EUROCODE VERSION



Version 9.0 29.05.2015.

www.consteelsoftware.com



CONTENT

1 General description
1.1 Installing the software
1.2 Design standard4
1.3 Joint Types
2 Create Joint
2.1 The Startup window
2.2 Create joint by model10
2.2.1 Place a joint
2.2.1.1 Place Joint directly from Joint details dialog11
2.2.1.2 Place Joints from Structural members tab
2.3 Create joint without model
2.3.1 Joint management
2.3.1.1 The Menu
2.3.2 Joint wizard17
3. Manipulate different joint types
3.1 Main window19
3.2 Parts of the Tree structure
3.2.1 Joint loading
3.2.2 Column
3.2.3 Beam
3.2.4 Stiffeners
3.2.5 End plate/Base plate
3.2.5 End plate/Base plate 27 3.2.6 Finplate 29
3.2.5 End plate/Base plate 27 3.2.6 Finplate 29 3.2.7 Connection plates 30
3.2.5 End plate/Base plate 27 3.2.6 Finplate 29 3.2.7 Connection plates 30 3.2.8 Brace 31
3.2.5 End plate/Base plate 27 3.2.6 Finplate 29 3.2.7 Connection plates 30 3.2.8 Brace 31 3.2.9 Plates 33
3.2.5 End plate/Base plate 27 3.2.6 Finplate 29 3.2.7 Connection plates 30 3.2.8 Brace 31 3.2.9 Plates 33 3.2.10 Splice plate 34



3.2.12 Welds
3.2.13 Foundation
3.3 Create joint types
3.3.1 Create beam-to-column joint
3.3.2 Create Beam splice Joint 41
3.3.3 Create Beam-to-beam Joint 43
3.3.4 Create Bracing Joint
3.3.5 Create column base Joint 47
3.3.6 Create hollow section Joint 49
3.3.7 Create splice plate component
3.3.8 Design of pad foundation
3.3.9 Default joint settings
4 Analysis of connections
5. Results and documentation
5.1 Documentation module57
60 Structure-Joint interaction



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:













		Gusset Plate	
Double flange plate	Middle edge	Upper and lower edge	Double flange plate
Splice plate	Brace	e section: tube, hollow, or round bar	Splice plate
Angle brace Bolted endplate			Angle brace
		Column Base	
		Rivid connection	
	Base Plate	with ground beam	





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







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.

Joint details	-	A			
Joint 1	General data				
Column	Project:				
- Stiffeners - Joint loading	Name:	Twint 1			
Right flange - Moment end-plate connection	nane.	Joint_1			
- End plate	Comment:			4	
- Welds	Engineer:				
	Date:	2015.03.11.			
	laint configuratio	20			
	Beam-to-col	umn joint			
	Type of connects	on(s)			
	Right flange:	Moment end-plate connection			
	Wah bark	No connection			
	THE DECK	no connección			
	Web front	No connection	area:		
	Left flange	No connection		Summary of results of the whole joint	
	Type of frame			Right flange - Moment end-plate connection	
	Braced	Comment: A steel frame is qualified as stiff	fened, when	Warning: The haunch flange thickness is less then the beam flan Error-0 Warning-1 Hint-0	nge thickness!
	Unbraced	displacements of the frame with	n at least 80%.	Right flange - Joint results for positive loading	
				Moment resistance: Axial design resistance:	M ₁ Rd = 224,25 kNm N ₁ Rd = 535,18 kN
				Shear resistance: Initial stiffness	V _{j,Rd} = 273,18 Mi Si is = 122945 53 Mim/rad
				Secant stiffness for applied moment:	Sj.sec = 122945,53 kNm/rad
				Stiffness class:	Semi-rigid (84,1%)
				class of strength.	Partial strength
				Beam flange and web in compression (6.2.6.7):	Fo.ns.Rd = 470,00 kN
				Dominant tension component: Bot row 1	
				Column flange in bending: Mode 3: Bolt failure	Ft1.Rd = 141,12 kN
	1	(m)		Calculation III Automatic calculation	Class OX
Document				Calcuation Automatic calculation	Close OK
	r			~	
	Csor	nópont műveletek			
				End joint placement	
		Select the joint!			
			_		

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:





- Copy (
): copy the selected joint
- Delete (): delete the selected joint
- **Document** (
- **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 (

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.



Currently applied N.		Resistance of bolts, rivets, pins, welds, plates in bearing	y M2 Joint	1,25
EN Recommend		Slip registance at ultimate limit state (Category C)	* M3	1.25
EuroCode	- 4.3.2(1) - Terrain category		7100	1.10
EN Recommend	4.3.3(1) - Ortography factor	Slip resistance at serviceability limit state (Category C)	γ M3ser	1,10
EN Hungarian N	4.4(1) - Turbulence factor	Bearing resistance of an injection bolt	γ M4	1,00
EN Finnish NA	- Other	Resistance of joints in hollow section lattice girder	γ M5	1,00
EN Singapore N	- 7.1 - Recommended values of external pressure	Resistance of pins at serviceability limit state	γ M6ser	1,00
EN Portuguese	7.2 - External pressure coefficients for flat roofs	Preload of high strength holts	v M7	1.10
EN Swedish NA	7.3.a - External pressure coefficients for monopi		7	1 50
EN Austrian NA	7.3.b - External pressure coefficients for monopi	Resistance of concrete	γc	1,50
EN Polish NA	7.4.a - External pressure coefficients for duopito			
Spanish standard	EN1993 Eurocode 3: Design of steel structures			
Spanish	Part 1-1: General rules and rules for buildings			
& User	- 6.1(1) - Safety factors			
	6.3.2.2(2) - Imperfection factors for lateral-torsional			
	— 6.3.2.3(1) - Parameters for LTB curves E			
	Rolled I or H shape (wide parallel flange)			
	Rolled I shape (sloped flange)			
	Welded I or H section			
	6.3.3(5) - Alternative methods for interaction			
	6.3.4(1) - General method for stability design			
	- Part 1-2: General – Structural fire design - 2.2 - Safety factors			
	- Part 1-8: General - Design of joints			
	3.1.1(3) - Bolt grade			
	EN 1998 Eurocode 8: Design of structures for earthquake re:			
	- Part 1: General rules, seismic actions and rules for buildin			
	···· 3.2.2.3(1)P - Parameters of vertical elastic response			
	3.2.2.5(4)P - Lower bound factor for the design spec			
	4			
ate or delete user defined	d standard parameters			
New				
			Cancel	OK

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.

pint identification		
Name:		
Joint_8		
Comment:		
Default settings of the join	t:	
		1.0
Default		- ×
Default		 ×
Default		 _ X
Default		 X
Default		 _ - ≈
Default		 X

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**.



elect joint type	x)
Selected joint configuration:	
Beam-to-column joint	
	Cancel < Previous Next >

▶ 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 *csJoint*, with showing all the possible adjustable parameters.

The *main window* consists of four separate parts.

n splice	General data				
Joint loading eft beam - Moment end-pla	Project:		3		
Beam Fod plate	Name:	Beam splice			
Welds	Comment:				
Beam	Engineer:				
Welds	Date:	2015.03.16.			
	Joint configuratio	n			
	Beam splice j	oint			
	Type of connection	on(s)			
	Left beam:	Moment end-plate connection	Summary of results of	the whole joint	•
	Right beam:	Moment end-plate connection	Left beam - Joint re Moment resistance: Axial design resistance: Shear resistance: Initial stiffness: Secant stiffness for ag	sults for positive loading e: pplied moment:	Mj.Rd = 55,59 kNm Nj.Rd = 393,69 kN Vj.Rd = 236,92 kN Sj.in = 132122,54 kNm/rad Sj.sec = 132122,54 kNm/rad
	Type of frame		Stiffness class:		Semi-rigid (65,3%)
	Braced	A steel frame is qualified as stiffened, w the bracing system decreases the berize	en Dominant compression	component:	Nominal pinned
	Our Contract Output Contrac	displacements of the frame with at least	B0%. Beam flange and v Dominant tension comp	web in compression (6.2.6.7): ponent:	F _{c,fb,Rd} = 782,70 kN
			Bolt row 1 End-plate in bendin Mode 3: Bolt failure	ng e	Ft1,Rd = 141,12 kN
4 111			Right beam - Joint r Moment resistance: Axial design resistance	results for positive loading	Mj.Rd = 55,59 kNm Nj.Rd = 393,69 kN
				staars Kanada da Kaa	

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

DNSTEEL cs**JOINT**

CSJOINT 9.0 USER MANUAL

- 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.



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.

Net Image: transmit and states all states Image: transmit and states<	ners loading flange - Moment end-plate connectior eam nd plate	Cross-section HEA 00 * Type of member Uniform cross-section *	Haterial 5 235 EV 10025-2 Height of section [mm] 290		
Image: And Exponent to they / four if they Average height of a clamm, / store, Image: Booth of and image it is an exponent of the clammed of the clambe is an exponent of the expon	Veda	Paragentry of code-ecclon Property Value Daph 399 nm width 300 nm Web Hodress 8,5 nm Flange Hodress 14 nm Flange Hodress 27 nm Plater radue 27 nm			
		i and tomost swell politic Claim of an interaction stars, Aurospheric Column / story Le + 5000 mm for this mase Pactor of reference plane Lar = 100 mm	Ls	Summar of results of the shole part Right Range - Monreet end databa consection Service 7: A house the results of consection thread - Wannight Reservices (a lease barn the basen thread - Wannight I need - Reservices (a lease barn the basen Wannight Range) - Alexister for positive loading Wannight Range - Alexister for the second Barner and Range - Range - Alexister (a lease of the second Comment Composed: Bottym II Comment Insonge responses Reservice - Range	Arange Mickinesst N ₁ N ₄ = 222.425 With N ₁ N ₄ = 253.18 With V ₁ N ₄ = 254.28 With Mickiness Mickiness Seen-cipit (0.4.1%) Partial eterophic Partial

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.

base joint	Column				
mn base - Base plate connectior	Cross-section		Material		
Column	HEA 200	•	Update S 235 EN 10025-2		
Foundation	Type of member		Height of section [mm]		
Welds	Uniform cross-section	on v	190		
Joint loading	Parameters of cross-s	ection			
	Property	Value			
	Depth Width Web thickness Flange thickness Fillet radius	190 mm 200 mm 6,5 mm 10 mm 18 mm			
				Summary of results of the whole joint Column base - Joint results for positive loading	•
				Moment resistance: Axial design resistance:	Mj.Rd = 5,74 kNm Ni Rd = 0.00 kN
	End configuration		Height of column	Shear resistance:	Vj.Rd = 202,03 kN
	1		Lc = 6000 mm	Initial stiffness: Secant stiffness for applied moment:	Sj.in = 3551,23 kNm/rad Sj.sec = 3551,23 kNm/rad
			Set with mouse	Stiffness class: Dominant components:	Semi-rigid
	Apply shear stub			Left side Base plate in transverse bending:	Ft.pl.Rd = 23,91 kN
	G ss-section			Right side	
	[HEA 200			Concrete in compression under the column flange (6.2.6.9):	Fc.pl.Rd = 104,65 kN
	Length 300	mm			
	Weld 3	mm			
+ III					
Document Plac	e		in i	Calculation V Automatic calculation	Close OK

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.

11		
Haunch with ha	nge 🔹	
tw (mm)	10	
hw (mm)	400	
Lw (mm)	1500 hul et	
tf (mm)		
bf (mm)	150 bf Lf	1
Lf (mm)	0	-w}
Assign the st	iffener to haunch 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:

nt_1 Column Söffeners - Joint loading Right flange - Moment end-p Beam End plate Welds	Web stiffners L Upper transverse plate L Plate thickness P ts = 16 mm Size of welds S a w = 4 mm	ower transverse plate Use lower stiffener late thickness t s = 16 mm ize of welds a w = 4 mm		
	Flange stiffeners TP2 Position a1 aw IRb plate Right fla 400 4	ts hs Add 16 110 Delete Edt	Summary of results of the whole joint	Ţ
	Similar types (Construction) Use stifferer (Construction) (C	Intervalues of sufferer Intervolution table theorems table theorems table theorems table theorems table theorems table theorems table theorem table theorem	Hight Tange - Moment en glate connection Wight Tange - Norman (See Section 2014) Error-O Wormport Net-O Moment resultance: 2024/25 Mm 2024/25 Mm	nge thickness/ Mj.Rd = Nj.Rd = Vj.Rd = Sj.m = Sj.sac = Semi-rigid Partial
			Dominant compression component:	

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.

m solice	End-plate						
Joint loading	Width	Height	Thickness	Material			
Left beam - Moment end-plat	bo = 180 📥 bo =	400 📥 to	= 16 mm S 2	35 EN 10			
Beam	pp 100 (np	φ					
End plate	Bolt		Position of end-plat	te			
Welds	M2 (ISO 7412) - 4.6	Modify	1 ep = -5	🚍 mm			
Right beam - Moment end-pla			1 -				
- End plate	Bolt head is on the	plate side					
Welds	Dalta havinental annai						
	bons - nonzontai spaci	ng	h 15 h	47 - b			
	☑ One bolt column		δ=, 1 ⁻¹ -1	···++			
	Edge distance			50			
	w1 = 45	🗘 mm		-			
	Pitch of holes						
	w2 = 45	A mm					
	No. Kool and Kool Ch	la.			Summary of results of the whole joint	_	
	Vertical positions of bo	its		400	Saminary of resolution and whole joint	•	
	Number of bolt rows	3 🗘		249	Left beam - Joint results for positive loading		*
	Uniform bolt distri	bution	406		55 59 kNm	Mj.Rd =	
					Axial design resistance:	Nj.Rd =	
	Pitch of holes	mm			393,69 kN		
	el	50 📋			Shear resistance: 236.02 kN	Vj.Rd =	-
	p1	249 📋		51	Initial stiffness:	Sj.ini =	-
	p2	51 🕅			132122,54 kNm/rad		
	eln	50	л т	50	Secant stiffness for applied moment: 132122 54 kNm/red	Sj.sec =	
		50			132122,34 Millindu		
			180	i	Stiffness class:	Semi-rigid	
			d-45-d90-	-45-k	(65,3%) Class of strength:	Nominal	
					pinned	Nominal	
					Dominant compression component: Beam flance and web in compression (6.2.6.7):	E. O. D. F.	
	Equal end-plates				782,70 kN	- C, TD, H0 -	
···· •]				Dominant tension component:		Ŧ
			Ø 7				

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

ype of bolt		Designation:	Structural bolt	
tandard	ISO 7412 🔻	Size:	M20	
		Standard:	ISO 7412	
oits size	M20 •	Cross-section properties:		
enath	80 -	Bolt shank area:	Ab = 314,2 mm ²	
		Bolt area:	As = 245,0 mm ²	E
Prestressed b	olts	Other properties:		
K O		Weight of 1000 pcs:	m = 255,73 kg	
Ka -		Min. dist. of bolts:	Gt = 66,0 mm	
μ		Sizes:		
ماد مديد الماد ما		Bolt shank diameter:	d = 20,0 mm	
aterial grade		Bolt hole diameter:	d0 = 22,0 mm	
1.6	▼	Length of bolt shank:	L = 80,0 mm	
		Thread length:	b = 31,0 mm	
/asher		D II Nut Washer		
Use washer o	n nut	Bolt Nut Washer	App	ly
	a halt hand			



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.

ot 8	End-plate			
Main girder	Width Height	Thickness Material		
Joint loading	bp = 130 _ hp = 321 _ tm	= 9 mm S 235 EN 101 -		
Web front - Web finplate co				
Beam	Bolt	Position of end-plate		
Finplate	M207,50 7412) - 4.6 Modify	ep = 42 📩 mm	0.0	
··· Welds		Gap g = 10 _ mm		
	Bolt head is on the plate side	Vertical positions of bolts		
	Shear plane in threaded part	Number of bolt rows	0.0	
	Bullion I have been store			
	Boits - nonzontal spacing	Uniform bolt distribution		
	One bolt columns			
	Edge distance	Pitch of holes mm		
	e ₂₆ 37 🔶 mm	el 80 🗊		
		p1 161 🗊		
	e2, 2/ mm	e1n 80	Summary of results of the whole joint	-
			Web front _ loint results for negative load	ing
			Shear resistance:	Vi.Rd = 42,41 kN
	10-120	*	Beduced bearing resistance on fin plate:	En to Rd = 42.41 kN
	321 161			
	+ ¥			
	-b71+66-	27*		
4 III	-130-	*		
· · · · · · · · · · · · · · · · · · ·			1	
	Diace		Calculation Automatic calculation	Close OK
→ [m]	Place	*	Calculation	Close

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.

tandard ISO 7412 Size: M20 standard ISO 7412 Standard: ISO 7412 olts size M20 Standard: ISO 7412 ength 80 ISO 7412 Standard: ISO 7412 Prestressed bolts Wight of 1000 pcs: Ab = 314.2 mm ² Bolt area: Ab = 314.2 mm ² Verstressed bolts Wight of 1000 pcs: m = 255.73 kg Min. dist of bolts: Gt = 66,0 mm Min. dist of bolts: Gt = 66,0 mm Sizes: Bolt shank diameter: d = 22,0 mm Iaterial grade Bolt bole diameter: d = 22,0 mm Length of bolt shank: L = 80,0 mm Vasher Will bole diameter: b = 31.0 mm = 31.0 mm Vertice of length: Ø bolt Nut Washer Apply Apply	ype of bolt	Designation:	Structural bolt	
Standard: ISO 7412 Standard: ISO 7412 Instantation ISO 7412 Standard: ISO 7412 Cross-section properties: Boil shark sree: Boil shark sree: Ab = 314.2 mm ² Boil shark sree: Ab = 314.2 mm ² Boil shark sree: Ab = 245.0 mm ² Weight of 1000 pcs: m = 255.73 kg Min dist of boits: Gt = 60.0 mm Sizes: Boil shark diameter: d = 20.0 mm Boit shark diameter: d0 = 22.0 mm Longth of boil shark: L = 80.0 mm Thread length: b = 31.0 mm Boit Nut Washer	tandard ISO 7412 🔻	Size:	M20	
olts size M20 V ength 80 V Boit shank area: Ab = 314.2 mm ² Boit area: As = 245,0 mm ² Prestressed bolts Other properties: Weight of 1000 pcs: m = 255,73 kg Min. dist. of bolts: Gt = 66,0 mm Jaterial grade Boit shank diameter: d = 20,0 mm 4.6 V: Sizes: Jasher Boit Nut Washer b = 31,0 mm Bolt Nut Washer Apply		Standard:	ISO 7412	
ength B0 Cross-section properties: Ab = 314.2 mm ² Ab = 34.2 mm ² Boit shank area: Ab = 314.2 mm ² As = 245,0 mm ² Prestressed bolts Other properties: Weight of 1000 pcs: m = 255,73 kg Min. dist. of bolts: Gt = 66,0 mm Sizes: Boit shank diameter: d = 20,0 mm Iaterial grade Boit shank diameter: d = 20,0 mm Length of bolt shank: L = 80,0 mm //asher Boit Nut Washer A pply	Bolts size M20 🔻	Cross section properties:		
Boil sharks area. All = 3 4 ± 1 1 1 + 1 +		Patt abank areas	Ab = 244.2 mm ²	
Prestressed bolts Construct	Length 80 V	Bolt eree:	Au = 314,2 mm ²	
Prestressed bolts Other properties: weight of 1000 pcs: m = 255,73 kg μ 0 Weight of 1000 pcs: Gt = 66,0 mm μ 0 Bolt shank diameter: d = 20,0 mm katerial grade Bolt shank diameter: d = 22,0 mm Length of bolt shank: L = 80,0 mm Vasher ✓ Bolt Nut Washer Apply		Don area.	Aa - 243,0 mili-	=
K₀ 0 Weight of 1000 pcs: m = 255,73 kg μ 0 Min. dist. of bolts: Gt = 66,0 mm sizes: Sizes: Bolt shank diameter: d = 20,0 mm solt hole diameter: d = 22,0 mm Length of bolt shank: L = 80,0 mm Vasher Ølt shank: L = 80,0 mm ▼ Ølse washer on nut Bolt Nut Washer Apply	Prestressed bolts	Other properties:		
K₀ U Image: Margin and State St		Weight of 1000 pcs:	m = 255 73 kg	
μ 0 Sizes: Sizes: Sizes: Solid Shank diameter: d = 20,0 mm Bolt hole diameter: d0 = 22,0 mm Length of bolt shank: L = 80,0 mm Thread length: b = 31,0 mm ▼ Solid Shank diameter: d0 = 22,0 mm Length of bolt shank: L = 80,0 mm Thread length: b = 31,0 mm ▼	K _a U	Min dist of holts:	Gt = 66.0 mm	
μ □ Sizes: Bolt shark diameter: d = 20,0 mm Bolt shark diameter: d = 22,0 mm Length of bolt shank: L = 80,0 mm Thread length: b = 31,0 mm Vasher Bolt Nut Washer Apply		Min. dist. of bolts.	01 - 00,0 mm	
laterial grade Boit shank diameter: d = 20,0 mm 4.6 ▼ icentified diameter: d0 = 22,0 mm Length of boit shank: L = 80,0 mm L = 80,0 mm Vasher Boit Nut b = 31,0 mm Ø boit Nut Washer Apply	μ 0	Sizes:		
Iaterial grade Bolt hole diameter: d0 = 22.0 mm 4.6 ▼ Bolt hole diameter: d0 = 22.0 mm Length of bolt shank: L = 80.0 mm Thread length: b = 31.0 mm Vasher Bolt Nut Washer Apply		Bolt shank diameter:	d = 20,0 mm	
4.6 ▼ Image: Constraint of the sense: L = 80,0 mm //asher Image: Constraint of the sense: b = 31,0 mm ▼ Ø Use washer on nut Bolt Nut Washer Apply	Material grade	Bolt hole diameter:	d0 = 22,0 mm	
Vasher Thread length: b = 31,0 mm Z Use washer on nut Bolt Nut Washer	4.6 👻 🛄	Length of bolt shank:	L = 80,0 mm	
Vasher / Washer Apply		Thread length:	b = 31.0 mm	-
Bolt Nut Washer Apply	Nasher			
Apply	I les washer en aut	Bolt Nut Washer		
	v Use washer on hut		App	лу



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.

ing joint	Brace			
Main girder	Cross-section	Material		
Joint loading	Claw 30.3x3.2 V Up	date S 235 EN 10025-2		
Web back - Gusset plate				
Connection plates	Parameters of cross-section			
Welds	and the ters of the case section			
Upper edge - Splice plat	Property Value			
Brace	Diameter 60.3 mm			
Plates	Thickness 3.2 mm			
Splice plate				
···· Weld				
Middle edge - Splice plat				
Brace				
Plates				
- Splice plate				
Weld				
 Lower edge - Splice plati 				
Brace			Summary of results of the whole joint	
Plates			Web back upper edge - Splice plate connection	
weld			Warning: Gusset plate buckling is not checked!	÷.
web	Rotation of section		Web back middle edge - Splice plate connection	
			Warning: Gusset plate buckling is not checked!	
	-90° -90°	Mirror	Web back lower edge - Splice plate connection	
			Warning: Gusset plate buckling is not checked!	=
	Decision of houses		Error-0 Warning-5 Hint-0	
	Position of brace		Web back upper edge - Joint results for positive loading	
	lx = 0			
			Summary	
	Ly = -15 🐳 mm	150 50		
	a= 45 A .	A B	Shear resistance of the dominant bolt: Fv,Db,Rd = 21,71 kN	
			Bearing resistance of dominant bolt: Fv,Rd = 00,00 KN Es ps ps = 57.36 kN	
	$\beta = 0$ \wedge		Brace tension and compression resistance of net cro:Nt.Rd.net = 147,84 kN	
			Block tearing resistance: Veff,Rd,sp= 184,28 kN	
			Resistance of shear welds: Fw,Rd = 239,45 kN	
4 111			Web back middle edge - Joint results for positive loading	-
		🔊 🕖 🖓 📿		_

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.





_12 hord	Cross-section	Material	
oint loading lane of braces	CHS 60.3x3.2 •	S 235 EN 10025-2	
. Brace	Parameters of cross-section		
Brace	Property Value		
Brace	Diameter 60,3 mm		
	Thickness 3,2 mm		
		Summary of results of the whole joint	•
	Placement	- Joint results for negative loadin	ng
	Angle of brace 54,73 🚖 °	Summary	
	Eccentricity 0 🔺 mm	Dominant member	Brace 1
	Rotation of section by 90°	Dominant component:	Chord face failure
		Resistance:	NRd = 67,32 kN
	Weld data	Type of joint:	K, N
	Fillet weld Butt wel	d - Joint results for positive loadin	g
	Weld size 3 mm	Summary	
	Mathod of wald design	Dominant member:	Brace 1
	Section resistance O Design	Dominant component:	Chord face failure
		Resistance.	NRG - 07,47 KM
		Type of joint:	K, N
	Place	Calculation V Automatic calculat	ion Close OK
Document		- Hardenadori	2.000

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.

racing joint	Plate 1	
Main oirder	Width Height Thickness	
Web back - Current plate		
Connection plates	1 Weddie is not of the online plate width	
Walda	Vividuris equal to spice plate widur	
Webs		
B opper edge - spice plat	material T	
Brace		
Plates	Type	
Splice plate		
···· weid		
⊢- Middle edge - Splice plat		
Brace		
Plates		
- Splice plate		
· Weld		
Lower edge - Splice plate		
··· Brace	Summary of results of	the whole joint 👻
Plates		
··· Splice plate	Web back upper ed	ge - Splice plate connection
Weld	Warning: Gusset plat	Duckling is not checked!
	Werping: Gunnet plat	buckling is not checked!
	Web back lower en	ne - Splice plate connection
	Warping: Gusset plat	buckling is not checked!
	Error-0 Warnino-3 H	int-0
	Web back upper ed	ge - Joint results for positive loading
	Summary	
	Characteristeness of th	a deminant halt E. m. m 24 74 M
	Shear resistance of a	bote: FV,Db,Rd = 21,71 KN
	Bearing resistance of	forminant bolt: Eb Db Rd = 57.36 kN
	Brace tension and con	pression resistance of net crotNt.Rd.net = 147,84 kN
	Block tearing resistant	e: Veff,Rd,sp= 184,28 kN
	Resistance of shear v	relds: F _{w,Rd} = 239,45 kN
	Web back middle e	dge - Joint results for positive loading
d Document	Place Via Line Calculation V Au	tomatic calculation Close OK

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,

pe of bolt		Designation:	Structural bolt	
andard	ISO 7412 🔻	Size:	M12	
		Standard:	ISO 7412	
ts size	M12 🔻			
		Cross-section properties:		
nath	75 👻	Bolt shank area:	Ab = 113,1 mm ²	
-		Bolt area:	As = 84,3 mm ²	E
Prestresse	d bolts	Other properties:		
		Weight of 4000 perce	- 05 04 he	
Ka 0		Min diat of holta:	01 = 00,01 Kg	
		Win. dist. of boils.	Gt = 58,0 mm	
μο		Sizes:		
		Bolt shank diameter:	d = 12.0 mm	
terial grade		Bolt hole diameter:	d0 = 13.0 mm	
6	▼	Length of bolt shank:	L = 75.0 mm	
• •		Thread length:	b = 23,0 mm	-
sher		·		
Use washe	er on nut	Bolt Nut Washer	Apr	lv
				·/



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.

bint_9 Man girder ⊃ont lading Web front - Moment end-ple → Resembler → Weds → Front end → End Joke → Welds	<pre>terial \$235 EN 10025-2 *** V/eb stiffener Total height Leb plate thickness is equal to the beam web thickness Plate thickness Is = 9 *** mm Cut of corners to anc: Size/radius of the cut: 0 *** mm Stiffener Use stiffener Use stiffener Use stiffener Use stiffener Stiffener Ls = 14 *** mm End-plate End-plate End-plate S*** mm Width</pre>	Summary of results of the whole joint Web front Joint results for negative loading Memory is a state of the state of the state of the state Avail design resistance: Shear resistance: Shear resistance: Bean flage and web in compression (8.2.6.7): Dominant tension component: Bean flage and web in compression (8.2.6.7): Dominant tension component: Shali gride state and plate in transverse bending: Mode 1.: Complete yielding of the flage	M. Ref = 77.24 KMM N. Ref = 77.24 KMM V. Ref = 125.16 KM V. Ref = 125.16 KM Sijase = 31181.97 KM/m/rad Sijase = 31181.97 KM/m/rad Fer. fb. Ref = 794.77 KM Frit.Ref = 83.49 KM
Document	Pace	Calculation V Automatic calculation	Close OK

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.

Automatic weld optimization Linear stress distribution is considered on the web weld for bending resistance Weld properties Upper flange weld aw.fu = $3 \div mm$ Size of weld on web aw.w = $3 \div mm$ Lower flange weld aw.fl = $3 \div mm$	
Lower haunch welds	Upper haunch welds
Size of weld on flange	Size of weld on flange
aw.f = 3	aw,f = 3

- 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

olumn base joint Golumn base - Base plate co Golumn — Foundation — End plate — Welds — Joint loading	Pourvision WZ	
	bb = 1400 mm Deep	▼ Mj.Rd = 5,74 Nj.Rd = 0,00 Vj.Rd = Sj.in =
	Secant stiffness for applied moment: 3551.23 ktm/sad Thickness Material grade tg = 20 mm C12/15 EN 1992-1 Image: C12/15 EN 1992-1 Mode 2. Bot failure with yielding of the flange	Sj.sec = Semi-rigid Ft.pl.Rd =
III Document	Place Calculation V Automatic calculation Cit	ose OK

- 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:

1	2	3
Fype of the selected or	mection:	×
Welded connection		
Cross-section	IPE 400	•
		Cancel OK

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 B**EAM)
- 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.

Select joint type - cs/oint 9.0000	Beam splice joint - csJoint 9.0.000
	Type of the selected connection:
Beam splice joint Cancel < Previous Next >	Cross-section HEA 200 ▼ Cancel <previous create<="" td=""></previous>

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**.

	Beam-to-beam joint - csJoint 9.0.001
	Left web
ected joint configuration:	Main girder section
zam-to-beam joint Cancel < Previous Next >	Cancel < Previous Create

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:

1	2	3	
>			
Type of the selected No connection	connection:		
Cross-section		-	

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 B**EAM)
- 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.

elect joint type - csJoint 9.0.000		
ted joint configuration:		
ing joint		
	Cancel < Previous	Next >
	Liner Ranne	
Left web	Upper flange	l
Left web Joint position along the mer At start or end point of	Upper flange Upper flange Right web Lower flange ber the member the member	
Left web Joint position along the mer O At start or end point of O At intermediate point of	Upper flange Upper flange Right web Lower flange ber the member the member the member	
Left web Joint position along the mer At start or end point of At intermediate point of Brace beam section	Upper flange Upper flange Right web Lower flange uber the member the member the member	

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 **B**RACE is selected in the connection tree)

Gusset Plate joint	— X —
	Upper edge
	Upper edge
	Middle edge
	Middle edge
	Lower edge
	Lower edge
	Cancel OK
	52
Type of connection	
	3
×	
Type of the selected connection:	
Double plate flange connection	
Cross-section HEA 200	
	Cancel OK

- 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)

: selected connection:	_	
section CHS 60.3x3.2 V		
tion CHS 60.3x3.2 •		Selected joint configuration: Column base joint
Cancel < Previous Create	< Previous Next >	Cancel <prev< td=""></prev<>
Cancel < Previous Crea	< Previous Next >	Cancel <prev< td=""></prev<>

► 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):

	Splice plate component - csloint
	Type of the selected connection:
Selected joint configuration:	Two sides, simple splice plate
Splee plate component Cancel Revisor Next >	Cancel <previous create<="" td=""></previous>

- 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 M_{AIN} 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

Lay er	Name	Start depth [m]	End depth [m]	Color	Reference soil for the calculation of bearing capacity
1	Finom homok	0,00	2,00		✓
	Kavicsos homok	2,00	4,00		
3	Sárga homokos i	4,00	5,50		
4	Sárga sovány ag	5,50	8.00		
•					
Le	vel of foundation:	FL	1	m	

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).

Layer	Name	Thickness [m]	Color	γ₅ [kN/m ³]	γa [kN/m ³]	¢u [°]	E [kN/m ²]	c [kN/m ²]	Cu [kN/m ²]	v [-]	Delete soil
1	Fine sand	2,00		22	20	35 <mark>,</mark> 0	57500,0	0,0	0,00	0,28	layer
2	Gravelly sand	2,00		20	18	28,0	20000,0	0,0	0,00	0,15	New soil layer
3	Yellow fine gravelly	1,50		21	20	32,0	18000,0	50,0	50,00	0,25	•
4	Yellow lean clay	2,50		20	19	22,0	15000,0	60,0	60,00	0,35	

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

Design settings

Design method: Based on selected	NA 🔻	- 1
Orained		- 1
Undrained		
Settlement limit	50	mm
Consider passive earth resistance in cal	lculation of sliding	
Angle of internal friction between foundati	ion 32	D
		_

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

Model	based joint	t loading					
ta of joint k	hads:	, out of the second					
igin of joint	load						
rrent load	1000.	1	Loadcase				
mentioau:		1				`	
Generate	load comb	ination	New Delete				
nbined valu	es of force	s without s	afety fact	tors			
me of N [kN] Vy		Vy [kN]	Vz [kN]	Mx [k	My [k	Mz [k	
ersistent	tent 0		0	0 0	0	0	
ransient	0	0	0	0	0	0	
)SLS ⊚ l	JLS @	0 A1 ○/	42 (N]				

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.

V	Elastic	analysis	of basic	values	ofreactions	for	foundation d	hecks

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 con.

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).

Default joint settings General Stiffeners	Setting1 Web stiffeners				
Bolts	Upper transverse plate Plate thickness		Lower transverse p Plate thickness	olate	
Welds	ts = 10	mm	ts = 10	mm	
- Plates	Size of welds		Size of welds		
Gusset plate	a w = 4 r	nm	a w = 4	mm	
	Shear stiffeners				
	Stiffener types		Dimensions of	of stiffener	
				2	
			Size of we	ide .	
			aw :	= 4	mm
			F	1	
	Size of weld		2	hs a ₁	
	aw 4 mm		t,		
	Thickness of plate				
	ts 10 mm				
eate or delete new default settin	gs				
etting1	Solution		_		
			G	ancel	Ōĸ

oint identification			
Name:			
Joint_15			
Comment:			
Default settings of the joint	:		
Default			- 🗶
Default			
Setting1			
Load sections	Cancel	< Previous	Next >



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
 - o Block tearing resistance of plates
 - Weld resistance and capacity to the applied load or section strength
- ► Hollow section connections:
 - o Chord face failure, web failure, shear failure and punching shear failure
 - Brace failure
 - $_{\odot}$ 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.

Summary of results of the whole joint	-	
Summary of results of the whole joint		
Summary of results of the selected connection		
Detailed results of the selected connection	nge thickness!	ĥ
Dominant load case		
Positive loading		
Negative loading		
internet in the second se	= 267,82 kNm	
Axial design resistance:	Nj.Rd = 705,60 kN	
Shear resistance:	Vj.Rd = 322,36 kN	
kNm/rad	5j,ini = 592040,70	
Secant stiffness for applied moment:	Silsec = 392646.70	=
kNm/rad	, ,	
Stiffness class:	Rigid	
Class of strength:	Partial strength	
Dominant compression component:		
Beam flange and web in compression (6.2.6.7):	Fc, fb, Rd = 587, 50 kN	
Dominant tension component:		
Bolt row 1	E.,	
Mode 3: Bolt failure	Ft1,Rd = 141,12 KN	-



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)

oint document	ation		23
Default langua 1 Er	age of documenta nglish	ation:	
Load cases 2 Dominal Real loa Basic lo	nt load case ad cases ad cases		
Detail level of	fresults nary results	Oetailed results	
	of load ange		
		Cancel	Create



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 is 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.

mport stiffness of semi-rigid connections in the analysis by load combinations Load combinations Apply connection stiffness	ad combinations Load cases	Joints		
Load combinations Apply connection stiffness	mport stiffness of semi-rigid c	onnections	in the analysis by loa	d combinations -
Load combinations Apply connection stiffness				
and combination 1	Load combinations		Apply connection :	tiffness
Load complication-1	Load combination-1		✓	