

CONSTEEL 12 USER MANUAL EUROCODE VERSION

Version 12.0



CONTENT

1 General description1	1
1.1 Installing the software1	1
1.2 The main window	2
1.2.1 The startup window1	.3
1.2.2 The graphical window1	.3
1.2.3 The menu	.4
1.2.4 Tabs 2	2
1.2.5 The side bar	6
1.2.6 csPI panel 2	:6
1.2.7 The status bar 2	7
1.2.8 The windows of Object tree, Diagnostics results and Object properties	3
1.3 General structure of dialogue windows	6
1.4 General functions for tables	7
1.5 Hot keys	9
2 File handling4	1
2.1 Basics	1
2.2 File types	1
2.3 Tekla Structures model import, export and update4	2
2.3.1 Import structural modell from Tekla Structures 4	2
2.3.1.1 Limitations of the import of bar members4	5
2.3.2 Complete model export into tekla structures 4	6
2.3.2.1 Limitations of the export of Joints and bar members	8
2.3.3 Change management between Tekla and ConSteel: Tekla update	9
2.4 IFC import and export	1
2.4.1 Import of IFC files5	1
2.4.1.1 Available objects for the import process5	4
2.4.2 Export of IFC files	6
2.4.2.1 Available objects for the export process5	7
2.5 DXF import	0
	1

STEEL-

2.6 Export of reinforcement into DXF	61
2.7 DLUBAL Rstab and SAP2000 model import(.xlsx)	
2.8 Exporting model data from Dlubal Rstab in xlsx format	65
3 Model view	67
3.1 Basics	67
3.2 Model views	67
3.3 Selection	70
3.4 Portions Manager	72
3.5 Object names and Renumbering	74
3.6 Dimensions	76
3.7 Measure	
3.8 Model grid	
4 Drawing geometry	
4.1 Basics	
4.2 Coordinate systems	
4.2.1 User coordinate system	
4.2.2 Local coordinate system of bar elements	
4.2.3 Local coordinate system of surface elements	
4.3 Defining action points	
4.4 Drawing	
4.5 Modifying	91
4.6 Layers	95
5 Structural modelling	97
5.1 Basics	97
5.2 Line members	97
5.2.1. Steel members	100
5.2.2. Reinforced concrete members	
5.2.2.1 Reinforced concrete cross sections with design option	103
5.2.2.1.1 Beam reinforcement definition	103
5.2.2.1.2 Column reinforcement definition	106
5.2.2.2. Reinforced concrete cross sections without design option	107

STEEL -----

5.2.3. Composite columns
5.2.4 Composite beams
5.2.5 Haunched members
5.2.6 Tapered members
5.3 Surface members
5.3.1 Plate reinforcement
5.4 Diaphragm
5.5 Rigid body
5.5.1 Modifying rigid body
5.6 Convert members to plates
5.7 Frame corner wizard
5.8 Materials
5.9 Supports
5.9.1 Point support ()
5.9.2 Line support (
5.9.3 Surface support (🔀)138
5.10 Link elements
5.10.1 Connect two structural points (
5.10.2 Connect two structural edges that are equal length and parallel ()
5.11 Shear field
5.11.1 Definition of shear stiffness141
5.11.2 Calculation of shear stiffness143
5.11.2.1 Determination of shear stiffness in case of panels produced by hoesch
5.11.2.2 Determination of shear stiffness in case of panels produced by Fischer
5.11.2.3 Determination of shear stiffness in case of panels produced by Achelor
5.11.2.4 Determination of shear stiffness according to Eurocode 3
5.11.3 Shear field stiffness definition146
5.11.3.1 Shear field stiffness calculation148
6 Structural Loads

STEEL-

6.1 Basics	151
6.2 Load cases and load groups	151
6.3 Load combination	153
6.4 Load types	154
6.4.1 Point load (155
6.4.2 Line load (156
6.4.2.1 Line wind load	159
6.4.2.2 Line Snow load	162
6.4.3 Surface load (163
6.4.4 Load transfer surface (🗾)	
6.4.4.1 Modifying load transfer surface	167
6.4.5 Temperature load (168
6.4.6 Prescribed displacement (
6.4.7 Change in length (, tension force (F)	169
6.5 Global imperfections	
6.5.1 Notional Load	171
6.5.2 Initial sway	171
6.5.3 Application of eigenshape	172
6.5.3.1 Apply eigenshapes	172
6.5.3.2 Automatic amplitude calculation	174
6.5.3.3 Managing eigenshapes	176
6.6 Moving load	
6.6.1 Moving load path (178
6.6.2 Influence graph (
6.6.2.1 Managing influence graphs	180
6.6.2.2 Analyze influence graph	
6.6.3 Crane load ()	180
	4

STEEL ----

6.6.3.1 Standard
6.6.3.2 Standard load based
6.6.3.3 User defined
6.6.3.4 Placing crane load
6.6.4 Train load (🧭)
6.7 Meteorological load generator
6.7.1 Load groups and load cases
6.7.2 Meteorological surfaces
6.7.2.1 Wind surface
6.7.2.2 Snow surface
6.7.2.3 Placing meteorological surface
6.7.3 Meteorological load generator197
7 Masses
7.1 Basics
7.2 Mass cases and mass groups198
7.2.1. Converting loads into masses
7.2.2. Manual definition of mass groups and mass cases
7.2.3. Considered direction of masses
7.3 Definition of mass combinations
7.4 Creating point masses
7.5 Seismic effect
7.6 Creating response spectrums
7.6.1. New standard elastic response spectrum
7.6.2. New user defined response spectrum
7.6.2.1 Manual definition of response spectrum 205
7.6.2.2 Manual definition of response spectrum, based on a cloned standard spectrum 207
8 Structural analysis
8.1 Basics
8.2 Finite elements
8.2.1 Basics
8.2.2 Line elements
8.3 Model check (diagnostics)



8.4 Analysis settings	
8.4.1 Overview page	
8.4.2 Subpage of global settings	
8.4.3 Subpage of load cases	215
8.4.4 Subpage of load combinations	216
8.4.5 Subpage of dynamic analysis	217
8.5 Analysis types	
8.5.1 Elastic analysis	
8.5.1.1 First order	219
8.5.1.2 Second order	219
8.5.1.3 Static eigenvalue – buckling analysis	220
8.5.1.3.1 Buckling sensitivity	221
8.5.1.4 Dynamic eigenvalue – vibration analysis	221
8.5.2 Plastic analysis	222
8.5.2.1 Basic	222
8.5.2.2 First order	223
8.5.2.3 Second order	223
8.5.2.4 Static eigenvalue – buckling analysis	223
8.6 Analysis results	
8.6.1 Result types	
8.6.2 Visualization options	227
8.6.3 Result markers	229
8.6.4 Envelope figures	
8.6.4.1 User defined envelope figures	232
8.6.4 Show original shape of structure	234
8.6.5 Table of dominant values	235
8.6.6 Animation of dynamic shapes	236
9 Standard Design	
9.1 Basics	
9.2 Steel design	
9.2.1 Global checks	238
9.2.1.1 Basics	238
9.2.1.2 Cross section check	238
	6

STEEL-

9.2.1.3 Stable length check	238
9.2.1.4 Buckling check	239
9.2.1.5 Results	240
9.2.2 Member checks	
9.2.2.1 Basics	
9.2.2.2 Individual Member Design	
9.2.2.3 Member Group Design	
9.3 Composite design	247
9.3.1 Composite column design	
9.3.1.1 Results	
9.3.2 Composite beam design	249
9.3.2.1 Analysis model	
9.3.2.2 Design of shear studs	249
9.3.2.3 Shear buckling resistance of composite beams	251
9.3.2.4 Bending and vertical shear resistance	251
9.3.2.5 Lateral-torsional buckling resistance of composite beams	251
9.3.2.6 Sheeting with ribs parallel to the supporting beam	251
9.3.2.7 Sheeting with ribs perpendicular to the supporting beam	252
9.4 Reinforced concrete design	
9.4.1 Reinforced concrete beam design	255
9.4.2 Reinforced concrete column design	255
9.4.3 Reinforced concrete slab design	256
9.5 Serviceability checks	258
9.5.1 Basics	258
9.5.2 Global horizontal deflection check	259
9.5.3 Global vertical deflection check	259
9.5.4 Local deflection check by members	
9.5.5 Relative deflection check for 2 points	260
10 Section Module	
10.1 Basics	261
10.2 Section administration	261
10.3 Section modelling	
10.3.1 The EPS model	262
	7

STEEL-

10.3.2 The GSS model	
10.4 Section Tools	
10.4.1 Geometrical calculations	265
10.4.2 Loads	266
10.4.3 Stresses	
10.4.4 Effective cross section	
10.4.5 Standard resistance calculations	
10.4.5.1 Steel cross sections	
10.4.5.2 Composite and reinforced concrete cross sections	270
10.4.5.3 Results	271
10.5 Documentation	
11 Fire Design	
11.1 Fire load group and load case	
11.2 Fire effect and protection	
11.3 Analysis	
11.4 Design	
12 Earthquake analysis	
12.1 Basics	
12.2 Steps of the seismic analysis	
12.3 Seismic effect	
12.3.1. Seismic settings	282
12.3.2. Additional effects	285
12.4 Seismic loadcases and loadcombinations	
12.4.1. Seismic loadgroup and loadcase	
12.4.2. Load combinations in seismic design situations	
13 Documentation	
13.1 Basics	
13.2 The document tab	
13.2.1 Start documentation	290
13.2.1.1 Documentation module	292
13.2.2 Creating snapshots	294
13.2.3 Snapshot manager	295
	8

	12.0 USER MANUAL
13.2.4 Model information	
13.3 MS Word export of the documentation	
13.3.1 Creating custom templates for MS Word export	
14 Joint module	
14.1 Basics	
14.2 Joint Types	
14.3 Create Joint	
14.3.1 Create joint without model	
14.3.2 Create joint by model	
14.3.3 Place joint in the global model	
14.3.4 Manipulating joints	
14.3.4.1 Joint details dialog	
14.3.4.2 Manipulating join	
12.3.5 Default joint settings	
14.4 Analysis of connections	
14.5 Results and documentation	
14.6 Structure-Joint interaction	
14.7 Joint Export	
15 csPI – ConSteel Programming Interface	
15.1 What is csPI?	
15.2 Functionalities of the csPI panel	
15.2.1 Minimized panel size	
15.2.2 Maximized panel size	
15.3 Basic tools of csPI	
15.3.1 Variables	
15.3.1 Mathematical operators	
15.3.2 Cycles, For loops	
15.3.3 If else statements	
15.3.4 Available commands of csPI	
15.4 Troubleshooting and debug functions of csPI	

-





1 GENERAL DESCRIPTION

1.1 INSTALLING THE SOFTWARE

In the followings hardware and software requirements are defined for the *ConSteel* software.

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 7, 8, or 10 64-bit

Recommended hardware:

- Processor: Intel Core i7 or equivalent
- Memory: 32 GB
- ▶ Video-card: 1 GB MB non-integrated

You 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 *ConSteel* 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 *ConSteel* install file (ConSteel_12_x64_install_xxxx_xx_xx.exe) and follow the instructions. As a first step the language of the setup has to be set. The *ConSteel* will use this language for the first time you start it, 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\ConSteel 12.0), 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 have to wait until it finishes.



1.2 The main window

Geometry Structural members Layers Document 31 1 1 1 1 1 6 1 1 1 1 1 1		New model - Design standard: EN Recommended Values - ConSteel 12 - x64 Edit View Options Standards Help	– 🗆 X
I I i i i i i i i i i i i i i i i i i i	Ge	cometry Structural members Loads Mass Finite element Analysis Global checks Member checks Serviceability checks Layers Document	
	∎ ♪		(귀 (귀) 2 (가 (라) 2 (가 () ((((
			Show hidden parts of the structure translucently
💌 CsPleditor / 🗈 ▶ Run 💮 🚱 📐 火 2750 Y 15750 Z 0 L 16373.8	Ra G		↓ ↓ シ 和 Q n /4 Step 250

The main window consists of six separate parts containing different functionalities. The graphical window is the area for the 3D structural modeling (#1); the menu on the top contains some important commands (#2); the tabs from left to right lead the user through the steps of structural design (#3); the side bar on the left contains functions for grids, views, most commonly used transformations and selections (#4); the status bar on the bottom makes the drawing phases easier (#5) and the object and parameter tables at the right gives always sophisticated information about the model, making fast modifications possible (#6).



1.2.1 The startup window

The starting of *ConSteel* opens a startup dialog window which allows to create and open 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.

	F	≪ ⊂	ONSTE	≣L
atest models:				
Model name	Last modified $~~ \bigtriangledown$	Path	Description	
Model 1	2010.03.17. 15:01:38	C:\ConSteelModels		
Model 2	2010.03.17.15:01:28	C:\ConSteelModels		
Model 3	2010.03.17.15:01:18	C:\ConSteelModels		=
Model 4	2010.03.17.15:01:06	C:\ConSteelModels		
Model 5	2010.03.17. 15:00:52	C:\ConSteelModels		
Model 6	2010.03.17. 15:00:42	C:\ConSteelModels		
Model 7	2010.03.17. 15:00:31	C:\ConSteelModels		
Model 8	2010.03.17. 15:00:20	C:\ConSteelModels		
Model 9	2010.03.17. 15:00:08	C:\ConSteelModels		
Model 10	2010.03.17. 14:59:39	C:\ConSteelModels		- -

The first big button (#1) is for creating a new model, the second (#2) is for opening model from the folder.

1.2.2 The graphical window

The structural model appears always in the graphical window. There are no other window opening options; however there are lots of viewing possibilities in this single window. The graphical window supports the modeling by the global coordinate system (*GCS*) and a moveable, rotatable and size adjustable grid. The coordinate system at the left bottom corner denotes always the unchangeable *GCS*; the origin of the user coordinate system (*UCS*) takes place at the middle point of the grid which is always the plane "XY" of the *UCS*. The following moving possibilities and quick view settings can be used during the structural model manipulation:



Move:

push and hold down the middle mouse button or use the four arrow buttons on the keyboard to move the model on the screen

Rotate:

hold down the ALT key and the left mouse button. The center of rotation is always adapting the actual model view

Scaling:

spin the middle mouse button forth and back or use the + and - buttons on the keyboard or hold down the *ALT* key and the right mouse button

Zoom window:

window drawing by the left mouse button, while pressing SHIFT+ALT keys

Hotkeys for vie:

- Ctrl+1: Switch to top view
- Ctrl+2: Switch to front view
- Ctrl+3: Switch to side view
- Ctrl+4: Switch to axonometric view
- Ctrl+5: Perpendicular to actual raster plane
- Ctrl+0: Quick zoom all

1.2.3 The menu

The *ConSteel* menu does not contain many commands since the main functionality is placed to the structured tabs and side bars and the applicability and modeling efficiency does not really need to duplicate the functions. However six important function groups appear here: the *FILE* handling, some *EDIT* options (undo-redo), *VIEW* and diagnostics, *OPTIONS* for settings (for saving, updating, selecting language, and for model diagnostics), *STANDARDS* for reviewing and defining standard parameters for design and *HELP*.



FILE menu:

File	Edit	View	Options	Standar			
	New n	nodel		- 1			
	Model attribute						
	Open		Ct	trl+O			
	Save		C	trl+S			
	Save A	\s					
	Impor	t		>			
	Export	:		>			
	Updat	e		>			
	Exit						
				_			

New model

A new structural model can be created.

Model attribute

The model description and the EN National Annex, or design standard can be changed. (These parameters can be set at first by creating a new model.)

Model description	
Design standard	[* EN Recommended Values *] 🗸
	American Standard 2010
	EN Recommended Values
	EN Hungarian NA
	EN Dutch NA
	EN Finnish NA
	EN Singapore NA
	EN Portuguese NA
	EN Swedish NA
	EN Austrian NA
	EN Polish NA
	EN German NA
	EN Bulgarian NA
	EN Greek NA
	EN UK NA
	EN Romanian NA
	EN Cyprian NA EN Croatian NA
	Spanish standard Italian standard
	MSZ 27003-1-5 + EN Hungarian NA
	[* EN Recommended Values *]

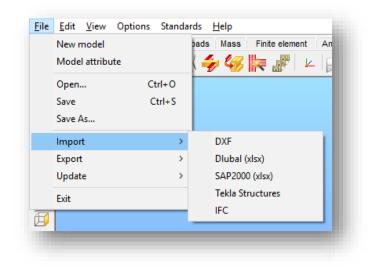
Open, Save, Save As

The functions are according to their names.



Import

Import menu contains a wide range of import possibilities from the very basic DXF wireframe import to the widespread IFC, furthermore the complex Dlubal and SAP2000 model import inlcuding the loadings as well.



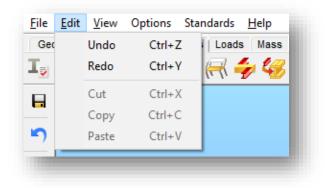
Export

The selected part of the model can be exported to *TEKLA Structures* or to IFC. In case of slab reinforcement result, the colored result view can be exported to DXF format as well.

Update

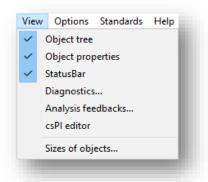
In case of connected Tekla modell (previously Tekla import or export is needed), connected Tekla model can be updated and modified acc. to the actual ConSteel model.

In the *EDIT* menu the common functions can be found:





VIEW menu:



With the first four functions the visibility of the dockable windows on the right side of the main screen, can be switched on/off. With the *SIZES OF OBJECTS*... function, the size of the objects can be changed.

oint load	¢			é			1	⇒
Distributed load	¢			Ā			1	⇒
Support	¢			Ā			1	⇒
Pin	4						1	⇒
Joint	¢			Ā			I	⇒
Local system	¢			Ā				⇒
Arrow end	4			-			1	⇒

OPTIONS menu:

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

Save

tions			×
Options Save Update Update Language Styles Diagnostics Export	☑ AutoSave: ☑ Backup	5 🚖 in minute	
		_	Cancel OK



12.0 USER MANUAL

If **Autosave** is clicked, the program automatically performs a save periodically in accordance with the adjusted number of minutes. If **Backup** is clicked, *ConSteel* creates a backup file *after a manual save is performed*. The backup model file can be used as a normal model by removing the **.bak** extension.

Update

ConSteel 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 (

ptions	×
 □ Options □ Save □ Update □ Language □ Styles □ Diagnostics □ Export 	 ✓ Automatic search for updates Search for updates now ✓ Send usage statistics ✓ Send usage statistics 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. KESZ Holding cRc uses this information to improve the products and features customers use most often and to help solve problems.
	Cancel OK

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.

ConSteel has to be restarted after a change.

- Options		
Save Update Language Styles Diagnostics Export	The language of the user interface after next start.	Imagiyar Imagiyar
		Cancel OK

Styles

Color of the objects can be changed.

··· Update	Name	According to the attribute	Color	~
···Language ··· <mark>Styles</mark>	Point			
··· Diagnostics	Line			
Export	Arc			
	Two-dimensional figure			
	Steel			
	Concrete			
	Reinforcement			
	Bolt			
	Section	Material	-	
	Bar member	Section	-	
	Plate element	Material	-	
	Rigid body			
	Diaphragm			~



Diagnostics

Options		×
Options Save Update Language Styles Diagnostics	Minimal distance of structural elements (1 <error <1000)<="" <warning="" td=""> Error 20 Warning 50</error>	
- Export		
	Cancel	ОК

Before the analysis the program checks possible modeling mistakes.

There are two different levels of worn the user for possible modeling mistakes:

- Error: if the distance between the endpoints of two members is more than 0 but less than the adjusted value, the program sends an error message and shows with red sign the relevant members in the diagnostics window and does not perform the analysis.
- Warning: if the distance between the endpoints of two members is more than the defined error level distance, but less than the adjusted value here, the program sends a warning message, and shows with yellow sign the relevant members in the diagnostics window.

Export

There is some third parties program which required direct (such as IDEA StatiCA) connection for Export.

Options	X
Options Save Update Language Styles Diagnostics Export	IDEA StatiCa interface Installation path of IDEA StatiCa C:\Program Files\IDEAStatiCa\StatiCa8 Notes ConSteel - IDEA Connection interface can be operated only with the Version 7.1 and 8.1 of IDEA StatiCa! Please keep in mind the interface was verified with the Version 7.1.2.40942 and 8.1.15 44633 of IDEA StatiCa. Please always check- especially ifyou use another IDEA StatiCa version - all of the geometry parameters and loadings of the exported connection(s)!

On this menu, all of the necessary settings can be made.

STANDARDS menu:

5

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	₽-EN1990 Eurocode 0: Basis of structural design	^	Action type	ψ0	ψ1	ψ2
EN Recommend			Imposed loads in buildings, category (see EN 1991-1-1)			
EN Recommend			Category A: areas in residential buildings	0,70	0,50	0,30
EN Hungarian N	EN 1991 Eurocode 1: Actions on structures		Category B: office areas	0.70	0.50	0.30
MSZ 27003-1-5	Part 1-3: General actions – Snowload		Category C: congregation areas	0.70	0.70	0.60
EN Dutch NA	Table 5.1 - Values of Ce for different topographies 5.2(8) - Thermal coefficient Ct		Category D: shopping areas	0,70	0,70	0.60
EN Singapore N	Part 1-4: General actions – Wind actions			1,00	0,90	0,80
EN Portuguese	4.2(1) - Basic wind velocity		Category E: storage areas			
EN Swedish NA			Category F: traffic area, vehicle weight ≤ 30 kN	0,70	0,70	0,60
EN Austrian NA			Category G: traffic area, 30 kN < vehicle weight ≤ 160kN	0,70	0,50	0,30
EN German NA			Category H: roofs	0,00	0,00	0,00
EN Bulgarian NA	4.5(1) - Air density		Snow loads on buildings (see EN 1991-1-3)			
EN Greek NA	Other 7.1 - Recommended values of external pressure		H>1000m	0,70	0,50	0,20
			H≤1000m	0,50	0,20	0,00
EN Romanian N			Wind loads on buildings (see EN 1991-1-4)	0,60	0,20	0,00
EN Croatian NA	7.3.b - External pressure coefficients for monop 7.4.a - External pressure coefficients for duopit		Temperature (non-fire) in buildings (see EN 1991-1-5)	0,60	0,50	0,00
Spanish	 7.4.b - External pressure coefficients for duopit 7.10 - Frictional coefficients Cfr for walls, parap EN 1992 Eurocode 2: Design of concrete structures -2.4.2.4 - Partial factors for materials -3.1.6 - Design compressive and tensile strengths -6.4 - Punching EN1993 Eurocode 3: Design of steel structures Part 1-1: General rules and rules for buildings -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 1				
Create or delete user defined	standard parameters			Cance	el	ОК



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

For creating a new, user defined standard:

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

Create or dele	te user defined stand	ard parameters		
New	As a copy of	EN Recommended Value 🔻 Nam	e user_def	- 📀 😳 📗
	_		_	_

- 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

1.2.4 TABS



The tabs contain the systematically collected functions of modeling, analysis and design, leading the engineers through the logical steps of the structural design.

By approaching any of the icons with the cursor, the short name of the icon will be appeared.

GEOMETRY TAB



It contains all the important CAD drawing and modification functions as well as dimensioning and measuring tools.



STRUCTURAL MEMBERS TAB

🤹 New mo	del - Design standard: EN	Recommende	d Values - ConSt	eel 12 - x64					
File Edit	View Options Standa	rds Help							
Geometry	Structural members	oads Mass	Finite element	Analysis	Global checks	Member checks	Servicea	bility checks	
I, I	💋 📒 🛹 🐚 🥽	1 🗳 🎸	🐂 🐙 🔺		> 🖽 📠	* 🖍 🖗	🥔 🛅		₹

The functions related to cross-sections; structural columns, beams, slabs and walls, frame corner wizard, diaphragm and rigid body, supports and connections are collected on this tab. The automatic members to plate conversion function can be also found on this tab.

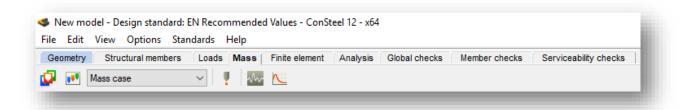
LOADS TAB



All types of loading including load cases, load groups, load combinations and unique loads placed on the structure can be created with the *Loaps* tab. The functions of meteorological load generation, crane and train load, and fire effect and protection can be found also here.

MASS TAB

Mass cases, mass groups, point masses, response spectrums and seismic effects can be created on the *Mass TAB*.



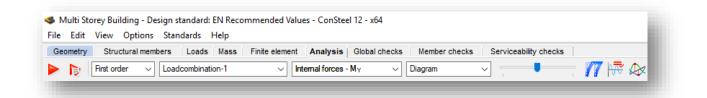


FINITE ELEMENT TAB

e Edit Vi	iew Options Star	ndards Help					
Beometry	Structural members	Loads Mas	Finite element	Analysis	Global checks	Member checks	Serviceability checks
	ConSteel	~	Loadcase	~	1		

It contains functions connected with the finite element mesh generation and modification and a model check option indicating possible modeling mistakes.

ANALYSIS TAB



The types of the structural analysis can be set and executed, the results can be viewed and labeled in various forms.

GLOBAL CHECKS TAB

ile Edit	View Options Star	ndards l	Help					
Geometry	Structural members	Loads	Mass	Finite element	Analysis	Global checks	Member checks	Serviceability checks
Steel	- Dominant calculation			v 0	ία _{cr}	~		~

The **GLOBAL CHECKS** tab contains the options and results of possible cross sections checks as well as the global buckling standard checks.

MEMBER CHECKS TAB

eom	etry	Structural members	Loads	Mass	Finite element	Analysis	Global checks	Member checks	Serviceability checks
P		First order	~	Internal fo	orces - N	~ Contou	ır surface 🗸		- 77

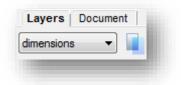


Steel member design related checks functions (buckling, lateral-torsional buckling and interaction stability) and the composite beam design related functions can be found on the **MEMBER DESIGN** tab.

Serviceability checks tab

All serviceability related functions (horizontal and vertical deflection checks) can be found on the **SERVICEABILITY CHECKS** tab.

LAYERS TAB



It provides functions for adjusting the layer properties.

DOCUMENT TAB



The functions of the **DOCUMENT** tab allow the flexible creation of a detailed statical documentation and contain a model information tool.



1.2.5 THE SIDE BAR

The side bar at the left of the main screen contains the mostly used functions for modeling.

	Save model
5	Undo/Redo
Ŷ₽\$>	Settings for the coordinate system, and snapping grids
	Model views: Top, Front, Side, Isometric, Perpendicular to raster
	Line view visualization of the model
	Wireframe visualization of the model
	Hidden line view visualization of the model
	Solid view visualization of the model
	Move point and edge
20	Move/Copy the selected objects
Ъ	Mirror the selected objects
	Rotate the selected objects
⊵a Ģa	Make a section (works on shell models only)
	Select all objects
	Deselect all objects
	Invert selection
	Select by properties
	Fit view (Ctrl+0)
]

1.2.6 CSPI PANEL

At the bottom left of the screen, the csPI control panel can be found. csPI is a programming interface made specially for ConSteel, with which parametric models can be built. All of the objects which can be found in ConSteel can be created through csPI, and all of the parameters of these objects are available to set from csPI. Please see CHAPTER 15 CSPI-CONSTEEL **PROGRAMMIN INTERFACE** to know more!





1.2.7 The status bar

∆ <	X	500	Y 60000	Z 0	L 60002,1	 L.ヅ @ ٩	n 4	Step 250
				,				-

The first field is a progress bar showing the progress of actions.

The next 3 icons on the status bar ($\textcircled{A} \land \textcircled{A} \land \textcircled{A}$) specify the meaning of the coordinates during drawing/modeling actions, while values are entered manually after pressing the appropriate letter ("X, Y, Z, L" for the coordinate axis or a length into a direction, or "a, b, L" for polar coordinates). The user can change the interpretation of the manually entered values with the following settings:



Even if you are in Descartes coordinate system, you have already picked the first point of a new structural beam, while approaching another point with your cursor in the model space, after typing "L" (lowercase), you can enter the length of your beam manually into the L field. By pressing the Enter key the beam is added to the model with the length "L" and the direction defined by the two points.

By moving or clicking the slider (\square) with the right mouse button, the size-representation of the objects can be changed. The next four icons ($\stackrel{\text{T}}{}$ $\stackrel{\checkmark}{}$ $\stackrel{\checkmark}{}$ $\stackrel{\checkmark}{}$ $\stackrel{\checkmark}{}$) allow the sophisticated visibility adjustment. By approaching any of these icons with the cursor a group of graphic symbols will appear ordered into a matrix shape.

Clicking the first icon on each of these four visibility setting matrix the scope of the settings will be changed between global (0) (valid for all tabs), or valid only for the current tab ($\fbox{0}$).

By setting this option on any of the 4 visibility matrixes, the selected scope will change on all the other three accordingly.

Changing this scope will also change the visibility between the previously adjusted global and the current tab visibility settings.



Visibility options of graphic symbols ($\frac{1}{2}$)

-	
	- 🔣 L. 🏏 🗉 🥵

- Visibility of lines, created with line, circle and arc function of **GEOMETRY** tab
- Visibility of structural members created with the beam or column function in the STRUCTURAL MEMBERS tab; switching these members to non-visible their centerlines might still be visible accordingly to the status of line visibility settings (
- Visibility of pins (end releases)
- Visibility of the two dimensional shapes; switching these two dimensional shapes to non-visible their surrounding lines might still be visible accordingly to the status of line visibility settings (/)
- Visibility of the 3 D objects (plates, slabs and walls); switching these 3D objects to non-visible the surrounding lines and 2 D shape components might still be visible accordingly to the status of line visibility settings of lines (\checkmark) and 2d shapes (\Box)
- [▲] Visibility of supports
- The Visibility of link elements
- **F** Visibility of placed joints symbols
- Visibility of initial bow imperfection
- ↓ Visibility of point, line and surface loads



- Visibility of load transfer surfaces; switching these load transfer surfaces to nonvisible their surrounding lines might still be visible accordingly to the setting of visibility of the lines (
- 🐸 Visibility of distributed surface loads
 - Visibility of masses

Visibility options of the grid and the local coordinate system ($\stackrel{[]}{\vdash}$)

· · /	-11
	- 81
田 🕹	- 8
🛂 🗽 🖄 📠	R
	~

:::: Visibility of the grid

- Visibility of the local coordinate systems of structural members (beams and columns)
- Visibility of the local coordinate systems of three dimensional objects (plates, slabs and walls)
- △ Visibility of the local coordinate systems of supports
- Visibility of the local coordinate systems of link elements
- Visibility of the local coordinate systems of the load transfer surfaces

When the visibility of an object is off, the local coordinate system of the object also will not be visible!

Visibility options of object names ($\stackrel{\checkmark}{\sim}$)

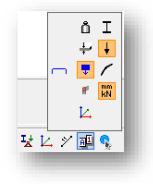
	• /	ы
	수 스	ы
\square	⊳ ≖	ш
	€ €	
	≪ #	
-L V: 14	💓 A	R



- Visibility of the number of finite elements (visible only on the *FINITE ELEMENT* and *ANALYSES* tabs)
- Visibility of names of bar elements
- Visibility of names of three dimensional objects (plates, slabs and walls)
- △ Visibility of names of supports
- Visibility of names of pins (end releases)
- The Visibility of names of link elements
- **F** Visibility of the name of placed joints symbols
- Visibility of the name of the load transfer surfaces
- Visibility of the name of the shear fields
- Visibility of the name of the reinforcement objects

When the visibility of an object is off, the name of it also will not be visible!

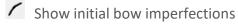
Visibility options of labels (🕮)



ů Show material grades



- ✤ Show the thickness of plates in mm
- ↓ Show the load intensity





- Show the units of quantities
- Show labels of the local cordinate axes
 - Show labels of frame corner zones

Show labels of masses

Action point sets (🛸)

The **Action point sets** offer a wide range of setting the point snapping functions:



Snapping the end points of graphical (lines, arcs) and structural (beams, columns) objects

Switch divide function ON / OFF

► ON:

When the Divide snap mode is turned ON, a new field will appear on the status bar. There are 3 options for the divide snap point specification. The user can switch them by clicking on the icon before the numeric field:

- Solution with the second se
- Image: Market Market
- n 4, clicking to the d icon, the number of division can be specified. The created snapping points are the intersection points of

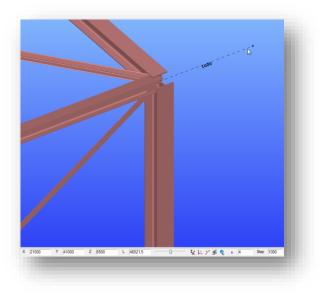


subdivided lengths. There is no rest distance at the end of the element

- ▶ OFF: The field of division will disappear from the bottom status bar
- Snapping the intersection points of graphical (lines, circles, arcs) and structural (beams, columns) objects
- Snapping the parallel point to a linear object
- Snapping the nearest element point of an object
- Snapping the lengthened point of a linear object. The system is showing the actual distance of the snapping point in [mm] from the endpoint of a linear element.
- Switches ON/OFF the snap points of the raster.
- Snapping point to create a tangent of an Arc / Circle from one point
- A Snapping point to create a perpendicular from one point to an object (Line or Arc/Circle)
- Snapping the Center point of Arc / Circle.

For snapping the center of linear elements, you have to use the appropriate settings of snap divided points, see above!

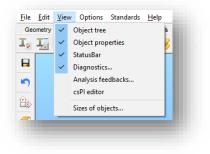
The last field of the Status bar is the STEP field. Here the given number in mm is the snapping distance towards the length direction of line and bar elements, when the lengthening snapping point is ON.





1.2.8 The windows of Object tree, Diagnostics results and Object properties

The visibility of the windows of the main screen can be switched ON / OFF in the **VIEW** menu.



Object tree window

The Object window has two tabs:

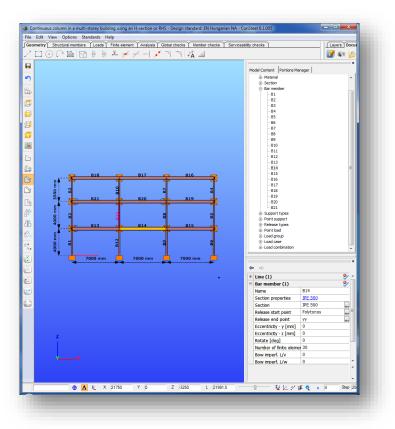
MODEL CONTENT tab

Model Content	Portions Manager	
🔅 - Layer		
🗄 - Materia		
Section		
🔅 - Bar mer	nber	
🔅 - Elemen	group	
🔅 - Haunch		
Support	types	
🔅 - Point su	pport	
Release	types	
🕀 Line loa	d	
🕀 - Load gr		
Load ca		
🗄 - Load co		
Placed :	oint	

By default the **MODEL CONTENT** tab contains all the predefined basic object types which can be used to build up a structural model.

While modeling, each new object (materials, sections, loads, members, supports etc.) will appear in the tree object structure in the appropriate group.

Selecting any of the objects, it will be highlighted (selected)





in the model graphic. Multiple selections of objects are possible in this tree.

PORTIONS MANAGER tab

Model Content Portions	
A 🖛 者	' 🚅 🚔 😤 🐘 📲
Portions	
Parame 5	
- Frame 8	
🖂 🛱 Column	
🗌 🛱 Beam	
🛄 📄 Storeys	
Show hidden parts of t	the structure translucently

Please find the description of the **PORTIONS MANAGER** tab in **3.2 MODEL VIEWS**, and in **3.4 PORTIONS MANAGER** chapters below!

Diagnostic window

In case of geometrical or modeling mistakes (like overlapping two or more object, missing supports, etc..) are detected, the Diagnostic windows visibility will switch ON automatically showing the name of problematic objects in a tree structure.



The first type of diagnostic results are the error messages appearing in red color.

The second type of the diagnostic result are possible errors in yellow color.

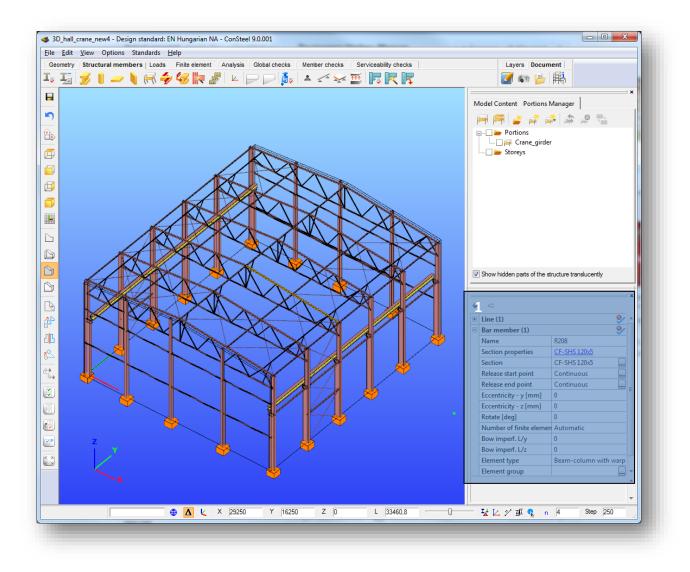


By clicking on any of the object name in tree structure, and pressing the **SELECT** button, the selected object will be highlighted in the model graphic. You can select more objects at the same time clicking their name while pressing the **SHIFT** or **CTRL** key.

The selected object can be erased by pressing **Delete** button at the bottom of the Diagnostic window or using the **Delete** key on the keyboard.

Object properties window

By selecting one object in the model space all the relevant properties of it will appear in this window (#1). The values of the parameters can be overviewed and in most cases these parameters can be changed.



All selected objects (and sub-object) appear in the table. By expanding one object all it's parameters can be seen and modified if possible. In the case of a multiple objects selection only the identical parameters of all objects appear, however the different ones can also be changed to identical.



With the unselect button ($^{\textcircled{}}$) previously selected object type can be removed from the selection.

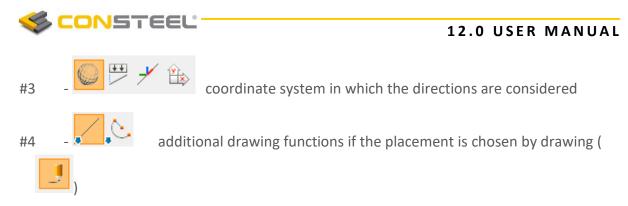
1.3 General structure of dialogue windows

Several dialogue windows – especially the most important ones on the *STRUCTURAL MEMBERS* and *LOADS* tabs – show the same structure in order to make the orientation in the labyrinth of tools and functions easy. The usual parts of these dialogues are described below:

q5(X,Y,Z)	kN/m	0 kN/m	-10 kN/m
q ₂ (X, Y, Z)	0 kN/m	0 kN/m	-10 kN/m
Position of q ₁ al	ong the line (x1)	From end A	√ 0 mm
		From end B	✓ 0 mm
Position of q ₂ al	ong the line (x _{2,r})	From end B	
-		From end B	
Position of q ₂ al Local eccentr Type		From end B	×

#1 - the method of placement (assign to an element or draw from one point to another)

#2 - extract data from a previously placed object. All the parameters are copied from the selected object



The main parameters of an object (#5) are placed in the middle part of the dialog.

1(X,Y,Z)	0	kN/m	0	kN/m	-10	kN/m
2(X,Y,Z)	0	kN/m	0	kN/m	-10	kN/m
osition of q ₂ a		×2,r)	From e	end B 🔻 🔻	0	mm
Туре		erence line			•	
	y [mm]	0	z [mm] 0	-	

The information field (#6) always gives information about the next required step of the placement.

i	Select the action line of load!	

Every icon on these dialogues has a tooltip with the name of the function. The text will appear when the mouse approaches the icon.

1.4 GENERAL FUNCTIONS FOR TABLES

Basically there are two types of tables in *ConSteel*: the input tables and the output tables. Since tables are usually for working with large amount of data, both table types have special features making the data managing more comfortable.



Input tables (used in the *LOAD COMBINATIONS* and *ANALYSIS PARAMETERS* dialogues) have the following common features:

- clicking on the heading cells the whole column below will be selected
- if there is a checking option in the table the multiple checking is possible for the selected cells clicking on the right mouse button

I Name	First or	1	Number of bucklin	Number of dynam	Earthquake calculation	
VULS) LC-1	~	V				
 (ULS) LC-2 	~					
 (ULS) LC-3 	~	✓				
✓ (ULS) LC-4	~					
 (ULS) LC-5 	~					
✓ (ULS) LC-6	 Image: A start of the start of					
 (ULS) LC-7 	 Image: A start of the start of					
 (SLS) SLS-1 						
 (SLS) SLS-2 		V				
 (SLS) SLS-3 						
 (SLS) SLS-4 						
 (SLS) SLS-5 						

if there is a value entry option in the table cells, multiple value entry is possible for the selected cells clicking on the right mouse button and defining the common value

Name	First or	Second or	Number of bucklin	Number of dynam	Earthquake calculation
 (ULS) LC-1 	 	 Image: A start of the start of			
(ULS) LC-2	 Image: A set of the set of the	✓			
(ULS) LC-3	 Image: A set of the set of the	✓			
(ULS) LC-4	~	~			
(ULS) LC-5	 	✓		Give common	value
(ULS) LC-6	~	~			
(ULS) LC-7	 	~			10
 (SLS) SLS-1 		✓			
(SLS) SLS-2		✓		Cancel	ОК
(SLS) SLS-3		✓			
(SLS) SLS-4					
 (SLS) SLS-5 					

► In case of editable tables, cell contents can be copied by pressing *CTRL+C* and pasted by pressing *CTRL+V* in the table or to MS Excel or to other spreadsheet applications



Output tables (used on the *ANALYSIS* and *STANDARD RESISTANCE* tabs and in the *DOMINANT VALUES* and *MODEL INFORMATION* dialogues) have the following common features:

- the rows of the table can be ordered according to a column (increasing, decreasing or ABC order) clicking to its heading cell
- the actual table can be saved to file for Excel
- the actual table can be saved for the own documentation which can be inserted into the final report in the documentation module (see CHAPTER 13 DOCUMENTATION)

1.5 Нот кеуѕ

The hot keys can significantly speed up the modeling work. *ConSteel* provides the following hot keys:

- **CTRL+N: new model**
- CTRL+O: open model
- CTRL+S: save model
- CTRL+Z: undo last action
- CTRL+Y: redo
- CTRL+A: select all
- **CTRL+I:** inverse selection
- CTRL+1: switch to XY view
- CTRL+2: switch to XZ view
- CTRL+3: switch to YZ view
- CTRL+4: switch to XYZ view
- **CTRL+5:** switches the view perpendicular to the raster
- CTRL+C: copy the selected cell contents from the editable tables
- CTRL+V: past the copied cell contents
- ESC: unselect all or terminate (or abort) the last action
- **Delete: delete selected objects**
- F1: open Help system



- **F3:** runs analysis with the actual settings
- **X**: manual definition of X coordinate value
- > Y: manual definition of Y coordinate value
- **Z**: manual definition of Z coordinate value
- > a: manual definition of alpha polar coordinate value
- **b**: manual definition of beta polar coordinate value
- L: manual definition of a length from the previous point in a defined direction
- R: switch between the global and last defined user coordinate system
- Middle mouse button: move model
- ALT+left mouse button: rotate model
- ALT+right mouse button: zooming model
- ► ↑: move model up
- ▶ ↓: move model down
- \rightarrow : move model right
- ► ←: move model left
- middle mouse button: scale model
- +: scale up model
- -: scale down model
- SHIFT+left mouse button: unselect
- SHIFT+ALT+left mouse button: window scale

CONSTEEL

2 FILE HANDLING

2.1 BASICS

The file handling of *ConSteel* follows the same usual and simple way of the MS Windows standard. The saved files contains all information about the model, it can be relocated to another folder and opened. By default the **.csm** file extension is associated with *ConSteel* and these files can be opened by double click.

2.2 FILE TYPES

The following file types are handled by *ConSteel*:

.csm: the ConSteelModel file, native binary file type. It can be opened with or without results. The result file is saved to a separate file with **.csr** (ConSteelResult) extension, but this file cannot be opened solely. For open and save.

.csm~: the ConSteelModel file created by the autosave functionality. AutoSave settings can be edited in the **OPTIONS** menu.

.csm.bak: the ConSteelModel backup file. Backup save settings can be edited in the **OPTIONS** menu. The Backup file is created at every manual save and stores the previous saved version of the model. If necessary the .bak extension can be deleted and backup model used as a normal model.

.anf: StruCad text file type. Only for open.

.csf: old ConSteel version binary file type (ConSteel 2.x, 3.x). Only for open.

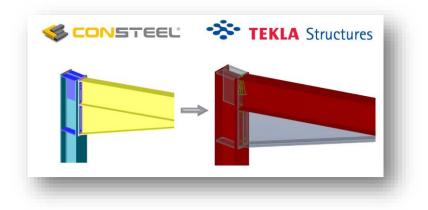
.asc: Tekla Structures ASCII file. Only for open.

- .dxf: AutoCad text file type. For open and save.
- .sc1: BoCad text file type. For open and save. (optional module is needed for Bocad!)
- .snf: StruCad text file type. For open and save.

STEEL

2.3 TEKLA STRUCTURES MODEL IMPORT, EXPORT AND UPDATE

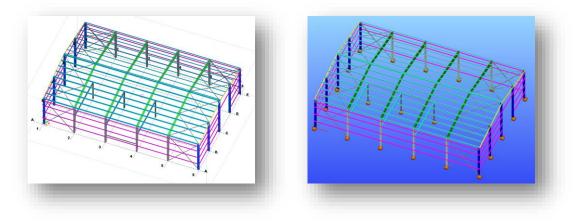
The smooth and quick transition from the structural model to the detailing model can save significant time and cost. Therefore *ConSteel* has a high level interface module to *Tekla Structures 64-bit*. Besides the export of the global structural model (ie. beams, columns, slabs, walls) all the joint models of *ConSteel* have the corresponding *Tekla* component. The designed structural model with the placed joint models can be converted into one detailed model without additional modeling efforts for the joints. This unique interface can save significant detailing time and therefore reduce appreciably the project costs.



ConSteel uses *Tekla* Open API (Application Programming Interface), also known as the .NET API. This provides an interface for third party applications to interact with the model in *Tekla Structures*. Therefore both applications need to have to be installed in order to export model from *ConSteel* to *Tekla Structures*.

2.3.1 IMPORT STRUCTURAL MODELL FROM TEKLA STRUCTURES

During the process, we have the possibility to import the structural members defined in Tekla Structures to ConSteel. From the structural parts of Tekla Structures, bar members can be imported (beams, columns, arched bars).





Import function is compatible with the following versions of Tekla Structures:

- 19.0, 19.1 64 bit
- 20.0, 20.1 64 bit
- 21.0, 21.1 64 bit
- 2016, 2016i 64 bit
- ▶ 2017, 2017i

For the import process from Tekla Structures, it is indispensable to run Tekla, and ConSteel in parallel. In Tekla Structure the model what is desired to be exported has to be opened, alongside with an empty model opened in ConSteel. For the proper importation process, it is recommended to use the Default environment of Tekla Structures. To start the import procedure, open *FILE* menu select the *IMPORT* tools and click on *TEKLA STRUCTURES*.

New model		bads Mass Finite element
Model attrib	ute	🖆 🌠 🐂 🖉 🔟
Open	Ctrl+O	/
Save	Ctrl+S	
Save As		
Import	>	DXF
Export	>	Dlubal (xlsx)
Update	>	SAP2000 (xlsx)
Exit		Tekla Structures
		IFC

On the opened import dialogue, we have the possibility to choose if the whole model, or just the selected parts of the model is desired to be imported to ConSteel from the parallelly running Tekla Structures model. If only the selected model parts are desired to be imported, these parts already has to be selected in Tekla Structures.

Furthermore, the layer handling has to be selected. By default the the imported model will use ConSteel layers, but optionally it is possible to use the layers defined by the classes in Tekla Structures. In this case, members of a Tekla Structure class can be imported to a new layer in ConSteel, what will use the colors of that class.

With the *IMPORT* button, the process can be initiated.

STEEL		12	2.0	USER	MANUAL
	🕃 TEKLA IMPORT				

All members		mport grid		
Selected members				
LAYERS				
Use the layers of 1	Tekla Structu	ires		
Import to ConSteel	ConSteel layer Layer_1		\sim	
_				
_		_	_	
C I	mport	X Ca	ncel	

In the next step, reference point has to be clicked in Tekla Structures. It is important to note, that it is important to check the alignment of the coordinate system in Tekla Structures. After selecting the reference point in *Tekla* the *Section IMPORT MANAGER* window appears. It is possible to change the conversion file for grades and for sections or edit the section name and grade manually.

	Import content	(ConSteel conversion			
C:\Program Files\ConSteel 12 Update 1 315\Data\Exp		MATERIAL	NEW NAME	^		
	WI600-12-20*200	S 235 H EN 10219-1	WI600-12-20*200-20*100 (S 235 H EN 10219-1)	×		
C:\Program Files\ConSteel 12 Update 1 315\Data\Exp	HEB400	S355	HEB 400 (S 355 EN 10025-2)	·		
	WI1000-20-25*50	S 235 H EN 10219-1	WI1000-20-25*500-25*500 (S 235 H EN 10219-1)	• _		
	WI200-753-12-17*.	. S 235 H EN 10219-1	WI200-753-12-17*263-17*263 (S 235 H EN 10219-1)	• _		
	HEA400	S355	HEA 400 (S 355 EN 10025-2)	-		
	WI600-200-12-19*.	. S 235 H EN 10219-1	WI600-200-12-19*220-19*220 (S 235 H EN 10219-1)	-		
	WI753-12-17*263	. S 235 H EN 10219-1	WI753-12-17*263-17*263 (S 235 H EN 10219-1)	-		
	IPE600	S355	IPE 600 (S 355 EN 10025-2)	-		
	WI1000-400-20-2	S 235 H EN 10219-1	WI1000-400-20-25*500-25*500 (S 235 H EN 10219-1)	-		
	SHS200*10	S355	SHS 200x10 (S 355 EN 10025-2)	-		
	0610*25	S 235 H EN 10219-1	0610*25 (S 235 H EN 10219-1)	- -		
	IPE200	S 235 H EN 10219-1	IPE 200 (S 235 H EN 10219-1)	-		
	WI753-200-12-17*.	. S 235 H EN 10219-1	WI753-200-12-17*263-17*263 (S 235 H EN 10219-1)	- -		
				ve to file	▷ Cont	inue 🗙 Cancel

The two main cases of the conversion:

- Automatic conversion:
 - Standard section library based conversion: Based on the conversion files, sections coming from Tekla structures can automatically assigned to a section from the Standard section library of ConSteel



- Standard macro based conversion: For those sections, which can not be converted directly to a standard library section, ConSteel will try to allign a so called "Standard macro" section based on the parameters of the imported section.
- Manual conversion: If based on the conversion file, ConSteel does not able to allign any standard library section for the imported section, and the parameters does not fit to any of the standard library cross sections, section conversion has to be performed manually. This can be done by either selecting a previously loaded cross section from the dropdown menu, or by clicking on the (...) button and selecting a proper section from the library. These automatically unconvertible sections are represented with their shapes at the properties part of the CONVERSION DIALOGUE.

If manual changes have been made the list can be saved and can be used as a conversion file in future exports. After clicking **OK** button the model will be exported.

2.3.1.1 LIMITATIONS OF THE IMPORT OF BAR MEMBERS

Bar members

- ► I and H profiles:
 - Hot rolled
 - Symmetric welded section with constant and variable height
 - Not-symmetric welded section with constant and variable height
- ► L profiles:
 - ▶ Hot rolled, with equal and unequal plates
 - Cold formed, with equal and unequal plates
- ► U profiles:
 - Hot rolled
- ► T profiles:
 - Hot rolled
- Box sections:
 - Symmetric welded section with constant height
 - Not-symmetric weldedsection with constant height (section will be defined with the thicker flange thickness)
 - WQ section can not be imported
- Roundbars:
 - Constant height
 - Sections with different diameter in the main and sub axes is not possible
- Hollow sections:
 - ▶ Hot rolled square and rectangular , constant height
 - In case of tapered member, parameters has to be set manually



- ► Tube sections:
 - ▶ Hot rolled sections with constant and variable height
 - Ellipse shaped section can not be imported
- Z profiles
 - Cold formed, folded edges
 - Cold formed sections without folded edges can not be imported
- C profilok
 - Cold formed, folded edges
 - Cold formed sections without folded edges can not be imported
- Multi-line ,continous beam can not be imported

Plates

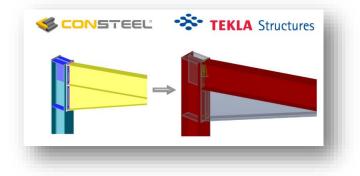
Contour plates can not be imported

Connections

Macros defined in Tekla Structures can not be imported

2.3.2 COMPLETE MODEL EXPORT INTO TEKLA STRUCTURES

Besides the export of the global structural model (ie. beams, columns, slabs, walls) all the joint models of *ConSteel* have the corresponding *Tekla* component. The designed structural model with the placed joint models can be converted into one detailed model without additional modeling efforts for the joints. This unique interface can save significant detailing time and therefore reduce appreciably the project costs.



The *ConSteel* export to *Tekla Structures* can be used from version 19 onward.

To export a model from ConSteel it is important to run both programs at the same time and to have a model opened in *Tekla Structures*. Then click on the *FILE* menu and select *Export* then select *TEKLA STRUCTURES*.



ile	Edit View	Options Standa	rds Help
	New model		pads Mass Finite element Analysis Glo
	Model attribute	•	+ * + + - * ¬ ¬
	Open	Ctrl+O	
	Save	Ctrl+S	
	Save As		
	Import	>	\sim 1
	Export	>	Tekla Structures
	Update	>	Reinforcement DXF export
	Exit		IFC

A dialog window opens asking for selecting model parts to be exported. Keep in mind that only those joints will be exported which are placed in the model and selected before exporting. Exportation of the model grid is also possible if the checkbox is checked. The next step is to select a reference point and to click on the **Export** button.

🕑 TEKLA EXPORT	
All members	
Selected members Export connection	
Overwrite Tekla model	
Add to Tekla model	
Pick the reference point!	
⊘ Export <mark>× C</mark> ancel	

After selecting the reference point in *Tekla* the *Section conversion* dialogue appears. It is possible to change the conversion file for grades and for sections or edit the section name and grade manually.



	ConSteel content	 Import 	conversion	PROPERT	
Program Files\ConSteel 11\Data\Export\SecnameC	EXPORT NAME	MATERIAL	NEW NAME	PARAMETER	
	IPE 200	S235JR	IPE200	Name	
Program Files\ConSteel 11\Data\Export\MatnameC	IPE 300	S235JR	IPE300	Material	
	HEA 300	S235JR	HEA300		
	SHS 180x5	S235JR	SHS180*5		
			ि Save t	o file 🛛 🖒 Cor	ntinue X Cancel

If manual changes have been made the list can be saved (Save to file) and can be used as a conversion file in future exports. At the *PROPERTIES* part of the dialogue, the parameters of the selected member is shown. After clicking **OK** button the model will be exported.

2.3.2.1 LIMITATIONS OF THE EXPORT OF JOINTS AND BAR MEMBERS

Bar members

- Arched members are not exported
- Compound sections

Semi rigid or pinned base plate \rightarrow Base Plate (1042)

- No haunch can be exported
- Anchor rod type must be set manually in *Tekla* (*ConSteel* settings has no effect on the export)
- Anchor rod properties which have effect on the export are the followings: diameter, grade
- Foundation will not be exported
- Upper and lower flange weld thickness would be the same in *Tekla* model(different *ConSteel* settings has no effect on the export)

Beam to column, moment end plate with haunch \rightarrow Haunch (40)

- Upper and lower haunch dimensions are the same (upper haunch dimensions come from the lower)
- No shear stiffeners are exported into Tekla only supplementary web plate



Flange stiffeners are not exported into Tekla

Beam to column, simple end plate \rightarrow Haunch (144)

Column top height modification must be set manually

Beam to column, simple end plate \rightarrow Haunch (146)

Beam splice connected with a pitch, simple or moment end plate without haunch \rightarrow Haunch (40)

Left and right sided macros are exported separately, explosion of components and redefinition of bolt parts are needed in Tekla

Beam splice connected in line, simple or moment end plate without haunch \rightarrow Joining plates (14)

Beam splice, moment end plate with haunch \rightarrow Apex haunch (106)

No upper haunch

Beam splice plate \rightarrow Splice connection (77)

Bolted circular flange plate \rightarrow Round joining plates (124)

Stiffened end plate (27)

> There is no web stiffener plate in Tekla

Beam to beam, simple end plate \rightarrow End plate (144)

Beam to beam, web fin plate \rightarrow Fin plate (146)

Gusset plate for hollow and tube brace sections \rightarrow Tube gusset (20)

Bolted gusset plate for angle brace \rightarrow Bolted gusset (11)

2.3.3 Change management between Tekla and ConSteel: Tekla update

Besides the import and export functions *ConSteel* can check and update *Tekla Structures* models during the detailing. Update of *Tekla Structures* model is always based on *ConSteel* model with the following attributes:

- Element name
- Material name
- Section name
- Section parameters
- Element geometry and position



For doing model update a complete model import (or export) process is necessary before.

To update Tekla model it is important to run both programs at the same time and to have a model opened in *Tekla Structures*. Then click on the *FILE* menu and select *Export* then select *TEKLA STRUCTURES UPDATE*.

🕑 TEKLA UPDAT	E		
• All members	📕 Export	grid	
Selected members	📃 Export	conne	ction
N	0	~	0 1
	Continue	×	<u>C</u> ancel

A dialog window opens and we have the possibility to choose if the whole model, or only the selected parts of the model is desired to be exported. Then click on the **CONTINUE** button.

During the process *Tekla Structures* model and parts will be updated. In case of conflict the following dialog will be appeared and users can make the decision and applied which model parts will be used.

	Model co	ontent				Search D	PROPERT	
4	CONSTEEL STATUS 🔻	TEKLA STATUS 🔻	NAME ¥	MATERIAL ¥	ТҮРЕ 🔻	CONVERSION STATUS ¥	PARAMETER	
Change level	CS UNCHANGED	18 DELETED	B1200	S355 EN 10025-2	Column	No conversion 🔻	Name	
	CS UNCHANGED	B DELETED	B3241	S355 EN 10025-2	Column	No conversion 🔻	Section	
	CS UNCHANGED	18 DELETED	B0241	S355 EN 10025-2	Column	No conversion 🔻	Eccentricity Y	
HANGES IN TEKLA	CS UNCHANGED	18 DELETED	B3041	S355 EN 10025-2	Column	No conversion 🔻	Eccentricity Z	
113 NEW elements will be created.	CS UNCHANGED	TS DELETED	B2941			No conversion 🔹		
	CB CHANGED	TS UNCHANGED	B2941	S235 EN 10025-2	Beam	No conversion 🔻		
3 elements will be MODIFIED.	CS CHANGED	TS CHANGED	B2941	S235 EN 10025-2	Beam	No conversion 🔻		
200 elements will be DELETED.	CHANGED	18 DELETED	B2941	S235 EN 10025-2	Beam	No conversion 🔻		
JELETED.	CB CHANGED	TS DELETED	B2941	S235 EN 10025-2	Beam	No conversion 🔻		
TERS	CS DELETED	TS UNCHANGED	B2941	S235 EN 10025-2	Column	No conversion 🔻		
NMODIFIED ×	CS DELETED	TS UNCHANGED	B2941	S235 EN 10025-2	Column	No conversion 🔻		
HANGED X	CS DELETED	TS UNCHANGED	B2941	S235 EN 10025-2	Column	No conversion 🔻		
ELETED X	CS DELETED	(IS) CHANGED	B2941	S235 EN 10025-2	Column	No conversion 🔻		
2941×	CS DELETED	(B) CHANGED	B2941	S235 EN 10025-2	Column	No conversion 🔻		

Clicking on **UPDATE** button the whole process will be finished. New elements created in *ConSteel* will be automatically exported to *Tekla Structures*. The next table consists the merge cases.



STEEL	두 Tekla	UPDATE / MERGE
CS UNCHANGED	TS UNCHANGED	-
CS UNCHANGED	TS CHANGED	Merge
CS UNCHANGED	B DELETED	Merge
CS CHANGED	19 UNCHANGED	UPDATE
CS CHANGED	T S CHANGED	Merge
CS CHANGED	B DELETED	Merge
CS DELETED	19 UNCHANGED	Merge
CS DELETED	B CHANGED	Merge
CS DELETED	B DELETED	-

2.4 IFC IMPORT AND EXPORT

IFC is a global standard for data exchange in the building industries. IFC is both a common data model and an open file format. Building industry professionals can use IFC to share data regardless of what software application they use to get their job done.

The IFC specification is developed and maintained by buildingSMART International as its "Data standard".

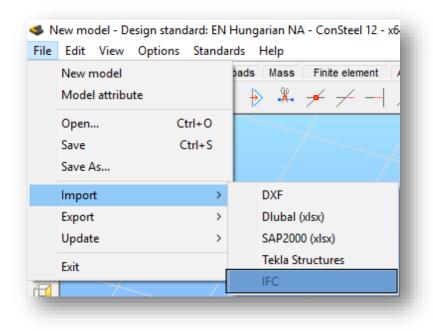
2.4.1 IMPORT OF IFC FILES

The IFC import function of ConSteel supports the importation of those ifc files, which were exported with the following format scheme:

► IFC 2x3

The IFC import function can be initiated from the *IMPORT* submenu of *FILE* menu, with the *IFC* button:





After selecting the IFC function, the import dialogue will appear. The path of the ifc file which should be imported can be given directly, or it can be browsed with the folders button. At the scale options, scaling of the imported model can be adjusted. Origo of the placement can be defined if it is desired, but if it is not, ConSteel will place the imported file into the global origo. For the plane of the placement the main planes, and user defined coordinate systems are also available to use.



After clicking on the *IMPORT* button, the *SECTION CONVERSION* dialogue will appear, where section and material data from the ifc file can be assigned to ConSteel sections and materials. ConSteel will automatically try to assign sections to all members, using the section and



material conversion table (located in the installation folder of ConSteel), but manual assignment is also possivle. In the left column of the conversion table, section names from the ifc file can be seen, followed by the material names coming from the ifc file. The next column stands for the assigned ConSteel sections and materials.

	Import content	\rightarrow	ConSteel conversion			
C:\Program Files\ConSteel 12 Update 1 315\Data\Exp	IMPORT NAME	MATERIAL	NEW NAME	^		
		. S 235 H EN 10219-1	WI600-12-20*200-20*100 (S 235 H EN 10219-1)	.		
C:\Program Files\ConSteel 12 Update 1 315\Data\Exp	WI1000-20-25*50	S 235 H EN 10219-1	WI1000-20-25*500-25*500 (S 235 H EN 10219-1) .		
	HEA400	S 355 MH/MLH EN 10219-1	HEA 400 (S 355 EN 10025-2)			
	WI753-12-17*263	S 235 H EN 10219-1	WI753-12-17*263-17*263 (S 235 H EN 10219-1)			
	IPE600	S 355 MH/MLH EN 10219-1	IPE 600 (S 355 EN 10025-2)	_		
	SHS200*10	S 355 MH/MLH EN 10219-1	SHS 200x10 (S 355 EN 10025-2)	_		
	HEB400	S 355 MH/MLH EN 10219-1	HEB 400 (S 355 EN 10025-2)	_		
	IPE200	S 235 H EN 10219-1	IPE 200 (S 235 H EN 10219-1)			
	WB1000-16-25*6	S 235 H EN 10219-1	WB1000-16-25*600-25 (S 235 H EN 10219-1)			
	IPE750*137	S 355 MH/MLH EN 10219-1	IPE 750x137 (S 355 EN 10025-2)			
	HEB800		HEB 800 (S 355 EN 10025-2)	_		
	WB1000-20-30*6	S 235 H EN 10219-1	WB1000-20-30*600-30 (S 235 H EN 10219-1)	_		
	HEB600	S 355 MH/MLH EN 10219-1	HEB 600 (S 355 EN 10025-2)			
	1		· · · · · · · · · · · · · · · · · · ·	Save to file	D Cont	inue 🗙 Cancel

There is two possible for the assignment of sections:

- Straightforward assignment: In this case the sections in the ifc file can be found int he conversion tables, and ConSteel will automatically assign the correct section
- > Partially straightforward assignment:
 - *"Standard macro sections" case*: If the section int he ifc file can not be found int he section conversion table, but the parameters of it fits to a standard type of section (IPE for example), ConSteel will create a so called *"standard macro section"*, and assign itt o the section from the ifc file.
 If ConSteel is not able to assign any standard macro section to a section, using
 - the (...) button, sections can be loaded from the section library at any time
 - Macro section assignment: In case of sections, where the assignment can not be performed, ConSteel will drav the shape of the section at the **PROPERTIES** side of the dialogue, providing some guidance about the type of the section, so the correct macro section can be loaded from the section administrator, using the (...) button.

After the assignments of the sections are performed, the table can be saved into a user conversion file with the (Save to file) button, and can be used later at any time. By pressing the **CONTINUE** button, the import process will be performed.



2.4.1.1 AVAILABLE OBJECTS FOR THE IMPORT PROCESS

Steel members:

- ► I profiles:
 - Members in general positions, straight or curved, hot rolled sections with constant flange thickness symmetric I sections
 - Members in general positions, straight or curved, welded sections with constant flange thickness symmetric I sections
 - Members in general positions, straight or curved, welded sections with constant flange thickness asymmetric I sections
 - Members in general positions, straight or curved, hot rolled sections with sloped flange symmetric I sections
- H profiles:
 - Members in general positions, straight or curved, hot rolled sections with constant flange thickness H sections
 - Members in general positions, straight or curved, welded sections with constant flange thickness H sections
- ► T profiles:
 - Members in general positions, straight or curved hot rolled T sections
 - Members in general positions, straight or curved welded T sections
- ► U profiles:
 - Members in general positions, straight or curved hot rolled U sections
 - Members in general positions, straight or curved welded U sections
 - Members in general positions, straight or curved hot rolled, sloped flange U sections
- Hot rolled hollow sections:
 - Members in general positions, straight or curved hot rolled hollow sections
- Cold formed hollow sections:
 - Members in general positions, straight or curved cold formed hollow sections
- Hot rolled tube sections:
 - Members in general positions, straight or curved hot rolled tube sections
- Cold formed tube sections:
 - Members in general positions, straight or curved cold formed tube sections
- Hot rolled L profiles:
 - Members in general positions, straight or curved hot rolled, equal or unequal flange length L sections
- Cold formed L profiles:
 - Members in general positions, straight or curved cold formed, equal or unequal flange length L sections
- Cold formed C profiles:

CONSTEEL

12.0 USER MANUAL

- Members in general position, straight or curved cold formed C sections with edge
- Members in general position, straight or curved cold formed C sections without edge
- Members in general position, straight or curved cold formed C sections with edge and sloped flanges
- Lindab C profiles:
 - Members in general position, straight or curved cold formed asymetric C sections with edge
- Welded C profiles:
 - Members in general position, straight or curved welded C sections without edge
- Cold formed J profiles:
 - Members in general position, straight or curved, cold formed J sections
- Cold formed U profiles:
 - Members in general position, straight or curved, cold formed U sections
- Cold formed Z profiles:
 - Members in general position, straight or curved cold formed Z sections with edge
 - Members in general position, straight or curved cold formed Z sections without edge
 - Members in general position, straight or curved cold formed Z sections with edge and sloped flanges
- Plate profiles:
 - Members in general position, straight or curved plate sections
- ► Box profiles:
 - Members in general position, straight or curved welded box sections
- ► Maltese/Half maltese cross profiles:
 - Members in general position, straight or curved welded maltese/half maltese cross sections
- ► WQ profiles:
 - Members on general position, straight or curved, welded WQ sections

Reinforced concrete members:

- Members in general position, straight, T shaped sections
- Members in general position, straight, circle shaped sections
- Members in general position, straight, hollow circle shaped sections
- Members in general position, straight, rectangle shaped sections
- Members in general position, straight, hollow rectangle shaped sections
- Members in general position, straight, trapezoid shaped sections
- Members in general position, straight, I shaped sections



Plates

- Horizontally positioned rectangle plates without holes, with or without chamfer or rounding
- Horizontally positioned poligonal shaped plates without holes, with or without chamfer or rounding
- Horizontally positioned circle shaped plates without holes, with or without chamfer or rounding
- Horizontally positioned rectangle plates with general (inside the plate) holes, with or without chamfer or rounding
- Horizontally positioned poligonal shaped plates with general (inside the plate) holes, with or without chamfer or rounding
- Horizontally positioned circle shaped plates with general (inside the plate) holes, with or without chamfer or rounding

Wall plates

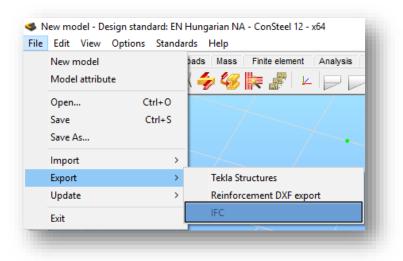
Vertically positioned plates without holes, with or without chamfer or rounding

2.4.2 EXPORT OF IFC FILES

Members from ConSteel can be exported from into ifc format, too. The export process uses the following file format:

► IFC 2x3

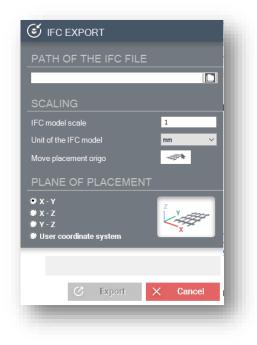
To export a model to *IFC* format, press the IFC button in the *Export* submenu of *FILE* menu.



After pressing the *Export* button, the *IFC Export DIALOGUE* will appear, where the path of the saved file can be set, besides the scaling parameters and the planes of the exported model.



Pressing the **EXPORT** button again, will initiate the process, and the whole model will be exported.



2.4.2.1 AVAILABLE OBJECTS FOR THE EXPORT PROCESS

Steel members:

- ► I profiles:
 - Members in general positions, straight or curved, hot rolled sections with constant flange thickness symmetric I sections
 - Members in general positions, straight or curved, welded sections with constant flange thickness symmetric I sections
 - Members in general positions, straight or curved, welded sections with constant flange thickness asymmetric I sections
 - Members in general positions, straight or curved, hot rolled sections with sloped flange symmetric I sections
- ► H profiles:
 - Members in general positions, straight or curved, hot rolled sections with constant flange thickness H sections
 - Members in general positions, straight or curved, welded sections with constant flange thickness H sections
- T profiles:
 - Members in general positions, straight or curved hot rolled T sections
 - Members in general positions, straight or curved welded T sections
- U profiles:

- Members in general positions, straight or curved hot rolled U sections
- Members in general positions, straight or curved welded U sections
- Members in general positions, straight or curved hot rolled, sloped flange U sections
- Hot rolled hollow sections:

CONSTEEL

- Members in general positions, straight or curved hot rolled hollow sections
- Cold formed hollow sections:
 - Members in general positions, straight or curved cold formed hollow sections
- Hot rolled tube sections:
 - Members in general positions, straight or curved hot rolled tube sections
- Cold formed tube sections:
 - Members in general positions, straight or curved cold formed tube sections
- Hot rolled L profiles:
 - Members in general positions, straight or curved hot rolled, equal or unequal flange length L sections
- Cold formed L profiles:
 - Members in general positions, straight or curved cold formed, equal or unequal flange length L sections
- Cold formed C profiles:
 - Members in general position, straight or curved cold formed C sections with edge
 - Members in general position, straight or curved cold formed C sections without edge
 - Members in general position, straight or curved cold formed C sections with edge and sloped flanges
- Lindab C profiles:
 - Members in general position, straight or curved cold formed asymetric C sections with edge
- Welded C profiles:
 - Members in general position, straight or curved welded C sections without edge
- Cold formed J profiles:
 - Members in general position, straight or curved, cold formed J sections
- Cold formed U profiles:
 - Members in general position, straight or curved, cold formed U sections
- Cold formed Z profiles:
 - Members in general position, straight or curved cold formed Z sections with edge
 - Members in general position, straight or curved cold formed Z sections without edge
 - Members in general position, straight or curved cold formed Z sections with edge and sloped flanges
- ▶ Plate profiles:
 - Members in general position, straight or curved plate sections
- Box profiles:



- Members in general position, straight or curved welded box sections
- Maltese/Half maltese cross profiles:

CONSTEEL

- Members in general position, straight or curved welded maltese/half maltese cross sections
- ► WQ profiles:
 - Members on general position, straight or curved, welded WQ sections

Reinforced concrete members:

- Members in general position, straight, T shaped sections
- Members in general position, straight, circle shaped sections
- Members in general position, straight, hollow circle shaped sections
- Members in general position, straight, rectangle shaped sections
- Members in general position, straight, hollow rectangle shaped sections
- Members in general position, straight, trapezoid shaped sections
- Members in general position, straight, I shaped sections

Plates

- Horizontally positioned rectangle plates without holes, with or without chamfer or rounding
- Horizontally positioned poligonal shaped plates without holes, with or without chamfer or rounding
- Horizontally positioned circle shaped plates without holes, with or without chamfer or rounding
- Horizontally positioned rectangle plates with general (inside the plate) holes, with or without chamfer or rounding
- Horizontally positioned poligonal shaped plates with general (inside the plate) holes, with or without chamfer or rounding
- Horizontally positioned circle shaped plates with general (inside the plate) holes, with or without chamfer or rounding

Wall plates

Vertically positioned plates without holes, with or without chamfer or rounding



2.5 DXF IMPORT

To import a dxf drawing file, click on *FILE* menu, *IMPORT* tools. Choose *DXF* from the available import functions.

New model		bads	Finite element	Analysis
Model attribute			, 🧐 😽 🚽	P 🗠
Open	Ctrl+O			
Save	Ctrl+S			
Save As				
Import	•		DXF	
Export	×.		Dlubal (xlsx)	
Exit			SAP2000 (xlsx)	
			Tekla Structures	

A windows file opening dialog will pop-up, where the dxf file has to be selected on the computer. After clicking OK, the drawing file will appear in the modeling area of *ConSteel*, and dxf import dialog will be opened automatically.

))	KF import			23
	Scaling			
	Unit of DXF model		mm	•
	DXF model scale		1	
	Move placement origo		k	
	Plane of placement			
	۲	X - Y		
	z y L T O	X - Z		
	I AA O	Y - Z		
	×	User coordina	te system	
	Layers			
	Import the layers of the E	XF model		
	Import to ConSteel layer			
	Fólia_1			-
		ОК	Car	ncel
	i			



On the DXF import dialog the following modifications can be made:

- Unit of dxf model (mm/cm/m)
- DXF model scale with scale factor
- Move placement origo (origo of the dxf file)
- Plane of placement

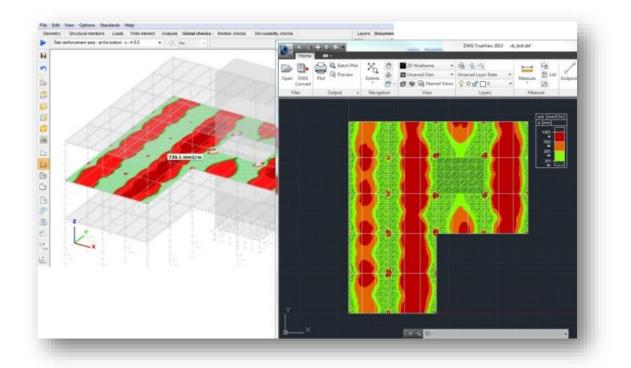
X-Y, X-Z, Y-Z of te global coordinate system, or User defined coordinate system)

Layer handling

Layers of the dxf drawing can be imported to ConSteel with its properties, or the whole dxf can be imported to an already existing ConSteel layer

With the **OK** button, the modifications can be applied, and the dxf drawing will be placed in the modeling area.

2.6 EXPORT OF REINFORCEMENT INTO DXF



With the export function, contour surface of the required reinforcements with the corresponding colur palette can easily be exported into dxf file format.

During the export, reinforcements contour surfaces for different directions (+x, -x, +y, -y) and their corresponding colour palettes will be placed on different layers

Objects, which will be exported with the funcion:



- ▶ Grid: Line, dashed line, or dotted line with the corresponding captions
- Plates with their contour lines
- ► FE division
- Labels such as name, plate thickness, material grade, if the visibility is turned on
- Colour palettes

The function can be initiated from the *FILE* menu's *EXPORT* functions:

New model		pads Mass Finite element Analysis
Model attribute		🖻 🧍 🗲 🥕 📝 🦄
Open	Ctrl+O	
Save	Ctrl+S	
Save As		
Import	>	
Export	>	Tekla Structures
Update	>	Reinforcement DXF export
Exit		IFC

After clicking ont he **REINFORCEMENT DXF EXPORT** function, the Reinforcement export dialogue will pop-up:

Reinforcement export	×
1 Export settings Export to one file Export portions separatley Coordinate system Global (XY) ~ Concrete palette unit cm2/m ~ Scale Unit of the DXF model mm ~ Scale of the DXF model 1/	Considered model portions Portions Prescription Portions Prescription Prescriptio
	Cancel Export

On the right (#2) side of the dialogue, the model portions are shown, while on the left (#1) side, export related adjustable settings are listed, such as:





- Export settings:
 - Export to one file
 - Export portions separately, into individual dxf files
- Coordinate system:
 - o Global coordinate system
 - o User defined
- Unit setting for the dxf file [mm, cm, m]
- Unit of the reinforcement colour palette [cm2/m, mm2/m]
- Scale of the dxf model by ratio

At the Considered model portions (#2) field, those portions can be selected which should be exported.

After clicking on the *EXPORT* button, the *SAVE* dialogue will appear, where path of the dxf file has to be given.

2.7 DLUBAL RSTAB AND SAP2000 MODEL IMPORT(.XLSX)

ConSteel is able to import model data, contained in an Excel (.xlsx) file, where different model data is stored on different worksheets. These table files can be exported from the softwares above.

To import the model data, click on *FILE* menu, *IMPORT* tools. Choose *DLUBAL(XLSX)* or *SAP2000(XLSX)* from the available import functions. Import settings for the two softwares works the same. After selecting the import function, the *IMPORT* dialog will appear:

Filename	4
Jnit of the model	mm
Direction of model Z axis	+Z •
Cross sections Cross sections Coss sections Code Code Concentrated forces	ts on members
	Cancel OK



For the import process, the following settings has to be made:

- Path of the file on the users coordinate has to be set by clicking on the sutton
- ▶ Unit of the original model file in the other software has to be set (mm/cm/m)
- Direction of the model Z axis has to be set, since it may be different in the other software
- ► The following model data can be imported
 - Cross sections (cross section parameters, rotation, eccentricities)
 - Nodal supports(rotation, eccentricities)
 - Loadcases
 - Nodal forces
 - o Concentrated forces on members
 - Member forces

It is important to know for the proper import process, that only those checkboxes should be checked in, which data is available in the xlsx file, or in other words, the model includes it.

After the settings are made, and clicked on the OK button, the import process will be initiated, and the **Section IMPORT MANAGER** will be shown. It is possible to change the conversion file for grades and for sections or edit the section name and grade manually.

Path of the section conversio	n file						
C:\Program Files\ConSteel 1	0\Data\Export\Secna	meConvDlubal.csv					
Path of the grade conversion file							
C:\Program Files\ConSteel 1		meConvDlubal.csv	(m)				
ease select the appropriate	section						
nport name	Material	New name					
PE 500 British Steel	\$355	IPE 500 (S 355 EN 10025-2)	▼				
PE 360 British Steel	\$355	IPE 360 (S 355 EN 10025-2)	·				
IPE 450 British Steel	\$355	IPE 450 (S 355 EN 10025-2)	×				
HE A 200 ArcelorMittal (\$355	HEA 200 (\$ 355 EN 10025-2)	·				
IPE 300 British Steel	\$355	IPE 300 (S 355 EN 10025-2)	×				
QRO 80x5.0 British Steel	\$355	RO 88.9x2.5 (S 355 EN 10025-2)	▼				
RD 24 ArcelorMittal	\$355	CHS 26.9x2 (S 355 EN 10025-2)	·				

If manual changes have been made the list can be saved () and can be used as a conversion file in future imports. After clicking **OK** button the model will be exported.

After clicking OK on the section import tab, the model will appear in the modeling area. It is also important to know, that the xlsx model export file has to be created using English language, otherwise the import process will not work.

2.8 EXPORTING MODEL DATA FROM DLUBAL RSTAB IN XLSX FORMAT

In order to export model data to an .xslx file, the following steps are necessary in Dlubal Rstab:

Click on *FILE* menu, *EXPORT* tools, and choose *MICROSOFT EXCEL* (*XLSX) format from the available options:

mat ormats for Frameworks DSTV Format - Members product interface for ated constructions ("#p) erg for Bocad, frie ESK/RS) Dentey ProStructure ("#p) Tekla Structures ("#p) Ortegraph ("#p) Advance Skeel ("#p) Ocadvak ("#p) ormats for Spreadsheet Økocoeft Kceel (#p)	General Formats for CAD Programs ASCII Format - Model Graphics of model to ASCII Fie DXF (* dd) Industry Foundation Classes - IFC (* fc) (#natytical Model (FC 23, e.g. for Softek, HofGraph) Bertley ISM (* ism dgn, * dgn) SDNF Format Steel detailing neutral file (* dat)	Direct Exports O Tekis Structures O Autodesk AutoCAD from vers. 2010)
) OpenOffice.org Calc (*ods)) CSV (*.cev)		

After clicking the **OK** button, the export setting dialog will appear. On the **EXPORT OF TABLES** part, data for the export has to be chosen. At **TABLES**, the "With table headers" and "Only filled rows" has to be checked in. At the **EXPORT TYPE**, "rewrite existing worksheet" is suggested to checked in.

Export of Tables		Tables	
🗹 1. Model Data		With table headers	
All		Only marked cells	
Only selected tables	2	Only selected obje	ots
		🗹 Only filled rows	
2. Loading		Only filled tables	
All	_		
Only selected tables	2	Export Type	
🗹 3. Load Data		Export tables to ack workbook	tive
 All 		Rewrite existing wa	orksheet
Only selected tables			
4. Results			
Only selected tables	2		
Formulas and Parameters			
Formulas in table			
Parameters to additional worksheet			
Formulas to additional worksheet			
2		Export	Cancel
		Lapon	Cancor



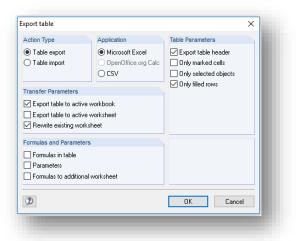
After clicking on the Export button, the Excel (xlsx) file will be generated, and opened. It is important to know, that if loads are desired to be exported too, the excel file should not be closed yet.

To export loads from a model, loadcases has to be exported one-by-one manually to the existing excel file. Export of Nodal loads and member loads can be performed on the bottom part of Dlublab main window:

. 1	~	0		Load	Load	Reference		Member Loa	ad Parameters			Distance	Over Tota	0
	Reference to	On Members No.	Load Type	Distribution	Direction	Length	p [kN/m]	P2		A	в	in %	Length	Comment
	Members	1	Eorce	Uniform	у	True Length	-1.566							Aus Windlast q: veränderlich (WZ 2, h: 8.000 m); cpe,A: -1.200; cpe,B: -0.800; cpe,C: -0.500; cpe,D: 0.720; cpe,E: -0.
Т	Members	1	Eorce	Uniform	z	True Length	1.055							Aus Windlast q: veränderlich (WZ 2, h: 8.000 m); cpe,A: -1.200; cpe,B: -0.800; cpe,C: -0.500; cpe,D: 0.720; cpe,E: -0.
T	Members	6	Eorce	Uniform	у	True Length	-0.842							Aus Windlast q: veränderlich (WZ 2, h: 8.000 m); cpe,A: -1.200; cpe,B: -0.800; cpe,C: -0.500; cpe,D: 0.720; cpe,E: -0.
Т	Members	6	Eorce	Uniform	z	True Length	-0.498							Aus Windlast g: veränderlich (WZ 2, h: 8.000 m); cpe.A: -1.200; cpe.B: -0.800; cpe.C: -0.500; cpe.D: 0.720; cpe.E: -0.
T	Members	7	Force	Trapezoidal	2	True Length	2.757	2.75	7	0.000	6.500			Aus Windlast g: veränderlich (WZ 2, h: 8.000 m); cpe.A: -1.200; cpe.B: -0.800; cpe.C: -0.500; cpe.D: 0.720; cpe.E: -0.

By chosing a loadcase at Nodal loads or Member loads, and clicking on the export to excel (

button, the window below will appear:



On the **EXPORT TABLE** dialogue "*Microsoft Excel*" format has to be choosen. Also, "*Export table to active workbook*" has to be checked in, so the data will be exported to the already opened model data. At **TABLE PARAMETERS**, the "With table headers" and "Only filled rows" has to be checked in.

After clicking on the **OK** button, the xlsx file can be imported to ConSteel.



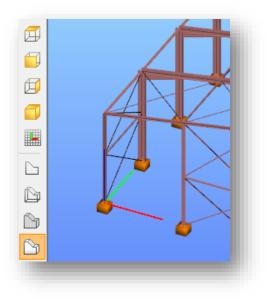
3 MODEL VIEW

3.1 BASICS

The transparent and multipurpose model views are very important to the user, since these views provide the first graphical model survey opportunity. Accordingly the first and most important objective of the model views to feedback the user with the objects placed, yield a visual proof that the right objects are at the right place.

3.2 MODEL VIEWS

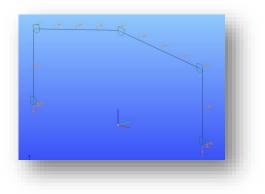
The model viewing options are on the left side bar.

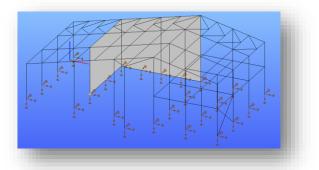


Beyond the usual model views (top view, front view, side view, axonometric view, perpendicular to raster view) there are fore visualizing options for the objects used:

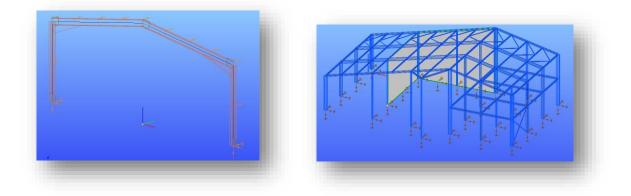
Ine view : the simplest model view, the bars are represented by single lines, the surfaces by a two-dimensional figure without thickness; the supports are line types. Recommended to use in the model building phases, since the clear visualization of the member snap points making the placement of supports and loads easier.



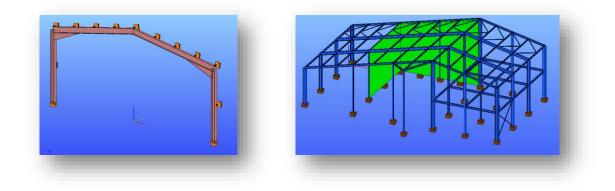




wireframe view : in the wireframe view the lines of the cross sections and plate thickness appear; the supports are line types.

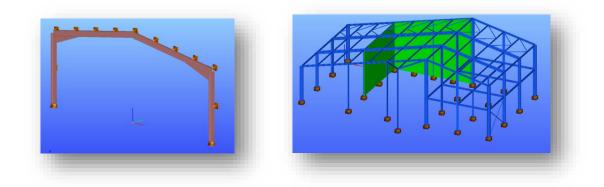


hidden line view : the cross section and plate thickness appear in a solid form without shading and sparkling effects; the supports are solid types.

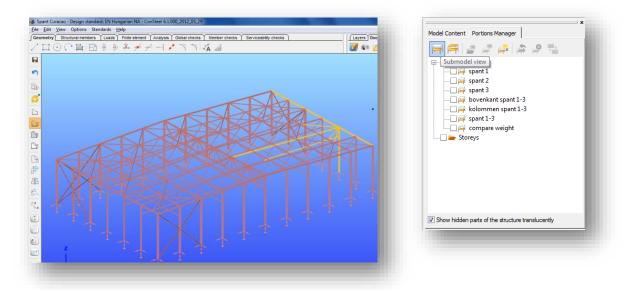


solid view : the cross section and plate thickness appear in a solid form with shading and sparkling effects; the supports are solid types.

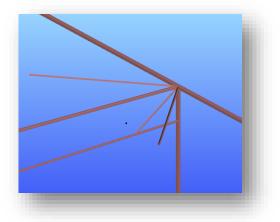




When the full model tends to be more complicated and parts of the model are in focus then the **SUBMODEL VIEW** should be applied.

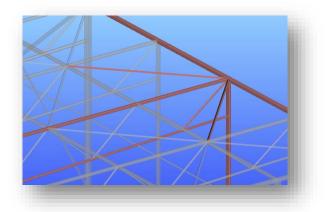


This option shows only the selected parts of the model,



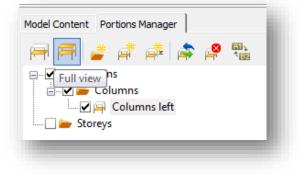
or checked the *Show hidden parts of the structure translucently* checkbox at the bottom of the **PORTIONS MANAGER**.





From **GEOMETRY** to **GLOBAL CHECKS** in every tab the selected submodel will be visible.

To make the whole model visible again click the *FULL VIEW* icon.



3.3 Selection

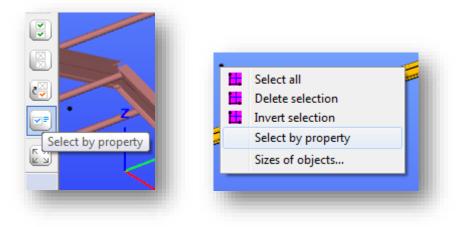
Objects can be selected in the graphical window, in the object tree, or by the additional selection options. As a general rule only the visible objects can be selected graphically, accordingly the same action can result in different selections on different tabs or model views. A selected object is changed to unselect by clicking on it while pressing the *SHIFT* button. The *ESC* button always unselects all objects.

The basic selection modes and their operation are the followings:

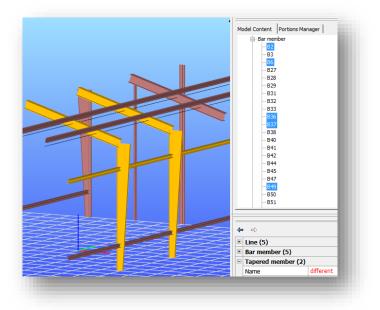
simple graphical selection: click on the object on the graphical window by the left mouse button. All objects will be selected which graphical symbol intersects the imaginary line set up perpendicular to the viewing (camera) plane at the clicking point. It means that the selection depends on the size of the graphical symbol of the objects and note that the covered objects will be selected as well!



- window selection: holding down the left button and moving the mouse will create a window. It is completed when the mouse button is released. The actual size of the window is continually seen on the screen. While drawing the window from left to right, only those members are selected that are within the window with their full size (inclusion window selection). When drawing the windows from right to left every member is selected no matter how small part is found in the rectangle (section and inclusion window selection).
- selection by property: this option can be reached on the left side bar or by right mouse button click on the graphical window. The selection can be made by choosing the desired object and further narrowing is possible by sorting the appropriate parameters out.



b object tree selection: select objects by clicking on the elements of the Object Tree.



By clicking the object names in the object tree while holding down the *CTRL* or *SHIFT* key results multiple objects selection. To unselect any of the selected object: hold down the *CTRL* key and click on the object name again in the object tree.

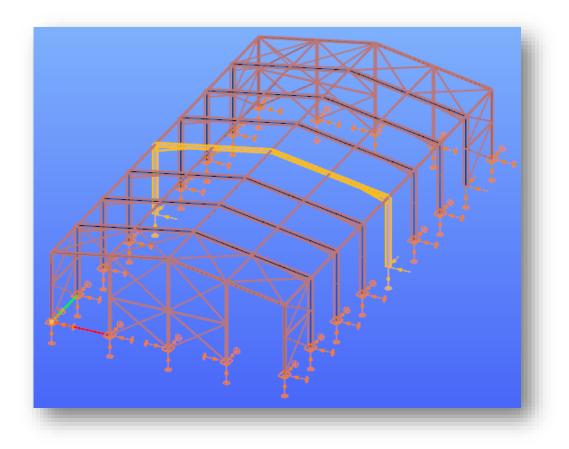


The additional generalized selection options (select all, unselect all, inverse selection) can be found on the left side bar or they can be reached by right mouse button click on the graphical window.

3.4 PORTIONS MANAGER

The **PORTIONS MANAGER** is a great tool to manage different but related parts of the model ie. floors, columns, bracings, beams, etc. This function can be achieved on the **PORTIONS MANAGER** tab.

New portions can be added by clicking the *New Portion* icon (#2). Before do so, select the parts of the model which will belong to that portion.



The portions can be arranged into folders. New folder can be created by clicking the **NEW FOLDER** icon (#1).



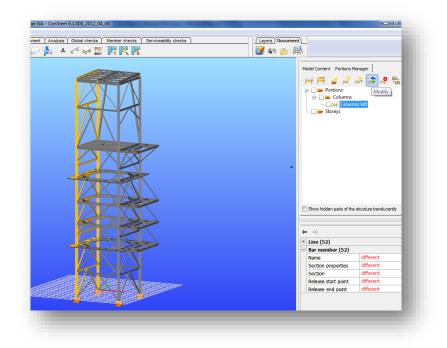
Model Content Portions Manager
🖃 🖳 🦢 Portions
Columns
🔲 🛱 Beams+bracings
Bracings
🗄 🗋 🖢 Storeys
🖸 🛱 Level (+12000)
🖂 🛱 Level (+9000)
🖂 🛱 Level (+6000)
🖸 🛱 Level (+4500)
🗌 🚘 Level (+3000)
G Show hidden parts of the structure translucently

To turn on a portion, check in the check-box in front of the name of the portion. More portions can be turned on at the same time.

If the *Show hidden parts of the structure translucently* check box (#6) is clicked at bottom of the *Portions Manager* tab, then the hidden parts of the model are also shown but those parts are translucent.

Any objects of the structural model can be part of several portions at the same time.

To modify a previously defined portion, first select the relevant parts of the model which will be the subject of the modification, then click on the name of the portion you are going to modify and finally click on the **MODIFY** icon (#3).





The **MODIFY PORTION** window will appear with the following options:

Modify portion	×
Add selected elements to portion	
\bigcirc Remove selected elements from portion	
\bigcirc Assign only selected elements to portion	
O Modify height	
Cancel OK	

- ► Add selected elements to portion: By clicking the *OK* button, the selected members will be added to the active portion. When in the selection some members already belong to the actually modified portion, then it will have no additional effect.
- Remove selected elements from portion: All those selected elements will no longer belong to the modified portion.
- Assign only the selected elements to portion: The modified portion will contain only the selected elements.

Renaming the Portion can be done by clicking the name of the portion and click on *RENAME* icon (#5).

Portion can be deleted by clicking the name of the portion and click the **D**ELETE button (#4).

3.5 OBJECT NAMES AND RENUMBERING

Names and labels are used for the better organization of the different objects in the model and for the better transparency of the documentation. The names and numbers of the objects are generated automatically while modeling. In some cases (for example creating Joints), the user has the opportunity to name the objects.

The following name and renumbering options are available:

- numbering of finite element points generated fully automatically, the user can't influence the process.
- names of structural element these are generated automatically while the user creates or copies structural objects (beam, column, plate, wall). Names consist a prefix and a



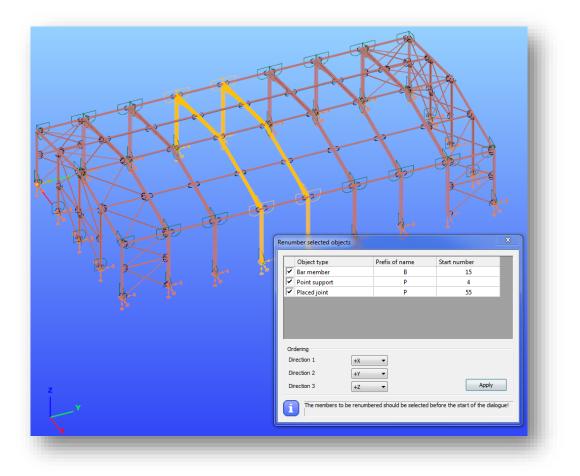
12.0 USER MANUAL

number (for instance: "B1" for a bar element). These names appear on the object tree in the right side table. The names can be modified by the **Renumber Selected OBJECTS** tool on the **STRUCTURAL MEMBERS** tab. Using this function a specific prefix and a start number can be defined, and a direction can be set in which the renumbering will be processed. On the finite element view these names denote the numbering of the finite elements of the members.

- names of structural supports, releases and links these names show the type of the placed support, release or link objects (for instance: "Fixed" or "yy,zz,w")
- names of joints these names show the user defined names of a placed joint (for instance: "Beam-to-column 01")

Renumbering the objects

To renumbering the objects first select the relevant objects, then click on the **RENUMBER SELECTED OBJECTS** icon on the **STRUCTURAL MEMBERS** tab.



The **RENUMBER SELECTED OBJECT** window will appear, with the automatically recognized object types contained by the set of selection. With the checkboxes can be selected the different types objects for renumbering.



The following functions can be used for renumbering:

- Prefix of the name can be defined
- Start number can be defined
- Priority of the renumbering can be set in the 3 main directions (X, Y Z)

The new names of the objects will built up from the given prefix and the new serial number, except the placed joint object type.

Label's visibility

The visibility of the labels can be set on the bottom status bar as it was discussed above (Chapter 1.2.6). The following label options are available:

- **material** the used material name of structural members (for instance: "S235")
- shape the name of the used cross-sections of structural bar members (for instance: "HEA 200")
- ▶ thickness the thickness of the used structural plates or walls
- load intensity
- units the units of load and thickness if set (for instance: "kN" or "mm")
- **initial curvature** the value of the initial curvature on structural bar members
- **coordinate system** the names of the local coordinate axes (X,Y,Z)

Additionally arbitrary text can be assigned to structural elements manually on the **DIMENSIONING** dialogue using the last tool.

3.6 DIMENSIONS

For the documentation of the calculated structural model it is very important to be aware of the exact dimensions of the model. The dimensioning tool can be found on the *GEOMETRY* tab.

Geometry	Structural members	Loads	Mass	Finite element	Analysis	Global checks	Member checks	Serviceability checks
/ 🏿 🤆			Å.	≁ ≁	1	<u> </u> 🖉		
8							Place dimension	



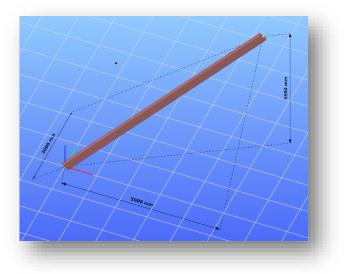
This provides a great number of different dimensioning options:

▶ Projected dimension of a linear object ^I The projected length of the selected linear object onto the global X or Y or Z axis will be dimensioned.

Dimensioning	×
Parameters	
Auto	
Auto According to global axis X According to global axis Y	
According to global axis 7	

Parameters:

- Auto: moving the mouse, the system detects the desired reference axis.
- According to global axis X: the X axis projection of the length of the linear object.
- According to global axis Y: the Y axis projection of the length of the linear object.
- According to global axis Z: the Z axis projection of the length of the linear object.



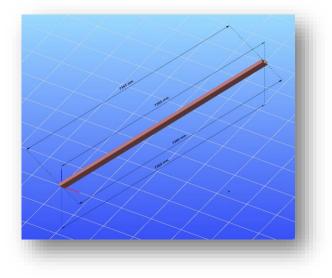
Lengthwise dimension of a linear object : The length of the selected linear object can be placed in one of the planes which is defined by centerline of the object and the global coordinate system X,Y plane, and a plane perpendicular to this plane and containing the centerline of the object.



12.0 USER MANUAL

Parameters:

• Auto: moving the mouse, the system is detects the desired plane.



Parameters:

- Auto: moving the mouse, the system is detects the desired reference axis.
- According to global axis X: the X axis projection of the distance between two selected points.
- According to global axis Y: the Y axis projection of the distance between two selected points.
- According to global axis Z: the Z axis projection of the distance between two selected points.
- Lengthwise dimension between two points
 The distance between the selected two points could be placed in one of the following planes.
 - Plane defined by centerline of the object and the X,Y plane global coordinate system
 - The plane perpendicular to the above plane and containing the centerline of the object.

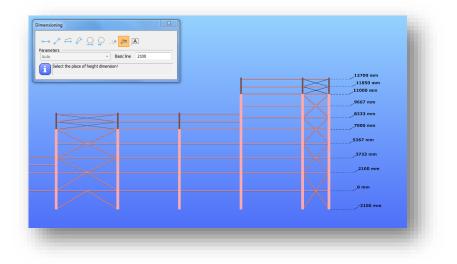
Parameters:

- Auto: moving the mouse, the system detects the desired reference axis.
- Diameter of a circle

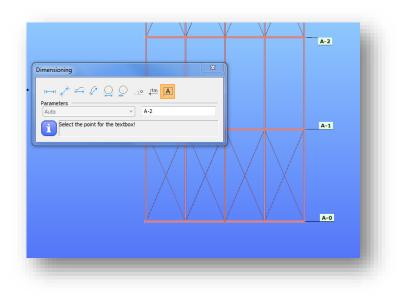


- Radius of a circle
- ► Angle of two lines After selecting two lines any of the four angles could be dimensioned.
- Height dimension ^{1m}: This function helps for quick dimensioning the height positions of selected points relative to global Z direction related to the given value in *Basic line* filed.

The content of the **Basic line** field means the height of the 0 level in the global Z direction. All the picked points will be measured from to this value as a 0.



Textbox to a given line A: The given text (A-2) in the script field can be placed into the model.





3.7 MEASURE

During modeling it is helpful to measure distances and angles. The **MEASURE** tool is found at the end of the **GEOMETRY** tab.

Geometry	Structural members	Loads	Finite element	Analysis	Global checks	Member checks
/ 🗆 🤇			* 🗲 🗡	/ 🍬	• ¬ ¬ •	À
						Measure

The following window offers the measuring functions:

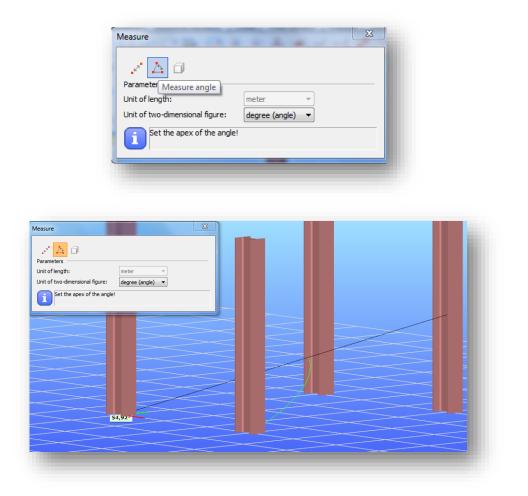
Parameters Unit of length: Meter Met	
-	-
_	•
Set the start point!	

Measure distance Shows on the screen the measured distance between two picked points in meter or in millimeter, selected from the "unit of length" list box.

	Masser M A G Parantes Ust disph meter
Measure	Unit of two-diversional figure: Bet the start point
Parameters Unit of length: Unit of two-dimensional figure: Set the start point!	



Measure angle : The function measures the angle between two lines defined by clicking three snap points in the 3D space in the plane of the two lines in degree or in radian.

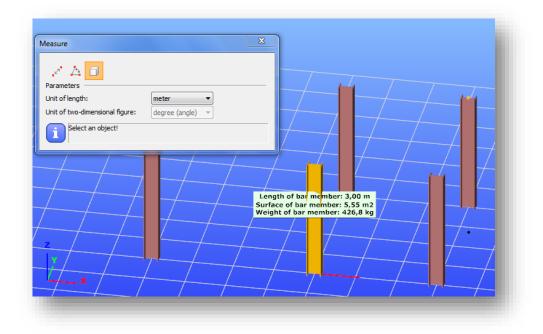


Data of structural member : The function shows on the screen the length, the surface and the weight of the selected structural member.

arameters Data of structural member	× A 0	
	Parameters Data of strue	
	Unit of two-dimensional figure:	degree (angle) 🔻







3.8 MODEL GRID

On the GEOMETRY TAB, it is possible to create custom MODEL GRID.

By clicking on the function, the **STRUCTURAL GRID** dialogue will be opened.

Grid 1 🔽 🔀	Structural grid properties Division in X direction:	Division in Y direction:	Set caption properties X caption	Y caption
	1500 2*2000	1500 2*2000	Initial value: 1 V Prefix: Overhang of the displayed lines:	1 V 0 mm V @ Ascending
	Color of grid lines: Grid position X 0 Display grid on the	mm Y ₀ 0	mm Z ₀ 0 mm oc	O Descending



Grid manager functions:

- Create a new structural grid with the button
- Accept grid properties and finalize grid with the Mutton
- Modify previously created grid with the ² button
- Delete previously created grid with the button

Model grid properties:

- Division of the grid in X and Y direction has to be set.
 - New row for the division can be created with the [□] button.
 - \circ Defined rows can be deleted with the \checkmark button.
 - The sequence can be modified with the arrows button. Values can be defined with single values (1500) or as multiplicated values (2*2000)
- Caption properties can be set both for X and Y directions.
 - It is possible to choose from capital letters, lowercase letters, decimal numbers, and roman numbers too.
 - Initial values can be set from the dropdown menu.
 - \circ $\;$ Prefix of the captions can be set manually.
 - Captions can be choosed if they desired to be in ascending or descending order both for X and Y directions
- Grid position can be set at the bottom of the dialogue.
 - Coordinates of the insertion point can be set manually according to the global coordinate system (X0,Y0,Z0), or with a graphical selection on the screen

using the arrow 통 button

- Rotation of the grid can be set with the definition of the alpha value, according to the global X axis
- It can also be choosed if the grid is desired to be shown on the lowest visible storey of the model (previously defined storeys are necessary)

<mark>S CON</mark>STEEĽ

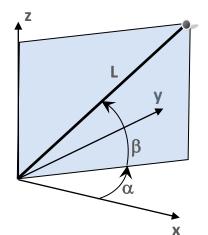
4 DRAWING GEOMETRY

4.1 BASICS

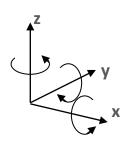
The first step of the modelling phase is the geometry drawing of the structure. In *ConSteel* all the geometrical objects can be easily created and modified in a 3D space. Among the basic CAD drawing, modifying and moving functionality advanced snap options make the modelling efficient. All the CAD functionality is placed on the *GEOMETRY* tab, the further view, select and snap options are on the side and status bar. Additionally all the relevant modelling functions are placed on the dialogues of structural members, supports and loads.

4.2 COORDINATE SYSTEMS

For the appropriate modelling and interpretation of the analysis and design results it is of high importance to be aware of the applied coordinate systems. This section summarizes in detail all the coordinate systems used in the *ConSteel*.



Two different type of coordinate systems can be used: the rectangular Descartes system (XYZ) and the polar system ($\alpha\beta$ L). In the polar system " α " denotes the angle between the axis "X" and the vertical plane defined by the axis "Z" and the point, " β " denotes the angle between the section line of that plane and the coordinate plane "XY" and the line from the origin to the point, and "L" denotes the distance from the origin. The polar coordinate system is less frequently used so the rest of this section describes the Descartes system.



As a general convention all the Descartes coordinate systems (axes directions and rotation signs) follow the most commonly applied right hand rule. This rule is valid for the definition of geometry and loads as well. The positive moments and rotations are defined as counter clockwise about the axis if it is viewed in front (towards the origin) in both the global and local system.



12.0 USER MANUAL

In the further text the following denotations are used:

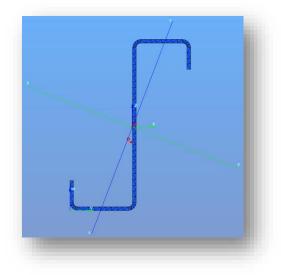
In case of global geometry (3D structural modelling):

- X, Y,Z: global coordinate system
- x, y, z: local coordinate system

In case of cross sections (2D modelling of sections):

- ▶ v, w: section edit coordinate system
- Y, Z: section edit coordinate system with its origin in the centre of gravity of the section
- y, z: section principal coordinate system

4.2.1 USER COORDINATE SYSTEM



The User coordinate system (UCS) is used for making the modelling easier and faster. The UCS is a specially positioned Descartes system (XYZ).

The tools for creating and modifying the coordinate systems can be reached by the **SET GRID AND COORDINATE SYSTEM** button on the side bar.

Grid and c	coordinate system X
Parameters Origin	x txx txz tyz ↔ txt Xt Yt Zt
Grid	X 0 mm Y 0 mm 2 0 mm X 1000 mm Y 1000 mm Size 20000 mm
Step	X 250 mm Y 250 mm t origin of coordinate system!

The User coordinates system can be switched on (or switch back to the global system) at the bottom status bar





The following creation options are available (from left to right):

- Switch to global system
- Set new origin for the UCS
- The "XY" plane of the global system will be the "XY" plane of the UCS
- The "XZ" plane of the global system will be the "XY" plane of the UCS
- The "YZ" plane of the global system will be the "XY" plane of the UCS
- Set the UCS by 3 points: the origin, the direction of "X" and "Y" axis
- Set the standing of the UCS identical to the local coordinate system of the selected plane with origin placed into the first node of this plane
- Set the direction of the axis "X" by two points X
- Set the direction of the axis "Y" by two points Y1
- Set the direction of the axis "Z" by two points Z1
- Reverse the direction of the selected axis

On the **STRUCTURAL MEMBERS TAB** and in the **SET LOCAL COORDINATE SYSTEMS** (



Icons for the modifications on the **SET LOCAL COORDINATE SYSTEMS** dialog are the following (from left to right):

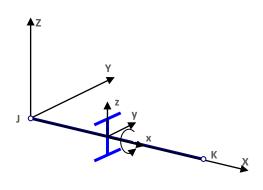
- Set "X" axis of UCS X[†]
- Set "Y" axis of UCS Y[†]
- Set "Z" axis of UCS Z[†]
- Set the "Z" axis perpendicular to the surface formed by the connecting beams

S CONSTEEL

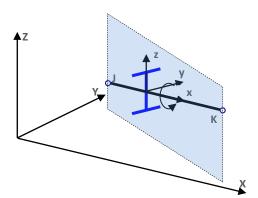
12.0 USER MANUAL

4.2.2 LOCAL COORDINATE SYSTEM OF BAR ELEMENTS

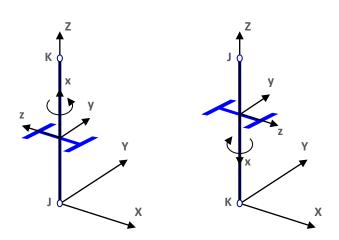
The reference line of bar members defines the axis "x" of the local coordinate system of the bars. The direction of the axis "x" is given by the start (J) and end (K) node of the reference line.



In the basic case the local coordinate system of a bar can be determined considering its reference line ("x") identical to the global axis "X". In this case the direction of local axes "y" and "z" are identical to the global axes "Y" and "Z". The position of the cross section on the bar is the following: the reference line goes through its centre of gravity, and the section "YZ" system coincides with the local "yz" system of the bar.



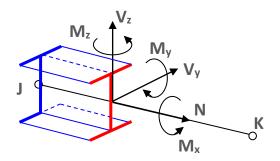
In case of a generally positioned beam member (not column with vertical reference line) the plane defined by the local "xz" system takes always vertical position and the axis "z" points towards the positive "Z" direction. For beams with (single) curvature the axis "x" is always the tangent.



In case of vertical reference line (column members) the local axis " γ " has the same direction as the global axis "Y", the direction of the local axis "z" depends on the placement of the column (position of the start and end points).



The interpretation and signs of the inner forces on a member is defined by its local coordinate system and the mentioned general sign convention as described in the following figure:



4.2.3 LOCAL COORDINATE SYSTEM OF SURFACE ELEMENTS

For surface elements the local plane "xy" is defined in the plane of the surface, the axis "z" is perpendicular to that plane and the direction of "+z" is in accordance with the right hand role of coordinate systems. The roles of the directions of the local axes of the plane surface elements:

By creating a rectangle shaped plane surface element:

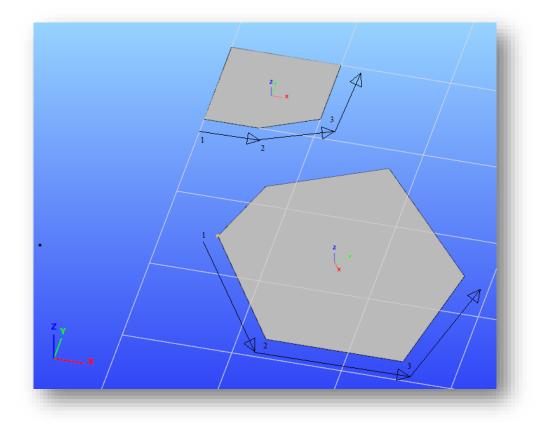
the axis "x" and "y" are parallel with the edges of the rectangle plane element. The "z" direction created in accordance with the right hand role of the coordinate systems.

By creating a plane surface element with polyline border:

- the axis "x" is parallel with the line defined by the first and the second picked points. It is the first edge of the plane surface element.
- the axis "y" points accordingly to the direction of the third picked point. It is the second edge of the created plane surface element.
- The "z" direction created in accordance with the right hand role of the coordinate systems.







In case of surface elements, the local coordinate system can be modified using the tools presented in Chapter 4.2.1.

4.3 Defining action points

Action points (snap points) can be set by moving the cursor on the sicon on the right side of the bottom status bar. You can find the description about the snap point settings above in **1.2.7** THE STATUS BAR chapter.

4.4 DRAWING

The functions for drawing described below are collected on the **GEOMETRY** tab.

Draw line

Simple line drawing. First select the start point. Then select the end point.

Polyline drawing. First select the start point. Then select the next points. Press **Esc** to interrupt line drawing.



Draw rectangle with lines



Draw rectangle. Select the corner point. Select the opposite corner.

Draw leaning rectangle in the space. Select the corner point. Select end point of the side of the rectangle. Select third point of the rectangle.

Draw Circle

Draw circle by radius. Select the centre-point of the circle. Select the radius of the circle. The circle will be created in the plane containing the centre-point and parallel to the actual UCS.



Draw circle by diameter. Select the start point. Select the end point. The circle can be created out of the actual UCS system.

Draw circle by 3 points. Select first point. Select second point. Select third point. The circle will be created in the plane defined by the three definition points.

Draw Arc

Arcs could be created out of the actual UCS.

Draw arc by centre-point, start point and angle. Select the centre of the arc. Select the start point of the arc. Select the angel of the arc or type it to the dialog window



Draw arc by 3 points. Select the start point. Select the end point. Select a point in the arc.

Draw arc by start point, end point and tangent. Select the start point. Select the end point. Select the other point of the tangent.

Draw two-dimensional plane elements



Draw two-dimensional plane element. Define the geometry of the plane element.

Γ	_		
I	L		

Draw hole. Select the plane element. Define the geometry of the hole (Rectangle, closed polyline or circle).

CONSTEEL

Change the direction of the local "x" axes of a plane element by defining two points.

Select the surface. Select the start point. Set the direction with the second point.

The construction modes of the plane elements:



The construction mode of a rectangle type plane element can be selected: rectangle or leaning rectangle.



The construction mode of a circular type of the plane element can be selected: draw circle by radius, by diameter, or circle by 3 points.

Draw polygon (a closed polyline). Set the first point. Set the next points. To close the polygon press right click on the mouse.

4.5 MODIFYING

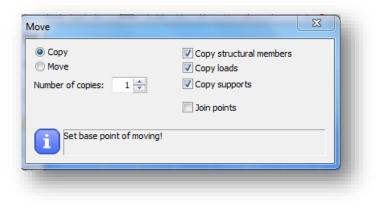
The four mostly used modifying functions are placed on the left sidebar:

Move point and edge. Select the point and (or) edge to be moved. Set the new position of the point (edge). This function is used for change the position of any of the end point of a linear or circular element, or to change the position of an edge or a corner point of a polygon, or of hole in a polygon.



Move selected objects. The selected objects can be moved or copied

Select the objects, and click the *Move selected objects* icon:





Select the transformation (Move or Copy). The copy function can be activated by the radio button near the Copy function. The number of copies can be selected.

Set start point of the vector for moving/copying. Set the end point of vector for moving/copying.

The selected members, loads and supports will be moved or copied to the position which is defined by the given vector accordingly to the selected options. The Move/Copy function is independent from the actual *UCS*.

Effect of the options:

Copy structural members:

Unchecked: Use when only the loads and/or the supports of the structural member need to be coped or moved.

Checked: Default. Structural members will be copied or moved.

Copy loads:

Unchecked: The loads will not be copied with the selected structural members.

Checked: The loads will be copied with the selected structural members.

Copy supports:

Unchecked: The supports of the selected structural members will not be copied.

Checked: The supports of the selected structural members will be copied. (In case of the selection set does not contain at least one selected member, than the selected supports will not be copied.)

Join point:

If the **Join points** box is checked then the end points of the copied objects will be connected with lines.

Mirror selected objects. The two given points represent mirroring axis will be projected into the actual *UCS* system and the mirroring will be performed in the actual *UCS* symmetrically to the axis of mirroring.

As first step select the UCS for mirroring (if it is needed), select objects to be mirrored and click on the **MIRROR SELECTED OBJECTS** icon.



Mirror	X
 © Сору © Моve 	 Copy structural members Copy loads Copy supports Join points
Set the first point of	the mirror axis!

Selecting the mirror line by picking two points for the mirror axis, the selected objects will be moved or copy to the mirrored position.

The effects of the options are the same as it was described above.



Rotate selected objects. Select the objects for rotation.

otate	(x
 Copy Move 	Copy structural members Copy loads
Number of copies:	Copy supports
α Apply	Join points
Set center of rotation!	

Select the Move or Copy option for the transformation. The number of copies can be selected. Set the centre of rotation. Set reference point of rotation. Set the direction point or type in the rotation angle and click on the **APPLY** button.

The effects of the options are the same as it was described above.

The further modifying functions are on the **GEOMETRY** tab:

Sizing of the selected objects. Select objects for sizing. Set the sizing centre, the sizing reference point and the sizing multiplier point (the distances of the sizing reference point and the sizing multiplier point to the sizing origin point determines the sizing factor). The sizing will be performed. The length (and depending on the sizing origin maybe the position) of the objects will be changed.

The length of the line-loads will be changed proportionally.

Curve selected line element. Select the edge for bend. Set the insert point.

CONSTEEL

Refract the selected line element. Select the edge to refract and the insert point along the axis of the selected object. Set the new position of the selected point. The original object will be broken into two parts.

The coordinates of the new position of the selected point also can be given manually accordingly to the selected coordinate system.

Divide selected objects. Lines and bars can be divided into equal or unequal parts.

By selecting the **To equal parts** option the number of the segments or the length of the segments can be set.

To equal parts	🔘 To unequal length parts
arameters	
Number of line segments	2
Length of line segment	1000
	Apply
1	

By selecting the two unequal parts option a series of relative or absolute lengths can be defined measured from A or B endpoint of the object.

🔘 To equal parts	To unequal length parts
arameters	
From point A	Relative distance values
From point B	Absolute distance values
Values: 150,2	00,50
	Apply

Break two selected objects. Select the two intersected objects to break. Both object will be broken by the intersection point. It works with linear and curved objects.



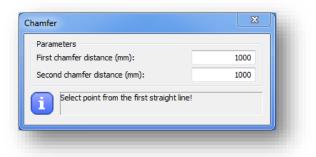
11

Trim selected element by cutting edge. Select cutting edge. Select the part to be trimmed. It works both with linear and curved objects.

Extend selected line elements to limit line. Selecting the limit line and click on t	he
inear object to extended to the limit line.	

Divide by trim. Cut out a part of the selected element. Select object for cut out. Select start point and the end point of the segment of the cut out.

Chamfer of two selected linear element. Select the first and the second linear element. The chamfer will be applied according to the order of selection.



Filleting of the edge of the section of two line element. Select the first and the second linear element. Chose a fillet radius.



4.6 LAYERS

The layers dialog window can be used for:



Name 🛆	Color	Line	Thickn	Transp		Fre	Obj
leams			• 1	0	\checkmark		
Iracing			▼ 3	0	~		
Columns			▼ 3	0	 Image: A start of the start of		
Crain-1			▼ 2	0	•		
Crain-2			▼ 2	0	-		
labs			▼ 3	0	•		

- Create new layers
- Copy existing layer-definition to a new layer
- Delete layers
- Edit properties of existing layers (colour, line style, thickness and transparency)
- ► Turn the visibility of layers on and off
- Freeze layers from activity whilst keeping them visible
- Turn on and off own style. If Objects of own style checkbox is turned OFF then all the objects on the layer use the selected colour, line style, thickness etc. If it is turned ON (default setting) then all the objects use the global style.

All CAD systems have a layer concept of some sort. Objects are placed on appropriate layers as a practical way of managing the objects within the model.

SCONSTEEL

5 STRUCTURAL MODELLING

5.1 BASICS

From the beginning of *ConSteel* great emphasis was taken on the user friendly structural modelling to build the structural model using real structural elements. Accordingly the *user model*, using whole structural members (beams with haunches, tapered columns, tension braces, slabs etc.) is completely separated from the *calculation model*, which is an automatically generated sophisticated finite element model used by the structural analysis and design. It follows that the engineer does not need to deal with the calculation model; he/she can concentrate solely on the building the real structural model.



IT SHOULD BE KEPT IN MIND THAT THE ENGINEER SHOULD BE AWARE OF THE FEATURES, POSSIBILITIES AND LIMITATIONS OF THE APPLIED ANALYSIS MODEL ALREADY AT THE MODELLING STAGE BECAUSE THE MISUNDERSTANDING OF THE MODELLING OPPORTUNITIES CAN LEAD TO UNEXPECTED ANALYSIS AND DESIGN RESULTS.

All the functionality connecting with the structural modelling are placed on the **STRUCTURAL MEMBERS** tab. Due to the high importance and wide range of modelling possibilities the functions of the cross-sections and joints are collected in separate modules **CSSECTION** (Chapter 10) and **CSJOINT** (Chapter 14).

5.2 LINE MEMBERS

The line members can be modelled with steel thin-walled, reinforced concrete, and composite cross-section. The members – depending on the placement – can be columns or beams. The column is a special positioned line member, which is always vertical, so the placement is simpler, because only the height should be defined.

The dialogue allows the following modelling options for creating members:

Draw the reference line of the member

draw as a line segment or continuous line

draw as an arc by centerpoint, startpoint and angle, 3 points or startpoint, endpoint and tangent





Select an existing line for the section as a reference line of a member

Read out the parameter of an existing member by clicking on it and assigning those parameters to the one to be created

Member parameters can be defined in the middle part of the dialog:

Beam Column Column Parameters

Section: before creating a beam or column member the cross-sections should be loaded, and the appropriate section can be selected from the combo including the loaded sections. If no section has been loaded in advance the SECTION dialogue can be called by pressing the three dots button (....) next to the combo. For more information about section administration see Chapter 10.2.

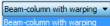
lame	△ Parameters	Material	Туре	Source	State	Load section
HEA 300	h=290; bf=300; tw=8.5; tf=	S 235 EN	Steel - standard	Standard	Not used	From library_
HEA 450	h=440; bf=300; tw=11.5; tf	S 235 EN	Steel - standard	Standard	Not used	Macro section
HEA 600	h=590; bf=300; tw=13; tf=2	S 235 EN	Steel - standard	Standard	Used	
HEA 800	h=790; bf=300; tw=15; tf=2	S 235 EN	Steel - standard	Standard	Not used	Modify
HEB 300	h=300; bf=300; tw=11; tf=1	S 235 EN	Steel - standard	Standard	Not used	Remove
RHS 300x100x6	h=300; b=100; t=6; r=12	S 235 EN	Steel - standard	Standard	Not used	Сору
Tapered	b1=100; tf1=10; h=200; tw=	S 235 EN	Steel - welded	In the m	Not used	
rc Rect + tab	bc=400; hc=700; l=150; t=1	C12/15	Concrete	In the m	Not used	
						Properties
						Into library
						Close

Release (start, end): predefined releases can be assigned to the ends of the members. If new release type is needed the *RELEASE* dialogue can be called by pressing the button next to the combo.



12.0 USER MANUAL

- Initial bow imp.: half-sine wave shaped initial crookedness can be defined in the two local direction perpendicular to the member reference axis ("y,z") with the given amplitude at the mid-length.
- Element type:
 <u>Section made from steel:</u>



Two choices are possible Tension bar Beam-column with warping and Tension bar, these types influence the finite element type used in the analysis. The beam-column with warping is a special element with 14 degree of freedom, including the warping of the thin-walled cross section. It is an important effect in case of structures with standard, open steel profiles.

The *tension bar* can only resist tensional axial force (no bending or torsional moments and shear) if it got compression the analysis neglects its effect.

Reinforce concrete or Composite sections:

There is only one choice, General beam-column the General beam-column type this is the commonly used 12 degree of freedom finite element.

Element group: the members can be sorted out into element groups for various purposes (selection, dominant results, sensitivity analysis etc.).
 Phase-1

Phase-2 if new element group is needed the *GROUPS OF STRUCTURAL MEMBERS* dialogue can be called by pressing the button next to the combo.



- Number of finite elements: ¹⁶/₁₆ the required number of finite elements used on the member in the analysis can be set. The default *automatic* option gives a sufficient result in the most cases.
- Local eccentricity: member eccentricity can be defined in the local coordinate system ("y, z") of the member, and the section can be rotated about the local coordinate "x".
- At the bottom of this dialog there is an instruction and command area for guiding the user.



It should be noted that not only the eccentric axial force produces influence in the analysis (additional bending moments) but the eccentric bending and torsional moments (additional bending and torsional moments and **bimoment)**!



5.2.1. STEEL MEMBERS

As the first step of line member modelling, cross sections has to be loaded into the model. This can be performed either from the **SECTION ADMINISTRATOR** directly, either from the **BEAM** dialogue. In ConSteel, for steel cross sections both standard cross section libraries and macro sections can be used.

In **CONSTEEL 12**, the following standard cross section libraries are available:

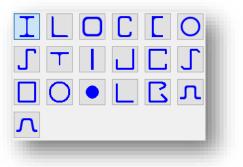
- Standard European cross section library
- American section library
- British section library
- Chinese section library
- Continental Steel section library
- Russian section library

In **CONSTEEL 12**, the following macro sections are available:

Welded from plates type

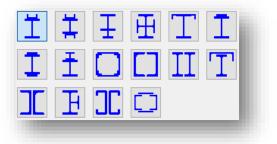
- \circ Welded I or H section
- Welded box section
- Welded maltese cross section
- o Welded half maltese cross section
- o Welded hat section
- Welded T
- \circ WQ section
- Slant flange welded I or H section
- Cold formed or hot rolled type





- Rolled I or H shape (with parallel flange)
- Rolled angle (parallel leg)
- RHS (cold formed shape)
- C shape (cold formed)
- Rolled chanel (parallel flange)
- CHS shape (hot rolled)
- Z shape (cold formed)
- T shape (half I)
- o Plate
- \circ $\,$ Cold formed unsymmetric U section $\,$
- Cold formed C section
- Cold formed Z section
- \circ $\,$ Cold formed or hot rolled rectangular hollow section
- o Cold formed or hot rolled circular hollow section
- o Round bar
- o Cold formed L profile
- o C section (cold formed, edge folded back) section
- \circ Cold formed omega section
- Cold formed slope omega section

Compound type



- o I or H shape reinforced by chanel at flange
- I or H shape reinforced by chanels at flanges
- I or H shape reinforced by half I or H



- I or H shape reinforced by half I or H shapes at web
- I or H shape reinforced by angels at flange
- I or H shape reinforced by plates at flanges
- o I or H shape reinforced by I or H shape at flange
- o Closed shape with angles and plates
- o Closed 2U
- Closed 2I
- o I plus U
- o I from 2U
- Half maltese
- o 2backed C
- Battened section

It is important to note, that in ConSteel, the material grade is an attribute of the cross section, therefore it has to be defined when choosing the type of the cross section.

On the beam dialogue, when defining the parameters of the line member, two options can be choosen for the element type. The *beam-column with warping* is a special element with 14 degree of freedom, including the warping of the thin-walled cross section. It is an important effect in case of structures with standard, open steel profiles.

The *tension bar* can only resist tensional axial force (no bending or torsional moments and shear) if it got compression the analysis neglects its effect.

5.2.2. REINFORCED CONCRETE MEMBERS

Definition of reinforced macro sections can be performed from the *MACRO DIALOGUE*. There is two main types of reinforced concrete sections. *Sections with Design Option* (#1) and *SECTIONS WITHOUT DESIGN OPTION* (#2).

|--|

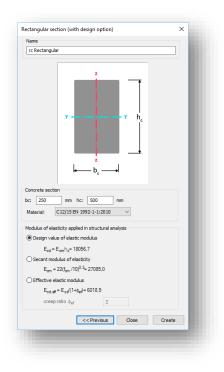


5.2.2.1 REINFORCED CONCRETE CROSS SECTIONS WITH DESIGN OPTION

REINFORCED CONCRETE CROSS SECTIONS WITH DESIGN OPTION, are represented with a gray cross section pictogram on the *MACRO* **SECTION** dialogue. In ConSteel 12, there are two section types with design option:

- Solid circle section
- Rectangular section

After choosing one of the available cross sections, on the section dialogue, only the overall dimensions, the concrete material grade and the modulus of elasticity applied in the structural analysis has to be set.



Because the reinforcement distribution my varie along the span of the concrete member, definition of the reinforcement settings has to be made through a dedicated **REBAR EDITOR** (III) function, which can be found on the **STRUCTURAL MEMBERS TAB**.

5.2.2.1.1 BEAM REINFORCEMENT DEFINITION

Creation of beam reinforcement objects can be created, if the Beam rebars option is selected on the left panel of the **REBAR EDITOR** dialogue. The **REBAR EDITOR** (III) dialogue can be found on the **STRUCTURAL MEMBERS TAB**



B 500 A 🗸 Ribbed	rc Rectangular 🗸
Ct	Section
Top cover - c _{t, nom} 3 cm	600 x 400
Bottom cover - c _{b, nam} : 3 cm	Material C25/30 EN 1992-1-1:2010
53 cover - c _{s, ran} , 3 cm c _b	Max aggregate size
Diameter of stirrup:	d _g 16 mm
Strutinclination $\phi[^{o}]$ 45	
Consider torsional rebars	
Area 0 mm ²	
Longitudinal reinforcement V Load combination-1 V	All beams V Diameter øfmml V
₹ 7,0	∩m ^
	T III
4 [kNm] MEd	
MRd 71,9 100.0	71,9
• -0,0 kNm • 97,3 kNm	
+ 6 \$\$ - 3,8cm	
2 ¢12 - 3,8cm	

- #2 If there are reinforced concrete cross sections with design option are loaded into the model, new beam reinforcement objects can be created by pressing the *ADD* button. Switching between different reinforcement objects works by selecting one from the dropdown menu. If a reinforcement object is copied, the copied object will inherit the parameters of the original object. The *DELETE* button will delete the actively selected reinforcement object only.
- #3 On the middle part of the dialogue, the cross section can be selected, for which the reinforcement settings are desired to be set on the other parts of the dialogue. The overall dimensions, and the material grade will be written below the dropdown menu of the selected cross section.
- #4 The bottom part of the dialogue, is a multifunctional panel. Here, the longitudinal and transverse directional rebars can be defined, envelope diagrams for bending and shear is defined, predesign values like (*Mrd and Vrd*) is shown and detailing rules are checked. Main functions of the panel in details:
 - Graphical display: After a reinforcement object is placed on a concrete member using the *PLACE* button, the length of the member on which the object is applied is highlighted. If there are available analysis results for the member, the bending moment and shear diagrams are shown on the panel, both for individual load combinations with the dropdown menus, and as envelope



diagrams. Switching between the bending and shear diagrams can be performed with the upper left dropdown menu. Med and Ved values are also shown

Mrd, Vrd Vrdc and Vrdmax values are also represented on the panel, if there are applied longitudinal and shear reinforcements (more on that below).

Unfilled detailing rules are also highlighted as warnings at the bottom of the panel.

Adding longitudinal rebars: If the LONGITUDINAL REINFORCEMENT option is selected in the upper left dropdown menu, using the + button it is possible to add rebars to the member. On the ADD BEAM REBAR dialogue, position, quantity and diameter settings has to be given. After pressing the ADD button, the graphical representation of the rebars will appear on the graphical display Deletion of a rebar can be performed by pressing the 10 button and choosing the rebar with the Sign.

bottom rebar 🗸 🗸	Rebar distance from the 3,8 cm
	section edge
Reinforcement	2 pcs \$16 \$
Start position	From end A 🗸 0 mm
End position	From end B V 0 mm
	Add Cancel

Adding stirrups: If the TRANSVERSE REINFORCEMENT option is selected in the upper left dropdown menu, using the ⁺ button it is possible to add stirrups to the member. On the ADD BEAM REBAR dialogue, only the spacing and the start-end position has to be defined. Diameter of the stirrups has to be given on the main rebar editor dialogue. After pressing the ADD button, the graphical representation of the rebars will appear on the graphical display Deletion of a rebar can be performed by pressing the ¹/₁ button and choosing the rebar with the ¹/₁ sign.

Stirrup spacing	100 mm
Start position	From end A 🗸 0 mm
End position	From end B 🗸 0 mm
	Add Cancel

CONSTEEL

A reinforcement object can be placed on more members, if the members has the same parameters (length and cross section) using the **PLACE** button.

If a reinforcement object is placed on more members with different lengths, a copy of the original reinforcement object will automatically be created for each member with a different length.

5.2.2.1.2 COLUMN REINFORCEMENT DEFINITION

By choosing the *COLUMN REBAR* option (#1) on the left side panel of the *REBAR EDITOR* dialogue, column reinforcement related data will be available on the dialogue.

Column rebars	Cumn reinforcement V Add	Copy Delete	
	y side ↓16 ven y distance ↓2 side ↓12 ven	itudinal side reinforcement 21 pcs start distance 40 mm	Section recectangular Section 500 x 250 Material C12/15 EN 1992-1-1-2010 Pasign parameters Design parameters Total

- #2 If there are loaded cross sections in the model with design option, new reinforcement object can be created using the *ADD* button. A reinforcement object is created by default, but any number of objects can be created. Switching between them is possible using the dropdown menu, *DELETION* and *COPY* can be performed using the dedicated buttons.
- #3 In the dropdown menu the available reinforced concrete cross sections will be listed, which has design option. Overall dimensions and material grade is listed below the choosen cross section, while the graphical representation of the section is shown at the middle panel (#5).



- #4 At the left side of the dialogue, parameters of the main reinforcement (concrete cover, diameter, material grade), and paremeters of the stirrups can be adjusted (diameter, spacing)
- #5 At the middle part of the dialogue, the cross section which is selected at the right panel (#3) is shown, with the dynamic representation of the reinforcements. On this panel, additional reinforcements can be added both for y and z directions, by checking the checkboxes at the bottom part, and defining the parameters for spacings, and diameters and quantity.
- #6 In order to be able to run the design process (discussed in details at CHAPTER 9.4 REINFORCE CONCRETE DESIGN) for the column, additional design parameters has to be set for the column. By clicking the (...) button, the design parameters will be opened, where all of the necessary parameters are listed, and can be set individually. Design parameters are separate objects, what means that more sets of parameters can be created, saved and used for the reinforcement object. The required parameters are buckling, curvature calculation and shear check parameters:

ckling parameters					
uckling length coefficient around y xis $(β_y)$	1		Buckling length coefficient around z axis (β _z)	1	
olumn length around y axis			Column length around z axis		
Length of member			Length of member		
OCustom length	1000	mm	OCustom length	1000	mm
arameter of curvature calculation (Eq 5.	34)				
actor K, calculated acc. to standard					
Auto					
OUser defined	1				
actor Køy calculated acc. to standard			Factor Ktp calculated acc. to standard	i	
Auto			Auto		
OUser defined	1		OUser defined	1	
ar check parameters					
Strutinclination	45				
			_		

5.2.2.2. REINFORCED CONCRETE CROSS SECTIONS WITHOUT DESIGN OPTION

Members in the model, which are defined with **REINFORCED CONCRETE CROSS SECTIONS WITHOUT DESIGN OPTION** are not allowed to use in the standard checks at the global checks tab. Members with these types of cross sections can provide analysis results (see **CHAPTER 8.6**), and the resistance surface (see **CHAPTER 10.4.5.2**), which of course takes into account the effect of the applied reinforcements of the cross section on the side of stiffness. Available cross sections are:

- Solid circle section
- Ring shaped section





- Double symmetric I section
- Singly symmetric I section
- I section with tapered flange width
- Rectangular section
- Rectangular section with hole
- Rectangular section with tab
- T shaped section
- ▶ T shaped section with tapered flange width
- Trapezoid section
- Converse T shaped section

When defining the sections, overall dimensions, concrete material grade, reinforcement parameters and the applied stiffness during the analysis has to be set.

ame		
rc I 1x symm.		•bt•
concrete section t: 400 mm b: 300 mm t: 600 mm t: 600 mm t: 600 mm t: 600 mm t: 500 A Ipper bars - number of rows: Mem Diameter of re 1. row 16	150 mm Material: 150 mm C12/158H 15 150 mm ✓ Cover - aby: 2 Cover - aby: 1 1 Number of r Cover - a zt t 4 20	
Iumber of lower bar rows: Mem Diameter of re 1. row 16	1 V Number of r Cover - az b 4 20	(m
Nodulus of elasticity applied in		E.m = 22(f.m /10) ^{0.3} = 27085.0
Design value of elastic mo	·	$E_{cd} = E_{cm}/\gamma_c = 18056.7$
Effective elastic modulus		$E_{cd,eff} = E_{cd}/(1+q_{ed}) = 6018.9$
creep ratio ϕ_{ef}	2	aataa aa taa
	on accorinding to EN 1992-1-1 5	. 21 formula

5.2.3. COMPOSITE COLUMNS

The first step is to define a composite column cross-section. Five types of cross-section are available: fully encased I section in concrete, partially encased I section in concrete, encased maltese cross-section in concrete, fully encased I section in hollow section and fully encased I section in circular hollow section. The cross-section macros can be found among the *MACRO SECTIONS*.



Steel Welded from plates Cold formed or hot rolled Compound Reinforced concrete Composite Composite Beam Budding Restrained Brace	
	Fully encased I section in concrete
<< Previous Close	Next >>

During the creation of the cross-section the parameters of the concrete section, reinforcement and the encased steel section have to be set.

Name				
comp Part. en	cased I			Upperibars
Reinforcement			Steel section	az
Diameter:	16	• mm	Name:	
Material:	B 500 A	•	HEA 300 -	Leftbars
Jpper bars:	4	pcs	Concrete section	<u> </u>
ower bars:	4	pcs	c: 20 mm	
eft bars:	0	pcs	Material:	Rightbars
Right bars:	0	pcs	C12/15 EN 1992-1-1:201 -	
Cover - ay:	20	mm		Lower bars 2
Cover - az:	20	mm		ay a ay
Secant mode Design value	ticity applied in ulus of elasticit e of elastic mod	y Iulus		$E_{cm} = 22(f_{cm}/10)^{0.3} = 27085,0$ $E_{cd} = E_{cm/c} = 18056,7$ ideration the shrinkage, creep and cracking
Apply L/200	initial sway in initial sway in initial sway in 2	r directio	'n	neraun un stirtikaye, u eep anu u adong
			ſ	<< Previous Close Create

For the structural analysis two types of elastic modulus can be used which can be selected on the middle part of the cross-section creation dialog.

Standard initial sways also can be defined in y and z directions for the cross-section. If the initial sways are applied than the cross-section check is enough according to the EuroCode. Stability check is not necessary.



5.2.4 COMPOSITE BEAMS

The first step is to define a composite beam cross-section. There are two types available: composite beam with solid concrete slab and composite beam with profiled steel sheeting. The cross-section macros can be found among the *MACRO SECTIONS*.

Steel Welded from plates Cold formed or hot rolled Compound Composite Composite Control Comm Beam Buckling Restrained Brace		
	Composite beam with constant slab thickness	
<< Previous Close		Next >>

When creating a new composite cross-section the effective breadth must be defined. It is important to know that during the analysis the actual effective breadth is automatically calculated based on the design parameters. The effective breadth which must be set at the cross-section creation is used in the graphics and for determining the self-weight of the beam. It is also possible to create an edge composite beam with different width in left and right side.

Rebars can be easily defined either with pitch or with area (mm^2/m) .

lame			a la b	d
Composite beam			= - b _{e1} →+ b _{eff}	
Concrete flange ◎ b ₄₁ 500 mm b ₄₂ 0 mm b ₄₂ 0 mm b ₄ 300 mm b ₄ 4 300 mm b ₄ 300 mm	Steel section Name: HEA 300 Headed shear studs newd 1 v D 20 h ₂₀ h ₂₀ f _v 450	mm mm N/mm		• h _c
	V Upper rebars V Bottom rebars at 150 mm at 335,103 mm ² /m ete surface	Transversal r Diameter: Pitch amor Area of rel The rebar	8 mm ng bars at 150	mm 3 mm²/m
Modulus of elasticity applied in struct Secant modulus of elasticity Design value of elastic modulus	ural analysis	_	$E_{cm} = 22(f_{cm}/10)^{0.3} = 29962,0$ $E_{cd} = E_{cm/c} = 19974,7$: Previous Close	Create



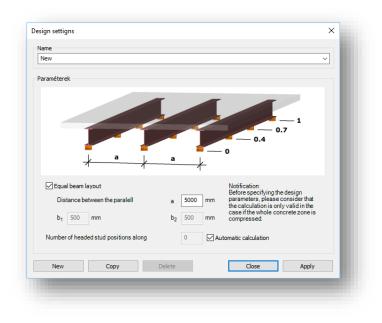
After the composite cross-section is defined it can be used for member creation the same way as a steel cross-section.

Before running the analysis select the composite beam(s) and set the **DESIGN PARAMETERS** on the **OBJECT PROPERTIES**.

+	Line (1)		8
-	Bar member (1)		8
	Name	B133	
	Section properties	Composite beam 2	
	Section	Composite beam 2	
	Release start point	yy,zz	
	Release end point	yy,zz	
	Eccentricity - y [mm]	0	
	Eccentricity - z [mm]	0	
	Rotate [deg]	0	
	Number of finite eleme	n Automatic	
	Bow imperf. L/v	0	
	Bow imperf. L/w	0	
	Element type	General beam-column	
	Element group		
	Design parameters	Internal 2	
	Layer	Layer_1	

In the **DESIGN SETTINGS** window you can set the distance between parallel beams, the number of shear studs, the type of analysis, moment redistribution and the support point positions along the member. The support position table can be automatically filled up by clicking on the button above the table.

You can create more design parameters and assign them to different members. You can also use more design parameters to easily switch between them to find the best parameters or to simply compare different settings.





5.2.5 HAUNCHED MEMBERS

The created members can be strengthened, if necessary, by using the **HAUNCH** function (

The haunch can only be used for members with I type (IPE, HEA, welded I) sections, and the shape of the haunch is considered as half of a welded or hot-rolled I section (with one flange) with decreasing web height.

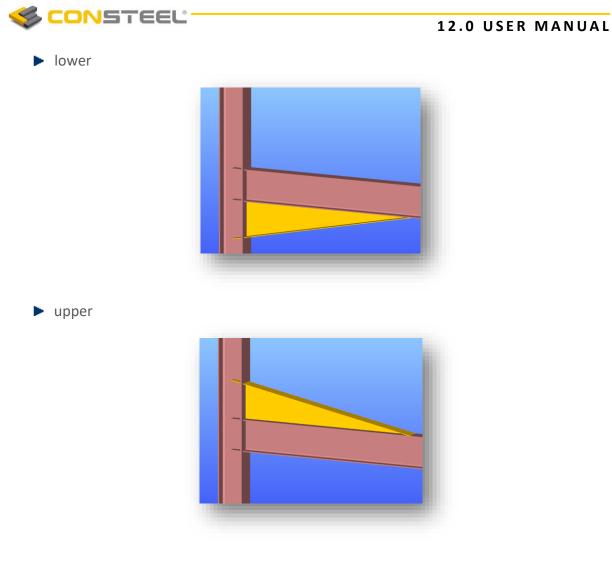
In case of welded haunch, on the dialogue panel the length (L), start height (hs), end height (he) web thickness (tw), flange width (b), flange thickness (tf) and the characteristics of the haunch should be set.

In case of hot rolled haunch, on the dialogue panel the Length (L), start height (hs), end height (he) ,the section (roll radius) and the characteristics of the haunch should be set. With a click on the (....) button, sections can be loaded from the section administrator.

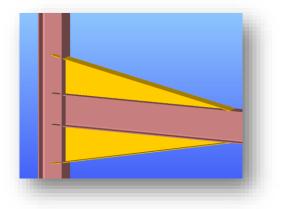
With the black arrow button ()) the geometrical parameters of the haunches can be loaded from the selected beam.

J i k Parameters	
Hot rolled O Welded	
L (mm) hs (mm) he (mm) Section 1000 300 100	▼
Position of the haunch	-
Select start point of the haunch on the beam!	

The position of the haunch is set by simply clicking on the member the start point (where the height of the haunch is full) and the direction point of the haunch on the member. It should be noted that if the start point is a common end point of more members (this is the usual case, for instance at a beam-to-column connection point) then this point should be approached and clicked on the member to be haunched. The characteristics of the haunch denote the appropriate side of the haunch on the member according to the direction of its local "z" axis. In the case of usual beam position it results the followings:



► symmetric

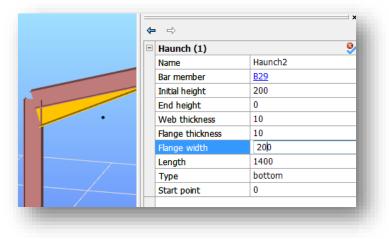




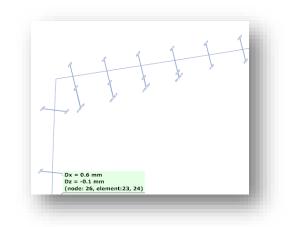
The haunches can be selected and modified in the parameter table as separate objects.



12.0 USER MANUAL



IMPORTANT TO KNOW THAT FOR THE HAUNCHED PART OF THE MEMBER NEW SECTIONS ARE CREATED DURING THE AUTOMATIC FINITE ELEMENT GENERATION WHICH CONSIST OF THE ORIGINAL SECTION AND THE HAUNCH WITH APPROPRIATE WEB HEIGHT. THESE NEW SECTIONS ARE PLACED ECCENTRICALLY ON THE REFERENCE LINE OF THE MEMBER (EXCEPT THE SYMMETRICAL HAUNCH TYPE).



THIS ECCENTRICITY CAUSES ADDITIONAL EFFECTS IN THE ANALYSIS RESULTS DUE TO THE ECCENTRIC POSITION OF THE SECTIONAL FORCES (FOR INSTANCE AT THE BEAM-TO-COLUMN CONNECTION POINT OF A FRAME WITH HAUNCHED BEAMS AND/OR COLUMNS THE EQUILIBRIUM OF THE IN-PLANE BENDING MOMENTS EXISTS ONLY IF THE ADDITIONAL MOMENTS FROM THE ECCENTRIC AXIAL FORCES ARE TAKEN INTO ACCOUNT)

5.2.6 TAPERED MEMBERS

Tapered members are frequently used in the economic design of steel framed structures, so the fast and simple modelling of tapered members is of high importance. For the definition of a tapered member first a line member with welded I or H, box, or cold formed C section should be created in the model.



Hot rolled and other shape of Macro section can't be tapered.

With the **TAPERED MEMBER** function () the section height of these sections can be set to linearly varying along the member length.

Tapered member	×
k i Parameters	
H1 (mm)	H2 (mm)
250	600
Set beam eccentricity	
Place the centroid of the	smaller section to the axis of the beam
Form	
z - Zy	$z = \begin{bmatrix} z \\ z \end{bmatrix} x = z \\ z \end{bmatrix} x = z \begin{bmatrix} z \\ z \end{bmatrix} x = z \\ z \end{bmatrix} x = z \begin{bmatrix} z \\ z \end{bmatrix} x = z \\ z \end{bmatrix} x = z \begin{bmatrix} z \\ z \end{bmatrix} x = z \\ z $
Select member fo	or the tapered member!

First the user has to specify the parameters and beam eccentricity options for the member to be tapered on the **TAPERED MEMBER** dialogue.

Specify the start (H1) and end section height (H2) independently of the original section height of the cross-section. The start value for section height applied at the start point of the member, the end value applied at the other end. To change the H1 and H2 values click the

icon. The values will be changed, and selecting again the member for tapering, the sizes of the tapering will change adequately.

The rules for the beam eccentricity of the tapered member relates to the axis of the originally positioned member to be tapered.

Set beam eccentricity	
Place the centroid of the bigger section to the ax	is of the beam 🔹 🔻
Place the centroid of the smaller section to the ax	is of the beam
Place the centroid of the bigger section to the ax	is of the beam
Place the centroid of the original section to the as	kis of the beam

- 1. *Place the centroid of the smaller section to the axis of the beam:* the centre of smaller H value of the tapered member will be positioned to the axis of the original beam
- 2. *Place the centroid of the bigger section to the axis of the beam:* the centre of bigger H value of the tapered member will be positioned to the axis of the original beam

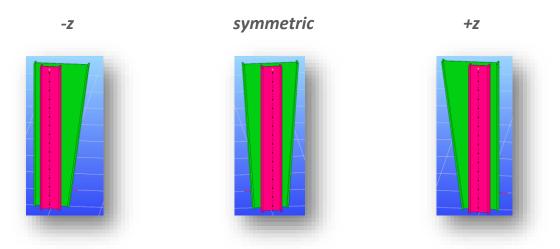


3. *Place the centroid of the original section to the axis of the beam:* the edge of the tapered member is coincident with the original member end and the tapering will start from this position.

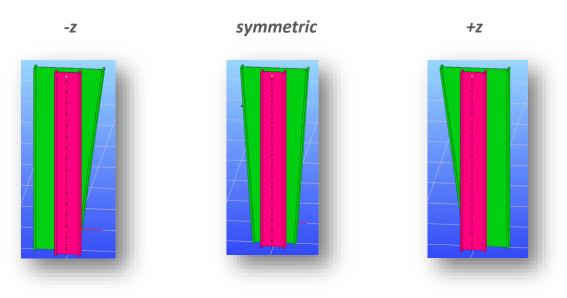
The relative position of the tapering can be -z (the left side of the tapered member will be parallel to the axis of the originally placed member), symmetric or +z (the right side of the tapered member will be parallel to the axis of the originally placed member). These definitions regulate the directions of the offset of the given height values along the local "z" axis of the tapered beam.

Below shown the effect of the different relative positioning by the chosen eccentricity:

 Place the centroid of the smaller section to the axis of the beam: the centre of smaller H value of the tapered member will be positioned to the axis of the original beam

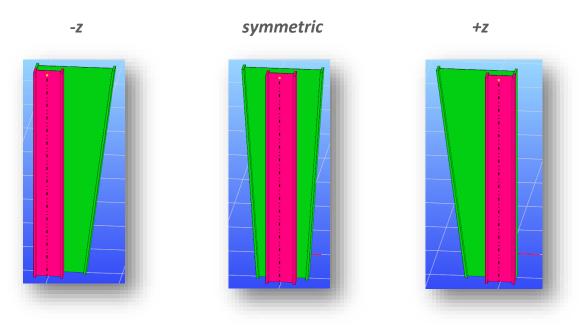


2. *Place the centroid of the bigger section to the axis of the beam:* the centre of bigger H value of the tapered member will be positioned to the axis of the original beam





3. *Place the centroid of the original section to the axis of the beam:* The edge of the tapered member is coincident with the original member end the tapering starts from this position



Activating the **1** icon, by clicking an earlier created tapered member in the model, the values, the eccentricity and the relative positioning will be read out of that clicked, and appear in the dialog box. By clicking another member for taper, these parameters will be applied.



IN CASE OF SELECTING A TAPERED MEMBER NOT ONLY THE MEMBER BUT THE TAPERING WILL BE SELECTED AUTOMATICALLY AND CAN BE MODIFIED IN THE PARAMETER TABLE AS SEPARATE OBJECT.

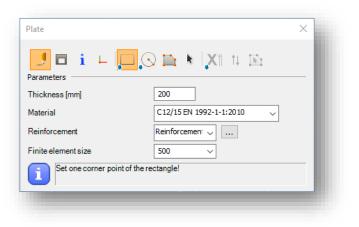
IMPORTANT TO KNOW THAT FOR THE TAPERED MEMBERS NEW SECTIONS ARE CREATED DURING THE AUTOMATIC FINITE ELEMENT GENERATION WITH APPROPRIATE SECTION HEIGHTS. THESE NEW SECTIONS ARE PLACED ECCENTRICALLY ON THE REFERENCE LINE OF THE MEMBER (EXCEPT THE SYMMETRICAL TAPERING). THIS ECCENTRICITY CAUSES ADDITIONAL EFFECTS IN THE ANALYSIS RESULTS DUE TO THE ECCENTRIC POSITION OF THE SECTIONAL FORCES (FOR INSTANCE AT THE BEAM-TO-COLUMN CONNECTION POINT OF A FRAME WITH TAPERED BEAMS AND/OR COLUMNS THE EQUILIBRIUM OF THE IN-PLANE BENDING MOMENTS EXISTS ONLY IF THE ADDITIONAL MOMENTS FROM THE ECCENTRIC AXIAL FORCES ARE TAKEN INTO ACCOUNT)



5.3 SURFACE MEMBERS

Plane surface members with uniform thickness can be modelled. Two options are available for

surface member modelling: the arbitrary shaped and positioned *plate member* (); and the vertical and square *wall member* (). The latter one is a modelling aid for the frequently used definition of walls since only the reference base line and the two height values should be defined.



Vall					x
🤳 i 🖊 k					
Parameters					
Height (start, end)	3000	mm	3000	mm	
Thickness (mm)	200	mm			
Material	C12/15 E	N 1992-	1-1:2010	•	
Finite element size	500	•]		
Set the start point!					

In the **PLATE** and **WALL** definition panels the drawing functions are collected in order to create the plate with the desired shape. Arbitrarily shaped holes can be defined in existing plate members. Three parameters can be set: the thickness, the material and the finite element size of the surface member. The finite element size is an approximate target maximum size for the finite element mesh used for the calculation of surface members.



5.3.1 PLATE REINFORCEMENT

In case of concrete *PLATES*, reinforcements can be added to the model at the reinforcement dropdown menu, or by clicking on the (...) button. Diameters, covering and material can be set on the *REINFORCEMENT* dialogue:

Name:	Reinforcement	1		\sim
Reinforcement				
Diameter of top rebars in	Xdirection	8	∼ mm	
Diameter of bottom rebai	rs in X direction	8	∼ mm	
Diameter of top rebars in	Ydirection	8	→ mm	
Diameter of bottom rebai	rs in Y direction	8	→ mm	
laterial:		B 500 A		\sim
laterial: Main rebar direction:		B 500 A		~
	Top cover		10	∽ mm
_	Top cover Bottom co	ring:		✓ mm

5.4 DIAPHRAGM

In-plane rigid element can be defined on the model in arbitrary planar. The added members' distance from each other will be same in the plane of the diaphragm element. With diaphragm element the in-plane rigid structural elements for example slab, trapezoid sheet can be modelled.

After clicking on the *DIAPHRAGM* (I) icon on the *STRUCTURAL MEMBERS* tab a dialog window appears.



liaphragm			x
Ν	lame	DIA 2	
Draw surface Select me	mbers		
) 🗖 🖉 🗖 🗖		<u>}</u> k [X1	
Set first point of t	the two-dimension	al figure!	
Close		Apply	ОК

Just like in the two-dimensional figure drawing there are different possibilities to draw the surface: draw a rectangle (), draw leaning rectangle (), draw circle (), draw polygon (). With the select function () previously created *LOAD TRANSFER SURFACE* can be select to use as a diaphragm also.

After drawing the surface it is possible to select the members which are added to the diaphragm or all of the planer members can add.

DIAPHRAGM element can be used also as a **LOAD TRANSFER SURFACE** and surface loads can be placed on it.

5.5 RIGID BODY

Arbitrary rigid body line can be defined on the model. Separate structural members can be linked with the rigid body.

Before clicking on the *RIGID BODY* () icon on the *STRUCTURAL MEMBERS* tab, the lines, members and surface edges have to be selected which should be converted to rigid body. By clicking on the *APPLY* button, the rigid body is created.

5.5.1 MODIFYING RIGID BODY

RIGID BODY can be modified by selecting it and changing the lines, members and surface edges selection on the property bar. With the black arrow, previously selected members can be removed from the selection or new members can be added.

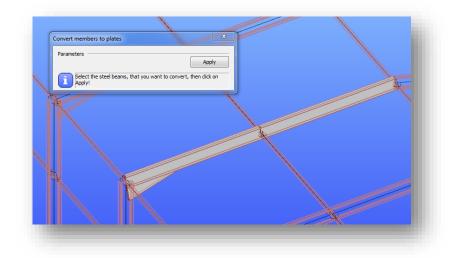


8		Rigid body (1)
	Rigid body2	Name
)	Rigid body2	Name

5.6 CONVERT MEMBERS TO PLATES

The selected members can be automatically converted to plates with **CONVERT MEMBERS**

TO PLATES function (



The following members with section can be converted to plates:

- ▶ Hot-rolled I, H and tube section
- Cold formed C, Z and tube section
- Welded I, H and box section

By clicking on the **APPLY** button the selected members will be converted to plates. During the conversion the given eccentricities are considered therefore the eccentric loads and supports will be in the same position after the conversion. Attaching nodes of the connecting members, haunches and tapered members are automatically converted.

In case of hot-rolled sections not only the flanges and web are converted to plates but also the neck area with a suitable additional plate element. Therefore the section properties of the original member and the converted are the same.



12.0 USER MANUAL

5.7 FRAME CORNER WIZARD

Geometry	Structural members	Loads	Mass	Finite element	Analysis	Global checks	Member checks	Serviceability checks	
	💋 🛯 🛹 🔪	R 🗳	43	📐 🦓 🔁		# 🌆	* 🖍 😪 (🦔 🎰 📑 属	7
			/	Frame corner	wizard 🛌				

The *FRAME CORNER WIZARD* is an optional function for modelling and calculation of the corner regions of structural models more realistically considering the overlapping of connected members. The function is automatically identifies the corner zones and applies special treatment for the geometrical modelling, buckling analysis, plastic analysis and global checks.

The main mechanical background is that these – generally stiffened – corner zones have usually significantly different behaviour than the connected beam members. Since these zones are very short the behaviour is influenced dominantly by the shear effect while the connected members have beam-column behaviour with dominant bending effect.

Functions of the frame corcer wizard dialogue:



#1 - With the button frame corner recognition can be turned on, and off.: By turning on frame corner recognition, frame corner definition can be started. Already defined



frame corners can be turned off by clicking the button, and turned on again if it is desired.

- #2 With the more about the function button, a new panel will open, where detailed information about the function is available.
- #3 Displays the model portion tree, in which it can be decided that what type of frame corner should be applied on which portion. It is important to note, that special frame corners (#5, #6, #7) can only be placed on model portions. The default frame corner type can be applied ont he whole model. To place a special frame corner, it is necessary to select a model portion, and than with a click on a type of frame corner and ont he apply button, the the frame corner will be placed ont he model.
- #4 Default frame corner: 7. DOF displacements of endnodes of neighboring elements are transferred independently of the topology of the joint
- #5 Box-type stiffened bolted or welded joint: Placement of a constraint element between neighboring elements, resulting displacements with opposite sign of 7. DOF of their nodes located at the boundary of the corner zones.
- #6 Bolted or welded joint with diagonal endplate: Placement of a constraint element between neighboring elements, resulting equal displacements of 7. DOF of their nodes located at the boundary of the corner zones.
- #7 Boksz-type stiffened bolted or welded joint with additional 1 or 2 diagonal stiffener(s): Placement of a constraint element between neighboring elements, resulting no displacements of 7. DOF of their nodes located at the boundary of the corner zones.

ing			Frame corner zones
	Joint topological type	Warping transfer	Special handling of frame corner zones for cases shown in figures below, according to the - Leaving out beam segments of frame corner zones - plastic hings
	-	7. DOF displacements of endnodes of neighboring elements are transferred independently of the topology of the joint	- standard - Adjusting haunches to the flanges of the - Forces of beams are taken from the flange of the columns for steel joint checks
and	box-type stiffened bolted or welded joint	placement of a constraint element between neighboring elements, resulting displacements with opposite sign of 7. DOP of their nodes located at the boundary of the correr zones	
and	bolted or welded joint with diagonal endplate	placement of a constraint element between neighboring elements, resulting equal displacements of 7. DOF of their nodes located at the boundary of the corner zones	The kinematical constraints between 7. DOF displacements of neighboring elements are defined in accordance with the following:
	box-type stiffened bolted or welded joint with additional 1 or 2 diagonal stiffener(s)	placement of a constraint element between neighboring elements, resulting no displacements of 7. DOF of their nodes located at the boundary of the comer zones	with the following . [[Basagillo Catel - Torsion warging transmission at thin-walled frame joints. Kinematics, modelling and structural response. J Constel Sciel Res (2011) [] Shyberg S- Chaptel - A model for warging transmission through joints of steel frames. Research Report (2002, July 2011) [] Kindmann R - Steel Structures Besign using FEM [] Statzberger G- Modell forung von Konstruktionsdetals in risminichen Stabberechnungen mittels geometreche Zwergegehungen. 1999

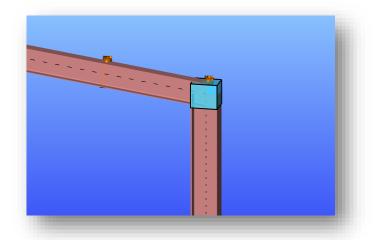


12.0 USER MANUAL

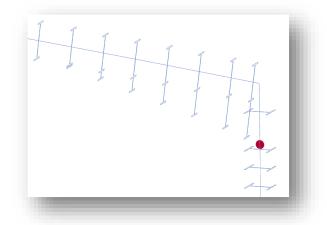
After clicking on the **APPLY** button the program detects the overlapping zones of the members in the identified connection nodes including occurrent eccentricities, haunches or tapered members.

The member segments inside the overlapping zones are separated and handled specifically:

 Geometry: the haunch is started at the end of the truncated member (at the end of the overlapping member segment). In this way the real dimensions and position of the haunch is modelled



- Buckling analysis: the finite elements belonging to the beam segments inside the overlapping nodes are left out from the buckling analysis (as it would be an unselected model portion). The reason is that in this zone member buckling cannot occur, and the effective buckling lengths of the members actually start outside of this zone
- Plastic analysis: the plastic hinges cannot be formed inside the corner zone. Accordingly the plastic hinges are formed at the theoretically exact positions (for instance on the column below the haunch) and not in the connection node where the "virtual" bending moment is the highest

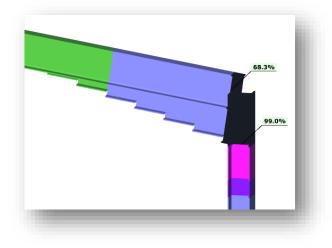


 Global checks: the global checks (including section and stability checks) are performed only for the member parts outside of the corner zone. In this way the "virtual" peaks



12.0 USER MANUAL

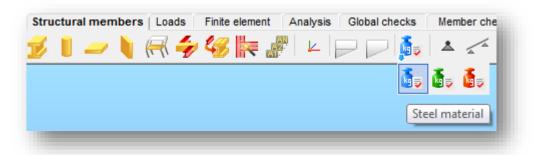
of the bending moments at the connection nodes inside the corner zones (which has no real mechanical meaning) are left out from the design checks and members are checked for the real, considerably bending moment (and the corresponding other internal force) values.



 Connection design: similarly to the earlier point the design bending moment and shear force values for the connection design are taken from the end of the truncated members where a beam-to-column connection is applied to the connection node where corner zone is defined

5.8 MATERIALS

New material grades can be defined as one of the three different types of material: steel, concrete and concrete reinforcement.



The latter is only used for the rebar reinforcement of concrete or composite cross sections. The material parameters are taken from the appropriate Structural Eurocode chapters (EN 1993-1-1 for steel and EN 1992-1-1 for concrete and reinforcement), and the default – indelible – material types are set with the standard values. New materials can also be created with arbitrary parameters.



Name	
Elastic modulus [N/mm2]	210000
Poisson factor	0,30
Thermal expansion coefficient [1/°C]	0,000012
Thermal expansion coefficient in fire situatio.	0,000014
Density [kg/m3]	7850
Fy1 [N/mm2]	235
Fy2 [N/mm2]	215
Fu1 [N/mm2]	360
Fu2 [N/mm2]	360
Thickness Y [mm]	40
Thickness U [mm]	40

In ConSteel only elastic material is considered in the analysis, so the calculation results are only affected by the elastic modulus, Poisson factor, density (if the self-weight of the structure is considered) and temperature expansion factor (if temperature load or fire is applied). In case of concrete material the *Effective elastic modulus* is used in the analysis as it defined in the EC 2.

Name	*
Secant modulus of elasticity [N/mm2]	27085,00
Elastic safety factor	1,20
Compression strength [N/mm2]	12,00
Compression strength modification factor	1,00
Tension strength [N/mm2]	1,10
Tension strength modification factor	1,00
Safety factor	1,50
Final creep	2,10 🗉
Poisson factor	0,20
Thermal expansion coefficient [1/°C]	0,000010
Density [kg/m3]	2500,00
Material model	Rectang 💌
Yield strain #1	0,00070
Ultimate strain #1	0,00350
Compression strength reduction factor	1,00
Yield strain #2	0,0020
Ultimate strain #2	0,00350
Yield strain #3	0,001750 👻

CONSTEEL

ame of reinforcement: B	240	•
Name		*
Poisson factor	0,30	
Thermal expansion coefficient [1/°C]	0,000010	
Density [kg/m3]	7850	=
Fyk [N/mm2]	240	-
EpsSu	0,03	
GammaS	1,15	-



12.0 USER MANUAL



IMPORTANT TO NOTE THAT IN CASE OF BAR MEMBERS (BEAMS, COLUMNS) THE MATERIAL IS THE PARAMETER OF THE CROSS SECTION OF THE MEMBER, SO THE CURRENT MODIFICATION SHOULD BE APPLIED FOR THE APPROPRIATE CROSS SECTION. THIS FEATURE ALLOWS THE DEFINITION OF CROSS SECTIONS WITH MULTIPLE MATERIALS (E.G. COMPOSITE SECTION). ACCORDINGLY IF IDENTICAL CROSS SECTIONS WITH DIFFERENT MATERIAL ARE LIKED TO BE USED IN ONE MODEL THEN MULTIPLE CROSS SECTION DEFINITION IS NEEDED!

5.9 SUPPORTS

There are three types of supports in *ConSteel*: point support, line support, and surface support.



Supports can be placed according to the Point loads can be placed according to the Global (



Placing supports according to the member local coordinate system is very useful feature when working with sloping members. The visibility of the local coordinate system can be turned on with the *Visibility of the coordinate systems* option and also the name of the axes can be shown on the screen using the appropriate options of the *Visibility of labels* functions.

21
2

S	CONSTEEL-	

12.0 USER MANUAL

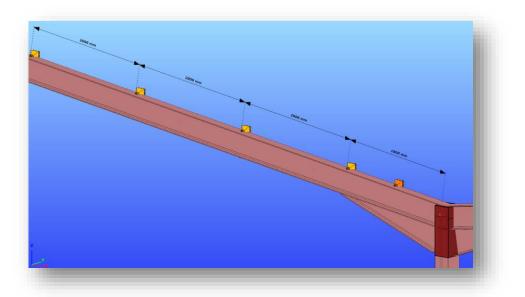
5.9.1 POINT SUPPORT (

Parameters	i 🔍 🛩 🏤	
Support Local eccentricity	Befogott	•
Туре	0 - Reference line y [mm] 0	▼ z [mm] 0
Relative distances	(e.g.: 300, 5*2200) [mm]	
Select the po	int for support!	

Point supports can be placed on any part of a line member or surface member (predefined points are not needed; end points or snap points can be used). With the black arrow button (

) **MULTIPLE PLACING** is possible with window selection.

With the **MULTIPLE SUPPORT** placement function (), more supports can be placed along a bar member at once by defining the relative distances between the support points. Support placement starts from the start point (End A) of the member. If there is an applied frame corner on the member, the first distance will be measured from the edge of the corner zone.



Supports placed with multiple support function will act like one object. Relative distances can be edited later at any time at the object properties by selecting the multiple support.

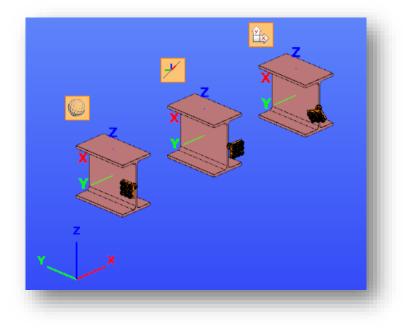


Point support (1)		- 8/
Name	P84	
Support	Încastrat	
Action point		
Ref. line of ecc.	main model, id=463	
Eccentricity reference	0 - Reference line	
Eccentricity - y	0	
Eccentricity - z	0	
Distances	300 5*2200	

After selecting the coordinate system, the type of the support has to be selected from the listbox. There are several point supports types predefined.

By positioning, the used coordinate system determines the orientation of constrains of the

support. Using the **Global (**), **Local (**) or the **User coordinate system (**) the orientations of the constrains represented by the support will be different:



By changing the eccentricity of the supported object, the support will keep the orientation to the coordinate system which was used by placing the support.

If any special supports are needed during the modelling process different from the predefined support types then click on the support definition button (....). With the **NEW** button new type of point support can be defined. 7 DOFs (Degrees of Freedom) can be set to free, fix or semi-rigid. For semi-rigid DOF the stiffness must also be set in kN/mm, or kNmm/rad.

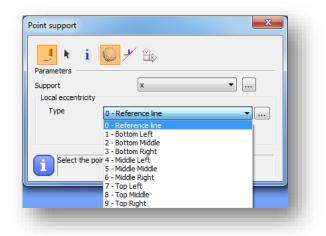


legr	ees of fre	edom		
	Free	Fix	Semi-rigid	Stiffness(kN/mm, kNm/rad)
C	۲	\odot	\odot	
(۲	\bigcirc	\bigcirc	-
z	\odot	۲	\odot	•
xx	۲	\odot	\odot	
RY	۲	\bigcirc	\bigcirc	
RΖ	\odot	۲	\odot	
Rw	\bigcirc	۲	\bigcirc	
	lew	Delete		Close Apply

The support type names can be clearly understood. For instance "x,y,z, xx" means any movement is fixed in x, y and z direction and the rotation around x axe is also fixed. All the rest DOFs are free.

Local eccentricity can also be defined to the supports:

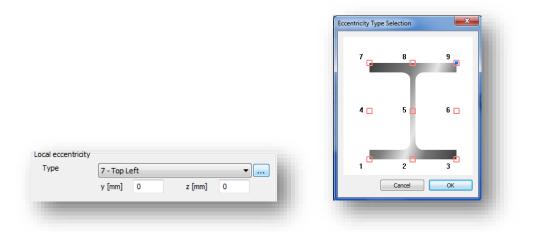
(This feature can be used for example for modelling the support effect of the bracing which is not connected to the reference line of the member but supporting the flange of the beam.)



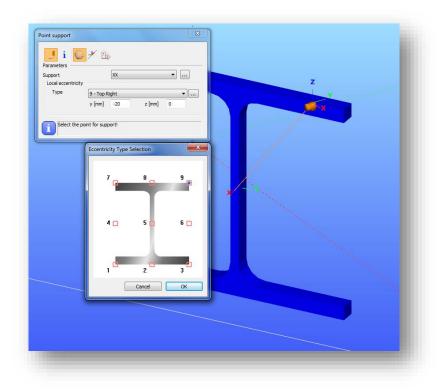
The eccentricity of the support can be defined **relative to the reference line** of a section, or **relative to the section geometry**. The "0 - Reference line" local eccentricity type can be selected only from this list-box.

Support eccentricity **relative to the section geometry** can be specified in two ways. The first way is: select one of the typical point of the section geometry (1-9) from the list, the second way is select one of the typical point (1-9) of the section relative position of the support by clicking the position on the graphical imitation of a section (click the ... button left to the list):





Giving value for the y or z parameters for local eccentricity these values will be added to the above selected position.



Changing the eccentricity of the supported object the new position of the support will be calculated accordingly.

Depending on the type of the selected local eccentricity used by placing the support the transformation rules are the following:

By selecting the "0 – Reference line" eccentricity type:

Placing a support with this eccentricity type, for example changing the "y" eccentricity of the supported object, the support will keep its position relative to the reference line,



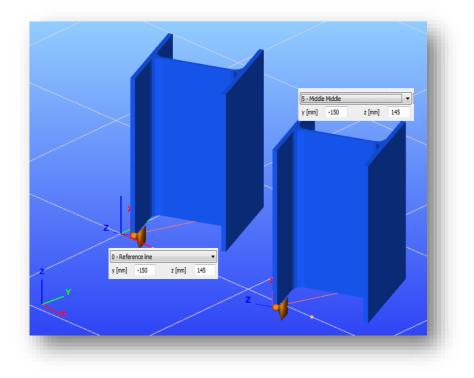
not to the section. By changing the "Rotation angle" attribute of the supported object, the reference line also rotates, and the position of the support will rotate too.

By selecting the 1 -9 eccentricity type:

Placing a support with any of these eccentricity types, for example changing the "y" eccentricity of the supported object, the support will keep its position relative to the section. By changing the "Rotation angle" attribute of the supported object, the position of the support will rotate too.

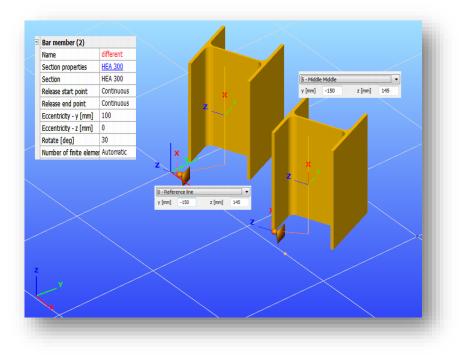
Example for the different types of eccentricity:

1. Example: Both supports were placed with -150mm eccentricity in the y direction, but left case was defined relative to the reference line of a section and the right case was defined relative to the middle-middle point of the section geometry.



If the eccentricity of the columns is changed to y= 100mm, the new position of the supports is as follows:





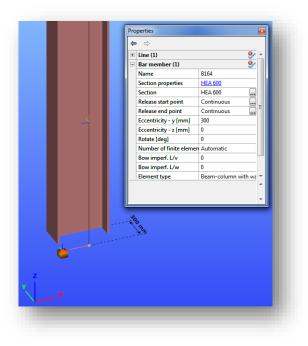
In case of the left case the position of the support did not change, because the position of the reference line of the column also did not change, but in case of the right case the support was moved with the points of the section of the column.

2. Example: The support is placed according to the global coordinate system; the eccentricity is 300 mm in the z direction of the local coordinate system of the supported column from the reference line.

Point support		×	
Parameters Support Local eccentricity Type	Fixed 0 - Reference line y [mm] 0 z	▼ ▼ z [mm] 300	
Select the po	int for support!		
Z			
× ×			

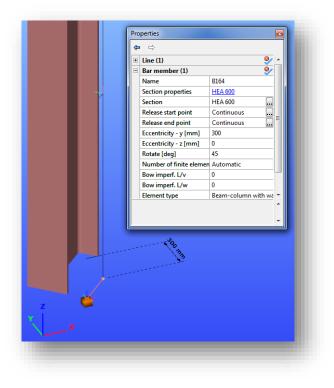


If the column is moved with 300mm in the y direction in the local coordinate system, then the position of the support will no change.



The support keeps its original relative position to the centreline of the column.

If the column is rotated with 45 degrees, the support is also rotated accordingly, but it keeps the directions of its local axis parallel to the axis of the global coordinate system, which was selected by for positioning the support.





3. Example: The support is placed according to the global coordinate system, the eccentricity is 300mm in the z direction of the local coordinate system of the supported column relative to the middle-middle point of the section geometry.

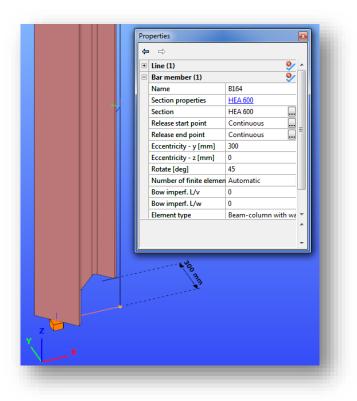
oint support	≁ ₩
Support Local eccentricity Type	
TYPE	5 - Middle Middle v y [mm] 0 z [mm] 300
z	

If the column is moved with 300mm in the y direction in the local coordinate system, then the support moves with the column.

Pro	perties		
	Line (1)	8/	
	Bar member (1)		
	Name	B164	
	Section properties	HEA 600	
	Section	HEA 600	
	Release start point	Continuous	
	Release end point	Continuous	=
	Eccentricity - y [mm]	300	
	Eccentricity - z [mm]	0	
	Rotate [deg]	0	
	Number of finite elemen	Automatic	
	Bow imperf. L/v	0	-
	Bow imperf. L/w	0	
	Element type	Beam-column with wa	Ŧ
	100 mm		•
z x			



If the column is rotated with 45 degree, then the support moves together with the column.



Selecting any of the above eccentricity types for positioning a support, changing the eccentricity of the supported object, the support will keep the originally given relative position from the new position of the supported object, and will keep the orientation defined by the applied coordinate system for placing this support.



WARNING!

BE AWARE THAT SINCE THE FINITE ELEMENT FOR THE STEEL BEAMS AND COLUMNS HAS 7 DOFS THE POINT SUPPORT ALSO HAS 7 DOFS, THE 7TH DOF REPRESENTS THE WARPING OF THE CROSS SECTION. ACCORDINGLY IF A CROSS SECTION ON A CERTAIN PLACE IS CONSIDERED TO FIXED FOR WARPING (FOR INSTANCE IN HEAVILY STIFFENED JOINTS) THE 7TH DOF SHOULD BE FIXED. IN JOINTS CONSISTING SEVERAL MEMBERS (ESPECIALLY WHEN MEMBER ECCENTRICITIES ARE PRESENT) IT IS RECOMMENDED TO APPLY WARPING SUPPORT!

5.9.2 LINE SUPPORT (



Line supports can be assigned to existing lines (line members or edge of surface members) or to lines drawn for the support only (for instance on a surface).



ne support	
👌 🤳 i	🔘 🗾 🏤 🛴 🖸
Parameters	
Support	Fixed 🔻 🗔
Local eccentricity	Fixed
Туре	0 - Reference line
	y [mm] 0 z [mm] 0
Select the lir	ne for support!

There are two default types for line supports the Fixed and the Pinned. With the button it is also possible to define a new line support by giving the attributes of constrains as it is needed.

lame of support model			S-x,y,zz,w	•
Degr	ees of fre Free	edom Fix	Semi-rigid	Stiffness(kN/mm/m, kNm/rad/m)
(\bigcirc	۲	\odot	•
Y	\bigcirc	۲	\odot	· · · · · · · · · · · · · · · · · · ·
z	۲	\odot	\odot	
RX	۲	\odot	\odot	
RY	۲	\odot	\odot	
RZ	\bigcirc	۲	\odot	•
Rw	\bigcirc	۲	\odot	•

By positioning a line support the effect of the applied coordinate system and local eccentricity type are the same as for the point support.



The line support will be placed to the selected edge of a surface (an edge of boundary or an edge of a hole), or to an axis of a beam.

The draw function:





The line support will be placed between the first and the second point defined by the draw function on the surface or on an axis of a beam.



Surface supports can be assigned to existing surfaces.

There are no predefined supports for supporting surfaces. By pressing the button, the user can create new surface support types for specifying 3 constrains.

▶ J i 🥥 🗡 🏤	Surface support Name of support model SFS-x, y, z	
Parameters	Degrees of freedom Free Fix Semi-rigid Stiffness(kN/mm/m ² , kNm/rad/n	n ²)
	x • • •	-
Select the surface for support!	Y 💿 🔘 🗌	•
	Z 💿 💿 💿	•
	New Delete Close Apply	

By clicking on a plate object, all the surface of this selected object will be supported by the support.

5.10 LINK ELEMENTS

Link element () can be used to connect elements which are not directly connected to each other.



Parameters of the link element can be defined in the middle part of the dialog:

- Position of connection: this parameter defines the place where the continuity acts in the percentage of the length of the link element. Value can be between 0 and 1
- Rotation: rotation value defines the angle of the link element in degrees
- Release: predefined releases can be assigned to the specified position. If new release type is needed the RELEASE dialogue can be called by pressing the ... button next to the combo

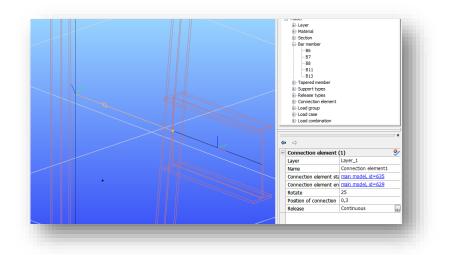
There are two different types of link element.

5.10.1 CONNECT TWO STRUCTURAL POINTS (

This type of link element can be used to connect two structural points.

t ttt	
Position of connection 0,3	Rotation 0
Release yy,zz	▼
Set the start point!	

For example if there is a cantilever on a relatively high tapered column (see image below) it is advisable to connect it not directly to the reference line of the column but with the link element. If it is connected the way like that the analysis and design results are more accurate and the modelling is more exact.



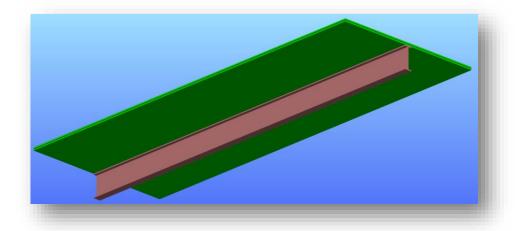


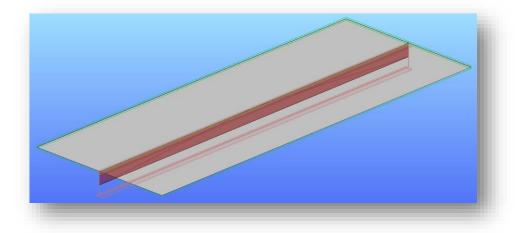
12.0 USER MANUAL

5.10.2 Connect two structural edges that are equal length and parallel (

Equal length and parallel edges and members can be connected with this kind of link element.

This type of link element can be used for example when connecting a slab eccentrically to a beam.





5.11 Shear field

With the new object, the stiffening effect of trapezoid sheeting can be considered in buckling analysis.

It is important to note, that the trapezoidal decking has to be fixed in an appropriate way to the understructure (all 4 sides on the trapezoidal decking) to be able to calculate proper results.



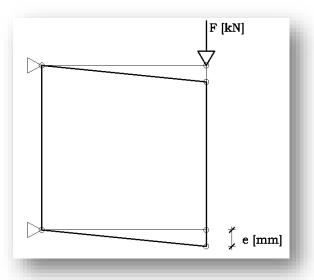
5.11.1 DEFINITION OF SHEAR STIFFNESS

On the picture below, the interpretation of shear stiffness can be seen, where "e" is the displacement of the shear diaphragm on the effect of a given F [kN] force. Shear elasticity by definition:

$$c = \frac{e}{F}$$
: Shear elasticity [mm/kN]

Reciprocal of the shear elasticity gives the shear stiffness, which is signed by S:

$$S = \frac{F}{e}$$
: Shear stiffness [kN/mm]



Shear diaphragm deformation to a given force [1]

A nyírási rugalmasságot és nyírási merevséget legtöbbször trapézlemezeknél értelmezik. Egy ilyen lemez esetén a teljes nyírási rugalmasság (C) az alábbi komponensekből tevődik össze:

Shear stiffness of a trapezoidal sheet consists of the following components:

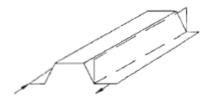
• Deformation of the profile:

 (h) profile distortion : every trough fostened

 (i) profile distortion : alternate troughs fastened

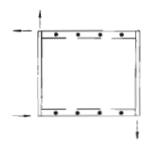


• Shear deformation in the sheet:

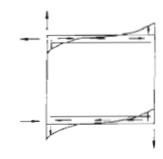


(j) shear strain in sheeting

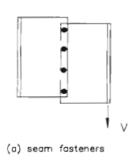
• Slip of the fastening elements:



(c) sheet/purlin fasteners:4 sides fastened



- (d) sheet/purlin fasteners:
 2 sides only fastened
- Displacement of the connections



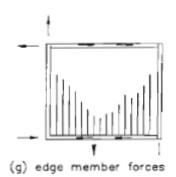
• Displacement of the beam-purlin connection



(k) purlin/rafter connection



• Axial deformation in the longitudinal edge members:



The summation of the 6 different components above, gives the value of the shear elasticity (C). Reciprocal of the shear elasticity is the shear stiffness (S), which can be taken into consideration during the buckling analysis.

5.11.2 CALCULATION OF SHEAR STIFFNESS

DNSTEEL

To the determination of shear stiffnesses, different German trapezoidal sheet manufacturers developed their own calculation methodologies for their own products. Each method can only be used for the product for which it was developed for.

The following methods are implemented in the software:

- Hoesch
- Fischer
- Arcelor
- EuroCode

5.11.2.1 DETERMINATION OF SHEAR STIFFNESS IN CASE OF PANELS PRODUCED BY HOESCH

The recommended method uses the following formula (DIN 18807, Schardt/Strehl method):

$$S = \frac{10^4}{K_1 + \frac{K_2}{L}} * a$$
 (1)

S: stiffness of the shear field [kN]

 K_1 : parameter specific to the selected panel [m/kN]

 K_2 : parameter specific to the selected $[m^2/kN]$

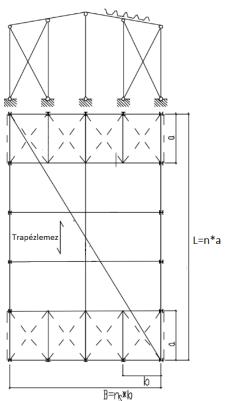
- L: length of the shear field parallel to the direction of the panel ribs [m]
- a: applicable effective width [m]



The figure shows the sketch of a general building showing the dimensions used in the previous formula. The method assumes that panels on all 4 edges along the boundary of the considered shear field are fixed to supporting structures with adequate spacing.

The values K_1 and K_2 have been specified by the producer for each panel type in function of its thickness. Thesevalues can be found on the website of the producer or in official application certificates. Important to note, that such certificates have a certain validity of application, therefore it is always recommended to double check the validity of the values considered by ConSteel.

The S value determined with formula (1) is valid if the trapezoidal sheet is fixed at each rib to the supporting structure. The S value shall be multiplied with 0,2 in case of fixations in every second rib only.



5.11.2.2 DETERMINATION OF SHEAR STIFFNESS IN CASE OF PANELS PRODUCED BY FISCHER

The recommended method uses the following formula (improved Schradt/Strehl method) (2). The formula contains 3 additional parameters ($K1^*$, $K2^*$ and eL) in comparison with the classical method, to consider the effect of the fixations of the panels.

$$S = \frac{1}{(K_1 + K_1^* e_L) + \frac{K_2 + K_2^*}{L}} * a \quad (2)$$

- S: stiffness of the shear field [kN]
- K_1 : parameter specific to the selected panel [10^{-4*}m/kN]
- K₂: parameter specific to the selected panel $[10^{-4*}m^2/kN]$
- K_1^* : parameter specific to the selected panel [10⁻⁴*1/kN]
- K_2^* : parameter specific to the selected panel [10^{-4*}m²/kN]
- **e**_L: distance between fixations lengthwise [m]
- **a**: applicable effective width [m]

The method assumes that panels on all 4 edges along the boundary of the considered shear field are fixed to supporting structures with adequate spacing.

The S value determined with formula (2) is valid if the trapezoidal sheet is fixed at each rib to the supporting structure. The S value shall be multiplied with 0,2 in case of fixations in every second rib only.

5.11.2.3 DETERMINATION OF SHEAR STIFFNESS IN CASE OF PANELS PRODUCED BY ACHELOR

The recommended method by Alcerol uses the following formula (3) (Bryan/Davies):

$$S = \frac{10^4}{\left[(K_1' * \alpha_2 + K_1^* e_L) + \frac{(K_2' * \alpha_1 * \alpha_4 + K_2^* * \alpha_3)}{L_s} \right]} * \alpha$$
(3)

S: stiffness of the shear field [kN]

CONSTEEL

K1': parameter specific to the selected panel [m/kN]

K₂': parameter specific to the selected panel [m²/kN]

K₁*: parameter specific to the selected panel [1/kN]

 K_2^* : parameter specific to the selected panel [m²/kN]

L_s: length of the shear field parallel to the direction of the panel ribs [m]

 α_1 , α_2 , α_3 : additional parameters depending on the number of panel spans defined in tables

 α_4 : additional parameter depending on the number of panel splices lengthwise

a: applicable effective width [m]

The method assumes that panels on all 4 edges along the boundary of the considered shear field are fixed to supporting structures with adequate spacing.

The S value determined with formula (2) is valid if the trapezoidal sheet is fixed at each rib to the supporting structure. The S value shall be multiplied with 0,2 in case of fixations in every second rib only.

5.11.2.4 DETERMINATION OF SHEAR STIFFNESS ACCORDING TO EUROCODE 3

The recommended method uses the following formula (4)

$$S = \left(1000 * \sqrt{t^3} * \left(50 + 10 * \sqrt[3]{b_{roof}}\right) * \frac{1}{h_w}\right) * a \tag{4}$$

- **S**: stiffness of the shear field [kN]
- t: thickness of the panel [mm]
- h_w : depth of the panel [mm]
- **a**: applicable effective width [m]



b_{roof}: length of the shear field parallel to the direction of the panel ribs (width of the roof) [mm]

The method doesn't require that panels are fixed on all 4 edges along the boundary of the considered shear field are fixed to supporting structures. As a minimum fixation to directly stabilized structures (2 sides) with adequate spacing is required.

The S value determined with formula (4) is valid if the trapezoidal sheet is fixed at each rib to the supporting structure. The S value shall be multiplied with 0,2 in case of fixations in every second rib only.

5.11.3 Shear field stiffness definition

The dialog of the **SHEAR FIELD STIFFNESS DEFINITION** can be opened with the ^{SHEAR} button of the **STRUCTURAL MEMBERS** tab.

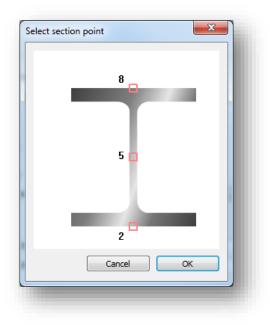
Shear field stiffness	value		
1 s	Set value manual		
	0	kN/m	
Effective width (a)	1	m	
Local eccentricity Type	0 - Reference line	z [mm] 0	
Position of S along the	e line (X1)	From end A 🔹 🔍) mm
Position of S along the	e line (X ₂)	From end B 💌 🛛) mm
4 A		B	



#1 - Value of shear stiffness can be defined manually, or if the (....) button is clicked, on the **SHEAR FIELD STIFFNESS CALCULATION** (see chapter **5.11.3.1**) it can be calculated automatically.

- #2 Definition of effective width parameter
- #3 Local eccentricity of the shear field to the member on which it will be placed:
 - 0. Reference line
 - 2. Bottom Middle
 - 5. Middle Middle
 - 8. Top Middle

By clicking on the <u>button</u>, action point of eccentricity can be defined graphically on a section illustration.

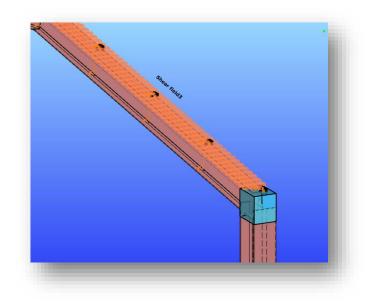


- #4 -Position of the *SHEAR FIELD* along the bar member on which it will be placed:
 - 1. X₁:
 - a. From end A of the member: Distance X1 is measured from the start point of the member
 - b. From end B of the member: Distance X1 is measured from the end point of the member
 - 2. X₂:
- a. From end A of the member: Distance X1 is measured from the start point of the member



- b. From end B of the member: Distance X1 is measured from the end point of the member
- c. Relative: Defining X2 according relatively to X1

#5 - With the button, members can be selected in the model, on which the *SHEAR FIELD* should be applied.



5.11.3.1 SHEAR FIELD STIFFNESS CALCULATION

ame of shear field	ShearField_ 1	•	
ofile of trapezoid plate	FI 165/250/Special fi	•	Jan 1
oplied calculation	Schardt/Strehl improv	ed 🔻	S
roperties of shear field			
Fixing	each rib		
	\bigcirc each second rib		
Shear panel length (L ₂)	6	m	
Fixing distance (e _L)	3	m	
lumber of bays	1 *		· · · ·
lumber of longitudinal splices (n' $_{\rm b}$)	1		
ear field stiffness value	576	kN/m	
ethod can only be used, if all 4 edge	es of the shear papel are		
nnected to the substructure.			L _s =b _{roof}
New Delete			Close Apply



Definition of the parameters of a new *SHIER FIELD* object can be started after the *NEW* button is clicked.

As a first step, a trapeziodal plate profile has to be loaded into the model on the **SELECT TRAPEZOID PLATE SECTION** dialogue by clicking on the three dots button (....). The parts

of the dialogue are the following:

ection bank:		Section parameters								
Trapezoidal sheets	*	⊉ ame	Paran	neters	K1	K2	K1*	K2*	Lsmin	Note
i∰- Fisher ⊟- Hoesch	E	T 35.1-1	hw=3	32,0 mm, t= 1,0 mm, b	0,1	2,7	0,0	0,0	1,4	valid unti
Positive										
T 35.1-0.63 T 35.1-0.75		Loaded sections								Load
T 35. 1-0.88		Name		Manufacturer		Calcul	lation m	ethod		Layer
		FI 165/250/Special fixing-1		Fisher		Schare	lt/Strehl	improv	ed	Positive
		3 New Delete		Modify						Close

#1 -Trapezoidal sheet section bank: Sections are collected by manufacturers in the tree structure.

#2 -Section parameters: Properties of the selected trapezoidal sheet in the tree structure(#1) is represented in this table.

#3 -After a section is selected in the section tree (#1), the *LOAD* button has to be clicked to load into the *LOADED SECTIONS* table. Loaded sections can be used for the calculation of sshear stiffness on the *SHEAR FIELD STIFFNESS CALCULATION* dialogue.

With the **DELETE** button, previously loaded trapezoidal sheet cross sections can be deleted.

By clicking the **NEW** button, manual trapezoidal sheets can be defined. On the **MACRO SECTION** dialogue, geometric parameters can be defined, and the custom sheet can be renamed, and saved.



Name		Trapezoid	Trapezoidal sheets						
Manufactur	er	Manufact	lanufacturer						
Fixing		Positive	Positive layer						
Note		Valid	Valid						
Calculation	method	Schardt/S	trehl/DIN 188	07	•				
Geometrie	cal paramete	ers							
	bu								
+			_		_				
hp					$-\sqrt{2}$				
1-1-1	1				¥-				
			+		+ T				
			b _b		tp				
	+		b _b		tp				
Ьь	33	mm		0,242	tp m/kN				
bь bu	33 106	mm	br	0,242					
			b _г К1		m/kN				
bu	106	mm	b _r К1 К2	7,871	m/kN m²/kN				
bu br	106 207	mm mm	b _г К1 К2 К1 [*]	7,871 0	m/kN m²/kN m/kN				
bu br t	106 207 0,63	mm mm mm	br K1 K2 K1* K2*	7,871 0 0 0	m/kN m²/kN m/kN m²/kN				

The following parameters has to be defined for the shear stiffness calculation:

- Applied method for the calculation:
 - o Schardt/Strehl/DIN 18807 method
 - o Schardt/Strehl improved method
 - Bryan/Davies method
 - EC based method
- Fixing:
 - Positive layer
 - Negative layer
- Geometric and section specific parameters

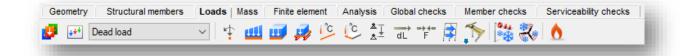
According to the properties of the manually defined or loaded trapezoidal sheet section, shear field stiffness value will be calculated automatically on the **SHEAR FIELD STIFFNESS CALCULATION** dialogue.

SCONSTEEL

6 STRUCTURAL LOADS

6.1 BASICS

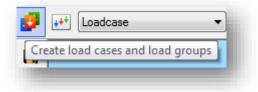
The definition of loading on a structural model is one of the most important modelling phases. Contrary to the modelling of structural members the load modelling is minutely controlled and supported by the structural codes and standards since the appropriate definition of loads ensures the major part in the reliability of structural performance. In *ConSteel* several types of loading options help the engineer in this work. In accordance with the modelling of structural member the engineer can work with the load types and options defined and handled in the structural standards and the applied loads are automatically converted into finite element loads for the calculation model. All the loading functionality connecting with the load modelling is placed on the *LOADS* tab.



6.2 LOAD CASES AND LOAD GROUPS

Before place any loads in the model load cases and load groups (¹) must be defined. As a default, one load group and one load case have been defined in ConSteel for quick calculations.

In a Load group several load cases can be defined. Properly specified load groups needed to automatically generate load combinations.



There are 9 different load group types in *ConSteel*: Permanent, Variable, Meteorological, Wind, Snow, Crane, Accidental, Accidental snow, Fire, Seismic.



By clicking on the **NEW** button the standard load-groups will appear. Selecting one of the name, a new load group is created.

Ų Persistent	U Permanent	
└── <u>↓</u> Dead load ↓ Live	Variable stent	
Live load	Meteorological	
Snow	Wind le 2	6 sup 1,35
Snow load	Snow	
	Crane 3	G.inf 1
	Accidental	
	Accidental snow	
	Fire	
	E Seismic	
	Cancel	Delete
	Load group Color	
	New load case Copy	Delete
	Dead load	\sim

For each load group safety and combination factors must be given. The combination factors can be edited manually or can be selected on the *STANDARD VALUES* of factors dialog. On the dialog typical cases can be selected which are provided by the EuroCode and the Spanish EAE standard. Factor values are dependent from the selected NA.

Action type	Ψο	Ψ1	Ψ2 _
Imposed loads in buildings, category (see EN 1991-1-1)			
Category A: areas in residential buildings	0,70	0,50	0,30
Category B: office areas	0,70	0,50	0,30
Category C: congregation areas	0,70	0,70	0,30 ≘
Category D: shopping areas	0,70	0,70	0,60
Category E: storage areas	1,00	0,90	0,80
Category F: traffic area, vehicle weight ≤ 30 kN	0,70	0,70	0,60
Category G: traffic area, 30 kN < vehicle weight ≤ 160kN	0,70	0,50	0,30
Category H: roofs	0,00	0,00	0,00
Snow loads on buildings (see EN 1991-1-3)			
sk < 2,75 kN/m2	0,70	0,40	0,20 -
III	_		•
	Cancel		Apply

CONSTEEL

ConSteel can add the structure's dead load to one of the load cases if necessary. In order to do so please select the appropriate load case at the bottom right corner of the dialogue.

6.3 LOAD COMBINATION

According to the actual structural design code load combinations must be set from load cases.

Load combinations () can be created manually by giving the safety and combination factors manually for each load case or can be created automatically by using the *AUTOMATIC*

LOAD COMBINATION GENERATION (¹¹⁴) function of *ConSteel*.

It is possible to edit multiple safety factors at a time: select safety factors and right click over it.

Cell contents can be copied and pasted. On the table, select the cells that you want to copy and press CTRL+C. Select the upper-left cell of the paste area and press CTRL+V. Cell contents can be copied from MS Excel and from other spreadsheet applications also.

lame	Limit state		Wind load y	Wind load x	Dust load	Eigen gewicht	^
oad combination-2	(ULS) Ultimate	•	0	1,05	0	1	
oad combination-3	(ULS) Ultimate	•	0	0	1,05	1	
oad combination-4	(ULS) Ultimate	-	1,5	0	0	0,85	
oad combination-5	(ULS) Ultimate	•	0	1,5	0	0,85	
oad combination-6	(ULS) Ultimate	•	0	0	1,5	0,85	E
erviceability-1	(SLS) Serviceability	•	0,5	0	0	1	
erviceability-2	(SLS) Serviceability	-	0	0,5	0	1	
erviceability-3	(SLS) Serviceability	-	0	0	0,5	1	
oad combination-7	(ULS) Ultimate	•	1,1	0	0	1	
oad combination-8	(ULS) Ultimate	Ŧ	0	1,5	0	1,1	
				÷			-

The yellow rows in the load combination table are the automatically generated combinations. The white ones are manually edited.

Huge number of load combinations needs long period of time to calculate in the analysis. So it is good to keep the number of the load combinations to the minimum. If the automatic load combination feature is used in *ConSteel* it is advisable to filter the unnecessary loadcases



before the generation. At the *TREE CONTROL* of the right side of the *AUTOMATIC GENERATION OF LOAD COMBINATIONS* dialogue, consideration of the loadcases for the load combination generation can be decided. A *TICKED CHECKBOX* means, that the loadcase will be used for the load combination generation:

timate limit states Serviceability limit states	Considered load cases:
Load combinations in ultimate limit states Load combinations in ultimate limit states Load combinations in persistent and transient design situations - (6. 10.) $\sum_{j \geq 1} \gamma_{G_{ij}} G_{k,j}^{-\mu} * \gamma_{Q_{i}} Q_{k,l}^{-\mu} * \sum_{j \geq 1} \gamma_{Q_{i}} \psi_{0,i} Q_{k,j}$ Load combinations in persistent and transient design situations - (6. 10. a,b) $\begin{cases} \sum_{j \geq 1} \gamma_{G_{ij}} G_{k,j}^{-\mu} * \gamma_{Q_{i}} \psi_{0,i} Q_{k,l}^{-\mu} * \sum_{j \geq 1} \gamma_{Q_{i}} \psi_{0,j} Q_{k,j} \\ \sum_{j \geq 1} \xi_{j} \gamma_{G_{ij}} G_{k,j}^{-\mu} * \gamma_{Q_{i}} Q_{k,l}^{-\mu} * \sum_{j \geq 1} \gamma_{Q_{i}} \psi_{0,j} Q_{k,j} \\ \sum_{j \geq 1} \xi_{j} q_{G_{ij}} G_{k,j}^{-\mu} * A_{i}^{-\mu} * (\psi_{1,i}, \psi_{2,i}) Q_{k,l}^{-\mu} * \sum_{j \geq 1} \psi_{2,j} Q_{2,j} Q_{k,i} \end{cases}$	Image: Section of the section of th
$\begin{array}{c} j \geq 1 & \qquad \text{is1} \\ \hline \hline \\ \hline \\ \hline \\ \hline \\ \\ \\ \\ \\ \\ \\ \\ \\ \\ $	
	Cancel Apply

After the generation of load combinations, deletion of those rows which cannot be relevant is also a possible way to minimize calculation time. To do so, select the load combination (multiple LC-s can be selected with using *CTRL* or *SHIFT* buttons) and click on the *DELETE LOAD COMBINATION* icon.

It is important to set the limit state correctly because design checks will be made to Ultimate (*ULS*) load combinations and also serviceability check are performed for Serviceability (*SLS*) combinations.

6.4 LOAD TYPES

Any placed load will belong to the load case selected from the list.

👫 Dead	load	v 🔯 📶	🔟 🗾 🕓		🖹 🥎 👫 🌄	6
Dead Live lo Snow	ad		/	7		



6.4.1 POINT LOAD (

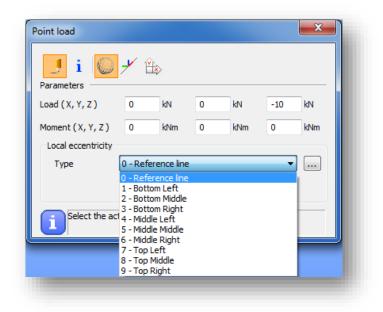
Point loads can be placed on any part of a line member or surface member (predefined points are not needed; end points or snap points can be used).

Point loads can be placed according to the **Global (**), Local () or the User coordinate



<u>।</u> i 🔘	≁ 🖒					
Load (X, Y, Z)	0 k	đN	0	kN	-10	kN
Moment (X, Y, Z)	0 k	۵Nm	0	kNm	0	kNm
Local eccentricity						
Type (0 - Referen	ice line			•	
١	/ [mm] 0			z [mm]	D	
Select the act	on point of	load!				

Eccentricity can be applied on the point load also. The effect of the selected coordinate system, the applied local eccentricity type and values are the same as point support (see chapter 5.9.1).







Line loads can be assigned to existing lines (line members or edge of surface members) or to lines drawn for the loads only (for instance on a surface).

Line loads can be placed according to the Global (¹⁰⁰), Local (¹¹¹), User coordinate system

(E) and **Global projection system** (E) which is commonly used when modeling the snow load.

	i 😡 🖽 🛩		1
q1(X,Y,Z)	0 kN/m	0 kN/m	-10 kN/m
q2(X,Y,Z)	0 kN/m	0 kN/m	-10 kN/m
Position of q ₁ alor Position of q ₂ alo	ng the line (x _{2,r})	From end A	0 mm 0 mm
Туре	0 - Reference line		
	y [mm] 0	z [mm] 0	j
Select the	action line of load!		

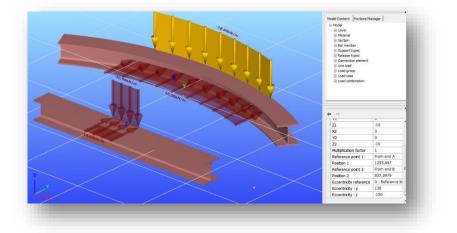
In order to place a line load click on the member using the select function (

can be applied to multiple members at the same time by using *PLACE LOADS* icon (). This case the members must be selected before clicking on the *LINE LOAD* icon on *LOADS* tab.

If just a partial line load need to be applied along the member then there are two ways to apply:

The first way is to use the draw function () and set the start point and the end point of the line load. This function can be used both for linear and curved members in accordance with the selected drawing function ().

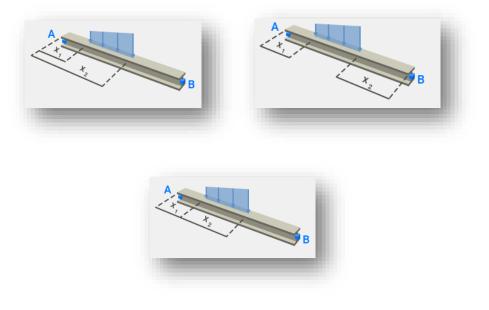




kN/m -10 kN/n kN/m -10 kN/n
kN/m -10 kN/n
В
A ~ 0 mm
B ~ 0 mm
×
0

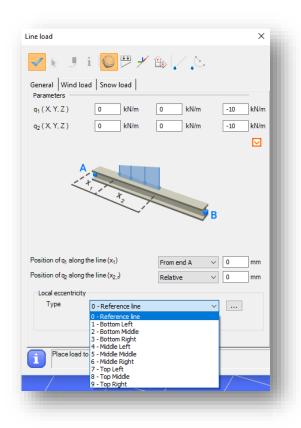
2. The second way is to give the exact distances of the start and end points of the line load from the member's end points. Using the setting of Position of q₁, q₂, there are 2x3 = 6 possible option to define a partial line load. The "Position of q1" set to "From end A" has 3 options, and setting it to "From end B" has also 3 options:





This is a flexible way for numerical input the line load positions.

Eccentricity can be applied on the line load also. The effect of the selected coordinate system, the applied local eccentricity type and values are the same as point support (see chapter 5.9.1).



By clicking the **i** icon, the attributes of the selected line load fill out the parameters of the line load window, even if it was set by the draw function.



6.4.2.1 LINE WIND LOAD

Line wind load is a great tool to apply wind load to members easily. It saves time because only the basic data must be added and *ConSteel* automatically calculates the appropriate wind load.

Before clicking on the *LINE LOAD* icon, as a first step the member(s) has to be selected where the line wind load will be placed.

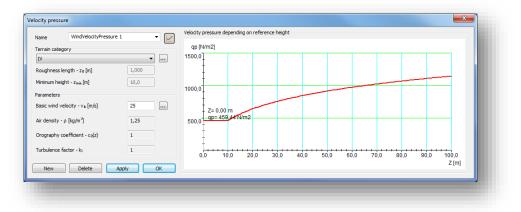
The line wind load related functions can be found on the **WIND LOAD** tab of **LINE LOAD** dialog.

ara	ral Wind action Snow load	
	ity pressure - q _P [kN/m ²] 0,459	
Ir	nternal pressure - c _{pi} 0,2	
E	xtrenal pressure - cpe	
	Load intensity [kN/m]	Length [m]
~	-4,234	1,460
~	-2,203	13,597
~	-2,203	1,460
~	-2,203	13,597
Loc	ection of placement Axis z © ral eccentricity Type 0 - Reference line	Axis y
	y [mm] 0	z [mm] 0

As a first parameter, the *Velocity pressure* has to be added, or by clicking on the three dots button (\cdots), previously created Velocity pressure functions can be selected from the dropdown menu or by clicking on the three dots button (\cdots) a new EuroCode Velocity pressure function can be created according to the selected National Annex to use to calculate the proper *Velocity pressure*.







Internal pressure can be taken in to account by check in the check-box. C_{pi} value can be set manually or by clicking on the three dots button (\dots) and use the Internal pressure calculator function. To calculate the value, two EuroCode methods can be used.

External pressure can be set by clicking on the three dots button (....).

Global wind parameters Surface type			WindGlob	▼		
			Duopitch	Duopitch roof		
Cons	tant param	eter				
#		h [m]	di [i	m]	α [°]	
L	_	7,300	30,0	000	4,999	
	l direction d d direction Li		▼ b [m] A [m"]	50	d (n e (m	
		e figure with : f the frame (x		Frame distar	nce [m]	6
	•	4		d		
	e/4 ↓	F				
-		G F	н	J	I	þ
-	¥		Н	J ∢-e/10-	 ►	b
	_ * _	F ← 0/10-> es EN 1991-1	-4:2007 Tabl	e 7.4		
#	il Cpe valu	F ← e/10 → es EN 1991-1	-4:2007 Tabl	e 7.4 H	I	J
# 1	¥ al Cpe valu	F ← 0/10-> es EN 1991-1	-4:2007 Tabl	e 7.4		
Actu # 1	il Cpe valu α. 4,999	F ← e/10 → es EN 1991-1 F -1,700	-4:2007 Tabl G -1,200	e 7.4 H -0,600	I -0,600	J -0,600



To calculate the correct *External Pressure* values it is necessary to give geometrical parameters of the whole building. Geometrical parameters can be set on the *Global wind parameters* dialog.

ame	WindGlobal 1	•
Directions of basic wind direction ($\Xi=0^{\circ}$) in global Building dimensions to the basic wind direction	coordinate system	+X •
parallel - do [m]		20
perpendicular - b o [m]		50
Loaded area - A [m ²]		10
Ground level - Z [m]		0
	Apply	ОК

Besides the building dimensions, the direction of the basic wind in global coordinate system has to be selected from the dropdown menu and the ground level can be set if the lowest point of the model will not be on the ground.

As a next step one of the standard *Surface type* (wall, flat roof, duopitch roof, etc.) has to be selected from the dropdown menu. If the proper surface is selected, then all of the necessary parameters from the previously selected member(s) are loaded and the actual c_{pe} values are calculated.

To create line wind load it is necessary to set the *Position of the frame in the building* and to set the *Frame distance*.

As a last step the proper wind load cases has to be selected from the dropdown menu, on the bottom of the *EXTERNAL PRESSURE COEFFICIENTS DIALOG*.

By clicking on the **OK** button the line wind loads are created.

Pressing the **PLACE LOADS** icon () on the **LINE LOAD** dialog the created line wind loads are placed on the selected member(s).



	Line load 🛛 🔀
-2.20kN/m -2.20kN/m	General Wind action Snow load Parameters
	Direction of placement Axis z Axis y Local eccentricity Type 0 - Reference line y [mm] 0 z [mm] 0
	Place load to the selected beams!



For the correct line wind load placement, it is necessary to set the direction of the used local coordinate axis (Axis z, or y) to point out from the building.

6.4.2.2 LINE SNOW LOAD

Line snow load is a great tool to apply snow load to members easily. It saves time because only the basic data must be added and *ConSteel* automatically calculates the appropriate snow load.

Before clicking on the *LINE LOAD* icon, as a first step the member(s) has to be selected where the line snow load will be placed.

The line snow load related functions can be found on the **SNOW LOAD** tab of **LINE LOAD** dialog.

The snow load parameters are automatically loaded from the selected NA, but all of these can be modified manually.

/ k 🙂	i ©	** 7	K 👜 📝	<u>c</u>	
eneral Wind act arameters	ion Snov	v load			— (
haracteristic snow	load - s _k	1,25	Snow load	Exceptional	I snow load
/ind factor - C.		1	Exceptional fac	tor - Ceal	2
emperature factor	- Ct	1	Shape factor -	μ	
	y [kN/m]		ntensity [kN/m]		
 -3,000 		-3,000		15,057	
7					
-3,000		-3,000		15,057	
Direction of placen	nent	-3,000	○ Avis v	15,057	
Direction of placen	nent	-3,000	O Axis y	15,057	
Direction of placen			O Axis y		
Direction of placen Axis z Local eccentricity Type	0 - Refere	ence line			
Direction of placen a Axis z Local eccentricity Type	0 - Refere		C Axis y		
Direction of placen Axis z Local eccentricity Type	0 - Refere	ence line			
Direction of placen a Axis z Local eccentricity Type	0 - Refere y [mm]	ence line 0	z [mm]		



As a first step the *Shape factor* has to be set, by clicking on the three dots button (....).

Surface t Constant	ype t parameter	Duopitch	roof	•
#	h [m]	b _i [m]	α1 [°]	α2 [°]
1	7,300	30,000	4,999	4,999
			Frame distance [r	n] <u>3</u>
	μ1(α1)			μ1(α2)
	_			_
	values EN 19			
#		μ1(α1)	μ1(α2)
1	_	0,800	0,800	
ase 1		•	Clos	e OK

To define the shape factor it is necessary to select the *Surface type* (flat roof, duopitch roof, etc.) from the dropdown menu. If the *Surface* type is selected, then the parameters of the previously selected member(s) are automatically loaded and the shape factor is calculated.

As a last step the proper snow load cases has to be selected from the dropdown menu, on the bottom of the **SNOW LOAD DIALOG**.

By clicking on the **OK** button the line snow loads are created.

Pressing the *PLACE LOADS* icon () on the *LINE LOAD* dialog the created line snow loads are placed on the selected member(s).

6.4.3 SURFACE LOAD (



Surface loads can be defined using the same method like in point or in line loads. No surface moment loads can be placed.



i i i i i i General Wind action Parameters Load (X, Y, Z) 0 kN/m2 -10 kN/m2	eate surface load
Parameters Load (X, Y, Z) 0 ktV/m2 0 ktV/m2 -10 ktV/m2	▶ J i 💟 🤊 🗡 🆦 📮 🛇 🖆
Load (X, Y, Z) 0 kN/m2 0 kN/m2 -10 kN/m2	General Wind action
	Parameters
Select the surface!	Load (X, Y, Z) 0 kN/m2 0 kN/m2 -10 kN/m2
	i Select the surface!

6.4.4 LOAD TRANSFER SURFACE (

Load transfer surface is a special surface which distributes surface loads to members as line loads. This is very useful in those cases when surface load need to be distributed to members, like floor loads, snow and wind load, etc.

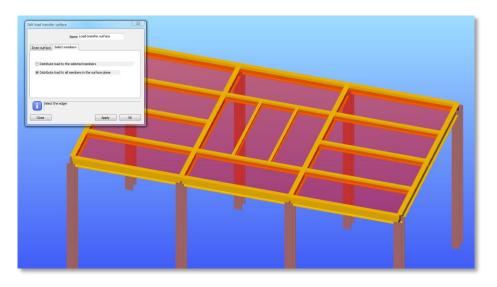
After clicking on the *LOAD TRANSFER SURFACE* (*Joint Contexpension*) icon on the *LOADS* tab a dialog window appears.

Edit load transfer surface
Name Load transfer surface
Draw surface Select members
💺 🗾 🗆 🗆 🛄 💽 🛅 💺 🗶 î 🕮 🖄
Set one corner point of the rectangle!
Close OK OK

Just like in the two-dimensional figure drawing there are different possibilities to draw the surface: draw a rectangle, draw leaning rectangle, draw circle, draw polygon. After drawing the surface it is possible to select the members to which the surface distributes the load. There are two options: distribute load to the selected members or distribute load to all members covered by the surface in the surface plane. If second option is clicked then the appropriate members will be highlighted. If none of the options is clicked, then the second one will be applied as a default.

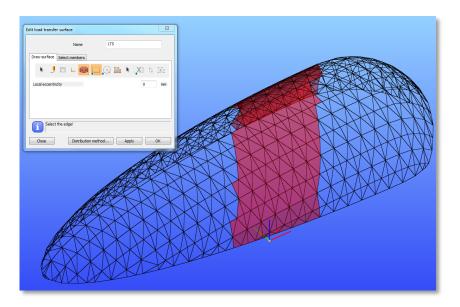






It is possible to choose the first option and select members for carrying the surface load or remove members from the selection using the SHIFT + Left click. After the corresponding members have been selected and OK is clicked then the surface is created.

It is also possible to use the **MULTIPLE PLACEMENT OF LOAD TRANSFER SURFACES** by clicking on the (^{SSD}) icon. This function is used to define multiple load transfer surfaces with a simple selection (rectangle selection, or by clicking the border edges). Edges of a load transfer surface has to be in the one plane.

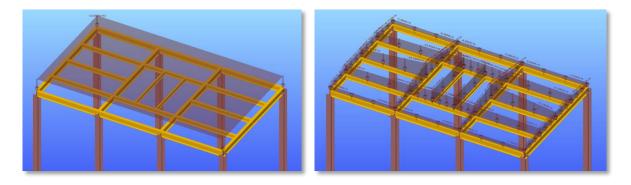


It is important to know, that the maximum number of edges that can be selected in one step, is currently limited in 200. If the selected edges exceeds the limit of 200, a notification will be shown on the *EDIT LOAD TRANSFER SURFACE DIALOG*.

Surface load can be placed by using the method described in chapter 6.4.3.

There are two visibility options: view the surface load or view the distributed load.





Changing the views is possible by clicking on the dedicated icon which can be found among the visibility of graphics symbols setting.

EFZ —
« 🕐 🖡
🕹 🕹 🔻 🔛
- 🔣 🗠 🖄 🗉

There are two possible methods to convert surface load to the selected members:

Converting surface loads to uniform line loads is performed by a meeting the following requirements:

- ▶ the resultant force of all the line loads is the same as for the surface load
- the line loads are *constant* on all the selected members

Oistribute surface load to uniform line loads
Distribute surface load based on structural points
$\hfill \hfill $
Intersection points of loaded bars
$\hfill \hfill $
☐ Intersection points of loaded and unloaded bars not ☐ laying in the surface
Save as default Close Apply



Converting the surface load to line loads based on structural points has the following background:

- (1) the surface load is first converted to concentrated point loads acting on the selected structural points using the Delaunay triangulation technique
- (2) the concentrated loads are then converted to line loads on the selected members meeting with the following requirements:
 - the resultant force of all the line loads is the same as for the surface load
 - the line loads are *linear* on all the selected members
 - the end value of the line loads on the selected structural points for all the selected members are equal

Nethod of surface load distri	ibution 🛛 📉
O Distribute surface load to u	uniform line loads
Oistribute surface load bas	sed on structural points
Endpoints of loaded bars too)	bars (common points of continuous
Intersection points o	of loaded bars
Intersection points o the surface	of loaded and unloaded bars laying in
Intersection points o laying in the surface	of loaded and unloaded bars not
Save as default	Close Apply

The basic working method is the following:

First create the load transfer surface. Select the members which it distributes load to, or accept the default setting which is distribute loads to all planar members. Apply surface loads to the surface at every load case where it is necessary.

6.4.4.1 MODIFYING LOAD TRANSFER SURFACE

Load transfer surface can be modified by selecting it and changing the properties in the property bar. Member selection can be easily modified from the planar members to selected members. It is possible to highlight the previously selected members by clicking on the blue thick.



Ξ	Load transfer surface (1)			
	Name of load transfer s	Load transfer surface		
	Two-dimensional figure	main model, id=2524		
	Member selection	Selected membe 🧹 🔖		
	Visible	Yes		

If selected members option is chosen and then the black arrow is clicked then previously selected members can be removed from the selection or new members can be added.

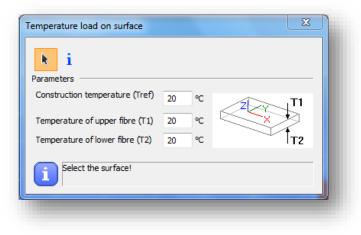
6.4.5 TEMPERATURE LOAD (°C	<u>°</u>

Temperature loads can be defined to members () or to surfaces ().

First choose the type of the temperature load: the temperature can change along the cross section's height or the width. Then add the construction temperature (reference temperature) and the upper and lower temperature of the cross section. The last step is to select the member where you would like to apply the load.

i		k i
meters		Parameters
oe of temperature load		Type of temperature load
Varying along the height of the section	1	Varying along the height of the section
Varying along the width of the section	Z †	Varying along the width of the section
truction temperature (Tref) 20 °C		Construction temperature (Tref) 20 °C
erature of upper fibre (T1) 20 °C	-2 1	Temperature of left fibre (T 1) 20 °C
erature of lower fibre (T2) 20 °C	2	Temperature of right fibre (T2) 20 °C
Select the line for temperature load!		Select the line for temperature load!





6.4.6 PRESCRIBED DISPLACEMENT (

Prescribed displacement can be applied for point supports.

It can be placed according to the **Global** (¹⁰⁰), or the **User coordinate system** (¹¹⁰) and can be given with components or by vector.

visplacement (X, Y, Z) 0 mm 0 mm -10 mm	🤳 i 🔘 🗄 Parameters	\$					
otation (X, Y, Z) 0 ° 0 ° 0 °	with components			🔘 by ve	ctor		
ocadon (X, Y, Z) 0 0 0 0	Displacement (X, Y, Z)	0	mm	0	mm	-10	mm
Select the point for prescribed displacement!	Rotation (X, Y, Z)	0	•	0	۰	0	•
	Select the point	for preso	ribed dis	placemen	t!		

The change of the original member system length can be set and applied as a load on bar members with the *CHANGE IN LENGTH* (dL) function. With this function prestress effect can be modelling.

F

6.4.7 CHANGE IN LENGTH (, TENSION FORCE (



Change in length of bar
i Parameters Change in length (dL) 15 mm Select the member for elongation!

Tension force can be applied on bar members as a load with the **TENSION FORCE** (function.

ension force on bar		×
Parameters		-
Tension force (F)	0 kN	
Select the me	ember for pretension!	

6.5 GLOBAL IMPERFECTIONS

Geometry Structural members	Loads Mass Finite e	element Analysis	Global checks Memi	ber checks Serv	viceability checks
🤑 <table-cell-rows> Dead load</table-cell-rows>	🖂 💠 📶 🗾 ;	🗄 🙂 💐	dL F	1 👯 💨 💧	
		7-	Global	imperfections	

Three types of global imperfection can be applied on the model with the **GLOBAL IMPERFECTION** (F) function. Several numbers of imperfections can be created, but only one can be used for the analysis at the same time. Used imperfection can be set for the analysis at the **ANALYSIS TAB's SET ANALYSIS PARAMETERS** dialog (chapter 8.4).



6.5.1 NOTIONAL LOAD

Initial imperfections can be taken into account by the application of notional loads.

On the basis of the selected load groups, the notional loads will be generated automatically in the selected load combinations. Direction of the notional loads can be defined automatically or can be set for every load combinations.

Multiplication factor for magnitude of notional load (Ni) can be selected from the dropdown menu, or can be type manually. This factor means how much percentage of the selected loads will be act as a notional load. Value can be between 0 and 1.

Pressing the **APPLY** or the **OK** button the notional load will be created.

6.5.2 INITIAL SWAY

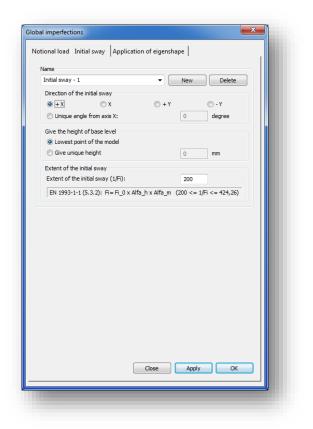
Initial sway can be set for the whole model.

With the **NEW** button several initial sway can be created or can be deleted by the **DELETE** button.

To create an initial sway, the direction, height of base level and the extent of initial sway (1/Fi) have to be given.

Pressing the **APPLY** or the **OK** button the initial sway will be created.

tion	al load Initial sway Appli	cation of eige	enshape	1
/ Ap	ply notional load			
ultic	lication factor for magnitude of	notional load ((Ni)	0,002 -
	dered load groups			
01131				
	Load group			
~	DL			
~	Snow			
	dered load combinations and dir Load combinations	ections	Direction	<u>^</u>
_				^
<u>~</u>	Load combination-1		+X	• E
~ ~	Load combination-2		Automatic	<u>•</u>
~	Load combination-3		Automatic -Y	
_	Load combination-4		-Y Automatic	<u> </u>
	Load combination-5		Automatic	-
	Load combination 6			
~	Load combination-6			
~	Load combination-7		-X	•
~				
~	Load combination-7 Load combination-8		-X Automatic	▼ ▼
~	Load combination-7 Load combination-8 Load combination-9		-X Automatic Automatic	• •
~	Load combination-7 Load combination-8 Load combination-9 Load combination-10		-X Automatic Automatic Automatic	- - -
	Load combination-7 Load combination-8 Load combination-9 Load combination-10 Load combination-11		-X Automatic Automatic Automatic Automatic	• • •
~	Load combination-7 Load combination-8 Load combination-9 Load combination-10 Load combination-11 Load combination-12		-X Automatic Automatic Automatic Automatic Automatic	• • •
~	Load combination-7 Load combination-8 Load combination-9 Load combination-10 Load combination-11 Load combination-12 Load combination-13		-X Automatic Automatic Automatic Automatic Automatic Automatic	V V V V V V V V V V V
~	Load combination-7 Load combination-8 Load combination-9 Load combination-10 Load combination-11 Load combination-13 Load combination-14	Close	-X Automatic Automatic Automatic Automatic Automatic Automatic Automatic	V V V V V V V V V V V V





6.5.3 APPLICATION OF EIGENSHAPE

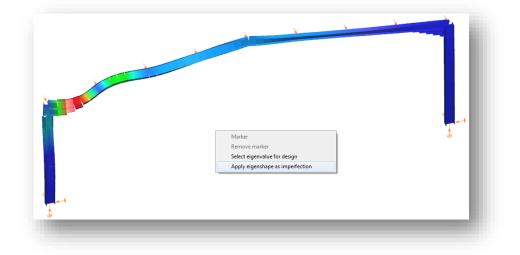
Previously calculated buckling modes can be applied universally as a geometric imperfection for all of the load combinations. Even more buckling modes, as an imperfection case, can be easily superposed as one imperfection.

6.5.3.1 APPLY EIGENSHAPES

Previously performed buckling analysis is needed to apply an eigenshape as an imperfection.

On the *ANALYSIS* tab's Buckling result view either calculated buckling mode can set as imperfection.

After selecting the proper eigenshape, and by clicking in the graphical area with right mouse button and choosing the *Apply eigenshapes as an imperfection* option, the parameters of the imperfection can set.



On the setting dialog the following parameters can be set:

- Imperfection group: any eigenshape will belong to an imperfection group, which can be selected from the list. One imperfection group can contain more than one eigenshapes. If more than one eigenshapes are in an imperfection group, than the effect of the eigenshapes will be added together during the application. With the NEW GROUP button a new group can be created.
- ▶ *Name of imperfection case:* unique name can be added for the imperfection
- *Type of imperfection:*
 - Overall: imperfection case will be used for all of the load combination
 - Local: imperfection will be used just for the origin load combination which occurred the selected buckling mode
- Modelling of imperfection:



- Complete: buckling mode will be applied on the model with all of the freedom (node displacement, rotation, warping)
- Nodal disp.: only the nodal displacement will be applied as imperfection
- Sign of amplitude:
 - Automatic: direction of the imperfection will be set properly for each of the load combinations; in case of normal way selection, for local imperfection type, only the manual option is available
 - Manual: direction of the imperfection will be accordingly to the sign of the given amplitude value
- **•** Type of amplitude:
 - *mm value:* maximum displacement value of the eigenshape can be given in mm. The other displacement values will be commensurable to the given amplitude.
 - *multiplication factor:* the calculated displacement values can be multiplied with this factor
- ▶ Value of amplitude: value of the previously selected amplitude type has to be given

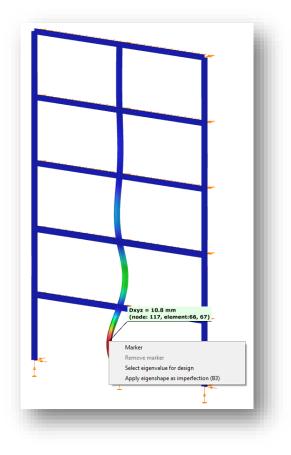
Apply eigenshape as imperf	ection	×
Imperfection group		
Global imperfection		~
Name of imperfection case	Imperfection case_2	
Type of imperfection	Overall	~
Modelling of imperfection	Complete	\sim
Sign of amplitude	Automatic	\sim
Type of amplitude	mm value	\sim
Value of amplitude	1	
Type of imperfection - Overall: In all load combinat - Local: In current load combin		
Modelling of imperfection - Nodal disp.: Nodal displace	ments only	
Sign of amplitude - Manual: The amplitude valu negative	e can be positive and	
New group	Cancel OK	

Pressing the **OK** button the eigenshape is applied as an imperfection with the set parameters.



6.5.3.2 AUTOMATIC AMPLITUDE CALCULATION

Applying any imperfection it is always a problem how to define the proper amplitude to have safe but economic results. The Eurocode EN 1993-1-1 gives some limited guidance for certain cases, based on these rules the program offers an automatic calculation of the imperfection amplitude. Since these rules are strongly connected to member parameters (cross-section type, shape, properties, buckling curve etc.) the first task is the selection of a reference member which would be the suitable base for the amplitude calculations.



This can be done by right click on the selected member and then choose the *Apply eigenshape as imperfection* where as a feedback the name of the selected member is also appearing (B3). The selected member should be the dominant one in the certain buckling mode. For the automatic amplitude definition the three dots (....) button next to the *Value of amplitude* input field should be pressed on the dialogue presented in the previous chapter.

There are two choices for the amplitude calculation:

Equivalent initial bow imperfection

This option is based on the table EN 1993-1-1 5.3.2 (3)b where initial bow amplitude ratios are defined depending on the corresponding buckling curve of the selected member. In order to evaluate the proper amplitude the equivalent member length



should also be specified (which is not necessary equal to the system length of the selected member...) and the calculated value is displayed in mm. For lateral torsional buckling problem an additional multiplication factor (k) can be defined for the amplitude. It is important to note that these values are officially valid only for simply supported straight, prismatic members with uniform compression (or bending), for other situation the user has to consider the special support, geometry or loading effects when defining the initial bow. Finally the calculated value (e₀) is accepted by the *OK* button which automatically gets into the *Value of amplitude* input field.

Equivalent imperfection based on the elastic critical buckling shape

This option is based on the rules of EN 1993-1-15.3.2 (11). This method calculates fully automatically the amplitude value based on the selected axis of buckling with the following steps:

- 1. Calculation of the internal forces at the FE nodes of the member from the deformed shape equal to the selected buckling mode
- 2. Determination of the critical position where the relevant bending moment (M_{γ} or M_z calculated in 1.) is the highest
- 3. Read the following information at the critical position:
 - i. Normal force value [N_{Ed}] from the first order analysis (without imperfection)
 this value must be compression

ii. Relevant bending moment [My or M_z later displayed as M^{II} denoted as $EI\eta_{cr,max}$ in the standard]

iii. Cross section properties [class for compression, yield stress, area, relevant section modulus (W_y or W_z), relevant buckling curve]

Finally the calculated value ($\eta_{init,max}$) is accepted by the **OK** button which automatically gets into the 'Value of amplitude' input field. This calculation is only applicable to buckling problems of a dominantly compressed member but with arbitrary support condition.

S CONSTEEL	
	12.0 USER MANUAL

		 Equivalent initial bow im 		
alent initial bow imperfection		Used part of standard	: EN 1993-1-1 5.3.2 (3) b)	
part of standard: EN 1993-1-1 5.3.2 (Equivalent member len	ngth L _{eq} [m]	5
valent member length L	. _{oq} [m] 5	Design value for initial	bow amplitude Leg/	250 -
gn value for initial bow amplitude	L _{eq} / 250	Apply for lateral to		
pply for lateral torsional buckling				0,5
	k 0,5			
	o [mm] 20		e ₀ [mm]	20
			1 265,0 kN 100,9 kNm 1,598	
		CLuit CLor	5,080	
		λ	0,561	
		α	0,340	
		χ e0	0,856 9,79 mm	
		MI	18,6 kNm	
		ηinit.max	23,10 mm	
				*
				~

6.5.3.3 MANAGING EIGENSHAPES

Previously created imperfection groups and imperfection cases (applied eigenshapes) can be managed on the **GLOBAL IMPERFECTIONS** dialog's **APPLICATION OF EIGENSHAPE** tab.

By the first dropdown menu, imperfection groups can be switched, and whole imperfection group can be deleted with the **Delete** button.

In the middle part of the dialog can be seen the imperfection cases (applied eigenshapes) in the selected imperfection group. With the checkboxes in the first column each imperfection cases can be turn off and on. In the following columns, all of the previously setted parameters can checked and modified.

With the **DELETE ROW** button the selected imperfection cases can be deleted from the imperfection group.



Glob	al imperfection									 Delete 	•
т	Name	Load combination	Buckling eigenshapes	Member Ty	ype of amp	Valu	Туре	Modelli	ng	Sign	
~	Imperfection case	Load combination-101	ıckling eigenshapes - 1	B2 m	nm value 💌	1,000				Automatic	
~	Imperfection cas	Load combination-101	ıckling eigenshapes - 2	- m	nm value 🗵	1,000	Overall	Comple	te 🗵	Automatic	-
											L
											l
											l
											l
											l
											l
											l
											l
											l



6.6 MOVING LOAD

EuroCode based or user defined crane and train load can be created and placed on the model manually or by the loading of an influence graph.

Geo	metry Structural members	Loads Mass	Finite element	Analysis	Global checks	Member checks Serviceability check	ks
ø	↓++ Dead load	× 💠 📶	🗾 鐞 🕑	°c ≛∓	dL F	🏂 🎇 🌏	
H		/		7-	•	ta 🕂 🏷 🌮	

6.6.1 MOVING LOAD PATH (

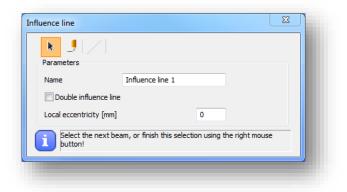
To place a moving load on the model or to calculate influence graph, as a first step it is necessary to create **MOVING LOAD PATH** on those members where the load will be placed or moved.

Members have to be lain in one line and have to be connected.

Moving load path can be defined with selection (

In case of selection, the selection function (**I**) has to be selected and all of the members have to selected with the left mouse where the **MOVING LOAD PATH** should be applied. At the end the selection function has to be closed by clicking with the right mouse button.

In case of drawing, the drawing function () has to be selected and the start and the end point of the drawing line has to be picked. **MOVING LOAD PATH** will be applied on all of members which are lying under the drawing line.



Double moving load path can be created by clicking in the check-box.



To run *Influence graph* analysis, it is necessary to set the *INFLUENCE GRAPH* parameters.

nfluence graph	X
Calculation Name	
Influence gr	jraph 1
Influence gr	jraph 2
Parameters	
Influence line	Influence line 1
Place of investigation	
 Cross section 	B359 7388
Structural member	
Submodel	Braced part 🔍
Type of influence graph	
 Displacement 	e _x 👻
Beam force	V ₂
Reaction force	Rx
Beam stress	GN 🖌
Direction of unit load	
📝 -Y	✓ -Z □ -X
-+X	✓ +Z ✓ +Y
New De	Delete Cancel Apply

As a first step one of the previously created influence line has to be selected from the dropdown menu (#1) where the unit load will move. If there is not any influence line, then with the three dot button (....), a new one can be created.



12.0 USER MANUAL

As a next step, the place of the investigation (#2) has to be selected with black arrow button (

). In the current version of *ConSteel*, a cross section of an arbitrary member can be selected. Not just those members can be selected that are part of the selected influence line.

Three types of influence graph can be analyzed (#3):

- Displacement
 - Displacement in local x, y and z axis
 - Rotation around local x, y and z axis
- Internal force
 - Major (My) and minor (Mz) axis bending moment
 - Normal force (N)
 - Major (V_y) and minor (V_z) axis shear
 - Torsion (T)
 - Bimoment (B)
- Reaction force
 - Reaction force in the global X, Y and Z axis
 - Reaction moment around the global X, Y and Z axis

As a last step the direction of the unit load has to be chosen. Simultaneously more direction can be chosen, but separate graph will belong to for each directions.

By clicking on the *APPLY* button, the influence graph setting is created.

With the *NEW* button another influence graph setting can be created.

6.6.2.1 MANAGING INFLUENCE GRAPHS

Previously created influence graphs can be managed in the table part of the dialog (#4). All of the previously created influence graphs can be seen and selected.

After the selection, all parameters can be modified with the **APPLY** button.

The selected influence graph(s) can be deleted by pressing **D**ELETE button.

With the checkboxes in the *Calculation* column, influence graph(s) can be switched on/off for calculation.

6.6.2.2 ANALYZE INFLUENCE GRAPH

To run Influence graph analysis see **CHAPTER 8 STRUCTURAL ANALYSIS** below!



Three types of overhead traveling crane load can be defined:



- Standard: fully EN 1991-3 based
- Standard load based: EN 1991-3 standard defined wheel loads have to set. Group of loads are created by ConSteel
- User defined: direct wheel loads have to be set for each wheels

				_
ype of crane	Standard	Standard load based	O User defined	

6.6.3.1 STANDARD

For the fully EN 1993-3 based standard crane load, the geometrical parameters (#1), loadings (#2), and the driving properties (#3) have to be set. The wheel loads are automatically calculated.

Name Cran	e load 1											•
Туре	of crane		۲	Standard	ł	0) Standar	d load ba	sed	🔘 Use	r defined	
Geome	etrical pro	perties										
Crane	span - L				þ1	,3 m	- F	<u> </u>			77	
Minima	l distance	e of the t	rolley - em	in	0	m	Lb				Ξ.	
u mba	er of axes		4 -				**	e _{min}	L			
	1 2		<u> </u>									
	0,0 1,2											
Loadin	ig proper	ties									_	
Self we	eight of c	rane brid	ge - Qci		70 kN	1		Con	25	ALL P		
Self w	eight of t	rolley - Q	202		15 kN	1 18	-		3	(1)		
Elevat	ed weight	t-Qh			180 kN	10		Æ				
D	ynamic fa	ctors					-	X a _h				
Driving) properti	ies										
Numb	er of sing	le wheel	drives		2							
Whee	l drive		() C	entral		(Crane sys	tem			CFF	-
			I (ndividual			Friction fa	ctor			0,2	
Guidin	g device t	type) F	anged w	heel		🗇 Gudie p	in				
	nce of pin ion - a _{sxt}		he driving				tance of p					
			e pin and	the rail	10 r		ng the driv litional gap				5	mm
- x	st gap be	weena	ic pin and	dic rui	10 n		and the ra			in or are	5	
Wieel	loads						1. Loado	case				•
	1, 1	1, 2	2, 3	2, 4	3, 5	3, 6	4, 7	4, 8				
F ₈	2,338	2,338	0,000	0,000	0,000	0,000	0,000	0,000				
-	1,423	9,352	0,000	0,000	0,000	0,000	-1,423	-9,352				
Fz	-63,250	-9,625	-63,250	-9,625	-63,250	-9,625	-63,250	-9,625				
	New		Delete		Place lo	bad			Cancel		App	oly



The following geometrical properties (#1) can be set:

- Crane span L [m]: span of the crane bridge
- Minimal distance of the trolley e_{min} [m]: minimum distance between the trolley and the crane girder
- Number of axes: Numbers of the axes of the crane bridge have to be set from the dropdown menu. Minimum number of the axis is 2, maximum is 10. Distances between the axes also have to be set.

In case of loading properties (#2), the self weight of the crane bridge (Q_{c1}) and the trolley (Q_{c2}) and the elevated weight (Q_h) have to be set. By pressing the **DYNAMIC FACTORS** button, the default value of the factors can be seen. Factors can be modified manually.

The following driving properties (#3) have to be set:

- Wheel drive: number of the driven wheels and the system of the wheel drive (central or individual)
- Crane system: to select the proper EuroCode care system, please see the definition of the crane systems in the relevant part of the EN 1991-3
- Friction factor: friction factor between the rail and the wheels
- Guiding device: two types of guiding device can be chosen, flanged wheels and guide pins. According to the selection, the necessary geometrical parameters of the guiding device have to be set

The automatically calculated wheel loads for each load case can be seen in the table (#4). The load cases can be changed with the dropdown menu.

In the name of the wheels, the first prefix means the number of the axis; the second means the number of the wheel.



6.6.3.2 STANDARD LOAD BASED

For the standard load based crane load, the numbers of the axis and the standard wheel loads have to be set.

							•
Type of crane	0) Standard	State	indard load ba	sed 🔘	User defined	
Geometrical prop							
umber of axes	2	•					
Axis 1	2						
i [m] 0,0	2,5						
Vheel loads							
21,1 1,2	2, 3 2, 4						
Qc 0,000 0,00	0,000 0,00	00					
Qh 0,000 0,00	0,000 0,00	00					
Ht 0,000 0,00							
lı 0,000 0,00							
Hs. 0,000 0,00							
Hs. 0,000 0,00							
Ht3 0,000 0,00	0 0,000 0,00	00					
Dynamic fact	ors						
Dynamic fact	ors						
	ors		1.	Loadcase			r
		2, 4	1.	Loadcase			
Vheel loads	2 2, 3	2, 4	1.	Loadcase			•
Interview Interview 1, 1 1, 1 x 0,000 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0	2 2, 3 000 0,000 000 0,000	0,000 0,000	<u>l.</u>	Loadcase			
Interview Interview 1, 1 1, 1 x 0,000 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0	2 2, 3 000 0,000	0,000 0,000	<u>[1.</u>	Loadcase			
Interview Interview 1, 1 1, 1 x 0,000 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0	2 2, 3 000 0,000 000 0,000	0,000 0,000	1.	Loadcase			

In the geometrical properties field (#1) the number of the axes of the crane bridge have to be set from the dropdown menu. Minimum number of the axis is 2, maximum is 10. Distances between the axes also have to be set.

In the second field (#2) the EuroCode based wheel loads have to be set for each wheels. By pressing the **DYNAMIC FACTORS** button, the default value of the factors can be seen. Factors can be modified manually.

The automatically calculated wheel loads for each load case can be seen in the table (#3). The load cases can be changed with the dropdown menu.

S CONSTEEL

In the name of the wheels, the first prefix means the number of the axis; the second means the number of the wheel.

6.6.3.3 USER DEFINED

For the user defined crane load the following parameters have to be set:

- The numbers of the axes of the crane bridge have to be set from the dropdown menu. Minimum number of the axis is 2, maximum is 10. Distances between the axes also have to be set.
- ► Three directions (F_x, F_y and F_z) of the wheel loads have to be set for each wheels

Nam											
Cra	ane load 1										-
Тур	e of crane		\odot	Standard	ł	0) Standar	d load ba	sed 🤅	🖲 User de	fined
Geor	metrical pro	operties									
Numb	ber of axe	s	2 -	•]							
Axis	1	2									
		0,4		1							
	n] 0,0 eel loads	0,4	2, 3	2, 4	3, 5	3, 6	4, 7	4, 8			_
Whe	el loads		2, 3 0,000	2, 4 0,000	3, 5 0,000	3, 6 0,000	4, 7 0,000	4, 8 0,000			
Whe F _x	el loads	1, 2									
Whe Fx Fy	el loads 1, 1 2,338	1, 2 2,338 9,352	0,000	0,000 0,000	0,000	0,000 0,000	0,000	0,000			
ai [m Whe Fx Fy Fz	el loads 1, 1 2,338 1,423	1, 2 2,338 9,352	0,000 0,000	0,000 0,000	0,000 0,000	0,000 0,000	0,000 -1,423	0,000 -9,352	i	i	
Whe Fx Fy	el loads 1, 1 2,338 1,423	1, 2 2,338 9,352	0,000 0,000	0,000 0,000	0,000 0,000	0,000 0,000	0,000 -1,423	0,000 -9,352	i		

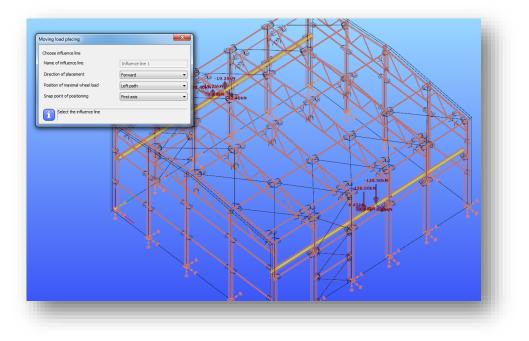
6.6.3.4 PLACING CRANE LOAD

There is two ways to place crane load on the model, but before placing with the **APPLY** button the defined crane load has to be saved.

Placing crane load manually

By pressing the **PLACE LOAD** button on the bottom of the **CRANE LOAD** dialog, the selected loadcase can be place manually on the model.





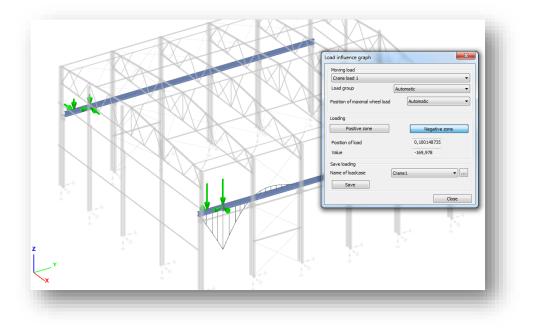
At first an influence line has to be chosen with the left mouse button. After the loads have to be placed along selected influence line by pressing the left mouse button.

Direction of the placement, position of the maximal wheel load and the snap point of the positioning can be selected from the dropdown menus.

Placing crane load by loading influence graph

Crane load can be placed on the model by loading an influence graph.

To analyze an influence graph please see Chapter 6.6.2 and Chapter 8.





To place a crane load by loading an influence graph, please select the *Influence graph* analysis type from the first dropdown menu on the *ANALYSIS* tab and click with the right mouse button on the graphical area and select *LOAD INFLUENCE* line from the appeared menu.

Moving load	•
Load group	Automatic
Position of maximal wheel load	Automatic 💌
oading	
Positive zone	Negative zone
Position of load	0,100148735
Value	-169,978
Save loading	1
Nace of loadcase Cr	rane1 🔻
Save	
	Close

In the *Moving load* field (#1) the loading properties can be set:

- From the first dropdown menu, one of the previously created crane load can be selected to place
- Load group: in case of standard or standard load base crane, EuroCode load groups are automatically created. Influence graph can be loaded by the selected load groups or by the dominant if the Automatic function is selected from the dropdown menu
- Position of maximal wheel load: vertical wheel loads depend on the position of the trolley. Maximum vertical wheel load could be on the left or the right path, or the dominant position can be automatically detected if the Automatic function is selected from the dropdown menu

With the **POSITIVE ZONE** and **NEGATIVE ZONE** buttons in the Loading field (#2), the influence graph can be loaded. If the **POSITIVE ZONE** button is used then the loading is give the maximum positive value in the section, if the **NEGATIVE ZONE** button is used then the loading is give the maximum negative value in the section. The position of the load and the current value in the section are appeared.

The placed load can be saved to a load case by the function of the Save loading field (#3). With the dropdown menu, one of the previously created load case can be select to save or with the three dot button a new one can be created. With the **SAVE** button the placed load can be saved to the selected load case.





Two types of train load can be created and placed on the model.

Standard

From the dropdown menu three types of EuroCode defined train load can be selected. In the tables the properties of the selected one can be seen.

Standard LM71 User bined SW/0 SW/2 Imm Fx [kN] Fy [kN] Fz [kN] Position of load [m] 0,00 0,00 -250,00 0,000 0,00 0,00 -250,00 3,200	/								
Standard LM71 User defined SW/2 Concentrated forces Ium. Fg [kN] Fg [kN] Fg [kN] Position of load [m] 0,00 0,00 -250,00 0,000 0,00 0,00 -250,00 1,600 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800 + - Member forces Num. gat [kN/ gg [kN/.	ain loa	d 1							-
User defined Imm 71 SW/0 SW/2 Concentrated forces Fz [kN] Fz [kN] Position of load [m] 0,00 0,00 -250,00 0,000 0,00 0,00 -250,00 1,600 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800	us								
User defined SW/0 SW/2 Concentrated forces Fig [kN] Fig [kN] Position of load [m] 0,00 0,00 -250,00 0,000 0,00 0,00 -250,00 1,600 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800) Stand	lard				•]		
Concentrated forces Lum. F ₂ [kN] F ₂ [kN] Position of load [m] 0,00 0,00 -250,00 0,000 0,00 0,00 -250,00 1,600 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800) User o	defined	SW/0				1		
$\begin{tabular}{ c c c c c c c c c c c c c c c c c c c$	Con	centrated for			_		1		
0,00 0,00 -250,00 0,000 0,00 0,00 -250,00 1,500 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800 ***********************************	_			E. ILNI		E. ILNI		Position of I	and [m]
0,00 0,00 -250,00 1,600 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800									
0,00 0,00 -250,00 3,200 0,00 0,00 -250,00 4,800									
+ - 7 Member forces Num. q _{x1} [kN/ q _{x2} [kN/ q _{y1} [kN/ q _{y2} [kN/ q _{x1} [kN/ q _{x2} [kN/ Position Position 0,00 0,00 0,00 0,00 -80,00 -80,00 -0,80,00 -100,000 2 0,00 0,00 0,00 -80,00 -80,00 5,600 104,800		-							
Member forces Num. q _{s1} [kN/ q _{s2} [kN/ q _{s1} [kN/ q _{s2} [kN/ q _{s1} [kN/ position <	Ļ	0,00						4,800	
			·						
					-10.05M	-18.001N			

User defined

In case of *User defined*, the concentrate and line load tables are editable. With the plus (+) and minus (-) buttons a new row can be added to the table and the selected row can be removed from the table.

In case of concentrate load the three directions and the position of the load can be set.



In case of line load the intensity of load can be set on the start and the end point in the three directions, and the position of the end points can be set.

The placing of the train load is the same as for the crane load. Please see Chapter 6.6.3.4.

6.7 METEOROLOGICAL LOAD GENERATOR

All of the meteorological load generator related functions (METEOROLOGICAL SURFACE

METEOROLOGICAL LOAD GENERATOR (Can be found on the LOADS tab.

Meteorological load generator can generate wind and snow load for the most common type of structures according to the following standards:

► Wind

- EuroCode 1 (EN 1991-4)
- Spanish standard (SE-AE)
- Snow
 - EuroCode 1 (EN 1991-3)



HEIGHTS OF THE STRUCTURE (h) FOR GENERATING WIND LOADS ARE CALCULATED FROM THE GLOBAL z=0 LEVEL. THEREFORE THE STRUCTURAL MODEL HAS TO BE PLACED TO ITS REAL HEIGHT IN THE MODELLING AREA MEASURED FROM THE GLOBAL z=0 LEVEL. FOR EXAMPLE LOWEST EDGE OF A WINDWARD WALL CANNOT BE PLACED BELOW THE GLOBAL z=0 LEVEL, OR IF THE MODEL IS A ROOF OF A BUILDING, THEN THE ROOF MUST BE PLACED IN THE REAL HEIGHT MEASURED FROM THE GLOBAL z=0 LEVEL.

6.7.1 LOAD GROUPS AND LOAD CASES

Load groups and load cases are automatically created during the generation process according to the placed meteorological surfaces and the selected wind directions.

Safety factors and coefficient factors are loaded from the selected national NA.



U Persistent	Load group (Meteoro							
↓ Dead load ↓ Live	Name	Snow						
↓ Live load ↓ Snow		Standard valu	es of factors					
Snow load	Safety factor		γ ₀ 1,5					
	Combination	actor	Ψ ₀ 0,7					
	Factor of freq	uent load level	Ψ ₁ 0,5					
	Factor of quas	i permanent load level	Ψ ₂ 0,3					
	The load cases ca	an be applied along with e	each other					
	New load grou	ıp	Delete					
	Load case							
	Name	Snow load						
	Load group	Snow	~					
	Color							
	The actions can be favorable in the combination							
	Can't be the main	in permanent and variab	le combination					
	Can't be the main	in accidental combination	ı					
	New load cas	е Сору	Delete					
	Load case including s	elf weight						
	Dead load		~					
	Clo	se Apply	ОК					

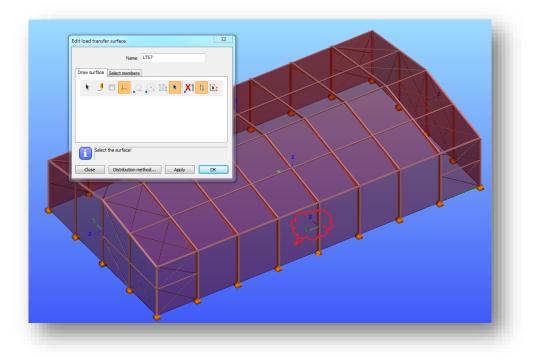
6.7.2 METEOROLOGICAL SURFACES

The first step to generate meteorological load is to place meteorological surfaces on the model. Meteorological surfaces contain all of the information and parameters what are necessary for the load generation.

Meteorological surfaces can be placed on any load transfer surface. It is advised to cover all meteorological load attacked sides of the model with load transfer surfaces.

It is important to check the local axes directions of the load transfer surfaces. All z (blue) axes must point out of the building. If modification is necessary then it is recommended to use the *invert z axis* tool on the *EDIT LOAD TRANSFER SURFACE* dialog (see image below).





Before placing the meteorological surfaces, all of the necessary parameters have to be set on

the *METEOROLOGICAL SURFACE* () dialog.

<u> </u>
Wall
WindVelocityPressure 1 🔹
WindInternalPressure 1 🔹
WindFriction 1
WindGlobal 1 💌
SnowParameter 1 💌



6.7.2.1 WIND SURFACE

Velocity pressure

lame WindVelocityPressu	re 1 🔹 📈	Velocity pressure depending on refere	ence height			
errain category II		qp [N/m2] 1500,0]				_
Roughness length - z ₀ [m]		ļ				
Minimum height - z _{min} [m]	2,0	1000,0				
arameters Зас wind velocity - vь [m/s]	20,06		Z= 32,00 m qp= 789,72 N/m2			
Air density - ρ [kg/m ³]	1,25	500,0				_
Drography coefficient - co(z)	1	F I				
urbulence factor - kı	1	0,0 10,0 20,0	30,0 40,0 50,0	60,0 70,0	80,0 90,0	100,0
New Delete	Apply OK	0,0 10,0 20,0	50,0 40,0 50,0	00,0 10,0	00,0 00,0	Z [m]
Delete						

For the velocity pressure the following two parameters have to be set:

Terrain category (#1)

From the dropdown menu the proper category can be selected or by clicking the three dot button ($\overline{$) the standard definition of the categories can be seen.

Roughness length (z_0) and the minimum height (z_{min}) are loaded from the selected national NA.

Parameters (#2)

All of the parameters are loaded from the selected national NA, only the basic wind velocity (v_b) has to be set. By clicking the three dot button (\dots) the fundamental value $(v_{b,0})$ of wind velocity also can set manual or by using one of the implemented win map. In this case the basic wind velocity pressure is automatically calculated.









VELOCITY PRESSURE DIALOG AND THE NECESSARY PARAMETERS COULD BE DIFFERENT ACCORDING TO THE SELECTED EUROCODE NATIONAL ANNEX

Internal pressure

Internal pressure can be set for the generation:

- Pressure could be different in all wind directions
- Pressure could be equal in all wind directions
- Or, pressure could be equal in couples

Name	WindIntern	alPressure 1 🔹
Wind	All direction	s are different 🔹
Internal pressu		
	Case 1	Case 2
Cpi,0	+0,200	▼0,300 ▼
Cpi,90	-0,300	▼ +0,200 ▼
Cpi, 180	+0,200	▼ +0,200 ▼
Cpi,270	-0,300	▼ +0,200 ▼
Use alternat	tive internal p	ressure factors
New	Delete	Apply OK



The value can set manually, or the extreme values can be selected from the dropdown menus, or by clicking the three dot button (....) one of the EuroCode internal pressure calculator

With the checkbox (#1) alternative internal pressure can be set in all wind direction.

Wind friction

method can be used.

For wind friction, just the friction coefficient has to be set by selecting one of the national NA defined category or selecting the user defined.

Name	WindFriction	11		-
Wind friction				
🔘 Smooth (i.e	e. steel, smoot	h concrete)		
🔘 Rough (i.e	. rough concre	te, tar-boards)	
Very rough	ı (i.e. ripples, r	ibs, folds)		
🔘 User defin	ed			
Friction coeffi	cient - C _{fr}		0,0400	
New	Delete	Apply	ОК	

Geometrical parameters

The following geometrical parameters have to be set:

- Direction of the basic wind in global coordinate system. The basic wind direction is the 0° wind direction.
- Building dimensions to the basic wind direction
- Loaded area (A) of the wind load. If the loaded area is equal or bigger than 10m² then ConSteel use c_{pe10} in the generation. If the area is equal or smaller than 1m² then ConSteel use c_{pe1} in the generation. Between the two values ConSteel interpolates between c_{pe10} and c_{pe1} acc. to the EuroCode rule.
- Ground level. ConSteel measures the height of the structure from the global Z=0 level for the wind load generation. Therefore the structure base level is not equal with the ground level, but the structure is placed on the global Z=0 level in ConSteel, the difference has to be set manually in the ground level field.



Vindload generation geometric parameters	WindGlobal 1	
Directions of basic wind direction (Ξ=0°) in global coor		+X -
Building dimensions to the basic wind direction		
parallel - do [m]		21,4
perpendicular - b o [m] Loaded area - A [m ²]		23,75
Ground level - Z [m]		0
	Apply	ОК

6.7.2.2 SNOW SURFACE

On the Snow load parameter dialog, the following parameters have to be set:

- Snow load zone (if it is necessary according to the selected NA)
- Site altitude (if it is necessary according to the selected NA)
- ▶ Wind (C_e) and thermal coefficient (C_t)
- Coefficient for exceptional snow load

The basic values are loaded from the selected national NA.

Name SnowParameter 1	•	Sk [kN/m2]	
Parameters]		
Snow load zone 1. Zor	ne 🔻	15,0	
Site altitude - A [m]	0		
Characteristic snow load - s _k [kN/m ²]	1,25		
Snow load		10,0	
Wind coefficient - C e	1,0 🔻		
Thermal coefficient - Ct	1		
Snow load - S [kN/m ²]	1,25	5,0 Z= 1200 09 m Sk= 3 25 kN/m2	
Exceptional snow load			
Coefficient for exceptional snow load - C	2		
Exceptional snow load - S_{Ad} [kN/m ²]	2,5	0,0 400,0 800,0 1200,0 1600,0 2000,0 2400,0 2800,0 3200,0 3600,0 4000,0	
New Delete Apply	ОК	Z [m]	1

6.7.2.3 PLACING METEOROLOGICAL SURFACE

After all of the necessary parameters were set, meteorological surfaces can be placed on the 3D model.

From the dropdown menu (#1), the relevant standard surface category has to be chosen and a load transfer surface has to be selected with the left button of the mouse.



The following symbol will be appeared on the load transfer surface, after the successful placement:

- ► Wind surface:
- Snow surface: (just on roofs)

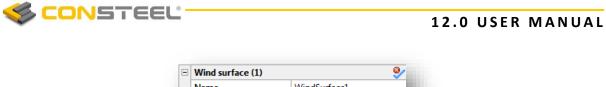
Spandard surface category	Wall
Wind surface	
Velocity pressure	WindVelocityPressure 1 🔹 🔍
Internal pressure	WindInternalPressure 1 🔹
Wind friction	WindFriction 1
Geometric parameters	WindGlobal 1 🔹
Z Snow surface	
Snow load parameter	SnowParameter 1
i Select the surface!	

The following standard categories can be selected from the menu (#1):

- Wall
- Flat roof (sharp eaves)
- Flat roof (with parapets)
- Flat roof (curved eaves)
- Flat roof (mansard eaves)
- Monopitch roof
- Duopitch roof

In case of duopitch roof not just one load transfer surface, but two connected load transfer surfaces have to be selected.

By selecting the symbol, meteorological load surface can be modified in the **OBJECT PROPERTIES** window.



]	Wind surface (1)		8
	Name	WindSurface1	
	Surface type	Duopitch roof	
	Define zones	Standard	
	Geometric parameters	WindGlobal 1	
	Internal pressure	WindInternalPressure 1	
	Wind friction	WindFriction 1	
	Velocity pressure [kN/m/	WindVelocityPressure 1	
-	Snow surface (1)		9/
	Name	SnowSurface1	
	Surface type	Duopitch roof	
	Geometric parameters	SnowParameter 1	

In case of Wind surface, by clicking the three dots button in the Define zone row, the **WIND ZONE VIEWER** dialog open, where the calculated wind zones and c_{pe} values can be seen and checked.

	t para	meter								
oaded a	area - /	4 [m²]		10						
luilding h	neight	- h [m]		12,05	Pit	ch ang	le α [°]		5,	71059
Wind dire	ection	depende	nt							
Vind dire	ection [ଂ] 🛛		·	b [m]	23,75		e [m] 23	3,75
asic win	d direc	tion 0			d [m]	21,4		h/	d 0,	563084
Schema	atic zor	ne figure	with si	zes						
		-			d				-	
	€/4	F								1
	¥	· ·								
	7	G		н		J		1		b
	7	Ŭ				Ŭ		•		ī
_	4	F								
	e/4 _▼	F								¥
		- −e/1	0-►		-	-e/10				
	d Cpe t	table EN	1991-1	-4:2007	Table 7	.4				
Standar	- C.									
	F,10	F,1	G,10	G,1	H,10	H,1	I,10	I,1	J,10	J,1
α	-	F,1 -2,5	G,10 -1,2	G,1 -2,0	H,10 -0,6	H,1 -1,2	I,10 -0,6	I,1 -0,6	J,10 0,2	J,1 0,2
α 5,000	F,10									
α. 5,000 5,000	F,10 -1,7	-2,5	-1,2	-2,0	-0,6	-1,2	-0,6	-0,6	0,2	0,2
α 5,000 5,000 15,000	F,10 -1,7 0,0	-2,5 0,0	-1,2 0,0	-2,0 0,0	-0,6 0,0	-1,2 0,0	-0,6 -0,6	-0,6 -0,6	0,2 -0,6	0,2 -0,6
α 5,000 5,000 15,000 15,000 Actual C	F,10 -1,7 0,0 -0,9 0,2	-2,5 0,0 -2,0 0,2	-1,2 0,0 -0,8	-2,0 0,0 -1,5	-0,6 0,0 -0,3	-1,2 0,0 -0,3	-0,6 -0,6 -0,4	-0,6 -0,6 -0,4	0,2 -0,6 -1,0	0,2 -0,6 -1,5
α 5,000 5,000 15,000 15,000 Actual C	F,10 -1,7 0,0 -0,9 0,2	-2,5 0,0 -2,0 0,2 Jes	-1,2 0,0 -0,8	-2,0 0,0 -1,5	-0,6 0,0 -0,3	-1,2 0,0 -0,3	-0,6 -0,6 -0,4	-0,6 -0,6 -0,4	0,2 -0,6 -1,0	0,2 -0,6 -1,5
α 5,000 5,000 15,000 15,000 Actual C α	F,10 -1,7 0,0 -0,9 0,2 pe valu	-2,5 0,0 -2,0 0,2 Jes	-1,2 0,0 -0,8 0,2	-2,0 0,0 -1,5	-0,6 0,0 -0,3 0,2	-1,2 0,0 -0,3 0,2	-0,6 -0,6 -0,4 0,0	-0,6 -0,6 -0,4 0,0	0,2 -0,6 -1,0 0,0	0,2 -0,6 -1,5 0,0
α. 5,000 5,000 15,000 15,000	F,10 -1,7 0,0 -0,9 0,2 pe valu	-2,5 0,0 -2,0 0,2 Jes	-1,2 0,0 -0,8 0,2	-2,0 0,0 -1,5 0,2	-0,6 0,0 -0,3 0,2 H	-1,2 0,0 -0,3 0,2	-0,6 -0,6 -0,4 0,0	-0,6 -0,6 -0,4 0,0	0,2 -0,6 -1,0 0,0	0,2 -0,6 -1,5 0,0
α 5,000 15,000 15,000 Actual C α 5,711	F,10 -1,7 0,0 -0,9 0,2 pe valu	-2,5 0,0 -2,0 0,2 ues	-1,2 0,0 -0,8 0,2 G -1,:	-2,0 0,0 -1,5 0,2	-0,6 0,0 -0,3 0,2 H -0,57	-1,2 0,0 -0,3 0,2	-0,6 -0,6 -0,4 0,0 I -0,58	-0,6 -0,6 -0,4 0,0	0,2 -0,6 -1,0 0,0 J -0,62	0,2 -0,6 -1,5 0,0



6.7.3 METEOROLOGICAL LOAD GENERATOR

Meteorological load generator function can be found on the *LOADS* tab.

Geometry Structural members	Loads Mass	Finite element	Analysis Glol	bal checks	Member checks	Serviceability checks
🔑 <table-cell-rows> Dead load</table-cell-rows>	 	🔟 🦊 🕑	C Å∓ dL	+ =++= 🛱	🦘 🎇 🕷	ان 👌
		/	7			Meteorological load generator

The function can be used to automatically generate wind and snow loads based on the wind surface settings (internal pressure, velocity pressure, wind friction, geometric parameters) and snow surface settings (snow load parameters).

Each direction and surface can be turned on and off for generation by clicking on the checkboxes.

Name	Categ	ory	Туре	Status	✓ 0	9	0 🔽 180	2	70 Generate	
Internal pressure										
Wind friction										
WindSurface1	Wall		Standard	Valid						
WindSurface2	Wall		Standard	Valid					 Image: A start of the start of	
WindSurface3	Wall		Standard	Valid					 Image: A start of the start of	
WindSurface4	Wall		Standard	Valid					 Image: A start of the start of	
WindSurface5	Duopi	tch roof	Standard	Valid					v	
Generate exception	onal I									_
		Duopitch r	oof	Standard	Valid				✓	
SnowSurface1										

By clicking on the **OK** button the meteorological load generation will be performed. The necessary load groups and load cases will be created automatically.

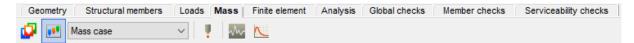




7 MASSES

7.1 BASICS

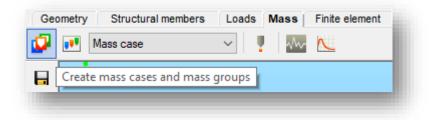
In ConSteel 12, the upper bar has changed, since a new tab, Masses were added, which contains such functions, which are necessary for dynamic calculations. Mass cases, and mass combinations can be created for masses which were placed on the model manually, or converted from loads into masses.



7.2 Mass cases and mass groups

Masses on the structure, just like in the case of loads, has to be grouped together in mass cases and mass groups. Before defining masses on the structure, groups and cases should be

defined with using the **CREATE MASS CASES AND MASS GROUPS** (



The dialogue can be separated into two main parts:

On the left side of the dialogue (#1), loadcases which were defined on the *LOADS* tab can be converted automatically into masses for the dynamic calculations. On the right side of the dialogue (#2), mass groups and cases can be defined manually.



Convert load cases	Manually defin		Mass group (Permane	int)	
⊡- Permanent └── <u>↓</u> Loadcase	G U Perm	anent lass case	Mass group (Permane	Permanent	
1			New 2 Mass case Name Maas group Color		Delete
Considered direction					
In case of Seismic analysis	Direction 1	Direction 2			
In case of Free vibration analysis		Y	New mass case	сору	Delete
			Clos	e Apply	ОК

7.2.1. CONVERTING LOADS INTO MASSES

Loads placed on the Loads tab can be converted into masses on the left side of the Mass cases and mass groups dialogue. The tree panel will automatically filled up with the previously created permanent, variable and snow loadgroups and their corresponding loadcases. If a checkbox of a loadcase is clicked, it will be converted into mass, after pressing the **OK** or **APPLY** button on the dialogue.

7.2.2. MANUAL DEFINITION OF MASS GROUPS AND MASS CASES

MASS CASES AND MASS GROUPS can be created manually, on the right side of the dialogue, just like in the cases of load cases and load groups.



Convert load cases	Manually defi		 Mass group (Permane	nt)	
⊡- <mark>□↓</mark> Permanent └□↓ Loadcase	Pern	nanent Mass case	Name	Permanent	
				2	
		1	New Mass case		Delete
			Name		
			Mass group		~
			Color		
				3	
Considered direction					
In case of Seismic analysis	Direction 1	Direction 2			
In case of Free vibration analysis	x		New mass case		Delete
			Clos	e Apply	ОК

With the **NEW** button on the *Mass groups* (#2) part of the dialogue, new mass groups can be added to the *Manually defined* tree panel. Permanent, Variable and Snow mass groups can be created. After choosing the correct type, a new element will be added to the tree panel (#1), as a mass group.

With the **New MASS CASE** button on the Mass case panel (#3), cases can be added to a previously selected mass group in the tree panel. Any number of mass cases can be added to a mass group. Colour of each mass case can be set manually with the colour palette, which will be used for the graphical object representation int he modeling area.

Each mass case can be moved from a mass group to another, using the drag-and-drop feature of the tree panel (#1).

7.2.3. CONSIDERED DIRECTION OF MASSES

Both for masses converted from loadcases, and for masses defined manually, the direction in which they will be taken into consideration during the dynamic calculations can be set, on the bottom of the *MASS CASES AND MASS GROUPS DIALOGUE* using the checkboxes, after the selection of a mass case in the tree. In case of seismic analysis, only the global Z direction can be turned off, while in the case of vibration analysis, any direction is free to switch on or off (X,Y,Z).



7.3 DEFINITION OF MASS COMBINATIONS

Definition of *MASS COMBINATIONS* works like the *LOAD COMBINATION* definition (*CHAPTER* **6.3**), but without the possibility to generate mass combinations automatically. Factors for each masscases has to be set manually to create a mass combination.

The grid of the mass combinations dialogue can be separated into two parts:

- #1 List of mass cases converted from loads
- #2 –.List of mass cases defined manually

lass combinations				- 🗆 X
👬 눩 🤔 🗗				
		Converted from	Load	Manually defined
Name	Loadcase	Load case	Load case	Mass case
Mass combination	0	0	0	1
1				2
-			Close	Apply OK

7.4 CREATING POINT MASSES

POINT MASS () objects can be placed on any point of the structure. Every placed point load will be automatically assigned to the active masscase of the masscase dropdown menu.



Geometry	Structural members	Loads	Mass	Finite element
💋 💌	Mass case 1	\sim	Ŵ	\sim
	Mass case 1			
	Mass case 2		•	

Point mass can be placed on any point of any bar members, snap points, start and end points and intermediate points are all available. After giving the value of the mass in kg dimension, it can be placed with a simple click on the desired member.

🥊 i	
ameters	
	-
ss 100 kg	
Select the action point of mass!	-

7.5 Seismic effect

To see the detailed description of the usage of the **SEISMIC EFFECT**, please go to **CHAPTRER 12 EARTHQUAKE ANALYSIS**!

7.6 CREATING RESPONSE SPECTRUMS

With the **CREATE RESPONSE SPECTRUM** () function, two type of response spectrums can be defined:

- Standard elastic spectrum: Response spectrum definition according to the selected standard and national annex. Parameters for the definition may varies in case of different annexes.
- User defined response spectrum: Response spectrum definition by defining the points of the spectrum manually



Type of response spectrum	×
• New Standard elastic spectrum	
○ New User defined response spectrum	
Clone an existing response spectrum	
	\sim

7.6.1. New standard elastic response spectrum

By choosing the **New Standard ELASTIC SPECTRUM** option, the spectrum can be created according to the active national which can be used later for response spectrum analysis, and earthquake calculations.

According to the recommended values of *EuroCode*, the following parameters are necessary for the definition of an elastic response spectrum:

- Type of spectrum: the dropdown menu is filled up with the possible types of the active annex
- Importance class: with the dropdown menu, the importance class of the structure can be selected. Value for each selected class is written into the editbox, next to the dropdown menu, according to the selected annex
- Ground type: with the (....) button, the type of the ground below the structure can be selected. At the bottom of the pop-up dialogue, all of the parameters are represented with their values, according to the active annex

Give ground type			×
Other ground ty	pe		
O Type A - Rock o m of weaker ma	r other rock-like ge terial at the surfac	ological formation, i e.	including at most 5
 several tens of 		and, gravel, or very ;, characterized by ; h.	
Type C - Deep c day with thickne	leposits of dense o ess from several te	r medium dense sar ns to many hundred	nd, gravel or cliff ds of meters.
		dium cohesionless so ominantly soft-to-fin	
O of type C or D a		ng between about 5	ayer with vs values 5 m and 20 m,
Horizontal:			
S = 1,5	TB = 0,1	TC = 0,25	TD = 1,2
Vertical:			
S = 1	TB = 0,05	TC = 0,15	TD = 1
			OK

Ground acceleration - a_{gR} [m/s²]: can be defined manually through the editbox, or by clicking on the (...) button, it can be choosen by clicking on a region on the built-in maps:





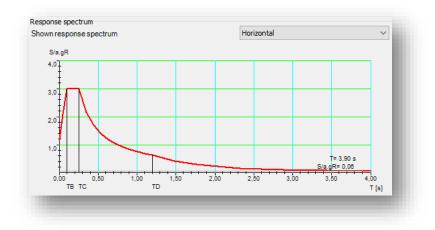
 Vertical acceleration - a_{vg}/a_g: can not be edited, it is for informative purposes. It shows the actual used value according to the active annex.

					~
esponse spectrum parameters					
Type of spectrum	2 ~				
mportance class	I ~ 0,8				
Ground type	C				
Ground acceleration - a _{gR} [m/s2]	0,981				
Vertical acceleration a _{vg} /a _g	0,45				
esponse spectrum		Horizonta			~
Shown response spectrum		Horizonta			, i i i i i i i i i i i i i i i i i i i
S/a.gR 4,0]					- I
‡ l					
3,0					-
2,0					
2,0					
		T= 2,40 s a.g.R= 0,18			
			3,00	3,50	4,00 T [s]

On the bottom of the Elastic response spectrum dialogue, the defined spectrum is given graphically too. By hovering the cursor above the graph, the value of each point can be



obtained. Switching between vertical/horizontal spectrums can be performed using the dropdown menu.



7.6.2. New user defined response spectrum

ConSteel provides two ways for the definition of **NEW USER DEFINED RESPONSE SPECTRUMS**:

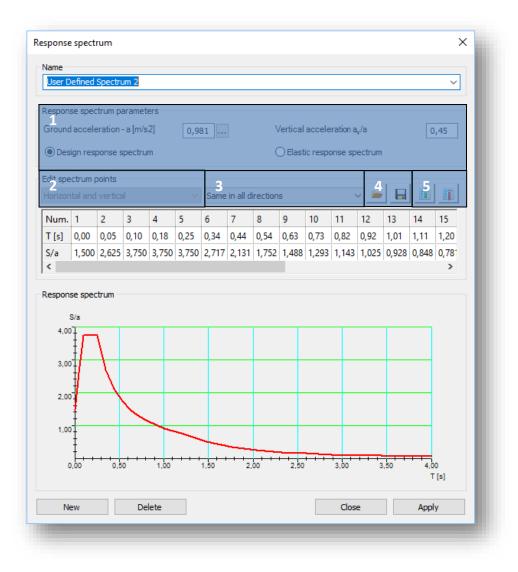
Type of response spectrum		×
New Standard elastic spectrum		
New User defined response spectrum		
Clone an existing response spectrum		
Standard Spectrum 1	~	
Standard Spectrum 1		
	Cancel	Create

- 1. Response spectrum can be created manually from scratch, by simply clicking on the *CREATE* button
- 2. Response spectrum creation based on an existing standard spectrum, by cloning and modifying it. If the *CLONE AN EXISTING RESPONSE SPECTRUM* checkbox is checked in, the dropdown menu of the previously defined spectrums will become active. By choosing a standard response spectrum, and clicking on the *CREATE* button, modification of the selected spectrum can be started.

7.6.2.1 MANUAL DEFINITION OF RESPONSE SPECTRUM

After selecting the **New User DEFINED RESPONSE SPECTRUM** option, and clicking on the **CREATE** button, the creation dialogue will appear.





On the *Response spectrum parameters* part (#1), the following parameters has to be set:

- Ground acceleration a [m/s²]: can be defined manually through the editbox, or by clicking on the (...) button, it can be choosen by clicking on a region on the built-in maps.
- Vertical acceleration a_v/a: ratio of the vertical/horizontal acceleration can be set int he editbox
- Type of the response spectrum: Design or Elastic response spectrum can be choosen.
 Design response spectrum will be used in design situations, while elastic response spectrum will be used in case of serviceability situations

In the *Edit spectrum points* part of the dialogue, the graph of the response spectrum can be defined, by giving the time parameter in seconds, and the corresponding spectrum value divided by the acceleration.



Any number of columns can be added to the spectrum definition grid. By pressing the \square

(#5), new columns can be added. Using the III (#4), columns can be deleted.

A newly defined user response spectrum is the same in all (X,Y,Z) directions. If it is necessary, different spectrums can be defined for each direction, using the dropdown menu on the dialogue.

The data of the spectrum can be saved in *.csv and *.txt format, using the 📕 (#4) button.

Previously created and saved spectrums can be imported using the 💻 (#4) button

7.6.2.2 MANUAL DEFINITION OF RESPONSE SPECTRUM, BASED ON A CLONED STANDARD SPECTRUM

After selecting the **New USER DEFINED RESPONSE SPECTRUM** by cloning an existing response spectrum, and pressing the **CREATE** button, the modification of the selected standard spectrum can be started. All points of the standard design spectrum will automatically be loaded into the table of the definition dialogue, where any point and parameter is free for to edit.

All of the modification options are the same as it is written in chapter **MANUAL DEFINITION OF RESPONSE SPECTRUM, CHAPTER 7.6.2.1**

8 STRUCTURAL ANALYSIS

8.1 BASICS

The analysis of the structural model can be the most "black box" type phase of the design process for the engineer, and additionally the modern structural standards usually define the appropriate and required analysis type for the used design formula. The ConSteel applies the finite element method for all the calculations using two beam-column element types (traditional 12 DOF Timoshenko and 14 DOF thin-walled including warping of the section) and two shell elements (triangle and quadrilateral). The great variety of calculation possibilities includes first and complete second order (not only P- δ effect) analysis; flexural-, torsional-, and lateral-torsional buckling solutions; local buckling analysis of slender steel members built from shell elements; static and dynamic eigenvalue analysis and earthquake calculation based on elastic response spectrum and modal analysis. The exceptionally fast and robust equation solvers yield optimal calculation time even for unusually huge models. The functionalities connected with the structural analysis are placed on two tabs: the **FINITE ELEMENT** and **ANALYSIS** tab. On the **FINITE ELEMENT** tab mesh generation options are available for surface elements, and the generated finite element model can be viewed. Usually this step can be skipped since in the first step of the analysis the finite element mesh generation is automatically performed.

8.2 FINITE ELEMENTS

8.2.1 BASICS

The mechanical performance of the calculated model is always highly influenced and limited by the applied finite element type. The considered displacements, forces, shape functions, second order effects within the finite element model basically determine the quality of the analysis, the expectable results and accordingly the applicable standard verification methods. On the other hand the engineer should be aware of the important characteristics of the used finite elements already at the model building phase in order to avoid the non-expected structural behavior and calculate the mechanical performance of the imagined structure as accurate as possible. In *ConSteel* all the finite element models always have the real 3D behavior; there are no options for special reduced degree of freedom calculations (plane frames, plate or membrane surfaces etc.) if it is required appropriate support and/or load system should be applied. All the defined loads and supports are converted to nodal forces by



load cases and nodal supports, no forces or supports are considered along the finite elements, this feature is taken into account during the automatic FE mesh generation.

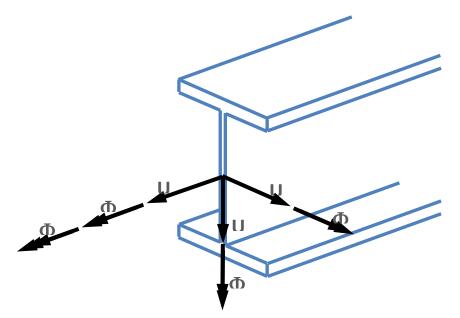
The subsequent sections do not intend to introduce the deep theory behind the applied finite elements – it can be found in the literature – only the most significant features are presented and explained which are important to know for the appropriate interpretation of the results.

8.2.2 LINE ELEMENTS

There are three options for line elements:

- a 6 degrees-of-freedom (6DOF) general beam-column element for the bars subjected to axial force, shear force, bending moment, torsion or any interaction of these
- a 7 degrees-of-freedom (7DOF) beam-column with warping element for the bars subjected same as 6DOF element + warping effect
- > a tension-only (TO) element for bars subjected by tension force

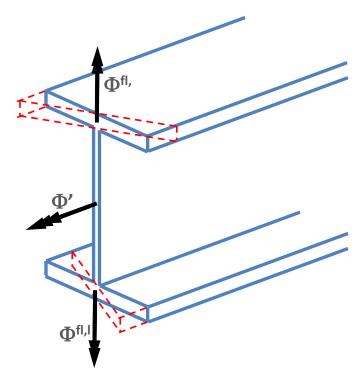
The 7DOF element is specially developed for thin-walled members where the warping of the cross-section is of high importance in the behavior, this effect is considered by the 7th DOF. In the following figure the considered nodal displacements are illustrated:



The first 6 DOF are the conventional displacements (U_x, U_y, U_z) and rotations (Φ_x , Φ_y , Φ_z) according to the local coordinate system of the member (see section **4.2 COORDINATE SYSTEMS**). What needs more explanation is the 7th DOF which is mathematically the first derivative of the twist about the longitudinal axis (Φ'_x); mechanically it represents the warping of the section which is straight consequence of torsion on thin-walled members. The next figure illustrates the warping effect of I shaped cross-section when the flanges step out of the original plane of the section.







In this case the warping DOF can be considered as a dual and opposite rotation of the flanges about the axis perpendicular to their width (in this case the local axis "z").

Since steel members are usually relatively slender various modes of global stability failure can occur: flexural buckling, torsional buckling, lateral-torsional buckling and any interactions of these, all modes can be calculated by the 7DOF element. It is a quite important and advantageous feature in the stability design of these members but since the accurate calculation of all the torsional modes is highly dependent on the 7th warping DOF it is essential to consider this effect already in the modeling phase (see chapter *5 STRUCTURAL MODELLING* for the modeling warnings).

The TO elements have only 1 DOF which is the longitudinal displacement (U_x). These elements are considered in the calculation only if they are subjected to tension accordingly the calculations are iterative in nature. At first an analysis is performed considering all the TO elements modeled by one finite element. Next a force check is executed on the TO elements and the compressed ones are neglected from the model and a new calculation is initiated. It is continued until all the TO elements have tension. This model configuration is taken into account in the eigenvalue calculations.



Since the eigenvalue calculations can not be performed iteratively it may happen that in the eigenshape the TO element gets compression i.e. its length shortens. In this case if this effect is significant and should be avoided a new eigenvalue analysis should be run without the TO element.



8.3 MODEL CHECK (DIAGNOSTICS)

In *ConSteel* there is a possibility to perform a model check previous to executing any calculations. This function automatically runs before starting the finite element mesh generation or analysis but can be initiated any time (switching the "Diagnostics" on the *VIEW* menu and run) examining the recent state of the model. There are two kinds of diagnostics messages:

- **ERROR**: the errors make the calculations impossible or meaningless to execute so the detected errors stop further calculations
- ▶ **WARNING**: the warnings allow the calculations but notice the possible errors

One part of the model checks is performed on the user model (basic check), where basic requirements for the normal performance of a model are controlled; the other part is performed on the generated finite element model (pre-calculation check).

The following basic checks are performed:

- existence of load on the structure
- existence of support on the structure
- length of bars, line loads and line supports
- > value for the thickness and finite element size for surface members
- overlap, length and compatibility of haunches
- multiple supports on the same place
- compatibility of tension bars

The following pre-calculation checks are performed:

- overhang of line loads and line supports
- point loads and point supports are not on the model
- overlap of surface members
- overlap of bar members
- very small distance (< 5 mm) between points or lines of surfaces, bars, loads or supports (the limit distance can be set in the **OPTIONS** menu)

The object which the errors or warnings are detected can be selected and deleted from the diagnostics results table (in the middle of the right tables).



8.4 ANALYSIS SETTINGS

The required analysis types can be set on the **SET ANALYSIS PARAMETERS** dialogue, which has one **OVERWIEW PAGE**, and four subpages for the three type of separated analysis settings for the better transparency. These four types of analysis settings are:

- **GLOBAL SETTINGS** (with Subpage of global settings **8.4.2.**)
- ► *STATICAL ANALYSIS* (with Subpage of load cases *8.4.3*, Subpage of load combinations *8.4.4.*)
- **DYNAMIC ANALYSIS** (with Subpage of dynamic analysis **8.4.5**.)

8.4.1 OVERVIEW PAGE

On the so called **OVERVIEW PAGE** settings of the analysis are applied on all load cases/load combinations etc.

For a better transparency the analysis setting are ordered in separated groups.

nalysis parameters		<u> </u>
Global settings Global imperfection Include connection stiffne	- No	->
Statical analysis Load cases I Elastic analysis of load ca Elastic analysis of basic v	ses alues of reactions for foundation checks	>
Load combinations Type of calculation Ultimate limit states V First order analysis Second order analysis V Budding analysis V Budding sensitivity Influence graphs	Elastic Serviceability limit states First order analysis Second order analysis	->
Dynamic analysis Free vibration analysis Response spectrum analysis	:e graphs	->
	Cancel Calculatio	n Apply

Settings of the **Global settings** group (#1) are applied for the whole model:

- With the global imperfection dropdown menu previously created Global imperfection (see. chapter 6.5) can be set for the model. Only one global imperfection group can be applied in the same time
- With the checkbox previously created and placed joints' stiffness can be used during the calculation. By turning on the checkbox, connection stiffnesses will be applied on all of the load combinations in the model.

For further settings, the **SUPBAGE OF GLOBAL SETTINGS** can be opened with the button.

Settings of the **Statical analysis** group (#2):

CONSTEEL

The two checkboxes of Load cases are valid only for elastic analysis: calculation can be executed for the separate load cases. For the special load combinations necessary for the foundation design in CSJOINT, the second checkbox should be checked. SUPBAGE

OF LOAD CASES can be accessed with the button.

- In the Load combination group the type of analysis can be selected from the dropdown menu. Basically two types of analysis can be performed: 1) elastic: all the elements have linear elastic material following the Hook-law, 2) plastic: plastic hinge analysis considering the concentrated plastification of special cross sections due to bending moment, detailed description is in chapter 8.5.2. Necessary analysis types (first or second order) can be set separately for the ULS and the SLS load combinations.
- With the Buckling analysis checkbox the calculation of the buckling analysis can be decided. If buckling analysis is turned on 10 eigenshapes will be calculated for each loadcombinations by default. The number of the eigenshapes can be modified at the

SUPBAGE OF LOAD COMBINATIONS with the button. With the last checkbox the buckling sensitivity analysis can be turn on and off.

Settings of the Dynamic analysis group (#3:)

In the Dynamic analysis group the parameters of the dynamic analysis can be set. Free vibration analysis and Modal Response Spectrum Analysis (Seismic MRSA) can be calculated.

For further settings, the **SUPBAGE OF DYNAMIC ANALYSIS** can be opened with the button.



8.4.2 SUBPAGE OF GLOBAL SETTINGS

import stiffness of semi-rigid connection:			<u> </u>
Load combinations	Apply connection stiffness	Use Si.sd instead of Si.ini/ŋ	^î
(ULS) Load combination-1			
(ULS) Load combination-2			
(ULS) Load combination-3			
(ULS) Load combination-4			
(ULS) Load combination-5			=
(ULS) Load combination-6			
(ULS) Load combination-7			
(ULS) Load combination-8			
(ULS) Load combination-9			
(ULS) Load combination-10			
(ULS) Load combination-11			
(ULS) Load combination-12			
(ULS) Load combination-13			
(ULS) Load combination-14			
(ULS) Load combination-15			
(ULS) Load combination-16			
(ULS) Load combination-17			
(ULS) Load combination-18			
(ULS) Load combination-19			
(ULS) Load combination-20			
(ULS) Load combination-21			
(ULS) Load combination-22			
(ULS) Load combination-23			
(ULS) Load combination-24			
(ULS) Load combination-25			
(ULS) Load combination-26			
(ULS) Load combination-27			
0101111111000			· · · ·

On this subpage, previously created and placed joints' stiffness can be applied in the calculation in the selected load combinations. With the checkboxes in the second column secant stiffness can be used instead of initial stiffness.



8.4.3 SUBPAGE OF LOAD CASES

Name	Use common settings	
Sheeting		
Wind_0FGHmax_IJmax		
Wind_0FGHmax_Umin	Beam results Shell results	
Wind_0FGHmin_JJmax		
Wind_0FGHmin_Umin	Reactions	
Wind_0_Int1_FGHmax_IJmax		
Wind_0_Int1_FGHmax_IJmin		
Wind_0_Int1_FGHmin_IJmax		
Wind_0_Int1_FGHmin_IJmin		
Wind_90_FGHmax_IJmax		
Wind_90_Int1_FGHmax_IJmax		
Snow_0		
Snow_1		
Snow_2		

This subpage has two main parts:

The load cases table (#1)

In this table, all of the loadcases are listed. With the checkboxes, it can be decided that which of the loadcases should be calculated during the analysis. Results for load cases will be calculated according to the settings in the result tree (#2)

The result tree (#2)

In the result tree, it can be decided, that which result type should be calculated for the load cases. Calculated results can be a common setting for all of the load cases, if the **USE COMMON SETTINGS** checkbox is clicked. If it is unclicked, calculated results can be set differently for each loadcases by clicking on the name of a loadcase, and selecting the proper result types.



CONSTEEL

lel p	ortion for second order analysis		- Who	le model -		•	Use common settings			
tel p	ortion for buckling analysis		- Who	le model -		•	Results			
er lir	mit of relevant buckling eigen values		10				Deformations			
5	Name	First o	Secon	Number of	Sensiti	~	Shell results			
2	(ULS) Load combination-1			10	V		Reactions			
	(ULS) Load combination-2	V		10	V					
	(ULS) Load combination-3	V								
	(ULS) Load combination-4	V		10	v	Ξ				
	(ULS) Load combination-5	V								
	(ULS) Load combination-6	V								
	(ULS) Load combination-7	V								
	(ULS) Load combination-8	×								
	(ULS) Load combination-9	V								
	(ULS) Load combination-10	V								
	(ULS) Load combination-11	×								
	(ULS) Load combination-12	V								
	(ULS) Load combination-13	V								
	(ULS) Load combination-14	×								
	(ULS) Load combination-15	V								
	(ULS) Load combination-16	V								
	(ULS) Load combination-17	V								
	(ULS) Load combination-18	Image: A start and a start								
	(ULS) Load combination-19	V								
	(ULS) Load combination-20	Image: A start and a start								
	(ULS) Load combination-21	V								
	(ULS) Load combination-22	V								
	(ULS) Load combination-23	V								
	(III S) Load combination 24					-				

This subpage has three main parts:

Portion and relevant buckling eigenvalues settings (#1)

In the dropdown menu previously created model portion or whole model can be selected for second order and buckling analysis. The analysis will be performed just for the selected portion. Here can be entered also the upper limit of the relevant buckling eigen values. All of the eigen values which are higher than the limit will be skipped. Because of this the final number of the calculated eigenshapes could be less then the set.

The load combinations table (#2)

In this table, all of the loadcases are listed. With the checkboxes in the first column, it can be decided that which of the loadcombinations should be calculated during the analysis. First and second order analysis calculation can be turned on or off for each combination, as well as the calculation of buckling sensitivity. Number of calculated eigenshapes can be given for each combinations individually Results for load cases will be calculated according to the settings in the result tree (#3)



The result tree (#3)

In the result tree, it can be decided, that which result type should be calculated for the load combinations. Calculated results can be a common setting for all of the load combinations, if the **USE COMMON SETTINGS** checkbox is clicked. If it is unclicked, calculated results can be set differently for each loadcombinations by clicking on the name of a loadcombination, and by selecting the proper result types.

8.4.5 SUBPAGE OF DYNAMIC ANALYSIS

) First order stiffness		econd order stiffness		3	
Name Averification_effect (M B Mass combination-1	la: 10	Load combination		Results	
Response spectrum analys Relative modal damping Consideration of modal s	factor	0,05			
Consider all modal sha	es ε _{lX} (τ ε ₁₂ 0,05		

This subpage has three main parts:

The dynamic calculation settings table (#1)

Above the table, it can be decided, if first, or second order stiffness matrix should be used for the dynamic calculations. In case if the second order stiffness checkbox is checked, an additional Load combination column will appear in the table below, where it has to be decided that which load combination's second order stiffness should be used for each seismic or dynamic calculation.



In the table, the previously defined seismic effect, and all of the mass combinations are being listed as individual rows. Number of dynamic eigenshapes can be defined for each effect and mass combination, the default value is 10.

The rows of the table can be separated into two groups:

- A, Row for Modal Response Spectrum Analysis (MRSA)
 For seismic MRSA calculation, it is necessary to have at least one *SEISMIC EFFECT* defined and assigned to the seismic load group in the model. For the possibilities of definition of seismic effects, please check out *CHAPTER* 12
 EARTHQUAKE ANALYSIS.
- B, Rows for dynamic and response spectrum analysis

Each defined *MASS COMBINATION* is listed as an individual row in the table. According to the settings on the analysis Overview page in chapter **8.4.1**, free vibration and response spectrum analysis can be performed.

In case if the check box of "To calculate" is turned on, and "Response spectrum analysis" is turned off for a mass combination, only a *FREE VIBRATION ANALYSIS* will be performed.

In case if bot of the "To calculate" and "Response spectrum analysis" checkboxes are turned on for a mass combination, **RESPONSE SPECTRUM ANALYSIS** will be added to the free vibration calculation. Predefined response spectrum is necessary to be defined previously in the model.

▶ The Response spectrum analysis settings (#2)

Covers the relative modal damping factor, what can be overwritten manually, and the consideration of modal shapes in each direction. By default, all of the modal shapes will be considered during the calculation, what can be changed using the dropdown menu.

The result tree (#3)

In the result tree, it can be decided, that which result type should be calculated for the mass combinations. Calculated results can be a common setting for all of the mass combinations, if the **USE COMMON SETTINGS** checkbox is clicked. If it is unclicked, calculated results can be set differently for each mass combinations by clicking on the name of a masscombination, and by selecting the proper result types.

8.5 ANALYSIS TYPES

8.5.1 ELASTIC ANALYSIS

8.5.1.1 FIRST ORDER

The first order analysis calculates the structural response considering the initial stiffness of the model. It is advisable to run a first order analysis in order to check the model performance before executing more costly calculations (for instance eigenvalue analysis). The steps of the first order analysis are the following:

- Calculation of first order (initial) stiffness matrices (K^{el}s) of the finite elements in their local coordinate system
- 2. Compiling the global stiffness matrix (K_s) and nodal force vector (P) of the whole model by transforming the element stiffness matrices into the global coordinate system
- 3. Modifying the global stiffness matrix and nodal force vector considering the special boundary conditions (supports, continuity releases between the elements, prescribed displacements, temperature loads etc.)
- 4. Solution of the basic linear system of equations which writes the relationship between the applied nodal forces – known variables generated from the loads – and the nodal displacements – unknown variables (U) – in the global coordinate system for the global model (see section 8.2 FINITE ELEMENTS for the interpretation of nodal displacements and forces):

$$\mathbf{K}_{s}\mathbf{U}=\mathbf{P} \quad \rightarrow \quad \mathbf{U}=\mathbf{K}_{s}^{-1}\mathbf{P}$$

(1)

 Calculation of the internal forces (and stresses) of the elements (f^{el}) in their local coordinate system by transforming the global nodal displacements of the element into local system (u^{el}) using their local stiffness matrices:

$$\mathbf{f}^{el} = \mathbf{K}_{s}^{el} \mathbf{u}^{el}$$

(2)

8.5.1.2 SECOND ORDER

The second order calculations take into account that the loaded and deformed structure can behave differently than the initial configuration. This effect can be considered as if the initial stiffness was changing during the loading history. The steps of the second order analysis are the following:

- 1. Performing the whole first order analysis as described in the previous section
- 2. Calculation of geometric stiffness matrices (K^{el}_g) of the finite elements in their local coordinate system by the internal forces of the elements (f^{el})

CONSTEEL

- 3. Compiling the second order global stiffness matrix $(K_s + K_g)$ and nodal force vector (P) of the whole model by transforming the element stiffness matrices into the global coordinate system
- Modifying the second order global stiffness matrix and nodal force vector considering the special boundary conditions (supports, continuity releases between the elements, prescribed displacements, temperature loads etc.)
- 5. Solution of the basic linear system of equations which writes the relationship between the applied nodal forces – known variables generated from the loads – and the nodal displacements – unknown variables (U) – in the global coordinate system for the loaded and deformed global model (see section 8.2 FINITE ELEMENTS for the interpretation of nodal displacements and forces):

$$(\mathbf{K}_{s} + \mathbf{K}_{g})\mathbf{U} = \mathbf{P} \quad \rightarrow \quad \mathbf{U} = (\mathbf{K}_{s} + \mathbf{K}_{g})^{-1}\mathbf{P}$$
(3)

 Calculation of the internal forces (and stresses) of the elements (f^{el}) in their local coordinate system by transforming the global nodal displacements of the element into local system (u^{el}) using their local stiffness matrices:

$$\mathbf{f}^{el} = \mathbf{K} \, {}^{el}_{s} \mathbf{u}^{el}$$

7. If the difference between the new nodal displacements and the ones obtained earlier exceeds a certain limit repeat the calculations from step 2

8.5.1.3 Static eigenvalue – buckling analysis

In a mechanical interpretation the eigenvalue analysis approximates the elastic critical load levels where the structure is subjected to some modes of loss of stability. Mathematically it means that the second order equation of (3) has no unique solution because the second order stiffness matrix is singular. In *ConSteel* linear eigenvalue analysis is performed considering one parameter, conservative loading and that the geometric stiffness matrix depends linearly on the load factor (λ):

$$\mathbf{K}_{g}(\lambda \mathbf{f}) = \lambda \mathbf{K}_{g}(\mathbf{f})$$
(4)

In this case the eigenvalue analysis can be written in the following form:

$$\left(\mathbf{K}_{s} + \lambda \mathbf{K}_{g}\right)\mathbf{U} = 0 \tag{5}$$

The solutions are certain critical load factors (buckling loads factor λ_{cr}^{i}) which make the second order stiffness matrix singular and corresponding displacements (buckling shapes **U**ⁱ).

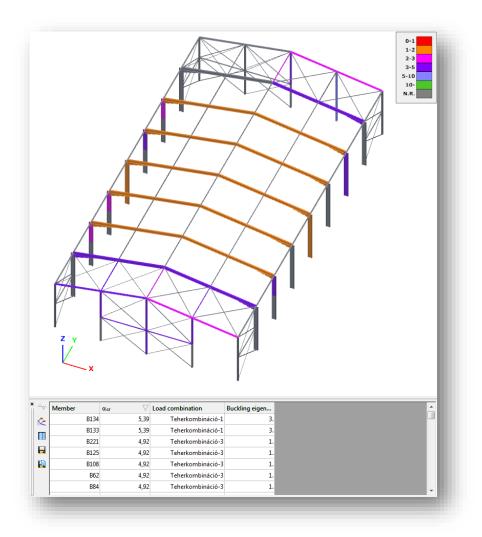
The possible buckling shapes which can be calculated by this eigenvalue analysis are basically influenced by the considered second order effects which are determined by the applied finite element. In case of beam-column structures the 7 DOF finite elements have the capability to



consider all modes of global buckling shapes: flexural buckling, torsional buckling, lateraltorsional buckling and any interactions of these.

8.5.1.3.1 BUCKLING SENSITIVITY

Buckling sensitivity analysis is a very useful function to assist the global stability design. Buckling sensitivity gives a review about the eigen shapes of the structure, and gives for every member the relevant eigen shape which is the best for the buckling analysis.



8.5.1.4 DYNAMIC EIGENVALUE – VIBRATION ANALYSIS

In a mechanical interpretation the dynamic eigenvalue analysis approximates the eigenfrequencies where the structure is subjected to some modes of free vibration. In *ConSteel* linear eigenvalue analysis is performed considering the second order stiffness matrix and consistent mass matrix (M)

$$\left(\left(\mathbf{K}_{s}+\mathbf{K}_{g}\right)+\omega^{2}\mathbf{M}\right)\mathbf{U}=0$$
(6)

(0)



The solutions are certain eigenfrequencies (ω) and corresponding displacements (free vibration shapes **U**ⁱ).

The possible vibration shapes which can be calculated by this eigenvalue analysis are basically influenced by the considered stiffness and mass matrices which are determined by the applied finite element. In case of beam-column structures the 7 DOF finite elements have the capability to consider all modes of global vibration shapes: flexural, torsional, lateral-torsional vibration and any interactions of these.

8.5.2 PLASTIC ANALYSIS

8.5.2.1 BASIC

When plastic analysis is selected a plastic hinge analysis is performed considering planar hinges which may develop about the local *y* axis of the cross sections due solely to the major axis bending moment (M_y) – no other effects (like shear, normal force or minor axis bending) influence the formation of the hinge. The basic settings for the plastic analysis are available by pressing the three dots (....) button next to the *Type of calculation* combo on the *SET ANALYSIS PARAMETERS* dialogue.

onditions of development of planar plastic hinge			
Model portion considered in plastic hinge analysis	ole model -		•
✓ The cross section is at least one line symmetric			
\fbox The cross section is compact - class 1 or 2			
Accuracy of condition check of development of planar plastic hinge - N	ipl% (0,01	
Behaviour of plastic hinges			
V Plastic rotation about major axis			
V Plastic rotation about minor axis			
☑ Plastic warping deformation			
Warnings			
The rotation in the plastic hinge exceeds limit	6	5	0
✓ Normal force exceeds limit	C),25	* Npl
Minor axis bending moment exceeds limit	C),1	* Mpl
☑ Minor axis shear force exceeds limit	C),5	* Vpl
Results of plastic hinge stages			
Deformations			
Beam results			
Shell results			
Reactions			
	Cancel		OK
	Cancer		UK



The first group of options controls the conditions for the development of the plastic hinge. The proper model portion can be selected where the hinges are searched. Two options are available for the cross section where hinges may develop: limited to at least singly symmetric (about the local 'z' axis) and to compact (AISC) or class 1 (EC) sections. According to the usual rules of design standards these last two restrictions are necessary for the plastic hinge analysis. The accuracy of the hinge calculation can also be set as the percentage of the plastic bending resistance (M_{pl}).

In the second group of options the mechanical behavior of the plastic hinges after formation. Since the planar hinges are part of a spatial model (the "planar" word refers only to the source of the hinge formation) the further mechanical behavior should be defined as a spatial hinge object (spatial release). Accordingly the hinge can rotate about the major axis (in the original plane of source bending moment M_y), about the minor axis together with free warping deformation. The two latter options are realistic considering that a planar plastic hinge in an I-shaped section practically causes fully yielded flanges which obviously implies the minor axis rotation and warping hinge behavior. These options influence however only the out-of-plane behavior of the model (for instance buckling of members next to the hinge out-of-the-plane of the planar hinge).

The third group of options offers warning displays. These warnings are indicating if the plastic hinge is significantly influenced by other effects than the major axis bending moment (M_y) during the whole hinge history.

In the fourth group of options the important results can be selected which have to be saved and displayed for all the stages. In case of larger models and possibility for large number of hinges the amount of results can increase significantly so it is advisable to carefully choose the really important ones.

8.5.2.2 FIRST ORDER

Coming soon!

8.5.2.3 SECOND ORDER

Coming soon!

8.5.2.4 STATIC EIGENVALUE – BUCKLING ANALYSIS

Coming soon!

8.6 ANALYSIS RESULTS

8.6.1 RESULT TYPES

For the two types of finite elements (line and surface elements) the following analysis results are available:

- 1. Line elements Elastic and plastic analysis
- First and second order calculations:
 - ▶ Deformations on the finite element nodes in the global coordinate system
 - XYZ all components
 - X only displacements in the "X" direction
 - Y only displacements in the "Y" direction
 - Z only displacements in the "Z" direction
 - R_x only rotation around the local "x" axis of the member
 - Ry only rotation around the local "x" axis of the member
 - R_z only rotation around the local "x" axis of the member
 - W only warping of the member [1/rad]
 - R_{xx} only rotation of the member [degree]
 - Internal forces on the end nodes of each finite element in the local coordinate system
 - N axial force
 - V_y shear force in the local "y" direction
 - V_z shear force in the local "z" direction
 - T torsional moment
 - M_y bending moment about the local "y" axis
 - M_z bending moment about the local "z" axis
 - B bimoment

Reactions – on the support nodes in the local coordinate system of the support

- R all the reaction forces and moments
- R_R all the reaction forces
- R_{RR} all the reaction moments
- R_x reaction force in the in the global "X" direction
- R_y reaction force in the in the global "Y" direction
- R_z reaction force in the in the global "Z" direction
- R_{xx} reaction moment about the global "X" direction
- R_{yy} reaction moment about the global "Y" direction



- R_{zz} reaction moment about the global "Z" direction
- Static eigenvalues and corresponding eigenshapes on the finite element nodes in the global coordinate system

Only elastic analysis

Dynamic eigenvalues and corresponding eigenshapes – on the finite element nodes in the global coordinate system

Only elastic analysis

- Plastic hinges plastic hinge history with the positions of the plastic hinges on the proper finite element ends and the corresponding stages by load factors
- 2. Surface elements
- First and second order calculations:
 - ▶ Deformations on the finite element nodes in the global coordinate system
 - XYZ all components
 - X only displacements in the "X" direction
 - Y only displacements in the "Y" direction
 - Z only displacements in the "Z" direction
 - R_x only rotation around the local "x" axis of the member
 - R_y only rotation around the local "x" axis of the member
 - R_z only rotation around the local "x" axis of the member
 - W only warping of the member [1/rad]
 - R_{xx} only rotation of the member [degree]
 - Equilibrium signed summation of the internal nodal force components and the appropriate external force and reaction components, all the values should be zero which means the model is in equilibrium in the calculated deformed state
 - Forces on surface edge forces and moments per unit length on the end nodes of each finite element in the local coordinate system
 - m_x bending moment on the section perpendicular to the local "x" axis
 - m_y bending moment on the section perpendicular to the local "y" axis
 - m_{xy} torsional moment
 - n_x normal force in the local "x" direction
 - n_y normal force in the local "y" direction
 - n_{xy} membrane shear force
 - v_{xz} plate shear force on the section perpendicular to the local "x" axis
 - v_{yz} plate shear force on the section perpendicular to the local "y" axis
 - v_{Rz} resultant plate shear force

α_n – principal directions of the membrane forces

$$\alpha_n = \frac{1}{2} \arctan\left(\frac{2n_{xy}}{n_x - n_y}\right) - 90^\circ < \alpha_n \le +90^\circ$$

α_m – principal directions of the plate moments

$$\alpha_m = \frac{1}{2} \arctan\left(\frac{2m_{xy}}{m_x - m_y}\right) - 90^\circ < \alpha_m \le +90^\circ$$

Stresses on surface – on upper, middle and lower plane of the surface on the end nodes of each finite element in the local coordinate system

- σ_x normal stress in the local "x" direction
- σ_y normal stress in the local "y" direction
- σ_{xy} shear stress

CONSTEEL

σ₁ – maximum principal stress

$$\sigma_1 = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \sigma_{xy}^2}$$

σ₂ – minimum principal stress

$$\sigma_2 = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \sigma_{xy}^2}$$

σ_{HMH} – Huber-Mises-Hencky stress

$$\sigma_{HMH} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\sigma_{xy}^2}$$

α_σ – principal directions of the stresses

$$\alpha_{\sigma} = \frac{1}{2} \arctan\left(\frac{2\sigma_{xy}}{\sigma_{x} - \sigma_{y}}\right) - 90^{\circ} < \alpha_{\sigma} \le +90^{\circ}$$

- Reactions on the support nodes in the global coordinate system
 - R all the reaction forces and moments
 - R_R all the reaction forces
 - R_{RR} all the reaction moments
 - R_x reaction force in the in the global "X" direction
 - R_y reaction force in the in the global "Y" direction
 - R_z reaction force in the in the global "Z" direction
 - R_{xx} reaction moment about the global "X" direction
 - R_{yy} reaction moment about the global "Y" direction
 - R_{zz} reaction moment about the global "Z" direction
- Static eigenvalues and corresponding eigenshapes on the finite element nodes in the global coordinate system



Dynamic eigenvalues and corresponding eigenshapes – on the finite element nodes in the global coordinate system

8.6.2 VISUALIZATION OPTIONS

CONSTEEL

The results of the performed analysis are usually a huge amount of data so the efficient handling of it has great importance. There should be opportunities for global overview and for obtaining accurate, detailed information about a certain part or problem. There are two main possibilities for the demonstration of the analysis results: the *graphical visualization* and the *result tables* (for the general functionality of tables used in *ConSteel* see section **1.4** *GENERAL FUNCTIONS FOR TABLES*). The two visualization options are obviously in strong connection and can be manipulated by the functions placed on the upper part of the *ANALYSIS* tab. The four combos contain the following selection options (from left to right):

First order	combination-1	✓ Defor	mation - XYZ	 Diagram 🔽	
				View	
				Finite element	
				Diagram Contour line	
				Discrete colors	
				Contour surface	

- selection of analysis type (discussed in section 8.5 ANALYSIS TYPES)
- selection of load combination or load case or envelope figures
- selection of result type (discussed in section 8.6.1 RESULT TYPES)
- selection of type of view (finite element/diagram/contour line/ discrete colors/contour surface)

If choosing the *DISCRETE COLORS* in the dropdown menu of the type of view, there is a possibility to modify the default settings of the color palette. To open up the *PALETTE*

4,00 -	
4,00 -	
2,00 -	
1,10 -	
0,20	
0,00	c

SETTINGS dialogue, a right click on the palette summary (



7,000	۵ 📾	Name	Palette	2		-
4,000	ి 📾	Result type	Deformat	ion		
2,000	۵ 🔒	Scale division			4	
0,000	0	Scale unision			7	•
			Reset automa	tic palette		
		Save	Reset automa	tic palette		
				tic palette	_	Save
		Save		tic palette	_	Save

On the palette settings dialogue, the division of the scale can be set with the up and down arrow buttons (). With the delete button () on the dialogue, division values can be deleted. Division values are calculated automatically by default between the minimum and maximum values of the results, but these values can be set manually at any time, with a manual input of a new division value. If a value is modified manually, a refresh button () will appear next to it, what can be used to recalculate it according to the division scale.

Modifications can be saved for further use by clicking on the save button. The settings will be saved to the UserConfig.xml, what can be found in the Documents/ConSteel folder.

Also, the modified user palette can be set as a default palette.



All the model view options discussed in section **3.2 MODEL VIEWS** are applicable on the result graphics. Moving the slide beside the combos the scaling of the results can be adjusted on the graphics. In case of partial or submodel view the graphics and the tables show only the results of the actual model part. Three types of result tables can be viewed:



Extreme values by members: select the maximum and minimum values for each structural members

K.

User defined values: show the values belonging to the markers defined by the user (discussed more deeply in section **8.6.3 RESULT MARKERS**)



All values.

The result tables arrange the values according to the current result type:

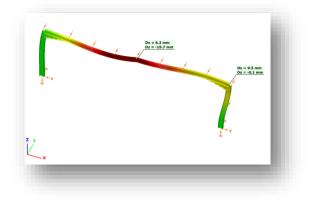
- displacement type results (first or second order deformations, static or dynamic eigenshapes): finite element node number, displacement components
- force type results (internal forces, forces or stresses on surface): finite element node number, finite element number, force components
- reaction type results (reactions, equilibrium): finite element node number, reaction components.

8.6.3 RESULT MARKERS

On the graphical interpretation of the results markers can be placed in order to show certain significant values. When moving the mouse along the structural elements (or more correctly the finite elements) the result markers continuously appear showing the actual values. These markers can be fixed by clicking on the right mouse button and choosing the 'Marker' option.

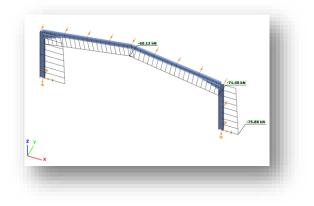
The fixed markers will appear on every result views where it has interpretable value:

Deformation

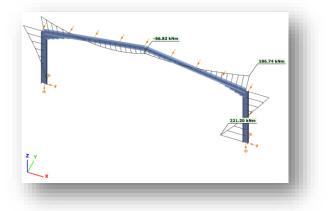


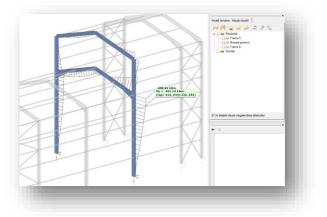


Normal force



Bending moment





When there are fixed markers on the model the 'User defined values' table contains the appropriate (view dependent) values of the marked points. In this table the markers can be switched to disabled by uncheck the proper row.



	5	37	-60,130	0,000	4,900	0,000	-56,820	0,000	0,000
	4	56	-71,400	0,000	52,980	0,000	186,740	0,000	0,000
V	3	9	-75,880	0,000	-78,680	0,000	221,200	0,000	0,000

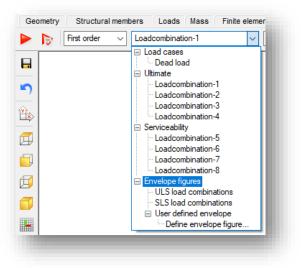
The markers can be deleted by:

- click on the proper row in the table by the right mouse button
- click on the marked point on the graphics by the right mouse button and chose 'Remove marker'

Extreme values can automatically marked by the check boxes on the 'Extreme values' table rows.

8.6.4 ENVELOPE FIGURES

In ConSteel, *min. max and min-max* envelope figures are available both for *bar members*, and *shell models* too. Envelope visualization of results can be accessed through the *LOAD COMBINATION* dropdown menu of the *ANALYSIS* tab, after a successful calculation. Envelope figures are available both for first, and second order results. By default, there are two envelope figures to choose from, one for all of the calculated ULS and one for all of the calculated SLS combinations. It is possible to create custom envelope figures, which is explained in details in the next *CHAPTER* 8.6.4.1.

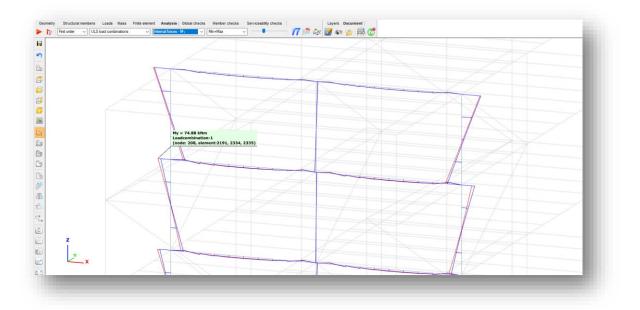


The following result types can be displayed through envelope figures (detailed explanation on *Result Types* can be found in *CHAPTER 8.6.1*):

- Internal forces for bar members
- Forces on shell
- Stresses on shell



There is a colour scheme for the envelope figures. Minimal values represented with blue, while maximal values with a red diagram. When hovering the cursor above a point of an envelope, (either min or max), the value, and the load combination from which it is coming from, is displayed in a flag.



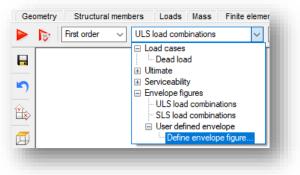
Placing flags on the envelope figures values is also possible. In this case, the table on the bottom of the main window will contain every envelope information about the flagged point.

-				Load combination	N [kN]						B [kNm ²]
'	Minimal	199	2360	Loadcombination-1	-631,880	5,100	- 50, 560	0,011	-75,890	-7,350	-0,019
~	Minimal	208	2334	Loadcombination-3	-578,371	4,510	39,530	-0,010	58,240	-6,420	0,019
~	Maximal	208	2334	Loadcombination-1	-624,410	5,430	49,730	-0,012	74,880	-7,850	0,021

8.6.4.1 USER DEFINED ENVELOPE FIGURES

Besides the default, built in envelope options (all ULS and SLS combinations), it is possible to create user defined envelope figures. This means, that the results, from which the envelope function can build up the figure, can be specified, and saved as a new **USER DEFINED ENVELOPE FIGURE**. Definition can be initiated from the **LOAD COMBINATION** dropdown menu's **DEFINE ENVELOPE FIGURE**... option, after a successful first and/or second order analysis calculation.





After clicking on the function, the **USER DEFINED ENVELOPE** dialogue will appear. On this dialogue, any number of custom envelope figures can be created, using the **NEW** button.

Load c	combinations	ULS load combinations	
O Load c	ases	SLS load combinations	
and comb	pinations or Load cases		
All load co	ombinations		~
To cal	Load combinations		
 Image: A start of the start of	(ULS) Loadcombination-1		
 Image: A start of the start of	(ULS) Loadcombination-2		
▼ ▼	(ULS) Loadcombination-3		
 Image: A start of the start of	(ULS) Loadcombination-4		

If the Load combinations is selected at the **PARAMETERS** part of the dialogue, the elements of the dropdown menu below **LOAD COMBINATIONS OR LOAD CASES**, and the table below will be different.

If the Load combinations is selected at the parameters, the options of the dropdown menu (#1) are:

- All load combinations: All of the calculated load combinations will be used for the envelope figure
- Selected load combinations: Calculated load combinations can be selected with checkboxes to be taken into account for the envelope figure



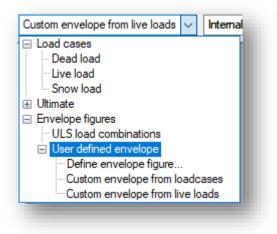
Load combinations based on selected load cases: Only those load combinations will be taken into account, which are generated based on the selected loadcases, using the checkboxes

d combinations based on selected load ca	ases	~
Persistent	Load combinations	
······ ✓↓ Dead load	(ULS) Loadcombination-1	
Live load	(ULS) Loadcombination-2	
Inow ↓ Snow load	(ULS) Loadcombination-3	
	(ULS) Loadcombination-4	

If the Load case is selected at the parameters, the options of the dropdown menu (#1) are:

- ▶ All loadcases: All of the loadcases will be used for the envelope figure
- Selected loadcases: Only the selected loadcases will be used for the envelope figures

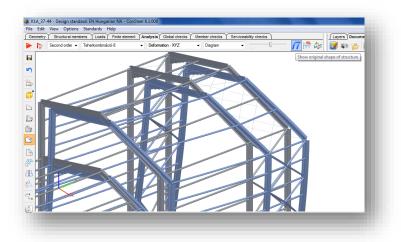
After the user defined envelopes are created, they will be listed at the *LOAD COMBINATIONS* dropdown menu of the *ANALYSIS TAB*, and will be listed at the *USER DEFINED ENVELOPE* part:



8.6.4 Show original shape of structure

SHOW ORIGINAL SHAPE OF STRUCTURE function will show the original shape of structure and the deformed structure at the same time:





8.6.5 TABLE OF DOMINANT VALUES

The last icon on the **ANALYSIS** tab activates the table of dominant values. The table is view sensitive which means that the dominant values are always selected from the current result type. Only the values at the ends of structural members are examined. The functionality of the table is as follows:

Limit state		Extreme	values	Sort		E	Bar end				
Ultimate		Absolute	Maximum	▼ For I	Dars	•	A, B separately	•			
📝 If the dom	inant value was	0, the table does	NOT include it.	🕅 S	ieparate rows						
Designation	Section	Dominant	N [kN]	Vy[kN]	V _z [kN]	M ₈ [kNm	My [kNm]	M ₂ [kNm]	B [kNm²]	Load combination	Â
648 A end	IPE 400	My	104,8	-0,1	27,7	0,1	-223,3	-0,4	0,2	Teherkombináció-	
648 B end	IPE 400	My	-4,9	-0,1	156,3	0,1	505,5	0,4	0,0	Teherkombináció-	-
649 A end	IPE 400	My	23,6	0,2	-150,2	-0,1	329,4	0,6	-0,1	Teherkombináció-	
649 B end	IPE 400	My	103,6	0,1	-31,8	-0,1	-223,3	-0,4	0,2	Teherkombináció-	
650 A end	IPE 400	My	5,9	-0,4	26,8	0,1	-150,5	-0,2	0,1	Teherkombináció-	
650 B end	IPE 400	My	-36,3	0,7	139,6	0,0	446,0	-0,7	-0,1	Teherkombináció-	
651 A end	IPE 400	My	-29,7	-0,1	-135,5	-0,1	293,7	-0,3	-0,1	Teherkombináció-	
651 B end	IPE 400	My	18,9	0,3	19,3	-0,1	-150,4	-0,2	0,1	Teherkombináció-	
533 B end	HEA 300	My	-111,1	-0,4	36,1	0,0	131,8	0,1	-0,1	Teherkombináció-	
786 A end	HEB 340	My	-90,5	0,3	-38,0	0,0	-226,7	0,2	0,0	Teherkombináció-	
786 B end	HEB 340	My	-80,0	1,0	-0,9	0,0	-322,9	-2,3	-0,2	Teherkombináció-	
788 A end	HEB 340	My	-280,3	0,4	-54,8	0,0	-274,1	0,6	0,0	Teherkombináció-	
788 B end	HEB 340	My	-109,1	-0,3	-59,4	0,4	-238,3	-0,3	-0,3	Teherkombináció-	
790 A end	HEB 300	My	-184,5	0,8	-38,7	0,0	-190,6	0,9	-0,1	Teherkombináció-	
790 B end	HEB 300	My	-172,4	0,5	-21,9	-0,1	-337,4	-0,2	0,0	Teherkombináció-	
792 A end	HEB 300	My	-171,9	-0,5	-22,1	-0,1	-336,8	-0,2	0,0	Teherkombináció-	
792 B end	HEB 300	Mg	-168,1	-0,1	-8,4	0,2	-397,9	-0,4	-0,2	Teherkombináció-	
534 B end	HEA 300	My	-124,4	-0,6	-46,0	0,0	-206,2	0,1	0,1	Teherkombináció-	
535 B end	HEA 300	My	-239,4	-0,3	87,9	0,0	328,7	0,0	0,0	Teherkombináció-	
526 D and	LIEA 200	м	- 250 2	-0.6	.81 1	0.0	-366.1	0.0	0.0	Teherkomhináció-	-

Limit state: selection of appropriate load combinations belonging to resistance or serviceability limit states (the deformations are automatically appear for the serviceability limit state, while the internal forces and reactions for the resistance limit state)



Extreme values: setting the type of extreme values

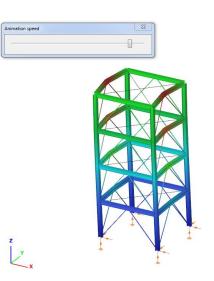
- Sort: sorting the values for bars (structural members), bar groups or cross sections
- **Bar end:** handle the ends of the members separately (A, B ends) or together (Both ends)

Single or multiple component selection is possible, the dominant values are always highlighted by colored cells, the 'Dominant' column contains the dominant components.

Limit state Ultimate					par sections		ar end 30th ends	T	
Designation	Dominant	N [kN]		Vz[kN]	M ₈ [kNm]	My [kNm]	Mz [kNm]	B [kNm²]	Load combination
CHS 114.3x3.6	Vz	0,2	0,0	-0,4	0,0	0,0	0,0	0,0	Teherkombináció
IPE 400	Vz	-152,4	-0,2	-205,3	0,0	304,7	0,0	0,0	Teherkombináció
IPE 400	My	-39,8	0,2	17,8	-0,1	-338,0	-0,5	0,2	Teherkombináció
KA_20	Vz	0,0	0,0	-0,1	0,0	0,0	0,0	0,0	Teherkombináció
HEA 300	Vz	-245,0	0,6	-93,3	0,0	0,0	0,0	0,0	Teherkombináció
HEA 300	My	-239,2	-0,6	-81,1	0,0	-366,1	0,0	0,0	Teherkombináció
HEB 340	Vz	-300,5	0,3	-94,2	0,0	0,0	0,0	0,0	Teherkombináció
HEB 340	My	-222,1	-0,2	-36,4	0,0	-530,2	0,1	0,1	Teherkombináció
HEB 300	Vz	-204,6	0,2	-53,3	0,1	0,0	0,0	0,0	Teherkombináció
HEB 300	Mų	-168,1	-0,1	-8,4	0,2	-397,9	-0,4	-0.2	Teherkombináció

8.6.6 ANIMATION OF DYNAMIC SHAPES

In case of vibration analysis, the animation of the dynamic shapes can be reached to click on the reached to click on the reached to click on the reached with the slide bar.





9 STANDARD DESIGN

The design functionalities are placed on three tabs: the *GLOBAL CHECKS*, *MEMBER CHECKS* and *SERVICEABILITY CHECKS* tab.

9.1 BASICS

The required code checks of the different types of structural components – cross sections, members – can be conveniently evaluated based on the following standard:

Design of steel structures:

- EuroCode
 - EN 1993-1-1: General rules and rules for buildings
 - EN 1993-1-2: General rules. Structural fire design
 - EN 1993-1-5: Plated structural elements (particular)
- Spanish standards
 - Structural Steel Code (EAE)

Design of composite steel and concrete structures:

- EuroCode
 - EN 1994-1-1: General rules and rules for buildings

Design of concrete structures:

- EuroCode
 - EN 1992-1-1: General rules and rules for buildings

Several National Annexes are implemented and user annex can also be defined and saved.

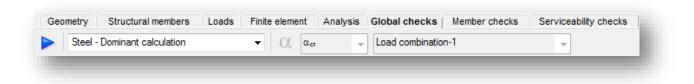


9.2 STEEL DESIGN

Steel ultimate state design related functions can be found on the *GLOBAL CHECKS* and the *MEMBER CHECKS* tabs.

9.2.1 GLOBAL CHECKS

9.2.1.1 BASICS



Cross-section and global buckling checks (according to the Section 6.3.4 (General method) of EN 1993-1-1) which are independent of any specific member parameter can be fully automatically evaluated on the *GLOBAL CHECKS* tab.

As basic settings the analysis result type (first or second order) should be selected and the load combinations which should be evaluated.

9.2.1.2 CROSS SECTION CHECK

el - 🔻
- 1

Cross-section check can be done according to the EN 1993-1-1 6.2 or EN 1993-1-2 4.2 (if the combination contains fire effect).

Check can be performed for the whole model or partial model.

Used safety factor can be selected. If it is necessary γ_{M1} can be used instead of γ_{M0} by clicking on the checkbox.

9.2.1.3 STABLE LENGTH CHECK

According to the Eurocode (1993-1-1 BB3), stable length check can be defined using the results of a plastic hinge analysis. During the calculation, the distance between the plastic hinge and the nearest lateral (Lub) or torsional restraint (Lm, Ls) is being compared with the calculated stable length of the corresponding plastic segment. If the stable length of a plastic segment is



bigger than the distance between the plastic hinge and the lateral/torsional restraint, lateral torsional buckling of the member can be neglected.

During the stable length check, the following checks are performed by ConSteel:

- 1. Automatic recognition of the plastic hinges on the structure
- 2. Searching for the nearest supports of each plastic hinges
 - a. Lateral restraint
 - b. Torsional restraint
- 3. Defining the distance between the plastic hinge and the nearest supports
- 4. Calculating the stable length (depending on the type of the support) for the plastic segments, between the plastic hinge and the nearest supports

On the elastic segments global stability resistance will be evaluated according to the General method of Eurocode.

9.2.1.4 BUCKLING CHECK

Buckling check can be evaluated for steel structures according to the EN 1993-1-1 6.3.4.

$$\frac{\chi_{op}\alpha_{ult,k}}{\gamma_{M1}} \ge 1,0$$

To run the buckling check, it is required to have buckling analysis results.

Model portion:	- Whole model - 🔻
lastic critical factor $\alpha_{cr,op}$	Ultimate resistance factor $\alpha_{ult,k}$
First buckling eigenvalue	Image: minimum value by members
Selected eigenvalue	minimum value in model portion
\bigcirc Automatic (based on sensitivity analysis)	
Reduction factor χ_{op}	
a) minimum of (χ; χ_{LT})	
(γ) b) interpolated between (γ;γιτ)	

For the buckling check the following parameters can be set:

Elastic critical factor α_{cr,op}: eigenvalue of the buckling analysis. It can be the first or the selected eigenvalue according to the design situation or can be automatically selected for each members based on the results of the sensitivity analysis.
 Eigenvalue can be selected on the *ANALYSIS* tab to click in the model area with right click.

• Ultimate resistance factor $\alpha_{ult,k}$: reciprocal of the utilization of the conservative interaction resistance. Value of the utilization can be considered by members or by the model portion.

$$\overline{\lambda}_{\text{op}} = \sqrt{\frac{\alpha_{\text{ult},k}}{\alpha_{\text{cr,op}}}}$$

Reduction factor χ_{op}: reduction factor for the λ_{op} slenderness. It can be determined from the minimum of (χ, χ_{LT}) or from the interpolated value of (χ, χ_{LT}).

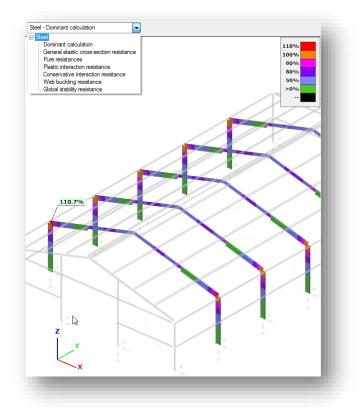
Check can be performed for the whole model or partial model.

9.2.1.5 RESULTS

CONSTEEL

The results of the performed checks are demonstrated with graphical visualization and result tables (for the general functionality of tables used in *ConSteel* see section **1.4** *GENERAL FUNCTIONS FOR TABLES*).

The shown results can be changed in the drop down menu.



When moving the mouse along the structural elements the result markers continuously appear showing the actual utilization.



The detail of the results can be seen in the Section module (see section **10 SECTION** *MODULE*). Section module can be opened directly from the *GLOBAL CHECKS* tab to click on the proper cross section on the model area or in the result table with right click.

9.2.2 MEMBER CHECKS

9.2.2.1 BASICS

Geometry	Structural members	Loads	Finite element	Analysis	Global checks	Member checks	Serviceability checks
🖉 📘	First order	•	Internal forces - N	•	Contour surface		
		-			_		

All steel member design related functions can be found on the **MEMBER DESIGN** tab.

The following parts of the Eurocode 3 (EN 1993-1-1) can be evaluated:

- 6.3.1 Uniform members in compression
- 6.3.2 Uniform members in bending
- 6.3.3 Uniform members in bending and axial compression

9.2.2.2 INDIVIDUAL MEMBER DESIGN

To run steel member design go to **MEMBER CHECKS** tab and select **STEEL** tab in the table at the bottom of the screen. First the members have to be selected and then added to the list. It can be performed by clicking on the **ADD** button. After that one member has to be chosen and **SELECT** button is clicked.

Name	Parameters	State	Validity	Add Add
B1 (selected)	L=10000mm, IPE 600	Unexamined	No	Delete
B5	L=12000mm, IPE 330	Unexamined	No	
B2	L=10000mm, IPE 600	Unexamined	No	= Select
B9	L=6000mm, IPE 330	Unexamined	No	Select (+)
B3	L=10000mm, IPE 600	Unexamined	No	Select (-)
B7	L=6000mm, IPE 330	Unexamined	No	
B4	L=6000mm, IPE 330	Unexamined	No	Document

If a member is selected, the corresponding row highlighted with green and analysis results are automatically loaded. Member design can be run if there is analysis result (first and second order) and cross check for the member. Same like in the analysis the type of the result view can be set by three drop down menus.

	NSTEEĽ					12.0 USER	MANUAL
Geometry	Structural members	Loads	Finite element	Analysis	Global checks	Member checks	Serviceability checks
🖌 [First order	•	Internal forces - N	•	Contour surface	•	. 77

Special analysis can be run by clicking on the second icon (#1). Only one load combination can be selected at the same time. Buckling and lateral torsional buckling analysis is performed just for the selected member. If the normal force is negligible then no buckling analysis can be run. On the basis of these special analysis, *ConSteel* can determine the factors (effective length factors, warping factor etc.) for the buckling and the lateral-torsion buckling design.

То	Name	Number of buckling eigenshapes	Number of LT buckling eigenshapes
~	Load combination-1	5	5
	Load combination-2	5	5
	Load combination-3	5	5
	Load combination-4	5	5
	Serviceability-1	5	5
	Serviceability-2	5	5
	Serviceability-3	5	5
	Serviceability-4	5	5
			Cancel

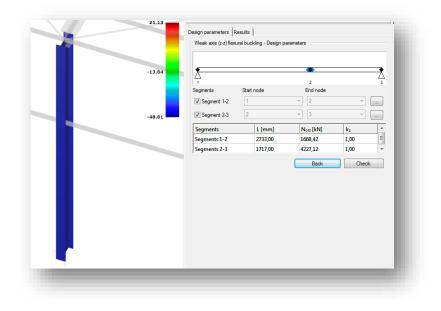
The next step is to choose the load combination and design type from the options: pure cases (strong axis buckling, weak axis buckling, lateral-torsional buckling) and interaction stability (interaction of buckling and lateral torsional buckling, interaction of buckling and bending, interaction of lateral-torsional buckling and bending). The dominant load combination is automatically selected and marked with a * symbol.

Design parameters Results	
Load combinations:	Load combination-2*
Pure cases	
Buckling	
✓ Strong axis buckling	Ig
✓ Weak axis bucklin	g
 Lateral-torsional buckling 	ng
Interaction stability	
Interaction of buckling and lateral-torsic (members susceptible to torsional defor	mations)
N _b + M _{b,y}	
N _b + M _{b,y} + M _{c,z}	
Interaction of buckling and bending (members not susceptible to torsional d	eformations)
N _b + M _{c,y}	
N _b + M _{c,a}	
N _b + M _{c,y} + M _{c,a}	
Interaction of lateral-torsional buckling	and bending
M _{b,y} + M _{c,z}	
	Next

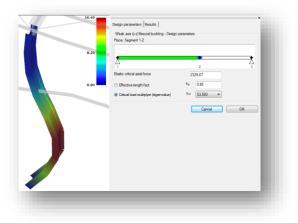


Depending on the member not all checks can be performed. If the normal force is negligible then strong or weak axis buckling design cannot be run.

If **strong or weak axis buckling option** is selected then first the design parameters has to be set. The program automatically identifies the supports which could have effect on the buckling check.



Among the supports the member is divided to segments. Segments can also be turned off in order not to calculate them during the analysis. The design parameters can be individually set for each segment by clicking on the three dots icon.



The actual segment is highlighted with green. There are two options to set the design parameters: set the effective length factor manually by typing in the appropriate value or by choosing the right critical load multiplier. The second option is possible if the special analysis has already been run. In both cases the elastic critical axial force is automatically calculated.



If all the design parameters has been properly set then design check can be performed by clicking on the *CHECK* button. Results will be displayed.

Number of the dominant finite element: 1 Place of the dominant FE node: j Class of the dominant rcoss section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 43.5 kN Ned 964,3 kN Nor 2229,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	roblems						
Land Combination 2 Buckling case Examined system length(s) Examined system length(s) Yeak axis buckling Summary of results Used capacity in lateral buckling: Diace of the dominant cross section for compres I used part of standard: 6.3.1(6.46-6.49) Detailed calculation Ned 43,5 kN Ned 964,3 kN Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	R13						
Buckling case Case 1 Examined system length(e)	ation Load combinat	2					
Examined system length(6) Image: system length(6)		-2					
Weak axis buckling Summary of results Used capacity in lateral buckling: 4,5% Place of the dominant cross section: 0 mm from the fir Number of the dominant finite element: 1 Place of the dominant FE node: 1 Place of the dominant FE node: 1 Class of the dominant FE node: 1 Class of the dominant FE node: 1 Detailed calculation Ned Nead 964,3 kN Nor 2733 mm k 0,737 α 0,340 Φ	e Case 1						
Δ 2 Weak axis buckling 4,5% Summary of results 0 mm from the fir Number of the dominant cross section: 0 mm from the fir Number of the dominant FE node: 1 Class of the dominant FE node: 1 Class of the dominant FE node: 1 Detailed calculation 6.3.1(6.46-6.49) Detailed calculation 1 Nes 43,5 kN NesA 964,3 kN Nor 22329,9 kN L 2733 mm k 0,737 α 0,340 Φ 0,863	stem length(s)						
Δ 2 Weak axis buckling 4,5% Summary of results 0 mm from the fir Number of the dominant cross section: 0 mm from the fir Number of the dominant FE node: 1 Class of the dominant FE node: 1 Class of the dominant FE node: 1 Detailed calculation 6.3.1(6.46-6.49) Detailed calculation 1 Nes 43,5 kN NesA 964,3 kN Nor 22329,9 kN L 2733 mm k 0,737 α 0,340 Φ 0,863	2.07						
Δ 2 Weak axis buckling 4,5% Summary of results 0 mm from the fir Number of the dominant cross section: 0 mm from the fir Number of the dominant FE node: 1 Class of the dominant FE node: 1 Class of the dominant FE node: 1 Detailed calculation 6.3.1(6.46-6.49) Detailed calculation 1 Nes 43,5 kN NesA 964,3 kN Nor 22329,9 kN L 2733 mm k 0,737 α 0,340 Φ 0,863							
Weak axis buckling Summary of results Used capacity in lateral buckling: Place of the dominant cross section: 0 mm from the fir Number of the dominant finite element: 1 Place of the dominant finite element: 1 Class of the dominant FE node: 1 Class of the dominant FE node: 1 Obstaled calculation Ned NeA 964,3 kN Ner 2733 mm k 0,737 α 0,340							
Summary of results 4,5% Used capacity in lateral buckling: 4,5% Place of the dominant cross section: 0 mm from the fir Number of the dominant finite element: 1 Place of the dominant FE node: i Class of the dominant rcross section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation Nea NEa 43,5 kN Nea 964,3 kN Nor 2232,9 kN L 2733 mm k 0,737 α 0,340 Φ 0,863		2					
Used capacity in lateral buckling: 4,5% Place of the dominant cross section: 0 mm from the fir Number of the dominant finite element: 1 Place of the dominant finite element: 1 Class of the dominant Fire node: 1 Class of the dominant cross section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 43,5 kN Ne ₂ A 964,3 kN Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,863	Weak axis buckling						
Place of the dominant cross section: 0 mm from the fir Number of the dominant finite element: 1 Place of the dominant finite element: 1 Class of the dominant FE node: j Class of the dominant cross section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 1 NEd 43,5 kN Ned 2329,9 kN L 2733 mm k 0,846 λ 0,737 α. 0,340 Φ 0,863	y of results						
Number of the dominant finite element: 1 Place of the dominant FE node: j Class of the dominant rcss section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 43,5 kN Ned 964,3 kN Nor 2229,9 kN L 2733 mm k 0,846 λ 0,737 α 0,840 Φ 0,863	capacity in lateral buckling:	4,5%					
Place of the dominant FE node: j Class of the dominant cross section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 43,5 kN NEd 43,5 kN Nerd 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	of the dominant cross sect	a: 0 mm from the first ne					
Class of the dominant cross section for compres 2 Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 43,5 kN Nb.Fad 964,3 kN Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	er of the dominant finite e	nent: 1					
Used part of standard: 6.3.1(6.46-6.49) Detailed calculation 43,5 kN Nead 43,5 kN Nb,Fid 964,3 kN L 2733 mm k 0,846 λ 0,737 α. 0,340 Φ 0,863	of the dominant FE node:	j					
Detailed calculation 43,5 kN NEd 43,5 kN Nb,Rd 964,3 kN Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	of the dominant cross sect	for compress 2					
NEd 43,5 kN Nb,Rd 964,3 kN Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	part of standard:	6.3.1(6.46-6.49) form					
Nb,Rd 964,3 kN Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863	calculation						
Nor 2329,9 kN L 2733 mm k 0,846 λ 0,737 α. 0,340 Φ 0,863		43,5 kN					
L 2733 mm k 0,846 λ 0,737 α 0,340 Φ 0,863		964,3 kN					
k 0,846 λ 0,737 α 0,340 Φ 0,863		2329,9 kN					
λ 0,737 α 0,340 Φ 0,863		2733 mm					
α 0,340 Φ 0,863		0,846					
Ф 0,863		0,737					
		0,340					
0.702		0,863					
χ 0,763		0,763					
A 5381,2 mm ²		5381,2 mm ²					
fy 235,0 N/mm ²		235,0 N/mm ²					

There are three drop down menus in order to view different results. The first one can be used to change member, the second one is to change load combinations and the third one is to change the buckling case. The corresponding case is highlighted with green in the image. A short explanation can be read in the bottom when any of the rows is selected.

Lateral torsional buckling design is more or less the same as the buckling design in *ConSteel* usage, there are only small differences. New cases can be defined and unnecessary cases can be deleted. For each segment start and end node can be set.

By clicking on the three dots button the design parameters can be altered. The elastic critical bending moment can be calculated from the analytic formula or from the critical load

3.67	<u>×</u>				2					2
	New case	D	elete uncheck	ed cas						
	Segments	Start nor	de		End no	ode				- 1
0.00	Segment 1-2	1		•	2				•	
	Segment 2-3	2		•	3				•	
	Segments	L [mm]	Mer [kNm]	k ₂	kw	C ₁	C ₂	C 3	Zg	<u>^</u>
	Segments 1-2	2733,00	548,96	1,00	1,00	1,88	1,00	0,94	0,00	E
	Segments 2-3	1717,00	805,51	1,00	1,00	1,21	1,00	1,00	0,00	*
					Ba	sk –		Ch	eck	



multiplier. When the first method is used the factors can be typed in or a special automatic C factor calculator can be used by clicking on the **AUTOC** button. The second option is to choose the corresponding critical load multiplier from the list.

		•	<u> </u>
1		2	3
astic critical bending moment [kNm]	618,31		
Analytic formula			
Effective length factor	k _a	1,30	
Warping factor	kw	0,84	
Cifactors	C ₁	2,44	
	C2	1.00	
AutoC	C₃	0,21	
Eccentricity of load [mm]	Zq	0,00	
Critical load multiplyer (eigenvalue)	0.cr		

Interaction stability design goes through the steps of the pure cases.

After the check is made the state of the member in the list at the bottom changes to 'Examined' from 'Unexamined'.

9.2.2.3 MEMBER GROUP DESIGN

By using the **SELECT** (+) button it is possible to select more members for the member design. **SELECT** (-) can be used for remove members from the selection. Member group design works nearly the same as individual member design there are just minor differences. No special analysis can be run in member group design therefore the critical load factor cannot be selected.

If the members are selected then the next step is to select the design type and set the design parameters, then click *CHECK* button similar to the individual member design.



-	Load combinations	
ŧ	✓ Load combination-1	
Ŧ	 Load combination-2 	
Ŧ	Load combination-3	
ŧ	 Load combination-4 	
Ŧ	 Load combination-5 	
ŧ	 Load combination-6 	
		Back Next

C k	mber	R6 (57,76 %)		-		
.0	ad combination	Load combination-2		•		
3 u	ckling case	Case 2		•		
-	camined system length(s)					
	Strong axis buckling			-		
1	strong axis booking					
1				t		
	ateral-torsional buckling		-	2		
1	aterat-torsional booking					
1	<u></u>	٠		t		
-	-	2	-	3		
-	Summary of results	1		^		
	Used capacity		57,8%			
	Used part of standard	:	6.3.3 (6.61-6.62) form			
	Strong axis buckling					
	Summary of results					
	Used capacity in la	6,2%				
	Place of the domin	0 mm from the first no				
	Number of the do	1				
	Place of the domin	j				
	Class of the dominant cross section for compress 2					
	Used part of stand	ard:	6.3.1(6.46-6.49) form			
	Detailed calculation					
		Lateral-torsional buckling				
-	Summary of results Used capacity in lateral-torsional buckling: 50,2%					
	Place of the domin		50,2% 4800 mm from the firs			
		minant finite element:	8			
	Place of the domin		k			
		ant cross section for compress				
	Used part of stand		6.3.2(6.54-6.56) form			
ŧ	Detailed calculation					
	Results of minor a	is bending				
	Capacity of minor axi	-	4,5%			
	Place of the dominan		4800 mm from the firs			
	Number of the domin	ant finite element:	8 mm from the first no	μ		
	Place of the dominan	t FE node:	1			
	Class of the dominant	cross section for bending:	1			
	Used part of standard	:	6.2.5 (6.12-6.15) form			
	Mz,Ed		1,3 kNm			
	Maja,Rid		28,3 kNm			
	Wpta		5381,2 mm ³			
-	Interaction factors			-		

For each load combination member design can be turned off. It is important to know that not all the design options can be run for all members in load combinations. It depends on the analysis result.

In the results tab the dominant member is automatically shown. The number next to the member number shows the capacity.



9.3 Composite design

Composite column design related functions can be found on the **GLOBAL CHECKS** tab, beam design related function can be found on the **MEMBER CHECKS** tab.

9.3.1 Composite column design

Cross-section of the composite columns can be evaluated on the *GLOBAL CHECKS* tab by clicking on the blue arrow. On the *DESIGN...* dialog the *Composite column design* settings has to be chosen by the check box. Checks can be performed according to the first or the second order analysis and for the whole or a previously created portion model.

Name							L. F	¢	-
comp Fully end	ased I								
Reinforcement		Concrete section			a	a Vpper bars			
Diameter:	16 🔻	mm	bc:	400	mm	+	•	 † [
Material: B 500 A 🔹			hc:	hc: 400 mm			Leftbars		
lpper bars:	4	pcs	Material:			Y	Y hc		
ower bars:	4	pcs	C20/2	25 EN 1993	2-1-1:2010 🔻				
eft bars:	4	pcs	Steel :	section				Rightbars	
Right bars:4pcsCover - a:20mm			Name			+		i	
			HEA 200 🔻			a†	a Z Lower bars		
							∐a ∎		
Modulus of elas	ticity applied	d in stru	tural ana	alysis					
Secant modulus of elasticity						$E_{cm} = 22(f_{cm}/10)^{0.3} = 29962,0$			
O Design value of elastic modulus						$E_{cd} = E_{cm}$	/c= 19974,7	7	
V Special calcu	lation of the	e effecti	ve flexur	al stiffness	taking into consi	ideration the	shrinkage,	creep and cracking	
Apply L/200	initial sway	in Y dire	ction						
Apply L/150	initial sway	in Z dire	ction						
						<< Previous	Clos	e Create	

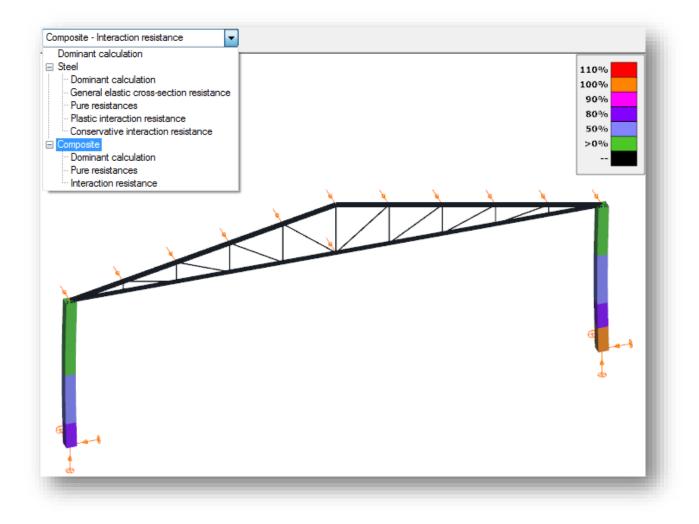
If the initial sways in the y and z directions were applied on the checked cross-section then the stability check of the composite columns can be neglected. Cross-section checks are enough.



9.3.1.1 RESULTS

The results of the performed checks are demonstrated with graphical visualization and result tables (for the general functionality of tables used in *ConSteel* see section **1.4** *GENERAL FUNCTIONS FOR TABLES*).

The shown results can be changed in the drop down menu.



When moving the mouse along the structural elements the result markers continuously appear showing the actual utilization.

The detail of the results can be seen in the Section module (see *CHAPTER 10 SECTION MODULE*). Section module can be opened directly from the *GLOBAL CHECKS* tab to click on the proper cross section on the model area or in the result table with right click and select the *CALCULATE SECTION* menu.



9.3.2 COMPOSITE BEAM DESIGN

To run composite beam design go to **MEMBER CHECKS** tab and select **COMPOSITE BEAM** tab in the table at the bottom of the screen. Then select the composite beams and click on **ADD** button. Now the selected beams are added to the table. Select the beam(s) from the list which you would like to design and click on **SELECT** (**SELECT+**) button. You can select which load combinations and first or second order results to use for the design. Then click on the **CHECK** button and the design will be performed.

Composite beam design in *ConSteel* is carried out according to EN 1994-1-1:2010 standard.

The design bending resistance is determined using plastic theory according to 6.2.1.2 or 6.2.1.3 where the cross-section is in Class 1 or 2. The section class is determined acc. to 5.5.2. Class 3 and 4 cross-sections are not covered by *ConSteel* composite beam design. The composite beam is checked for plastic bending, vertical shear, and shear buckling, crushing of the concrete flange and longitudinal shear at all the critical cross-sections. The critical cross sections are determined according to 6.1.1. (4). The following methods are not included in *ConSteel*: non-linear resistance to bending (6.2.1.4.) and elastic resistance to bending (6.2.1.5.). The profiled steel decking is assumed not to contribute to the resistance.

9.3.2.1 ANALYSIS MODEL

In the **analysis model** the calculation of the effective width of concrete flange is based on the method defined in a previous chapter (in the dependence of the span and the distance between the shear studs in the cross section). Based on practice experiences, composite beams are usually created as two-pinned one span beams. ConSteel can design composite beams with one span formation. In case of multi span beams, and where the tension is in the concrete flange, design can not be performed.

In the **ANALYSIS** tab the analysis model is displayed.

9.3.2.2 DESIGN OF SHEAR STUDS

The allocation of the studs is plastic based, where the studs has to be distributed uniformly in both directions from the dominant cross section. The dominant cross section, where the bending is maximal, is located automatically.

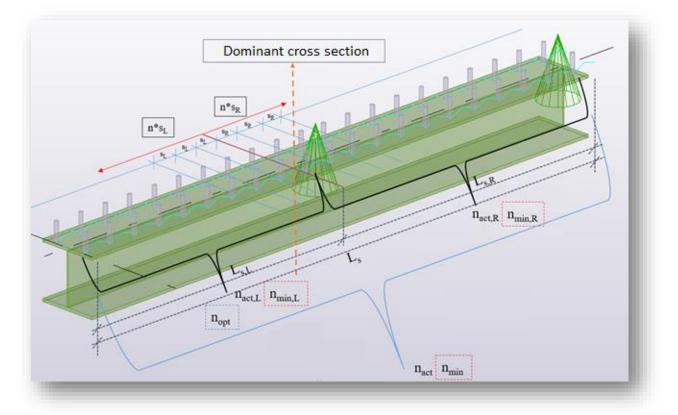
If the number of shear studs is desired to be calculated automatically, ConSteel willcalculate the bending resistance of the beam with the minimal required number of studs. ConSteel will increase than the number of shear studs, until the bending resistance exceeds the acting bending on the beam. The green section on the schematic figure below shows, where the optimal distribution of shear studs is calculated. The following parameters are evaluated for the examined part of the beam:

CONSTEEL

 n_{opt} : optimal number of shear stud positions along the dominant phase of the beam n_{act} : number of shear stud positions along the examined member

 s_L : shear stud distribution in the left direction from the dominant cross section [mm] $n_{act,L}$:number of shear stud positions in the left direction from the dominant cross section

 s_{R} : shear stud distribution in the right direction from the dominant cross section [mm] $n_{act,R}$: number of shear stud positions in the right direction from the dominant cross section



n_{stud}: number of shear studs in one cross section

n_{min}: minimum number of shear stud positions along the member

 $n_{\text{min},\text{L}}$: minimal number of shear stud positions in the left direction from the dominant cross section

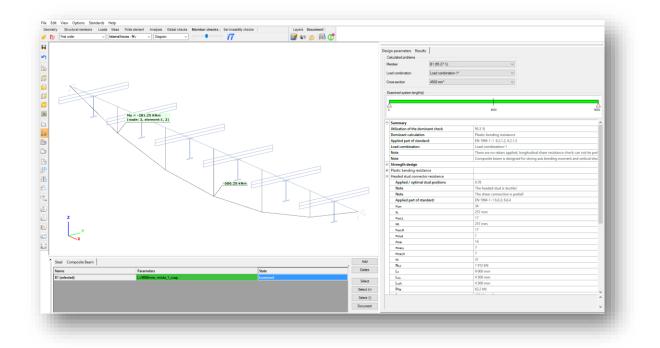
 $n_{\mbox{\scriptsize min,R}}$: minimal number of shear stud positions in the left direction from the dominant cross section

Instead of using the automatic evaluation of the number of the shear studs, manual definition of the number of studs is available too. In this case, the software checks, if the distribution fits



to the detailing rules, such as the minimal and the maximal distance between the studs. The number of the studs has to be given for the whole length of the beam. During the check of the beam, ConSteel will uniformly distribute the studs along the member (In case of automatic evaluation of required shear stud numbers, non-uniform distribution is also possible).

Calculation of the utilizations of the shear studs are evaluated by the dividing the applied number of shear studs with the optimal number of shear studs (n_{act} / n_{opt})



9.3.2.3 Shear buckling resistance of composite beams

The shear buckling resistance of the web is determined in accordance with section 5 and Appendix A of EN 1993-1-5:2006. The shear resistance contribution of the flanges is ignored.

9.3.2.4 BENDING AND VERTICAL SHEAR RESISTANCE

The influence of vertical shear on the bending resistance is ignored if the vertical shear force is less than the half of the shear resistance.

For Class 1 or 2 cross-sections, if the vertical shear force exceeds the half of the shear resistance, the reduced design steel strength is used to calculated the bending strength of the member as defined in 6.2.2.4 (2)

9.3.2.5 LATERAL-TORSIONAL BUCKLING RESISTANCE OF COMPOSITE BEAMS

Lateral-torsional buckling of composite beams is not included.

9.3.2.6 Sheeting with RIBS parallel to the supporting beam



The design shear resistance of studs is taken as the resistance of the studs in solid slab multiplied by the reduction factor as defined in 6.6.4.1 (2).

If the sheeting is discontinuous ConSteel assumes that it is welded to the flange of the beam but the distance between two ribs ('a' parameter) can be set.

9.3.2.7 Sheeting with ribs perpendicular to the supporting beam

The design shear resistance of studs is taken as the resistance of the stud in the solid slab multiplied by the reduction factor as defined in equation 6.23, limited to the maximum value as defined in Table 6.2. While calculating the resistance of studs in solid slabs the stud ultimate tensile strength is limited to 450 N/mm².



9.4 Reinforced concrete design

Design of concrete members are available at at the **GLOBAL CHECKS** tab. By clicking on the **b** button, the **DESIGN**... dialogue will appear, where below the *Steel* and *Composite design*, the **CONCRETE DESIGN EN 1992-1-1** panel can be found. On the panel, **BEAM REINFORCEMENT**, **COLUMN REINFORCEMENT** and **SLAB REINFORCEMENT** calculations are available. If the necessary input parameters for these calculations are available (cross sections, reinforcement data, design parameters and analysis results) the checkboxes are active, and choosing the desired item, **CALCULATION** can be initiated to obtain utilizations.

oadcombination	s to check			-
First order e	lastic	Second (order elastic	
To calculate	Load combinations		Selected eigenvalue	
>	Loadcombination-1		-	
Steel design EN	1993-1-1			+
	l 1993-1-1 mn check EN 1994-1-1		_	+
	mn check EN 1994-1-1		- Whole model -	
Composite colu Model portio	mn check EN 1994-1-1 n:		- Whole model - V	
Composite colu	mn check EN 1994-1-1 n:		- Whole model - V	
Composite colu Model portio	mn check EN 1994-1-1 n: n EN 1992-1-1		- Whole model - V	-
Composite colu Model portio Concrete desig	mn check EN 1994-1-1 n: n EN 1992-1-1		- Whole model - V	-
Composite colu Model portio Concrete desig Beam reinford Considered	mn check EN 1994-1-1 n: n EN 1992-1-1 cement model portions			-
Composite colu Model portio Concrete desig Beam reinford Considered	mn check EN 1994-1-1 n: n EN 1992-1-1 cement model portions orcement		- Whole model - V	-
Composite colu Model portio Concrete desig Beam reinfor Considered Column reinfor Considered	mn check EN 1994-1-1 n: n EN 1992-1-1 cement model portions orcement model portions			-
Composite colu Model portio Concrete desig Beam reinfor Considered Column reinfor Considered	mn check EN 1994-1-1 n: n EN 1992-1-1 cement model portions orcement model portions ement		- Whole model - V	-
Composite colu Model portio Concrete desig Beam reinfor Considered Column reinfor Considered	mn check EN 1994-1-1 n: n EN 1992-1-1 cement model portions orcement model portions		- Whole model - V	-

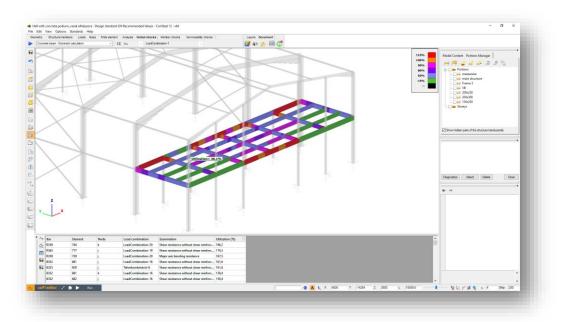
Required input parameters for concrete design are the following (see *CHAPTER 5.2.2 CONCRETE REINFORCED MEMBERS* for details):

- In case of concrete beams and columns
 - Reinforced concrete cross sections with design option (see CHAPTER 5.2.2.1)
 - Beam/column reinforcement object assigned to the concrete member (see *CHAPTER 5.2.2.1.1* and *5.2.2.1.2*)



- First or second order analysis results (see *CHAPTER 8.5.1.1* and *8.5.1.2*)
- In case of reinforced concrete slabs
 - Defined reinforcement schemes in the main directions (see *CHAPTER 5.3.1*)
 - First or second order shell results (see *CHAPTER 8.6.1*)

During the design, **CONSTEEL** will perform the available standard checks according to EN1992-1-1 for all of the cross sections automatically. Cross sections with their dominant utilizations will be collected into the results grid, and will be shown using the color-coded visualization in the modeling area.



Results of the individual standard checks can also be obtained using the dropdown menu in the upper left corner of the design tab:

Dominant calculation	-
Dominant calculation	
Major axis bending resistance	
 Shear resistance without shear reinforcement 	
Resistance of interction of shear and torsion without	ut long
Concrete column	
··· Dominant calculation	
 Resistance of interaction of compression and bend 	ling
Slab reinforcement area	
🖃 at the top - x	
Ø 8,0	
🖃 at the top - y	
Ø 8,0	
at the bottom - x	
Ø 8,0	
at the bottom - y	
Ø 8.0	



9.4.1 Reinforced concrete beam design

In case of concrete beam design, the following checks of **EN1992-1-1** will be performed:

- Longitudinal reinforcement
 - EN1992-1-1 9.2.1.1 (1) (9.1.n) formula Minimum longitudinal reinforcement
 - o EN1992-1-1 9.2.1.1 (3) formula Maximum longitudinal reinforcement
 - EN1992-1-1 6.1 Major axis bending
- Shear reinforcement
 - EN1992-1-1 9.2.2. (6) (9.6.N) formula Maximim stirrup distance
 - EN1992-1-1 9.2.2 (5) (9.5.n) formula Minimum percentage of reinforcement
 - EN1992-1-1 6.2.2. (1) (6.2.a) and (6.2.b) formula Shear resistance without shear reinforcement
 - EN1992-1-1 6.2.3. (3) (6.8) and (6.9) formula Shear resistance with shear reinforcement
- Interaction of shear and torsion
 - EN1992-1-1 6.3.2. (4) (6.29) formula Resistance of inclined compression concrete bars
 - EN1992-1-1 6.3.2. (5) (6.31) formula Required longitudinal torsion reinforcement
 - EN 1992-1-1 6.3.2. (3) (6.28) formula Applied longitudinal torsion reinforcement

9.4.2 REINFORCED CONCRETE COLUMN DESIGN

In case of concrete column design, the following checks of **EN1992-1-1** will be performed:

- Longitudinal reinforcement
 - EN1992-1-1 9.5.2 (1) formula Minimum diameter of longitudinal reinforcement
 - EN1992-1-1 9.5.2 (2) formula Minimum longitudinal reinforcement
 - EN1992-1-1 9.5.2 (3) formula Maximum longitudinal reinforcement
 - EN1992-1-1 5.8.3 and 5.8.8 Biaxial bending
- Shear reinforcement
 - EN1992-1-1 9.5.3 (1) formula Minimum diameter of stirrup
 - EN1992-1-1 9.5.3 (2) formula Maximum distance ot stirrups
 - EN1992-1-1 6.2.2 (1) (6.2.a) and (6.2.b) formula Shear resistance without shear reinforcement

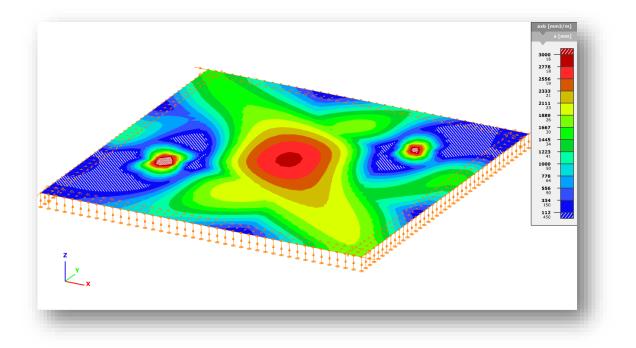


 EN1992-1-1 6.2.3 (3) (6.8) and (6.9) formula – Shear resistance with shear reinforcement

9.4.3 REINFORCED CONCRETE SLAB DESIGN

If there is a concrete slab placed in the model, with applied reinforcement in the main directions (see *CHAPTER 5.3*), the slab reinforcement calculation option will be available in the *DESIGN*... dialogue.

Result of the calculation is the required area of reinforcement [mm2/m] in each of the main directions, both in upper and lower positions. Change between the visualization of the directions is possible with the dropdown menu in the upper left corner. Results are represented with contour surface visualization:



By clicking ont he colour palette with the right mouse button, properties (such as scale division, values of divisions) can be modified:



718	🙆 🔒	Name	Színskála 1	
628	a 🔒	Resulttype	Slab reinforcem	entarea at the
559	🙆 🔒	6 J F		
503	a 🔒	Scale division		19
457	a 🔒	Reinforceme	nt area [mm2/m]	
419	0 💼	Reinforceme	nt distance [mm]	
387	0 💼			
335	0 💼			
296	0			
265	0			
251	0			
223	0			
201	0			
183	0			
168	Ø 🔒			
155	ⓐ ⓐ			
144	ⓐ ⓐ			
126	ⓐ ⓐ		Reset automatic pale	tte
112	ⓐ ⓐ	Save		
		Set as defaul	t palette	Save
			Delete Close	e Apply

The result of the calculation, the contour surface with the colour palette itself can be exported into dxf form from *FILE/EXPORT* menu (see *CHAPTER 2.6*).

9.5 Serviceability checks

9.5.1 BASICS

All serviceability related functions can be found on the **SERVICEABILITY** tab. By clicking on the blue arrow icon a new dialog appears. The previously defined serviceability checks can be found and it is also possible to open or delete the selected checks.

Global vertica Clocal deflection	ntal deflection check (42,18%) il deflection check (58,82%) on check by members (16,06%) ction check for 2 points (12,19%)	
<u> </u>		

In order to create a new serviceability check **New CHECK** button is clicked. The following checks can be performed:

- Global horizontal deflection check
- Global vertical deflection check
- Local deflection check by members
- Relative deflection check for 2 points



$9.5.2 \; \text{Global horizontal deflection check}$

Global horizontal deflection check can be used to check the biggest horizontal deflection in the model or in the model portion. First or Second order analysis can be selected just like the serviceability load combinations that need to be taken into account.

Reference height can be typed in or two points can be selected and program calculates the height distance between them. Check is done after clicking the *CALCULATION* button. The dominant bar is automatically selected in the model to make the interpretation easier.

9.5.3 GLOBAL VERTICAL DEFLECTION CHECK

Global vertical deflection check works similar to the global horizontal deflection check. There is just minor difference which is the following: reference length has to be set instead of reference height. This can be set manually or by clicking two points on the model.

bal horizontal deflect	ion check				×
Check name:	Global horizo	ntal deflection	check		
Type of design					
First order analysis		© Ser	cond orde	r analysis	
To cal Load com	binations				
	bination-2				
-	bination-3 bination-4				
-	ibination-4 ibination-5				
-	bination-1				
Parameters					
Model portion:		- W	/hole mode	el - 🔻	
Reference height H [r	nm]:	60			
First structural point:		X: 0 Y: 0 Z:			
Second structural poi	nt: 🔭	X: 0 Y: 0 Z:			
Deflection limit:		H/ 10	0		
			Calcu	ulation	
Results					
Dominant load combin	ation:	Load	d combinat		
Bar name:				B1	
Finite element point: Dominant horizontal d	affaction [m-1			2 252,575	
Dominant norizontal d	enection (mm)			252,575 60	
Capacity [%]:				420,958	_
			Mégse		ж
			Mégse		ж
bal vertical deflection) check		Mégse		ж
bal vertical deflection	check Global vertica	al deflection ch			
Check name:		al deflection ch			
	Global vertica				
Check name: Type of design	Global vertica		neck		
Check name: Type of design First order analysis To cal Load com	Global vertica binations		neck	r analysis	
Check name: Type of design First order analysis To cal Load com	Global vertica binations bination-1 bination-2		neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com	Global vertica binations		neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com Load com	Global vertica binations bination-1 bination-2 bination-3		neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com Load com	Global vertica binations bination-1 bination-2 bination-3 bination-4		neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com Load com	Global vertica binations bination-1 bination-2 bination-3 bination-4		neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com Load com	Global vertica binations bination-1 bination-2 bination-3 bination-4		neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com	Global vertica binations bination-1 bination-2 bination-3 bination-4		neck	r analysis	
Check name: Type of design First order analysis To calue. Load com Load com Load com Load com Load com	Global vertica binations bination-1 bination-2 bination-3 bination-4	@ Ser	neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com	Global vertice	@ Ser	neck	r analysis	
Check name: Type of design First order analysis To cal Load com Load com Load com Load com Load com Load com Parameters Model portion:	Global vertic.	@ Sec	neck cond order	r analysis	
Check name: Type of design First order analysis First order analysis Load com Load com Load com Load com Parameters Model portion: Reference length L [n	Global vertic. binations bination-1 bination-2 bination-3 bination-3 bination-3 bination-5	• Sec • W 700	hole mode	r analysis 	
Check name: Type of design First order analysis First order analysis Load com Load com Load com Check name Parameters Reference length L [n First structural point:	Global vertic. binations bination-1 bination-2 bination-3 bination-3 bination-5 wm]:	© Sec 	hele mode 00 00 0 2 2: 6000	r analysis 	
Check name: Type of design First order analysis Grid Load com Load com Load com Load com Load com Load com Adde partion: Reference length L (In First structural point: Second structural point:	Global vertic. binations bination-1 bination-2 bination-3 bination-3 bination-5 wm]:	© Sec 	hole mode 00 6000 0	r analysis Δ	
Check name: Type of design First order analysis To cal Load corr Load corr Load corr Load corr Load corr Load corr Analysis Parameters Model portion: Reference length L [n First structural point: Second structural point: Second structural point: Second structural point:	Global vertic. binations bination-1 bination-2 bination-3 bination-3 bination-5 wm]:	© Sec 	hole mode 00 6000 0	r analysis 	
Check name: Type of design First order analysis Grid Load com Load com Load com Load com Load com Load com Adde partion: Reference length L (In First structural point: Second structural point:	Global vertic. binations bination-1 bination-2 bination-3 bination-3 bination-5 wm]:	© Sec 	hole mode 00 6000 0	r analysis Δ	
Check name: Type of design First order analysis To cal Load corr Load corr Load corr Load corr Load corr Check nameters Model portion: Reference length L [n First structural point: Second structural point: Seco	Global vertic. bibinations bination-1 bination-2 bination-3 bination-3 bination-3 bination-5 m]:	© Set 	hole mode 00 6000 0	r analysis 2 al- • al- • ion-2	
Check name: Type of design © First order analysise © First order analysise Load com © Load com Exercise Reference length L [m First structural point: Second structural point: Secon	Global vertic. bibinations bination-1 bination-2 bination-3 bination-3 bination-3 bination-5 m]:	© Set 	hele mode bele mode 0 0 0 0 0 0 0 0 0 0 0 0 0	r analysis 2 al- al-	
Check name: Type of design.	Global vertic. binations bination-1 bination-2 bination-3 bination-4 bination-5 m]: m]:	© Set 	hele mode bele mode 0 0 0 0 0 0 0 0 0 0 0 0 0	r analysis <u>2</u> <u>2</u> <u>2</u> <u>4</u> <u>4</u> <u>4</u> <u>4</u> <u>4</u> <u>4</u> <u>4</u> <u>4</u>	
Check name: Type of design First order analysis First order analysis Load com Load com Load com Load com First Load com First structural point: Second structural po	Global vertic. binations bination-1 bination-2 bination-3 bination-4 bination-5 m]: m]:	© Set 	hele mode bele mode 0 0 0 0 0 0 0 0 0 0 0 0 0	r analysis	
Check name: Type of design First order analysis First order analysis Load com Load com Load com Load com Parameters Model portion: Reference length L [n First structural point: Second structural point: Second structural point: Dominant load combin Bar name: Finite element point: Dominant vertical defi Deflection limit (rm):	Global vertic. binations bination-1 bination-2 bination-3 bination-4 bination-5 m]: m]:	© Set 	hele mode bele mode 0 0 0 0 0 0 0 0 0 0 0 0 0	r analysis () al- v alation ion-2 B2 3 291,489 70	
Check name: Type of design First order analysis First order analysis Load com Load com Load com Load com First Load com First structural point: Second structural po	Global vertic. binations bination-1 bination-2 bination-3 bination-4 bination-5 m]: m]:	© Set 	hele mode bele mode 0 0 0 0 0 0 0 0 0 0 0 0 0	r analysis	
Check name: Type of design First order analysis First order analysis Load com Load com Load com Load com Parameters Model portion: Reference length L [n First structural point: Second structural point: Second structural point: Dominant load combin Bar name: Finite element point: Dominant vertical defi Deflection limit (rm):	Global vertic. binations bination-1 bination-2 bination-3 bination-4 bination-5 m]: m]:	© Set 	hele mode bele mode 0 0 0 0 0 0 0 0 0 0 0 0 0	r analysis (al	



9.5.4 LOCAL DEFLECTION CHECK BY MEMBERS

Local deflection check by members goes through all the beams in the model, and checks the local deflection, and then selects the maximum. This check is suitable only for horizontal beams i.e. members in XY plane.

Local deflection check by members is not adequate for cantilever beams just for beams which are supported by two other beams or supports in both ends.

9.5.5 Relative deflection check for 2 Points

Relative deflection check for 2 points can be used for two clicked points. Relative horizontal or vertical check can be selected and performed.

ocal deflection check by members	X
Check name: Local deflection check l	oy members
Type of design	
First order analysis	Second order analysis
To cal Load combinations	
Load combination-2	
Load combination-3 Load combination-4	
Load combination-4	
Load combination-1	
Parameters	
Model portion:	- Whole model - 🔻
Deflection limit: L /	100 Calculation
	Calculation
Results	
	.oad combination-1
Bar name:	B2
Finite element point:	10
Dominant relative vertical deflection [mm]:	0,701502
Deflection limit [mm]:	50
Capacity [%]:	1,403
	Mégse OK
lative deflection check for 2 points	
Check name: Relative deflection check	
Check name: Relative deflection check	k for 2 points
Check name: Relative deflection check	
Check name: Relative deflection check Type of design First order analysis So So To cal Load combinations	k for 2 points
Check name: Relative deflection check Type of design First order analysis So To cal Load combinations Load combination-2	k for 2 points
Check name: Relative deflection check Type of design First order analysis So To cal Load combinations Load combination-2 Load combination-3	k for 2 points
Check name: Relative deflection check Type of design First order analysis So To cal Load combinations Load combination-2	k for 2 points
Check name: Relative deflection check Type of design First order analysis Sec To cal Load combinations Load combination-3 Load combination-4	k for 2 points
Check name: Relative deflection check Type of design First order analysis Solution Control Conditioners Condi	k for 2 points
Check name: Relative deflection check Type of design First order analysis Control Load combinations Load combination-3 Load combination-4 Load combination-5 Load combination-1 Read combination-1	k for 2 points
Type of design First order analysis To cal Load combinations Load combination-2 Load combination-3 Load combination-4 Load combination-1 Parameters First structural point: X: 0 Y:	k for 2 points
Check name: Relative deflection check Type of design First order analysis Set To cal Load combination-2 Load combination-3 Load combination-4 Load combination-5 Load combination-1 Parameters First structural point: : X: 0 Y:	k for 2 points cond order analysis
Check name: Relative deflection check Type of design First order analysis To cal Load combinations Load combination-2 Load combination-4 Load combination-5 Load combination-1 Parameters First structural point: X: 5000 	k for 2 points cond order analysis
Check name: Relative deflection check Type of design First order analysis Contained combinations Contained combination-2 Contained combination-3 Contained combination-4 Contained combination-4 Contained combination-5 Contained combination-1 Contained combinatio	k for 2 points kcond order analysis 0 Z: 6000 Y: 0 Z: 6000
Check name: Relative deflection check Type of design First order analysis To cal Load combination-2 Load combination-3 Load combination-4 Load combination-4 Load combination-5 Load combination-1 Relative horizontal deflection Deflection limit [rmm]:	k for 2 points k for 2 points cond order analysis 0 2: 6000 Y: 0 2: 6000 10
Check name: Relative deflection determined of design Type of design First order analysis To cal Load combination-2 Load combination-3 Load combination-3 Load combination-4 Load combination-5 Load combination-1 Parameters First structural point: To the structural point: To	k for 2 points k for 2 points kcond order analysis 0 Z: 6000 Y: 0 Z: 6000 10 Load combinations
Check name: Relative deflection check Type of design First order analysis First order analysis Contact combination-2 Load combination-3 Load combination-4 Load combination-4 Load combination-1 Load combination-1 Contact combination-1 Relative horizontal deflection Deflection limit: [mm]: Dominant load combination: Dominant horizontal deflection [mm];	k for 2 points k for 2 points cond order analysis 0 2: 6000 Y: 0 2: 6000 10 Load combinations 0
Check name: Relative deflection check Type of design First order analysis For cal Load combinations Load combination-2 Load combination-4 Load combination-4 Load combination-5 Load combination-1 Parameters First structural point: X: 5000 Relative horizontal deflection Deflection limit (nm): Dominant load combination: Dominant horizontal deflection [nm]: Capacity [%]:	k for 2 points cond order analysis 0 0 2: 6000 Y: 0 2: 6000 10 Load combinations 0 0
Check name: Relative deflection check Type of design First order analysis For cal Load combinations Load combination-2 Load combination-3 Load combination-4 Load combination-5 Load combination-1 Parameters First structural point: X: 5000 Parameters First structural point: X: 5000 Relative horizontal deflection Dominant load combination: Dominant horizontal deflection [rmm]: Capacity [%]: Relative vertical deflection Deflection limit [rmm]: 	k for 2 points cond order analysis cond order analysis 0 0 2: 6000 Y: 0 2: 6000 10 Load combinations 0 0 100
Check name: Relative deflection check Type of design First order analysis For cal Load combinations Load combination-2 Load combination-4 Load combination-4 Load combination-5 Load combination-1 Parameters First structural point: X: 5000 Relative horizontal deflection Deflection limit (nm): Dominant load combination: Dominant horizontal deflection [nm]: Capacity [%]:	k for 2 points cond order analysis 0 0 2: 6000 Y: 0 2: 6000 10 Load combinations 0 0
Check name: Relative deflection check Type of design Image: Second Structure Seco	k for 2 points cond order analysis cond order analysis 0 0 2: 6000 Y: 0 2: 6000 10 Load combinations 0 0 100
Check name: Relative deflection check Type of design Image: Second S	

10 SECTION MODULE

10.1 BASICS

The wide functionality of the *ConSteel* connected with cross-sections suggested detaching a separate module: Section Module (*csSection*). This module collects all the tools relate to the creation, property calculation, stress analysis and standard design of a cross-section. Two parallel modeling possibilities exist: the *General Solid Section* (*GSS*) is an accurate finite element based surface model for any kind of cross-sections; the *Elastic Plate Segment*(*EPS*) is a thin-walled model especially for steel profiles. The steel standard design formulas are easy to evaluate on the *EPS* model including the real effective section generation of class 4 cross-sections based on the sectional forces resulted from the structural analysis.

Section administration and Section module can be launched by clicking the icons shown above. These icons can be found in the **STRUCTURAL MEMBERS** tab.

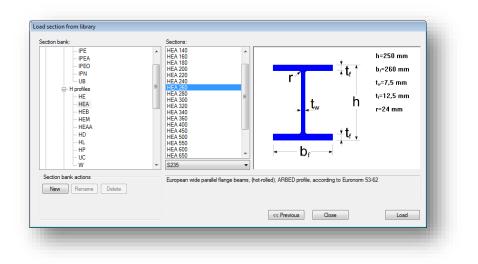
10.2 Section administration

Section must be loaded into the model database in order to use it in the model. This can be done in the section administration dialogue. Here you can see all the previously loaded sections and new sections can be loaded or created by clicking the appropriate buttons on the right.

Name 🛆	Parameters	Material	Туре	Source	State	Load section
CF-SHS 100x4	b=h=100; t=4; r=8	S 355 EN	Steel - standard	Standard	Used	From library
CF-SHS 120x5	b=h=120; t=5; r=10	S 355 EN	Steel - standard	Standard	Used	Macro section
CF-SHS 40x4	b=h=40; t=4; r=8	S 355 EN	Steel - standard	Standard	Used	
CF-SHS 60x4	b=h=60; t=4; r=8	S 355 EN	Steel - standard	Standard	Used	Modify
CF-SHS 80x4	b=h=80; t=4; r=8	S 355 EN	Steel - standard	Standard	Used	Remove
Column_lower	b1=250; tf1=16; h=600; tw=	S 235 EN	Steel - welded	Modellb	Used	Сору
Column_upper	b1=250; tf1=12; h=400; tw=	S 235 EN	Steel - welded	Modellb	Used	
020	d=20	S 355 EN	Steel - welded	Modellb	Used	
030	d=30	S 355 EN	Steel - welded	Modellb	Used	
Endwall_column	b1=220; tf1=14; h=200; tw=	S 235 EN	Steel - welded	Modellb	Used	
Endwall_column_b	b1=220; tf1=14; h=350; tw=	S 235 EN	Steel - welded	Modellb	Used	_
HEA 400	h=390; bf=300; tw=11; tf=1	S 235 EN	Steel - standard	Standard	Used	
column_cantilever	b1=250; tf1=14; h=400; tw=	S 235 EN	Steel - welded	Modellb	Used	Select



Sections can be loaded from the library into the model. In the section library all the most used standard European sections can be found in a tree structure. If a section is selected the all the properties can be seen at the right. Multiple sections can be loaded at the same time by using the typical Windows selection tools: *CTRL* or *SHIFT* buttons.



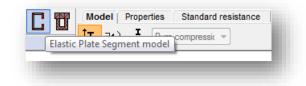
It is very important to know that the material grade of the member must be chosen when the section is loaded into the database.

So called macro sections can also be created in *ConSteel*. The following macro section types can be created:

- Steel
 - Welded from plates
 - Cold formed or hot rolled
 - Compound
- Reinforced concrete
- Composite
- Buckling Restrained Brace

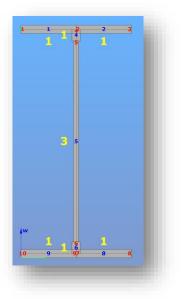
10.3 Section modelling

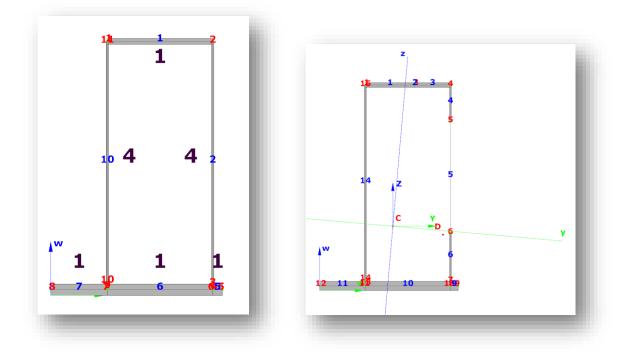
10.3.1 THE EPS MODEL



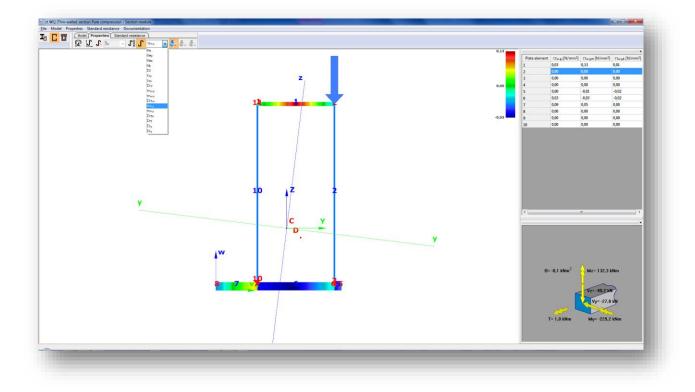


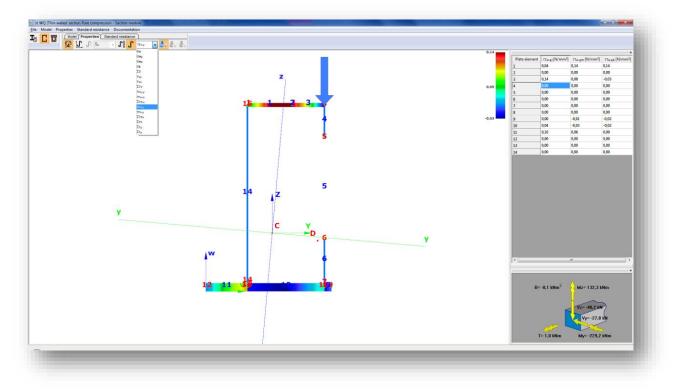
The *Elastic Plate Segment (EPS)* model is a thin-walled simplification of the usually steel profiles from welded, hot-rolled or cold-formed sections. The model is very easy and fast to calculate and has accurate results for thin-walled sections compared with the GSS model, which can be used and run for the same section side by side. The model yields the same general, torsional and shear properties and stresses and moreover it calculates the static moments and sectional classes according to the EuroCode. The EPS model is also very suitable for the evaluation all of the standard design formulas in a very sophisticated way. In case of Class 4 profile the effective cross-section is automatically generated based on the proper sectional forces, and the standard procedures can be evaluated on this effective section.





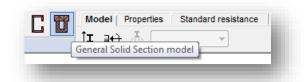




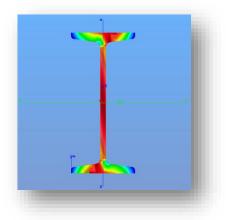




10.3.2 THE GSS MODEL



The *General Solid Section (GSS)* was created in order to model any kind of cross sections as accurate as possible. The model dissolves the section with arbitrary shape into triangle elements, and calculates the sectional properties by the finite element method. The section can be assembled with any number of different materials and the automatic finite element mesh generates separate triangles for the certain materials controlling the mesh size according to the elastic properties of the material. Beyond the usual properties (sectional area, second moment of inertia, etc.) the GSS model precisely calculates the torsional and shear properties like St. Venant and warping moment of inertia, shear areas and factors, etc., and these accurate properties are used in the global analysis. The elastic normal, torsional and shear stresses can be also calculated on this model using a spectacular colored view.



10.4 Section Tools

10.4.1 GEOMETRICAL CALCULATIONS

There are three coordinate systems in Section module: main axis system (y,z), section edit system (v,w) and the section edit system moved to the centre (C) of the section (Y,Z). The point D is the centre of rotation.

The section properties can be viewed according to these coordinate systems.



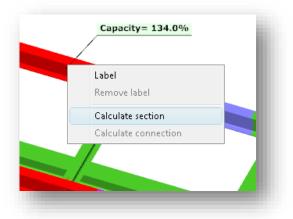
10.4.2 LOADS

If the section module is opened from the model, then all the load combination from the model are automatically imported into the section model. The relevant is chosen but any other can be selected and checked. It is also possible to add load combinations manually by clicking on

the **New Load Combination** icon

+												
Lcomb.	Current	NEd [kN]	My.Ed [kNm]	Mz.Ed [kNm]	Vy.ed [kN]	Vz.Ed [kN]	Bed [kNm ²]	Tt.Ed [kNm]	Tw.Ed [kNm]	TEd [kNm]	βer.op [-]	Bult [-]
Load Combination 1 (first order)		0,0	-162,3	0,0	-0,0	93,8	-0,1	-0,1	0,1	0,0	1,370	0,937
Load Combination 1 (second order)	 	-0,5	-162,3	-0,0	-0,2	93,8	-0,1	-0,0	0,0	0,0	1,370	0,937

csSection can be launched from the model if the *ANALYSIS* or *GLOBAL CHECK* tab is chosen and first or second order analysis or standard design had been run. There are two ways to open *csSection* directly for a selected section. First is to press right click over the section which needs to be analyzed and select *CALCULATE SECTION*.



The second way is to press right click over the element in the result sheet which can be found in the bottom of the screen and click on *CALCULATE SECTION*.

e	lement	Joint	Load combination	Examination	Capacity [%]	
1	45	k	Load Combination 1	Global stability resistance	95,7	
	31	j	Load Combination 1	Global stability resistance	95,7	
1	30	k	Load Combination 1	Global stability resistance	95,7	
1	66	k	Load Combination 1	Global stability resistance	95,7	
1	67	j	Load Combination 1	Global stability resistance	95,7	Calculate section
4	34	j	Load Combination 1	Global stability resistance	95,5	



10.4.3 STRESSES

Stresses can be checked by selecting the **PROPERTIES** tab and clicking on the **STRESSES**

icon **I**. Stresses can be viewed both in the EPS and the GSS model.

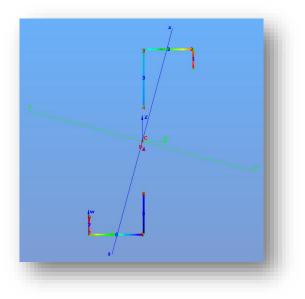
The following stresses are calculated by *CsSection*:

Stress Symbol	Stress name
σ _N	Normal stress due to axial force
σ _{My}	Normal stress due strong axis bending
σ _{Mz}	Normal stress due to weak axis bending
σ _B	Normal stress due to bimoment
Σσ	Resultant normal stress
τ _{Vy}	Shear stress due to strong axis shear force
τ _{Vz}	Shear stress due to weak axis shear force
Στν	Resultant shear stress due to shear forces
τ _{Tsv.y}	Strong axis shear stress due to St. Venant torsion
τ _{Tsv.z}	Weak axis shear stress due to St. Venant torsion
Σ τ _{Τsv}	Resultant shear stress due to St. Venant torsion
τ _{τω.y}	Strong axis shear stress due to warping torsion
τ _{τω.z}	Weak axis shear stress due to warping torsion
Σ τ _{τω}	Resultant shear stress due to warping torsion
Σ τ _Τ	Resultant shear stress due to torsion
Σ τ _γ	Strong axis resultant shear stress
Στ _z	Weak axis resultant shear stress

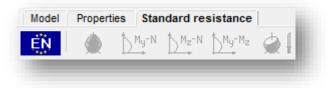


10.4.4 EFFECTIVE CROSS SECTION

If the first icon **14** is clicked and EPS model is selected then effective section take into consideration.



10.4.5 STANDARD RESISTANCE CALCULATIONS



For the given loading case the module determines the necessary and possible checks, performs all the standard calculations and evaluates the dominant case.

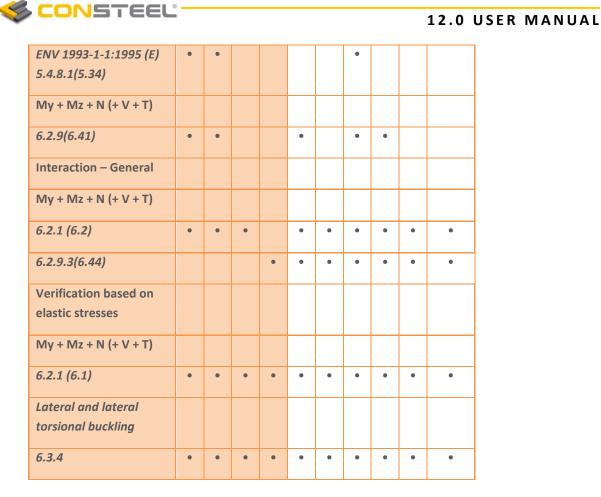
10.4.5.1 STEEL CROSS SECTIONS

The standard design procedures performed on the *EPS* model cover the major part of steel design methods in the Eurocode (the whole EN 1993-1-1 and parts of EN 1993-1-5) including the calculations of general elastic resistance, examination of pure cases, plastic and conservative interaction resistances of certain section types, web buckling analysis.



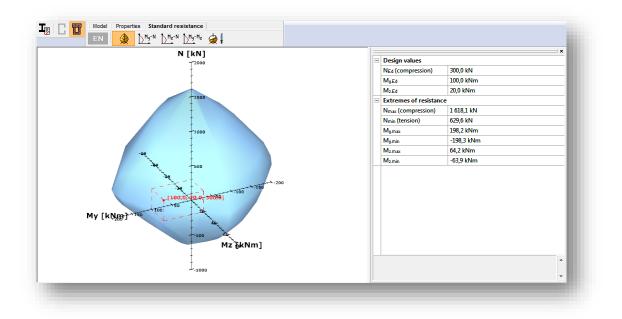
The following design checks are calculated by **CSSECTION**:

Verification type	S	ectio	n clas	S			Sha	pe ty	pe	
	1	2	3	4	I	U	0		•	Other
Pure forces										
6.2.3	•	•	•	•	•	•	•	•	•	٠
6.2.4	•	•	•	•	•	•	•	•	•	٠
6.2.5	•	•	•	•	٠	٠	•	•	•	٠
6.2.6(2-3)	•	•			•	•	•	•	•	٠
6.2.6(4-5)			•	•	٠	٠	•	•	•	•
6.2.7	•	•	•	•	٠	٠	•	•	•	•
EN 1993-1-5	•	•	•	•	٠	٠		•		
Interactions – plastic										
V + T										
6.2.7(6.26)	•	•			•					
6.2.7(6.27)	•	•				•				
6.2.7(6.28)	•	•					•	•		
M + V (+ T)										
6.2.8	•	•			•	•	•	•	•	٠
My + N (+ V + T)										
6.2.9(6.32)	•	•							•	
6.2.9(6.33 - 6.34, 6.36)	•	•			•					
6.2.9(6.39)	•	•						•		
ENV 1993-1-1:1995 (E) 5.4.8.1(5.34)	•	•					•			
Mz + N (+ V + T)										
6.2.9(6.32)	•	•							•	
6.2.9(6.35, 6.37 - 6.38)	•	•			•				-	
6.2.9(6.40)	•	•			-			•		
0.2.9(0.40)	•	•						•		



10.4.5.2 Composite and reinforced concrete cross sections

In case of reinforced concrete section the *CSSECTION* module can perform just the resistance surface of the section. It is also possible to view the resistance line along three different axes and the arbitrary section.



CONSTEEL

For the composite sections the resistance surface and the part of the EC 4 are performed.

The following design checks are calculated by *csSection* for the composite sections:

Verification type		SI	pe				
Pure forces							
6.7	٠	٠	•	•	٠		
6.7.3.2 (6.30)	٠	٠	•	•	٠		
6.7.3.2 (figure 6.19), 6.7.3.4	٠	٠	•	•	٠		
6.7.1 (9), table 6.3		٠	•	•	٠		
Interactions	٠	٠	٠	٠	٠		
6.7.3.6 (6.45)	٠	٠	٠	٠	٠		
6.7.3.7 (6.46-6.47)	٠	٠	٠	٠	٠		

10.4.5.3 RESULTS

There is a short summary about the calculations on the right and also the name of the dominant case, formula and load combination can be found there.

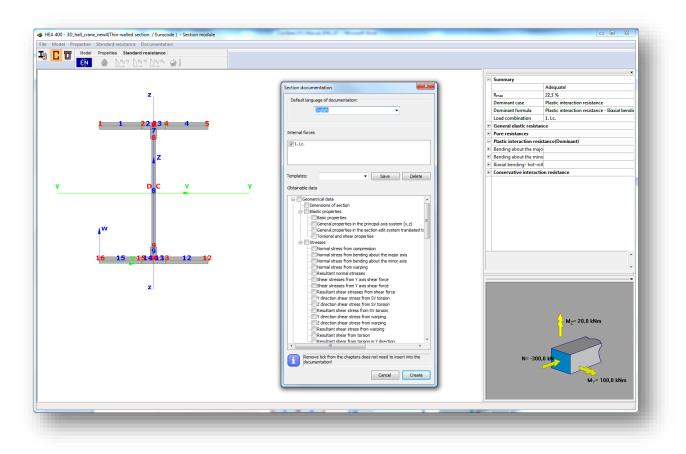
-	Summary			
		Adequate!		
	Rmax	81,1 %		
	Dominant case	Interaction check		
	Dominant formula	Compression + Bending about the major axis		
	Load combination	1. Lc.		
-	Pure resistances			
+	Compression			
-	Bending about the majo			
	Utilization	89,3 %		
	Applied part of stand	EN 1994-1-1 6.7.3.2 6.19 figure 6.7.3.5		
	My,Ed	200,0 kNm		
	Ieff,ll,y	2 561 406 630,0 mm ²		
	Mpl,y,Rd	224,1 kNm		
-	Interaction check			
-	Compression + Bending			
	Utilization	81,1 %		
	Applied part of stand	EN 1994-1-1 6.7.3.6 (6.45) formula		
	NEd	1 000,0 kN		
	My,Ed	200,0 kNm		
	Mpl,y,Rd	224,1 kNm		
	Mpl,y,N,Rd	274,1 kNm		
	μdy	1,22		
	α.Μι	0,90		



10.5 DOCUMENTATION

Each analyzed cross-section can be separately documented in detail for both section design including the shape parameters, geometrical, mechanical and design results, and the generated documents can be attached to the main documentation of the global structure.

After clicking the **Documentation** option on the menu bar, the way for creating documentation is the same as in *ConSteel* main module (see *CHAPTER 13 Documentation* for more detailed instructions).





11 FIRE DESIGN

The Fire design process is an integrated part of the *ConSteel* automated and efficient design process. In case of fire design situation *ConSteel* automatically uses and applies propel formulas of EuroCode for steel elements. General information about the design functionalities can be found in the *CHAPTER* **9**.

This chapter describes the workflow of the Fire design in *ConSteel* and highlits the differences compared to the general rules.

The required code checks of the different types of steel structural components – cross sections, members – can be conveniently evaluated based on the following standard:

▶ EN 1993-1-2: General rules. Structural fire design

11.1 FIRE LOAD GROUP AND LOAD CASE

For fire design situation, a dedicated *Fire load* group was created that handle the specialities of fire effect.



STEEL-

Fire load group has two options:

Consider indirect actions calused by thermal expansion:
 By default ConSteel neglect the effect of thermal expansion during the analysis, but by clicking on the checkbox there is the possibility to take into account

Value of variable factor for main variable load:
 By deafault ConSteel applies the Ψ_{2,1} but switching the radio button Ψ_{1,1} also can be used for load combination generation

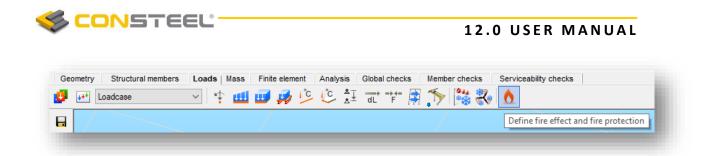
Permanent ↓ Loadcase	Load group (Fire) Name Fire Consider indirect actions caused by thermal expansion Representative value of Q1 variable action (EN 1991-1-2:2005 4.3.1(2)) $\bigcirc \Psi_{1,1} Q_1$ $\textcircled{O} \Psi_{2,1} Q_1$
	New load group Delete Load case
	New load case Copy Delete Load case including self weight

Atleast one load case must be added to the fire load group to place the fire effect on the model.

11.2 FIRE EFFECT AND PROTECTION

The fire effect and protection can be defined by clicking the **DEFINE FIRE EFFECT AND FIRE**

PROTECTION () icon on the **LOADS** tab.



The first option is to set the nominal fire curve. Three EuroCode deifned fire curves are implemented and can be used to define fire effect: Standard, External and Hydrocarbon.

The second one is to set the fire resistance time that can be varied between 15-200 min.

Define fire effect and fire prote	ection X
Parameters	
Nominal fire curve:	Standard \checkmark
Period of time (min)	15 ~
Fire protection data:	Protection 1 ~
Ψ1+Qa: Ψ2+Qa:	Ψ_1 1 Ψ_2 1
Design parameters - EN 1993	-1-2 4.2.3.3 (7)-(8)
~G 1	~G 1
Select the beams to se	t fire effect to them!

By clicking on the three dot button next the *Fire protection data* label, different fire protection can be created.

Fire protection can be *Unprotected* and *Protected*. If Protected is chosen then the insulation material and the thickness also must be specified. From the dropdown menu, the section grading for fire effect, according to EN 1993-1-2-4.5. chapter, also must be selected.



Define fire protection		×
Name:	Protection 1	~
O Protected (EN 13381-	1,2 and 4)	
- insulation material	Board Fiber Cement $~~~\cdots~~$	
- thickness	10 mm	
Grading according to EN	1993-1-24.2.5 table 4.2 and 4.3	
Open (I) section exposed	to fire on all sides	\sim
New	Delete Close Apply	

By clicking on the **APPLY** button, fire protection will be saved. With the **NEW** button, several other protection can be created. With **DELETE** button, the selected and created protection can be deleted.

The fire effect can be set to a different intensity at the top and the bottom of the member by modifying Ψ_1 and Ψ_2 multiplication factors. In case of varying intensity the design parameters acc. to the EN 1993-1-2 4.2.3.3 (7)-(8) also can be set.

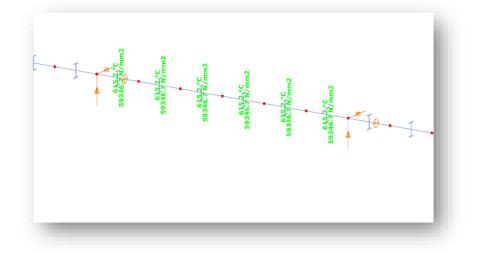
Created fire effect can be placed on the members with a simple click on the model. It is possible to apply fire effect to more members at the same time. This case the members should be selected before clicking the fire icon.

11.3 ANALYSIS

ONSTEEL

During the finite element generation *ConSteel* calculates the steel temperature and the reduced modulus of elasticity. These values can be viewed in the *FINITE ELEMENT* tab by selecting the appropriate load case.





The analysis for the fire load combinations are performed in global level. Therefore the calculated deformations and internal forces possibly include the effect of the expansion and decreased elasticity modulus of those members which are subjected to fire.

11.4 DESIGN

ConSteel performs cross section resistance and buckling checks for all steel members which are subjected to fire effect by using the adequate formulas from the EuroCode 3 standard.

Classification of cross-section (EN 1993-1-2 4.2.2)

Cross-sections are classified as for normal tempereature (EN 1993-1-1) but using the reduced value for ϵ .

$$\varepsilon = 0.85 \cdot \left[\frac{235}{f_{\mathcal{Y}}}\right]^{0.5}$$

Tension (EN 1993-1-2 4.2.3.1)

Permanent temperature ((1))

$$\begin{split} N_{t,fi,\theta,Rd} &= k_{y,\theta} \cdot \frac{\gamma_{M,\theta}}{\gamma_{M,fi}} \cdot N_{pl,Rd} & \text{(Class 1-4)} \\ \text{where} \\ k_{y,\theta} & \text{value based on a function according to Table 3.1} \\ N_{pl,Rd} & \text{based on normal case} \end{split}$$



Varying temperature ((2))

$$N_{t,fi,t,Rd} = A_{non,t} \cdot \frac{f_y}{\gamma_{M,fi}}$$
(Class 1-4)

Compression (Class 1-3: EN 1993-1-2 4.2.3.2, Class 4: + Annex E.2)

$$N_{c,ft,t,Rd} = A \cdot \frac{k_{y,\theta} \cdot f_y}{\gamma_{M,ft}}$$
(Class 1-3)
$$N_{c,ft,t,Rd} = A_{eff} \cdot \frac{k_{p02,\theta} \cdot f_y}{\gamma_{M,ft}}$$
(Class 4)

where

 $k_{p0.2,\theta}$ value based on a function according to Table 3.1

Varying temperature ((6))

Conservative way, case (1) where $\theta_a = \theta_{a,max}$

Bending (EN 1993-1-2 Class 1-2: 4.2.3.3; Class 3: 4.2.3.4; Class 4: + Annex E.2)

Permanent temperature ((1))

$$\begin{split} M_{fi,\theta,Rd} &= k_{y,\theta} \cdot \frac{\gamma_{M,\theta}}{\gamma_{M,fi}} \cdot M_{Rd} \\ \text{where } M_{Rd} &= M_{pl,Rd} \quad \text{or in case of shear } M_{Rd} = M_{V,Rd} \quad \text{(Class 1-2)} \\ M_{Rd} &= M_{el,Rd} \quad \text{or in case of shear } M_{Rd} = M_{V,Rd} \quad \text{(Class 3)} \\ M_{Rd} &= M_{eff,Rd} \quad \text{(Class 4)} \\ k_{y,\theta} \quad \text{value based on a function according to Table 3.1} \end{split}$$

Varying temperature ((2))

$$M_{fi,t,Rd} = W_{pl,non,t} \cdot \frac{f_y}{\gamma_{M,fi}}$$
(Class 1-2)
$$M_{fi,t,Rd} = k_{y,\theta,max} \cdot \frac{\gamma_{M,\theta}}{\gamma_{M,fi}} \cdot \frac{1}{\kappa_1 \cdot \kappa_2} M_{el,Rd}$$
(Class 3)

where $\kappa_l = 0.7$ if protection=1 and m_factor_section=2.3 or 10 $\kappa_l = 0.85$ if protection=2 and m_factor_section=3 or 4 $\kappa_l = 1.0$ in other cases $\kappa_2 = 1.0$ conservative



$$M_{fi,t,Rd} = k_{p0.2,\theta,max} \cdot \frac{\gamma_{M,0}}{\gamma_{M,fi}} \cdot \frac{1}{\kappa_{I} \cdot \kappa_{2}} M_{eff,Rd} \qquad (Class 4)$$

Shear (EN 1993-1-2 Class 1-2: 4.2.3.3(6); Class 3: 4.2.3.4(4); Class 4: + Annex E.2)

Permanent temperature

$$V_{fi,t,Rd} = k_{y,\theta,web} \cdot \frac{\gamma_{M,\theta}}{\gamma_{M,fi}} \cdot V_{Rd}$$
(Class 1-4)

Varying temperature

 $k_{y,\theta,web}$ the hottest point in the web

In case of complex internal forces *ConSteel* use the conservative interaction formula neglecting the effect of shear:

$$\frac{N_{fi,Ed}}{N_{fi,\theta,Rd}} + \frac{M_{y,fi,Ed}}{M_{y,fi,\theta,Rd}} + \frac{M_{z,fi,Ed}}{M_{z,fi,\theta,Rd}} \le 1$$

Global stability resistance

To calculate the global stability resistance for fire design situation, *ConSteel* uses EuroCode General method (EN 1993-1-1 6.3.4) as for normal temperature but using the proper buckling curves:

For compression

$$\chi_{fi} = \frac{1}{\varphi_{\theta} + \sqrt{\varphi_{\theta}^2 - \bar{\lambda}_{\theta}^2}}$$

For bending

$$\chi_{LT,fi} = \frac{1}{\phi_{LT,\theta,com} + \sqrt{\phi_{LT,\theta,com}^2 - \bar{\lambda}_{LT,\theta,com}^2}}$$



12 EARTHQUAKE ANALYSIS

12.1 BASICS

ConSteels can perform earthquake analysis is based on *Modal response spectrum analysis* (MRSA), and it supports three different analysis types:

- All modal shapes, CQC summation:
 ConSteel calculates the modal loads for all dynamic shapes, for all directions, and calculates the analysis results (displacements, internal forces) for each dynamic shapes using first order analysis. After this, it summarizes them with Eurocode's CQC method.
 CQC summation method can give the highest expectable deformations and internal forces, based on a statistical method. Envelope diagrams of deformation and internal forces without signs are the result of the CQC summation
- Single dominant mode:
 From the calculated dynamic shapes, ConSteel chooses the dominant one for each direction, and performs the calculation for one dominant shape for each direction. Modal loads for each direction are the results of the method.
- Selected modes, linear summation:
 With the modal shapes, choosed by the user, and with the combination factors for each shapes ConSteel creates a linear combination of vibration shapes, and gives the results for all directions using the combined shapes. Modal loads for each direction are the results of the method.

	All modal shapes, CQC summation	Single dominant mode	Selected modes, linear summation
Dynamic calculation	Required	Required	Required
Second order analysis	Not possible	Yes	Yes
Stability calculations	Not possible	Yes	Yes
Global check – Cross section check	Yes	Yes	Yes

Comparision of the three methods:



Global check – Stability check	Not possible	Yes	Yes
Member check	Not possible	Yes	Yes
Masses taken into consideration	Masses corresponding for each vibration shapes for all directions	Masses for the choosen vibration shapes with the consideration of the combination factors in all directions	100% of the masses in all directions
Automation	Yes	Manual input required	Yes
Second order sensitivity	Yes	Yes	Yes

Results of the modal response spectrum analysis with the 100% "+" 30% rule are combined with the other effects acting in sesmic sisutation, by ConSteel.

a) E_{Edx} "+" 0,30 E_{Edy} "+" 0,30 E_{Edz}
b) 0,30 E_{Edx} "+" E_{Edy} "+" 0,30 E_{Edz}
c) 0,30 E_{Edx} "+" 0,30 E_{Edy} "+" E_{Edz}

12.2 Steps of the seismic analysis

Modal response spectrum based seismic analysis can be performed with the following steps in ConSteel:

- 1. Definition of modal loads/Mass tab
 - Definition of mass cases and mass groups
 - Mass definition manually on the structure, or by converting existing load cases
 - Mass combination definitions
- 2. Seismic effect definition/Masses tab
 - Definition of Standard or User defined response spectrum
 - Definition of Seismic effect
 - Setting the analysis method
- 3. Seismic loadgroup and load combination definition / Loads tab



- Seismic loadgroup and loadcase creation
- Assignment of seismic effect to seismic loadcase
- Definition of considered shapes for the analysis Only in the case of Selected modes, linear summation analysis method!
- 4. Analysis / Analysis tab
 - Setting the statical calculation parameters; For the comparision of available analysis types, please check chapter 13.1
 - Setting the parameters for Dynamic calculations
- 5. Design / Global checks and Member checks tab
 - For the available design checks, please go to the comparision of different analysis types at *chapter 13.1*

12.3 Seismic effect

Seismic effects can be created with the **SEISMIC EFFECT** (**Description**) function which can be found on the **MASS** tab. On the Seismic effect dialogue, all parameters can be set, which are required for the seismic analysis.

For the seismic effect creation, existence of pre-created mass combination and response spectrum is required. To see how to define these, please go to *CHAPTER 7 MASSES*!

New seismic effect can be created by pressing the **NEW** button. More seismic effects can be created and saved. Change between different saved seismic effects is possible with the *Name* dropdown menu.

Options of the **SEISMIC EFFECT** dialogue has been separated into two tabs:

- Seismic settings
- Additional effects

12.3.1. SEISMIC SETTINGS

In the Settings (#1) panel, main parameters of the seismic analysis can be set

 Mass combination: Choice of the used mass combination for the dynamic shape evaluation during the modal response spectrum analysis has to be done. The dropdown menu is automatically filled up with the pre-defined mass combinations. Seismic analysis will only be performed for the mass combination which is selected here!



- Calculation method: Choice of the calculation method to be used for the seismic analysis. To see the differences between each calculation methods, please see chapter 12.1
- Main direction of the structure from X: The main direction of the structure can be se tusing the black arrow () tool, by clicking the start and end point of a vector in the modeling area. The other option, is to set the angle between the main direction and the X axis manually in the editbox with direct input. The direction, which is given will be the "1" direction, perpendicularly to this will be the "2" direction of the effect. The q1 and q2 factors corresponds with these directions too.

smic settings Additional effects			
ettings Mass combination		Mass combination-1	~
Calculation method		All modes, CQC summation	
Main direction of the structure from X (a+)		
	-11		
Pesponse spectrums Response spectrum for ULS		Spectrum 1	× …
Elastic response spectrum for SLS		Spectrum 1	×
 ULS design spectrum and gd factors 	s for SLS	Lower limit of spectrum - ß	0,2
ehaviour factors			
Consider vertical seismic effect] Same behaviour in Horizontal XY d	lirections
Behaviour factors		isplacement behaviour factors	
q ₁	1,5	9d1 9d2	1,5
42 9 _z	1,5	4d2 9dz	1,5
-			2/0
Shown response spectrum	Design respons	se spectrum - Horizontal	~
Sd/a.gR			
3,0			
4			
2,0			
1,5			
1.0			
0.5		T= 2,70 s	
		Va.gR= 0,20	
0,00 0,50 1,00 TB TC TD	1,50 2,00	2,50 3,00 3,50	4,00 T [s]



On the response spectrums panel (#2), the response spectrum can be selected which will be used for the seismic analysis.

Response spectrum for ULS	Spectrum 1	~
Elastic response spectrum for SLS	Spectrum 1	~
ULS design spectrum and qd factors for SLS	Lower limit of spectrum - B	0,2

Handling of the response spectrums in ultimate and serviceability limit states can be the following:

- ULS response spectrum, and use of q_d factors
 In this case choice of response spectrum (standard or user defined) is possible only for the ULS checks. Deformations, which are necessary for the SLS checks are evaluated from the ULS results, using the q_d displacement behaviour factors
- response spectrum for ULS and SLS limit states
 Choosing the *Elastic response spectrum for SLS* option, different response spectrum can be set for ULS and SLS limit states. For ULS limit state, only standard or user defined design spectrum, while for SLS limit state only standard or user defined elastic response spectrum can be choosen from the dropdown menu

Clicking on the (....) button will bring up the spectrum definition dialogue, where parameters of a spectrum can be checked, or even new ones can be created (see CHAPTER 7.6 CREATING RESPONSE SPECTRUMS)

At the *Behaviour factors (#3)* panel, behaviour and displacement behaviour factors can be set for the main directions (1,2,z).

Number of adjustable behaviour factors are depending of the type of the applied response spectrum(s):

	Behaviour factor (q)	Displacement behaviour factor (q _d)
ULS design spectrum and qd factors for SLS	Adjustable in case of standard response spectrum In case of user defined spectrum, the spectrum itself contains it, can not be adjusted	Adjustable



Different response	Adjustable in case of standard response spectrum	
spectrum for ULS and SLS limit states	In case of user defined spectrum, the spectrum itself contains it, can not be adjusted	Can not be adjusted

Behaviour factors besides the type of the spectrum, are dependent on the following options:

- Consider vertical seismic effect: If the option is turned on, vertical behaviour factors can be set, and vertical seismic effects will be taken into consideration for the seismic analysis
- Same behaviour in Horizontal XY directions: If the option is turned on, it is enough to define only one behaviour parameter, it will be applied for both X and Y directions

At the bottom of the dialogue, in the Shown response spectrum panel (#4) the selected response spectrum *—design or elastic-* of the dropdown menu can be seen. By hovering the mouse over the points of the graph, its value will be displayed graphically.

12.3.2. Additional effects

On the *ADDITIONAL EFFECTS* tab of the *SEISMIC EFFECT* dialogue, the following settings can be adjusted:

- Accidental torsional effects: In case of spatial 3D models, effects of the uncertainities
 of load placements causing torsional effects, and also has to be taken into account.
 ConSteel calculates the effect automatically. For the calculation, storeys has to be
 defined, where torsion effect is acting, and eccentricity value, with which the
 masspoints will be shifted.
- Second order effect: Second order effects has to be taken into account, if the coefficient of the shift difference (θ) between the storeys is greater than 0,1. Consteel automatically calculates it, and takes it into accound. Storeys has to be pre defined.

Considered storeys has to be choosen in the *Considered storeys for accidental torsional and second order effects (#1)* panel. If there are storeys defined in the model (see *CHAPTER 3.4 PORTION MANAGER*), they will be automatically loaded into the table. If the checkbox is turned on next to a storey, it will be taken into consideration during the seismic analysis.



Seismic			~
	-		· ·
eismic	settings Additional effects		
Consid	ered storeys for accidental torsional	and second order effects	
1			
	Storeys Level (+ 5000)		
	Level (+3000)		
	Level (+0)		
	· · · ·		
Accide	ntal torsional effects		
Negleo	t		~
A	ental eccentricity [%] 5		
Accid	ental eccentricity [76]		

At the *Accidental torsional effects* (#2), consideration of the effect can be turned on or off. The following combinations can be choosen for the shift of the mass ponts:

- +e_{aX}+e_{aY}, e_{aX}-e_{aY}: Mass points will be shifted both for positive and negative directions in the global X and Y directions with the given value, but not mixed. It is a simplification, and it is reducing the number of seismic combinations
- +e_ax+e_ay, +e_ax-e_ay, -e_ax+e_ay, e_ax-e_ay: Mass points will be shifted both for positive and negative directions in the global X and Y directions, including mixed signs too, with the given value. It is the recommended way by Eurocode, but it increases the number of seismic combinations.

The value of the shift of the mass points can be given in the editbox with a percentage.

12.4 Seismic loadcases and loadcombinations

12.4.1. SEISMIC LOADGROUP AND LOADCASE



Going back to the *LOADGCASES AND LOADGROUPS* dialogue, a new type of *SEISMIC LOADGROUP* has to be defined. After creating a seismic loadgroup, th epre-defined seismic loadcase will automatically assigned to the group as a seismic loadcase.

Id cases and load groups	Permanent Variable Variable Meteorological Wind Snow Crane Accidental Accidental Accidental Grine Seismic Cancel Load case	ic 1 v
	Name Load group New load case	Seismic 1 Seismic Copy Delete ght
	Loadcase	Apply OK

If the calculation method for the seismic effect is set to **SELECTED MODES**, **LINEAR SUMMATION**, an additional button will appear on the **LOAD CASES AND LOAD GROUPS** dialogue when the seismic loadcase is selected in the tree control. The **LINEAR COMBINATION OF MODAL EFFECTS** (#1) button stands for the option to set the combination factors for the calculated vibration shapes. After clicking on the button, a new dialogue will pop-up, which contains all of the calculated vibration shapes, with the corresponding mass participation for each one, in all (1,2,z) main directions. By setting the combination factors, mass participation can be adjusted for each vibration modes, 1 which will be considered during the next seismic analysis.

If all of the combination factor cells of the table are empty, ConSteel will perform seismic analysis with **SINGLE DOMINANT MODE** calculation method.



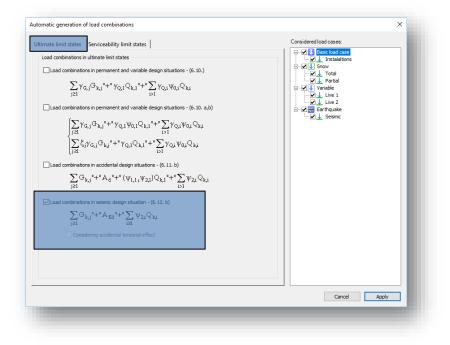
0	Load group (Seismi	c)
Basic load case Loadcase	Name	Earthquake
Earthquake		
Load case	Туре	Response spectrum based $\qquad \lor$
	Seismic	Earthquake \checkmark
	New load gro	Delete Delete
	Load case	
	Name	Load case
	Load group	Earthquake \vee
	Free vibration anal	ysis is required!
	Linea	ar combination of modal effects
	1	
	New load ca	ase Copy Delete
	IVEW IOAD CA	copy Delete
	Load case including	self weight
	None	~
	-	lose Apply OK

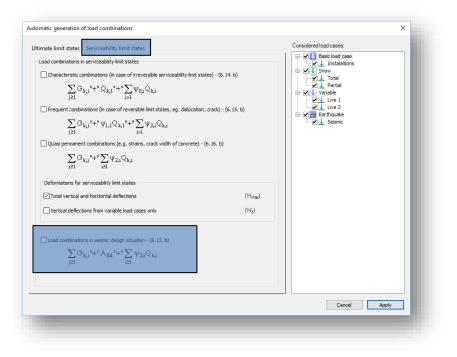
Modal shapes (Time [s])	1			2		Z
	Mass p.	Comb f.	Mass p.	Comb f.	Mass p.	Comb f.
(1.) 0,692	0,92	1	0,00		0,00	
(2.) 0,257	0,00		0,00		0,01	
(3.) 0,248	0,08		0,00		0,00	
(4.) 0,236	0,00		0,00		0,99	
(5.) 0,000	0,00		0,54		0,00	
(6.) 0,000	0,00		0,46		0,00	

12.4.2. LOAD COMBINATIONS IN SEISMIC DESIGN SITUATIONS

On the *AUTOMATIC GENERATION OF LOAD COMBINATIONS* dialogue, all combinations can be automatically generated for seismic situations too, including ultimate and serviceability limit states.









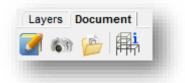
13 DOCUMENTATION

13.1 BASICS

In *ConSteel* there is a powerful and easy-to-use documentation engine in order to create precise, easily editable static documents that meet the requirements of the structural engineers.

The static documentation contains the full description of the model and also the results of the analysis and design.

13.2 The document tab



13.2.1 START DOCUMENTATION

All icons which related to the documentation are placed in the **Document** tab. The documentation can be generated if first icon is clicked. The documentation wizard guides you through the documentation generating process. On the first window the name of the documentation can be set. The name appears on every page of the documentation. Other fields like the name of the project, the name of the engineers and the date of the creation can also be set but would appear only on the front page.

Name of document:	Document 1
Name of project:	
Names of engineers:	
Date of creation:	2008.11.10.
Default language of documentation:	English
i Click into required field to define p Adding name of document is comp separated by semicolon.	roperty ulsory! The names of engineers should be



The header and the footer pictures of the document can be customized so your own company logo can be added. In order to do so the default header and footer images need to be replaced. These files can be found in the software's installation directory; the default path is the following: C:\Program Files\ConSteelversion\Pic. The "Header_pic.png" and the "Footer_pic.png" need to be replaced by your own images.

New document (3/2)	EX
Analysis results First order Second order First and second order	< Previous Next > Cancel
-	

The second step is to choose which of the analysis results need to be included in the documentation. Any of the analysis results can be chosen if had been calculated previously.

emplates:	Select all	✓ Save	Dele	te
Model of				*
√ Main √ Poin				
🖃 🔽 Bar r	nembers			
	Parameters of bar members			
	Sections			=
	Parameters of tapered members			
	Parameters of haunch Releases (pins)			
	Holes			
	transfer surface			
	oad transfer surface			
	Placement data			
⊡ . VLine				
	Placement data			
⊡ V Surf	ace supports			Ψ.
Ultimate limit st	ate design parameters			
Summary of	standard checks by members			
V Order l	y member name	utilization limit	10	%
	,			
Order I	y utilization value	utilization limit	10	%
Detailed res	ults of standard checks by membe			
		utilization limit	50	%
Demosius	1			Feel
1 Remove	tick from the chapters does not n	eed to insert into the do	cumenta	uon!
	Cancel	< Previous	Creat	e



The third window of the wizard includes the name of the chapters which can be added to or can be removed from the documentation.

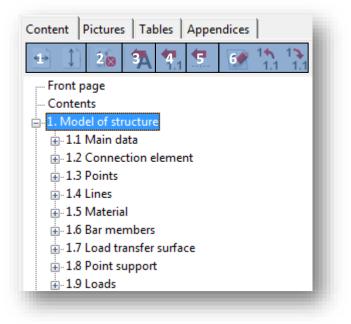
For the ULS design documentation utilization limit can be set at the bottom of the dialog. All of the members will be included the generated documentation that are equal or higher than the set limits. Separate limits can be set for the summary and for the detailed documentations.

Documentation templates can be saved with the *SAVE* button. Most used documentation structures can be saved as templates. Based on the templates, documentation can be created with one click.

If you click on the **CREATE** button the program will generate the documentation.

13.2.1.1 DOCUMENTATION MODULE

CONSTEEL



After creating the documentation 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 by drag-and-drop function. It is also possible to delete content from document by clicking the third icon (#2).

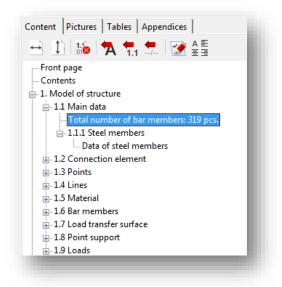
Text can be inserted anywhere in the document if you click on the fourth icon (#3) and type in the desired text into the window. It is also possible to insert headings by clicking on the fifth icon (#4).

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



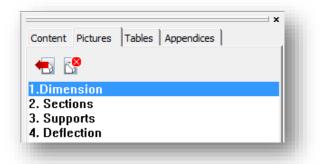
The last two icons on the right are different if you click on a chapter or in a paragraph. If you click on a chapter then the icons are as described above. If a paragraph is clicked where there

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.



With the first two icons (#1) the fitting of the pages can be set. Pages can be fit with width (

By clicking the **Pictures** tab the image handling icons can be seen. Any of the taken snapshots can be inserted into the documentation or can be deleted. The image will be inserted below the selected paragraph.



The third tab is the **Tables**. There are lot of possibilities in *ConSteel* where tables can be saved. If you see the icon during analysis and design process where the results appears in a



table, then that table is saveable. Any previously saved tables can be inserted to the documentation or can be deleted from the model database. The table will be inserted below the selected paragraph.

The last tab is the **Appendices**. If any joint or section documentation had been generated previously, these can be found here and can be added to the end of the documentation. The appendices can also be deleted from the database.



IT SHOULD BE BEAR IN MIND THAT THE DOCUMENTATION NEED TO BE REGENERATED TO BE UP-TO-DATE AFTER ANY CHANGES HAS BEEN MADE ON THE MODEL. IF THERE ARE SNAPSHOTS IN THE DOCUMENTATION THESE SNAPSHOTS SHOULD BE TAKEN AGAIN AND NEED TO BE ADDED TO THE DOCUMENTATION.

13.2.2 CREATING SNAPSHOTS

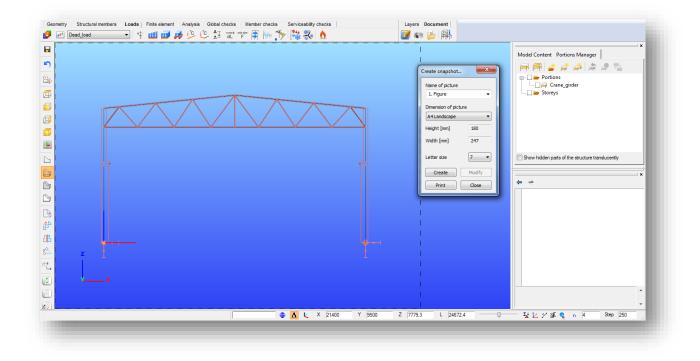
The second icon on the **DOCUMENT** tab can be used to take a snapshot of the model. The snapshots can be easily added to the desired part of the documentation (see 11.2.1.1 chapter for detailed information).

Name of picture	2
1. Figure	•
Dimension of pic	cture
Height [mm]	247
Width [mm]	180
Letter size	7 🔻
Create	Modify
Print	Close

The size of the snapshot can be set and a dashed rectangle shows which part of the model would appear on the picture. The model can be moved or rotated to fit into the dashed rectangle.

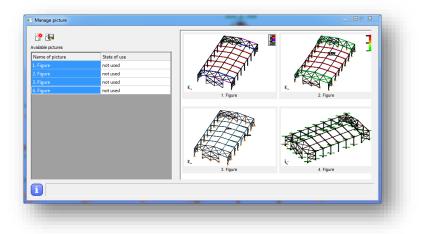
With the *PRINT* button the actual view of the model can be printed directly from the dialog.





13.2.3 SNAPSHOT MANAGER

The third icon on the documentation tab is the Snapshot manager is where the taken snapshots can be exported to a file or can be deleted from the model (except if the picture was inserted into the documentation) in order to reduce model file size. Multiple selected pictures can be saved and deleted at the same time. The name of the images can also be edited by double clicking the name of the image.



13.2.4 MODEL INFORMATION

With using the fourth icon on the **DOCUMENT** tab the model information can be viewed or can be exported to a file. The model information window shows the most important features



of the model like Number of bars, Number of supports, Total weight of the structure, etc. The mass, surface and the length can be viewed for each section in the model. The model information can be shown based on the full model or on the current view if the model view shows only a part of the full model using the sub model view.

Based	on the full model	Based on the	current view
Main data			
Number of bars	4	Total weight of structure	e [t] 3,46
Number of support	mber of support 13 Total surface to be painted[m2]		nted[m2] 61,05
		Total length [m]	30,25
Name	Mass	: (t) Surface (m2)	Length (m)
PE 500	1,955	38,04	18,25
1EA 400	1,503	23	12
2			Close

13.3 MS WORD EXPORT OF THE DOCUMENTATION

Documentations can be exported into an editable *.docx* format Word file. The export itself is a complete process, which means every table, picture and appendices will be exported. The *MICROSOFT WORD* export function can be found in the documentation module's *DOCUMENT* menu, under the *EXPORT* submenu.

🤹 Docume	nt 1 - New model -	Documentation module
Document	Edit Document	ation language
Open		
New d	ocument	
Close		
Delete		
Print		
Export	>	Microsoft Word
Exit		



After pressing the *MICROSOFT WORD* button, the Documentation export dialogue will appear:

	×
Cancel	OK
	Cancel

On the dialogue the path of the exported file has to be given, by browsing it with the (...) button. It is available (and required) to choose a *TEMPLATE* for the exported document. Templates are a collection of styling settings, which will be applied on the exported Word file. By default, a ConSteel template is provided in the installation package, and it contains the same styling settings as the main ConSteel documentation. By selecting the *USE ConSTEEL TEMPLATE*, and pressing the *OK* button, ConSteel will generate a Word file of your documentation, which is the same in content, and is as close as it can be in styling as the original one. Custom templates can also be used, please check *CHAPTER 13.3.1* to know more!

When opening an exported documentation in Word for the first time, the table of content has to be refreshed, by right clicking on the heading of the table and pressing the refresh button:

*	Documentation
Contonto	Arial - 8,5 - A A 💉 🗲
Contents	B I ∐ a ^b → A → = → 1 → Styles
Update fields to refresh table of	content ⁱ & _{Cut}
	Paste Options:
	Ê
	Dupdate Field
	Edit Field
	Toggle Field Codes
	A <u>F</u> ont
	<u>≡</u> ¶ <u>P</u> aragraph



$13.3.1\ CREATING\ CUSTOM\ TEMPLATES\ FOR\ MS\ WORD\ EXPORT$

As it was mentioned in the previous chapter, the installation package of ConSteel contains a default template, which is filled up with the same styling settings, as the main documentation. This template can be found in the installation folder of ConSteel, which is *"C:\Program Files\ConSteel 12\Data\Export\ConSteel.dotx"* by default.

If a new template, with customized styling settings is desired to be created, the **ConSteel.dotx** file should be copied first. After opening the copied file, the change in stylings can be performed in the **STYLES** dialogue of MS Word:

Clear All	
Normal	T
Heading 1	<u>¶a</u>
Heading 2	<u>¶а</u>
Title	<u>¶а</u>
Subtitle	<u>¶a</u>
Headpage Document	Ту <u>; ¶а</u>
Headpage Title	<u>¶a</u>
Headpage Content	<u>¶a</u>
Table Text	<u>¶a</u>
TOC Heading	T
TOC 1	T
Footer	<u>¶a</u>
Header	<u>¶а</u>
Show Preview Disable Linked Styles	5
Ma 🖇 🛠	Options

After the settings are made, the file should be saved in **.dotx** format (as the original was), and it can be used for word exports in the future.

< CONSTEEL

14 JOINT MODULE

14.1 BASICS

In the *ConSteel* design approach great emphasis is placed on the integrated modelling and calculation of the global structure and its connections therefore a powerful joint analysis engine is developed as a separate module: *csJoint*. A great range of different joint types can be created easily applying several special connection elements, the design tools covers practically the whole joint standard Eurocode 3 Part 1-8. A joint can be defined alone or based on the global model by the automatic joint identification tool. A prepared joint can be placed several different places in the global model to make the joint integral part of it. A placed joint is always automatically rechecked based on the current analysis results, and the connection stiffness is also updated modifying the global model consequently.

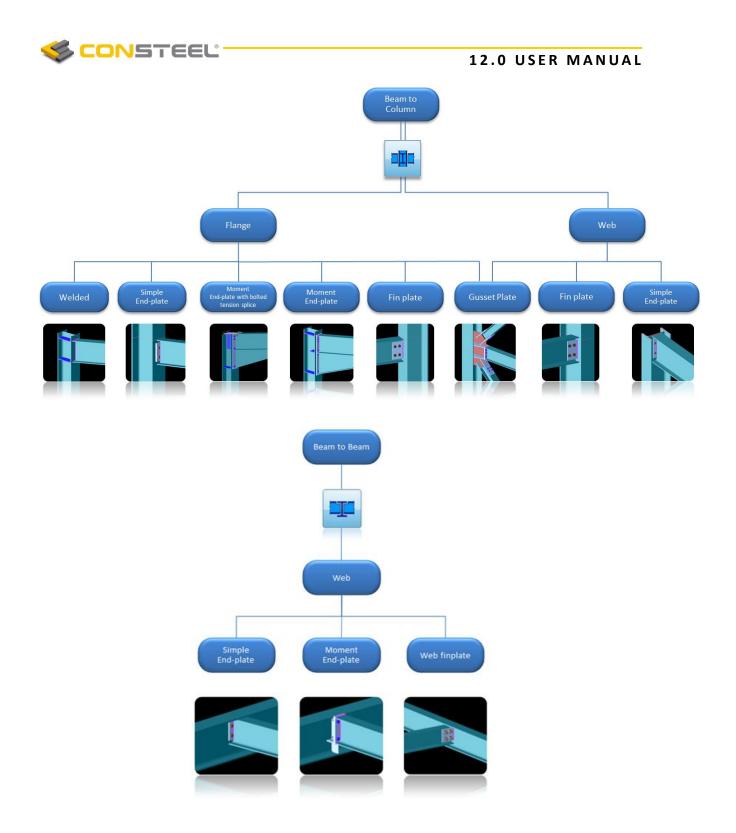


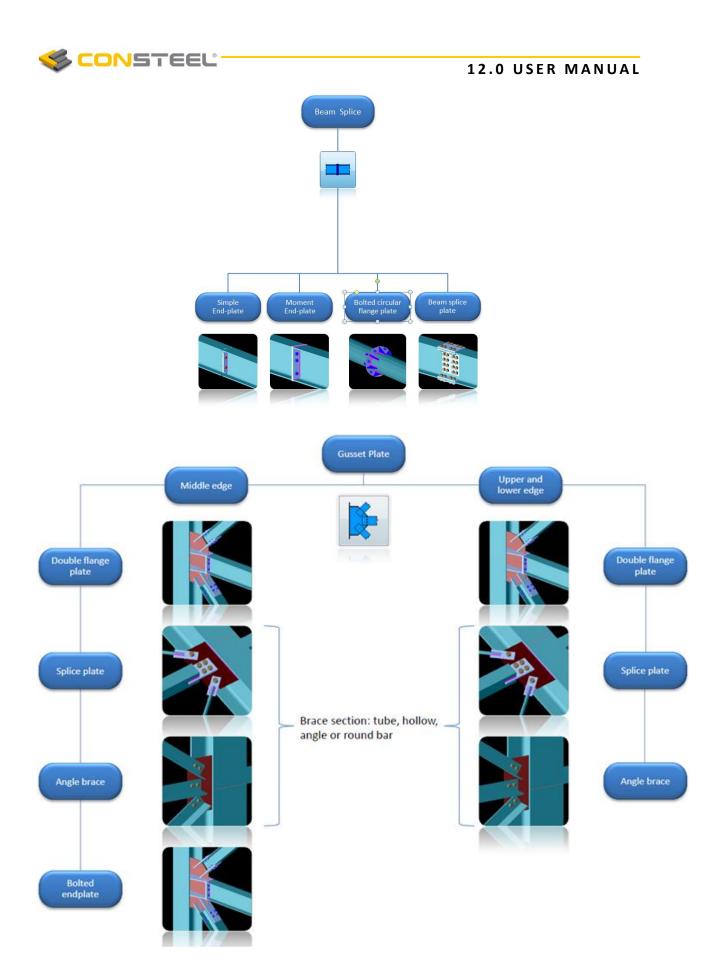
All the joint related commands can be found on the *STRUCTURAL MEMBERS* tab. These are the followings: Edit joint, Create joint by model, Place joint.

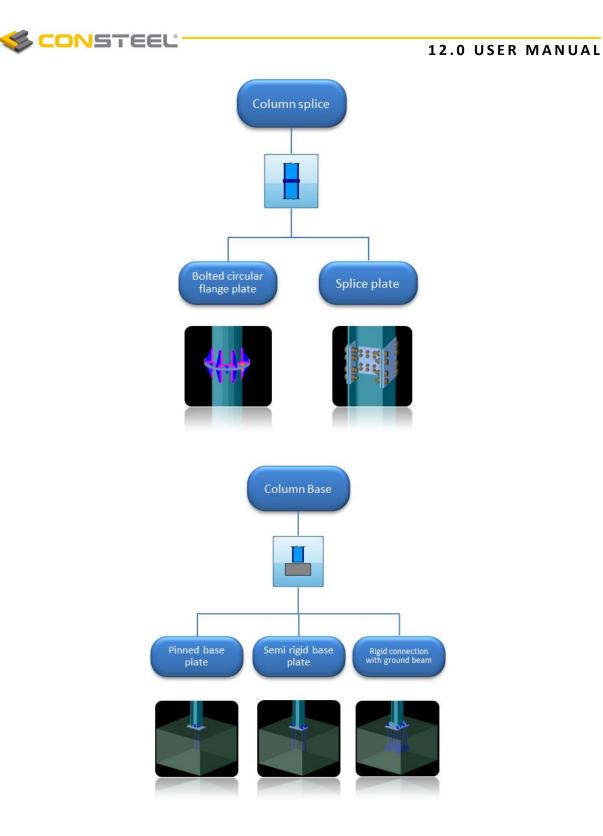
14.2 JOINT TYPES

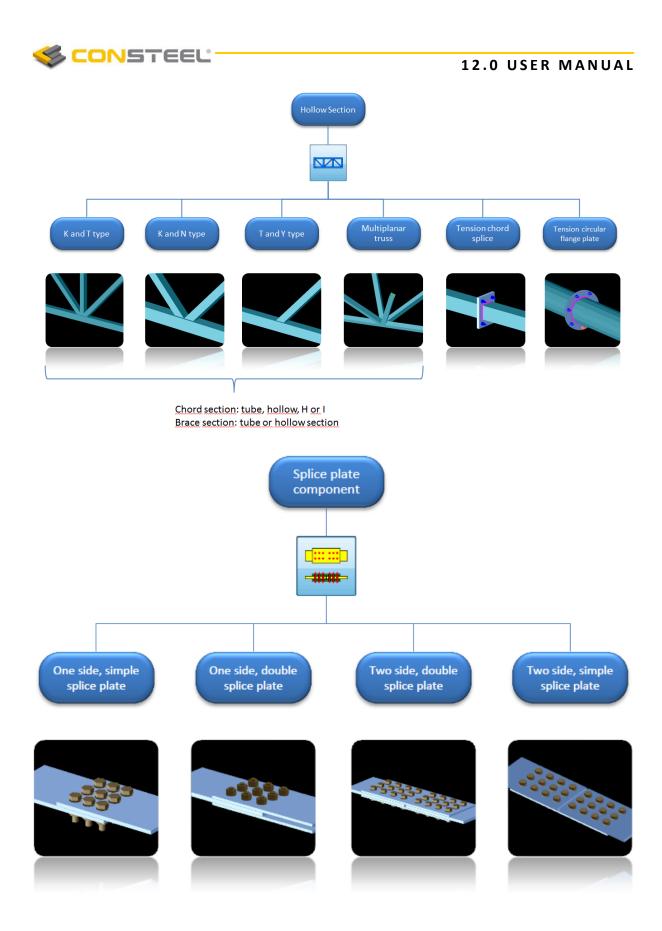
The number of the joint types is one of the most dynamically developing parts of the software, usually considering the claims of the users. The module offers a number of strengthening possibilities such as: lower or/and upper haunch with or without flange, transverse web stiffeners, flange stiffeners and flange backing plates, shear stiffeners (supplementary web plate, Morrison stiffener, single or double skew plate stiffener) etc.

At the moment the *csJoINT* module has the following joint and connection types:









14.3 CREATE JOINT

14.3.1 CREATE JOINT WITHOUT MODEL

Joint models can be created without global model by clicking on the **EDIT JOINT** icon on the **STRUCTURAL MEMBERS** tab.

Geometry	Structural members	Loads	Finite element Analysis	Global checks	lember checks	Serviceability checks
I, I	💋 🛯 🛹 🔪 (R 🐓	🎸 🐂 💒 🖂		* 🖍 🗙 1	e Fr R
						Edit joint

In this case the joint type, geometry and the loads need to be set manually. One of the advantages of *ConSteel*, that in *csJoint* it is possible to handle several connections inside of a joint. By this, a large scale of customised joints can be designed. In the following, a creation of a beam-to-column joint will be demonstrated.

Step 1 - Joint identification

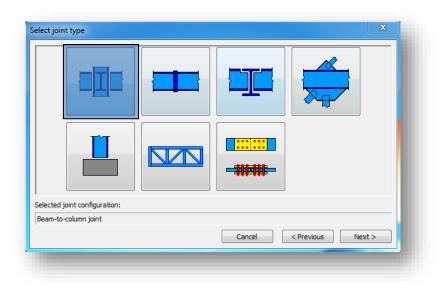
After clicking on the *CREATE* button () on the top left corner the name of the joint must be given. Comment for the joint can be optionally given, and *Default settings* of the joint can be modified (see later in *CHAPTER 14.3.5 DEFAULT JOINT SETTINGS*).

pint identification		
Name:		
Joint_15		
Comment:		
Default settings of the joint	:	
Setting1		- 🗶
betangi		9″ `
	Cancel < Prev	vious Next >
Load sections		

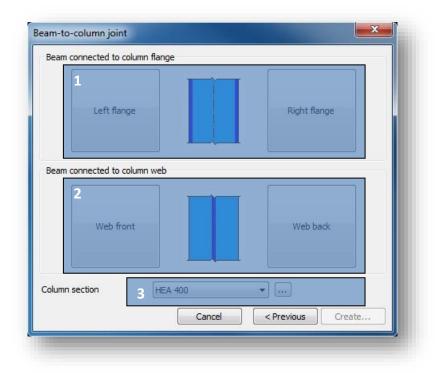


Step 2 – Joint configuration

The type of the joint has to be defined on the next dialog. *csJoint* can handle great variety of joint types, which covers nearly the whole joint standard Eurocode 3 Part 1-8. (see joint types in *CHAPTER 14.2*)



Step 3 – Connections configuration



Here (picture above) it is possible to set connections both to the flanges (#1) and the web (#2). For eligible list of connections, please see *CHAPTER* **14.2**. In this dialog, the section of the

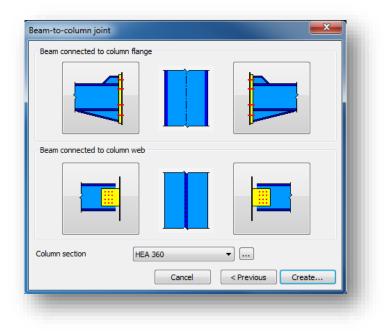


column is also has to be set. It can be chosen from the drop down menu, or loaded from the section administrator by clicking the three dot button (....).

1			
		×	
ype of the selected co			
Moment end-plate con	IPE 330	•	
		Cancel OK	

When setting a connection (picture above), the type (#1) and the cross section of the connected element (#2) has to be set. It can be chosen from the drop down menu, or loaded from the section administrator by clicking the three dot button ($\overline{\cdots}$). For eligible list of connections, please see **CHAPTER 14.2**.

After the desired connections have been set (picture below), by clicking the *CREATE* button the joint will be created.





Joint loads can be given manually and the manipulations of the connection elements can be made as a last step.

	General data					_
nt_15						
Column Stiffeners	Project:					
Joint loading						
Left flange - Moment end-pl	Name:	Joint_15				
Right flange - Moment end-c	Comment:					
Web front - Web finplate co	connerrer					
Web back - Web finplate cor	Engineer:					
	Date:	2015.03.12.				
	Joint configuration	n				
	Beam-to-col	umn joint				
	Type of connecti	on(s)				
	Left flange:	Moment end-plate connection				
				Summary of results of the whole joint		
	Right flange:	Moment end-plate connection			· · · ·	
				Left flange - Joint results for positive loading		*
	Web front:	Web finplate:		Moment resistance: Axial design resistance:	Mj.Rd = 67,21 kNm	
	Web back:	Web finplate:		Shear resistance:	Nj.Rd = 421,49 kN Vi.Rd = 195,75 kN	
	web back.	web implace:		Initial stiffness:	Si.ini = 31615,99 kNm/rad	
	Type of frame			Secant stiffness for applied moment:	S _{j.sec} = 31615,99 kNm/rad	=
	.,,	Comment:		Stiffness class:	0	
	Braced	A steel frame is qualified as stiffen	ed, when	Class of strength:	Semi-rigid (30,7%) Partial strength	
		the bracing system decreases the		onder er er er er gan.	r andar on origin	
	Our Contract Output Contrac	displacements of the frame with at	: least 80%.	Dominant compression component:		
				Column web in transverse compression (6.2.6	i.2):Fc.wc.Rd = 554,27 kN	
				Dominant tension component: Bolt row 1		
				Column flange in bending:	Ft1,Rd = 141,12 kN	
				Mode 3: Bolt failure		
				Right flange - Joint results for positive load	ina	
				Moment resistance:	Mi,Rd = 67,21 kNm	
				Axial design resistance:	Nj.Rd = 421,49 kN	
				Shear resistance:	Vj.Rd = 195,75 kN	
4 11				Initial stiffness:	Sj.ini = 31615,99 kNm/rad	Ŧ
Desmark	Place		a 🕝 🧖	Calculation V Automatic calculation	Close OK	
Document	Place		ø (27) (28)		Close OK	

For complete information about manipulating the connection, please see **CHAPTER 14.3.4 MANIPULATING CONNECTIONS**.

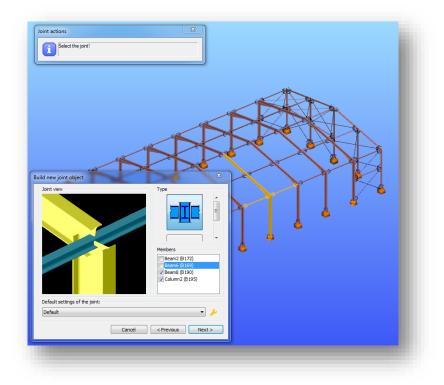
14.3.2 CREATE JOINT BY MODEL

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 members from the identified connection by removing the appropriate tick.





14.3.3 PLACE JOINT IN THE GLOBAL MODEL

After creating the joint it is advisable to place it in the global model. There are two options to do this:

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 of **JOINT DETAILS** dialog (see picture below). By choosing it, it is possible to place the joint to several places by single clicks on the 3D structural model where the same geometry exists. By clicking **END JOINT PLACEMENT** button (see picture below), loads will automatically imported from the model if analysis results are available.





	General data			<u>.</u>		
t_1 Column	Project:					
Stiffeners Joint Koading Right flange – Moment end-plate connection Beam End plate Welds						
	Name:	Joint_1				
	Comment:					
	Engineer:					
	Date:	2015.03.11.				
	Joint configuratio	on				
	Beam-to-col	umn joint				
		,				
	Type of connect					
	Right flange:	Moment end-plate connection				
	Web back	No connection				
	Web front	No connection				
	Left flange	No connection		Summary of results of the whole joint		
	Type of frame			Right flange - Moment end-plate connection		
	C Braced	Comment: A steel frame is qualified as st	ffroed when	Warning: The haunch flange thickness is less then the bear Error-0 Warning-1 Hint-0	m flange thickness!	
	Unbraced	the bracing system decreases displacements of the frame wi	the horizontal	and the second second second second		
	Character			Right flange - Joint results for positive loading Moment resistance:	Mj.Rd = 224,25 kNm	
				Axial design resistance: Shear resistance:	Nj.Rd = 535,18 kN Vi.Rd = 273,18 kN	
				Initial stiffness: Secant stiffness for applied moment:	Sj.in = 122945,53 kN Sj.sec = 122945,53 k	m/rad Nm/rad
				Stiffness class:	Semi-rigid (84,1%)	
				Class of strength:	Partial strength	
				Dominant compression component:		
				Beam flange and web in compression (6.2.6.7): Dominant tension component:	Fo.no.Rd = 470,00 kN	
				Bot row 1 Column flange in bending:	Fit.Rd = 141,12 kN	
				Mode 3: Bolt failure		
m. 1						
Document Place]	i 💭 🛄	i i i i i i i i i i i i i i i i i i i	Calculation		Close OK

The other way of placing a joint, is to use **PLACE JOINT** function, which can be found on the **STRUCTURAL MEMBERS** tab. This function is useful when more previously created joints have to be placed. One joint can be placed to several places in the model where the same geometry exists. After the joint is placed, *csJoint* automatically import the loads from the global model if analysis results are available.



It should be bear in mind that the settings in the csJoint module (profiles sizes, material grades, etc.) will not effect the 3D model. The changed values will be considered in the joint calculation only! The user has 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.

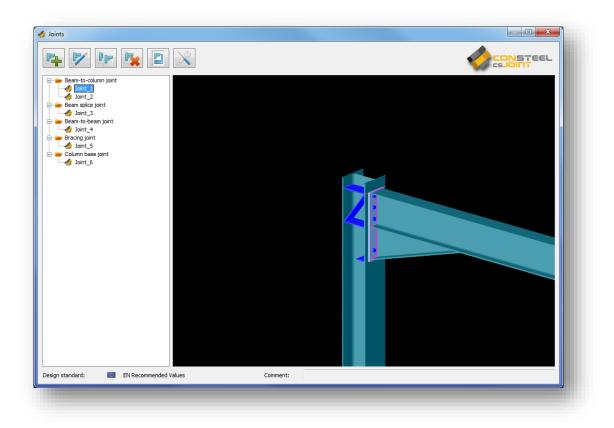
14.3.4 MANIPULATING JOINTS

No matter if a joint is created with or 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 an overview of the *csJoint* module using a beam-to-column connection as an example.





Select the joint from the tree structure on the **JOINT** dialog, and press **DETAILS** (**IDENTION**) button, or double click on the name of the joint.



14.3.4.1 JOINT DETAILS DIALOG

After the creation of a new joint, or opening an existing one for modification, the **JOINT DETAILS** dialog appears, which can be separated into four parts. The window contains the general information about the joint: Joint name, project, engineer, date, etc. These properties will appear in the joint documentation and also useful to distinguish joints from each other.



Detail to Encl plate Parameters of cross section Property Value Depth 220 mm Width 220 mm Width 220 mm Web Vickness 8 mm Flage thickness 13 mm Filet radue 24 mm Obtion of column Image thickness Ockum of topmost storey / portal frame Image thickness is less then the beam Column of an intermedate storey Average height of colum / storey Lc = 6000 mm Set with mouse Position of reference plane Lsr = 100 mm Lsr = 100 mm Share resistance: Nine = 51783 NI T7% Adequate Axial design resistance: Nine = 51783 NI T7% Adequate Axial design resistance: Nine = 5073 NI T7% Adequate Share utilization: 57 % Adequate Axial design resistance: Nine = 7446,6 Kinvriad Share utilization: 50 % Adequate Axial design resistance: Nine = 7446,6 Kinvriad Share utilization: 50 % Adequate Class of strength: Simmer veid utilizaton: Axial design resistance: </th <th>t_1 Column Stiffeners Joint loading Right flange - Moment end-p i Beam</th> <th>Column Coss-section HEA 280 Type of member Uniform cross-section</th> <th>Material S 235 EN 10025-2 Height of section [mm] 270</th> <th>3</th> <th></th>	t_1 Column Stiffeners Joint loading Right flange - Moment end-p i Beam	Column Coss-section HEA 280 Type of member Uniform cross-section	Material S 235 EN 10025-2 Height of section [mm] 270	3	
Postor of column Column of topmost storey / portal frame Column of an intermediate storey Average height of column / storey Lc = 6000 mm Set with mouse Position of reference plane Lsr = 100 mm Lsr = 100 mm Shear resistance: Vi, Rd = 517,63 kN Shear resistance: Vi, Rd = 617,63 kN Shear resistance: Vi, Rd = 617,63 kN Shear resistance: Vi, Rd = 817,63 kN Shear resistance: Vi, Rd = 817,64 kNm/rad Sistiffness in applied moment: Sistiffness in a	End plate	Property Value Depth 270 mm Width 280 mm Web thickness 8 mm Flange thickness 13 mm			
	Ш	Column of topmost storey / portal Column of an intermediate storey Average height of column / storey Lc = 6000 mm Set with mouse Position of reference plane	Lsr	A Right flange - Moment end-plate connect Warning: The upper transverse web stiffener the flange thickness! Warning: The lower transverse web stiffener the flange thickness! Error-0 Warning-2 Hint-0 Right flange - 1. Tehereset Moment resistance: Moment utilization: Axial design resistance: Shear resistance: Shear resistance: Shear utilization: Dominant weld component: halus tiffness: Secant stiffness: Scant stiffness: Class of strength: Dominant compression component:	hickness is less then the beam ickness is less then the beam Mj.Rd = 174,44 kNm S7 % Adequate Nj.Re = 517,63 kN Vj.Rd = 260,97 kN 77 % Adequate Flange weld on the lower 90 % Adequate Sj.im = 74446,46 kNm/rad Sj.sec = 74446,46 kNm/rad Semi-rigid (72,3%) Partial strength

- (#1) The first part shows the tree structure about the joint: all components of the connection can be seen and selected.
- (#2) On the second 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.
- (#3) The third part is the graphical window. Predefined views of the joint can be selected with the view tools:
- (#4) The fourth 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.

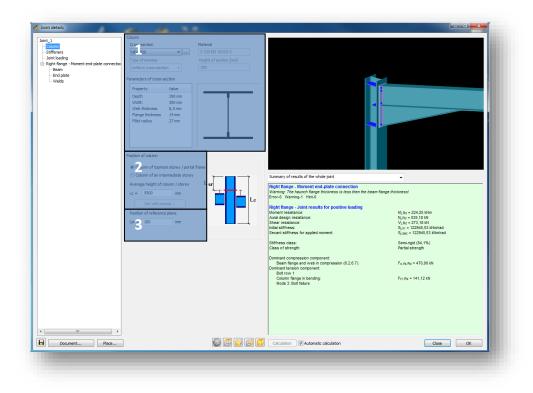
CONSTEEL

Documentation of the Joint can be started with the **DOCUMENT**... button on the bottom left corner of the dialog.

14.3.4.2 MANIPULATING JOIN

Column

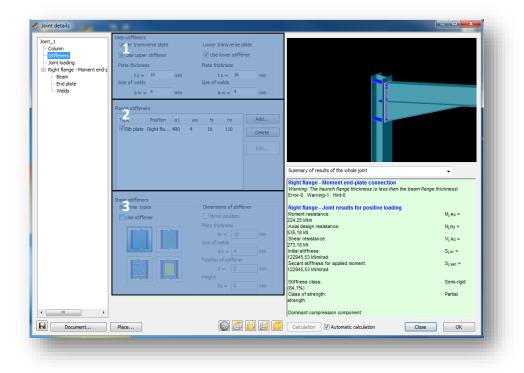
If the *Column* is selected in the connection tree, then column related properties of the connection can be set like the section size and material grade (#1). Setting the position of the column is also very important (#2). Furthermore the position of the reference plane is essential for the joint calculation.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



Stiffeners

Stiffeners can be added if *Stiffeners* is clicked in the connection tree. Upper and lower stiffener plate properties can be seen in the #1 part of the image below. Additional web and flange stiffeners can also be defined (#2). In order to place shear stiffener in the joint click on 'Use stiffener' check box (#3). Bear in mind that the position of the stiffeners are measured from the reference plane of the beam, which can be set if the *Beam* is clicked in the joint tree (see below!).





Joint loading

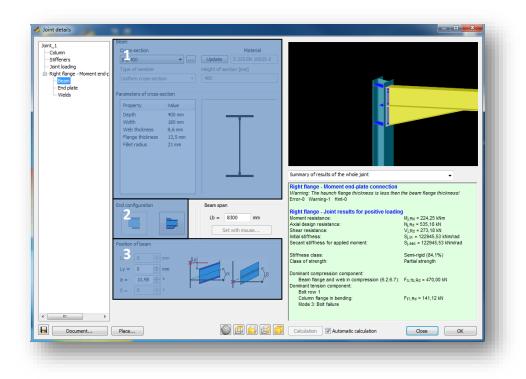
If *Joint loading* is selected, it is possible to choose from two options (#1). When "User defined joint loading" is selected, joint loads can be given manually by defining load cases, and internal forces (#2). If "Model based joint loading" is selected, *csJoint* automatically imports the internal forces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

xint_1 Column Stiffeners Joint loading ⊰-Right flange - Moment end-;	Joint laading I put of joint loading User defined joint loading Model based joint loading		
- Deam - End plate - Welds	Data of joint bads: Current load: Unrent load: Internal forces of beams connected to the joint Name of N [kr] Vy [kr] Vz [kr] Mx [k My [k Mz [k]	Summary of results of the whole joint Right flange - Moment end-plate connection Warning: The hauch flange thickness is less then Error-3 Warning: The hauch flange thickness is less then Error-3 Warning: The hauch flange thickness is less them resistance: Axial design resistance: India stiffness for applied moment: Stiffness class: Class of strength: Dominant compression component: Beam flange and web in compression (6.2.6.7): Dominant compression component: Beam flange and web in compression (6.2.6.7): Dominant compression component: Column flange in bendrag: Mode 3: 806 tailure	$\begin{split} M_{I_{1}}R_{4} &= 224.25 kHm \\ H_{1}R_{2} &= 535.16 kH \\ V_{1}R_{4} &= 273.18 kH \\ S_{1}m &= 122445,53 kHm/rad \\ S_{2}me &= 122445,53 kHm/rad \\ Semi-tigid (24,1%) \\ Partial strength \end{split}$
Document	Place	Calculation	Close OK



Beam

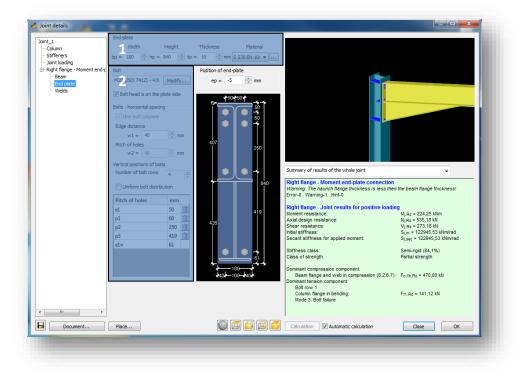
If the *Beam* is selected in the connection tree, then beam related properties of the connection can be set like the section size and material grade (#1). Beam end configuration can be set such as haunch (#2). Furthermore the beam span and position of the beam are essential for the joint calculation (#3).



End plate

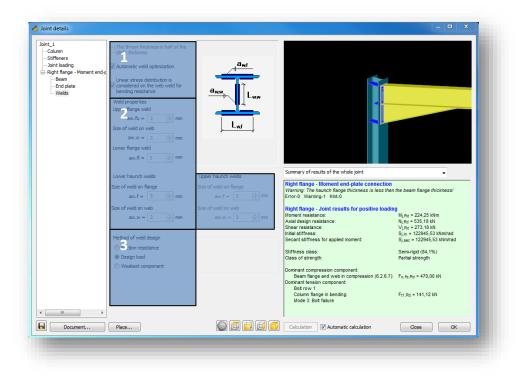
End plate properties (geometry, material) can be edited in the #1 part of the image below. Bolt properties can be seen on #2 part.





Welds

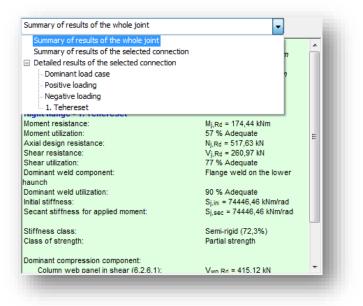
At *Welds*, it is possible to choose if welds should be optimized automatically or not (#1). If not, weld properties can be set manually (#2). Method of weld design also can be set in this window (#3)





Any changes made on the joint will take place either when pressing **ENTER** or by clicking to another editable field on the window. If *Automatic calculation* is turned on and any changes take place then the calculation of the joint is automatically rerun. It can be turned off. In that case click on the **CALCULATION** button to run the calculation.

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.

12.3.5 DEFAULT JOINT SETTINGS

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

Default joint settings can be changed and saved by clicking the **DEFAULT JOINT SETTINGS** (



icon on the **JOINTS** dialog.



⊡ Default joint settings ··· General	Setting1					
Stiffeners Bolts	Upper transverse plate Plate thickness		Lower transv Plate thickne			
Foundation bolts Welds		mm	ts =		mm	
Plates	Size of welds		Size of weld	3		
Moment end-plate conne Gusset plate	a w = 4 n	nm	a w =	4	mm	
	Shear stiffeners					
	Stiffener types			ons of stiff thickness	ener	
] [1	ts = 2	mm	
			Size	of welds		
			-	aw = 4	mm	
	Stiffener of column flange					
				F	Ť a,	
	Size of weld			×=	1	
	aw 4 mm			LS .		
	Thickness of plate					
	ts 10 mm					
reate or delete new default setti						
Setting1	Solution			Cancel		ж
				Gaucer		2

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

14.4 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:

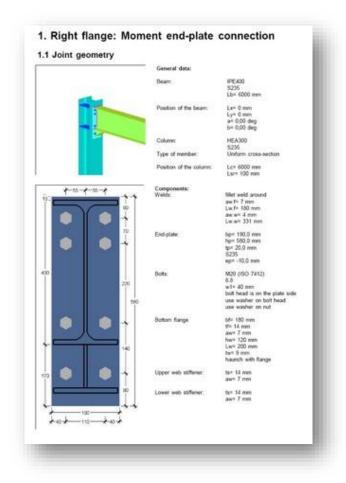
- Moment connections:
 - Moment resistance based on the plastic tension components of the individual or grouped bolt rows and compression components including the effect of axial load
 - Shear resistance of the bolts in combined shear and tension
 - Bearing resistance of plates
 - Web and flange weld 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

14.5 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.



CONSTEEL

The way for creating documentation is the same as in *ConSteel* main module (see Chapter 11 for more detailed instructions).

14.6 STRUCTURE-JOINT INTERACTION

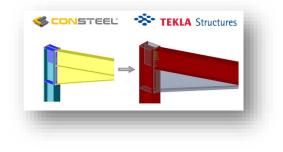
The most up-to-date structural design procedures take into account the mechanical interaction between the global structural model and its connections (rigid, semi-rigid or pinned) which generally makes the results more economic and realistic. However this approach requires a more complicated relationship 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** (**IS**) 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.

d combinations		s in the analysis by load combinations
Load combinati	ions	Apply connection stiffness
Load combinati	on-1	✓

14.7 JOINT EXPORT

All joints can be exported to *TEKLA Structures* steel detailing software. See chapter 2.4 for more detailed information.



15 CSPI – CONSTEEL PROGRAMMING INTERFACE

15.1 WHAT IS CSPI?

csPI is a new, sophisticated tool, a programming interface introduced in *ConSTEEL 11*, with which code based parametric models can be built easily and run at any time.

By writing the commands of *csPI*'s own "easy-to-read", very straightforward code language, any kind of ConSteel objects (members, plates, support, loads etc.) can be created. Basic tools of programming are also included in csPI, like the usage of *VARIABLES*, *FOR LOOPS* and *IF*-*ELSE STATEMENTS* to make parametric structure building more effective.

For complete documentation of *csPI*, with detailed **command descriptions**, **copy&paste sample codes**, downloadable **csPI scripts** visit our wiki page:

http://wiki.consteelsoftware.com/index.php/Main Page

15.2 Functionalities of the csPI panel

csPI panel can be used in 2 sizes, minimized and maximized size. When ConSteel is started, by default csPI panel appears on the bottom left part of your window in "minimized" size and it can be maximized at any time.

15.2.1 MINIMIZED PANEL SIZE

In minimized state of the panel, code can not be written, only the following functionalities are available:

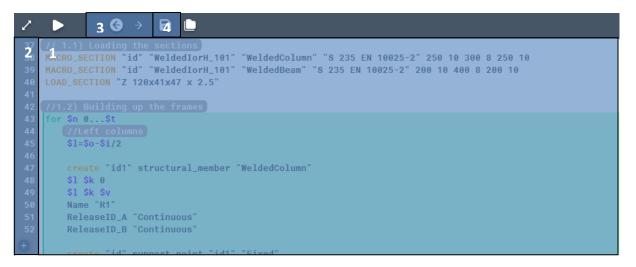


- 1. The size of the *csPI* panel can be "maximized or minimized" with the *OPEN/CLOSE* button, and model writing can be started
- 2. Existing *CSPI* file (.cspi) can be imported by clicking on the *LOAD* button of the panel.
- 3. Loaded *csPI* files can be executed with the *Run* button



15.2.2 MAXIMIZED PANEL SIZE

After the *CSPI* panel size is maximized with the *OPEN/CLOSE BUTTON*, further functionalities become available, and code writing can be started:



- 1. The **CODE INPUT AREA** is the main part of the panel. This is where variables, cycles, commands can be defined, and will be executed line-by-line after the Run button is clicked. It has more features, like:
 - a. Automatic colouring of commands, variables comments, cycles etc.
 - b. Right click on the panel will bring up the **INSERT** possibility, with which sample codes can be imported automatically
 - c. Right click on the panel will bring up the *HELP* possibility, which will direct you to the wiki. If right click is performed on a command, it directs you to the clicked command in the wiki.
- The *NUMBER OF LINES* has two main functions. It helps to navigate in the code, and helps to locate errors in the code. If the code can not be executed with the run button because of any kind of mistake (*syntax error for instance*), a dialogue which locates the exact line where the problem is, will appear.
- 3. The **UNDO/REDO BUTTON** does what is expected
- 4. The **SAVE BUTTON** can be used to save the script of the Code input area into a *.cspi* file, which can be opened later in **CONSTEEL**
- 5. Docking the *csPI* panel is also possible, which can be useful if for example a secondary monitor is available to use. To dock the *csPI* panel, *click&HolD* on the heading of the panel in maximized size, and *DRAG&DROP* to a place where the panel is desired to be moved.

15.3 BASIC TOOLS OF CSPI

15.3.1 VARIABLES

VARIABLES in a computer program are analogous to "Buckets" or "Envelopes" where information can be maintained and referenced. On the outside of the bucket is a name. When referring to the bucket, we use the name of the bucket, not the data stored in the bucket.

When creating or modifying objects with different kind of *COMMANDS*, they will require different input parameters (*such as start-end coordinates of a member, combination factors, ID's etc...*). These parameters can be defined locally as numerical values, or as a variable.

In *csPI*, definition of a variable starts with a "*\$*" sign, followed by the name of the variable, and after an equal sign, the value assignment: *\$x=1000*

Value of a variable can be:

- numerical
- text variable (string of characters) [SOON]

If *VARIABLES* are defined, they are free to use them in the following lines of the code. They can either used as command attributes, and object attributes too.

15.3.1 MATHEMATICAL OPERATORS

MATHEMATICAL OPERATORS can be used with numerical values and variables too. They can also be used inside of a command in object attributes

The following mathematical operations are implemented in csPI:

- + → Summation
- → Subtraction
- * \rightarrow Multiplication
- $/ \rightarrow$ Division
- % → Modulo
- ^ → Exponent
- sin \rightarrow Sine
- $\cos \rightarrow$ Cosine
- tan → Tangent



- $\log \rightarrow$ Logarithm (base 10)
- **round** \rightarrow Rounding
- floor \rightarrow Rounding down
- ceil \rightarrow Rounding up
- **nint** \rightarrow Nearest integer
- sqrt \rightarrow Square root

15.3.2 CYCLES, FOR LOOPS

CYCLE is the basic tool to perform repeating (same or similar) operations. csPi handles the for cycle, for which the parameters are the:

- Definition of the running parameter
- Start value of the running parameter
- ▶ End value of the running parameter

15.3.3 IF ELSE STATEMENTS

An *IF-ELSE STATEMENT* in programming is a conditional statement that runs a different set of statements depending on whether an expression is true or false.

csPi supports **one-way** (*if-endif*) and **two-way** (*if-else-endif*) if selections also.



if [expression]
statement
endif

- o if and endif is reserved word
- o expression must evaluate to true or false
- o statement1 executed if value of expression is true

► Two-way if selection:

if [expression] statement1 else statement2 endif



- if and endif is reserved word
- o expression must evaluate to true or false
- o statement1 executed if value of expression is true
- o statement2 executed if value of expression is false

In the expression, after the if command, the following operators can be used:

==	\rightarrow	equals
!=	\rightarrow	Not equal
<	\rightarrow	Less than
<=	\rightarrow	Less equal
>	\rightarrow	Greater than

 \rightarrow Greater equal

15.3.4 AVAILABLE COMMANDS OF CSPI

In this chapter, only the list of commands are collected. For a complete documentation of *csPI*, with detailed **command descriptions**, **copy&paste sample codes**, downloadable **csPI scripts** visit our wiki page:

http://wiki.consteelsoftware.com/index.php/Main Page

Load Section

With the LOAD_SECTION command, sections can be loaded from the standard library

Macro Section

With the MACRO_SECTION command, different macro sections can be created and loaded into the model

Create

With the CREATE command, the following list of *-so called-* conventional and unconventional objects can be created.

- LoadGroup
- \circ LoadCase
- LoadTransferSurface
- Structural_Member
- Structural_Plate



- Tapered_Member
- \circ Link
- Support_Point
- NodeForce
- o MemberForce
- o PlateForce

Set

With the SET command previously created *—so called-* conventional or unconventional object attributes can be setted, OR modified.

- LoadGroup
- LoadCase
- Structural_Member
- Structural_Plate
- Tapered_Member
- o Link
- Support_Point
- o NodeForce
- o MemberForce
- o PlateForce
- Get Release Name

With the GET_RELEASE_NAME command, end-releases can be found, and it's settings can be stored in a variable. If the release to be found does not exists, a new release type will be created, and also stored in a variable.

Get Support Name

With the GET_SUPPORT_NAME command, supports can be found by support conditions, and it's settings can be stored in a variable. If the support type to be found does not exists, a new one will be created, and also stored in a variable.

Select

With the select command, previously created objects can be selected, which objects can be used and modified in further commands

- Select all/Deselect all
- Select by_ID/Deselect by_ID
- Select by_Type/Deselect by_Type



- Select by_Attribute/Deselect by_Attribute
- Select by_User
- o Select invert
- Move & Copy

With the mirror and copy commands, previously created and selected objects can be moved or copied with the commands below.

- Move_selected_point_or_edge
- Move_selected
- Copy_selected
- Rotate

With the rotate commands, previously created and selected objects can be rotated with the command options below:

- Rotate_selected
- Rotate_copy_selected
- Mirror

With the mirror commands, previously created and selected objects can be mirrored with the command options below:

- Mirror_selected
- Mirror_copy_selected
- Get user input

With the Get_user_input commands, input of variables can be performed through custom defined dynamic dialogues. On the custom dialogues, different kind of controls can be placed to obtain data in a dynamic way from the user, during the execution of the code:

- TEXT Place an editbox on the dialogue, which will accept letters as input parameter
- NUMBER Place an editbox on the dialogue, which will accept numbers as input parameter
- BOOL Place a checkbox on the dialogue, which will have a value of 1 if checked, or a value of 0 if unchecked
- OPTION Place a dropdown menu on the dialogue filled up with predifened values
- POINT Place a selection button on the dialogue, with which a selected points coordinates can be obtained



- OBJECT Place a selection button on the dialogue, with which the IDs of the selected objects can be obtained and stored
- Excel command package

CONSTEEL

The excel command package covers the commands below, with which data of an excel file can be accessed from csPI, so it can be read stored and used in the script:

- EXCEL_OPEN Opens the Excel file found on the specified path in it's command attribute
- EXCEL_CLOSE Closes the lastly opened Excel file
- EXCEL_READ_CELL Reads a value from a specified cell of a specified Excel worksheet, and stores it's value in a csPI variable
- EXCEL_COUNT_FILLED_ROWS Counts the filled rows in a specified column of a specified Excel worksheet, and stores it's value in a csPI variable

15.4 TROUBLESHOOTING AND DEBUG FUNCTIONS OF CSPI

Debug function can be used to bug fix csPI codes or follow how the code is working.

Debug function has two parallel function:

Automatic code check: code check is automatically and continuously running and evaluate the written code.

Code check has two signs:

- Warning subjected line is yellow highlighted and after the code, a warning message appears. Warning sign does not block the code running
 LOAD_SECTION "HEA 200" (Warning: Section was not loaded. There is a section with the same name already loaded in
- Error: subjected line is red highlighted and after the code, an error message appears. Error sign blocks the code running
 LOAD_SECION "IPE 500" Error: Unknown command: LOAD_SECION
- Debugging function: debugging function is a very useful function to step-by-step evaluate and follow the wrote code.

Debug functions can be found on the top line of the csPI editor



Add debug point: debug points stop the code during the debugging. Several debug points can be placed. To place a debug point, step to the line with CONSTEEL

12.0 USER MANUAL

the cursor and push the button. Place debug point highlights the number of the raw (🤨).

 \odot Delete debug point: if the cursor is standing on a line where a debug point is, then the Add debug button is switched to Delete debug point button.



Delete all debug point: with a simple click, all of the placed debug points can be deleted



Turn debug ON: by clicking on the button, debugging will be started and the first running will stop at the debug point. In case of switched ON debug function, there are some extra debugging option:



Go to next row: by pushing, debugging steps line-to-line

 $| \psi |$ Go to next debug point: by pushing, debugging jumps to the next debug point

 $|\psi|$ Run till end: by pushing, debugging neglects the following debug point(s) and run till end of the code

C Turn debug OFF: by clicking on the button, debugging will be finished

Watch window: Watch window shows the actual value of the variables over the actual debugging positon

