CONSTEEL 8.0 USER MANUAL

EUROCODE VERSION



CONSTEEL

Version 8.0

08.05.2014.

www.consteelsoftware.com

CONTENT

1 General description 8
1.1 Installing the software 8
1.2 The main window 9
1.2.1 The startup window 10
1.2.2 The graphical window 10
1.2.3 The menu 11
1.2.4 The tabs 18
1.2.5 The side bar 21
1.2.6 The status bar 21
1.2.7 The windows of Object tree, Diagnostics results and Object properties 27
1.3 General structure of dialogue windows 31
1.4 General functions for tables 32
1.5 Hot keys 34
2 File handling 36
2.1 Basics 36
2.2 File types 36
2.3 Model export into StruCad 37
2.3.1 Complete model export 37
2.3.2 Joint export 40
2.4 Complete model export into Tekla Structures 41
2.4.1 Joint export limitations 43
3 Model view 45
3.1 Basics 45
3.2 Model views 45
3.3 Selection 48
3.4 Portions Manager 50
3.5 Object names and Renumbering 52
3.6 Dimensions 54
3.7 Measure 59

4 Drawing geometry 62
4.1 Basics 62
4.2 Coordinate systems 62
4.2.1 User coordinate system 63
4.2.2 Local coordinate system of bar elements 64
4.2.3 Local coordinate system of surface elements 65
4.3 Defining action points 67
4.4 Drawing 67
4.5 Modifying 69
4.6 Layers 73
5 Structural modelling 75
5.1 Basics 75
5.2 Line members 75
5.2.1 Creating line members 75
5.2.1.1 Composite column modelling 78
5.2.1.2 Composite beam modelling 79
5.2.2 Haunched members 81
5.2.3 Tapered members 83
5.3 Surface members 87
5.4 Diaphragm 88
5.5 Rigid body 88
5.5.1 Modifying rigid body 89
5.6 Convert members to plates 89
5.7 Materials 90
5.8 Supports 92
5.8.1 Point support 92
5.8.2 Line support 101
5.8.3 Surface support 102
5.9 Link elements 103
5.9.1 Connect two structural points 104
5.9.2 Connect two structural edges that are equal length and parallel 105

6 Structural Loads 106
6.1 Basics 106
6.2 Load cases and load groups 106
6.3 Load combination 108
6.4 Load types 109
6.4.1 Point load 109
6.4.2 Line load 110
6.4.2.1 Line wind load 113
6.4.3 Surface load 114
6.4.4 Load transfer surface 115
6.4.4.1 Modifying load transfer surface 118
6.4.5 Temperature load 118
6.4.6 Prescribed displacement 119
6.4.7 Change in length, tension force 119
6.5 Global imperfections 120
6.5.1 Notional Load 120
6.5.2 Initial sway 121
6.5.3 Application of eigenshape 122
6.5.3.1 Apply eigenshapes 123
6.5.3.2 Managing eigenshapes 124
6.6 Earthquake 125
6.6.1 Seismic load group and load case 126
6.6.2 Earthquake effect 127
6.6.2.1 Standard design response spectrum 127
6.6.2.2 User defined design response spectrum 128
6.6.2.3 Seismic analysis settings 129
6.7 Wind load generator 130
6.7.1 Wind load cases 130
6.7.2 Wind surfaces 131
6.7.3 Wind Generator 134
7 Structural analysis 136
7.1 Basics 136

7.2 Finite elements 136
7.2.1 Basics 136
7.2.2 Line elements 137
7.3 Model check (diagnostics) 139
7.4 Analysis settings 140
7.4.1 Basic settings tab 140
7.4.2 Load combinations tab 142
7.4.3 Load cases tab 142
7.4.4 Joints tab 142
7.5 Analysis types 143
7.5.1 First order 143
7.5.2 Second order 143
7.5.3 Static eigenvalue – buckling analysis 144
7.5.3.1 Buckling sensitivity 145
7.5.4 Dynamic eigenvalue – vibration analysis 145
7.6 Analysis results 146
7.6.1 Result types 146
7.6.2 Visualization options 149
7.6.3 Result markers 150
7.6.4 Show original shape of structure 152
7.6.5 Table of dominant values 152
7.6.6 Animation of dynamic shapes 153
8 Standard Design 154
8.1 Basics 154
8.2 Steel design 154
8.2.1 Global checks 154
8.2.1.1 Basics 154
8.2.1.2 Cross section check 155
8.2.1.3 Buckling check 155
8.2.1.4 Results 156
8.2.2 Member checks 157
8.2.2.1 Basics 157

8.2.2.2 Individual Member Design 157
8.2.2.3 Member Group Design 162
8.3 Composite design 164
8.3.1 Composite column design 164
8.3.1.1 Results 165
8.3.2 Composite beam design 166
8.3.2.1 Analysis model 166
8.3.2.2 Design model 166
8.3.2.3 Design of shear studs 167
8.3.2.4 Additional comments on composite beam design 168
8.4 Serviceability checks 169
8.4.1 Basics 169
8.4.2 Global horizontal deflection check 170
8.4.3 Global vertical deflection check 170
8.4.4 Local deflection check by members 171
8.4.5 Relative deflection check for 2 points 171
9 Section Module 172
9.1 Basics 172
9.2 Section administration 172
9.3 Section modelling 174
9.3.1 The EPS model 174
9.3.2 The GSS model 177
9.4 Section Tools 177
9.4.1 Geometrical calculations 177
9.4.2 Loads 178
9.4.3 Stresses 179
9.4.4 Effective cross section 180
9.4.5 Standard resistance calculations 180
9.4.5.1 Steel cross sections 180
9.4.5.2 Composite and reinforced concrete cross sections 182
9.4.5.3 Results 183
9.5 Documentation 184

10 Fire Design 185
10.1 Fire effect and protection 185
10.2 Analysis 186
10.3 Design 187
11 Documentation 190
11.1 Basics 190
11.2 The document tab 190
11.2.1 Start documentation 190
11.2.1.1 Documentation module 192
11.2.2 Creating snapshots 194
11.2.3 Snapshot manager 195
11.2.4 Model information 196
12 Joint module 197
12.1 Basics 197
12.2 Joint Types 197
12.3 Create Joint 208
12.3.1 Create joint without model 208
12.3.2 Create joint by model 209
12.3.3 Manipulating Connections 210
12.3.4 Default joint settings 215
12.4 Analysis of connections 216
12.5 Results and documentation 216
12.6 Structure-Joint interaction 217
12.7 Joint Export 218

1 GENERAL DESCRIPTION

1.1 INSTALLING THE SOFTWARE

In the followings we give those hardware and software requirements the *ConSteel* software cannot be run without, or their lack may result in a slow operation:

Minimum hardware requirements:

- Processor: Intel Pentium IV or equivalent
- Memory: 2 GB
- HDD: 100 MB
- Video-card: 128 MB non-integrated
- Operating-system: MS Windows 2000 / XP

Recommended hardware:

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

You will need to have administrative or power user rights for the installation. Without administrative rights you cannot install the hard lock driver and those **.dll** files, which are indispensable for the system. You will have to set the language of setup. The ConSteel will use this language for the first time you start, but you may switch to another language later. During installation the instructions of the setup shall be followed step by step. The installing program will copy the elements of the software into the Directory you have specified, then it will position the Menu of the program onto the selected place of the „START”-menu. Finally the install shield will place the starting-icon onto the desktop. The last step of the setup is the hard lock driver installation. This has no progress signal, so you will have to wait while it finishes.

As a default both 32 and 64 bit Windows operating systems are able to provide 2 GB RAM for the 32 bit applications, like ConSteel is today. When the computer has more than 2 GB RAM physically, than with the following changes in the configuration of the boot.ini file of the Windows operating system approximately 3 GB will be accessible for ConSteel.

Change the configuration for Windows Vista and Windows 7 operating systems:

To extend the available memory run the following from the command prompt:

bcdedit /set increaseuserva 3072

to switch back, run the following from the command prompt:

bcdedit /deletevalue increaseuserva

The configuration steps for the following operation systems:

- Windows XP Professional
- Windows Server 2003
- Windows Server 2003, Enterprise Edition
- Windows Server 2003, Datacenter Edition
- Windows 2000 Advanced Server
- Windows 2000 Datacenter Server
- Windows NT Server 4.0, Enterprise Edition:

backup the original **boot.ini** file and then put the "***/3GB***" switch in your **boot.ini** !

here is the sample boot.ini file with 3GB switch.

[boot loader]
timeout=30
default=multi(0)disk(0)rdisk(0)partition(2)\WINNT
[operating systems]
multi(0)disk(0)rdisk(0)partition(2)\WINNT="????" /3GB

The "????" in the above line represents the name of the operating system, you have to add the "/3GB" text at the end of that line.

1.2 THE MAIN WINDOW

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

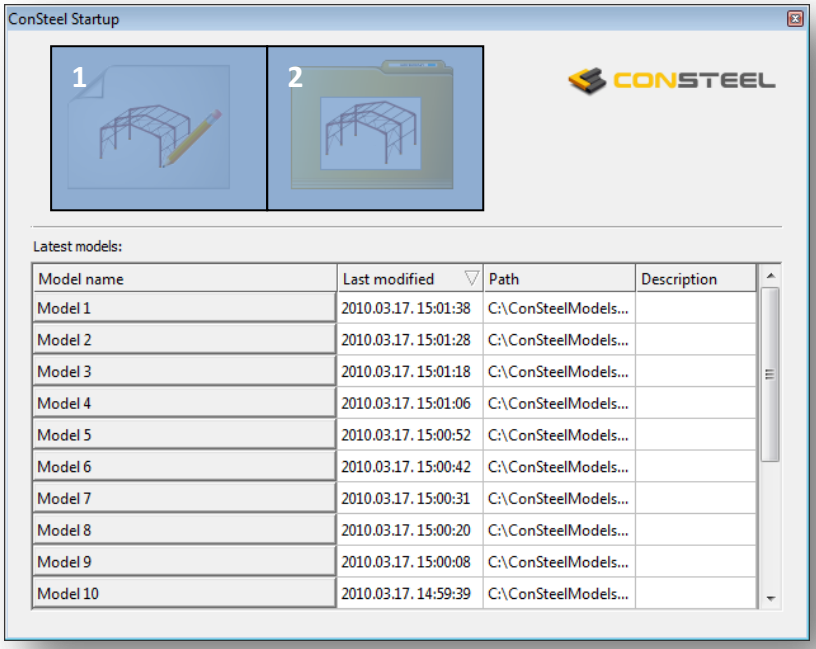


Q: What's the difference between mechanical engineers and civil engineers?

A: Mechanical engineers build weapons, civil engineers build targets.

1.2.1 THE STARTUP WINDOW

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



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

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 helps the modeling by the global coordinate system (*GCS*) and a moveable, rotatable and size adjustable grid, which is the main area for drawing. 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 should be used during the structural model manipulation:

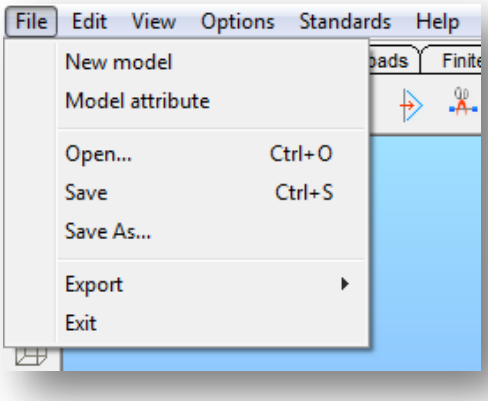
- **Move:**
  push and hold down the middle mouse button or use the four arrow buttons on the keyboard to move the model on the screen
- **Rotate:**
  hold down the ***ALT*** key and the left mouse button. The center of rotation is always adapting the actual model view
- **Scaling:**
  spin the middle 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

In *ConSteel* the 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***.

In ***FILE*** menu:

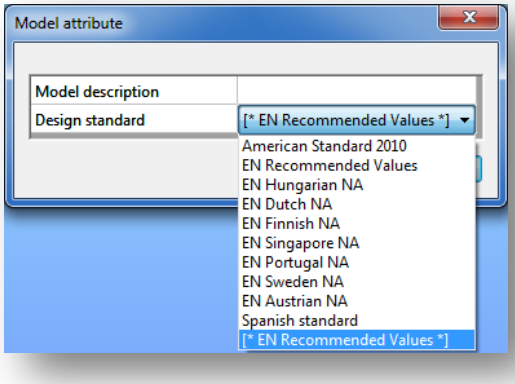


New model

New model can be created.

Model attribute

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.)

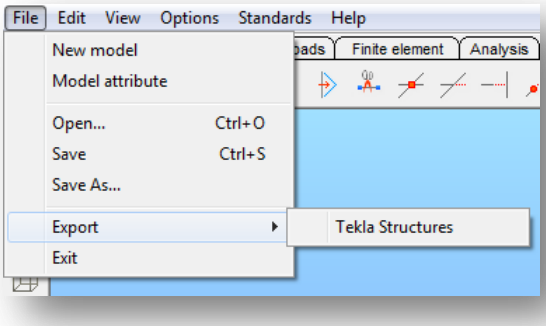


Open, Save, Save As

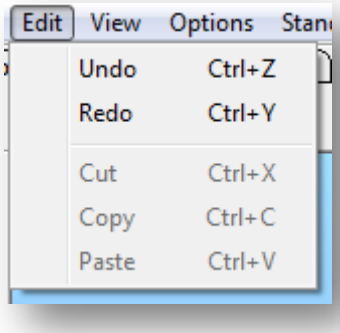
The functions are according to their names.

Export

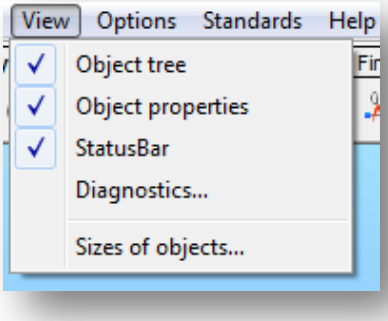
Selected part of the model can be exported to *TEKLA Structures*.



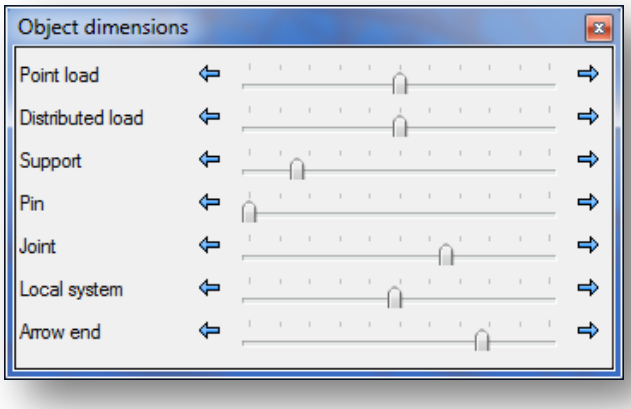
In ***EDIT*** menu the common functions can be found:



In ***VIEW*** menu:

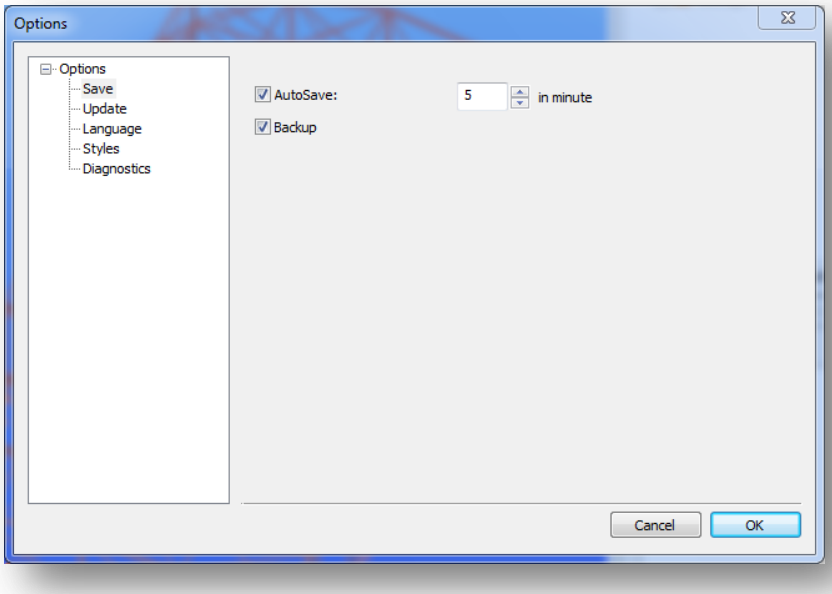


With the first four functions the visibility of the dockable windows can be switched. With the ***SIZES OF OBJECTS…*** function, the size of the objects can be changed.



In ***OPTIONS*** menu, the following settings can be found: ***SAVE***, ***UPDATE***, ***LANGUAGE***, ***STYLES***, and ***DIAGNOSTICS***.

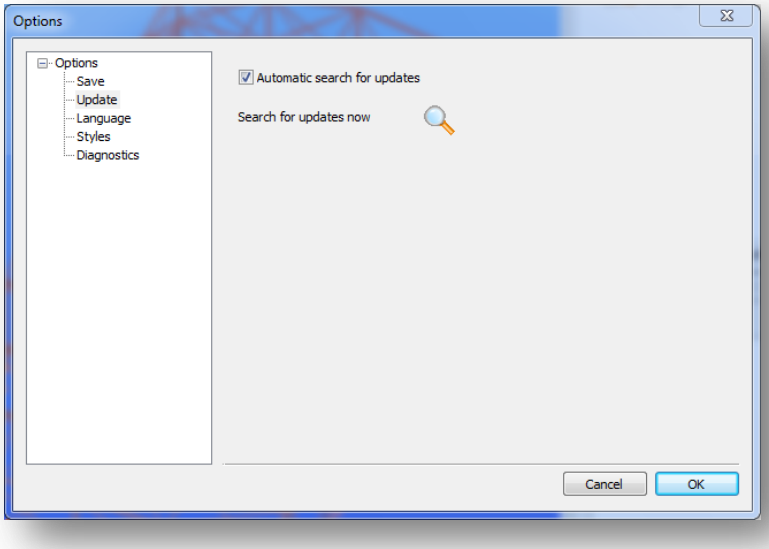
Save



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

Update

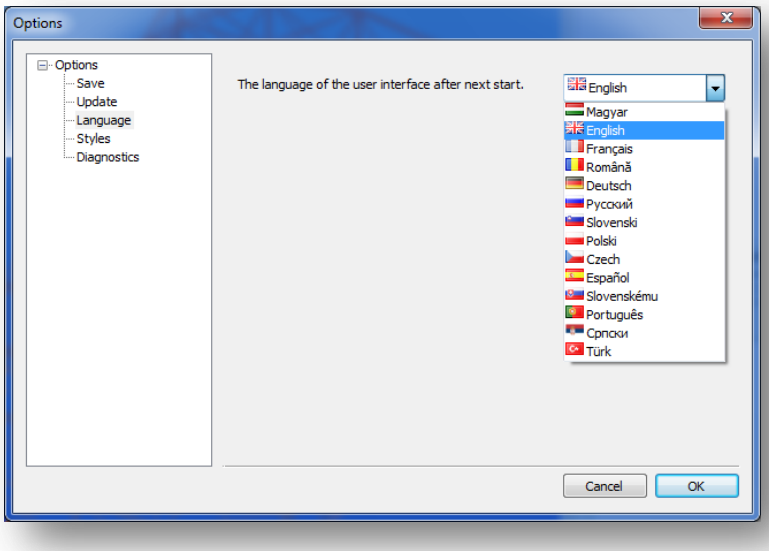
ConSteel looks for an available new version on the web at every startup. It can be turned off. The check for a new version can be performed manually by clicking on the **Search for updates now** icon.



Language

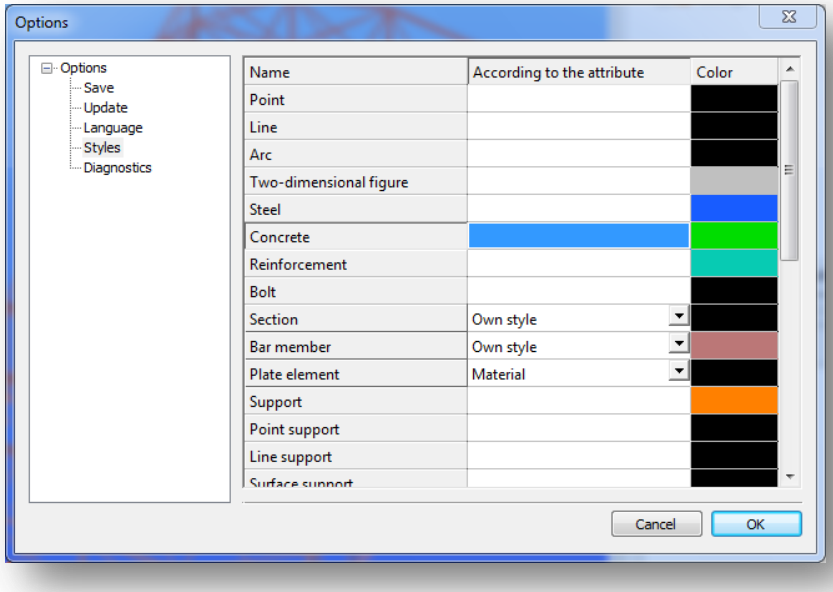
The language of the user interface can be set here from the following languages: English, German, French, Spanish, Portuguese, Russian, Chinese, Turkish, Hungarian, Polish, Czech, Slovak, Slovenian, Rumanian, Bulgarian, and Serbian.

ConSteel has to be restarted after changing.

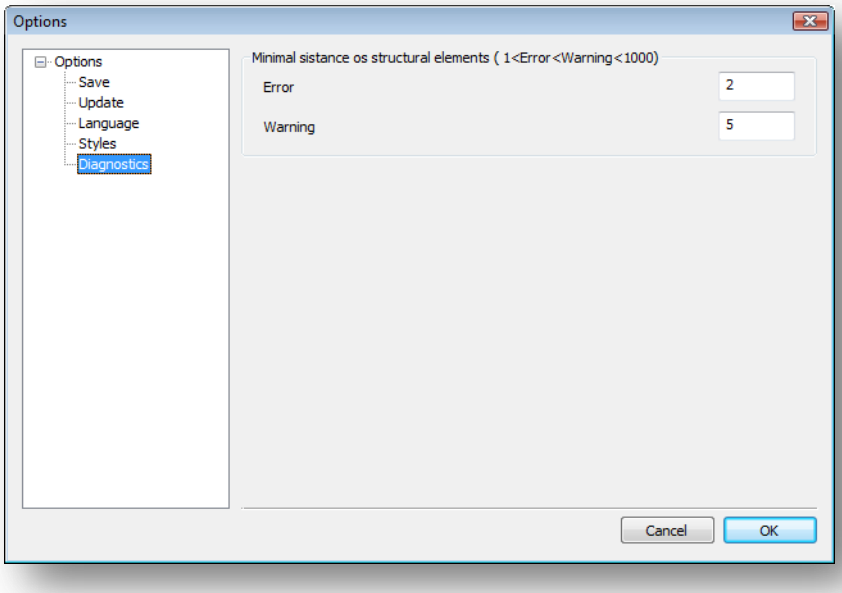


Styles

Colour of the objects can be changed.



Diagnostics



Before the analysis, the program checks the possible modeling mistakes.

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

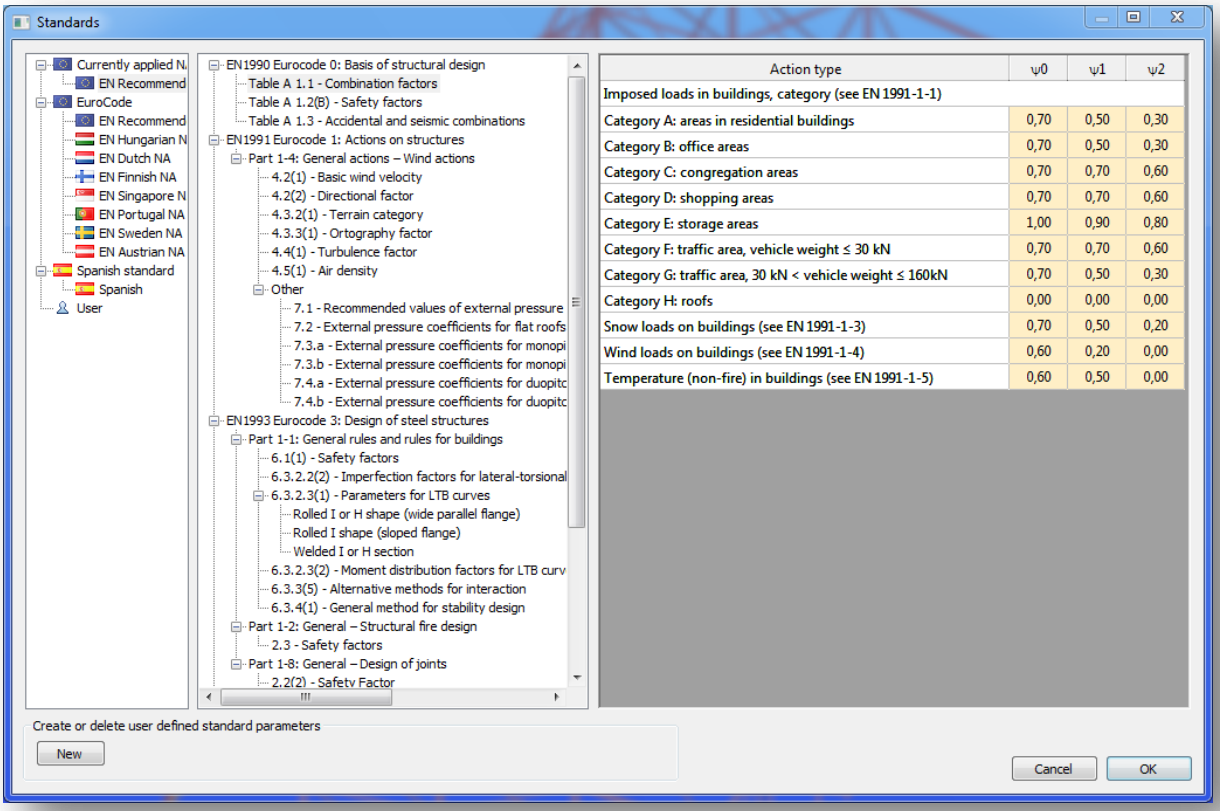
- **Error:** In case of the distance between the endpoints of two members is more than 0 but less than the adjusted value, the program sends an error message, and shows

with red sign the relevant members in the diagnostics window and not perform the analysis.

- **Warning:** In case of 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.

The ***STANDARDS*** menu

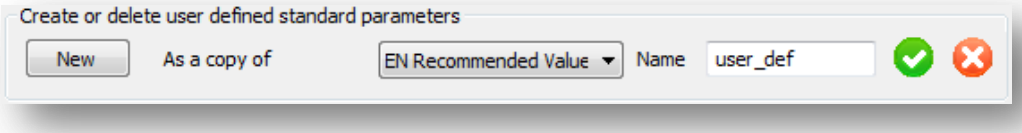
The menu provides a great opportunity to view existing and to create new standard parameters in an easy way. The used standard can be set in the ***FILE*** menu.



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

For creating a user defined standard:

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

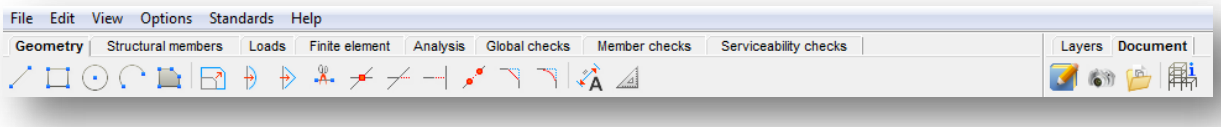


- name the new defined standard
- parameters for the new defined standard can be changed in the chapters

User defined National Annex saved to the following file:

\Documents\ConSteel\UserStandard.xml

1.2.4 THE TABS



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

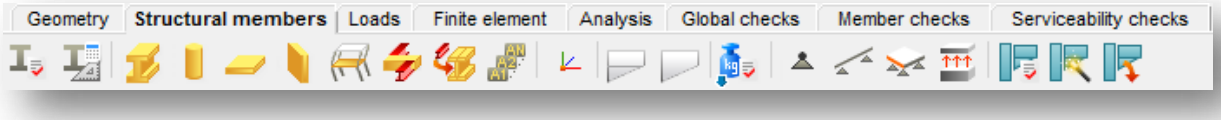
By approaching any of the icons with the cursor, the short name of the icon will be appeared.

GEOMETRY TAB



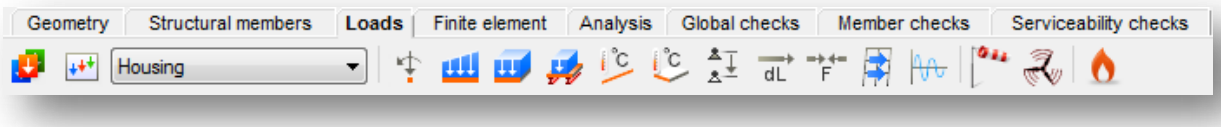
Contains all the important CAD drawing, modification functions, dimensioning and measuring tools.

STRUCTURAL MEMBERS TAB



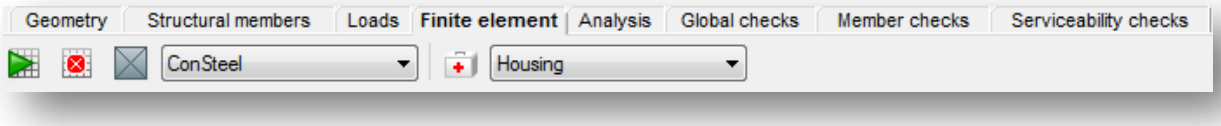
The functions related to cross-sections; structural columns, beams, slabs and walls, diaphragm and rigid body, supports and connections are collected on this tab. The automatic members to plate conversion function can be also found on this tab.

LOADS TAB



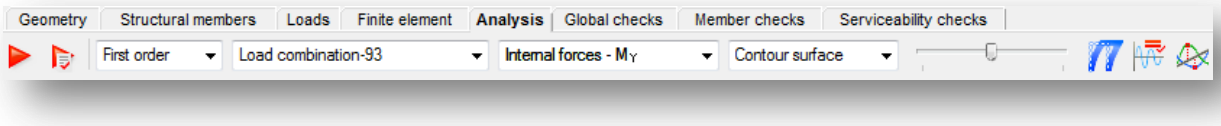
All types of loading including load cases, load groups, load combinations and unique loads placed on the structure can be created on the **LOADS** tab. Functions of wind-load generation, and fire effect and protection can be found also here.

FINITE ELEMENT TAB



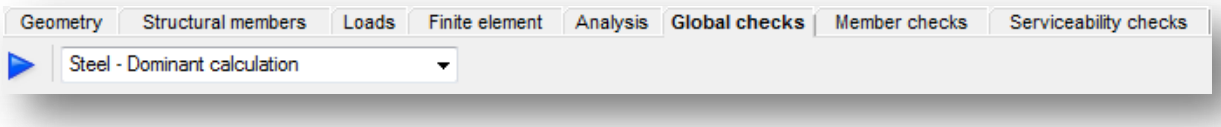
Contains functions connected with finite element mesh generation and modification, and a model check option indicating the possible modeling mistakes.

ANALYSIS TAB



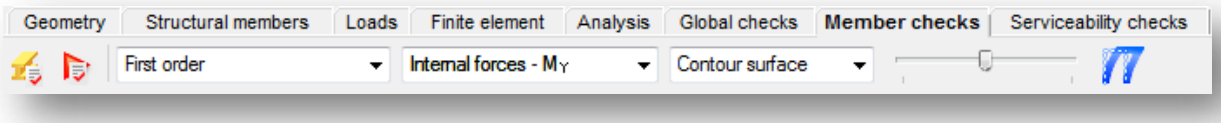
The structural analysis types can be set and executed, the results can be viewed and labeled in various forms.

GLOBAL CHECKS TAB



The **GLOBAL CHECKS** tab contains the options and results viewing possibilities of cross sections and global buckling standard checks.

MEMBER CHECKS TAB

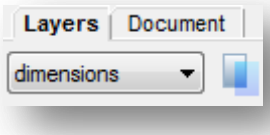


Steel member design related functions (buckling, lateral-torsional buckling and interaction stability) and the composite beam design functions can be found on the **MEMBER DESIGN** tab.

SERVICEABILITY CHECKS TAB

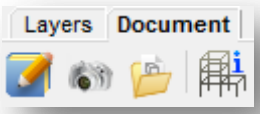
All serviceability related functions (horizontal and vertical deflection checks) can be found on the **SERVICEABILITY CHECKS** tab.

LAYERS TAB



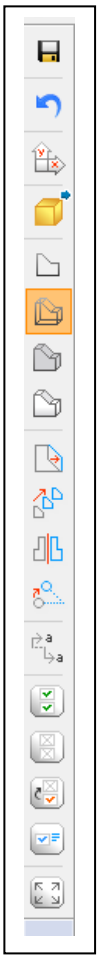
Provides functions for adjusting the layer properties.

DOCUMENT TAB



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

1.2.5 THE SIDE BAR



Save model

Undo/Redo

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

Move/Copy the selected objects

Mirror the selected objects

Rotate the selected objects

Make a section (works on shell models only)

Select all objects

Deselect all objects

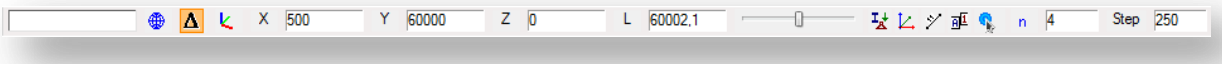
Invert selection

Select by properties

Fit view (Ctrl+0)

The side bar contains the mostly used functions for modeling.

1.2.6 THE STATUS BAR



The first field is a progress bar, shows the progress of the actions.

The next 3 icons on the status bar  are for specifying the interpretation of the coordinates during drawing/modeling actions, while values are entered manually after pressing the appropriate letter (“X, Y, Z, L” for coordinate axis or length into a direction, or “a, b, L” for polar coordinates). The user could influence the interpretation of the manually entered values with the following settings:

 **Global**

 **User**

 **Absolute**

 **Relative**

Descartes Polar

Even if you are in Descartes coordinate system, and you have picked the first point of a new structural beam, 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. Than press the Enter key.

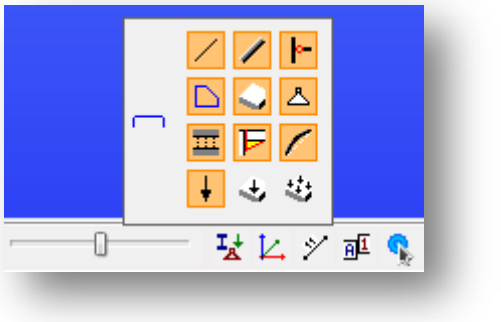
By moving or clicking the slider with the right mouse button, the size-representation of the objects can be changed. The next four icons allow the sophisticated visibility adjustment. Approaching any of these icons with the cursor a group of graphic symbols will appear ordered into a matrix shape.

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

Setting this option on any of the 4 visibility matrix, 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



Visibility of lines, created with line, circle and arc function of Geometry menu.

Visibility of the structural members created with beam or column function in Structural members menu. Switching these members non visible their centerlines might still be visible accordingly to the status of line visibility settings .

Visibility of pins (end releases).

Visibility of the 2 dimensional shapes. Switching these 2 dimensional shapes 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 non visible the surrounding lines and 2d shape components might still be visible accordingly to the status of line visibility settings of lines and 2d shapes .

Visibility of supports.

Visibility of link elements.

Visibility of placed joints symbols.

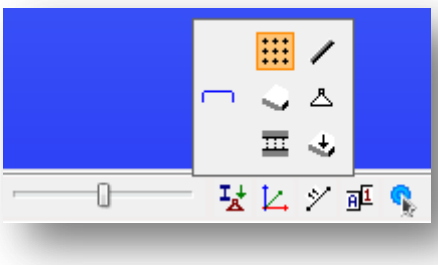
Visibility of initial bow imperfection.

Visibility of point, line and surface loads.

Visibility of load transfer surfaces. Switching these load transfer surfaces non visible their surrounding lines might still be visible accordingly to the setting of visibility of the lines .

Visibility of distributed surface loads.

Visibility options of the grid and the local coordinate system



Visibility of the grid

Visibility of the axes of the local coordinate systems of structural members (beams and columns)

Visibility of the axes of the local coordinate systems of 3 dimensional objects (plates, slabs and walls)

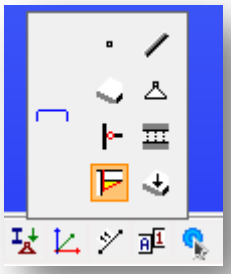
Visibility of the axes of local coordinate systems of supports.

Visibility of the axes of local coordinate systems of link elements.

Visibility of the axes of the local coordinate systems of the load transfer surfaces.

When the visibility of the object is off, the axes of the object also will not be visible!

Visibility options of object names



Visibility of the number of finit elements. (Visible only on the **Finite element** and **Alnalysis** tabs!)

Visibility of the name of bar elements.

Visibility of the name of 3 dimensional objects (plates, slabs and walls).

Visibility of the name of supports.

Visibility of the name of pins (end releases).

Visibility of the name of link elements.

Visibility of the name of placed joints symbols.

Visibility of the name of the load transfer surfaces.

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

Visibility options of labels



Show materialsgrades.

Show shapenames.

Show the thickness of plates in mm.

Show the load intensity.

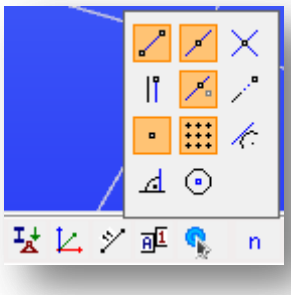
Show initial bow imperfection.

Show the units of quantities.

Show labels of the local cordinate axes.

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:

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

  - % 25, clicking the n icon, the percentage will be calculated to a length of the approached element, and from the approached end this length will be measured by the snapping points. Usually there is a rest distance at the end of the element
  - d 1000, clicking to the % icon, a distance can be specified. This length will be measured by the snapping points from the approached end of the object. Usually there is a rest distance at the end of the element
  - n 4, clicking to the d icon, the number of division can be specified. This snapping points show the endpoint of the subdivided length. 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 to 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.

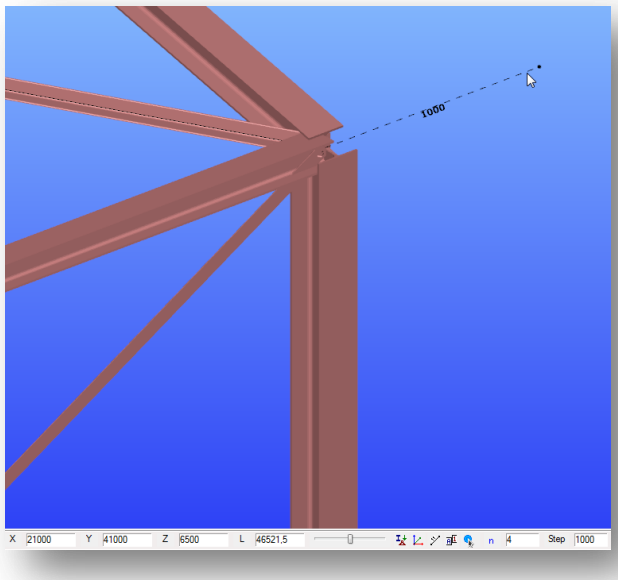
Switches On/OFF the snap point of the raster.

Snapping point to a tangent of an Arc / Circle from one point

Snapping point to a perpendicular from one point to an object (Line or Arc/Circle)

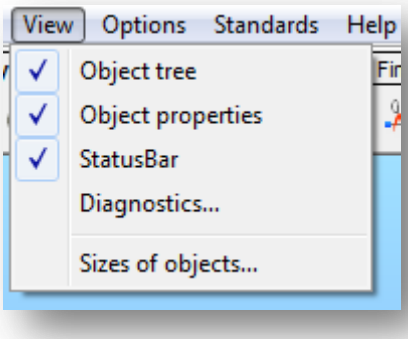
Snapping to Center point of Arc / Circle.
For snapping the center of linear elements, you have to use the appropriate settings of snap divided points, see above!

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.



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

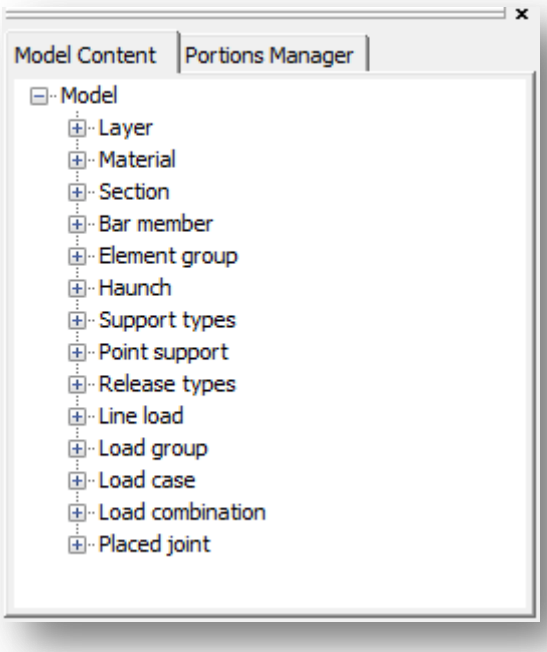
The visibility of these windows can be switched ON / OFF in the ***View*** menu.



Object tree window

Object window has two tabs:

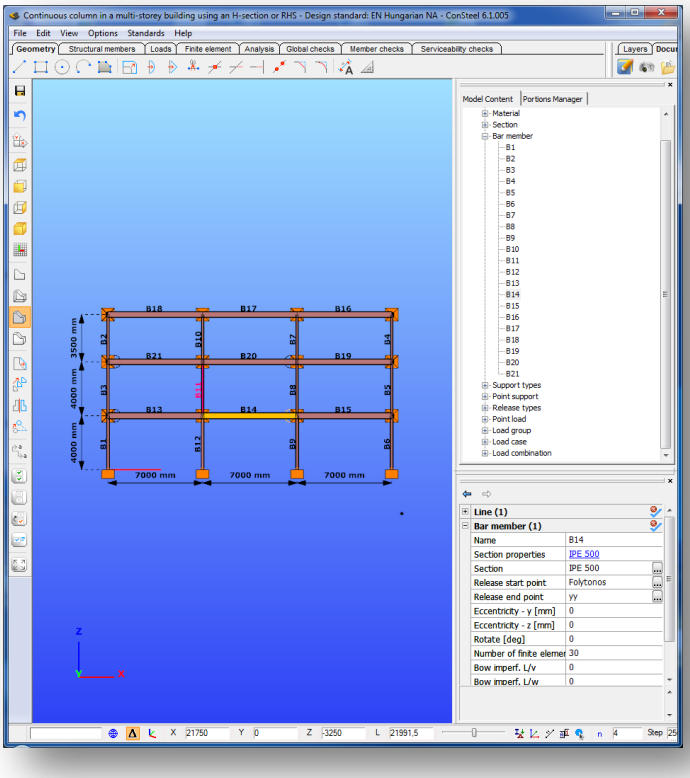
MODEL CONTENT tab



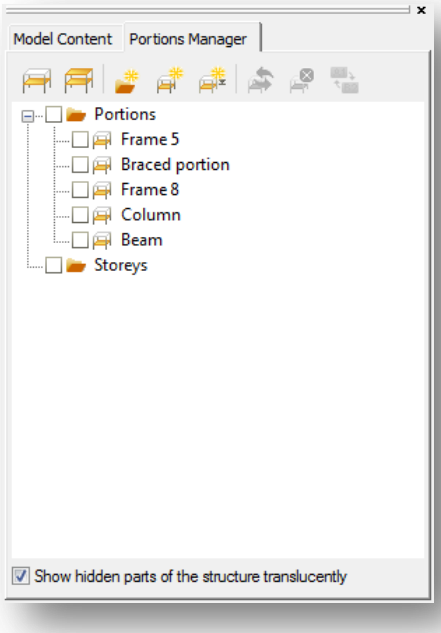
By defaults this tab contains all the predefined basic object types which are necessary for building up a structure.

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 space. Multiple selections of objects are possible in this tree.



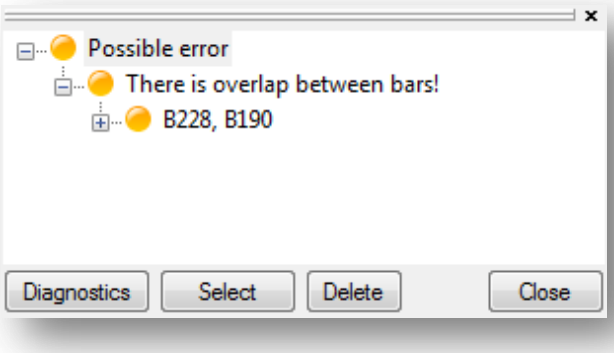
PORTIONS MANAGER tab



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

Diagnostic window

In case of a 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 is the error messages appearing in red color.

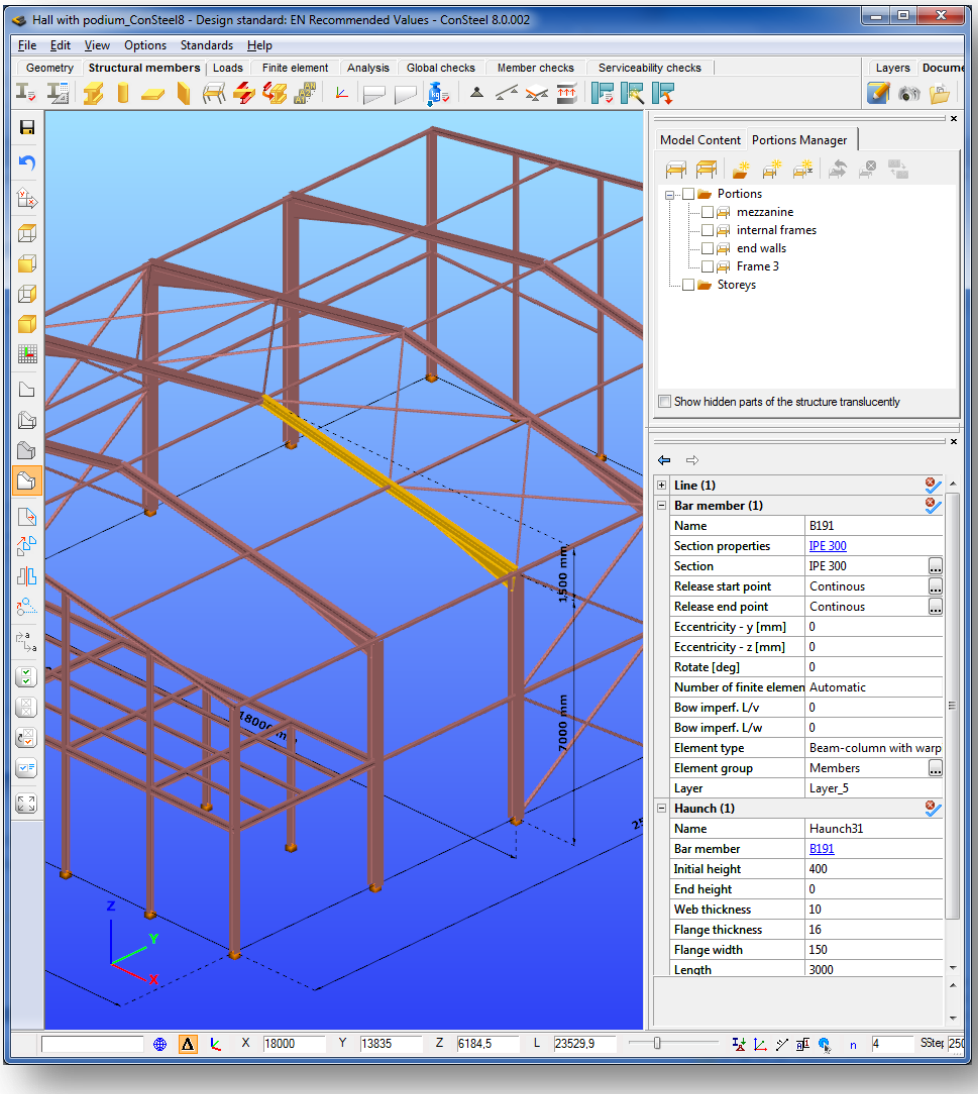
The second type of the diagnostic result is the possible errors in yellow color.

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 space. 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

Selecting one object in the model space, all the relevant properties will appear in this window. The values of the parameters can be overviewed, and in most cases these parameters can be changed.



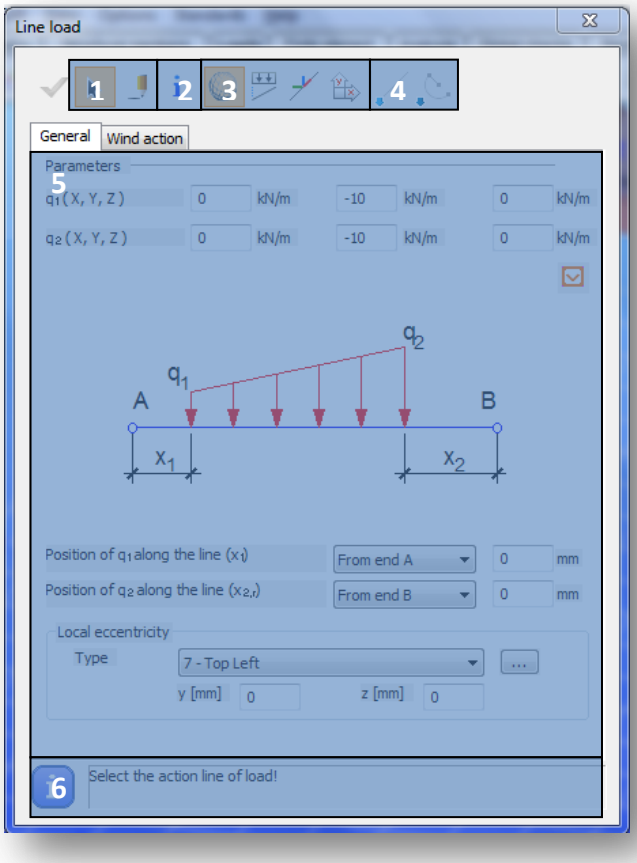
In the lower table all the selected objects (and sub-object) appear, and after expanding one object, all the parameters of it can be seen, and modified if possible. In case of multiple

objects selection only the identical parameters appear, however the different ones can also be changed to identical.

With the unselect button ( ) 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 same structure in order to make easy the orientation in the labyrinth of tools and functions. The usual parts of these dialogues are described below:



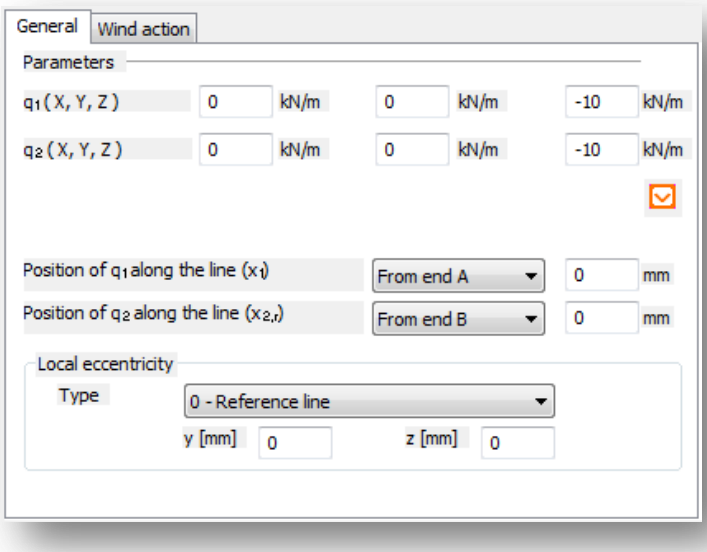
1 - the method of placement (assign to an element or draw from one point to another)

2 - extract data from a previously placed object. All the parameters are set to the same as for the selected object

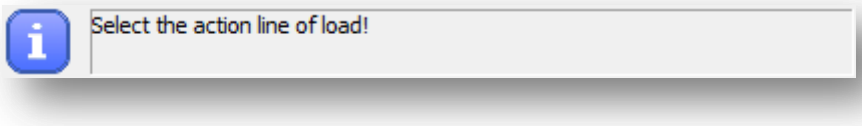
3 -  coordinate systems in which the directions are considered

4 -  additional drawing functions if the placement is by drawing ()

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



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



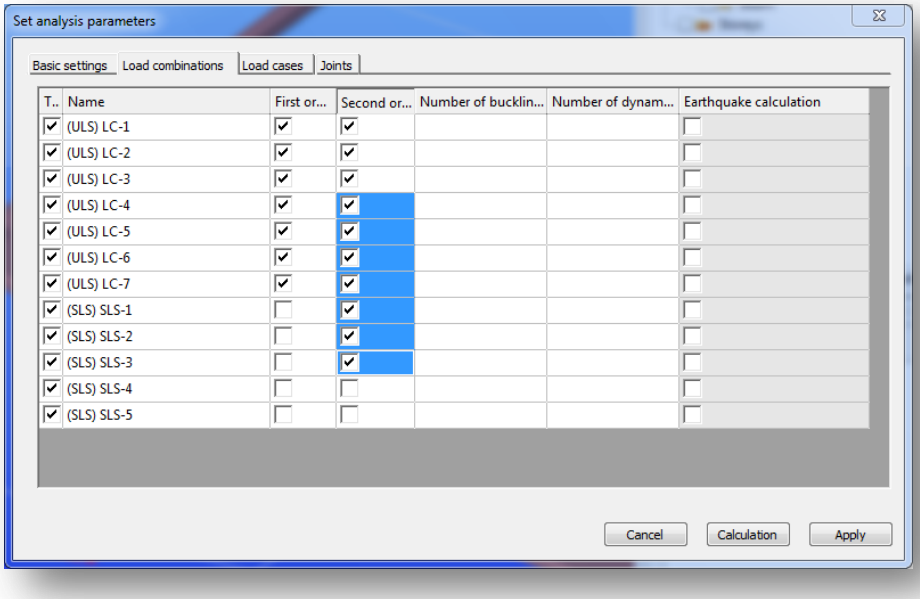
Every icon on these dialogues has a tooltip with the name of the function on it. The text will appear when the mouse approaches the icon

1.4 GENERAL FUNCTIONS FOR TABLES

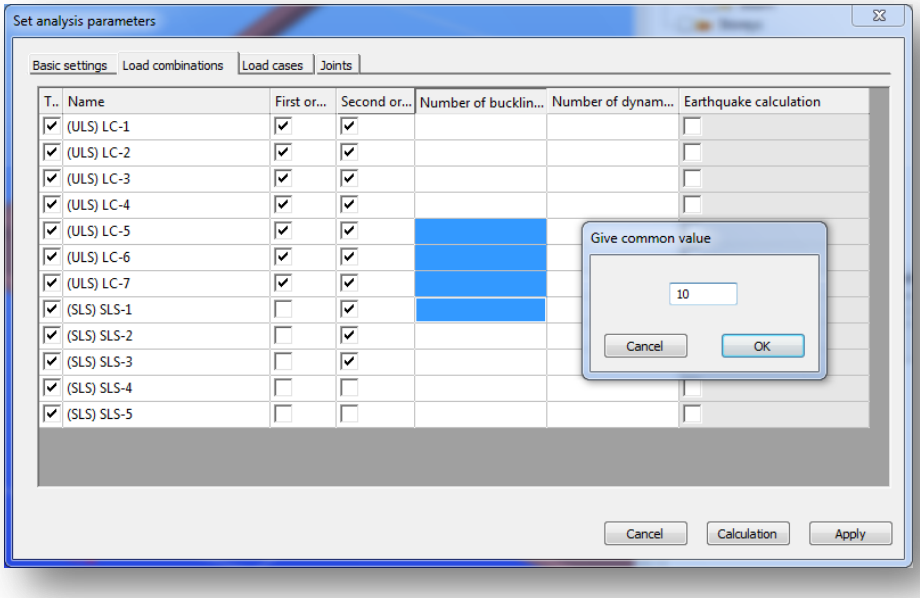
Basically there are two types of tables in *ConSteel*: the input tables and the output tables. Since tables are usually used for working with large amount of data, both table types have special features making the data managing more comfortable.

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 the there is a checking option in the table the multiple checking is possible for the selected cells clicking on the right mouse button



- if the there is 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



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 11)

1.5 HOT KEYS

The hot keys can significantly speed up the modeling work. In *ConSteel* the following hot keys can be used:

- **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**
- **ESC: unselect all or terminate (or abort) the last action**
- **Delete: delete selected objects**
- **F1: open Help system**
- **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 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
- →: 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

2 FILE HANDLING

2.1 BASICS

The file handling in *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 other folder and opened. Normally the **.csm** file extension is associated with the *ConSteel*, these files can be opened by double click.

2.2 FILE TYPES

The following file types are handled in *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. Open and save.

.csm~: the ConSteelModel file created by 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. Backup file is created at every manual save and stores the previous saved version of the model. If necessary .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.

2.3 MODEL EXPORT INTO STRUCAD

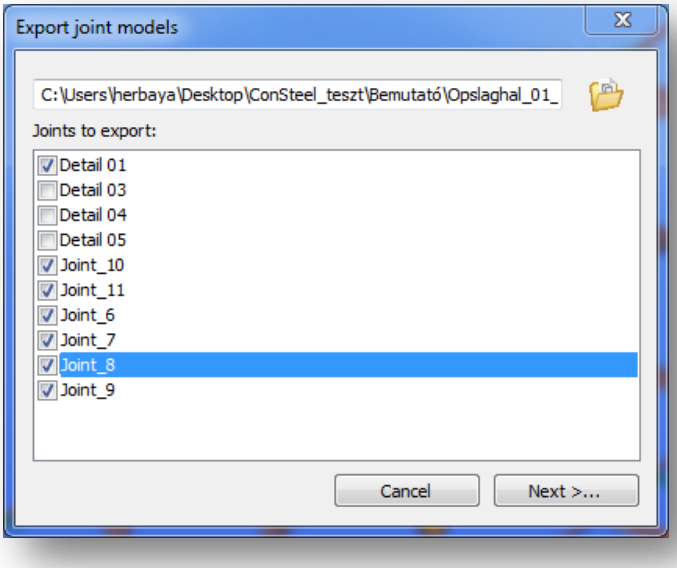
A special interface option has been developed in order to make the *StruCad* export functionality complete. Using this unique feature not only the beam and bar objects of the structural model can be exported to *StruCad* but all the details of the calculated joints. Moreover if a joint is placed to the structural model, the export process identifies these joints and the model will appear with their completely developed and detailed connections in *StruCad*.

2.3.1 COMPLETE MODEL EXPORT

To export the complete model into *StruCad*, follow the steps below. First click on the ***FILE*** menu and select ***SAVE AS*** option. Add the name of the StruCAD **.snf** file. The length of the filename is maximized to 8 characters and does not contain spaces or non-English alphabet letters (for example letters with accent). Select the **.snf** file type and click on the ***SAVE*** button.

As the first phase of the export, ConSteel will check the defined and the placed joints. When there are some defined joints which were not placed in the model, the system will worn the user listing all the defined joints in a window, where the placed joints are checked, and the unplaced joints are unchecked. This window will appear only when there are placed and unplaced defined joints at the same time.

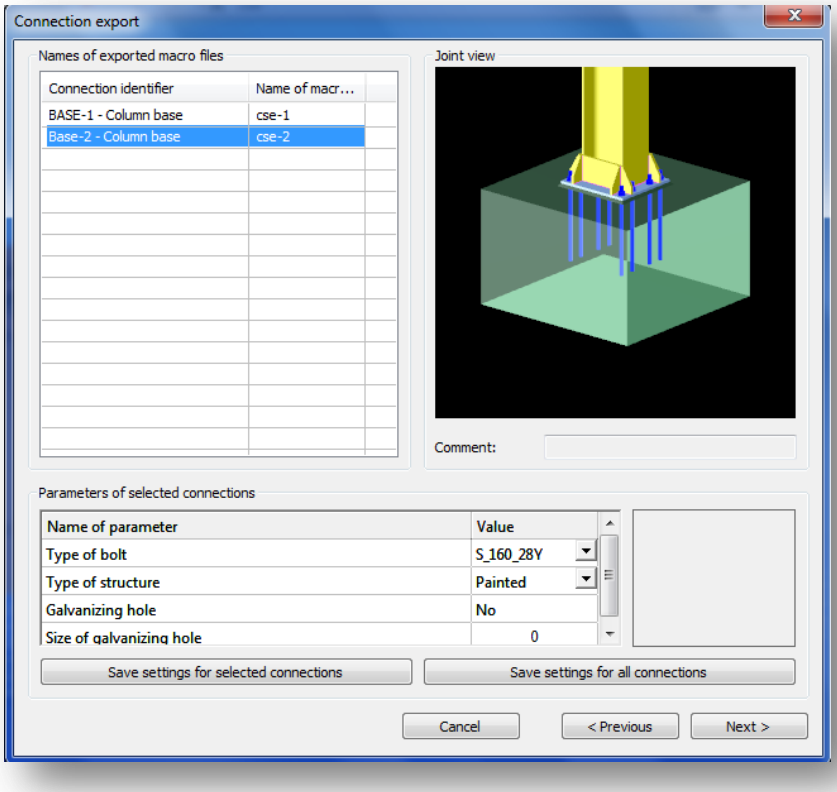
When created joints are not placed (but at least one joint has been placed) in the model than the joints can be selected to be exported in the following dialog:



In the first field you can specify a folder for the connection macros on the top of the window.

In case of a placed macro is unchecked, than this macro will not be exported!

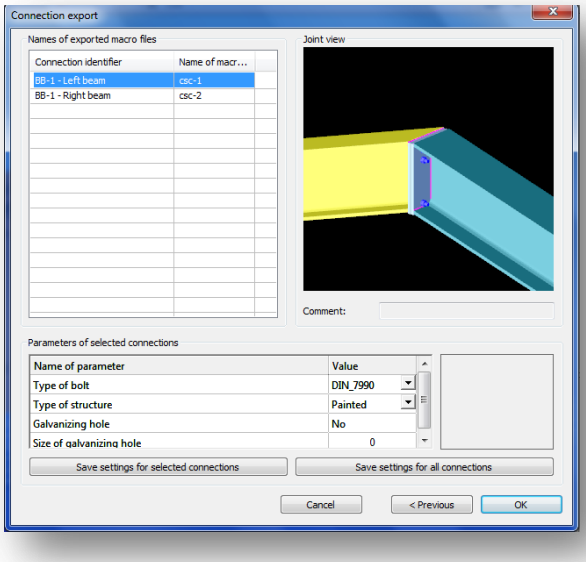
Click on the ***NEXT >…*** button, the user could specify additional parameters to the different types of connections.



A special dialogue window appears for every joint type where the following settings can be made: name of the exported joint macro file; other joint parameters which are needed for the *StruCad* joint macro but not necessary for the joint calculation in *csJoint*. These parameters can be different for each connection. If you click on the ***SAVE SETTINGS FOR SELECTED CONNECTIONS*** then the parameters will be saved ONLY for the selected connection. The other option is the ***SAVE SETTINGS FOR ALL CONNECTIONS*** button; in this case the parameters will be the same for all the connections in the list.

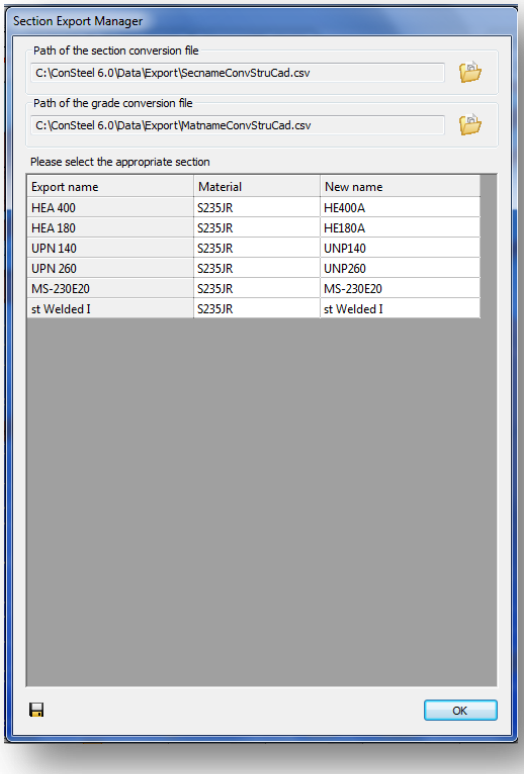
By clicking the ***NEXT >*** button, the selected connection of the next type will appear for editing their parameters.

Clicking the ***< PREVIOUS*** button, the user could return to the previous screen.



When the window of the last connection type appears, after pressing the ***OK*** button the export process will continue.

The following task is to check - and in case of need, to change - the names of sections and the material grades, for the case these names are used differently in ConSteel and in the used default settings in StruCAD. It could be done manually or using some prepared files.



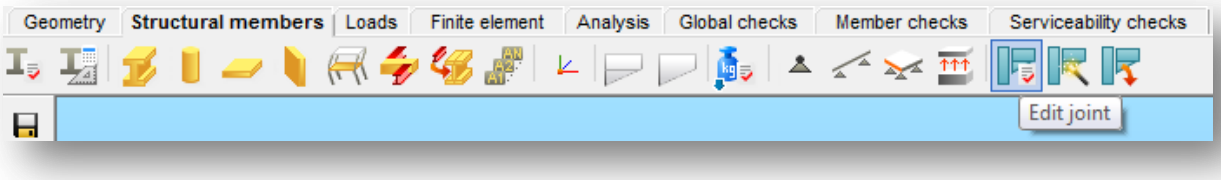
The user could select the right “name conversion” files by clicking the appropriate icon. After changing any of the section names or material names, the user can save the changes clicking the icon at the bottom left of the above window for further use.

In order to import the model into *StruCad*, click on the dedicated *ConSteel* icon in *StruCad* or use the ***csIMPORT*** command.

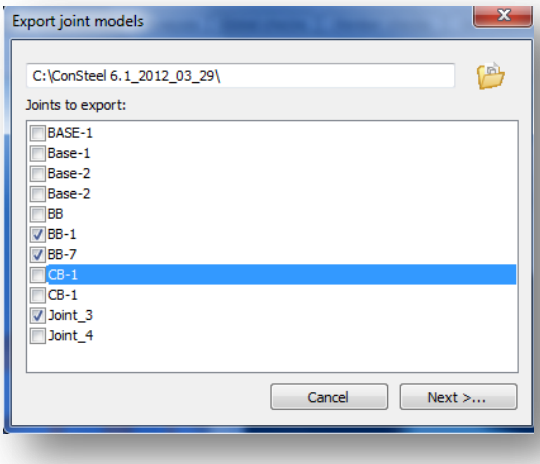
The exported macro files of the joints must be copied into the relevant *StruCad* model’s **mac** folder. Afterwards these joints will appear among the Local Macro end types in *StruCad*.

2.3.2 JOINT EXPORT

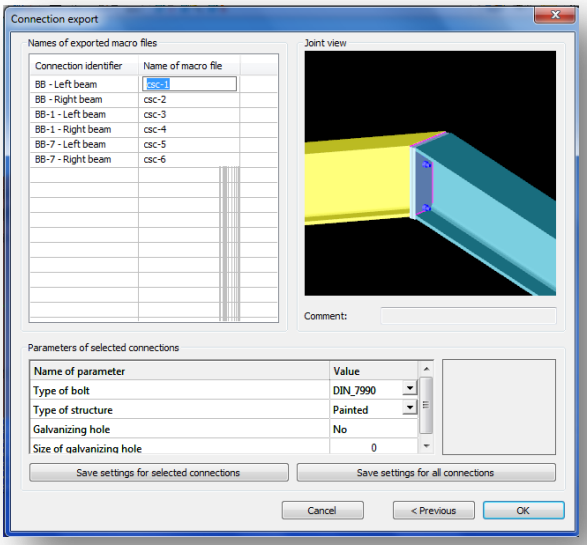
For exporting only joints, the ***EDIT JOINT*** function has to be chosen from the ***STRUCTURAL MEMBERS*** tab.



By pressing ***CTRL+SHIFT+E*** keys, all the defined connection will appear in a window:



To choose a destination folder and to give the optional parameters are same as above.



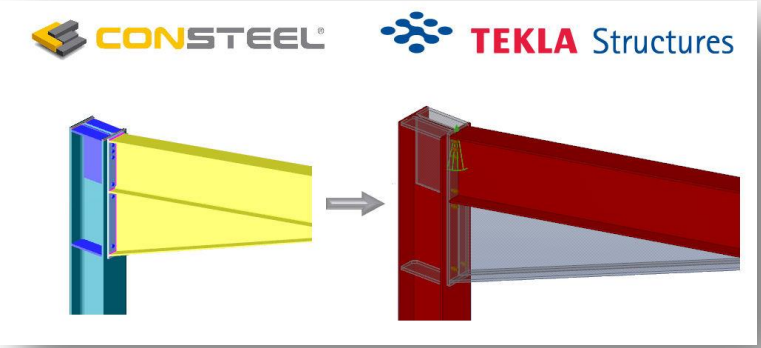
The name of the *StruCad* macro file can also be changed but bear in mind the following naming conventions:

- The name of the exported joint macro file has a prefix (for example csb-). This prefix calls the relevant macro in *StruCad*. Do not change or delete this prefix! (If you do so a warning appears on the screen.)
- The number next to the prefix can be changed but must not be longer than 6 characters. This suffix should contain only numbers!
- Make sure not to use a macro file name more than once in a model. Each macro name should be unique.

After the export the files could be sent to the detailer.

2.4 Complete model export into Tekla Structures

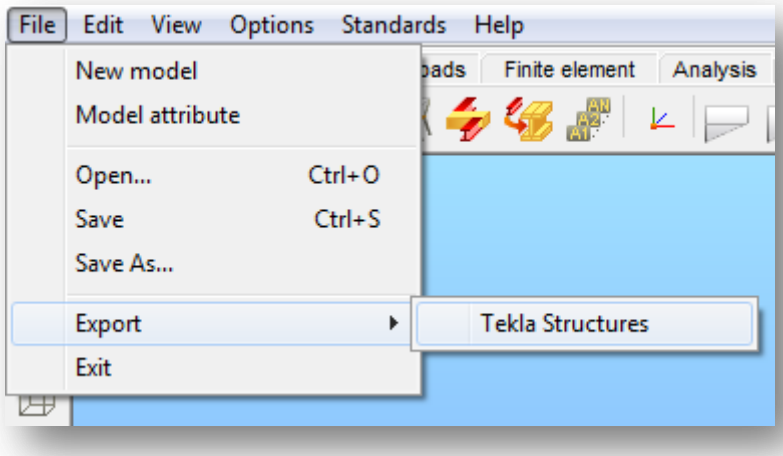
The smooth and quick transition from the structural model to the detailing model can save significant time and cost, therefore *ConSteel* has a high level interface module to *Tekla Structures*. Besides the export of the global structural model (ie. beams, columns, slabs, walls) all the joint models appear in *ConSteel* have the corresponding *Tekla* component, so the designed structural model with the placed joint models can be converted in one piece easily into detailing model without additional modeling efforts on the joints. This unique interface can save significant detailing time and therefore can appreciably reduce 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 installed in order to export model from *ConSteel* to *Tekla Structures*.

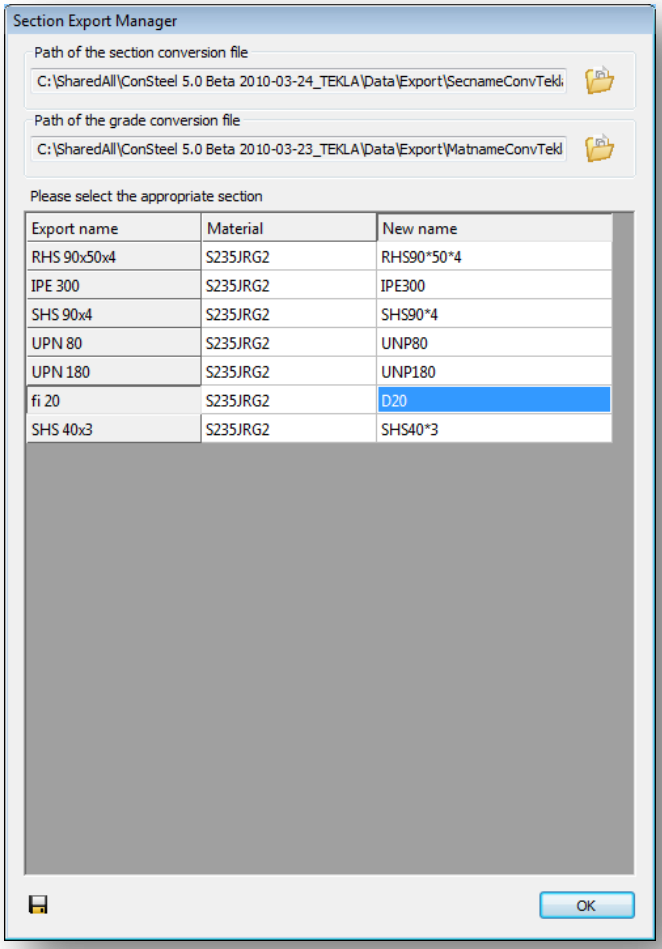
ConSteel export to *Tekla Structures* can be used from version 14 onward.

To export a model from ConSteel it is important to run both program 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***.



A dialog window opens asking for selecting model parts to be exported. Bear in mind that only those joints will be exported which are placed in the model and selected before exporting. The next step is to select a reference point and click on the ***EXPORT*** button.

After selecting the reference point in *Tekla* the ***SECTION EXPORT MANAGER*** window appears. It is possible to change the conversion file for grades and for sections, or edit the section name and grade manually.



If manual changes have been made then 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.

2.4.1 JOINT EXPORT LIMITATIONS

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)

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*

Apex haunch (106)

- No upper haunch

Stiffened end plate (27)

- There is no web stiffener plate in *Tekla*

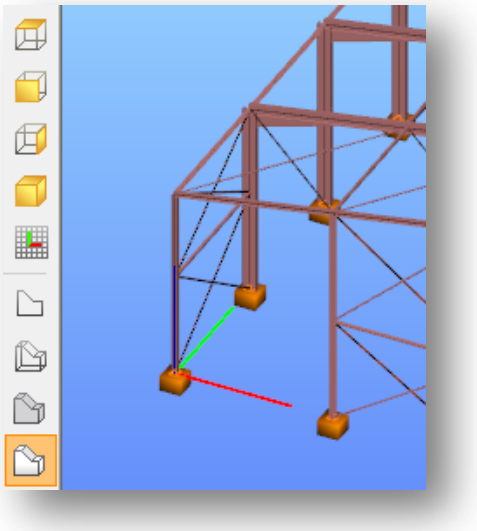
3 MODEL VIEW

3.1 BASICS

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

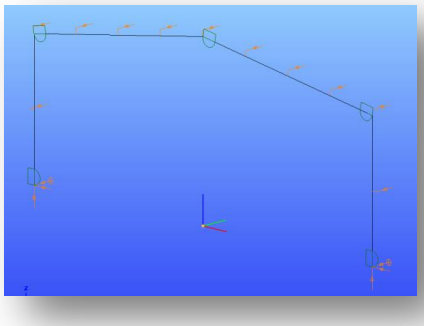
3.2 MODEL VIEWS

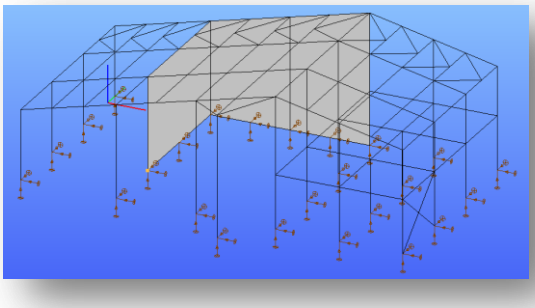
The model viewing options can be found on the left side bar.



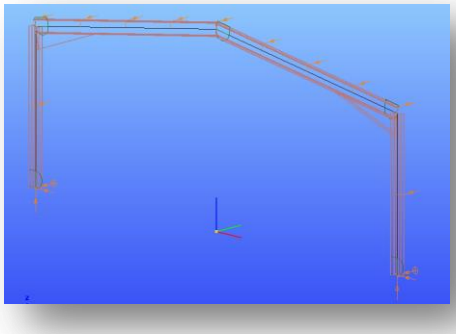
Beyond the usual model views (top view, front view, side view, axonometric view, perpendicular to raster view) there are fore visualizing options for the objects used:

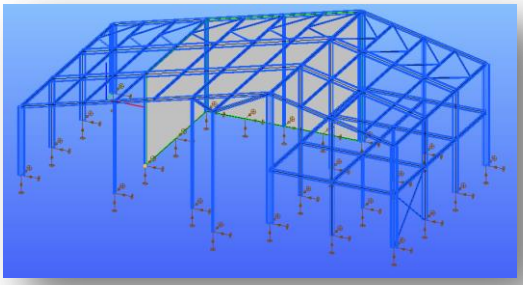
- **line view** : the simplest model view, the bars are represented by a single line, the surfaces by a two-dimensional figure without thickness, the supports are line types. *Recommended to use in the model building phases, since the clear visualization of the member snap points making the placement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



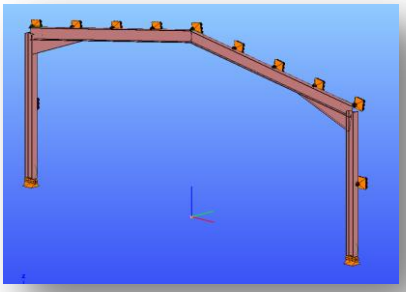


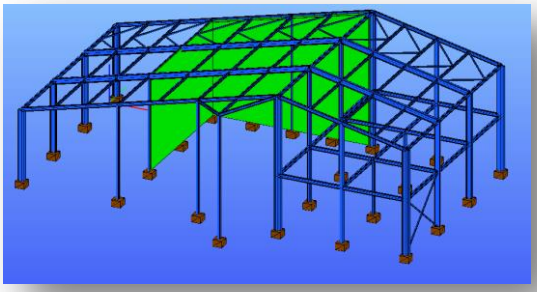
- **wireframe view** : on the wireframe view the lines of the cross sections and plate thickness appear, the supports are line types.



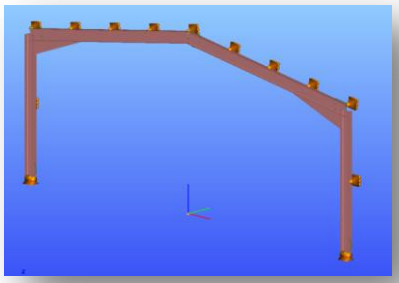


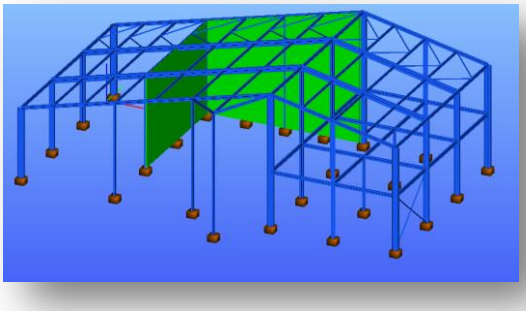
- **hidden line view** : the cross section and plate thickness appear in a solid form without shading and sparkling effects, the supports are solid types.



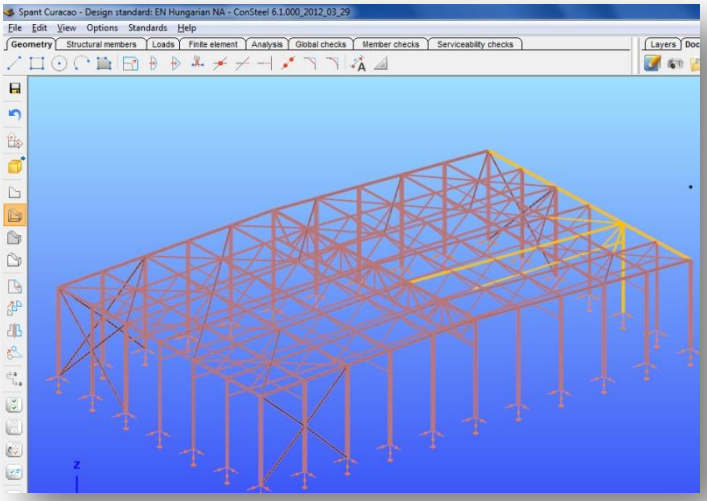


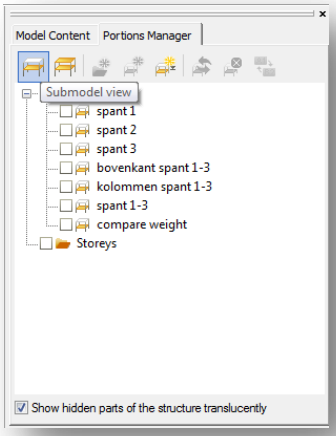
- **solid view** : the cross section and plate thickness appear in a solid form with shading and sparkling effects, the supports are solid types.



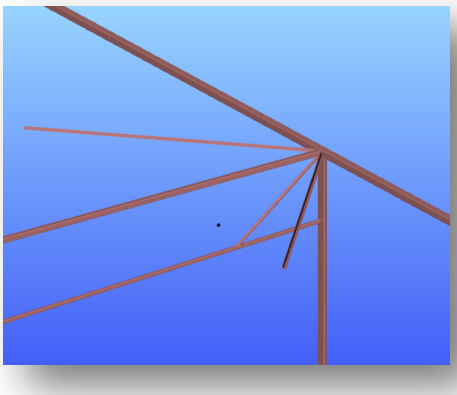


When the full model tends to be more complicated and parts of the model are in focus then the ***SUBMODEL VIEW*** should be applied.

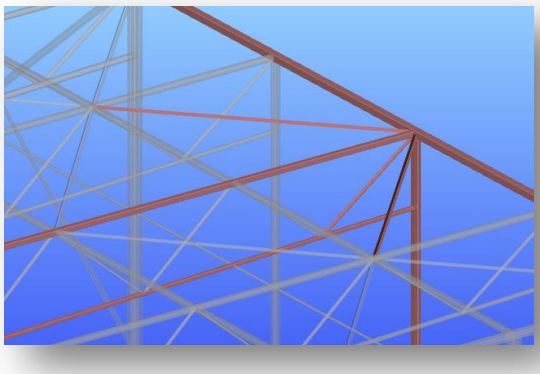




This option shows only the selected parts of the model,

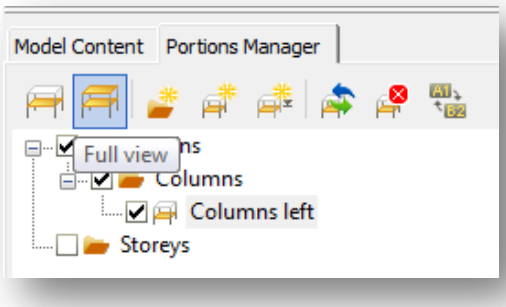


or checked the *Show hidden parts of the structure translucently* checkbox at the bottom of the ***PORTIONS MANAGER***.



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

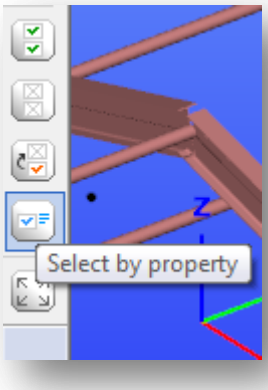
Objects can be selected on the graphical window, in the object tree, or by the additional selection options. As a general rule only the visible objects can be selected graphically, accordingly the same action can result in different selection on different tabs or model views. A selected object change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

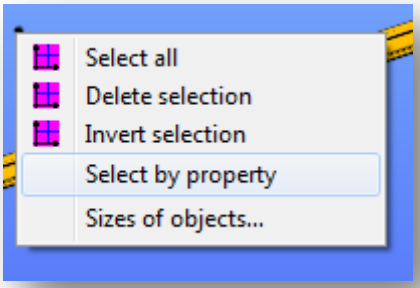
The basic selection modes and their operation are the followings:

- **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

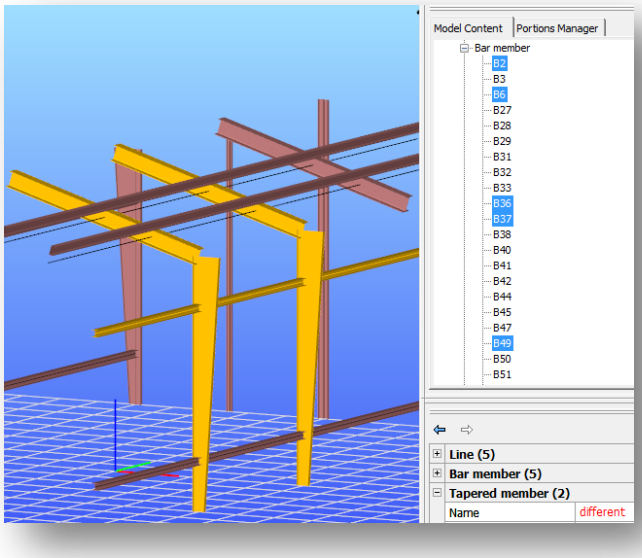
the selection depends on the size of the graphical symbol of the objects and note that the covered objects will be selected as well!

- **window selection**: holding down the left button and moving the mouse will create the window. It is completed when the mouse button is released. The actual size of the window is continually seen on the screen. Drawing the window from left to right, then only those members are selected that are within the window with their full size (inclusion window selection). Drawing the windows from right to left then every member is selected no matter with 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.





- **object tree selection**: selecting objects by clicking on the elements of the Object Tree.



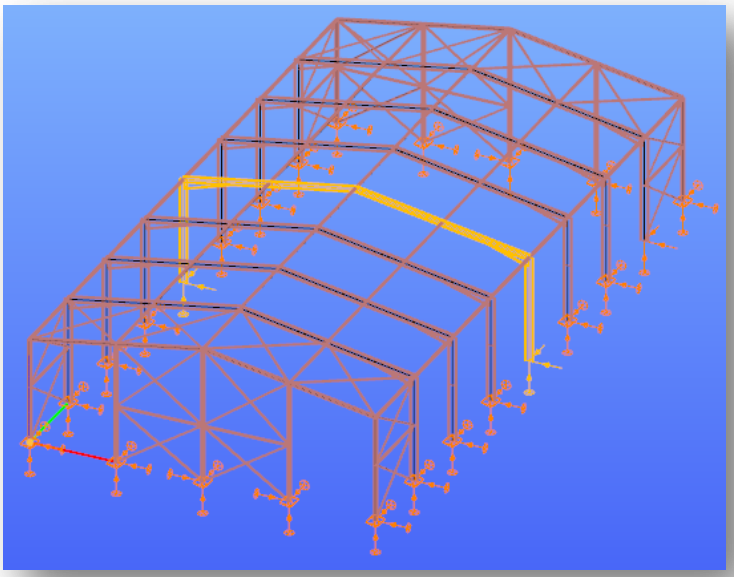
By clicking the object names in the object tree while holding down the ***CTRL*** or ***SHIFT*** key results multiple objects selection. To reselect any of the selected object: hold down the ***CTRL*** key and click on the object name again in the object tree.

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.

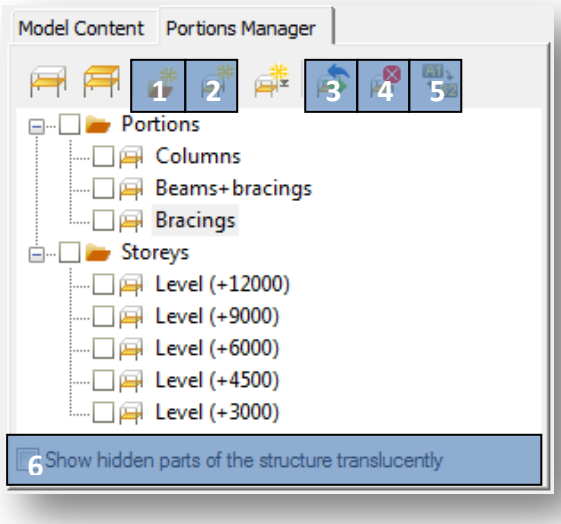
3.4 PORTIONS MANAGER

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 the portion.



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

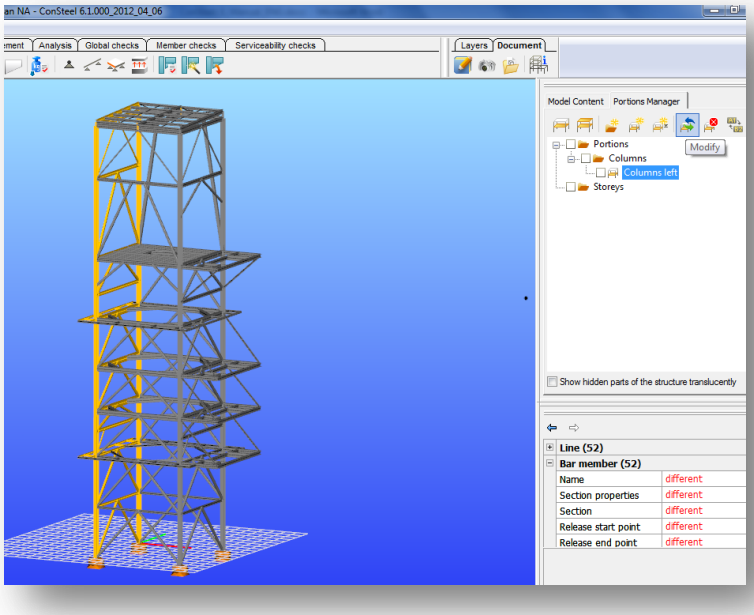


To turn on portion, check in the check-box in front of name of the portion. More portions can be turned on at the same time.

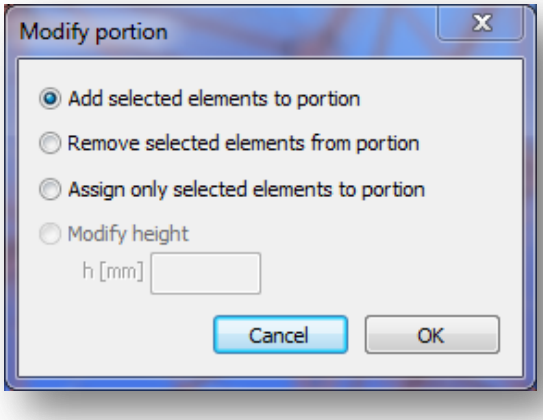
If the "*Show hidden parts of the structure translucently*" check box (#6) is clicked at bottom of the ***PORTIONS MANAGER*** tab, then the hidden parts of the model are also shown but those parts a translucent.

Any objects of the structural model can be part of more 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 part-model you are going to modify and finally click on the ***MODIFY*** icon (#3).



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



- **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 member already belongs 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 by clicking the name of the portion and click on ***DELETE*** icon (#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 give a name for the objects.

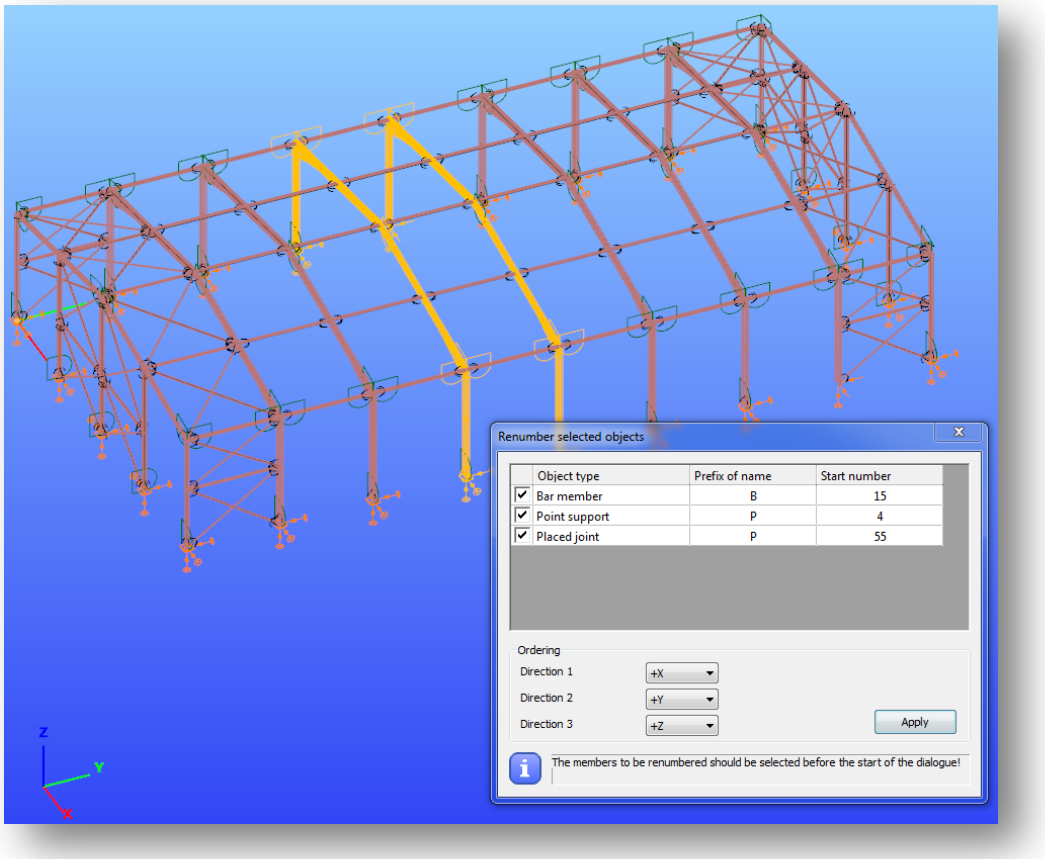
The following name and renumbering options are available:

- **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, copies structural objects (beam, column, plate, wall) consisting of a prefix and a 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 could 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

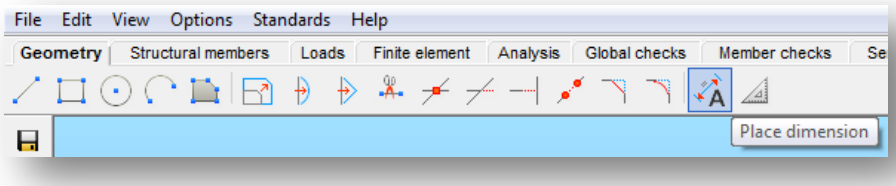
The visibility of the labels can be set on the bottom status bar as it was discussed above (Chapter 1.2.6). The following label options are available:

- **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.

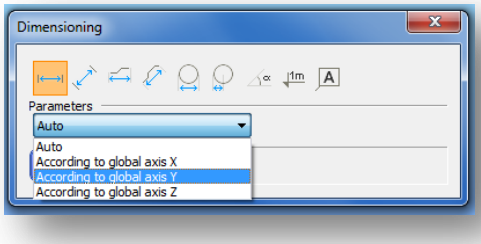
3.6 DIMENSIONS

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.



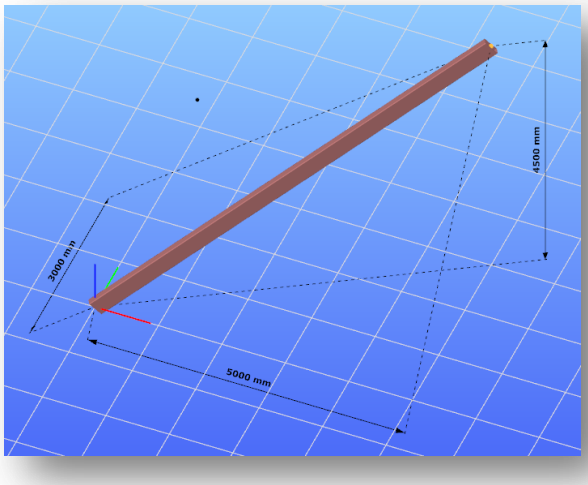
This provides a great number of different dimensioning options:

- **Projected dimension of a linear object** : The projected length of the selected linear object onto the global X or Y or Z axis will be dimensioned.

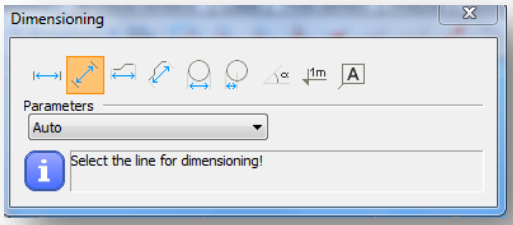


Parameters:

- 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.

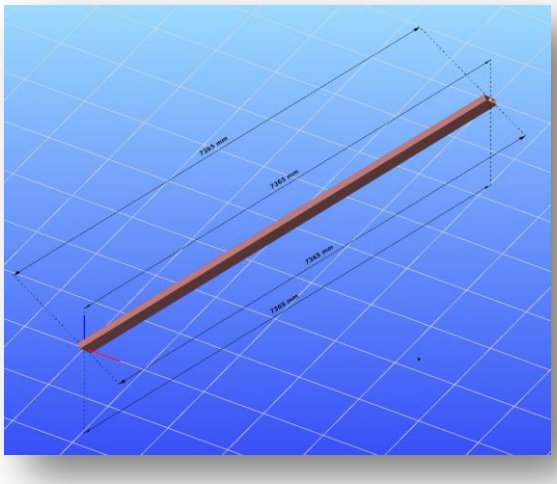


- **Lengthwise dimension of a linear object** : The length of the selected linear object could be placed in one of the plane which is defined by centerline of the object and the global coordinate system X,Y plane, and a plane perpendicular to this plane and containing the centerline of the object.

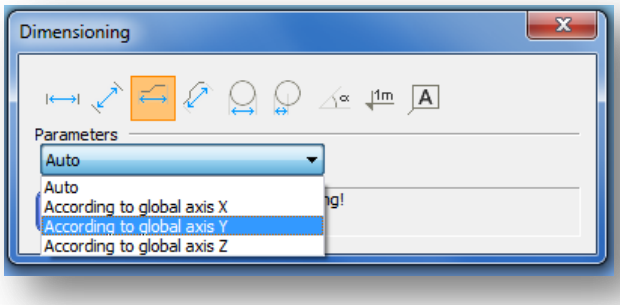


Parameters:

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



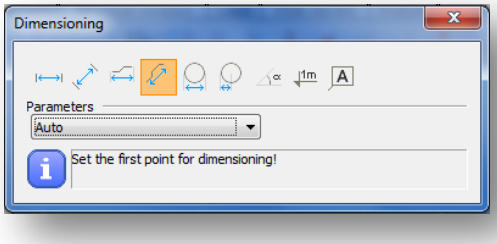
- **Projected dimension of the distance between two selected points** : The length of the distance between two selected points will be projected onto the global X or Y or Z axis.



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 plane.
  - 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 is detects the desired reference axis.

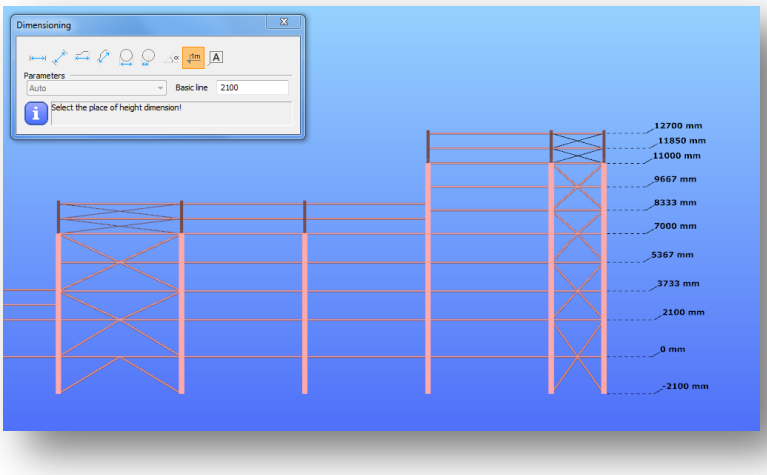
- **Diameter of a circle**

- **Radius of a circle** 

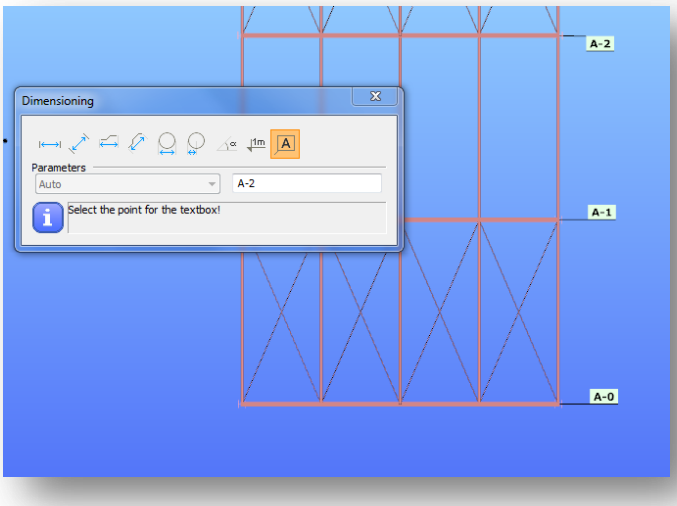
- **Angle of two lines** **:** After selecting two lines any of the four angle could be dimensioned.

- **Height dimension** **:** This function helps for quick dimensioning the high positions of the selected points of a construction 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.

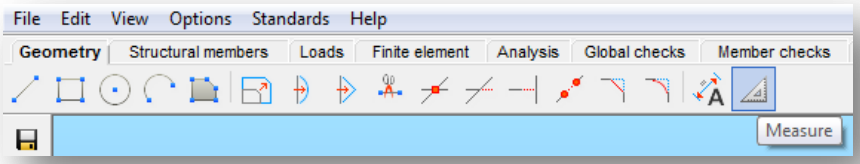


- **Textbox to a given line** **:** The given text (A-2) in the script field can be placed into the model.

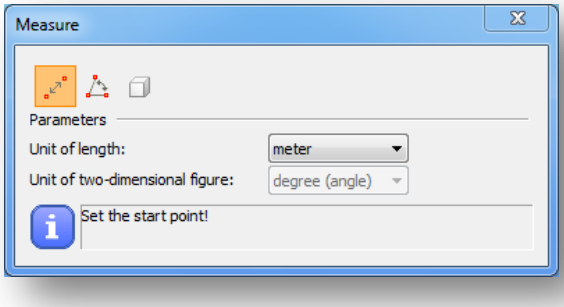


3.7 MEASURE

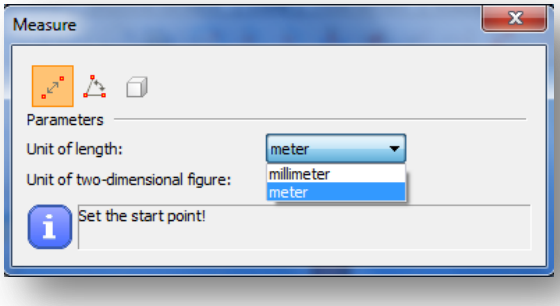
During modeling, sometime necessary to measure distances, and angles. The ***MEASURE*** tool can be found at the end of the ***GEOMETRY*** tab.

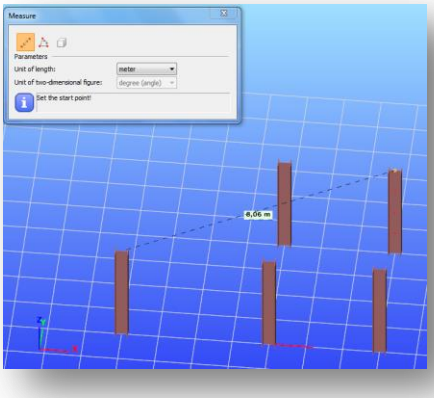


The following window offers the measuring functions:

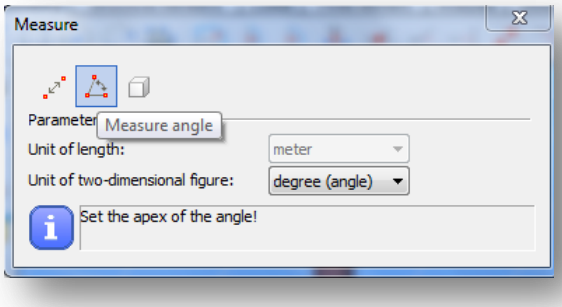


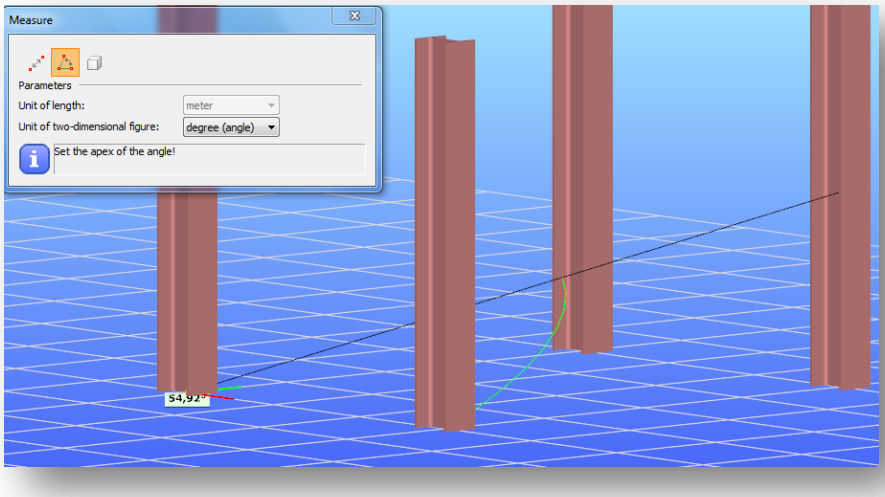
- **Measure distance** : Shows on the screen the measured distance between two picked points in meter or in millimeter, selected from the "unit of length" list box.



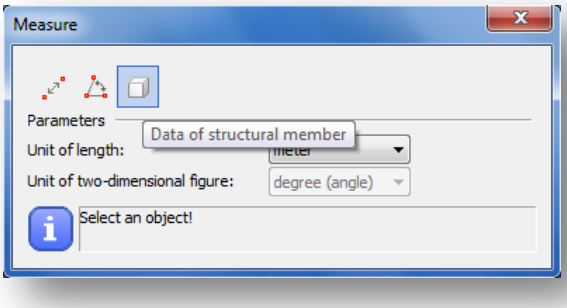


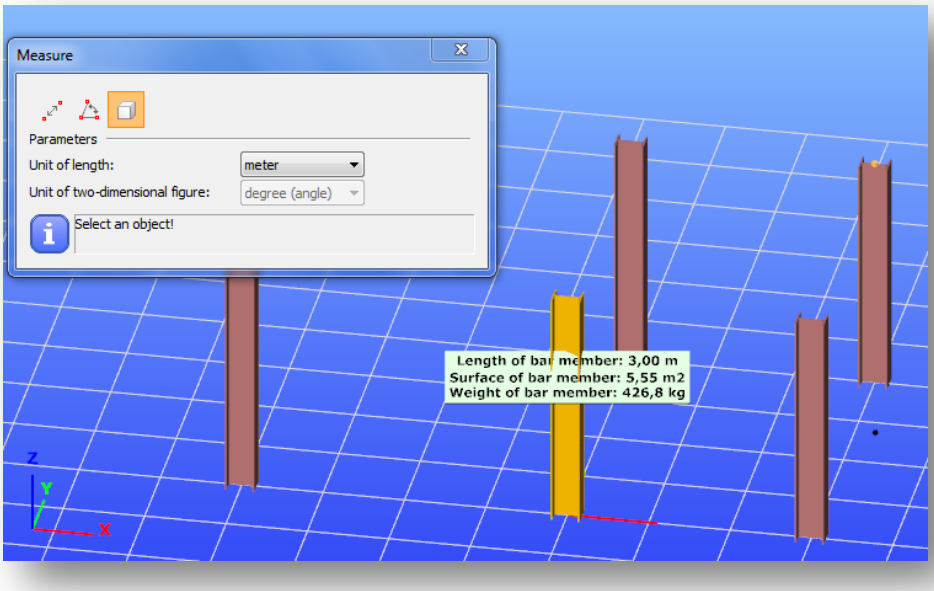
- **Measure angle** : Function measure the angle between two lines was defined by clicking three snap points in the 3D space in the plane of the two lines in degree or radian in.





- **Data of structural member** : Function shows on the screen the length, the surface and the weight of the selected structural member.




Measure
Parameters
Unit of length:
meter
Unit of two-dimensional figure:
degree (angle)
Select an object!
Length of bar member: 3,00 m
Surface of bar member: 5,55 m2
Weight of bar member: 426,8 kg
Z
Y
X

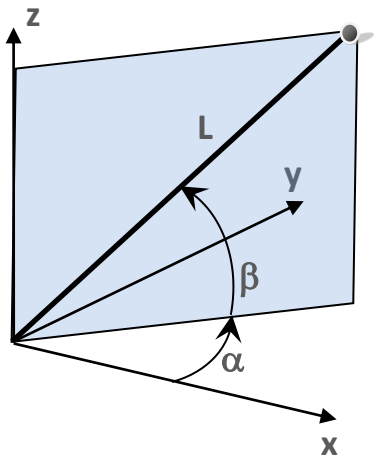
4 DRAWING GEOMETRY

4.1 BASICS

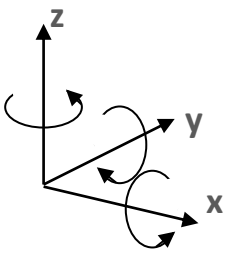
The first step of the modelling phase is the geometry drawing of the structure. In *ConSteel* all the geometrical objects can be easily created and modified in a 3D space. Among the basic CAD drawing, modifying and moving functionality advanced snap options make the modelling efficient. All the CAD functionality 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 (αβL). 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.

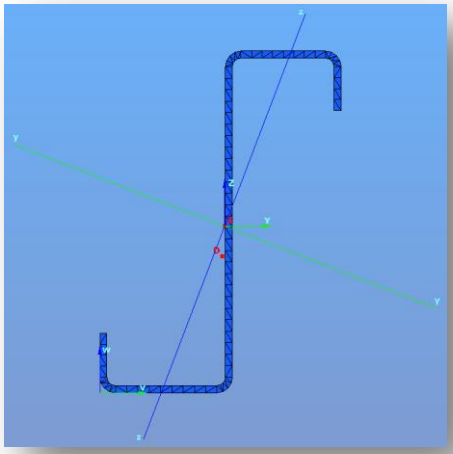


As a general convention all the Descartes coordinate systems (axes directions and rotation signs) follow the most commonly applied right hand rule. This rule is valid for the definition of geometry and loads as well. The positive moments and rotations are defined as counter clockwise about the axis if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

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



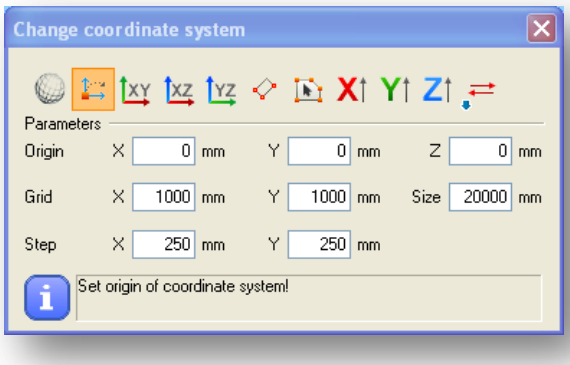
In case of cross sections (2D modelling of sections):

- 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

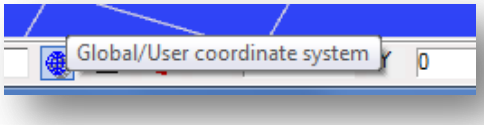
4.2.1 User coordinate system

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.



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

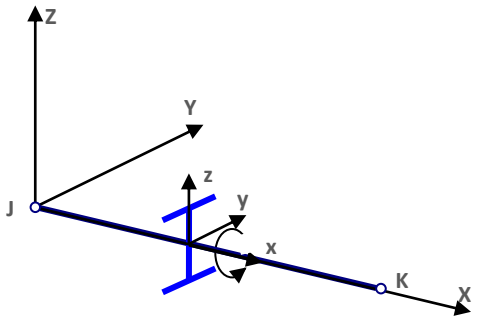


The following creation options are available (from left to right):

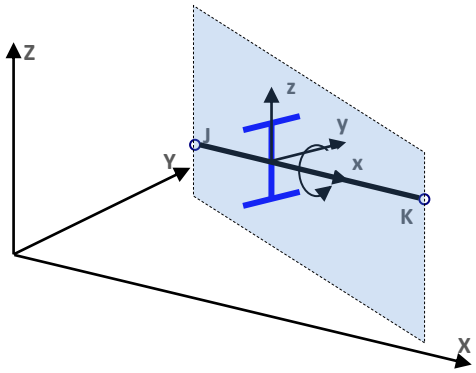
- Switch to global system
- Set new origin for the UCS
- The “XY” plane of the global system will be the “XY” plane of the UCS
- The “XZ” plane of the global system will be the “XY” plane of the UCS
- The “YZ” plane of the global system will be the “XY” plane of the UCS
- Set the UCS by 3 points: the 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
- Set the direction of the axis “Y” by two points
- Set the direction of the axis “Z” by two points
- Reverse the direction of the selected axis

4.2.2 LOCAL COORDINATE SYSTEM OF BAR ELEMENTS

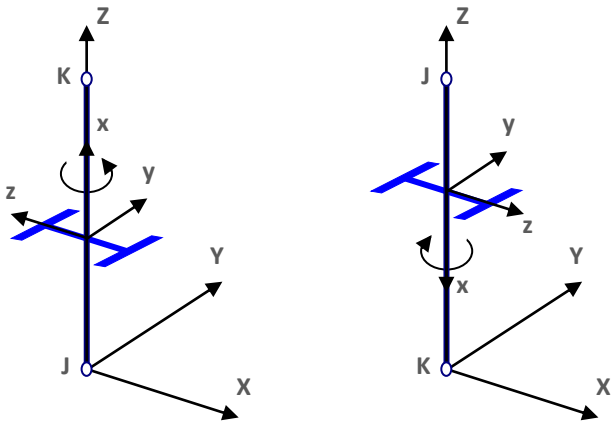
The reference line of bar members defines the axis “x” of the local coordinate system of the bars. The direction of the axis “x” is given by the start (**J**) and end (**K**) node of the reference line.



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.

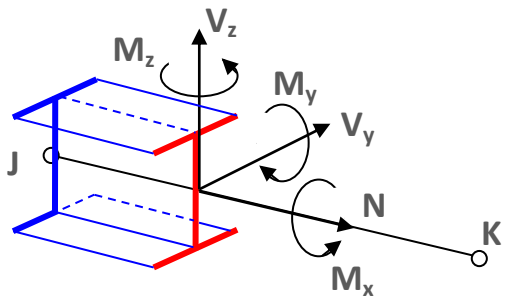


In case of a generally positioned beam member (not column with vertical reference line) the plane defined by the local "xz" system takes always vertical position and the axis "z" points towards the positive "Z" direction. For beams with (single) curvature the axis "x" is always the tangent.



In case of vertical reference line (column members) the local axis "y" 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

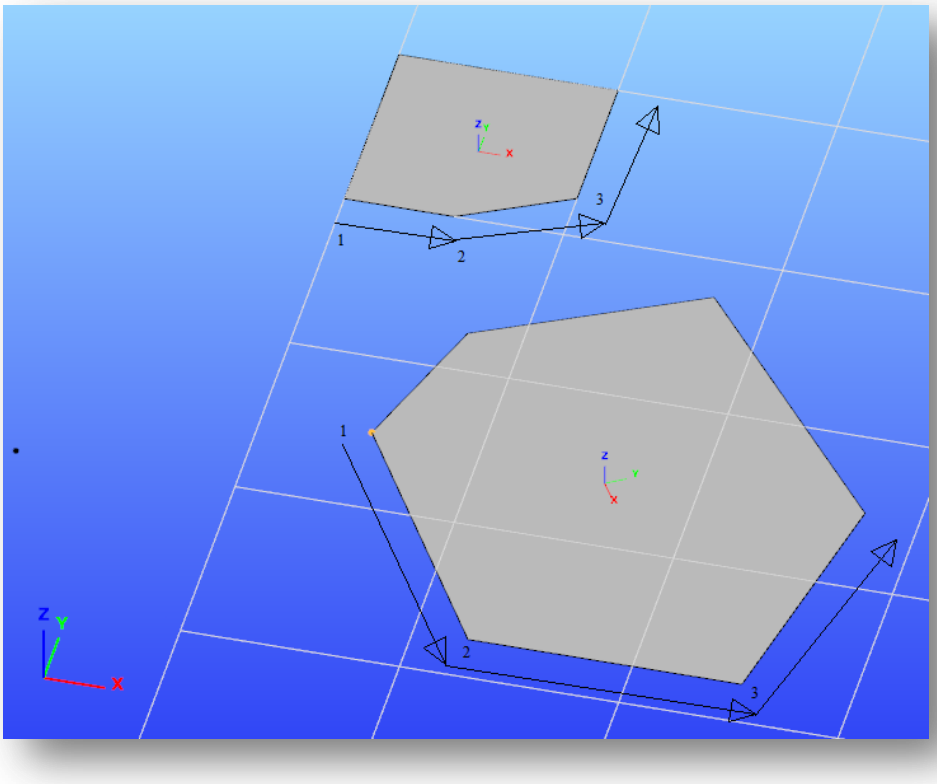
For surface elements the local plane "xy" is defined in the plane of the surface, the axis "z" is perpendicular to that plane and the direction of "+z" is in accordance with the right hand role of coordinate systems. The roles of the directions of the local axes of the plane surface elements:

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  icon on the right side of the bottom status bar. You can find the description about the snap point settings above in **Status bar** section (Chapter 1.2.6).

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.

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

 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:

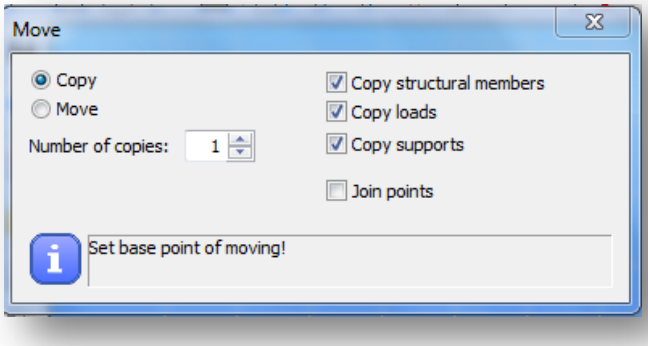


Move point and edge. Select the point and (or) edge to be moved. Set the new position of the point (edge). This function is used for change the position of any of the end point of a linear or circular element, or to change the position of an edge or a corner point of a polygon, or of hole in a polygon.



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 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.

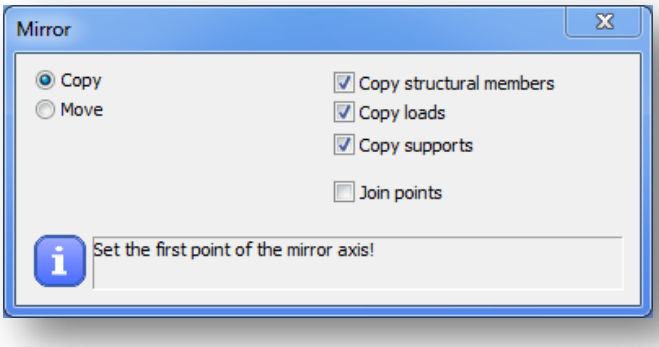
Checked: The supports of the selected structural members will be copied. (In case of the selection set does not contain at least one selected member, than the selected supports will not be copied.)

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.

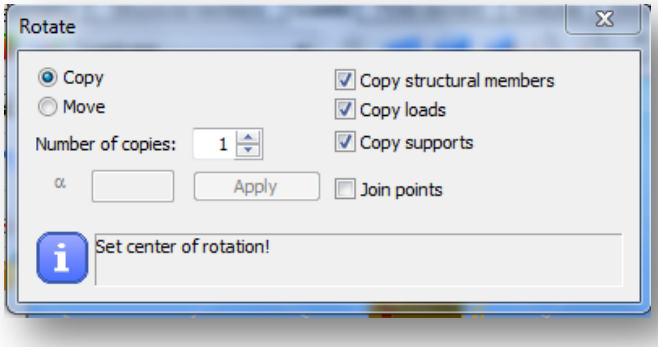
As first step select the *UCS* for mirroring (if it is needed), select objects to be mirrored and click on the ***MIRROR SELECTED OBJECTS*** icon.



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.



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 Effect 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 to 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.

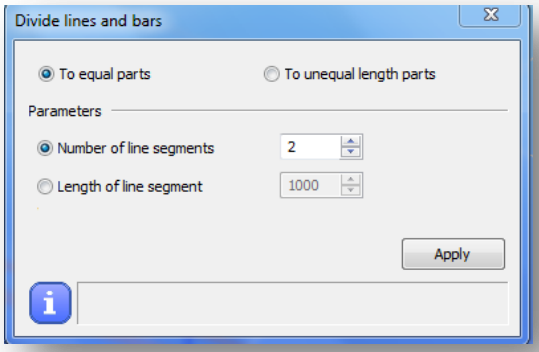
Curve selected line element. Select the edge for bend. Set the insert point.

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.

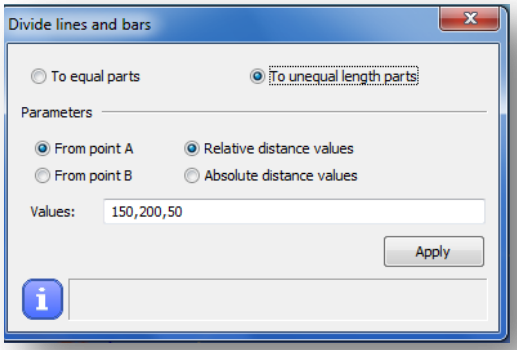
The coordinates of the new position of the selected point also can be given manually accordingly to the selected coordinate system.

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.



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.



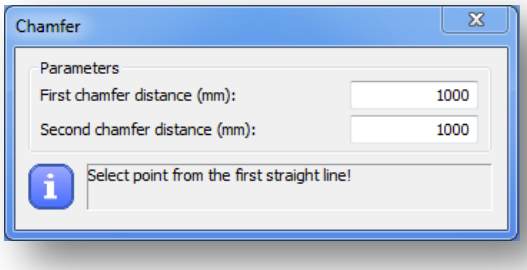
Break two selected objects. Select the two intersected objects to break. Both object will be broken by the intersection point. It works with linear and curved objects.

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

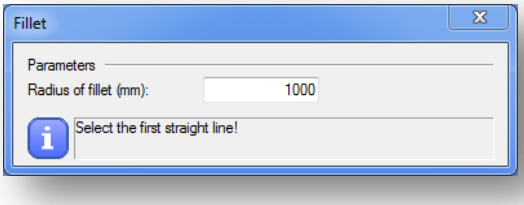
Extend selected line elements to limit line. Selecting the limit line and click on the linear object to extended to the limit line.

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.

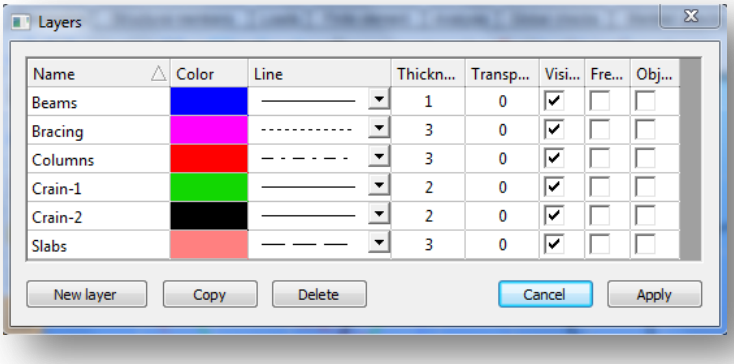


Filleting of the edge of the section of two line element. Select the first and the second linear element. Chose a fillet radius.



4.6 LAYERS

The layers dialog window can be used for:



- 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.

5 STRUCTURAL MODELLING

5.1 BASICS

From *ConSteel 5.0* great emphasis was taken on the user friendly structural modelling in which the engineer can build the structural model using real structural elements. Accordingly the *user model*, built by the engineer 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 direct productive labour 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 9) and ***CSJOINT*** (Chapter 12).

5.2 LINE MEMBERS

5.2.1 CREATING LINE MEMBERS

From the *ConSteel 6.0* version the line members with steel thin-walled, reinforced concrete, and composite cross-section can be modelled. The members – depending on the placement – can be columns or beams. The column is a special position 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

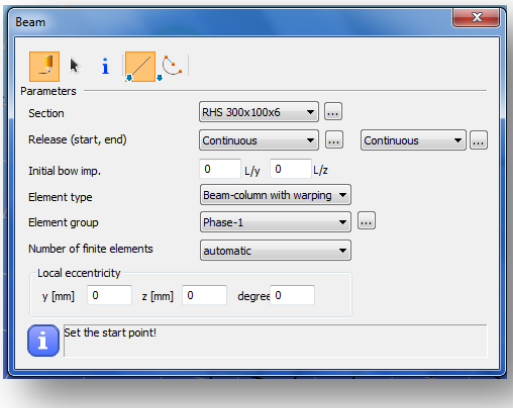
 draw as a line segment or continuous line

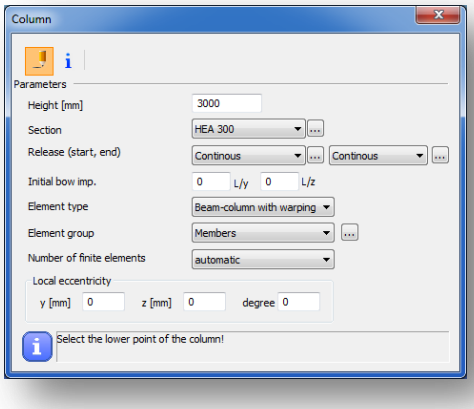
 draw as an arc by centerpoint, startpoint and angle, 3 points or startpoint, endpoint and tangent

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

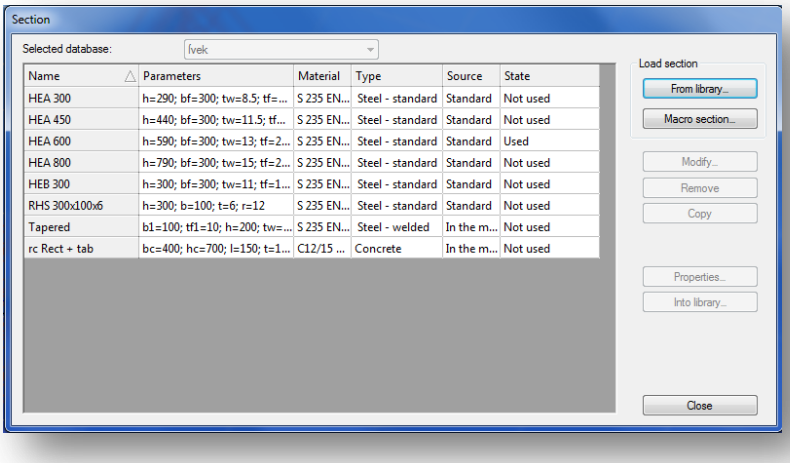
 Read out the parameter of an existing member by clicking on it and assigning those parameters to the one to be created

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



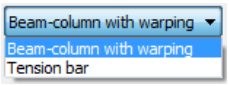


- *Section:* before creating a beam or column member cross-sections should be loaded, and the appropriate section can be selected from the combo including the loaded sections. If no section has been loaded in advance the ***SECTION*** dialogue can be called by pressing the  button next to the combo. For more information about section administration see Chapter 9.2.



- *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.
- *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*:

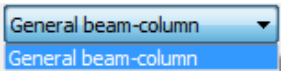
  Section made from steel:

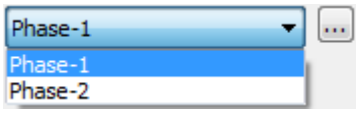
  Two choices are possible *Beam-column with warping* and *Tension bar*, these types influence the finite element type used in the analysis. The *beam-column with warping* is a special element with 14 degree of freedom, including the warping of the thin-walled cross section. It is an important effect in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

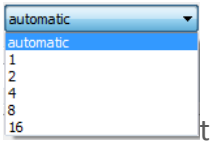
  Reinforce concrete or Composite sections:

  There is only one choice, 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.).

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

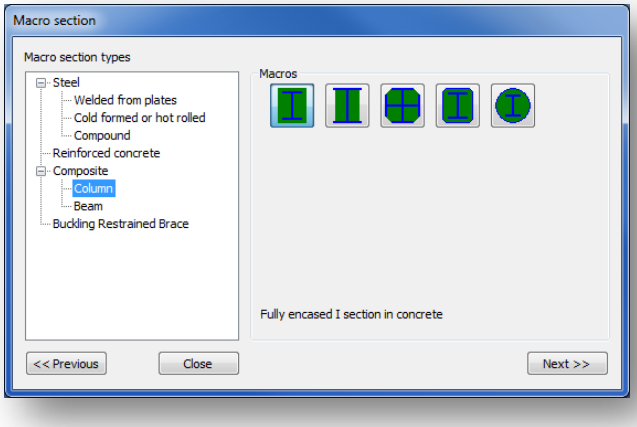
  

- *Number of finite elements*: the required number of finite elements used on the member in the analysis can be set. The default *automatic* option gives a sufficient result in the most cases.
- *Local eccentricity*: member eccentricity can be defined in the local coordinate system ("y,z") of the member, 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.

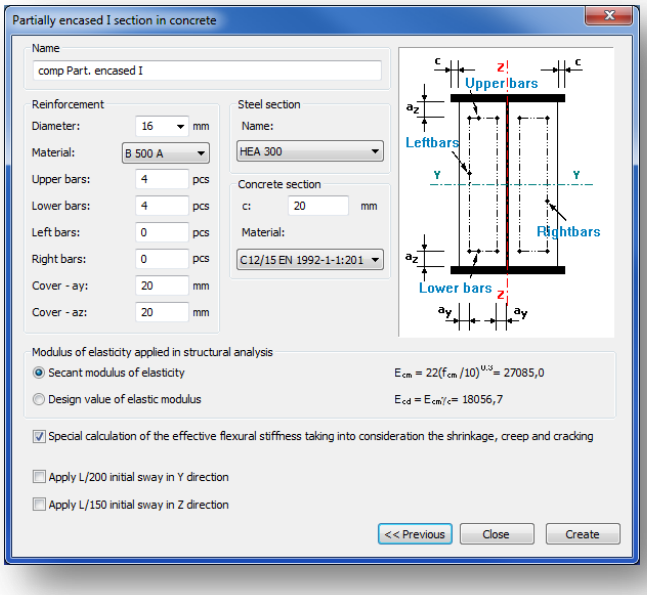
IT SHOULD BE NOTED THAT NOT ONLY THE ECCENTRIC AXIAL FORCE PRODUCES INFLUENCE IN THE ANALYSIS (ADDITIONAL BENDING MOMENTS) BUT THE ECCENTRIC BENDING AND TORSIONAL MOMENTS (ADDITIONAL BENDING AND TORSIONAL MOMENTS AND **BIMOMENT**)!

5.2.1.1 COMPOSITE COLUMN MODELLING

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***.



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

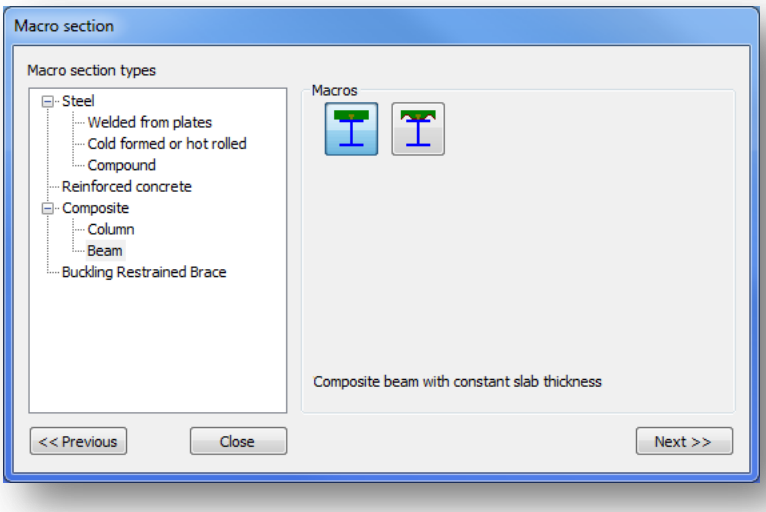


For the structural analysis two types of elastic modulus can be used which can be selected on the middle part of the cross-section creation dialog.

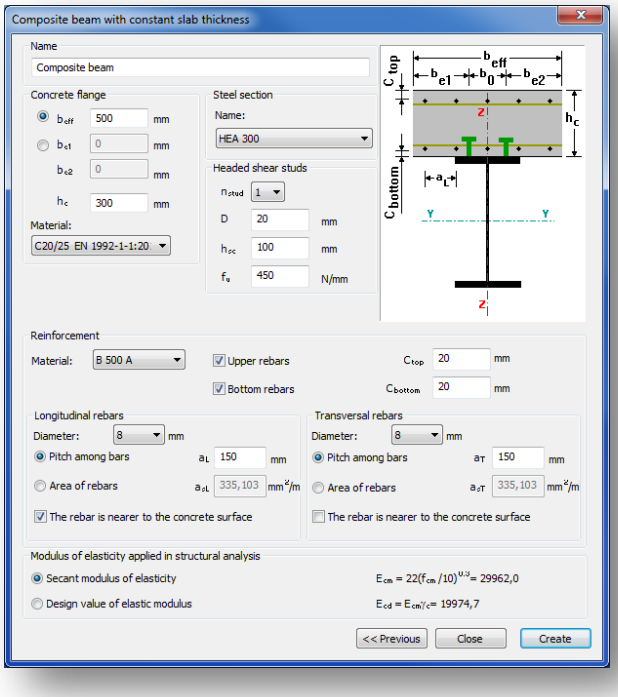
Standard initial sways also can be defined in y and z directions for the cross-section. If the initial sways are applied than the cross-section check is enough according to the EuroCode. Stability check is not necessary.

5.2.1.2 COMPOSITE BEAM MODELLING

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***.

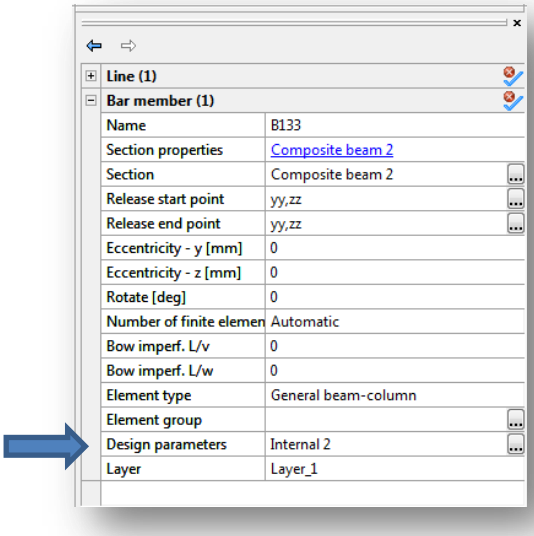


When creating a new composite cross-section the effective breadth must be defined. It is important to know that during the analysis the actual effective breadth is automatically calculated based on the design parameters. The effective breadth which must be set at the cross-section creation is used in 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).



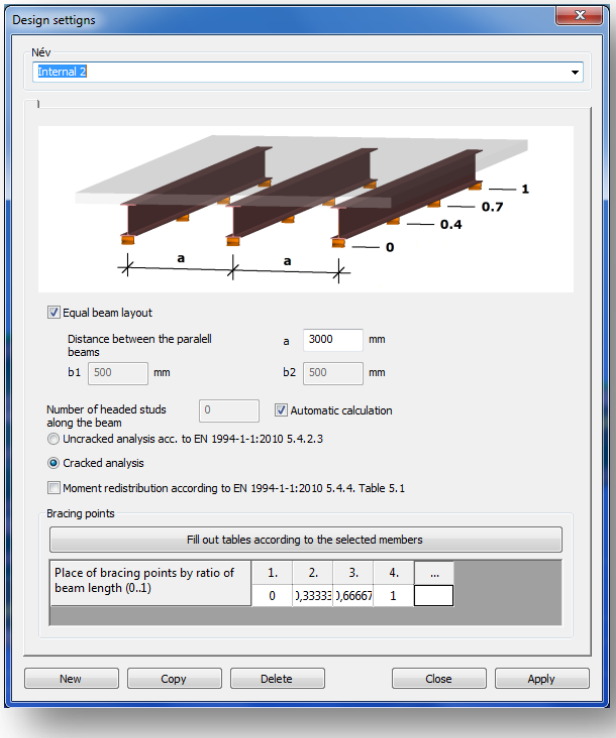
After the composite cross-section is defined it can be used for member creation the same way as a steel cross-section.

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



In the design parameters 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

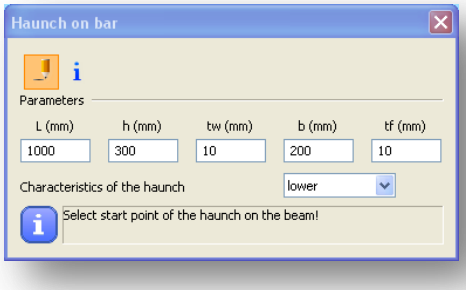
clicking on the button above the table. You can create more design parameters and assign them to different members. You can also use more design parameters to easily switch between them to find the best parameters or to simply compare different settings.



5.2.2 HAUNCHED MEMBERS

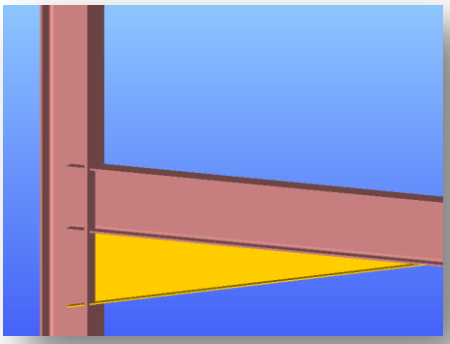
The created members can be strengthened, if necessary, by using ***HAUNCH*** function ().

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 I section (with one flange) with decreasing web height. On the dialogue panel the length (L), height (h), web thickness (tw), flange width (b), flange thickness (tf) and the characteristics of the haunch should be set.

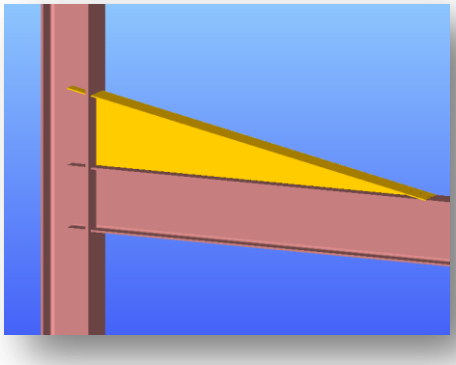


The position of the haunch is set by simply clicking on the member the start point (where the height of the haunch is full) and the direction point of the haunch on the member. It should be noted that if the start point is a common end point of more members (this is the usual case, for instance at a 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:

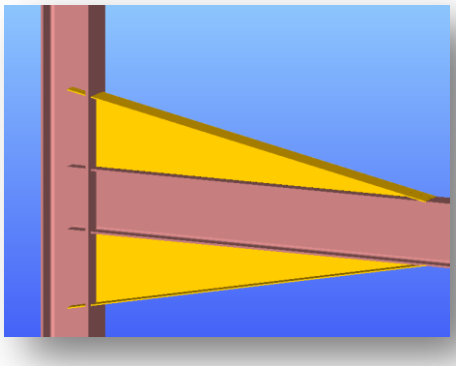
- lower



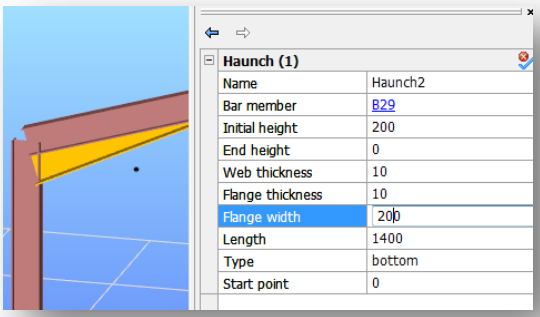
- upper



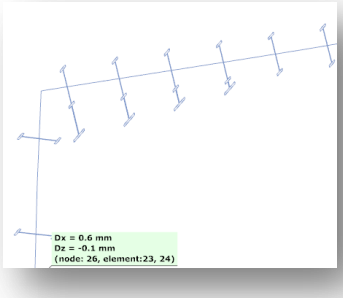
- symmetric



THE HAUNCHES CAN BE SELECTED AND MODIFIED IN THE PARAMETER TABLE AS SEPARATE OBJECTS.



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).



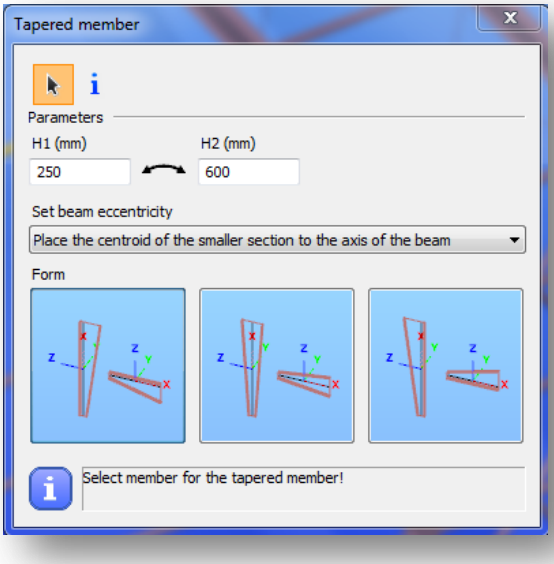
THIS ECCENTRICITY CAUSES ADDITIONAL EFFECTS IN THE ANALYSIS RESULTS DUE TO THE ECCENTRIC POSITION OF THE SECTIONAL FORCES (FOR INSTANCE AT THE BEAM-TO-COLUMN CONNECTION POINT OF A FRAME WITH HAUNCHED BEAMS AND/OR COLUMNS THE EQUILIBRIUM OF THE IN-PLANE BENDING MOMENTS EXISTS ONLY IF THE ADDITIONAL MOMENTS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

5.2.3 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 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.

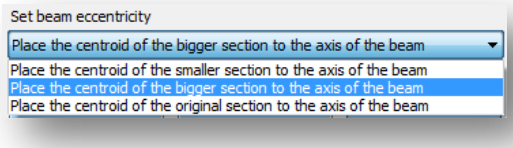
With the ***TAPERED MEMBER*** function () the section height of these sections can be set to linearly varying along the member length.



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.



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

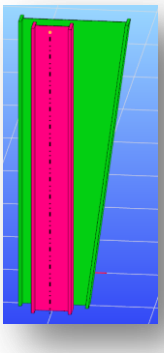
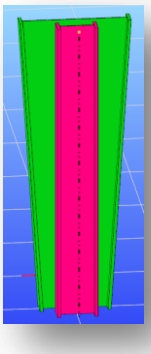
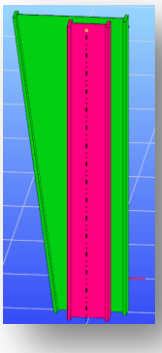
3. *Place the centroid of the original section to the axis of the beam:* the edge of the tapered member is coincident with the original member end and the tapering will start from this position.

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:

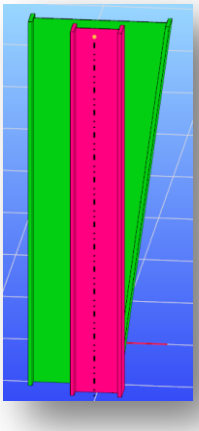
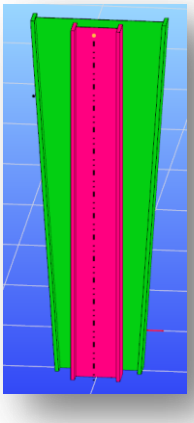
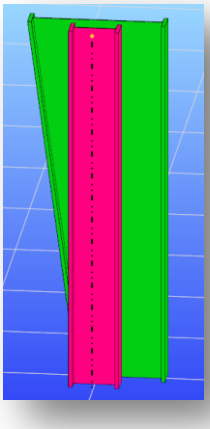
1. *Place the centroid of the smaller section to the axis of the beam:* the centre of smaller H value of the tapered member will be positioned to the axis of the original beam

-z ***symmetric*** ***+z***

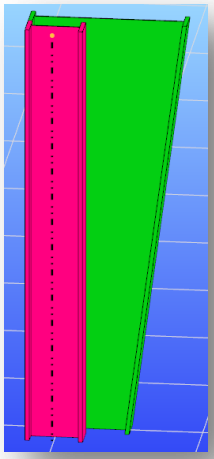
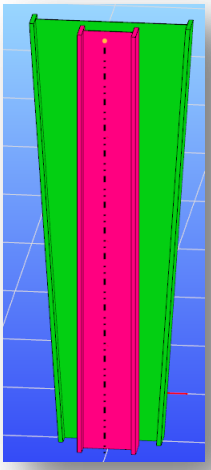
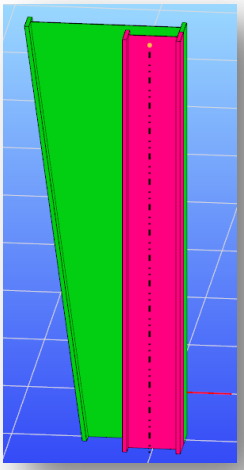
2. *Place the centroid of the bigger section to the axis of the beam:* the centre of bigger H value of the tapered member will be positioned to the axis of the original beam

-z ***symmetric*** ***+z***

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

-z ***symmetric*** ***+z***

Activating the  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.

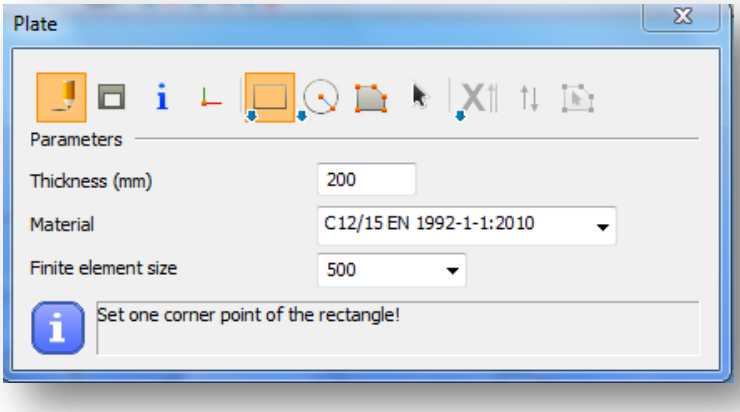


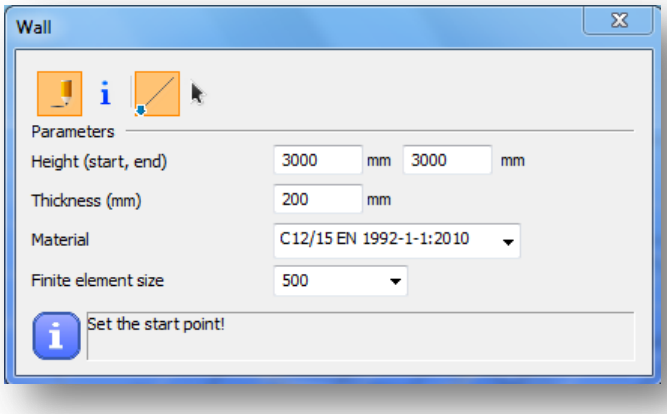
IN CASE OF SELECTING A TAPERED MEMBER NOT ONLY THE MEMBER BUT THE TAPERING WILL BE SELECTED AUTOMATICALLY AND CAN BE MODIFIED IN THE PARAMETER TABLE AS SEPARATE OBJECT.

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.



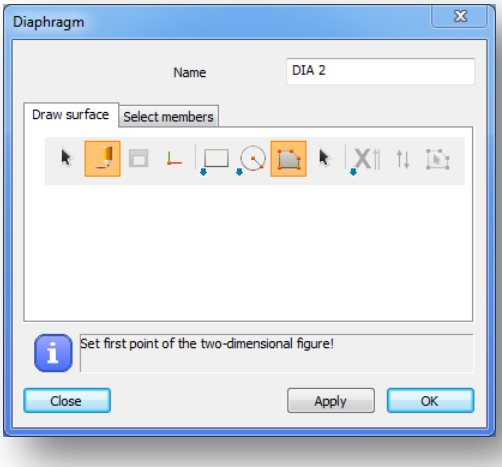


In the ***PLATE*** and ***WALL*** definition panels the drawing functions are collected in order to create the plate with the desired shape. Arbitrarily shaped holes can be defined in existing plate members. Three parameters can be set: the thickness, the material and the finite element size of the surface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

5.4 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  icon on the ***STRUCTURAL MEMBERS*** tab a dialog window appears.



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.

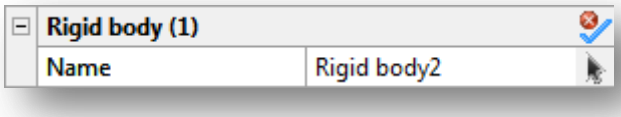
5.5 RIGID BODY

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

Before clicking on the  icon, 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.

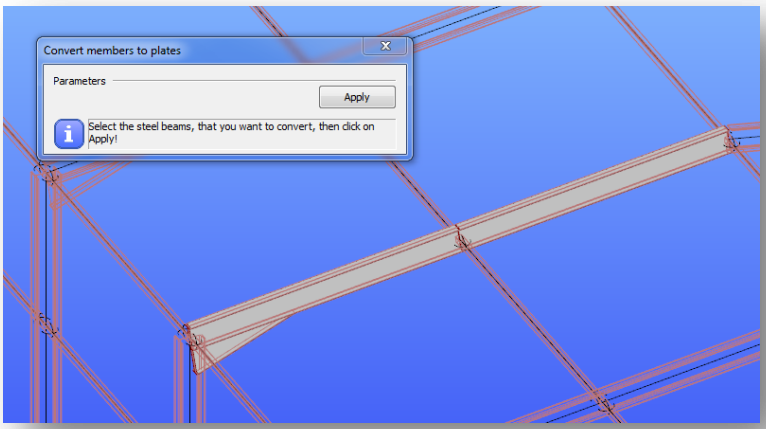
5.5.1 MODIFYING RIGID BODY

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.



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:

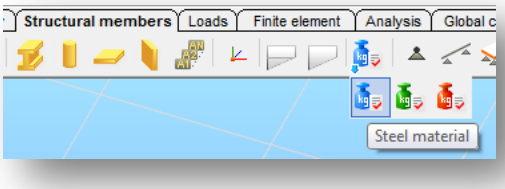
- Hot-rolled I, H and tube section
- Cold formed C, Z and tube section
- Welded I, H and box section

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.

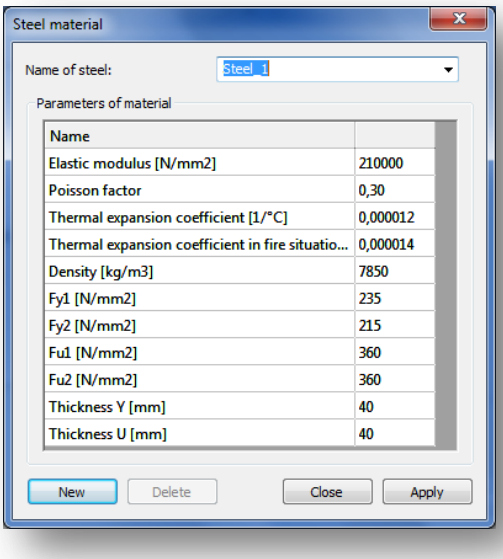
In case of hot-rolled sections not only the flanges and web are converted to plates but also the neck area with a suitable additional plate element. Therefore the section properties of the original member and the converted are the same.

5.7 MATERIALS

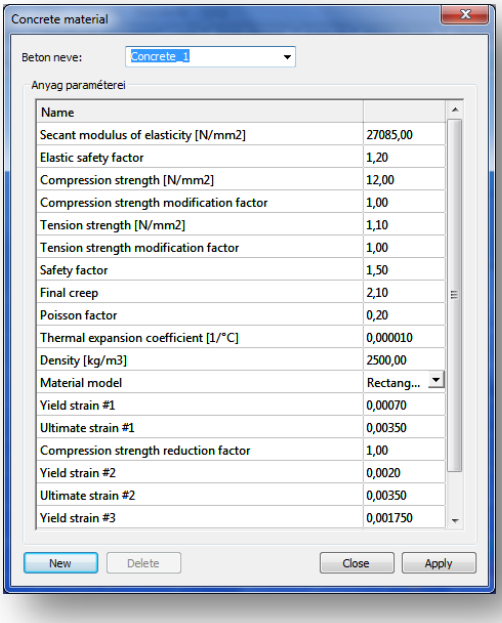
New material grades can be defined as one of the three different types of material: steel, concrete and concrete reinforcement.

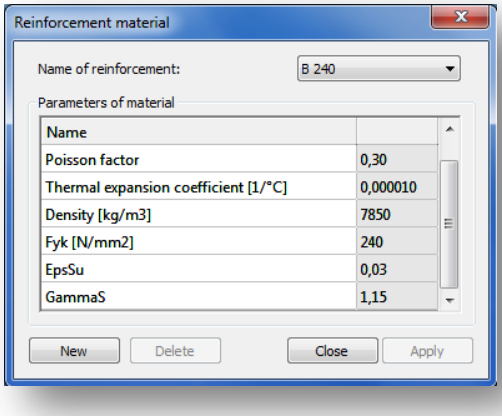


The latter is only used for the rebar reinforcement of concrete or composite cross sections. The material parameters are taken from the appropriate Structural Eurocode chapters (EN 1993-1-1 for steel and EN 1992-1-1 for concrete and reinforcement), and the default – indelible – material types are set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



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.



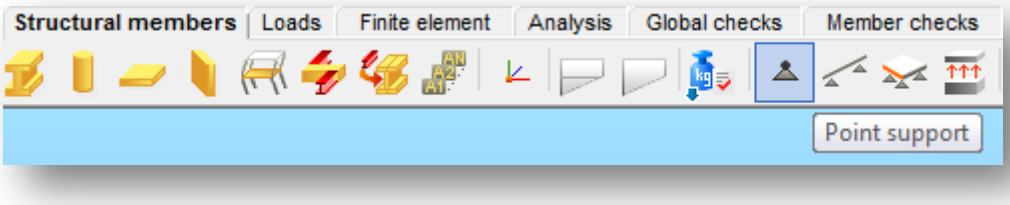




IMPORTANT TO NOTE THAT IN CASE OF BAR MEMBERS (BEAMS, COLUMNS) THE MATERIAL IS THE PARAMETER OF THE CROSS SECTION OF THE MEMBER, SO THE CURRENT MODIFICATION SHOULD BE APPLIED FOR THE APPROPRIATE CROSS SECTION. THIS FEATURE ALLOWS THE DEFINITION OF CROSS SECTIONS WITH MULTIPLE MATERIALS (E.G. COMPOSITE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

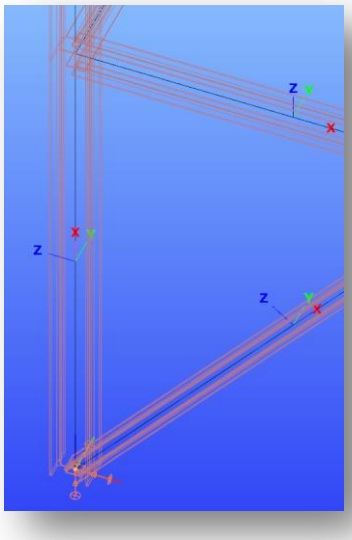
5.8 SUPPORTS

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



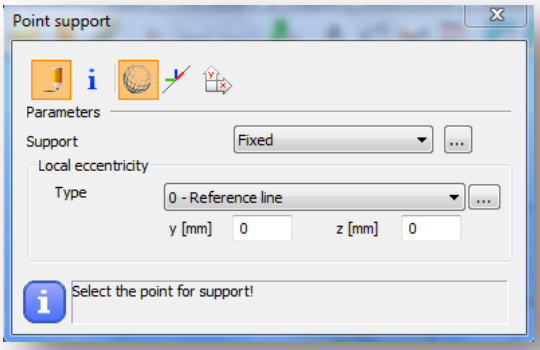
Supports can be placed according to the Point loads can be placed according to the **Global (**  **), Local (**  **)** or the **User coordinate system (**  **).**

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.



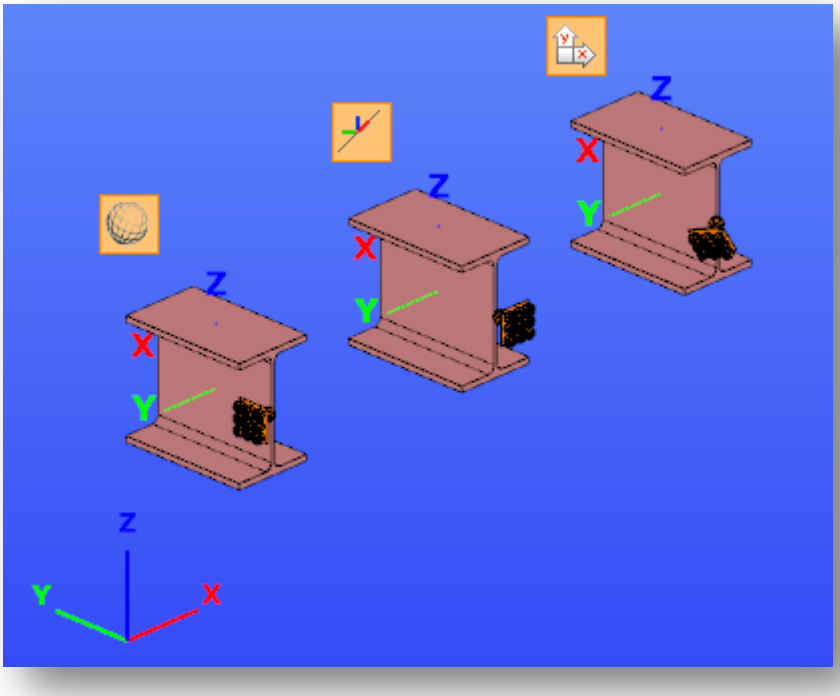
5.8.1 POINT SUPPORT ()

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).



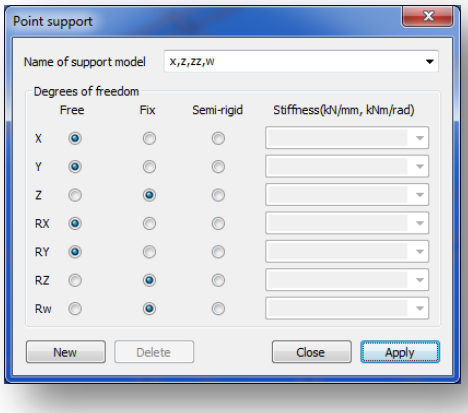
After selecting the coordinate system, the type of the support has to be selected from the list-box. There are several point supports types predefined.

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.

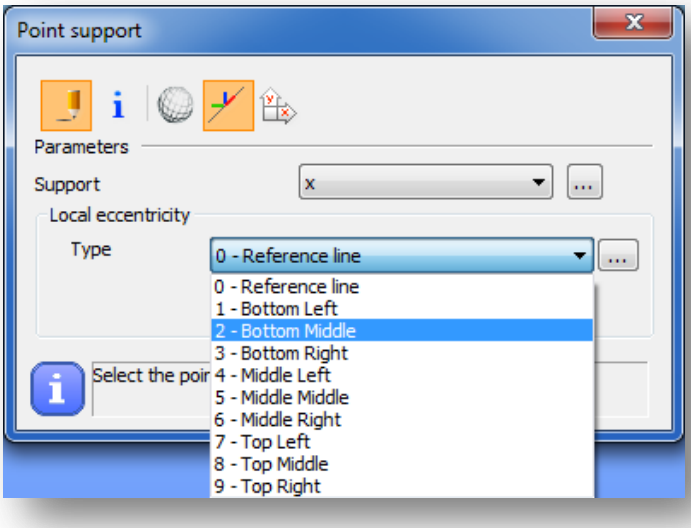
If any special supports are needed during the modelling process different from the predefined support types then click on the support definition icon (...). With the ***NEW*** button new type of point support can be defined. 7 DOFs (Degrees of Freedom) can be set to free, fix or semi-rigid. For semi-rigid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



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.

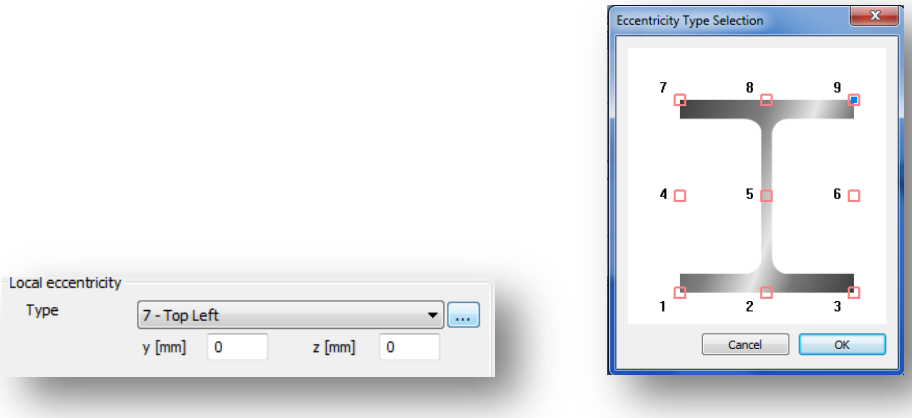
Local eccentricity can also be defined to the supports:

(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.)

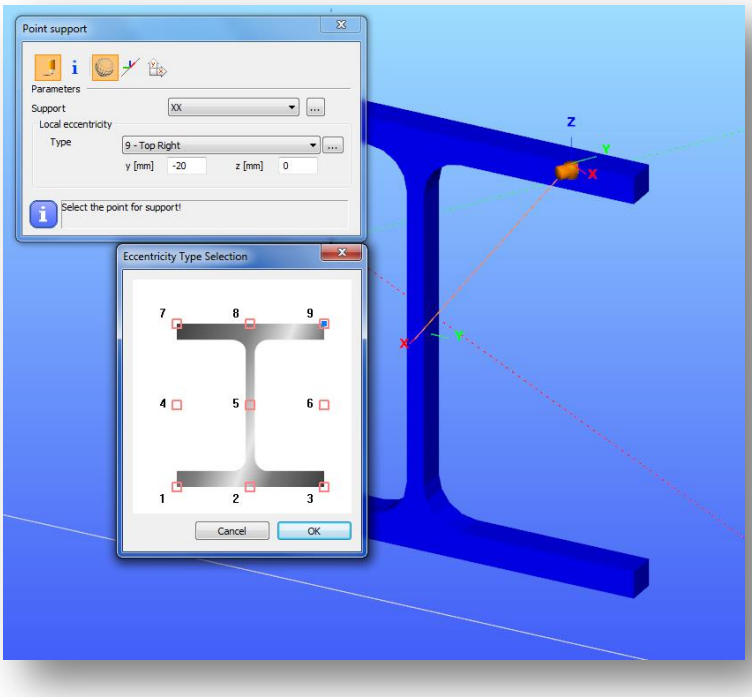


The eccentricity of the support can be defined **relative to the reference line** of a section, or **relative to the section geometry**. The "0 - Reference line" local eccentricity type can be selected only from this list-box.

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:

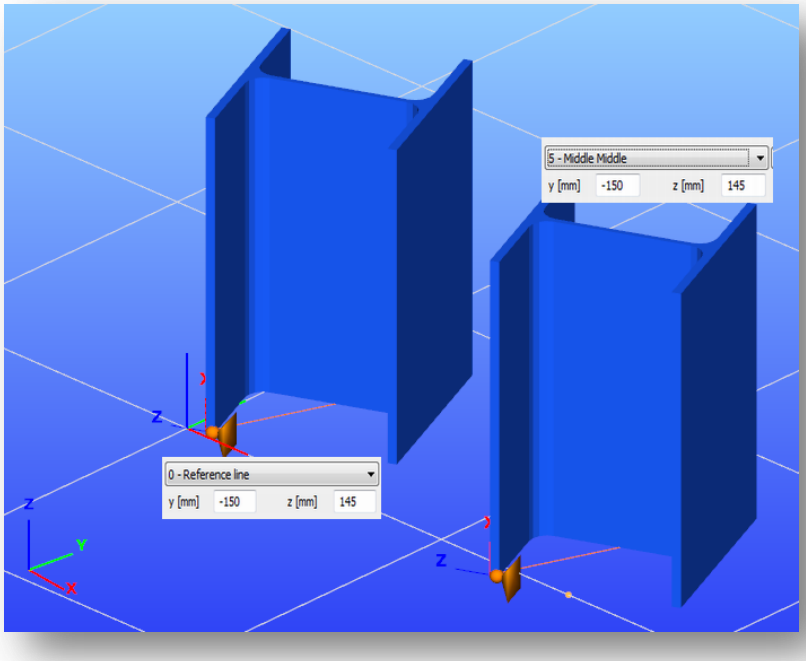
Placing a support with this eccentricity type, for example changing the “y” eccentricity of the supported object, the support will keep its position relative to the reference line, not to the section. By changing the “Rotation angle” attribute of the supported object, the reference line also rotates, and the position of the support will rotate too.

By selecting the 1 -9 eccentricity type:

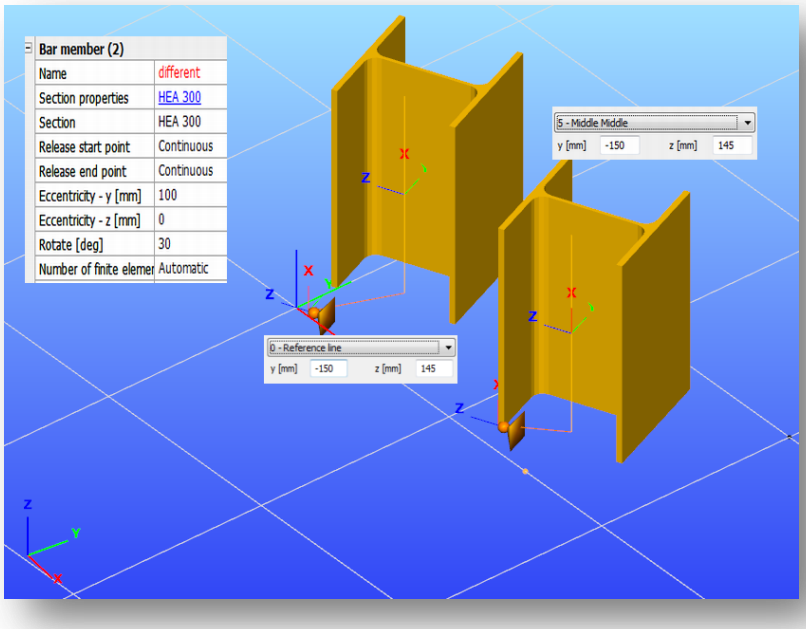
Placing a support with any of these eccentricity types, for example changing the “y” eccentricity of the supported object, the support will keep its position relative to the section. By changing the “Rotation angle” attribute of the supported object, the position of the support will rotate too.

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.

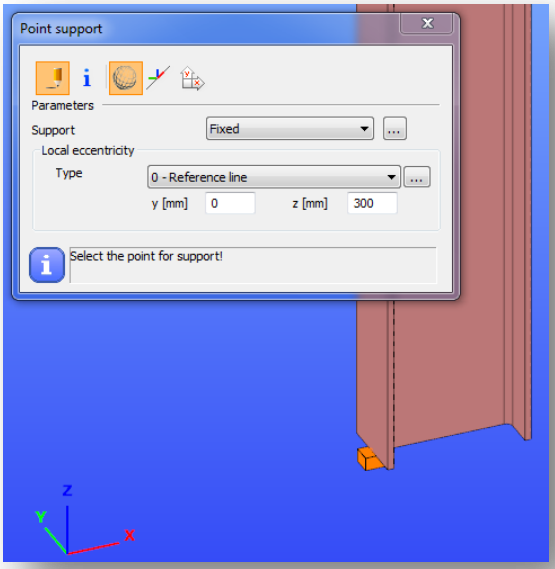


If the eccentricity of the columns is changed to y= 100mm, the new position of the supports is as follows:

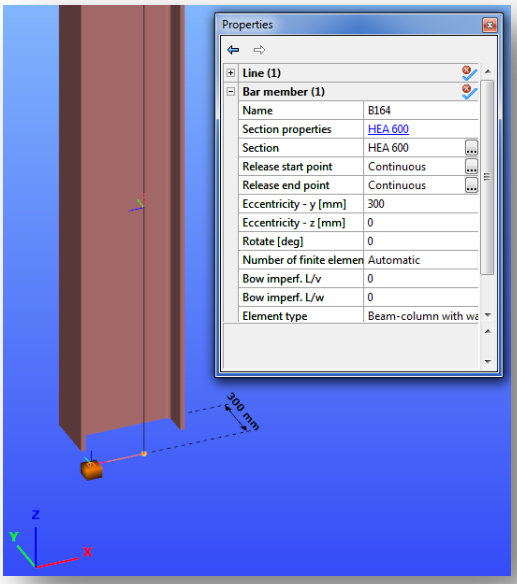


In case of the left case the position of the support did not change, because the position of the reference line of the column also did not change, but in case of the right case the support was moved with the points of the section of the column.

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.

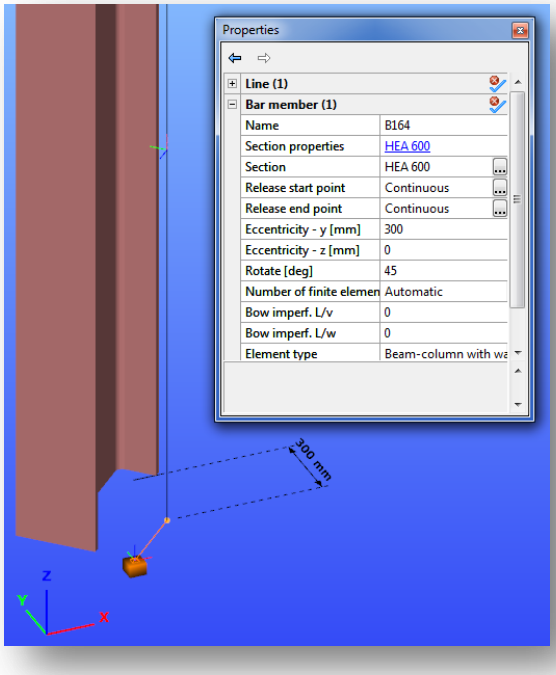


If the column is moved with 300mm in the y direction in the local coordinate system, then the position of the support will no change.

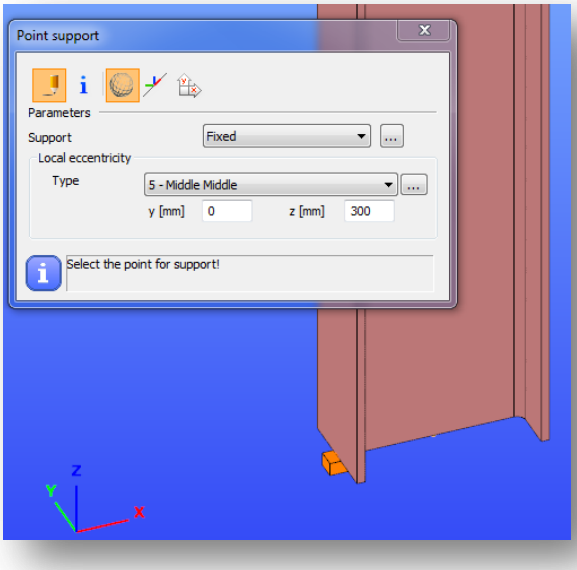


The support keeps its original relative position to the centreline of the column.

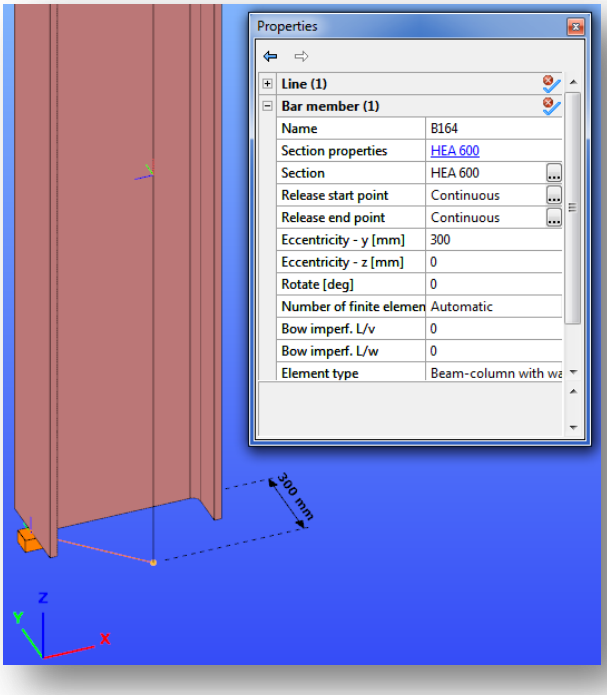
If the column is rotated with 45 degrees, the support is also rotated accordingly, but it keeps the directions of its local axis parallel to the axis of the global coordinate system, which was selected by for positioning the support.



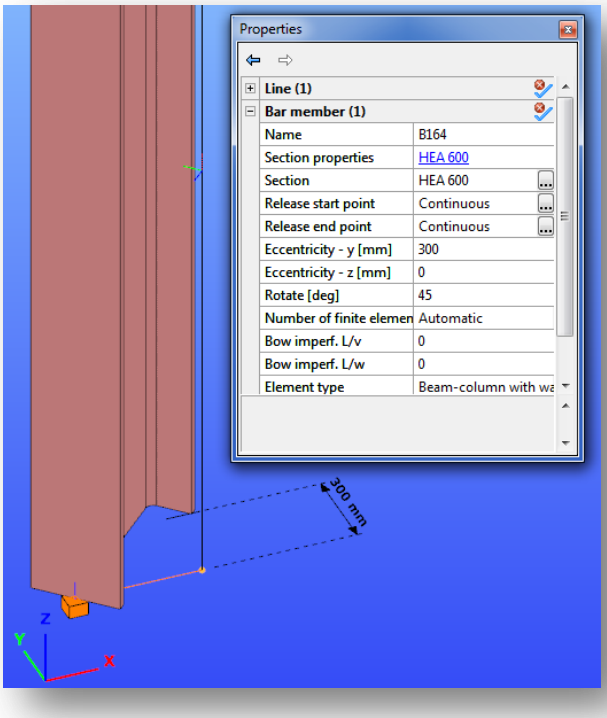
3. Example: The support is placed according to the global coordinate system, the eccentricity is 300mm in the z direction of the local coordinate system of the supported column relative to the middle-middle point of the section geometry.



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



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.

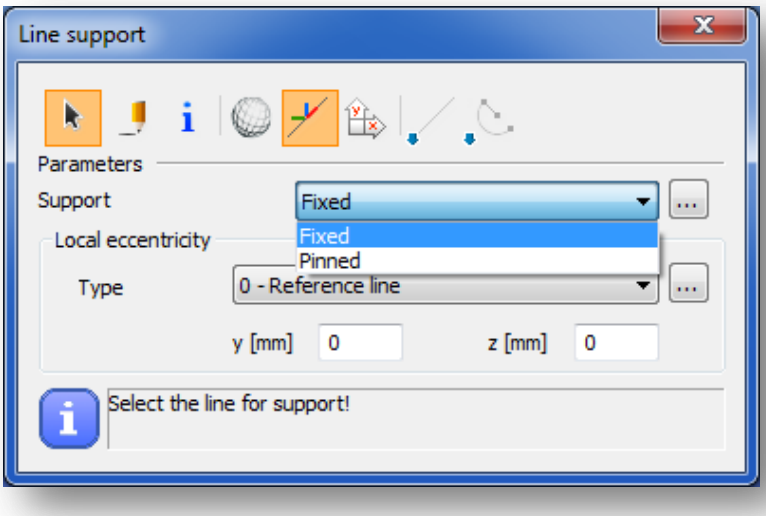


WARNING!

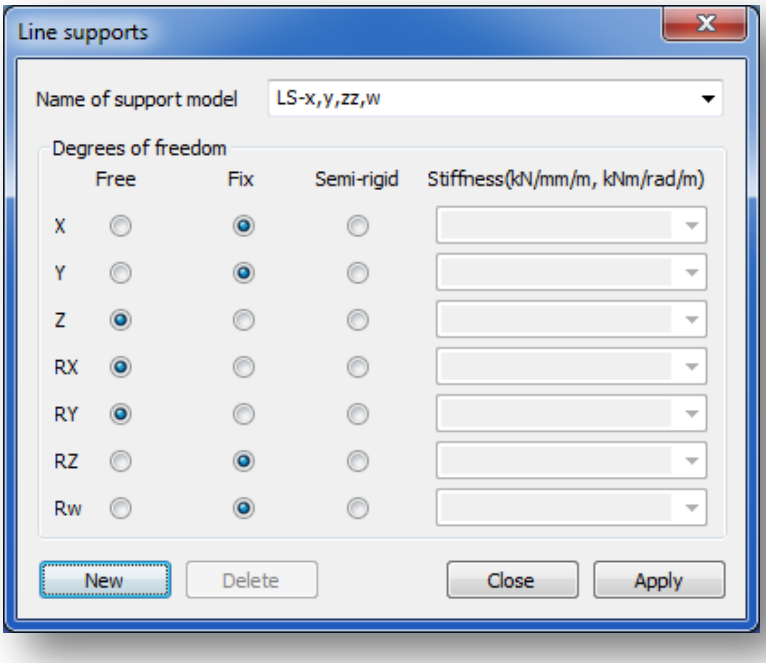
BE AWARE THAT SINCE THE FINITE ELEMENT FOR THE STEEL BEAMS AND COLUMNS HAS 7 DOFS THE POINT SUPPORT ALSO HAS 7 DOFS, THE 7TH DOF REPRESENTS THE WARPING OF THE CROSS SECTION. ACCORDINGLY IF A CROSS SECTION ON A CERTAIN PLACE IS CONSIDERED TO FIXED FOR WARPING (FOR INSTANCE IN HEAVILY STIFFENED JOINTS)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

5.8.2 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).



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.



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 select function: 

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.

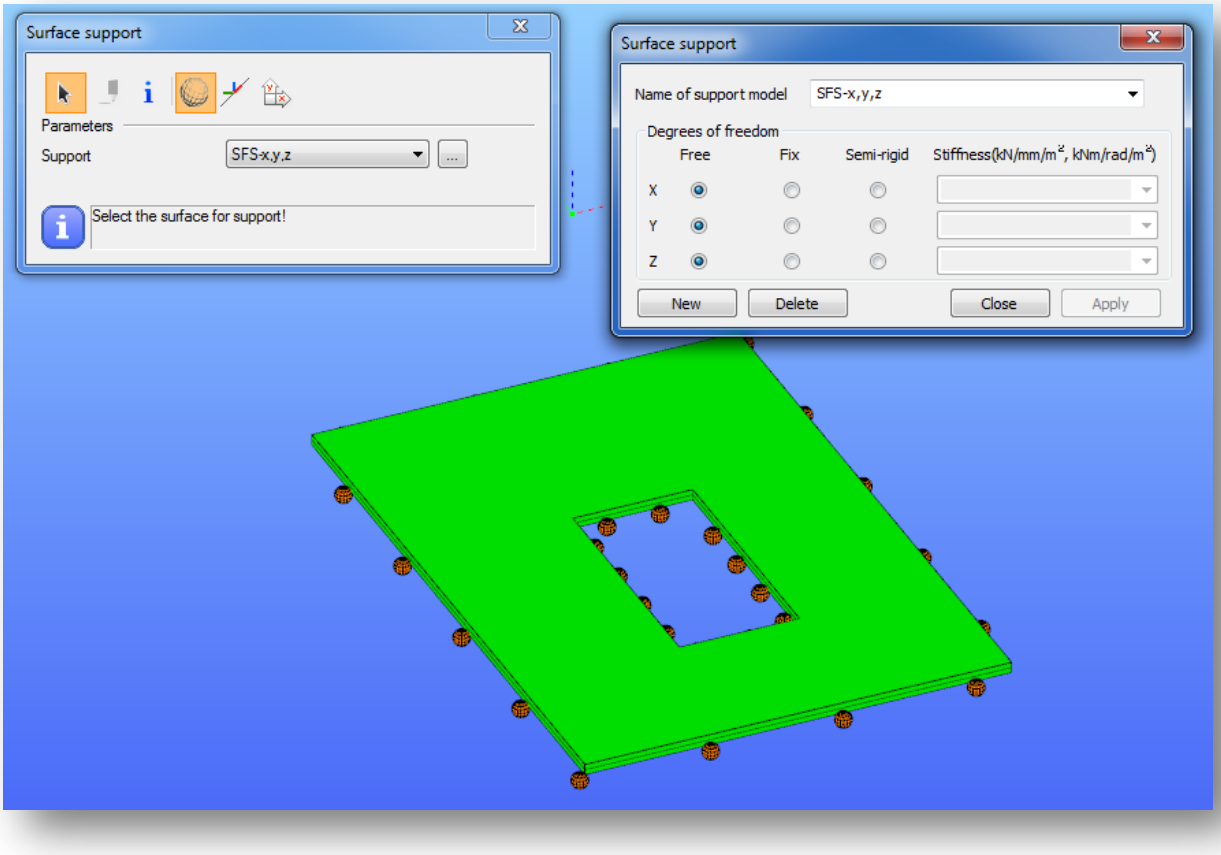
The draw function: 

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.

5.8.3 SURFACE SUPPORT ()

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.



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

5.9 LINK ELEMENTS

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

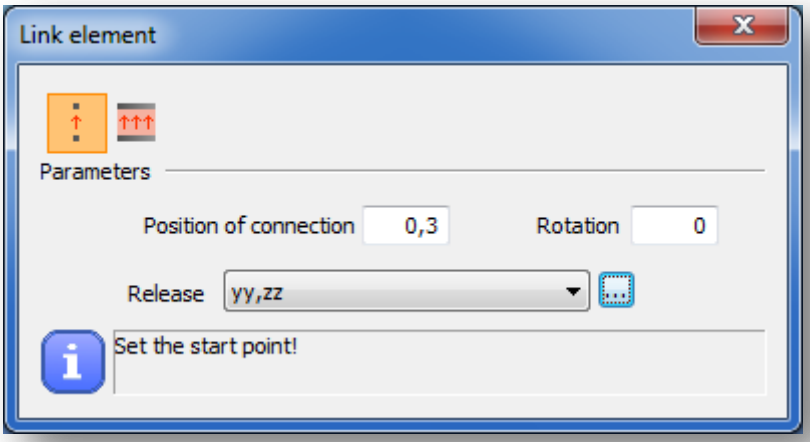
Parameters of the link element can be defined in the middle part of the dialog:

- *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.

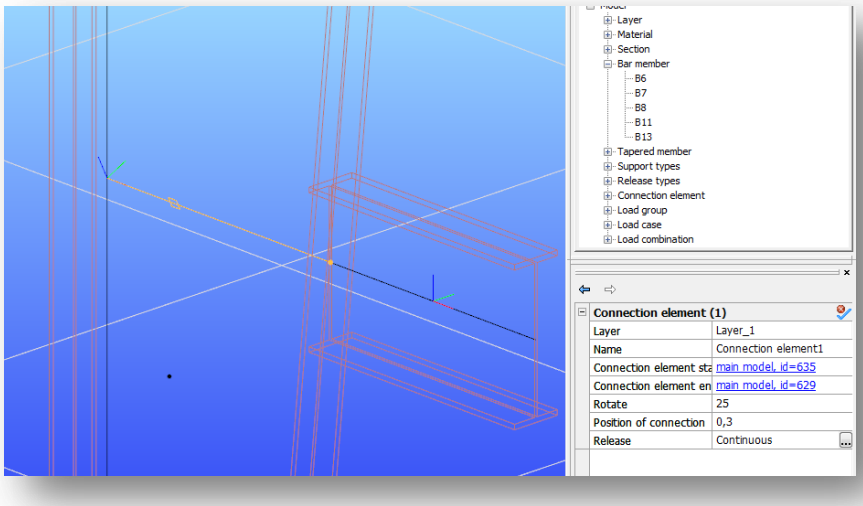
There are two different types of link element.

5.9.1 CONNECT TWO STRUCTURAL POINTS ( )

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



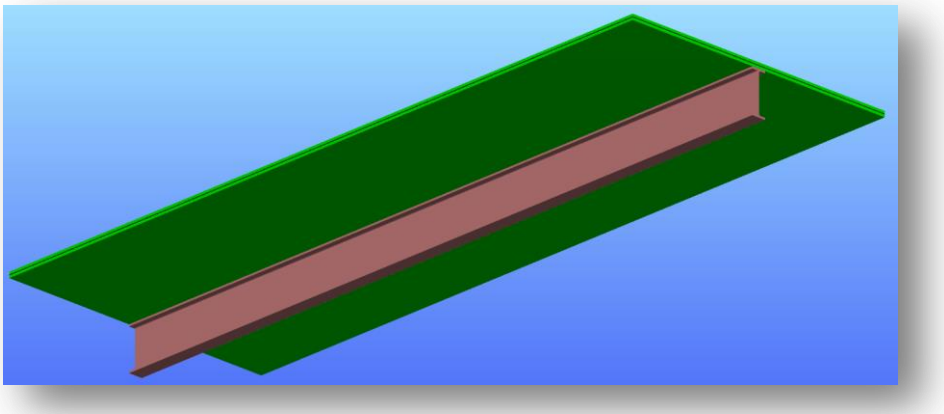
For example if there is a cantilever on a relatively high tapered column (see image below) it is advisable to connect it not directly to the reference line of the column but with the link element. If it is connected the way like that the analysis and design results are more accurate and the modelling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

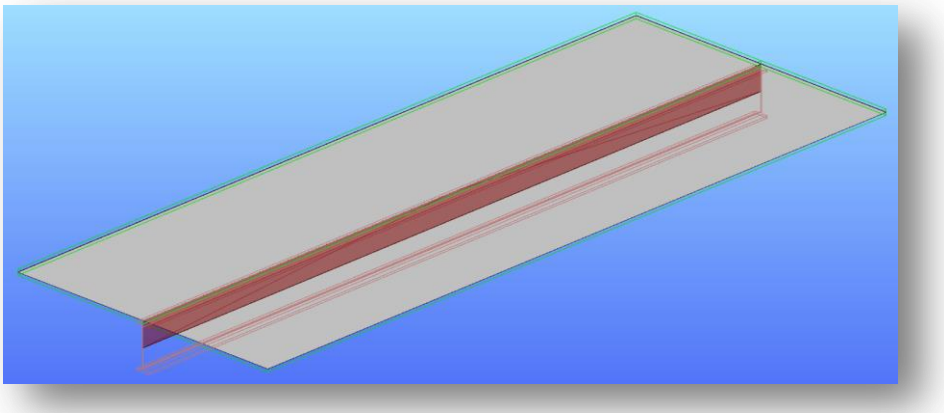


5.9.2 CONNECT TWO STRUCTURAL EDGES THAT ARE EQUAL LENGTH AND PARALLEL ( )

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.

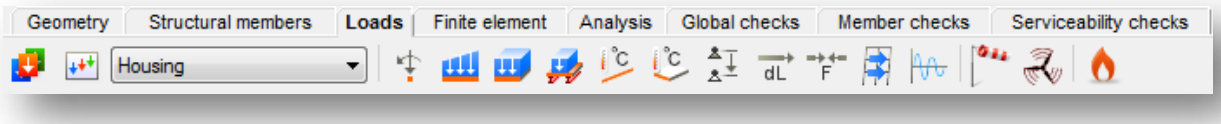




6 STRUCTURAL LOADS

6.1 BASICS

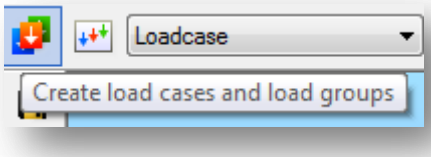
The definition of loading on a structural model is one of the most important modelling phases. Contrary to the modelling of structural members the load modelling is minutely controlled and supported by the structural codes and standards since the appropriate definition of loads ensures the major part 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

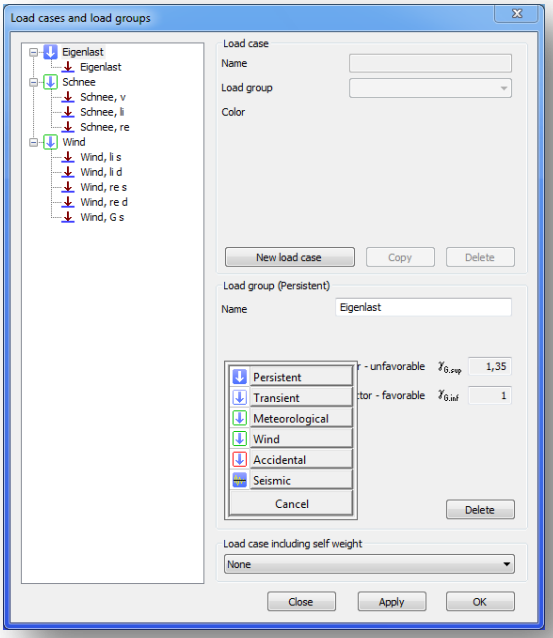
Before place any loads in the model load cases and load groups must be defined. As a default, one load case has been defined in ConSteel for quick calculations.

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

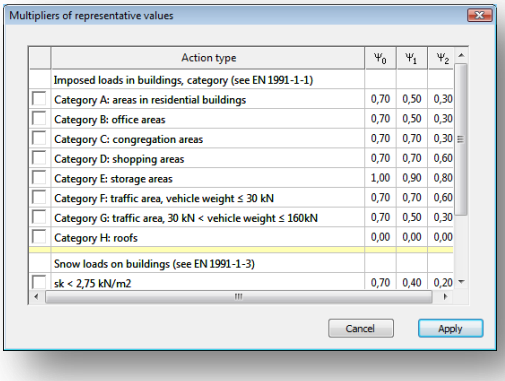


There are 5 different load group types in *ConSteel*: Persistent, Transient, Meteorological, Accidental and Seismic. As a special type of meteorological load group, there is a Wind- load group. This has to be used for the automatic wind load generation.

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



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.

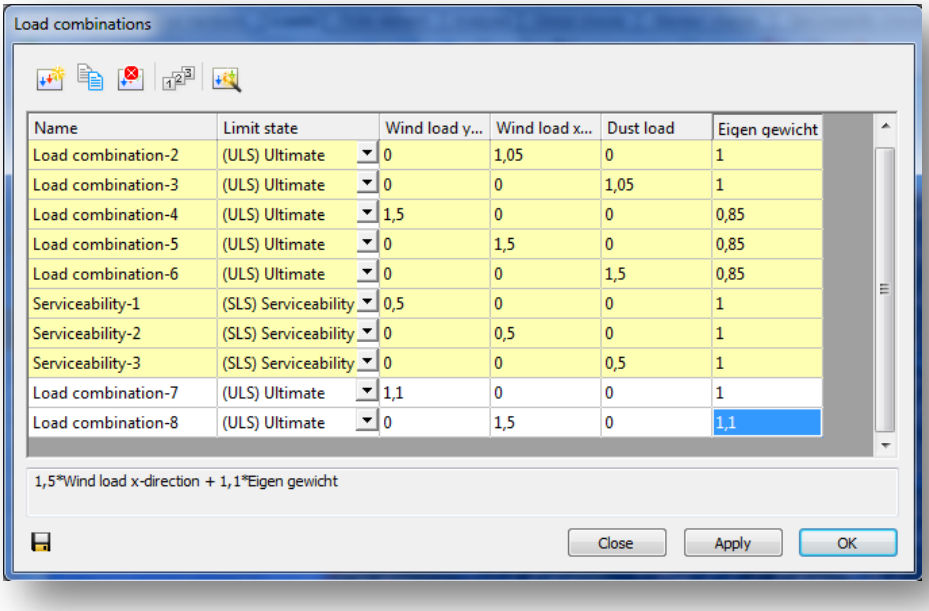


ConSteel can add the structure's dead load to one of the load cases if necessary. In order to do so please select the appropriate load case at the bottom right corner of the dialogue.

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*** function of *ConSteel*.

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

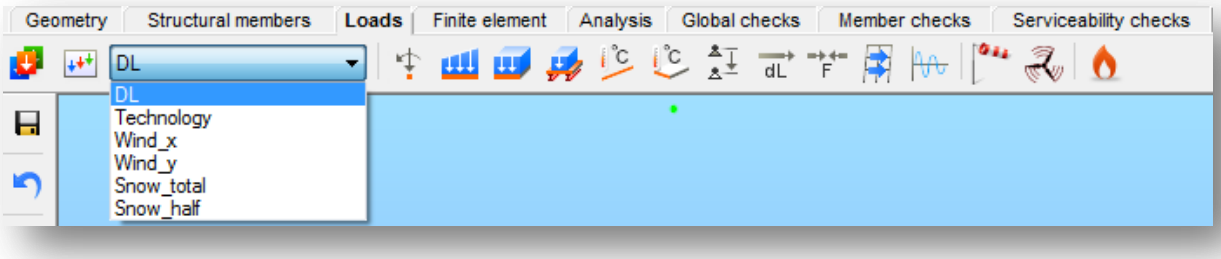


The yellow rows in the load combination table are the automatically generated combinations. The white ones are manually edited.

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 delete those rows which cannot be relevant. 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

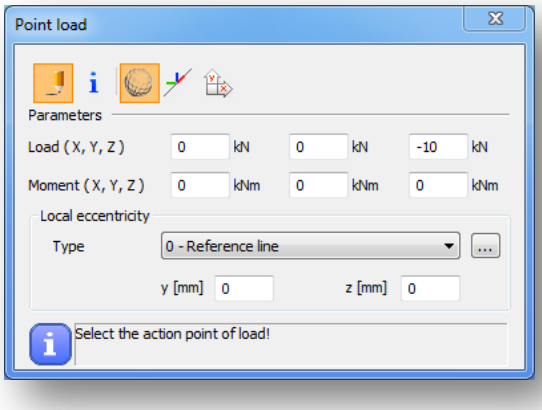
Any placed load will belong to the load case selected from the list:



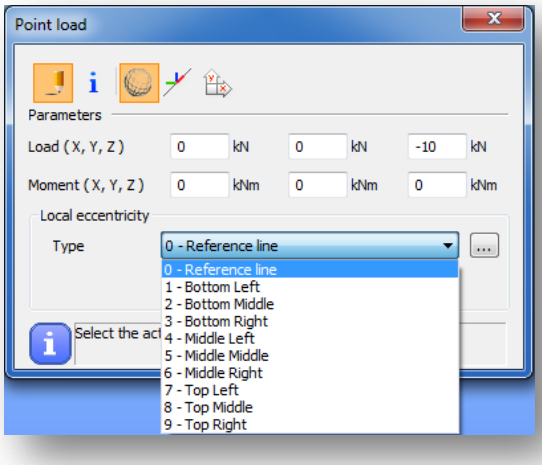
6.4.1 POINT LOAD ( )

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 system ( ).**



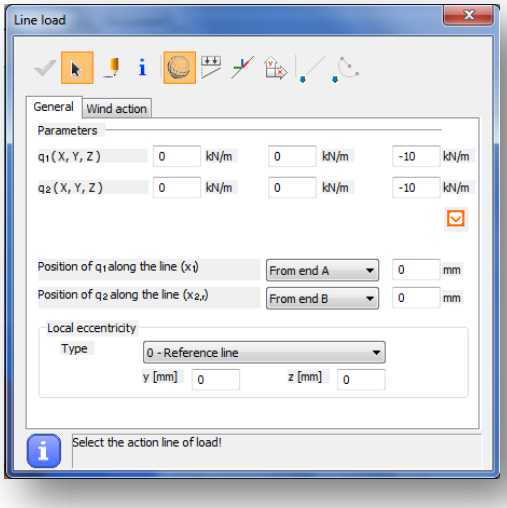
Eccentricity can be applied on the point load also. The effect of the selected coordinate system, the applied local eccentricity type and values are the same as point support (see chapter 5.8.1).



6.4.2 LINE LOAD ( )

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 ( )**, **Local ( )**, **User coordinate system ( )** and **Global projection system ( )** which is commonly used when modeling the snow load.

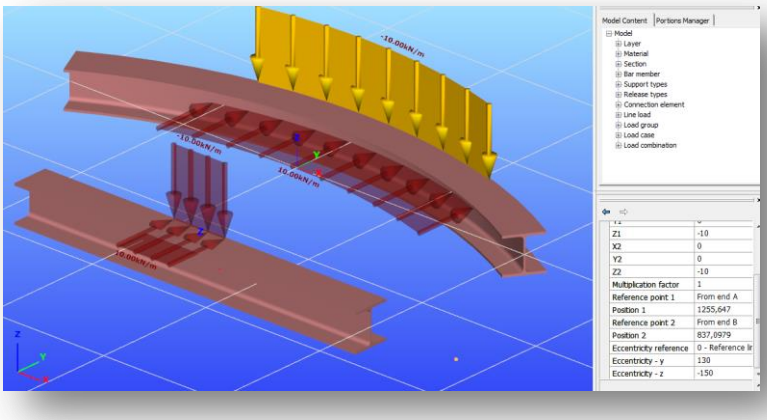


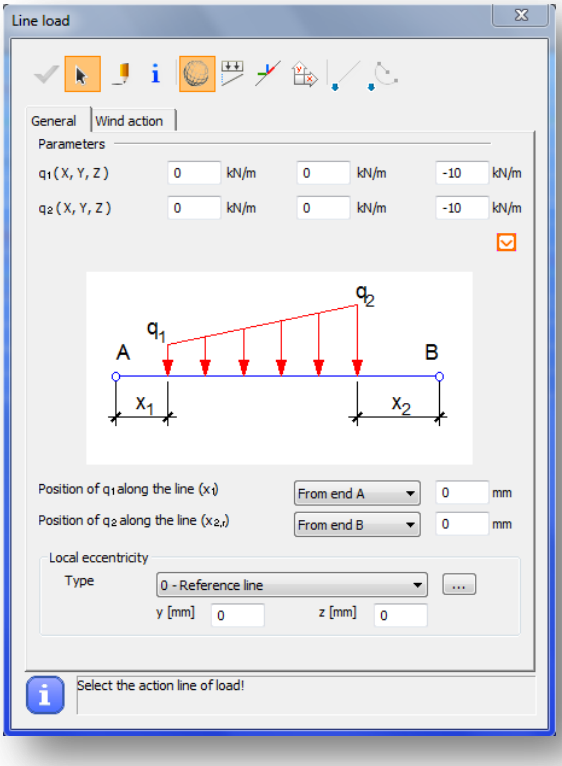
In order to place a line load click on the member using the select function ( ). Line load can be applied to multiple members at the same time by using ***PLACE LOADS*** icon ( ).

This case the members must be selected before clicking on the ***LINE LOAD*** icon on ***LOADS*** tab.

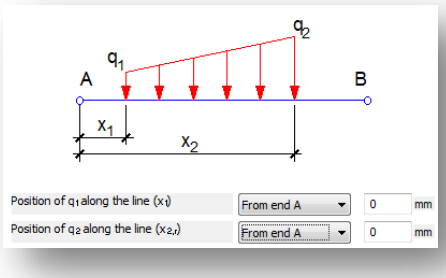
If just a partial line load need to be applied along the member then there are two ways to apply:

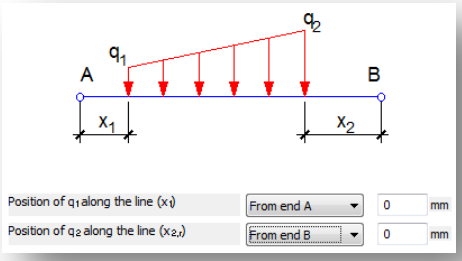
1. The first way is to use the draw function by selecting the  icon and set the start point and the end point of the line load. This function can be used both for linear and curved members in accordance with the selected drawing function .

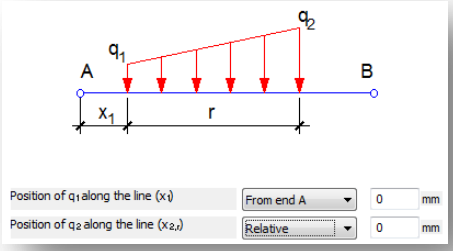




2. The second way is to give the exact distances of the start and end points of the line load from the member's end points. Using the setting of Position of q1, q2, there are 2x3 = 6 possible option to define a partial line load. The "Position of q1" set to "From end A" has 3 options, and setting it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

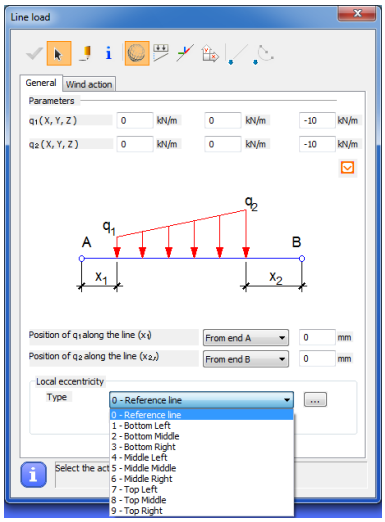






This is a flexible way for numerical input the line load positions.

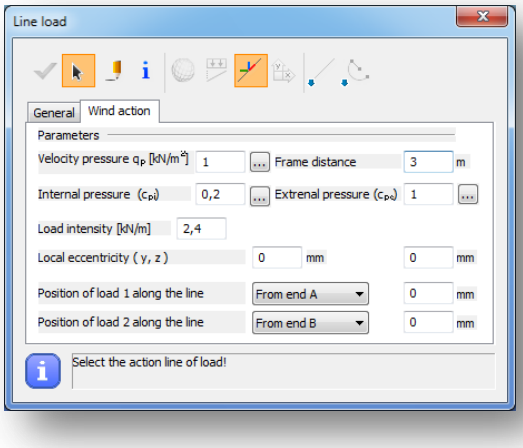
Eccentricity can be applied on the line load also. The effect of the selected coordinate system, the applied local eccentricity type and values are the same as point support (see chapter 5.8.1).



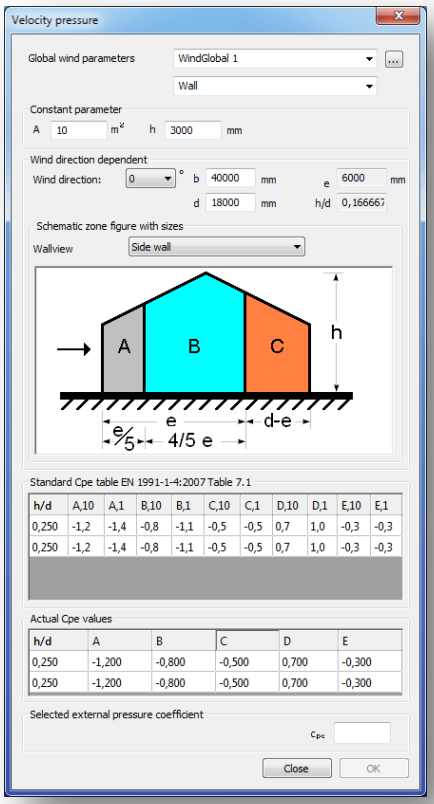
By clicking the  icon, the attributes of the selected line load fill out the parameters of the line load window, even if it was set by the draw function.

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. The three dots icons provide help determining the correct values of the parameters. It is important to know that load intensity is calculated using the parameters added on the window and these parameters can be modified later but every line wind load is independent from the others. Therefore modifying the parameter of a line wind load will not have effect on the other line loads. If one or more parameter(s) need to be modified for all line wind loads then all must be selected and the appropriate parameter(s) must be modified.

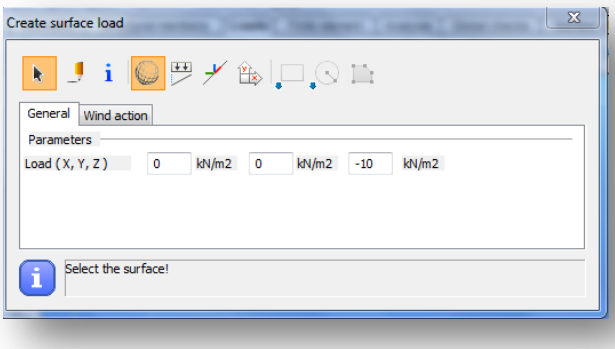


By clicking on the *Internal Pressure* the window below will appear. It is possible to view and select the C_{pe} values after choosing the type of the wind surface (wall, flat roof, duopitch roof,etc).



6.4.3 SURFACE LOAD

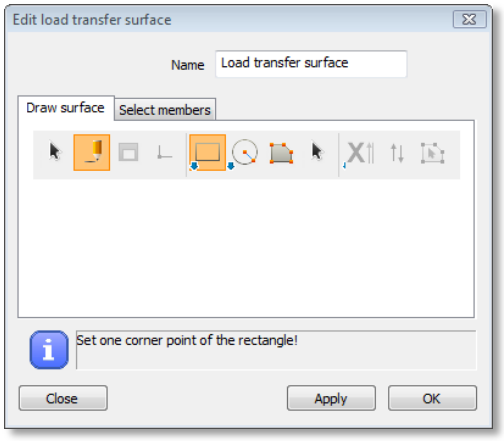
Surface loads can be defined using the same method like in point or in line loads. No surface moment loads can be placed.



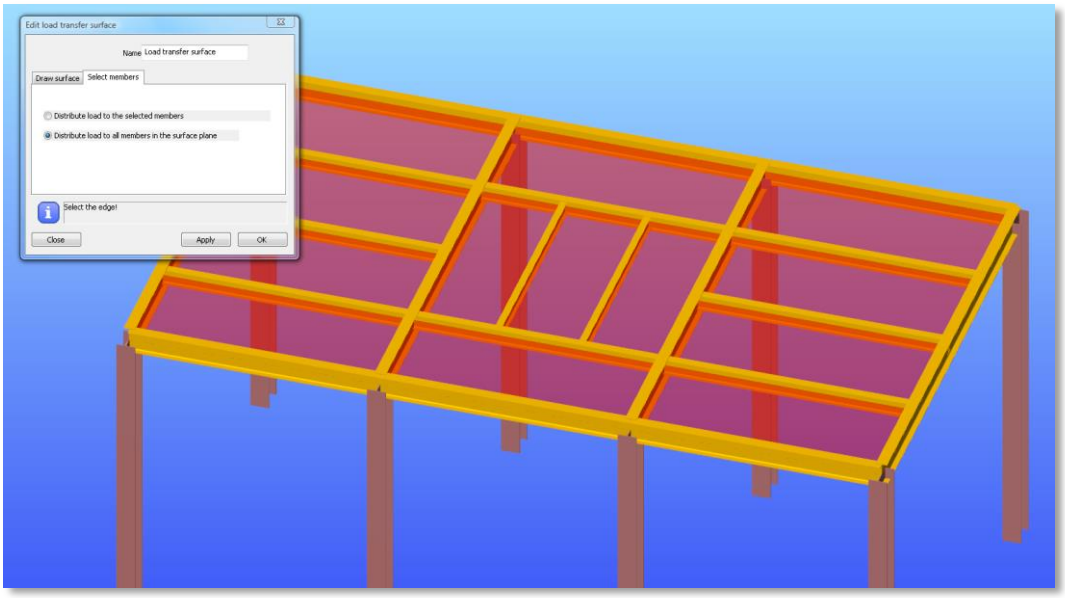
6.4.4 LOAD TRANSFER SURFACE ( )

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 icon on the ***LOADS*** tab a dialog window appears.



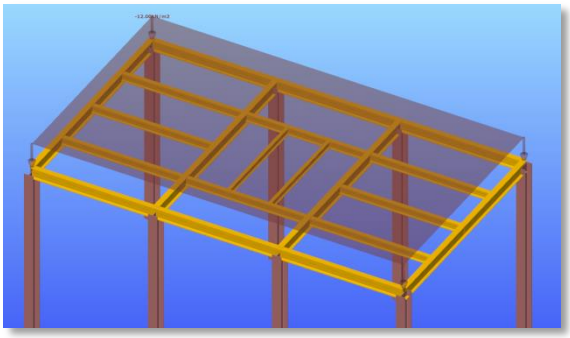
Just like in the two-dimensional figure drawing there are different possibilities to draw the surface: draw a rectangle, draw leaning rectangle, draw circle, draw polygon. After drawing the surface it is possible to select the members to which the surface distributes the load. There are two options: 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.

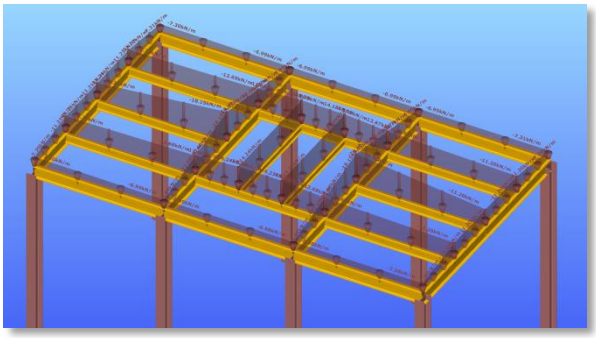


It is also possible to choose the first option and select members for carrying the surface load or remove members from the selection using the SHIFT + Left click. After the corresponding members have been selected and ***OK*** is clicked then the surface is created.

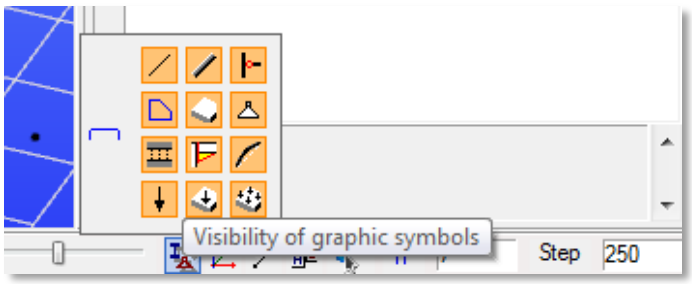
Surface load can be placed by using the method described in chapter 6.4.3.

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.

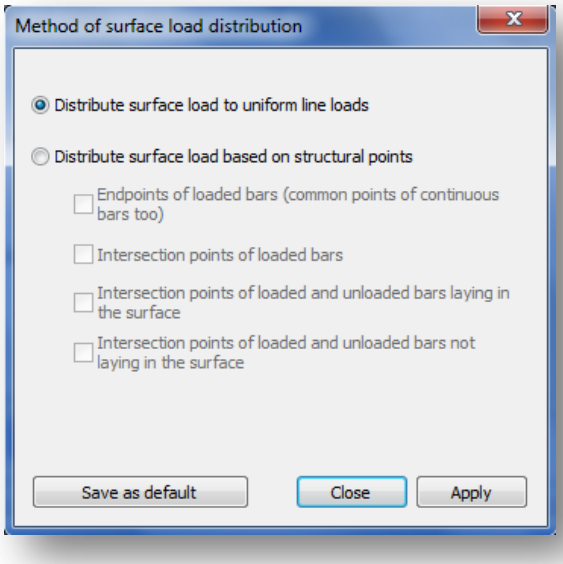




There are two possible methods to convert surface load to the selected members:

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

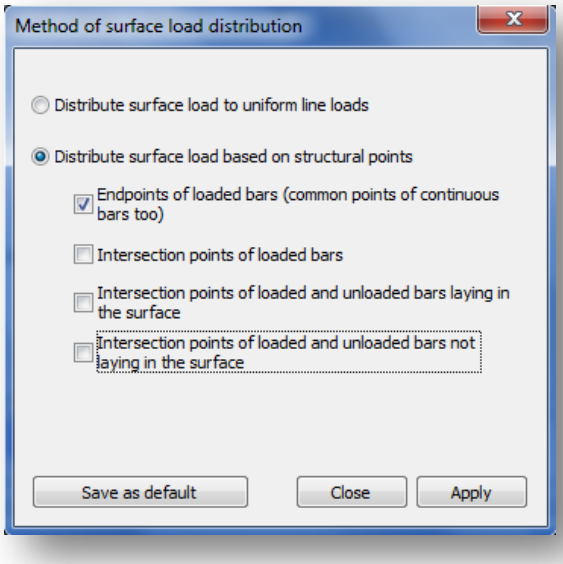


Converting the surface load to line loads based on structural points has the following background:

(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:

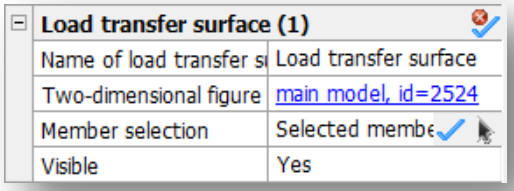
- the resultant force of all the line loads is the same as for the surface load
- the line loads are *linear* on all the selected members
- the end value of the line loads on the selected structural points for all the selected members are equal



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.

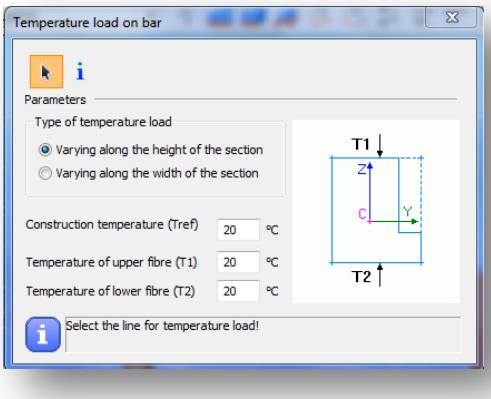


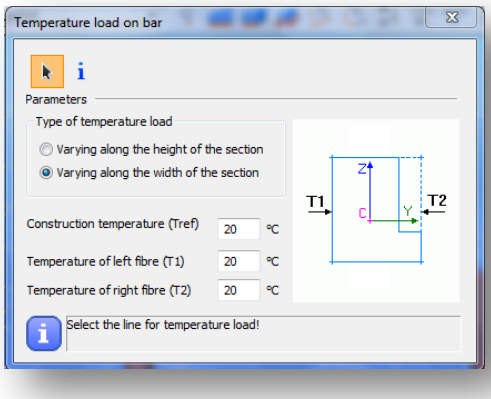
If selected members option is chosen and then the black arrow is clicked then previously selected members can be removed from the selection or new members can be added.

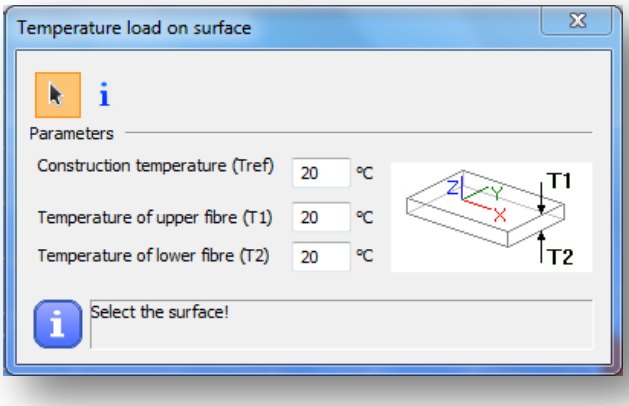
6.4.5 TEMPERATURE LOAD ( )

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.



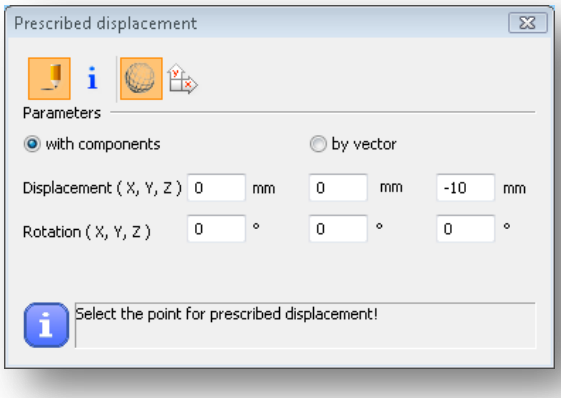




6.4.6 PRESCRIBED DISPLACEMENT ( )

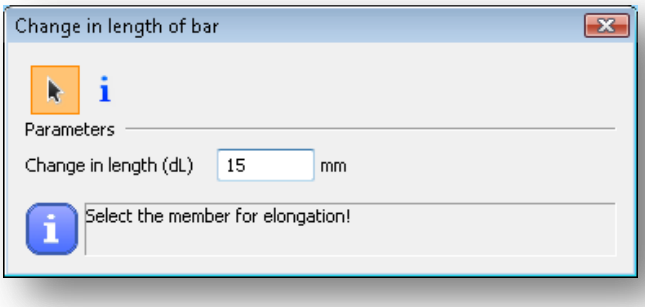
Prescribed displacement can be applied for point supports.

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

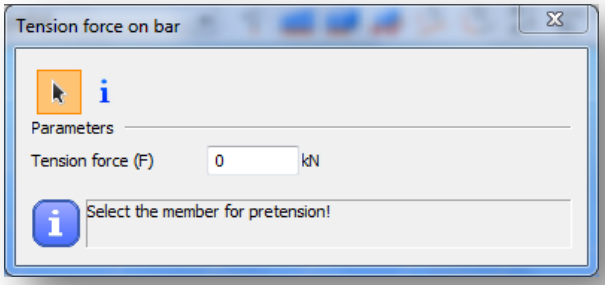


6.4.7 CHANGE IN LENGTH ( ), TENSION FORCE ( )

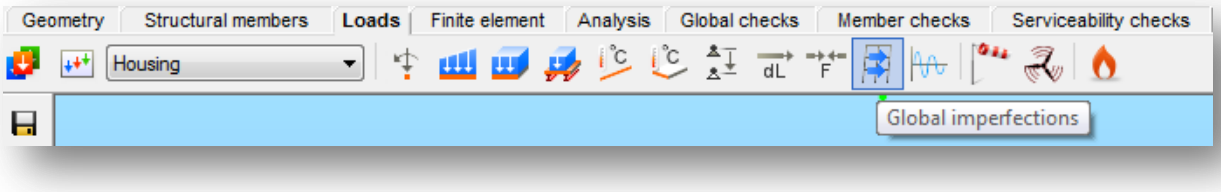
The change of the original member system length can be set and applied as a load on bar members with the change in length function. With this function prestress effect can be modelling.



Tension force can be applied on bar members as a load.



6.5 GLOBAL IMPERFECTIONS



Three types of global imperfection can be applied on the model with the ***GLOBAL IMPERFECTION*** 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 7.4).

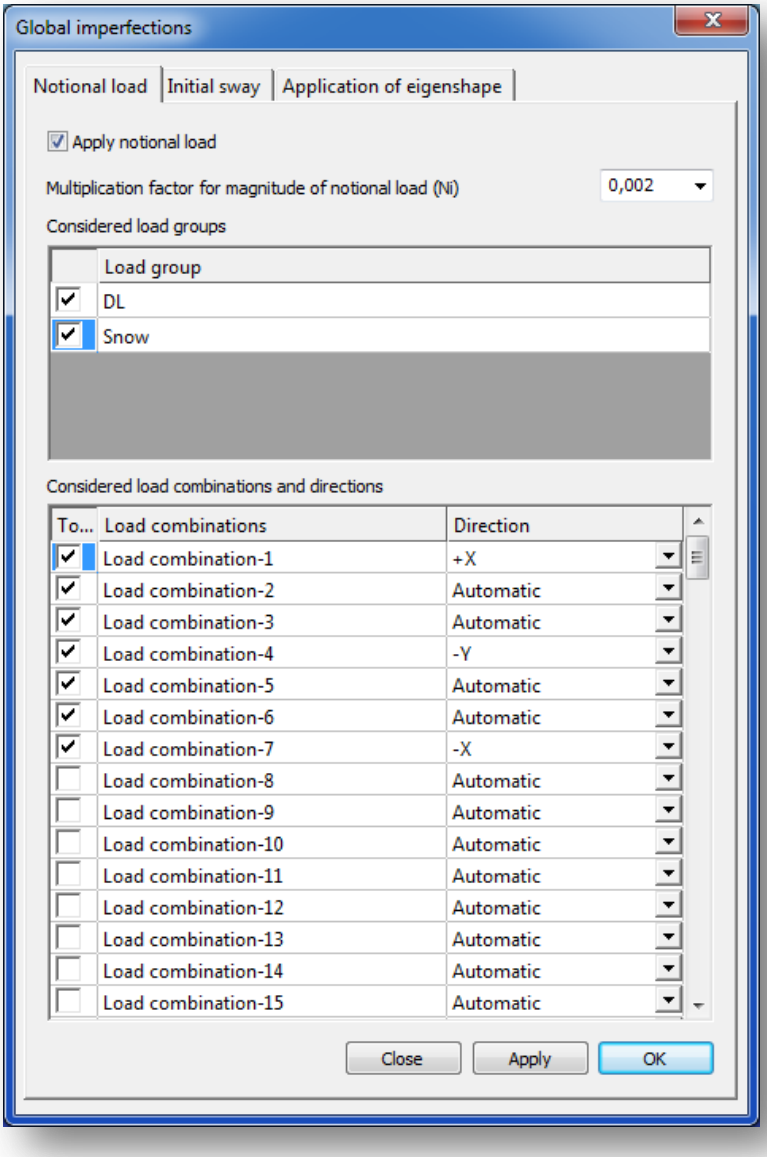
6.5.1 NOTIONAL LOAD

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 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.



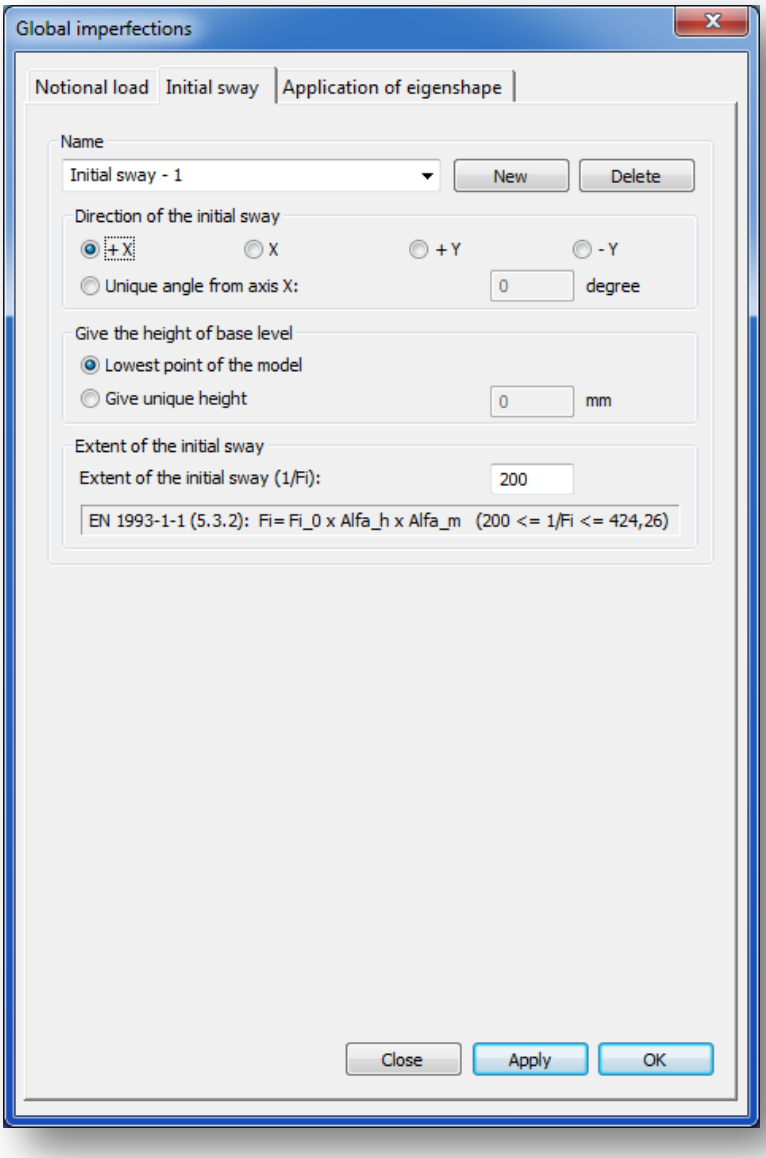
6.5.2 INITIAL SWAY

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.



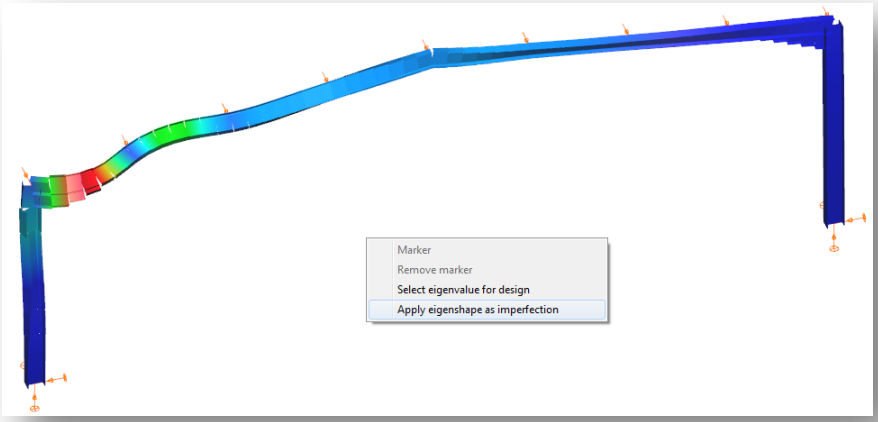
6.5.3 APPLICATION OF EIGENSHAPE

Previously calculated buckling modes can be applied as a geometric imperfection with user defined amplitude. Even more buckling modes can be easily superposed as one imperfection.

6.5.3.1 APPLY EIGENSHAPES

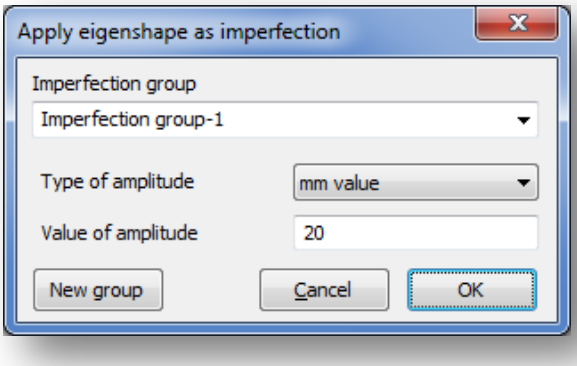
To apply eigenshapes as an imperfection previously the buckling analysis has to be run.

On the ***ANALYSIS*** tab from the dropdown menu the proper eigenshape has to be chosen. To apply the eigenshape as an imperfection, clicking in the graphical area with right mouse button and choosing the 'Apply eigenshapes as an imperfection' option.



In the appeared dialog the following settings have to be set:

- *Imperfection group*: any eigenshape will belong to an imperfection group, which can be selected from the list. With the ***NEW GROUP*** button a new group can be created. One load 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.
- *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



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

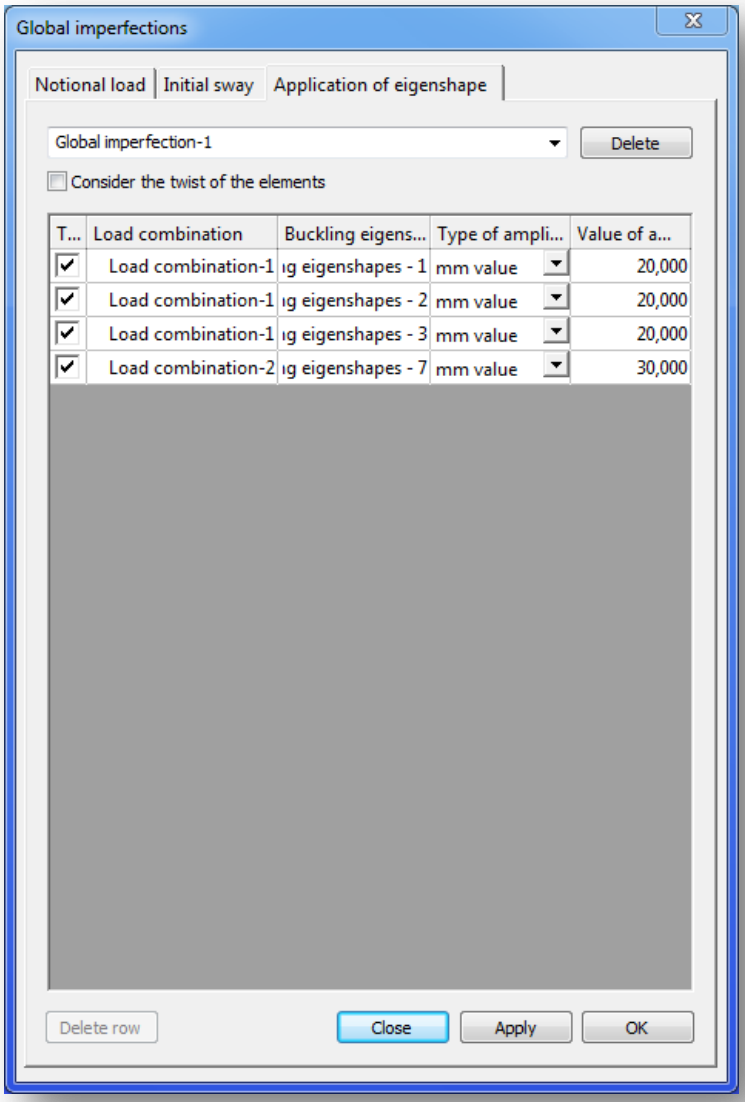
6.5.3.2 MANAGING EIGENSHAPES

Previously created imperfection groups and 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 applied eigenshapes in the selected imperfection group. With the checkboxes in the first column each eigenshape can be turn off and on. With the last two columns the type of the amplitude and the value of the amplitude can be modified.

With the ***DELETE ROW*** button the selected eigenshapes can be deleted from the imperfection group.



6.6 EARTHQUAKE

Seismic load can be determined according to the modal response spectrum analysis.

For each vibration shapes the equivalent static loads are generated. Combination of the seismic loads in one direction is calculated according the CQC (Complete Quadratic Combination) method. Combination of the components of the seismic actions is computed using all of the three following combinations:

a) E_{Edx} "+" 0,30 E_{Edy} "+" 0,30 E_{Edz}

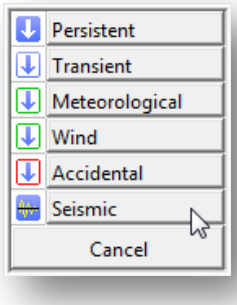
b) 0,30 E_{Edx} "+" E_{Edy} "+" 0,30 E_{Edz}

c) 0,30 E_{Edx} "+" 0,30 E_{Edy} "+" E_{Edz}

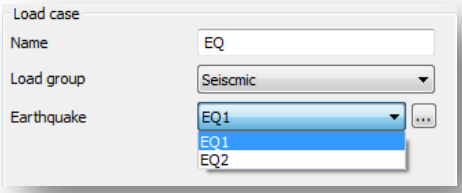
6.6.1 SEISMIC LOAD GROUP AND LOAD CASE

The first step is to create the seismic load group and the necessary number of seismic load case. For one seismic effect one load case is necessary.

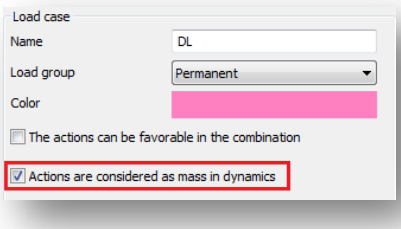
For the seismic loads the dedicated seismic load group must be used.



The proper seismic effect must be set in each load cases.



In case of each load cases (permanent, variable etc.) must be set to consider the loads as mass, if it should be taken in to account in the calculation. The setting can be done on the field of the load cases.



6.6.2 EARTHQUAKE EFFECT ( )

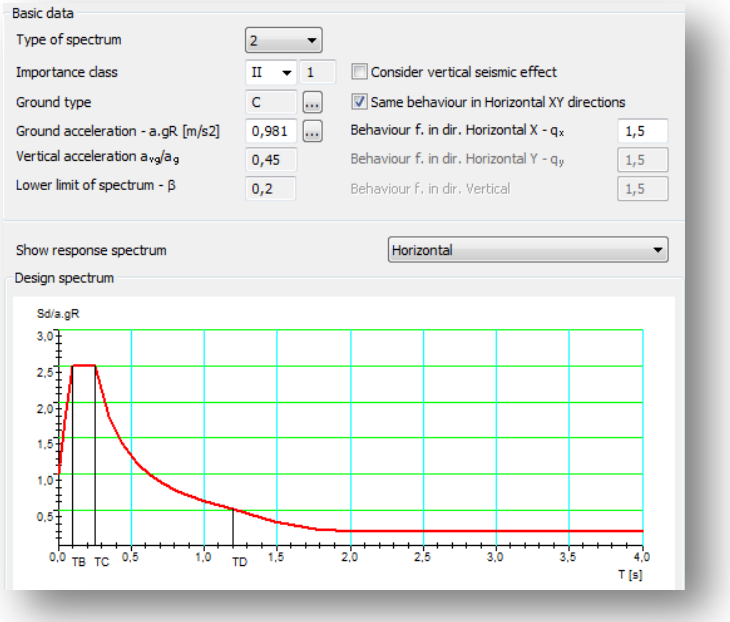
EARTHQUAKE EFFECT dialog can be opened in two ways:

1. Directly from the ***LOAD CASES AND LOAD GROUPS*** dialog. Click on the tree dots button on the seismic load case's field.
2. With the dedicated ***EARTHQUAKE EFFECT*** function on the ***LOADS TAB***

Response spectrum can be defined in two ways:

1. According to the Eurocode 8
2. Entering the points of the spectrum by the user

6.6.2.1 STANDARD DESIGN RESPONSE SPECTRUM

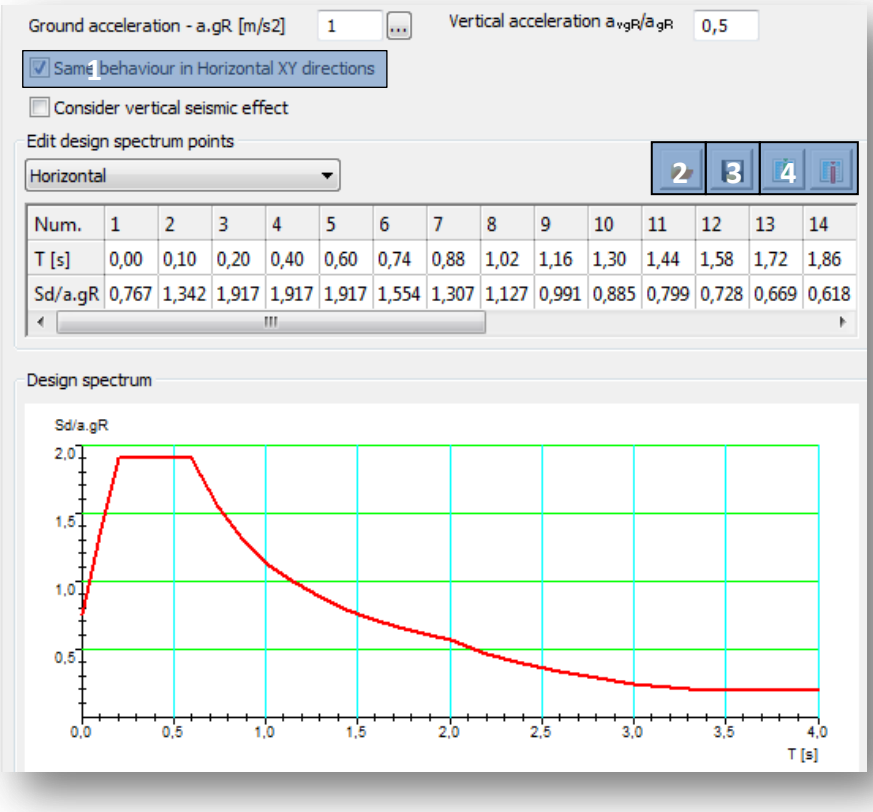


The standard design response spectrum can be defined according to the EuroCode 8. The parameters of the spectrum are automatically loaded from the selected national annex.

The following standard parameters can be changed on the dialog:

- Type of the spectrum (if the selected nation annex allow)
- Importance class
- Ground acceleration
- Behaviour factor (it can be same in the horizontal XY directions or different)

6.6.2.2 USER DEFINED DESIGN RESPONSE SPECTRUM



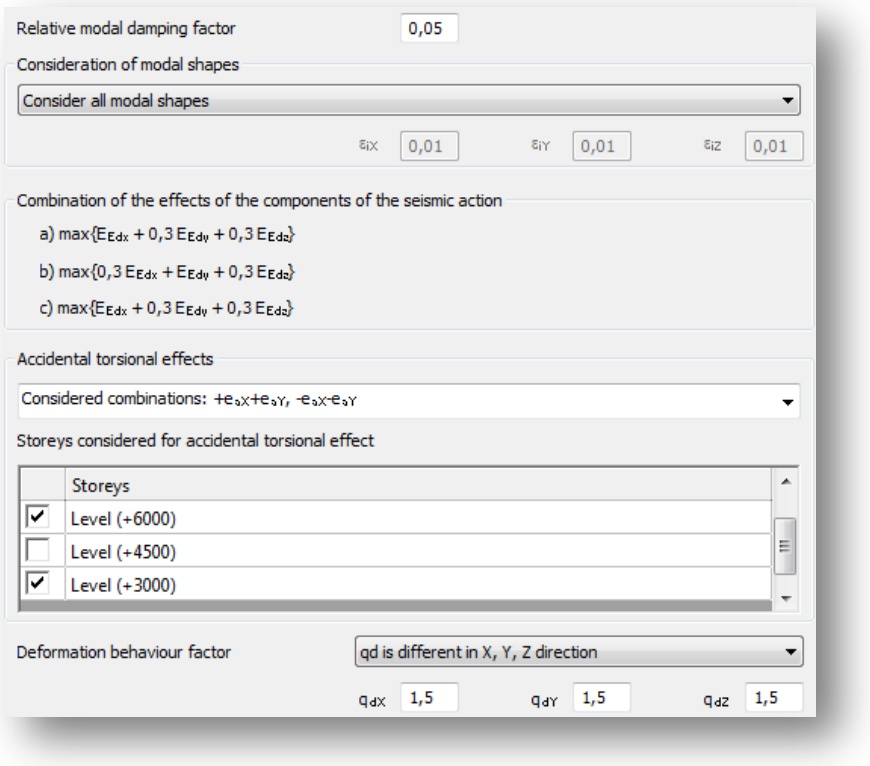
User defined response design spectrum can be set on the following ways:

1. Entering the point of the spectrum (S_d/a_{gR}) in the proper time
2. Formerly entered and saved user spectrum can be loaded
3. Existing standard spectrum can be cloned as a user defined and can be modified in the various time

Different user defined design spectrum can be set in the three directions (X, Y, Z). If in the horizontal X, Y directions the behaviour of the structure is same, it is possible to set one spectrum in the two directions (#1).

The set design spectrum can be saved (#3) to **.csv** or **.txt** file and can be loaded with the open function (#2). Number of the columns of the table can be increased and decreased with the dedicated bottoms (#4).

6.6.2.3 SEISMIC ANALYSIS SETTINGS



Analysis settings can be found on the second tab on the earthquake dialog. On this tab can be set the following settings:

- Relative modal damping factor
- Consideration of the modal shapes: two options can be chosen. Consider all modal shapes or neglect modal shapes under the threshold values. Different values can be set in the three directions (X, Y, Z).
- Accidental torsion effect: effect of the uncertainty of the location of masses can be considered in the calculation
- Considered storey: for the accidental torsion effect calculation must be selected the storeys where should be worked the effect. Storeys can be added in the portions manager.
- Deformation behaviour factor: value of the deformation behaviour factor can be set. Deformation factor can be equal with q, equal in X, Y, Z directions, or different in X, Y, Z directions.

6.7 WIND LOAD GENERATOR

Wind load generator can generate wind load for the most common type of structures according to the following standards:

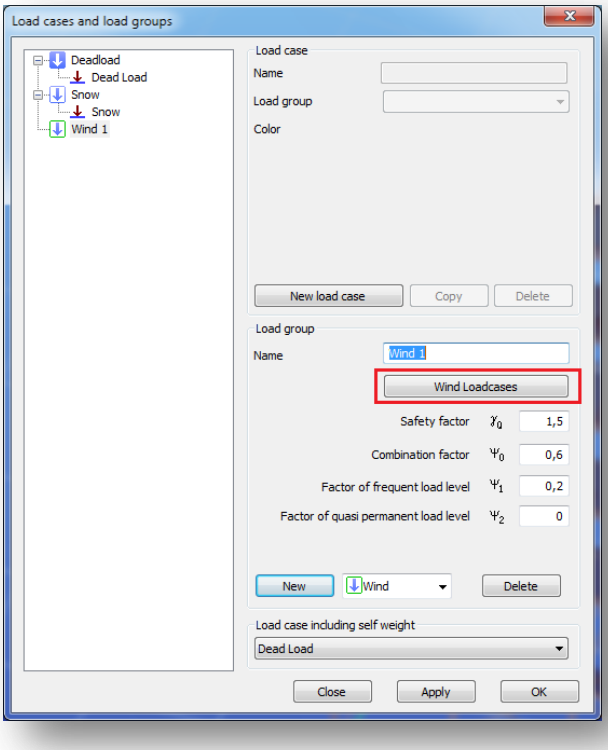
- EuroCode 1 (EN 1991-4)
- Spanish standard (SE-AE)



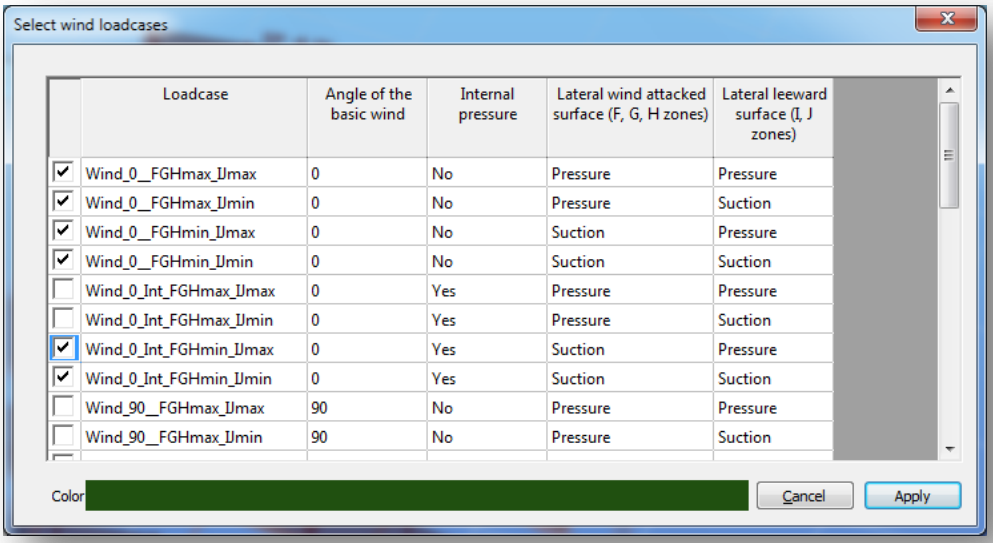
HEIGHTS OF THE STRUCTURE (h) FOR GENERATING WIND LOADS ARE CALCULATED FROM THE GLOBAL Z=0 LEVEL. THEREFORE THE STRUCTURAL MODEL HAS TO BE PLACED TO ITS REAL HEIGHT IN THE MODELLING AREA MEASURED FROM THE GLOBAL Z=0 LEVEL. FOR EXAMPLE LOWEST EDGE OF A WINDWARD WALL CANNOT BE PLACED BELOW THE GLOBAL Z=0 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 WIND LOAD CASES

Wind load cases must be defined as a first step of the wind load generation. If a wind load-group is created a dedicated button will be active to create the wind load cases.



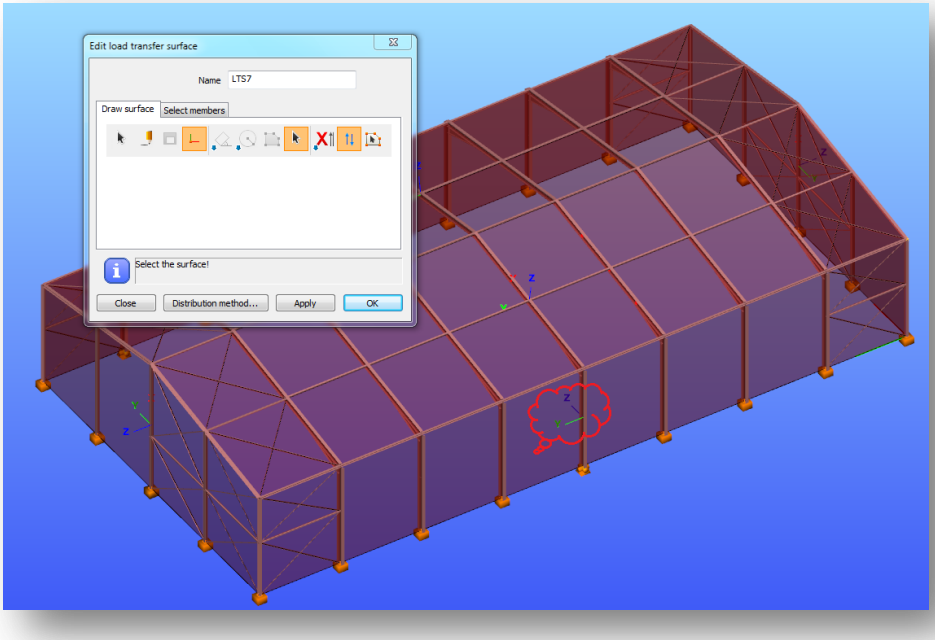
The dialog window contains all the possible EC wind load cases. Multiple selections are possible by pressing the ***SHIFT*** buttons.



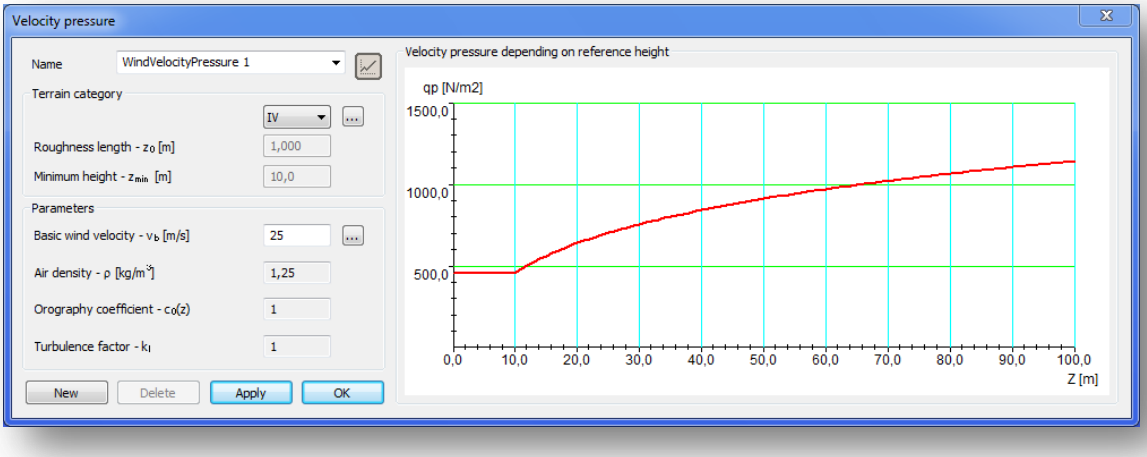
6.7.2 WIND SURFACES

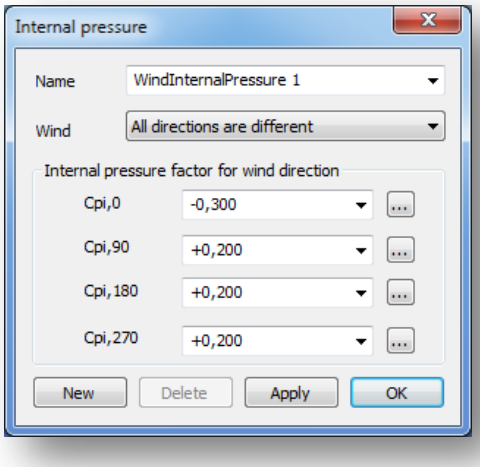
Any load transfer surface can be selected as a wind surface. It is advised to cover all wind attacked sides of the model with load transfer surfaces.

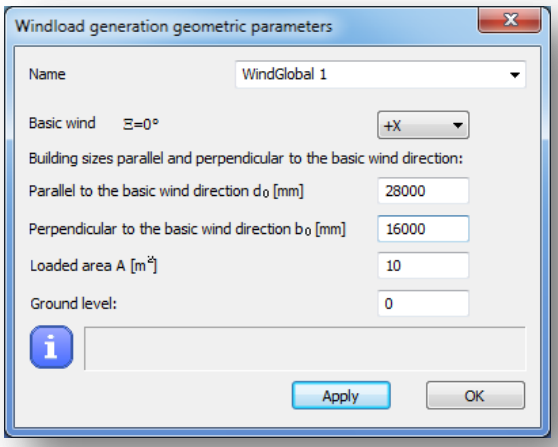
It is important to check the local axes directions of the load transfer surfaces. All Z axes must point out of the building. If modification is necessary then it is recommended to use the *invert z axis* tool on the ***EDIT LOAD TRANSFER SURFACE*** dialog (see image below).



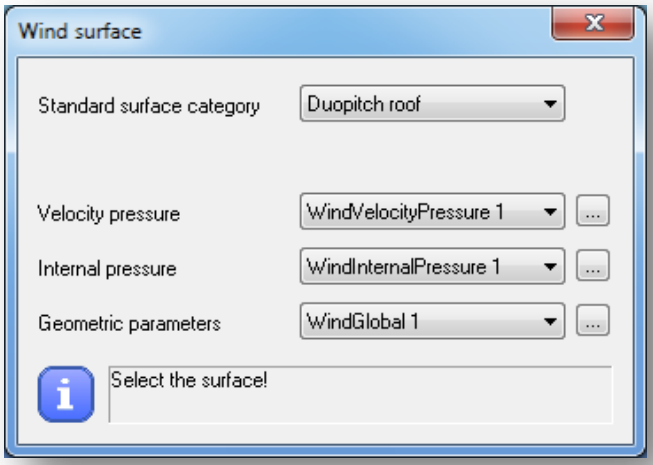
Before selecting the wind surfaces, velocity pressure, internal pressure and geometric parameters must be set.







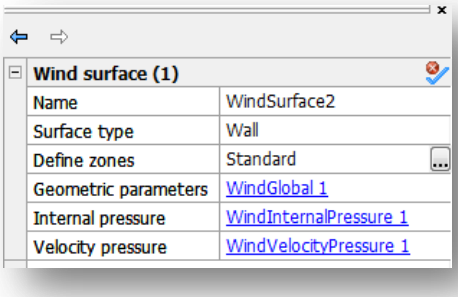
The next step is to assign standard categories to the load transfer surfaces. There are 7 categories: wall, flat roof (sharp eaves), flat roof (with parapets), flat roof (curved eaves), flat roof (mansard eaves), monopitch roof, duopitch roof. If duopitch roof is chosen two surfaces must be selected directly one after the other.



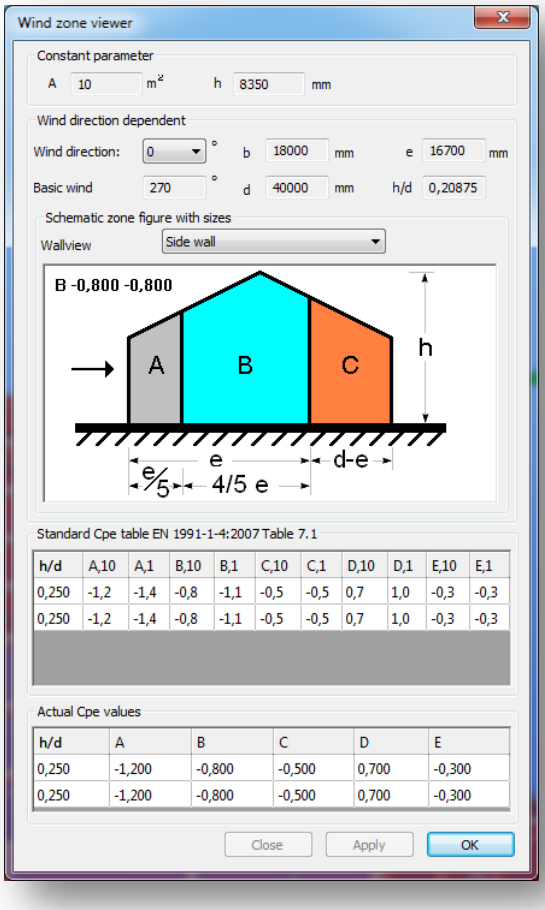


Wind surface has a symbol of an air sleeve.

Wind surface can be modified by selecting and setting the parameters

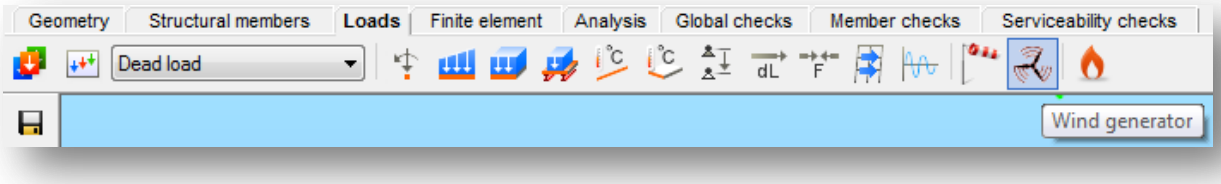


If the three dots icon is clicked the ***WIND ZONE VIEWER*** window is opened and the standard zones and Cpe values can be checked.

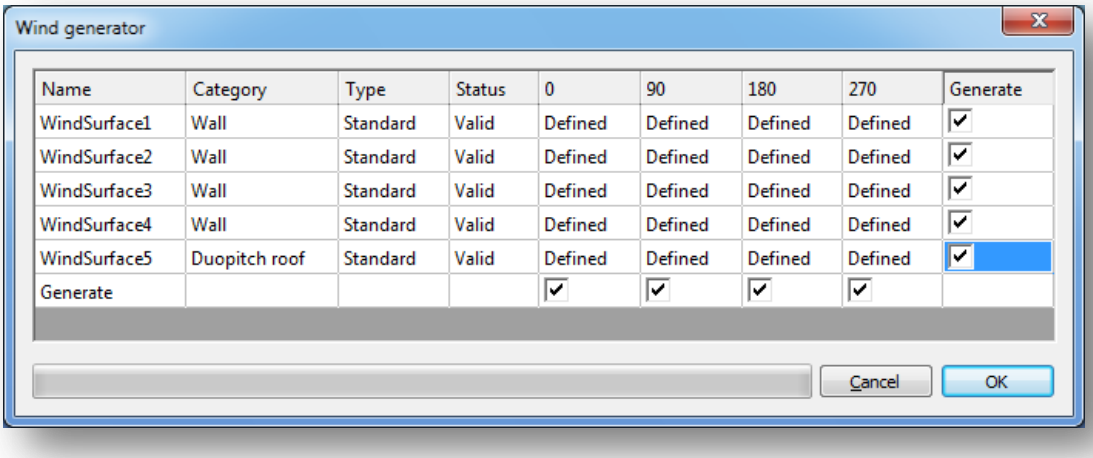


6.7.3 WIND GENERATOR

Wind generator function can be found on the ***LOADS TAB***.



The function can be used to automatically generate wind load based on the wind surface settings (internal pressure, velocity pressure, geometric parameters). Each direction and wind surface can be turned on and off for analysis.


Wind generator
Name
Category
Type
Status
0
90
180
270
Generate
WindSurface1
Wall
Standard
Valid
Defined
WindSurface2
WindSurface3
WindSurface4
WindSurface5
Duopitch roof
Generate
Cancel
OK

7 STRUCTURAL ANALYSIS

7.1 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.

7.2 FINITE ELEMENTS

7.2.1 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 elements model 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.

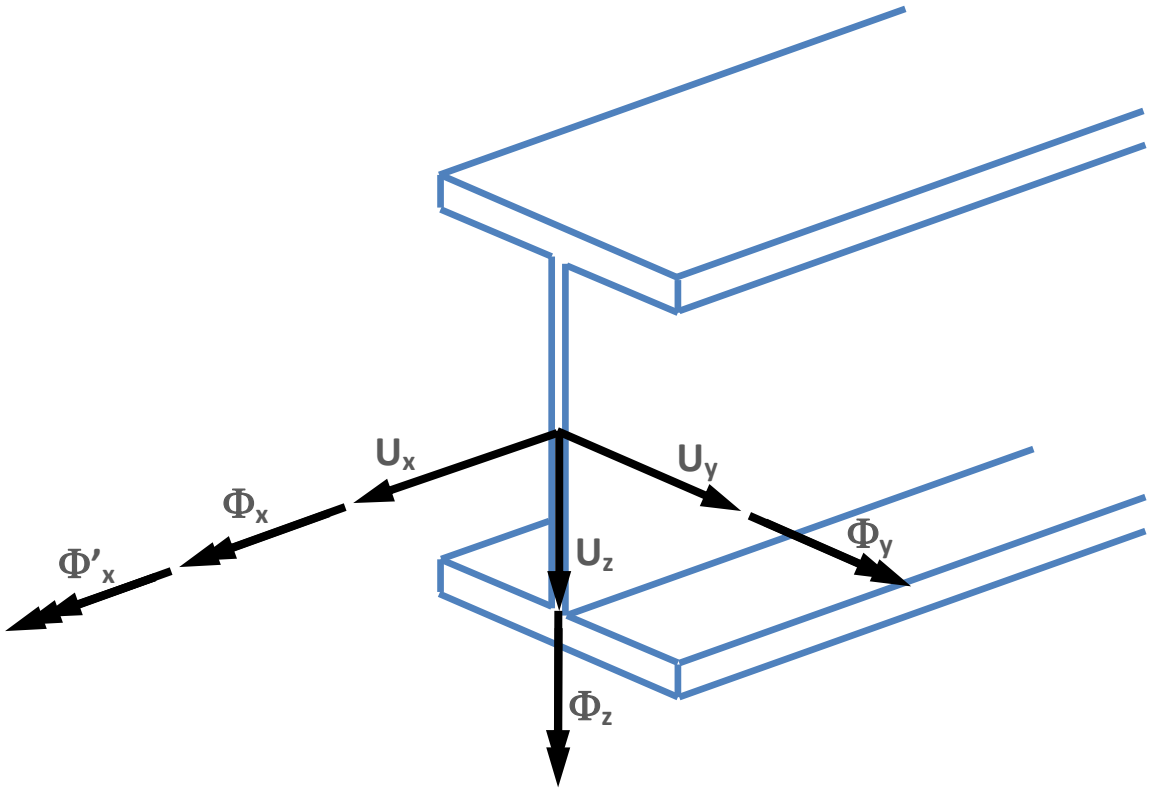
The subsequent sections do not intend to introduce the deep theory behind the applied finite elements – it can be found in the literature – only the most significant features are presented and explained which are important to know for the appropriate interpretation of the results.

7.2.2 LINE ELEMENTS

There are three options for line elements:

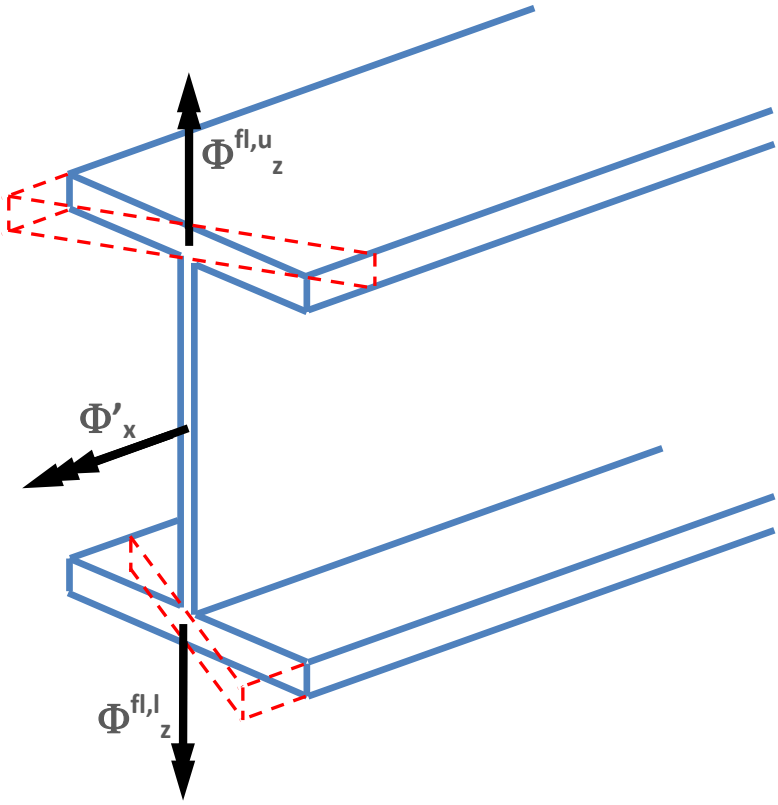
- a 6 degrees-of-freedom (6DOF) general beam-column element for the bars subjected to axial force, shear force, bending moment, torsion or any interaction of these
- a 7 degrees-of-freedom (7DOF) beam-column with warping element for the bars subjected same as 6DOF element + warping effect
- a tension-only (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 has 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.

7.3 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

Part of the model checks is performed on the user model (basic check), these are basic requirements for the normal performance of a model; 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 on can be selected and deleted from the diagnostics results table (in the middle of the right tables).

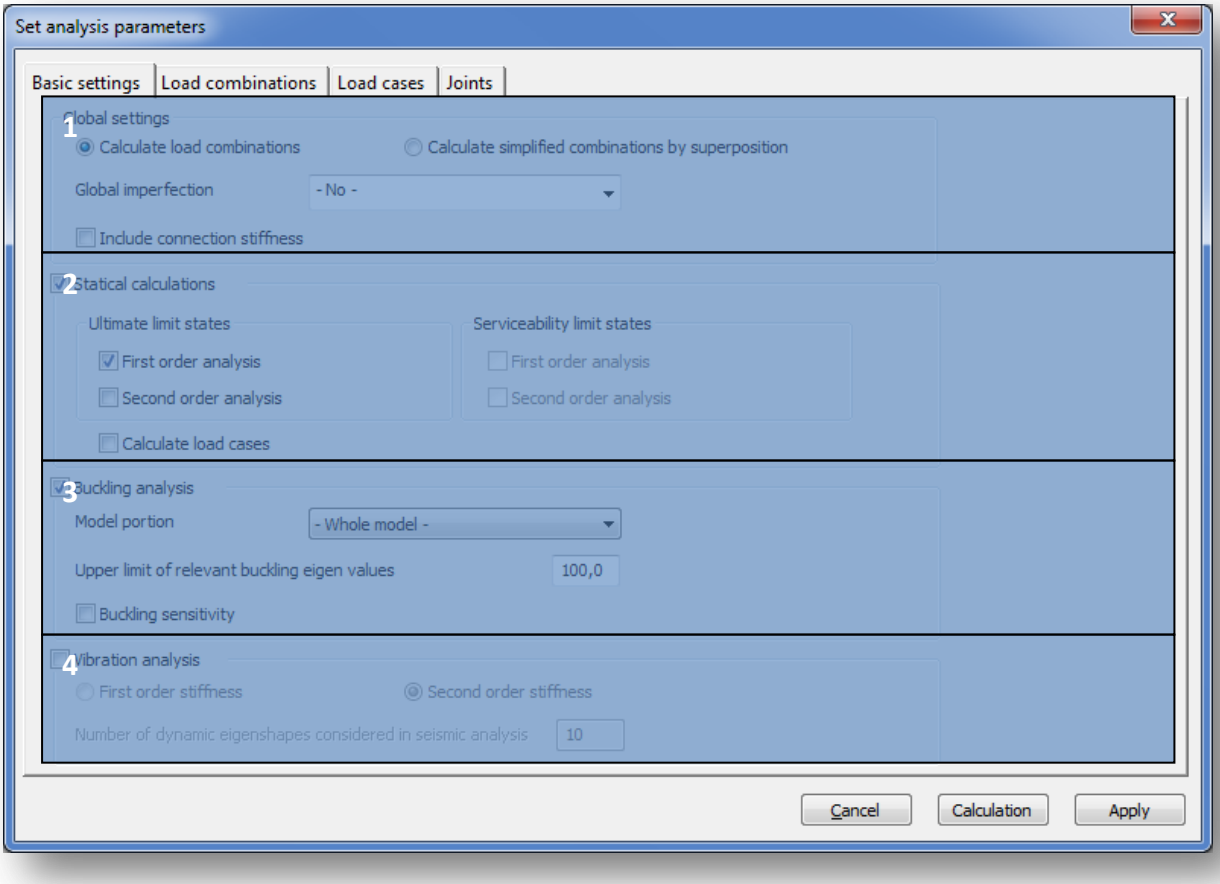
7.4 ANALYSIS SETTINGS

The required analysis types can be set on the ***ANALYSIS PARAMETERS*** panel. The analysis settings are separated in four tabs.

7.4.1 BASIC SETTINGS TAB

On the ***BASIC SETTINGS*** tab set analysis will be performed for all load combination.

For a better transparency the analysis setting are ordered in separated groups.



Settings of the Global settings group (#1) are applied for the whole model:

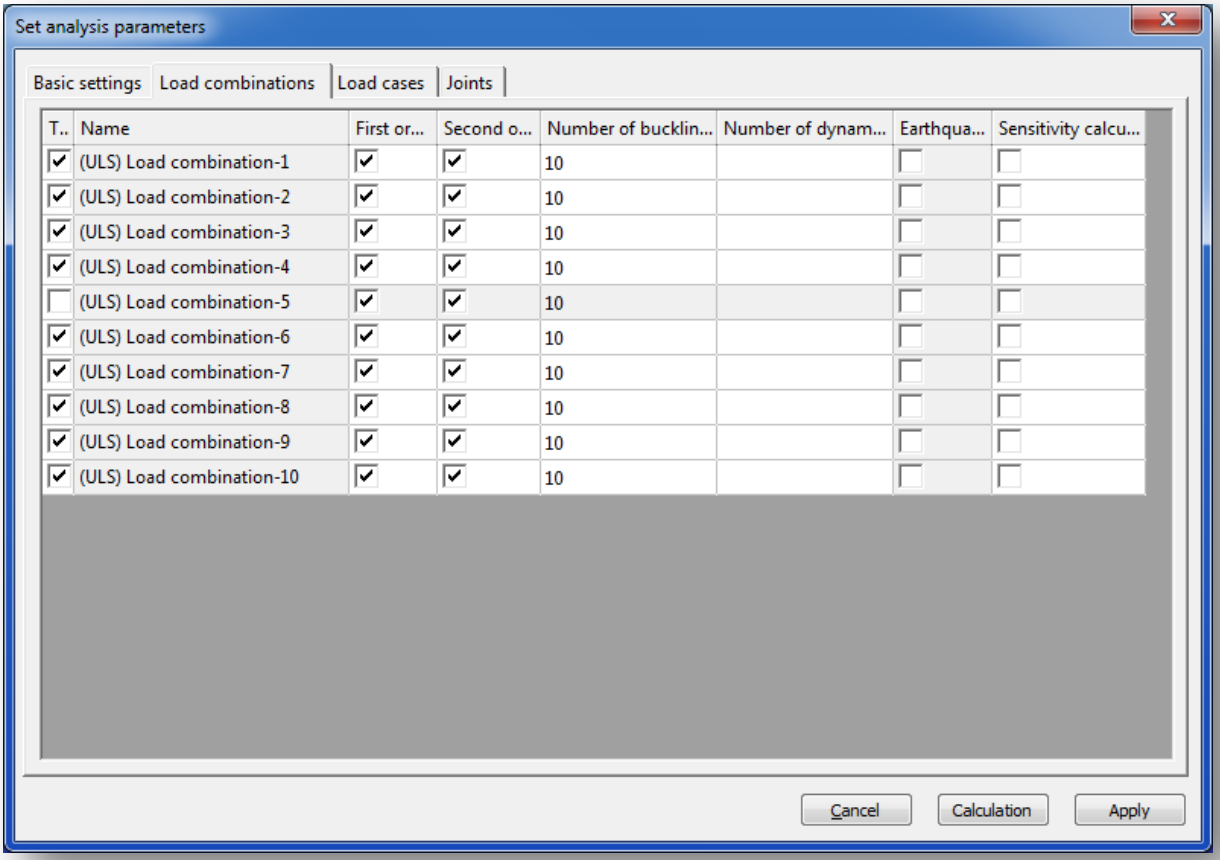
- For the calculations two types of load combination can be used: 1) previously created load combinations (see chapter 6.2) or 2) simplified load combinations by superposition of the load cases. In case of second option just the first order analysis type is available, but the running time of the calculation can be much more shorter instead of the first option
- With the global imperfection dropdown menu previously created Global imperfection (see. chapter 6.5) can be set for the model. Only one global imperfection can be applied in the same time
- With the checkbox previously created and placed joints' stiffness can be used during the calculation

In the Statical calculation group (#2) the necessary analysis (first or second order) can be set separately for the ULS and the SLS load combinations. Here can be set the calculation of the load cases also with the last checkbox. Statical calculation can be turn on/off with the checkbox near the name of the group.

In the Buckling analysis group (#3) the parameters of the buckling analysis can be set. Buckling analysis can be turn on/off with the checkbox near the name of the group. If buckling analysis is turned on by default 10 eigenshapes will be calculated for each load cases. Number of the eigenshapes can be modified at the Load combinations tab. In the dropdown menu previously created model portion or whole model can be selected for buckling analysis. Buckling 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 les then the set. With the last checkbox the buckling sensitivity analysis can be turn on and off.

In the Vibration analysis group (#4) the parameters of the vibration analysis can be set. Type of the stiffness matrix (first or second order) can be chosen and in case of earthquake load combinations the number of dynamic eigenshapes can be given. Vibration analysis can be turn on/off with the checkbox near the name of the group. If vibration analysis is turned on by default 10 eigenshapes will be calculated for each load cases. Number of the eigenshapes can be modified at the Load combinations tab.

7.4.2 LOAD COMBINATIONS TAB



On the ***LOAD COMBINATIONS*** tab every analysis parameters and settings can be set for each load combinations uniquely.

Load combinations can be turn on/off with the checkboxes in the first column.

7.4.3 LOAD CASES TAB

On the ***LOAD CASES*** tab load cases can be turn on/off for calculation with the checkboxes in the first column.

7.4.4 JOINTS TAB

On the ***JOINT*** tab 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.

7.5 ANALYSIS TYPES

7.5.1 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:

1. Calculation of first order (initial) stiffness matrices ($\mathbf{K}^{el}_{s}$) of the finite elements in their local coordinate system
2. Compiling the global stiffness matrix ($\mathbf{K}_s$) and nodal force vector ($\mathbf{P}$) 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 ($\mathbf{U}$) – in the global coordinate system for the global model (see section ***7.2 FINITE ELEMENTS*** for the interpretation of nodal displacements and forces):

$$\mathbf{K}_s\mathbf{U} = \mathbf{P} \quad \rightarrow \quad \mathbf{U} = \mathbf{K}_s^{-1}\mathbf{P} \tag{1}$$

5. Calculation of the internal forces (and stresses) of the elements ($\mathbf{f}^{el}$) in their local coordinate system by transforming the global nodal displacements of the element into local system ($\mathbf{u}^{el}$) using their local stiffness matrices:

$$\mathbf{f}^{el} = \mathbf{K}^{el}_{s}\mathbf{u}^{el} \tag{2}$$

7.5.2 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 ($\mathbf{K}^{el}_{g}$) of the finite elements in their local coordinate system by the internal forces of the elements ($\mathbf{f}^{el}$)

3. Compiling the second order global stiffness matrix ($\mathbf{K_s + K_g}$) and nodal force vector (**P**) of the whole model by transforming the element stiffness matrices into the global coordinate system
4. 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 ***7.2 FINITE ELEMENTS*** for the interpretation of nodal displacements and forces):

$$\left(\mathbf{K}_s + \mathbf{K}_g\right)\mathbf{U} = \mathbf{P} \quad \rightarrow \quad \mathbf{U} = \left(\mathbf{K}_s + \mathbf{K}_g\right)^{-1}\mathbf{P} \tag{3}$$

6. Calculation of the internal forces (and stresses) of the elements ($\mathbf{f}^{el}$) in their local coordinate system by transforming the global nodal displacements of the element into local system ($\mathbf{u}^{el}$) using their local stiffness matrices:

$$\mathbf{f}^{el} = \mathbf{K}_s^{el}\mathbf{u}^{el}$$

7. If the difference between the new nodal displacements and the ones obtained earlier exceeds a certain limit repeat the calculations from step 2

7.5.3 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):

$$\mathbf{K}_g(\lambda\mathbf{f}) = \lambda\mathbf{K}_g(\mathbf{f}) \tag{4}$$

In this case the eigenvalue analysis can be written in the following form:

$$\left(\mathbf{K}_s + \lambda\mathbf{K}_g\right)\mathbf{U} = 0 \tag{5}$$

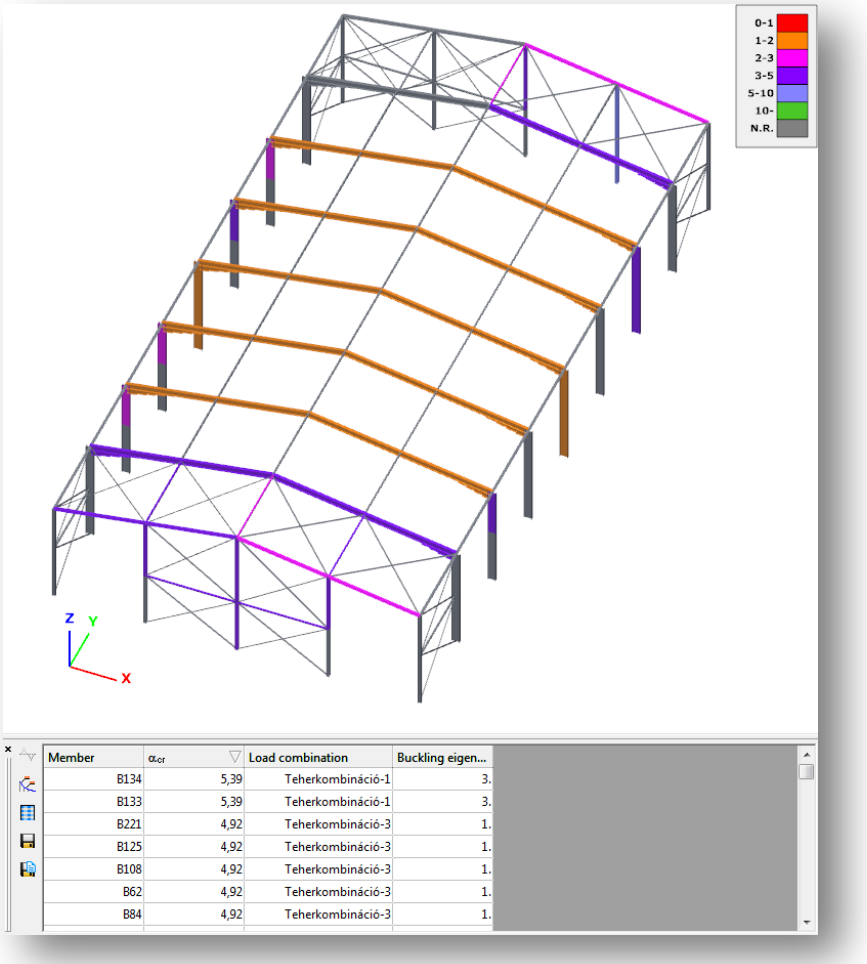
The solutions are certain critical load factors (buckling loads factor λ_{cr}^{i}) which make the second order stiffness matrix singular and corresponding displacements (buckling shapes $\mathbf{U}^{i}$).

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, lateral-torsional buckling and any interactions of these.

7.5.3.1 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.



Member	α_{cr}	Load combination	Buckling eigen...
B134	5,39	Teherkombináció-1	3.
B133	5,39	Teherkombináció-1	3.
B221	4,92	Teherkombináció-3	1.
B125	4,92	Teherkombináció-3	1.
B108	4,92	Teherkombináció-3	1.
B62	4,92	Teherkombináció-3	1.
B84	4,92	Teherkombináció-3	1.

7.5.4 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**)

$$\left(\left(\mathbf{K}_s + \mathbf{K}_g\right) + \omega^2 \mathbf{M}\right)\mathbf{U} = 0 \qquad (6)$$

The solutions are certain eigenfrequencies (ω^j) and corresponding displacements (free vibration shapes $\mathbf{U}^j$).

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.

7.6 ANALYSIS RESULTS

7.6.1 RESULT TYPES

For the two types of finite elements (line and surface elements) the following analysis results are available:

1. **Line 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
  - 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
  - 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
    - M_x – 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 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

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
  - 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

$$\alpha_n = \frac{1}{2} arctan\left(\frac{2n_{xy}}{n_x - n_y}\right) \quad -90° < \alpha_n \le +90°$$

- α_m – principal directions of the plate moments

$$\alpha_m = \frac{1}{2} arctan\left(\frac{2m_{xy}}{m_x - m_y}\right) \quad -90° < \alpha_m \le +90°$$

- 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
  - σ_1 – maximum principal stress

$$\sigma_1 = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \sigma_{xy}^2}$$

  - σ_2 – 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

$$\sigma_{HMH} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\sigma_{xy}^2}$$

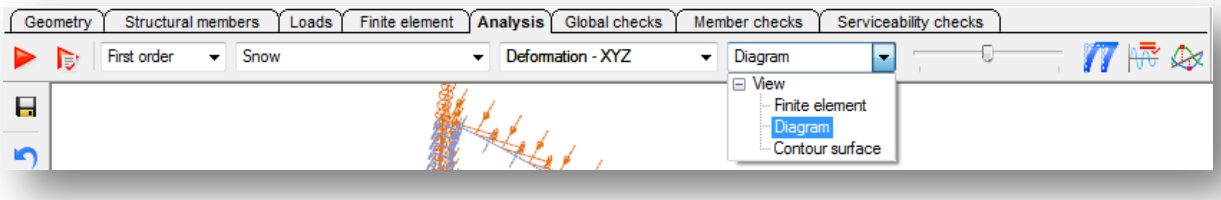
  - α_σ – principal directions of the stresses

$$\alpha_\sigma = \frac{1}{2} arctan\left(\frac{2\sigma_{xy}}{\sigma_x - \sigma_y}\right) \quad -90° < \alpha_\sigma \le +90°$$

- 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

7.6.2 VISUALIZATION OPTIONS

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):



- selection of analysis type (discussed in section ***7.4 ANALYSIS TYPES***)
- selection of load combination or load case
- selection of result type (discussed in section ***7.5.1 RESULT TYPES***)
- selection of type of view

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



User defined values: show the values belonging to the markers defined by the user (discussed more deeply in section ***7.5.3 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

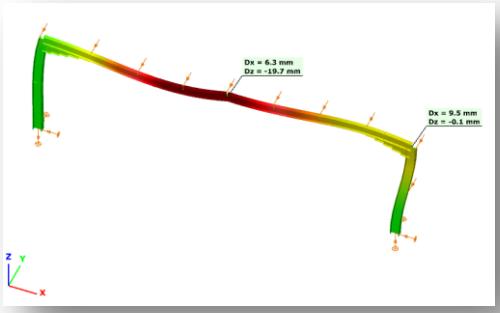
- force type results (internal forces, forces or stresses on surface): finite element node number, finite element number, force components
- reaction type results (reactions, equilibrium): finite element node number, reaction components.

7.6.3 RESULT MARKERS

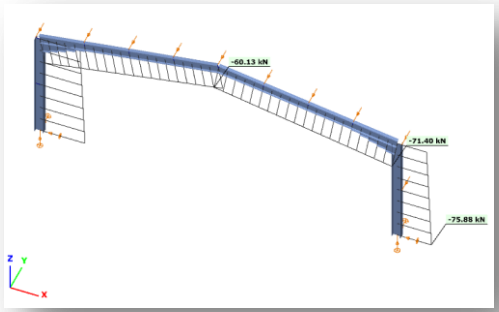
On the graphical interpretation of the results markers can be placed in order to show certain significant values. When moving the mouse along the structural elements (or more correctly the finite elements) the result markers continuously appear showing the actual values. These markers can be fixed by 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:

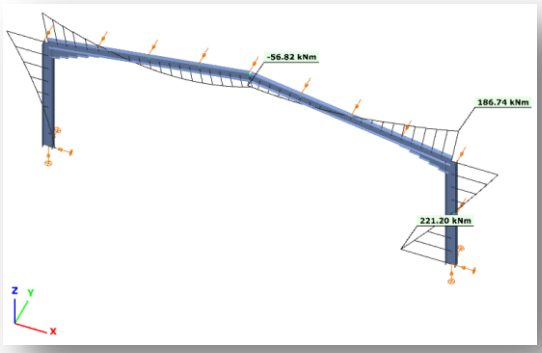
- Deformation

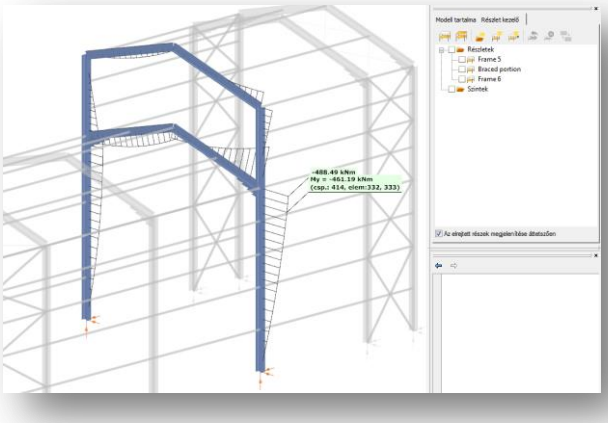


- Normal force



- 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
☑	3	9	-75,880	0,000	-78,680	0,000	221,200	0,000	0,000

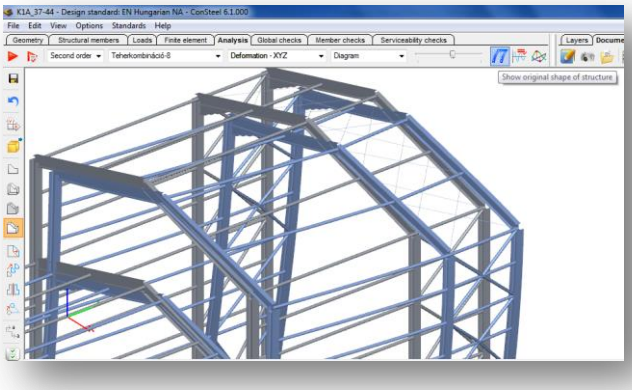
The markers can be deleted by:

- 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.

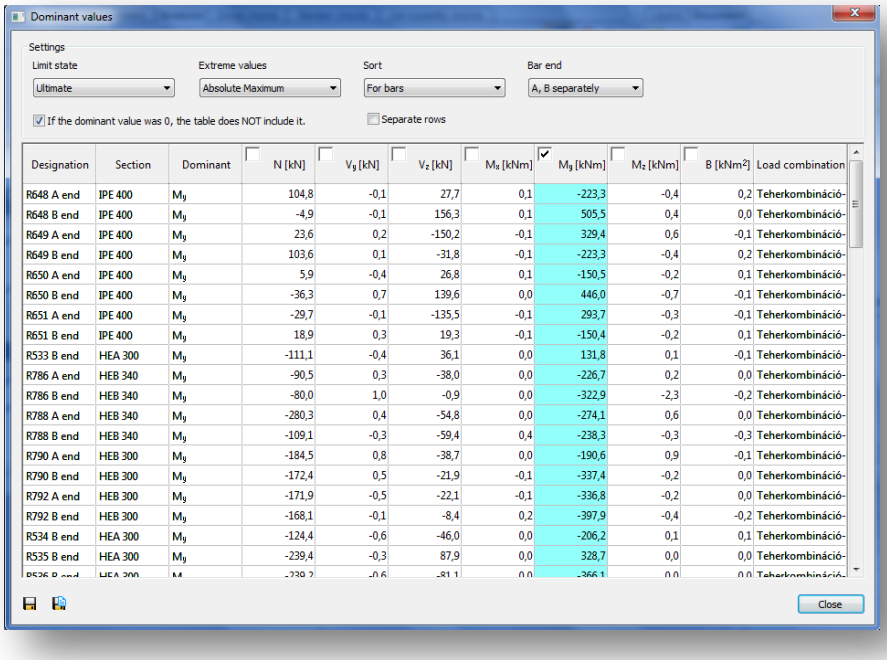
7.6.4 SHOW ORIGINAL SHAPE OF STRUCTURE

SHOW ORIGINAL SHAPE OF STRUCTURE function  will show the original shape of structure and the deformed structure at the same time:



7.6.5 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:



Designation	Section	Dominant	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]	B [kNm²]	Load combination
R648 A end	IPE 400	M_y	104,8	-0,1	27,7	0,1	-223,3	-0,4	0,2	Teherkombináció-
R648 B end	IPE 400	M_y	-4,9	-0,1	156,3	0,1	505,5	0,4	0,0	Teherkombináció-
R649 A end	IPE 400	M_y	23,6	0,2	-150,2	-0,1	329,4	0,6	-0,1	Teherkombináció-
R649 B end	IPE 400	M_y	103,6	0,1	-31,8	-0,1	-223,3	-0,4	0,2	Teherkombináció-
R650 A end	IPE 400	M_y	5,9	-0,4	26,8	0,1	-150,5	-0,2	0,1	Teherkombináció-
R650 B end	IPE 400	M_y	-36,3	0,7	139,6	0,0	446,0	-0,7	-0,1	Teherkombináció-
R651 A end	IPE 400	M_y	-29,7	-0,1	-135,5	-0,1	293,7	-0,3	-0,1	Teherkombináció-
R651 B end	IPE 400	M_y	18,9	0,3	19,3	-0,1	-150,4	-0,2	0,1	Teherkombináció-
R533 B end	HEA 300	M_y	-111,1	-0,4	36,1	0,0	131,8	0,1	-0,1	Teherkombináció-
R786 A end	HEB 340	M_y	-90,5	0,3	-38,0	0,0	-226,7	0,2	0,0	Teherkombináció-
R786 B end	HEB 340	M_y	-80,0	1,0	-0,9	0,0	-322,9	-2,3	-0,2	Teherkombináció-
R788 A end	HEB 340	M_y	-280,3	0,4	-54,8	0,0	-274,1	0,6	0,0	Teherkombináció-
R788 B end	HEB 340	M_y	-109,1	-0,3	-59,4	0,4	-238,3	-0,3	-0,3	Teherkombináció-
R790 A end	HEB 300	M_y	-184,5	0,8	-38,7	0,0	-190,6	0,9	-0,1	Teherkombináció-
R790 B end	HEB 300	M_y	-172,4	0,5	-21,9	-0,1	-337,4	-0,2	0,0	Teherkombináció-
R792 A end	HEB 300	M_y	-171,9	-0,5	-22,1	-0,1	-336,8	-0,2	0,0	Teherkombináció-
R792 B end	HEB 300	M_y	-168,1	-0,1	-8,4	0,2	-397,9	-0,4	-0,2	Teherkombináció-
R534 B end	HEA 300	M_y	-124,4	-0,6	-46,0	0,0	-206,2	0,1	0,1	Teherkombináció-
R535 B end	HEA 300	M_y	-239,4	-0,3	87,9	0,0	328,7	0,0	0,0	Teherkombináció-

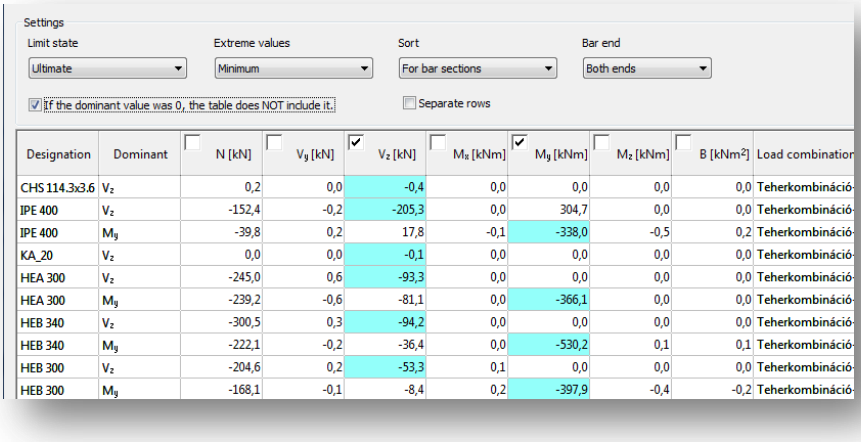
Limit state: selection of appropriate load combinations belonging to resistance or serviceability limit states (the deformations are automatically appear for the serviceability limit state, while the internal forces and reactions for the resistance limit state)

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)

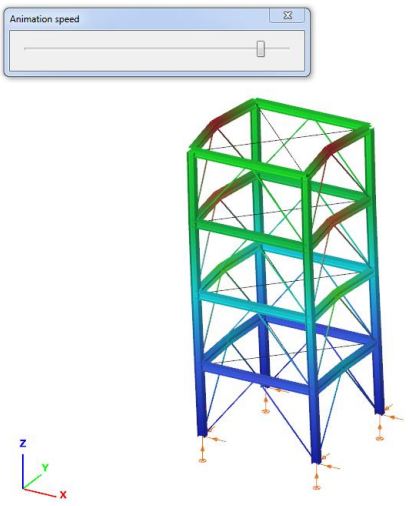
Single or multiple component selection is possible, the dominant values are always highlighted by colored cells, the 'Dominant' column contains the dominant components.



Designation	Dominant	N [kN]	V_y [kN]	V_z [kN]	M_x [kNm]	M_y [kNm]	M_z [kNm]	B [kNm²]	Load combination
CHS 114.3x3.6	V_z	0,2	0,0	-0,4	0,0	0,0	0,0	0,0	Teherkombináció
IPE 400	V_z	-152,4	-0,2	-205,3	0,0	304,7	0,0	0,0	Teherkombináció
IPE 400	M_y	-39,8	0,2	17,8	-0,1	-338,0	-0,5	0,2	Teherkombináció
KA_20	V_z	0,0	0,0	-0,1	0,0	0,0	0,0	0,0	Teherkombináció
HEA 300	V_z	-245,0	0,6	-93,3	0,0	0,0	0,0	0,0	Teherkombináció
HEA 300	M_y	-239,2	-0,6	-81,1	0,0	-366,1	0,0	0,0	Teherkombináció
HEB 340	V_z	-300,5	0,3	-94,2	0,0	0,0	0,0	0,0	Teherkombináció
HEB 340	M_y	-222,1	-0,2	-36,4	0,0	-530,2	0,1	0,1	Teherkombináció
HEB 300	V_z	-204,6	0,2	-53,3	0,1	0,0	0,0	0,0	Teherkombináció
HEB 300	M_y	-168,1	-0,1	-8,4	0,2	-397,9	-0,4	-0,2	Teherkombináció

7.6.6 ANIMATION OF DYNAMIC SHAPES

In case of vibration analysis, the animation of the dynamic shapes can be reached to click on the  icon. Animation spees can be increased and decreased with the slide bar.



8 STANDARD DESIGN

The design functionalities are placed on three tabs: the ***GLOBAL CHECKS***, ***MEMBER CHECKS*** and ***SERVICEABILITY CHECKS*** tab.

8.1 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

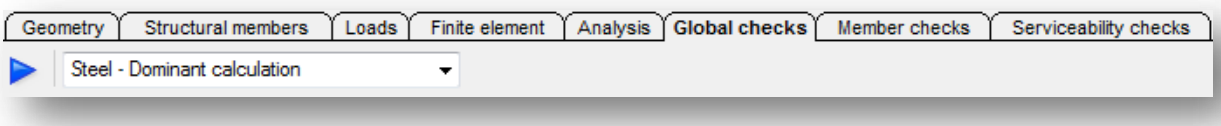
Several National Annexes are implemented and user annex can also be defined and saved.

8.2 STEEL DESIGN

Steel design related functions can be found on the ***GLOBAL CHECKS*** and the ***MEMBER CHECKS*** tabs.

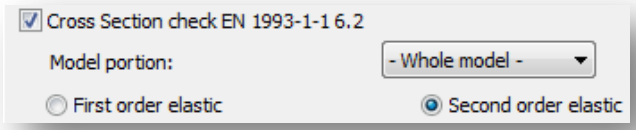
8.2.1 GLOBAL CHECKS

8.2.1.1 BASICS



Cross-section and global buckling check (according to the Section 6.3.4 (General method) of EN 1993-1-1) can be evaluated on the ***GLOBAL CHECKS*** tab.

8.2.1.2 CROSS SECTION CHECK



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).

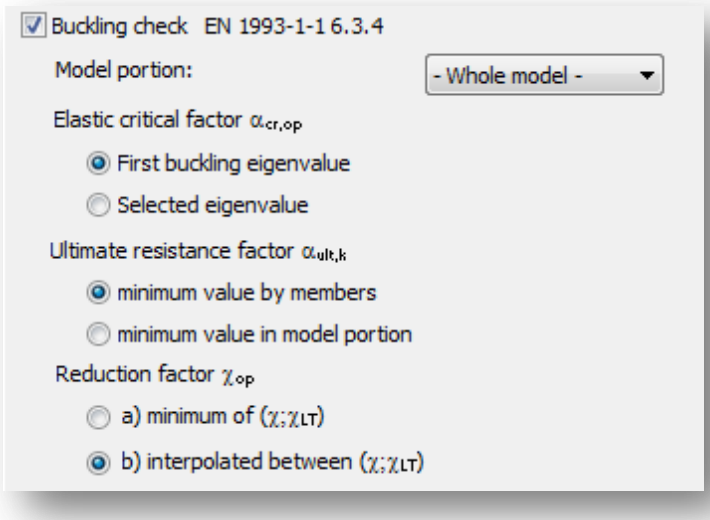
Check can be performed for the whole model or partial model.

8.2.1.3 BUCKLING CHECK

Buckling check can be evaluated for steel structures according to the EN 1993-1-1 6.3.4.

$$\frac{\chi_{op}\alpha_{ult,k}}{\gamma_{M1}} \geq 1{,}0$$

To run the buckling check, it is required to have buckling analysis results.



For the buckling check the following parameters can be set:

- Elastic critical factor $\alpha_{cr,op}$: eigenvalue of the buckling analysis. It can be the first or the selected eigenvalue according to the design situation. 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.

$$\bar{\lambda}_{op} = \sqrt{\frac{\alpha_{ult,k}}{\alpha_{cr,op}}}$$

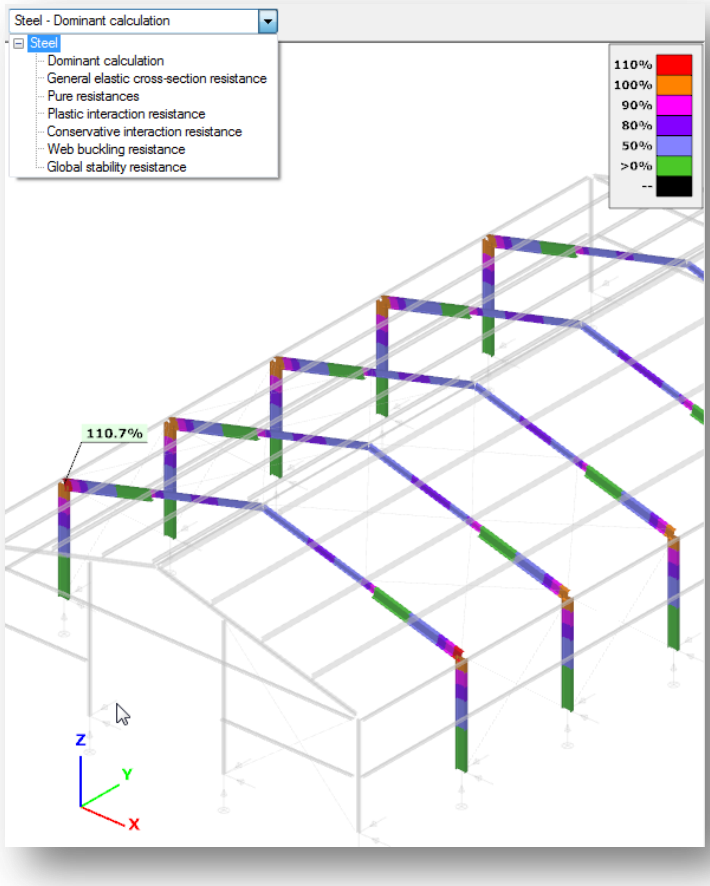
- Reduction factor χ_{op}: reduction factor for the λ_{op} slenderness. It can be determined from the minimum of (χ,χ_{LT}) or from the interpolated value of (χ,χ_{LT}).

Check can be performed for the whole model or partial model.

8.2.1.4 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.

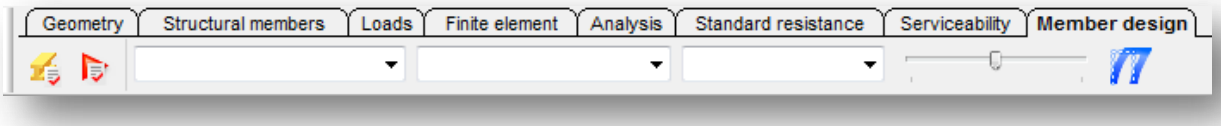


When moving the mouse along the structural elements the result markers continuously appear showing the actual utilization.

The detail of the results can be seen in the Section module (see section ***9 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.

8.2.2 MEMBER CHECKS

8.2.2.1 BASICS



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:

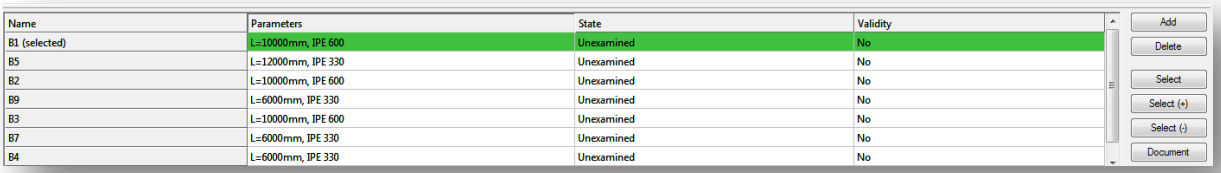
6.3.1 Uniform members in compression

6.3.2 Uniform members in bending

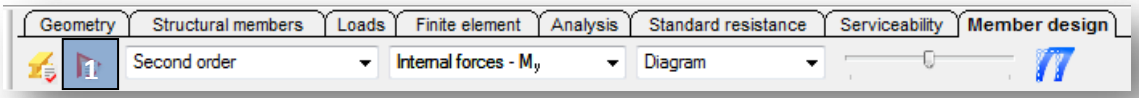
6.3.3 Uniform members in bending and axial compression

8.2.2.2 INDIVIDUAL MEMBER DESIGN

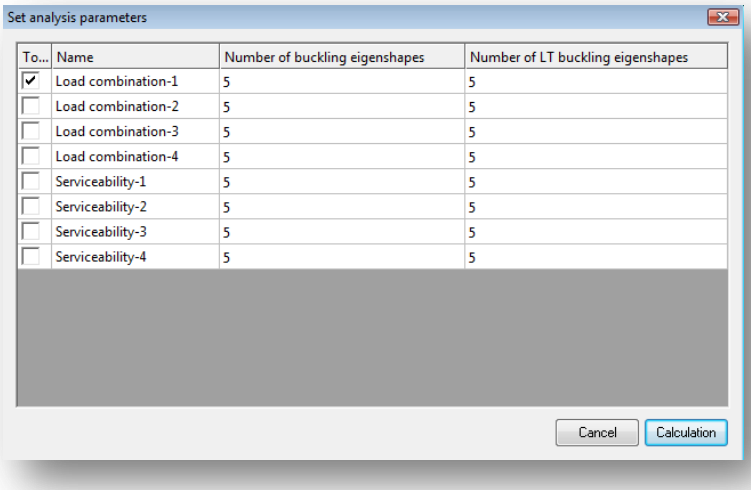
To run member design, first the members have to be selected and then added to the list. It can be performed by clicking on the ***ADD*** button. After that one member has to be chosen and ***SELECT*** button is clicked.



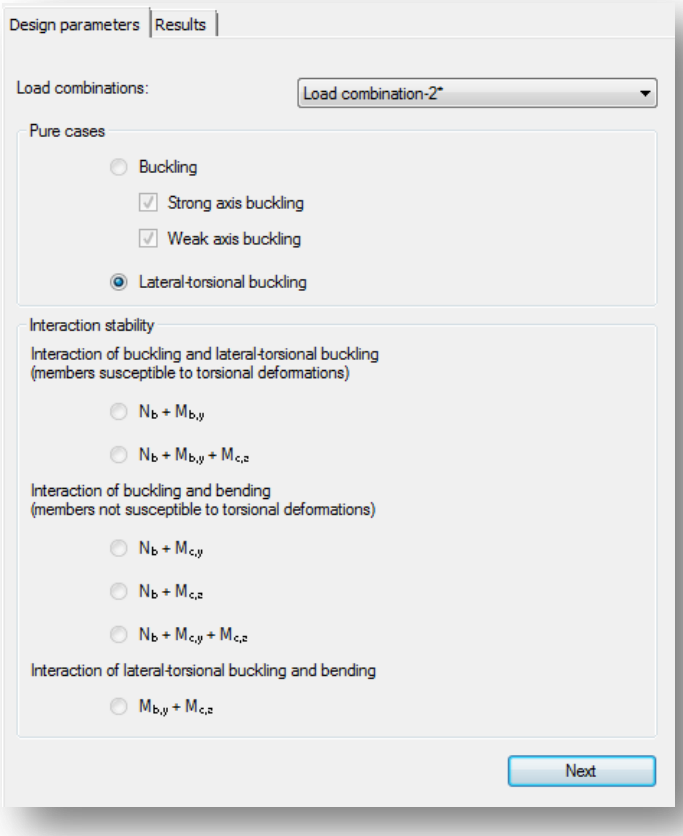
If a member is selected, the corresponding row highlighted with green and analysis results are automatically loaded. Member design can be run if there is analysis result (first and second order) and cross check for the member. Same like in the analysis the type of the result view can be set by three drop down menus.



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.

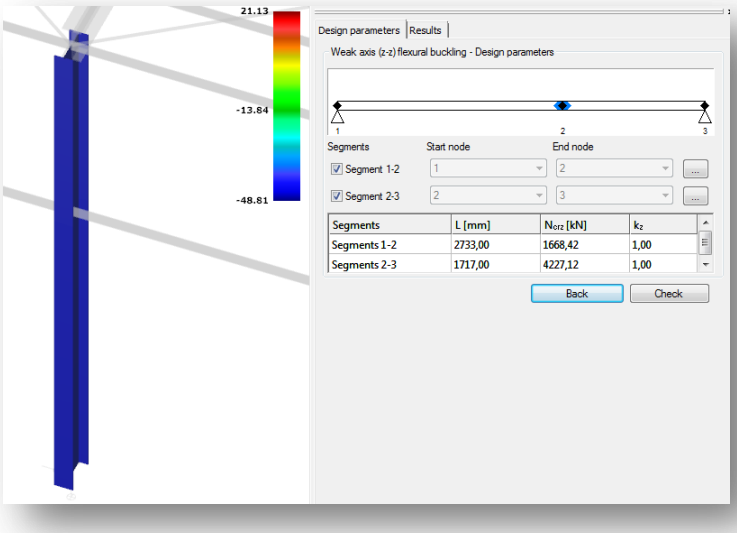


The next step is to choose the load combination and design type from the options: pure cases (strong axis buckling, weak axis buckling, lateral-torsional buckling) and interaction stability (interaction of buckling and lateral torsional buckling, interaction of buckling and bending, interaction of lateral-torsional buckling and bending). The dominant load combination is automatically selected and marked with a * symbol.

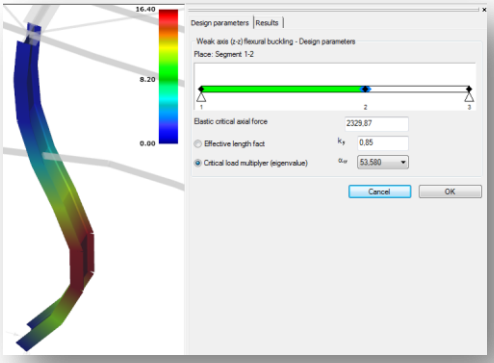


Depending on the member not all checks can be performed. If the normal force is negligible then strong or weak axis buckling design cannot be run.

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.

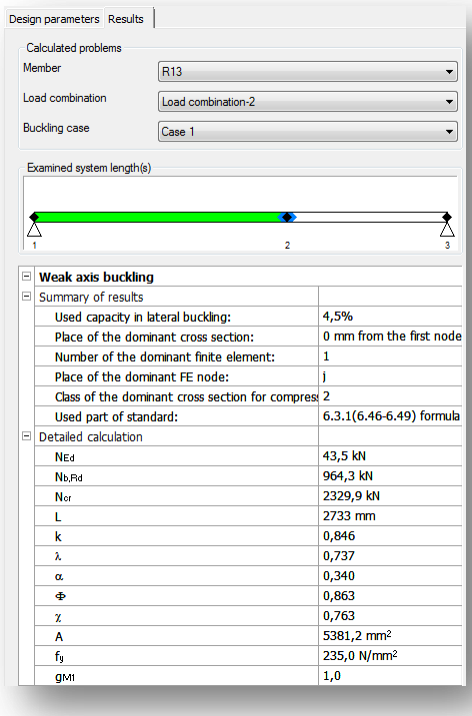


Among the supports the member is divided to segments. Segments can also be turned off in order not to calculate them during the analysis. The design parameters can be individually set for each segment by clicking on the three dots icon.



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.

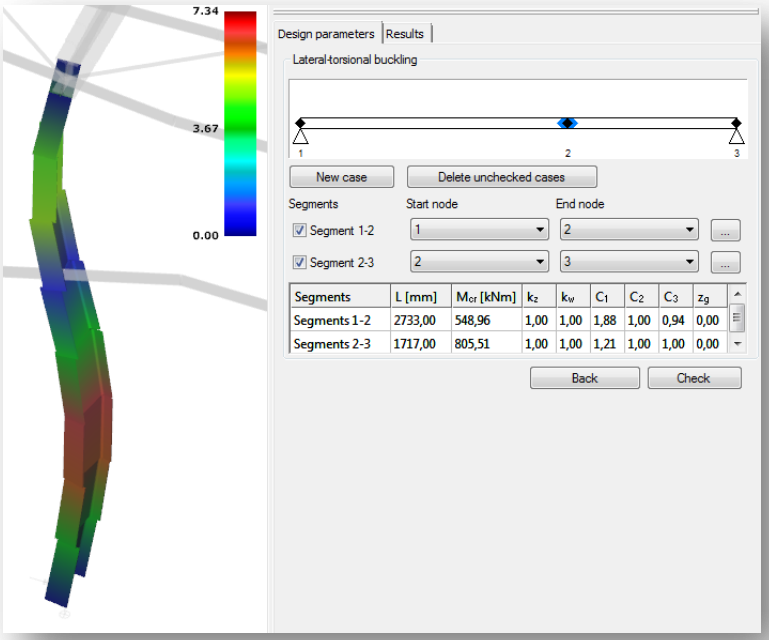
If all the design parameters has been properly set then design check can be performed by clicking on the ***CHECK*** button. Results will be displayed.



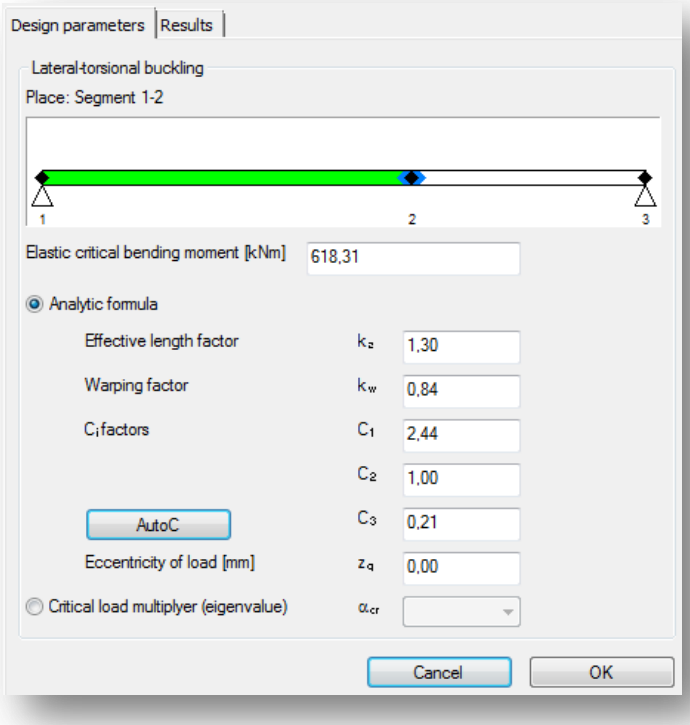
Weak axis buckling	
Summary of results	
Used capacity in lateral buckling:	4,5%
Place of the dominant cross section:	0 mm from the first node
Number of the dominant finite element:	1
Place of the dominant FE node:	j
Class of the dominant cross section for compres	2
Used part of standard:	6.3.1(6.46-6.49) formula
Detailed calculation	
N_{Ed}	43,5 kN
$N_{b,Rd}$	964,3 kN
N_{cr}	2329,9 kN
L	2733 mm
k	0,846
λ	0,737
α	0,340
Φ	0,863
χ	0,763
A	5381,2 mm²
f_y	235,0 N/mm²
γ_{M1}	1,0

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 icon the design parameters can be altered. The elastic critical bending moment can be calculated from the analytic formula or from the critical load 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.



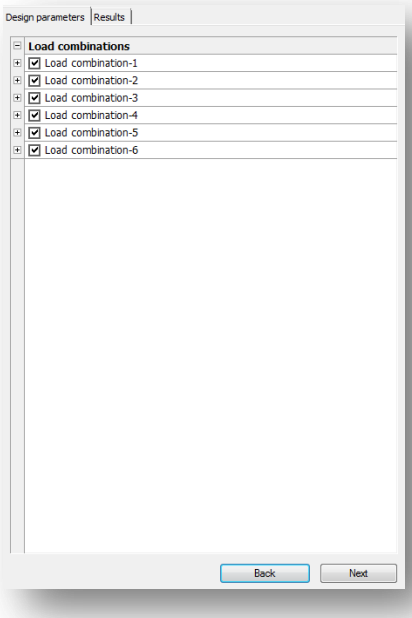
Interaction stability design goes through the steps of the pure cases.

After the check is made the state of the member in the list at the bottom changes to 'Examined' from 'Unexamined'.

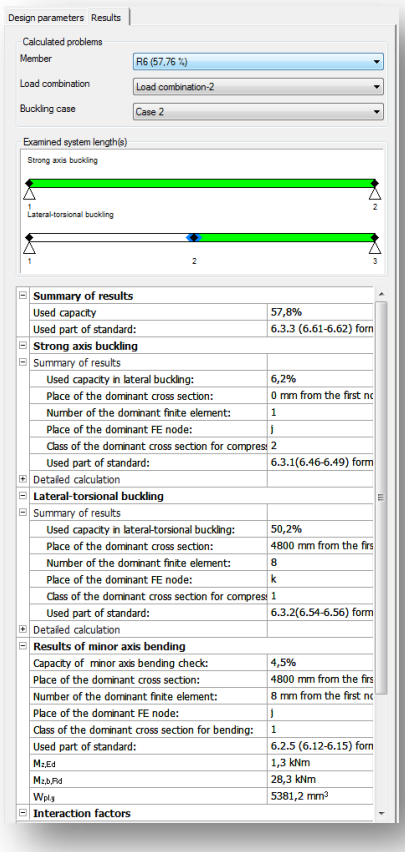
8.2.2.3 Member Group Design

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.



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.



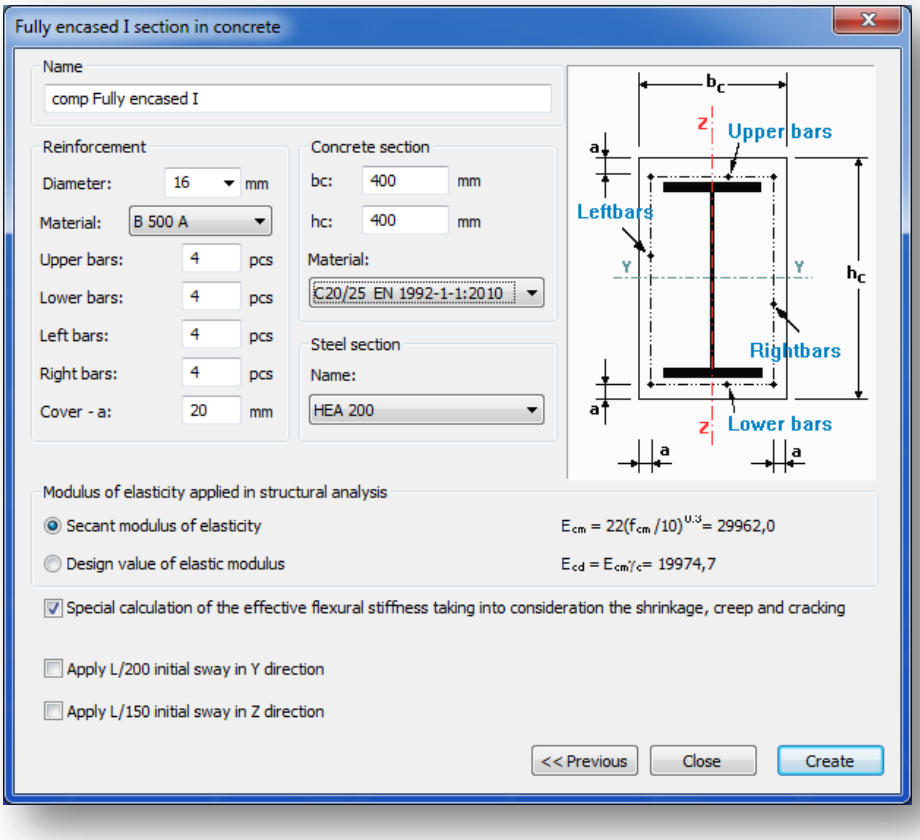
In the results tab the dominant member is automatically shown. The number next to the member number shows the capacity.

8.3 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.

8.3.1 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 Cross Section check 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.

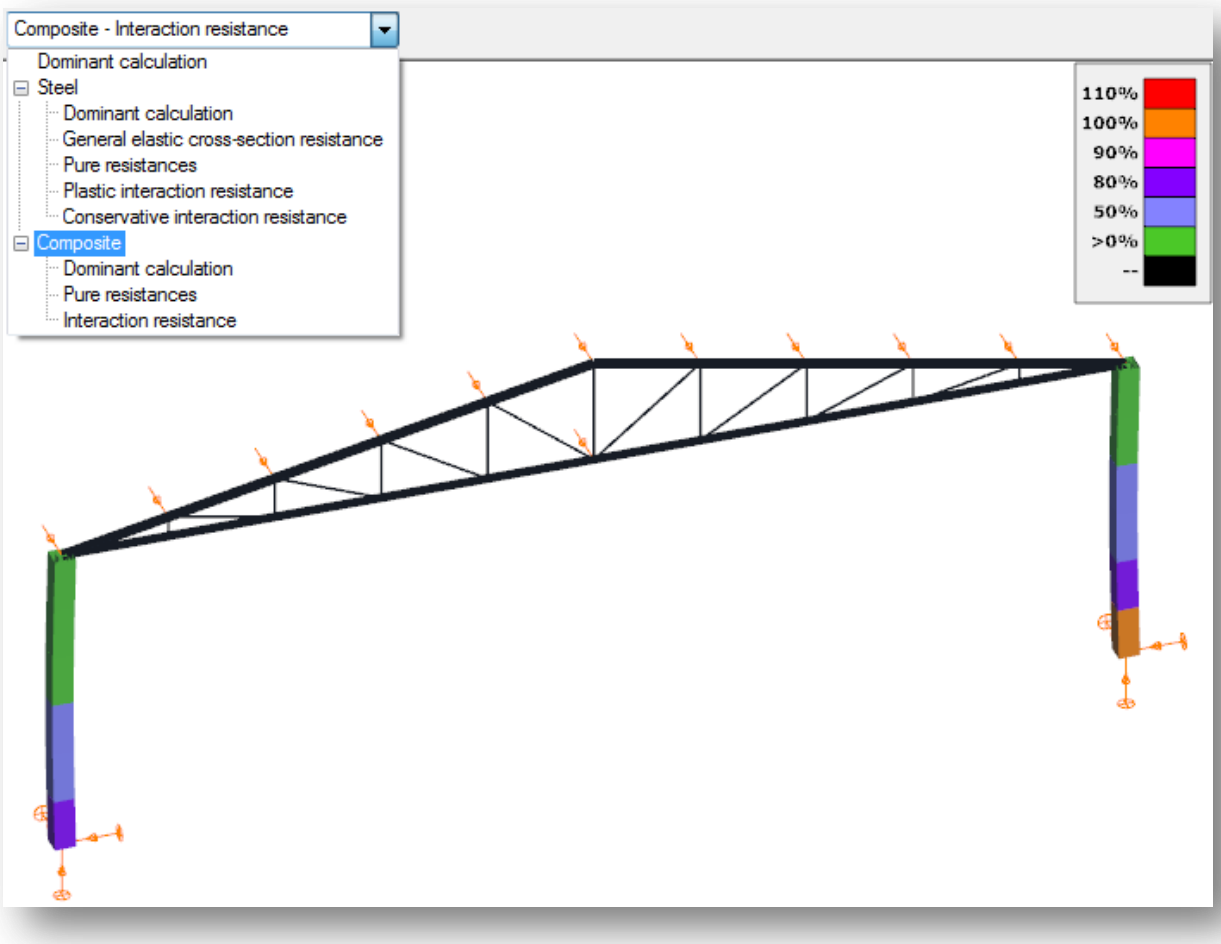


If the initial sways in the y and z directions were applied on the checked cross-section then the stability check of the composite columns can be neglected. Cross-section checks are enough.

8.3.1.1 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 section ***9 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.

8.3.2 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.

8.3.2.1 ANALYSIS MODEL

In the **analysis model** the calculation of the effective width of concrete flange is based on the method defined in 5.4.1.2. During the elastic analysis constant effective width is assumed over the whole length of each span (5.4.1.2. (4)). The spans are calculated according to the support positions along the beam which can be set in the design parameters window. Equivalent spans are calculated according to the method described in 5.4.2.1

In the analysis tab the analysis model is displayed.

8.3.2.2 DESIGN MODEL

For the **design model** there is a special algorithm which analyse the bending moment diagram of the beam to recognise the support positions and also calculates the lengths between points of zero bending moment (i.e. equivalent spans – L_e).

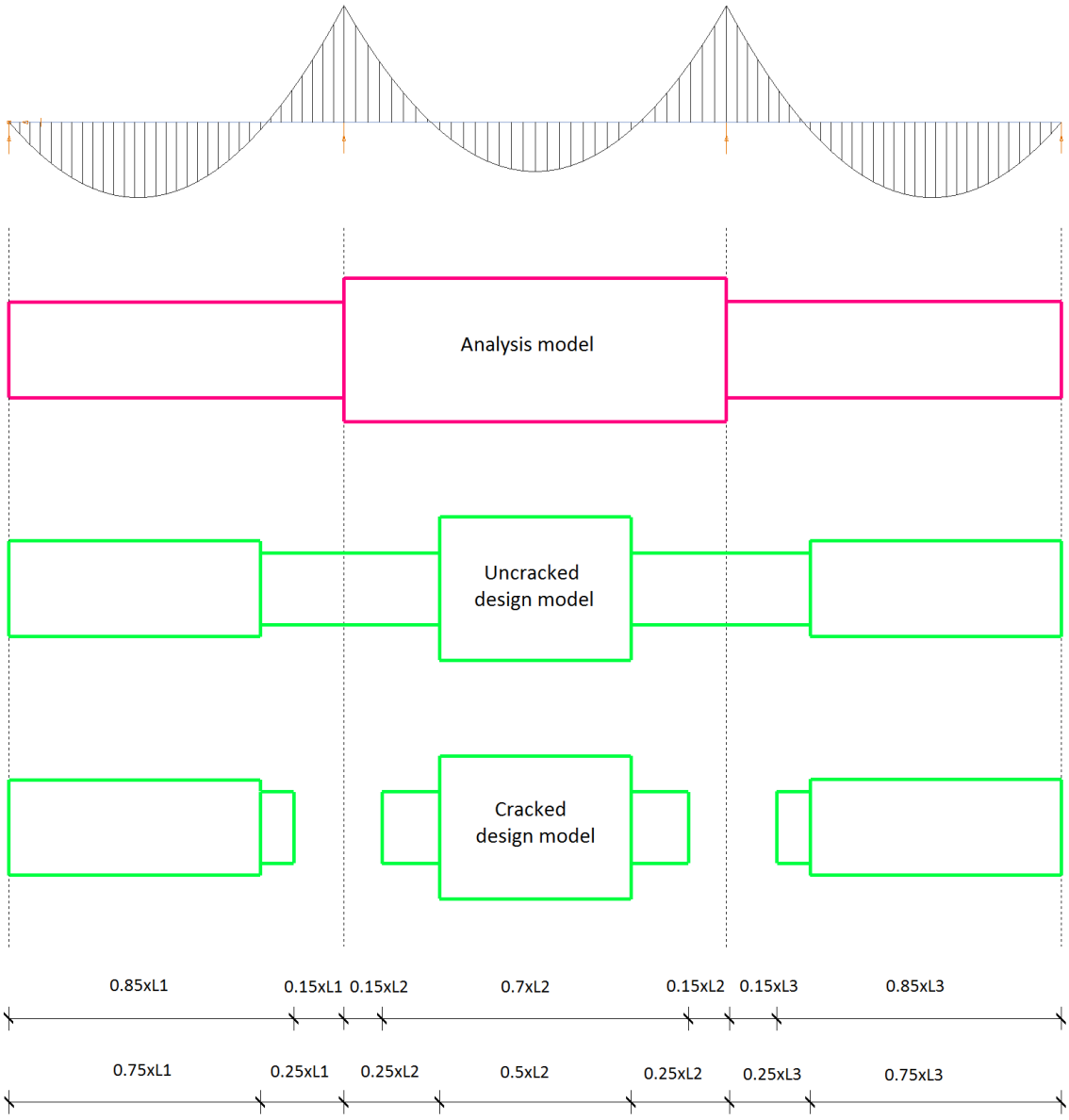


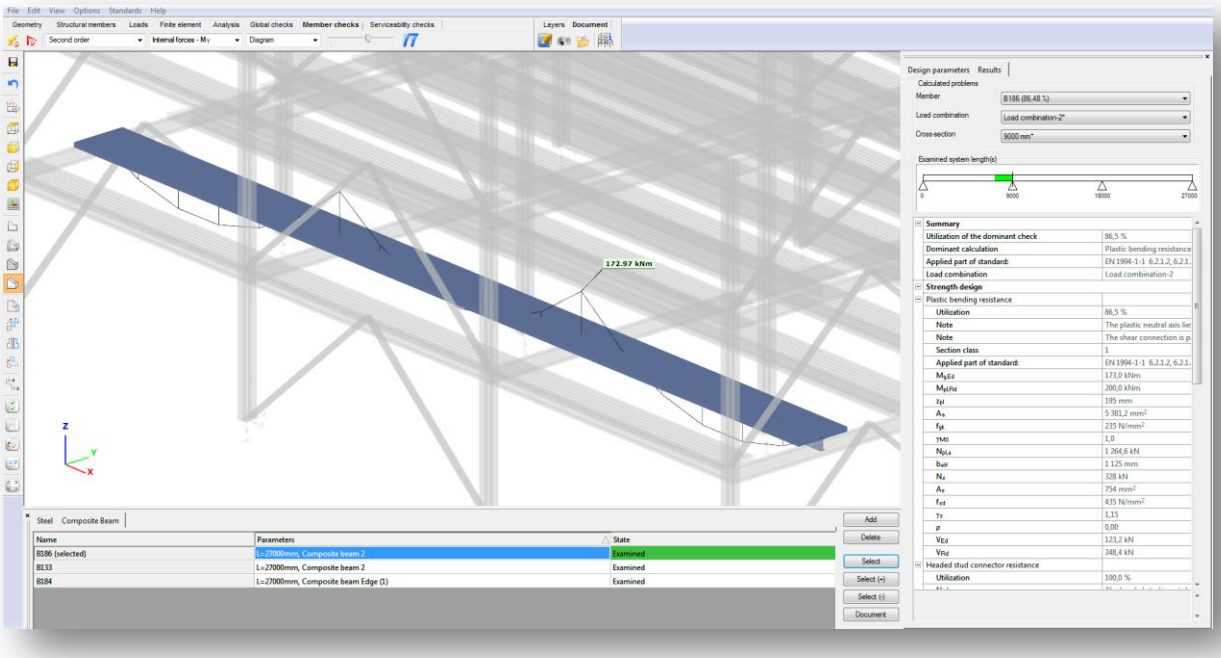
FIGURE: EFFECTIVE WIDTH IN ANALYSIS AND DESIGN MODELS

8.3.2.3 DESIGN OF SHEAR STUDS

If the 'Automatic number of shear studs' option is selected (by default it is turned on) the number of studs and the layout will be determined by an iteration algorithm which is based on the minimum number of shear studs that also meet the plastic bending capacity requirements (i.e. $M_{Pl,Rd} > M_{y,Ed}$). The utilization in this case is always 100% because the optimal number of studs are used. The green segment shows the segment on which the optimal number of studs and layout are calculated.

If the exact number of shear stud is added, the program checks whether it meets the code specified strength and spacing requirements. The number of shear studs must be defined for the whole composite beam and during the design process ConSteel calculates with the number of studs over the length under consideration assuming that the studs are placed uniformly.

The utilization of the shear studs is calculated by dividing the optimal number of the studs by the applied number of studs: Utilization= n_{opt} / n_{act}



8.3.2.4 ADDITIONAL COMMENTS ON COMPOSITE BEAM DESIGN

6.2.2.3 Shear buckling resistance

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.

6.2.2.4 Bending and vertical shear

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)

6.4. Lateral-torsional buckling of composite beams

Lateral-torsional buckling of composite beams is not included.

6.6.4.1 Sheeting with ribs parallel to the supporting beams

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.

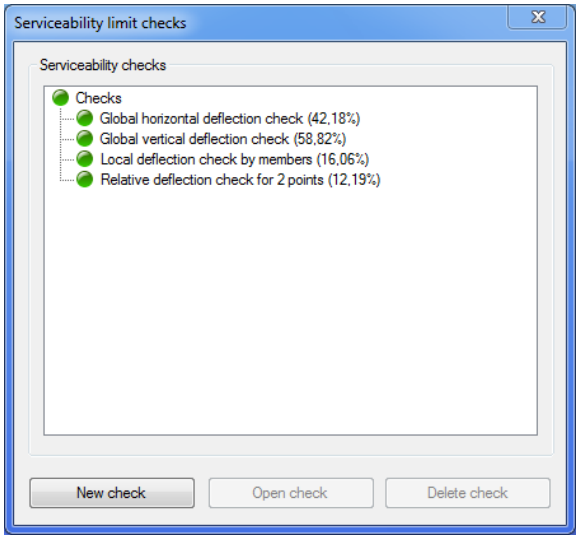
6.6.4.2 Sheeting with ribs perpendicular to the supporting beams

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/mm^2.

8.4 SERVICEABILITY CHECKS

8.4.1 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.

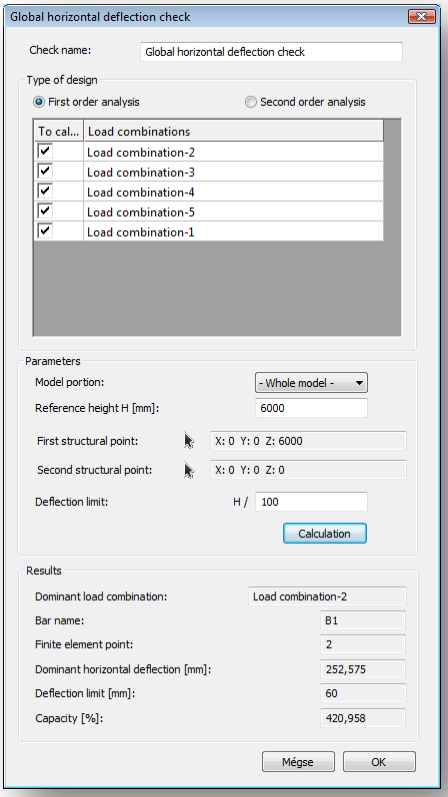


In order to create a new serviceability check ***NEW CHECK*** button is clicked. The following checks can be performed:

- Global horizontal deflection check
- Global vertical deflection check

- Local deflection check by members
- Relative deflection check for 2 points

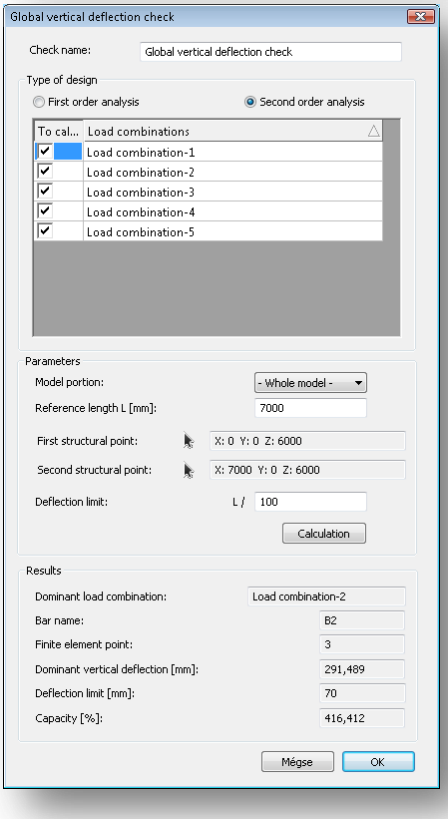
8.4.2 GLOBAL HORIZONTAL DEFLECTION CHECK



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.

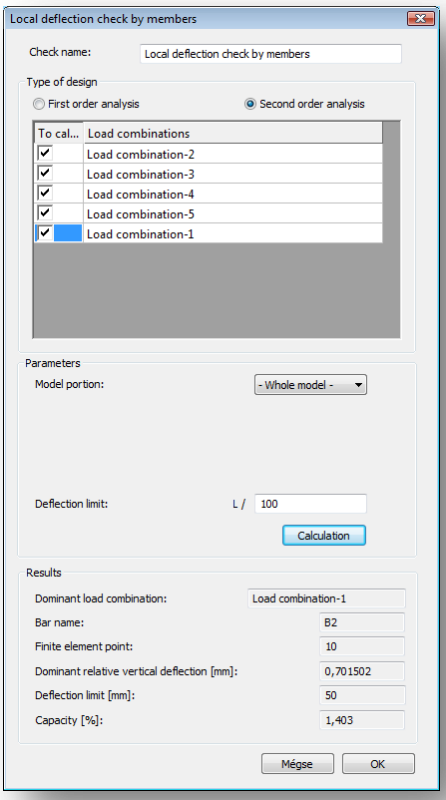
Reference height can be typed in or two points can be selected and program calculates the height distance between them. Check is done after clicking the ***CALCULATION*** button. The dominant bar is automatically selected in the model to make the interpretation easier.

8.4.3 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.

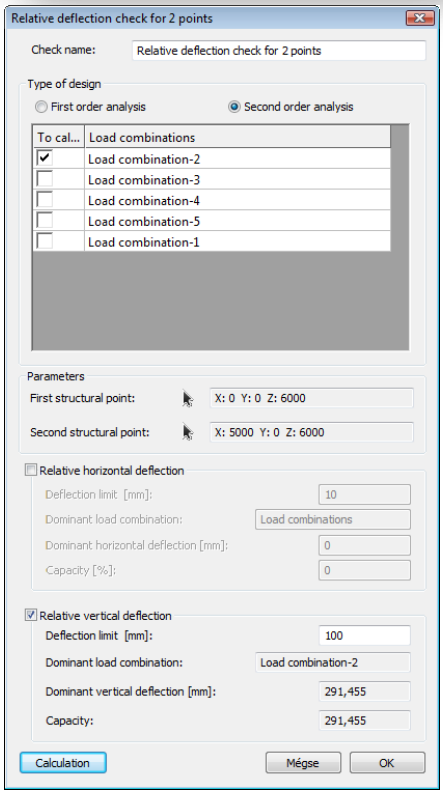
8.4.4 LOCAL DEFLECTION CHECK BY MEMBERS



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.

8.4.5 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.

9 SECTION MODULE

9.1 BASICS

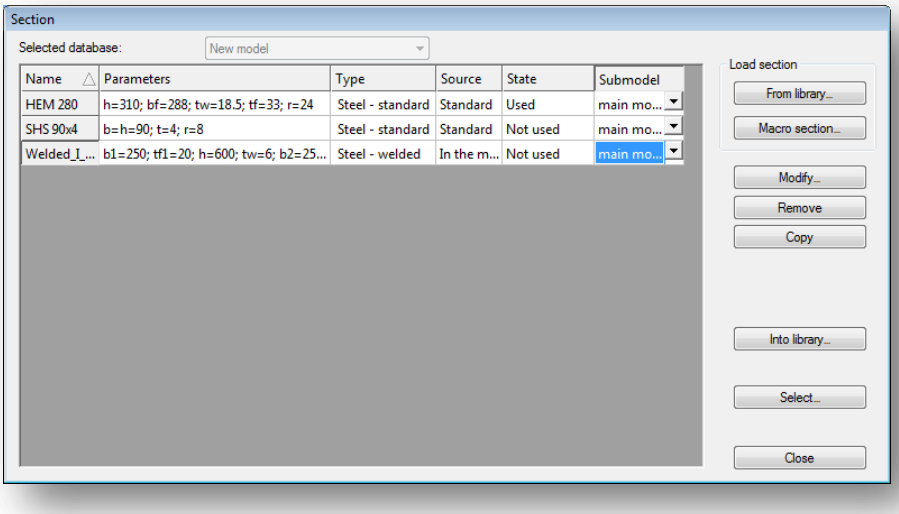
The wide functionality of the *ConSteel* connected with cross-sections suggested detaching a separate module: Section Module (***CSSECTION***). This module collects all the tools relate to the creation, property calculation, stress analysis and standard design of a cross-section. Two parallel modeling 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.



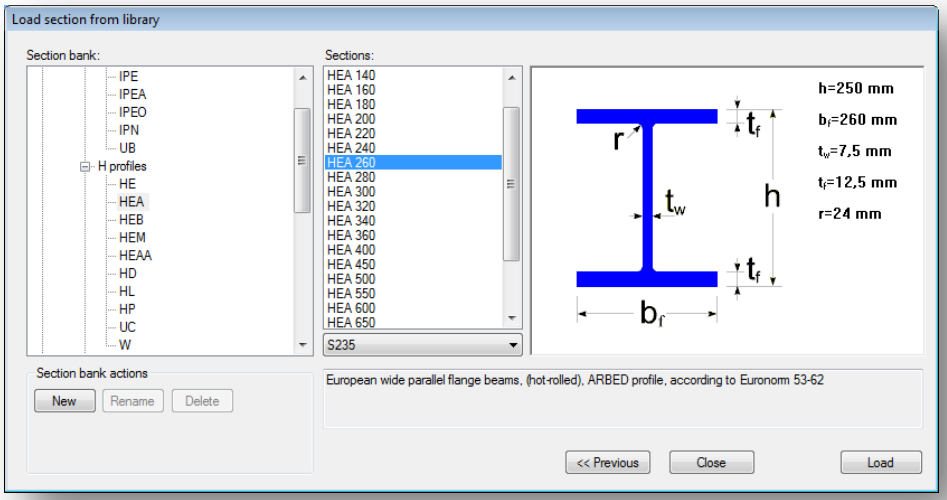
Section administration and Section module can be launched by clicking the icons shown above. These icons can be found in the ***STRUCTURAL MEMBERS*** tab.

9.2 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.



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.



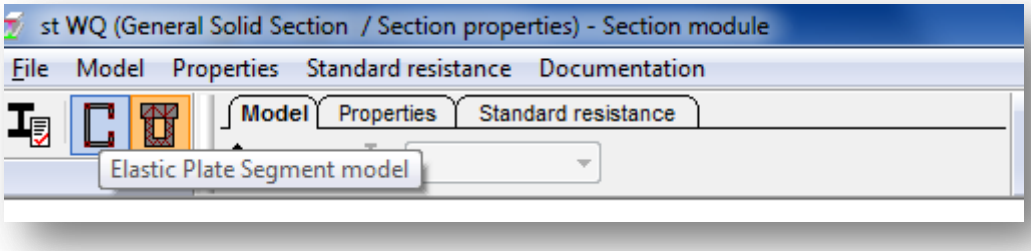
It is very important to know that the material grade of the member must be chosen when the section is loaded into the database.

So called macro sections can also be created in *ConSteel*. The following macro section types can be created:

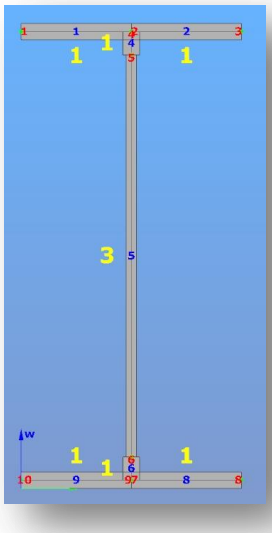
- Steel
  - Welded from plates
  - Cold formed or hot rolled
  - Compound
- Reinforced concrete
- Composite
- Buckling Restrained Brace

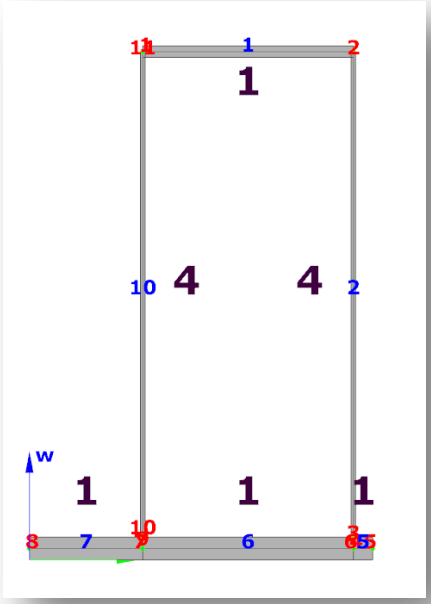
9.3 SECTION MODELLING

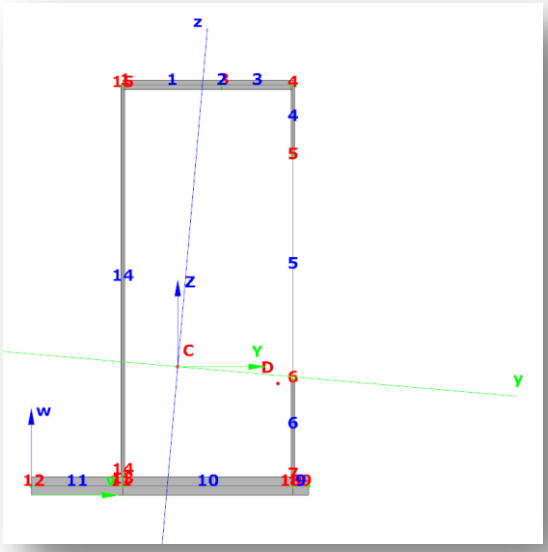
9.3.1 THE EPS MODEL

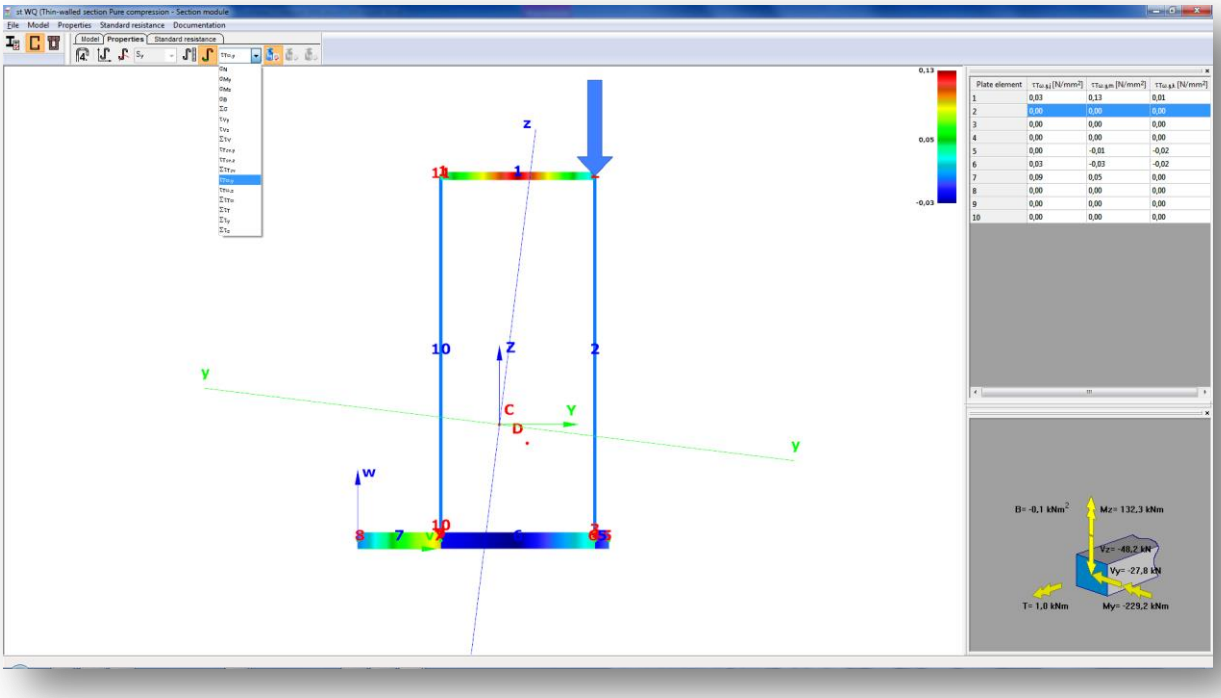


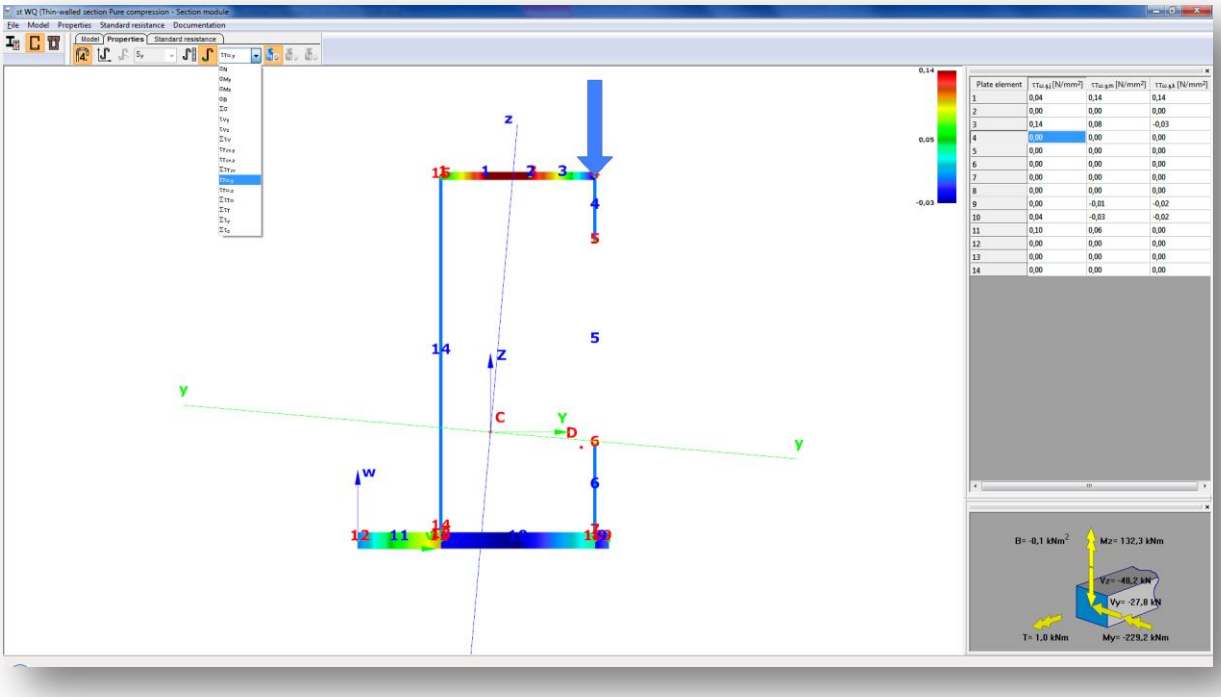
The Elastic Plate Segment (EPS) model is a thin-walled simplification of the usually steel profiles from welded, hot-rolled or cold-formed sections. The model is very easy and fast to calculate and has accurate results for thin-walled sections compared with the GSS model, which can be used and run for 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.



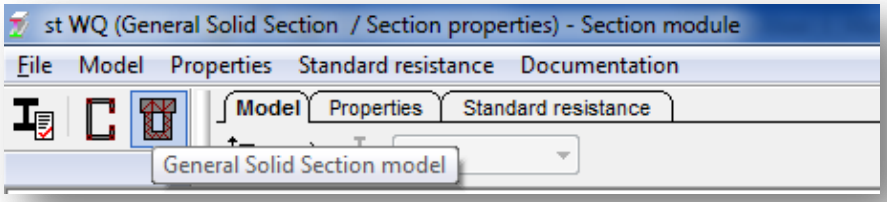

1
2
1
4
4
10
2
w
1
1
1
8
7
6
5


z
1
2
3
4
4
5
5
14
Z
C
Y
D
6
y
w
6
12
11
10

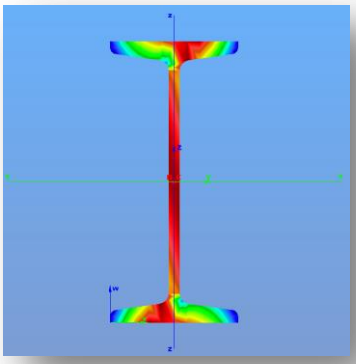




9.3.2 THE GSS MODEL



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.



9.4 SECTION TOOLS

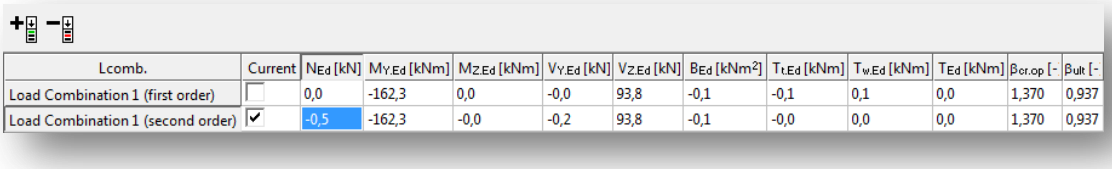
9.4.1 GEOMETRICAL CALCULATIONS

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.

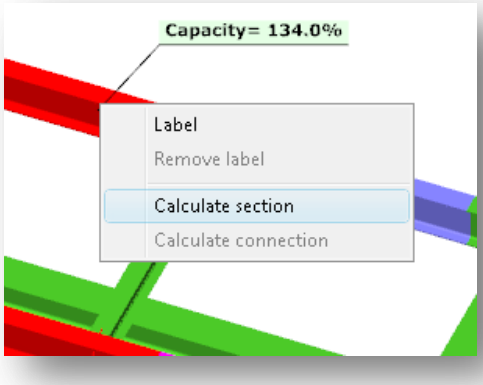
9.4.2 LOADS

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 .

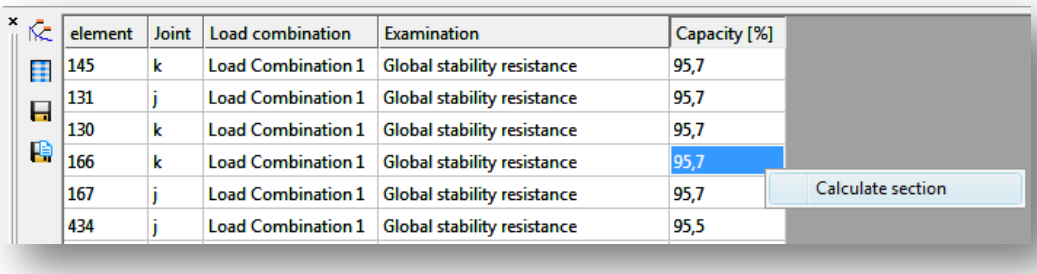


Lcomb.	Current	N_{Ed} [kN]	$M_{Y,Ed}$ [kNm]	$M_{Z,Ed}$ [kNm]	$V_{Y,Ed}$ [kN]	$V_{Z,Ed}$ [kN]	B_{Ed} [kNm²]	$T_{t,Ed}$ [kNm]	$T_{w,Ed}$ [kNm]	T_{Ed} [kNm]	$\beta_{cr,op}$ [-	β_{ult} [-
Load Combination 1 (first order)	☐	0,0	-162,3	0,0	-0,0	93,8	-0,1	-0,1	0,1	0,0	1,370	0,937
Load Combination 1 (second 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***.



element	Joint	Load combination	Examination	Capacity [%]
145	k	Load Combination 1	Global stability resistance	95,7
131	j	Load Combination 1	Global stability resistance	95,7
130	k	Load Combination 1	Global stability resistance	95,7
166	k	Load Combination 1	Global stability resistance	95,7
167	j	Load Combination 1	Global stability resistance	95,7
434	j	Load Combination 1	Global stability resistance	95,5

Calculate section

9.4.3 STRESSES

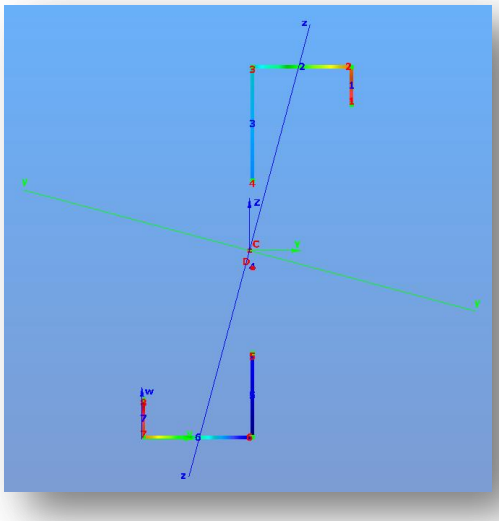
Stresses can be checked by selecting the ***PROPERTIES*** tab and clicking on the ***STRESSES*** icon . Stresses can be viewed both in the EPS and the GSS model.

The following stresses are calculated by ***CSSECTION***:

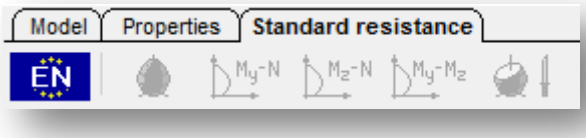
Stress Symbol	Stress name
σ_N	Normal stress due to axial force
σ_{My}	Normal stress due strong axis bending
σ_{Mz}	Normal stress due to weak axis bending
σ_B	Normal stress due to bimoment
$\Sigma\sigma$	Resultant normal stress
τ_{Vy}	Shear stress due to strong axis shear force
τ_{Vz}	Shear stress due to weak axis shear force
$\Sigma\tau_V$	Resultant shear stress due to shear forces
$\tau_{Tsv.y}$	Strong axis shear stress due to St. Venant torsion
$\tau_{Tsv.z}$	Weak axis shear stress due to St. Venant torsion
$\Sigma\ \tau_{Tsv}$	Resultant shear stress due to St. Venant torsion
$\tau_{T\omega.y}$	Strong axis shear stress due to warping torsion
$\tau_{T\omega.z}$	Weak axis shear stress due to warping torsion
$\Sigma\ \tau_{T\omega}$	Resultant shear stress due to warping torsion
$\Sigma\ \tau_T$	Resultant shear stress due to torsion
$\Sigma\ \tau_y$	Strong axis resultant shear stress
$\Sigma\ \tau_z$	Weak axis resultant shear stress

9.4.4 EFFECTIVE CROSS SECTION

If the first icon  is clicked and EPS model is selected then effective section take into consideration.



9.4.5 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.

9.4.5.1 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.

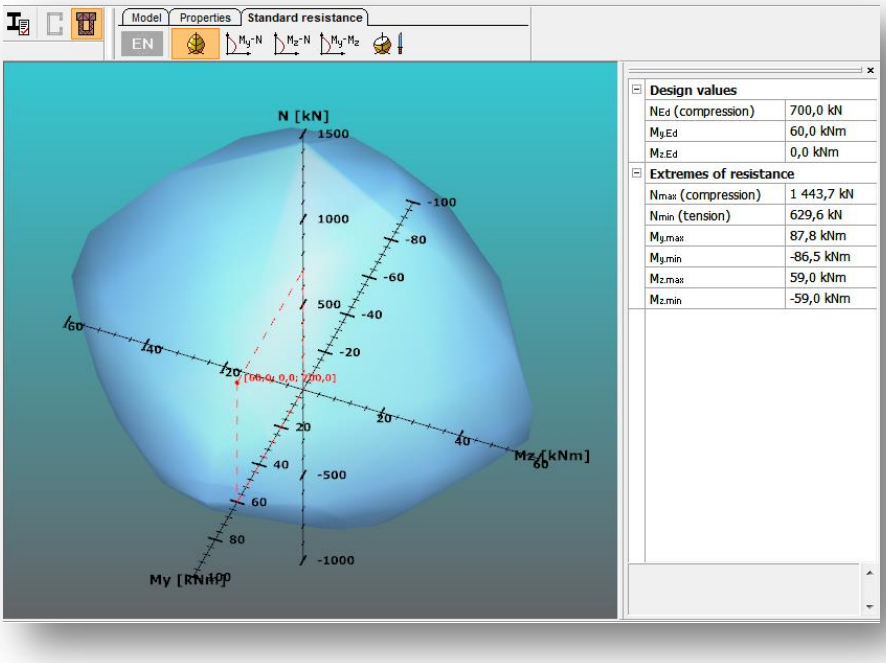
The following design checks are calculated by *csSECTION*:

Verification type	Section class				Shape type					
	1	2	3	4	I	U	O	□	■	Other
Pure forces										
6.2.3	•	•	•	•	•	•	•	•	•	•
6.2.4	•	•	•	•	•	•	•	•	•	•
6.2.5	•	•	•	•	•	•	•	•	•	•
6.2.6(2-3)	•	•			•	•	•	•	•	•
6.2.6(4-5)			•	•	•	•	•	•	•	•
6.2.7	•	•	•	•	•	•	•	•	•	•
EN 1993-1-5	•	•	•	•	•	•		•		
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	•	•			•	•	•	•	•	•
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)	•	•						•		
ENV 1993-1-1:1995 (E)	•	•					•			

5.4.8.1(5.34)										
My + Mz + N (+ V + T)										
6.2.9(6.41)	•	•			•		•	•		
Interaction – General										
My + Mz + N (+ V + T)										
6.2.1 (6.2)	•	•	•		•	•	•	•	•	•
6.2.9.3(6.44)				•	•	•	•	•	•	•
Verification based on elastic stresses										
My + Mz + N (+ V + T)										
6.2.1 (6.1)	•	•	•	•	•	•	•	•	•	•
Lateral and lateral torsional buckling										
6.3.4	•	•	•	•	•	•	•	•	•	•

9.4.5.2 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.



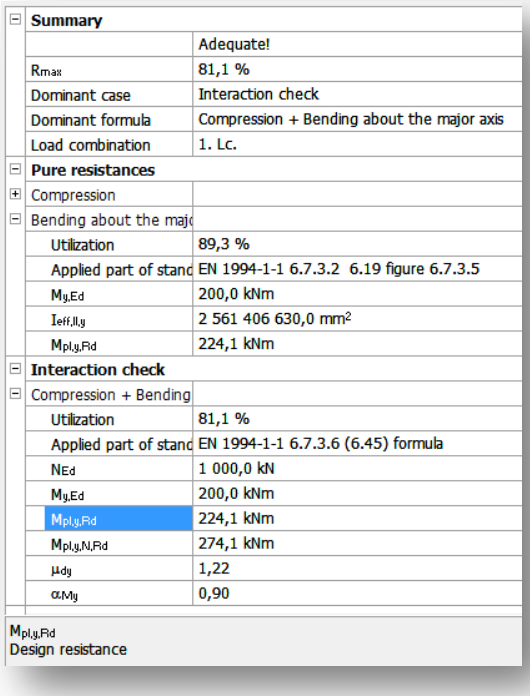
For the composite sections the resistance surface and the part of the EC 4 are performed.

The following design checks are calculated by *csSECTION* for the composite sections:

Verification type	Shape type				
					
Pure forces					
6.7	•	•	•	•	•
6.7.3.2 (6.30)	•	•	•	•	•
6.7.3.2 (figure 6.19), 6.7.3.4	•	•	•	•	•
6.7.1 (9), table 6.3		•	•	•	•
Interactions	•	•	•	•	•
6.7.3.6 (6.45)	•	•	•	•	•
6.7.3.7 (6.46-6.47)	•	•	•	•	•

9.4.5.3 RESULTS

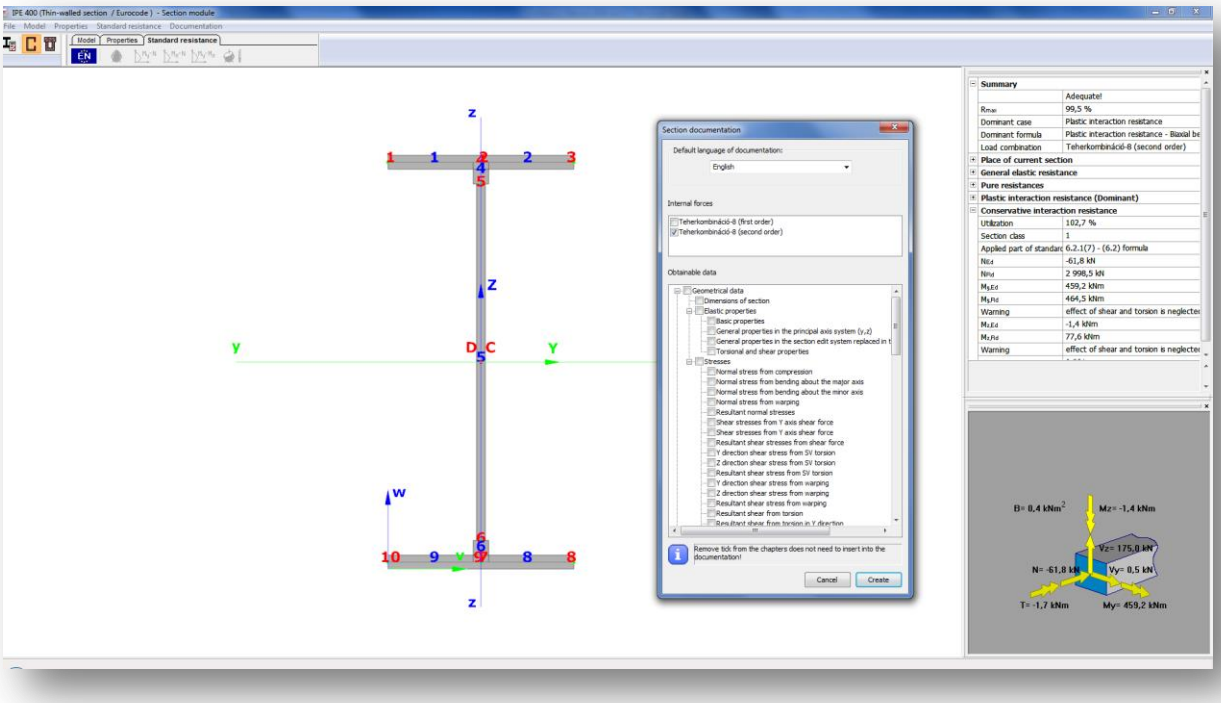
There is a short summary about the calculations on the right and also the name of the dominant case, formula and load combination can be found there.



9.5 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.

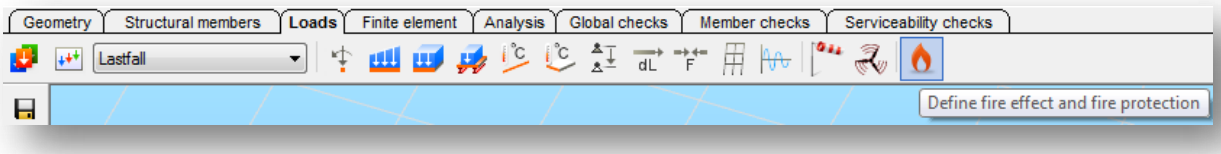
After clicking the **Documentation** option on the menu bar, the way for creating documentation is the same as in *ConSteel* main module (see Chapter 11 for more detailed instructions).



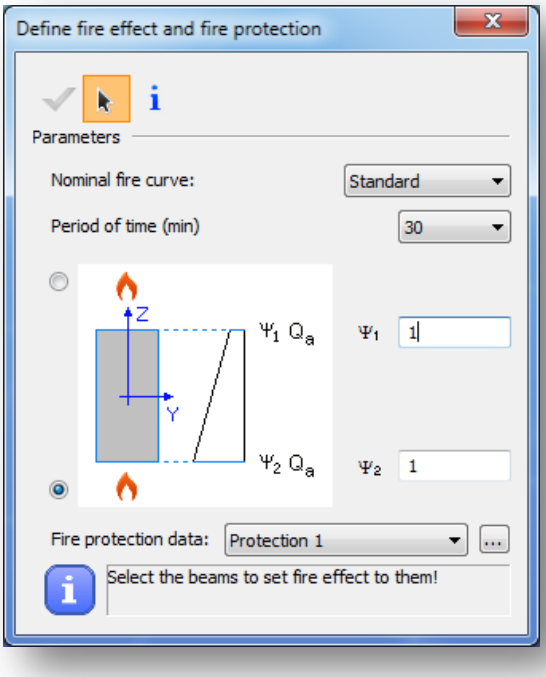
10 FIRE DESIGN

10.1 FIRE EFFECT AND PROTECTION

Before define fire effect an accidental load group and at least one accidental load case must be created. The fire effect and protection can be defined by clicking the appropriate icon  on the ***LOADS*** tab.

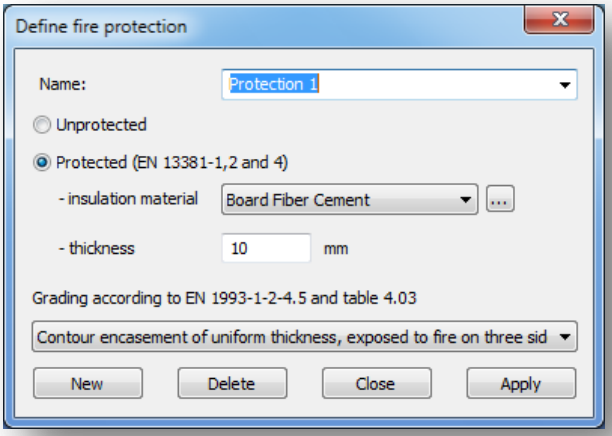


The first option is to set the nominal fire curve. The second one is to set the fire resistance time. The fire effect can be set to different intensity in the top and the bottom of the section.



Different fire protection can be created by clicking on the three dot icon next the Fire protection data. In this window it is possible to select between Unprotected and Protected.

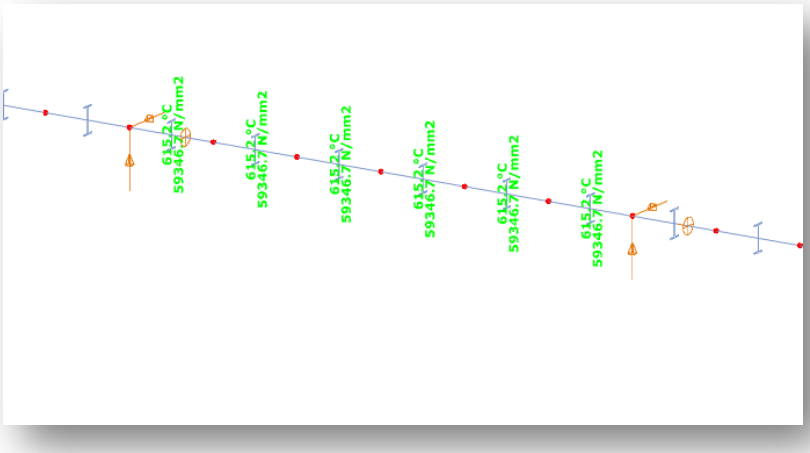
If Protected is chosen then the insulation material and the thickness must be specified. The third parameter to set is the grading according to EN 1993-1-2-4.5.



If the fire protection is set then the members must be selected. 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.

10.2 Analysis

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 accidental (fire) load combinations are performed in global level. Therefore the calculated deformations and internal forces include the effect of the expansion and decreased elasticity modulus of those members which are subjected to fire.

10.3 DESIGN

ConSteel performs cross section resistance checks for all members which are subjected to fire effect by using the adequate formulas from the EuroCode 3 standard.

Tension (EN 1993-1-2 4.2.3.1)

Permanent temperature ((1))

$$N_{t,fi,\theta,Rd} = k_{y,\theta} \cdot \frac{\gamma_{M,0}}{\gamma_{M,fi}} \cdot N_{pl.Rd}$$ (Class 1-4)

where

$k_{y,\theta}$ value based on a function according to Table 3.1

$N_{pl,Rd}$ based on normal case

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)

Permanent temperature ((1))

$$N_{c,ft,t,Rd} = A \cdot \frac{k_{y,\theta} \cdot f_y}{\gamma_{M,fi}}$$ (Class 1-3)

$$N_{c,fi,t,Rd} = A_{eff} \cdot \frac{k_{p0.2,\theta} \cdot f_y}{\gamma_{M,fi}}$$ (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))

$$M_{fi,\theta,Rd} = k_{y,\theta} \cdot \frac{\gamma_{M,0}}{\gamma_{M,fi}} \cdot M_{Rd}$$

where $M_{Rd} = M_{pl.Rd}$ or in case of shear $M_{Rd} = M_{V.Rd}$ (Class 1-2)

$M_{Rd} = M_{el.Rd}$ or in case of shear $M_{Rd} = M_{V.Rd}$ (Class 3)

$M_{Rd} = M_{eff.Rd}$ (Class 4)

$k_{y,\theta}$ value based on a function according to Table 3.1

Varying temperature ((2))

$$M_{fi,t,Rd} = W_{pl,non,t} \cdot \frac{f_y}{\gamma_{M,fi}} \quad \text{(Class 1-2)}$$

$$M_{fi,t,Rd} = k_{y,\theta,max} \cdot \frac{\gamma_{M,0}}{\gamma_{M,fi}} \cdot \frac{1}{\kappa_1 \cdot \kappa_2} M_{el,Rd} \quad \text{(Class 3)}$$

where $\kappa_1 = 0{,}7$ if protection=1 and m_factor_section=2,3 or 10

$\kappa_1 = 0{,}85$ if protection=2 and m_factor_section=3 or 4

$\kappa_1 = 1{,}0$ in other cases

$\kappa_2 = 1{,}0$ conservative

$$M_{fi,t,Rd} = k_{p0.2,\theta,max} \cdot \frac{\gamma_{M,0}}{\gamma_{M,fi}} \cdot \frac{1}{\kappa_1 \cdot \kappa_2} M_{eff,Rd} \quad \text{(Class 4)}$$

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

$$V_{fi,t,Rd} = k_{y,\theta.web} \cdot \frac{\gamma_{M,0}}{\gamma_{M,fi}} \cdot V_{Rd} \quad \text{(Class 1-4)}$$

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:

$$\frac{N_{fi,Ed}}{N_{fi,\theta,Rd}} + \frac{M_{y,fi,Ed}}{M_{y,fi,\theta,Rd}} + \frac{M_{z,fi,Ed}}{M_{z,fi,\theta,Rd}} \le 1$$

A typical cross section check in *ConSteel* looks like below.

Compression	
Utilization	5,4 %
Section class	1
Applied part of standard	EN 1993-1-2: 4.2.3.2
N_{Ed}	-30,0 kN
$N_{fi,c,Rd}$	558,8 kN
$k_{y,\Theta}$	0,772
Θ_A	502,6 °C
f_y	235,0 N/mm^2
γ_{M0}	1,0
$\gamma_{M,fi}$	1,0

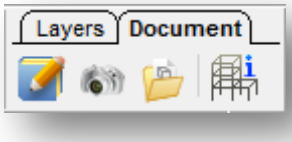
11 DOCUMENTATION

11.1 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.

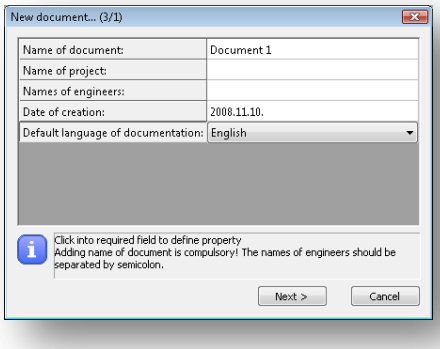
The static documentation contains the full description of the model and also the results of the analysis and design.

11.2 THE DOCUMENT TAB

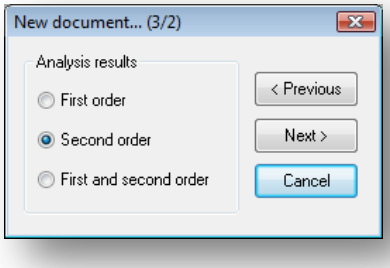


11.2.1 START DOCUMENTATION

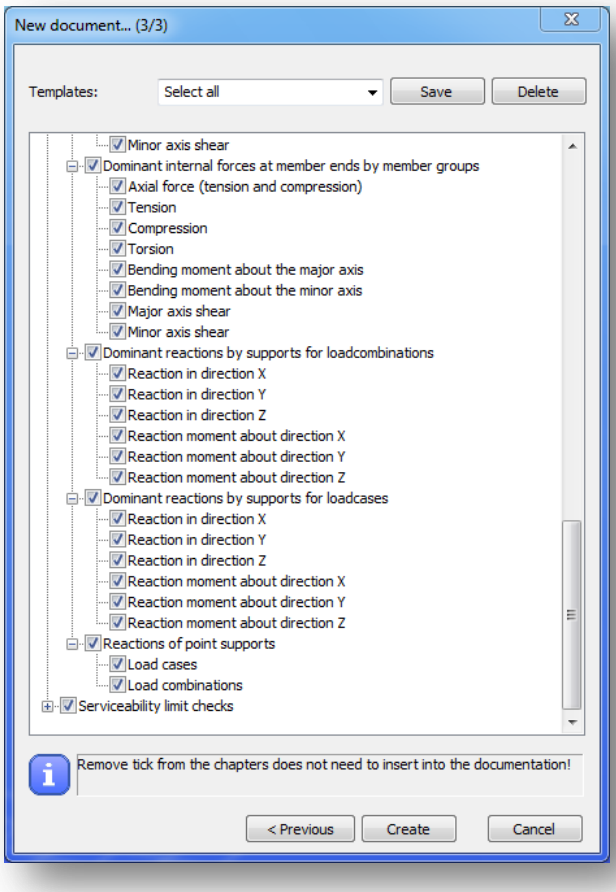
All icons which related to the documentation are placed in the ***DOCUMENT*** tab. The documentation can be generated if first icon is clicked. The documentation wizard guides you through the documentation generating process. On the first window the name of the documentation can be set. The name appears 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.



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.



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.

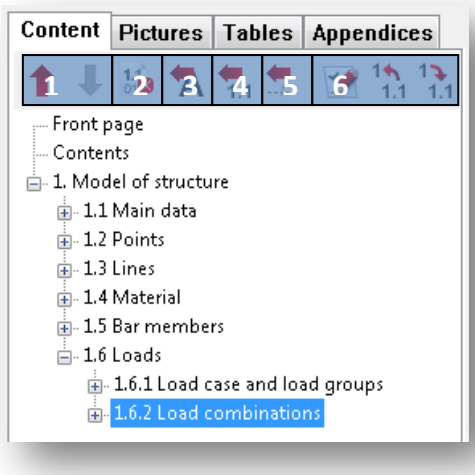


The third window of the wizard includes the name of the chapters which can be added to or can be removed from the documentation.

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.

11.2.1.1 DOCUMENTATION MODULE

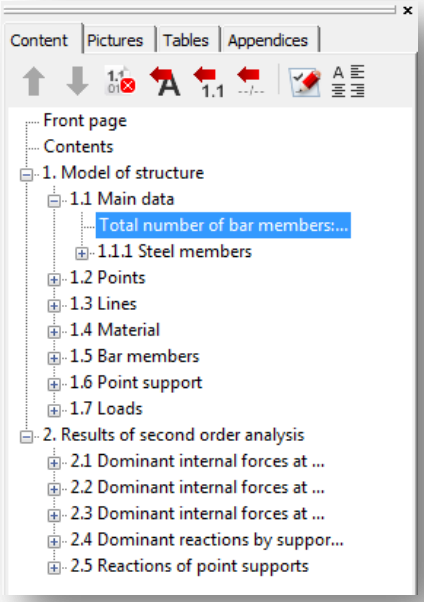


After creating the documentation it will be opened in a new window. At the top right of the window the chapters can be seen. The chapters can be moved up or down using the red arrow icons (#1) if a chapter is selected. It is also possible to delete content from document by clicking the third icon (#2).

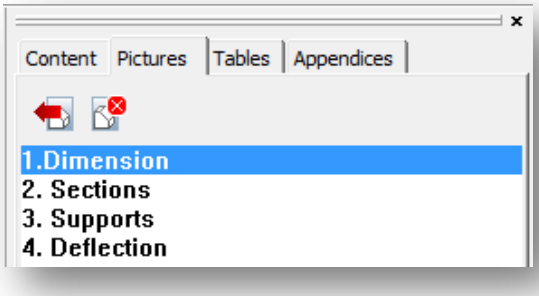
Text can be inserted anywhere in the document if you click on the fourth icon (#3) and type in the desired text into the window. It is also possible to insert headings by clicking on the fifth icon (#4).

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

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



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.



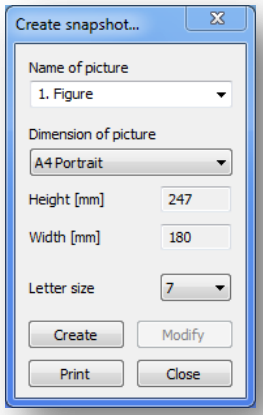
The third tab is the **Tables**. There are lot of possibilities in *ConSteel* where tables can be saved. If you see the  icon during analysis and design process where the results appears in a table, then that table is saveable. Any previously saved tables can be inserted to the documentation or can be 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.

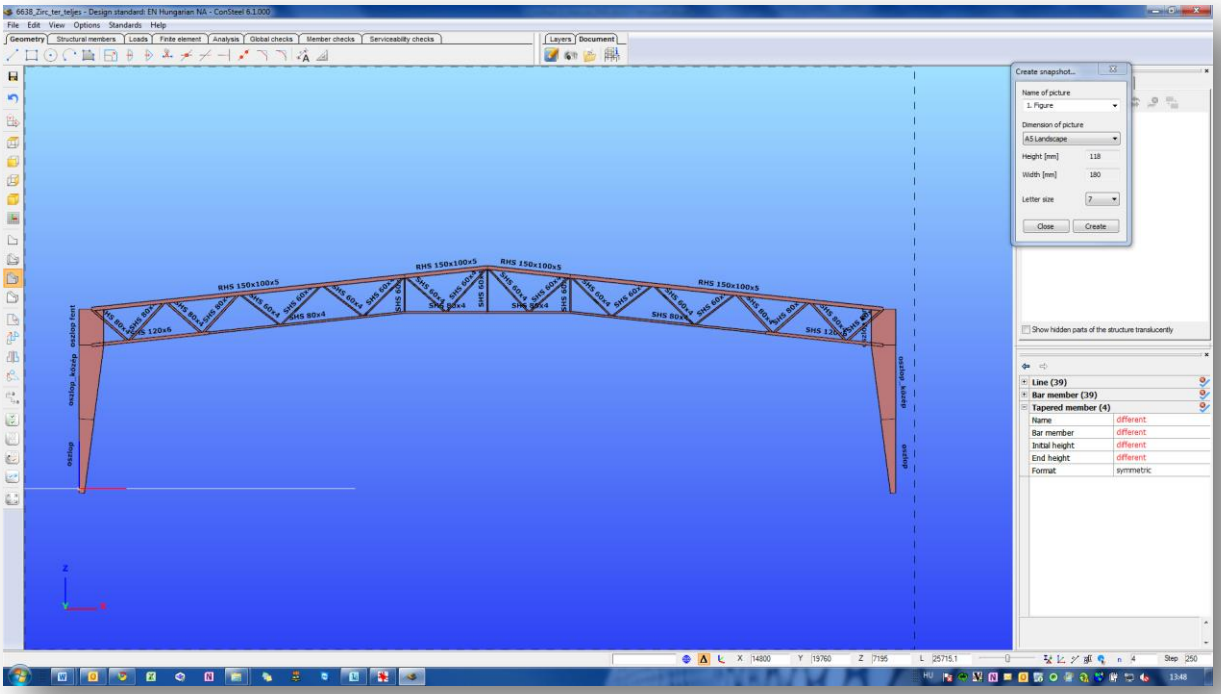
11.2.2 CREATING SNAPSHOTS

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).



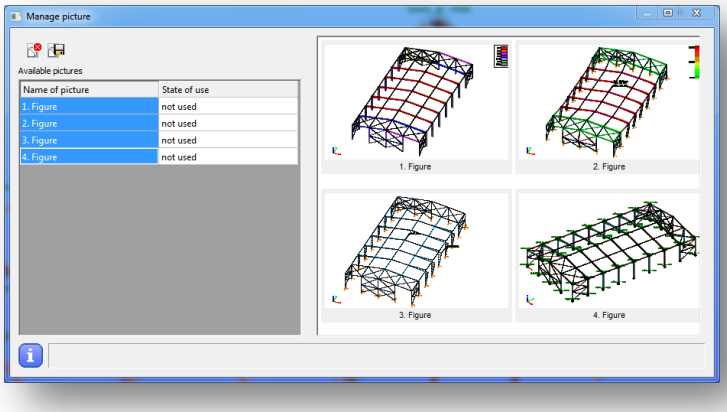
The size of the snapshot can be set and a dashed rectangle shows which part of the model would appear on the picture. The model can be moved or rotated to fit into the dashed rectangle.

With the ***PRINT*** button the actual view of the model can be printed directly from the dialog.



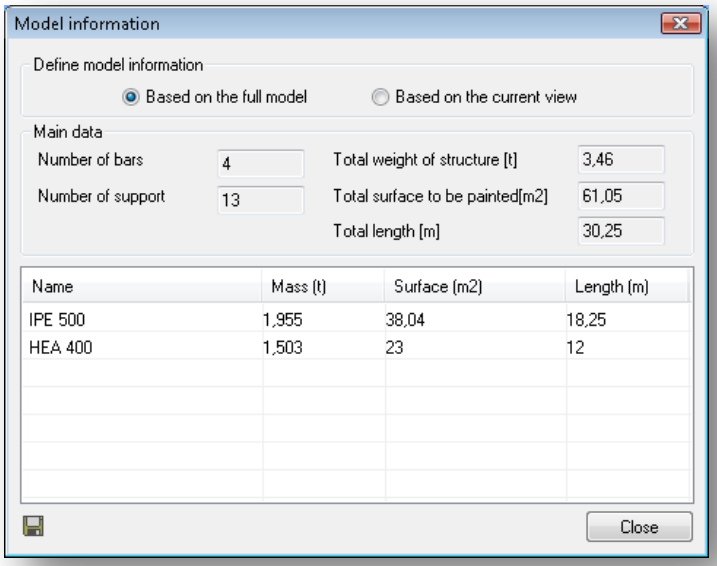
11.2.3 SNAPSHOT MANAGER

The third icon on the documentation tab is the Snapshot manager  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.



11.2.4 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.



12 JOINT MODULE

12.1 BASICS

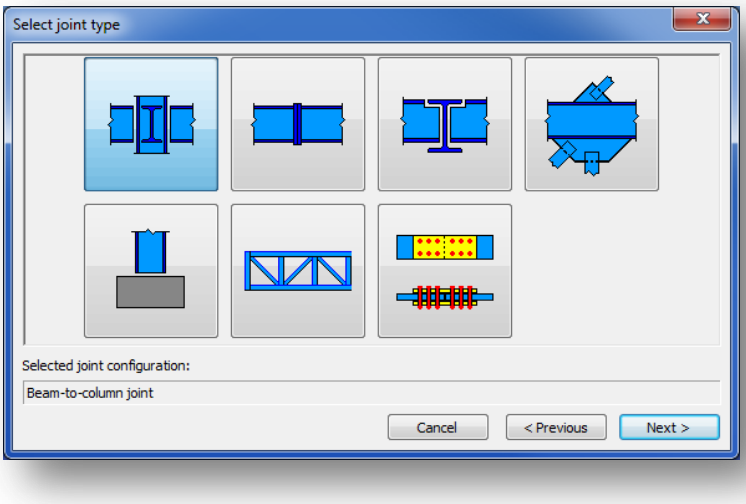
In the *ConSteel* design approach great emphasis is placed on the integrated modelling and calculation of the global structure and its connections therefore a powerful joint analysis engine is developed as a separate module: *csJoint.* A great range of different joint types can be created easily applying 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.



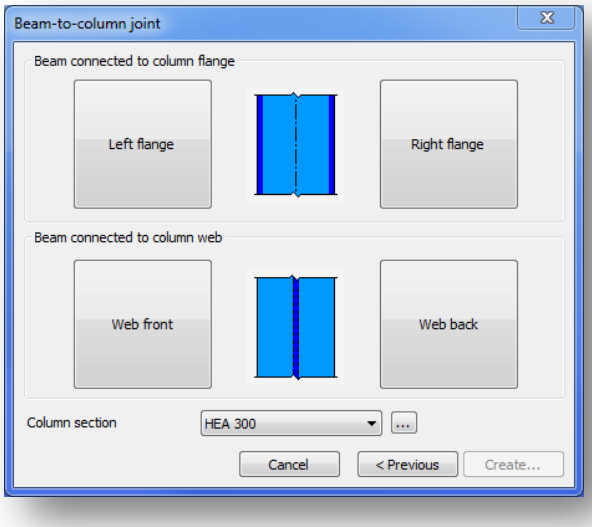
All the joint related commands can be found on the ***STRUCTURAL MEMBERS*** tab. These are the followings: Edit joint, Create joint by model, Place joint.

12.2 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. At the moment the ***CSJOINT*** module has the following joint and connection types:

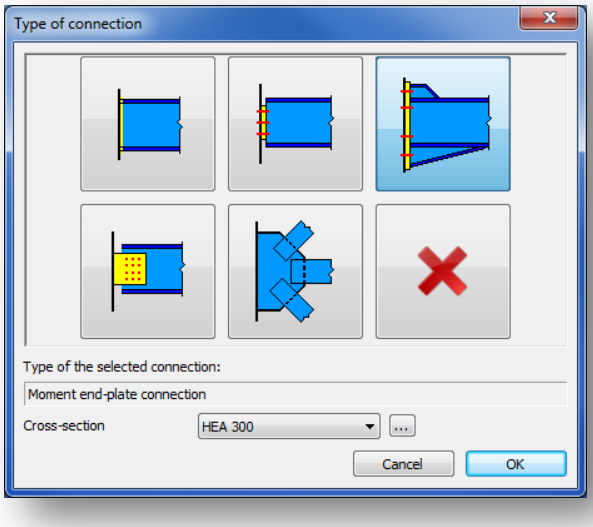


- Beam-to-column joints:



  - Connections on the flanges:

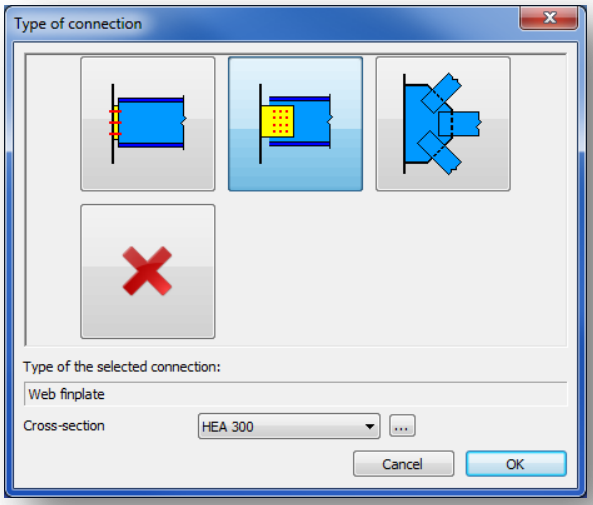
    - Welded moment connection
    - Simple shear connection
    - Bolted, end-plate moment connection
    - Web finplate connection
    - Gusset plate connection with double plate flange splice of I sections, plate splice of hollow sections, angle brace or bolted end-plate of I sections



  - Connections on the web:

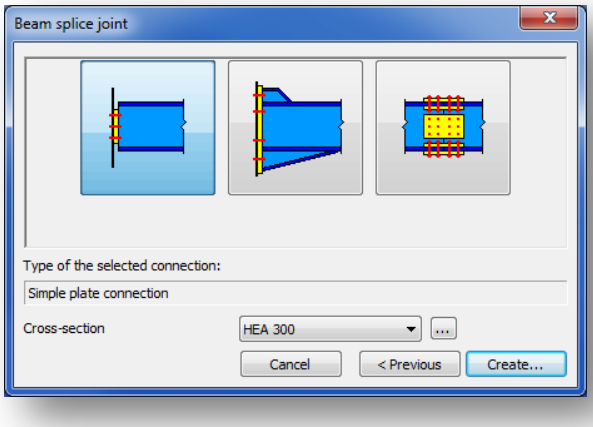
    - Simple shear connection

- Web finplate connection
- Gusset plate connection with double plate flange splice of I sections, plate splice of hollow sections, angle brace or bolted end-plate of I sections



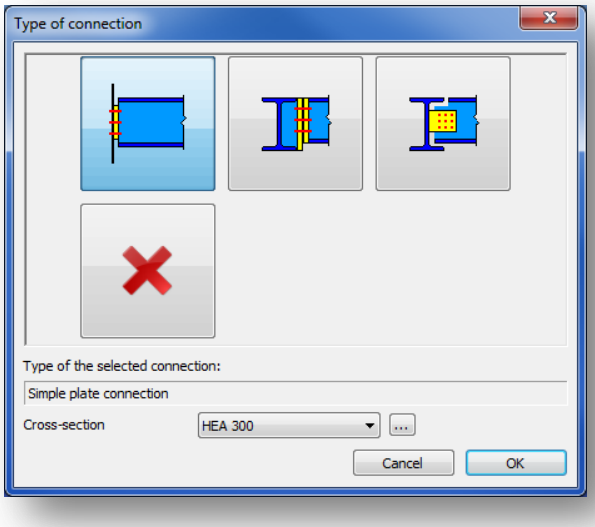
- Beam splice joints with

  - Simple shear connection
  - Bolted, end-plate moment connection
  - Beam splice plate connection

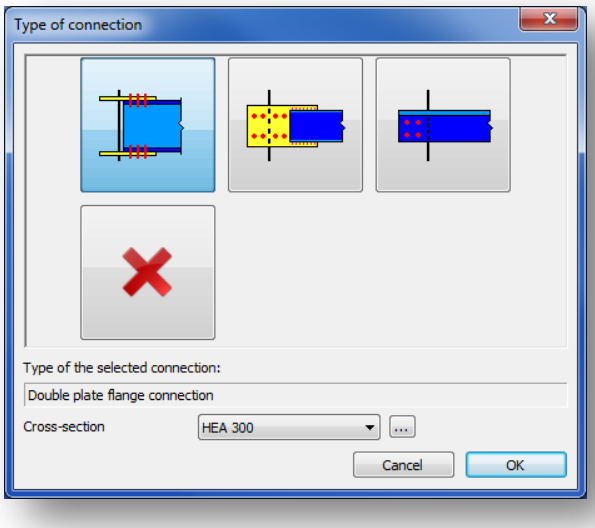


- Beam-to-beam (web) joints with

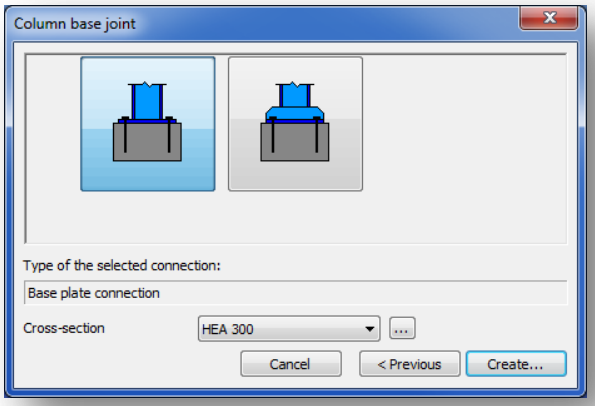
  - Simple shear connection
  - Bolted, end-plate moment connection
  - Web finplate



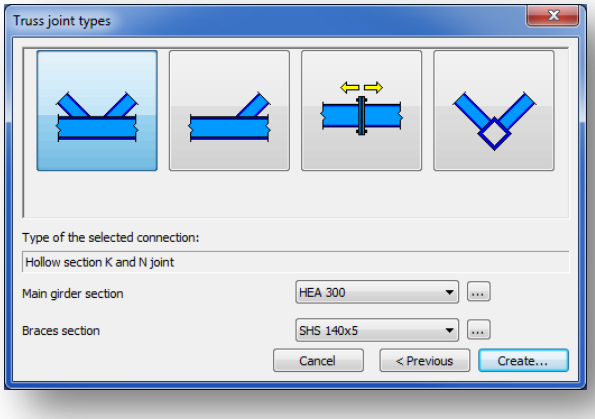
- Bracing gusset plate joint to beam flange and web with
  - double plate flange splice of I sections connection
  - plate splice of hollow sections connection
  - angle brace connection



- Column base joints with
  - Bolted, base-plate moment connection
  - Rigid connection with ground beam

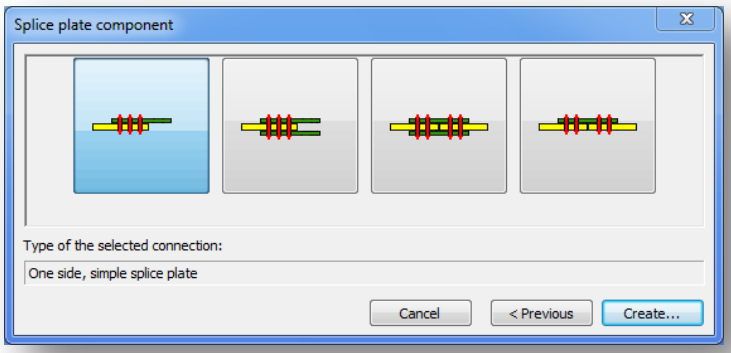


- Hollow section (truss) joints with
  - K and N connection
  - T and Y connection
  - Multiplanar truss
  - Tension chord splice connection

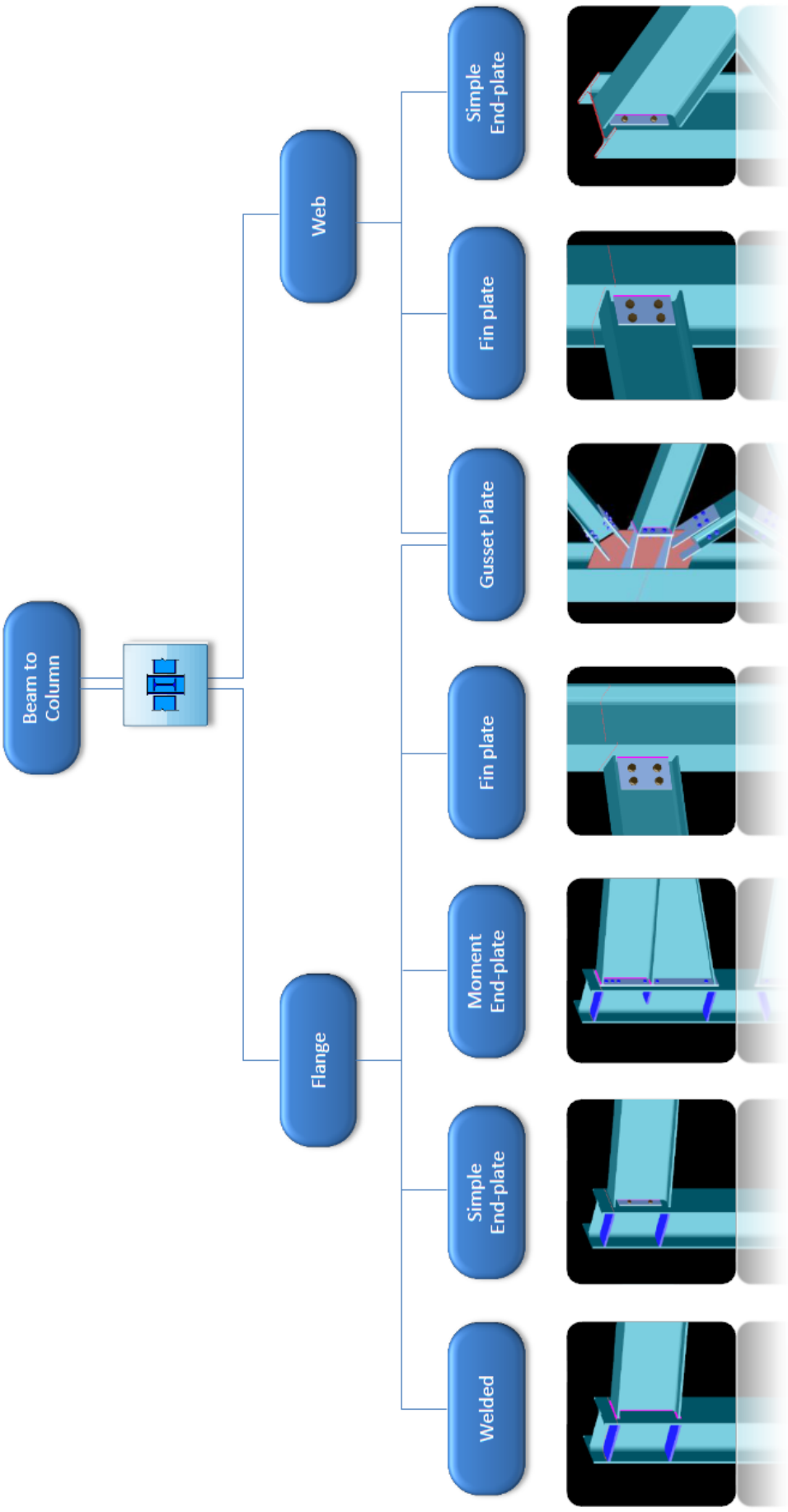


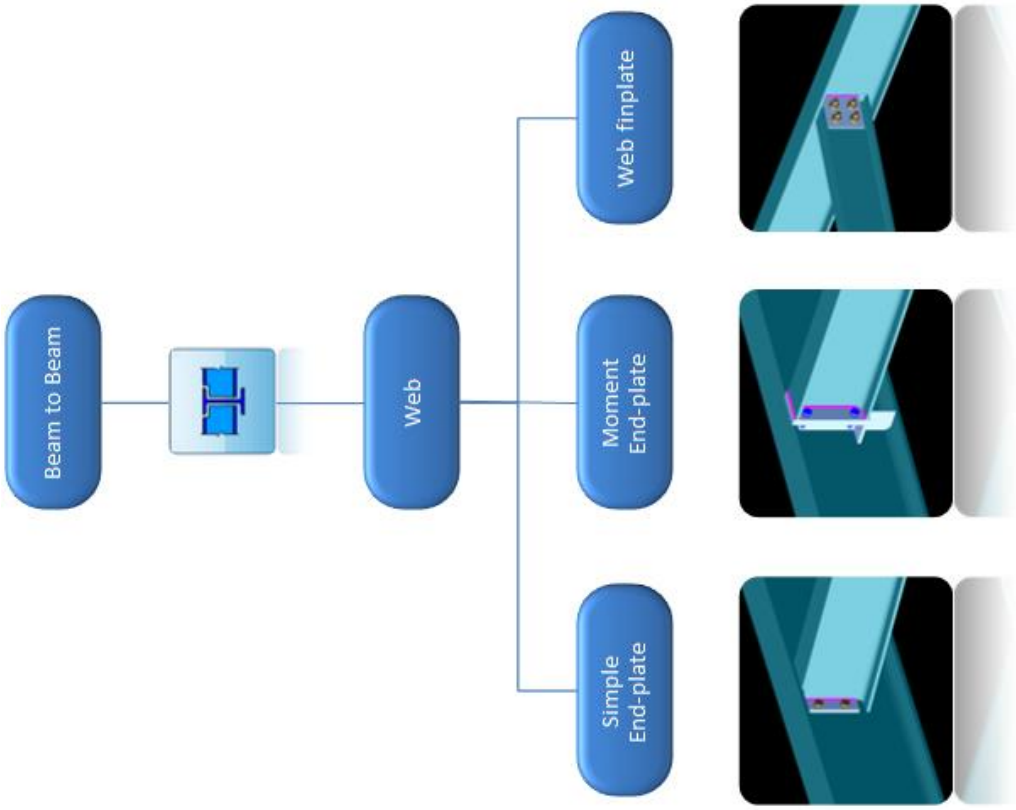
- Splice plate component

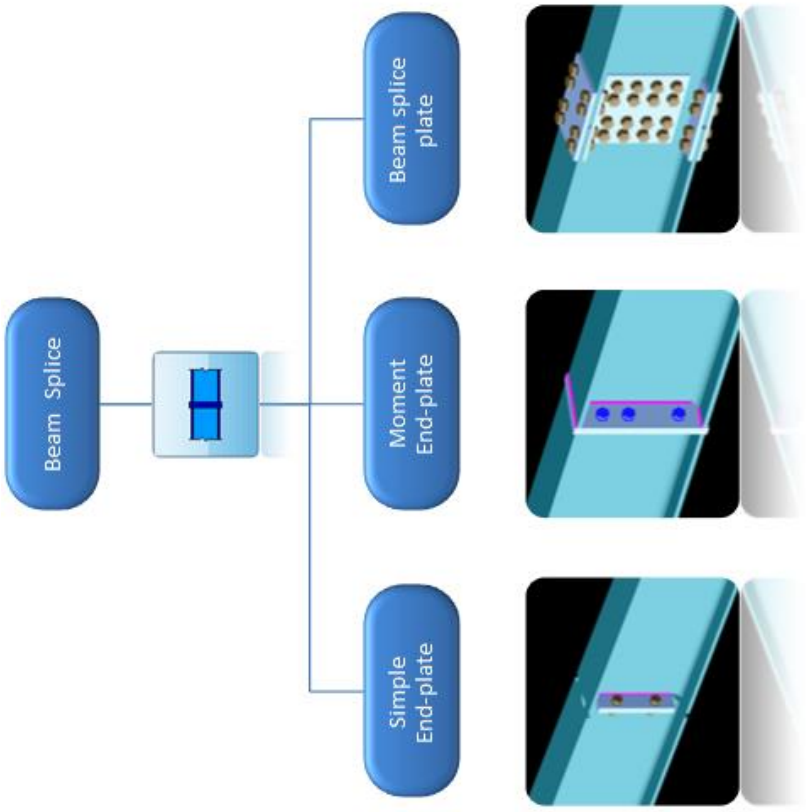
  - One side, simple splice plate
  - One side, double splice plate
  - Two side, double splice plate
  - Two side, simple splice plate

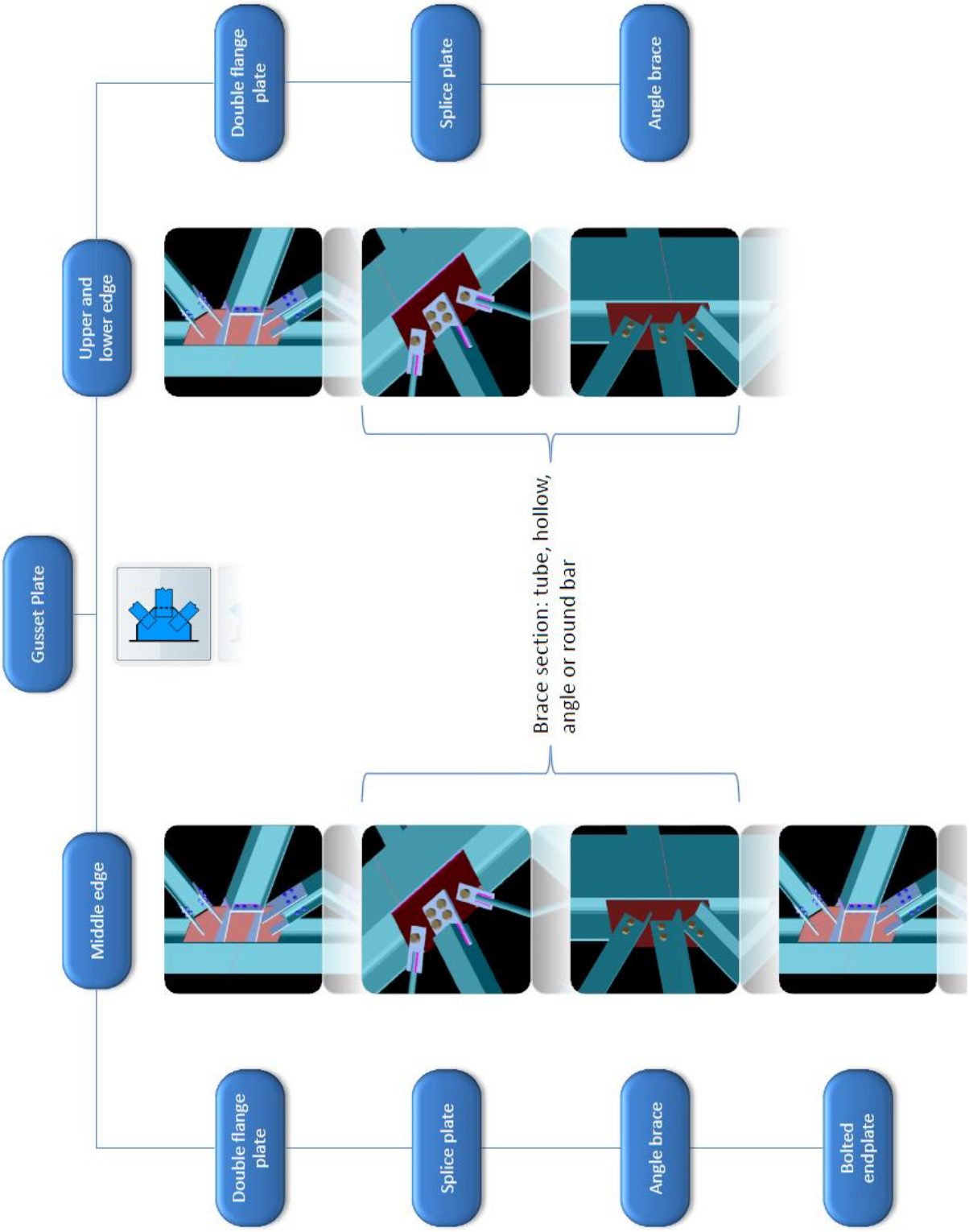


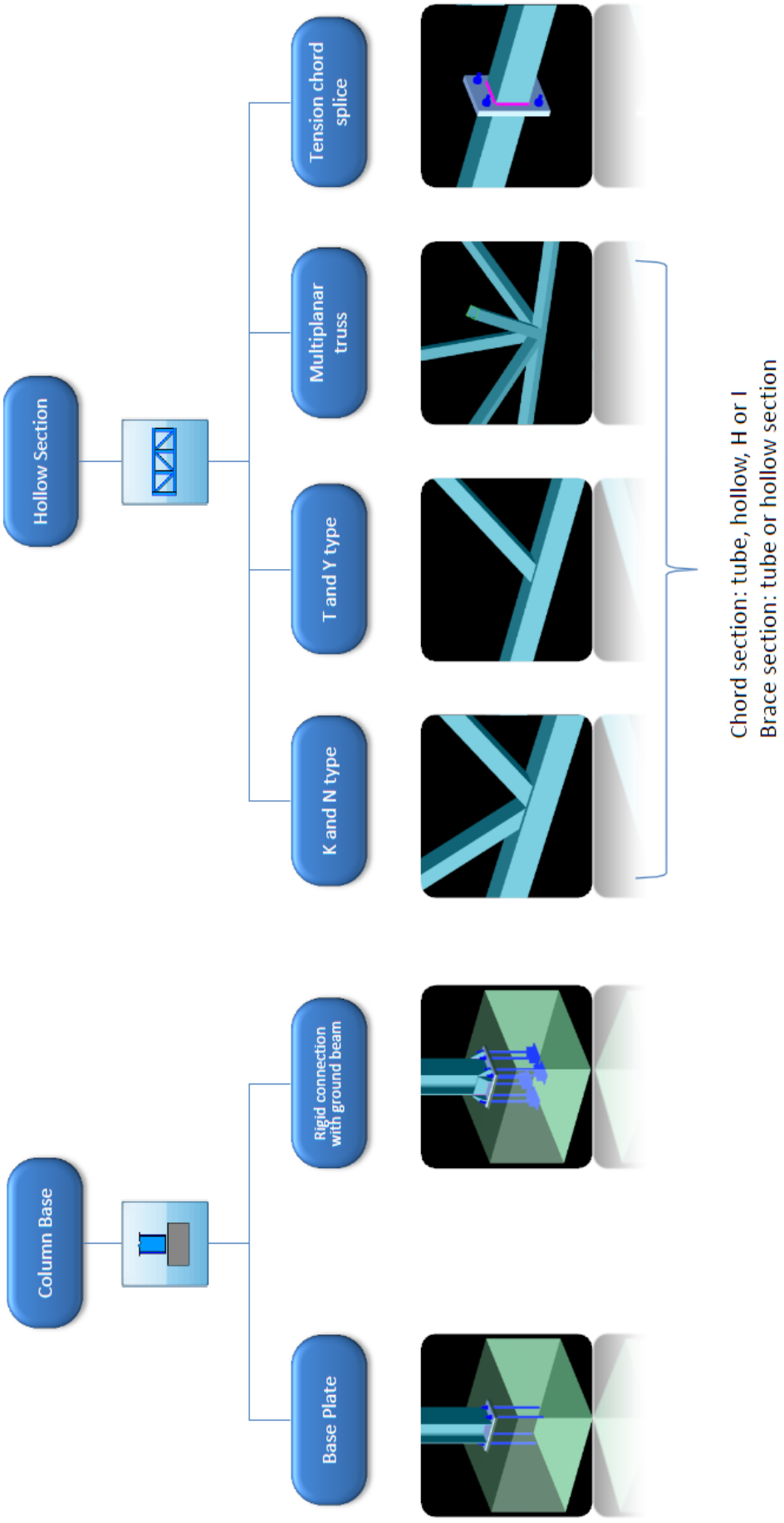
The module offers a number of strengthening possibilities such as: lower or/and upper haunch with or without flange, transverse web stiffeners, flange stiffeners and flange backing plates, shear stiffeners (supplementary web plate, Morrison stiffener, single or double skew plate stiffener) etc.

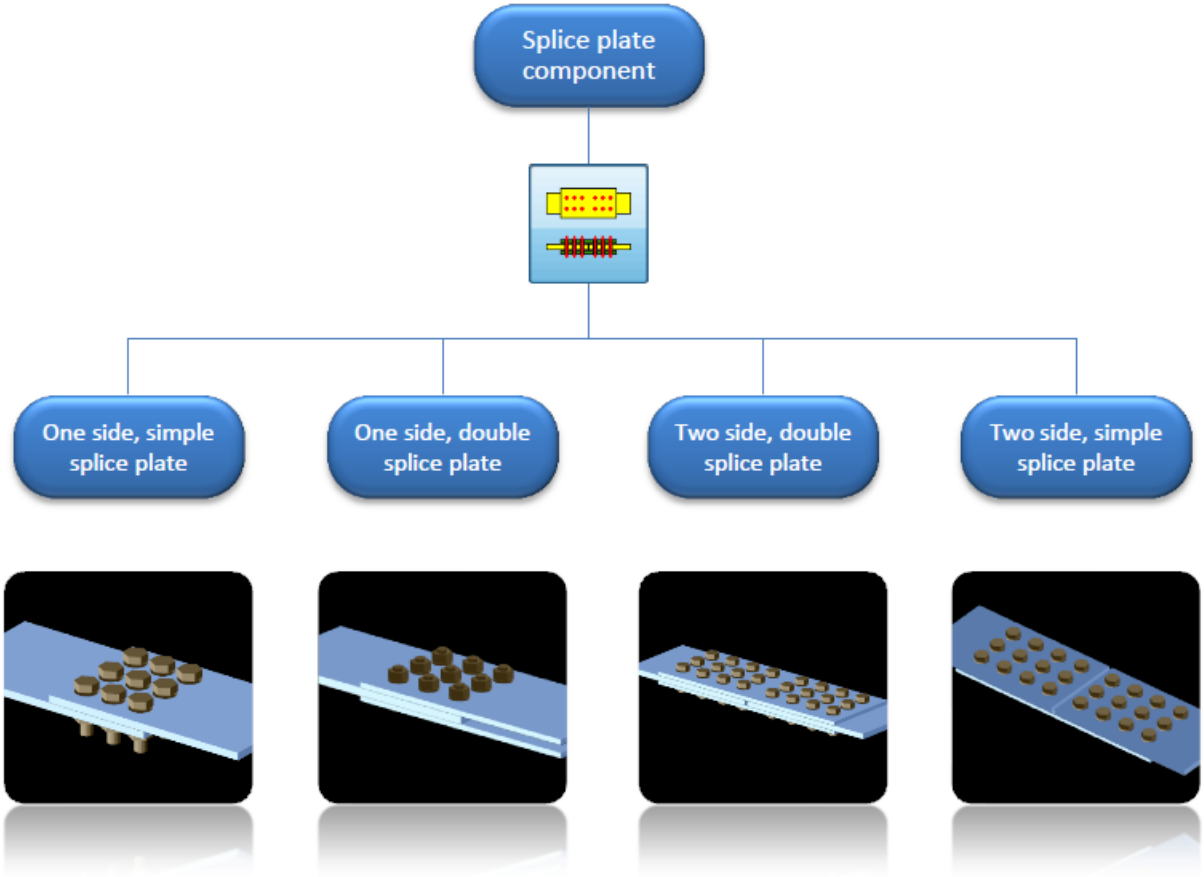

Beam to Column
Web
Flange
Simple End-plate
Fin plate
Gusset Plate
Fin plate
Moment End-plate
Simple End-plate
Welded


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


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


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

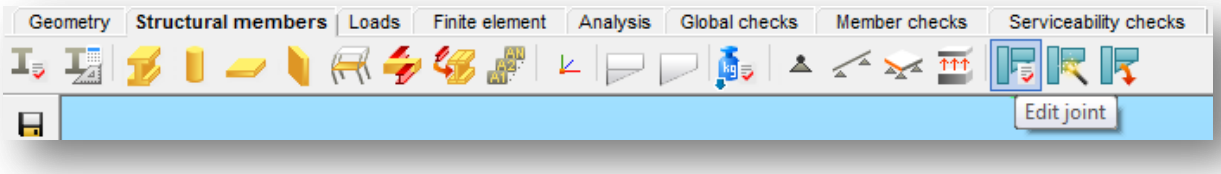

Hollow Section
Tension chord splice
Multiplanar truss
T and Y type
K and N type
Chord section: tube, hollow, H or I
Brace section: tube or hollow section
Column Base
Rigid connection with ground beam
Base Plate


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

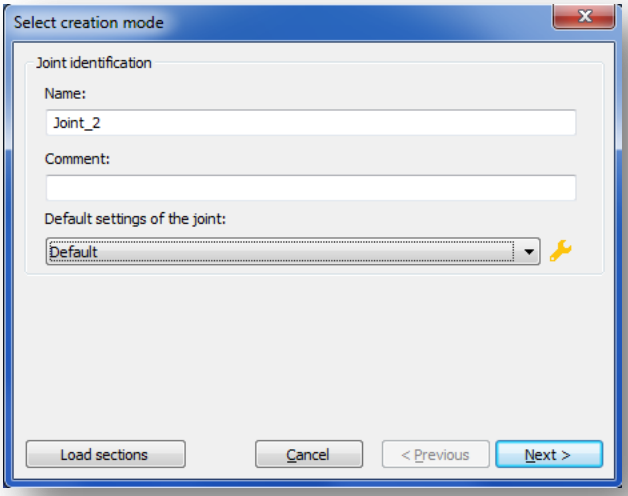
12.3 CREATE JOINT

12.3.1 CREATE JOINT WITHOUT MODEL

Joint models can be created without global model by clicking on the ***EDIT JOINT*** icon on the ***STRUCTURAL MEMBERS*** tab.



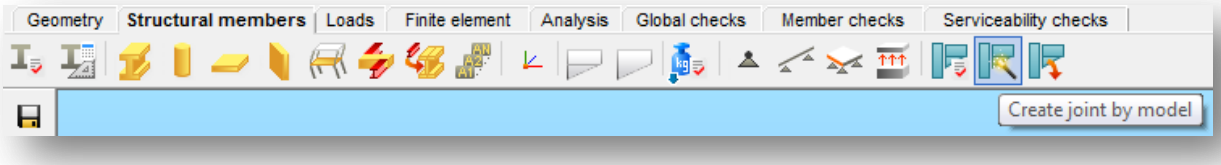
In this case the geometry and the loads need to be set manually. After clicking on the ***CREATE BUTTON*** () on the top left corner the name of the joint must be given. There is an option to select one of the predefined parameter file. These files contain the preferred basic parameters of the joint, which will be used for the creation. The user could prepare several parameter files.



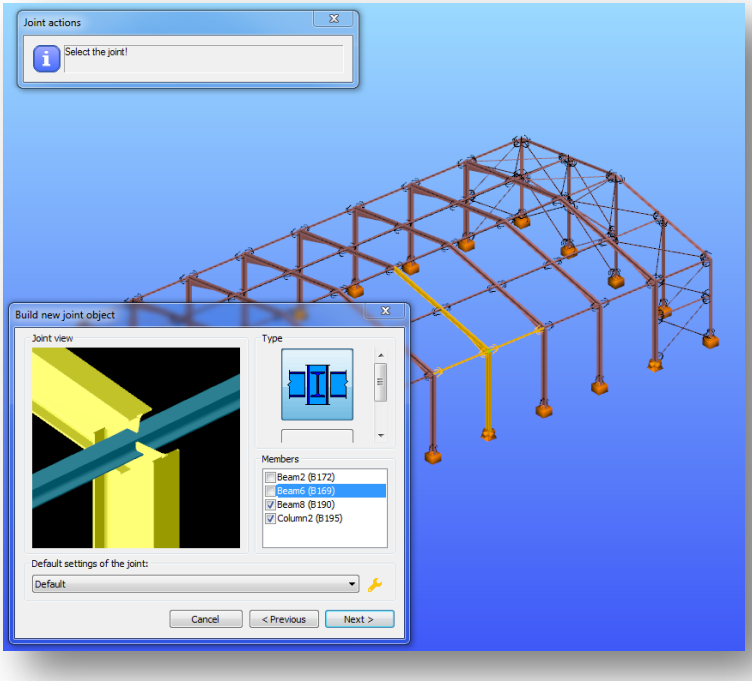
Then the type of the joint needs to be chosen (see chapter 12.2 for the possible joint types). The next step is to choose the geometry of the connection and then the type of connection.

12.3.2 CREATE JOINT BY MODEL

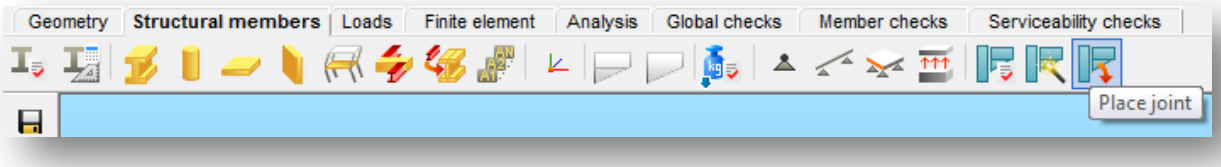
Creating joint by the model is simple and easy. Just click on the ***CREATE JOINT BY MODEL*** button and select the joint.



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



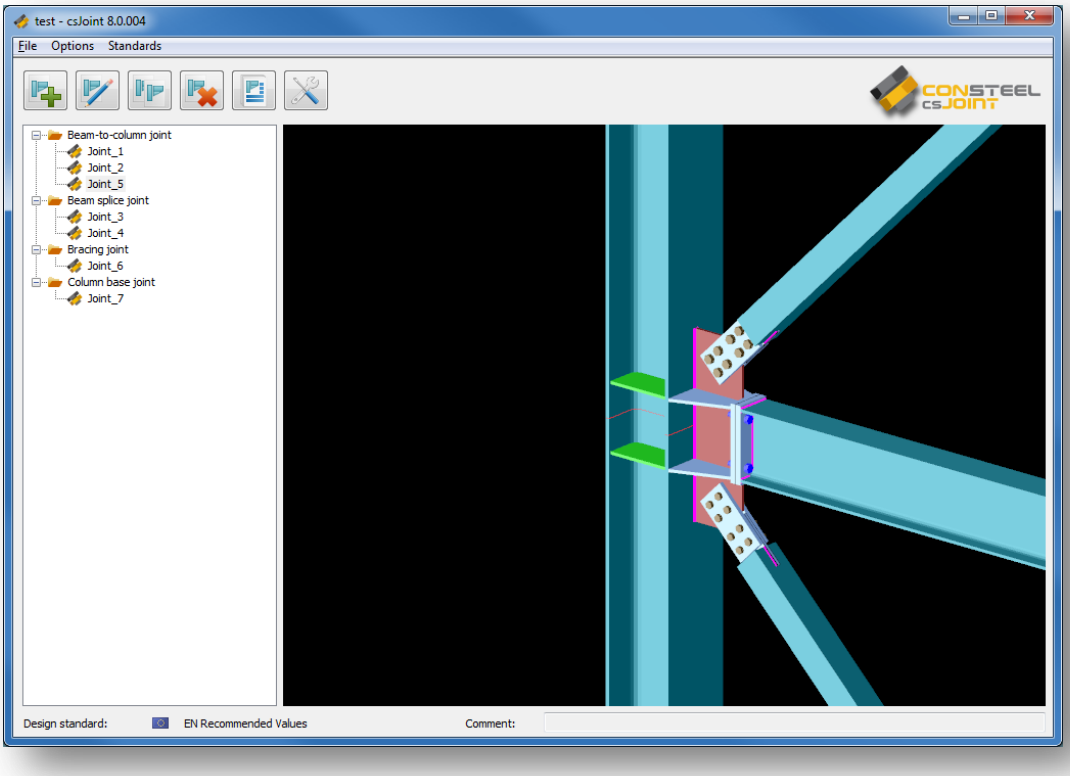
After creating the joint it is advisable to place it in the global model. To do so, use the ***PLACE JOINT*** function of ConSteel.



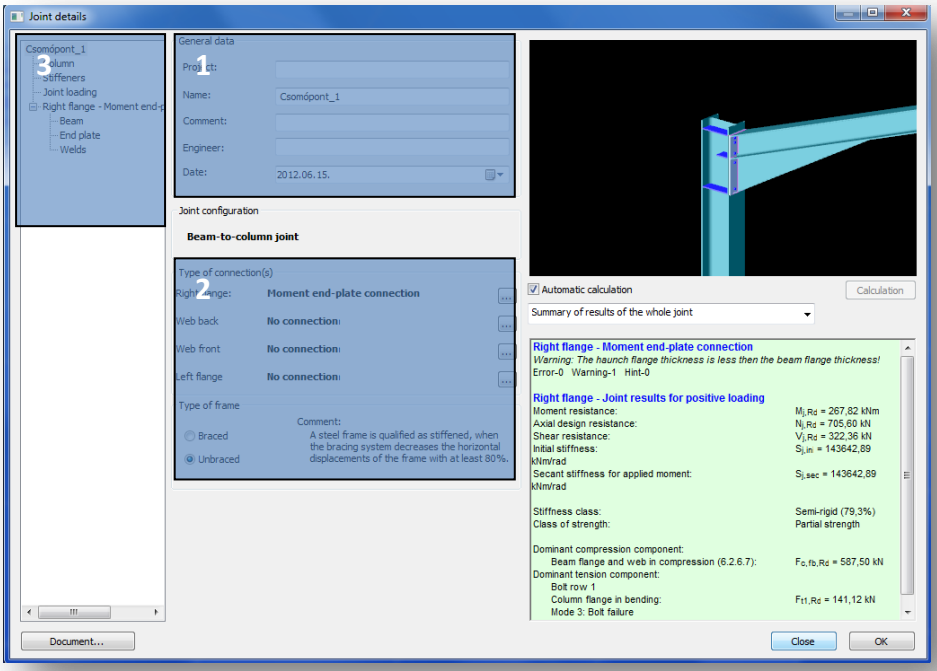
One joint can be placed to several places in the model where the same geometry exists. After the joint is placed, *ConSteel* automatically import the loads from the global model.

12.3.3 MANIPULATING CONNECTIONS

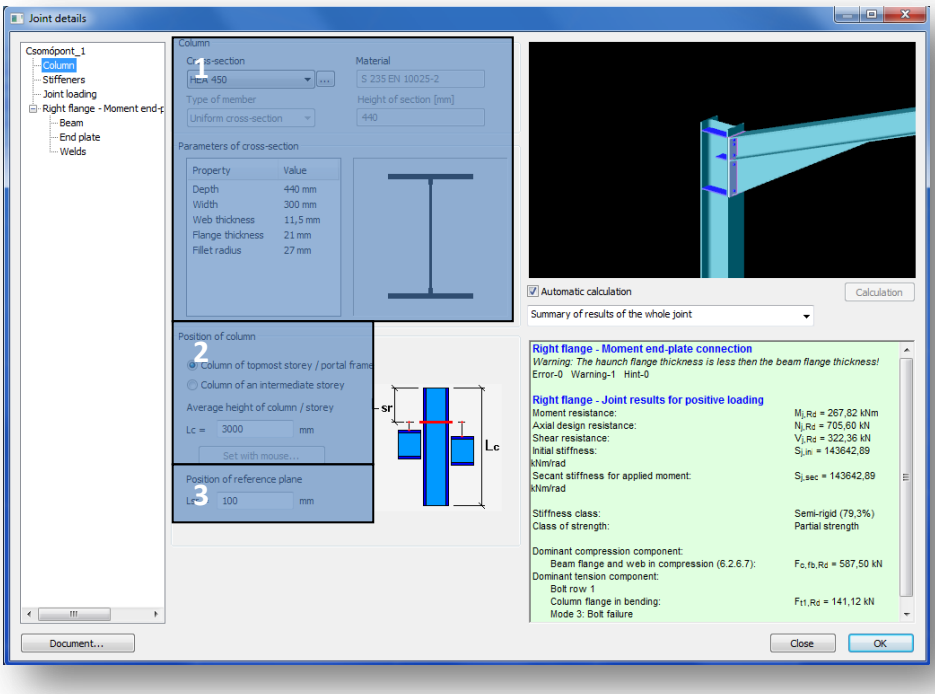
No matter if a joint is created with our without model manipulating the joint is the same. Joint details appear in a new window where numerous settings can be made. In the following pages we would like to give you an overview of the Joint module (***CSJOINT***) using a beam-to-column connection as an example. Select the connection from the tree structure on the ***CSJOINT OPEN*** dialog, and press ***DETAILS*** () button.



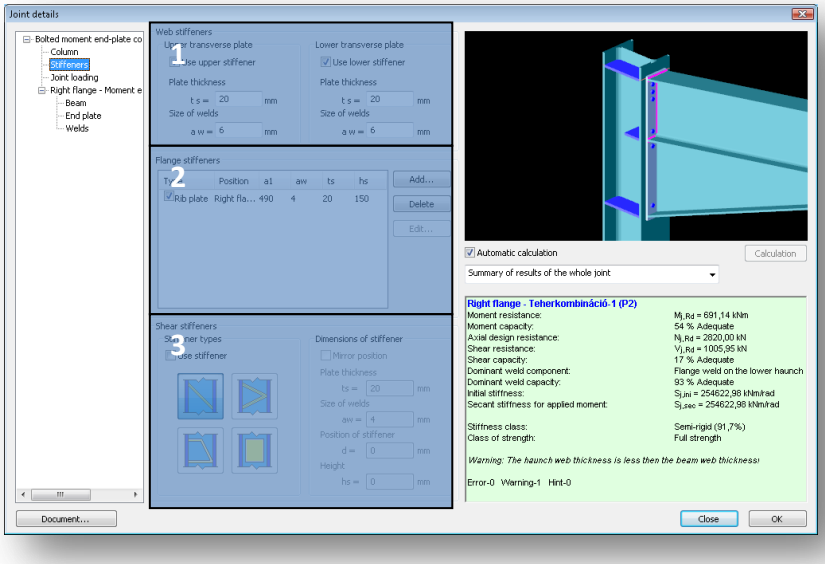
The #1 part in the below contains the general information about the joint: Joint name, project, engineer, date, etc. These properties will appear in the joint documentation and also useful to distinguish joints from each other. The #2 part in the image below shows the general type of the connection. Additional connections can be added or removed with the three dots buttons if necessary. The #3 part shows the tree structure about the joint: all components of the connection can be seen and selected here.



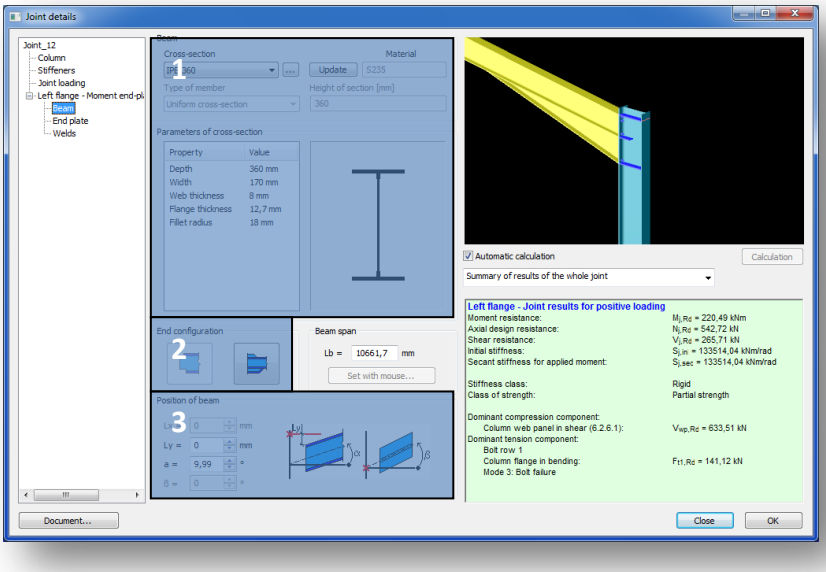
If the ***COLUMN*** is selected in the connection tree, then column related properties of the connection can be set like the section size and material grade (#1). Setting the position of the column is also very important (#2). Furthermore the position of the reference plane is essential for the joint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



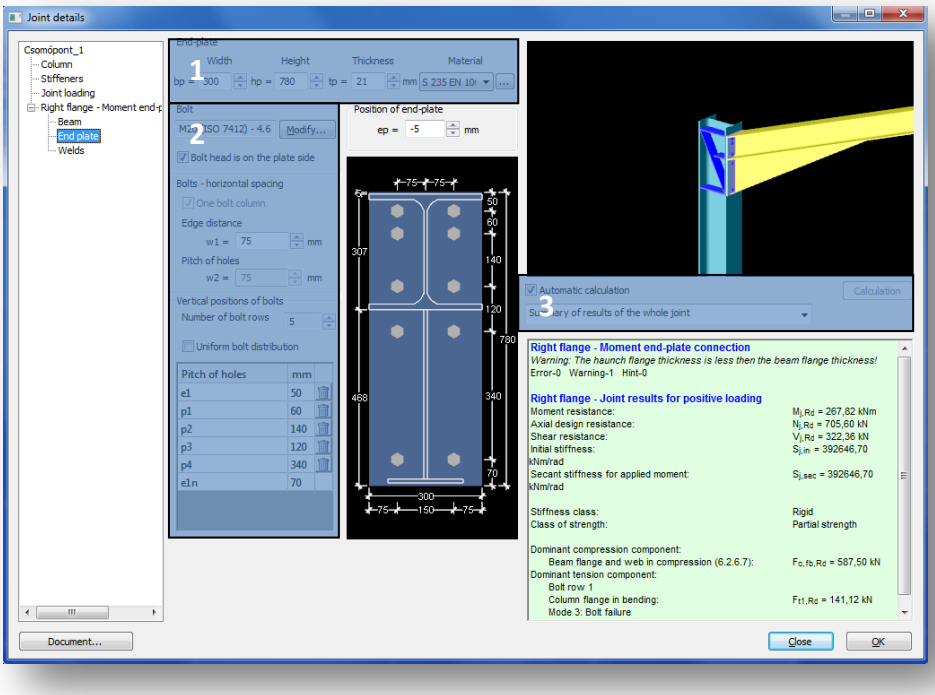
Stiffeners can be added if ***STIFFENERS*** is clicked in the connection tree. Upper and lower stiffener plate properties can be seen in the #1 part of the image below. Additional web and flange stiffeners can also be defined (#2). In order to place shear stiffener in the joint click on 'Use stiffener' 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!).



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).



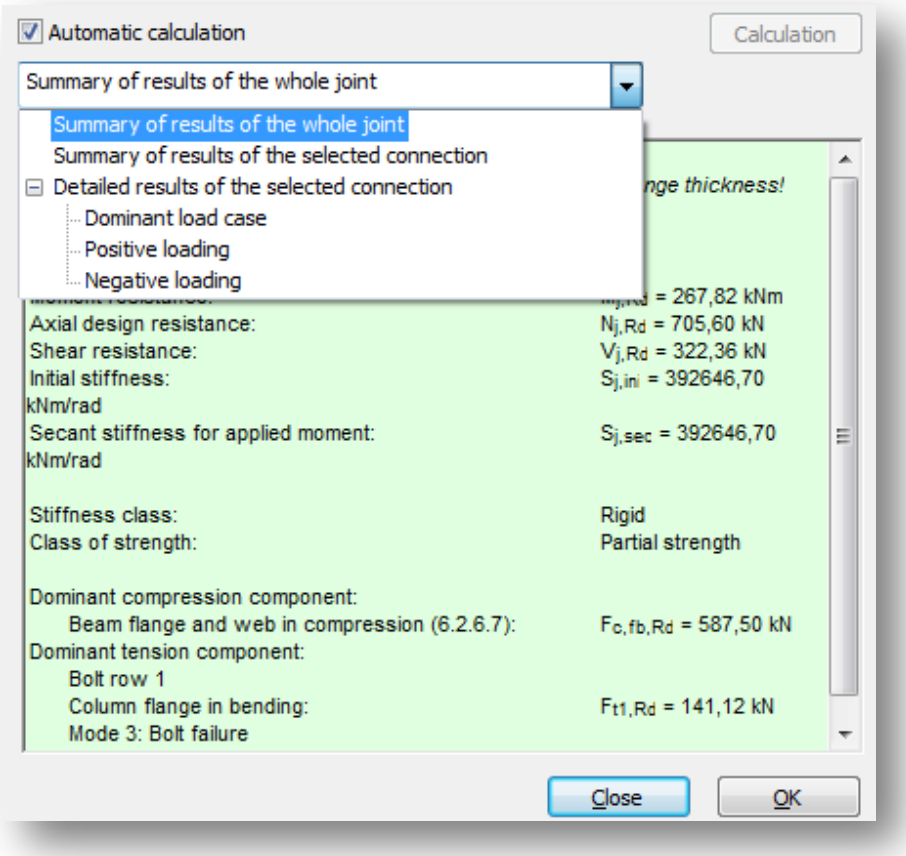
END PLATE properties (geometry, material) can be edited in the #1 part of the image below. Bolt properties can be seen on #2 part.



The size of the welds and the method of weld design can be set if the ***WELDS*** is clicked in the joint tree.

Any changes made on the joint will take place either when pressing Enter on the keyboard 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 (see previous picture #3 part).

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.



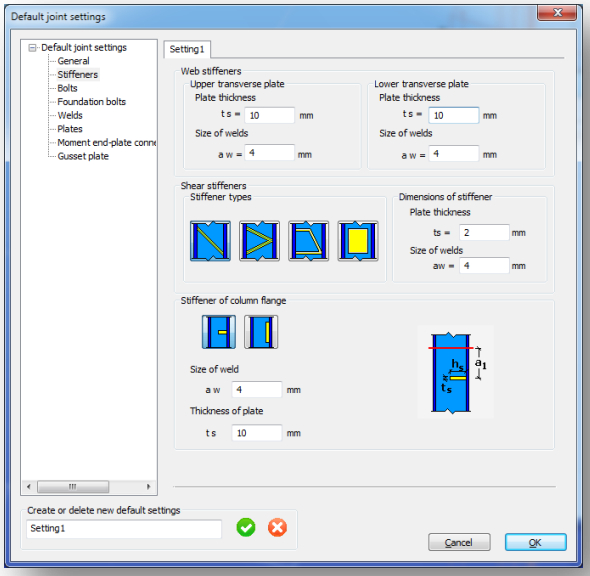
The result of the joint calculation can be seen in the highlighted part of the window. If the detailed result is selected then all the results of the necessary calculations can be seen according to the selected load combination or according to the dominant load case.



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

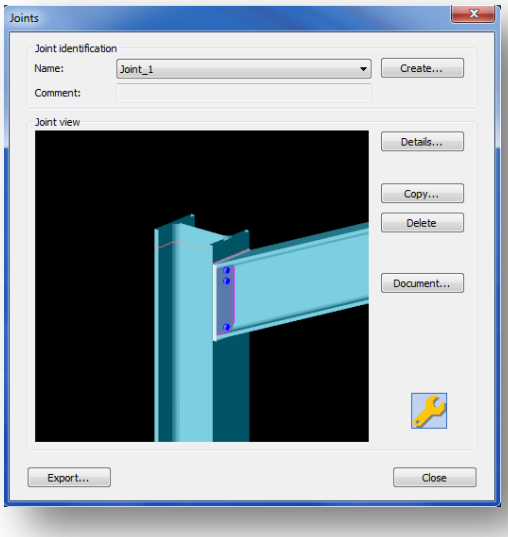
12.3.4 DEFAULT JOINT SETTINGS

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



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

For saving the data of a connection as a default setting click the  icon.



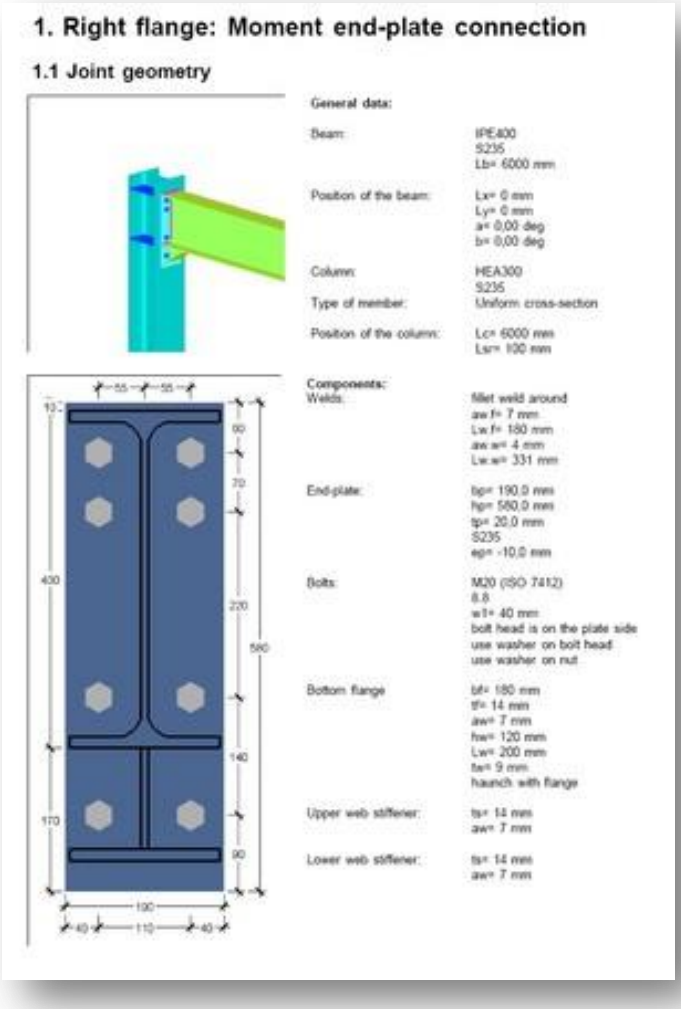
12.4 ANALYSIS OF CONNECTIONS

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

12.5 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.


1. Right flange: Moment end-plate connection

1.1 Joint geometry

General data:

Beam: IPE400, S235, Lb= 6000 mm

Position of the beam: Lx= 0 mm, Ly= 0 mm, a= 0,00 deg, b= 0,00 deg

Column: HEA300, S235

Type of member: Uniform cross-section

Position of the column: Lc= 6000 mm, Lsr= 100 mm

Components:

Welds: fillet weld around, aw,f= 7 mm, Lw,f= 180 mm, aw,w= 4 mm, Lw,w= 331 mm

End-plate: bp= 190,0 mm, hp= 580,0 mm, tp= 20,0 mm, S235, ep= -10,0 mm

Bolts: M20 (ISO 7412) 8.8, w1= 40 mm, bolt head is on the plate side, use washer on bolt head, use washer on nut

Bottom flange: bf= 180 mm, tf= 14 mm, aw= 7 mm, hw= 120 mm, Lw= 200 mm, tw= 9 mm, haunch with flange

Upper web stiffener: ts= 14 mm, aw= 7 mm

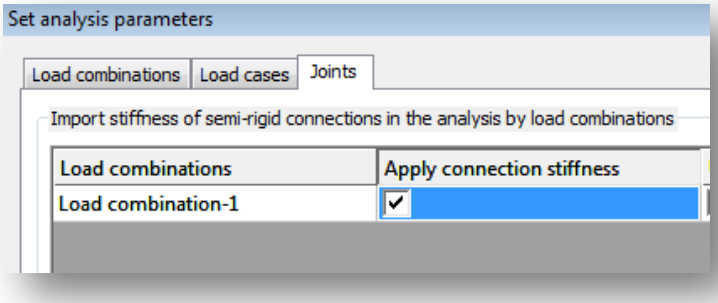
Lower web stiffener: ts= 14 mm, aw= 7 mm

The way for creating documentation is the same as in *ConSteel* main module (see Chapter 11 for more detailed instructions).

12.6 Structure-Joint interaction

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

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



12.7 JOINT EXPORT

All joints can be exported to *TEKLA* structural steel detailing software. See chapter 2.4 for more detailed information.