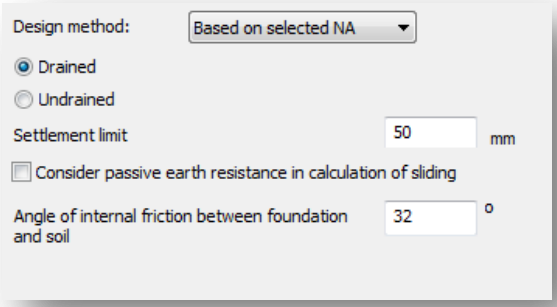
Level of the foundation and water can also be set on this dialog.

New soil layers can be added or deleted from the actual soil stratigraphy, or completely new stratigraphy can be made, using the *New Soil Stratigraphy* button, after clicking the ***EDIT SOIL LAYERS*** button (picture below).



The user defined stratigraphies are saved to the Documents/ConSteel/Foundation folder in **.CSM** files format.

- Design settings

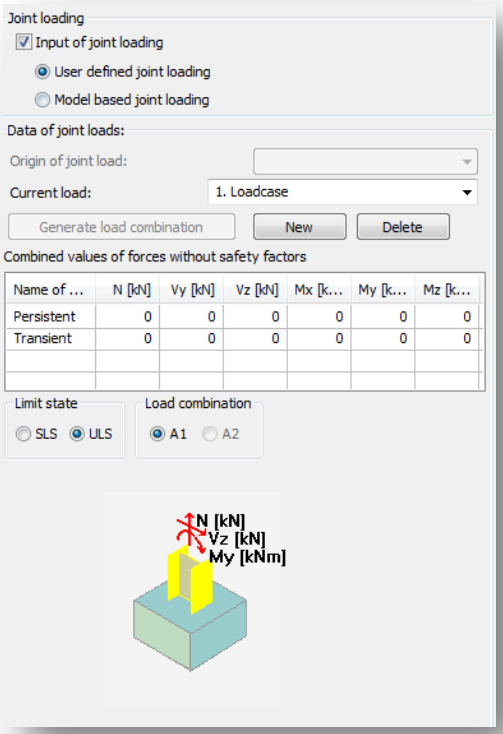


By default, according to the chosen National Annex, the program automatically decides which Design Approach should be used, but it can be chosen manually too.

Drained or undrained circumstances, settlement limit and angle of internal friction between foundation and soil can be set too on the dialog.

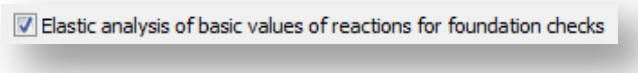
It is has to be chosen if the passive earth resistance should be taken into consideration under the calculation.

- Loads of foundation



There are two ways of load input to a Joint model. Loads can be imported from the model (only in ***ConSteel***), or it can be defined manually (***ConSteel & csJoint***).

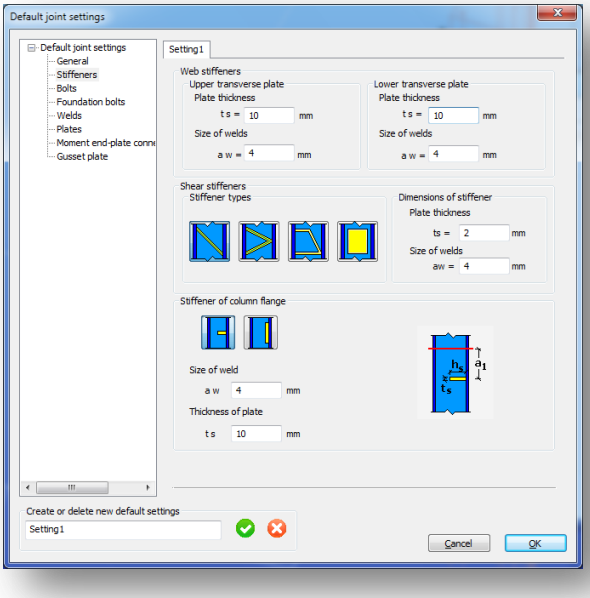
- When using ConSteel and loads wanted to be loaded from the model, a new setting has to be made on the Analysis panel. If the checkbox is ticked in (picture below), ***ConSteel*** will generate the necessary reaction forces for foundation checks (picture below). After Analysis run, these reactions can be seen on the Analysis results as well as in the Joint modules ***Joint loading*** dialog.



- When using ***csJoint***, the only option is to define the loads of the joint manually. After *User defined joint loading* is selected, load can be defined by clicking the *New* button. For each loadcase, the limit state has to be set if it is SLS or ULS, since ULS combinations are used to the Bearing and Sliding, and SLS combinations are used to settlement calculations. When defining a load, the illustration on the bottom of the dialog helps define directions correctly.

3.3.9 DEFAULT JOINT SETTINGS

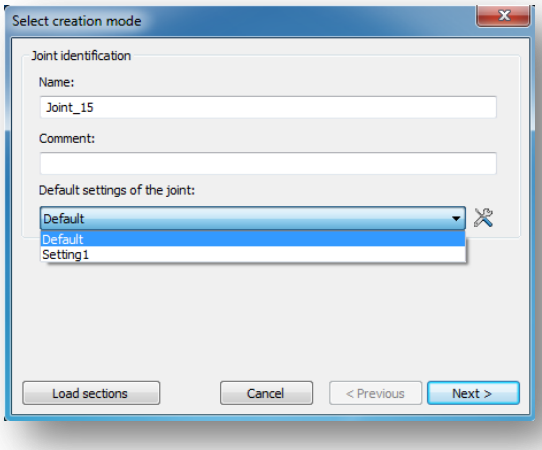
The default joint setting like weld sizes, bolt diameter, etc. can be changed and saved.



The default joint settings like weld sizes, bolt diameters, etc. can be changed and saved by clicking the  icon.

The default joint configuration file is saved to the Documents\ConSteel folder as UserConfig.xml.

When creating a new joint manually, user defined default joint setting can be chosen in the dropdown menu (see the picture below).



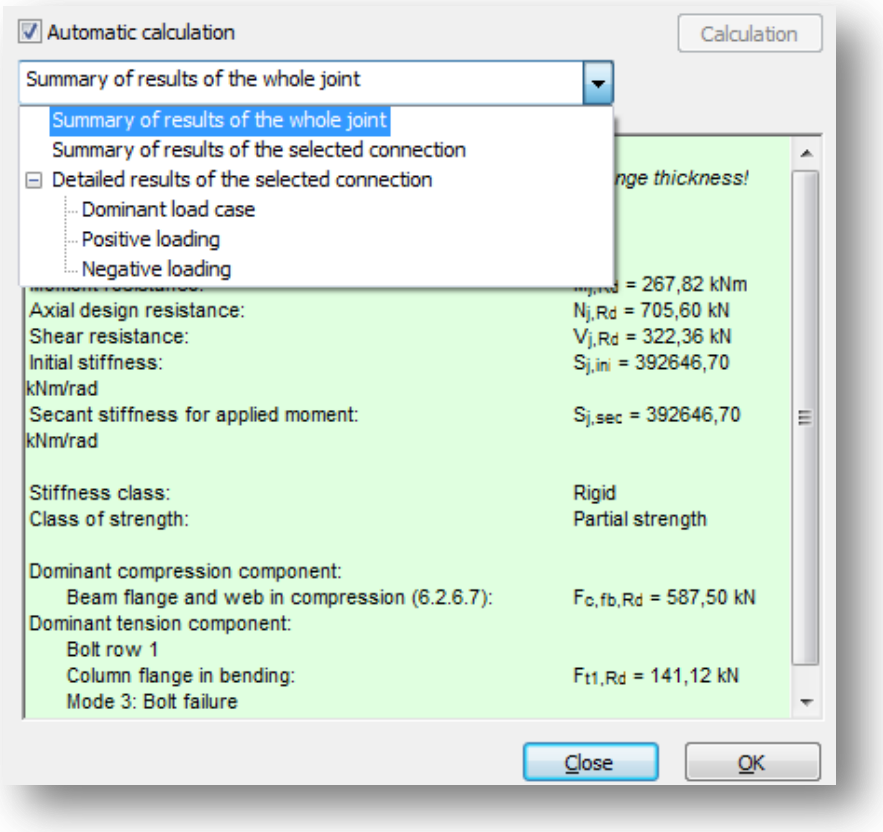
4 ANALYSIS OF CONNECTIONS

All the analysis in *CSJOINT* module is based on the standard procedures of Eurocode 3 Part 1-8, these procedures are almost entirely covered by the module. For the different types of connections the following analyses are performed:

- Moment connections:
  - Moment resistance based on the plastic tension components of the individual or grouped bolt rows and compression components including the effect of axial load
  - Shear resistance of the bolts in combined shear and tension
  - Bearing resistance of plates
  - Web and flange weld resistance and capacity to the applied load, the section strength or the appropriate component
  - Initial and secant stiffness based on the tension and compression components and applied moment
- Shear connections:
  - Shear resistance of bolts and plates
  - Bearing resistance of plates
  - Block tearing resistance of plates
  - Weld resistance and capacity to the applied load or section strength
- Hollow section connections:
  - Chord face failure, web failure, shear failure and punching shear failure
  - Brace failure
  - Local buckling of the members

5. RESULTS AND DOCUMENTATION

The analysis results are displayed in two main forms: a summarized view containing the main resistances, stiffness and capacities; and a detailed view showing the components of the main results which is comprehensive enough to see what are the weakest point of the connections and what type of strengthening would be the most efficient to apply. A joint is calculated for all the loads coming from different places or different combinations, and the dominant place and combination is automatically highlighted. The spectacular documentation shows the detailed geometry of the joint with its connections, and the summarized and detailed results for all cases can be flexibly documented.



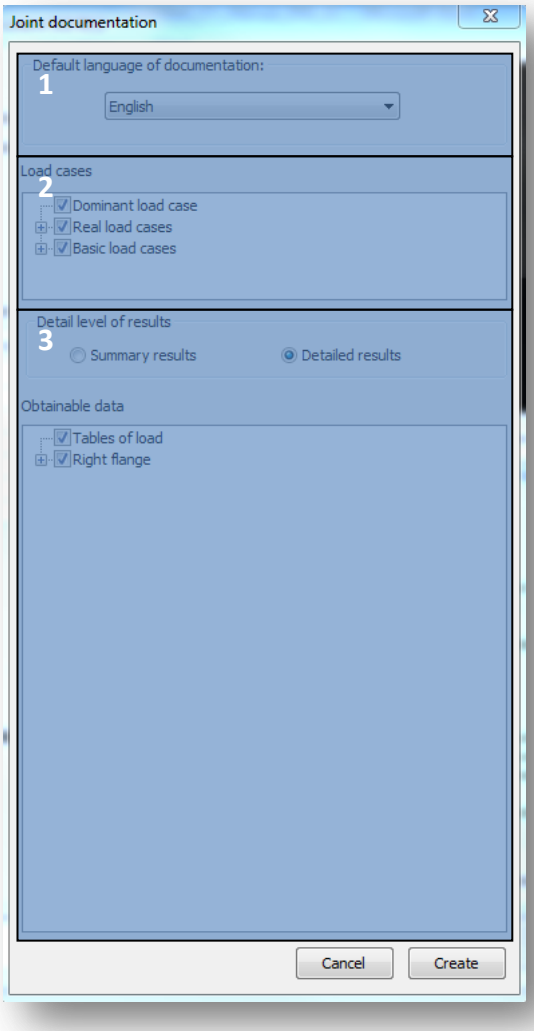
5.1 DOCUMENTATION MODULE

Each created Joint can be separately documented in detail for both Joint design including geometrical, mechanical parameters, and design results. in ***CONSTEEL*** the generated Joint documentation can be attached to the main documentation of the global structure.

Document generation can be initiated from 2 places in ***CSJOINT.***

- From the Joint manager with the  icon. (see chapter ***2.2.1 JOINT MANAGEMENT***)
- From the ***MAIN WINDOW*** of the created Joint with the ***DOCUMENT…*** button (see chapter ***3.1 MAIN WINDOW***)

After clicking the ***DOCUMENT…*** button, ***JOINT DOCUMENTATION*** dialog will appear. (see picture below)



- Default language of documentation (#1)

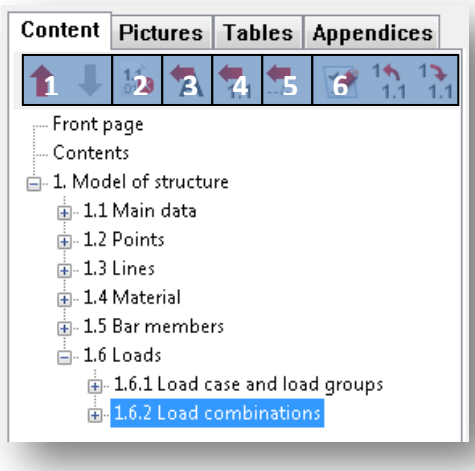
Document creation will use the selected language in the dropdown menu. Language of the document can be changed later too.

- Load cases (#2)

By ticking the checkboxes, it can be chosen which loadcases should be documented.

- Detail level of results (#3)

With summary results only the relevant resistance will be documented, while with detailed results all of the resistances are shown. In the Obtainable data it is possible to clarify the documented data by clicking the checkboxes.



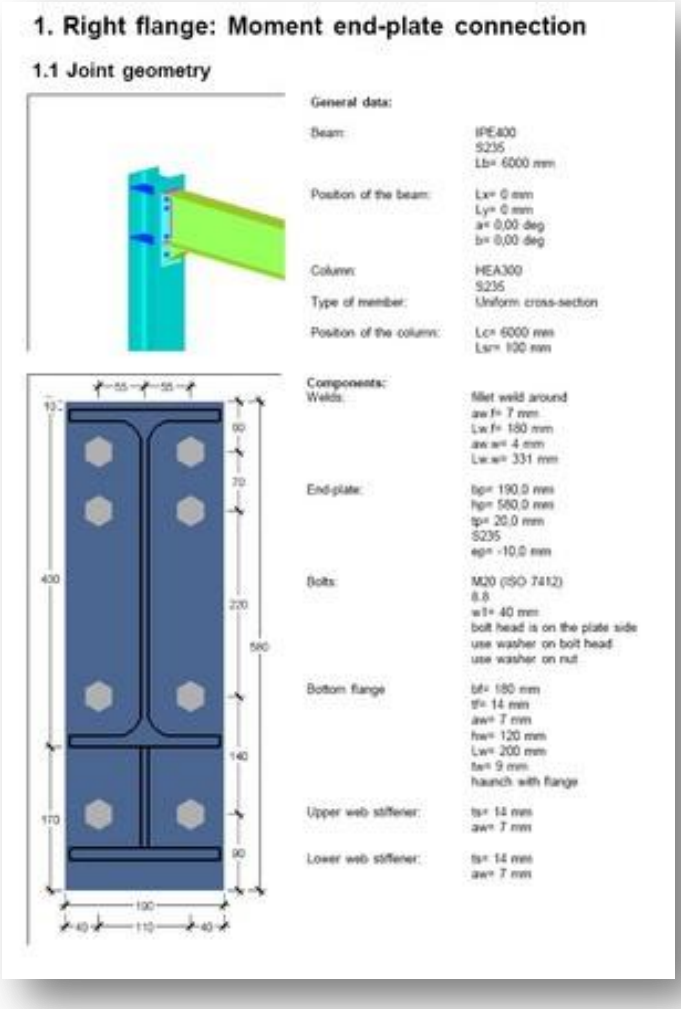
After the documentation is generated, it will be opened in a new window. At the top right of the window the chapters can be seen. The chapters can be moved up or down using the red arrow icons (#1) if a chapter is selected. It is also possible to delete content from document by clicking the third icon (#2).

Text can be inserted anywhere in the document if you click on the fourth icon (#3) and type in the desired text into the window. It is also possible to insert headings by clicking on the fifth icon (#4).

You can also insert page break anywhere in the document using the sixth icon (#5). The last three icons (#6) can be used for the following purposes: Edit heading and increase or decrease level of heading.

The last two icons on the right are different if you click on a chapter or in a paragraph. If you click on a chapter then the icons are as described above. If a paragraph is clicked where there is a table the last icon  can be used to delete any column from the table. If you

click on a paragraph which contains text then the last two icons on the right can be used to edit the paragraph content and the alignment.



6 STRUCTURE-JOINT INTERACTION

The most up-to-date structural design procedures take into account the mechanical interaction between the global structural model and its connections (rigid, semi-rigid or pinned) which generally makes the results more economic and realistic. However this approach requires a more complicated relationship between the joints and the structure and accordingly more serious modelling effort from the engineer. In the *ConSteel* all the joint types can be defined freely or based on the global model geometry using the automatic identification tool, which examines the position of the connected members and the proper cross-sections and offer the possible joint types. After defining the joint it can place back to the global model and the appropriate connection stiffness can be automatically used in the global analysis, and a placed joint is always rechecked based on the current analysis results.

In order to place a joint to the global model click on the ***PLACE JOINT***  icon on the ***STRUCTURAL MEMBERS*** tab. One joint can be placed to several places in the model if these meet the geometrical requirements. The stiffness of the connection can also be taking into account during the analysis if desired. In order to do so click on the analysis parameters and put a tick to ***APPLY CONNECTION STIFFNESS***. Rerun the analysis to the changes take place.