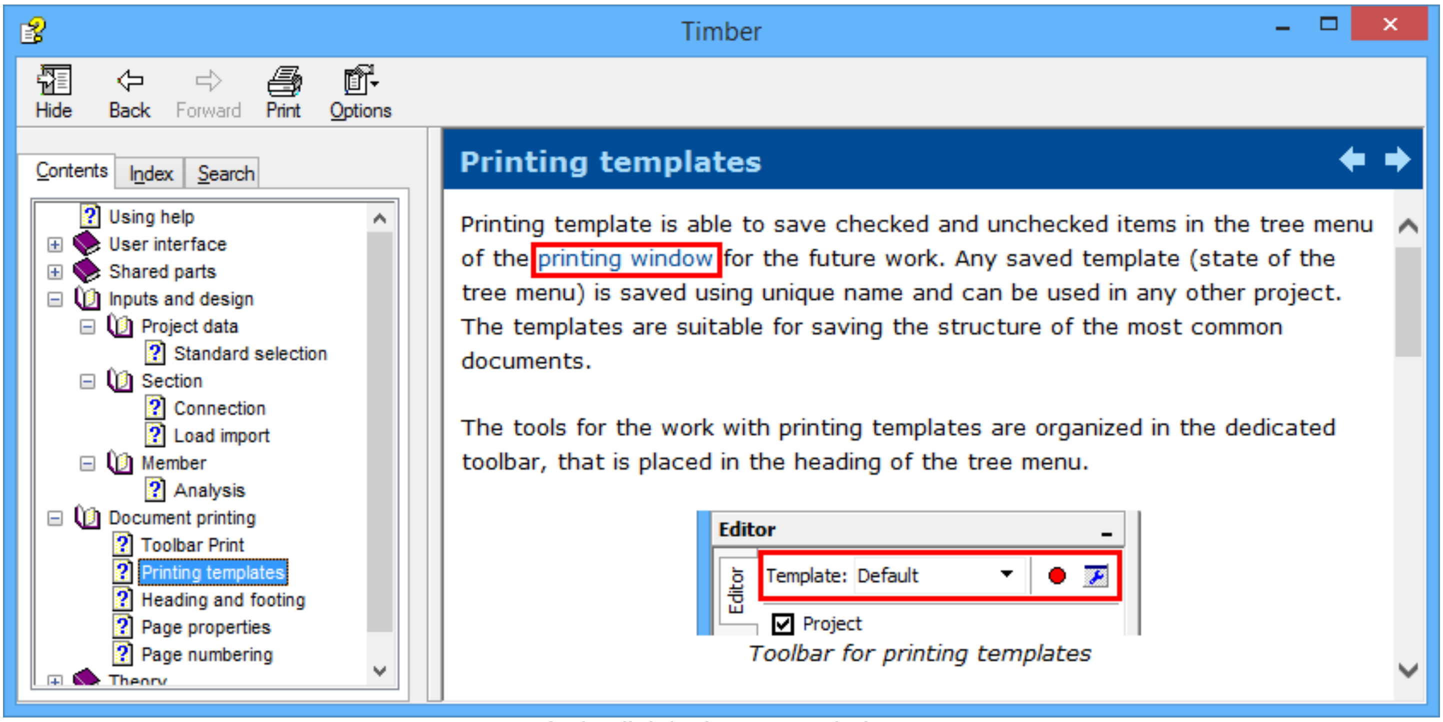
# Using help

Help window can either be launched by directing through the program menu (items "**Help**", "**Content**") or by pressing "**F1**" button anywhere in the program. If using "**F1**" button, the dedicated page for active program mode or window will be displayed.

These tools can be used for navigation in Help window:

- **Tree menu** - List of the topics organized in the tree structure. Left-button click on the topic launches corresponding page in the right part of the window. The branches of the tree menu can be hidden by clicking the symbols "+/-" in front of the topic's name. The tree menu can be hidden by clicking the button "**Hide**" in the main toolbar.
- **Buttons "Back" and "Forward"** - These buttons in the main toolbar allow listing through pages, which have been recently opened.
- **Links** - The help pages contain links to the related pages. These links are highlighted by blue colour.
- **Index** - List of topics organized in the alphabetical order. The index can be displayed using tab "**Index**" above the tree menu.

Active page can be printed using the "**Print**" button in the main toolbar. The printing command is also included in the context menu of displayed page. The context menu can be opened by right-button click.




*Active link in the menu window*

# What's new

This chapter contains changes and new features sorted according to the releases:

- **Version 4**
- **Version 5**
- **Edition 2017**
- **Edition 2018**
- **Edition 2019**
- **Edition 2020**
- **Edition 2021**

# Version 4

The version 4 was released on 31 October 2014. It brings these features:

## New program Concrete Fire

- Fire resistance of RC members according to EN 1992-1-2
- Determination of temperature in concrete and reinforcement bars
- General combination of actions (normal and shear forces, bending and torsional moments)
- Nominal and parametric fire exposure curves

- Basic and accidental loads, import of loads from file
- Analysis according to 500°C isotherm method or Zone method
- Import of projects created in programs "**Concrete 2D**" and "**Concrete 3D**"

### Fin 2D, Fin 3D

- Members with excluded tension or compression
- Deformation in all points of structure
- Auto save option
- Copy of load case including loads
- Input of loads in the local coordinate system of the cross-section in Fin 3D

### Concrete Beam

- Spring supports, internal hinges, analysis nodes
- Tapers

### Concrete 2D, Concrete 3D

- Input of effective modulus of elasticity for stress verification
- Stress and deformation diagrams

### Loading

- Envelope of wind loads on roof
- Localization based on force (calculation of anchors)

### Steel

- Manual input of buckling curve
- Plastic analysis of general cross-sections created in program "**Section**"
- Figure of effective cross-section for class 4

### Masonry

- New verification of pillars and walls
- New load template – pillar/wall
- Effective thickness of wall

### Other features

- Covering page for graphical documents
- Timber – wood-based materials
- Timber Fire – analysis of built-up cross-sections
- Section – import of DXF file

## Version 5

Version 5 of the program "**Fin EC**" was released on 24th March 2015. It contains these new features:

### Enhanced user interface in programs Fin 2D and Fin 3D (spring 2016)

Main improvements of user interface are:

- Graphical input of joints and members in "**Fin 3D**"
- Context menus for tables and workspace
- Easier selections in the workspace and tables
- Changes in the tree menu
- Easier editing of joints and members

All improvements are described in a dedicated chapter.

### Figures in documentation

The figures can be added into text documentation in programs "**Fin 2D**" and "**Fin 3D**". These figures are updated regularly and show already the latest results.

### 3D interaction diagram

The software "**Concrete**" contains 3D interaction diagram for cross-sections. The program shows both 3D view and significant planar cuts.

## Column heads

The details of column-slab connection in program "**Punching**" may be reinforced by column heads. Several types are available (step, oblique, general shape).

## Detail with fin plates in "Steel Connections"

Connection of beam to column or girder with the help of fin plate.

## Wind load on canopy roofs

The program "**Loading**" creates the loading reports for wind on canopy roofs.

## Other features

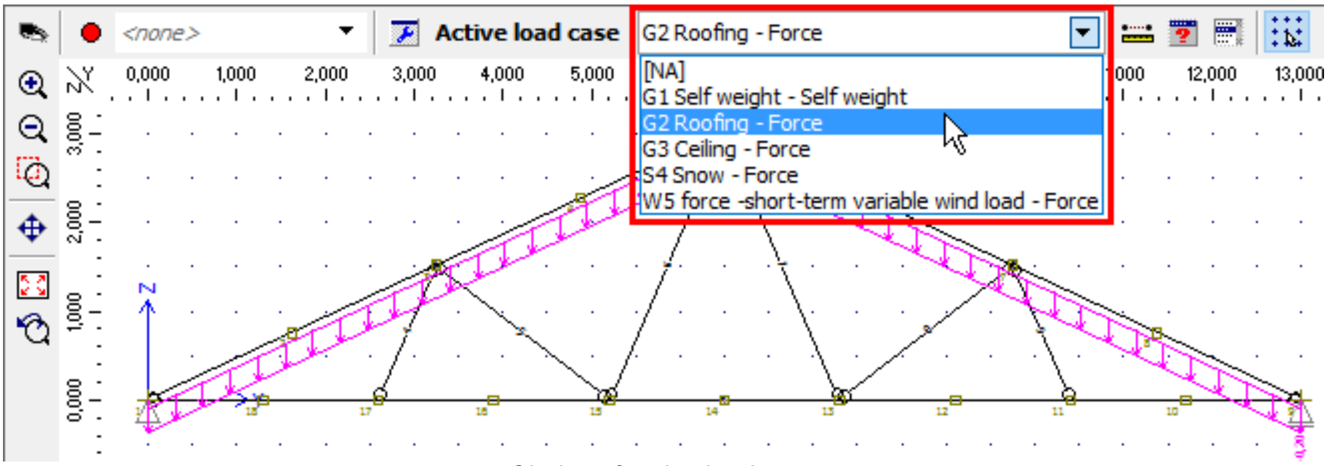
- "**Concrete**" – choice of reinforcement ductility class
- "**Steel**" - general built-up cross-section with 4 L-profiles
- New appearance of programs
- Read-only mode

# User interface in Fin 2D and Fin 3D

These changes in user interface were made:

## Simplification of the tree menu

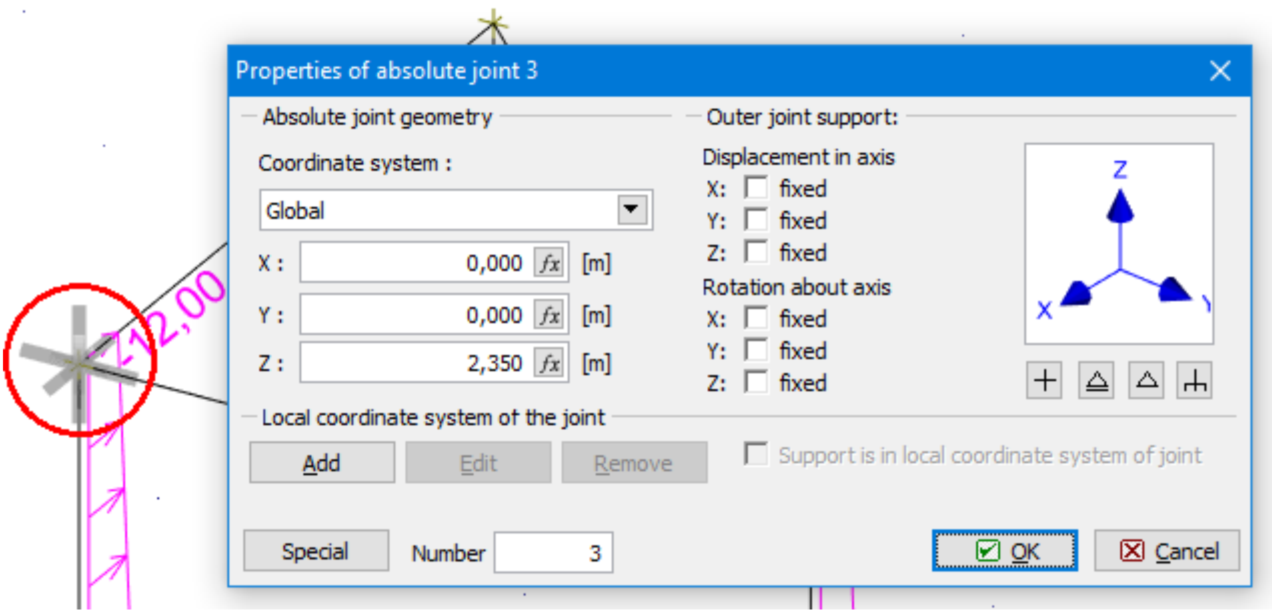
- Choice of active load case was moved to the heading of workspace. The new position corresponds to the selection of results in postprocessor.




*Choice of active load case*

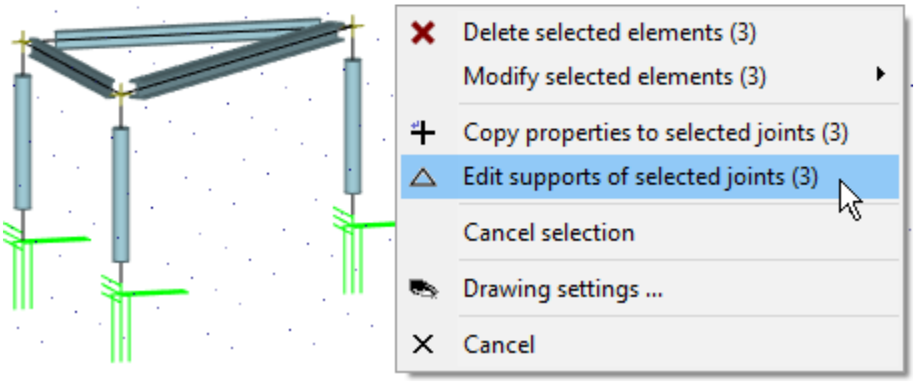
The selection of active load case was also added into the input frame in the mode "**Topology**" of the tree menu.

- Modes "**Edit**" and "**Delete**" can be used both for a work with joints and members. The edited element is highlighted by grey colour in the workspace.



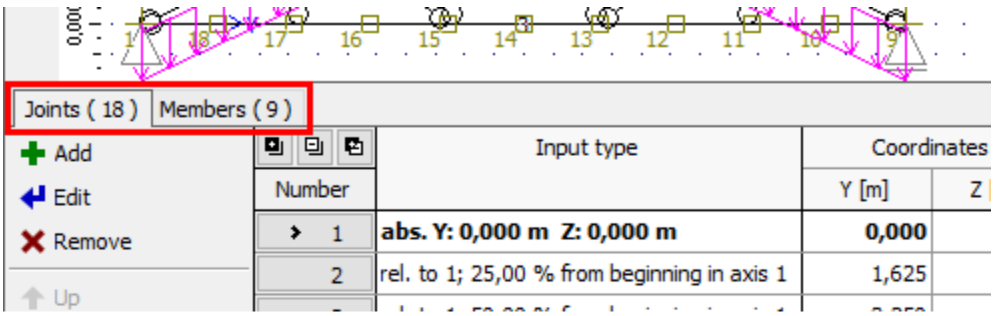
*Highlighted joint in the workspace*

- The commands for the work with selected items (copy properties, edit supports, add loads etc.) are located only in the context menu, that can be opened on the workspace with the help of right mouse button. These tools are available only in cases when some elements are selected (highlighted by green colour in the workspace).




*Context menu for selected joints*

- Tools "**Add scissor joint**", "**Convert joint to absolute**" and "**Divide members**" were moved to the mode "**Tools**" of the tree menu. Alternatively, they are located in context menus for joints and members in the workspace.
- Tables of joint and members are organized into tabs in the bottom frame for the mode "**Topology**". Identical solution is used also for tables of joint and member loads and for several types of combinations in the mode "**Load**".




*Tabs with joints and members tables*

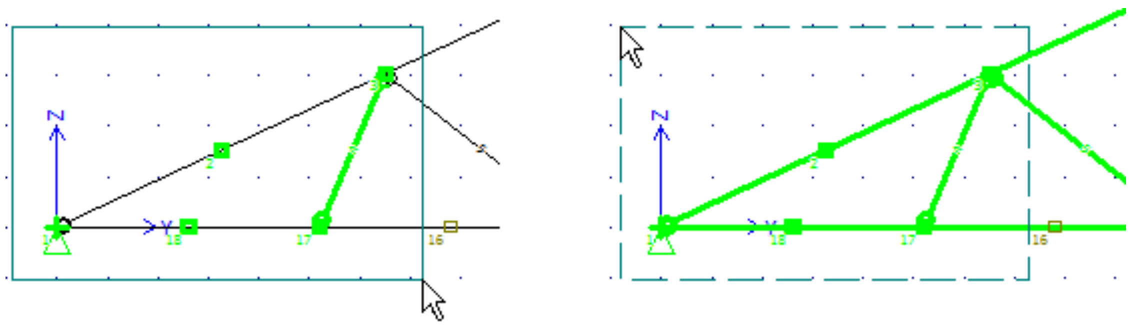
- Preprocessor and postprocessor were merged into one tree menu
- Design members are sorted into tabs according to the type of material in the part "**Design**".

## Easy edit of joints and members

Any member or joint can be edit by double-click on the element in the workspace. This procedure opens corresponding editing window.

## Work with selections

Selection of objects was modified according to the rules used in CAD applications. The selection is done by rectangular area. Dragging from left to right selects objects that are entirely enclosed in rectangle, dragging from right to left selects objects that are crossed by rectangle (the rectangle is drawn by dashed line in this case). The selection can be cancelled completely with the help of the key "**Esc**" or by the command "**Cancel selection**" in the context menu on the workspace. Certain objects can be removed from the selection by repeated selection with pressed key "**Shift**".



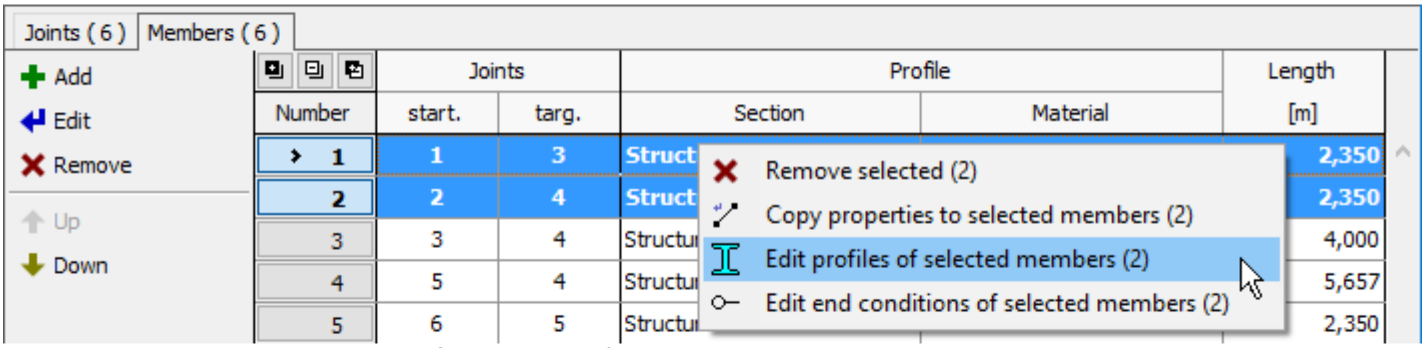
*Difference in the dragging from left to right (left figure) and from right to left (right figure)*

The behaviour may be changed with the help of the setting "**Use Shift to add to selection**", which is located in the tab "**General**" of the window "**Options**". If checked, any new selection cancels previous selection. Adding of new objects to the existing selection can be performed with pressed key "**Shift**".

Also selection in tables was improved. Following shortcuts can be used:

- **"Ctrl"+"A"** - Selects all items in table
- **Selection of range using "Shift"** - Range (more following items) can be selected by clicking on the first and last item. When clicking on the last one, the key "**Shift**" has to be pressed.
- **Addition of items to selection using "Ctrl"** - Items can be added to existing selection with pressed key "**Ctrl**".

Right mouse button click on the selected items in the table opens context menu with tools for the work with selected items.

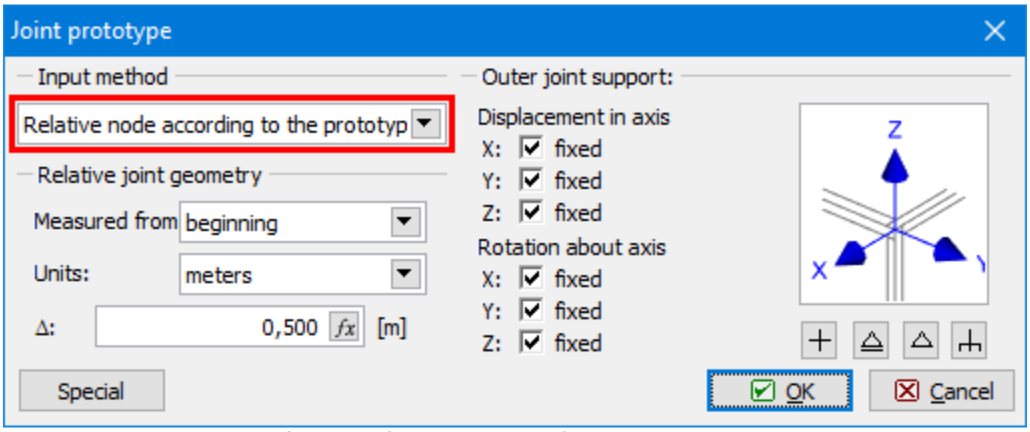



*Context menu for selected members in the table*

## Simplified input of joints

Both absolute and relative joints can be inserted using the same tool "**Add joint**". The joint entered on the existing member is automatically recognized as relative one. Otherwise, it is entered as an absolute one. Both types use the supporting method given in the prototype of joint.

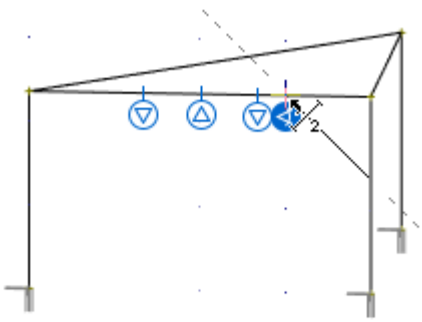
The relative joint can be inserted into the location given by the clicking or the position can be specified parametrically. The setting "**Relative joint according to the prototype**" has to be selected in the prototype properties in these cases.




*Choice of input method for relative joints*

## Graphical input in the program Fin 3D

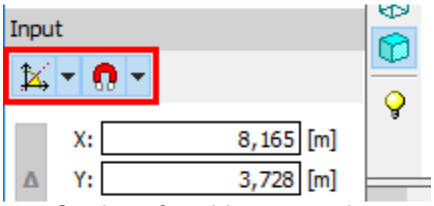
The program "**Fin 3D**" features a new method of graphical input, that corresponds methods common in CAD applications. The range of functions is identical to existing graphical input in the program "**Fin 2D**". It is possible to use snap points and snapping to significant directions.



*Snap points on a member*

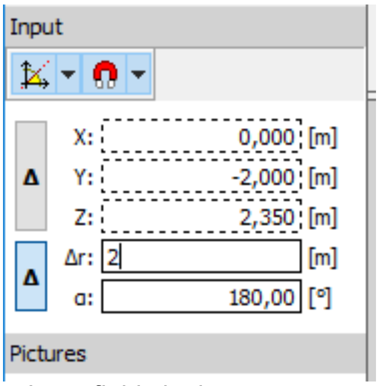
Settings for snap points and snapping to directions are located above the input fields in the bottom part of the tree menu.



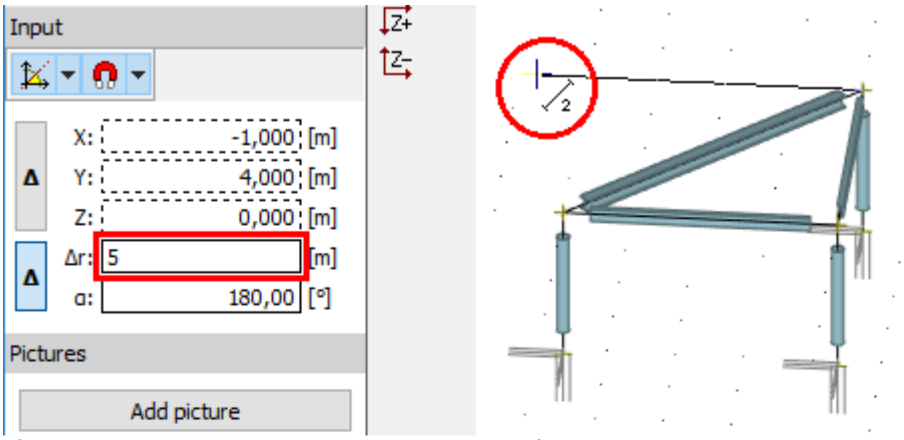
*Settings for object snapping*

These input fields can be used both for the input of joints and members. It is possible to jump into input fields with the help of cursor or by keyboard entries "**x**", "**y**" and "**z**". For example, the expression *x0y2z1.3* fills *0* into the input line "**x**", *2* into the input line "**y**" and *1.3* into the field "**z**".



*Input fields in the tree menu*

The easiest way of entering end point of a member is to specify the member direction by cursor and specify the length on keyboard. The length will be automatically filled into the field "**Δr**". The input has to be confirmed by the key "**Enter**". The program automatically snaps the cursor into directions *45°*. This behaviour may be changed (switch off or change it to *30°*) using the button "".



*Definition of member direction by the cursor and input of member length with the help of keyboard*

Alternatively, it is possible to use extended options of input fields in tree menu. Input fields "**X**" "**Y**" and "**Z**" are usable for the definition of the end point with the help of global coordinates, fields "**r**" and "**α**" define the end point by the member length and rotation about the axis *y*. Buttons "**Δ**" changes whether the input is considered in the global coordinate system or relatively to the beginning of the member. It is possible to jump into input fields with the help of cursor or by keyboard entries "*x*", "*y*", "*z*", "*r*" or "*a*". For example, the expression "*r2a15*" fills *2m* into the field "**r**" and *15°* into the field "**α**".

# Edition 2017

## General improvements

- New window for printing documents
- Autosave option for all programs
- Support of high resolution screens (UHD, 4K)

## Steel, Steel Fire, Timber, Timber Fire

- New function "**Beam**" for comprehensive analysis of horizontal beams with specified loads.

### Fin 2D, Fin 3D

- Display of utilization for all members of the structure
- Merged windows with drawing settings
- Workspace show changes in drawing settings
- National annexes for Austria, Slovenia and Bulgaria
- Improved work with tools for manipulation (move, copy, mirror)
- Extraction of member from design group
- Extended options for input of loads

### Programs for RC structures

- Support of new standards CSN EN 1992-1-1 Z3 and STN EN 1992-1-1 A1/NA

### Concrete Beam

- Improved brief analysis report

### Concrete

- Improved analysis speed

### Loading

- National annexes for Austria, Slovenia and Bulgaria

### Punching

- New details "**Wall end**" a "**Wall corner**"
- General procedure for calculation the factor $\beta$
- User-defined value of the factor $k_{max}$

## Edition 2018

### General improvements

- Improved appearance
- Editable font size in documents
- Import of projects using XML format

### Fin 2D, Fin 3D

- Members with variable cross-sections (haunches)
- Alignment of members on screen
- Easier selection of existing cross-sections and materials

### Designing programs

- Import of forces from clipboard
- New window for import of forces
- Haunches for task types "**Beam**" and "**Member**"
- Input of cross-section in segments (tasks "**Beam**" and "**Member**")

### Punching

- Punching of foundation slab

### Loading

- Wind load for vertical walls in two directions
- Consideration of the lack of correlation of wind pressures for walls

## Edition 2019

### All programs

- BIM support (IFC file format)
- Font selection in protocols
- Improved support of operating systems Windows 7, Windows 8.1, Windows 10

# Edition 2020

## All programs

- The new tool Annotations
- Customizable colour scheme of output documents
- New style of tables with sorting option

## Fin 2D, Fin 3D

- Simplified list of combinations

## Fin 2D

- Redesigned editing tools

## Concrete, Concrete Fire, Concrete Beam, Punching

- National annex for the United Kingdom

## Concrete

- Option to disable minimum eccentricity for plain concrete

## Punching

- Enlarged column footing in a foundation slab

## Steel, Steel Fire

- Welded HSQ cross-sections
- Extended database of cold-formed profiles

## Steel

- Verification of beam deformation for frequent combinations

## Loading

- Wind load on free-standing walls and signboards

# Edition 2021

## All programs

- Italian national annex for Eurocodes
- Italian localization
- Annexes - a new tool for entering text and importing photos or documents into the output protocols
- Import of *.dwg file format
- Printing window - new navigation in the document with the help of tree menu

## Fin 2D, Fin 3D

- Display cross-section name on the workspace

## Verification programs

- new node type "**Internal hinge with support**" for the task type "**Beam**"

## Steel, Steel Fire

- User-defined selection of lateral torsional buckling curve

## Corbel

- Consideration of horizontal force eccentricity in the verification of tensile reinforcement

## Punching

- Option to change the range of considered reinforcement for foundation slabs

# User interface

User interface of Fin EC respects general conventions of Windows applications. Basic work with the software is described in part "**Main application window**". Programs use also these non-standard components:

- Tables
- Tree menu
- Toolbars
- Active dimensions and objects
- Calculator

## System requirements

Fin EC supports these operating systems:

- (Windows XP SP3)
- (Windows Vista)
- Windows 7
- Windows 8.1
- Windows 10

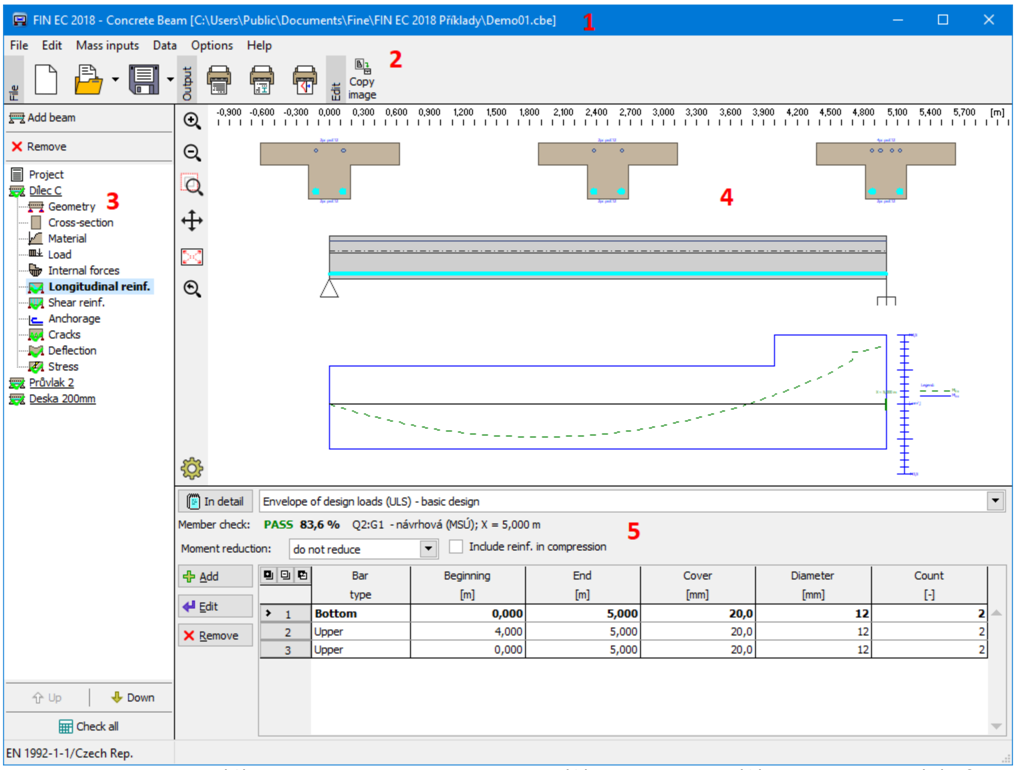
Minimum hardware requirements are:

- display resolution 1024x768
- USB port (for hardware key)

# Main application window

Main application window respects general rules of Windows graphical user interface. It contains these parts:

| | |
|---|---|
| **Window heading** | • Heading contains buttons for minimizing, maximizing and closing the application window. It contains also the name of project file including complete file directory. Star behind the project name signifies, that the project was modified since the last saving of the file. |
| **Main menu** | • The basic menu, that is located at the top of the window. The items in the menu are sorted into several parts. Part "**Data**" is a mirror of the tree menu. Main menu is also accessible using "**Alt**" key on the keyboard. |
| **Toolbars** | • Toolbars with frequently used commands (documentation printing, view adjustments). |
| **Tree menu** | • The main graphical control element, that divides the work into logical parts. |
| **Workspace** | • Graphical display of the project. |
| **Frame** | • Frame is located in the bottom part of the window, it contains control elements for data inputs. |
| **Status bar** | • Status bar is placed along the bottom edge of the application window. Some additional information is displayed there (design standard, cursor position etc.). |

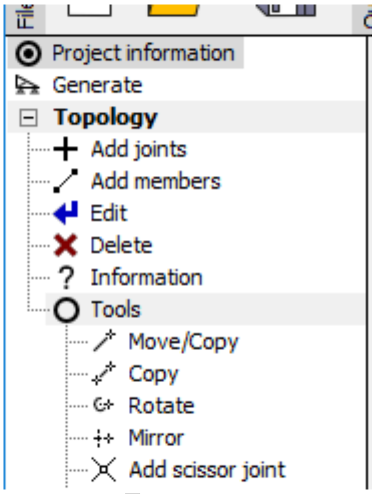


*Main window: Heading (1), main menu with toolbars (2), tree menu (3), workspace (4), frame (5)*

# Tree menu

The tree menu is the main graphical control element of programs Fin EC. It is used for segmentation of the work into logical parts. The order of items in tree menu respects the course of the work, users should start at the top of the tree menu and continue in downward direction. The tree menu is located in the left part of the main application window.

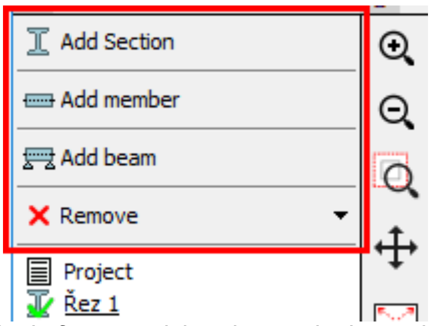



*Tree menu*

Navigation in the tree menu can be done with the help of mouse or keyboard (arrows "**Up**"and "**Down**" for switching between items in the same level, arrows "**left**" and "**right**" for switching from one level to another one). Part "**Data**" of the main menu contains a mirror of the tree menu and can be used as an alternative way for navigation.

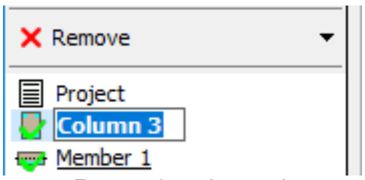
## Dimensioning programs

The tree menu is used also for organizing the particular tasks of the project in the dimensioning programs. The heading contains buttons for manipulation with tasks (add, remove). Button "**Remove**" can be used in more modes (remove active tasks, remove all tasks of the same type, remove all tasks), the mode selection appears after clicking on the black arrow on the right side of the remove button.




*Tools for organizing the particular tasks*

Particular tasks can be renamed by clicking on the task name in the tree menu.




*Renaming the task*

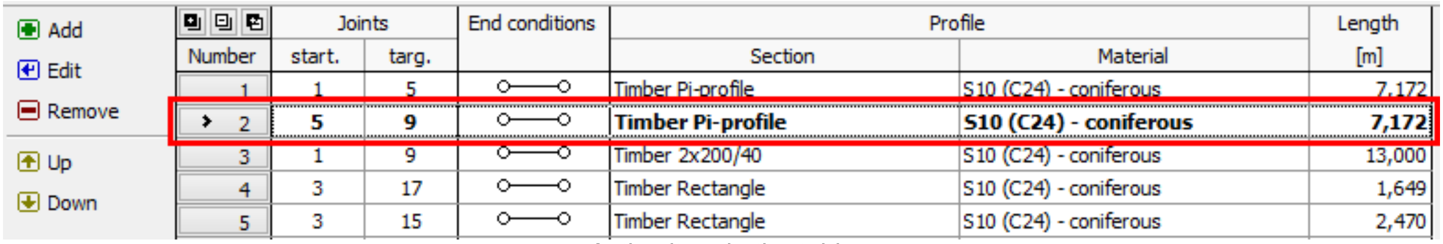
The order of tasks in the tree menu can be changed with the help of buttons "**Up**" and "**Down**" in the bottom part of the tree menu. These buttons are changing the position of the active task. Button "**Check all**" verifies all tasks in a batch.

# Tables

Function of tables is to maintain entered data in a transparent way. Data (items) are organized in rows. Columns display values or properties assigned to the particular items.

Items in tables can be modified using buttons, that are placed in the toolbar on the left side of the table. The certain item shall be set as active before editing or removing it. Active item can be set by clicking the left mouse button on the given row or by using graphical selection in the workspace (only for displayed items). Active item is highlighted by bold font and sign "**>**" before the number of the item (first column of the table).



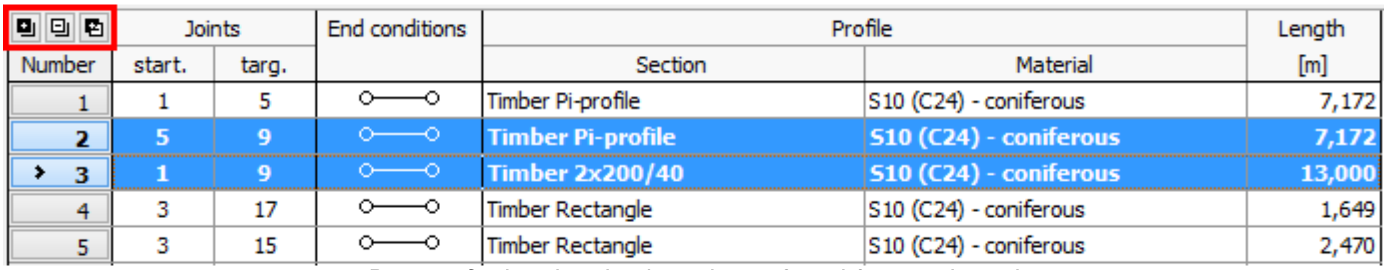

| Number | Joints start. | Joints targ. | End conditions | Profile Section | Profile Material | Length [m] |
|---|---|---|---|---|---|---|
| 1 | 1 | 5 | o——o | Timber Pi-profile | S10 (C24) - coniferous | 7,172 |
| 2 | 5 | 9 | o——o | Timber Pi-profile | S10 (C24) - coniferous | 7,172 |
| 3 | 1 | 9 | o——o | Timber 2x200/40 | S10 (C24) - coniferous | 13,000 |
| 4 | 3 | 17 | o——o | Timber Rectangle | S10 (C24) - coniferous | 1,649 |
| 5 | 3 | 15 | o——o | Timber Rectangle | S10 (C24) - coniferous | 2,470 |

*Active item in the table*

The order of the items in the table can be changed with the help of "**Up**"/"**Down**" buttons in the toolbar in certain cases.

## Items selection

The majority of tables is able to store also status "**selected**" for every item. This status can be used for batch changes (e.g. removing) or for other following operations (printing only for selected items etc). Selected items have blue background in the table. Items can be selected using buttons in the first column of the table (column with the items numbers).



| Number | Joints start. | Joints targ. | End conditions | Profile Section | Profile Material | Length [m] |
|---|---|---|---|---|---|---|
| 1 | 1 | 5 | o——o | Timber Pi-profile | S10 (C24) - coniferous | 7,172 |
| 2 | 5 | 9 | o——o | Timber Pi-profile | S10 (C24) - coniferous | 7,172 |
| 3 | 1 | 9 | o——o | Timber 2x200/40 | S10 (C24) - coniferous | 13,000 |
| 4 | 3 | 17 | o——o | Timber Rectangle | S10 (C24) - coniferous | 1,649 |
| 5 | 3 | 15 | o——o | Timber Rectangle | S10 (C24) - coniferous | 2,470 |

*Buttons for batch selections, items 2 and 3 are selected*

Left upper corner of the table also contains three additional buttons, that can be used for batch selection:

- select all items in the table
- cancel the selection of all selected items
- invert selection (select unselected items and cancel the selection of selected ones)

Status of selected items corresponds to the display of items in the workspace. Active item is highlighted in the workspace. Selected items are coloured in green in the workspace, they turn into red before removing.

# Toolbars

Toolbars improve the accessibility of the most important tools ad functions. Toolbars are usually located in the upper part of the window between the main menu and workspace. Programs Fin EC use these toolbars:

**Files** • Basic functions for the work with the project

**Scale** • Adjustment of the structure view in the workspace

**Selections** • Objects selection in the workspace (only certain programs)

**Edit** • Tools for the work with objects in the workspace (only certain programs)

# Files

Toolbar "**Files**" contains basic functions for the work with the project:


*Toolbar "Files"*

The toolbar contains these buttons:

**New file** • Create new project. If there is any opened project in the program, the window for saving the project will appear before the start of the new project.

**Open file** • Open existing project using standard navigation window. If there is any opened project in the program, the window for saving the project will appear before opening the new project.

**Save** • Save current project into the file. If saving for the first time, the window "**Save as**" will appear. The target directory and file name can be specified in this window.

**Print**
- Open printing window, the window is opened in the mode for printing text documents.

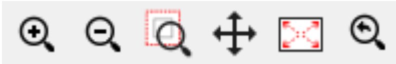
**Print brief document**
- Open printing window, the window is opened in the mode for printing brief documentation.

**Print current view**
- Open printing window, for printing the current view (content of the workspace)

**Copy into clipboard**
- Copy current view into the clipboard. The picture can be inserted from clipboard into graphical editors (Paint etc.) or text editors (Word, Writer) using shortcut "**Ctrl**"+"**V**".

## Scale

Buttons in this toolbar are designated for modifications of the structure view in the workspace (zoom, view move etc).



*Toolbar "Scale"*

The toolbar contains these buttons:

**Zoom in**
- Mode for enlargement of the structure view. The mode can be terminated by clicking the right mouse button.

**Zoom out**
- Mode for reduction of the structure view. The mode can be terminated by clicking the right mouse button.

**View selected region**
- Mode for zooming the selected rectangle of the workspace.

**Move the view**
- Pan tool for moving the structure view. Drag & drop operation.

**View all**
- Optimizes the scale to see complete structure in the workspace.

**Use previous scale**
- Sets the last state of the view (before last zoom).

## Selections

Buttons in this toolbar change the mode of the graphical object selection in the workspace.

*Toolbar "Selections"*

The toolbar contains these buttons:

**Select via clicking**
- Objects are selected or deselected by clicking on the object.

**Select using a line**
- Objects, that are crossed by entered line, are added into (or removed from) the selection.

**Select using a rectangle**
- Objects, that are within the entered rectangle, are added into (or removed from) the selection. The consideration of objects overhanging the borders is based on the setting "**Overhangs in selections**".

**Select using a diamond**
- Objects, that are within the entered parallelogram (diamond), are added into (or removed from) the selection. The consideration of objects overhanging the borders is based on the setting "**Overhangs in selections**".

**Overhangs in selections**
- This setting affects the consideration of the objects that are overhanging the selection area. These objects are included in the selection if this button is pressed. The setting is effective only for selections using rectangle and parallelogram.

**Mode "Add"**
- Mode for selecting the objects. The objects can't be removed from the selection in this mode, change of the mode is necessary first.

**Mode "Remove"**
- Mode for deselecting the objects. The objects can't be added into the selection in this mode, change of the mode is necessary first.

**Mode "Invert"**
- Mode for changing the state of objects. It's possible to convert selected objects into deselected ones and deselected ones into selected ones. First click on the object adds the object into the selection, second click on the same object removes the object from selection

**Select all**
- Add all objects into selection.

**Deselect all**
- Deselect all objects.

| | | |
|---|---|---|
| **Invert all** | • | Convert selected objects into deselected ones and deselected objects into selected ones. |

# Edit

Buttons in this toolbar activate basic modes for the graphical work in the workspace.

*Toolbar "Edit"*

The toolbar contains these buttons:

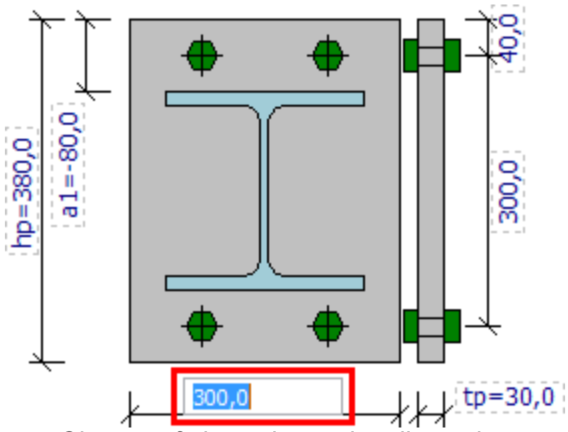
| | | |
|---|---|---|
| **Add** | • | Clicking the left mouse button adds a new objects into the cursor position. |
| **Edit** | • | Mode for editing the object properties. Left mouse button click opens a window with object properties. |
| **Remove** | • | Clicking the left mouse button removes the objects. |

# Active dimensions and objects

Active dimensions and objects make the editing of proportions and properties much easier, as users are able to modify these parameters directly in the workspace.

## Active dimensions

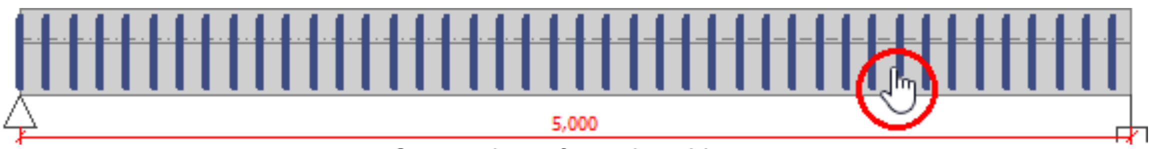
Active dimensions can be used for direct change of proportions in the workspace. Active dimensions are highlighted by the dashed rectangle around the dimension value. The input line for the appropriate dimension appears after clicking on the active dimension. Entered value has to be confirmed by "**Enter**" or by clicking elsewhere in the workspace.




*Change of size using active dimension*

## Active objects

Active objects work in the similar way as the active dimensions. Active objects aren't highlighted in the workspace, however, the appearance of the cursor changes when crossing them. The cursor turns into the "hand" shape. Clicking on the active objects opens a window with properties of the object.

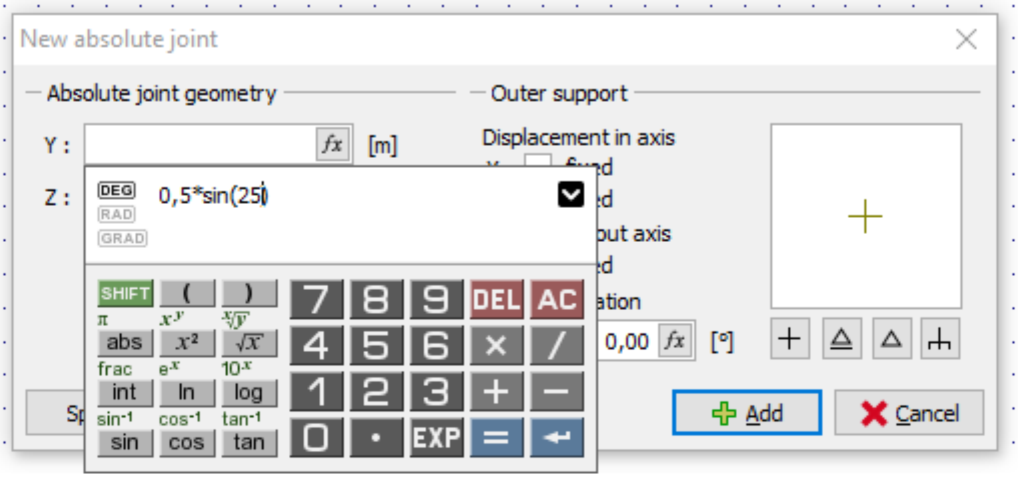



*Cursor shape for active objects*

# Calculator

Certain input lines have extended range of input methods. These lines are indicated by the button "***fx*** " on the right side of the input line. This button launches a calculator window, which allows user to calculate needed input value. The calculated value will be transferred into input line after pressing the button "**Enter**". Simple formulas can be also entered directly into the input line. Example: ***3*sin(30)+5*** . The entered formula is calculated automatically when leaving the input line.




*Input line with calculator*

# Shared parts

# Settings

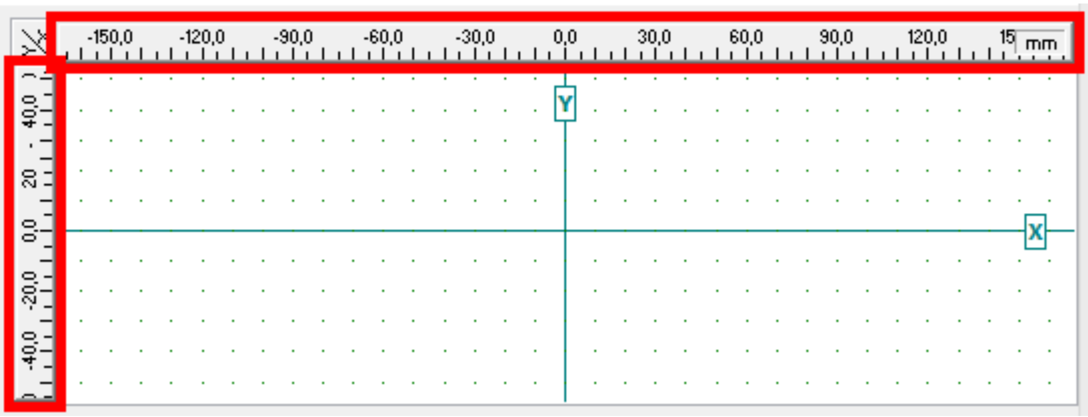
## Options

This window contains settings that may change the appearance and behaviour of the software. Settings are organized into following tabs:

### General

This tab contains properties of grid displayed in the workspace. It's possible to specify the origin and steps both in vertical and horizontal directions. Cursor movements may be limited only to grid points with the help of setting "**Snap to grid**".

There is also an option to switch on/off rulers for the workspace.




*Workspace with rulers*

### Units

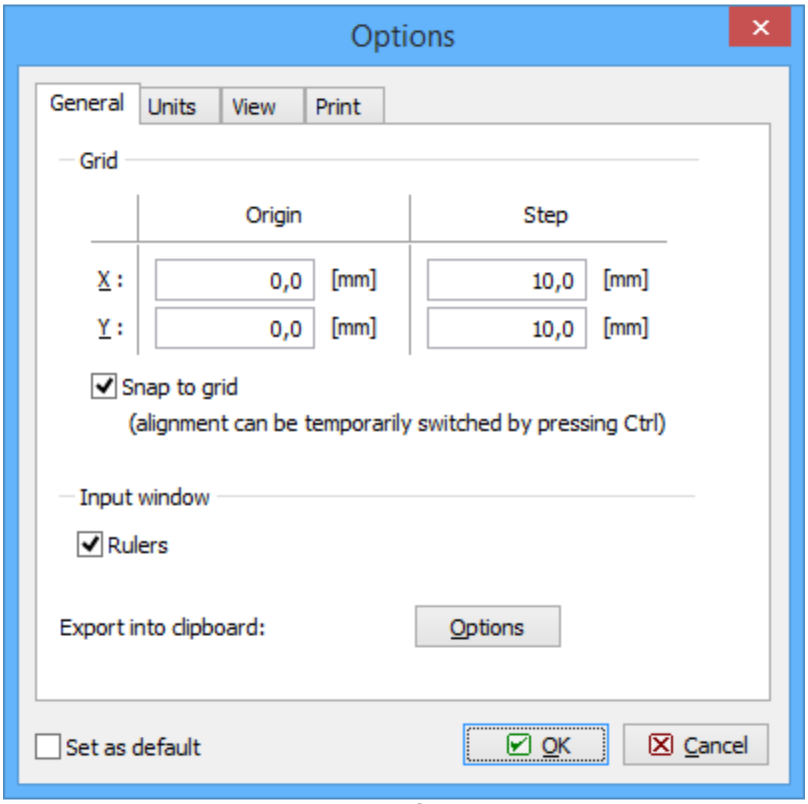
Units and number of decimal places can be specified in this part.

### View and Print

These tabs contains settings for changing the appearance of the workspace and figures in the documentation. These tabs substitutes the window "**Drawing settings**" that is used in other Fin EC programs.

The option "**Set as default**" sets the current settings as default ones for new projects.



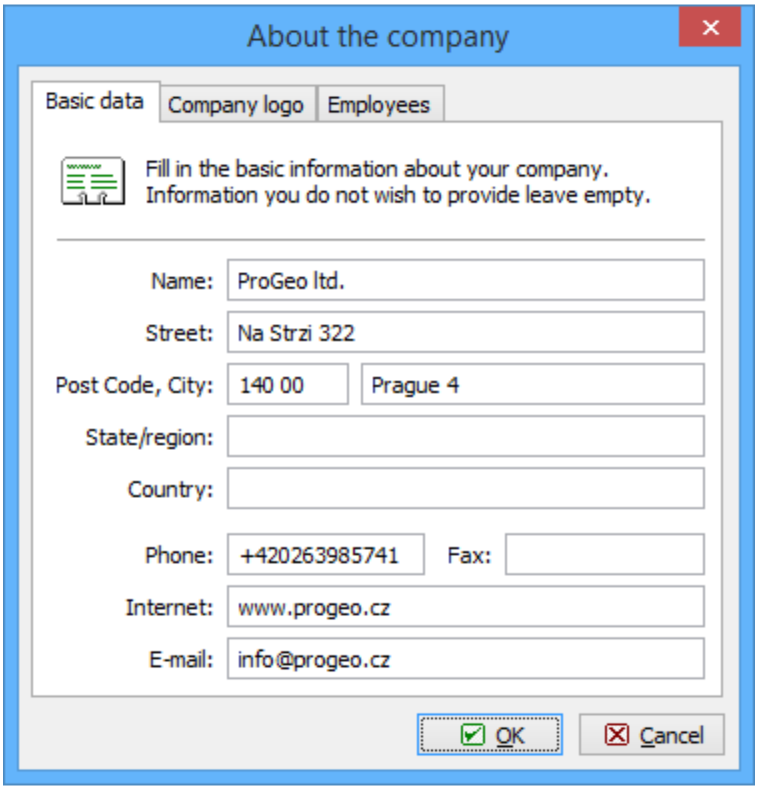

*Window "Options"*

## About the company

This window can be launched using main menu (part "**Options**", item "**Company...**"). Entered data can be used in heading and footing in final documentation. Entries are organized into these tabs:

### Basic data

Tab "**Basic data**" contains main identification and contact data of the company. These data can be used in headings and footings of the documents.




*Tab "Basic data"*

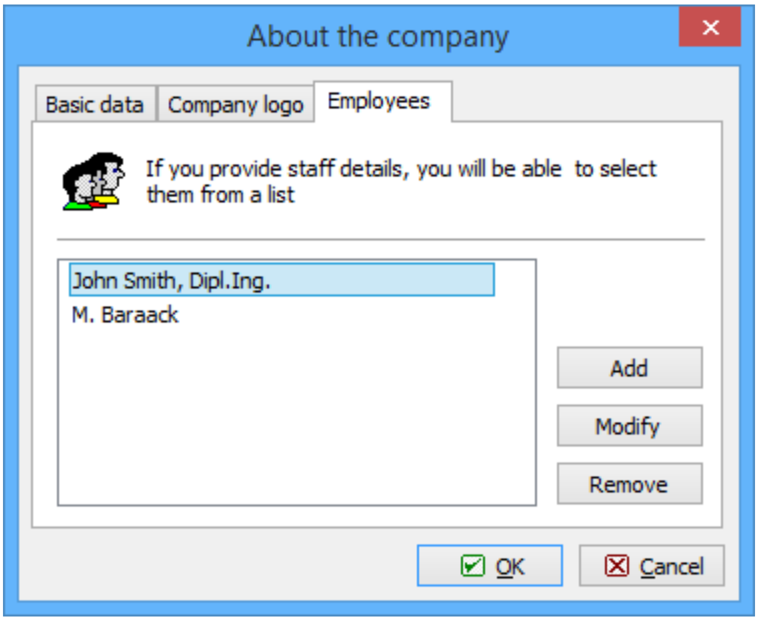
 

## Company logo

The company logo can be loaded in this tab. The window for loading the logo can be launched by the button "**Add**". These picture formats are supported: **.JPG* , **.JPEG* , **.JPE* , **.BMP* , **.ICO* , **.EMF* , **.WMF* .

## Designers

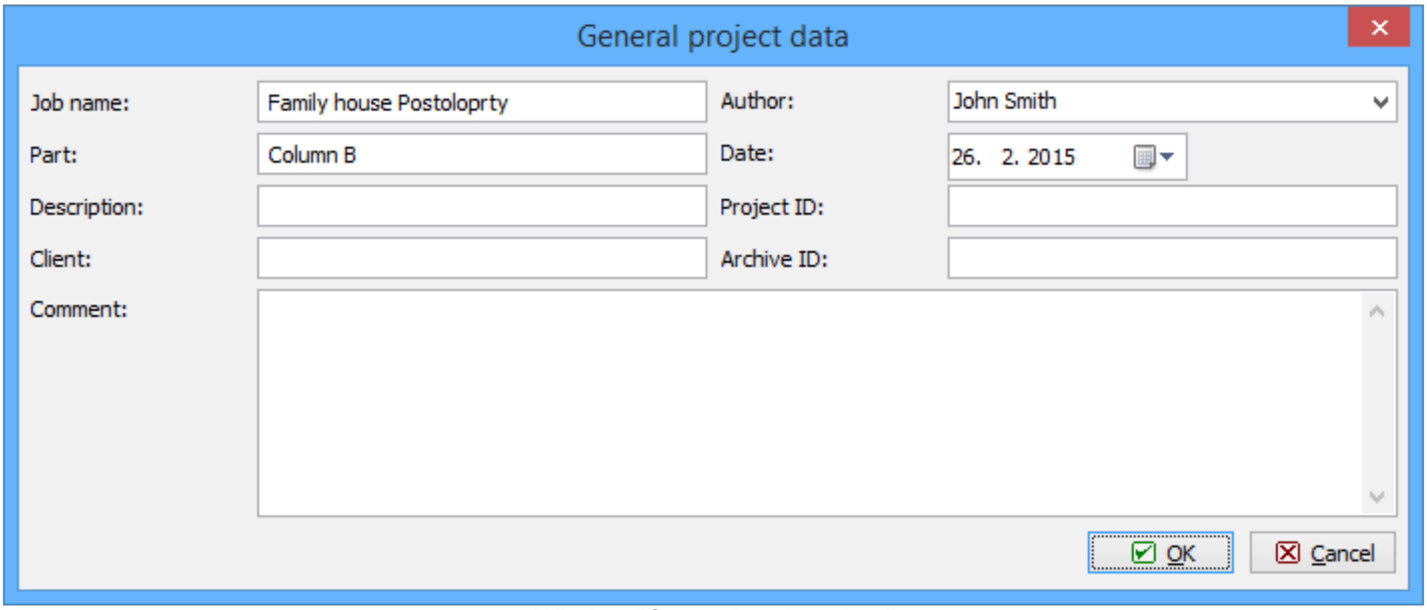
The list of designers can be created in this tab. The names can be used as entry "**Author**" in the window "**General project data**".




*Tab "Designers"*

# General project data

Basic project information can be specified in this window. Inserted data can be used in heading and footing of final documentation. The item "**Author**" can be filled with the help of list of designers, that can be specified in the window "**About the company**".

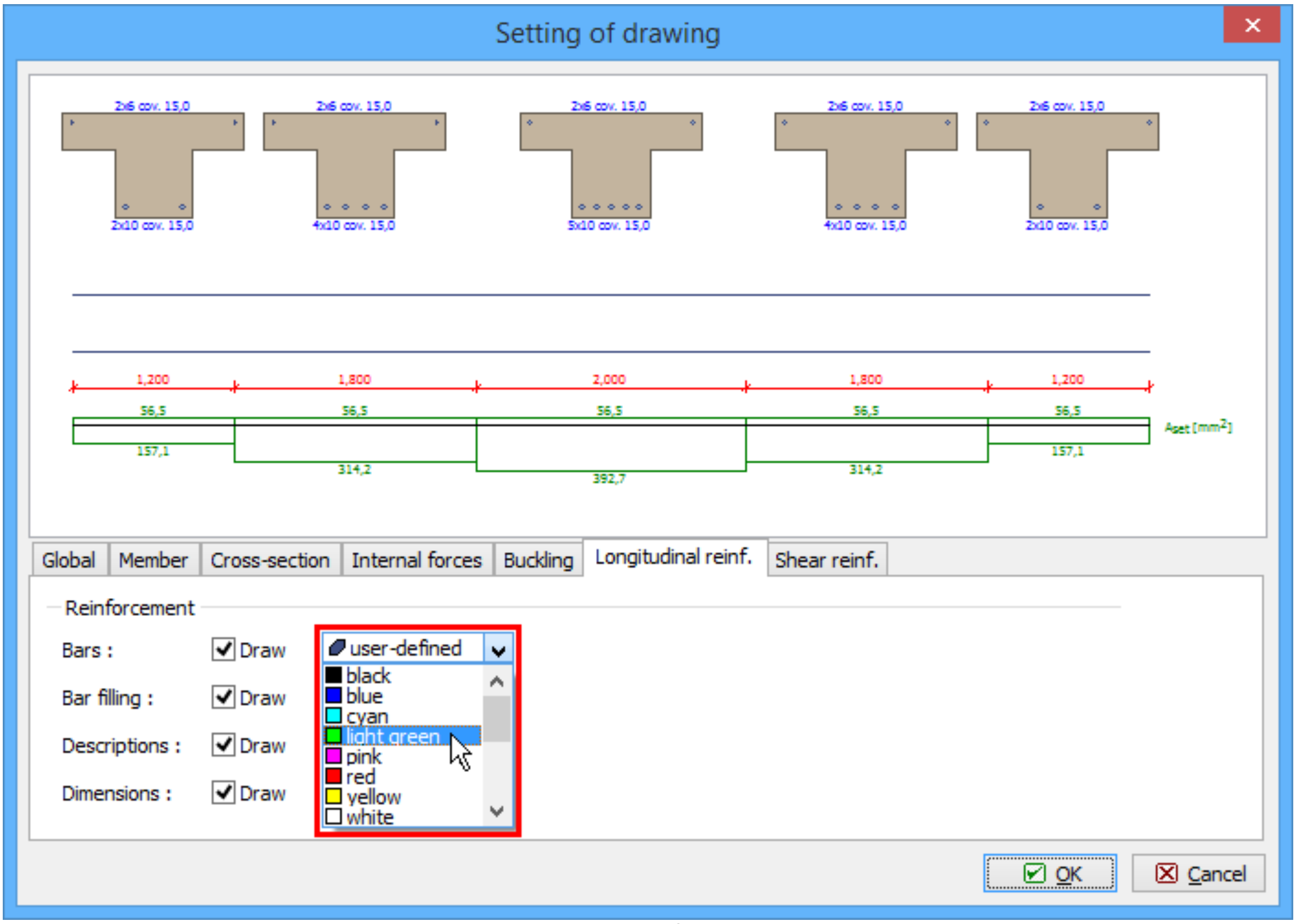



*Window "General project data"*

# Drawing settings

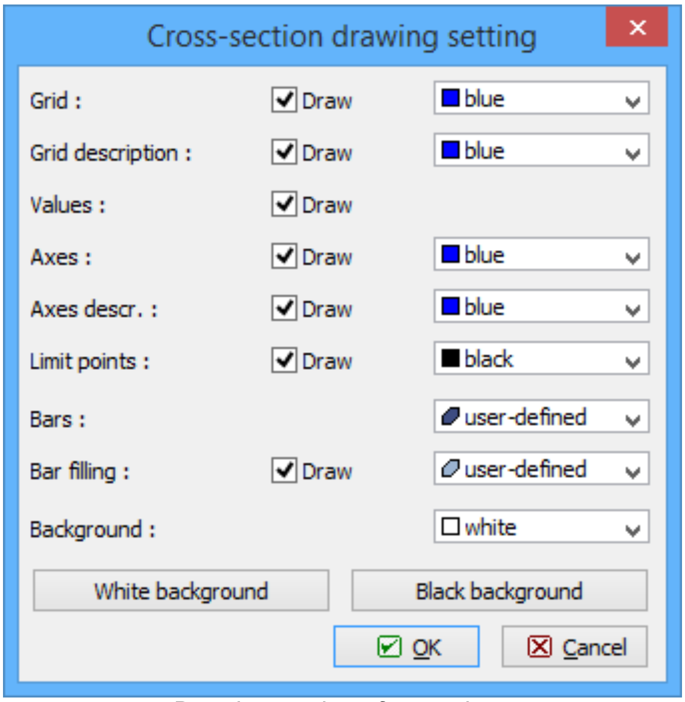
This window contains display settings for the software. These settings can be applied both in the workspace and in the documentation (options "**Drawing settings**" and "**Output drawing settings**"). The window appearance may vary slightly for these two modes. The main part of the window is covered by the preview, that shows the final appearance of the workspace. Bottom part contains options for changing colours of particular objects in the drawing. These options are organized into tabs in accordance with the input structure in the tree menu. Colours can be changed using drop-down menu.

The tab "**Global**" contains two pre-defined colour schemes: "**Black background**" and "**White background**". Font size can be also defined here.




*Drawing settings for members*

The window is simplified for some modes (e.g. for "**Section**" tasks) and some parts (tabs or preview) may be missing. The general work with the settings is identical.




*Drawing settings for sections*

# Fin

# Project information

This window contains following tabs: "**Project**", "**Standard**" and "**Analysis**".

## Project

Basic project information can be specified in this tab. Inserted data can be used in heading and footing of final documentation. The item "**Author**" can be filled with the help of list of designers, that can be specified in the window "**About the company**".

## Standard

This tab contains an option to specify the designing standard including the national annex. The standard is necessary for the determination of partial safety factors and for creation of load combinations. Following standards are supported:

| | |
|---|---|
| **EN 1990** | • The fundamental design standard for Eurocodes. A range of national annexes is available for this standard. Combination factors $\psi_0$, $\psi_1$ and $\psi_2$ can be specified in the window "**Load case edit**". The change of combination factors in the window "**Combination properties**" isn't possible. |
| **CSN 73 0035/Z3** | • The Czech national standard (incl. Amendment *Z3/2006*) that isn't valid any more. The combination factors can be specified in the "**Combination properties**" or in the "**Table of combinations**". As these combinations don't meet requirements of Eurocodes, the verification programs may provide incorrect results. |
| **STN 73 0035** | • The Slovakian national standard that isn't valid any more. The combination factors can be specified in the "**Combination properties**" or in the "**Table of combinations**". As these combinations don't meet requirements of Eurocodes, the verification programs may provide incorrect results. |
| **General** | • The general standard without pre-defined default values of partial factors. The combination factors can be specified in the "**Combination properties**" or in the "**Table of combinations**". As these combinations don't meet requirements of Eurocodes, the verification programs may provide incorrect results. |

The national annex "**Default EC**" creates the combinations according to the fundamental Eurocode without any national annex.

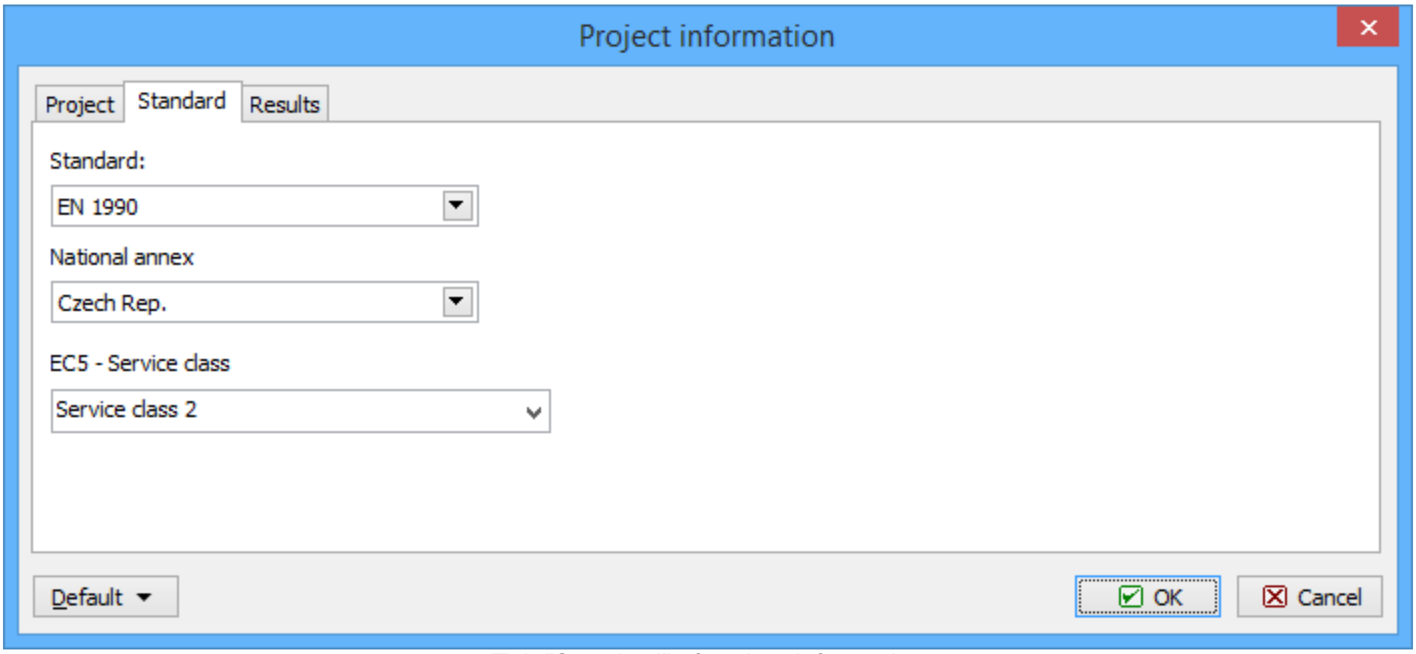
## Analysis

This tab contains options to switch on advanced analysis modules:

| | |
|---|---|
| **2nd order analysis** | • The analysis according to the second order theory (the matrix of initial stresses is added into analysis). The dedicated items "**2nd order combination ULS/SLS**" appear in the tree menu. The analysis is described in the chapter "**2nd order analysis**". |
| **Linear stability** | • The calculation of linear stability. The dedicated item "**Combination linear stability**" appears in the tree menu. The analysis is described in the chapter "**Linear stability**". Only in program "**Fin 3D**". |
| **Dynamics - Eigenmodes** | • The dynamic analysis of eigenmodes. The dedicated item "**Concentrated weights**" appears in the tree menu. The analysis is described in the chapter "**Eigenmodes**". Only in program "**Fin 3D**". |

These modes are disabled as a default, as such types of analysis aren't common part of the design.




*Tab "Standard" of project information*

# Tools

Following tools are available in this part of the main menu:

**Measure distance**
- Measure the distance specified by two points in the workspace. Available also with the help of shortcut *Ctrl+M*. Only in the program "**Fin 2D**".

**Selections**
- The option "**Special selections**" is dedicated for an easy selection of elements according to the defined rules (members with identical cross-sections, joints with specified load etc.). The item "**Saved selections**" opens the window with the list of user defined selections.

**Load cases and combinations**
- The template (list) of load cases and combinations can be loaded or exported in this part. This tool may be used for an easy transfer of load case/combination parameters from one project to another. The file of the template has an extension **.flc*.

**Load**
- This tool increases or reduces the load in load cases with the help of specified multiplication factor. The factor may be applied to all or selected loads, the tool may be limited to active or selected load cases. This tool is suitable e.g. for modification of input after the change of loading width of structural elements.

**Weight and painting surface**
- Calculates the total weight of the structure and the total surface area of all members

**Continuity analysis**
- Test that checks whether the structure is divided into more parts or not. Hidden division into more parts caused by overlapping joints or members causes collapses during analysis very often. The partial segments of the structure aren't usually supported in a sufficient way and the singularity appears in these cases. There is an option to highlight the certain part of the structure.

**Abs. joints on members**
- Test that checks the coordinates of all absolute joints and compare them with positions of members. Absolute joints lying on members aren't connected to the members and may cause the singularity of the structure. Such joints may be converted into relative ones. This conversion automatically creates a connection between member and joint.

**Error list**
- Shows the list of errors that appear during the analysis

# Options

This window contains settings that affect the appearance of the application window and output documentation. It contains three tabs: "**General**", "**Outputs**" and "**Schemes**".

## General

The frame "**Grid**" contains the parameters of the grid displayed in the workspace (beginning and step). This grid can be used for snapping the cursor during the input of structure. The setting "**Snap to grid**" has to be switched on in this case. This behaviour can be switched off (or switched on) temporarily when pressing the key *Ctrl* during the input.

The frame "**Windows**" contains the settings that affect the appearance of the workspace. The colour schemes set the objects colours (background, members, loads, marks etc.). The program contains few pre-defined schemes, new ones may be defined in the tab "**Schemes**". The button "**->**" switches the window into this tab. Both for preprocessor and postprocessor the rulers may be switched on or off.

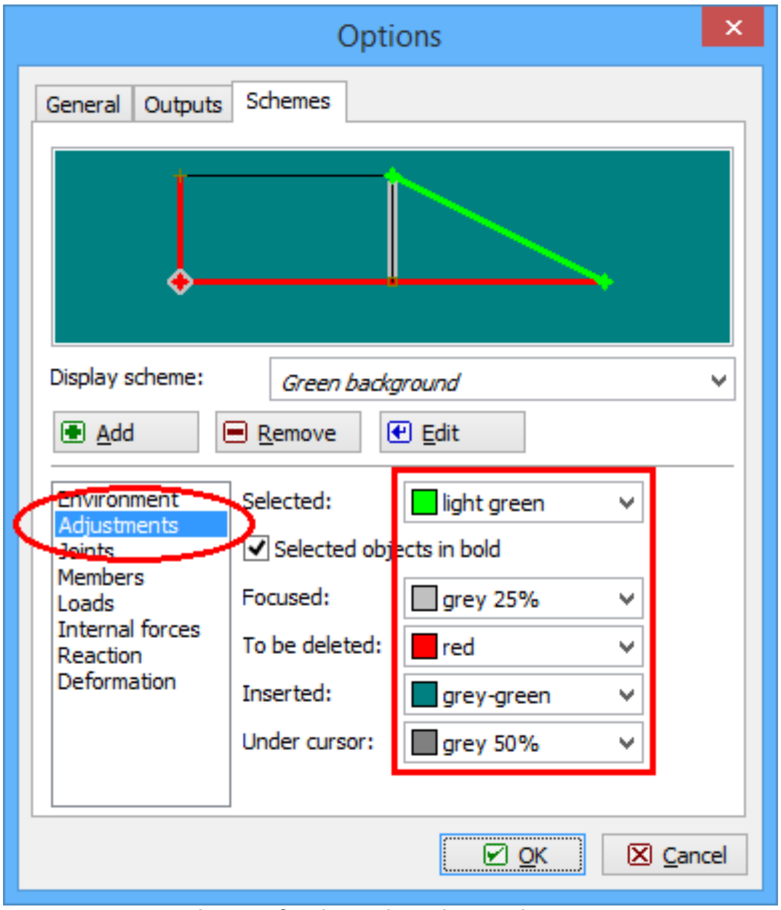
## Outputs

The colour schemes for documentation (the frame "**Print**") and for graphical exports (the frame "**Clipboard**") may be defined in this tab. The colour schemes set the objects colours (background, members, loads, marks etc.). The program contains few pre-defined schemes, new ones may be defined in the tab "**Schemes**". The button "**->**" switches the window into this tab.

The properties of the figure copied into the clipboard (e.g. with the help of the shortcut *Ctrl+C* may be changed here. The properties consist of the figure size, resolution, boundary, heading etc. This option is available only in the program "**Fin 2D**".

## Schemes

The colour schemes can be added and modified in this tab. The colour schemes may be used in the workspace, documents and clipboard. The tools for the work with schemes are available with the help of buttons "**Add**", "**Edit**" and "**Remove**". The active colour scheme (selected in the list "**Display scheme**") is shown in the preview in the upper part of the window. The selection of colours is performed in the bottom part of the window. The colours may be specified only for user defined schemes (highlighted by italics in the list). Pre-defined schemes can't be changed.

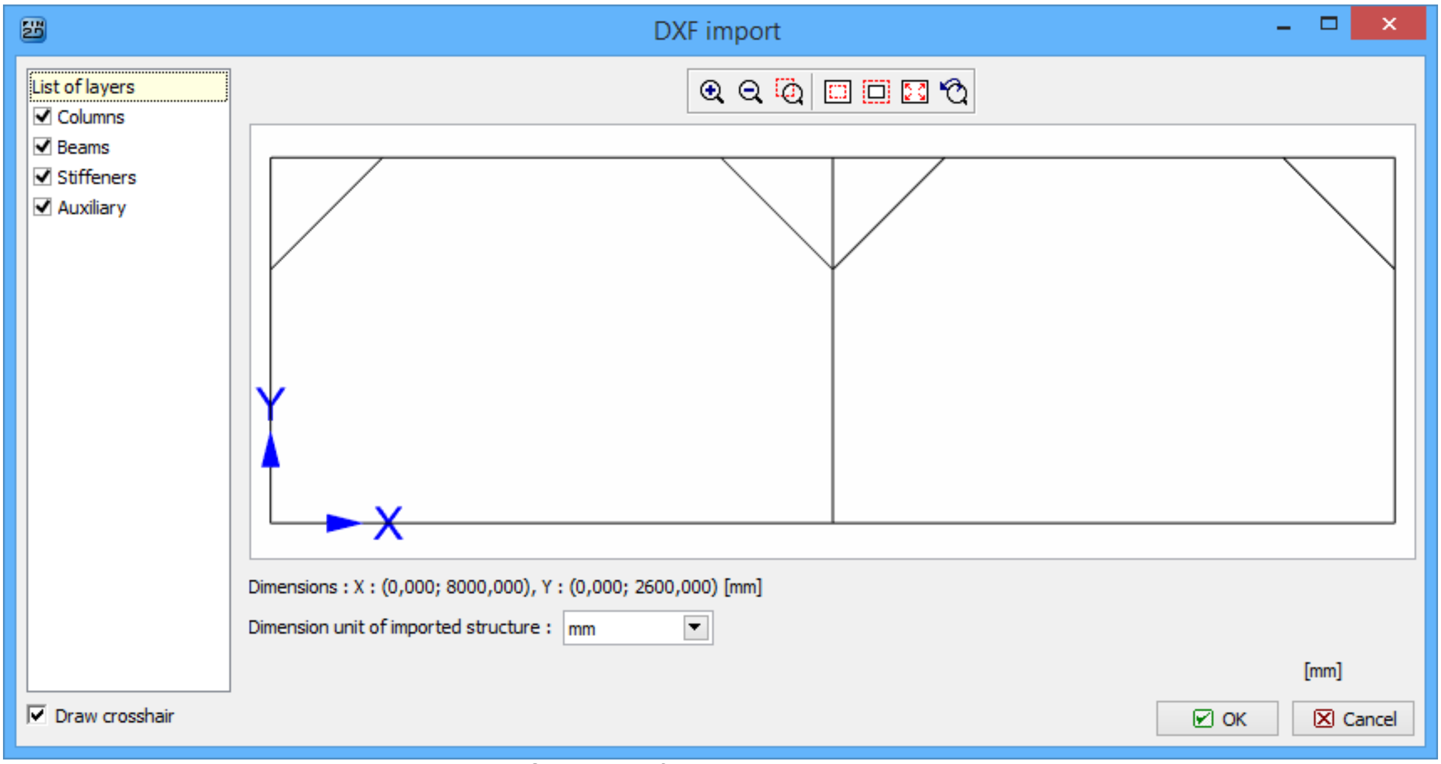



*Input of colours in colour scheme*

# Import of *.dxf file

This window contains the properties of imported **.dxf* file. The left part shows the list of layers. Certain layers may be disabled for import with the help of corresponding check boxes. The length unit of original drawing can be specified in the bottom part of the window. The default selection is done according to the objects dimensions, however, this procedure isn't accurate for large structures.



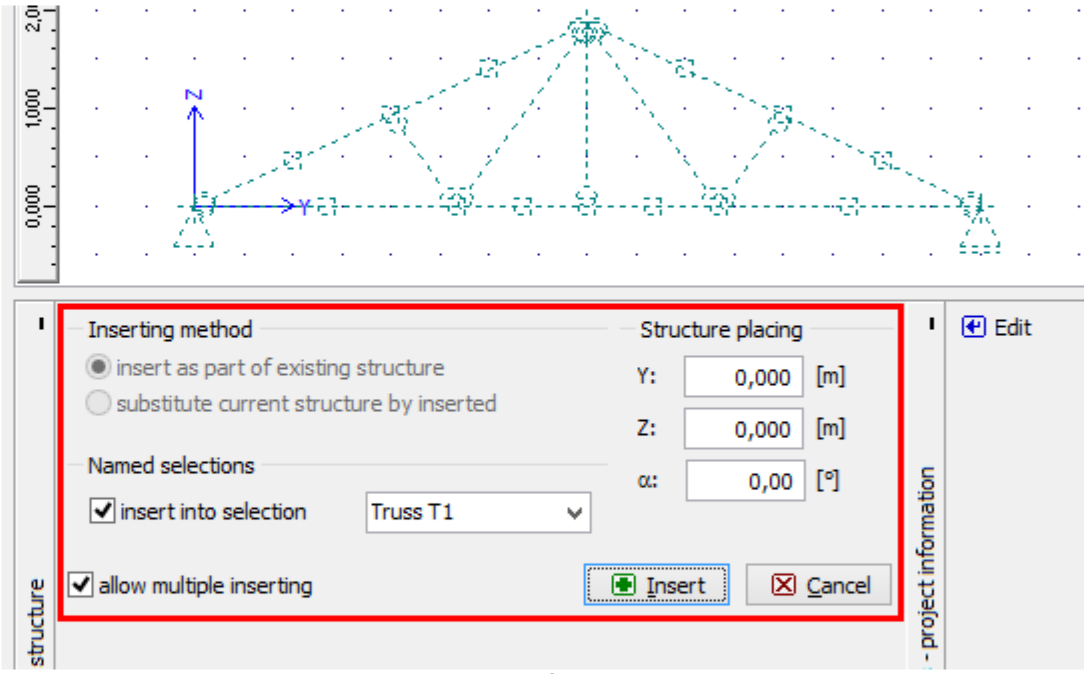

*Selection of layers in the drawing*

## Insertion of structure

When inserting the structure from **.dxf* file or from "**Generator of 2D structures**", the insertion point (coordinates of the left bottom corner of the structure in the workspace) and structure rotation should be specified. The structure may be added to the existing members (the option "**Add to existing structure**" or may replace the existing elements (the option "**Replace existing structure**").

The inserted structure may be also stored as saved selection. The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". The saved selection is beneficial for cases, when the certain operation (copy, change of cross-sections, addition of load) should be applied to all elements of inserted structure. The check box "**Insert into selection**" is able to create such saved selection.

The setting "**Allow multiple inserting**" gives the option to insert the structure more times into different parts of the workspace.

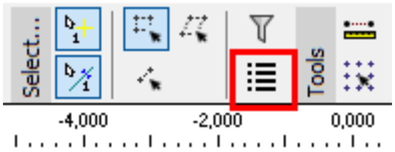



*Insertion of structure*

# Saved selections

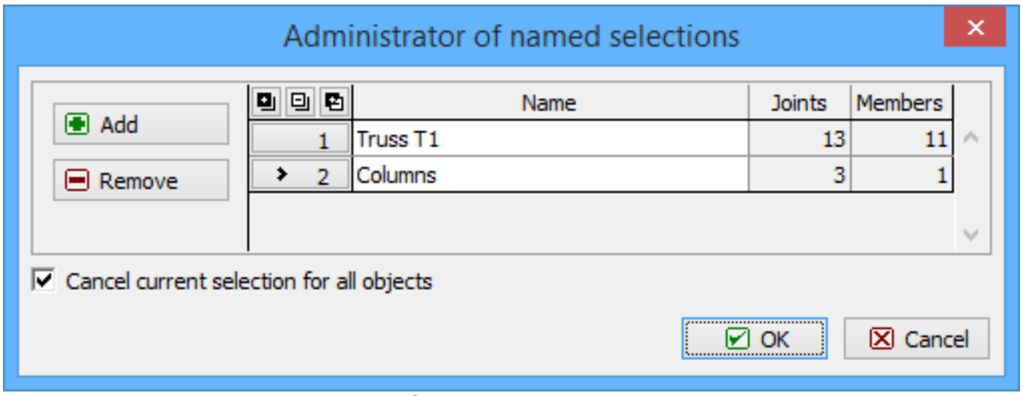
The "**Saved selections**" are suitable for the repetitive selection of joints and members (e.g. for application of certain operation). The list of selected items is saved in the table using the unique name. The active selection (highlighted by the mark "**>**" in the first column) is applied in the workspace after closing the window with the help of the button "**OK**". The window "**Saved selections manager**" may be launched from the main menu (part "**Tools**" - "**Selections**") or with the help of the button in the toolbar "**Selections**".




*Button for saved selections*

## Addition of saved selection

The input of new selection has to start with the selection of elements in the workspace. After that, the "**Saved selections manager**" can be launched. The selection will be saved with the help of the button "**Add**".




*Window "Saved selections manager"*

# Special selections

"**Special selections**" are dedicated for the fast selection of structural elements (joints or members) with the specified properties. The window may be launched from the main menu ("**Tools**" - "**Selections**"), using the shortcut *Ctrl+L* or the button "" in the toolbar "**Selections**". Following parameters may be used for the definition of selections:

**Member selections**

**Loaded in current load case**
- The selection of all members that are loaded in the active load case. This option may be used for the edit or removal of member loads.

**Unloaded in current load case**
- The selection of all members that aren't loaded in the active load case. This option may be used for the input of member loads.

**Identical profiles**
- The selection of all members with the identical member profile (end conditions, cross-section, member type). This option may be used for the batch edit of end conditions, member type etc.

**Identical sections**
- The selection of all members with the identical cross-section. This option may be used for the batch edit of cross-section.

**Identical materials**
- The selection of all members with the identical material.

**Identical types**
- The selection of all members with the specified type ("**Beam**" or "**Beam on elastic subsoil**").

**For selected joints**
- The selection of all members that begin or ends in the selected joints.

**Between selected joints**
- The selection of all members that begin and ends in the selected joints.

**Joint selections**

**Loaded in current load case**
- The selection of all joints that are loaded in the active load case. This option may be used for the edit or removal of joint loads.

**Unloaded in current load case**
- The selection of all joints that aren't loaded in the active load case. This option may be used for the input of joint loads.

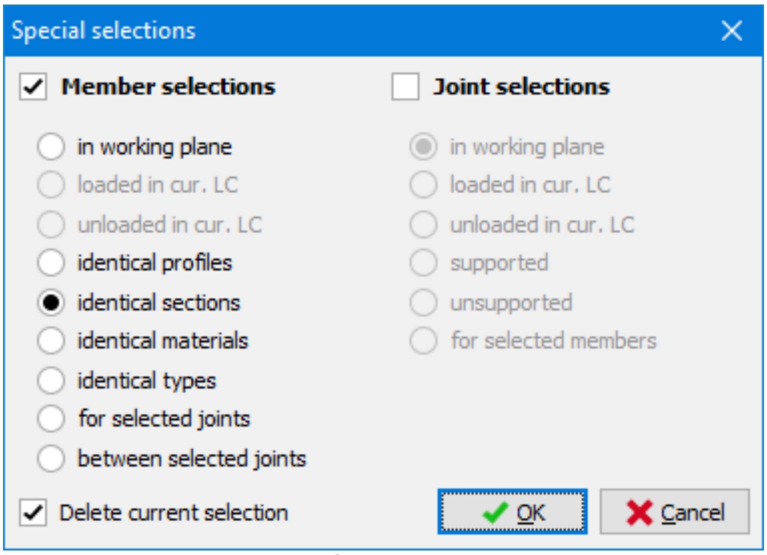
**Supported**
- The selection of all joints that are supported. This selection may be used for edit or removal of supports.

**Unsupported**
- The selection of all joints that aren't supported. This selection may be used for input of supports.

**For selected members** • The selection of all joints that are lying on the selected members.

The setting "**Delete current selection**" turns all selected items from previous selection into unselected ones.




*Window "Special selections"*

# Edit profile

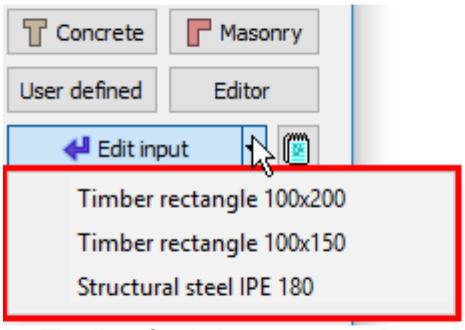
This window contains the parameters of the member cross-section and material.

The cross-section may be specified using three different methods:

- The cross-section can be selected from the pre-defined database with the help of dedicated buttons "**Steel**", "**Timber**", "**Concrete**", "**Masonry**". The database for any material contains different shapes of cross-sections (rectangle, T-shape etc.), the input is done in the window "**Cross-section editor**".
- The shape of the cross-section may be created with the help of the external programs "**Section**" and "**Sector**". These programs calculates all needed cross-sectional characteristics and transfer them into the analysis. This input method is available after clicking on the button "**Editor**".
- All cross-sectional characteristics may be specified numerically in the dedicated window, that can be launched by the button "**User defined**".

The existing cross-section can be changed easily by the button "**Edit input**" or by clicking on the cross-section view. The window "**Cross-section editor**" opens in both cases.

The drop-down list on the right side of the button "**Edit input**" can be used for fast selection of existing cross-section, as this list contains all cross-sections already defined in the structure.




*The list of existing cross-sections*

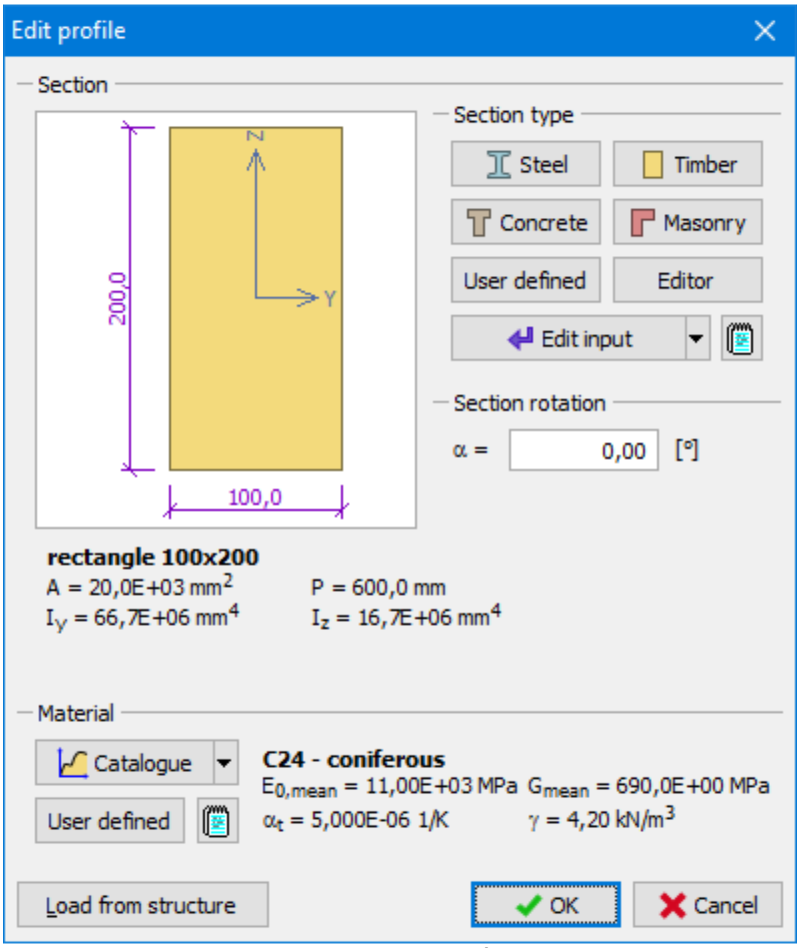
The input field "**Section rotation**" is able to rotate the cross-section about the member axis. This option is beneficial for the design of members, that are loaded in the direction, which isn't parallel to the main axes (e.g. purlins).

Material can be specified with the help of the pre-defined database in the window "**Materials catalogue**" (button "**Catalogue**") or can be specified numerically (button "**User defined**").

The cross-section and material may be copied from the existing member using the button "**Load from structure**".

The cross-sectional characteristics are described in the chapter "**Cross-sectional characteristics**".

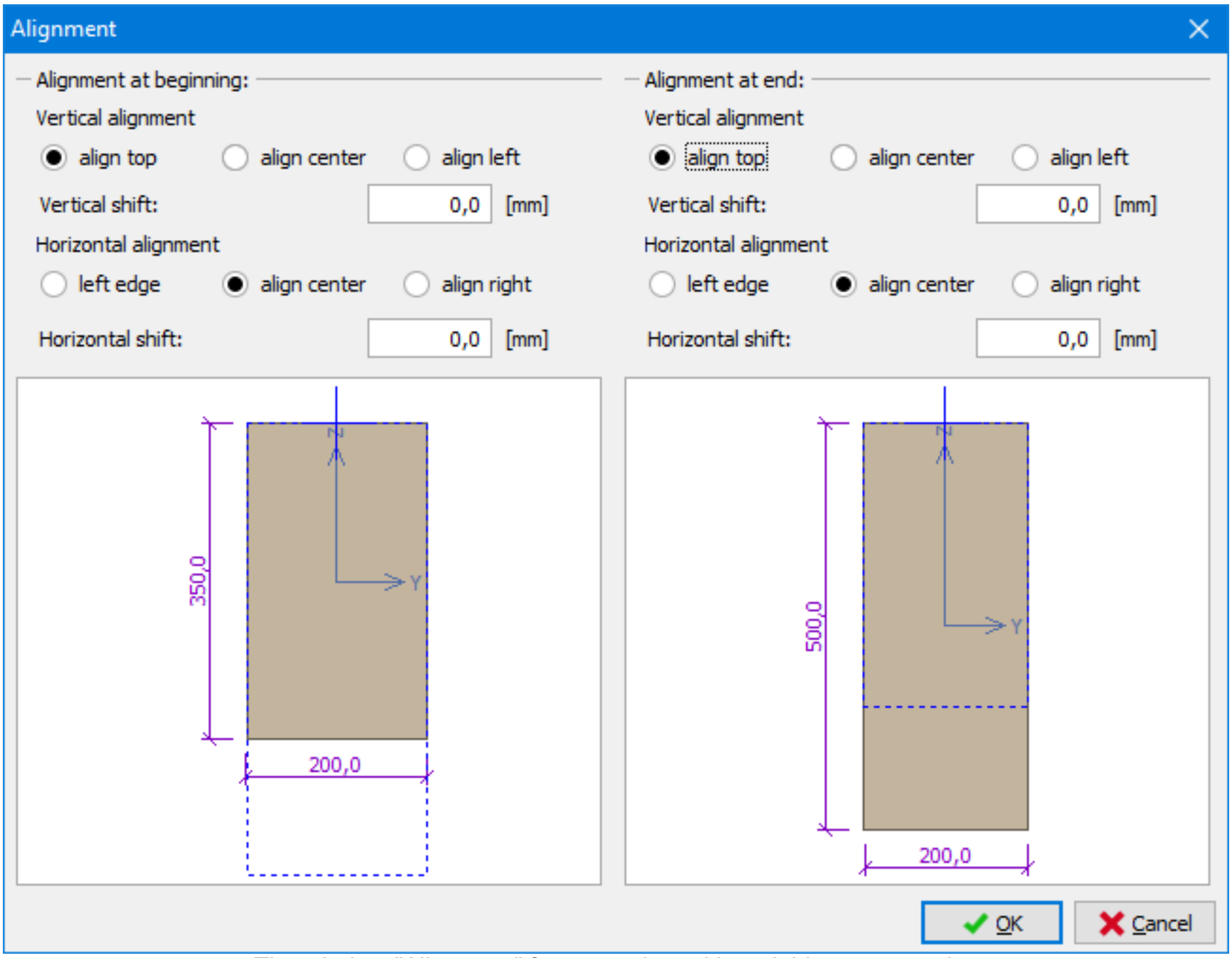


*Window "Edit profile"*

## Alignment

This window contains an option to specify the position of member mass relatively to reference line of the member. This alignment is used only visual appearance of the structure, it does not affect the stiffness matrix during analysis.

Both vertical and horizontal alignment can be specified for the cross-section. Members with variable cross-section (tapers) require input of alignment both at the beginning and end of the member.



*The window "Alignment" for a member with variable cross-section*

## Drawing settings

The window "**Drawing settings**" contains the settings that affect the display of the structure in the workspace. The window contains two tabs in the upper part: "**Structure topology and loading**" and "**Results**" (post-processor only). The bottom part contains settings, which affects both display of structure and results:

| | |
|---|---|
| **Load, internal forces and deformation only on selected members** | • Displays loads, internal forces and deformations only for selected members (highlighted by green in the workspace). |
| **Show units for load, internal force and deformation values** | • The values of loads, forces and deformations are displayed including corresponding units |
| **Correction of display size for load, forces and schematic deformations** | • Modifies the size of displayed diagrams. This setting is suitable for cases, when the default view doesn't provide readable results. |
| **Text size** | • Modifies the font size in the workspace |
| **Size of support symbols** | • Modifies the size of support marks and symbols of end conditions for members in the workspace |

### Structure topology and loading

Part "**Common**" contains following parameters:

| | |
|---|---|
| **Global coordinate system** | • Shows the axes of the coordinate system in it's origin |
| **User coordinate systems** | • Shows the axes of the coordinate system in it's origin |
| **Draw grid** | • Shows the snapping grid for the input of members and joints. The properties of the grid are located in the window "**Options**". |

Next group of settings relates to joints and members:

| | |
|---|---|
| **Numbering** | • Shows the numbers of joints |
| **Symbols** | • Shows the symbols of joints, that differentiate the absolute and relative joints |
| **Supports** | • Shows the support marks |
| **Numbering** | • Shows the numbers of members |

**Local axes** • Shows the local coordinate systems of members. The orientation of local axes is important for the input of load.

**End conditions** • Shows the style of connections to the start and end joints of members. There are dedicated symbols for pinned and special (e.g. spring) connections. The fixed connection isn't highlighted by any symbol.

**Beginnings** • The beginning of the member is highlighted by the arrow. This information is helpful for the input of entities like relative joints or loads.

**Sections** • Shows the cross-sections of members

**Equalize cross-sections (Fin 2D only)** • Equalizes the size of displayed cross-sections. This setting is helpful in structures with significant differences in cross-sections dimensions (e.g. combination of thin steel and massive concrete cross-sections), as some cross-sections wouldn't be recognizable.

**Scale (Fin 2D only)** • This value is used as a size multiple for displayed cross-sections.

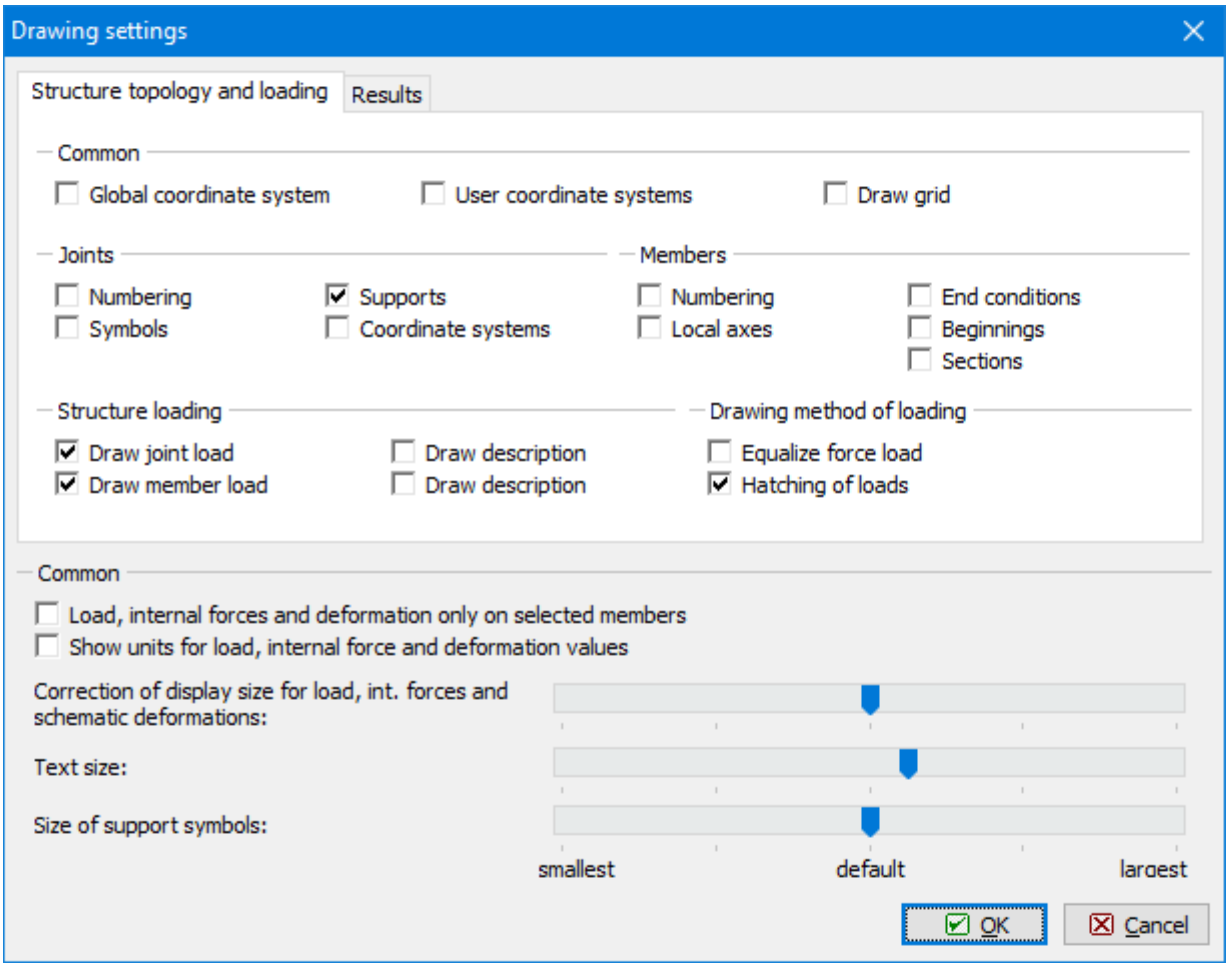
Last part affects the display of loads:

**Draw joint load** • Shows joint loads

**Draw member load** • Shows member loads

**Equalize force load** • The loads can be equalized using this setting. This is helpful in structures with significant differences in load values, as some loads wouldn't be recognizable.

**Hatching of loads** • This setting fills the displayed load with vertical hatching including orientation arrows.




*Window "Drawing settings"*

## Results

This tab is available only for the part "**Results**" of the tree menu (post-processor). The displayed quantities can be switched on or off with the help of corresponding check boxes here. The description style can be selected for every quantity:

**No description** • Shows only diagram without any displayed values

**Describe** • Shows also values of corresponding quantity

**Highlight maxima** • Shows diagram and values of corresponding quantity. The member, which shows the extreme value of the quantity, is highlighted.

The internal forces can be displayed relatively to the member or cross-sectional coordinate system:

| | |
|---|---|
| **Internal forces in member coordinate system** | • The internal forces are displayed in the local coordinate systems of members (axes described $2$ , $3$ ), without any consideration of cross-section rotation. The gravity load causes only bending moment $M_2$ in these cases. These values are suitable for checking the global behaviour of the structure. |
| **Internal forces in coordinate system of cross-section** | • The internal forces are displayed in the local coordinate systems of cross-sections (axes described $y$ , $z$ ), which respect the rotation of the cross-section. The gravity load is divided into two components (e.g. moments $M_y$ and $M_z$ ) in these cases. These values are important for the analysis of particular members. |

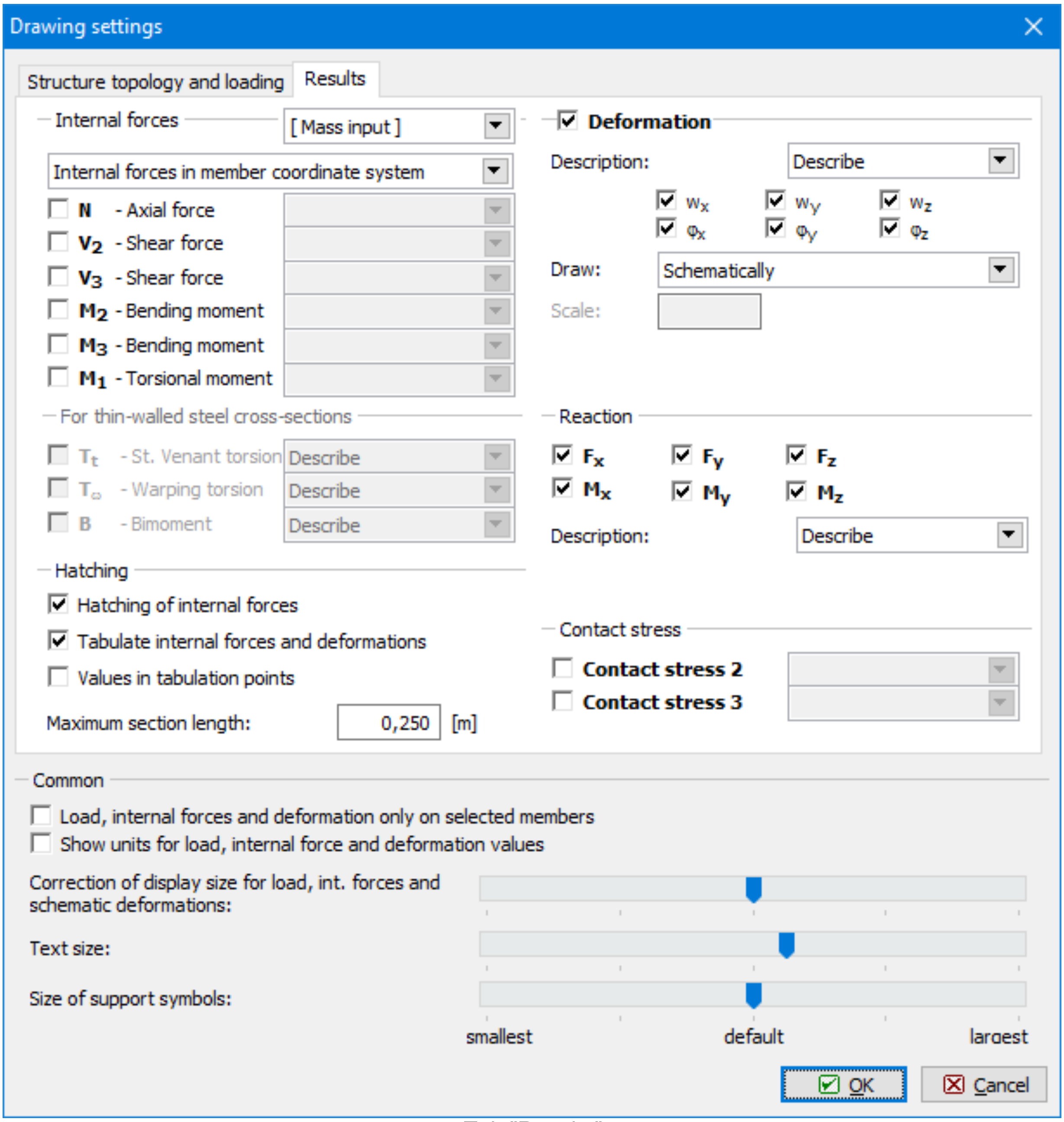
Deformations can be displayed using defined scale (option "**Scale**") or the scale can be select by the software automatically. This way provides clear view in all cases (option "**Schematically**").

Part "**Hatching**" contains following settings:

| | |
|---|---|
| **Hatching of internal forces** | • Fills the displayed diagrams of internal forces with vertical hatching |
| **Tabulate int. forces and deformations** | • Shows the values of internal forces along diagrams (between maximum values), the maximum distance between two values may be also specified |
| **Drawing method of deformations** | • The option "**Schematically**" displays deformations are displayed with respect to the readability of results, no scale is used. The option "**Scale**" displays deformations in given scale. |

The bottom part of the window contains shared settings, which are described in the chapter "**Drawing settings**".

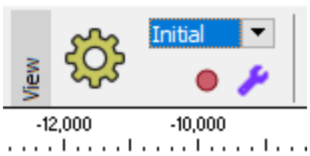
 




*Tab "Results"*

## View manager

The style of drawing in the workspace and list of displayed quantities can be saved as a template with the help of the "**View manager**". Such template may be restored again. The templates store settings from the window "**Drawing settings**". The workspace contains dedicated toolbar for the work with view templates.

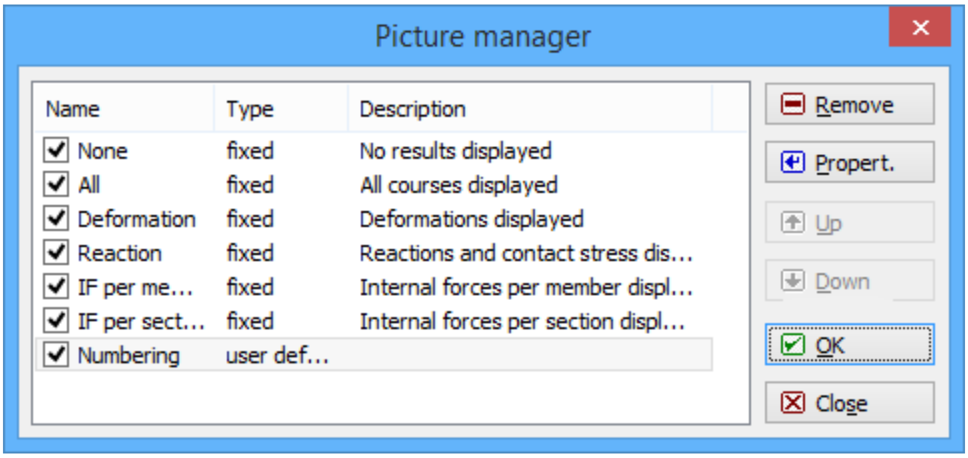



*Toolbar for the management of view templates*

The toolbar contains following buttons:


- Saves a new template


- Runs the window "**View manager**". This window contains tools for editing or deleting view templates.

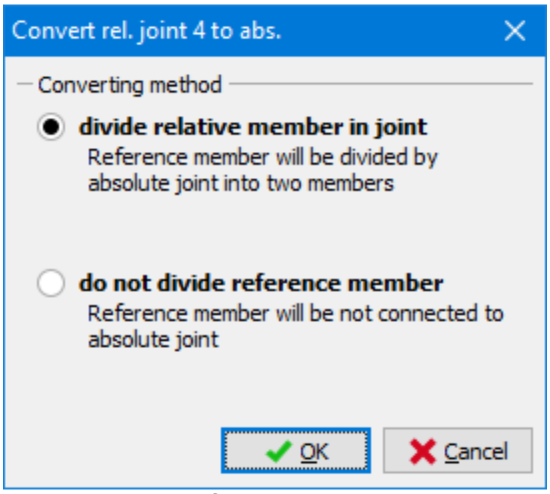



*Window "View manager"*

## Conversion of relative joint to absolute

This tool is suitable for the conversion of relative joint into absolute one. After this conversion, the position of this joint may be changed in any direction (position of the relative joint may be changed only in the direction of the reference member). This tool can be activated by the link "**Convert to abs.**" in the part "**Joints**" of the tree menu. After that, conversion is done by clicking on the joint in the workspace. There are two ways, how to convert the joint:

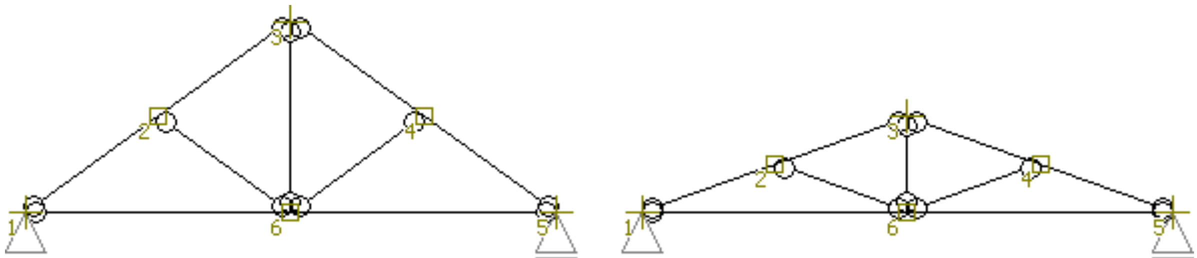
- **Divide reference member in joint** - the reference member will be divided into two parts, the joint will be the reference joint of both parts
- **Do not divide reference member** - the reference member remains untouched, the converted joint won't be connected to the member




*Window "Convert relative joint"*

### Brief example of the conversion

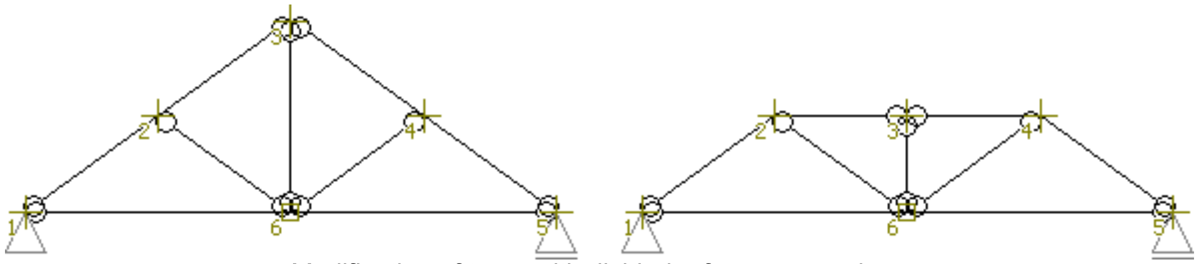
In the first case, joints *2* and *4* are the relative joints placed on top chords. Their position is automatically changed after the change of *Z*-coordinate of the apex. The truss is still duo-pitched.




*Modification of truss with relative joints*

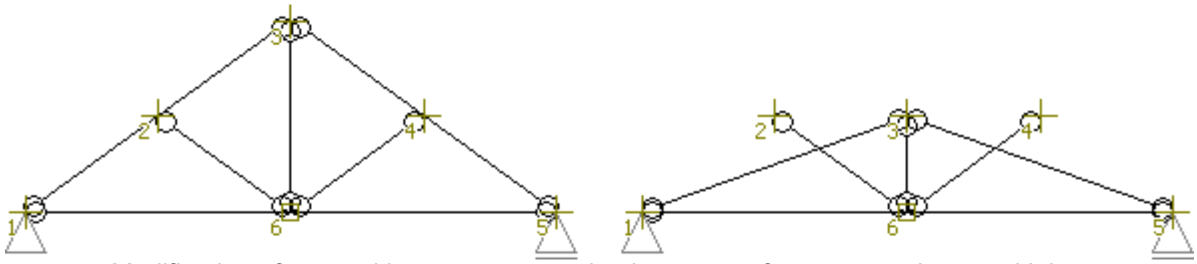
In the second case, joints *2* and *4* were converted into absolute ones, reference members were divided into two parts. The truss shape was changed into hipped one after the change of *Z*-coordinate of the apex, as the position of relative joints is defined by the coordinates.



*Modification of truss with divided reference members*

The last case shows the joints *2* and *4* that were converted into absolute ones without any division of reference members. The shape of truss is similar to the first case after the change of *Z*-coordinate of the apex, however, webs aren't connected to the top chords any more.

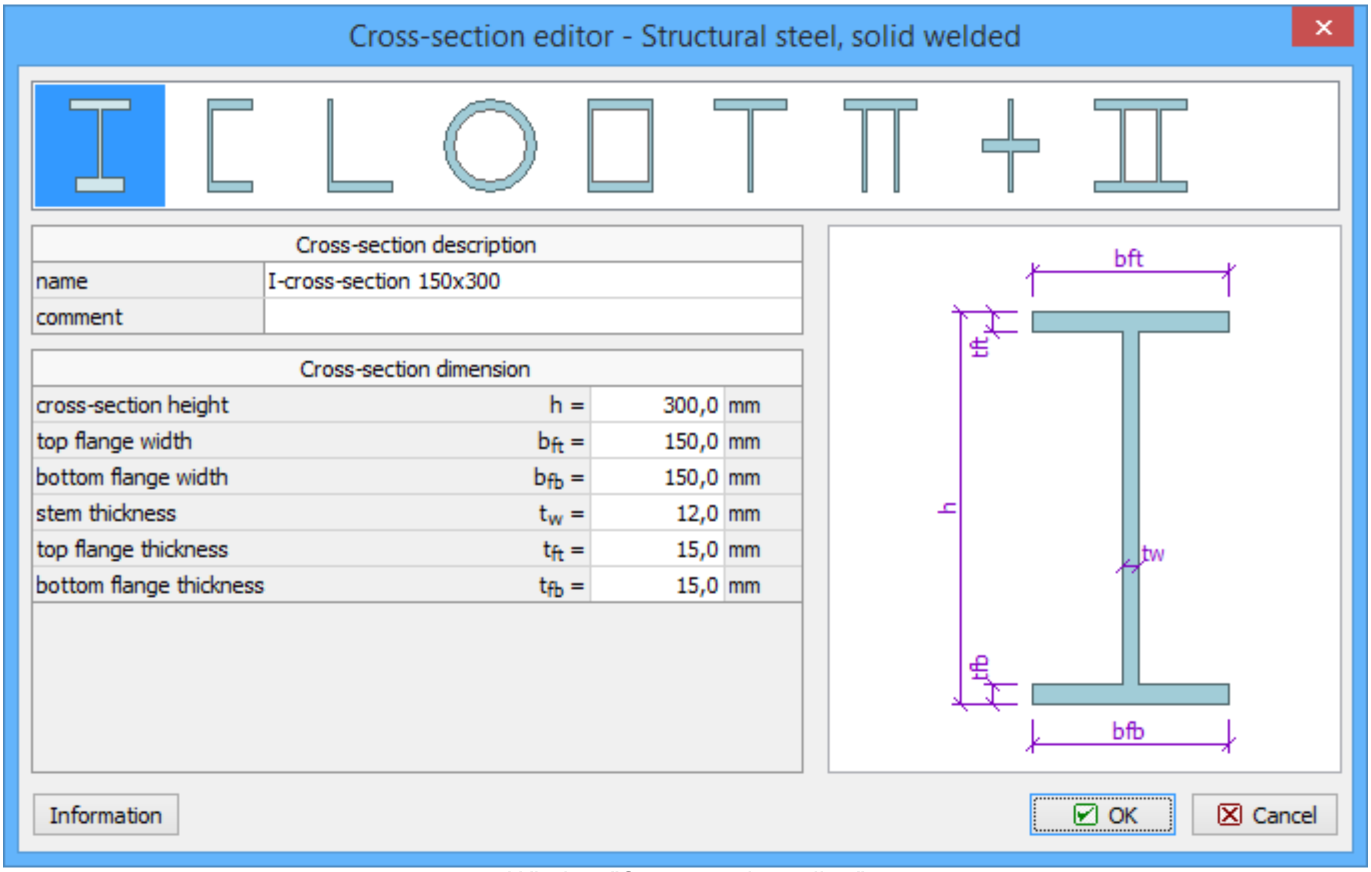


*Modification of truss without any connection between reference members and joints*

## Cross-section editor

The member cross-section can be modified in this window. The upper part contains library of available shapes (range differs according to the material and cross-section type). Dimensions or profile type can be entered in the table in the left part of the window. The meaning of dimensions is shown in the cross-section view in the right part of the window.

"**Information**" button in the left bottom corner shows complete list of cross-section characteristics.

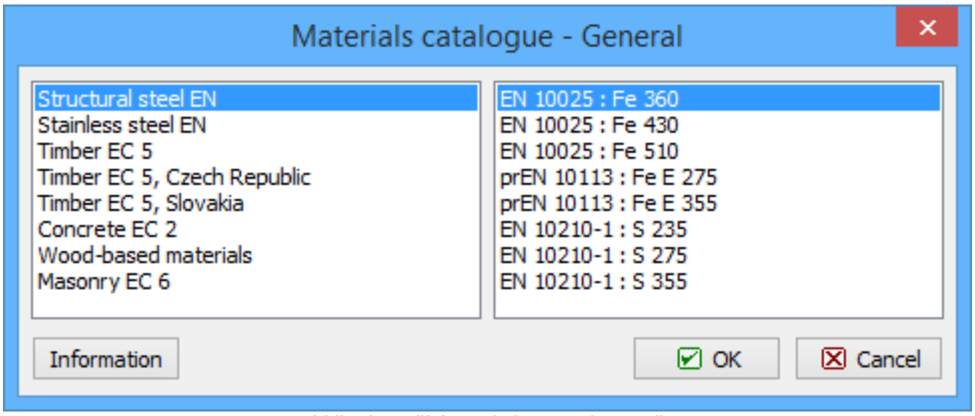



*Window "Cross-section editor"*

## Materials catalogue

This window contains the database of structural materials. The database is sorted according to the type of material (left list). Right list contains the corresponing strength grades for the selected material type.

The complete list of material characteristics for selected grade can be opened using "**Information**" button.



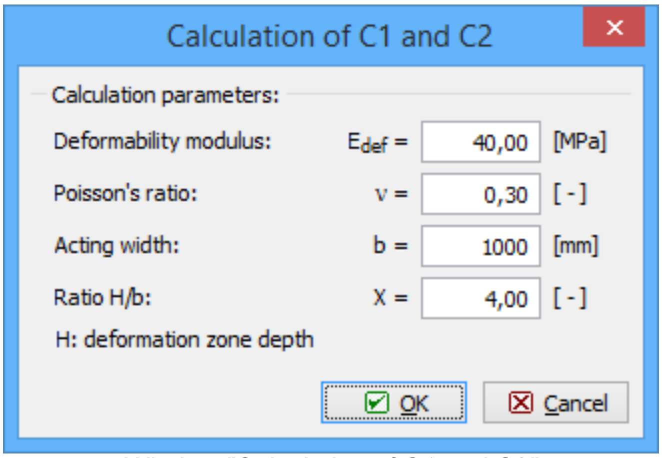

*Window "Materials catalogue"*

## Calculation of C1 and C2

The subsoil constants $C_1$ and $C_2$ may be calculated with the help of following inputs:

| | |
|---|---|
| **Deformability modulus** | • The modulus of deformation. It's a subsoil parameter, that can be obtained from tests. |
| **Poisson's ratio** | • The dimensionless value from interval *(0;0.5)*. Subsoil parameter. |
| **Acting width** | • The width of structure, that is in contact with the subsoil. This value is usually equal to the cross-sectional width, however, may be different for structures with additional parts between the member and subsoil. |
| **Ratio H/b** | • The ratio between the depth of deformation zone and the acting width. The deformation zone is the layer of subsoil, which is affected by the deformation caused by the member. The value is usually between *1.5* and *5.0*. |

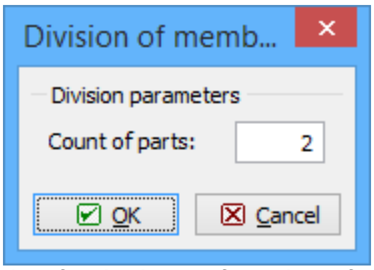
The calculation of Winkler-Pasternak constants is described in the chapter "**Subsoil model**".




*Window "Calculation of C1 and C2"*

## Division

This tool divides the member into identical parts with the help of the relative joints. These inserted joints may be used for input of point loads or for connection of another members. The tool is launched in the mode "**Members**" - "**Divide**" of the tree menu. The number of parts has to be specified in the dedicated window. The number of inserted joints is smaller by *1* comparing to the number of parts.




*Window for the input of number of parts*

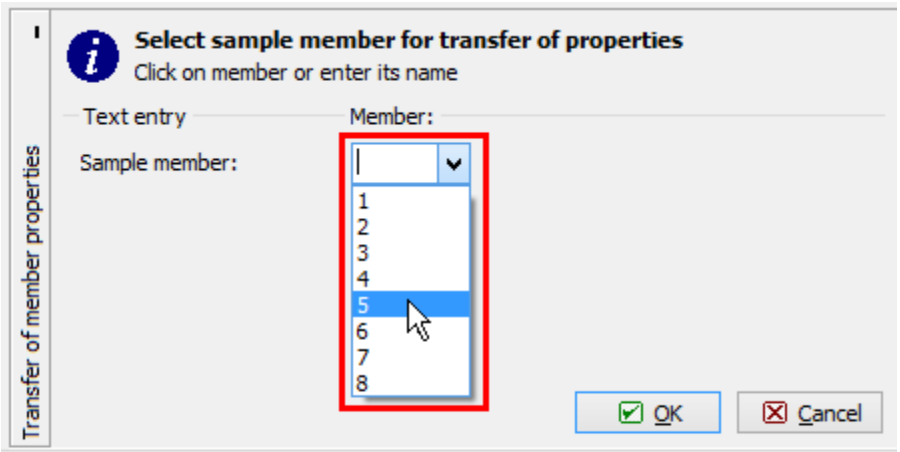
The relative joints are added after closing the window by the button "**OK**".

*Member before and after using the tool "Divide"*

The tool "**Conversion of relative joint to absolute**" has to be also used if the member should be divided into more individual members (e.g. for input of different cross-sections). Inserted relative joints will be converted into absolute ones with the help of this tool.

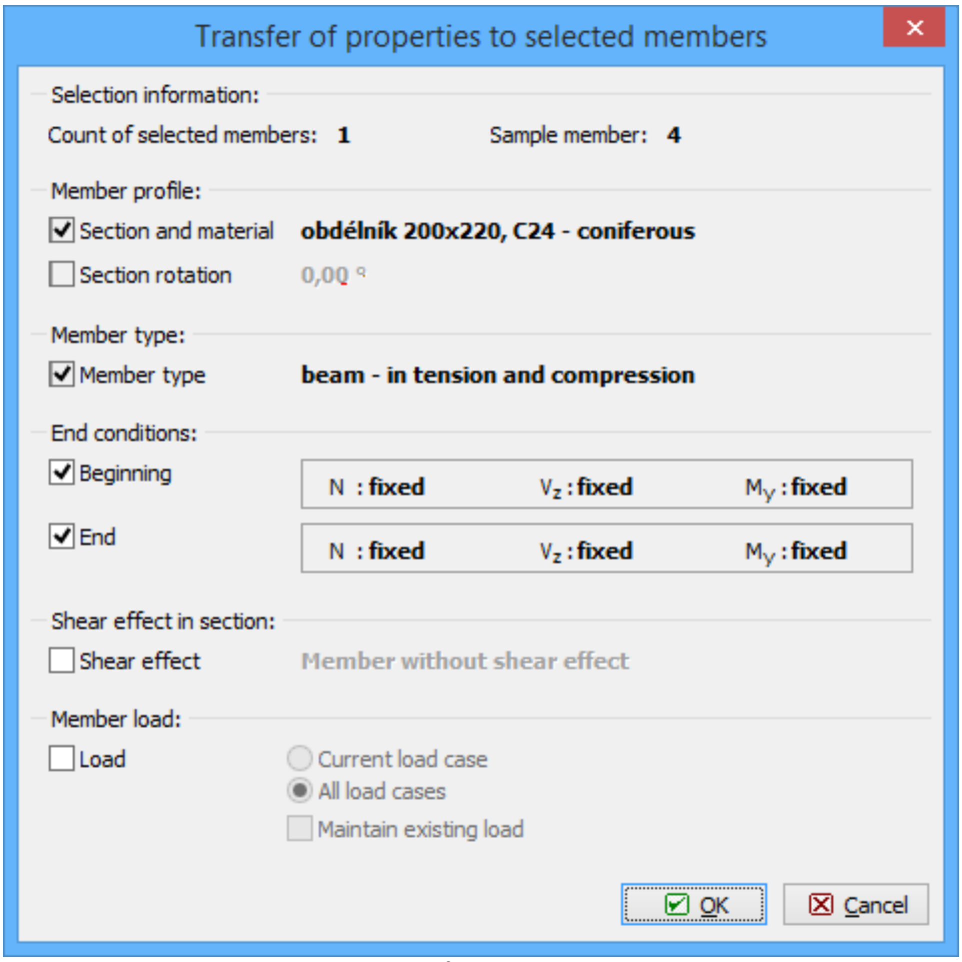
## Copy properties

This tool is able to copy properties of the sample member to the selected members. The fundamental condition is, that the structure contains at least one selected member (highlighted by the green colour in the workspace). The frame for the input of sample member appears in the bottom part of the window after running the tool in the part "**Members**" "**Selected**" "**Copy properties**". The choice has to be confirmed by the button "**OK**". The choice of sample member is possible also by clicking on member in the workspace.




*Selection of sample member in the bottom frame*

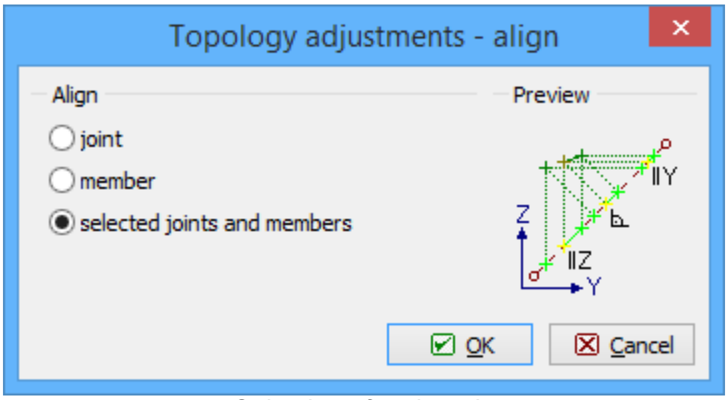
The properties, that should be copied to the selected members, can be selected in the following window. It is possible to copy cross-section, material, end conditions, member type and load applied to the member.




*Window "Copy properties"*

## Align

This tool aligns structural elements into the specified line. The tool may be applied to joint, member or selected elements. One of these three modes has to be selected in the first window "**Topology adjustment - align**".

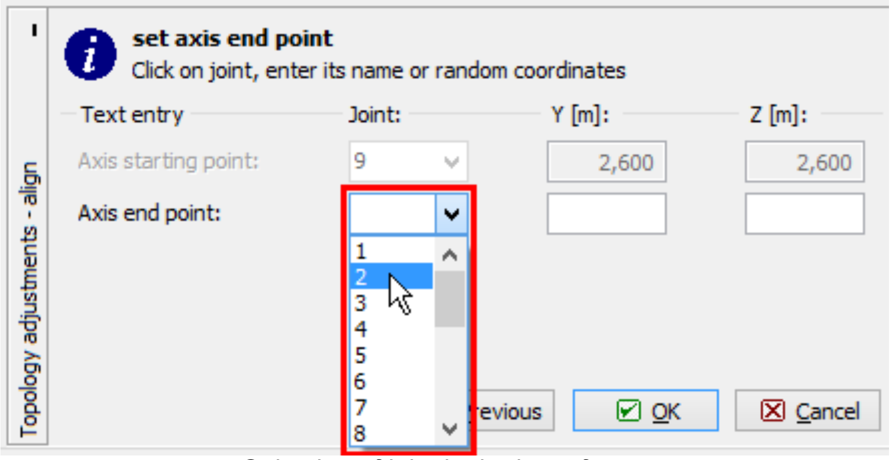



*Selection of tool mode*

### Selection of elements

The input of alignment line follows after running the alignment tool. The line is defined by two points, these points may be specified by coordinates or by choosing the existing joints. The selection of joints may be done by clicking in the workspace or with the help of input lines in the input frame.

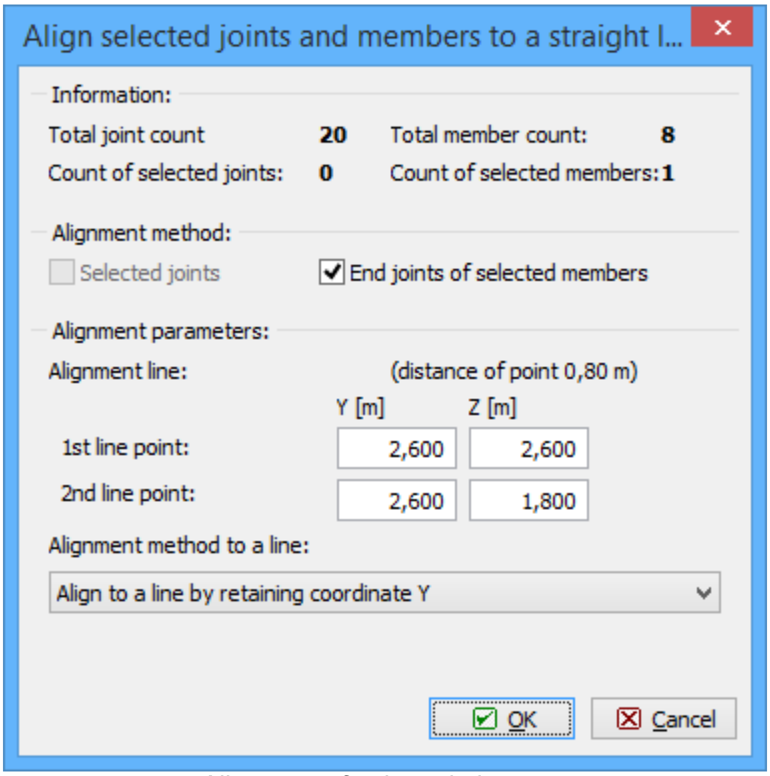


*Selection of joint in the input frame*

The joint of member for alignment will be selected in a similar way.

## Alignment parameters

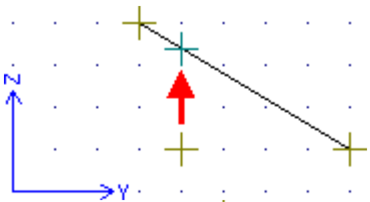
The window contains an option to modify the position of alignment line and also select the mode of manipulation.



*Alignment of selected elements*

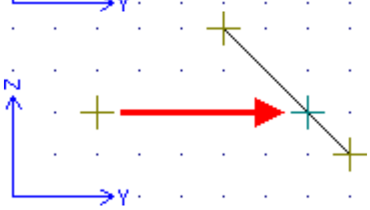
Following modes are available for joints:

**Align to a line by retaining coordinate *Y***




- Alignment of joint into point lying on alignment line. The horizontal coordinate *Y* of the new position is identical to the coordinate of original position.

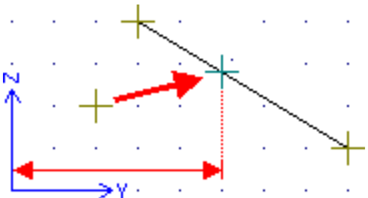
**Align to a line by retaining coordinate *Z***




- Alignment of joint into point lying on alignment line. The horizontal coordinate *Z* of the new position is identical to the coordinate of original position.

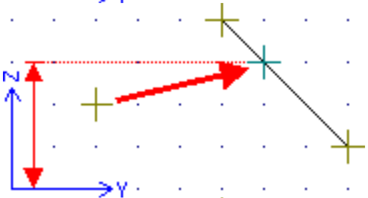
 

**Align to a line by setting coordinate *Y***



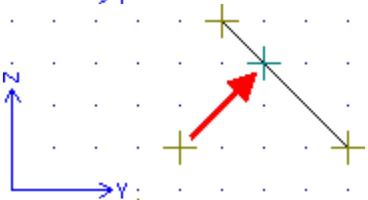

- Alignment of joint into point lying on alignment line. The horizontal coordinate *Y* of the new position is specified by the user.

**Align to a line by setting coordinate *Z***



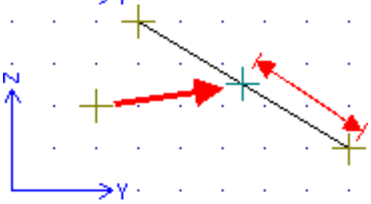

- Alignment of joint into points lying on alignment line. The horizontal coordinate *Z* of the new position is specified by the user.

**Align to a line by perpendicular line lead through joint**




- Alignment of joint into point lying on alignment line. The new position is the closest point on the line (measured from original position).

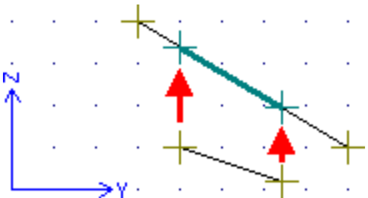
**Align to a line in distance from the first point**




- Alignment of joint into point lying on alignment line. The position is given by the distance from the first specified point of the alignment line.

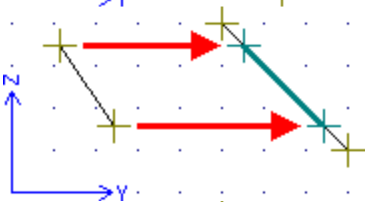
Following modes are available for members and selected elements:

**Align to a line by retaining coordinate *Y***



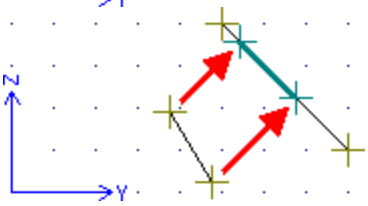

- Alignment of start and end joints of the member (or selected joints, end joints of selected members) into points lying on alignment line. The horizontal coordinate *Y* of the new position is identical to the coordinate of original position.

**Align to a line by retaining coordinate *Z***




- Alignment of start and end joints of the member (or selected joints, end joints of selected members) into points lying on alignment line. The horizontal coordinate *Z* of the new position is identical to the coordinate of original position.

**Align to a line by perpendicular line lead through joint**




- Alignment of start and end joints of the member (or selected joints, end joints of selected members) into points lying on alignment line. The new position is the closest point on the line (measured from original position).

# Loading

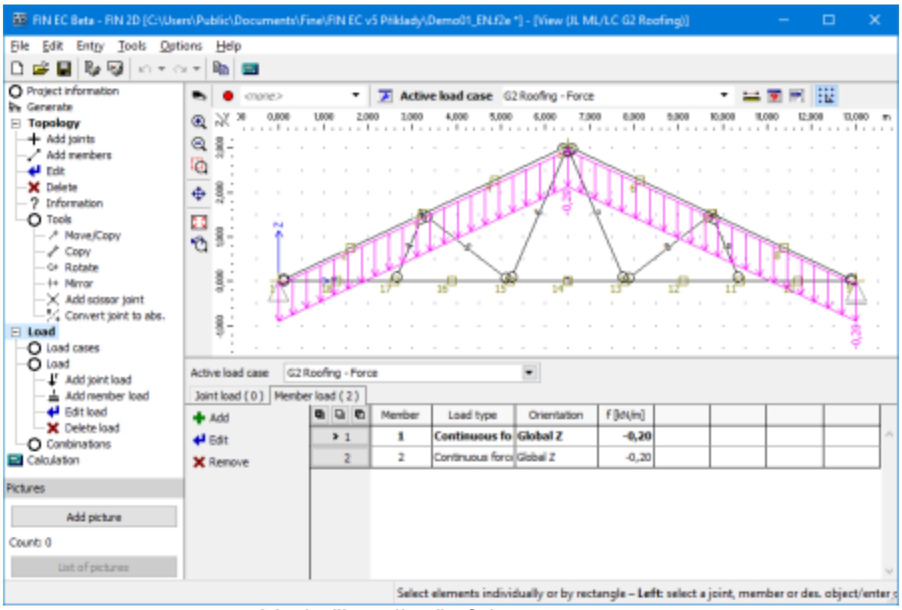
This mode of the tree menu contains tools for input and edit of loads in the structure. Following types of loads are supported:

- **Forces and bending moments** - The fundamental types of loads, supported are several types of linear load and also point loads.
- **Deformations of supports** - The option to specify enforced deformation of supports. This load can be added only into joints with defined support. The input is permitted only for load cases with code "**Deformation**".
- **Warming or cooling of structure** - The exposition of members to thermal changes. The input is permitted only for load cases with code "**Thermal**".

This mode is divided into following parts in the tree menu:

- **Load cases** - Input of load cases and their properties
- **Loads** - Input of particular loads into existing load cases
- **Combinations** - Mutual action of individual load cases in given design situations

The input frame in the bottom part of the window contains a drop down menu "**Active load case**", where the load case displayed on the workspace can be selected. The identical choice is also available in the heading of the workspace. The list of available load cases has to be specified in the part "**Load cases**" first. The bottom frame also contains tables of member and joint loads for the active load case. These tables are organized into tabs. The input and modifications of member and joint loads are described in the chapter "**Loads**", such work can be done easily with the help of tools in tree menu or in context menus in the workspace.

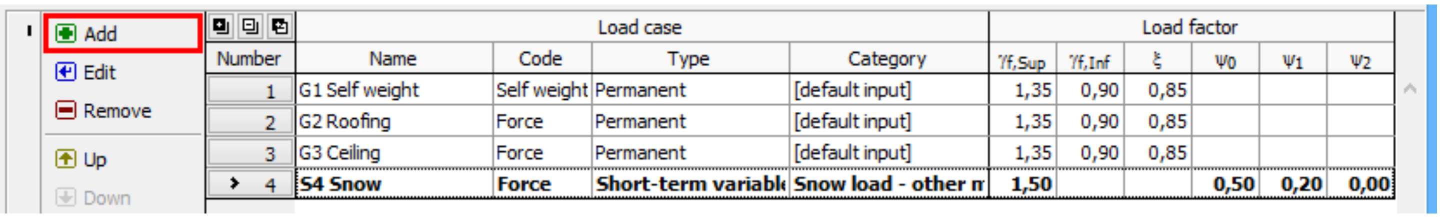


*Mode "Loading" of the tree menu*

## Load cases

Load cases are suitable for merging the loads, that have the same basis with regards to the standard and appear in the same time. Examples are self-weight of the structure or snow load. The loads, that have different properties according to the standard (e.g. permanent and variable loads), cannot be included in one load case.

The list of load cases for the project can be specified in this part. The toolbar for load cases input is located on the left side of the table of load cases. Available are tools for insertion of new load case, for load case edit and removing.



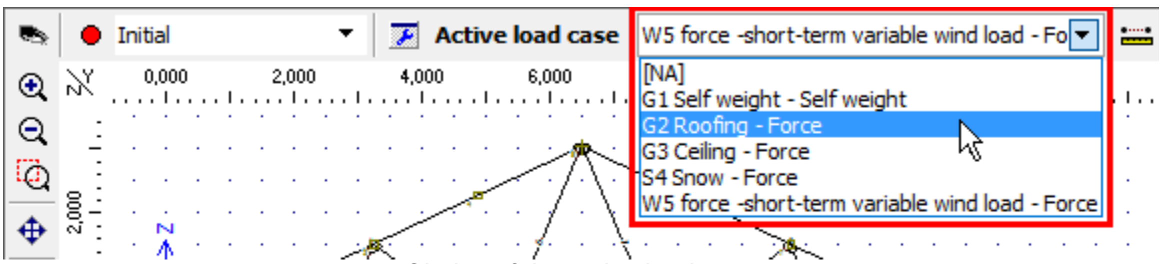
| | Load case | | | | Load factor | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| Number | Name | Code | Type | Category | $\gamma_{f,Sup}$ | $\gamma_{f,Inf}$ | $\xi$ | $\psi_0$ | $\psi_1$ | $\psi_2$ |
| 1 | G1 Self weight | Self weight | Permanent | [default input] | 1,35 | 0,90 | 0,85 | | | |
| 2 | G2 Roofing | Force | Permanent | [default input] | 1,35 | 0,90 | 0,85 | | | |
| 3 | G3 Ceiling | Force | Permanent | [default input] | 1,35 | 0,90 | 0,85 | | | |
| 4 | S4 Snow | Force | Short-term variabl | Snow load - other n | 1,50 | | | 0,50 | 0,20 | 0,00 |



*Add a new load case*

The general properties of the load case are organized in the window "**Load case edit**". The load case name, type, load duration and combination factors may be specified in this window. Insertion of loads into these load cases is described in the chapter, that belongs to the part "**Load**" of the tree menu.

If the loads in the certain load case should be displayed in the workspace, it is necessary to set this load case as an active one in the list box in the heading of the workspace.




*Choice of the active load case*

The list of load cases and combinations may be transferred between projects with the help of the **.flc* templates. The tools for the import and export of templates are included in the main menu in the part "**Tools**" - "**Load cases and combinations**".

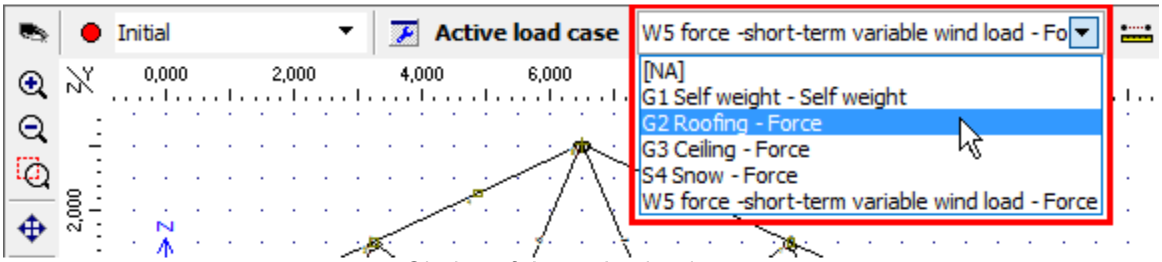
Theoretical background is described in the chapter "**Load cases**".

# Loads

The load applied to the structure can be specified in this part. The loads can be assigned both to members and joints. Such loads may contain forces and bending moments (load cases with the code "**Force**"), heating/cooling of structure (load cases with the code "**Thermal**") or deformations of supports (load cases with the code "**Deformation**"). The load is connected to the element, it means that the change of joint or member position causes the change of load position. Removal of element causes the removal of load. The load is added into the load case, that is selected in the heading of the workspace. It isn't possible to insert any load into the load case "**Self-weight**", as this load case contains only automatically generated load induced by the weight of the structure.

Theoretical information regarding load is here.



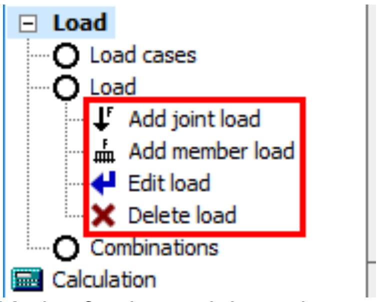
*Choice of the active load case*

The drop down menu "**Active load case**" is also placed in the heading of the bottom input frame in mode "**Load**" of the tree menu.

The load may be added (and modified or deleted) into the structure using two different ways: graphically in the workspace or numerically with the help of the load table in the bottom part of the window.

## Graphical modes

The load may be inserted/modified/removed graphically by clicking on the corresponding in the workspace. The tree menu has to be set into the appropriate mode.

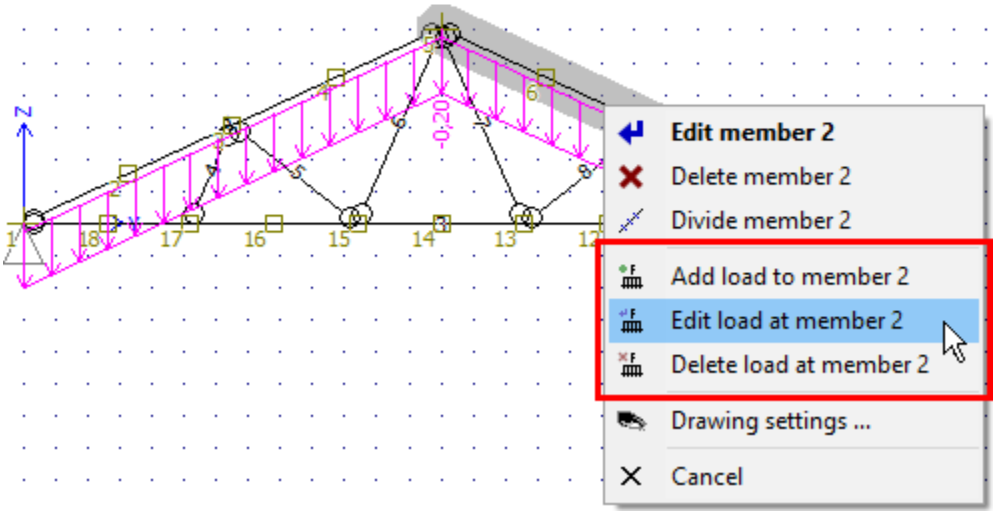


*Modes for the work in workspace*

The window "**Prototype of joint/member load**" appears when starting the graphical input of load. This window contains properties of load that will be applied to defined element. This window is moved into the bottom frame of the application window after the confirmation of input by the button "**OK**". It is possible to define loaded elements in the workspace after the input of load prototype. The prototype properties may be changed arbitrarily in the bottom frame during the input of loads.

For editing the load, it is necessary to switch the tree menu into the mode "**Edit load**". After that, the dedicated window "**Joint load properties**" or "**Member load properties**" with load properties appears after clicking on the joint or member.

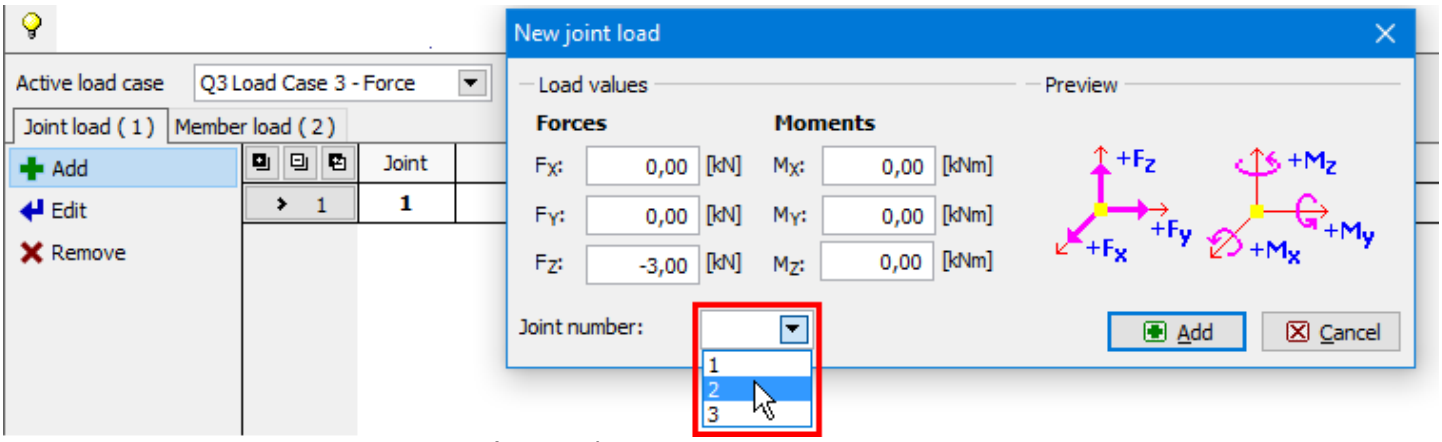
As an alternative way to functions in the tree menu, it is possible to use tools in the context menus for joints and members in the workspace. These tools bring an option to work with load also in the part "**Topology**" of the tree menu. The context menu fr a joint or a member can be opened by clicking on the element by right mouse button.




*Tools for a work with loads located in the context menu for a member*

## Work with load tables

Joint and member loads may be also added with the help of the load tables that appears in the bottom part of the window in mode "**Load**" of the tree menu. Tables contain a toolbar with buttons "**Add**", "**Edit**" and "**Remove**". The new load has to be specified in the window "**New joint/member load**". Not only the load values, but also the number of an element has to be specified in this window. The load will be applied to the structure using the button "**Add**", the input can be finished by the button "**Cancel**".

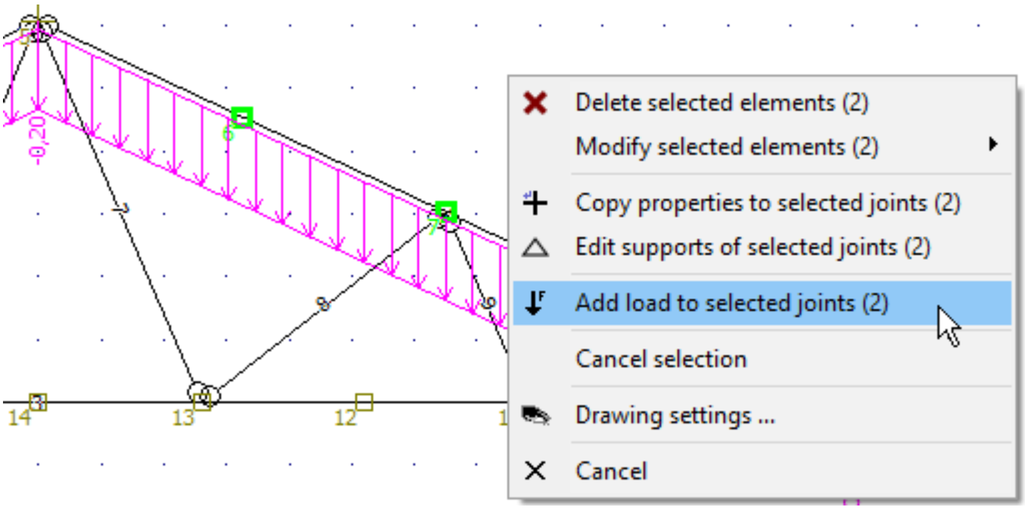



*Choice of joint number during load input*

The existing load in the table may be modified with the help of the double-click on corresponding table row. Alternatively, single click (this click sets the load as an active one and highlight it by the bold font) and button "**Edit**" may be used.

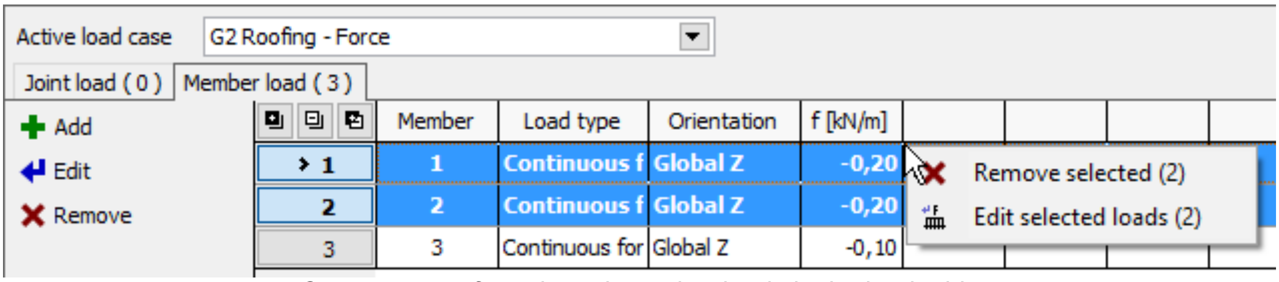
## Input and edit in a batch

The load may be also inserted or modified in a batch for selected joints or members (highlighted by the green colour in the workspace). The context menu opened in the workspace contains tools for batch work with loads in these cases. It is possible to add new loads, edit loads with same values or delete loads.



*Input of load for selected joints*

Alternatively, it is possible to modify or delete selected loads in load tables in the bottom frame (visible in the mode "**Load**" of the tree menu). These tables also contain context menus with appropriate tools.




*Context menu for selected member loads in the load table*

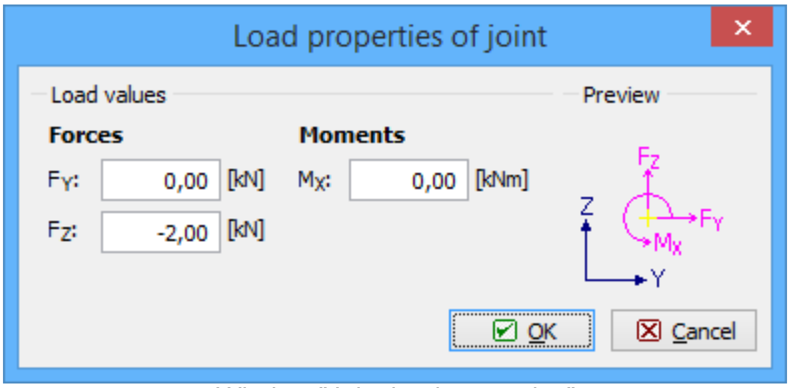
Values of all or selected loads may be also multiplied by specified factor. This option is suitable for example in case that it is necessary to change the loading width of the member. This option is included in the main menu in the part "**Tools**" - "**Load**". This operation may be applied to active, selected Or all load cases in the structure.

## Joint load properties

This window contains the values of joint load. The range of values differs according to the code of the active load case.

Forces and moments can be specified in load cases with code "**Force**". The input of forces and moments is done separately for directions according to the main axes of the global coordinate system. The orientation of positive values is displayed in the figure in the right part of the window.

For load cases with the code "**Deformation**", the deformation of the support may be specified. This load can be applied only to joints that are supported in corresponding directions.




*Window "Joint load properties"*

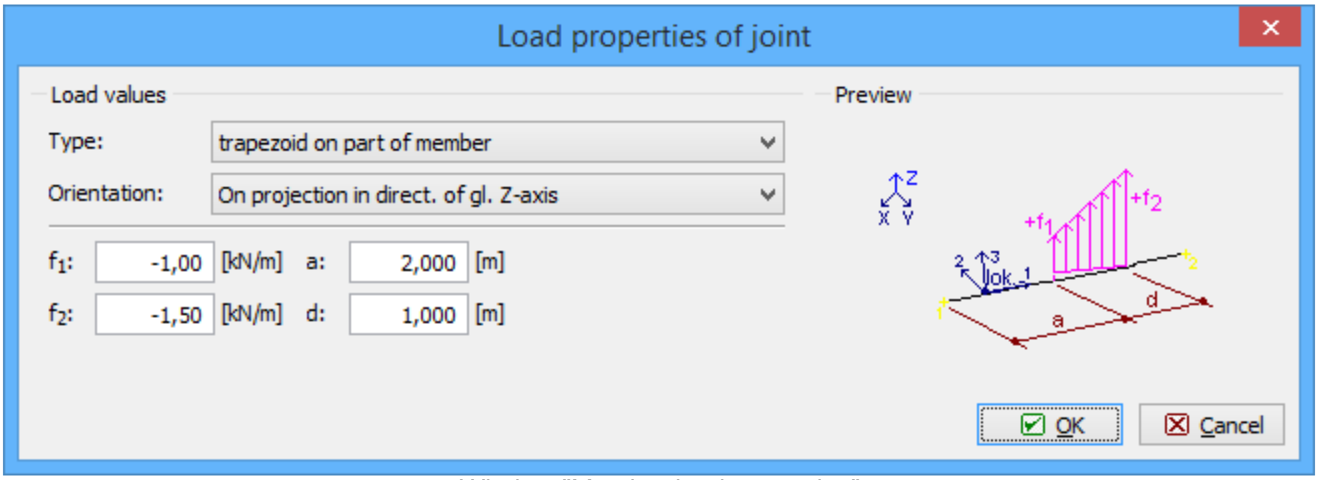
## Member load properties

This window contains the properties of member load. The range of values differs according to the code of the active load case.

Forces and moments can be specified in load cases with code "**Force**". For load cases with the code "**Temperature**", the uniform or non-uniform warming/cooling can be specified.

The right part of the window contains the figure that shows positive orientation of inserted load.




*Window "Member load properties"*

Following load types are supported:

## Individual force

Type "**Individual force**" is the point load that acts in given position elsewhere along the member length. This load may represent e.g. reaction from supported beam. The input consists of load position and load magnitude including correct orientation.

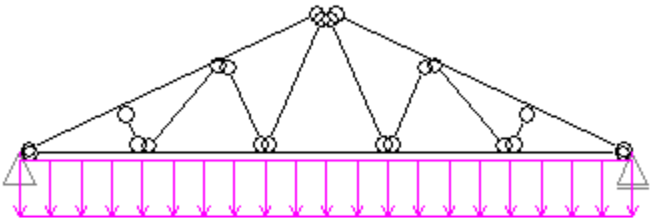
*Point load in the middle of the single beam*

## Individual moment

The type that is similar to "**Individual force**", however, the member is loaded by bending moment.

## Distributed on entire member

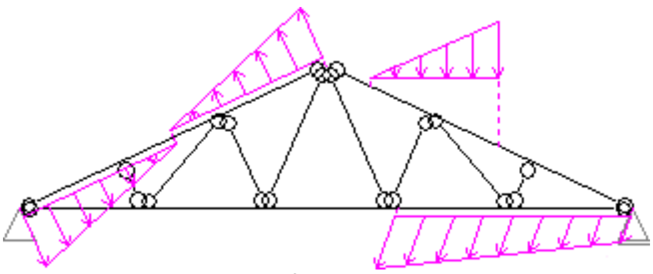
This type represents linear load with constant value along the whole member length. This is the most common member load. The load value and orientation has to be specified for this type.



*Linear load on bottom chord*

## Trapezoid on part of member

Almost arbitrary linear load can be created with the help of this type. The load is given by the values at the beginning and end of load. It means, that this type may be used for input of constant, trapezoidal or triangular load. The position is given by the load length and distance from the member beginning.



*Examples of trapezoidal loads*

## Load orientation

Following options may be used for the definition of load direction:

**Orientation along local axis 1**
- The load acts in the member direction.

**Orientation along local axis 2**
- The load acts perpendicular to the member, the load is in the horizontal direction for angle *α=90°*.

**Orientation along local axis 3**
- The load acts perpendicular to the member, the load plane is vertical. The angle *α* may change the orientation of the load in this plane.

**Orientation along global axes X,Y,Z**
- The load acts everytime in the direction of the global axis, without any respect to the member direction. The load has to respect orientation of axes. E.g. gravity load has to be specified as a negative value, as gravity acts against the direction of the global axis *Z*.

**On projection in the direction of global axes X,Y,Z**
- The load acts always in the direction of the global axis, without any respect to the member direction. The length of load is equal to the length of member projection to the corresponding global axis.

**Orientation along section axes y,z**
- The load acts in directions of cross-sectional axes. This option is useful for cases, where cross-section is rotated and cross-sectional axes aren't identical to the member axes *2, 3*.

## Thermal load

Thermal load may be used for considering situations, when the member is exposed to the significant changes of temperatures, that may cause additional stresses in the structure. Warming or cooling is defined as a difference of the temperature comparing to the normal state, where no additional stresses occur. The thermal load may be equal or unequal. For unequal thermal load, the temperatures on particular edges of the thermal area has to be specified. The thermal area may respect size of the cross-section (the option "**Use from member section**" or may be specified manually (including the position of the centre of gravity).

# Combinations

The load combinations are used for the mutual action of different load cases. The description of any combination contains the list of included load cases including corresponding load case factors and combination factors. The combination factors may be obtained from load cases characteristics (design standard "**EN 1990**") or specified manually (other design standards).

The combinations are divided into particular tables according to their application. These tables are organized into tabs in the bottom frame. Following tabs are available:

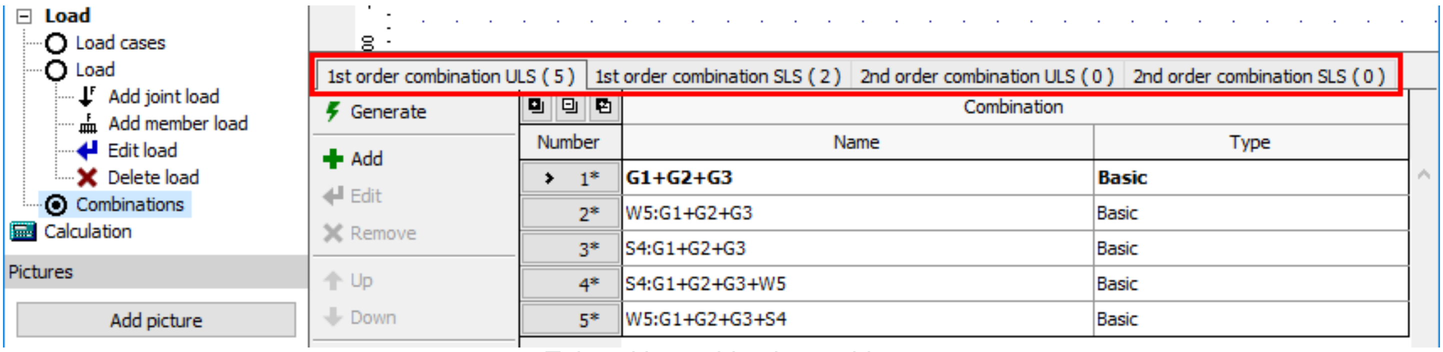
**1st order combination ULS**
- The fundamental combinations for the analysis of ultimate limit states. Both combination factors and load factors *γ* are applied in these combinations. Results of these combinations are transferred into design modules for the verification of the bearing resistance.

**1st order combination SLS**
- The combinations for the analysis of serviceability limit states. Load factors *γ* aren't applied in these combinations. These combinations shows the values of structure deformations.

**2nd order combination ULS**
- The combinations for the analysis of ultimate limit states according to the 2nd order theory. The rules are identical to the "**1st order combination ULS**". The combinations can be copied from 1st order or specified manually. Available only if the analysis according to the 2nd order is switched on in the window "**Project information**".

**2nd order combination SLS**
- The combinations for the analysis of serviceability limit states according to the 2nd order theory. The rules are identical to the "**1st order combination SLS**". The combinations can be copied from 1st order or specified manually. Available only if the analysis according to the 2nd order is switched on in the window "**Project information**".

**Linear stability combination**
- The combinations for the analysis of linear stability. These combinations don't contain any factors. Available only if the analysis of linear stability is switched on in the window "**Project information**".



*Tabs with combinations tables*

The list of combinations in the table can be created with the help of dedicated toolbar on the left side of the table. The toolbar contains the button "**Generate**" for batched input of combinations in the window "**Combinations generator**" and the button "**Add**" for the input of single combination in the window "**Combination**". The list of combinations may be displayed in the "**Table of combinations**" with the help of the button "**Table**".

The list of load cases and combinations may be transferred between projects with the help of the **.flc* templates. The tools for the import and export of templates are included in the main menu in the part "**Tools**" - "**Load cases and combinations**".

Theoretical background is described in the chapter "**Combinations**".

# Calculation

The analysis can be run by the command "**Calculation**" in the tree menu. Alternatively, it is possible to use the main menu or key *F9*. The window "**Calculation properties**" is launched always before running the analysis. The window contains the settings that may influence the type and parameters of the calculation. At the end of the analysis, the calculation report is created. This report may be opened again with the help of the command "**Error list**", that can be found in the main menu, part "**Tools**". After the analysis, the tree menu is automatically switched from the input mode into the mode "**Results**" (postprocessor).

The analysis is also described in the chapters "**1st order analysis**" and "**2nd order analysis**".

# Calculation properties

This window is launched always before running the analysis. The window contains the settings that may influence the type and parameters of the calculation. The settings are organized into two tabs.

## Calculation

The particular calculations (e.g. dynamics, linear stability) may be switch on in this tab. The tab contains also the setting, that automatically saves the project before running the analysis. This feature may be beneficial mainly for large unstable structures.

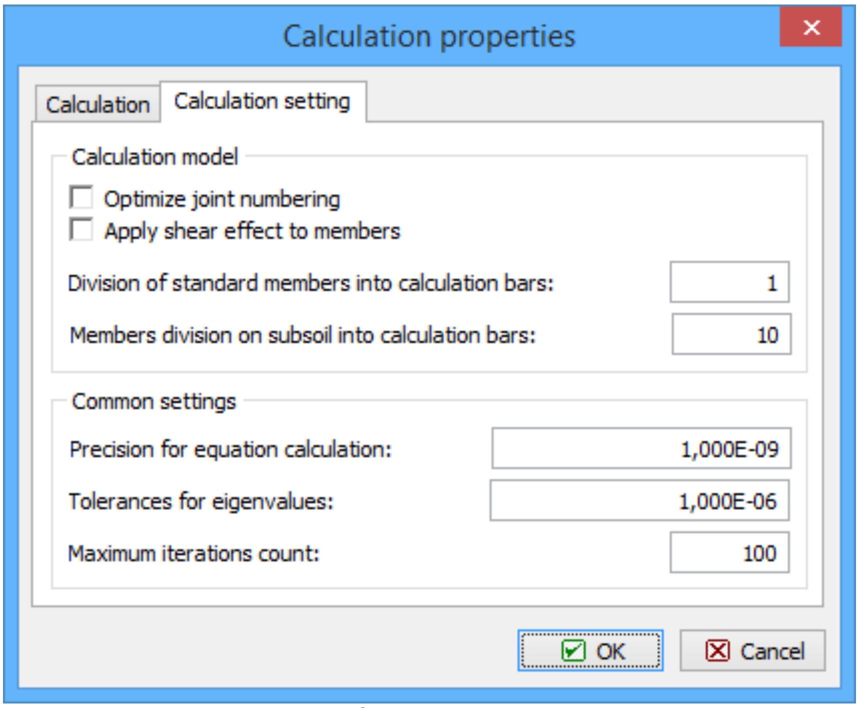
## Calculation setting

Following settings are included:

| | |
|---|---|
| **Optimize joint numbering** | • The software renumbers the joints in the structure in that way, that the equations may be solved faster. The renumbering is done internally, it doesn't affect the numbering of joints in the user interface. This feature doesn't work for structures divided into more separated parts. |
| **Apply shear effect to members** | • This setting changes the theoretical model for members. This setting is recommended cases, where member length isn't significantly longer than cross-section dimensions. The theoretical background is described in the chapter "**Special member characteristics**". |
| **Division of standard members for analysis** | • This setting affects the division of members into particular analysis members. There is usually no significant reason to apply higher value than *2*, as higher number doesn't improve results, only increases the analysis time. The higher value may be beneficial only in special cases (e.g. unsymmetrical load applied to the member), when the analysis may provide more precise values of deformations along the member length. |
| **Division of members on subsoil for analysis** | • The division of members with specified subsoil. The more precise results are obtained for higher values of this parameter, therefore the default value *10* is used. If the division is done already in preprocessor (more particular members are specified instead of one long member), it is possible to reduce this value. |
| **Precision for equation calculation** | • The default value is recommended. |
| **Tolerances for eigenvalues** | • The default value is recommended. |
| **Maximum iterations count** | • This parameter sets the maximum number of calculation cycles for structures with semi-rigid joints. |

The analysis is also described in the chapters "**1st order analysis**" and "**2nd order analysis**".




*Window "Calculation properties"*

## Results display

The workspace in the mode "**Results display**" shows the results of the calculation. This mode may be also used for organizing members for the verification in verification programs. The tree menu contains following parts:

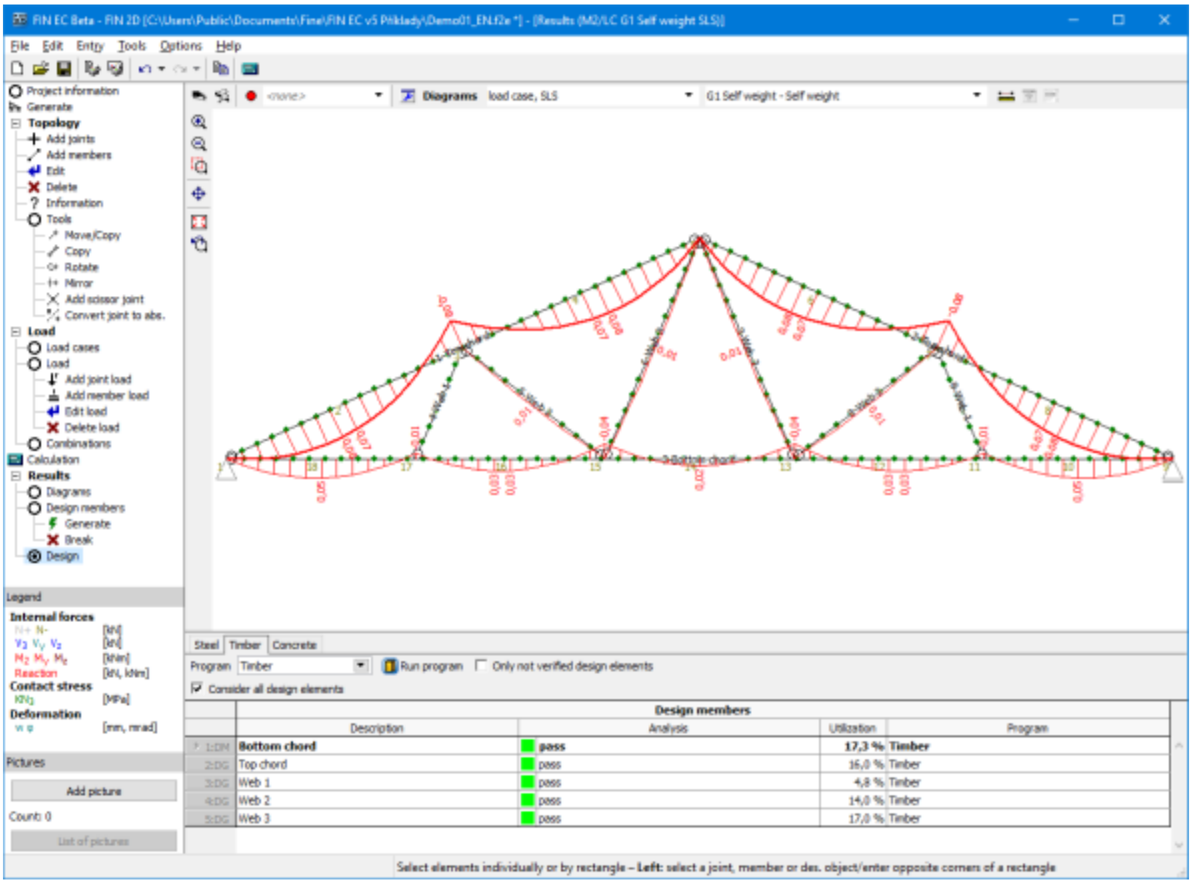
**Diagrams**
- This part is able to display results (forces, moments, reactions, deformations) for specified part of the structure. The results are displayed in the table in the bottom part of the window.

**Design members**
- This part is suitable for organizing the members before the final verification. Members may be merged into design members or groups. This operation reduces the verification, as merged members are verified together as one member.

**Design**
- The member verification can be done in this part, including the choice of appropriate verification program (basic design, fire resistance etc.). The lists of verified members and considered combinations may be reduced by the user.

The displayed results can be modified with the help of the toolbar above the workspace. The button "" launches the window "**Results view settings**", where the displayed quantities (forces, moments, deformations, stresses etc.) may be selected. The displayed items (marks, description of joints and members, font size) can be specified with the help of the button "" in the window "**Drawing settings**". The toolbar also contains the "**View manager**". The list box "**Diagrams**" defines, whether the results will be displayed for load cases, combinations or an envelope. The parameters of the envelope has to be specified in the window "**Envelope**". This window can opened with the help of the button "**Set**".

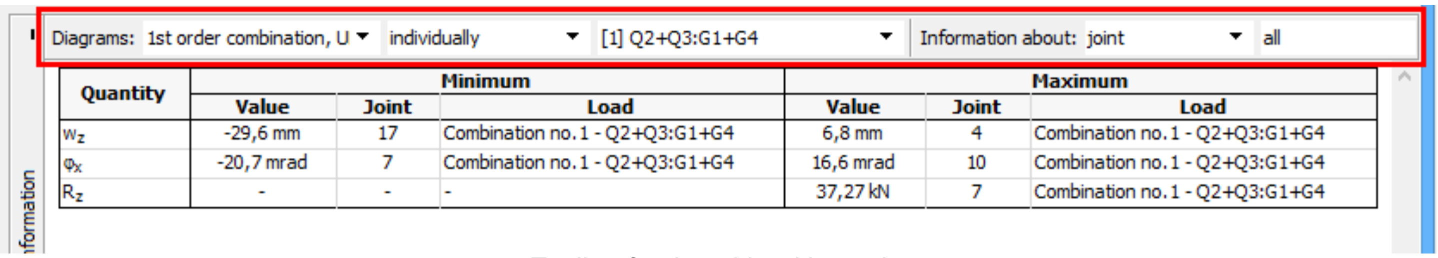
 




*Bending moments on members*

## Information about results

This part shows the most important results in the table in the bottom part of the window. Minimum and maximum values of internal forces are displayed for members, extreme values of deformations and reactions are shown for joints. The range of displayed values can be specified in the dedicated toolbar in the heading of the table.




| Quantity | Minimum | | | Maximum | | |
|---|---|---|---|---|---|---|
| | Value | Joint | Load | Value | Joint | Load |
| $w_z$ | -29,6 mm | 17 | Combination no. 1 - Q2+Q3:G1+G4 | 6,8 mm | 4 | Combination no. 1 - Q2+Q3:G1+G4 |
| $\varphi_x$ | -20,7 mrad | 7 | Combination no. 1 - Q2+Q3:G1+G4 | 16,6 mrad | 10 | Combination no. 1 - Q2+Q3:G1+G4 |
| $R_z$ | - | - | - | 37,27 kN | 7 | Combination no. 1 - Q2+Q3:G1+G4 |

*Toolbar for the table with results*

## Envelope

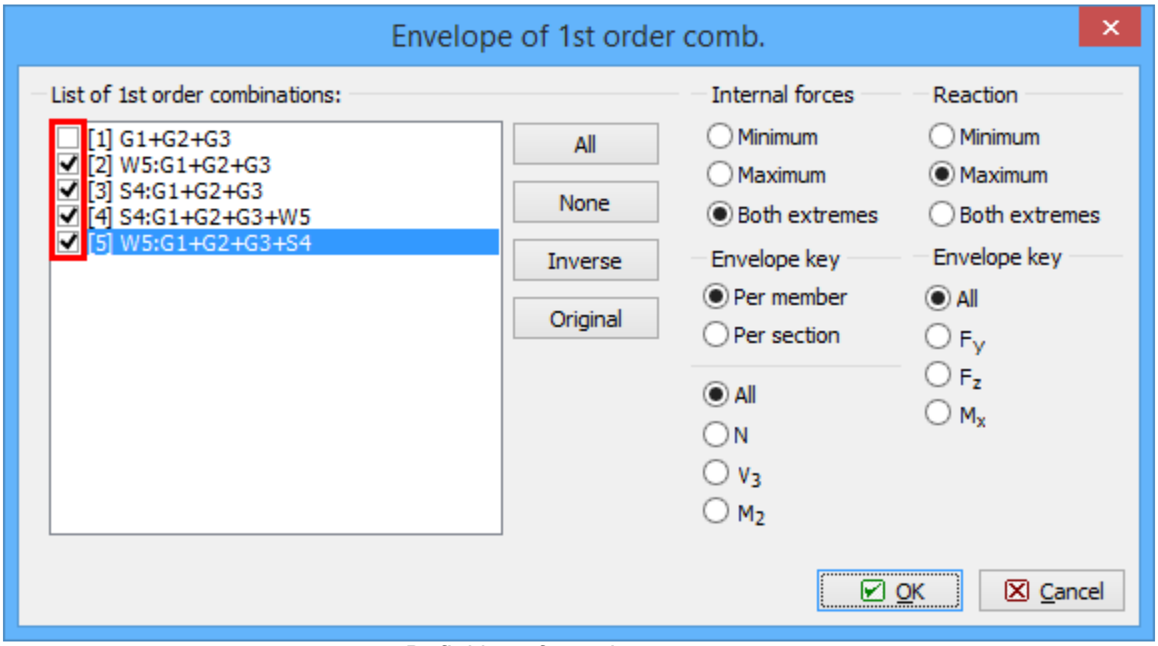
This window contains properties of envelope of combinations or load cases. The envelope is the diagram of quantity (mainly internal forces), that shows the maximum and minimum values of certain quantity for all combinations (load cases) included in the envelope in any point of the structure.

Left part shows complete list of available combinations (load cases), that may be included in the envelope. The selection can be done with the help of check boxes in front of the combination numbers or with the help of buttons in the toolbar on the right side of the list. The toolbar contains following buttons:

- **All** • Selects all combinations or load cases in the list.
- **None** • Cancels the selection of combinations or load cases.
- **Inverse** • Selects the combinations (load cases) that weren't selected and deletes the selection for combinations (load cases) that were selected.
- **Original** • Sets the selection that was active during the opening this window.

### Internal forces and Reactions

The frames "**Internal forces**" and "**Reactions**" contain the parameters for making the envelopes of internal forces and reactions. Only maximum, minimum or both extreme values may be shown in envelopes. The "**Envelope key**" defines, whether the envelope will be shown for all components (the option "**All**") or only for one component and other components will show only values in corresponding combination (or load case), where the maximum of key component appears.




*Definition of envelope content*

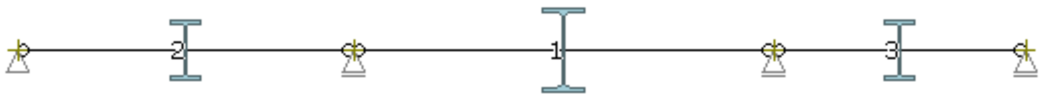
## Design elements

This part is suitable for optimizing the number of structural elements before the verification process. The members may be merged into design members and groups, it is also possible to rename the elements to simplify the identification in the project.

### Design members and design groups

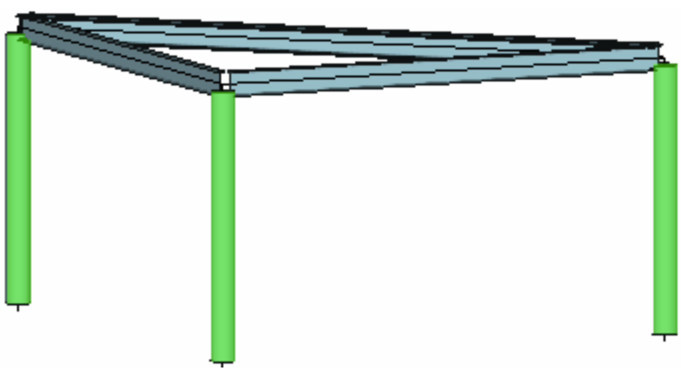
Any structural member is listed as an individual element in the part "**Design**" of the tree menu. All the necessary parameters for verification (buckling parameters etc) have to be specified for any of these members. This procedure may be long lasting for more complicated structures. To simplify the work, the members may be merged into design members and groups. The number of members for verification may be reduced significantly with the help of this procedure.

**Design members (DM)** may be used for merging the members of the same material (timber, steel, concrete) that lie in the same line. Merged members may have different cross-sections (except concrete structures). Only one member with the length equal to the total length of merged members will be transferred into the designing program when the design member is created. As the designing programs are using calculated diagrams of internal forces as an input, it is possible to merge members connected through inner hinges. The typical example of design member is the steel column with variable cross-section or the RC beam with more supports.




*Connected beams that may be merged into one design element*

**Design groups (DG)** is suitable for merging the members (members or design members) that have identical topology and may be verified as one member. The merged members have to have identical cross-section, length and orientation. The members in design group are transfferred into designing program as one member, however, number of verified cases is multiplied by the count of members in the group. The opposite orientation of members in group is permitted. The software shows a warning, as it may cause wrong results for members with unsymmetrical designing parameters (buckling properties etc.) along the member length.

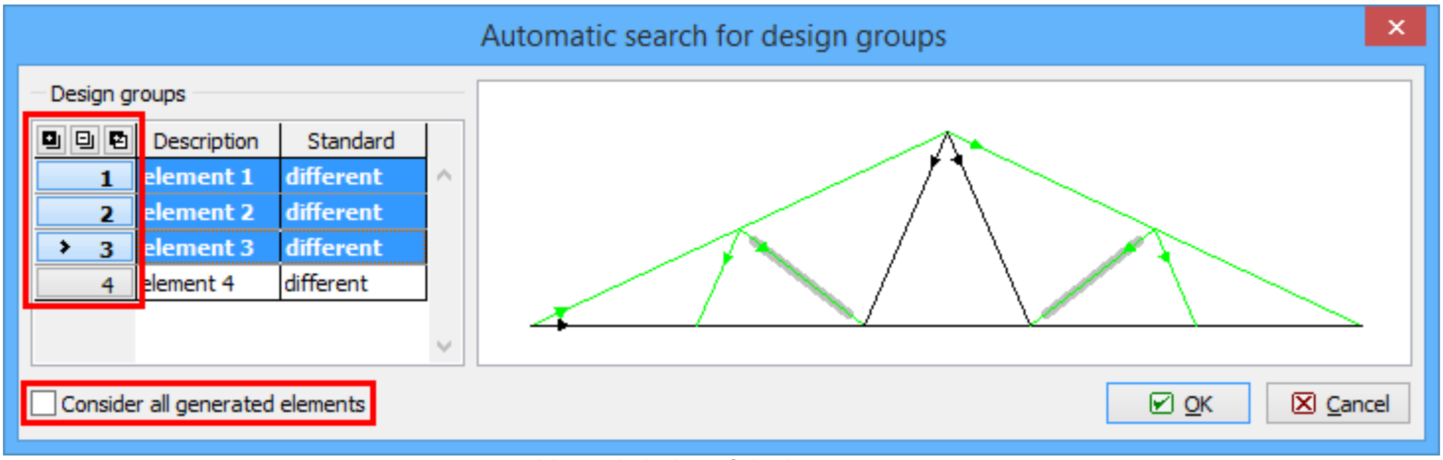


*Design group of columns (highlighted by green)*

## Input of design members and groups

The design members should be created before design groups, as design members may be merged into design groups. The design elements can be created manually (part "**Selected**" in the tree menu) or automatically (part "**Generate**"). For the manual work, the members for merging into design member or group has to be selected first (highlighted by green in the workspace). After that, corresponding tool ("**Join to DM**" or "**Join to DG**") can be used.

Automatic creation of design members and groups launches the dedicated window, that shows design elements which were recognized by the software. Table in the left part shows the list of proposed design elements, active one is highlighted also in the structure view. The table of design groups also shows the information regarding the direction of merged members (problem described above). The window contains the setting "**Consider all generated elements**", which provides an option to create only few of suggested elements. The choice of elements for creation can be done with the help of buttons in the first column of the table.



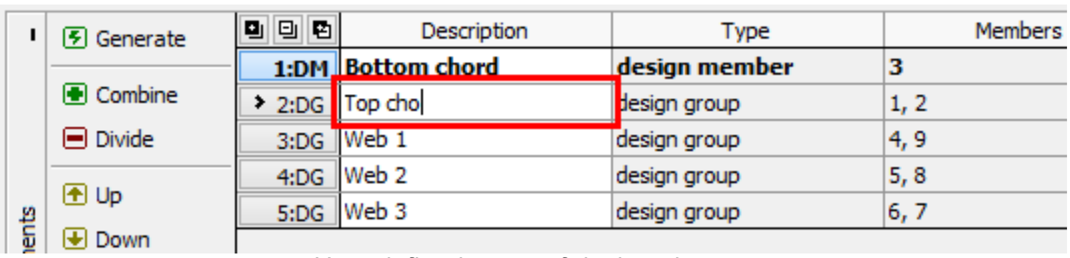

*Manual choice of design groups*

## Breaking design members and groups

Already created design elements may be broken back into the individual members. The active group or member (the active element is highlighted in the workspace and also by the bold font in the table of design elements) may be deleted with the help of the tool "**Individually**" - "**Break**". Similar way ("**Selected**" - "**Break**") can be used for selected elements. The tool "**Break into members**" in the tree menu deletes all design members and groups in the structure.

## Renaming the dimensioning elements

Any design element may be described to simplify the identification in the structure. The name can be specified in the column "**Description**" of the table with design elements.




*User defined name of design element*

# Design

The structural analysis of members can be done in this part. The analysis is done in verification modules and can't be performed if these programs aren't installed on the computer. The design members are organized into three groups according to the material: "**Steel**", "**Timber**", "**Concrete**".

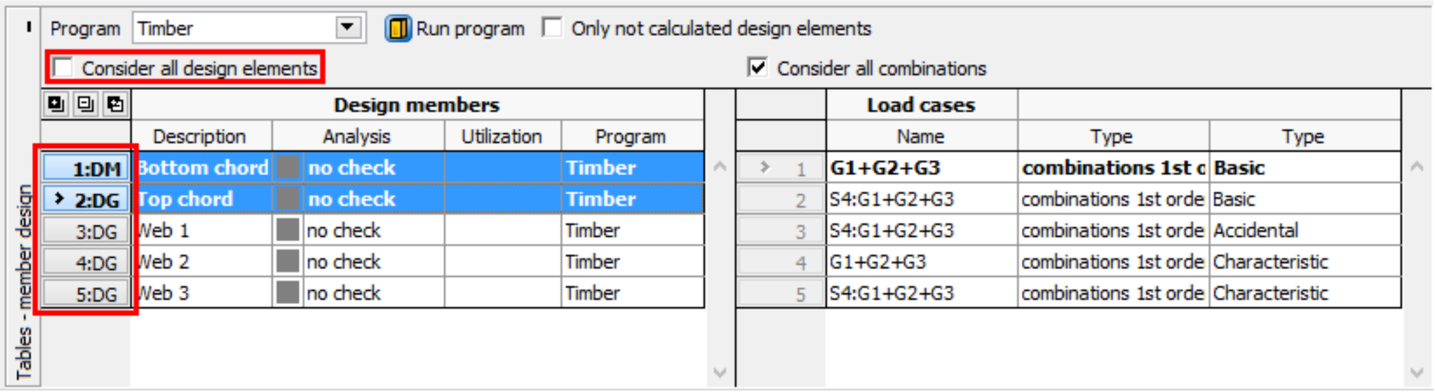
The transfer of members into verification programs is done with the help of the table in the bottom part of the window. The appropriate program for the analysis can be selected in the list box "**Program**". The design members will be transferred into this program after slicking on the button "**Run program**". The analysis may be limited to certain members or combination. Such behaviour may be beneficial for more complicated structures with a lot of members. Following options are available:

| | |
|---|---|
| **Only not verified design elements** | • Only elements, that aren't verified, will be transferred into verification program |

**Consider all design elements**
- If this setting is switched off, it is possible to select, which design elements should be transferred into the verification programs. The choice can be done with the help of dedicated buttons in the first column of the table.

**Consider all combinations**
- If this setting is switched off, it is possible to select, which combinations should be transferred into the verification programs. The choice can be done with the help of dedicated buttons in the first column of the table with combinations.




*Manual choice of design elements for verification*

The member cross-sections may be changed during the work in verification programs. If some cross-sections were changed in this way, the software automatically offers the recalculation of internal forces after the return from verification program. This recalculation updates the internal forces according to the latest stiffness distribution in the structure.

Following verification programs may be used for the analysis:

**Verification programs for steel structures**

**Steel**
- This program verifies steel members according to the standards EN 1993-1-1 and EN 1993-1-4 (stainless steel). Both fundamental and accidental combinations can be verified in this program.

**Steel Fire**
- This program verifies fire resistance of steel members according to EN 1993-1-2. Only accidental combinations are checked in this program.

**Verification programs for timber structures**

**Timber**
- This program verifies timber members according to the standard EN 1995-1-1. Both fundamental and accidental combinations can be verified in this program.

**Timber Fire**
- This program verifies fire resistance of timber members according to EN 1995-1-2. Only accidental combinations are checked in this program.

**Verification programs for concrete structures**

**Concrete**
- This program verifies concrete members according to the standards EN 1992-1-1 and EN 1992-2 (Concrete bridges). Both ultimate limit states (fundamental and accidental combinations) and serviceability limit states (stress limitation, crack control) can be verified in this program.

**Concrete Beam**
- This program verifies horizontal concrete members according to the standards EN 1992-1-1 and EN 1992-2 (Concrete bridges). Both ultimate limit states (fundamental and accidental combinations) and serviceability limit states (stress limitation, crack control, deflection control) can be verified in this program.

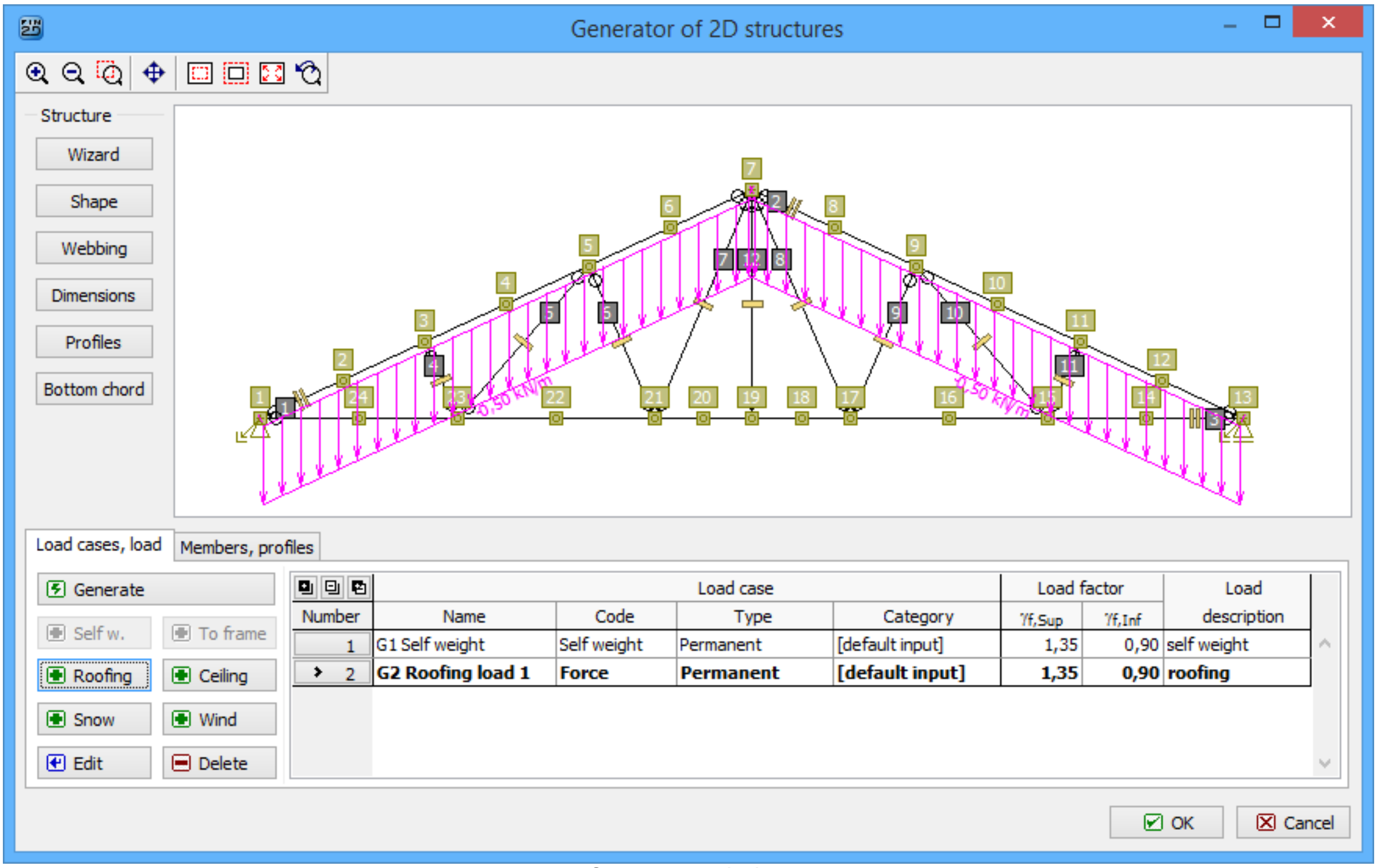
**Concrete Fire**
- This program verifies fire resistance of concrete members according to EN 1992-1-2. Only accidental combinations are checked in this program.

## Generator of 2D structures

The generator of 2D structures is suitable for the fast input of the most common structures (trusses, attic structures, frames) including member properties and basic loads. The window consists of the workspace, vertical toolbar "**Structure**" and the input frame in the bottom part of the window.

The workspace shows the shape of the structure including specified cross-section of members. These cross-sections may be switched off with the help of the setting "**Draw sections**" that is placed in the input frame in the tab "**Members, profiles**". Also the size of these cross-sections may be changed in the same place with the help of appropriate slider. Workspace also displays the loads specified in the load case that is selected as an active one in the table of load cases in the tab "**Load cases, load**".




*Generator window*

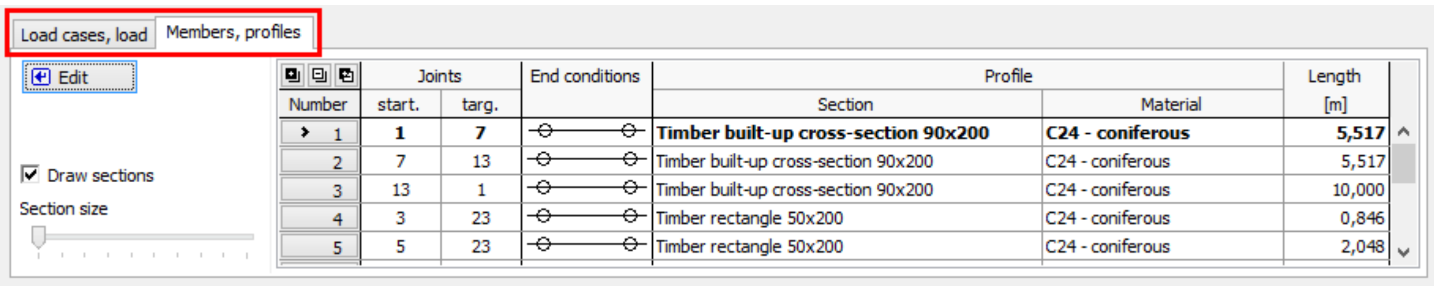
Input of the structure may be done with the help of the wizard that may be launched by the button "**Wizard**" in the vertical toolbar "**Structure**". Any part of the wizard may be launched again with the help of the dedicated buttons "**Shape**", "**Webbing**", "**Dimensions**" and "**Profiles**" in the toolbar. Also button "**Bottom chord**" for the input of camber is included in the toolbar.

## Load cases, load

The table in the tab "**Load cases, load**" is suitable for the input of the basic load cases including loads. Input is disabled for empty structure. The basic load cases can be added in the dedicated window, that can be launched by the appropriate button ("**Self-weight**", "**To frame**", "**Roofing**", "**Ceiling**", "**Snow**" and "**Wind**". Load cases may be modified or removed using the buttons "**Edit**" and "**Delete**".

## Members, profiles

This tab contains a table with member properties that were specified in the part "**Profiles**". The profile (cross-section and material) of any member may be changed in the window "**Edit profile**" that may be launched by double-click in the table or by the button "**Edit**".



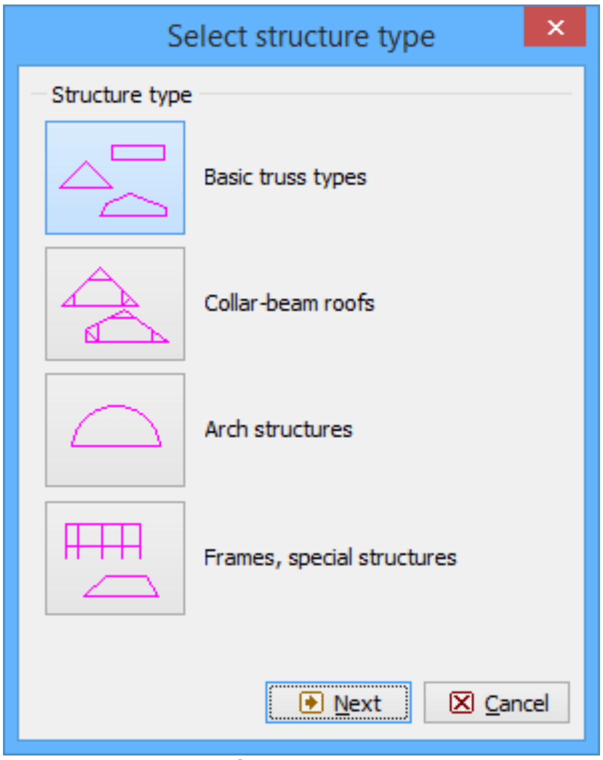

*Tabs for input of loads and edit of member properties*

The structure will be transferred into the program "**Fin 2D/3D**" after closing the window by the button "**OK**". The window may be also closed without transfer of the structure with the help of the button "**Cancel**". The inputs are stored in the memory, it is possible to launched the generator again and continue with the input. Any parameters specified in the generator may be changed later in the main application.

# Type, webbing, dimensions

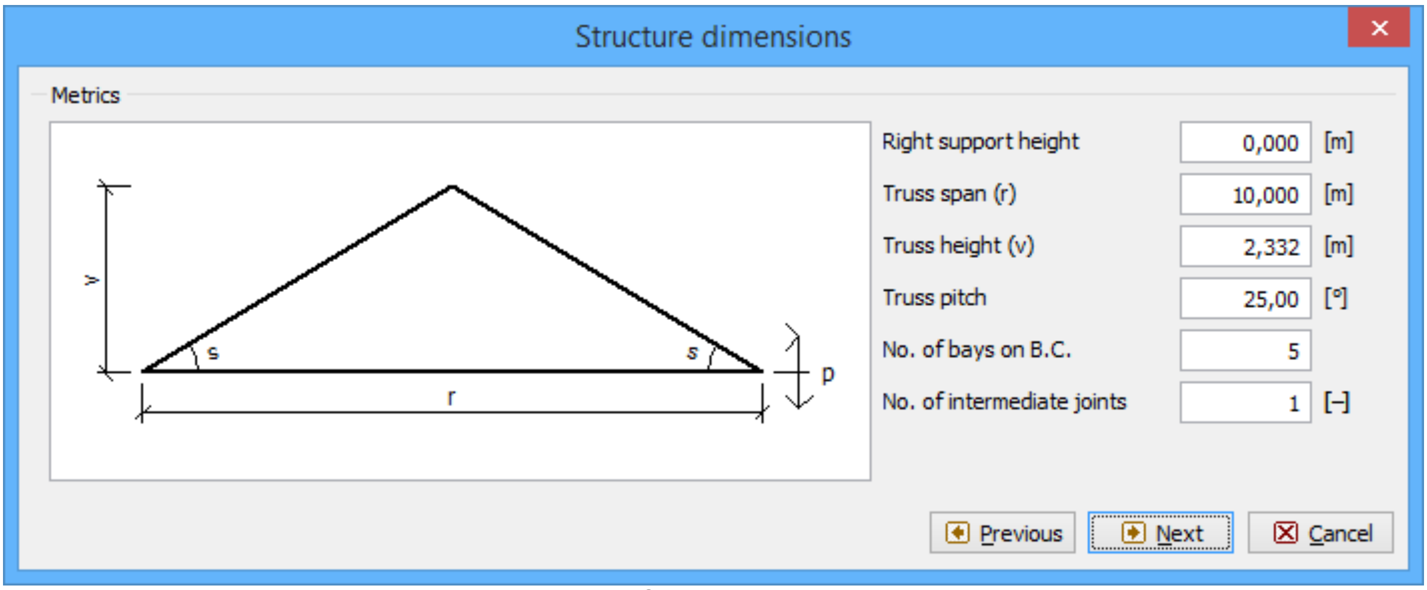
The wizard consists of few windows with the input of geometry and dimensions of the structure. The navigation between the windows can be done by the buttons "Next" and "Previous". First window "**Select structure type**" contains an option to select basic type of the structure (truss, attic structure, frame etc.). The windows "**Selection of structure form**" and "**Selection of webbing**" follow.




*Window "Select structure type"*

## Structure dimensions

Following window contains dimensions of the structure. Some input fields are connected togehther, input of one value will specify also the second value. For example, truss height is calculated automatically according to the specified pitch and span. The button "**Next**" switches the wizard into the next window "**Profiles in groups**".




*Window "Structure dimensions"*

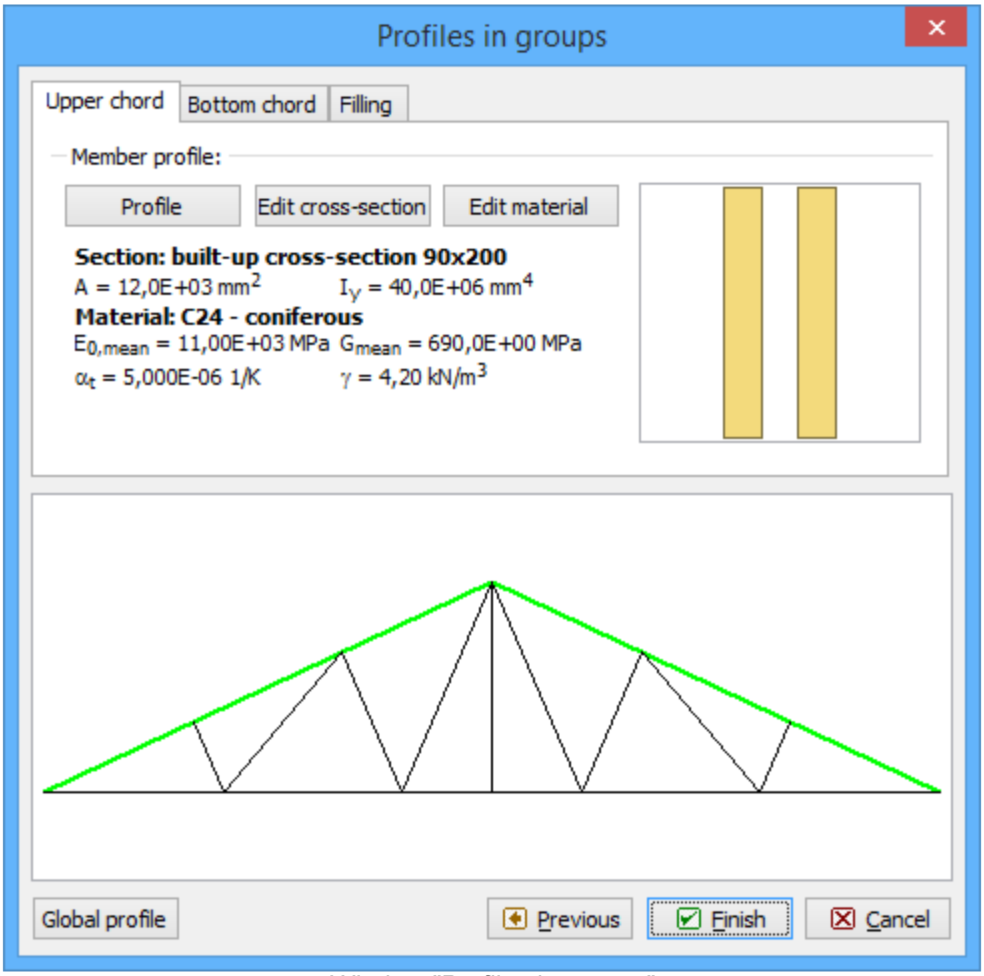
# Profiles

The window "**Profiles in groups**" is dedicated for batched input of material and cross-sectional characteristics for structural members. The characteristics may be specified in two ways:

- The input of characteristics for certain groups (e.g. top chords/bottom chords/webs for trusses) may be done in the upper part of the window. Any group has its own tab with dedicated tools for input. The characteristics are organized in the window "**Edit profile**" that may be launched by the button "**Profile**". The already specified cross-section may be changed in the window "**Cross-section editor**" that may be launched by the button "**Edit cross-section**". The material may be changed in the window "**Materials catalogue**" (button "**Edit material**".
- The identical characteristics for all members may be specified with the help of the button "**Global profile**" in the left bottom corner of the window. The input is done in the window "**Edit profile**". The specified profile is automatically copied into all tabs in the upper part of the window. It may be changed there for certain group without any limits.

The characteristics are transferred automatically into the main window after clicking on the button "**Finish**". The button "**Previous**" navigates back to the window "**Structure dimensions**".

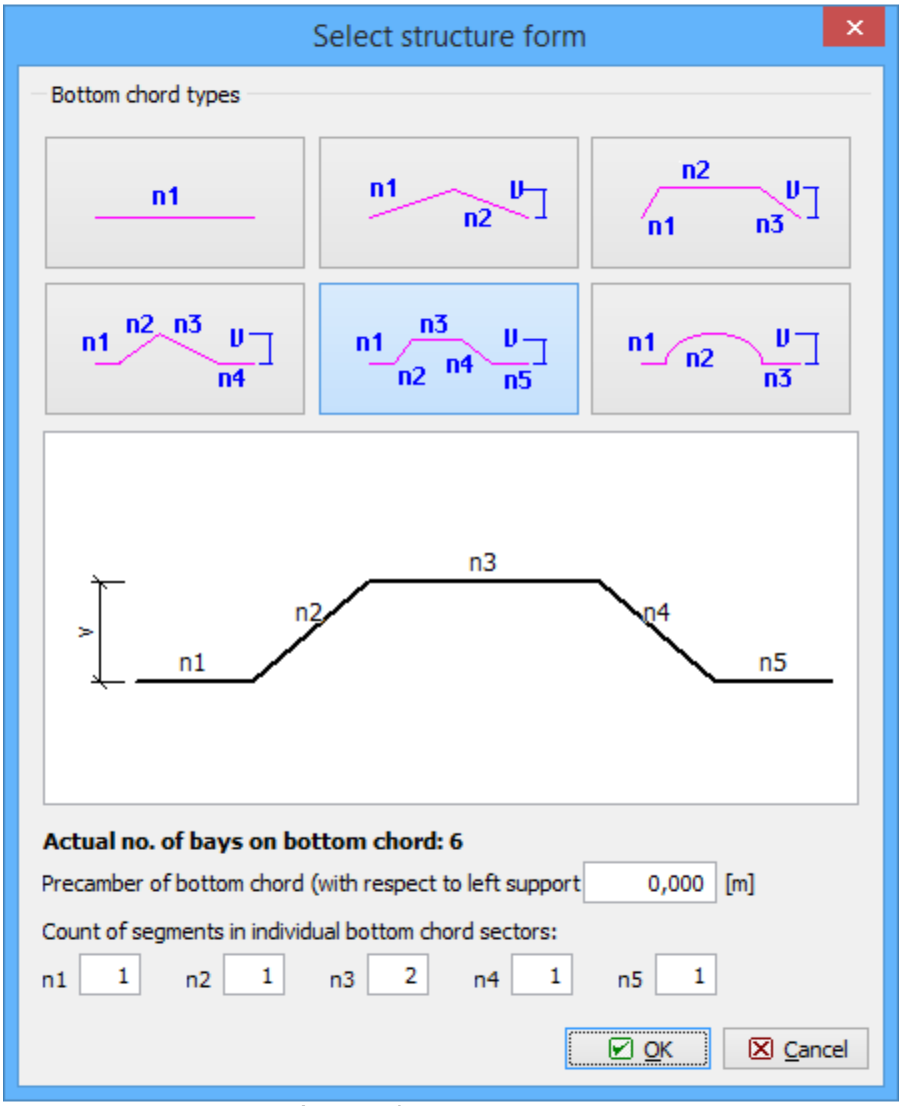



*Window "Profiles in groups"*

## Bottom chord

The camber of trusses may be specified in this window. Upper part contains few buttons with types of camber, bottom part contains the input fields for the input of camber value and number of bays for any sector of the bottom chord. The total number of bays depends on the value specified in the window "**Structure dimensions**". The sum of number of bays in all sectors has to be identical to the total number of bays. The software automatically modifies the number of bays in following sector during the input. The truss shape in the main window is updated automatically after clicking on "**OK**".




*Shape of the bottom chord*

# Load

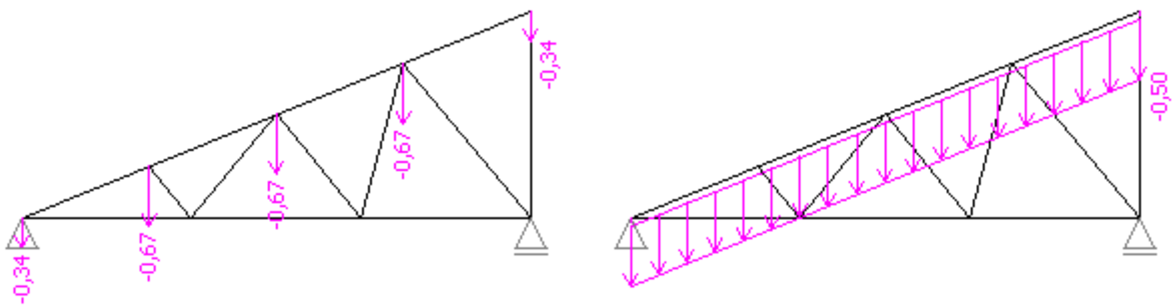
The new load cases including specified load may be entered easily with the help of this window. The input is divided into two tabs: "**Load case**" and "**Load**".

## Load case

This tab contains general parameters of the new load case. These properties are described in the chapter "**Load case**". The range of available options for any setting depends on the load type (self-weight, roofing, snow, wind etc.). These settings aren't limited during following work.

## Load

The values of loads may be specified in this tab. The load may be specified as a point load (option "**Per joint**") or as a linear load (option "**Per member**"). For the option "**Per joint**", the point loads are inserted both into joints with connected webs and into into intermediate joints. These joints are inserted into the structure with the help of the setting "**Number of intermediate joints**" in the window "**Structure dimensions**".

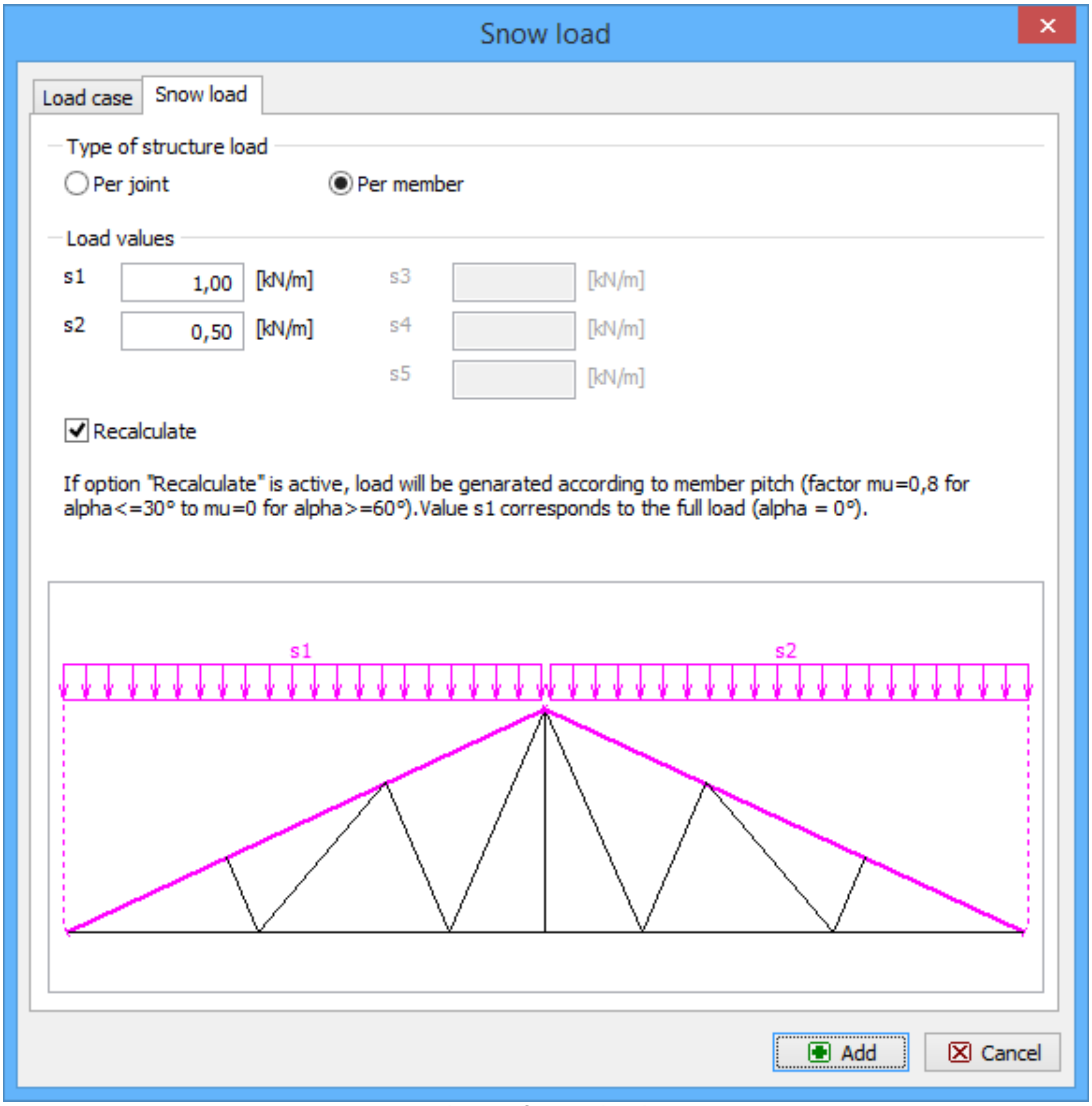



*Loading per joint and per member*

The values should be specified in *kN/m*, orientation of the positive direction is displayed in the structure view. Gravitational load should be entered as the positive value, wind pressure should be entered as a negative value of the load.

The snow load may be reduced automatically by the factor $\mu_1$ according to the figure *5.1* of EN 1991-1-3 with the help of the setting "**Recalculate**". The value of this factor depends on the pitch of the roof.




*Input of snow load*

# Polygons

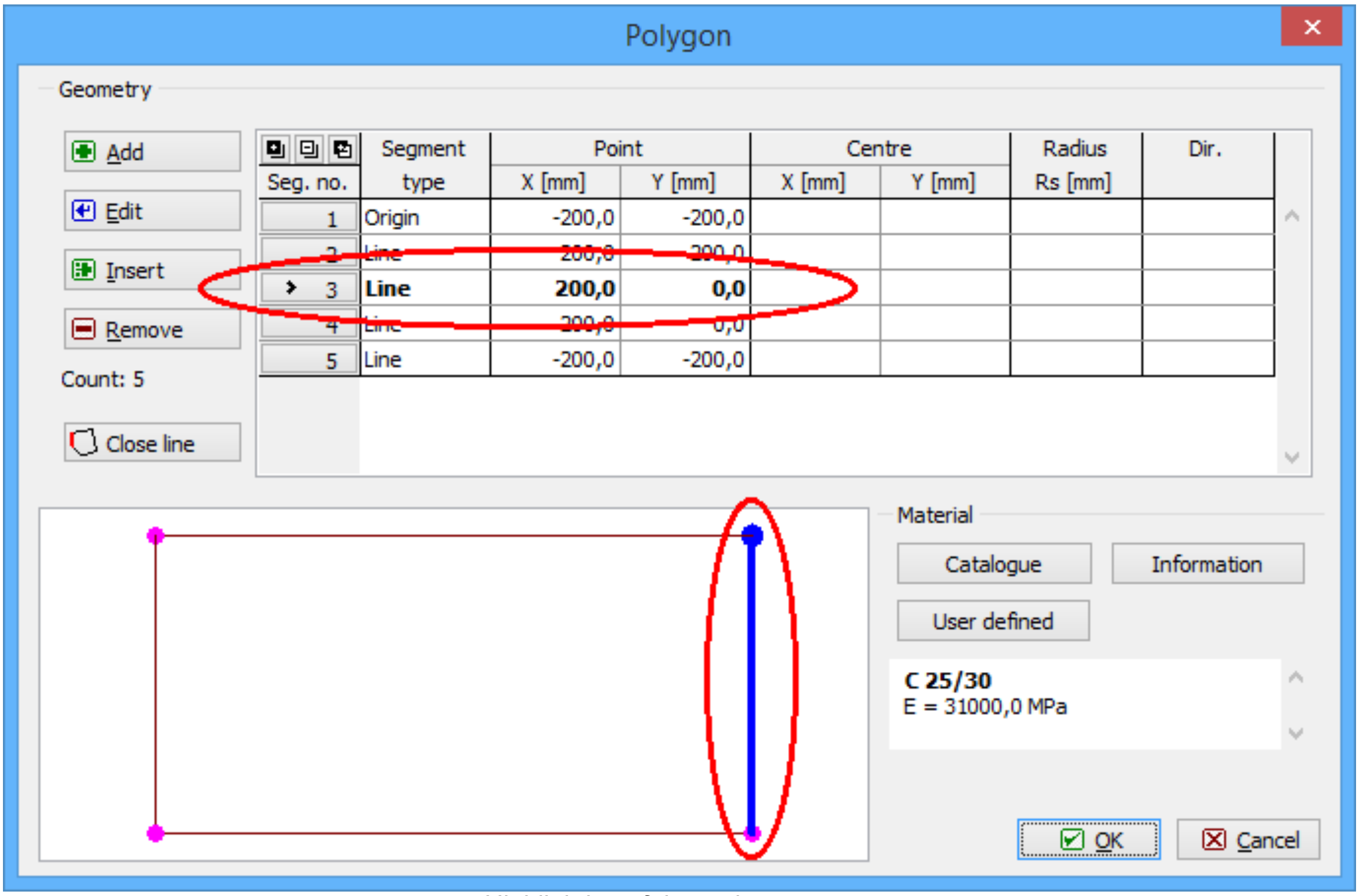
## Polygon

This window contains tools for the input of polygonal objects. Any polygon may consist of unlimited number of segments, it means lines or arches. Any segment is specified by the end point and the type. The end point of previous segment is automatically considered as the beginning of the following one. The segments are organized into the table "**Geometry**". Segment number *1* is the beginning of the polygon, the last segment has to be identical with the beginning.

Segments may be entered and edit with the help of toolbar on the left side of the table. Segment properties are organized in the window "**Edit segment**". The button "**Add**" inserts new segments to the end of the list, button "**Insert**" adds the segment above the active one. The polygon may be closed automatically with the help of the button "**Close line**".

The left bottom corner shows the polygon shape. The active segment in the table is highlighted by the blue colour in the polygon view.




*Highlighting of the active segment*

## Edit segment

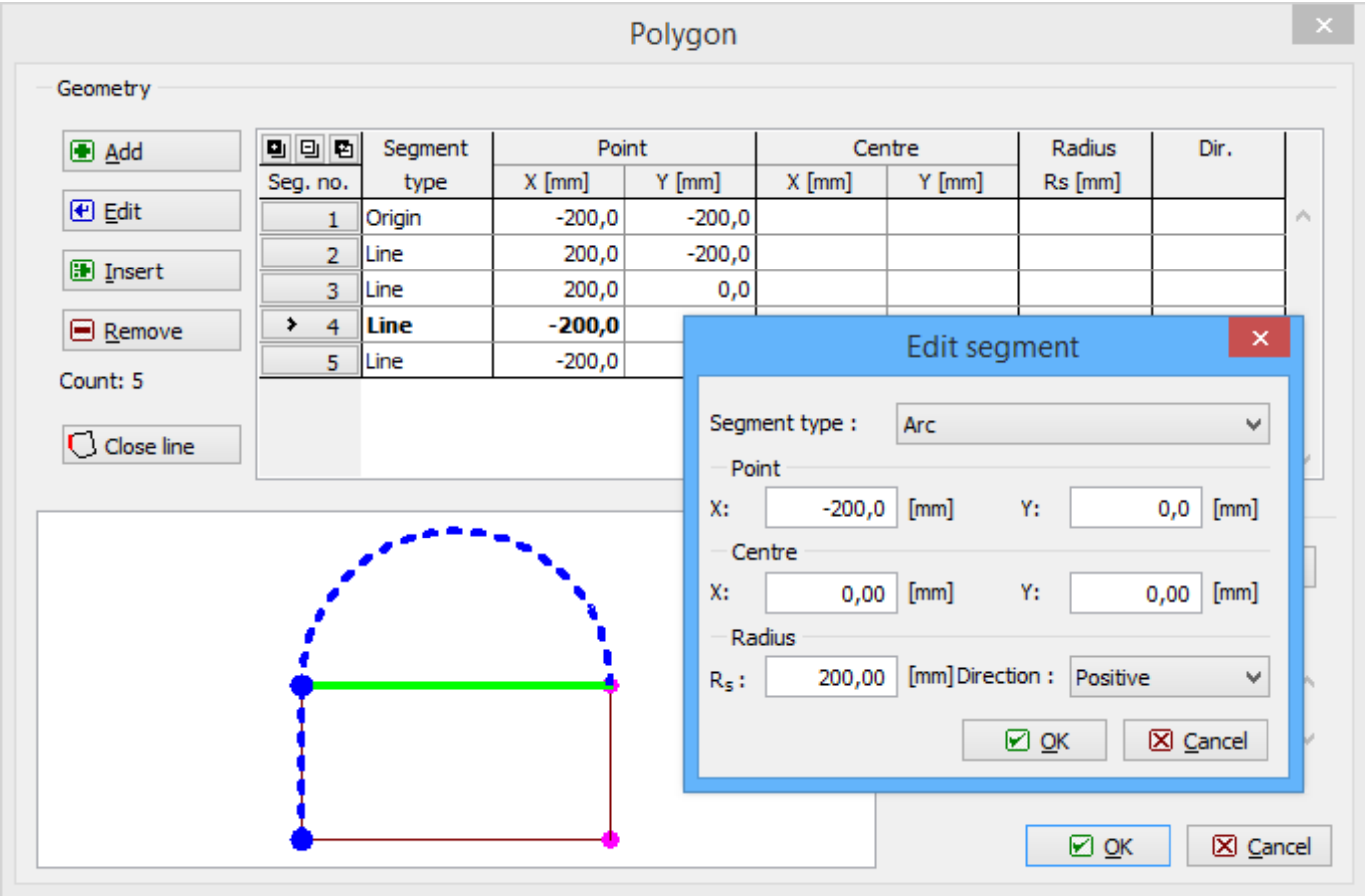
The geometry of particular polygon segment may be changed with the help of this window. Any segment is specified by the end point and the type. The end point of previous segment is automatically considered as the beginning of the following one.

These segment parameters may be specified here:

- **Segment type** – The geometry of the segment. Available options are "**Line**" and "**Arc**".
- **Point** – The end point of the segment specified with the help of coordinates *X, Y*
- **Centre** – The arc centre specified with the help of coordinates *X, Y*. This option is enabled only for the segment type "**Arc**".
- **Radius** – The radius of the arc. The value "**Direction**" changes the orientation of the arc (positive value creates convex shape when using input in anti-clockwise direction). This option is enabled only for the segment type "**Arc**".

Any changes are automatically shown in the polygon view in the window "**Polygon**".

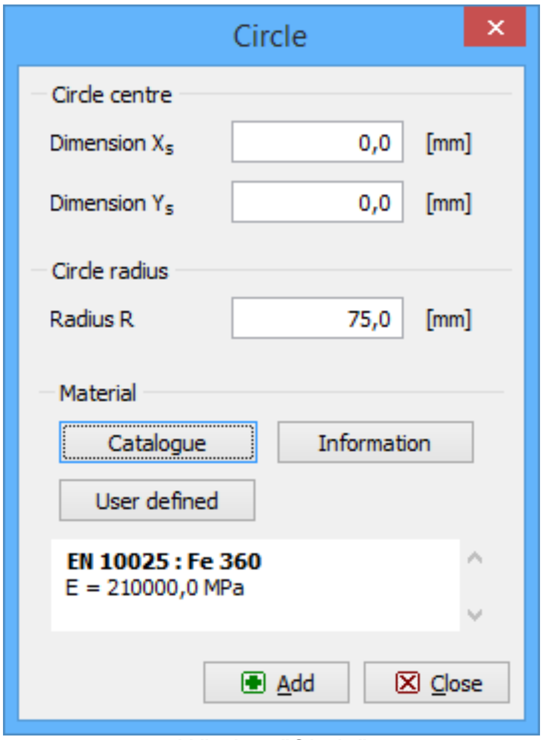



*Window "Edit segment" including the updated polygon view in the parent window*

## Circle

This window is dedicated for the input of the circular object. The geometry is specified by the circle centre (coordinates $X_S$ and $Y_S$ according to the global coordinate system) and radius $R$.

The bottom part may contain input of material in certain cases (not for openings).




*Window "Circle"*

# Loading

# Load case

This window contains complete properties of the entered load case. Every load case is characterized by the name and by the ID code that is placed in front of the name. This ID code consists of the unique load case number and prefix. The prefix represents the type of load:

| | |
|---|---|
| **G** | • Permanent load |
| **Q** | • Variable load |
| **A** | • Accidental load |
| **W** | • Variable load - wind |
| **S** | • Variable load - snow |

It means that first load case may have ID code *G1* providing that the type of this load case is "permanent". ID code *Q5* means that the load case represents the variable load and is placed on the fifth position in the list of load cases. ID code is used in the description of load combinations.

## Code of load case

Code of load case determine the type of load that can be entered into the load case. These options are available:

| | |
|---|---|
| **Self-weight** | • Loads in this load case represent the self weight of the structure. These loads are generated automatically by the software. Only one load case with this code per project is permitted. |
| **Force** | • Any forces or moments may be added into these load cases. The majority of loads are defined in the load cases with this code. The number of these load cases isn't limited. |
| **Deformation** | • Deformations of supports may be added into these load cases. The number of these load cases isn't limited. Available only in programs "**Fin 2D**" and "**Fin 3D**". |
| **Temperature** | • Thermal loads may be added into these load cases. The number of these load cases isn't limited. Available only in programs "**Fin 2D**" and "**Fin 3D**". |

## Load type

The load type represents the load character with regards to the variability in time. The range is based on the classification according to the chapter *4.1.1.* of EN 1990. Variable loads are also divided in accordance with the table *2.1* of EN 1995-1-1 (design of timber structures).

## Load factors

Partial load factors $\gamma_f$ may be specified in this part. This factor takes account of the possibility of unfavourable deviations from the representative values of the load. Favourable ($\gamma_{f,inf}$) and unfavourable ($\gamma_{f,sub}$) effect of permanent loads shall be distinguished. Default values of factors are based on the table *A1.2(B)* of EN 1990.

## Category

Range of available categories respects the sorting in the table *A1.1* of EN 1990. This category is determinative for the selection of combination factors $\psi_0$, $\psi_1$ a $\psi_2$ for variable loads. Category "**User defined input**" gives the option to specify arbitrary values of the combination factors.

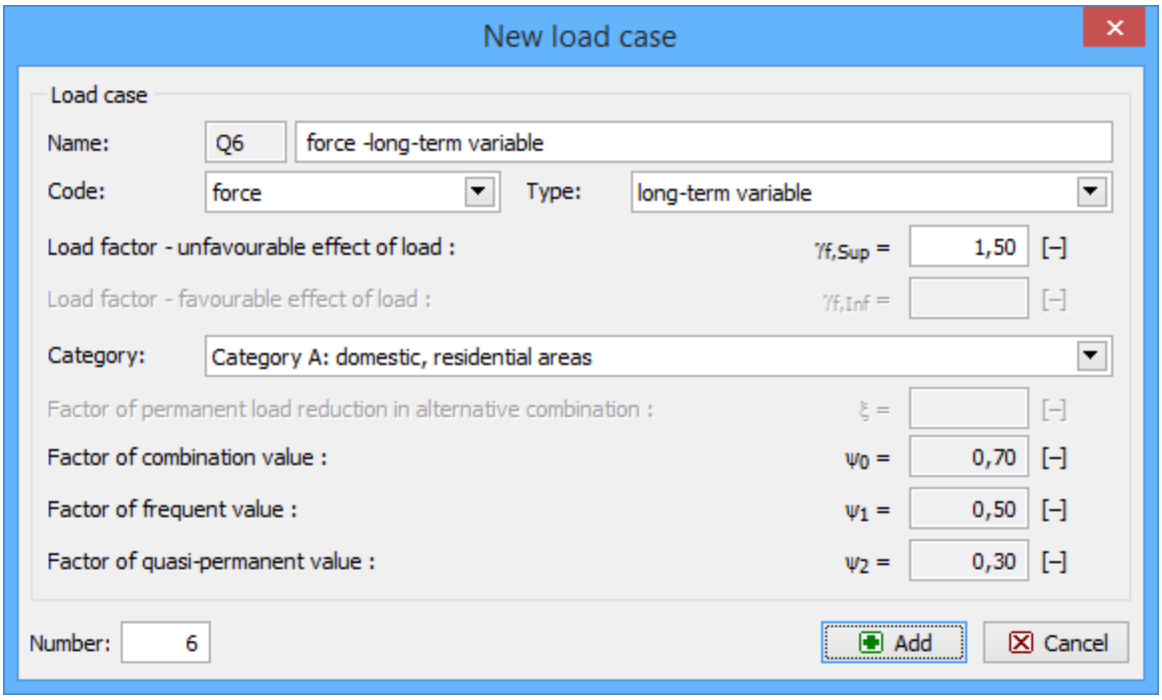
## Combination factors

The default values of these factors are based on the standard EN 1990 and depend on the category of the load case. Arbitrary values may be specified for the user defined input. Following factors are included:

| | |
|---|---|
| $\xi$ | • **Factor of permanent load reduction in alternative combination** – factor for permanent loads that is used in the alternative combinations for ultimate limit states (combinations according to the formula *(6.10b)* of EN 1990). |
| $\psi_0$ | • **Factor of combination value of the variable load** – combination factor for variable loads that is used in the combinations both for ultimate and serviceability limit states. |
| $\psi_1$ | • **Factor of frequent value of the variable load** – combination factor for variable loads that is used in the accidental combinations and in the combinations for serviceability limit states. |
| $\psi_2$ | • **Factor of quasi-permanent value of the variable load** – combination factor for variable loads that is used in the accidental combinations and in the combinations for serviceability limit states. |

Left bottom corner shows the ordinal number of the load case. The order of load cases in the list of load cases may by changed with the help of this value.




*Window "Load case"*

# Combination

This window contains properties of a combination of load cases. The input field in front of the combination name shows the content of the combination. This brief description consists of the ID codes of the load cases. Main variable loads are at the beginning and are separated by the colon.

## Type

These combinations are available for ultimate limit state:

**Basic** • The fundamental combination according to the formula *(6.10)* of EN 1990

**Alternative** • The combinations according to the formulas *(6.10a)* and *(6.10a)* of EN 1990. Two combinations are considered in the analysis for this option. First combination contains reduced permanent load cases, second one contains reduced main variable load.

**Accidental** • The accidental combination according to the formula *(6.11)* of EN 1990

Following combinations are available for serviceability limit state:

**Characteristic** • The combination according to the formula *(6.14)* of EN 1990

**Frequent** • The combination according to the formula *(6.15)* of EN 1990

**Quasi-permanent** • The combination according to the formula *(6.16)* of EN 1990

**Final deformation** • The combination for the calculation of final deflection for timber structures. These combinations are based on the chapter *2.2.3(5)* of EN 1995-1-1. These combinations shows relevant values only for deformations. Internal forces are increased due to creep effect and shouldn't be used for the design.

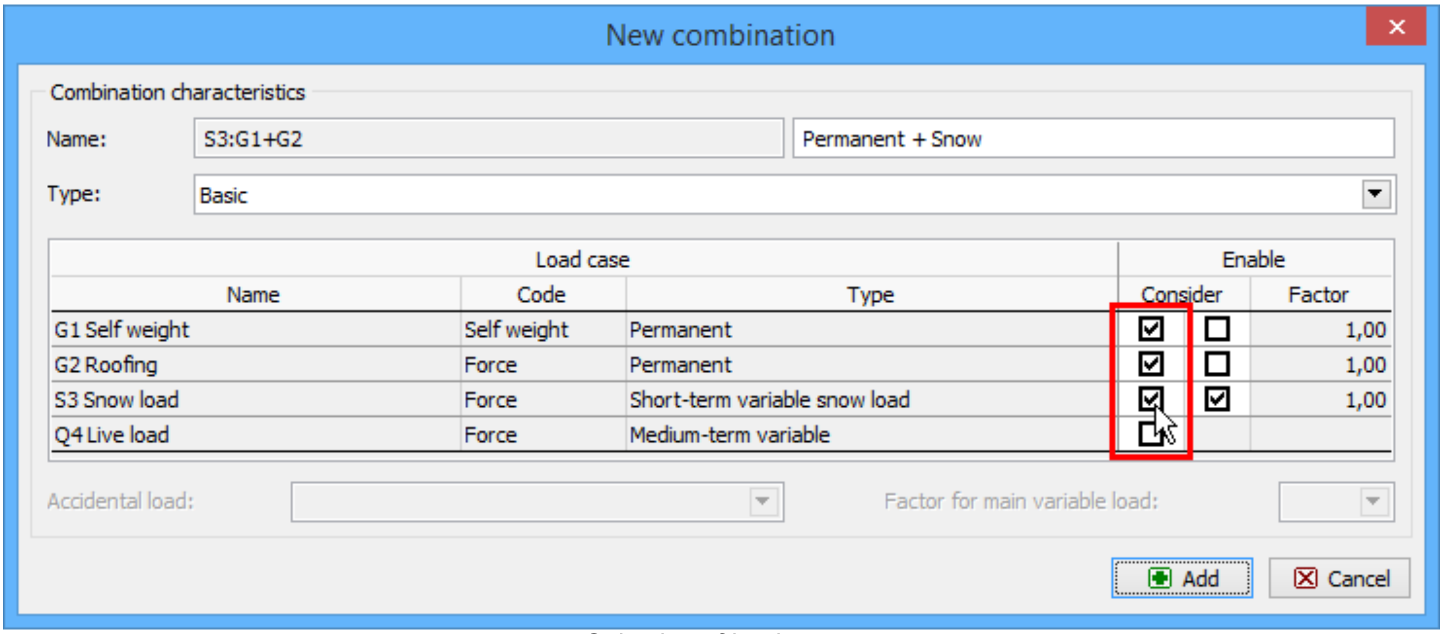
## Selection of load cases

The selection of the load cases included in the combination can be done in the table with the list of load cases. Any load case may be added into the combination using the first check box in the column "**Consider**". Second check box sets the favourable effect of permanent loads (partial factor $\gamma_{f,inf}$) and alternatively sets the variable load as a main one in the combination. Number of main variable loads for one combination isn't limited.

The accidental load may be selected for combination type "**Accidental**". This load case can be selected in the bottom part from the list of entered accidental load cases. Accidental combinations may be entered also without any accidental load case. This option is common for the analysis of fire resistance of the structures. The option "**Not specified**" in the list of accidental load cases has to be selected in these cases. The combination factor ($\psi_1$ or $\psi_2$) has to be specified for the accidental load:

- $\boldsymbol{\psi_1}$ - Factor of frequent value of the variable load
- $\boldsymbol{\psi_2}$ - Factor of quasi-permanent value of the variable load



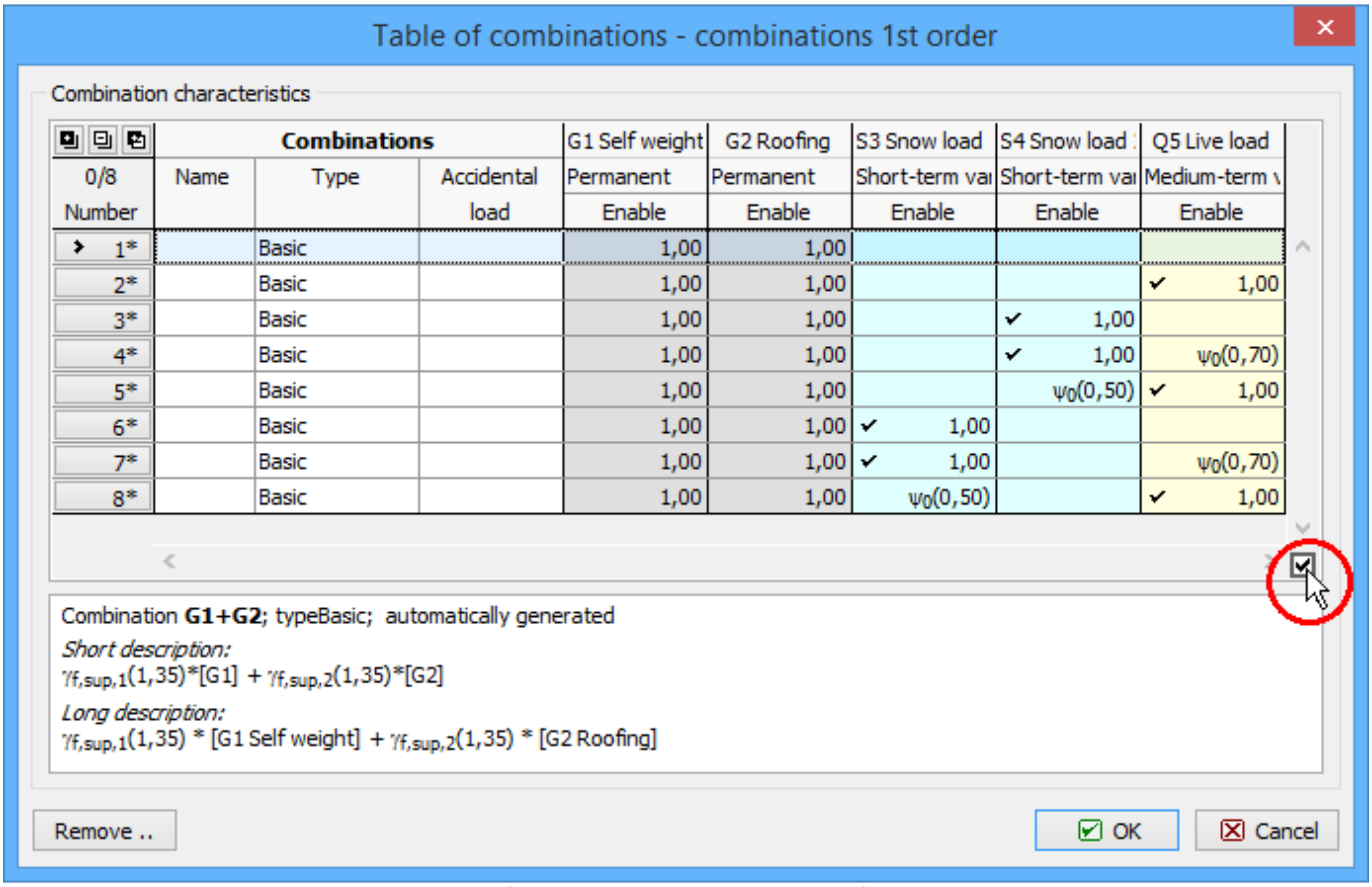

*Selection of load cases*

## Table of combinations

This window shows the overview of all combinations in the project including their combination factors. Combinations are organized in rows, columns represent particular load cases. The name, combination type and content are shown for any combination. Bottom part shows the detailed description including all partial factors for the active combination.

The load case types may be highlighted in the table with the help of check box in the right bottom corner of the table. This highlighting makes the table more transparent.

The active or selected load cases may be deleted with the help of button "**Remove...**".




*Highlighting of load case types in the table of combinations*

## Generator of combinations

This window contains parameters of batch input of load combinations. Due to character of the standard EN 1990, large number of load combinations may be created by the generator. As such number may cause slowing down the program work, the expected number of generated combinations is shown in the right bottom corner of the window. User is able to check this number and modify the specified rules to get acceptable number of generated combinations. Upper part of the window contains tools for input of combination rules, bottom part contains generator settings.

## Mutually interacting load cases

Selected load cases may be merged together into groups. Load cases in these groups will be in all combinations together. Permanent and variable loads can't be merged into one group. The groups of permanent load cases won't be considered if the setting "**All permanent loads in combination**" in "**Characteristics of generator**" is switched on, as any combination will contain all permanent loads. The merged permanent load cases will be considered only for distinguishing the favourable and unfavourable effects of loads (provided that the setting "**Permanent loads act only unfavourably**" isn't switched on).

## Excluded interaction of load cases

The load cases that can't appear together in one combination may be specified in this part. Both load cases and groups of load cases may be excluded here. The rules aren't limited by the type of load cases

Exclusion may be specified using two ways:

**Mutual exclusion**
- More load cases can be added in such group. Generated combination will contain no more than one load case from this group. This option is suitable e.g. for more variants of snow load applied to the structure.

**Exclusion by couples**
- This option may be used for cases where one load case should be excluded with more different load cases. The load case that should be excluded with others has to be selected in the first column. Arbitrary number of load cases may be selected in the second column. These load cases won't be considered in combinations together with the load case specified in the first column. This option is suitable e.g. for exclusion of construction load and loads applied to the finished structure.

## Load cases and groups acting as the main variable load

Any variable load case is considered as a main variable load in the default mode. This mode may be switched off with the help of the check box "**Automatically create main variable loads**". After that, certain load cases may be removed from the list of main variable loads or may be merged together. These merged load cases will be considered as main variable load in one combination.

## Characteristics of generator

**Keep existing combinations**
- The existing combinations will remain in the project and new combinations will be added after closing the window with the help of button "**Generate**".

**Remove all combinations**
- The existing combinations will be replaced by the new combinations after closing the window with the help of button "**Generate**".

**Remove generated combinations**
- The existing automatically generated combinations will be replaced by the new generated combinations after closing the window with the help of button "**Generate**". The existing manually specified combinations will remain in the project.

**Remove all combinations of the current type**
- The existing combinations of the selected type (in part "**Generate combinations**") will be replaced by the new combinations after closing the window with the help of button "**Generate**".

**Remove generated combinations of the current type**
- The existing combinations of the selected type (in part "**Generate combinations**") will be replaced by the new generated combinations after closing the window with the help of button "**Generate**". The existing manually specified combinations will remain in the project.

## Generate combinations

These combinations are available for ultimate limit state:

**Basic**
- The fundamental combination according to the formula *(6.10)* of EN 1990

**Alternative**
- The combinations according to the formulas *(6.10a)* and *(6.10a)* of EN 1990. Two combinations are considered in the analysis for this option. First combination contains reduced permanent load cases, second one contains reduced main variable load.

**Accidental**
- The accidental combination according to the formula *(6.11)* of EN 1990. The accidental load may be selected for combination type "**Accidental**". This load case can be selected in the bottom part from the list of entered accidental load cases. Accidental combinations may be entered also without any accidental load case. This option is common for the analysis of fire resistance of the structures. The option "**Not specified**" in the list of accidental load cases has to be selected in these cases. The combination factor ($\psi_1$ or $\psi_2$) has to be specified for the accidental load

Following combinations are available for serviceability limit state:

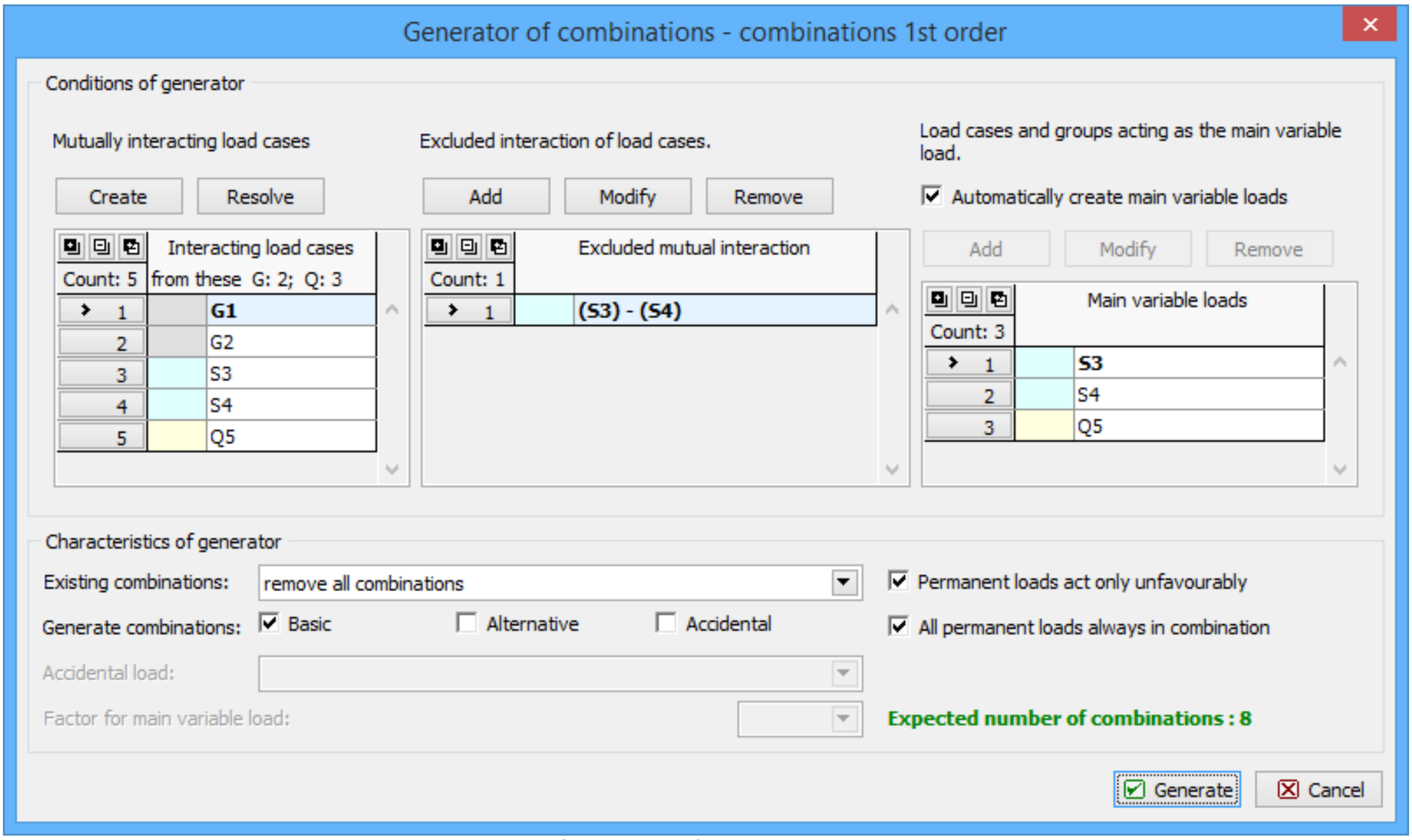
**Characteristic**
- The combination according to the formula *(6.14)* of EN 1990

**Frequent** • The combination according to the formula *(6.15)* of EN 1990

**Quasi-permanent** • The combination according to the formula *(6.16)* of EN 1990

**Final deformation** • The combination for the calculation of final deflection for timber structures. These combinations are based on the chapter *2.2.3(5)* of EN 1995-1-1. These combinations shows relevant values only for deformations. Internal forces are increased due to creep effect and shouldn't be used for the design.




*Generator of combinations*

# Concrete

## Standard selection

The design standard can be selected in this window. These design standards are available: "**EN 1992-1-1**" (Design of concrete buildings) and "**EN 1992-2**" (Design of concrete bridges). Different national annexes can be selected for both design standards. The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex. The values of certain parameters and partial factors $\gamma_M$ both for basic and accidental design situations can be specified for option "**User defined**".

Values of factors and parameters for all available national annexes are written in the chapter "**National annexes**".

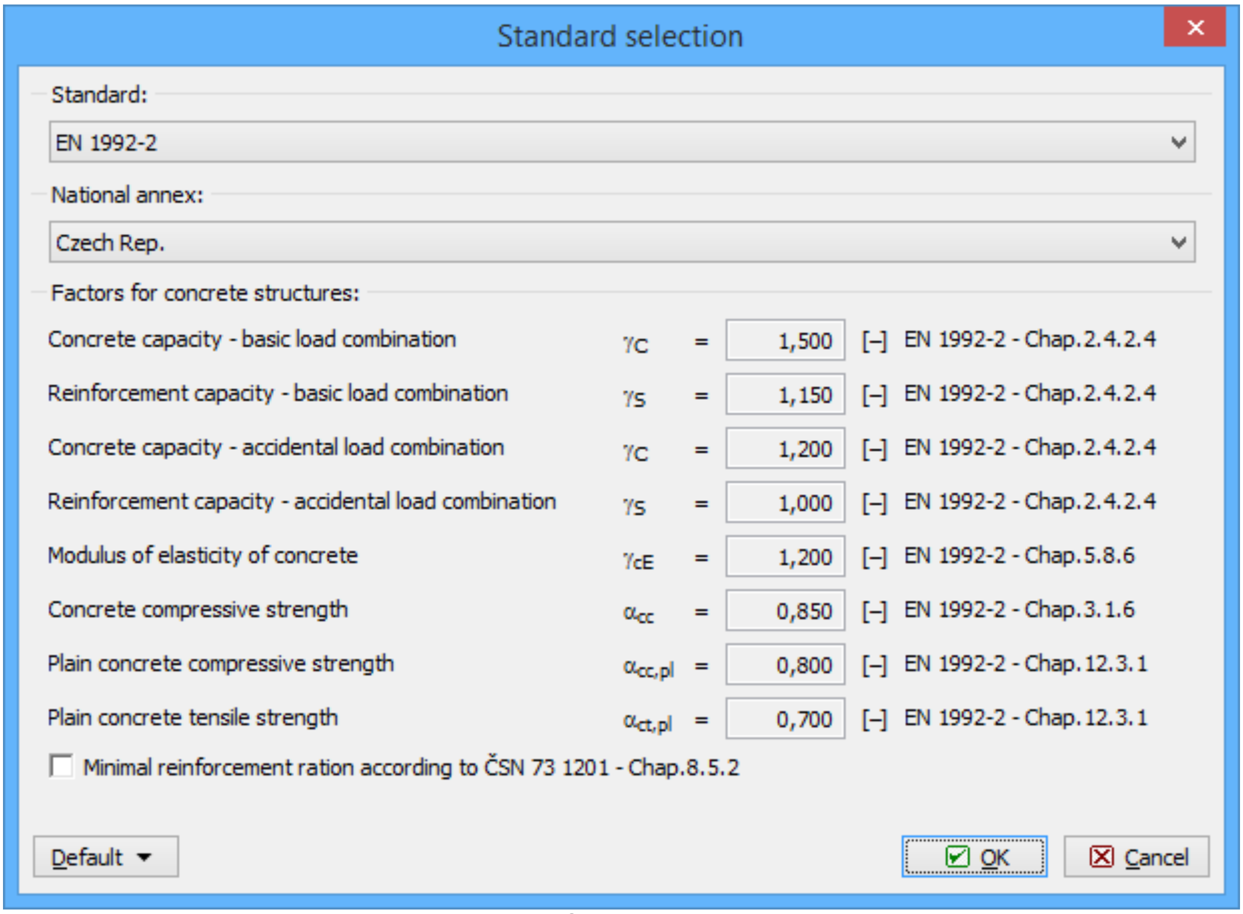
The minimum reinforcement area may be checked for member type "**Slab**" if the setting "**Minimum reinforcement ratio according to CSN 73 1201 - Chap.8.5.2**" is switched on. This verification is described in the chapter "**Structural rules**" of theoretical help.

Button "**Default**" contains these two tools:

**Adopt default settings** • Set the default values for all parameters.

**Save settings as default** • Set entered parameters as defaults for new projects.

Partial factors are described in the theoretical part of the help.



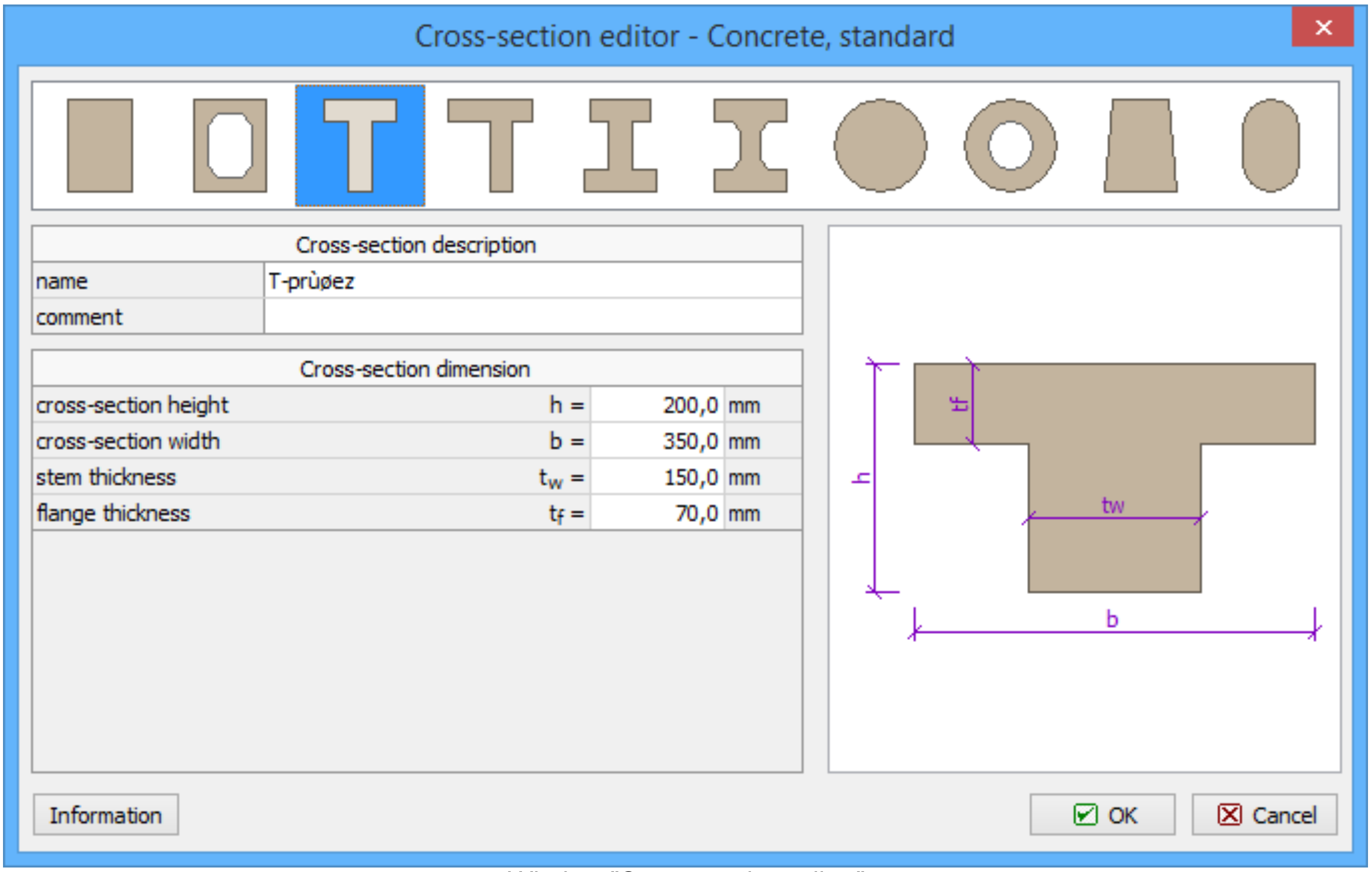

*Window "Standard selection"*

## Cross-section editor

The member cross-section can be modified in this window. The upper part contains library of available shapes (range differs for certain programs and task types). Dimensions can be entered in the table in the left part of the window. The meaning of dimensions is shown in the cross-section view in the right part of the window.

"**Information**" button in the left bottom corner shows complete list of cross-sectional characteristics.




*Window "Cross-section editor"*

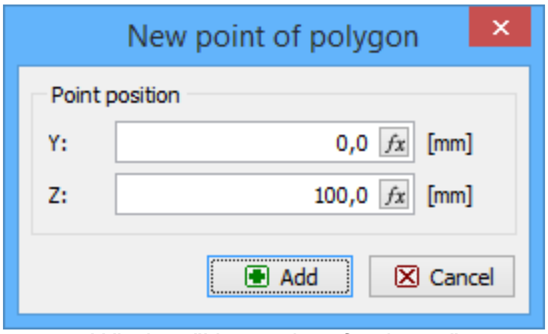
# General cross-section

The geometry of general cross-section may be specified in this window. There are two basic ways of geometry input that may be combined: graphical input in the workspace and numerical input using the coordinates of nodes. The geometry may be also imported from **.dxf* file using the function "**Import DXF**" in the part "**File**" - "**Import**".

The cross-sections entered with the help of database of pre-defined shapes may be modified in this window.

## Numerical input

New nodes of the cross-section may be entered numerically with the help of table "**Cross-section geometry** " in the left part of the window. The input is performed in the window "**New point of polygon**" that may be launched by the button "**+**" in the toolbar under the table. Modifications and deletion may be done with the help of the buttons "**Edit**" a "**Remove**". The calculator may be used for the calculation of the accurate coordinates. This calculator may be launched by the button "*fx*" in the input line.



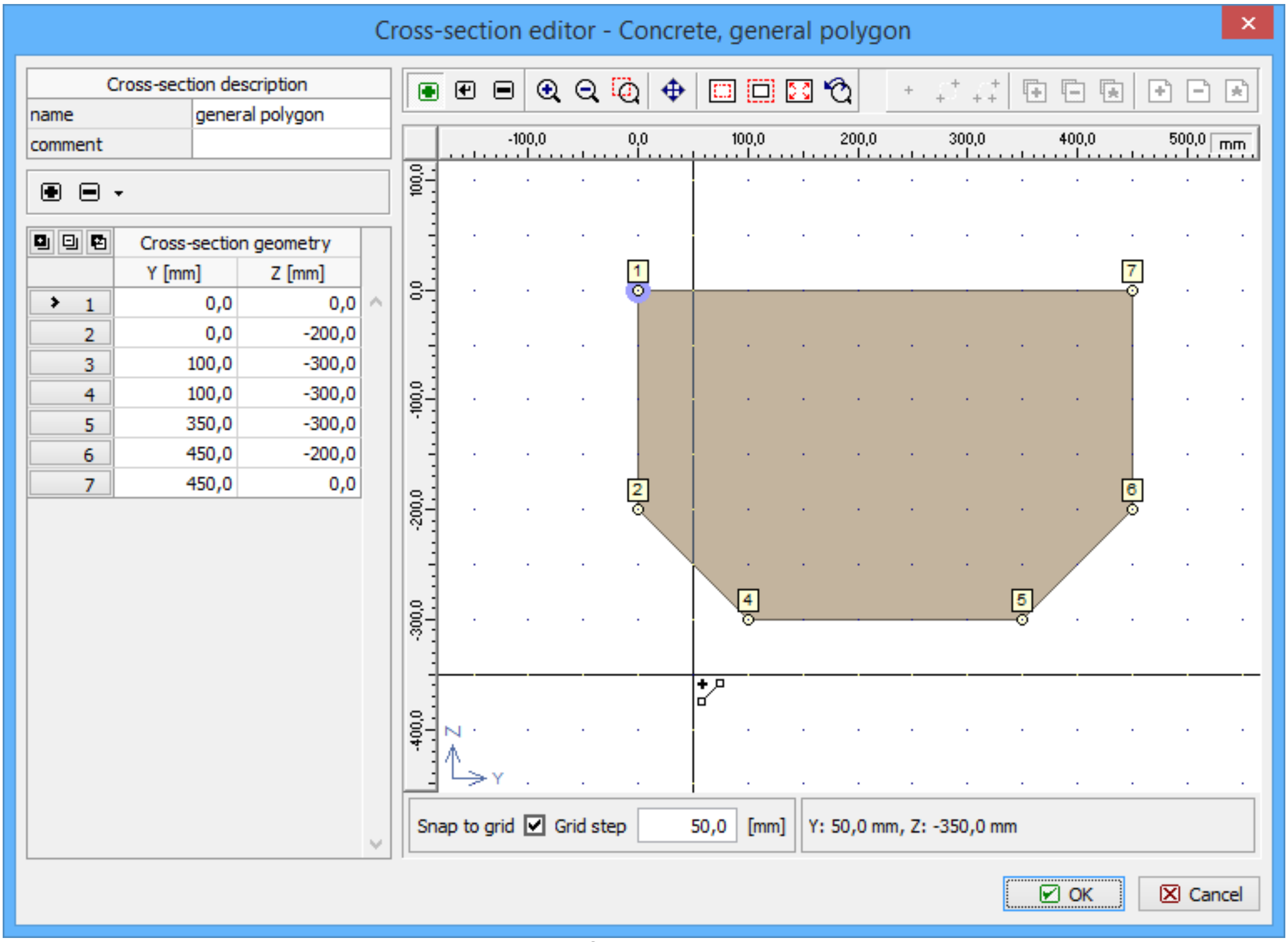

*Window "New point of polygon"*

The coordinates may be edit directly in the table. The nodes may be deleted using the button "**-**" in the toolbar above the table.

## Graphical input

The new nodes may be also specified on the workspace in the right part of the window. The appropriate mode may be selected in the toolbar "**Edit**". The new bars may be added by the cursor in the mode "**Add**". The grid may be used during the nodes input after switching on the setting "**Snap to grid**". Mode "**Remove**" deletes the existing nodes after clicking on them. The tools from toolbars "**Selections**" and "**Scale**" may be helpful during the work.




*Window "Cross-section editor"*

# Materials

The material properties of concrete and reinforcement can be specified in this window. It contains following inputs:

**Environment**
- The button "**Edit**" runs the window "**Environment**" where is possible to select the environmental conditions for analysed member. The environmental conditions affects the indicative strength class and the calculation of minimum cover.

**Concrete**
- This part contains buttons for input of concrete properties. The properties can be specified by using the strength classes from pre-defined database in window "**Materials catalogue - concrete**" (button "**Catalogue**") or by entering the properties numerically in the window "**Material editor - concrete**" (button "**User defined**").

**Longitudinal reinforcement**
- This part contains buttons for input of longitudinal reinforcement properties. The properties can be specified by using the strength classes from pre-defined database in window "**Materials catalogue - steel**" (button "**Catalogue**") or by entering the properties numerically in the window "**Material editor - steel**" (button "**User defined**").

**Shear reinforcement**
- This part contains buttons for input of transverse reinforcement properties. The properties can be specified by using the strength classes from pre-defined database in window "**Materials catalogue - steel**" (button "**Catalogue**") or by entering the properties numerically in the window "**Material editor - steel**" (button "**User defined**").

## Indicative strength class

This part shows the indicative strength class according to the table *E.1N* that is considered as a minimum one for specified environmental conditions. This requirement takes into consideration corrosion protection of reinforcement and protection of concrete attack.

Check box "**Aeration >4%**" affects the classification of the structure in accordance with the table *4.3N* that is necessary for the calculation of minimum cover. If ticked on, the limiting concrete strength grade for reduction of structural classification is reduced by *1*. This setting also affects the indicative strength class of structures exposed to rain and freezing (*XF1*, *XF2*, *XF3*) for certain national annexes (Czech republic).

The maximum size of used aggregate should be specified in the input field "**Maximum size of aggregate**". This value affects the minimum cover (the minimum cover should be increased for aggregate greater than *32mm* in accordance with the table *4.2*) and also the minimum distance between bars (chapter *8.2(2)* of EN 1992-1-1). This setting is available only for projects that have the setting "**Check bar spacing**" switched on in the main screen of the program.

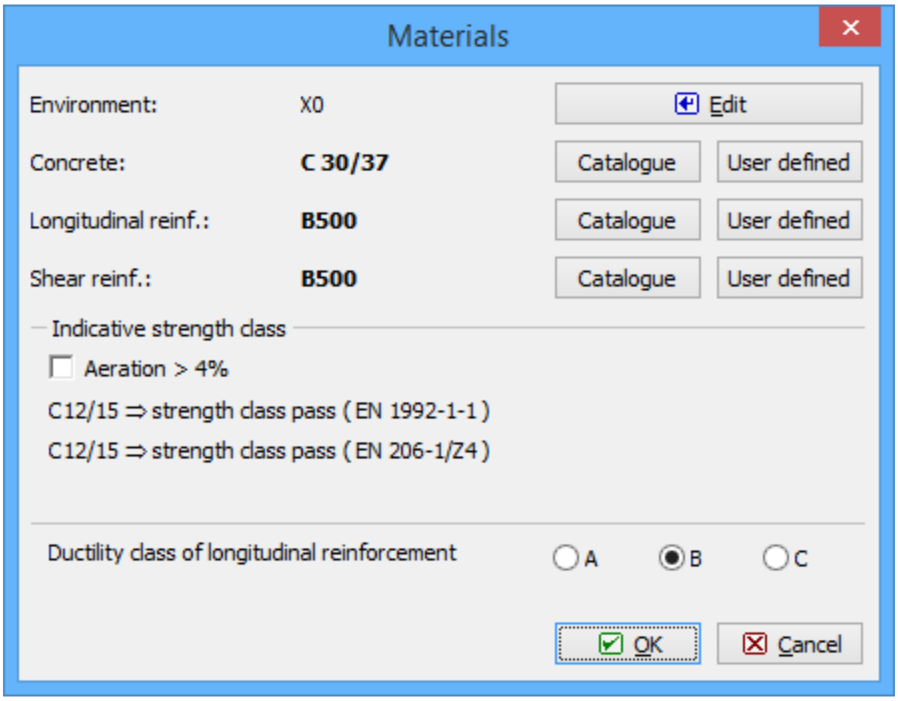
## Ductility class of longitudinal reinforcement

The ductility class can be specified in this part. This choice affects the value of the characteristic strain at maximum force $\varepsilon_{uk}$ and the minimum value of the factor $k=(f_t/f_y)_k$ for calculation of stress according to the table *C.1* of EN 1992-1-1.

## Fire *(only program Concrete Fire)*

These material parameters affects the fire resistance of the member:

| | |
|---|---|
| **Aggregate type** | • The aggregate type ("**Siliceous**" or "**Calcareous**") affects the stress-strain relationships of concrete under compression at elevated temperatures (chapter *3.2.2.1* of EN 1992-1-2) and thermal and physical properties of concrete (chapter *3.3*) |
| **Reinforcement type** | • The reinforcement type ("**Hot rolled**" or "**Cold worked**") affects the stress-strain relationships of steel at elevated temperatures (chapter *3.2.3* of EN 1992-1-2) |
| **Concrete moisture** | • The moisture content of concrete affects the value of specific heat of concrete in accordance with *3.3.2* of EN 1992-1-2 |
| **Parameter of thermal conductivity** | • This parameter is used for the calculation of the thermal conductivity $\lambda_c$ with the help of linear interpolation between the upper and lower limits. These limits are determined according to the chapter *3.3.3* of EN 1992-1-2. The lower limit of the thermal conductivity is used for the parameter equal to *0*, the upper limit of the thermal conductivity is used for the parameter equal to *1.0*. |




*Window "Materials"*

# Environment

The environmental conditions for a member can be specified in this window. The range of conditions is based on the chapter *4.2* of EN 1992-1-1. The conditions influence the calculation of minimum (indicative) strength class and minimum reinforcement cover. These procedures are described in the chapters "**Indicative strength class**" and "**Minimum cover**" of theoretical help.

Description of options:

## Corrosion induced by carbonation

| | |
|---|---|
| **X0 - No risk of carbonation** | • Concrete inside buildings with very low air humidity |
| **XC1 - Dry or permanently wet** | • Concrete inside buildings with low air humidity, concrete permanently submerged in water |
| **XC2 - Wet, rarely dry** | • Concrete surfaces subject to long-term water contact. Examples: most of foundations |
| **XC3 - Moderate humidity** | • Concrete inside buildings with moderate or high air humidity, external concrete sheltered from rain |

| | |
|---|---|
| **XC4 - Cyclic wet and dry** | • Concrete surfaces subject to water contact, not within exposure class XC2. Examples: cantilever walls, external structures |

## Corrosion induced by chlorides

| | |
|---|---|
| **X0 - No risk** | • No risk of corrosion induced by chlorides |
| **XD1 - Moderate humidity** | • Concrete surfaces exposed to airborne chlorides |
| **XD2 - Wet, rarely dry** | • Swimming pools, concrete components exposed to industrial waters containing chlorides |
| **XD3 - Cyclic wet and dry** | • Parts of bridges exposed to spray containing chlorides, pavements, car park slabs |

## Corrosion induced by chlorides from sea water

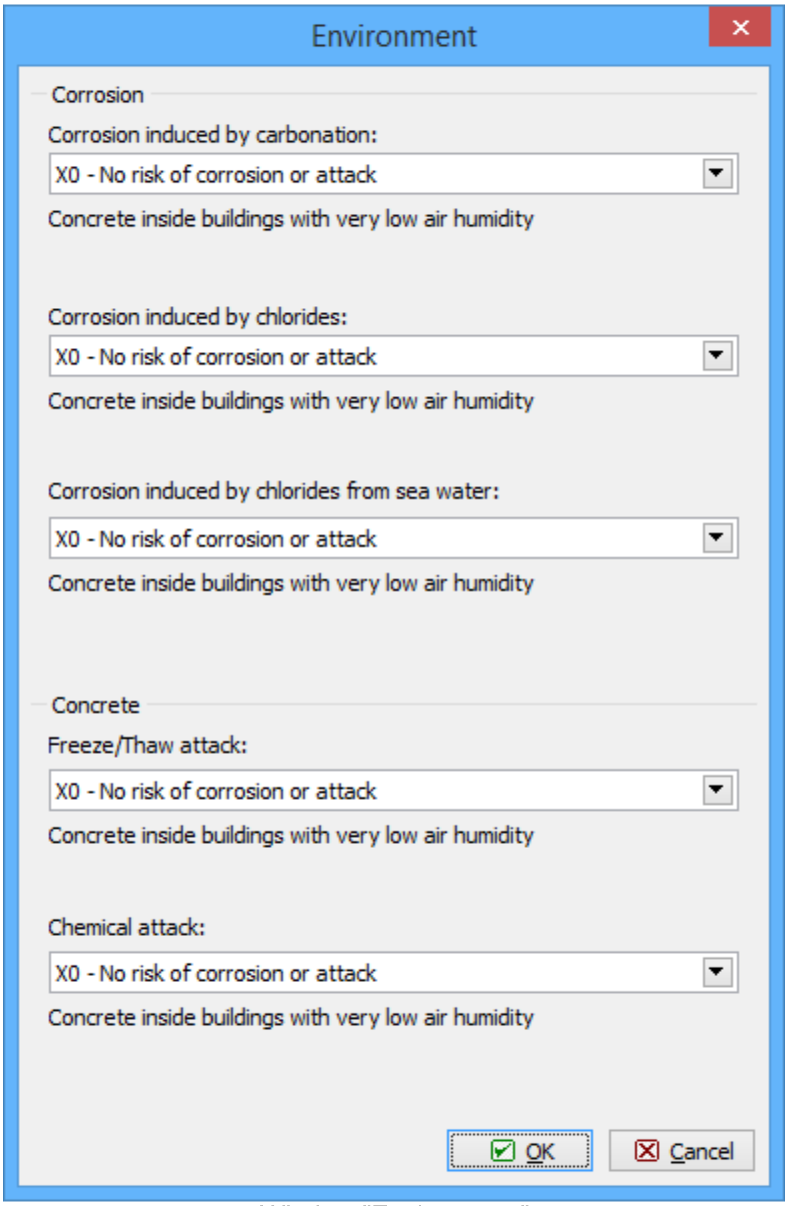
| | |
|---|---|
| **X0 - No risk** | • No risk of corrosion induced by chlorides from sea water |
| **XS1 - Exposed to airborne salt but not in direct contact with sea water** | • Structures near to or on the coast |
| **XS2 - Permanently submerged** | • Parts of marine structures |
| **XS3 - Tidal, splash and spray zones** | • Parts of marine structures |

## Freeze/Thaw Attack

| | |
|---|---|
| **X0 - No risk** | • Concrete inside buildings not exposed to rain and freezing |
| **XF1 - Moderate water saturation, without de-icing agent** | • Vertical concrete surfaces exposed to rain and freezing |
| **XF2 - Moderate water saturation, with de-icing agent** | • Vertical concrete surfaces of road structures exposed to freezing and airborne de-icing agents |
| **XF3 - High water saturation, without de-icing agents** | • Horizontal concrete surfaces exposed to rain and freezing |
| **XF4 - High water saturation with de-icing agents or sea water** | • Road and bridge decks exposed to de-icing agents, concrete surfaces exposed to direct spraycontaining de-icing agents and freezing, splash zone of marine structures exposed to freezing |

## Chemical attack

| | |
|---|---|
| **X0 - No risk** | • Concrete not exposed to aggressive chemical environment |
| **XA1 - Slightly aggressive chemical environment according to EN 206-1, Table 2** | • Natural soils and ground water |
| **XA2 - Moderately aggressive chemical environment according to EN 206-1, Table 2** | • Natural soils and ground water |
| **XA3 - Highly aggressive chemical environment according to EN 206-1, Table 2** | • Natural soils and ground water |



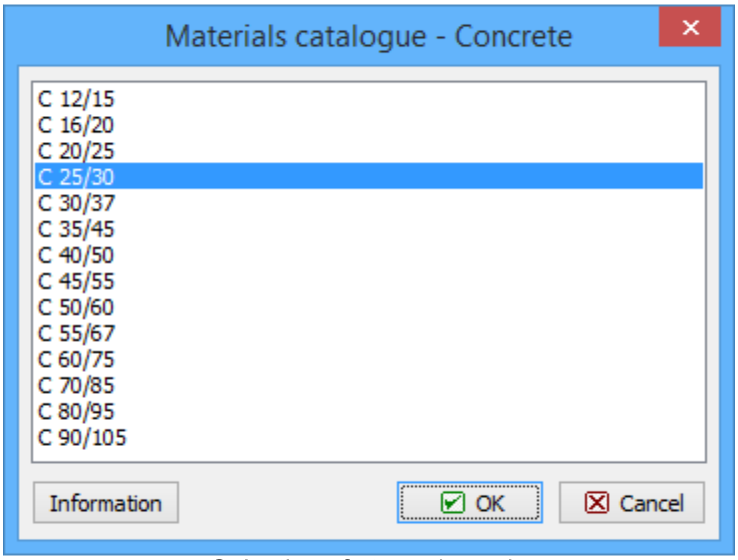

*Window "Environment"*

## Materials catalogue - concrete

This window contains the database of strength grades for concrete. The name of any grade consists of the characteristic compressive cylinder strength of concrete at *28* days ($f_{ck}$) in front of the slash and the characteristic compressive cube ($f_{ck,cube}$) behind the slash.

The complete list of material characteristics for selected grade can be opened using "**Information**" button.

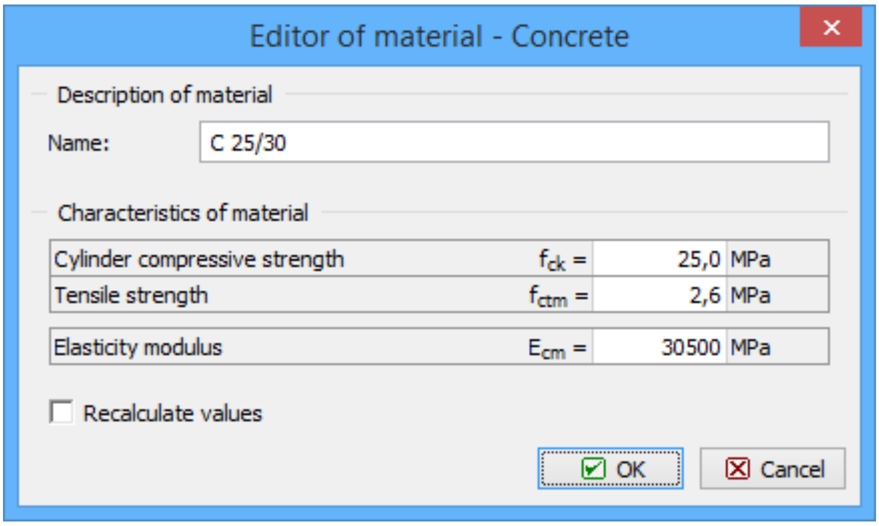
 




*Selection of strength grade*

## Material editor - concrete

The arbitrary material characteristics can be specified in this window. The quantities $f_{ctm}$ (mean value of axial tensile strength of concrete) and $E_{cm}$ (secant modulus of elasticity of concrete) may be obtained automatically using the characteristic compressive cylinder strength $f_{ck}$. This procedure can be switched on using the setting "**Recalculate values**". Calculations are based on the chapter *3.1* of EN 1992-1-1 and are described in the chapter "**Material characteristics**" of the theoretical part of the help.

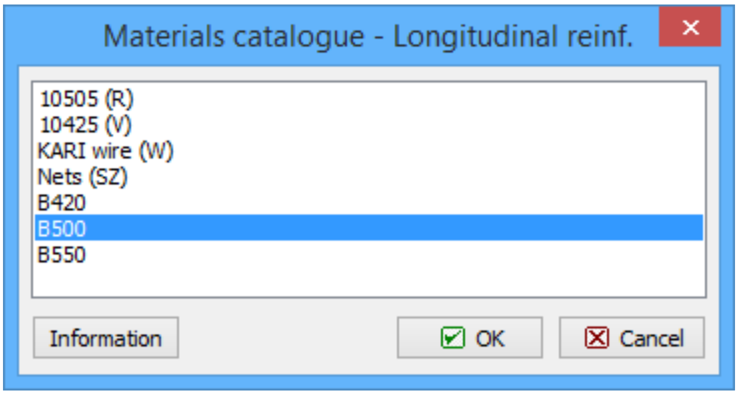



*Window "Material editor"*

## Materials catalogue - reinforcement

This window contains the database of steel reinforcement grades.

The complete list of material characteristics for selected grade can be opened using "**Information**" button.

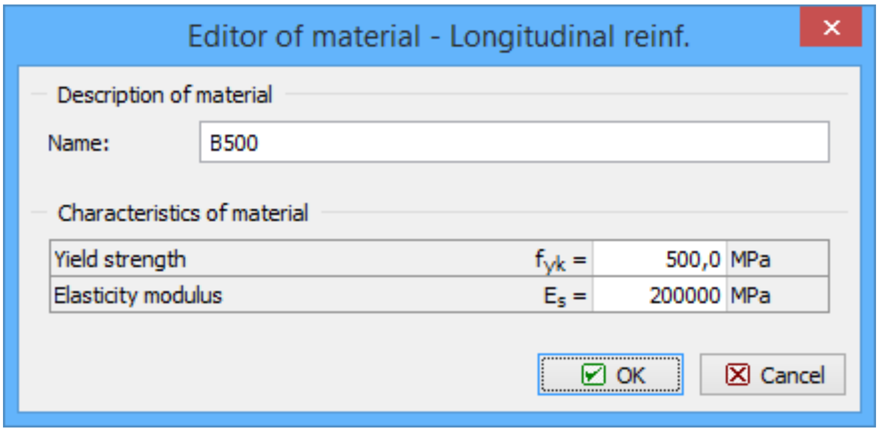



*Window "Materials catalogue"*

# Material editor - reinforcement

The arbitrary material characteristics can be specified in this window. The characteristics are described in the chapter "**Material characteristics**" of the theoretical part of the help.




*Window "Material editor"*

# Reinforcement cover

This window contains inputs for the calculation of minimum cover in accordance with EN 1992-1-1, chapter *4.4.1*.

## Environment

The window shows exposure class specified in the window "**Materials**", as this input has an impact on the calculation of minimum cover. The exposure class can be change in the window "**Environment**" that can be launched by the button "**Edit**".

## Structure class

The structure class is used for the determination of the value $c_{min,dur}$ (minimum cover due to environmental conditions). Recommended class for the structures with design working life *50* years is *S4*. This class may be increased or reduced according to the rules given in the table *4.3N*. The class shall be increased for design working life *80* or *100* years, reduction is possible for slabs or for members with special quality control of th production. The determination of the value $c_{min,dur}$ is done in accordance with table *4.4N* of EN 1992-1-1.

## Other influences

This part contains parameters that affects minimal cover $c_{min}$ and allowance in design for deviation $\Delta c_{dev}$.

The minimum cover of elements exposed to abrasion should be increased by sacrificial layer in accordance with *4.4.1.2.(13)*. The "**Abrasion class**" has to be specified in this case. Available options are:

**No abrasion**
- The surface isn't exposed to abrasion, the minimum cover isn't increased

**XM1 - moderate abrasion**
- The surfaces with moderate abrasion (e.g. members of industrial sites frequented by vehicles with air tyres) should be increased by $k_1$. Recommended value is *5mm*.

**XM2 - heavy abrasion**
- The surfaces with heavy abrasion (e.g. members of industrial sites frequented by fork lifts with air or solid rubber tyres) should be increased by $k_2$. Recommended value is *10mm*.

**XM3 - extreme abrasion**
- The surfaces with extreme abrasion (e.g. members industrial sites frequented by fork lifts with elastomer or steel tyres or track vehicles) should be increased by $k_3$. Recommended value is *15mm*.

The window contains also following settings:

**Max accregate diameter is greater than 32mm**
- The minimum cover due to bond requirement $c_{min,b}$ should be increased by *5mm* if the nominal maximum aggregate size is greater than *32mm* in accordance with *4.4.1.2.(3)* of EN 1992-1-1.

**Uneven surface**
- The minimum cover should be increased by at least *5mm* for uneven surfaces (e.g. exposed aggregate) in accordance with *4.4.1.2.(11)* of EN 1992-1-1.

**The additive safety element**
- The concrete cover should be increased by the additive safety element $\Delta c_{dur,\gamma}$ in accordance with *4.4.1.2.(6)*. Recommended value is *0mm*.

**Stainless steel**
- The minimum cover may be reduced by $\Delta c_{dur,st}$ according to *4.4.1.2.(7) in cases* where stainless steel is used or where other special measures have been. Recommended value of $\Delta c_{dur,st}$ is *0mm*.

**Additional protection**
- The minimum cover may be reduced by $\Delta c_{dur,add}$ for concrete with additional protection (e.g. coating) in accordance with *4.4.1.2.(8)* of EN 1992-1-1. Recommended value is *0mm*.

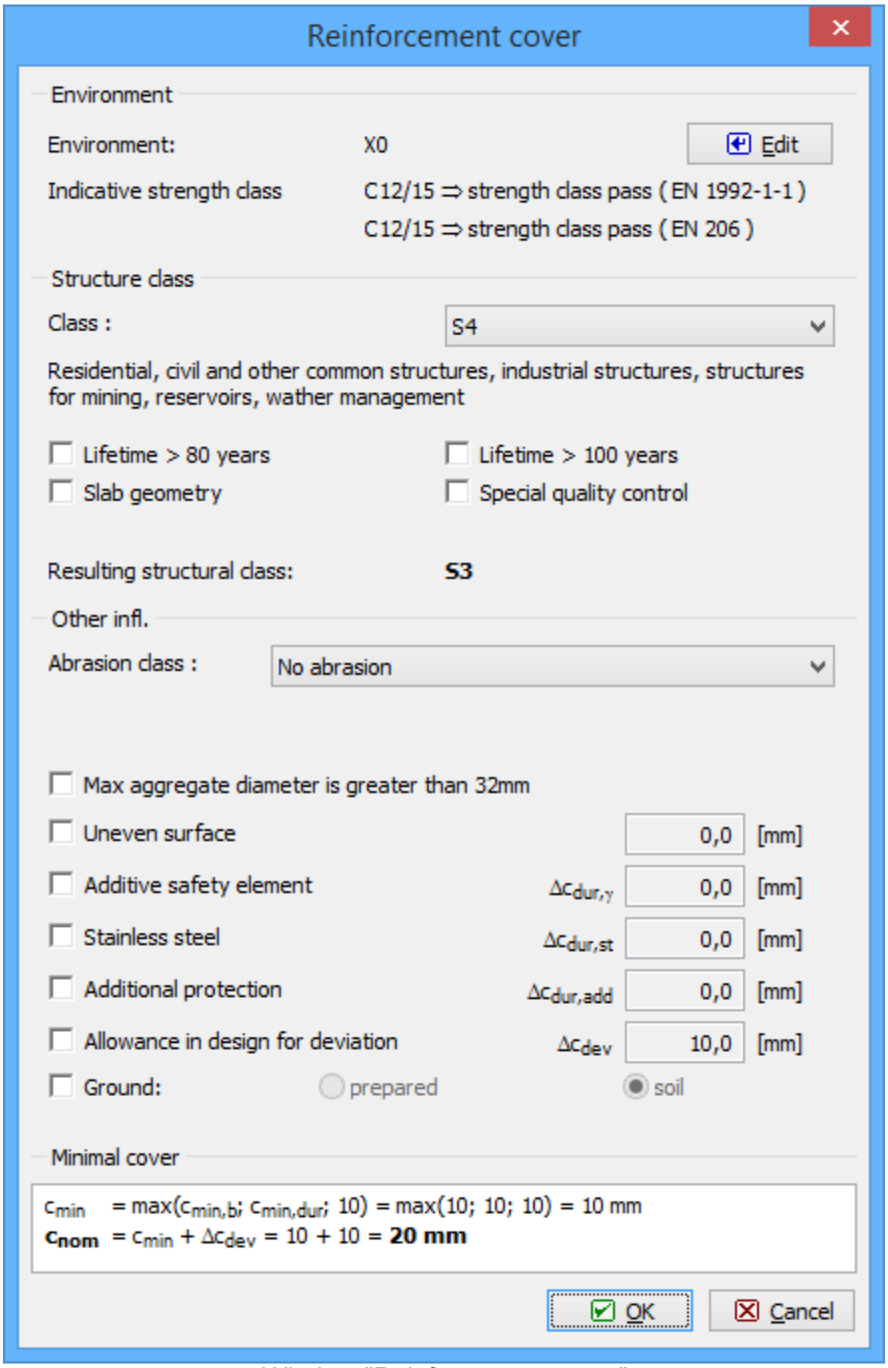
**Allowance in design for deviation**
- The minimum cover shall be increased in design to allow for the deviation $\Delta c_{dev}$ in accordance with *4.4.1.3(1)*. Recommended value is *10mm*. This value may be reduced for fabrication with a quality assurance system (includes measurements of the concrete cover). Permissible value should be within the interval *<10mm, 5mm>*. The allowance $\Delta c_{dev}$ may be reduced to *<10mm, 0mm>* for quality assurance system with very accurate measurement device and with rejection of non conforming members (e.g. precast elements).

The minimum cover should be increased for concrete cast against uneven surfaces in accordance with *4.4.1.3(4)*. These options are available:

**Concrete cast against prepared ground**
- The minimum cover should be at least $k_1$ for concrete cast against prepared ground (including blinding). The value of $k_1$ should be considered as *40mm*.

**Concrete cast against soil**
- The minimum cover should be at least $k_2$ for concrete cast against soil. The value of $k_2$ should be considered as *75mm*.

Calculation can be checked in the bottom part of the window. Procedures are described in the chapter "**Minimum cover**".




*Window "Reinforcement cover"*

# Edit reinforcement - general cross-section

The arbitrary reinforcement for general cross-section may be specified in this window. There are three basic ways of reinforcement input that may be combined: batched input with the help of the button "**Generate**", graphical input in the workspace and numerical input using the coordinates of bars. Reinforcement may be also imported from **.dxf* file.

## Automatic generation

The bars may be entered in an automatic way with the help of the window "**Edit reinforcement**" that may be launched by the button in the left bottom corner of the window. Any existing reinforcement is deleted when adding the new bars from that window.

## Numerical input

New bars may be entered numerically with the help of table "**General reinforcement**" in the left part of the window. The input is performed in the window "**Edit reinforcement**" that may be launched by the button "**Add**" in the toolbar under the table. Modifications and deletion may be done with the help of the buttons "**Edit**" a "**Remove**".

## Graphical input

The new bars may be also specified on the workspace in the right part of the window. The appropriate mode may be selected in the toolbar "**Edit**". The new bars may be added by the cursor in the mode "**Add**". The bar diameter may be specified in the input field on the right side of the toolbar. The window "**Edit reinforcement**" with bar properties (position, diameter) can be launched for existing bar after clicking on the bar in the mode "**Edit**". Mode "**Remove**" deletes the existing bars after clicking on them. The tools from toolbars "**Selections**" and "**Scale**" may be helpful during the work.

## Import of reinforcement

Position and diameter of the reinforcement may be imported from the **.dxf* file using the button "**Import**" in the left bottom corner of the window. All circles with diameters up to *50mm* are automatically recognized as reinforcement bars. The origin of the coordinate system has to be respected when preparing the input file. The dedicated window is shown during the import of **.dxf* file. User is able to switch on/off certain layers in the file and specify the input units.

## Cover

Required cover of longitudinal reinforcement can be calculated in this part. these options are available:

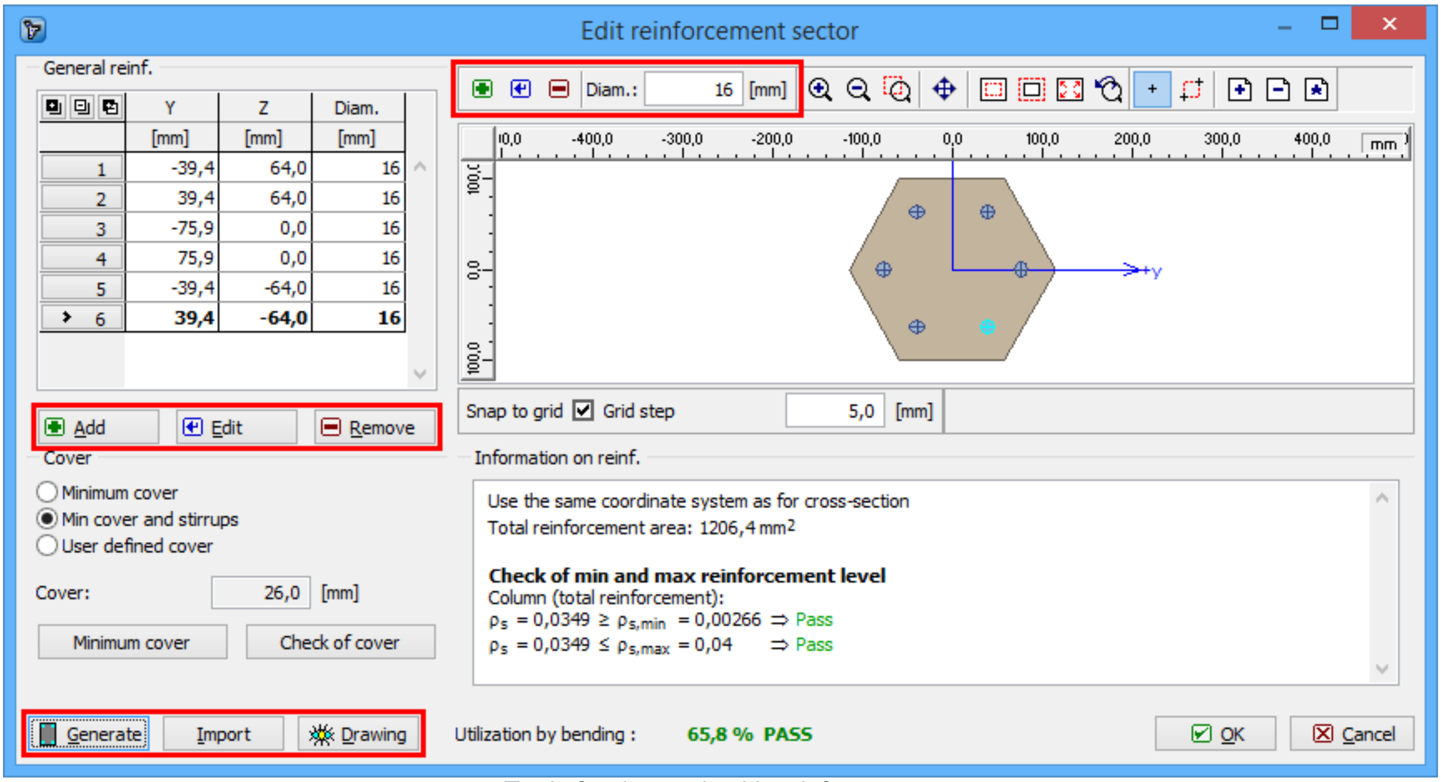
| | |
|---|---|
| **Minimum cover** | • The minimum cover calculated in the window "**Reinforcement cover**" will be used. The calculation in this window can be changed after clicking on the button "**Minimum cover**". |
| **Minimum cover and stirrups** | • The sum of stirrups' diameter (specified in part "**Shear reinforcement**") and minimum cover calculated in the window "**Reinforcement cover**" will be used. The calculation in this window can be changed after clicking on the button "**Minimum cover**". |
| **User defined cover** | • The user defined value of the reinforcement cover can be specified for this option. |

Button "**Check of cover**" runs the control of minimum cover for the bar.

## Information

Bottom part of the window shows the verification of structural rules (described in the chapter "**Structural rules**"). Bottom edge of the window shows the value "**Utilization by bending**" that shows actual result of longitudinal reinforcement verification. This value is updated automatically adfter any change in the reinforcement input.

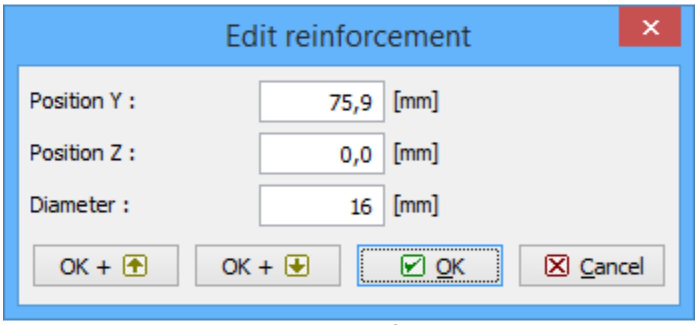
 



*Tools for the work with reinforcement*

# Edit reinforcement

The position and diameter of the reinforcement bar may be specified or modified in this window. The position is defined relatively to the origin of the coordinate system that is shown in the window "**Edit reinforcement - general cross-section**".



*Window "Edit reinforcement"*

# Edit reinforcement

This window is suitable for the input of longitudinal reinforcement in pre-defined shapes (rectangular shape, T-shape, circle etc.). The reinforcement is defined with the help of rows specified by the cover from upper (part "**Upper reinforcement**") or lower (part "**Bottom reinforcement**") edge of the cross-section. Any changes are automatically visible in the cross-section view in the right part of the window. Bars may be specified with the help of bars count ("**Number**") or bars spacing ("**Distance**"). In this case, the bars count for the member is calculated as a ratio of cross-section width in given height and the distance between bars. This input style is common mainly for slabs and walls, where the one linear meter of the structure is usually analysed and the bar spacing doesn't respect this notional dimension

Only one reinforcement row along the cross-section perimeter is available for circular cross-sections (circle, annulus).

## Cover

Required cover of longitudinal reinforcement can be calculated in this part. these options are available:

**Minimum cover** • The minimum cover calculated in the window "**Reinforcement cover**" will be used. The calculation in this window can be changed after clicking on the button "**Minimum cover**".

**Minimum cover and stirrups** • The sum of stirrups' diameter (specified in part "**Shear reinforcement**") and minimum cover calculated in the window "**Reinforcement cover**" will be used. The calculation in this window can be changed after clicking on the button "**Minimum cover**".

**User defined cover**
- The user defined value of the reinforcement cover can be specified for this option.

Button "**Check of cover**" runs the control of minimum cover for the bar.

## Information

Bottom part of the window shows the verification of structural rules (described in the chapter "**Structural rules**").

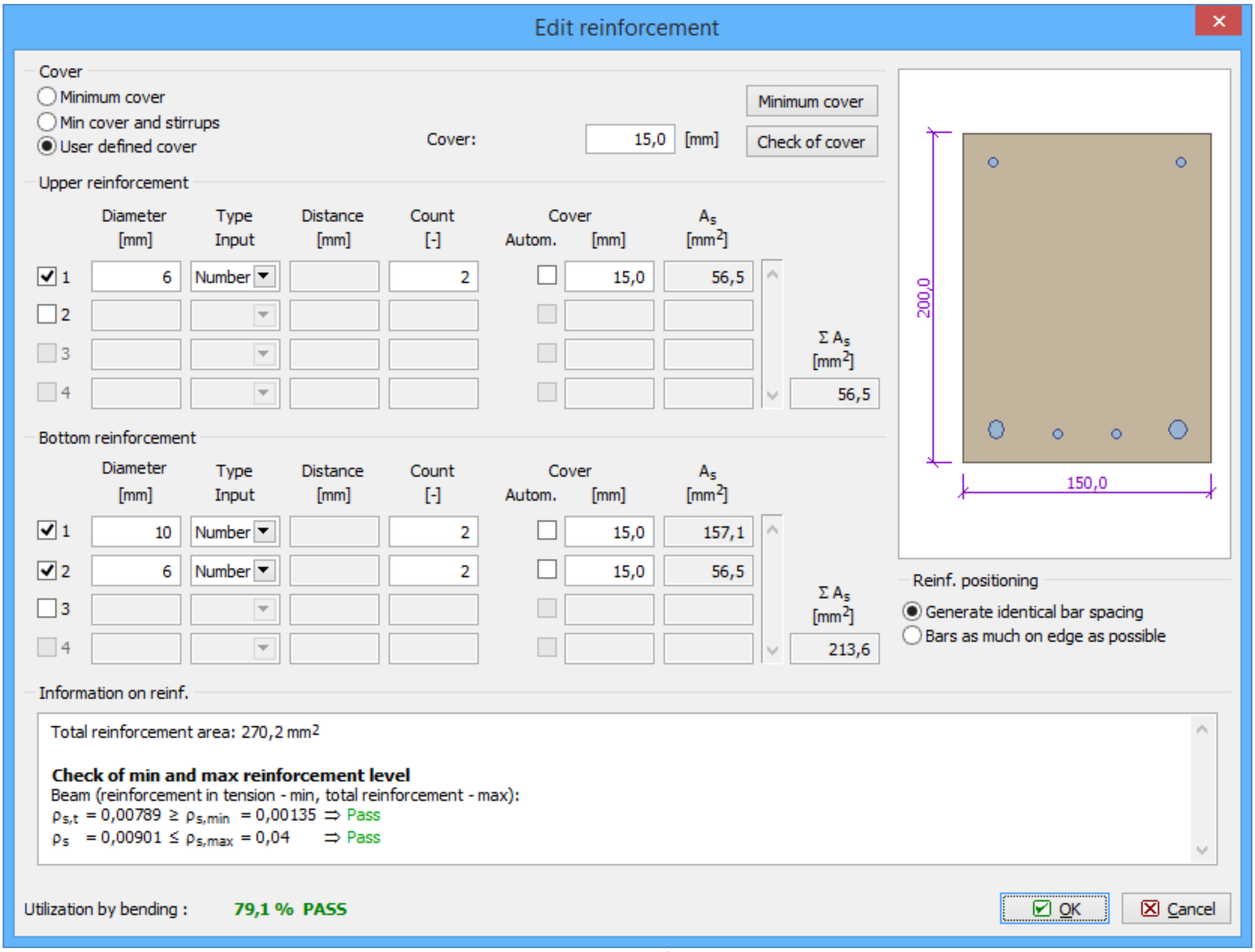
## Reinforcement positioning

The bars in rows have may follow these two options of horizontal alignment:

**Generate identical bar spacing**
- The edge bars are placed according to the value of cover. Intermediate bars are placed in that way that the bar spacing is identical between all bars.

**Bars as much on edge as possible**
- The edge bars are placed according to the value of cover. Intermediate bars are placed as much on edge as possible in that way that the bar spacing equal to the minimum value of bar spacing.

Bottom edge of the window shows the value "**Utilization by bending**" that shows actual result of longitudinal reinforcement verification. This value is updated automatically adfter any change in the reinforcement input.




*Window "Edit reinforcement"*

## Shear reinforcement

This window contains properties of transverse reinforcement and performs verification of shear forces and torsional moments. Cross-sections without specified shear reinforcement are analysed as plain concrete members. Properties of shear reinforcement are placed in the upper part of the window. Following types of reinforcement are supported:

- **Boundary stirrups** - closed stirrups along the cross-sectional perimeter that are able to resist both shear forces and torsional moment (included only in certain programs)
- **Ties, inner stirrups** - inner parts of stirrups or ties between upper and bottom longitudinal reinforcement (may be specified in two directions in certain programs)

- **Bent-up bars** - shear reinforcement with angle between the reinforcement and the member axis in the interval *<45,0;90,0>*

Characteristics of individual reinforcement types:

## Boundary stirrups

This reinforcement is considered as a basic transverse (shear) reinforcement of the member. Shape of bars respects the geometry of cross-section, distance from the edge is equal to the specified cover. The stirrups are specified by the bar diameter and spacing between bars along the member axis. Two profiles per bar are considered automatically.

Analysis of torsion can be influenced by the setting "**Torsion**". These options are available:

- **Consider only to shear resistance** - Complete bearing capacity of stirrups is considered in the analysis of shear. Stirrups aren't considered during the analysis of torsion. This option should be used for open stirrups without any torsional resistance.
- **Split to shear and torsion automatically** - Bearing capacity of stirrups is divided between analysis of shear and torsion in that way that both analysis provide the same utilization.
- **Consider to torsion resistance with ratio** - Bearing capacity of stirrups is divided between analysis of shear and torsion according to the specified ratio. Input value is the bearing capacity of stirrups for analysis of torsion.

## Ties, inner stirrups

Inner bars of stirrups and ties can be specified in this part. The reinforcement is specified by the diameter, number of bars and spacing along the member axis. Spacing and diameter can be automatically copied from boundary stirrups using the setting "**Same as boundary stirrups**". This behaviour is suitable for reinforcement made of inner parts of boundary stirrups.

## Bent-up bars

The difference between bent-up bars and inner stirrups is the angle between bar and member axis. Bent-up bars are specified by the diameter, number of bars, pitch and spacing along the member axis. Bent-up bars are considered to be placed only in one point of the member length (not in the row along the member length). For other cases, the setting "**As row of bent-up bars**" shall be switched on. Spacing of bent-up bars along the member length has to be specified in this case.

## Inner lever arm

The user defined value of inner lever arm can be specified in this part. This value is one of the fundamental inputs for shear analysis. Available is automatic calculation (described in the theoretical part of help) or manual input as a portion of effective depth of cross-section $d$. The inner lever arm may be considered as *0,9d* according to *6.2.3(1)* of EN 1992-1-1 (provided that the member isn't loaded by normal force).

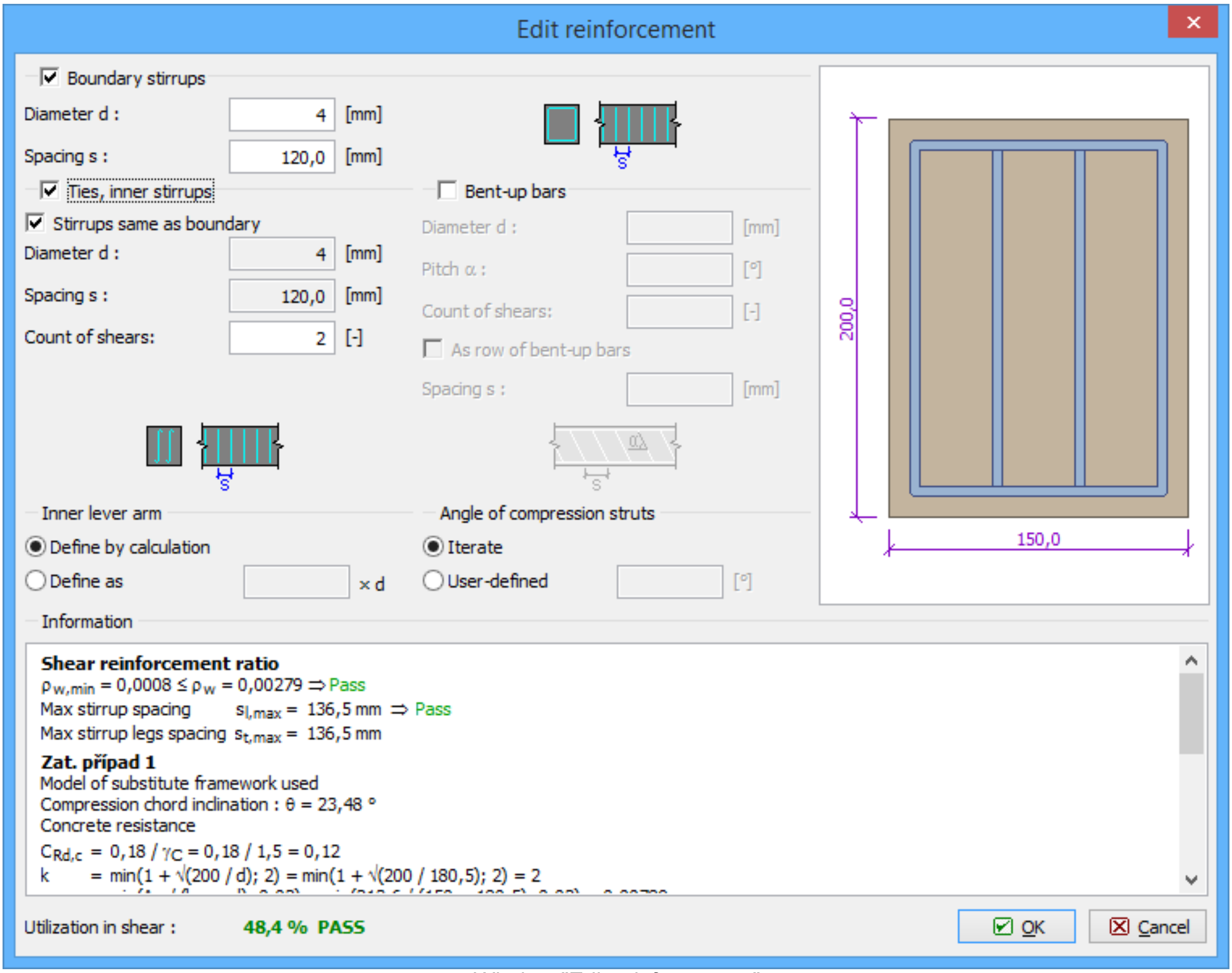
## Angle of compression struts

Angle of compression struts can be calculated automatically by the software or specified manually by the user. Automatic iteration is based on finding the pitch for which the maximum shear $V_{Rd,max}$ is equal to the bearing capacity of the shear reinforcement $V_{Rd,s}$. Permissible interval for the pitch is *<21,8°;45°>* according to *6.2.3(2)* of EN 1992-1-1.

## Information

Bottom part of the window shows both the results of the analysis (described in the chapter "**Ultimate limit state - shear**") and verification of structural rules (described in the chapter "**Structural rules**").



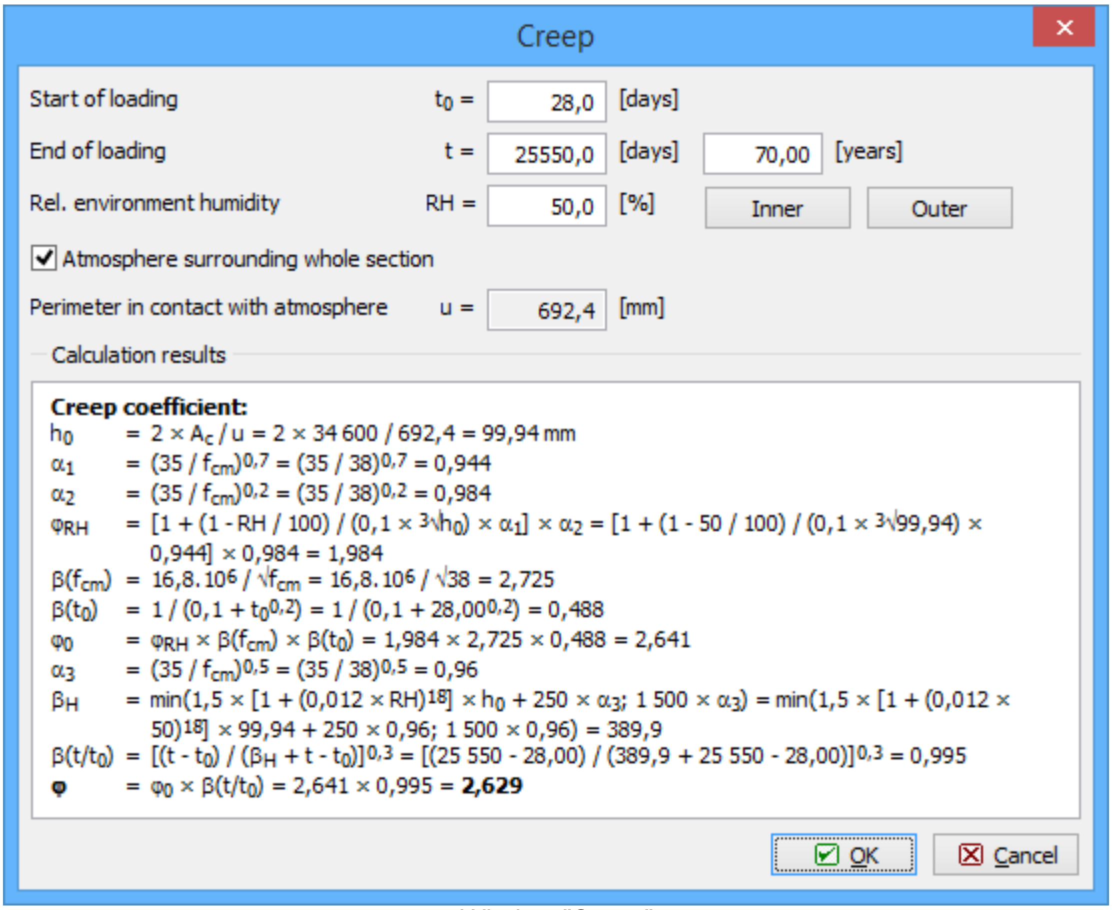

*Window "Edit reinforcement"*

## Creep

This window contains parameters that are necessary for the calculation of the creep factor in accordance with *3.1.4* of EN 1992-1-1. These parameters should be specified:

| | |
|---|---|
| **Start of loading** | • The age of concrete at loading |
| **End of loading** | • The age of concrete at the needed moment (working life). The value may be specified in days or years. |
| **Relative environment humidity** | • The relative humidity of the ambient environment. The humidity *50%* (inside conditions) or *80%* (outside conditions) can be selected with the help of buttons "**Inner**" and "**Outer**". |
| **Atmosphere surrounding whole section** | • This setting refer to the perimeter of the cross-section that is in contact with atmosphere. Whole perimeter is considered as a default, it's possible to specify only part of the perimeter with the help of this setting. |

Calculations connected with creep are described in the part "**Creep factor**" of theoretical help.




*Window "Creep"*

## Buckling

The upper part of the window contains buckling parameters (consideration in the analysis, fundamental lengths $L_x$ and supporting style) for directions *Y* and *Z* (in analysis type "**Concrete 2D**" only direction *Y*). The pinned supporting style is considered as a default, the buckling length is equal to the fundamental length in this case. The different supporting style for directions *Y* and *Z* may be selected in the window "**Buckling length**" that is available after clicking on the button with a calculator.

### Creep coefficient

The buckling analysis is affected by the creep coefficient that may be specified in the window "**Creep**". This window may be opened by the button with a calculator.

### Method based on

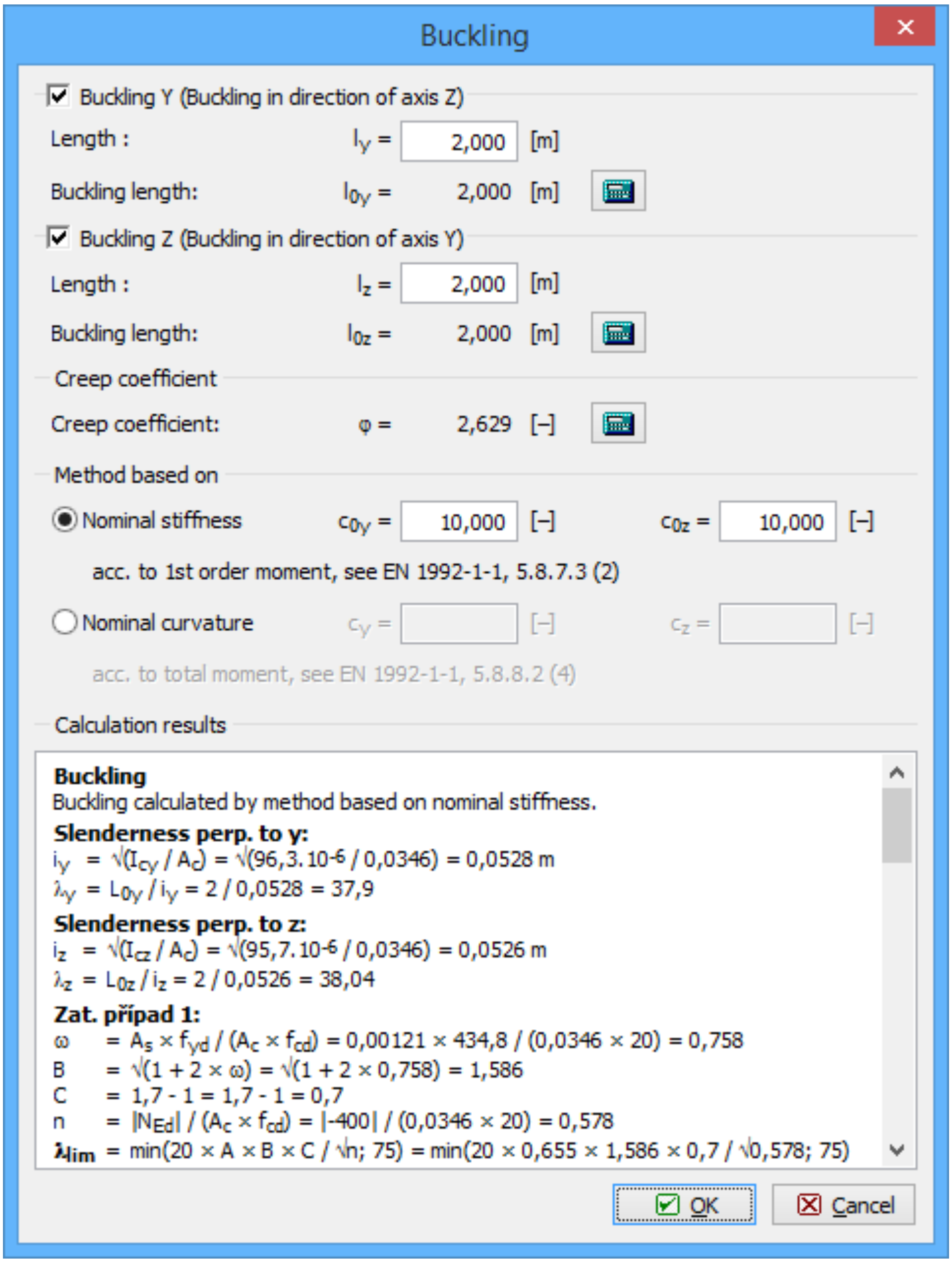
The main parameter of the buckling analysis is the method of verification. Following options are available:

**Method based on the nominal stiffness**
- This method is suitable both for partial members and whole structures. It is based on the nominal stiffness that takes into account the effects of cracking, material non-linearity and creep on the overall behaviour. The coefficient that depends on the distribution of first order moment $c_0$ has to be specified for each direction according to the chapter *5.8.7.3(2)*. Recommended values are described in the chapter "**Buckling**" of the theoretical help.

**Method based on the nominal curvature**
- This method is primarily suitable for isolated members with constant normal force. The nominal second order moment is calculated according to a deflection that is based on the effective length and an estimated maximum curvature. The method is based on the chapter *5.8.8*. The coefficient that depends on the curvature distribution *c* has to be specified for each direction according to the chapter *5.8.8.2(4)*. Recommended values are described in the chapter "**Buckling**" of the theoretical help. Not available for the members made of plain concrete.

**Simplified method based on *12.6.5.2* of the standard**
- Simplified method according to the chapter *12.6.5.2* of EN 1992-1-1. Only for members made of plain concrete.

The calculation details are displayed in the bottom part of the window.




*Window "Buckling"*

# Loads

This window contains options and settings for load input.

## Load

The load name and combination type can be specified in this part. The combination type influences the analysis method. These types are available:

**Basic design (ULS)**
- The fundamental load type, assumption is that the internal forces are based on basic (or alternative) load combination in accordance with *6.4.3.2* of EN 1990. These loads are analysed for ultimate limit states. Internal forces in these loads are reduced by reduction factor $\eta_{fi}$ in the program "**Concrete Fire**" (described below).

**Accidental design (ULS)**
- The assumption for this type is that the internal forces are based on accidental load combination in accordance with *6.4.3.3* of EN 1990. These loads are analysed for ultimate limit states using partial factors for accidental situations.

**Characteristic (SLS)**
- The assumption for this type is that the internal forces are based on characteristic load combination in accordance with *6.5.3a* of EN 1990. These loads are analysed for serviceability limit states - stress control. Not available in the software "**Concrete Fire**".

**Quasi-permanent (SLS)**
- The assumption for this type is that the internal forces are based on quasi-permanent load combination in accordance with *6.5.3c* of EN 1990. These loads are analysed for serviceability limit states - crack control. Not available in the software "**Concrete Fire**".

**Frequent (SLS)**
- The assumption for this type is that the internal forces are based on frequent load combination in accordance with *6.5.3b* of EN 1990. These loads may be analysed for serviceability limit states - deflection control. Available only in the software "**Concrete Beam**".

If the check box "**Forces calculated acc. to 2nd order theory**", the analysis for this load will be performed without any consideration of buckling (the forces are already calculated on deformed structure.

## Force on cross-section

This part contains input fields for internal forces. Following forces are supported:

- **N** – Normal force
- **$M_y$** – Bending moment about axis *y* (positive values represent tension in the bottom edge of the cross-section)
- **$M_z$** – Bending moment about axis *y* (positive values represent tension in the left edge of the cross-section)
- **$V_z$** – Shear force in vertical direction (parallel with axis *z*)
- **$V_y$** – Shear force in horizontal direction (parallel with axis *y*)
- **T** – Torsional moment about member axis *1*

The input range depends on program and task type.

## Reduction coefficient for design load *(only program "Concrete Fire")*

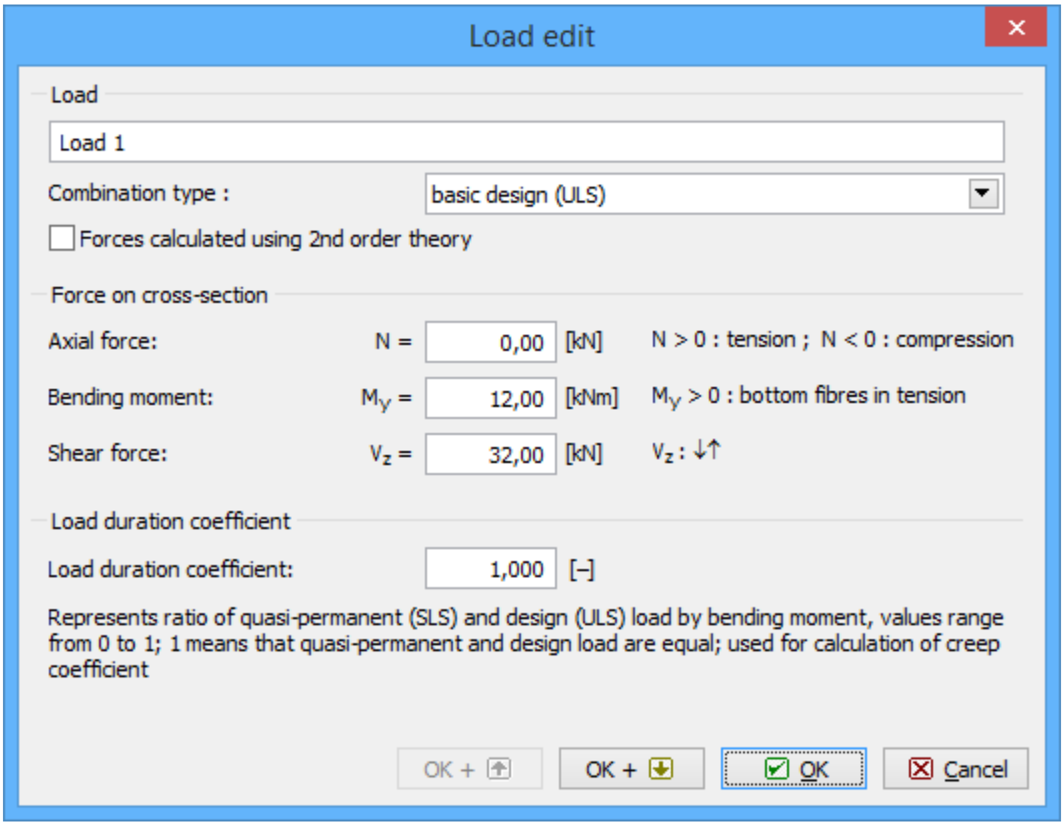
Reduction coefficient for the design load $\eta_{fi}$ recalculates the load determined for fundamental design combination into the design values for the fire situation. This coefficient should be obtained in accordance with *2.4.2* of EN 1992-1-2. As a simplification, value of $\eta_{fi} = 0.7$ may be used.

## Load duration coefficient

This coefficient represents the portion of quasi-permanent component in the design load. Value *0* means that the load doesn't contain any quasi-permanent part, value *1.0* means that complete load is quasi-permanent. This coefficient is important for the calculation of the creep factor.

## Proportion of reinforcement to concrete stiffness

This factor is available for design standard EN 1992-2, load types "**Quasi-permanent**" and "**Characteristic**". The value represents the ratio of stiffness of reinforcement and concrete. For standard conditions, this ratio is calculated with the help of moduli of elasticity. This solution doesn't respect the degradation of modulus of elasticity of concrete due to creep and similar effects.

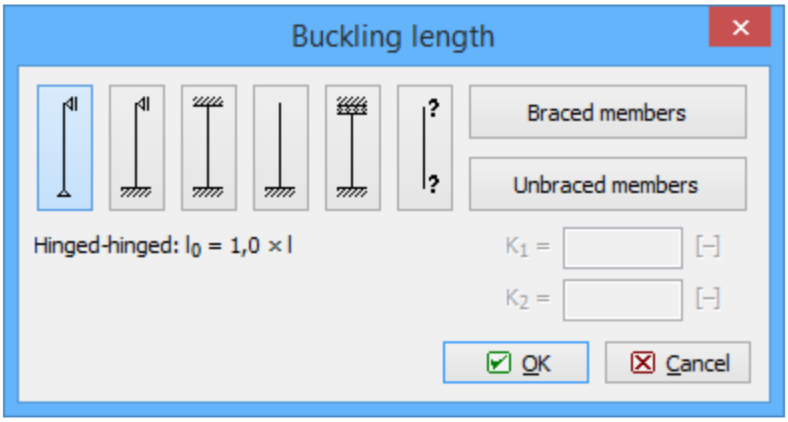



*Window "Load edit"*

# Buckling length

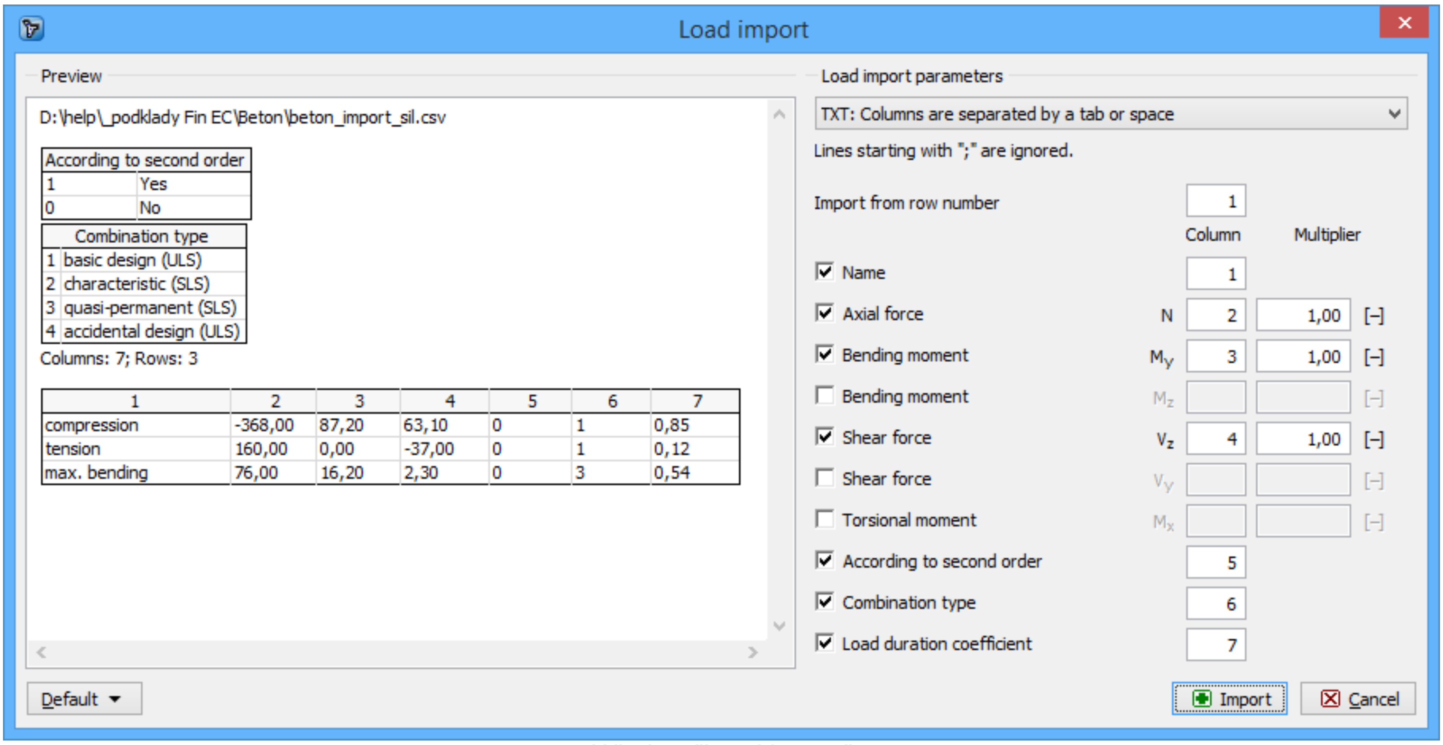
The factor for calculation of the effective member length for buckling analysis can be specified in this window. The factor is selected according to the support style of the member. Range of supporting styles and factor values are based on general theories for stability analysis. Arbitrary value of the factor can be specified for the sixth option (figure with question marks). Options "**Braced members**" and "**Unbraced members**" refer to the frame structure according to the chapter *5.8.3.2(3)* of EN 1992-1-1.




*Window "Buckling length"*

# Load import

Window "**Load import**" appears after loading the text or **.csv* source file. The data included in the source file can be arranged in this window. Left part shows content of imported file. The assignment of the certain column to the load entry (force or property) can be done in the right part of the window. Numerical entries can be also multiplied by specified multiplier. This multiplier can be used mainly for conversion caused by different units in the source file and in the program. Initial rows can be skipped using setting "**Import from row number**". This setting is helpful for source files with headings. Default units required by the software are *[kN]* and *[kNm]*.




*Window "Load import"*

## Preparing text file

Text file can be created in any text editor (e.g. *Notepad*, *Word*, *Writer*). File format requires that every row contains one load case. Every row can contain all values of internal forces separated by space or tabulator. The order may differ comparing to the order of forces in the software, however, order has to be identical for all rows of the document. The load name, load duration and second order effect may be also specified for every load. The combination type, load duration coefficient and second order effect has to be specified using this numerical codes. This scheme is used for consideration of analysis in accordance with II. order theory:

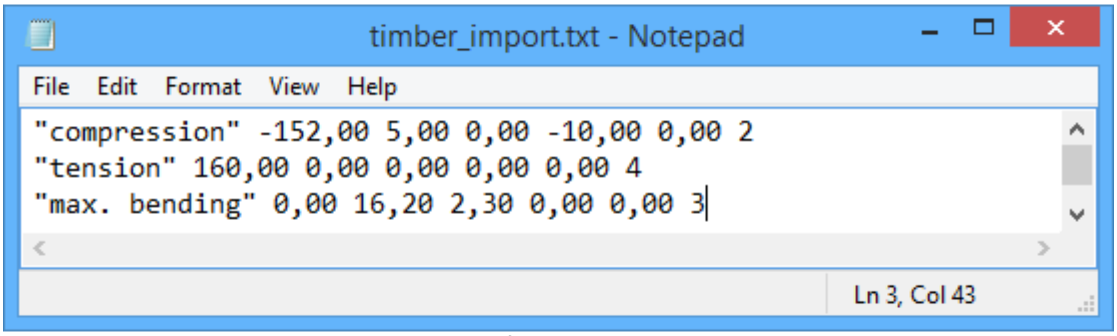
| | |
|---|---|
| 1 | Forces calculated in accordance with II. order theory |
| 0 | Forces aren't calculated in accordance with II. order theory |

The scheme for combination type:

| | |
|---|---|
| 1 | Basic design (ULS) |
| 2 | Characteristic (SLS) |
| 3 | Quasi-permanent (SLS) |
| 4 | Accidental design (ULS) |

Also load duration coefficient may be imported from the file. This coefficient represents the portion of quasi-permanent component in the design load. Value *0* means that the load doesn't contain any quasi-permanent part, value *1.0* means that complete load is quasi-permanent. This coefficient is important for the calculation of the creep factor.

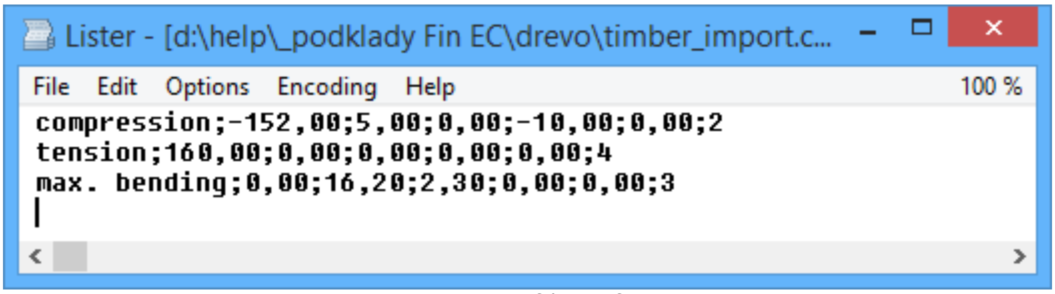
The file can be also created using part of analysis documentation from arbitrary structural analysis software.




*Text file in Notepad*

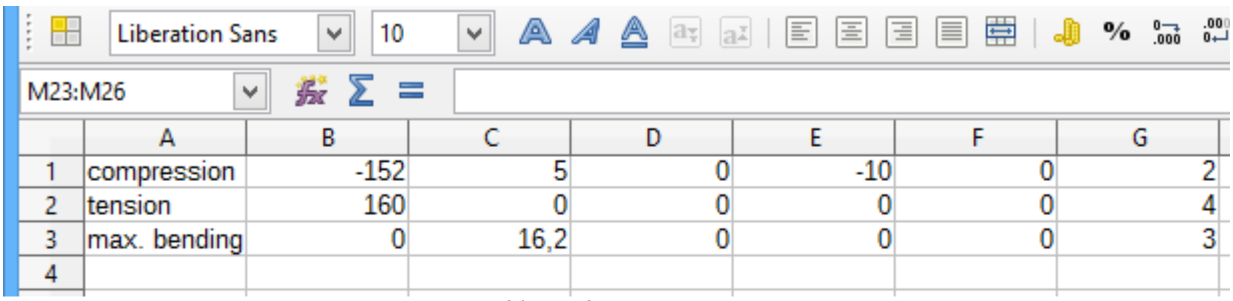
## Preparing **.csv* file

The rules for **.csv* (comma-separated values) files are almost identical . Main difference is, that the particular values are separated by semicolon ";".




*Example of *.csv file*

This file type can be easily created using spreadsheet programs like *Excel* or *Calc*. Created document can be saved as **.csv* file with appropriate separator.




*Edit of *.csv file in spreadsheet*

# Cross-section

This part contains tools for the input of cross-section geometry. Following options are available:

**Basic** • Selection of the cross-section geometry from pre-defined database. The button launches the window "**Cross-section editor**".

**Polygon** • Input of arbitrary geometry of the cross-section using general polygon. The button launches the window "**General cross-section**". Not available for analysis type "**Concrete 2D**".

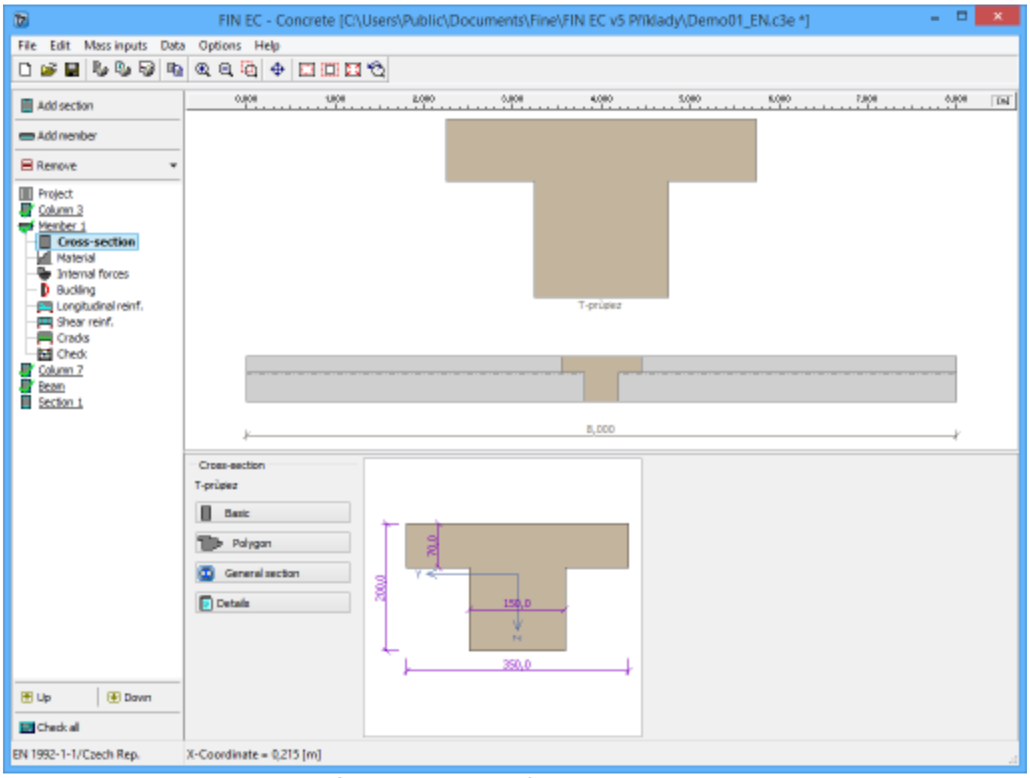
 

**General section** • Input of general cross-section in the program "**Section**". This option contains additionally the ability to specify the cross-section with holes. Not available for analysis type "**Concrete 2D**".

**Details** • Shows the cross-sectional properties (area, moment of inertia etc.) for entered cross-section.

The corresponding window may be opened also using double-click in the active workspace.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the material will be automatically copied from this program.




*Part "Cross-section" of the member design*

# Material

The materials of member and reinforcement can be specified in this part. There is a dedicated window "**Materials**" for materials input. This window can be launched using "**Material**" button. Both pre-defined strength grades and user defined input of material characteristics are available in this window.

## Plain concrete

Input of material for task type "**Plain concrete**" contains these options:

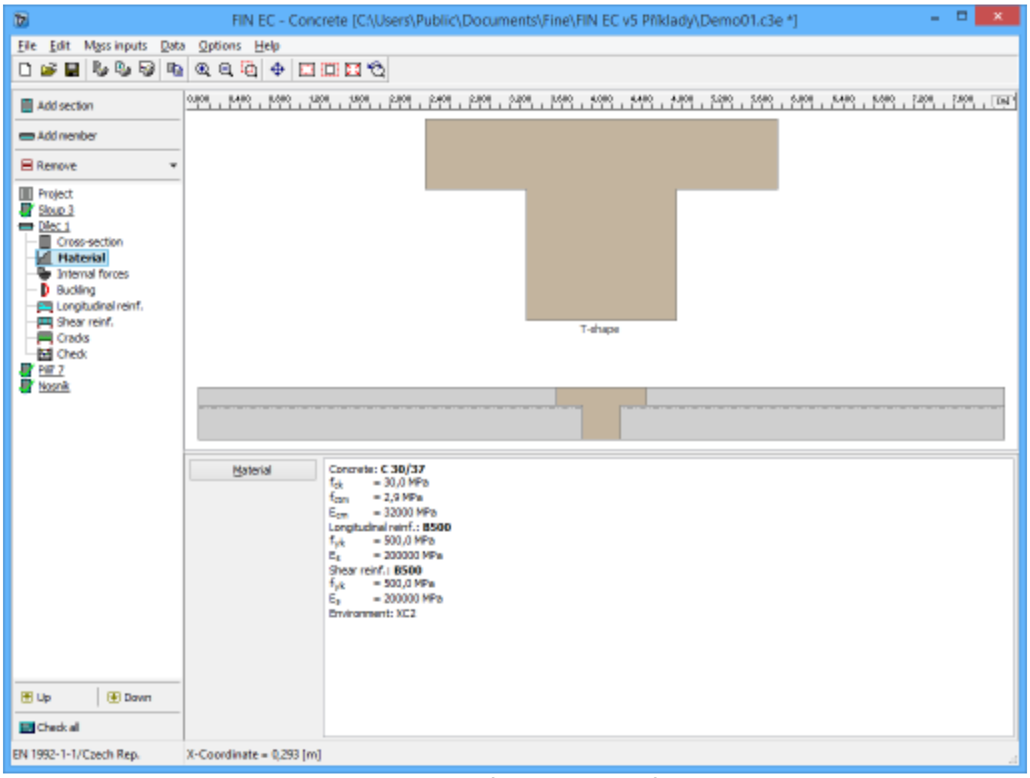
**Material** • Pre-defined database of strength classes in the window "**Materials catalogue - concrete**"

**User defined** • Input of user defined characteristics in the window "**Material editor - concrete**"

**Include concrete in tension** • Option to calculate the capacity of the cross-section including that part of cross-section where the tensile stress is lower than the tensile strength of concrete. Otherwise, only sompressive part of cross-section is considered during the analysis.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the material of concrete will be automatically copied from this program. Reinforcement materials have to be specified in this part.



*Part "Material" of member verification*

## Internal forces

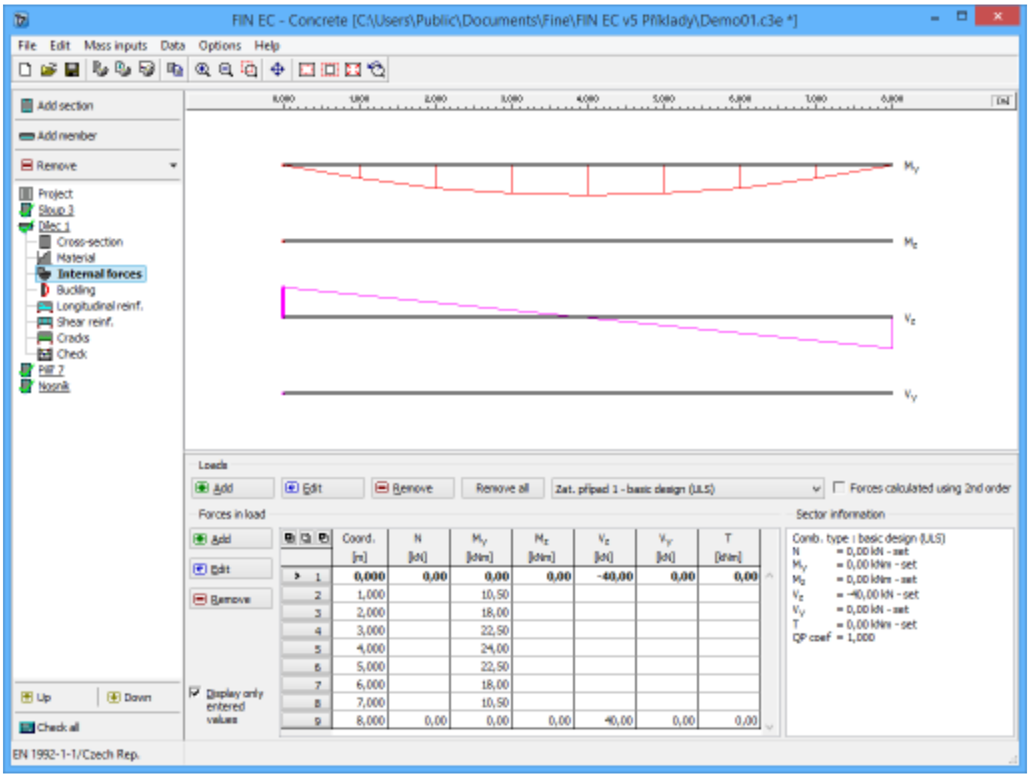
The internal forces along the member length can be specified in this part. More loads (load combinations) can be specified for any member.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the internal forces will be automatically copied from this program.

### Loads

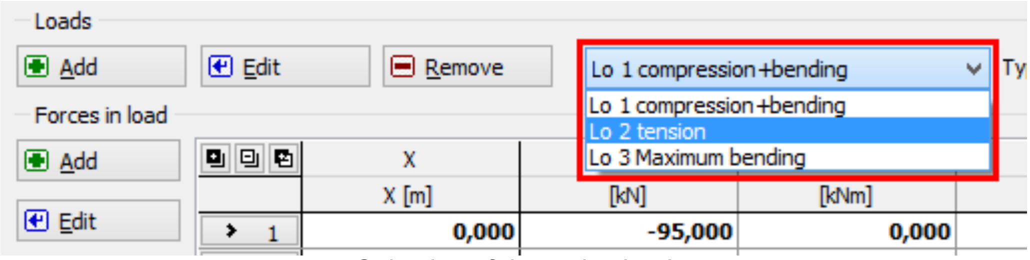
The upper part of the input frame contains buttons for input and edit of loads. The load is a set of internal forces (design values), that corresponds to the results of design combinations. The basic properties of the load can be specified in the window "**Loads**".

Any load contains a setting "**Forces calculated acc. to 2nd order**". The load won't be verified including buckling consideration, if this setting is switched on for the certain load.



## Input of internal forces

The active load has to be selected before starting the input of internal forces. The active load can be selected using list box above the table with values of internal forces.



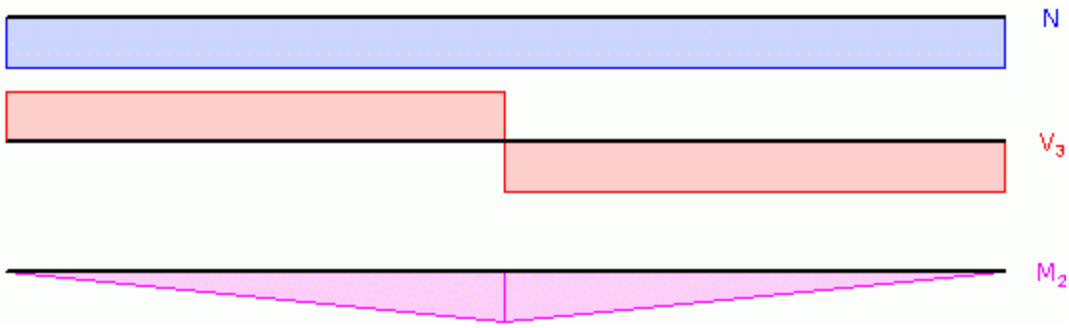
*Selection of the active load*

The internal forces are entered with the help of values in the certain points along the member length. These points should be specified mainly in the positions of local minima or maxima or in the inflection points. Intermediate values are determined automatically using linear interpolation. These points are organized in the table in the bottom part of the application window and can be entered with the help of dedicated window.

The table shows both specified and calculated values of internal forces for each point. The automatically calculated values can be hidden using setting "**Show only entered values**".

## Example of entered internal forces

The following figure shows diagram of normal force $N$, shear force $V_3$ and bending moment $M_2$. The table shows the entered internal forces, that has to be entered in the certain positions.



*Diagrams of internal forces*

These diagrams can be created using these values:

| Section number | Position *x* [m] | N [kN] | $V_3$ [kN] | $M_2$ [kN] |
|---|---|---|---|---|
| 1 | 0,00 | 20,00 | -10,00 | 0,00 |
| 2 | 2,50 | | -10,00 (left) | 25,00 |
| 3 | 2,50 | | 10,00 (right) | |
| 4 | 5,00 | 20,00 | 10,00 | 0,00 |

Intermediate values are calculated automatically using linear interpolation.

# Force edit

The internal forces in the certain member point can be entered with the help of this window. These properties can be specified:

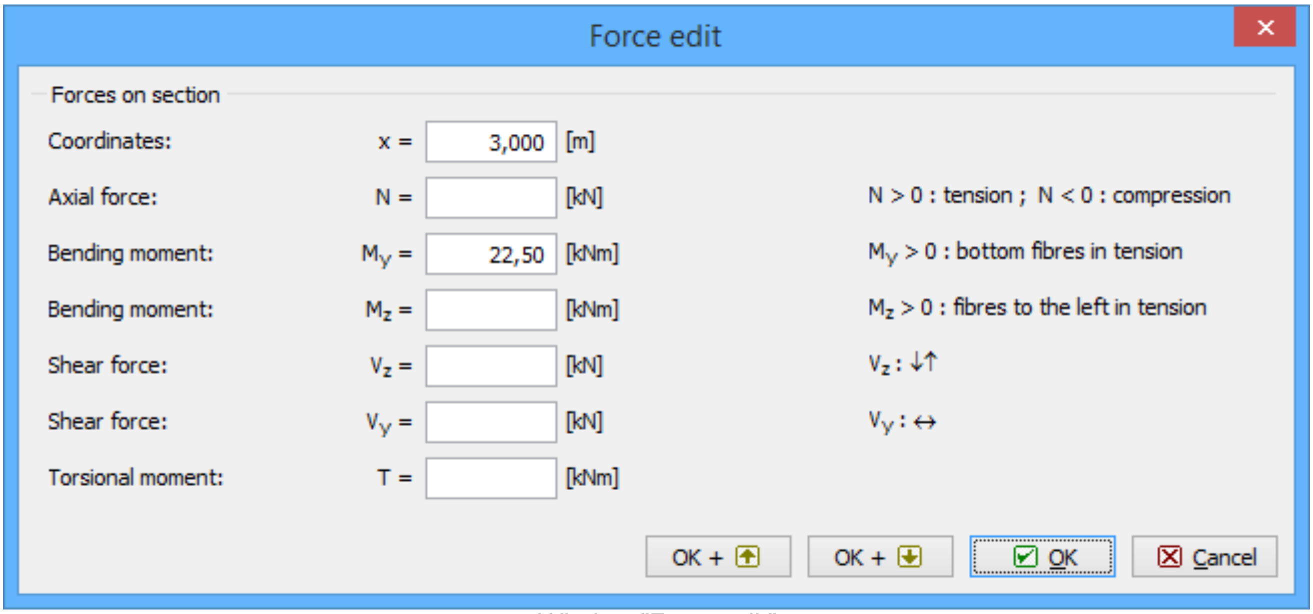
| | |
|---|---|
| **x** | • Basic input, that specifies the position of the point along the member length. The position is measured from the left end of the member |
| **N** | • Normal force |
| **$M_y$** | • Bending moment about axis $y$ (positive values represent tension in the bottom edge of the cross-section) |
| **$M_z$** | • Bending moment about axis $z$ (positive values represent tension in the left edge of the cross-section) - only selected programs and task types |
| **$V_z$** | • Shear force in vertical direction (parallel with axis $z$) |

**$V_y$** • Shear force in horizontal direction (parallel with axis *y*) - only selected programs and task types

**T** • Torsional moment about member axis *1* - only selected programs and task types

Values of all internal forces shall be specified only in the first (*x=0*) and last (*x* is equal to member length) sections. Intermediate sections may contain unfilled certain internal forces. The values of these forces are obtained with the help of linear interpolation using the closest values that were specified in other sections.




*Window "Force edit"*

## Buckling

This part contains parameters of buckling and imperfection. The setting "**Use imperfection**" adds an and effect of imperfection *$l_0$/400* according to *5.2(9)* into the analysis. Imperfection is calculated using the basic member length $l_0$ that may be changed in the table in the bottom part of the window. The fundamental length $l_0$ is the real length, not the buckling length.

The buckling analysis may be switched off for certain applications with the help of setting "**Calculate with buckling**".

The buckling analysis is affected by the creep coefficient that may be specified in the window "**Creep**". This window may be opened by the button "".

The main parameter of the buckling analysis is the method of verification. Following options are available:

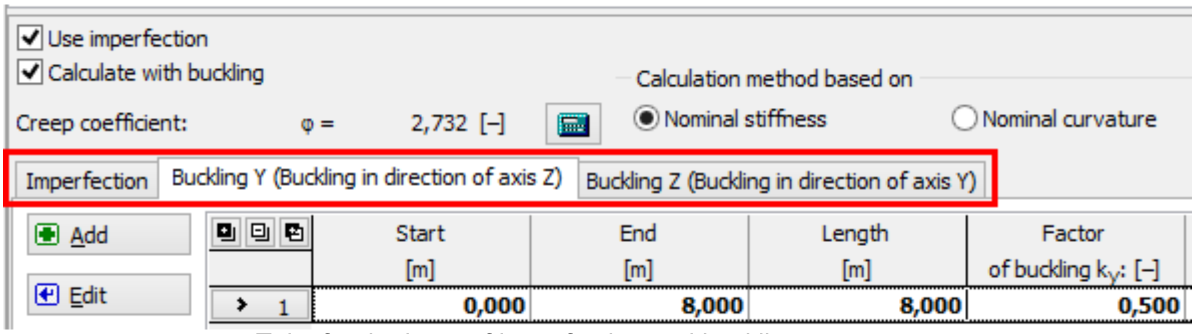
**Method based on the nominal stiffness** • This method is suitable both for partial members and whole structures. It is based on the nominal stiffness that takes into account the effects of cracking, material non-linearity and creep on the overall behaviour. The coefficient that depends on the distribution of first order moment $c_0$ has to be specified for each direction according to the chapter *5.8.7.3(2)*. Recommended values are described in the chapter "**Buckling**" of the theoretical help.

**Method based on the nominal curvature** • This method is primarily suitable for isolated members with constant normal force. The nominal second order moment is calculated according to a deflection that is based on the effective length and an estimated maximumcurvature. The method is based on the chapter *5.8.8*. The coefficient that depends on the curvature distribution *c* has to be specified for each direction according to the chapter *5.8.8.2(4)*. Recommended values are described in the chapter "**Buckling**" of the theoretical help. Not available for the members made of plain concrete.

**Simplified method based on 12.6.5.2 of the standard** • Simplified method according to the chapter *12.6.5.2* of EN 1992-1-1. Only for members made of plain concrete.

The imperfection and bucking parameters can be specified in tables in the bottom part of the input frame. The tables for imperfection and buckling in two directions are organized into tabs.

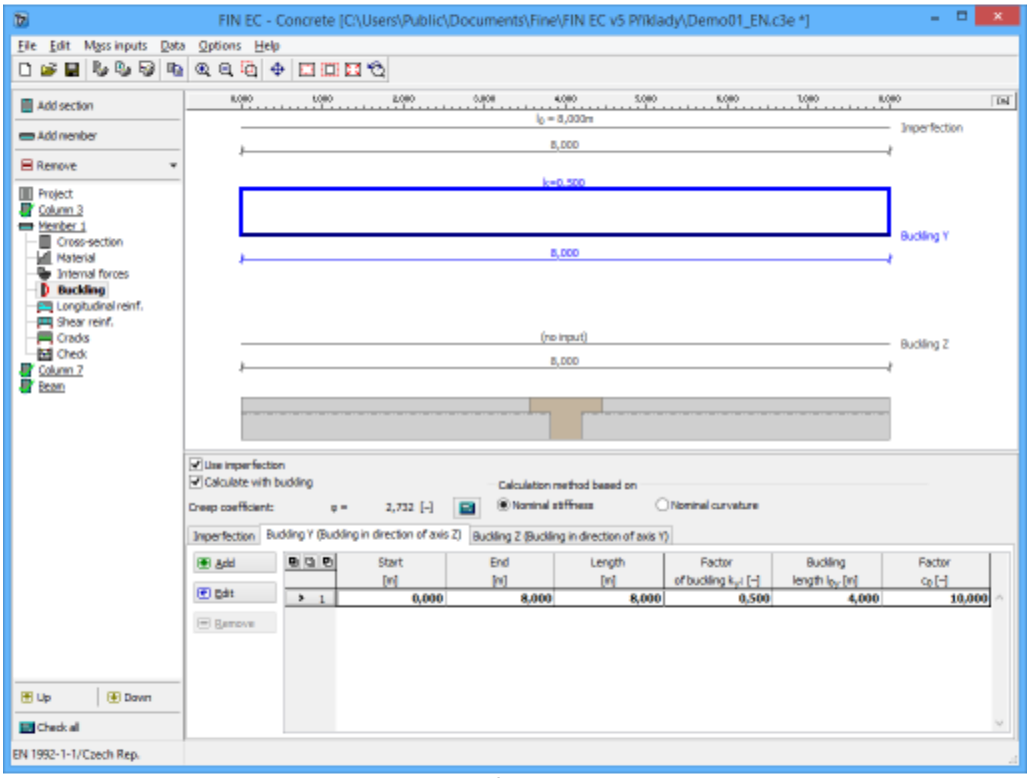
 




*Tabs for the input of imperfection and buckling parameters*

The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of the sector are organized in the windows "**Imperfection**" and "**Buckling Y/Z**". More sectors can be added (button "**Add**") for the input of different imperfection/buckling properties along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. The sector lengths may be edit by using the active dimensions, double-click on certain sector launches the appropriate window for sector edit.



*Part "Buckling" of the member design*

# Imperfection

This window contains parameters that are necessary for the determination of imperfection for given member sector. The fundamental parameter is the coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector and is used for the calculation of the sector length.

## Imperfection parameters

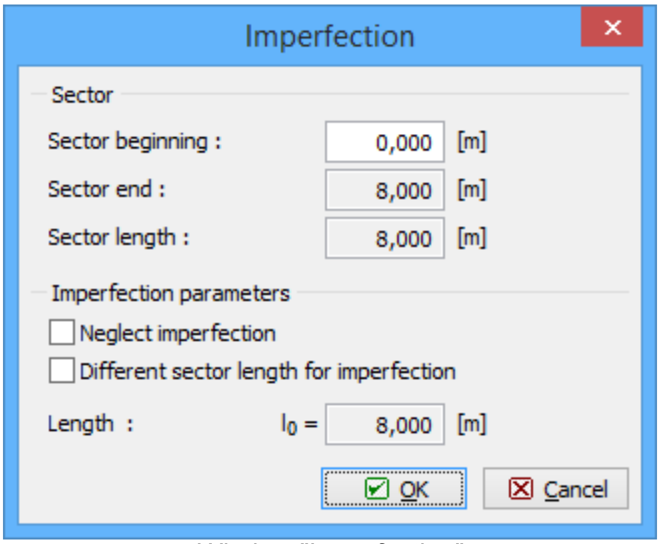
Following parameters shall be specified for the imperfection determination:

**Neglect imperfection**
- The consideration of the imperfection may be switched off with the help of this setting. Following parameters won't be enabled in that case.

**Different sector length for buckling**
- If the fundamental length for the imperfection calculation differs from the sector length *l*, this check box should be switched on and the fundamental length should be defined in the input field "**Length**". In other cases, the sector length is considered as a fundamental length for the imperfection calculation. The sector length is updated automatically after any modification (e.g. insertion of new sector).

**Length**
- The fundamental length $l_0$ that is used for the calculation of the imperfection.



*Window "Imperfection"*

# Buckling Y/Z

This window contains parameters that are necessary for the determination of buckling lengths $l_{0y}$ and $l_{0z}$ for given member sector. The fundamental parameter is the coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector and is used for the calculation of the sector length.

## Buckling parameters

Following parameters shall be specified for the buckling analysis:

**Neglect buckling**
- The buckling analysis may be switched off with the help of this setting. Following parameters won't be enabled in that case.

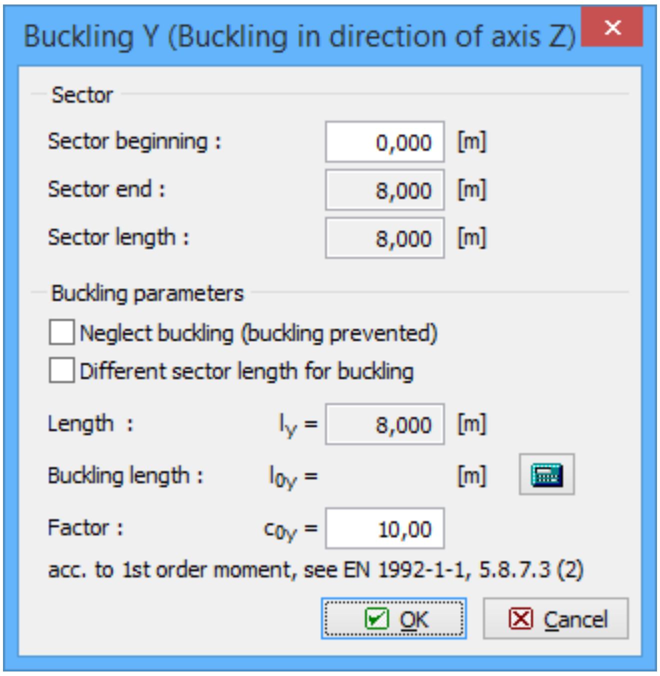
**Different sector length for buckling**
- If the fundamental length for the buckling analysis differs from the sector length $l$, this check box should be switched on and the fundamental length should be defined in the input field "**Length**". In other cases, the sector length is considered as a fundamental length for buckling analysis. The sector length is updated automatically after any modification (e.g. insertion of new sector).

**Length**
- The fundamental length $l$ that is used as an input for the calculation of the buckling length $l_0$.

**Buckling length**
- The buckling length in the corresponding direction that is calculated with the help of the fundamental length for buckling analysis and the supporting style of the sector that may be changed in the window "**Buckling length**" (launched by the button ""). The hinged supporting style is considered as a default, the buckling length is equal to the fundamental length in this case.

**Factor**
- The coefficient that depends on the distribution of first order moment $c_0$ has to be specified for the analysis method based on the nominal stiffness according to the chapter *5.8.7.3(2)*. The coefficient that depends on the curvature distribution $c$ has to be specified for the analysis method based on the nominal curvature according to the chapter *5.8.8.2(4)*. Recommended values are described in the part "**Buckling**" of the theoretical help.




*Window "Buckling Y"*

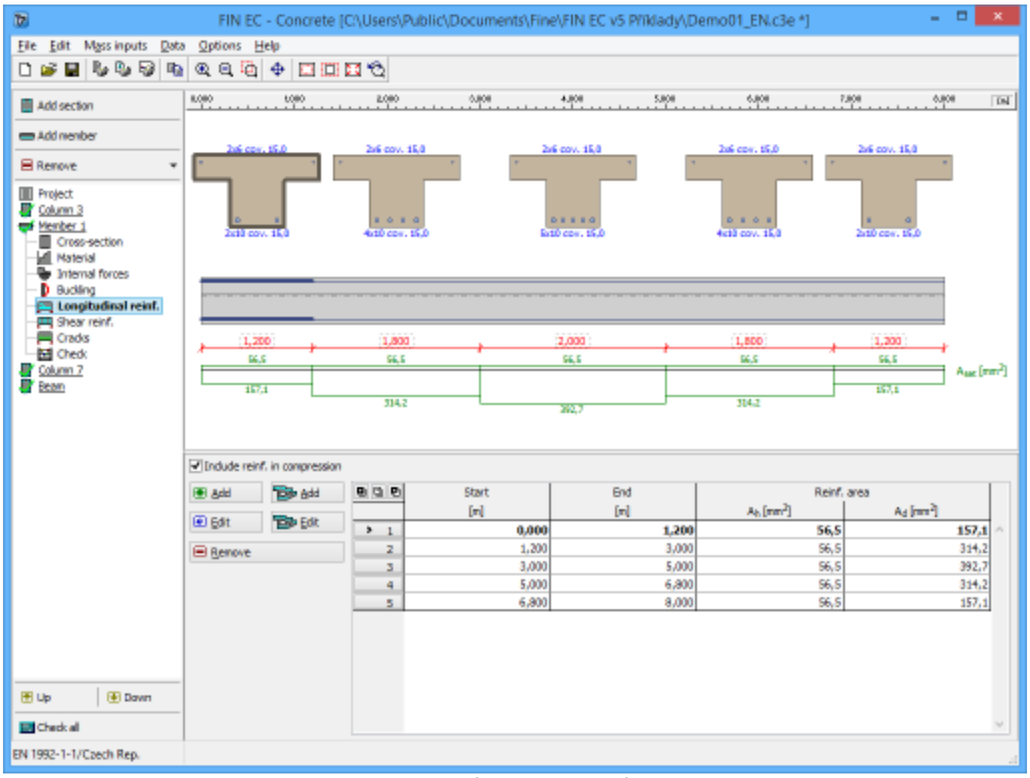
## Longitudinal reinforcement

The longitudinal reinforcement can be specified in this part of the tree menu. The reinforcement parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different reinforcement parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of the longitudinal reinforcement (type, diameter, number etc.) are organized in the window "**Edit reinforcement sector**". The longitudinal reinforcement may be also specified in a general way for check type "**3D**". The reinforcement has to be entered and modified in the window "**Longitudinal reinforcement - general cross-section**" that can be launched by the button "" in this case. More sectors can be added (button "**Add**") for input of different longitudinal reinforcement along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The longitudinal reinforcement in compression may be also included in the analysis. The consideration of this reinforcement in the analysis depends on the setting "**Include reinforcement in compression**".

The particular sectors are displayed also in the active workspace. The sector lengths may be edit by using the active dimensions, double-click on certain sector launches the appropriate window for sector edit.

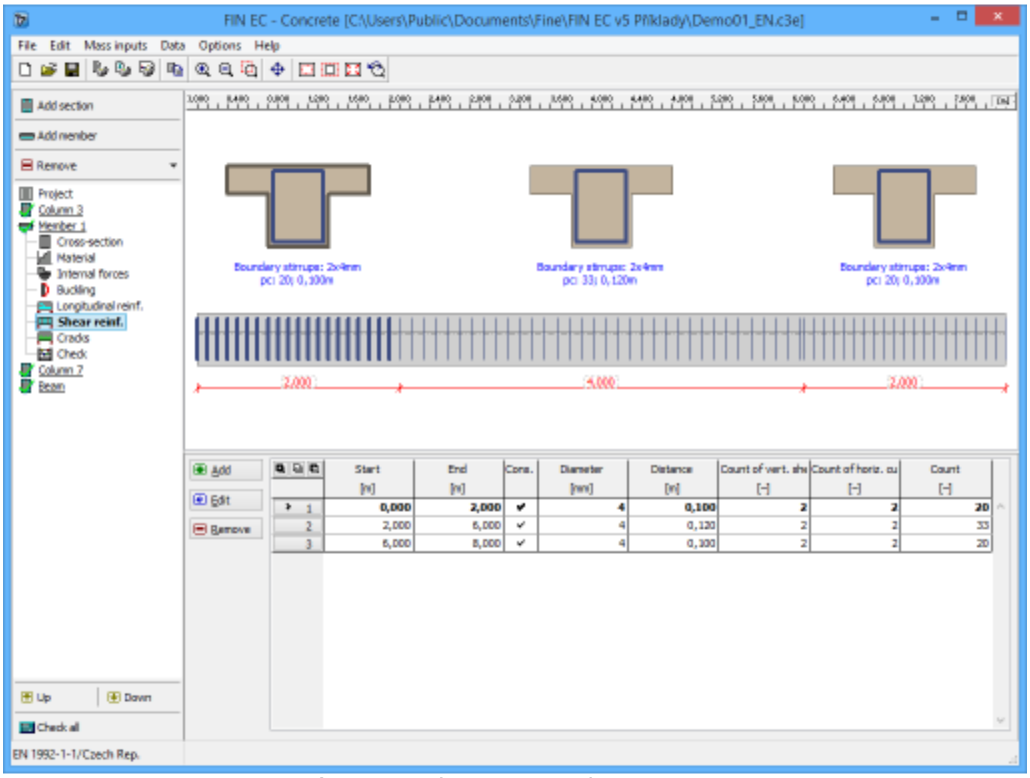


*Part "Longitudinal reinforcement" of member design*

## Shear reinforcement

The shear reinforcement can be specified in this part of the tree menu. The reinforcement parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different reinforcement parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of the shear reinforcement (type, diameter, number etc.) are organized in the window "**Edit reinforcement sector**". More sectors can be added (button "**Add**") for input of different shear reinforcement along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. The sector lengths may be edit by using the active dimensions, double-click on certain sector launches the appropriate window for sector edit.

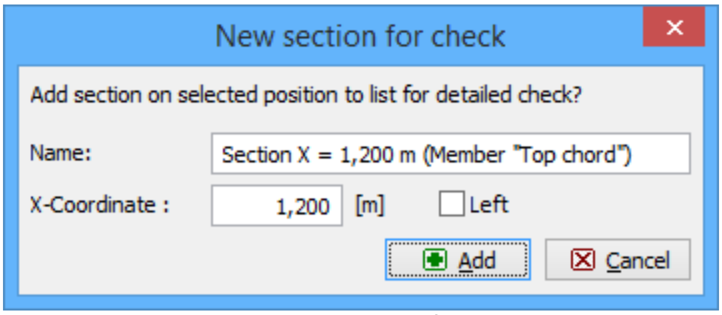


*Part "Shear reinforcement" of member design*

# Edit section for check

The detailed results in certain point can be displayed with the help of verification sections. These sections can be added or modified using window **"Edit section for check**". This window contains input lines for specification of the name and the section position (measured from the member beginning). Check box "**Left**" can be used in the points of discontinuity. The results on the left side of this point will be displayed if the check box is switched on.




*Window "Edit section for check"*

# Drawing settings

This window contains settings related to the appearance of workspace and displayed quantities.

Left part "**Drawing settings**" contains list of quantities that can be displayed in the workspace. Individual quantities can be displayed with the help of check box "**Draw**". Colour and description (setting "**Values**") may be specified for any quantity. Any quantity can be selected for mode "**Draw individually**" in part "**Overall setting**". Only few quantities are available for other modes (for example only $M_{Ed}$ and $M_{Rd}$ can be displayed for mode "**Draw moments**"). The part "**Overall setting**" contains also following options:

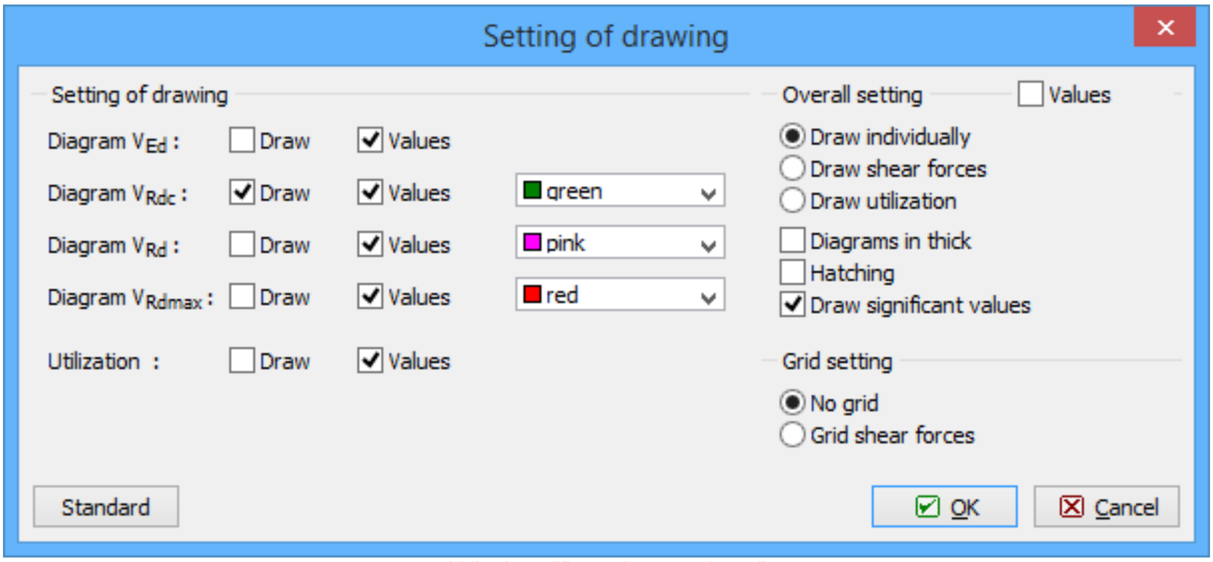
**Diagrams in thick** • Diagrams of quantities will be highlighted by thick line.

**Hatching** • Diagrams of quantities will be filled by hatching.

**Draw significant values** • Only extreme values will be described.

Grid and it's units can be switched on in the part "**Grid setting**".

Default settings can be loaded with the help of button "**Standard**".




*Window "Drawing settings"*

# Steel

# Cross-section edit

The geometry of steel cross-section can be specified in this window. The window contains cross-section preview and these buttons for cross-section input:

**Rolled, Welded, Solid, Rolled composite**
- The input of cross-sections that are considered as compact during the design. The geometry is defined in the window "**Cross-section editor**". The cross-section properties are also described in the chapter "**Cross-sections**" of theoretical part of the help.

**Rolled built-up, Welded built-up**
- The input of built-up cross-sections (built-up members consist of two or four partial cross-sections that are connected by battens or lacing). The geometry of partial cross-sections is defined in the window "**Cross-section editor**". The cross-section properties of built-up cross-sections are also described in the chapter "**Cross-sections**" of theoretical part of the help.

**User defined**
- The general input of cross-sectional characteristics.

**Editor**
- The option for the input of arbitrary geometry of the cross-section in the program "**Section**". Analysis of these cross-sections is described in the theoretical part of help.

**Details**
- Shows detailed cross-section characteristics in a new window

The cross-section can be rotated about its axis *1* using input line "**Rotation**". This feature can be used for cases where the load isn't applied in the directions of the main cross-section axis (e.g. purlins). The cross-section properties are also described in the chapter "**Cross-sections**" of theoretical part of the help.

The analysis method may be also specified here:

**Plasticity according to class**
- The verification method is performed according to the automatic classification of the cross-section. Available only for cross-sections from pre-defined database.

**Non-plastic calculation**
- The plastic resistance of the cross-section isn't considered during the design, even if the cross-section is classified as class *I.* or *II*. Analysis is performed according to the rules for classes *III.* or *IV*. Available only for cross-sections from pre-defined database.

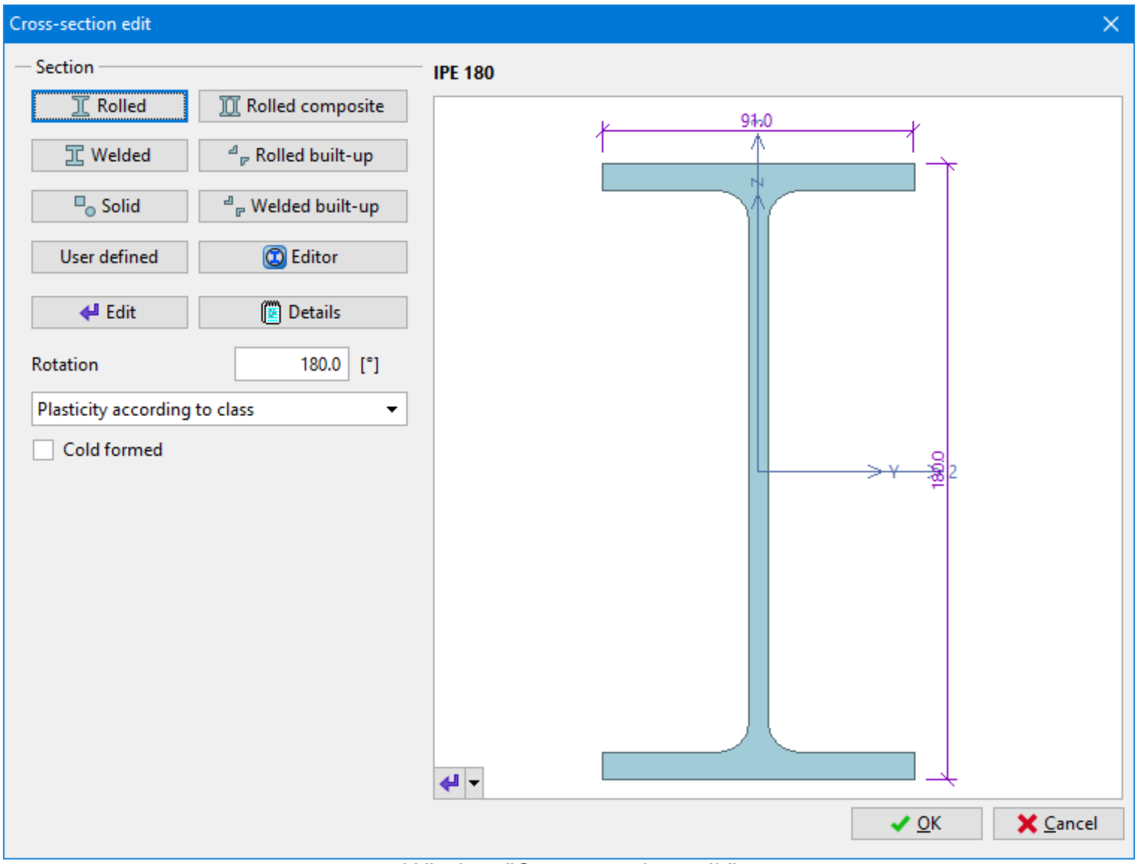
**Elastic calculation**
- The verification of the cross-section with the help of the elastic resistance. Available only for cross-sections created in the software "**Section**".

**Plastic calculation**
- The verification of the cross-section with the help of the plastic resistance. Available only for cross-sections created in the software "**Section**".

The setting "**Cold formed**" affects the selection of an appropriate buckling curve in accordance with the table *6.2* of EN 1993-1-1.

The shear areas $A_{vy}$, $A_{vz}$ for shear verification in directions $y$ and $z$ have to be specified for cross-sections created in the program "**Section**". The setting "**Estimate shear areas**" divides the total area of the cross-section uniformly.

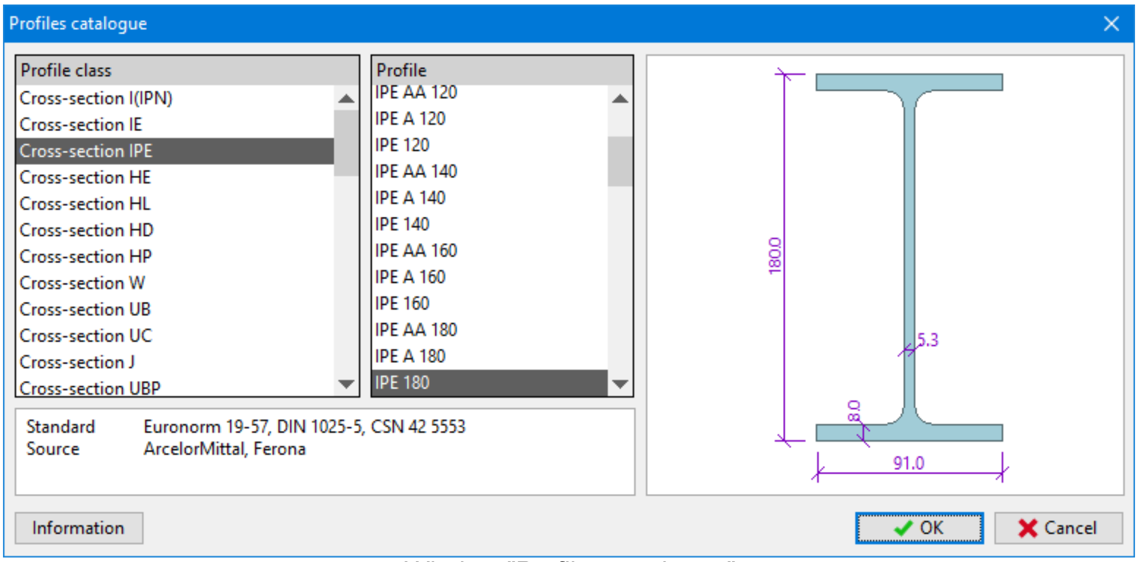



*Window "Cross-section edit"*

## Cross-section editor

The member cross-section can be modified in this window. The upper part contains library of available shapes (range differs according to the cross-section type). Dimensions or profile type can be entered in the table in the left part of the window. The meaning of dimensions is shown in the cross-section view in the right part of the window.

"**Information**" button in the left bottom corner shows complete list of cross-section characteristics.



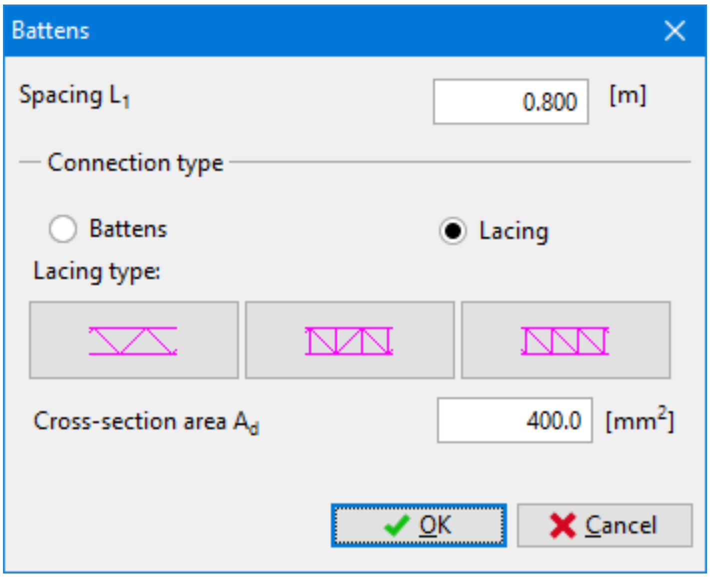

*Window "Profiles catalogue"*

## Battens

The properties of connection of built-up members may be specified in this window. The fundamental input is the "**Spacing $L_1$**" that represents the distance between the lacings or battenings. The following setting is the type of connection. The design standard contains in the chapter *6.4* two types of connections: "**Battens**" and "**Lacing**". Battens are specified by the height and width of their cross-section, lacing by the geometry and the cross-sectional area of bracings.

The analysis of built-up members is described in the theoretical part of help.



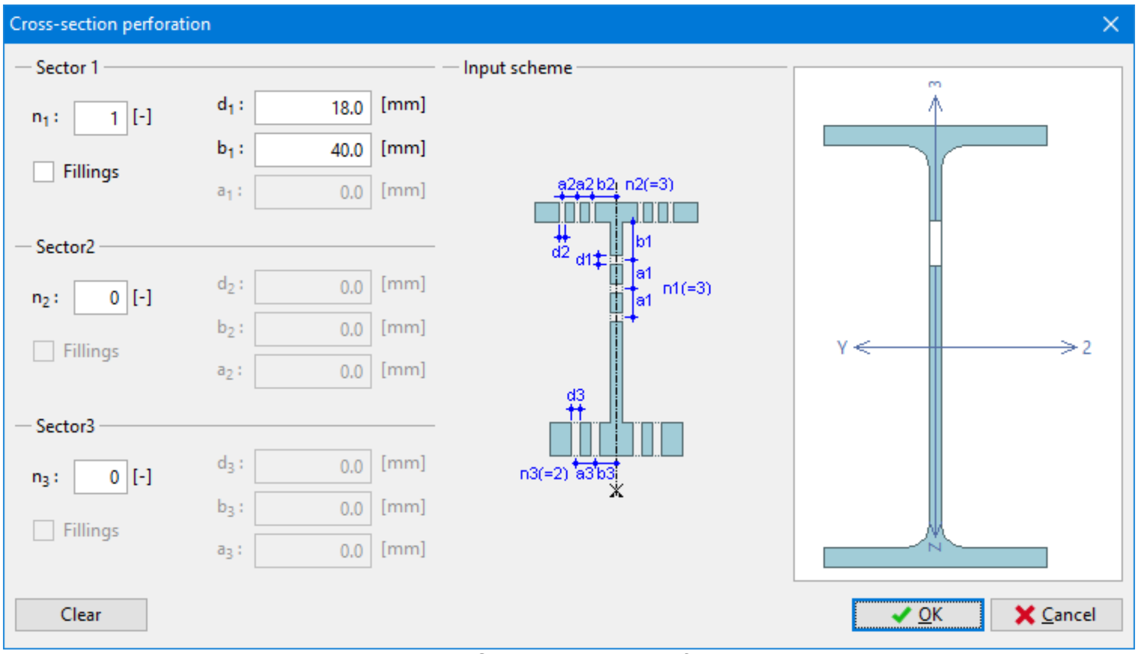

*Window "Battens"*

## Perforation edit

The perforation of the cross-section (caused for example by holes for connectors) may be specified in this window. Holes may be specified in all parts of the cross-section. The perforation is specified by the number of holes, their diameter, spacing and distance from edge. The window contains a cross-section view with explanation of all inputs. Right part of the window shows the cross-section with entered holes. Input of holes is limited in the way, that it isn't possible to specify unsymmetrical holes for symmetrical cross-section (e.g. left and right parts of flange have to be perforated in the same way). If the holes are filled with fasteners, the settings "**Fillings**" may be switched on. These holes aren't considered in the verification of compressive and bending resistance, if the holes are within compressive part of the cross-section.

This procedure is described in the part "**Perforation of cross-sections**" of the theoretical help.

The button "**Clear**" deletes all specified parameters.



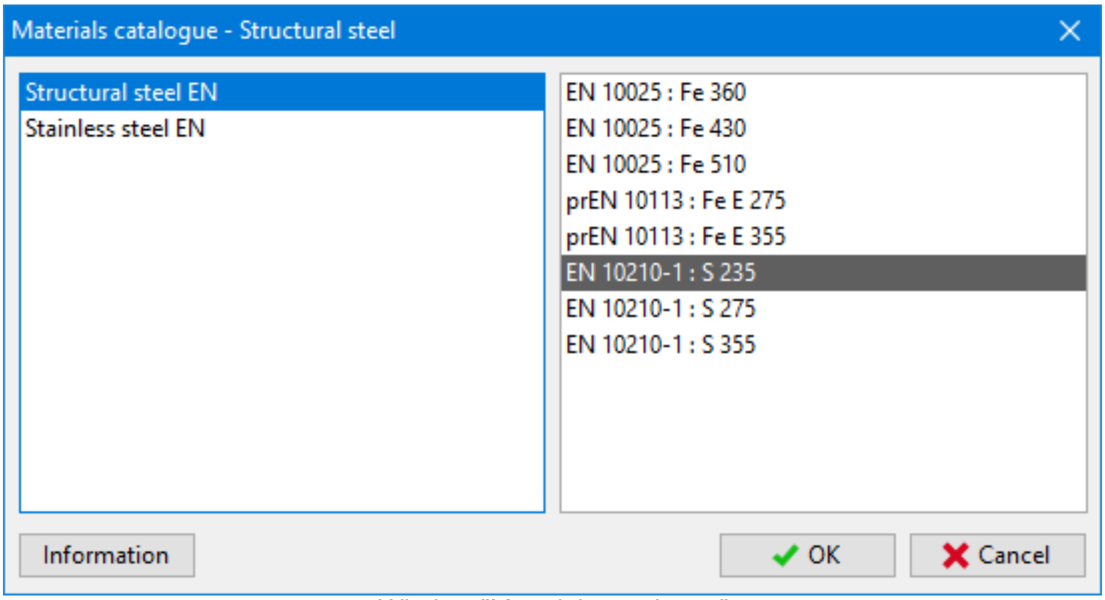

*Window "Cross-section perforation"*

## Materials catalogue

This window contains the database of steel strength grades. The grade names conatin also the name of corresponding standard.

The complete list of material characteristics for selected grade can be opened using "**Information**" button.

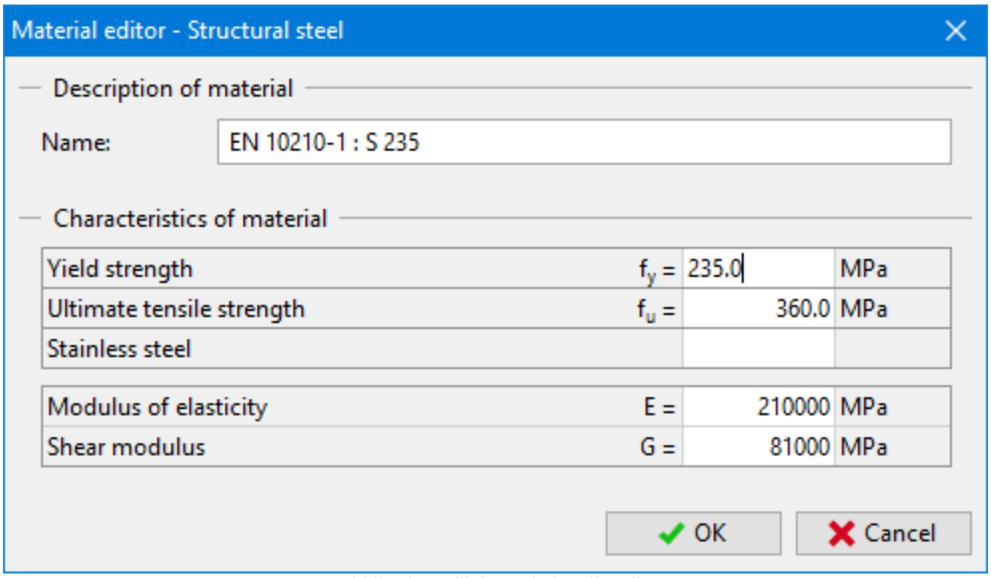
 




*Window "Materials catalogue"*

## Material editor

The arbitrary material characteristics can be specified in this window. The characteristics are described in the chapter "**Material characteristics**" of the theoretical part of the help.




*Window "Material editor"*

## Load edit

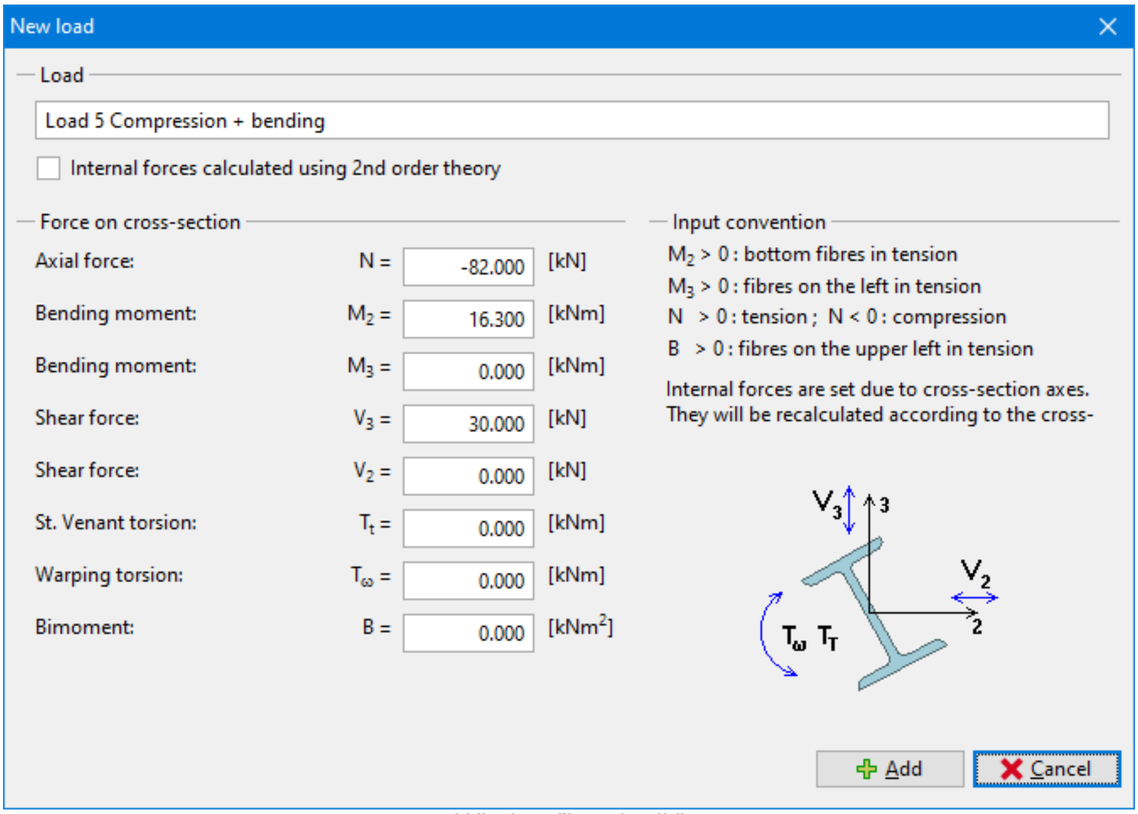
The internal forces and other properties of the load can be entered with the help of this window. These properties can be specified:

| | |
|---|---|
| **Load** | • Name of the load |
| **N** | • Normal force |
| **$M_2$** | • Bending moment about axis *2* (positive values represent tension in the bottom edge of the cross-section) |
| **$M_3$** | • Bending moment about axis *3* (positive values represent tension in the left edge of the cross-section) - only selected programs and task types |
| **$V_3$** | • Shear force in vertical direction (parallel with axis *3*) |
| **$V_2$** | • Shear force in horizontal direction (parallel with axis *2*) |
| **$T_t$** | • Torsional moment about member axis *1* - St. Venant torsion |
| **$T_\omega$** | • Torsional moment about member axis *1* - Warping torsion |
| **B** | • Bimoment |

If the check box "**Forces calculated acc. to 2nd order theory**", the analysis for this load will be performed without any consideration of buckling (the forces are already calculated on deformed structure.

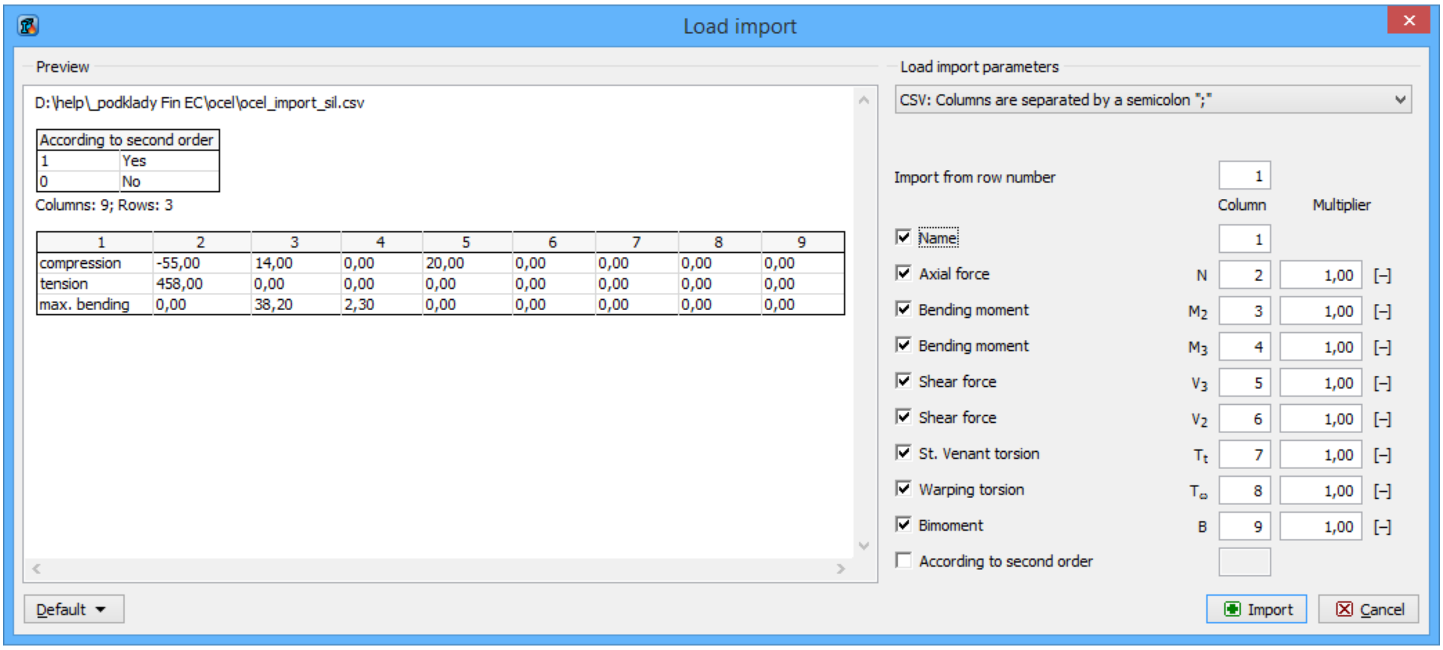
The range of available internal forces may differ according to the type of cross-section (e.g. torsional moments and bimoment aren't supported for built-up cross-sections).




*Window "Load edit"*

# Load import

Window "**Load import**" appears after loading the text or **.csv* source file. The data included in the source file can be arranged in this window. Left part shows content of imported file. The assignment of the certain column to the load entry (force or property) can be done in the right part of the window. Numerical entries can be also multiplied by specified multiplier. This multiplier can be used mainly for conversion caused by different units in the source file and in the program. Initial rows can be skipped using setting "**Import from row number**". This setting is helpful for source files with headings. Default units required by the software are *[kN]* and *[kNm]*.




*Window "Load import"*

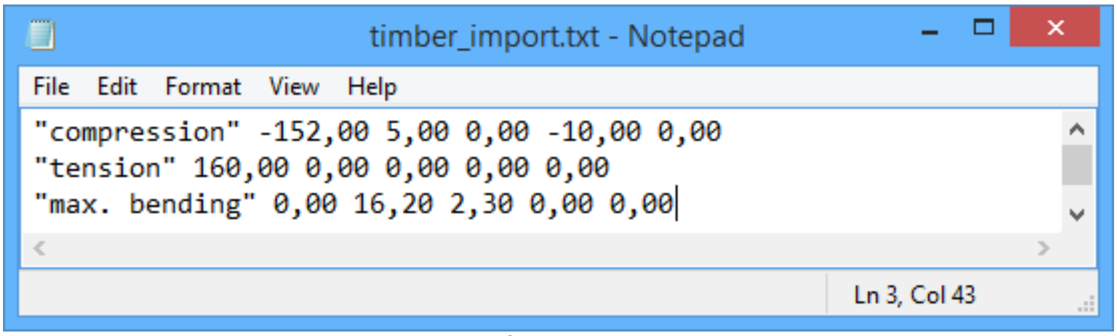
## Preparing text file

Text file can be created in any text editor (e.g. *Notepad*, *Word*, *Writer*). File format requires that every row contains one load case. Every row can contain all values of internal forces separated by space or tabulator. The order may differ comparing to the order of forces in the software, however, order has to be identical for all rows of the document. The load name and second order effect may be also specified for every load.

This scheme is used for consideration of analysis in accordance with II. order theory:

| | |
|---|---|
| 1 | Forces calculated in accordance with II. order theory |
| 0 | Forces aren't calculated in accordance with II. order theory |

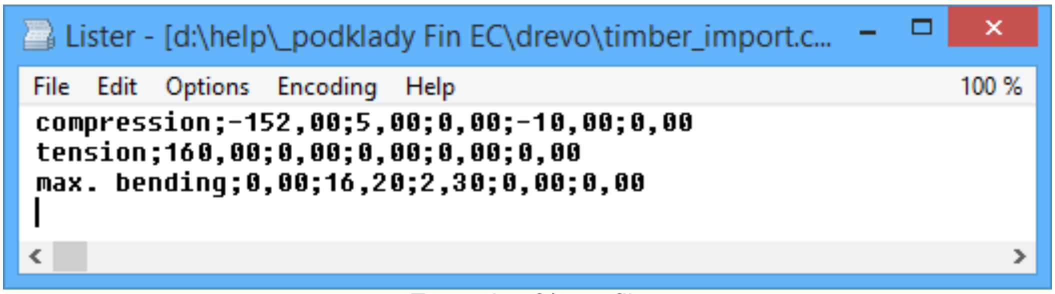
The file can be also created using part of analysis documentation from arbitrary structural analysis software.




*Text file in Notepad*

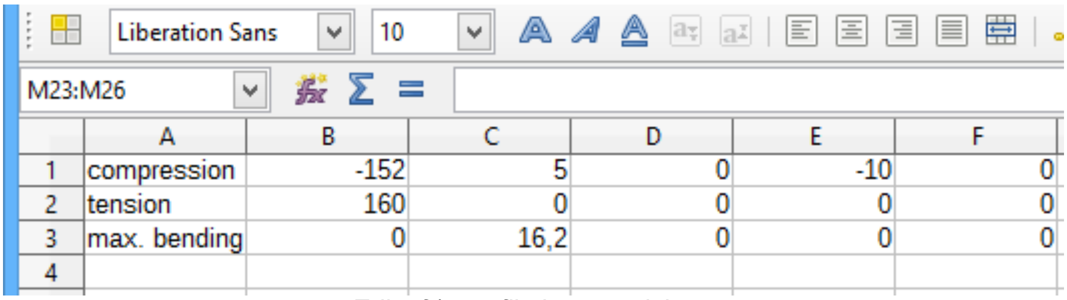
## Preparing **.csv* file

The rules for **.csv* (comma-separated values) files are almost identical . Main difference is, that the particular values are separated by semicolon ";".




*Example of *.csv file*

This file type can be easily created using spreadsheet programs like *Excel* or *Calc*. Created document can be saved as **.csv* file with appropriate separator.



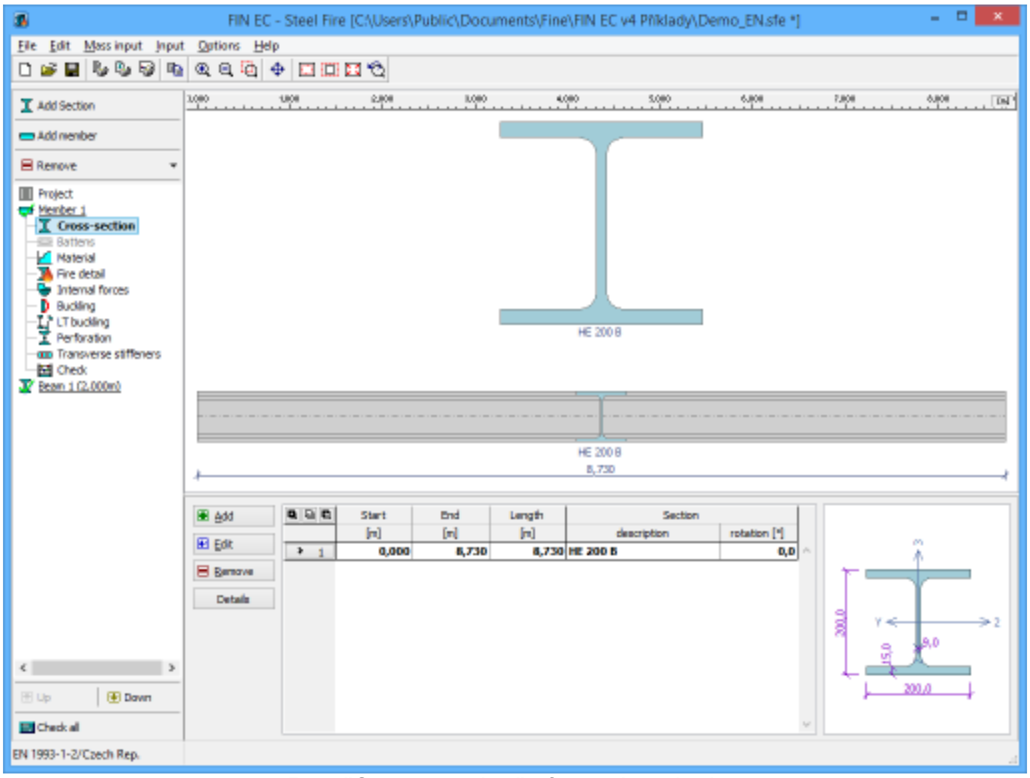

*Edit of *.csv file in spreadsheet*

# Cross-section

The member cross-section may be specified in this part of the tree menu. The cross-sectional parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sector may contain different cross-section. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of cross-section are organized in the window "**Cross-section edit**". More sectors can be added (button "**Add**") for input of different fire resistance parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

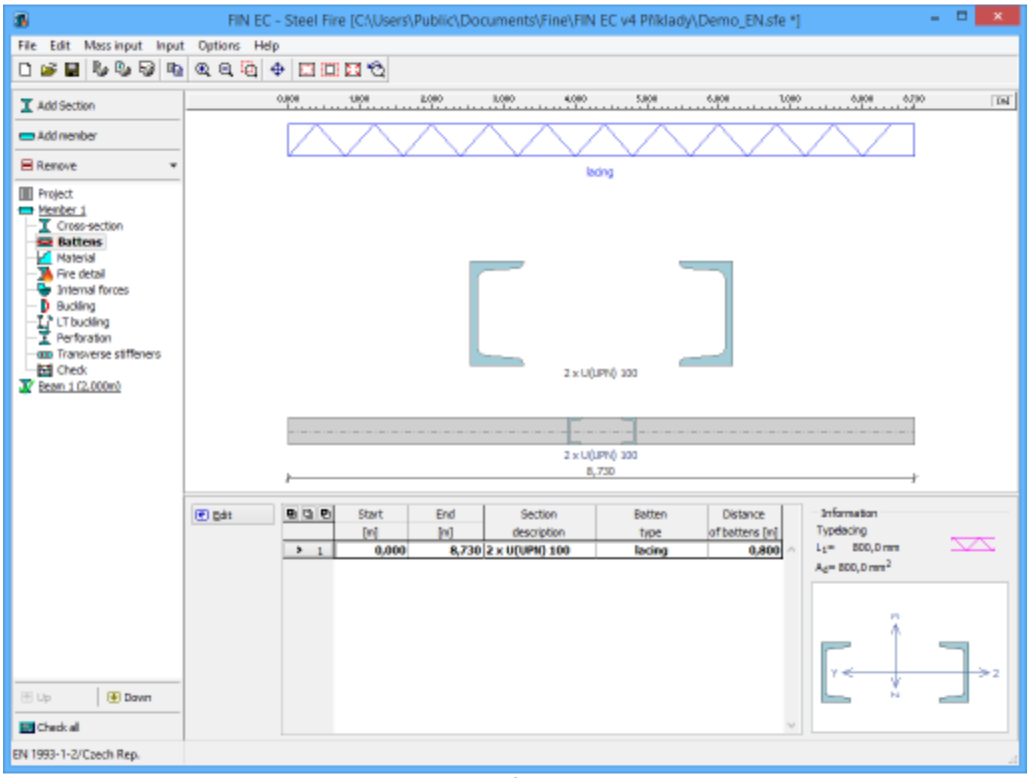
If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the cross-section geometry will be automatically copied from this program.




*Part "Cross-section" of member design*

## Battens

The connection of built-up members can be specified here. This part is disabled if built-up cross-section isn't selected in the part "**Cross-section**". The sectors for the input of connection parameters correspond to the sectors in the part "**Cross-section**". Edit of sector count and their length isn't possible here. The parameters of the connection are organized in the window "**Battens**", that can be launched by the button "**Edit**". Parameters of connection for the active sector are displayed in the right part of the input frame.




*Part "Battens" of member design*

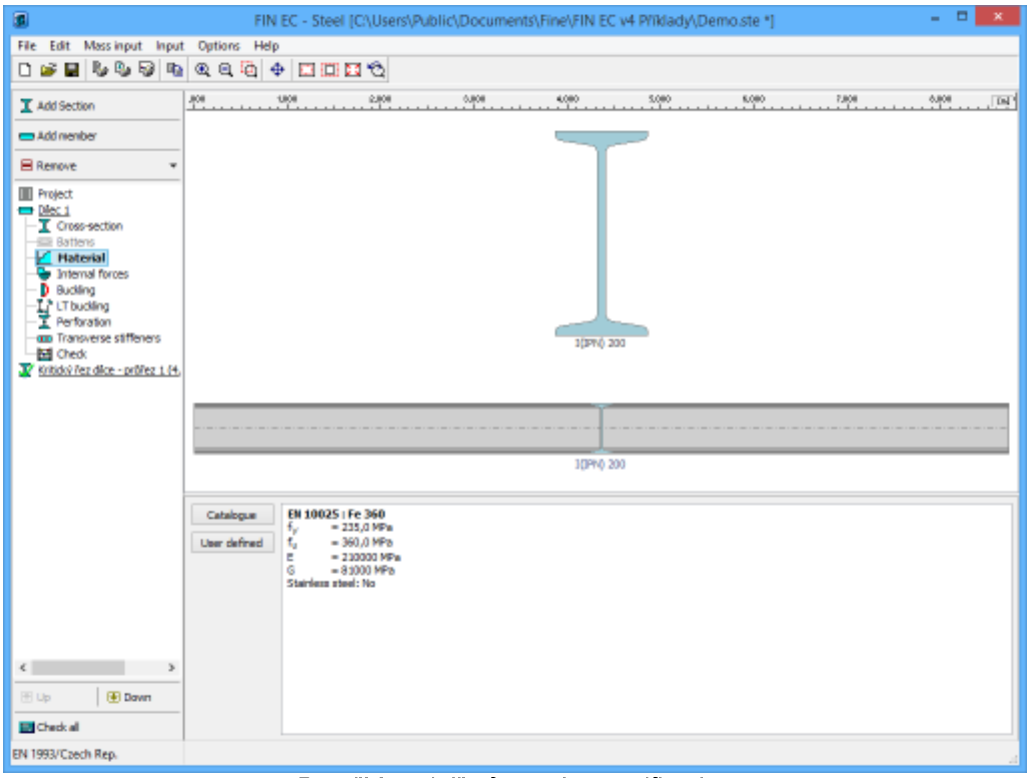
 

# Material

The material of the member can be specified in this part. Timber grade can be selected in the window "**Materials catalogue**", which can be launched using "**Catalogue**" button. The material with non-standard values of properties can be specified in the window "**Material editor**" using "**User defined**" button. The strength properties of entered timber are automatically shown in dedicated frame.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the material will be automatically copied from this program.

The characteristics are described in the chapter "**Material characteristics**" of the theoretical part of the help.




*Part "Material" of member verification*

# Internal forces

The internal forces along the member length can be specified in this part. More loads (load combinations) can be specified for any member.

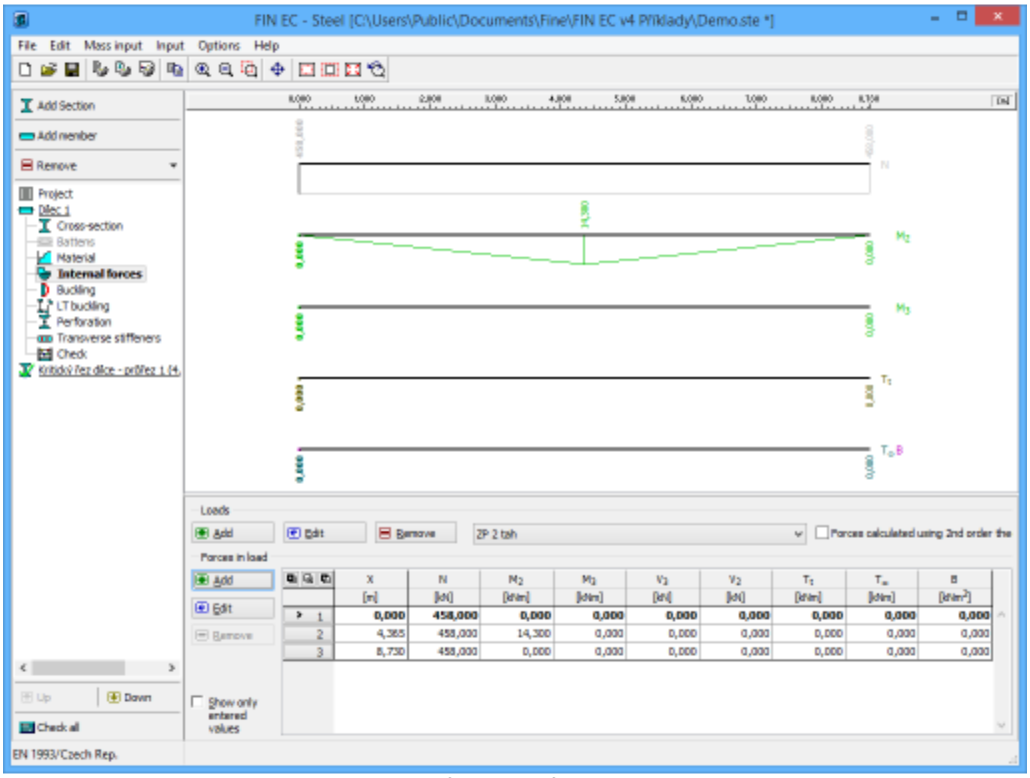
If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the internal forces will be automatically copied from this program.

## Loads

The upper part of the input frame contains buttons for input and edit of loads. The load is a set of internal forces (design values), that corresponds to the results of design combinations. The basic properties of the load can be specified in the window "**Loads**".

Any load contains a setting "**Forces calculated acc. to 2nd order**". The load won't be verified including buckling consideration, if this setting is switched on for the certain load.

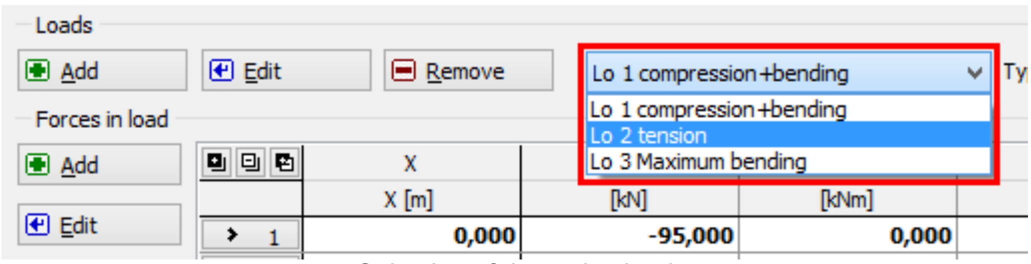
 



*Part "internal forces" of member design*

## Input of internal forces

The active load has to be selected before starting the input of internal forces. The active load can be selected using list box above the table with values of internal forces.



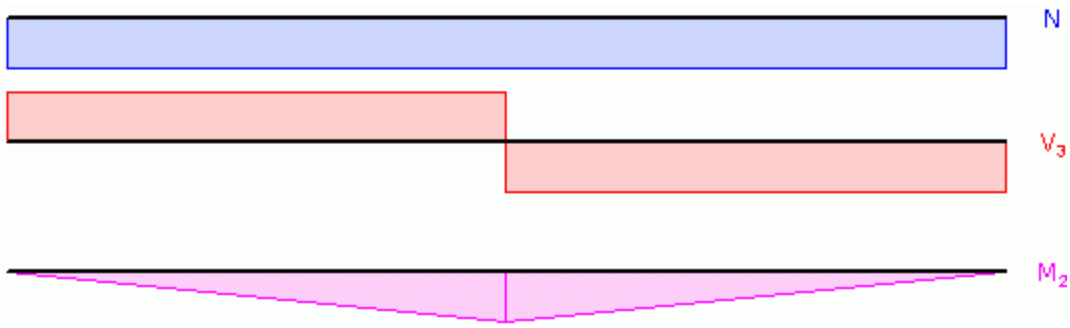

*Selection of the active load*

The internal forces are entered with the help of values in the certain points along the member length. These points should be specified mainly in the positions of local minima or maxima or in the inflection points. Intermediate values are determined automatically using linear interpolation. These points are organized in the table in the bottom part of the application window and can be entered with the help of dedicated window.

The table shows both specified and calculated values of internal forces for each point. The automatically calculated values can be hidden using setting "**Show only entered values**".

## Example of entered internal forces

The following figure shows diagram of normal force $N$, shear force $V_3$ and bending moment $M_2$. The table shows the entered internal forces, that has to be entered in the certain positions.




*Diagrams of internal forces*

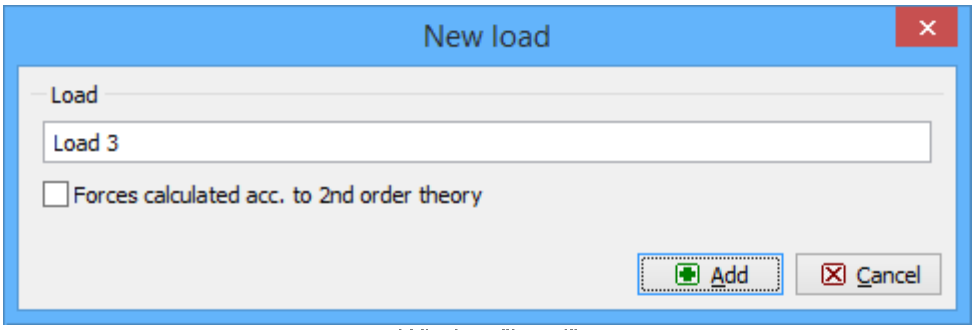
These diagrams can be created using these values:

| Section number | Position *x* [m] | N [kN] | $V_3$ [kN] | $M_2$ [kN] |
|---|---|---|---|---|
| 1 | 0,00 | 20,00 | -10,00 | 0,00 |
| 2 | 2,50 | | -10,00 (left) | 25,00 |
| 3 | 2,50 | | 10,00 (right) | |
| 4 | 5,00 | 20,00 | 10,00 | 0,00 |

Intermediate values are calculated automatically using linear interpolation.

## Loads

The new member load and its properties can be entered with the help of this window. If the check box "**Forces calculated acc. to 2nd order theory**", the analysis for this load will be performed without any consideration of buckling (the forces are already calculated on deformed structure.




*Window "Load"*

## Force edit

The internal forces in the certain member point can be entered with the help of this window. These properties can be specified:

**x** • Basic input, that specifies the position of the point along the member length. The position is measured from the left end of the member

**N** • Normal force

**$M_2$** • Bending moment about axis *2* (positive values represent tension in the bottom edge of the cross-section)

**$M_3$** • Bending moment about axis *3* (positive values represent tension in the left edge of the cross-section) - only selected programs and task types

**$V_3$** • Shear force in vertical direction (parallel with axis *3* )

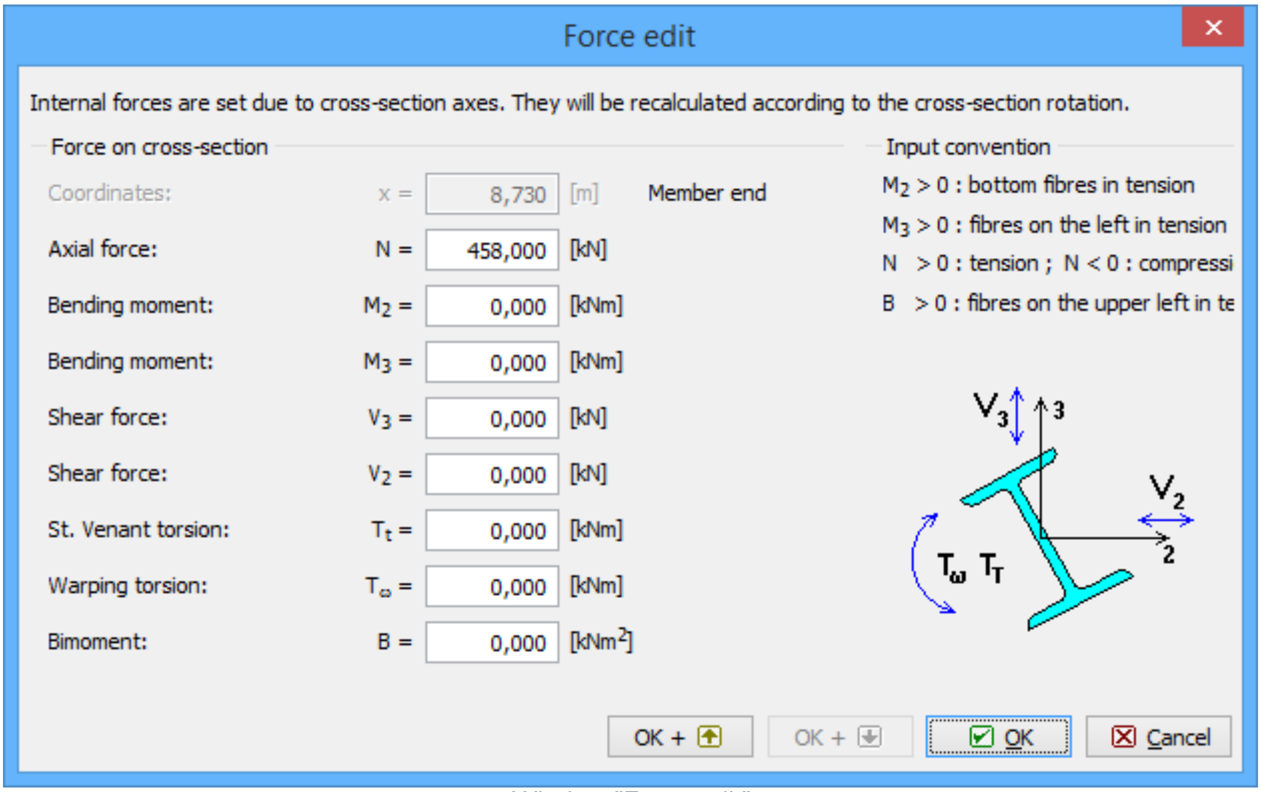
**$V_2$** • Shear force in horizontal direction (parallel with axis *2* )

**$T_t$** • Torsional moment about member axis *1* - St. Venant torsion

**$T_\omega$** • Torsional moment about member axis *1* - Warping torsion

**B** • Bimoment

Values of all internal forces shall be specified only in the first (*x=0* ) and last (*x* is equal to member length) sections. Intermediate sections may contain unfilled certain internal forces. The values of these forces are obtained with the help of linear interpolation using the closest values that were specified in other sections.




*Window "Force edit"*

## Buckling

The buckling parameters can be specified in this part of the tree menu. List box "**Buckling for calculation**" contains these three styles of buckling verification:

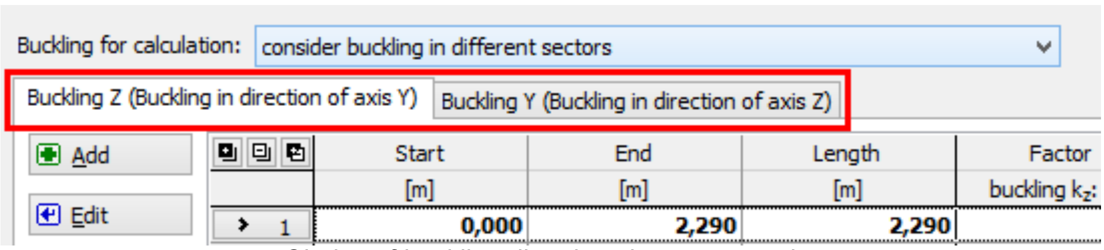
**Consider buckling in identical sectors**
- The buckling effect will be considered during the analysis, buckling parameters (lengths, end conditions) have to be specified in the same length sectors for both buckling directions.

**Neglect buckling**
- The buckling effect won't be considered during the member analysis. Can be used for members, where the buckling is prevented.

**Consider buckling in different sectors**
- The buckling effect will be considered during the analysis, buckling parameters (lengths, end conditions) are specified in the different length sectors for both buckling directions.

If the axes *y* and *z* aren't the main axes of the cross-section (e.g. *L*-profiles), the buckling is considered in directions of the main cross-sectional axes $\eta$ and $\zeta$ during the design. The buckling analysis in directions *y* and *z* may be forced by switching off the setting "**Buckling to main axes η, ζ**".

The bucking parameters can be different along the member length. In this case, the member has to be divided into buckling sectors. The buckling sectors can be specified in the table in the bottom part of the main window. The buckling parameters are organized into two tabs "**Buckling Z**" and "**Buckling Y**" for analysis style "**Consider buckling in different sectors**".

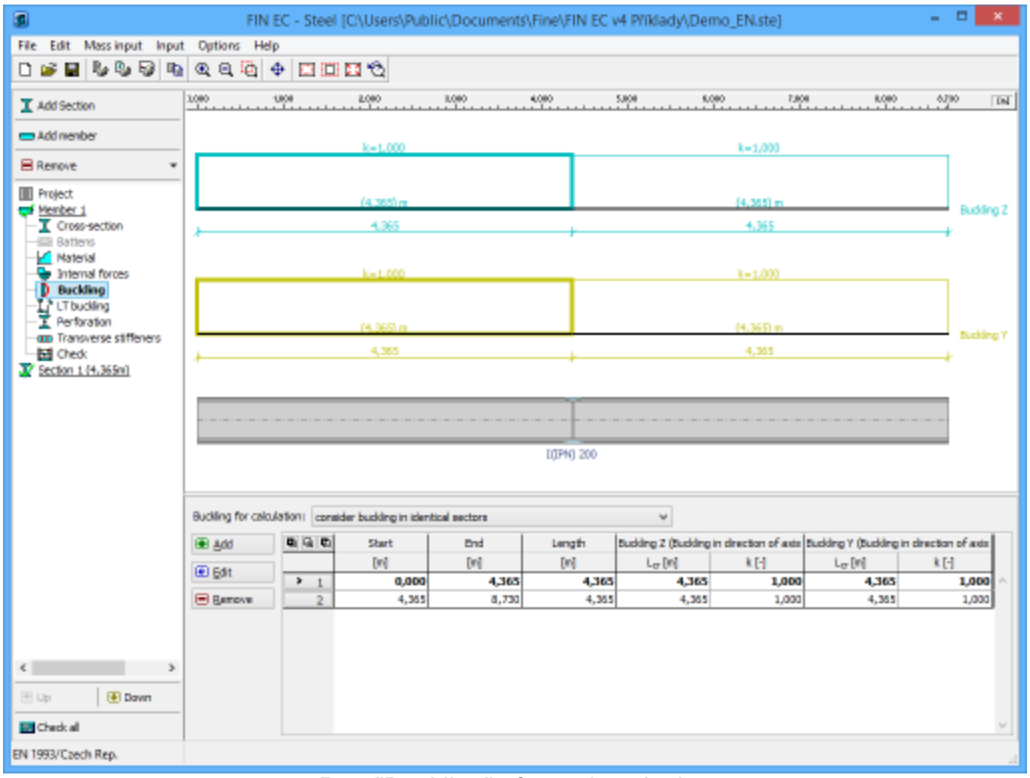



*Choice of buckling directions in separate tabs*

The table contains particular member sectors, every sector may contain different buckling parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The buckling parameters are organized in the window "**Buckling**". More sectors can be added (button "**Add**") for input of different buckling parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

The input is performed in the similar way for style "**Consider buckling in identical sectors**", however only one sectors table is available. The sector properties are organized in the window "**Edit buckling sector**".

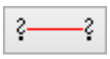



*Part "Buckling" of member design*

## Buckling

First part "**Buckling check**" of the window contains basic settings for the certain buckling direction. Buckling verification can be switched of with the help of "**Neglect buckling**" setting. It's also possible to specify different basic sector length for the calculation of buckling length.

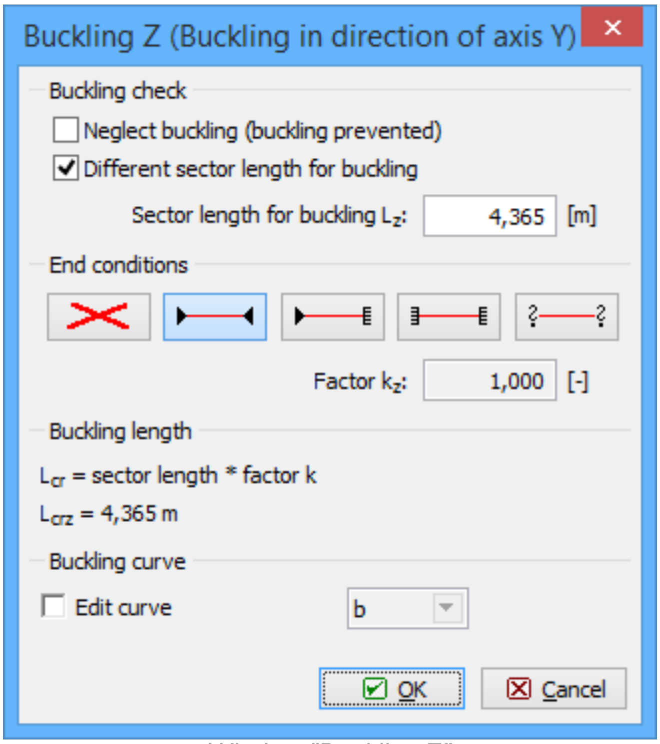
The part "**End conditions**" contains several options for specifying the end conditions for buckling length calculation. The appropriate value of factor $k$ is used according to the selected conditions. This factor multiplies the basic sector length for buckling $L_x$. The result of this multiplication is the buckling length $l_{cr}$, that is used in the buckling analysis. These end conditions are available:

| | | |
|---|---|---|
|  | **Not specified** | • The default setting of the program, factor $k$ isn't specified, buckling length $l_{cr}$ isn't calculated. The analysis of buckling cannot be performed for this option. |
|  | **Pinned-pinned** | • Both ends are considered as fixed. The value of factor $k$ is equal to *1.0*. The most common option for timber structures. |
|  | **Pinned - fixed** | • One end is considered as pinned, another one as fixed. The value of factor $k$ is equal to *0.7*. |
|  | **Fixed-fixed** | • The member is considered as fixed on the both ends. The value of factor $k$ is considered as *0.5*. |
|  | **User input** | • The arbitrary value of factor $k$ can be specified using this option. For example, the value *2.0* can be specified for cantilever. |

Part "**Buckling length**" shows final value of buckling length. This value is used as $l_{cr}$ in the stability analysis. This part is described in the chapter "**Buckling**" of the theoretical help.

The part "Buckling curve" contains ans ability to specify manually the buckling curve according to the table *6.2* of EN 1993-1-1. This option may be used for settings that aren't considered in the software (e.g. thick welds for welded boxes) or fir non-standard cases (e.g. welded *I*-profile used for the input of hot-rolled cross-section).

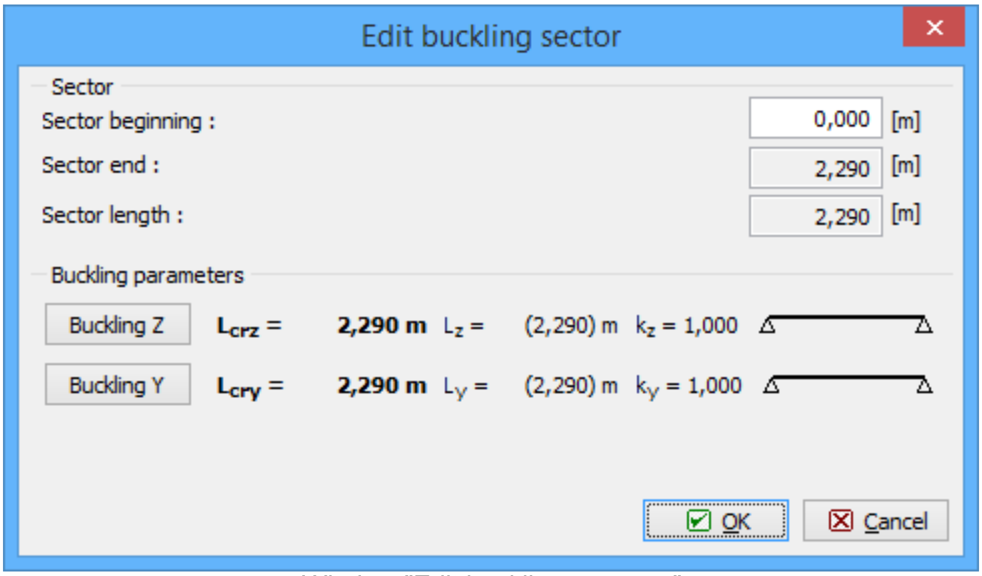



*Window "Buckling Z"*

## Edit buckling sector

The data necessary for calculation of buckling lengths $l_{cry}$ and $l_{crz}$ for particular sector of a member length can be specified in this window. Basic parameter is "**Sector beginning**", that specifies also the end of previous member sector. This value is measured from the member beginning. The sector end and its length are also displayed.

Part "**Buckling parameters**" contains buttons "**Buckling Z**" and "**Buckling Y**" for input of parameters, that are used for calculation of buckling lengths $l_{cry}$ and $l_{crz}$ for active sector. Parameters are specified in the window "**Buckling**". All these parameters (supporting style, basic length and factor *k*) are displayed also in this window on the right side of the buttons.



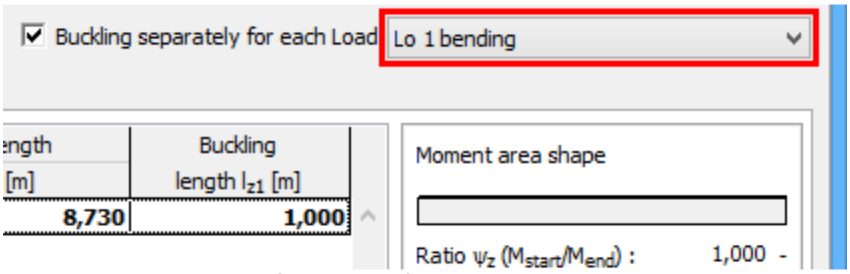

*Window "Edit buckling segment"*

## LT buckling

The lateral torsional buckling parameters can be specified in this part of the tree menu. List box "**LT buckling for calculation**" contains these two styles of LT buckling verification:

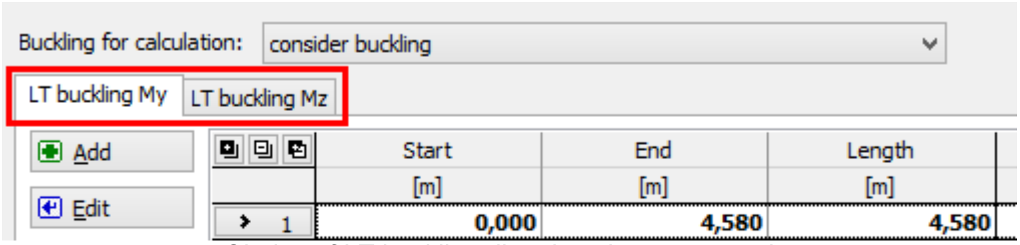
| | |
|---|---|
| **Consider buckling** | • The lateral torsional buckling will be considered during the analysis |
| **Do not consider buckling** | • The lateral torsional buckling won't be considered during the analysis. Can be used for members, where the LT buckling is prevented. |

As the parameters of lateral torsional buckling depend on the moment distribution, the parameters may differ for individual loads. The same buckling parameters are considered for all loads as a default. The unique parameters for individual loads may be entered after using the setting "**Buckling separately for each load**". If the setting is switched on, the list box with all entered loads appears on the right side of the setting. The buckling parameters has to be specified individually for all loads (the load displayed in the list box is the active one for the parameters input) in this case.



*Selection of active loading*

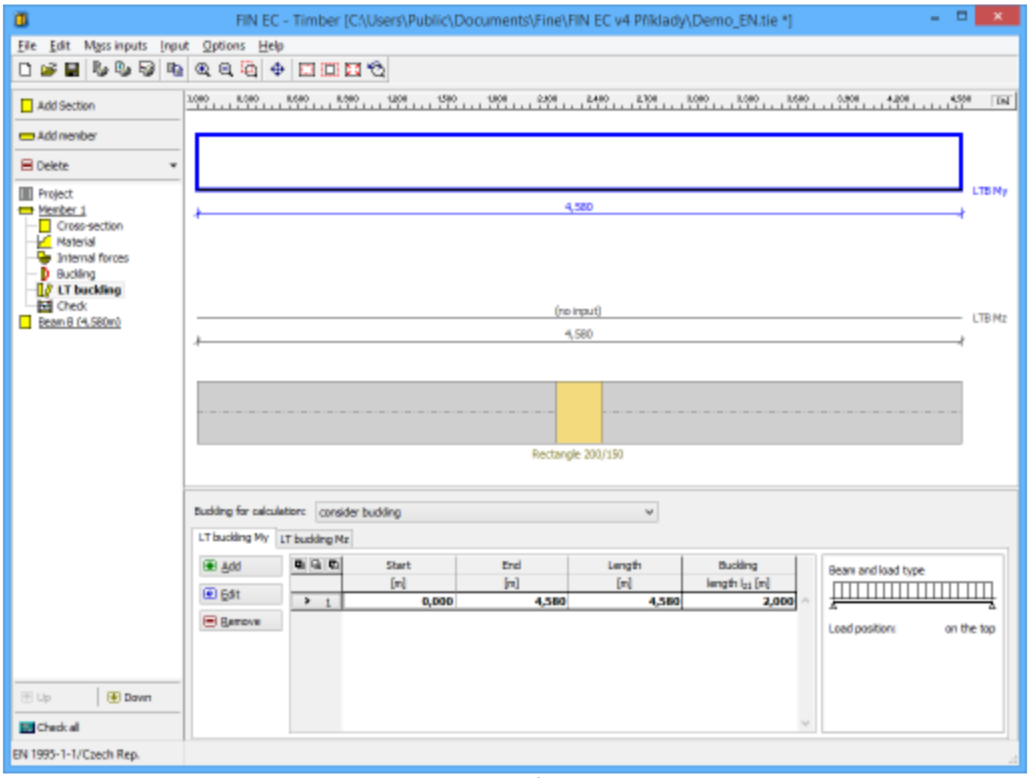
The LT bucking parameters can be different along the member length. In this case, the member has to be divided into particular sectors. The LTB sectors can be specified in the table in the bottom part of the main window. The LTB parameters are organized into two tabs "**LT buckling My**" and "**LT buckling Mz**" according to the buckling direction.



*Choice of LT buckling directions in separate tabs*

The table contains particular member sectors, every sector may contain different LTB parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The buckling parameters are organized in the window "**LT buckling parameters**". More sectors can be added (button "**Add**") for input of different LTB parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.



*Part "LT buckling" of member design*

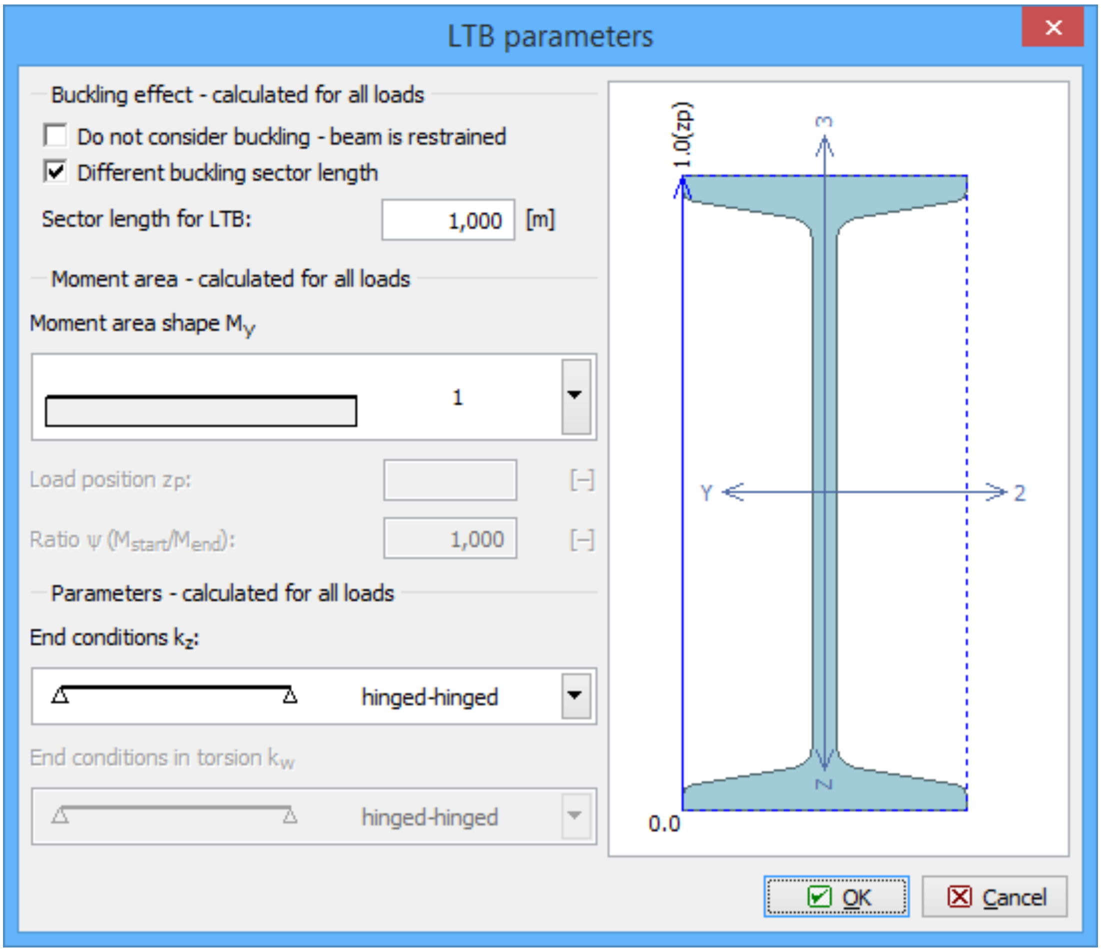
 

# LT buckling parameters

First part "**Buckling effect**" of the window contains basic settings for the certain lateral torsional buckling direction. LT buckling verification can be switched of with the help of "**Neglect LTB**" setting. It's also possible to specify different basic sector length $l_1$ for the calculation of LT buckling length.

The following parameter is the moment distribution according to the table *6.6* of EN 1993-1-1. The appropriate value of the correction factor $k_c$ is selected according to the moment distribution. There are some additional parameters for certain moment distribution styles. The proportion $\psi$ of bending moments at the beginning and end pf the sector has to be specified for trapezoidal distribution. The load position with respect to the centre of gravity has to be specified in the most of cases. The position is specified as a factor with the interval *<0;1>*, the value *0* means bottom (left) edge and the value *1* means upper (right) edge of the cross-section.

The part "**Parameters**" contains an option for input of supporting method of the sector (end conditions). The end conditions affect the coefficient $k_z$. This factor reduces the fundamental buckling length and is used for the calculation of the critical moment. The value of the coefficient $k_z$ is *1.0* for end conditions "**hinge-hinge**" and *0.5* for the end conditions "**fixed-fixed**". The end conditions for torsion have to be specified for moment areas with variable direction. This parameter affects the coefficient $k_w$, that is used for the calculation of the parameter $\kappa_{wt}$. The values of the factor $k_w$ are identical to the values of $k_z$. The analysis of lateral torsional buckling is described in the theoretical part of help.




*Window "LTB parameters"*

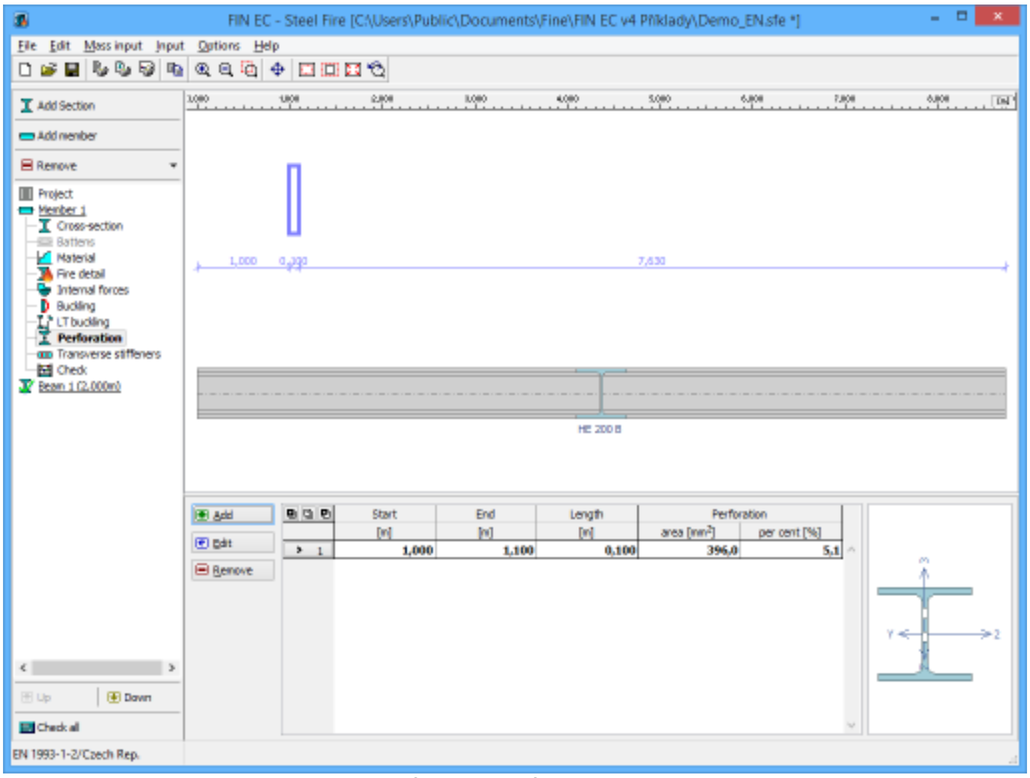
# Perforation

The perforation of the cross-section (caused for example by holes for connectors) may be specified in this part of the tree menu. Perforation can be specified in particular sectors, any sector is specified by the beginning and length. Sectors can be added with the help of the button "**Add**", perforation properties are organized in the window "**Perforated sector edit**". Modifications of existing sectors may be done with the help of buttons "**Edit**" and "**Remove**".

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

Specified perforation reduces the cross-sectional characteristics of the member, however, the resistance of the cross-section may be higher, as the ultimate strength $f_u$ is used in the analysis. This procedure is described in the part "**Perforation of cross-sections**" of the theoretical help.

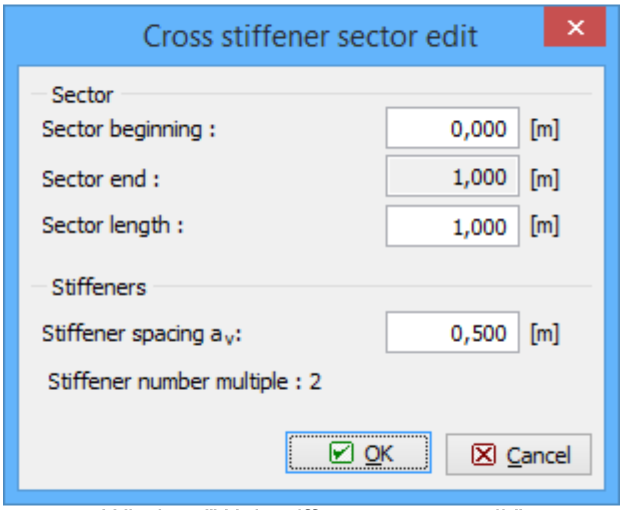


*Part "Perforation" of member design*

## Web stiffeners

The web stiffeners may be specified in this part of the tree menu. Stiffeners can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different parameters of stiffeners. Sectors can be added with the help of the button "**Add**". Modifications of existing sectors may be done with the help of buttons "**Edit**" and "**Remove**".

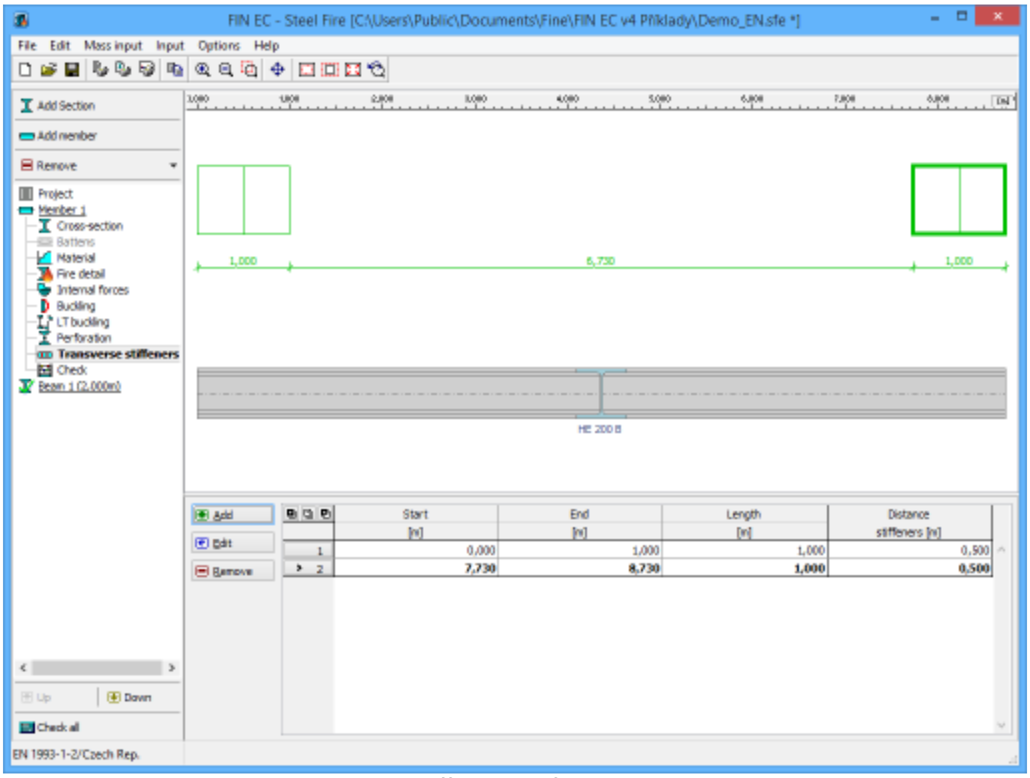
The sector properties are organized in the window "**Web stiffeners sector edit**" and consist of following parameters: sector beginning measured from the member origin, sector length and spacing of web stiffeners.




*Window "Web stiffeners sector edit"*

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

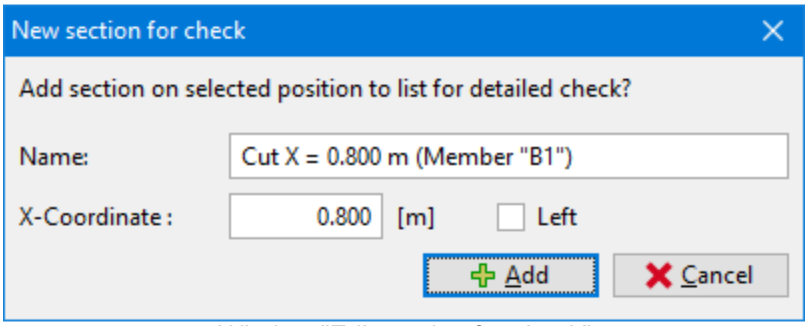
 



*Part "Web stiffeners" of member design*

## Edit section for check

The detailed results in certain point can be displayed with the help of verification sections. These sections can be added or modified using window **"Edit section for check**". This window contains input lines for specification of the name and the section position (measured from the member beginning). Check box "**Left**" can be used in the points of discontinuity. The results on the left side of this point will be displayed if the check box is switched on.




*Window "Edit section for check"*

# Timber

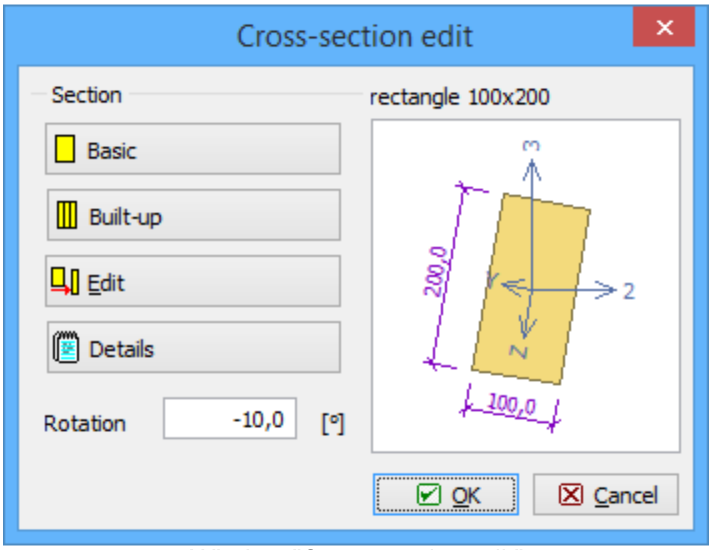
## Cross-section edit

The geometry of timber cross-section can be specified in this window. The window contains cross-section preview and these buttons for cross-section input:

| | |
|---|---|
| **Basic** | • The basic range of the cross-sections (rectangular, circular, *I*- , *T*- and *Pi*-shapes) can be entered with the help of this window. The input of cross-section is performed in the "**Cross-section editor**" window. |
| **Built-up** | • The built-up cross-sections (cross-sections made of more particular parts) can be entered with the help of this window. The input of cross-section is performed in the "**Cross-section editor**" window. |
| **Edit** | • The dimensions of existing cross-section can be changed easily with the help of this button. The "**Cross-section editor**" is launched in appropriate mode (for basic or built-up cross-sections) and the current dimensions are predefined. |
| **Details** | • Shows detailed cross-section characteristics in a new window |

The cross-section can be rotated about its axis *1* using input line "**Rotation**". This feature can be used for cases where the load isn't applied in the directions of the main cross-section axis (e.g. purlins).

The cross-section properties are also described in the chapter "**Cross-sections**" of theoretical part of the help.

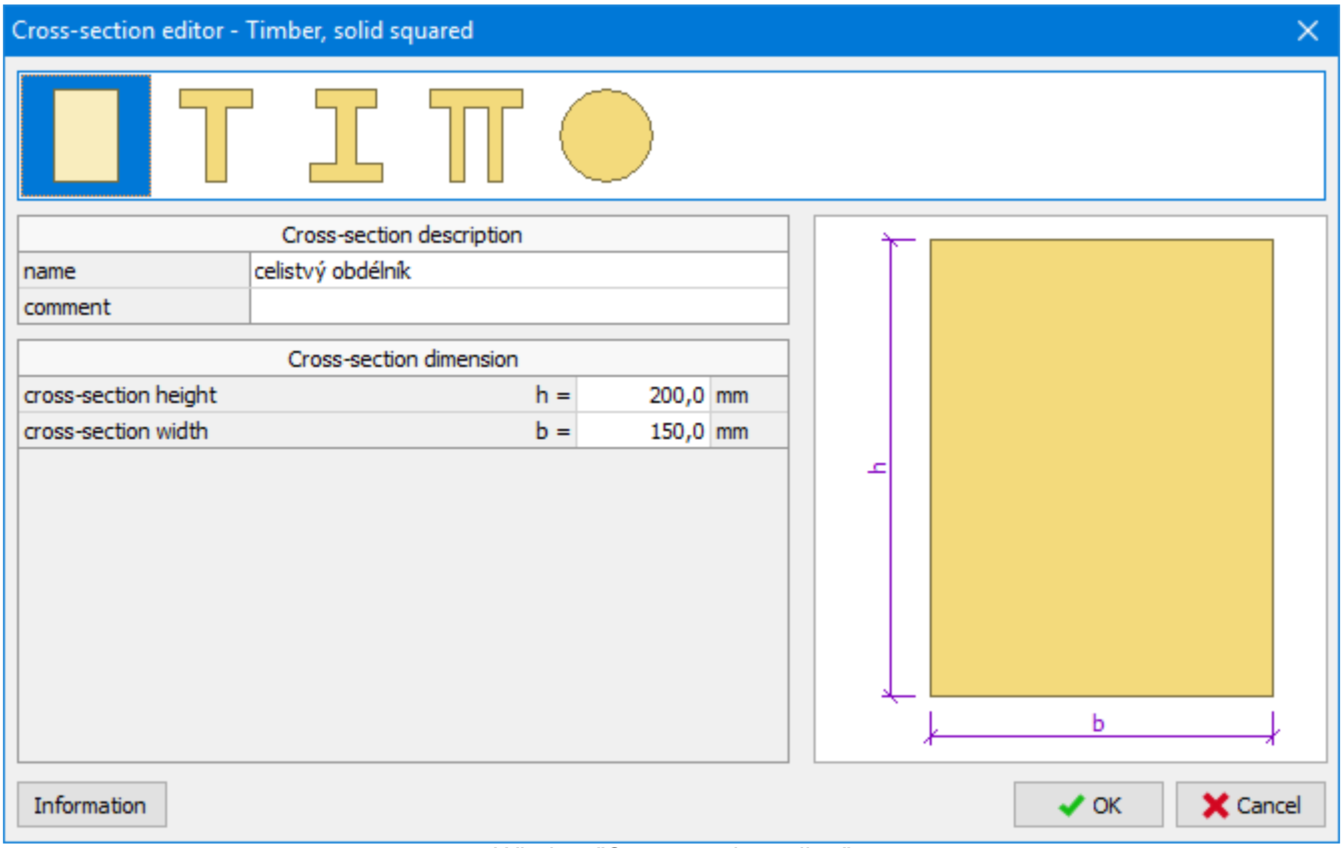



*Window "Cross-section edit"*

## Cross-section editor

The member cross-section can be modified in this window. The upper part contains library of available shapes (range is different for "**Basic**" and "**Built-up**" cross-sections). Dimensions can be entered in the table in the left part of the window. The meaning of dimensions is shown in the cross-section view in the right part of the window.

"**Information**" button in the left bottom corner shows complete list of cross-sectional characteristics.



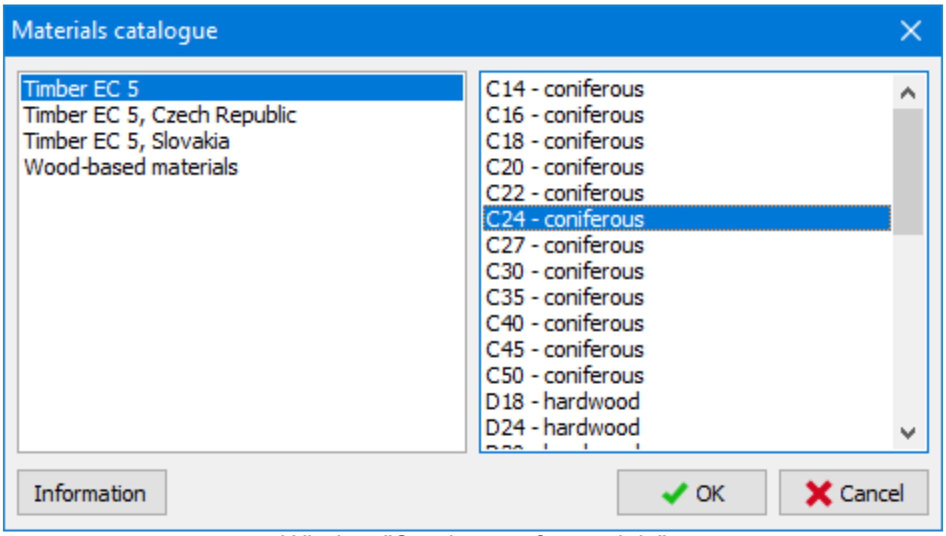

*Window "Cross-section editor"*

## Materials catalogue

This window contains the database of timber strength grades. Basic grades in accordance with EN 338 (softwood, hardwood) and EN 1194 (glued laminated timber) are included in the list "**Timber EC5**". National timber grading is defined in accordance with EN 1912. The characteristics are described in the chapter "**Material characteristics**" of the theoretical part of the help.

The complete list of material characteristics for selected grade can be opened using "**Information**" button.

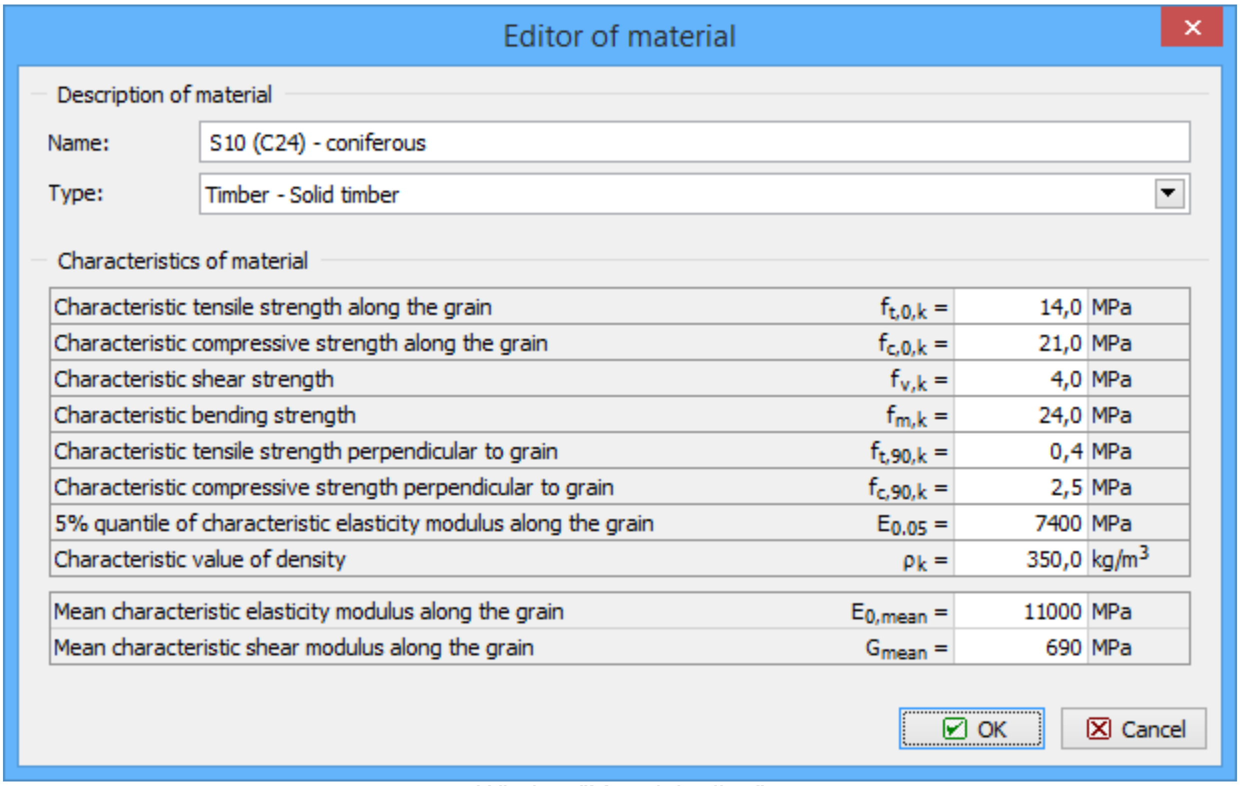
 



*Window "Catalogue of materials"*

## Material editor

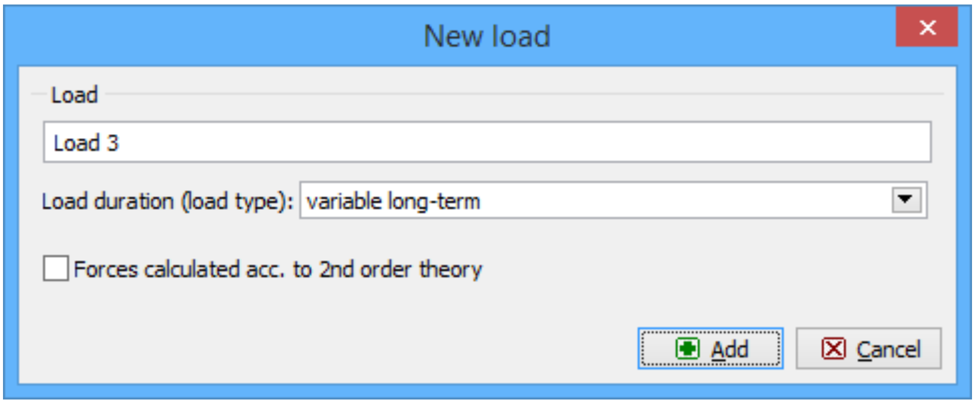
The arbitrary material characteristics can be specified in this window. The characteristics are described in the chapter "**Material characteristics**" of the theoretical part of the help.



*Window "Material editor"*

## Loads

The new member load and its properties can be entered with the help of this window. The load should be assigned to one of the load duration classes in accordance with table *2.1* of EN 1995-1-1 (program "**Timber**" only). This load duration class affects the value of $k_{mod}$ factor (chapter "**Material characteristics**" of theoretical part of help). If the check box "**Forces calculated acc. to 2nd order theory**", the analysis for this load will be performed without any consideration of buckling (the forces are already calculated on deformed structure.



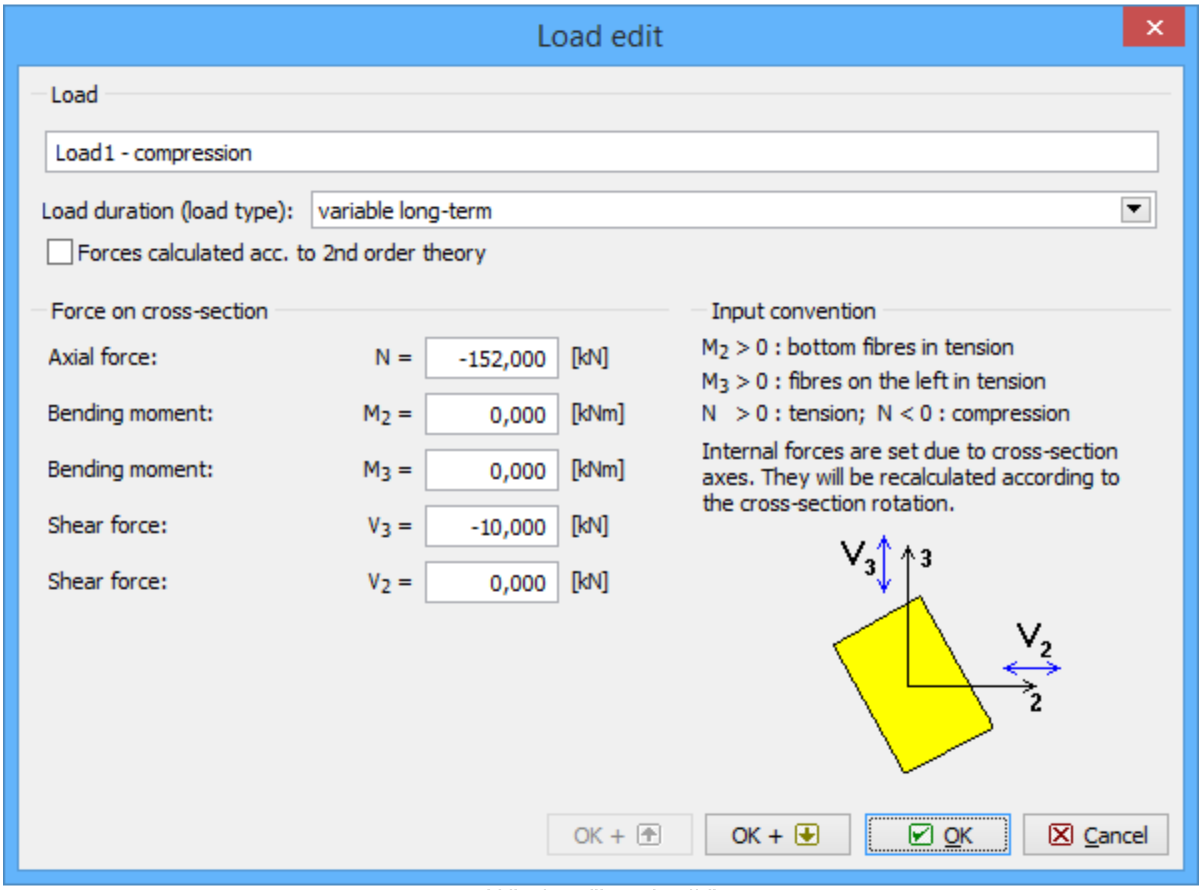

*Window "Load"*

## Load edit

The internal forces and other properties of the load can be entered with the help of this window. These properties can be specified:

| | | |
|---|---|---|
| **Load** | • | Name of the load |
| **Load duration** | • | Load duration class in accordance with tables *2.1* and *2.2* of EN 1995-1-1. The load duration class affects the value of the factor $k_{mod}$. This factor is described in the chapter "**Material characteristics**" of theoretical help. Available only for the program "**Timber**". |
| **N** | • | Normal force |
| **$M_2$** | • | Bending moment about axis *2* (positive values represent tension in the bottom edge of the cross-section) |
| **$M_3$** | • | Bending moment about axis *3* (positive values represent tension in the left edge of the cross-section) |
| **$V_3$** | • | Shear force in vertical direction (parallel with axis *3*) |
| **$V_2$** | • | Shear force in horizontal direction (parallel with axis *2*) |

If the check box "**Forces calculated acc. to 2nd order theory**", the analysis for this load will be performed without any consideration of buckling (the forces are already calculated on deformed structure.




*Window "Load edit"*

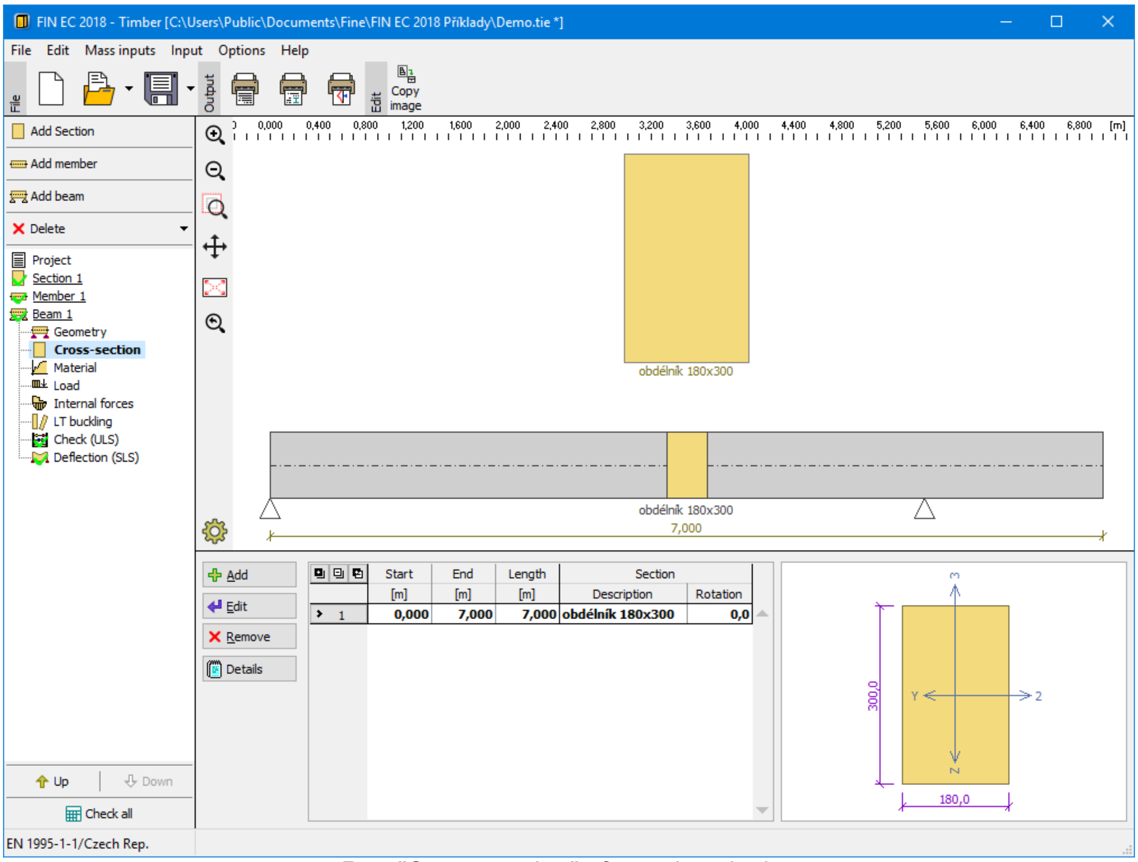
# Cross-section

The member cross-section may be specified in this part of the tree menu. The cross-sectional parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sector may contain different cross-section. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of cross-section are organized in the window "**Cross-section edit**". More sectors can be added (button "**Add**") for input of different fire resistance parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the cross-section geometry will be automatically copied from this program.

The cross-sections properties are also described in the chapter "**Cross-sections**" of the theoretical part of the help.




*Part "Cross-section" of member design*

# Material

The material of the member can be specified in this part. Timber grade can be selected in the window "**Materials catalogue**", which can be launched using "**Catalogue**" button. The material with non-standard values of properties can be specified in the window "**Material editor**" using "**User defined**" button. The strength properties of entered timber are automatically shown in dedicated frame.

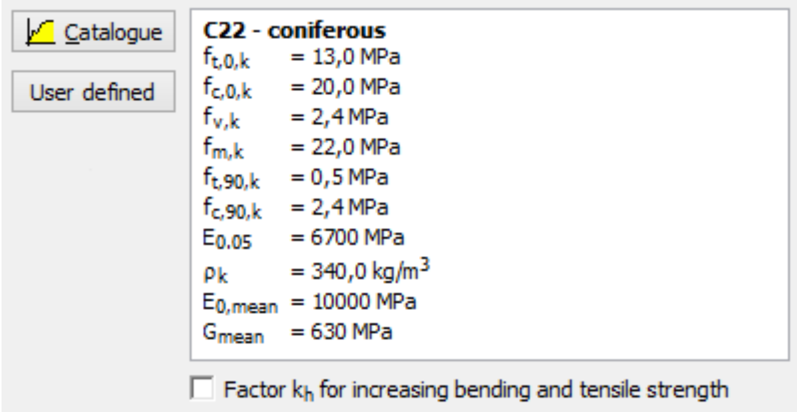
Standard EN 1995-1-2 defines different charring rates for softwood and hardwood in the table *3.1*. Only exception is beech, that should be considered as a softwood in accordance with *3.4.2.(6)*. The radio button "**beech/other hardwood**" is available for *Dxx* grades in these cases.

The timber strengths in tension and bending of members with maximum dimension less than 150mm can be increased in accordance with chapter *3.2* of EN 1995-1-1 using $k_h$ factor. This factor can be applied with the help of check box "**Factor $k_h$ for increasing bending and tensile strength**".

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the material will be automatically copied from this program.

The characteristics are described in the chapter "**Material characteristics**" of the theoretical part of the help.




*Part "Material" of member verification*

# Internal forces

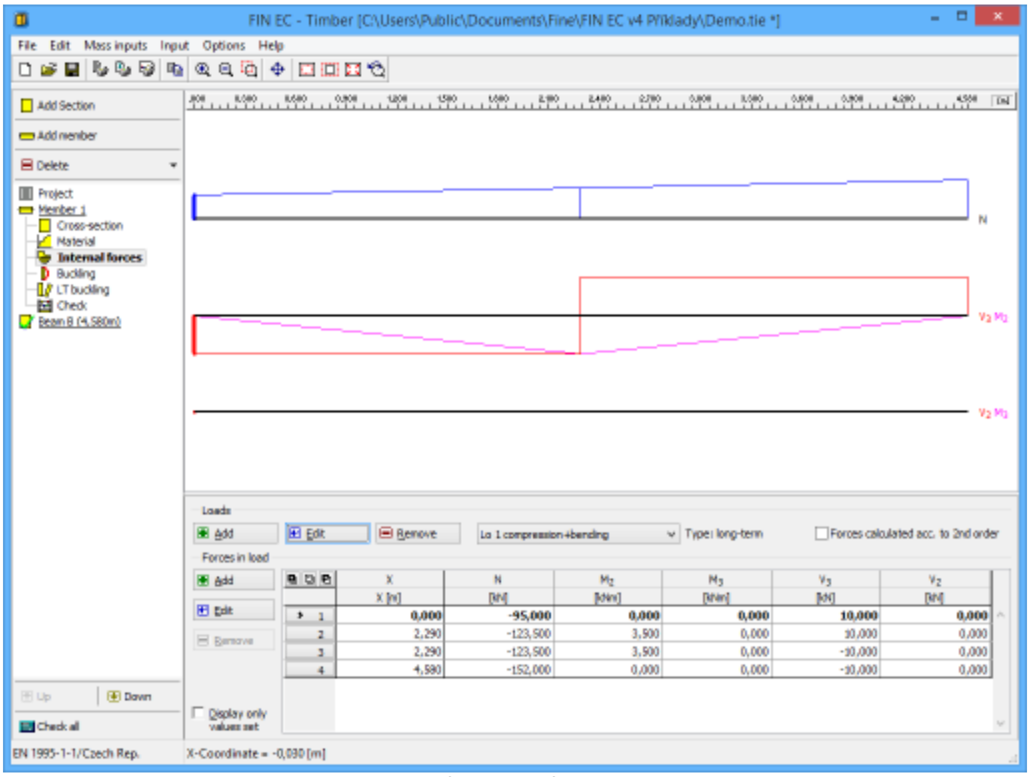
The internal forces along the member length can be specified in this part. More loads (load combinations) can be specified for any member.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the internal forces will be automatically copied from this program.

## Loads

The upper part of the input frame contains buttons for input and edit of loads. The load is a set of internal forces (design values), that corresponds to the results of design combinations. The basic properties of the load can be specified in the window "**Loads**".

Any load contains a setting "**Forces calculated acc. to 2nd order**". The load won't be verified including buckling consideration, if this setting is switched on for the certain load.

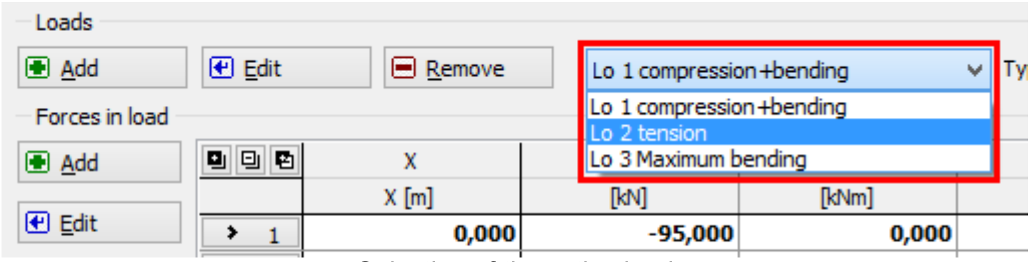



*Part "internal forces" of member design*

## Input of internal forces

The active load has to be selected before starting the input of internal forces. The active load can be selected using list box above the table with values of internal forces.



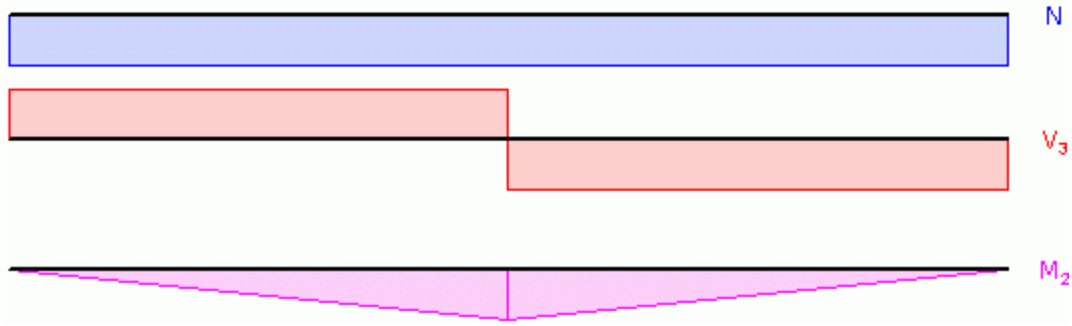
*Selection of the active load*

The internal forces are entered with the help of values in the certain points along the member length. These points should be specified mainly in the positions of local minima or maxima or in the inflection points. Intermediate values are determined automatically using linear interpolation. These points are organized in the table in the bottom part of the application window and can be entered with the help of dedicated window.

The table shows both specified and calculated values of internal forces for each point. The automatically calculated values can be hidden using setting "**Show only entered values**".

## Example of entered internal forces

The following figure shows diagram of normal force *N*, shear force $V_3$ and bending moment $M_2$. The table shows the entered internal forces, that has to be entered in the certain positions.



*Diagrams of internal forces*

These diagrams can be created using these values:

| Section number | Position *x* [m] | N [kN] | $V_3$ [kN] | $M_2$ [kN] |
|---|---|---|---|---|
| 1 | 0,00 | 20,00 | -10,00 | 0,00 |
| 2 | 2,50 | | -10,00 (left) | 25,00 |
| 3 | 2,50 | | 10,00 (right) | |
| 4 | 5,00 | 20,00 | 10,00 | 0,00 |

Intermediate values are calculated automatically using linear interpolation.

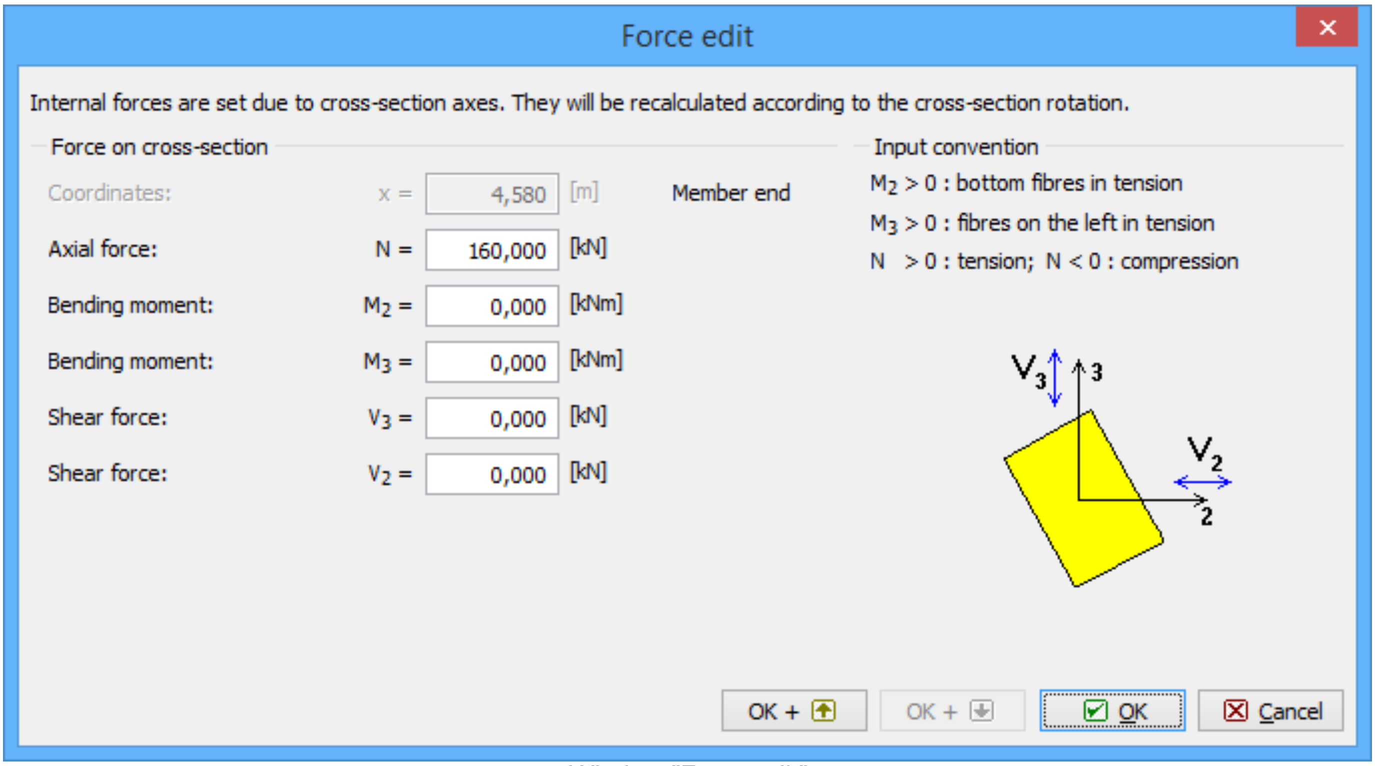
# Forces

The internal forces in the certain member point can be entered with the help of this window. These properties can be specified:

| | |
|---|---|
| **x** | • Basic input, that specifies the position of the point along the member length. The position is measured from the left end of the member |
| **N** | • Normal force |
| $\mathbf{M_2}$ | • Bending moment about axis *2* (positive values represent tension in the bottom edge of the cross-section) |
| $\mathbf{M_3}$ | • Bending moment about axis *3* (positive values represent tension in the left edge of the cross-section) |
| $\mathbf{V_3}$ | • Shear force in vertical direction (parallel with axis *3*) |
| $\mathbf{V_2}$ | • Shear force in horizontal direction (parallel with axis *2*) |

Values of all internal forces shall be specified only in the first (*x=0*) and last (*x* is equal to member length) sections. Intermediate sections may contain unfilled certain internal forces. The values of these forces are obtained with the help of linear interpolation using the closest values that were specified in other sections.




*Window "Force edit"*

## Buckling

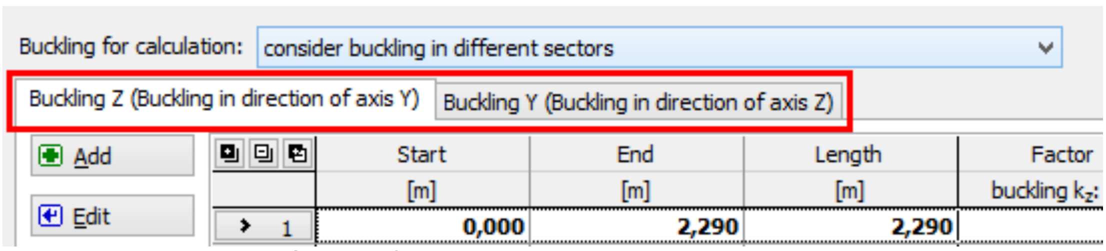
The buckling parameters can be specified in this part of the tree menu. List box "**Buckling for calculation**" contains these three styles of buckling verification:

**Consider buckling in identical sectors**
- The buckling effect will be considered during the analysis, buckling parameters (lengths, end conditions) have to be specified in the same length sectors for both buckling directions.

**Neglect buckling**
- The buckling effect won't be considered during the member analysis. Can be used for members, where the buckling is prevented.

**Consider buckling in different sectors**
- The buckling effect will be considered during the analysis, buckling parameters (lengths, end conditions) are specified in the different length sectors for both buckling directions.

The bucking parameters can be different along the member length. In this case, the member has to be divided into buckling sectors. The buckling sectors can be specified in the table in the bottom part of the main window. The buckling parameters are organized into two tabs "**Buckling Z**" and "**Buckling Y**" for analysis style "**Consider buckling in different sectors**".



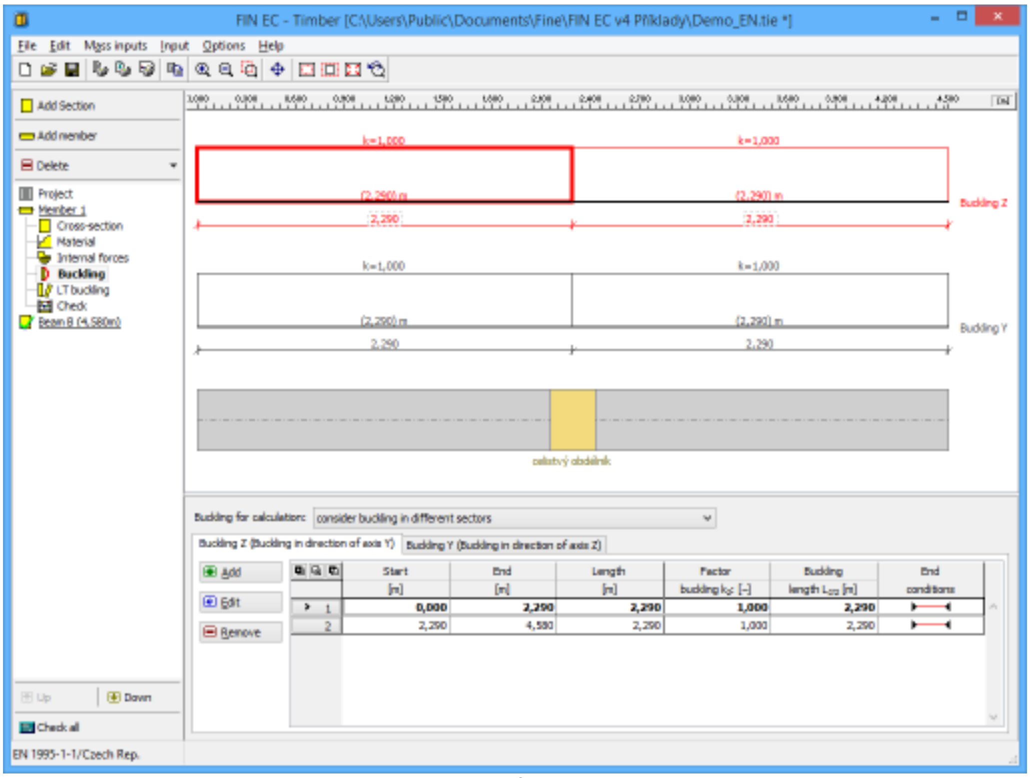

*Choice of buckling directions in separate tabs*

The table contains particular member sectors, every sector may contain different buckling parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The buckling parameters are organized in the window "**Buckling**". More sectors can be added (button "**Add**") for input of different buckling parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

The input is performed in the similar way for style "**Consider buckling in identical sectors**", however only one sectors table is available. The sector properties are organized in the window "**Edit buckling sector**".



*Part "Buckling" of member design*

## Buckling

First part "**Buckling check**" of the window contains basic settings for the certain buckling direction. Buckling verification can be switched of with the help of "**Neglect buckling**" setting. It's also possible to specify different basic sector length for the calculation of buckling length.

The part "**End conditions**" contains several options for specifying the end conditions for buckling length calculation. The appropriate value of factor $k$ is used according to the selected conditions. This factor multiplies the basic sector length for buckling $L_x$. The result of this multiplication is the buckling length $l_{cr}$, that is used in the buckling analysis. These end conditions are available:

 **Not specified**
- The default setting of the program, factor $k$ isn't specified, buckling length $l_{cr}$ isn't calculated. The analysis of buckling cannot be performed for this option.

 **Pinned-pinned**
- Both ends are considered as fixed. The value of factor $k$ is equal to *1.0*. The most common option for timber structures.

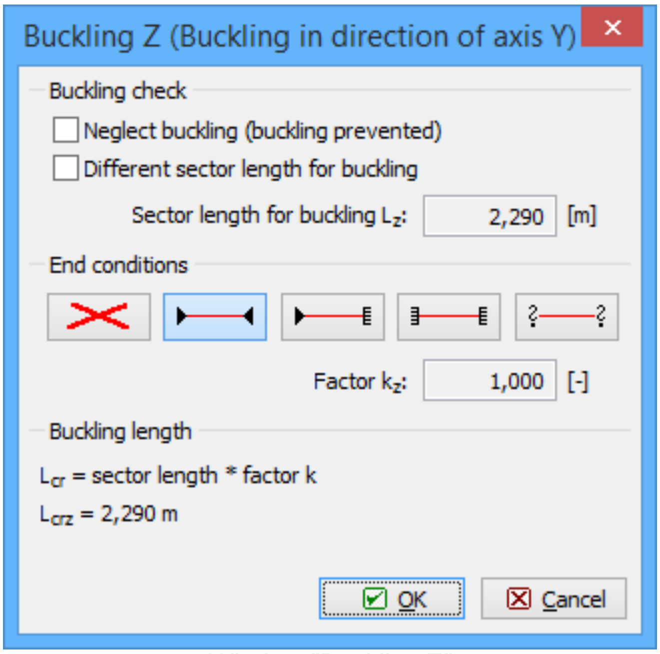
 **Pinned - fixed**
- One end is considered as pinned, another one as fixed. The value of factor $k$ is equal to *0.7*.

 **Fixed-fixed**
- The member is considered as fixed on the both ends. The value of factor $k$ is considered as *0.5*.

 **User input**
- The arbitrary value of factor $k$ can be specified using this option. For example, the value *2.0* can be specified for cantilever.

Part "**Buckling length**" shows final value of buckling length. This value is used as $l_{cr}$ in the stability analysis. This part is described in the chapter "**Buckling**" of the theoretical help.

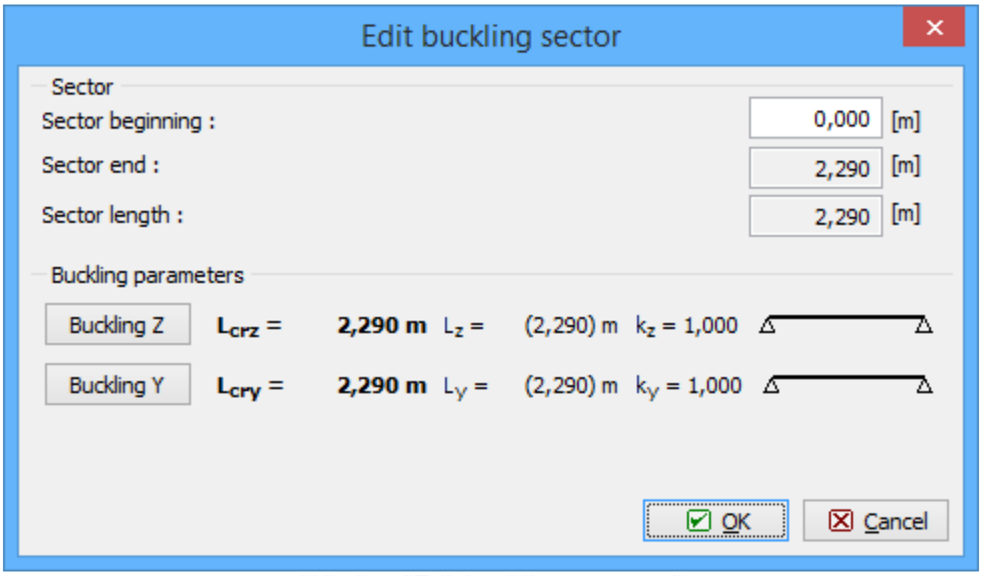


Window "Buckling Z"

## Edit buckling sector

The data necessary for calculation of buckling lengths $l_{cry}$ and $l_{crz}$ for particular sector of a member length can be specified in this window. Basic parameter is "**Sector beginning**", that specifies also the end of previous member sector. This value is measured from the member beginning. The sector end and its length are also displayed.

Part "**Buckling parameters**" contains buttons "**Buckling Z**" and "**Buckling Y**" for input of parameters, that are used for calculation of buckling lengths $l_{cry}$ and $l_{crz}$ for active sector. Parameters are specified in the window "**Buckling**". All these parameters (supporting style, basic length and factor $k$) are displayed also in this window on the right side of the buttons.



Window "Edit buckling segment"

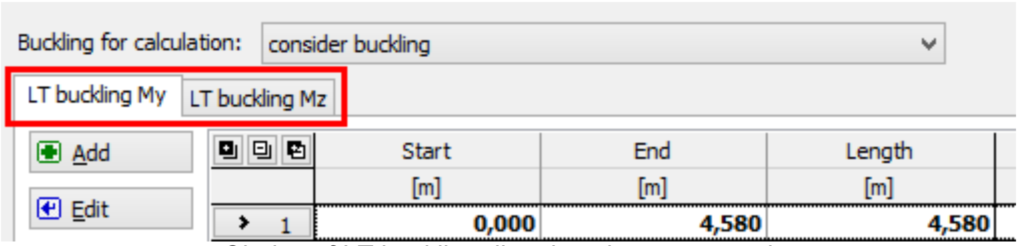
## LT buckling

The lateral torsional buckling parameters can be specified in this part of the tree menu. List box "**LT buckling for calculation**" contains these two styles of LT buckling verification:

| | |
|---|---|
| **Consider buckling** | • The lateral torsional buckling will be considered during the analysis |
| **Do not consider buckling** | • The lateral torsional buckling won't be considered during the analysis. Can be used for members, where the LT buckling is prevented. |

The LT bucking parameters can be different along the member length. In this case, the member has to be divided into particular sectors. The LTB sectors can be specified in the table in the bottom part of the main window. The LTB parameters are organized into two tabs "**LT buckling My**" and "**LT buckling Mz**" according to the buckling direction.

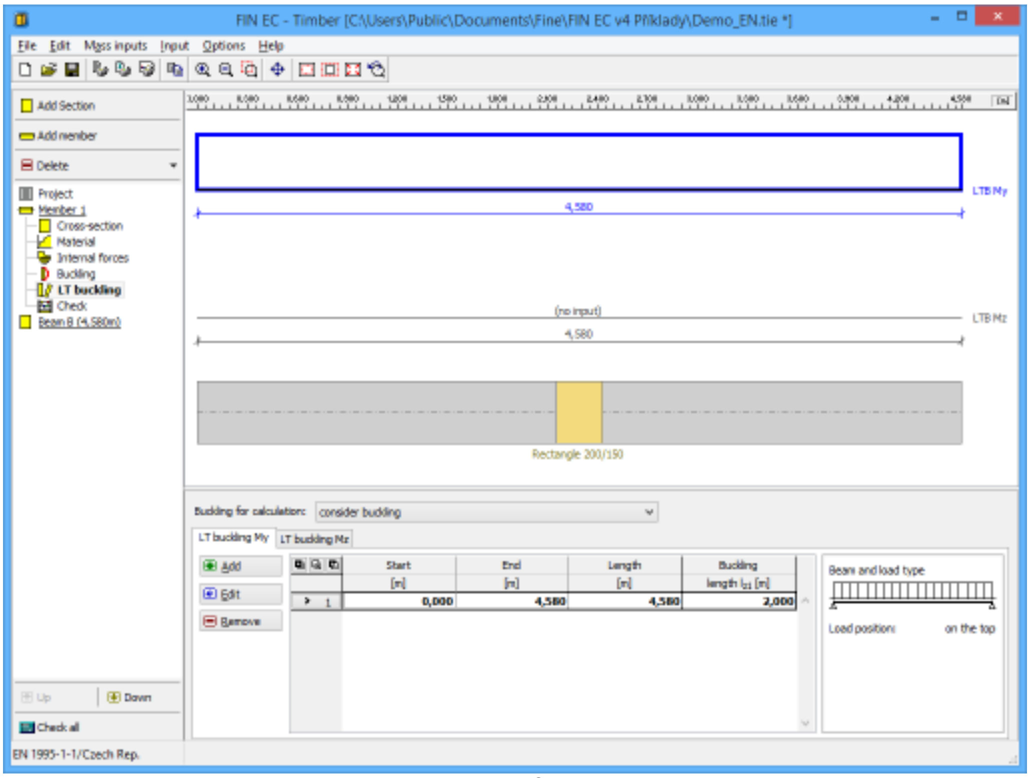



*Choice of LT buckling directions in separate tabs*

The table contains particular member sectors, every sector may contain different LTB parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The buckling parameters are organized in the window "**LT buckling parameters**". More sectors can be added (button "**Add**") for input of different LTB parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

The LTB analysis is described in the chapter "**Lateral torsional buckling**" of the theoretical part of the help.




*Part "LT buckling" of member design*

## LT buckling parameters

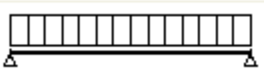
First part "**LTB effect**" of the window contains basic settings for the certain lateral torsional buckling direction. LT buckling verification can be switched of with the help of "**Neglect LTB**" setting. It's also possible to specify different basic sector length $l_1$ for the calculation of LT buckling length.

The part "**Beam and load type**" contains several options for specifying the analysis parameters in accordance with table *6.1* of EN 1995-1-1 (end conditions and load position). The value of ratio between the effective length $l_{ef}$ and the span $l$ is selected according to these inputs. These options are available:



- The default setting of the program, end conditions and load type aren't specified, effective length $l_{ef}$ isn't calculated. The analysis of buckling cannot be performed for this option.

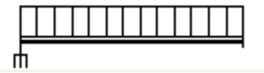
- Simply supported beam with constant moment. Ratio $l_{ef}/l$ is *1.0*.



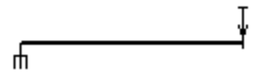
- Simply supported beam with uniformly distributed load. Ratio $l_{ef}/l$ is *0.9*.

- Simply supported beam with concentrated force at the middle of the span. Ratio $l_{ef}/l$ is *0.8*.



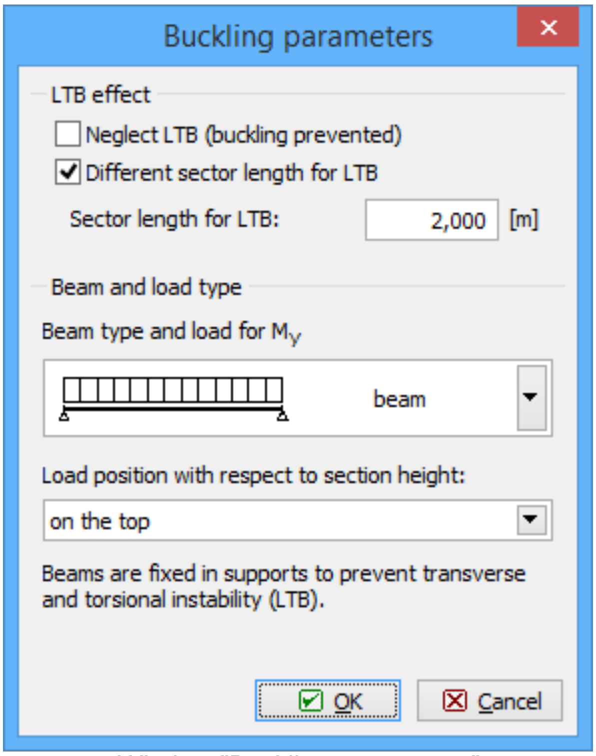
- Cantilever with uniformly distributed load. Ratio $l_{ef}/l$ is *0.5*.



- Cantilever with concentrated force at the free span. Ratio $l_{ef}/l$ is *0,8*.

The load position with respect to the centre of gravity ("**on the top**", "**in the middle**", "**at the bottom**") has to be specified in the most of cases. These options respect the note in table *6.1*. The effective length $l_{ef}$ is increased by *2h* for the load applied at the compression edge and decreased by *0,5h* for the load applied at the tension edge of the beam.

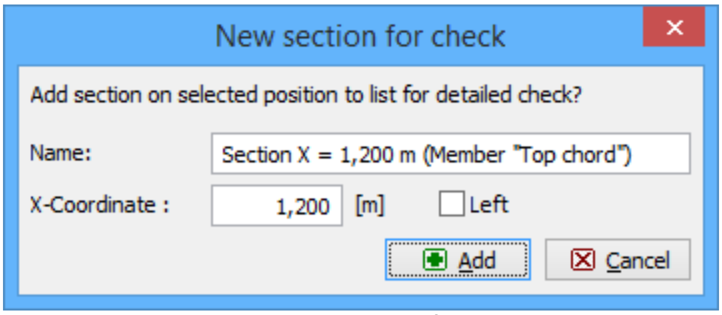
The values specified in the table *6.1* are valid only providing that the torsional rotation is prevented at member supports.




*Window "Buckling parameters"*

## Edit section for check

The detailed results in certain point can be displayed with the help of verification sections. These sections can be added or modified using window **"Edit section for check**". This window contains input lines for specification of the name and the section position (measured from the member beginning). Check box "**Left**" can be used in the points of discontinuity. The results on the left side of this point will be displayed if the check box is switched on.




*Window "Edit section for check"*

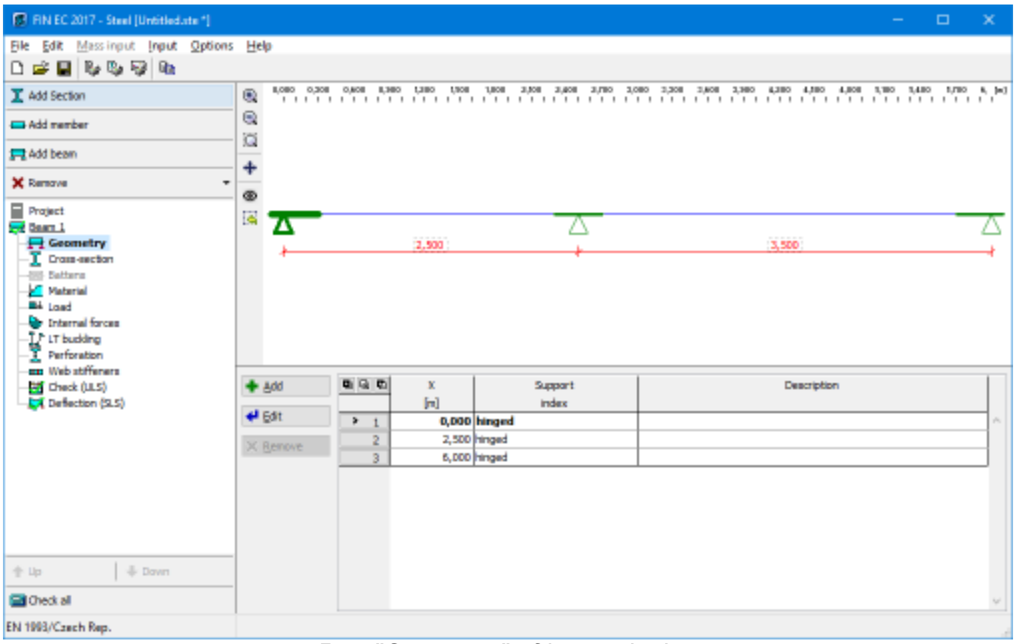
## Beams

## Geometry

This part contains tools for input of nodes into the structure. Nodes like different types of supports, middle hinges and calculation nodes (points with detailed results) can be added with the help of the table in the input frame. Node properties are organized in the window "**Edit** " that can be launched using buttons "**Add**" and "**Edit**".

The workspace contains active dimensions that are able to change spans and support widths without launching appropriate window.



*Part "Geometry" of beam design*

## Edit member node

Properties of node (support) can be defined in this window. Basic parameter is the position of the node (marked as "**X-coordinate**" in the window) that is measured from the beginning of the member. These node types are supported:

**Calculation node** • Nodes that show exact values of forces and moments in the part "**Internal forces**" and also values of deflection in the part "**Deflection (SLS)**" (programs "**Steel**" and "**Timber**"). Calculation nodes influence neither topology neither internal forces.

**Hinged** • Support fixed in vertical direction, free in rotation

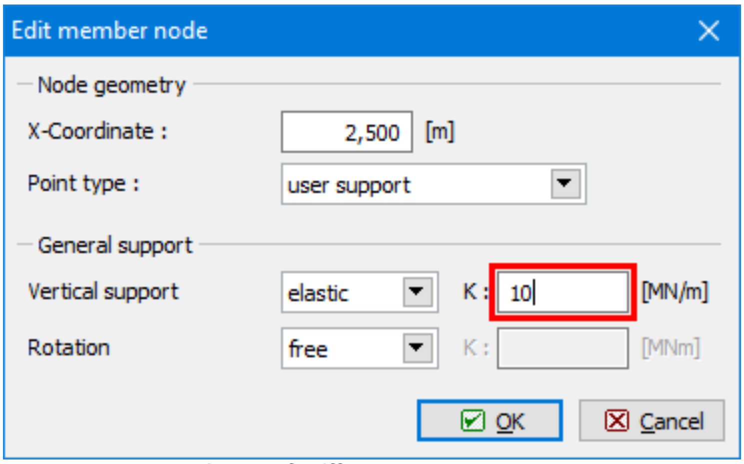
**Fixed** • Support fixed both in vertical direction and in rotation

**User support** • Support with possibility to define the stiffness both in vertical direction and in rotation

**Middle hinge** • Middle hinge in a beam bay. Only shear forces are transported in this point, bending moment is equal to *0*.

### General support

This part contains stiffness characteristics for node type "**User defined support**". The stiffness is defined by spring constants $K$ .




*Input of stiffness parameters*

## Load

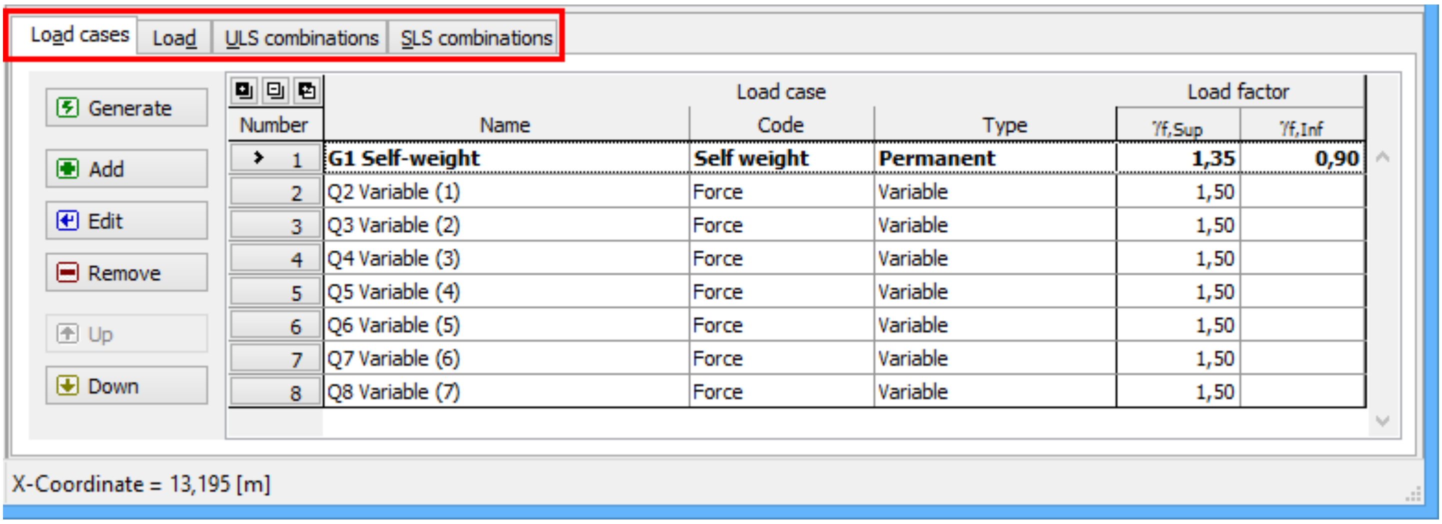
Part "**Load**" contains tools for input of load cases and combination including final diagrams of internal forces. The input is divided into four parts (tabs): "**Load cases**", "**Load**", "**ULS combinations**" a "**SLS combinations**".

Load cases, loads and combinations can be added manually or can be created automatically wth the help of buttons

"**Generate**". This automatic input simplifies the work and is suitable for the most of applications.

Any changes in this part cause removal of results and internal forces. New internal forces are calculated in accordance with the tables of combinations for ULS and SLS.




| Number | Load case Name | Code | Type | Load factor $\gamma_{f,Sup}$ | $\gamma_{f,Inf}$ |
|---|---|---|---|---|---|
| 1 | G1 Self-weight | Self weight | Permanent | 1,35 | 0,90 |
| 2 | Q2 Variable (1) | Force | Variable | 1,50 | |
| 3 | Q3 Variable (2) | Force | Variable | 1,50 | |
| 4 | Q4 Variable (3) | Force | Variable | 1,50 | |
| 5 | Q5 Variable (4) | Force | Variable | 1,50 | |
| 6 | Q6 Variable (5) | Force | Variable | 1,50 | |
| 7 | Q7 Variable (6) | Force | Variable | 1,50 | |
| 8 | Q8 Variable (7) | Force | Variable | 1,50 | |

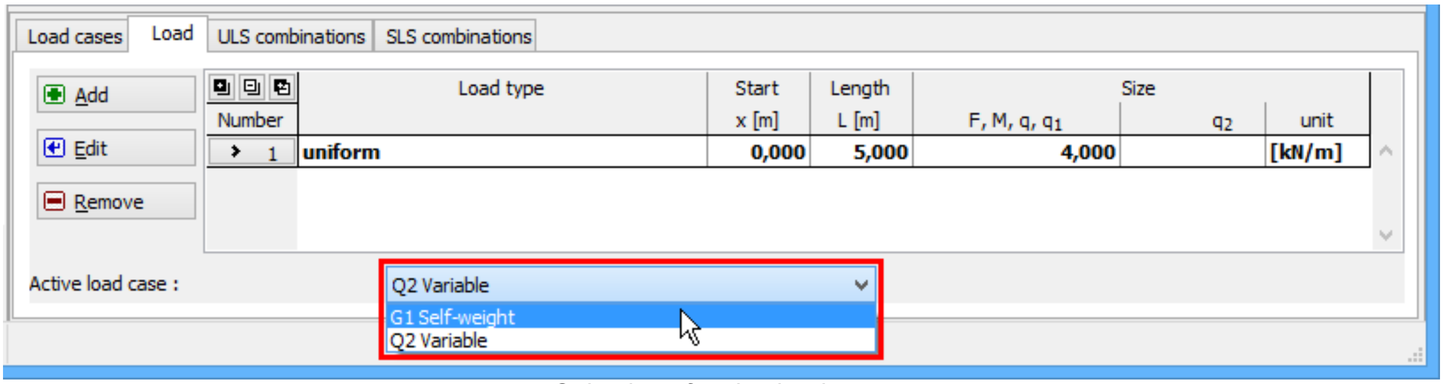
*Tabs with input tables*

## Load cases

Load case is a group of load with identical nature. Load cases can be included in the unlimited number of combinations. New load cases can be created automatically in window "**Load case generator**" using the button "**Generate**" or can be added manually using button "**Add**". The properties of the load case are organized in the window "**Load case**" in this case.

## Load

This part is dedicated for the input of loads (continuous and point loads, bending moments) into active load case. Active load case can be selected in the drop-down menu above the table.




| Number | Load type | Start x [m] | Length L [m] | Size F, M, q, $q_1$ | $q_2$ | unit |
|---|---|---|---|---|---|---|
| 1 | uniform | 0,000 | 5,000 | 4,000 | | [kN/m] |

*Selection of active load case*

New load can be added using the button "**Add**". Load properties (type, position, value) are organized in the window "**Load edit**". Load case "**Self-weight**" is updated automatically and it isn't possible to edit existing loads or add new ones.

Automatically generated load cases already contain loads according to the input in the window "**Load case generator**".

## ULS combinations

These combinations are suitable for the analysis of states associated with collapse or with other similar forms of structural failure. These combinations are used for the verification of longitudinal and shear reinforcement. The design values of loads are used during the analysis. New combinations can be added manually with the help of the button "**Add**" (launches the window "**Combination**") or created automatically in "**Generator of combinations**". Entered combinations can be checked in the "**Table of combinations**" that shows well arranged overview of all combinations

## SLS combinations

These combinations are suitable for the analysis of states that correspond to conditions beyond which specified service requirements for astructure or structural member are no longer met. These combinations are used for analysis of deflection, cracks and stresses. Input rules are identical.

# Load case generator

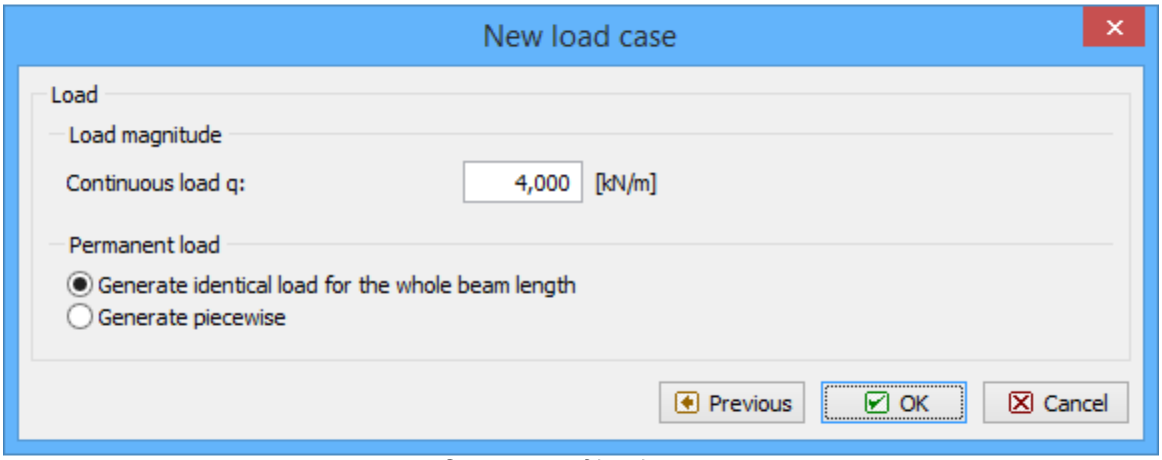
This tool is able to add a new load case including load very easily. The load generation is done with respect of member geometry. More load cases are created for continuous beams with more supports. These load cases ensure that maximum values of bending moments and shear forces were found for any part of the member. Input is divided into two parts. It's possible to switch from one part to another one with the help of buttons "**Next**" and "**Previous**":

### Load case

Main parameters of the load case can be specified here (name, type, load duration, combination factors). These properties are described in the chapter "**Load case edit**".

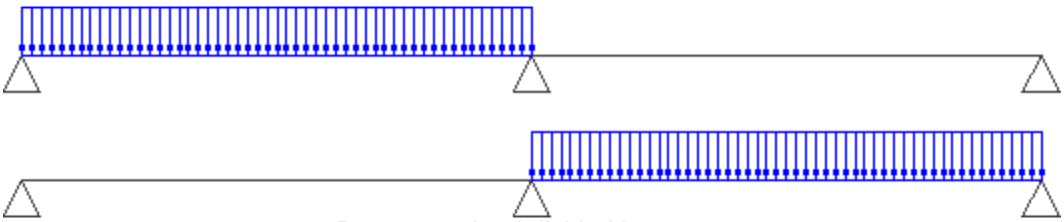
### Load magnitude

The value of the load can be specified in this part. Positive value of the load means that the load acts in the gravity direction.



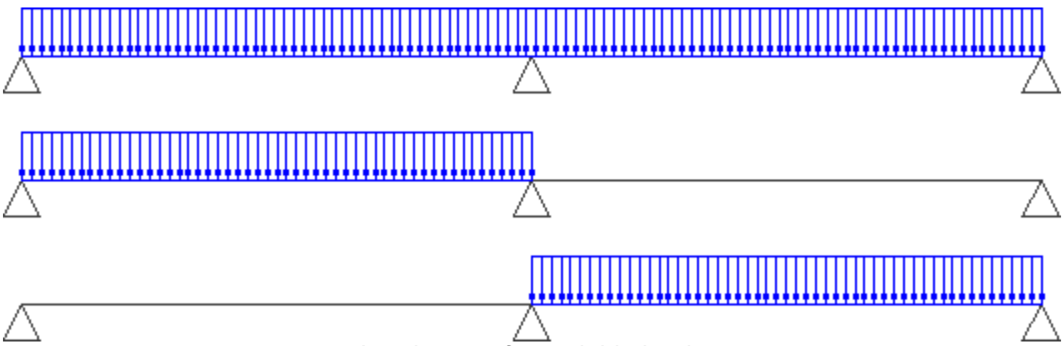

*Generator of load cases*

Permanent loads can be divided into more parts according to the member bays for the setting "**Generate piecewise**". The number of created load cases is equal to the number of bays. Load just for one bay is included in any load case. The distinction between favourable and unfavourable effects ofpermanent loads can be made with the help of this setting.



*Permanent load divided into parts*

More load cases are created automatically for variable load to ensure that maximum values of the bending moment and shear force will be achieved in any bay of the member.

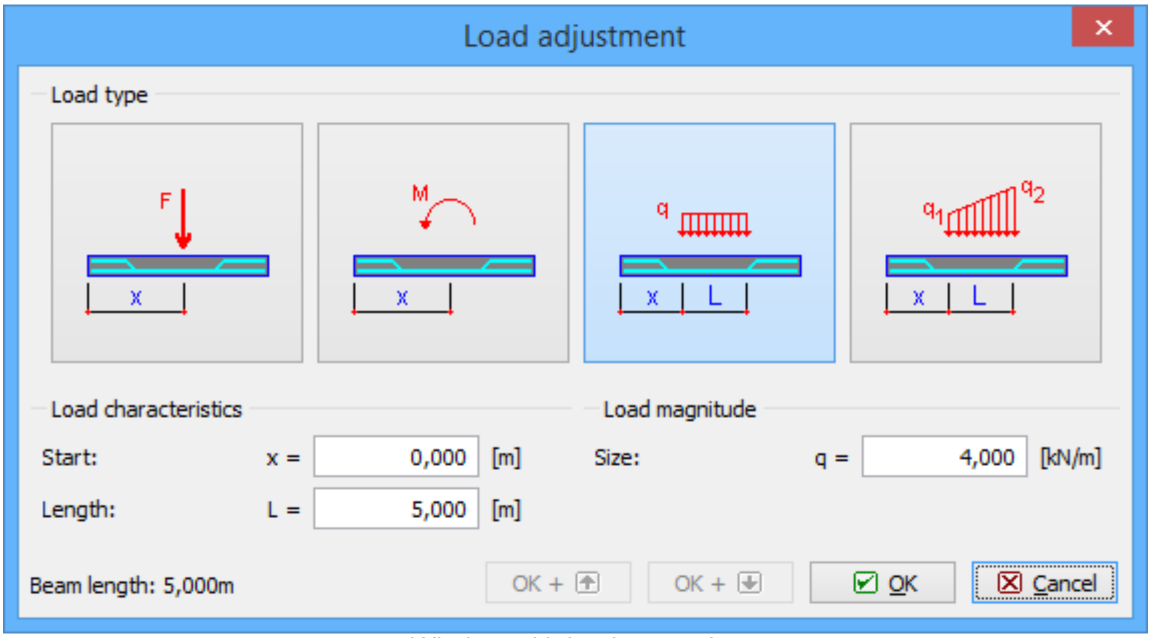


*Load cases for variable loads*

## Load edit

The load properties can be specified in this window. The load type can be selected with the help of buttons in the upper part of the window. There are four available load types: force, bending moment, linear load, trapezoid load. The position is specified by the distance from the beam start *x*.




*Window with load properties*

# Xml

# General rules for XML import

## General rules XML

The following rules shall be respected when writing XML file:

- All elements have to be bounded by a root element
- The elements must have opening and closing tags. These tags can be merged into one for an empty element.
- All elements have to be properly nested
- Attributes have to be marked by characters (') or (").
- The names of elements are case sensitive

# XML import for programs "Fin 2D" and "Fin 3D"

The file format for import of structures is identical for programs **"Fin 2D**" and **"Fin 3D**". Both general rules for XML files and following rules given for this import should be respected:

- The name of fundamental element is not limited (for example <FINE>)
- All values are defined using base SI units
- Angles are defined in degrees

## File structure

These parts can be included in the XML file:

| | |
|---|---|
| **<header>** | • Optional chapter with project properties (design standard, |
| **<coordinate Systems>** | • Optional part with local coordinate systems of joints |
| **<supports>** | • The part which contains types of supports or member connections |
| **<materials>** | • The part which contains list of materials. Optional |
| **<sections>** | • Optional part with cross- sections |
| **<linetypes>** | • The part with member profiles. Optional |
| **<points>** | • Obligatory part with joints and their properties |
| **<lines>** | • Obligatory part with member properties (reference joints etc.) |
| **<loadcases>** | • The part with load cases and loads |
| **<combinations>** | • Optional part which contains load combinations |

The file may contain also parameters for verification of steel and concrete members:

| | |
|---|---|
| **<dimSteel>** | • The part with analysis parameters for steel members. Optional |
| **<dimConcrete>** | • The part with analysis parameters for RC members. Optional |

```
<FINE>
  <header/>
  <coordinateSystems/>
  <supports/>
  <materials/>
  <sections/>
  <linetypes/>
  <points/>
  <lines/>
  <loadcases/>
  <combinations/>
  <dimSteel/>
  <dimConcrete/>
  <dimTimber/>
</FINE>
```

*Structure of XML file*

## <header>

This element is an optional part of XML file. It contains fundamental project information. Following attributes are supported:

| | |
|---|---|
| **name** | • Project name (optional) |
| **company** | • Company name (optional) |
| **description** | • Description (optional) |
| **investor** | • investor (optional) |
| **standard** | • The design standard. This setting affects mainly parameters of load cases and combinations. This attribute is optional. The default value is "**ec**" (Eurocode). This attribute supports following values: "**ec**" (Eurocode), "**csn**" (ČSN 73 0035), "**stn**" (STN 73 0035), "**sans**" (SANS 10160), "**other**" (general standard with an option to specify partial load and combination factors manually) |
| **annex** | • The national annex for the standard "**ec**". Supported values: "**none**" (no national annex), "**czech**" (Czechia), "**slovak**" (Slovakia), "**poland**" (Poland), "**france**" (France) |

```
<FINE>
  <header name="Projekt 01" company="Fine s.r.o."
description="Testovací projekt" investor="Investor" standard="ec"
annex="czech"/>
  <coordinateSystems/>
```

*Elelement <header> with project information*

## <coordinateSystems>

This part contains properties of local coordinate systems which may be used for definition of support rotation. It is an optional part of XML file. Every coordinate system is marked by the element **<coordinateSystem>**. This element supports following attributes:

| | |
|---|---|
| **id** | • unique joint ID (obligatory) |
| **x_x** | • $x$ -component of the vector which defines the axis $x$ of local coordinate system, obligatory |
| **x_y** | • $y$ -component of the vector which defines the axis $x$ of local coordinate system, obligatory |
| **x_z** | • $z$ -component of the vector which defines the axis $x$ of local coordinate system, obligatory |
| **xy_x** | • $x$ -component of the plane which is defined by the axes $x$ and $y$ , obligatory |
| **xy_y** | • $y$ -component of the plane which is defined by the axes $x$ and $y$ , obligatory |
| **xy_z** | • $z$ -component of the plane which is defined by the axes $x$ and $y$ , obligatory |

```
  <header name="Projekt 01" company="Fine s.r.o." description="Testovací
projekt" investor="Investor" standard="ec" annex="czech"/>
  <coordinateSystems>
    <coordinateSystem id="1" x_x="0,5" x_y="0,5" x_z="0" xy_x="5"
xy_y="0" xy_z="0" />
    <coordinateSystem id="2" x_x="-0,5" x_y="0,5" x_z="0" xy_x="-5"
xy_y="0" xy_z="0" />
  </coordinateSystems>
  <supports/>
```

*Part <coordinateSystems> with two coordinate systems*

## <supports>

This part contains properties of support types. It is an optional part of XML file. Every support type is marked by the element **<support>**. This element supports following attributes:

| | |
|---|---|
| **id** | • Unique joint ID (obligatory) |
| **x** | • The support type in the direction of the axis $x$ , optional |
| **y** | • The support type in the direction of the axis $y$ , optional |
| **z** | • The support type in the direction of the axis $z$ , optional |
| **mx** | • The support type for the rotation about the axis $x$ , optional |
| **my** | • The support type for the rotation about the axis $y$ , optional |
| **mz** | • The support type for the rotation about the axis $z$ , optional |

The support type in the given direction can be defined using following values:

| | |
|---|---|
| "**free**" | • Free support |
| "**fixed**" | • Fixed support |
| **number** | • Spring support, the number defines the s tiffness of the support in ***[MN/m]*** (shift) or ***[MNm]*** (rotation). |

The default value "**free**" is used for directions which are not specified in the XML file.

```
</coordinateSystems>
<supports>
  <support id="1" x="free" y="fixed" z="fixed" mx="fixed"/>
  <support id="2" x="free" y="67.8" z="fixed" mx="fixed"/>
</supports>
<materials/>
```

*Part <supports> of XML file*

## <materials>

This part contains properties of materials. It is an optional part of XML file. Every material is marked by the element **<material>**. This element supports following attributes:

| | |
|---|---|
| **id** | • Unique material ID (obligatory) |
| **type** | • The material typy, obligatory. The material can be selected from the database (values of material type are listed in the tables below) or can be defined manually (the value "**numerically**" has to be used for this attribute). In this case, the properties of the material have to be specified manually. |
| **name** | • Material name, obligatory |
| **standard** | • The name of corresponding standard, optional |
| **E** | • The modulus of elasticity in ***[Pa]*** , obligatory |
| **G** | • The shear modulus in ***[Pa]*** , obligatory |
| **alpha_t** | • The coefficient of thermal expansion defined in $[K^{-1}]$ , obligatory |
| **gama** | • The specific weight defined in $[N/m^3]$ , obligatory |

The material can be defined using two different ways:

- Choice from materials catalogue. Only attributes "**id**" and "**type**" are required in this case.
- Input of material using manual input. The value "**numerically**" has to be used for the attribute "**type**" in this case. Additionally, it is necessary to specify the modulus of elasticity (attribute "**E**"), the shear modulus (attribute "**G**"), the coefficient of thermal expansion (attribute "**alpha_t**") and the specific weight (attribute "**gama**").

Following tables contain list of materials in the internal database including corresponding values of the attribute "**type**":

## Steel

| Material | Value for XML |
|---|---|
| EN 10025: Fe 360 | SS_EC3_EN10025_Fe360 |
| EN 10025: Fe 430 | SS_EC3_EN10025_Fe430 |
| EN 10025: Fe 510 | SS_EC3_EN10025_Fe510 |
| prEN 10113: Fe E 275 | SS_EC3_prEN10113_FeE275 |
| prEN 10113: Fe E 355 | SS_EC3_prEN10113_FeE355 |
| EN 10210-1: S 235 | SS_EC3_EN10210_S235 |
| EN 10210-1: S 275 | SS_EC3_EN10210_S275 |
| EN 10210-1: S 355 | SS_EC3_EN10210_S355 |

## Stainless steel

| Material | Value for XML | Material | Value for XML |
|---|---|---|---|
| Stainless steel 1.4003 | SS_SL_1_4003 | Stainless steel 1.4432 | SS_SL_1_4432 |
| Stainless steel 1.4016 | SS_SL_1_4016 | Stainless steel 1.4435 | SS_SL_1_4435 |
| Stainless steel 1.4512 | SS_SL_1_4512 | Stainless steel 1.4311 | SS_SL_1_4311 |
| Stainless steel 1.4306 | SS_SL_1_4306 | Stainless steel 1.4406 | SS_SL_1_4406 |
| Stainless steel 1.4307 | SS_SL_1_4307 | Stainless steel 1.4439 | SS_SL_1_4439 |
| Stainless steel 1.4541 | SS_SL_1_4541 | Stainless steel 1.4529 | SS_SL_1_4529 |
| Stainless steel 1.4301 | SS_SL_1_4301 | Stainless steel 1.4547 | SS_SL_1_4547 |
| Stainless steel 1.4401 | SS_SL_1_4401 | Stainless steel 1.4318 | SS_SL_1_4318 |
| Stainless steel 1.4404 | SS_SL_1_4404 | Stainless steel 1.4362 | SS_SL_1_4362 |
| Stainless steel 1.4539 | SS_SL_1_4539 | Stainless steel 1.4462 | SS_SL_1_4462 |
| Stainless steel 1.4571 | SS_SL_1_4571 | | |

## Timber

| Material | Value for XML | Material | Value for XML |
|---|---|---|---|
| C14 - softwood | WO_EN_C14 | C40 - softwood | WO_EN_C40 |
| C16 - softwood | WO_EN_C16 | C45 - softwood | WO_EN_C45 |
| C18 - softwood | WO_EN_C18 | C50 - softwood | WO_EN_C50 |
| C20 - softwood | WO_EN_C20 | D30 - hardwood | WO_EN_D30 |
| C22 - softwood | WO_EN_C22 | D35 - hardwood | WO_EN_D35 |
| C24 - softwood | WO_EN_C24 | D40 - hardwood | WO_EN_D40 |
| C27 - softwood | WO_EN_C27 | D50 - hardwood | WO_EN_D50 |
| C30 - softwood | WO_EN_C30 | D60 - hardwood | WO_EN_D60 |
| C35 - softwood | WO_EN_C35 | D70 - hardwood | WO_EN_D70 |

## Timber, Czechia

| Material | Value for XML |
|---|---|
| S13 (C30) - softwood | WO_EN_CZ_S13 |
| S10 (C24) - softwood | WO_EN_CZ_S10 |
| S7 (C18) - softwood - spruce, pine | WO_EN_CZ_S7 |
| S7 (C16) - softwood - fir, larch | WO_EN_CZ_S7 |

## Timber, Slovakia

| Material | Value for XML |
|---|---|
| S0 (C30) - softwood | WO_EN_CZ_S13 |
| SI (C24) - softwood | WO_EN_CZ_S10 |
| SII (C16) - softwood | WO_EN_CZ_S7 |

## Concrete

| Material | Value for XML | Material | Value for XML |
|---|---|---|---|
| C12/15 | CO_EC2_C12 | C45/55 | CO_EC2_C45 |
| C16/20 | CO_EC2_C16 | C50/60 | CO_EC2_C50 |
| C20/25 | CO_EC2_C20 | C55/67 | CO_EC2_C55 |
| C25/30 | CO_EC2_C25 | C60/75 | CO_EC2_C60 |
| C30/37 | CO_EC2_C30 | C70/85 | CO_EC2_C70 |

| | | | |
|---|---|---|---|
| C35/45 | CO_EC2_C35 | C80/95 | CO_EC2_C80 |
| C40/50 | CO_EC2_C40 | C90/105 | CO_EC2_C90 |

## Plastics

| Material | Value for XML |
|---|---|
| Vinyl | PL_VINYL |

## Wood-based materials

| Material | Value for XML | Material | Value for XML |
|---|---|---|---|
| OSB/2 (6-10)mm parallel | WB_OSB2_R_6_10_mm | Practicleboard P4 (25-32)mm | WB_PB_P4_25_32_mm |
| OSB/2 (6-10)mm perpendicular | WB_OSB2_K_6_10_mm | Practicleboard P4 (32-40)mm | WB_PB_P4_32_40_mm |
| OSB/2 (10-18)mm parallel | WB_OSB2_R_10_18_mm | Practicleboard P4 (40-50)mm | WB_PB_P4_40_50_mm |
| OSB/2 (10-18)mm perpendicular | WB_OSB2_K_10_18_mm | Practicleboard P5 (6-13)mm | WB_PB_P5_6_13_mm |
| OSB/2 (18-25)mm parallel | WB_OSB2_R_18_25_mm | Practicleboard P5 (13-20)mm | WB_PB_P5_13_20_mm |
| OSB/2 (18-25)mm perpendicular | WB_OSB2_K_18_25_mm | Practicleboard P5 (20-25)mm | WB_PB_P5_20_25_mm |
| OSB/3(6-10)mm parallel | WB_OSB3_R_6_10_mm | Practicleboard P5 (25-32)mm | WB_PB_P5_25_32_mm |
| OSB/3 (6-10)mm perpendicular | WB_OSB3_K_6_10_mm | Practicleboard P5 (32-40)mm | WB_PB_P5_32_40_mm |
| OSB/3 (10-18)mm parallel | WB_OSB3_R_10_18_mm | Practicleboard P5 (40-50)mm | WB_PB_P5_40_50_mm |
| OSB/3 (10-18)mm perpendicular | WB_OSB3_K_10_18_mm | Practicleboard P6 (6-13)mm | WB_PB_P6_6_13_mm |
| OSB/3 (18-25)mm parallel | WB_OSB3_R_18_25_mm | Practicleboard P6 (13-20)mm | WB_PB_P6_13_20_mm |
| OSB/3 (18-25)mm perpendicular | WB_OSB3_K_18_25_mm | Practicleboard P6 (20-25)mm | WB_PB_P6_20_25_mm |
| OSB/4 (6-10)mm parallel | WB_OSB4_R_6_10_mm | Practicleboard P6 (25-32)mm | WB_PB_P6_25_32_mm |
| OSB/4 (6-10)mm perpendicular | WB_OSB4_K_6_10_mm | Practicleboard P6 (32-40)mm | WB_PB_P6_32_40_mm |
| OSB/4 (10-18)mm parallel | WB_OSB4_R_10_18_mm | Practicleboard P6 (40-50)mm | WB_PB_P6_40_50_mm |
| OSB/4 (10-18)mm perpendicular | WB_OSB4_K_10_18_mm | Practicleboard P7 (6-13)mm | WB_PB_P7_6_13_mm |
| OSB/4 (18-25)mm parallel | WB_OSB4_R_18_25_mm | Practicleboard P7 (13-20)mm | WB_PB_P7_13_20_mm |
| OSB/4 (18-25)mm perpendicular | WB_OSB4_K_18_25_mm | Practicleboard P7 (20-25)mm | WB_PB_P7_20_25_mm |
| Practicleboard P4 (6-13)mm | WB_PB_P4_6_13_mm | Practicleboard P7 (25-32)mm | WB_PB_P7_25_32_mm |
| Practicleboard P4 (13-20)mm | WB_PB_P4_13_20_mm | Practicleboard P7 (32-40)mm | WB_PB_P7_32_40_mm |
| Practicleboard P4 (20-25)mm | WB_PB_P4_20_25_mm | Practicleboard P7 (40-50)mm | WB_PB_P7_40_50_mm |

## Masonry

| Material | Value for XML |
|---|---|
| Tile masonry (hollow bricks) | MA_EC6_BurntHollow |
| Brickwork masonry E=2GPa | MA_EC6_Burnt_2 |
| Brickwork masonry E=4GPa | MA_EC6_Burnt_4 |
| Brickwork masonry E=6GPa | MA_EC6_Burnt_6 |
| Pore-concrete masonry E=1GPa | MA_EC6_Porous_1 |
| Pore-concrete masonry E=2GPa | MA_EC6_Porous_2 |
| Stonework masonry | MA_EC6_Stone |

```
    </supports>
    <materials>
      <material id="1" type="SS_EC3_EN10210_S275" />
      <material id="2" type="NUMERICALY" name="Nový materiál" E="210e9"
G="81e9" alpha_t="12e-6" gama="78500"/>
    </materials>
    <sections/>
```

*Part <materials> of XML file*

## <sections>

This part contains properties of cross-section. It is an optional part of XML file. Every cross-section is marked by the element **<section>**. This element supports following attributes:

| | |
|---|---|
| **id** | • Unique ID, obligatory |
| **type** | • The cross-section type, obligatory. Available values are listed in the table below. |
| **name** | • The cross-section name. Obligatory for hot-rolled cross-sections (the name corresponds to the name in the database). |
| **material** | • The material ID, obligatory |

The range of available attributes differ cross-section to cross-section.

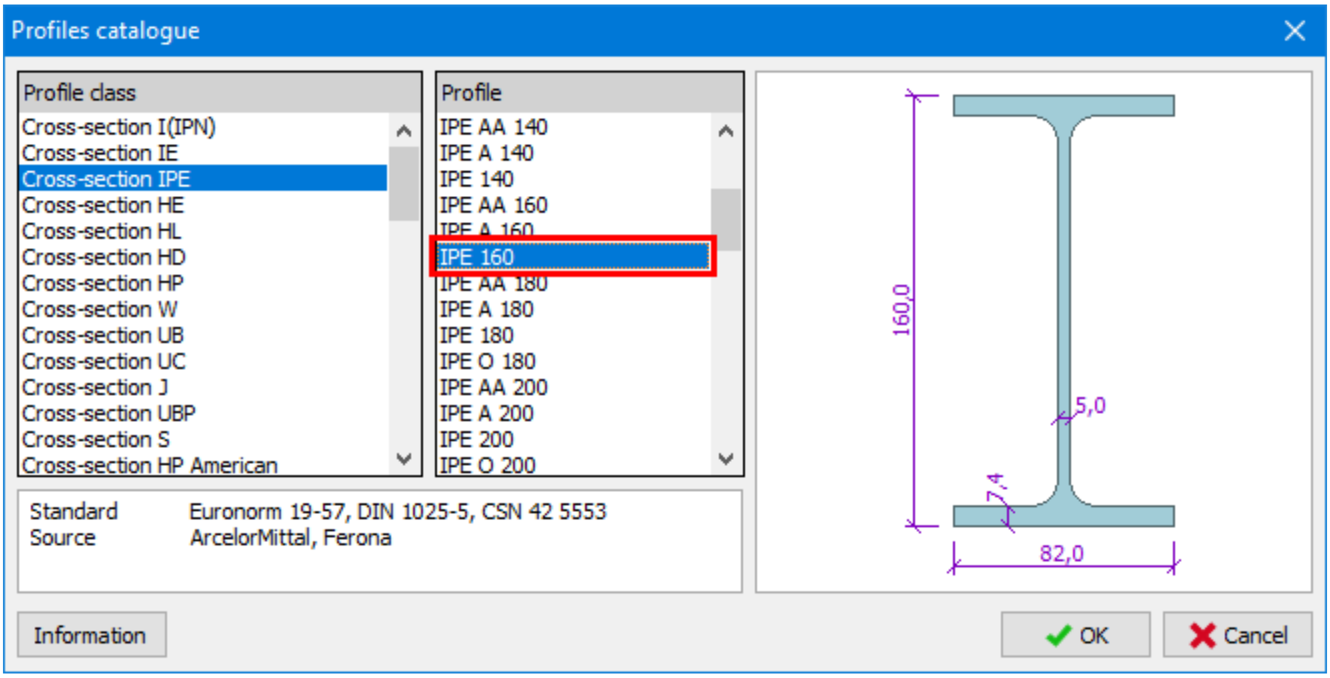
### Steel

Hot-rolled cross-sections are defined using the attribute "**type**" (group of cross-sections) and "**name**" (the cross-section within the group). Available values for the attribute "**type**":

| The group name | Value for "type" | The group name | Value for "type" |
|---|---|---|---|
| Cross-section I(IPN) | STEEL_I_ROLLED_I | Cross-section UPE | STEEL_U_ROLLED_UPE |
| Cross-section IE | STEEL_I_ROLLED_IE | Cross-section UAP | STEEL_U_ROLLED_UAP |
| Cross-section IPE | STEEL_I_ROLLED_IPE | Cross-section L equilateral | STEEL_L_ROLLED_EQUAL |
| Cross-section HE | STEEL_I_ROLLED_HE | Cross-section L unequilateral | STEEL_L_ROLLED_UNEQUAL |
| Cross-section HL | STEEL_I_ROLLED_HL | Seamless tube CHS | STEEL_CH_ROLLED_CSN |
| Cross-section HD | STEEL_I_ROLLED_HD | MSH circular cross-section | STEEL_CH_ROLLED_MSH |
| Cross-section HP | STEEL_I_ROLLED_HP | Seamless tube RHS | STEEL_RH_ROLLED_CSN |
| Cross-section W | STEEL_I_ROLLED_W | MSH rectangular cross-section | STEEL_RH_ROLLED_MSH |
| Cross-section UB | STEEL_I_ROLLED_UB | Seamless tube square RHS | STEEL_RH_ROLLED_CSN_SQ |
| Cross-section UC | STEEL_I_ROLLED_UC | MSH square RHS | STEEL_RH_ROLLED_MSH_SQ |
| Cross-section U(UPN) | STEEL_U_ROLLED_U | Cross-section T | STEEL_T_ROLLED |
| Cross-section UE | STEEL_U_ROLLED_UE | | |

The attribute "**name**" corresponds to the cross-section name in the database. This value can be copied from the window "**Cross-section catalogue**" using the shortcut "**Ctrl**" + "**C**".



*The cross-section name for the attribute **<name>***

This attribute has to be identical to the name in the database.

```
</materials>
<sections>
  <section id="1" type="STEEL_I_ROLLED_HE" name="HE 160 B" material="1"/>
</sections>
<linetypes>
```

*hot-rolled cross-section in xml*

The values for the attribute "**type**" are listed in the following table for cross-section group "**Solid welded**". Additional attributes are also shown there. Values should be defined in meters.

| **Name** | **Value for "type"** | **Attributes** |
|---|---|---|
| I-cross-section | STEEL_I_FABRICATED | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness |
| U-cross-section | STEEL_U_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness |
| L-cross-section | STEEL_L_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**t1**" - thickness of vertical part; "**t2**" - thickness of horizontal part |
| CHS | STEEL_CH_FABRICATED | "**D**" - outer diameter; "**t**" - thickness |
| RHS | STEEL_RH_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - thickness of horizontal wall; "**tw**" - thickness of vertical wall |
| T-cross-section | STEEL_T_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness |
| Pi-cross-section | STEEL_Pi_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness; "**c**" - distance between centre and vertical wall |
| Cross | STEEL_CROSS_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - thickness of horizontal wall; "**tw**" - thickness of vertical wall |
| II-cross-section | STEEL_II_FABRICATED | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness; "**c**" - distance between centre and web |

"**Solid**" cross-sections can be defined using following values:

| **Name** | **Value for "type"** | **Attributes** |
|---|---|---|
| Circle | STEEL_CS | "**D**" - outer diameter |

| | | |
|---|---|---|
| Rectangle | STEEL_RS | "**h**" - cross-section height; "**b**" - cross-section width |

```
<sections>
  <section id="1" type="STEEL_I_ROLLED_HE" name="HE 160 B" material="1"/>
  <section id="2" type="STEEL_RS" name="Plech" material="2" h="0.01" b="0.1" />
</sections>
<linetypes>
```

*Rectangular cross-section in XML*

"**Built-up hot-rolled**" cross-sections contain additional attribute "**typeinternal**". This attribute defines the name of particular cross-section. Values for this attribute are identical to attribute "**type**" for welded cross-sections.

Attribute "**type**" for built-up hot-rolled cross-sections:

| **Name** | **Value for "type"** |
|---|---|
| Built-up 2xU | STEEL_RH_COMPOSED |
| Built-up I-profiles | STEEL_II_COMPOSED |

"**Built-up rolled**" cross-sections contain additional attributes "**typeinternal**" and "**distance**". The attribute "**typeinternal**" defines the name of particular cross-section. Values for this attribute are identical to attribute "**type**" for welded cross-sections. The attribute "**distance**" defines the distance between particular cross-sections.

Attribute "**type**" for built-up rolled cross-sections:

| **Name** | **Value for "type"** | **Name** | **Value for "type"** |
|---|---|---|---|
| Built-up rolled, 2xI | STEEL_BUILTUP_II | Built-up rolled, 2xL short | STEEL_BUILTUP_L2_LOW |
| Built-up rolled, 2xU closed | STEEL_BUILTUP_UU_O | Built-up rolled, 2xL cross | STEEL_BUILTUP_L2_CROSS |
| Built-up rolled, 2xU open | STEEL_BUILTUP_UU_I | Built-up rolled, 2xL closed long | STEEL_BUILTUP_LU_HIGH |
| Built-up rolled, 4xL | STEEL_BUILTUP_L4 | Built-up rolled, 2xL closed short | STEEL_BUILTUP_LU_LOW |
| Built-up rolled, 2xL long | STEEL_BUILTUP_L2_HIGH | | |

```
    <section id="1" type="STEEL_I_ROLLED_HE" name="HE 160 B" material="1"/>
    <section id="2" type="STEEL_RS" name="Plech" material="2" h="0.01" b="0.1" />
    <section id="3" type="STEEL_BUILTUP_II" typeinternal="STEEL_I_ROLLED_HE"
name="HE 160 B" material="2" distance="0.3" />
  </sections>
```

*Example*

## Timber

The values for the attribute "**type**" are listed in the following table for timber cross-sections. Additional attributes are also shown there. Values should be defined in meters.

| **Name** | **Value for "type"** | **Attributes** |
|---|---|---|
| Rectangle | WOOD_SQUARED_RECT | "**h**" - cross-section height; "**b**" - cross-section width |
| T-cross-section | WOOD_SQUARED_T | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness |
| I-cross-section | WOOD_SQUARED_I | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness |
| Pi-cross-section | WOOD_SQUARED_Pi | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness; "**c**" - distance between centre and vertical wall |
| Circle | WOOD_SQUARED_CIRCLE | "**D**" - outer diameter |
| Rectangle composite | WOOD_COMPOSED_RS | "**h**" - cross-section height; "**b**" - cross-section width; "**n**" - count of particular cross-sections; |
| Built-up section | WOOD_COMPOSED_RO | "**h**" - cross-section height; "**b**" - cross-section width; "**n**" - count of particular cross-sections; "**bm**" - distance between particular cross-sections |

## Concrete

The values for the attribute "**type**" are listed in the following table for RC cross-sections. Additional attributes are also shown there. Values should be defined in meters.

| Name | Value for "type" | Attributes |
|---|---|---|
| Rectangle | CONCRETE_RS | "**h**" - cross-section height; "**b**" - cross-section width |
| Hollow rectangle | CONCRETE_RH | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness; "**hnb**" - bottom haunch height; "**hnt**" - upper haunch height; "**bn**" - haunch width |
| T-cross-section | CONCRETE_T_BASIC | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness |
| T-cross-section, general | CONCRETE_T_GENERAL | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness; "**c**" - distance between centre and web |
| I-cross-section | CONCRETE_I_BASIC | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness |
| I-cross-section, general | CONCRETE_I_GENERAL | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness; "**hnb**" - bottom haunch height; "**hnt**" - upper haunch height; "**bnb**" - bottom haunch width; "**bnt**" - top haunch width |
| Circle | CONCRETE_CS | "**D**" - outer diameter |
| Ring | CONCRETE_CH | "**D1**" - outer diameter; "**D2**" - inner diameter |
| Trapezoid | CONCRETE_TRAPEZE | "**h**" - cross-section height; "**bt**" - cross-section width at the top; "**bb**" - cross-section width at the bottom |
| Rectangle with arches | CONCRETE_RS_ARC | "**h**" - cross-section height; "**b**" - cross-section width |

## Masonry

The values for the attribute "**type**" are listed in the following table for masonry cross-sections. Additional attributes are also shown there. Values should be defined in meters.

| Name | Value for "type" | Attributes |
|---|---|---|
| Rectangle | MASONRY_RS | "**h**" - cross-section height; "**b**" - cross-section width |
| Hollow rectangle | MASONRY_RH | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness |
| T-cross-section | MASONRY_T | "**h**" - cross-section height; "**b**" - cross-section width; "**tf**" - flange thickness; "**tw**" - web thickness |
| Cross | MASONRY_L | "**b1**" - cross-section width; "**h1**" - height of centre part; "**b2**" - width of upper part; "**h2**" - height of upper part; "**s1**" - distance of upper part from the right edge; "**b3**" - width of bottom part; "**h3**" - height of bottom part; "**s2**" - distance of bottom part from the right edge |
| I-cross-section | CONCRETE_I_BASIC | "**h**" - cross-section height; "**bft**" - top flange width; "**bfb**" - bottom flange width; "**tw**" - web thickness; "**tft**" - top flange thickness; "**tfb**" - bottom flange thickness |
| Circle | CONCRETE_CS | "**D**" - outer diameter |
| Ring | CONCRETE_CH | "**D1**" - outer diameter; "**D2**" - inner diameter |

# <linetypes>

This part contains properties of member profiles. It is an optional part of XML file. Every member profile is marked by the element **<linetype>**. This element supports following attributes:

**id** • Unique ID, obligatory

**startSupport, endSupport** • The member connection on the beginning and end. Available values: "**fixed**" (fixed connection), "**free**" (without connection), "**joint**" (joint) or ID from the part **<supports>**, optional, if not defined, the fixed connection is considered

**startDeplanation, endDeplanation**
- The number in the range *<0,1>* which defines the the rate of warping prevention at the beginning and end, optional, if not defined, the value *0* is used

**section**
- ID of the cross-section, obligatory (only for members with constant cross-section)

**sectionPri, sectionSec**
- ID of the cross-section at the beginning and at the end, obligatory (only for members with variable cross-section)

**rotation**
- Rotation of the cross-section defined in the anti-clockwise direction in degrees. Optional, if not defined, the value *0* is used

**considerShear**
- Adds the effect of shear forces for given member into analysis. Available values: "**true**" (including shear effect) or "**false**" (without shear effect), optional, if not defined, the value "**true**" is used

**typeCT**
- The option to specify whether the member acts in compression or tension only, optional. Available values: "**both**" (member carries compressive and tensile forces), "**tension**" (member transfers only tensile forces), "**compression**" (member transfers only compressive forces), the option "**both**" is considered if not defined

```
  </sections>
  <linetypes>
    <linetype id="1" startSupport="fixed" endSupport="fixed" section="3"
considerShear="false"/>
    <linetype id="2" startSupport="1" endSupport="2" section="2" />
  </linetypes>
  <points>
```

*Part <linetypes> of XML file*

## Elastic subsoil

The elastic subsoil can be assigned to the member using the element **<subsoil>**. Following attributes are supported by this element:

**c1, c2**
- Winkler-Pasternak constants, obligatory

**width**
- The member width, optional

**opposite**
- The setting whether the subsoil acts against the given axis. Optional, the vaule "**false**" is used as a default

**before**
- Switches on the shear effect at the beginning of the member. Optional, the value "**false**" is used if not defined

**behind**
- Switches on the shear effect at the end of the member. Optional, the value "**false**" is used if not defined

**direction**
- The axis direction, available values are "**2**" or "**3**", obligatory

```
    <linetype id="2" startSupport="1" endSupport="2" section="2" />
    <linetype id="1" startSupport="fixed" endSupport="free" section="1">
      <subsoil c1="11" c2="2" width="25.3" opposite="true" before="false"
behind="true" direction="2" />
      <subsoil c1="11" c2="2" width="5.3" before="false" behind="true"
direction="3" />
  </linetypes>
```

*Member profile with subsoil defined in two directions*

# <points>

This part contains properties of joints. It is an obligatory part of XML file. Every member is marked by the element **<line>**. The range of supported attributes differs according to the joint type:

## Absolute joints

Absolute joints support following attributes:

**id**
- unique joint ID (obligatory)

**x**
- The $x$ -coordinate of global coordinate system. Optional. If not defined, the coordinate $x=0$ is used.

**y**
- The $y$ -coordinate of global coordinate system. Optional. If not defined, the coordinate $y=0$ is used.

**z**
- The $z$ -coordinate of global coordinate system. Optional. If not defined, the coordinate $z=0$ is used.

**support** • The number of support type, which corresponds to the ID in the part **<supports>**. Additionally, values "**joint**" (hinged support) and "**fixed**" (fixed support) are allowed. Optional .

**coordinateSystem** • The number of local coordinate system which defines the rotation of support. The number corresponds to the unique ID in the part **<coordinateSystems>**. Optional.

## Relative joints

Relative joints support following attributes:

**id** • unique joint ID (obligatory)

**line** • The number of a reference member. The number corresponds to the ID in the part **<lines>** (optional)

**fromStart/ fromEnd** • The attributes which define the position of the relative joint on the reference member. If the attribute "**fromStart**" is used, the position is measured from the begining of the member. If the attribute "**fromEnd**" is used, the position is measured from the end of the member. The value of the attribute defines the distance from reference point. The value with the character *%* means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory.

**support** • The number of support type, which corresponds to the ID in the part **<supports>**. Additionally, values "**joint**" (hinged support) and "**fixed**" (fixed support) are allowed. Optional.

**coordinateSystem** • The number of local coordinate system which defines the rotation of support. The number corresponds to the unique ID in the part **<coordinateSystems>**. Optional.

Only structures with more than two absolute joints can be imported into the programs "**Fin 2D**" and "**Fin 3D**".

```
</linetypes>
<points>
  <point id="1" x="-5" y="0" z="0" support="joint"/>
  <point id="2" x="5" y="3,6" z="1" support="1" coordinateSystem="1" />
  <point id="3" line="1" fromStart="0,5"/>
  <point id="4" line="1" fromEnd="0,5"/>
  <point id="5" line="1" fromStart="50%"/>
</points>
<lines>
  <line id="1" start="3" end="5" linetype="1"/>
```

*Part <points> with two absolute and three relative joints*

# <lines>

This part contains properties of members. It is an obligatory part of XML file. Every member is marked by the element **<line>**. This element supports following attributes:

**id** • Unique member ID (obligatory)

**start** • Number of start joint, the number corresponds to the ID in the part **<points>** (obligatory)

**end** • Number of end joint, the number corresponds to the ID in the part **<points>** (obligatory)

**linetype** • Number of member profile, which corresponds to ID in the part **<linetypes>** (optional)

Only structures with more than one member can be imported into the programs "**Fin 2D**" and "**Fin 3D**".

```
  <point id="7" line="1" fromStart="70%"/>
</points>
<lines>
  <line id="1" start="3" end="5" linetype="1"/>
  <line id="2" start="5" end="7" linetype="3"/>
  <line id="3" start="7" end="1" linetype="1"/>
</lines>
<loadcases/>
```

*Part <lines> with three members*

# <loadcases>

This part contains properties of load cases including inserted loads. It is an optional part of XML file. Every load case is marked by the element **<loadcase>**. This element supports following attributes:

**id** • Unique ID, obligatory

**name** • The load case name, obligatory

**code** • The load case code. Possible values are: "**force**" (load case with forces and moments), "**deformation**" (load case with support deformation), "**dead**" (load case with self-weight, this load case cannot contain any inserted load) or "**temperature**" (load case with load defined by temperature difference)

**type** • The load case type (obligatory), avialable values: "**permanent**" (permanent load), "**longTimeLive**" (variable long term), "**middleTimeLive**" (variable middle term), "**shortTimeLive**" (variable short term), "**momentTimeLive**" (variable instantaneous), "**accidentalLive**" (accidental), "**shortTimeLiveWind**" (short term wind load), "**shortTimeLiveSnow**" (short term snow load), "**mediumTimeLiveSnow**" (middle term snow load), "**accidentalWind**" (accidental wind load) and "**accidentalSnow**" (accidental snow load)

**category** • Category of variable load case. Available values: "**user**" (user-defined combination factors), "**standard**" (standard permanent load), "**A**" (variable cat. A), "**B**" (variable cat. B), "**C**" (variable cat. C), "**D**" (variable cat. D), "**E**" (variable cat. E), "**F**" (variable cat. F), "**G**" (variable cat. G), "**H_RoofOnly**" (variable cat. H), "**I_RoofPerA**" (variable roof I for cat. A), "**I_RoofPerB**" (variable roof I for cat. B), "**I_RoofPerC**" (variable roof I for cat. C), "**I_RoofPerD**" (variable roof I for cat. D), "**K_RoofSpecial**" (variable cat. K), "**wind**" (vítr), "**snow_FINS**" (snow Scandinavia), "**snow_Greater1000**" (snow > 1000 a.s.l.), "**snow_LessEqual1000**" (snow < 1000 a.s.l.), "**temperature**" (thermal load).

**gamaSup** • The partial factor for upper design value, optional

**gamaInf** • The partial factor for lower design value, optional

**ksi** • The factor for permanent loads in "**alternative**" combinations. Available only for design standards "**ec**" or "**sans**", optional.

**psi0** • Factor for combination value. Available only for design standards "**ec**" or "**sans**", optional.

**psi1** • Factor for frequent value. Available only for design standards "**ec**" or "**sans**", optional.

**psi2** • Factor for quasi-permanent value. Available only for design standards "**ec**" or "**sans**", optional.

**coefficient** • The setting which defines partial load case factor for standards "**csn**", "**stn**" or "**other**". Optionak, default value is ***1.0***

Individual load cases (except the load case with the code "**dead**") may contain joint and member loads. The joint loads are defined using the element **<pointLoad>** and member loads are defined using the element **<lineLoad>**.

## Point loads

Point loads are defined using the element **<pointLoad>**. Following attributes are available:

**x**, **y**, **z** • Forces or deformations in given directions, optional

**mx**, **my**, **mz** • Moments and rotations in given directions, optional

**points** • List of points where is the load assigned. Obligatory. Example of syntax: the value "**1,6,9-12, 3**" adds the load into points 1, 3, 6, 9, 10, 11, 12

```
<loadcases>
  <loadcase id="1" name="ZS1" code="force" type="longTimeLive" category="A">
    <pointLoad y="34" z="75,3" mz="15.3" points="1,3,5,7"/>
  </loadcase>
</loadcases>
```

*Point load*

## Member load

Member loads are defined by the element **<lineLoad>**. The attribute "**lines**" defines member IDs, which should be loaded by this load. Syntax is described above (example: "**1,6,9-12, 3**"). Individual types of member loads are defined using following elements:

**<singleForce>** • Single force on member

**<singleMoment>** • Single moment on member

**<continuous>** • Line load for the whole member length

**<continuousPart>** • Line load on the part of member length

**<trapezoidal>** • Trapezoidal load with defined beginning and end

The element **<singleForce>** for **single force** supports following attributes:

**direction** • The load direction. Available values: "**x**" (global axis $x$ ), "**y**" (global axis $y$ ), "**z**" (global axis $z$ ), "**1**" (local axis $1$ ), "**2**" (local axis $2$ ), "**3**" (local axis $3$ )

| | |
|---|---|
| **value** | • Value of load |
| **angle** | • The angle between the force and member axis. Optional, if not defined, the value $90$ is used |
| **fromStart, fromEnd** | • The position from the member beginning (attribute "**fromStart**") or member end (the attribute "**fromEnd**"). The value of the attribute defines the distance from reference point. The value with the character $\%$ means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory. |

```
<combination id="1" name="Kombinace 1" kind="1" type="basic">
  <loadcase id="1">
    <lineLoad lines="1,5">
      <singleForce direction="x" value="34,6" fromStart="2,5"/>
    </lineLoad>
  </loadcase>
  <loadcase id="2"/>
```

*Load case with point loads on members 1 and 5*

The element **<singleMoment>** for **single moment** supports following attributes:

| | |
|---|---|
| **direction** | • The load direction. Available values: "**x**" (global axis $x$ ), "**y**" (global axis $y$ ), "**z**" (global axis $z$ ), "**1**" (local axis $1$ ), "**2**" (local axis $2$ ), "**3**" (local axis $3$ ) |
| **value** | • Value of moment |
| **fromStart, fromEnd** | • The position from the member beginning (attribute "**fromStart**") or member end (the attribute "**fromEnd**"). The value of the attribute defines the distance from reference point. The value with the character $\%$ means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory. |

The element **<continuous>** for **continuous load on the whole member length** supports following attributes:

| | |
|---|---|
| **direction** | • The load direction. Available values: "**x**" (global axis $x$ ), "**y**" (global axis $y$ ), "**z**" (global axis $z$ ), "**1**" (local axis $1$ ), "**2**" (local axis $2$ ), "**3**" (local axis $3$ ) |
| **value** | • Value of load |
| **angle** | • The angle between the force and member axis. Optional, if not defined, the value $90$ is used |

```
    </lineLoad>
    <lineLoad lines="3-6">
      <continuous direction="z" value="12,3" />
    </lineLoad>
  </loadcase>
```

*Continuous load on members 3, 4, 5 a 6*

The element **<continuousPart>** for **continuous load on the member part** supports following attributes:

| | |
|---|---|
| **direction** | • The load direction. Available values: "**x**" (global axis $x$ ), "**y**" (global axis $y$ ), "**z**" (global axis $z$ ), "**1**" (local axis $1$ ), "**2**" (local axis $2$ ), "**3**" (local axis $3$ ), "**yProjection**" (direction of global axis $x$ on member projection) "**zProjection**" (direction of global axis $z$ on member projection) |
| **value** | • Value of load |
| **angle** | • The angle between the load and member axis. Optional, if not defined, the value $90$ is used |
| **fromStart1, fromEnd1, fromSecond** | • The position of load beginning from the member start (attribute "**fromStart1**") or member end (the attribute "**fromEnd1**"). The load beginning can be also defined using the position of load end (the attribute "**fromSecond**"). The value of the attribute defines the distance from reference point. The value with the character $\%$ means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory. |
| **fromStart2, fromEnd2, fromFirst** | • The position of load beginning from the member start (attribute "**fromStart2**") or member end (the attribute "**fromEnd2**"). The load beginning can be also defined using the position of load end (the attribute "**fromFirst**"). The value of the attribute defines the distance from reference point. The value with the character $\%$ means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory. |

The element **<trapezoidal>** for **trapezoidal load on the member part** supports following attributes:

| | |
|---|---|
| **direction** | • The load direction. Available values: "**x**" (global axis $x$ ), "**y**" (global axis $y$ ), "**z**" (global axis $z$ ), "**1**" (local axis $1$ ), "**2**" (local axis $2$ ), "**3**" (local axis $3$ ), "**yProjection**" (direction of global axis $x$ on member projection) "**zProjection**" (direction of global axis $z$ on member projection) |
| **startValue** | • Value of load at the begining, obligatory |
| **endValue** | • Value of load at the end, obligatory |

**angle** • The angle between the load and member axis. Optional, if not defined, the value *90* is used

**fromStart1, fromEnd1, fromSecond** • The position of load beginning from the member start (attribute "**fromStart1**") or member end (the attribute "**fromEnd1**"). The load beginning can be also defined using the position of load end (the attribute "**fromSecond**"). The value of the attribute defines the distance from reference point. The value with the character *%* means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory.

**fromStart2, fromEnd2, fromFirst** • The position of load beginning from the member start (attribute "**fromStart2**") or member end (the attribute "**fromEnd2**"). The load beginning can be also defined using the position of load end (the attribute "**fromFirst**"). The value of the attribute defines the distance from reference point. The value with the character *%* means that the distance is defined using per cents of the member length. Otherwise, the position is defined in meters. Obligatory.

```
<lineLoad lines="2">
  <trapezoidal direction="z" startValue="12,3" endValue="1.5"
fromStart1="1,5" fromFirst="3"/>
</lineLoad>
```

*Trapezoidal load*

## <combinations>

This part contains properties of load combinations. It is an optional part of XML file. Every combination is marked by the element **<combination>**. This element supports following attributes:

**id** • Unique ID, obligatory

**name** • The combination name, obligatory

**kind** • The combination kind, obligatory. Available values: "**1**" (combination for I.order analysis), "**2**" (combination for II. order analysis), "**linearStability**" (combination for linear stability)

**type** • The combination type, obligatory. Available values for standards "**ec**" and "**sans**" (defined in the part **<header>**): "**basic**" (basic combination ULS), "**basicAlternate**" (alternative combination ULS), "**accidental**" (accidental combination ULS), "**characteristic**" (charakteristic combination SLS), "**frequent**" (frequent combination SLS), "**quasipermanent**" (quasi permanent combination SLS). Values for standards "**csn**", "**stn**" and "**other**": "**extreme**" (ULS), "**service**" (SLS) or "**fire**" (combination for fire resistance).

The list of load cases is defined with the help of elements **<loadcase>**. Following attributes are supported:

**id** • ID of load case given in the part **<loadcases>** , obligatory

**isFavourable** • Only for standards "**ec**" or "**sans**" in the part **<header>**. The setting affects whether upper or lower combination value will be used. Available options "**true**" (lower value) or "**false**" (upper value). Optional, the value "**false**" is used as a default.

**isMainVariable** • Only for standards "**ec**" or "**sans**" in the part **<header>**. The setting affects whether the load case will be considered as main variable load or not. Available options "**true**" (is main variable) or "**false**" (is not main variable). Optional, the value "**false**" is used as a default.

**coefficient** • This setting defines the combination factor of the load case (only for standards "**csn**", "**stn**" or "**other**")

```
</loadcases>
<combinations>
  <combination id="1" name="Kombinace 1" kind="1" type="basic">
    <loadcase id="1"/>
    <loadcase id="2"/>
    <loadcase id="3" isMainVariable="true"/>
  </combination>
</combinations>
<dimSteel/>
```

*Combination for ULS with three load cases*

# XML import for steel members

Data for verification of steel structures are organized in the part which is defined by the element **<dimSteel>**. Particular members are defined by the element **<member>** which supports following attributes:

**id** • Unique ID of the member

**bucklingYZ** • The part with buckling properties. If not defined, buckling is not considered

These parts are necessary for the direct import into the program "**Steel**":

**<section>** • The part with cross-section properties
**<material>** • The part with material properties
**<load>** • The part with diagrams of internal forces

Following parts are identical for direct import into the program "**Steel**" and for import of structures into "**Fin 2D**" or "**Fin 3D**":

**<buckling>** • The part with properties of buckling. If not defined, buckling is not considered
**<lateralBuckling>** • The part with properties of lateral torsional buckling. If not defined, the lateral torsional buckling is not considered
**<weakening>** • Optional part with holes etc.
**<stiffeners>** • Optional part with web stiffeners

```
<dimSteel>
  <member id="1" bucklingYZ="true">
    <buckling/>
    <lateralBuckling/>
    <weakening/>
    <stiffeners/>
  </member>
</dimSteel>
```

*Fundamental structure of XML for verification of steel member*

## <buckling>

This part contains segments with parameters of buckling. It is an optional part of XML file. Every segment is marked by the element **<buckling>**. This element supports following attributes:

**type** • The direction of considered buckling. Available values: "**y**" (buckling Y), "**z**" (buckling Z)
**start** • The beginning of a segment with web stiffeners. If not defined, the member beginning is considered
**end** • The end of a segment with web stiffeners. If not defined, the member end is considered
**length** • The fundamental length for the buckling analysis. If not defined, the segment length is considered
**support** • The end conditions of the member. Available values: "**jj**" (joint-joint), "**jf**" (joint-fixed), "**ff**" (fixed-fixed), "**user**" (user-defined buckling length, the attribute "**k**" - buckling length factor)
**use** • The setting which switches on or off the consideration of buckling. Available values: "**true**" (buckling considered), "**false**" (buckling not considered). If not defined, the value "**true**" is used.

```
<member id="1" bucklingYZ="true">
  <buckling type="y" start="3,4" end="4,5" length="5" support="jj" use="true"/>
  <lateralBuckling type="my" start="3,4" length="5" shape="8" position="0,3"
```

*Buckling parameters*

## <lateralBuckling>

This part contains segments with parameters of lateral torsional buckling. It is an optional part of XML file. Every segment is marked by the element **<lateralBuckling>**. This element supports following attributes:

**type** • The direction of considered bending moment. Available values: "**my**" (LT buckling for moment $M_y$ ), "**mz**" (LT buckling for moment $M_z$ )
**start** • The beginning of a segment with web stiffeners. If not defined, the member beginning is considered
**end** • The end of a segment with web stiffeners. If not defined, the member end is considered
**length** • The fundamental length for the calculation of lateral torsional buckling. If not defined, the segment length is considered
**shape** • The shape of the moment area. Available values: "**1**" - "**7**" (according to the shape of moment area). The additional attribute "**ksi**" (ratio between bending moment at the beginning and end, permitted range is $<0, 1>$ ) has to be specified for the value "**3**". The additional attributes "**position**" (the position of the load, range $<0, 1>$ ), "**k**" and "**kw**" (available values "**jj**" joint-joint and "**ff**" fixed-fixed) has to be specified for values between "**4**" and "**7**".

**use** • The setting which switches on or off the consideration of lateral torsional buckling. Available values: "**true**" (LT buckling considered), "**false**" (LT buckling not considered). If not defined, the value "**true**" is used.

```
    <buckling type="y" start="3,4" end="4,5" length="5" support="jj" use="true"/>
    <lateralBuckling type="my" start="3,4" length="5" shape="8" position="0,3"
k="ff" kw="ff" use="false"/>
    <weakening start="3,4" n1="3" a1="2" b1="3" d1="5"/>
```

*Lateral torsional buckling for moment $M_y$*

## <weakening>

This part contains segments with perforation of the cross-section. It is an optional part of XML file. Every segment with perforation is marked by the element **<weakening>**. This element supports following attributes:

**start** • The beginning of a segment with web stiffeners. If not defined, the member beginning is considered

**end** • The end of a segment with web stiffeners. If not defined, the member end is considered

**n1, n2, n3, d1, d2, d3, b1, b2, b3, a1, a2, a3** • Dimensions which are described in the corresponding window. If not defined, the value $\mathbf{0}$ os considered.

**fill1, fill2, fill3** • Setting which switches on or off filling of holes. Available values: "**true**" (holes are filled), "**false**" (holes are not filled)

```
k="ff" kw="ff" use="false"/>
    <weakening start="3,4" n1="3" a1="2" b1="3" d1="5"/>
    <stiffeners start="3,4" av="0,5"/>
  </member>
```

*Member weakening*

## <stiffeners>

This part contains segments with web stiffeners. It is an optional part of XML file. Every segment with web stiffeners is marked by the element **<stiffeners>**. This element supports following attributes:

**start** • The beginning of a segment with web stiffeners. If not defined, the member beginning is considered

**end** • The end of a segment with web stiffeners. If not defined, the member end is considered

**av** • Distance between web stiffeners

```
    <weakening start="3,4" n1="3" a1="2" b1="3" d1="5"/>
    <stiffeners start="3,4" av="0,5"/>
  </member>
```

*Part with web stiffeners*

# XML import for RC members

The XML import may be used for input of "**member**" tasks. Both general rules for XML files and following rules given for this import should be respected:

- The name of fundamental element is not limited (for example <FINE>)
- All values are defined using base SI units
- Angles are defined in degrees

## File structure

The XML file may contain general properties of the project defined in the element **<header>**. Member properties are stored in the element **<dimConcrete>**. Every member in the part **<dimConcrete>** is defined by the element **<member>**. This element supports following attributes:

**name** • The member name, obligatory

**length** • The member length. Not used in files for "**Fin 2D**" and "**Fin 3D**"

**kind** • The structure type, obligatory. Available values: "**beam**", "**column**", "**slab**", "**wall**"

**aeration** • The setting whether the aeration exceeds $4\%$ . Available values: "**true**", "**false**". If not defined, the value "**false**" is used.

**dg** • The diameter of aggregate, optional parameter. Default value is $0.016$ .

**bucklingMethod** • The method for buckling analysis. Available values: "**stiffness**" - the method based on nominal stiffness, "**curvature**" - the method based on nominal curvature. Optional, default value is "**stiffness**".

**reinforcementCompressed** • Consider reinforcement in compression. Available values: "**true**", "**false**", if not defined, the value "**true**" is considered

The element **<member>** may contain following parts:

**<section>** • The part with diagrams of internal forces. Not used in files for "**Fin 2D**" and "**Fin 3D**"

**<material>** • The with . Not used in files for "**Fin 2D**" and "**Fin 3D**"

**<materialReinforcementLongitudinal>** • The material of longitudinal reinforcement

**<materialReinforcementTransverse>** • The material of transverse reinforcement

**<environment>** • The properties of environment

**<creep>** • The creep properties

**<load>** • The part with diagrams of internal forces. Not used in files for "**Fin 2D**" and "**Fin 3D**"

**<imperfection>** • The effect of imperfection

**<buckling>** • The buckling properties

**<cover>** • The calculation of reinforcement cover

**<reinforcement/>** • The part with longitudinal reinforcement

**<transversereinforcement/>** • The part with transverse reinforcement

```
<FINE>
  <header/>
  <dimConcrete>
    <member>
      <section>
      <material/>
      <materialReinforcementLongitudial/>
      <materialReinforcementTransverse/>
      <environment/>
      <creep/>
      <load/>
      <imperfection/>
      <buckling/>
      <cover/>
      <reinforcement/>
      <transversereinforcement/>
    </member>
  </dimConcrete>
</FINE>
```

*File structure*

## <materialReinforcementLongitudinal>, <materialReinforcementTransverse>

These parts contains material of longitudinal (the element **<materialReinforcementLongitudinal>**) and transverse (the element **<materialReinforcementTransverse>**) reinforcement. It is an optional part of the XML file. These parts contain obligatory attribute "**type**" which defines the material class. Following values are supported:

| **Material name** | **Value for XML** |
|---|---|
| EN 10025: Fe 360 | CS_EC2_10505R |
| EN 10025: Fe 430 | CS_EC2_10425V |
| EN 10025: Fe 510 | CS_EC2_KARIW |
| prEN 10113: Fe E 275 | CS_EC2_MeshSZ |
| prEN 10113: Fe E 355 | CS_EC2_B420 |
| EN 10210-1: S 235 | CS_EC2_B500 |
| EN 10210-1: S 275 | CS_EC2_B550 |

```
<material type='CO_EC2_C45'/>
<materialReinforcementLongitudial type='CS_EC2_B500'/>
<materialReinforcementTransverse type='CS_EC2_B420'/>
<environment xc='2' xa='1'/>
```

*Materials of reinforcement*

## <environment>

This part contains details of environment. It is an optional part of XML file defined by the element **<environment>**. This element supports following attributes:

**xc** • Corrosion induced by carbonation, available values "**1**" - "**4**"
**xd** • Corrosion induced by chlorides, available values "**1**" - "**3**"
**xs** • Corrosion induced by chlorides from sea, available values "**1**" - "**3**"
**xf** • Freeze/thaw attack, available values "**1**" - "**4**"
**xa** • Chemical attack, available values "**1**" - "**3**"

```
<materialReinforcementTransverse type='CS_EC2_B420'/>
<environment xc='2' xa='1'/>
<creep t0='30' t='30000' rh='0.6' u='1'/>
```

*Part <environment>*

## <creep>

This part contains creep properties. It is an optional part defined by the element **<creep>**. This element supports following attributes:

**t0** • The beginning of loading, optional. The default value is ***28***
**t** • The end of loading, optional. The default value is ***29200***
**rh** • relative environmental humidity $<0, 1>$ , the default value is ***0,5***
**u** • The perimeter in contact with an atmosphere. If not defined, the whole perimeter of the cross-section is considered

```
<environment xc='2' xa='1'/>
<creep t0='30' t='30000' rh='0.6' u='1'/>
<load title='Kombinace 1' secondOrder='true'
```

*Part <creep>*

## <load>

The element **<load>** contains diagrams of internal forces in the certain combination. More elements **<load>** have to be specified if more load combinations should be considered. This element supports following attributes:

**title** • The name, obligatory
**secondOrder** • The setting which defines whether the forces were calculated using second order theory. Available values: "**true**" (inner forces according to II. order), "**false**" (inner forces according to I. order theory). Optional, default value is "**false**".
**combinationKind** • The type of combination, available values: "**basic**", "**characteristic**", "**frequent**", "**quasipermanent**"
**QPCoef** • The load duration factor, permitted range is $<0,1>$ . Default value is ***1.0***

Diagrams of inner forces are defined using elements **<force>**. These elements contain the values of forces and moments in individual points along the member length. The element supports following attributes:

**x** • The position measured from the beginning of the member, obligatory.
**N** • The value of axial force
**Vy** • The value of shear force in the direction of the axis $y$
**Vz** • The value of shear force in the direction of the axis $z$
**My** • The value of bending moment about the axis $y$
**Mz** • The value of bending moment about the axis $z$
**T** • The value of torsional moment

The intermediate values are calculated using rules given in the software (linear interpolation). Value defined in the format like "**1230;1350**" refer to different value of the force on the left and on the right.

```
<creep t0='30' t='30000' rh='0.6' u='1'/>
<load title='Kombinace 1' combinationKind='basic' loadcaseKind=''>
  <force x='0' N='12100' Vy='12000' My='3000'/>
  <force x='5,1' N='12300;13500'/>
  <force x='11,2' N='13400' Vz='11'/>
</load>
<imperfection start="3,4" end="4,5" length="5" use="true"/>
```

*The load with specified diagrams of inner forces*

## <imperfection>

This part contains segments with parameters of imperfection. It is an optional part of XML file. Every segment is marked by the element **<imperfection>**. This element supports following attributes:

| | |
|---|---|
| **start** | • The beginning of a segment with imperfection. If not defined, the member end is considered |
| **end** | • The end of a segment with imperfection. If not defined, the member end is considered |
| **length** | • The fundamental length for the imperfection. If not defined, the segment length is considered |
| **use** | • The setting which switches on or off the consideration of imperfection. Available values: "**true**" (imperfection considered), "**false**" (imperfection not considered). If not defined, the value "**true**" is used. |

```
</load>
<imperfection start="3,4" end="4,5" length="5" use="true"/>
<buckling type="y" start="3,4" end="4,5" length="5" support="jj"
```

*The part <imperfection>*

## <buckling>

This part contains segments with parameters of buckling. It is an optional part of XML file. Every segment is marked by the element **<buckling>**. This element supports following attributes:

| | |
|---|---|
| **type** | • The direction of considered buckling. Available values: "**y**" (buckling Y), "**z**" (buckling Z) |
| **start** | • The beginning of a segment with web stiffeners. If not defined, the member beginning is considered |
| **end** | • The end of a segment with web stiffeners. If not defined, the member end is considered |
| **length** | • The fundamental length for the buckling analysis. If not defined, the segment length is considered |
| **support** | • The end conditions of the member. Available values: "**jj**" (joint-joint), "**jf**" (joint-fixed), "**ff**" (fixed-fixed), "**f**" (cantilever), "**fp**" (fixed-fixed with shift), "**user**" (user-defined buckling length, the attribute "**beta**" - buckling length factor) |
| **use** | • The setting which switches on or off the consideration of buckling. Available values: "**true**" (buckling considered), "**false**" (buckling not considered). If not defined, the value "**true**" is used, "**braced**" (braced structure), "**unbraced**" (unbraced structure). |

```
<imperfection start="3,4" end="4,5" length="5" use="true"/>
<buckling type="y" start="3,4" end="4,5" length="5" support="jj" use="true"/>
<cover class='s3' life80='true' life100='true' slab='true' quality='true'
```

*Part <buckling>*

## <cover>

This part contains parameters which affect the calculation of reinforcement cover:

| | |
|---|---|
| **class** | • The structure class, available values are: "**s1**","**s2**", "**s3**", "**s4**", "**s5**", "**s6**". The default value is "**s4**". |
| **life80** | • The design lifetime exceeds ***80 years*** . Available values: "**true**", "**false**", default value is "**false**" |
| **life100** | • The design lifetime exceeds ***100 years*** . Available values: "**true**", "**false**", default value is "**false**" |
| **slab** | • The setting which defines that the member has slab geometry. Available values: "**true**", "**false**", default value is "**false**" |
| **quality** | • Special quality control. Available values: "**true**", "**false**", default value is "**false**" |

**abbrasion** • Abrasion class. Available values: "**XM1**", "**XM2**", "**XM3**", the abrasion is not considered as a default

**surface** • The uneven surface correction, optional, default value is $\boldsymbol{0}$

**cgama** • Additive safety element $\Delta c_{dur,\gamma}$, default value is $\boldsymbol{0}$

**cst** • Stainless steel $\Delta c_{dur,st}$, default value is $\boldsymbol{0}$

**cadd** • Additional protection $\Delta c_{dur,add}$, default value is $\boldsymbol{0}$

**cdev** • Allowance in design for deviation $\Delta c_{dev}$, default value is $\boldsymbol{0}$

**ground** • The ground type. Available values: "**prepared**" (concrete cast against prepared ground), "**soil**" (concrete cast against soil). The concrete cast against ground is not considered as a default.

```
    <buckling type="y" start="3,4" end="4,5" length="5" support="jj" use="true"/>
    <cover class='s3' life80='true' life100='true' slab='true' quality='true'
abbrasion='XM1' surface='0.02' cgama='0.01' cst='0.005' cadd='0.011' cdev='0.015'
ground='prepared'/>
    <reinforcement start="3,4" end="4,5" cover='min'>
```

*Part <cover>*

## <reinforcement>

This part contains segments with longitudinal reinforcement. It is an optional part of XML file. Every segment is marked by the element **<reinforcement>**. This element supports following attributes:

**start** • The beginning of a segment. If not defined, the member beginning is considered

**end** • The end of a segment. If not defined, the member beginning is considered

**cover** • The reinforcement cover. Available values: "**min**" (minimum cover), "**stirrup**" (minimum cover and stirrups), "**user**" (user-defined value, the value of cover has to be defined for the attribute "**user**", if not specified, the value is identical to the option "**stirrup**")

Other attributes differ according to the cross-section shape:

### Rectangular cross-sections

The additional attribute "**positioning**" can be defined for the element **<reinforcement>**. This attribute affects positions of bars in particular rows. Available values: "**even**" (identical distance between bars), "**side**" (bar placed as close to edge as possible). The particular rows or bars are specified with the help of elements **<row>** (rows) or **<inlet>** (bars).

Following attributes are supported by the element **<row>**:

**kind** • Row type, values: "**top**", "**bottom**"

**diameter** • Bar diameter

**distance** • Input style based on distance between bars, the value of the attribute defines this distance

**count** • Input style based on count of bars. The count is entered as a value of this attribute

**cover** • Cover. The minimum cover is considered if not specified

```
<reinforcement start="3,4" end="4,5" cover='min'>
  <row kind='top' diameter='0.012' distance='0.2'/>
  <row kind='top' diameter='0.012' count='3' cover='0.1'/>
</reinforcement>
```

*Part <reinforcement> with two rows*

Individual bars are specified by elements **<inlet>**. Rows are ignored in this case. The element supports following attributes:

**y** • Bar position (coordinate in the direction of the axis $y$ )

**z** • Bar position (coordinate in the direction of the axis $z$ )

**diameter** • Bar diameter

```
  </reinforcement>
  <reinforcement start="3,4" end="4,5" cover='min'>
    <inlet y='0,08' z='0.23' diameter='0.02'/>
    <inlet y='0,22' z='0.23' diameter='0.02'/>
  </reinforcement>
</member>
```

*Part <reinforcement> with two bars*

## Circular cross-section

Following attributes are supported by the element **<reinforcement>** for circular cross-section:

**top** • Insert bar at the top, available value: "**true**", "**false**", default value is "**true**"
**count** • Count of bars
**diameter** • Bar diameter

## Oval cross-section

Following attributes are supported by the element **<reinforcement>** for oval cross-section:

**count1** • Count of bars in arch
**diameter1** • Bar diameter in arch
**count2** • Count of bars in straight part
**diameter2** • Bar diameter in straight part

```
    </reinforcement>
    <reinforcement start="3,4" end="4,5" cover='stirrup' diameter1='0.02'
count1='8' diameter2='0.02' count2='4'/>
  </member>
```

*Part <reinforcement> for oval cross-section*

# <transverseReinforcement>

This part contains segments with transverse reinforcement. It is an optional part of XML file. Every segment is marked by the element **<transverseReinforcement>**. This element supports following attributes:

**start** • The beginning of a segment. If not defined, the member beginning is considered
**end** • The end of a segment. If not defined, the member beginning is considered
**diagonalAngle** • The angle of compression struts, if not specified, the value is calculated by the software
**ratio_zd** • The ratio $z/d$ , range $(0;1)$ , if not specified, the value is calculated by the software

Other attributes differ according to the reinforcement type:

## Boundary stirrups

The element **<boundary>** is used for this type of reinforcement. It supports following attributes:

**diameter** • Diameter
**distance** • Distance between sitrrups
**kind** • Style of torsion analysis. Available values: "**shear**" (consider stirrups only for shear), "**auto**" (split between shear and torsion automatically), "**user**" (user-defined ration of split between shear and torsion. The ratio has to be defined by the additional attribute "**ratio**")

## Vertical ties (inner stirrups)

The element **<zstirrups>** is used for this type of reinforcement. It supports following attributes:

**diameter** • Diameter
**distance** • Distance between sitrrups, default value is equal to the distance of boundary stirrups
**count** • Count of ties

## Horizontal ties (inner stirrups)

The element **<ystirrups>** is used for this type of reinforcement. It supports following attributes:

**diameter** • Diameter
**distance** • Distance between sitrrups, default value is equal to the distance of boundary stirrups
**count** • Count of ties

## Vertical bent-up bars

The element **<zbends>** is used for this type of reinforcement. It supports following attributes:

**diameter** • Diameter
**distance** • distance between bent-up bars along the member length. If not specified, one bent-up bar is considered
**count** • Count of bars on one position
**angle** • Pitch

### Horizontal bent-up bars

The element **<ybends>** is used for this type of reinforcement. It supports following attributes:

| | |
|---|---|
| **diameter** | • Diameter |
| **distance** | • distance between bent-up bars along the member length. If not specified, one bent-up bar is considered |
| **count** | • Count of bars on one position |
| **angle** | • Pitch |

```
  </reinforcement>
  <transversereinforcement start="3,4" diagonalAngle='45' ratio_zd='0.9'>
    <boundary diameter='0.012' distance='0.2'/>
    <zstirrups diameter='0.012' distance='0.2' count='2'/>
    <ystirrups diameter='0.012' distance='0.2' count='2'/>
    <zbends diameter='0.012' distance='0.2' count='2' angle='45'/>
    <ybends diameter='0.012' distance='0.2' count='2' angle='45'/>
  </transverseReinforcement>
</member>
```

*Part <transverseReinforcement> with more types of shear reinforcement*

# Inputs and design

# Program Fin 2D

The program "**Fin 2D**" is suitable for the structural analysis of 2D truss and frame structures with the help of finite element method. The particular members of the structure may be designed in the verification programs.

The main parts of the user interface are the workspace, the tree menu, the main menu and the input frame in thee bottom part of the window. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

## Tree menu

The tree menu is the fundamental navigation object of the program. It contains all important functions for the work with the structure. The tree menu contains these parts:

- **Topology** - Contains tools for input of structure topology (joints and members)
- **Load** - Loads, load cases and combinations can be specified in this part
- **Calculation** - Runs the calculation of internal forces
- **Results** - Shows the results and contains tools for verification of members

The bottom part of tree menu contains buttons for insertion and administration of pictures for output documentation. The structure view in the workspace may be saved and used in output documentation. The pictures are updated continuously, the pictures in the documentation shows the latest state of structure including corresponding results. The new picture can be added with the help of the button "**Add picture**". After using this button, the window "**Picture properties**" appears. The parameters like description, orientation and position in document structure can be defined in this window. The pictures can be modified in the window "**List of pictures**", that can be opened by the button of the same name.

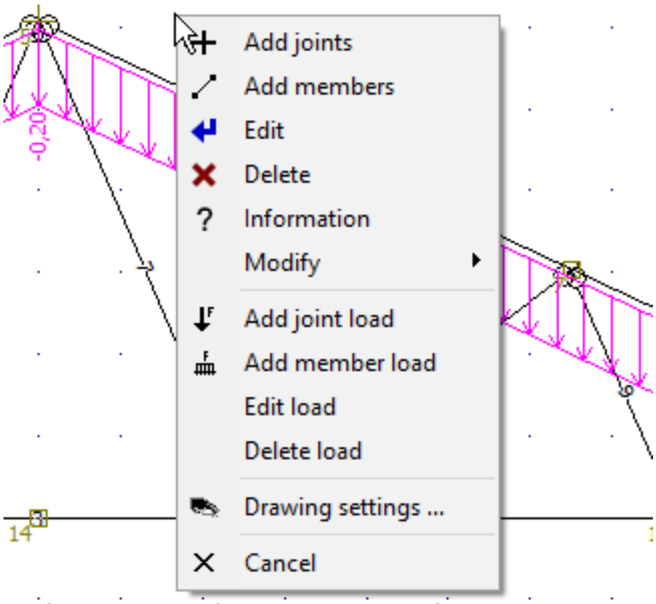
## Workspace

The workspace shows the created structure and can be used both for the graphical input and results display. The displayed items can be switched on or off in the window "**Drawing settings**", that can be launched by the button "" in the toolbar above the workspace. The view configuration may be saved or restored with the help of pre-defined templates, the work with templates is described here.


*Choice of view template*

The workspace also contains an option to use context menus, that can be opened by right mouse button click. This menu contains the most common commands. The range of commands differ according to the element type. Menus are different for joints, members and general point in the workspace. Additional commands are available for structures with selected elements (highlighted by green in the workspace).



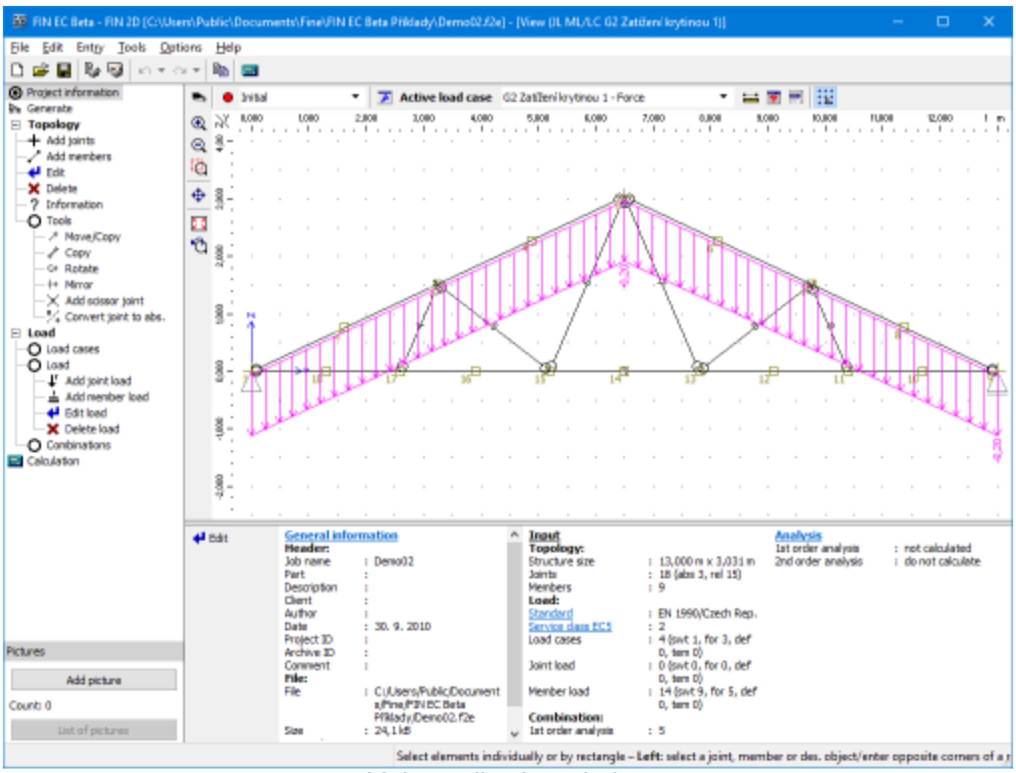

*Context menu for general place of the workspace*

## Main menu

The main menu contains complete range of available tools and functions, it may substitute the work with the tree menu. Additionally, the main menu contains part "**Tools**" with some useful tools.

## Input frame

The input frame is located in the bottom part of the window. The default view shows the general information about the project (number of members and joints, load cases etc.), status of analysis and also project details specified in the window "**Project information**", that may be used in the heading or footing of the documentation. This window can be opened with the help of the button "**Edit**".




*Main application window*

## Topology

The structure can be created in the program Fin 2D using these three ways:

- **Direct input of the structure** - The fundamental input of topology and loading. The topology is defined by the particular joints and members. Loads are organized into load cases and load combinations, it is possible to enter joint load or member load.
- **Generator of 2D structures** - The most common structures (trusses, attic roofs, frames) can be created easily with the help of the "**Generator of 2D structures**". This generator is able to create both the structure topology and the fundamental load. This tool is described in the chapter "**Generator of 2D structures**".
- **Import from **.dxf* file** - The structure topology may be also imported from **.dxf* in the window "**Import dxf**". This option is available in the main menu, part "**File**", "**Import**".

All these options may be combined (the basic part of the structure can be created in generator or imported from **.dxf*, the second part of the work can be done with the available tools).

## Input and editing methods

The methods of input and editing are described in chapters

- **Input element**
- **Edit elements**
- **Delete elements**

The program in the mode "**Tools**" of the tree menu contains additional tools for the manipulation with elements and structures (move, copy, rotate etc.).

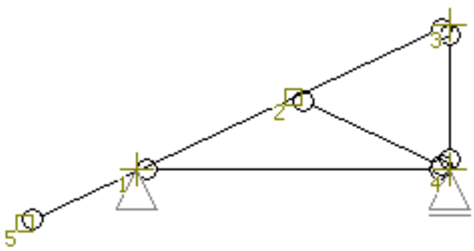
## Types of elements

The structures consist of two basic types of structural elements: joints and members.

Two different types of joints are included: "**absolute**" and "**relative**". Special type of relative joint is the "**scissor joint**", that can be specified in the intersection of two members.

**Absolute joints** are the joints, that have the position specified by the coordinates *[Y,Z]* in the global coordinate system. Their position can be changed only by editing the coordinates. Absolute joints are usually used for input of fundamental shape of the structure. These joints are marked with cross in the workspace.

**Relative joints** have the position specified relatively to the reference member. The position on member is given by the distance from the beginning or end joint of the member. The distance may be specified in length unit (metres) or in a proportional unit. The joints may be placed between beginning and end joints of the member or may lie outside this segment and extend the member length. The position has to be specified by a negative value in this case. The relative joints are used for connections of members to the point placed on the other member (typically connection of webs to chords).




*An overhang created by the relative joint 5 that is placed in front of the reference joint 1*

The position of the relative joint can be changed only in the direction of the local axis of the reference member. Relative joints are usually used in trusses for connections of webs to chords. These joints are marked with square in the workspace.

The differences between absolute and relative joints are also described in the chapter "**Conversion of relative joint to absolute**".

**Scissor joint** is a special type of relative joint, as it has two reference members. The position of such joint is given by the intersection of these reference members. This joint creates hinged connection of intersecting members.

**Member** is specified by start and end joints, cross-section, material and end conditions. The material and cross-sectional characteristics can be specified with the help of pre-defined databases, user input or by the import from programs "**Section**" and "**Sector**".

The member is connected to the reference joints by a connection, that consists of three components (two shifts and one rotation). Any of these components may be defined as free or spring.

Any member contains local coordinate system, that is used for the input of relative joints and load. The origin of this local coordinate system is the beginning of the member, local axis *1* is given by the member direction.

## Member type

Program uses two basic member types: "**Beam**" and "**Beam on elastic subsoil**". The "**Beam**" is the fundamental member type locally supported in joints, "**Beam on elastic subsoil**" is the member, that is supported along the whole length by a subsoil (e.g. foundations).

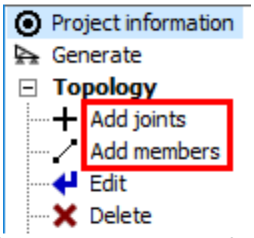
Members are also described in the theoretical chapter "**Structural elements**".

# Input elements

Joints may be inserted graphically in the workspace or with the help of the table in the bottom part of the window.

## Graphical input

Joints and members can be inserted directly by clicking in the workspace. The tree menu has to be switched into the mode "**Add joints**" or "**Add members**".

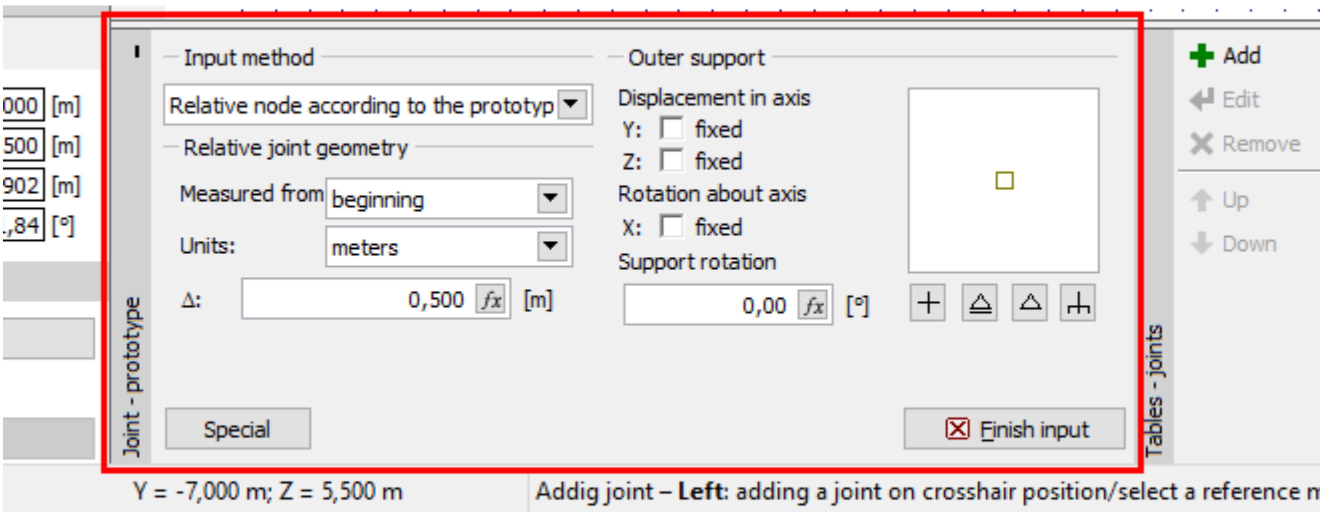



*Modes for graphical input of elements*

When adding **joints**, the window "**Joint prototype**" appears first. This window contains support properties and also input method for relative joints. Two methods are included:

- **Relative joint located by mouse click** - The position of joint will be determined by the position of cursor.
- **Relative joint according to the prototype** - The position of joint will be selected according to inputs "**Relative joint geometry**", which are located in the prototype frame.

The entered data in this window has to be confirmed by the button "**OK**". After that, the window is docked in the bottom frame.

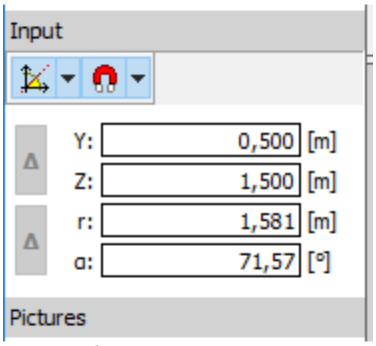



*Prototype properties anchored in the input frame*

The input is performed by clicking in the workspace. The snapping grid may be used for easier input, the properties of the grid are specified in the window "**Options**" (main menu part "**Tools**"). The snapping can be switched off (or switched on) temporarily when pressing the key "**Ctrl**" during the input. Program offers also snapping points like mid point of member or intersection of two members. Snapping points may be switched on and off in the list, that can be opened by the button "".

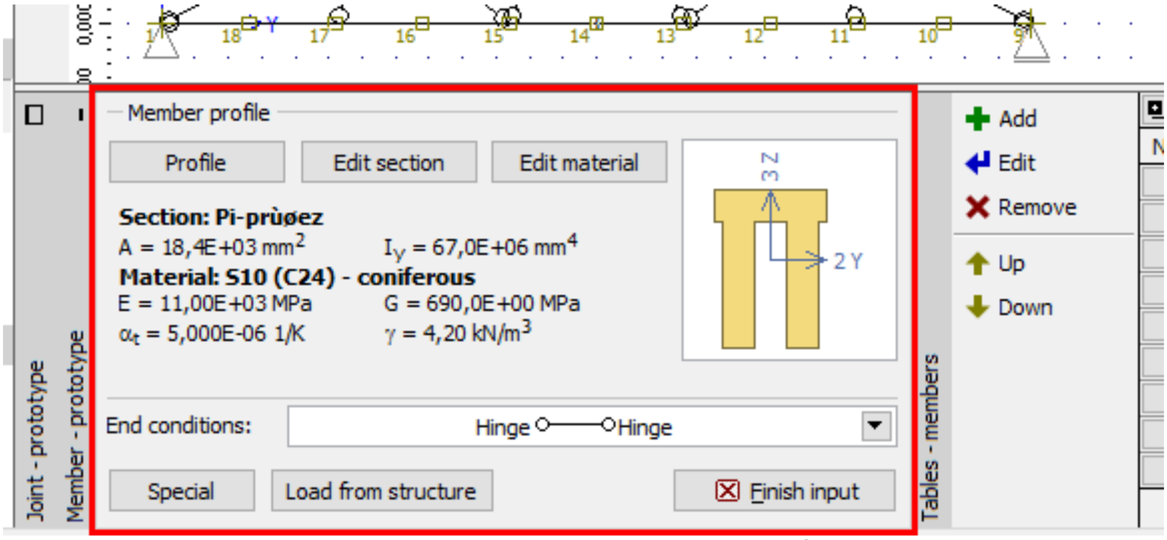
The input fields in the bottom part of the tree menu can be also used for the input of joints. It is possible to jump into input fields with the help of cursor or by keyboard entries "**y**" and "**z**". For example, the expression *y2z1.3* fills *2* into the input line "**y**" and *1.3* into the field "**z**".



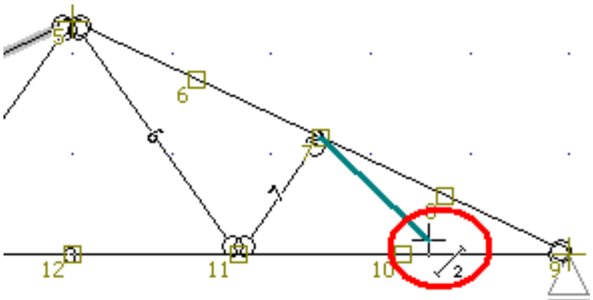
*Input fields in the tree menu*

The input of **members** is based on similar procedures. After the selection of an appropriate mode, the window "**Member prototype**" appears. This window contains properties (cross-section, material, end conditions etc.) that will be assigned to new members. The properties correspond to the parameters in the window "**Member properties**" (parameters may be copied from existing member using the tool "**Load from structure**"). Also the window "**Prototype of joint**", that contains the parameters of newly created joint (beginnings and ends of new members). The data in these windows has to be confirmed by the button "**OK**". After that, prototypes are anchored in the input frame, where the parameters may be changed.



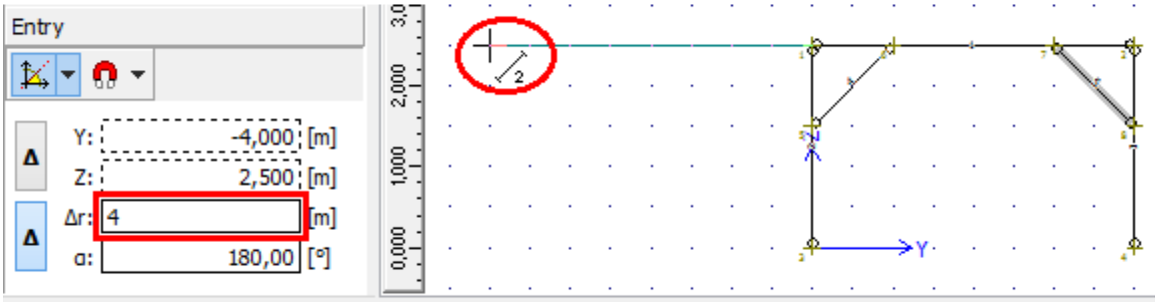
*Prototype properties anchored in the input frame*

The member input is done by clicking on the position of member beginning and end. The cursor shows number "**1**" in the mode for input of the member beginning and number "**2**" in the mode for input of the member end.


*Input of member in the workspace*

The member beginning can be entered using methods for joints. The end joint can be specified in a similar way. The easiest way is to specify the member direction by cursor and specify the length on keyboard. The length will be automatically filled into the field "**Δr**". The input has to be confirmed by the key "**Enter**". The program automatically snaps the cursor into directions *45°*. This behaviour may be changed (switch off or change it to 30°) using the button "".

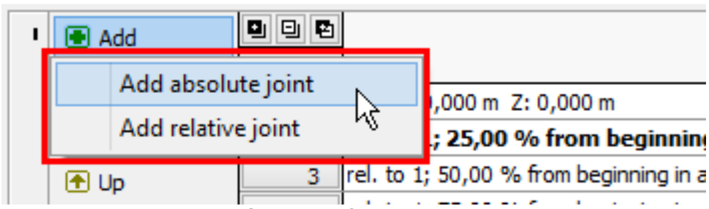



*Definition of member direction by the cursor and input of member length with the help of keyboard*

Alternatively, it is possible to use snapping to existing joints and snapping points (described above) or extended options of input fields in tree menu. Input fields "**Y**" and "**Z**" are usable for the definition of the end point with the help of global coordinates, fields "**r**" and "**α**" define the end point by the member length and rotation about the axis *y*. Buttons "**Δ**" changes whether the input is considered in the global coordinate system or relatively to the beginning of the member. It is possible to jump into input fields with the help of cursor or by keyboard entries "*y*", "*z*", "*r*" or "*a*". For example, the expression "*r2a15*" fills *2m* into the field "**r**" and 15° into the field "**α**".

## Input in tables

Joints and members may be also added with the help of the button "**Add**" in the toolbar on the left side of the tables, which appears in the bottom frame for the mode "**Topology**" of the tree menu. For the input of joints, one of available options ("**Add absolute point**" or "**Add relative point**") has to be selected first. Input is performed in the windows "**Properties of absolute joint**" and "**Properties of relative joint**". Input of relative joints isn't allowed until at least one member is specified.




*Choice of joint type*

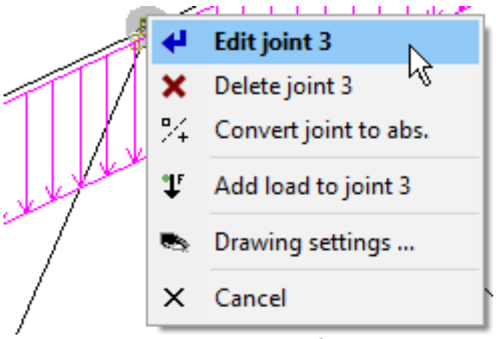
Members can be entered in a similar way. The window for input of new members is identical to "**Member properties**", the reference joints, cross-section and material have to be specified there.

# Edit elements

Joints and members may be modified individually or in a batch. Editing is available only in preprocessor (parts "**Topology**" and "**Load**" of the tree menu).

## Editing in the workspace

The individual joint or member can be edited by double-click on the element in the workspace. The editing of properties is done in the windows "**Properties of absolute joint**" or "**Properties of relative joint**" for joints or in the window "**Member properties**" for members. An alternative way is to use a command "**Edit joint**" or "**Edit member**" in a context menu, that can be opened by right mouse button in the workspace. The same procedure can be used also for selected elements (highlighted by green in the workspace).




*The context menu for joint*

The appropriate window for editing an element can be also opened using double-click on an appropriate row in tables of joints and members in the bottom frame. Alternatively, the button "**Edit**" on the left side of the table can be used.

## Graphical mode "Edit"

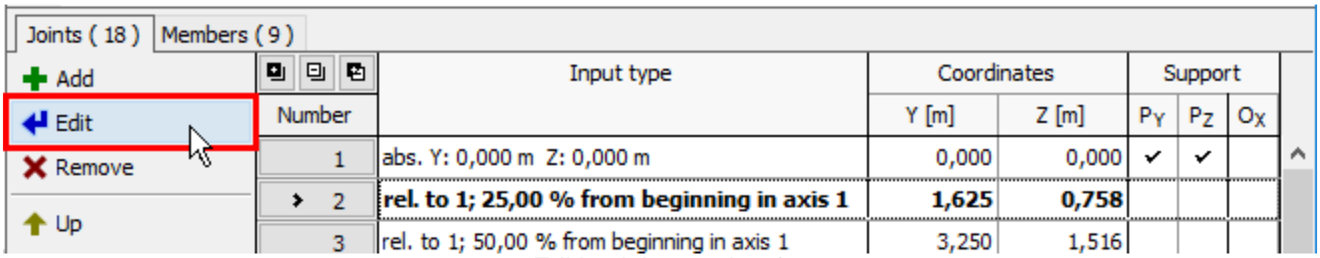
The tree menu has to be switched into the mode "**Topology**" "**Edit**" and after that, it is possible to edit joints and members by clicking on the appropriate element in the workspace. The software also considers close surrounding of the element for a correct click. The considered surrounding is indicated by the change of the cursor appearance.

*Appearance of cursor for joint edit*

The mode can be terminated by selection of other mode or by right mouse button click.

## Editing joints and members in the table

Joints and members may be also edited with the help of the button "**Edit**" in the toolbar on the left side of the table. The program modifies the active element (highlighted by bold font and mark "**>**" in front of the joint number).

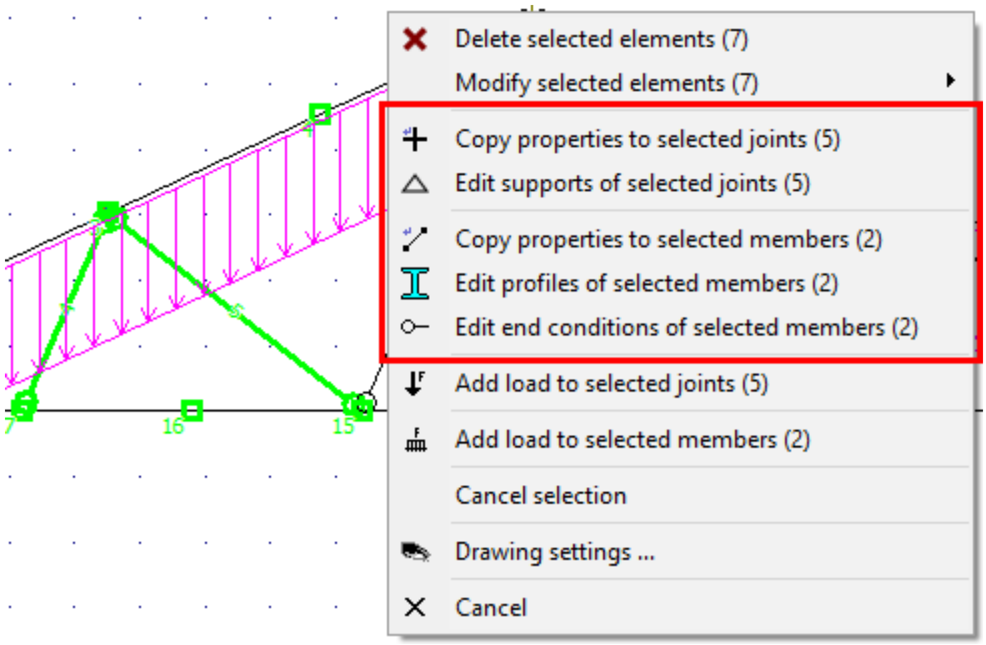



*Editing joint number 2*

An alternative way is to use the command "**Edit**" in the context menu for the appropriate table row.

## Editing selected joints and members

Joints and members can be also selected and modified in a batch. These elements can be selected in the workspace or in tables in the bottom frame. Selected joints are highlighted by green in the workspace and by blue and bold font in the table. Tools for batch edit are available in the context menu, that can be opened by right mouse button click in the workspace (provided that some joints or members are selected).




*Tools for batch edit of selected elements*

Following tools may be used for batch editing of joints:

| | |
|---|---|
| **Copy properties** | • Copy of specified joint properties to selected joints |
| **Edit supports** | • Batch edit of supports for selected joints |

Following tools may be used for batch editing of members:

| | |
|---|---|
| **Copy properties** | • Copy of specified member properties to selected members |

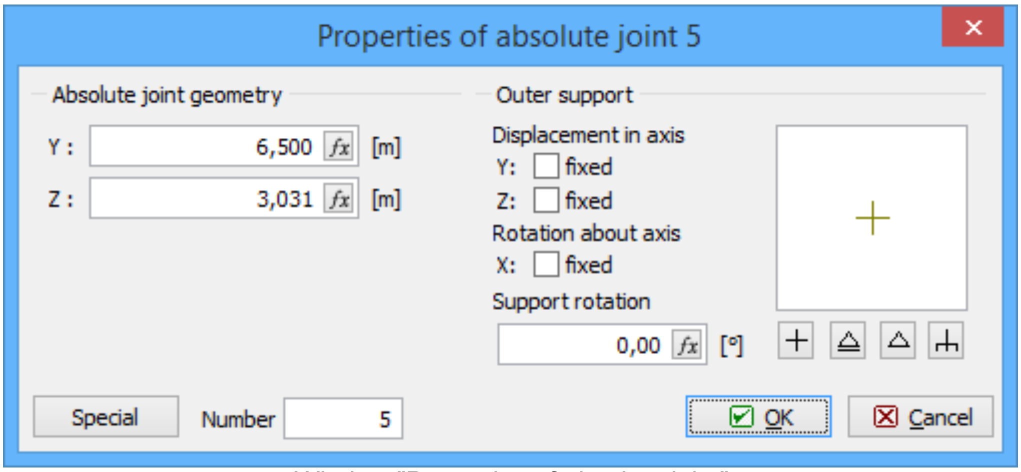
**Edit profiles**
- This tool is able to modify cross-section and material of selected member. The range of available properties is equal to the range described in the chapter "**Edit profile**".

**Edit end conditions**
- Batch edit of end conditions for selected joints

## Properties of absolute joint

The position and supporting style may be changed for absolute joint. The position of the absolute joint is given by the coordinates *[Y,Z]* in the global coordinate system. The accurate values of coordinates may be calculated with the help of the built-in calculator.

The right part "**Outer support**" contains the properties of the support. These properties are described in the chapter "**Supports**". The spring supports have to be specified in the window **"Special properties of joints**", that can be opened by the button "**Special**" in the left bottom corner.

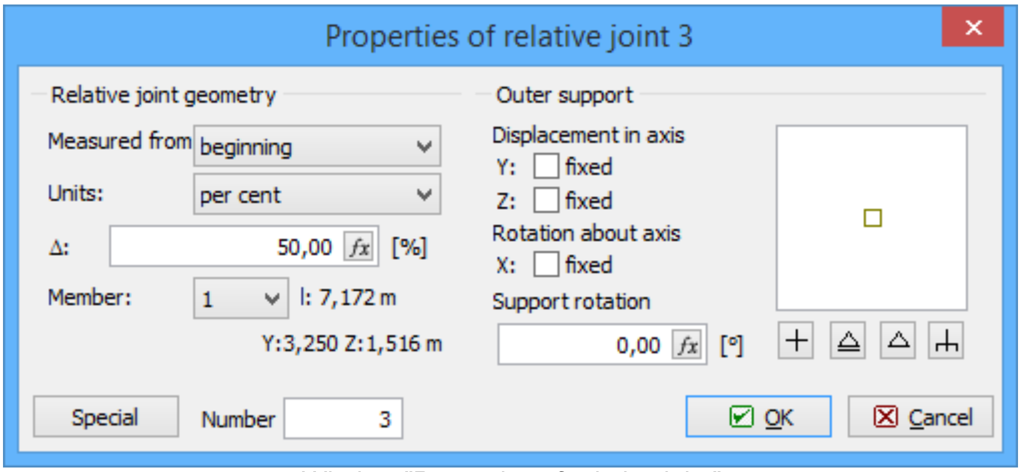



*Window "Properties of absolute joint"*

## Properties of relative joint

The position and supporting style may be changed for relative joint. The position of the relative joint is given by the number of reference member, by the position in metres or per cents and by reference point (beginning or end of the member).

The right part "**Outer support**" contains the properties of the support. These properties are described in the chapter "**Supports**". The spring supports have to be specified in the window **"Special properties of joints**", that can be opened by the button "**Special**" in the left bottom corner.




*Window "Properties of relative joint"*

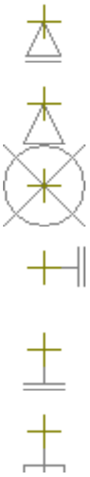
## Supports

The joint support consists of three components: prevented displacements in directions of global axes *Y* and *Z* and prevented rotation about axis *X*. The support in the certain direction can be switch on by the corresponding check box "**fixed**". Following symbols are used in the workspace for different types of supports:

- The joint isn't supported in any direction

- The joint is supported in the direction of the axis *Y* (sliding support with sliding in vertical direction)



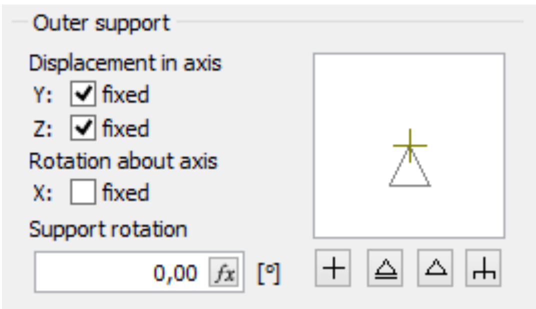
- The joint is supported in the direction of the axis $Z$ (sliding support with sliding in horizontal direction)
- The joint is supported in the directions of the axes $Y$ and $Z$ (hinge)
- The rotation in the joint prevented
- The joint is supported in the direction of the axis $Y$ and the rotation is prevented (fixed support with permitted sliding in the vertical direction)
- The joint is supported in the direction of the axis $Z$ and the rotation is prevented (fixed support with permitted sliding in the horizontal direction)
- Fixed support (rotation and displacements in both directions prevented)

The most common support types can be entered with the help of dedicated buttons in the right bottom corner of the frame.

The support properties also contain the input line "**Support rotation**". This option is useful for cases, where the reactions in the support has to be obtained in the rotated coordinate system or where the structure is supported in certain direction that isn't parallel to main axes.

The button "**Special**" opens the window "**Special properties of joint**", where the spring supports may be defined.

The support styles are also described in the theoretical chapter "**Supports**".



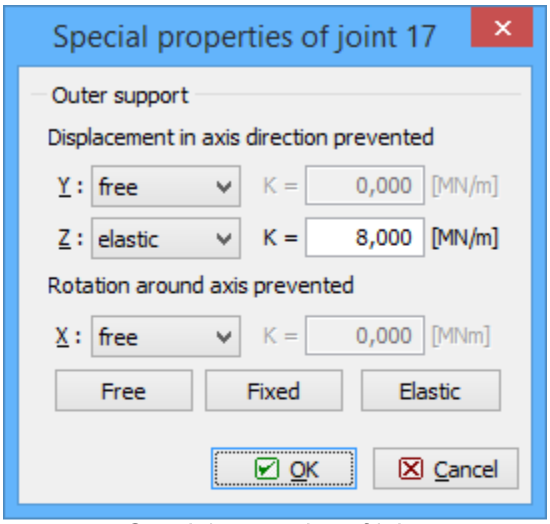

*Support properties*

## Special properties of joint

This window contains extended properties of joint support. It brings an option to specify a spring support in any direction. The spring constant $K$ has to be specified in this case.

The buttons "**Free**", "**Fixed**" and "**Elastic**" will assign corresponding support method to all directions.

The support styles are also described in the theoretical chapter "**Supports**".




*Special properties of joint*

## Member properties

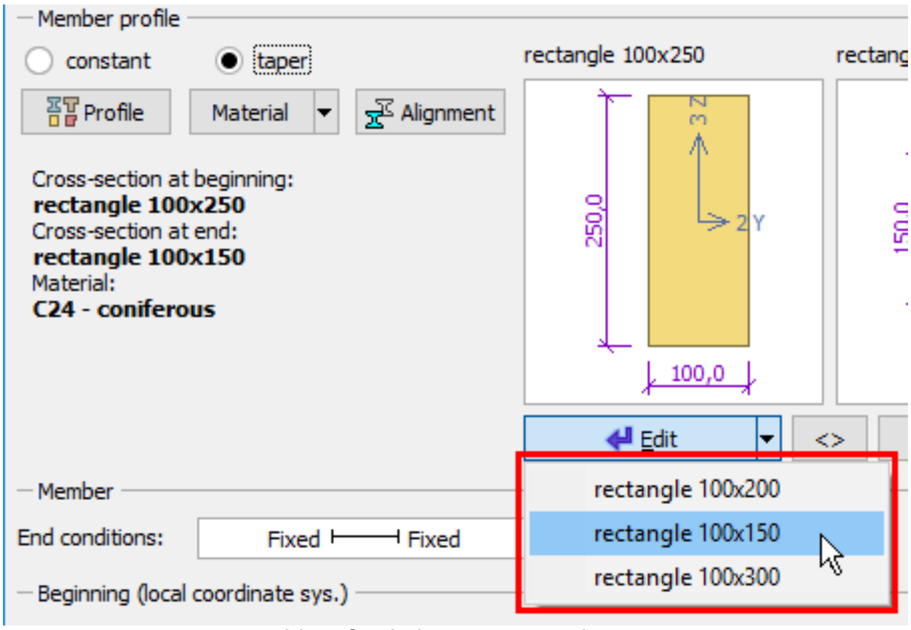
This window contains basic properties of member: beginning and end joint, cross-section, material and end conditions.

The button "**Special**" in the left bottom corner launches the window **"Special properties of member**", that contains advanced properties (subsoil parameters, excluded tension or compression, spring end connections).

The position of the member is given by the numbers of beginning and end joints. These joints can be changed with the help of the list box or by the direct input of the number on the keyboard. The button "**<>**" changes the member orientation (switches the beginning and end joint), the positions of relative joints and loads are preserved.

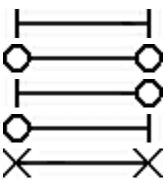
The frame "**Member profile**" contains properties of cross-section and material. The member may have identical cross-section along the whole length (option "**constant**") or the cross-section vary along the member length (the option "**taper**"). The cross-section and material can be changed in the window "**Edit profile**" which can be launched by the button "**Profile**". Parameters of the existing cross-section (e.g. dimensions) or material can be changed in the windows "**Cross-section editor**", (the button "**Edit**" under the cross-section view) or "**Materials catalogue**" (the button "**Material**"). Members with variable cross-section have two buttons for cross-section edit, the first one for member beginning and the second one for member end. Buttons for edit of cross-sections and materials contain drop down menu with existing cross-sections and materials. These list can be also used for fast input of these properties.




*List of existing cross-sections*

The visual appearance of the structure can be modified by an alignment of the member which can be specified in a dedicated window using the button "**Alignment**". This alignment defines the position of member mass relatively to reference line of the member. It affects only structure view not the analysis.

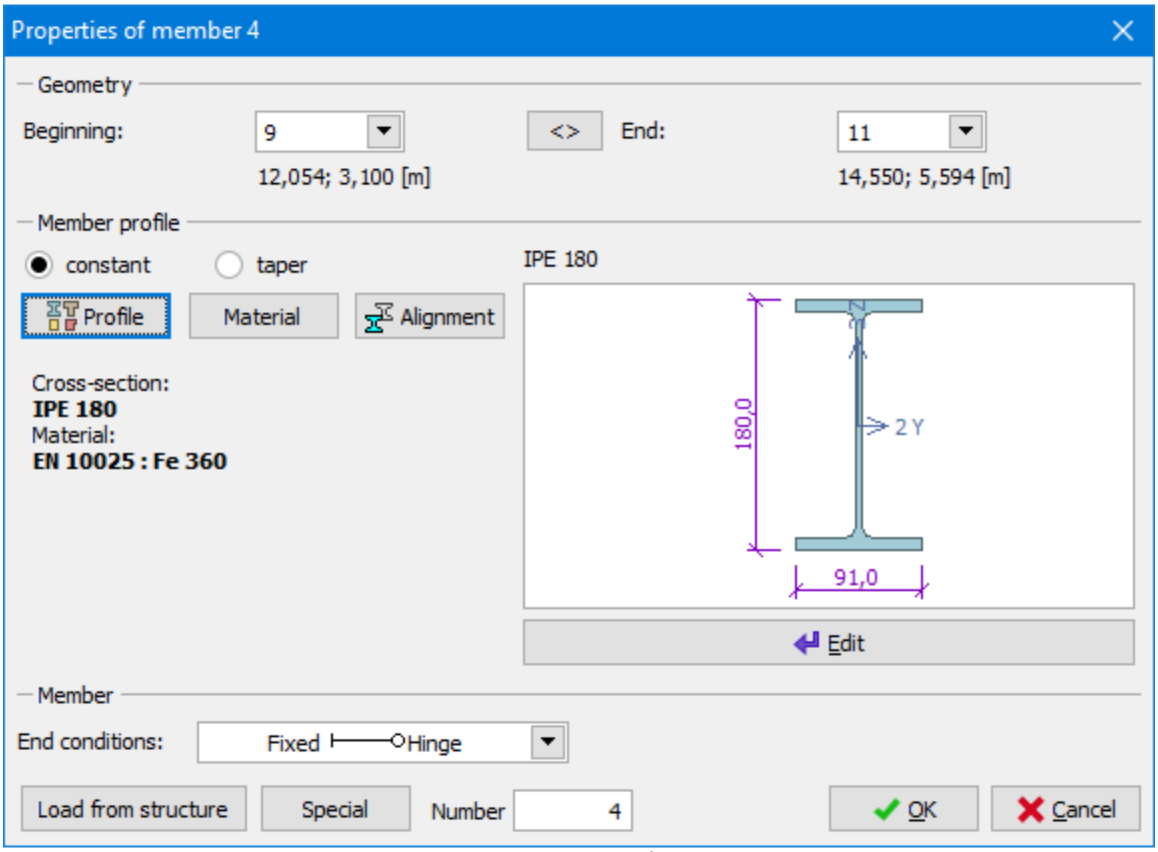
The frame "**Member**" contains the choice of end conditions (pinned, fixed):



- The member is fixed on both ends
- The member is pinned on both ends
- The member has fixed beginning and pinned end
- The member has pinned beginning and fixed end
- The member has special conditions (e.g. spring connections) at the beginning and end. The properties are organized in the window "**Special properties of member**".

Parameters (cross-section, material, end conditions) may be copied from existing member using the tool "**Load from structure**".

End conditions are also described in the theoretical chapter "**End conditions**".




*Window "Properties of member"*

# Special properties of member

This window contains advanced properties of member

## Member type

Program uses two basic member types:

**Beam** • Fundamental member type supported in joints.

**Beam on elastic subsoil** • The member, that is supported along the whole length by a subsoil (spring support).

Additionally, it is possible to exclude compressive or tensile stresses in the member.

## Subsoil parameters

The subsoil is defined by Winkler-Pasternak constants $C_1$ and $C_2$. The constants $C_1$ and $C_2$ may be calculated from the general parameters of subsoil in the window "**Calculation of C1 and C2**". The subsoil acts as general spring, that acts both in compression and tension. As this model usually does not correspond to the real conditions, the appearance of tensile stress should be checked.

The setting "**Use member width**" uses the cross-section width as the width of member contact with subsoil. If not used, the contact width $b$ may be specified manually.

The default assumption is, that the subsoil acts in the direction of the local axis *3* (against gravity). The setting "**Acts in axis *3***" is able to change this behaviour.

The shear effects at the member ends may be taken into the account with the help of settings "**In front of member**" and "**Behind member**".

The subsoil properties are described in the theoretical chapter "**Subsoil model**".

## Shear effect

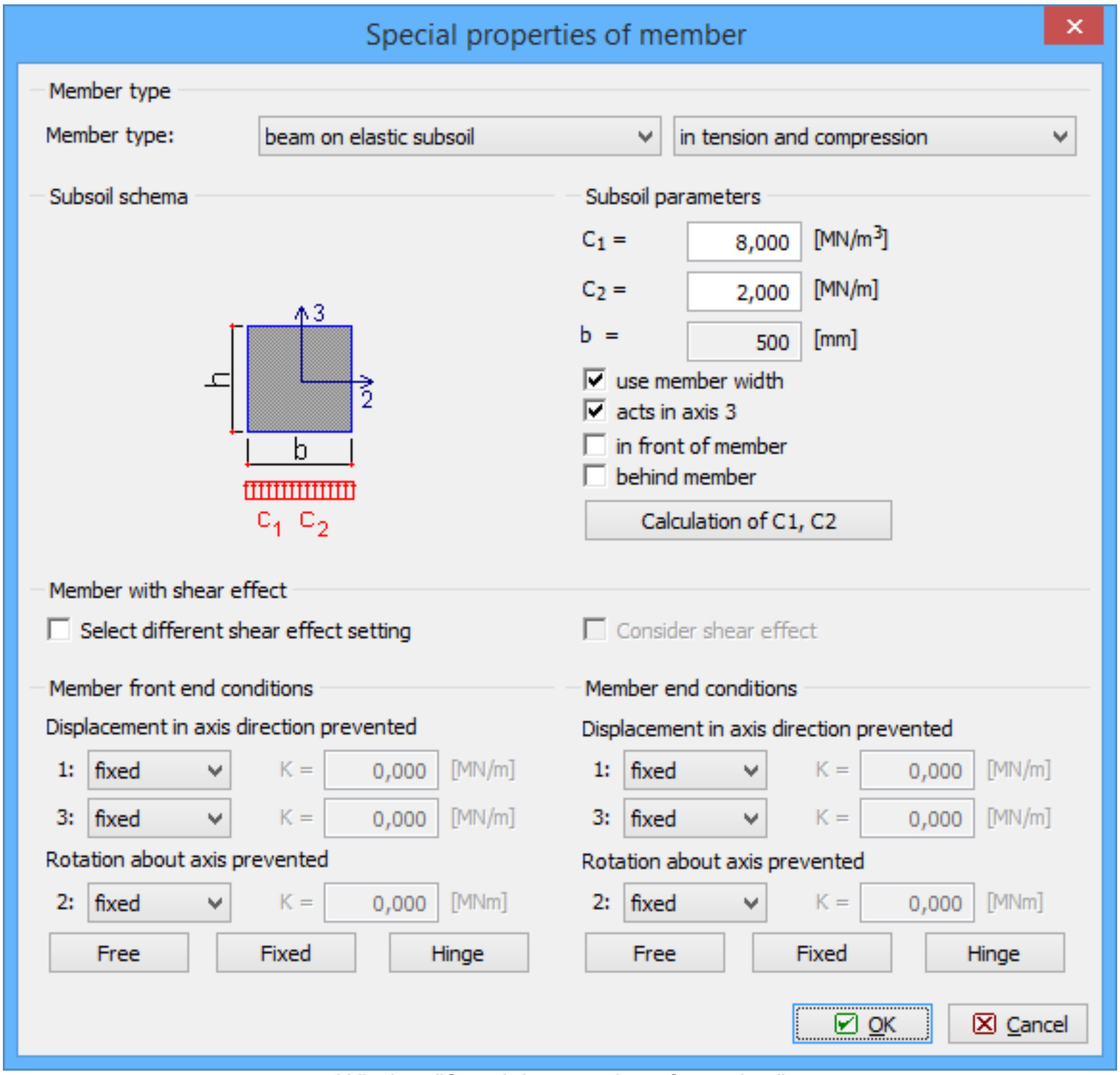
The part "**Member with shear effect**" may rewrite the global settings regarding the consideration of shear effect on deformations. This theoretical model with shear consideration is recommended cases, where member length isn't significantly longer than cross-section dimensions. The theoretical background is described in the chapter "**Special member characteristics**".

## Member end conditions

This frame contains extended options for end conditions. It brings an option to specify a spring connections in any direction (not only fixed/free). The spring constant $K$ has to be specified in this case. The buttons "**Free**", "**Fixed**" and "**Elastic**" will create corresponding end condition. End conditions are also described in the theoretical chapter "**End**

**conditions**".




*Window "Special properties of member"*

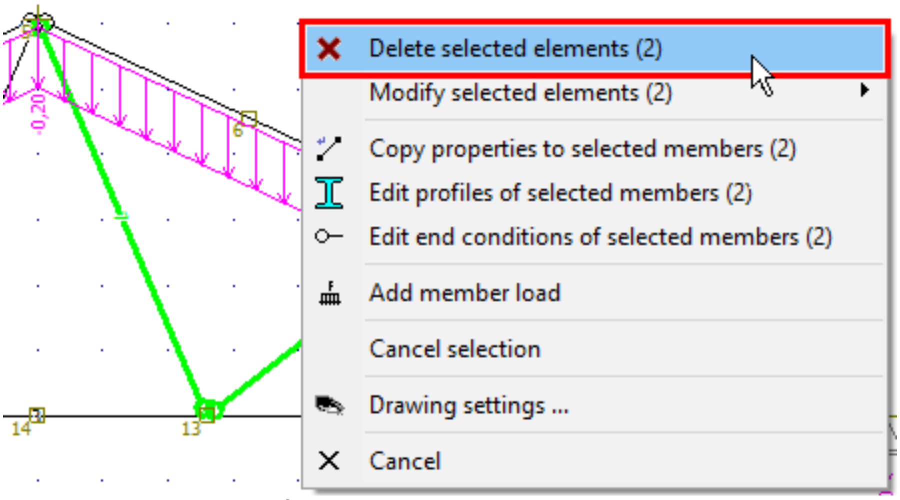
# Delete elements

Joints and members may be deleted individually or in a batch, using the table in the bottom part of the window or graphical mode for workspace. Any deletion of joint removes also all members connected into this joint. Any deletion of member removes also all relative joints on the member and members connected into these joints.

## Using context menu

Any joint or member can be deleted using command "**Delete joint**" or "**Delete member**" in a context menu, that can be opened by right mouse button in the workspace. The same procedure can be used also for selected elements (highlighted by green in the workspace).

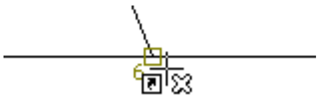
 




*Deletion of selected member using context menu*

## Graphical mode "Delete"

The tree menu has to be switched into the mode "**Topology**" "**Delete**" and after that, it is possible to delete joints or members by clicking on the appropriate element in the workspace. The software also considers close surrounding of the element for a correct click. The considered surrounding is indicated by the change of the cursor appearance.

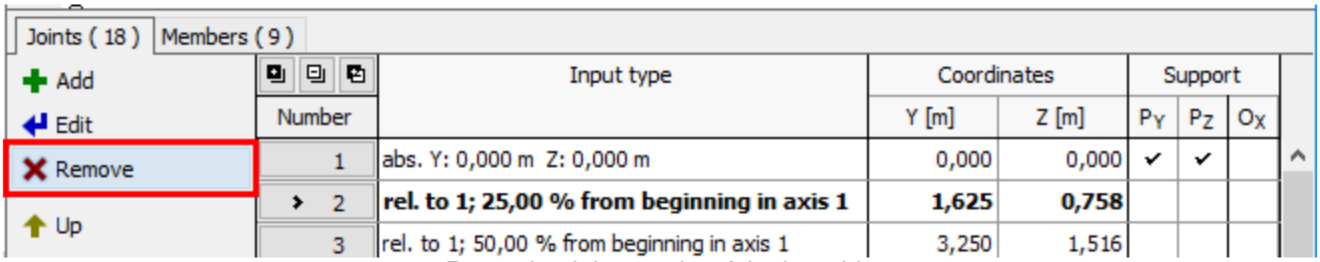


*Appearance of cursor for deletion*

The mode can be terminated by selection of other mode or by right mouse button click.

## Removing elements in the table

Joints and members may be also deleted with the help of the button "**Remove**" in the toolbar on the left side of the table. The program deletes the active element (highlighted by bold font and mark "**>**" in front of the joint number) or selected ones (highlighted by blue in the table).




| Number | Input type | Coordinates Y [m] | Z [m] | Support Py | Pz | Ox |
|---|---|---|---|---|---|---|
| 1 | abs. Y: 0,000 m Z: 0,000 m | 0,000 | 0,000 | ✔ | ✔ | |
| > 2 | **rel. to 1; 25,00 % from beginning in axis 1** | **1,625** | **0,758** | | | |
| 3 | rel. to 1; 50,00 % from beginning in axis 1 | 3,250 | 1,516 | | | |

*Removing joint number 2 in the table*

# Tools

The mode "**Tools**" contains functions and commands that can be used for editing created structures.

## Tools

These tools can be used both with complete structure and selected parts. Most of the tools can be used in two modes: as a simple transformation (change of position or shape) or as a copy tool (keeps original structure). Following tools are included:

| | |
|---|---|
| **Move** | • The tool that moves the structure in given direction |
| **Copy** | • The tool that copies the structure in given direction |
| **Enlarge/Shrink** | • The tool that enlarges or shrinks the structure |
| **Rotate** | • The tool that rotates or copies the structure using specified angle |
| **Mirror** | • The tool that mirrors existing structure, including copy option |
| **Align** | • The tool that aligns structural elements (joints, members) into given line |

### Joints

**Add scissor joint**

- Inserts a scissor joint. The scissor joint is a special type of relative joint, as it has two reference members. The position of such joint is given by the intersection of these reference members. This joint creates hinged connection of intersecting members.

**Convert joint to absolute**

- Converts a relative joint to an absolute one.

**Absolute joints on members**

- Test that checks the coordinates of all absolute joints and compare them with positions of members. Absolute joints lying on members aren't connected to the members and may cause the singularity of the structure. Such joints may be converted into relative ones. This conversion automatically creates a connection between member and joint.

**Absolute joints on members convert to relative**

- A tool that finds absolute joints placed on members or in their surroundings and converts these joints into relative ones. The considered surroundings can be defined by user. The tool can be applied both to all and selected joints.

### Members

**Divide members**

- Tool that adds specified number of relative joints with uniform distribution along the member length.

**Continuity analysis**

- Test that checks whether the structure is divided into more parts or not. Hidden division into more parts caused by overlapping joints or members causes collapses during analysis very often. The partial segments of the structure aren't usually supported in a sufficient way and the singularity appears in these cases. There is an option to highlight the certain part of the structure.

### Load

**Member load multiplication**

- This tool increases or reduces member loads in load cases with the help of specified multiplication factor. The factor may be applied to all or selected loads, the tool may be limited to active or selected load cases. This tool is suitable e.g. for modification of input after the change of loading width of structural elements.

**Joint load multiplication**

- This tool increases or reduces joint loads in load cases with the help of specified multiplication factor. The factor may be applied to all or selected loads, the tool may be limited to active or selected load cases. This tool is suitable e.g. for modification of input after the change of loading width of structural elements.

### Load cases and combinations

**Load template**

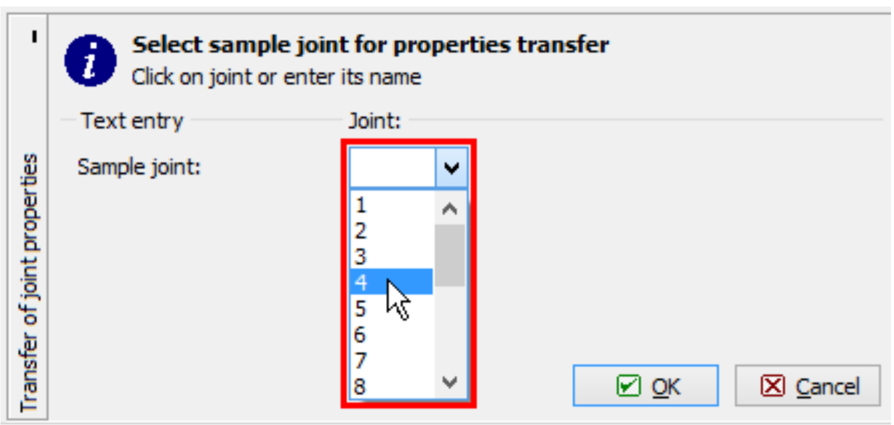
- The import of template (list) of load cases and combinations. This tool may be used for an easy transfer of load case/combination parameters from one project to another. The file of the template has an extension **.flc*.

**Save template**

- The export of template (list) of load cases and combinations. This tool may be used for an easy transfer of load case/combination parameters from one project to another. The file of the template has an extension **.flc*.

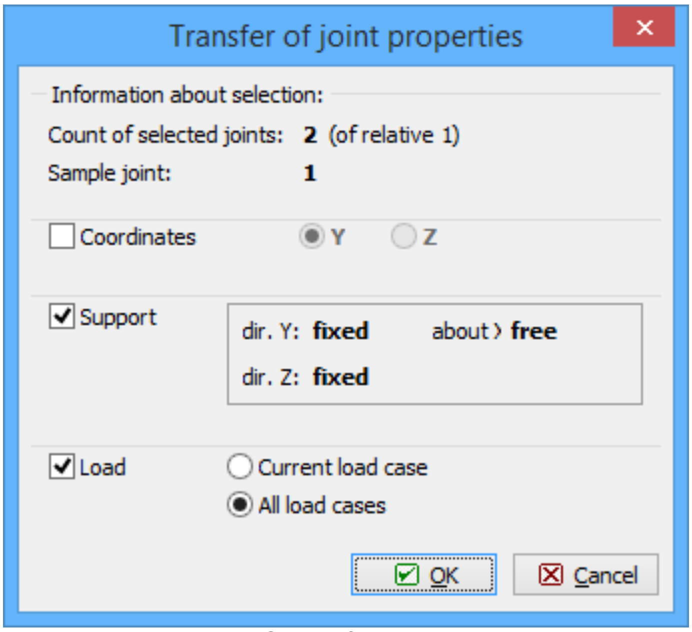
## Copy properties

This tool can be used for copying the joint properties to another joints. This tool is enabled if at least one joint in the structure is selected (highlighted by green in the workspace). The option for the input of sample joint appears after the choice of the mode "**Joints**" "**Selected**" "**Copy properties**" in the tree menu. The properties of this joint will be copied to selected joints. The choice of the sample joint has to be confirmed by the button "**OK**". The sample joint can be also selected graphically by clicking in the workspace.




*Choice of sample joint*

The window, that appears after the choice of the sample joint, contains the options to specify properties, that will be assigned to selected joints. It is possible to copy one of coordinates (joints will be aligned into horizontal or vertical line), support style and load in active or all load cases.

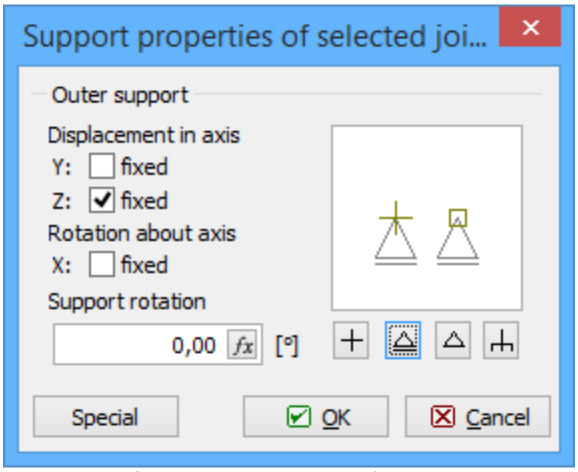



*Window "Copy of joint properties"*

## Edit supports

This tool may be used for batch edit of supports for selected joints. This tool is enabled if at least one joint in the structure is selected (highlighted by green in the workspace). The window "**Support properties of selected joints**" appears after the choice of the mode "**Joints**" "**Selected**" "**Edit supports**" in the tree menu. The support properties that will be assigned to all selected joints can be specified in this window.

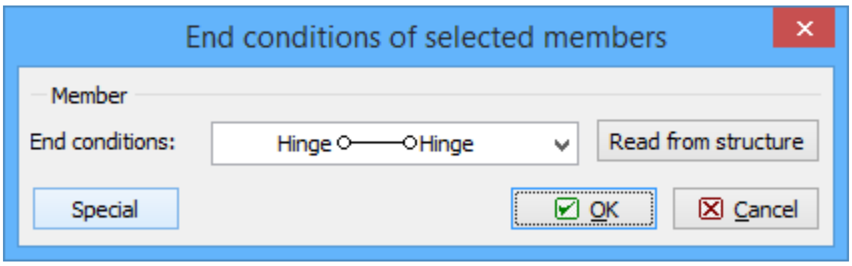
The support styles are also described in the chapter "**Supports**".




*Window "Support properties of selected joints"*

## Edit end conditions

This tool may be used for batch edit of end conditions for selected members. This tool is enabled if at least one member in the structure is selected (highlighted by green in the workspace). The window "**End conditions of selected members**" appears after the choice of the mode "**Members**" "**Selected**" "**Edit end conditions**" in the tree menu. The end conditions that will be assigned to all selected members can be specified in this window. Parameters may be copied from existing member using the tool "**Load from structure**".



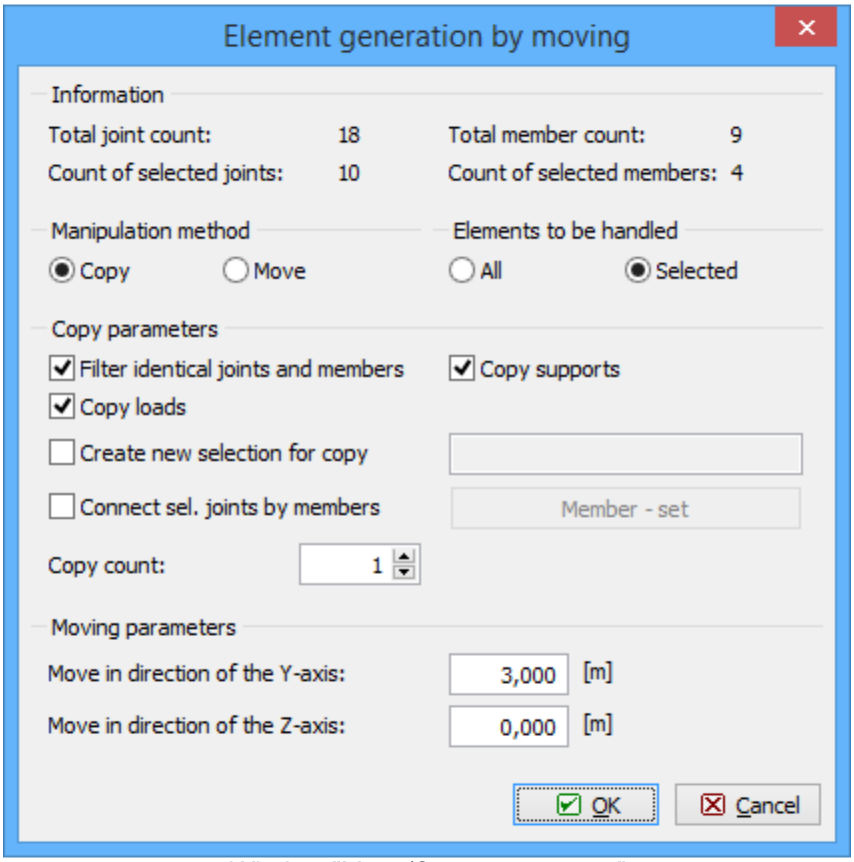
 

*Window for change of end conditions*

The spring end conditions have to be specified with the help of the button "**Special**" in the left bottom corner. The parameters correspond to the content of the window "**Special properties of member**".

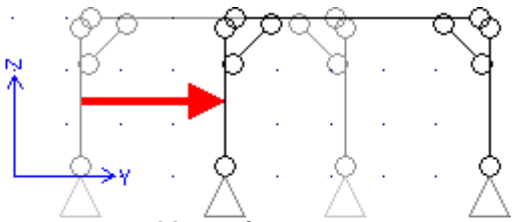
## Move/Copy

The tool "**Move/Copy**" can be used for shift or copy of structure (or its part). The window contains parameters, that affects the final behaviour of the tool.




*Window "Move/Copy parameters"*

The "**Manipulation method**" sets, whether the tool will only change the position of the structure (or its part) or will keep existing structure and create a new copy. The most of following settings are disabled for the option "**Move**" (only movement vector can be specified).

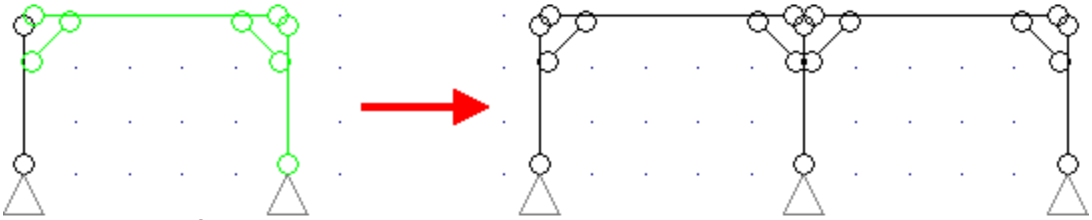



*Move of structure*

This tool is able to move or copy whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

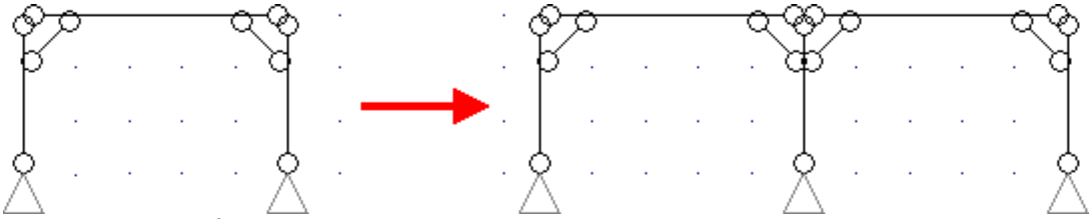
The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. Following example shows the behaviour. If this setting is switched off, it is necessary to copy only selected members (highlighted by green) in this structure. Otherwise, two frames (old one and new one) without any connection would be created.



*Copy without the setting "Filter identical joints and members"*

The setting "**Filter identical joints and members**" treats these problems. If switched on, the overlapping elements are checked and the frame can be copied as a whole structure. The overlapping column in the middle will be deleted, new frame and the old one will be connected into one structure.



*Copy with the setting "Filter identical joints and members"*

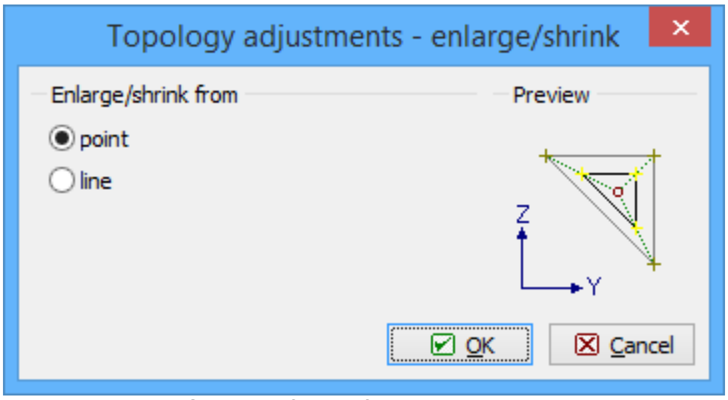
The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "" in the toolbar above the workspace.

The next entry is the number of copies. The bottom part of the window contains the vector of transformation divided into two components according to the global axes $Y$ and $Z$.

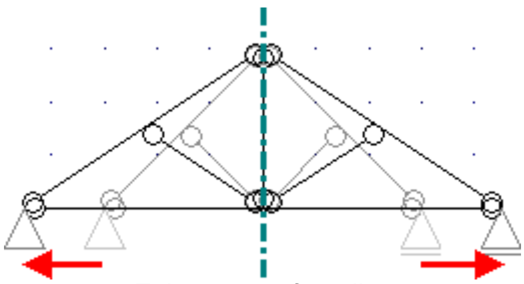
## Enlarge/Shrink

This tool can be used for enlargement or shrinkage of structure (or its part). There are two main modes of this tool: transformation relatively to point or line. The choice of the mode has to be done in the window, that appears after clicking on the tool in the tree menu.




*Choice of transformation mode*

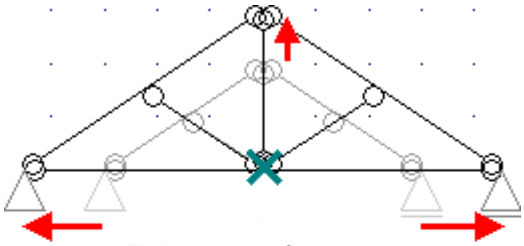
The transformation from line enlarges or shrinks the structure only in the direction perpendicular to the given line. The structure size in the direction of the line is not affected. The shape of the new structure differs from the shape of the original structure.



*Enlargement from line*

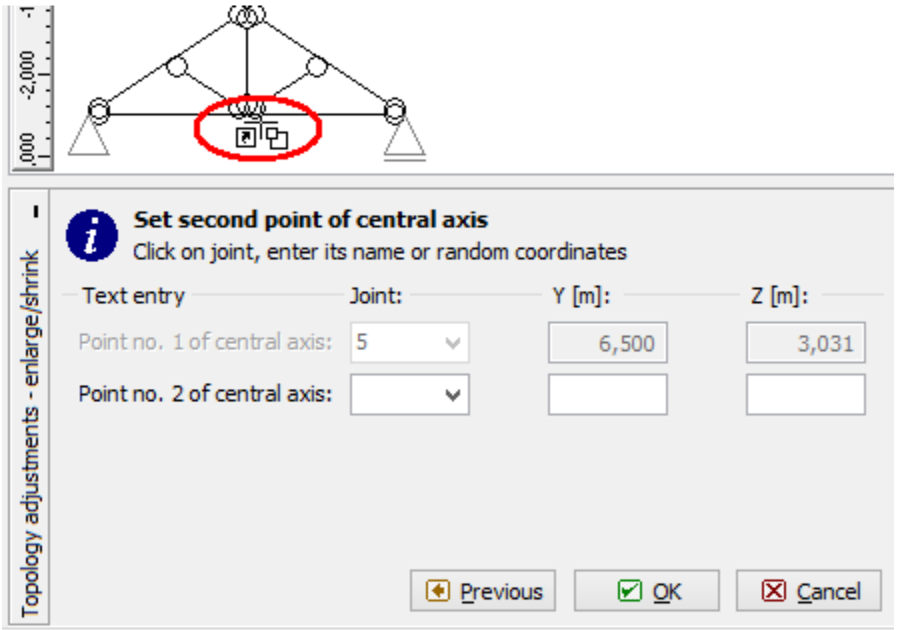
 

The transformation from point enlarges or shrinks the structure in all directions. The shape of the new structure is identical to the shape of the original structure.



*Enlargement from point*

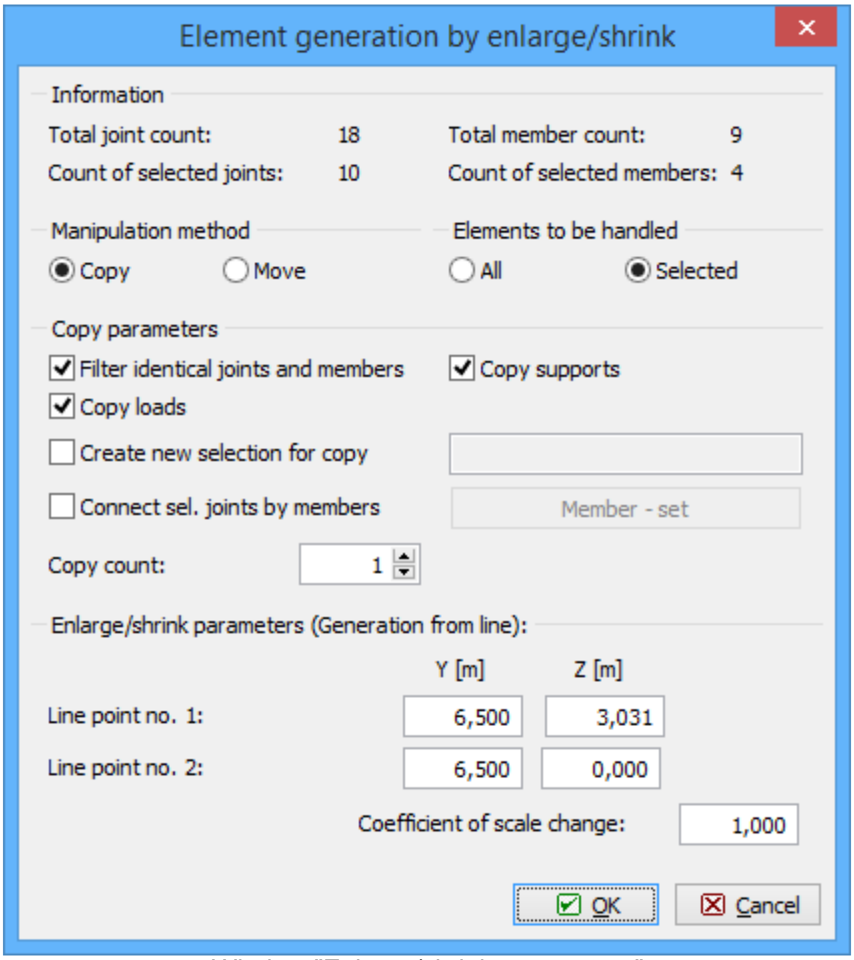
The input of reference point or line follows. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.




*Input of reference line in the workspace*

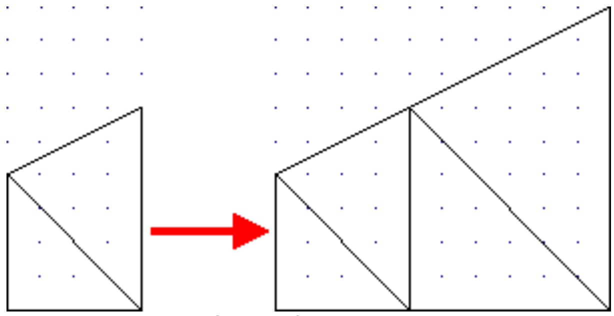
After that, the window with parameters of transformation appears.




*Window "Enlarge/shrink parameters"*

The "**Manipulation method**" sets, whether the tool will only change the shape of the structure (or its part) or will keep existing structure and create a new modified copy. The most of following settings are disabled for the option "**Move**" (only transformation parameters can be specified).



*Copy of structure*

This tool is able to enlarge or shrink whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. This setting is recommended. Otherwise, two structures (old one and new one) without any connection may be created.

The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

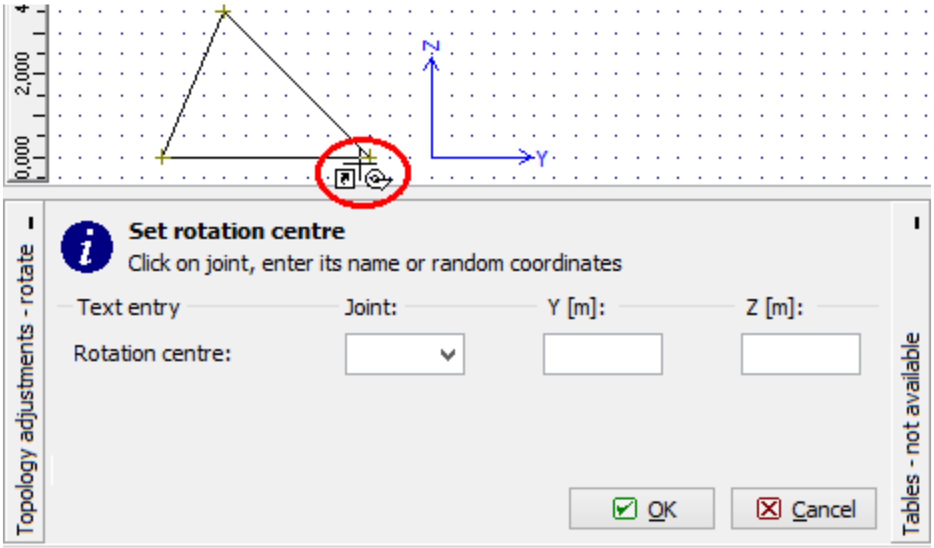
The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "" in the toolbar above the workspace.

The next entry is the number of copies. The bottom part of the window contains the reference point/line coordinates and scale factor. Value greater than *1.0* enlarges the structure, smaller value reduces structure.

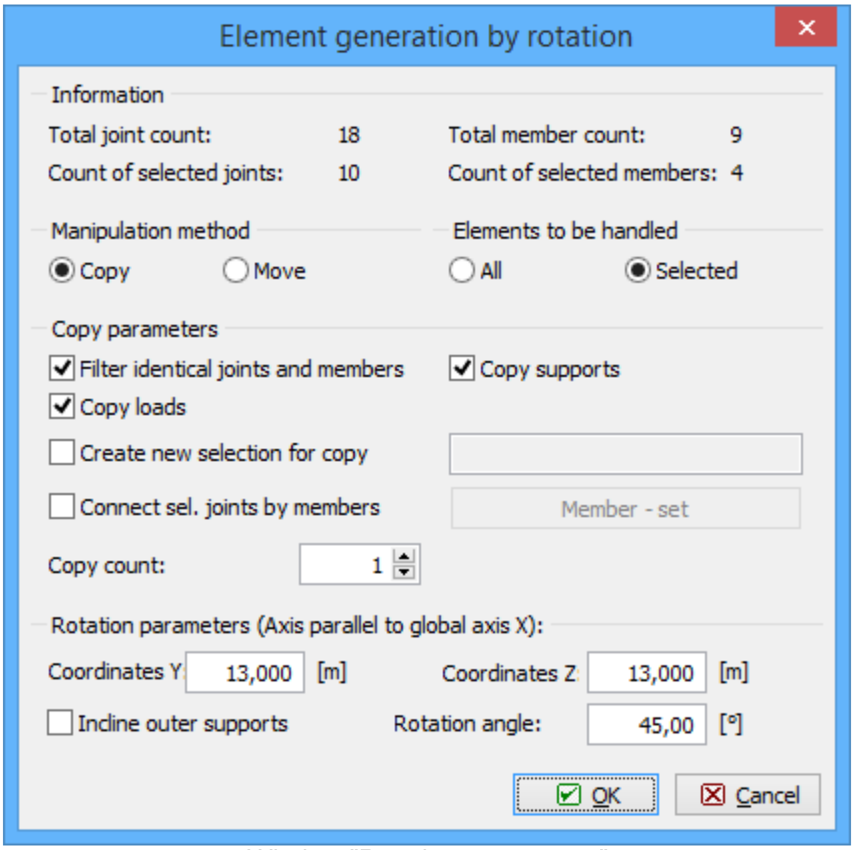
# Rotate

This tool can be used for rotation of structure (or its part). The rotation centre has to be specified first. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.



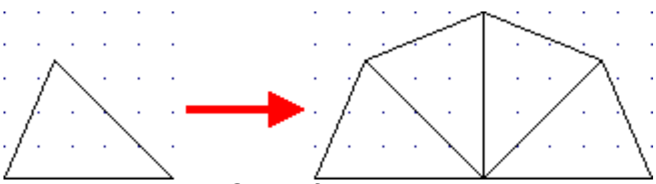

*Input of rotation centre in the workspace*

After that, the window with parameters of transformation appears.




*Window "Rotation parameters"*

The "**Manipulation method**" sets, whether the tool will only change the rotation of the structure (or its part) or will keep existing structure and create a new rotated copy (or copies). The most of following settings are disabled for the option "**Move**" (only transformation parameters can be specified).

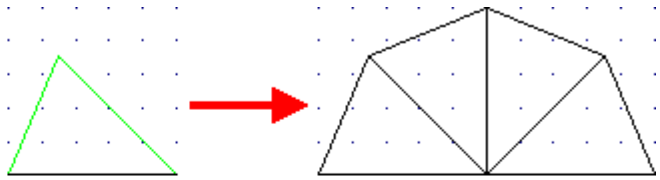


*Copy of structure*

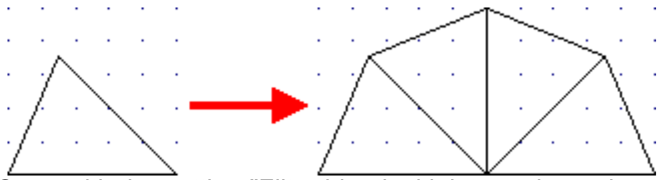
This tool is able to rotate or copy whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. Following example shows the behaviour. If this setting is switched off, it is necessary to copy only selected members (highlighted by green) in this structure. Otherwise, four parts without any connection would be created.



*Copy without the setting "Filter identical joints and members"*

The setting "**Filter identical joints and members**" treats these problems. If switched on, the overlapping elements are checked and the frame can be copied as a whole structure. The overlapping rays will be deleted, new frames and the old one will be connected into one structure.



*Copy with the setting "Filter identical joints and members"*

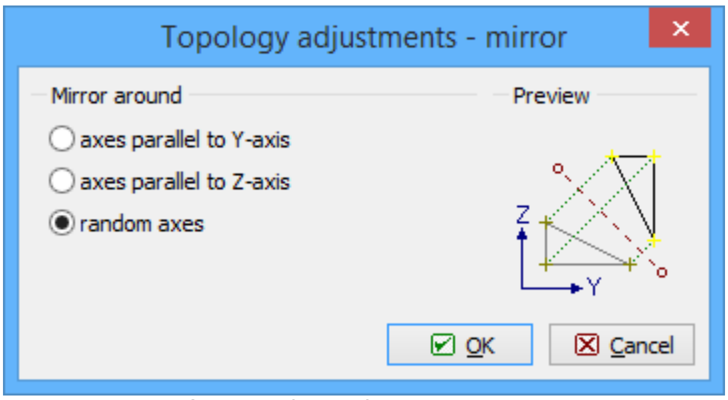
The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "" in the toolbar above the workspace.

The next entry is the number of copies. The bottom part of the window contains the rotation centre coordinates and the rotation angle. Positive value means rotation in anti-clockwise direction.

## Mirror

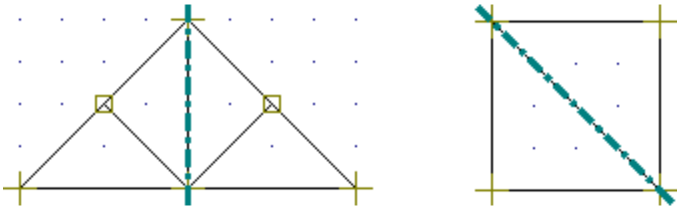
This tool can be used for mirror of structure (or its part). There are three main modes of this tool: mirror using axis parallel to *Y* or *Z* axis and using general axis. The choice of the mode has to be done in the window, that appears after clicking on the tool in the tree menu.




*Choice of transformation mode*

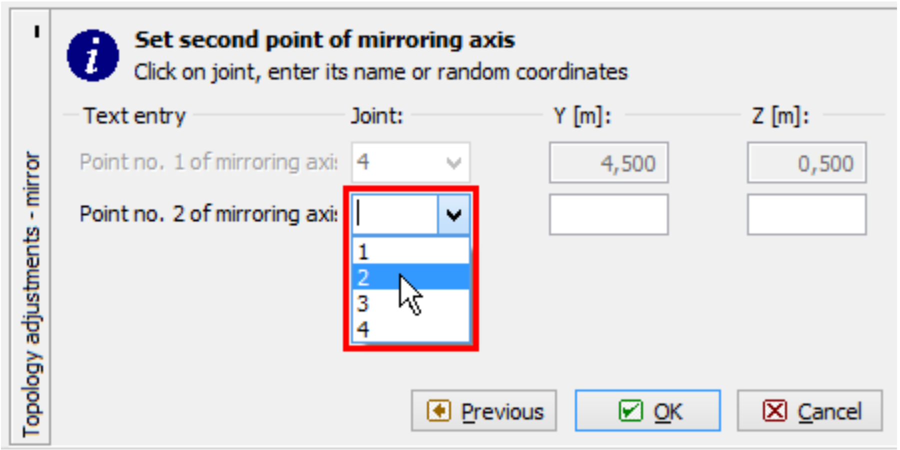
The axis for first and second option is specified by the distance from corresponding global axis. The axis for third option is given by two points.



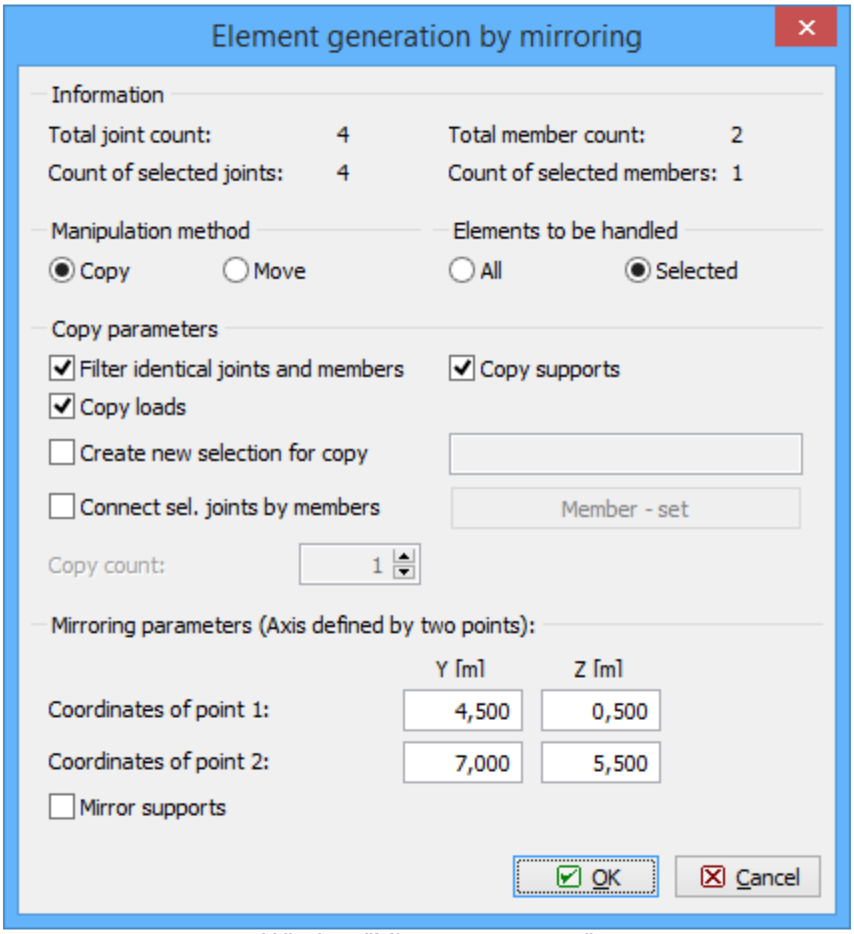
*Mirror using axis parallel to Z-axis and general axis*

The input of axis follows. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.




*Choice of joint number in the input frame*

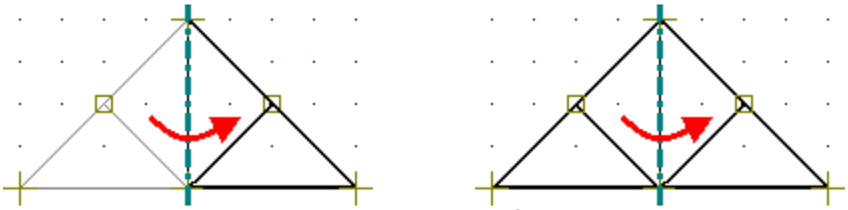
After that, the window with parameters of transformation appears.




*Window "Mirror parameters"*

The "**Manipulation method**" sets, whether the tool will only mirror the structure (or its part) or will keep existing structure and create a new copy. The most of following settings are disabled for the option "**Move**" (only transformation parameters can be specified).



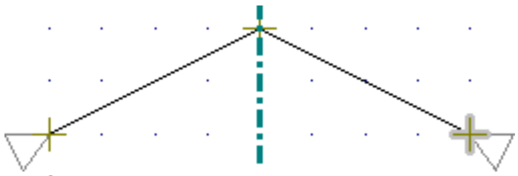
*Modes "Move" and "Copy"*

This tool is able to mirror whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. This setting is recommended. Otherwise, two structures (old one and new one) without any connection may be created.

The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "" in the toolbar above the workspace.



*Structure mirrored including supports*

# Program Fin 3D

The program "**Fin 3D**" is suitable for the structural analysis of 3D truss and frame structures with the help of finite element method. The particular members of the structure may be designed in the verification programs.

The main parts of the user interface are the workspace, the tree menu, the main menu and the input frame in thee bottom part of the window. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

## Tree menu

The tree menu is the fundamental navigation object of the program. It contains all important functions for the work with the structure. The tree menu contains these parts:

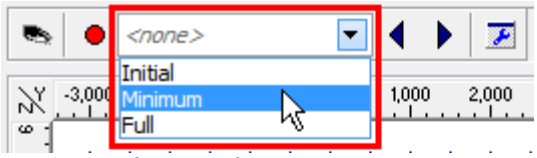
- **Topology** - Contains tools for input of structure topology (joints and members)
- **Load** - Loads, load cases, combinations and concentrated weights (in the part "**Dynamics**") can be specified in this part.
- **Calculation** - Runs the calculation of internal forces
- **Results** - Shows the results and contains tools for verification of members

The bottom part of tree menu contains buttons for insertion and administration of pictures for output documentation. The structure view in the workspace may be saved and used in output documentation. The pictures are updated continuously, the pictures in the documentation shows the latest state of structure including corresponding results. The new picture can be added with the help of the button "**Add picture**". After using this button, the window "**Picture properties**" appears. The parameters like description, orientation and position in document structure can be defined in this window. The pictures can be modified in the window "**List of pictures**", that can be opened by the button of the same name.

## Workspace

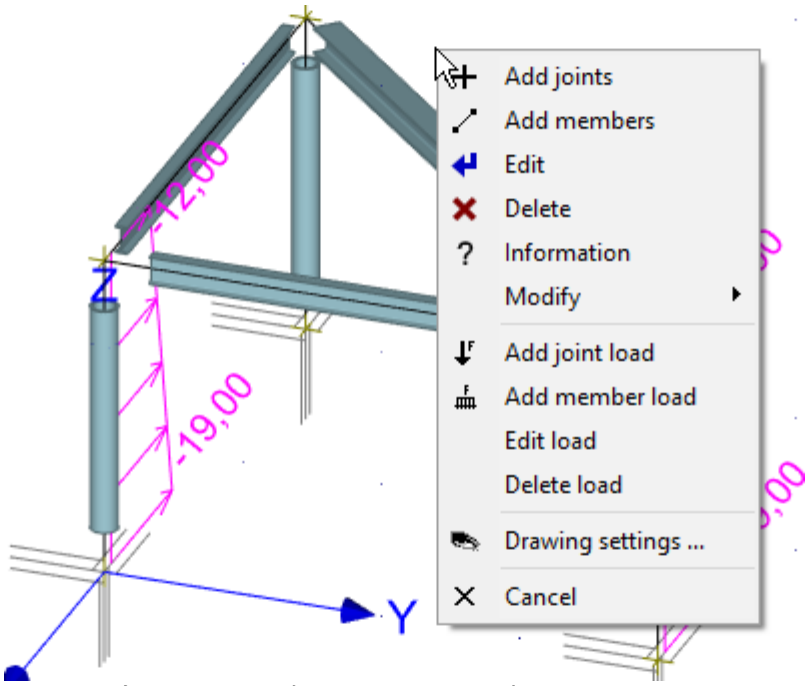
The workspace shows the created structure and can be used both for the graphical input and results display. The displayed items can be switched on or off in the window "**Drawing settings**", that can be launched by the button "" in the toolbar above the workspace. The view configuration may be saved or restored with the help of pre-defined templates, the work with templates is described here.



*Choice of view template*

The workspace also contains an option to use context menus, that can be opened by right mouse button click. This menu contains the most common commands. The range of commands differ according to the element type. Menus are different for joints, members and general point in the workspace. Additional commands are available for structures with selected elements (highlighted by green in the workspace).



*Context menu for general place of the workspace*

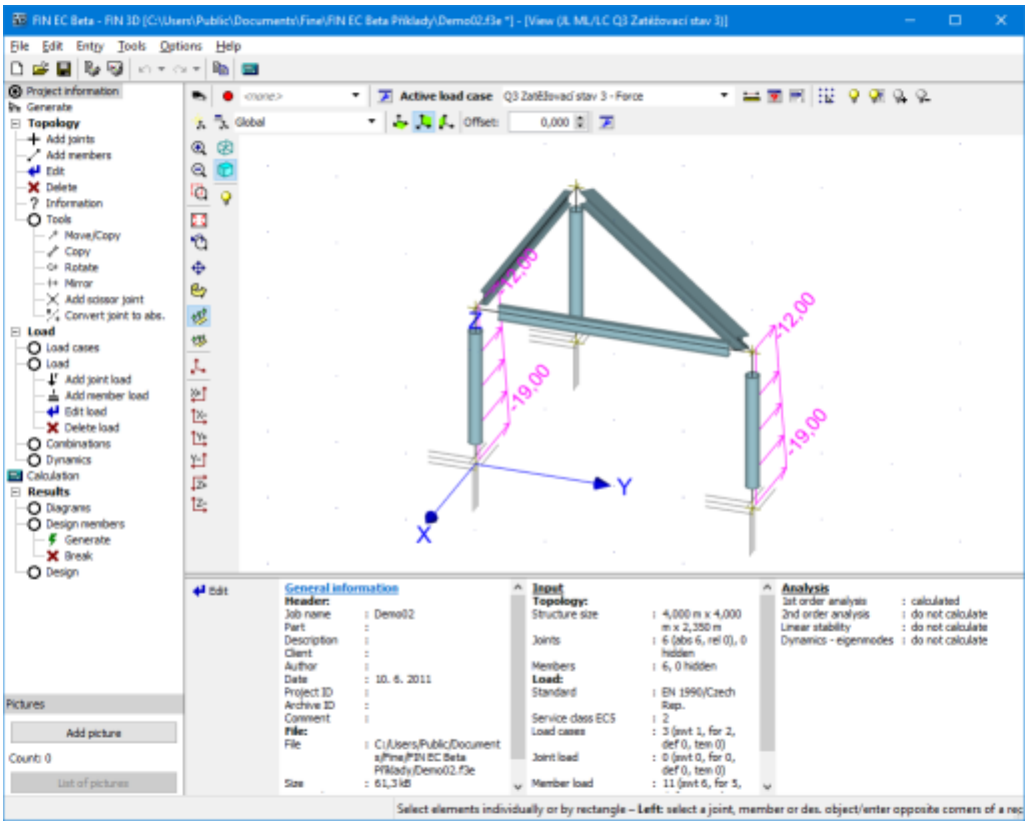
## Main menu

The main menu contains complete range of available tools and functions, it may substitute the work with the tree menu. Additionally, the main menu contains part "**Tools**" with some useful tools.

## Input frame

The input frame is loacated in the bottom part of the window. The default view shows the general information about the project (number of members and joints, load cases etc.), status of analysis and also project details specified in the window "**Project information**", that may be used in the heading or footing of the documentation. This window can be opened with the help of the button "**Edit**".



*Main application window*

# Topology

The structure can be created in the program Fin 3D using these three ways:

- **Direct input of the structure** - The fundamental input of topology and loading. The topology is defined by the particular joints and members. Loads are organized into load cases and load combinations, it is possible to enter joint load or member load.
- **Generator of 2D structures** - The most common structures (trusses, attic roofs, frames) can be created easily with the help of the "**Generator of 2D structures**". This generator is able to create both the structure topology and the fundamental load. This tool is described in the chapter "**Generator of 2D structures**".
- **Import from **.dxf* file** - The structure topology may be also imported from **.dxf* in the window "**Import dxf**". This option is available in the main menu, part "**File**", "**Import**".

All these options may be combined (the basic part of the structure can be created in generator or imported from **.dxf*, the second part of the work can be done with the available tools).

## Input and editing methods

The methods of input and editing are described in chapters

- **Input element**
- **Edit elements**
- **Delete elements**

The program in the mode "**Tools**" of the tree menu contains additional tools for the manipulation with elements and structures (move, copy, rotate etc.).

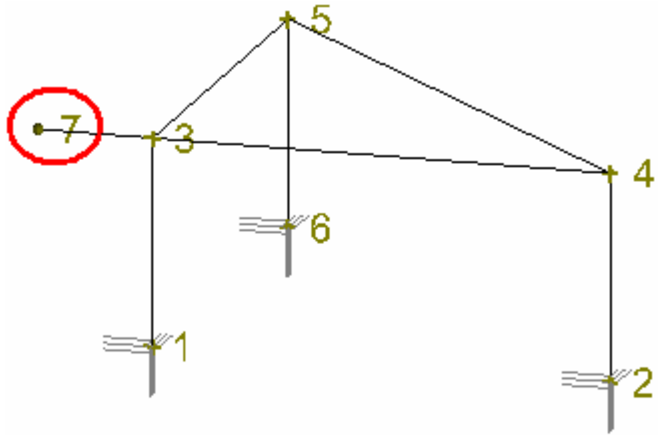
## Types of elements

The structures consist of two basic types of structural elements: joints and members.

Two different types of joints are included: "**absolute**" and "**relative**". Special type of relative joint is the "**scissor joint**", that can be specified in the intersection of two members.

**Absolute joints** are the joints, that have the position specified by the coordinates *[X,Y,Z]* in the global coordinate system. Their position can be changed only by editing the coordinates. Absolute joints are usually used for input of fundamental shape of the structure. These joints are marked with cross in the workspace.

**Relative joints** have the position specified relatively to the reference member. The position on member is given by the distance from the beginning or end joint of the member. The distance may be specified in length unit (metres) or in a proportional unit. The joints may be placed between beginning and end joints of the member or may lie outside this segment and extend the member length. The position has to be specified by a negative value in this case. The relative joints are used for connections of members to the point placed on the other member (typically connection of webs to chords).




*A cantilever created by the relative joint 7 that is placed in front of the reference joint 3*

The position of the relative joint can be changed only in the direction of the local axis of the reference member. Relative joints are usually used in trusses for connections of webs to chords. These joints are marked with square in the workspace.

The differences between absolute and relative joints are also described in the chapter "**Conversion of relative joint to absolute**".

**Scissor joint** is a special type of relative joint, as it has two reference members. The position of such joint is given by the intersection of these reference members. This joint creates hinged connection of intersecting members.

**Member** is specified by start and end joints, cross-section, material and end conditions. The material and cross-sectional characteristics can be specified with the help of pre-defined databases, user input or by the import from programs "**Section**" and "**Sector**".

The member is connected to the reference joints by a connection, that consists of six components (three shifts and three rotations). Any of these components may be defined as free or spring.

Any member contains local coordinate system, that is used for the input of relative joints and load. The origin of this local coordinate system is the beginning of the member, local axis *1* is given by the member direction.

## Member type

Program uses two basic member types: "**Beam**" and "**Beam on elastic subsoil**". The "**Beam**" is the fundamental member type locally supported in joints, "**Beam on elastic subsoil**" is the member, that is supported along the whole length by a subsoil (e.g. foundations).

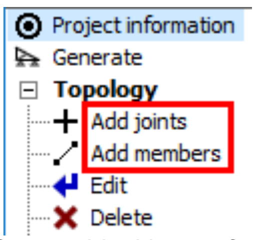
Members are also described in the theoretical chapter "**Structural elements**".

# Input elements

Joints may be inserted graphically in the workspace or with the help of the table in the bottom part of the window.

## Graphical input

Joints and members can be inserted directly by clicking in the workspace. The tree menu has to be switched into the mode "**Add joints**" or "**Add members**".

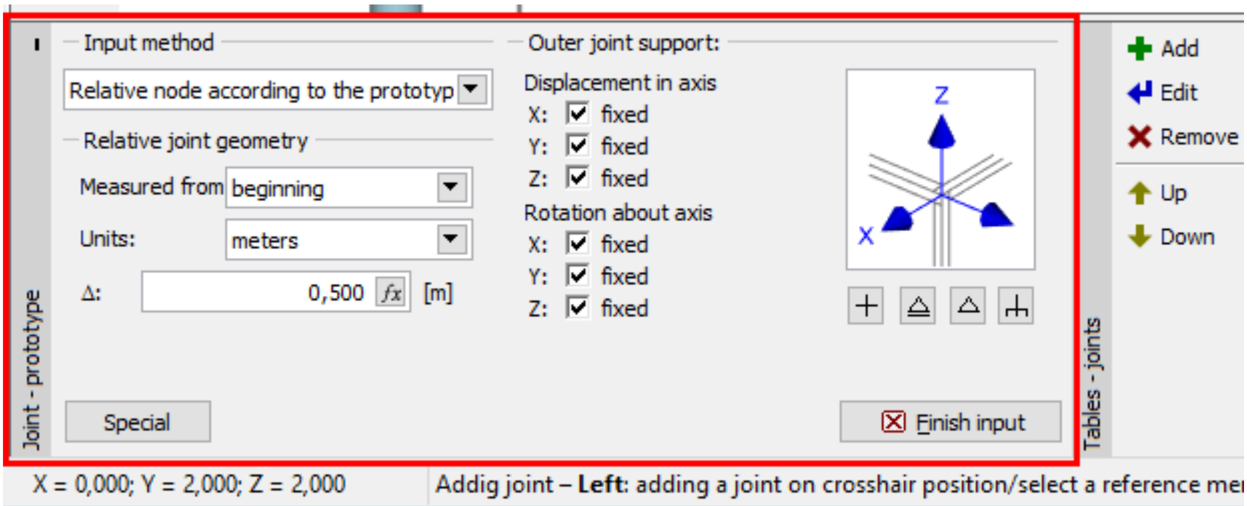



*Modes for graphical input of elements*

When adding **joints**, the window "**Joint prototype**" appears first. This window contains support properties and also input method for relative joints. Two methods are included:

- **Relative joint located by mouse click** - The position of joint will be determined by the position of cursor.
- **Relative joint according to the prototype** - The position of joint will be selected according to inputs "**Relative joint geometry**", which are located in the prototype frame.

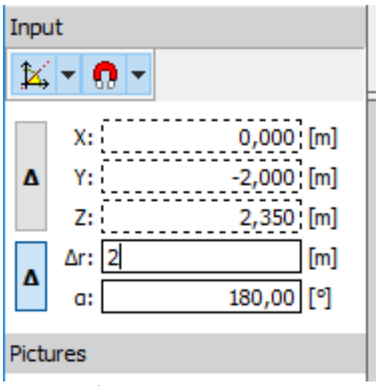
The entered data in this window has to be confirmed by the button "**OK**". After that, the window is docked in the bottom frame.




*Joint prototype anchored in the input frame*

The input is performed by clicking in the workspace. The snapping grid may be used for easier input, the properties of the grid are specified in the window "**Options**" (main menu part "**Tools**"). The snapping can be switched off (or switched on) temporarily when pressing the key "**Ctrl**" during the input. Program offers also snapping points like mid point of member or intersection of two members. Snapping points may be switched on and off in the list, that can be opened by the button "".

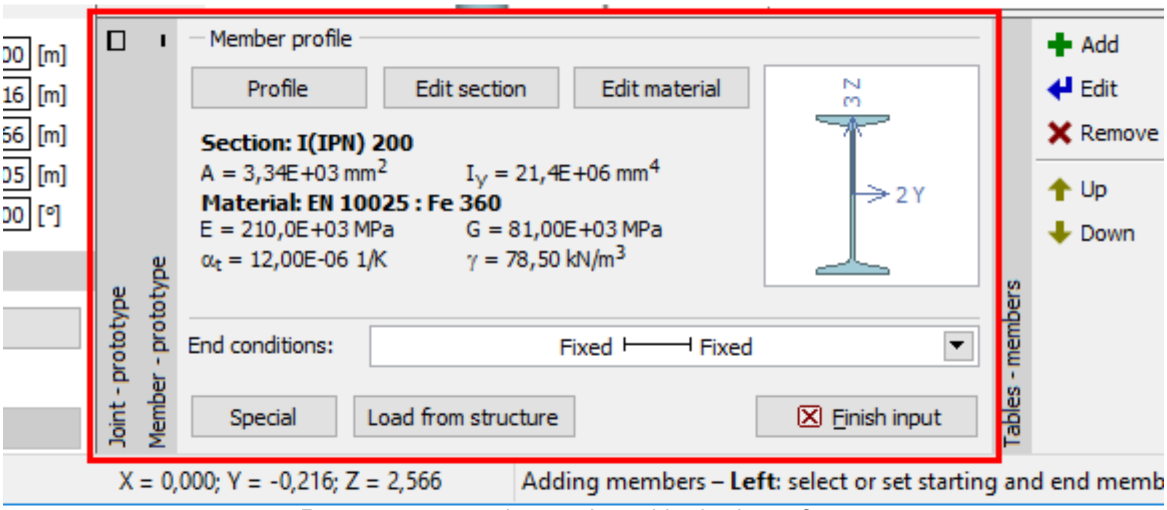
The input fields in the bottom part of the tree menu can be also used for the input of joints. It is possible to jump into input fields with the help of cursor or by keyboard entries "**x**", "**y**" and "**z**". For example, the expression *x0y2z1.3* fills *0* into the input line "**x**", *2* into the input line "**y**" and *1.3* into the field "**z**".




*Input fields in the tree menu*

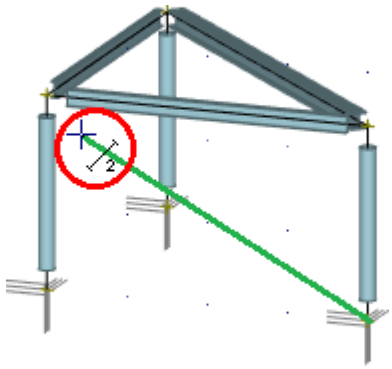
The input of **members** is based on similar procedures. After the selection of an appropriate mode, the window "**Member prototype**" appears. This window contains properties (cross-section, material, end conditions etc.) that will be assigned to new members. The properties correspond to the parameters in the window "**Member properties**" (parameters may be copied from existing member using the tool "**Load from structure**"). Also the window "**Prototype of joint**", that contains the parameters of newly created joint (beginnings and ends of new members). The data in these windows has to be confirmed by the button "**OK**". After that, prototypes are anchored in the input frame, where the parameters may be changed.

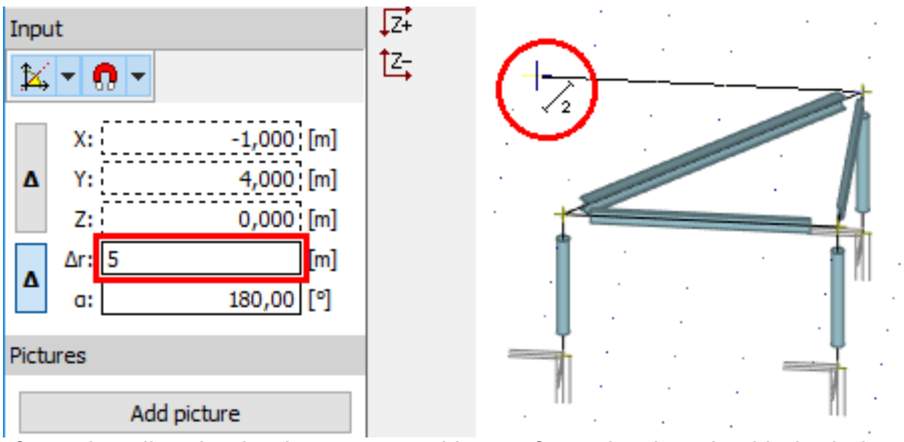


*Prototype properties anchored in the input frame*

The member input is done by clicking on the position of member beginning and end. The cursor shows number "**1**" in the mode for input of the member beginning and number "**2**" in the mode for input of the member end.


*Input of member in the workspace*

The member beginning can be entered using methods for joints. The end joint can be specified in a similar way. The easiest way is to specify the member direction by cursor and specify the length on keyboard. The length will be automatically filled into the field "**Δr**". The input has to be confirmed by the key "**Enter**". The program automatically snaps the cursor into directions *45°*. This behaviour may be changed (switch off or change it to *30°*) using the button "".



*Definition of member direction by the cursor and input of member length with the help of keyboard*

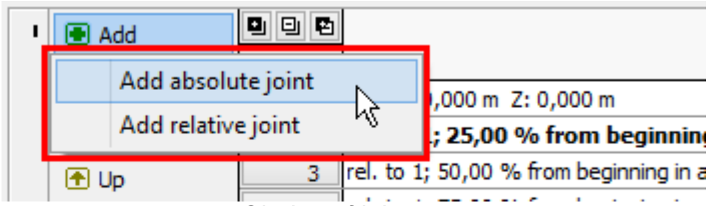
Alternatively, it is possible to use snapping to existing joints and snapping points (described above) or extended options of input fields in tree menu. Input fields "**X**" "**Y**" and "**Z**" are usable for the definition of the end point with the help of global coordinates, fields "**r**" and "**α**" define the end point by the member length and rotation about the axis *y*. Buttons "**Δ**" changes whether the input is considered in the global coordinate system or relatively to the beginning of the member. It is possible to jump into input fields with the help of cursor or by keyboard entries "*x*", "*y*", "*z*", "*r*" or "*a*". For example, the expression "*r2a15*" fills *2m* into the field "**r**" and *15°* into the field "**α**".

## Input in tables

Joints and members may be also added with the help of the button "**Add**" in the toolbar on the left side of the tables, which appears in the bottom frame for the mode "**Topology**" of the tree menu. For the input of joints, one of available options

("**Add absolute point**" or "**Add relative point**") has to be selected first. Input is performed in the windows "**Properties of absolute joint**" and "**Properties of relative joint**". Input of relative joints isn't allowed until at least one member is specified.




*Choice of joint type*

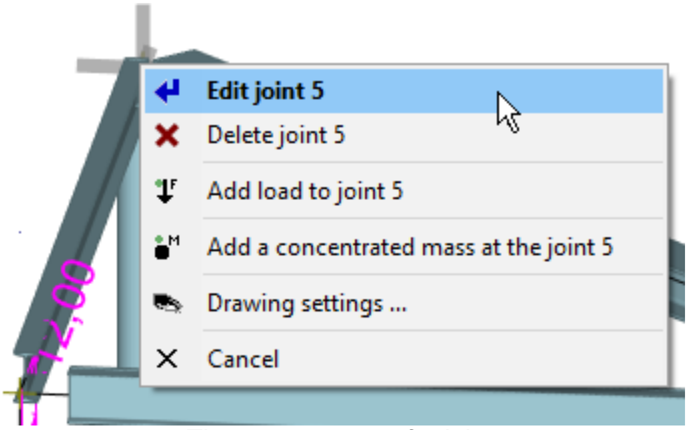
Members can be entered in a similar way. The window for input of new members is identical to "**Member properties**", the reference joints, cross-section and material have to be specified there.

# Edit elements

Joints and members may be modified individually or in a batch. Editing is available only in preprocessor (parts "**Topology**" and "**Load**" of the tree menu).

## Editing in the workspace

The individual joint or member can be edited by double-click on the element in the workspace. The editing of properties is done in the windows "**Properties of absolute joint**" or "**Properties of relative joint**" for joints or in the window "**Member properties**" for members. An alternative way is to use a command "**Edit joint**" or "**Edit member**" in a context menu, that can be opened by right mouse button in the workspace. The same procedure can be used also for selected elements (highlighted by green in the workspace).

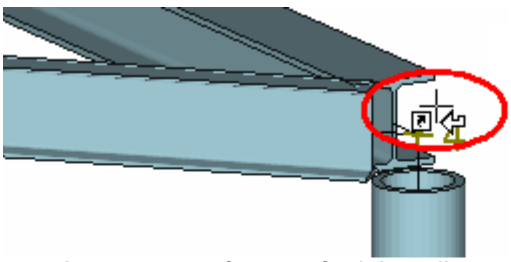



*The context menu for joint*

The appropriate window for editing an element can be also opened using double-click on an appropriate row in tables of joints and members in the bottom frame. Alternatively, the button "**Edit**" on the left side of the table can be used.

## Graphical mode "Edit"

The tree menu has to be switched into the mode "**Topology**" "**Edit**" and after that, it is possible to edit joints and members by clicking on the appropriate element in the workspace. The software also considers close surrounding of the element for a correct click. The considered surrounding is indicated by the change of the cursor appearance.



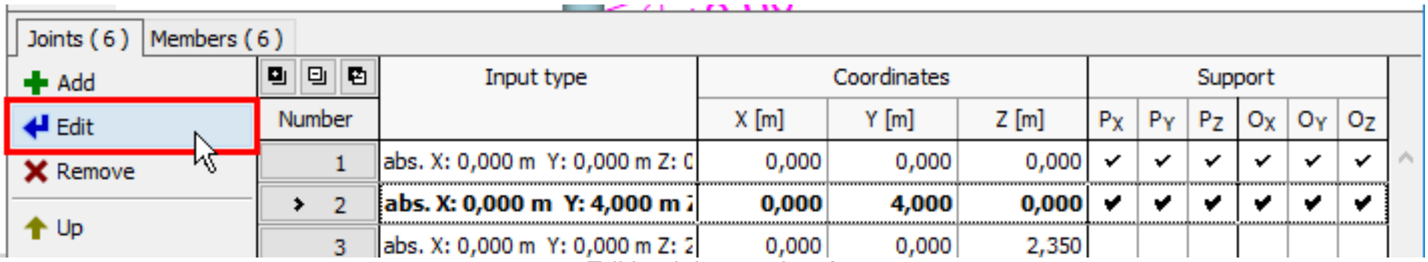
*Appearance of cursor for joint edit*

The mode can be terminated by selection of other mode or by right mouse button click.

## Editing joints and members in the table

Joints and members may be also edited with the help of the button "**Edit**" in the toolbar on the left side of the table. The program modifies the active element (highlighted by bold font and mark "**>**" in front of the joint number).

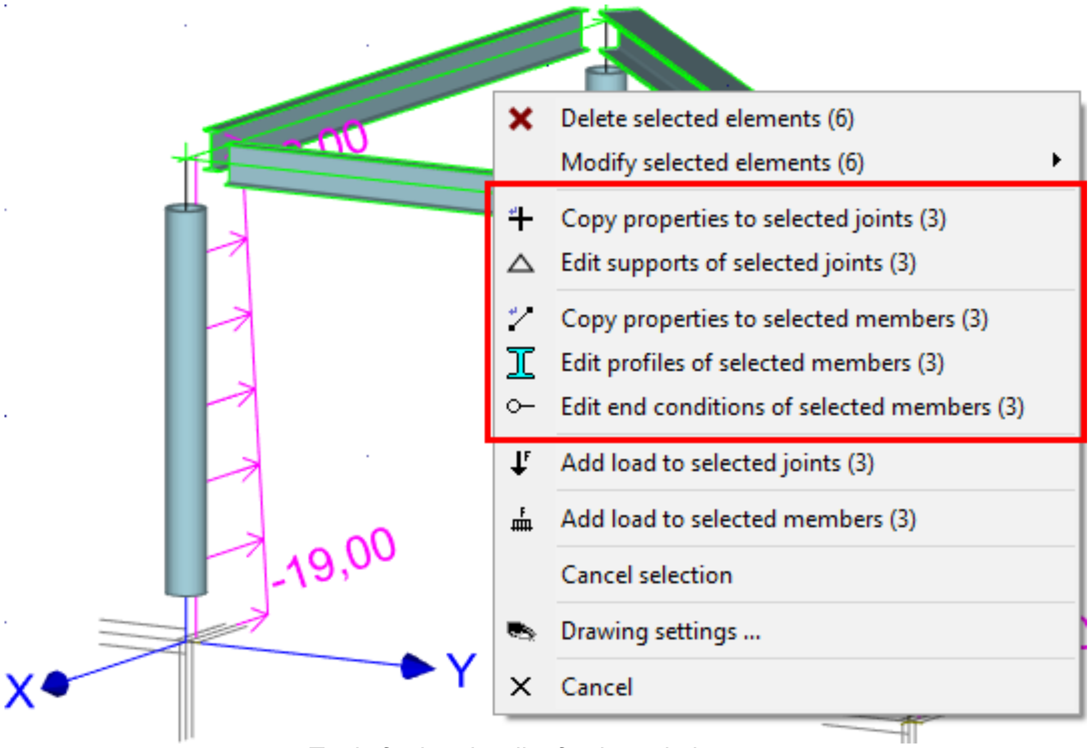


*Editing joint number 2*

An alternative way is to use the command "**Edit**" in the context menu for the appropriate table row.

## Editing selected joints and members

Joints and members can be also selected and modified in a batch. These elements can be selected in the workspace or in tables in the bottom frame. Selected joints are highlighted by green in the workspace and by blue and bold font in the table. Tools for batch edit are available in the context menu, that can be opened by right mouse button click in the workspace (provided that some joints or members are selected).



*Tools for batch edit of selected elements*

Following tools may be used for batch editing of joints:

| | | |
|---|---|---|
| **Copy properties** | • | Copy of specified joint properties to selected joints |
| **Edit supports** | • | Batch edit of supports for selected joints |

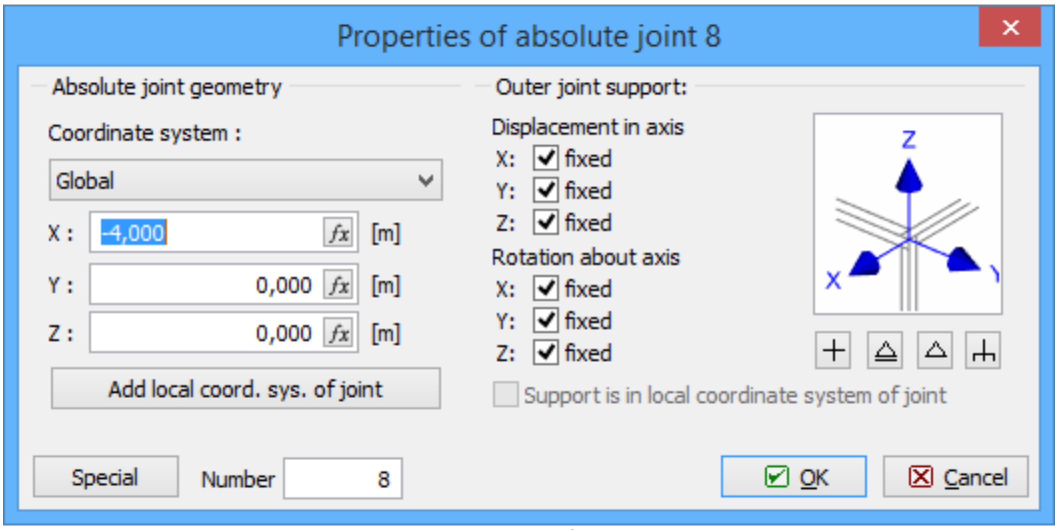
Following tools may be used for batch editing of members:

| | | |
|---|---|---|
| **Copy properties** | • | Copy of specified member properties to selected members |
| **Edit profiles** | • | This tool is able to modify cross-section and material of selected member. The range of available properties is equal to the range described in the chapter "**Edit profile**". |
| **Edit end conditions** | • | Batch edit of end conditions for selected joints |

## Properties of absolute joint

The position and supporting style may be changed for absolute joint. The position of the absolute joint is given by the coordinates *[X,Y,Z]* in the global coordinate system. The accurate values of coordinates may be calculated with the help of the built-in calculator. The button "**Add local coordinate system of joint**" opens the window "**Coordinate system of joint**", where the rotated coordinate system for support may be defined.

The right part "**Outer support**" contains the properties of the support. These properties are described in the chapter "**Supports**". The spring supports have to be specified in the window **"Special properties of joints**", that can be opened by the button "**Special**" in the left bottom corner.

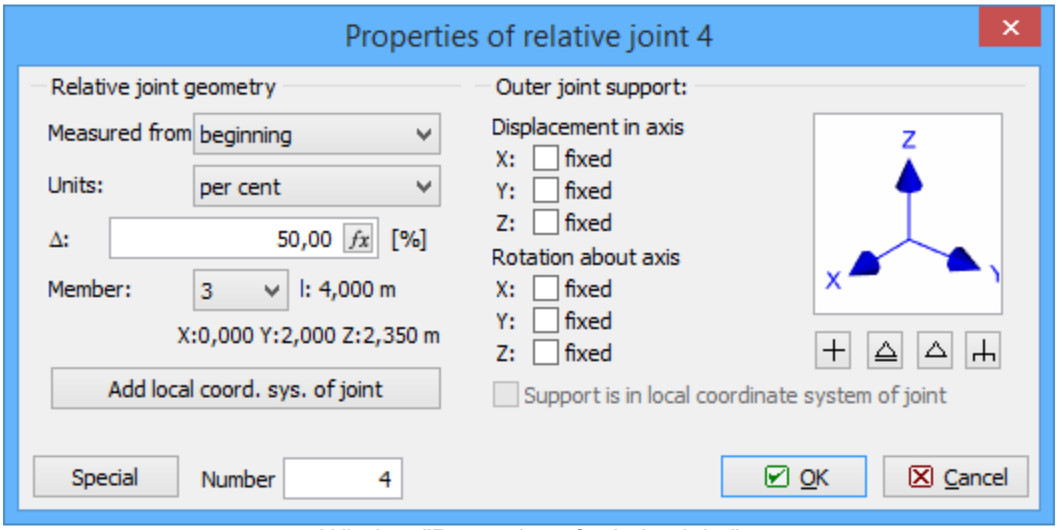



Window "Properties of absolute joint"

## Properties of relative joint

The position and supporting style may be changed for relative joint. The position of the relative joint is given by the number of reference member, by the position in metres or per cents and by reference point (beginning or end of the member). The button "**Add local coordinate system of joint**" opens the window "**Coordinate system of joint**", where the rotated coordinate system for support may be defined.

The right part "**Outer support**" contains the properties of the support. These properties are described in the chapter "**Supports**". The spring supports have to be specified in the window **"Special properties of joints**", that can be opened by the button "**Special**" in the left bottom corner.




Window "Properties of relative joint"

## Supports

The joint support consists of three components: prevented displacements in directions of global axes $Y$ and $Z$ and prevented rotation about axis $X$. The support in the certain direction can be switch on by the corresponding check box "**fixed**". Following symbols are used in the workspace for different types of supports:

- The joint isn't supported in any direction

- The joint is supported in all directions, rotation-free joint (hinge)

- The rotation is fixed in all directions, displacements are not restricted.

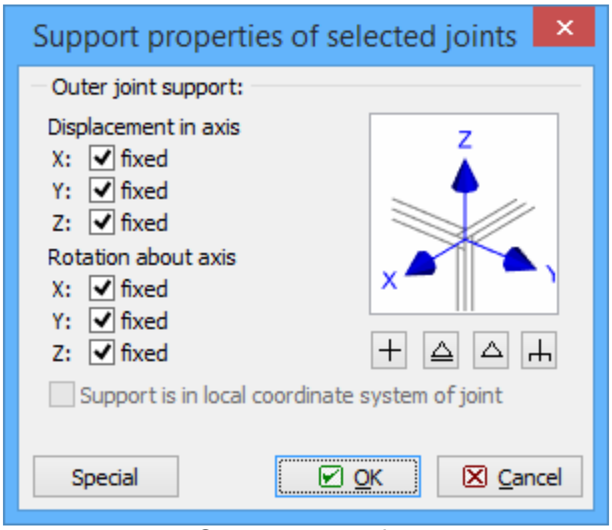
- The fixed support (rotations and displacements in all directions prevented)

The most common support types can be entered with the help of dedicated buttons in the right bottom corner of the frame.

The support properties also contain the setting "**Support is in local coordinate system**". This option is useful for cases, where the reactions in the support has to be obtained in the rotated coordinate system or where the structure is supported in certain direction that isn't parallel to main axes. The local coordinate system of joint has to be specified first in the joint properties.

The button "**Special**" opens the window "**Special properties of joint**", where the spring supports may be defined.

The support styles are also described in the theoretical chapter "**Supports**".



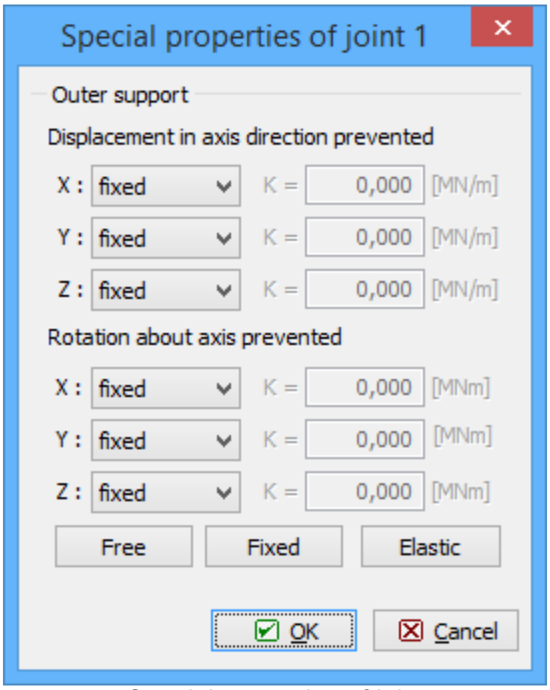

*Support properties*

## Special properties of joint

This window contains extended properties of joint support. It brings an option to specify a spring support in any direction. The spring constant *K* has to be specified in this case.

The buttons "**Free**", "**Fixed**" and "**Elastic**" will assign corresponding support method to all directions.

The support styles are also described in the theoretical chapter "**Supports**".




*Special properties of joint*

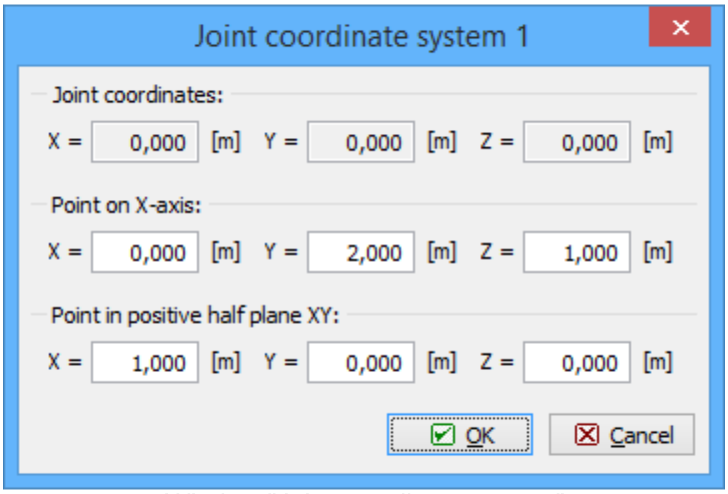
 

# Coordinate system of joint

The coordinate systems of joints are used for the definition of rotated supports in joints. These coordinate systems have axes $X_S$,$Y_S$,$Z_S$. The origin of this coordinate system is in the joint. The coordinate system is defined by the point in the positive part of the axis $X_S$ and by the point, that lies in the plane $X_S Y_S$ (where the coordinate $Y_S$ is positive). The axis $Zs$ is given by the rules for a right-hand Cartesian coordinate system. Any joint may contain only one coordinate system.

These three points that specify the coordinate system cannot be in line or have identical coordinates.

This option is useful for cases, where the reactions in the support has to be obtained in the rotated coordinate system or where the structure is supported in certain direction that is not parallel to the global axes.



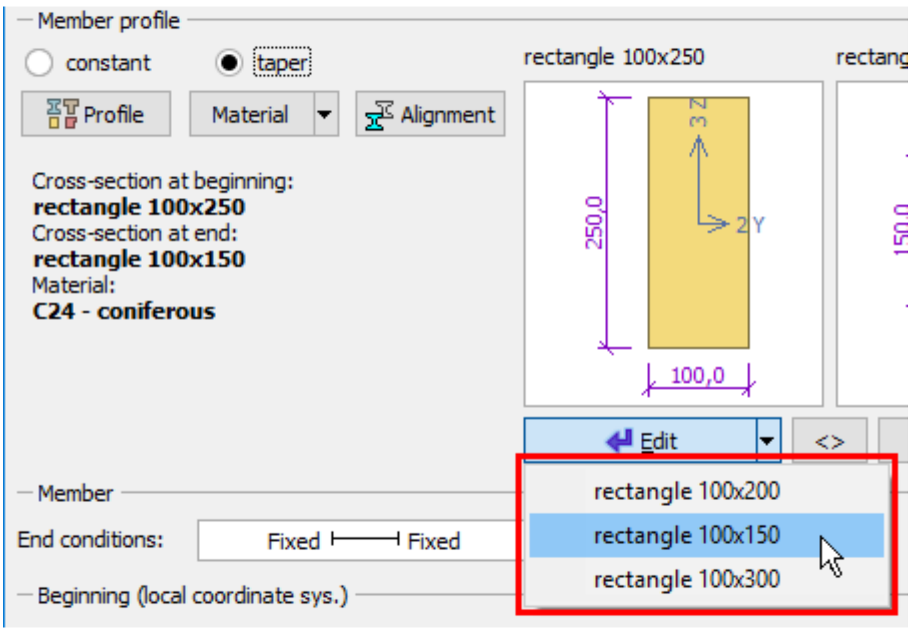

*Window "Joint coordinate system"*

# Member properties

This window contains basic properties of member: beginning and end joint, cross-section, material and end conditions. The button "**Special**" in the left bottom corner launches the window **"Special properties of member**", that contains advanced properties (subsoil parameters, excluded tension or compression, spring end connections).

The position of the member is given by the numbers of beginning and end joints. These joints can be changed with the help of the list box or by the direct input of the number on the keyboard. The button "**<>**" changes the member orientation (switches the beginning and end joint), the positions of relative joints and loads are preserved.

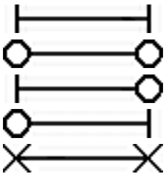
The frame "**Member profile**" contains properties of cross-section and material. The member may have identical cross-section along the whole length (option "**constant**") or the cross-section vary along the member length (the option "**taper**"). The cross-section and material can be changed in the window "**Edit profile**" which can be launched by the button "**Profile**". Parameters of the existing cross-section (e.g. dimensions) or material can be changed in the windows "**Cross-section editor**", (the button "**Edit**" under the cross-section view) or "**Materials catalogue**" (the button "**Material**"). Members with variable cross-section have two buttons for cross-section edit, the first one for member beginning and the second one for member end. Buttons for edit of cross-sections and materials contain drop down menu with existing cross-sections and materials. These list can be also used for fast input of these properties.



*List of existing cross-sections*

The visual appearance of the structure can be modified by an alignment of the member which can be specified in a dedicated window using the button "**Alignment**". This alignment defines the position of member mass relatively to reference line of the member. It affects only structure view not the analysis.

The frame "**Member**" contains the choice of end conditions (pinned, fixed):



- The member is fixed on both ends
- The member is pinned on both ends
- The member has fixed beginning and pinned end
- The member has pinned beginning and fixed end
- The member has special conditions (e.g. spring connections) at the beginning and end. The properties are organized in the window "**Special properties of member**".

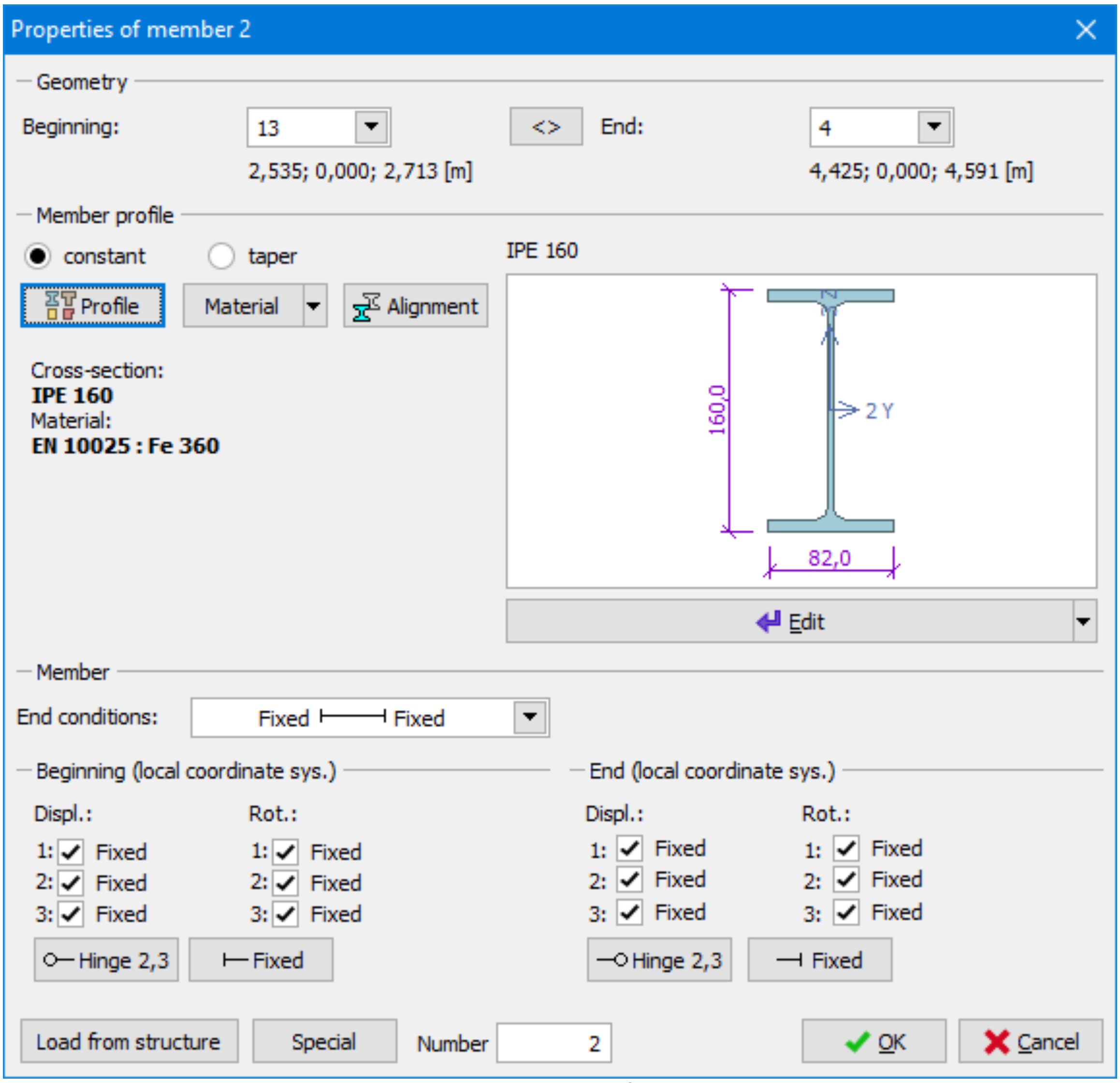
The end conditions can be also specified with the help check boxes "**Fixed**" for corresponding directions or rotations. The end conditions are specified in the local coordinate system of member.

Parameters (cross-section, material, end conditions) may be copied from existing member using the tool "**Load from structure**".

## Pinned connection

The rotation about the member axis (local axis *1* ) should be considered when entering pinned end conditions on both ends of member. Completely pinned connection (no rotation fixed on both ends) causes singularity during analysis and collapse of the calculation. Therefore, rotation about the local axis *1* should be fixed at one end of a member. Such solution ensures analysis stability and does not cause torsional stiffness of the member. This solution is not necessary in cases, where the rotation about the axis *1* is prevented by another connected member.

End conditions are also described in the theoretical chapter "**End conditions**".



*Window "Properties of member"*

# Special properties of member

This window contains advanced properties of member.

## Member type

Program uses two basic member types:

| | |
|---|---|
| **Beam** | • Fundamental member type supported in joints. |
| **Beam on elastic subsoil** | • The member, that is supported along the whole length by a subsoil (spring support). |

Additionally, it is possible to exclude compressive or tensile stresses in the member.

## Subsoil parameters

The subsoil can be specified in directions of local axes *3* or *2*. The subsoil is defined by Winkler-Pasternak constants $C_1$ and $C_2$. The constants $C_1$ and $C_2$ may be calculated from the general parameters of subsoil in the window "**Calculation of C1 and C2**". The subsoil acts as general spring, that acts both in compression and tension. As this model usually does not correspond to the real conditions, the appearance of tensile stress should be checked.

The setting "**Use member width**" uses the cross-section width as the width of member contact with subsoil. If not used, the contact width $b$ may be specified manually.

The default assumption is, that the subsoil acts in the direction of the local axis *3* (against gravity). The setting "**Acts in axis *3***" is able to change this behaviour.

The shear effects at the member ends may be taken into the account with the help of settings "**In front of member**" and "**Behind member**".

The subsoil properties are described in the theoretical chapter "**Subsoil model**".

## Warping prevention

The part "**Rate of warping prevention**" contains the setting, that affects behaviour of member in torsion. Torsion causes both the deformation of cross-section in its plane and in the perpendicular direction (parallel to the member axis). This behaviour is called warping. If the warping is not prevented in the structure, torsion induces only shear stresses and the cross-section is deformed in both directions. Such behaviour is called St.Venant torsion. If the warping is prevented, the torsion induces shear and axial stresses and such torsion is called warping torsion. Warping does not appear for all cross-sections. Warping is common mainly for steel cross-sections with warping coordinate $\omega$ and warping constant $I_\omega$ greater than *0*. Warping parameters can be specified only for these cross-sections.

Warping prevention can be specified with the help of the constant with the interval *<0;1>*, where *0* means free warping and *1* means warping absolutely prevented. The intermediate values describe combined behaviour. Three different internal forces induced by torsion may appear on members subjected to warping: St.Venant torsional moment $T_t$, bimoment $B$ and warping torsional moment $T_o$. Moments $T_t$ and $T_o$ induce shear stresses in cross-section, bimoment $B$ induces axial stress.

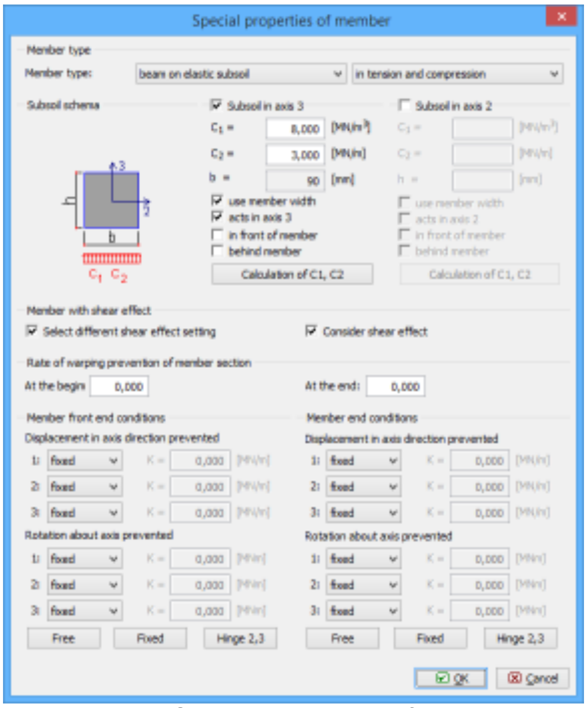
## Shear effect

The part "**Member with shear effect**" may rewrite the global settings regarding the consideration of shear effect on deformations. This theoretical model with shear consideration is recommended cases, where member length isn't significantly longer than cross-section dimensions. The theoretical background is described in the chapter "**Special member characteristics**".

## Member end conditions

This frame contains extended options for end conditions. It brings an option to specify a spring connections in any direction (not only fixed/free). The spring constant $K$ has to be specified in this case. The buttons "**Free**", "**Fixed**" and "**Elastic**" will create corresponding end condition. End conditions are also described in the theoretical chapter "**End conditions**".



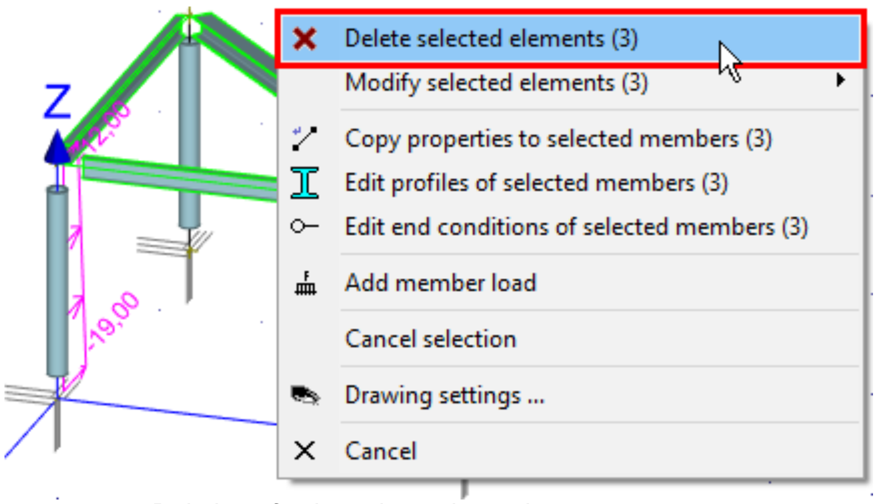
*Window "Special properties of member"*

# Delete elements

Joints and members may be deleted individually or in a batch, using the table in the bottom part of the window or graphical mode for workspace. Any deletion of joint removes also all members connected into this joint. Any deletion of member removes also all relative joints on the member and members connected into these joints.

## Using context menu

Any joint or member can be deleted using command "**Delete joint**" or "**Delete member**" in a context menu, that can be opened by right mouse button in the workspace. The same procedure can be used also for selected elements (highlighted by green in the workspace).

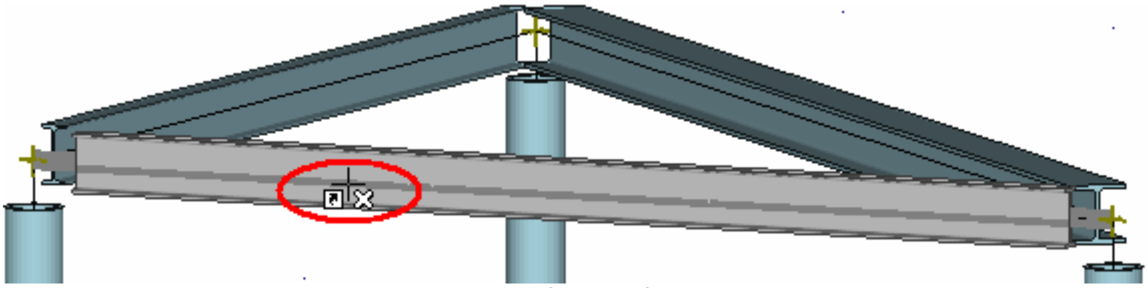



*Deletion of selected member using context menu*

## Graphical mode "Delete"

The tree menu has to be switched into the mode "**Topology**" "**Delete**" and after that, it is possible to delete joints or members by clicking on the appropriate element in the workspace. The software also considers close surrounding of the element for a correct click. The considered surrounding is indicated by the change of the cursor appearance.

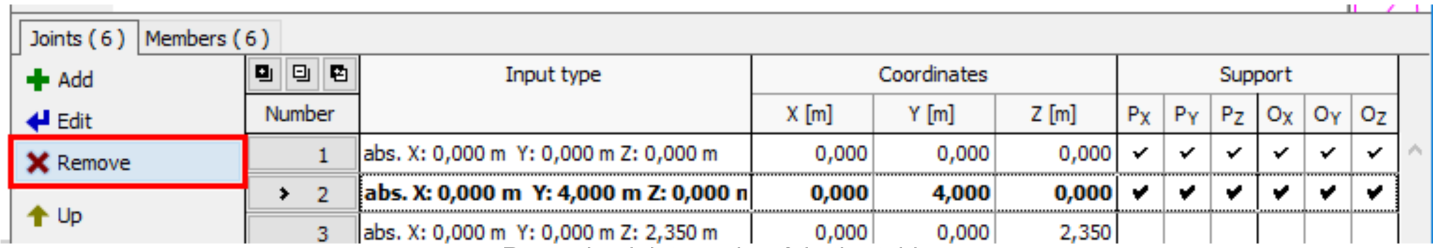


*Appearance of cursor for deletion*

The mode can be terminated by selection of other mode or by right mouse button click.

## Removing elements in the table

Joints and members may be also deleted with the help of the button "**Remove**" in the toolbar on the left side of the table. The program deletes the active element (highlighted by bold font and mark "**>**" in front of the joint number) or selected ones (highlighted by blue in the table).




| Number | Input type | Coordinates X [m] | Coordinates Y [m] | Coordinates Z [m] | Support Px | Py | Pz | Ox | Oy | Oz |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | abs. X: 0,000 m Y: 0,000 m Z: 0,000 m | 0,000 | 0,000 | 0,000 | ✔ | ✔ | ✔ | ✔ | ✔ | ✔ |
| > 2 | **abs. X: 0,000 m Y: 4,000 m Z: 0,000 n** | **0,000** | **4,000** | **0,000** | ✔ | ✔ | ✔ | ✔ | ✔ | ✔ |
| 3 | abs. X: 0,000 m Y: 0,000 m Z: 2,350 m | 0,000 | 0,000 | 2,350 | | | | | | |

*Removing joint number 2 in the table*

# Tools

The mode "**Tools**" contains functions and commands that can be used for editing created structures.

## Tools

These tools can be used both with complete structure and selected parts. Most of the tools can be used in two modes: as a simple transformation (change of position or shape) or as a copy tool (keeps original structure). Following tools are included:

| | |
|---|---|
| **Move** | • The tool that moves the structure in given direction |
| **Copy** | • The tool that copies the structure in given direction |
| **Enlarge/Shrink** | • The tool that enlarges or shrinks the structure |
| **Rotate** | • The tool that rotates or copies the structure using specified angle |
| **Mirror** | • The tool that mirrors existing structure, including copy option |
| **Align** | • The tool that aligns structural elements (joints, members) into given line |

## Joints

| | |
|---|---|
| **Add scissor joint** | • Inserts a scissor joint. The scissor joint is a special type of relative joint, as it has two reference members. The position of such joint is given by the intersection of these reference members. This joint creates hinged connection of intersecting members. |
| **Convert joint to absolute** | • Converts a relative joint to an absolute one. |
| **Absolute joints on members** | • Test that checks the coordinates of all absolute joints and compare them with positions of members. Absolute joints lying on members aren't connected to the members and may cause the singularity of the structure. Such joints may be converted into relative ones. This conversion automatically creates a connection between member and joint. |
| **Absolute joints on members convert to relative** | • A tool that finds absolute joints placed on members or in their surroundings and converts these joints into relative ones. The considered surroundings can be defined by user. The tool can be applied both to all and selected joints. |

## Members

| | |
|---|---|
| **Divide members** | • Tool that adds specified number of relative joints with uniform distribution along the member length. |

**Continuity analysis**

- Test that checks whether the structure is divided into more parts or not. Hidden division into more parts caused by overlapping joints or members causes collapses during analysis very often. The partial segments of the structure aren't usually supported in a sufficient way and the singularity appears in these cases. There is an option to highlight the certain part of the structure.

## Load

**Member load multiplication**

- This tool increases or reduces member loads in load cases with the help of specified multiplication factor. The factor may be applied to all or selected loads, the tool may be limited to active or selected load cases. This tool is suitable e.g. for modification of input after the change of loading width of structural elements.

**Joint load multiplication**

- This tool increases or reduces joint loads in load cases with the help of specified multiplication factor. The factor may be applied to all or selected loads, the tool may be limited to active or selected load cases. This tool is suitable e.g. for modification of input after the change of loading width of structural elements.

## Load cases and combinations

**Load template**

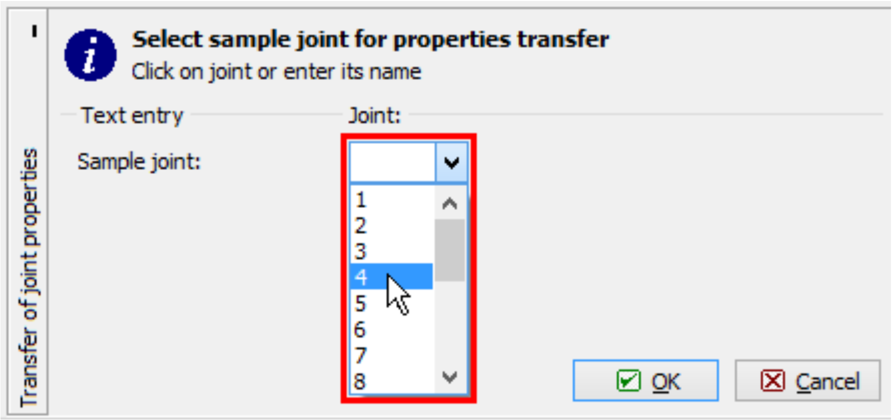
- The import of template (list) of load cases and combinations. This tool may be used for an easy transfer of load case/combination parameters from one project to another. The file of the template has an extension **.flc*.

**Save template**

- The export of template (list) of load cases and combinations. This tool may be used for an easy transfer of load case/combination parameters from one project to another. The file of the template has an extension **.flc*.

# Copy properties

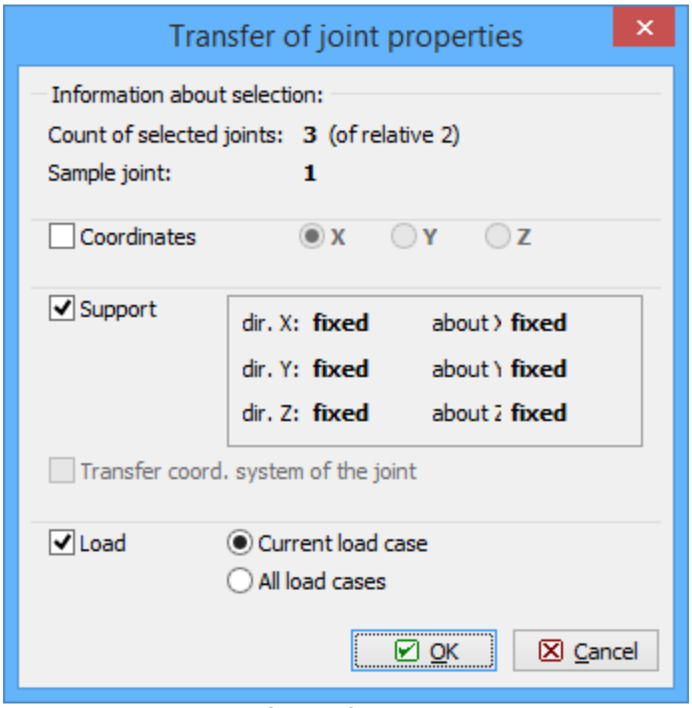
This tool can be used for copying the joint properties to another joints. This tool is enabled if at least one joint in the structure is selected (highlighted by green in the workspace). The option for the input of sample joint appears after the choice of the mode "**Joints**" "**Selected**" "**Copy properties**" in the tree menu. The properties of this joint will be copied to selected joints. The choice of the sample joint has to be confirmed by the button "**OK**". The sample joint can be also selected graphically by clicking in the workspace.




*Choice of sample joint*

The window, that appears after the choice of the sample joint, contains the options to specify properties, that will be assigned to selected joints. It is possible to copy one of coordinates (joints will be aligned into a plane perpendicular to the given global axis), support style (including local coordinate system) and load in active or all load cases. The concentrated loads can be copied, if the module "**Dynamics**" is enabled.

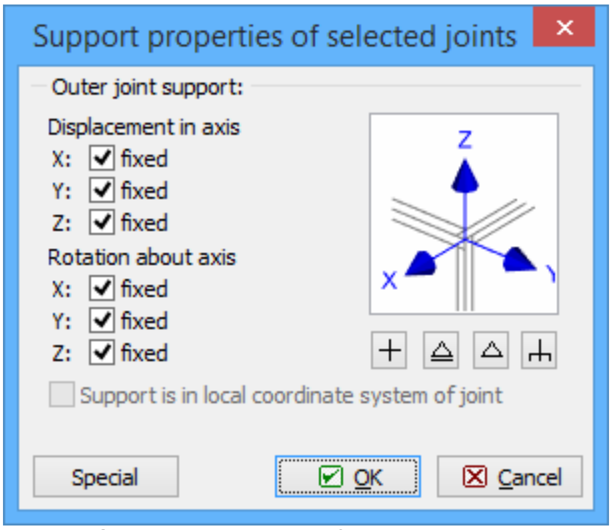
 



*Window "Copy of joint properties"*

## Edit supports

This tool may be used for batch edit of supports for selected joints. This tool is enabled if at least one joint in the structure is selected (highlighted by green in the workspace). The window "**Support properties of selected joints**" appears after the choice of the mode "**Joints**" "**Selected**" "**Edit supports**" in the tree menu. The support that will be assigned to all selected joints can be specified in this window. The range of inputs corresponds to the properties described in the chapter "**Supports**".

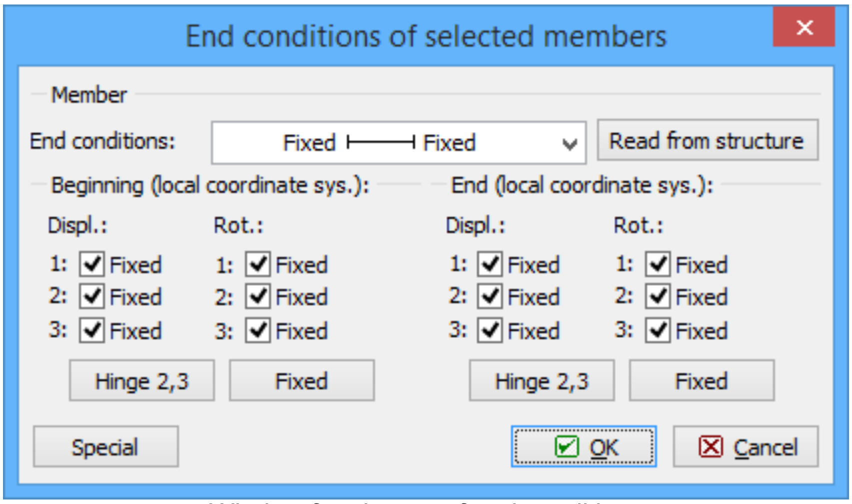


*Support properties of selected joints*

## Edit end conditions

This tool may be used for batch edit of end conditions for selected members. This tool is enabled if at least one member in the structure is selected (highlighted by green in the workspace). The window "**End conditions of selected members**" appears after the choice of the mode "**Members**" "**Selected**" "**Edit end conditions**" in the tree menu. The end conditions that will be assigned to all selected members can be specified in this window. The range of inputs corresponds to the window "**Member properties**". Parameters may be copied from existing member using the tool "**Load from structure**".

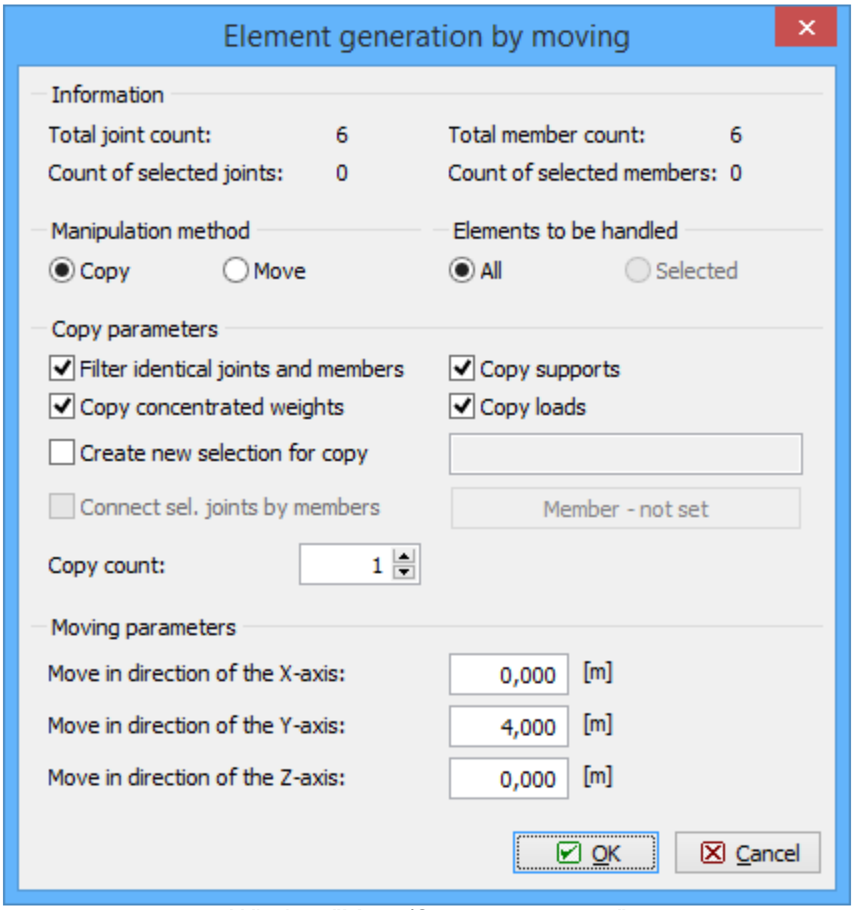



*Window for change of end conditions*

The spring end conditions have to be specified with the help of the button "**Special**" in the left bottom corner. The parameters correspond to the content of the window **"Special properties of member"**.

## Move/Copy

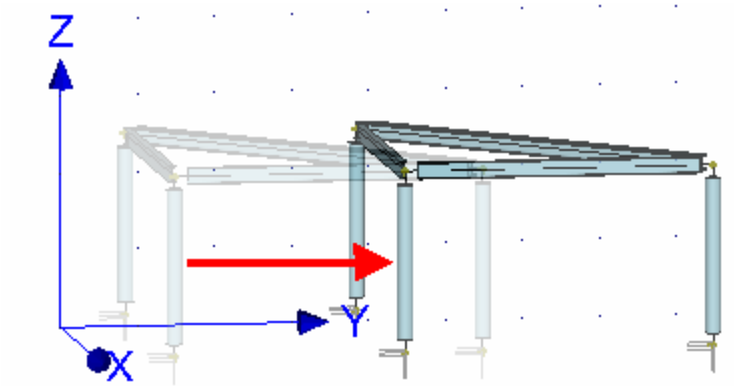
The tool "**Move/Copy**" can be used for shift or copy of structure (or its part). The window contains parameters, that affects the final behaviour of the tool.




*Window "Move/Copy parameters"*

The "**Manipulation method**" sets, whether the tool will only change the position of the structure (or its part) or will keep existing structure and create a new copy. The most of following settings are disabled for the option "**Move**" (only movement vector can be specified).

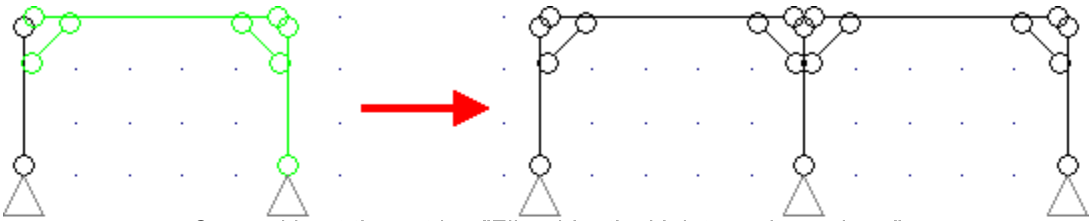
 




*Move of structure*

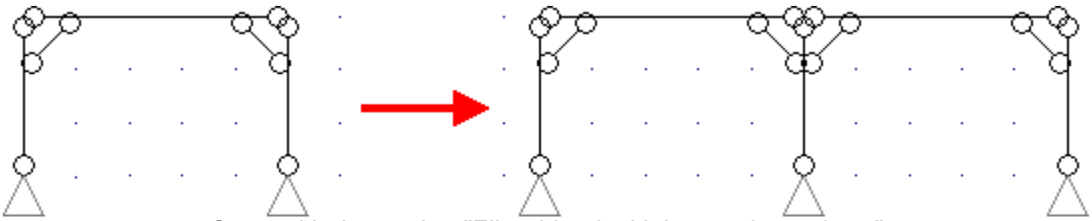
This tool is able to move or copy whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. Following example shows the behaviour. If this setting is switched off, it is necessary to copy only selected members (highlighted by green) in this structure. Otherwise, two frames (old one and new one) without any connection would be created.



*Copy without the setting "Filter identical joints and members"*

The setting "**Filter identical joints and members**" treats these problems. If switched on, the overlapping elements are checked and the frame can be copied as a whole structure. The overlapping column in the middle will be deleted, new frame and the old one will be connected into one structure.



*Copy with the setting "Filter identical joints and members"*

The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

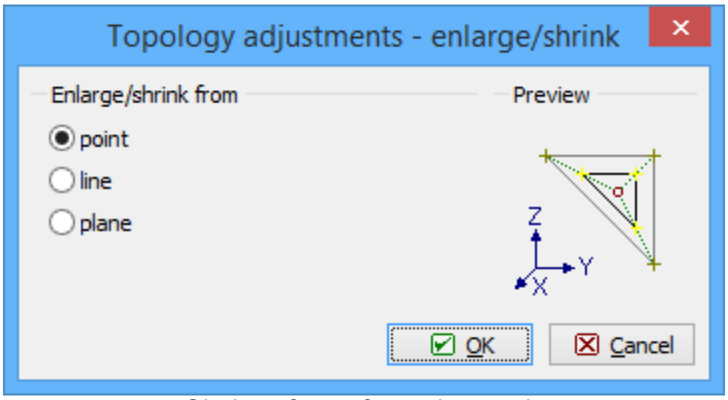
The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "" in the toolbar above the workspace.

The next entry is the number of copies. The bottom part of the window contains the vector of transformation divided into two components according to the global axes $X$, $Y$ and $Z$.

## Enlarge/Shrink

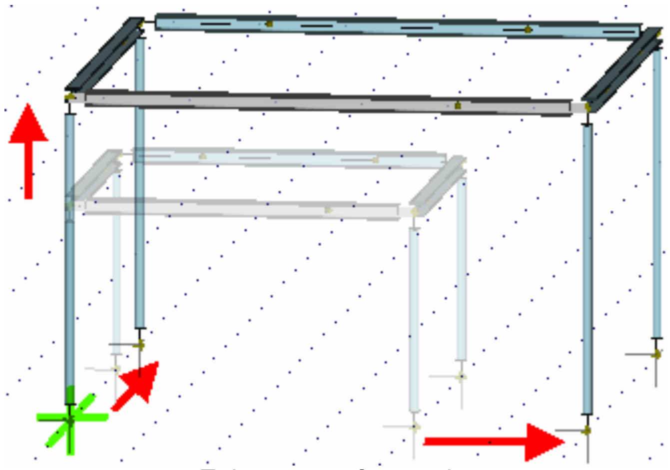
This tool can be used for enlargement or shrinkage of structure (or its part). There are three main modes of this tool: transformation relatively to point, line or plane. The choice of the mode has to be done in the window, that appears after clicking on the tool in the tree menu.



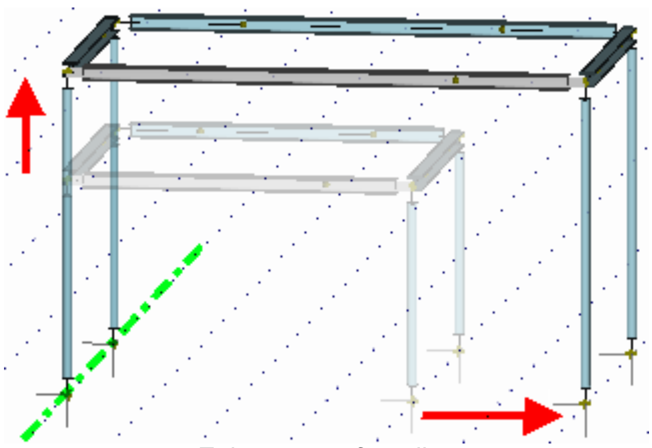

*Choice of transformation mode*

The transformation from point enlarges or shrinks the structure in all directions. The shape of the new structure is identical to the shape of the original structure.



*Enlargement from point*

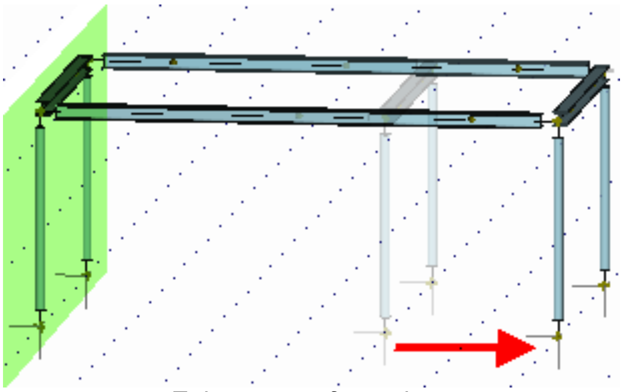
The transformation from line enlarges or shrinks the structure only in the directions perpendicular to the given line. The structure size in the direction of the line is not affected. The shape of the new structure differs from the shape of the original structure.



*Enlargement from line*

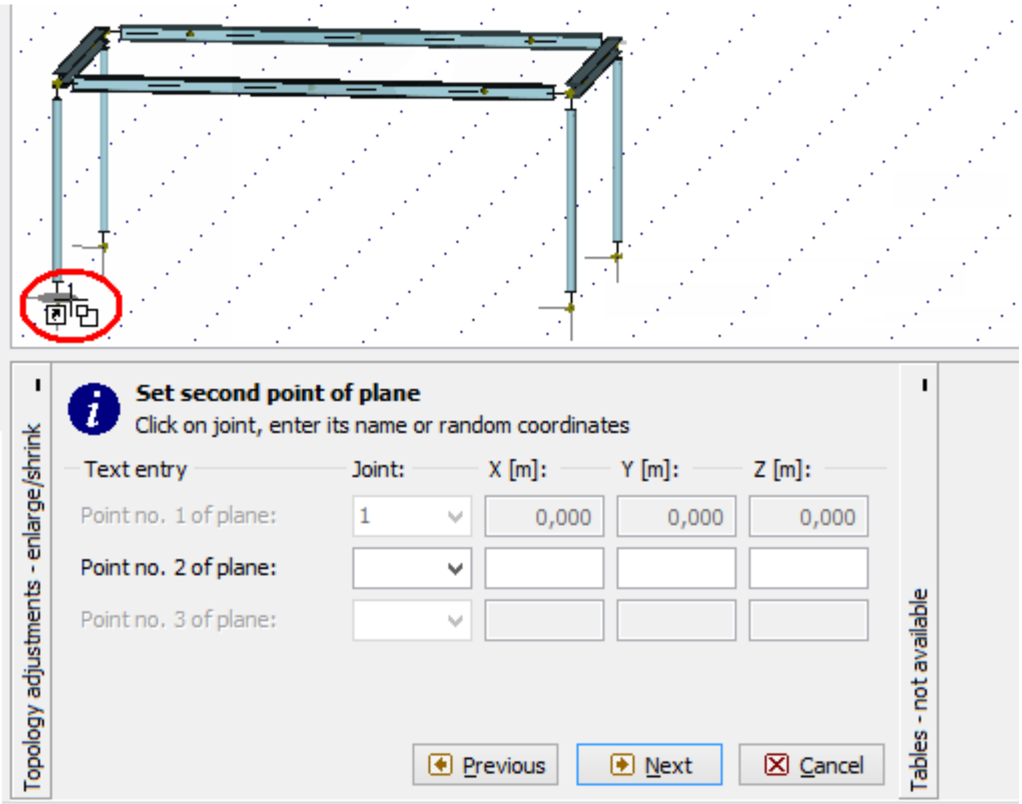
The transformation from plane enlarges or shrinks the structure only in the direction perpendicular to the defined plane. The structures sizes in plane are not affected. The shape of the new structure differs from the shape of the original structure.



*Enlargement from plane*

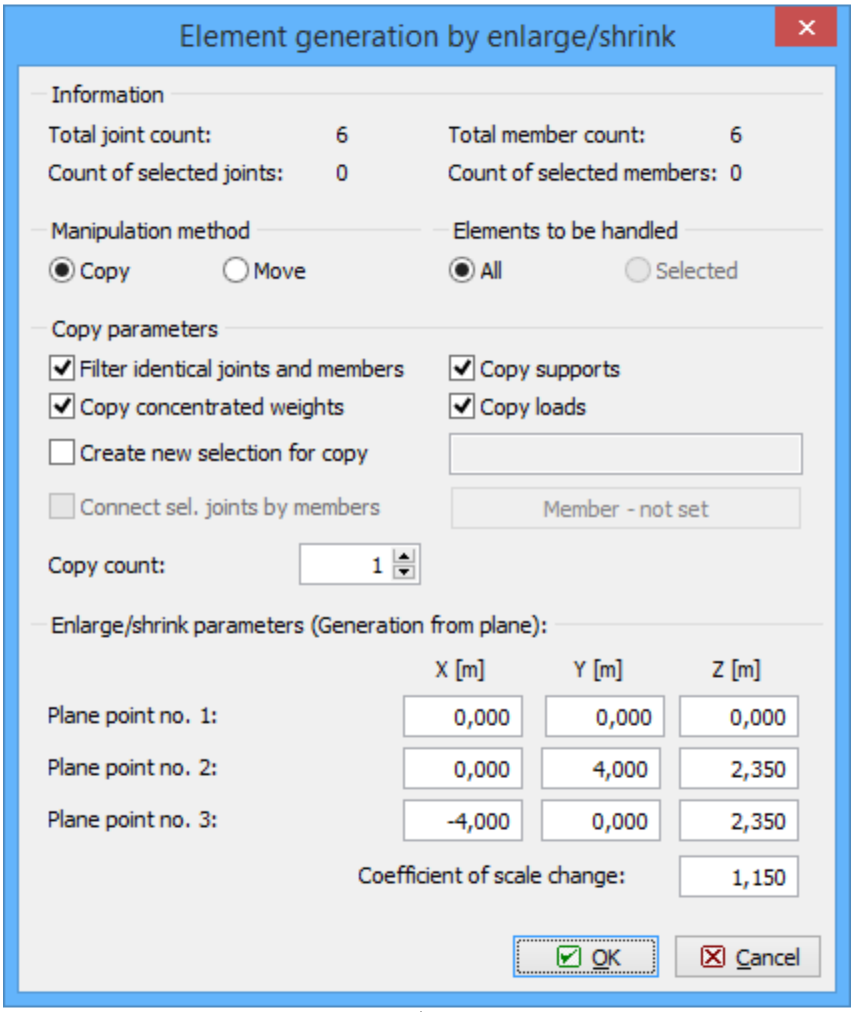
The input of reference point or line follows. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.




*Input of reference points in the workspace*

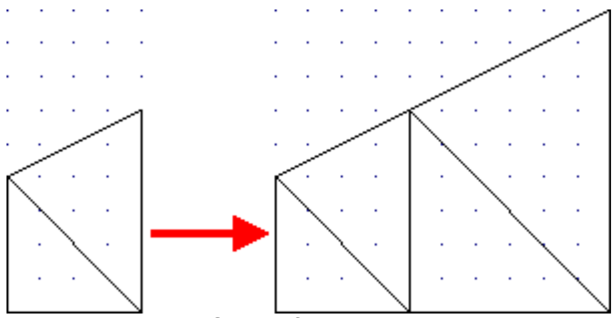
After that, the window with parameters of transformation appears.




*Window "Enlarge/shrink parameters"*

The "**Manipulation method**" sets, whether the tool will only change the shape of the structure (or its part) or will keep existing structure and create a new modified copy. The most of following settings are disabled for the option "**Move**" (only transformation parameters can be specified).



*Copy of structure*

This tool is able to enlarge or shrink whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. This setting is recommended. Otherwise, two structures (old one and new one) without any connection may be created.

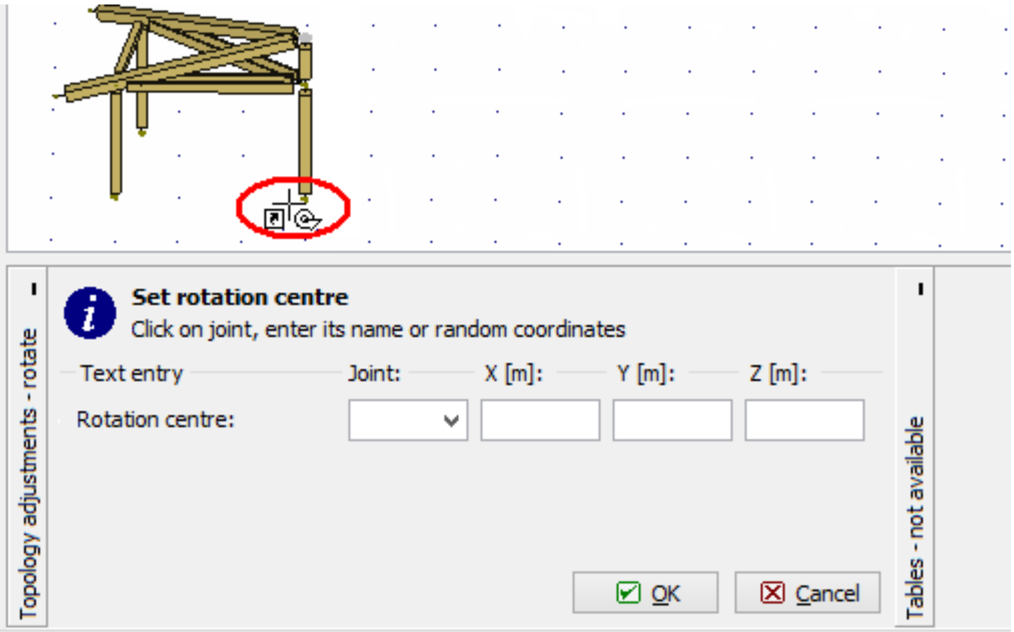
The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "" in the toolbar above the workspace.

The next entry is the number of copies. The bottom part of the window contains the reference point/line/plane coordinates and scale factor. Value greater than *1.0* enlarges the structure, smaller value reduces structure.

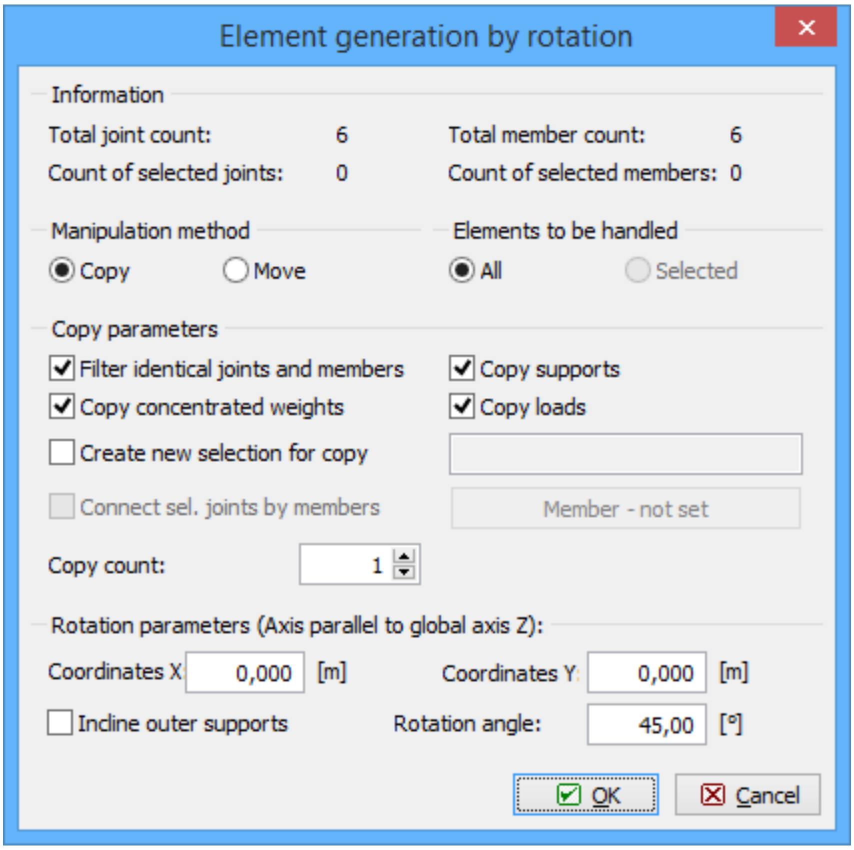
## Rotate

This tool can be used for rotation of structure (or its part). The rotation axis may be parallel to global axes *X*, *Y*, *Z* or defined with the help of two points. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.




*Input of rotation centre in the workspace*

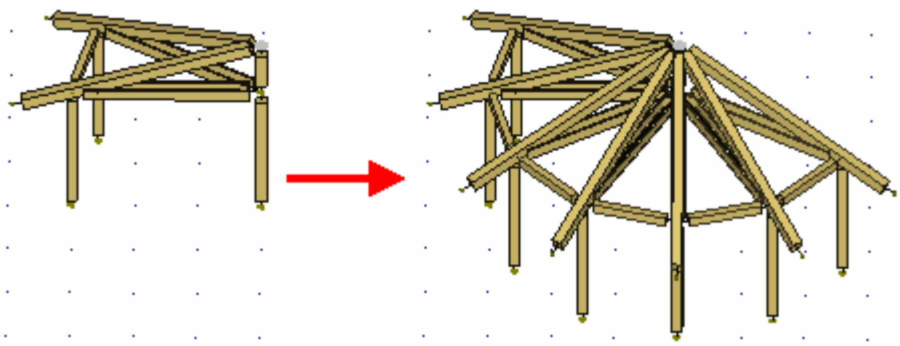
After that, the window with parameters of transformation appears.




*Window "Rotation parameters"*

The "**Manipulation method**" sets, whether the tool will only change the rotation of the structure (or its part) or will keep existing structure and create a new rotated copy (or copies). The most of following settings are disabled for the option "**Move**" (only transformation parameters can be specified).

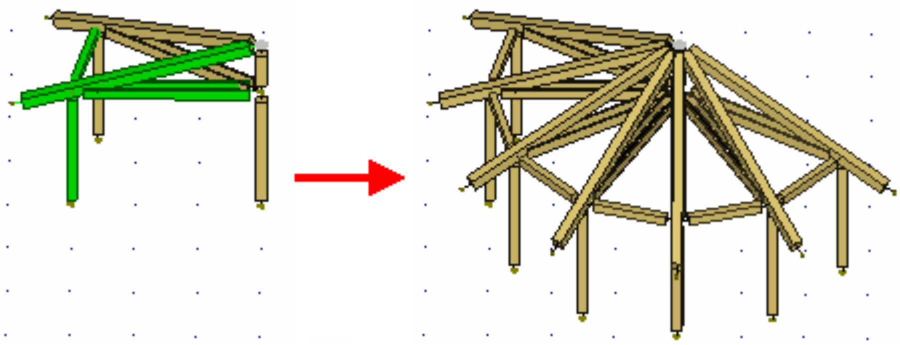
 



*Multiple copy of structure*

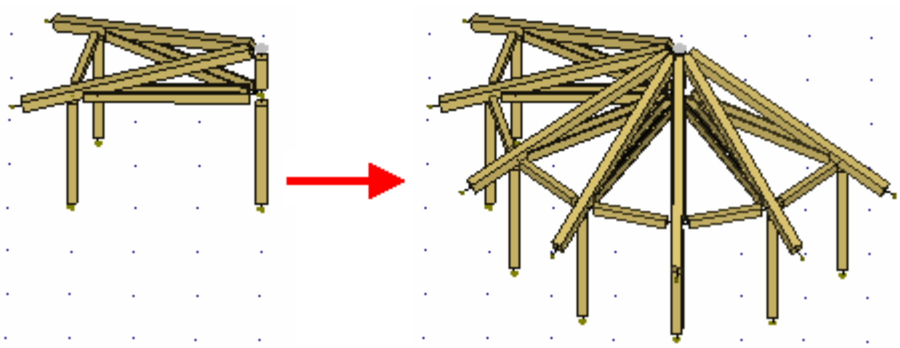
This tool is able to rotate or copy whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. Following example shows the behaviour. If this setting is switched off, it is necessary to copy only selected members (highlighted by green) in this structure. Otherwise, four parts without any connection would be created.



*Copy without the setting "Filter identical joints and members"*

The setting "**Filter identical joints and members**" treats these problems. If switched on, the overlapping elements are checked and the frame can be copied as a whole structure. The overlapping rays will be deleted, new frames and the old one will be connected into one structure.



*Copy with the setting "Filter identical joints and members"*

The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

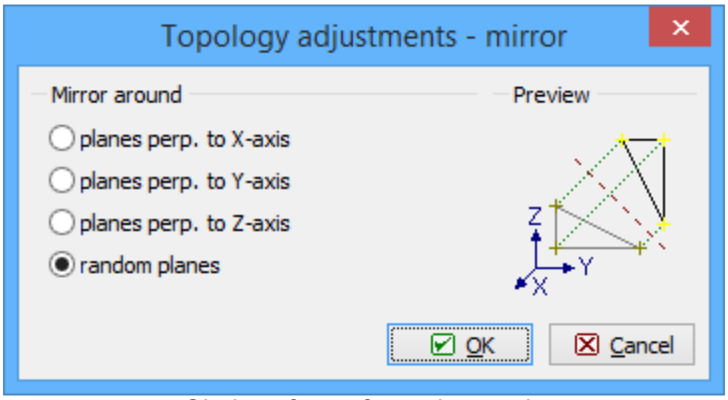
The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button " " in the toolbar above the workspace.

The next entry is the number of copies. The bottom part of the window contains the rotation axis coordinates and the rotation angle. Positive value means rotation in anti-clockwise direction.

## Mirror

This tool can be used for mirror of structure (or its part). There are four main modes of this tool: mirror using the planes perpendicular to main axes $X$, $Y$ or $Z$ or using a general mirror plane. The choice of the mode has to be done in the window, that appears after clicking on the tool in the tree menu.

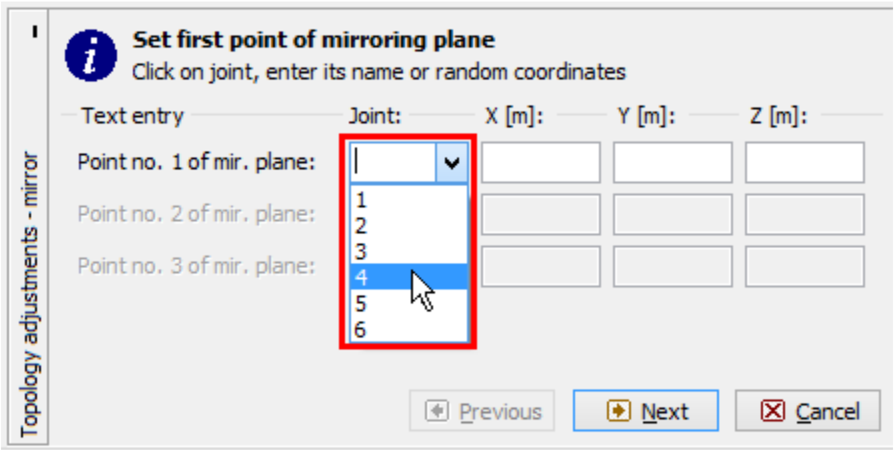
 



*Choice of transformation mode*

The plane for first three options is specified by the distance from the coordinate system origin on the corresponding global axis. The plane for the fourth option is given by three points.

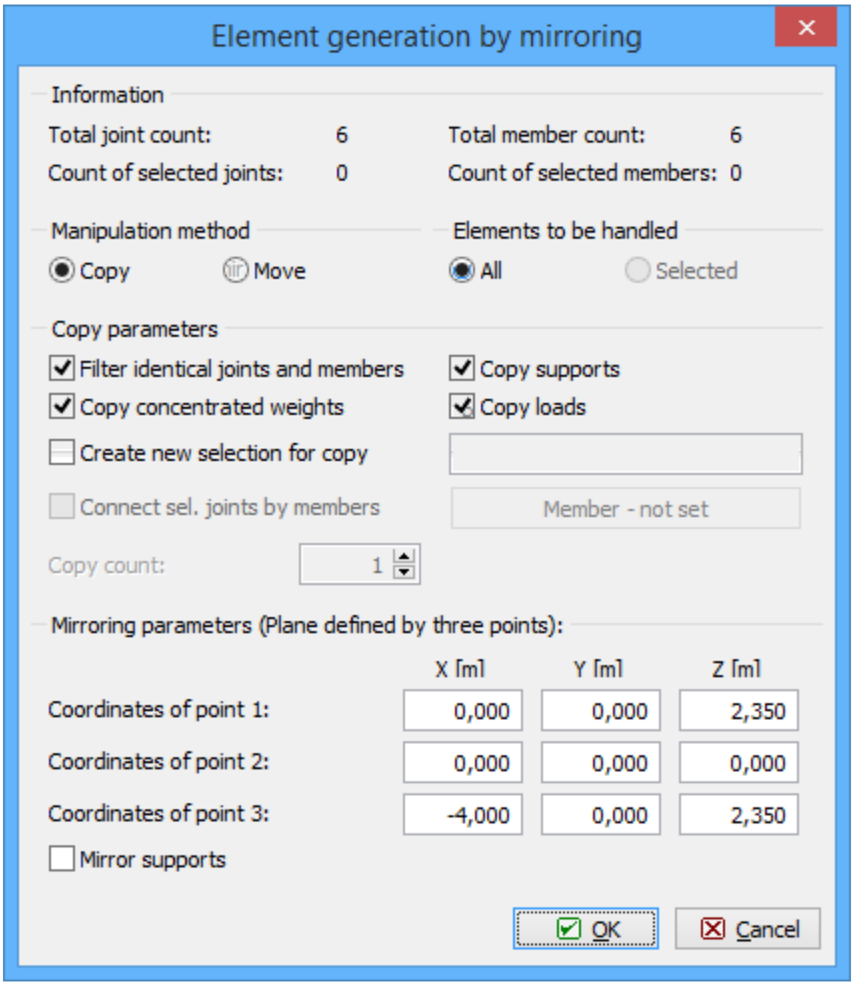
The input of plane follows. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.



*Choice of joint number in the input frame*

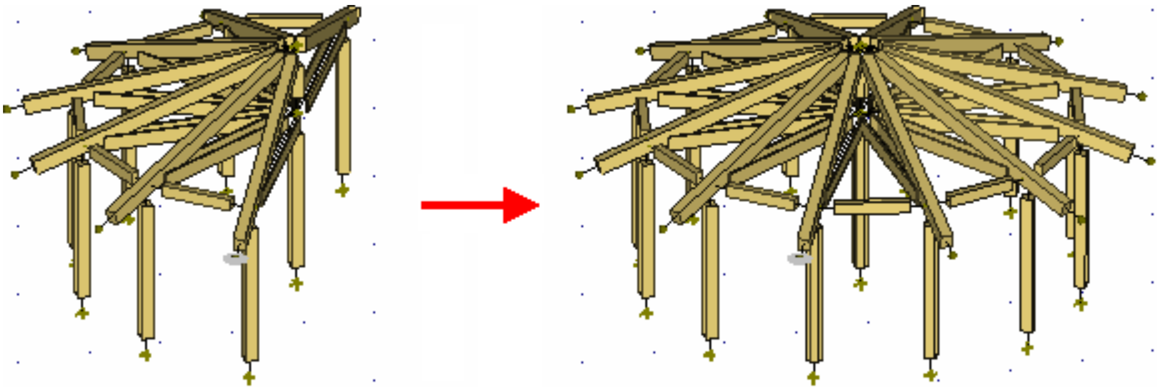
After that, the window with parameters of transformation appears.




*Window "Mirror parameters"*

The "**Manipulation method**" sets, whether the tool will only mirror the structure (or its part) or will keep existing structure and create a new copy. The most of following settings are disabled for the option "**Move**" (only transformation parameters can be specified).



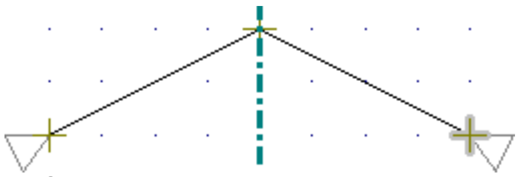
*Mode "Copy"*

This tool is able to mirror whole structure or only selected part. This behaviour can be specified in the part "**Elements for manipulation**". The option "**Selected**" is available only for structures, where are some selected members or joints (highlighted by green).

The setting "**Filter identical joints and members**" automatically filters and deletes overlapping joints and members, that may appear in the structure after applying the tool. This setting is recommended. Otherwise, two structures (old one and new one) without any connection may be created.

The load and supports may be also copied with members and joints when using appropriate settings "**Copy supports**" and "**Copy loads**".

The transferred structure may be also stored as a saved selection with the help of the setting "**Create new saved selection**". The saved selection is the list of joints and members, that may be selected in a batch easily with the help of the window "**Saved selections manager**". This window can be opened with the help of the button "  " in the toolbar above the workspace.



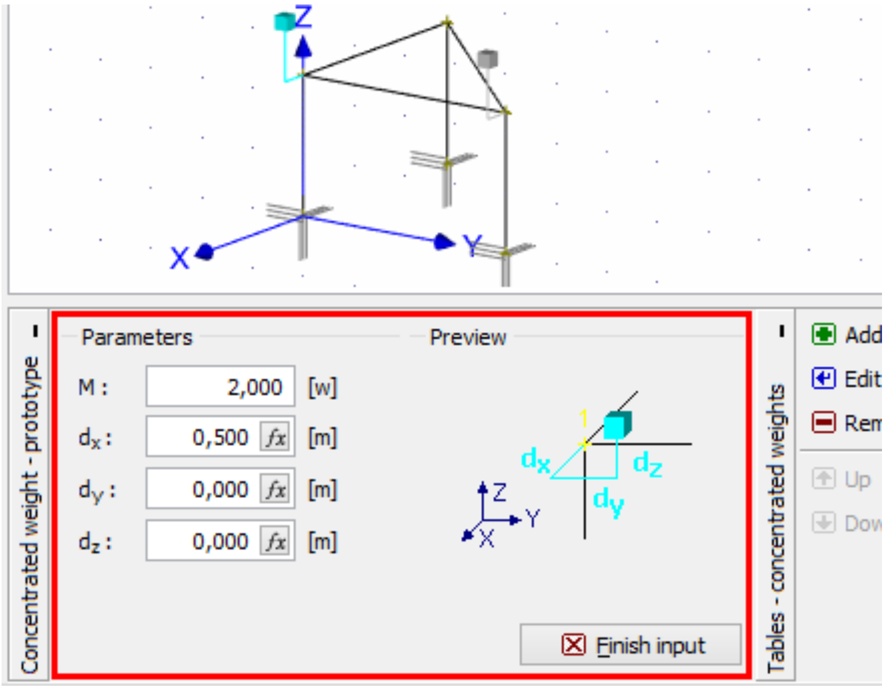
*Structure mirrored including supports*

# Dynamics

The concentrated masses are the masses that does not relate to the fundamental structure and affects the dynamic behaviour of the structure (heavy machinery etc.). The concentrated mass may be specified in arbitrary joint of the structure. It is defined by the additional weight in tons and the eccentricities in directions parallel to the main axes. Concentrated masses are taken into account in dynamic analysis (determination of eigenmodes).

## Input of concentrated masses

The concentrated loads can be inserted by clicking on the corresponding joint in the workspace. The mode "**Add**" in the part "**Dynamics**" of the tree menu has to be activated. The window "**Prototype of concentrated mass**" appears when starting the graphical input. This window contains properties of concentrated mass (mass and eccentricities) that will be applied to joints. The input range corresponds to the window "**Properties of concentrated mass**". The prototype window is moved into the bottom frame of the application window after the confirmation of input by the button "**OK**". It is possible to specify the joints in the workspace after that. The prototype properties may be changed arbitrarily in the bottom frame during the work.




*Prototype properties anchored in the input frame*

The concentrated masses may be also specified with the help of the table in the bottom part of the window. The range of inputs is identical, however, the reference joint number has to be entered additionally.

## Editing and removing concentrated masses

The concentrated masses may be modified or removed in the workspace after the selection of the appropriate graphical mode in the tree menu or with the help of the toolbar in the table in the bottom part of the window. The mass parameters are organized in the window "**Properties of concentrated mass**". More selected concentrated masses may be modified in a batch with the help of the tool "**Remove selected**" in the context menu of masses table.

# Properties of concentrated weight

This window contains parameters of concentrated weight. Concentrated weights are taken into account in dynamic analysis (determination of eigenmodes). The concentrated weights are weights, that does not relate to the fundamental structure, but affects the dynamic behaviour of the structure (heavy machinery etc.). The window contains following parameters:

**M** • The additional weight in tons

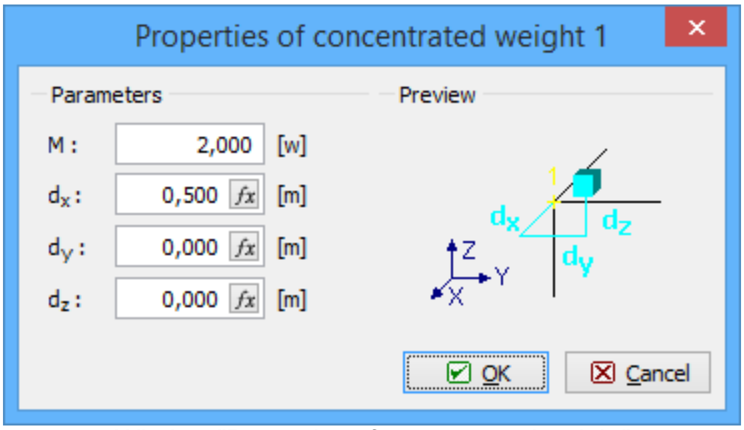
$\mathbf{d_x}$ • The eccentricity in the direction of the global axis $x$, relatively to the reference joint

**$d_y$** • The eccentricity in the direction of the global axis $y$, relatively to the reference joint

**$d_z$** • The eccentricity in the direction of the global axis $z$, relatively to the reference joint

The weight, that does not relate to the fundamental structure, but affects the dynamic behaviour of the structure, may be added with the help of concentrated weights. The specified weight is added into the joints. The weights may be entered including eccentricities relative to these joints.



*Window "Properties of concentrated weight"*

# Program Concrete

The software "**Concrete**" verifies reinforced concrete cross-sections of any shape according to EN 1992-1-1 and EN 1992-2.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Data**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Two types of particular tasks can be used in the software:

**Section** • Fast analysis of RC cross-section with unlimited number of loads.

**Member** • Analysis of the whole member with specified diagrams of internal forces. This type is suitable for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

These tasks can be added with the help of the buttons "**Add section**" and "**Add member**" in the heading of the tree menu.

## Main screen

Default screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

Part "**Calculation options**" contains settings that may influence the analysis:

**Check bar spacing** • The check of bar spacing is performed both for longitudinal and shear reinforcement. Verification is described in the part "**Structural rules**" of the theoretical help.

**Check of detailing only informative** • Structural rules (reinforcement area, bar spacing) are checked, however, they don't affect the final result "**Pass/Fail**". The member is considered as passed, even though some structural rules aren't fulfilled.

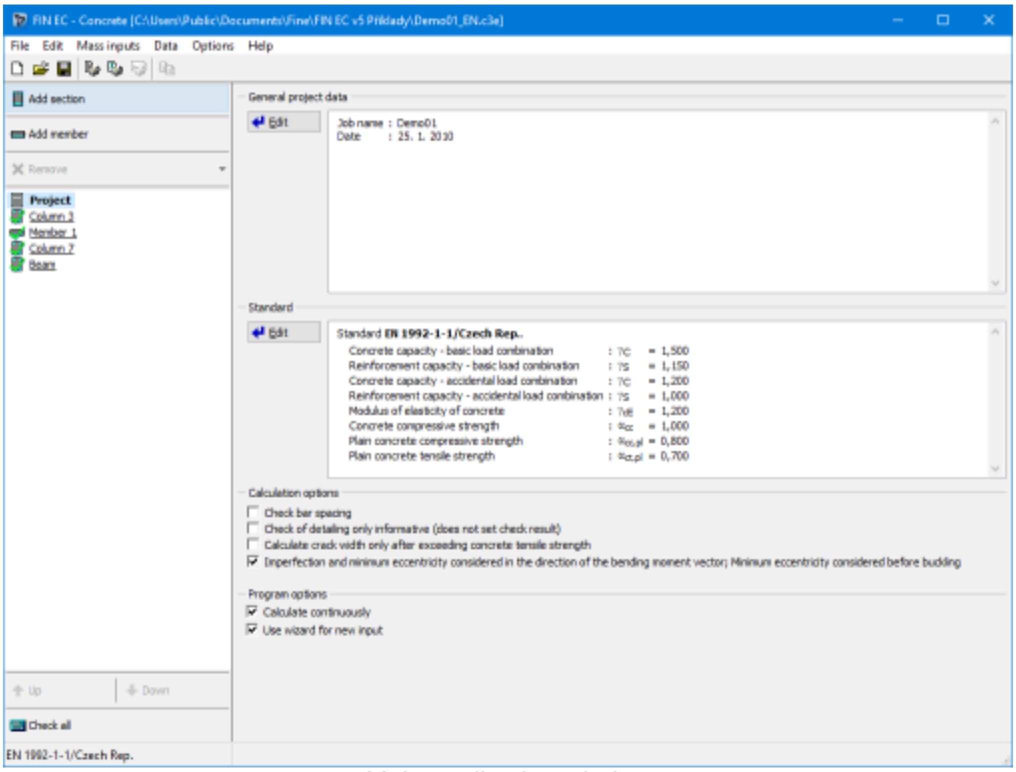
**Calculate crack width only after exceeding concrete tensile strength** • The crack width is calculated after the exceeding of limiting value $f_{ctm}$ (tensile strength of concrete). Cracks that may appear due to technological reasons aren't considered.

**Imeprfection and minimum eccentricity considered in the direction of the bending moment vector, minimum eccentricity considered before buckling** • This settings enables alternative way of analysis, which was used in older versions of the software as a default. Details are described in the part "**Ultimate limit state**" of the theoretical help.

| | |
|---|---|
| **Calculate continuously** | • The setting "**Calculate continuously**" recalculates the results after any change immediately. This may be quite time consuming and limiting for members with a lot of load combinations. When switched off, any part is recalculated after clicking on it in the tree menu. |
| **Use wizard for new input** | • The setting that runs all important windows at the beginning of the input of new task. The input is faster. |

The setting "**Calculate continuously**" recalculates the results after any change immediately. This may be quite time consuming and limiting for members with a lot of load combinations. When switched off, any part is recalculated after clicking on it in the tree-menu.



*Main application window*

# Section

Task type "**Section**" is suitable for the fast verification of the concrete cross-section, that is loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".

## Check type

The type of verification may be specified in this part. Following options are available:

- **2D** - Input and analysis are simplified to the one-axis bending. Graphical result of the analysis is the *M-N* diagram. This analysis is suitable for members which don't require biaxial analysis (e.g. beams, slabs, walls).
- **3D** - Advanced biaxial verification of the RC member including the torsional effect. Graphical result of the analysis is the $M_y$-$M_z$ diagram for specified $N$. This analysis is suitable for members which require biaxial analysis (e.g. columns, beams).
- **Plain** - Biaxial verification of the member made of plain concrete including the torsional effect. Graphical result of the analysis is the $M_y$-$M_z$ diagram for specified $N$.

## Section, Material, Reinforcement

This part contains the main characteristics of the cross-section that shall be specified first.

The fundamental parameter is the member type. The member type influences both the analysis and structural rules. Differences are described in the theoretical part of the help in the chapter "**Member types**".

The following characteristics are organized into the dedicated windows that may be launched by corresponding buttons. Some of them (mainly reinforcement) are disabled at the beginning. They require input of the previous parameters (geometry of the cross-section and material) as the launching mode of these windows depends on these characteristics.

| | |
|---|---|
| **Section** | • The input of geometry of the cross-section with the help of the database of pre-defined shapes in the window "**Cross-section editor**". |

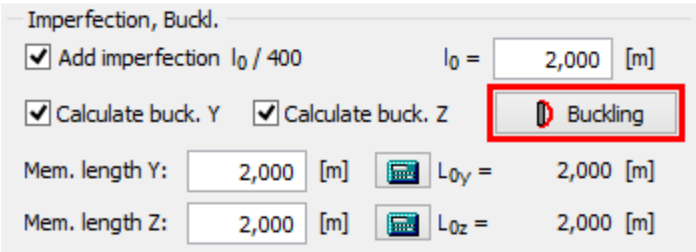
 

| | |
|---|---|
| **Polygon** | • The input of general geometry of the cross-section in the window "**General cross-section**". Not available for check type "**2D**". |
| **Editor** | • The input of general geometry of the cross-section in the program "**Section**". The general geometry including holes may be defined in this program. Not available for check type "**2D**". |
| **Material** | • The input of concrete strength class and steel grade in the window "**Materials**" |
| **Reinforcement** | • The parametric input of longitudinal reinforcement in the window "**Reinforcement**" |
| **General reinforcement** | • The general input of longitudinal reinforcement in the window "**Reinforcement - general cross-section**". Not available for check type "**2D**". |
| **Shear reinforcement** | • The input of the shear reinforcement in the window "**Shear reinforcement**" |
| **Include reinforcement in compression** | • The longitudinal reinforcement in compression may be also included in the analysis with the help of this setting. |

Cross-section view is active, mouse click on the cross-section launches the window for cross-section edit.

## Imperfection, Buckling

This part contains parameters of imperfection and buckling. The imperfection of $l_0/400$ may be considered in accordance with the chapter *5.2(9)* of EN 1992-1-1. The fundamental length $l_0$ has to be specified for the imperfection. This fundamental length $l_0$ is the real length of the member, not the buckling length. If the buckling analysis is switched on for certain direction, this value is automatically copied to the input fields for fundamental lengths for buckling analysis "**Mem. length Y**" and "**Mem. length Z**". These values may be rewritten without any change of the value $l_0$. The pinned supporting style is considered as a default, the buckling length is equal to the fundamental length in this case. The different supporting style for directions $Y$ and $Z$ may be selected in the window "**Buckling length**" that is available after clicking on the button "". The button "**Buckling**" opens the window "**Buckling**" that contains complete buckling parameters including the analysis method, creep factor etc.




*The button for opening the buckling properties*

## Cracks

This part of the member design contains inputs related to the crack calculation (serviceability limit state). The verification is performed only for load combinations "**Quasi-permanent (SLS)**". The maximum crack width $w_{max}$ is considered in accordance with table *7.1N*. Option for user defined value is also included. Setting "**Calculate crack width only at upper/bottom edge**" switches off the crack control on the cross-section sides. This setting is suitable for the verification of part of the structure (for example one linear meter of slab).

## Interaction diagram

This part shows the bearing capacity of the cross-section as the 3D object in the chart with axes $N$, $M_y$ and $M_z$ (or in the planar chart with axes $N$ and $M_y$ for verification type "**2D**"). Verification of loads within the displayed area is OK, verification of loads located outside the displayed area fails. The program is also able to show the most significant sections:

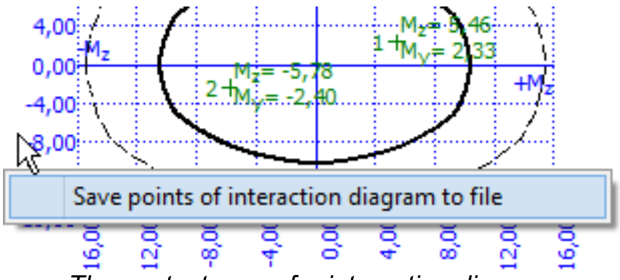
- **Interaction diagram $M_y$-$M_z$** - the horizontal section of the 3D diagram. The section is created for given axial force, which can be specified in the right upper corner. If not specified, the axial force for the active load is used.
- **Interaction diagrams N-$M_y$ or N-$M_z$** - the vertical sections of the 3D diagram. The sections are created for given bending moments $M_z$ or $M_y$, which can be specified in the right upper corner. If not specified, the corresponding bending moment for the active load is used.
- **Interaction diagram N-M** - the vertical sections of the 3D diagram. This section is given by the point *[0,0,0]* and by the point which represents the active load.

The button "**View**" in the right bottom part of the diagram opens the window, which contains the options for displaying the diagram with or without the effect of buckling and for changing the appearance of the diagram. In sections, the dashed line shows the bearing capacity of the cross-section, thick line shows the bearing capacity including the buckling consideration.

The interaction diagram is active and may be used for the insertion of new loads. The click into the interaction diagram inserts new load with the combination of internal forces according to the coordinates of the click. Cursor position is also displayed in the status bar. Other forces in the inserted load (shear forces, torsional moment) are equal to $0$.

The right button click in the interaction diagram opens a context menu that contains tool for exporting the coordinates of interaction diagram into **.csv* file.




*The context menu for interaction diagram*

## Loads - internal forces

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads may be also imported from text or **.csv* file. This feature is suitable for import of large number of loads that were calculated with the help of another structural engineering program. Import may be performed using the window "**Load import**" that may be launched by button "**Import**".

## Results

The overview of results for all loads are displayed in the right bottom part of the main window. Detailed results for the active load in the loads table may be displayed using button "**In detail**". These results are displayed in the new window, text in this window may be copied into clipboard using shortcut *Ctrl+C* and pasted into a document.

Analysis is described in the parts "**Ultimate limit state**" and "**Serviceability limit state**".

Results

**ULTIMATE LIMIT STATE**

| no. | Name | $N_{Ed}$ / $N_{Rd}$ [kN] | $M_{Edy}$ / $M_{Rdy}$ [kNm] | $M_{Edz}$ / $M_{Rdz}$ [kNm] | $V_{Edz}$ / $V_{Rdz}$ [kN] | $V_{Edy}$ / $V_{Rdy}$ [kN] | Utilization [%] | Analysis |
|---|---|---|---|---|---|---|---|---|
| 1 | Load 1 | -400,00 | 2,33 → 3,96 | 5,46 → 9,13 | 0,00 | 0,00 | 63,7 | Pass |
| | | -692,00 | 6,22 | 14,33 | 0,00 | 0,00 | | |

**STRESS LIMITATION**

| no. | Name | $N_{Ed}$ [kN] | $M_{Edy}$ [kNm] | $M_{Edz}$ [kNm] | $\sigma_c$ [MPa] | $\sigma_{s,max}$ [MPa] | $\sigma_{s,min}$ [MPa] | Utilization [%] | Analysis |
|---|---|---|---|---|---|---|---|---|---|
| 2 | Load 2 | -400,00 | -2,40 → -4,00 | -5,78 → -9,46 | 18,82 | -20,74 | 94,95 | 0,0 | Pass |
| Limit values $k_3 \times f_{yk}$ | | | | | | 400,00 | | | |

*Part "Results" of the cross-section design*

# Member

Task type "**Member**" is suitable for the detailed verification of the concrete member (e.g. beam, slab, column wall), that is loaded by unlimited number of loads. This task type is suitable mainly for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**". The fundamental input is the type of verification. Following options are available:

- **2D** - Input and analysis are simplified to the one-axis bending. This analysis is suitable for members which don't require biaxial analysis (e.g. beams, slabs, walls).
- **3D** - Advanced biaxial verification of the RC member including the torsional effect. This analysis is suitable for members which require biaxial analysis (e.g. columns, beams).
- **Plain** - Biaxial verification of the member made of plain concrete including the torsional effect.

The main frame of member design contains also these inputs:

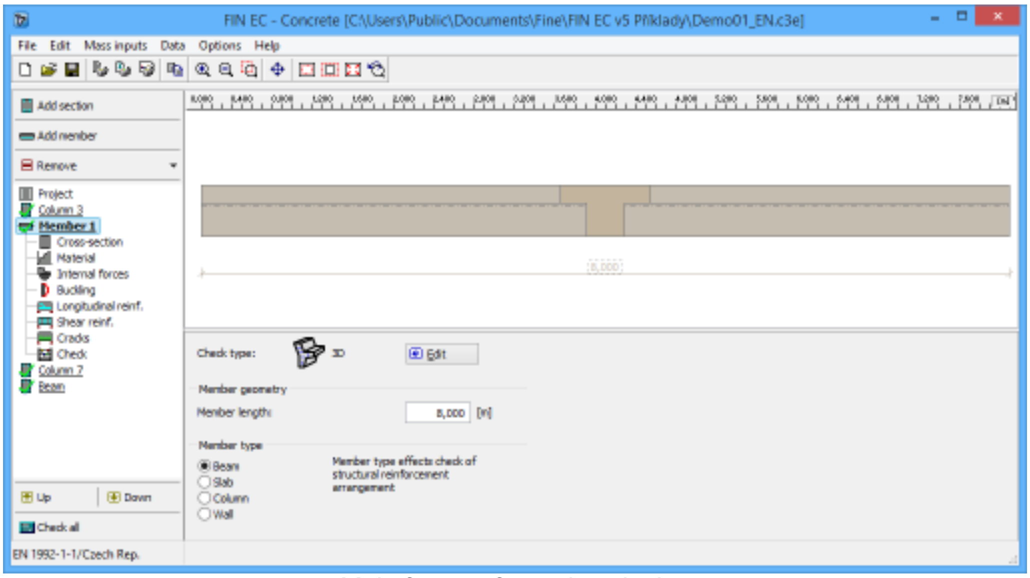
| | |
|---|---|
| **Member length** | • Total member length specified in metres. |
| **Member type** | • The member type influences both the analysis and structural rules. Differences are described in the theoretical part of the help in the chapter "**Member types**". |

The member design contains these parts:

- **Cross-section**
- **Material**
- **Internal forces**
- **Buckling**
- **Longitudinal reinforcement**
- **Shear reinforcement**
- **Cracks**
- **Analysis**

General work with members (addition, manipulation) is described in the chapter "**Tree menu**".



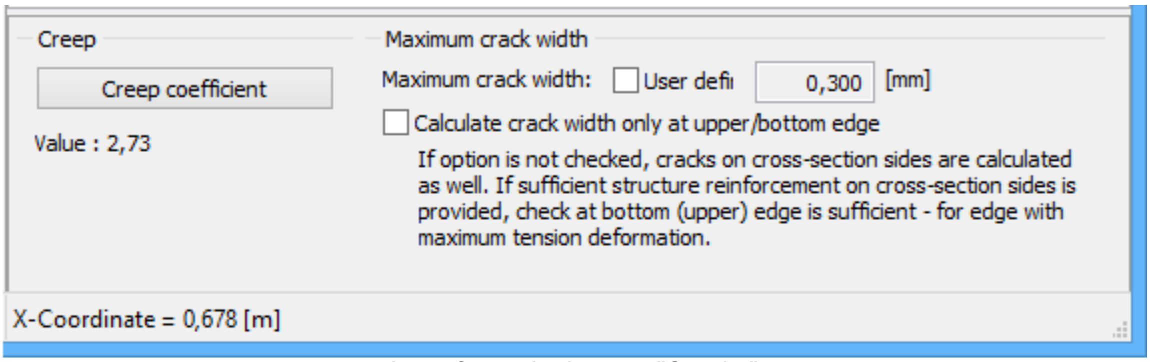

*Main frame of member design*

## Cracks

This part of the member design contains inputs related to the crack calculation (serviceability limit state). The verification is performed only for load combinations "**Quasi-permanent (SLS)**". The maximum crack width $w_{max}$ is considered in accordance with table *7.1N*. Option for user defined value is also included. Setting "**Calculate crack width only at upper/bottom edge**" switches off the crack control on the cross-section sides. This setting is suitable for the verification of part of the structure (for example one linear meter of slab).

Cracks may also arise from other causes such as plastic shrinkage or expansive chemical reactions within the hardened concrete (creeping). These factors are taken into account with the help of creep coefficient. The value of this coefficient can be changed in the window "**Creep**" that can be launched by the button "**Creep coefficient**".

Calculation and verification of cracks are described in the chapter "**Serviceability limit state**" of the theoretical help.

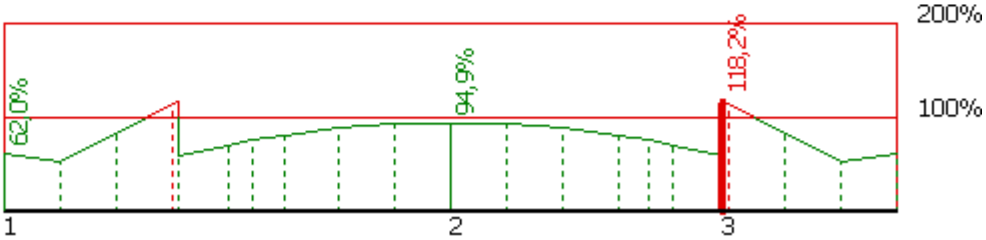



*Input frame in the part "Cracks"*

## Analysis

This part shows the results of structural analysis for the member. The results are displayed with the help of utilization diagram in the workspace. Passed member is coloured by green colour. The parts where the utilization exceeds 100% are coloured by red colour.




*Utilization of the member, red parts failed*

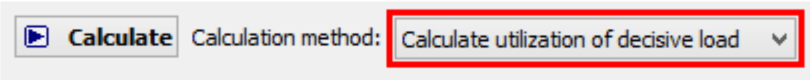
The frame in the bottom part contains tools for changing the analysis method and for the work with analysis sections (positions with detailed results).

## Analysis method

Analysis method can be selected in the upper part of the input frame. The analysis style and considered loads are selected according to the certain applied method. These options are available:

| | |
|---|---|
| **Calculate utilization of decisive load** | • Display the results for the decisive load (the load with the maximum utilization). All entered loads are considered in this option. |
| **Calculate envelope of maximum utilizations from all loads** | • Display envelope of the maximum utilization in every point of the member length. All entered loads are considered in this option. |
| **Individual loads** | • Show results for selected load. |

The analysis have to be run by the button "**Analyse**" after the change of the analysis method.




*Selection of analysis method*

## Verification sections

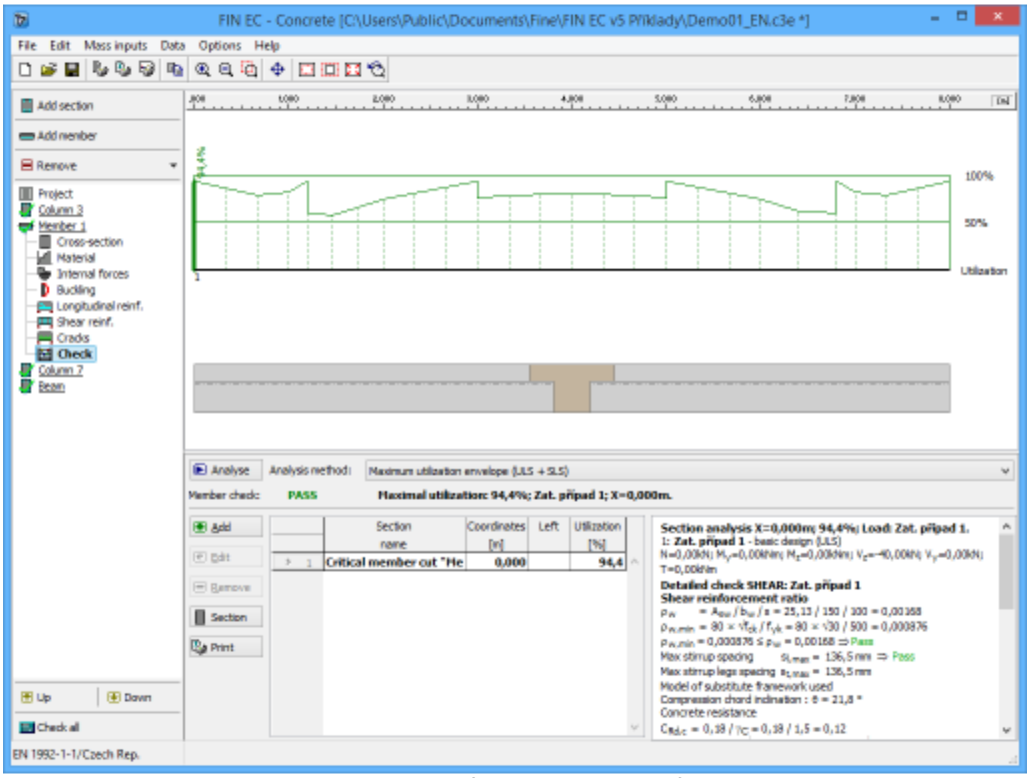
Verification sections are used for display of detailed results in certain points along the member length. These verification sections can be converted into standalone "**Section**" tasks, the results can be printed using graphical outputs. The verification section is created automatically in the position with the worst utilization, other sections can be added manually.

These tools are available for the work with sections:

| | |
|---|---|
| **Section** | • Converts the active section on member into standalone task of "**Section**" type. All member properties (cross-section, material, internal forces, parameters of buckling) are copied into the new task. |
| **Add** | • Inserts a new section on member. The detailed results can be displayed for this section. Properties of the new section have to be specified in the window "**New section for analysis**". |
| **Edit** | • Edit of the active cross-section properties (name, coordinate). |
| **Remove** | • Remove the active section from the list. |
| **Printing** | • Print results for all entered sections using printing window. |

The new section for analysis can be also added using active workspace. New section can be added by double-click on the needed position on the member.



*Part "Analysis" of the member verification*

# Program Concrete Fire

The software "**Concrete Fire**" verifies fire resistance of reinforced concrete cross-sections according to EN 1992-1-2.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Data**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Two types of particular tasks can be used in the software:

| | |
|---|---|
| **Section** | • Fast analysis of RC cross-section with unlimited number of loads. |
| **Member** | • Analysis of the whole member with specified diagrams of internal forces. This type is suitable for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**". |

These tasks can be added with the help of the buttons "**Add section**" and "**Add member**" in the heading of the tree menu.

## Main screen

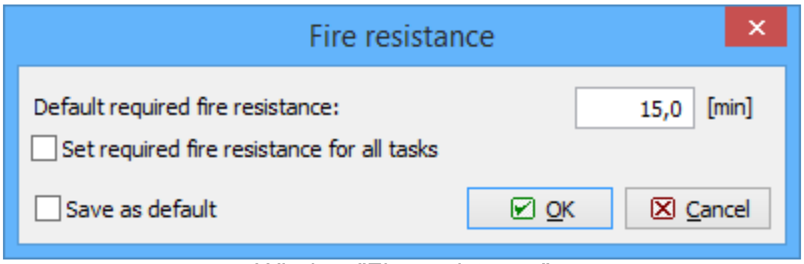
Default screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

The part "**Fire resistance parameters**" contains an option for input of default value of fire resistance for all tasks of the project. The input can be done in the window "**Fire resistance**", that can be launched by using the button "**Edit**". The check box **"Set required fire resistance for all tasks**" sets the specified fire resistance also to all existing tasks in the project. The setting "**Save as default**" will set specified fire resistance as a default for all new projects.



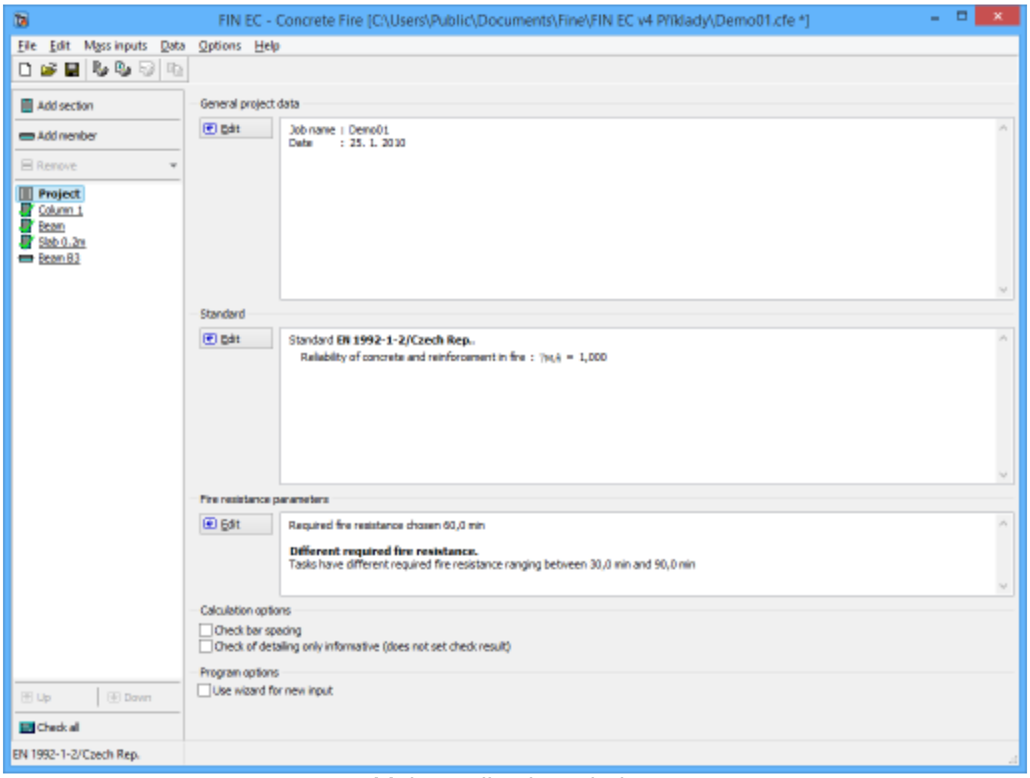

*Window "Fire resistance"*

Part "**Calculation options**" contains settings that may influence the analysis:

**Check bar spacing** • The check of bar spacing is performed both for longitudinal and shear reinforcement. Verification is described in the part "**Structural rules**" of the theoretical help.

**Check of detailing only informative** • Structural rules (reinforcement area, bar spacing) are checked, however, they don't affect the final result "**Pass/Fail**". The member is considered as passed, even though some structural rules aren't fulfilled.

The setting "**Use wizard for new input**" runs all important windows at the beginning of the input of new task. The input is faster.




*Main application window*

## Standard selection

The national annex for the standard EN 1992-1-2 can be selected in this window. Option "**Default EC**" sets the partial factors according to the design standard without any national annex. The value of the partial factor $\gamma_{M,fi}$ may be specified by user for the option "**User defined**". Values of factors for all available national annexes are written in the chapter "**National annexes**".

The minimum reinforcement area may be checked for member type "**Slab**" if the setting "**Minimum reinforcement ratio according to CSN 73 1201 - Chap.8.5.2**" is switched on. This verification is described in the chapter "**Structural rules**" of theoretical help.

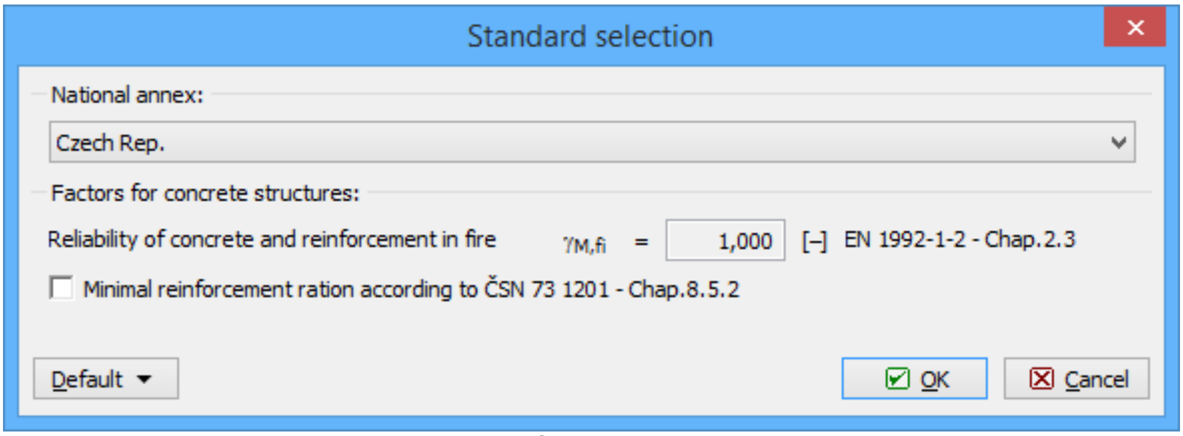
Button "**Default**" contains these two tools:

**Adopt default settings** • Set the default values for all parameters.

**Save settings as default** • Set entered parameters as defaults for new projects.

Partial factors are described in the theoretical part of the help.



*Window "Standard selection"*

# Section

Task type "**Section**" is suitable for the fast verification of the concrete cross-section, that is loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".

## Section, Material, Reinforcement

This part contains the main characteristics of the cross-section that shall be specified first.

The fundamental parameter is the member type. The member type influences both the analysis and structural rules. Differences are described in the theoretical part of the help in the chapter "**Member types**".

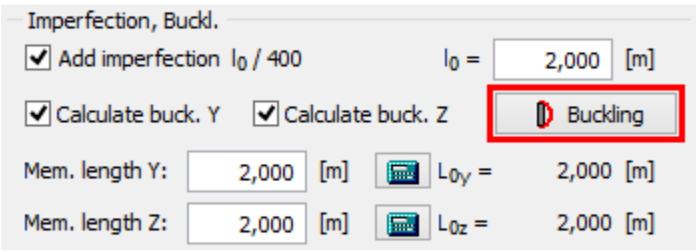
The following characteristics are organized into the dedicated windows that may be launched by corresponding buttons. Some of them (mainly reinforcement) are disabled at the beginning. They require input of the previous parameters (geometry of the cross-section and material) as the launching mode of these windows depends on these characteristics.

| | |
|---|---|
| **Section** | • The input of geometry of the cross-section with the help of the database of pre-defined shapes in the window "**Cross-section editor**". |
| **Material** | • The input of concrete strength class and steel grade in the window "**Materials**" |
| **Reinforcement** | • The parametric input of longitudinal reinforcement in the window "**Reinforcement**" |
| **General reinforcement** | • The general input of longitudinal reinforcement in the window "**Reinforcement - general cross-section**". Not available for check type "**2D**". |
| **Shear reinforcement** | • The input of the shear reinforcement in the window "**Shear reinforcement**" |
| **Include reinforcement in compression** | • The longitudinal reinforcement in compression may be also included in the analysis with the help of this setting. |

Cross-section view is active, mouse click on the cross-section launches the window for cross-section edit.

## Imperfection, Buckling

This part contains parameters of imperfection and buckling. The imperfection of $l_0/400$ may be considered in accordance with the chapter *5.2(9)* of EN 1992-1-1. The fundamental length $l_0$ has to be specified for the imperfection. This fundamental length $l_0$ is the real length of the member, not the buckling length. If the buckling analysis is switched on for certain direction, this value is automatically copied to the input fields for fundamental lengths for buckling analysis "**Mem. length Y**" and "**Mem. length Z**". These values may be rewritten without any change of the value $l_0$. The pinned supporting style is considered as a default, the buckling length is equal to the fundamental length in this case. The different supporting style for directions *Y* and *Z* may be selected in the window "**Buckling length**" that is available after clicking on the button "". The button "**Buckling**" opens the window "**Buckling**" that contains complete buckling parameters including the analysis method, creep factor etc.



*The button for opening the buckling properties*

## Fire

This part contains parameters related to the fire resistance analysis:

**Limit fire resistance period**
- Fire resistance in minutes, the verification is performed for this time.

**Method**
- The choice of the fire resistance method. There are two available methods according to the annex *B* of EN 1992-1-2: "**500° isotherm method**" and "**Zone method**". Both methods are described in the part "**Methods for fire resistance analysis**" of the theoretical help.

**Temperature curve**
- The choice of the temperature curve, that is used for the determination of the temperature of gas in time, in the window "**Temperature curve**". Properties are described in the theoretical part of the help in the chapter "**Temperature curve**". The range of available temperature curves according to the methods of analysis is described in the chapter "**Methods for fire resistance analysis**".

**Fire detail**
- The fire detail (number of edges that are exposed to fire) may be selected here. The button launches new window "**Fire detail**".

## Interaction diagram

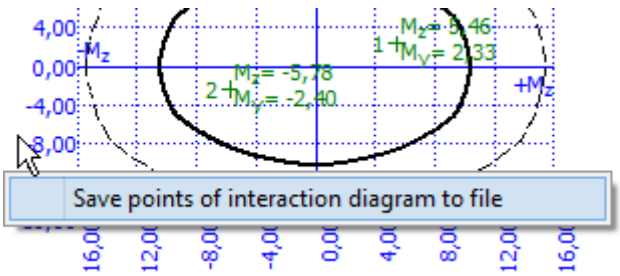
This part shows the bearing capacity of the cross-section as the 3D object in the chart with axes $N$, $M_y$ and $M_z$ (or in the planar chart with axes $N$ and $M_y$ for verification type "**2D**"). Verification of loads within the displayed area is OK, verification of loads located outside the displayed area fails. The program is also able to show the most significant sections:

- **Interaction diagram $M_y$-$M_z$** - the horizontal section of the 3D diagram. The section is created for given axial force, which can be specified in the right upper corner. If not specified, the axial force for the active load is used.
- **Interaction diagrams N-$M_y$ or N-$M_z$** - the vertical sections of the 3D diagram. The sections are created for given bending moments $M_z$ or $M_y$, which can be specified in the right upper corner. If not specified, the corresponding bending moment for the active load is used.
- **Interaction diagram N-M** - the vertical sections of the 3D diagram. This section is given by the point *[0,0,0]* and by the point which represents the active load.

The button "**View**" in the right bottom part of the diagram opens the window, which contains the options for displaying the diagram with or without the effect of buckling and for changing the appearance of the diagram. In sections, the dashed line shows the bearing capacity of the cross-section, thick line shows the bearing capacity including the buckling consideration.

The interaction diagram is active and may be used for the insertion of new loads. The click into the interaction diagram inserts new load with the combination of internal forces according to the coordinates of the click. Cursor position is also displayed in the status bar. Other forces in the inserted load (shear forces, torsional moment) are equal to *0*.

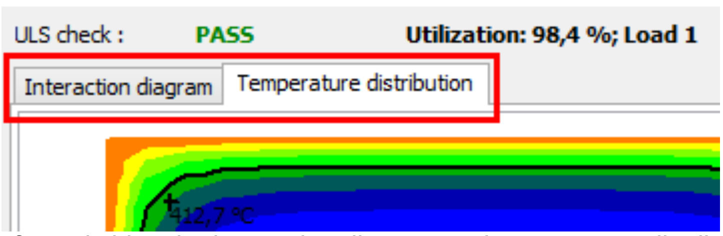
The right button click in the interaction diagram opens a context menu that contains tool for exporting the coordinates of interaction diagram into **.csv* file.




*The context menu for interaction diagram*

## Temperature distribution

The thermal distribution in the cross-section may be displayed instead of interaction diagram with the help of the tab "**Temperature distribution**".



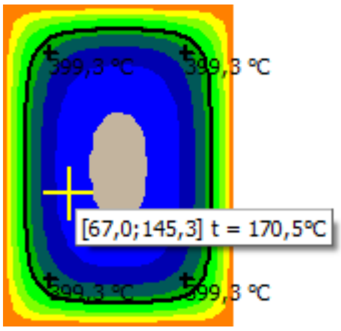

*Tabs for switching the interaction diagram and temperature distribution*

The temperature distribution for specified fire resistance is displayed with the help of isolines. Following settings may effect the drawing:

| | |
|---|---|
| **Draw effective section** | • The shape of the cross-section that is considered in the analysis is drawn by the bold line |
| **Draw reinforcement temperature** | • The temperatures for all longitudinal bars are displayed |
| **Draw stirrups temperature** | • The temperatures for all transverse bars (stirrups, links) are displayed |

The cursor is able to show the temperature for specified fire resistance in any point of the cross-section.




*The displayed temperature in the point of cursor*

## Loads - internal forces

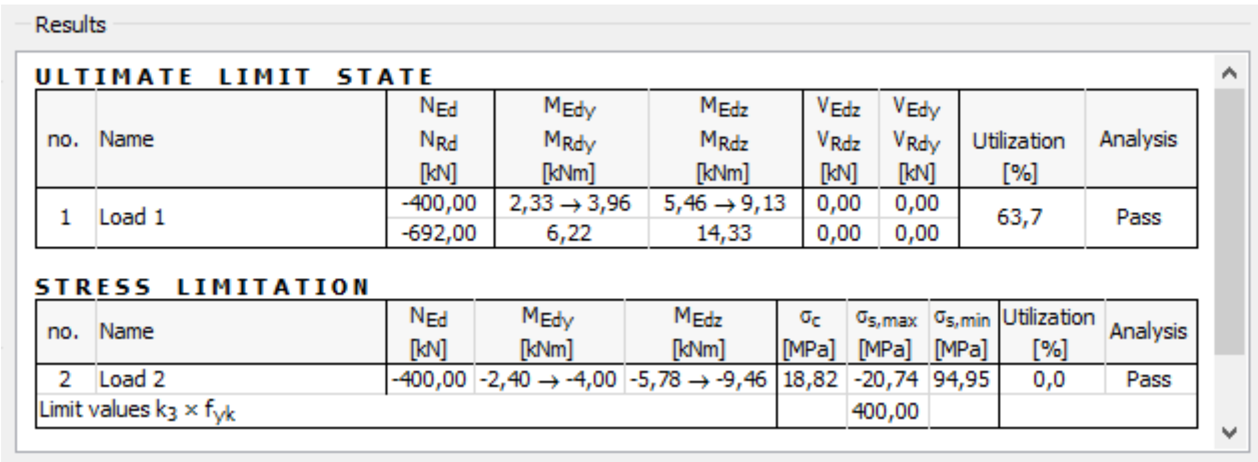
This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads may be also imported from text or **.csv* file. This feature is suitable for import of large number of loads that were calculated with the help of another structural engineering program. Import may be performed using the window "**Load import**" that may be launched by button "**Import**".

## Results

The overview of results for all loads are displayed in the right bottom part of the main window. Detailed results for the active load in the loads table may be displayed using button "**In detail**". These results are displayed in the new window, text in this window may be copied into clipboard using shortcut *Ctrl+C* and pasted into a document.

Analysis is described in the theoretical part of the help.



Results

ULTIMATE LIMIT STATE

| no. | Name | $N_{Ed}$ / $N_{Rd}$ [kN] | $M_{Edy}$ / $M_{Rdy}$ [kNm] | $M_{Edz}$ / $M_{Rdz}$ [kNm] | $V_{Edz}$ / $V_{Rdz}$ [kN] | $V_{Edy}$ / $V_{Rdy}$ [kN] | Utilization [%] | Analysis |
|---|---|---|---|---|---|---|---|---|
| 1 | Load 1 | -400,00 | 2,33 → 3,96 | 5,46 → 9,13 | 0,00 | 0,00 | 63,7 | Pass |
| | | -692,00 | 6,22 | 14,33 | 0,00 | 0,00 | | |

STRESS LIMITATION

| no. | Name | $N_{Ed}$ [kN] | $M_{Edy}$ [kNm] | $M_{Edz}$ [kNm] | $\sigma_c$ [MPa] | $\sigma_{s,max}$ [MPa] | $\sigma_{s,min}$ [MPa] | Utilization [%] | Analysis |
|---|---|---|---|---|---|---|---|---|---|
| 2 | Load 2 | -400,00 | -2,40 → -4,00 | -5,78 → -9,46 | 18,82 | -20,74 | 94,95 | 0,0 | Pass |
| Limit values $k_3 \times f_{yk}$ | | | | | | 400,00 | | | |

*Part "Results" of the cross-section design*

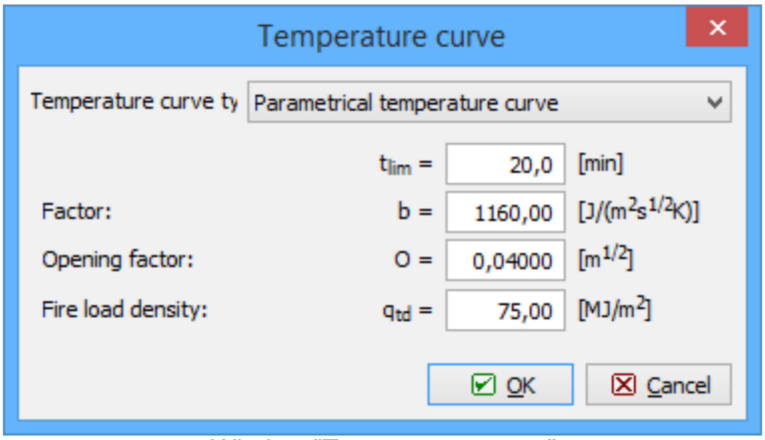
# Temperature curve

The temperature curve that is used for the determination of the temperature of gas in time may be selected here. Following options are available:

- **Standard temperature curve** - nominal curve defined in EN 13501-2. This curve describes the fully developed fire.
- **Parametric temperature curve** - this curve is effected by the physical parameters that describe the conditions in the fire compartment. This curve isn't available for "**Zone method**".

The expressions that represent temperature curves are described in the chapter "**Temperature curves**" of theoretical help.

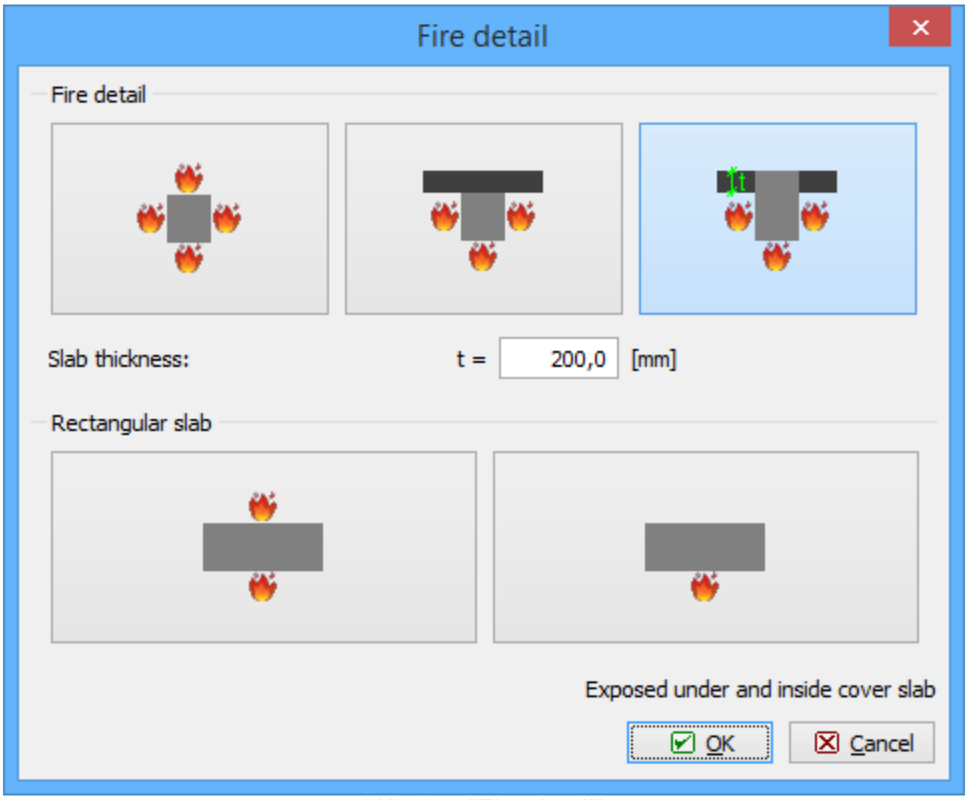
 




*Window "Temperature curve"*

# Fire detail

The type of fire detail may be specified in this window. Details differ according to the number of sides exposed to fire. Individual category "**Rectangular slab**" is created for cases where the verification is done only for part of the structure (for example one linear meter of slab). Vertical edges aren't exposed to fire in these cases.




*Window "Fire detail"*

# Member

Task type "**Member**" is suitable for the detailed verification of the concrete member (e.g. beam, slab, column wall), that is loaded by unlimited number of loads. This task type is suitable mainly for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

The main frame of member design contains also these inputs:

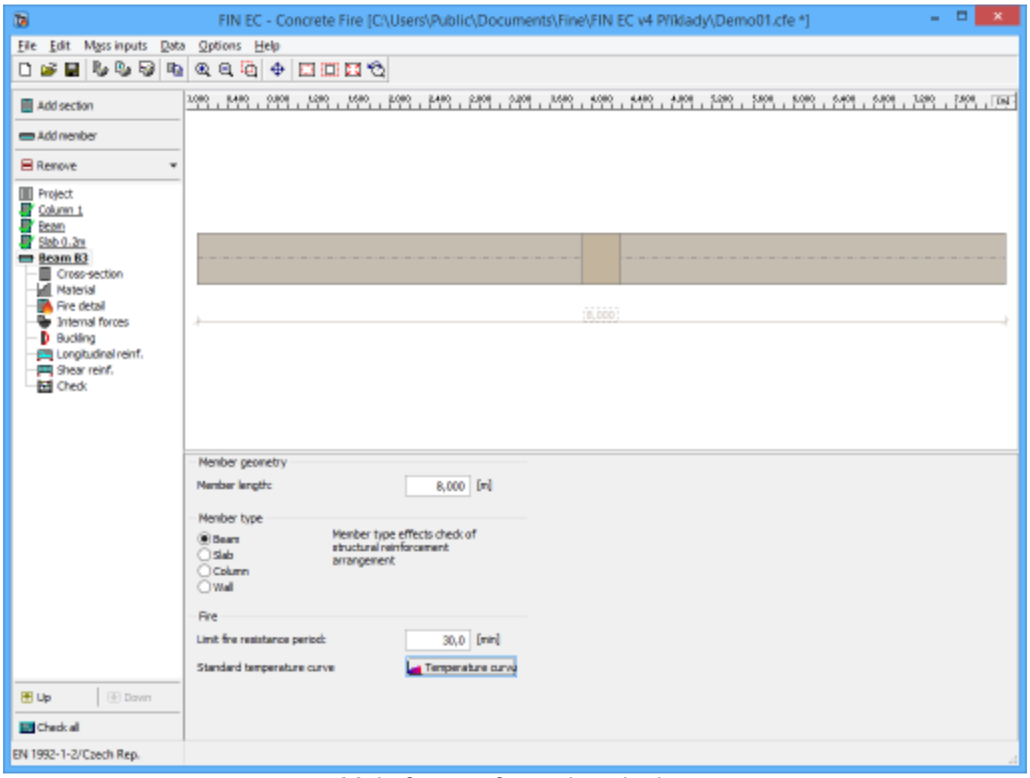
| | |
|---|---|
| **Member length** | • Total member length specified in metres. |
| **Member type** | • The member type influences both the analysis and structural rules. Differences are described in the theoretical part of the help in the chapter "**Member types**". |
| **Limit fire resistance period** | • Fire resistance in minutes, the verification is performed for this time. |

**Temperature curve**
- The choice of the temperature curve, that is used for the determination of the temperature of gas in time, in the window "**Temperature curve**". Properties are described in the theoretical part of the help in the chapter "**Temperature curves**". The range of available temperature curves according to the methods of analysis is described in the chapter "**Methods for fire resistance analysis**".

The member design contains these parts:

- **Cross-section**
- **Material**
- **Fire detail**
- **Internal forces**
- **Buckling**
- **Longitudinal reinforcement**
- **Shear reinforcement**
- **Analysis**

General work with members (addition, manipulation) is described in the chapter "**Tree menu**".



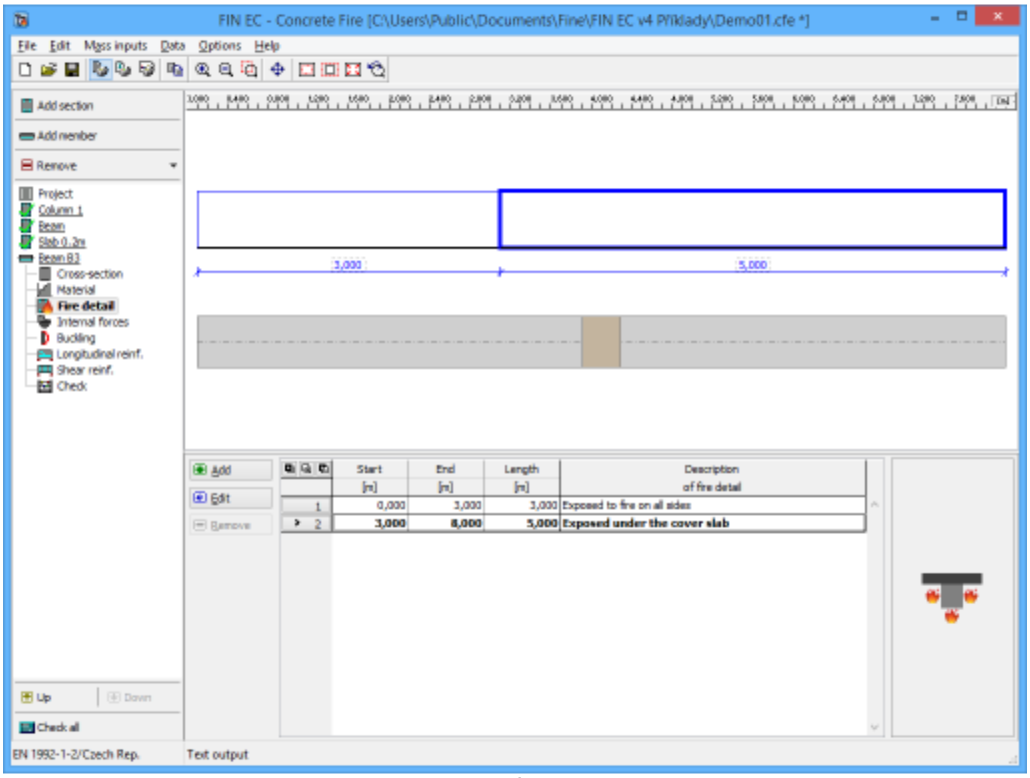

*Main frame of member design*

# Fire detail

The fire detail may be specified in this part of the tree menu. The detail can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different fire detail. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The fire detail properties are organized in the window "**Fire detail**". More sectors can be added (button "**Add**") for input of different fire detail properties along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

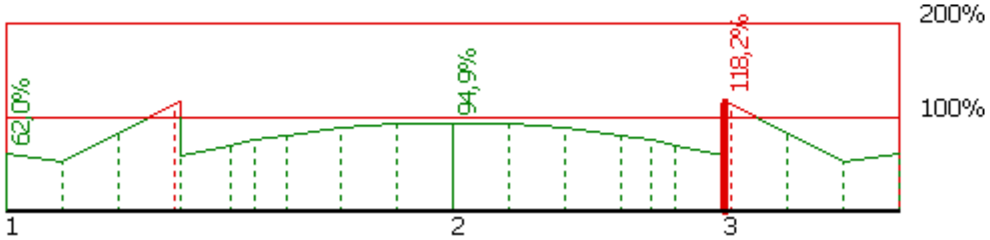
 



*Part "Fire detail" of member analysis*

# Analysis

This part shows the results of structural analysis for the member and specified time of fire resistance. The results are displayed with the help of utilization diagram in the workspace. Passed member is coloured by green colour. The parts where the utilization exceeds 100% are coloured by red colour.




*Utilization of the member, red parts failed*

The frame in the bottom part contains tools for changing the analysis method and for the work with analysis sections (positions with detailed results).

## Analysis method

The fire resistance method and the analysis method may be selected in the upper part of the input frame. There are two methods for the analysis of fire resistance according to the annex *B* of EN 1992-1-2: "**500° isotherm method**" and "**Zone method**". Both methods are described in the part "**Methods for fire resistance analysis**" of the theoretical help.

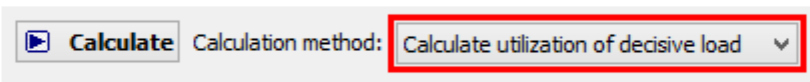
Analysis method can be selected in the upper part of the input frame. The analysis style and considered loads are selected according to the certain applied method. These options are available:

**Calculate utilization of decisive load**
- Display the results for the decisive load (the load with the maximum utilization). All entered loads are considered in this option.

**Calculate envelope of maximum utilizations from all loads**
- Display envelope of the maximum utilization in every point of the member length. All entered loads are considered in this option.

**Individual loads**
- Show results for selected load.

The analysis have to be run by the button "**Analyse**" after the change of the analysis method.




*Selection of analysis method*

## Verification sections

Verification sections are used for display of detailed results in certain points along the member length. These verification sections can be converted into standalone "**Section**" tasks, the results can be printed using graphical outputs. The verification section is created automatically in the position with the worst utilization, other sections can be added manually.

These tools are available for the work with sections:

**Section** • Converts the active section on member into standalone task of "**Section**" type. All member properties (cross-section, material, internal forces, parameters of buckling) are copied into the new task.

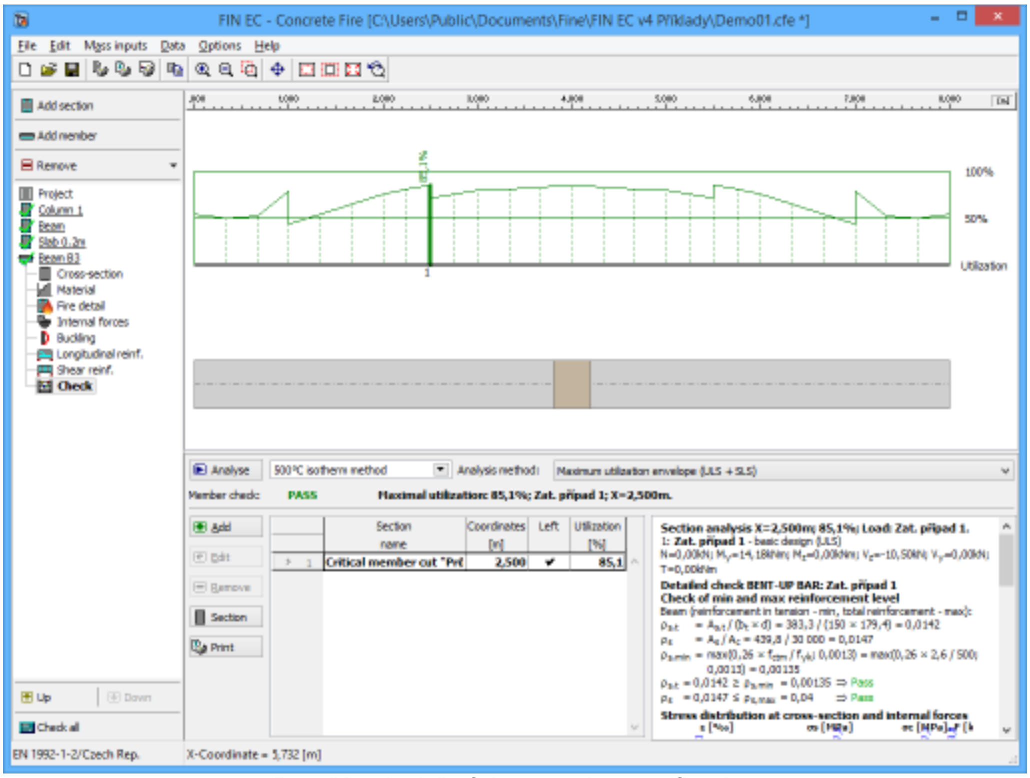
**Add** • Inserts a new section on member. The detailed results can be displayed for this section. Properties of the new section have to be specified in the window "**New section for analysis**".

**Edit** • Edit of the active cross-section properties (name, coordinate).

**Remove** • Remove the active section from the list.

**Printing** • Print results for all entered sections using printing window.

The new section for analysis can be also added using active workspace. New section can be added by double-click on the needed position on the member.



*Part "Analysis" of the member verification*

# Program Concrete Beam

The software "**Concrete Beam**" verifies horizontal reinforced concrete structures according to EN 1992-1-1.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Data**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

The particular tasks ("**Members**") can be added with the help of the button "**Add member**" in the heading of the tree menu.

## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

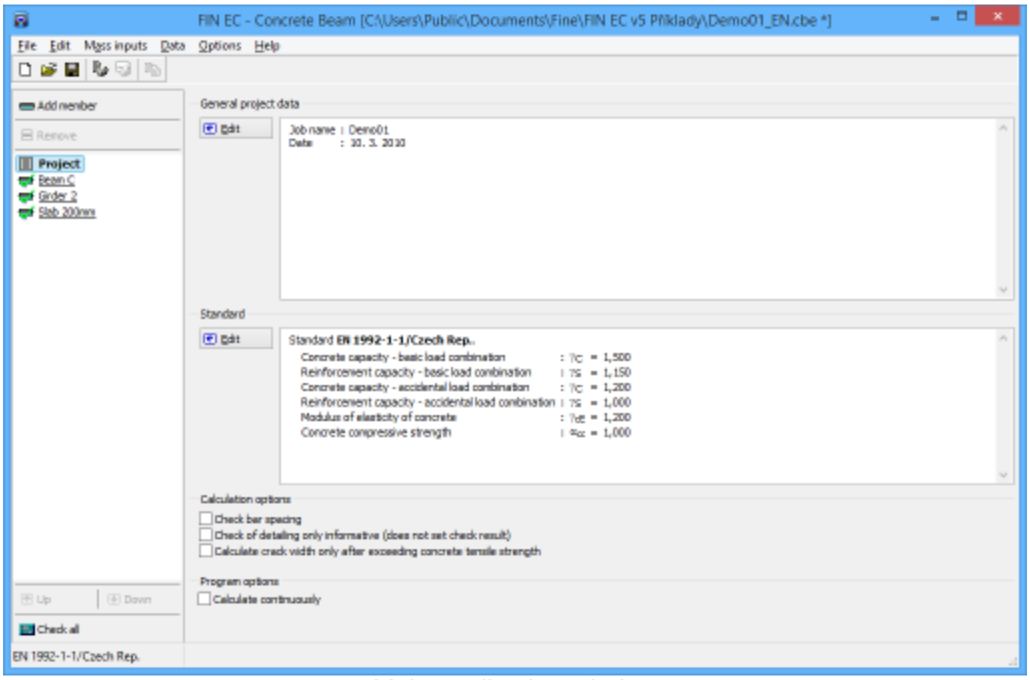
Part "**Calculation options**" contains settings that may influence the analysis:

**Check bar spacing**
- The check of bar spacing is performed in parts "**Longitudinal reinforcement**" and "**Shear reinforcement**". Verification is described in the part "**Structural rules**" of the theoretical help.

**Check of detailing only informative**
- Structural rules (reinforcement area, bar spacing) are checked, however, they don't affect the final result "**Pass/Fail**". The member is considered as passed, even though some structural rules aren't fulfilled.

**Calculate crack width only after exceeding concrete tensile strength**
- The crack width is calculated after the exceeding of limiting value $f_{ctm}$ (tensile strength of concrete). Cracks that may appear due to technological reasons aren't considered.

The setting "**Calculate continuously**" recalculates the results after any change immediately. This may be quite time consuming and limiting for members with a lot of load combinations. When switched off, any part is recalculated after clicking on it in the tree-menu.



*Main application window*

# Member

Task type "**Member**" is suitable for the detailed verification of horizontal RC member (beam, girder, slab) that is loaded by unlimited number of loads.

The main frame of member design contains these inputs:

**Member length**
- Total member length specified in metres.

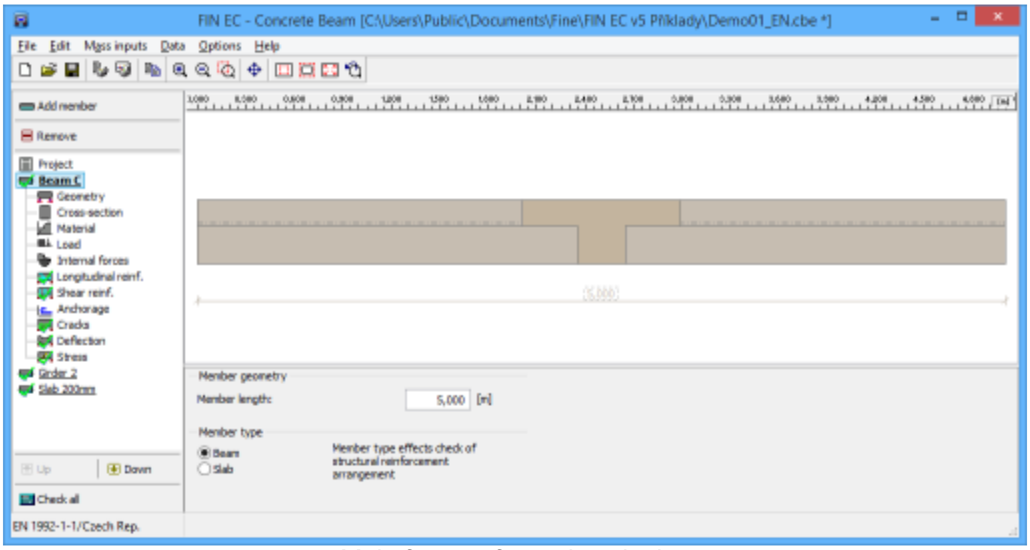
**Member type**
- Member type influences both structural rules and analysis. Differences are described in the part "**Member types**" of theoretical help. Member type should be selected according to the real function of member in the structure.

Beam properties are organized into following parts:

- **Geometry**
- **Cross-section**
- **Material**
- **Load**
- **Internal forces**
- **Longitudinal reinforcement**
- **Shear reinforcement**
- **Anchorage**
- **Cracks**
- **Deflection**
- **Stress**

General work with members (addition, manipulation) is described in the chapter "**Tree menu**".



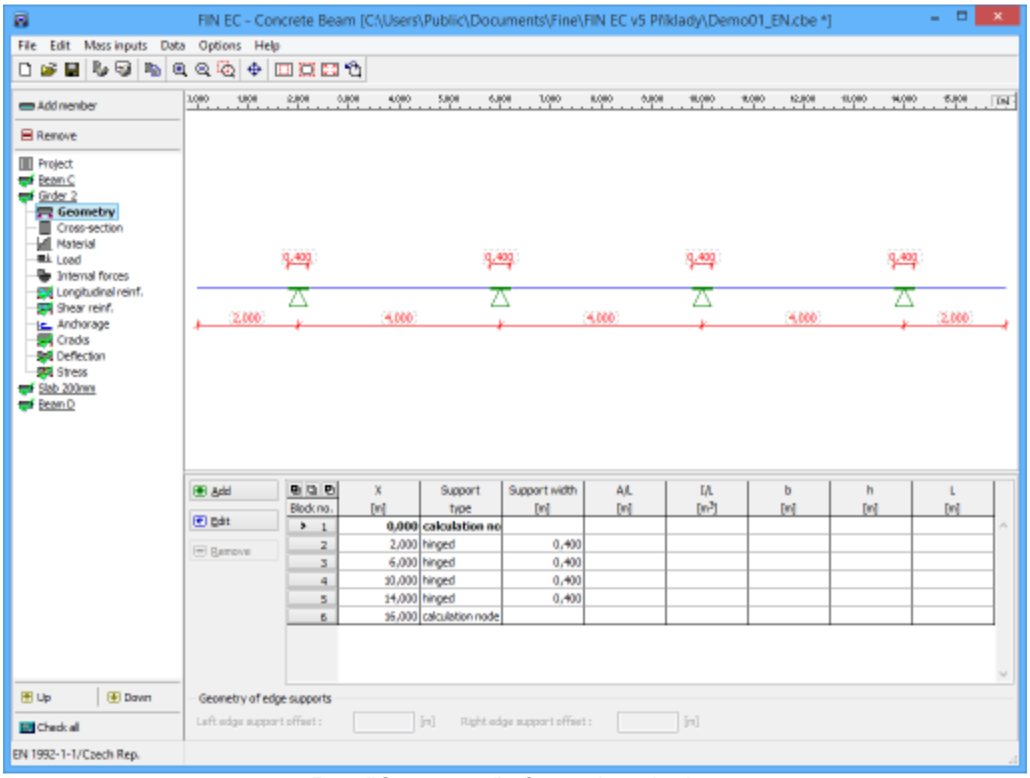

*Main frame of member design*

## Geometry

This part contains tools for input of nodes into the structure. Nodes like different types of supports, middle hinges and calculation nodes (points with detailed results) can be added with the help of the table in the input frame. Node properties are organized in the window "**Edit** " that can be launched using buttons "**Add**" and "**Edit**".

Offset can be defined for edge supports in the part "**Geometry of edge supports**". This value affects the distance between analysis support and support edge. This distance is significant for the reduction of bending moments. The offset doesn't affect analysis span.

The workspace contains active dimensions that are able to change spans and support widths without launching appropriate window.




*Part "Geometry" of member design*

## Edit member node

Properties of node (support) can be defined in this window. Basic parameter is the position of the node (marked as "**X-coordinate**" in the window) that is measured from the beginning of the member. These node types are supported:

**Calculation node**
- Nodes that show results in parts "**Longitudinal reinforcement**", "**Shear reinforcement**", "**Cracks**", "**Deflection**" and "**Stress**". The results in needed points can be gained with the help of calculation nodes. Calculation nodes influence neither topology neither internal forces.

**Hinged**
- Support fixed in vertical direction, free in rotation

**Fixed**
- Support fixed both in vertical direction and in rotation

**User support**
- Support with possibility to define the stiffness both in vertical direction and in rotation

**Middle hinge**
- Middle hinge in a beam bay. Only shear forces are transported in this point, bending moment is equal to *0*.

Next input is the "**Support width**" that is used for example for the reduction of bending moments in the part "**Longitudinal reinforcement**".

## General support

This part contains parameters for the calculation of support stiffness for node type "**User defined support**". These values should be entered when the setting "**Enter A/L, I/L**" is switched on:

**A/L**
- Ratio of total cross-sectional area of vertical structural members supporting the beam (*A*) and their length (*L*). Typical examples of such structural members are columns. This ratio is used for the calculation of support stiffness in the vertical direction. The assumption is that the supporting members are made of concrete. For any other material, recalculation of area to equivalent area for concrete material is necessary. The ratio of moduli of elasticity should be used in these cases.

**I/L**
- Ratio of total moment of inertia of vertical structural members supporting the beam (*I*) and their length (*L*). Typical examples of such structural members are columns. This ratio is used for the calculation of rotational support stiffness. The assumption is that the supporting members are made of concrete. For any other material, recalculation of area to equivalent area for concrete material is necessary. The ratio of moduli of elasticity should be used in these cases.

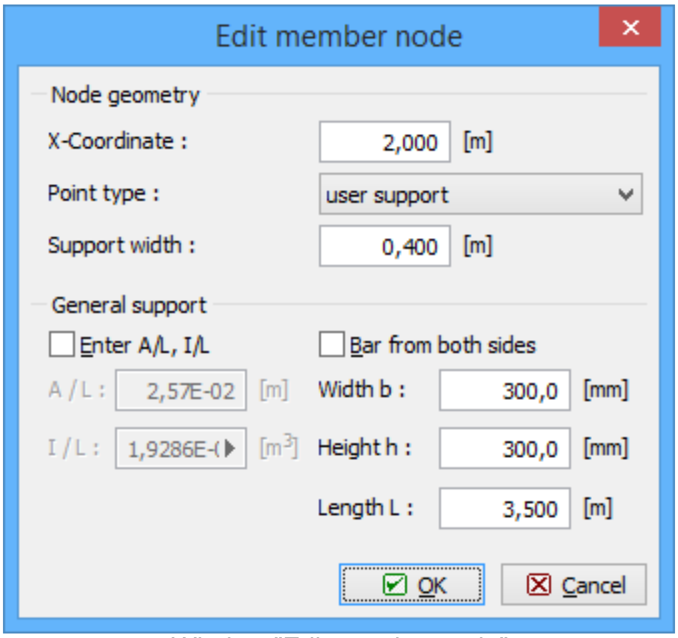
In other cases, these values are automatically calculated using following inputs:

**Width *b***
- Width of cross-section of supporting members (columns). The cross-sectional dimension perpendicular to the beam direction should be used for this input.

**Height *h***
- Height (Depth) of cross-section of supporting members (columns). The cross-sectional dimension in the beam direction should be used for this input.

**Length *L***
- Length (height) of structural members (columns) that support the beam

These values are used for calculation of *A/L* and *I/L* ratios. The area and moment of inertia are multiplied by two for setting "**Member from both sides**" as the connections from both (bottom and upper) sides are considered.



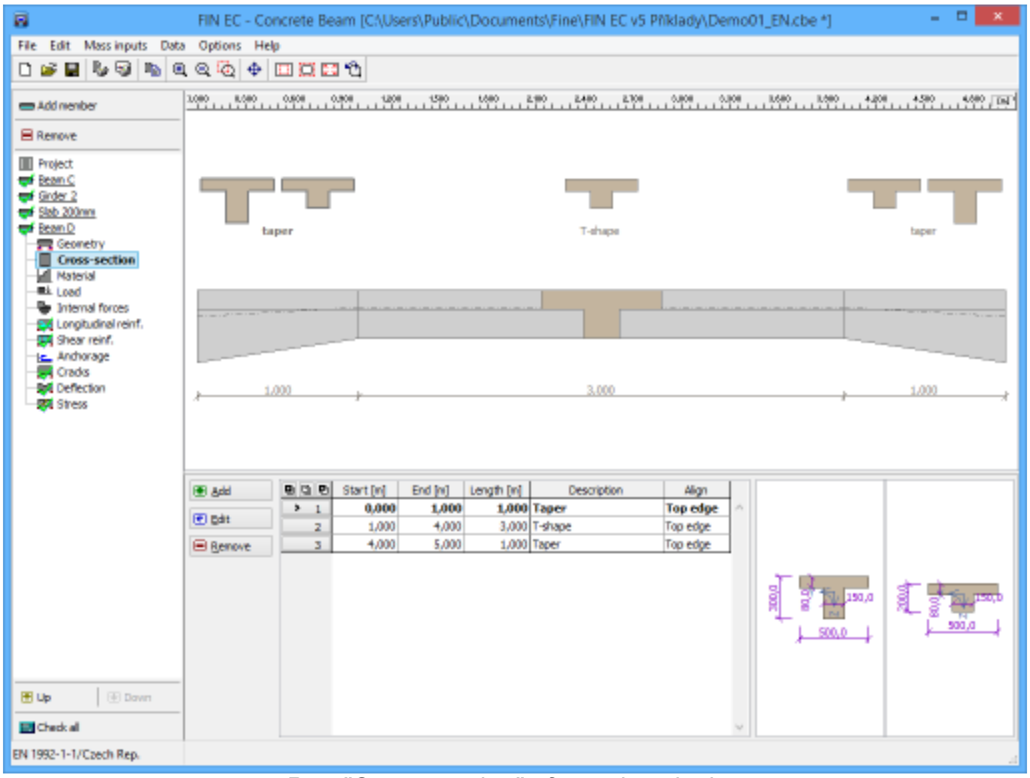
*Window "Edit member node"*

# Cross-section

The cross-section geometry can be specified in this part of the tree menu. The cross-section can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different cross-section parameters. Sectors with variable cross-sections (taper) may be also entered. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of the cross-section (type, geometry, dimensions etc.) are organized in the window "**Edit cross-section sector**". More sectors can be added (button "**Add**") for input of different cross-section geometry along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

If the member is loaded from "**Fin 2D**" or "**Fin 3D**", the cross-section geometry will be automatically copied from this program.



*Part "Cross-section" of member design*

# Edit cross-section sector

The geometry of cross-section in particular sector of a member length can be specified in this window. Basic parameter is "**Sector beginning**", that specifies also the end of previous member sector. This value is measured from the member beginning. The sector end and its length are also displayed.

## Sector type

The sector type can be specified in this part. These options are available:

- **Cross-section** - sector with constant cross-section along its whole length
- **Taper** - variable cross-section along the sector length. Geometry is defined by the cross-section at the beginning and end of the sector.

## Alignment

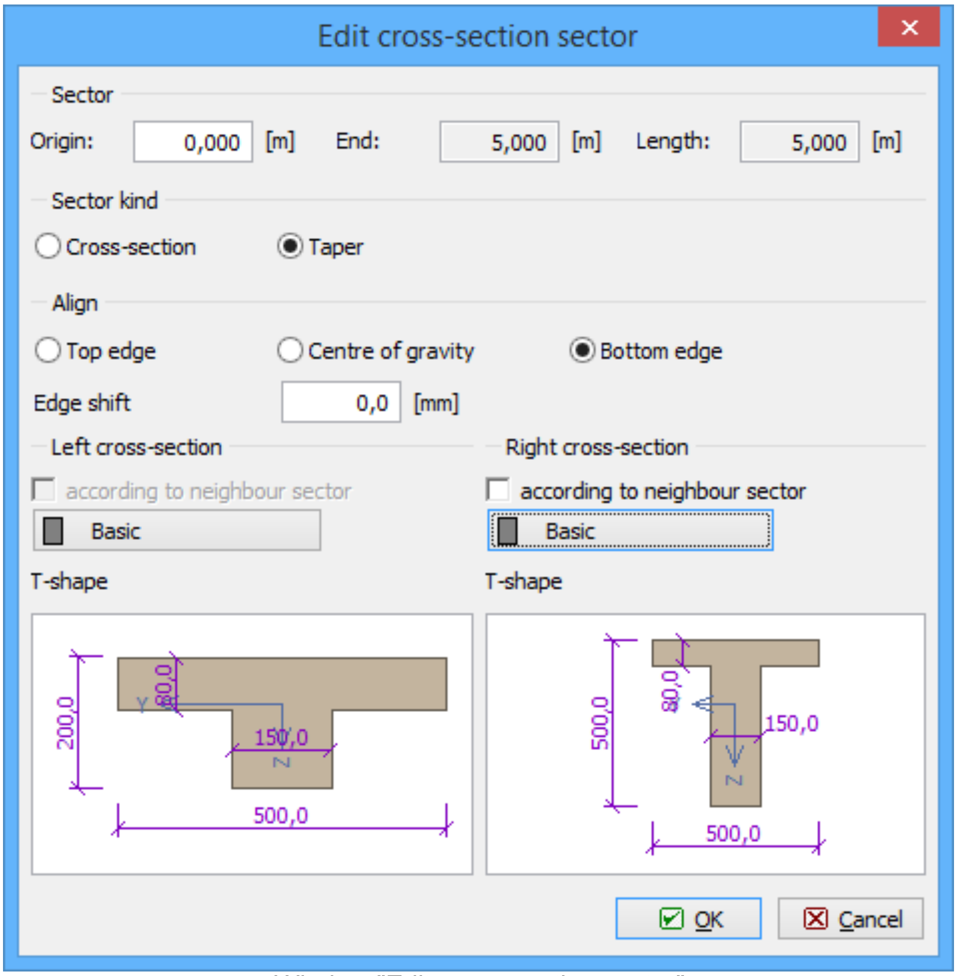
Sector alignment relative to the theoretical member axis can be specified here. The alignment changes only graphical view of the member, the axis bends aren't taken into consideration during the analysis.

## Cross-section

Shape and dimensions of cross-section can be specified here. These properties are organized in the window "**Cross-section editor**" that can be launched by the button "**Basic**". The cross-section has to be specified at the beginning and end for sector type "**Taper**". Both cross-sections have to have identical geometry type. Taper that changes the geometry from T-shape to rectangle should be specified as a T-shape with the identical width of web and flange at the end.

The geometry of the cross-sections at the beginning and end of taper can be copied from preceding or following sector using the setting "**according to neighbour sector**".




*Window "Edit cross-section sector"*

## Longitudinal reinforcement

This part is dedicated for the input of longitudinal reinforcement. The individual bars can be inserted with the help of the table in the bottom part of the screen. The reinforcement properties are organized in the window "**Edit bar**", that appears after clicking on the button "**Add**" (or "**Edit**" for existing bar) in the toolbar on the left side of the table. This window contains complete properties of the bar (beginning, end, shape, diameter, cover etc.).

Detailed results of the analysis may be displayed in a new window after using the button "**In detail**". The workspace is able to show diagrams of bending moments, Cross sectional area of reinforcement and utilization. Displayed quantities can be switched on/off in the window "**Drawing settings**", that can be launched using the button "**Diagrams**". The results for envelope of all relevant load combinations are displayed as a default. Results for certain load combination can be displayed with the help of drop-down menu in the heading of input frame.

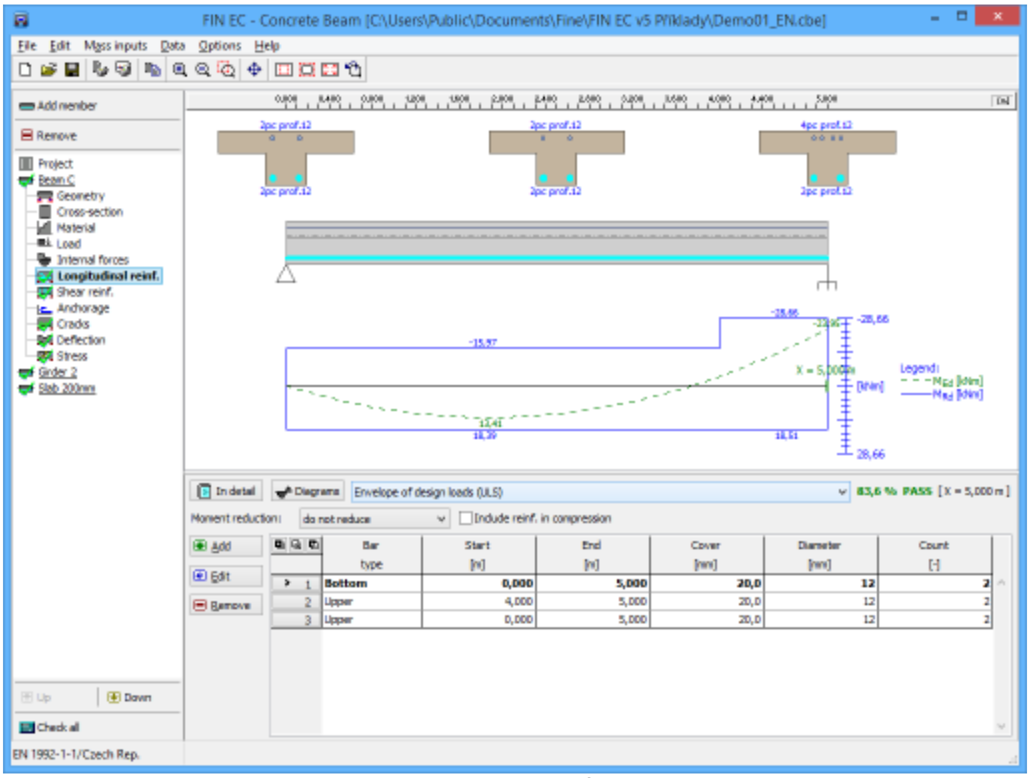
The longitudinal reinforcement in compression may be also included in the analysis. The consideration of this reinforcement in the analysis depends on the setting "**Include reinforcement in compression**".

The reduction of bending moments at the supports can be switched on with the help of drop-down menu "**Moment reduction**". These options are available:

**Do not reduce**
- Bending moments over the supports aren't reduced. Longitudinal reinforcement is verified using internal forces, that were calculated in part "**Internal forces**".

**Reduce to face**
- The values of bending moments at the edges of supports are used also for the verification along the whole support length. Higher values of bending moments in these positions are ignored.

**Reduce as continuous beam**
- Bending moment and shear force are reduced at the support using assumption, that the stress in the support is constant along the whole support length. This procedure is based on chapter *3.3.3.* of CSN 73 1201. This theory is valid for continuous beams with simple supports (e.g. slab supported by masonry walls).



*Part "Longitudinal reinforcement"*

# Edit bar

This window contains parameters of reinforcement bar. The bar shape (straight bar, bent, double bent) can be selected in the part "**Bar type**", it's length, position and diameter in the part "**Bar position**".

## Cover

Required cover of longitudinal reinforcement can be calculated in this part. these options are available:

**Minimum cover**
- The minimum cover calculated in the window "**Reinforcement cover**" will be used. The calculation in this window can be changed after clicking on the button "**Minimum cover**".

**Minimum cover and stirrups**
- The sum of stirrups' diameter (specified in part "**Shear reinforcement**") and minimum cover calculated in the window "**Reinforcement cover**" will be used. The calculation in this window can be changed after clicking on the button "**Minimum cover**".

**User defined cover**
- The user defined value of the reinforcement cover can be specified for this option.

Button "**Check of cover**" runs the control of minimum cover for the bar.

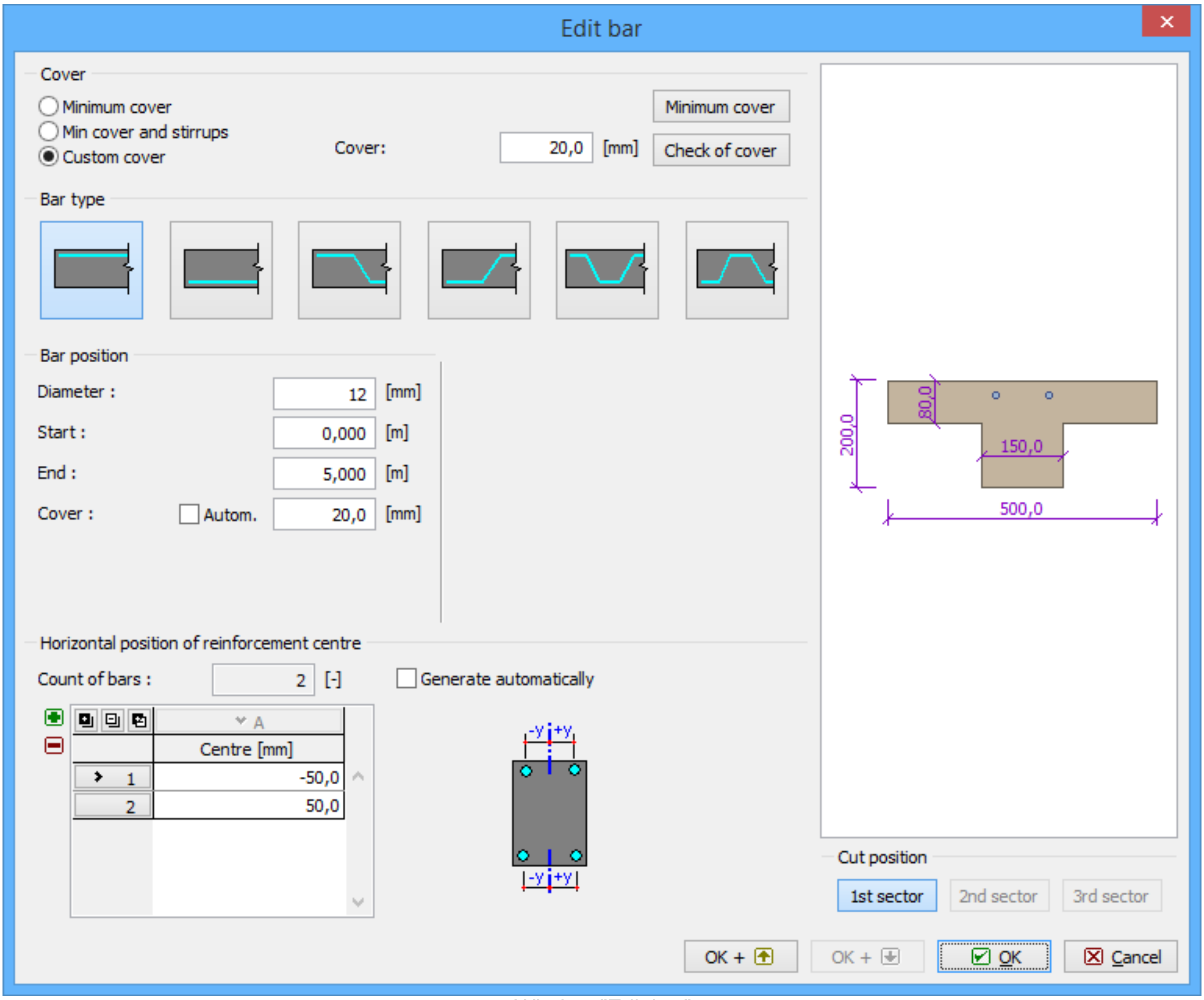
## Horizontal position of reinforcement centre

The horizontal position of bars can be specified in this part. The bars are organized automatically as a default. The longest bars are inserted into corners according to the specified cover, intermediate bars are placed uniformly between these two bars. Bars positions can be specified manually after switching off the setting "**Generate automatically**".

## Cut position

The cross-section view shows bars positions in the first sector for bent bars. Bars position in following sectors can be shown using buttons "**2nd sector**" and "**3rd sector**".




*Window "Edit bar"*

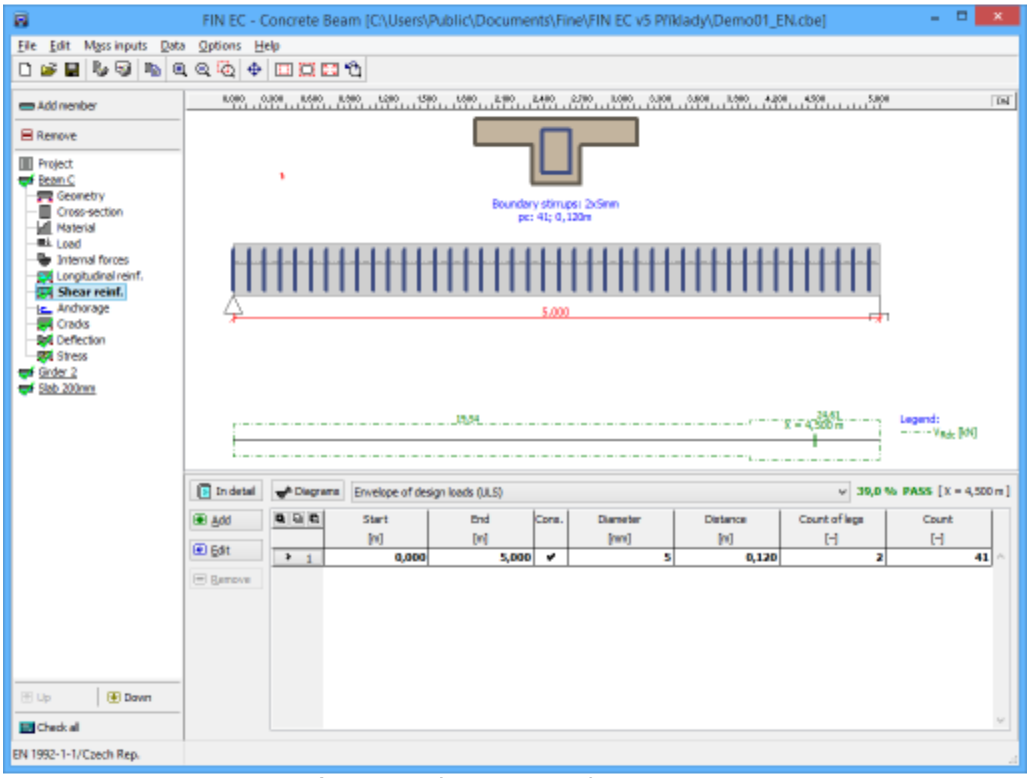
## Shear reinforcement

The shear reinforcement can be specified in this part of the tree menu. The reinforcement parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different reinforcement parameters. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The properties of the shear reinforcement (type, diameter, number etc.) are organized in the window "**Edit reinforcement sector**". More sectors can be added (button "**Add**") for input of different shear reinforcement along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

Complete analysis of shear reinforcement is also performed in this part. Detailed results can be displayed in a new window after using the button "**In detail**". The workspace is able to show diagrams of shear forces, shear resistance and utilization. Displayed quantities can be switched on/off in the window "**Drawing settings**", that can be launched using the button "**Diagrams**". The results for envelope of all relevant load combinations are displayed as a default. Results for certain load combination can be displayed with the help of drop-down menu in the heading of input frame.



*Part "Shear reinforcement" of member design*

## Anchorage

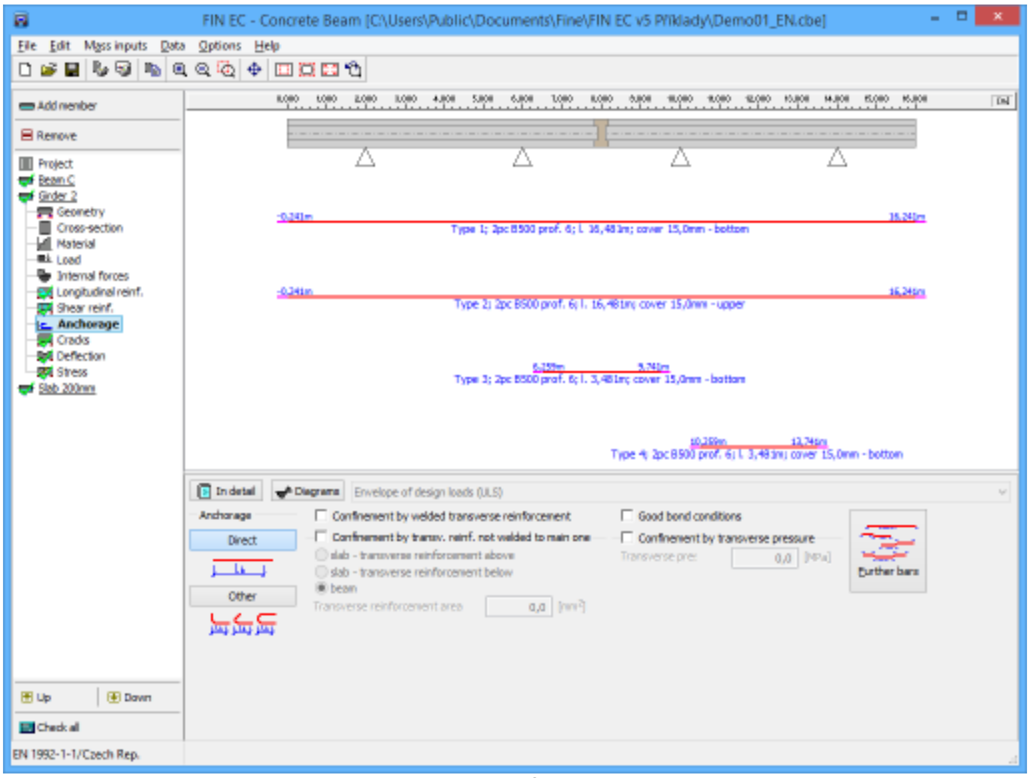
This part shows anchorage lengths for all reinforcement bars in member. Anchorage parameters can be specified in the input frame in the bottom part of the window:

| | |
|---|---|
| **Anchorage** | • The anchorage method (straight bar or other anchorage like bend, hook or loop) |
| **Confinement by welded transverse reinforcement** | • The reduction of anchorage length in accordance with table *8.2* (coefficient $\alpha_4$). |
| **Confinement by transv. reinf. not welded to main one** | • The reduction of anchorage length in accordance with table *8.2* (coefficient $\alpha_3$). Additional inputs are member type (the coefficient $K$ depends on this input) and total area of transverse reinforcement |
| **Good bond conditions** | • The anchorage conditions in accordance with figure *8.2*. Good bond conditions can be considered for all members with height up to *250mm*, in the bottom layer of *250mm* thickness for members up to *600mm* and along the whole height except upper layer of *300mm* thickness for higher cross-sections. |
| **Confinement by transverse pressure** | • The reduction of anchorage length in accordance with table *8.2* (coefficient $\alpha_5$) |

The table with complete overview of anchorage lengths can be displayed in a new window after using the button "**In detail**". Anchorage lengths are also shown in the workspace. Parameters of drawing can be changed in the window "**Drawing settings**", that can be launched using the button "**Diagrams**".

Calculation and verification of stress are described in the chapter "**Anchorage**" of the theoretical help.




*Part "Anchorage" of member design*

## Cracks

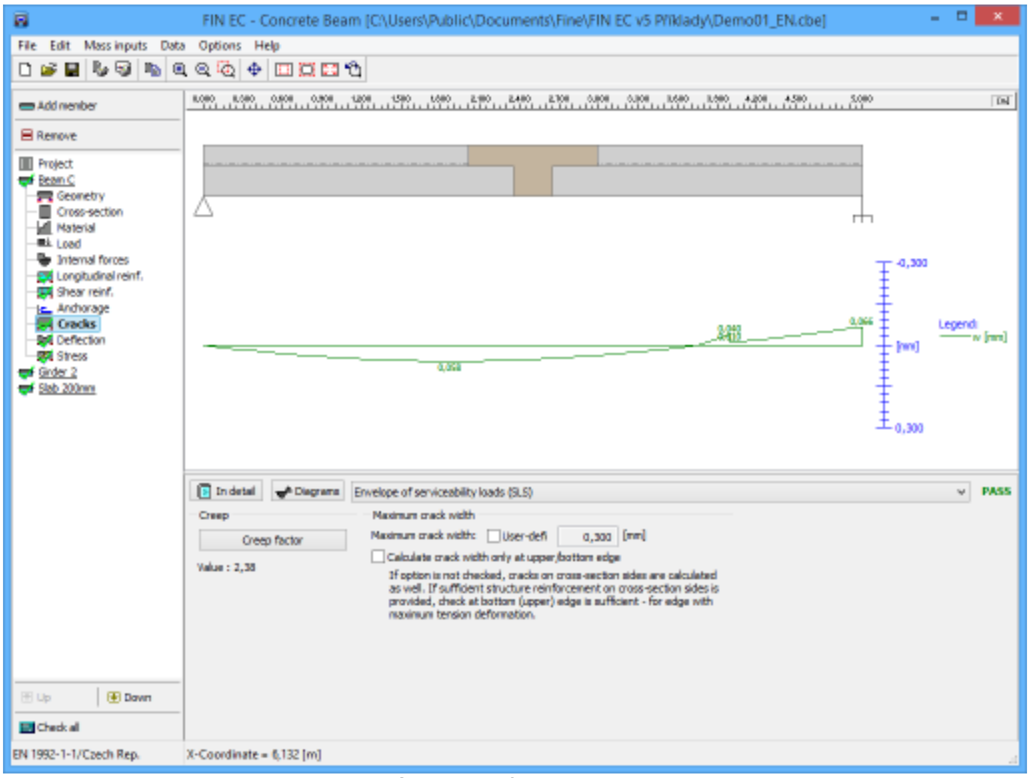
This part of the beam design shows results of crack calculation (serviceability limit state). The width of cracks along the whole member length is calculated and compared with the maximum value according to the standard or according to the user's input. The verification is performed only for load combinations "**Quasi-permanent (SLS)**".

The maximum crack width is considered in accordance with table *7.1N*. Option for user defined value is also included. Setting "**Calculate crack width only at upper/bottom edge**" switches off the crack control on the cross-section sides. This setting is suitable for the verification of part of the structure (for example one linear meter of slab). Complete results can be displayed in a new window after using the button "**In detail**". The workspace is able to show diagrams of bending moments and crack width. Displayed quantities can be switched on/off in the window "**Drawing settings**", that can be launched using the button "**Diagrams**". The results for envelope of all relevant load combinations are displayed as a default. Results for certain load combination can be displayed with the help of drop-down menu in the heading of input frame.

Cracks may also arise from other causes such as plastic shrinkage or expansive chemical reactions within the hardened concrete (creeping). These factors are taken into account with the help of creep coefficient. The value of this coefficient can be changed in the window "**Creep**" that can be launched by the button "**Creep coefficient**".

Calculation and verification of cracks are described in the chapter "**Serviceability limit state**" of the theoretical help.




*Part "Cracks" of member design*

# Deflection

This part of the beam design shows results of deflection verification (serviceability limit state) along the member length. The verification is performed only for load combinations that are dedicated to the serviceability limit state design. If there isn't any load combination for serviceability limit state, the results aren't available. The frame in the bottom part contains these settings for deflection verification:

## Quasi-permanent combinations

Limiting value for deflection caused by the combinations with specified type "**Quasi-permanent (SLS)**" can be specified in this part. Deflection control for quasi-permanent combinations is defined in the chapter *7.4.1* of EN 1992-1-1. These options are available:

- **l/250 - Common requirements** — The limit defined in the chapter *7.4.1.(4)* of EN 1992-1-1. This limit shall be considered for deflection that could impair the appearance and general utility of the structure.
- **l/500 - Strict requirements** — The limit defined in the chapter *7.4.1.(5)* of EN 1992-1-1. This limit shall be considered for deflection that could damage adjacent pars of the structure like partition walls.
- **l/ - User defined requirements** — User defined limit specified as span-depending value
- **Deflection** — User defined limit specified as an absolute value in *mm*

## Characteristic (Frequent) combinations - user defined requirements

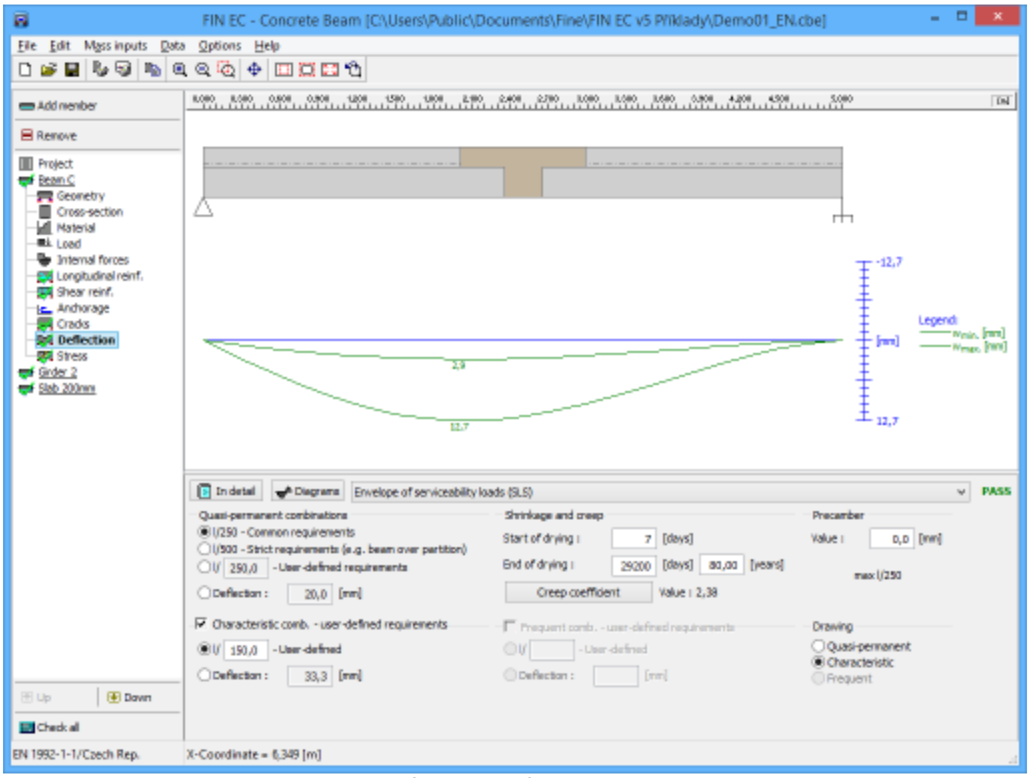
Requirements for deflection caused by characteristic and frequent combinations aren't specified in the design standard. User defined limits for such deformations can be defined in these parts. The limits can be entered as span depending or absolute values.

## Shrinkage and creep

This part contains basic inputs for the calculation of creep coefficient. The start and end of drying shrinkage can be specified here. More inputs and detailed results can be shown in the window "**Creep**" that can be launched by the button "**Creep coefficient**".

Complete results can be displayed in a new window after using the button "**In detail**". The workspace is able to show diagrams of bending moments and deformations (maximum and minimum values, particular components). Displayed quantities can be switched on/off in the window "**Drawing settings**", that can be launched using the button "**Diagrams**". The results for envelope of all relevant load combinations are displayed as a default. Results for certain load combination can be displayed with the help of drop-down menu in the heading of input frame. Some quantities (particular components of deflection) cannot be displayed for envelope of load combinations.

Calculation and verification of stress are described in the chapter "**Serviceability limit state**" of the theoretical help.

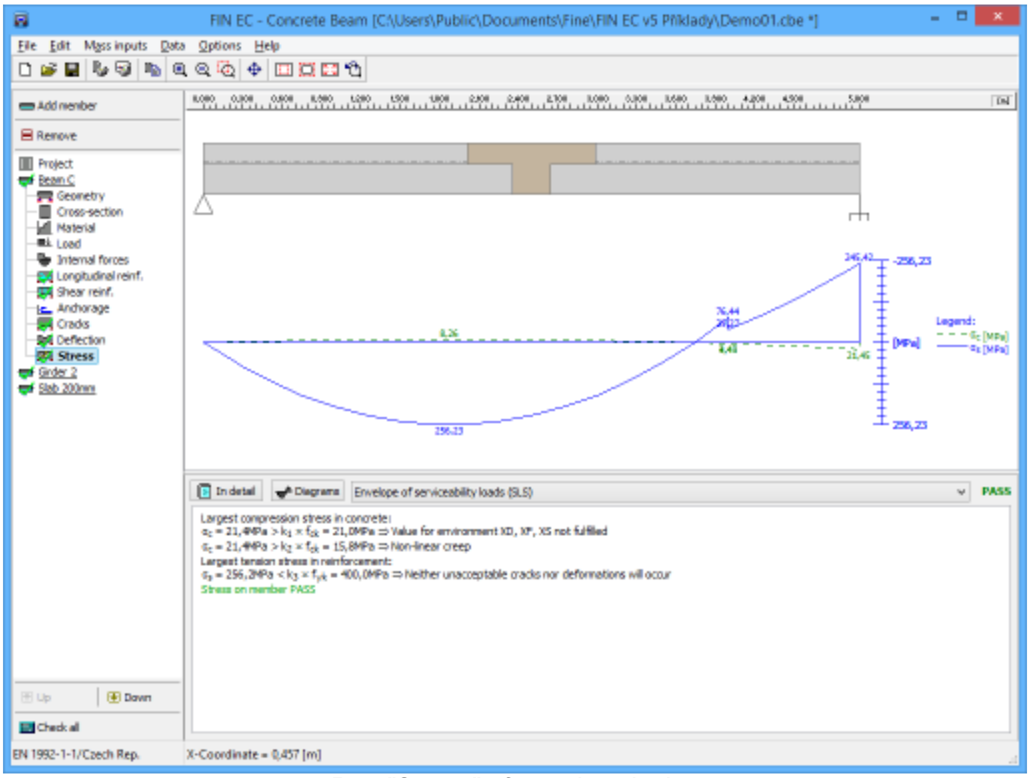


*Part "Deflection" of member design*

## Stress

This part of the beam design shows results of stress verification (serviceability limit state) along the whole member length. Stress both in concrete and reinforcement steel is calculated and checked. The verification is performed only for load combinations "**Characteristic (SLS)**". Results are shown in the frame in the bottom part. Complete results can be displayed in a new window after using the button "**In detail**". The workspace is able to show diagrams of bending moments and stresses in steel and concrete. Displayed quantities can be switched on/off in the window "**Drawing settings**", that can be launched using the button "**Diagrams**". The results for envelope of all relevant load combinations are displayed as a default. Results for certain load combination can be displayed with the help of drop-down menu in the heading of input frame.

Calculation and verification of stress are described in the chapter "**Serviceability limit state**" of the theoretical help.



*Part "Stress" of member design*

# Program Corbel

The software "**Corbel**" verifies directly supported (corbels on columns) and indirectly supported (corbels on beams) corbels according to EN 1992-1-1.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Data**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Two types of particular tasks ("**Corbels**") can be verified in the software:

| | |
|---|---|
| **Directly supported corbel** | • The detail where the load is directly transferred into the structure. Typical example is the corbel placed on the column. The detail is analysed as CCC joint. |
| **Indirectly supported corbel** | • The detail that is hanging on the structure. Typical example is the corbel placed on the beam. The detail is analysed as CCT joint. The tensile stress in the joint is transferred by stirrups in beam. |

These tasks can be added with the help of the button "**Add**" in the heading of the tree menu.

## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

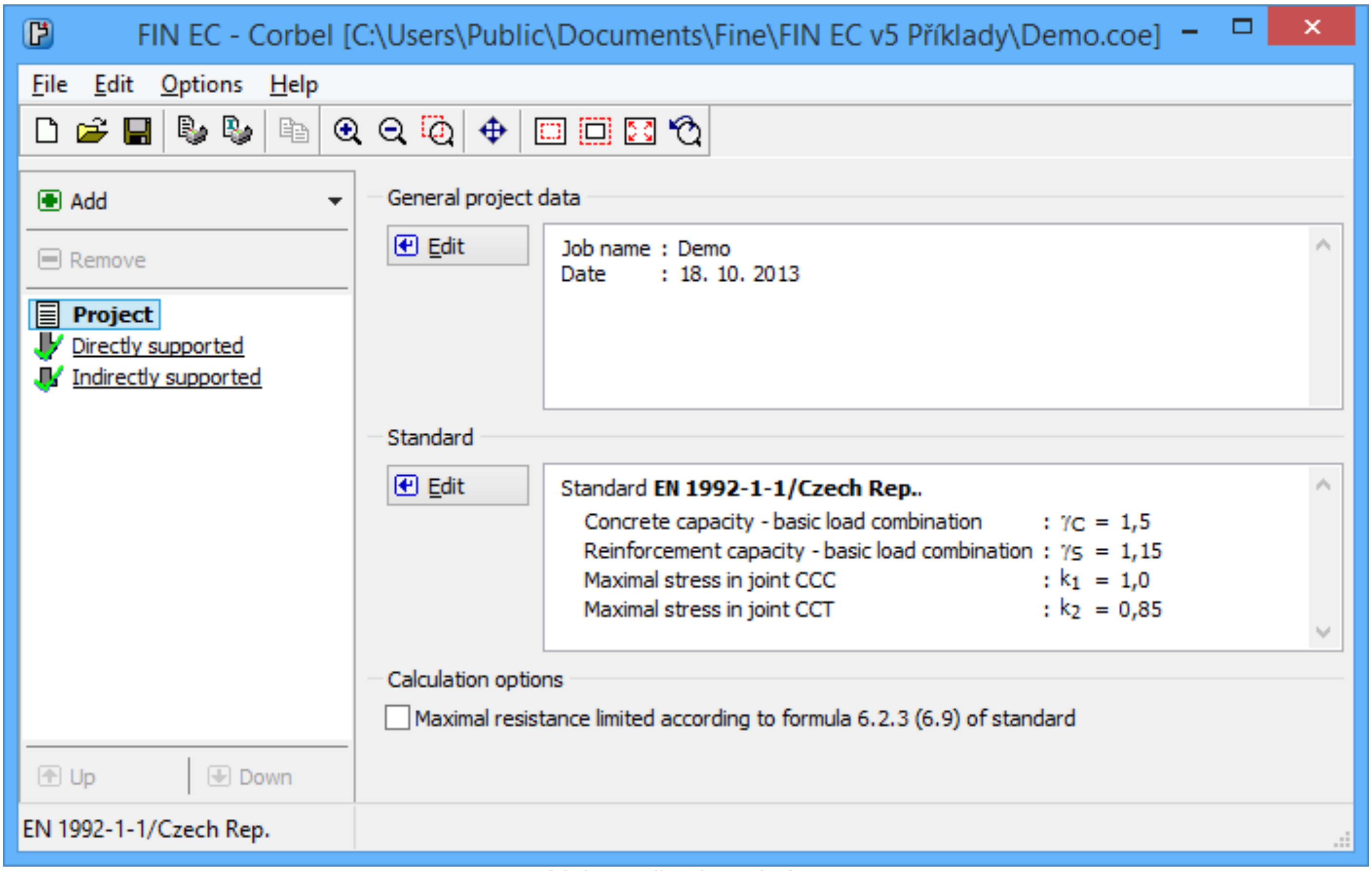
Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

Part "**Calculation options**" contains settings that may influence the analysis:

| | |
|---|---|
| **Maximal resistance limited according to formula *6.2.3(6.9)* of standard** | • The possibility to limit the design value of the compressive resistance due to crushing of the compression struts |

The program appearance may be changed in the window "**Global settings**" that is available in the part "**Options**" of the main menu.




*Main application window*

# Global settings

The appearance of the workspace and figures in documents can be changed in this window. The window contains these two tabs: "**View**" a "**Schemes**".

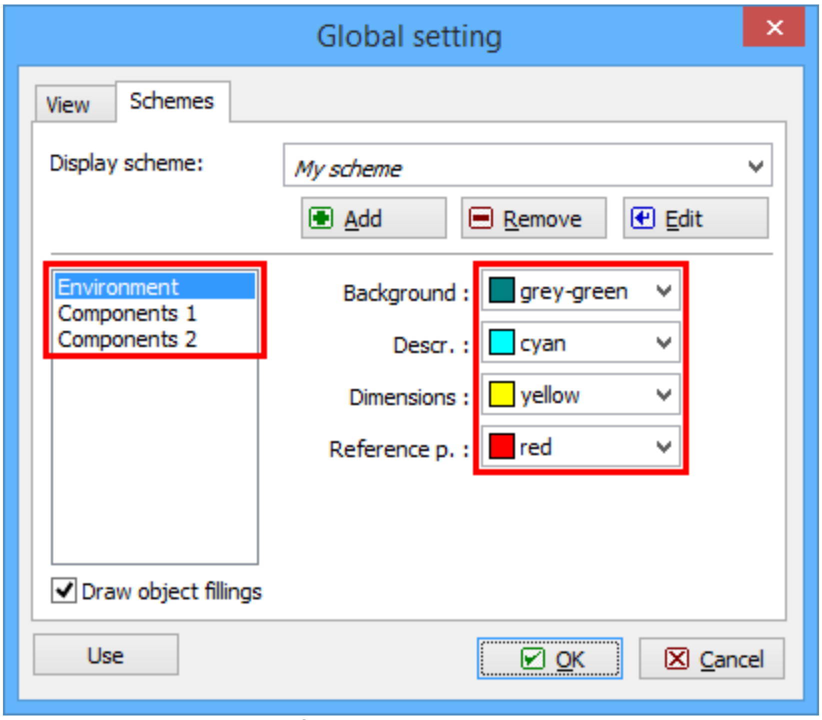
## View

This tab contains settings for changing the appearance of workspace and figures both in documents and clipboard (saving figure using *Ctrl+C* ). There is an option to change the font size in the workspace with the help of "**Text size**" setting. The selection of colour schemes changes the appearance of figures in the workspace, documents and clipboard. It means, that the connection in the workspace can be coloured, however, the figure copied using clipboard will be in black and white only. Few schemes are pre-defined, however it's possible to add new ones in the tab "**Schemes**".

Button "**Options**" launches the window with properties of figures copied into clipboard (size, borders etc.).

## Schemes

User defined colour schemes can be defined in this tab. These schemes can be used in the tab "**View**" for workspace, documents or figures in clipboard. Buttons "**Add**", "**Edit**" and "**Remove**" are available for the work with schemes. The colours for particular items can be specified in the bottom part of the window. Colours can be modified only for user defined schemes (highlighted by italics in the list of schemes). Pre-defined schemes can't be modified.




*Input of colours into new scheme*

# Corbel

This screen contains input fields in the left part, corbel view in the right upper corner and results in the right bottom corner. The appearance of the workspace may be change in the window "**Global settings**" that is available in the part "**Options**" of the main menu. Following inputs should be specified for the detail:

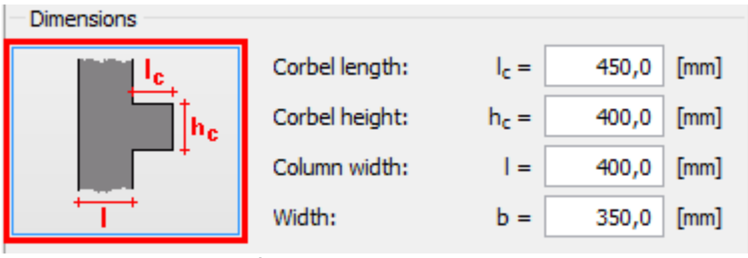
## Materials

The materials of member and reinforcement can be specified in this part. There is a dedicated window "**Materials**" for materials input. This window can be launched using "**Material**" button. Both pre-defined strength grades and user defined input of material characteristics are available in this window.

## Dimensions

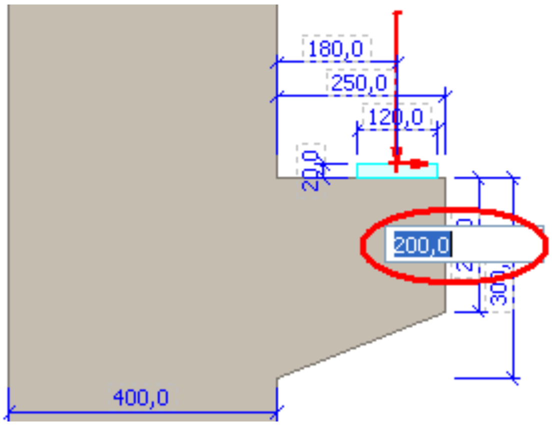
Part "**Dimensions**" is dedicated for the selection of the corbel shape and specifying its dimensions.




*Button for changing the corbel type*

The dimensions may be specified in the corresponding input fields or directly in the workspace with the help of active dimensions.




*Editing dimensions in the workspace*

## Loading

The corbel is loaded mainly by the vertical force with certain eccentricity. This part contains following inputs:

| | | |
|---|---|---|
| **Eccentricity $a_C$** | • | The distance between the force and the edge of column or beam |
| **Vertical force $F_{Ed}$** | • | The design value of the vertical force |
| **Horizontal force $H_{Ed}$** | • | The additional horizontal component. The value is usually calculated as a part of the vertical force, however an arbitrary value may be specified. This component increases the area of the main reinforcement. |
| $H_{Ed}/F_{Ed}$ | • | The ratio of the vertical and horizontal component of the load. Minimum recommended value is *0.2* |

## Slide plate

Following data may be specified for the sliding plate:

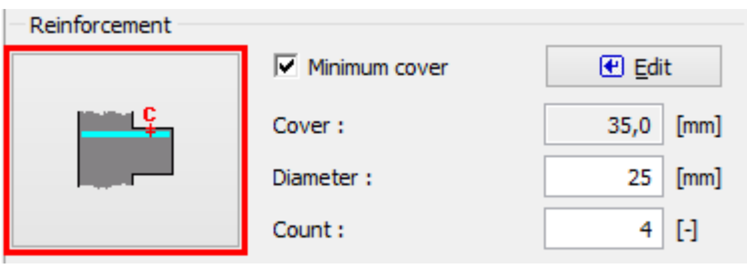
| | | |
|---|---|---|
| **Length $l_p$** | • | The length of the slide plate. The internal edge is the important data for the placement of the shear reinforcement. |
| **Height $\Delta h$** | • | The height of the slide plate. This value increases the eccentricity of the loaded point and thus also the tensile force in the main reinforcement. |
| **Width $b_p$** | • | The width of the slide plate. |

## Reinforcement

The main tensile reinforcement may be specified here. The reinforcement is defined by the number of bars and diameter.

If the setting "**Minimum cover**" is switched on, the software automatically calculates the minimum cover according to the stirrups diameter and the parameters specified in the window "**Reinforcement cover**" (the window may be opened using the button "**Edit**"). The user defined value may be specified if the setting is switched off.

The button with corbel scheme launches the window "**Reinforcement**" that contains extended options for reinforcement input (second layer, different diameters per layer).



*Button for detailed input of reinforcement*

## Stirrups

The reinforcement by horizontal and vertical stirrups should be also specified for the corbel. Horizontal stirrups are necessary for the elimination of transverse tension in the compression struts, vertical stirrups also create the tension ties in the strut and tie model for long corbels.

The stirrups are defined by the diameter, number of stirrups (number of rows along the corbel length) and number of bars (number of bars in one row). The steel grade may be specified in the part "**Materials**" (described above).

## Results

The right bottom part of the window shows the results of the analysis and also the error messages. The analysis is described in the theoretical part of the help.

# Reinforcement

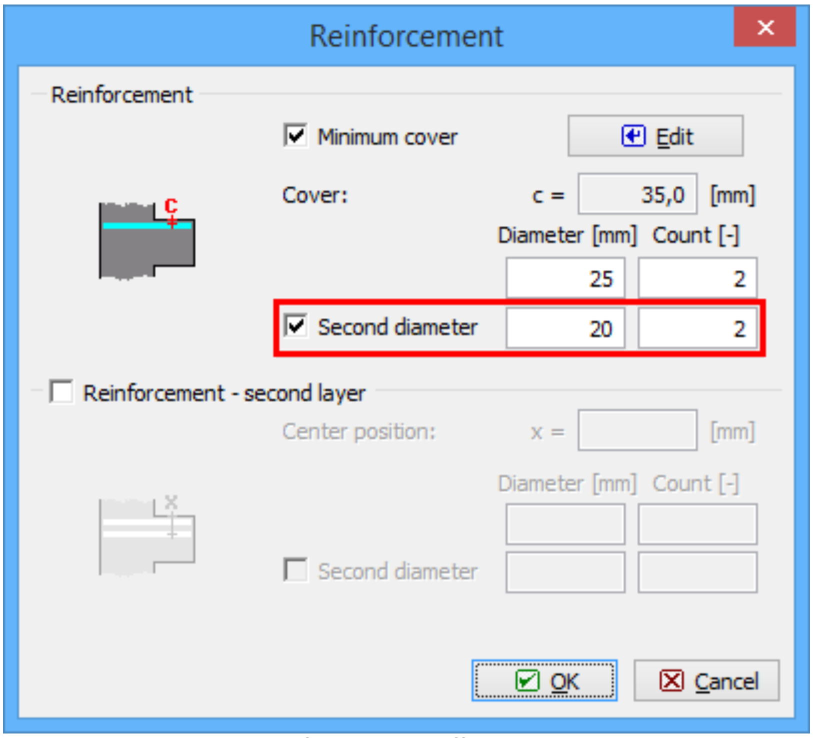
This window contains extended input options for the main reinforcement in the corbel. The second layer and different diameters in one layer may be specified here comparing to the main window.

If the setting "**Minimum cover**" is switched on, the software automatically calculates the minimum cover according to the stirrups diameter and the parameters specified in the window "**Reinforcement cover**" (the window may be opened using the button "**Edit**"). The user defined value may be specified if the setting is switched off.

The reinforcement is specified by the number of bars and the diameter.

The check box "**Reinforcement - second layer**" may add an additional reinforcement layer. The position is specified by the distance between the top edge of the corbel and the centre of the layer (unlike the primary layer that is specified by the reinforcement cover).

Additional bars with different diameter may be specified for both layers with the help of setting "**Second diameter**".




*Insertion of bars with different diameter*

# Program Punching

The software "**Punching**" verifies directly supported (corbels on columns) and indirectly supported (corbels on beams) corbels according to EN 1992-1-1.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for switching between parts of an input. The tree menu can be alternated by the part "**Entry**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

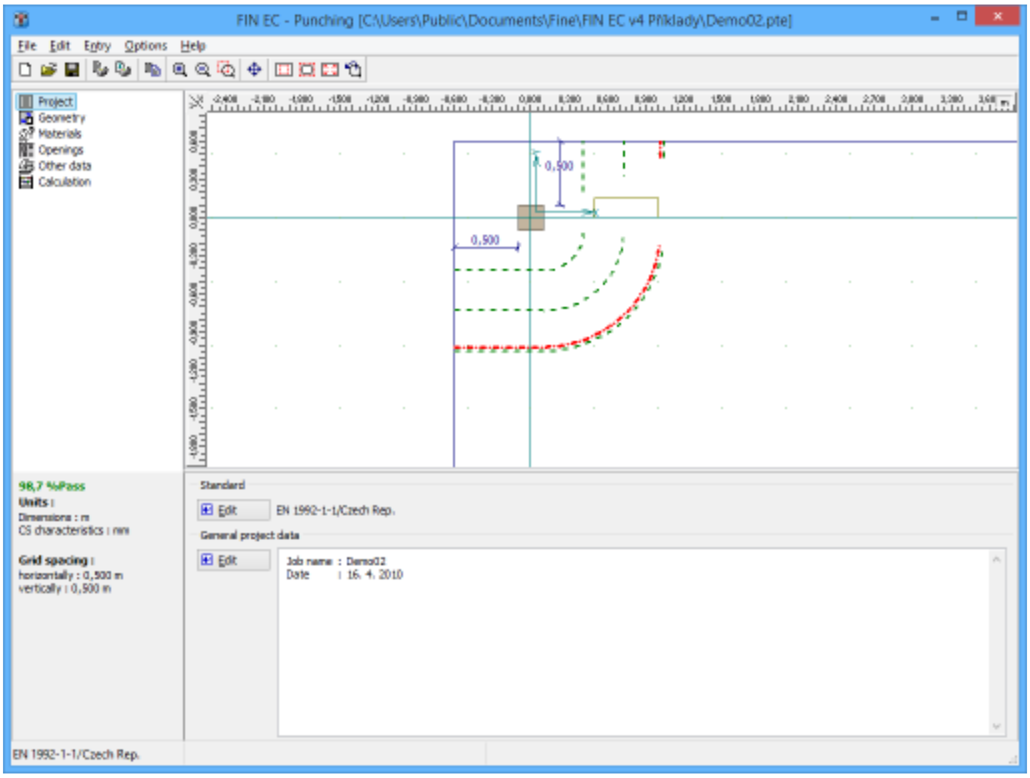
## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

The tree menu in the left part can be used for switching between particular sections of the input and verification (geometry, materials, analysis). The structure of the tree menu is duplicated in the main menu, part "**Entry**". The right upper part of the window contains the workspace that show the analysed detail. Workspace appearance may be changed in the window "**Options**" that is accessible from the main menu. The workspace shows not only the geometry of the detail, but also particular control perimeters $u_x$ (green or blue colour) and also the perimeter $u_{out}$, where the shear reinforcement is not required (red colour).




*Main application window*

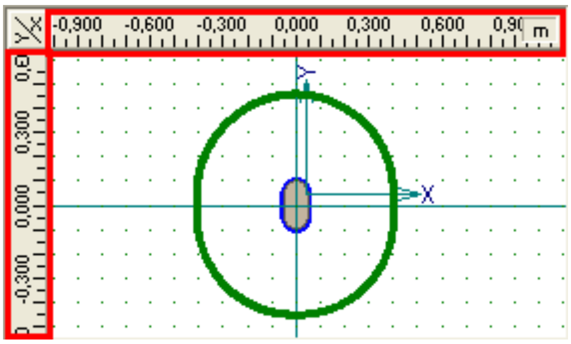
# Options

This window contains settings that affect the appearance of the workspace and final documentation. The window contains following tabs:

## General

This tab contains properties of the grid that may be displayed in the workspace and that may be used for the input of openings. The origin and step of the grid can be entered here. The mouse cursor will by aligned automatically according to the grid after using the setting "**Snap to grid**".

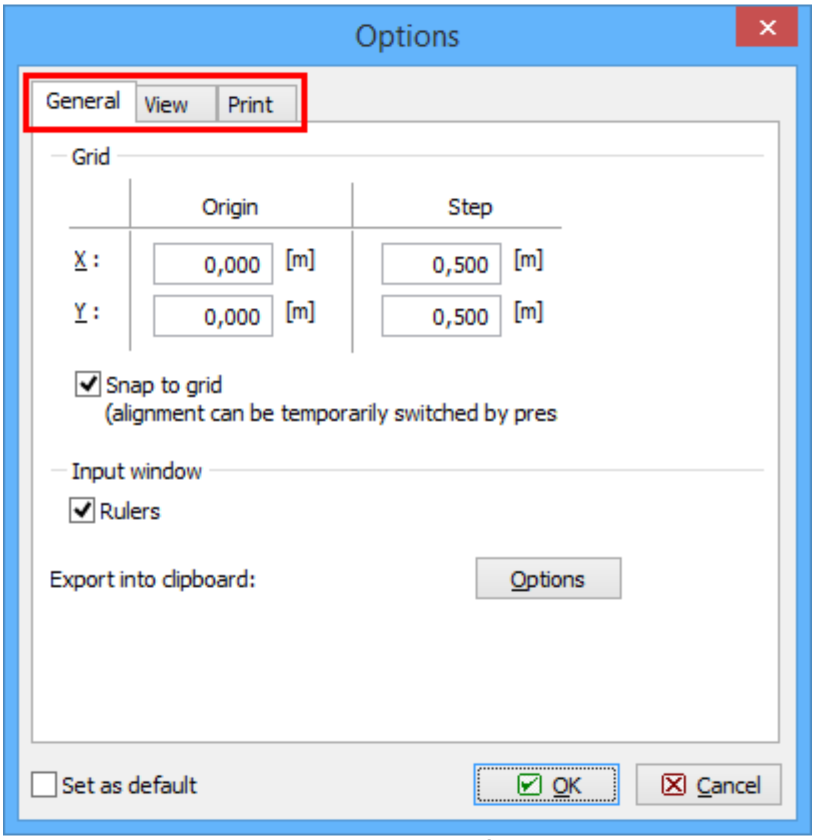
Rulers may be also switched on/off for the workspace.



*Workspace with grid and rulers*

## View and Print

These tabs contain settings that may change the appearance of the workspace and also set colours for printing. The tabs substitute function of the window "**Drawing settings**" that is included in other Fin EC programs.




*Tabs in the window "Options"*

# Geometry

This part contains basic geometric parameters of the project:

## Geometry

This frame contains an option to specify the structure type ("**foundation slab**" or "**floor slab**") and also other properties of the detail. The slab thickness $h_s$ and column position may be specified here. Following positions are available:

**Internal** • The column isn't affected by any slab edge.

**Edge** • The column is placed close to the one edge of the slab. The distance $a_1$ from the slab edge has to be specified in this case. The software determinates during the design, whether the analysis is or isn't affected by the edge.

**Internal** • The column is placed close to the corner of the slab. The distances $a_1$ and $a_2$ from the slab edges has to be specified in this case. The software determinates during the design, whether the analysis is or isn't affected by the edges.

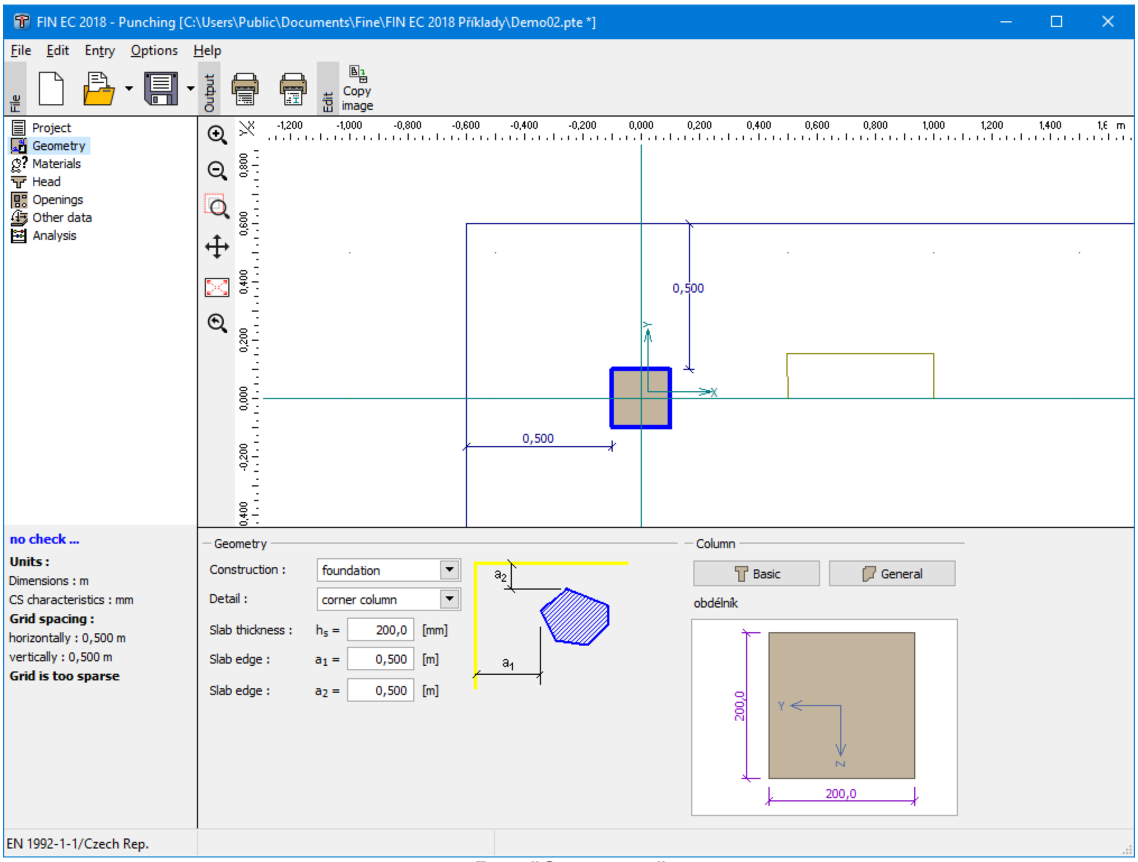
## Column

The geometry of the column cross-section may be specified here. Following options are available:

**Basic** • Input of basic shapes with the help of pre-defined database of cross-sections. Shapes like rectangle, circle, I-profile, T-profile are available for this option.

**General** • Input of general shape of the column in the dedicated window.



*Part "Geometry"*

# Materials

The material characteristics of concrete and steel may be specified in this part:

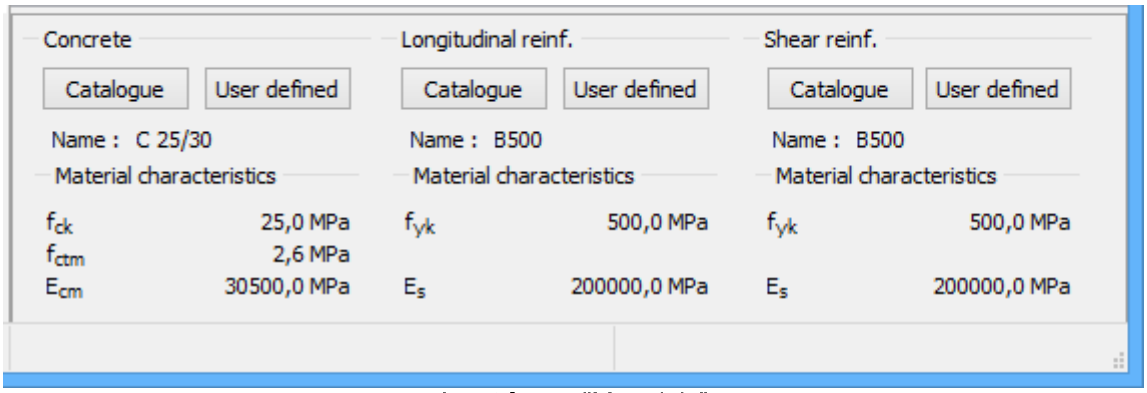
## Concrete

This part contains buttons for input of concrete properties. The properties can be specified using the strength classes from pre-defined database in the window "**Materials catalogue - concrete**" (button "**Catalogue**") or by entering the properties numerically in the window (button "**User defined**").

## Longitudinal and shear reinforcement

This part contains buttons for input of materials for longitudinal reinforcement in slab and shear reinforcement above the column. The properties can be specified using the strength classes from pre-defined database in the window "**Materials catalogue - steel**" (button "**Catalogue**") or by entering the properties numerically (button "**User defined**").

Selected materials including their characteristics are displayed in the bottom part of the frame.



*Input frame "Materials"*

# Head

The column head that increased the slab resistance in punching can be specified in this part.

## Head

The part "**Head**" contains the choice of head type and dimensions. Following types are available:

**No head** • The column does not have any head

**Column-shaped head** • The shape of the head respects the shape of column cross-section. The dimensions are given only by the distance from the column edge to head edge $x$.

**Rectangular head** • The head has rectangular shape. The dimensions are given by the orthogonal distances from the outer points of column cross-section.

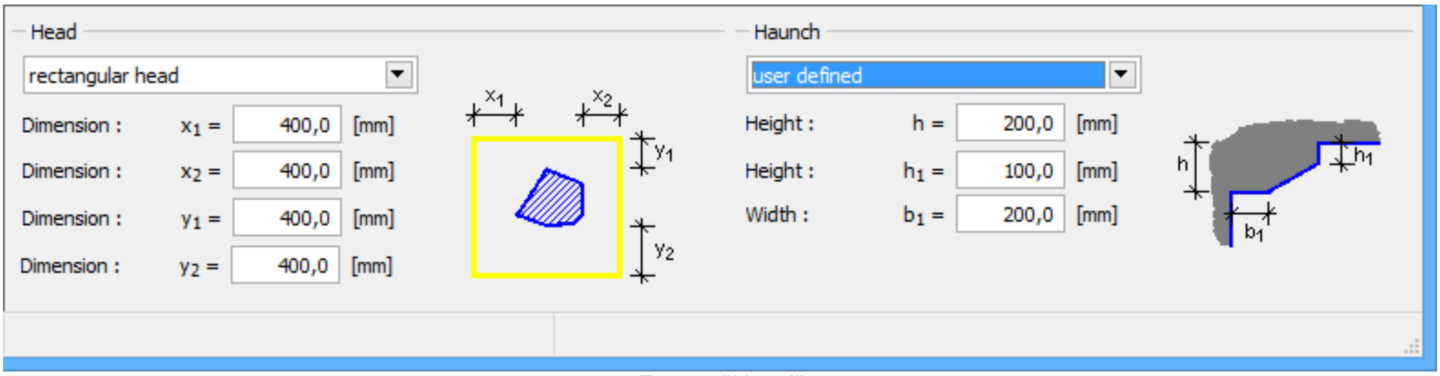
## Haunch

This part contains the geometry of head section. Following options may be chosen:

**Step** • The bottom surface of the head is horizontal, the head is ended with vertical edge.

**Oblique** • The bottom surface of the head is inclined.

**User defined** • Combination of previous options. This type contains horizontal, inclined and vertical parts. Both the length of horizontal part and the height of the vertical part may be defined as *0*.




*Frame "Head"*

# Openings

This part is dedicated for the input of openings in slab. The number of inserted openings isn't limited. The openings may be added numerically (using dedicated windows) or graphically in the workspace.

## Numerical input

Numerical input may be performed with the help of table in the input frame. The general shapes may be inserted in the window "**Polygon**", that may be launched by the button "**Add polygon**". Circular openings may be entered by the button "**Add circle**". The input is performed in the window "**Circle**" in this case.

## Graphical input

The graphical input is possible after selection of an appropriate input mode in the tree menu. After that, the workspace may be used for the input of openings. Following modes are available:

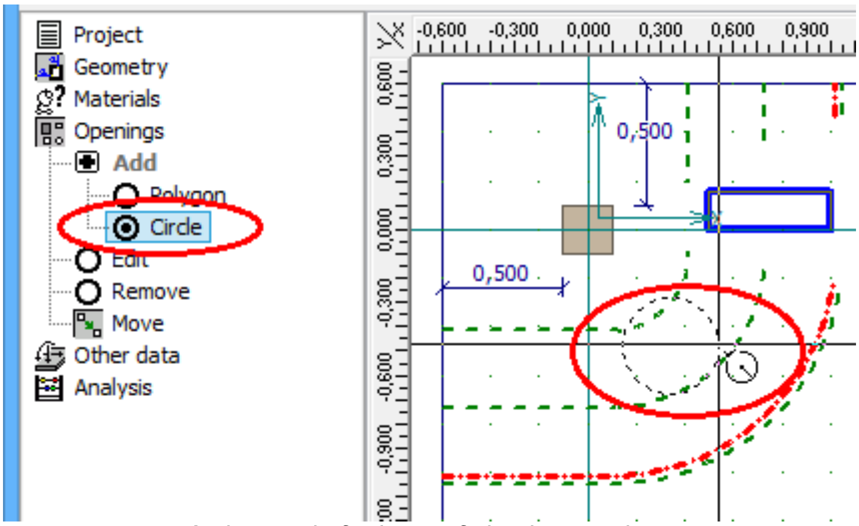
**Add - Polygon** • Insertion of general polygonal opening. Any click on the workspace will create a new node of the polygon. As the polygon shall be closed, the last node in polygon has to be identical to the first node in polygon.

**Add - Circle** • Insertion of circular opening. First click on the workspace defines the centre of the opening, second one a point on a circle (radius is the distance between these two points).

**Edit** • The existing openings may be modified in this mode. Clicking on existing opening launches appropriate window for opening edit. The window "**Polygon**" is launched for polygonal openings, the window "**Circle**" for circular ones.

**Remove** • Any existing opening may be deleted by clicking in this mode.

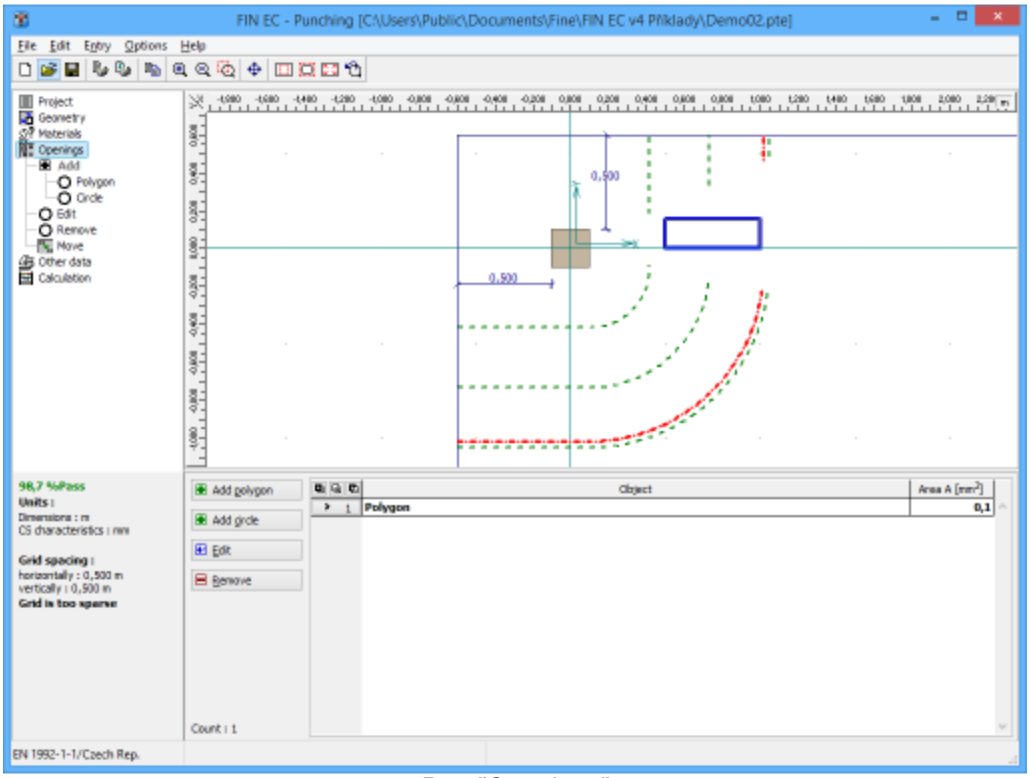



*Active mode for input of circular openings*

The cursor is automatically aligned according to the displayed grid. This alignment may be switched off or modified (e.g. change of grid step) in the window "**Options**" that may be launched with the help of part "**Options**" in the main menu. The alignment may be also switched off temporarily during the work with the help of the key "**Ctrl**" on the keyboard.

## Move and copy

The existing openings may be moved or copied with the help of the tool "**Move**" in the tree menu. The properties of the modification has to be specified in the window "**Move**". This tool may be used for all, active or selected openings. More openings may be selected with the help of the table in the input frame.

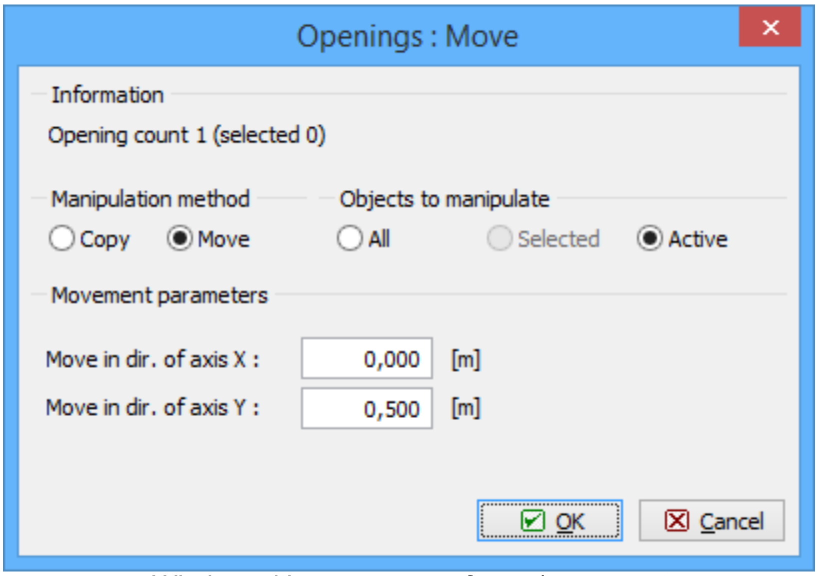



*Part "Openings"*

# Move

One or more openings may be moved or copied with the help of this window. Modes "**Copy**" or "**Move**" may be selected in the part "**Manipulation method**". The manipulation may be performed for all openings, selected ones or only for the active one. The bottom part of the window contains entering fields for the input of movement vector (specified with the help of components parallel to the axes *x* and *y* ).




*Window with parameters of copy/movement*

# Other data

This part contains load input and properties of the slab reinforcement.

## Loading

Following forces and moments may be specified for the detail:

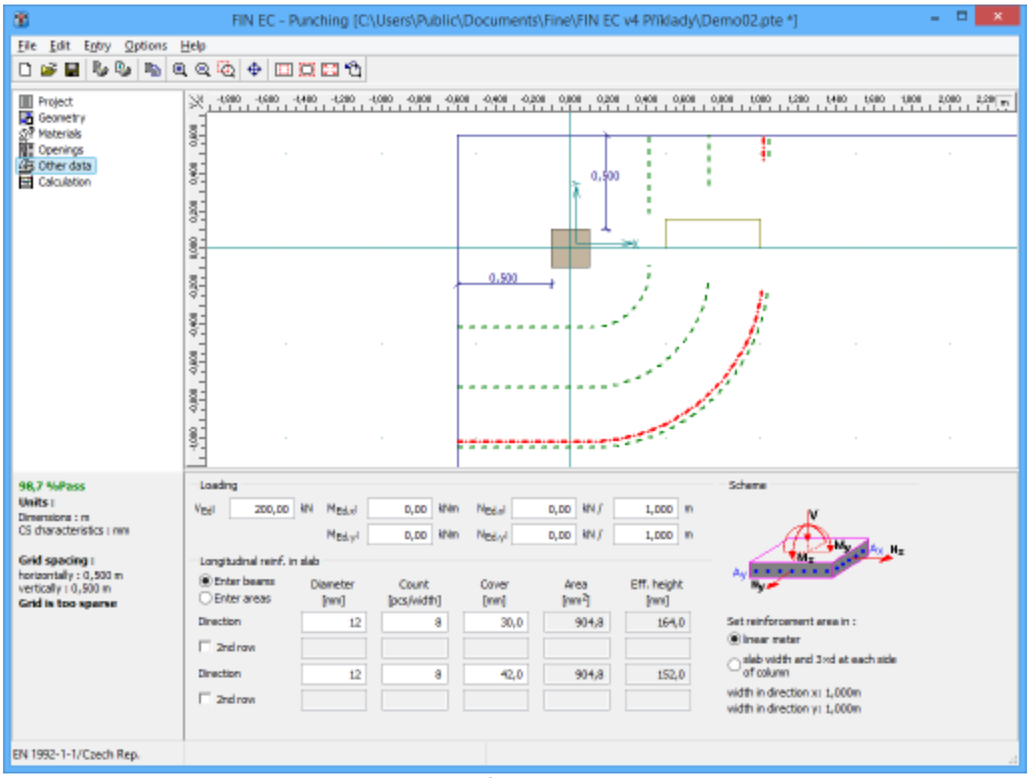
| | |
|---|---|
| $V_{Ed}$ | • The design value of shear force, positive value means the orientation in the gravity direction. |
| $\sigma$ | • The stress under the slab which is used for calculation of reaction inside the controlled perimeter. |
| $M_{Ed,x}$ | • The design value of bending moment about the axis $x$. The orientation of the positive value is displayed in the figure "**Scheme**". |
| $M_{Ed,y}$ | • The design value of bending moment about the axis $y$. The orientation of the positive value is displayed in the figure "**Scheme**". |
| $N_{Ed,x}$ | • The design value of axial force parallel to the axis $x$ in the slab. Such forces may be caused e.g. by the prestressing. The corresponding width for stress calculation should be also defined. |
| $N_{Ed,y}$ | • The design value of axial force parallel to the axis $y$ in the slab. Such forces may be caused e.g. by the prestressing. The corresponding width for stress calculation should be also defined. |

## Longitudinal reinforcement in slab

The longitudinal reinforcement in the slab may be defined for directions $x$ and $y$. The reinforcement volume may be defined with the help of reinforcement area or with the help of two reinforcement rows. Second row may have different (reinforcement is organized in two layers) or identical (for combining the regular reinforcement in slab and additional reinforcement above the supports) cover.

The corresponding slab width may be specified in the right part. The width may be specified as one linear meter or as the total considered slab width (column width + *3d* on both sides of the column) according to the chapter *6.4.4* of EN 1991-1-1. The reinforcement specified for one linear meter will be multiplied by the total considered slab width during the analysis.



*Part "Other data"*

# Analysis

This part contains tools for the input of shear reinforcement and also displays results of the analysis.

## Calculation parameters

The bottom part contains an option to specify the input method of the coefficient $\beta$ . Following options are available:

**Consider $\beta$ according to 6.4.3(6)**
- The value $\beta$ is set according to the figure *6.21N* and doesn't depend on the real value of bending moment. This simplified procedure may be used in cases where the neighbouring bays have similar spans and where the lateral stability of the building doesn't depend on frame action between slabs and columns.

**Calculate $\beta$ according to 6.4.3(3-5) - in moment direction**
- The procedure, where the factor $\beta$ is calculated according to the entered bending moments. The eccentricity is calculated in the direction of bending moment.

**Calculate $\beta$ according to 6.4.3(3-5) - in axes directions**
- The procedure, where the factor $\beta$ is calculated according to the entered bending moments.

**User defined value $\beta$**
- The option to input arbitrary value of the factor $\beta$ .

**Always consider reinforcement in the range 0 to 2d**
- If the distance of the control perimeter from a column is shorter than $2d$ , all shear reinforcement in the range $0$ to $2d$ will be considered in the calculation of the bearing capacity. This setting is available only for foundation slabs.

The setting "**Maximum resistance of reinforced perimeter**" affects the value of the factor $k_{max}$ . This factor is a ratio of maximum resistance of reinforced perimeter and a slab resistance.

**Consider $k_{max}$ according to 6.4.5(1)**
- The factor $k_{max}$ is calculated automatically according to the slab thickness. The calculation is based on the chapter NA 2.52a (CSN EN 1992-1-1 Z3) and NA 2.130 (STN EN 1992-1-1 A1/NA).

**Consider $k_{max}$ for double headed studs**
- The value of $k_{max}$ is *1.9* (according to STN EN 1992-1-1 A1/NA).

**User defined value $k_{max}$**
- An option to specify arbitrary value of $k_{max}$ .

The procedures used during the analysis are described in the chapter "**Punching**" of the theoretical help.

## Reinforcement input

Three different types of shear reinforcement are available for the input:

- **radial stirrups** - the shear reinforcement is organized in radial rows (figure *6.22A* in EN 1992-1-1)
- **concentrated stirrups** - - the shear reinforcement is organized in concentric rows

- **bent-ups** - shear reinforcement made of bent-up bars

The structural rules are also checked during the analysis. This control may be switched off with the help of the setting "**Check structural rules**". The structural rules are described in the theoretical part of help in the chapter "**Punching - structural rules**".

Two basic ways may be used for the input of the reinforcement: automatic design and manual input. The automatic design may be run by the button "**Design**". Parameters of the automatic design has to be specified in the window "**Reinforcement generation**", that appears after clicking on the button. The manual input of individual rows of the reinforcement can be done with the help of tools in the input frame. The parameters differ according to the reinforcement type. The diameter, spacing, position of the first link leg and number of branches has to be specified for radial stirrups. The reinforcement for other types is organized in the table, every row means one perimeter of the reinforcement. The number of bars per one perimeter means number of link legs in all cases. it means that number of double-sided bent-up bars for real structure is the half of the number specified for the design. The reinforcement in the table may be added and modified with the help of the toolbar on the left side of the table. Reinforcement properties are organized in the window "**Reinforcement edit**".

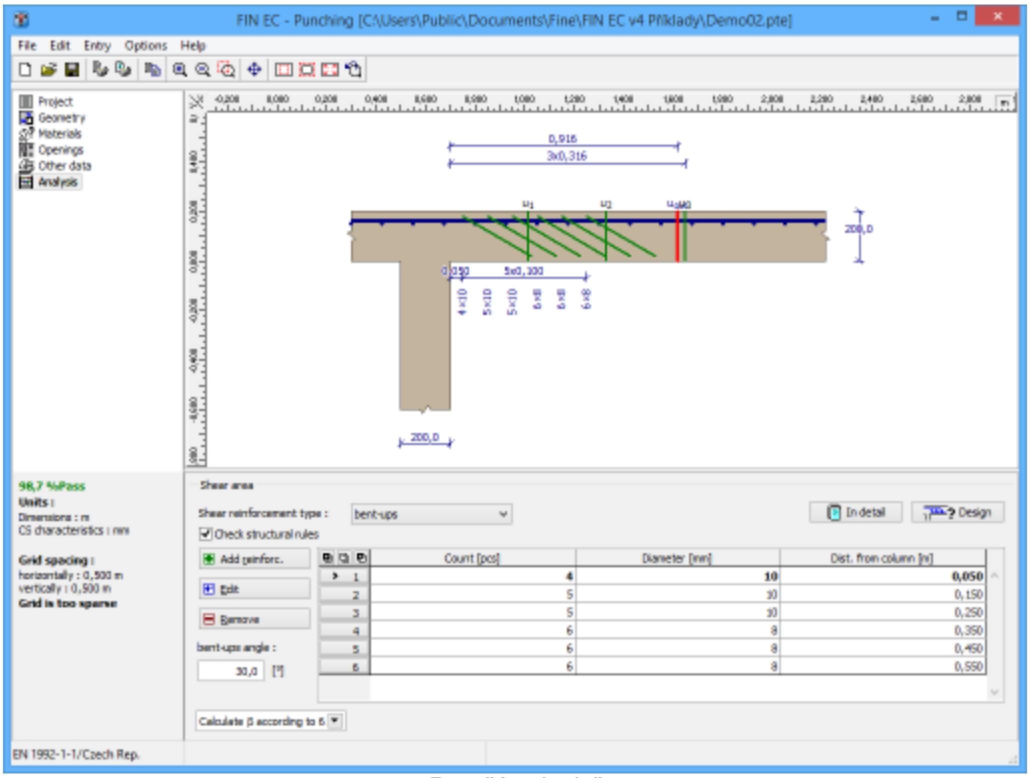
## Analysis

Results of the analysis are displayed in the left part of the main window. Detailed results may be displayed using button "**In detail**". These results are displayed in the new window, text in this window can be copied into clipboard using shortcut *Ctrl+C* and pasted into a document. Following three situation may be the result of the analysis:

**Reinforcement not possible**
- The maximum punching shear resistance $V_{Rd,max}$ is exceeded for some control perimeter. The slab isn't capable to transfer specified loading. The slab thickness or column's perimeter has to be increased.

**Fail**
- The condition $V_{Ed,max}$ < $V_{Rd,max}$ is fulfilled, slab is able to transfer entered loading. As the condition $V_{Ed}$ < $V_{Rd,cs}$ or $V_{Rd,c}$ isn't fulfilled for certain control perimeter, the shear reinforcement has to be added.

**Pass**
- The condition $V_{Ed,max}$ < $V_{Rd,max}$ is fulfilled, , slab is able to transfer entered loading. Also the condition $V_{Ed}$ < $V_{Rd,cs}$ or $V_{Rd,c}$ is fulfilled for all control perimeters, the resistance of the detail is OK.

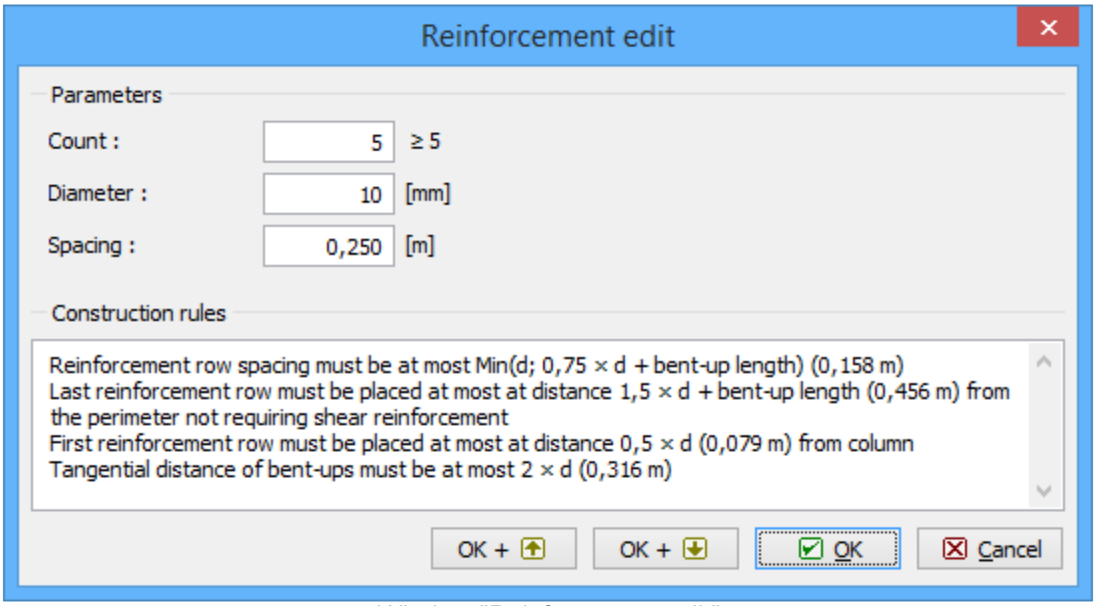


*Part "Analysis"*

# Reinforcement edit

The parameters of the reinforcement in one row may be specified in this window. The reinforcement is specified by the bars count and diameter and the position of the row measured as a distance from the column edge. The number of bars means the number of steel profiles along the whole perimeter. If the setting "**Check structural rules**" is switched on in the part "**Analysis**", the software checks the minimum number of bars in row (required by the maximum distance between bars).




*Window "Reinforcement edit"*

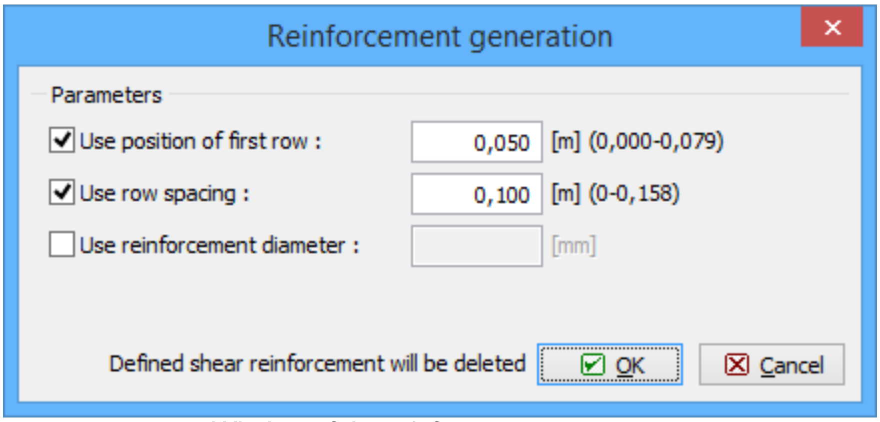
## Reinforcement generation

The parameters of the automatic design of the reinforcement may be specified in this window. The values, that should be respected during the automatic design, may be specified manually:

- Position of the first row
- Row spacing
- Reinforcement diameter

If all these parameters are specified, the software designs only number of rows and number of bars per every row. Limiting values according to the standard for all parameters are shown in brackets. These values are based on structural principles.

The existing reinforcement will be automatically deleted after closing the window with the help of the button "**OK**".




*Window of the reinforcement generator*

## Program Steel

The software "**Steel**" verifies steel elements according to EN 1993-1-1 and EN 1993-1-4.

### User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Input**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Three types of particular tasks can be used in the software:

**Section** • Fast analysis of steel cross-section with unlimited number of loads.

**Member** • Analysis of the whole member with specified diagrams of internal forces. This type is suitable for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

**Beam** • Comprehensive analysis of horizontal beam with unlimited number of supports. Both ultimate and serviceability limit states are considered during analysis.

These tasks can be added with the help of the buttons "**Add section**", "**Add member**" and "**Add beam**" in the heading of the tree menu.

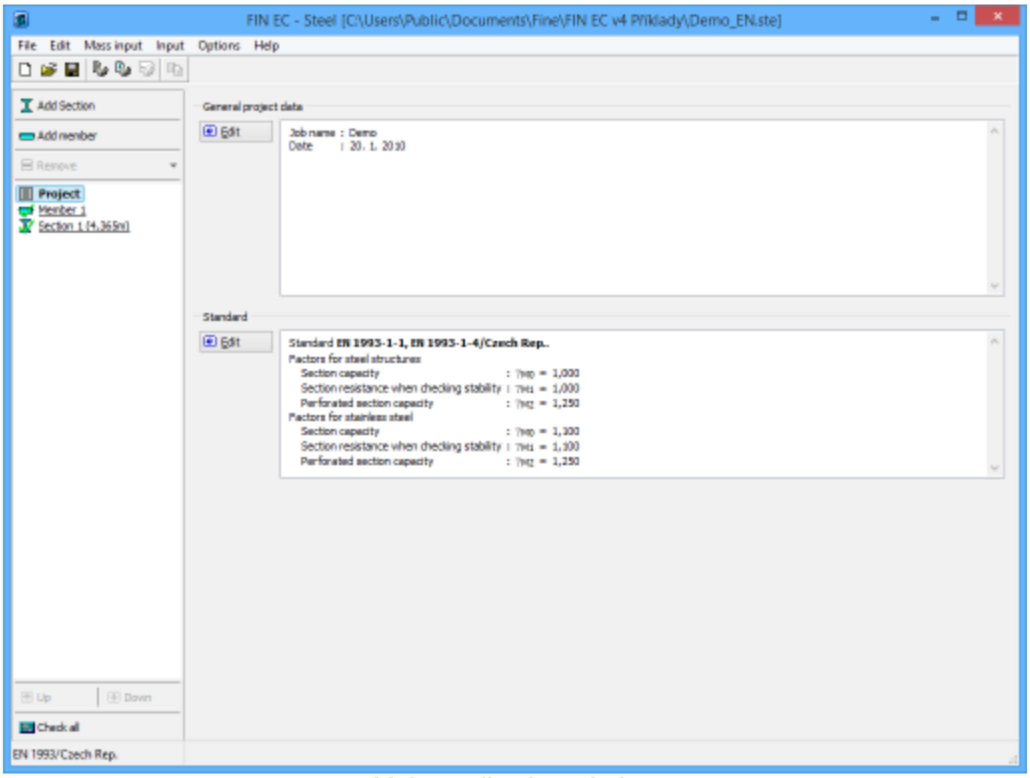
## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

Analysis and verifications used in the software are described in the theoretical part of the help.




*Main application window*

## Standard selection

The national annex and other properties of the design standard can be selected in this window. The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex. The values of partial factors $\gamma_M$ can be specified for option "**User defined**". Default values are based on the chapters *6.1* of EN 1993-1-1 and *5.1* of EN 1993-1-4.

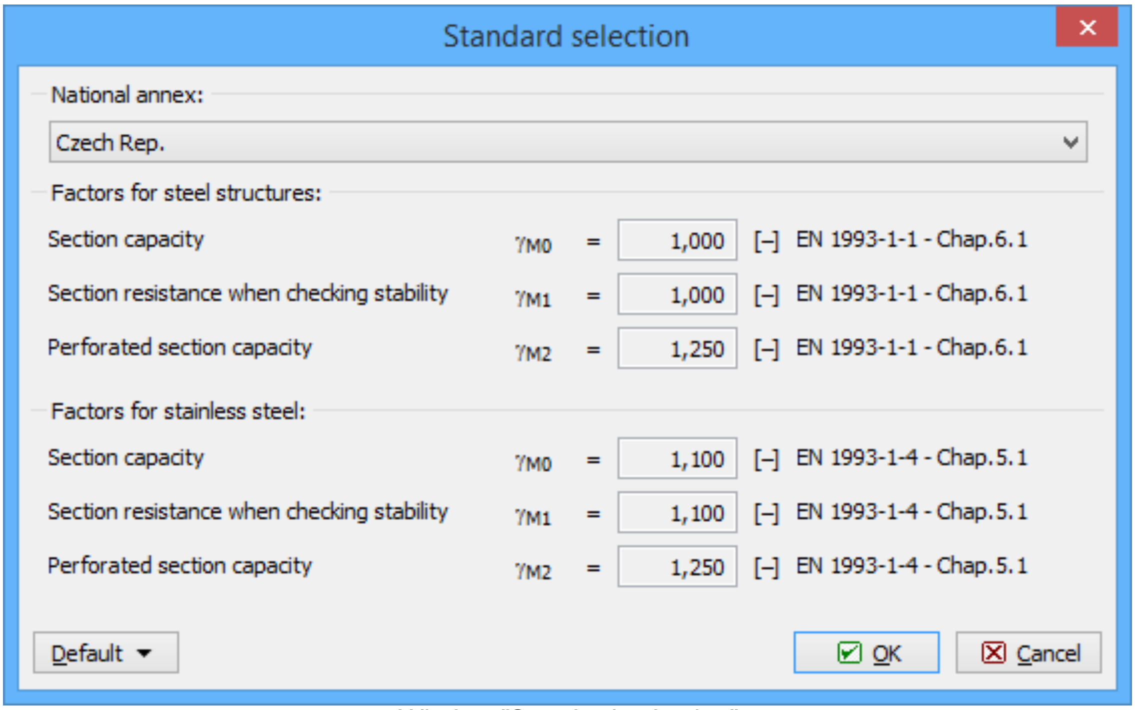
Partial factors are described in the theoretical part of help.

Button "**Default**" contains these two tools:

**Adopt default settings** • Set the default values for all parameters.

**Save settings as default** • Set entered parameters as defaults for new projects.

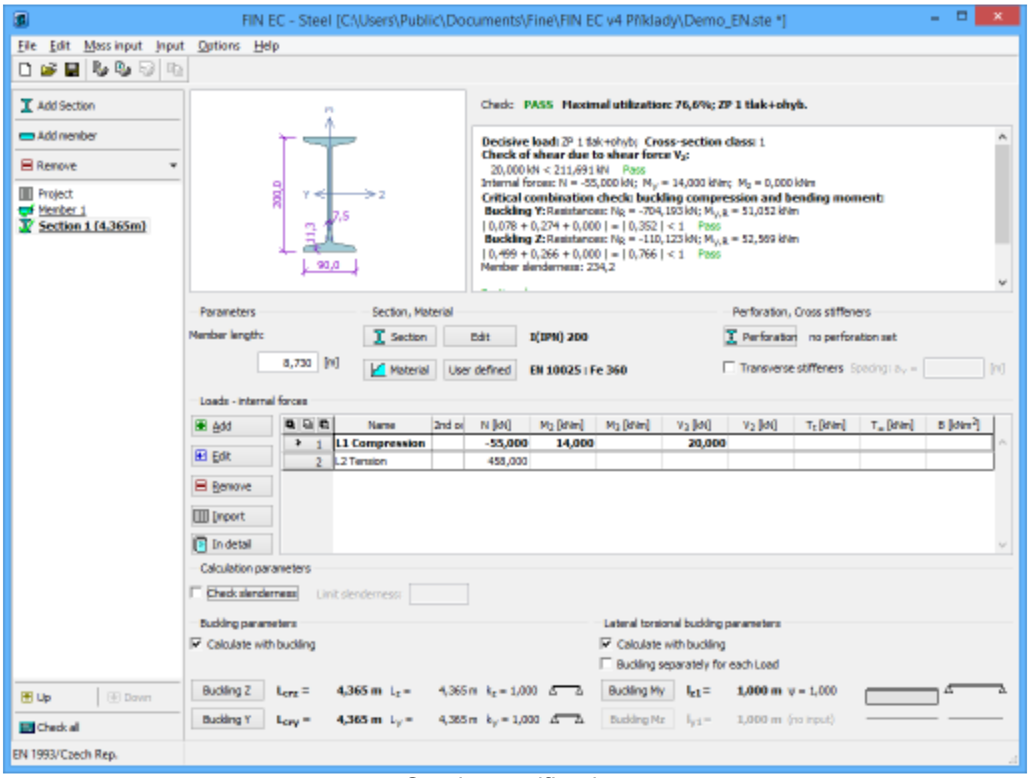
 




*Window "Standard selection"*

## Section

Task type "**Section**" is suitable for the fast verification of fire resistance of the steel cross-section, that may be loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".



*Section verification*

The window contains these parts:

### Parameters

The member length that is used in buckling and lateral torsional buckling verifications.

### Section, Material

Following buttons are placed in this part:

| | |
|---|---|
| **Section** | • Input of cross-section geometry in the window "**Cross-section edit**". |
| **Edit** | • Runs "**Cross-section editor**" in appropriate mode. If the cross-section isn't specified yet, the window "**Cross-section edit**" is opened.. |
| **Material** | • The steel grade can be selected in the window "**Materials catalogue**". Database contains strangth grades according to EN 10025, prEN 10113 a EN 10210-1. |
| **User defined** | • Input of arbitrary values of material characteristics in the window "**Material editor**". |

The background in described in the chapters "**Cross-sections**" and "**Material characteristics**" of the theoretical help.

## Perforation, Web stiffeners

The perforation of the cross-section (caused for example by holes for connectors) and web stiffeners may be specified in this part of the tree menu. Perforation can be entered with the help of the button "**Perforation**", which launches the window "**Perforation edit**". Specified perforation reduces the cross-sectional characteristics of the member, however, the resistance of the cross-section may be higher, as the ultimate strength $f_u$ is used in the analysis. This procedure is described in the part "**Perforation of cross-sections**" of the theoretical help. Web stiffeners are able to increase the resistance of thin webs, where the web crippling may appears. The spacing of stiffeners has to be specified in this case. The stiffness of these stiffeners isn't checked. The assumption is, that the stiffness is sufficient. Perforation and stiffeners can't be specified for built-up cross-sections.

## Battens

The connection of built-up members can be specified here. The parameters of the connection are organized in the window "**Battens**", that can be launched by the button "**Battens**".

## Loads - internal forces

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads can be also imported from text or **.csv* file. This feature can be used for import of large number of loads, that were calculated with the help of another structural engineering program. Import can be performed using window "**Load import**", that can be launched by button "**Import**".

## Calculation parameters

The slenderness verification can be switched on in this part. The maximum permitted value of slenderness ratio has to be specified by the user. The verification is described in the part "**Slenderness verification**" of the theoretical help.

## Buckling parameters

The buckling parameters can be specified in this part. The parameters are organized into two different directions $z$ and $y$ and can be specified in the window "**Buckling**", that can be launched by using the buttons "**Buckling Z**" and "**Buckling Y**". The main inputs (the buckling length, end conditions, basic length) are displayed on the right side of the buttons. If the axes $y$ and $z$ aren't the main axes of the cross-section (e.g. $L$-profiles), the buckling is considered in directions of the main cross-sectional axes $\eta$ and $\zeta$ during the design. The buckling analysis in directions $y$ and $z$ may be forced by switching off the setting "**Buckling to main axes η, ζ**". The buckling parameters input is enabled only for tasks with at least one load, that contains compressive force. The buckling analysis is described in the chapter "**Buckling resistance**" and "**Buckling resistance of built-up cross-sections**".

## LTB parameters

The parameters of lateral torsional buckling can be specified in this part. The lateral torsional buckling may be induced by bending moments $M_y$ or $M_z$. Only lateral torsional buckling in one direction may appear for one combination of loads. As the parameters of lateral torsional buckling depend on the moment distribution, the parameters may differ for individual loads. The same buckling parameters are considered for all loads as a default. The unique parameters for individual loads may be entered after using the setting "**Buckling separately for each load**". If the setting is switched on, the list box with all entered loads appears on the right side of the setting. The buckling parameters has to be specified individually for all loads (the load displayed in the list box is the active one for the parameters input) in this case. The parameters are organized into two different directions $z$ and $y$ and can be specified in the window "**LT buckling parameters**", that can be launched by using the buttons "**Buckling My**" and "**Buckling Mz**". The main inputs (the basic length, beam and load types) are displayed on the right side of the buttons. The parameters input is enabled only for tasks with at least one load, that contains corresponding bending moment. The analysis of lateral torsional buckling is described in the chapter "**Bending resistance**".

## Results

Results of the analysis for the worst load are displayed in the right upper part of the main window. Results consist of critical temperature and the fire resistance period.

The critical temperature is calculated as the temperature, for which the utilization of the member is equal to 100%. The value of critical temperature is calculated using iteration procedures. If the member fails for the temperature *20°C*, this temperature is signed as a critical one and the calculation stops. As the temperature *350°C* is set as a maximum one for members, that belong to the class *4*, according to the designing standard, this temperature is considered as a limiting value of the critical temperature for the class *4*.

Detailed results for the active load in the loads table can be displayed using button "**In detail**". These results are displayed in the new window, text in this window can be copied into clipboard using shortcut *Ctrl+C* and pasted into a document.

Analysis is described in the theoretical part of the help.

# Member

Task type "**Member**" is suitable for the detailed verification of the steel member, that is loaded by unlimited number of loads. This task type is suitable mainly for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

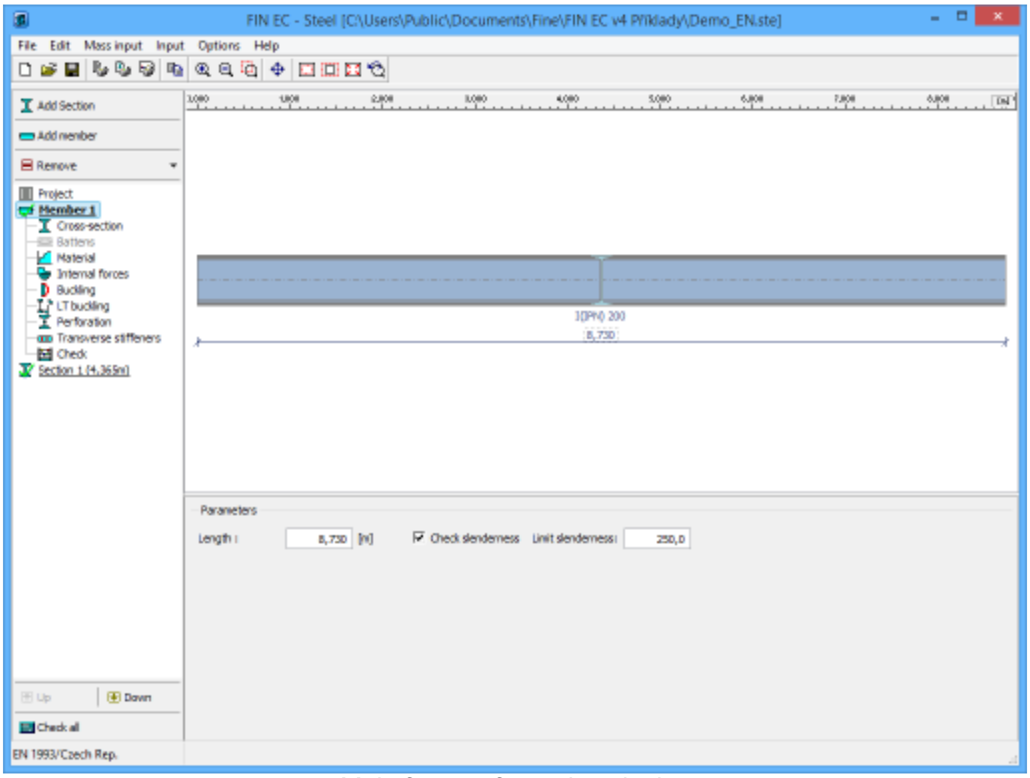
The main frame of member design contains these inputs:

| | |
|---|---|
| **Member length** | • Total member length specified in metres. |
| **Limit slenderness** | • Option for input of maximum slenderness ratio for the member. The verification is described in the chapter "**Slenderness verification**" of theoretical part of help |

The member design contains these parts:

- **Cross-section**
- **Battens**
- **Material**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Perforation**
- **Transverse stiffeners**
- **Analysis**

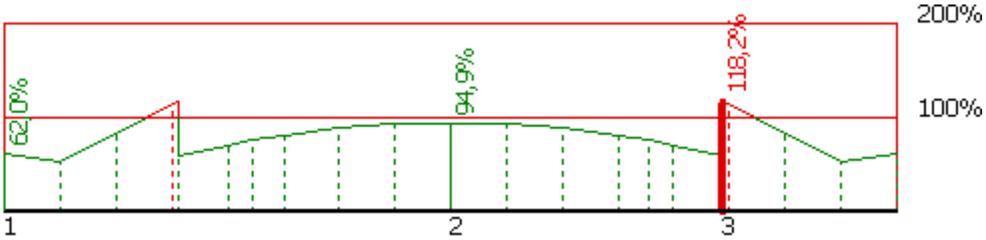
General work with members (addition, manipulation) is described in the chapter "**Tree menu**".


*Main frame of member design*

# Analysis

This part shows the results of structural analysis for the member. The results are displayed with the help of utilization diagram in the workspace. Passed member is coloured by green colour. The parts where the utilization exceeds 100% are coloured by red colour.



*Utilization of the member, red parts failed*

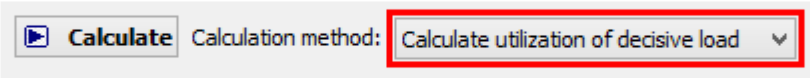
The frame in the bottom part contains tools for changing the analysis method and for the work with analysis sections (positions with detailed results).

## Analysis method

Analysis method can be selected in the upper part of the input frame. The analysis style and considered loads are selected according to the certain applied method. These options are available:

| | |
|---|---|
| **Calculate utilization of decisive load** | • Display the results for the decisive load (the load with the maximum utilization). All entered loads are considered in this option. |
| **Calculate envelope of maximum utilizations from all loads** | • Display envelope of the maximum utilization in every point of the member length. All entered loads are considered in this option. |
| **Individual loads** | • Show results for selected load. |

The analysis have to be run by the button "**Analyse**" after the change of the analysis method.



*Selection of analysis method*

## Verification sections

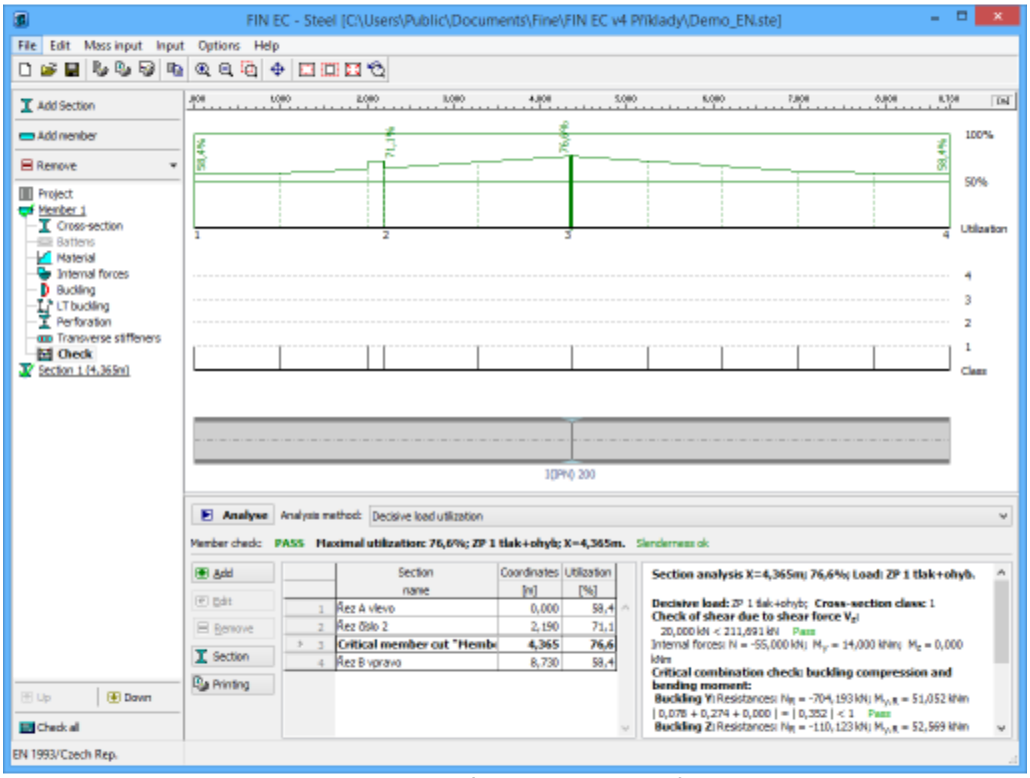
Verification sections are used for display of detailed results in certain points along the member length. These verification sections can be converted into standalone "**Section**" tasks, the results can be printed using graphical outputs. The verification section is created automatically in the position with the worst utilization, other sections can be added manually.

These tools are available for the work with sections:

| | |
|---|---|
| **Section** | • Converts the active section on member into standalone task of "**Section**" type. All member properties (cross-section, material, internal forces, parameters of buckling and LT buckling) are copied into the new task. |
| **Add** | • Inserts a new section on member. The detailed results can be displayed for this section. Properties of the new section have to be specified in the window "**New section for analysis**". |
| **Edit** | • Edit of the active cross-section properties (name, coordinate). |
| **Remove** | • Remove the active section from the list. |
| **Printing** | • Print results for all entered sections using printing window. |

The new section for analysis can be also added using active workspace. New section can be added by double-click on the needed position on the member.



*Part "Analysis" of the member verification*

# Beam

The task type "**Beam**" is suitable for the comprehensive analysis of general horizontal beam. Only single axis bending is supported in this task type.

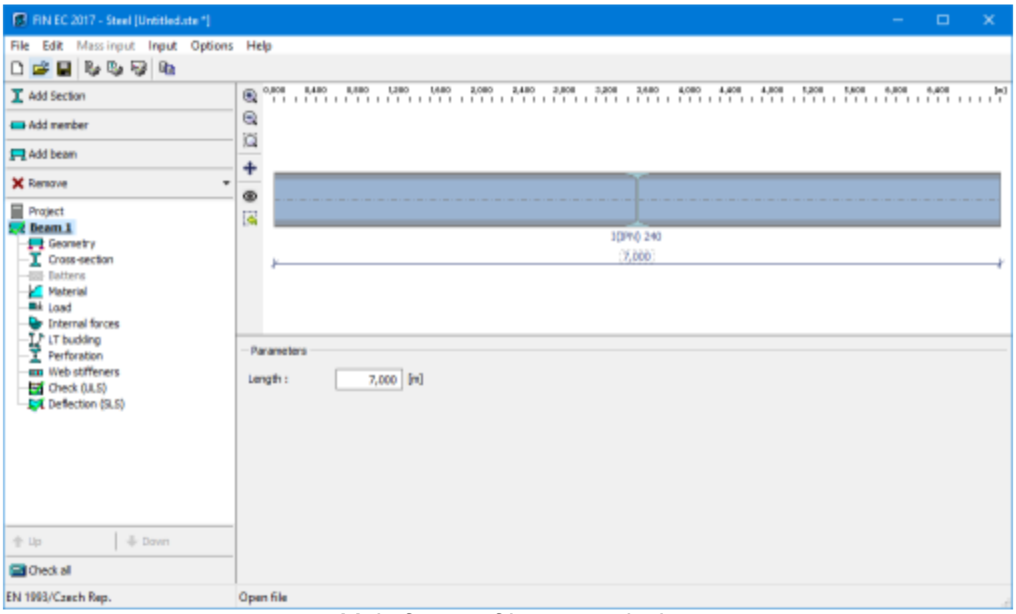
The main frame of member design contains an input line for changing the total beam length.

The beam analysis contains these parts:

- **Cross-section**
- **Battens**
- **Material**
- **Load**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Perforation**
- **Transverse stiffeners**
- **Analysis (ULS)**
- **Deflection (SLS)**

General work with beams (addition, manipulation) is described in the chapter "**Tree menu**".

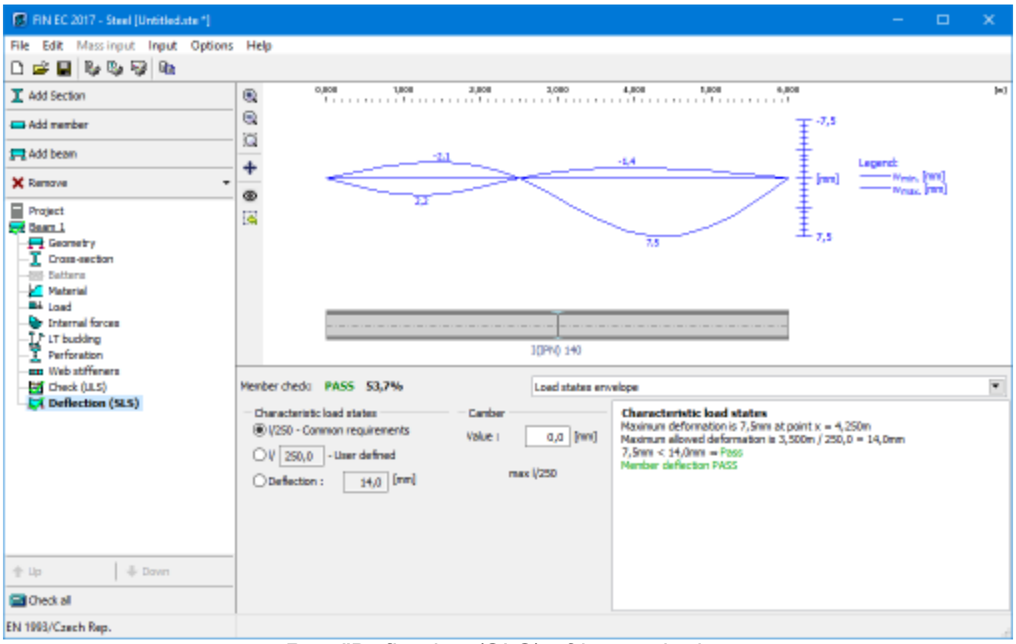



*Main frame of beam analysis*

## Deflection (SLS)

This part of the beam design shows results of deflection verification (serviceability limit state) along the member length. The verification is performed only for load combinations that are dedicated to the serviceability limit state design. If there isn't any entered load combination for serviceability limit state in the part "**Load**", the results aren't available. The frame in the bottom part contains these settings for deflection verification:

| | |
|---|---|
| **l/250 - Common requirements** | • This limit shall be considered for deflection that could impair the appearance and general utility of the structure. |
| **l/ - User defined requirements** | • User defined limit specified as span-depending value |
| **Deflection** | • User defined limit specified as an absolute value in *mm* |




*Part "Deflection (SLS) of beam design*

## Program Steel Fire

The software "**Steel Fire**" verifies fire resistance of reinforced concrete cross-sections according to EN 1992-1-2.

### User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Input**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Three types of particular tasks can be used in the software:

| | |
|---|---|
| **Section** | • Fast analysis of steel cross-section with unlimited number of loads. |
| **Member** | • Analysis of the whole member with specified diagrams of internal forces. This type is suitable for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**". |
| **Beam** | • Comprehensive analysis of horizontal beam with unlimited number of supports. |

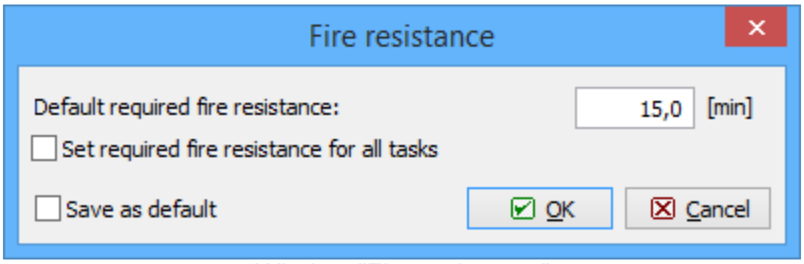
These tasks can be added with the help of the buttons "**Add section**", "**Add member**" or "**Add beam**" in the heading of the tree menu.

## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

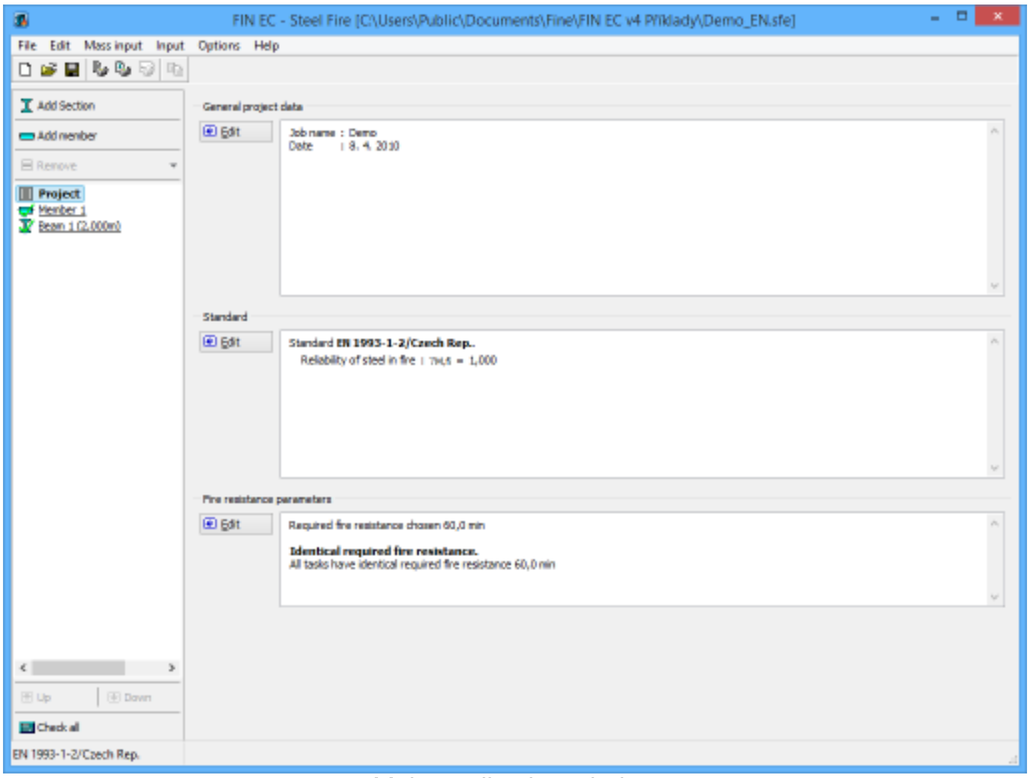
The part "**Fire resistance parameters**" contains an option for input of default value of fire resistance for all tasks (sections and members) of the project. The input can be done in the window "**Fire resistance**", that can be launched by using the button "**Edit**". The check box **"Set required fire resistance for all tasks**" sets the specified fire resistance also to all existing tasks in the project. The setting "**Save as default**" will set specified fire resistance as a default for all new projects.




*Window "Fire resistance"*

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

Analysis and verifications used in the software are described in the theoretical part of the help.




*Main application window*

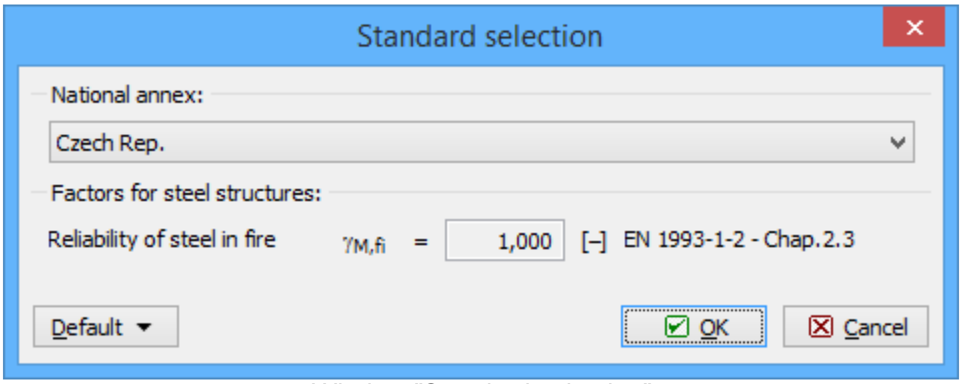
## Standard selection

The national annex and other properties of the design standard can be selected in this window. The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex. The value of partial factor $\gamma_{M,fi}$ can be specified for option "**User defined**". Default value is based on chapter *2.3* of EN 1993-1-2.

Partial factors are described in the theoretical part of help.

Button "**Default**" contains these two tools:

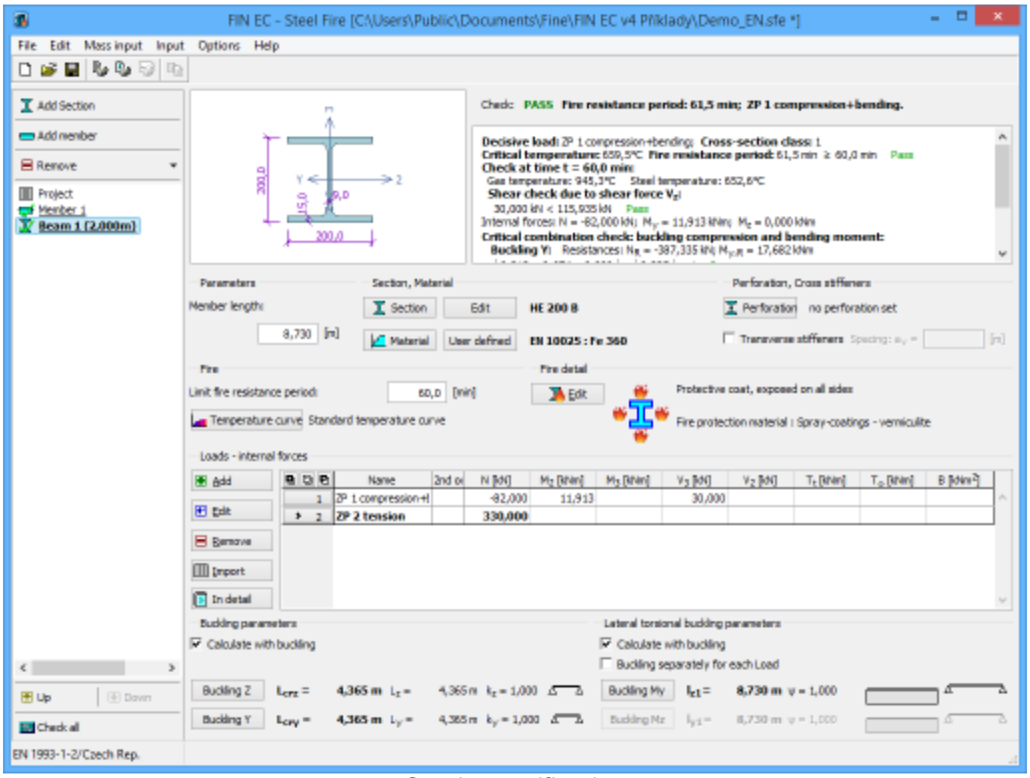
| | |
|---|---|
| **Adopt default settings** | • Set the default values for all parameters. |
| **Save settings as default** | • Set entered parameters as defaults for new projects. |




*Window "Standard selection"*

## Section

Task type "**Section**" is suitable for the fast verification of fire resistance of the steel cross-section, that may be loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".




*Section verification*

The window contains these parts:

### Parameters

The member length that is used in buckling and lateral torsional buckling verifications.

### Section, Material

Following buttons are placed in this part:

| | |
|---|---|
| **Section** | • Input of cross-section geometry in the window "**Cross-section edit**". |
| **Edit** | • Runs "**Cross-section editor**" in appropriate mode. If the cross-section isn't specified yet, the window "**Cross-section edit**" is opened.. |
| **Material** | • The steel grade can be selected in the window "**Materials catalogue**". Database contains strength grades according to EN 10025, prEN 10113 a EN 10210-1. |
| **User defined** | • Input of arbitrary values of material characteristics in the window "**Material editor**". |

The background in described in the chapters "**Cross-sections**" and "**Material characteristics**" of the theoretical help.

## Perforation, Web stiffeners

The perforation of the cross-section (caused for example by holes for connectors) and web stiffeners may be specified in this part of the tree menu. Perforation can be entered with the help of the button "**Perforation**", which launches the window "**Perforation edit**". Specified perforation reduces the cross-sectional characteristics of the member, however, the resistance of the cross-section may be higher, as the ultimate strength $f_u$ is used in the analysis. This procedure is described in the part "**Perforation of cross-sections**" of the theoretical help. Web stiffeners are able to increase the resistance of thin webs, where the web crippling may appears. The spacing of stiffeners has to be specified in this case. The stiffness of these stiffeners isn't checked. The assumption is, that the stiffness is sufficient. Perforation and stiffeners can't be specified for built-up cross-sections.

## Battens

The connection of built-up members can be specified here. The parameters of the connection are organized in the window "**Battens**", that can be launched by the button "**Battens**".

## Fire

The temperature curve and required time of fire resistance can be specified in this part. The value defined in the main application screen is used as a default. The calculated critical temperature is compared with this value.

The temperature curve is used for the determination of the temperature of gas in time. The curve may be changed in the window "**Temperature curve**" (button "**Curve**"). The expressions that represent temperature curves are described in the chapter "**Temperature development**" of theoretical help.

## Fire detail

The fire protection parameters can be specified in this part. The fire resistance parameters are organized in the window "**Fire detail**" that can be launched by the button "**Edit**".

## Loads - internal forces

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads can be also imported from text or **.csv* file. This feature can be used for import of large number of loads, that were calculated with the help of another structural engineering program. Import can be performed using window "**Load import**", that can be launched by button "**Import**".

## Buckling parameters

The buckling parameters can be specified in this part. The parameters are organized into two different directions $z$ and $y$ and can be specified in the window "**Buckling**", that can be launched by using the buttons "**Buckling Z**" and "**Buckling Y**". The main inputs (the buckling length, end conditions, basic length) are displayed on the right side of the buttons. If the axes $y$ and $z$ aren't the main axes of the cross-section (e.g. $L$-profiles), the buckling is considered in directions of the main cross-sectional axes $\eta$ and $\zeta$ during the design. The buckling analysis in directions $y$ and $z$ may be forced by switching off the setting "**Buckling to main axes η, ζ**". The buckling parameters input is enabled only for tasks with at least one load, that contains compressive force. The buckling analysis is described in the chapter "**Verification of solid cross-sections**" and "**Verification of built-up cross-sections**".

## LTB parameters

The parameters of lateral torsional buckling can be specified in this part. The lateral torsional buckling may be induced by bending moments $M_y$ or $M_z$. Only lateral torsional buckling in one direction may appear for one combination of loads. As the parameters of lateral torsional buckling depend on the moment distribution, the parameters may differ for individual loads. The same buckling parameters are considered for all loads as a default. The unique parameters for individual loads may be entered after using the setting "**Buckling separately for each load**". If the setting is switched on, the list box with all entered loads appears on the right side of the setting. The buckling parameters has to be specified individually for all loads (the load displayed in the list box is the active one for the parameters input) in this case. The parameters are organized into two different directions $z$ and $y$ and can be specified in the window "**LT buckling parameters**", that can be launched by using the buttons "**Buckling My**" and "**Buckling Mz**". The main inputs (the basic length, beam and load types) are displayed on the right side of the buttons. The parameters input is enabled only for tasks with at least one load, that contains corresponding bending moment. The analysis of lateral torsional buckling is described in the chapter "**Verification of solid cross-sections**".

### Results

Results of the analysis for the worst load are displayed in the right upper part of the main window. Results consist of critical temperature and the fire resistance period.

The critical temperature is calculated as the temperature, for which the utilization of the member is equal to 100%. The value of critical temperature is calculated using iteration procedures. If the member fails for the temperature *20°C*, this temperature is signed as a critical one and the calculation stops. As the temperature *350°C* is set as a maximum one for members, that belong to the class *4*, according to the designing standard, this temperature is considered as a limiting value of the critical temperature for the class *4*.

Detailed results for the active load in the loads table can be displayed using button "**In detail**". These results are displayed in the new window, text in this window can be copied into clipboard using shortcut *Ctrl+C* and pasted into a document.

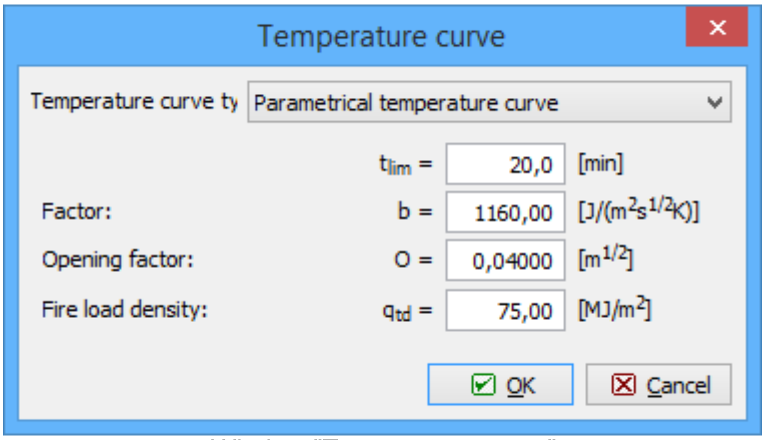
Analysis is described in the theoretical part of the help.

## Temperature curve

The temperature curve that is used for the determination of the temperature of gas in time may be selected here. Following options are available:

- **Standard temperature curve** - nominal curve defined in EN 13501-2. This curve describes the fully developed fire.
- **External fire curve** - nominal curve intended for the outside of separating external walls that can be exposed to fire from different parts of the facade (directly from the inside of the corresponding fire compartment or from a compartment situated below or adjacent to the respective external wall)
- **Hydrocarbon fire curve** - nominal curve for representing effects of a hydrocarbon type fire
- **Parametric temperature curve** - this curve is effected by the physical parameters that describe the conditions in the fire compartment.

The expressions that represent temperature curves are described in the chapter "**Temperature development**" of theoretical help.




*Window "Temperature curve"*

## Fire detail

This window contains a range of available fire details (style of fire protection). Details are divided into two basic categories: unprotected ones and protected ones. The sorting according to the number of exposed sides follows for both categories.

Unprotected cross-sections may be exposed to fire from all sides or only from three sides (in case that the cross-section is covered from one side e.g. by a slab). Cross-sections may be also protected by concrete slab partially (part of cross-section height is fixed with concrete). For such cases, protected height ($h_{pr}$) or exposed height ($h_{exp}$) has to be specified.

There are two general types of fire protection: coatings (the thickness $d_p$ has to be specified) and protected boxes (inputs are the thickness $d_p$ and box size). Protected details are also differentiated according to the number of exposed sides.

Materials of a fire protection may be divided into two main categories: coatings and board materials for protected boxes. The materials database contains wide range of items for both categories. Any other material may be specified manually with the help of user defined material characteristics.

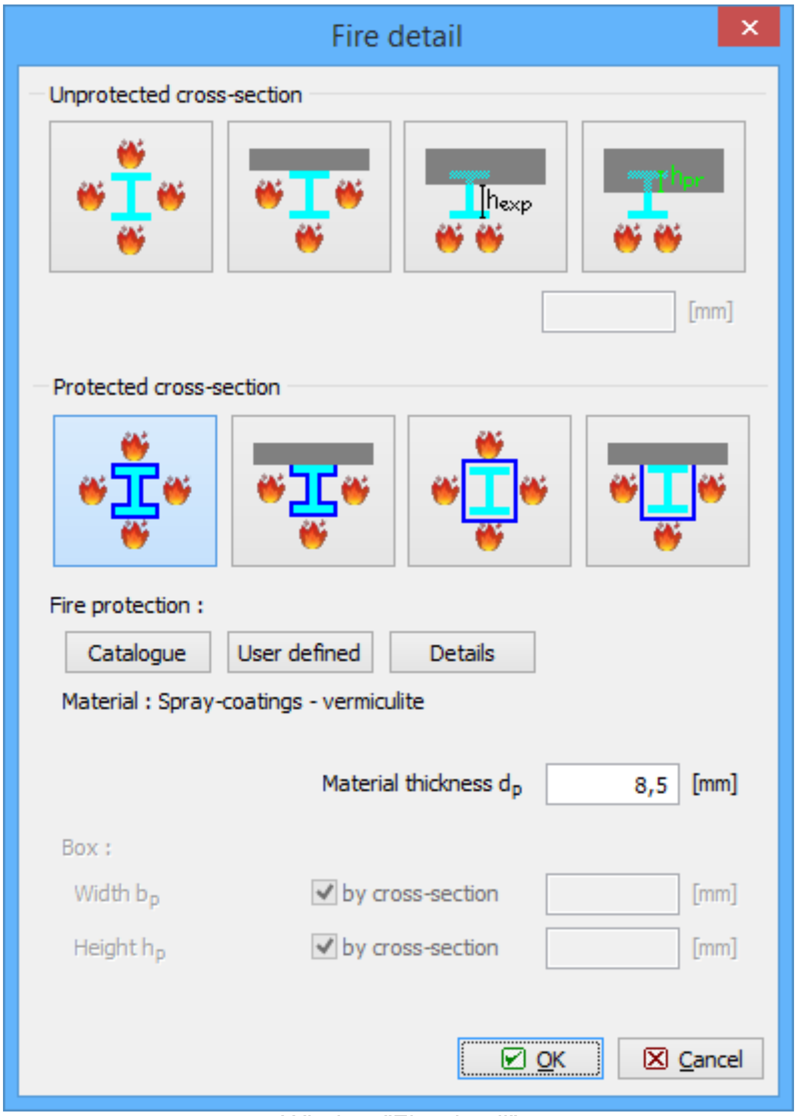
Following buttons are available for the input of fire protective material:

**Catalogue** • The selection of material from pre-defined database in the window "**Materials catalogue**".

**User defined** • The input of arbitrary material with the help of material characteristics in the window "**Material editor**".

**Details** • The window that shows complete material characteristics of the specified material.

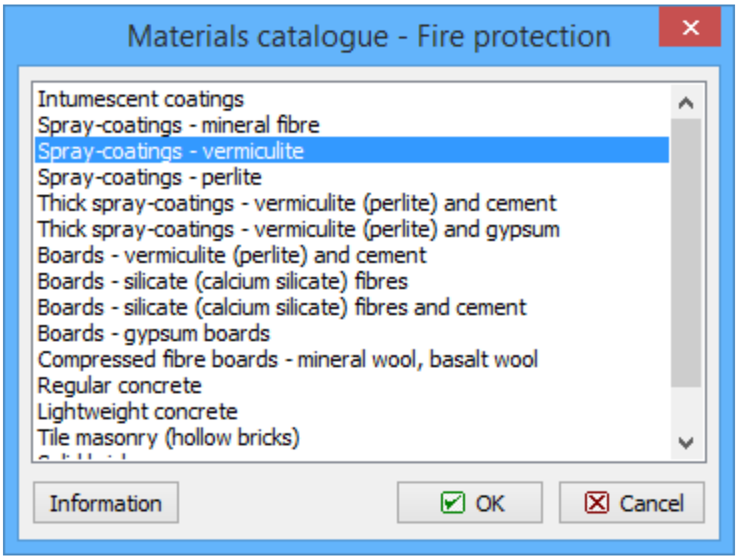



*Window "Fire detail"*

## Materials catalogue - Fire protection

This window contains the database of fire protective materials.

The complete list of material characteristics for selected material may be opened using the "**Information**" button.

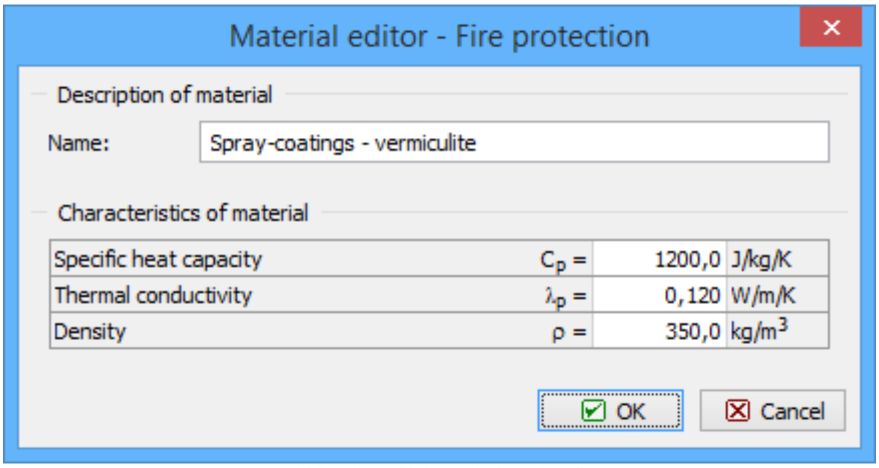



*Window "Materials catalogue"*

## Material editor - Fire protection

The arbitrary material characteristics can be specified in this window. The characteristics are described in the chapter "**Temperature development**" of the theoretical part of the help.




*Window "Material editor"*

## Member

Task type "**Member**" is suitable for the detailed verification of the steel member, that is loaded by unlimited number of loads. This task type is suitable mainly for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

The main frame of member design contains these inputs:

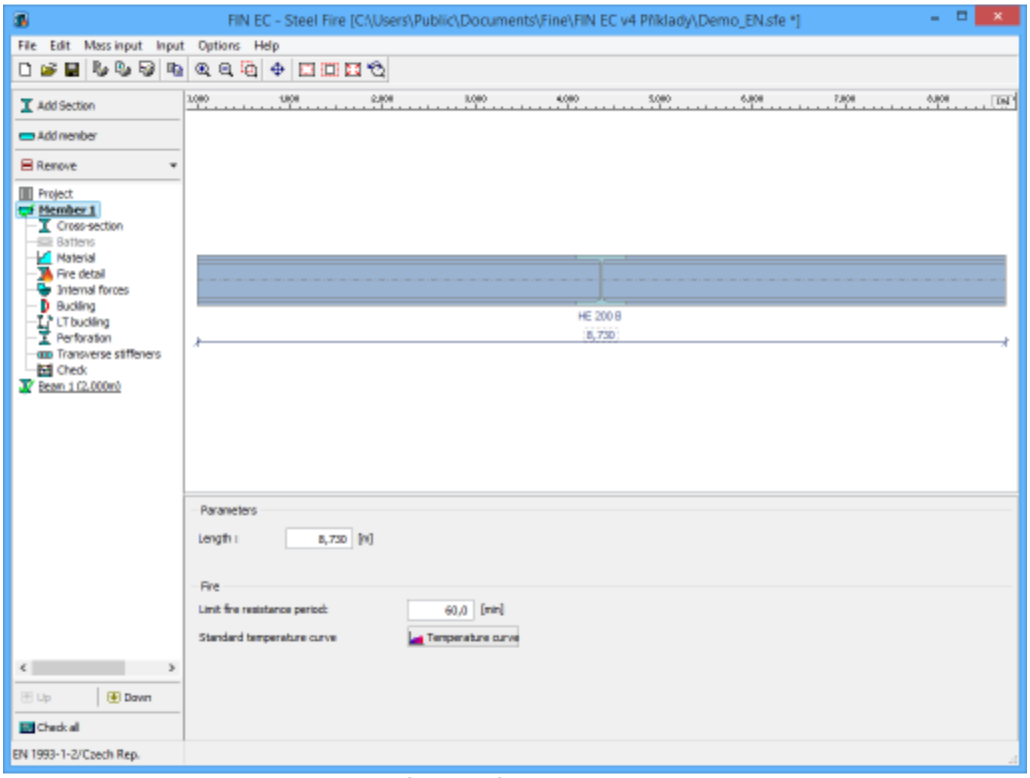
| | |
|---|---|
| **Member length** | • Total member length specified in metres. |
| **Limit fire resistance period** | • Fire resistance in minutes, results in part "**Analysis**" are compared with this value. |
| **Temperature curve** | • The choice of the temperature curve, that is used for the determination of the temperature of gas in time, in the window "**Temperature curve**". |

The member design contains these parts:

- **Cross-section**
- **Battens**
- **Material**
- **Fire detail**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Perforation**
- **Transverse stiffeners**
- **Analysis**

General work with members (addition, manipulation) is described in the chapter "**Tree menu**".

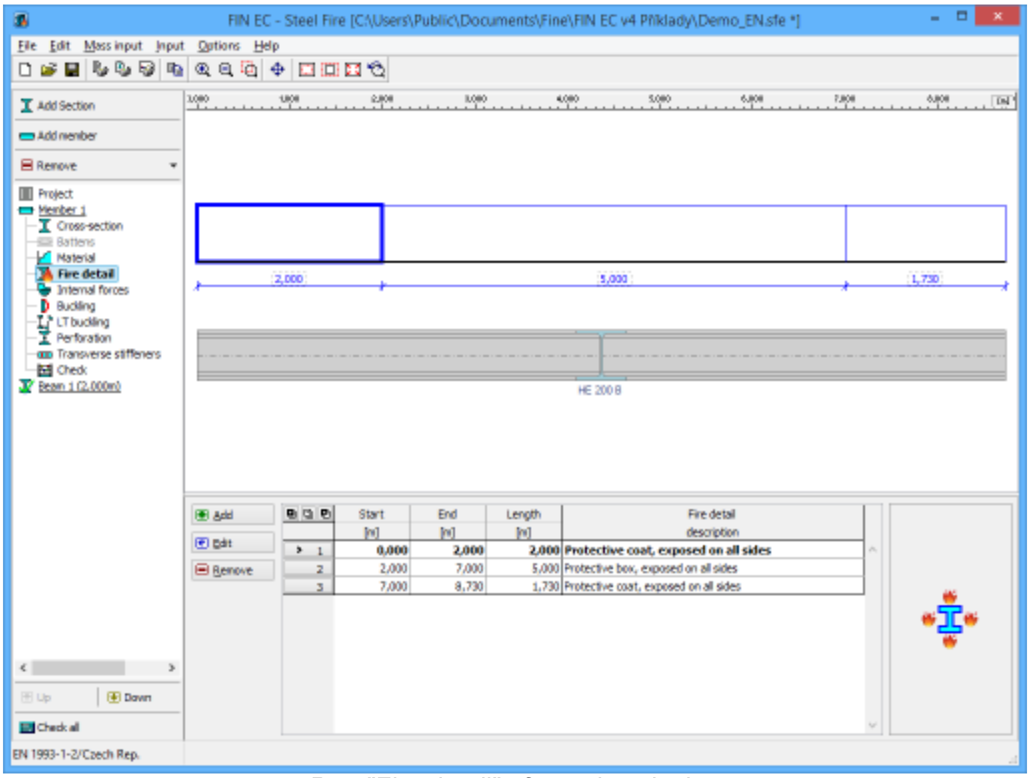


Main frame of member design

## Fire detail

The fire protection parameters can be specified in this part of the tree menu. These parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different parameters of fire resistance. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The fire resistance parameters are organized in the window "**Fire detail**". More sectors can be added (button "**Add**") for input of different fire resistance parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

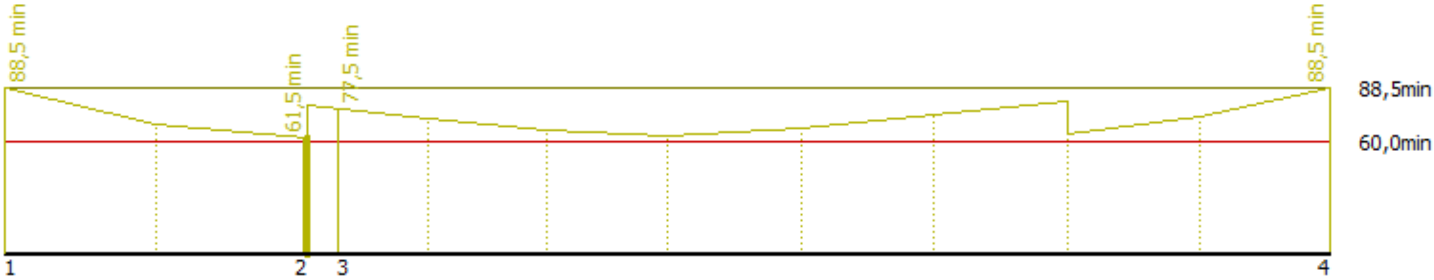


Part "Fire detail" of member design

# Analysis

This part shows the results of structural analysis for the member. The results are displayed with the help of temperature diagram in the workspace. The parts where the limit fire resistance isn't fulfilled are coloured by red colour.




*Fire resistance period along the member length*

The frame in the bottom part contains tools for changing the analysis method and for the work with analysis sections (positions with detailed results).

## Analysis method

The analysis method and considered loads are selected according to the certain applied method. These options are available:

| | |
|---|---|
| **Fire resistance of decisive load** | • Display the results for the decisive load (the load with the minimum value of fire resistance period). All entered loads are considered in this option. |
| **Envelope of minimum fire resistance period** | • Display envelope of the fire resistance period in every point of the member length. All entered loads are considered in this option. |
| **Individual loads** | • Show results for selected load. |

The analysis have to be run by the button "**Analyse**" after the change of the analysis method.

## Verification sections

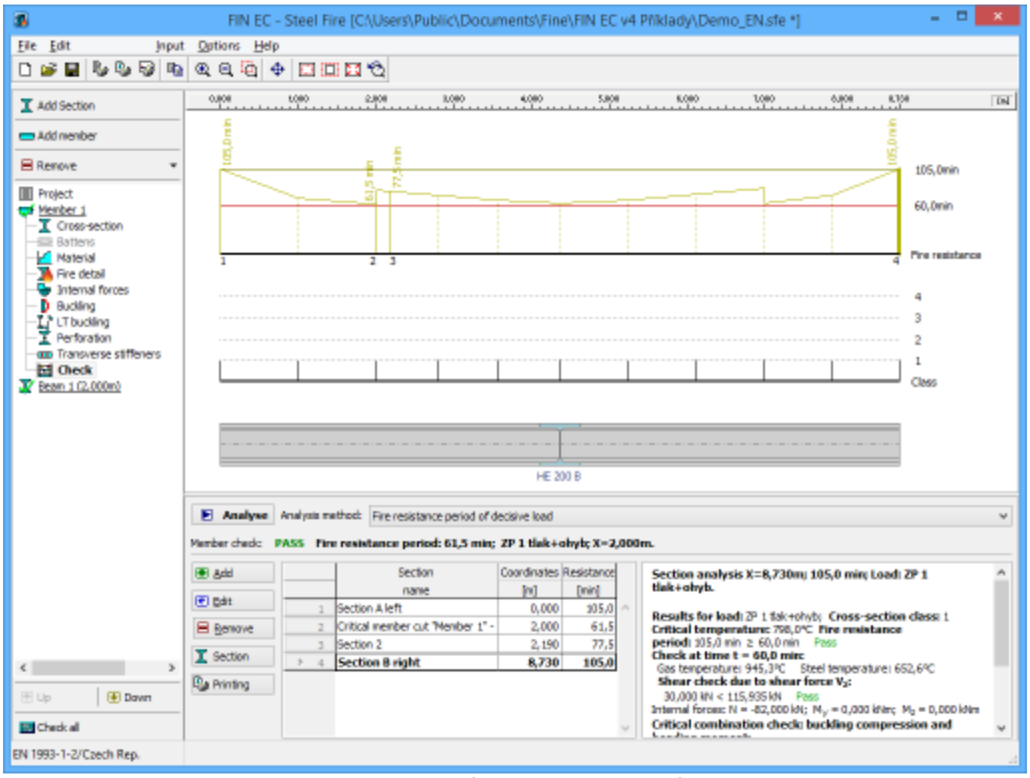
Verification sections are used for display of detailed results in certain points along the member length. These verification sections can be converted into standalone "**Section**" tasks, the results can be printed using graphical outputs. The verification section is created automatically in the position with the lowest critical temperature, other sections can be added manually.

These tools are available for the work with sections:

| | |
|---|---|
| **Section** | • Converts the active section on member into standalone task of "**Section**" type. All member properties (cross-section, material, internal forces, parameters of buckling and LT buckling) are copied into the new task. |
| **Add** | • Inserts a new section on member. The detailed results can be displayed for this section. Properties of the new section have to be specified in the window "**New section for analysis**". |
| **Edit** | • Edit of the active cross-section properties (name, coordinate). |
| **Remove** | • Remove the active section from the list. |
| **Printing** | • Print results for all entered sections using printing window. |

The new section for analysis can be also added using active workspace. New section can be added by double-click on the needed position on the member.



*Part "Analysis" of the member verification*

# Beam

Task type "**Beam**" is suitable for the comprehensive analysis of general horizontal beam. Only single axis bending is supported in this task type.

The main frame of beam analysis contains these inputs:

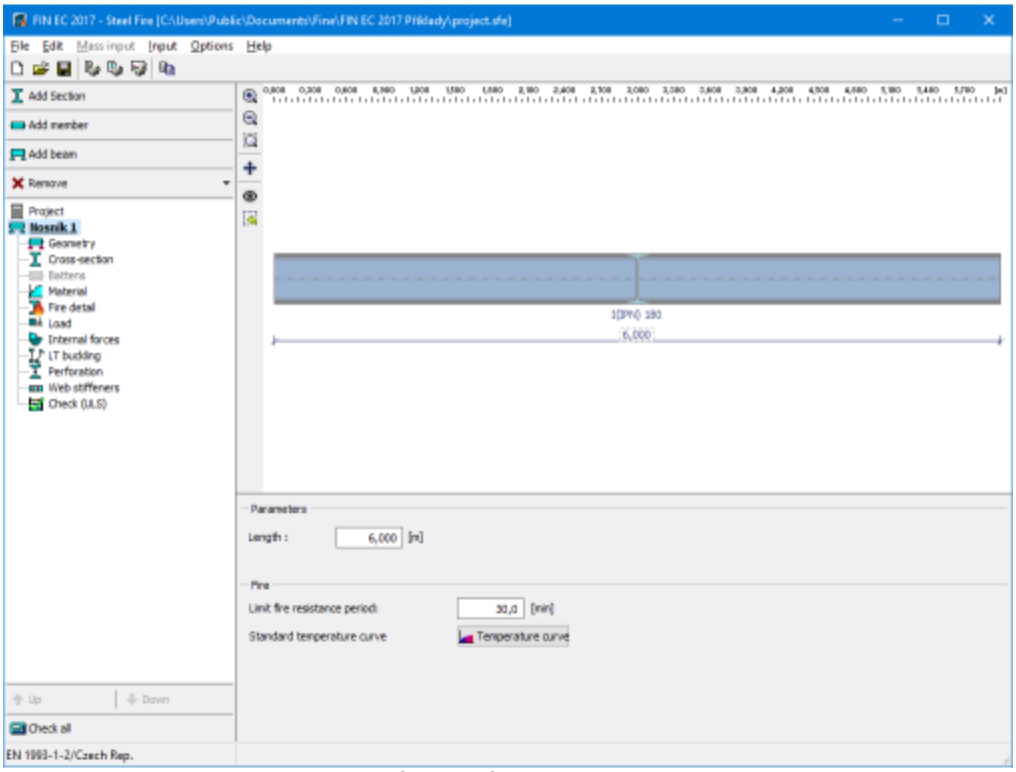
| | |
|---|---|
| **Member length** | • Total member length specified in metres. |
| **Limit fire resistance period** | • Fire resistance in minutes, results in part "**Analysis**" are compared with this value. |
| **Temperature curve** | • The choice of the temperature curve, that is used for the determination of the temperature of gas in time, in the window "**Temperature curve**". |

The beam design contains these parts:

- **Geometry**
- **Cross-section**
- **Battens**
- **Material**
- **Fire detail**
- **Load**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Perforation**
- **Transverse stiffeners**
- **Analysis**

General work with beams (addition, manipulation) is described in the chapter "**Tree menu**".




*Main frame of member design*

# Program Steel Connections

The software "**Steel connections**" verifies steel connections according to EN 1993-1-8.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Entry**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Following particular tasks ("**Joints**") can be verified in the software:

| | |
|---|---|
| **Beams connection to column** | • The joint that connects beams or truss elements to column. Connections can be applied from all four sides of the column. |
| **Splice joint of beam** | • The splicing joint of two beam parts with the help of end plates. Both pinned and stiff details are available. |
| **Beam connection to girder** | • The connection of beam to the another beam (girder). The detail can be designed using end plate or fin plate. The joint can be both sided. |
| **Truss joint** | • The joint of truss structure made of RHS or L-profiles |
| **Column base** | • Connection of steel column to foundation using bolts or concreting. The stiffness of the detail with base plate is calculated according to the joint geometry. |

These tasks can be added with the help of the button "**Add**" in the heading of the tree menu.

## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

The frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

## Calculation parameters

This frame contains the choice of structure type. Available are two options:

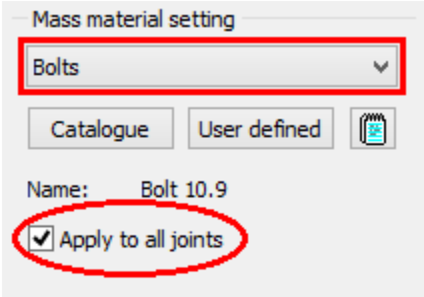
- **Frame with sway mode failure** - frames with significant horizontal displacements

- **Frame with no sway mode failure** - frames where the bracing system reduces significantly the horizontal displacements

This parameter is used in the classification of joints.

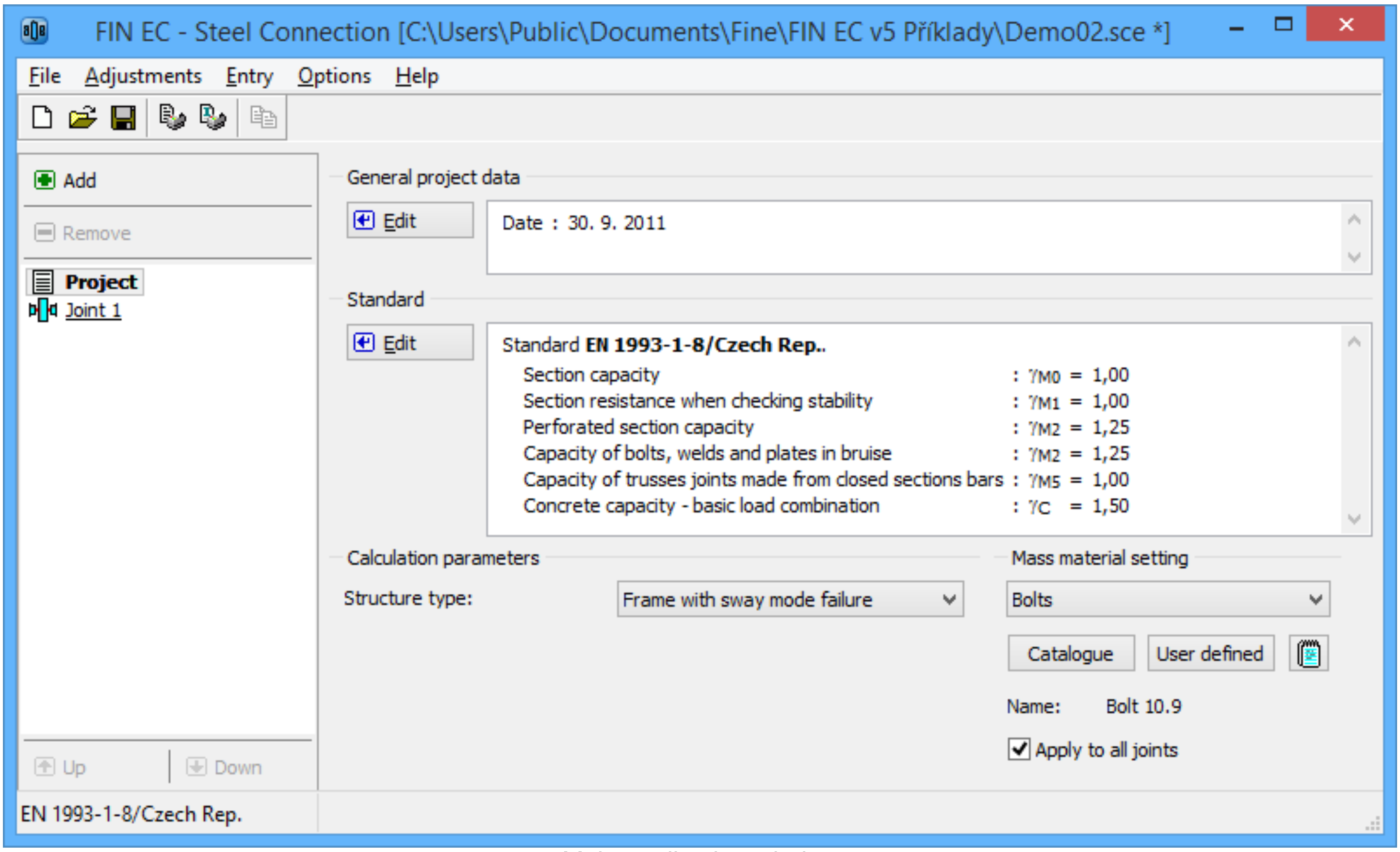
### Global material input

This part can be used for the input of identical material for all tasks in the project. For such input, it is necessary to select appropriate group (bolts, structural elements etc.), check the setting "**Apply to all joints**" and select the material from pre-defined database (button "**Catalogue**") or specify the material numerically (button "**User defined**").




*Bolt material for all joints*

The appearance of the workspace may be modified in the window "**Global settings**", that can be opened from the main menu.




*Main application window*

## Standard selection

The national annex and other properties of the design standard can be selected in this window. The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex. The values of partial factors $\gamma_M$ can be specified for option "**User defined**".

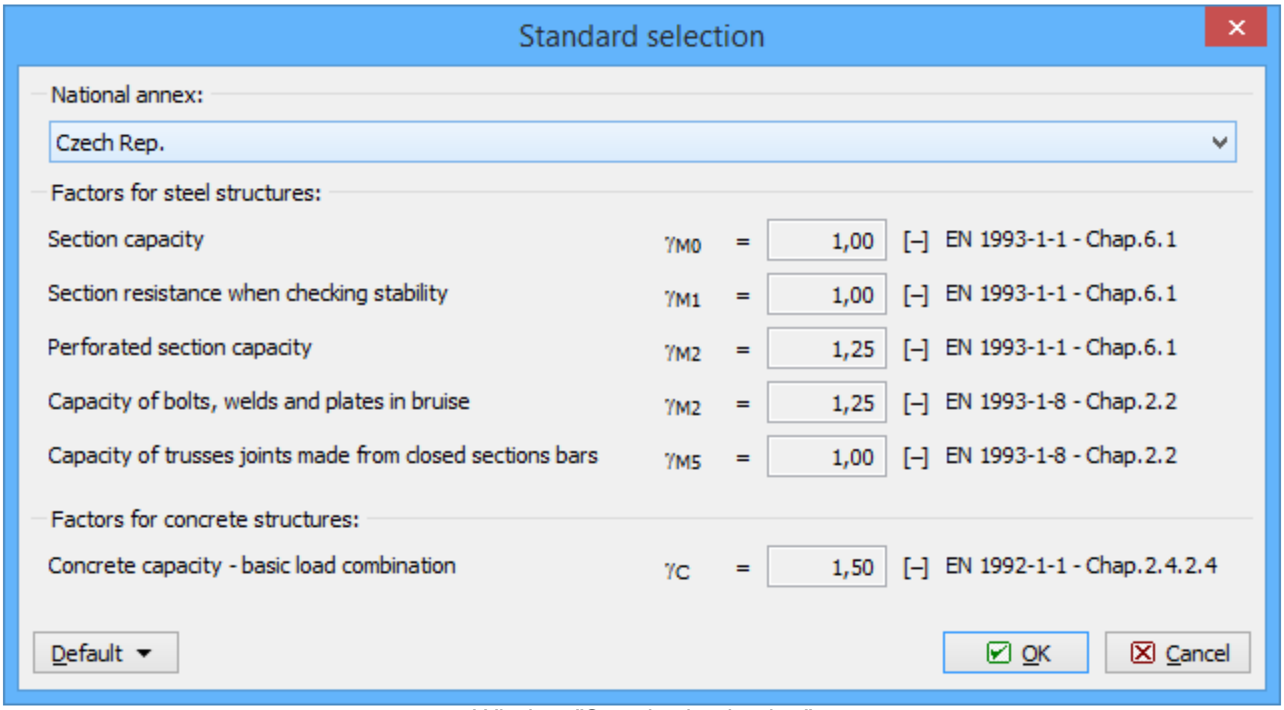
Partial factors are described in the theoretical part of help.

Button "**Default**" contains these two tools:

**Adopt default settings** • Set the default values for all parameters.

**Save settings as default** • Set entered parameters as defaults for new projects.



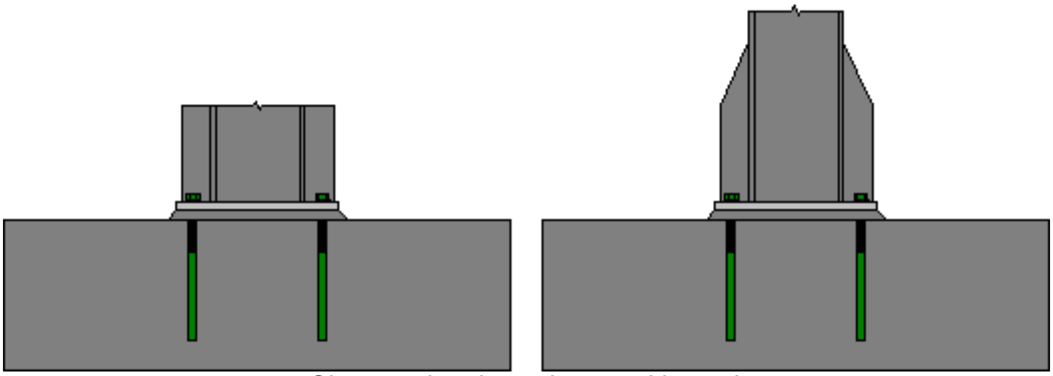

*Window "Standard selection"*

## Global settings

The appearance of the workspace and figures in documents can be changed in this window. The window contains these two tabs: "**View**" a "**Schemes**".

### View

This tab contains settings for changing the appearance of workspace and figures both in documents and clipboard (saving figure using *Ctrl+C* ). There is an option to change the font size in the workspace with the help of "**Text size**" setting. The items description and dimensions can be switched off using the setting "**Draw dimensions of structural items**". The setting "**Draw unshortened haunch**" influences the display of haunch in figures.



*Shortened and not shortened haunch*

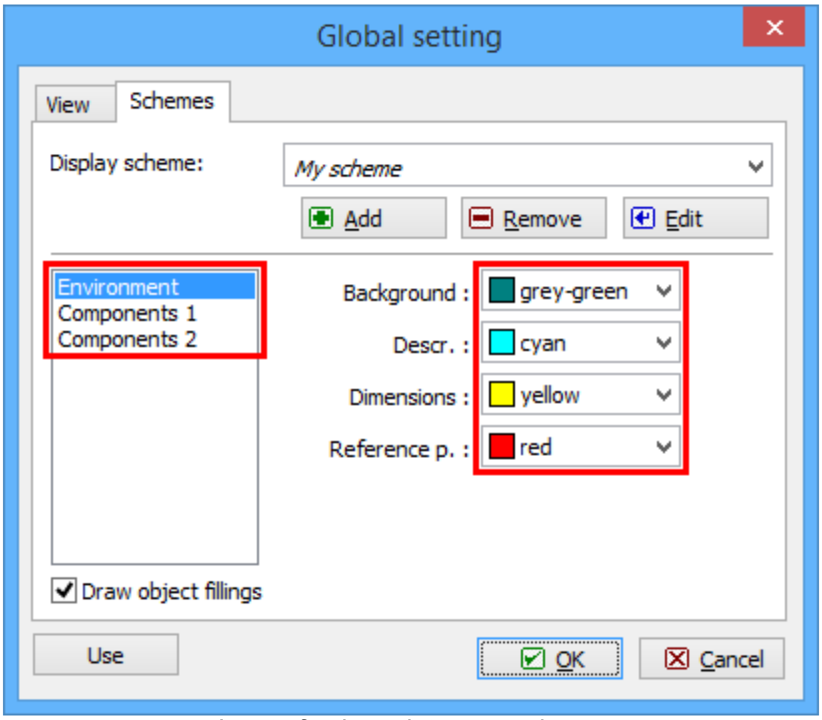
The selection of colour schemes changes the appearance of figures in the workspace, documents and clipboard. It means, that the connection in the workspace can be coloured, however, the figure copied using clipboard will be in black and white only. Few schemes are pre-defined, however it's possible to add new ones in the tab "**Schemes**".

Button "**Options**" launches the window with properties of figures copied into clipboard (size, borders etc.).

### Schemes

User defined colour schemes can be defined in this tab. These schemes can be used in the tab "**View**" for workspace, documents or figures in clipboard. Buttons "**Add**", "**Edit**" and "**Remove**" are available for the work with schemes. The colours for particular items can be specified in the bottom part of the window. Colours can be modified only for user defined schemes (highlighted by italics in the list of schemes). Pre-defined schemes can't be modified.

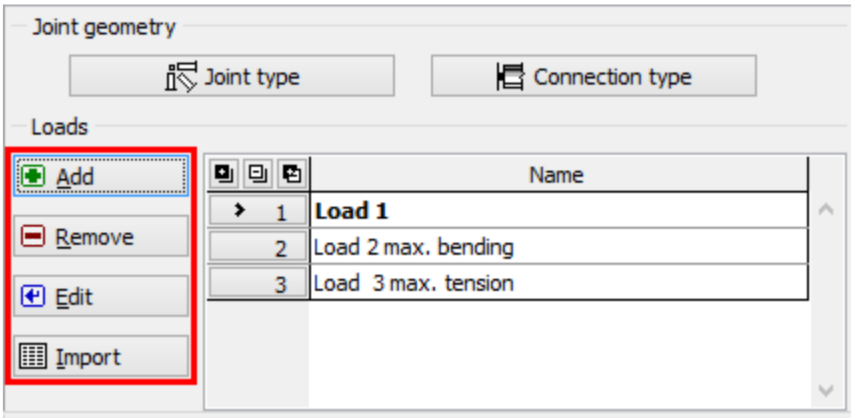
 




*Input of colours into new scheme*

## Joint

The main screen of the detail can be used for the input of loading list and for the change of detail type. Loading means the combination of design values of internal forces and moments, that acts in the structure in the same time. These loads are used for the verification of bearing capacity of the joint. This part contains only the input of loading list, the particular load values have to be specified in the part "**Loading**". The buttons "**Add**" and "**Remove**" in the toolbar on the left side of the loading table can be used for insertion or deletion of individual loadings. The properties of existing loading can be modified with the help of the button "**Edit**". The button "**Import**" can be used for the import of loading values from **.txt* or **.csv* files. The import settings can be specified in the window "**Load import**".



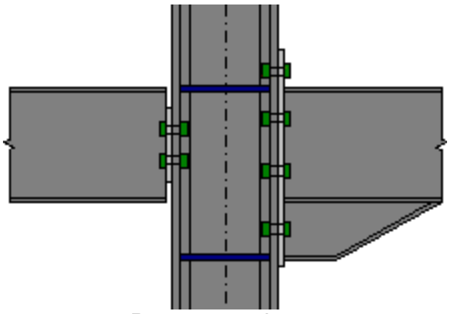

*Toolbar for the work with loads*

The button "**Joint type**" can be used for the change of joint type, the button "**Connection type**" for the specification of the number and type of particular connections in the joint. The change of joint type or connections automatically deletes all existing inputs.

The range of input differs according to the detail type:

## Beam to column joint



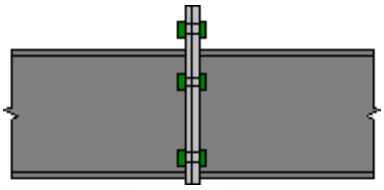
*Beam to column*

Beam to column joint consists of these parts:

- **Loading**
- **Column**
- **Stiffeners**

The input of particular connections follow.

## Beam splice



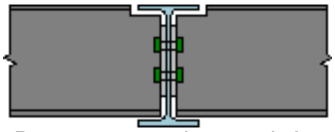
*Beam splice*

These parts are included in the beam splice detail:

- **End plate**
- **Bolts**

The input of particular connections follow. Both pinned and stiff end plates are allowed for this type of joint.

## Beam connection to girder



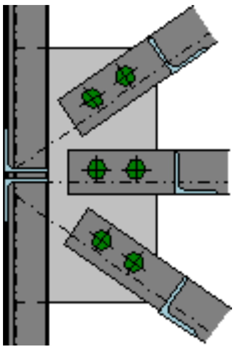
*Beam connection to girder*

Beam connection to girder contains these parts:

- **Primary beam**

The input of particular connections follow. Only pinned end plate and fin plate are allowed for this type of joint.

## Truss joint



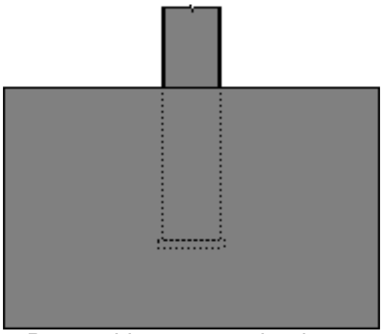
*Truss joint*

Following parts are included in the truss joint:

- **Loading** (only joints with RHS chord)
- **Truss load**
- **Chord**
- **Plate**
- **Members**
- **Results**

## Base with concreted column

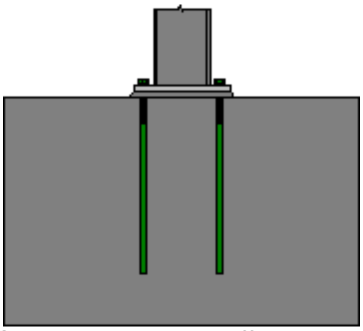


*Base with concreted column*

Base with concreted column consists of these parts:

- **Loading**
- **Column base**
- **Column position**
- **Column**
- **Results**

## Column base with stiff end plate



*Column base with stiff end plate*

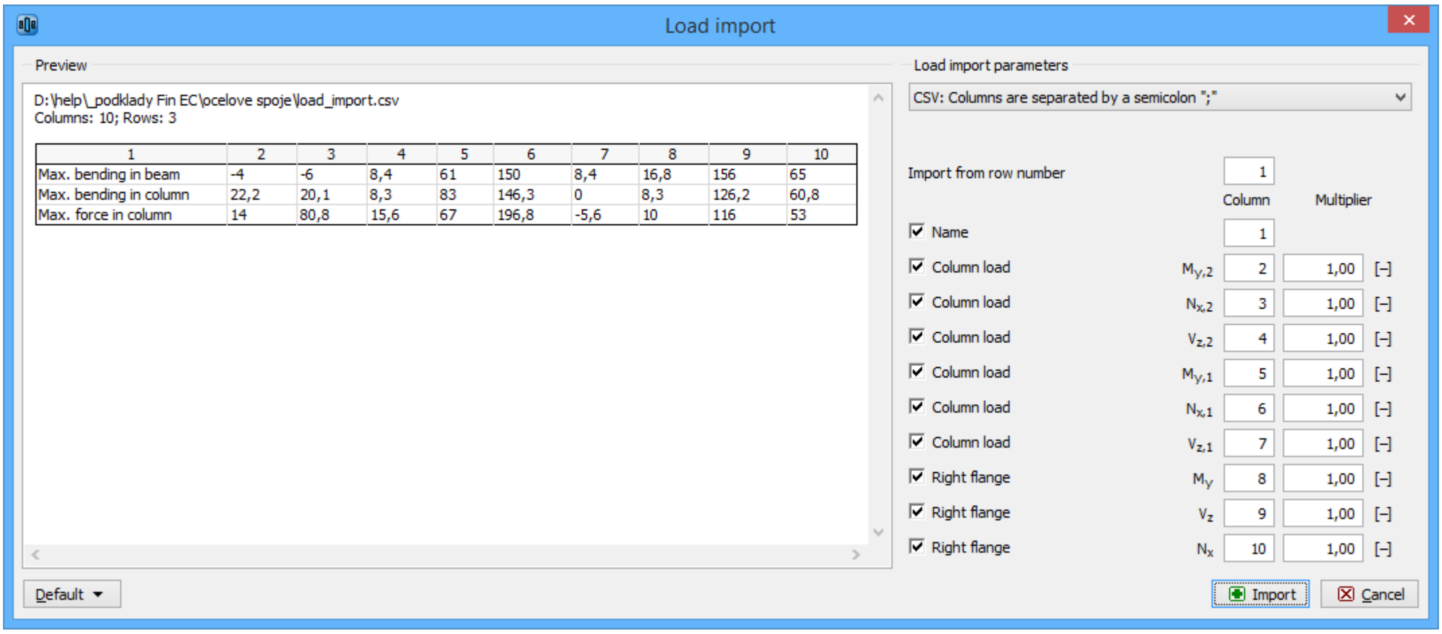
This detail contains following parts:

- **Loading**
- **Column base**
- **Column position**
- **Column**
- **Base plate**
- **Welds**
- **Bolts**
- **Results**

# Load import

Window "**Load import**" appears after loading the text or **.csv* source file. The data included in the source file can be arranged in this window. Left part shows content of imported file. The assignment of the certain column to the corresponding force can be done in the right part of the window. Numerical entries can be also multiplied by specified multiplier. This multiplier can be used mainly for conversion caused by different units in the source file and in the program. Initial rows can be skipped using setting "**Import from row number**". This setting is helpful for source files with headings. Default units required by the software are *[kN]* and *[kNm]*.

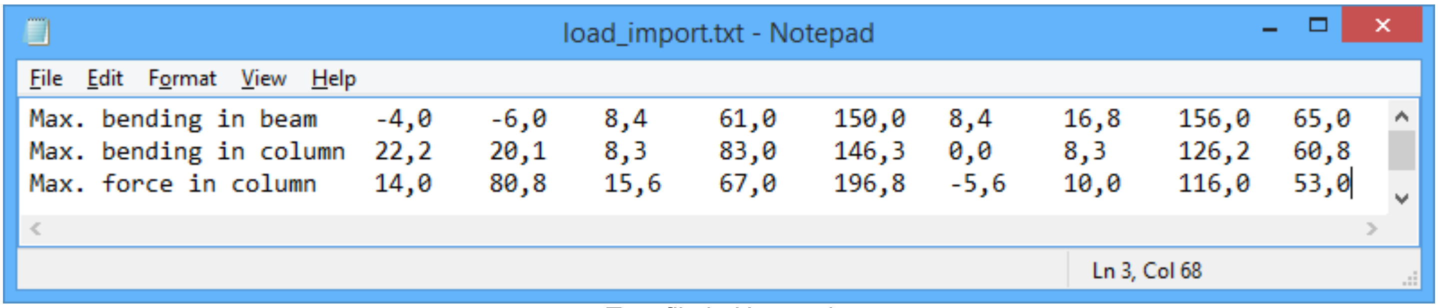


*Window "Load import"*

## Preparing text file

Text file can be created in any text editor (e.g. *Notepad*, *Word*, *Writer*). File format requires that every row contains one load case. Every row can contain all values of internal forces separated by space or tabulator. The order may differ comparing to the order of forces in the software, however, order has to be identical for all rows of the document. The load name may be also specified for every load.

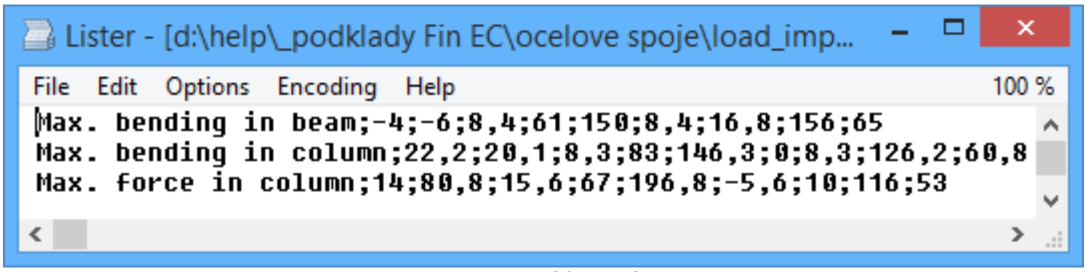
The file can be also created using part of analysis documentation from arbitrary structural analysis software.



*Text file in Notepad*

## Preparing *.csv file

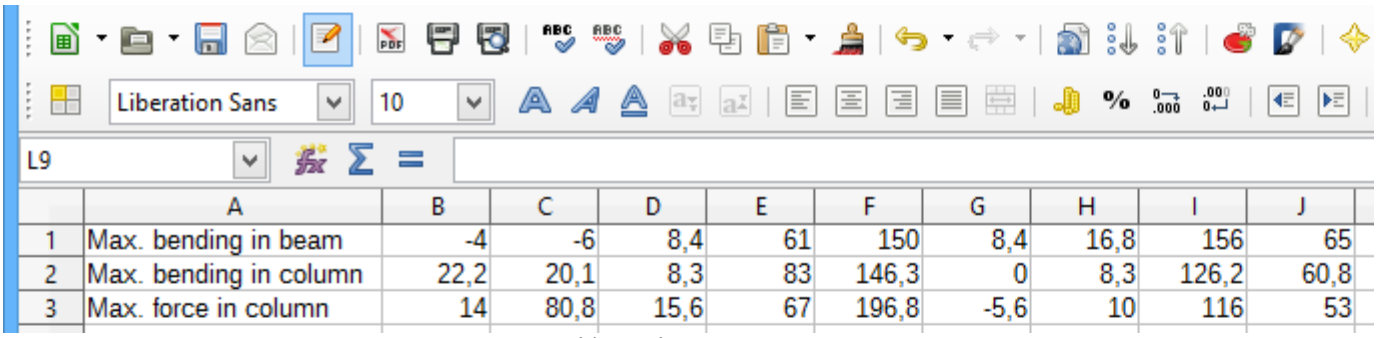
The rules for **.csv* (comma-separated values) files are almost identical . Main difference is, that the particular values are separated by semicolon ";".



*Example of *.csv file*

This file type can be easily created using spreadsheet programs like *Excel* or *Calc*. Created document can be saved as **.csv* file with appropriate separator.

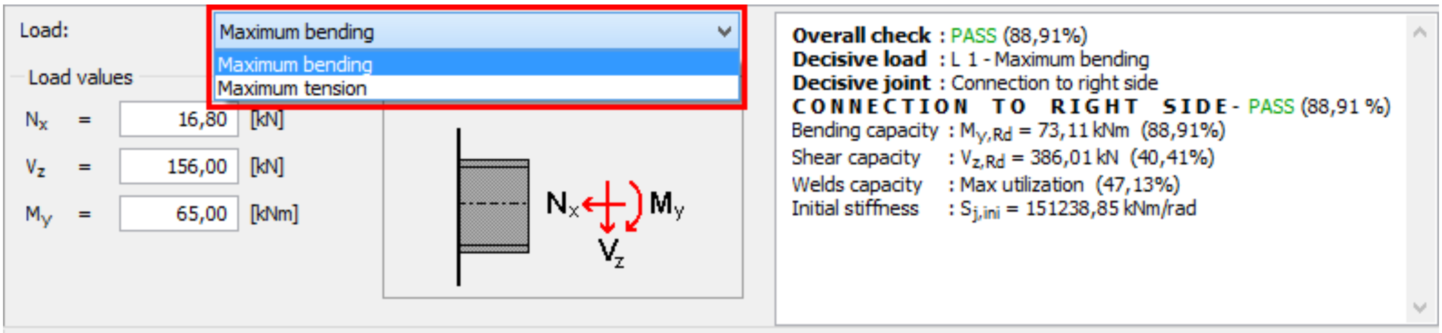


| | A | B | C | D | E | F | G | H | I | J |
|---|---|---|---|---|---|---|---|---|---|---|
| 1 | Max. bending in beam | -4 | -6 | 8,4 | 61 | 150 | 8,4 | 16,8 | 156 | 65 |
| 2 | Max. bending in column | 22,2 | 20,1 | 8,3 | 83 | 146,3 | 0 | 8,3 | 126,2 | 60,8 |
| 3 | Max. force in column | 14 | 80,8 | 15,6 | 67 | 196,8 | -5,6 | 10 | 116 | 53 |

*Edit of *.csv file in spreadsheet*

## Loading

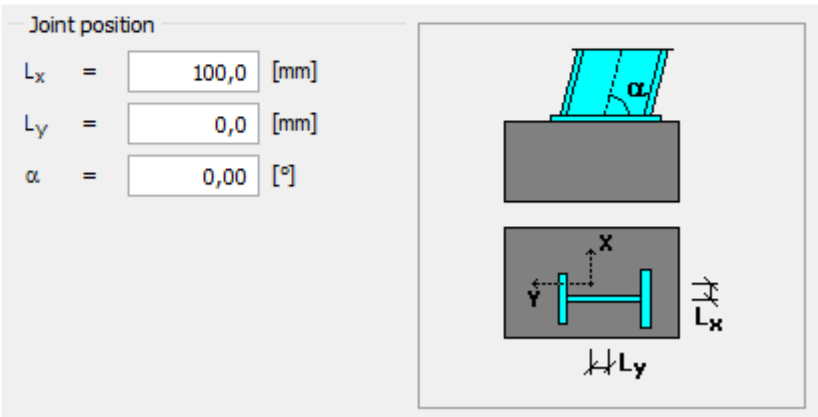
The forces for individual loads can be specified in this part. The active load has to be selected first in the upper part of the frame. List of loads can be specified in the main screen of the joint. Bottom part of the frame contains input of forces and moments. The orientations of positive values are shown in the scheme, which is placed in the middle of the input frame. The design values of forces and moments should be entered. The equilibrium of forces in joint is checked during the input.




*Choice of active load*

## Joint position

This part contains the inputs that specify the position (eccentricity) of connection relatively to the centre of joint or base. The marking of inputs corresponds to the detail scheme on the right side of the inputs.




*Joint position*

## Section

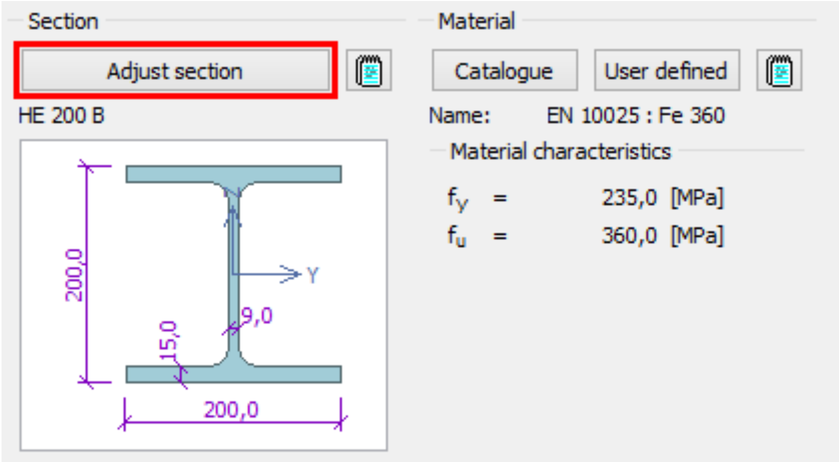
The cross-section and material of the chord can be specified in this part.

### Section

The cross-section of beam or column can be specified in the window "**Beam/column properties**". This window can be opened with the help of the button "**Adjust section**". This window also contains the input of haunches. The button "" shows the table with cross-sectional characteristics.

### Material

Chord material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program).




*The button for the edit of cross-section*

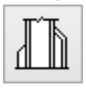
## Beam/column properties

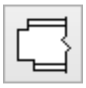
This window contains geometry of column/beam. The left part shows the cross-section at the member end, right part shows the side view. Properties can be changed using following buttons:

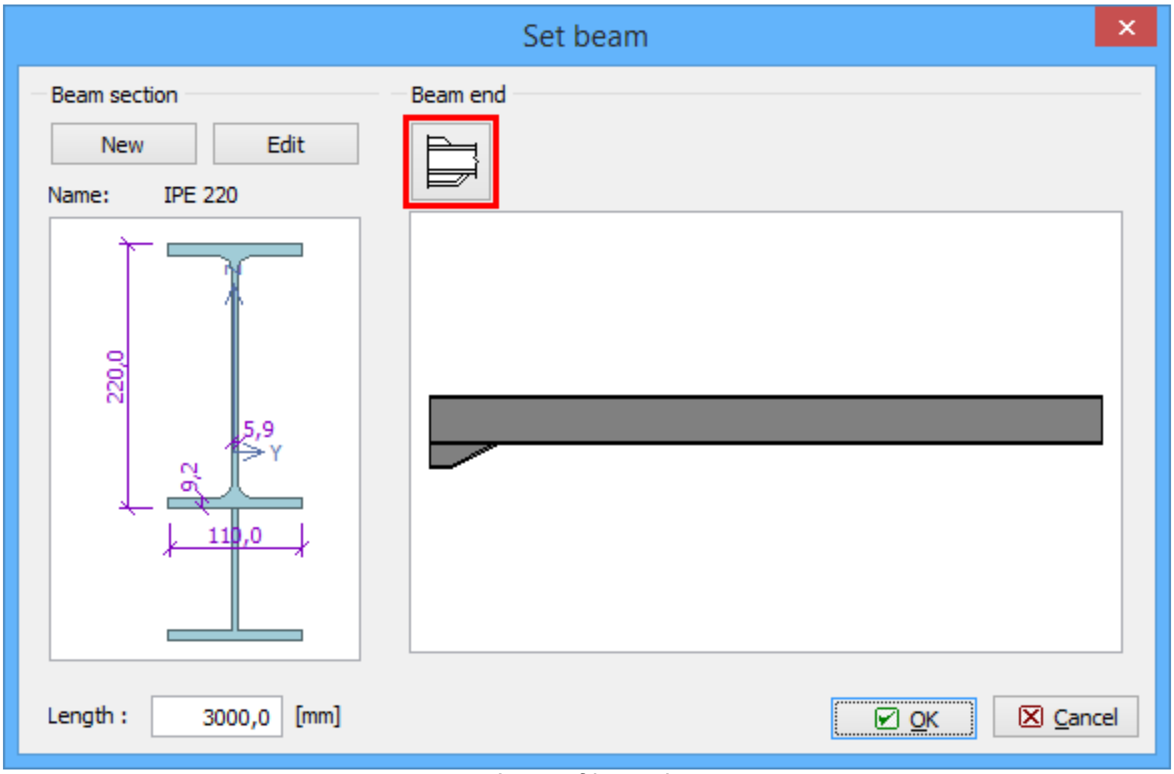
**New** • The button for the new input of cross-section. The section can be selected from pre-defined database of hot-rolled profiles or can be specified as welded profile with user defined dimensions. The input is done in the window "**Cross-section editor**".

**Edit** • The button for the edit of existing cross-section. It is possible either to change the hot-rolled profile or modify dimensions of welded profile. The input is done in the window "**Cross-section editor**".

**Length** • The input of member length


• The button for the input of haunches in the joint of column to base or in the beam connection. The haunches create the more stiff connection. The properties are organized in the window "**Haunches**". Available only for fixed connections.


• The button for the input of cuts at the end of pinned beam. These cuts can be used in cases, where the beam flange intersects with girder or column. The properties are organized in the window "**Beam end**". Available only for pinned connections.




*Input of haunch*

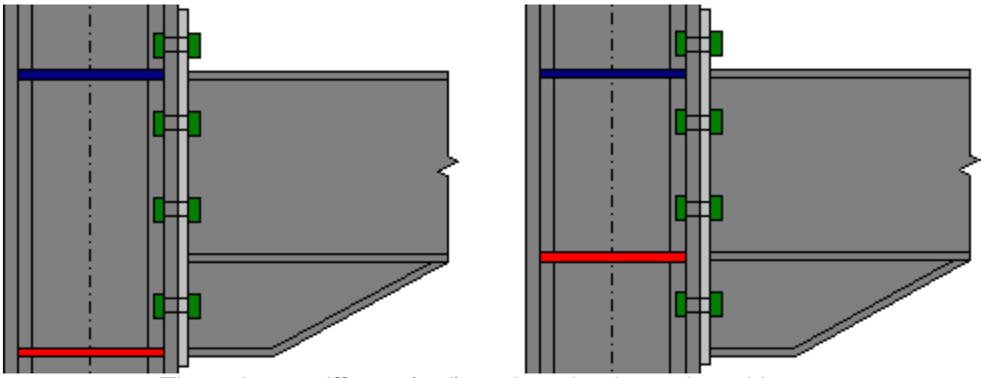
 

## Haunches

This window can be used for the input of haunches for beam or column. The haunches increase the stiffness of the connection. The input of haunches for top and bottom (or left and right) flanges is divided into two tabs. Haunches can be specified including the flange. Following dimensions have to be specified:

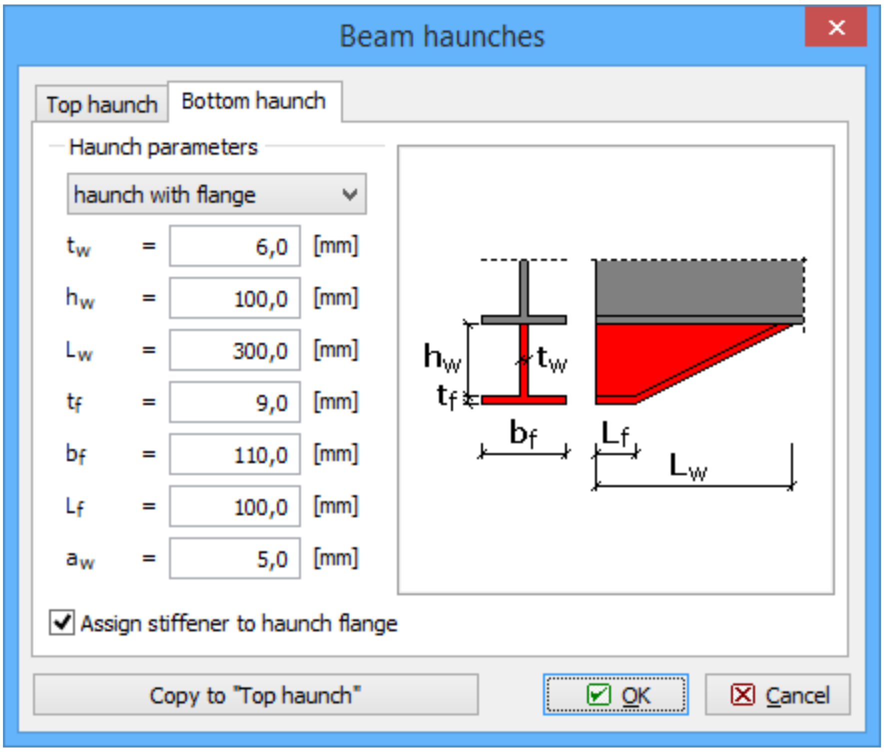
| | |
|---|---|
| $t_w$ | • The thickness of haunch web |
| $h_w$ | • The height of haunch |
| $L_w$ | • The total length of haunch |
| $t_f$ | • The thickness of haunch flange (only for haunch with flange) |
| $b_f$ | • The width of haunch flange (only for haunch with flange) |
| $L_f$ | • The length of straight flange |
| $a_w$ | • The weld thickness |

If the setting "**Assign stiffener to haunch flange**" is switched on, the corresponding column stiffener is automatically placed into the position of haunch flange. Otherwise, the stiffener remains in the position of beam flange.



*The column stiffener (red) assigned to haunch and beam*

The button "**Copy to...**" copies the haunch properties from one side of beam or column to another one.

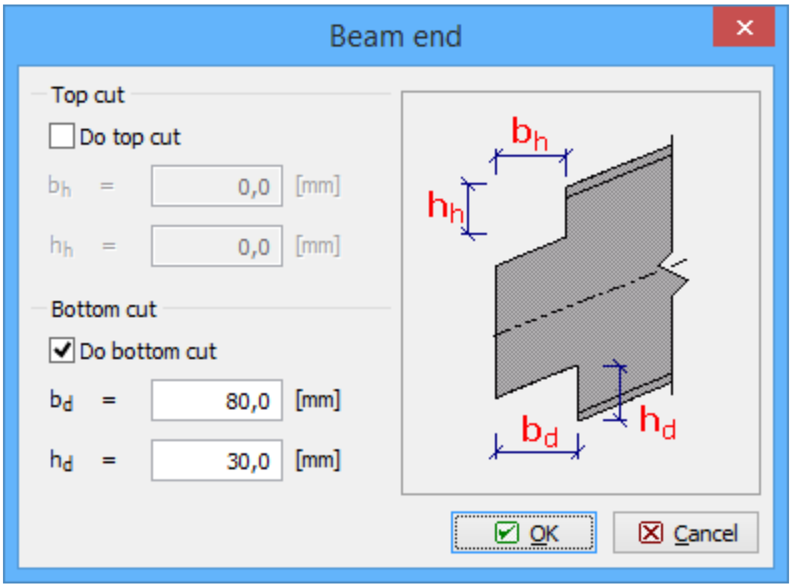



*Window "Beam haunches"*

## Beam end

This window can be used for the input of cuts at the end of pinned beam. These cuts can be used in cases, where the beam flange intersects with girder or column. The meaning of entered dimensions is described in the scheme in the right part of the window.

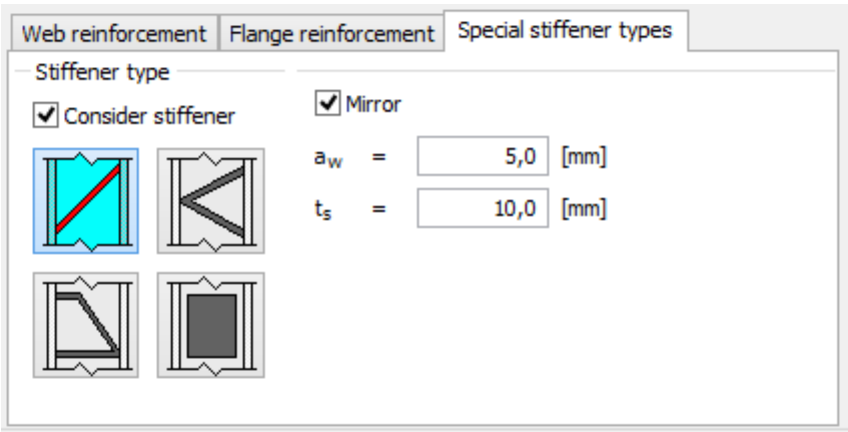



*Window "Beam end"*

## Stiffeners

The additional column stiffeners, which increase the bearing resistance in tension, compression and bending, can be specified in this part. The input is divided into three tabs according to the stiffener type:

- **Web stiffeners** - horizontal stiffeners increasing the bearing capacity of web in tension or compression
- **Flange stiffeners** - stiffeners increasing the bending capacity of column flange
- **Special stiffener types** - stiffeners, that increase the shear bearing capacity of column




*Input frame for stiffeners*

### Web stiffeners

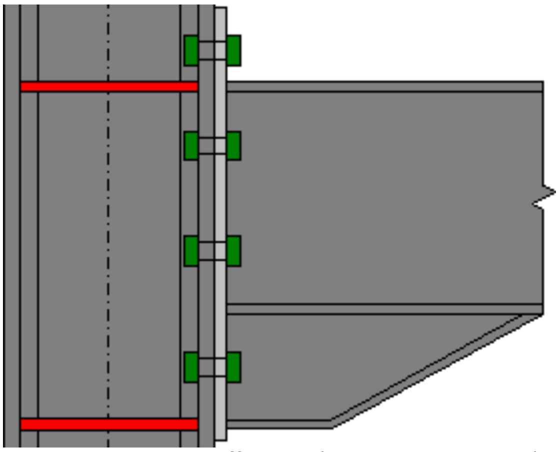
These stiffeners are horizontal plates in the levels of beam flanges. The stiffeners for top and bottom flange can be specified separately. The beams with haunch can be restrained either in the level of beam flange or in the level of the haunch flange (can be defined in the window "**Beam haunches**"). The following inputs have to be specified for stiffeners:

**$a_w$** • The weld thickness

**$t_s$** • The stiffener thickness

The theory is described in chapter "**Column web in tension and compression**".

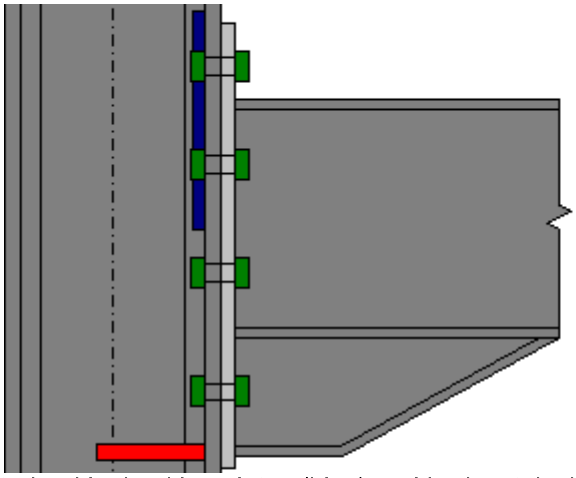


*Horizontal web stiffeners (highlighted by red)*

## Flange stiffeners

This tab contains properties of column flange stiffeners and backing plates. Stiffeners and backing plates can be specified in the window "**Column flange stiffener**", that can be launched by the button "**Add**". Stiffeners and plates are organized in the table. The buttons "**Edit**" and "**Remove**" are also available for the work with these objects.

The theory is described in chapters "**End plate or column flange in bending, bolts in tension**" and "**Column web in tension and compression**".



*Flange restrained by backing plates (blue) and horizontal stiffener (red)*

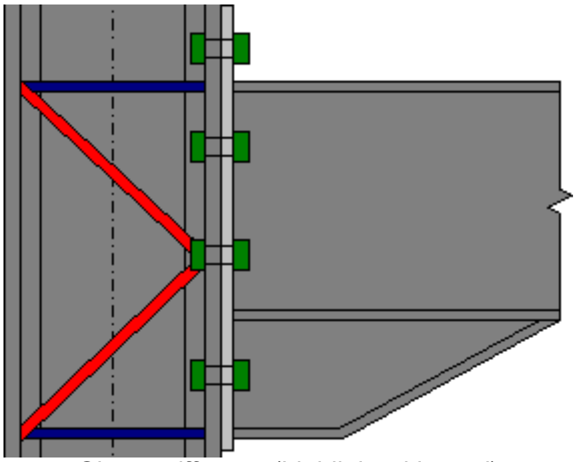
## Special stiffener types

The shear stiffeners of column web can be defined in this part. More types are available, it is also possible to use the additional web plates. Stiffeners are given by following dimensions:

| | | |
|---|---|---|
| $a_w$ | • | The weld thickness |
| $t_s$ | • | The plate thickness |
| $h_s$ | • | The distance from beam flange (only for Z-stiffener) |
| $a_1$ | • | The distance of top edge from reference level (only for web plates) |
| $h_s$ | • | The plate height (only for web plates) |

The theory is described in chapter "**Column web in shear**".

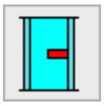
 



*Shear stiffeners (highlighted by red)*

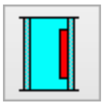
## Column flange stiffeners

This window contains properties of column flange stiffeners. Following stiffener types are available:



**Partial horizontal stiffener of column web**

- The horizontally placed stiffener, that is welded both to the column web and flange. The stiffener increases the column flange bearing capacity in bending. It may also increase the bearing resistance of column web in tension or compression.



**Stiffener of column flange**

- The flange stiffening of vertically placed plate. It increases the bearing resistance of column flange in bending.

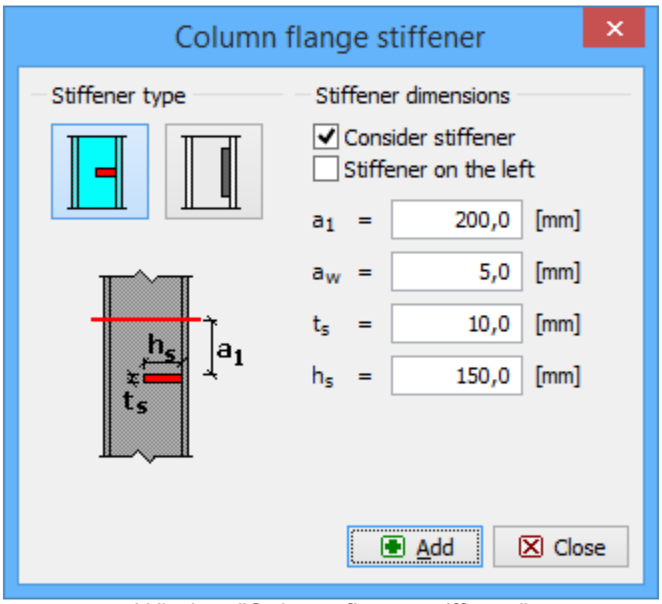
### Stiffener dimensions

Following dimensions have to be specified (the meaning of entered dimensions is described in the scheme in the left part of the window):

**$a_1$** • The distance of top edge from reference level
**$a_w$** • The weld thickness
**$t_s$** • The stiffener thickness
**$h_s$** • The stiffener length

The stiffener can be temporarily switched off with the help of the button "**Consider stiffener**". The stiffener dimensions remain in the window, however, the stiffener is not considered in the analysis. This feature can be used for fast review of bearing capacity with and without stiffener. The setting "**Stiffener on the left**" mirrors the stiffener on the left column flange.

The theory is described in chapters "**End plate or column flange in bending, bolts in tension**" and "**Column web in tension and compression**".




*Window "Column flange stiffener"*

# End/Fin plate

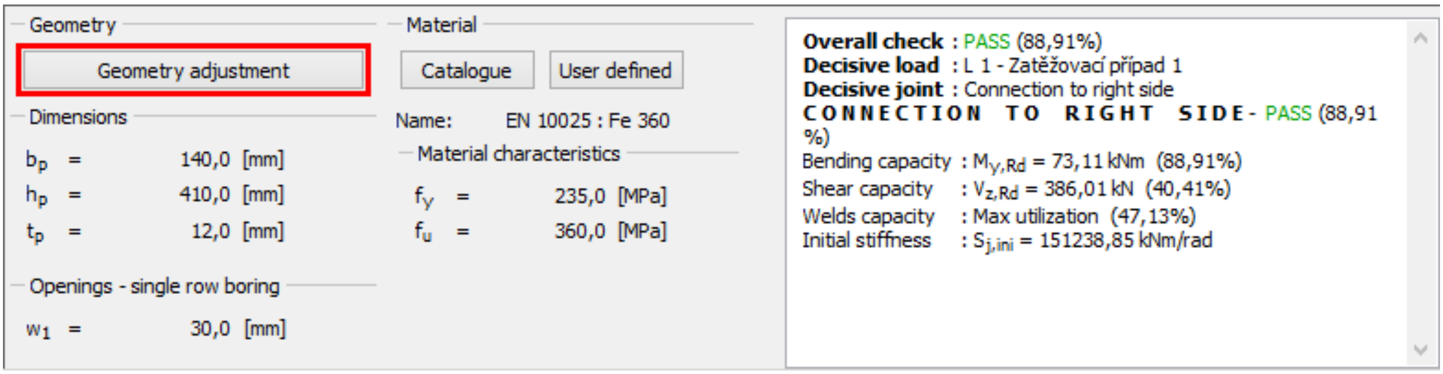
This part contains properties of end plate or fin plate.

## Geometry

The button "**Geometry adjustment**" opens the window "**Edit end plate**" or "**Edit fin plate**". These windows contains properties of plates like dimensions or positions of bolts.

## Material

Plate material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program).




*The button for the edit of plate geometry*

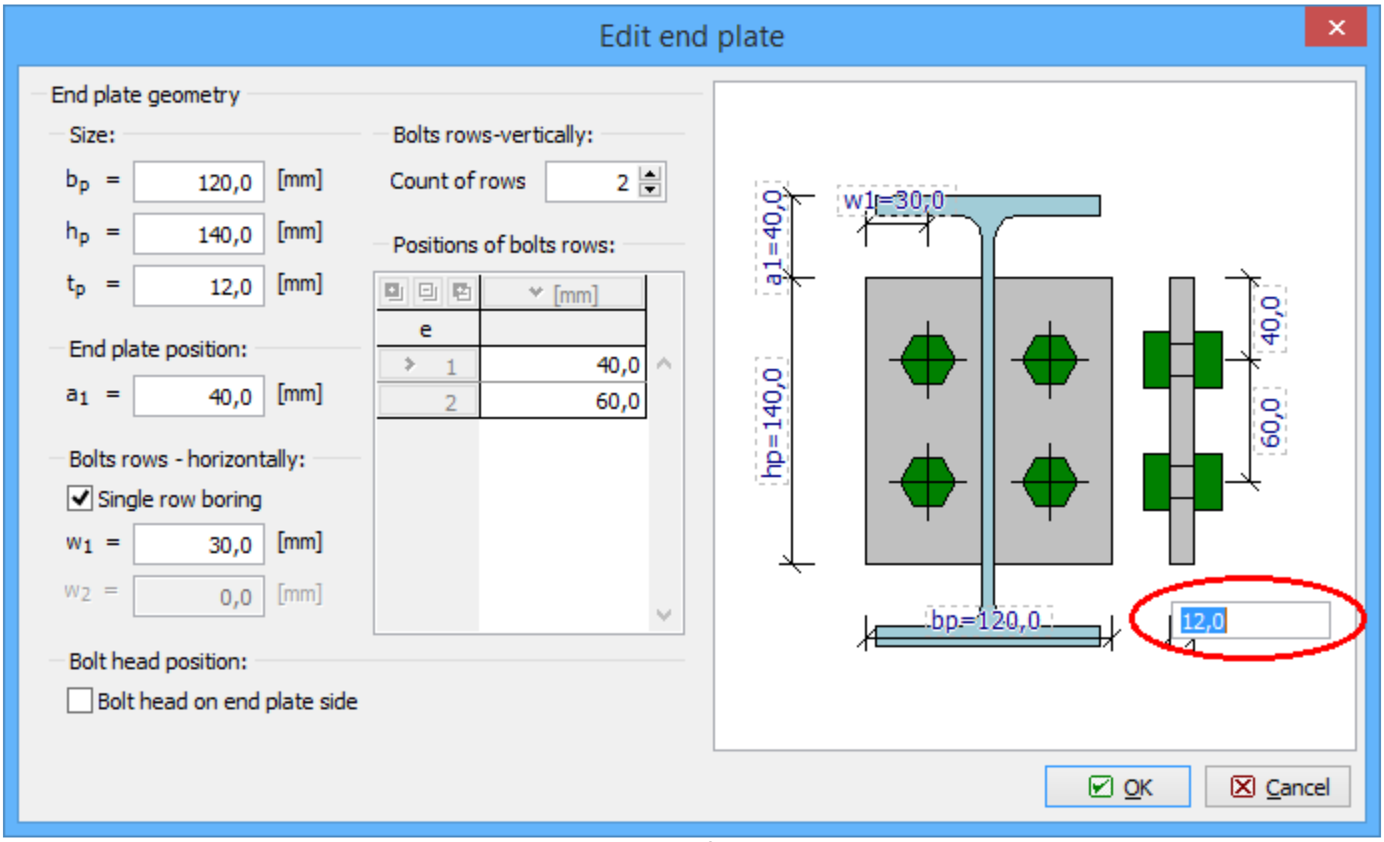
# Edit end plate

This window contains properties of end plate including dimensions and bolts count. Following values can be specified:

| | |
|---|---|
| **$b_p$** | • The end plate width |
| **$h_p$** | • The end plate height |
| **$t_p$** | • The plate thickness |
| **$a_1$** | • The position relatively to the top flange edge |
| **Single row** | • If this setting is switched off, it is possible to add one more vertical row of bolts. Available only for pinned connections. |
| **$w_1$, $w_2$** | • The distances of columns measured from the vertical edge of plate |
| **Count of rows** | • The number of horizontal rows of bolts |
| **Positions of bolts rows** | • The distance of first bolt centre from top plate edge and vertical distances of following rows |
| **Bolt head on end plate side** | • This setting affects the orientation of bolts. Available only for connections of end plates to column. |

The parameters can be also changed using the active dimensions in the detail layout:




*Input with the help of active dimension*

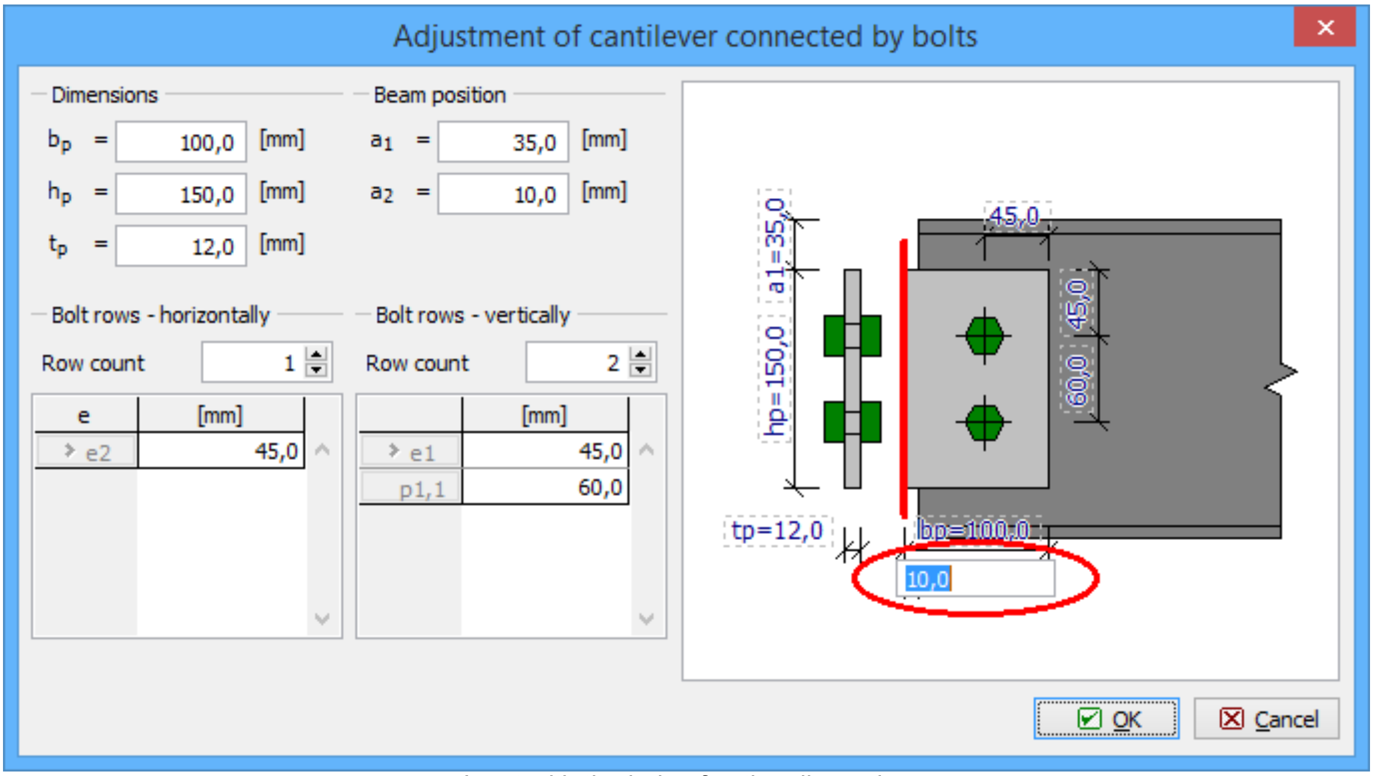
## Edit fin plate

This window contains properties of fin plate including dimensions and bolts count. Following values can be specified:

- **$b_p$** – The end plate width
- **$h_p$** – The end plate height
- **$t_p$** – The plate thickness
- **$a_1$** – The position relatively to the top flange edge
- **$a_2$** – The distance between beam end and girder/column edge
- **Count of rows horizontally** – The number of vertical rows of bolts
- **Count of rows vertically** – The number of horizontal rows of bolts
- **$e_1$, $p_1$** – The distance of first bolt centre from top plate edge and vertical distances of following rows
- **$e_2$, $p_2$** – The distance of first bolt centre from vertical plate edge and horizontal distance of following row

The parameters can be also changed using the active dimensions in the detail layout:




*Input with the help of active dimension*

## Bolts

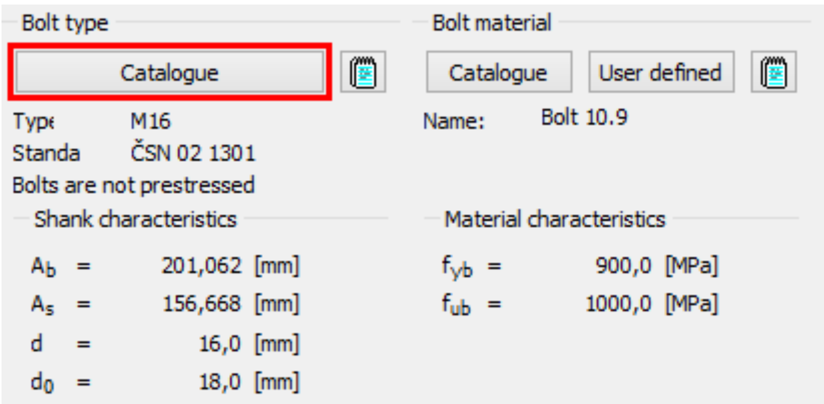
The bolts properties can be specified in this part.

### Bolt type

The window "**Catalogue**" opens the window "**Bolts catalogue**", where the bolts properties (type, dimensions) can be specified.

### Bolt material

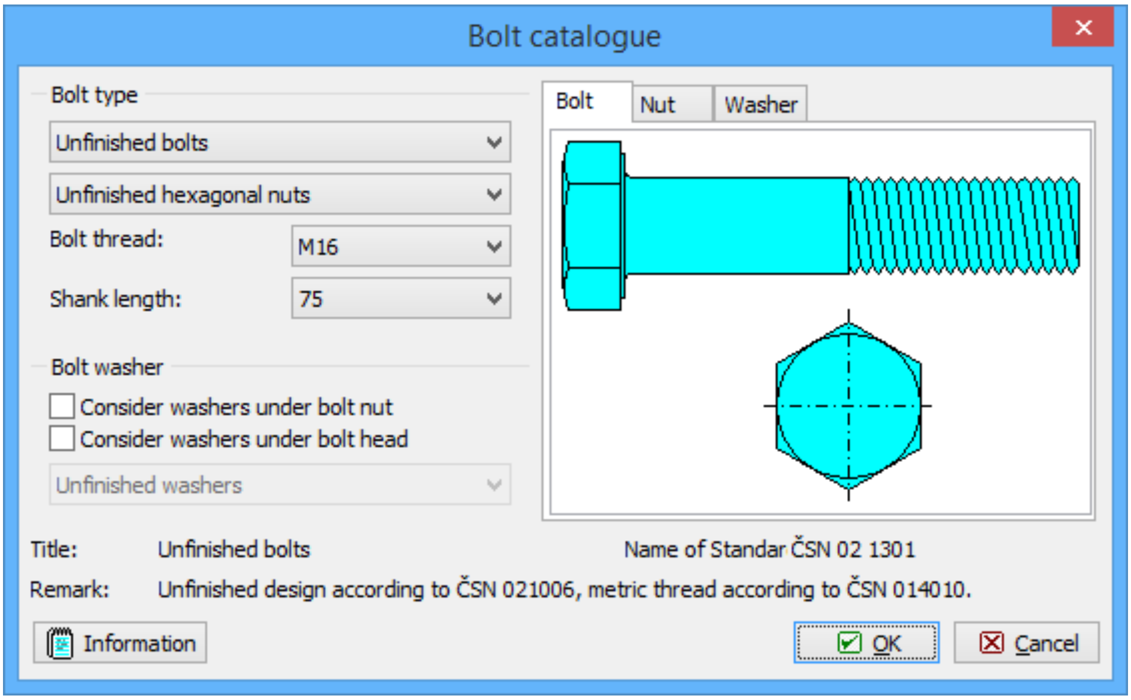
Bolt material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program).




*The button for launching bolts properties*

## Bolt catalogue

This window contains the properties of bolts, nuts and washers. Right part of the window shows the schemes of bolt, nut and washer, schemes are organized into particular tabs. Complete geometric characteristics can be displayed with the help of the button "**Information**".




*Window "Bolts catalogue"*

## Welds

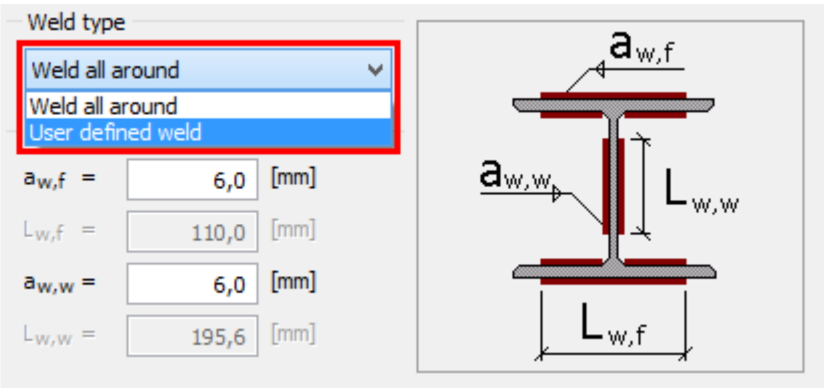
This part contains properties of welds, that are used for connection of beam or column to end plate or column. Two weld types are available:

| | |
|---|---|
| **Weld all around** | • The welds are defined only by the thickness. Welding lengths are given by the cross-sectional dimensions. |
| **User defined weld** | • The welding lengths for flanges and web can be specified for this type. The program automatically checks, whether the lengths are within the limits given by the cross-sectional dimensions. |

Following dimensions of welds can be specified:

| | |
|---|---|
| $\mathbf{a_{w,f}}$ | • The welds thickness for flanges |
| $\mathbf{L_{w,f}}$ | • The welds length for flanges |
| $\mathbf{a_{w,w}}$ | • The welds thickness for web |
| $\mathbf{L_{w,w}}$ | • The welds length for web |

The marking of inputs corresponds to the scheme on the right side of the inputs.




*Choice of weld type*

## Results

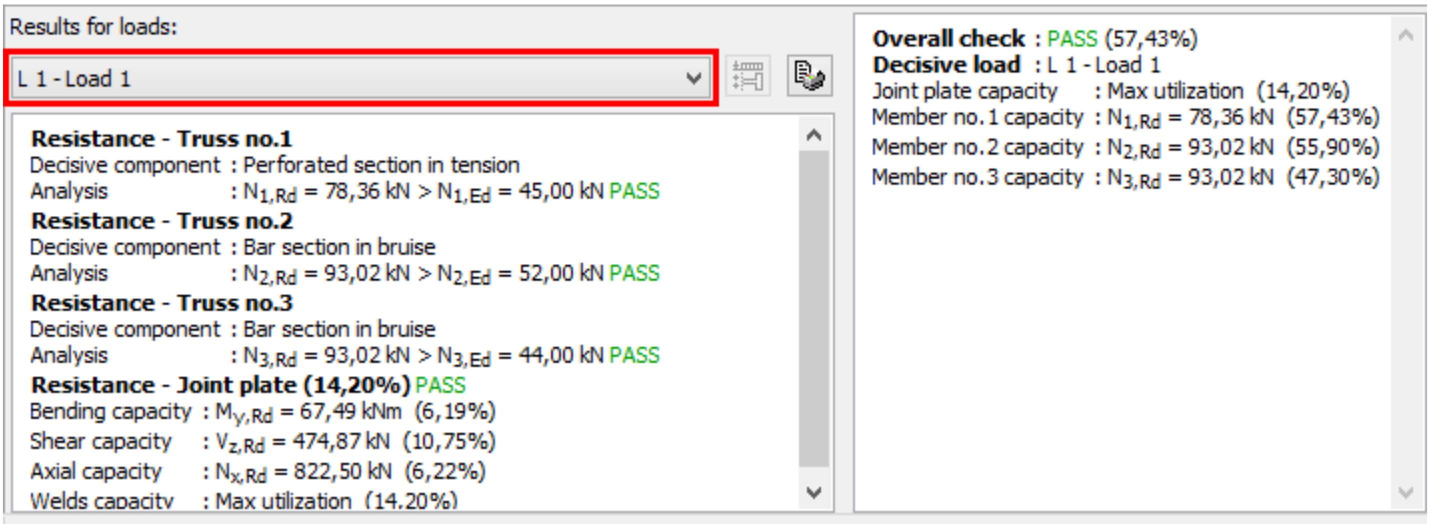
The left part of the input frame shows the detailed results for specified load. The needed load has to be selected in the list, that is placed above the results view. The list also contains the item "**Null load**". This option shows ultimate bearing capacity of the detail, without any verification of specified load. This frame contains two buttons for printing the detailed results:

 • Shows the forces distribution in the contact joint

- Prints the detailed analysis report for selected load

Right part shows the results of overall analysis for complete connection. As the analysis contains also checks of geometry and analysis presumptions, the bottom part may contain the list of warnings and errors. The errors are considered as significant problems and the software is not able to calculate results.




*Choice of load for results display*

## Chord

The cross-section and material of the chord can be specified in this part.

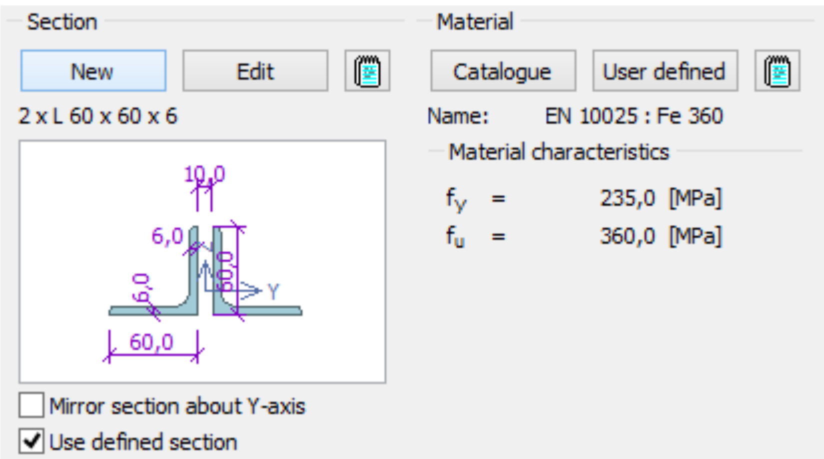
### Section

| | |
|---|---|
| **New** | • This button launches the window "**Cross-section editor**" that contains pre-defined database of hot-rolled and welded cross-sections. |
| **Edit** | • The button for the editing of existing cross-section. It is possible to change profile type or dimensions of welded cross-section in the window "**Cross-section editor**". |
| **Mirror section about Y-axis** | • The setting that changes the orientation of the chord cross-section. This option is not available for RHS cross-sections. |
| **Consider chord** | • The setting that removes chord from the joint design. The analysis is done only for joint plate and connected members. This option is not available for RHS cross-sections. |

The button "" shows the table with cross-sectional characteristics.

### Material

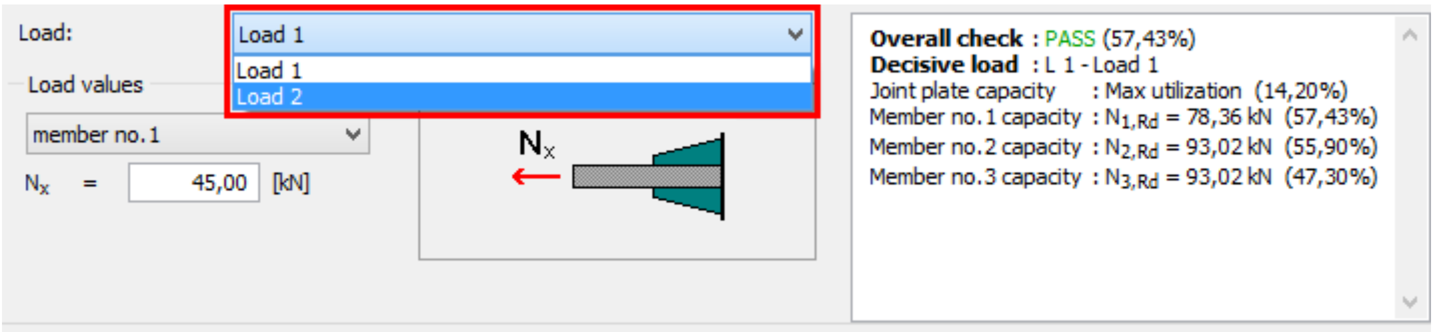
Chord material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program).




*Input frame "Chord"*

## Truss load

The member forces in individual loads can be specified in this part. The active load has to be selected first in the upper part of the frame. List of loads can be specified in the main screen of the joint. Bottom part of the frame contains input of axial force for individual members. The corresponding member has to be selected in the list box. The design values should be entered.




*Choice of active load*

# Plate

This part contains properties of joint plate.

## Geometry

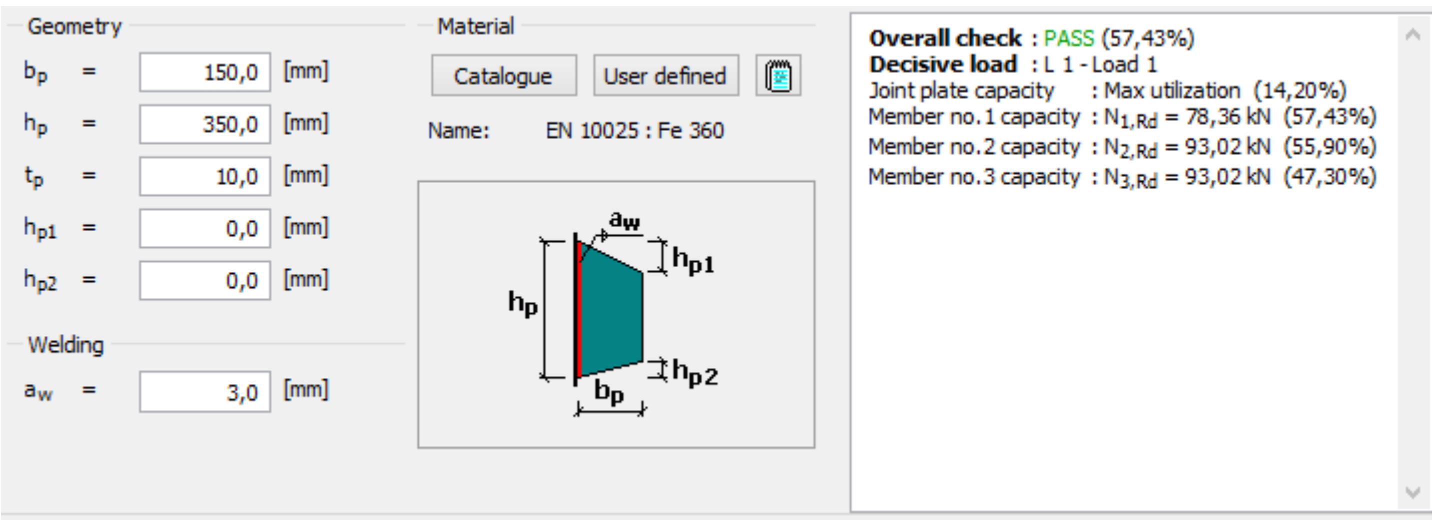
Following dimensions are included:

- $\mathbf{b_p}$ The plate width
- $\mathbf{h_p}$ The plate height
- $\mathbf{t_p}$ The plate thickness
- $\mathbf{h_{p1}}$ The vertical difference of upper edge
- $\mathbf{h_{p2}}$ The vertical difference of bottom edge
- $\mathbf{a_w}$ The weld thickness

The dimensions corresponds to the detail scheme on the right side of the inputs.

## Material

Plate material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program).

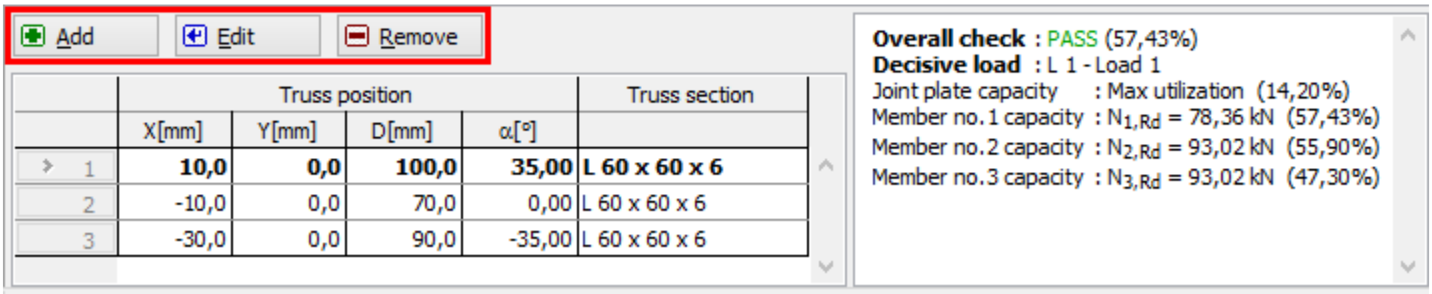



*Input frame "Plate"*

# Members

The members connected to the joint can be specified in this part. The input contains exact position, cross-section and material and connection type. New member can be added with the help of the button "**Add**". The member properties are organized in the window "**Member properties**". Existing members are stored in the table. The toolbar above the table also contains buttons for editing or removing members.




*Toolbar for the work with members*

## Member properties

This window contains properties of member connected to the truss joint. The properties are organized into two tabs:

### Member

This part contains inputs of geometry and material. The member cross-section can be specified in the window "**Cross-section editor**", that can be launched by the button "**New**". The cross-sectional dimensions can be modified quickly with the help of the button "**Edit**". The button "" is able to load cross-sectional properties from existing member. The button "" shows the table with cross-sectional characteristics.

The material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program). The button "" is able to load material from existing member. The button "" shows the table with material characteristics.

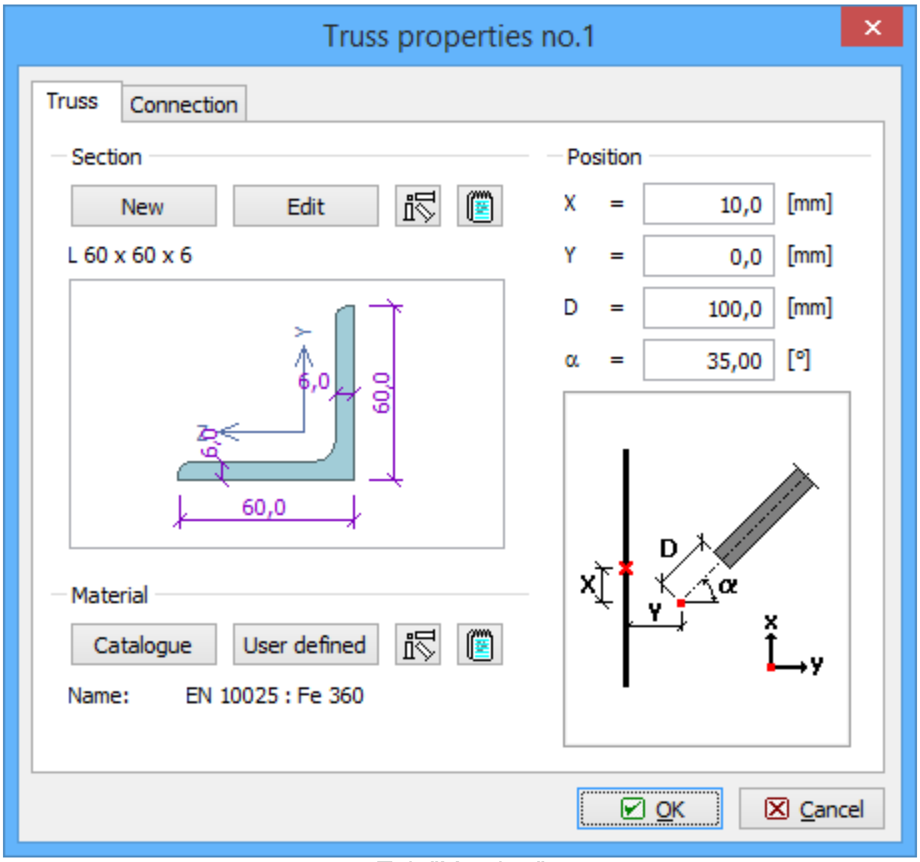
The member position is given by following dimensions:

**X** • The position of member insertion point measured along the chord length

**Y** • The distance of the member insertion point measured from the chord edge

**D** • The distance of the member beginning measured from the member insertion point

**α** • The member rotation




*Tab "Member"*

## Connection

The members can be connected in these ways:

- **Bolted connection**
- **Welded connection**

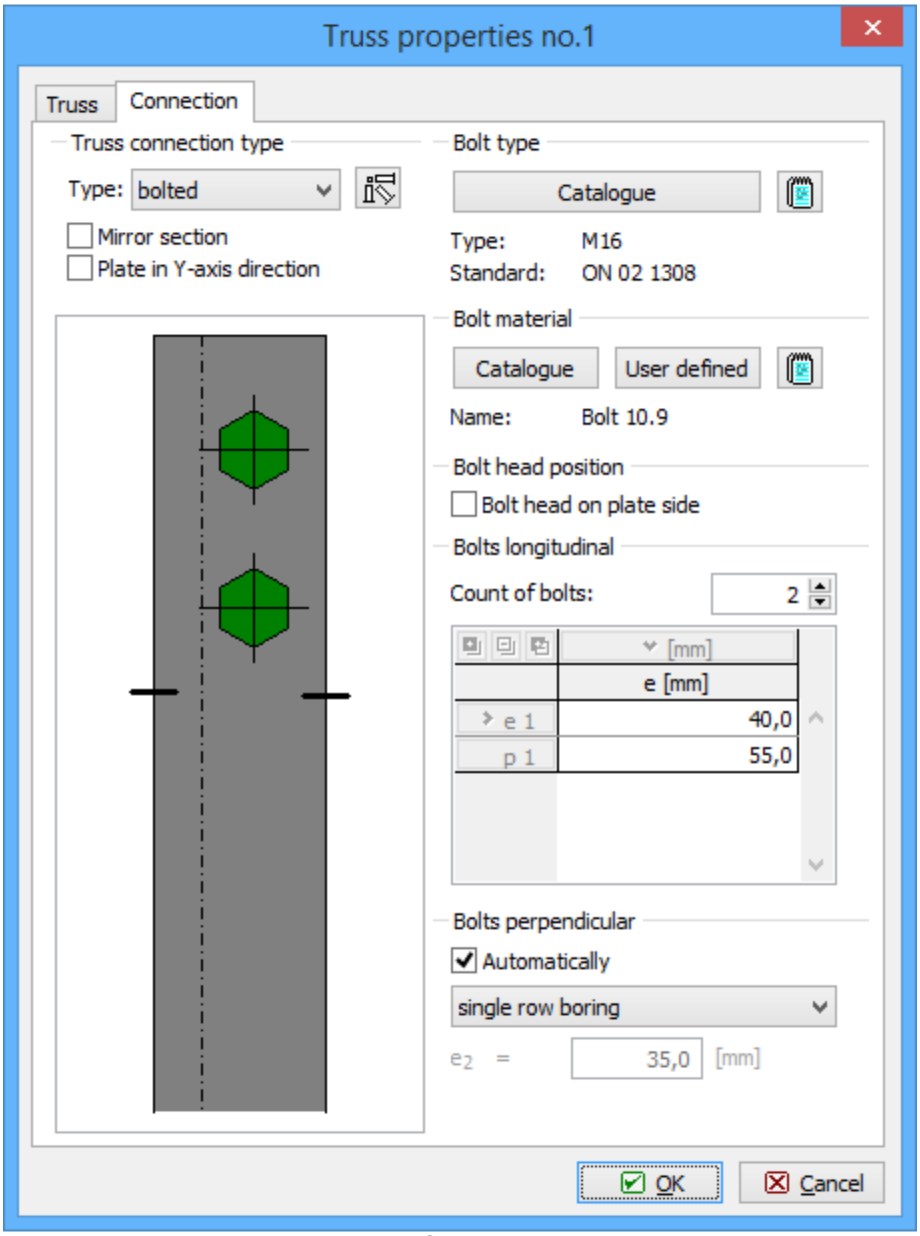
This choice can be done with the help of the setting "**Type**". The button "" is able to load properties of connection from existing member. The setting "**Mirror section**" changes the position of protruding part of the cross-section. The conflict of two members can be solved with the help of the setting "**Plate in Y-axis direction**", that moves the member to opposite surface of the joint plate.

The bolt properties have to be specified for **bolted** connection. The type and size of bolt has to be selected from pre-defined database, that can be opened with the help of the button "**Catalogue**". Bolt material can be selected from database (the button "**Catalogue**") or can be specified numerically with the help of yield and ultimate tensile strengths (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints in the default screen of the program.

The rows count can be also specified. The distances between bolts can be calculated automatically or entered by the user.

Following dimensions are available for **welded** connections:

| | |
|---|---|
| $\mathbf{a_{w,p1}}$ | • The thickness of weld close to the protruding arm of the cross-section |
| $\mathbf{L_{w,p1}}$ | • The length of weld close to the protruding arm of the cross-section |
| $\mathbf{a_{w,p2}}$ | • The thickness of weld close to the adjacent arm of the cross-section |
| $\mathbf{L_{w,p2}}$ | • The length of weld close to the adjacent arm of the cross-section |
| $\mathbf{a_{w,v}}$ | • The thickness of ending weld |
| $\mathbf{L_{w,v}}$ | • The length of ending weld |



*Tab "Connection"*

## Column base

This part contains properties of base and grouting.

### Geometry

Following dimensions are included:

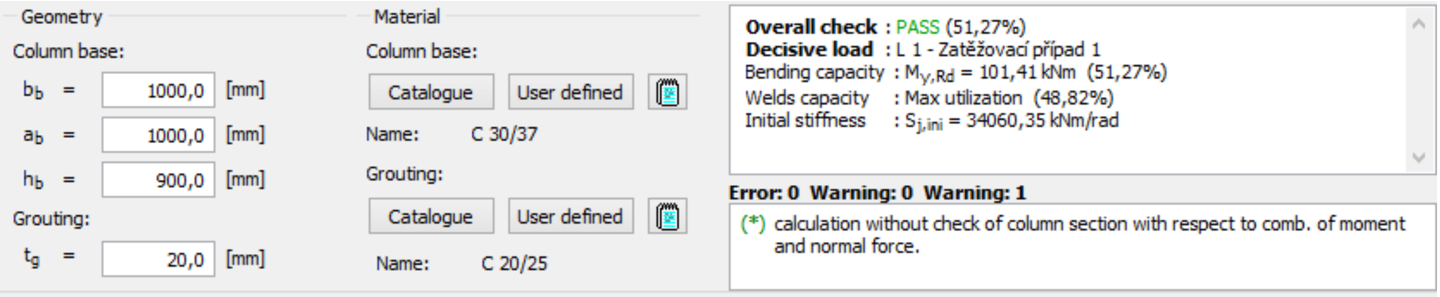
| | | |
|---|---|---|
| $\mathbf{b_b}$ | • | The dimension *b* of base |
| $\mathbf{a_b}$ | • | The dimension *a* of base |
| $\mathbf{h_b}$ | • | The base height |
| $\mathbf{t_g}$ | • | The grouting thickness (between end plate and concrete base) |

The detail view in the workspace updates automatically after the change of dimensions.

### Material

Base and grouting material can be selected from pre-defined database (the button "**Catalogue**") or can be specified numerically with the help of characteristic value of compressive strength $f_{ck}$ (the button "**User defined**"). These buttons aren't enabled, if the global material is assigned to all joints (in the default screen of the program).

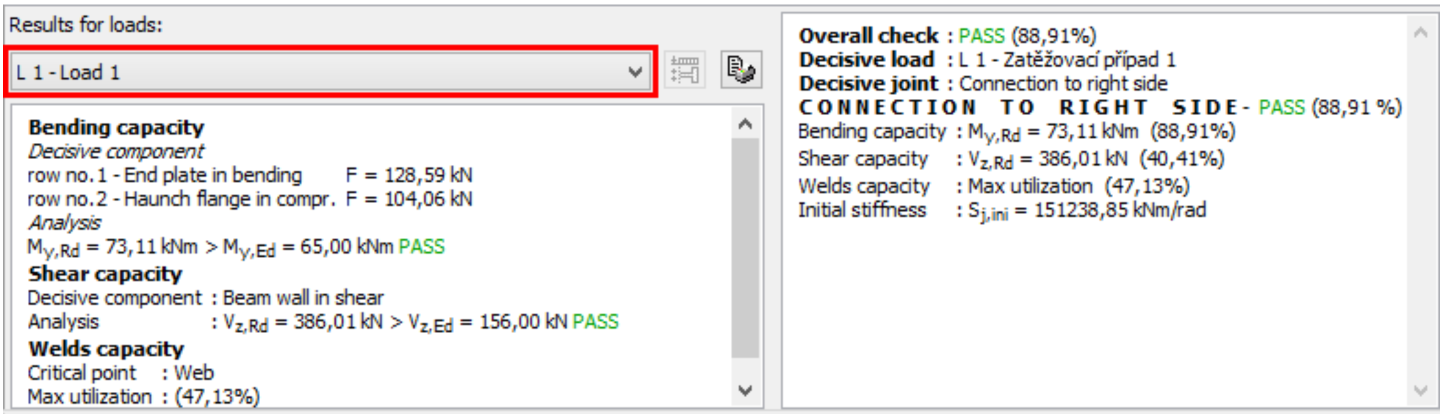



*Input frame "Column base"*

# Connection

This part displays detailed results for the connection and selected load. The load can be selected in the list box in the left upper corner of the bottom frame. The program also automatically creates the load "**Zero load**". This option contains the information regarding the maximum bearing capacity. The detailed results can be printed with the help of the button "".

Right part of the frame shows the overall results for the joint. This part also contains the results of geometric controls and other checks. Warnings are divided into three categories according to the seriousness. The most serious warnings (errors) cause termination of the analysis.




*Load choice for results display*

Following connection types are available in the program:

## Fin plate

Fin plate detail contains these parts:

- **Loading**
- **Joint position**
- **Beam**
- **Welds**
- **Fin plate**
- **Bolts**

## Hinged end plate

This detail contains following parts:

- **Loading**
- **Joint position**
- **Beam**
- **Welds**
- **End plate**
- **Bolts**

## Stiff end plate

The range of inputs is identical to the hinged end plate

## Welded connection

Welded connection contains following parts:

- **Loading**
- **Joint position**
- **Beam**
- **Welds**

### Webs to joint plate

Truss connection consists of these parts:

- **Loading**
- **Joint position**
- **Plate**
- **Members**

## Program Timber

The software "**Timber**" verifies timber elements according to EN 1995-1-1.

### User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Input**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Three types of particular tasks can be used in the software:

| | |
|---|---|
| **Section** | • Fast analysis of timber cross-section with unlimited number of loads. |
| **Member** | • Analysis of the whole member with specified diagrams of internal forces. This type is suitable for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**". |
| **Beam** | • Comprehensive analysis of horizontal beam with unlimited number of supports. Both ultimate and serviceability limit states are considered during analysis. |

These tasks can be added with the help of the buttons "**Add section**", "**Add member**" and "**Add beam**" in the heading of the tree menu.

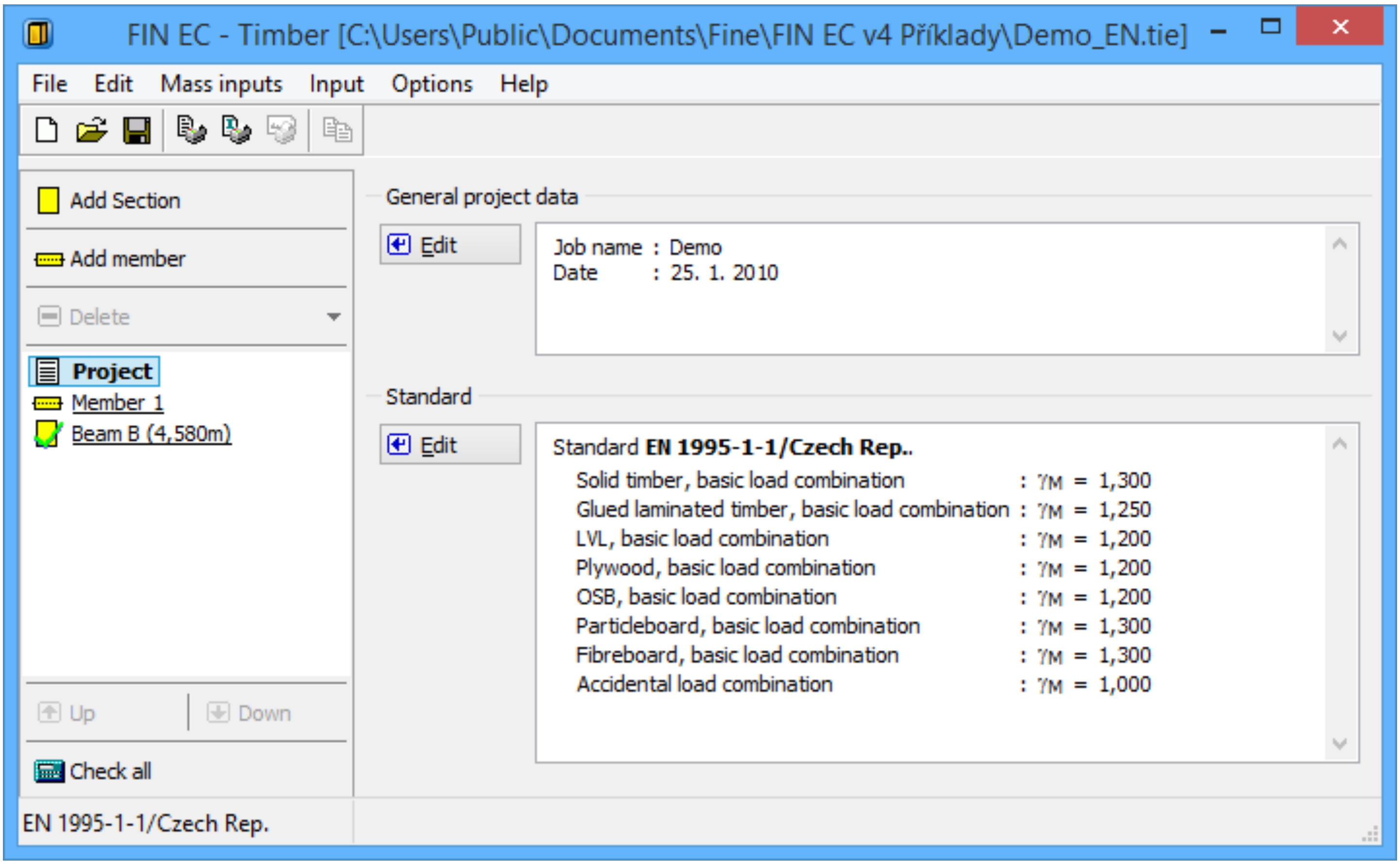
### Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

Analysis and verifications used in the software are described in the theoretical part of the help.




*Main application window*

## Standard selection

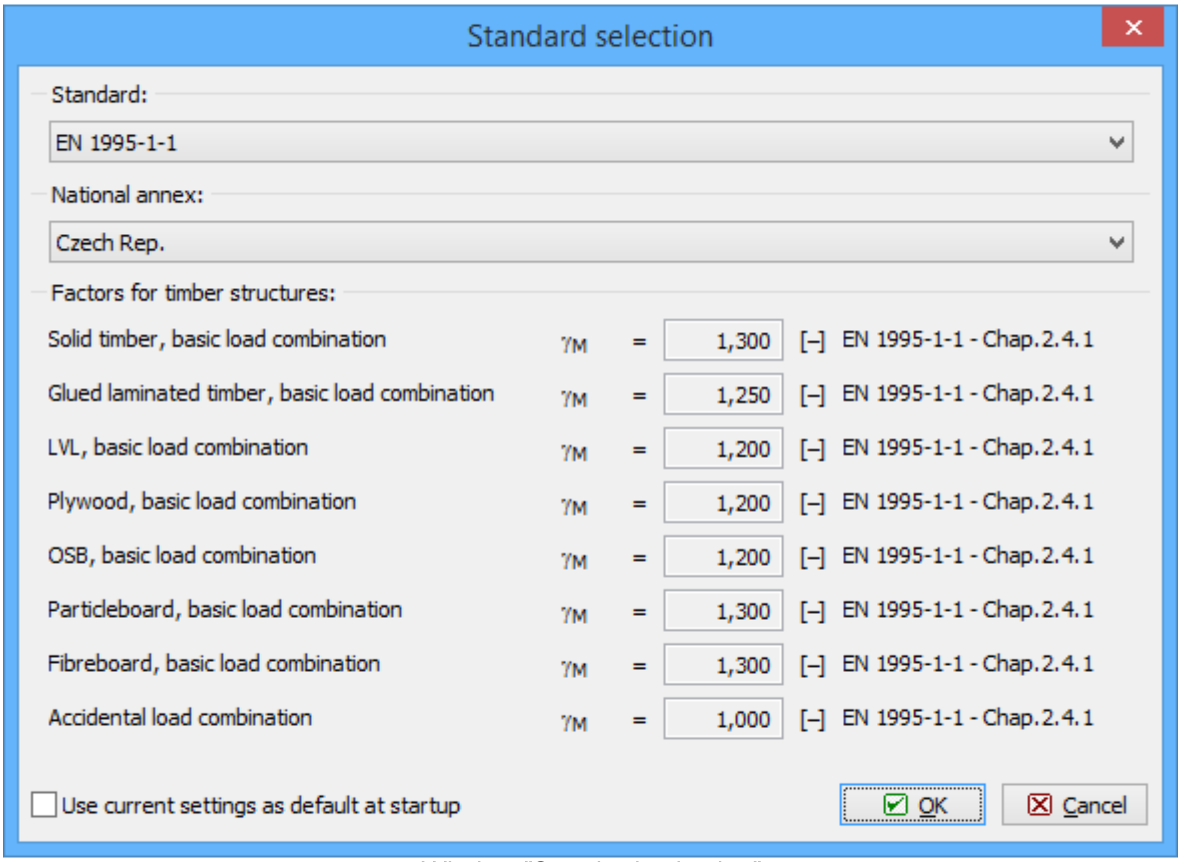
The design standard can be selected in this window. These design standards are available: "**EN 1995-1-1**" and "**CSN 73 1702**". Different national annexes can be selected for design standard "**EN 1995-1-1**". The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex. The values of partial factors $\gamma_M$ both for basic and accidental design situations can be specified for option "**User defined**". Default values are based on chapter *2.4.1* of EN 1995-1-1 (table *2.3*).

Partial factors are described in the chapter "**Ultimate limit state**" of theoretical help.

Button "**Default**" contains these two tools:

| | | |
|---|---|---|
| **Adopt default settings** | • | Set the default values for all parameters. |
| **Save settings as default** | • | Set entered parameters as defaults for new projects. |

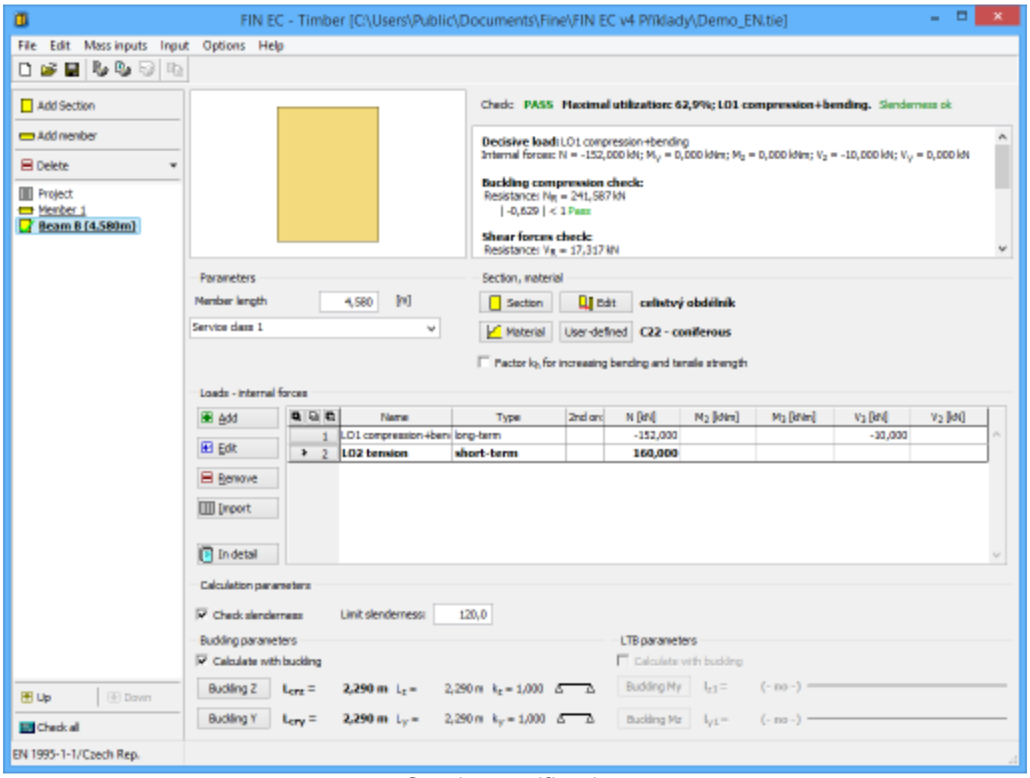
 



*Window "Standard selection"*

## Section

Task type "**Section**" is suitable for the fast verification of the timber cross-section, that is loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".


*Section verification*

The window contains these parts:

### Parameters

The member length (is used in buckling, LT buckling and slenderness verifications) and the service class can be selected in this part. The service class takes the moisture content into consideration and influences the value of factor $k_{mod}$ (described in the part "**Material characteristics**" of the theoretical help).

## Section, Material

Following buttons are placed in this part:

| | |
|---|---|
| **Section** | • Input of cross-section geometry in the window "**Cross-section edit**". |
| **Edit** | • Runs "**Cross-section editor**" in appropriate mode. If the cross-section isn't specified yet, the window "**Cross-section edit**" is opened. |
| **Material** | • The timber grade can be selected in the window "**Materials catalogue**". Both basic strength classes according to EN 1912 (C24 etc.) and local timber grades are available. |
| **User defined** | • Input of arbitrary values of material characteristics in the window "**Materials editor**". |

The tensile and bending strengths can be increased in accordance with the chapter 3.2 of EN 1995-1-1 for cross-section dimensions less than *150mm* using factor $k_h$. This rule can be applied using check box "**Factor $k_h$ for increasing bending and tensile strength**". This factor is described in the part "**Material characteristics**" of the theoretical help.

## Loads - internal forces

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads can be also imported from text or **.csv* file. This feature can be used for import of large number of loads, that were calculated with the help of another structural engineering program. Import can be performed using window "**Load import**", that can be launched by button "**Import**".

## Calculation parameters

The slenderness verification can be switched on in this part. The maximum permitted value of slenderness ratio has to be specified by the user. The verification is described in the part "**Slenderness verification**" of the theoretical help.

## Buckling parameters

The buckling parameters can be specified in this part. The parameters are organized into two different directions *z* and *y* and can be specified in the window "**Buckling**", that can be launched by using the buttons "**Buckling Z**" and "**Buckling Y**". The main inputs (the buckling length, end conditions, basic length) are displayed on the right side of the buttons. The buckling parameters input is enabled only for tasks with at least one load, that contains compressive force. Buckling calculations are described in the chapter "**Buckling**" of the theoretical help.

## LTB parameters

The parameters of lateral torsional buckling can be specified in this part. The parameters are organized into two different directions *z* and *y* and can be specified in the window "**LT buckling parameters**", that can be launched by using the buttons "**Buckling My**" and "**Buckling Mz**". The main inputs (the basic length, beam and load types) are displayed on the right side of the buttons. The parameters input is enabled only for tasks with at least one load, that contains corresponding bending moment. Lateral torsional buckling calculations are described in the chapter "**Lateral torsional buckling**" of the theoretical help.

## Results

Results of the analysis for the worst load are displayed in the right upper part of the main window. Detailed results for the active load in the loads table can be displayed using button "**In detail**". These results are displayed in the new window, text in this window can be copied into clipboard using shortcut *Ctrl+C* and pasted into a document.

Analysis is described in the part "**Ultimate limit state**" of the theoretical help.

# Connection

This window contain properties of the connection between particular shafts/flanges of the built-up cross-section. These options are available:

- **Not connected** - no connection between particular cross-sections is considered during the analysis. The total capacity of the member is calculated as a sum of capacities of particular parts. This style will be used if the setting "**Cross-section elements are joined**" is switched on.
- **Packs** - Particular shafts of the member are connected by intermediate packs. The joints may be either nailed or glued. The distance and size of packs have to be specified. The analysis is based on chapter *C.3* of EN 1995-1-1.
- **Battens** - Particular shafts of the member are connected by battens. The joints may be either nailed or glued. The distance and size of battens have to be specified. The analysis is based on chapter *C.3* of EN 1995-1-1.

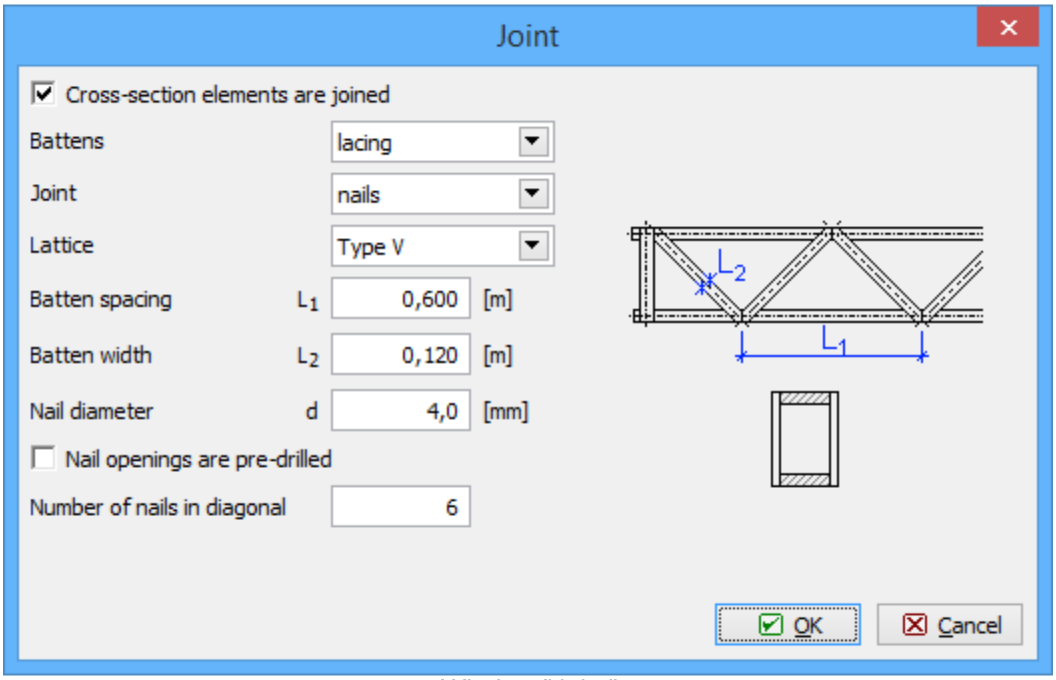
- **Lattice** - Flanges of the member are connected by *V*- or *N*-lattice. The joints may be either nailed or glued. Necessary inputs are distance between joints on flange and diagonal size. The number and diameter of nails per diagonal (if a diagonal consists of two or more pieces, thesum of the nails should be used). The check box "**Pre-drilled holes**" influences the value of slip modulus for nails. Analysis is based on *C.4* of EN 1995-1-1.

Analysed spaced columns with packs or gussets columns should fulfil conditions in accordance with *C.3.1(2)* of EN 1995-1-1, mainly:

- The number of bays is at least three, i.e. the shafts are connected at least at the ends and at the third points.
- The distance $b_m$ between shafts isn't greater than three times the shaft thickness (for packs) or six times the shaft thickness (for battens).
- The packs length $L_2$ satisfies the condition $L_2/b_m \geq 1,5$, The gussets satisfies the condition $l_2/b_m \geq 2$.

Analysed lattice columns should fulfil conditions in accordance with *C.4.1(2)* of EN 1995-1-1, mainly:

- there are at least three bays
- the slenderness ratio of the individual flange (node length $l_1$) is not greater than *60*
- no local buckling occurs in the flanges corresponding to the column length $l_1$

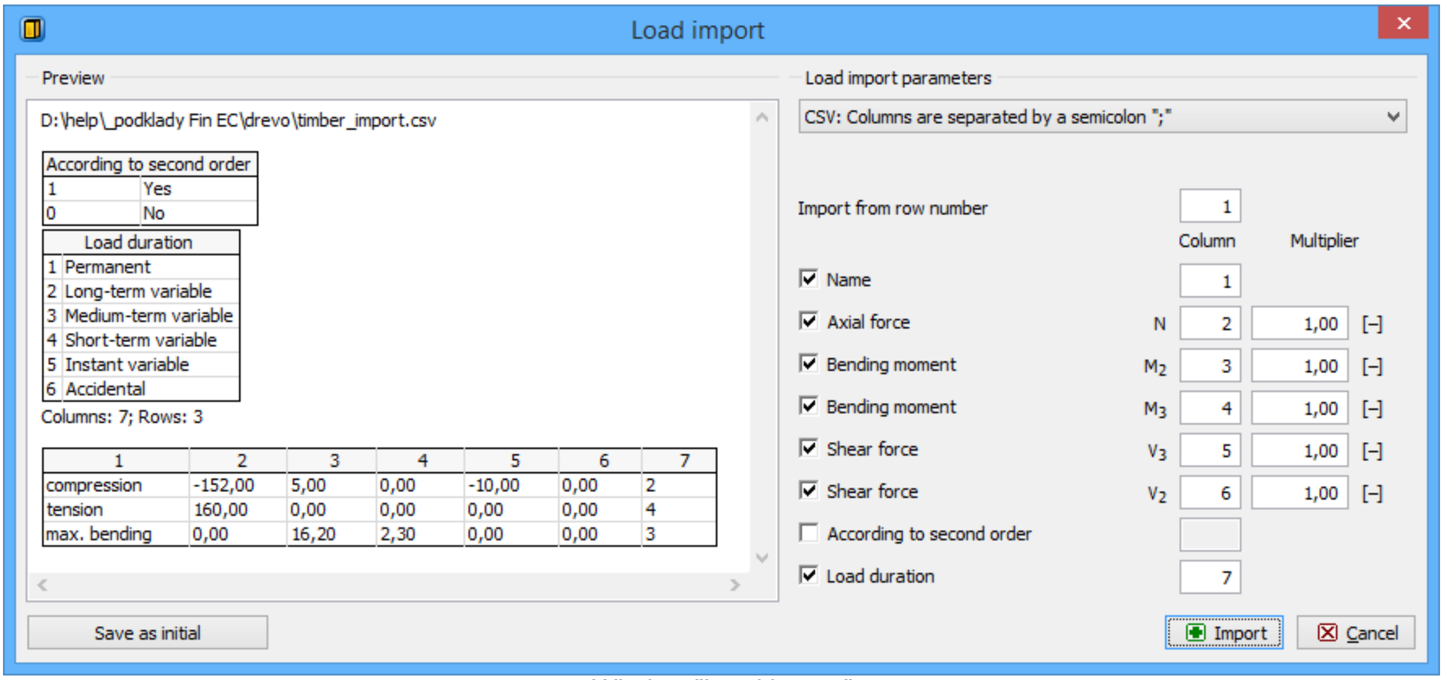



*Window "Joint"*

## Load import

Window "**Load import**" appears after loading the text or **.csv* source file. The data included in the source file can be arranged in this window. Left part shows content of imported file. The assignment of the certain column to the load entry (force or property) can be done in the right part of the window. Numerical entries can be also multiplied by specified multiplier. This multiplier can be used mainly for conversion caused by different units in the source file and in the program. Initial rows can be skipped using setting "**Import from row number**". This setting is helpful for source files with headings. Default units required by the software are *[kN]* and *[kNm]*.



*Window "Load import"*

## Preparing text file

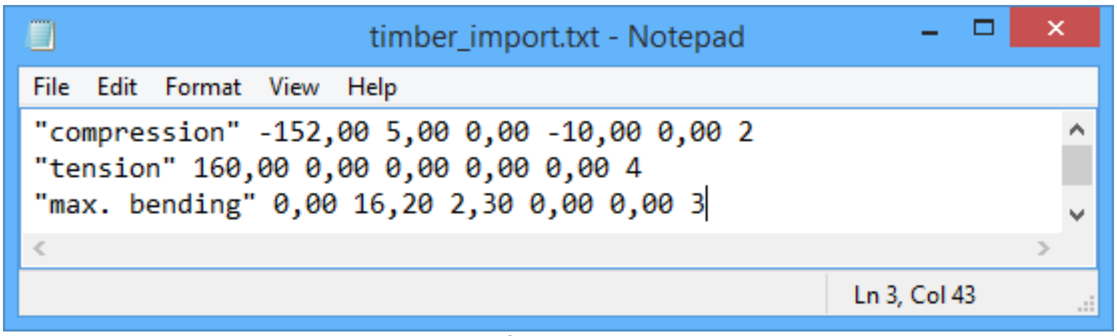
Text file can be created in any text editor (e.g. *Notepad*, *Word*, *Writer*). File format requires that every row contains one load case. Every row can contain all values of internal forces separated by space or tabulator. The order may differ comparing to the order of forces in the software, however, order has to be identical for all rows of the document. The load name, load duration and second order effect may be also specified for every load. The load duration and second order effect has to be specified using this numerical codes:

| | |
|---|---|
| 1 | Permanent |
| 2 | Variable long-term |
| 3 | Variable middle-term |
| 4 | Variable short-term |
| 5 | Variable instantaneous |
| 6 | Accidental |

This scheme is used for consideration of analysis in accordance with II. order theory:

| | |
|---|---|
| 1 | Forces calculated in accordance with II. order theory |
| 0 | Forces aren't calculated in accordance with II. order theory |

The file can be also created using part of analysis documentation from arbitrary structural analysis software.

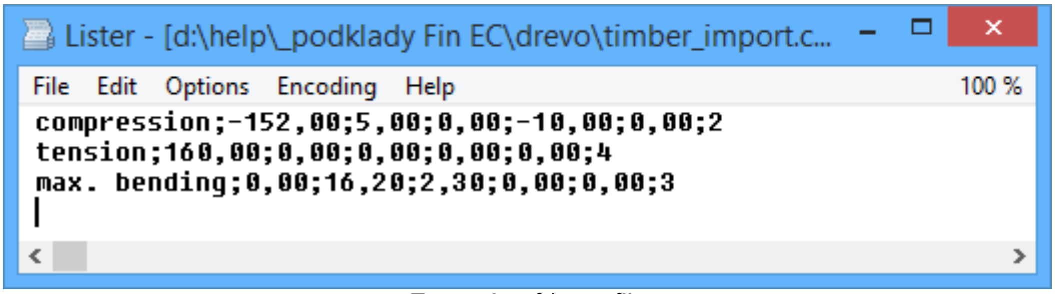


*Text file in Notepad*

## Preparing *.csv file

The rules for **.csv* (comma-separated values) files are almost identical . Main difference is, that the particular values are separated by semicolon ";".

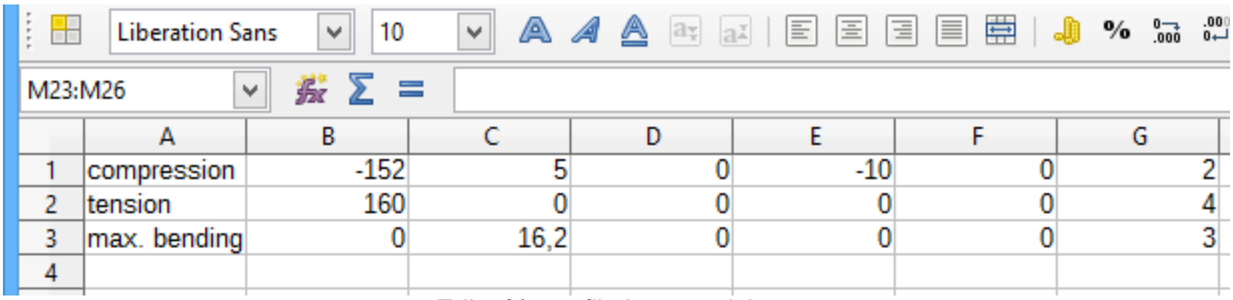
 




```
compression;-152,00;5,00;0,00;-10,00;0,00;2
tension;160,00;0,00;0,00;0,00;0,00;4
max. bending;0,00;16,20;2,30;0,00;0,00;3
```

*Example of *.csv file*

This file type can be easily created using spreadsheet programs like *Excel* or *Calc*. Created document can be saved as **.csv* file with appropriate separator.




| | A | B | C | D | E | F | G |
|---|---|---|---|---|---|---|---|
| 1 | compression | -152 | 5 | 0 | -10 | 0 | 2 |
| 2 | tension | 160 | 0 | 0 | 0 | 0 | 4 |
| 3 | max. bending | 0 | 16,2 | 0 | 0 | 0 | 3 |
| 4 | | | | | | | |

*Edit of *.csv file in spreadsheet*

# Member

Task type "**Member**" is suitable for the detailed verification of the timber member, that is loaded by unlimited number of loads. This task type is suitable mainly for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

The main frame of member design contains these inputs:

**Member length** • Total member length specified in metres.

**Limit slenderness** • Option for input of maximum slenderness ratio for the member.

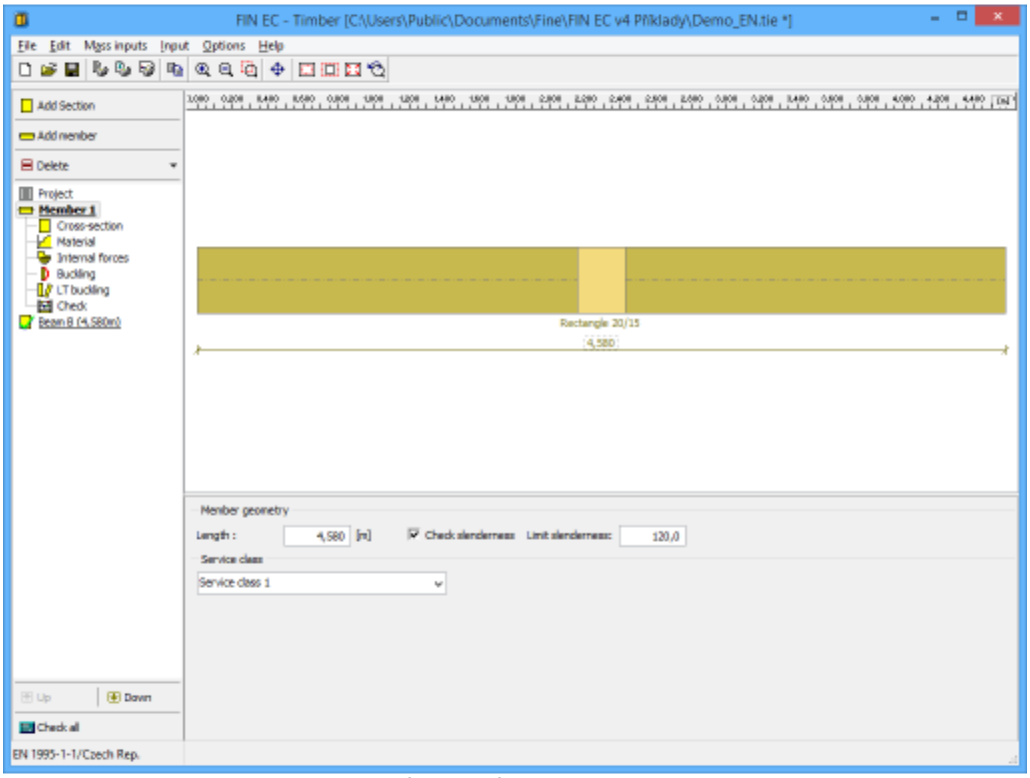
**Service class** • The service class takes the moisture content into consideration and influences the value of factor $k_{mod}$ (described in the part "**Material characteristics**" of the theoretical help).

The member design contains these parts:

- **Cross-section**
- **Material**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Analysis**

General work with members (addition, manipulation) is described in the chapter "**Tree menu**".

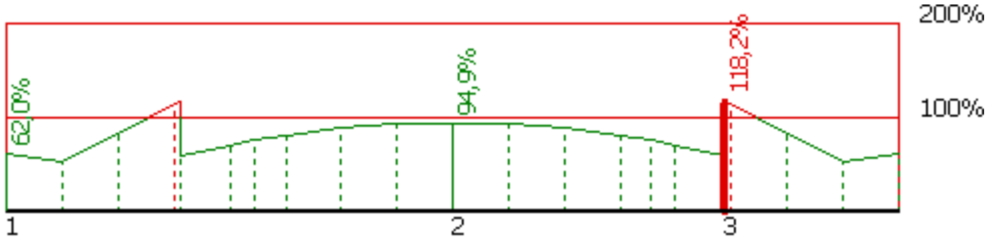
 



*Main frame of member design*

## Analysis

This part shows the results of structural analysis for the member. The results are displayed with the help of utilization diagram in the workspace. Passed member is coloured by green colour. The parts where the utilization exceeds 100% are coloured by red colour.



*Utilization of the member, red parts failed*

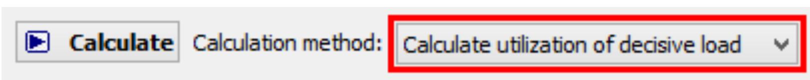
The frame in the bottom part contains tools for changing the analysis method and for the work with analysis sections (positions with detailed results).

### Analysis method

Analysis method can be selected in the upper part of the input frame. The analysis style and considered loads are selected according to the certain applied method. These options are available:

| | |
|---|---|
| **Calculate utilization of decisive load** | • Display the results for the decisive load (the load with the maximum utilization). All entered loads are considered in this option. |
| **Calculate envelope of maximum utilizations from all loads** | • Display envelope of the maximum utilization in every point of the member length. All entered loads are considered in this option. |
| **Individual loads** | • Show results for selected load. |

The analysis have to be run by the button "**Analyse**" after the change of the analysis method.



*Selection of analysis method*

### Verification sections

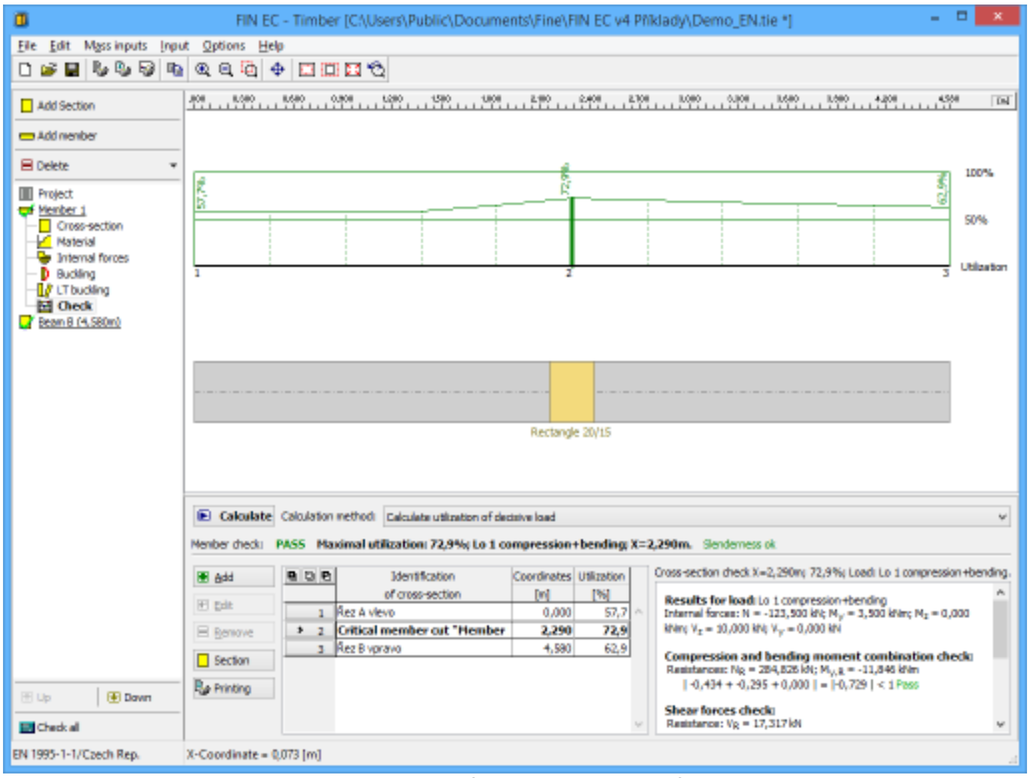
Verification sections are used for display of detailed results in certain points along the member length. These verification sections can be converted into standalone "**Section**" tasks, the results can be printed using graphical outputs. The verification section is created automatically in the position with the worst utilization, other sections can be added manually.

These tools are available for the work with sections:

| | | |
|---|---|---|
| **Section** | • | Converts the active section on member into standalone task of "**Section**" type. All member properties (cross-section, material, internal forces, parameters of buckling and LT buckling) are copied into the new task. |
| **Add** | • | Inserts a new section on member. The detailed results can be displayed for this section. Properties of the new section have to be specified in the window "**New section for analysis**". |
| **Edit** | • | Edit of the active cross-section properties (name, coordinate). |
| **Remove** | • | Remove the active section from the list. |
| **Printing** | • | Print results for all entered sections using printing window. |

The new section for analysis can be also added using active workspace. New section can be added by double-click on the needed position on the member.



*Part "Analysis" of the member verification*

# Beam

The task type "**Beam**" is suitable for the comprehensive analysis of general horizontal beam. Only single axis bending is supported in this task type.

The main frame of beam design contains these inputs:

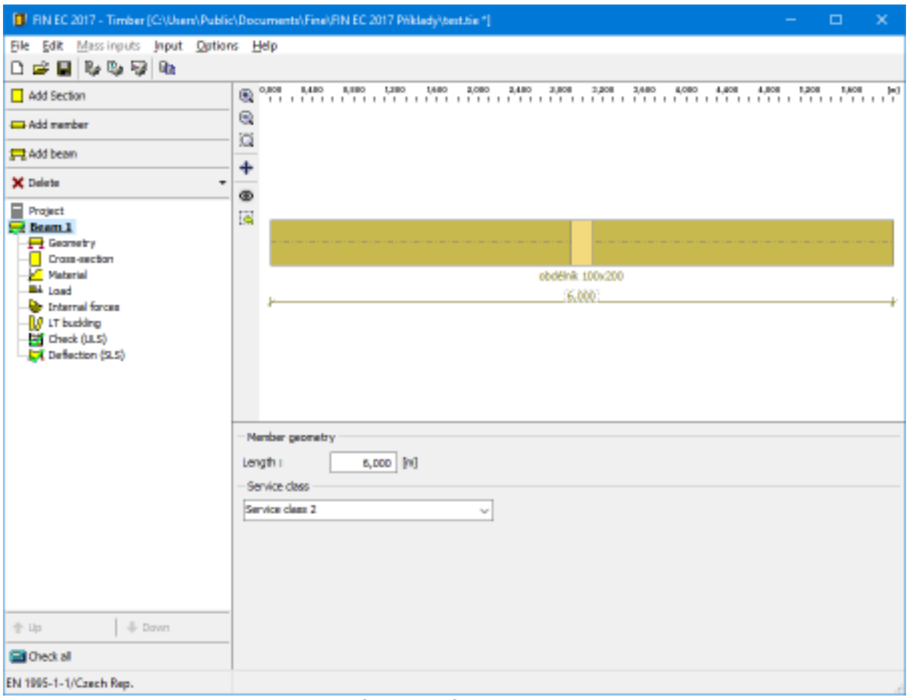
| | | |
|---|---|---|
| **Member length** | • | Total beam length specified in metres. |
| **Service class** | • | The service class takes the moisture content into consideration and influences the value of factor $k_{mod}$ (described in the part "**Material characteristics**" of the theoretical help). |

The beam analysis contains these parts:

- **Geometry**
- **Cross-section**
- **Material**
- **Load**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Analysis (ULS)**
- **Deflection (SLS)**

General work with beams (addition, manipulation) is described in the chapter "**Tree menu**".




*Main frame of beam design*

# Deflection (SLS)

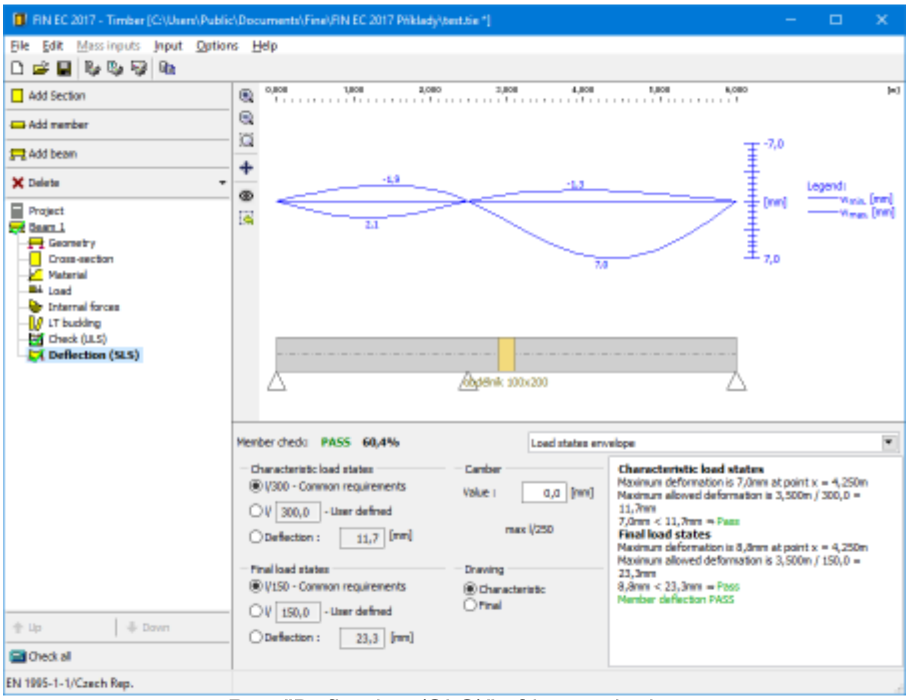
This part of the beam design shows results of deflection verification (serviceability limit state) along the member length. The verification is performed only for load combinations that are dedicated to the serviceability limit state design. If there isn't any entered load combination for serviceability limit state in the part "**Load**", the results aren't available. The frame in the bottom part contains these settings for deflection verification:

**Common requirements** • This limit shall be considered for deflection that could impair the appearance and general utility of the structure. The values are based on the table *7.2* of EN 1995-1-1.

**User defined requirements** • User defined limit specified as span-depending value

**Deflection** • User defined limit specified as an absolute value in *mm*

Limits both for characteristic combinations (used for verification of $w_{inst}$ ) and for combinations, which are dedicated for calculation of final deflection $w_{net,fin}$ can be specified in the input frame in the bottom part of the window.




*Part "Deflection (SLS)" of beam design*

# Program Timber Fire

The software "**Timber Fire**" verifies fire resistance of timber cross-sections according to EN 1995-1-2.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Input**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Three types of particular tasks can be used in the software:

**Section** • Fast analysis of timber cross-section with unlimited number of loads.

**Member** • Analysis of the whole member with specified diagrams of internal forces. This type is suitable for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

**Beam** • Comprehensive analysis of horizontal beam with unlimited number of supports.

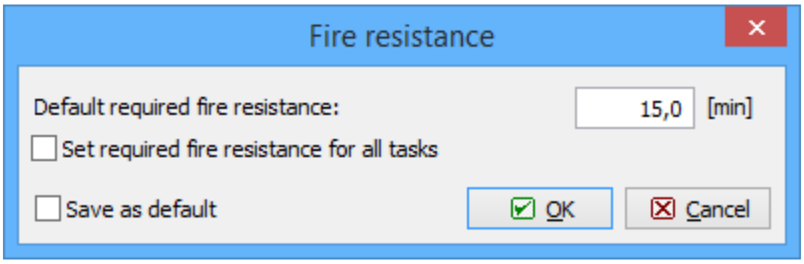
These tasks can be added with the help of the buttons "**Add section**", "**Add member**" or "**Add beam**" in the heading of the tree menu.

## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The part "**Fire resistance parameters**" contains an option for input of default value of fire resistance for all tasks (sections and members) of the project. The input can be done in the window "**Fire resistance**", that can be launched by using the button "**Edit**". The check box **"Set required fire resistance for all tasks**" sets the specified fire resistance also to all existing tasks in the project. The setting "**Save as default**" will set specified fire resistance as a default for all new projects.

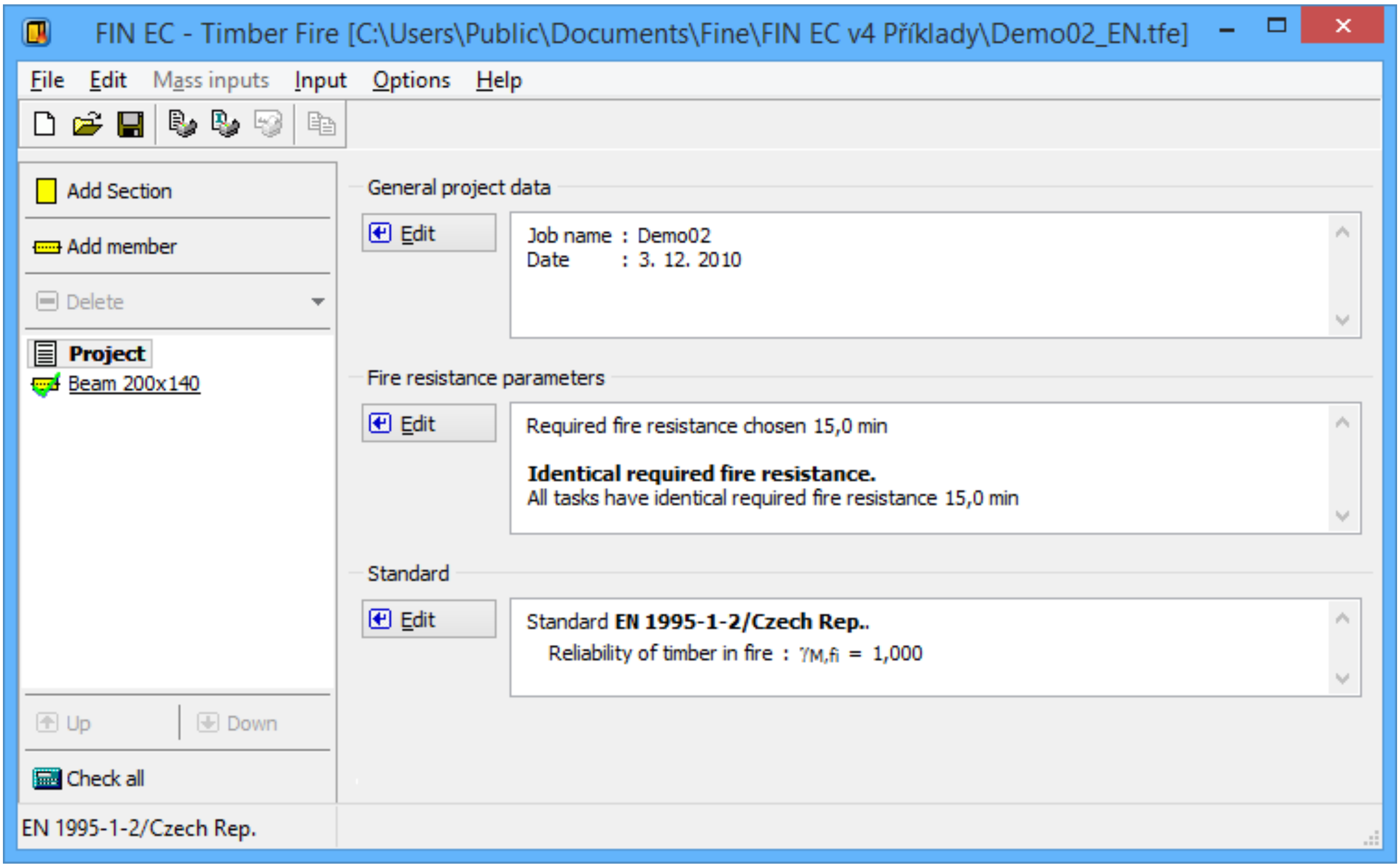



*Window "Fire resistance"*

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "**Standard selection**", that can be launched by the button "**Edit**".

Analysis and verifications used in the software are described in the theoretical part of the help.




*Main application window*

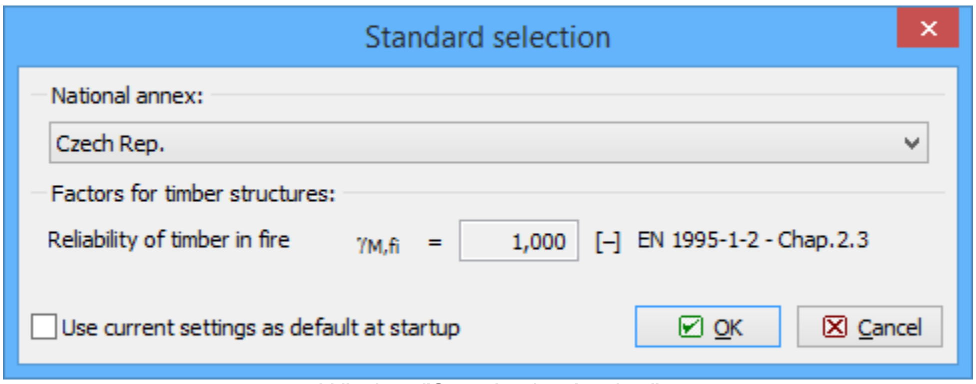
## Standard selection

The design standard can be selected in this window. These design standards are available: "**EN 1995-1-1**" and "**CSN 73 1702**". Different national annexes can be selected for design standard "**EN 1995-1-1**". The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex. The value of partial factor $\gamma_{M,fi}$ can be specified for option "**User defined**". Default value is based on chapter *2.3* of EN 1995-1-2.

Partial factors are described in the chapter "**Ultimate limit state**" of theoretical help.

Button "**Default**" contains these two tools:

| | | |
|---|---|---|
| **Adopt default settings** | • | Set the default values for all parameters. |
| **Save settings as default** | • | Set entered parameters as defaults for new projects. |

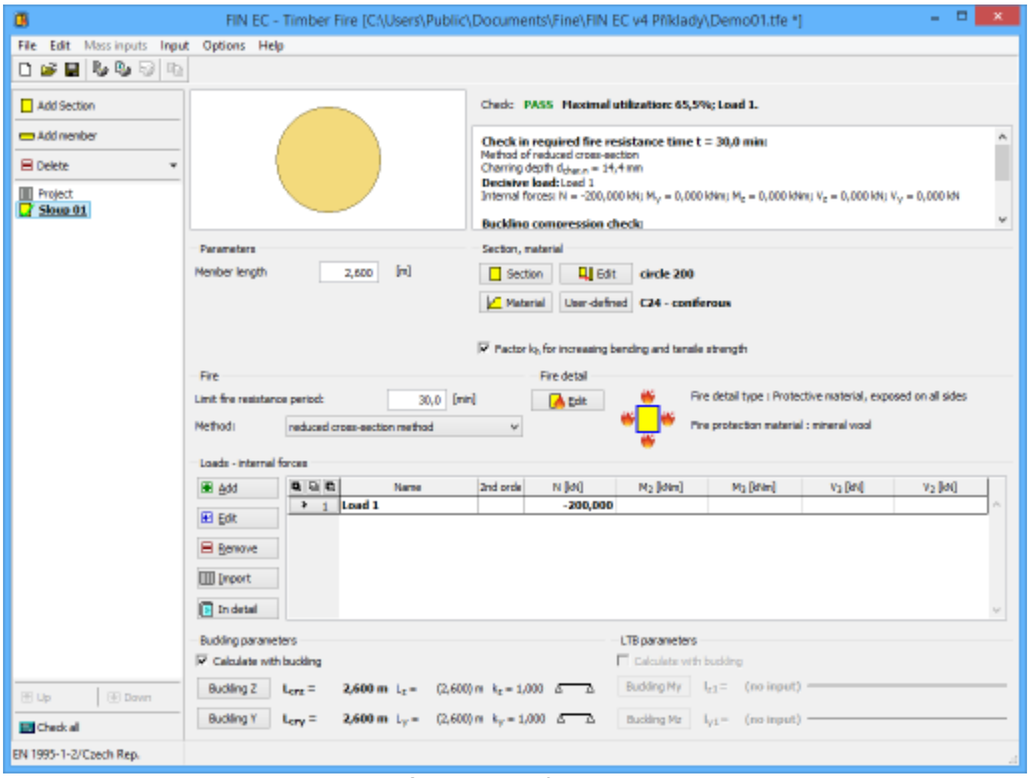



*Window "Standard selection"*

## Section

Task type "**Section**" is suitable for the fast verification of the timber cross-section, that is loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".




*Section verification*

The window contains these parts:

## Parameters

The member length that is used in buckling and lateral torsional buckling verifications.

## Fire

The design method and required time of fire resistance can be specified in this part. The value defined in the main application screen is used as a default.

The design procedure can be selected in accordance with chapter *4.2* of EN 1995-1-2. Available are "**Reduced cross-section method**" and "**Reduced properties method**". Recommended procedure is the "**Reduces cross-section method**" according to the standard. Both procedures are described in the chapter "**Methods for fire resistance**".

## Section, Material

Following buttons are placed in this part:

| | | |
|---|---|---|
| **Section** | • | Input of cross-section geometry in the window "**Cross-section edit**". |
| **Edit** | • | Runs "**Cross-section editor**" in appropriate mode. If the cross-section isn't specified yet, the window "**Cross-section edit**" is opened. |
| **Material** | • | The timber grade can be selected in the window "**Materials catalogue**". Both basic strength classes according to EN 1912 (C24 etc.) and local timber grades are available. |
| **User defined** | • | Input of arbitrary values of material characteristics in the window "**Materials editor**". |

The different charring rates are defined for softwood and hardwood in the table 3.1 of EN 1995-1-2. The only exception is the beech that shall be considered as a softwood in accordance with *3.4.2(6)* . The radio button "**hardwood beach**"/"**other hardwood**" takes this rule into account. This setting is available for *Dxx* timber grades.

tensile and bending strengths can be increased in accordance with the chapter 3.2 of EN 1995-1-1 for cross-section dimensions less than *150mm* using factor $k_h$. This rule can be applied using check box "**Factor $k_h$ for increasing bending and tensile strength**". This factor is described in the part "**Material characteristics**" of the theoretical help.

## Fire protection

The fire protection parameters can be specified in this part. The fire resistance parameters are organized in the window "**Fire detail**" that can be launched by the button "**Edit**".

## Loads - internal forces

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads can be also imported from text or **.csv* file. This feature can be used for import of large number of loads, that were calculated with the help of another structural engineering program. Import can be performed using window "**Load import**", that can be launched by button "**Import**".

## Buckling parameters

The buckling parameters can be specified in this part. The parameters are organized into two different directions *z* and *y* and can be specified in the window "**Buckling**", that can be launched by using the buttons "**Buckling Z**" and "**Buckling Y**". The main inputs (the buckling length, end conditions, basic length) are displayed on the right side of the buttons. The buckling parameters input is enabled only for tasks with at least one load, that contains compressive force.

## LTB parameters

The parameters of lateral torsional buckling can be specified in this part. The parameters are organized into two different directions *z* and *y* and can be specified in the window "**LT buckling parameters**", that can be launched by using the buttons "**Buckling My**" and "**Buckling Mz**". The main inputs (the basic length, beam and load types) are displayed on the right side of the buttons. The parameters input is enabled only for tasks with at least one load, that contains corresponding bending moment.

## Results

Results of the analysis for the worst load are displayed in the right upper part of the main window. Detailed results for the active load in the loads table can be displayed using button "**In detail**". These results are displayed in the new window, text in this window can be copied into clipboard using shortcut *Ctrl+C* and pasted into a document.

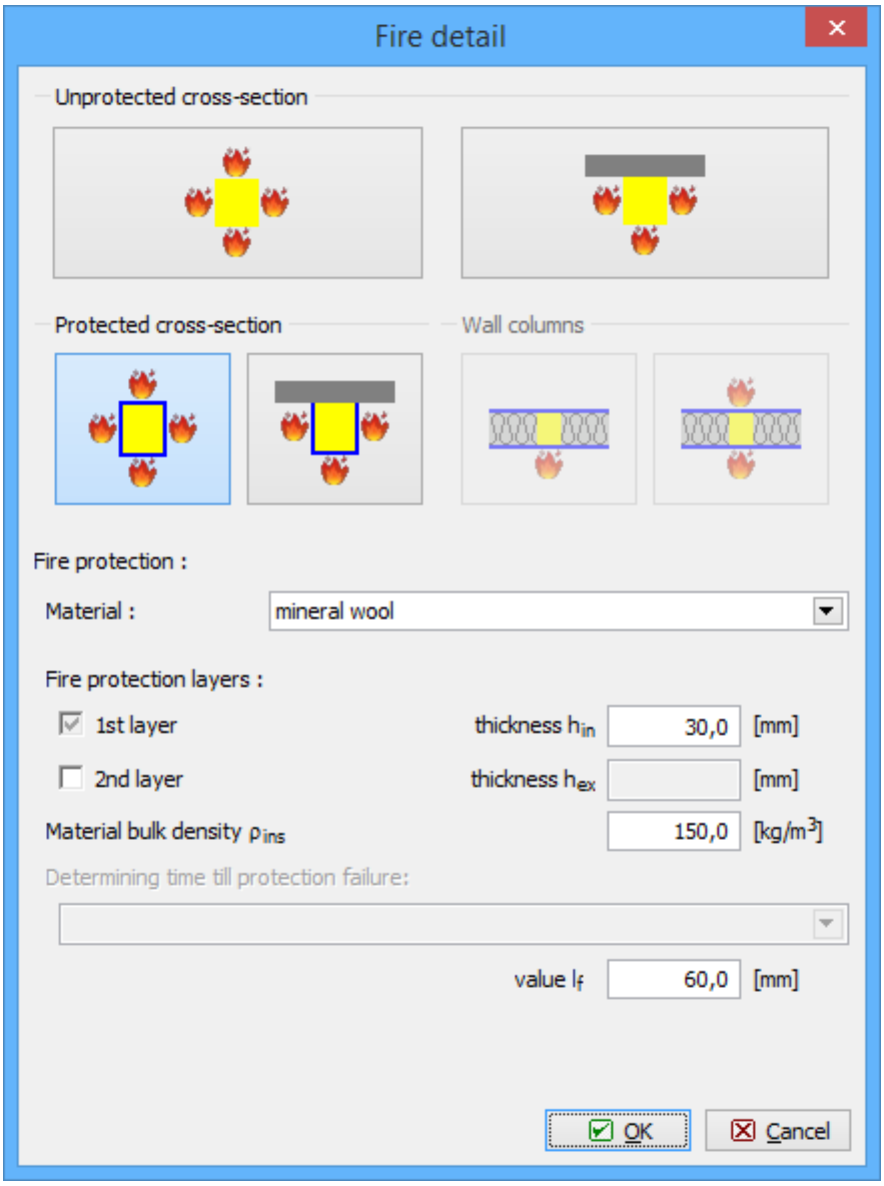
Analysis is described in the part "**Ultimate limit state**" of the theoretical help.

# Fire detail

The style of fire exposition and fire protection can be defined in this window. There are two types of unprotected detail: cross-section exposed from all four sides or only from three sides (mainly for beams). The identical options are available for protected cross-sections. Columns in wall (protected on two opposite sides) can be exposed from one or two sides.

The cross-sections can be protected by fire protective claddings. The fire protection delays the start of charring $t_{ch}$ and changes the charring rate. The timber, wood-based panels, plasterboards and mineral wood can be used as a fire protective material. The necessary properties like density, thickness and ply number have to be specified for any material. The gaps properties have to be specified for plasterboards additionally. The fasteners length have to be specified for mechanically connected claddings, as influences the failure time of protection $t_f$. Other protective material with user defined characteristics can be also entered for the detail. These characteristics are described in the chapter "**Calculation of char layer**".

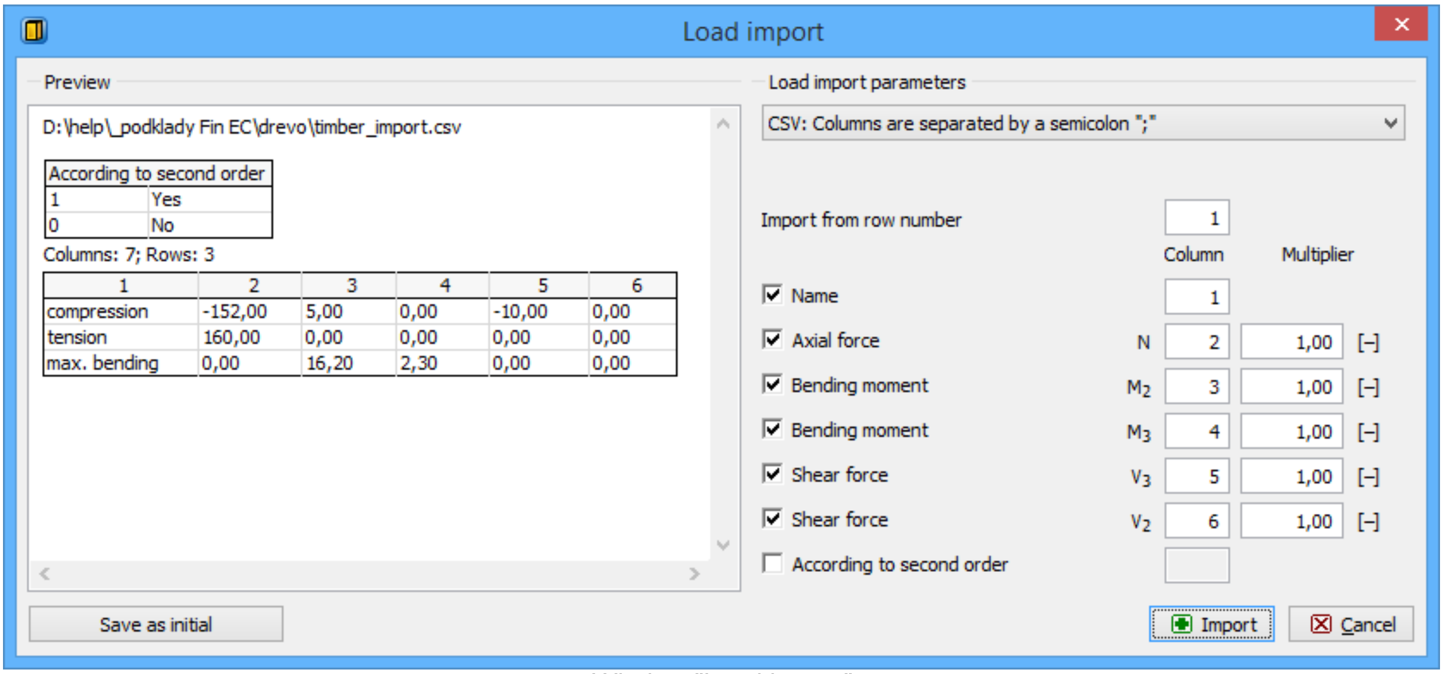



*Window "Fire detail"*

## Load import

Window "**Load import**" appears after loading the text or **.csv* source file. The data included in the source file can be arranged in this window. Left part shows content of imported file. The assignment of the certain column to the load entry (force or property) can be done in the right part of the window. Numerical entries can be also multiplied by specified multiplier. This multiplier can be used mainly for conversion caused by different units in the source file and in the program. Initial rows can be skipped using setting "**Import from row number**". This setting is helpful for source files with headings. Default units required by the software are *[kN]* and *[kNm]*.




*Window "Load import"*

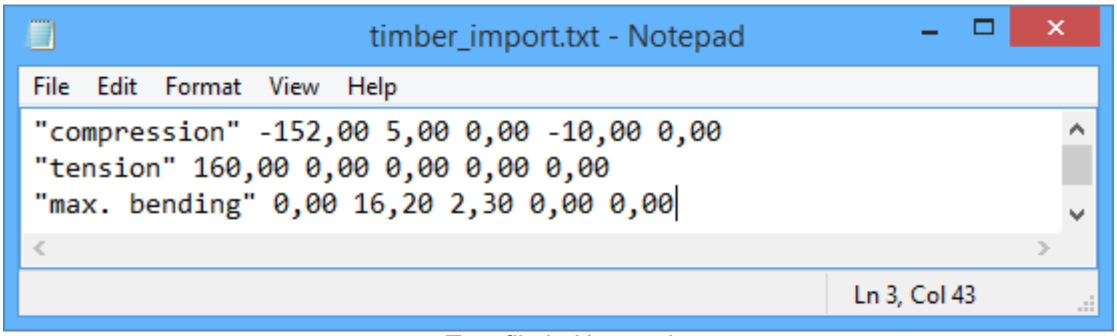
## Preparing text file

Text file can be created in any text editor (e.g. *Notepad*, *Word*, *Writer*). File format requires that every row contains one load case. Every row can contain all values of internal forces separated by space or tabulator. The order may differ comparing to the order of forces in the software, however, order has to be identical for all rows of the document. The load name and second order effect may be also specified for every load.

This scheme is used for consideration of analysis in accordance with II. order theory:

| | |
|---|---|
| 1 | Forces calculated in accordance with II. order theory |
| 0 | Forces aren't calculated in accordance with II. order theory |

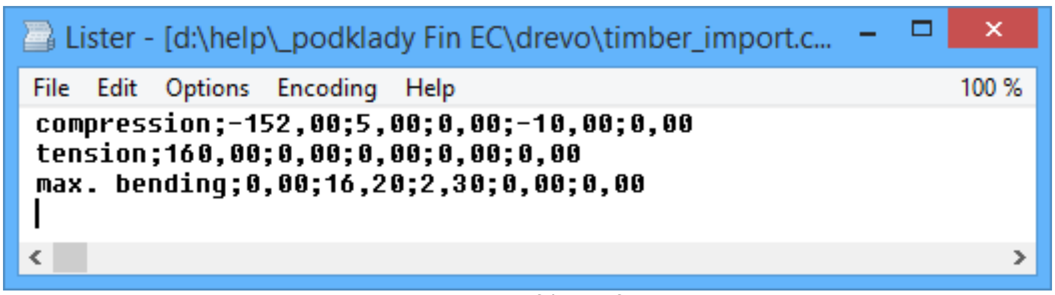
The file can be also created using part of analysis documentation from arbitrary structural analysis software.




*Text file in Notepad*

## Preparing **.csv* file

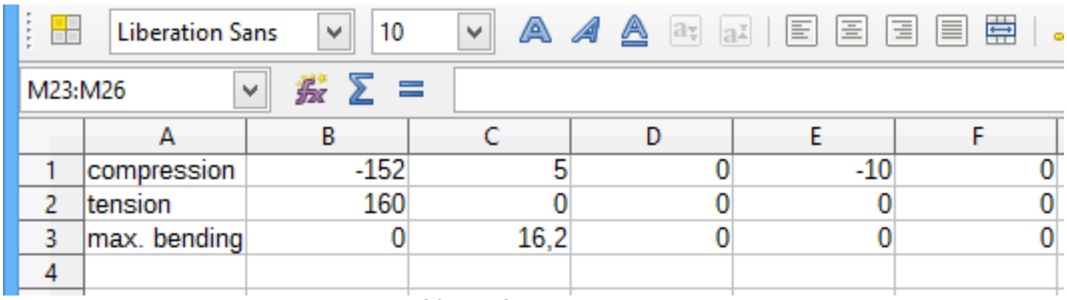
The rules for **.csv* (comma-separated values) files are almost identical . Main difference is, that the particular values are separated by semicolon ";".




*Example of *.csv file*

This file type can be easily created using spreadsheet programs like *Excel* or *Calc*. Created document can be saved as **.csv* file with appropriate separator.




*Edit of *.csv file in spreadsheet*

# Member

Task type "**Member**" is suitable for the detailed verification of the timber member, that is loaded by unlimited number of loads. This task type is suitable mainly for the batch analysis in programs "**Fin 2D**" a "**Fin 3D**".

The main frame of member design contains these inputs:

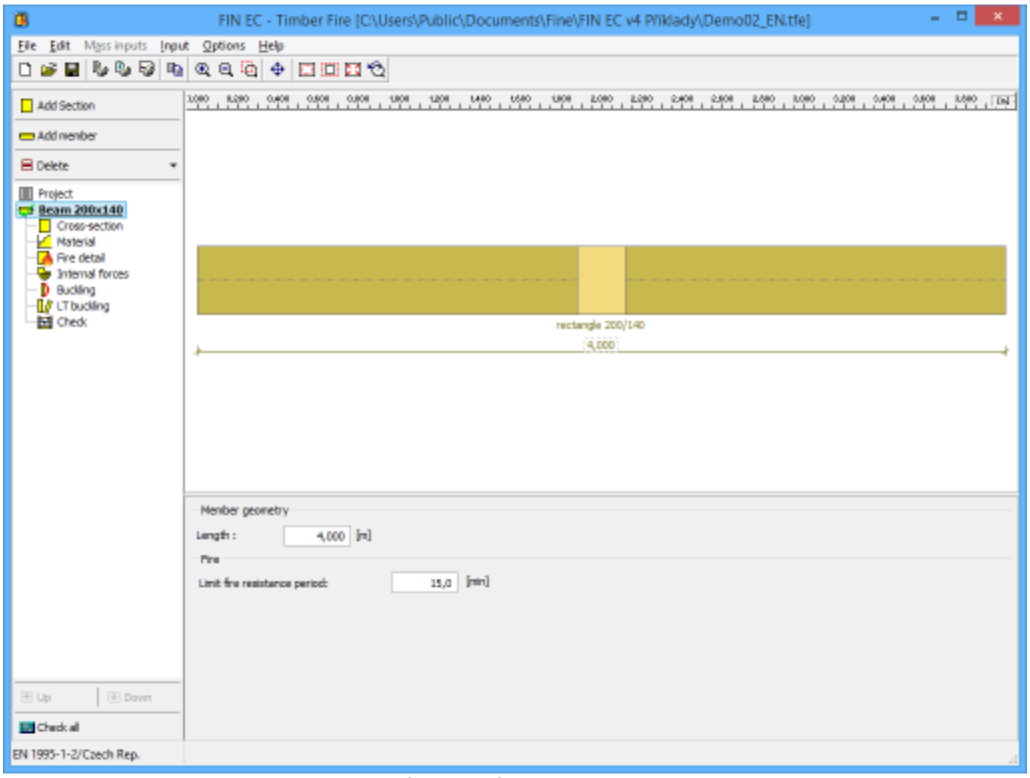
**Member length**
- Total member length specified in metres.

**Limit fire resistance period**
- Fire resistance in minutes, the verification is performed for this time.

The member design contains these parts:

- **Section**
- **Material**
- **Fire detail**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Analysis**

General work with members (addition, manipulation) is described in the chapter "**Tree menu**".



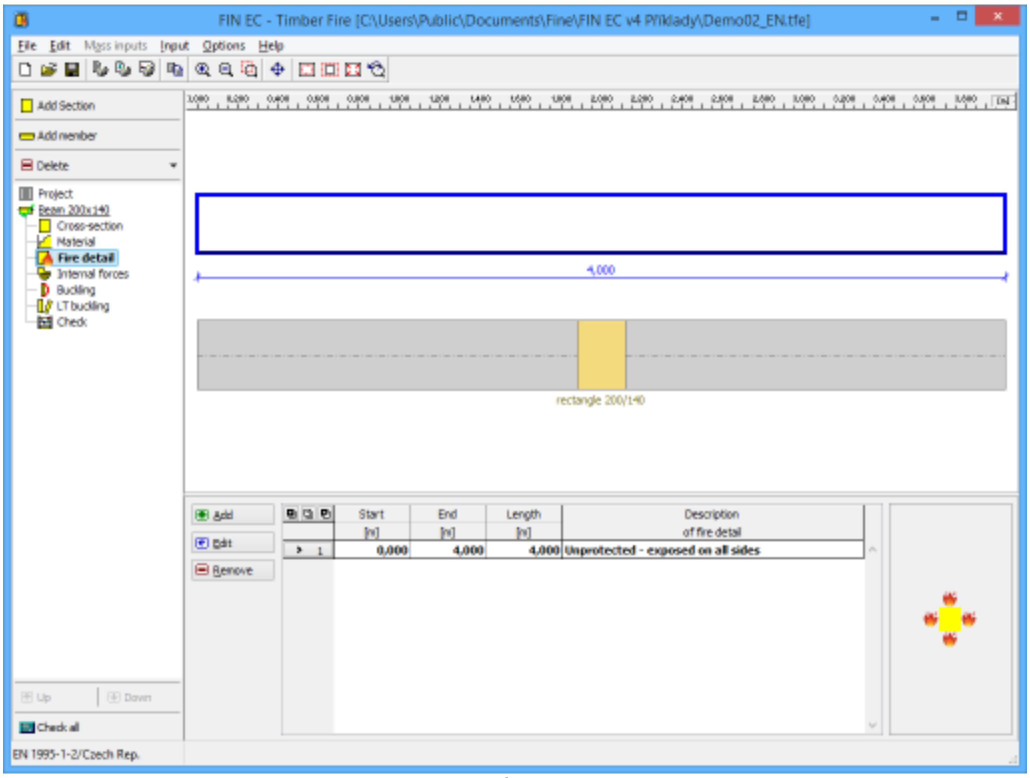

*Main frame of member design*

# Fire detail

The fire protection parameters can be specified in this part of the tree menu. These parameters can be specified for the whole member or can vary along the member length. In this case, the member has to be divided into particular sectors, every sectors may contain different parameters of fire resistance. The table contains one sector along the whole member length as a default for every new member. This sector can be modified using button "**Edit**" or by double-click on the table row. The fire resistance parameters are organized in the window "**Fire detail**". More sectors can be added (button "**Add**") for input of different fire resistance parameters along the member length. The new sectors are automatically added behind the first sector according to the start coordinate called "**Sector beginning**". This point is automatically considered as the end of previous sector.

The particular sectors are displayed also in the active workspace. Double-click on certain sector launches the appropriate window for sector edit.

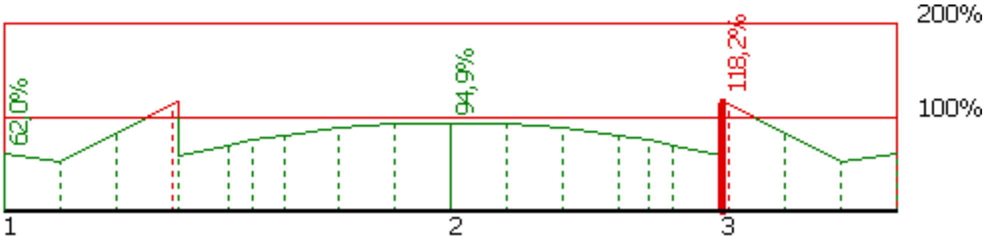
The fire protection is described in the chapter "**Calculation of char layer**" of the theoretical part of the help.



*Part "Fire detail" of the member design*

# Analysis

This part shows the results of structural analysis for the member. The results are displayed with the help of utilization diagram in the workspace. Passed member is coloured by green colour. The parts where the utilization exceeds 100% are coloured by red colour. The verification of the fire resistance is performed using the residual cross-section of the member. The residual cross-section is the calculated as an original cross-section reduced by the char layer.




*Utilization of the member, red parts failed*

The frame in the bottom part contains tools for changing the analysis method and for the work with analysis sections (positions with detailed results).

## Analysis method

The fire resistance method and the analysis method can be selected in the upper part of the input frame. There are two methods for the analysis of fire resistance according to *4.2* of EN 1995-1-2: "**Reduced cross-section method**" and "**Reduced properties method**". The "**Reduced cross-section method**" is recommended in the standard. Both methods are described in the part "**Methods for fire resistance analysis**" of the theoretical help.

The analysis style and considered loads are selected according to the certain applied method. These options are available:

| | |
|---|---|
| **Calculate utilization of decisive load** | • Display the results for the decisive load (the load with the maximum utilization). All entered loads are considered in this option. |

**Calculate envelope of maximum utilizations from all loads**
- Display envelope of the maximum utilization in every point of the member length. All entered loads are considered in this option.

**Individual loads**
- Show results for selected load.

The analysis have to be run by the button "**Analyse**" after the change of the analysis method.

The analysis is described in the part "**Ultimate limit state**" of the theoretical help.

## Verification sections

Verification sections are used for display of detailed results in certain points along the member length. These verification sections can be converted into standalone "**Section**" tasks, the results can be printed using graphical outputs. The verification section is created automatically in the position with the worst utilization, other sections can be added manually.

These tools are available for the work with sections:

**Section**
- Converts the active section on member into standalone task of "**Section**" type. All member properties (cross-section, material, internal forces, parameters of buckling and LT buckling) are copied into the new task.

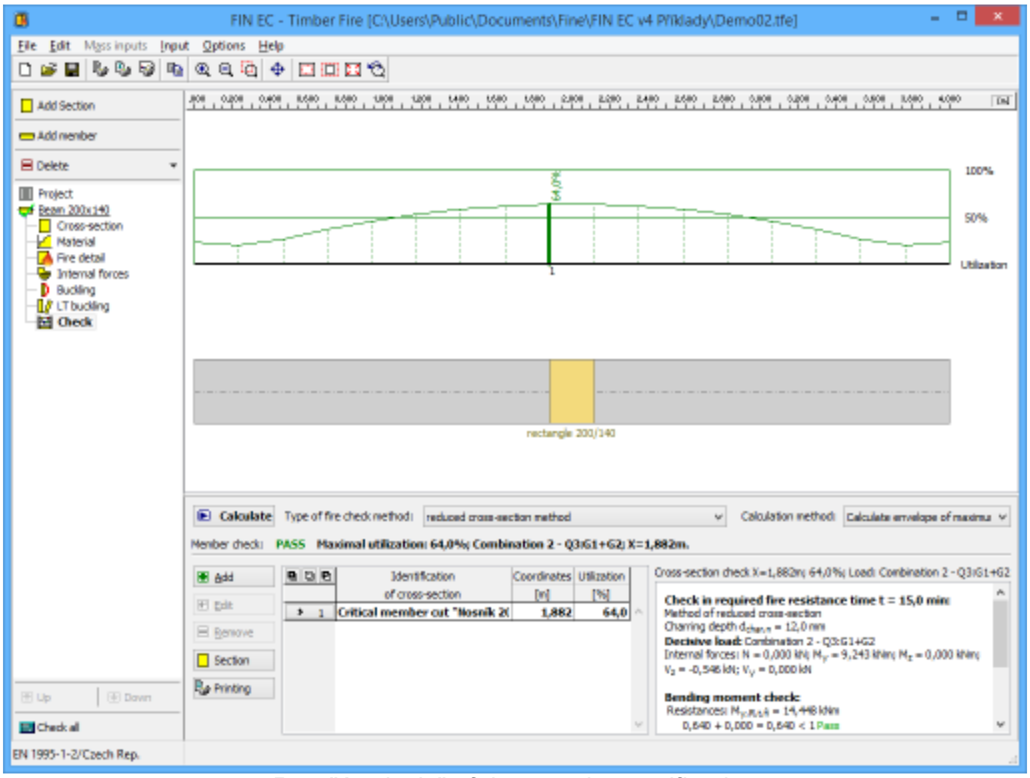
**Add**
- Inserts a new section on member. The detailed results can be displayed for this section. Properties of the new section have to be specified in the window "**New section for analysis**".

**Edit**
- Edit of the active cross-section properties (name, coordinate).

**Remove**
- Remove the active section from the list.

**Printing**
- Print results for all entered sections using printing window.

The new section for analysis can be also added using active workspace. New section can be added by double-click on the needed position on the member.




*Part "Analysis" of the member verification*

# Beam

Task type "**Beam**" is suitable for the comprehensive analysis of general horizontal beam. Only single axis bending is supported in this task type.

The main frame of beam analysis contains these inputs:

**Member length**
- Total member length specified in metres.

**Limit fire resistance period**
- Fire resistance in minutes, the verification is performed for this time.

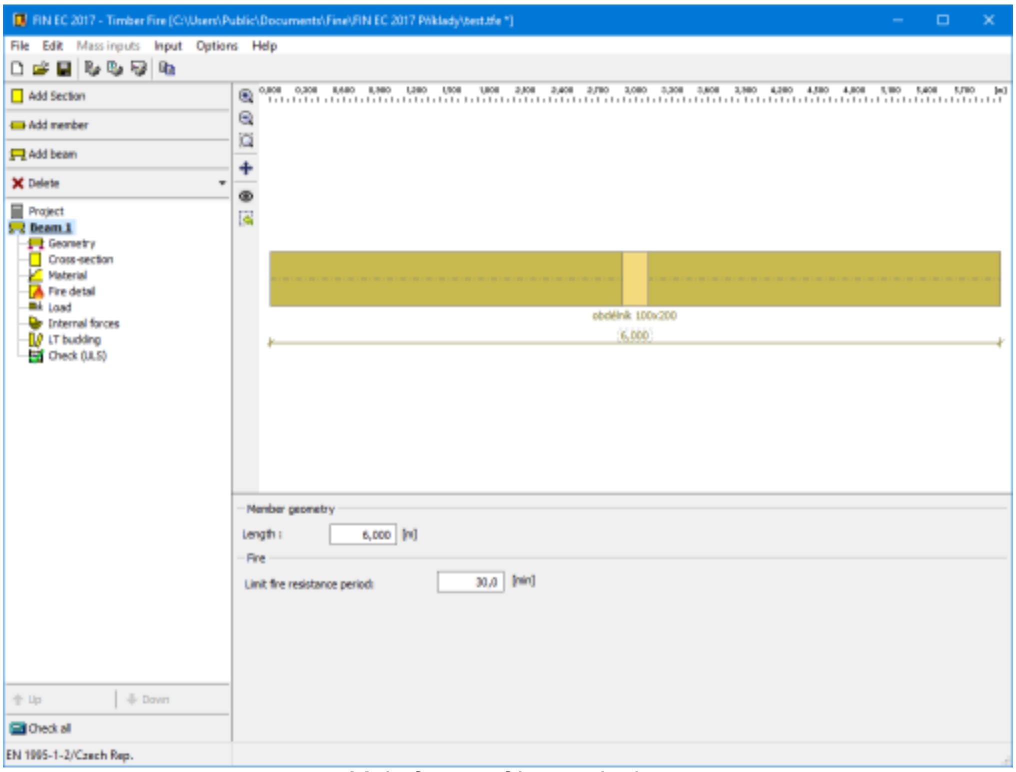
The beam design contains these parts:

- **Geometry**
- **Section**

- **Material**
- **Fire detail**
- **Load**
- **Internal forces**
- **Buckling**
- **LT buckling**
- **Analysis**

General work with beams (addition, manipulation) is described in the chapter "**Tree menu**".



*Main frame of beam design*

# Program Masonry

The software "**Masonry**" verifies walls and columns according to EN 1996-1-1.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". The tree menu can be alternated by the part "**Input**" of the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

Two types of particular tasks can be used in the software:

| | |
|---|---|
| **Wall** | • Verification of masonry wall loaded by normal and shear force and bending moment. |
| **Column** | • Verification of masonry column loaded by normal force and biaxial bending. |

These tasks can be added with the help of the buttons "**Add wall**" and "**Add column**" in the heading of the tree menu.

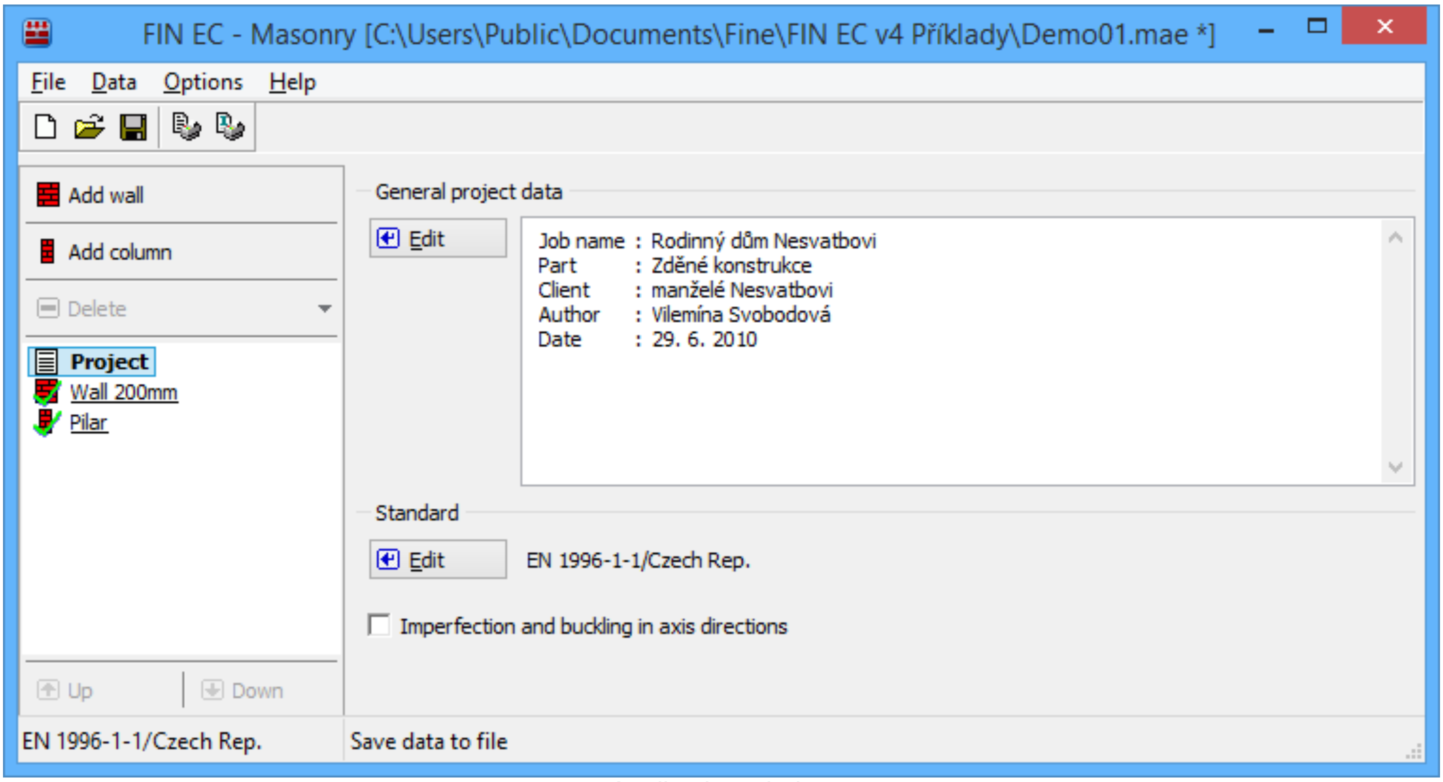
## Main screen

Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

The design standard including national annex can be changed in the part "**Standard**". These settings are placed in the window "Standard selection", that can be launched by the button "**Edit**". Setting "**Imperfection and buckling in axis directions**" adds an additional way of buckling verification for columns. This setting should be used only in special occasions and isn't recommended for general work.

Analysis and verifications used in the software "**Masonry**" are described in the theoretical part of the help.



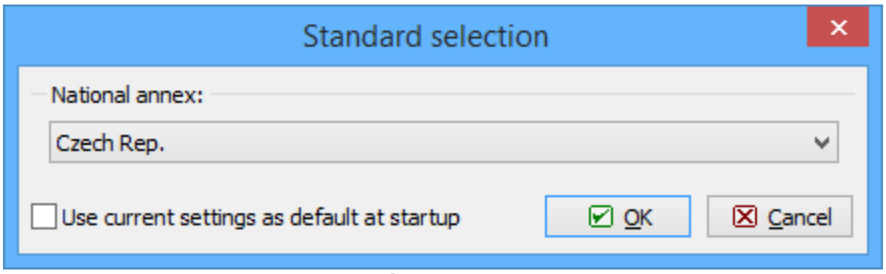
*Application window*

## Standard selection

The national annex for the standard EN 1996-1-1 can be selected in this window. Option "**Default EC**" sets the partial factors according to the design standard without any national annex.

Button "**Default**" contains these two tools:

| | |
|---|---|
| **Adopt default settings** | • Set the default values for all parameters. |
| **Save settings as default** | • Set entered parameters as defaults for new projects. |

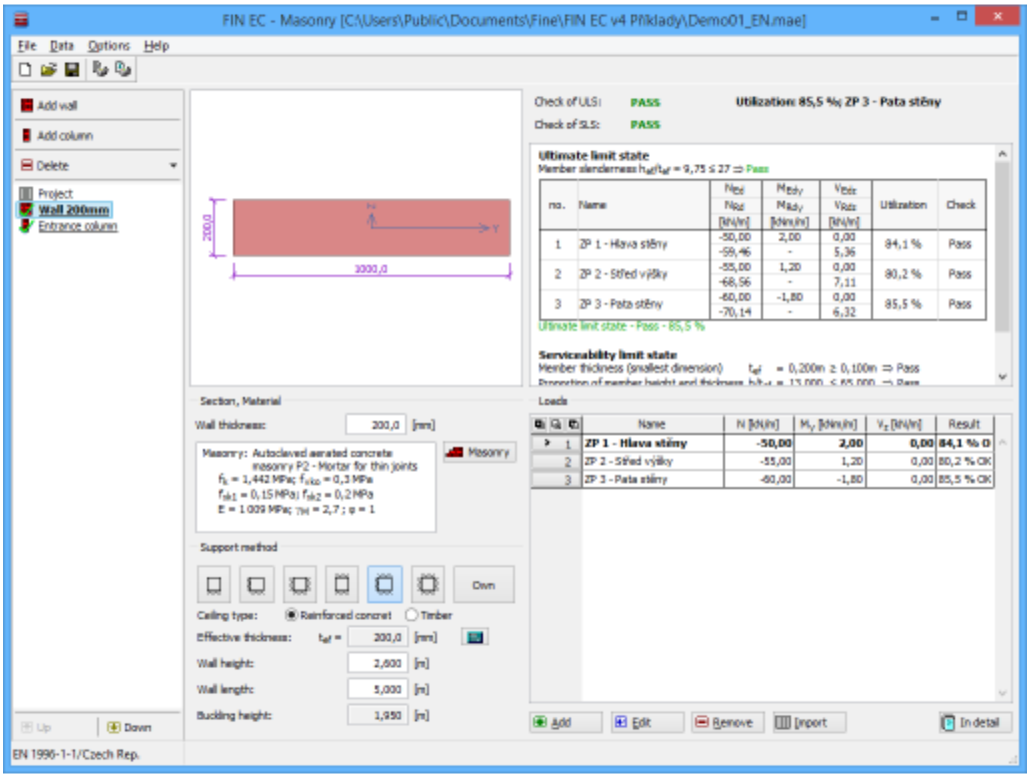


*Window "Standard selection"*

## Wall

Task type "**Wall**" is suitable for the verification of the wall, that is loaded by unlimited number of loads. The one linear meter of the wall is considered during the design. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".




*Task "wall"*

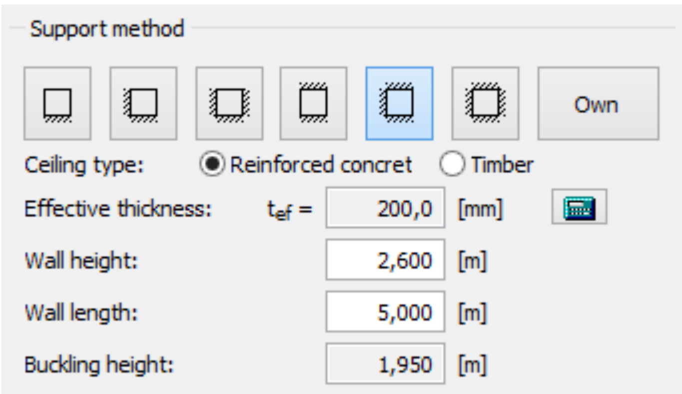
The window contains these parts:

## Section, Material

The wall thickness and material can be input in this part. Material characteristics of blocks and mortar are placed in the window "**Masonry**", that can be launched using the button of the same name.

## Buckling

The parameters for the calculation of the buckling length are included in this part. The supporting type can be selected with the help of dedicated buttons. The appropriate formula from chapter *5.5.1.2* of EN 1996-1-1 is selected according to this supporting method. The ceiling type has to be selected for walls supported at the top and at the bottom. The appropriate reduction factor $\rho_n$ is selected according to this selection. Ceiling made of RC slab should be selected only for cases, where the RC slabs are situated from both sides or from one side with bearing of at least *2/3* thickness of the wall. Also real wall height and length (for walls supported on sided) are necessary for the calculation of the buckling height. The arbitrary value of buckling height can be specified manually when using "**Own**" button. The calculation of buckling height is described in the chapter "**Effective height of masonry walls**" of theoretical part of the help.




*Part "Support method"*

## Loads

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

Loads can be also imported from text or **.csv* file. This feature can be used for import of large number of loads, that were calculated with the help of another structural engineering program. Import can be performed using window "**Load import**", that can be launched by button "**Import**".

## Results

This part contains analysis results for all entered loads. Results for ultimate limit state are arranged in the table. The entered internal forces (index starting with *E*) and calculated resistances (index starting with *R*) are displayed for every load. All forces that exceed the corresponding resistance are highlighted by red colour. Last column contains final result of the verification (pass/fail). Serviceability verification contains geometry controls including final result (pass/fail).

Detailed results can be displayed for active load using "**In detail**" button. The results are shown in new window. The text can be copied into clipboard using *Ctrl+C* and can be inserted into another document.

Methods and formulas applied during the analysis are described in chapters "**Ultimate limit state**" and "**Serviceability limit state**" of the theoretical part of the help.

**Ultimate limit state**

Member slenderness $h_{ef}/t_{ef}$ = 9,75 ≤ 27 ⇒ Pass

| no. | Name | $N_{Ed}$ / $N_{Rd}$ [kN/m] | $M_{Edy}$ / $M_{Rdy}$ [kNm/m] | $V_{Edz}$ / $V_{Rdz}$ [kN/m] | Utilization | Check |
|---|---|---|---|---|---|---|
| 1 | ZP 1 - Hlava stěny | -50,00 | 2,00 | 0,00 | 84,1 % | Pass |
| | | -59,46 | - | 5,36 | | |
| 2 | ZP 2 - Střed výšky | -55,00 | 1,20 | 0,00 | 80,2 % | Pass |
| | | -68,56 | - | 7,11 | | |
| 3 | ZP 3 - Pata stěny | -60,00 | -1,80 | 0,00 | 85,5 % | Pass |
| | | -70,14 | - | 6,32 | | |

Ultimate limit state - Pass - 85,5 %

**Serviceability limit state**

Member thickness (smallest dimension) $t_{ef}$ = 0,200m ≥ 0,100m ⇒ Pass

Proportion of member height and thickness $h/t_{ef}$ = 13,000 ≤ 65,000 ⇒ Pass

Proportion of member length and thickness $l/t_{ef}$ = 25,000 no limit

Serviceability limit state - Pass

*Results view for wall*

# Masonry

Material characteristics of masonry can be specified in this window. The characteristics can be obtained from manufacturers' catalogues or can be specified according to the requirements of designer. These inputs are necessary for determination of strength properties:

## Masonry units

**Catalogue**
- List of the most common manufacturers of masonry units. Arbitrary properties of masonry units can be selected for group "**General**".

**Product**
- The production range of selected manufacturer is displayed here. Following properties are filled automatically according to the manufacturer's documentation.

**Kind**
- Material of masonry units can be selected for "**General**" group in "**Catalogue**". This setting affects the determination of strength properties.

**Group**
- Group in accordance to the paragraph *3.1.1* of EN 1996-1-1 can be selected for "**General**" group in "**Catalogue**". Autoclaved aerated concrete, manufactured stone and dimensioned natural stone units areconsidered to be *Group 1*. Clay, calcium silicate and aggregate concrete units are sorted into *Groups 2-4* in accordance with table *3.1* in EN 1996-1-1.

**Category**
- Category in accordance to the appropriate standard (EN 771-1 až EN 771-6) can be selected for "**General**" group in "**Catalogue**".

**Strength $f_b$**
- Value of the normalised mean compressive strength of masonry units. This value can be declared by the manufacturer or can be obtained by converting the compressive strength by using given procedure.

**Load acts parallel to bed joints**
- This setting affects the calculation of characteristic value compressive strength of masonry in accordance with paragraph *3.6.1.2.(3)*.

**Perpend joints unfilled**
- This setting affects the calculation of characteristic value compressive strength of masonry in accordance with paragraph *3.6.1.2.(5)*.

**Masonry with longitudinal joint**
- Factor *K* is reduced by *0,8* for masonry with mortar joint parallel to the face of the wall through all or any part of the length of the wall par. *3.6.1.2.(6)*).

## Mortar

**Kind**
- Mortar type that affects formula for calculation of compressive strength $f_k$ and also value of factor $K$.

**Mortar type**
- Selection whether the composition and manufacturing method is chosen in order to achieve specified ( "**designed mortar**") or whether the properties of which are assumed from the stated proportions of the constituents ("**prescribed mortar**"). This setting influences value of partial factor $\gamma_M$.

**Strength $f_m$**
- Compressive strength of mortar

**Shell bedded masonry**
- Setting for determination of strength reduction due to shell bedded masonry, where the units are bedded on two or more equal strips of general purpose mortar (paragraph *3.6.2.(5)* of standard EN 1996-1-1). The ratio *g/t* has to be input, where *g* is the total width of the mortar strips and *t* is the thickness of the wall.

Strength properties of masonry are calculated according to these inputs. Determination of material properties is described in the chapter "**Material characteristics**" of theoretical part of the help. The material properties can be also specified manually with the help of setting "**Own values**". These characteristics can be entered:

$f_k$
- characteristic value compressive strength of masonry

$f_{vko}$
- characteristic value of initial shear strength of masonry, under zero compressive stress

$f_{xk1}$
- Characteristic value of flexural strength of masonry having the plane of failure parallel to the bed joints

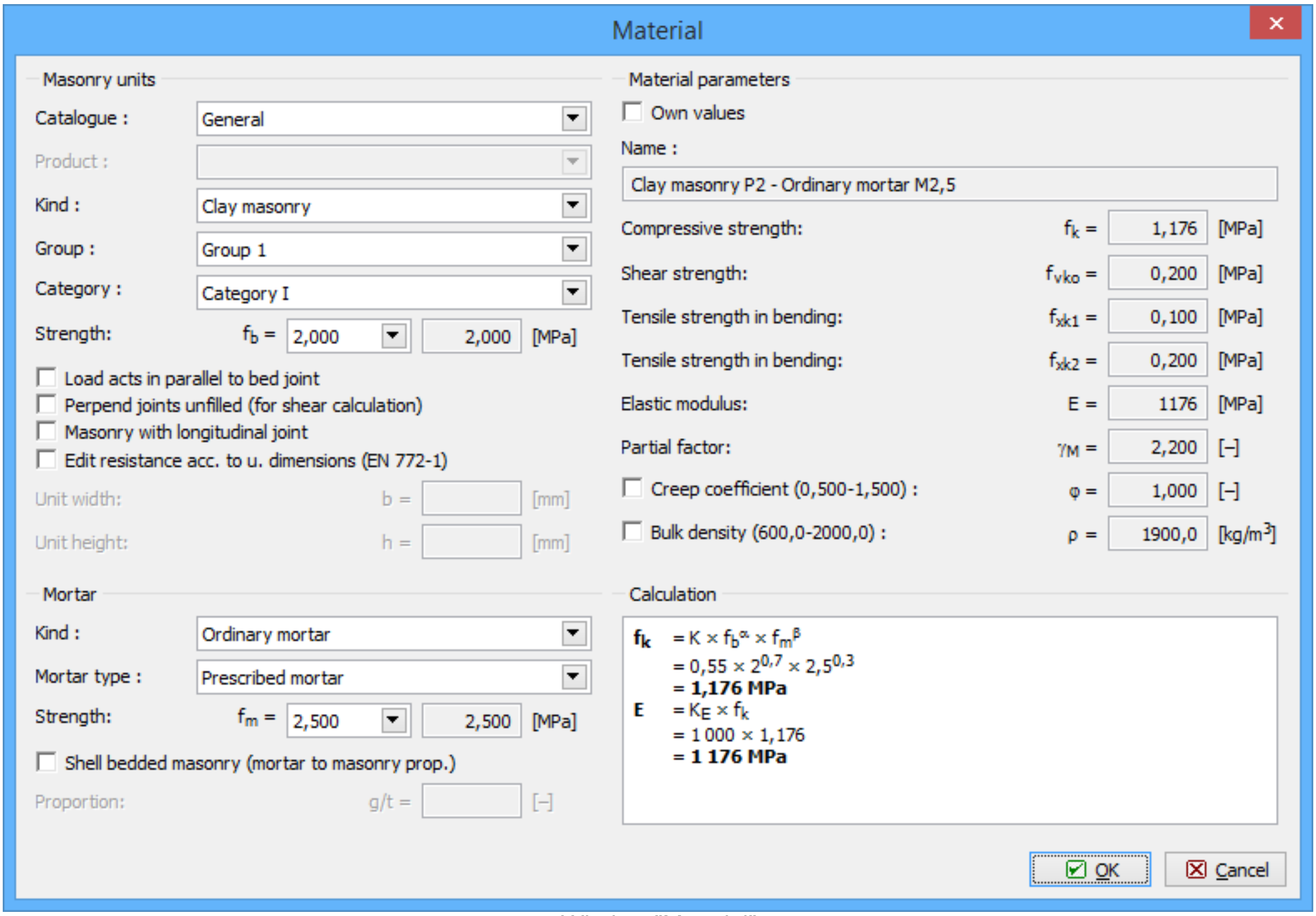
$f_{xk2}$
- Characteristic value of flexural strength of masonry having the plane of failure perpendicular to the bed joints

$E$
- Elastic modulus

$\gamma_M$
- Partial material factor

$\varphi_M$
- Creep coefficient, default values are calculated as an average of the ranges specified in the note for the chapter *3.7.4(2)*

$\rho$
- Bulk density, this value is used for calculation of self-weight for load type "**Vertically loaded wall/column**". Default values are based on production catalogues' of producers

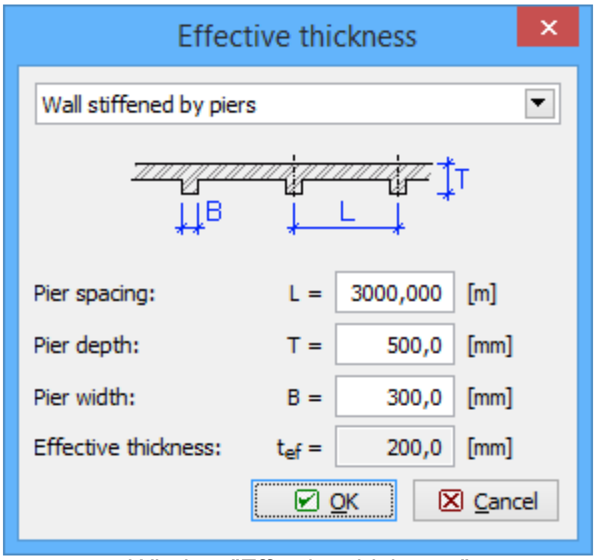



*Window "Material"*

## Effective thickness

The calculation of the effective wall thickness can be changes in this window. Effective thickness is used for analysis of ultimate limit state. These options are available:

- **Own** - the value of the effective thickness of the wall can be entered manually
- **Simple wall** - the real thickness of the wall is used as the effective one
- **Wall stiffened by piers** - effective thickness of the wall restrained by piers is calculated in accordance with chapter 5.5.1.3(2) of standard EN 1996-1-1



*Window "Effective thickness"*

## Load edit

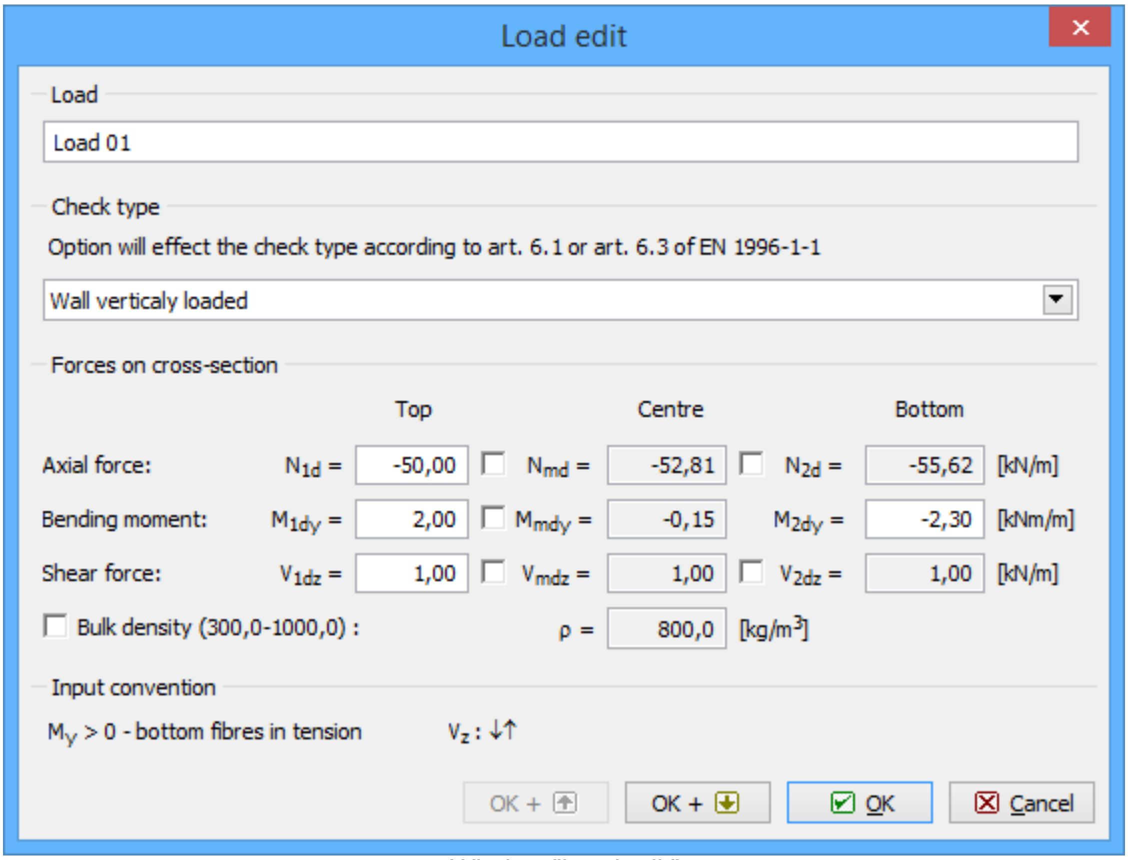
The internal forces and check type for individual load case can be input in this window. The appropriate equations for verification are selected according to the specified check type. These options are available:

| | |
|---|---|
| **Wall vertically loaded** | • This verification style is suitable for the verification of whole structural member that is loaded mainly by vertical load. Verification is performed at underside of floor, at mid height and at top of floor (chapter *6.1.2.* of EN 1996-1-1). |
| **Wall vertically loaded - top** | • This verification style is suitable for the verification of structural member at underside of floor (chapter *6.1.2.* of EN 1996-1-1). Member should be loaded mainly by vertical load. The eccentricity is calculated using equation *(6.5)* . |
| **Wall vertically loaded - centre** | • This verification style is suitable for the verification of structural member at mid height (chapter *6.1.2.* of EN 1996-1-1). Member should be loaded mainly by vertical load. The eccentricity is calculated using equation *(6.7)* . |
| **Wall vertically loaded - bottom** | • This verification style is suitable for the verification of structural member at top of floor (chapter *6.1.2.* of EN 1996-1-1). Member should be loaded mainly by vertical load. The eccentricity is calculated using equation *(6.5)* . |
| **Wall laterally loaded - bending about horizontal axis** | • This verification style is suitable for the verification of laterally loaded structural members (chapter *6.3.1* of EN 1996-1-1). The failure in plane parallel to bed joints is expected for this case (figure *3.1a* in EN 1996-1-1). The flexural strength having a plane of failure parallel to the bedjoints $f_{xk1}$ will be used in analysis. |
| **Wall laterally loaded - bending about vertical axis** | • This verification style is suitable for the verification of laterally loaded structural members (chapter *6.3.1* of EN 1996-1-1). The failure in plane perpendicular to bed joints is expected for this case (figure *3.1b* in EN 1996-1-1). The flexural strength having a plane of failure perpendicular to the bed joints $f_{xk2}$ will be used in analysis. |

All internal forces at underside of floor, at mid height and at top of floor can be input using load type "**Wall vertically loaded**". Some values can be calculated automatically using these rules:

- normal forces at mid height of wall $N_{md}$ and at top of floor $N_{2d}$ are calculated as a sum of normal force at underside of floor $N_{1d}$ and self-weight of wall or column. Self-weight is calculated using cross-sectional area, material density (can be input in the window "**Masonry**" or using the input line "**Bulk density**" in this window) and partial load factor $\gamma_G = 1{,}35$ .
- shear forces at mid height of wall $V_{mdz}$ and $V_{mdy}$ and at top of floor $V_{2dz}$ and $V_{2dy}$ use the values of shear forces $V_{1dz}$ and $V_{1dy}$ that were specified at underside of floor.

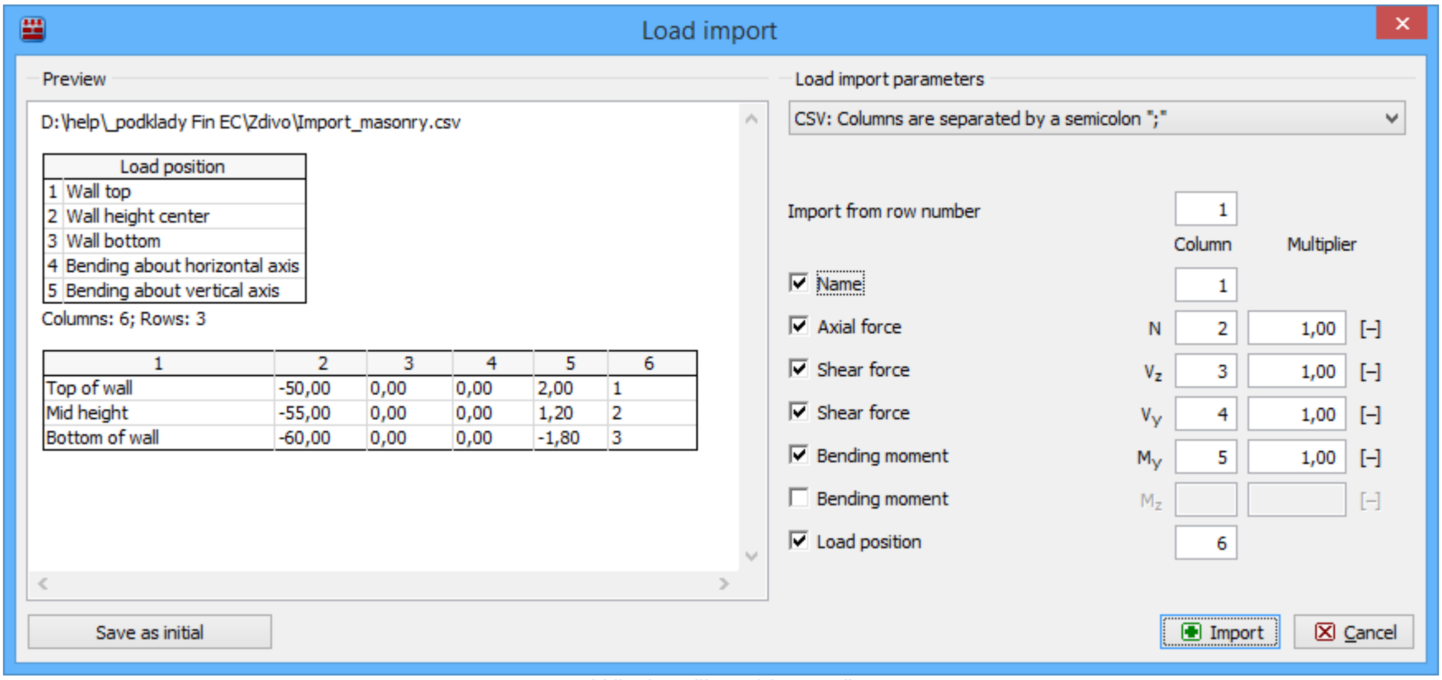
- Bending moments at mid height of wall $M_{mdz}$ and $M_{mdy}$ are automatically calculated using linear interpolation between bending moments at top of floor $M_{2dz}$, $M_{2dy}$ and at underside of floor $M_{1dz}$, $M_{1dy}$.




*Window "Load edit"*

## Load import

Window "**Load import**" appears after loading the text or **.csv* source file. The data included in the source file can be arranged in this window. Left part shows content of imported file. The assignment of the certain column to the load entry (force or property) can be done in the right part of the window. Numerical entries can be also multiplied by specified multiplier. This multiplier can be used mainly for conversion caused by different units in the source file and in the program. Initial rows can be skipped using setting "**Import from row number**". This setting is helpful for source files with headings. Default units required by the software are *[kN]* and *[kNm]*.



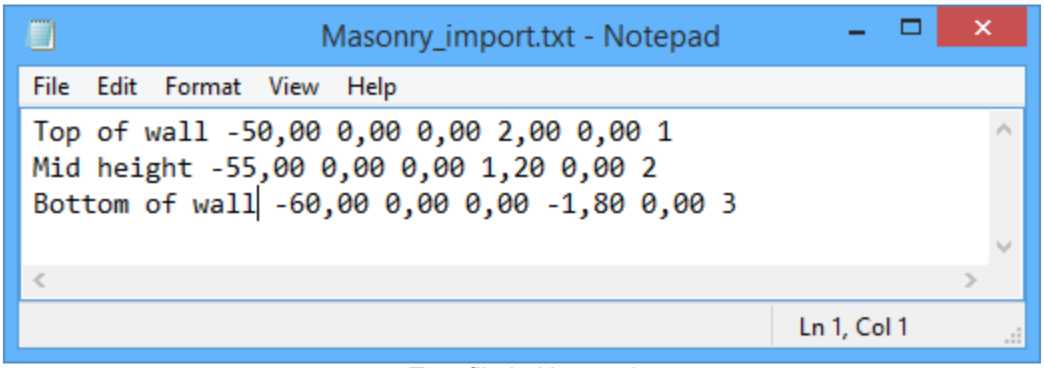

*Window "Load import"*

## Preparing text file

Text file can be created in any text editor (e.g. *Notepad*, *Word*, *Writer*). File format requires that every row contains one load case. Every row can contain all values of internal forces separated by space or tabulator. The order may differ comparing to the order of forces in the software, however, order has to be identical for all rows of the document. The load name and check type (verification style) may be also specified for every load. The check type "**Vertically loaded wall**" isn't supported in the import. Such load has to be separated into three different loads "**Top of wall**", "**Mid height**" and "**Bottom of wall**". The check type has to be specified using this numerical codes:

| | |
|---|---|
| 1 | Top of wall |
| 2 | Mid height of wall |
| 3 | Bottom of wall |
| 4 | Bending about horizontal axis |
| 5 | Bending about vertical axis |

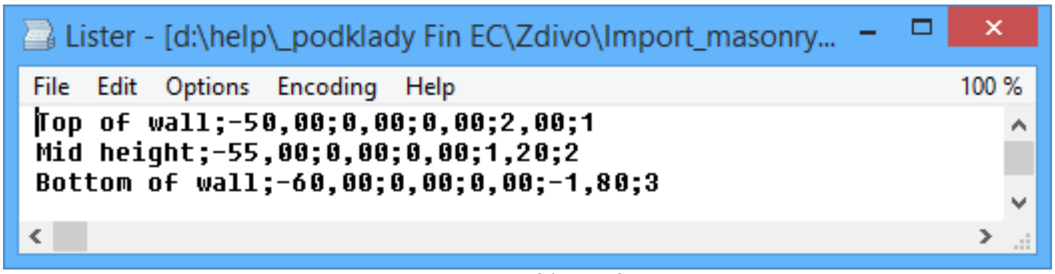
The file can be also created using part of analysis documentation from arbitrary structural analysis software.




*Text file in Notepad*

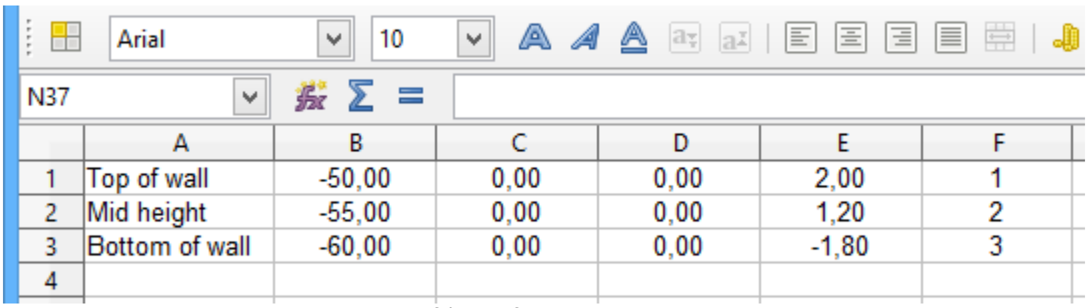
## Preparing **.csv* file

The rules for **.csv* (comma-separated values) files are almost identical . Main difference is, that the particular values are separated by semicolon ";".




*Example of *.csv file*

This file type can be easily created using spreadsheet programs like *Excel* or *Calc*. Created document can be saved as **.csv* file with appropriate separator.

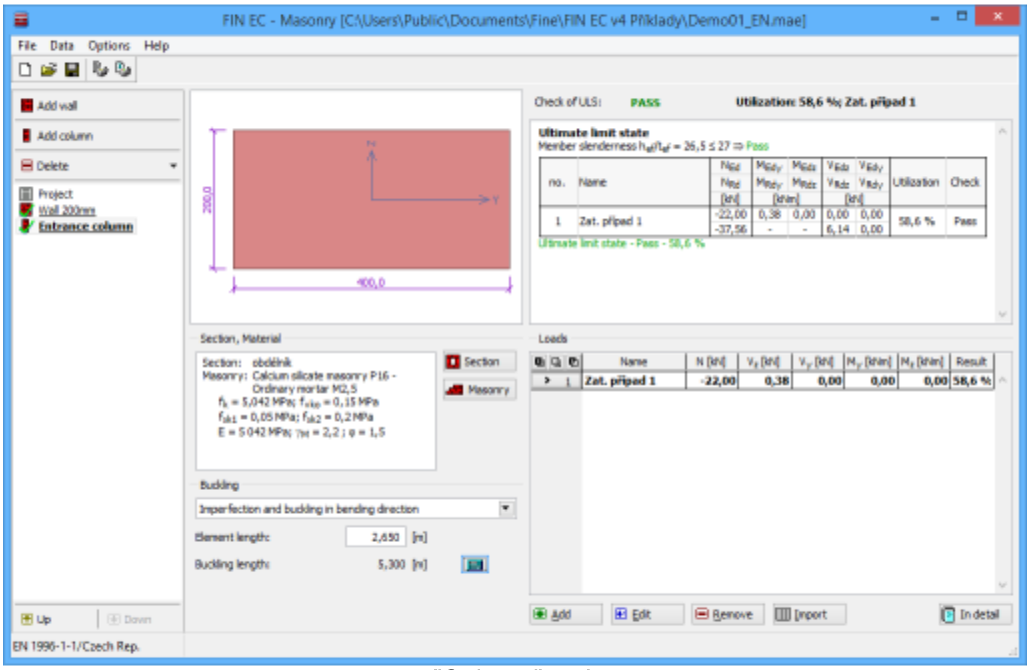



*Edit of *.csv file in spreadsheet*

# Column

Task type "**Column**" is suitable for the verification of the masonry pillar, that is loaded by unlimited number of loads. General work with particular tasks of the project (addition, manipulation) is described in the chapter "**Tree menu**".




*"Column" task*

The window contains these parts:

## Section, Material

The cross-section geometry and material characteristics can be specified in this part. These properties can be input using these buttons:

| | | |
|---|---|---|
| **Section** | • | Input of column cross-section shape and dimensions with the help of predefined database |
| **Masonry** | • | Input of material characteristics of blocks and mortar in the window "**Masonry**" |

The cross-section preview is active, the window for cross-section edit can b launched by clicking on the preview.

## Buckling

The method of buckling analysis will be available if the setting "**Imperfection and buckling in axis directions**" is switched on on the initial screen of the program. These methods can be used:

| | | |
|---|---|---|
| **Imperfection and buckling in the bending direction** | • | Buckling verification and imperfection calculation are performed in the bending direction. The bending direction is defined by vector sum of bending moments in directions $y$ and $z$. This method may provide non-standard results in certain cases. |
| **Imperfection and buckling in the axis directions** | • | Buckling verification and imperfection calculation are performed separately for both directions $y$ and $z$. Recommended method. |

This part also includes input line "**Element length**" for the input of basic member length. This length is used for the calculation of buckling lengths in the stability analysis. The buckling lengths are calculated with the help of buckling factors specified in the window "**Buckling length determination**". This button can be launched by "" button.

## Loads

This part contains list of loads (combinations of internal forces and moments), that are checked during the verification. Loads can be added in the table using buttons "**Add**", "**Modify**" and "**Remove**". Table shows the most important information for each load (mainly internal forces and result of analysis). Load properties are entered with the help of window "**Load edit**".

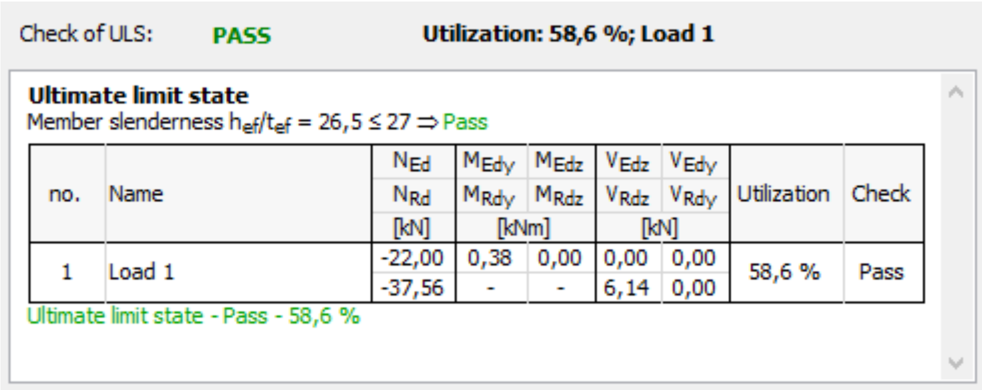
Loads can be also imported from text or **.csv* file. This feature can be used for import of large number of loads, that were calculated with the help of another structural engineering program. Import can be performed using window "**Load import**", that can be launched by button "**Import**".

## Results

This part contains analysis results for all entered loads. Results for ultimate limit state are arranged in the table. The entered internal forces (index starting with $E$) and calculated resistances (index starting with $R$) are displayed for every load. All forces that exceed the corresponding resistance are highlighted by red colour. Last column contains final result of the verification (pass/fail).

Detailed results can be displayed for active load using "**In detail**" button. The results are shown in new window. The text can be copied into clipboard using *Ctrl+C* and can be inserted into another document.

Analysis is described in the chapter "**Ultimate limit state**" of the theoretical part of the help.



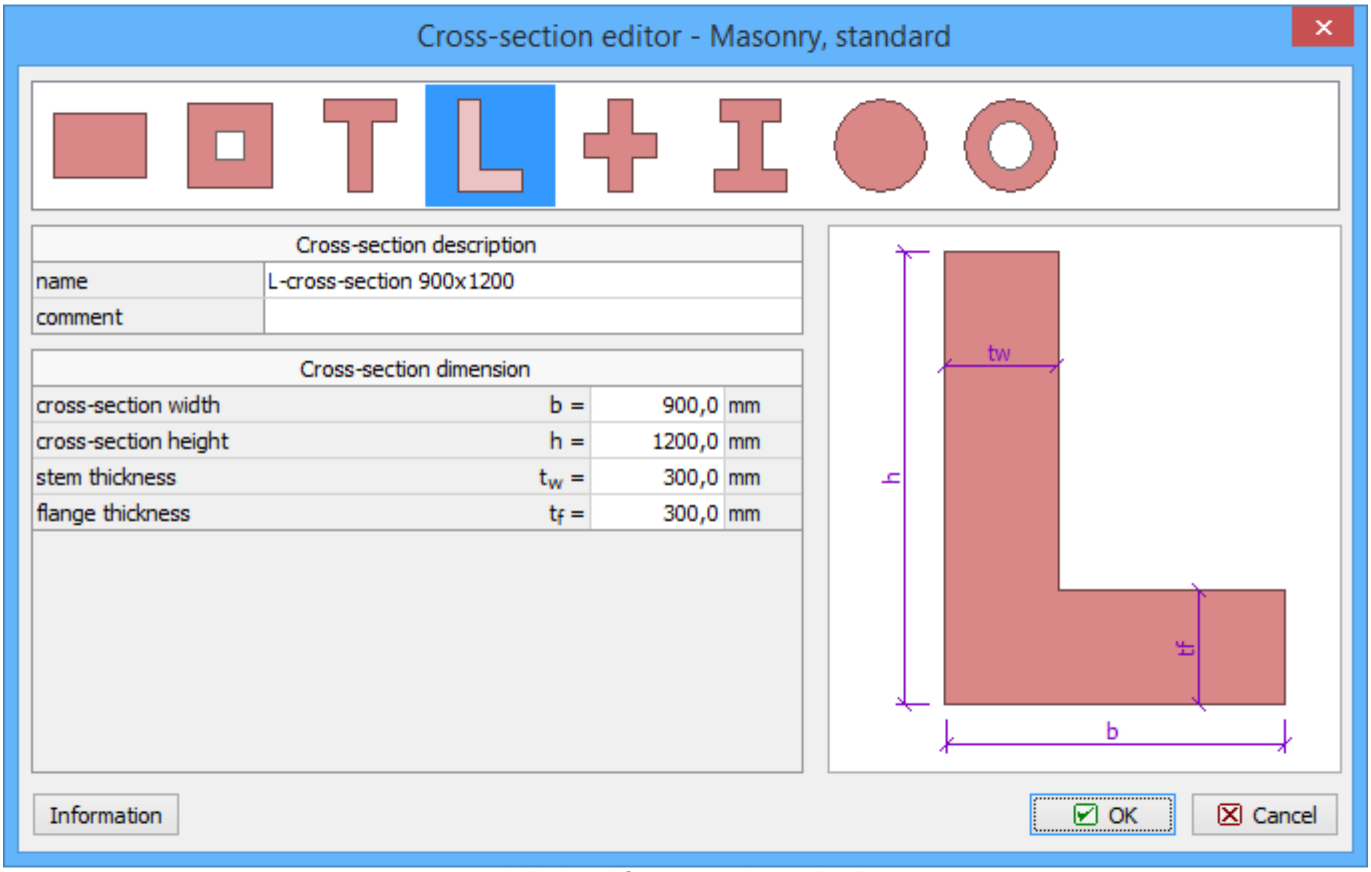

| no. | Name | $N_{Ed}$ / $N_{Rd}$ [kN] | $M_{Edy}$ / $M_{Rdy}$ [kNm] | $M_{Edz}$ / $M_{Rdz}$ [kNm] | $V_{Edz}$ / $V_{Rdz}$ [kN] | $V_{Edy}$ / $V_{Rdy}$ [kN] | Utilization | Check |
|---|---|---|---|---|---|---|---|---|
| 1 | Load 1 | -22,00 / -37,56 | 0,38 / - | 0,00 / - | 0,00 / 6,14 | 0,00 / 0,00 | 58,6 % | Pass |

*Part "Results" of column verification*

## Cross-section editor

The member cross-section can be modified in this window. The upper part contains library of available. Dimensions can be entered in the table in the left part of the window. The meaning of dimensions is shown in the cross-section view in the right part of the window.

"**Information**" button in the left bottom corner shows complete list of cross-sectional characteristics.

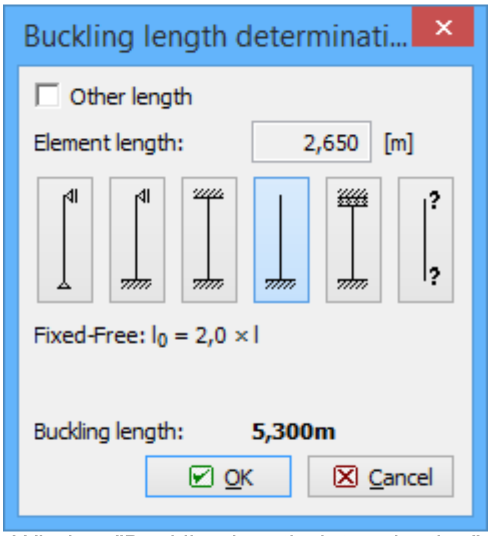



*Window "Cross-section editor"*

## Buckling length determination

The factor for calculation of the effective member length for buckling analysis can be specified in this window. The factor is selected according to the support style of the member. The check box "**Other length**" gives an option for entering a basic member length that differs from the member length specified on the main screen of the task. Range of supporting styles and factor values are based on general theories for stability analysis. Arbitrary value of the factor can be specified for the last option.




*Window "Buckling length determination"*

# Program Loading

The software "**Loading**" is able to create load reports in accordance with standards EN 1991-1-1, EN 1991-1-3 a EN 1991-1-4.

## User interface

The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for the administration of individual project tasks as well as for switching between parts of an input. The work with the tree menu is described in the chapter "**Tree menu**". Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

These types of reports can be created in the software:

**Area** • The load report that consists of more items (e.g. self weight of roof structure). The particular items of the report may be arbitrary construction materials, structural elements, variable loads or user defined item. Resulting units are $kN/m^2$.

**Linear** • The load report similar to the type "**Area**", the load is recalculated to one liner meter. Resulting units are $kN/m$.

**Point** • The load report similar to the type "**Area**", the load is recalculated to specified area. Result is a force in $kN$.

**Snow** • The report of snow load for selected type of structure (different roof types, local effects). As a result, load report contains design values including transparent figure.

**Wind** • The report of wind load for selected type of structure (different roof types, walls). As a result, load report contains design values including transparent figure.

All particular tasks create an individual report of certain part of structural design (self weight of structural part, snow load for roof, wind load for walls etc.). Any report may be converted into linear or point load with the help of "**localization**". Any task may contain unlimited number of localizations. Localizations are automatically updated after any modification of the original report. Reports can be added with the help of the button "**Add**" in the heading of the tree menu.

## Main screen

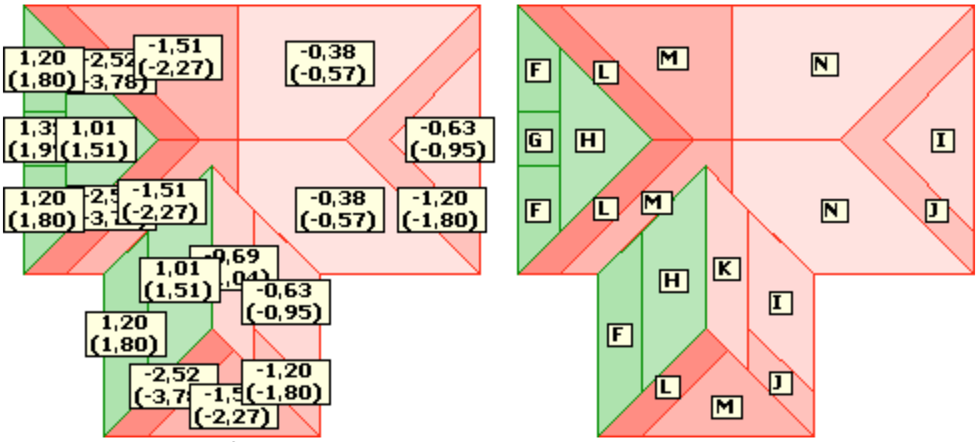
Basic screen of the software contains general data of the project (identification details, design standard).

Frame "**General project data**" shows data, that can be input in the window "**General project data**". The window can be opened by using the button "**Edit**". The entered data can be used in heading and footing of documents.

This screen also contains an option for changing the national annex for design standards. National annexes affect certain calculations and also loading maps are selected according to this setting. The national annex "**Default EC**" performs the design according to the fundamental Eurocode without any national annex.

The following setting "**Values of wind load on roof in table**" avoids non-transparent reports for more complicated roofs. When using this setting, such cases may contains the roof scheme only with region marks, load values will be organized in the table under the roof scheme.

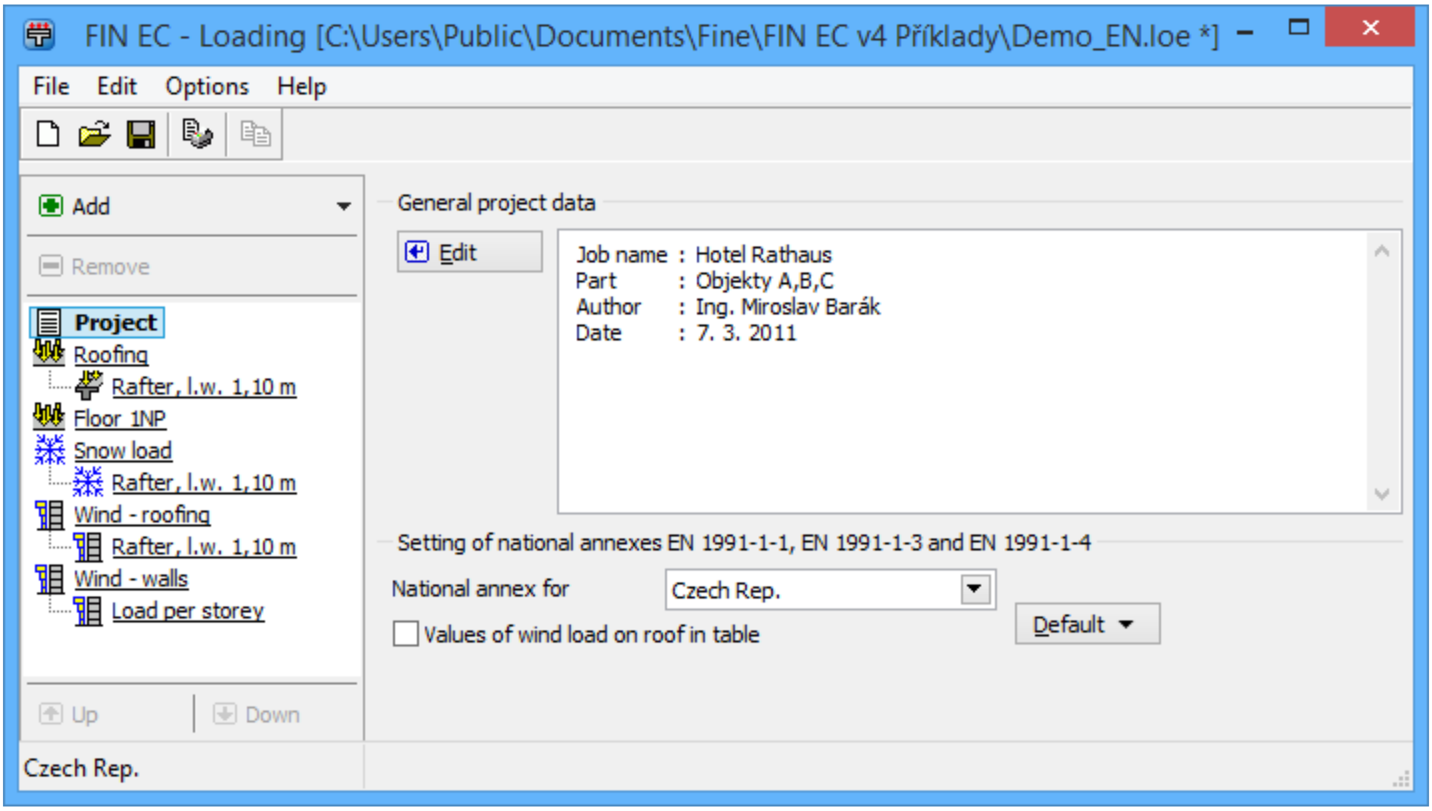



*Roof plan with displayed load values or region marks*

Button "**Default**" contains these two tools:

**Adopt default settings** • Set the default values for all parameters.

**Save settings as default** • Set entered parameters as defaults for new projects.




*Main application window*

## Area, Linear and Point loads

Load reports for area, linear and point loads are suitable for calculation of load that consists of more items (e.g. self weight of roof structure). The report structure is displayed in the right part of the window, functions for the work with the report are organized into toolbar that is placed along the tree menu. The particular items of the report may be arbitrary construction materials, structural elements, variable loads or user defined item. Resulting units are *kN/m²* (area load), *kN/m* (linear load) or *kN* (point load).

The toolbar for the report contains following buttons:

**Report** • The name, description and content (sorting, summaries etc.) of the report may be specified in the window "**Report settings**" with the help of this button

**Add** • This button adds a new item into the report. Item parameters has to be specified in the window "**New report item**".

**Insert** • This button adds also a new item into the report, the item is inserted before the active one (highlighted by blue colour) in the report.

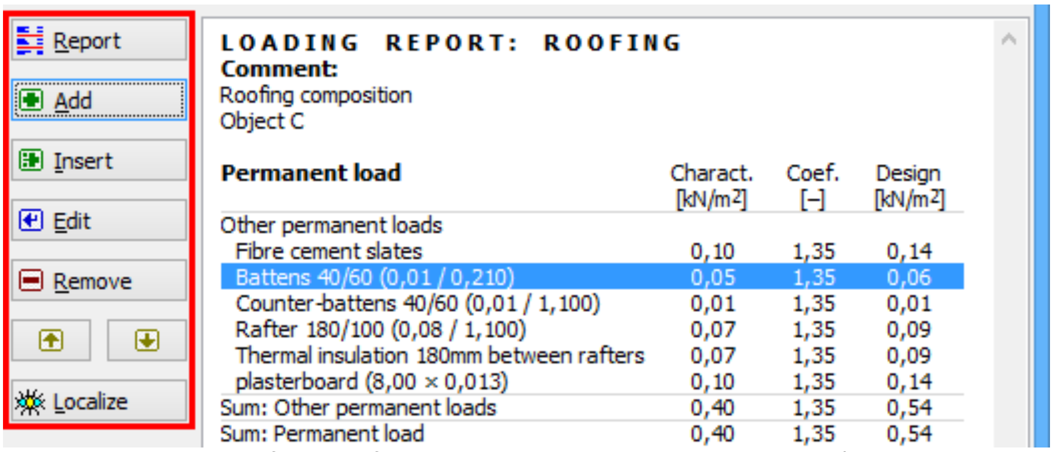
 

**Edit**
- This button launches the window "**Edit report item**" for modification of the active report item. Alternatively, it is possible to use double-click on the item in report.

**Remove**
- This button removes an active item (highlighted by blue colour) from the report

**Up/Down**
- The order of items in the report may be changed with the help of these buttons

**Localize**
- This button creates a new report, that contains all items from the active one. Localization may be of the same or lower type (any type of localization may be created for report type "**Area**", linear or point localization may be created for report type "**Linear**", only point localization is permitted for type "**Point**"). Localizations may be used for calculation of the load for linear element (rafter, beam) in planar structure. As the localization may contain also new items in the list, they may be also used for making the load reports with identical basis (e.g. roofing composition) and different additional parts (variable load). Properties of the localization are organized in the window "**Load localization**".




*Toolbar for edit of report items, active item is "Battens 40/60"*

## Report settings

This window contains a name and description of the load report and also settings that may modify the final content of the report:

**Sort according to type (permanent, variable...)**
- Sorting of items according to the selected "**Load type**" ("**Permanent**", "**Variable**", "**Prestress**", "**Accidental**"). The load type may be defined for items in the window "**Edit report item**". Otherwise, the order depends on the user's input.

**Sort according to category**
- Sorting of items according to the selected "**Category**".

**Recapitulation of permanent loads**
- Recapitulation of permanent loads according to the "**Category**" (may be defined for items in the window "**Edit report item**").

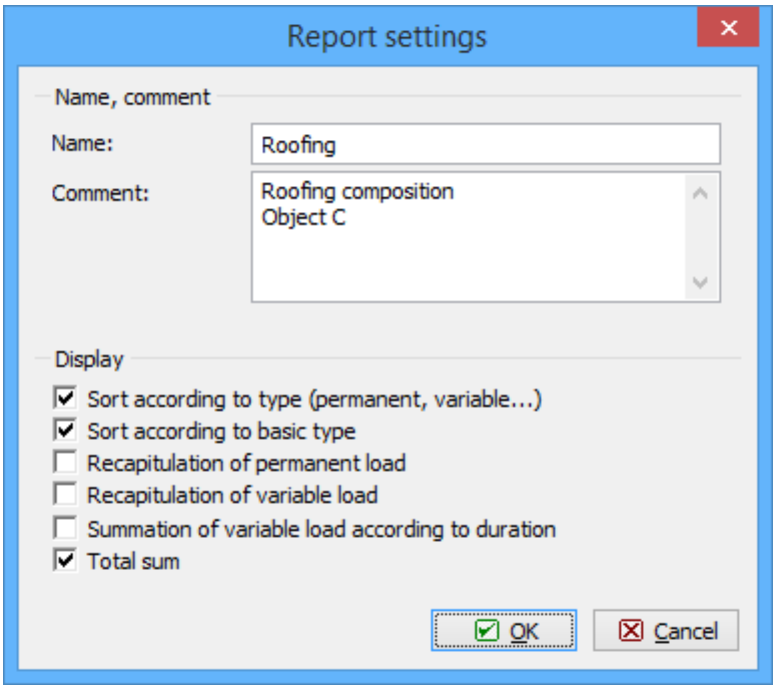
**Recapitulation of variable loads**
- Recapitulation of variable loads according to the "**Category**" (may be defined for items in the window "**Edit report item**").

**Summation of variable load according to duration**
- Recapitulation of variable loads according to the load duration (may be defined for items in the window "**Edit report item**").

**Total sum**
- The sum of permanent and variable loads

This window also contains input field for loading/distribution width or area when launched for a localization.




*Window "Report settings"*

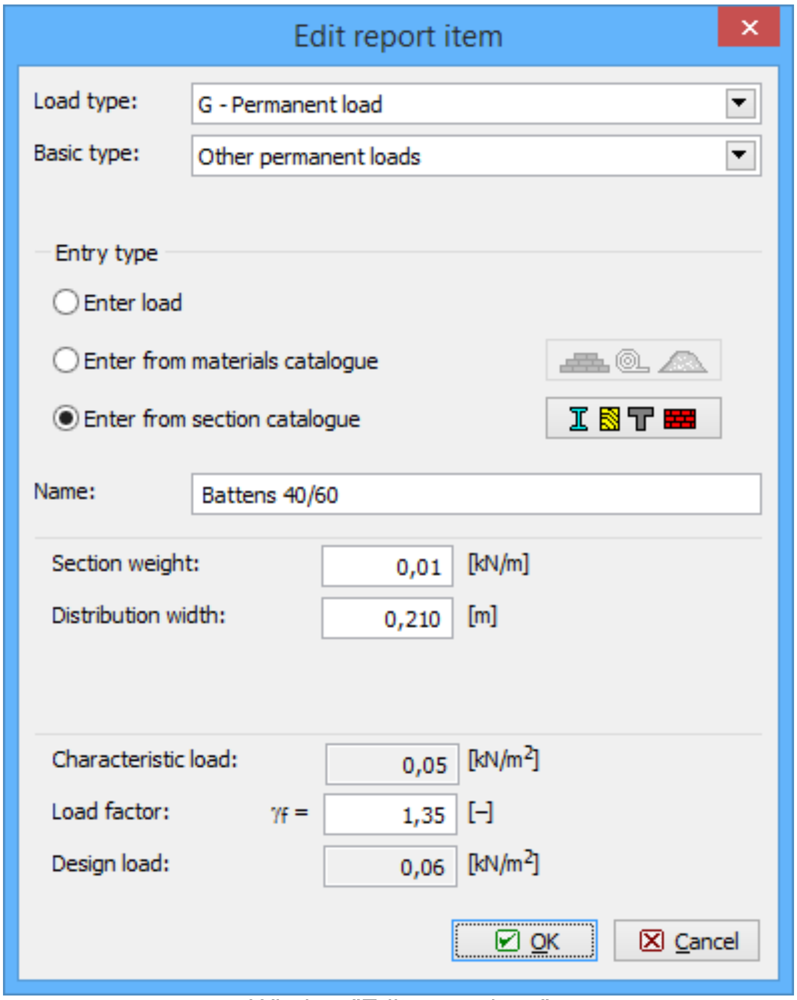
## Edit report item

This window contains parameters of individual report item. Upper part contains sorting of the item. The "**Load type**" ("**Permanent**", "**Variable**", "**Prestress**", "**Accidental**") defines the default value of the factor $\gamma$ and also is used for the basic sorting of the report. The "**Category**" may be used for detailed sorting of items in the list. This parameter doesn't influence any calculation procedures. The final document may be sorted according to the type, category, load duration etc. These options are placed in the window "**Report settings**" (button "**Report**" in the toolbar).

There are three ways how to specify the load value:

| | |
|---|---|
| **Enter load** | • Input of user defined item (specified manually) |
| **Enter from materials catalogue** | • The characteristic value of load is obtained from the catalogue of materials. This catalogue contains wide range of materials with pre-defined weight for permanent loads. Database may be extended manually by inserting user defined items. The obtained characteristic value of the load may be changed manually. The catalogue for variable loads contains loads according to the standard EN 1991-1. |
| **Enter from section catalogue** | • The characteristic value of load is obtained from the catalogue of cross-sections. The catalogue contains the most common steel, timber, concrete and masonry cross-sections including wide range of hot-rolled steel profiles. The characteristic value may be changed. The distribution width has to be specified, if the item is inserted into the report of type "**Area**". This distribution width recalculates the linear load to the area load. Analogously, load width (length of the linear member) has to be specified for items inserted into the "**Point**" report. |

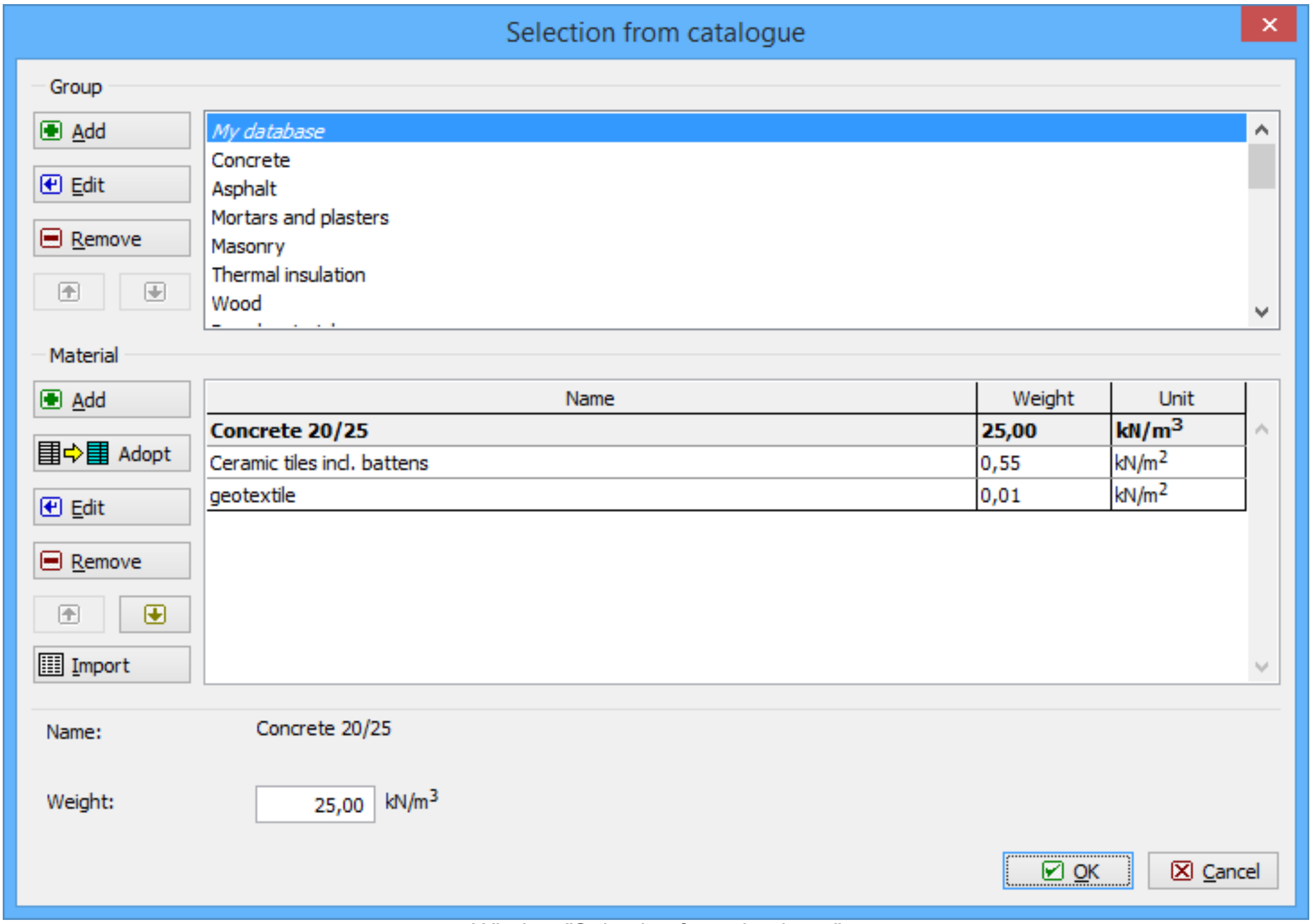
The load factor $\gamma_F$ has to be specified for any item. This partial factor recalculates the characteristic values of load to design ones. The values according to the table *A1.2(B)* of EN 1990 (Set B) are used as a default.




*Window "Edit report item"*

## Materials catalogue

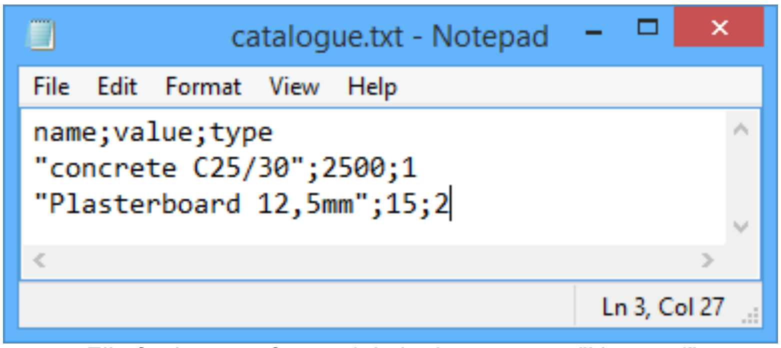
This window contains a database of pre-defined (and user defined) materials. The database is organized into groups, upper table contains list of available groups, bottom table list of items in selected group. User defined groups and items may be added into the database. Groups "**Stored materials**" contain values according to the tables *A.7* to *A.12* of *EN 1991-1-1* . The database of variable loads contains loads according to the chapter *6.3* of *EN 1991-1* .




Window "Selection from database"

## User defined catalogue

The catalogue may be extended by the user defined items. First, new group has to be created (the button "**Add**" in the toolbar along the table of groups). In such group, user is able to add, modify or delete materials. The user defined group may contain both completely new items (the button "**Add**") and existing items from other groups (the button "**Adopt**"). Items may be also imported from **.csv* file. This type of file may be created in a variety of text editors (Notepad, Word, Writer) or spreadsheet editors (e.g. Excel, Calc). Individual items are organized into rows, any row contains name, material weight and type. The name has to be bounded by quotation marks, the type determinates weight unit. Items with the type *1* have units *kg/m*$^3$, items with the type *2* are considered as planar materials and have units *kg/m*$^2$. Semicolon should be used as a separator.

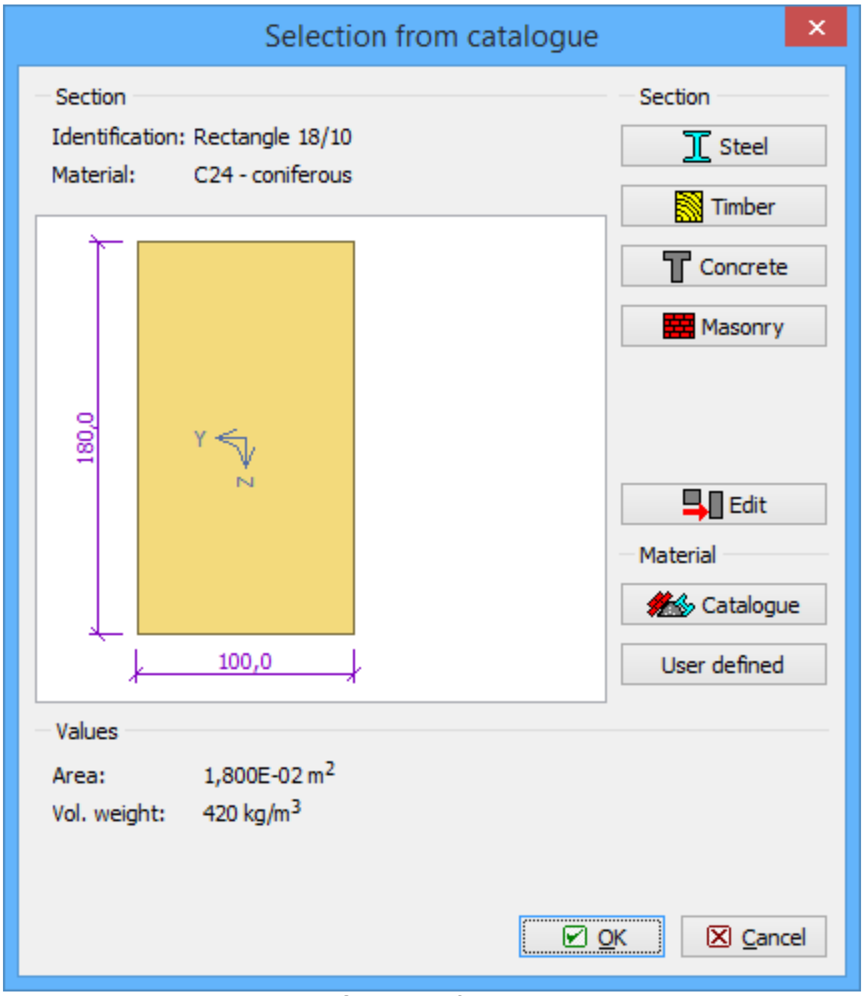



File for import of materials in the program "Notepad"

## Cross-sections catalogue

The item that represents linear structural member may be added into report with the help of this window. The cross-section of the element may be entered with the help of the corresponding buttons "**Steel**", "**Timber**", "**Concrete**" or "**Masonry**" in the right part of the window. Dimensions of existing cross-section may be modified with the help of the button "**Edit**". The material properties may be defined with the help of pre-defined database (button "**Catalogue**" or manually (button "**User defined**").




*Window "Selection from catalogue"*

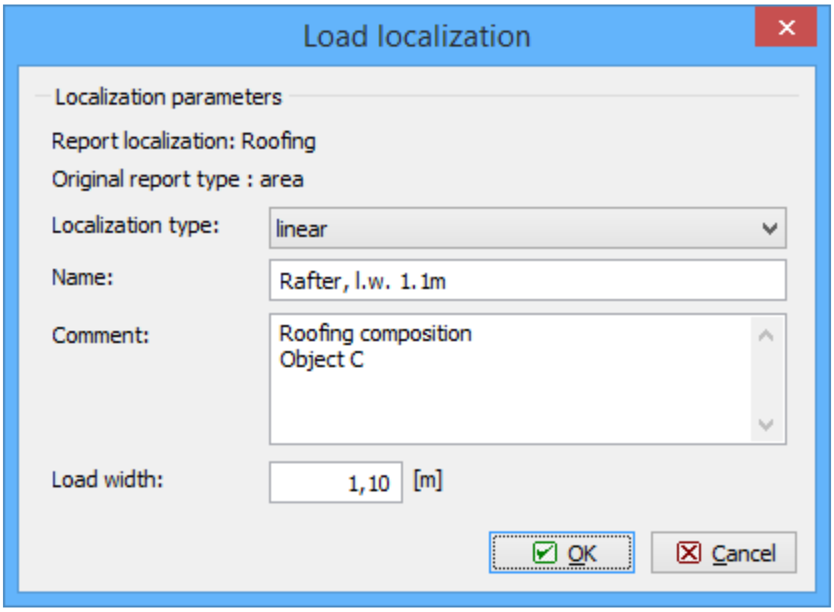
## Load localization

Load localization is a new load report based on the original report. While entering the new localization, the type ("**Area**", "**Linear**", "**Point**", "**To force**"), name and description may be specified in this window. The localization uses all items of original report. Localization may be of the same or lower type (any type of localization may be created for report type "**Area**", linear or point localization may be created for report type "**Linear**", only point localization is permitted for type "**Point**"). Localizations may be used for calculation of the load for linear element (rafter, beam) in planar structure. As the localization may contain also new items in the list, they may be also used for making the load reports with identical basis (e.g. roofing composition) and different additional parts (variable load).

The localization type "**To force**" is suitable for the recalculation of original report to the number of anchorage elements that should carry the entered load. The resulting units depend on the original report (*pc* for original report "**Point**", *pc/m* for "**Linear**", $pc/m^2$ for "**Area**"). The anchoring force (resistance of one anchoring element) has to be specified in this case.

The general properties (loading width or area, description) may be changed later in the window "**Load report**" for corresponding localization.




*Window "Load localization"*

## Snow

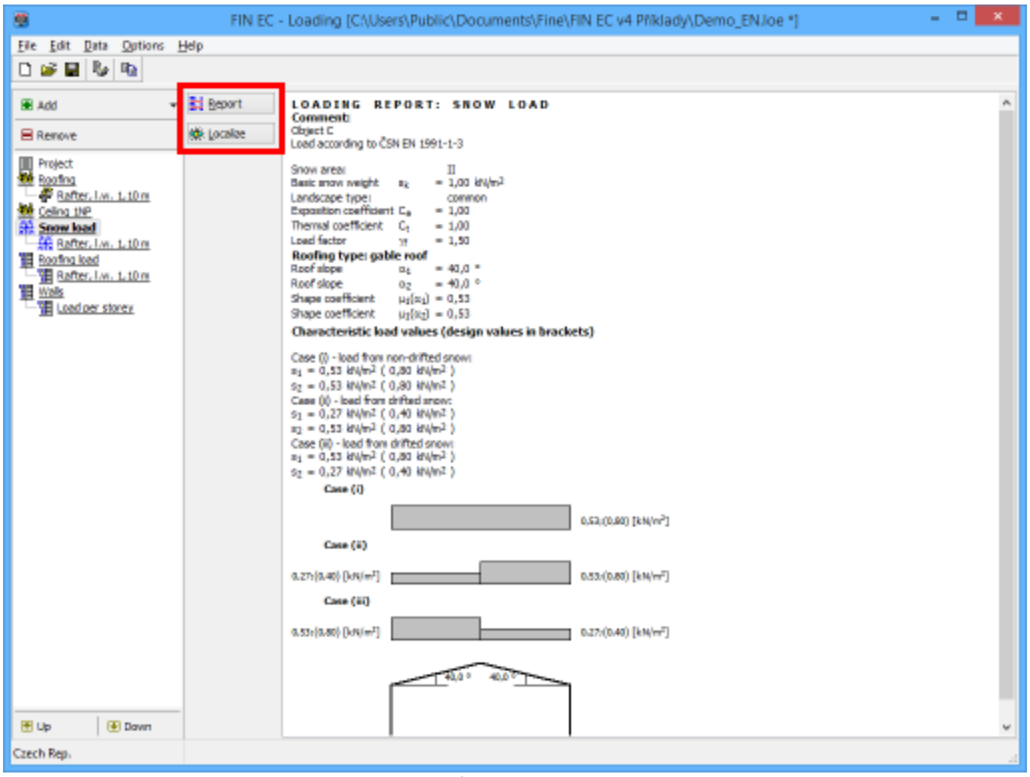
This type of load report shows the values of snow load for the selected type of structure. Both characteristic and design (in brackets) values of the load are calculated according to the specified snow region and other inputs. The parameters of the snow load has to be specified in the window "**Edit snow load**", that appears automatically after addition of the new load report. Resulting values are displayed in *kN/m²*. Conversion to the linear or point load may be done with the help of localization.

Following buttons are available in the toolbar on the right side of the tree menu:

**Report** • The snow parameters and load description may be changed in the window "**Edit snow load**" with the help of this button.

**Localize** • The basic load values in the active load report may be recalculated into linear or point load with the help of this button. The recalculation is done according to the specified loading width or loading area. Resulting values are displayed in *kN/m* or *kN*. Localization type and the needed parameters has to be specified in the window "**Load localization**".

The calculations are described in the chapter "**Snow load**".




*Toolbar for load report edit*

# Edit snow load

This window contains parameters of the snow load. The following settings are available in the frame "**Snow load properties**" (the range may differ according to the selected national annex):

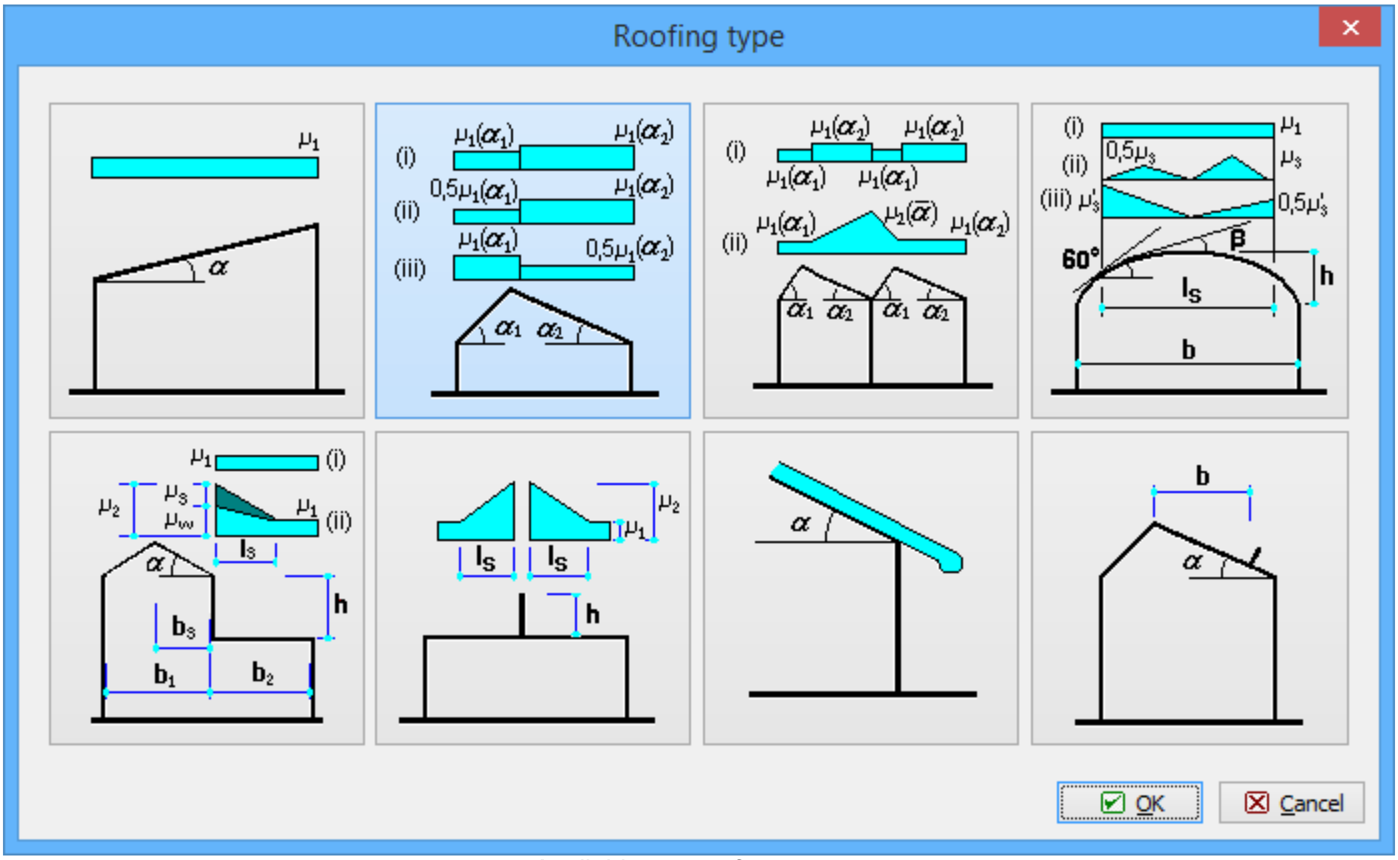
**Snow region**
- The wind region may be selected with the help of drop-down menu or map (may be opened using the button "**Map**"). The value of the characteristic value of snow load $s_k$ is selected according to the snow region. An arbitrary value of $s_k$ is possible for "**user defined**" snow region.

**Terrain type**
- The terrain type affects the exposure coefficient $C_e$. The values are based on the table *5.1*. of EN 1991-1-3. The future development around the site should be considered when selecting the terrain type.

**Thermal coefficient**
- This factor reduces snow load on roofs with high thermal transmittance (> *1W/m²K*). The coefficient $C_t$ should be considered as *1.0* in other cases.

**Consider accidental load**
- The exceptional snow load according to the chapter *4.3* of EN 1991-1-3 may be added into the load report with the help of this setting. Regions with accidental snow load should be described in national annexes. The coefficient for exceptional snow loads $C_{esl}$ has to be specified. Recommended value according to EN 1991-1-3 is *2,0*.

The type of snow load and necessary geometric parameters of the roof may be specified in the frame "**Roofing type**". The task type may be changed after clicking on the button with roof scheme.

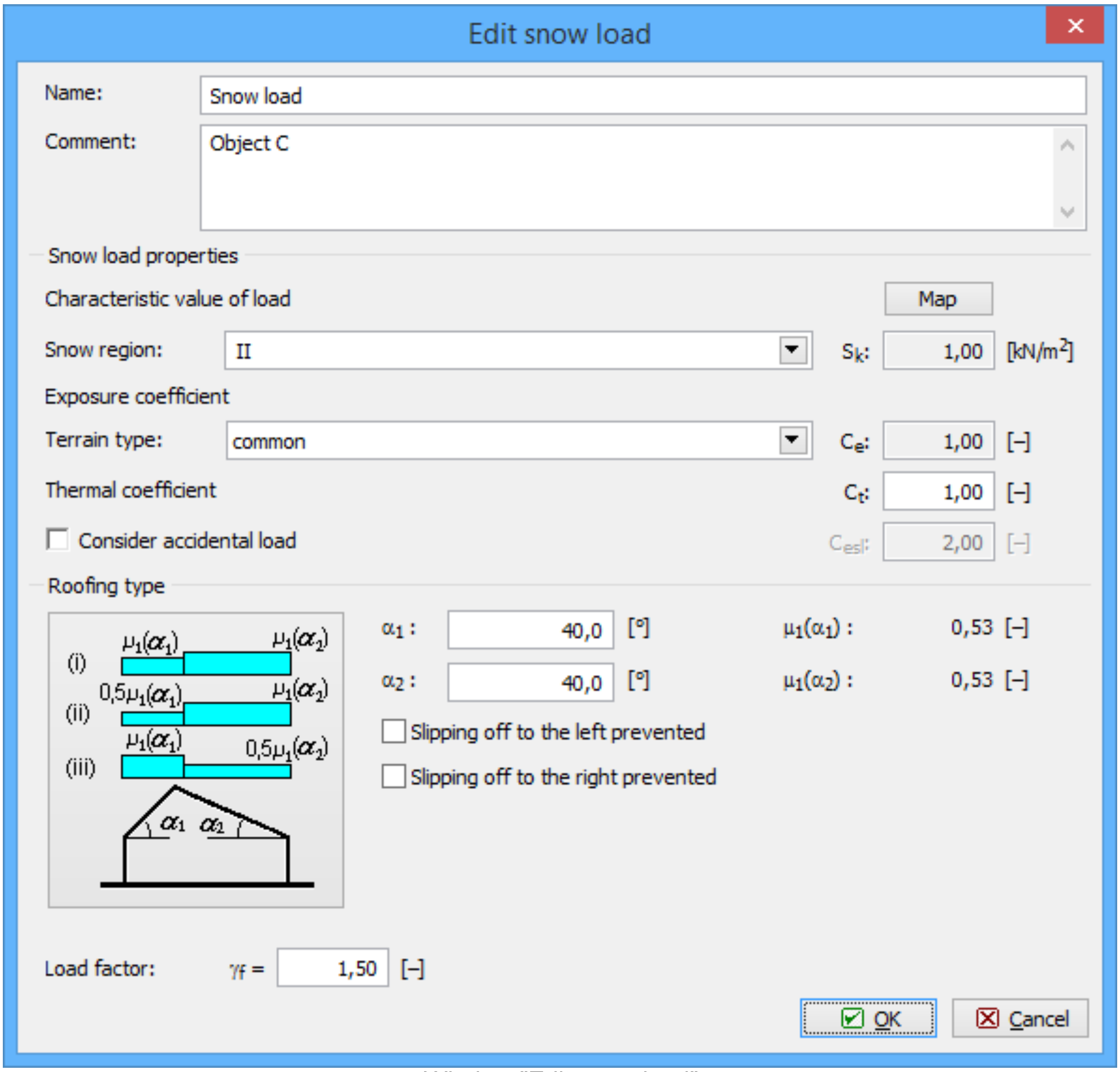



*Available types of structures*

If the setting "**Sliding off prevented**" is switched on, the snow load shape coefficient $\mu_1$ isn't reduced according to the pitch of the roof (figure *5.1* of EN 1991-1-3).

The load factor $\gamma_F$ is the partial factor accounting for model uncertainties and dimensional variations in accordance with EN 1990. The values according to the table *A1.2(B)* of EN 1990 (Set B) are used as a default.

The calculations are described in the chapter "**Snow load**".




*Window "Edit snow load"*

## Load localization

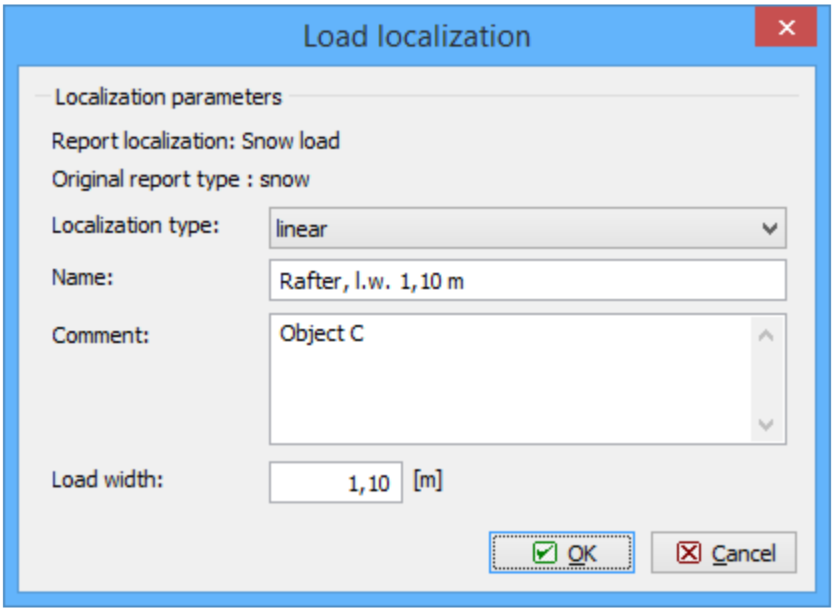
This window contains parameters of the localization that converts the load per area to the load per one linear meter.

Following localization types are available:

**Linear** • The load per area will be converted into the linear load. The loading width of the loaded element has to be specified. The resulting values will be calculated as a product of basic values and loading width.

**Point** • The load per area will be converted into the point load. The loading area of the loaded element has to be specified. The resulting values will be calculated as a product of basic values and loading area.




*Window "Load localization"*

## Wind

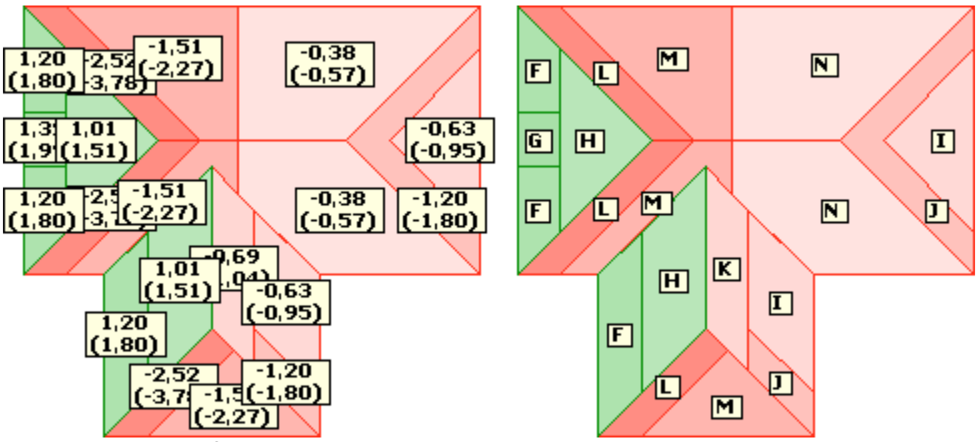
This type of load report shows the values of wind load for the selected type of structure. Both characteristic and design (in brackets) values of the load are displayed including transparent structure view. The parameters of the wind load has to be specified in the window "**Edit wind load**", that appears automatically after addition of the new load report. Resulting values are displayed in *kN/m²*. Conversion to the linear or point load may be done with the help of localization.

Following buttons are available in the toolbar on the right side of the tree menu:

**Report** • The wind parameters and load description may be changed in the window "**Edit wind load**" with the help of this button.

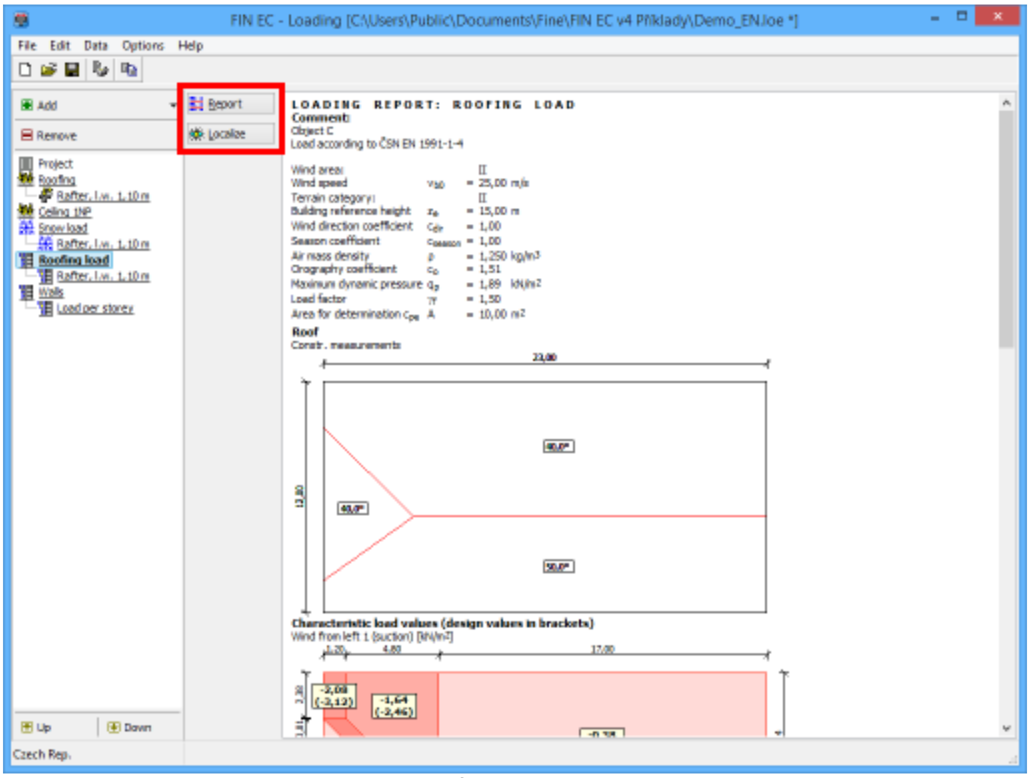
**Localize** • The basic load values in the active load report may be recalculated into linear or point load with the help of this button. The recalculation is done according to the specified loading width or loading area. Resulting values are displayed in *kN/m* or *kN*. Localization type and the needed parameters has to be specified in the window "**Load localization**".

The load report with load values written in the roof scheme may be non-transparent for more complicated roofs. In that cases, the setting "**Values of wind load on roof in table**" in the main application window may be used. The Roof scheme will contain also region marks, load values will be organized in the table under the roof scheme.



*Roof plans with displayed load values and region marks*

The calculations are described in the chapter "**Wind load**".



*Toolbar for load report edit*

## Edit wind load

This window contains parameters of the wind load. It is divided into two parts: First part contains parameters that are necessary for the calculation of the maximum dynamic pressure, second part is dedicated for the choice of wind load type and its parameters. Switching between these two parts can be done with the help of buttons "**Previous**"/"**Next**". Following parameters are placed in the first part:

**Wind region**
- The wind region may be selected with the help of drop-down menu or map (may be opened using the button "Map"). The value of the fundamental wind velocity $v_{b,0}$ is selected according to the wind region. An arbitrary value of $v_{b,0}$ is possible for "**user defined**" wind region.

**Terrain category**
- The terrain category refers to the terrain roughness on site. The detailed description of all categories is described in the "**Annex A**" of EN 1991-1-4. The parameters assigned to the selected category are based on the table *4.1* . The surrounding terrain with different roughness should be respected during the choice of the terrain category. Rules are given in the chapter *A.2* of the standard.

**Reference height of building**
- The reference height of the building $Z_e$ is a basic parameter for the calculation of maximum dynamic pressure.

**Directional factor**
- This factor $c_{dir}$ is defined in the chapter *4.2* of EN 1991-1-4.

**Season factor**
- The season factor $c_{season}$ may reduce wind load for temporary structures, that won't exist during the season with highest wind pressure.

**Air density**
- The recommended value of the air density $\rho$ is $1{,}25 kg/m^3$.

**Orography factor**
- The orography factor $c_o$ increases the wind velocity at isolated hills and ridges or cliffs and escarpments. The calculation may be performed in the dedicated window "**Orography factor**", that may be launched by the button "**Edit**". The calculation is based on the chapter *A.3* of EN 1991-1-4.

**Maximum dynamic pressure**
- The software shows the value of the maximum dynamic pressure $q_p$ according to the formula *(4.8)* of EN 1991-1-4. The user defined value of the pressure may be entered after using the setting "**User defined max. dynamic pressure**".

**Size of loaded area**
- The loaded area $A$ is used for the calculation of external pressure coefficient. The values $c_{pe,1}$ are used for loaded areas lower than $1 m^2$. The values $c_{pe,10}$ are used for loaded areas greater than $10 m^2$. The linear interpolation according to the chapter *7.2.1* is used for intermediate values.

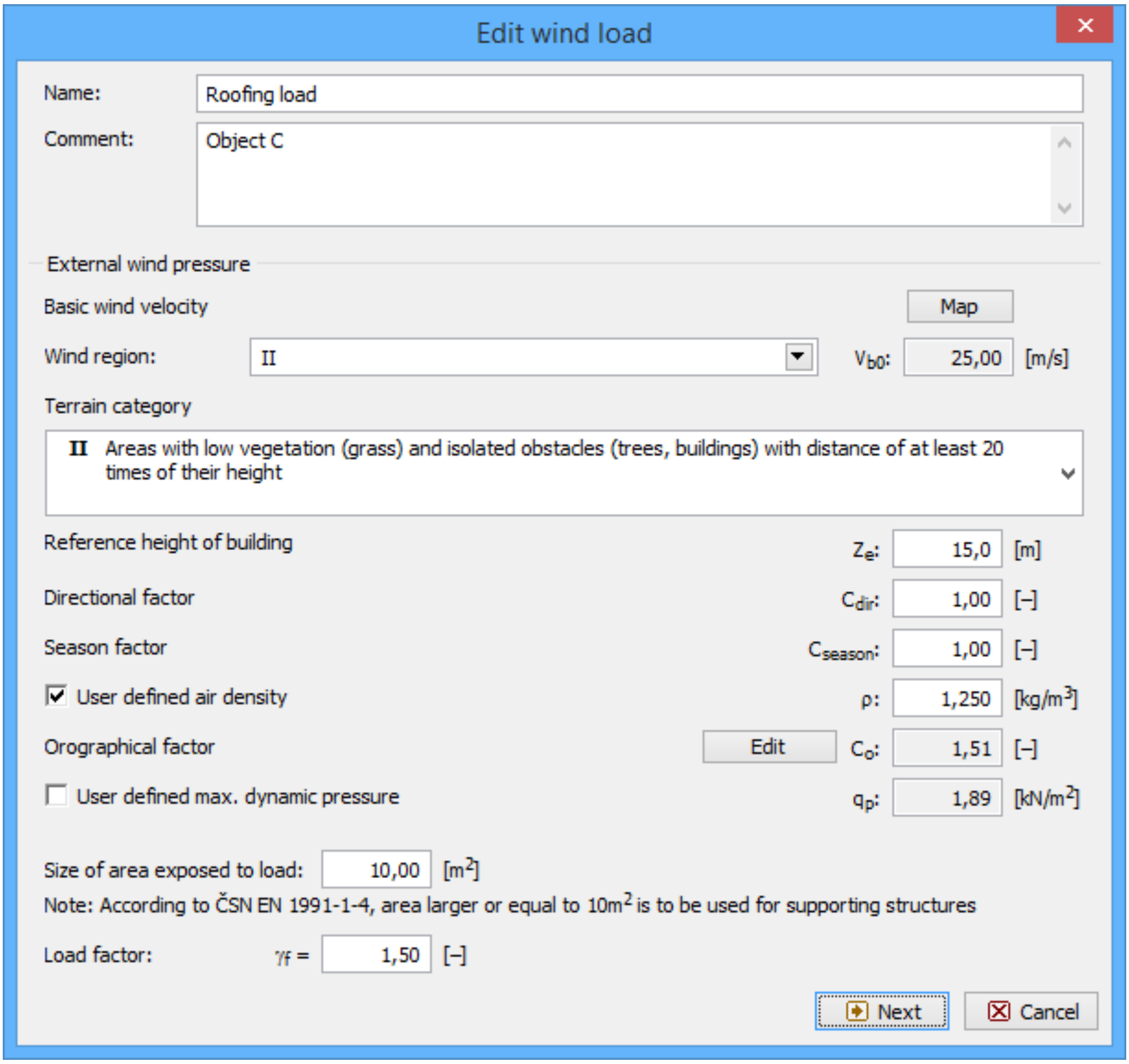
The load factor $\gamma_F$ is the partial factor accounting for model uncertainties and dimensional variations in accordance with EN 1990. The values according to the table *A1.2(B)* of EN 1990 (Set B) are used as a default.

The calculation of the maximum dynamic pressure is described in the chapter "**Wind load**" of the theoretical help.

The button "**Next**" switches the window into the second part, where the type of load and its parameters may be specified. Following types are available:

| | |
|---|---|
| **Roof** | • This options creates the load report for roof. The variety of plans and roof types are available in the software. The rules given in the chapters *7.2.3* (flat roofs), *7.2.4* (mono-pitched roofs), *7.2.5* (duo-pitched roofs) and *7.2.6* of EN 1991-1-4 are considered. |
| **Canopy roof** | • This options creates the load report for canopy roof. The variety of plans and roof types are available in the software. The rules given in the chapter *7.3* (canopy roofs) of EN 1991-1-4 are considered. |
| **Walls of rectangular plan object** | • This options creates the load report for walls of rectangular plan buildings. The erbitrary number of calculation levels may be specified for the report. The calculation is based on the chapter *7.2.2*. |
| **Vault roof** | • This options creates the load report for circular cylindrical roofs according to the chapter *7.2.8* of EN 1991-1-4. |
| **Dome** | • This options creates the load report for domes according to the chapter *7.2.8* of EN 1991-1-4. |



*Window "Edit wind load"*

## Roof/Canopy roof

The load report for roofing structure or canopy roof may be created for this type. Supported are different types of roofs (duo-pitched, hipped etc.) and few basic building plans. The linear load on arbitrary section may be calculated with the help of localization.

### Roof type

Following types of roofs are available:

**User defined**
- The roofs with non-standard topology (different values for pitch, combinations of gable and hip roofs etc.) may be created using this roof type. The modelling is described below. The values of factors $c_{pe}$ are based on the chapters *7.2.3*, *7.2.4*, *7.2.5* and *7.2.6* (or *7.3*) of EN 1991-1-4 depending on the roof topology. The worst option is used for cases, where more types are combined in one roof (e.g. rectangular roof with one hip).

**Hip**
- The hipped roof with constant pitch may be created with the help of this type. The hipped roof with different pitches has to be created with the help of the type "**User defined**". The values of factors $c_{pe}$ are based on the chapters *7.2.6* or *7.3*. The type "**Flat**" should be used for roofs with pitch lower than *5°*.

**Duo pitch**
- The pressure coefficients $c_{pe}$ for duo-pitched roofs are selected according to the chapter *7.2.5* or the table *7.7*. The type "**Flat**" should be used for roofs with pitch lower than *5°*. The duo-pitched roof with different pitches has to be created with the help of the type "**User defined**".

**Flat**
- The roof structures with pitch lower than *5°* should be considered as flat roofs. The values of factors $c_{pe}$ are based on the chapter *7.2.3*. The eaves type (sharp, curved, mansard and with parapets) should be selected for this type of roof.

**Mono pitch**
- This type is available only for rectangular plan. The calculation is based on the chapter *7.2.4* or the table *7.6*.

## Modelling options for "User defined" roof

The table "**Wall properties**" is enabled for the type "**User defined**". The pitch and eaves height may be changed for any wall. Also the corresponding roof surface may be deleted completely. The structure view on the right side of the table is updated automatically after any change.

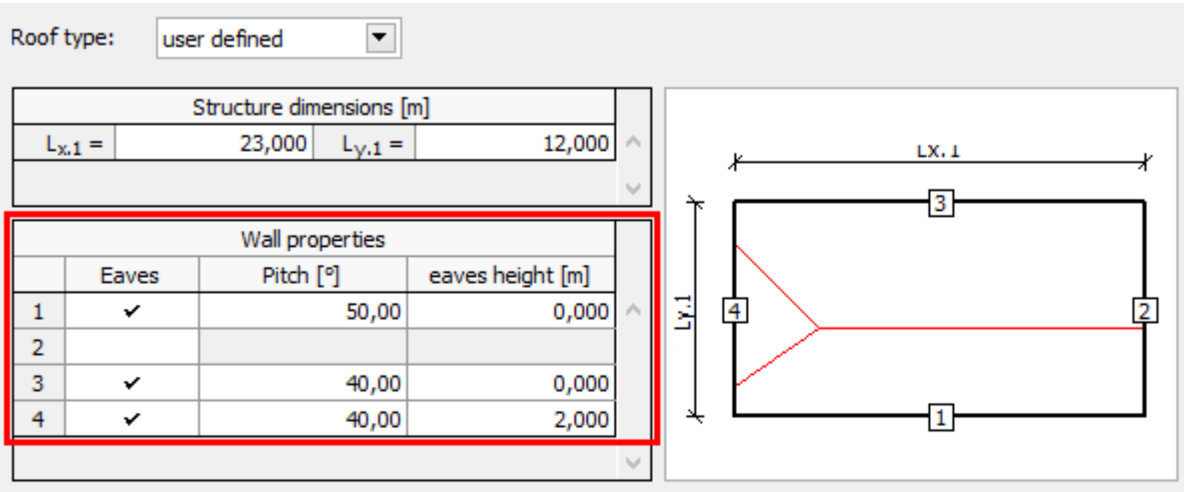
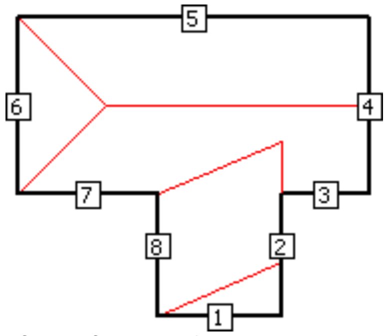



*Table "Wall properties"*

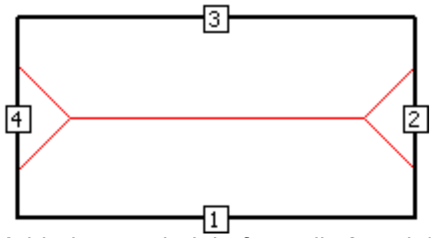
The table "**Wall properties**" contains roof edges organized into rows. The order in the table respects the numbering in the roof view. First column "**Eaves**" contains check boxes that may switch off the corresponding roof surface. For example, hip may be converted into gable with the help of this setting.




*Roof surfaces for lines 2 and 4 are switched off*

The column "**Pitch**" contains the input of roof pitch for corresponding surface. The vertical level of the eaves may be changed with the help of the column "**Eaves height**". This setting may be used for example for the input of dutch hip.



*Added eaves height for walls 2 and 4*

## Wind action on roof

The values of maximum pressure and maximum suction are written in the standard for certain types of roofs. This setting may influence, which values will be displayed in the load report. Both variants will be shown for the option "**Compression and suction**".

## Anchorage

The load per square meter may be transferred to the number of anchorage elements per square meter with the help of this part. The anchoring force (resistance of one anchoring element) has to be specified in this case. This may be used for the calculation of anchorage elements for flat roofs due to suction. This setting is usually combined with "**Envelope**" in the part "**Direction of wind action**".

## Direction of wind action *("Roof" type only)*

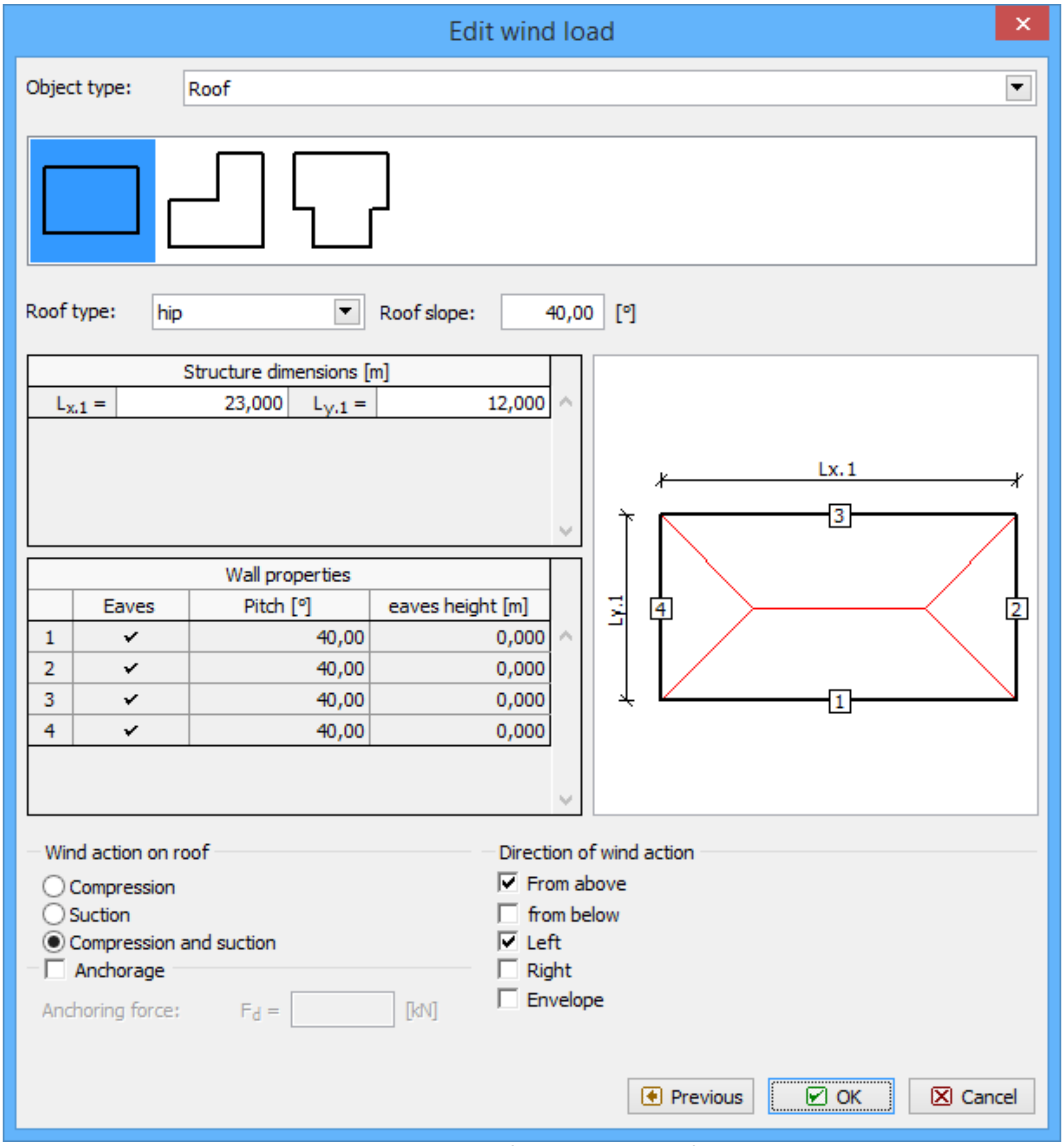
The direction of wind may be specified in this part. The orientation is based on the roof view. Four directions and an envelope are available. The envelope creates the scheme with the highest values of the load in any point of the roof, all four directions are considered. This setting is not available for canopy roofs, as the chapter *7.3* contains shape factors calculated for all directions.

## Canopy roof *("Canopy" type only)*

This part contains the option to specify the range of considered blockage ratios in accordance with the figure *7.15* of EN 1991-1-4. The default range *<0;1.0>* contains all possible cases. Limiting values are free-standing canopy (*φ=0*) and blocked canopy (*φ=1.0*). For example, the values for blocked canopy may be obtained with the help of the range *<1.0;1.0>*.

The parameters specified in the first part of the window (wind region, terrain category etc.) may be changed after clicking on the button "**Previous**".



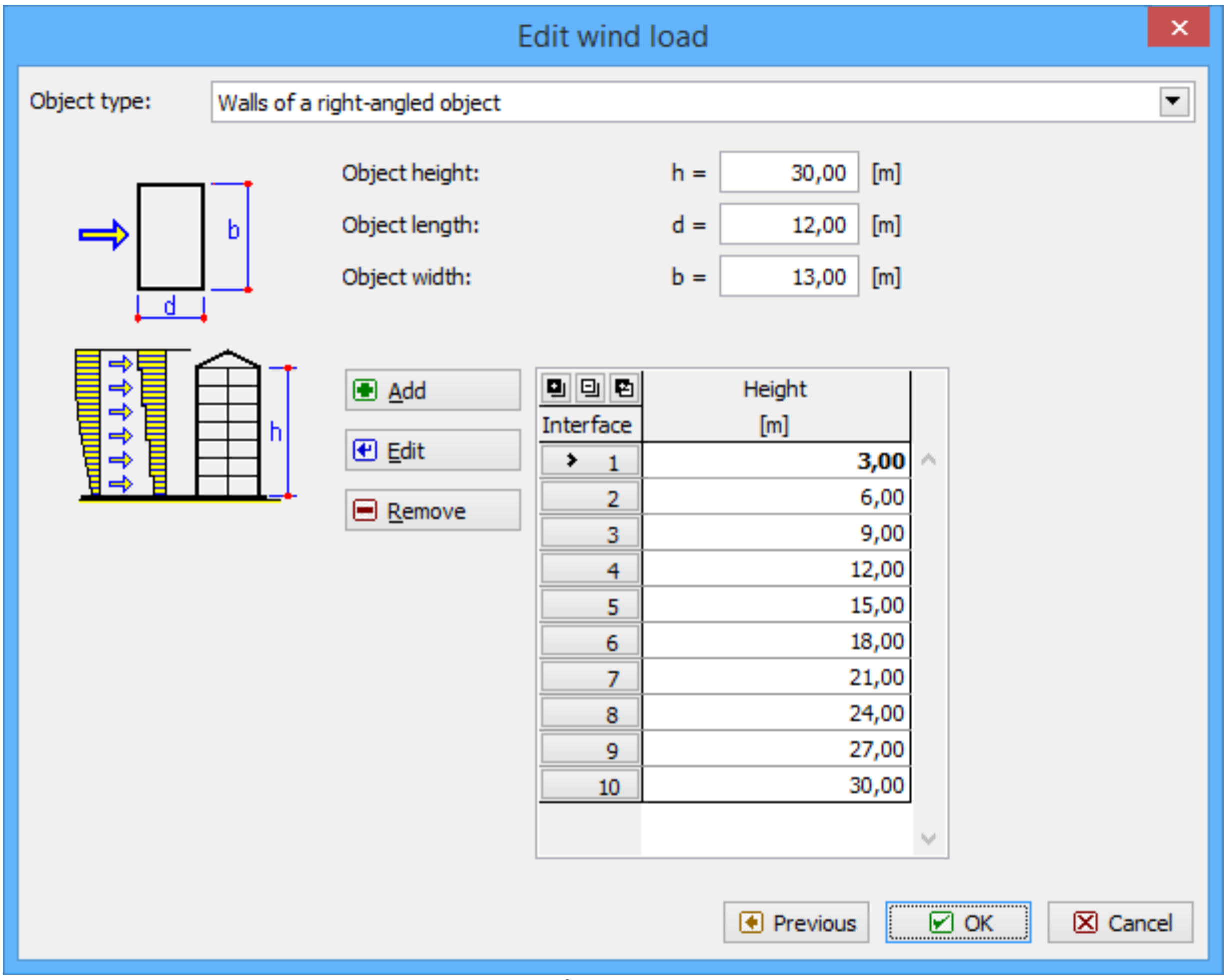
*Parameters of wind load on roof*

## Walls of rectangular plan object

The object height *h*, the object length *d* and the object width *b* have to be specified for walls of rectangular plan buildings. The meaning of entries is shown in the figure in the left part of the window. Bottom part of the window contains the table for the input of analysis levels, where the wind pressure will be calculated. Any level is characterized by the height above the terrain level. This height can't exceed the object height.

The parameters specified in the first part of the window (wind region, terrain category etc.) may be changed after clicking on the button "**Previous**".

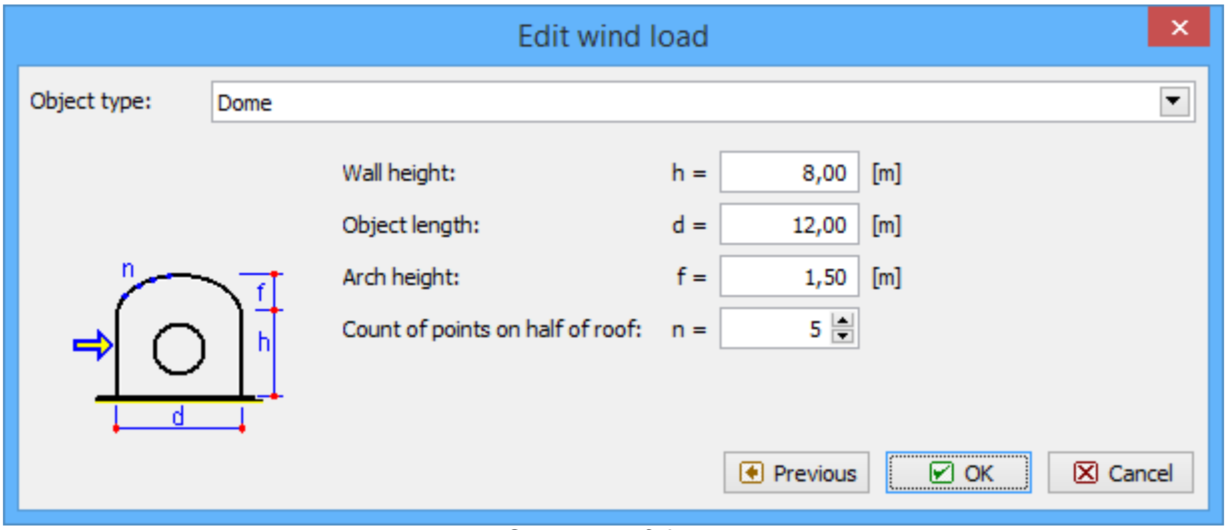



*Input of wall properties*

## Vault roof and dome

The wall height *h*, span *d* and arch height *f* have to be specified for vaulted roofs and domes. The meaning of entries is shown in the figure in the left part of the window. The number of calculation points per half of the roof has to be also specified for domes. The load values will be provided for these points. The values are organized into a table that contains also horizontal distance from the upwind edge of the roof.

The parameters specified in the first part of the window (wind region, terrain category etc.) may be changed after clicking on the button "**Previous**".

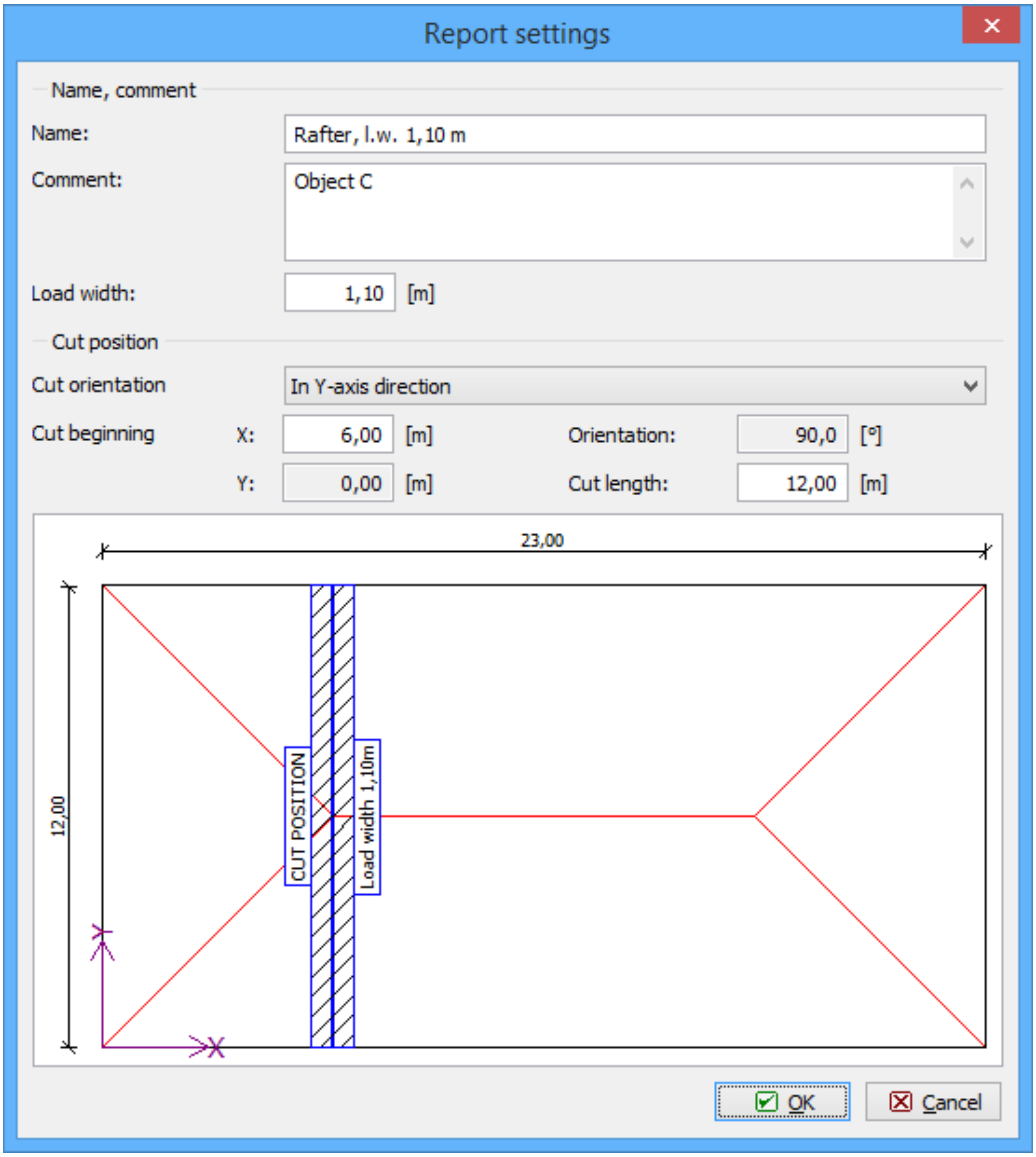



*Geometry of dome*

## Load localization

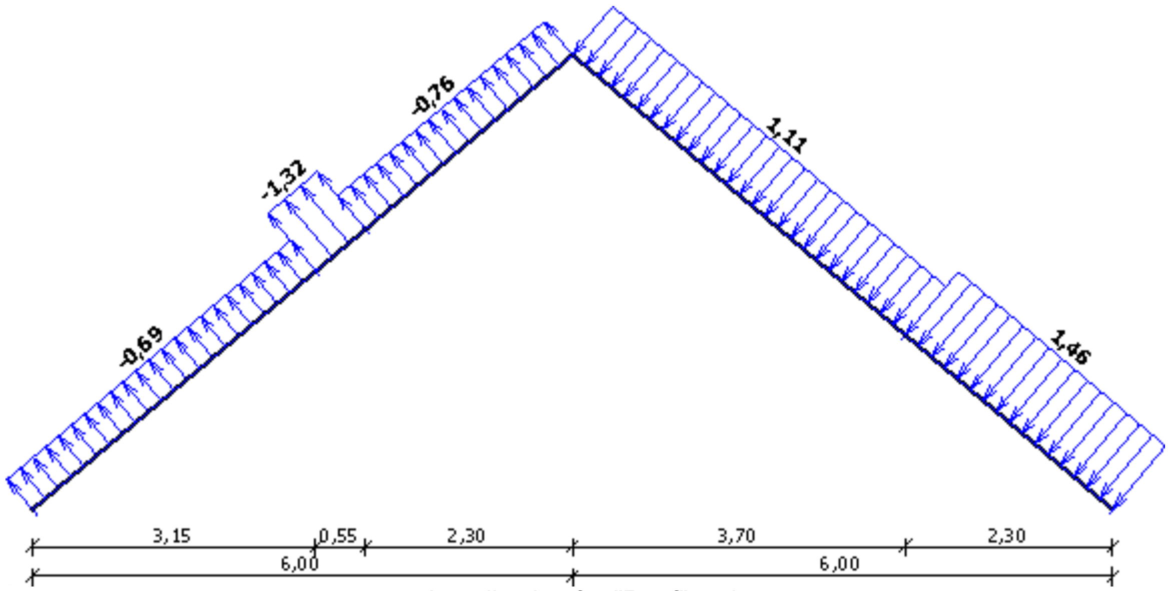
This window contains parameters of the localization that converts the load per area to the load per one linear meter. The load width of the structural member has to be specified for this conversion.




*Window "Load localization"*

If the localization is created for the task type "**Roof**", the localization is created as a section of the structure. The position, length and direction of the section may be specified by user. Bottom part of the window shows structure plan with position of specified section, including load area.




*Localization for "Roof" task*

# Program Sector

The program "**Sector**" calculates cross-sectional characteristics of sections made of walls. Walls are parts of cross-section, that have significantly longer length than the thickness. Typical examples of such structures are steel cross-sections, cross-sections of concrete bridges or cores of height buildings. Calculated characteristics are important mainly for verification of simple torsion and warping. The program calculates following characteristics:

- The coordinates of centre of gravity
- The cross-sectional area
- The moments of inertia
- The rigidity moment in simple torsion - $I_k$
- The main warping coordinate - $\omega$
- The position of shear centre
- The polar moment and radius of inertia - $I_p$, $i_p$
- The warping constant $S_\omega$
- The warping moment of inertia $I_\omega$

## User interface

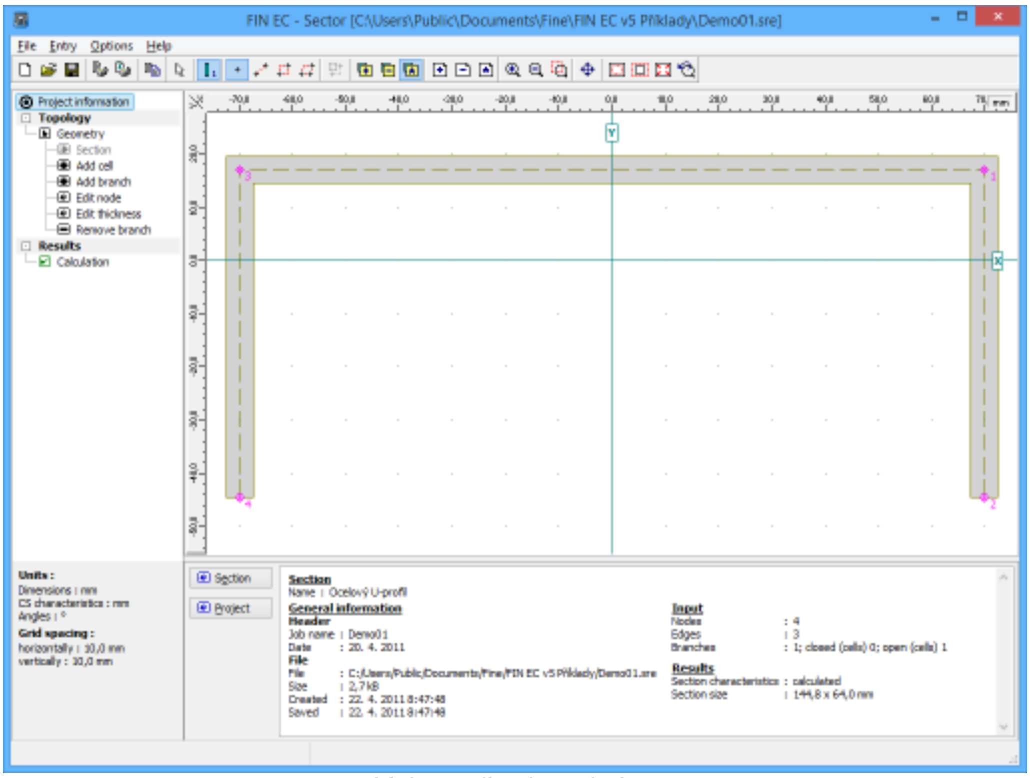
The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for switching between parts of an input. The tree menu can be alternated by the part "**Entry**" of the main menu. The appearance of the workspace may be modified in the window "**Options**", that can be opened from the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

The tree menu contains two main parts:

- **Topology** - This part contains tools for modelling the cross-section
- **Results** - This part shows the results of calculation

## Main screen

The default screen contains general information of the project. The cross-section name and additional notes can be specified with the help of the button "**Section**". The button "**Project**" opens the window "**General project data**", which contains additional data of the project, that can be used in heading and footing of documents.



*Main application window*

# Topology

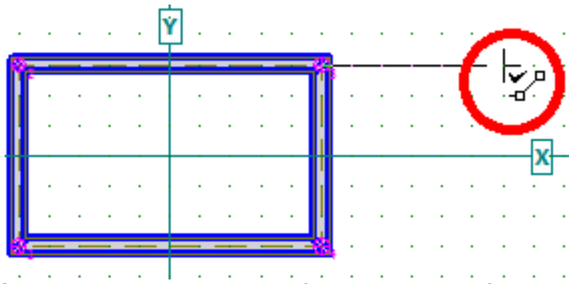
This part contains tools for input of cross-section geometry. The topology is entered as a scheme, particular sectors are defined as lines with given thickness. The modelling principles should be respected during the input.

## Input of cross-section

Following tools are available for the input of cross-section topology:

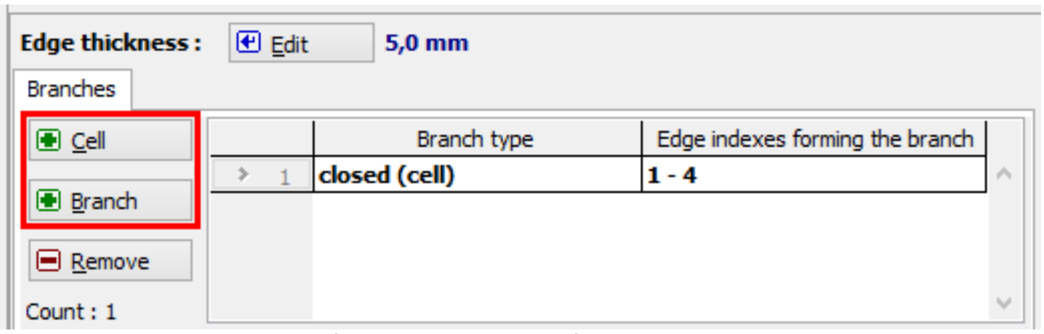
- **Profile** - The cross-section from pre-defined database can be used as a basis for modelling. The cross-section can be selected in the window "**Insert section**". Both hot rolled and welded profiles are available. This procedure can be used when calculating e.g. characteristics of I-profile with haunch. This option is available only for empty project without any cells or branches.
- **Add cell** - The closed branches (cells) can be added in this mode. Cells cannot be added if there is already any open branch in the project (including pre-defined cross-section that includes an open branch). This limitation is described in the theoretical part of help.
- **Add branch** - The branches can be inserted both numerically in the table and graphically on the workspace in this mode.

Graphical input is performed in the workspace. The nodes are entered by mouse clicking (left mouse button). The snapping grid can be used during the input, grid properties are organized in the window "**Options**". The input finish has to be confirmed by clicking on the first node (cells) or by repeated click on the last node (open branches). The cursor appearance changes during the confirmation:



*Cursor appearance when finishing input of branch*

Also numerical input of topology is available. This is done in the window "**Add branch/cell**", that contains tools for input of nodes coordinates. This window can be opened with the help of corresponding buttons "**Cell**" and "**Branch**" in the bottom part of the window:



*Buttons for numerical input of cells and branches*

## Edit of cross-section

The cross-section can be modified with the help of following tools:

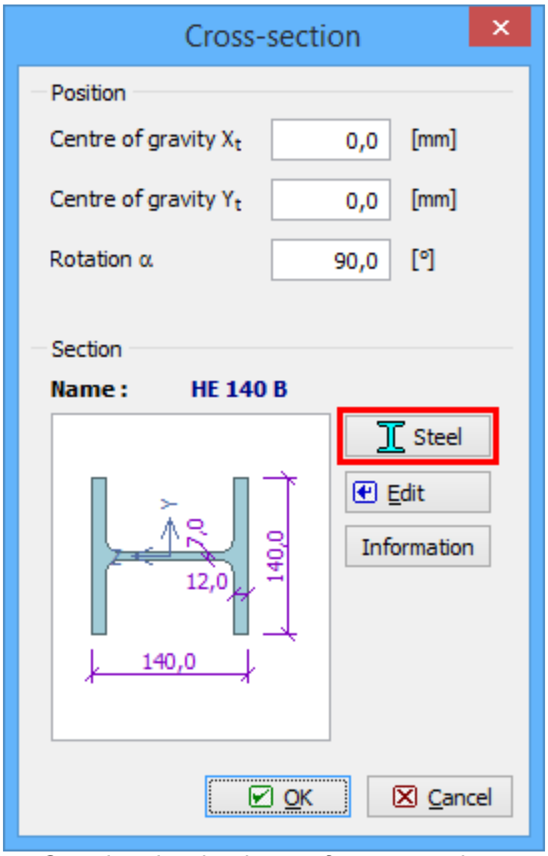
- **Edit node** - The tool opens the window "**Node properties**" for edit of node coordinates.
- **Edit thickness** - This tool is able to change the sector thickness in the window "**Sector properties**". The thickness can be changed for active, selected or all sectors.
- **Remove branch** - The tool for deleting branches. The tools automatically deletes also branches, that begin in some node on deleted branch.

After finishing the input, it is possible to switch the tree menu into the part "**Results**" and view the calculated cross-sectional characteristics.

# Insert section

The steel cross-section may be selected from pre-defined database as the basis for modelling with the help of this window. The database can be opened by the button "**Steel**", the selection of the cross-section is done in the window "**Cross-section editor**". The database contains both a wide range of hot-rolled profiles and a variety of welded cross-sections with arbitrary dimensions. The cross-section may be change with the help of the button "**Edit**", the cross-sectional characteristics can be displayed after using the button "**Information**".

The window also contains the input lines for insertion coordinates and rotation of the cross-section $\alpha$.



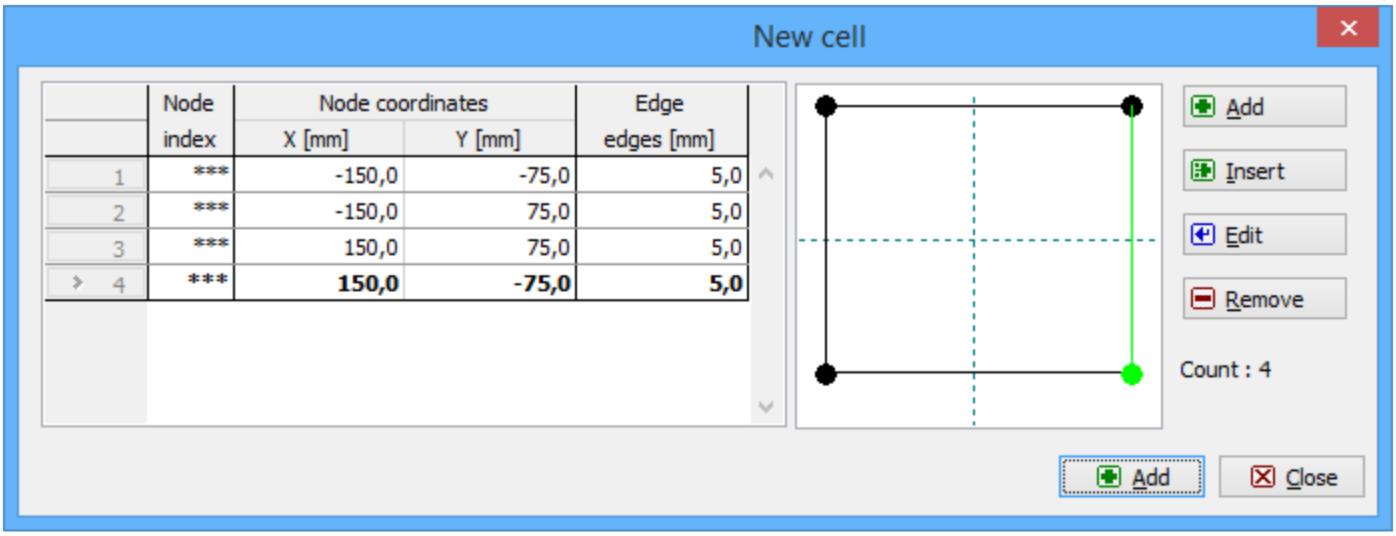

*Opening the database of cross-sections*

## Add branch/cell

Branches of the cross-section can be specified in this window. Left part of the window contains table with branch nodes, right part contains branch preview and dedicated toolbar with following buttons:

- **Add** - Opens the window "**Add node**" for entering the position of new node. The node is added at the end of the branch.
- **Insert** - Adds a new node in front of specified node. First, the active node has to be selected in the table before using this button (active node is highlighted by bold font in the table and green colour in the preview). After that, the new node will be specified also in the window "**Add node**".
- **Edit** - Opens the window with properties of active node (coordinates, sector thickness)
- **Remove** - Deletes the active node (highlighted by bold font in the table and green colour in the preview)

The minimum number of nodes in table is three for closed cells. Minimum number for open branches is two (beginning and end).

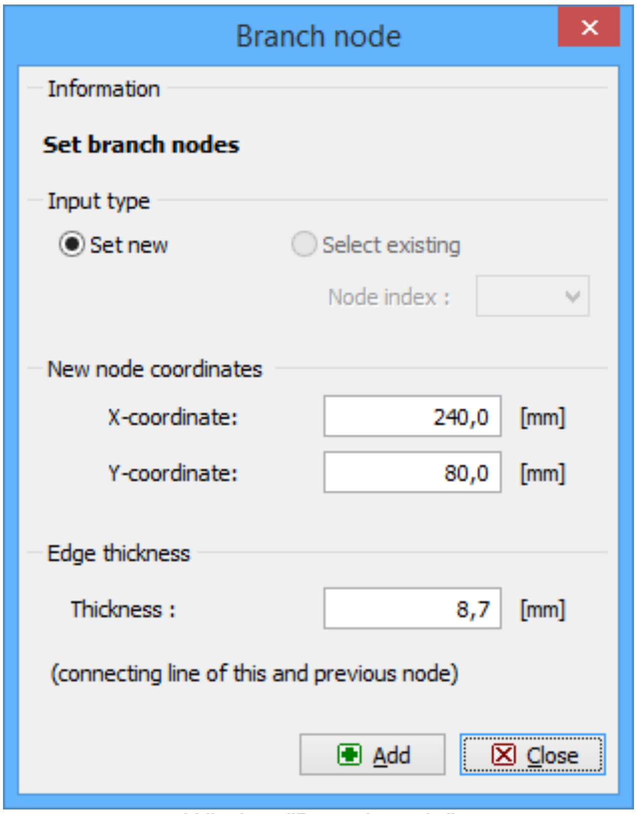


*Window "New cell"*

## Add node

This window can be used for the input of the branch topology. First node of the branch (excluding first branch in the project) has to be selected from existing nodes. Following ones can be specified as new nodes or selected from the list of existing nodes using these two options:

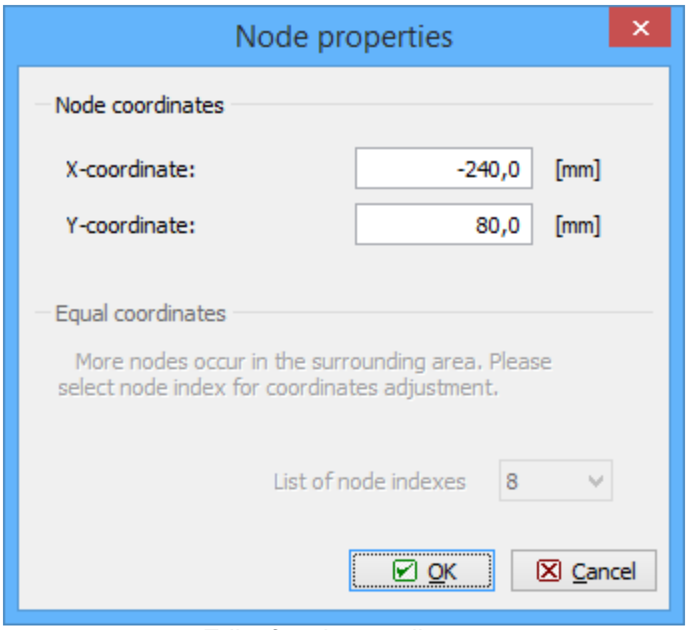
- **Set new** - The node position can be specified using coordinates *[X,Y]*. Simultaneously, the thickness of the sector, that ends in this node, can be entered.
- **Select existing** - The node position is defined by the position of selected existing node.




*Window "Branch node"*

## Node properties

This window contains coordinates of edited node. The coordinates are in the global coordinate system. If there is more nodes with the same coordinates, it is necessary to select the number of modified node.

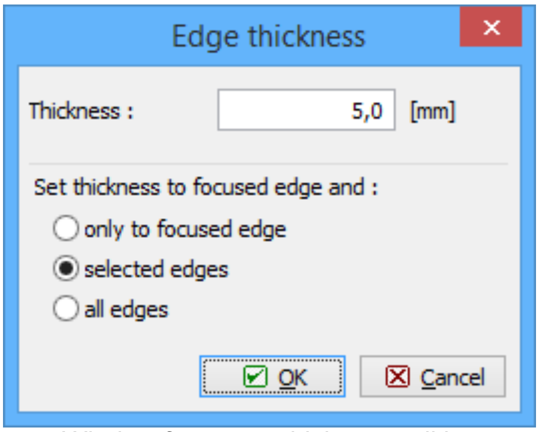



*Edit of node coordinates*

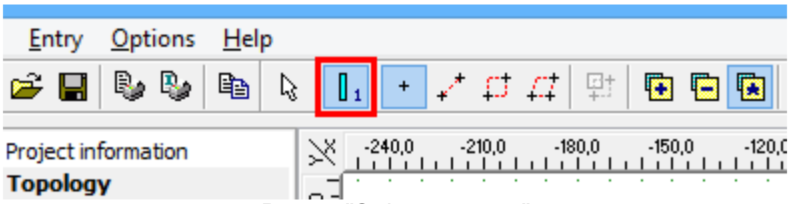
## Sector properties

The sector thickness can be changed in this window. This change may be applied to active sector, selected sectors or all sectors in the cross-section.



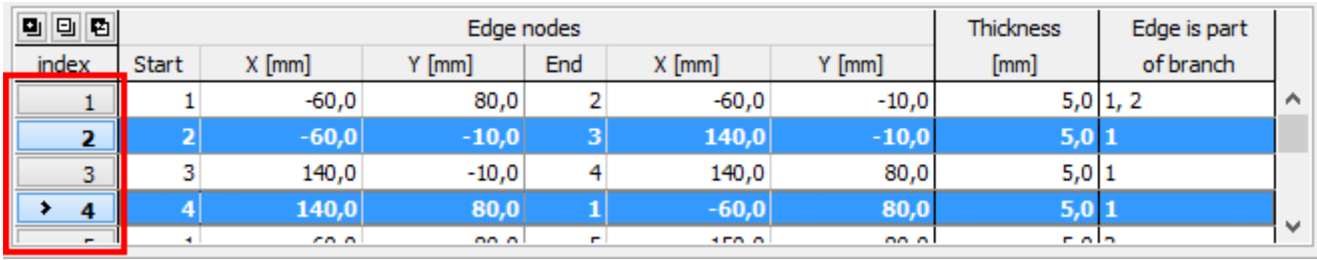

*Window for sector thickness editing*

If the change should be applied to more selected sectors, these sectors have to be selected first. The mode for selecting the sectors in the workspace has to be switched on with the help of corresponding button in the toolbar above the workspace. After that, it is possible to select sectors by clicking in the workspace. The selected sectors are highlighted by green colour. The toolbar "**Selections**" contains tools that make the work easier.




*Button "Select sectors"*

Alternatively, the sectors can be also selected with the help of buttons in the first column of the sectors table in the bottom part of the window.



| index | Edge nodes Start | X [mm] | Y [mm] | End | X [mm] | Y [mm] | Thickness [mm] | Edge is part of branch |
|---|---|---|---|---|---|---|---|---|
| 1 | 1 | -60,0 | 80,0 | 2 | -60,0 | -10,0 | 5,0 | 1, 2 |
| 2 | 2 | -60,0 | -10,0 | 3 | 140,0 | -10,0 | 5,0 | 1 |
| 3 | 3 | 140,0 | -10,0 | 4 | 140,0 | 80,0 | 5,0 | 1 |
| 4 | 4 | 140,0 | 80,0 | 1 | -60,0 | 80,0 | 5,0 | 1 |

*Buttons for sectors selection*

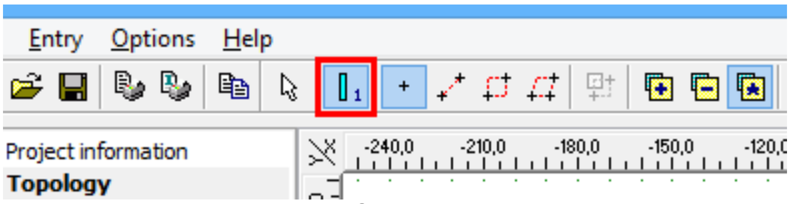
## Results

This part shows the calculation results. The workspace is able to show diagrams of some characteristics, the table in the bottom part shows the list of calculated results.

### Results in the workspace

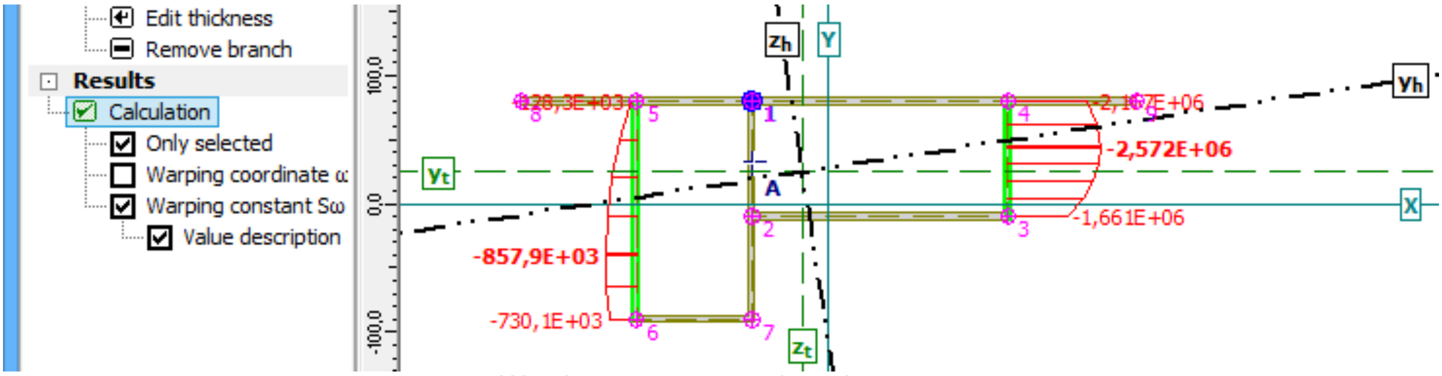
The workspace may display warping coordinate $\omega$ and warping constant $S_\omega$. The displayed characteristics may be switched on or off in the tree menu. The font size can be defined in the window "**Options**", tab "**View**". Warping characteristics can be drawn for the whole cross-section or only for selected sectors. The option "**Only selected**" in the tree menu has to be ticked on in this case. The mode for selecting the sectors in the workspace has to be switched on with the help of corresponding button in the toolbar above the workspace. After that, it is possible to select sectors by clicking in the workspace. The selected sectors are highlighted by green colour.




*Button "Select sectors"*

The toolbar "**Selections**" can be used when selecting the sectors. Selected sectors are highlighted by green colour.




*Warping constant on selected sectors*

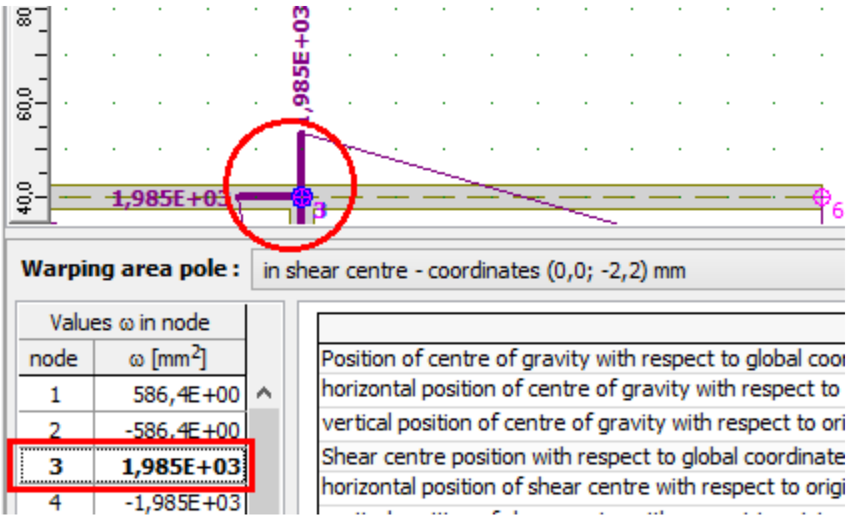
The workspace shows also these results:

- **The centre of gravity** - this point is located in the intersection of axes $y_t$ and $z_t$ (dashed lines)
- **The shear centre** - marked as a point *A*
- **The main axes of the cross-section** - bold axes marked $y_h$ and $z_h$

The picture in the workspace can be copied into clipboard with the help of shortcut *Ctrl+C* or the button " " in the toolbar above the workspace. The picture in the clipboard can be pasted into arbitrary document or graphic editor.

## Warping coordinates *ω* in table

Warping coordinates *ω* in individual nodes are also listed in the table in the left bottom corner. Clicking in the table highlights the appropriate node in the workspace.



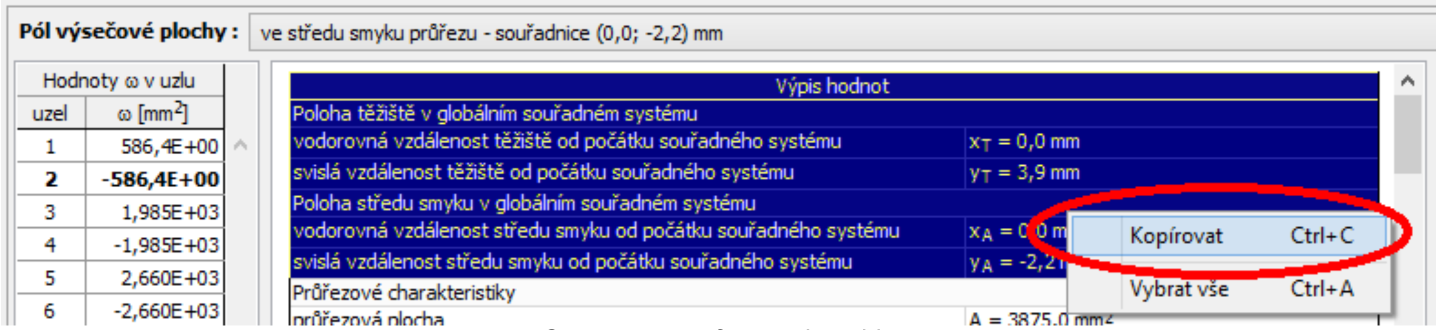

*Warping coordinates in the table and in the workspace*

## Table with results

The bottom frame also contains the table with all calculated characteristics. The data in table can be selected by the cursor and copied into another program (text editor). The table contains a context menu, that can be opened with the help of right mouse button.

The calculated cross-sectional characteristics are described also in the theoretical part of help.



*Context menu for results table*

## Program Section

The program "**Section**" calculates cross-sectional characteristics, that may be used as inputs for structural analysis and verification. The geometry of the cross-section is not limited, it is possible to combine pre-defined cross-sections from database (e.g. steel hot rolled cross-sections) and arbitrary shapes including openings and gaps. The program calculates following characteristics:

- The coordinates of centre of gravity - $x_t$, $y_t$
- The cross-sectional area - $A$
- The perimeter - $P$
- The coordinates of centre of gravity measured from left bottom corner - $y_{cg}$, $z_{cg}$
- The moments of inertia - $I_y$, $I_z$, $D_{yz}$
- The rotation of main axes - $\varphi$
- The radii of gyration - $i_y$, $i_z$
- The polar moment and radius of inertia - $I_p$, $i_p$
- The rigidity moment in simple torsion - $I_k$
- The cross-sectional moduli - $W_{y1}$, $W_{y2}$, $W_{z1}$ , $W_{z2}$

Cross-sectional characteristics are also described in the theoretical part of help.

### User interface

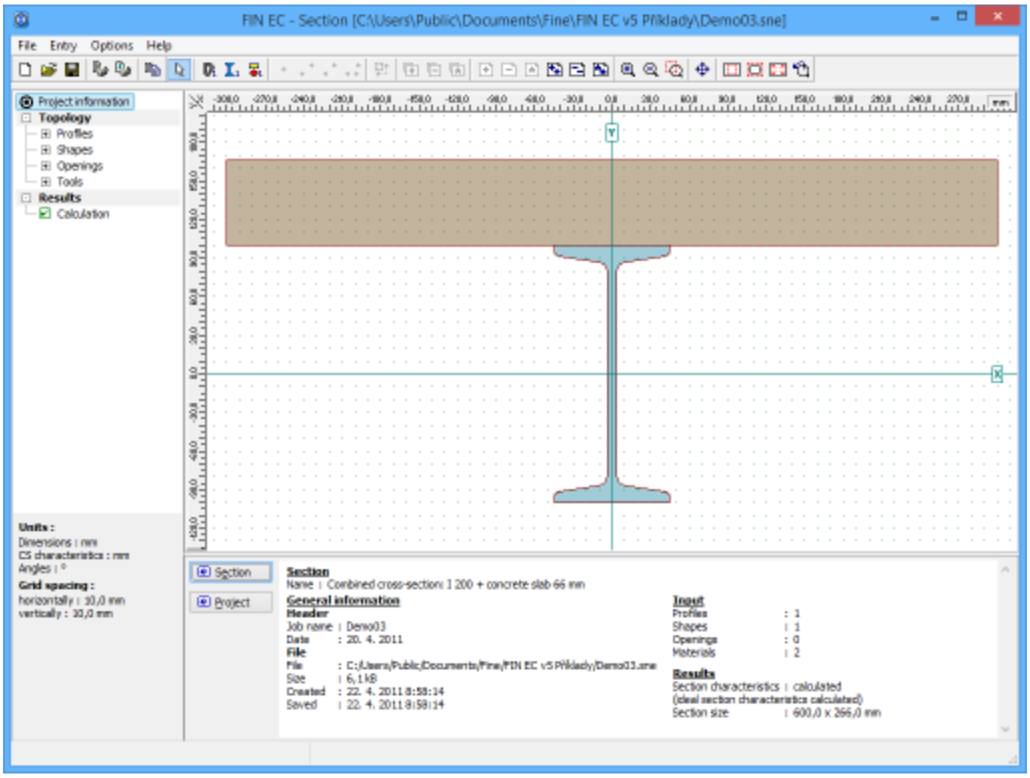
The user interface consists of a main menu with toolbars in the upper part of the window, tree menu on the left and input/display part on the right side of the window. The main menu contains all tools and functions, which can be used during the work. The tree menu is used for switching between parts of an input. The tree menu can be alternated by the part "**Entry**" of the main menu. The appearance of the workspace may be modified in the window "**Options**", that can be opened from the main menu. Tools for documents printing are organized in the window "**Print and export document**", which can be opened using the printing icon in the toolbar "**Files**" or using the appropriate link in the part "**File**" of the main menu.

The tree menu contains two main parts:

- **Topology** - This part contains tools for modelling the cross-section
- **Results** - This part shows the results of calculation

The default screen contains general information of the project. The cross-section name and additional notes can be specified with the help of the button "**Section**". The button "**Project**" opens the window "**General project data**", which contains additional data of the project, that can be used in heading and footing of documents.



*Main application window*

# Topology

This part contains tools for input of cross-section geometry. The topology is not limited, it is possible to combine arbitrary shapes and materials. The input is divided into four parts according to the object types:

- **Profiles** - This part contains pre-defined database of the most common cross-sections including wide range of steel hot-rolled profiles
- **Shapes** - The objects with arbitrary geometry can be added or modified in this part
- **Openings** - This part contains tools for input of openings
- **Tools** - This part contains tools for manipulation with objects (copy, move, mirror etc.)

The modelling principles should be respected during the input.

After finishing the input, it is possible to switch the tree menu into the part "**Results**" and view the calculated cross-sectional characteristics.

# Profiles

This part contains tools for the work with profiles. Profiles are pre-defined shapes organized in a default database. The database is sorted according to the material and it isn't possible to assign material, that does not correspond to the material group (e.g. concrete strength class cannot be assigned to hot-rolled I-profile). Profiles can be converted into general shapes or openings.

## Input of profiles

Profiles can be added graphically in the workspace with the help of the mode "**Add**" of the tree menu or numerically using the button "**Add**" in the toolbar on the left side of the bottom table. The choice of geometry and material is done in the window "**Profile**". The insertion point is specified by clicking in the workspace (graphical mode) or by entering coordinates (numerical input).

## Editing profiles

Following tools are available for editing profiles:

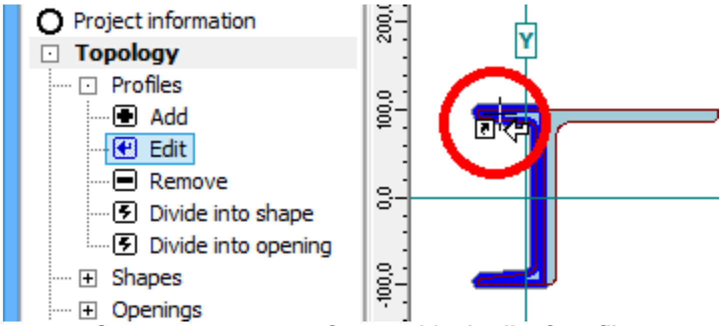
| | |
|---|---|
| **Edit** | • This tool provides an option to change the cross-section type or dimension in the window "**Profile**". This window contains also additional parameters like position, rotation and material. |
| **Divide into shape** | • This tool converts the profile into the object type "**Shape**". This conversion can be used in cases, when the pre-defined shape should be modified (add or remove vertex etc.) or when the material type does not suit user's needs. This conversion is non-reversible, general shape cannot be converted back into profile. |

**Divide into opening**

- This tool converts the profile into the object type "**Opening**". This conversion can be used in cases, when the opening has a shape, that is similar or identical to a profile from pre-defined database. This conversion is non-reversible, opening cannot be converted back into profile.

Modification can be done graphically with the help of tools in the tree menu (profile for editing is selected by clicking on the workspace) or with the help of toolbar along the table in the bottom frame (active profile for editing is highlighted by bold font). The focused profile for editing in the workspace is highlighted and also the cursor changes its appearance.

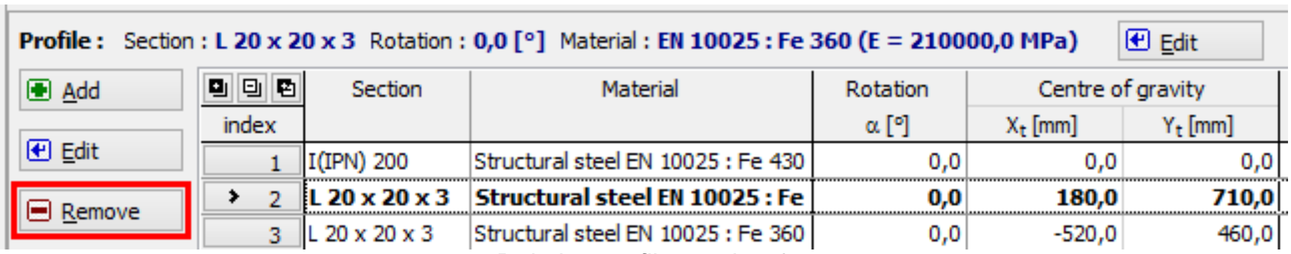



*Cursor appearance for graphical edit of profile*

The profiles may be also edited with the help of the functions in the part "**Tools**".

## Removing profiles

Profiles can be also deleted using two different ways: The tool "**Remove**" in the tree menu can be used for deleting the profiles in the workspace, the button "**Remove**" close to the profiles table deletes active profile (highlighted by bold font in the table and by blue colour in the workspace).



**Profile :** Section : **L 20 x 20 x 3** Rotation : **0,0 [°]** Material : **EN 10025 : Fe 360 (E = 210000,0 MPa)** Edit

Add / Edit / Remove

| index | Section | Material | Rotation α [°] | Centre of gravity $X_t$ [mm] | Centre of gravity $Y_t$ [mm] |
|---|---|---|---|---|---|
| 1 | I(IPN) 200 | Structural steel EN 10025 : Fe 430 | 0,0 | 0,0 | 0,0 |
| 2 | **L 20 x 20 x 3** | **Structural steel EN 10025 : Fe** | **0,0** | **180,0** | **710,0** |
| 3 | L 20 x 20 x 3 | Structural steel EN 10025 : Fe 360 | 0,0 | -520,0 | 460,0 |

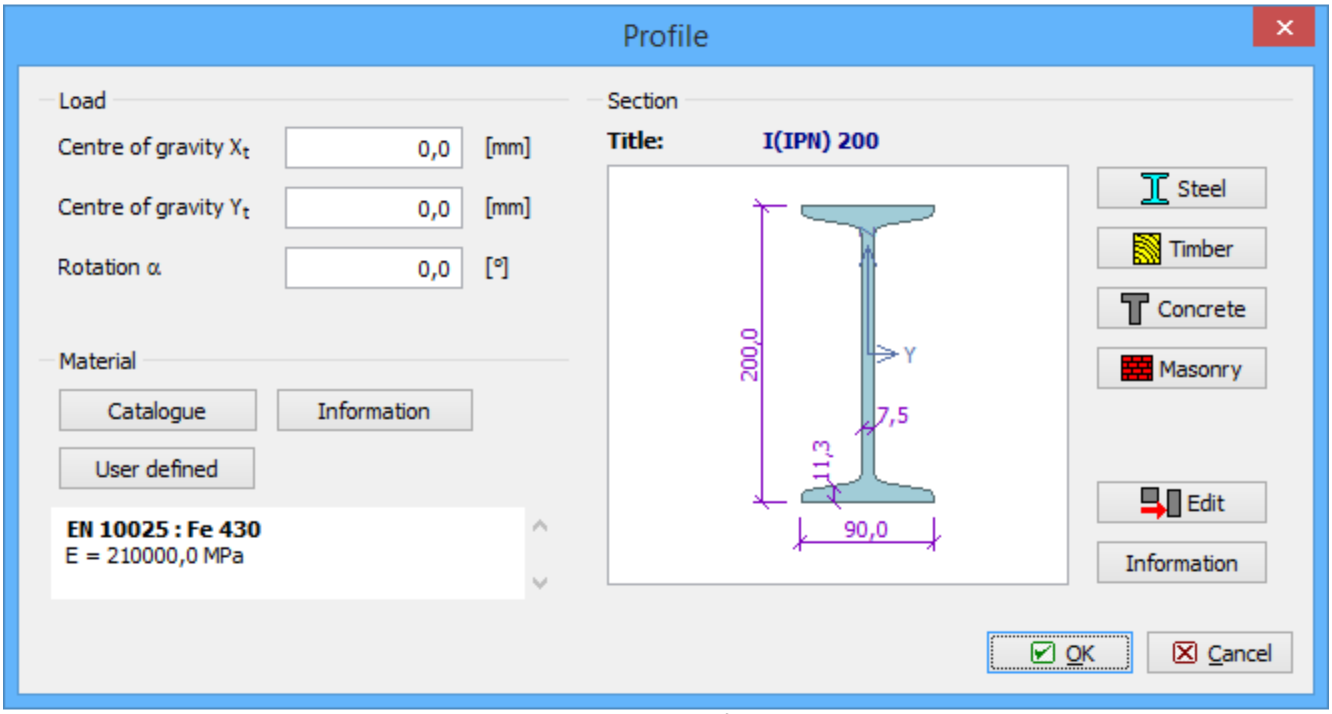
*Deleting profile number 2*

# Profile

The cross-section from pre-defined database can be selected in this window. The database is sorted according to the material type, buttons for opening corresponding part of the database (e.g. "**Steel**") are placed in the right part of the window. The choice of appropriate cross-section is done in the window "**Cross-section editor**". The existing cross-section can be edited by the button "**Edit**", cross-sectional characteristics can be viewed by the button "**Information**".

The window also contains the input fields for insertion point coordinates and profile rotation. Also material can be specified here. According to the structural rules, it is not possible to assign a material, that does not correspond to the profile type group (e.g. concrete strength class cannot be assigned to hot-rolled I-profile).




*Window "Profile"*

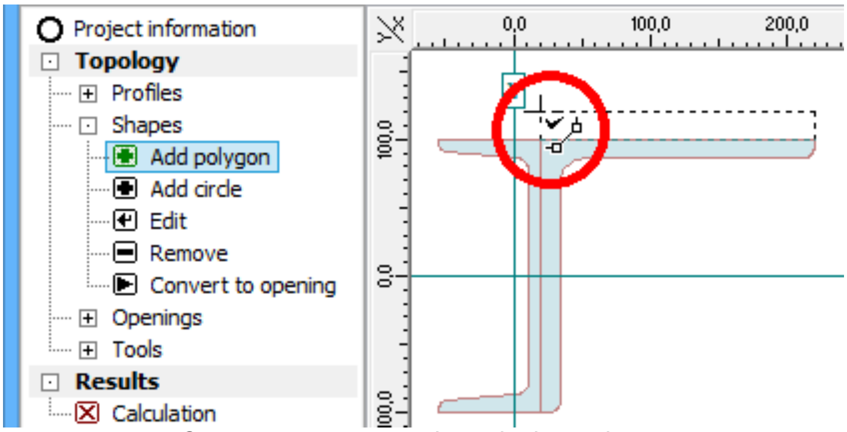
## Shapes

This part contains tools for the work with general polygons ("**Shapes**"). Shapes are divided according to the input style into two main types: general polygon and circle. The general polygons are not limited, only crossed polygons are not permitted. Any material can be assigned to a shape. Shapes can be converted into openings.

### Input of shapes

The input is divided into two options: input of general polygon and circle.

**General polygon** can be added graphically in the workspace with the help of the mode "**Add polygon**" of the tree menu or numerically using the button "**Polygon**" in the toolbar on the left side of the bottom table. In this case, the input of geometry and material is done in the window "**Polygon**". The graphical input is done by clicking into the positions of polygon vertexes on the workspace. The polygon has to be closed by clicking on the first vertex of the polygon. When closing the polygon, the cursor appearance changes.




*Cursor appearance when closing polygon*

The circle can be entered in a similar way: For graphical input (option "**Add circle**" in the tree menu), it is necessary to specify the centre and arbitrary point on the circle by clicking on the workspace. Alternative way is to use the button "**Circle**" in the table and specify the circle properties in the corresponding window.

### Editing shapes

Following tools are available for editing shapes:

| | | |
|---|---|---|
| **Edit** | • | This tool provides an option to change the geometry in the window "**Polygon**" or "**Circle**". This window contains also additional parameters like position, rotation and material. |
| **Convert to opening** | • | This tool converts the shape into the object type "**Opening**". This conversion is reversible, opening can be converted back into shape. |

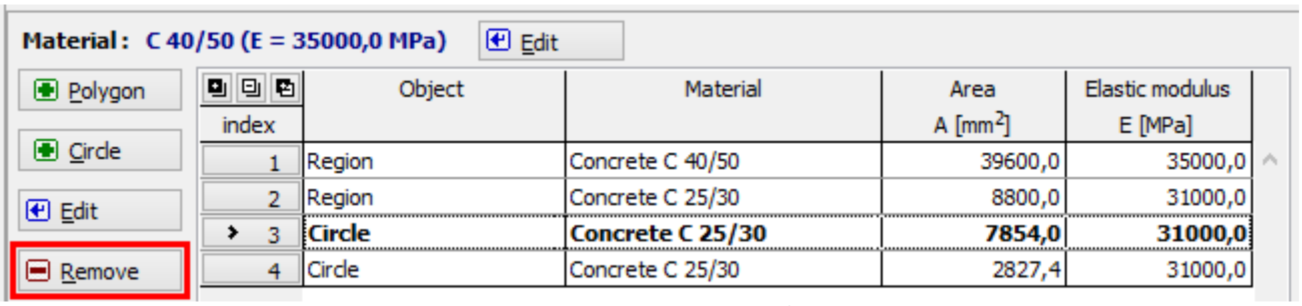
 

Modification can be done graphically with the help of tools in the tree menu (shape for editing is selected by clicking on the workspace) or with the help of toolbar along the table in the bottom frame (active shape for editing is highlighted by bold font).

The shapes may be also edited with the help of the functions in the part "**Tools**".

## Removing shapes

Shapes can be also deleted using two different ways: The tool "**Remove**" in the tree menu can be used for deleting the shapes in the workspace, the button "**Remove**" close to the shapes table deletes active shape (highlighted by bold font in the table and by blue colour in the workspace).




| index | Object | Material | Area A [mm$^2$] | Elastic modulus E [MPa] |
|---|---|---|---|---|
| 1 | Region | Concrete C 40/50 | 39600,0 | 35000,0 |
| 2 | Region | Concrete C 25/30 | 8800,0 | 31000,0 |
| 3 | **Circle** | **Concrete C 25/30** | **7854,0** | **31000,0** |
| 4 | Circle | Concrete C 25/30 | 2827,4 | 31000,0 |

*Deleting shape number 3*

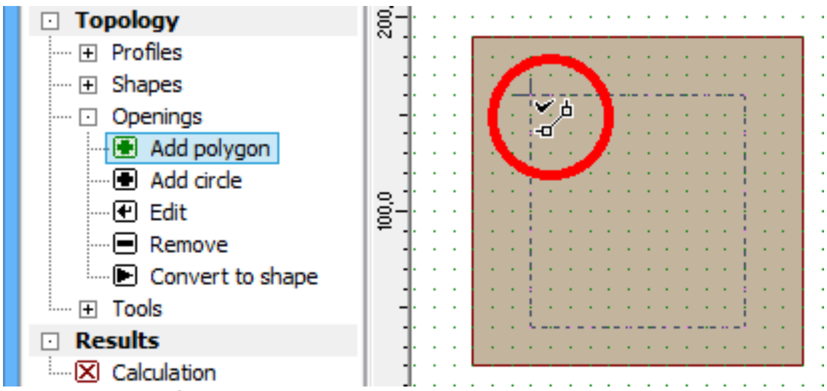
# Openings

This part contains tools for the work with openings (general polygons placed inside of "**profiles**" or "**shapes**"). Similarly to "**shapes**", they are divided according to the input style into two main types: general polygon and circle. The general polygons are not limited, only crossed polygons are not permitted.

## Input of openings

The input is divided into two options: input of general polygon and circle.

**General polygon** can be added graphically in the workspace with the help of the mode "**Add polygon**" of the tree menu or numerically using the button "**Polygon**" in the toolbar on the left side of the bottom table. In this case, the input of geometry and material is done in the window "**Polygon**". The graphical input is done by clicking into the positions of polygon vertexes on the workspace. The polygon has to be closed by clicking on the first vertex of the polygon. When closing the polygon, the cursor appearance changes.




*Cursor appearance when closing polygon*

The circle can be entered in a similar way: For graphical input (option "**Add circle**" in the tree menu), it is necessary to specify the centre and arbitrary point on the circle by clicking on the workspace. Alternative way is to use the button "**Circle**" in the table and specify the circle properties in the corresponding window.

## Editing profiles

Following tools are available for editing openings:

| | | |
|---|---|---|
| **Edit** | • | This tool provides an option to change the geometry in the window "**Polygon**" or "**Circle**". This window contains also coordinates of the position. |
| **Convert to shape** | • | This tool converts the opening into the object type "**Shape**". This conversion is reversible, shape can be converted back into opening. |

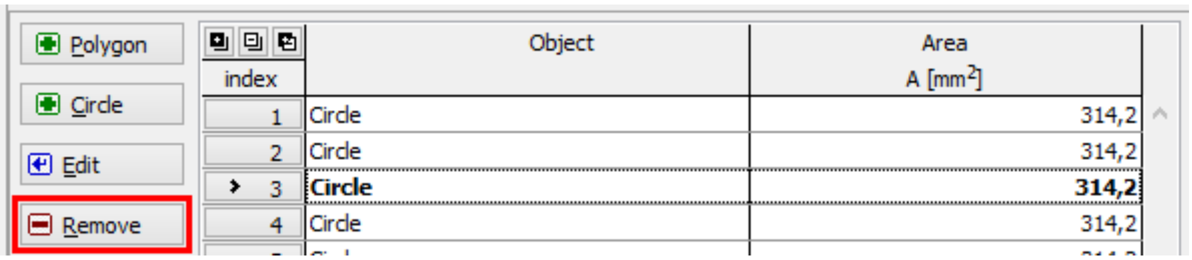
Modification can be done graphically with the help of tools in the tree menu (opening for editing is selected by clicking on the workspace) or with the help of toolbar along the table in the bottom frame (active opening for editing is highlighted by bold font).

The openings may be also edited with the help of the functions in the part "**Tools**".

### Removing openings

Openings can be also deleted using two different ways: The tool "**Remove**" in the tree menu can be used for deleting the openings in the workspace, the button "**Remove**" close to the table of openings deletes active opening (highlighted by bold font in the table and by blue colour in the workspace).




*Deleting opening number 3*

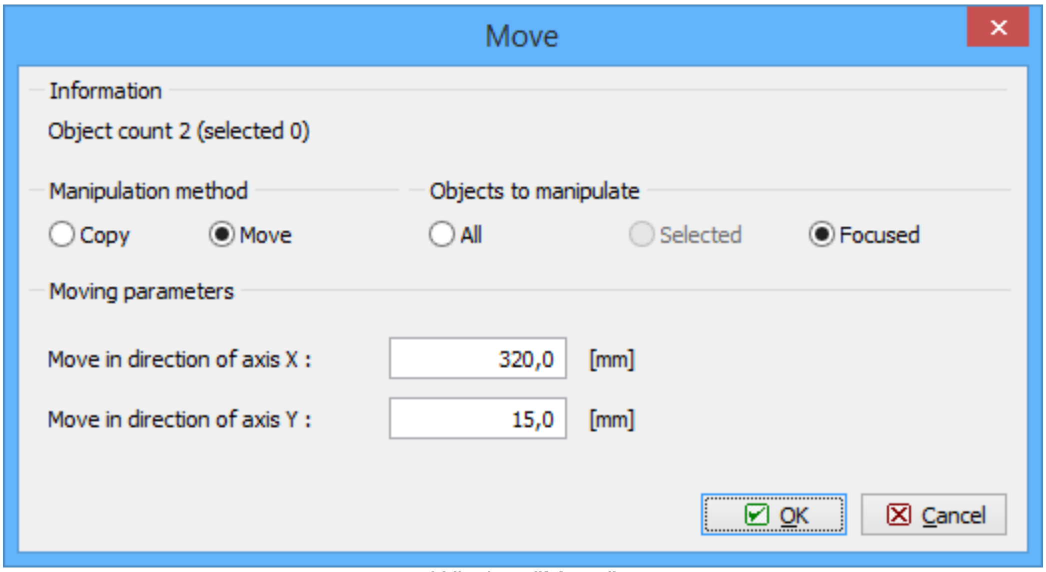
## Tools

This part contains tools that can be used for manipulation with objects (profiles, shapes and openings) from part "**Topology**". Following tools are available:

- **Move** - This tool moves selected object in given direction. The tool is able to work also in copy mode.
- **Rotate** - This tool rotates the specified object about the given centre point. The tool is able to work also in copy mode.
- **Mirror** - This tool mirrors the specified object with respect to mirror axis. The tool is able to work also in copy mode.
- **Align** - This rotates the object in that way, that its edge is parallel to the edge of sample object.
- **Incline** - This tool moves the object in that way, that its edge will be touching the edge of sample object.
- **Divide sector** - This tool inserts new nodes into selected sector
- **Remove vertex** - This tool removes specified vertex from shape or opening.
- **Remove** - This tool deletes given object.

## Move

The tool "**Move**" can be used for shift or copy of objects. The parameters of the manipulation can be specified in the window "**Move**", that appears after the selection of this tool.




*Window "Move"*

The "**Manipulation method**" sets, whether the tool will only change the position of an object or will keep existing object and create a new copy.

This tool is able to move or copy whole structure or only selected part. This behaviour can be specified in the part "**Objects to manipulate**". The option "**Selected**" is available only for structures, where are some selected objects (highlighted by green). The toolbar under the main menu contains buttons for switching on/off selections of openings, profiles and shapes.

*Toolbar for selections according to object types*

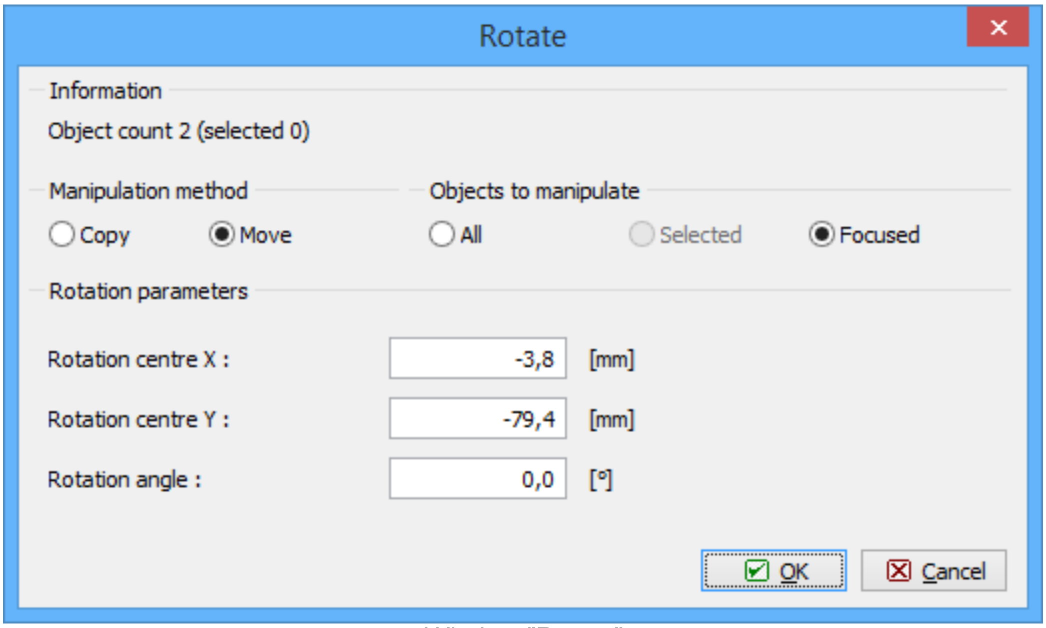
The objects have to be selected before running the tool in the tree menu. Otherwise, this option is not available.

*Copy of selected objects*

The last entry is the vector of transformation divided into two components according to the global axes *X* and *Y*.

## Rotate

The tool "**Rotate**" can be used for rotation or rotated copy of objects. The parameters of the manipulation can be specified in the window "**Rotate**", that appears after the selection of this tool. The rotation centre has to be specified first. The input may be performed in the bottom frame by entering joint numbers or coordinates or by clicking in the workspace.



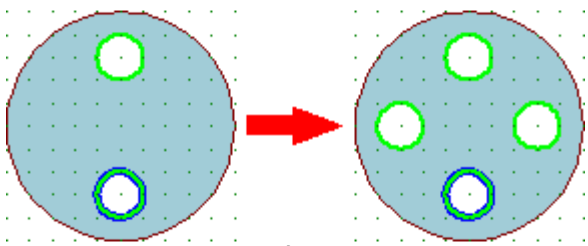

*Window "Rotate"*

The "**Manipulation method**" sets, whether the tool will only change the position of an object or will keep existing object and create a new copy.

This tool is able to rotate or copy whole structure or only selected part. This behaviour can be specified in the part "**Objects to manipulate**". The option "**Selected**" is available only for structures, where are some selected objects (highlighted by green). The toolbar under the main menu contains buttons for switching on/off selections of openings, profiles and shapes.

*Toolbar for selections according to object types*

The objects have to be selected before running the tool in the tree menu. Otherwise, this option is not available.



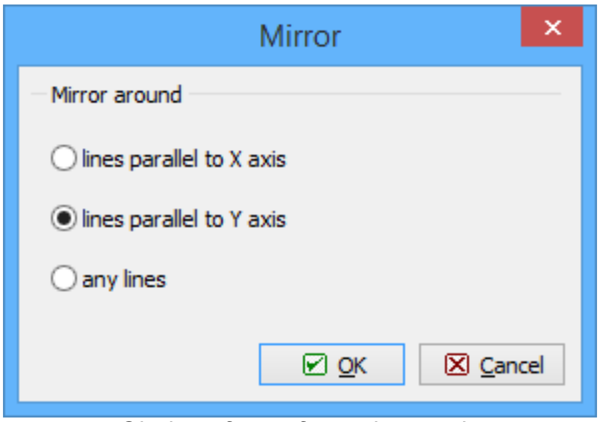
*Rotated copy of selected objects*

The last entries are the rotation centre and the rotation angle. Positive value means rotation in anti-clockwise direction.

## Mirror

This tool can be used for mirror of objects. There are three main modes of this tool: mirror using axis parallel to axes *X* or *Y* axis and using general axis. The choice of the mode has to be done in the window, that appears after clicking on the tool in the tree menu.

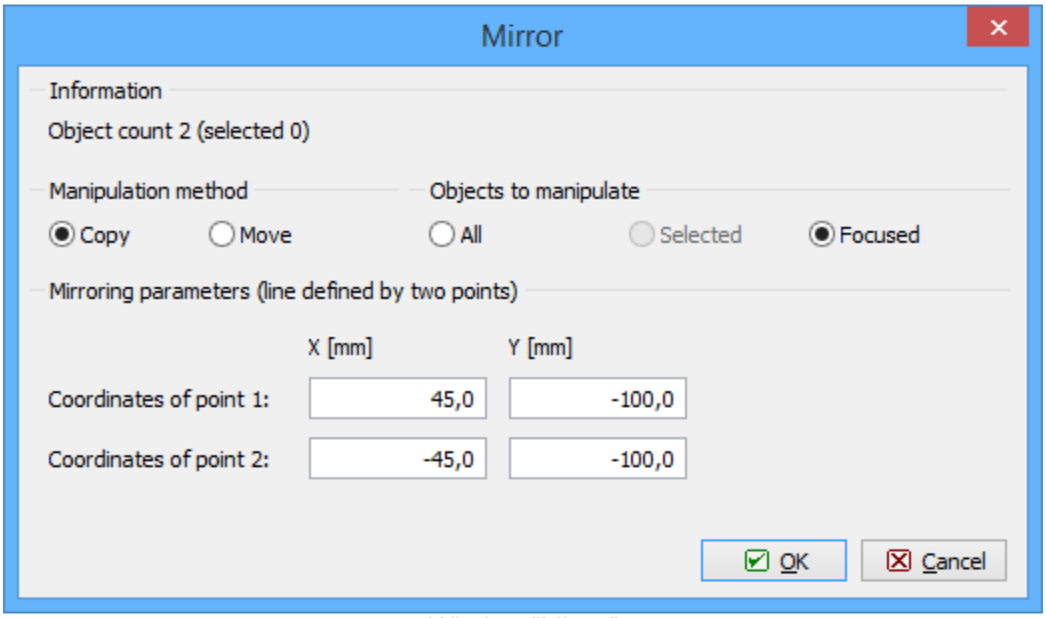
 




*Choice of transformation mode*

The mirror axis for first and second option is specified by the distance from corresponding global axis. The axis for third option is given by two points, that can be specified both by clicking in the workspace and by coordinates in the bottom frame.

The parameters of the manipulation can be specified in the window "**Mirror**", that appears after the selection of this tool.



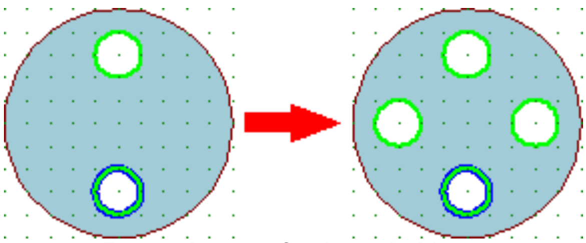

*Window "Mirror"*

The "**Manipulation method**" sets, whether the tool will only change the position of an object or will keep existing object and create a new copy.

This tool is able to mirror whole structure or only selected part. This behaviour can be specified in the part "**Objects to manipulate**". The option "**Selected**" is available only for structures, where are some selected objects (highlighted by green). The toolbar under the main menu contains buttons for switching on/off selections of openings, profiles and shapes.

*Toolbar for selections according to object types*

The objects have to be selected before running the tool in the tree menu. Otherwise, this option is not available.



*Rotated copy of selected objects*

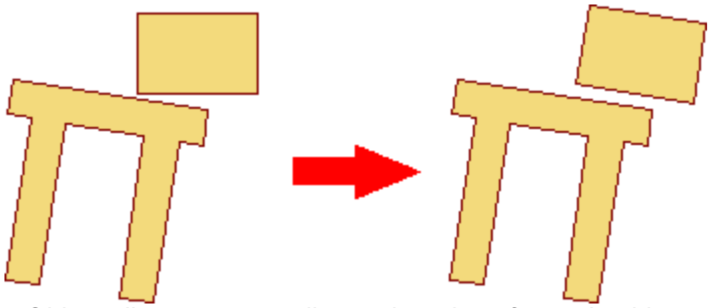
The last entry is the position of mirror axis. This axis was already specified, but it is possible to change the position.

# Align

This tool can be used for alignment of the object according to the angle given by a sample object.

For the alignment, it is necessary to select two vertexes on the object for the alignment and two vertexes on the sample object. The tool rotates the object for the alignment about its centre of gravity in that way, that the line given by two vertexes on aligned object is parallel to the line given by vertexes on the sample object. As two points define an oriented line, it is necessary to respect the order of input.



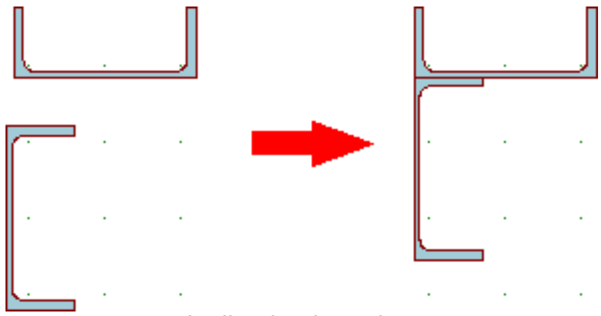
*Object alignment according to the edge of sample object*

# Incline

This tool can be used for moving the object in that way, that reference points on both objects will have identical coordinates. It is possible to use vertexes or general points on polygon sectors as reference points.

## Inclination by points

This option is faster, however, it can be used only in cases, when the reference points on both objects are vertexes. As an input, it is necessary to specify the reference vertex on the object for manipulation and after that the reference vertex on the sample object. The tool moves the object in that way, that reference vertexes will have identical position.

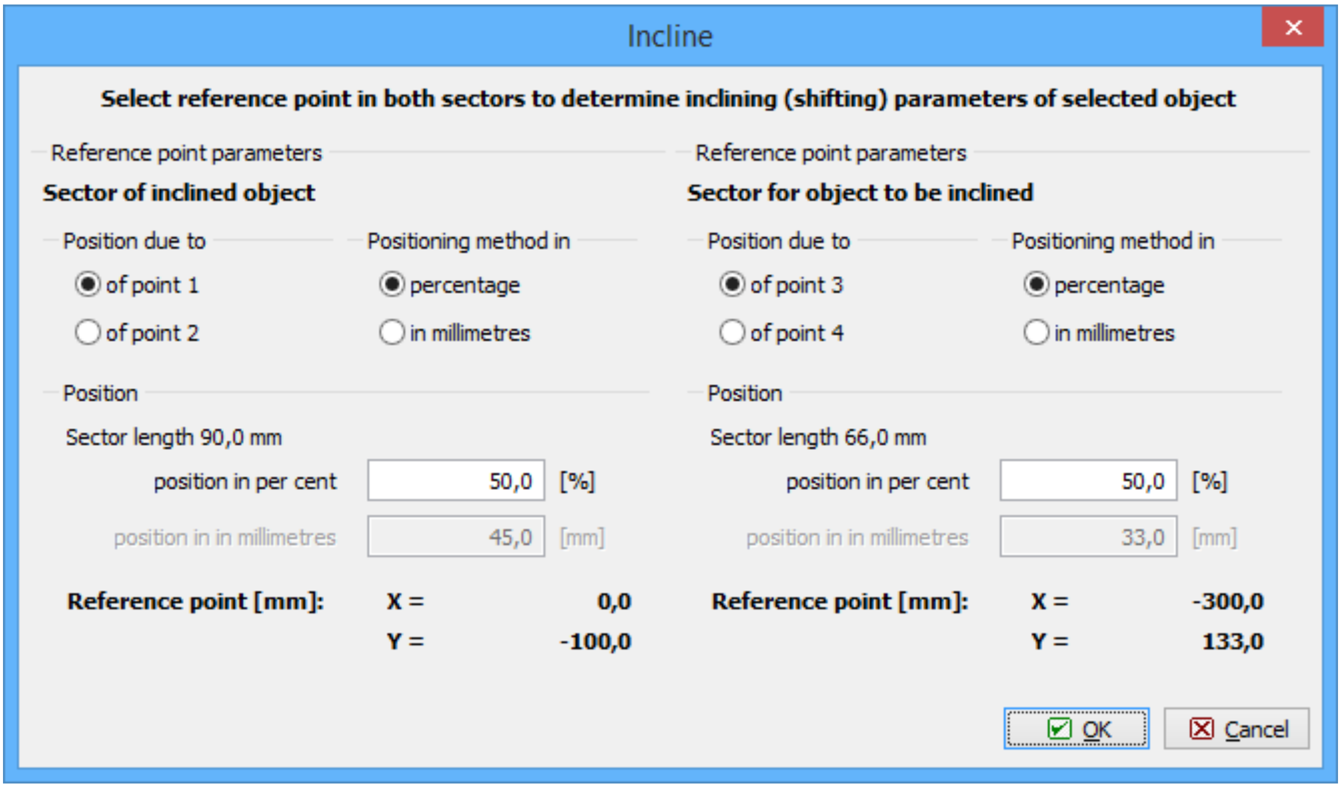


*Inclination by points*

## Inclination by sectors

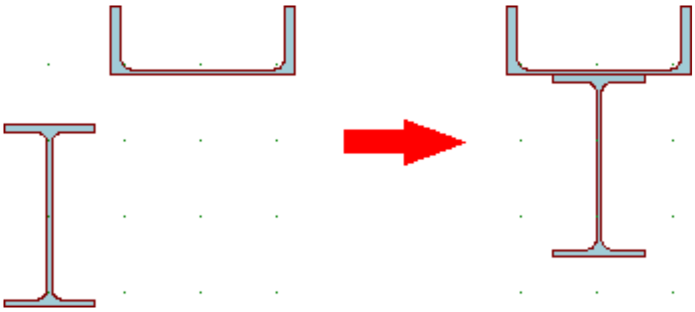
Inclination by sectors can be used in cases, where the objects won't have any vertexes with identical coordinates. The positions of reference points are specified in the window "**Incline**".




*Window "Incline"*

Reference points may be specified in per cents or in millimetres, positions can be measured from beginning or end of the sector.

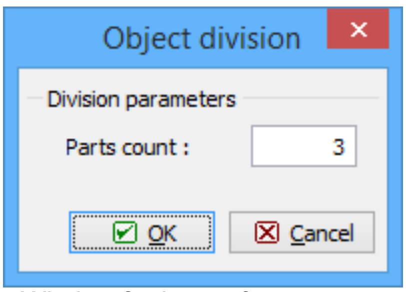


*Inclination by sectors*

## Divide sector

This tool adds new vertexes into selected sector of polygon. This tool can be used in cases, when the object should be reduced (create semi circle from circle) or extended (create T-shape from rectangle).

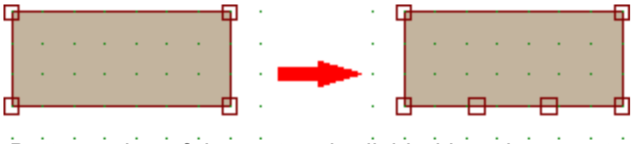
The specified sector is divided by inserted vertexes into parts with identical lengths. After secting the mode "**Divide sector**" in the tree menu, the window "**Object division**" appears. The number of created parts can be specified in this window.




*Window for input of parts count*

The input has to be confirmed by the button "**OK**". The software creates new nodes on selected edge.

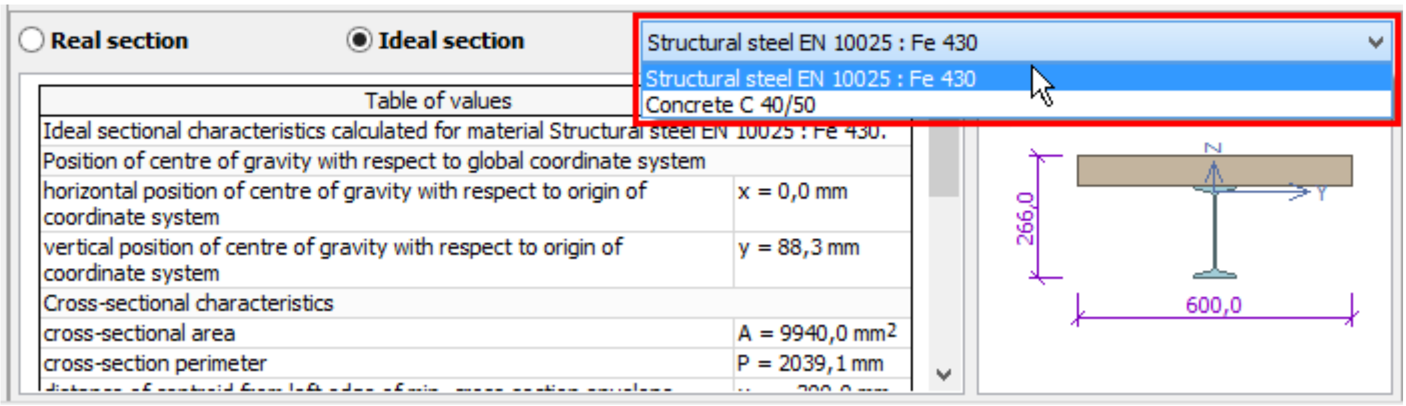


*Bottom edge of the rectangle divided into three parts*

## Results

This part shows the calculation results. The cross-sectional characteristics can be calculated using two different ways:

- **Real cross-sectional characteristics** are calculated according to the geometry of the cross-section and are not affected by the material.
- **Ideal cross-sectional characteristics** are important for combined cross-sections with more different materials (e.g. concrete and steel). Ideal characteristics are the characteristics recalculated for the material, that can be specified in the upper part of the frame with results. Recalculation is based on the proportion of moduli of elasticity.



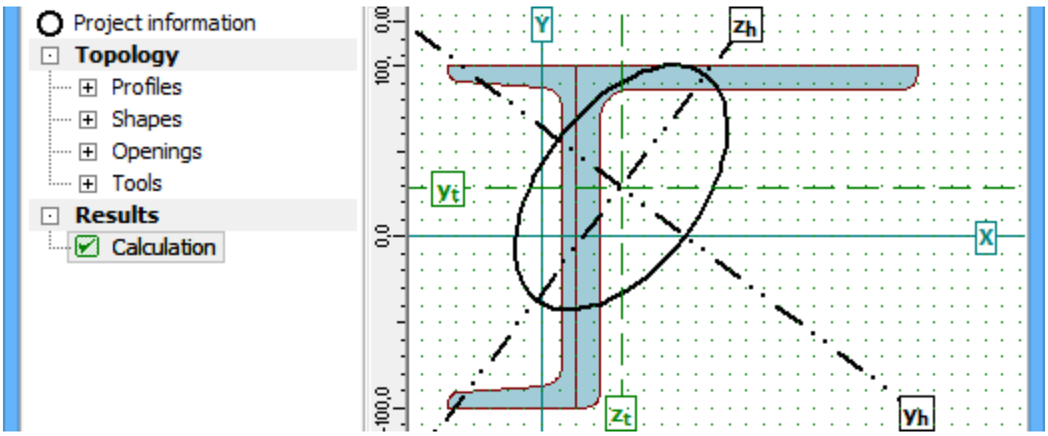

*Choice of material for ideal cross-section*

The procedure of calculating the ideal cross-sectional characteristics is described in the theoretical part of help.

### Results in the workspace

The workspace shows following results:

- **The centre of gravity** - this point is located in the intersection of axes $y_t$ and $z_t$ (dashed lines)
- **The ellipse of inertia**
- **The main axes of the cross-section** - bold axes marked $y_h$ and $z_h$




*Results in the workspace*

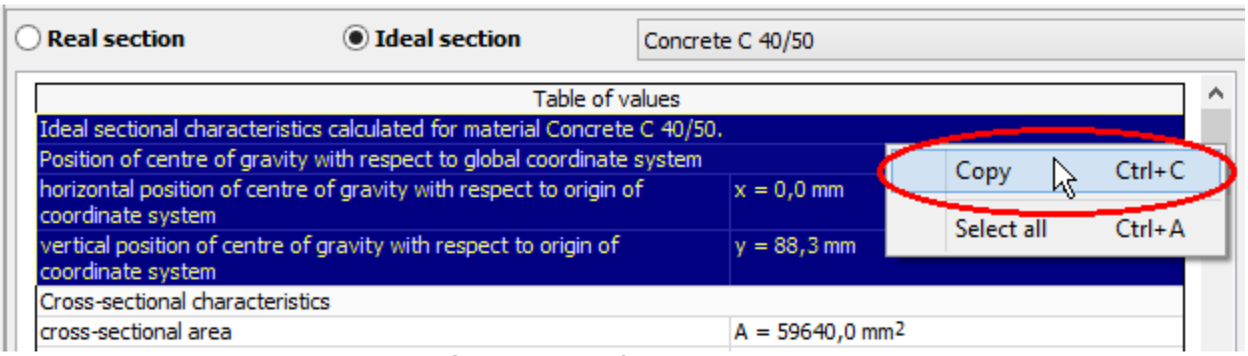
The picture in the workspace can be copied into clipboard with the help of shortcut *Ctrl+C* or the button "" in the toolbar above the workspace. The picture in the clipboard can be pasted into arbitrary document or graphic editor.

### Table with results

The bottom frame contains the table with all calculated characteristics. The data in table can be selected by the cursor and copied into another program (text editor). The table contains a context menu, that can be opened with the help of right mouse button.

The calculated cross-sectional characteristics are described also in the theoretical part of help.




*Context menu for results table*

# Program Parametric temperature curve

This program calculates the characteristics of the parametric temperature curve according to the specified fire load and topology of the fire compartment.

## General rules

The window contains input part on the left side, the compartment topology in the right upper corner and the results in the bottom part of the window. Documentation can be printed with the help of the window "**Print and export document**", that may be launched from main menu ("**File**" - "**Print**"), from the horizontal toolbar or with the help of the shortcut *Ctrl+P*.

There's an option to specify general data of the project (identification details, design standard), that may be used in heading and footing of documents. These data are organized in the window "**General project data**", which can be launched with the help of the main menu ("**Options**" - "**Project**"). The program also contains the window "**About the company**" with identification data of the company (contacts, list of designers etc.). Content of this window is shared for all Fin EC programs. Also these inputs may be added into documentation. This window can be opened with the help of link "**Company**" in the part "**Options**" of the main menu.

Following inputs affects the calculation of parametric temperature curve:

## Fire zone walls

This part contains a table for the input of compartment topology. The shape of the compartment isn't limited. The maximum wall height is *4m*, which is the fundamental limit for parametric curve according to the EN 1991-1-2. Any wall may contain one opening. More openings in one wall has to be specified as one opening with the width, that is equal to the sum of widths of all openings in the wall, and width the area, which is equal to the sum of areas of all openings. The materials of walls are described by the density, specific heat and the thermal conductivity. The materials of particular walls may differ.

## Materials of floor and ceiling

These parts contain characteristics of floor and ceiling, the characteristics are identical to the properties of the walls material.

## Fire parameters

Following parameters are necessary for the calculation:

**Time of fire development $t_{lim}$**

- It depends on the velocity of the fire development, following values are written in the standard: *15min* for high velocity of the fire development (e.g. theatres, cinemas, shopping centres, libraries), *20min* for medium velocity of the fire development (e.g. living areas, hospitals, hotels, office areas, schools) and *25min* for low velocity of the fire development (e.g. transport areas).

**Characteristic fire load density per unit floor area $q_{f,k}$**

- This value depends on the type of occupancy. The value is based on the table *E.4*, the range of recommended values is *50-2000 MJ/m²*.

**Factor related to the fire activation risk due to the size of the compartment $\delta_{q1}$**

- This factor describes the dependency of fire activation risk and the size of fire compartment. The values are based on the table *E.1* and are between *1.1* (compartment area *25m²*) and *2.13* (area *10000m²*).

**Factor related to the fire activation risk due to the type of occupancy $\delta_{q2}$**

- This factor describes the fire activation risk for certain occupancy types. The value is between *0.78* (galleries, museums, swimming pools) and *1.66* (production areas of fireworks and paints) according to the table *E.1*.

**Factor related to the different active fire fighting measures $\delta_n$**

- This factor takes into account the different active fire fighting measures like sprinkler, detection, automatic alarm transmission, firemen etc. The values are based on the table *E.2* and may be considered as *1.0* for normal fire fighting measures.

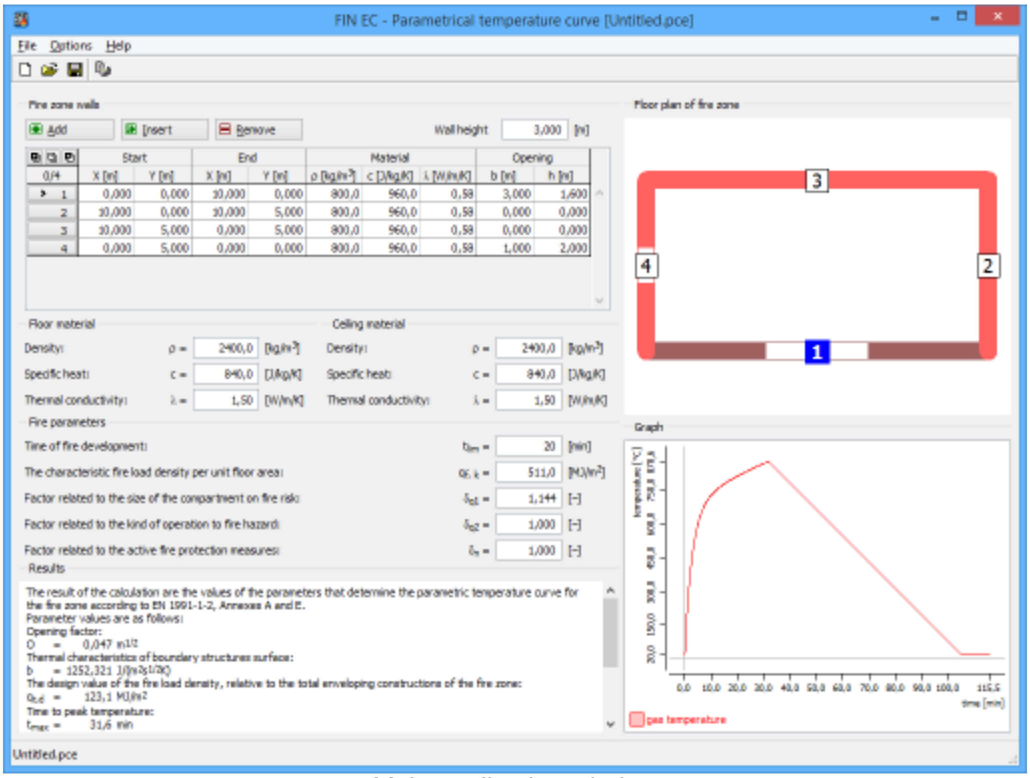
 

## Results

The results are displayed in three different ways. First, the parametric temperature curve is displayed as a graph, which shows the dependency of gas temperature in time. Second option are following four parameters, that describe this curve:

| | | |
|---|---|---|
| ***O*** | • | The opening factor |
| **b** | • | The thermal absorptivity for the total enclosure |
| $q_{t,d}$ | • | The design value of the fire load density related to the total surface area |
| $t_{max}$ | • | The time for maximum gas temperature |

These parameters are suitable for following calculations in other programs (e.g. "**Steel Fire**", "**Concrete fire**"). The third way shows the expressions, that describe the curve course. First phase (heating) of the curve is described by the exponential curve, second phase (cooling) is a straight line. These expression may be used for the calculation of gas temperature for arbitrary time.




*Main application window*

# Program Heat transfer

This program calculates the heat transfer into steel structure during the fire exposition. The results (temperature of steel and gas depending on time) are displayed in a graph.

## General rules

The window contains input part on the left side and graph with results on the right side. The graph is regularly updated according to the specified inputs. Documentation can be printed with the help of the window "**Print and export document**", that may be launched from main menu ("**File**" - "**Print**"), from the horizontal toolbar or with the help of the shortcut *Ctrl+P*.

There's an option to specify general data of the project (identification details, design standard), that may be used in heading and footing of documents. These data are organized in the window "**General project data**", which can be launched with the help of the main menu ("**Options**" - "**Project**"). The program also contains the window "**About the company**" with identification data of the company (contacts, list of designers etc.). Content of this window is shared for all Fin EC programs. Also these inputs may be added into documentation. This window can be opened with the help of link "**Company**" in the part "**Options**" of the main menu.

Following inputs affects the calculation of heat transfer:

## Cross-section properties

The cross-section of the member can be specified in this part. Following options are available:

| | | |
|---|---|---|
| **Hot-rolled** | • | The input of *I*-profile using the pre-defined database |
| **Welded** | • | The input of *I*-profile with user defined dimensions |

**General** • The input of general cross-section with the help of cross-sectional area and perimeter

The database and profile dimensions are organized in the window "**Cross-section editor**", that appears after clicking on the button "**Edit**".

## Temperature curve properties

The temperature curve that is used for the determination of the temperature of gas in time may be selected here. Following options are available:

- **Standard temperature curve** - nominal curve defined in EN 13501-2. This curve describes the fully developed fire.
- **External fire curve** - nominal curve intended for the outside of separating external walls that can be exposed to fire from different parts of the facade (directly from the inside of the corresponding fire compartment or from a compartment situated below or adjacent to the respective external wall)
- **Hydrocarbon fire curve** - nominal curve for representing effects of a hydrocarbon type fire
- **Parametric temperature curve** - this curve is effected by the physical parameters that describe the conditions in the fire compartment.

The expressions that represent temperature curves are described in the chapter "**Temperature development**" of theoretical help.

## Fire protection properties

This part contains a selection of fire detail (style of fire protection). Details are divided into two basic categories: unprotected ones and protected ones. The sorting according to the number of exposed sides follows for both categories.

Unprotected cross-sections may be exposed to fire from all sides or only from three sides (in case that the cross-section is covered from one side e.g. by a slab). Cross-sections may be also protected by concrete slab partially (part of cross-section height is fixed with concrete). For such cases, protected height ($h_{pr}$) or exposed height ($h_{exp}$) has to be specified.

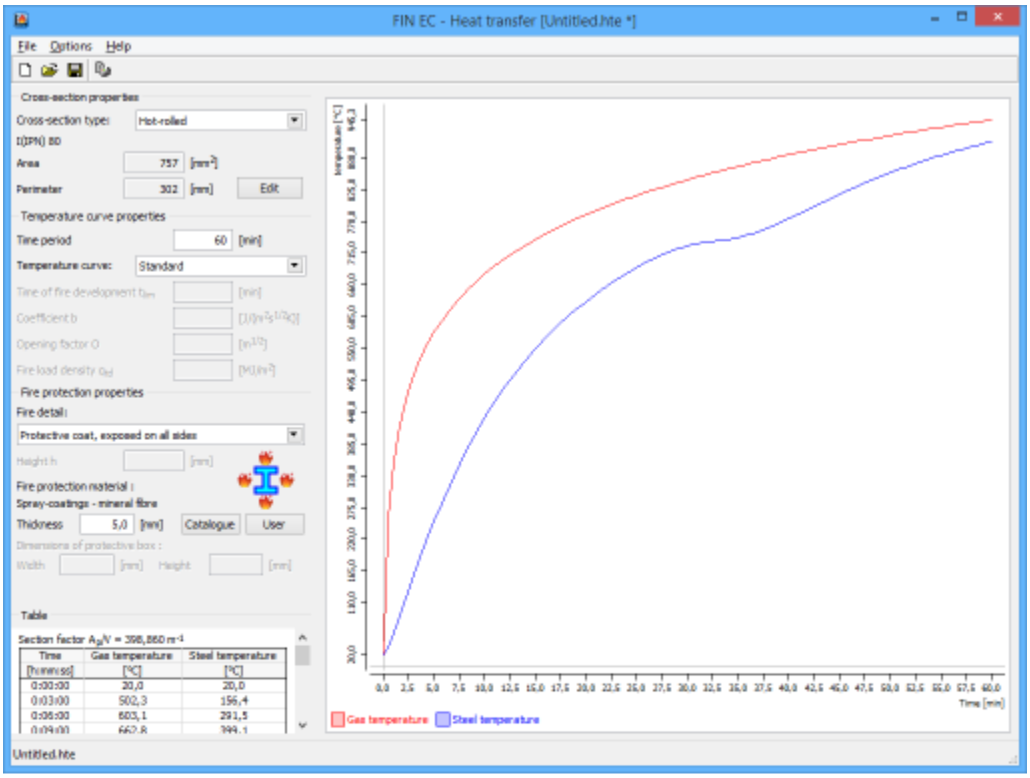
There are two general types of fire protection: coatings (the thickness $d_p$ has to be specified) and protected boxes (inputs are the thickness $d_p$ and box size). Protected details are also differentiated according to the number of exposed sides.

Materials of a fire protection may be divided into two main categories: coatings and board materials for protected boxes. The materials database contains wide range of items for both categories. Any other material may be specified manually with the help of user defined material characteristics.

Following buttons are available for the input of fire protective material:

**Catalogue** • The selection of material from pre-defined database in the window "**Materials catalogue**".

**User defined** • The input of arbitrary material with the help of material characteristics in the window "**Material editor**".




*Main application window*

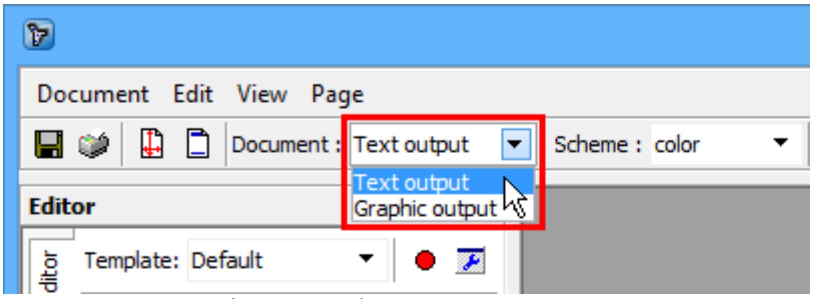
# Document printing

Window "**Print and export document**" contains the tools for printing the documentation of the project. Document structure can be defined in the tree menu, that is placed in the left part of the window. Other tools (print, heading and footing, document browsing) are available in the horizontal toolbar or in the main menu. The content of the document is updated automatically, all values and results correspond to the latest state of the project. Created document can be printed directly or saved as a file. Supported file formats are **.pdf* or **.rtf*. Documents created in the programs "**Fin 2D**" and "**Fin 3D**" may contain user defined pictures. The work with these pictures is described in the chapter "**Pictures printing**".

The most of the programs are able to create these types of documentation:

- **Text output** - Detailed document with variable content. The document can contain not only all inputs and results, but also detailed properties of materials and cross-sections and intermediate results of the analysis. Content of the document can be modified using the tree menu.
- **Graphic output** - brief document with one page per task. All inputs and results for each task are organized in a clear form.

Document type can be changed in the drop-down menu "**Document**" in the horizontal toolbar.

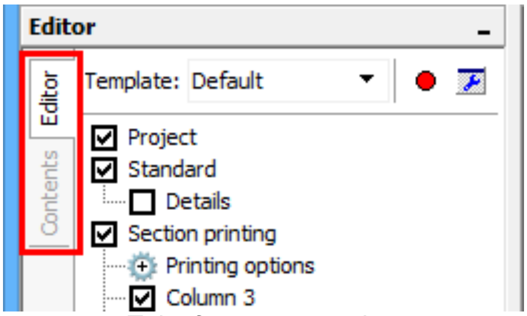



*Selection of the document type*

## Document content

Content of the document can be modified using tree menu in the left part of the printing window. The particular chapters of the document can be added or removed using appropriate check boxes. Any state of checked and unchecked chapters can be stored as a template. These templates will be available for all other projects.

Content of the final document can be displayed in place of the tree menu. The tabs for switching the tree menu and content view are placed on the left side of the tree menu.




*Tabs for a content view*

Browsing through the document is possible with the help of the mouse wheel or buttons in the toolbar.

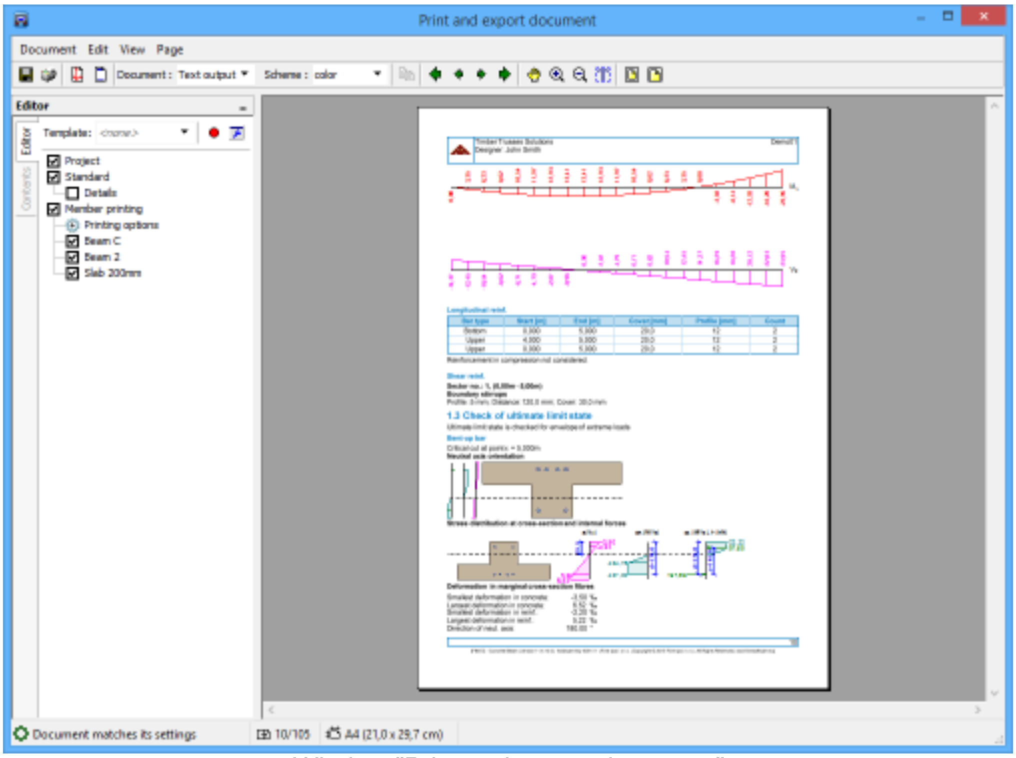
## Document appearance

Printing window also contains tools for modifying the appearance of the final document:

- **Heading and footing** - The heading and footing can be added in the window "**Heading and footing**".
- **Paper size and orientation** - The paper size, it's orientation and margins can be changed in the window "**Page setup**".
- **Font size** - The font size in documents can be also modified in the window "**Page setup**".
- **Page numbering** - The numbering style including prefix and suffix can be selected in the window "**Page numbering**".

These tools are available in the main menu or in the toolbar.

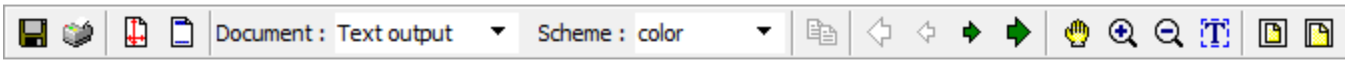
 



*Window "Print and export document"*

## Toolbar Print

The toolbar in the window "**Print and export document**" contains tools for changing the document properties, browsing the document and printing.



*Toolbar "Printing"*

The toolbar contains these buttons:

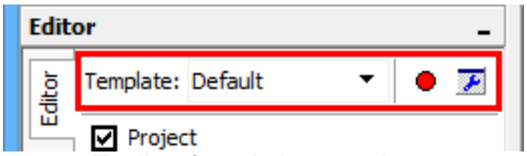
| | |
|---|---|
| **Save as** | • Save created document as a file. The document can be saved using **.PDF* or **.RTF* (editable) file format. |
| **Print** | • Print created document. |
| **Page properties** | • Change of the page size and orientation in the "**Page properties**" window. |
| **Heading and footing** | • Specify the heading and footing of the document with the help of window "**Heading and footing**". |
| Document : Text output — **Document type** | • Select the document type (text document or simple graphical output) |
| Scheme : color — **Colour scheme** | • Select the colour scheme (color, grey scale, black/white) |
| **Copy** | • Copy selected text into clipboard. |
| **First page** | • Display first page of the document |
| **Previous page** | • Display previous page of the document |
| **Next page** | • Display next page of the document |
| **Last page** | • Display last page of the document |
| **Pan** | • Pan tool for moving the displayed section |
| **Zoom in** | • Mode for enlargement of the document view. The mode can be terminated by clicking the right mouse button. |
| **Zoom out** | • Mode for reduction of the document view. The mode can be terminated by clicking the right mouse button. |
| **Text selection** | • Select the text in the document. |
| **Fit page** | • Optimize the scale to see whole page. |
| **Fit width** | • Optimize the scale according to the width of the page. |

# Printing templates

Printing template is able to save checked and unchecked items in the tree menu of the printing window for the future work. Any saved template (state of the tree menu) is saved using unique name and can be used in any other project. The templates are suitable for saving the structure of the most common documents.

The tools for the work with printing templates are organized in the dedicated toolbar, that is placed in the heading of the tree menu.




*Toolbar for printing templates*

## Saving templates

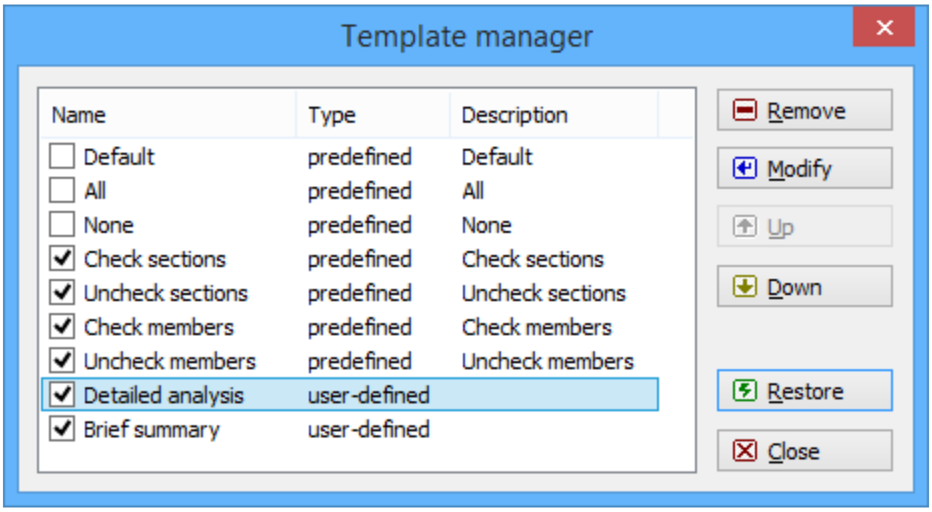
New template (state of all check boxes in the tree menu) can be saved using button "●" in the toolbar. The template name has to be specified before saving.

## Templates selection and management

The template can be selected using the drop-down menu "**Template**" in the toolbar that is placed in the heading of the tree menu. The list contains these predefined templates:

- **Default** - standard content of the document. The document contains all inputs and results. Detailed information isn't included.
- **All** - check all items in the tree menu
- **None** - uncheck all items in the tree menu
- **Check sections/members** - check all tasks of type "**section**" or "**member**"
- **Uncheck sections/members** - uncheck all tasks of type "**section**" or "**member**"

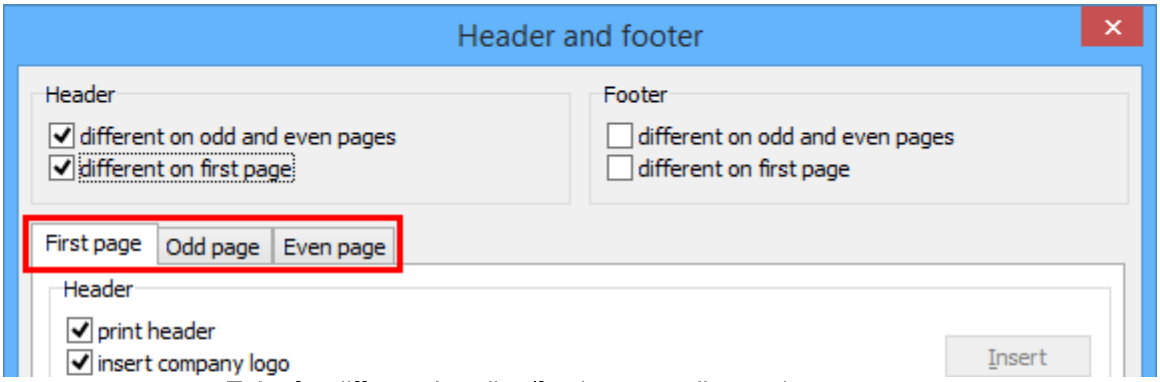
The "**Template manager**" window can be opened using button "" in the template toolbar. This window provides tools for making changes in the list of templates (renaming, removing). The predefined templates can't be modified or removed. Any template can be hidden in the drop-down menu of the template toolbar using check box in front of the item's name.




*Window "Template manager"*

# Heading and footing

The heading and footing including the content can be enabled with the help of this window. Settings in the upper part can set different rules for headings and footings in the odd and even pages and also for the first page. If used, the headings and footings are specified in unique tabs for each page type.



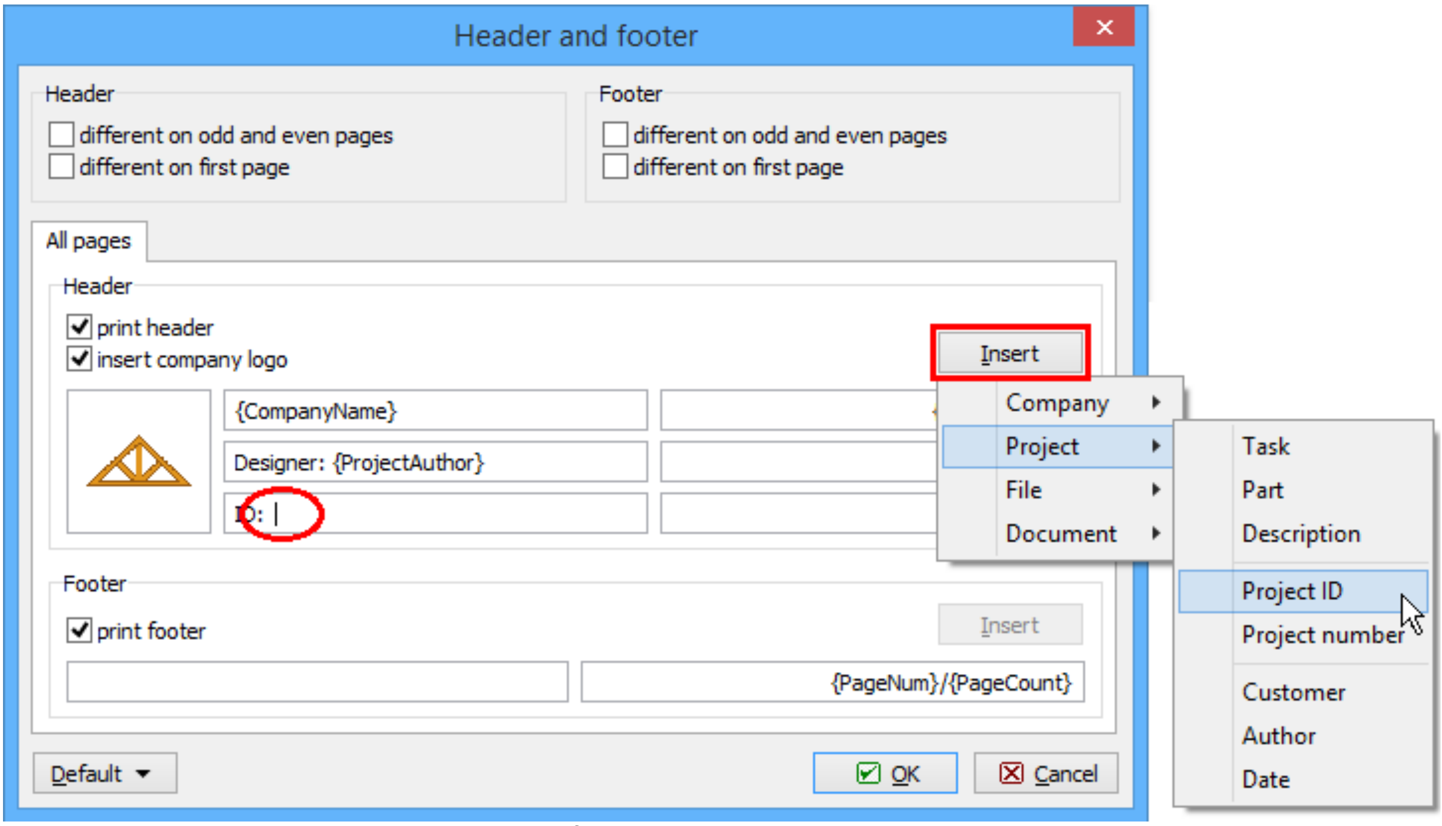

*Tabs for different heading/footing according to the page type*

Check boxes "**Print header/footer**" enable or disable the heading or footing of the document. The company logo can be displayed in the heading using check box "**Insert company logo**". Logo has to be loaded in the "**About the company**" window.

Every row of the heading or footing may contain any text or project variables. The project variables are entries, that are already specified in the another part of the project. These variables are supported:

- window "**About the company**" - basic company properties (name, address, contacts etc.)
- window "**General project data**" - project properties (project name, designer)
- system data - general properties of the document (date, time, page numbering)

Variables can be inserted using button "**Insert**" (the list of variables appears). The button will be enabled, if the cursor is places in an input line of heading or footing. Inserted variables are written in the internal format and are separated by curly brackets. Variables can be combined arbitrarily.




*Insertion of the variable into the cursor position*

Button "**Default**" contains these two tools:

| | | |
|---|---|---|
| **Adopt default settings** | • | Set the default values for all parameters. |
| **Save settings as default** | • | Set entered parameters as defaults for new projects. |

The default settings are saved separately for text documents and for screen prints. Settings are shared for all Fin EC programs.

## Page properties

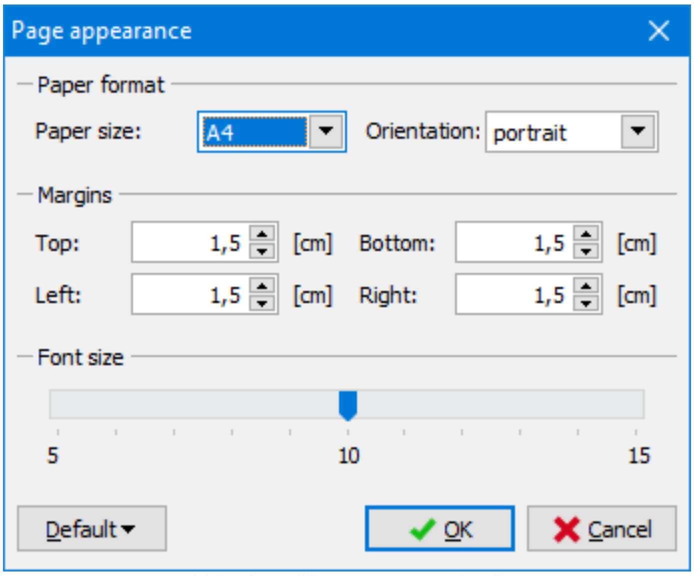
The page properties (paper size and orientation, margins) and font size can be specified in this window.

Button "**Default**" contains these two tools:

| | |
|---|---|
| **Adopt default settings** | • Set the default values for all parameters. |
| **Save settings as default** | • Set entered parameters as defaults for new projects. |

The default settings are saved separately for text documents and for screen prints. Settings are shared for all Fin EC programs.




*Window "Page properties"*

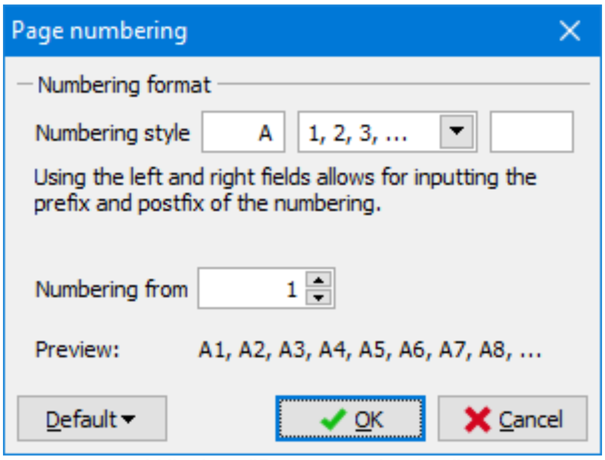
## Page numbering

The rules for page numbering can be specified in this window. The type of numerals can be selected in "**Numbering style**". There's also an option for insertion of prefix or suffix there. The number of the first page can be specified with the help of the setting "**Numbering from**".

Button "**Default**" contains these two tools:

| | |
|---|---|
| **Adopt default settings** | • Set the default values for all parameters. |
| **Save settings as default** | • Set entered parameters as defaults for new projects. |

The default settings are shared for all Fin EC programs.



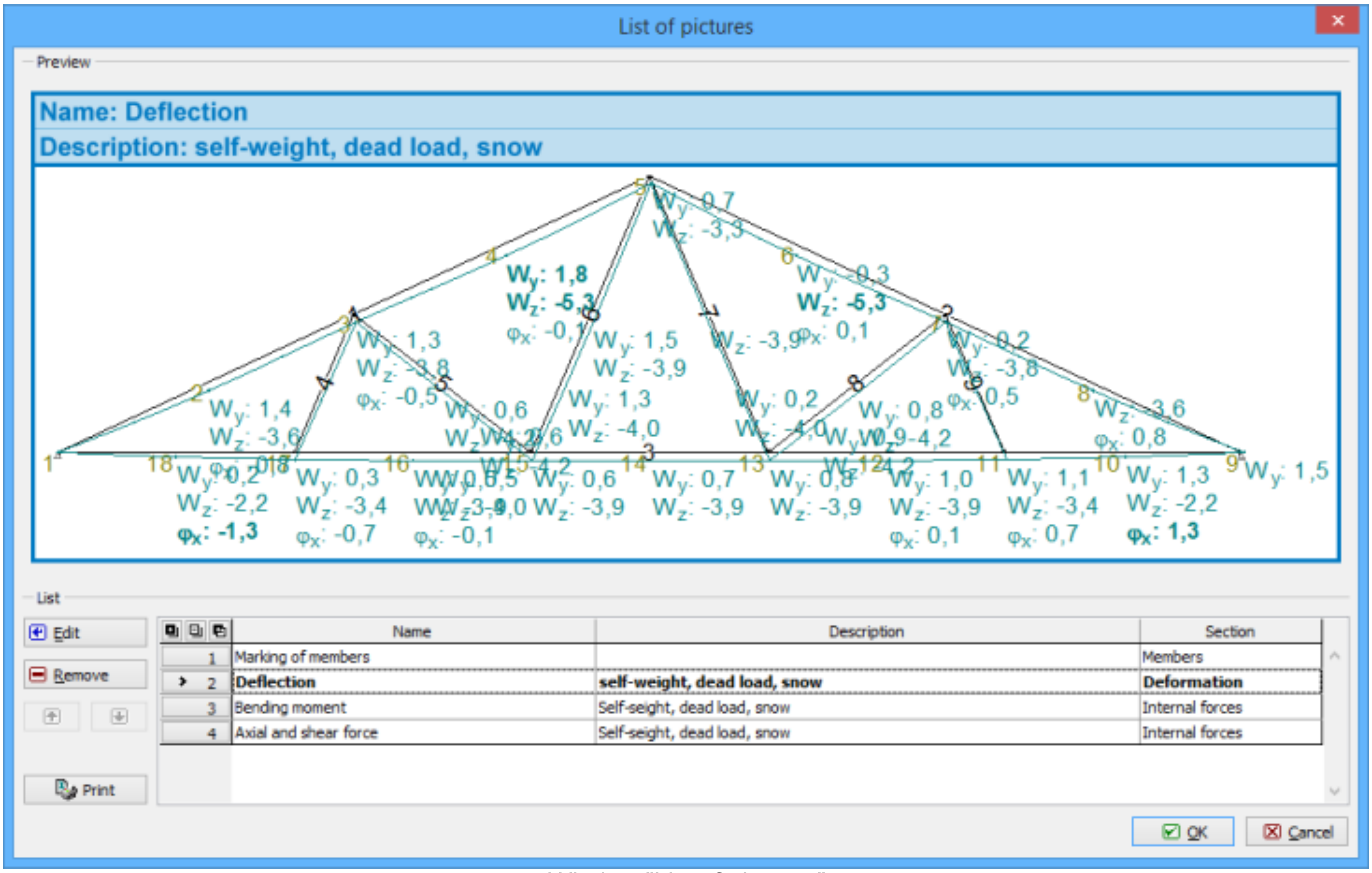

*Window "Page numbering"*

## Pictures printing

Programs "**Fin 2D**" and "**Fin 3D**" contains an option to insert user-defined figures into text documents. The programs store the drawing parameters (displayed items, description) and type of displayed results. The figures are generated automatically with the help of these settings before the printing. Therefore, the pictures show every time the current structure and latest results. The pictures can be added into documents with the help of the button "**Add picture**", that can be found in the bottom part of the tree menu. The pictures can be also modified (it is possible to change appearance, description or position in the document structure) later in the window "**List of pictures**".

### Window "List of pictures"

This window can be used for organizing the pictures for printing. The window contains pictures table in the bottom part and preview of an active picture in the upper part. The properties of active picture may be changed with the help of the button "**Edit**" in the window "**Picture properties**". This window can be also opened using double-click in the table. Pictures also can be deleted with the help of the button "**Remove**". The button "**Print**" is able to print directly the active picture.



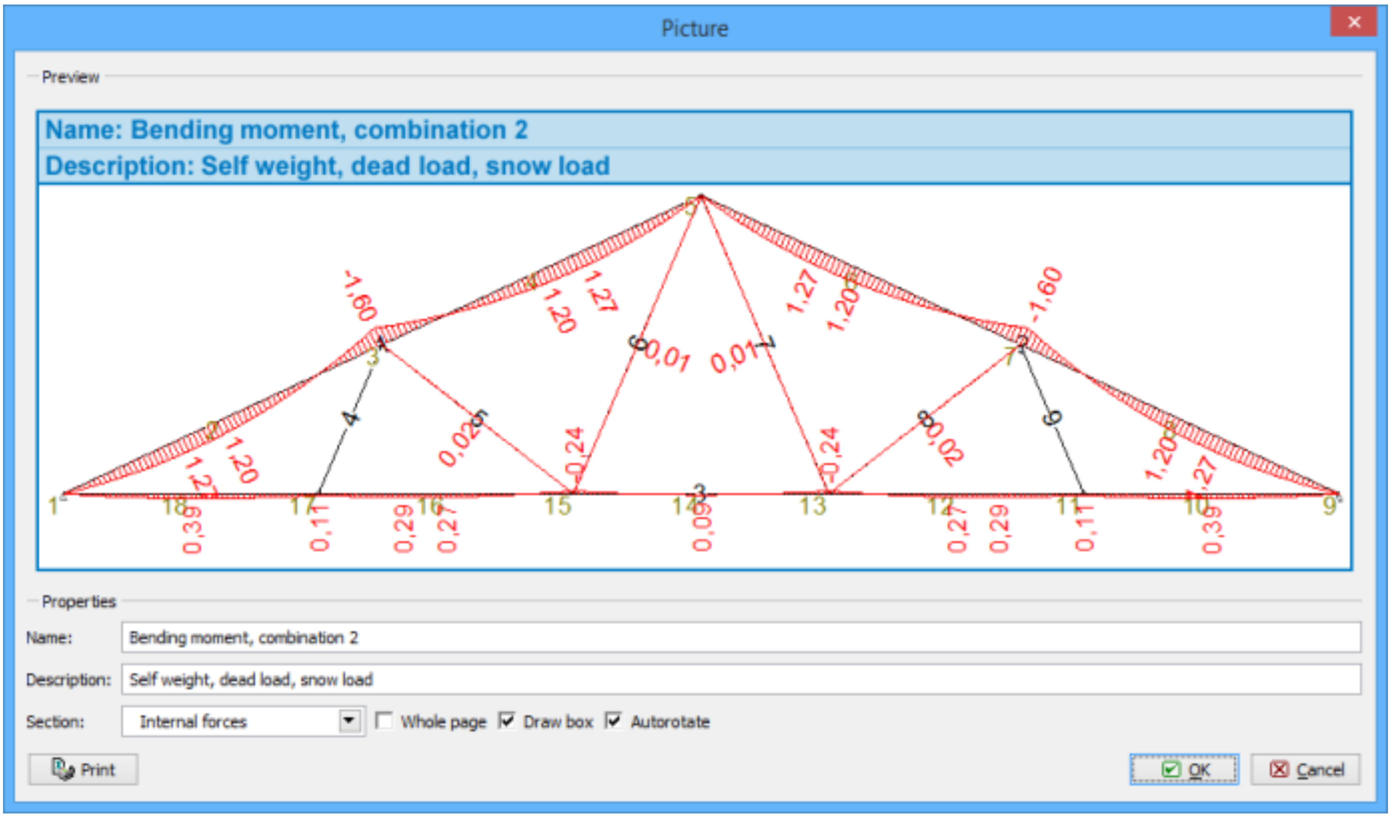

*Window "List of pictures"*

## Picture properties

This window shows the properties of picture, that is stored as a part of output documentation. The window contains both the picture view and some settings that may affect the appearance of the picture:

- **Name** – The name in heading of the picture can be specified here.
- **Description** – The detailed description of the picture can be inserted here.
- **Section** – The chapter (section) of the document, where the picture will be inserted, can be specified with the help of this setting.
- **Whole page** – This setting enlarges the picture to cover full page of the document
- **Draw box** – The picture borders can be switched on or off with the help of this setting.
- **Autorotate** – The setting, which automatically rotates the structure view according to the outer dimensions of the structure. This rotation optimizes the free space in the document. The structure with significant length will be rotated for full page view. The structure with significant height will be rotated in the picture that covers only part of the page.
- **Print** – The button in the left bottom corner is able to print directly the window. The printing is processed in the standard printing window.




*Picture properties*

# Theory

# Fin

## Coordinate systems

Few types of coordinate systems are used in programs Fin 2D and Fin 3D:

### Global coordinate system

The global coordinate system is the fundamental coordinate system and and has axes *X,Y,Z*. It is used for the input of the structure geometry. This system is a right-hand Cartesian coordinate system. The main attribute is, that the *Z*-axis is oriented in the upward direction. It means, that the gravity loads act against the global axis *Z*.

The global coordinate system consists of axes *Y,Z* in the software Fin 2D.

### Local coordinate systems

Any member has local coordinate system. It is necessary for the definition of cross-section orientation, for load input and for display of internal forces along the member length. Local coordinate system uses axes *1,2,3*. Following rules are used for local coordinate systems:

- The axis *1* is identical to the centre line of the member, beginning is in the start joint of the member, positive direction is given by the end joint of the member.
- The axis *3* lies in the vertical plane (parallel to global axis *Z*). Orientation is identical to the orientation of axis *Z* (upward direction). Only exception are the members parallel with the global axis *Z* (vertical members). In these cases, local axis *3* is parallel to the global axis *Y*.

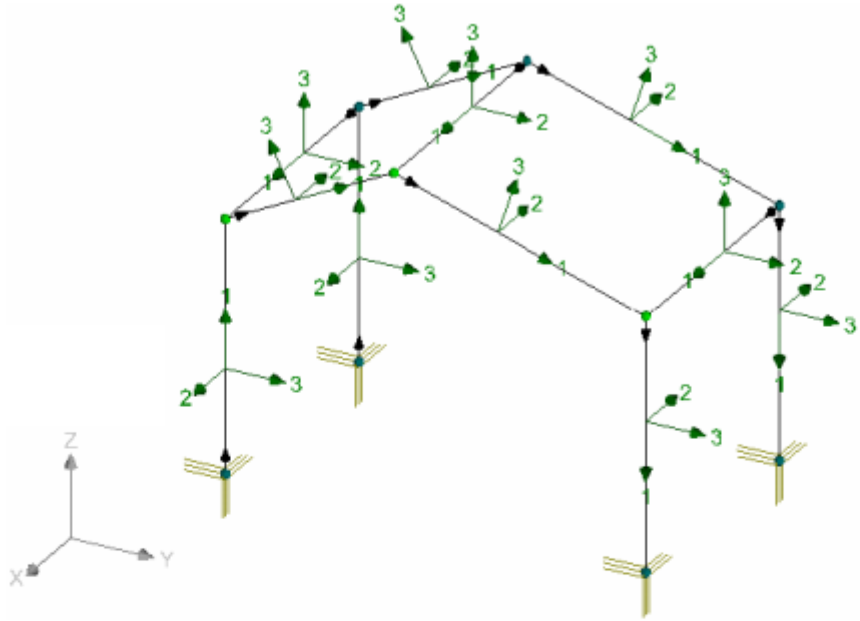
### Cross-sectional coordinate systems

The cross-sectional coordinate system is used for the definition of cross-section geometry. It has axes *y,z* and for rotation angle *0°* the axis *y* is equal to the axis *2* and the axis *z* is equal to the local axis *3*. This local coordinate system is used for rotation of cross-section or for calculation of internal forces along members with rotated cross-sections.

### Coordinate systems of joints *(FIN 3D only)*

The coordinate systems of joints are used for the definition of rotated supports in joints. These coordinate systems have axes $X_S, Y_S, Z_S$. The origin of this coordinate system is in the joint. The coordinate system is defined by the point in the positive part of the axis $X_S$ and by the point, that lies in the plane $X_S Y_S$ (where the coordinate $Y_S$ is positive). Any joint may contain only one coordinate system.




*Global coordinate system and local coordinate systems of members*

# Structural elements

The structures contain two basic types of elements: Joints (nodes) and members. Joints are divided into absolute and relative ones.

## Absolute joints

Absolute joints are spaceless points in a plane, that represent individual connections of structural members. The position is given by absolute (aren't depending on any other object) coordinates in global coordinate system.

Absolute joints in "**Fin 3D**" have six degrees of freedom (3 movements and 3 rotations), any of these degrees can be eliminated by support (or reduced by spring support). The supports can be defined both in the global coordinate system or in the coordinate systems of joints.

Absolute joints in "**Fin 2D**" have three degrees of freedom (2 movements and 1 rotation), any of these degrees can be eliminated by support (or reduced by spring support). Joint supports can be defined in the global coordinate system or according to the joint rotation.

## Members

The members are structural elements that connect joints in the structure. From geometric point of view, members are oriented line segments given by two joints. The member has reference beginning and end joints. The axis *1* of local coordinate system is defined by the orientation of the line from beginning to end joint.

Cross-sectional and material characteristics have to be assigned to any member. The cross-sectional characteristics may be defined with the help of the pre-defined database, manually or with the help of external program (programs "**Section**" or "**Sector**"). In a similar ways, material characteristics may be specified.

Next attribute is the member type. Two basic types are included: "**Beam**" and "**Beam on elastic subsoil**". The "**Beam**" is the fundamental member type locally supported in joints, "**Beam on elastic subsoil**" is the member, that is supported along the whole length by a subsoil (e.g. foundations).

The connection at the beginning and end is given by end conditions. Both connections have *6* degrees of freedom (3 movements and 3 rotations) in "**Fin 2D**" or *3* degrees of freedom (*2* movements and *1* rotation) in "**Fin 2D**". Any of these degrees can be switched on/off or converted into spring connection. When switching off, the occurrence of mechanism may appear.

The advanced characteristics are marked as "**Special**" in the software. These characteristics contain the consideration of shear effect on deformation, excluded tension or compression, warping parameters at the reference joints. These parameters are described in the chapter "**Special member characteristics**".

## Relative joints

Relative joints aren't given by the coordinates in the global coordinate system. They have the position specified relatively to the reference member. The position on member is given by the distance from the beginning or end joint of the member. The distance may be specified in length unit (metres) or in a proportional unit. The joints may be placed between beginning and end joints of the member or may lie outside this segment and extend the member length.

The relative joints may be supported in the same way as absolute joints. The identical is also number of degrees of freedom. The supports may be specified both in the global coordinate system and in the coordinate systems of joints.

The relative joints are used for connections of members to the point placed on the other member (typically connection of webs to chords). They may be also used for display of results in given position.

Scissor joint is a special type of relative joint, as it has two reference members. The position of such joint is given by the intersection of these reference members. This joint creates hinged connection of intersecting members.

# Supports

Supports ensure the stability of the structure in the space. They can be placed both in absolute and relative joints.

## Support types

The joints can be either released (the movement of the joint is possible for this direction in this case) or supported in each of its six degrees of freedom. They can be supported in two ways: fixed or spring. The fixed support completely prevents movement of the joint. There is zero deformation and, as a response to prevent deformation, the reaction arises in these cases. The elastic supports are deformed partially and the reactions are usually smaller than the reactions in fixed supports. The stiffness of the support is characterized by the value of the spring constant $K$. The units have to be respected when specifying the value of the spring constant.

## Support rotation

Supports can be rotated. In Fin 3D, the local coordinate system of the joint has to be created first. If the joint coordinate system is created, the coordinate system used in the support (global or local coordinate system of joint) can be selected.

In the program Fin 2D, support rotation is given only by the specified angle.

# End conditions

End conditions describe the properties of connections between member and reference joints. The number of considered degrees of freedom can be specified by the user.

## Free, fixed and spring connection

All six degrees of freedom at the beginning and end of the joint are fixed as a default. Any of these connection components may be switched off or replaced by a spring. In these cases, the occurrence of mechanism may appear when switching off certain components. It isn't possible to get results for structures with mechanism.

Spring end conditions are included in "**Special**" properties of member. The spring constant $K$ has to be specified for this type of connection.

The end conditions are defined in the local coordinate system of member. During the analysis, the stiffness of springs is combined with stiffness of member, that is defined relatively to the cross-sectional axes. The analysis provides exact results only for structures, where the local member axes $2,3$ and cross-sectional axes $y,z$ are identical. Otherwise (structures with rotated cross-sections), the results are only approximate.

# Cross-sectional characteristics

Cross-section is the fundamental parameter of the member. Cross-sectional parameters may be specified with the help of pre-defined databases, using external programs or entered manually.

## Cross-sectional characteristics

Following characteristics are considered in the analysis:

| | |
|---|---|
| $\boldsymbol{b}$ | • The maximum horizontal direction [mm] |
| $\boldsymbol{h}$ | • The maximum vertical dimension [mm] |
| $\boldsymbol{y_{cg}}$ | • The distance of the centre of gravity from the leftmost point of cross-section *[mm]* |
| $\boldsymbol{z_{cg}}$ | • The distance of the centre of gravity from the bottommost point of cross-section *[mm]* |
| $\boldsymbol{A}$ | • The cross-sectional area $[mm^2]$ |
| $\boldsymbol{P}$ | • The cross-sectional perimeter *[mm]* |
| $\boldsymbol{A_y}$ | • The shear area for shear parallel to the axis $y$ $[mm^2]$ |
| $\boldsymbol{A_z}$ | • The shear area for shear parallel to the axis $z$ $[mm^2]$ |
| $\boldsymbol{I_y}$ | • The moments of inertia about axis $y$ $[mm^4]$ |
| $\boldsymbol{I_z}$ | • The moments of inertia about axis $z$ $[mm^4]$ |
| $\boldsymbol{D_{yz}}$ | • The mixed moment of inertia $[mm^4]$ |
| $\boldsymbol{I_k}$ | • The rigidity moment in simple torsion $[mm^4]$ |
| $\boldsymbol{I_\omega}$ | • The sectional moment of inertia $[mm^4]$ |

## Input of cross-section

The cross-section may be defined using these options:

- The cross-section can be selected from pre-defined database, the characteristics will be calculated automatically by the software according to the specified dimensions.

- The cross-section can be created with the help of programs "**Section**" and "**Sector**". The characteristics are calculated by these programs and are transferred into the software. This option provides only limited number of characteristics. Therefore, shear areas and rigidity moments in torsion are equal to zero.
- The cross-sectional characteristics can be specified manually by the user.

The cross-sectional characteristics are checked after the manual input. The cross-sectional area and moments of inertia can't be zero. Also the rigidity moment in simple torsion $I_k$ has to be bigger than *0*. As this value isn't significant in the most of cases, the estimation of the value is available. The estimation is based on the St. Venant expression, that works fine for massive cross-sections, however does not provide sufficient results for thin walled cross-sections.

The part of cross-sectional characteristics is also the rotation of the cross-section relatively to the local coordinate system of the member. The rotation is the angle between axes *y*,*z* in cross-sectional coordinate system and axes *2*,*3* of local coordinate system of the member. This angle is positive if the cross-section is rotated in anti-clockwise direction for the view against the axis *1* of local coordinate system of member.

# Material

Material is the parameter, that significantly affects the stiffness of member. The material is given by material characteristics (constants that describe behaviour of material under applied load).

## Material characteristics

Following material characteristics are used in the analysis:

| | | |
|---|---|---|
| ***E*** | • | The modulus of elasticity |
| ***G*** | • | The shear modulus |
| ***α*** | • | The coefficient of thermal expansion |
| ***γ*** | • | The specific weight |

All material characteristics use certain units. These units have to be respected when entering the values manually.

## Input of material

The material can be specified using following ways:

- The material can be selected from pre-defined database. The material characteristics loaded from the database will be used.
- The material characteristics can be specified manually by the user.

# Subsoil model

The subsoil model is a part of members with type "**Member on subsoil**". It's a spring support along the whole member length. Subsoil acts in the local coordinate system of member and can be described as a row of springs with given stiffness that support the member. The subsoil usually acts in the direction of local axis *3* (direction of gravity), however, also subsoil in the direction of local axis *2* may be defined (Fin 3S).

Subsoil is considered as springs that act both in compression and tension. As this model usually does not correspond to the real conditions, the appearance of tensile stress should be checked.

The spring subsoil substitutes outer supports of the structure and contact stress is the substitution of joint reactions. Outer supports have to secure only movements perpendicular to the subsoil direction.

This model of subsoil provides accurate results for symmetric cross-sections without any rotation (concrete rectangle, steel RHS etc.). Rotated or unsymmetrical cross-section don't meet all assumptions of the model, results should be considered as approximate in these cases. However, such results usually describe the behaviour of the structure in a sufficient way.

## Subsoil characteristics

The subsoil stiffness is characterized by two constants $C_1$ and $C_2$. The constant $C_1$ it the typical spring stiffness in the direction of the corresponding axis, the constant $C_2$ acts like the shear connection between particular springs $C_1$. The model contains also shear effects at the beginning or end of the member. It means, that the subsoil exists also in front of the member beginning and behind the member end and affects the structure. Consideration of these effects can be switched on by the user. It should be used only in cases, where the subsoil isn't affected by another member or structure (neither in the member direction, nor in the perpendicular direction).

Next parameter is the width of contact between structure and member. As a default, contact width is considered as the corresponding dimension of the cross-section, however, arbitrary value may be specified.

The fundamental parameters of the subsoil are constants $C_1$ and $C_2$, however, they aren't known very often. The values can be calculated by the software. This calculation is based on subsoil parameters $E_{def}$ (the deformation modulus), $v$ (The Poisson's ratio) and $h_d$ (the depth of deformation zone).

The deformation modulus $E_{def}$ is the value usually obtained from in situ measurements. Sometimes, only the oedometric modulus of deformation $E_{oed}$ is available. Following expression describes the relation between these two values:

$$E_{oed} = E_{def}\frac{1-\nu}{(1+\nu)(1-2\nu)}$$

Where is: $\nu$ • Poisson's ratio in interval (0;0.5)

The depth of deformation zone describes the subsoil depth, that is deformed by the beam. This value is given as a ratio of the deformation zone depth $h_d$ and beam width $b$. For example, ratio *3* means the deformation zone depth $h_d=3b$ . Recommended values are between *1.5* and *5.0*.

# Special member characteristics

The member parameters, that aren't important for the most of structures, are marked as "**Special**" in the software. These parameters are mainly the member model choice, excluded stresses and spring end conditions.

## Shear effect

Two theoretical models of members can be used during analysis. First model is based on Bernoulli - Navier theory, where the planar cross-section perpendicular to the member axis remains planar and perpendicular to deformed member axis also after the deformation. This model ignores the effect of shear forces on deformations. It is suitable for typical trusses and frame structures, where member lengths are significantly longer than cross-section dimensions. Members with larger cross-sections (massive beams) should be analysed with the help of the second model based on Mindlin theory. According to this theory, the planar cross-section perpendicular to the member axis remains planar after the deformation, however, isn't perpendicular to deformed member axis any more. The stiffness of the member is decreased due to shear impact on deformation.

## Warping prevention at member ends

Torsion causes both the deformation of cross-section in its plane and in the perpendicular direction (parallel to the member axis). This behaviour is called warping. If the warping is not prevented in the structure, torsion induces only shear stresses and the cross-section is deformed in both directions. Such behaviour is called St.Venant torsion. If the warping is prevented, the torsion induces shear and axial stresses and such torsion is called warping torsion. Warping does not appear for all cross-sections. Warping is common mainly for steel cross-sections with warping coordinate $\omega$ and warping constant $I_\omega$ greater than *0*. Warping parameters can be specified only for these cross-sections.

Warping prevention can be specified with the help of the constant with the interval *<0;1>*, where *0* means free warping and *1* means warping absolutely prevented. The intermediate values describe combined behaviour.

Three different internal forces induced by torsion may appear on members subjected to warping: St.Venant torsional moment $T_t$, bimoment $B$ and warping torsional moment $T_o$. Moments $T_t$ and $T_o$ induce shear stresses in cross-section, bimoment $B$ induces axial stress.

## Special end conditions

The end conditions can be set for all degrees of freedom at the beginning and end of the structure. Free, fixed and spring connections can be combined.

## Excluded tension and compression

The tensile or compressive stresses can be excluded for any member. In these cases, the superposition principle for calculation of internal forces is not valid any more. Internal forces for all combinations have to be calculated by the direct analysis, not as a sum of particular load cases. The calculation is much more time consuming and it can't provide results in real time for more complicated structures with a lot of combinations.

The analysis uses the iteration principle. The members with unallowed stress are excluded from stiffness matrix gradually. Important note, that exclusion is done completely including the self-weight of the member and load applied to this member.

# Load cases

Load cases are collections of loads, that have the same basis with regards to the standard and appear in the same time. Examples are self-weight of the structure or snow load. The loads, that have different properties according to the standard (e.g. permanent and variable loads), cannot be included in one load case. Load case properties are described by code, type and partial nd combination factors.

## Code of load case

The program uses four different load case codes: "**Self-weight**", "**Force**", "**Deformation**" and "**Temperature**".

Load cases with the code "**Force**" can contain point or linear loads, forces or moments. Loads can be assigned to joints or members. The code "**Deformation**" is dedicated for enforced movements or rotations of supports. "**Self-weight**" is a special type of load case, it contains automatically generated self-weight of the structure. These loads can't be modified, only one load case with this code is permitted in the project. The code "**Temperature**" is used for actions induced by thermal changes.

It isn't possible to combine different types of load (e.g. self-weight and thermal action) in one load case. The change of load case code isn't permitted for existing load cases.

## Type of load case

The load case type describes the load duration in time. This parameter is required by certain design standards (verification of timber or RC structure).

## Factors

Any load case has own values of partial load factors and combination factors.

**Partial load factors** are used for the calculation of design values of loads according to the following expression

$$F_d = \gamma_f F_{rep}$$

| | | |
|---|---|---|
| Where is: | $F_d$ | • The design value of load |
| | $\gamma_f$ | • The partial factor, that takes into account deviations from representative values |
| | $F_{rep}$ | • The representative value of load |

Two different values of partial factor are used::

| | | |
|---|---|---|
| $\gamma_{f,Sup}$ | **The factor for unfavourable effect** | • The partial factor for loads, that induce unfavourable effects in structure |
| $\gamma_{f,Inf}$ | **The factor for favourable effect** | • The partial factor for loads, that induce favourable effects in structure. This factor is equal to $0$ for variable loads |

**Combination factors** are used for the calculation of representative values according to the following expression

$$F_{rep} = \psi F_k$$

| | | |
|---|---|---|
| Where is: | $F_{rep}$ | • The representative value of load |
| | $\psi$ | • The combination factors $\psi_0$, $\psi_1$ or $\psi_2$ or value $1.0$. The choice of appropriate factor depends on the combination type and load (permanent, main variable, variable, accidental). |
| | $F_k$ | • The characteristic value of load |

The combination factors takes account of simultaneous occurrence of more variable loads in load combinations and may be used for the calculation of long-term effect for serviceability limit states.

| | | |
|---|---|---|
| $\psi_0$ | **Factor for combination value** | • This factor is used in combinations for ULS and for non-reversible SLS. |
| $\psi_1$ | **Factor for frequent value** | • This factor is used in combinations for ULS (with accidental loads) and for reversible SLS. |
| $\psi_2$ | **Factor for quasi-permanent value** | • This factor is used in combinations for ULS and for reversible SLS. The value is also used for calculation of long-term effects on structure. |

**The reduction factor** $\xi$ is parameter of permanent loads. This factor is used in combinations for ULS according to the expression *6.10b* of EN 1990.

## Category

Categories of load cases are used for correct choice of combination factors $\psi$ for variable loads and the factor $\xi$ for permanent loads. Categories are based on the table *A1.1* of EN 1990.

# Load

Loading is a model of physical influences acting on the real structure. The program can model the physical effects of several types: the forces and moments, self-weight of the structure, imposed deformation of the structure and the effect of temperature changes on the structure. Loads are arranged into load cases.

## Force loads

The force loads can act on members or joints. These loads can be point forces and moments and continuous loads. For the exact determination of the load, the load case, in which the load belongs, the element (member or joint) to which the load is applied, the position and the load value have to be specified.

## Deformation loads

The load induced by the support deformation can be applied only to joints that are supported in corresponding directions. This is valid both for movements and rotations.

## Self-weight

Self-weight load is generated automatically when the load case with type "**Self-weight**" is created. The self-weight is generated as a continuous load in a negative direction of the global axis $Z$. The value of this load is determined by the cross sectional area of the member and the specific density of the material. No additional loads can be added into this load case. Additional loads have to be specified in the load case with the code "**Force**"

### Temperature loads

The temperature load can be used for modelling the effects of temperature change on structure. The temperature load is defined as a temperature increment relative to the common conditions, where no stresses induced by thermal load appear. The positive values of thermal load mean temperature increase, the negative values mean temperature drop.

The general input of thermal change in space isn't simple and consist of two parts. First part describes the temperature development along the rectangular plane (perpendicular to member), second part defines the position of cross-section in this plane. The rectangular plane is given by dimensions *dy* and *dz*, the temperature increments are marked as $t_h$-upper, $t_d$-bottom, $t_l$-left and $t_p$-right side of rectangle. At least one value has to be specified. The maximum number of entered values is three. The thermal distribution along the rectangular plane is calculated using following rules: The thermal increment is constant for whole area if one value is specified. For two specified values, the temperature distribution is calculated with the help of plane, that has gradient in the direction of line between specified values. Three specified values define the thermal distribution unambiguously. Fourth value is calculated according to the plane given by three specified values.

The position of the cross-section in the rectangular plane is given by the coordinates of the centre of gravity in this plane. The left bottom corner is considered as an origin.

As a default, dimensions of rectangular plane are equal to the maximum dimensions of member cross-section.

## Combinations

The load combinations are used for the mutual action of different load cases (e.g. self-weight of the structure, snow load and wind load). The combinations are created separately for ultimate limit states, serviceability limit states (both also divided according to the theories of first and second order) and for linear stability.

The description of any combination contains the number and name and the list of included load cases including corresponding load case factors and combination factors. The variable load cases may be considered as main ones.

### Ultimate limit states

Following types are available for ultimate limit states:

**Basic**
- The fundamental combinations according to the expression *6.10* of EN 1990

**Alternative**
- The alternative combinations according to the expressions *6.10a* and *6.10b* of EN 1990. This option creates two times higher number of combinations than the basic ones. These combinations aren't allowed in certain countries.

**Accidental**
- The accidental combinations according to the expression *6.11* of 1990. The accidental load including partial factor $\psi_1$ or $\psi_2$ may be specified for these combinations. The input of accidental load isn't necessary (e.g. fire resistance analysis).

### Serviceability limit states

Following types are available for serviceability limit states:

**Characteristic**
- The combinations according to the expression *6.14* EN 1990

**Frequent**
- The combinations according to the expression *6.15* EN 1990

**Quasi-permanent**
- The combinations according to the expression *6.16* EN 1990

**Final deflection**
- The combinations for the final deformation of timber structures. They are based on the chapter *2.2.3(5)* of EN 1995-1-1. These combinations provide relevant values of deflection. Internal forces are misleading as these combinations simulate the effect of creep.

### Linear stability

The combinations for linear stability don't use partial safety factors $\gamma_f$. The variable loads may be reduced with the help of the combination factor $\psi_0$.

## Analysis according to I. order theory

Analysis according to I. order theory is the fundamental function of the program. It consists of several parts, continuously following each other. The first part is the control of inputs. It checks whether the input of necessary parts is complete and the structure meets assumptions of analysis.

The next step is the optimization (optional), which is an operation that should significantly accelerate the analysis of complex structures.

After that, composition of stiffness matrix follows. It is compiled from the partial stiffness matrices of individual members. An essential element for the analysis is a member sector. The member sector is a part of member between two joints. Each member consists of adjacent sectors lying on one straight line.

The composition of vectors on the right sides, that contain joint loads, is the next step. These values contain also member loads that has been previously converted into joint loads. Number of vectors is equal to the number of load cases. The program compiles the stiffness matrix in elements.

The system of equations is solved with the help of method Sky-Line, which is effective for frame and topologically inhomogeneous structures. The advantage of this method is also the minimization of zero elements in the matrix, thereby the numerical inaccuracies is reduced.

Overall, the calculation may be time consuming when using iterative methods for dynamic and stability calculations, especially for larger structures where the numbering optimization has not been applied. As a result of the analysis, the values of deformations in the joints are obtained. These values are the key data for all calculations in post processor (internal forces, reactions, member deformations, stresses).

The last part of the analysis is the preparation and saving of values needed for fast results view. It means internal forces at the endpoints of members, reactions in supports and extreme values of internal forces.

### Numbering optimization

Numbering optimization is a process that allows for greater speed of calculation. If optimization is enabled, the joint are renumbered in order to accelerate solving the system of equations before running the analysis. The purpose of renumbering is that the stiffness matrix have nonzero elements as much as possible centered around the main diagonal. This significantly reduces the number of operations undertaken in solving the system of equations. Renumbering is done only within the calculation, so the user's originally assigned numbering retains in the program. Optimization algorithm does not perform renumbering on structures that are divided into more separated parts.

### Singularity during analysis

Singularity is the most common error that occurs during analysis. It is usually caused by insufficient support of structure or its part. It is necessary to check the supports of complete structure, as well as integrity of individual structure parts and eventually review end conditions of particular members. This error indicates that some part of the structure can move freely in space. The most often case is the member rotation about the member axis or insufficient support of planar structures in space.

## Deformations, reactions, contact stresses

Deformations, reactions and contact stresses in joints are the results of calculations, that are considered as accurate. Other values are calculated using these results as inputs.

### Deformations

Joints deformations are the fundamental results of calculation. All other results like reactions, internal forces and contact stress are calculated with the help of these values.

The program displays the diagrams of deformation for members or structure. The exact values along members are displayed in positions of relative joints or in inflexion points. If the user is interested in finding the exact value of the deformation in a specific point, it is necessary to insert a relative joint into this position. Such procedure is also important on members with free or spring end conditions.

### Reactions

The program calculates reactions induced by loading for all degrees of freedom in joints, that are replaced by the external support. Supports may be rigid or spring ones. Reactions can be forces (for movements fixed by an external support) and moments (for rotations fixed by an external support).

### Contact stresses

Contact stress is similar to reactions in external supports. It is the pressure which is transmitted from the structure to the subsoil and it is necessary to ensure that the subsoil is able to resist these stresses.

Contact stress is calculated from the displacements perpendicular to the subsoil and from the rotation in the plane perpendicular to the contact area about the axis perpendicular to the member axis. Calculation is based on deformations at nodes. And thus the stress is calculated in nodes and linear distribution is considered between nodes. It means that the denser division of member will be used, the more accurate results will be obtained. The oation of node may cause the discontinuity of results (stresses on the left and right sides of the node aren't equal). The average value should be considered in these cases. This inaccuracy can be reduced significantly by higher density of nodes for members on subsoil.

## Internal forces

Internal forces are the final results of the structural analysis. They can be used for verification of members in designing modules. Following internal forces can be calculated with the help of the program:

| | |
|---|---|
| **Normal force** | • The axial force, causes tensile or compressive stresses |
| **Shear forces** | • The forces perpendicular to the member axis, cause shear stresses. They can act in two parallel components in spatial structures. |
| **Bending moments** | • The moments that appear in members exposed to bending. They can act in two parallel components in spatial structures. |
| **Torsional moment** | • The torsional moment about the member axis, causes shear stresses in the cross-section |

**Warping moment** • The torsional moment in members with prevented warping. It causes shear stresses.
**Bimoment** • The moment in members with prevented warping. It causes axial stresses.

There are more options how to display internal forces for structure.

## Internal forces in coordinate systems of member and cross-section

Internal forces in local coordinate system of member are marked in accordance with axes of this coordinate system. $N$ is the axial force, $V_2$ and $V_3$ are shear forces in directions of local axes $2$ and $3$, $M_1$ is the torsional moment and $M_2$, $M_3$ are the bending moments about local axes $2$ and $3$.

Sometimes, it is necessary to see internal forces, that are transferred into the designing modules. It means internal forces in the local coordinate system of cross-section. These internal forces are signed $N$, $V_z$, $V_y$, $M_1$, $M_y$ and $M_z$, where subscripts $y$ and $z$ mean axes of cross-sectional coordinate system. The internal forces in the cross-sectional coordinate system contain also torsional characteristics like bimoment $B$, warping moment $T_\omega$ and St. Venant torsional moment $T_t$. These characteristics can be displayed only for steel cross-sections with *I*- and *U*-shape, RHS, $\pi$-shape or for built-up cross-sections made of more *I*-profiles. These cross-sections have warping coordinate $\omega$ and warping constant $I_\omega$ greater than $0$.

## Internal forces for load cases and combinations

Internal forces are calculated both for load cases and combination. Designing programs are able to work only with results of combinations. Therefore, at least one combination has to be created when transferring members into designing programs.

Mathematically, the values of forces for combination are the sums of the values of each load case multiplied by corresponding combination and load factors.

## Envelopes of internal forces

Envelopes are used to show the extreme values of internal forces in range of load cases or combinations. Any point of envelope shows the maximum or minimum value of internal force, that appears in one of defined load cases or combinations. Therefore, the envelope is only fictitious diagram, that can be used for finding the most stressed parts of the structure.

The envelope is defined by the list of considered load cases or combinations. Next input is the "**Envelope key**" that defines, whether the envelope will be shown for all internal forces or only for one force (or moment) and other forces will show only values in corresponding combination (or load case), where the maximum of key force appears.

# Eigenmodes

In addition to standard calculations of linear structural analysis, software Fin 3D has ability to determine the natural frequencies and eigen shapes of structures. The aim of this module is partly to allow the user to predict the behaviour of structures where the inertial effects of the mass of the structure can not be ignored and partly to attract attention to cases when any of the natural frequencies of the considered structures are located close to the frequency of the excitation.

In terms of the structural mechanics, the finding natural frequencies and eigen shapes is characterized as a general problem of eigenvalues described by the equation

$$\left(K-\omega^2 M\right) r=0$$

Where is:
- ***K*** • The stiffness matrix
- ***M*** • The mass matrix
- ***r*** • The eigenshape corresponding to the natural frequency

If the order of matrices $\boldsymbol{K}$ and $\boldsymbol{M}$ is $\boldsymbol{n}$, then the above equation allows software to calculate $\boldsymbol{n}$ natural frequencies $\boldsymbol{\omega_i}$ and $\boldsymbol{n}$ eigen shapes $\boldsymbol{r_i}$. The equation also shows that the absolute value of the components of the vector $\boldsymbol{r}$ is not decisive for the description of the shape oscillations. Eigenvectors $\boldsymbol{r_i}$ are standardized in the program, while the size of the individual components of the displacement is not displayed.

The stiffness matrix is assembled as in the case of linear analysis. The consistent formulation is used to build the mass matrix $\boldsymbol{M}$. Mass matrix is then diagonal, but generally full. Calculating the weight of the matrix elements is based on the material density of the individual members. Weight, which does not directly relate to a given structure, but has influence on the dynamic behavior of the structure, can be entered using concentrated masses. Entered mass (weight) is assigned to the nodes (joints), it is possible to define also the eccentricity.

As mentioned above, the equation allows to calculate only the amount of natural frequency, that is equal to the number of degrees of freedom. The literature describes a number of methods for finding a complete solution to the problem. In many cases it is impractical, because only the first few natural frequencies and eigenmodes are important from engineering point of view. Additionally, higher frequencies and mode shapes are usually burdened with considerable error resulting from discretization structures on individual finite elements. Program therefore supports only two the most commonly used methods, as only the first few eigen shapes and frequencies are needed.

### Subspace iteration method

The most common is the subspace iteration method. This method counts the number of selected shapes and lowest natural frequency, while in order to increase the speed of convergence of the iteration, it is performed on the higher plurality of eigenvectors, than is required. Therefore we recommend to enter the structure to be at least double the number of degrees of freedom than the required number of custom shapes. Unfortunately, this method does not guarantee that the calculated natural frequencies are the lowest ones. Therefore, the program is equipped by Sturm control informing the user about the possible omission of any of the required eigen shapes. This method fails sometimes for structures which are characterized by clusters with more frequencies. On the other hand, this method is easy to handle with eigen shapes belonging to multiple natural frequencies.

### Lanczos method

This method is suitable mainly for structures with a large number of degrees of freedom. If the solution does not require usage of the harddisc, this method is significantly faster than the subspace iteration method. Although this is a very reliable method, it is similarly to the method of subspace iteration completed with Sturm control. Also this method calculates only selected number of the lowest natural frequencies. This method unfortunately does not allow the separation of the eigenmodes belonging to multiple natural frequencies. In this case, we recommend to use the method mentioned above.

### Analysis accuracy and convergence of the calculation

As we mentioned in the introduction to this chapter, the accuracy of calculation of the individual natural frequencies depends on the selected level of discretization. Theoretically it is possible to find as many custom shapes and frequencies as the number of degrees of freedom in the structure. In reality, however, shapes, whose level is close to the number of physical degrees of freedom, burdened with considerable numerical error due to very coarse division into elements. In this case it is necessary to choose a finer division of members into individual analysis elements. In addition, the rate of convergence to lower natural modes of oscillation for subspace iteration method depends on the number of vectors that are used in the calculation, and the iteration works, if possible, always with a number of eigen shapes slightly higher than the required number.

Insufficient number of physical degrees of freedom is one of the main reasons for the termination of calculation without founding all requested eigen shapes. Another reason may be inadequate maximum number of iterations or the required accuracy for calculating the natural frequencies. While the maximum number of iterations, which allows the calculation is equal to *200*. The required tolerance of accuracy should be higher than $10^{-4}$.

At this point, we would like to draw the attention to the special importance of the concept of iteration for Lanczos method. From a theoretical point of view, this is a generation-base vector used in a Rayleigh-Ritz method. In practice, this means that the maximum number of iterations can not be higher than the number of physical degrees of freedom. Users should therefore not be surprised that the calculation of the structure with 6 degrees of freedom was stopped after the sixth iteration, although the specified number of iterations was elected *100*. The sufficient attention should be given to the selection of the number of iterations in the Lanczos method, because it influences the speed of calculation. The reason is, that the section of internal memory is allocated for each iteration and such space may stay completely unused. The number of specified iterations should be therefore as close as possible to the number of iterations that are required for the convergence to the specified number of natural frequencies. Unfortunately, there is no general rule, how to proceed in such cases, and the user has to rely on his own judgement and experience. The maximum number of iterations is equal to *200* for this method.

## Linear stability

The stability analysis provides important information regarding the behaviour of a structure. The main result is the proportion between applied load and critical load, which causes loss of structural stability. The buckling shape shows the mechanism of failure and can be used for estimation of character of geometric imperfections, that are decisive for the structure.

The problem of linear stability can be described as the problem of determining critical loads of ideal structure as the factor for the load, which really acts on this structure. For this purpose it is necessary to write the equilibrium conditions for the deformed structure and include the effect of axial forces on the lateral stiffness of members. Equilibrium equation can be described with the help of following expression:

$$(K - \lambda K_\sigma)\, r = 0$$

| Where is: | | |
|---|---|---|
| $\boldsymbol{K}$ | • | The stiffness matrix |
| $\boldsymbol{K_\sigma}$ | • | The initial stress matrix (effect of axial forces) |
| $\boldsymbol{r}$ | • | The buckling shape |

Like the eigen shapes, the buckling form is also standardized, the standardized values of joint deformations are not displayed. The above equation is similar to equation describing the eigen shapes. Unlike eigenmodes, the stability problem is interested only in the lowest eigen number of the equations. This number is marked as $\lambda_{crit}$. The vector $\boldsymbol{R}$ corresponds to the load distributed along the structure. Following expression may be used for determination of critical load:

$$R_{crit} = \lambda_{crit} R$$

This vector $\boldsymbol{R_{crit}}$ corresponds to the critical load that causes the loss of stability of the whole structure. If the value of $\lambda_{crit}$ is negative, the structure is stable and loss of stability may occur only for load with opposite direction.

## Subspace iteration method and Lanczos method

This equation is a general problem of eigenvalues. Both subspace iteration and Lanczos methods. Subspace iteration method was modified in order to solve the structures, which contain both tensile and compressive members. The Lanczos method is more conservative in this respect and requires that the initial stress matrix is positive definite. In other words, it allows to solve only the structure, which contain only unloaded members or members in compression. The advantages and disadvantages of both methods are described in detail in the previous chapter.

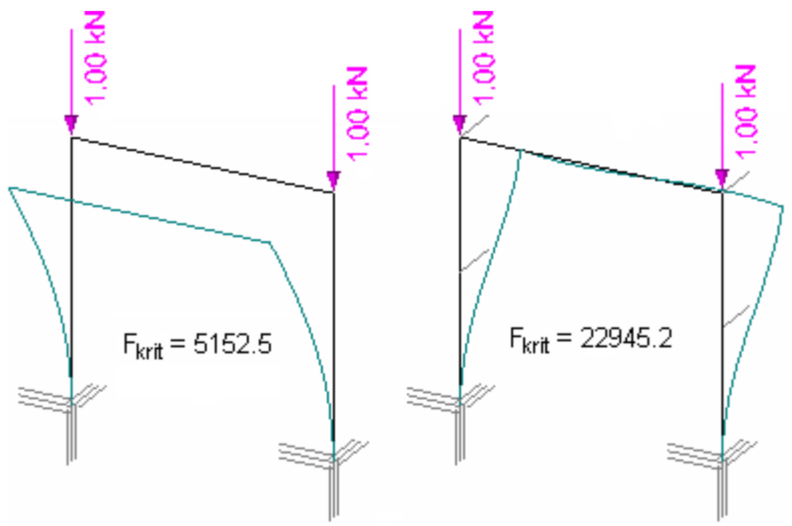
## Inverse matrix method

One of the best-known methods, which is solely intended to find the lowest eigenvalue of an equation is the method of inverse iteration. As is already evident from the name is an inverse iteration method by iteration. This method is relatively reliable and accurate in the sense that repeating the iteration cycle increases the precision closer to the correct solution. Unfortunately, in some cases, the speed of convergence is very slow, and the maximum number of possible iterations, which is equal to *200*, may not be always sufficient. Therefore, the results are supplemented with information about the achieved accuracy. In case that the convergence was not achieved, the user can either increase the number of iterations, or to reduce the required accuracy, or to select another method.

## Recommendation

As the superposition principle can't be used in stability analysis, the calculation is performed only for the defined combinations. The results are characterized by a combination of critical load factor $\lambda_{crit}$ (marked as $\boldsymbol{F_{crit}}$) and its buckling shape. The value of $\lambda_{crit}$ should be greater than *4* for all combinations. Otherwise, the designer should seriously review the design concept and the static model, reformulate it or consider possible design modifications.

Following notes refer to the difference between spatial and planar structures. The user should respect that the planar structure loaded in a plane tends to deviate from the plane of the structure. If the deviation from the plane of the structure is prevented, the critical load factor $\lambda_{crit}$ is significantly higher. This fact is demonstrated on the example of a simple plane frame whose columns are loaded in the frame plane by axial forces. The result is shown in the following figure. First case shows out of plane buckling, the second one shows in plane buckling, as the out of plane buckling was prevented. The $\lambda_{crit}$ is four times greater.




*Buckling out of plane and in plane*

The calculation of buckling lengths is a rather complicated issue and there is no general rule in this area. Users have to respect engineering experience and common sense. The stability analysis provides simple input for the estimation of buckling lengths. The following procedure may be used for significantly compressed members: In the first step, it is necessary to determine the axial forces in the structure. In the second step, it is possible to estimate the value $\lambda_{crit}$ and determine the axial forces corresponding to the critical load using formula

$$N_{crit} = \lambda_{crit} N$$

Where is: ***N*** • The axial force in the member

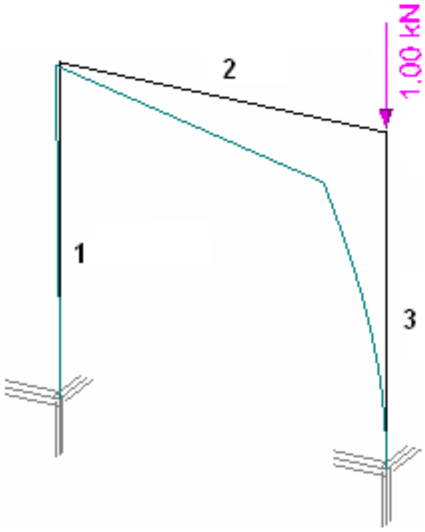
The buckling length can be calculated using the Euler expression:

$$l_h^2 = \pi^2 \frac{EI}{N_{crit}}$$

| | | |
|---|---|---|
| Where is: | $l_h$ | • The buckling length |
| | $EI$ | • The bending stiffness of the member |
| | $N_{krit}$ | • The critical axial force |

This formula was derived assuming axially loaded straight member. Its use for estimation of buckling lengths of generally loaded structures should therefore be subject to a thorough analysis, as the applicability of this formula should refer only to the significantly compressed members with the behaviour of a lone straight bar. This fact corresponds to the rod *3* on the following figure, while the rods *1* and *2* only substitute the flexible support of the upper joint of member *3* and their buckling verification is pointless. The rod *3* acts as a spring supported cantilever. Therefore, the calculation of buckling lengths of the rods *1* and *2* would be pointless.




*Axially loaded member*

# 2nd order analysis

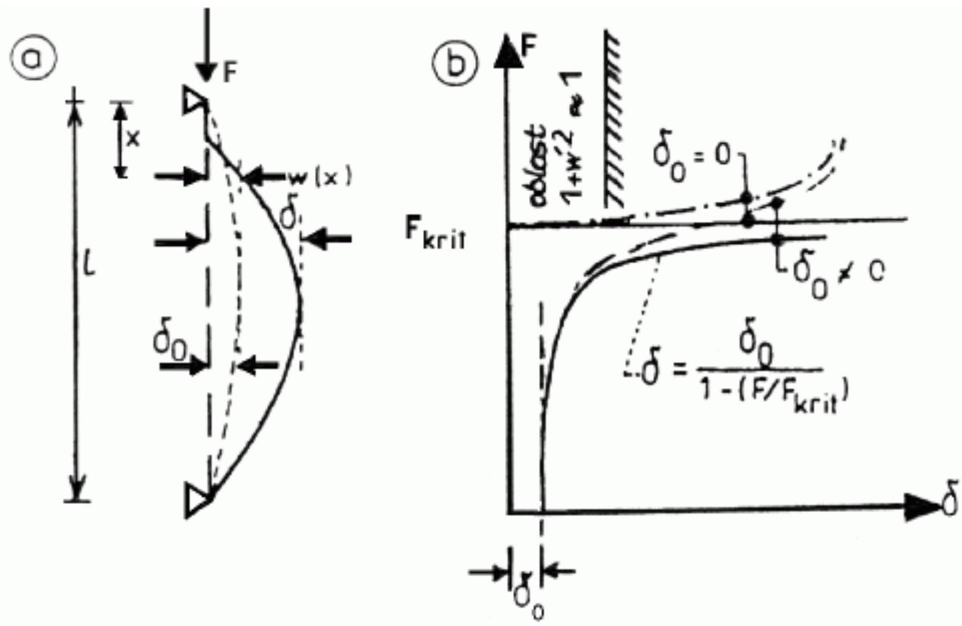
## Theory

The second-order analysis is a powerful tool for engineers when designing slender frame structures. Views on the scope of its use, however, differs along the engineering community. Therefore, this chapter clarifies some important aspects of this theory both from practical and theoretical point of view.

The theory of second-order is a tool for stress analysis of slender structures with significant axial forces that are either under the effect of lateral forces, or are exposed to initial imperfections. Imperfections can be material ones (eg. Uneven distribution of stiffness in the cross-section) or geometric ones (curved member axis eccentricities in supports etc.). Material imperfections can be usually converted into geometric ones. In this case, the member axis in imperfect structures means a line connecting stiffness centres of heterogeneous sections.

The second order theory is a simplification method of geometrically non-linear analysis of structures. The assumptions upon which this theory can be used, are listed below. The difference between the geometrically non-linear and linear approach can be shown in the following figure:




*Loading curves of ideal ($\delta_0$=0) and imperfect ($\delta_0$≠0) member*

In the first part of the figure, the initial imperfections are expressed by the parameter $\delta_0$ and the final state by the parameter $\delta$. The initial imperfections $\delta_0$ may be caused also by the effect of lateral forces. They can be determined by the common calculation according to the first order theory. Second part shows the loading curves of ideal ($\delta_0=0$) and imperfect ($\delta_0 \neq 0$) member corresponding to the geometrically non-linear (dashed curves) and linear (solid curve) solutions. They can be obtained by integrating the differential Euler equation:

$$\frac{1}{\rho} - \frac{1}{\rho_0} = \frac{M}{EI}$$

For the geometrically non-linear analysis following expression can be used:

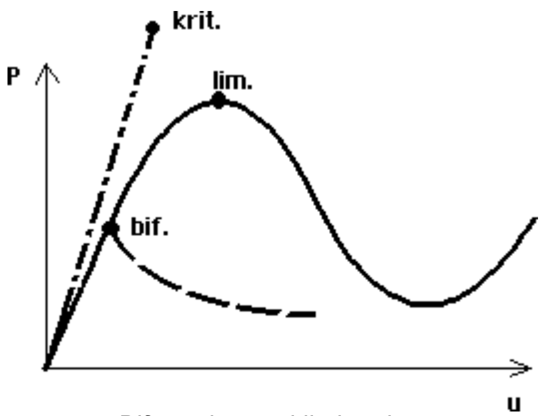
$$\frac{1}{\rho} = -\frac{w''}{\left(1 + w'^2\right)^{\frac{3}{2}}}$$

This expression is valid for linear analysis:

$$\frac{1}{\rho} \cong -w''$$

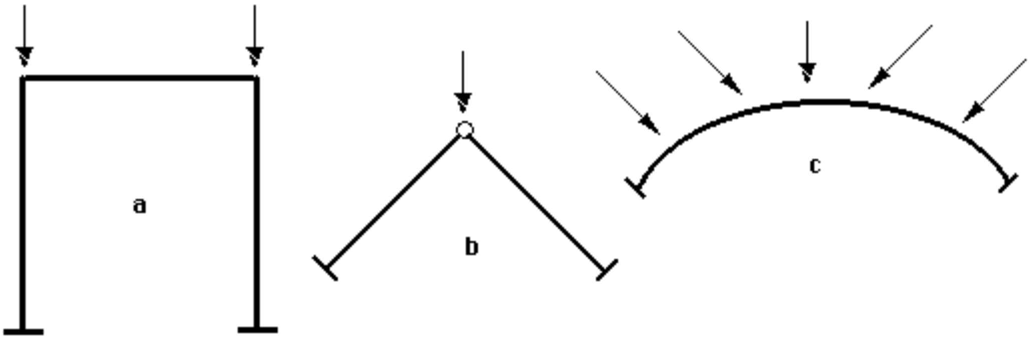
Apostrophe means the derivation of deflection. As the integrating of the Euler differential equation is difficult, the approximate methods are usually used. Here, the deformation variant of FEM is used.

**Geometrically non-linear calculations** are necessary to find critical states (bifurcation or limit ones) on the diagram "load-displacement" especially for curved structures, such as arches and meshed structures arranged in the shape of a curved surface (e.g. spherical or cylindrical). In a limit point, load reaches its maximum or minimum value. The bifurcation points leads to branching of balance (e.g. intersection of symmetrically and non-symmetrically deformed arch). As the linear calculation (dotted line) is not able to cover these relationships, it significantly overestimates the stability capacity.




*Bifurcation and limit points*

**Linear calculations** can be applied with certainty to the common frame structures forming an orthogonal system (option *a* in the following figure). Contrary to structure *a* , the structure *b* acts closer to the arch *c* and the linearisation is inadmissible. The decision whether the design can use linear or non-linear calculation requires not only the theoretical knowledge but also some practical experiences.




*Validity of linear theory*

## Assumptions of second order theory

The second order theory is a simplification of non-linear analysis and is based on following assumptions:

- The relation between deformations and movements are linear
- The internal forces don't change during deformation
- Equilibrium is calculated on deformated structure

With the classical formulation based on the integration of basic equations, the relationship between end member forces $f$ and corresponding movements $r$ can be expressed using formula

$$K\left(\lambda\right)r = f$$

| | | |
|---|---|---|
| Where is: | $\lambda$ | • The load parameter |

All external forces grow proportionally to this parameter. Linearity of the analysis means that the stiffness matrix $K$ is not a function of the joint deformations vector $r$ . There are two ways how to solve the problem.

First way is, that the stiffness matrix $K(\lambda)$ is converted using expression

$$K\left(\lambda\right) = K_0 - \lambda K_\sigma$$

| | | |
|---|---|---|
| Where is: | $K_0$ | • The stiffness matrix according to the first order theory |
| | $K_\sigma$ | • The matrix of initial stresses |

The solution is close to the exact results, if the member division is satisfactory. This approach is the basis of programs. In accordance with the above formulation, the following calculation is used. The first step is to determine the axial forces on the structure using the first order theory. Knowledge of the axial forces is used for composition of initial stress matrix $K_\sigma$ . The calculation is repeated after that, however, with modified matrix $K(\lambda)$ .

The basic equation assumes ideal shape of structure, but with the possibility of lateral load forces. In the case of imperfect structure loaded with axial forces, the expression is modified accordingly:

$$\left(K_0 - \lambda K_\sigma\right)r = K_0 r_0$$

| | | |
|---|---|---|
| Where is: | $r_0$ | • The vector of initial joint imperfections |

The expression $K_0 r_0$ is equal to the effect of shear forces. In case of combination of both effects, the impacts of imperfections $K_0 r_0$ and shear forces $R$ are added up. If both expressions $K_0 r_0$ and $R$ are equal to *0*, the analysis turns into the problem of linear stability.

Second solution is based on the classic principles of analysis of slender imperfect structures.

The fundamental equation is

$$K_0 r = f$$

The stiffness matrix is updated with the respect of new geometry, therefore

$$K_1 r \neq f$$

The inbalance $f \neq K_1 r$ can be developed by this expression

$$K_1 \Delta r = f - K_1 r$$

or

$$K_1 (r + \Delta r) = f$$

The calculation is repeated until

$$\|\Delta r\| \le \varepsilon$$

## Second order theory and standards

Designing standards are based on the determination of buckling coefficients, that correspond to the member slenderness (buckling lengths). Used expressions are based on the fundamental assumption that the member is isolated. However, members are usually part of a structure and are affected by surrounding members. Therefore, buckling lengths are usually different. As the second order theory provides internal forces on deformed structure, there is no need to calculate buckling coefficients. It can be said that the stability problem is thus converted into the strength analysis.

Finally, one important conclusion. Knowing the value of the critical load factor $\lambda_{crit}$, we can easily estimate the deformation according to the second order theory with respect to initial imperfections $\delta_0$ or deformations calculated according to the first order theory (providing that the load is proportional and the decisive members are compressed).

$$\delta = \frac{\delta_0}{1 - \frac{1}{\lambda_{crit}}}$$

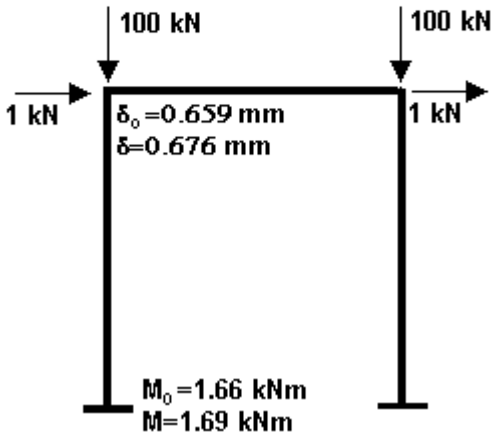
Similar expressions may be used (with lower precision) also for internal forces like bending moments etc.

$$M = \frac{M_0}{1 - \frac{1}{\lambda_{crit}}}$$

Where is: $M_0$ • The bending moment calculated according to the first order theory

## Example

Short example shows the validity of the relationships mentioned above. The example is a simple plane frame loaded by point loads in the corners of the frame. The geometric imperfections were simulated by lateral forces with the magnitude *1/100* of the vertical force. Horizontal movement of beam $\delta_0$ calculated according to the first order theory is equal to *0.659mm*. Second order analysis calculates the horizontal displacement $\delta$ equal to $\mathbf{0.675mm}$. Similar results were obtained for the bending moment in supports. Moment $M_0$ was equal to *1.66kNm*, while moment $M$ calculated according to the theory of second-order rose to *1,69kNm*. Using the equations above, the lateral displacement $\delta$ is equal to *0.676 mm*, which practically coincides with the calculation using the FIN 3D. The value of the bending moment is equal *1,70kNm*. The difference is less than *0.6%*.



*Structure scheme*

# TeorieStatikaNarodniPrilohy

# Concrete

# Member types

Member type influences both structural rules and analysis. These member types are supported:

## Beam

The member for which the span is not less than *3* times the overall section depth should be considered as a beam in accordance with *5.3.1(3)*.

Check that the maximum strain in concrete *3,5‰* can't be reached before reaching the yield strength of reinforcement $f_{yk}$.

The most distant reinforcement from compressive edge of concrete cross-section is considered during this analysis. This analysis ensures, that the concrete crushing can't appear without plastic elongation of the reinforcement. Any collapse of the structure should be preceded by increased deflection and significant cracks in these cases. This analysis is based on limitation of ratio $\xi$:

$$\xi \leq \xi_{max}$$

where $\xi$ is given by formula

$$\xi = \frac{x}{d}$$

Where is: ***x*** • The neutral axis depth

***d*** • The distance between compressive edge of cross-section and the most distant tensile bar

Maximum value $\xi_{lim}$ is calculated using formula

$$\xi_{lim} = \frac{3.5 \cdot 10^{-6} E_s}{3.5 \cdot 10^{-6} E_s + f_{yk}}$$

Where is: $E_s$ • The design value of modulus of elasticity of reinforcing steel

$f_{yk}$ • The characteristic yield strength of reinforcement

## Slab

The member for which the minimum panel dimension is not less than *5* times the overall slab thickness should be considered as a slab according to *5.3.1(4)*.

Check that the maximum strain in concrete *3,5‰* can't be reached before reaching the yield strength of reinforcement $f_{yk}$ is performed for this type of member. This analysis is described above.

## Column

The members for which the section depth does not exceed *4* times its width and the height is at least *3* times the section depth should be considered as columns in accordance with *5.3.1(7)*.

## Wall

Vertical members, that don't meet requirements for columns shall be considered as walls according to *5.3.1(7)*.

# Material characteristics

## Concrete

The material can be selected from the database of strength grades or specified manually by the user. The database contains strength grades according to the table *3.1* of EN 1992-1-1. Following characteristics have to be specified for user defined material:

- $f_{ck}$ • The characteristic compressive cylinder strength of concrete at 28 days
- $f_{ctm}$ • The mean value of axial tensile strength of concrete
- $E_{cm}$ • The secant modulus of elasticity of concrete

Automatic determination of $f_{ctm}$ and $E_{cm}$ according to the table *3.1* can be used for user defined material. For $f_{ck}$ lower than or equal to *50MPa*, $f_{ctm}$ is calculated using formula

$$f_{ctm} = 0.3 f_{ck}^{\frac{2}{3}}$$

following formula is used for other cases:

$$f_{ctm} = 2.12 ln\left(1 + \frac{f_{cm}}{10}\right)$$

The secant modulus of elasticity of concrete $E_{cm}$ is calculated using following formula

$$E_{cm} = 22\left(\frac{f_{cm}}{10}\right)^{0.3}$$

Where the mean value of concrete cylinder compressive strength $f_{cm}$ is calculated using following formula

$$f_{cm} = f_{ck} + 8$$

## Reinforcement steel

The materail can be selected from the database of strength grades or specified manually by the user. Following characteristics have to be specified for user defined material:

| | | |
|---|---|---|
| $f_{yk}$ | • | The characteristic yield strength of reinforcement |
| $E_s$ | • | The design value of modulus of elasticity of reinforcing steel |

The design values of material characteristics are used during the analysis. The design values are calculated by division of characteristic values (index starts with *k*) by partial factor for material properties $\gamma_M$. Values of $\gamma_M$ are written in chapter "**National annexes**". Index of design values starts with *d*.

# Material characteristics at elevated temperatures

## Concrete

The compressive strength of concrete is calculated in accordance with *4.2.4.2*:

$$f_{c,\Theta} = k_c f_{ck}$$

| | | | |
|---|---|---|---|
| Where is: | $f_{c,\Theta}$ | • | The compressive strength of concrete at temperature $\Theta$ |
| | $k_c$ | • | The coefficient |
| | $f_{ck}$ | • | The characteristic compressive cylinder strength of concrete at 28 days |

## Reinforcement steel

The characteristic strength of reinforcement steel is calculated according to the chapter *4.2.4.3*:

$$f_{sy,\Theta} = k_s f_{yk}$$

| | | | |
|---|---|---|---|
| Where is: | $f_{sy,\Theta}$ | • | The strength of reinforcement steel at temperature $\Theta$ |
| | $k_s$ | • | The coefficient |
| | $f_{yk}$ | • | The characteristic yield strength of reinforcement |

# Indicative strength class

The indicative (minimum) strength class of the concrete depends on the exposure class, that is defined in the table *4.1* of EN 1992-1-1. The indicative strength class is given in the table *E.1N*:

| Exposure class | Indicative strength class |
|---|---|
| X0 | C12/15 |
| XC1 | C20/25 |
| XC2 | C25/30 |
| XC3 | C30/37 |
| XC4 | C30/37 |
| XD1 | C30/37 |
| XD2 | C30/37 |
| XD3 | C35/45 |
| XS1 | C30/37 |
| XS2 | C35/45 |
| XS3 | C35/45 |
| XF1 | C30/37 (C25/30) |
| XF2 | C25/30 (aeration min.4%) |
| XF3 | C30/37 (C25/30 aeration min.4%) |
| XF4 | C30/37 (aeration min.4%) |
| XA1 | C30/37 (C25/30) |
| XA2 | C30/37 |
| XA3 | C35/45 |

*Note: values in brackets are valid for the Czech republic (table E1.CZ). The higher strength class shall be used for aeration lower than the minimum value.*

# Minimum cover

The minimum cover of reinforcement is calculated in accordance with the chapter *4.4.1* of EN 1992-1-1.

The nominal cover is calculated using formula *4.1*:

$$c_{nom} = c_{min} + \Delta c_{dev}$$

Where is: $c_{nom}$ • The nominal cover

$c_{min}$ • The minimum cover

$\Delta c_{dev}$ • The allowance in design for deviation

The minimum cover is calculated using formula *4.2*:

$$c_{min} = max \begin{Bmatrix} c_{min,b} \\ c_{min,dur} + \Delta c_{dur,\gamma} - \Delta c_{dur,st} - \Delta c_{dur,add} \\ 10mm \end{Bmatrix}$$

Where is: $c_{min,b}$ • The minimum cover due to bond requirement according to the table *4.2*

$c_{min,dur}$ • The minimum cover due to environmental conditions according to the table *4.4N*

$\Delta c_{dur,\gamma}$ • The additive safety element in accordance with *4.4.1.2(6)*

$\Delta c_{dur,st}$ • The reduction of minimum cover for use of stainless steel in accordance with *4.4.1.2(7)*

$\Delta c_{dur,add}$ • The reduction of minimum cover for use of additional protection in accordance with *4.4.1.2(8)*

## Creep factor

The creep coefficient takes creep and shrinkage into account during the analysis. The creep coefficient is calculated in accordance with *Annex B* of EN 1992-1-1. Cement grade *N* is considered during the analysis.

The creep coefficient $\varphi$ is calculated according to *B.1*:

$$\varphi = \varphi_0 \beta_c(t, t_0)$$

Where is: $\varphi_0$ • The notional creep coefficient

$\beta_c(t,t_0)$ • The coefficient to describe the development of creep with time after loading

The notional creep coefficient $\varphi_0$ is calculated according to *B.2*:

$$\varphi_0 = \varphi_{RH} \beta(f_{cm}) \beta(t_0)$$

Where is: $\varphi_{RH}$ • The notional creep coefficient

$\beta(f_{cm})$ • The factor to allow for the effect of concrete strength

$\beta(t_0)$ • The factor to allow for the effect of concrete age at loading

The notional creep coefficient $\varphi_{RH}$ is calculated using following formulas:

$$\varphi_{RH} = 1 + \frac{1 - RH/100}{0.1 \cdot \sqrt[3]{h_0}}$$ for $f_{cm} \leq 35MPa$

$$\varphi_{RH} = \left[1 + \frac{1 - RH/100}{0.1 \cdot \sqrt[3]{h_0}} \cdot \alpha_1\right] \cdot \alpha_2$$ for $f_{cm} < 35MPa$

Where is: $RH$ • The relative humidity of the ambient environment in *%*

$h_0$ • The notional size of the member

$\alpha_1, \alpha_2$ • The coefficients to consider the influence of the concrete strength

The notional size of the member $h_0$ is calculated according to the formula *(B.6)*:

$$h_0 = \frac{2A_c}{u}$$

Where is: $A_c$ • The area of cross-section

$u$ • The perimeter of the member in contact with the atmosphere

The coefficients to consider the influence of the concrete strength $\alpha_1$ and $\alpha_2$ are calculated using formulas *(B.8c)*:

$$\alpha_1 = \left[\frac{35}{f_{cm}}\right]^{0.7}, \alpha_2 = \left[\frac{35}{f_{cm}}\right]^{0.2}$$

Where is: $f_{cm}$ • The mean compressive strength of concrete at the age of 28 days

The factor for the effect of concrete strength $\beta(f_{cm})$ is calculated according to *(B.4)*:

$$\beta(f_{cm}) = \frac{16.8}{\sqrt{f_{cm}}}$$

The factor for allowance of the effect of concrete age at loading $\beta(t_0)$ is calculated according to *(B.5)*:

$$\beta(t_0) = \frac{1}{(0.1 + t_0^{0.2})}$$

| | | |
|---|---|---|
| Where is: | $t_0$ | • The age of concrete at loading |

The coefficient describing the development of creep with time after loading $\beta_c(t,t_0)$ iscalculated in accordance with *(B.7)*:

$$\beta_c(t, t_0) = \left[\frac{(t - t_0)}{(\beta_H + t - t_0)}\right]^{0.3}$$

| | | |
|---|---|---|
| Where is: | $t$ | • The age of concrete at the moment considered |
| | $t_0$ | • The age of concrete at loading |
| | $\beta_H$ | • The coefficient |

Coefficient βH is calculated using following formulas:

$$\beta_H = 1.5\left[1 + (0.012RH)^{18}\right] h_0 + 250 \leq 1500$$ for $f_{cm} \leq 35MPa$

$$\beta_H = 1.5\left[1 + (0.012RH)^{18}\right] h_0 + 250\alpha_3 \leq 1500\alpha_3$$ for $f_{cm} < 35MPa$

Coefficient $\alpha_3$ is calculated using formula *(B.8c)*:

$$\alpha_3 = \left[\frac{35}{f_{cm}}\right]^{0.5}$$

# Structural rules

Following structural rules are considered in the analysis:

- Minimum reinforcement area
- Maximum reinforcement area
- Minimum spacing of bars

## Minimum reinforcement area

The total area of tensile reinforcement for member types "**Beam**" and "**Slab**" is verified in accordance with *9.2.1.1(1)*:

$$A_{s,t} \geq max\begin{Bmatrix} 0.26\frac{f_{ctm}}{f_{yk}}b_t d \\ 0.0013 b_t d \end{Bmatrix}$$

| | | |
|---|---|---|
| Where is: | $A_{s,t}$ | • The total area of tensile reinforcement |
| | $f_{ctm}$ | • The mean value of axial tensile strength of concrete |
| | $f_{yk}$ | • The characteristic yield strength of reinforcement |
| | $b_t$ | • The mean width of the tension zone |
| | $d$ | • The effective depth of a cross-section |

Following verification is performed for member type "**Slab**", if the check of minimum reinforcement area according to CSN 73 1201, chapter *8.5.2* is switched on:

$$A_{s,t} \geq max\begin{Bmatrix} 0.0018\frac{f_{yk}}{500}A_c \\ 0.0014 A_c \end{Bmatrix}$$

| | | |
|---|---|---|
| Where is: | $A_{s,t}$ | • The total area of tensile reinforcement |
| | $A_c$ | • The area of cross-section |
| | $f_{yk}$ | • The characteristic yield strength of reinforcement |

The total area of vertical reinforcement for member type "**Column**" is verified in accordance with *9.5.2(2)*:

$$A_s \geq max \begin{Bmatrix} \frac{0.10N_{Ed}}{f_{yd}} \\ 0.002A_c \end{Bmatrix}$$

Where is:

- **$A_s$** • The total area of vertical reinforcement
- **$N_{Ed}$** • The design value of the compressive force
- **$f_{yd}$** • The design yield strength of reinforcement
- **$A_c$** • The area of cross-section

The area of vertical reinforcement for member type "**Walls**" is verified in accordance with *9.6.2(1)*:

$$A_s \geq 0.002A_c$$

Where is:

- **$A_s$** • The total area of vertical reinforcement
- **$A_c$** • The area of cross-section

### Maximum reinforcement area

All member types are verified in accordance with *9.2.1.1(3)*, *9.5.2(2)* and *9.5.2(2)*:

$$A_s \leq 0.04A_c$$

Where is:

- **$A_s$** • The total reinforcement area
- **$A_c$** • The area of cross-section

### Minimum spacing of bars

The minimum spacing of bars is checked in accordance with *8.2(2)*:

$$s_{min} = \begin{Bmatrix} k_1\phi \\ d_g + k_2 \\ 20mm \end{Bmatrix}$$

Where is:

- **$s_{min}$** • The clear distance between bars
- **$k_1, k_2$** • The constants, $k_1 = 1mm$, $k_2 = 5mm$
- **$\phi$** • The bar diameter
- **$d_g$** • the maximum size of aggregate

## Buckling

The second order effects are taken into account for the members loaded by compressive force in accordance with the chapter *5.8* of EN 1992-1-1. Following methods are avalable:

- Method based on the nominal stiffness
- Method based on the nominal curvature
- Simplified method based on 12.6.5.2 of the standard *(only plain concrete)*

### Slenderness criterion

The detailed analysis is performed if the slenderness is greater than limiting value $\lambda_{lim}$. The limiting slenderness is given by the expression *5.13N*:

$$\lambda_{lim} = 20 \cdot A \cdot B \cdot C\sqrt{n}$$

Where is:

- **$\lambda_{lim}$** • The limiting slenderness
- **$A, B, C$** • The coefficients
- **$n$** • The relative normal force

The coefficient *A* is given by the expression:

$$A = \frac{1}{1 + 0.2\varphi_{ef}}$$

Where is:

- **$\varphi_{ef}$** • The effective creep ratio

The coefficient *B* is given by the expression:

$$B = \sqrt{1 + 2\omega}$$

Where is: ***ω*** • The mechanical reinforcement ratio

The mechanical reinforcement ratio $\omega$ is given by the expression:

$$\omega = \frac{A_s f_{yd}}{A_c f_{cd}}$$

Where is:
- $A_s$ • The total area of longitudinal reinforcement
- $f_{yd}$ • The design yield strength of reinforcement
- $A_c$ • The cross-sectional area
- $f_{cd}$ • The design compressive strength of concrete

The coefficient $C$ is given by the expression:

$$C = 1.7 - r_m$$

Where is: $r_m$ • The moment ratio

The moment ratio $r_m$ is given by the expression:

$$r_m = \frac{M_{01}}{M_{02}}$$

Where is: $M_{01}$, $M_{02}$ • The first order end moments

The relative normal force $n$ is given by the expression:

$$n = \frac{N_{Ed}}{A_c f_{cd}}$$

Where is:
- $N_{Ed}$ • The design value of the normal force
- $A_c$ • The cross-sectional area
- $f_{cd}$ • The design compressive strength of concrete

## Method based on the nominal stiffness

This method is based on the chapter *5.8.7*. The nominal stiffness is given by expression *(5.21)*:

$$EI = K_c E_{cd} I_c + K_s E_s I_s$$

Where is:
- $K_c$ • The factor for effects of cracking, creep etc.
- $E_{cd}$ • The design value of the modulus of elasticity of concrete
- $I_c$ • The moment of inertia of concrete cross section
- $K_s$ • The factor for contribution of reinforcement
- $E_s$ • The design value of the modulus of elasticity of reinforcement
- $I_s$ • The second moment of area of reinforcement, about the centre of area of the concrete

The factor $K_c$ is given by expression *(5.22)*:

$$k_c = \frac{k_1 k_2}{(1 + \varphi_{ef})}$$

Where is:
- $k_1$ • The factor which depends on concrete strength class
- $k_2$ • The factor which depends on axial force and slenderness
- $\varphi_{ef}$ • The effective creep ratio

The factor $k_1$ is given by expression *(5.23)*:

$$k_1 = \sqrt{\frac{f_{ck}}{20}}$$

Where is: $f_{ck}$ • The characteristic compressive cylinder strength of concrete at 28 days

The factor $k_2$ is given by expression *(5.24)*:

$$k_2 = n\frac{\lambda}{170} \leq 0.2$$

| | | |
|---|---|---|
| Where is: | ***n*** | • The relative axial force |
| | ***λ*** | • The slenderness ratio |

The buckling load based on the nominal stiffness is calculated using expression:

$$N_B = \frac{\pi^2 EI}{{l_0}^2}$$

| | | |
|---|---|---|
| Where is: | $N_B$ | • The buckling load based on nominal stiffness |
| | ***EI*** | • The nominal stiffness |
| | $l_0$ | • The effective length for buckling analysis |

The total design moment, including second order moment, is calculated using formula *(5.28)*:

$$M_{Ed} = M_{0Ed}\left[1 + \frac{\beta}{\frac{N_B}{N_{Ed}} - 1}\right]$$

| | | |
|---|---|---|
| Where is: | $M_{0Ed}$ | • The first order moment |
| | $\beta$ | • The factor that depends on distribution of first and second order moments |
| | $N_B$ | • The buckling load based on nominal stiffness |
| | $N_{Ed}$ | • The design value of axial load |

The factor $\beta$ is given by expression *(5.29)*:

$$\beta = \frac{\pi^2}{c_0}$$

| | | |
|---|---|---|
| Where is: | $c_0$ | • The coefficient which depends on the distribution of first order moment |

The value of the factor $c_0$ is the input in the software. Following values are recommended according to the chapter *5.8.7.3(3)*:

| **Factor $c_0$** | **The distribution of first order moment** |
|---|---|
| 8.0 | constant |
| 9.6 | parabolic |
| 12 | symmetric triangular |

## Method based on the nominal curvature

This method uses procedures given in the chapter *5.8.8*. The nominal curvature is given by the expression *(5.34)*:

$$\frac{1}{r} = K_r \cdot K_\varphi \cdot \frac{1}{r_0}$$

| | | |
|---|---|---|
| Where is: | ***1/r*** | • The curvature |
| | $K_r$ | • The correction factor depending on axial load |
| | $K_\varphi$ | • The factor for taking account of creep |

The curvature $1/r_0$ is given by following formula:

$$\frac{1}{r_0} = \frac{\varepsilon_{yd}}{0.45d}$$

| | | |
|---|---|---|
| Where is: | $\varepsilon_0$ | • The strain of reinforcement at yield strength |
| | ***d*** | • The effective depth of a cross-section |

The strain of reinforcement at yield strength $\varepsilon_0$ is calculated using this formula:

$$\varepsilon_{yd} = \frac{f_{yd}}{E_s}$$

Where is: $f_{yd}$ • The design yield strength of reinforcement

$E_s$ • The design value of modulus of elasticity of reinforcing steel

The effective depth of a cross-section is given by the expression *(5.35)*:

$$d = \frac{h}{2} + i_s$$

Where is: $h$ • The height of a cross-section

$i_s$ • The radius of gyration of the total reinforcement area

The factor $K_r$ is given by the expression *(5.36)*:

$$K_r = min \begin{Bmatrix} \frac{n_u - n}{n_u - n_{bal}} \\ 1.0 \end{Bmatrix}$$

Where is: $n$ • The relative axial force

$n_{bal}$ • The value of the relative axial force at maximum moment resistance. The value *0.4* is usedin aaccordance with *5.8.8.3(3)*.

The relative axial force $n_u$ is calculated using the formula:

$$n_u = 1 + \omega$$

Where is: $\omega$ • The mechanical reinforcement ratio

The factor $K_\varphi$ is given by the expression *(5.37)*:

$$K_\varphi = max \begin{Bmatrix} 1 + \beta\varphi_{ef} \\ 1.0 \end{Bmatrix}$$

Where is: $\beta$ • The factor depending on the strength class of the concrete and the slenderness ratio

$\varphi_{ef}$ • The effective creep ratio

The factor $\beta$ is given by the expression:

$$\beta = 0.35 + \frac{f_{ck}}{200} - \frac{\lambda}{150}$$

Where is: $f_{ck}$ • The characteristic compressive cylinder strength of concrete at 28 days

$\lambda$ • The slenderness ratio

The nominal second order moment $M_2$ is given by the expression *(5.33)*:

$$M_2 = N_{Ed} e_2$$

Where is: $N_{Ed}$ • The design value of axial force

$e_2$ • The member deflection

he deflection $e_2$ is calculated using formula

$$e_2 = \frac{1}{r} \frac{{l_0}^2}{c}$$

Where is: $1/r$ • The curvature

$l_0$ • The effective length for buckling analysis

$c$ • The factor depending on the curvature distribution

The value of the factor $c$ is the input in the software. Following values are recommended according to the chapter *5.8.8.2(4)*:

| **Factor *c*** | **Curvature distribution** |
|---|---|
| 8.0 | constant |
| 10 | sinusoidal |

The design moment is given by the expression *(5.31)*:

$$M_{Ed} = M_{0Ed} + M_2$$

Where is: ***$M_{Ed}$*** • The design moment including the second order effect
***$M_{0Ed}$*** • The design value of the first order moment
***$M_2$*** • The nominal second order moment

## Simplified design method according to 12.6.5.2

This method may be used for plain concrete and lightly reinforced members according to the chapter *12.6.5.2* of EN 1992-1-1. The design axial resistance is given by the expression *(12.10)*:

$$N_{Rd} = b \cdot h_w \cdot f_{cd} \cdot \Phi$$

Where is: ***$b$*** • The overall width of the cross-section
***$h_w$*** • The overall depth of the cross-section
***$f_{cd}$*** • The design compressive strength of concrete
***$\Phi$*** • The factor taking into account eccentricity

The factor $\Phi$ is given by the expression *(12.11)*:

$$\Phi = 1.14\left(1 - \frac{2e_{tot}}{h_w}\right) - 0.02\frac{l_0}{h_w} \leq \left(1 - \frac{2e_{tot}}{h_w}\right)$$

The eccentricity $e_{tot}$ is calculated using following formula:

$$e_{tot} = e_0 + e_i$$

Where is: ***$e_0$*** • The first order eccentricity including, where relevant, the effects of floors (e.g. possible clamping moments transmitted to the wall from a slab) and horizontal actions
***$e_i$*** • The additional eccentricity covering the effects of geometrical imperfections
***$l_0$*** • The effective length for buckling analysis

# Anchorage

Anchorage lengths are calculated in accordance with the chapter *8.4* of EN 1992-1-1. The basic required anchorage length $l_{b,rqd}$ is calculated according to the formula *8.3* for any bar:

$$l_{b,rqd} = \left(\frac{\phi}{4}\right)\left(\frac{\sigma_{sd}}{f_{b,d}}\right)$$

Where is: ***$l_{b,rqd}$*** • The basic required anchorage length
***$\phi$*** • The bar diameter
***$\sigma_{sd}$*** • The design stress of the bar
***$f_{bd}$*** • The design value of the ultimate bond stress

The design value of the ultimate bond stress $f_{bd}$ is calculated using formula *8.2*:

$$f_{bd} = 2.25\eta_1\eta_2 f_{ctd}$$

Where is: ***$f_{ctd}$*** • The design value of concrete tensile strength
***$\eta_1$*** • The coefficient related to the quality of the bond condition and the position of the barduring concreting (value is *1.0* for good bond conditions and *0.7* for other cases)
***$\eta_2$*** • The coefficient related to the bar diameter. Value is *1.0* for all bars with diameter up to *32mm*. The coefficient is calculated using following formula for other cases:

$$\eta_2 = \frac{132 - \phi}{100}$$

The design anchorage length $l_{bd}$ is calculated using formula *8.4*:

$$l_{bd} = \alpha_1\alpha_2\alpha_3\alpha_4\alpha_5 l_{b,rqd} \geq l_{b,min}$$

Where is: ***$l_{b,min}$*** • The minimum anchorage length
***$\alpha_1$*** • The coefficient in accordance with table *8.2* considering the effect of the bars form assuming adequate cover

| | | |
|---|---|---|
| ***α₂*** | • | The coefficient in accordance with table *8.2* considering the effect of concrete minimum cover |
| ***α₃*** | • | The coefficient in accordance with table *8.2* considering the effect of confinement by transverse reinforcement |
| ***α₄*** | • | The coefficient in accordance with table *8.2* considering the influence of one or more welded transverse bars along the design anchorage length |
| ***α₅*** | • | The coefficient in accordance with table *8.2* considering the effect of the pressure transverse to the plane of splitting along the design anchorage length |

The minimum anchorage length $l_{b,min}$ is calculated using formula *8.6* for anchorages in tension:

$$l_{b,min} > max \begin{Bmatrix} 0.3 l_{b,rqd} \\ 10\phi \\ 100mm \end{Bmatrix}$$

The minimum anchorage length $l_{b,min}$ is calculated using formula *8.7* for anchorages in compression:

$$l_{b,min} > max \begin{Bmatrix} 0.6 l_{b,rqd} \\ 10\phi \\ 100mm \end{Bmatrix}$$

# Methods for fire resistance analysis

## 500°C isotherm method

This method is based on the chapter *B.1* of EN 1992-1-2. It may be used in combination with standard or parametric temperature curve. The minimum member width according to the table *B.1* of EN 1992-1-2 has to be respected.

Analysis procedure:

- The calculation of the temperature distribution with the help of finite element method
- The determination of the new effective cross-section by excluding the concrete outside the *500°C* isotherm. The material characteristics aren't reduced for this cross-section
- The determination of the reduced material characteristics for all bars of the reinforcement according to their temperatures
- The verification of the new effective cross-section using geometric and material characteristics mentioned above. The reinforcement placed outside the 500°C isotherm is considered in the analysis.

## Zone method

This method is based on the chapter *B.2*. This method provides provides more accurate results, mainly for the columns. It may be used only in combination with the standard temperature curve.

Analysis procedure:

- The cross-section is divided into specified number of parallel zones (default count is *100*)
- The mean temperature and the corresponding mean compressive strength $f_{cd}(\Theta)$ is calculated in every zone
- The mean reduction coefficient for a particular section $k_{c,m}$ is calculated using expression *(B.11)*:

$$k_{c,m} = \frac{(1 - 0.2/n)}{n} \sum_{i=1}^{n} k_c(\Theta_i)$$

| Where is: | | | |
|---|---|---|---|
| | ***(1-0.2)n*** | • | The factor which allows for the variation in temperature within each zone |
| | ***n*** | • | The number of parallel zones in the half of width |
| | ***$k_c(\Theta_i)$*** | • | The reduction factor for compressive strength in *i* zone for temperature *Θ* |

- The thickness of the damaged zone is calculated. The following expression *(B.12)* is used for member types "**Beam**" and "**Slab**":

$$a_z = w \left[ 1 - \frac{k_{c,m}}{k_c(\Theta_M)} \right]$$

The following expression *(B.13)* is used for member types "**Column**" and "**Wall**" and for all other members that have the buckling verification switched on:

$$a_z = w\left[1-\left(\frac{k_{c,m}}{k_c\left(\Theta_M\right)}\right)^{1.3}\right]$$

Where is:

- $w$ — The half of the cross-section width
- $k_c(\Theta_M)$ — The reduction factor for compressive strength in the centre of the cross-section

# Temperature curves

Following temperature curves are used for the description of the gas temperature according to EN 1991-1-2.

## Standard temperature curve

The fundamental nominal curve, the temperature of gas in the fire compartment is given by the expression

$$\Theta = 20 + 345 log_{10}\left(8t+1\right)$$

Where is:

- $\theta_g$ — The gas temperature in the fire compartment in *°C*
- $t$ — The time in minutes

## Parametric temperature curve

This curve is valid for fire compartments up to $500m^2$, without openings in the roof and for a maximum compartment height of *4m*. The curve consists of two phases: heating phase and cooling phase. Following parameters describe the geometry of the curve:

- The time $t_{lim}$ for maximum gas temperature in case of fuel controlled fire
- The thermal absorptivity for the total enclosure *b* given by the expression

$$b = \sqrt{\rho c \lambda}$$

Where is:

- $\rho$ — The density of boundary of enclosure in $kg/m^3$
- $c$ — The specific heat of boundary of enclosure in *J/(kg K)*
- $\lambda$ — thermal conductivity of boundary of enclosure in *W/(m K)*

- The opening factor *O* given by the expression

$$O = A_v\frac{\sqrt{h_{eq}}}{A_t}$$

Where is:

- $A_v$ — The total area of vertical openings on all walls in $m^2$
- $h_{eq}$ — The weighted average of window heights on all walls in *m*
- $A_t$ — The total area of enclosure (walls, ceiling and floor, including openings) in $m^2$

- The design value of the fire load density $q_{t,d}$ related to the total surface area $A_t$ of the enclosure.

This curve may be used only for "**500°C isotherm method**".

# Ultimate limit state

Bending moment with/without normal force is verified in accordance with EN 1992-1-1, chapter *6.1*

The verification of ultimate limit state is based on following assumptions:

- plane sections remain plane
- the strain in reinforcement is the same as in the surrounding concrete
- the tensile strength of the concrete is ignored
- the stresses in the concrete in compression are derived from the design stress-strain relationship (parabola-rectangle diagram)
- the stresses in the reinforcing steel are derived from the design curve with an inclined top branch with a strain limit

# Ultimate limit state - shear

The verification of the shear resistance is performed according to the chapter *6.2* of EN 1992-1-1.

## Members not requiring design shear reinforcement

The design value for the shear resistance $V_{Rd,c}$ is given according to the chapter *6.2.2(1)*:

$$V_{Rd,c} = \left[C_{Rd,c} k \left(100 \rho_l f_{ck}\right)^{1/3} + k_1 \sigma_{cp}\right] b_w d$$

| | | |
|---|---|---|
| Where is: | $\rho_l$ | • The reinforcement ratio for longitudinal reinforcement |
| | $f_{ck}$ | • The characteristic compressive cylinder strength of concrete at *28* days |
| | $b_w$ | • The smallest width of the cross-section in the tensile area |
| | $d$ | • The effective depth of a cross-section |

The minimum value of the shear resistance $V_{Rd,c}$ is given by expression *(6.2b)*:

$$V_{Rd,c} = \left(v_{min} + k_1 \sigma_{cp}\right) b_w d$$

The coefficient $k$ is given by

$$k = 1 + \sqrt{\frac{200}{d}}$$

| | | |
|---|---|---|
| Where is: | $d$ | • The effective depth of a cross-section |

The reinforcement ratio $\rho_l$ is calculated using following formula

$$\rho_l = min\left\{\frac{A_{sl}}{b_w d}; 0.02\right\}$$

| | | |
|---|---|---|
| Where is: | $A_{sl}$ | • The area of the tensile reinforcement |

The concrete compressive stress at the centroidal axis due to axial loading $\sigma_{cp}$ is given by expression

$$\sigma_{cp} = min\left\{\frac{N_{Ed}}{A_c}; 0.2 f_{cd}\right\}$$

| | | |
|---|---|---|
| Where is: | $N_{Ed}$ | • The design value of the applied axial force |
| | $A_c$ | • The cross sectional area of concrete |
| | $f_{cd}$ | • The design value of concrete compressive strength |

The value of $C_{Rd,c}$ is given by formula

$$C_{Rd,c} = \frac{0.18}{\gamma_c}$$

The value of $v_{min}$ is calculated using the formula *(6.3N)*:

$$v_{min} = 0.035 k^{3/2} \sqrt{f_{ck}}$$

## Members requiring design shear reinforcement

The shear resistance is calculated with the help of formula *(6.8)*:

$$V_{Rd,s} = \frac{A_{sw}}{s} z \cdot f_{yd} \cdot cot\Theta$$

| | | |
|---|---|---|
| Where is: | $A_{sw}$ | • The cross-sectional area of the shear reinforcement |
| | $s$ | • The spacing of the stirrups |
| | $f_{yd}$ | • The design yield strength of the shear reinforcement |
| | $z$ | • The inner lever arm. This value may be obtained from the analysis or specified by the user. |
| | $\Theta$ | • The angle between the concrete compression strut and the beam axis perpendicular to the shear force |

The shear resistance for members with inclined shear reinforcement is calculated with the help of formula *(6.13)*:

$$V_{Rd,s} = \frac{A_{sw}}{s} z \cdot f_{yd} \cdot \left(cot\Theta + cot\alpha\right) sin\alpha$$

The maximum shear force limited by crushing of the compression struts is calculated using formula *(6.9)*:

$$V_{Rd,max} = \frac{\alpha_{cw} b_w z \nu_1 f_{cd}}{cot\Theta + tan\Theta}$$

Where is:
- $\alpha_{cw}$ • The coefficient taking account of the state of the stress in the compression chord, value is *1.0*
- $b_w$ • The minimum width between tension and compression chords
- $v_1$ • The strength reduction factor for concrete cracked in shear
- $f_{cd}$ • The design value of concrete compressive strength
- $\Theta$ • The angle between the concrete compression strut and the beam axis perpendicular to the shear force

The maximum shear force limited by crushing of the compression struts for members with inclined shear reinforcement is calculated using formula *(6.14)*:

$$V_{Rd,max} = \alpha_{cw} b_w z \nu_1 f_{cd} \frac{cot\Theta + cot\alpha}{1 + cot^2\Theta}$$

The strength reduction factor for concrete cracked in shear is calculated using formula *(6.6N)*:

$$\nu_1 = 0.6\left(1 - \frac{f_{ck}}{250}\right)$$

The minimum of values $V_{Rd,s}$ and $V_{Rd,max}$ is considered as the design shear resistance $V_{Rd}$.

# Torsion (ULS)

The torsional resistance is calculated according to the chapter *6.3* of EN 1992-1-1. The analysis is based on the basis of a thin-walled closed section. The effective wall thickness is calculated in accordance with the chapter *6.3.2*:

$$t_{ef} = \frac{A_c}{u}$$

Where is:
- $t_{ef}$ • The effective wall thickness
- $A_c$ • The total area of the cross-section
- $u$ • The outer circumference of the cross-section

## The torsional resistance without reinforcement

The verification in accordance with the chapter *6.3.2(5)* is done for members without any torsional reinforcement:

$$\frac{T_{Ed}}{T_{Rd,c}} + \frac{V_{Ed}}{V_{Rd,c}} \leq 1.0$$

Where is:
- $T_{Ed}$ • The design torsional moment
- $T_{Rd,c}$ • The torsional cracking moment
- $V_{Ed}$ • The design shear force
- $V_{Rd,c}$ • The design value for the shear resistance

The resistance $T_{Rd,c}$ is given by the expression *(6.26)*:

$$T_{Rd,c} = 2 f_{ctd} A_k t_{ef}$$

Where is:
- $f_{ctd}$ • The design tensile strength of concrete
- $A_k$ • The area enclosed by the centre-lines of the connecting walls

## The resistance including torsional reinforcement

The capacity of the concrete struts is given by the formula *(6.29)*:

$$\frac{T_{Ed}}{T_{Rd,max}} + \frac{V_{Ed}}{V_{Rd,max}} \leq 1.0$$

Where is:
- $T_{Ed}$ • The design torsional moment
- $T_{Rd,max}$ • The design torsional resistance moment
- $V_{Ed}$ • The design shear force
- $V_{Rd,max}$ • The maximum design shear resistance

The design torsional resistance moment $T_{Rd,max}$ is given by the expression *(6.30)*:

$$T_{Rd,max} = 2\nu\alpha_{cw} f_{cd} A_k t_{ef} sin\Theta cos\Theta$$

| | | |
|---|---|---|
| Where is: | $\alpha_{cw}$ | • The coefficient taking account of the state of the stress in the compression chord, value is *1.0* |
| | $A_k$ | • The area enclosed by the centre-lines of the connecting walls |
| | $v$ | • The strength reduction factor for concrete cracked in shear |
| | $f_{cd}$ | • The design value of concrete compressive strength |
| | $\Theta$ | • The angle between the concrete compression strut and the beam axis perpendicular to the shear force |

The design shear force in the transverse reinforcement $V_{Edt}$ is given by the expression

$$V_{Edt} = \frac{T_{Ed}}{2A_k}$$

| | | |
|---|---|---|
| Where is: | $V_{Edt}$ | • The design shear force in the transverse reinforcement caused by torsional moment |
| | $T_{Ed}$ | • The design torsional moment |
| | $A_k$ | • The area enclosed by the centre-lines of the connecting walls |

The design resistance of the transverse reinforcement $V_{Rdt}$ is given by the expression:

$$V_{Rdt} = \frac{A_{sw}}{s} \cdot f_{yd} \cdot cot\Theta$$

| | | |
|---|---|---|
| Where is: | $A_{sw}$ | • The cross-sectional area of the transverse reinforcement considered for the torsional resistance |
| | $s$ | • The spacing of the stirrups |
| | $f_{yd}$ | • The design yield strength of the shear reinforcement |
| | $\Theta$ | • The angle between the concrete compression strut and the beam axis perpendicular to the shear force |

The verification is done using following expression:

$$\frac{V_{Edt}}{V_{Rdt}} \leq 1.0$$

# Serviceability limit state

Serviceability limit states are verified according to the chapter *7* of EN 1992-1-1. Analysis of serviceability limit states checks the specified service requirements for a structure or structural member during its working life.

Cross-sections are assumed to be cracked for any tensile stress during the analysis of serviceability limit states. The user defined setting is able to consider cross-sections with tensile stress up to the mean value of axial tensile strength $f_{ctm}$ as uncracked.

## Stress limitation

The verification is based on the chapter *7.2*. The analysis is performed for the load (combination) type "**Characteristic**".

The stress level under the characteristic load is be limited due to occurrence of longitudinal cracks. Maximum compressive stress is given by expression

$$\sigma_{c,lim} = k_1 f_{ck}$$

| | | |
|---|---|---|
| Where is: | $k_1$ | • The coefficient, the value is *0.6* |
| | $f_{ck}$ | • The characteristic compressive cylinder strength of concrete at *28* days |

The verification is done according to the chapter *7.2(2)* only for the environmental conditions *XD*, *XF* or *XS*.

Tensile stresses in the reinforcement is limited in order to avoid inelastic strain, unacceptable cracking or deformation. The maximum tensile stress is limited in accordance with *7.2(5)* by the formula

$$\sigma_{c,lim} = k_3 f_{yk}$$

| | | |
|---|---|---|
| Where is: | $k_3$ | • The coefficient, the value is *0.8* |
| | $f_{yk}$ | • The characteristic yield strength of reinforcement |

The ratio of stiffness of reinforcement and concrete may be specified for the design standard EN 1992-2. This ratio may take account of the degradation of modulus of elasticity of concrete due to creep and similar effects. Such procedure may be required by consequent standards (e.g CSN 73 6214, chapter *6*).

## Crack control

The verification is based on the chapter *7.3*. Cracking limitation ensures the proper functioning and durability of the structure and keep the appearance in acceptable state. . The analysis is performed for the load (combination) type "**Quasi-permanent**".

Crack width is calculated according to the chapter *7.3.4*. The crack width $w_k$ is given by formula *(7.8)*:

$$w_k = s_{r,max}\left(\varepsilon_{sm} - \varepsilon_{cm}\right)$$

| | | |
|---|---|---|
| Where is: | $s_{r,max}$ | • The maximum crack spacing |
| | $\varepsilon_{sm}$ | • The mean strain in the reinforcement under the relevant combination of loads. Only the additional tensile strain beyond the state of zero strain of the concrete at the same level is considered |
| | $\varepsilon_{cm}$ | • The mean strain in the concrete between cracks |

Expression $\varepsilon_{sm}$-$\varepsilon_{cm}$ is given by *(7.9)*:

$$\varepsilon_{sm} - \varepsilon_{cm} = \frac{\sigma_s - k_t \frac{f_{ct,eff}}{\rho_{peff}}\left(1 + \alpha_e \rho_{p,eff}\right)}{E_s} \geq 0.6\frac{\sigma_s}{E_s}$$

| | | |
|---|---|---|
| Where is: | $\sigma_s$ | • The stress in the tension reinforcement assuming a cracked section |
| | $\alpha_e$ | • The ratio $E_s/E_{cm}$ |
| | $k_t$ | • The factor dependent on the duration of the load. The value is *0.6* for short-term loads and *0.4* for long-term loads |

and

$$\rho_{p,eff} = \frac{A_s}{A_{c,eff}}$$

| | | |
|---|---|---|
| Where is: | $A_s$ | • The area of reinforcing steel |
| | $A_{c,eff}$ | • The effective area of concrete in tension surrounding the reinforcement |

The maximum crack spacing $s_{r,max}$ is given by *(7.11)*:

$$s_{r,max} = k_3 c + \frac{k_1 k_2 k_4 d}{\rho_{p,eff}}$$

| | | |
|---|---|---|
| Where is: | $k_1$ | • The coefficient which takes account of the bond properties of the bonded reinforcement. The value for high bond bars is *0.8*. |
| | $k_2$ | • The coefficient which takes account of the distribution of strain. The value is *1.0* for pure tension and *0.5* for bending. |
| | $k_3$ | • The coefficient, the value is *3.4* |
| | $k_4$ | • The coefficient, the value is *0.425* |
| | $c$ | • The cover to the longitudinal reinforcement |
| | $d$ | • The bar diameter. Where a mixture of bar diameters appears, an equivalent diameter is used. |

The equivalent diameter $d$ is given by expression *(7.12)*:

$$d = \frac{n_1 {d_1}^2 + n_2 {d_2}^2}{n_1 d_1 + n_2 d_2}$$

| | | |
|---|---|---|
| Where is: | $n_1$ | • The number of bars of the diameter $d_1$ |
| | $n_2$ | • The number of bars of the diameter $d_2$ |

The value of maximum crack width $w_{max}$ is based on the table *7.1N*.

## Deflection control *(only program Concrete Beam)*

The deflection is calculated using the exact analysis in accordance the recommendation in *7.4.3(7)*. First, the curvatures at frequent sections along the member are calculated. This is followed by the calculation of deflection using the numerical integration. The deformation parameters at points where the section isn't fully cracked are obtained using the expression *(7.18)* that is described in the chapter *7.4.3(3)*:

$$\alpha = \zeta \alpha_{||} + \left(1 - \zeta\right)\alpha_{|}$$

| | | |
|---|---|---|
| Where is: | $\alpha$ | • The deformation parameter (e.g. a strain, a curvature, a rotation) |
| | $\zeta$ | • The distribution coefficient |

| | | |
|---|---|---|
| | $\alpha_I$ | • The parameter calculated for the uncracked conditions |
| | $\alpha_{II}$ | • The parameter calculated for the fully cracked conditions |

The distribution coefficient $\zeta$ is given by *(7.19)*:

$$\zeta = 1 - \beta \left( \frac{\sigma_{sr}}{\sigma_s} \right)^2$$

| | | |
|---|---|---|
| Where is: | $\zeta$ | • The distribution coefficient |
| | $\beta$ | • The coefficient taking account of the influence of the duration of the loading. The value is *1.0* for pure tension and *0.5* for bending. |
| | $\sigma_{sr}$ | • The stress in the tension reinforcement calculated on the basis of a cracked section under the loading conditions causing first cracking |
| | $\sigma_s$ | • The stress in the tension reinforcement calculated on the basis of a cracked section |

The curvature due to shrinkage is given by the expression *(7.21)*:

$$\frac{1}{r_{cs}} = \varepsilon_{cs} \alpha_e \frac{S}{I}$$

| | | |
|---|---|---|
| Where is: | $1/r_{cs}$ | • The curvature due to shrinkage |
| | $\varepsilon_{cs}$ | • The free shrinkage strain |
| | $\alpha_e$ | • The effective modular ratio |
| | $S$ | • The first moment of area of the reinforcement about the centroid of the section |
| | $I$ | • The second moment of area of the section |

The effective modular ratio $\alpha_e$ is given by:

$$\alpha_e = \frac{E_s}{E_{c,eff}}$$

| | | |
|---|---|---|
| Where is: | $\alpha_e$ | • The effective modular ratio |
| | $E_s$ | • Tesign value of modulus of elasticity of reinforcing steel |
| | $E_{c,eff}$ | • The effective modulus of elasticity for concrete |

The effective modulus of elasticity for concrete $E_{c,eff}$ is calculated using the formula *(7.20)*:

$$E_{c,eff} = \frac{E_{cm}}{1 + \varphi(\infty, t_0)}$$

| | | |
|---|---|---|
| Where is: | $E_{c,eff}$ | • The effective modulus of elasticity for concrete |
| | $E_{cm}$ | • The secant modulus of elasticity of concrete |
| | $\varphi(\infty, t_0)$ | • The creep coefficient relevant for the load and time interval |

# Punching

The punching analysis is based on the chapter *6.4* of EN 1992-1-1. Openings are considered exactly according to the input (the condition, that the distance of the opening should be shorter than $6d$, isn't checked and consideration of distant openings depends on the designer's decision). Control perimeters are found according to the chapter *6.4.2(1)* using the distance between perimeters equal to $2d$.

## Shear stress in control perimeter

The maximum shear stress in control perimeter is given by the expression

$$v_{Ed} = \beta \frac{V_{Ed}}{u_i d}$$

| | | |
|---|---|---|
| Where is: | $\beta$ | • The coefficient |
| | $V_{Ed}$ | • The design value of shear force |
| | $d$ | • The mean effective depth of the slab |
| | $u_i$ | • The length of the control perimeter being considered |

The mean effective depth of the slab is calculated using formula

$$d = \frac{d_y + d_z}{2}$$

Where is: $d_y$ • The effective depth of the slab in direction *y*
$d_z$ • The effective depth of the slab in direction *z*

The coefficient $\beta$ may be entered manually, selected according to the figure *6.21N* or calculated using expression *(6.39)*:

$$\beta = 1 + k\frac{M_{Ed}}{V_{Ed}} \cdot \frac{u_1}{W_1}$$

Where is: $u_1$ • The length of the control perimeter being considered
$k$ • The coefficient dependent on the ratio between the column dimensions $c_1$ and $c_2$. The value is a function of proportions of unbalanced moment transmitted by uneven shear and by bending and torsion. Values are based on the table *6.1*. The factor is calculated in the direction of bending moment.
$W_1$ • The modulus that corresponds to the shear distribution in the figure *6.19*

The program provides also an option to use general formula for calculation of the factor $\beta$, (the option "**Calculate $\beta$ according to 6.4.3(3-5) - in axes directions**" in the part "**Analysis**").

$$\beta = 1 + \frac{u_1}{V_{Ed}} \cdot \left( k_x \frac{M_{Ed,x} + V_{Ed} \cdot e_{x,1}}{W_{x,1}} + k_y \frac{M_{Ed,y} + V_{Ed} \cdot e_{y,1}}{W_{y,1}} \right)$$

Where is: $u_1$ • The length of the control perimeter being considered
$k_x, k_y$ • The coefficients dependent on the ratio between the column dimensions $c_1$ and $c_2$. The values are functions of proportions of unbalanced moment transmitted by uneven shear and by bending and torsion. Values are based on the table *6.1*. The factor is calculated in the direction of bending moment.
$M_{Ed,x}, M_{Ed,y}$ • Bending moments in directions $x$ and $y$
$e_{x,1}, e_{y,1}$ • The eccentricities of centre of gravity of control perimeter relative to the centre of gravity of column
$W_{x,i}, W_{y,i}$ • The moduli in directions $x$ and $y$, recalculated relatively to the centre of gravity of control perimeter

The modulus $W_1$ is calculated according to the expression *(6.40)* using the numerical integration:

$$W_1 = \int_0^{u_i} |e| \, dl$$

Where is: $dl$ • The length increment of the perimeter
$e$ • The distance of $dl$ from the axis about which the moment $M_{Ed}$ acts

## Maximum punching shear resistance

The following verification is done for any control perimeter:

$$v_{Ed} < v_{Rd,max}$$

Where is: $v_{Ed}$ • The design value of shear stress
$v_{Rd,max}$ • The design value of the maximum punching shear resistance

The design value of the maximum punching shear resistance is calculated according to the chapter *6.4.5(3)*:

$$v_{Rd,max} = 0.4 \nu f_{cd}$$

where

$$\nu = 0.6 \left[ 1 - \frac{f_{ck}}{250} \right]$$

## Punching shear resistance of a slab without punching shear reinforcement

The punching shear reinforcement isn't necessary if the following condition is fulfilled:

$$v_{Ed} < v_{Rd,c}$$

Where is: $v_{Ed}$ • The design value of shear stress
$v_{Rd,c}$ • The design value of the punching shear resistance of a slab without punching shear reinforcement

The design value of the punching shear resistance of a slab without punching shear reinforcement is calculated in accordance with the chapter *6.4.4(1)*:

$$V_{Rd,c} = C_{Rd,c}k\left(100\rho_l f_{ck}\right)^{1/3} + k_1\sigma_{cp} \geq \left(v_{min} + k_1\sigma_{cp}\right)$$

where

$$k = 1 + \sqrt{\frac{200}{d}} \leq 2.0$$

and

$$\rho_l = \sqrt{\rho_{ly} \cdot \rho_{lz}} \leq 0.02$$

Where is: $\rho_{ly}, \rho_{lz}$ • The reinforcement ratios for bonded tension reinforcement in slab parallel to axis *y* and *z*

The compressive stress in the concrete from axial load $\sigma_{cp}$ is given by the expression

$$\sigma_{cp} = \frac{\sigma_{cy} + \sigma_{cz}}{2}$$

The normal concrete stresses in the critical section $\sigma_{cy}$ and $\sigma_{cz}$ are given by expressions:

$$\sigma_{cy} = \frac{N_{Ed,y}}{A_{cy}}; \sigma_{cz} = \frac{N_{Ed,z}}{A_{cz}}$$

Where is: $N_{Ed,y}$, $N_{Ed,z}$ • The longitudinal forces across the full bay (internal columns) or the longitudinal force across the control section (edge columns). The force may be from a load or prestressing.

$A_{cy}, A_{cz}$ • Corresponding areas of of concrete

The value $v_{min}$ is calculated according to the chapter *6.2.2(1)*:

$$v_{min} = 0.035k^{2/3} \cdot {f_{ck}}^{1/2}$$

## Punching shear resistance with shear reinforcement

If the shear reinforcement is required, the procedure according to *6.4.5(1)* is used:

$$v_{Rd,cs} = 0.75v_{Rd,c} + 1.5\frac{d}{s_r}A_{sw}f_{ywd,eff}\frac{1}{u_1 d}sin\alpha$$

Where is:
- $A_{sw}$ • The area of shear reinforcement in the perimeter
- $s_r$ • The radial spacing of perimeters of shear reinforcement
- $f_{ywd,eff}$ • The effective design strength of the punching shear reinforcement
- $d$ • The mean of the effective depths in the orthogonal directions
- $\alpha$ • The angle between the shear reinforcement and the plane of the slab

The previous expression requires constant value of $s_r$ between individual perimeters of shear reinforcement and also constant area $A_{sw}$ in all these perimeters. The expression was modified to allow input of different values of $s_r$ and $A_{sw}$:

$$v_{Rd,cs} = 0.75v_{Rd,c} + 0.75A_{sw,x}f_{ywd,eff}\frac{1}{u_1 d}sin\alpha$$

Where is: $A_{sw,x}$ • The real area of shear reinforcement between verified and previous control perimeter (the zone with the width equal to $2d$ ). The area of reinforcement between control perimeter and the column is used for perimeters with the distance shorter than $2d$ in the foundation slabs. The considered area may be extended to $2d$ when using the setting "**Always consider reinforcement in the range 0 to 2d**". This setting is not recommended.

The area $A_{sw,x}$ alternates the following expression in the formula *(6.52)*

$$2d\frac{A_{sw}}{s_r}$$

This expression describes the reinforcement area in the strip with the width *2d* for constant $A_{sw}$ and $s_r$.

The effective design strength of the punching shear reinforcement is given by the expression

$$f_{ywd,eff} = 250 + 0.25d \leq f_{ywd}$$

The length of the control perimeter where the shear reinforcement is not required is deinfed in the chapter *6.4.5(4)*:

$$u_{out,ef} = \frac{\beta V_{Ed}}{V_{Rd,c} d}$$

# Punching - structural rules

The structural rules are checked in accordance with the chapter *9.4.3*. This verification may be switched off in the software. Following rules are checked:

- The spacing of link leg perimeters should not exceed *0,75d*
- The first link leg perimeters should be within the interval *(0,3d;0,5d)*
- The spacing of link legs around a perimeter should not exceed *1,5d* for the first control perimeter (*2d* from loaded area)
- The spacing of link legs around a perimeter should not exceed *2,0d* for other control perimeters

where *d* is the effective height of the cross-section.

# Corbel

The analysis in the software "**Corbel**" is done according to the following procedures:

## Design value of the stress in joint

The value of the design compressive strength is defined as

$$f_{cd} = \alpha_{cc} f_{ck} / \gamma_c$$

| Where is: | $f_{cd}$ | • The design compressive strength |
|---|---|---|
| | $\alpha_{cc}$ | • The coefficient taking account of long term effects on the compressive strength |
| | $f_{ck}$ | • The characteristic compressive cylinder strength of concrete at 28 days |
| | $\gamma_c$ | • The partial safety factor for concrete |

The design values for the compressive stresses in nodes where no ties are anchored at the node (type CCC) is given by the following expression according to the chapter *6.5.4. (4)*:

$$\sigma_{Rd,max} = k_1 \nu' f_c d$$

The design values for the compressive stresses in compression - tension nodes with anchored ties provided in one direction (type CCT) is given by the following expression according to the chapter *6.5.4. (4)*:

$$\sigma_{Rd,max} = k_2 \nu' f_c d$$

The design values for the compressive stresses in compression - tension nodes with anchored ties provided in more than one direction (type CTT) is given by the following expression according to the chapter *6.5.4. (4)*:

$$\sigma_{Rd,max} = k_3 \nu' f_c d$$

| Where is: | $\sigma_{Rd,max}$ | • The maximum stress which can be applied at the edges of the node |
|---|---|---|
| | $k_1$ | • The coefficient for CCC joints |
| | $k_2$ | • The coefficient for CCT joints |
| | $k_3$ | • The coefficient for CTT joints |
| | $v'$ | • The coefficient |
| | $f_{cd}$ | • The design compressive strength |

The reduction coefficient *v'* is calculated in accordance with the chapter *6.5.2 (2)*:

$$\nu' = 1 - f_{c,k}/250$$

| Where is: | $f_{ck}$ | • The characteristic compressive cylinder strength of concrete at 28 days |
|---|---|---|

## Strut and tie model

The width (horizontal projection) of the compressive zone in the node is given by the expression:

$$x_1 = \frac{F_{Ed}}{b \cdot \sigma_{Rd,max}}$$

| Where is: | $x_1$ | • The width of the compressive zone |
|---|---|---|
| | $F_{Ed}$ | • The design value of vertical force |
| | $b$ | • The corbel width |

| | | |
|---|---|---|
| | $\sigma_{Rd,max}$ | • The design value for the compressive stress in node |

The outer lever arm is calculated using formula:

$$a = a_c + 0.5x_1 + \frac{H_{Ed}}{F_{Ed}}(d' + \Delta h)$$

| | | |
|---|---|---|
| Where is: | $a$ | • The outer lever arm |
| | $a_c$ | • The eccentricity of the vertical force |
| | $x_1$ | • The width of the compressive zone |
| | $H_{Ed}$ | • The design value of horizontal force |
| | $F_{Ed}$ | • The design value of vertical force |
| | $d'$ | • The difference between the corbel height and the effective height |
| | $\Delta h$ | • The height of slide plate |

The width (vertical projection) of the compressive zone in the node is given by the expression:

$$y_1 = d - \sqrt{d^2 - 2x_1 a}$$

| | | |
|---|---|---|
| Where is: | $a$ | • The outer lever arm |
| | $d$ | • The effective height of the cross-section |
| | $x_1$ | • The width of the compressive zone |

The inner lever arm is given by the expression:

$$z = d - 0.5y_1$$

The tensile force in the main reinforcement is given by the expression:

$$F_t = F_{Ed}\frac{a}{z} + H_{Ed}(1 + d'/z)$$

| | | |
|---|---|---|
| Where is: | $F_{Ed}$ | • The design value of vertical force |
| | $a$ | • The outer lever arm |
| | $z$ | • The inner lever arm |
| | $H_{Ed}$ | • The design value of horizontal force |
| | $d'$ | • The difference between the corbel height and the effective height |

The force in the compression strut is given by the expression:

$$F = F_{Ed}/sin(\Theta)$$

| | | |
|---|---|---|
| Where is: | $\Theta$ | • The angle between the concrete compression strut and the beam axis perpendicular to the shear force |
| | $F_{Ed}$ | • The design value of vertical force |

## The shear resistance without transverse reinforcement

The shear resistance of the cross-section without stirrups is verified according to the chapter *6.2.2(1)* of EN 1992-1-1.

## Reinforcement design

The value of the design compressive strength is defined as

$$f_{yd} = f_{yk}/\gamma_s$$

| | | |
|---|---|---|
| Where is: | $f_{yd}$ | • The design yield strength of reinforcement |
| | $f_{ck}$ | • The characteristic yield strength of reinforcement |
| | $\gamma_s$ | • The partial safety factor for reinforcement steel |

The required area of the main reinforcement is given by the formula:

$$A_{sl,req} = F_t/f_{yd}$$

The transverse tensile force for the calculation of vertical and horizontal stirrups is given by the chapter *6.5.3(3)*.

$$T = \frac{1}{4}\left(1 - 0.7\frac{a}{h}\right)F$$

Vertical and horizontal components of the tensile force are calculated using following formulas:

$$T_{vert} = T \cdot cos(\theta), T_{horz} = T \cdot sin(\theta)$$

Where is:

- $T_{vert}$ • The vertical component of the tensile force
- $T_{horz}$ • The horizontal component of the tensile force
- $T$ • The transverse force in the compression strut
- $\theta$ • The angle between the concrete compression strut and the beam axis perpendicular to the shear force

The vertical and horizontal components of the tensile force are increased by *20%* due to different direction of the force and reinforcement.

The minimum shear reinforcement for long corbels is calculated according to the chapter *6.2* of EN 1992-1-1

$$A_{sv,req} = \beta \cdot F_{Ed} / f_{yd}$$

Where is:

- $A_{sv,req}$ • The required area of shear reinforcement
- $\beta$ • The coefficient
- $F_{Ed}$ • The design value of the force in compression strut
- $f_{yd}$ • The design yield strength of reinforcement

This reinforcement should be placed in the middle three quarters of the distance between the clumn and slide plate.

Minimum vertical and horizontal reinforcement are given by the expression

$$A_{sv,req} = \frac{T_{vert}}{f_{y,d}}; A_{sh,req} = \frac{T_{hor\,z}}{f_{y,d}}$$

## Stress under slide plate

The stress under slide plate is verified in accordance with the chapter *6.7* of EN 1992-1-1:

$$F_{Rdu} = A_{c0} \cdot f_{cd} \cdot \sqrt{A_{c1}/A_{c0}} \leq 3.0 \cdot f_{cd} \cdot A_{c0}$$

Where is:

- $A_{c0}$ • The loaded area
- $A_{c1}$ • The maximum design distribution area with a similar shape to $A_{c0}$

The area $A_{c1}$ is calculated according to the chapter *6.7(3)* using the maximum angle of *26.56°* provided that design distribution area can't overlap the edge of corbel.

# National annexes

Following values of coefficients are used for the national annexes to EN 1992-1-1:

| **Coefficient** | **EN 1992-1-1** | **Czechia** | **Slovakia** | **Poland** | **Bulgaria** |
|---|---|---|---|---|---|
| **$\gamma_c$ - basic load combination** | 1.50 | 1.50 | 1.50 | 1.40 | 1.50 |
| **$\gamma_s$ - basic load combination** | 1.15 | 1.15 | 1.15 | 1.15 | 1.15 |
| **$\gamma_c$ - accidental load combination** | 1.20 | 1.20 | 1.20 | 1.20 | 1.20 |
| **$\gamma_s$ - accidental load combination** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |
| **$\gamma_{m,fi}$ - accidental load combination** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |
| **$\gamma_{cE}$ - modulus of elasticity** | 1.20 | 1.20 | 1.20 | 1.20 | 1.30 |
| **$\alpha_{cc}$ - concrete compressive strength** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |
| **$\alpha_{cc}$ - plain concrete compressive strength** | 0.80 | 0.80 | 0.80 | 0.80 | 0.80 |
| **$\alpha_{cc}$ - plain concrete tensile strength** | 0.80 | 0.70 | 0.80 | 0.80 | 0.80 |
| **$k_1$ - stress in joint CCC** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |
| **$k_2$ - stress in joint CCT** | 0.85 | 0.85 | 0.85 | 0.85 | 0.85 |
| **$k_3$ - stress in joint CTT** | 0.75 | 0.75 | 0.75 | 0.75 | 0.75 |

Following values of coefficients are used for the national annexes to EN 1992-2:

| **Coefficient** | **EN 1992-2** | **Czechia** | **Slovakia** | **Poland** | **Bulgaria** |
|---|---|---|---|---|---|
| **$\gamma_c$ - basic load combination** | 1.50 | 1.50 | 1.50 | 1.40 | 1.50 |
| **$\gamma_s$ - basic load combination** | 1.15 | 1.15 | 1.15 | 1.15 | 1.15 |
| **$\gamma_c$ - accidental load combination** | 1.20 | 1.20 | 1.20 | 1.20 | 1.20 |
| **$\gamma_s$ - accidental load combination** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |

| | | | | | |
|---|---|---|---|---|---|
| **$\gamma_{m,fi}$ - accidental load combination** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |
| **$\gamma_{cE}$ - modulus of elasticity** | 1.20 | 1.20 | 1.20 | 1.20 | 1.30 |
| **$\alpha_{cc}$ - concrete compressive strength** | 0.85 | 0.85 | 0.85 | 0.85 | 0.85 |
| **$\alpha_{cc}$ - plain concrete compressive strength** | 0.80 | 0.80 | 0.80 | 0.80 | 0.80 |
| **$\alpha_{cc}$ - plain concrete tensile strength** | 0.80 | 0.70 | 0.80 | 0.80 | 0.80 |
| **$k_1$ - stress in joint CCC** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |
| **$k_2$ - stress in joint CCT** | 0.85 | 0.85 | 0.85 | 0.85 | 0.85 |
| **$k_3$ - stress in joint CTT** | 0.75 | 0.75 | 0.75 | 0.75 | 0.75 |

# Steel

## Cross-sections

The program contains a pre-defined database of cross-sections, which contains a wide range of rolled cross-sections and 11 shapes with arbitrary dimensions. Nine of these cross-sections are walled shapes (cross-section segments have thickness much smaller than the length), two of them are solid shapes. These pre-defined shapes are automatically considered as welded ones. The hot-rolled cross-section, that isn't included in the database, may be entered with the help of these shapes. However, the correct analysis parameters (plastic or elastic resistance, buckling curve) have to be specified manually in these cases.

The cross-sections are defined in dextrorotary coordinate system, the positive direction of the axis $y$ is oriented to the left and the positive direction of the axis $z$ is oriented downward. The origin is in the centre of gravity

Following cross-sectional characteristics are used in the analysis:

| | |
|---|---|
| ***$b, h, t_1, t_2$ ...*** | • The dimensions *[mm]* |
| ***$A$*** | • The cross-sectional area *[$mm^2$]* |
| ***$I_y, I_z$*** | • The moments of inertia about axes $y$ and $z$ *[$mm^4$]* |
| ***$i_y, i_z$*** | • The radius of gyration for axes $y$ and $z$ [m] |
| ***$D_{yz}$*** | • The mixed moment of inertia *[$mm^4$]* (only $L$-shapes) |
| ***$I_\eta, I_\zeta$*** | • The moments of inertia about main axes $\eta$ and $\zeta$ *[$mm^4$]* (only $L$-shapes) |
| ***$i_\eta, i_\zeta$*** | • The radius of gyration for main axes $\eta$ and $\zeta$ *[m]* (only $L$-shapes) |
| ***$I_t$*** | • The rigidity moment in simple torsion *[$mm^4$]* |
| ***$I_\omega$*** | • The sectional moment of inertia *[$mm^6$]* |
| ***$y_t, z_t$*** | • The coordinates of the centre of gravity *[mm]* |
| ***$a_y, a_z$*** | • The coordinates of shear centre *[mm]* |
| ***$y_p, z_p$*** | • The coordinates of load position *[mm]* |
| ***$h_{pl}, b_{pl}$*** | • The levels of horizntal and vertical neutral axes for plastic resistance *[mm]* |
| ***$W_{pl,y}, W_{pl,z}$*** | • The plastic cross-sectional moduli *[$mm^3$]* |

The cross-sectional moduli are calculated including the effect of unsymmetry with the help of the mixed moment of inertia $D_{yz}$. Following expressions are used:

$$W_y = \frac{I_y I_z - D_{yz}^2}{z I_z - y D_{yz}}, W_z = \frac{I_y I_z - D_{yz}^2}{z D_{yz} - y I_y}$$

The rigidity moment in simple torsion is calculated using following expression for walled cross-sections:

$$I_t = \frac{1}{3} \sum b_i t_i^3$$

The simplified theory de Saint Venant is applied for solid cross-sections:

$$I_t = \frac{A^4}{40(I_y + I_z)}$$

The sectional moment of inertia is defined by expression:

$$I_\omega = \iint_F \omega^2 dF$$

| | | |
|---|---|---|
| Where is: | ***$F$*** | • The cross-sectional area |
| | ***$\omega$*** | • The sectional ordinate |

## Coordinate systems

Two coordinate systems are defined in the cross-section: the local coordinate system of member (axes *2*,*3*) and the coordinate system of cross-section (axes $y,z$). Input (buckling etc.) is done usually in the local coordinate system of member. The rotation of the cross-section is the rotation relatively to the local axes *2*,*3*. Forces are also defined in the local coordinate system. The analysis is performed in the coordinate system of cross-section. Load is automatically recalculated from the local coordinate system to the coordinate system of the cross-section.

# Material characteristics

The material can be selected from pre-defined database. the database contains a range of common strength classes. Both materials defined in Eurocode 3 and CSN are included.

Also user defined material may be specified in the software. Following characteristics are necessary for the analysis:

- $E$ • The modulus of elasticity
- $G$ • The shear modulus
- $f_y$ • The yield strength
- $f_u$ • The ultimate strength

# Perforation of cross-sections

The perforation may be defined for all types of cross-sections except solid ones and CHS profiles. Holes may be specified in all parts of the cross-section. The perforation is specified by the number of holes, their diameter, spacing and distance from edge. Input of holes is limited in the way, that it isn't possible to specify unsymmetrical holes for symmetrical cross-section (e.g. left and right parts of flange have to be perforated in the same way). The reduced cross-sectional characteristics are considered in the design. The change of the centre of gravity is ignored in the design.

The resistance of perforated cross-section may be higher than the resistance of the original cross-section, as the ultimate strength of steel $f_u$ is considered in the analysis. The analysis of the cross-section without holes is based on the yield strength $f_y$. The cross-sectional characteristics are calculated in a different way for certain types of loads. All unfilled holes are considered for the combination of axial force and bending moments. Filled holes are considered only in cross-sectional parts, that are in tension. All holes without any exception are considered in the shear analysis.

# Classification of cross-sections

The classification of cross-sections is based on the chapter *5.5* of EN 1993-1-1. The classification is done according to the geometrical rules and the type of loading. The plastic resistance is considered for classes *1* and *2*, elastic resistance for class *3*. The cross-sections in class *4* are reduced due to the effects of local buckling to the effective cross-section. The elastic resistance of the effective cross-section is considered in the analysis of these cross-sections.

The classification is performed for any particular wall of the cross-section separately, the worst one is selected as a final class for the whole cross-section. The tensile parts are automatically classified as the class *1*. This isn't formally in compliance, however, it doesn't effect results and it brings better continuity of the work in the software.

The classification is done for the combination of loading (axial force and bending moments). The bending moment about axis parallel to the particular wall is transferred into the increment of axial force in that way, that the final stress is identical to the stress of original bending moment. The plastic resistance is considered for classes *1* and *2*, the elastic resistance for classes *3* and *4* in this recalculation. Brief example of the classification of *I*-profile with a positive value of bending moment $M_y$: The increment of axial force in the upper flange d$N$ would have a negative value, as the positive value of bending moments causes tensile stress in the upper flange. The increment d$N$ would be positive for bottom flange, as tension occurs there.

The plastic resistance may be excluded manually in the design by the user. The design according to classes *3* or *4* is used in these cases. The type of analysis (elastic or plastic resistance) has to be specified manually for cross-sections created in the program "**Section**".

The rectangular cross-sections can't be classified according to EN 1993-1-1. Following rules are used:

- Cross-sections with slenderness smaller than *9* are classified as class *1*
- Cross-sections with slenderness smaller than *10* are classified as class *2*
- Cross-sections with slenderness greater than *10* are classified as class *3*

As it isn't possible to calculate an effective cross-section for rectangular cross-sections, the class *4* isn't included for these cross-sections. Therefore, local buckling of significantly slender rectangles is ignored.

The stress caused by the warping torsion is calculated with the help of elastic theory. Therefore, these cross-sections are automatically classified as classes *3* or *4*.

# Slenderness verification

The engineering practice it is known that very thin elements may cause problems. Limit values aren't specified in the standard, but it is convenient to keep the slenderness ratio within certain limits very often. Therefore, the program always calculates and reports the value of the slenderness ratio for any member. This value can be also compared with the specified limit.

The slenderness ratios in the directions of $y$ and $z$ axis are calculated using following formulas:

$$\lambda_y = \frac{l_{cr,y}}{i_y}, \lambda_z = \frac{l_{cr,z}}{i_z}$$

| | | |
|---|---|---|
| where is: | $l_{cr,y}$, $l_{cr,z}$ | • Buckling lengths corresponding to bending about the $y$ and $z$ axis |
| | $i_y$, $i_z$ | • Radius of gyration corresponding to bending about the $y$ and $z$ axis |

Buckling lengths $l_{cr,y}$ and $l_{cr,z}$ are determined using these rules:

- **members in compression** - buckling length for corresponding direction is used.
- **members in tension** - "**Sector length for buckling**" is used. If not specified, basic member length will be used.

The greater value of $\lambda_y$ and $\lambda_z$ will be used for slenderness verification.

## Analysis of general cross-sections

The general cross-section created in the program "**Section**" may be analysed for arbitrary combination of axial force ($N$), bending moments ($M_y$ and $M_z$) and shear forces ($V_y$ and $V_z$). The elastic or plastic resistance may be considered in the analysis. The effective cross-section for the analysis of the class *IV.* can't be found by the software.

The buckling effect may be considered in the analysis. The effect of lateral-torsional buckling can't be considered.

The shear areas for directions $y$ and $z$ have to be specified by the user. The automatic estimation in the software divides the total cross-sectional area equally into both directions.

## Shear resistance of solid cross-sections

The shear resistance for directions $z$ and $y$ is calculated using expressions

$$V_{pl,Rd,z} = \frac{A_{V,z}\frac{f_y}{\sqrt{3}}}{\gamma_{M0}}, V_{pl,Rd,y} = \frac{A_{V,y}\frac{f_y}{\sqrt{3}}}{\gamma_{M0}}$$

| | | |
|---|---|---|
| Where is: | $A_{V,z}$, $A_{V,y}$ | • The shear areas for directions $z$ and $y$ |
| | $f_y$ | • The yield strength |
| | $\gamma_{M0}$ | • The partial safety factor |

If the following expression is fulfilled for perforated cross-section,

$$A_{V,z,osl} < \frac{f_y}{f_u}A_{V,z}, A_{V,y,osl} < \frac{f_y}{f_u}A_{V,y}$$

| | | |
|---|---|---|
| Where is: | $A_{V,z,osl}$, $A_{V,y,osl}$ | • The shear areas of perforated cross-section for directions $z$ and $y$ |
| | $A_{V,z}$ , $A_{V,y}$ | • The shear areas for directions $z$ and $y$ |
| | $f_y$ | • The yield strength |
| | $f_u$ | • The ultimate strength |

the shear resistance of perforated cross-section is calculated using expression:

$$V_{pl,Rd,z,osl} = \frac{A_{V,z,osl}\frac{f_u}{\sqrt{3}}}{\gamma_{M0}}, V_{pl,Rd,y,osl} = \frac{A_{V,y,osl}\frac{f_u}{\sqrt{3}}}{\gamma_{M0}}$$

| | | |
|---|---|---|
| kde je: | $A_{V,z,osl}$, $A_{V,y,osl}$ | • The shear areas of perforated cross-section for directions $z$ and $y$ |
| | $f_y$ | • The yield strength |
| | $\gamma_{M0}$ | • sThe partial safety factor |

The minimum of $V_{pl,Rd}$ and $V_{pl,Rd,osl}$ is considered as the final shear resistance. If the corresponding plate of the cross-section are fixed on both ends in the perpendicular directions, the effect of local buckling is taken into consideration. The effect of the local buckling may be reduced with the help of web stiffeners. The simple post-critical method based on the chapter *5.6* is used for the buckling of walls. The local buckling is considered for slenderness greater than $69\varepsilon$, where $\varepsilon=(235/f_y)^{0.5}$. The shear resistance including the local buckling is given by the formula

$$V_{ba,Rd,z} = \frac{dt_w\tau_{ba}}{\gamma_{M1}}, V_{ba,Rd,y} = \frac{dt_w\tau_{ba}}{\gamma_{M1}}$$

| | | |
|---|---|---|
| Where is: | $d$ | • The height of plate |
| | $t_w$ | • The thickness of plate |
| | $\tau_{ba}$ | • The simple post-critical shear resistance |
| | $\gamma_{M1}$ | • The partial safety factor |

The design shear resistance $V_{Rd,z}$ is calculated as the minimum of $V_{pl,Rd,z}$ and $V_{ba,Rd,z}$, the design shear resistance $V_{Rd,y}$ is calculated as the minimum of $V_{pl,Rd,y}$ and $V_{ba,Rd,y}$. The buckling caused by shear isn't considered for plates supported only on one end.

## Shear stress due to torsion

The St. Venant and warping torsion is considered during the analysis. For St. Venant torsion, the shear stress $\tau_t$ is calculated for torsional moment $T_t$. Following expression is used for open cross-sections:

$$\tau_t = \frac{T_t}{I_t} t$$

| | | |
|---|---|---|
| Where is: | $T_t$ | • The torsional moment |
| | $I_t$ | • The rigidity moment in St.Venant torsion |
| | $t$ | • The thickness of the plate in the considered point |

The stress $\tau_t$ for closed cross-sections is given by the expression

$$\tau_t = \frac{T_t}{\Omega_t}$$

| | | |
|---|---|---|
| Where is: | $T_t$ | • The torsional moment |
| | $\Omega_t$ | • The area bounded by the centre lines of plates increased by a factor of two |
| | $t$ | • The thickness of the plate in the considered point |

For warping torsion, the shear stress $\tau_\omega$ is calculated for torsional moment $T_\omega$ using following formula:

$$\tau_\omega = \frac{T_\omega S_\omega}{I_\omega t}$$

| | | |
|---|---|---|
| Where is: | $T_\omega$ | • The torsional moment |
| | $S_\omega$ | • The warping constant in the considered point |
| | $I_\omega$ | • The sectional moment of inertia |
| | $t$ | • The thickness of the plate in the considered point |

The shear stress is verified according to the following expression:

$$\tau = \tau_t + \tau_\omega \leq \frac{f_y}{\sqrt{3}\gamma_{M0}}$$

| | | |
|---|---|---|
| Where is: | $f_y$ | • The yield strength |
| | $\gamma_{M0}$ | • The partial safety factor |

The values of shear resistance $V_{Rd,y}$ and $V_{Rd,z}$ are reduced due to torsion for combination of shear forces and torsional moments. The reduction is based on following expression

$$V_{T,Rd} = \left[\sqrt{1 - \frac{\tau_t}{f_y/\sqrt{3}/\gamma_{M0}}} - \frac{\tau_\omega}{f_y/\sqrt{3}/\gamma_{M0}}\right] V_{Rd}$$

for $I$- and $U$-profiles and

$$V_{T,Rd} = \left[1 - \frac{\tau_t}{f_y/\sqrt{3}/\gamma_{M0}}\right] V_{Rd}$$

for other cross-sections.

## Low and high shear

The shear verification affects bending and axial resistances according to EN 1993-1-1. The "**low shear**" is defined by the following condition:

$$V_{Sd} \leq 0.5 V_{pl,Rd}$$

| | | |
|---|---|---|
| Where is: | $V_{Sd}$ | • The shear force |
| | $V_{pl,Rd}$ | • The shear resistance |

Otherwise, "**High shear**" appears. If there are shear shear forces due to torsion in the cross-section, the value of resistance $V_{pl,T,Rd}$ (reduced value due to torsion, described in the chapter "**Shear stress due to torsion**") is used instead of the resistance $V_{pl,Rd}$. The design values of the axial and bending resistances are reduced for "**High shear**". The reduction is based on the reduction factor for yield strength according to the following formula

$$(1-\rho) f_y$$

| | | |
|---|---|---|
| Where is: | $\rho$ | • The reduction factor making an allowance for the presence of shear forces |
| | $f_y$ | • The yield strength |

The reduction factor $\rho$ is given by formula

$$\rho = \left( 2 \frac{V_{Sd}}{V_{pl,Rd}} - 1 \right)^2$$

or by the expression

$$\rho = \left( 2 \frac{V_{Sd}}{V_{pl,T,Rd}} - 1 \right)^2$$

for shear due to torsion

## Tensile resistance

The tensile resistance is calculated using the expression

$$N_{pl,Rd} = \frac{A f_y}{\gamma_{M0}}$$

| | | |
|---|---|---|
| Where is: | $A$ | • The cross-sectional area |
| | $f_y$ | • The yield strength |
| | $\gamma_{M0}$ | • The partial safety factor |

The tensile resistance of the cross-section with holes is calculated using the expression

$$N_{u,Rd} = 0.9 \frac{A_{osl} f_u}{\gamma_{M2}}$$

| | | |
|---|---|---|
| Where is: | $A_{osl}$ | • The cross-sectional area of perforated cross-section |
| | $f_u$ | • The ultimate strength |
| | $\gamma_{M2}$ | • The partial safety factor |

The design value of tensile resistance $N_{t,Rd}$ is chosen as the minimum of these two values. The design value of the resistance is reduced for "**High shear**" (described in the chapter "**Low and high shear**"). The reduction factor is given by the expression

$$\rho = \frac{(1-\rho_z) A_{V,z} + (1-\rho_y) A_{V,y}}{A}$$

| | | |
|---|---|---|
| Where is: | $\rho_z, \rho_y$ | • The reduction factor making an allowance for the presence of shear forces in directions $z$ and $y$ |
| | $A_{V,z}, A_{V,y}$ | • The shear areas for directions $z$ and $y$ |
| | $A$ | • The cross-sectional area |

## Compressive resistance

The compressive resistance for classes *1*, *2*, and *3* are defined by the expression

$$N_{c,Rd} = \frac{A f_y}{\gamma_{M0}}$$

Following expression is used for the class *4*

$$N_{c,Rd} = \frac{A_{eff} f_y}{\gamma_{M0}}$$

| | | |
|---|---|---|
| Where is: | $A$ | • The cross-sectional area |
| | $A_{eff}$ | • The cross-sectional area of the effective cross-section |
| | $f_y$ | • The yield strength |
| | $\gamma_{M0}$ | • The partial safety factor |

The compressive resistance of the perforated cross-section for classes *1*, *2*, and *3* are defined by the expression

$$N_{u,Rd} = 0.9 \frac{A_{osl} f_u}{\gamma_{M2}}$$

Following expression is used for the class *4*

$$N_{u,Rd} = 0.9 \frac{A_{eff,osl} f_u}{\gamma_{M2}}$$

| | | |
|---|---|---|
| Where is: | $A_{osl}$ | • The cross-sectional area of the perforated cross-section |
| | $A_{eff,osl}$ | • The cross-sectional area of the effective perforated cross-section |
| | $f_u$ | • The yield strength |
| | $\gamma_{M2}$ | • The partial safety factor |

The design value of compressive resistance $N_{c,Rd}$ is chosen as the minimum of these two values. The design value of the resistance is reduced for "**High shear**" (described in the chapter "**Low and high shear**"). The reduction factor is given by the expression

$$\rho = \frac{(1-\rho_z) A_{V,z} + (1-\rho_y) A_{V,y}}{A}$$

| | | |
|---|---|---|
| Where is: | $\rho_z, \rho_y$ | • The reduction factor making an allowance for the presence of shear forces in directions *z* and *y* |
| | $A_{V,z}, A_{V,y}$ | • The shear areas for directions *z* and *y* |
| | $A$ | • The cross-sectional area |

# Buckling resistance

The buckling resistance is given by the formula

$$N_{b,Rd} = \chi \frac{A f_y}{\gamma_{M1}}$$

| | | |
|---|---|---|
| Where is: | $\chi$ | • The reduction factor for flexural buckling |
| | $A$ | • The cross-sectional area |
| | $f_y$ | • The yield strength |
| | $\gamma_{M1}$ | • The partial safety factor |

The area of the effective cross-section is used for the class *4*. The value of $A_{osl}$ is used for perforated cross-sections. The buckling resistance is calculated for directions *y* and *z* or for the main axes directions *η* and *ζ* (L-cross-sections).

The values of slenderness $\lambda_z$ and $\lambda_y$ for buckling perpendicular to the axes *z* and *y* are given by expression

$$\lambda_z = \frac{L_{cr,z}}{i_z}, \lambda_y = \frac{L_{cr,y}}{i_y}$$

| | | |
|---|---|---|
| Where is: | $L_{cr,z}$, $L_{cr,y}$ | • The buckling lengths for buckling perpendicular to axes *z* and *y* |
| | $i_z$, $i_y$ | • The radius of gyration for axes *z* and *y* |

The values of relative slenderness $\overline{\lambda}_z$ and $\overline{\lambda}_y$ are given by the expressions

$$\overline{\lambda}_z = \frac{\lambda_z}{\lambda_1}\sqrt{\frac{A_{eff}}{A}}, \overline{\lambda}_y = \frac{\lambda_y}{\lambda_1}\sqrt{\frac{A_{eff}}{A}}$$

Where is:

| | | |
|---|---|---|
| $\lambda_z$, $\lambda_y$ | • | The slenderness corresponding to the axes $z$ and $y$ |
| $\lambda_1$ | • | The slenderness value to determine the relative slenderness |
| $A_{eff}$ | • | The cross-sectional area of the effective cross-section |
| $A$ | • | The cross-sectional area |

The slenderness value to determine the relative slenderness $\lambda_1$ is given by the formula

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

Where is:

| | | |
|---|---|---|
| $E$ | • | The modulus of elasticity |
| $f_y$ | • | The yield strength |

The value of imperfection factor $\alpha$ is selected according to the values $\overline{\lambda_z}$ and $\overline{\lambda_y}$ and shape of the cross-section. This factor represents one of the buckling curves $a_0$, $a$, $b$, $c$, $d$. The buckling curve may be also selected by the user. The reduction factors for buckling $\chi_z$ and $\chi_y$ are calculated using expressions

$$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \overline{\lambda}_z^2}}, \chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \overline{\lambda}_y^2}}$$

however, following expressions have to be fulfilled

$$\chi_z \leq 1.0, \chi_y \leq 1.0$$

where ϕ is calculated according to the follwoing expressions for directions $z$ and $y$.

$$\phi = \frac{1}{2}\left(1 + \alpha\left(\overline{\lambda}_z - 0.2\right) + \overline{\lambda}_z^2\right), \phi = \frac{1}{2}\left(1 + \alpha\left(\overline{\lambda}_y - 0.2\right) + \overline{\lambda}_y^2\right)$$

The buckling resistances $N_{b,Rd,z}$ and $N_{b,Rd,y}$ are calculated with the help of reduction factors $\chi_z$ and $\chi_y$:

$$N_{b,Rd,z} = \chi_z\frac{Af_y}{\gamma_{M1}}, N_{b,Rd,y} = \chi_y\frac{Af_y}{\gamma_{M1}}$$

The design value of the buckling resistance is reduced for "**High shear**" (described in the chapter "**Low and high shear**"). The reduction factor is given by the expression

$$\rho = \frac{(1-\rho_z)A_{V,z} + (1-\rho_y)A_{V,y}}{A}$$

Where is:

| | | |
|---|---|---|
| $\rho_z$,$\rho_y$ | • | The reduction factor making an allowance for the presence of shear forces in directions $z$ and $y$ |
| $A_{V,z}$,$A_{V,y}$ | • | The shear areas for directions $z$ and $y$ |
| $A$ | • | The cross-sectional area |

## Bending resistance

The bending resistance is calculated for classes *1* and *2* using following formula

$$M_{c,Rd,y} = \frac{W_{pl,y}f_y}{\gamma_{M0}}, M_{c,Rd,z} = \frac{W_{pl,z}f_y}{\gamma_{M0}}$$

Where is:

| | | |
|---|---|---|
| $W_{pl,y}$, $W_{pl,z}$ | • | The plastic section moduli for axes $y$ and $z$ |
| $f_y$ | • | The yield strength |
| $\gamma_{M0}$ | • | The partial safety factor |

The bending resistance for classes *3* and *4* is calculated in four points of the cross-section. These points are located in the corners of the cross-section. The following expression is used for the class *3*

$${}^iM_{c,Rd,y} = \frac{{}^iW_yf_y}{\gamma_{M0}}, {}^iM_{c,Rd,z} = \frac{{}^iW_zf_y}{\gamma_{M0}}$$

Where is:

| | | |
|---|---|---|
| ${}^iW_y$, ${}^iW_z$ | • | The elastic section moduli of cross-section about the axes y and $z$ in the $i$-point |
| $f_y$ | • | The yield strength |

$\gamma_{M0}$ • The partial safety factor

The following expression is used for the class *4*

$$^{i}M_{c,Rd,y} = \frac{^{i}W_{y,eff} f_y}{\gamma_{M0}}, {}^{i}M_{c,Rd,z} = \frac{^{i}W_{z,eff} f_y}{\gamma_{M0}}$$

Where is:
- $^{i}W_{eff,y}$, $^{i}W_{eff,z}$ • The elastic section moduli of effective cross-section about the axes y and *z* in the *i*-point
- $f_y$ • The yield strength
- $\gamma_{M0}$ • The partial safety factor

The bending resistance of perforated cross-section is calculated for classes *1* and *2* using following formula

$$M_{c,Rd,y,osl} = 0.9\frac{W_{pl,y,osl} f_u}{\gamma_{M2}}, M_{c,Rd,z,osl} = 0.9\frac{W_{pl,z,osl} f_u}{\gamma_{M2}}$$

Where is:
- $W_{pl,y,osl}$, $W_{pl,z,osl}$ • The plastic section moduli of perforated cross-section for axes *y* and *z*
- $f_u$ • The yield strength
- $\gamma_{M2}$ • The partial safety factor

The following expression is used for the class *3*

$$^{i}M_{c,Rd,y,osl} = \frac{^{i}W_{y,osl} f_y}{\gamma_{M2}}, {}^{i}M_{c,Rd,z,osl} = \frac{^{i}W_{z,osl} f_y}{\gamma_{M2}}$$

Where is:
- $^{i}W_{y,osl}$, $^{i}W_{z,osl}$ • The elastic section moduli of perforated cross-section about the axes y and *z* in the *i*-point
- $f_u$ • The yield strength
- $\gamma_{M2}$ • The partial safety factor

The following expression is used for the class *4*

$$^{i}M_{c,Rd,y,osl} = \frac{^{i}W_{y,eff,osl} f_y}{\gamma_{M2}}, {}^{i}M_{c,Rd,z,osl} = \frac{^{i}W_{z,eff,osl} f_y}{\gamma_{M2}}$$

Where is:
- $^{i}W_{y,eff,osl}$, $^{i}W_{z,eff,osl}$ • The elastic section moduli of effective perforated cross-section about the axes y and *z* in the *i*-point
- $f_u$ • The yield strength
- $\gamma_{M2}$ • The partial safety factor

The minimum of the values $M_{c,Rd,y}$ and $M_{c,Rd,y,osl}$ or $M_{c,Rd,z}$ and $M_{c,Rd,z,osl}$ is used in the verification.

The design value of the resistance is reduced for "**High shear**" (described in the chapter "**Low and high shear**").

$$M_{c,Rd,y,red} = \frac{W_{pl,y,red} f_y}{\gamma_{M0}}, M_{c,Rd,z,red} = \frac{W_{pl,z,red} f_y}{\gamma_{M0}}$$

Where is:
- $^{i}W_{pl,y,red}$, $^{i}W_{pl,z,red}$ • The reduced plastic section moduli for axes *y* and *z*
- $f_y$ • The yield strength
- $\gamma_{M0}$ • The partial safety factor

The reduced plastic section moduli are calculated as plastic section moduli with reduced capacity in walls that are subjected to "**high shear**". The reduction is based on the factors $\rho_z$ and $\rho_y$ that are calculated in accordance with the chapter "**Low and high shear**". The minimum of the values $M_{c,Rd,y}$ and $M_{c,Rd,y,red}$ or $M_{c,Rd,z}$ and $M_{c,Rd,z,red}$ is used in the verification.

## Bending resistance including the effect of lateral torsional buckling

ÚThe bending resistance is given by the formula

$$M_{b,Rd,y} = \chi_{LT,y} M_{c,RD,y}, M_{b,Rd,z} = \chi_{LT,z} M_{c,RD,z}$$

Where is:
- $\chi_{LT,y}$, $\chi_{LT,z}$ • The reduction factors for lateral-torsional buckling
- $M_{c,Rd,y}$, $M_{c,Rd,z}$ • The bending resistances

The reduction factors for lateral-torsional buckling $\chi_{LT,y}$ and $\chi_{LT,z}$ depends on the value of the elastic critical moment for lateral-torsional buckling $M_{cr}$, that is calculated according to the expression

$$M_{cr} = \mu_{cr}\frac{\pi\sqrt{EI_zGI_t}}{L}, M_{cr} = \mu_{cr}\frac{\pi\sqrt{EI_yGI_t}}{L}$$

where the factor $\mu_{cr}$ is the critical moment given by following expressions

$$\mu_{cr} = \frac{C_1}{k_z}\left[\sqrt{1+\kappa_{wt}^2+\left(C_2\zeta_g-C_3\zeta_j\right)^2}-\left(C_2\zeta_g-C_3\zeta_j\right)\right]$$

$$\mu_{cr} = \frac{C_1}{k_y}\left[\sqrt{1+\kappa_{wt}^2+\left(C_2\zeta_g-C_3\zeta_j\right)^2}-\left(C_2\zeta_g-C_3\zeta_j\right)\right]$$

The non dimensional torsion factor $k_{wt}$ is calculated using formula

$$\kappa_{wt} = \frac{\pi}{k_\omega L}\sqrt{\frac{EI_\omega}{GI_t}}$$

$\zeta_g$ is the non dimensional parameter considereing the load position relatively to the shear centre

$$\zeta_g = \frac{\pi z_g}{k_z L}\sqrt{\frac{EI_z}{GI_t}}, \zeta_g = \frac{\pi y_g}{k_y L}\sqrt{\frac{EI_y}{GI_t}}$$

$\zeta_j$ is the non dimensional parameter considereing the unsymmetry of cross-sections

$$\zeta_j = \frac{\pi z_j}{k_z L}\sqrt{\frac{EI_z}{GI_t}}, \zeta_j = \frac{\pi y_j}{k_y L}\sqrt{\frac{EI_y}{GI_t}}$$

| | | |
|---|---|---|
| Where is: | ***$C_1$, $C_2$, $C_3$*** | • The parameters considering the load and end contions |
| | ***$k_z$, $k_y$, $k_w$*** | • The buckling length factors |
| | ***E*** | • The modulus of elasticity |
| | ***G*** | • The shear modulus |
| | ***$I_z$, $I_y$*** | • The moments of inertia about axes *y* and *z* |
| | ***L*** | • The distance between two points where the lateral torsional buckling is prevented |
| | ***$I_\omega$*** | • The sectional moment of inertia |
| | ***$I_t$*** | • The rigidity moment in simple torsion |
| | ***$z_g$, $y_g$*** | • The horizontal and vertical distances of load position and shear centre |

and $z_j$ and $y_j$ are given by expressions

$$z_j = z_s - \frac{0.5\int_A\left(y^2+z^2\right)z\,\mathrm{d}A}{I_y}, y_j = y_s - \frac{0.5\int_A\left(y^2+z^2\right)y\,\mathrm{d}A}{I_z}$$

| | | |
|---|---|---|
| Where is: | ***$z_s$, $y_s$*** | • The horizontal and vertical distances of centre of gravity and shear centre |

The relative slenderness $\overline{\lambda}_{LT}$ is calculated with the help of the critical moment $M_{cr}$:

$$\overline{\lambda}_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}}, \overline{\lambda}_{LT} = \sqrt{\frac{W_z f_y}{M_{cr}}}$$

| | | |
|---|---|---|
| Where is: | ***$W_y$, $W_z$*** | • The section moduli of cross-section about the axes y and *z* |

The value of the imperfection factor $\alpha_{LT}$ is set according to the buckling curves *a* , *b* , *c* , *d* . The factors $\chi_{LT,y}$ and $\chi_{LT,z}$ are given by expressions

$$\chi_{LT,z} = \frac{1}{\phi_{LT}+\sqrt{\phi_{LT}^2-\overline{\lambda}_{LT}^2}}, \chi_{LT,y} = \frac{1}{\phi_{LT}+\sqrt{\phi_{LT}^2-\overline{\lambda}_{LT}^2}}$$

however, following condition has to be fulfilled

$$\chi_{LT,z} \leq 1.0, \chi_{LT,y} \leq 1.0$$

where

$$\phi_{LT} = \frac{1}{2}\left(1 + \alpha_{LT}\left(\overline{\lambda}_{LT} - 0.2\right) + \overline{\lambda}_{LT}^2\right)$$

The effect of lateral torsional buckling isn't considered for cross-sections resistant to LT buckling (e.g. RHS) and for cases, where the bending moment acts about the weak axis of the cross-section.

# Bimoment resistance

The stress caused by the bimoment is calculated with the help of elastic theory. Therefore, these cross-sections are automatically designed as classes *3* or *4*. The resistance is calculated in four points (corners of the cross-sections), where are the worst results expected. The resistance is calculated using following expression:

$$^{i}B_{Rd} = \frac{I_\omega f_y}{^{i}\omega \gamma_{M0}}$$

| | | |
|---|---|---|
| Where is: | $I_\omega$ | • The sectional moment of inertia |
| | $\omega$ | • The warping coordinate |
| | $f_y$ | • The yield strength |
| | $\gamma_{M0}$ | • The partial safety factor |

# Verification of shear resistance

The verification of shear resistance is performed in two directions *y* and *z* according to the following expressions:

$$|V_z| \leq V_{T,Rd,z}, |V_y| \leq V_{T,Rd,y}$$

| | | |
|---|---|---|
| Where is: | $V_z$, $V_y$ | • The shear forces |
| | $V_{t,RD,z}$, $V_{t,RD,y}$ | • The shear resistances calculated according to the chapters "**Shear resistance of solid cross-sections**" and "**Shear stress due to torsion**". |

# Verification of axial stress in solid cross-sections

The axial stress in the cross-section may be caused by axial force *N*, bending moments $M_y$, $M_z$ and bimoment *B*. The combination of these forces and moments is verified.

The verification without consideration of buckling and lateral torsional buckling is done according to the following expression for the classes *1* and *2*:

$$\frac{N}{N_{Rd}} + \frac{M_y}{M_{c,Rd,y}} + \frac{M_z}{M_{c,Rd,z}} \leq 1$$

The expression for the class *3*:

$$\frac{N}{N_{Rd}} + \frac{M_y}{M_{c,Rd,y}} + \frac{M_z}{M_{c,Rd,z}} + \frac{B}{B_{Rd}} \leq 1$$

The expression for the class *4*:

$$\frac{N}{N_{Rd}} + \frac{M_y + Ne_{N_y}}{M_{c,Rd,y}} + \frac{M_z + Ne_{N_z}}{M_{c,Rd,z}} + \frac{B}{B_{Rd}} \leq 1$$

| | | |
|---|---|---|
| Where is | $N_{\theta,Rd}$ | • The tensile resistance $N_{t,Rd}$ or resistance in plain compression $N_{c,Rd}$ |
| | $M_{c,Rd,y}$, $M_{c,Rd,y}$ | • The bending resistance about the axes *y* and *z* |
| | $B_{Rd}$ | • The resistance for stresses due to the bimoment |
| | $e_{Ny}$, $e_{Nz}$ | • The difference between centres of gravity for total and effective cross-sections in directions *z* and *y* |

If the shear verification fails for particular directions *y* and *z*, the value *1.0* is used for corresponding direction in equations described above.

# Verification of shear resistance for built-up cross-sections

## Verification of the shear force $V_z$

If the axis *z* is perpendicular to the strong axis of the cross-section (this is fulfilled for the most of cases), the shear resistance for the force $V_z$ is calculated in the same way as for solid cross-sections:

$$V_{Rd,z} = \frac{A_{V,z}\frac{f_y}{\sqrt{3}}}{\gamma_{M0}}$$

Where is:
- $A_{V,z}$ • The sum of shear areas of all partial cross-sections for the direction $z$
- $f_y$ • The yield strength
- $\gamma_{M0}$ • The partial safety factor

The final verification is done according to the following expression:

$$|V_z| \leq V_{Rd,z}$$

### Verification of shear resistance $V_y$

The force $V_y$ is usually parallel to the strong axis of the cross-section. It means, that the shear resistance depends on the stiffness of battens. This behaviour affects the bending resistance (chapter "**Verification of axial resistance for built-up cross-sections**"), buckling (the chapter "**Verification of buckling resistance of built-up cross-sections**") and the verification of battens and lacing (the chapter "**Verification of connection in built-up members**").

## Verification of axial resistance for built-up cross-sections

The resistance of the partial member in tension or in plain compression is calculated using the expression

$$N_{t,Rd} = \frac{Af_y}{\gamma_{M0}}$$

or

$$N_{c,Rd} = \frac{Af_y}{\gamma_{M0}}$$

Where is:
- $A$ • The cross-sectional area
- $f_y$ • The yield strength
- $\gamma_{M0}$ • The partial safety factor

The bending resistance for bending moment $M_y$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,Rd,y} = \frac{W_{pl,y}f_y}{\gamma_{M0}}$$

The formula for the class *3*:

$$M_{c,Rd,y} = \frac{W_y f_y}{\gamma_{M0}}$$

The formula for the class *4*:

$$M_{c,Rd,y} = \frac{W_{y,eff} f_y}{\gamma_{M1}}$$

Where is:
- $W_{pl,y}$ • The plastic section modulus of partial cross-section about the axis $y$
- $W_y$ • The elastic section modulus of partial cross-section about the axis $y$
- $W_{y,eff}$ • The effective section modulus of partial cross-section about the axis $y$

The bending moment $M_z$ is recalculated into the increment of axial force in partial cross-section $dN$. The recalculation uses following expression

$$\mathrm{d}N = \frac{M_z}{h_0}$$

The formula for members with battens:

$$\mathrm{d}N = 0.5 M_z h_0 \frac{A}{\frac{A}{2}h_0^2 + 2I_z}$$

Where is:
- $h_0$ • The distance of points of inertia of partial cross-sections
- $A$ • The area

| | | |
|---|---|---|
| | $I_z$ | • The second moment of inertia of partial cross-section |

The similar recalculation is used for the shear force $V_y$ in the direction of the strong axis. The shear force will be transferred into bending moment $M_z$ for partial cross-section. Following expression is used:

$$M_{z,Sd} = \frac{V_y l_1}{4}$$

| | | |
|---|---|---|
| Where is: | $l_1$ | • The distance of battens |

The bending resistance of partial cross-section for bending moment $M_z$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,Rd,z} = \frac{W_{pl,z} f_y}{\gamma_{M0}}$$

The formula for the class *3*:

$$M_{c,Rd,z} = \frac{W_z f_y}{\gamma_{M0}}$$

The formula for the class *3*:

$$M_{c,Rd,z} = \frac{W_{z,eff} f_y}{\gamma_{M1}}$$

| | | |
|---|---|---|
| Where is: | $W_{pl,z}$ | • The plastic section modulus of partial cross-section about the axis *z* |
| | $W_z$ | • The elastic section modulus of partial cross-section about the axis *z* |
| | $W_{z,eff}$ | • The effective section modulus of partial cross-section about the axis *z* |

The verification of the combination of axial force and bending moments is done according to the rules similar to the verification of solid cross-sections. This expression is used:

$$\frac{\frac{|N|}{n} + |\mathrm{d}N|}{N_{Rd}} + \frac{\frac{|M_y|}{n}}{M_{c,Rd,y}} + \frac{|M_{z,Sd}|}{M_{c,Rd,z}} \leq 1$$

| | | |
|---|---|---|
| Where is: | $n$ | • The number of partial cross-sections |
| | d$N$ | • The increment of axial force due to bending moment $M_z$ |
| | $M_{z,Sd}$ | • The bending moment in partial cross-section due to shear force $V_y$ |

## Verification of buckling resistance of built-up cross-sections

The buckling resistance perpendicular to the strong axis is given by expression

$$N_{b,Rd,y} = \chi \frac{A f_y}{\gamma_{M1}}$$

| | | |
|---|---|---|
| Where is: | $\chi$ | • The reduction factor for flexural buckling |
| | $A$ | • The cross-sectional area |
| | $f_y$ | • The yield strength |
| | $\gamma_{M1}$ | • The partial safety factor |

The effective cross-sectional area is considered for class *4*.

The slenderness $\lambda_y$ in the direction perpendicular to the strong axis y is given by formula

$$\lambda_y = \frac{L_{cr,y}}{i_y}$$

| | | |
|---|---|---|
| Where is: | $L_{cr,y}$ | • The buckling length for buckling perpendicular to the axis *y* |
| | $i_y$ | • The radius of gyration for axis *y* |

The relative slenderness $\overline{\lambda}_y$ is given by the expression

$$\overline{\lambda}_y = \frac{\lambda_y}{\lambda_1}\sqrt{\frac{A_{eff}}{A}}$$

| Where is: | | |
|---|---|---|
| | ***$\lambda_y$*** | • The slenderness in the direction perpendicular to the axis y |
| | ***$\lambda_1$*** | • The slenderness value to determine the relative slenderness |
| | ***$A_{eff}$*** | • The effective cross-sectional area |
| | ***$A$*** | • The cross-sectional area |

The slenderness value $\lambda_1$ is given by the formula

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

| Where is: | | |
|---|---|---|
| | ***$E$*** | • The modulus of elasticity |
| | ***$f_y$*** | • The yield strength |

The value of the imperfection factor $\alpha$ is set according to the buckling curves *a* , *b* , *c* , *d* . The factor $\chi_y$ corresponds to the relative slenderness $\overline{\lambda_y}$ and is calculated using expression

$$\chi_y = \frac{1}{\phi + \sqrt{\phi^2 - \overline{\lambda}_y^2}}$$

however, following condition has to be fulfilled

$$\chi_y \leq 1.0$$

where

$$\phi = \frac{1}{2}\left(1 + \alpha\left(\overline{\lambda}_y - 0.2\right) + \overline{\lambda}_y^2\right)$$

The partial cross-section fails if the specified axial force is greater than the resistance $N_{b,Rd,y}$.

The calculation of buckling resistance perpendicular to the weak axis follows. The elastic flexural buckling force $N_{cr}$ is given by the expression

$$N_{cr} = \pi^2\frac{EI_{eff}}{l_{cr,z}^2}$$

| Where is: | | |
|---|---|---|
| | ***$l_{cr,z}$*** | • The buckling length for buckling perpendicular to the axis *z* |
| | ***$k_{E,\theta}$*** | • The reduction factor for the slope of the linear elastic range |
| | ***$E$*** | • The modulus of elasticity |
| | ***$I_{eff}$*** | • The effective value of the moment of inertia, that depends on the type of connection of partial cross-sections |

Following formula is used for $I_{eff}$ for lacing

$$I_{eff} = \frac{A}{2}h_0^2$$

| Where is: | | |
|---|---|---|
| | ***$h_0$*** | • The distance of points of inertia of partial cross-sections |
| | ***$A$*** | • The cross-sectional area of partial cross-section |

The second moment of area $I_1$ is calculated for built-up cross-sections with battens using the expression

$$I_1 = \frac{A}{2}h_0^2 + 2I_z$$

| Where is: | | |
|---|---|---|
| | ***$A$*** | • The cross-sectional area of partial cross-section |
| | ***$h_0$*** | • The distance of points of inertia of partial cross-sections |
| | ***$I_z$*** | • The second moment of area of partial cross-section |

The radius of gyration $i_0$ is given by the expression

$$i_0 = \sqrt{\frac{I_1}{2A}}$$

For the slenderness

$$\lambda = \frac{l_{cr,z}}{i_0}$$

the factor $\mu$ is selected. The effective value of the moment of inertia $I_{eff}$ is given by the expression

$$I_{eff} = \frac{A}{2}h_0^2 + 2\mu I_z$$

The partial cross-section fails if the specified axial force is greater than the resistance $N_{cr}$.

The verification of the shear stiffness $S_V$ follows. The shear stiffness is given by the follwoing formula for battens

$$S_v = 2\pi^2 \frac{EI_z}{l_1^2}$$

or

$$S_v = \frac{24EI_z}{l_1^2\left(1 + \frac{2I_z}{rI_b} + \frac{h_0}{l_1}\right)}$$

However, following expression has to be fulfilled

$$S_v \leq 2\pi^2 \frac{EI_z}{l_1^2}$$

| | | |
|---|---|---|
| Where is: | ***$l_1$*** | • The distance of battens |
| | ***r*** | • The number of planes of lacings |
| | ***$I_b$*** | • The in plane second moment of area of one batten |
| | ***$h_0$*** | • The distance of points of inertia of partial cross-sections |

The axial force shouldn't exceed the shear stiffness $S_V$. Also following expression has to be fulfilled

$$\frac{|N|}{N_{cr}} + \frac{|N|}{S_v} > 1$$

The force in the middle of the batten is calculated using formula

$$N_{f,Sd} = 0.5\left(N + M_s h_0 \frac{A}{I_{eff}}\right)$$

The force in lacing is

$$N_{f,Sd} = 0.5N + \frac{M_s}{h_0}$$

Where the moment $M_S$ is given by the expression

$$M_s = \frac{N_{e_0}}{1 - \frac{|N|}{N_{cr}} - \frac{|N|}{S_v}}$$

| | | |
|---|---|---|
| Where is: | ***$e_0$*** | • The bow imperfection given by the expression $l_{cr,z}/500$ |

The buckling resistance is given by expression

$$N_{b,Rd} = \chi_z \frac{Af_y}{\gamma_{M1}}$$

| | | |
|---|---|---|
| Where is: | ***$\chi_y$*** | • The reduction factor for flexural buckling |
| | ***A*** | • The cross-sectional area |
| | ***$f_y$*** | • The yield strength |
| | ***$\gamma_{M1}$*** | • The partial safety factor |

where the factor $\chi_z$ corresponds to the slenderness $\lambda$, that is given by the expression

$$\lambda = \frac{l_1}{i_{min}}$$

Where is:

| | | |
|---|---|---|
| $l_1$ | • | The distance of battens |
| $i_{min}$ | • | The minimum radius of gyration for partial cross-section |

The relative slenderness $\overline{\lambda}_z$ is given by the formula

$$\overline{\lambda}_z = \frac{\lambda_z}{\lambda_1}\sqrt{\frac{A_{eff}}{A}}$$

where

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

The value of the imperfection factor $\alpha$ is set according to the buckling curves $a$, $b$, $c$, $d$. The factor $\chi_z$ corresponds to the relative slenderness $\overline{\lambda}_z$ and is calculated using expression

$$\chi_z = \frac{1}{\phi + \sqrt{\phi^2 - \overline{\lambda}_z^2}}$$

however, following condition has to be fulfilled

$$\chi_z \leq 1.0$$

where

$$\phi = \frac{1}{2}\left(1 + \alpha\left(\overline{\lambda}_z - 0.2\right) + \overline{\lambda}_z^2\right)$$

where

$$\alpha = 0.65\sqrt{\frac{235}{f_y}}$$

The shear force $V_S$ is calculated for the batten

$$V_S = \pi\frac{M_S}{l_{cr,z}}$$

Where the moment $M_S$ is given by the expression

$$M_{z,Sd} = \frac{(V_S + V_y)l_1}{4}$$

Where is:

| | | |
|---|---|---|
| $l_1$ | • | The distance of battens |
| $V_y$ | • | The entered shear force |

The bending resistance of partial cross-section for bending moment $M_y$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,Rd,y} = \frac{W_{pl,y}f_y}{\gamma_{M0}}$$

The formula for the class *3*:

$$M_{c,Rd,y} = \frac{W_y f_y}{\gamma_{M0}}$$

The formula for the class *4*:

$$M_{c,Rd,y} = \frac{W_{y,eff} f_y}{\gamma_{M1}}$$

| | | | |
|---|---|---|---|
| Where is: | $W_{pl,y}$ | • | The plastic section modulus of partial cross-section about the axis *y* |
| | $W_y$ | • | The elastic section modulus of partial cross-section about the axis *y* |
| | $W_{y,eff}$ | • | The effective section modulus of partial cross-section about the axis *y* |

The bending resistance of partial cross-section for bending moment $M_z$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,Rd,z} = \frac{W_{pl,z} f_y}{\gamma_{M0}}$$

The formula for the class *3*:

$$M_{c,Rd,z} = \frac{W_z f_y}{\gamma_{M0}}$$

The formula for the class *4*:

$$M_{c,Rd,z} = \frac{W_{z,eff} f_y}{\gamma_{M1}}$$

| | | | |
|---|---|---|---|
| Where is: | $W_{pl,z}$ | • | The plastic section modulus of partial cross-section about the axis *z* |
| | $W_z$ | • | The elastic section modulus of partial cross-section about the axis *z* |
| | $W_{z,eff}$ | • | The effective section modulus of partial cross-section about the axis *z* |

The verification is done for two points: the mid point of the distance between two battens and in the connection of batten.

The verification in the mid point of the distance between two battens:

$$\frac{N_{f,Sd} + \mathrm{d}N}{N_{b,Rd}} + \frac{k_y \frac{M_y}{n}}{M_{c,Rd,y}} \leq 1$$

| | | | |
|---|---|---|---|
| Where is: | $n$ | • | The number of partial cross-sections |
| | d$N$ | • | The increment of axial force due to bending moment $M_z$ |
| | $k_y$ | • | The factor calculated according to the rules for solid cross-sections |

The verification in the connection of batten:

$$\frac{\frac{N}{n} + \mathrm{d}N}{N_{b,Rd}} + \frac{k_y \frac{M_y}{n}}{M_{c,Rd,y}} + \frac{k_z M_{z,Sd}}{M_{c,Rd,z}} \leq 1$$

## Verification of connection in built-up members

The axial force in the lacing without buckling consideration is given by the following expression

$$N_{Sp} = \frac{V_y d}{r h_0}$$

| | | | |
|---|---|---|---|
| Where is: | $V_y$ | • | The entered shear force |
| | $r$ | • | The number of planes of lacings |
| | $d$ | • | The length of lacing |
| | $h_0$ | • | The distance of points of inertia of partial cross-sections |

The resistance of lacing is calculated using formula

$$N_{Rd,Sp} = \frac{A_d f_y}{\gamma_{M0}}$$

| | | | |
|---|---|---|---|
| Where is: | $A_d$ | • | The cross-sectional area of lacing |
| | $f_y$ | • | The yield strength |
| | $\gamma_{M0}$ | • | The partial safety factor |

Following expression has to be fulfilled

$$N_{Sp} \leq N_{Rd,Sp}$$

The axial force in the lacing including buckling consideration is given by the following expression

$$N_{Sp} = \frac{(V_s + |V_y|)\, d}{r h_0}$$

| Where is: | | | |
|---|---|---|---|
| | $V_y$ | • | The entered shear force |
| | $V_S$ | • | The shear force in the point of lacing |
| | $d$ | • | The length of lacing |
| | $r$ | • | The number of planes of lacings |
| | $h_0$ | • | The distance of points of inertia of partial cross-sections |

The slenderness of the web is estimated according to the following formula:

$$\lambda_{Sp} = \frac{d}{3.5\sqrt{A_d}}$$

| Where is: | | | |
|---|---|---|---|
| | $d$ | • | The web length |
| | $A_d$ | • | The cross-sectional area of web |

The relative slenderness $\overline{\lambda}_{Sp}$ is given by the formula

$$\overline{\lambda}_{Sp} = \frac{\lambda_{Sp}}{\lambda_1}$$

where is

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

The value of the imperfection factor $\alpha$, is set according to the buckling curve $c$. The factor $\chi_{Sp}$ corresponds to the relative slenderness $\overline{\lambda}_{Sp}$ and is calculated using expression

$$\chi_{Sp} = \frac{1}{\phi + \sqrt{\phi^2 - \overline{\lambda}_{Sp}^2}}$$

however, this expression has to be fulfilled

$$\chi_{Sp} \leq 1.0$$

where

$$\phi = \frac{1}{2}\left(1 + \alpha\left(\overline{\lambda}_{Sp} - 0.2\right) + \overline{\lambda}_{Sp}^2\right)$$

The buckling resistance of the web is given by the expression

$$N_{Rd,Sp} = \chi_{Sp}\frac{A_d f_y}{\gamma_{M1}}$$

The webs are OK if the following expression is fulfilled:

$$N_{Sp} \leq N_{Rd,Sp}$$

## National annexes

These partial factors are applied for design standard EN 1993-1-1 and its annexes:

| Factor | EN 1993-1-1 | Czechia | Slovakia | Poland | Bulgaria | Italy |
|---|---|---|---|---|---|---|
| **$\gamma_{M0}$ - Section capacity** | 1.00 | 1.00 | 1.00 | 1.00 | 1.05 | 1.05 |
| **$\gamma_{M1}$ - Stability verification** | 1.00 | 1.00 | 1.00 | 1.00 | 1.05 | 1.05 |
| **$\gamma_{M2}$ - Perforated sections** | 1.25 | 1.25 | 1.25 | 1.10 | 1.25 | 1.25 |

These partial factors are applied for design standard EN 1993-1-8 and its annexes:

| Factor | EN 1993-1-8 | Czechia | Slovakia | Poland | Bulgaria | Italy |
|---|---|---|---|---|---|---|
| **$\gamma_{M2}$ - Bolts, welds, plates** | 1.25 | 1.25 | 1.25 | 1.25 | 1.25 | 1.25 |
| **$\gamma_{M5}$ - Joints made of CHS, RHS** | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 | 1.00 |

# Steel Fire

## Basic principles of the fire resistance analysis

The methods of verification of the fire resistance are described in the standard EN 1993-1-2. The main result is the time of fire resistance of the structure, that is compared with the required time of the fire resistance. The required fire resistance is given in corresponding regulations and differs according to the type of the structure.

The analysis is based on the specified loading for the fire design situation. The resistance of the structure during fire situation is based on the rules for common design, however, the characteristics are affected by the increased temperature. During the analysis, the critical temperature is calculated for entered load. After that, the fire resistance period is calculated. This period is calculated as a time, when the critical temperature in steel will be reached. It depends on the selected temperature curve and specified fire detail.

## Loads for fire resistance analysis

The forces, that are used for the analysis of fire resistance, should be determined according to the rules for accidental combinations given in EN 1990. The combinations are given by expression

$$\sum \gamma_{GA} G_k + \psi_{1,1} Q_{k,1} + \sum \psi_{2,i} Q_{k,i} + \sum A_d(t)$$

Where is:

| | | |
|---|---|---|
| $G_k$ | • | The characteristic value of permanent load |
| $Q_{k,1}$ | • | The characteristic value of main variable load |
| $Q_{k,i}$ | • | The characteristic value of other variable loads |
| $A_{d(t)}$ | • | The design value of accidental load in fire situation, dependant on time $t$ |
| $\gamma_{GA}$ | • | The partial load factor for permanent loads in accidental combination |
| $\psi_{1,1}, \psi_{2,i}$ | • | The combination factors |

The load may be determined for time $t=0$ and consider these conditions as constant for the whole length of accidental situation.

## Calculation of resistance

The resistance of structures in fire situations is affected by the reduction of strength and deformation properties caused by increased temperature. These changes are described by reduction factors $k_{y,\theta}$ and $k_{E,\theta}$. $k_{y,\theta}$ is the reduction factor for the yield strength and $k_{E,\theta}$ is the reduction factor for the slope of the linear elastic range. Values are given in EN 1993-1-2, interval is between *1.0* for *20°C* and *0.0* for *1200°C*. Thus, the verification of steel elements is based on reduced values of yield strength and modulus of elasticity.

### Calculation of shear resistance

The shear resistance for directions *z* and *y* is calculated using expressions

$$V_{fi,\Theta,Rd,z} = \frac{A_{V,z}\frac{k_{y,\Theta} f_y}{\sqrt{3}}}{\gamma_{M,fi}}, V_{fi,\Theta,Rd,y} = \frac{A_{V,y}\frac{k_{z,\Theta} f_y}{\sqrt{3}}}{\gamma_{M,fi}}$$

Where is:

| | | |
|---|---|---|
| $A_{V,z}$, $A_{V,y}$ | • | The shear areas for directions *z* and *y* |
| $k_{y,\theta}$ | • | The reduction factor for the yield strength of steel |
| $f_y$ | • | The yield strength |
| $\gamma_{M,fi}$ | • | The partial safety factor for fire design situation |

If the corresponding plate of the cross-section are fixed on both ends in the perpendicular directions, the effect of local buckling is taken into consideration. The effect of the local buckling may be reduced with the help of web stiffeners. The simple post-critical method is used for the buckling of walls:

$$V_{fi,\Theta,Rd,z} = k_{y,\Theta}\frac{d t_w \tau_{ba}}{\gamma_{M,fi}}, V_{fi,\Theta,Rd,y} = k_{z,\Theta}\frac{d t_w \tau_{ba}}{\gamma_{M,fi}}$$

Where is:

| | | |
|---|---|---|
| $d$ | • | The height of plate |
| $t_w$ | • | The thickness of plate |
| $\tau_{ba}$ | • | The simple post-critical shear resistance |
| $\gamma_{M,fi}$ | • | The partial safety factor for fire design situation |

The design shear resistance $V_{fi,\theta,Rd,z}$ is calculated as the minimum of $V_{fi,\theta,Rd,z}$ and $V_{fi,\theta,ba,Rd,z}$, the design shear resistance $V_{fi,\theta,Rd,y}$ is calculated as the minimum of $V_{fi,\theta,Rd,y}$ and $V_{fi,\theta,ba,Rd,y}$. The buckling caused by shear isn't considered for plates supported only on one end.

## Shear due to torsion

The St. Venant and warping torsion is considered during the analysis. For St. Venant torsion, the shear stress $\tau_t$ is calculated for torsional moment $T_t$. Following expression is used for open cross-sections:

$$\tau_t = \frac{T_t}{I_t} t$$

| | | |
|---|---|---|
| Where is: | $T_t$ | • The torsional moment |
| | $I_t$ | • The rigidity moment in St.Venant torsion |
| | $t$ | • The thickness of the plate in the considered point |

The stress $\tau_t$ for closed cross-sections is given by the expression

$$\tau_t = \frac{T_t}{\Omega_t}$$

| | | |
|---|---|---|
| Where is: | $T_t$ | • The torsional moment |
| | $\Omega_t$ | • The area bounded by the centre lines of plates increased by a factor of two |
| | $t$ | • The thickness of the plate in the considered point |

For warping torsion, the shear stress $\tau_\omega$ is calculated for torsional moment $T_\omega$ using following formula:

$$\tau_\omega = \frac{T_\omega S_\omega}{I_\omega t}$$

| | | |
|---|---|---|
| Where is: | $T_\omega$ | • The torsional moment |
| | $S_\omega$ | • The warping constant in the considered point |
| | $I_\omega$ | • The sectional moment of inertia |
| | $t$ | • The thickness of the plate in the considered point |

The shear stress is verified according to the following expression:

$$\tau = \tau_t + \tau_\omega \leq \frac{k_{y,\Theta} f_y}{\sqrt{3}\gamma_{M,fi}}$$

| | | |
|---|---|---|
| Where is: | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |

The values of shear resistance $V_{Rd,y}$ and $V_{Rd,z}$ are reduced due to torsion for combination of shear forces and torsional moments. The reduction is based on following expression

$$V_{fi,\Theta,T,Rd} = \left[\sqrt{1 - \frac{\tau_t}{k_{y,\Theta} f_y/\sqrt{3}/\gamma_{M,fi}}} - \frac{\tau_\omega}{k_{y,\Theta} f_y/\sqrt{3}/\gamma_{M,fi}}\right] V_{fi,\Theta,Rd}$$

for $I$- and $U$-profiles and

$$V_{fi,\Theta,T,Rd} = \left[1 - \frac{\tau_t}{k_{y,\Theta} f_y/\sqrt{3}/\gamma_{M,fi}}\right] V_{fi,\Theta,Rd}$$

for other cross-sections.

## Calculation of tensile resistance

The tensile resistance is calculated using the expression

$$N_{fi,\Theta,T,Rd} = \frac{A k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

| | | |
|---|---|---|
| Where is: | $A$ | • The cross-sectional area |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |

## Calculation of compressive resistance

The compressive resistance is calculated using the expression

$$N_{c,fi,\Theta,T,Rd} = \frac{A k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

| | | |
|---|---|---|
| Where is: | ***A*** | • The cross-sectional area |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |

## Calculation of compressive resistance including buckling consideration

The compressive resistance is calculated using the expression

$$N_{b,fi,\Theta,T,Rd} = \chi_{fi} \frac{A k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

| | | |
|---|---|---|
| Where is: | $\chi_{fi}$ | • The reduction factor for flexural buckling in the fire design situation |
| | ***A*** | • The cross-sectional area |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |

The resistance including buckling consideration is calculated in directions $y$ and $z$ or in directions of main axes $\eta$ and $\zeta$. The reduction factor for flexural buckling $\chi_{fi}$ corresponds to the relative slenderness $\bar{\lambda}_\theta$ and is given by the expression

$$\chi_{fi} = \frac{1}{\varphi_\Theta + \sqrt{\varphi_\Theta^2 - \bar{\lambda}_\Theta^2}}$$

where

$$\varphi_\Theta = \frac{1}{2}\left(1 + \alpha\bar{\lambda}_\Theta + \bar{\lambda}_\Theta^2\right)$$

where

$$\alpha = 0.65\sqrt{\frac{235}{f_y}}$$

The relative slenderness $\bar{\lambda}$ for the temperature $\theta$ is given by the expression

$$\bar{\lambda}_\Theta = \bar{\lambda}\sqrt{k_{y,\Theta}/k_{E,\Theta}}$$

| | | |
|---|---|---|
| Where is: | $\bar{\lambda}$ | • The relative slenderness for temperature *20°C* |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $k_{E,\theta}$ | • The reduction factor for the slope of the linear elastic range |

The minimum of values for both directions is considered as the compressive resistance $N_{b,fi,\theta,Rd}$.

## Calculation of bending resistance

The flexural resistance is calculated using the expression

$$M_{c,fi,\Theta,Rd} = \frac{W k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

| | | |
|---|---|---|
| Where is: | ***W*** | • The plastic cross-sectional modulus |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |

| | | |
|---|---|---|
| | $\kappa_1$ | • The adaptation factor for non-uniform temperature across the cross-section |
| | $\kappa_2$ | • The adaptation factor for non-uniform temperature along the beam |

The bending resistance is calculated in four points (corners of the cross-sections) for cross-sections in classes 3 and 4.

## Effect of lateral-torsional buckling

The flexural resistance including an effect of lateral-torsional buckling is calculated using the expression

$$M_{b,fi,\Theta,Rd} = \chi_{LT,fi} M_{c,fi,\Theta,Rd}$$

| | | |
|---|---|---|
| Where is: | $\chi_{LT,fi}$ | • The reduction factor for lateral-torsional buckling for the fire design situation |
| | $M_{c,fi,\theta,Rd}$ | • The bending resistance calculated in accordance with previous chapter |

The reduction factor for lateral-torsional buckling $\chi_{LT,fi}$ corresponds to the relative slenderness $\bar{\lambda}_{LT,\theta}$ and is given by the expression

$$\chi_{LT,fi} = \frac{1}{\varphi_{LT,\Theta} + \sqrt{\varphi_{LT,\Theta}^2 - \bar{\lambda}_{LT,\Theta}^2}}$$

where

$$\varphi_{LT,\Theta} = \frac{1}{2}\left(1 + \alpha\bar{\lambda}_{LT,\Theta} + \bar{\lambda}_{LT,\Theta}^2\right)$$

where

The relative slenderness $\bar{\lambda}_{LT,\theta}$ for the temperature $\theta$ is given by the expression

$$\bar{\lambda}_{LT,\Theta} = \bar{\lambda}_{LT}\sqrt{k_{y,\Theta}/k_{E,\Theta}}$$

| | | |
|---|---|---|
| Where is: | $\bar{\lambda}_{LT,\theta}$ | • The relative slenderness for temperature *20°C* |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $k_{E,\theta}$ | • The reduction factor for the slope of the linear elastic range |

## Calculation of resistance for bimoment

The stress caused by the bimoment is calculated with the help of elastic theory. Therefore, these cross-sections are automatically designed as classes 3 or 4. The resistance is calculated in four points (corners of the cross-sections), where are the worst results expected. The resistance is calculated using following expression:

$$B_{fi,\Theta,Rd} = \frac{I_\omega k_{y,\Theta} f_y}{\omega \gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

| | | |
|---|---|---|
| Where is: | $I_\omega$ | • The sectional moment of inertia |
| | $\omega$ | • The warping coordinate |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |
| | $\kappa_1$ | • The adaptation factor for non-uniform temperature across the cross-section |
| | $\kappa_2$ | • The adaptation factor for non-uniform temperature along the beam |

# Verification of solid cross-sections

## Verification of shear resistance

The verification of the shear resistance is done in two directions ($y$ and $z$). Following expression is used:

$$|Q| \le V_{fi,\Theta,Rd}$$

| | | |
|---|---|---|
| Where is: | $Q$ | • The entered shear force |
| | $V_{fi,\theta,Rd}$ | • The shear resistance of the cross-section |

## Verification of axial stress in cross-section

The axial stress in the cross-section may be caused by axial force $N$, bending moments $M_y$, $M_z$ and bimoment $B$. The combination of these forces and moments is verified.

The verification without consideration of buckling and lateral torsional buckling is done according to the following expression:

$$\frac{N}{N_{fi,\Theta,Rd}}+\frac{M_y}{M_{c,fi,\Theta,Rd,y}}+\frac{M_z}{M_{c,fi,\Theta,Rd,z}}+\frac{B}{B_{fi,\Theta,Rd}}\leq 1$$

Where is

| | | |
|---|---|---|
| $N_{fi,\theta,Rd}$ | • | The tensile resistance $N_{t,fi,\theta,Rd}$ or resistance in plain compression $N_{c,fi,\theta,Rd}$ |
| $M_{c,fi,\theta,Rd,y}$ | • | The bending resistance about the axis $y$ |
| $M_{c,fi,\theta,Rd,z}$ | • | The bending resistance about the axis $z$ |
| $B_{fi,\theta,Rd}$ | • | The resistance for stresses due to the bimoment |

The verification including the consideration of buckling is done according to the following expression:

The formula for classes *1* and *2*:

$$\frac{N}{N_{b,fi,\Theta,Rd}}+\frac{k_y M_y}{M_{c,fi,\Theta,Rd,y}}+\frac{k_z M_z}{M_{c,fi,\Theta,Rd,z}}\leq 1$$

Where is:

| | | |
|---|---|---|
| $N_{b,fi,\theta,Rd}$ | • | The buckling resistance |
| $k_y$, $k_z$ | • | The interaction factors |

The factors $k_y$, $k_z$ are given by the expression

$$k_y=1-\frac{\mu_y N}{\chi_y A f_y}, k_z=1-\frac{\mu_z N}{\chi_z A f_y}$$

but

$$k_y\leq 1.5, k_z\leq 1.5$$

Where is: $\chi_y$, $\chi_z$ • The reduction factors for flexural buckling

and factors $\mu_y$, $\mu_z$ are given by equations

$$\mu_y=\overline{\lambda}_y\left(2\beta_{M_y}-4\right)+\frac{W_{pl,y}-W_y}{W_y}, \mu_z=\overline{\lambda}_z\left(2\beta_{M_z}-4\right)+\frac{W_{pl,z}-W_z}{W_z}$$

but

$$\mu_y\leq 0.9, \mu_z\leq 0.9$$

Where is:

| | | |
|---|---|---|
| $\overline{\lambda}_y$, $\overline{\lambda}_z$ | • | The relative slenderness |
| $\beta_{My}$, $\beta_{Mz}$ | • | The equivalent uniform moment factors |

The formula for the class *3*:

$$\frac{N}{N_{b,fi,\Theta,Rd}}+\frac{k_y M_y}{M_{c,fi,\Theta,Rd,y}}+\frac{k_z M_z}{M_{c,fi,\Theta,Rd,z}}+\frac{B}{B_{fi,\Theta,Rd}}\leq 1$$

Where is:

| | | |
|---|---|---|
| $N_{b,fi,\theta,Rd}$ | • | The buckling resistance |
| $k_y$, $k_z$ | • | The interaction factors |

The factors $k_y$, $k_z$ are given by the expressions

$$k_y=1-\frac{\mu_y N}{\chi_y A f_y}, k_z=1-\frac{\mu_z N}{\chi_z A f_y}$$

but

$$k_y\leq 1.5, k_z\leq 1.5$$

Where is: $\chi_y$, $\chi_z$ • The reduction factors for flexural buckling

and factors $\mu_y$, $\mu_z$ are given by equations

$$\mu_y=\overline{\lambda}_y\left(2\beta_{M_y}-4\right), \mu_z=\overline{\lambda}_z\left(2\beta_{M_z}-4\right)$$

but

$$\mu_y \leq 0.9, \mu_z \leq 0.9$$

Where is: **$\overline{\lambda}_y$, $\overline{\lambda}_z$** • The relative slenderness

**$\beta_{My}$, $\beta_{Mz}$** • The equivalent uniform moment factors

The formula for the class *4*:

$$\frac{N}{N_{b,fi,\Theta,Rd}} + \frac{k_y\left(M_y + Ne_{N_y}\right)}{M_{c,fi,\Theta,Rd,y}} + \frac{k_z\left(M_z + Ne_{N_z}\right)}{M_{c,fi,\Theta,Rd,z}} + \frac{B}{B_{fi,\Theta,Rd}} \leq 1$$

Where is: **$N_{b,fi,\theta,Rd}$** • The buckling resistance

**$k_y$, $k_z$** • The interaction factors

The factors $k_y$, $k_z$ are given by the expressions

$$k_y = 1 - \frac{\mu_y N}{\chi_y A_{eff} f_y}, k_z = 1 - \frac{\mu_z N}{\chi_z A_{eff} f_y}$$

but

$$k_y \leq 1.5, k_z \leq 1.5$$

Where is: **$\chi_y$, $\chi_z$** • The reduction factors for flexural buckling

and factors $\mu_y$, $\mu_z$ are given by equations

$$\mu_y = \overline{\lambda}_y\left(2\beta_{M_y} - 4\right), \mu_z = \overline{\lambda}_z\left(2\beta_{M_z} - 4\right)$$

but

$$\mu_y \leq 0.9, \mu_z \leq 0.9$$

Where is: **$\overline{\lambda}_y$, $\overline{\lambda}_z$** • The relative slenderness

**$\beta_{My}$, $\beta_{Mz}$** • The equivalent uniform moment factors

The verification including the consideration of buckling and lateral torsional buckling is done according to the following expression:

The formula for classes *1* and *2*:

$$\frac{N}{N_{b,fi,\Theta,Rd,z}} + \frac{k_{LT} M_y}{M_{b,fi,\Theta,Rd,y}} + \frac{k_z M_z}{M_{c,fi,\Theta,Rd,z}} \leq 1$$

and

$$\frac{N}{N_{b,fi,\Theta,Rd,y}} + \frac{k_y M_y}{M_{c,fi,\Theta,Rd,y}} + \frac{k_{LT} M_z}{M_{b,fi,\Theta,Rd,z}} \leq 1$$

The formula for the class *3*:

$$\frac{N}{N_{b,fi,\Theta,Rd,z}} + \frac{k_{LT} M_y}{M_{b,fi,\Theta,Rd,y}} + \frac{k_z M_z}{M_{c,fi,\Theta,Rd,z}} + \frac{B}{B_{fi,\Theta,Rd}} \leq 1$$

and

$$\frac{N}{N_{b,fi,\Theta,Rd,y}} + \frac{k_y M_y}{M_{c,fi,\Theta,Rd,y}} + \frac{k_{LT} M_z}{M_{b,fi,\Theta,Rd,z}} + \frac{B}{B_{fi,\Theta,Rd}} \leq 1$$

Where is: **$N_{fi,\theta,Rd,z}$** • The buckling resistance for buckling perpendicular to the axis *z*

**$N_{fi,\theta,Rd,y}$** • The buckling resistance for buckling perpendicular to the axis *y*

**$M_{c,fi,\theta,Rd,y}$** • The bending resistance about the axis *y*

**$M_{c,fi,\theta,Rd,z}$** • The bending resistance about the axis *z*

**$M_{b,fi,\theta,Rd,y}$** • The bending resistance about the axis *y* including the consideration of lateral torsional buckling

**$M_{b,fi,\theta,Rd,z}$** • The bending resistance about the axis *z* including the consideration of lateral torsional buckling

**$B_{fi,\theta,Rd}$** • The resistance for stresses due to the bimoment

The factor $k_{LT}$ is given by the expression

$$k_{LT} = 1 - \frac{\mu_{LT} N}{\chi_z A f_y}, k_{LT} = 1 - \frac{\mu_{LT} N}{\chi_y A f_y}$$

but

$$k_{LT} \leq 1$$

Where is: $\chi_y, \chi_z$ • The reduction factors for flexural buckling

and the factor $\mu_{LT}$ is given by the expression

$$\mu_{LT} = 0.15\overline{\lambda}_z \beta_{M,LT} - 0.15, \mu_{LT} = 0.15\overline{\lambda}_y \beta_{M,LT} - 0.15$$

but

$$\mu_{LT} \leq 0.9$$

Where is:
- $\overline{\lambda}_y$, $\overline{\lambda}_z$ • The relative slenderness
- $\beta_{M,LT}$ • The equivalent uniform moment factor

The formula for the class *4*:

$$\frac{N}{N_{b,fi,\Theta,Rd,z}} + \frac{k_{LT}\left(M_y + Ne_{N_y}\right)}{M_{b,fi,\Theta,Rd,y}} + \frac{k_z\left(M_z + Ne_{N_z}\right)}{M_{c,fi,\Theta,Rd,z}} + \frac{B}{B_{fi,\Theta,Rd}} \leq 1$$

and

$$\frac{N}{N_{b,fi,\Theta,Rd,y}} + \frac{k_y\left(M_y + Ne_{N_y}\right)}{M_{c,fi,\Theta,Rd,y}} + \frac{k_{LT}\left(M_z + Ne_{N_z}\right)}{M_{b,fi,\Theta,Rd,z}} + \frac{B}{B_{fi,\Theta,Rd}} \leq 1$$

Where is:
- $N_{fi,\theta,Rd,z}$ • The buckling resistance for buckling perpendicular to the axis *z*
- $N_{fi,\theta,Rd,y}$ • The buckling resistance for buckling perpendicular to the axis *y*
- $M_{c,fi,\theta,Rd,y}$ • The bending resistance about the axis *y*
- $M_{c,fi,\theta,Rd,z}$ • The bending resistance about the axis *z*
- $M_{b,fi,\theta,Rd,y}$ • The bending resistance about the axis *y* including the consideration of lateral torsional buckling
- $M_{b,fi,\theta,Rd,z}$ • The bending resistance about the axis *z* including the consideration of lateral torsional buckling
- $B_{fi,\theta,Rd}$ • The resistance for stresses due to the bimoment

The factor $k_{LT}$ is given by the expression

$$k_{LT} = 1 - \frac{\mu_{LT} N}{\chi_z A_{eff} f_y}, k_{LT} = 1 - \frac{\mu_{LT} N}{\chi_y A_{eff} f_y}$$

but

$$k_{LT} \leq 1$$

Where is: $\chi_y, \chi_z$ • The reduction factors for flexural buckling

and the factor $\mu_{LT}$ is given by the expression

$$\mu_{LT} = 0.15\overline{\lambda}_z \beta_{M,LT} - 0.15, \mu_{LT} = 0.15\overline{\lambda}_y \beta_{M,LT} - 0.15$$

but

$$\mu_{LT} \leq 0.9$$

Where is:
- $\overline{\lambda}_y$, $\overline{\lambda}_z$ • The relative slenderness
- $\beta_{M,LT}$ • The equivalent uniform moment factor

If the shear verification fails for particular directions *y* and *z*, the value *1.0* is used for corresponding direction in equations described above.

## Verification of built-up cross-sections

The verification of built-up cross-sections starts with the classification. The classification is identical to the process for solid cross-sections. The verification of built-up members is done according to the following rules.

## Verification of shear resistance $V_z$

If the axis $z$ is perpendicular to the strong axis of the cross-section (this is fulfilled for the most of cases), the shear resistance for the force $V_z$ is calculated in the same way as for solid cross-sections:

$$V_{fi,\Theta,Rd,z} = \frac{A_{V,z}\frac{k_{y,\Theta} f_y}{\sqrt{3}}}{\gamma_{M,fi}}$$

Where is:

- $A_{V,z}$ • The shear area for direction $z$
- $k_{y,\theta}$ • The reduction factor for the yield strength of steel
- $f_y$ • The yield strength
- $\gamma_{M,fi}$ • The partial safety factor for fire design situation

The final verification is done according to the following expression:

$$|Q| \leq V_{fi,\Theta,Rd,z}$$

## Verification of shear resistance $V_y$

The force $V_y$ is usually parallel to the strong axis of the cross-section. It means, that the shear resistance depends on the stiffness of battens.

## Verification of the resistance for tension, compression and bending

The resistance of the partial member in tension or in plain compression is calculated using the expression

$$N_{fi,\Theta,Rd} = \frac{A k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

Where is:

- $A_{V,z}$ • The area of partial cross-section
- $k_{y,\theta}$ • The reduction factor for the yield strength of steel
- $f_y$ • The yield strength
- $\gamma_{M,fi}$ • The partial safety factor for fire design situation

The bending resistance for bending moment $M_y$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,fi,\Theta,Rd,y} = \frac{W_{pl,y} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *3*:

$$M_{c,fi,\Theta,Rd,y} = \frac{W_y k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *4*:

$$M_{c,fi,\Theta,Rd,y} = \frac{W_{y,eff} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

Where is:

- $W_{pl,y}$ • The plastic section modulus of partial cross-section about the axis *y*
- $W_y$ • The elastic section modulus of partial cross-section about the axis *y*
- $W_{y,eff}$ • The effective section modulus of partial cross-section about the axis *y*

The bending moment $M_z$ is recalculated into the increment of axial force in partial cross-section $dN$. The recalculation uses following expression

$$\mathrm{d}N = \frac{M_z}{h_0}$$

The formula for members with battens:

$$\mathrm{d}N = 0.5 M_z h_0 \frac{A}{\frac{A}{2} h_0^2 + 2 I_z}$$

Where is: $h_0$ • The distance of points of inertia of partial cross-sections

$A$ • The area

$I_z$ • The second moment of inertia of partial cross-section

The similar recalculation is used for the shear force $V_y$ in the direction of the strong axis. The shear force will be transferred into bending moment $M_z$ for partial cross-section. Following expression is used:

$$M_{z,Sd} = \frac{V_y l_1}{4}$$

Where is: $l_1$ • The distance of battens

The bending resistance of partial cross-section for bending moment $M_z$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,fi,\Theta,Rd,z} = \frac{W_{pl,z} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *3*:

$$M_{c,fi,\Theta,Rd,z} = \frac{W_z k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *4*:

$$M_{c,fi,\Theta,Rd,z} = \frac{W_{z,eff} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

Where is: $W_{pl,z}$ • The plastic section modulus of partial cross-section about the axis *z*

$W_z$ • The elastic section modulus of partial cross-section about the axis *z*

$W_{z,eff}$ • The effective section modulus of partial cross-section about the axis *z*

The verification of the combination of axial force and bending moments is done according to the rules similar to the verification of solid cross-sections. This expression is used:

$$\frac{\frac{|N|}{n} + |\mathrm{d}N|}{N_{fi,\Theta,Rd}} + \frac{\frac{|M_y|}{n}}{M_{fi,\Theta,c,Rd,y}} + \frac{|M_{z,Sd}|}{M_{fi,\Theta,c,Rd,z}} \leq 1$$

Where is: $n$ • The number of partial cross-sections

d$N$ • The increment of axial force due to bending moment $M_z$

$M_{z,Sd}$ • The bending moment in partial cross-section due to shear force $V_y$

## Verification of buckling resistance

The buckling resistance perpendicular to the strong axis is given by expression

$$N_{b,fi,\Theta,Rd,y} = \chi_{fi,y} \frac{\beta_A A k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

Where is: $\chi_{fi,y}$ • The reduction factor for flexural buckling in the fire design situation

$A$ • The cross-sectional area

$k_{y,\theta}$ • The reduction factor for the yield strength of steel

$f_y$ • The yield strength

$\gamma_{M,fi}$ • The partial safety factor for fire design situation

$\beta_A$ • The factor considering the class of cross-section, $\beta_A=A_{eff}/A$ for class *4* and $\beta_A=1$ for other classes

The slenderness $\lambda_y$ in the direction perpendicular to the strong axis y is given by formula

$$\lambda_y = \frac{L_{cr,y}}{i_y}$$

Where is: **$L_{cr,y}$** • The buckling length for buckling perpendicular to the axis *y*

**$i_y$** • The radius of gyration for axis *y*

The relative slenderness $\overline{\lambda_y}$ is given by the expression

$$\overline{\lambda}_y = \frac{\lambda_y}{\lambda_1}\sqrt{\beta_A}\sqrt{k_{y,\Theta}/k_{E,\Theta}}$$

Where is: **$\lambda_y$** • The slenderness in the direction perpendicular to the axis y

**$\lambda_1$** • The slenderness value to determine the relative slenderness

**$\beta_A$** • The factor considering the class of cross-section, $\beta_A=A_{eff}/A$ for class *4* and $\beta_A=1$ for other classes

**$k_{y,\theta}$** • The reduction factor for the yield strength of steel

**$k_{E,\theta}$** • The reduction factor for the slope of the linear elastic range

The slenderness value $\lambda_1$ is given by the formula

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

Where is: **$E$** • The modulus of elasticity

**$f_y$** • The yield strength

The reduction factor $\chi_{fi,y}$ corresponds to the relative slenderness $\overline{\lambda_y}$ and is given by the expression

$$\chi_{fi,y} = \frac{1}{\varphi_\Theta + \sqrt{\varphi_\Theta^2 - \overline{\lambda}_\Theta^2}}$$

where

$$\varphi_\Theta = \frac{1}{2}\left(1 + \alpha\overline{\lambda}_y + \overline{\lambda}_y^2\right)$$

where

$$\alpha = 0.65\sqrt{\frac{235}{f_y}}$$

The partial cross-section fails if the specified axial force is greater than the resistance $N_{fi,\theta,b,Rd,y}$.

The calculation of buckling resistance perpendicular to the weak axis follows. The elastic flexural buckling force $N_{cr}$ is given by the expression

$$N_{cr} = \pi^2\frac{k_{E,\Theta}EI_{eff}}{l_{cr,z}^2}$$

Where is: **$l_{cr,z}$** • The buckling length for buckling perpendicular to the axis *z*

**$k_{E,\theta}$** • The reduction factor for the slope of the linear elastic range

**$E$** • The modulus of elasticity

**$I_{eff}$** • The effective value of the moment of inertia, that depends on the type of connection of partial cross-sections

Following formula is used for $I_{eff}$ for lacing

$$I_{eff} = \frac{A}{2}h_0^2$$

Where is: **$h_0$** • The distance of points of inertia of partial cross-sections

**$A$** • The cross-sectional area of partial cross-section

The second moment of area $I_1$ is calculated for built-up cross-sections with battens using the expression

$$I_1 = \frac{A}{2}h_0^2 + 2I_z$$

| | | |
|---|---|---|
| Where is: | ***A*** | • The cross-sectional area of partial cross-section |
| | ***h₀*** | • The distance of points of inertia of partial cross-sections |
| | ***I_z*** | • The second moment of area of partial cross-section |

The radius of gyration $i_0$ is given by the expression

$$i_0 = \sqrt{\frac{I_1}{2A}}$$

For the slenderness

$$\lambda = \frac{l_{cr,z}}{i_0}$$

the factor $\mu$ is selected. The effective value of the moment of inertia $I_{eff}$ is given by the expression

$$I_{eff} = \frac{A}{2}h_0^2 + 2\mu I_z$$

The partial cross-section fails if the specified axial force is greater than the resistance $N_{cr}$.

The verification of the shear stiffness $S_V$ follows. The shear stiffness is given by the follwoing formula for battens

$$S_v = 2\pi^2\frac{k_{E,\Theta}EI_z}{l_1^2}$$

or

$$S_v = \frac{24k_{E,\Theta}EI_z}{l_1^2\left(1 + \frac{2I_z}{rI_b} + \frac{h_0}{l_1}\right)}$$

However, following expression has to be fulfilled

$$S_v \leq 2\pi^2\frac{k_{E,\Theta}EI_z}{l_1^2}$$

| | | |
|---|---|---|
| Where is: | ***l₁*** | • The distance of battens |
| | ***r*** | • The number of planes of lacings |
| | ***I_b*** | • The in plane second moment of area of one batten |
| | ***h₀*** | • The distance of points of inertia of partial cross-sections |

The axial force shouldn't exceed the shear stiffness $S_V$. Also following expression has to be fulfilled

$$\frac{|N|}{N_{cr}} + \frac{|N|}{S_v} > 1$$

The force in the middle of the batten is calculated using formula

$$N_{f,Sd} = 0.5\left(N + M_s h_0\frac{A}{I_{eff}}\right)$$

The force in lacing is

$$N_{f,Sd} = 0.5N + \frac{M_s}{h_0}$$

Where the moment $M_S$ is given by the expression

$$M_s = \frac{N_{e_0}}{1 - \frac{|N|}{N_{cr}} - \frac{|N|}{S_v}}$$

| | | |
|---|---|---|
| Where is: | ***e₀*** | • The bow imperfection given by the expression $l_{cr,z}/500$ |

The buckling resistance is given by expression

$$N_{b,fi,\Theta,Rd} = \chi_z \frac{\beta_A A k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

| | | |
|---|---|---|
| Where is: | $\chi_y$ | • The reduction factor for flexural buckling |
| | $A$ | • The cross-sectional area |
| | $k_{y,\theta}$ | • The reduction factor for the yield strength of steel |
| | $f_y$ | • The yield strength |
| | $\gamma_{M,fi}$ | • The partial safety factor for fire design situation |
| | $\beta_A$ | • The factor considering the class of cross-section, $\beta_A=A_{eff}/A$ for class *4* and $\beta_A=1$ for other classes |

where the factor $\chi_z$ corresponds to the slenderness $\lambda$, that is given by the expression

$$\lambda = \frac{l_1}{i_{min}}$$

| | | |
|---|---|---|
| Where is: | $l_1$ | • The distance of battens |
| | $i_{min}$ | • The minimum radius of gyration for partial cross-section |

The relative slenderness $\overline{\lambda}_z$ is given by the formula

$$\overline{\lambda}_z = \frac{\lambda}{\lambda_1}\sqrt{\beta_A}\sqrt{k_{y,\Theta}/k_{E,\Theta}}$$

where

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

The factor $\chi_z$ corresponds to the relative slenderness $\overline{\lambda}_z$ and is calculated with the help of following expression

$$\chi_z = \frac{1}{\varphi_\Theta + \sqrt{\varphi_\Theta^2 - \overline{\lambda}_z^2}}$$

where

$$\varphi_\Theta = \frac{1}{2}\left(1 + \alpha\overline{\lambda}_z + \overline{\lambda}_z^2\right)$$

where

$$\alpha = 0.65\sqrt{\frac{235}{f_y}}$$

The shear force $V_S$ is calculated for the batten

$$V_S = \pi\frac{M_S}{l_{cr,z}}$$

The moment $M_{z,Sd}$ for the partial cross-section is given by the formula

$$M_{z,Sd} = \frac{(V_S + V_y)l_1}{4}$$

| | | |
|---|---|---|
| Where is: | $l_1$ | • The distance of battens |
| | $V_y$ | • The entered shear force |

The bending resistance of partial cross-section for bending moment $M_y$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,fi,\Theta,Rd,y} = \frac{W_{pl,y}k_{y,\Theta}f_y}{\gamma_{M,fi}} / \left(\kappa_1\kappa_2\right)$$

The formula for the class *3*:

$$M_{c,fi,\Theta,Rd,y} = \frac{W_y k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *4*:

$$M_{c,fi,\Theta,Rd,y} = \frac{W_{y,eff} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

Where is:

| | | |
|---|---|---|
| $W_{pl,y}$ | • | The plastic section modulus of partial cross-section about the axis *y* |
| $W_y$ | • | The elastic section modulus of partial cross-section about the axis *y* |
| $W_{y,eff}$ | • | The effective section modulus of partial cross-section about the axis *y* |

The bending resistance of partial cross-section for bending moment $M_z$ is calculated for the classes *1* and *2* according to the following formula:

$$M_{c,fi,\Theta,Rd,z} = \frac{W_{pl,z} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *3*:

$$M_{c,fi,\Theta,Rd,z} = \frac{W_z k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

The formula for the class *4*:

$$M_{c,fi,\Theta,Rd,z} = \frac{W_{z,eff} k_{y,\Theta} f_y}{\gamma_{M,fi}} / (\kappa_1 \kappa_2)$$

Where is:

| | | |
|---|---|---|
| $W_{pl,z}$ | • | The plastic section modulus of partial cross-section about the axis *z* |
| $W_z$ | • | The elastic section modulus of partial cross-section about the axis *z* |
| $W_{z,eff}$ | • | The effective section modulus of partial cross-section about the axis *z* |

The verification is done for two points: the mid point of the distance between two battens and in the connection of batten.

The verification in the mid point of the distance between two battens

$$\frac{N_{f,Sd} + \mathrm{d}N}{N_{fi,\Theta,b,Rd}} + \frac{k_y \frac{M_y}{n}}{M_{fi,\Theta,c,Rd,y}} \leq 1$$

Where is:

| | | |
|---|---|---|
| $n$ | • | The number of partial cross-sections |
| d$N$ | • | The increment of axial force due to bending moment $M_z$ |
| $k_y$ | • | The factor calculated according to the rules for solid cross-sections |

The verification in the connection of batten

$$\frac{\frac{N}{n} + \mathrm{d}N}{N_{fi,\Theta,b,Rd}} + \frac{k_y \frac{M_y}{n}}{M_{fi,\Theta,c,Rd,y}} + \frac{k_z M_{z,Sd}}{M_{fi,\Theta,c,Rd,z}} \leq 1$$

## Verification of lacing

The axial force in the lacing without buckling consideration is given by the following expression

$$N_{Sp} = \frac{V_y d}{r h_0}$$

Where is:

| | | |
|---|---|---|
| $V_y$ | • | The entered shear force |
| $r$ | • | The number of planes of lacings |
| $d$ | • | The length of lacing |
| $h_0$ | • | The distance of points of inertia of partial cross-sections |

The resistance of lacing is calculated using formula

$$N_{fi,\Theta,Rd,Sp} = \frac{A_d k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

Where is:
- ***$A_d$*** • The cross-sectional area of lacing
- ***$k_{y,\theta}$*** • The reduction factor for the yield strength of steel
- ***$f_y$*** • The yield strength
- ***$\gamma_{M,fi}$*** • The partial safety factor for fire design situation

Following expression has to be fulfilled

$$N_{Sp} \leq N_{fi,\Theta,Rd,Sp}$$

The axial force in the lacing including buckling consideration is given by the following expression

$$N_{Sp} = \frac{(V_s + |V_y|)\, d}{r h_0}$$

Where is:
- ***$V_y$*** • The entered shear force
- ***$V_S$*** • The shear force in the point of lacing
- ***$d$*** • The length of lacing
- ***$r$*** • The number of planes of lacings
- ***$h_0$*** • The distance of points of inertia of partial cross-sections

The slenderness of the web is estimated according to the following formula:

$$\lambda_{Sp} = \frac{d}{3.5\sqrt{A_d}}$$

Where is:
- ***$d$*** • The web length
- ***$A_d$*** • The cross-sectional area of web

The relative slenderness $\overline{\lambda_{Sp}}$ is given by the formula

$$\overline{\lambda}_{Sp} = \frac{\lambda_{Sp}}{\lambda_1}$$

where is

$$\lambda_1 = \pi\sqrt{\frac{E}{f_y}}$$

The factor $\chi_{Sp}$ corresponds to the relative slenderness $\overline{\lambda_{Sp}}$ and is calculated using expression

$$\chi_{Sp} = \frac{1}{\varphi_\Theta + \sqrt{\varphi_\Theta^2 - \overline{\lambda}_{Sp}^2}}$$

where

$$\varphi_\Theta = \frac{1}{2}\left(1 + \alpha\overline{\lambda}_{Sp} + \overline{\lambda}_{Sp}^2\right)$$

where

$$\alpha = 0.65\sqrt{\frac{235}{f_y}}$$

The buckling resistance of the web is given by the expression

$$N_{fi,\Theta,Rd,Sp} = \chi_{Sp}\frac{A_d k_{y,\Theta} f_y}{\gamma_{M,fi}}$$

The webs are OK if the following expression is fulfilled:

$$N_{Sp} \leq N_{fi,\Theta,Rd,Sp}$$

## Critical temperature

The critical temperature is calculated as the temperature, for which the utilization of the member is equal to 100%. The value of critical temperature is calculated using iteration procedures. If the member fails for the temperature *20°C*, this temperature is signed as a critical one and the calculation stops. As the temperature *350°C* is set as a maximum one for members, that belong to the class *IV*, according to the designing standard, this temperature is considered as a limiting value of the critical temperature for the class *IV*.

## Fire details

The fire details (style of fire protection) are divided into two basic categories: unprotected ones and protected ones. The sorting according to the number of exposed sides follows for both categories.

Unprotected cross-sections may be exposed to fire from all sides or only from three sides (in case that the cross-section is covered from one side e.g. by a slab). Cross-sections may be also protected by concrete slab partially (part of cross-section height is fixed with concrete). For such cases, protected height or exposed height has to be specified.

There are two general types of fire protection: coatings (the thickness $d_p$ has to be specified) and protected boxes (inputs are the thickness $d_p$ and box size). Protected details are also differentiated according to the number of exposed sides.

Materials of a fire protection may be divided into two main categories: coatings and board materials for protected boxes. The materials database contains wide range of items for both categories. Any other material may be specified manually with the help of user defined material characteristics. The density, the thermal conductivity and the specific heat capacity have to be specified in these cases.

## Temperature development

Following temperature curves are used for the description of the gas temperature according to EN 1991-1-2.

### Standard temperature curve

The fundamental nominal curve, the temperature of gas in the fire compartment is given by the expression

$$\Theta = 20 + 345 log_{10}(8t + 1)$$

| | | |
|---|---|---|
| Where is: | $\theta_g$ | • The gas temperature in the fire compartment in *°C* |
| | $t$ | • The time in minutes |

### External fire curve

The external fire curve is defined by the expression

$$\Theta_g = 660\left(1 - 0.687e^{-0.32t} - 0.313e^{-3.8t}\right) + 20$$

| | | |
|---|---|---|
| Where is" | $\theta_g$ | • The gas temperature in the fire compartment in *°C* |
| | $t$ | • The time in minutes |

The gas temperature is limited by the value *680°C*. Therefore, the critical temperature won't be achieved in certain cases.

### Hydrocarbon fire curve

The hydrocarbon curve is defined by the expression

$$\Theta_g = 1080\left(1 - 0.325e^{-0.167t} - 0.675e^{-2.5t}\right) + 20$$

| | | |
|---|---|---|
| Where is: | $\theta_g$ | • The gas temperature in the fire compartment in *°C* |
| | $t$ | • The time in minutes |

Also this curve is limited, the limiting value is *1100°C*.

### Parametric temperature curve

This curve is valid for fire compartments up to $500m^2$, without openings in the roof and for a maximum compartment height of *4m*. The curve consists of two phases: heating phase and cooling phase. Following parameters describe the geometry of the curve:

- The time $t_{lim}$ for maximum gas temperature in case of fuel controlled fire
- The thermal absorptivity for the total enclosure *b* given by the expression

$$b = \sqrt{\rho c \lambda}$$

| | | |
|---|---|---|
| Where is: | $\rho$ | • The density of boundary of enclosure in $kg/m^3$ |
| | $c$ | • The specific heat of boundary of enclosure in *J/(kg K)* |

| | | |
|---|---|---|
| | ***λ*** | • thermal conductivity of boundary of enclosure in *W/(m K)* |

- The opening factor *O* given by the expression

$$O = A_v \frac{\sqrt{h_{eq}}}{A_t}$$

| | | |
|---|---|---|
| Where is: | $A_v$ | • The total area of vertical openings on all walls in $m^2$ |
| | $h_{eq}$ | • The weighted average of window heights on all walls in *m* |
| | $A_t$ | • The total area of enclosure (walls, ceiling and floor, including openings) in $m^2$ |

- The design value of the fire load density $q_{t,d}$ related to the total surface area $A_t$ of the enclosure.

## Temperature development

The temperature development differs for protected and unprotected fire details.

Following expression is used for unprotected members:

$$\Delta\Theta_{a,t} = \frac{A_m/V}{c_a \rho_a} h_{net} \Delta t$$

| | | |
|---|---|---|
| Where is: | $\Delta\theta_{a,t}$ | • The increment of temperature in an unprotected steel member |
| | $A_m/V$ | • The section factor for unprotected steel members |
| | $c_a$ | • The specific heat of steel |
| | $\rho_a$ | • The unit mass of steel |
| | $h_{net}$ | • The design value of the net heat flux |
| | $\Delta t$ | • The time period |

The specific heat of steel differs according to the temperature in accordance with the chapter *3.4.1.2* of EN 1993-1-2. The net heat flux $h_{net}$ is given according to the chapter *3.1* of EN 1991-1-2. The temperature of steel member is checked with the help of temperature increments, the time period specified for the temperature increment $\Delta t$ is *5* seconds.

The temperature development for protected members is defined by following expression:

$$\Delta\Theta_{a,t} = \frac{\lambda_p A_p/V}{d_p c_a \rho_a} \frac{\Theta_{g,t} - \Theta_{a,t}}{1 + \phi/3} \Delta t - \left(e^{\phi/10} - 1\right) \Delta\Theta_{g,t}$$

where

$$\phi = \frac{c_p \rho_p}{c_a \rho_a} d_p A_p/V$$

| | | |
|---|---|---|
| Where is: | $\Delta\theta_{a,t}$ | • The increment of temperature in a protected steel member |
| | $\lambda_p$ | • The thermal conductivity of the fire protection system |
| | $A_p/V$ | • The section factor for steel members insulated by fire protection material |
| | $d_p$ | • The thickness of the fire protection material |
| | $c_a$ | • The specific heat of steel |
| | $\rho_a$ | • The unit mass of steel |
| | $\theta_{g,t}$ | • The ambient gas temperature at time *t* |
| | $\theta_{a,t}$ | • The steel temperature at time *t* |
| | $\Delta t$ | • The time period |
| | $\Delta\theta_{g,t}$ | • The increase of the ambient gas temperature during the time interval $\Delta t$ |
| | $c_p$ | • The temperature independent specific heat of the fire protection material |
| | $\rho_p$ | • The unit mass of the fire protection material |

The specific heat of steel differs according to the temperature in accordance with the chapter *3.4.1.2* of EN 1993-1-2. The temperature of steel member is checked with the help of temperature increments, the time period specified for the temperature increment $\Delta t$ is *30* seconds.

The critical time is given as a sum of time increments $\Delta t$, that reach the critical temperature. The critical temperature is calculated in advance with the help of utilization of the member.

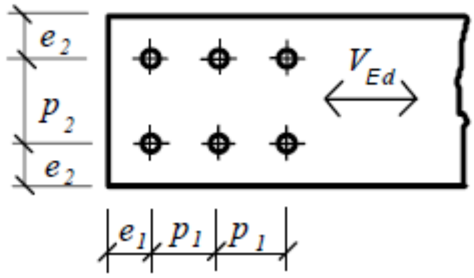
 

# Steel Connections

## Bolts

### Spacings

Following values are given in the standard as minimum and maximum spacings for bolts in shear:




*Spacing*

$$1.2d_0 \leq e_1 \leq max\,(12t, 150mm)$$
$$1.2d_0 \leq e_2 \leq max\,(12t, 150mm)$$
$$2.2d_0 \leq p_1 \leq max\,(14t, 150mm)$$
$$3.0d_0 \leq p_2 \leq max\,(14t, 150mm)$$

Where is:

| | | |
|---|---|---|
| $t$ | • | The minimum material thickness |
| $d_0$ | • | The hole diameter |
| $e_1$, $e_2$, $p_1$, $p_2$ | • | The distance given in the figure above |

Following values are recommended:

| **Bolts** | **Recommended spacing (mm)** | | |
|---|---|---|---|
| | $p$ ; $p_2$ | $e$ | $e$ |
| M12 | 44 | 30 | 25 |
| M16 | 55 | 40 | 30 |
| M20 | 70 | 50 | 40 |
| M24 | 80 | 60 | 50 |
| M27 | 90 | 70 | 55 |
| M30 | 100 | 75 | 60 |
| M36 | 120 | 90 | 70 |

### Shear resistance

The design value of shear resistance of *n* bolts in unthreaded portion is given by the expression:

$$F_{v,Rd} = n\frac{0.6 f_{ub} A_s}{\gamma_{M2}}$$

The resistance in threaded portion is calculated for bolt materials *4.6*, *5.6*, *8.8* using expression

$$F_{v,Rd} = n\frac{0.6 f_{ub} A_s}{\gamma_{M2}}$$

and for material *4.8*, *5.8*, *10.9* with the help of this formula:

$$F_{v,Rd} = n\frac{0.5 f_{ub} A_s}{\gamma_{M2}}$$

Where is:

| | | |
|---|---|---|
| $F_{v.Rd}$ | • | The shear resistance |
| $f_{ub}$ | • | The ultimate tensile strength |
| $A_s$ | • | The tensile stress area of bolt |

The program calculates only the shear resistance in threaded portion of the bolt. The shear resistance of the unthreaded portion is not considered in the analysis

## Bearing resistance

The bearing resistance in vertical direction is calculated for the minimum of coefficient *α*

$$\alpha = \frac{e_1}{3d_0}; \alpha = \frac{p_1}{3d_0} - \frac{1}{4}; \alpha = 1.0$$

and for minimum value of $k_1$

$$k_1 = 2.8\frac{e_2}{d_0} - 1.7; k_1 = 1.4\frac{p_2}{d_0} - 1.7; k_1 = 2.5$$

For the horizontal direction, the minimum value of α is selected from these formulas

$$\alpha = \frac{e_2}{3d_0}; \alpha = \frac{p_2}{3d_0} - \frac{1}{4}; \alpha = \frac{f_{ub}}{f_{u,p}}; \alpha = 1.0$$

the minimum value of $k_1$ is selected from following expressions for horizontal direction

$$k_1 = 2.8\frac{e_1}{d_0} - 1.7; k_1 = 1.4\frac{p_1}{d_0} - 1.7; k_1 = 2.5$$

The resistance is given by the formula

$$F_{b,Rd} = n\frac{k_1 \alpha f_u d t}{\gamma_{M2}}$$

| | | |
|---|---|---|
| Where is: | $F_{b.Rd}$ | • The design value of bearing resistance |
| | $d_0$ | • The hole diameter |
| | $e_1$, $e_2$, $p_1$, $p_2$ | • The spacing according to the figure above |
| | $d$ | • The bolt diameter |
| | $t$ | • The material thickness |

## Slip resistance

The preloading force is calculated using the expression *(3.7)*:

$$F_{p,C} = 0.7 f_{ub} A_s$$

The design value of slip resistance is given by the formula *(3.6)*

$$F_{s,Rd} = \frac{k_s n_{ub} \mu}{\gamma_{M3}} F_{p,C}$$

The connections may be preloaded only in serviceability limit state. In that cases, shear and bearing resistance of bolts are verified for ultimate limit state. The software considers the preloading in the ultimate limit state. In such cases, only bearing resistance is checked.

Following values of slip factor *μ* are used in the software (based on the table *3.7*):

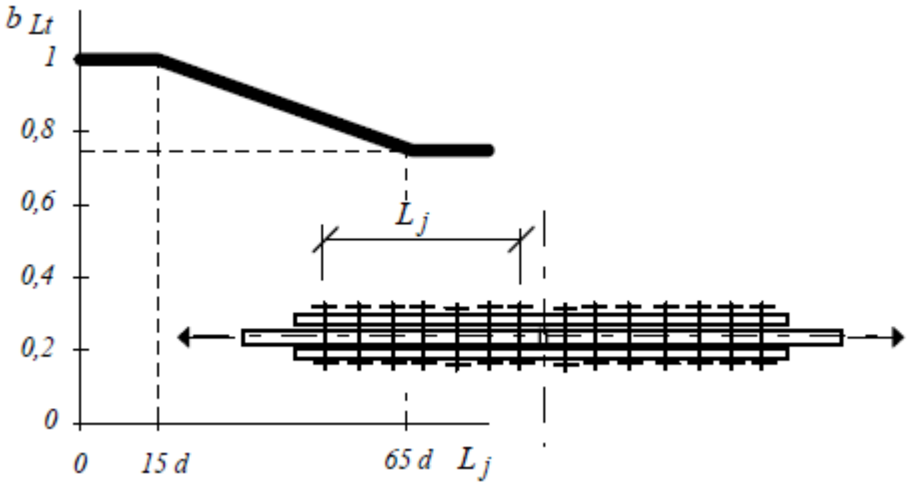
| Class | Description | *μ* |
|---|---|---|
| A | Blasted with shot or grit | 0.5 |
| B | Blasted with shot or grit (spray-metallised with aluminium or zinc based product or with alkali-zinc silicate paint with a given thickness) | 0.4 |
| C | Cleaned by wire brush or flame cleaning | 0.3 |
| D | Without any treatment | 0.2 |

## Long joints

The shear resistance $F_{v.Rd}$ in long joints, where the distance between the centres of the end fasteners is more than *15d*, is reduced with the help of the coefficient

$$\beta_{Lf} = 1 - \frac{L_j - 15d}{200d}; \beta_{Lf} \leq 1.0; \beta_{Lf} \geq 0.75$$

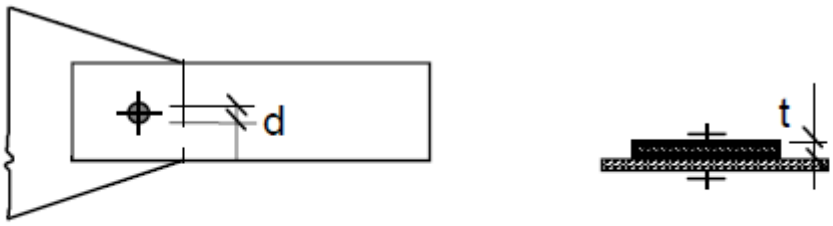
Where $L_j$ is the distance between the centres of the end fasteners.



*The resistance reduction for long joints*

The design resistance of single lap joint with only one bolt row is limited according the chapter *3.6.1(10)* using expression

$$F_{b,Rd} = \frac{1.5 f_u d t}{\gamma_{M2}}$$



*Single lap joint with only one bolt row*

## Angles connected by one leg

The resistance of single angle in tension, that is connected by a row of bolts in one leg, is calculated in accordance with the chapter *3.10.3* using the following expressions

joints with one bolt

$$N_{u,Rd} = \frac{2.0\,(e_2 - 0.5 d_0)\, t f_u}{\gamma_{M2}}$$

joints with 2 bolts

$$N_{u,Rd} = \frac{\beta_2 A_{net} f_u}{\gamma_{M2}}$$

joints with 3 or more bolts

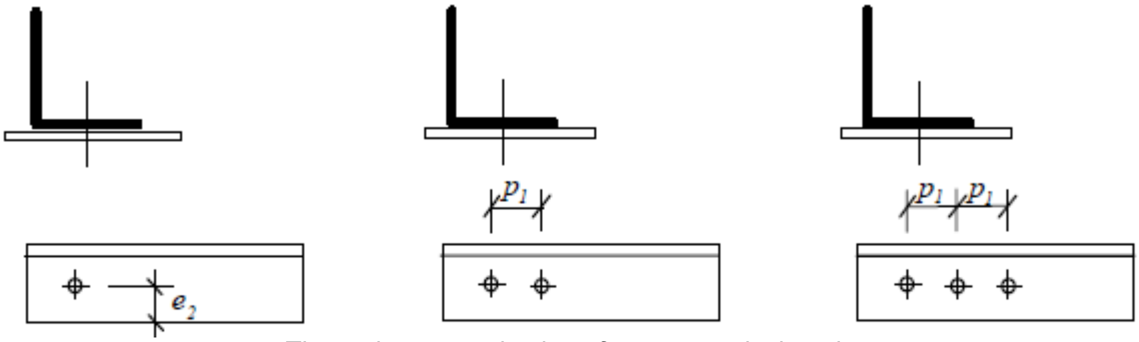
$$N_{u,Rd} = \frac{\beta_3 A_{net} f_u}{\gamma_{M2}}$$

Where is:
- $\gamma_{M2}$ • The partial safety factor of net section
- $A_{net}$ • The net area of angle
- $\beta_2$, $\beta_3$ • The reduction factors according to the chapter 3.10.3

The values of reduction factors $\beta_2$, $\beta_3$ are based on the table *3.8* of EN 1993-1-8:

| **Pitch** | **≤2,5$d_0$** | **≥2,5$d_0$** |
|---|---|---|
| Two bolts | 0,4 | 0,7 |
| Three bolts and more | 0,5 | 0,7 |




*The resistance reduction of unsymmetrical angle*

All angles in the software are analysed as unsymmetrical.

## Welds

Resistance of fillet weld is calculated using expression

$$F_{w,Rd} = \frac{f_u a_{we} 2 L_{we}}{\beta_w \gamma_{M2} \sqrt{2}}$$

Where is: ***L**$_{we}$* • The weld length

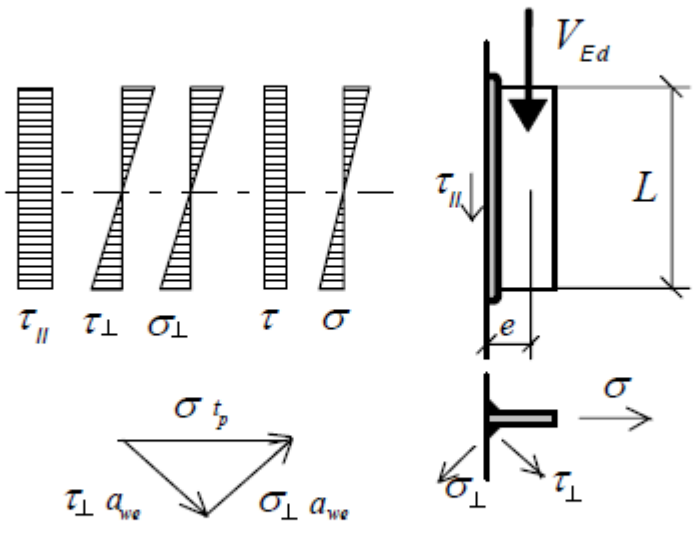
The particular stresses are calculated for fillet weld:

$$\sigma = \frac{V_{Ed} e}{\frac{t_p L^2}{6}}$$

$$\tau_{\perp} = \sigma_{\perp} = \frac{t_p \sigma}{2 a_{we} \sqrt{2}}$$

$$\tau_{||} = \frac{V_{Ed}}{2 a_{we} L}$$

The diagram of particular components




*Stresses in fillet weld*

Following expressions have to be satisfied in accordance with chapter *4.5.3.2*

$$\sqrt{\sigma_{\perp}^2 + 3\left(\tau_{\perp}^2 + \tau_{||}^2\right)} \leq \frac{f_u}{\beta_w \gamma_{M2}}$$

and

$$\sigma_{\perp} \leq 0.9 \frac{f_u}{\gamma_{M2}}$$

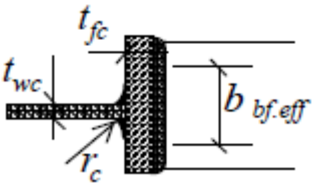
 

The joint of I-profile made of fillet welds is analysed using the plastic distribution of stresses. The web fillets transfer shear forces and flange welds transfer bending moments. This procedure may provide insufficient results for certain cases (e.g. beams with haunches). The elastic distribution may provide better resistance in these cases.

### Connections to unstiffened flanges

The resistance of weld in connection to unstiffened flange is reduced according to the chapter *4.10* with the help of the effective widths of an unstiffened T-joint. The effective width is calculated using formula

$$b_{bf,eff} = min(b_{fb}; t_{wc} + 2r_c + 7t_{fc})$$




*Effective width for unstiffened flange*

### Bending resistance

The welds are designed for increased value of design moments according to the chapter *6.2.3(5)*. The design moment is calculated using following expressions:

- **Braced frames** - $1.4M_{j.rd}$
- **Unbraced frames** - $1.7M_{j.rd}$

## Plate or beam web

The general assumption is, that the joint plate is partially fixed by connected members. Therefore, joint plates are verified according to the rules for the class *3* (elastic analysis without local buckling consideration). This assumption has to be respected during the design of joint.

### Shear

The design shear resistance is calculated using expression

$$V_{pl,Rd} = \frac{A_v f_y}{\sqrt{3}\gamma_{M0}}$$

| | | |
|---|---|---|
| Where is: | $A_v$ | • The area subjected to shear |

The holes are not considered if

$$\frac{A_{v,net}}{A_v} > \frac{f_y}{f_u}$$

Otherwise, the shear resistance is calculated using expression

$$V_{pl,Rd} = \frac{A_{v,net} f_u}{\sqrt{3}\gamma_{M3}}$$

| | | |
|---|---|---|
| Where is: | $A_{v,net}$ | • The net area subjected to shear |

### Beam web in shear

Beam web is verified with the help of the tearing effective length

$$L_1 = a_1 \leq 5d$$

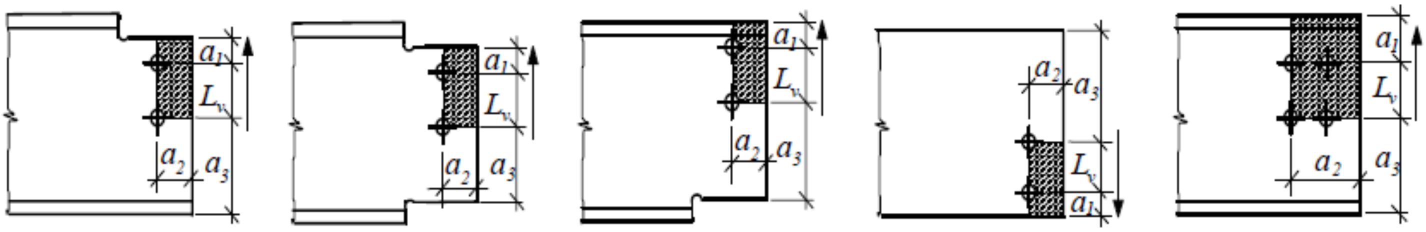
$$L_2 = (a_2 - 0.5d_0)\,\frac{f_u}{f_y}$$

$$L_3 = L_v + a_1 + a_3 \leq (L_v + a_1 + a_3 - nd_0)\,\frac{f_u}{f_y}$$

$$L_{v,eff} = L_v + L_1 + L_2$$

using expression

$$V_{u,Rd} = \frac{A_{v,eff} f_y}{\sqrt{3}\gamma_{M0}} = \frac{t_w L_{v,eff} f_y}{\sqrt{3}\gamma_{M0}}$$



*Beam web subjected to shear*

## Plate in bending

The design bending resistance is calculated according to the chapter *6.2.5* of EN 1993-1-1 (class *3*) using expression

$$M_{el,Rd} = W \frac{f_y}{\gamma_{M0}}$$

Holes are considered if

$$0.9 \frac{A_{v,net}}{A_v} n > \frac{f_y \gamma_{M2}}{f_u \gamma_{M0}}$$

## Combination of shear and bending

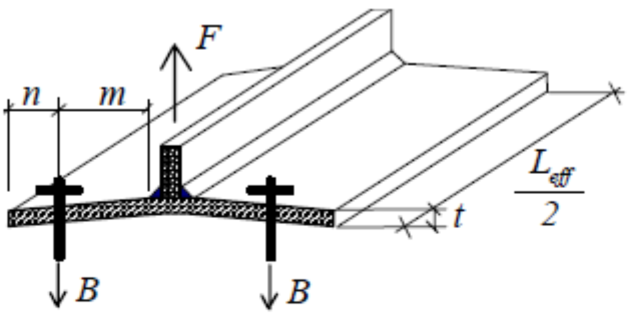
If the shear force is smaller than the half of the design shear resistance, the combination of shear and bending is not considered in accordance with the chapter *6.2.8* of EN 1993-1-1. Otherwise, the combination of shear and bending is checked.

The joint plate in joints with angles can be subjected to complicated stresses. In these cases, only horizontal section along the weld is checked. This section is checked for the combination of tension, bending and shear. The sum of forces is checked for the joints with more connected members.

# End plate or column flange in bending, bolts in tension

## Equivalent T-stub

Resistance and stiffness of column flange or end plate in bending is calculated with the help of equivalent T-stub with one row of bolts.




*Equivalent T-stub*

## Resistance

There are three collapse modes for equivalent T-stub:

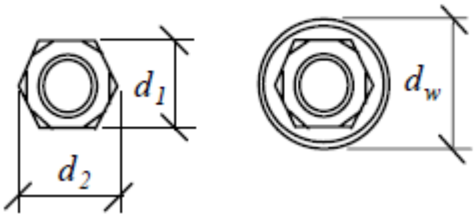
1. Complete yielding of the flange
2. Bolt failure with yielding of the flange
3. Bolt failure

Bolt resistance is based on the tensile resistance or stress under washer (or bolt head/nut):

$$2B_{t.Rd} = 2min \begin{Bmatrix} 0.9 A_s f_{ub} / \gamma_{M2} \\ 0.6 \pi d_w t f_u / \gamma_{M2} \end{Bmatrix}$$

Where the dimension $d_w$ is selected according to the size of washer or nut:




*Dimensions of nuts and washers*

The resistance of equivalent T-stub is calculated as a minimum resistance for three failure modes described above. The first failure mode (yielding of the flange) is given by the expression in accordance with table *6.2*:

$$F_{T,Rd} = \frac{4M_{pl,1,Rd}}{m}$$

The more exact method is

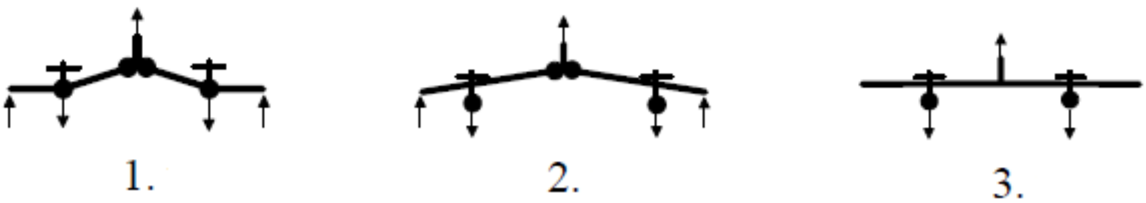
$$F_{T,Rd} = \frac{(8n - 2e_w)\,M_{pl,1,Rd}}{2mn - e_w\,(m+n)}$$

Second failure mode is given by the formula

$$F_{T,Rd} = \frac{2M_{pl,2,Rd} + 2nB_{t,Rd}}{m+n}$$

This expression is usef for the third failure mode:

$$F_{T,Rd} = 2B_{t,Rd}$$

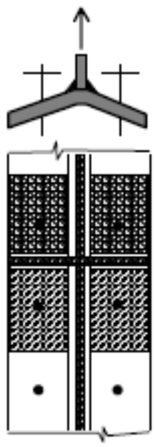



*Failure modes*

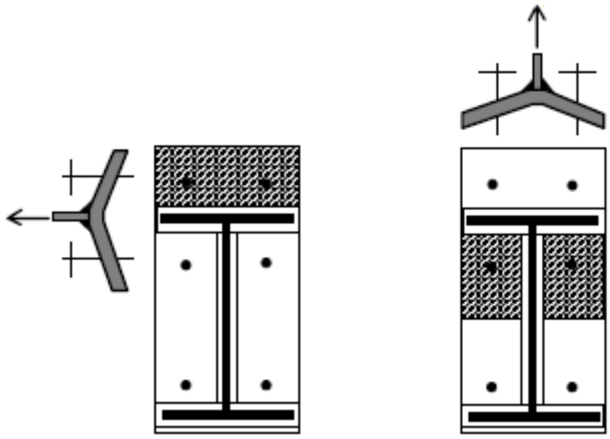
Where is:

$$n = e_{min} \leq 1.25m$$
$$M_{pl,1,Rd} = 0.25L_{eff,1}t_f^2 f_y/\gamma_{M0}$$
$$M_{pl,2,Rd} = 0.25L_{eff,2}t_f^2 f_y/\gamma_{M0}$$
$$e_w = d_w/4$$

The equivalent T-stub is selected in that way, that the bending about the beam web appears. Only exception are the bolts in unstiffened overhanging part of the plate. The bending is considered about the beam flange in this case.

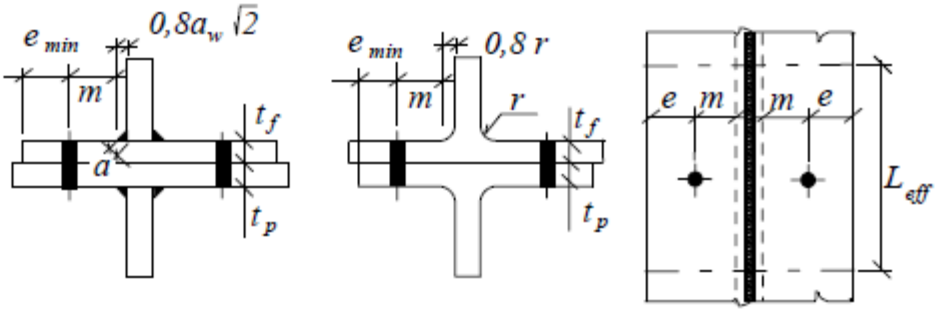


*The equivalent T-stub for stiffened column web*



*The equivalent T-stub for unstiffened overhanging part of the plate and fof bolts under the flange*

The resistance of equivalent T-stub is calculated for both sides of the connection (column flange and end plate). The minimum value is considered as the resulting resistance.




*Dimensions of welded and rolled T-stub*

## Effect of axial stress in column

The connection resistance may be affected for combination of transverse bending and axial stress (due to axial force and bending moment). If the stress in the column flange $\sigma_{com.Ed}$ is greater than *180MPa* (for *S235*), the resistance moment of the flange $M_{pl.Rd}$ is reduced by the factor

$$k_{fc} = (2f_{y,fc} - 180 - \sigma_{com,Ed})\,(2f_{y,fc} - 360) \leq 1.0$$

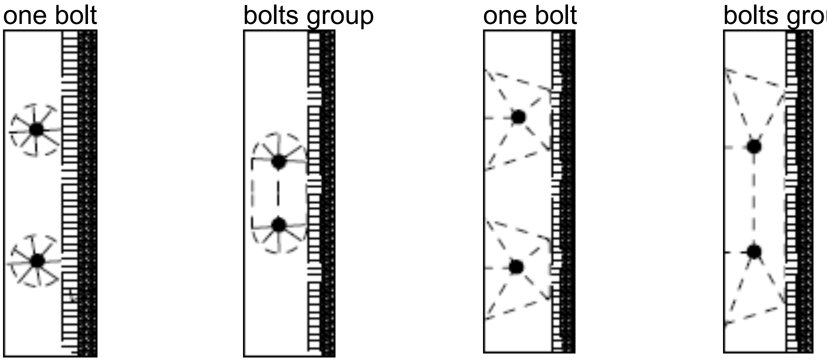
## The effective length of the equivalent T-stub

The effective length of the equivalent T-stub $L_{eff}$ is calculated for individual bolt rows using the method of plastic hinges. The worst value is considered when more failure modes may appear. As the second failure mode cannot appear for circular failure mode (length $L_{eff.cp}$), two different effective lengths according to the figure below are used.

$$L_{eff,1} = min\,(L_{eff,cp}; L_{eff,op})$$
$$L_{eff,2} = L_{eff,op}$$

Circular failure $L_{eff.cp}$ — one bolt, bolts group

Other failure $L_{eff.op}$ — one bolt, bolts group



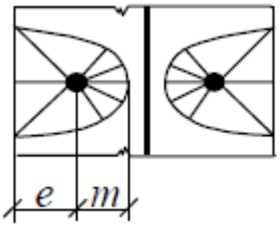
The effective length of the equivalent T-stub for circular failure is calculated as the length of one half of the T-stub, that has the same resistance as circular plate loaded in the middle and supported along its perimeter. The assumption $m = r \approx n$ was confirmed by tests. Therefore, the corresponding expression is

$$L_{eff.cp} = 2\pi m$$

The effective length for bolts close to the plate edge is calculated in a similar way:

$$L_{eff,op} = 4m + 1.25e$$




*Circular plastic hinge for bolts close to the plate edge*

The effective length $L_{eff}$ for bolts along the stiffener of flange or column (and for bolts under the beam flange) is calculated separately using following expression

$$L_{eff,op} = \alpha m$$

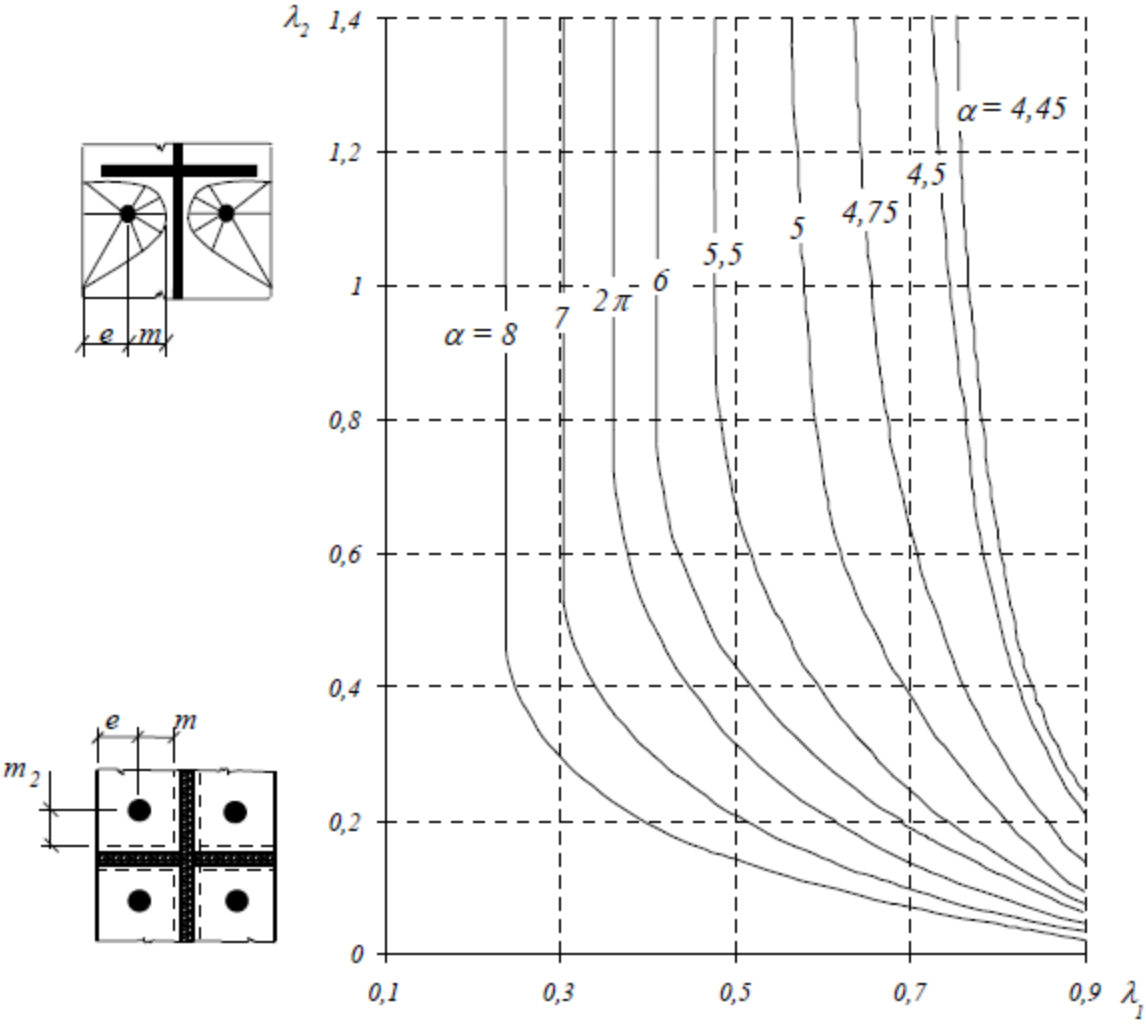
The effective length for the failure of bolts group is calculated according to the following formula

$$L_{eff,op} = 0.5p + \alpha m - 2m - 0.625e$$

The factor $\alpha$ is determined from the graph, that is based on analytical and numerical methods used for verification of the bolt in the corner. Following parameters are used:

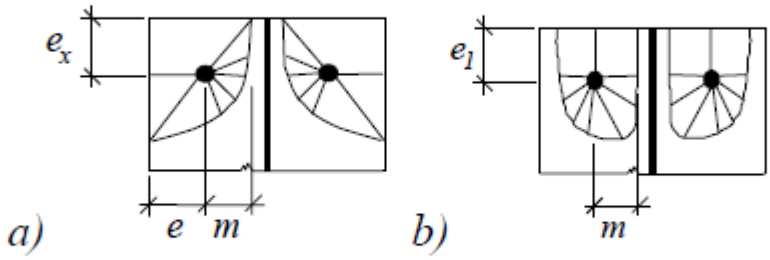
$$\lambda_1 = \frac{m}{m+e}; \lambda_2 = \frac{m_2}{m+e}$$

The dimensions $m$ , $m_2$ and $e$ of T-stub are shown in the following figure:




*Graph for calculation of effective length close to the stiffener*

The values of the factor $\alpha$ are in the interval between *4,45* and *8* depending on the bolt position in the corner. These values were found with the help of tests and analytical methods.




*Plastic hinges at the end of flange*

The effective length for bolts at the end of the flange (option *a* in the figure above) is calculated using expression

$$L_{eff,op} = 2m + 0.625e + e_x$$

The effective length for circular failure (option *b* in the figure above) is calculated using expression

$$L_{eff,cp} = \pi m + 2e_1$$

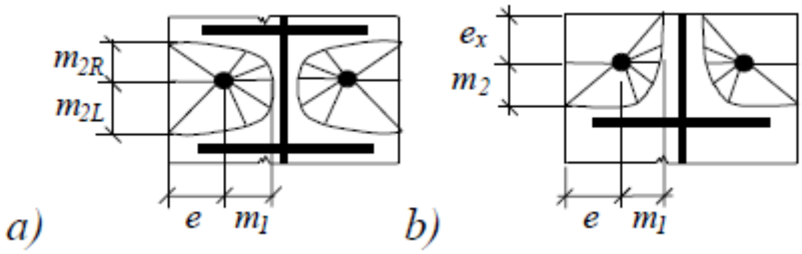
The effective length for bolts between two stiffeners (option *a* in the figure below) is calculated using expression

$$L_{eff,op} = \alpha_R m_1 + \alpha_L m_1 - (4m_1 + 1.25e)$$

Where $\alpha_R$ corresponds to $m_R$ and $\alpha_L$ is calculated for $m_L$.

The effective length for bolts between stiffener and end of flange (option *b* in the figure below) is calculated using expression

$$L_{eff,op} = \alpha m_1 - (2m_1 + 0.625e) + e_x$$




*Plastic hinges between two stiffeners and between stiffener and flange*

One of following failure modes is the decisive one for bolts at the end of plate:

- Plastic circular failure

$$L_{eff.cp} = 2\pi m_x$$

- Failure of individual bolts

$$L_{eff,op} = 4m_x + 1.25e_x$$

- Failure in the corner of the plate

$$L_{eff,op} = 2m_x + 0.625e_x + w/2$$

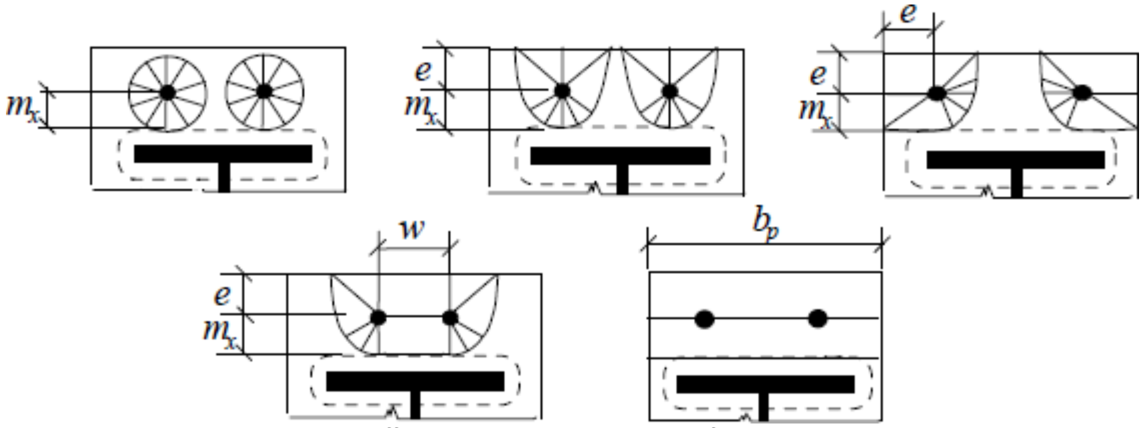
- Failure of the bolts group

$$L_{eff.op} = 2m_x + 0.625e_x + w/2$$

- Bending of plate

$$L_{eff,op} = \frac{b_p}{2}$$

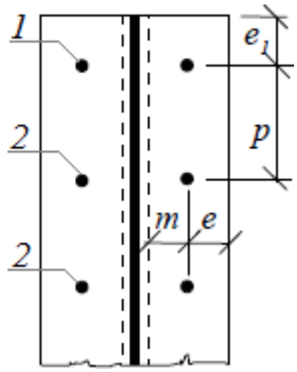
 




*Effective lengths at the end of plate*

The resistance of individual rows of bolts is calculated according to the following table for flanges of unstiffened columns:

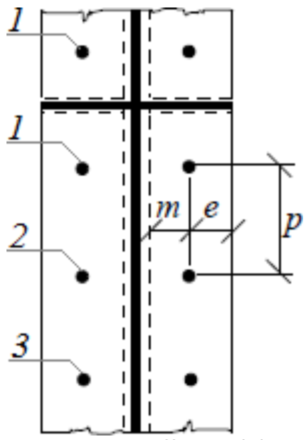
| Rows of bolts | Failure of single row | | Failure of rows group | |
|---|---|---|---|---|
| | circular ($L_{eff.cp}$) | other ($L_{eff.op}$) | circular ($L_{eff.cp}$) | other ($L_{eff.op}$) |
| Internal | $2\pi m$ | $4m+1{,}25e$ | $2p$ | $p$ |
| End | $2\pi m$<br>$\pi m+2e_1$ | $4m+1{,}25e$<br>$2m+0{,}625e+e_1$ | $\pi m+p$<br>$2e_1+p$ | $2m+0{,}625e+0{,}5p$<br>$e_1+0{,}5p$ |




*Unstiffened column flange: End (1) and internal (2) rows*

The resistance of individual rows of bolts is calculated according to the following table for flanges of stiffened columns:

| Rows of bolts | Failure of single row | | Failure of rows group | |
|---|---|---|---|---|
| | circular ($L_{eff.cp}$) | other ($L_{eff.op}$) | circular ($L_{eff.cp}$) | other ($L_{eff.op}$) |
| Close to stiffener | $2\pi m$ | $\alpha m$ | $\pi m+p$ | $0{,}5p+\alpha m-2m-0{,}625e$ |
| Internal | $2\pi m$ | $4m+1{,}25e$ | $2p$ | $p$ |
| End | $2\pi m$ | $4m+1{,}25e$ | $\pi m+p$ | $2m+0{,}625e+0{,}5p$ |

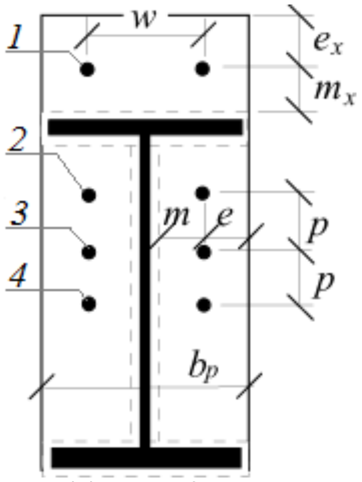



*Stiffened column flange: row close to stiffener (1), internal (2) and end (3) rows*

The resistance of individual rows of bolts is calculated according to the following table for end plates:

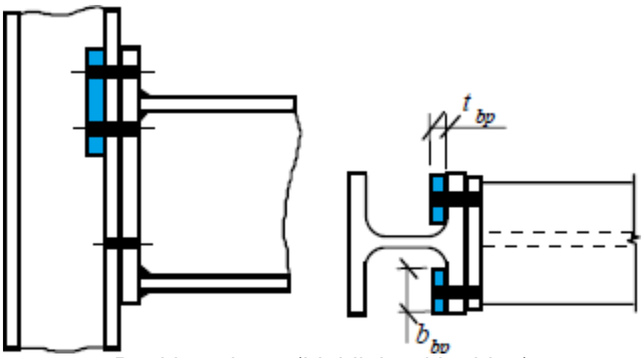
| Rows of bolts | Failure of single row | Failure of rows group |
|---|---|---|

| | circular ($L_{eff.cp}$) | other ($L_{eff.op}$) | circular ($L_{eff.cp}$) | other ($L_{eff.op}$) |
|---|---|---|---|---|
| Out of flange in tension | $2\pi m$<br>$\pi m_x+w$<br>$\pi m_x+2e$ | $4m_x+1{,}25ex$<br>$e+4m_x+0{,}625e_x$<br>$0{,}5b_p$<br>$0{,}5w+2m_x+0{,}625e_x$ | | |
| Under flange in tension | $2\pi m$ | $\alpha m$ | $\pi m+p$ | $0{,}5p+\alpha m-2m-0{,}625e$ |
| Internal | $2\pi m$ | $4m+1{,}25e$ | $2p$ | $p$ |
| End | $2\pi m$ | $4m+1{,}25e$ | $\pi m+p$ | $2m+0{,}625e+0{,}5p$ |



*End plate: Rows out of flange in tension (1), under flange in tension (2), internal (3) and end (4) rows*

The column flange may be restrained by backing plates:



*Backing plates (highlighted by blue)*

Backing plates increase the resistance in the failure *1* (4 plastic hinges in T-stub). The increase is equal to the bending resistance of backing plate

$$F_{T,Rd} = \frac{4M_{pl,Rd} + 2M_{bp,Rd}}{m}$$

Where the plastic moment is calculated using formula

$$M_{bp,Rd} = 0.25 \sum L_{eff} t_{bp}^2 f_{y,bp} / \gamma_{M0}$$

Where is: ***$t_{bp}$*** • The thickness of backing plate

***$f_{y.bp}$*** • The yield strength of backing plate

## Resistance of unstiffened column flange in the welded joint

The resistance of unstiffened column flange in the welded joint is calculated using formula

$$F_{t,fc,Rd} = (t_{wc} + 2s + 7kt_{fc})\, t_{fb} f_{y,fb} / \gamma_{M0}$$

but

$$F_{t,fc,Rd} \geq 0,7 b_{fb} t_{fb} f_{y,fb} / \gamma_{M0}$$

Where is: ***$t_{wc}$, $t_{fc}$*** • The thickness of column flange

***$f_{fb}$, $t_{fb}$*** • The dimensions of connected member

Ans $s = r_c$ for rolled sections and $s = a_{b.wf} 2^{0,5}$ for welded sections. The factor $k$ is calculated using formula

$$k = \frac{t_{fc} f_{y,fc}}{t_{fb} f_{y,fb}} \leq 1.0$$

## Stiffness

The assumption is that the stiffness of any component can be calculated separately. This assumption does not work for prying of bolts. In these cases, the stiffness depends also on the stiffness of T-stub. Only rough estimation is done in this case. The deformation of single row $\delta_b$ can be calculated uding formula

$$\delta_b = \frac{F_b L_b}{2 E A_s}$$

The stiffness of one row $k_b$ can be calculated using expression

$$k_b = \frac{F_b}{E \delta_b} = 2.0 \frac{A_s}{L_b}$$

The stiffness factor of plate in bending, where no prying occurs, is based on the expression for cantlever deformation:

$$\delta_p = \frac{F_p m^3}{3EI}$$

$$k_p = \frac{F_p}{E \delta_p} = \frac{F_p 3EI}{E F_p m^3} = \frac{3 \frac{L_{eff.ini} t^3}{12}}{m^3} = \frac{L_{eff.ini} t^3}{2 m^3}$$

EN 1993-1-8 uses initial spring value of T-stub length, that is estimated as $L_{eff.ini} = 0{,}9\, L_{eff}$.

The stiffness factor calculation of T-stub with prying is more complicated, as mutual influence of bolts in tension and T-stub occurs. This effect is not considered and following expression is used (subscripts $k_{4,5,6}$ correspond to the components according to EN 1993-1-8)

$$k_p = k_4 = k_5 = k_6 = 0.9 \frac{L_{eff} t^3}{m^3}$$

The backing plates do not increase the stiffness of T-stub significantly. Therefore backing plates are not considered in the calculation of stiffness.

# Column web in shear

The resistance of unstiffened column web in shear is calculated using formula

$$V_{wp,Rd} = \frac{0.9 f_{y,wc} A_{vc}}{\gamma_{M0} \sqrt{3}}$$

Where the shear area is considered for rolled column as

$$A_{vc} = A_c - 2 b_c t_{fc} + (t_w + 2 r_c) t_{fc}$$

The resistance is increased for web with horizontal stiffeners. The resistance increment is

$$V_{wp,add,Rd} = \frac{4 M_{pl,fc,Rd}}{d_s}$$

but

$$V_{wp,add,Rd} \leq \frac{2 M_{pl,fc,Rd} + 2 M_{pl,st,Rd}}{d_s}$$

Where is:

***$d_s$*** • The distance between stiffeners

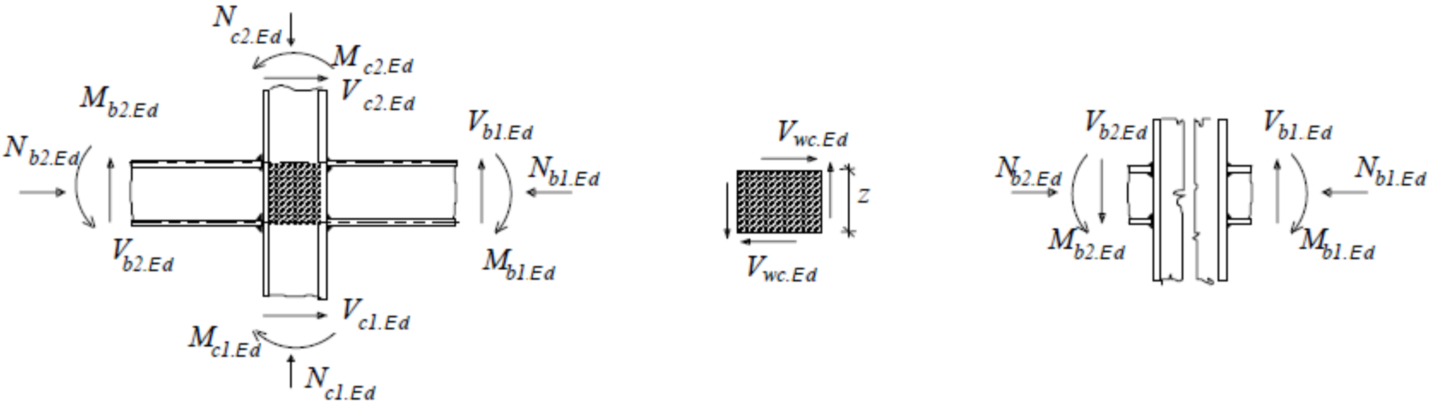
***$M_{pl.fc.Rd}$*** • The design moment resistance of column flange

***$M_{pl.st.Rd}$*** • The design moment resistance of column stiffener

The resistance of column web in shear has to be greater than the average shear force in the column. This value is calculated using expression

$$V_{wc,Rd} = \frac{M_{b1,Ed} - M_{b2,Rd}}{z} - \frac{V_{c1,Rd} - V_{c2,Ed}}{2}$$

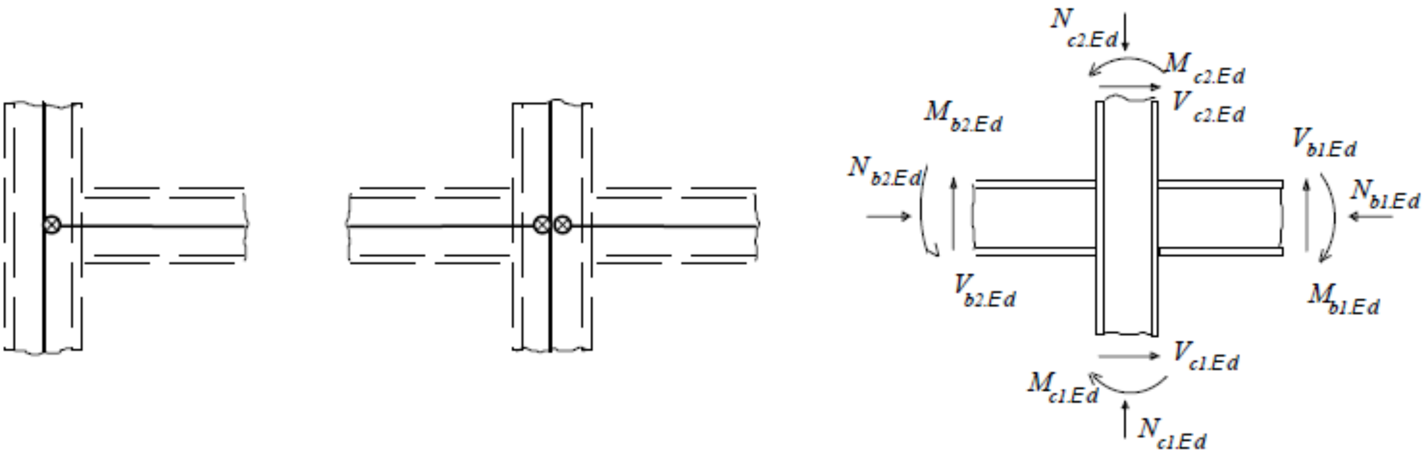
 



*Column web in shear*

## Shear stress induced by bending moment

The detail is converted into the analysis layout bending springs in bending.



*The analysis model with springs*

The unbalance of moments in the connection is added into account with the help of the transformation parameters $\beta$. The parameter $\beta_1$ for the right connection is calculated using expression

$$\beta_1 = \left| \left(1 - \frac{M_{b2,Ed}}{M_{b1,Ed}}\right) - \frac{z}{2M_{b1,Ed}} \left(V_{c1,Ed} - V_{c2,Ed}\right) \right|$$

The parameter $\beta_2$ for the left connection is based on the following formula

$$\beta_2 = \left| \left(1 - \frac{M_{b2,Ed}}{M_{b1,Ed}}\right) - \frac{z}{2M_{b2,Ed}} \left(V_{c1,Ed} - V_{c2,Ed}\right) \right|$$

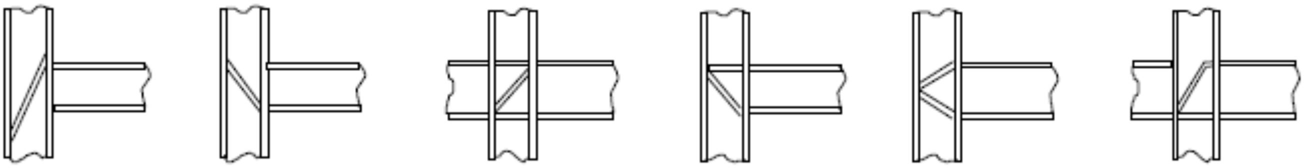
The shear effect is added with the help of the reduction factor $\omega$ according to the table *6.3*:

| **Transformation parameter $\beta$** | **Reduction factor $\omega$** |
|---|---|
| $0 \le \beta \le 0,5$ | $\omega = 1,0$ |
| $0,5 < \beta < 1,0$ | $\omega = \omega_1 + 2(1-\beta)(1-\omega_1)$ |
| $\beta = 1,0$ | $\omega = \omega_1$ |
| $1,0 < \beta < 2,0$ | $\omega = \omega_1 + (\beta-1)(\omega_2-\omega_1)$ |
| $\beta = 2,0$ | $\omega = \omega_2$ |

Where the values of $\omega_1$ and $\omega_2$ are calculated using following formulas

$$\omega_1 = \frac{1}{\sqrt{1 + 1.3\left(b_{eff} t_{wc} / A_{vc}\right)^2}}$$

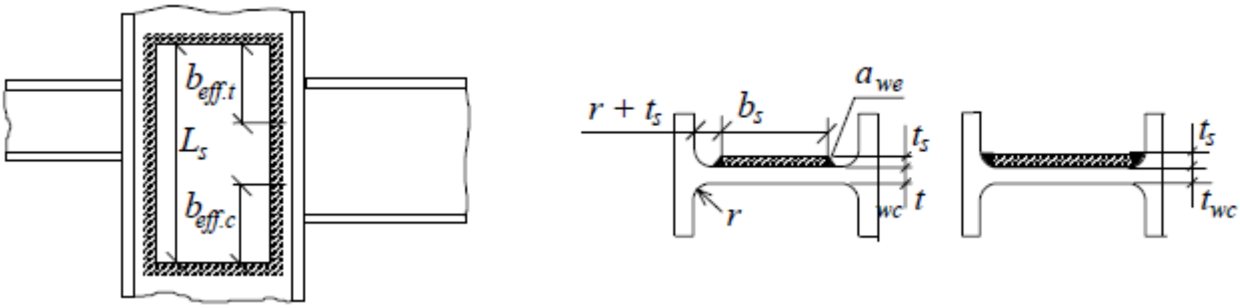
$$\omega_2 = \frac{1}{\sqrt{1 + 5.2\left(b_{eff} t_{wc} / A_{vc}\right)^2}}$$



*Shear stiffeners*

## Supplementary web plates

Supplementary web plates increase the web resistance in shear, tension and compression. The plate width (maximum $b_s \leq 40\varepsilon t_s$ without internal connection) should cover whole web height between roundings. The thickness should not be greater that the web thickness $(t_s \geq t_{wc})$. The plate height $L_s$ should cover complete effective web length in tension and compression. Welds should have sufficient effective thickness. For column web in shear and compression the value $a_{we} \geq t_s/2^{0,5}$ should be considered. The thickness $a_{we} \geq t_s$ for longitudinal welds and $a_{we} \geq ts/2^{0,5}$ for transverse welds should be used for column web in tension.




*Column web with supplementary web plate*

## Stiffness

The stiffness of unstiffened column web in the shear is calculated using expression

$$k_{wp} = k_1 = 0.38 \frac{A_{vc}}{\beta z}$$

| | | |
|---|---|---|
| Where is: | ***A*$_{vc}$** | • The shear area of column web |
| | ***β*** | • The transformation parameter |
| | ***z*** | • The inner lever arm |

# Column web in tension and compression

## Column web in tension

The design resistance of unstiffened column web is calculated using expression

$$F_{t,wc,Rd} = \frac{\omega b_{eff,t} t_{wc} f_{y,wc}}{\gamma_{M0}}$$

where the factor $\omega$ corresponds to the shear impact, table is shown in the chapter "**Column web in shear**".

The effective height in tension for welded connections is given by the expression

$$b_{eff,t} = t_{fb} + 2\sqrt{2} a_{b,wf} + 5\left(t_{fc} + s\right)$$

where

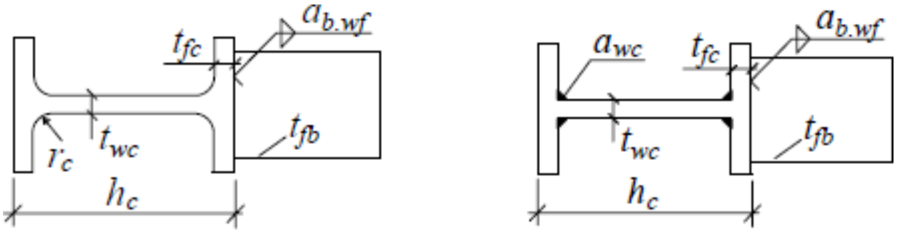
$$s = r_c$$

for hot-rolled cross-sections and

$$s = a_{b,wf}\sqrt{2}$$

for welded cross-sections.




*Beam connection to hot-rolled and welded columns*

The effective column web width $b_{eff,t}$ for bolted connections is equal to the effective length of eqivalent T-stub.

## Column web in compression

The design resistance of unstiffened column web is calculated using expression

$$F_{c,wc,Rd} = \frac{\omega b_{eff,c} t_{wc} f_{y,wc}}{\gamma_{M0}} \leq \frac{\omega \rho b_{eff,c} t_{wc} f_{y,wc}}{\gamma_{M1}}$$

where the factor $\omega$ corresponds to the shear impact, table is shown in the chapter "**Column web in shear**". The resistance is limited by its stability, that is taken into account with the help of the factor $\rho$.

The effective height in compression for welded connections is given by the expression

$$b_{eff,c,wc} = t_{fb} + 2\sqrt{2}a_p + 5\left(t_{fc} + s\right)$$

where

$$s = r_c$$

for hot-rolled cross-sections and

$$s = a_{wc}\sqrt{2}$$

for welded cross-sections.

The effective column web height $b_{eff,wc}$ for bolted connections with end plate thickness $t_p$ is calculated using expression .

$$b_{eff,c,wc} = t_{fb} + 2\sqrt{2}a_p + 5\left(t_{fc} + s\right) + 2t_p$$

The last part of previous expression ($2t_p$) may be limited by the cantilever dimension. Following formula may be used for connections with angles on flanges

$$b_{eff,c} = 2t_a + 0.6r_a + 5\left(t_{fc} + s\right)$$

| Where is: | | |
|---|---|---|
| | $t_a$ | • The thickness of angle |
| | $r_a$ | • The root radius |

## Column web stability

The slenderness of column web $\overline{\lambda_p}$ is given by the formula

$$\overline{\lambda_p} = 0.932\sqrt{\frac{b_{eff,c} d_{wc} f_{y,wc}}{E t_{wc}^2}}$$

The web height of hot-rolled column $d_{wc}$ is given by the formula

$$d_{wc} = h_c - 2\left(t_{fc} + r_c\right)$$

The expression for welded columns:

$$d_{wc} = h_c - 2\left(t_{fc} + \sqrt{2}a_c\right)$$

The reduction factor $\rho$ is given for

$$\overline{\lambda_p} \leq 0.72$$

by the formula

$$\rho = 1.0$$

and for

$$\overline{\lambda_p} > 0.72$$

by the formula

$$\rho = \frac{\overline{\lambda_p} - 0.22}{\overline{\lambda_p}^2}$$

### Effect of axial stress in column

The axial stress may affect the resistance of column flange and web when calculating the compressive stresses. If the axial stress $\sigma_{com,Ed}$ at the beginning of root radius or weld edge of column flange is greater than $0{,}5 f_{y,wc}$ , The compressive resistance of column web $F_{c,wc,Rd}$ is reduced by the factor $k_{wc}$

$$k_{wc} = 1.25 - 0.5\sigma_{com,Ed}/f_{y,wc} \leq 1.0$$

### Stiffness

The stiffness od unstiffened column web in compression $k_2$ or stiffened or unstiffened column web in tension $k_3$ is given by the formula

$$k_{c,wc(t,wc)} = k_{2(3)} = 0.7\frac{b_{eff,wc}t_{wc}}{d_{wc}}$$

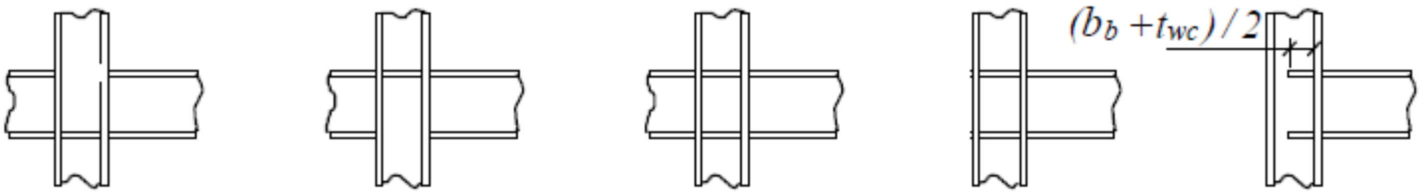
Where is:

- $b_{eff,wc}$ • The effective height of column web in compression or tension
- $d_{wc}$ • The free column web height
- $t_{wc}$ • The column web thickness

The value of $b_{eff,t,wc}$ for bolted connection with one row may be considered as the minimum of effective lengths $L_{eff}$ (individually or as a part of bolt group).

### Column web stiffeners

The column web may be stiffened both in tension and compression. The Stiffeners shall have the thickness equal to the beam flange thickness and identical or better steel grade. The full capacity of stiffeners may be reached also for partial stiffeners, if the stiffeners length is longer than $(b_b + t_{wc})/2$ , where $b_b$ is beam width and $t_{wc}$ is the column web thickness.




*Column web stiffeners*

## Beam flange and web in tension and compression

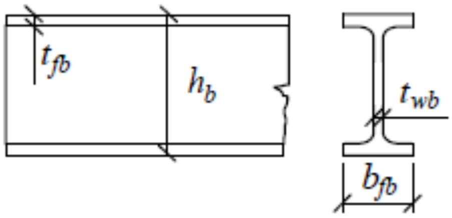
### Compressive resistance of beam web and flange

The design value in compression is calculated with the help of the fundamental beam resistance

$$F_{c,fb,Rd} = \frac{M_{c,Rd}}{(h_b - t_{fb})}$$

Where is:

- $M_{c,Rd}$ • The bending resistance of beam (the shear effect may be neglected in the most of cases, therefore $M_{c,Rd} \approx M_{b,Rd}$)
- $h_b$ • The beam height
- $t_{fb}$ • The beam flange thickness




*Connected beam*

## Tensile resistance of beam web

The column web resistance in tension for bolted connections may be calculated using expression

$$F_{t,wb,Rd} = b_{eff,t,wb} t_{wb} f_{y,wb} / \gamma_{M0}$$

Where is: ***b**$_{eff,t,wb}$*

- The length of equivalent T-stub of end plate for one or more bolt rows

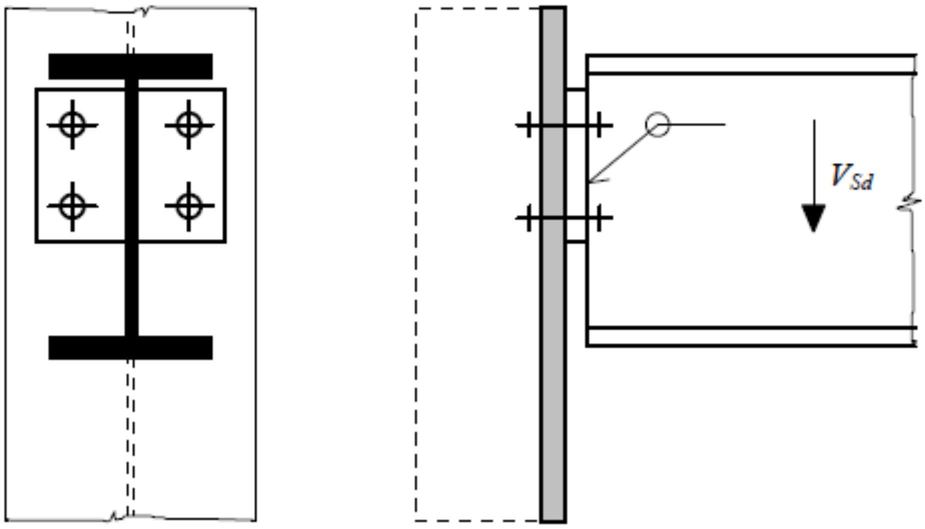
## Stiffness

The deformability of column flange in tension and compression is insignificant comparing to the stiffness of other components. Therefore, stiffness of this component is considered as infinity.

# TeorieOSBrit

# Pinned end plate to column

The detail scheme:



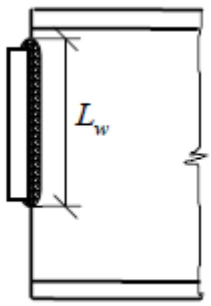

*Pinned end plate to column detail*

The bearing capacity of the connection is given by the minimum

$$F_{Ed} = min\left\{F_{vbw,Rd}; nF_{b,v,Rd}; nF_{b,b,Rd}; F_{w,v,Rd}; F_{p,v,Rd}\right\}$$

of following vales *:*

- The shear resistance of beam web $F_{vbw,Rd}$ (usually critical component for this type of connection)

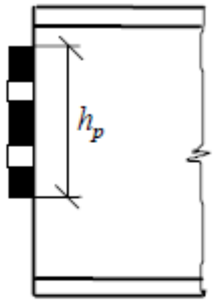



*The shear length of beam web*

- The resistance of *n* bolts $nF_{b,v,Rd}$
- The bearing resistance of bolts in end plate $nF_{b,b,Rd}$ (can be calculated as a sum of bearing resistances of end and intermediate bolts for materials with sufficient ductility)
- The shear resistance of welds $F_{w,v,Rd}$
- The shear resistance of end plate $V_{p,v,Rd}$ in the cross-section with gaps. This can be neglected if $A_{v,net}/A_v > f_y/f_u$ . In the end plate cross-section is considered only a half of the force in a beam.

The bending resistance and the interaction with shear may be limiting for long end plates.

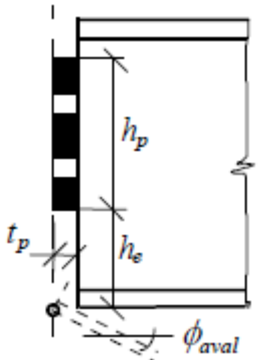



*Cross-section of end plate with gap*

The connection may be considered as a pinned one if the beam rotation does not cause the contact of bottom flange with a column The rotation capacity is greater than rotation of simple beam

$$\phi_{aval} = t_p / h_e \geq \phi_{req} = \frac{\gamma_{M1} L_b^3}{24 E I_b}$$

| | | |
|---|---|---|
| Where is: | $L_b$ | • The beam span |
| | $E$ | • The modulus of elasticity |
| | $I_b$ | • The second moment of inertia |




*Rotation of beam end*

The remaining tensile resistance for the interaction of tension and shear shall be able to transfer full plastic moment resistance of connected end plate. Therefore, the thickness of the plate should be limited

$$t \leq 0.36 d \sqrt{f_{ub} / f_y}$$

Where $d$ is bolts diameter. Welds are designed in that way, that they are able to transfer full plastic resistance moment of the plate in tension.

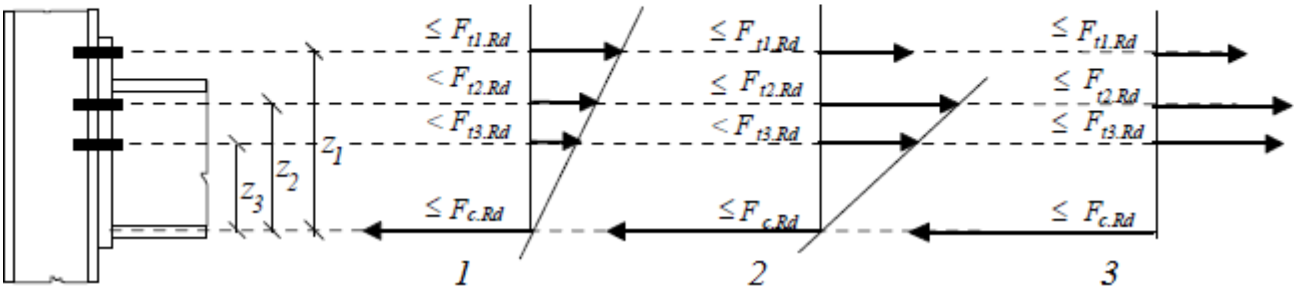
## Stiff end plate to column

### Bending resistance

The bending resistance of the joint is calculated as the minimum of component resistances in the joint part in tension, that is limited by the resistance of components in compressive part of the joint. The plastic stress distribution is considered in the most of cases, sometimes, the elastic distribution is used. The procedure depends on the certain joint type. Elastic distribution is based on the most stressed bolt under the top flange, sometimes it is considered as an elastic distribution along the whole height of the cross-section made of bolts and part in compression.

The shear forces are distributed uniformly for elastic distribution. In plastic distribution, the shear forces are assigned only to bolts, that do not transfer the bending, or the shear and bending resistances are calculated with the help of interaction diagram. Tensile force has to contain the impact of prying. The slip resistance of preloaded bolts is not affected by the tensile forces in bolts, as the friction is considered only in the compressive part.




*Stress distribution: elastic (1), elastic-plastic (2) and plastic model (3), resistance of i-row in tension $F_{t,i,Rd}$, the minimum resistance of components in compression $F_{c,Rd}$*

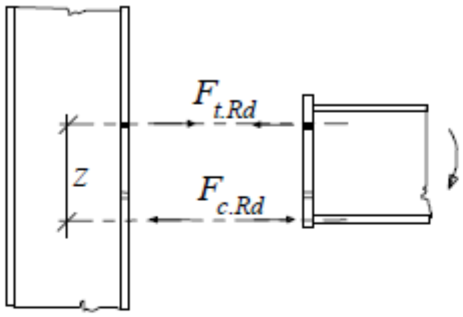
The bending resistance of bolted connection for plastic distribution is calculated using expression

$$M_{j,Rd} = \sum_i F_{ti,Rd} z_i$$

Where is:

- $F_{ti,Rd}$ • The resistance of *i*- row of bolts in tension
- $z_r$ • The distance of *i* -row measured from the centre of compression part

The axis of compressive part is considered in the axis of flange in compression. For one component in tension, the inner lever arm may be considered as the distance of axes of compressive and tensile components.




*Inner lever arm for bolted joints with one row in tension*

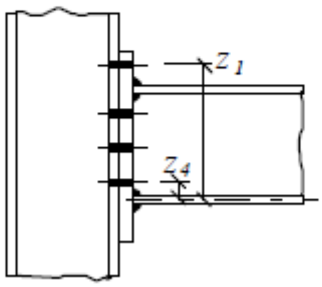
For more components (e.g. more rows of bolts) in tensile part, the equivalent stiffness of tensile part $k_{eq}$ is used. This stiffness contains both the effect of particular components $k_{eff.i}$ and the impact of arm of corresponding row on the stiffness of the tensile part.

$$k_{eq} = \frac{\sum_i k_{eff,i} z_i}{z}$$

$$k_{eff} = \frac{1}{\sum_i \frac{1}{k_i}}$$

Inner lever arm for more rows of bolts may be calculated using approximate method according to the figura above or ma be calculated using following expression

$$z = \frac{\sum_i k_{eff,i} z_i^2}{\sum_i k_{eff,i} z_i}$$




*Inner lever arm*

where $z_i$ is the distance of components in tension with stiffness $k_{eff,i}$ from the centre of the part in compression.

The resistance of one row in tension $F_{t,Rd}$ is calculated as a minimum of following values:

- The column flange in bending $F_{t,fc,Rd}$
- The column web in tension $F_{t,wc,Rd}$
- The end plate in bending $F_{t,ep,Rd}$
- The beam web in tension $F_{t,wb,Rd}$

The resulting resistance is limited by the resistance of part in compression and the resistance of web in shear:

- The column web in shear $V_{wp,Rd}/\beta$
- The column web in compression $F_{c,wc,Rd}$
- The beam flange in compression $F_{c,fb,Rd}$

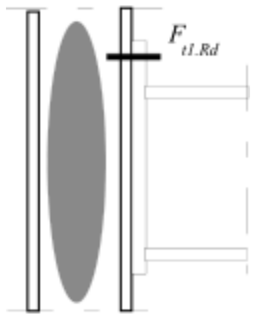
The resistance of any row of bolts has to be reduced in case of more than one row of bolts in tensile part. The bearing capacity cannot be greater than:

- The column flange in bending $F_{t,fc,Rd}$
- The column web in tension $F_{t,wc,Rd}$
- The end plate in bending $F_{t,ep,Rd}$
- The beam web in tension $F_{t,wb,Rd}$

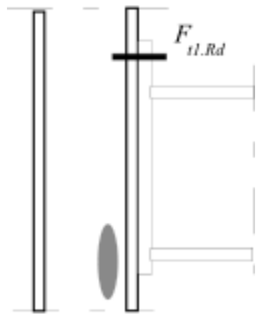
The effect of prying caused by the deformation of end plate has to be checked when designing the detail with end plate. The effect of prying is transferred into the analysis of equivalent T-stub. The increase of plate thickness avoid prying, as the decisive failure would be mode *3* in this case. This mode is not very suitable, as the resistance is given by the failure in bolts. In joints with full plastic hinges, the strengthening of material may affect the resistance of welds. The welds are designed using increased joint resistances in these cases. It means $1.4M_{j,Rd}$ for braced frames and $1.7M_{j,Rd}$ for unbraced frames. The effect of more rows is described in tables below:

The resistance of connection with three rows in tensile part: The resistance of first row $F_{t1,Rd}$ is calculated first, next two rows are neglected

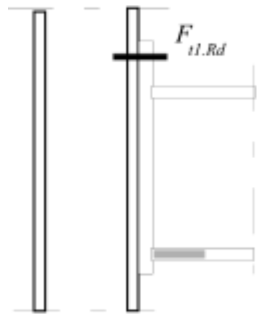
**Column web in shear and compression, beam flange in compression**




**Column web in shear**
$F_{t1,Rd} \leq V_{wp,Rd}/\beta$

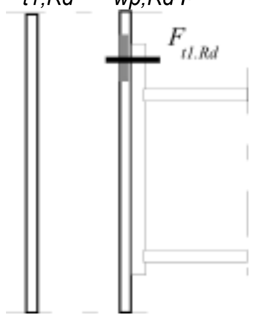



**Column web in compr.**
$F_{t1,Rd} \leq F_{c,wc,Rd}$

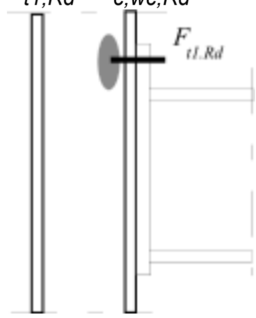



**Beam flange in compr.**
$F_{t1,Rd} \leq F_{c,fb,Rd}$

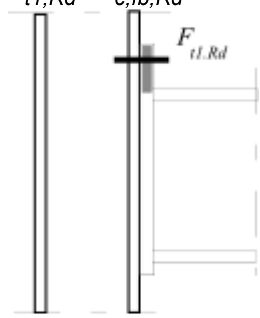
**Resistance of first row in tension**




**Column flange in bending**
$F_{t1,Rd} \leq F_{t,fc,Rd}$



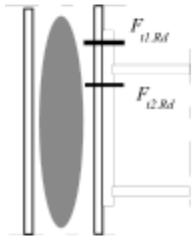

**Column web in tension**
$F_{t1,Rd} \leq F_{t,wc,Rd}$



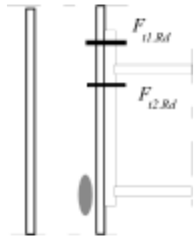

**End plate in bending**
$F_{t1,Rd} \leq F_{t,ep,Rd}$

The resistance of second row $F_{t2,Rd}$ follows, the third row is neglected

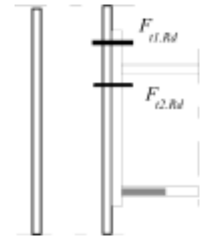
**Column web in shear and compression, beam flange in compression**



**Column web in shear**
$F_{t2,Rd} \leq V_{wp,Rd}/\beta - F_{t1,Rd}$

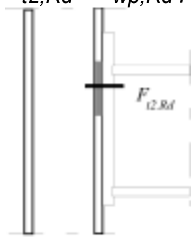


**Column web in compr.**
$F_{t2,Rd} \leq F_{c,wc,Rd} - F_{t1,Rd}$

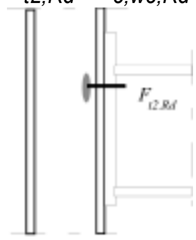


**Beam flange in compr.**
$F_{t2,Rd} \leq F_{c,fb,Rd} - F_{t1,Rd}$

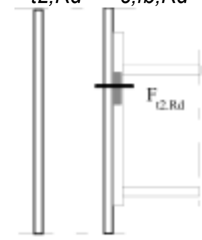
**Resistance of second row in tension**



**Column flange in bending**
$F_{t2,Rd} \leq F_{t2,fc,Rd}$

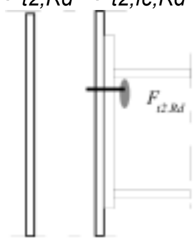


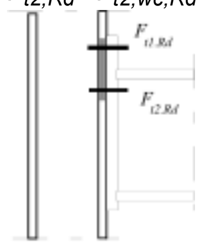
**Column web in tension**
$F_{t2,Rd} \leq F_{t2,wc,Rd}$

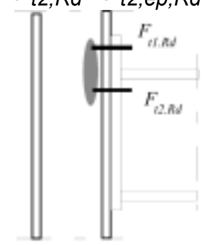


**End plate in bending**
$F_{t2,Rd} \leq F_{t2,ep,Rd}$

**Resistance of second row in tension**







**Resistance of two rows in tension**

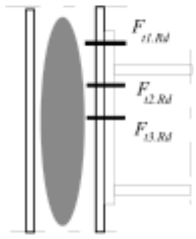
**Beam web in tension**
$F_{t2,Rd} \leq F_{t2,wb,Rd}$

**Column flange in bending**
$F_{t2,Rd} \leq F_{t(1+2),fc,Rd} - F_{t1,Rd}$

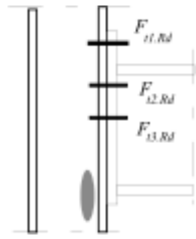
**Column web in tension**
$F_{t2,Rd} \leq F_{t(1+2),wc,Rd} - F_{t1,Rd}$

The resistance of third row $F_{t3.Rd}$ follows

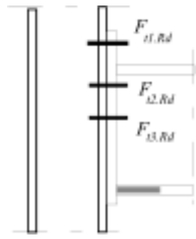
**Column web in shear and compression, beam flange in compression**



**Column web in shear**
$F_{t3,Rd} \leq V_{wp,Rd}/\beta - F_{t1,Rd} - F_{t2,Rd}$

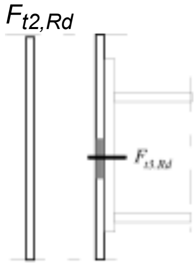


**Column web in compr.**
$F_{t3,Rd} \leq F_{c,wc,Rd} - F_{t1,Rd} - F_{t2,Rd}$

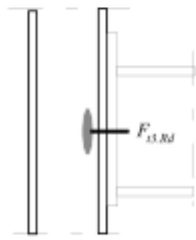


**Beam flange in compr.**
$F_{t3,Rd} \leq F_{c,fb,Rd} - F_{t1,Rd} - F_{t2,Rd}$

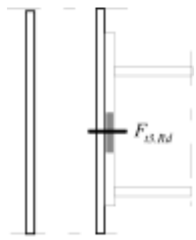
**Resistance of third row in tension**



**Column flange in bending**
$F_{t3,Rd} \leq F_{t3,fc,Rd}$

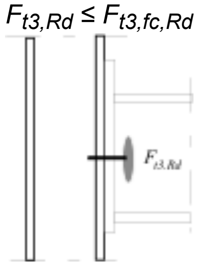


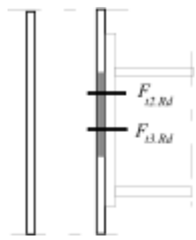
**Column web in tension**
$F_{t3,Rd} \leq F_{t3,wc,Rd}$

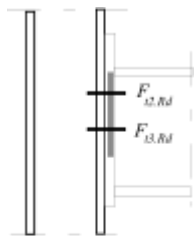


**End plate in bending**
$F_{t3,Rd} \leq F_{t3,ep,Rd}$

**Resistance of third row in tension**







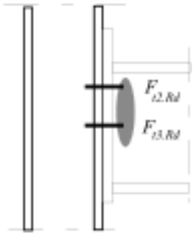
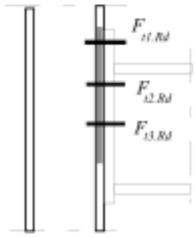
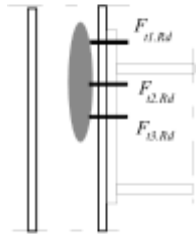
**Resistance of second and third rows in tension**

**Beam web in tension**
$F_{t3,Rd} \leq F_{t3,wb,Rd}$

**Column flange in bending**
$F_{t3,Rd} \leq F_{t(2+3),fc,Rd} - F_{t2,Rd}$

**End plate in bending**
$F_{t3,Rd} \leq F_{t(2+3),wc,Rd} - F_{t2,Rd}$

| | | | |
|---|---|---|---|
| **Resistance of second and third rows in tension** |  |  |  |
| **Resistance of three rows in tension** | **Beam web in tension** $F_{t3,Rd} \leq F_{t(2+3),wb,Rd} - F_{t2,Rd}$ | **Column flange in bending** $F_{t3,Rd} \leq F_{t(1+2+3),fc,Rd} - F_{t1,Rd} - F_{t2,Rd}$ | **Column web in tension** $F_{t3,Rd} \leq F_{t(1+2+3),wc,Rd} - F_{t1,Rd} - F_{t2,Rd}$ |

The impact of interaction between axial force and bending moment on resistance and stiffness is neglected up to the value *0.1* of beam resistance in compression ($0.1N_{pl.Rd}$). The interaction diagram should be done for greater forces and the reduction similar to the column bases is necessary.

## Bending stiffness

The tangential stiffness shows the dependency of bending moment on connection rotation

$$S_j = M/\phi$$

The initial stiffness $S_{j,ini}$ is calculated using a sum of deformations of particular components. The tangential stiffness $S_j$ is calculated for individual values of bending moments on non-linear part of moment-deformation diagram. The elastic deformation is calculated using formula

$$\delta_i = \frac{F_i}{k_i E}$$

| | | |
|---|---|---|
| Where is | $F_i$ | • The force in component |
| | $E$ | • The modulus of elasticity |

The rotation is calculated using following formula for the force $F_i$ with inner lever arm $z$

$$\phi_j = \frac{\sum_i \delta_i}{z}$$

The initial value of stiffness is calculated using expression:

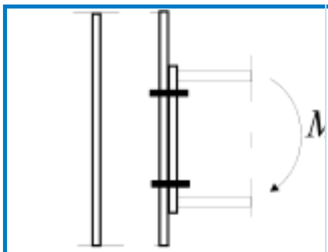
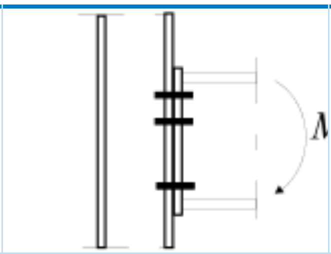
$$S_{j,ini} = \frac{M_j}{\phi_j} = \frac{F_i z}{\frac{\sum \delta_i}{z}} = \frac{F_i z^2}{\frac{F_i}{E}\sum \frac{1}{k_i}} = \frac{E z^2}{\sum \frac{1}{k_i}} \rightarrow \frac{E z^2}{\mu \sum \frac{1}{k_i}}$$

The expression can be modified for non-linear part of the curve with the help of factor $\mu$. The factor describes the difference between initial and tangential stiffness. The value of the factor is *1.0* in the linear part and is increasing with increasing value of bending moment.

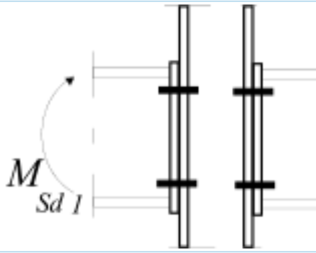
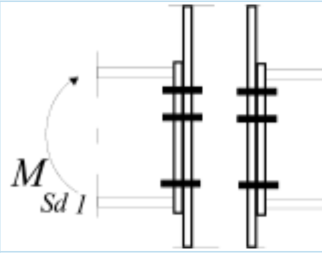
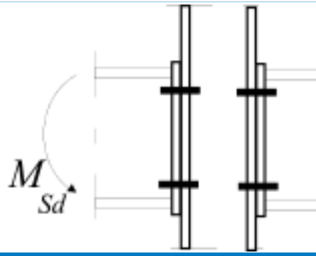
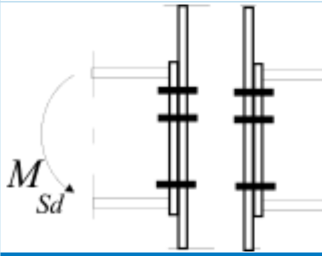
$$\mu = \frac{S_{j,ini}}{S_j} = \left(\kappa \frac{M_{Ed}}{M_{j,Rd}}\right)^{\psi} \geq 1.0$$

| | | |
|---|---|---|
| Where is: | $z$ | • The inner lever arm |
| | $\psi$ | • The curve factor (*2.7* for connections with end plate) |
| | $\kappa$ | • The factor of beginning of non-linear curve (value *1.5*) |
| | $M_{ed}$ | • The design value of bending moment |

Components for calculation of stiffness:

| | | | |
|---|---|---|---|
|  | **One row in tension:** $k_1, k_2, k_3, k_4, k_5, k_{10}$ |  | **Two and more rows in tension:** $k_1, k_2, k_{eq}$ ($k_3, k_4, k_5, k_{10}$) |

| | | | |
|---|---|---|---|
|  | **One row in tension:**<br>*$k_1$, $k_2$, $k_3$, $k_4$, $k_5$, $k_{10}$* |  | **Two and more rows in tension:**<br>*$k_1$, $k_2$, $k_{eq}$ ($k_3$, $k_4$, $k_5$, $k_{10}$)* |
|  | **One row in tension:**<br>*$k_2$, $k_3$, $k_4$, $k_5$, $k_{10}$* |  | **Two and more rows in tension:**<br>*$k_2$, $k_{eq}$ ($k_3$, $k_4$, $k_5$, $k_{10}$)* |

**Note:** Marking corresponds to EN 1993-1-8.

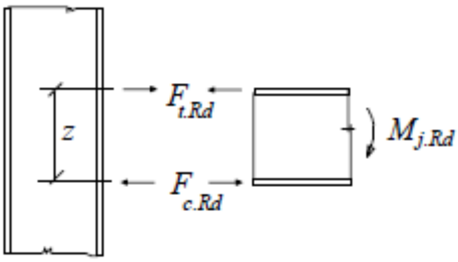
| | | |
|---|---|---|
| Where is: | $k_1$ | • The unstiffened column web in shear |
| | $k_2$ | • The stiffened column web in tension |
| | $k_3$ | • The unstiffened column web in tension |
| | $k_4$ | • The column flange in bending |
| | $k_5$ | • The end plate in bending |
| | $k_6$ | • The angle in bending |
| | $k_7$ | • The flange and web of beam in compression |
| | $k_8$ | • The beam web in tension |
| | $k_9$ | • The plate in tension or compression |
| | $k_{10}$ | • The bolts row in tension |

# Welded connecton to column

## Bending resistance

Binding resistance is calculated using formula

$$M_{j,Rd} = F_{t,Rd} z$$




*Inner lever arm for welded connection*

The tensile resistance $F_{t,Rd}$ is considered as a minimum of following values:

- Column flange in bending $F_{t,fc,Rd}$
- Column web in tension $F_{t,wc,Rd}$

The resulting resistance is limited by the resistance of part in compression and the resistance in shear:

- The column web in shear $V_{wp,Rd}/\beta$
- The column web in compression $F_{c,wc,Rd}$

## Bending stiffness

The initial value of stiffness is calculated using expression:

$$S_{j,ini} = \frac{M_j}{\phi_j} = \frac{E z^2}{\mu \sum \frac{1}{k_i}}$$

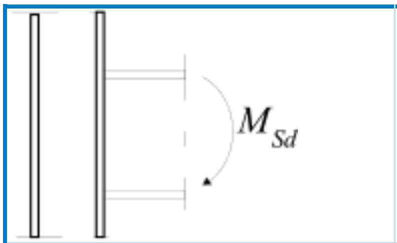
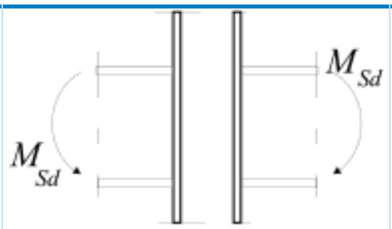
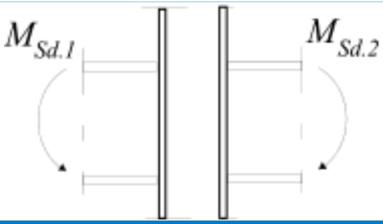
 

The expression can be modified for non-linear part of the curve with the help of factor $\mu$. The factor describes the difference between initial and tangential stiffness. The value of the factor is *1.0* in the linear part and is increasing with increasing value of bending moment.

$$\mu = \frac{S_{j,ini}}{S_j} = \left(\kappa \frac{M_{Ed}}{M_{j,Rd}}\right)^{\psi} \geq 1.0$$

| | | | |
|---|---|---|---|
| Where is: | ***z*** | • | The inner lever arm |
| | ***ψ*** | • | The curve factor (*2.7* for connections with end plate) |
| | ***κ*** | • | The factor of beginning of non-linear curve (value *1.5*) |
| | ***M<sub></sub>ed*** | • | The design value of bending moment |

Components for calculation of stiffness:

| | | | |
|---|---|---|---|
|   | *$k_1$, $k_2$, $k_3$* |   | *$k_2$, $k_3$* |
|   | *$k_1$, $k_2$, $k_3$* | | |

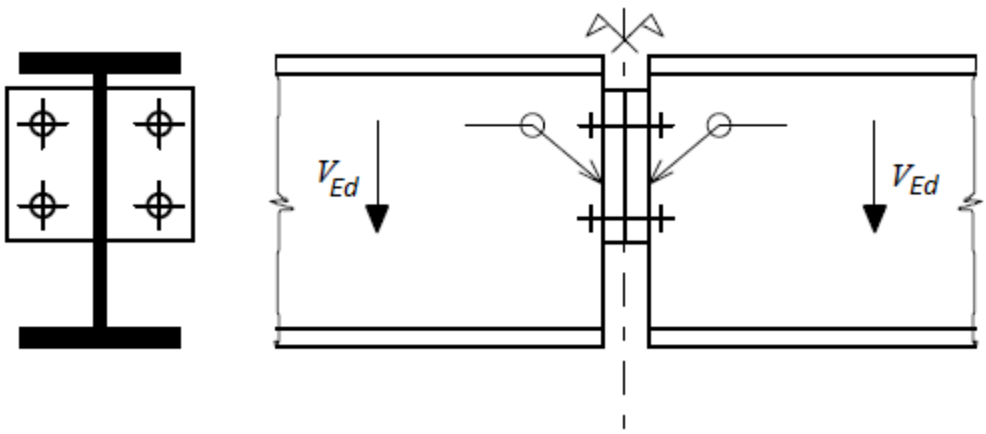
**Note:** Marking corresponds to EN 1993-1-8.

| | | | |
|---|---|---|---|
| Where is: | ***$k_1$*** | • | The unstiffened column web in shear |
| | ***$k_2$*** | • | The stiffened column web in tension |
| | ***$k_3$*** | • | The unstiffened column web in tension |

Unstiffened welded connections are considered as ductile, if the required rotation is smaller than $\phi_{CD.min}$ = *0,015* (EN 1993-1-8). Limiting value of rotation may be calculated for welded connections with stiffened web using formula

$$\phi_{Cd,min} = 0.025 h_c / h_b$$

## Pinned beam splice

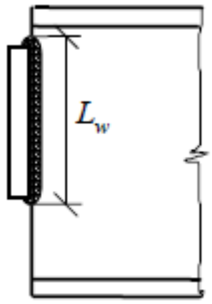



*Detail view*

The bearing resistance is based on the resistances of following components:

- The beam web in shear $F_{vbw,Rd}$ (usually the decisive component)

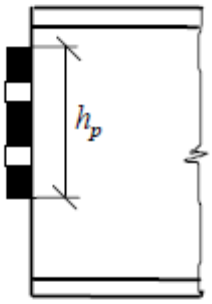
 




*The shear length*

- The shear resistance in thread of *n* bolts $nF_{b,v,Rd}$
- The bearing resistance of bolts in end plate $nF_{b,b,Rd}$ (the resistance may be calculated as a sum of resistances of inner and edge bolts)
- The shear resistance of welds $F_{w,v,Rd}$
- The shear resistance of end plate $V_{p,v,Rd}$ in the cross-section with gaps. This can be neglected if $A_{v,net}/A_v > f_y/f_u$ . In the end plate cross-section is considered only a half of the force in a beam.

The bending resistance and the interaction with shear may be limiting for long end plates.



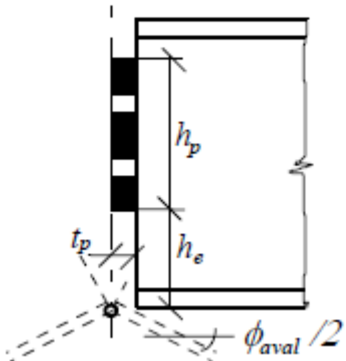

*Cross-section of end plate with gaps*

The bearing capacity of the connection is given by the minimum

$$F_{Ed} = min\left\{F_{vbw,Rd}; nF_{b,v,Rd}; nF_{b,b,Rd}; F_{w,v,Rd}; F_{p,v,Rd}\right\}$$

The connection may be considered as a pinned one if the beam rotation does not cause the contact of bottom flange with a second beam.

$$\phi_{aval} = 2t_p/h_e \geq \phi_{req} = \frac{\gamma_{M1} L_b^3}{12EI_b}$$




*Rotation of beam end*

The remaining tensile resistance for the interaction of tension and shear shall be able to transfer full plastic moment resistance of connected end plate. Therefore, the thickness of the plate should be limited

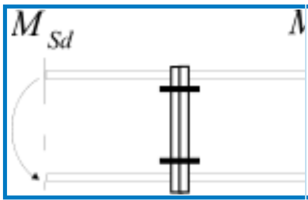
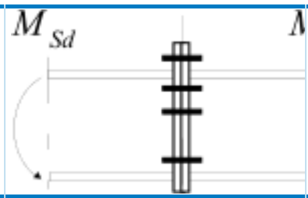
$$t \leq 0.36d\sqrt{f_{ub}/f_y}$$

Where *d* is bolts diameter. Welds are designed in that way, that they are able to transfer full plastic resistance moment of the plate in tension.

## Stiff beam splice

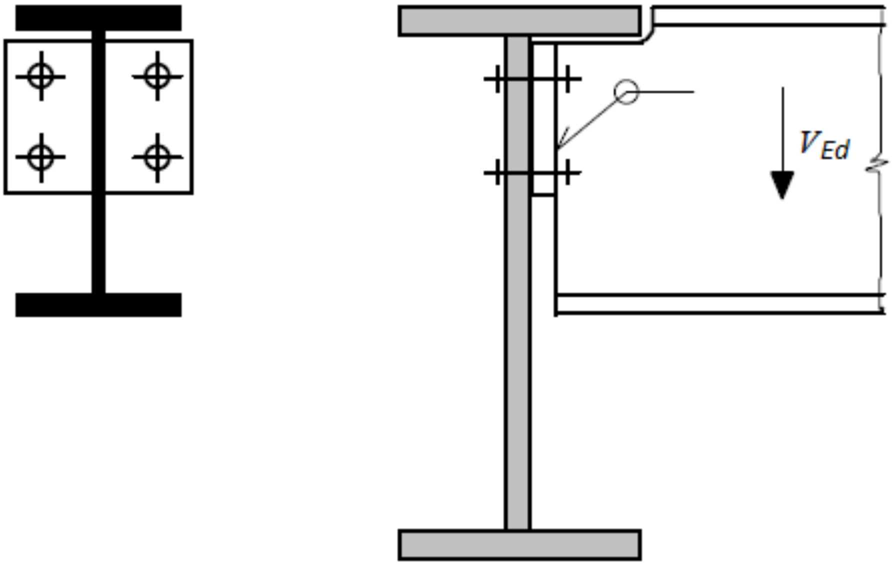
The bending resistance and stiffness of the splicing detail are calculated in the same way as for the beam - column joint. The difference is, that the column components are removed.

| | | | |
|---|---|---|---|
|  | **One bolts row in tension**<br>$k_5$ on left, $k_5$ on right, $k_{10}$ |  | **Two or more bolts rows**<br>$k_{eq}$ ($k_5$, $k_5$, $k_{10}$) |

**Note:** The marking respects the convention used in EN 1993-1-8

| | | |
|---|---|---|
| Where is: | $\boldsymbol{k_5}$ | • The end plate in bending |
| | $\boldsymbol{k_{10}}$ | • The bolts row in tension |

## Pinned end plate to beam



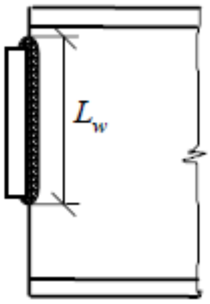

*Pinned end plate to beam detail*

The bearing capacity of the connection is given by the minimum

$$F_{Ed} = min\left\{F_{vbw,Rd}; nF_{b,v,Rd}; nF_{b,b,Rd}; F_{w,v,Rd}; F_{p,v,Rd}\right\}$$

of following vales:

- The shear resistance of beam web $F_{vbw,Rd}$ (usually critical component for this type of connection)



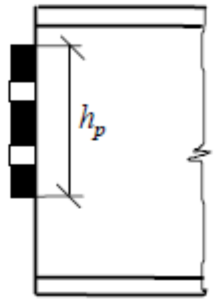

*The shear length of beam web*

- The bending resistance of beam web with gaps $M_{bw,v,Rd}$ and shear resistance $V_{bw,v,Rd}$ including an interaction of inner forces
- The resistance of $n$ bolts $nF_{b,v,Rd}$
- The bearing resistance of bolts in end plate $nF_{b,b,Rd}$ (can be calculated as a sum of bearing resistances of end and intermediate bolts for materials with sufficient ductility)
- The shear resistance of welds $F_{w,v,Rd}$

- The shear resistance of end plate $V_{p,v,Rd}$ in the cross-section with gaps. This can be neglected if $A_{v,net}/A_v > f_y/f_u$. In the end plate cross-section is considered only a half of the force in a beam.

The bending resistance and the interaction with shear may be limiting for long end plates.

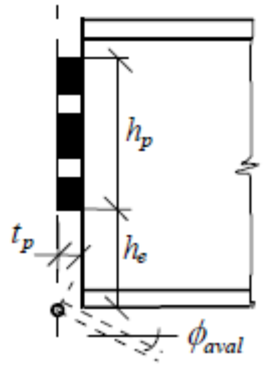



*Cross-section of end plate with gaps*

The connection may be considered as a pinned one if the beam rotation does not cause the contact of bottom flange with a column The rotation capacity is greater than rotation of simple beam

$$\phi_{aval} = t_p / h_e \geq \phi_{req} = \frac{\gamma_{M1} L_b^3}{24 E I_b}$$

| | | |
|---|---|---|
| Where is: | $L_b$ | • The beam span |
| | $E$ | • The modulus of elasticity |
| | $I_b$ | • The second moment of inertia |




*Rotation of beam end*

The remaining tensile resistance for the interaction of tension and shear shall be able to transfer full plastic moment resistance of connected end plate. Therefore, the thickness of the plate should be limited

$$t \leq 0.36 d \sqrt{f_{ub} / f_y}$$

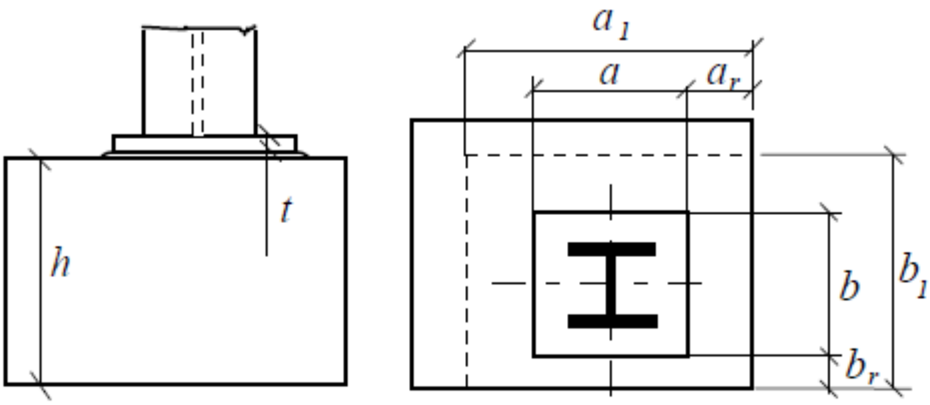
Where *d* is bolts diameter. Welds are designed in that way, that they are able to transfer full plastic resistance moment of the plate in tension.

## Base plate in compression

The resistance of concrete in compression is defined according to EN 1993-1-1 by the following formula

$$f_j = \beta_j k_j f_{cd}$$




*Base dimensions*

Where is:

- **$\beta_j$** • The factor, that is applied in case that the characteristic strength of grouting is higher than *20%* of the characteristic strength of concrete and the grouting thickness is smaller than 20% of minimum base dimension. The value of the factor is *2/3*.
- **$f_{cd}$** • The design value of concrete strength in compression
- **$k_j$** • The concentration factor

The factor $k_j$ is calculated using expression

$$k_j = \sqrt{\frac{a_1 b_1}{ab}}$$

Where is:

- **a, b** • The dimensions of base
- **$a_1$, $b_1$** • The effective dimensions
- **h** • The height of a base

The values $a_1$ and $b_1$ are calculated using following expressions

$$b_1 = min \begin{Bmatrix} b + 2b_r \\ 5b \\ b + h \\ 5a_1 \end{Bmatrix}, b_1 \geq b$$

$$a_1 = min \begin{Bmatrix} a + 2a_r \\ 5a \\ a + h \\ 5b_1 \end{Bmatrix}, a_1 \geq a$$

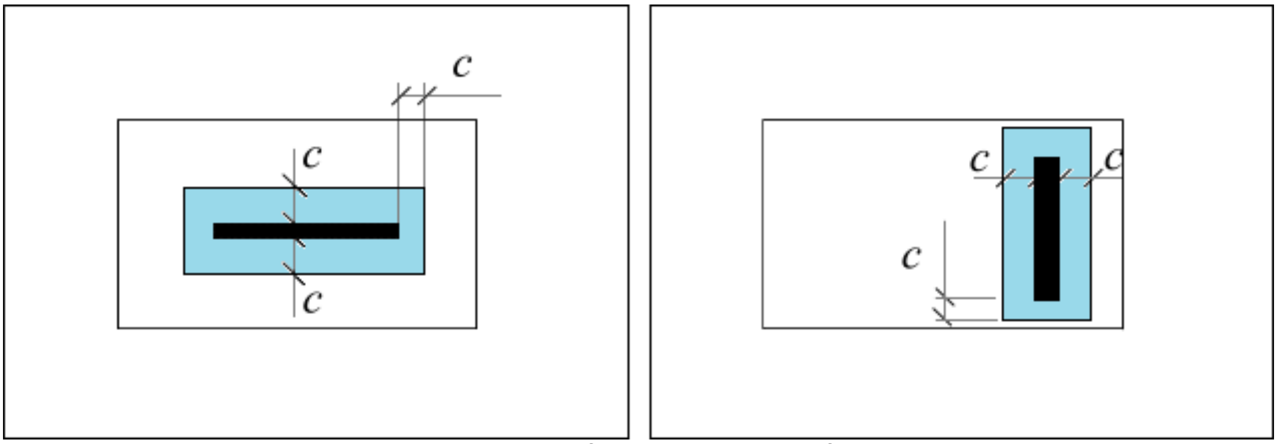
The assumption is, that the stress is distributed uniformly under the active area. This area consists of the column cross-section and surrounding are with maximum distance *c* from the edge of the cross-section. The thickness *c* is given by formula

$$c = t\sqrt{\frac{f_y}{3\gamma_{M0} f_j}}$$

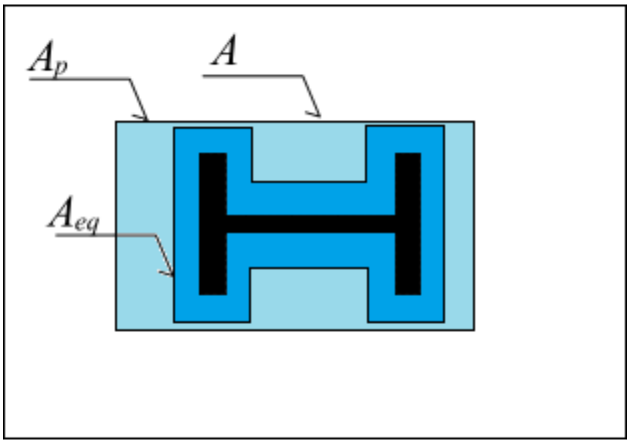
Where is:

- **t** • The thickness of the base plate
- **$f_y$** • The yield strength
- **$f_j$** • The concrete resistance in compression
- **$\gamma_{M0}$** • The partial safety factor




The active areas for column web and flange




The active area under the base plate

## Stiffness

The deformation between base plate and concrete surface for force *F* is calculated using expression

$$\delta_r = \frac{F \alpha a_r}{E_c A_r}$$

where the shape factor *α* depends on material characteristics. Following formula is used for concrete foundation and steel base plate according to (Wald, Sokol; 1996)

$$\delta_r = \frac{0.85F}{E_c \sqrt{L a_r}}$$

| | | |
|---|---|---|
| Where is: | $\delta_r$ | • The teformation under the stiff base plate |
| | $\alpha$ | • The shape factor |
| | $E_c$ | • The modulus of elasticity of concrete |
| | $L$ | • The plate length |
| | $a_r$ | • The width of effective base plate |

The stiffness is calculated in accordance with EN 1993-1-8 using formula

$$k_c = \frac{F}{\delta E} = \frac{E_c \sqrt{a_{eq,el} L}}{1.5 \cdot 0.85E} = \frac{E_c \sqrt{a_{eq,el} L}}{1.275E}$$

The effective width of T-stub $a_r$ can be replaced for elastic deformations by the following width (ed.Weynand; 1999):

$$a_{eq,el} = t_w + 2.5t \approx a_{eq,str} = t_w + 2t\sqrt{\frac{f_y}{3 f_j \gamma_{M0}}}$$

| | | |
|---|---|---|
| Where is: | $t_w$ | • The web thickness of effective T-stub |
| | $t$ | • The thickness of base plate |

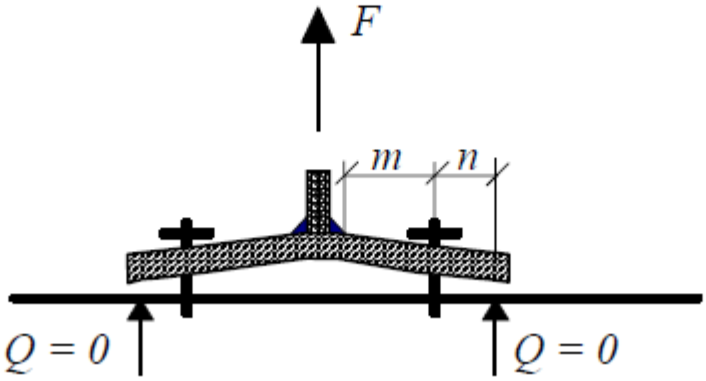
## References

Wald F., Sokol Z.: *Proposal of the Stiffness Design Model of the Column Bases*, Connection v Steel Structures II, Behaviour, Strength, and Design, ed. Bjorhovde, Colson, Zandonini, Pergamon Press, London 1996, p. 237 - 249, ISBN 0-08-042821-5.

COST C1: *Column Bases in Steel Building Frames*, ed. K. Weynand, EC, Brussels 1999.

# Base plate in bending

The behaviour of base plate and bolts is described with the help of the equivalent T-stub. The differences in the behaviour of base plate and end plate (Wald; 1993) were considered in the component method. The base plate is thick, as it is designed for the transfer of compressive forces into the concrete foundation. The bolts are longer, therefore, the stiffness is reduced. The impact of washers and nuts sizes is more significant. The impact on the overall stiffness may be neglected (Sokol, Wald; 1997).




*T-stub without any contact with concrete foundation*

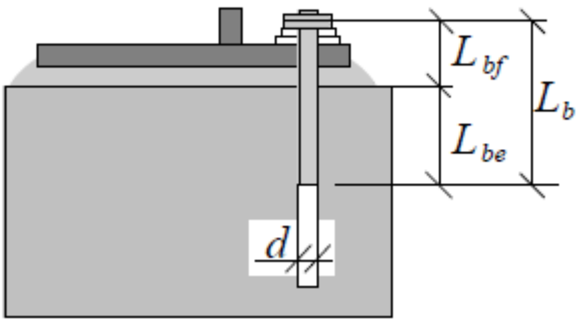
Simplified assumptions in accordance with (Weynand; 1999) are used for calculation of stiffness and resistance of bolts in tension and base plate in bending. The limiting value for the cases with prying and cases without any contact between plate and foundation is $n = 1{,}25m$ .

$$L_{b,min} = \frac{8.82 m^3 A_s}{l_{eff} t^3} \mathop{}^{<}_{>} L_b$$

Where is: ***A*$_s$** • The bolt area in thread

***L*$_b$** • The effective length of bolt (figure below)

The effective length of concreted bolts $L_b$ consists of the free part above the foundation surface $L_{bf}$ and the effective part under the surface $L_{be} \approx 8d$ . It means $L_b = L_{bf} + L_{be}$ (Wald; 1993). The prying does not occur for bolts lengths $L_b$ longer than $L_{b.lim}$.

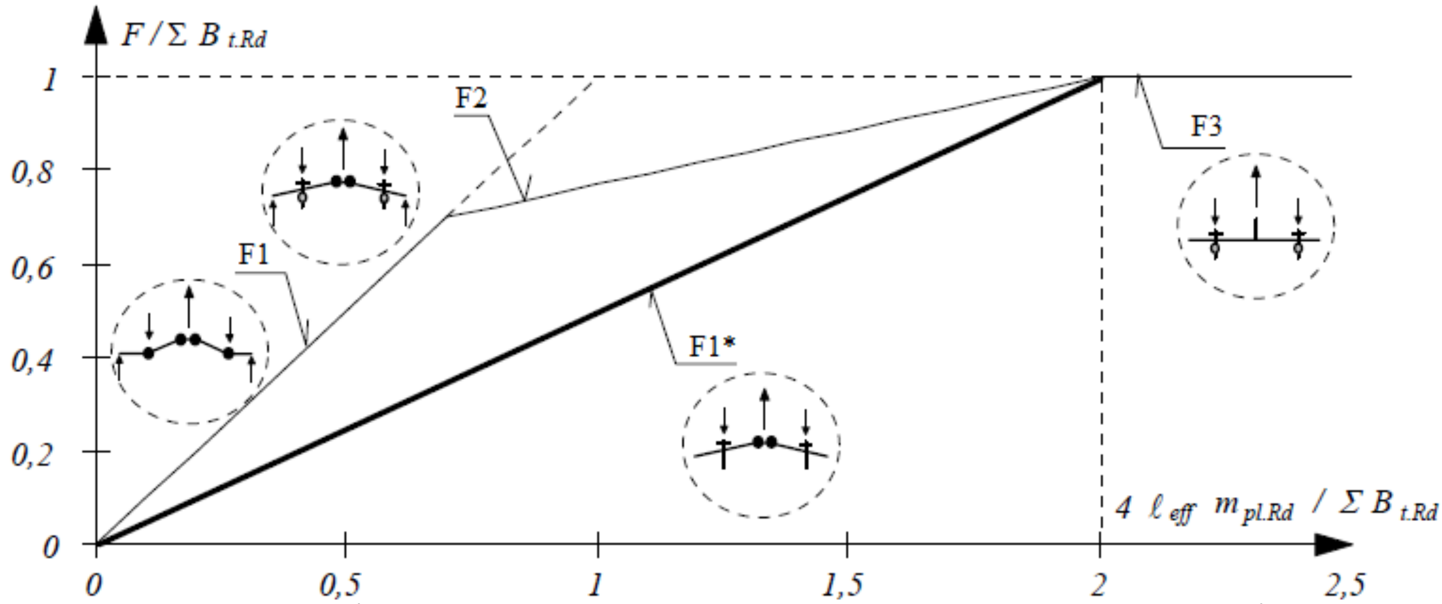



*The effective length of bolt*

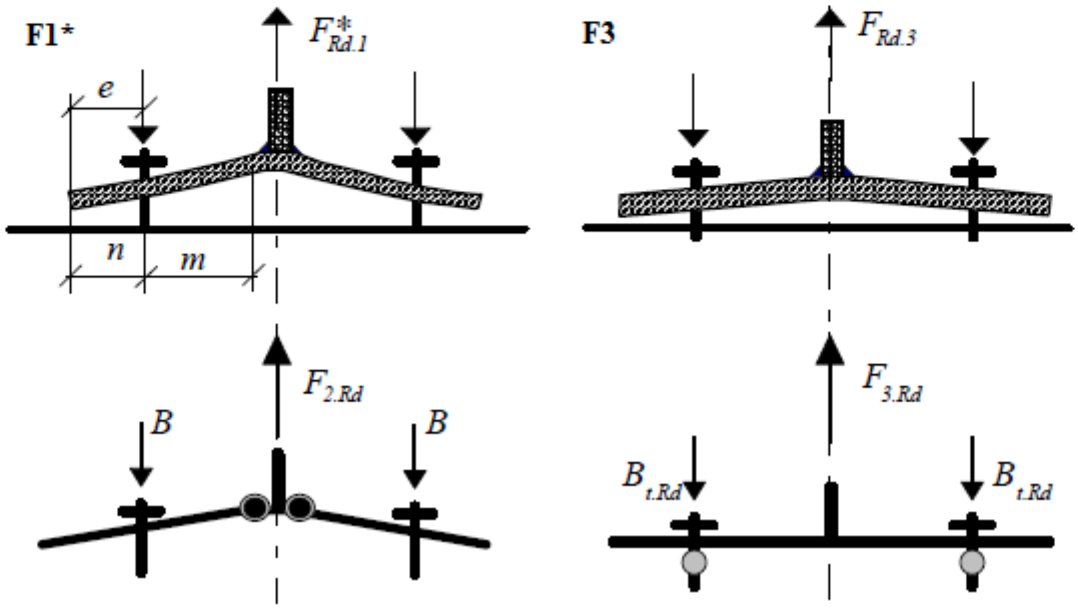
The resistance of effective T-stub is the minimum of three plastic failures. The failure *F1* corresponds to the failure of T-stub with thin base plate and long bolts. The mechanism with four plastic hinges appears in plate in this case.

The failure *F3* corresponds to the thick plate with weak bolts. The failure appears in bolts in this case. The failure *F2* is the intermediate case with two plastic hinges and failure in bolts.




Failure modes ($B_{t.Rd}$ is bolt resistance in tension and $m_{pl.Rd}$ is the plastic resistance moment)




Failures F1* (no contact with concrete foundation) and F3 (bolts)

For cases without prying, the failure *F1** appears instead of failure modes *F1* or *F2*. As the contact between plate and foundation usually appears after the appearance of plastic hinges, the failure modes *F1* or F2 appear. However, these modes are connected with significant deformations. Therefore, the failure mode with two hinges is checked. The resistance is given by the formula

$$F^*_{Rd,1} = \frac{2l_{eff,1}m_{pl,Rd}}{m}$$

The method of linear hinges is used for the calculation of the effective length of T-stub $l_{eff.1}$.

## Stiffness

The stiffness factor of the equivalent T-stub without any contact with foundation is calculated separately for plate (thickness $t$) and bolts:

$$k_p = \frac{0.425 l_{eff} t^3}{m^3}$$

$$k_b = 2.0\frac{A_s}{L_b}$$

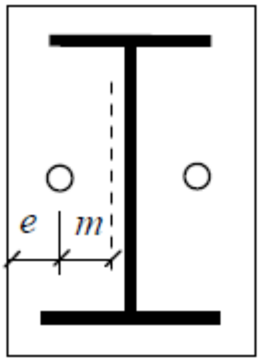
The stiffness of complete T-stub is calculated as a sum of these components. For cases with prying, following expressions are used in accordance with EN 1993-1-8:

$$k_p = \frac{0.85 l_{eff} t^3}{m^3}$$
$$k_b = 1.6 \frac{A_s}{L_b}$$

The effective length $l_{eff}$ of equivalent T-stub is equal to0 $l_{eff,1}$ or $l_{eff,2}$, according to the failure mode for ultimate limit state. $l_{eff}$ is calculated using the method of linear hinges:

Base plates with bolts between flanges:




*Bolts between flanges*

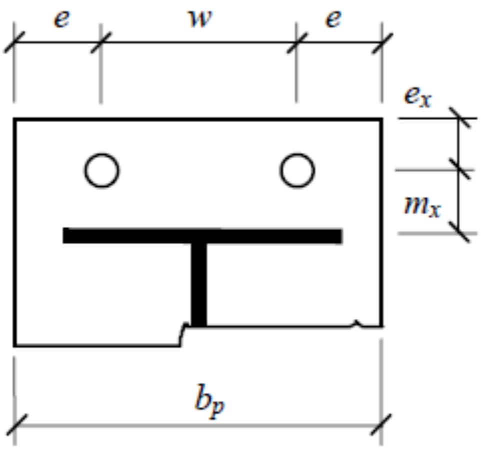
Values $l_{eff,1}$ and $l_{eff,2}$ are given by following expressions for cases with prying effect:

$$l_1 = 2\alpha m - (4m + 1.25e)$$
$$l_2 = 2\pi m$$
$$l_{eff,1} = min(l_1; l_2)$$
$$l_{eff,2} = l_1$$

Values $l_{eff,1}$ and $l_{eff,2}$ are given by following expressions for cases without prying effect:

$$l_1 = 2\alpha m - (4m + 1.25e)$$
$$l_2 = 4\pi m$$
$$l_{eff,1} = min(l_1; l_2)$$
$$l_{eff,2} = l_1$$

Base plates with bolts out of flanges:




*Bolts out of flanges:*

Values $l_{eff,1}$ and $l_{eff,2}$ are given by following expressions for cases with prying effect:

$$l_1 = 4m_x + 1.25e_x)$$
$$l_2 = 2\pi m_x$$
$$l_3 = 0.5b_p$$
$$l_4 = 0.5w + 2m_x + 0.625e_x$$
$$l_5 = e + 2m_x + 0.625e_x$$
$$l_6 = \pi m_x + 2e$$
$$l_7 = \pi m_x + p$$
$$l_{eff,1} = min(l_1; l_2; l_3; l_4; l_5; l_6; l_7)$$
$$l_{eff,2} = min(l_1; l_3; l_4; l_5)$$

Values $l_{eff,1}$ and $l_{eff,2}$ are given by following expressions for cases without prying effect:

$$l_1 = 4m_x + 1.25e_x)$$
$$l_2 = 4\pi m_x$$
$$l_3 = 0.5b_p$$
$$l_4 = 0.5w + 2m_x + 0.625e_x$$
$$l_5 = e + 2m_x + 0.625e_x$$
$$l_6 = 2\pi m_x + 4e$$
$$l_7 = 2\pi m_x + 2p$$
$$l_{eff,1} = min(l_1; l_2; l_3; l_4; l_5; l_6; l_7)$$
$$l_{eff,2} = min(l_1; l_3; l_4; l_5)$$

### References

Wald F., Obata M., Matsuura S., Goto Y.: *Prying of Anchor Bolts* , Nagoya University Report, Nagoya 1993, pp. 241-249.

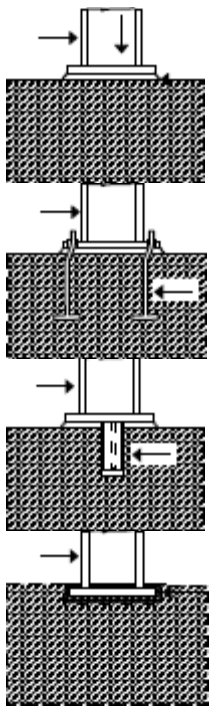
*Column Bases in Steel Building Frames* , ed. K. Weynand, EC, Brussels 1999.

Sokol Z., Wald F.: *Experiments with T-stubs in Tension and Compression* , Research Report, CVUT, Praha 1997.

## Base plate in shear

The shear forces between base plate and foundation aren't verified in the software. Shear forces may be transferred by:

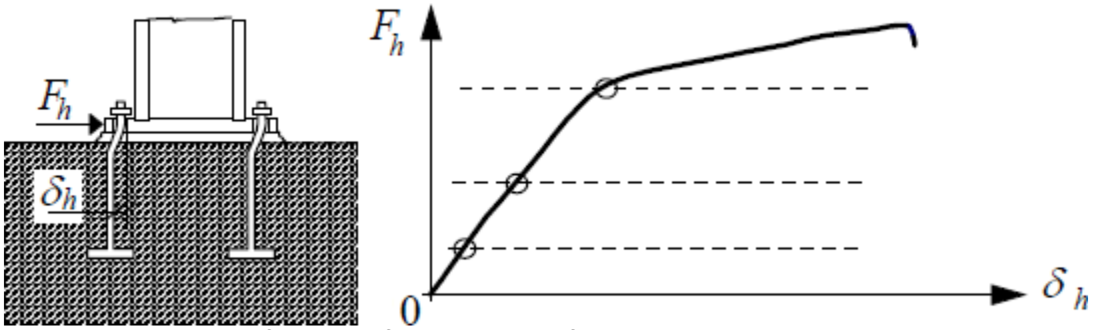
- **The friction between base plate, grouting and foundation**
- **The bolts (bolts in shear and bending)**
- **The additional steel piece**
- **The edge of base plate**



The value of friction factor is recommended as $\mu=0.4$ (CEB, 1997), combined with the partial safety factor $\gamma= 1,5$ . The friction may be increased due to pretension of bolts.

The anchoring bolts may be verified in accordance with (CEB, 1997). The interaction of bending and shear resistance of anchoring bolts is analysed in this case.



*Deformation-force diagram of anchoring bolts in shear*

The tensile failure appears after significant deformation (Weynand, 1999; Bowman 1989). The resistance may be calculated as reduced bolts resistance in tension, that is described in the same way as shear resistance:

$$F_{v,Rd} = \frac{\alpha f_{ub} A_s}{\gamma_{Mb}}$$

Where is:

| | | |
|---|---|---|
| $f_{ub}$ | • | The ultimate strength of bolts |
| $\gamma_{mb}$ | • | The equivalent length of anchoring bolt |

The bolts *4.6* have $\alpha = 0{,}375$, the bolts *5.6* have $\alpha = 0{,}250$. This expression is verified for grouting thickness till *60mm* (Bowman 1989). The shear resistance of bolts in foundation depends on the distance from base end and should be checked separately. Usually, bolts in tension and interaction of stresses are not considered.

## References

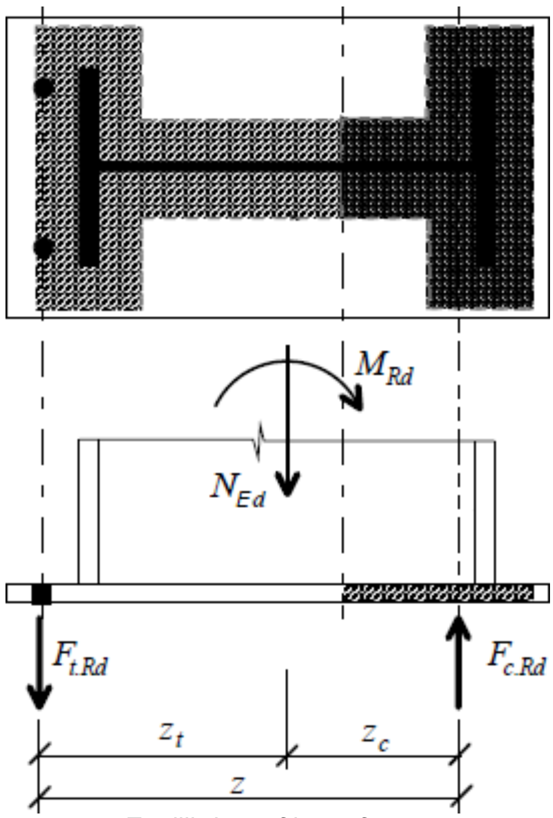
CEB: *Design of Fastenings in Concrete*, Design guide, Thomas Telford Services Ltd, London 1997, s. 83, ISBN 0 7277 2558 0.

COST C1: *Column Bases in Steel Building Frames*, ed. K. Weynand, EC, Brussels 1999

Bowman L P, Gresnight A. M., Romeijn, A: *Reserch into the connection of steel base plate to concrete foundation*, holandsky, Stevin Laboratory Report No.: 25.6.89.05/c6, Delft 1989

# Resistance of base

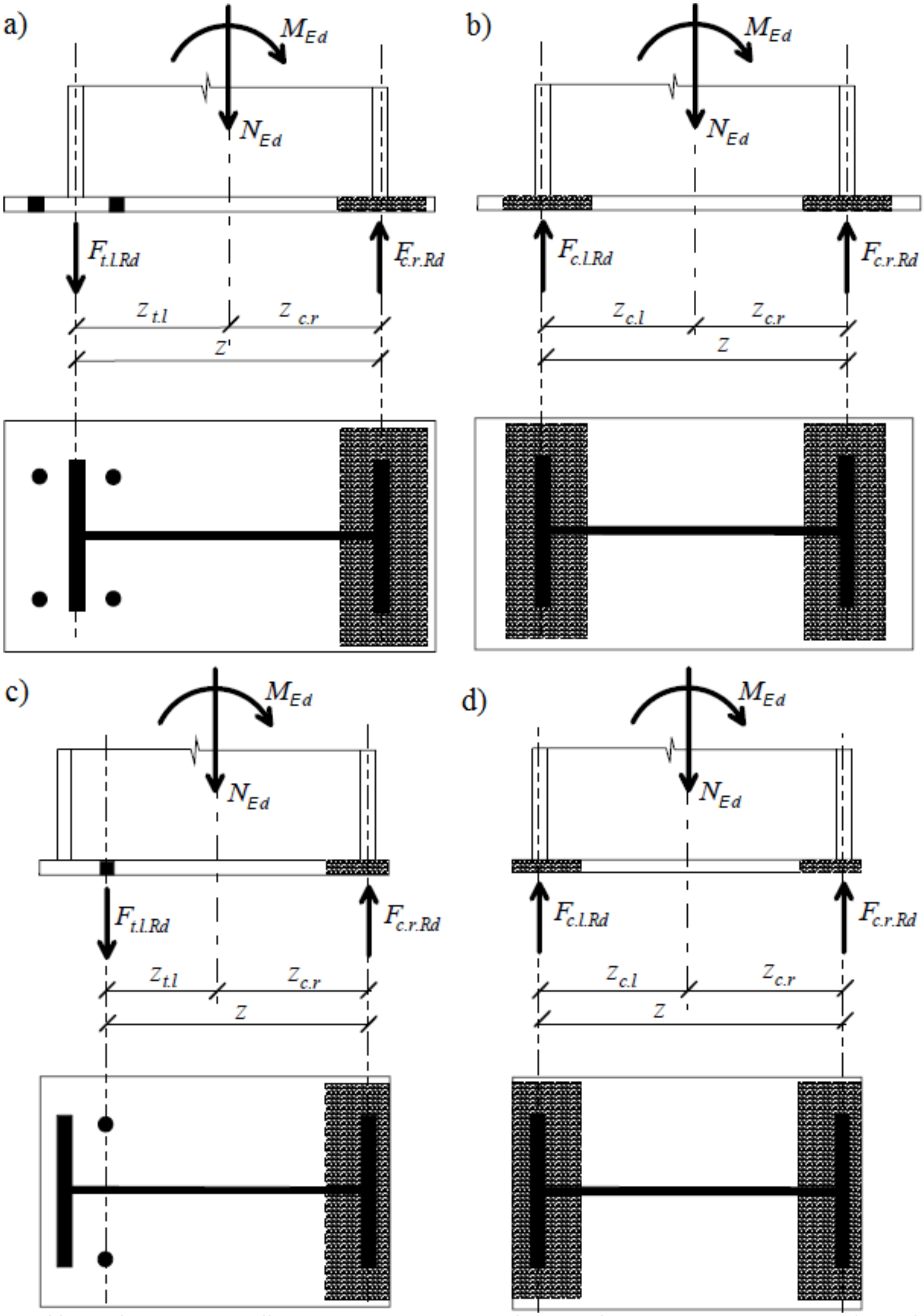
The calculation of base resistance loaded by axial force and bending moment is based on the equilibrium of forces in base plate. The tensile resistance is given by the bolts resistance $F_{t.Rd}$. With the help of this value, the position of neutral axis and bending resistance $M_{Rd}$ for axial force $N_{Ed}$ may be found. The plastic distribution of stress is considered



*Equilibrium of inner forces*

In case that only effective areas under flanges are considered (Steenhuis 1999) and only $N_{Ed}$, $M_{Ed}$ are used, the solution is easier, as the axis of compressive part is equal to flange axis. This assumption may be used also for short cantilevers, that are shorter than the effective width of plate. The tensile force acts in the bolts axis, the resultant is used for two rows of bolts.




*Equilibrium of forces for model with effective areas under column flanges; a) two bolts rows in tension, b) Two flanges in compression, c) one row and cantilever shorter than effective width, d) both flanges in compression and short cantilevers*

The resistances in tension $F_{t,l,Rd}$ and compression $F_{c,l,Rd}$, $F_{c,r,Rd}$ were found in previous chapters. The expression $e = M_{Ed}/N_{Ed} \geq z_{c,r}$ is valid for cases with bolts in tension. According to the options *a)* and *c)*, the base parts in tension and compression may be found using expressions:

$$\frac{M_{Sd}}{z} - \frac{N_{Sd} z_{c,r}}{z} = F_{t,1,Rd}$$

$$\frac{M_{Sd}}{z} + \frac{N_{Sd} z_{c,l}}{z} = F_{c,r,Rd}$$

The bending resistance of column $M_{Rd}$ for given $N_{Ed}$ may be calculated as the minimum of values $M_{Ed}$ in expressions above.

$$M_{Rd} = min \begin{Bmatrix} F_{t,l,Rd} z + z_{c,r} N_{Sd} \\ F_{c,r,Rd} z - z_{t,t} N_{Sd} \end{Bmatrix}$$

If the eccentricity $e = M_{Ed} / N_{Ed} < z_{c,r}$ (options *b)* and *d)*), there is not any tensile force in bolts. Bending resistance is given by the following expression in these cases

$$M_{Rd} = min \begin{Bmatrix} F_{c,l,Rd} z + z_{c,r} N_{Sd} \\ F_{c,r,Rd} z - z_{c,l} N_{Sd} \end{Bmatrix}$$

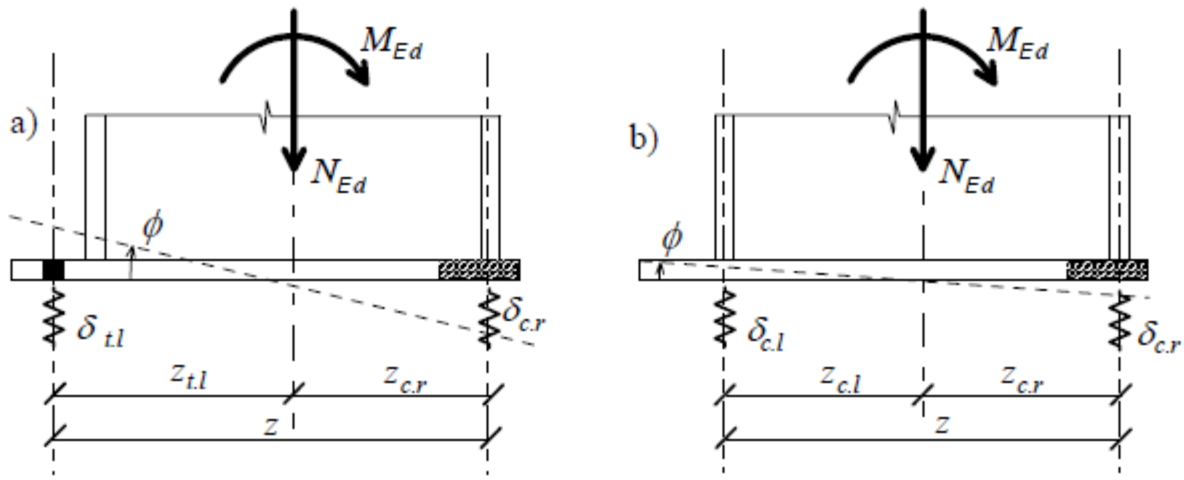
### References

Steenhuis C. M.: *Assembling procedure for base plates*, report 98-NOC-R0447 TNO, Delft 1999, p. 79.

## Bending stiffness

The initial stiffness is usually determined using proportional loading of the detail with constant eccentricity $e = M_{Ed}/N_{Ed} = const$. Sometimes, stiffness for constant axial force is used (Bowman, 1989).

The analysis model uses the simplification mentioned in previous chapters. This simplification deals with the centre of compressive stress under the column flanges. The dependency of deformations $\delta_t$, $\delta_c$ of particular components on inner forces is based on stiffness $k_t$ of part in tension and stiffness $k_c$ of part in compression. Following expressions are used:




*Analysis model*

$$\delta_{t,l} = \frac{\frac{M_{Ed}}{z} - \frac{N_{Ed} z_{c,r}}{z}}{E k_{t,l}} = \frac{M_{Ed} - N_{Ed} z_{c,r}}{E z k_{t,l}}$$

$$\delta_{t,l} = \frac{\frac{M_{Ed}}{z} + \frac{N_{Ed} z_{t,l}}{z}}{E k_{c,r}} = \frac{M_{Ed} + N_{Ed} z_{t,l}}{E z k_{c,r}}$$

The rotation of the foundation is calculated using expression:

$$\phi = \frac{\delta_{t,l} + \delta_{c,r}}{z} = \frac{1}{E z^2} \left( \frac{M_{Ed} - N_{Ed} z_{c,r}}{k_{t,l}} + \frac{M_{Ed} + N_{Ed} z_{t,l}}{k_{c,r}} \right)$$

Therefore, initial stiffness is

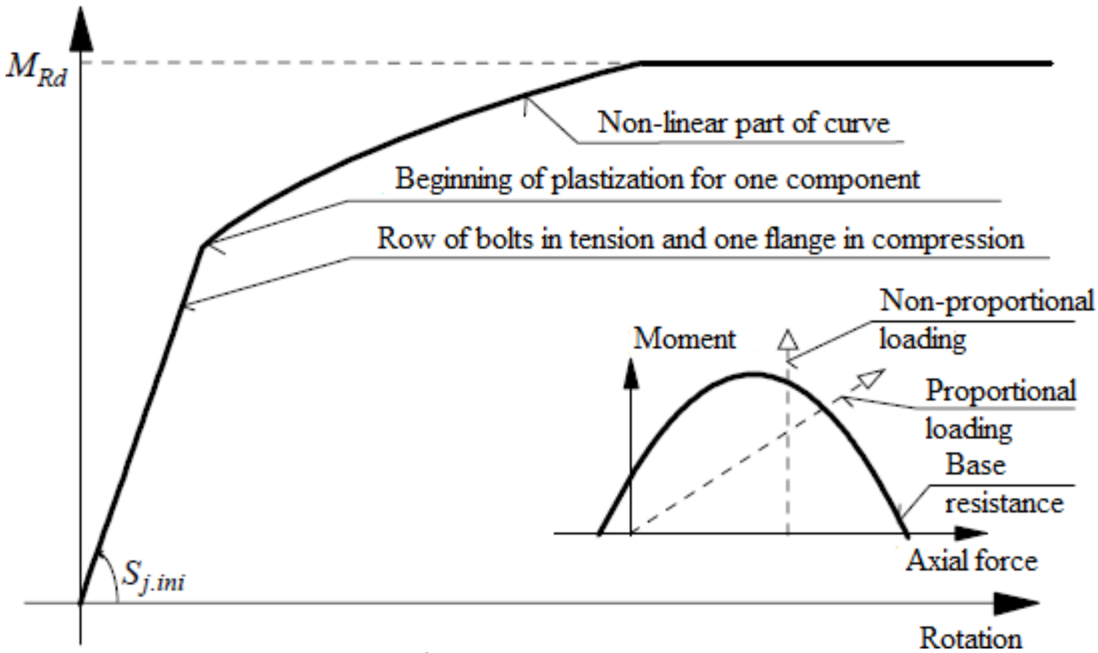
$$S_{j,ini} = \frac{Ez^2}{\frac{1}{k_{c,r}} + \frac{1}{k_{t,l}}} = \frac{Ez^2}{\sum \frac{1}{k}}$$

Non-linear part of moment-rotation dependency is based on the shape factor

$$\mu = \left(1.5\frac{M_{Ed}}{M_{Rd}}\right)^{2.7} \geq 1.0$$

The tangential bending stiffness is calculated using expression

$$S_j = \frac{Ez^2}{\mu \sum \frac{1}{k}}$$

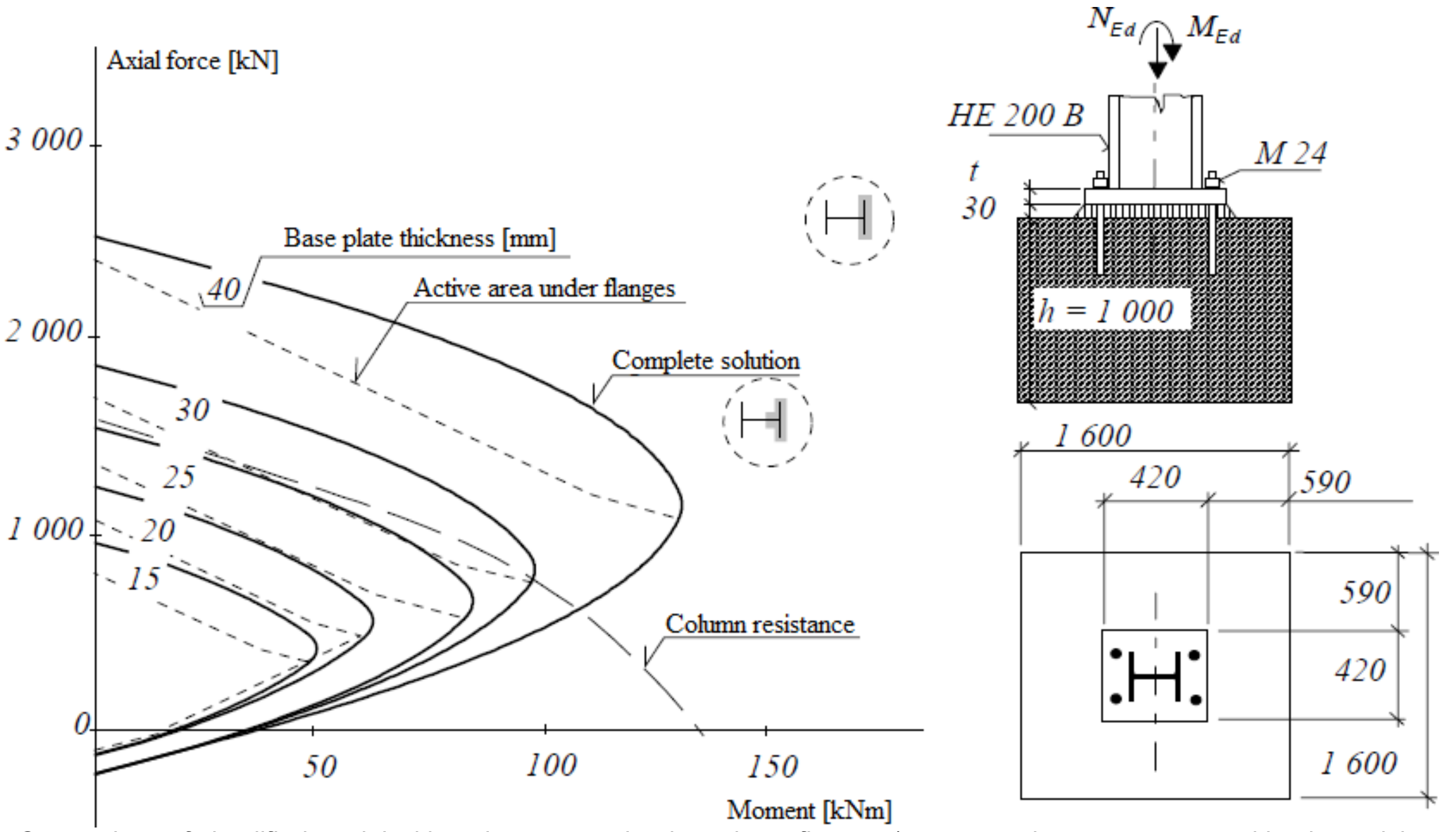



*Rotation-moment diagram for proportional loading with constant eccentricity*

The linear part of the curve represents bolts in tension and second flange in compression. Non-linear part describes plastic deformation of a component (end plate, bolts in tension or concrete in compression).

Following figure shows the effect of simplified model on resistance determination.




*Comparison of simplified model with active area under the column flanges (curvature changes are caused by the activity of bolts) and model with active area under the all cross-section*

## References

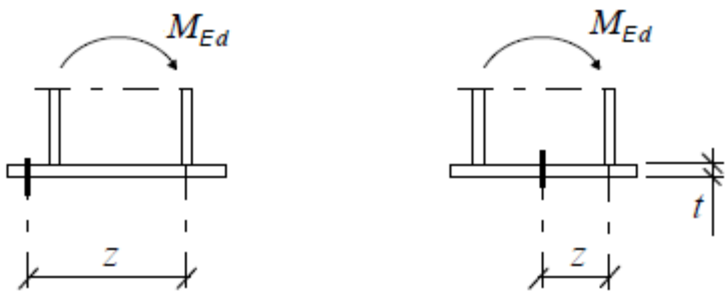
Bowman L P, Gresnight A. M., Romeijn, A: *Reserch into the connection of steel base plate to concrete foundation*, holandsky, Stevin Laboratory Report No.: 25.6.89.05/c6, Delft 1989.

# Preliminary prediction of stiffness

The preliminary prediction of detail stiffness may be beneficial for correct structural analysis of complete structure. Following expressions may be used. Initial stiffness of detail is calculated using expression

$$S_{j,ini,app} = \frac{Ez^2}{20}$$

where $z$ is the inner lever arm. It's a distance between the centre of compressive flange and bolts.




*Inner lever arm*

## References

Wald F., Bauduffe N., Sokol Z.: *Preliminary Prediction of the Column-Base Stiffness*, in Steel Structures of the 2000´s, Istanbul 2000

# Bending resistance of concreted column

These assumptions shall be fulfilled when designing detail with concreted detail:

Flange slenderness is limited by following expression:

$$b_c \leq 20t_{fc}$$

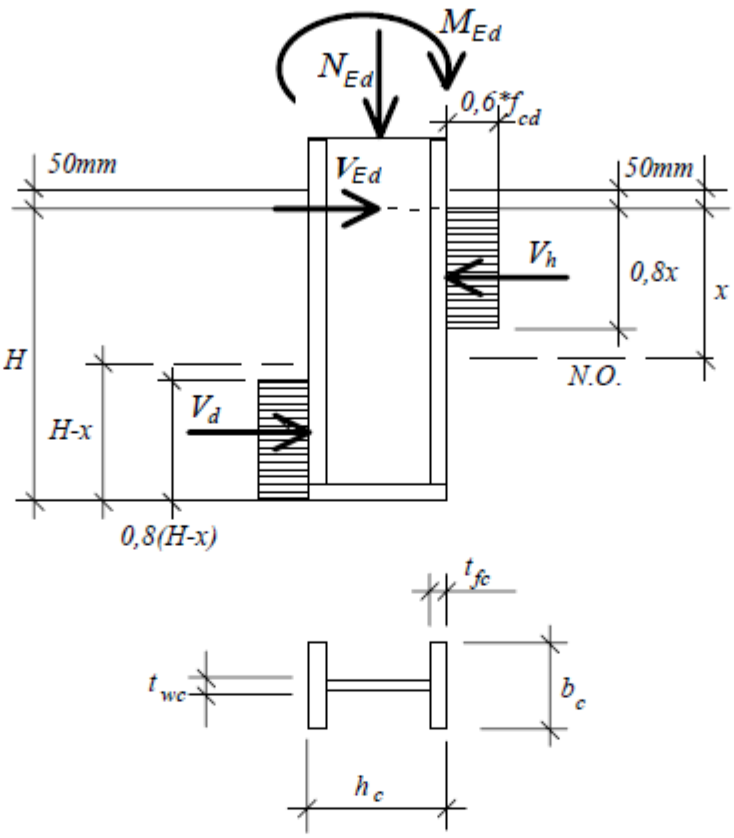
The concreting height shall be within following limits:

$$b_{eff} < H \leq 2b_{eff}$$

The effective flange width $b_{eff}$ is used for the design. This value take into the account the effect of inner edge of second flange for I and H profiles.

$$b_{eff} = min\{b_c + 0.5h_c; 2b_c\}$$

The analysis model uses following distribution of stresses in the detail:




*Stress distribution valid till H= 2beff.*

The horizontal forces $V_h$ and $V_d$ are given by expressions:

$$V_h = 0.8x0.6f_{cd}b_{eff}$$
$$V_d = 0.8(H-x)0.6f_{cd}b_{eff}$$

Horizontal equilibrium of forces:

$$V_{Ed} + V_d - V_h = 0$$
$$V_{Ed} + 0.8(H-x)0.6f_{cd}b_{eff} = 0$$

Therefore:

$$x = 1.0417\frac{V_{Ed} + 0.48b_{eff}f_{cd}H}{b_{eff}f_{cd}}$$

Moment equilibrium for the column end:

$$M_{Ed} + V_{Ed}h + V_d 0.4\left(H-x\right) - V_h\left(H-0.4x\right) = 0$$

The depth of concreting is calculated using following expression:

$$H = \frac{3.4722\left(0.5V_{Ed} + 0.1\sqrt{49V_{Ed}^2 + 57.6M_{Ed}b_{eff}f_{cd}}\right)}{b_{eff}f_{cd}}$$

## Shear resistance of concreted column

The distribution of shear forces along the column length is shown on the figure below. Following have have to be fulfilled:

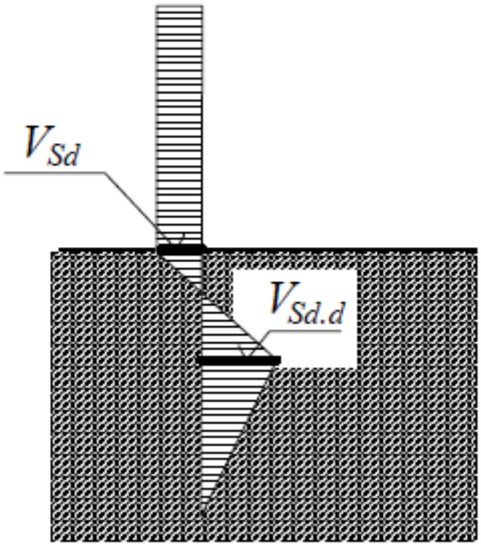
The column has to be able to transfer maximum force above the foundation $V_{Ed}$

$$V_{Ed} \leq V_{pl,Rd}$$

The column has to be able to transfer maximum shear force inside the foundation $V_{Ed,d}$

$$V_{Ed,d} \leq V_{wp,Rd}$$




*Shear distribution in detail*

$$V_{wp,Rd} = V_{pl,Rd} + V_{wp,c,Rd}$$

Where is:
- ***$V_{wp,Rd}$*** • The shear resistance of concreted steel column
- ***$V_{pl,Rd}$*** • The shear resistance of steel column
- ***$V_{wp,c,Rd}$*** • The shear resistance of concrete between flanges and column web

The maximum shear force within the foundation $V_{Ed.d}$ transferred by column web and concrete between flanges is given by the formula

$$V_{Ed,d} = V_{d1} - V_{d2} = 0.8\,(H - x)\,0.6 f_{ck}\,(2b - b_{eff})$$

The shear resistance of concrete between flanges $V_{wp,c,Rd}$ is calculated in that way, that it corresponds to the shear capacity of concreted columns.

$$V_{wp,c,Rd} = D_{c,Rd} sin\theta$$

$$D_{c,Rd} = \nu \frac{\alpha_d f_{ck}}{\gamma_c} A_{concr}$$

Where:

$$\alpha_d = 0.85$$

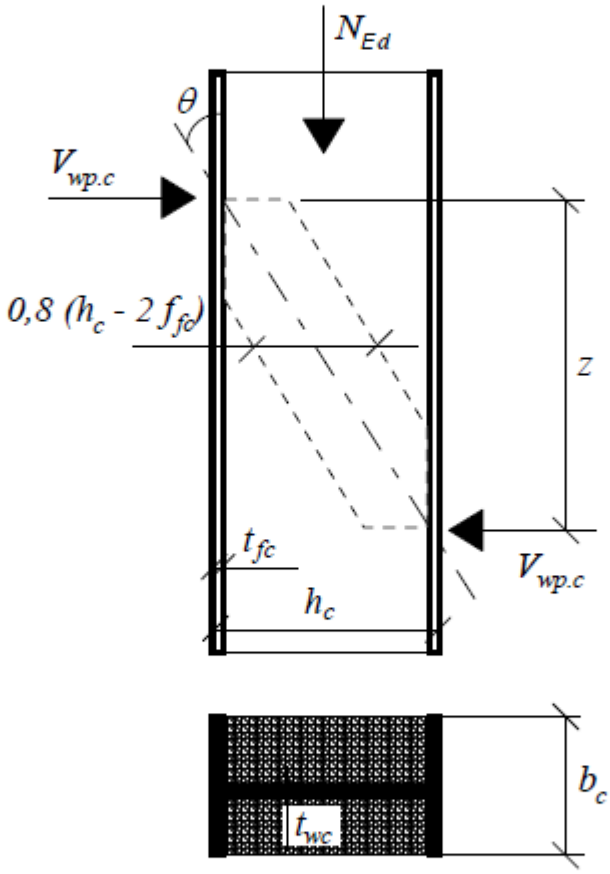
$$A_{concr} = [0.8\,(h_c - 2t_{fc})\cos\theta]\,(b_c - t_w)$$

$$\theta = arctan\left(\frac{h_c - 2t_{fc}}{z}\right)$$

$$z = 0.6H$$

$$\nu = 0.55$$



*Compressive strut in concrete*

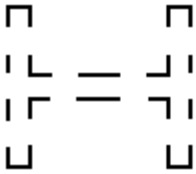
## Axial resistance of concreted column

The friction between column surface and concrete may be considered in the most of cases. The friction resistance $F_{rd}$ is calculated as the minimum value of failure along the steel surface and the combination of shear failure in concrete and friction on flanges:

$$F_{Rd} = min\left\{F_{s,Rd}; F_{c,s,Rd}\right\}$$

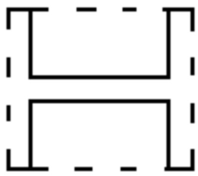
The friction resistance of column surface $F_{s,Rd}$ is given by the expression:

$$F_{s,Rd} = A_{steel,surf}\frac{0.36\sqrt{f_{ck}}}{\gamma_c}$$


*Considered column surface*

Combined failure of shear in concrete and friction on flanges $F_{c,s,Rd}$ is given by the expression

$$F_{c,s,Rd} = A_{con,surf}\frac{0.25f_{ctk,05}}{\gamma_c} + A_{steel1,surf}\frac{0.36\sqrt{f_{ck}}}{\gamma_c}$$


*Combined failure of shear in concrete and friction on flanges*

## Punching without base plate

The punching at the bottom end has to be checked if the friction resistance fails (or if cannot be considered). The punching resistance is based on EN 1992-1-1

$$V_{Rd} = \frac{u}{\beta}(\nu_{Rd,1}; \nu_{Rd3})$$

| | | |
|---|---|---|
| Where is: | $\beta$ | • The eccentricity factor ($\beta = 1,0$) |
| | $u$ | • The critical perimeter of the cross-section |
| | $v_{Rd1}$ | • The design shear resistance of concrete without reinforcement per one linear meter according to EN 1992-1-1 |
| | $v_{Rd3}$ | • The design shear resistance of concrete with reinforcement per one linear meter according to EN 1992-1-1 |

The compressive resistance is calculated as

$$F_{ber,Rd} = f_j A_{eff}$$

| | | |
|---|---|---|
| Where is: | $f_j$ | • The concrete strength in concentrated compression |
| | $A_c$ | • The cross-sectional area of column |

## Punching with base plate

The base resistance in punching is calculated in accordance with EN 1992-1-1 as

$$V_{Rd} = \frac{u_{bp}}{\beta}(\nu_{Rd1}; \nu_{Rd3})$$

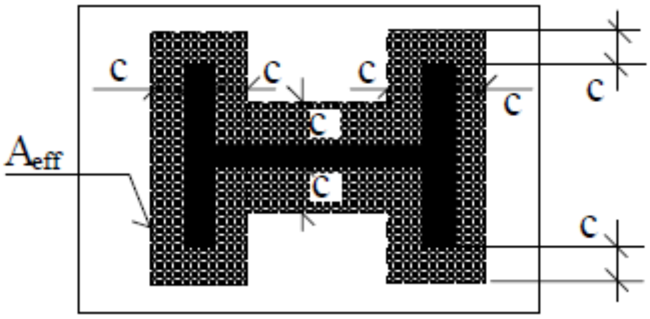
| | | |
|---|---|---|
| Where is: | $u_{bp}$ | • The perimeter of critical cross-section for column with base plate |

The compressive resistance is calculated as

$$F_{ber,Rd} = f_j A_{eff}$$

| | | |
|---|---|---|
| Where is: | $A_{eff}$ | • The effective area of column calculated in the same way as the effective area of base plate on foundation failure according to EN 1992-1-1 |

$$c = t\sqrt{\frac{f_y}{3\gamma_{M0} f_j}}$$




*The effective area for column*

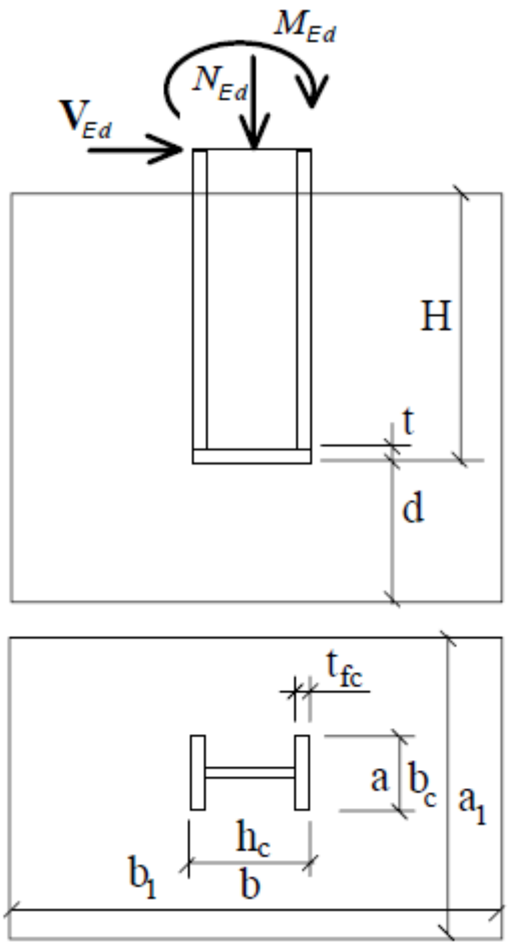
# Estimation of embedment depth

The preliminary estimation is, that for concrete strength classes *C20/25* and higher, the embedment depth $H = 3b_c$ is able to transfer full design plastic bending resistance of H-profile steel column $M_{pl,Rd,S235}$ (made of steel *S235*).

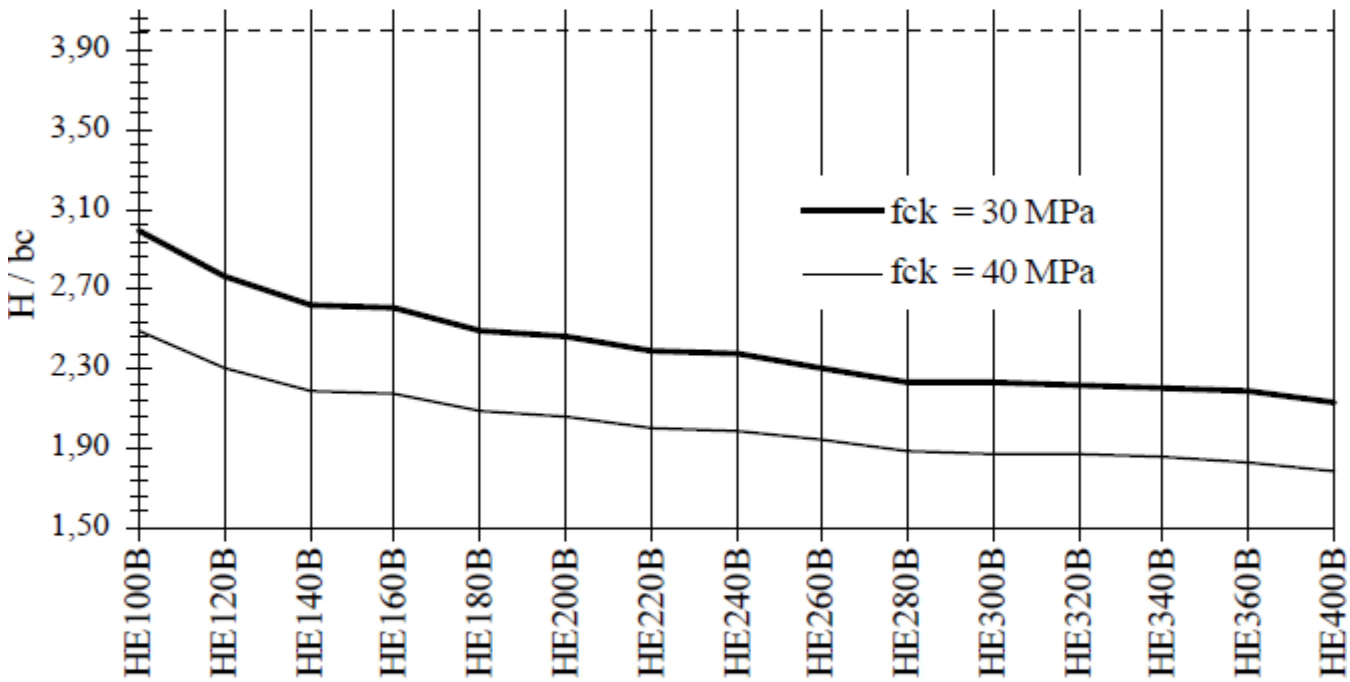
Also, the shear force $V_{Sd} \leq V_{pl,Rd}/4$ should be transferred automatically with the bending moment $M_{pl,Rd,S235}$.

The concrete foundation should be checked parallelly for shear stress and stress induced by soil.

The examples of design resistance for stiff axis of column are given in the chart and table below. The base plate has dimensions equal to the column cross-section. The plate thickness is selected in that way, that the resistance is equal to the compressive resistance of concrete. Dimensions are shown on the figure below. Following chart shows the proportional embedment depth $H/b_c$ limited by the column resistance in bending $M_{pl,Rd}$ and shear $V_{pl,Rd}$. The table shows the cohesion resistance $F_{Rd}$, punching resistance $V_{Rd}$, compressive resistance under the plate $F_{ber,Rd}$ and compressive resistance without base plate $F_{ber,0,Rd}$ for given concrete depth under the column $d$. The concrete base is without shear reinforcement. The chart and the table are calculated for steel grade *S235* and concrete classes *C12/15*, *C20/25*, *C30/37*, *C40/50*. Partial safety factors are $\gamma_{M0}=1,15$ and $\gamma_c=1,5$ .




*Foundation gemetry*




*The relative embedment depth for $M_{pl,Rd}$ and $V_{pl,Rd}$ for bending and shear, steel S235, concrete classes C30/37 a C40/50, partial factors $\gamma_{M0}$=1.15 and $\gamma_c$=1.5*

| Column | Base geometry | Design resistances |
|---|---|---|

| Cross-section | Concrete class | d [mm] | t [mm] | a [mm] | b [mm] | a1 [mm] | b1 [mm] | H [mm] | $N_{pl,Rd}$ [kN] | $V_{Rd}$ [kN] | $F_{Rd}$ [kN] | $F_{ber,Rd}$ [kN] | $F_{ber,0,Rd}$ [kN] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| HE100B | C12/15<br>C20/25<br>C30/37<br>C40/50 | 200 | 16 | 100 | 100 | 700 | 700 | 300 | 531 | 69<br>90<br>11<br>129 | 141<br>198<br>259<br>314 | 148<br>201<br>257<br>315 | 42<br>70<br>105<br>139 |
| HE200B | C 12/15<br>C20/25<br>C30/37<br>C40/50 | 400 | 30 | 200 | 200 | 1400 | 1400 | 600 | 1596 | 860<br>349<br>432<br>502 | 483<br>679<br>889<br>1077 | 542<br>692<br>902<br>1096 | 126<br>206<br>314<br>419 |
| HE300B | C12/15<br>C20/25<br>C30/37<br>C40/50 | 600 | 40 | 300 | 300 | 2100 | 2100 | 900 | 3045 | 594<br>775<br>959<br>1117 | 905<br>1272<br>1667<br>2020 | 1058<br>1371<br>1784<br>2163 | 240<br>399<br>599<br>799 |

Where is:

| | | |
|---|---|---|
| $N_{pl,Rd}$ | • | The vertical resistance |
| $V_{Rd}$ | • | The cohesion resistance |
| $F_{Rd}$ | • | The punching resistance |
| $F_{ber,Rd}$ | • | The resistance affected by the embedment depth and punching with base plate |
| $F_{ber,0,Rd}$ | • | The resistance affected by the embedment depth and punching without base plate |

## Stiffness classification

The classification of joints with regard to the stiffness is beneficial for modelling the stuctures, as rigid connections does not affect the distribution of internal forces any more. These limits depend on the structure type and type of analysis. The estimation is described in EN 1993-1-1.

Two types of structure have to be distinguish when classifying the anchoring joints: braced and unbraced structures. The accuracy is *3%* for ultimate limit state and *20%* for serviceability limit state (Wald, Jaspart; 1998). For common cases $(\overline{\lambda}_0 \leq 2 \div 3)$, the stiffness of braced structures may be calculated for

$$\bar{\lambda}_0 \leq 0.5$$

using formula

$$S_{j,ini} \geq 0$$

, for interval

$$0.5 < \bar{\lambda}_0 < 3.93$$

using formula

$$S_{j,ini} \geq 7\frac{(2\bar{\lambda}_0 - 1)EI_c}{L_c}$$

and for

$$\bar{\lambda}_0 \geq 3.93$$

using formula

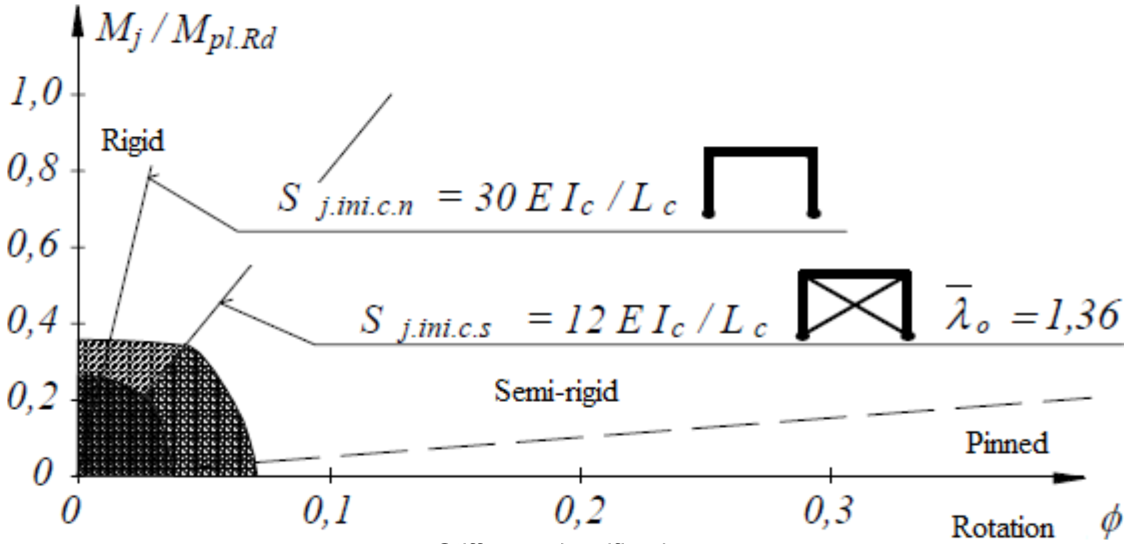
$$S_{j,ini} \geq 48\frac{EI_c}{L_c}$$

where $\overline{\lambda}_0$ is the relative slenderness of the column with hinged ends. This expression may be used for any slenderness. For relative slenderness limited by $\overline{\lambda}_0 = 1,36$ the limiting value may be considered as $12EI_c/L_c$.

Unbraced frames are more sensitive with regard to the stiffness of bases. For these structures, horizontal deformation is a decisive one (Wald, Jaspart; 1998), limiting value is

$$S_{j,ini} \geq 30\frac{EI_c}{L_c}$$




*Stiffness classification*

## References

Wald F., Jaspart J. P.: *Stiffness Design of Column Bases*, v 2nd World Steel Conference, San Sebastian No.: 135, Journal of Constructional Steel Research Vol. 46, Nos. 1-3, 1998, ISSN 0143-974X.

# Timber

# Cross-sections

Solid cross-sections can be rectangular or circular, or are composed of several rectangles into shapes *I*, *T* or *Pi*. For these cross-sections calculation assumes integrity and doesn't consider any slip between the different parts of the section.

Built-up cross-sections consist of several rectangular parts. Sub-sections are either rigidly attached (glued) or are unconnected.

## Coordinate Systems

Two coordinate systems are created for any cross-section: the local coordinate system of a member with axes marked *2* and *3* and cross-sectional coordinate system with axes marked *y* and *z*. Input of designing properties (buckling, LTB) is done mostly in the local coordinate system of the member. Thus rotation of the cross-section is measured relatively to the local axes *2* and *3*. Internal forces are indicated by subscripts *2* and *3*, as they are also related to the local coordinate system of the member. The verification is done in the coordinate system of the cross-section. Internal forces are always converted into the coordinate system of the cross-section before the verification.

# Material characteristics

The material can be selected from predefined database or specified manually by the user. the database contains basic strength classes for softwood and hardwood according to EN 338, strength classes for glued laminated timber and also local timber grades in accordance with EN 1912. These characteristics are necessary for the analysis:

| | | |
|---|---|---|
| $f_{m,k}$ | • | Characteristic bending strength |
| $f_{t,0,k}$ | • | Characteristic tensile strength along the grain |
| $f_{c,0,k}$ | • | Characteristic compressive strength along the grain |
| $f_{v,k}$ | • | Characteristic shear strength |
| $f_{t,90,k}$ | • | Characteristic tensile strength perpendicular to the grain |
| $f_{c,90,k}$ | • | Characteristic compressive strength perpendicular to grain |
| $E_{0,05}$ | • | Fifth percentile value of modulus of elasticity |
| $\rho_k$ | • | Characteristic density |

The design values of material characteristics are used during the analysis. The design values are calculated by multiplication of characteristic values by modification factor $k_{mod}$ and division by partial factor for material properties $\gamma_M$. Values of $\gamma_M$ are written in chapter "**National annexes**". Index of design values starts with *d*. Modification factor for duration of load and moisture content $k_{mod}$ depends on load duration and service class (table *3.1* of EN 1995-1-1).

The strength parameters in tension and bending can be increased for small cross-sections in accordance with chapter *3.2* of EN 1995-1-1. Depth factor $k_h$ according to the formula *(3.1)* is used for this increase.

# Slenderness verification

The engineering practice it is known that very thin elements may cause problems. Limit values aren't specified in the standard, but it is convenient to keep the slenderness ratio within certain limits very often. Therefore, the program always calculates and reports the value of the slenderness ratio for any member. This value can be also compared with the specified limit.

The slenderness ratios in the directions of $y$ and $z$ axis are calculated using following formulas:

$$\lambda_y = \frac{l_{cr,y}}{i_y}, \lambda_z = \frac{l_{cr,z}}{i_z}$$

where is: $l_{cr,y}$ , $l_{cr,z}$
- Buckling lengths corresponding to bending about the $y$ and $z$ axis

$i_y$ , $i_z$
- Radius of gyration corresponding to bending about the $y$ and $z$ axis

Buckling lengths $l_{cr,y}$ and $l_{cr,z}$ are determined using these rules:

- **members in compression** - buckling length for corresponding direction is used.
- **members in tension** - "**Sector length for buckling**" is used. If not specified, basic member length will be used.

The greater value of $\lambda_y$ and $\lambda_z$ will be used for slenderness verification.

## Buckling

These formulas are used for the calculation of relative slenderness ratios $\lambda_{rel,y}$ and $\lambda_{rel,z}$

$$\lambda_{rel,y} = \frac{\lambda_y}{\pi}\sqrt{\frac{f_{c,0,k}}{E_{0,05}}}, \lambda_{rel,z} = \frac{\lambda_z}{\pi}\sqrt{\frac{f_{c,0,k}}{E_{0,05}}}$$

where is: $\lambda_y$ , $\lambda_z$
- Slenderness ratios corresponding to bending about the $y$ and $z$ axis

$k_c$
- Instability factor, the lesser of $k_{c,y}$ *and* $k_{c,z}$ is considered

$f_{c,0,k}$
- Characteristic compressive strength along the grain

Slenderness ratios $\lambda_y$ , $\lambda_z$ are calculated using these formulas

$$\lambda_y = \frac{l_{cr,y}}{i_y}, \lambda_z = \frac{l_{cr,z}}{i_z}$$

where is: $l_{cr,y}$ , $l_{cr,z}$
- Buckling lengths corresponding to bending about the $y$ and $z$ axis

$i_y$ , $i_z$
- Radius of gyration corresponding to bending about the $y$ and $z$ axis

The instability factors $k_{c,y}$ and $k_{c,z}$ are calculated using following formulas

$$k_{c,y} = \frac{1}{k_y + \sqrt{{k_y}^2 - {\lambda_{rel,y}}^2}}, k_{c,z} = \frac{1}{k_z + \sqrt{{k_z}^2 - {\lambda_{rel,z}}^2}}$$

where is: $\lambda_{rel,y}$ , $\lambda_{rel,z}$
- Relative slenderness ratios corresponding to bending about the $y$ and $z$ axis

$k_y$ , $k_z$
- Instability factors calculated using following formulas

$$k_y = 0.5(1 + \beta_c(\lambda_{rel,y} - 0.3) + {\lambda_{rel,y}}^2), k_z = 0.5(1 + \beta_c(\lambda_{rel,z} - 0.3) + {\lambda_{rel,z}}^2)$$

where is: $\lambda_{rel,y}$ , $\lambda_{rel,z}$
- Relative slenderness ratios corresponding to bending about the $y$ and $z$ axis

$\beta_c$
- Straightness factor equal to *0.2* for solid timber and *0.1* for glued laminated timber

## Lateral torsional buckling

The factor used for lateral buckling $k_{crit}$ will be used in the analysis if the lateral torsional buckling is taken into consideration. This factor is calculated in accordance with chapter *6.3.3.* of EN 1995-1-1:

The relative slenderness for bending is calculated using this formula

$$\lambda_{rel,m} = \sqrt{\frac{f_{m,k}}{\sigma_{m,crit}}}$$

where is: $\lambda_{rel,m}$
- Relative slenderness for bending

$f_{m,k}$
- Characteristic bending strength

**$\sigma_{m,crit}$** • Critical bending stress

The critical bending stress is calculated using formula

$$\sigma_{m,crit} = \frac{M_{y,crit}}{W_y} = \frac{\pi\sqrt{E_{0,05} I_z G_{0,05} I_{tor}}}{l_{ef} W_y}$$

where is:
- **$E_{0,05}$** • Fifth percentile value of modulus of elasticity
- **$G_{0,05}$** • Fifth percentile value of shear modulus
- **$I_z$** • Second moment of area about the weak axis $z$
- **$I_{tor}$** • Torsional moment of inertia
- **$l_{ef}$** • Effective length
- **$W_y$** • Section modulus about axis $y$

The critical bending stress for softwood with solid rectangular cross-section is calulated in accordance with *(6.32)*

$$\sigma_{m,crit} = \frac{0.78 b^2}{h \cdot l_{ef}} E_{0,05}$$

where is:
- **$\sigma_{m,crit}$** • Critical bending stress
- **$b$** • Width of the member
- **$h$** • Depth of the member
- **$l_{ef}$** • Effective length
- **$E_{0,05}$** • Fifth percentile value of modulus of elasticity

The factor used for lateral buckling $k_{crit}$ is equal to *1.0* for $\lambda_{rel,m} \leq 0{,}75$ . Following formula is used for the range $0{,}75 < \lambda_{rel,m} \leq 1{,}4$

$$k_{crit} = 1.56 - 0.75\lambda_{rel,m}$$

Following formula is used for $\lambda_{rel,m} > 1{,}4$

$$k_{crit} = \frac{1}{\lambda_{rel,m}^2}$$

# Built-up members

The capacity of the member in the $z$-axis direction is calculated as the sum of the capacities of the individual parts. The capacity in the $y$-axis direction is calculated in accordance with *C.1.2(2)*

$$\sigma_{c,0,d} \leq k_c f_{c,0,d}$$

where

$$\sigma_{c,0,d} = \frac{f_{c,d}}{A_{tot}}$$

Where is:
- **$A_{tot}$** • Total area of cross-section
- **$k_c$** • Instability factor calculated using an effective slenderness ratio $\lambda_{ef}$ determined using following formulas

## Spaced members with packs or gussets

The total cross-sectional area is calculated in accordance with *C.3.1(3)*:

$$A_{tot} = 2A$$

The total moment of inertia is calculated in accordance with *C.3.1(3)*:

$$I_{tot} = \frac{b[(2h+a)^3 - a^3]}{12}$$

The buckling verification in the $z$-axis direction is calculated as the sum of the capacities of the individual parts. The buckling verification in the $y$-axis direction is calculated in accordance with *C.3.2(2)*

$$\lambda_{ef} = \sqrt{\lambda^2 + \eta\frac{n}{2}\lambda_1^2}$$

Where is:
- **$\lambda$** • The slenderness ratio for a solid member with the same length, the same area and the same second moment of area
- **$\lambda_1$** • The slenderness ratio for the shafts, minimum value is *30*
- **$n$** • The number of shafts
- **$\eta$** • The factor given in table *C.1*

The slenderness ratio $\lambda$ is calculated using formula

$$\lambda = l\sqrt{\frac{A_{tot}}{I_{tot}}}$$

The slenderness ratio for the shafts $\lambda_1$ is calculated using formula

$$\lambda_1 = \sqrt{12}\frac{l_1}{h}$$

The shear forces on the gussets or packs are calculated in accordance with formula *(C.13)*

$$T_d = \frac{V_d I_1}{a_1}$$

## Lattice members with glued or nailed joints

The effective slenderness ratio is calculated in accordance with *C.4.2(2)* using formula

$$\lambda_{ef} = max\begin{cases}\lambda_{tot}\sqrt{1+\mu} \\ 1.05\lambda_{tot}\end{cases}$$

Where is:
- **$\lambda_{tot}$** • The slenderness ratio for a solid member with the same length, the same area and the same second moment of area
- **$\mu$** • The factor calculated using following formulas

The factor $\mu$ is calculated for a glued $V$-truss using formula *(C.16)*

$$\mu = 4\frac{e^2 A_f}{I_f}\left(\frac{h}{l}\right)^2$$

Where is:
- **$e$** • The eccentricity of the joints
- **$A_f$** • The area of the flange
- **$I_f$** • The second moment of area of flange
- **$l$** • The span
- **$h$** • The distance of the flanges

The factor $\mu$ is calculated for a glued $N$-truss using formula *(C.17)*

$$\mu = \frac{e^2 A_f}{I_f}\left(\frac{h}{l}\right)^2$$

The factor $\mu$ is calculated for a nailed $V$-truss using formula *(C.18)*

$$\mu = 25\frac{hE_{mean}A_f}{l^2 n K_u sin2\Theta}$$

Where is:
- **$n$** • The number of nails in a diagonal. If a diagonal consists of two or more pieces, $n$ is thesum of the nails
- **$E_{mean}$** • Mean value of modulus of elasticity
- **$K_u$** • The slip modulus of one nail for ULS

The factor $\mu$ is calculated for a nailed $N$-truss using formula *(C.19)*

$$\mu = 50\frac{hE_{mean}A_f}{l^2 n K_u sin2\Theta}$$

Where is:
- **$n$** • The number of nails in a diagonal. If a diagonal consists of two or more pieces, $n$ is thesum of the nails
- **$E_{mean}$** • Mean value of modulus of elasticity
- **$K_u$** • The slip modulus of one nail for ULS

# Ultimate limit state

The verification equations differ according to the stress style.

## Tension

Tension is verified in accordance with chapter *6.1.2.* of EN 1995-1-1 using formula

$$\frac{N}{A \cdot f_{t,0,d}} \leq 1.0$$

Where is:

- $N$ • Normal force
- $A$ • Cross-sectional area
- $f_{t,0,d}$ • Design tensile strength

## Compression

Members in compression can be designed without or with consideration of buckling. Verification without buckling consideration is performed in accordance with chapter *6.1.4.* using formula

$$\frac{|N|}{A \cdot f_{c,0,d}} \leq 1.0$$

Where is: $f_{c,0,d}$ • Design compressive strength

Verification with buckling consideration is performed in accordance with chapter *6.3.2.* using formula

$$\frac{|N|}{k_c \cdot A \cdot f_{c,0,d}} \leq 1.0$$

Where is: $k_c$ • Instability factor, minimum value of $k_{c,y}$ and $k_{c,z}$ is used. These factors are calculated according to the rules described in chapter "**Buckling**".

## Biaxial bending

Biaxial bending is verified in accordance with chapter *6.1.6.* of EN 1995-1-1 using formulas

$$\left| \frac{M_y}{W_y \cdot f_{m,y,d}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d}} \right| \leq 1.0$$

$$\left| k_m \frac{M_y}{W_y \cdot f_{m,y,d}} + \frac{M_z}{W_z \cdot f_{m,z,d}} \right| \leq 1.0$$

kde je:

- $M_y$, $M_z$ • Bending moments about *y*- and *z*-axis
- $W_y$, $W_z$ • Section modulus about *y*- and *z*-axis
- $f_{m,y,d}$, $f_{m,z,d}$ • Design bending strength about *y*- and *z*-axis
- $k_m$ • Factor considering redistribution of bending stresses in cross-section

The verification is performed at the edge of the cross-section, in the point of the most significant stress.

Verification with consideration of lateral torsional buckling uses these formulas

$$\left| \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d}} \right| \leq 1.0$$

$$\left| k_m \frac{M_y}{W_y \cdot f_{m,y,d}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d}} \right| \leq 1.0$$

Where is: $k_{crit,y}$, $k_{crit,z}$ • Factor used for lateral buckling about *y*- and *z*-axis. These factors are calculated according to the rules described in chapter "**Lateral torsional buckling**".

The bending is verified only in strong axis direction for built-up cross-sections.

## Shear

Shear is verified in accordance with *6.1.7* using formulas

$$\frac{|V_z \cdot S_y|}{I_y \cdot t_y \cdot f_{v,d}} \leq 1.0$$

$$\frac{|V_y \cdot S_z|}{I_z \cdot t_z \cdot f_{v,d}} \leq 1.0$$

| | | |
|---|---|---|
| Where is: | $V_y$, $V_z$ | • Shear forces in *y-* and *z-* directions |
| | $S_y$, $S_z$ | • First moment of area about *y-* and *z-* axis for cross-section part above (under) point of verification |
| | $I_y$, $I_z$ | • Second moment of area about *y-* and *z-* axis |
| | $t_y$, $t_z$ | • Width of cross-section in verified position |
| | $f_{v,d}$ | • Design shear strength |

The verification is performed in the point of the most significant stress. Built-up cross-sections are verified in the centre of gravity.

The shear is verified only in strong axis direction for built-up cross-sections.

## Combined bending and axial tension

Combined bending and axial tension is verified in accordance with chapter *6.2.3.* using formulas

$$\left|\frac{N}{A \cdot f_{t,0,d}} + \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d}}\right| \leq 1.0$$

$$\left|\frac{N}{A \cdot f_{t,0,d}} + k_m \frac{M_y}{W_y \cdot f_{m,y,d}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d}}\right| \leq 1.0$$

## Combined bending and axial compression

Combined bending and axial compression is verified in accordance with chapter *6.2.4.* using formulas

$$\left|-\left(\frac{N}{A \cdot f_{c,0,d}}\right)^2 + \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d}}\right| \leq 1.0$$

$$\left|-\left(\frac{N}{A \cdot f_{c,0,d}}\right)^2 + k_m \frac{M_y}{W_y \cdot f_{m,y,d}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d}}\right| \leq 1.0$$

The verification is performed at the edge of the cross-section, in the point of the most significant stress.

Verification with buckling consideration is performed in accordance with chapter *6.3.2.*

$$\left|\frac{N}{A \cdot k_{c,y} \cdot f_{c,0,d}} + \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d}}\right| \leq 1.0$$

$$\left|\frac{N}{A \cdot k_{c,z} \cdot f_{c,0,d}} + k_m \frac{M_y}{W_y \cdot f_{m,y,d}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d}}\right| \leq 1.0$$

| | | |
|---|---|---|
| Where is: | $k_{c,y}$, $k_{c,z}$ | • The Instability factors for buckling perpendicular to *y*- and *z*-axis. These factors are calculated according to the rules described in chapter "**Buckling**". |

# National annexes

These partial factors are applied for design standard EN 1995-1-1 and its annexes:

| Factor | EN 1995-1-1 | Czechia | Slovakia | Poland |
|---|---|---|---|---|
| **$\gamma_M$ - solid timber** | 1,30 | 1,30 | 1,30 | 1,30 |
| **$\gamma_M$ - glued laminated timber** | 1,25 | 1,25 | 1,25 | 1,25 |
| **$\gamma_M$ - accidental situations** | 1,00 | 1,00 | 1,00 | 1,00 |

These partial factors are applied for design standard EN 1995-1-2 and its annexes:

| Factor | EN 1995-1-2 | Czechia | Slovakia | Poland |
|---|---|---|---|---|
| **$\gamma_{M,fi}$ - fire resistance** | 1,00 | 1,00 | 1,00 | 1,00 |

# Timber Fire

## Basic principles

The verification of the fire resistance is performed in accordance with EN 1995-1-2 (Eurocode EC5). Structural members are verified using residual cross-section, that is the cross-section reduced by the char layer. The analysis is performed for the members loaded by the combination of normal and shear forces and bending moments.

## Material characteristics

The material can be selected from predefined database or specified manually by the user. the database contains basic strength classes for softwood and hardwood according to EN 338, strength classes for glued laminated timber and also local timber grades in accordance with EN 1912. These characteristics are necessary for the analysis:

| | |
|---|---|
| $f_{m,k}$ | • Characteristic bending strength |
| $f_{t,0,k}$ | • Characteristic tensile strength along the grain |
| $f_{c,0,k}$ | • Characteristic compressive strength along the grain |
| $f_{v,k}$ | • Characteristic shear strength |
| $E_{0,mean}$ | • Mean value of modulus of elasticity |
| $E_{0,05}$ | • Fifth percentile value of modulus of elasticity |
| $G_{mean}$ | • Mean value of shear modulus |
| $\rho_k$ | • Characteristic density |

The verification of the fire resistance is performed using lower material reliability than the verification for common temperature. The reason is, that the fire exposure is considered as an accidental situation and only requirement is the bearing capacity during the escape of persons. Therefore 20% quantile is used instead of 5% quantile for common design situations. The 20% quantile properties are calculated as characteristic ones multiplied by the factor $k_{fi}$ that is greater than *1* .

The design values of material characteristics are used during the analysis. The design values are calculated by multiplication of characteristic values by modification factor $k_{mod,fi}$ and division by partial factor for material properties $\gamma_{M,fi}$. Index of design values starts with *d* . Modification factor for duration of load and moisture content $k_{mod,fi}$ depends on the design procedure.

$$f_{d,fi} = k_{mod,fi}\frac{k_{fi}f_k}{\gamma_{M,fi}}$$

The strength parameters in tension and bending can be increased for small cross-sections in accordance with chapter *3.2* of EN 1995-1-1. Depth factor $k_h$ according to the formula *(3.1)* is used for this increase.

## Calculation of charring layer

The depth of the charring layer $d_{char}$ is the thickness of the material layer at the surface of the timber cross-section, that loses its ability to transfer stresses due to the degradation of mechanical properties. The material in this layer is charred, with no strength or even fallen off. The charring depth depends on charring rate, time, after which charring occurs (ie, the desired duration of fire resistance), and on a fire protection properties. Determination of the charring depth is the first important part of the fire resistance verification.

There are two basic values of the char depth used during the fire resistance verification. First value is the charring depth for one-dimensional charring $d_{char,0}$, depending on the design charring rate for one-dimensional charring $\beta_0$. Second value is the notional charring depth $d_{char,n}$, which depends on the design notional charring rate under standard fire exposure $\beta_n$. One-dimensional charring occurs when the flat wood element is exposed to fire from one side. If the cross-section is exposed to a fire from several sides, the increased carbonization in corners occurs. Since the calculation of the curvature in corners is complicated, the notional charring rate $\beta_n$ was defined. This notional charring rate performs increased values comparing to the one-dimensional charring rate $\beta_0$, as the values are affected by the increased charring rate around the corners of the section.

### Unprotected cross-section

The charring depth for unprotected cross-section is calculated using formula

$$d_{char} = \beta t$$

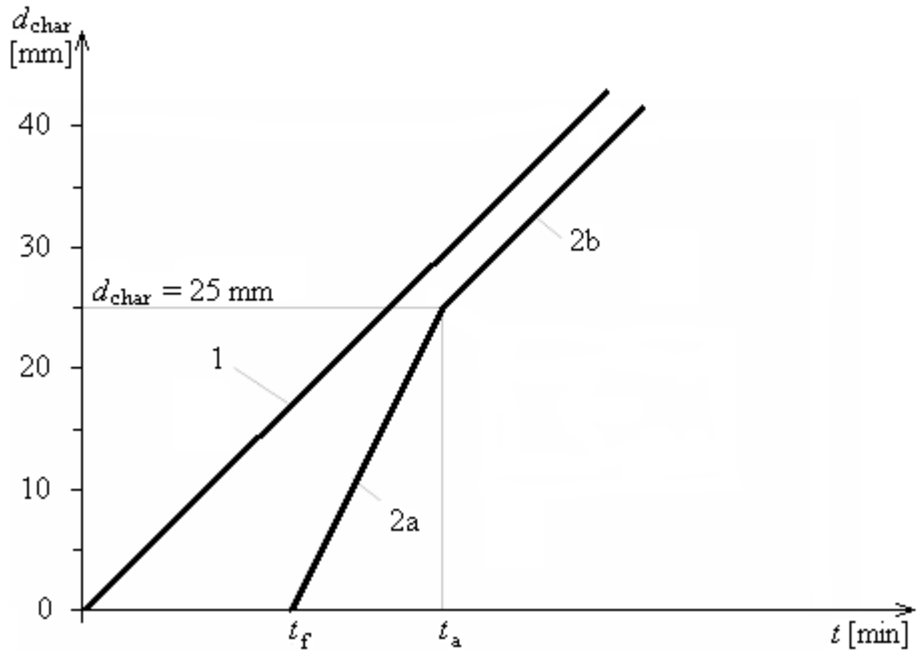
| | | |
|---|---|---|
| Where is: | $t$ | • The time of fire exposure |
| | $\beta$ | • The design charring rate |

The values of charring rate $\beta$ defined in EN 1995-1-2 corresponds to standard fire effects, that is described using standard temperature curve. For this type of fire, the charring rate $\beta$ is a constant value without any time dependency. It corresponds to the fact that the temperature increases theoretically to infinity for the standard temperature curve over time.

## Protected cross-section

The charring depth of protected cross-section doesn't rise linearly and doesn't begin immediately at time *0*, as the beginning is shifted by a certain period $t_{ch}$. The charring course contains some marginal moments in which changes of charring rate occur. The first of them is the beginning of charring $t_{ch}$. Another important point is the time $t_f$, that describes the time of protection failure. The values $t_{ch}$ and $t_f$ may be identical for cases when the charring starts in time of protection failure. The charring rate of protected cross-section in time between $t_{ch}$ and $t_f$ is slower comparing to the unprotected element. After the failure of fire protection the charring rate increases to a value, that is higher than the charring rate for unprotected element. Another important timestamp is the time $t_a$, which is determined as the minimum of the two values: First value is the time, when the charring depth is identical to the charring depth of unprotected element with the same dimensions. The second value is the time when the charring depth is equal to *25mm*. The charring rate $\beta$ is the same as for unprotected cross-section after reaching this value. Following situations may occur for protected cross-sections:

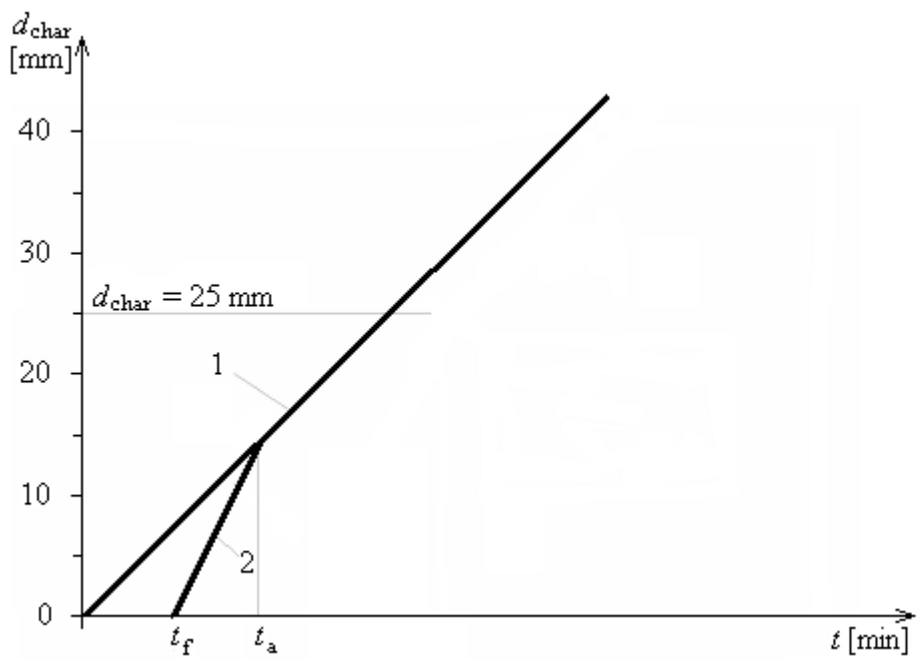



*Development of charring depth in time for $t_f$= $t_{ch}$, charring depth 25mm was reached earlier comparing to the unprotected cross-section*

Where is:

**1** • The charring depth for unprotected cross-section

**2** • The charring depth for protected cross-section (***2a*** - charring rate is faster after the protection failure; ***2b*** - charring rate is equal to the rate of unprotected cross-section after reaching the charring depth *25mm*)

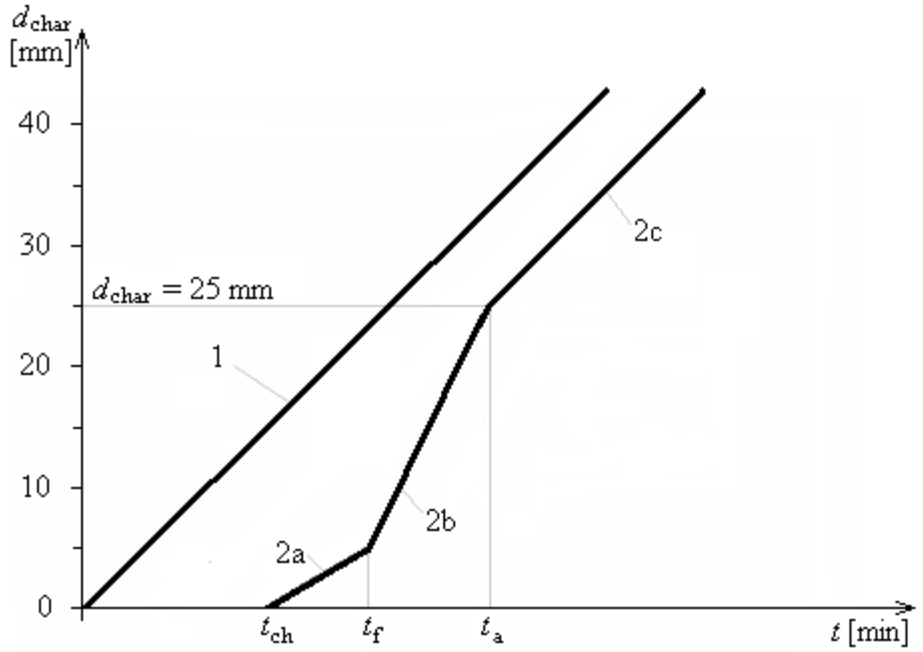



*Development of charring depth in time for $t_f$= $t_{ch}$, the charring depth equal to the charring depth of unprotected cross-section was reached earlier than the charring depth 25mm*

Where is: ***1*** • The charring depth for unprotected cross-section

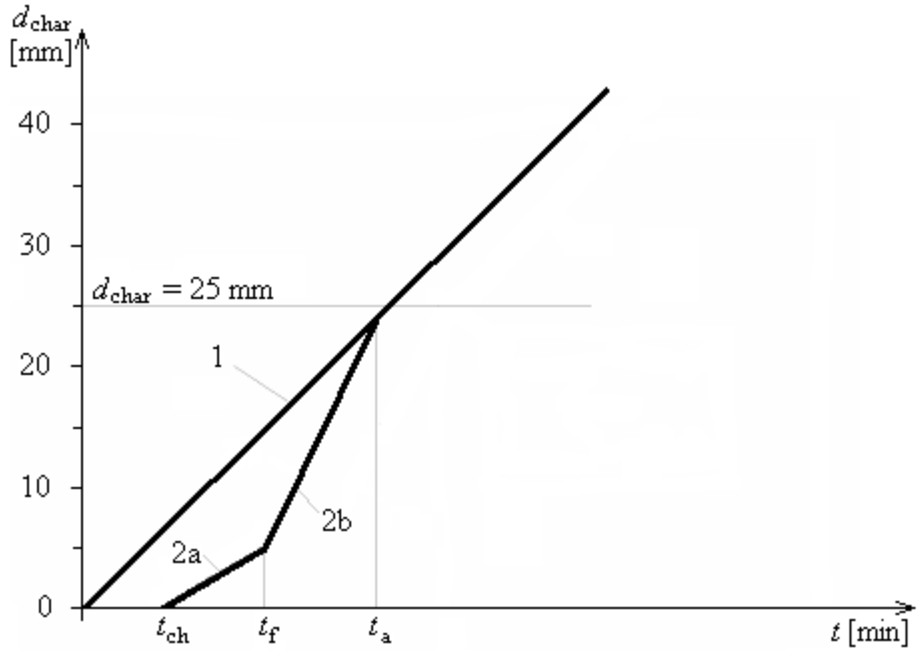
***2*** • The charring depth for protected cross-section, the charring rate is faster after the protection failure




*Development of charring depth in time for $t_f > t_{ch}$, charring depth 25mm was reached earlier comparing to the unprotected cross-section*

Where is: ***1*** • The charring depth for unprotected cross-section

***2*** • The charring depth for protected cross-section (***2a*** -charring rate is slower before the protection failure; ***2b*** - charring rate is faster after the protection failure; ***2c*** - charring rate is equal to the rate of unprotected cross-section after reaching the charring depth *25mm*)




*Development of charring depth in time for $t_f > t_{ch}$, the charring depth equal to the charring depth of unprotected cross-section was reached earlier than the charring depth 25mm*

Where is: ***1*** • The charring depth for unprotected cross-section

***2*** • The charring depth for protected cross-section (***2a*** -charring rate is slower before the protection failure; ***2b*** - charring rate is faster after the protection failure)

Time $t_a$ is calculated with the help of following procedures.

The increase of the charring depth for unprotected cross-section is calculated using this formula:

$$d = \beta t$$

| | | | |
|---|---|---|---|
| Where is: | $t$ | • | The time of fire exposure |
| | $\beta$ | • | The design charring rate |

Following formula is used for protected cross-sections with $t_f = t_{ch}$ for time between $t_f$ and $t_a$:

$$d = k_3 \beta (t - t_f)$$

| | | | |
|---|---|---|---|
| Where is: | $k_3$ | • | The post-protection coefficient ($k_3 > 1$) |

Following formula is used for protected cross-sections with $t_f > t_{ch}$ for time between $t_f$ and $t_a$:

$$d = d_f + k_3 \beta (t - t_f)$$

| | | | |
|---|---|---|---|
| Where is: | $k_3$ | • | The post-protection coefficient ($k_3 > 1$) |
| | $d_f$ | • | The charring depth for time $t_f$ calculated with the help of following formula: |

$$d_f = k_2 \beta (t_f - t_{ch})$$

| | | | |
|---|---|---|---|
| Where is: | $k_2$ | • | The insulation coefficient ($k_2 < 1$) |

# Methods for fire resistance analysis

The fire resistance of timber members can be verified using two different design methods according to the chapter *4.2* of EN 1995-1-2: "**Reduced cross-section method**" and "**Reduced properties method**". The "**Reduced cross-section method**" is recommended in the standard, the "**Reduced properties method**" provides more economical results.

## Reduced cross-section method

The effective cross-section is verified for this method. The effective cross-section is calculated by reducing the residual cross-section by the edge layer with the thickness $k_0 d_0$, where $k_0$ is the factor, that grows linearly from *0* (time *0 min*) to *1* (time *20 min* or $t_{ch}$ for $t_{ch} > 20$ *min*). Thickness $d_0$ is a constant with its value *7 mm*. Design values of material characteristics are calculated using factor $k_{mod,fi}$ equal to *1.0*.

## Reduced properties method

The residual cross-section is verified in this method, however the material properties are reduced by factor $k_{mod,fi}$. The factor $k_{mod,fi}$ is calculated in a different ways for different properties and depends on the perimeter and area of the residual cross-section.

The following formula is used for bending strength:

$$k_{mod,fi} = 1 - \frac{1}{200} \cdot \frac{p}{A_r}$$

For compressive strength:

$$k_{mod,fi} = 1 - \frac{1}{125} \cdot \frac{p}{A_r}$$

For tensile strength and modulus of elasticity:

$$k_{mod,fi} = 1 - \frac{1}{330} \cdot \frac{p}{A_r}$$

| | | | |
|---|---|---|---|
| Where is: | $p$ | • | Perimeter |
| | $A_r$ | • | Area of residual cross-section |

These values of $k_{mod,fi}$ are valid for time $t > 20$ *min*. For time *0-20 min* the value of $k_{mod,fi}$ is increasing linearly from 0 to the described value.

# Ultimate limit state

The verification equations differ according to the stress style.

## Tension

Tension is verified in accordance with chapter *6.1.2.* of EN 1995-1-1 using formula

$$\frac{N}{A \cdot f_{t,0,d,fi}} \leq 1.0$$

| Where is: | ***N*** | • Normal force |
|---|---|---|
| | ***A*** | • Cross-sectional area |
| | $f_{t,0,d,fi}$ | • Design tensile strength |

## Compression

Members in compression can be designed without or with consideration of buckling. Verification without buckling consideration is performed in accordance with chapter *6.1.4.* using formula

$$\frac{|N|}{A \cdot f_{c,0,d,fi}} \leq 1.0$$

| Where is: | $f_{c,0,d,fi}$ | • Design compressive strength |
|---|---|---|

Verification with buckling consideration is performed in accordance with chapter *6.3.2.* using formula

$$\frac{|N|}{k_c \cdot A \cdot f_{c,0,d,fi}} \leq 1.0$$

| Where is: | $k_c$ | • Instability factor, minimum value of $k_{c,y}$ and $k_{c,z}$ is used. These factors are calculated according to the rules described in chapter "**Buckling**". |
|---|---|---|

## Biaxial bending

Biaxial bending is verified in accordance with chapter *6.1.6.* of EN 1995-1-1 using formulas

$$\left| \frac{M_y}{W_y \cdot f_{m,y,d,fi}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d,fi}} \right| \leq 1.0$$

$$\left| k_m \frac{M_y}{W_y \cdot f_{m,y,d,fi}} + \frac{M_z}{W_z \cdot f_{m,z,d,fi}} \right| \leq 1.0$$

| kde je: | $M_y$, $M_z$ | • Bending moments about *y*- and *z*-axis |
|---|---|---|
| | $W_y$, $W_z$ | • Section modulus about *y*- and *z*-axis |
| | $f_{m,y,d,fi}$, $f_{m,z,d,fi}$ | • Design bending strength about *y*- and *z*-axis |
| | $k_m$ | • Factor considering redistribution of bending stresses in cross-section |

The verification is performed at the edge of the cross-section, in the point of the most significant stress.

Verification with consideration of lateral torsional buckling uses these formulas

$$\left| \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d,fi}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d,fi}} \right| \leq 1.0$$

$$\left| k_m \frac{M_y}{W_y \cdot f_{m,y,d,fi}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d,fi}} \right| \leq 1.0$$

| Where is: | $k_{crit,y}$, $k_{crit,z}$ | • Factor used for lateral buckling about *y*- and *z*-axis. These factors are calculated according to the rules described in chapter "**Lateral torsional buckling**". |
|---|---|---|

The bending is verified only in strong axis direction for built-up cross-sections.

## Shear

Shear is verified in accordance with *6.1.7* using formulas

$$\frac{|V_z \cdot S_y|}{I_y \cdot t_y \cdot f_{v,d,fi}} \leq 1.0$$

$$\frac{|V_y \cdot S_z|}{I_z \cdot t_z \cdot f_{v,d,fi}} \leq 1.0$$

Where is: **$V_y$, $V_z$** • Shear forces in *y-* and *z-* directions

**$S_y$, $S_z$** • First moment of area about *y-* and *z-* axis for cross-section part above (under) point of verification

**$I_y$, $I_z$** • Second moment of area about *y-* and *z-* axis

**$t_y$, $t_z$** • Width of cross-section in verified position

**$f_{v,d,fi}$** • Design shear strength

The verification is performed in the point of the most significant stress. Built-up cross-sections are verified in the centre of gravity.

The shear is verified only in strong axis direction for built-up cross-sections.

### Combined bending and axial tension

Combined bending and axial tension is verified in accordance with chapter *6.2.3.* using formulas

$$\left|\frac{N}{A \cdot f_{t,0,d,fi}} + \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d,fi}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d,fi}}\right| \leq 1.0$$

$$\left|\frac{N}{A \cdot f_{t,0,d,fi}} + k_m \frac{M_y}{W_y \cdot f_{m,y,d,fi}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d,fi}}\right| \leq 1.0$$

### Combined bending and axial compression

Combined bending and axial compression is verified in accordance with chapter *6.2.4.* using formulas

$$\left|-\left(\frac{N}{A \cdot f_{c,0,d,fi}}\right)^2 + \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d,fi}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d,fi}}\right| \leq 1.0$$

$$\left|-\left(\frac{N}{A \cdot f_{c,0,d,fi}}\right)^2 + k_m \frac{M_y}{W_y \cdot f_{m,y,d,fi}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d,fi}}\right| \leq 1.0$$

The verification is performed at the edge of the cross-section, in the point of the most significant stress.

Verification with buckling consideration is performed in accordance with chapter *6.3.2.*

$$\left|\frac{N}{A \cdot k_{c,y} \cdot f_{c,0,d,fi}} + \frac{M_y}{W_y \cdot k_{crit,y} \cdot f_{m,y,d,fi}} + k_m \frac{M_z}{W_z \cdot f_{m,z,d,fi}}\right| \leq 1.0$$

$$\left|\frac{N}{A \cdot k_{c,z} \cdot f_{c,0,d,fi}} + k_m \frac{M_y}{W_y \cdot f_{m,y,d,fi}} + \frac{M_z}{W_z \cdot k_{crit,z} \cdot f_{m,z,d,fi}}\right| \leq 1.0$$

Where is: **$k_{c,y}$, $k_{c,z}$** • The Instability factors for buckling perpendicular to *y*- and *z*-axis. These factors are calculated according to the rules described in chapter "**Buckling**".

# Masonry

## Material characteristics

Database of material characteristics is based on production sheets of the individual manufacturers. Characteristics of general masonry units are calculated in accordance with EN 1996-1-1.

### Characteristic compressive strength of masonry

Characteristic compressive strength of masonry is calculated using equation

$$f_k = K \cdot f_b^{\alpha} \cdot f_m^{\beta}$$

where: **$f_k$** • is the characteristic compressive strength of the masonry

**$K$** • is a constant based on table *3.3*

**$\alpha,\beta$** • are constants

**$f_b$** • is the normalised mean compressive strength of the units

**$f_m$** • is the compressive strength of the mortar

Modified equation for masonry made with general purpose mortar and lightweight mortar:

$$f_k = K \cdot f_b^{0.7} \cdot f_m^{0.3}$$

Modified equation for masonry made with thin layer mortar, in bed joints of thickness *0.5mm* to *3mm*, and clay units of Group *1* and *4*, calcium silicate, aggregate units and aac units:

$$f_k = K \cdot f_b^{0.85}$$

Modified equation for masonry units made with thin layer mortar, in bed joints of thickness *0.5mm* to *3mm*, and clay units of Group *2* and *3*:

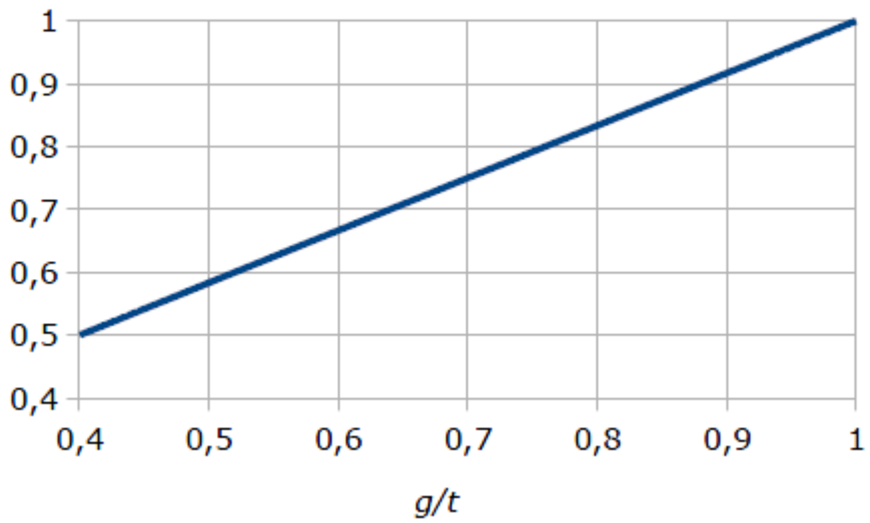
$$f_k = K \cdot f_b^{0.7}$$

Value of the constant *K* is given in the table *3.3* in *EN 1996-1-1*. Where load acts parallel to the direction of the bed joints, the constant *K* should then be multiplied by *0.5* for units of Group *2* and *3*. For masonry made with general purpose mortar where there is a mortar joint parallel to the face of the wall through the length of the wall, the constant *K* is reduced by *0.8*.

The value *K* is multiplied by reduction factor dependent on the ratio *g/t* for shell bedded masonry.

where: ***g*** • is the total width of the mortar strips
***t*** • is the wall thickness

Minimal value of this ratio is *0.4*. The value of the reduction factor is shown in the graph below:




## Characteristic shear strength of masonry

Characteristic shear strength of masonry $f_{vk}$ for general purpose mortar, thin layer mortar in beds of thickness *0.5mm* to *3.0mm* or lightweight mortar with all joints filled is calculated using equation

$$f_{vk} = f_{vko} + 0.4\sigma_d$$

but not greater than

$$f_{vk} \leq 0.065 f_b$$

where: $f_{vk}$ • is the characteristic shear strength of masonry
$f_{vko}$ • is the characteristic initial shear strength, under zero compressive stress
$\sigma_d$ • is the design compressive stress perpendicular to the shear in the member at the levelunder consideration, using the appropriate load combination based on the average verticalstress over the compressed part of the wall that is providing shear resistance
$f_b$ • is the normalised compressive strength of the masonry units

The characteristic shear strength of masonry using general purpose mortar or thin layer mortar in beds of thickness *0.5mm* to *3.0mm* or lightweight mortar having the perpend joints unfilled is calculated using equation

$$f_{vk} = 0.5 f_{vko} + 0.4\sigma_d$$

but not greater than

$$f_{vk} \leq 0.045 f_b$$

where: $f_{vk}$ • is the characteristic shear strength of masonry
$f_{vko}$ • is the characteristic initial shear strength, under zero compressive stress
$\sigma_d$ • is the design compressive stress perpendicular to the shear in the member at the levelunder consideration, using the appropriate load combination based on the average verticalstress over the compressed part of the wall that is providing shear resistance
$f_b$ • is the normalised compressive strength of the masonry units

The characteristic shear strength of shell bedded masonry is calculated using equation

$$f_{vk} = \frac{g}{t} f_{vko} + 0.4\sigma_d$$

but not greater than

$$f_{vk} \leq 0.045 f_b$$

where:
- $f_{vk}$ • is the characteristic shear strength of masonry
- $f_{vko}$ • is the characteristic initial shear strength, under zero compressive stress
- $g$ • is the total width of the mortar strips
- $t$ • is the wall thickness
- $\sigma_d$ • is the design compressive stress perpendicular to the shear in the member at the levelunder consideration, using the appropriate load combination based on the average verticalstress over the compressed part of the wall that is providing shear resistance
- $f_b$ • is the normalised compressive strength of the masonry units

### Characteristic flexural strength of masonry

Characteristic flexural strength of masonry is determined according to the chapter *3.6.3*.

### Modulus of elasticity

Modulus of elasticity is calculated using equation

$$E = K_E \cdot f_k$$

where:
- $f_k$ • is the characteristic compressive strength of masonry
- $K_E$ • is a constant; value is *1000*

## Effective height of masonry walls

Calculation of buckling lengths is based on the chapter *5.5.1.2* of EN 1996-1-1.

Effective height of masonry wall is calculated using equation

$$h_{ef} = \rho_n \cdot h$$

where:
- $h_{ef}$ • is the effective wall height
- $\rho_n$ • is a reduction factor; bottom index represents number of restrained edges of the wall
- $h$ • is the clear storey height of the wall

### Walls restrained at the bottom

Reduction factor $\rho_n$ is calculated using equation

$$\rho_1 = 2$$

### Walls restrained at the top and bottom

Reduction factor $\rho_n$ for walls restrained at the top and bottom is

$$\rho_2 = 1$$

Reduction factor $\rho_n$ of the walls restrained at the top and bottom by RC slabs from both sides (or from one side with bearing of at least *2/3* thickness of the wall) with the eccentricity of the load at the top smaller than 0.25 times the wall thickness is equal to

$$\rho_2 = 0.75$$

### Walls restrained at the bottom and stiffened on one vertical edge

Reduction factor $\rho_n$ for walls restrained at the bottom and stiffened on one vertical edge is calculated according to the following equations:

For

$$h \leq 3.5l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_2 = \frac{1}{1 + \left(\frac{h}{3l}\right)^2} \cdot \rho_1$$

For

$$h > 3.5l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_2 = \frac{3l}{h} \geq 0.6$$

where: ***h*** • is the clear storey height of the wall

***l*** • is the wall length

***$\rho_2$*** • is a reduction factor for walls restrained at the bottom and stiffened on one vertical edge

## Walls restrained at the top and bottom and stiffened on one vertical edge

Reduction factor $\rho_n$ for walls restrained at the top and bottom and stiffened on one vertical edge is calculated according to the following equations:

For

$$h \leq 3.5l$$

$$\rho_3 = \frac{1}{1 + \left(\frac{\rho_2 \cdot h}{3l}\right)^2} \cdot \rho_2$$

For

$$h > 3.5l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_3 = \frac{1.5l}{h} \geq 0.3$$

where: ***h*** • is the clear storey height of the wall

***l*** • is the wall length

***$\rho_3$*** • is a reduction factor for walls restrained at the top and bottom and stiffened on one vertical edge

## Walls restrained at the bottom and stiffened on both vertical edges

Reduction factor $\rho_n$ for walls restrained at the bottom and stiffened on both vertical edges is calculated according to the following equations:

For

$$h \leq 1.15l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_3 = \frac{1}{1 + \left(\frac{h}{l}\right)^2} \cdot \rho_1$$

For

$$h > 1.15l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_3 = \frac{l}{h}$$

where: ***h*** • is the clear storey height of the wall

***l*** • is the wall length

***$\rho_3$*** • is a reduction factor for walls restrained at the bottom and stiffened on both vertical edges

## Walls restrained at the top and bottom and stiffened on both vertical edges

Reduction factor $\rho_n$ for walls restrained at the top and bottom and stiffened on both vertical edges is calculated according to the following equations:

For

$$h \leq 1.15l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_4 = \frac{1}{1+\left(\frac{\rho_2 \cdot h}{3l}\right)^2} \cdot \rho_2$$

For

$$h > 1.15l$$

is the raduction factor $\rho_n$ calculated using equation

$$\rho_4 = \frac{0.5l}{h}$$

where: ***h*** • is the clear storey height of the wall
***l*** • is the wall length
***$\rho_4$*** • is a reduction factor for walls restrained at the top and bottom and stiffened on both vertical edges

## Ultimate load states

The ultimate limit state design is based on rules given in EN 1996-1-1, chapter *6*.

### Unreinforced masonry walls subjected to mainly vertical loading

Basic equation for design of unreinforced masonry walls subjected to mainly vertical loading is (in accordance with paragraph *6.1.2*):

$$N_{Ed} \leq N_{Rd}$$

where: ***$N_{Ed}$*** • is the design value of the vertical load
***$N_{Rd}$*** • is the design value of the vertical resistance of the wall

The design value of the vertical resistance $N_{Rd}$ is calculated using formula

$$N_{Rd} = \Phi \cdot A \cdot f_d$$

where: ***$\Phi$*** • is the capacity reduction factor
***A*** • is the total area of cross-section
***$f_d$*** • is the design compressive strength of the masonry

The design compressive strength $f_d$ of the cross-sections with area smaller than *$0.1m^2$* is multiplied in accordance with *6.1.2.1.(3)* by factor

$$0.7 + 3A$$

kde je: ***A*** • is the total area of cross-section

The calculation of reduction factor for slenderness and eccentricity $\Phi_i$ at the top and bottom of the wall is based on on a rectangular stress block:

$$\Phi_i = 1 - 2\frac{e_i}{t}$$

where: ***$\Phi_i$*** • is the reduction factor for slenderness and eccentricity
***$e_i$*** • is the eccentricity at the top or bottom of the wall
***t*** • is the wall thickness

The eccentricity at the top or bottom of the wall $e_i$ is calculated using equation

$$e_i = \frac{M_{id}}{N_{id}} + e_{init} \geq 0.05t$$

where: ***$M_{id}$*** • is the design value of the bending moment at the top or bottom of the wall caused by the eccentricity of the floor load at the support
***$N_{id}$*** • is the design value of the vertical force at the top or bottom of the wall
***$e_{init}$*** • is the initial eccentricity
***t*** • is the wall thickness

The initial eccentricity is calculated in accordance with *5.5.1.1(4)* using equation

$$e_{init} = \frac{h_{ef}}{450}$$

where: ***$h_{ef}$*** • is the effective height of the wall

The reduction factor within the middle height of the wall $\Phi_m$ is calculated according to the annex *G*. Following equation is used for the walls with rectagular cross-sections:

$$\Phi_m = A_1 e^{\frac{-u^2}{2}}$$

where is

$$A_1 = 1 - 2\frac{e_{mk}}{t}$$

and

$$u = \frac{\lambda - 0.063}{0.73 - 1.17\frac{e_{mk}}{t}}$$

where is

$$\lambda = \frac{h_{ef}}{t_{ef}}\sqrt{\frac{f_k}{E}}$$

The eccentricity at the middle height of the wall $e_{mk}$ is calculated using equation

$$e_{mk} = e_m + e_k \geq 0.05t$$

where:
- **$e_m$** • is the eccentricity due to loads
- **$e_k$** • is the eccentricity due to creep
- **$t$** • is the wall thickness

The eccentricity due to loads $e_m$ is calculated using equation

$$e_m = \frac{N_{md}}{M_{md}} \pm e_{init}$$

where:
- **$M_{md}$** • is the design value of the bending moment at the middle of the height of the wall resulting from the moments at the top and bottom of the wall
- **$N_{md}$** • is the design value of the vertical force at the middle height of the wall
- **$e_{init}$** • is the initial eccentricity

The eccentricity due to creep $e_k$ is calculated using equation

$$e_k = 0.002\Phi_\infty \frac{h_{ef}}{t_{ef}}\sqrt{t \cdot e_m}$$

where:
- **$h_{ef}$** • is the effective height of the wall
- **$t_{ef}$** • is the effective thickness of the wall
- **$\Phi_\infty$** • is the final creep coefficient
- **$t$** • is the wall thickness
- **$e_m$** • is the eccentricity due to loads

For walls fulfilling condition

$$\lambda_c \leq 15$$

is the eccentricity due to creep $e_k$ equal to zero.

The design value of the vertical force is calculated using iteration of the deformation along the cross-section area under conditions written in *6.1.1(2)* for more complicated shapes of the cross-sections. the stress-strain relationship diagram is taken to be rectangular. Normal force can't be equal to zero and can't be located outside the cross-section.

## Unreinforced masonry walls subjected to lateral loading

Basic equation for design of unreinforced masonry walls subjected to lateral loading is (in accordance with paragraph *6.3.1*):

$$M_{Ed} \leq M_{Rd}$$

where:
- **$M_{Ed}$** • is the design value of the moment
- **$M_{Rd}$** • is the design value of the bending resistance

The design value of the bending resistance $M_{Rd}$ is calculated using equation

$$M_{Rd} = f_{xd} \cdot Z$$

where:
- **$f_{xd}$** • is the design flexural strength appropriate to the plane of bending
- **$Z$** • is the elastic section modulus

When a vertical load is present, the favourable effect of the vertical stress is considered using equation in accordance with *6.3.1(4)(i)*:

$$f_{xdl.app} = f_{xdl} + \sigma_d$$

where:
- $f_{xd}$ • is the design flexural strength of masonry with the plane of failure parallel to the bed joints
- $\sigma_d$ • is the design compressive stress on the wall, not taken to be greater than $0.2f_d$

## Unreinforced masonry walls subjected to shear loading

Shear is analysed according to *6.2*, basic equation is

$$V_{Ed} \leq V_{Rd}$$

where:
- $V_{Ed}$ • is the design value of the shear force
- $V_{Rd}$ • is the design value of the shear resistance

The design value of the shear resistance $M_{Rd}$ is calculated using equation

$$V_{Rd} = f_{vd} \cdot A_c$$

where:
- $V_{Rd}$ • is the design value of the shear resistance
- $f_{vd}$ • is the design value of the shear strength of masonry
- $A_c$ • is the area of compressed part of the cross-section

## Buckling of columns with more complicated cross-sections

Buckling verification of columns, that have more complicated shapes of cross-sections, is performed with the help of an effective cross-section. The effective cross-section is a rectangle, that is selected according to the following rules:

- The area of the effective cross-section is identical to the area of the real cross-section
- The ratio $W_y/W_z$ is identical for the real and effective cross-sections

# Serviceability load states

Verification of serviceability limit state is based on standard EN 1996-1-1, paragraph *7.2(5)*. Dimension limits of the masonry walls are checked in accordance with Annex *F*. Verification of the serviceability limit state uses clear height of the wall, not the effective height.

Following dimensions are used for the verification of the serviceability limit state:

- $t$ • the wall thickness
- $h$ • the clear height of the wall
- $l$ • the wall length

Due to validity limits of the Annex *F* ,the minimal thickness of the wall is

$$t \geq 0.1m$$

## Walls restrained at the bottom

Walls restrained at the bottom should satisfy following formula:

$$\frac{h}{t} \leq 15$$

## Walls restrained at the top and bottom

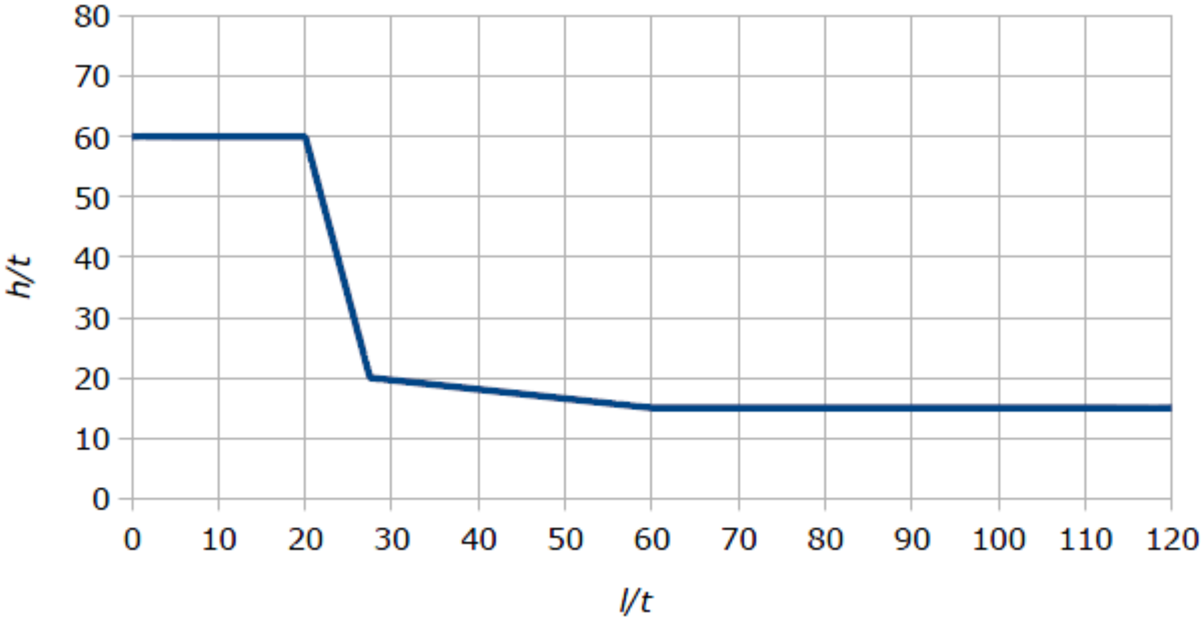
Walls restrained at the top and bottom should satisfy following formula:

$$\frac{h}{t} \leq 30$$

## Walls restrained at the bottom and stiffened on one vertical edge

The maximal values of *h/t* ratio for walls restrained at the bottom and stiffened on one vertical edge are shown in following graph:

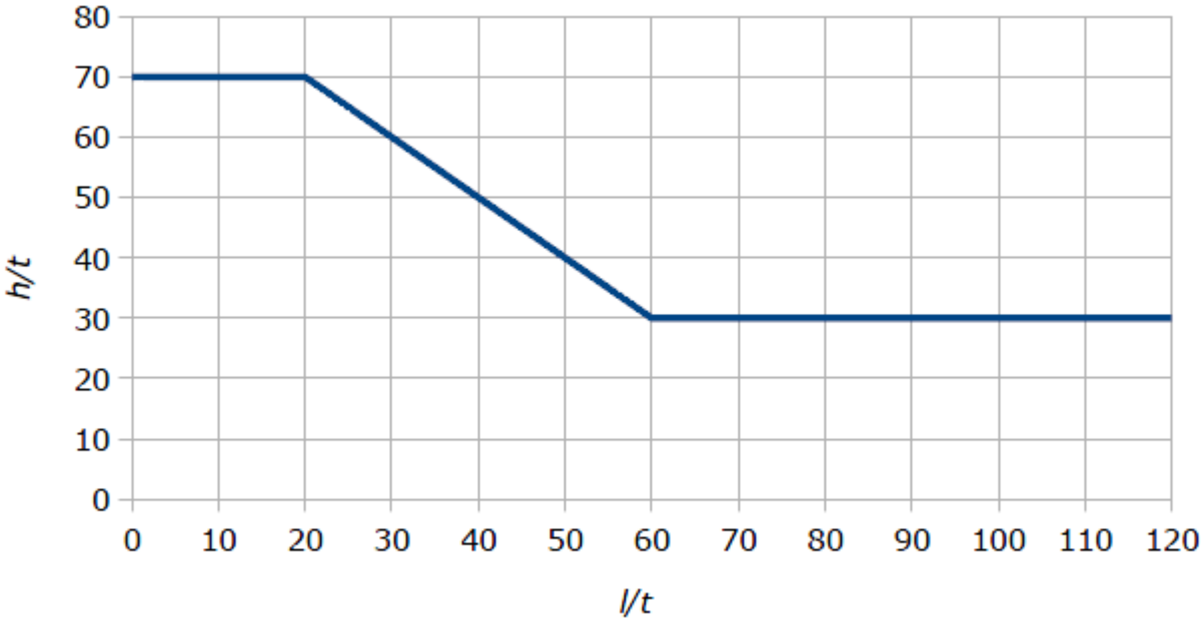
 




*Limiting values of h/t ratio depending on l/t ratio*

## Walls restrained at the top and bottom and stiffened on one vertical edge

The maximal values of *h/t* ratio for walls restrained at the top and bottom and stiffened on one vertical edge are shown in following graph:

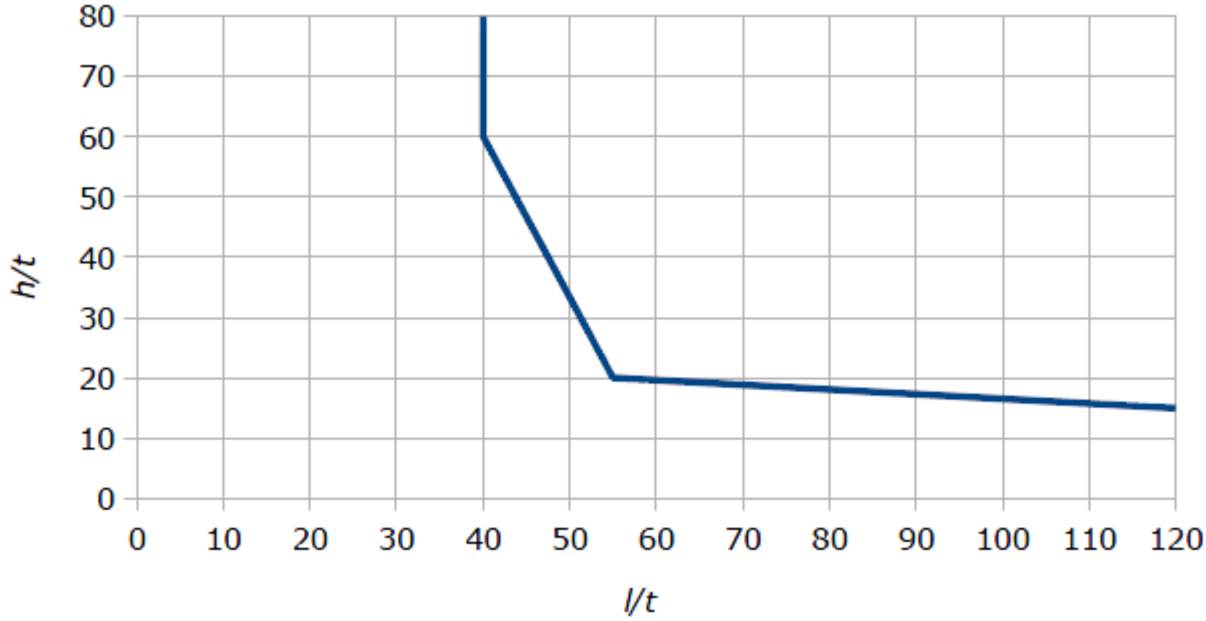



*Limiting values of h/t ratio depending on l/t ratio*

## Walls restrained at the bottom and stiffened on both vertical edges

The maximal values of *h/t* ratio for walls restrained at the bottom and stiffened on both vertical edges are shown in following graph:

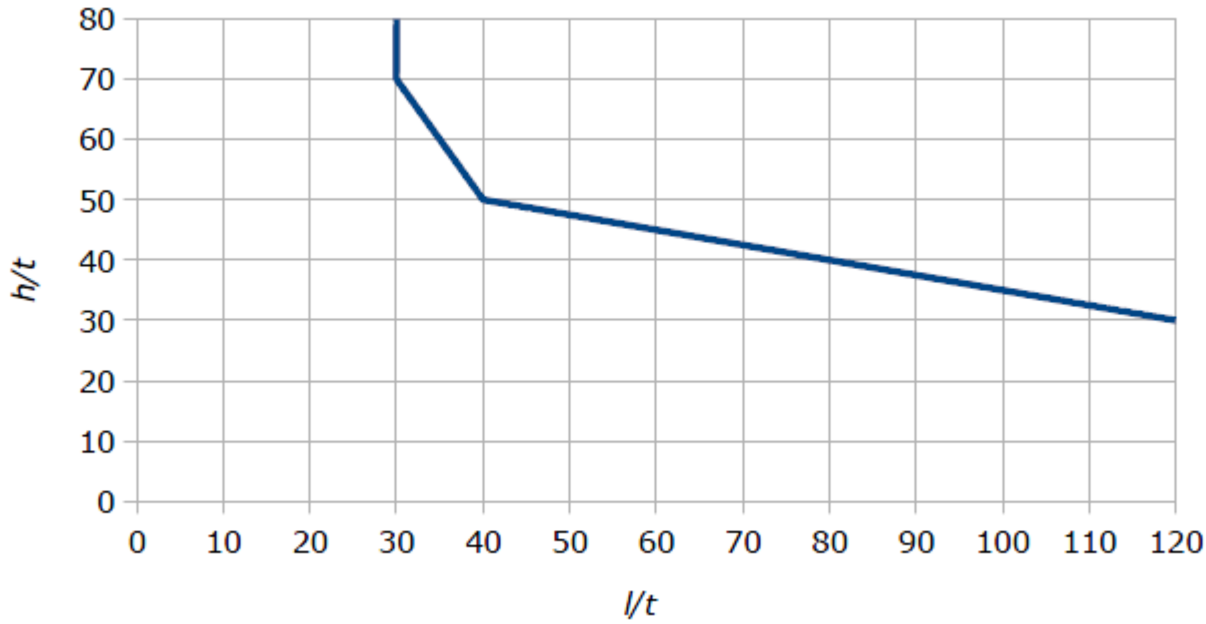
 




*Limiting values of h/t ratio depending on l/t ratio*

## Walls restrained at the top and bottom and stiffened on both vertical edges

The maximal values of *h/t* ratio for walls restrained at the top and bottom and stiffened on both vertical edges are shown in following graph:




*Limiting values of h/t ratio depending on l/t ratio*

# National annexes

Following partial factors are used for individual national annexes:

| **Factor** | **EN 1996-1-1** | **Czechia** | **Slovakia** | **Poland** |
|---|---|---|---|---|
| **$\gamma_M$ - Units of Category I, designed mortar** | 2,0 | 2,00 (2,50) | 2,00 | 1,70 |
| **$\gamma_M$ - Units of Category I, prescribed mortar** | 2,20 | 2,20 (2,70) | 2,20 | 2,00 |
| **$\gamma_M$ - Units of Category II, any mortar** | 2,50 | 2,50 (3,00) | 2,50 | 2,20 |

Values in brackets for the national annex "**Czech Republic**" are used for aac units.

# Loading

# Snow load

The calculations are performed in accordance with the standard EN 1991-1-3. These procedures are included:

## Snow load

The snow load on the roof is calculated using formula *(5.1)*:

$$s = \mu_i C_e C_t s_k$$

where is:

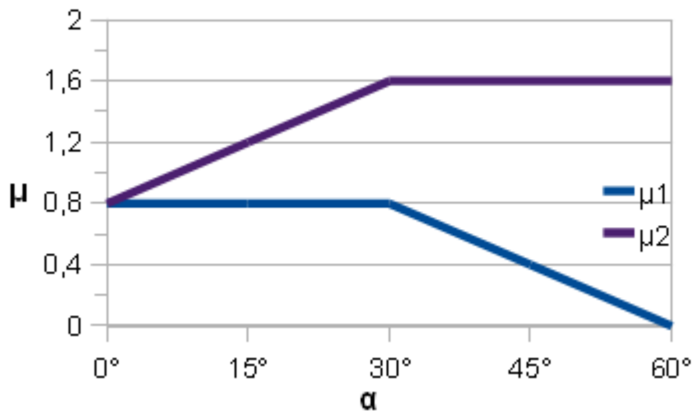
| | | |
|---|---|---|
| $\mu_i$ | • | Snow load shape coefficient, values are described below |
| $s_k$ | • | Characteristic value of the snow load on the ground. It can be entered with the help of snow map or entered manually. |
| $C_e$ | • | Exposure coefficient |
| $C_t$ | • | Thermal coefficient |

The exposure coefficient $C_e$ is obtained according to the selected topography. The values are in accordance with the table *5.1* of EN 1991-1-3:

| **Topography** | **Coefficient $C_e$** |
|---|---|
| Windswept | 0.8 |
| Normal | 1.0 |
| Sheltered | 1.2 |

## Mono-pitched roofs

The shape coefficient $\mu_1$ in accordance with figure *5.1* of EN 1991-1-3 is used for mono-pitched roofs. The value of this coefficient depends on the roof pitch:

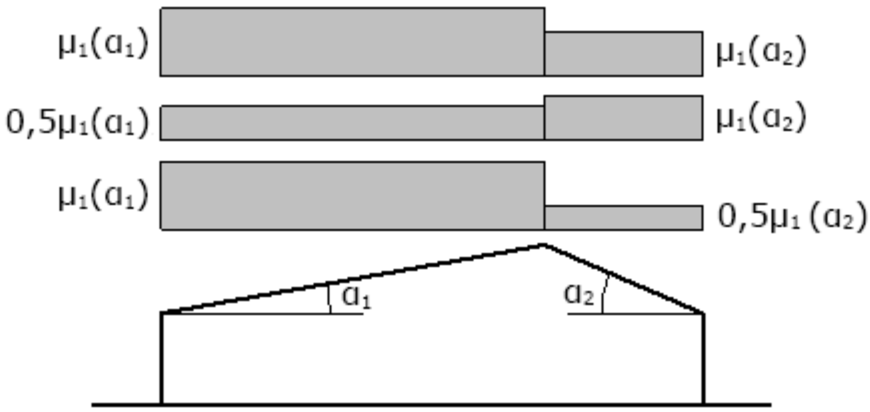



*Shape coefficients $\mu_1$ and $\mu_2$*

The value of the coefficient $\mu_1$ is equal to *0.8* if the sliding of the snow is prevented (snow fences etc.).

## Duo-pitched roofs

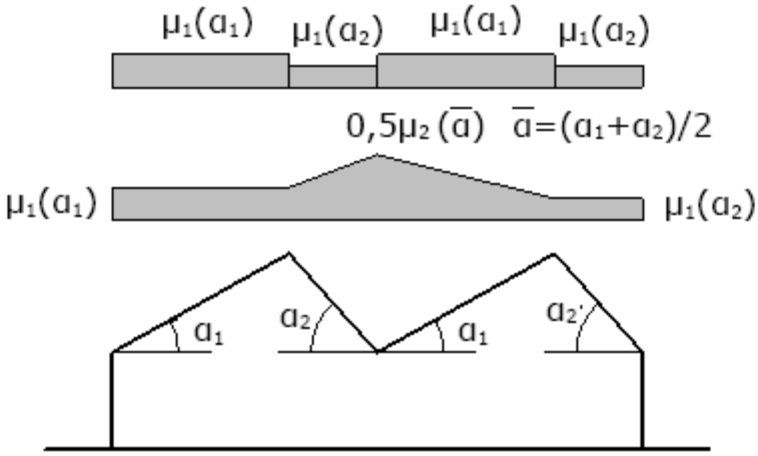
The shape coefficient $\mu_1$ in accordance with figure *5.1* of EN 1991-1-3 is used for duo-pitched roofs. The figure showing the dependency of the coefficient value and the roof pitch is shown in the chapter "**Mono-pitched roofs**". The value of the coefficient $\mu_1$ is equal to *0.8* if the sliding of the snow is prevented (snow fences etc.). These three load cases are created for duo-pitched roof in accordance with chapter *5.3.3*:



*Load cases for duo-pitched roofs*

## Multi-span roofs

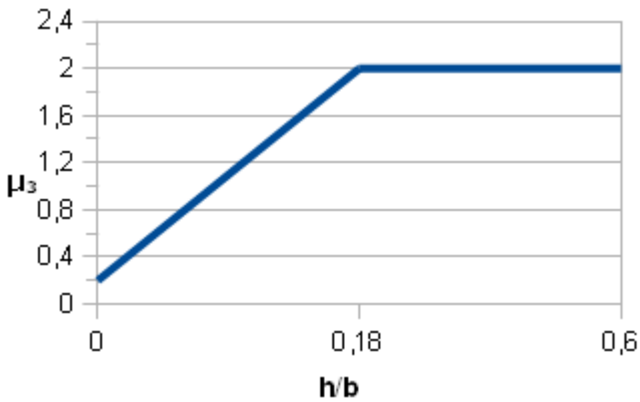
The shape coefficients $\mu_1$ and $\mu_2$ in accordance with figure *5.1* of EN 1991-1-3 is used for multi-span roofs. The figure showing the dependency of the coefficients values and the roof pitch is shown in the chapter "**Mono-pitched roofs**". The value of the coefficient $\mu_1$ is equal to *0.8* if the sliding of the snow is prevented (snow fences etc.). These load cases for undrifted and drifted snow are created in accordance with chapter *5.4*:




*Load cases for multi-span roofs*

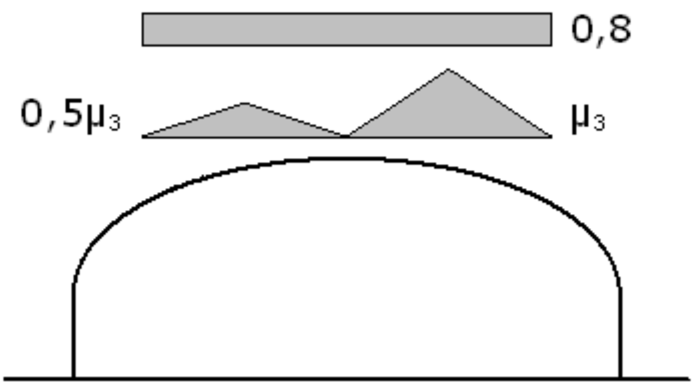
## Cylindrical roofs

The shape coefficient $\mu_3$ in accordance with figure *5.3.5(1)* of EN 1991-1-3 is used for cylindrical roofs. Its value is depending on ratio *h/b*, where *h* is the height of cylindrical roof and *b* is the roof span. The coefficient values are shown in the following figure:




*Coefficient $\mu_3$*

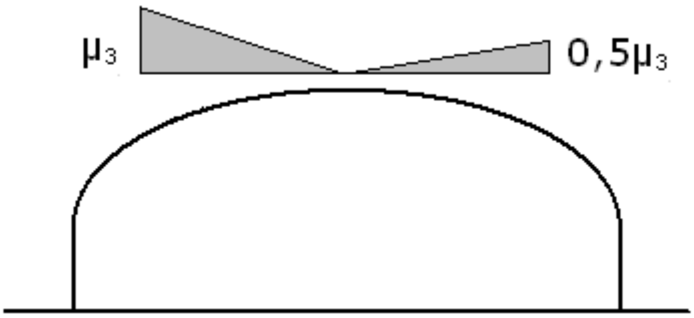
The snow load is considered in the parts where the roof pitch is smaller than *60°* according to the chapter *5.3.5*. Following load cases are considered:




*Load cases for cylindrical roofs*

Following load case based on the standards CSN/STN 73 0035 is considered additionally for Czech and Slovak nation annexes :



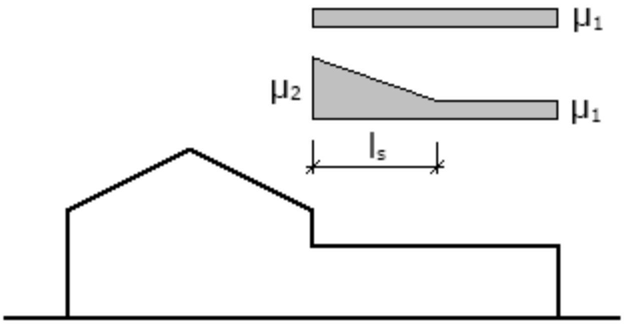

*Drifted snow in accordance with Czech and Slovak national annex*

This load case is considered under these circumstances:

- This scheme will be considered for all cylindrical roofs with ratio *h/b* greater than *1/8*.
- This scheme will be considered for all cylindrical roofs in snow areas *IV* and *V*.

## Roofs abutting and close to taller construction works

The shape coefficients for these roofs are calculated according to the chapter *5.3.6* of EN 1991-1-3. These load cases are considered:




*Load cases for abutting roofs*

The shape coefficients $\mu_1$ and $\mu_2$ are calculated using following formulas:

$$\mu_1 = 0.8$$

$$\mu_2 = \mu_s + \mu_w$$

| | | |
|---|---|---|
| Where is: | $\mu_s$ | • Snow load shape coefficient due to sliding of snow from the upper roof |
| | $\mu_w$ | • Snow load shape coefficient due to wind |

The value of the coefficient $\mu_s$ is equal to 0 for $\alpha \leq 15°$. The following formula is used for $\alpha > 15°$:

$$\mu_s = \frac{\mu b_s}{l_s}$$

| | | |
|---|---|---|
| Where is: | $\mu$ | • The shape coefficient for the upper roof, the value is *0.8* |
| | $b_s$ | • Horizontal distance of the roof ridge and fascia |
| | $l_s$ | • Length of drifted snow |

The coefficient $\mu_w$ is calculated using following formula:

$$\mu_w = \frac{b_1 + b_2}{2h} \leq \frac{\gamma h}{s_k}$$

| | | |
|---|---|---|
| Where is: | $b_1$ | • Span of upper roof |
| | $b_1$ | • Span of lower roof |
| | $h$ | • Vertical distance between lower roof and fascia of upper roof |
| | $\gamma$ | • The weight density $2 kN/m^3$ |

The drift length $l_s$ is calculated using formula

$$l_s = 2h$$

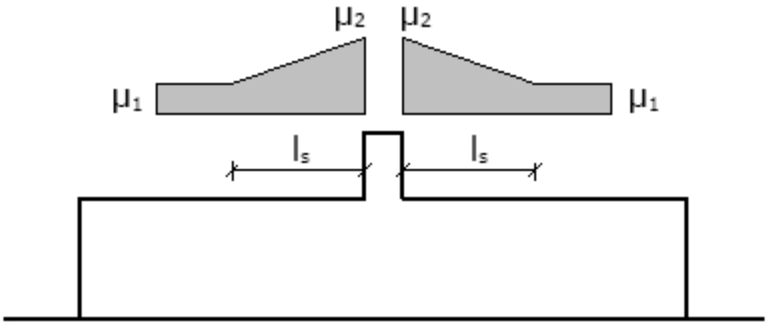
| | | |
|---|---|---|
| where is: | $h$ | • Vertical distance between lower roof and fascia of upper roof |

The drift length $l_s$ is limited by the interval *<5m;15m>*. The shape coefficient is calculated using lineat interpolation between $\mu_1$ and $\mu_2$ for buildings where $l_s$ is greater than span of lower roof.

### Drifting at projections and obstructions

The following loading figure is created in accordance with the chapter *6.2* of EN 1991-1-3 for the roofs with obstructions:



*Loading figure for obstructions*

The coefficients $\mu_1$ and $\mu_2$ are calculated using formula *(6.1)*:

$$\mu_1 = 0.8$$
$$\mu_2 = \gamma h / s_k$$

| | | | |
|---|---|---|---|
| where is: | ***γ*** | • | The weight density of snow *2kN/m³* |
| | ***h*** | • | The obstruction height |
| | ***$s_k$*** | • | The characteristic value of the snow load on the ground. |

### Snow overhanging the edge of a roof

Snow overhanging the edge of a roof is calculated in accordance with chapter *6.3*. Following formula is used:

$$S_e = \frac{ks^2}{\gamma}$$

| | | | |
|---|---|---|---|
| where is: | ***k*** | • | The coefficient to take account of the irregular shape of the snow |
| | ***s*** | • | The most onerous undrifted load case appropriate for the roof |
| | ***γ*** | • | The weight density of snow *3kN/m³* |

The coefficient *k* is calculated using formula:

$$k = 3d$$

But following expression has to be fulfilled:

$$k \leq d\gamma$$

| | | | |
|---|---|---|---|
| where is: | ***d*** | • | The depth of the snow layer |

### Snow load on snowguards and other obstacles

Snow load on snowguards and other obstacles is calculated according to the chapter *6.4* of EN 1991-1-3. Following formula is used:

$$F = s \cdot b \cdot sin(\alpha)$$

| | | | |
|---|---|---|---|
| where is: | ***s*** | • | The snow load on the roof relative to the most onerous undrifted snow load case |
| | ***b*** | • | The width on plan (horizontal) between the guards or obstacles |
| | ***α*** | • | The roof pitch |

## Wind load

The calculations are performed in accordance with the standard EN 1991-1-4. These procedures are included:

### Peak velocity pressure

The calculation of the peak velocity pressure is performed in accordance with chapter *4* of EN 1991-1-4.

The basic wind velocity $v_b$ is calculated using formula *(4.1)*:

$$v_b = c_{dir} \cdot c_{season} \cdot v_{b,0}$$

where is:

- $v_b$ • The basic wind velocity defined at *10m* above ground of the terrain category *II*
- $c_{dir}$ • The directional factor
- $C_{season}$ • The season factor
- $v_{b,0}$ • The fundamental value of the basic wind velocity (entered manually or selected from the map of wind regions)

The mean wind velocity $v_m(z)$ at the height $z$ above the terrain is calculated using following formula:

$$v_m(z) = c_r(z) \cdot c_0(z) \cdot v_b$$

where is:

- $c_r(z)$ • The roughness factor
- $c_o(z)$ • The orography factor calculated in accordance with annex *A.3* of EN 1991-1-4
- $v_b$ • The basic wind velocity defined at *10m* above ground of the terrain category *II*

For the interval

$$z_{min} \leq z \leq z_{max}$$

the roughness factor $c_r(z)$ is calculated using formula

$$c_r(z) = k_r \cdot ln\left(\frac{z}{z_0}\right)$$

For

$$z \leq z_{min}$$

the following formula is used:

$$c_r(z) = c_r(z_{min})$$

where is:

- $z_0$ • The roughness length according to the table *4.1* of EN 1991-1-4
- $z_{min}$ • The minimum height according to the table *4.1* of EN 1991-1-4
- $z_{max}$ • The maximum height *200m*
- $k_r$ • The terrain factor

The terrain factor $k_r$ is calculated using formula:

$$k_r = 0.19\left(\frac{z_0}{z_{0,II}}\right)^{0.07}$$

- $z_{0,II}$ • The roughness length for the terrain category *II*

The turbulence intensity $I_v(z)$ shall be calculated for the interval

$$z_{min} \leq z \leq z_{max}$$

using formula

$$I_v(z) = \frac{k_I}{c_0(z) \cdot ln(z/z_0)}$$

For

$$z \leq z_{min}$$

the following formula is used:

$$I_v(z) = I_v(z_{min})$$

where is:

- $k_I$ • The turbulence factor, the value is *1.0*
- $c_0$ • The orography factor
- $z_0$ • The roughness length according to the table *4.1* of EN 1991-1-4

The peak velocity pressure $q_p(z)$ is calculated using formula *(4.8)*:

$$q_p(z) = [1 + 7I_v(z)] \cdot \frac{1}{2} \cdot \rho \cdot {v_m}^2$$

where is: $\rho$ • The air density

### Wind pressure on surfaces

The wind pressure $w_e$ acting on the external surfaces is calculated using following formula:

$$w_e = q_p(z_e) \cdot c_{pe}$$

where is:
- $q_p(z_e)$ • The peak velocity pressure
- $z_e$ • The reference height
- $c_{pe}$ • The pressure coefficient

The pressure coefficient $c_{pe}$ depends on the loading are $A$. The values $c_{pe,1}$ for members with loaded area of $1m^2$ or less and $c_{pe,10}$ for members with loaded area of $10m^2$ or larger are defined in the standard. The linear interpolation is used for members with loaded area between $1m^2$ and $10m^2$:

$$c_{pe} = c_{pe,1} + (c_{pe,10} - c_{pe,1}) \cdot log_{10}(A)$$

### Pressure coefficients for roofs

The pressure coefficients for roofs are selected in accordance with chapters *7.2.3* (flat roofs), *7.2.4* (mono-pitched roofs), *7.2.5* (duo-pitched roofs) and *7.2.6* (hip roofs) of EN 1991-1-4. The most unfavourable value is used for the roofs with undeterminable topology (for example the rectangular building with one hip and one gable).

### Pressure coefficients for roofs

The pressure coefficients for walls are selected in accordance with chapter *7.2.2* of EN 1991-1-4.

### Pressure coefficients for vaulted roofs and domes

Pressure coefficients for vaulted roofs and domes are selected in accordance with chapter *7.2.8* of0 EN 1991-1-4.

## National annexes

### Czech Republic

Following rules are used for the determination of the **snow load** according to CSN EN 1991-1-3/NA:

- The map and list of snow regions is used according to the amendment *Z1 CSN EN 1991-1-3:2005*.
- The load figure according to the figure *NA.1* is used for cylindrical roofs with *h/b> 1/8* or with snowguards and in snow regions *IV* and *V*.

Following rules are used for the determination of the **wind load** according to CSN EN 1991-1-4/NA:

- The map and list of wind regions is used according to the *CSN EN 1991-1-4:2007*.

### Slovakia

Following rules are used for the determination of the **snow load** according to STN EN 1991-1-3/NA:

- Tha values of $s_k$ are calculated according to the chapter *NA.2.8* of *STN EN 1991-1-3/NA1:2012*.
- The map and list of snow regions is used according to the *Annex C* of *STN EN 1991-1-3/NA1:2012*.
- The map and list of regions with accidental load is used according to the *Annex C* of *STN EN 1991-1-3/NA1:2012*.

Following rules are used for the determination of the **wind load** according to STN EN 1991-1-4/NA:

- The map and list of wind regions is used according to the *STN EN 1991-1-4:2008*.
- Pressure coefficients $c_{pe}$ for flat roofs are selected according to the table *7.2/NA*.
- Pressure coefficients $c_{pe}$ for mono-pitched roofs are selected according to the table *7.3/NA*.
- Pressure coefficients $c_{pe}$ for duo-pitched roofs are selected according to the table *7.4/NA*.
- Pressure coefficients $c_{pe}$ for hip roofs are selected according to the table *7.5/NA*.

### Poland

Following rules are used for the determination of the **snow load** according to PN EN 1991-1-3/NA:

- The map and list of snow regions is used according to the *NB1.7* of *PN EN 1991-1-3:2005*.

Following rules are used for the determination of the **wind load** according to PN EN 1991-1-4/NA:

- The map and list of wind regions is used according to the *PN EN 1991-1-4:2008*.

# Sector

# Modelling principles

Following terminology is used in the software:

- **Sector** - The fundamental part of the cross-section. It is straight centre line of one part of the cross-section between two nodes. The sector is given by the beginning and end nodes and by the thickness.
- **Node** - The points on the beginning and end of the sector. The position is given by the coordinates *[X,Y]*.
- **Branch** - The group of connected sectors, the program uses two types: closed branches (cells) and open branches.
- **Cell** - Closed branch, that have beginning and end node with identical coordinates
- **Open branch** - Branch, that does not have identical beginning and end nodes

## Modelling principles

Following rules have to be respected when modelling a cross-section:

- The section cannot contain any separated parts
- The cells have to be entered before open branches. The cell cannot be inserted into the cross-section, where any open branch already exists.
- The cells have to be connected by at least one sector (two neighbouring nodes). The input of new cell has to start in some node of already existing cell.
- The existing nodes of closed cells are considered only at the beginning of the input of new cell. If some node in existing cell is selected after the input of new node, no connection between existing and new cell is considered in this point.
- If there is some existing cell, first open branch has to begin in some node placed on this cell. Position of first node of open branch is not limited for empty project.
- Nodes that follow the first node in the open branch are not considered as connected to other branches or cells, even if they have identical coordinates with another nodes. This rule avoids input of closed cells using the input of open branches.

If there are two nodes with identical coordinates, the calculation does not consider any connection in this point.

# Cross-sectional characteristics

Following characteristics are calculated:

## Warping coordinate

The warping coordinate is doubled area circumscribed by the radius vector between defined pole and point on the centre line of the branch. The values are modified in cells due to the effect of constant shear flow.

## Main warping coordinate

The warping coordinate φ is considered as a main coordinate $\omega$, if the beginning is selected in that way, that following formula is valid:

$$\int_A \phi \mathrm{d}A = 0$$

It means that the warping coordinate of whole cross-section is equal to *0*.

The warping coordinate depends on the pole position. Usually, poles in the centre of gravity or in the shear centre are used.

## The rigidity moment in simple torsion

The rigidity moment in simple torsion is calculated for open branches with the help of this expression:

$$I_K = \frac{1}{3}\sum_i I_i \delta_i^3$$

| | | |
|---|---|---|
| Where is: | $l_i$ | • The length of *i*-sector |
| | $\delta_i$ | • The thickness of *i*-sector |

Following expression is used for cells:

$$I_K = D\Omega$$

| | | |
|---|---|---|
| Where is: | $D$ | • The parameter dependent on the shear flow in the cell |
| | $\Omega$ | • The area surrounded by the centre line of the cell multiplied by two |

### Shear centre

The shear centre is a point, through which goes the resultant of inner shear forces in the cross-section. If the resultant of external transverse forces goes also through this point, the bending of the member does not cause the torsional stresses. The warping characteristics are calculated relatively to this point very often.

### Warping constant

The warping constant is necessary for calculation of shear stress in warping. It is calculated according to the expression

$$S_\omega(s) = \int_0^s \omega\delta \mathrm{d}s$$

| | | |
|---|---|---|
| Where is: | ***s*** | • The centre line of cross-section |
| | ***ω*** | • The main warping coordinate |
| | ***δ*** | • The sector thickness |

### Warping moment of inertia

The warping moment of inertia describes the stiffness in warping. It is necessary for calculation the normal stress induced by warping. It is calculated using expression

$$I_\omega = \int_A \omega^2 \mathrm{d}A$$

| | | |
|---|---|---|
| Where is: | ***A*** | • The cross-sectional area |
| | ***ω*** | • The main warping coordinate |

# Section

## Modelling principles

Following terminology is used in the software:

- **Profile** - pre-defined shape of cross-section from default database (e.g. hot-rolled steel profile). The geometry cannot be modified.
- **Shape** - general closed polygon with assigned material
- **Opening** - general closed polygon without assigned material, any opening has to be inside of one shape or profile. No overlapping is permitted.

### Modelling principles

Following rules have to be respected when modelling a cross-section:

- The number of particular objects is not limited
- The objects (profiles, shapes, openings) cannot intersect or overlap each other
- Any contact of two objects should be done from outer edge, the contact from inner edge may cause incorrect results.
- The profile material has to respect the database sorting (e.g. for profile *IPE200*, it is possible to select only steel grades, not concrete strength classes). This limitation is valid only for profiles, it is possible to convert profile into shape and change the material.
- It is not necessary to insert opening in cases, where one shape or profile is inserted into another one (e.g. steel I-profile in concrete column).

## Cross-sectional characteristics

The cross-sectional characteristics can be calculated in two different ways:

- **Real cross-sectional characteristics** are calculated according to the geometry of the cross-section and are not affected by the material.
- **Ideal cross-sectional characteristics** are important for combined cross-sections with more different materials (e.g. concrete and steel). Ideal characteristics are the characteristics recalculated for the material, that can be specified in the upper part of the frame with results. Recalculation is based on the proportion of moduli of elasticity. This ensures, that all parts are deformed in the same way when exposed to tension, compression or bending. The areas and moments of inertia are multiplied by the factor $n$:

$$n = \frac{E_0}{E_i}$$

| | | |
|---|---|---|
| Where is: | $E_0$ | • The modulus of elasticity of corresponding object |
| | $E_i$ | • The modulus of elasticity of ideal material |

## Available cross-sectional characteristics

Following characteristics are calculated:

- **The centre of gravity in coordinate system** - $x_T$ and $y_T$
- **The cross-sectional area** - A

$$A = \int_A \mathrm{d}A$$

- **The cross-sectional perimeter** - $P$
- **The centre of gravity with respect of the left bottom corner of minimum cross-sectional envelope** - $y_{cg}$, $z_{cg}$
- **Moments of inertia** - $I_y$, $I_z$, $D_{yz}$

$$I_y = \int_A z^2 \mathrm{d}A$$
$$I_z = \int_A y^2 \mathrm{d}A$$
$$D_{yz} = \int_A yz \mathrm{d}A$$

- **Rotation of main axes** - $\varphi$
- **Radii of inertia** - $i_y$, $i_z$
- **Polar moment and radius of inertia** - $I_p$, $i_p$
- **Rigidity moment in simple torsion** - $I_k$
- **Cross-sectional moduli in edges** - $W_{y1}$, $W_{y2}$, $W_{z1}$ , $W_{z2}$

# Parametric temperature curve

## Calculation of parametric temperature curve

The fundamental problem of the fire resistance analysis is the determination of the design fire model (dependency of gas temperature on time). The available options are described in the standard EN 1991-1-2 (Actions on structures exposed to fire). This standard contains basic models represented by nominal curves, simplified methods and advanced fire models. Problems of more complicated models are the difficulty of calculations and obtaining of adequate analysis parameters. The usage of nominal temperature curves is easy and efficient. However, results may be inaccurate, mainly for long lasting fires. The nominal curves, contrary to parametric curve, don't contain cooling phase, i.e. the reduction of gas temperature after burning all fire load. Another advantage of the parametric curve is that it takes into account the compartment size, size of openings in walls, structure materials, type of occupancy and active fire fighting measures.

The validity of parametric temperature curve is limited by the maximum floor area $500 m^2$ and the maximum wall height $4 m$. The ceiling can't have any openings.

The calculation of curve course is performed according to the Annex A of EN 1991-1-2.

# Engineering manuals

## Introduction

This User Manual is intended for all users of the Fin EC structural analysis programs. Simple examples are used to explain the principles of the basic structural analysis programs (Fin 2D and Fin 3D) as well as of the design programs for concrete, steel, timber and masonry structures.

The examples show procedures for analyses of particular tasks in selected programs. Due to similar principles of work, each example can be used as guidance for other programs of the same type. For instance, the example "**Reinforced concrete column**" shows work with the "**Concrete**", however the same principles of quick design of a section subject to a combination of internal forces can be applied in all other design programs for concrete, steel, timber and masonry structures.

The User Manual contains the following examples:

### Reinforced concrete column

This example shows design of a reinforced concrete cross-section subject to given combination of internal forces in the program "**Concrete**". The same principles of design are used in all other design programs – "**Concrete Fire**", "**Steel**", "**Steel Fire**", "**Timber**", "**Timber Fire**" and "**Masonry**". The example also shows the procedure for composition of output documentation.

### Punching shear

In this example, work procedure in the program "**Punching**" is explained. Apart from that, the procedure for editing headers of output documents, which can also be applied in other programs Fin EC, is described.

### Timber truss

The example shows the procedure for modelling a timber truss using the generator of structures in the program "**Fin 2D**" and subsequent design of members in the program "**Timber**". It also describes utilization of design groups and collective definition of calculation parameters in the design program. This example is also suitable for users of the program "**Fin 3D**".

### 3D structure

This example shows analysis of a 3D structure in the program "**Fin 3D**" and subsequent design of members in the program "**Steel**". Apart from gradual definition of the structure in the program "**Fin 3D**" using joints and members, work with the design groups is described. These procedures can also be used in the program "**Fin 2D**" in connection with any of the design programs.

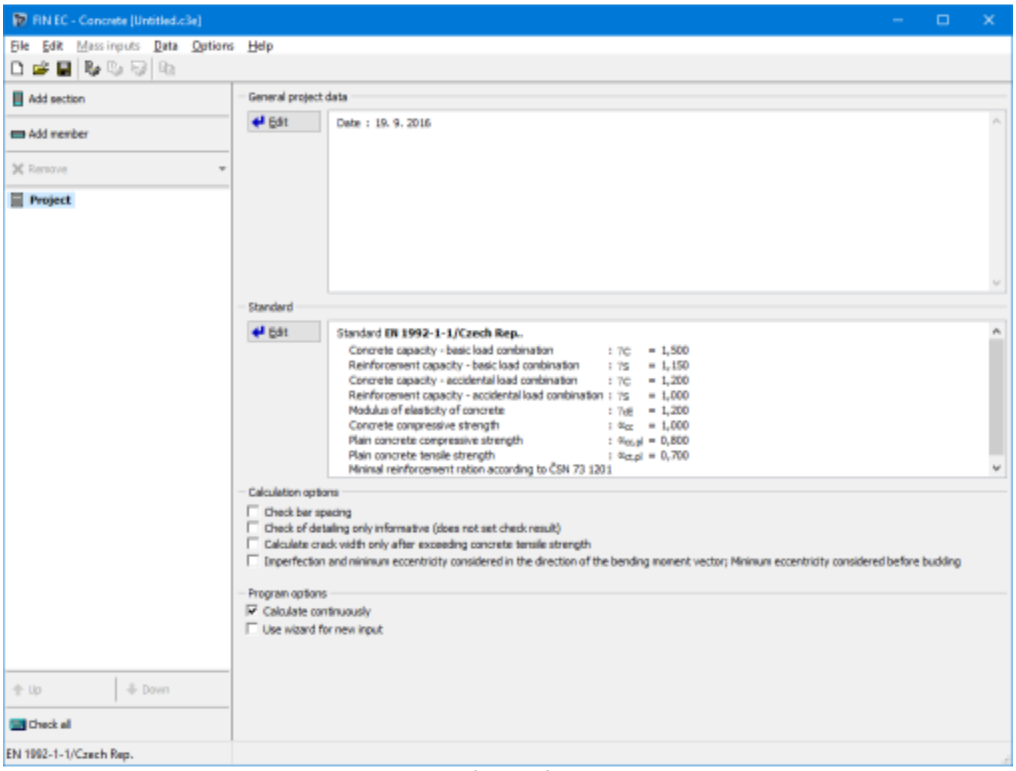
## RC column

### Introduction

This tutorial shows a design of RC column of a hexagonal cross-section. With *200mm* depth of the section and *2000mm* length, the column is subjected to axial compressive force and biaxial bending. The actions in the ultimate limit state are: $N_x$= *400kN*, $M_y$= *2.33kNm* and $M_z$= *5.46kNm*. The stress limitation (serviceability limit state) should be checked for $N_x$= *350kN* and $M_y$= *2.00kNm*. The strength class of the concrete is *C30/37 X0* and steel grade *B500* is used for reinforcement.

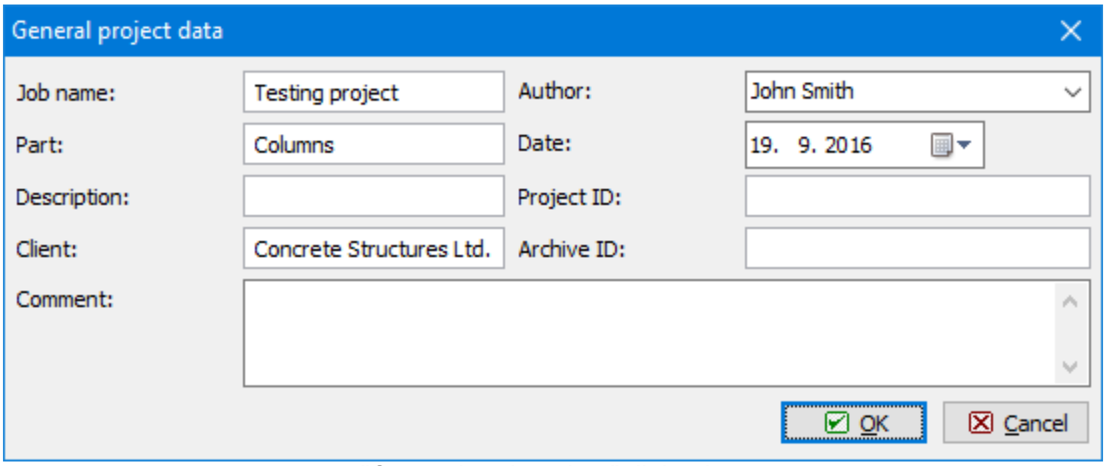
### Starting a new project

The following screen appears after running program "**Concrete**":




*The start screen of the "Concrete" program*

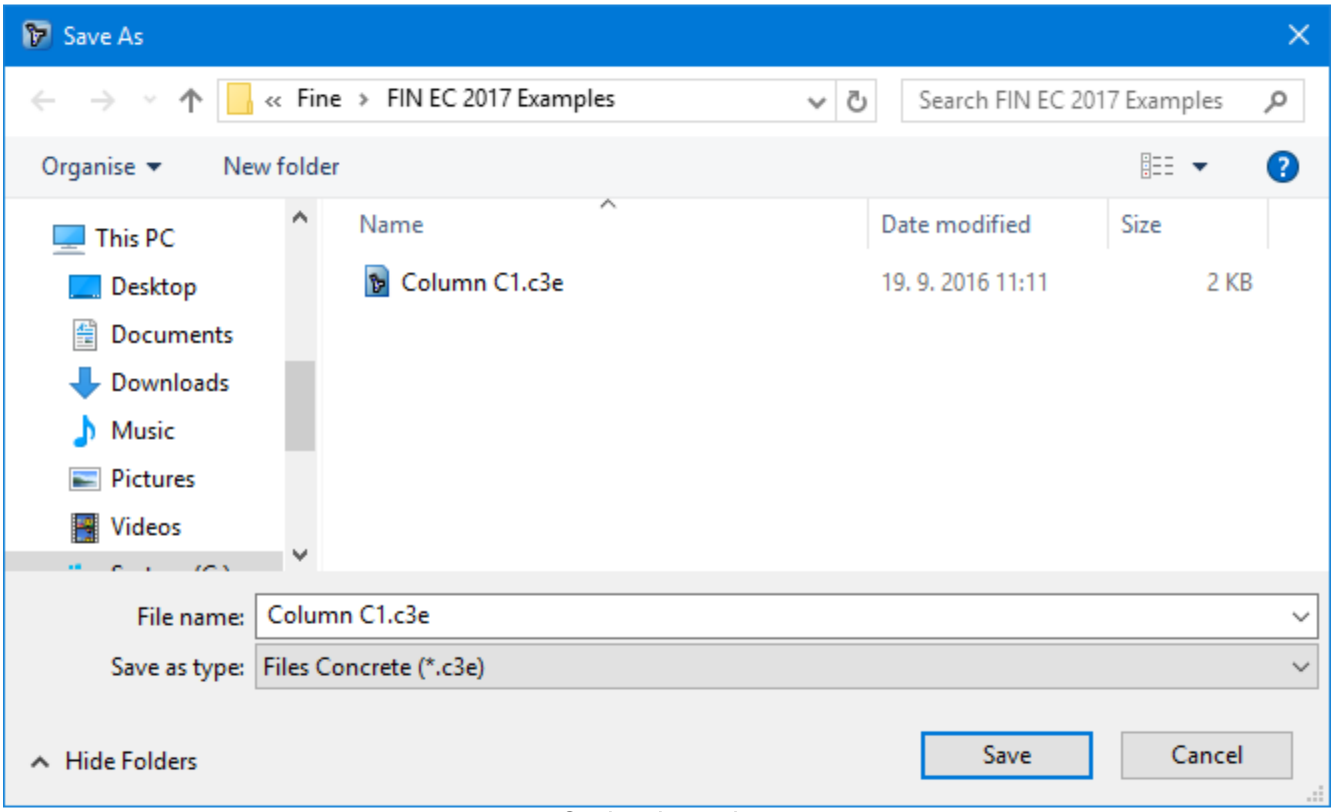
The program provides opportunity to calculate a unlimited number of partial tasks per project. There are two task types supported by the software: "**Section**" and "**Member**". The type "**Section**" is suitable for easy verification of RC cross-sections, the type "**Member**" is usually used for a verification of the structures, which were created in the programs "**Fin 2D**" and "**Fin 3D**". We will use the type "**Section**" for our analysis. The start screen contains a part "**General project data**", where the job name, description and other project identification data can be entered. After clicking the "**Edit**" button, we first enter the job name and other project details:




*"General project data" dialog box*

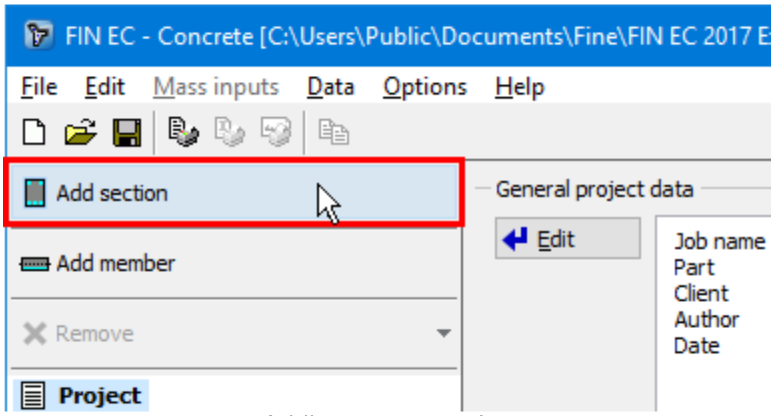
These data can be displayed in the header or footer of the final documentation.

Prior to commencing any work, it is advisable to save the job. This can be done either using "" button, or in the main menu clicking on "**File**" – "**Save As**", or using the "**Ctrl+S**" shortcut.



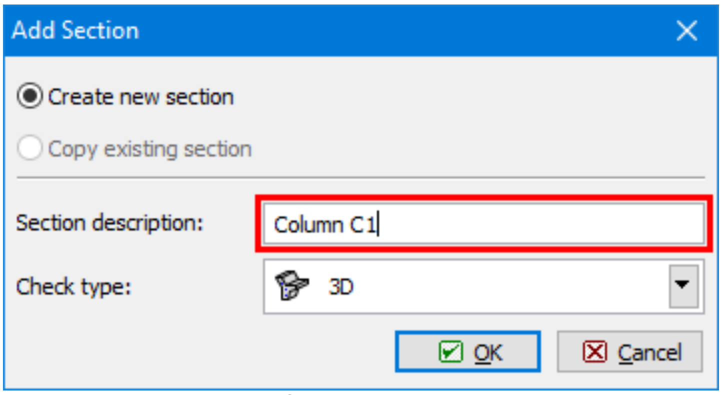

*Saving the project*

Now we can proceed to entering a new task by clicking the "**Add Section**" button in the upper part of the program's tree menu.



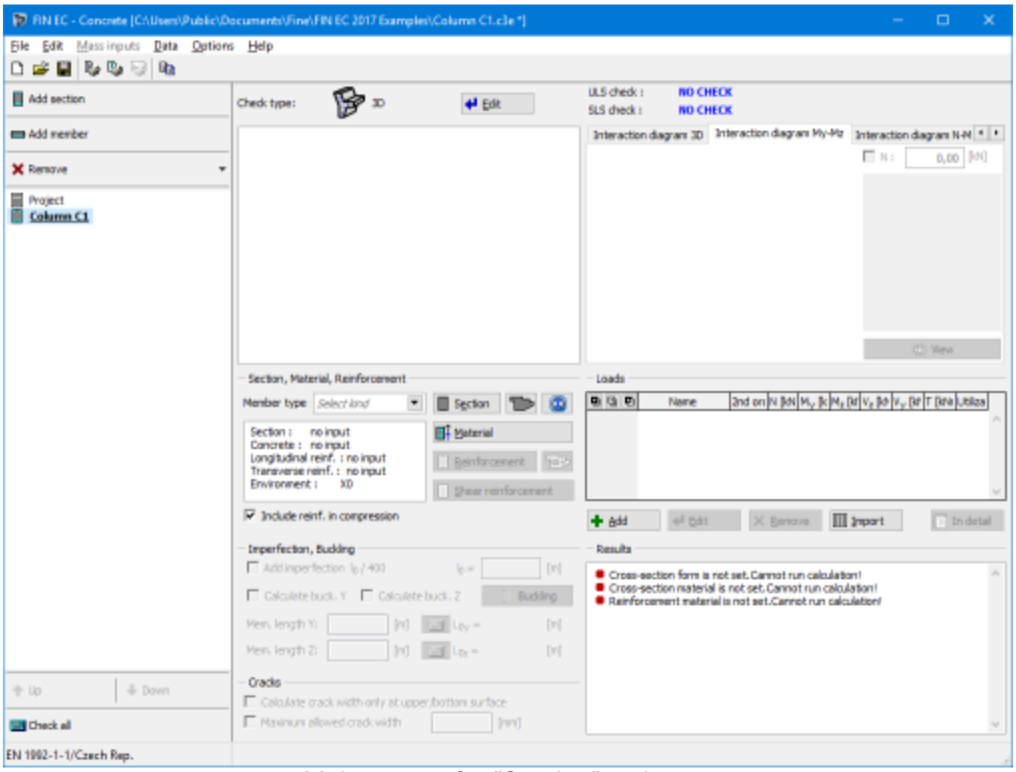
*Adding a new section*

The following dialog box appears, in which we can enter the section's name ("**Column**") into the "**Section description**" field, confirming by clicking the "**OK**" button.



*Dialog box for adding a new section*

A new item has been generated in the tree menu, representing the new added section ("**Column C1**"). The program has now automatically selected this item; therefore we can directly proceed to entering the section parameters.

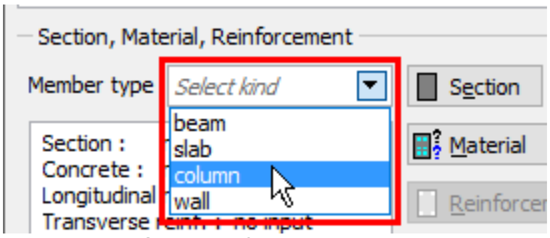

*Main screen for "Section" task type*

## Section, Material, Reinforcement

At first it is necessary to enter the basic geometrical and material characteristics of the section in the "**Section**, **Material**,

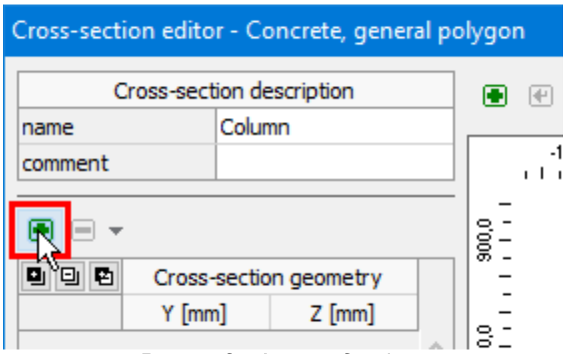
**Reinforcement**" frame. We select the function of the member in the structure from the "**Member type**" drop-down list. Available types are "**beam**", "**slab**", "**column**" or "**wall**".




*Choice of member type*

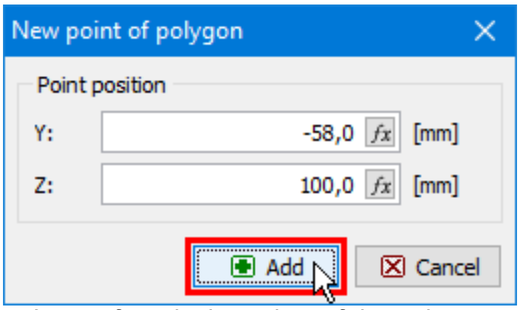
In our example we select member type "**column**". This selection affects the analysis and verifications of the reinforcement arrangement.

As a hexagon is not included in the library of pre-defined cross-sections, we need to use the "" button to define geometry of a general polygon. The polygon shape can be defined graphically or numerically in the window "**Cross-section editor**". The shape of cross-section is defined by six points, which have to be defined in the correct order. We can define each of the points numerically by clicking the "**+**" button, located in the bar in the left upper corner above the table of points.




*Button for input of points*

A new window with input lines for enetring coordinates appears. We specify the coordinates of the forst point *[-0,058;0,100]* and insert the point by the button "**Add**".

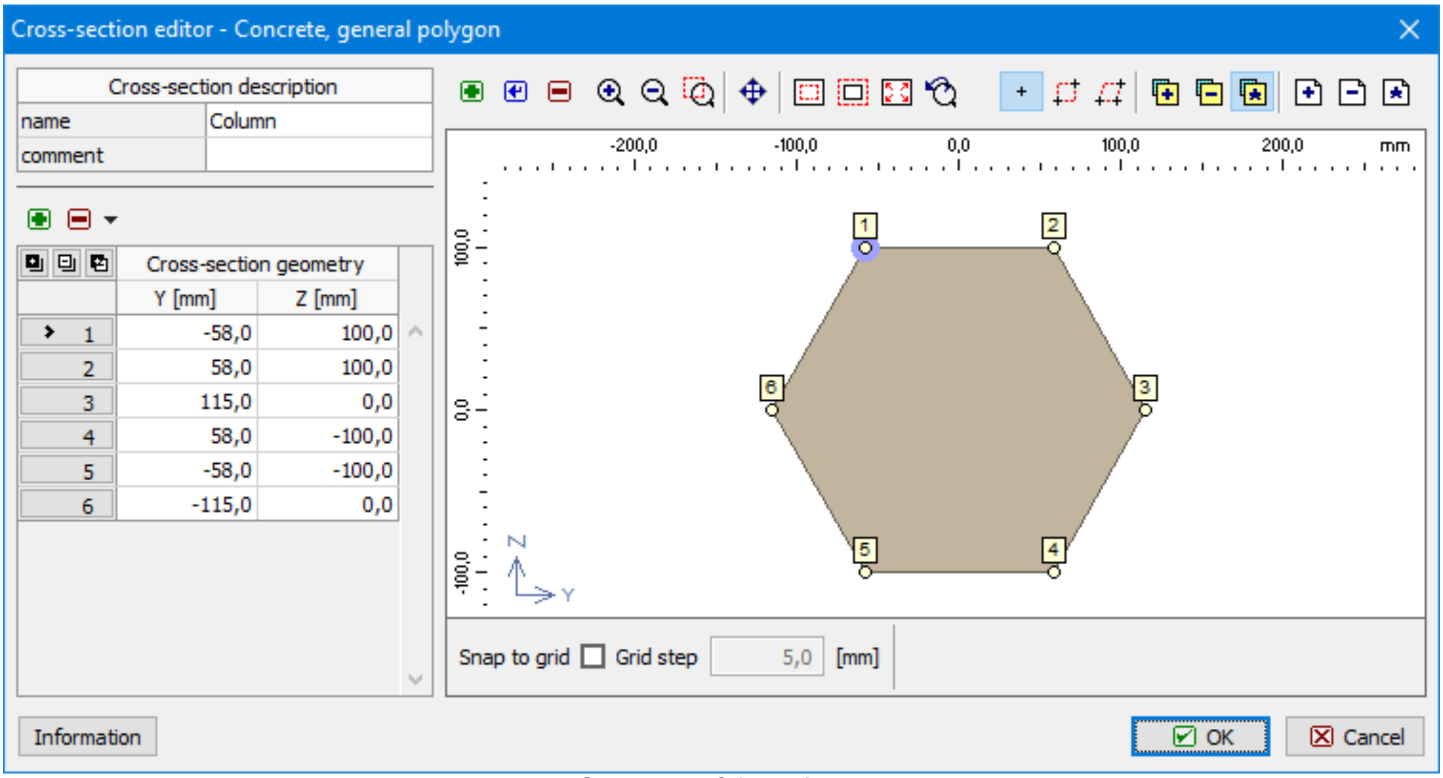



*Input of particular points of the polygon*

We will specify positions of following points *[-0,058;0,100]*, *[0,058;0,100]*, *[0,115;0,000]*, *[0,058;-0,100]*, *[-0,058;-0,100]*, *[-0,115;0,000]* in the same way. After entering coordinates of the last point, we return to the "**Cross-section editor**" dialog box by clicking the "**Cancel**" button.

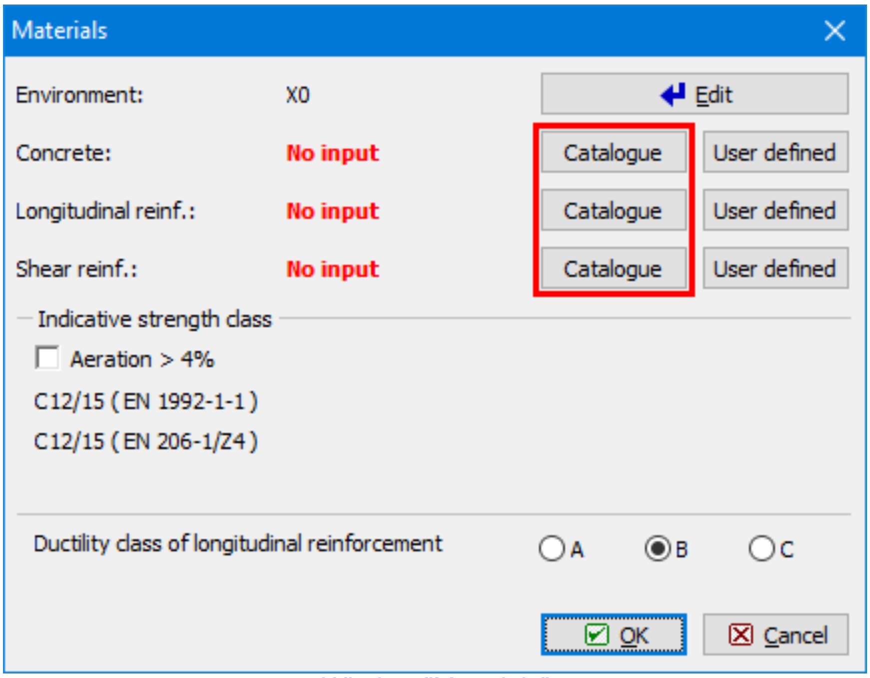
The geometry of the cross-section appears on the right side of the dialog box; we can change the points coordinates either directly in the table on the left side or graphically on the right side. We close the dialog box by clicking the "**OK**" button.




*Geometry of the polygon*

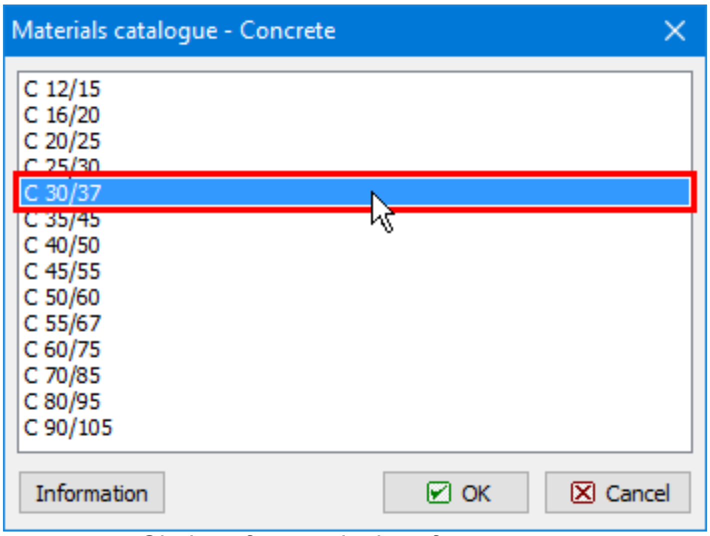
We proceed with defining the material properties in "**Materials**" dialog box which is run by clicking the "**Material**" button in the "**Section**, **Materials**, **Reinforcement**" fram. Assuming the column is located inside of the structure we select "**X0**" for "**Exposure class**", as the column is not in contact with outside environment. Subsequently we define the material properties of concrete and longitudinal and transversal reinforcement. We can select standardized materials from the library of pre-defined materials clicking the "**Catalogue**" button at relevant lines.




*Window "Materials"*

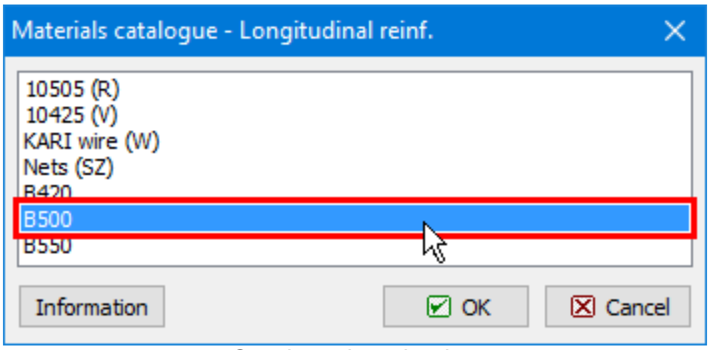
For concrete we select the strength class "**C30/37**" and close the dialog box by clicking "**OK**".



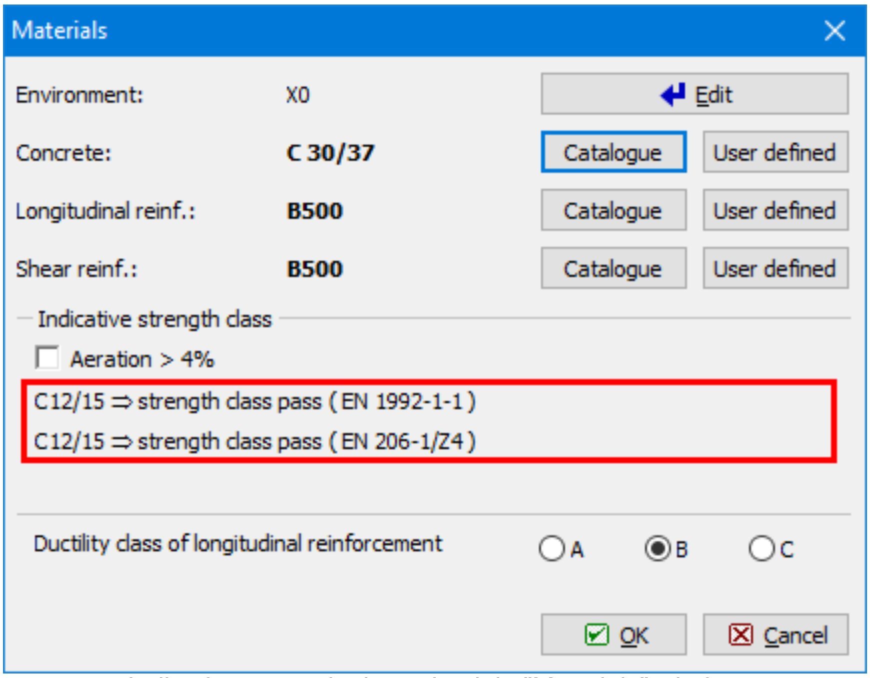

*Choice of strength class for concrete*

Proceeding to definition of the steel properties, we select the grade "**B500**" for both longitudinal and transversal reinforcement and close the dialog box by clicking "**OK**".




*Steel grade selection*

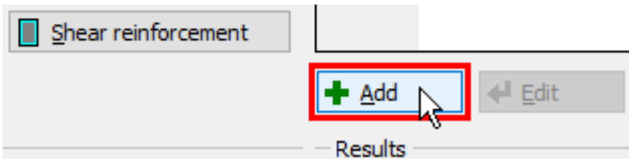
After returning to the "**Materials**" dialog box we can check the summary of the selected materials and confirm whether the selected class of concrete fulfils requirements on the "**Indicative strength class**" given by the selected exposure class. We exit the window "**Materials**" by clicking "**OK**".




*Indicative strength class check in "Materials" window*

## Loads

After defining the geometry of the section and material properties, we can proceed either with defining the reinforcement or loads. In our example we first define a load case because then we are able to check the results of the reinforcement assessment during its definition. To create a load case, we click the "**Add**" button located under the "**Loads**" table.




*Button for insertion of new loads*

In the "**New load**" window we select a "**Combination type**". This type should be selected according to the type of combination, which was used for determination of forces and moments. This input affects the type of verification. The following options are available:

**Basic design (ULS)**
- Forces and bending moments have been obtained from the basic combination for persistent and transient design situations according to EN 1990, equations ***6.10*** resp. ***6.10a*** and ***6.10b***. These loads are used for basic assessment of cross-section's capacity in the ultimate limit state.

**Accidental design (ULS)**
- Forces and bending moments have been obtained from the combination for accidental design situations according to EN 1990, equation ***6.11***. These loads are used for assessment of cross-section's capacity in accidental design situations in the ultimate limit state (partial safety and material factors for accidental design situations are used).

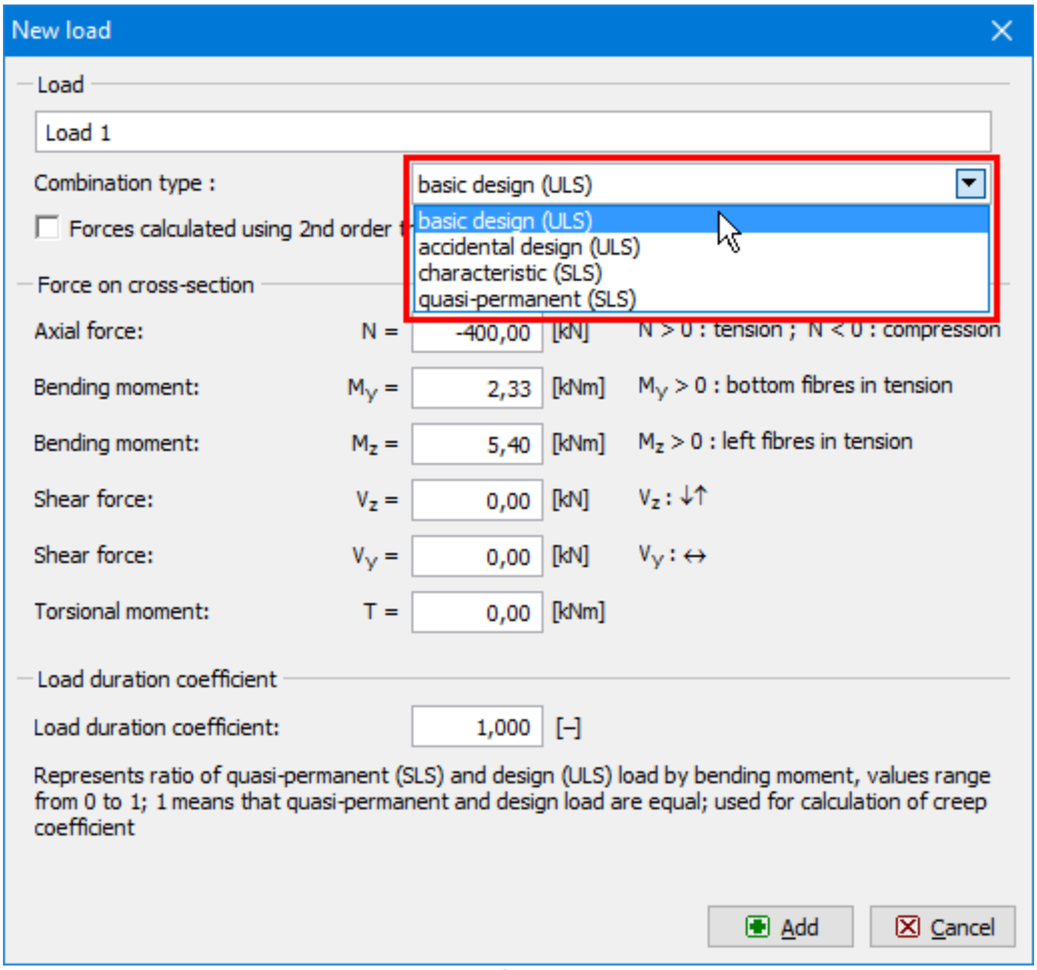
**Characteristic (SLS)**
- Forces and bending moments have been obtained from the characteristic combination according to EN 1990, equation ***6.14***. These loads are used for assessment of the stress limitation (serviceability limit state).

**Quasi-permanent (SLS)**
- Forces and bending moments have been obtained from the quasi-permanent combination according to EN 1990, equation ***6.16***. These loads are used for assessment of the crack widths in the serviceability limit state.

Subsequently we enter the forces and bending moments acting on the cross-section; in our example the axial force is $N = -400 kN$ (negative value denotes compression) and the bending moments are $M_y = 2{,}33 kNm$ and $M_z = 5{,}46 kNm$.

Also, we should enter the "**Load duration coefficient**", i.e. the ratio of quasi-permanent and total loads for calculation of the creep coefficient. If the exact value is not available, we can leave the conservative value of *1.00*, which means that the total load is considered quasi-permanent in the calculations. The new load is confirmed by clicking the "**Add**" button and "**Cancel**" to exit the dialog box.

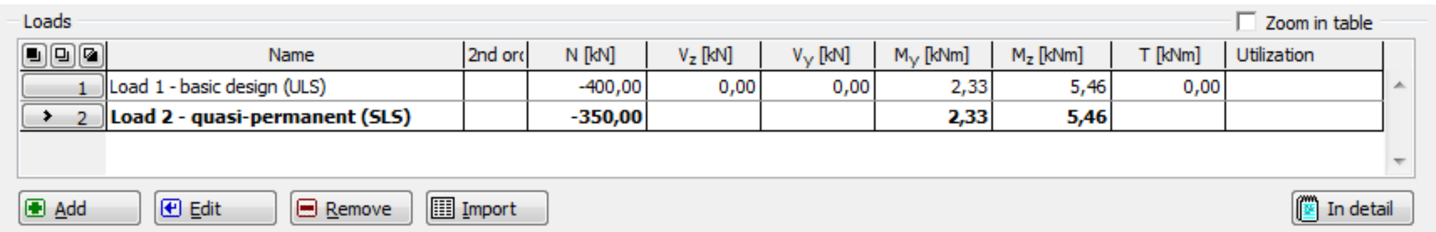
 




*Input of new load*

To define the loads for the serviceability limit state, analogical procedure is used as for the ultimate limit state loads. To assess the stress limitation, we select "**Combination type – characteristic (SLS)**" and enter the relevant axial compressive force *N= -350kN* and the bending moment $M_y$*= 2.00kNm*. After input confirmation by the button "**Add**", we will exit the window by using the button "**Cancel**".

As result, a table summarizing all defined load cases is generated in the dialog box.




*Loads*

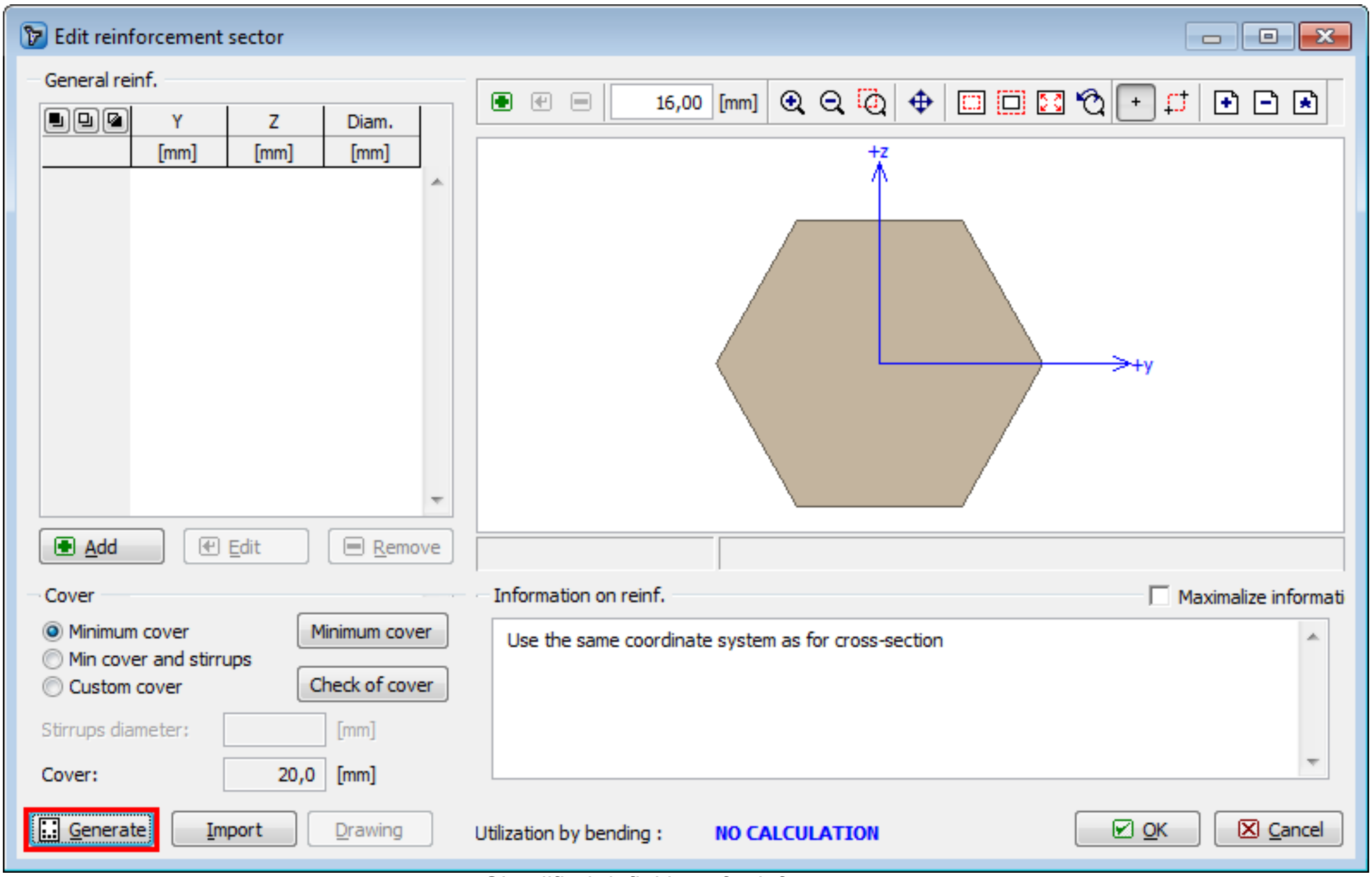
The number of loads is not limited in the software. The input can be done also in a batch using text or **.csv* file (button "**Import**").

## Reinforcement

After returning to the main dialog box, we can proceed to defining the longitudinal and transverse reinforcement. We open the dialog box "**Edit reinforcement sector**" for longitudinal reinforcement definition by clicking on the "**Reinforcement**" button in the "**Section**, **Material**, **Reinforcement**" frame. The upper part of the window contains an option to select a calculation method of the cover. We keep the method ""

In this dialog box, we can define the reinforcement either numerically in the table on the left-hand side or graphically in the right part of the dialog box.We can also take advantage of a simplified approach by clicking "" and then the "**Generate**" button in the left lower corner of the dialog box. In our example we will use this option.

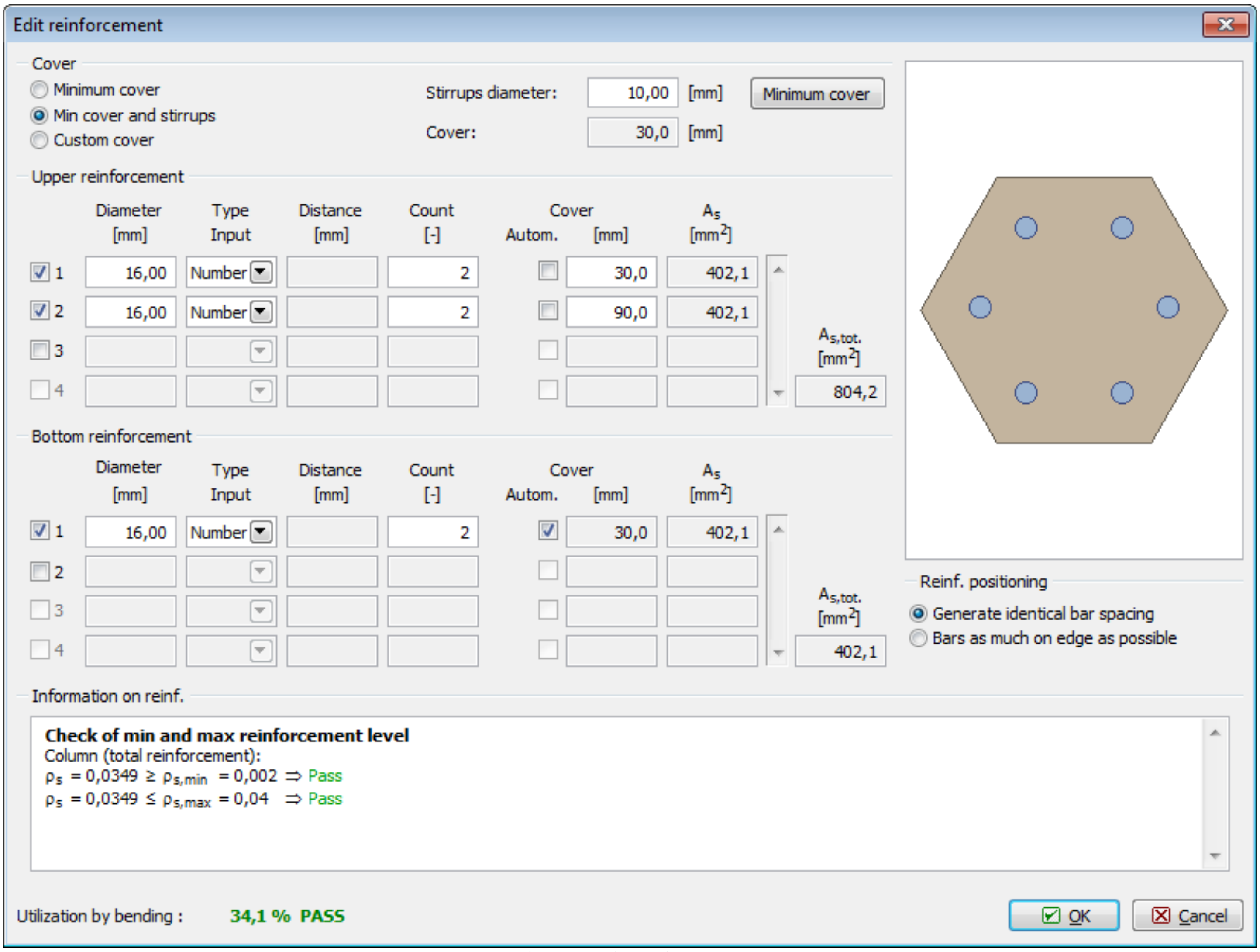
 




*Simplified definition of reinforcement*

We can now easily define required longitudinal reinforcement using *16mm* bars in each corner of the cross-section, entering 3 layers with appropriate covers:

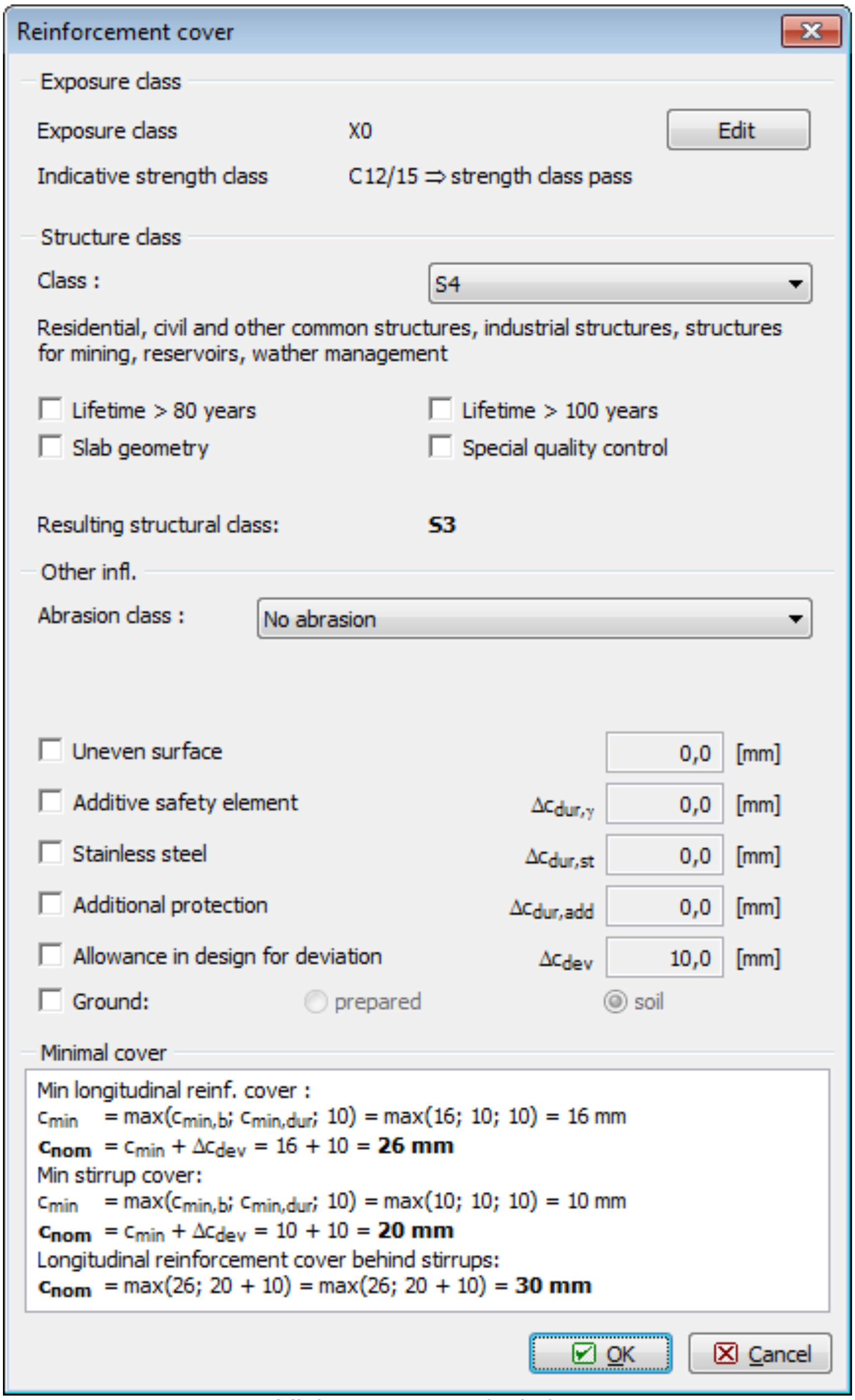
 




*Definition of reinforcement*

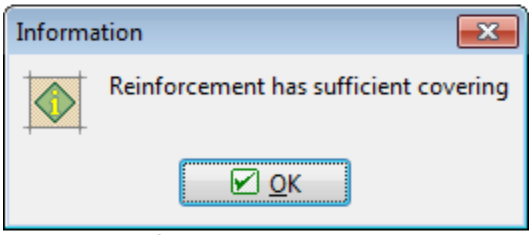
After defining reinforcement we can immediately check in the lower part of the dialog box that the area of the reinforcement is sufficient and passes design criteria with *34.1%* utilization by bending. In the section "**Information on reinforcement**" we can also confirm that the detailing requirements given by the code are satisfied. Finally we need to check if the covers are correctly defined. Having a column with stirrups, the option "**Min cover and stirrups**" is selected in the "**Cover**" section of the dialog box. The program will calculate the minimum required cover of the longitudinal reinforcement as sum of the minimum cover given by the code and the diameter of the stirrups. The calculation can be checked in the dialog box opened by clicking "**Minimum cover**" button:




*Minimum cover calculation*

As it is not necessary to change the settings of cover calculation, we can exit the dialog box by clicking "**OK**" and return to the dialog box for longitudinal reinforcement definition. To check whether the geometry of the longitudinal reinforcement satisfies requirements on minimum covers, we can run the assessment by clicking "**Check of cover**".




*Covers check result*

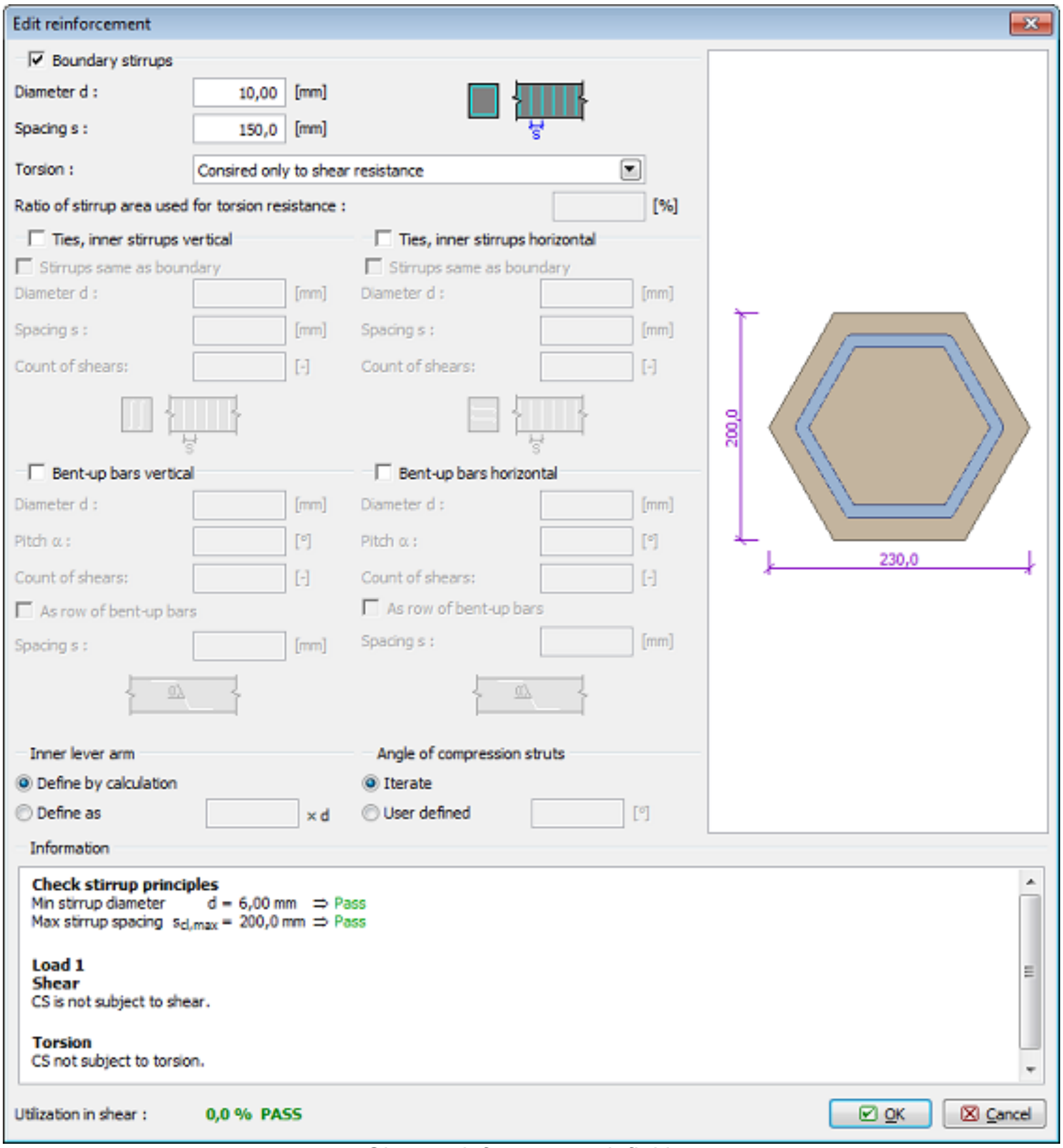
The check returned a positive result, therefore we can return to the main dialog box by clicking "**OK**".

## Shear reinforcement

We can proceed to defining the shear reinforcement by clicking the eponymous button in the main screen.

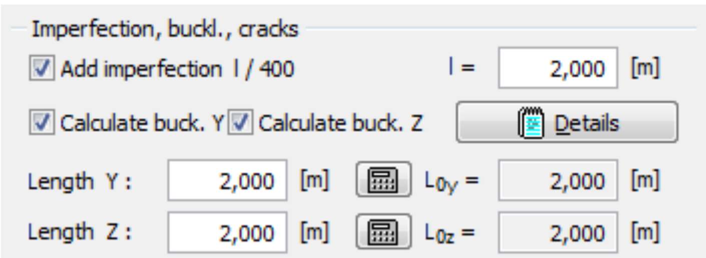
We specify *10mm* diameter boundary stirrups with *150mm* spacing:



*Shear reinforcement definition*

## Buckling

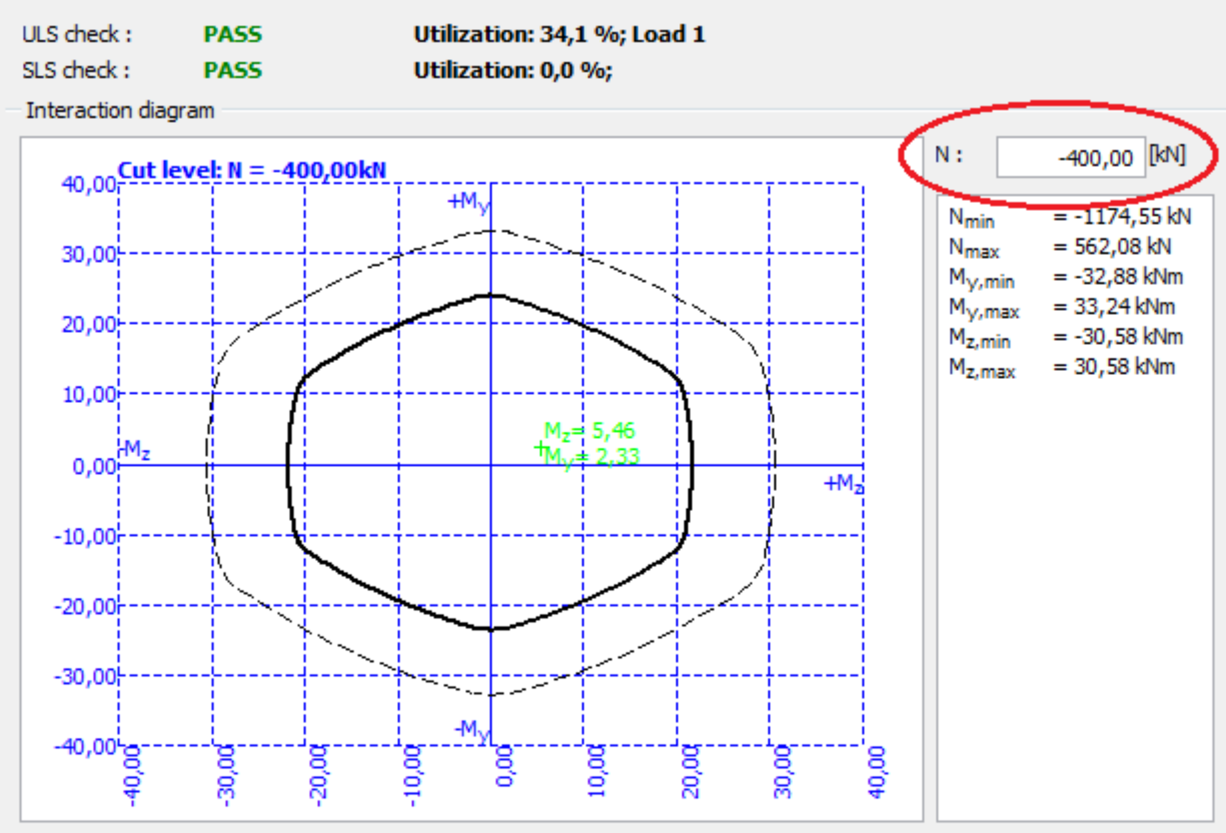
The next step is defining the buckling parameters. Firstly we need to tick "**Calculate buck. Y/Z**" boxes followed by entering the nominal lengths of the column for both directions, based on which the effective buckling lengths will be calculated. For a column simply supported on both ends, the effective buckling lengths equal to the nominal. Defining different boundary conditions can be done by clicking the "" buttons for each direction.



*Defined buckling parameters*

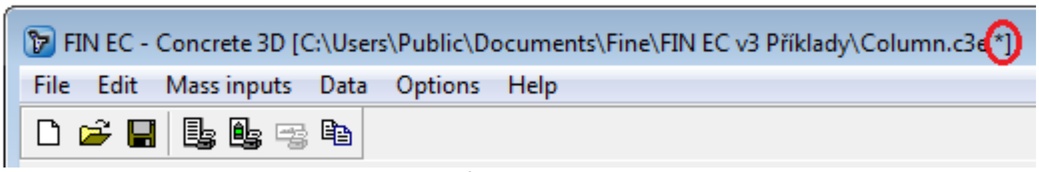
After defining the buckling parameters, two distinct areas of capacity are shown in the interaction diagram: thin dashed line denotes capacity of the member without influence of buckling and thick line denotes the capacity reduced by buckling effect. To check position of a defined load case in the "**Interaction diagram**" we need to enter the relevant axial force level; in our example *N= -400kN*. Program will create a cut through the interaction diagram on this level showing the position of the defined load case:



*Defining a cut through the interaction diagram*

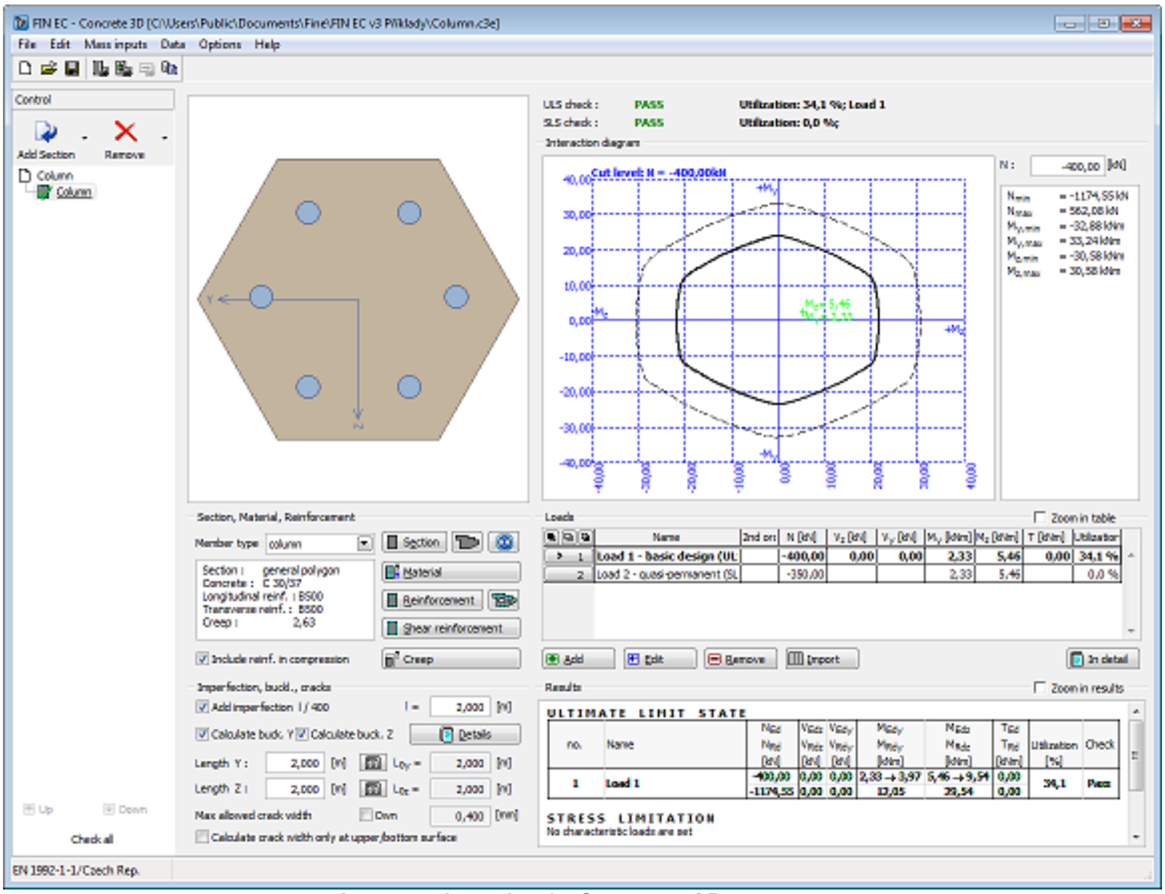
Because we completed definition of all parameters, it is recommended to save the job by clicking "" on the toolbar or using shortcut "**Ctrl+S**". The actual state of the job during work may not be identical to the one saved on disk; this is indicated by "*" in the program window header. In such case it is advisable to save the job.



*Indication of non-saved job state*

Indication of non-saved job stateAs all structural requirements have been checked and satisfied during the definition of parameters and as the main dialog box indicates that the section passes design checks in both ultimate and serviceability limit states, the job can be considered finished.

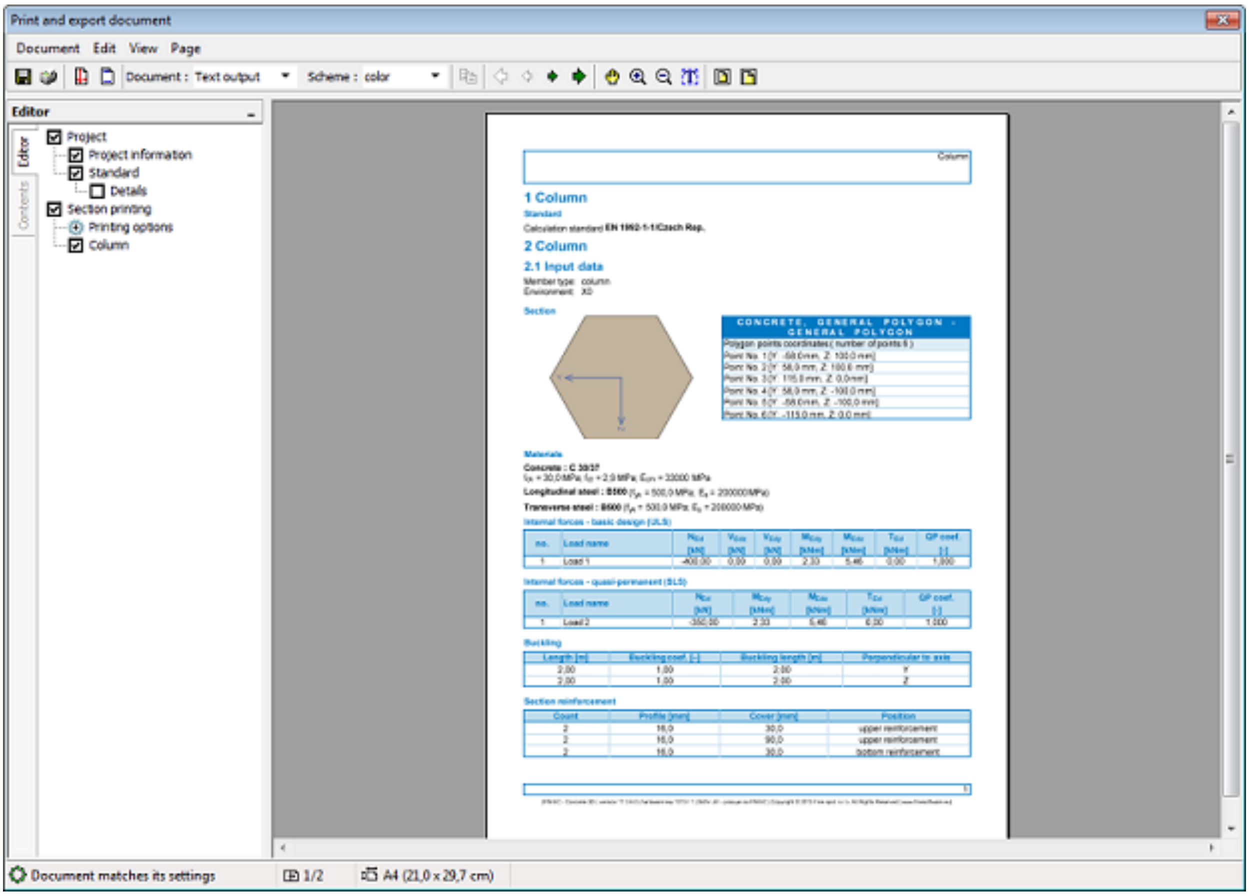
 



*Assessed section in Concrete 3D program*

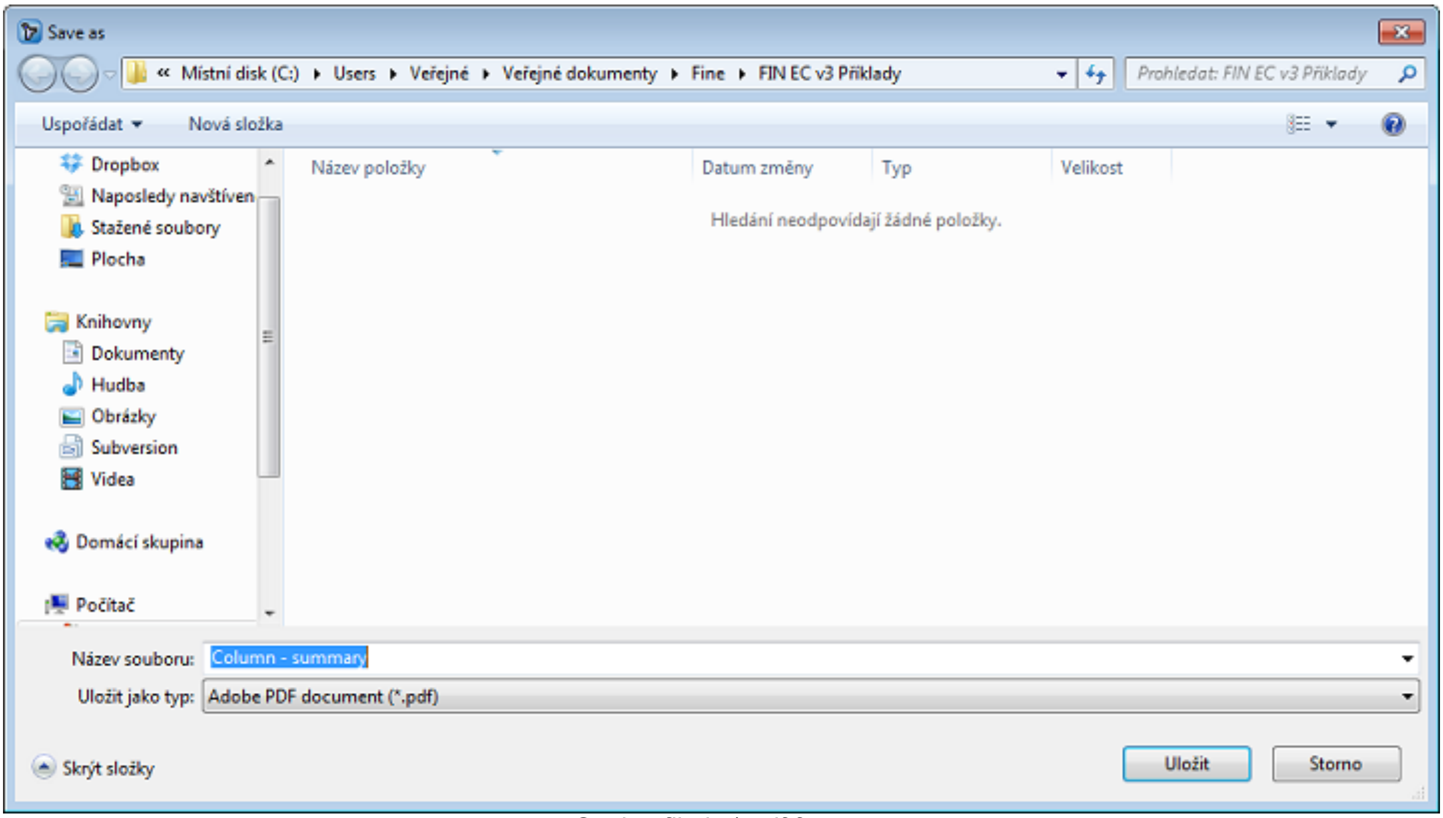
## Outputs

When the job is finished and saved, we can proceed to composing the output documentation. First we print out a concise single page output which summarizes all input data and design checks results. Composition of this output is run by clicking "**File**" and "**Graphic output**" or "" in the toolbar.



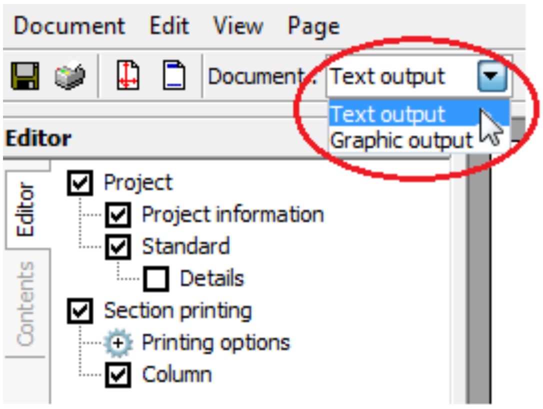
*Graphical output of Concrete 3D program*

We can print the document directly by clicking "" button or save it on disk as *.pdf or *.rtf file by clicking "" button. We use the second option and save the file on disk. In the dialog box "**Save as**" we can enter the file name and select the destination folder.



*Saving file in *.pdf format*

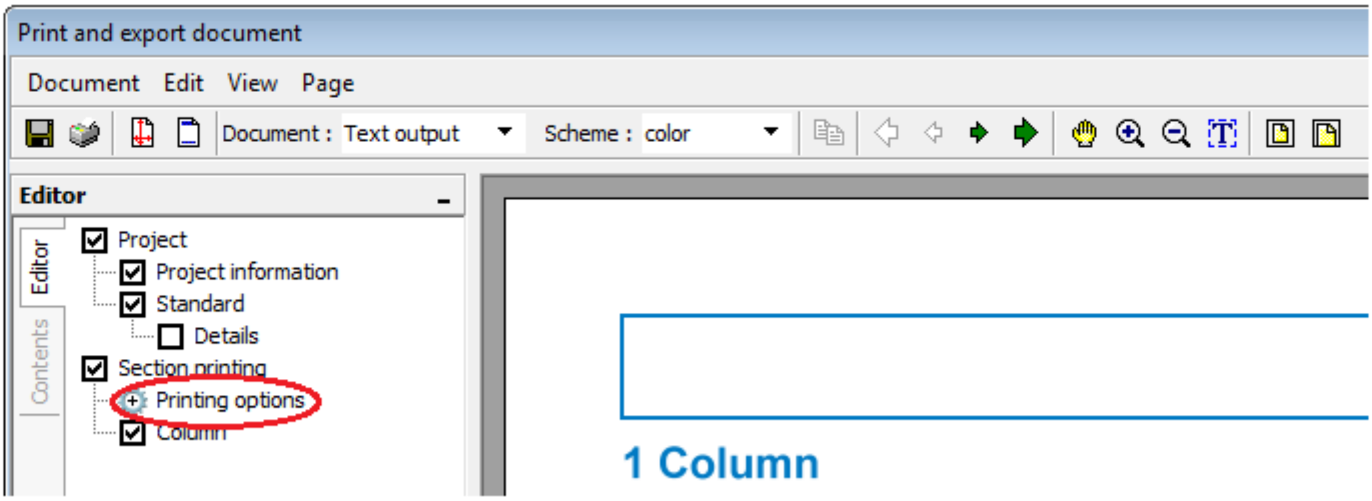
Apart from this concise document, we can also compose a detail text output by clicking the "" button in the toolbar or selecting "**File**" and "**Text/Graphic output**" in the main menu. However, as we are still in the print and export document dialog box we can change the document type to text output directly in the toolbar's "**Document**" drop-down list.



*Change of output type*

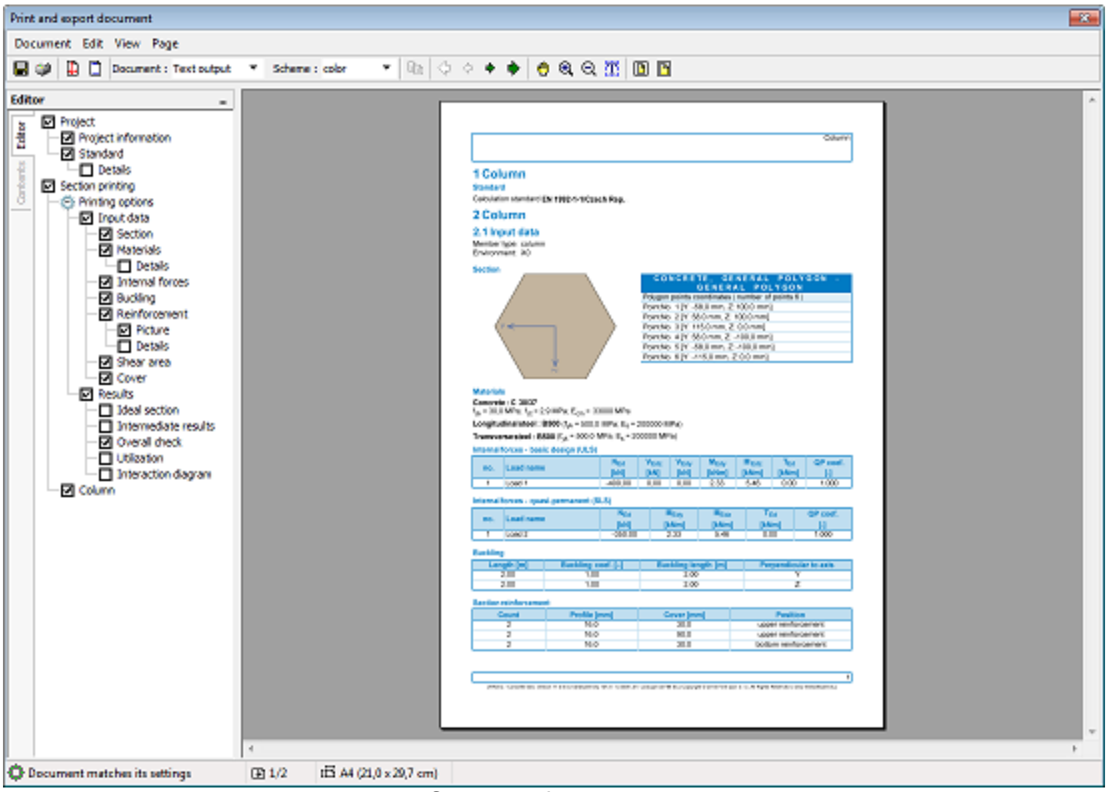
After switching to the "**Text output**" mode we can set in the "**Editor**" which parts of the assessment will be included in the output and how detailed the output shall be.




*Printing options of text outputs*

Program will immediately re-generate the output to reflect each change made in the settings in the tree on the left-hand side. Once the output contains all required information, we can again save the document on disk.



*Generated text output*

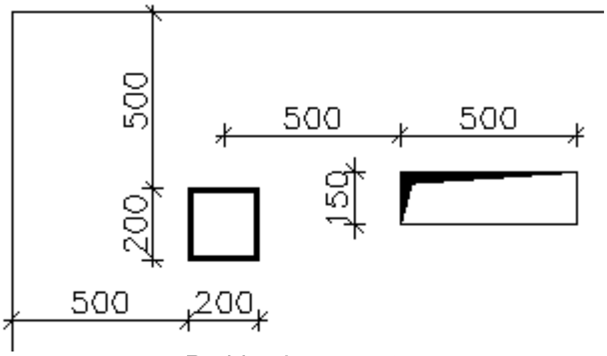
Completing the outputs generation, our work is done.

# Punching

## Task

In this example, the task is to design reinforcement against punching of a corner column of 200 x 200 mm square cross-section through a 200 mm thick reinforced concrete slab. The column is located 500 mm from the edges of the slab, which is weakened by an opening of 150 x 500 mm located as per the following sketch. Concrete strength class C25/30 is used in the design.

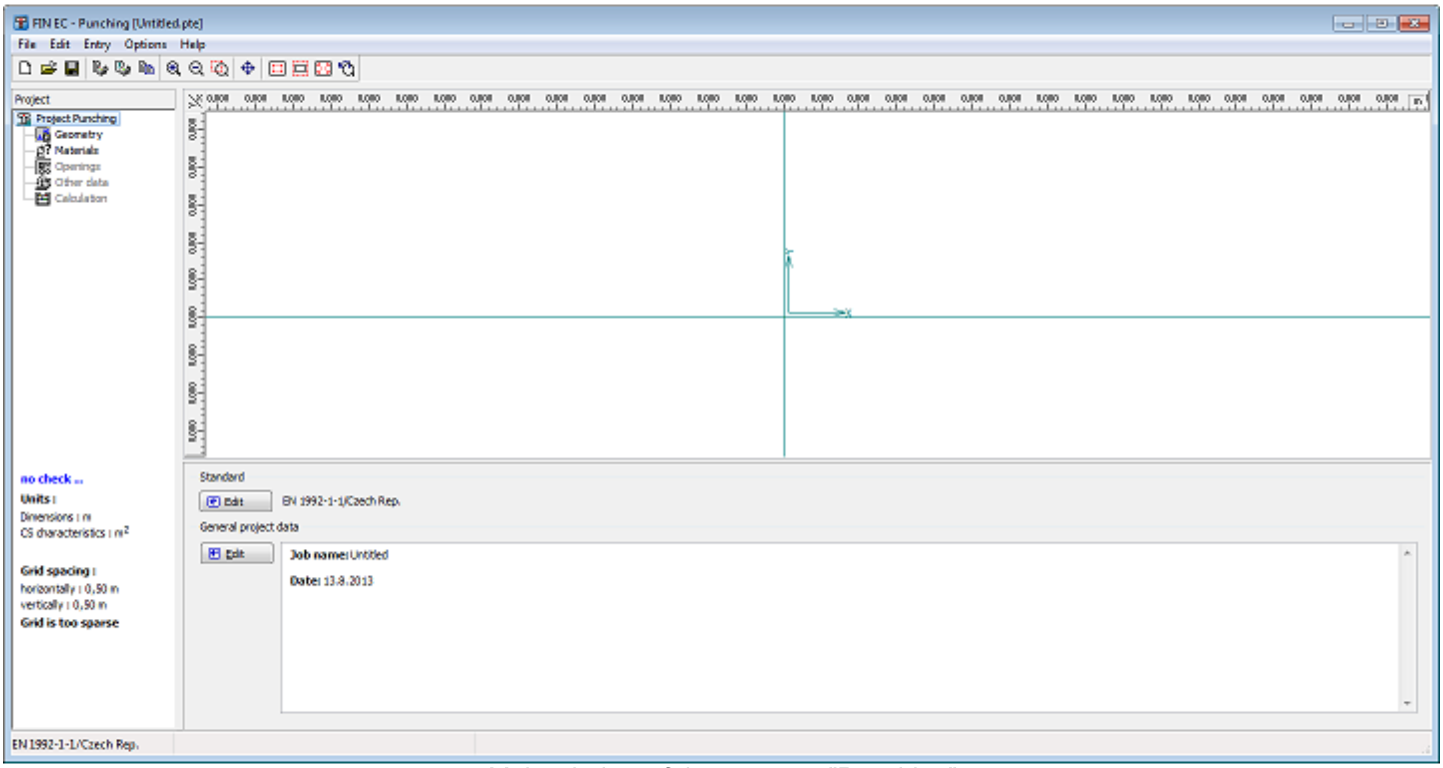
 




*Problem's geometry*

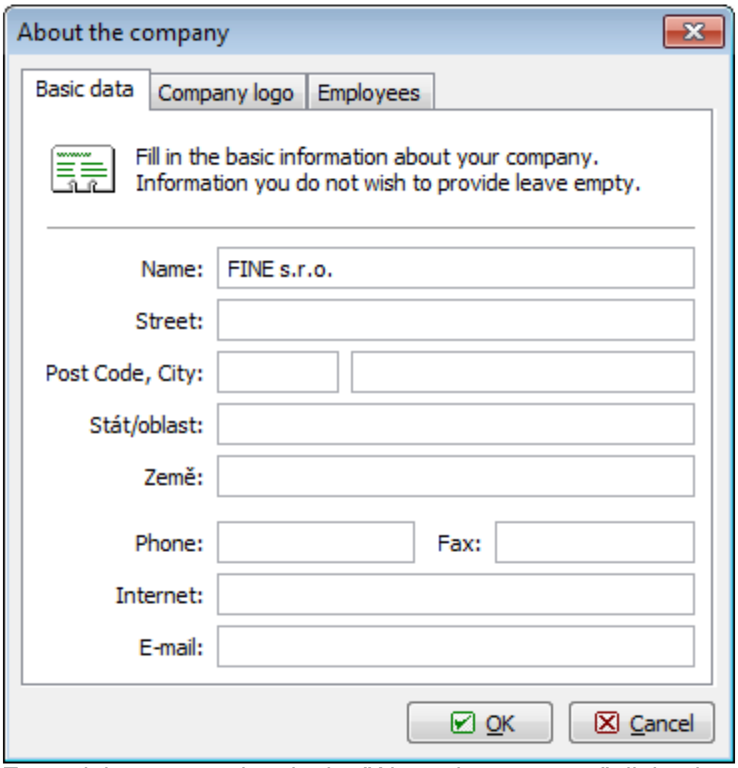
## Project data

In the main program window, all essential project data can be entered in the frame in the bottom part. Definition modes can be switched between using the tree on the left hand side; the rest of the window serves as a model space.



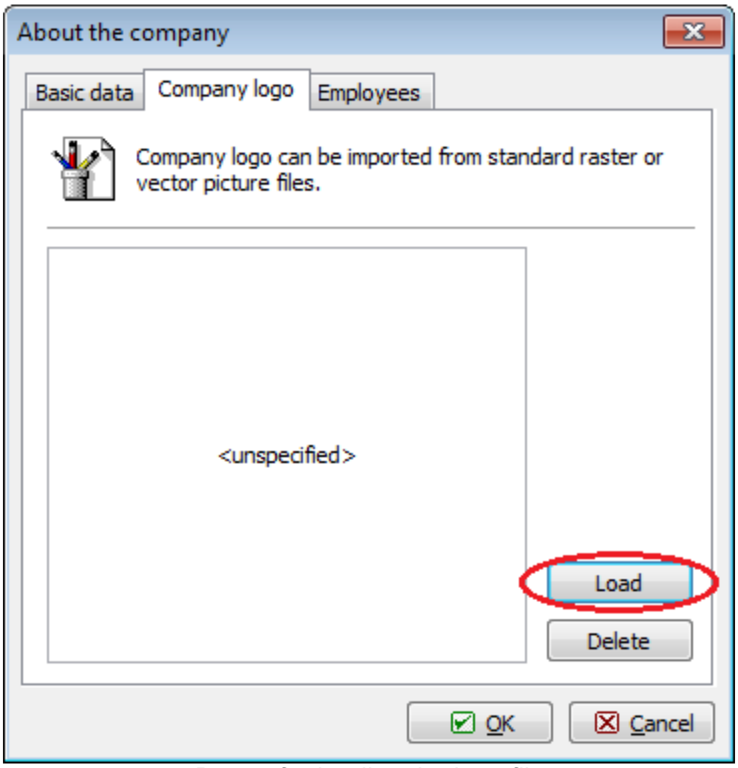

*Main window of the program "Punching"*

First we enter the company data which will be later used when composing the output documentation. The dialog box **"About the company"** can be run from the main menu selecting "**Settings**". In the first tab for instance, we can enter company's name and address.




*Essential company data in the "About the company" dialog box*

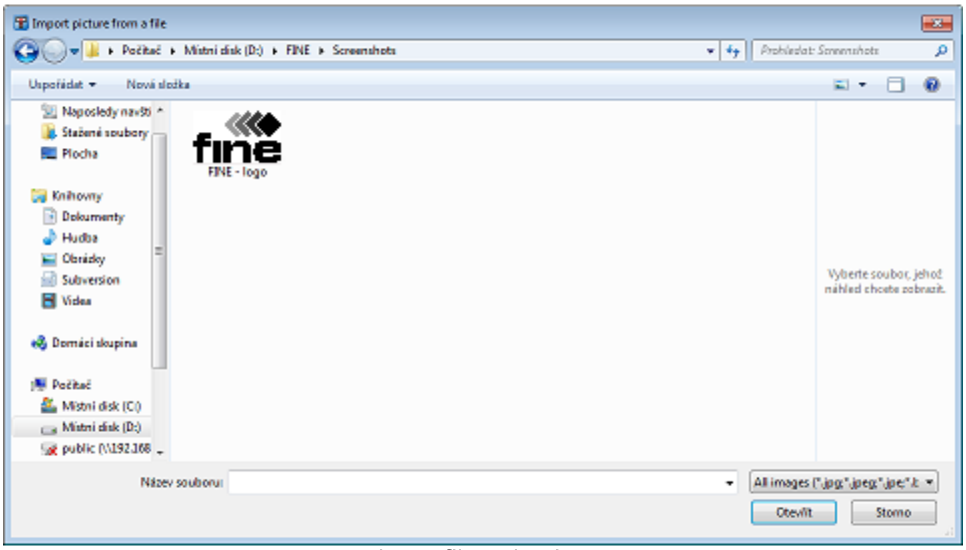
In the second tab we can enter the company's logo which can later be displayed in the header of the output documentation. The logo can be loaded as an image of any usual format such as *.bmp, *.jpg, *.ico etc. To import the logo we run the dialog box by clicking "**Load**".




*Button for loading the logo file*

For selecting files, the standard Windows user interface is used. After opening the folder containing the file, we load the file by clicking the "**Open**" button.



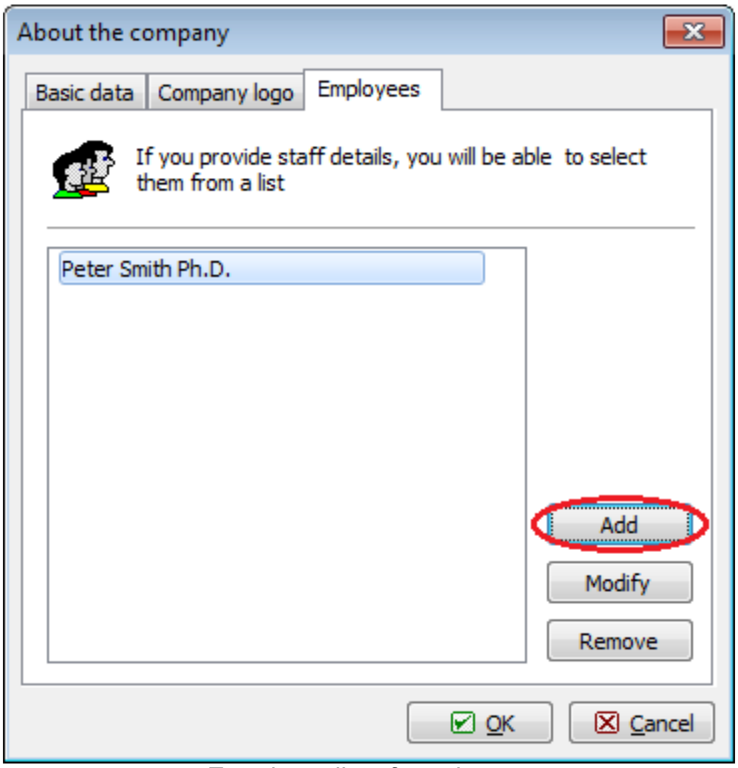

*Logo file selection*

A preview of the logo is subsequently displayed; its size will be automatically adjusted to the graphic layout of the program's output.


*A logo loaded to the program*

In the last tab "**Employees**", we can enter a list of company's employees from which we can select the author of the project. New employees are entered by clicking the "**Add**" button.

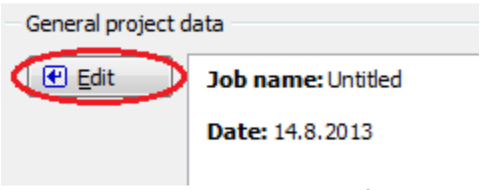



*Entering a list of employees*

We exit the dialog box "**About the company**" by clicking "**OK**". The company's data are shared with all EC programs and can be therefore used in other programs of the package.

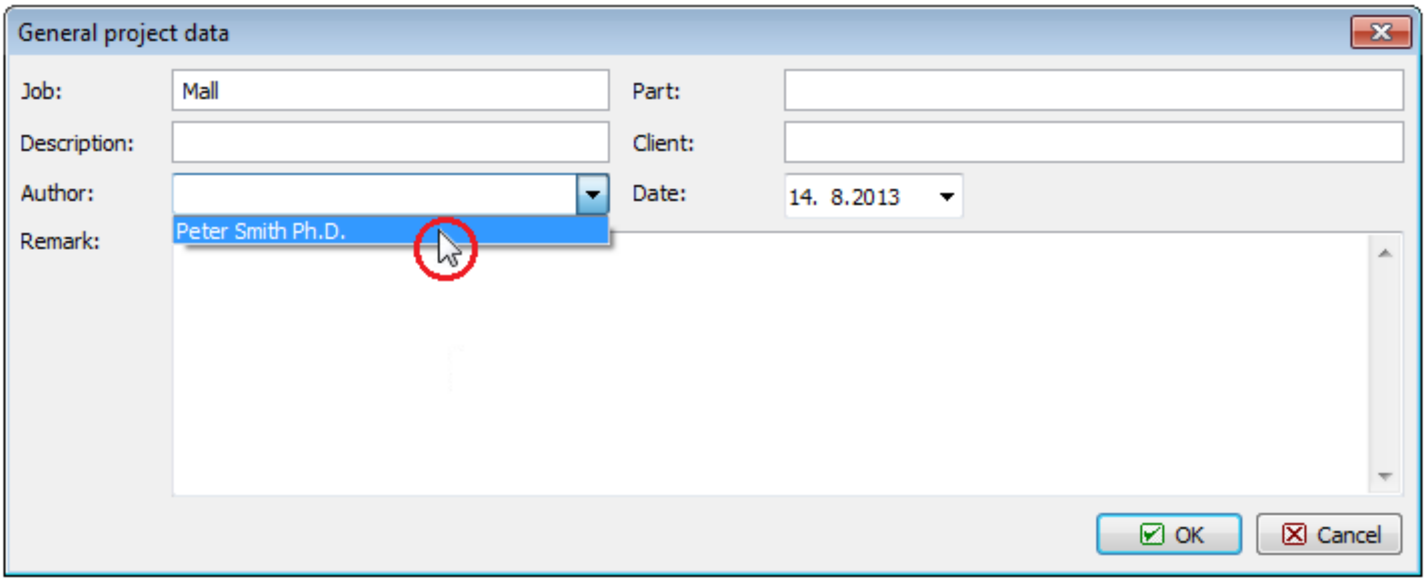
**Basic settings**

After entering data in the "**About the company**" dialog box we return to the main screen. Here we can enter essential information about the job in the "**General project data**" dialog box which opens by clicking the "**Edit**" button.




*Opening "General project data" dialog box from the main window*

In the "**General project data**" dialog box we can enter e.g. job's name, description, remarks etc. It is also possible to select the author of the project from the list of the employees which we created in the "**About the company**" dialog box. All this information can be later displayed in the headers or footers of the output documents.




*Selecting job's author from list in "General project data" dialog box*

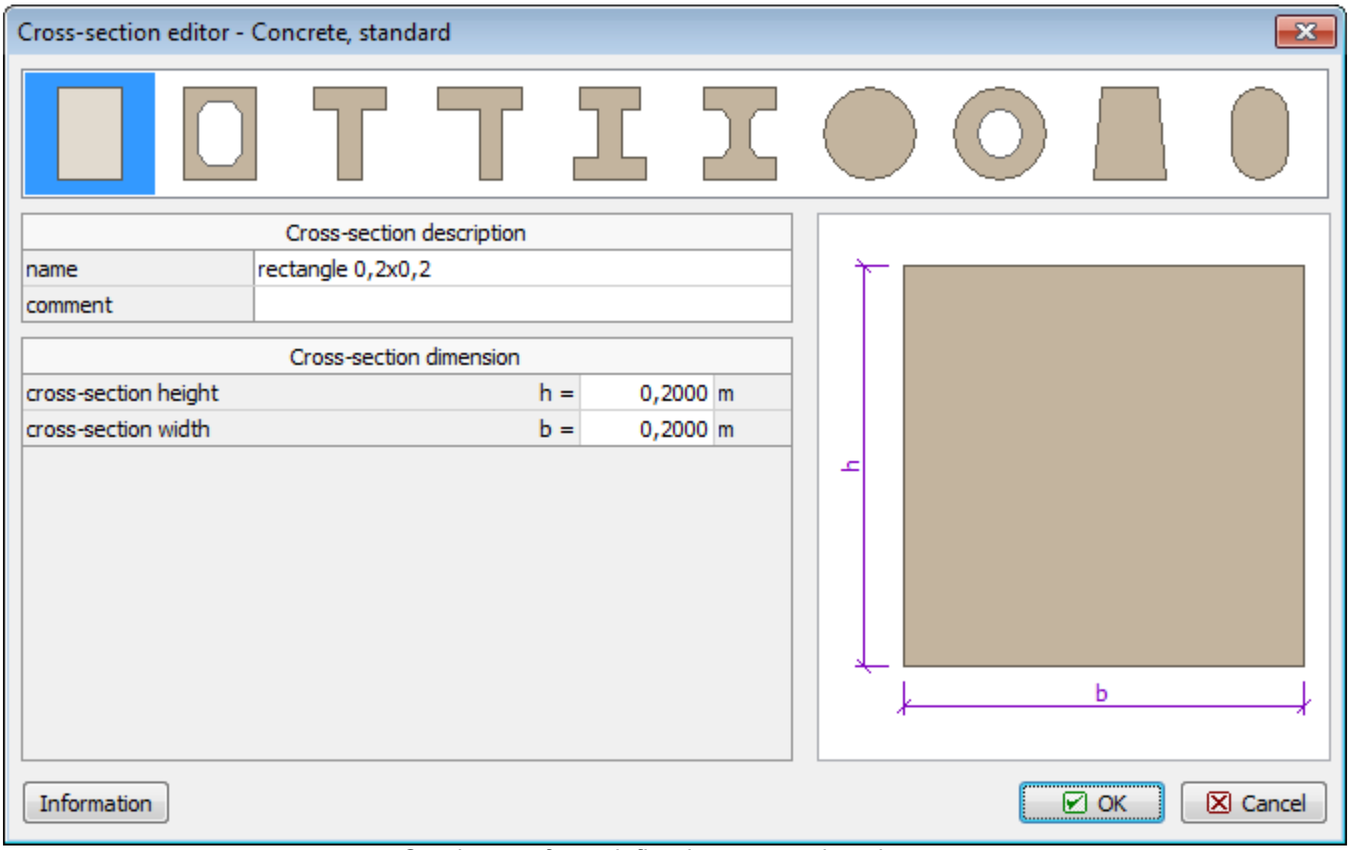
 

Once we have finished entering the project data, we exit the dialog box by clicking "**OK**". Then we can proceed to the next step by clicking the "**Geometry**" button in the tree on the left-hand side.

## Geometry

In this dialog box, we first select the column’s cross-section. Clicking the "

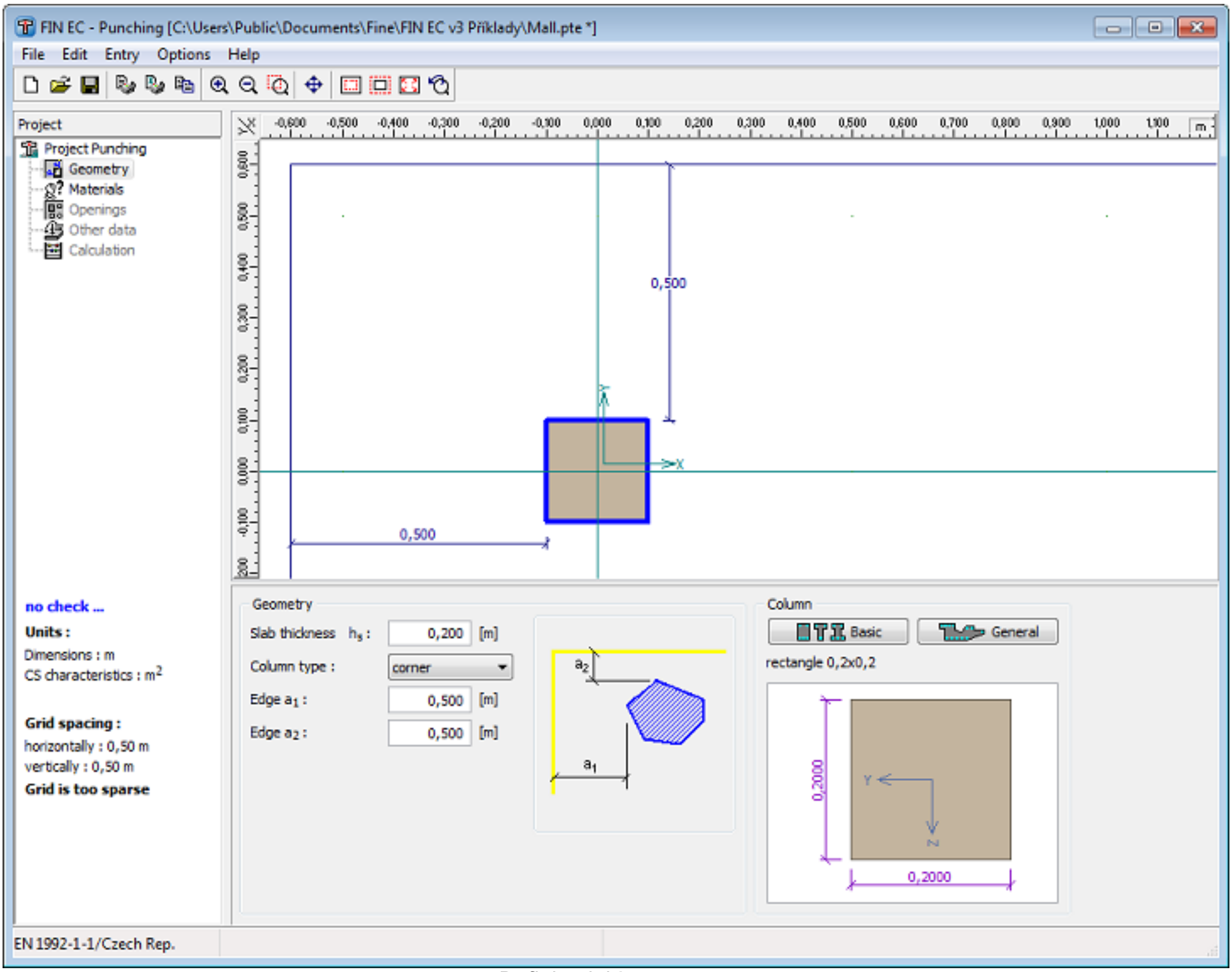
" button we open the catalogue of pre-defined shapes. Then we select a rectangle and enter the dimensions.




*Catalogue of pre-defined cross-section shapes*

In the "**Geometry**" section of the main screen we enter the slab thickness, select the "**corner**" column type and define the edge distances. The structural layout instantly appears in the model space.

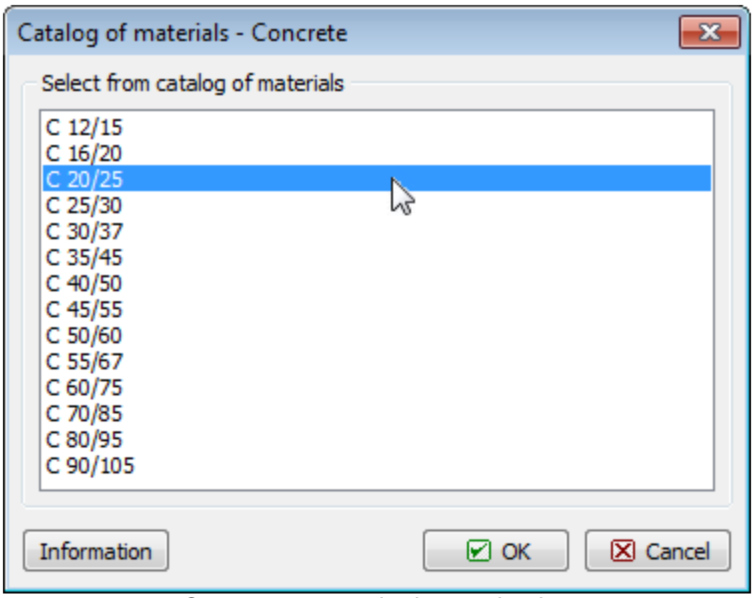
 



*Defining job's geometry*

## Materials

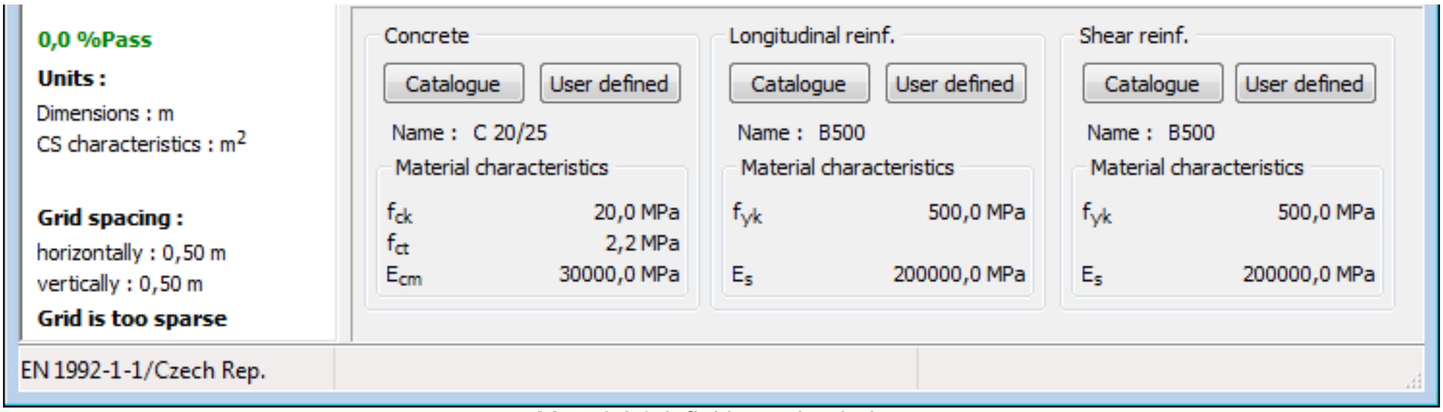
Proceeding to the next item of the tree - "**Materials**", another sub-window appears in the main window's bottom part. In this section we need to specify materials for concrete and slab's longitudinal and punching shear reinforcement. As we are using a standard concrete class, we click on the "**Catalogue**" button and open the list of available concrete strength classes. We select C20/25 and exit the dialog box by clicking "**OK**".




*Concrete strength class selection*

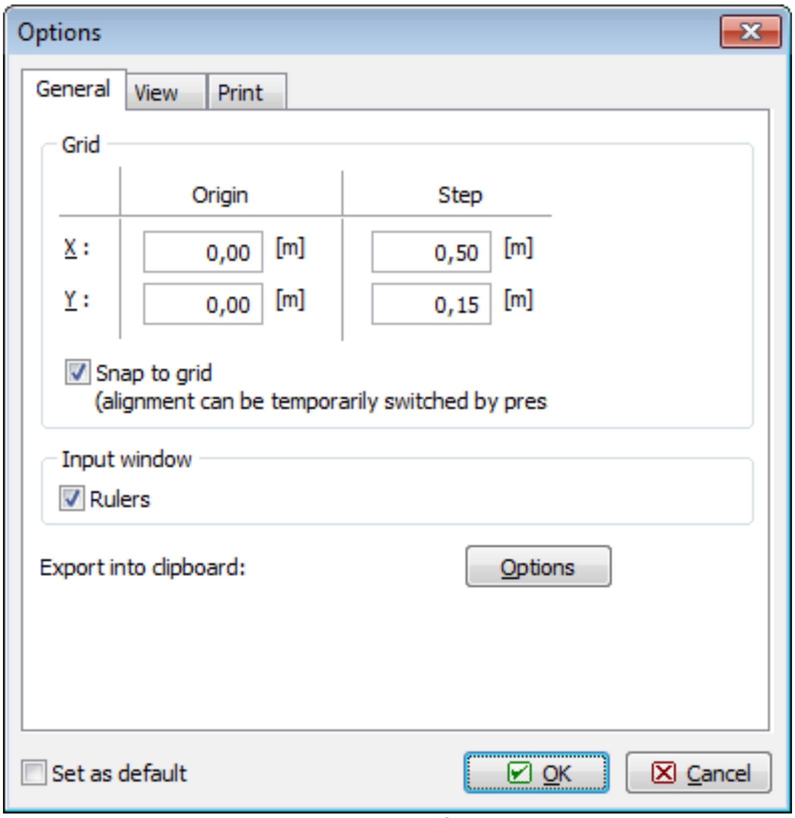
We proceed analogically to specifying reinforcement material. For both longitudinal and shear reinforcement we will select the "**B500**" grade.




*Materials' definition sub-window*

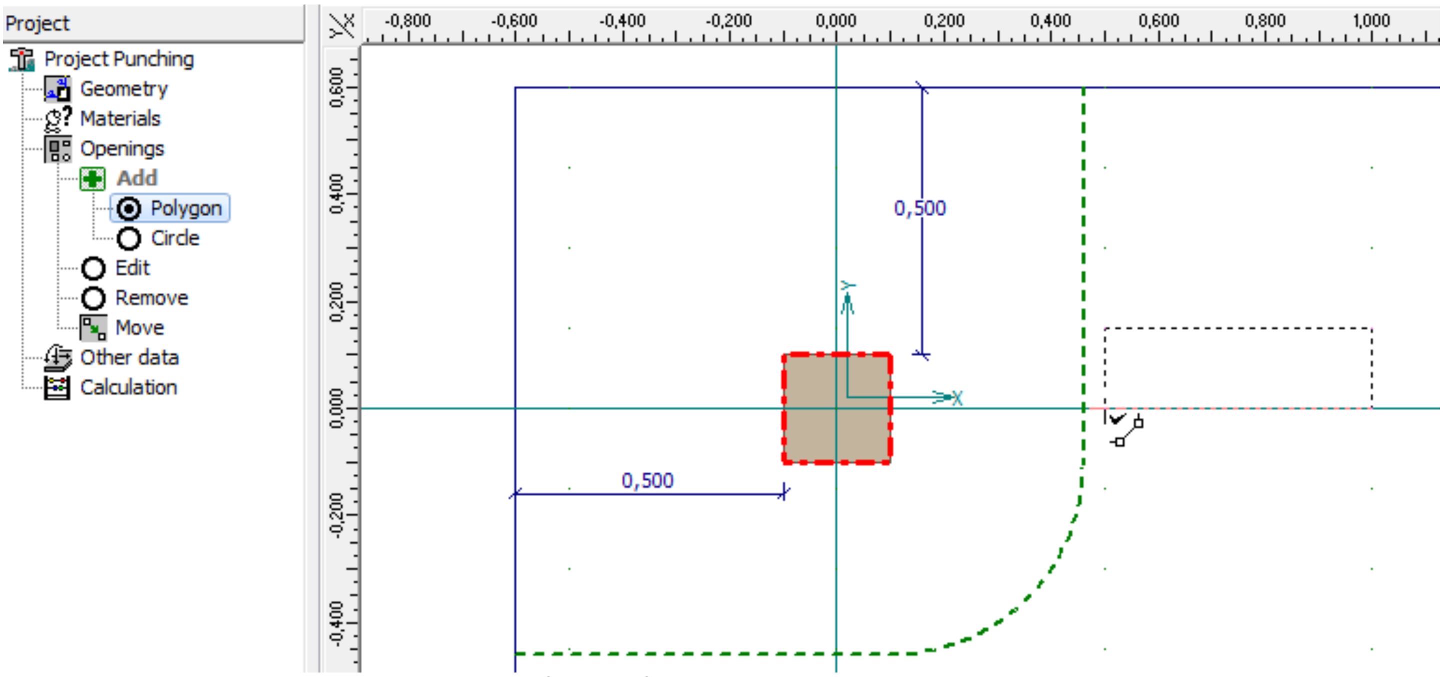
## Openings

To simplify definition of the opening we first open the "Options" dialog box by selecting "**Settings**" in the main menu. In the tab "**General**" we adjust the grid to suit our needs by defining the step as 0.5 m in the X direction and 0.15 m in the Y direction. We exit the dialog box by clicking "**OK**".




*Adjusting grid step in the "Options" dialog box*

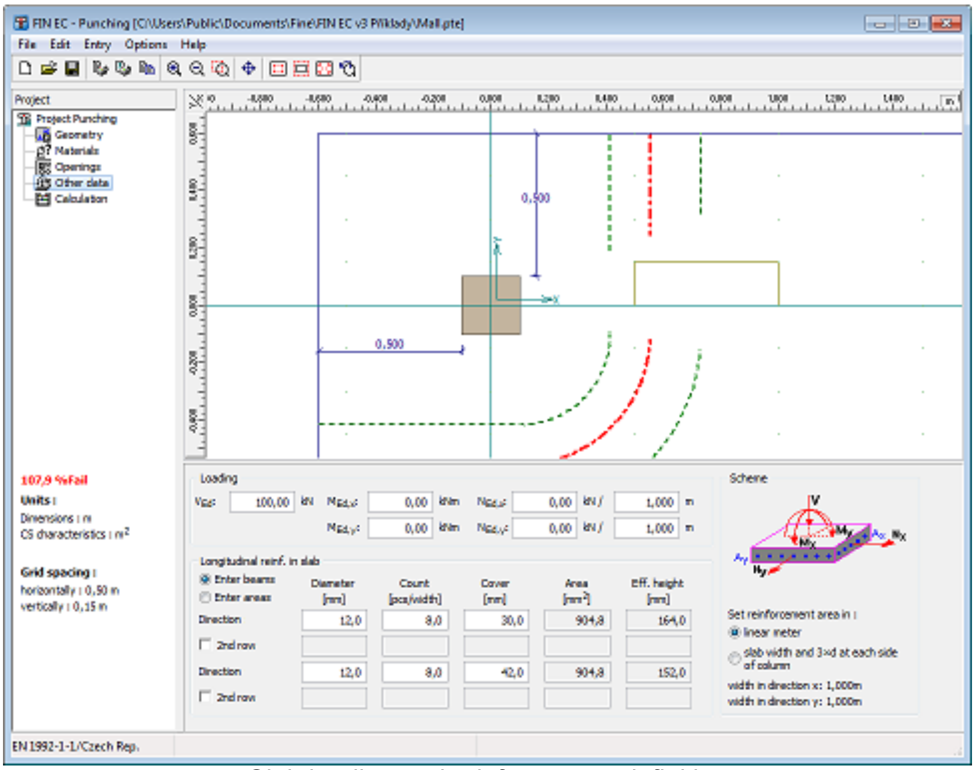
Now we can switch in the tree to "**Openings**" and "**Polygon**" and draw the contour of the opening directly in the model space clicking in each of its four corners. To finalize the contour we click again on the first corner.



*Definition of opening in model space*

## Other data

In this section of the tree we define the loading and longitudinal reinforcement of the slab as $V_{Ed}$ = 100 kN and 8 of 12 mm diameter bars per meter run.

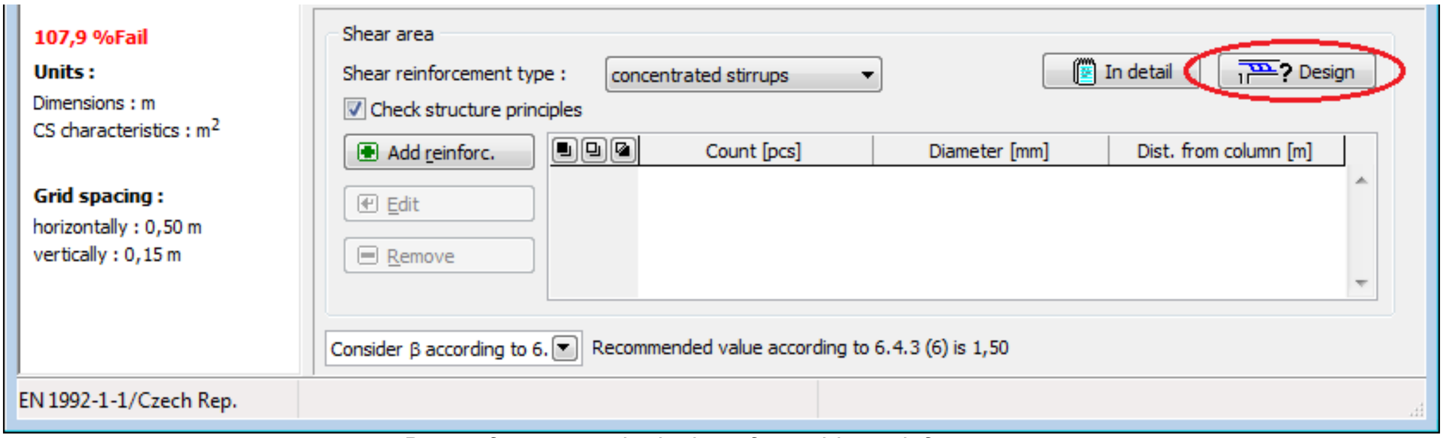


*Slab loading and reinforcement definition*

## Calculation

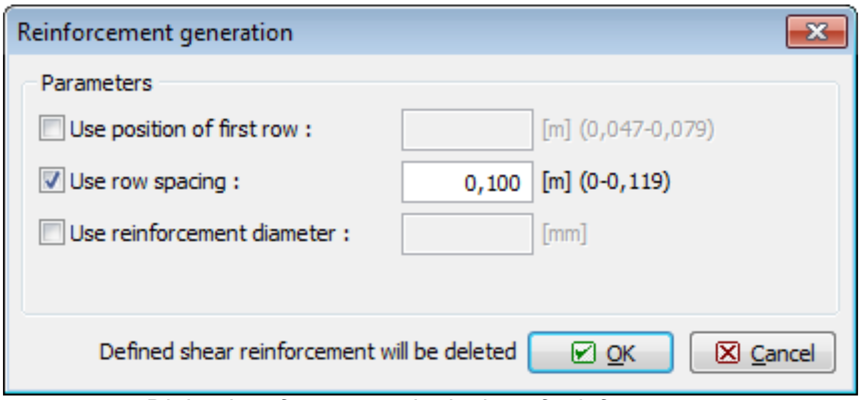
Switching to the last item of the control tree "**Calculation**" we can proceed to carrying out the design checks of the punching shear reinforcement. First we need to select the shear reinforcement type; in our example we will use "**concentrated stirrups**". Then we can either define the reinforcement manually in the table in the definition frame or we can let the program design the reinforcement automatically. Choosing the latter, we run the automatic reinforcement design by clicking the "**Design**" button.




*Button for automatic design of punching reinforcement*

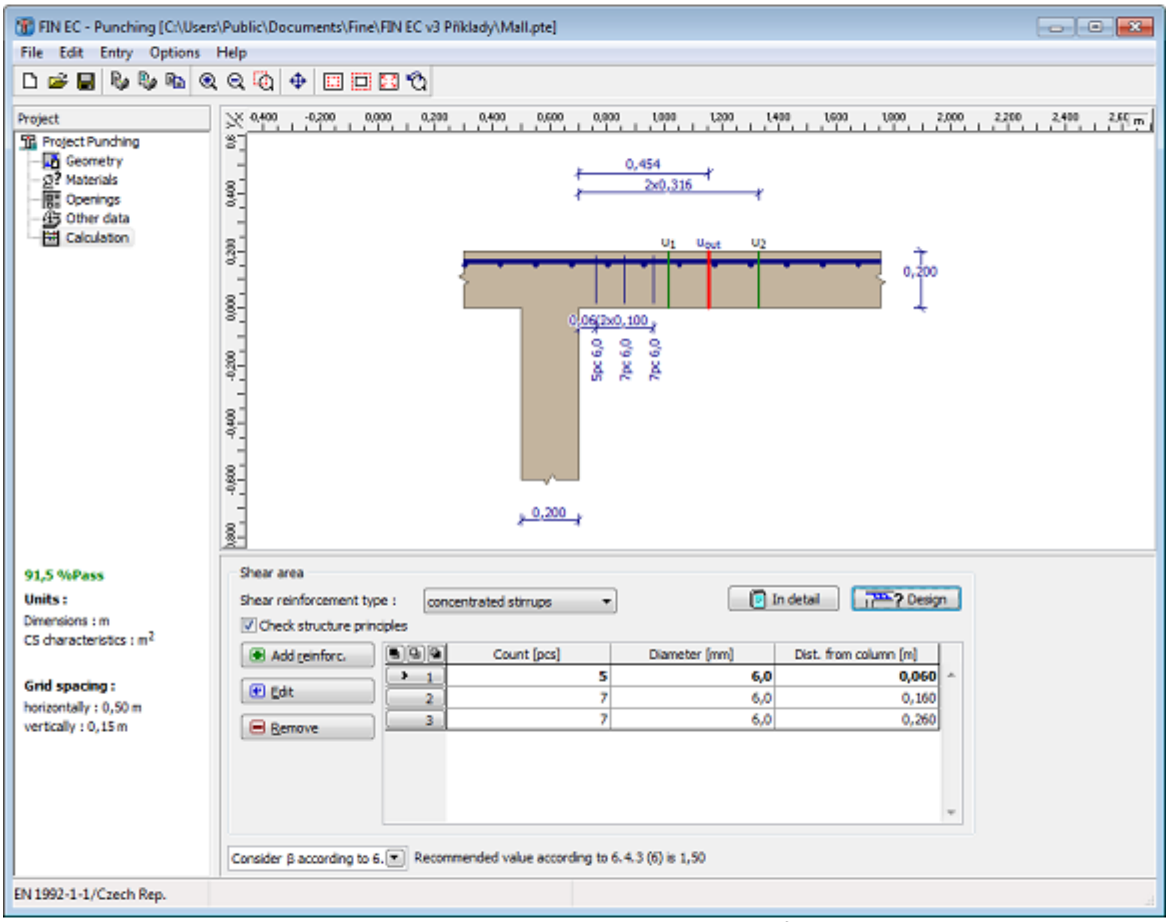
In the "**Reinforcement generation**" dialog box we can define invariable reinforcement parameters; other parameters will be defined automatically. Ticking the "**Use row spacing**" box and entering a value we define fixed stirrup spacing 100 mm; the position of the first row and the stirrups diameters we leave to be designed automatically.




*Dialog box for automatic design of reinforcement*

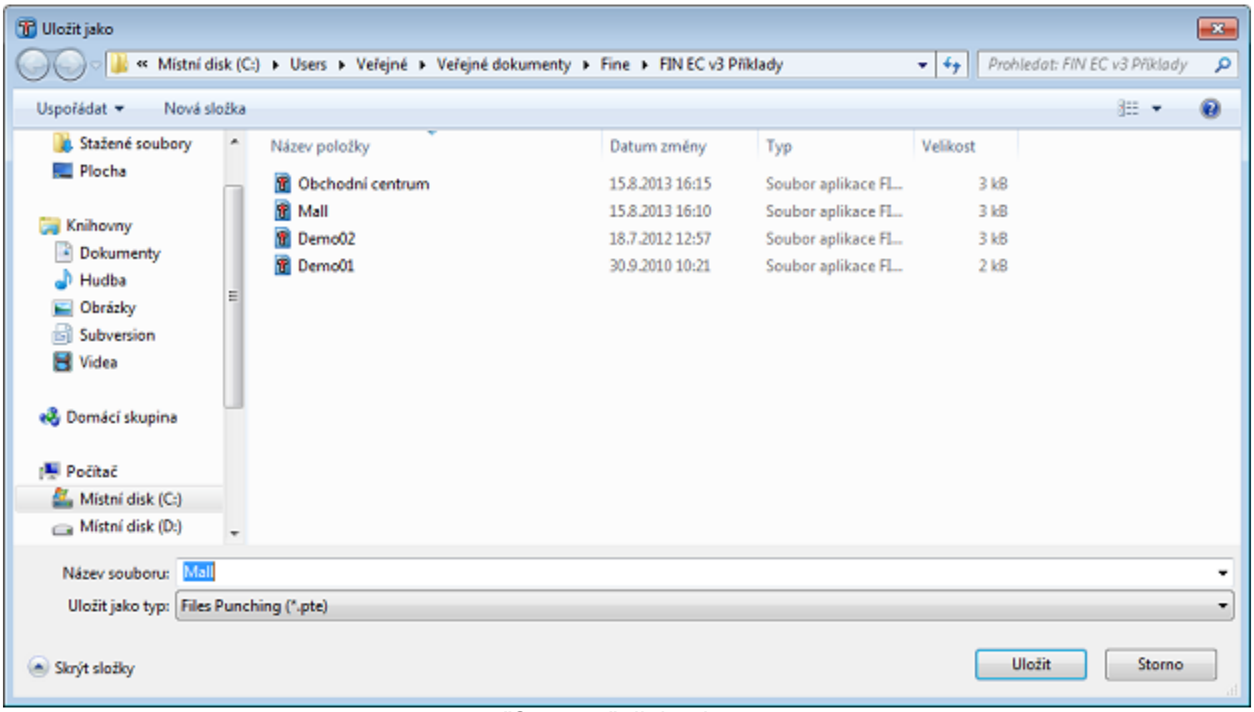
After closing the dialog box by clicking "**OK**" the reinforcement is designed to reach required capacity and fulfil all detailing requirements.




*Designed and checked punching shear reinforcement*

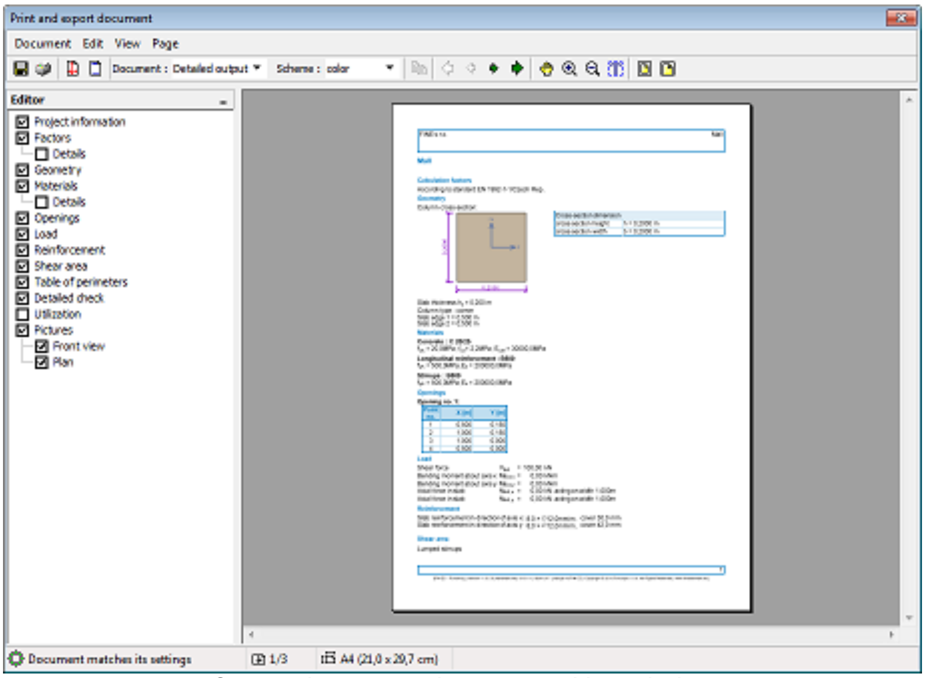
Finally, we save the job using e.g. the "**Ctrl+S**" shortcut. The program runs the standard "**Save as**" dialog box where we find and select the destination folder, enter the name and save the job.




*"Save as" dialog box.*

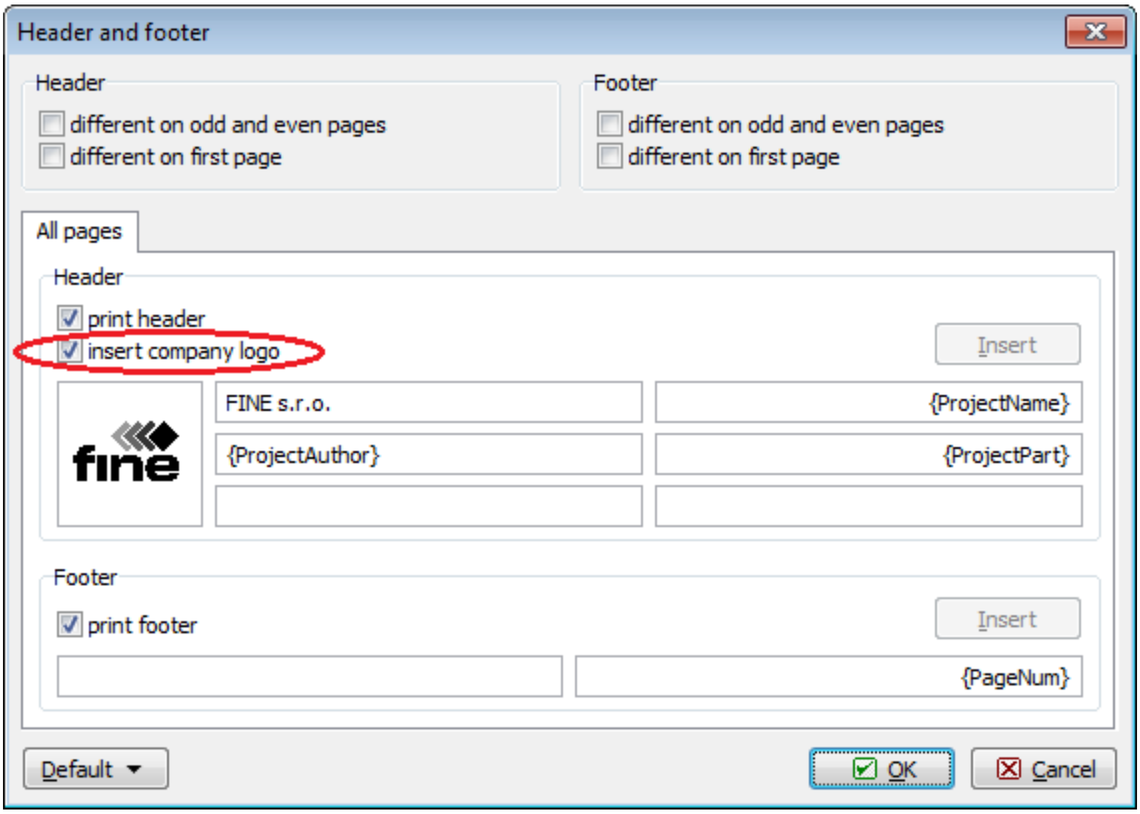
## Outputs

Composition of the output documentation follows the same procedure as for the other FIN EC programs. We can compose both graphical and/or text output; for the text output we can select in the control tree which chapters of the documentation are to be printed out. We can print the composed document directly by clicking the "" button or save it as a *.pdf or a *.rtf file by clicking the "" button.




*Output documentation composition window*

At the beginning we imported the company's logo into the program, now we can switch on displaying the logo in the documents header in the "**Header and footer**" dialog box. We run this dialog box by selecting it in the "**Document**" section of the main menu or directly by clicking the "" button in the toolbar.




*Displaying logo in document's header*

After ticking the "**insert company logo**" box, the logo appears in the headers of all documents.


*Header with the company's logo*

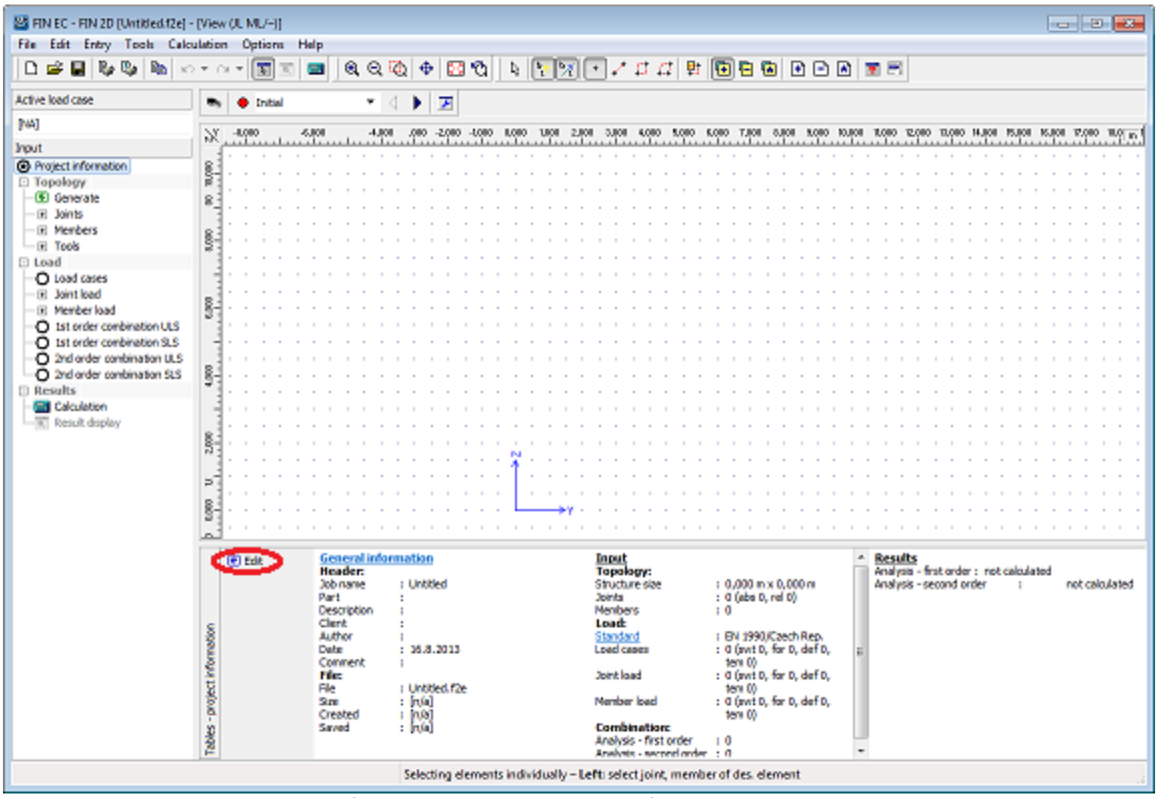
# Timber truss

## Task

In this example, the task is to design a symmetric roof timber truss of *13 m* length and *25 degrees* of the roof pitch. The truss consists of timber members of class *C24* and *40 mm* thickness; the spacing of the trusses is *1 m* centres. The truss is subject to dead load *0.2 kN/m* from both roofing and ceilings and to snow load *1.0 kN/m* according to the Snow area 2 of the Czech snow map.

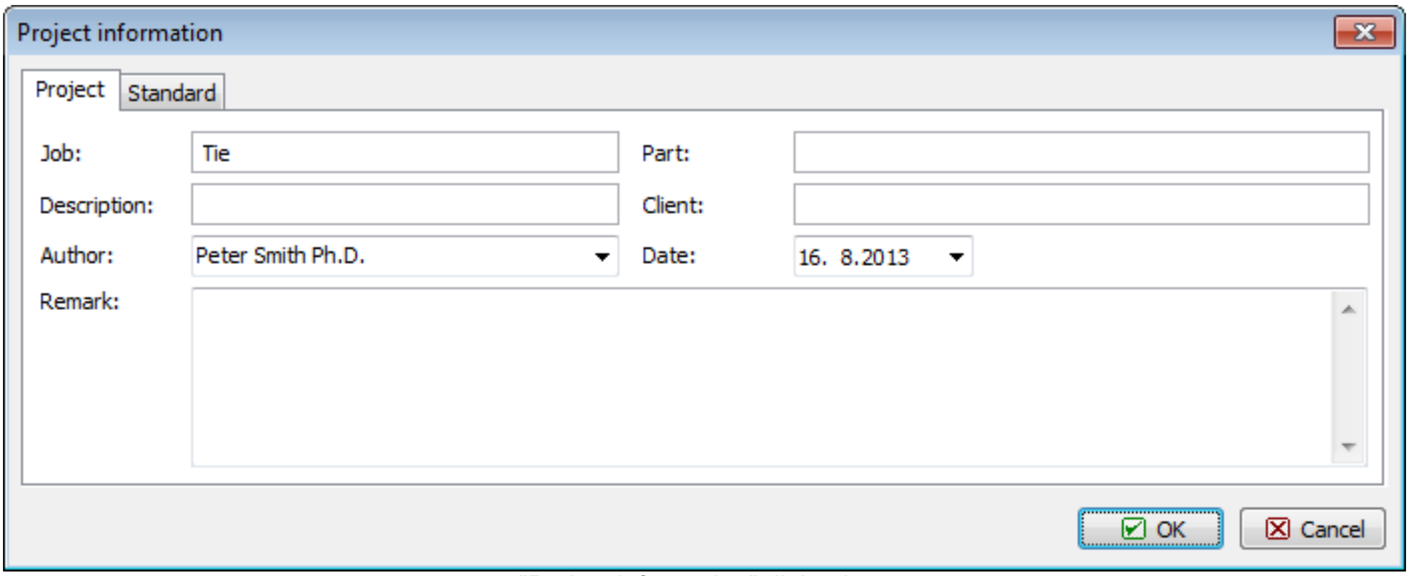
## Setting up project

After running the program 2D, the main screen appears, consisting of the model space on the right-hand side, the control tree on the left-hand side and the input table in the bottom part. The table displays the project information which can be later used in headers and footers of the output documents. To enter or edit the project information, we can run the relevant dialog box by clicking the "**Edit**" button.




*Button for running the "Project information" dialog box*

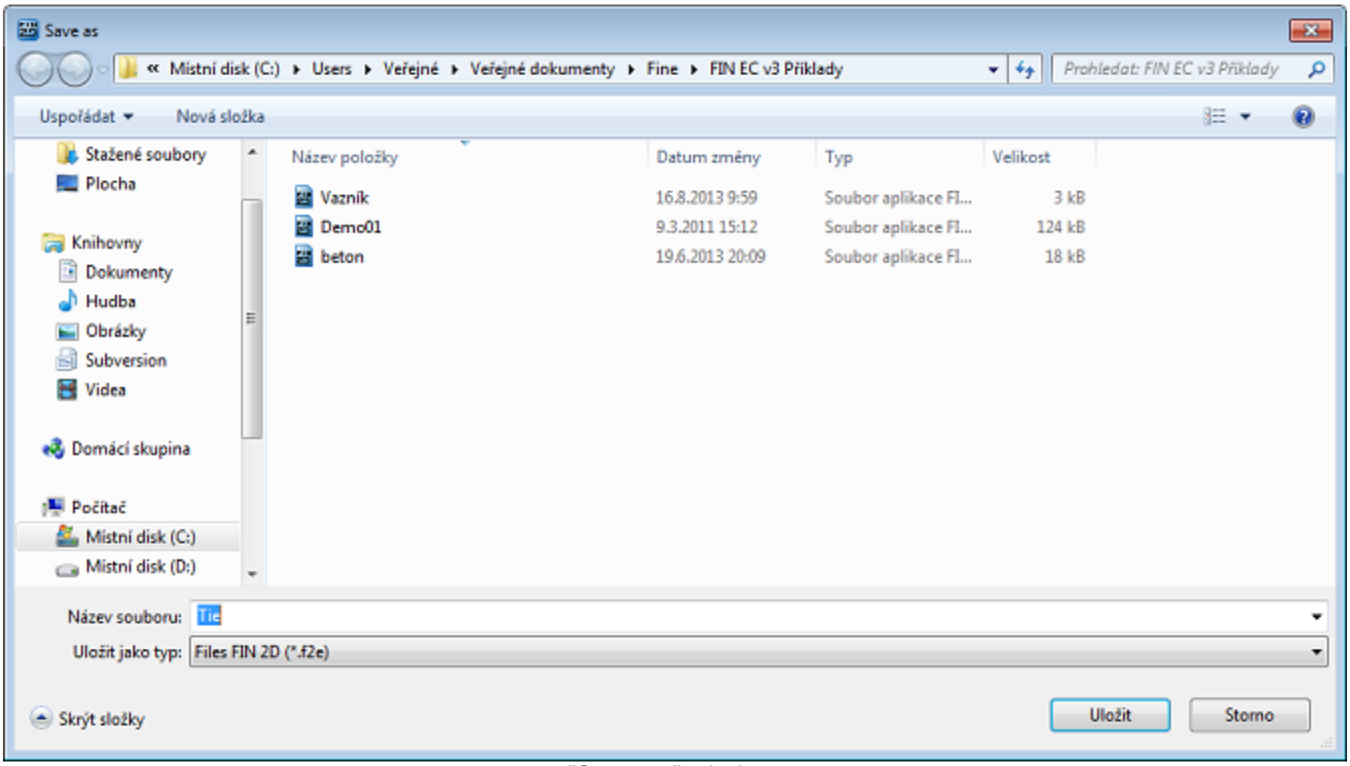
In the dialog box we can enter e.g. the job title or the project author. After entering all necessary data we exit the dialog box by clicking the "**OK**" button.




*"Project information" dialog box.*

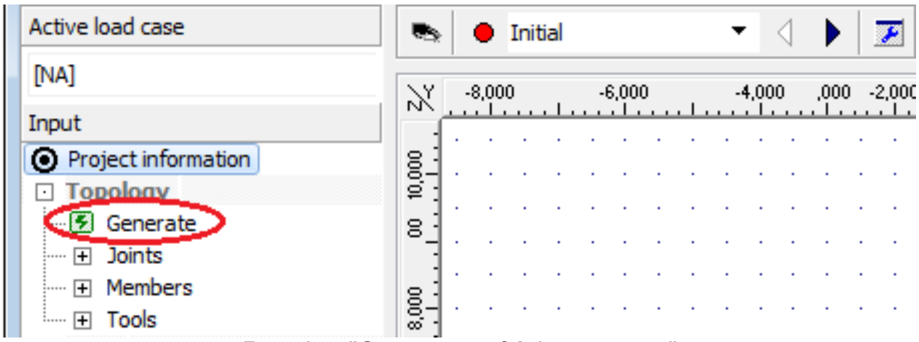
Before proceeding with work we should save the job using e.g. the "**Ctrl+S**" shortcut. The file name is entered and the destination folder selected in the standard 'Save as' window.




*"Save as" window.*

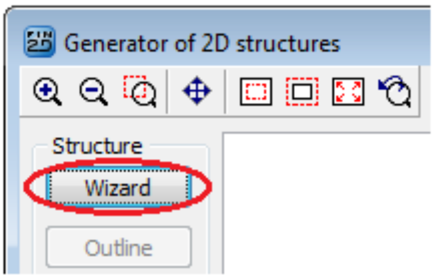
## Structure generation

We can define the truss geometry either by entering individual nodes and members or we can simply make use of the Generator of 2D structures. In our example we will choose the latter approach. The generator is run by clicking the "**Generate**" button in the "**Topology**" section of the control tree.



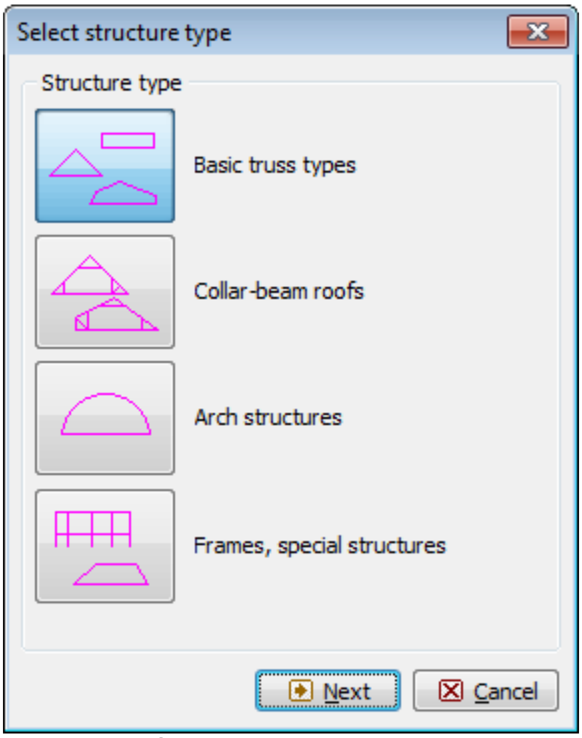

*Running "Generator of 2d structures"*

To generate the structure in the generator, we first click the "**Wizard**" button in the top left corner of the dialog box.



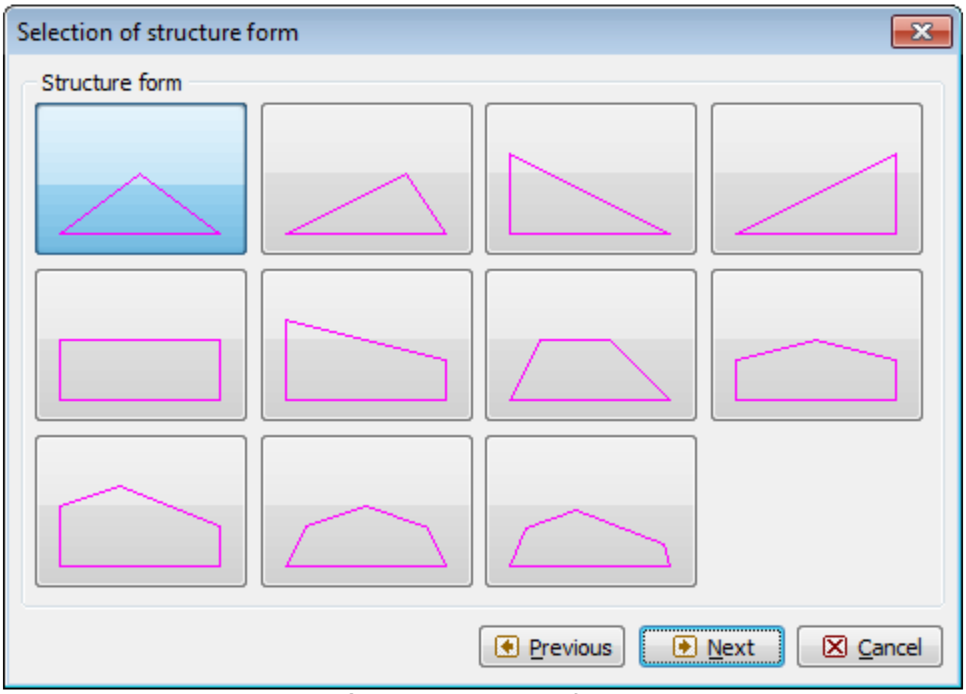

*Running wizard in Generator of 2D structures*

A dialog box appears enabling us to select one of the basic types of the structure. We select Basic truss types and proceed to the next step by clicking the "**Next**" button.




*Selecting structure type*

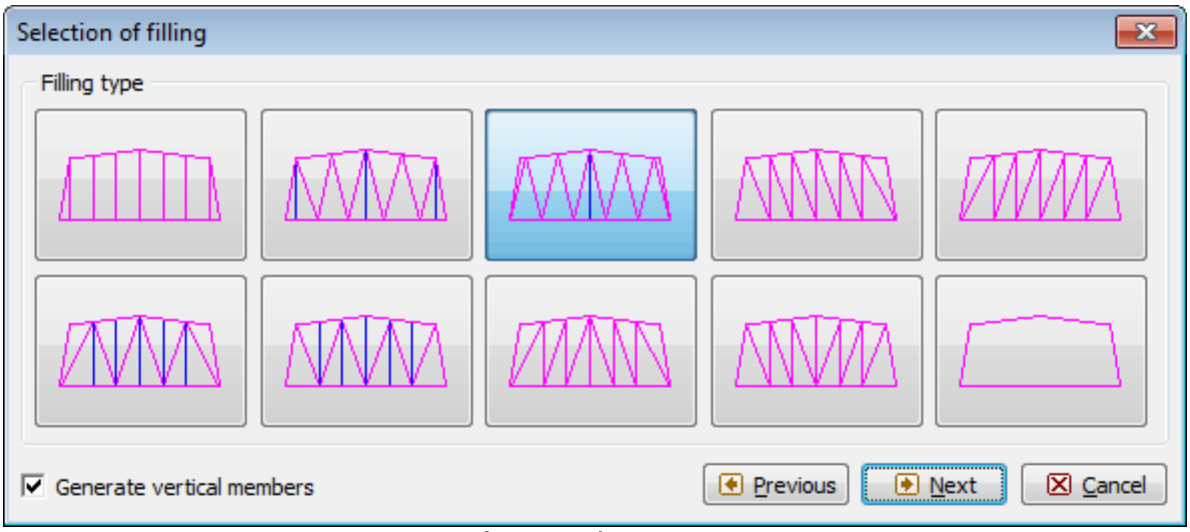
In the following dialog box, we select the desired form of the truss and click the "**Next**" button.




*Selecting structure form*

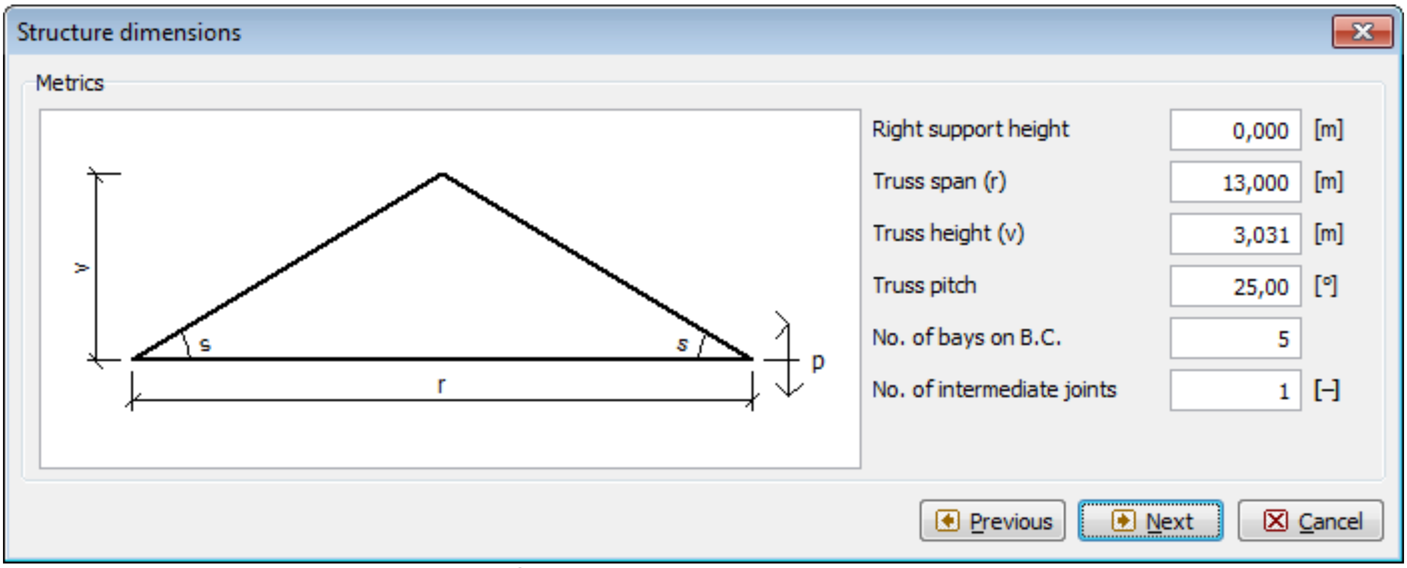
The following dialog box offers a selection of basic types of filling members' layouts. We select the desired filling type and in the bottom left corner we switch off automatic entering of verticals by unticking the "**Generate vertical members**" box.




*Selecting filling type*

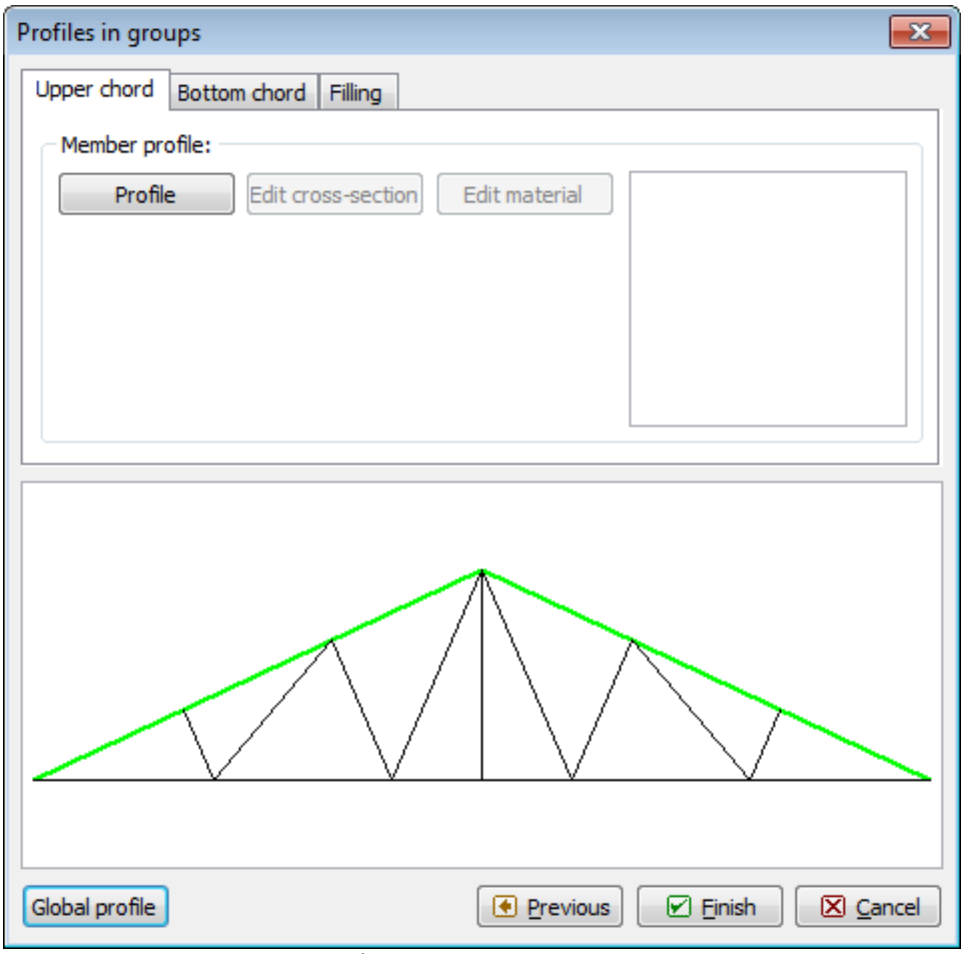
The next dialog box "**Structure dimensions**" enables defining the main dimensions of the truss. If we enter the pitch and the length, the program will automatically calculate the height of the truss. In the field "**No. of bays on B.C.**" we define into how many segments the bottom chord will be divided by nodes. Additional nodes may be inserted between the main nodes of the upper and the bottom chords. In these nodes, the exact values of internal forces and deformations will be calculated and additional nodal forces can be defined. In the field "**No. of intermediate joints**" we can define how many nodes will be added to each segment.




*"Structure dimensions" dialog box.*

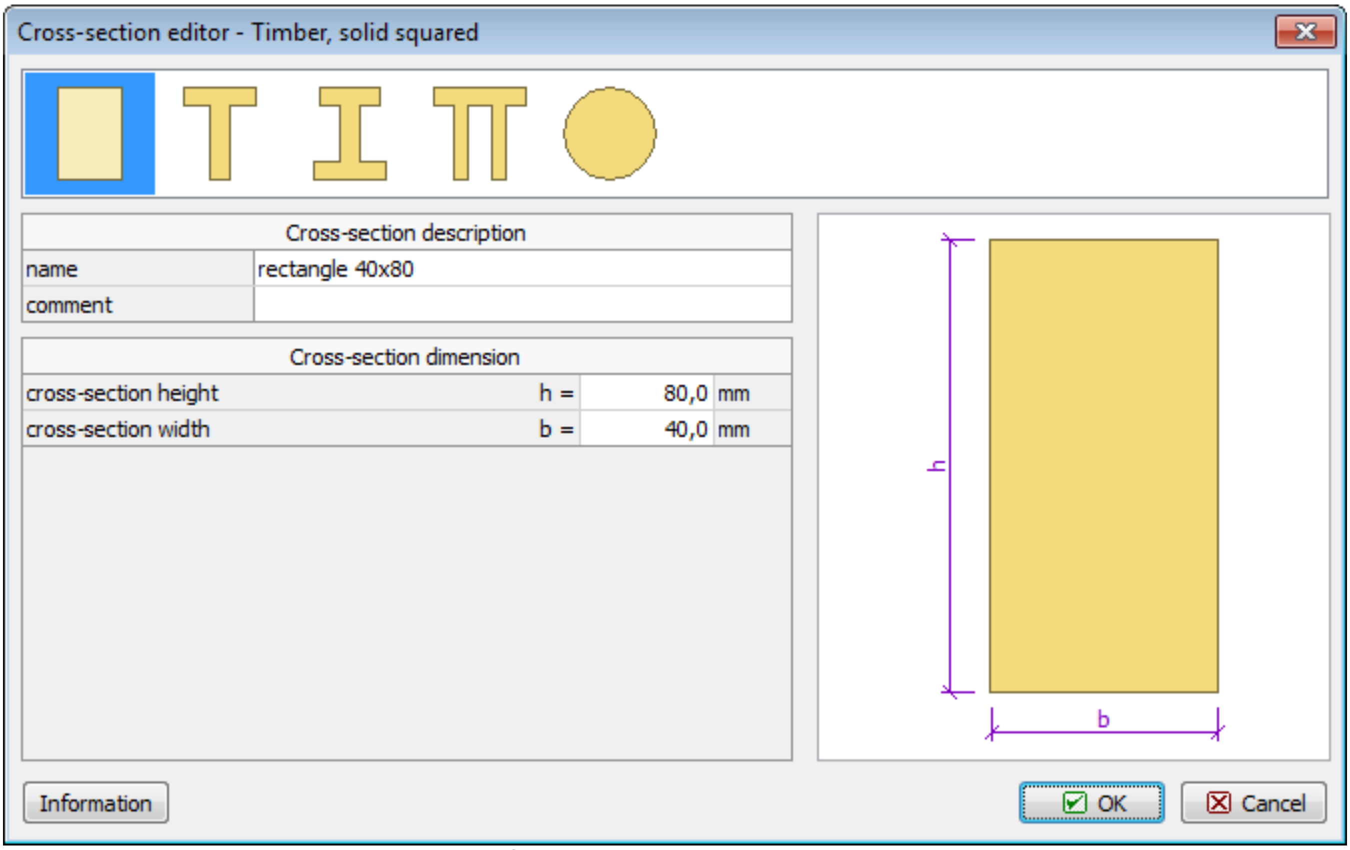
After definition of the truss dimensions, we need to specify cross-sections and materials of individual truss members. This will be done in the following dialog box "**Profiles in groups**" in which we can specify profiles and materials separately for the upper and the bottom chords and the filling studs by clicking the "**Profile**" button in the relevant tabs. However, we will use a quicker approach specifying first the material and profile for all members by clicking the "**Global profile**" and then only changing the profiles of the upper and the bottom chords.




*"Profiles in groups" dialog box*

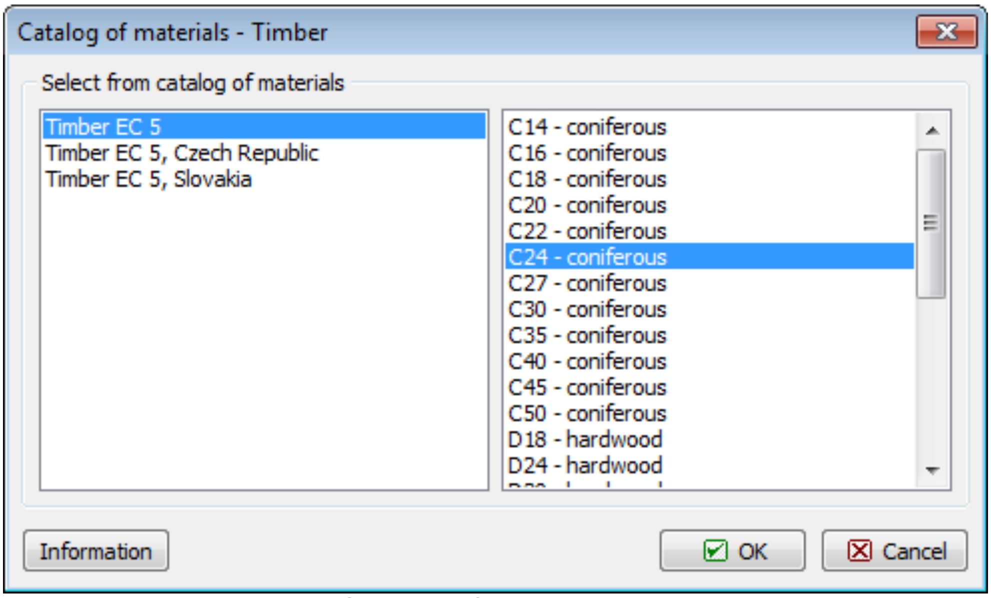
After clicking the "**Profile**" button, the "**Cross-section editor**" dialog box appears on screen. In this dialog box we select "**Timber**" and "**Solid squared**" type and define the dimensions of the rectangular profile as *h = 80 mm* and *b = 40 mm*.




*Defining cross-section dimensions*

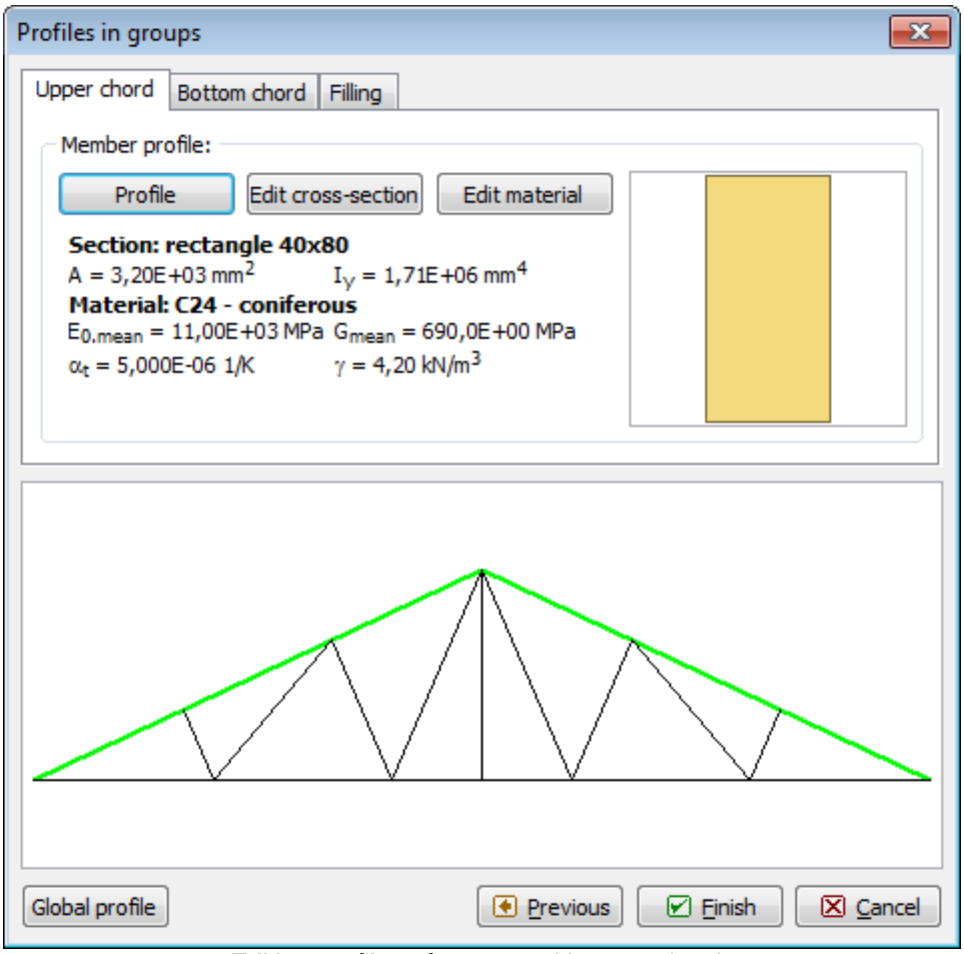
 

After confirming the dimensions by clicking "**OK**", the "**Catalogue of materials – timber**" dialog box appears, offering a selection of the standard timber strength classes. We select **C24** and confirm by clicking the "**OK**" button.




*Selection of strength class*

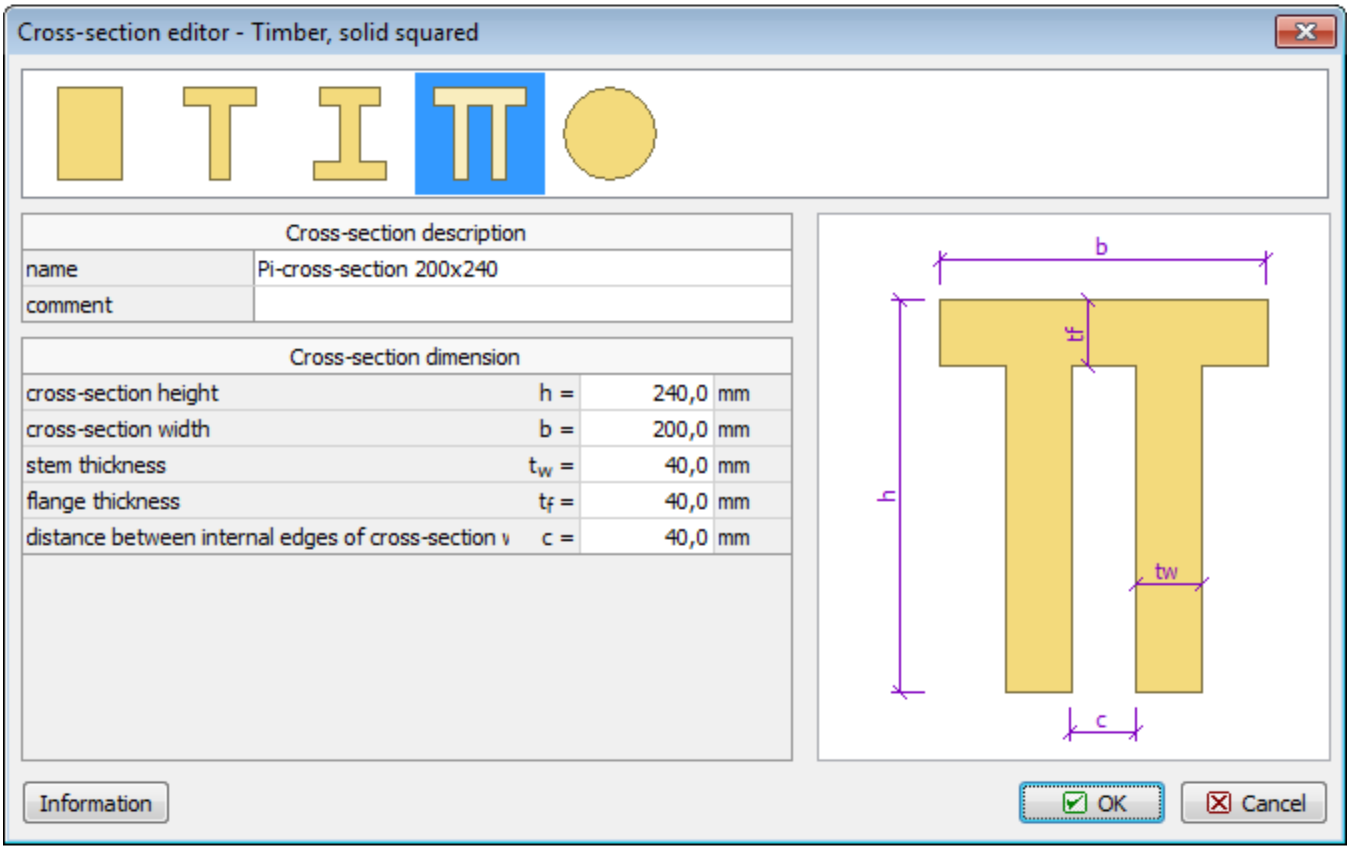
After confirming, the selected material and cross-section data appear in all three tabs of the "**Profiles in groups**" dialog box.




*Editing profiles of upper and bottom chords*

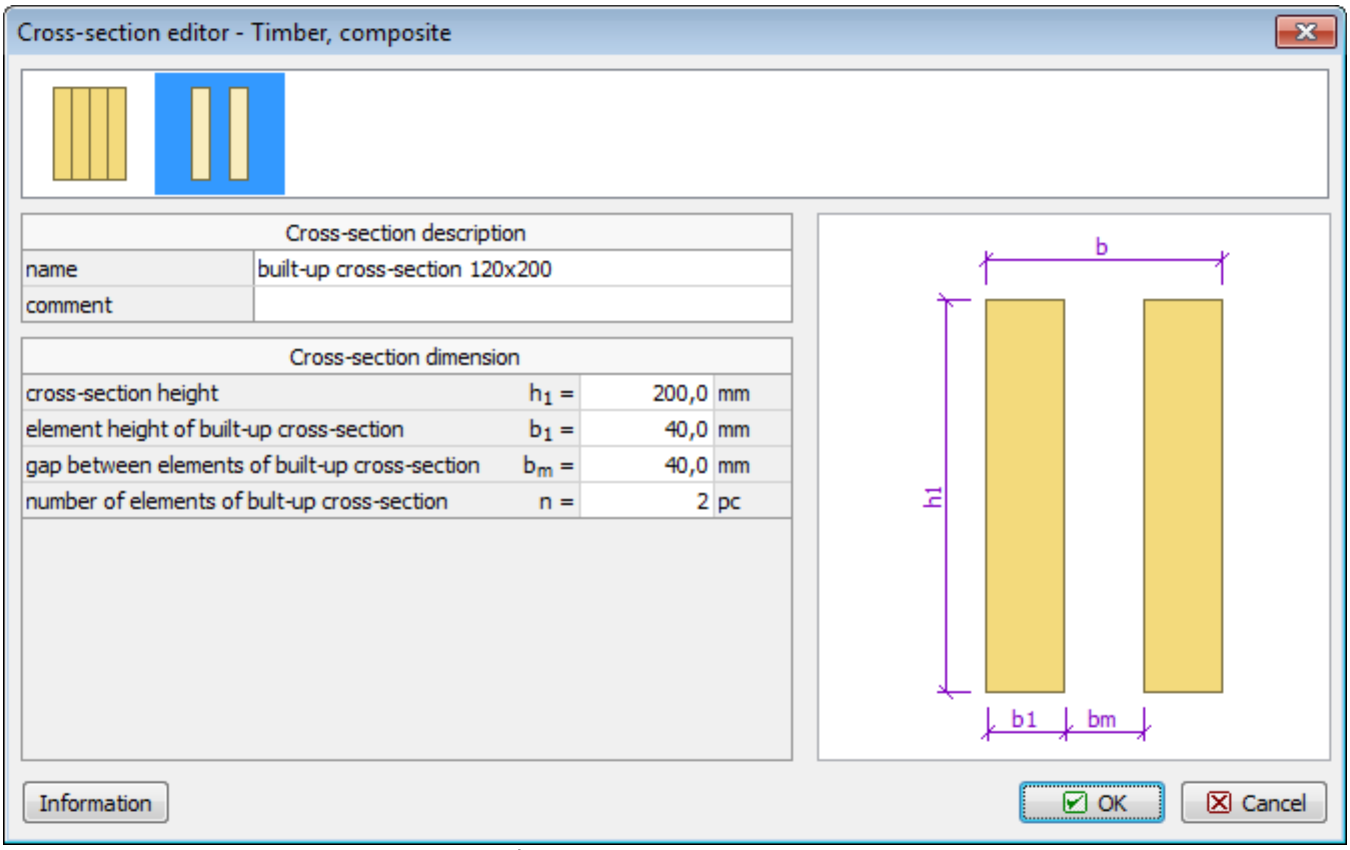
Now we change the profiles of the upper and the bottom chords. First, in the "**Upper chord**" tab using the "**Profile**" button, we set a Pi-shaped profile and enter the dimensions.



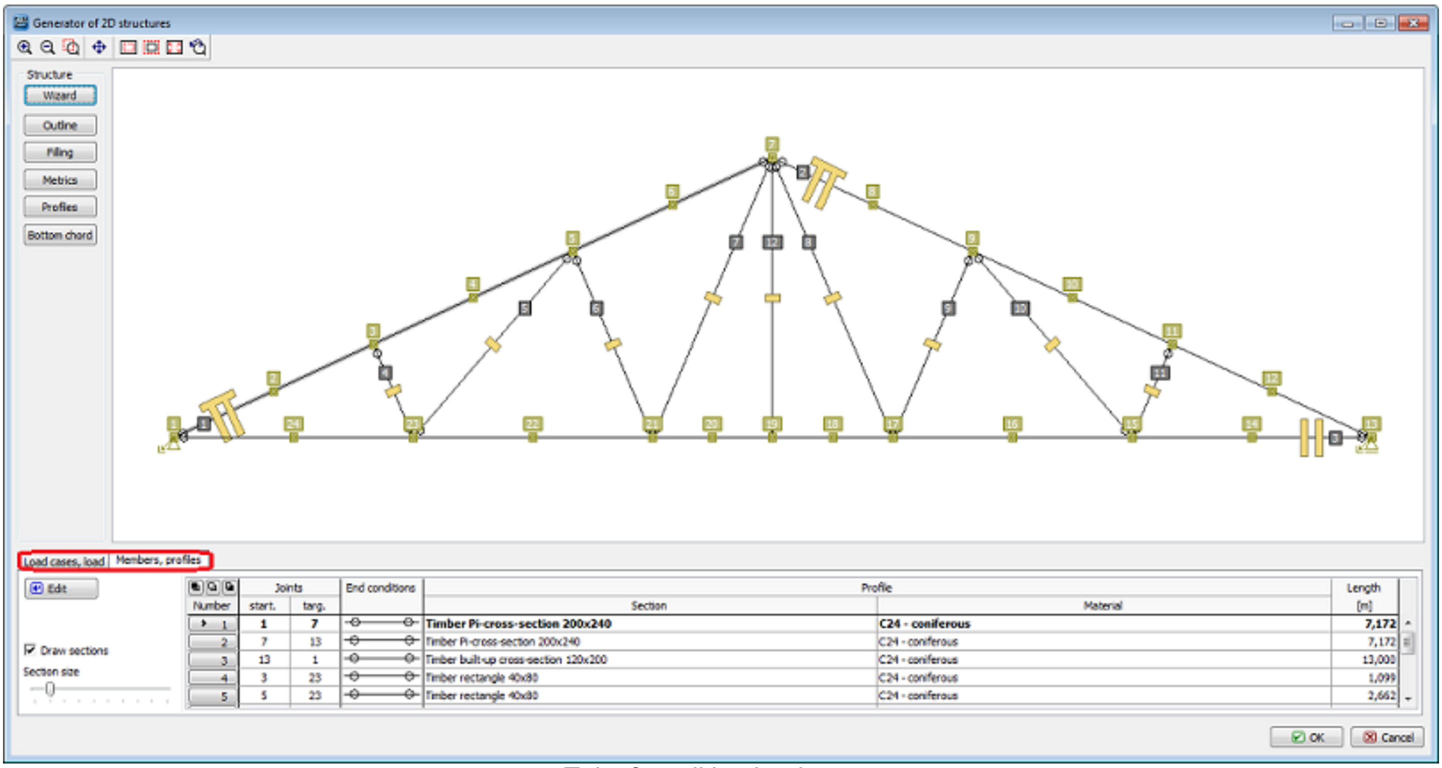

*Upper chord's cross-section dimensions*

Analogically, we define a compound cross-section of the bottom chord.



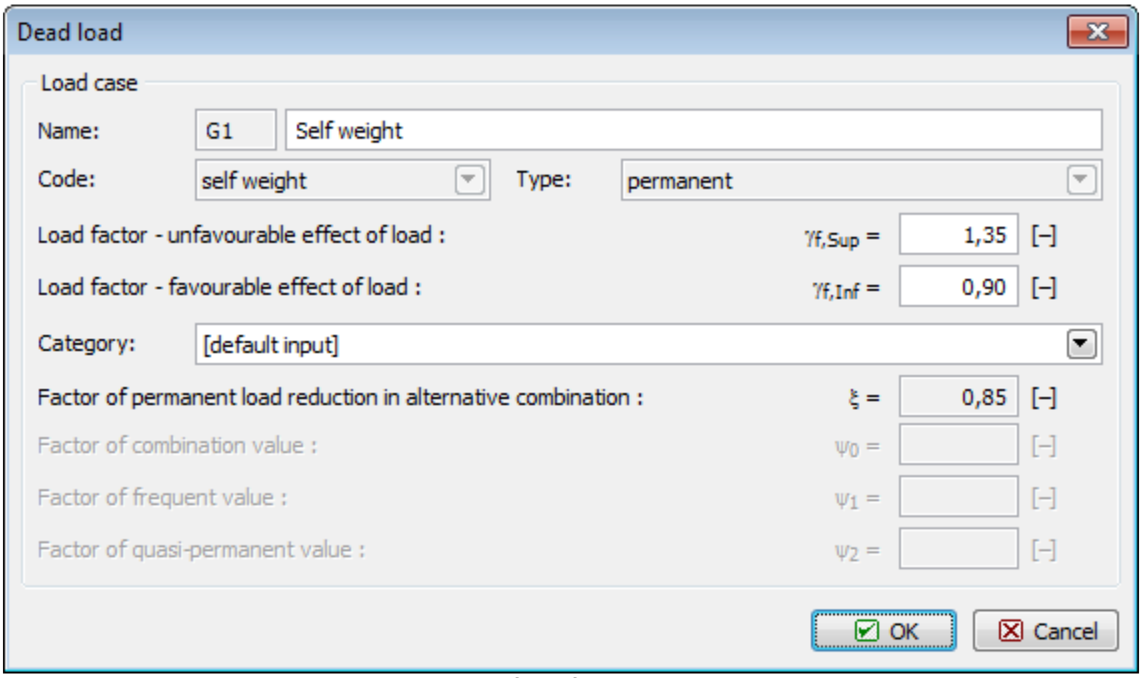

*Dimensions of bottom chord's compound cross-section*

After changing the chords' profiles we exit the "**Cross-section editor**" by clicking "**OK**". The generated truss is now displayed on screen. We can go back to any of the previous steps using relevant buttons in the "**Structure**" frame to the left from the model space. In the frame in the bottom of the screen, the tables for managing load cases and members are organized into tabs.



*Tabs for editing load cases*

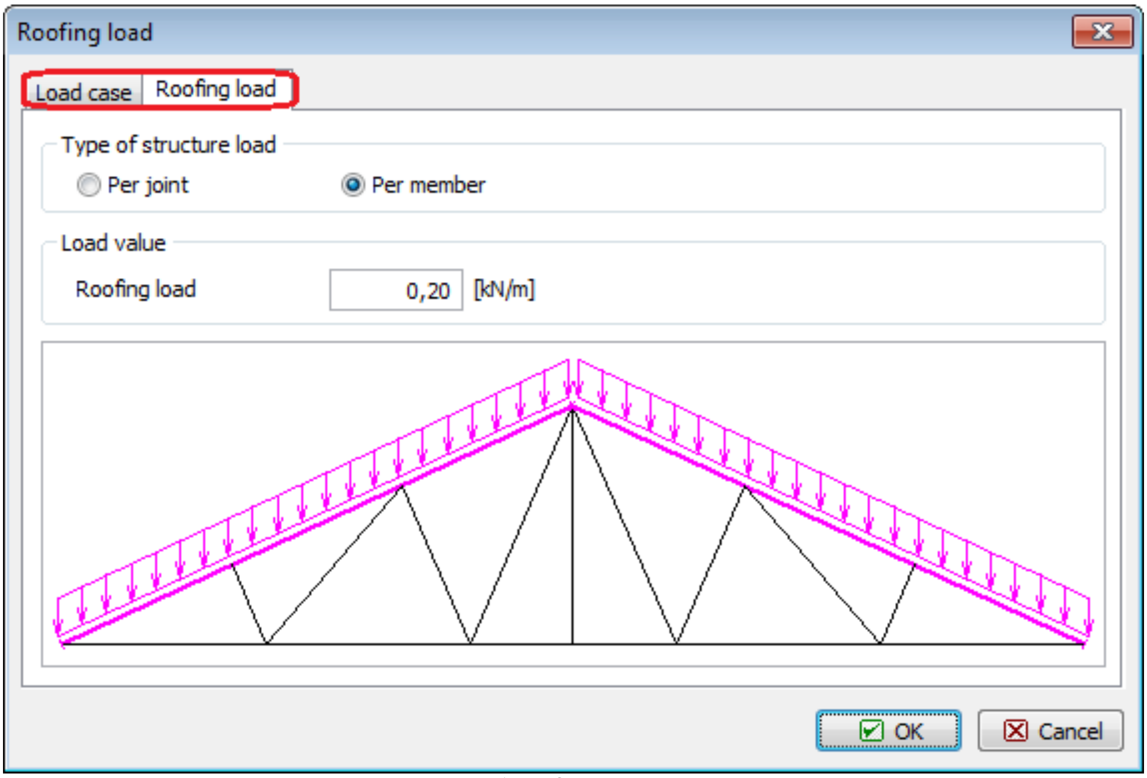
We select the "**Load cases, load**' tab and begin to define the load cases. Firstly, selecting "**Self weight**" we define a load case, which will contain automatically generated loads from the truss's self weight. In the dialog box we can edit the name of the load case or load factors.




*Parameters of "self weight" load case*

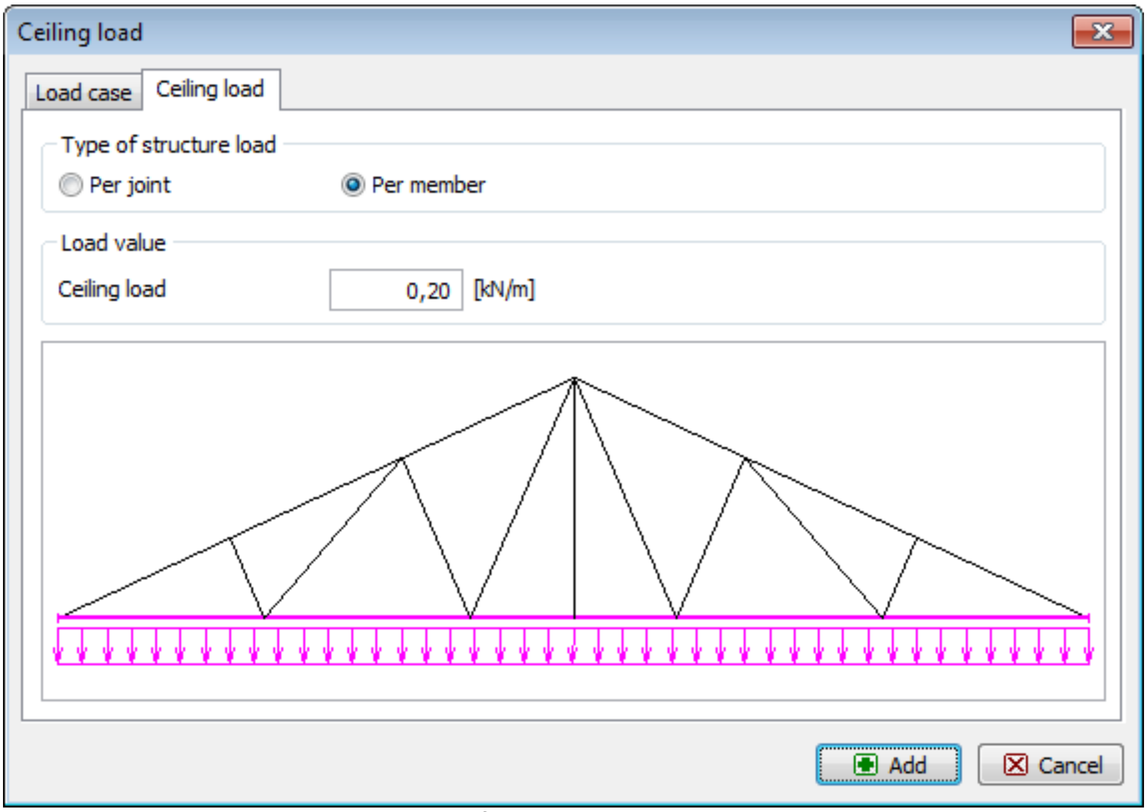
Adding a new load case is confirmed by clicking "**OK**" and the loads generated from the self weight into this load case are instantly displayed in the model space. We continue with adding loads from the roofing, using the "**Roofing**" button. The "**Roofing load**" dialog box contains two tabs; the first is for specifying the load case parameters (similarly to self weight), in the second the load magnitude is defined. We switch to the second tab to enter the value *0.2 kN/m* and add the load case by clicking "**OK**". Then we exit the dialog box by clicking the "**Cancel**" button.




*Tabs in the 'Roofing load' dialog box*

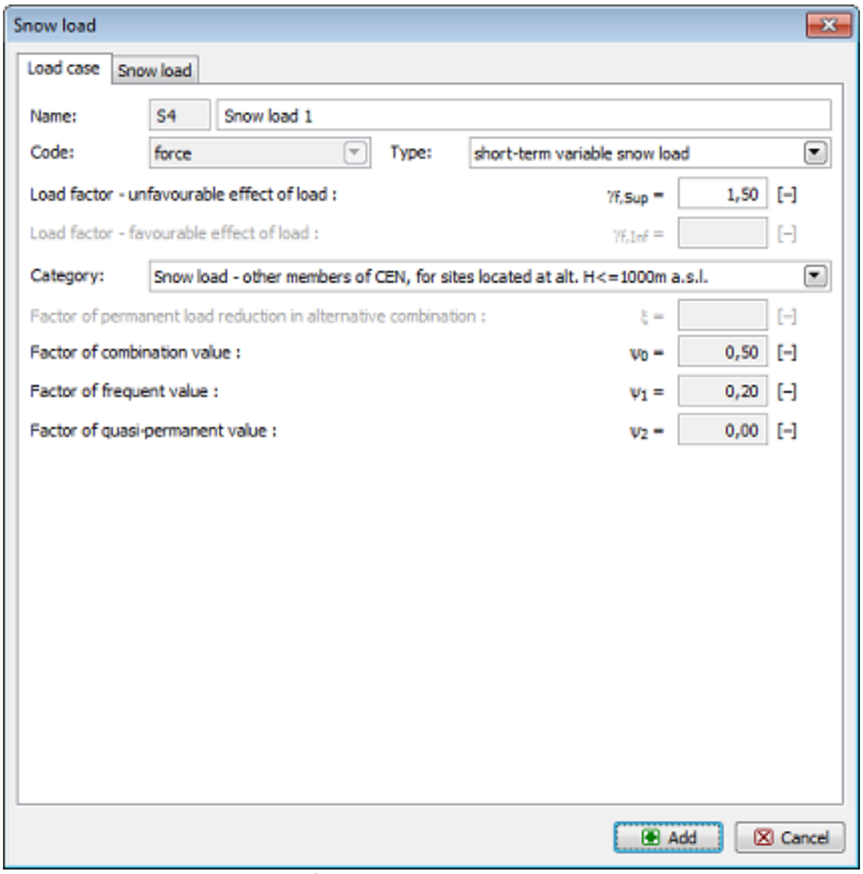
We repeat the same procedure to define the ceiling loads.




*Defining ceiling loads.*

For snow loads, due to the variable nature of the loading, the dialog box for the load case properties contains different data than that for the permanent loads. Short or medium term loading type can be selected, as well as the "**Category**" which sets the combination factors in accordance with EN 1990.

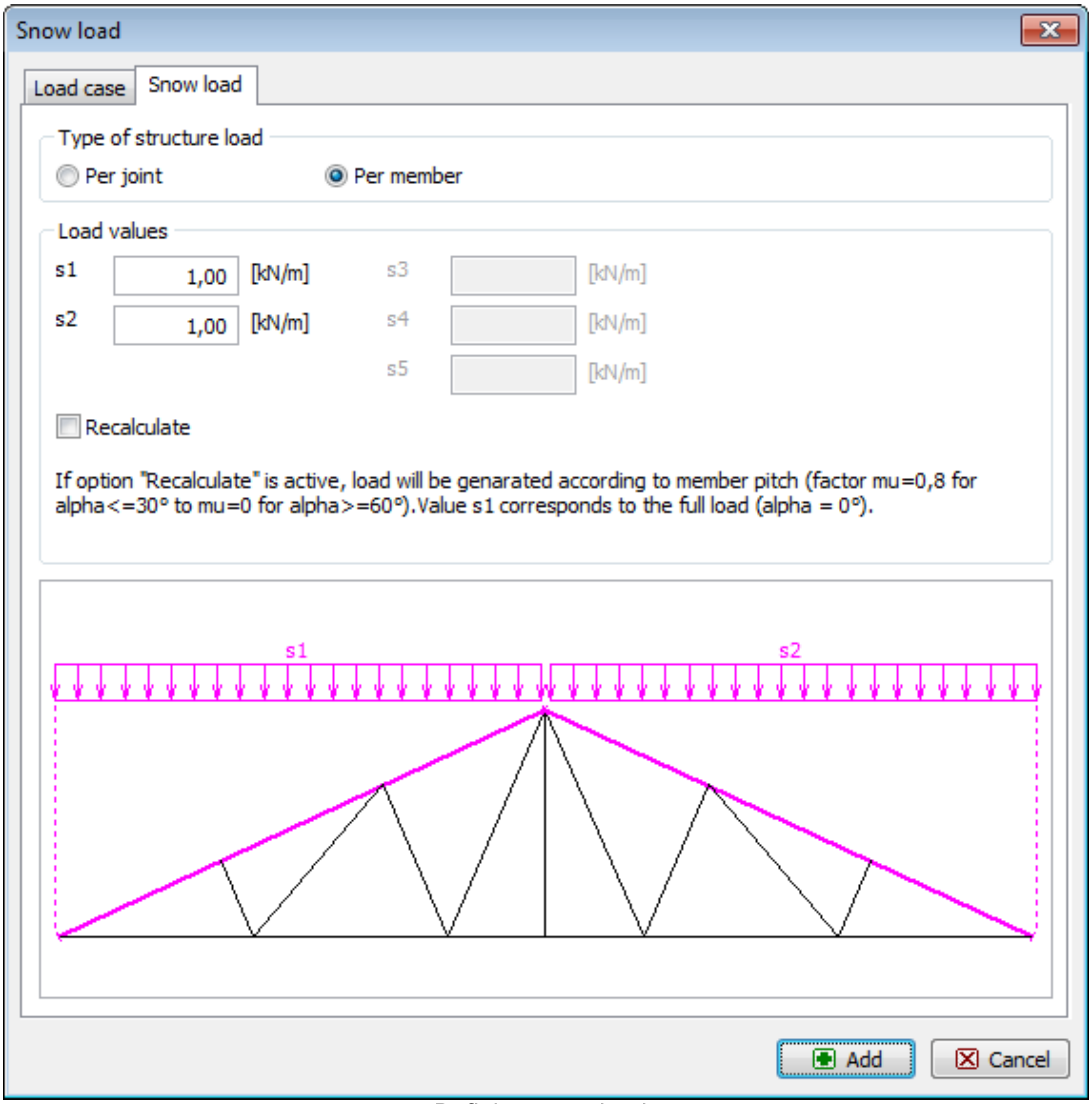
 




*Snow load properties*

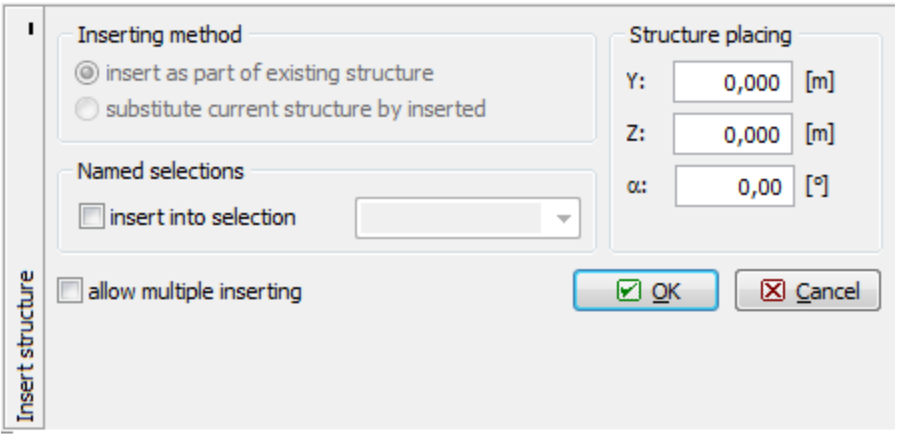
In the second tab, we can define the loads separately for the left and the right side of the truss; it is possible to define non-uniformly distributed load caused by snowdrifts. The magnitude of the loading can be entered as the basic value obtained from the snow map; automatic redistribution on the inclined plane is run by ticking the "**Recalculate**" box. First we define the load case with uniformly distributed loads *1.0 kN/m* applied to both halves of the truss; the load is applied to the structure by clicking the "**Add**" button. Then we can change the value *s1* to *0.5 kN/m*; thus we obtain a non-uniformly distributed load case which we again apply to the truss by clicking the "**Add**" button. Finally, we switch the values *s1* and *s2* to obtain a load case symmetric to the previous. We apply it by clicking the "**Add**" button and the "**Cancel**" button to exit the dialog box.




*Defining snow loads*

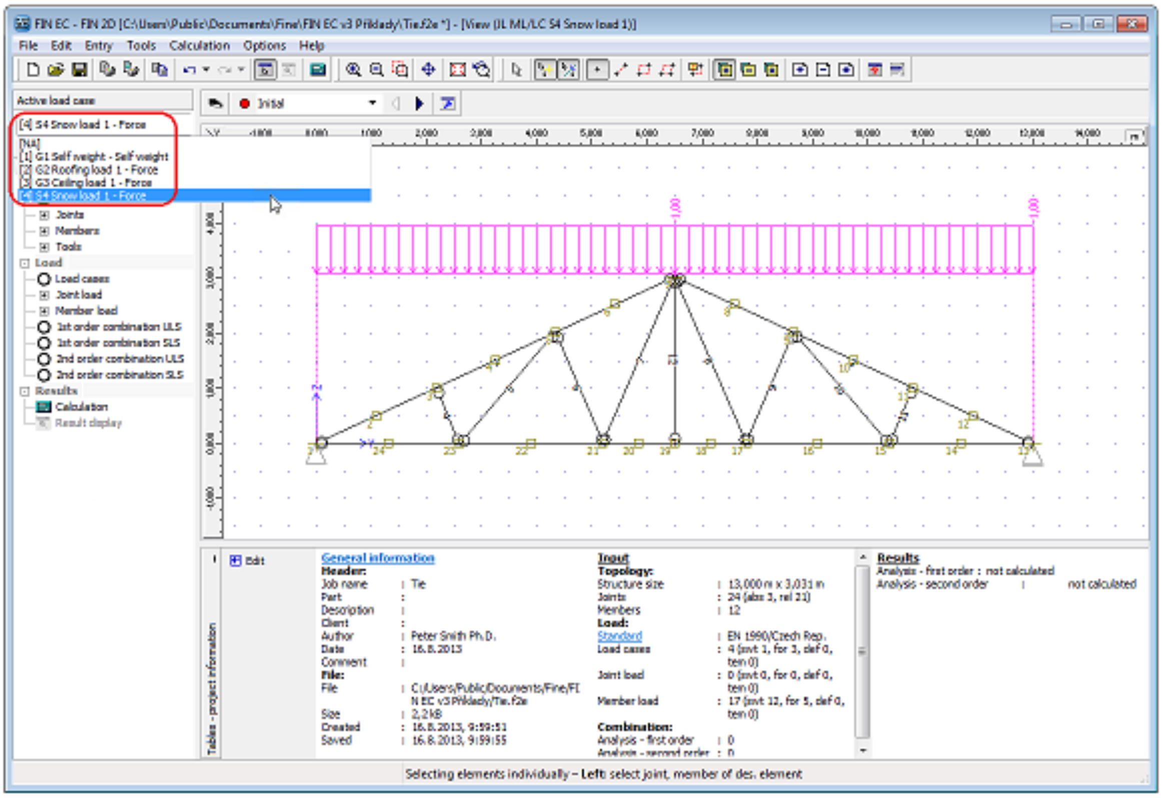
We can check and amend the defined load cases using the table in the bottom part of the Generator of 2D structures. If the load cases are correctly defined, we can insert the generated structure into the 2D program by clicking "**OK**". We can define structure placing and rotation in the table located in the bottom part of the main screen.




*"Insert structure" table.*

After inserting, the truss is displayed in the program's model space. Geometry can be further edited in the "**Topology**" part of the control tree, load cases and loads can be edited in the "**Loads**" part. Only the active load case selected from the drop down list in the upper part of the control tree is displayed in the model space.

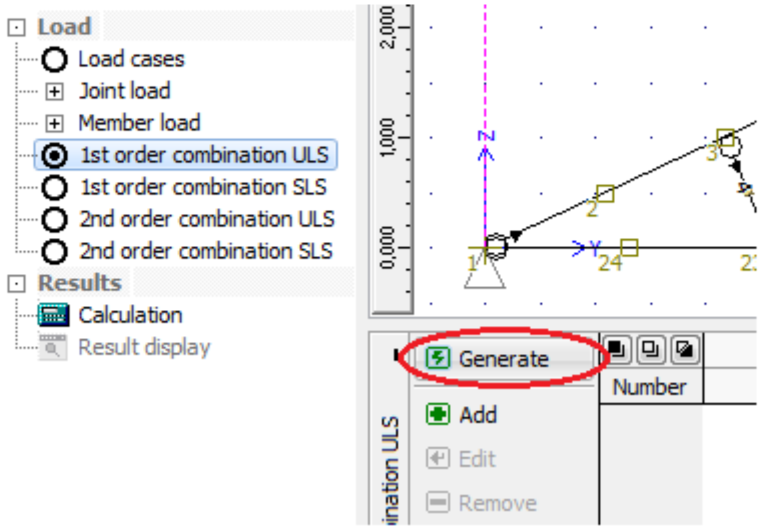
 



*Displaying particular load cases*

## Definition of combinations

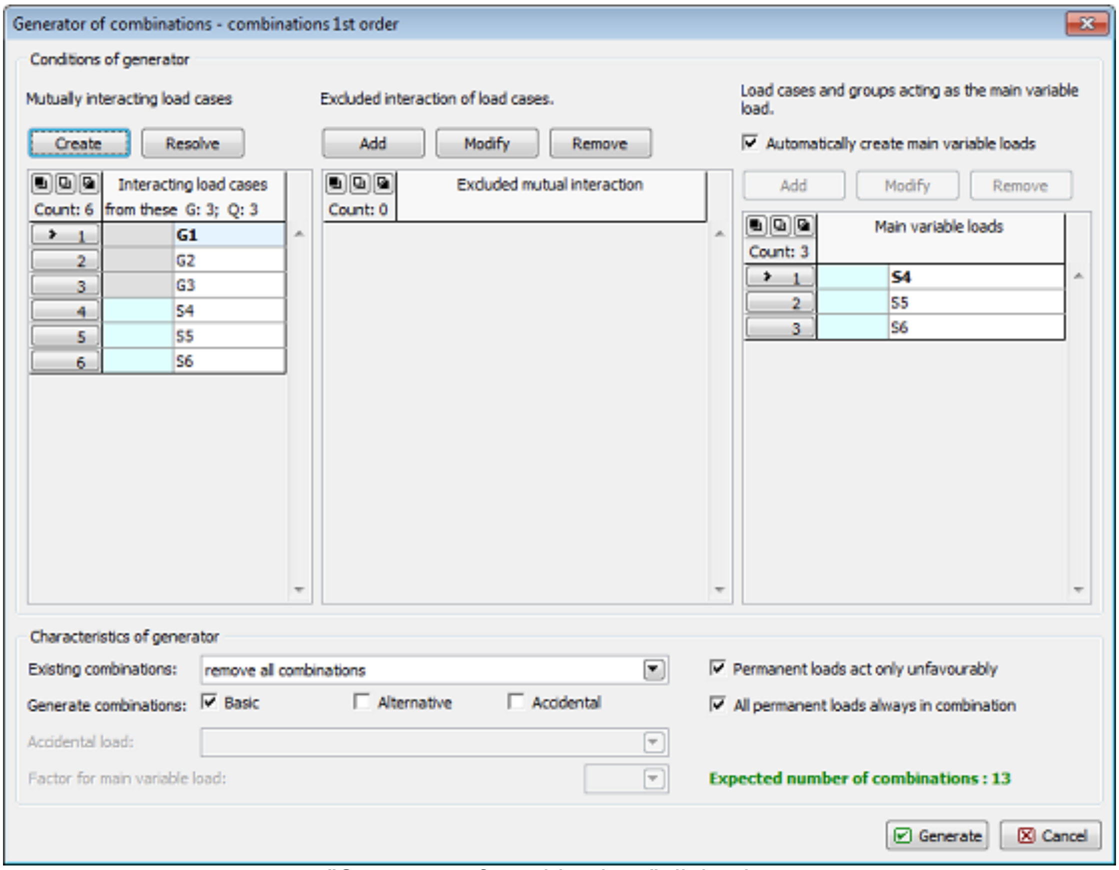
We can proceed to defining the load combinations which are defined separately for the ultimate and the serviceability limit states. First we define the combination for the ultimate limit state. We switch to "**1st order combination ULS**" in the control tree and run automatic definition of combination by clicking the "**Generate**" button in the table of combinations.




*Button for automatic definition of combinations.*

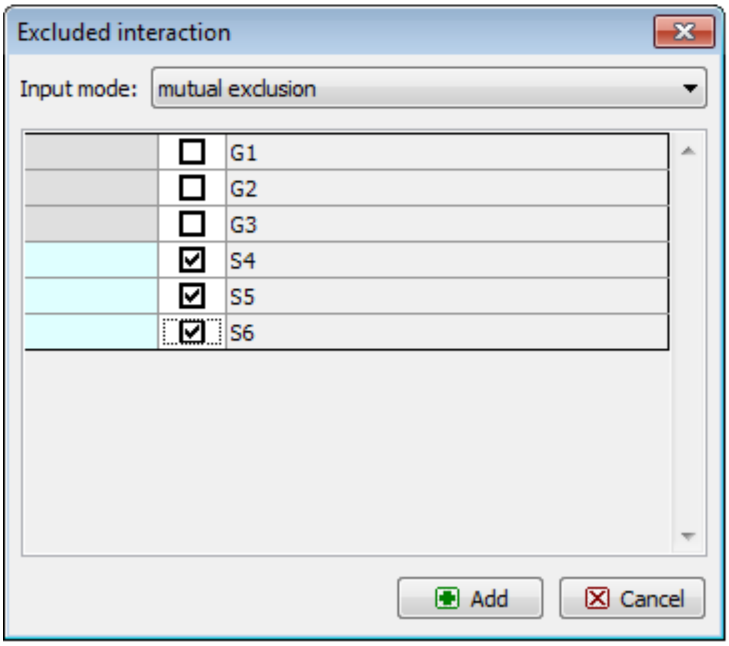
The automatic generation of combination is run in the "**Generator of combinations**" dialog box. The dialog box contains three tables. In the first, the load cases which act simultaneously are combined. In the second, we can set mutual exclusion of some load cases in one combination. The last table contains a list of variable loads which shall be considered main. In our example, we need to exclude simultaneous action of the defined snow load cases; hence we create a new exclusion group by clicking the "**Add**" button in the table "**Excluded interaction of load cases**".




*"Generator of combinations" dialog box.*

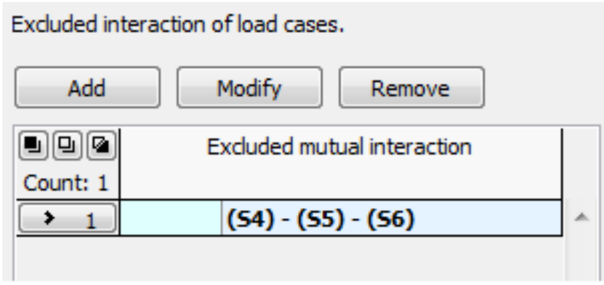
In the "**Excluded interaction**" dialog box we select the load cases *S4*, *S5* and *S6* and confirm selection by clicking the "**Add**" button.




*Defining mutual exclusion of load cases*

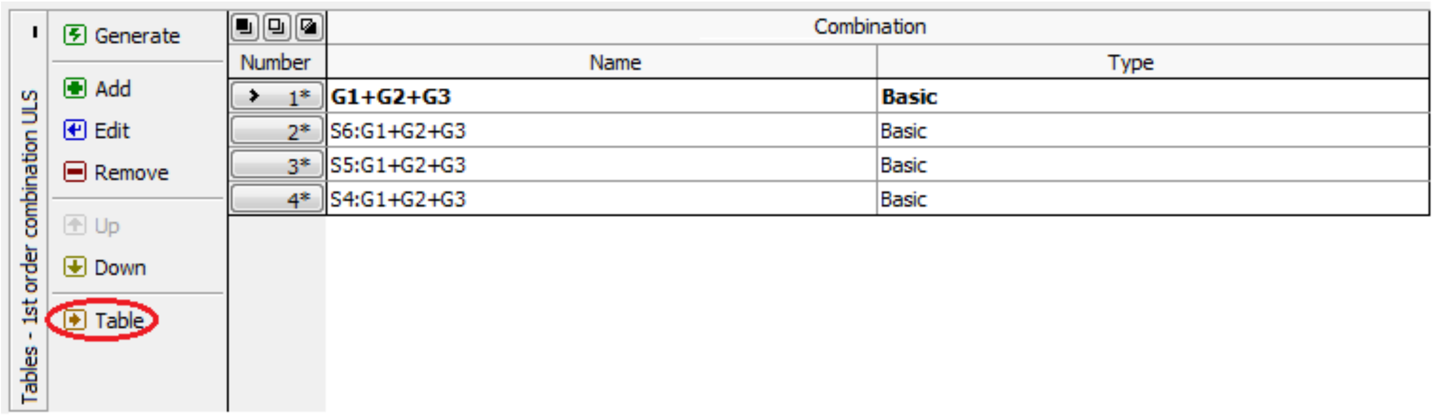
After closing the dialog box a new group of mutually exclusive load cases appears in the relevant table. Thus it is guaranteed that only one of these load cases can appear in one combination.




*Added group of mutually exclusive load cases*

Once we have finished entering data, we can create the combinations by clicking the "**Generate**" button. A list of the generated combinations appears in the table in the bottom part of the screen; we can add, edit or erase the combinations as necessary. We can also display the list in a comprehensive table by clicking the "**Table**" button.




*Button for running "Table of combinations"*

In the "**Table of combinations**" we can check the generated combinations; for the active combination, a detail description including the used load factors is displayed in the bottom part of the table.

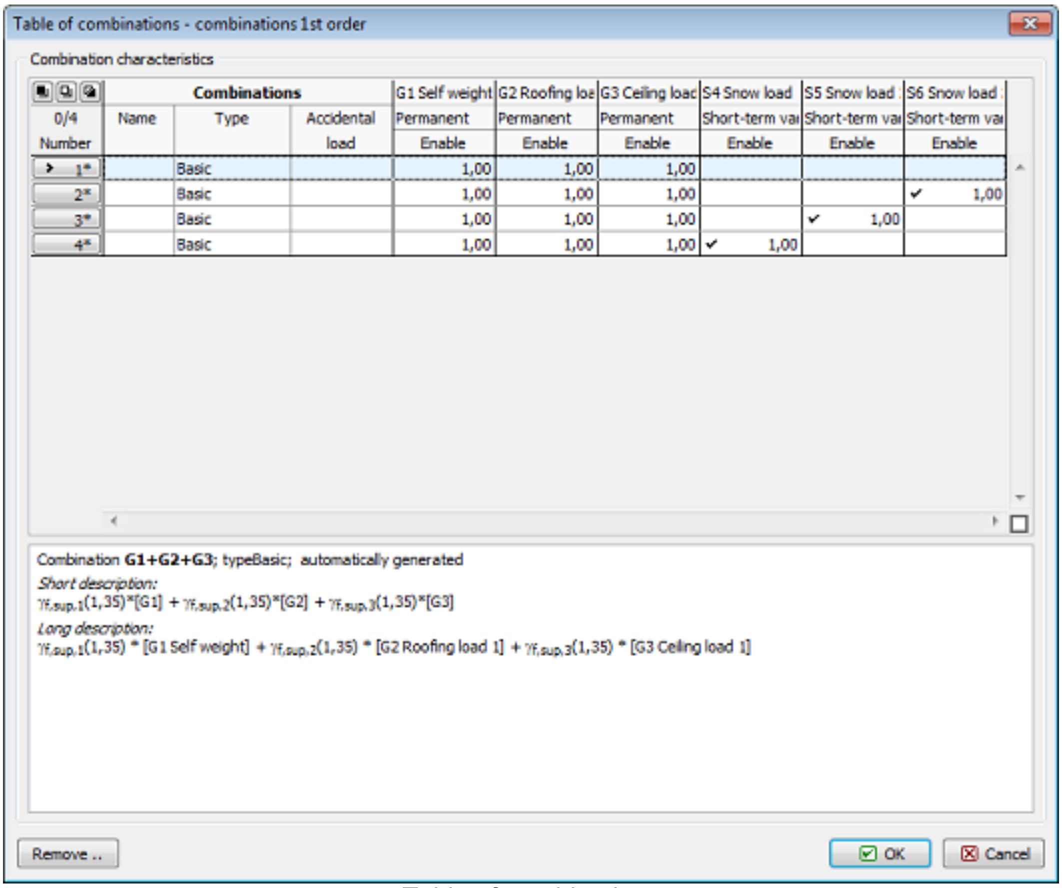
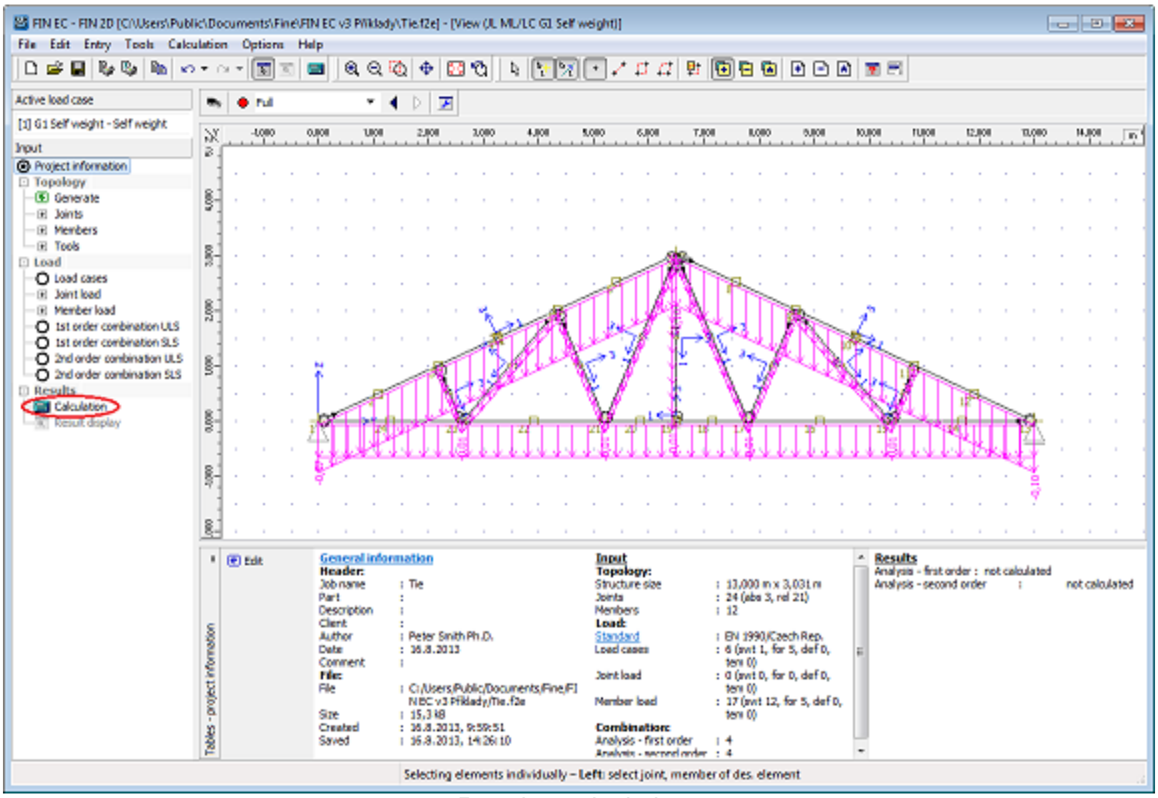



*Table of combinations*

Analogically we will generate the characteristic combinations switching to "**1st order combinations SLS**" in the control tree.

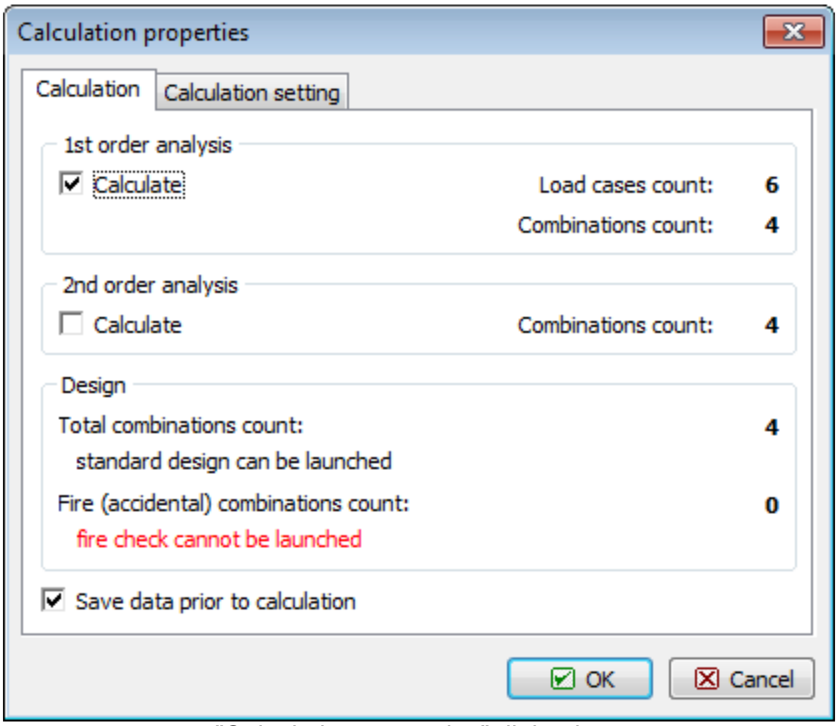
## Calculation and results display

Now we can finally proceed to running the "**Calculation**" of internal forces by clicking the eponymous button in the control tree.



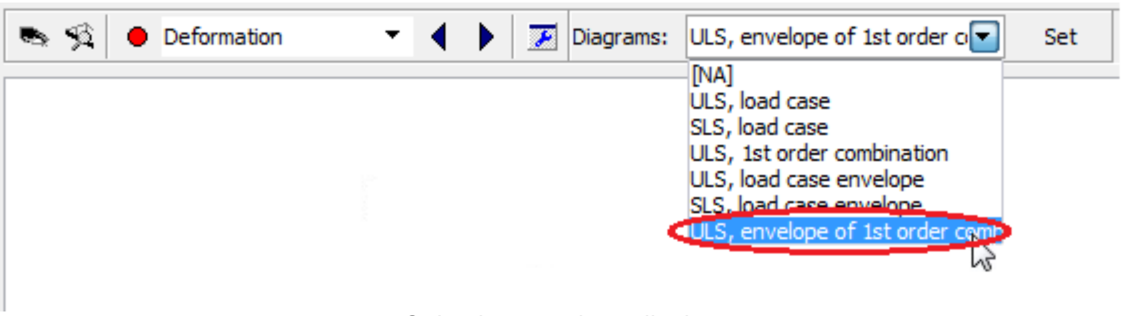

*Running calculations*

The "**Calculations properties**" dialog box appears; we can confirm the settings by clicking "**OK**" after which the calculation is executed and a window with information about the calculation process displayed. After clicking the "**Cancel**" button, program switches automatically to post-processor.



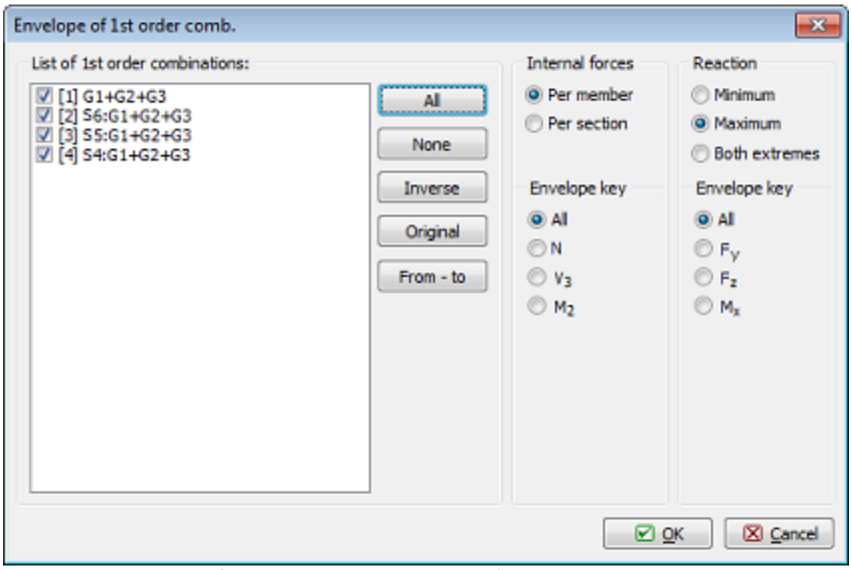

*"Calculation properties" dialog box.*

After finishing calculation, deformation resulting from the combination *No. 1* is displayed in the model space. The program offers, apart from many other functions, a variety of settings for results displaying, e.g. enables saving views into the "**Named selections**" and printing all views subsequently. In our example we will show how to display the envelope of the bending moments. First we select "**ULS, envelope of 1st order combinations**" in the drop down list.




*Selecting envelope display*

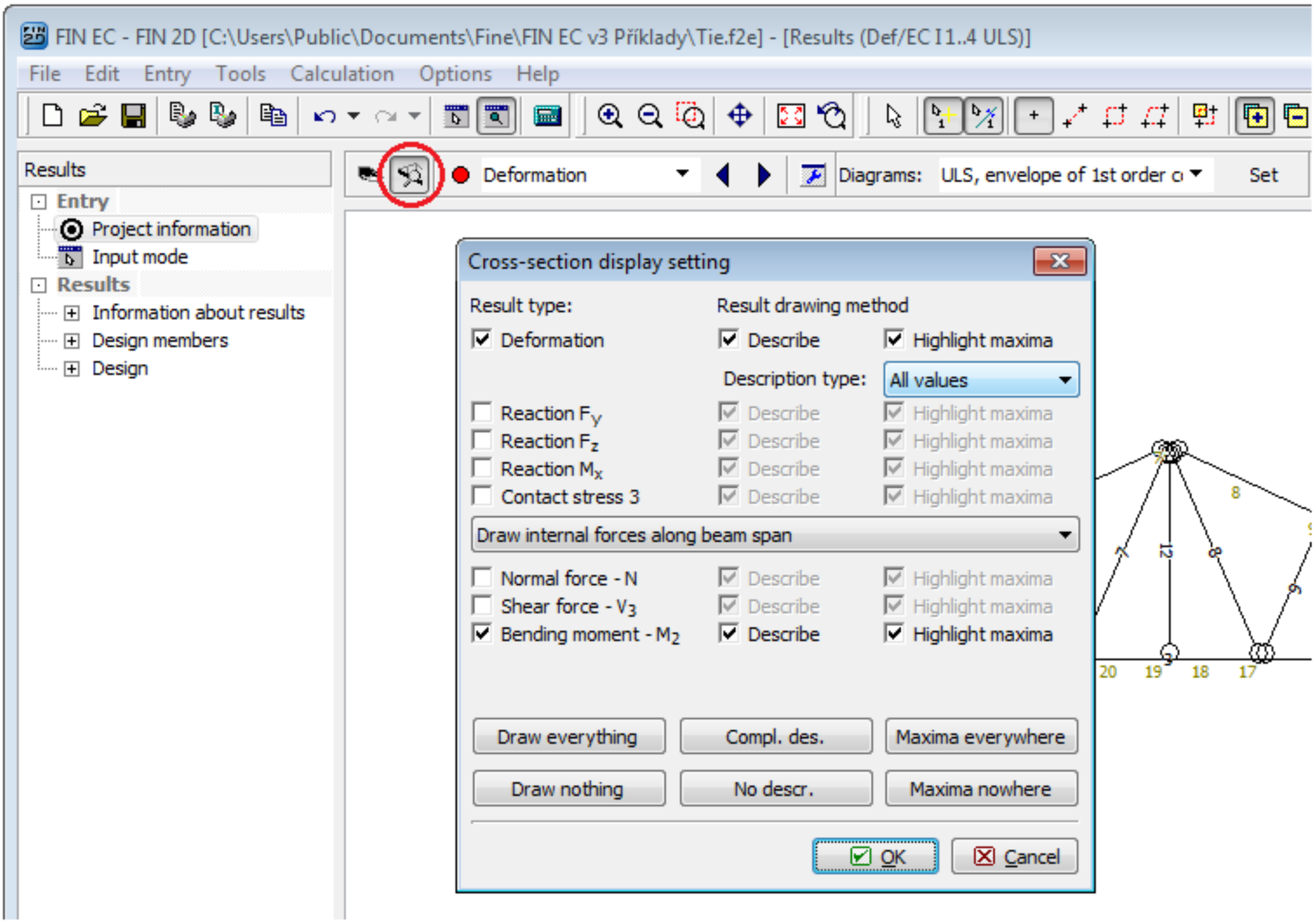
To define an envelope of all combinations we click the "**All**" button in the dialog box; thus all combinations in the list on the left-hand side are automatically selected.




*Selecting combinations for envelope*

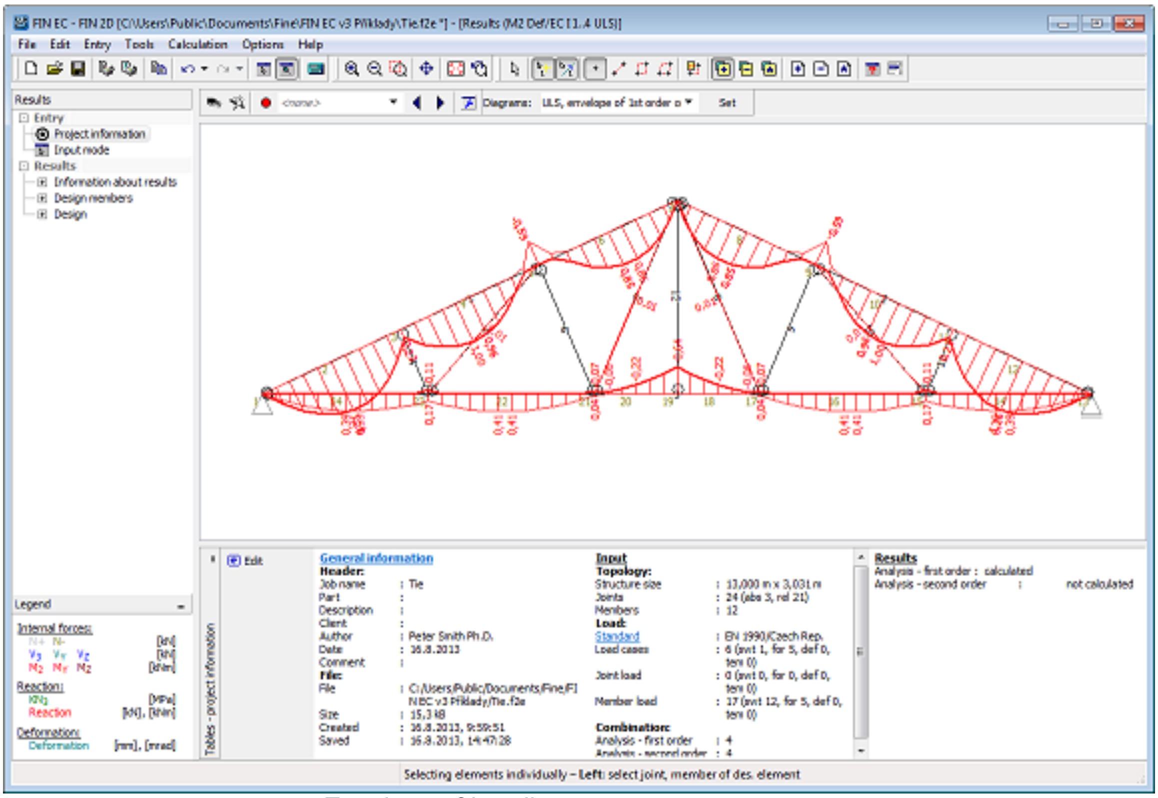
Then we run the "**Cross-section display settings**" dialog box and select "**Bending moment**".



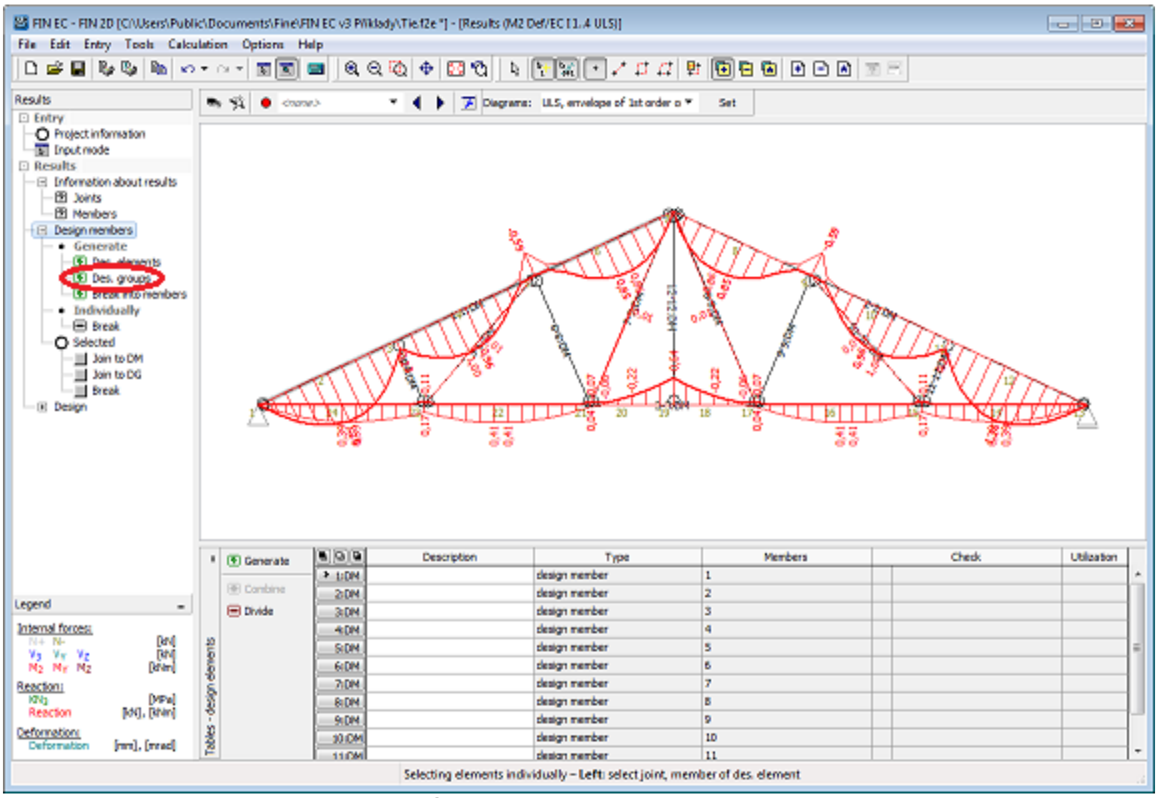

*"Cross-section display setting" dialog box.*

After confirming, the envelope of bending moments is displayed on the structure.



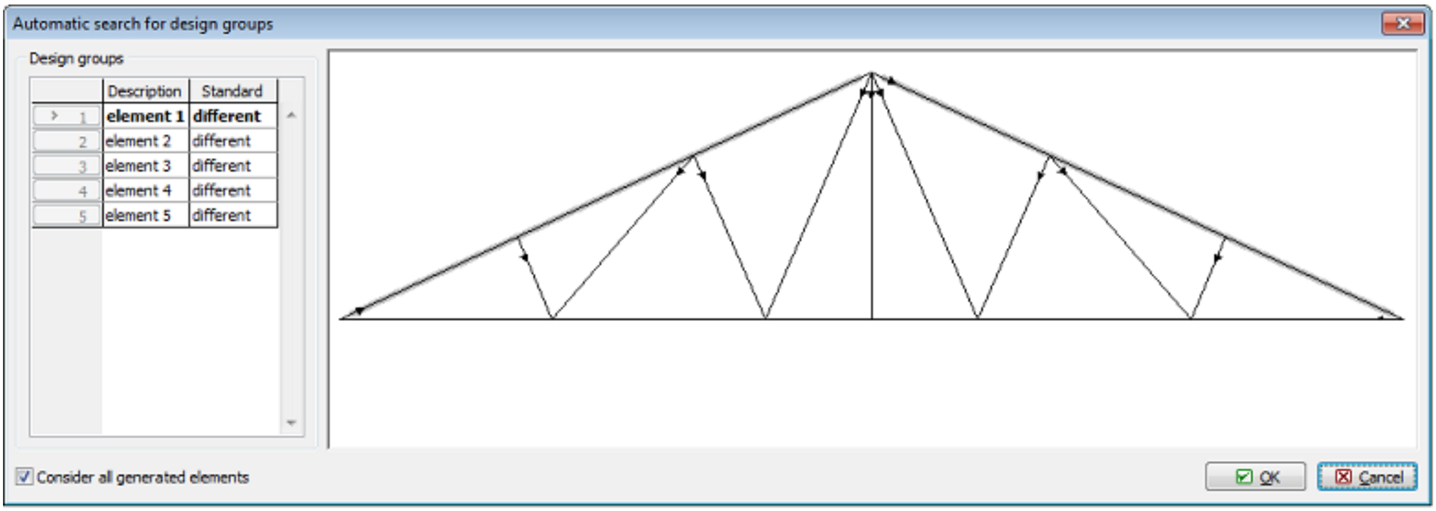

*Envelope of bending moments on structure.*

Now we proceed to design checks of the structure's cross-sections. First we switch to the "**Des. groups**" in the control tree. The structure consists of total of 11 elements, representing 11 design members. The program enables merging the members into design groups so that the assessment is as quick and straightforward as possible. Members merged into a design group are checked as one member; the loading is however considered on all members separately. This approach is beneficial for instance in case when we need to check a number of concrete columns in which we want to have unified reinforcement – it is sufficient to merge all of them into one design group. To create design groups automatically, we select "**Generate – Design Groups**" in the control tree.




*Generating design groups*

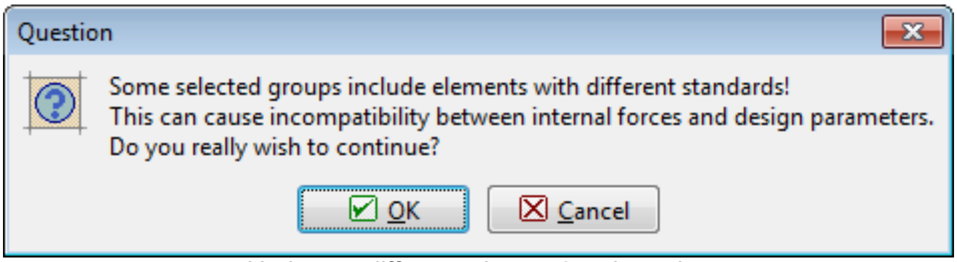
We can check in the dialog box which design groups were found by the program. If we want to create only some of the suggested groups, we can untick the "**Consider all generated elements**" box and then proceed only with selected design groups. In our example we will use all suggested groups, therefore we close the dialog box by clicking "**OK**".




*Suggested design groups*

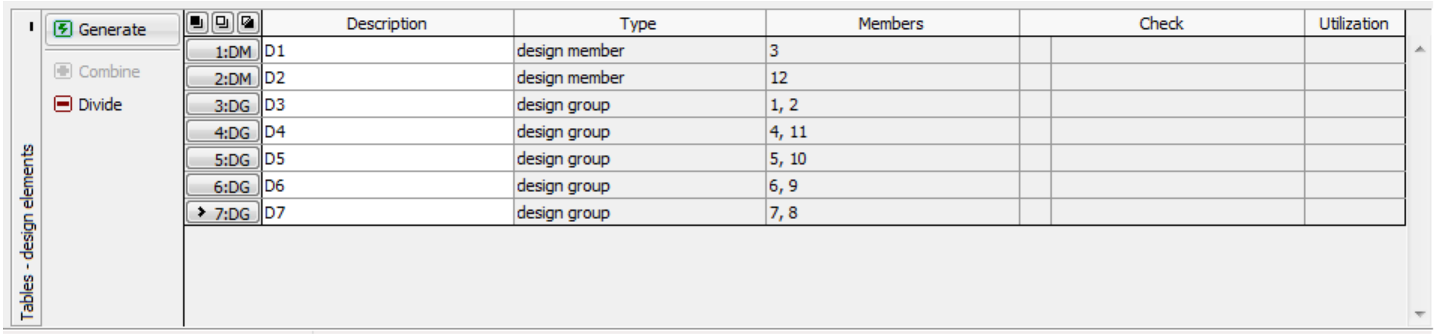
Because the orientation of particular elements vary we need to appreciate in case of which members this could cause difficulties – in our example it could be the upper chord. However; cross-sections and buckling parameters are constant along the length of the upper chords, therefore varying axes orientation should not influence the results of the assessment.




*Notice on different element's orientation*

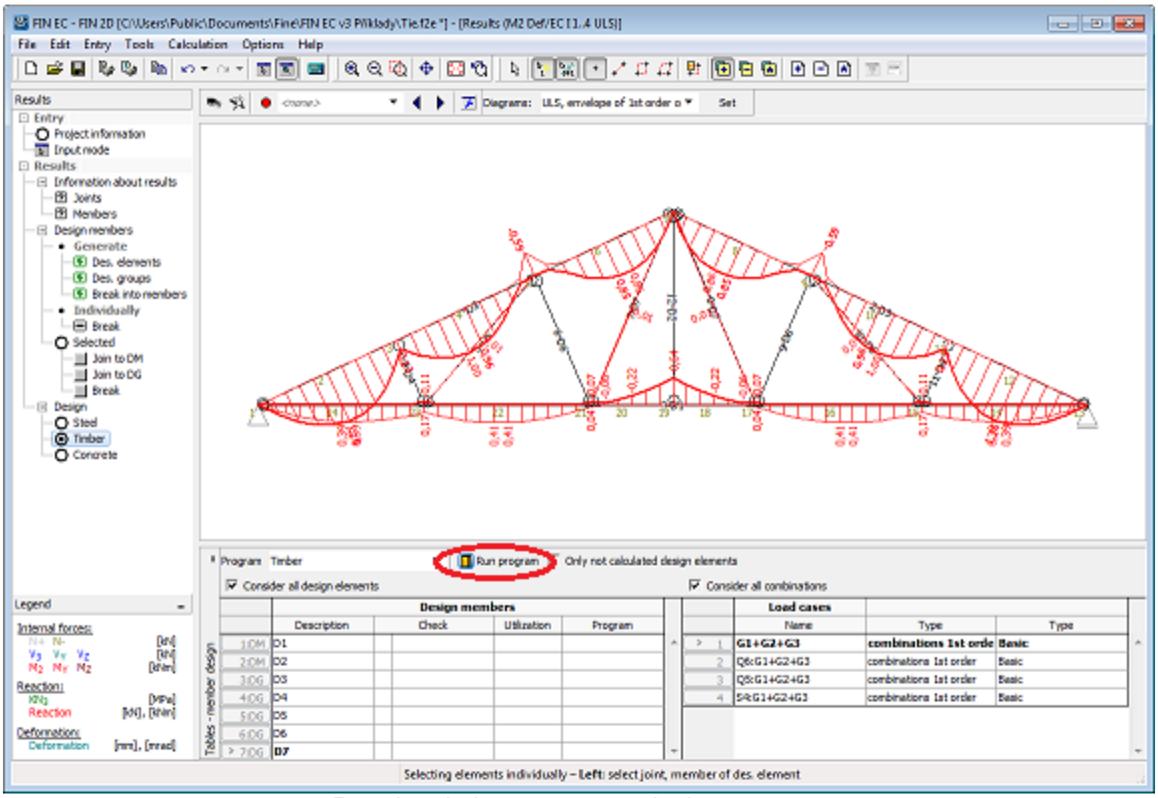
Individual elements have been merged into 5 design groups and one design member. We can name the members and groups in the table in the bottom part of the screen.



| | Description | Type | Members | Check | Utilization |
|---|---|---|---|---|---|
| 1:DM | D1 | design member | 3 | | |
| 2:DM | D2 | design member | 12 | | |
| 3:DG | D3 | design group | 1, 2 | | |
| 4:DG | D4 | design group | 4, 11 | | |
| 5:DG | D5 | design group | 5, 10 | | |
| 6:DG | D6 | design group | 6, 9 | | |
| 7:DG | D7 | design group | 7, 8 | | |

*Table with entered names of design members and groups*

Now we can proceed to the design itself. We select "**Design**" and "**Timber**" in the control tree and run the timber structures design program by clicking the "**Run program**" button.

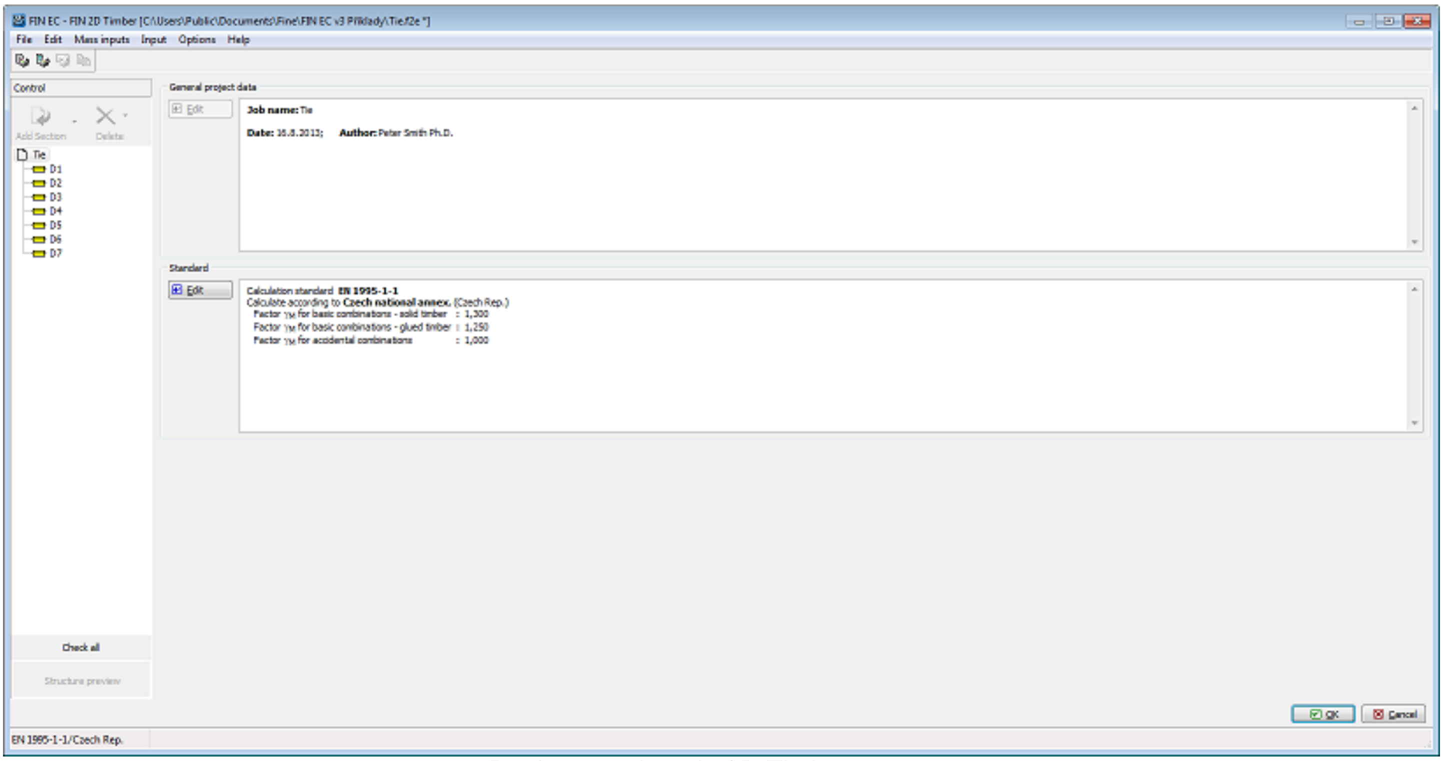



*Running timber structures design program*

## Members design

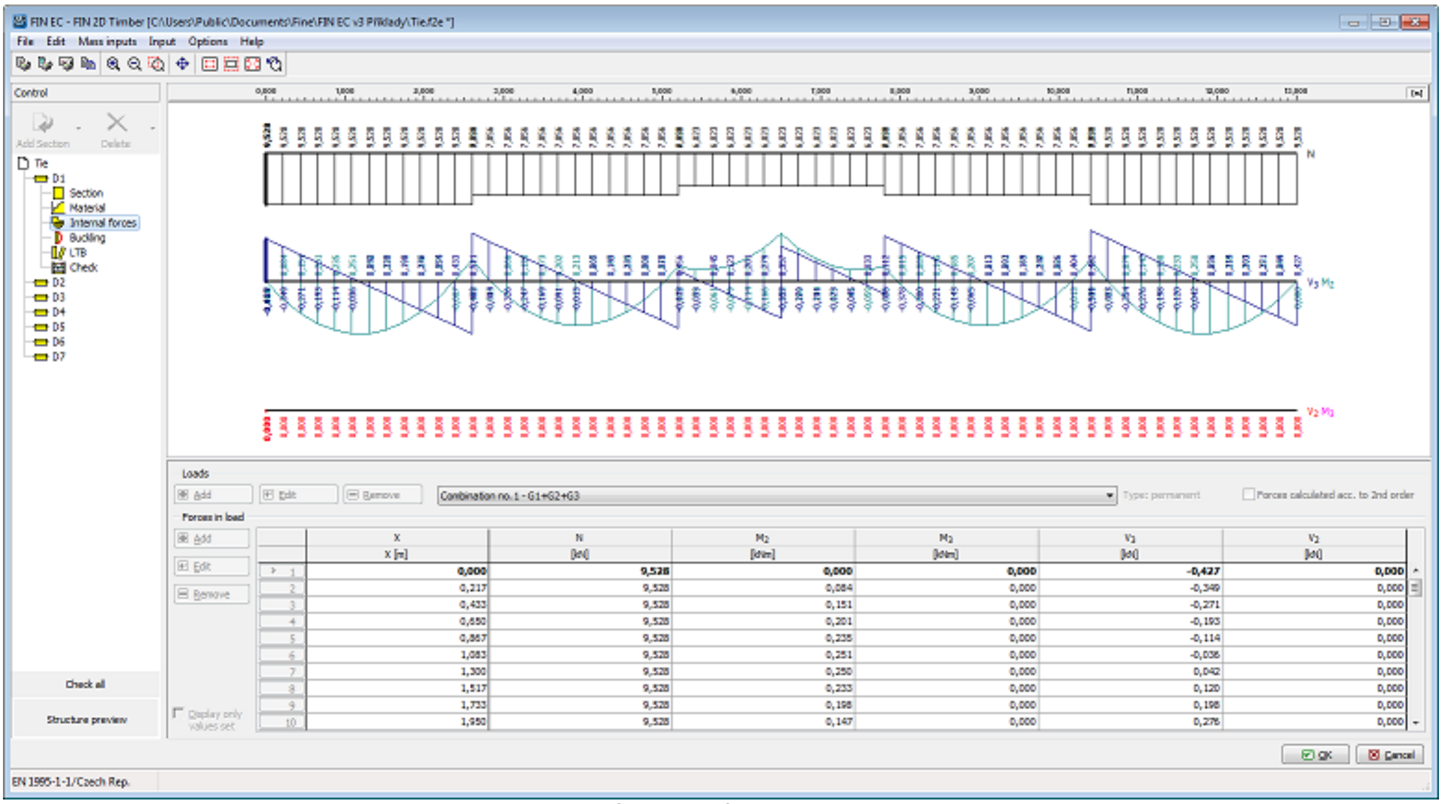
Program 2D Timber is run with all design members and groups automatically imported.



*Design members in 2D Timber program*

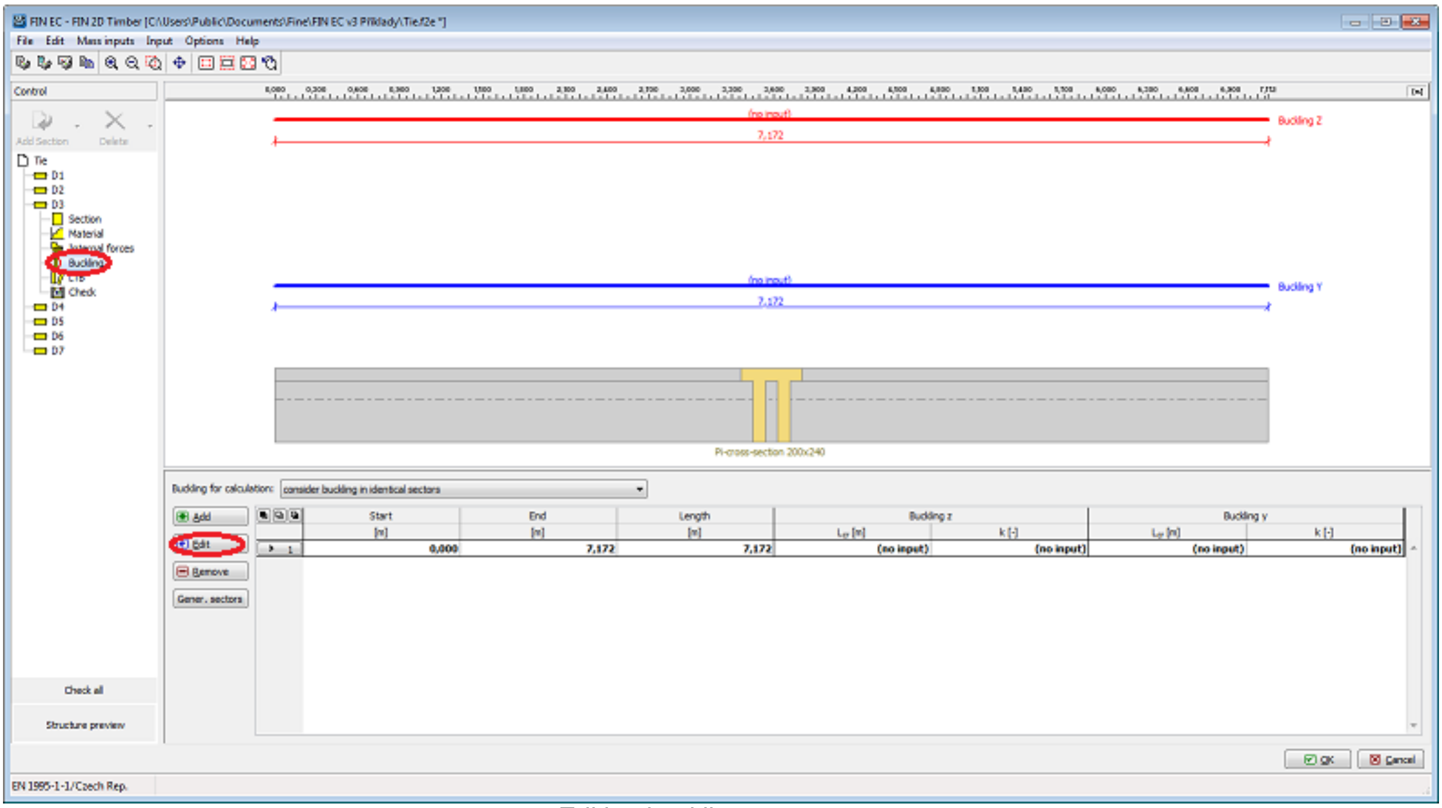
All data regarding geometry (members' lengths, cross-sections etc.) and loading (internal forces distribution for all combinations) have been imported into the program. The data can be checked in the relevant sections of the control tree. We can confirm position of a selected member in the structure by clicking the "**Structure preview**" button.



*Distribution of internal forces in bottom chord*

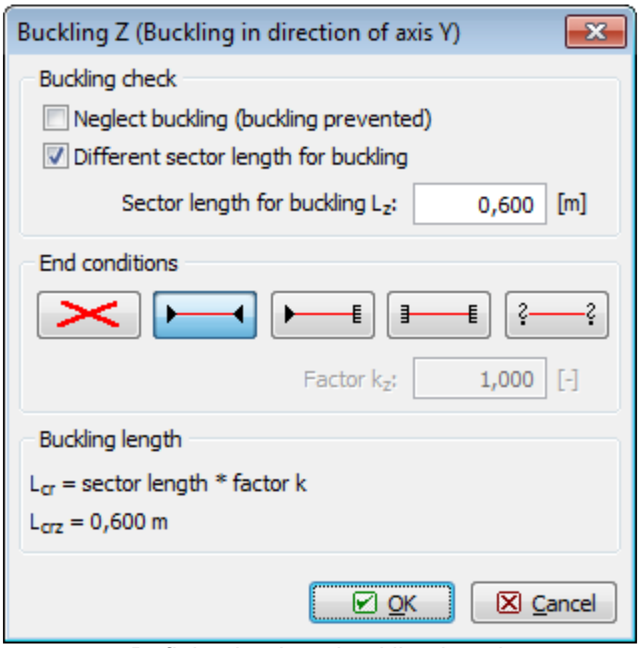
Proceeding to members design, we will demonstrate the procedure on the *upper chord* i.e. "**D1**" design group. The upper chord is subject to compression; therefore it is necessary to define the buckling parameters. In our example we assume that out-of-plane buckling is restrained by purlins at *0.6m* centres. We switch to "**Buckling**" in the upper chords section of the control tree and run the buckling parameters dialog box by clicking the "**Edit**" button.




*Editing buckling parameters*

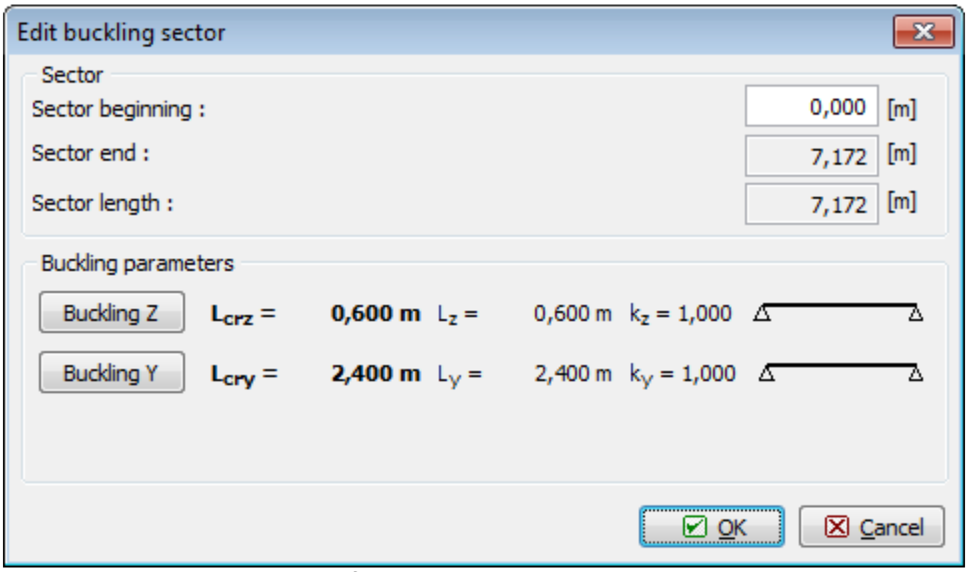
In the "**Edit buckling sector**" dialog box we can define parameters for out-of-plane buckling ("**Buckling Z**") and in-plane buckling ("**Buckling Y**"). For out-of-plane buckling we define simple end conditions and sector length for buckling $L_z = 0.6m$ ; for in-plane buckling we define simple end conditions as well with sector length 2.4 m. The parameters are defined in the "**Buckling Z**" dialog box.




*Defining in-plane buckling length*

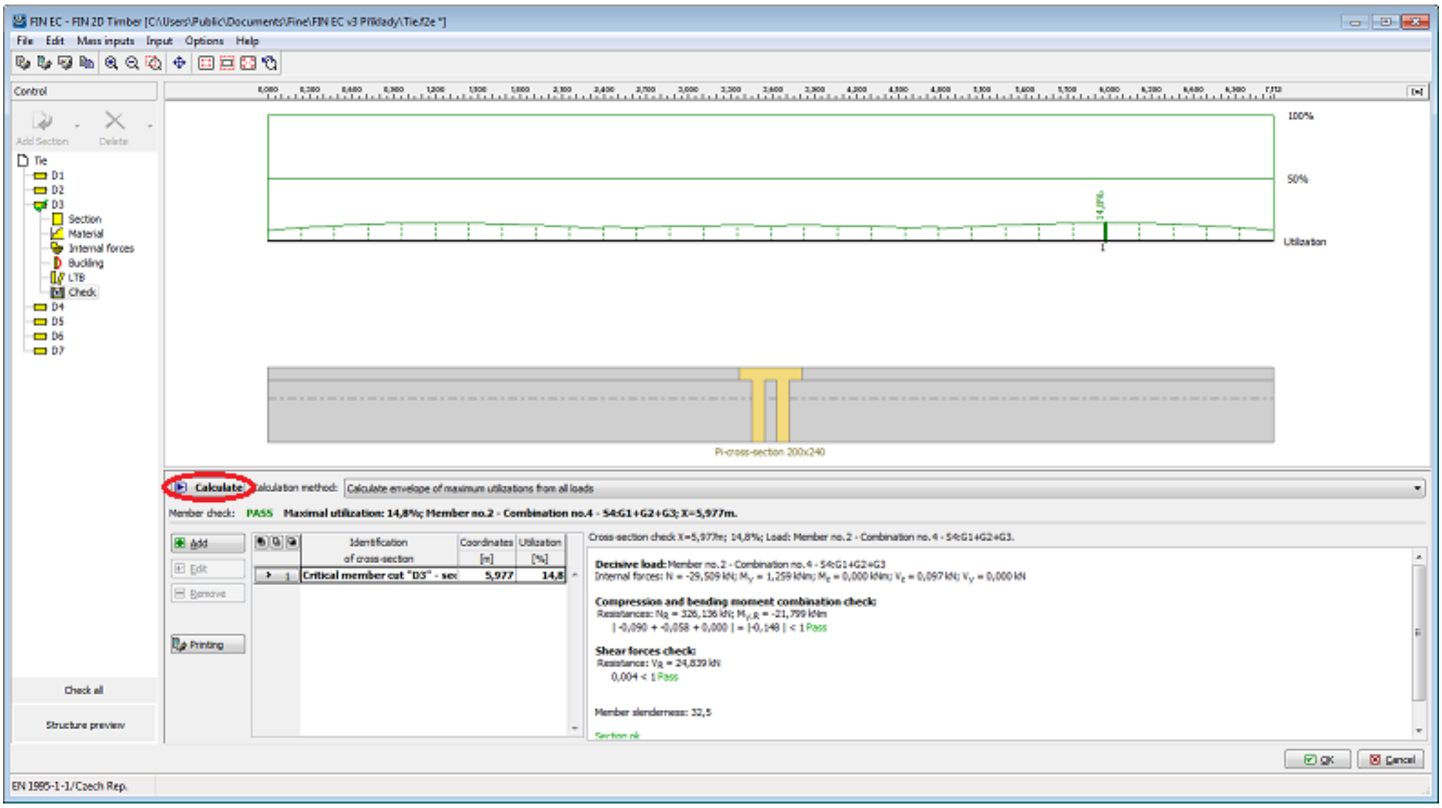
When the parameters are defined for both directions we can close the dialog box by clicking "**OK**".




*Defined buckling parameters*

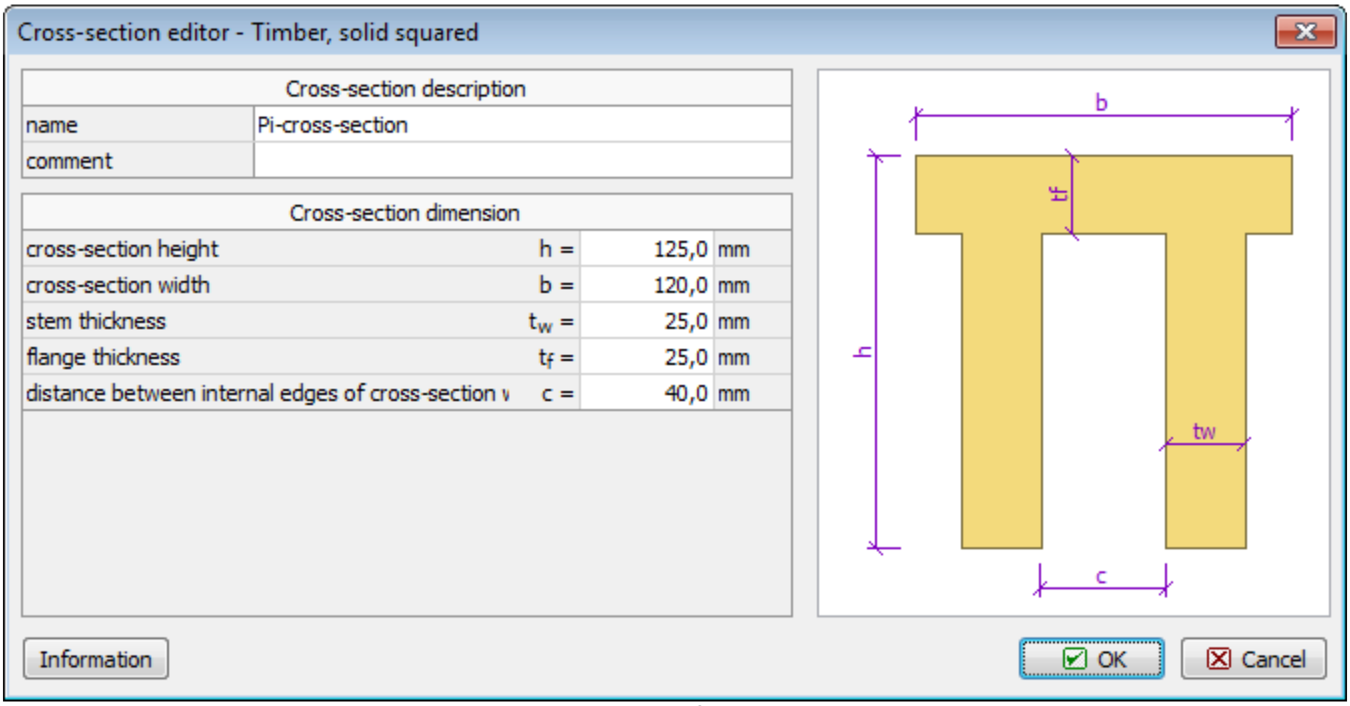
We switch to the "**Check**" section of the control tree and run the calculations by clicking the "**Calculate**" button. In the model space, the cross-section utilization curve is displayed along the length of the member; the critical section with the highest utilization is checked in the bottom right corner of the screen. If it is necessary to display detail checks in other sections, these can be added using the table in the bottom part of the screen or simply by double-clicking in the selected location of the utilization diagram in the model space.



*Members check*

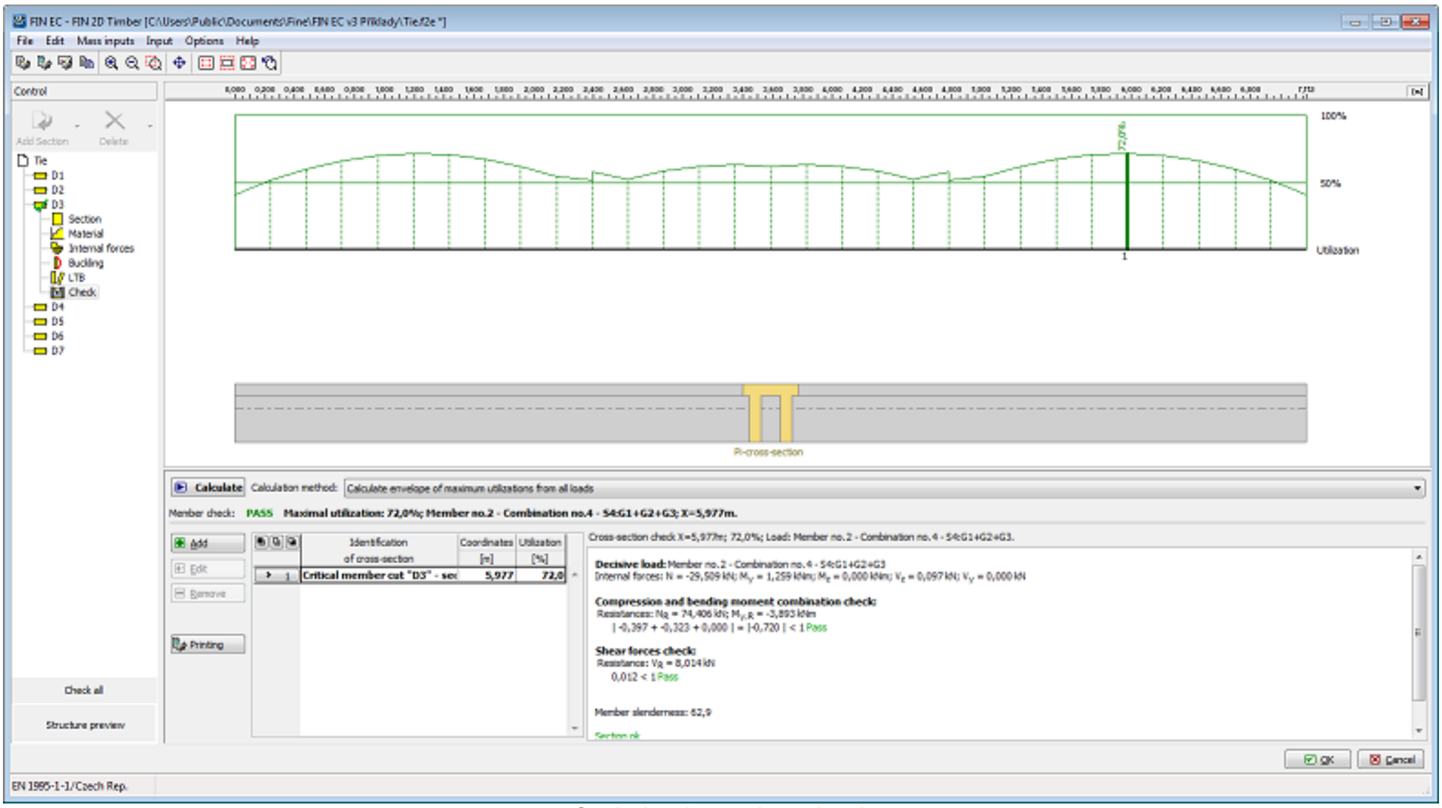
Because the maximum utilization of the member is very low, we can reduce the size of the cross-section. We switch to the "**Section**" part of the control tree and run the "**Cross-section editor**" dialog box where we can edit the cross-section geometry.



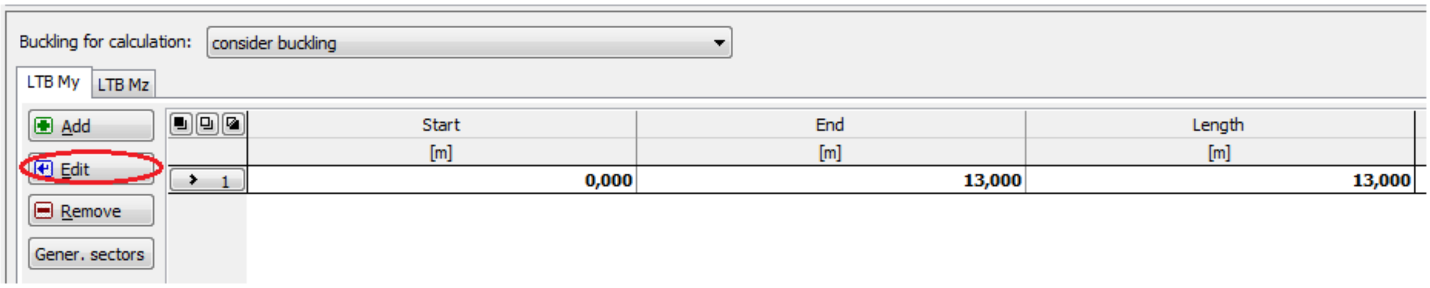

*Edited dimensions of the upper chord*

After editing the geometry as shown we return to the "**Check**" part of the control tree and re-calculate the structure, obtaining more acceptable check results.



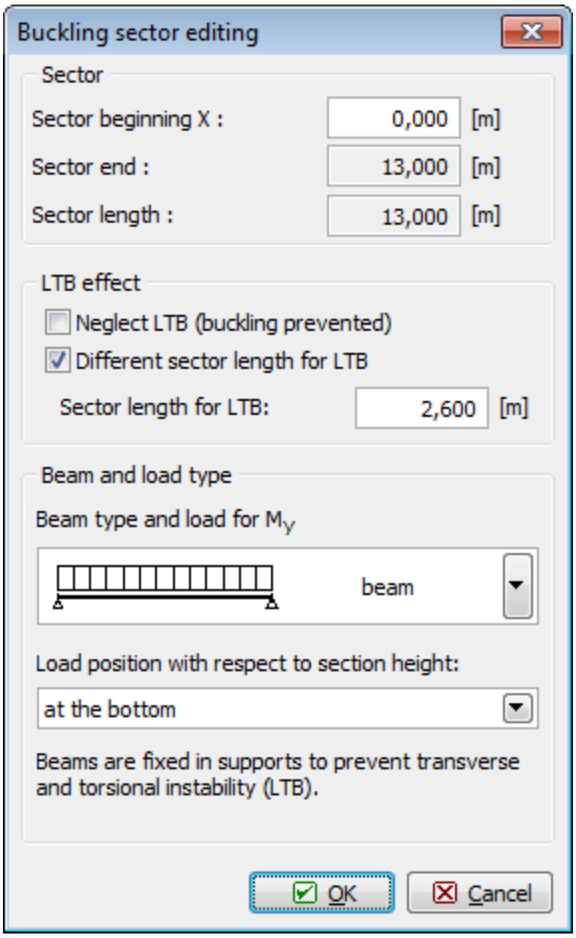

*Optimized member check*

Now we proceed with the "**bottom chord**". Because the bottom chord is in tension it is not necessary to define buckling parameters. However, we need to define lateral torsional buckling parameters as lateral and torsional stability should be checked in members subject to combination of tension and bending. We switch to the "**LTB**" section of the control tree and analogically to the buckling of the upper chord we define the buckling parameters for bending moment $M_y$ by clicking the "**Edit**" button.




*Editing buckling parameters*

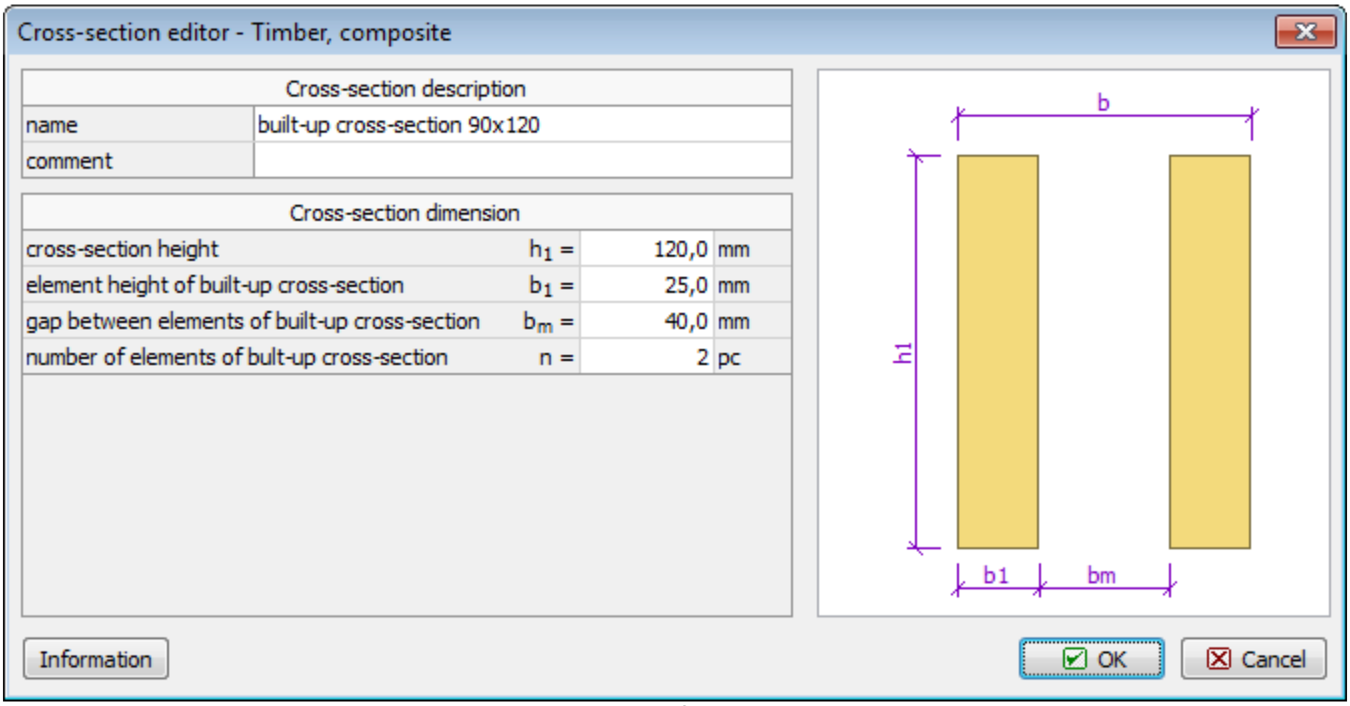
In the "**Buckling sector editing**" dialog box we define the LTB effective length and select appropriate beam and load type. We close the dialog box by clicking "**OK**".




*Buckling parameters*

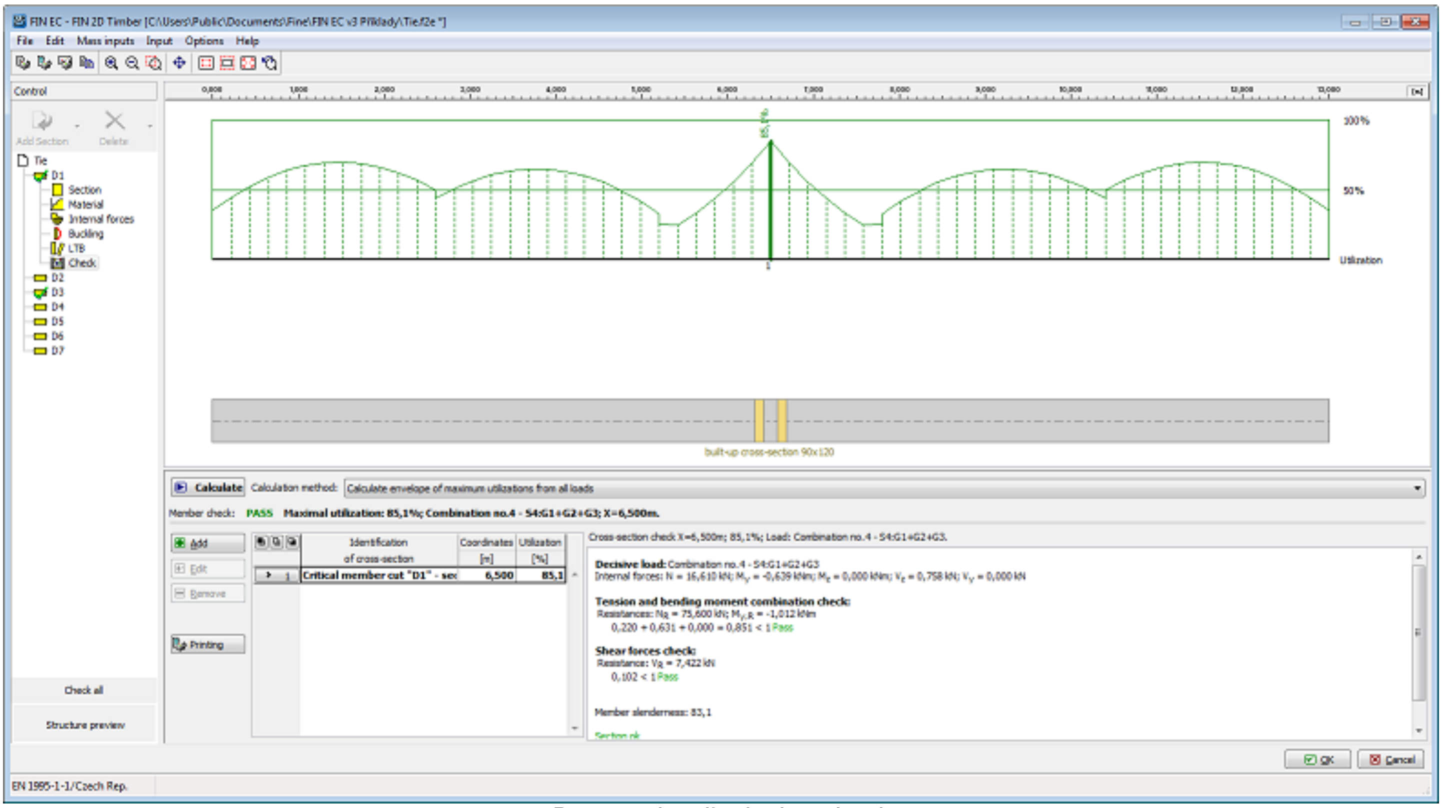
Then we switch to the "**Check**" section of the control tree again and carry out the members design check. Also for this member the utilization is too low therefore we edit the cross-section again.




*Editing geometry of the bottom chord*

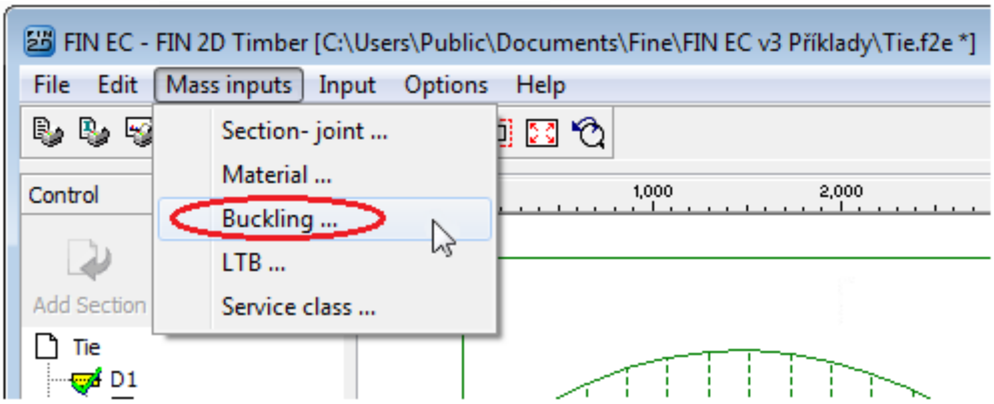
We carry out the design check again confirming more economic design of the member.




*Bottom chord's design check*

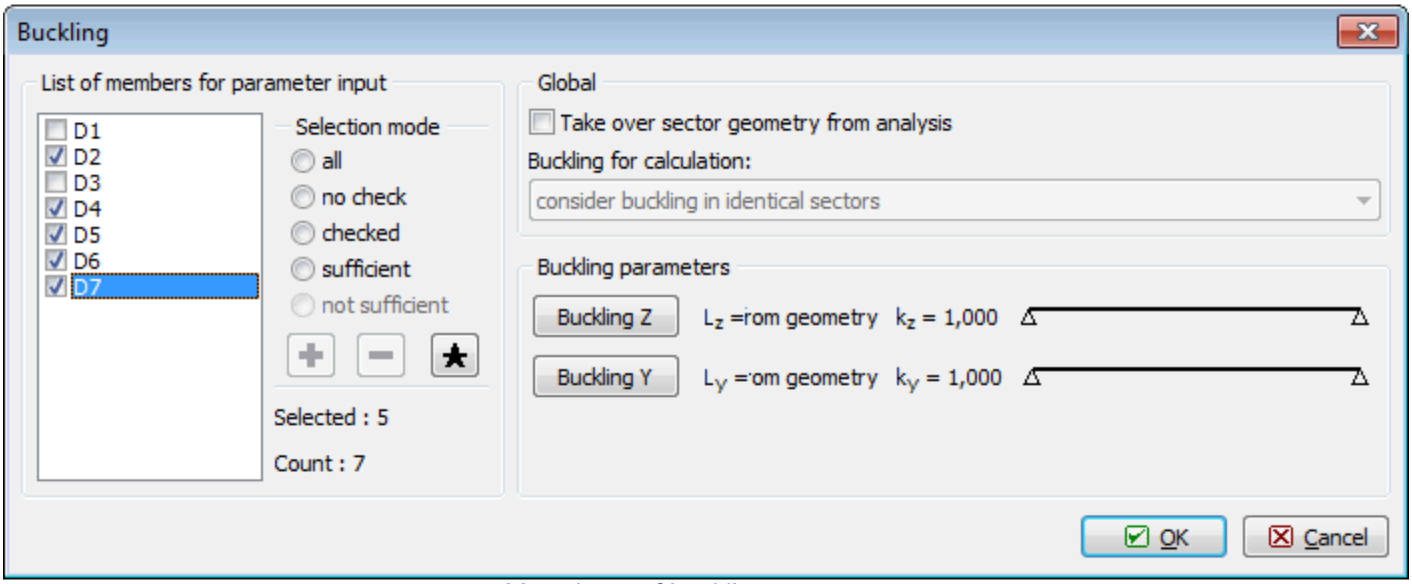
Finally, the diagonals remain to be checked. Because the diagonals' properties are almost identical, we can define the calculation parameters for all of them together. This can be done using a function in the "**Mass input**" part of the main menu. First we define the buckling parameters.




*Mass input of buckling properties*

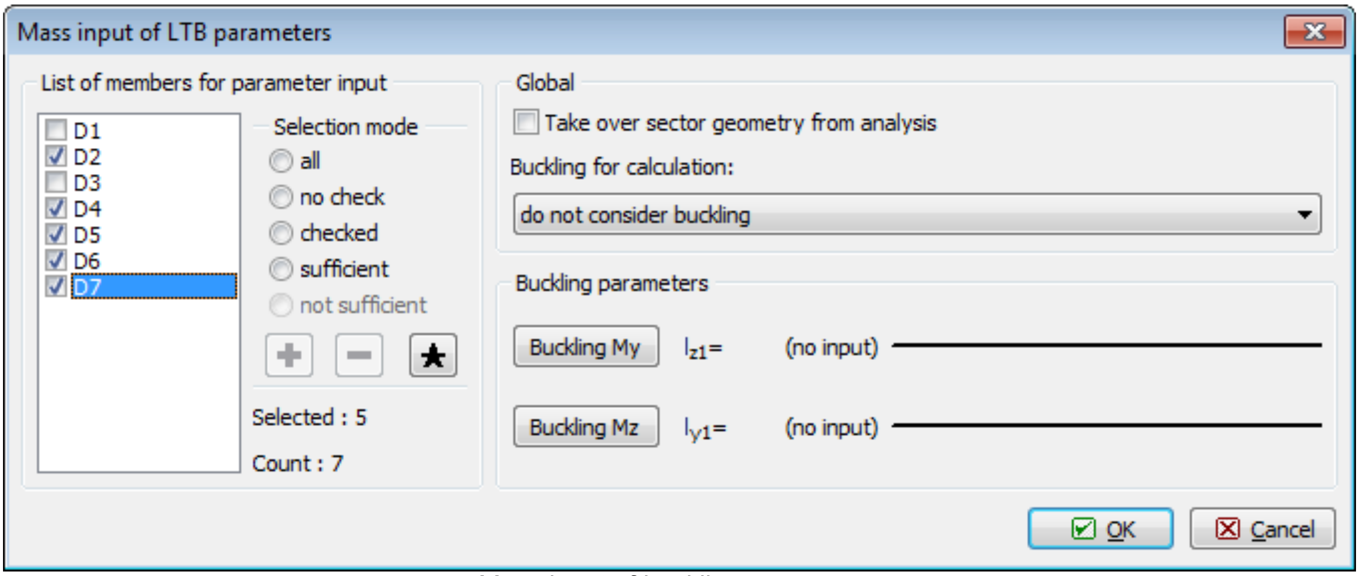
Even though it would be sufficient to define the buckling parameters for the compressive members only, it is easier to assign them to all diagonals. Hence we select members D1 to D4 on the left-hand side. In the right part we tick the "**Take over sector geometry from analysis**" so that the program will use the actual lengths of the members as buckling lengths. Finally we define simple supports at both ends for both y and z directions. After clicking "**OK**" the entered parameters are assigned to all diagonals.




*Mass input of buckling parameters*

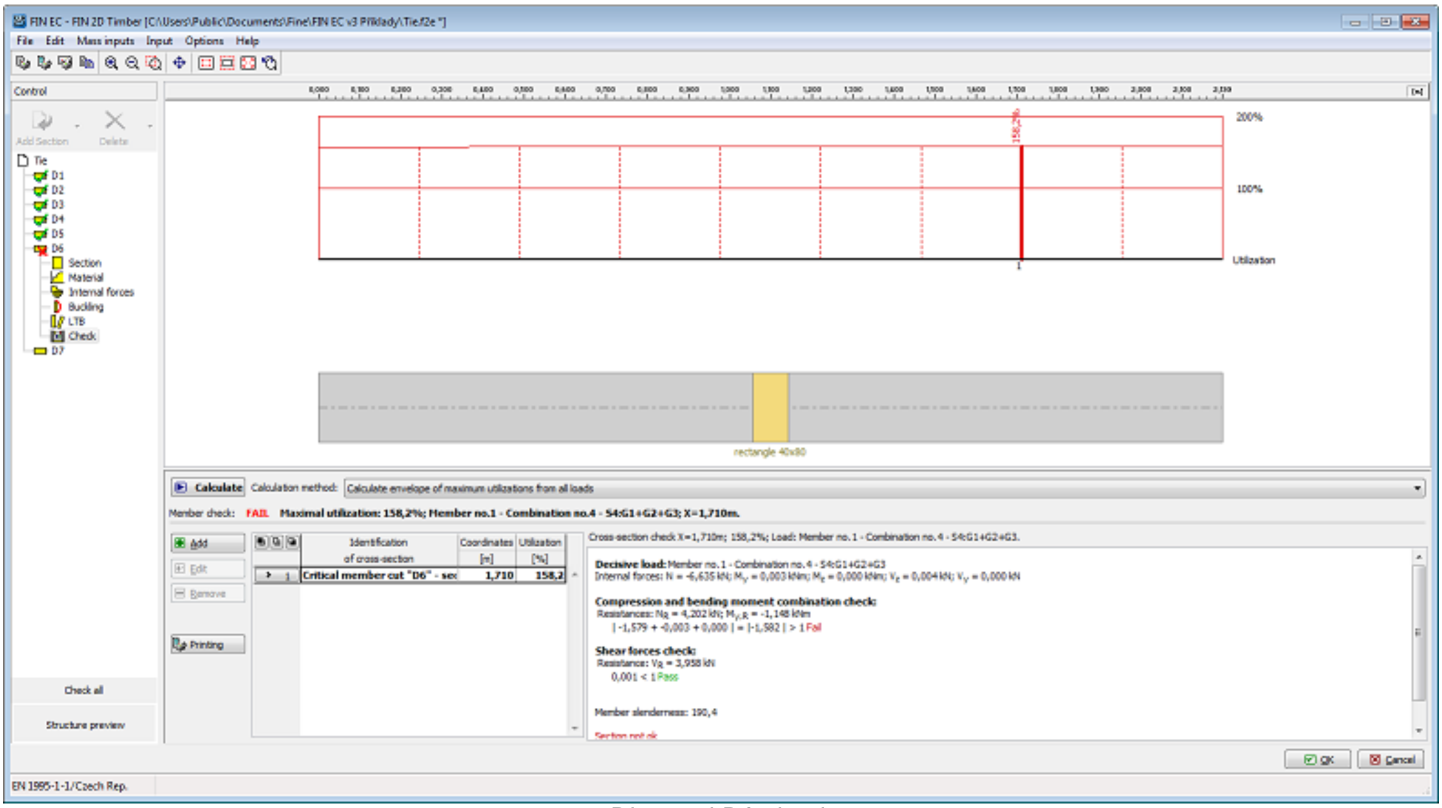
We continue with defining lateral torsional buckling parameters. Diagonals are generally only subject to axial forces therefore the LTB checks are not required; however, due to their self weight, small bending moments can occur. In such cases, the program demands lateral torsional buckling checks. In the "**Mass input**" section we select "**LTB**"; in the left part of the dialog box we again select all diagonals and from drop down list in the right part we select "**do not consider buckling**". Thus the influence of lateral torsional buckling will not be considered in the design checks of the selected members.




*Mass input of buckling parameters*

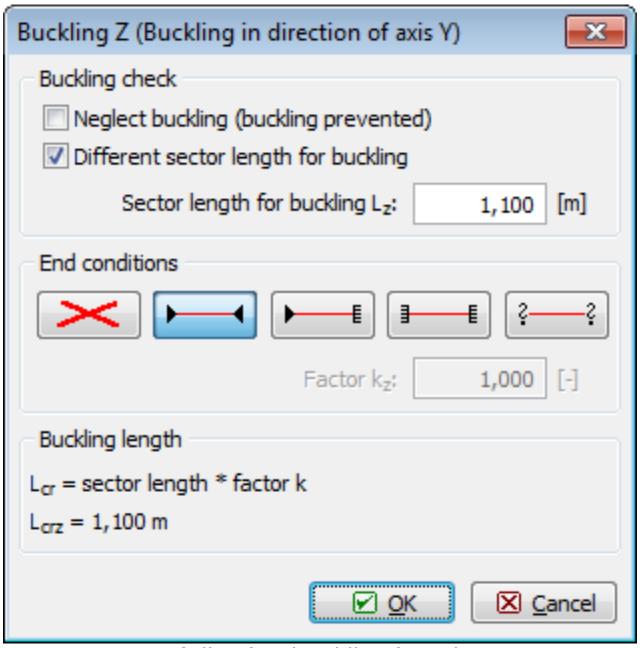
Now we can carry out the design checks for all diagonals. The diagonal D6 does not pass the buckling check, however we can increase its capacity by reducing its buckling length.



*Diagonal D6 check*

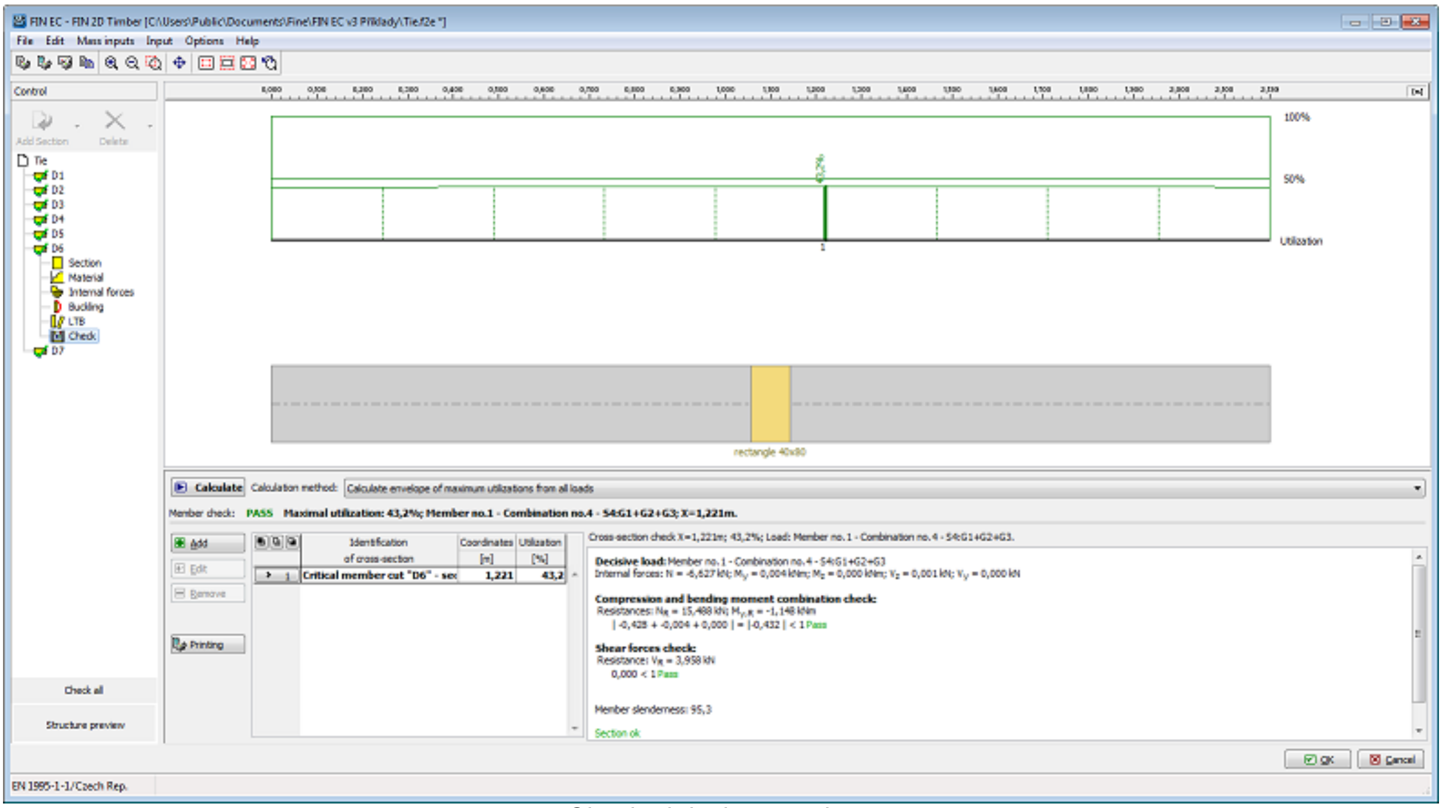
Therefore we design a longitudinal stiffener in the diagonal's centre which will reduce its buckling length to half. In the "**Buckling**" section of the control tree we adjust the buckling length in the "**Buckling Z**" dialog box.



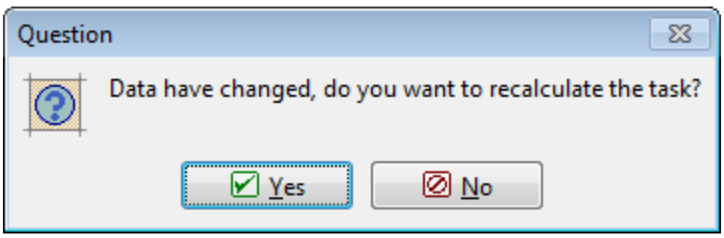

*Adjusting buckling length*

After re-calculating, the diagonal passes the design checks.




*Checked design members*

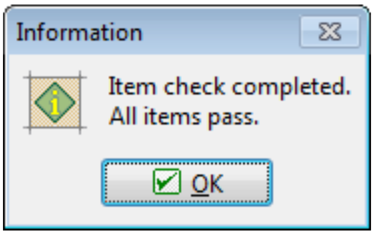
Now all design members and groups in the structure are checked, therefore we can exit the design module by clicking "**OK**". Program 2D has recognized that some members were changed. Because the stiffness of individual members changed and thus different internal forces can act on some of the them, it is necessary to re-calculate the structure. The program automatically offers this option. If we select "**No**", the program will erase results and return to the pre-processor. In our example, we select "**Yes**" after which the structure is re-calculated.



*Question on re-calculating structure*

After finishing the calculation, the program updates distribution of internal forces and deformations and erases the results in the design module. Therefore we run the design module again and make use of the command "**Check All**" located in the bottom part of the control tree. We do not have to re-define all parameters as they are saved after previous definition. The module will carry out design checks for all members and inform us about the results.



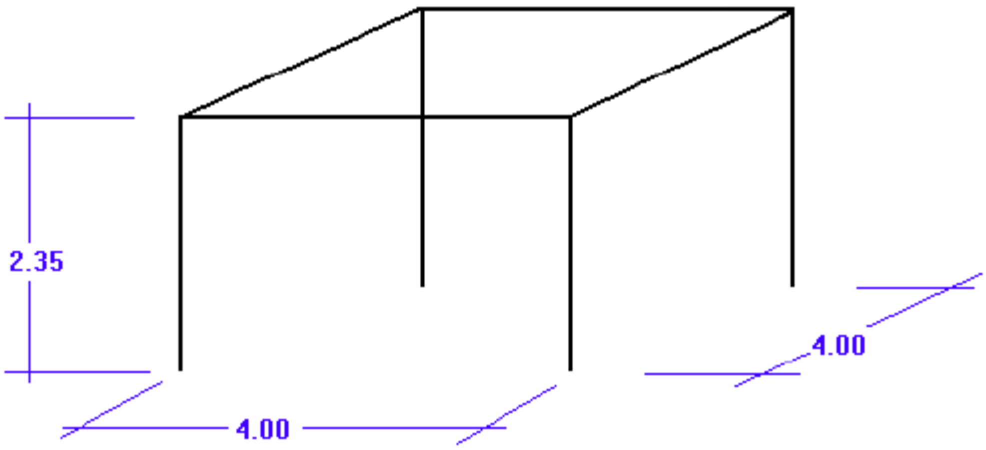

*Members' design checks results*

After returning to the program 2D, members which passed the checks are marked in green and the members which did not in red. In our case, all members are marked in green; therefore the structure's design is completed.

# 3D structure

## Introduction

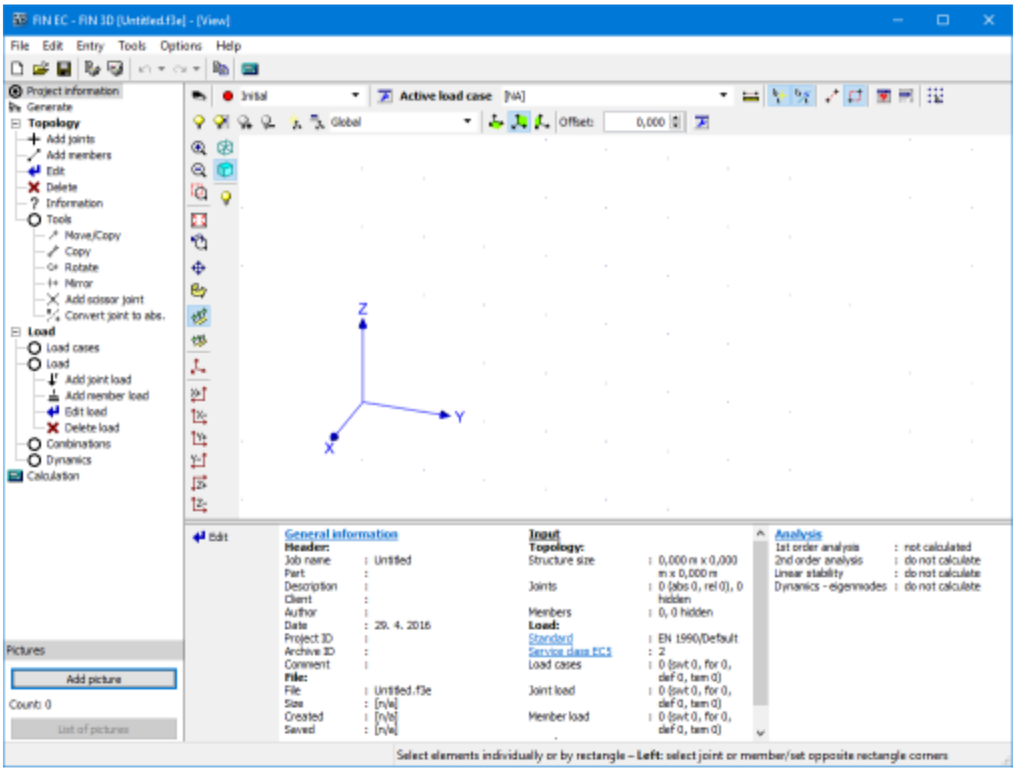
This tutorial shows the input and verification of the structure shown in the picture below. Columns are made of RHS profiles, beams are *I*-profiles. Material class *Fe 360* is used. Beams are loaded by linear load *18 kN/m*. Two columns are loaded by trapezoidal load (*12kN/m* at the top and *19kN/m* at the bottom). Supports are rigid. The programs "**Fin 3D**" and "**Steel**" will be used.




*Structural model*

The main window appears after running the program "**Fin 3D**".



*Main application window*

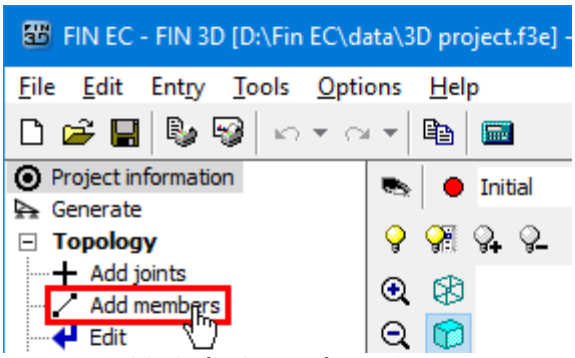
## Basic part of structure

The structure will be created with the help of manual input of joints and members. This input style shows most of particular procedures which appear during the work with the software. The input will be done using following steps:

- Input of a first frame (two columns and a beam)
- Input of load
- Copy of the existing frame
- Input of transverse beams and their load

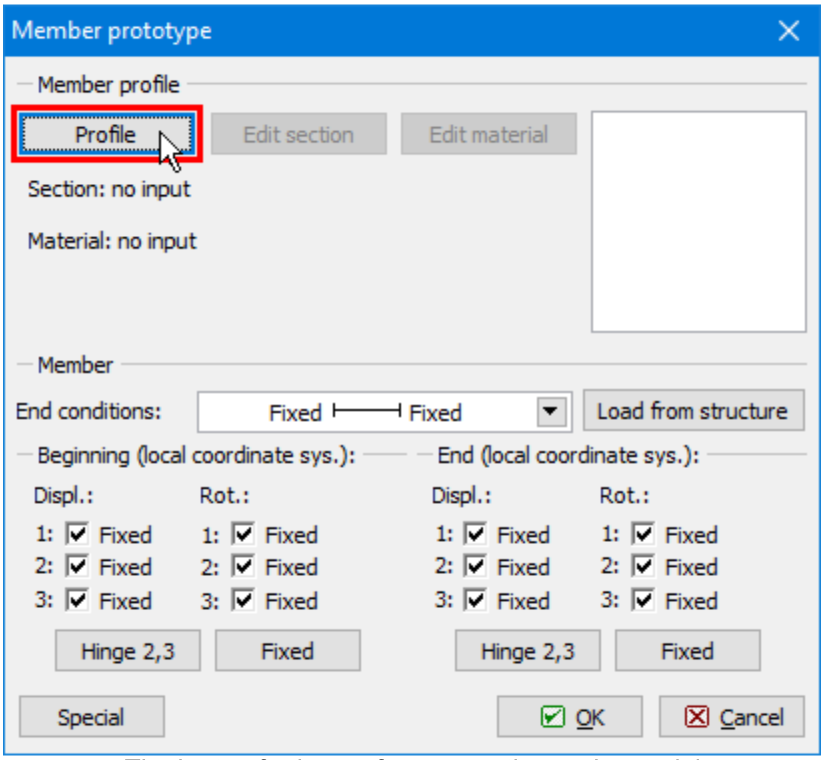
This procedure is not the easiest way, however, it shows variety of functions and tools included in the software. First, we save the project. The name (for example "**3D project**") appears in the heading of the window and is also copied into designing modules.

We select the mode "**Add members**".



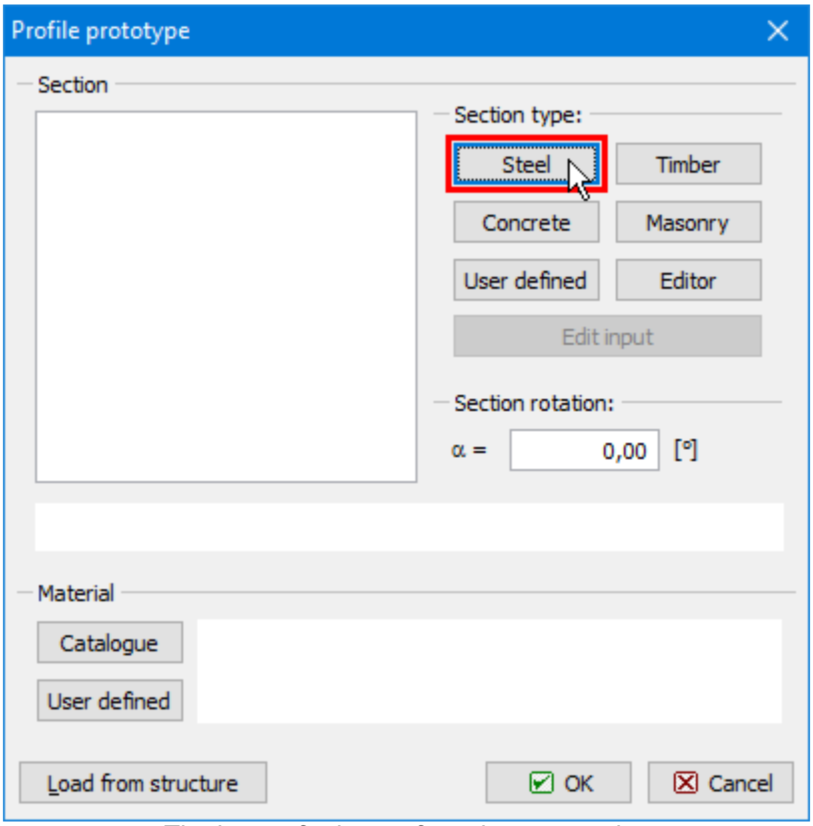

*Mode for input of members*

This mode launches the window "**Member prototype**". This window contains properties (cross-section, material, end conditions) that will be assigned to new members. We will continue with the help of the button "**Profile**".




*The button for input of cross-section and material*

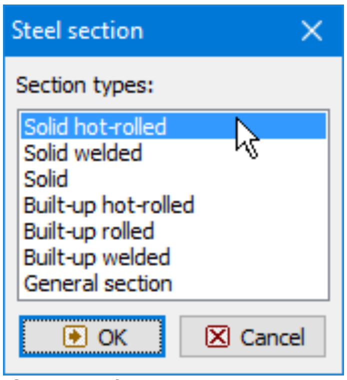
The window "**Profile prototype**" which contains options for input of cross-section and material. We will use the button "**Steel**".




*The button for input of steel cross-section*

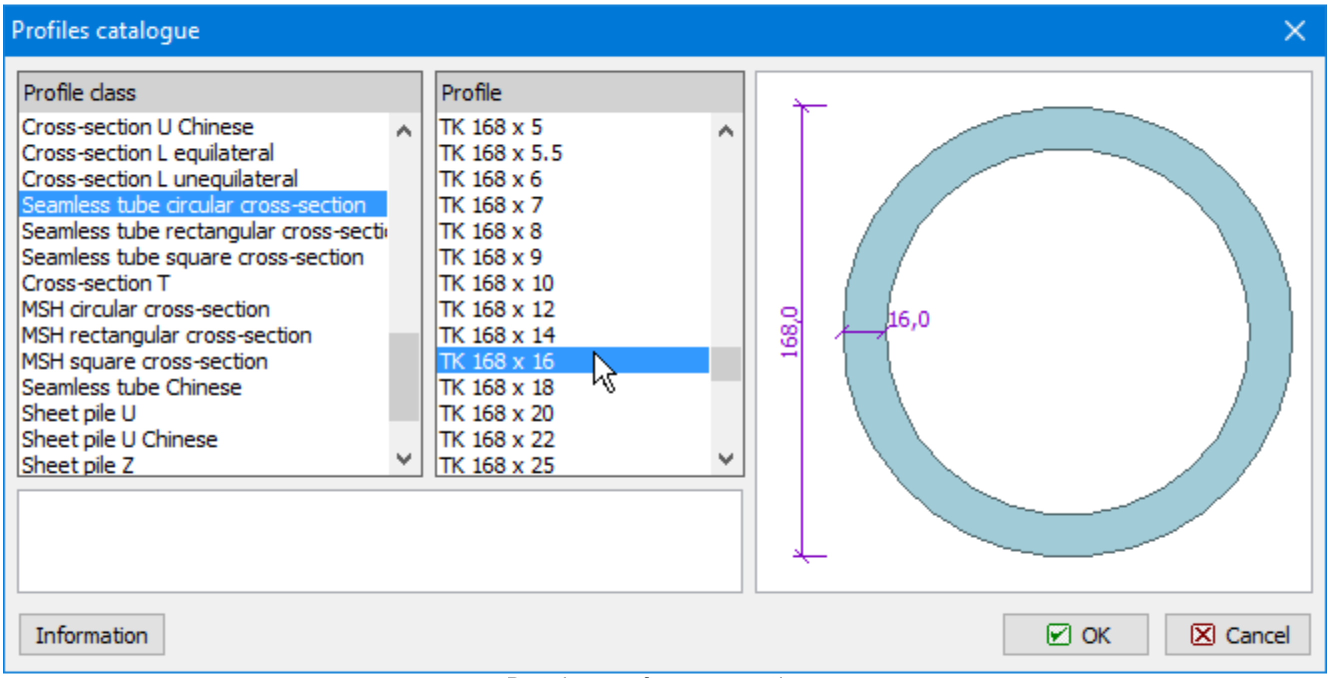
The window "**Steel section**" that appears after the clicking on the button contains an option to select type of cross-section. We select database of rolled cross-sections (the option "**Solid hot-rolled**") and open the window "**Profiles catalogue**" by pressing the button "**OK**".



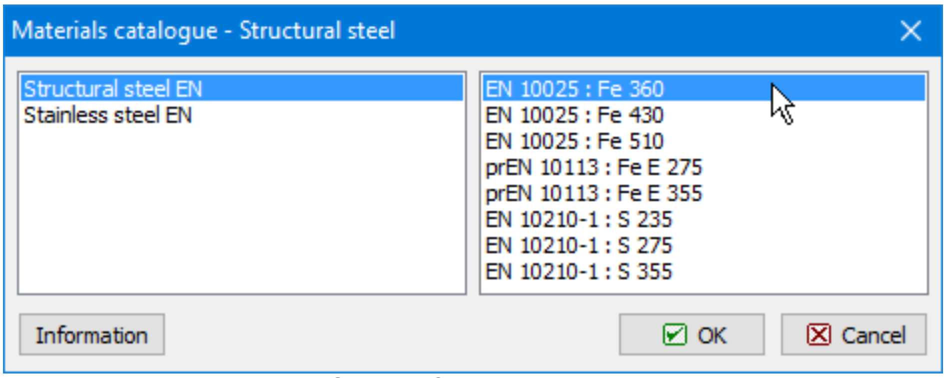

*Choice of cross-section type*

We select the section type "**Seamless tube circular cross-section**" (RHS) in the first column of the database and the item "**TK 168x16**" in the second column. The choice of cross-section has to be confirmed by the button "**OK**".




*Database of cross-sections*

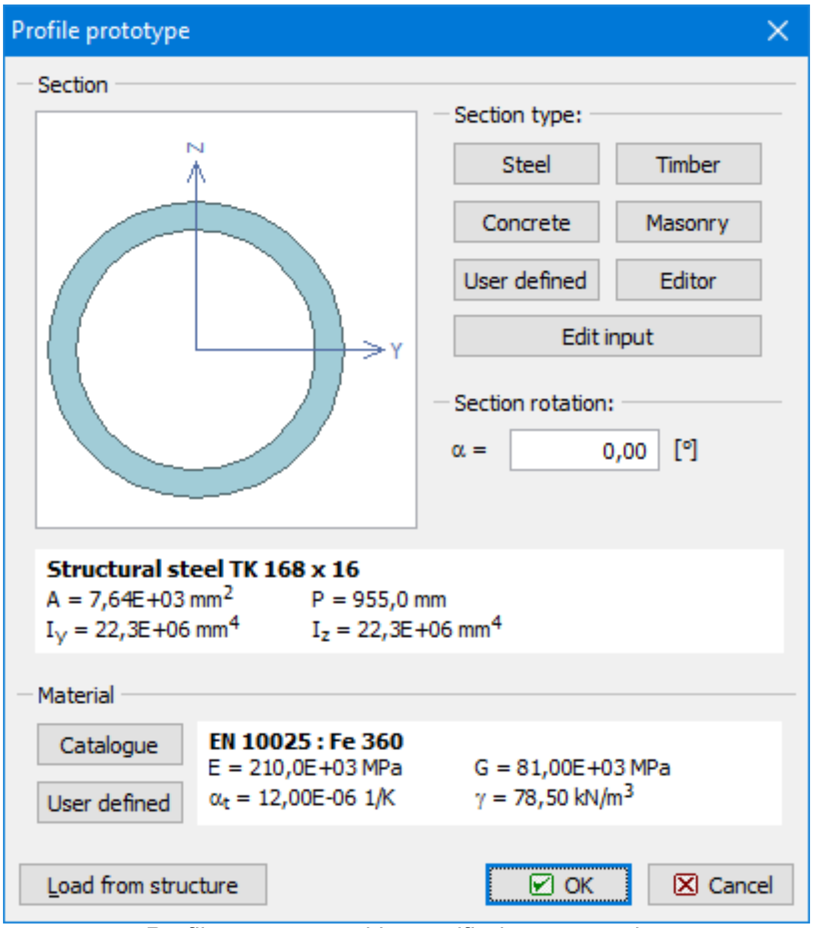
After the confirmation a window with material grades appears. We select the strength class "**EN 10025: Fe 360**".




*Choice of strength grade*

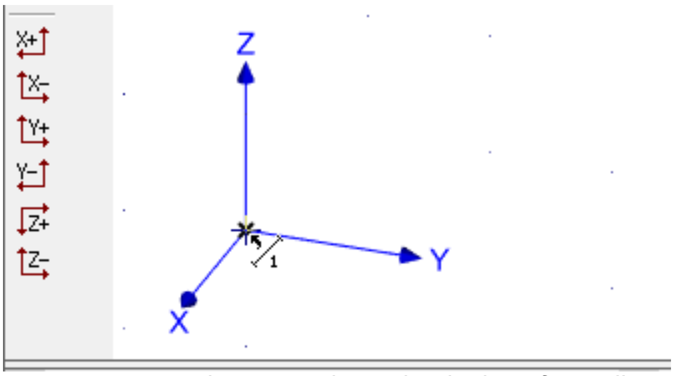
The properties entered in the window "**Profile prototype**" are displayed in the following figure.



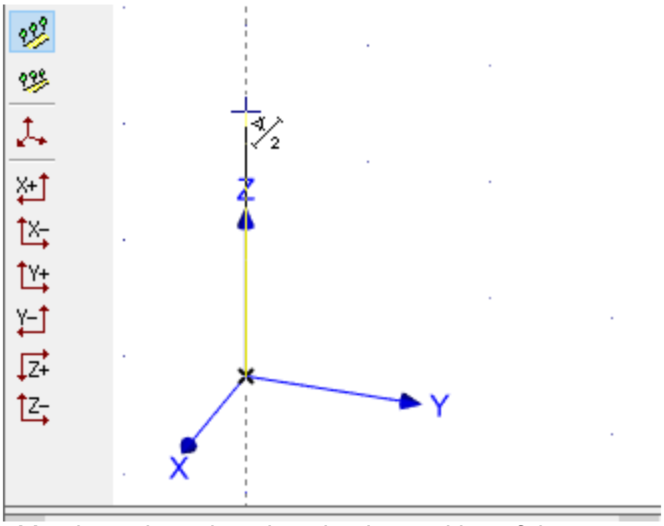

*Profile prototype with specified cross-section*

After the confirmation of the window "**Prototype profile**" by the button "**OK**", the prototype properties will be docked in the bottom frame. The input of members in the workspace is enabled now. First, we will insert a column. We specify the start of global coordinate system as a member beginning. The cursor snaps to this points automatically in close surrounding. The snapping is also indicated by the change of cursor appearance.



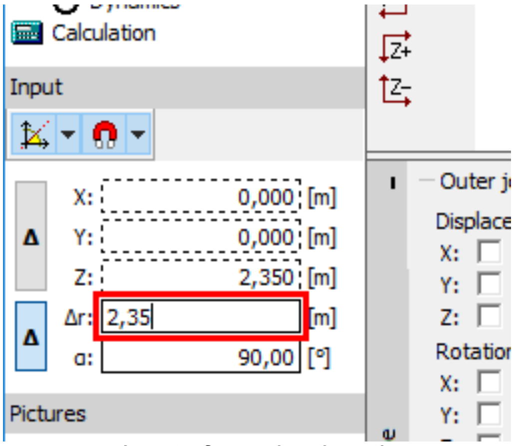

*The cursor appearance when snapping to beginning of coordinate system*

We show the orientation of the column by the cursor in the next step (the orientation should be in the direction of the axis "**Z**"). The alignment into this direction is automatically offered by the program (the program automatically snaps into *45°* directions as a default).



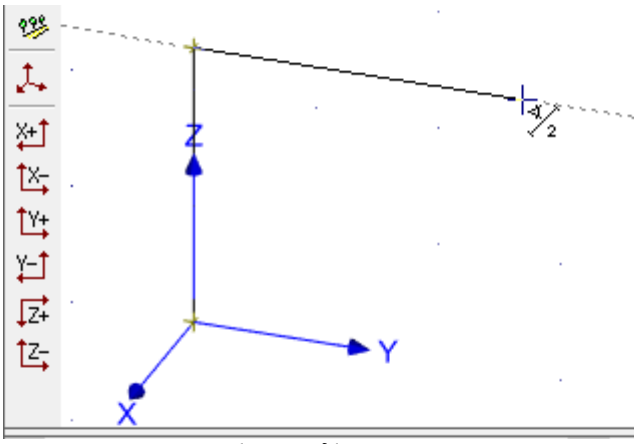

*Member orientation given by the position of the cursor*

Paralelly, we specify the length of member (column height) *2.35* using keyboard. The lengths have to be specified in metres. The length entered on the keyboard is automatically inserted into the input field "**Δr**" in the bottom part of the tree menu. The input has to be confirmed by the button "**Enter**".




*Input of member length*

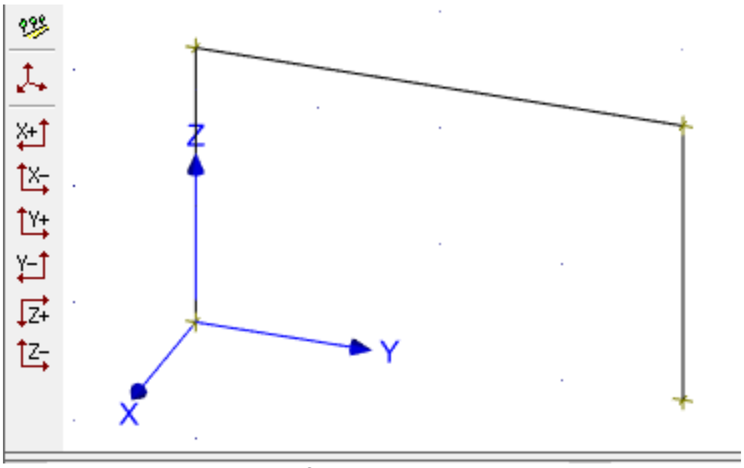
The input of beam follows. The beginning is identical to the end of the column, the end will be specified with the help of cursor (direction parallel to the axis "**Y**") and the length *4m*. We will finish the input of the length by the button "**Enter**".




*Input of beam*

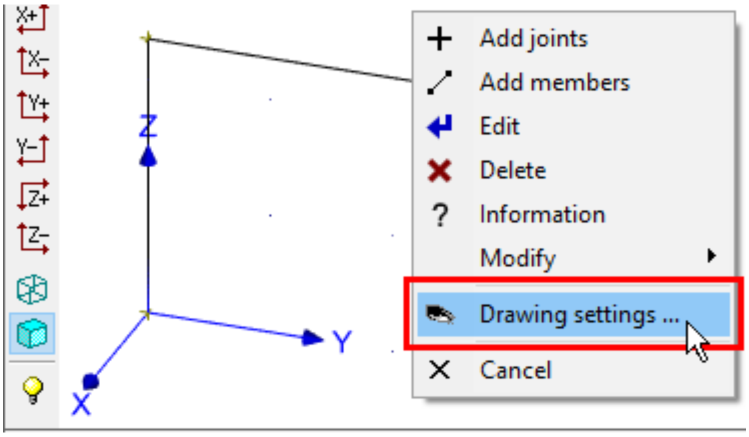
We will continue with the input of the second column. The input will be done from the end of the beam against the direction of the axis "**Z**".




*Frame made of two columns and a beam*

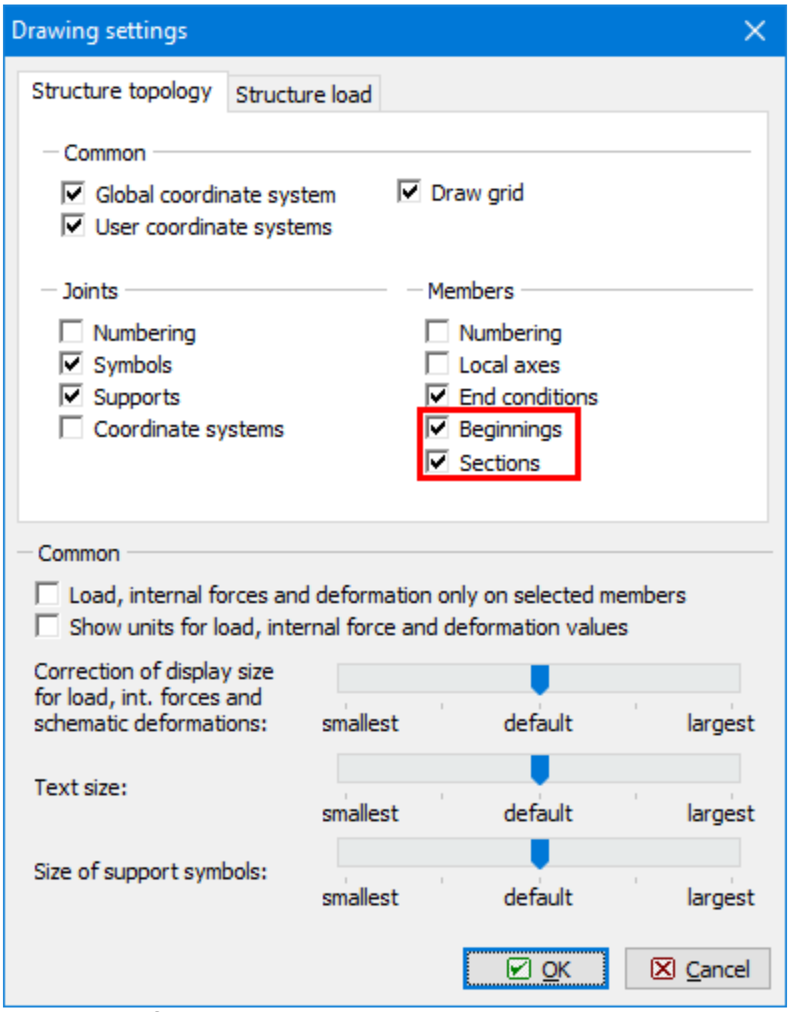
We terminate the input of members by a right mouse button click. After that, we change the display settings to improve the visibility of input. The window "**Drawing settings**" can be opened with the help of the context menu, which can be launched by a right mouse button click everywhere in the workspace.




*Drawing settings in the context menu*

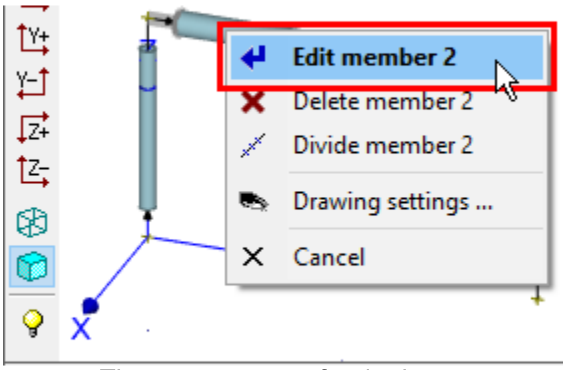
The window "**Drawing settings**" contains parameters, which affect the displayed objects in the workspace. We check settings "**Beginnings**" and "**Sections**". First setting highlights beginnings of members by black arrows, second shows member masses. The window has to be closed by the button "**OK**".




*Changes in the window "Drawing settings"*

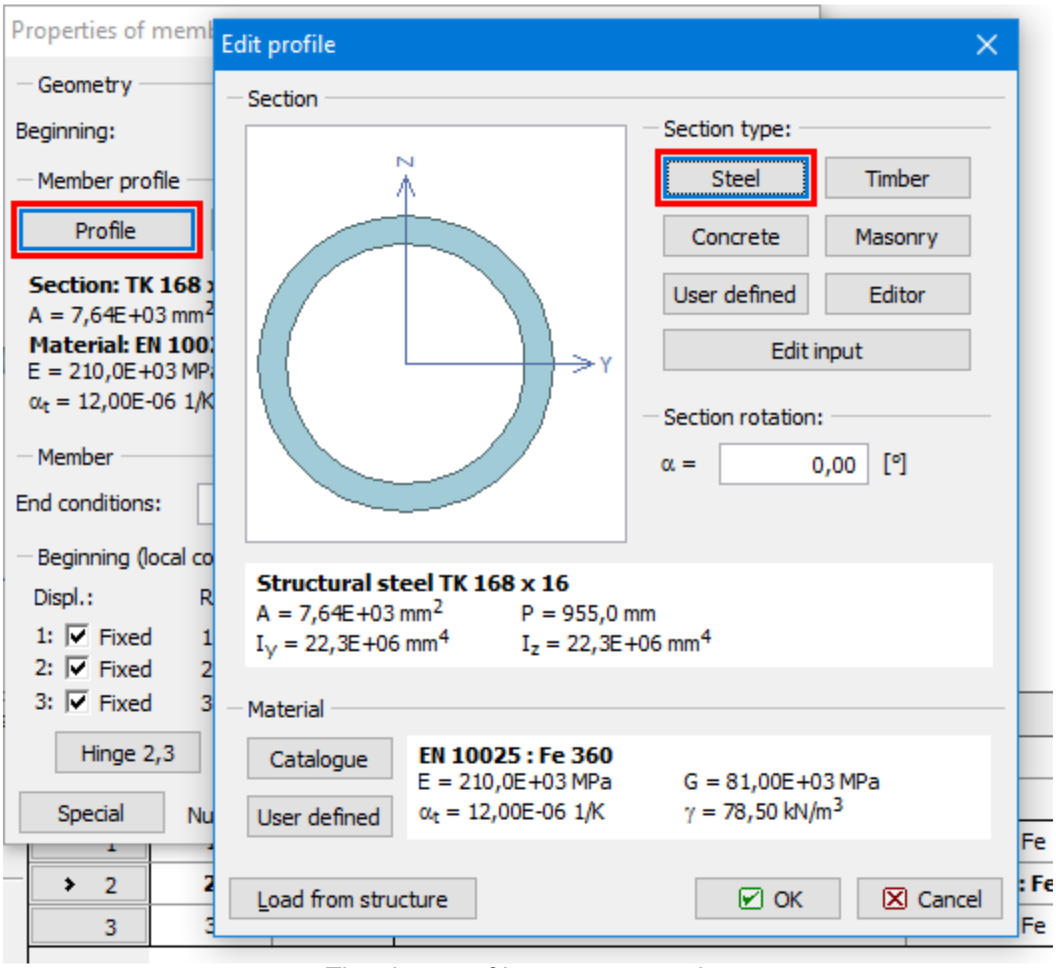
The workspace shows now, that both columns and beam have identical cross-section (RHS). Therefore, next step is to change the cross-section of the beam. We open the context menu for the beam by right button mouse click on this member. We select the option "**Edit member**" in this menu.




*The context menu for the beam*

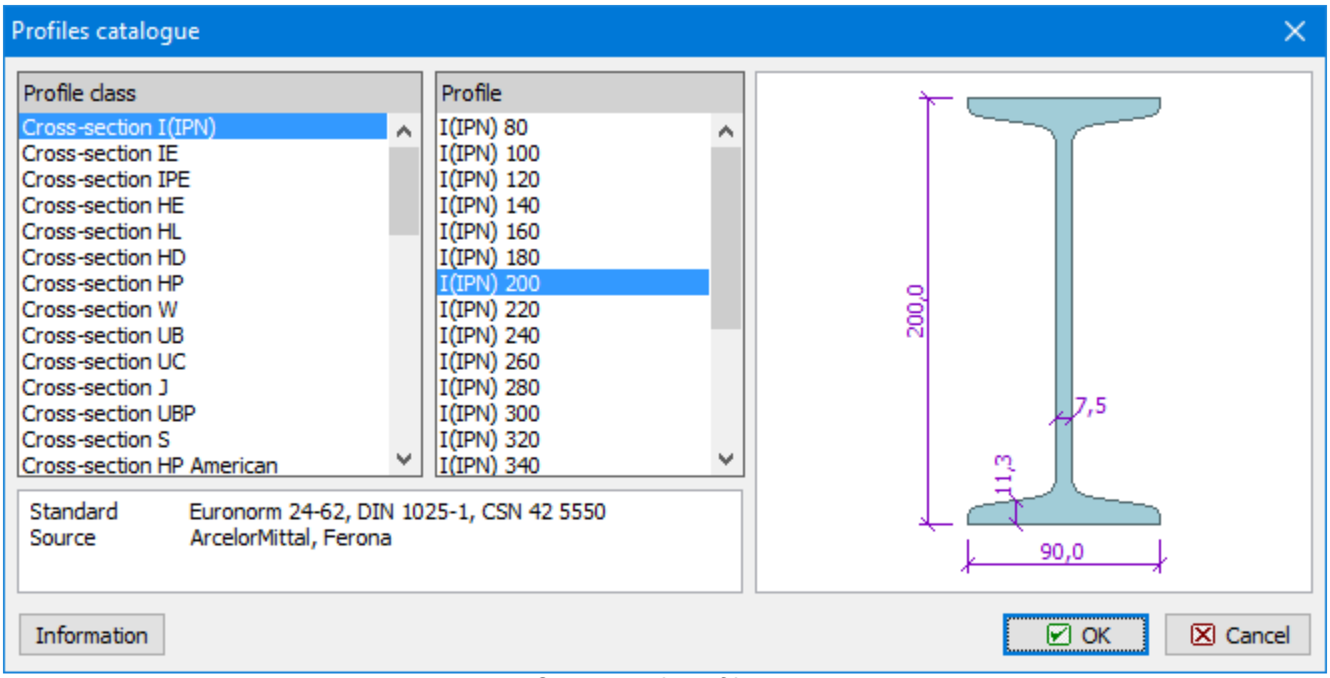
This action opens the window "**Properties of member**". We change the cross-section using method already described above. The button "**Profile**" launches the window "**Edit profile**". We select the option "**Steel**" in this window.




*The change of beam cross-section*

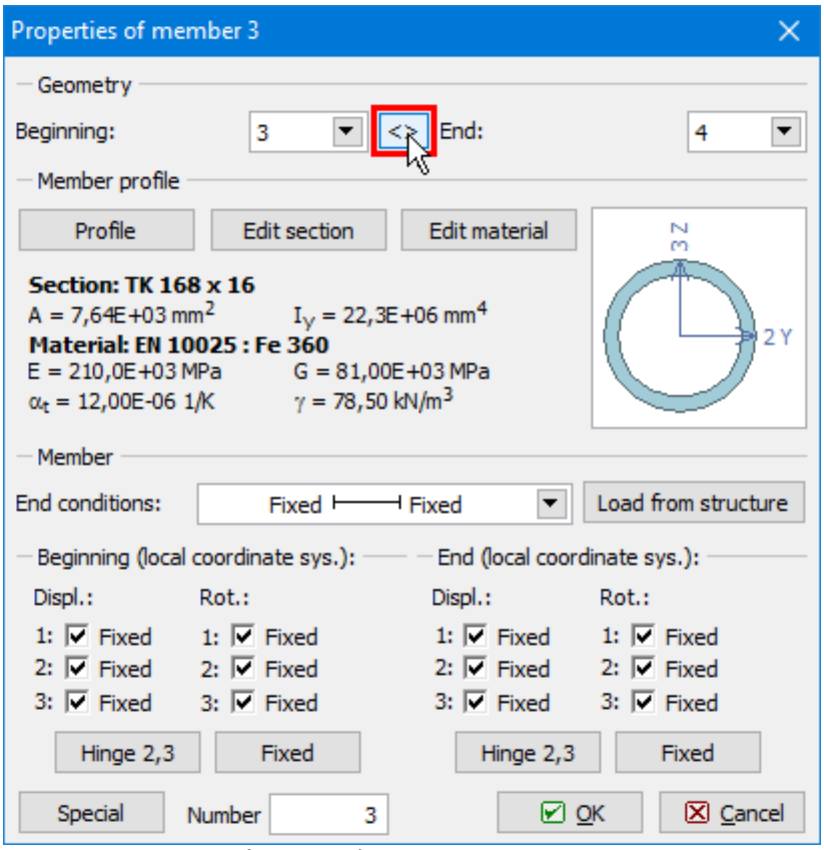
The cross-section type "**Solid hot-rolled**" should be selected for opening the database of rolled cross-sections. The beam should have the cross-section "**I(IPN) 200**".




*Cross-section of beam*

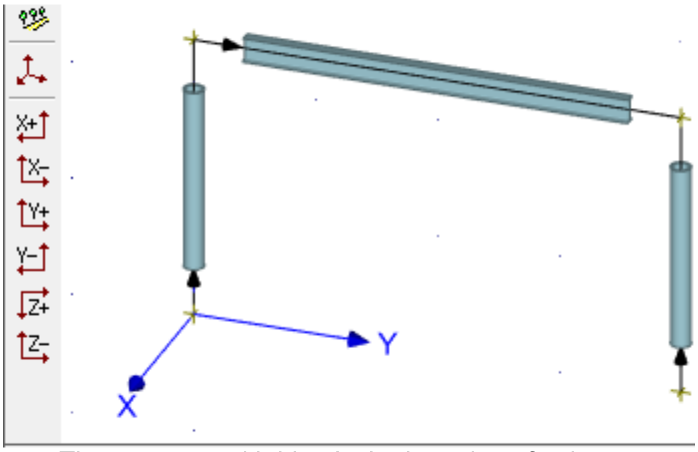
After finishing the changes of beam's cross-section, we will continue by modifying the second column. We open the window "**Properties of member 3**" for the right column and change the orientation of the member (the order of start and end joints). The orientation is not significant for an analysis of internal forces, however, may be critical for a correct verification in the designing module when using unsymmetrical design properties (parameters of buckling etc.). The orientation can be changed by the button "**<>**" between numbers of beginning and end joints. The weindow has to be closed by the button "**OK**".



*Change of column orientation*

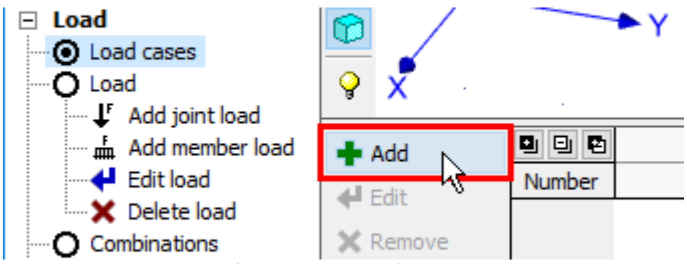
After that, both columns have identical orientation parallel to the axis "**Z**".



*The structure with identical orientation of columns*

## Load input

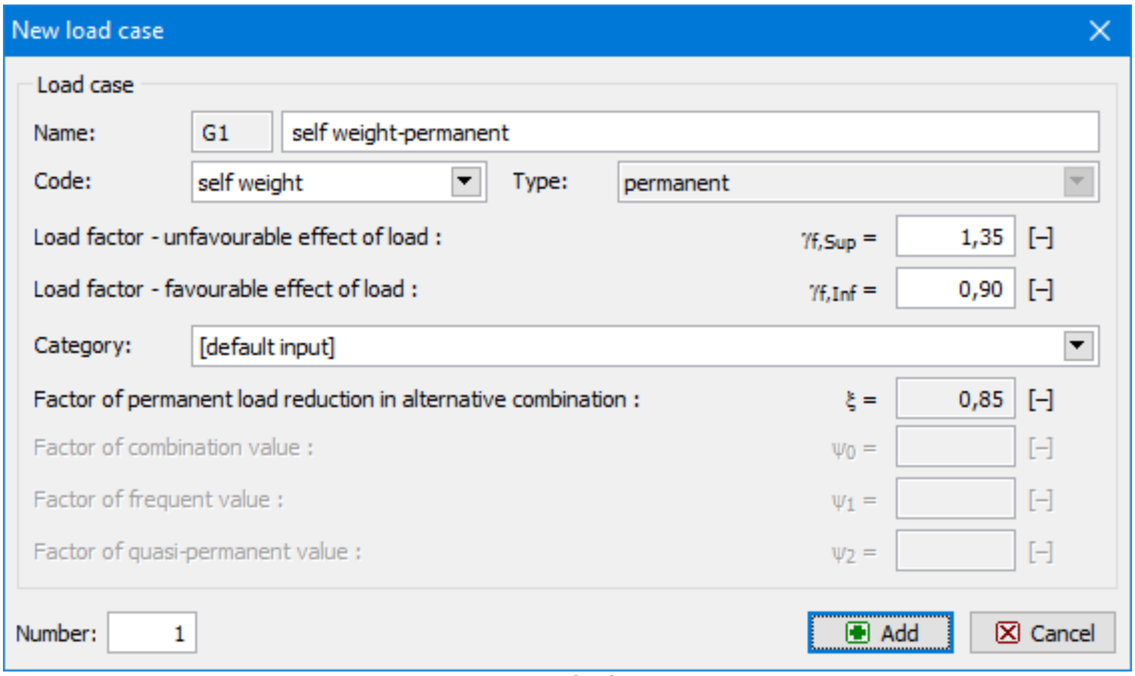
The input of loads follows. We switch the tree menu into the mode "**Load cases**" and add new load cases with the help of the button "**Add**" in the toolbar, that is located on the left side of the input frame.



*The button for insertion of new load cases*

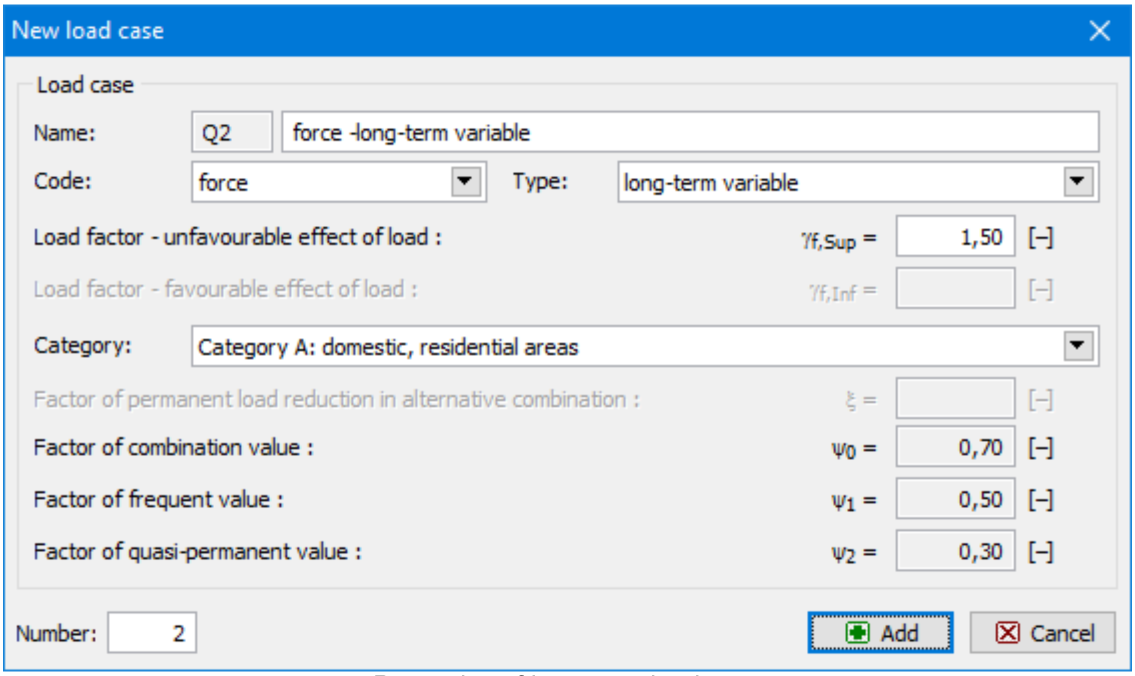
The window "**New load case**" automatically starts with properties of the load case "**Self weight - permanent**". This load case represents a self weight of the structure and the loads included in this load case are generated automatically according to the member properties. We insert this load case by the button "**Add**".



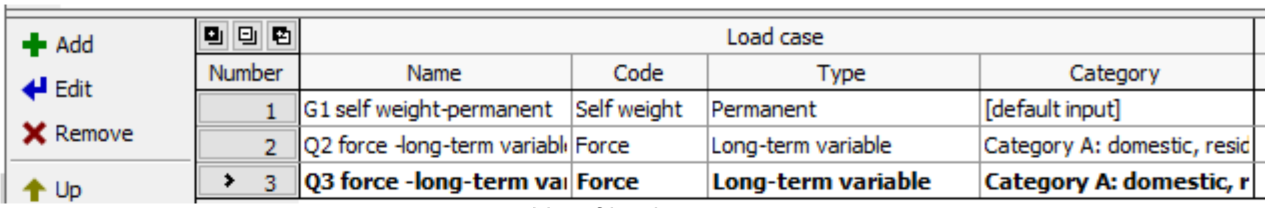

*Load case "Self-weight"*

Analogously, we add two other load cases with selected type "**Long-term variable**".




*Properties of long-term load case*

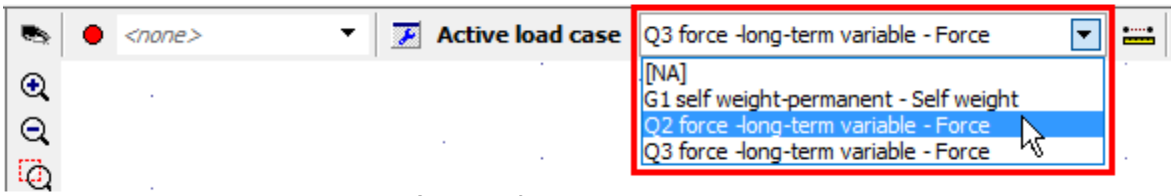
The list of specified load cases is displayed in the bottom frame.




*List of load cases*

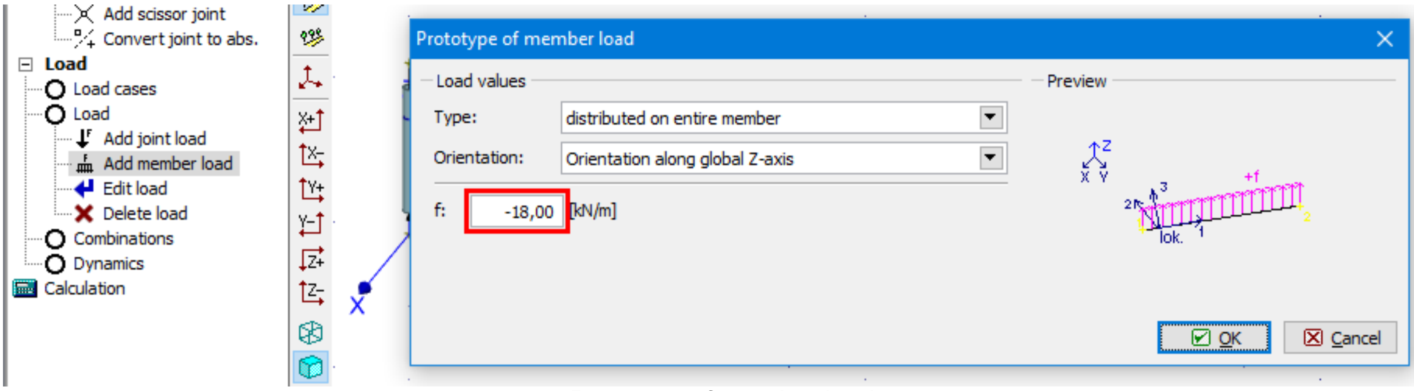
The next step is to insert loads into these load cases (except self weight). The active load case (we start with "**Q2**") has to be selected in the drop-down menu in the heading of the workspace.



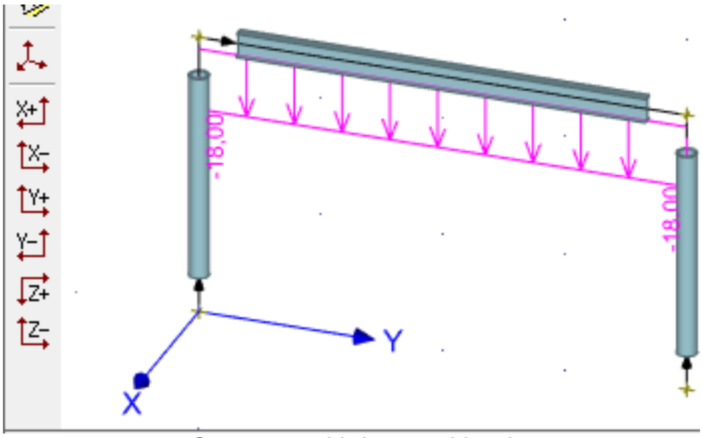

*Choice of the active load case*

We switch the tree menu into the mode "**Add member load**" and specify linear load with the value *-18 kN/m* in the window "**Prototype of member load**". The prototype properties has to be confirmed by the button "**OK**".



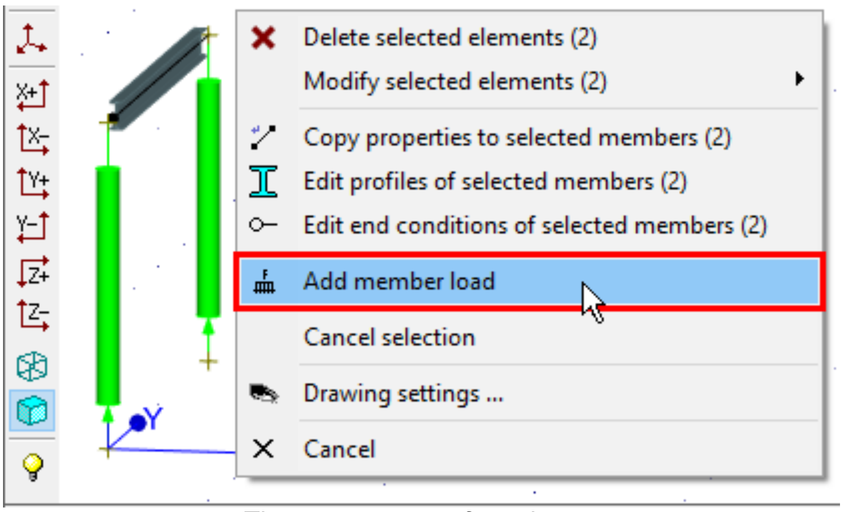

*Prototype of member load*

We assign this load to the beam by clicking on the member in the workspace. The load appears in the workspace immediately.




*Structure with inserted load*

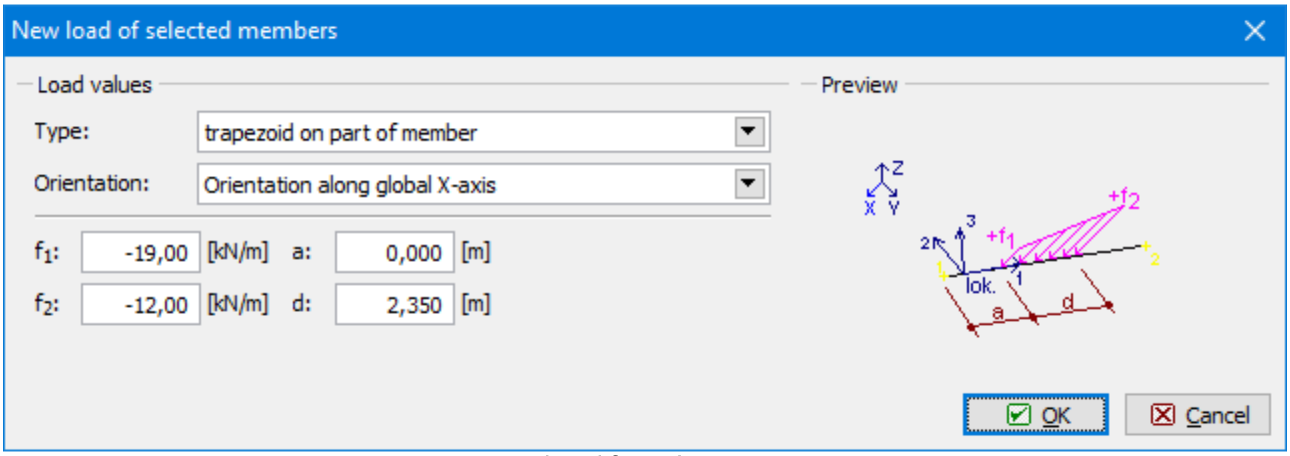
We change the active load case to "**Q3**" in the heading of the workspace. As the load will be identical for both columns, we will use a tool for insertion of loads to selected members. First, we select columns in the workspace. Selected members are highlighted by a green colour. After that, we select the option "**Add member load**" in the context menu.




*The context menu for columns*

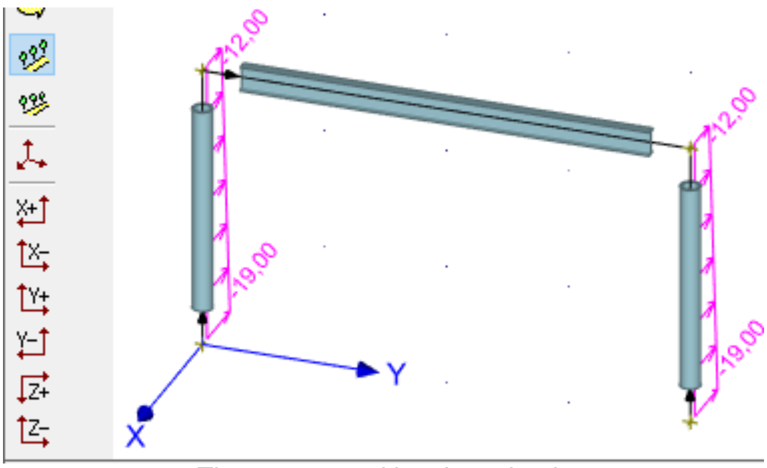
 

We select the load type "**trapezoid on part of member**" and the orientation "**Along global X-axis**". We also specify values at the beginning and end and the length.



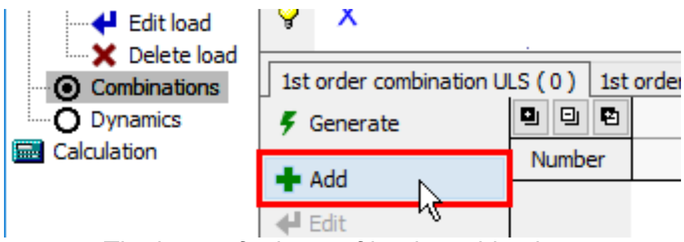
*Load for columns*

The load is applied to both columns after the confirmation by the button "**OK**". The selection of columns can be cancelled by the key "**Esc**".



*The structure with column loads*

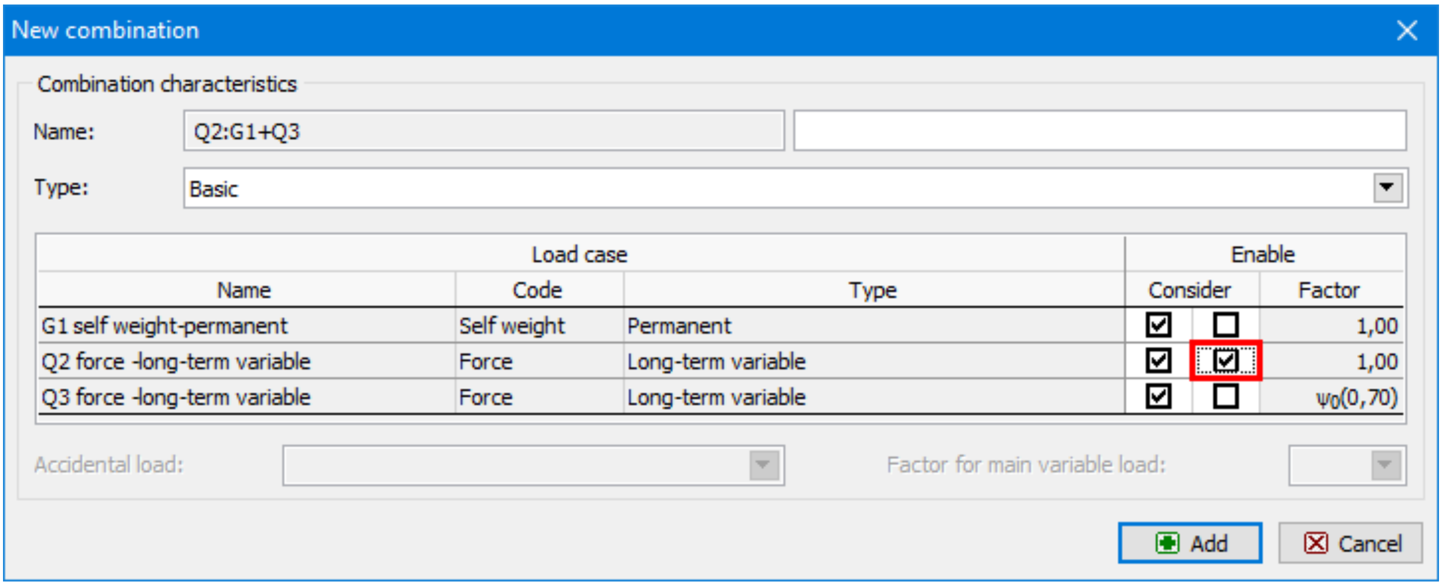
The last part of the load input is a creation of combinations. The combinations are organized separately for ultimate and serviceability limit states in the program. First, we specify combinations for ultimate limit states. We will create two combinations. Both of them will contain all three load cases, however, the main variable load will differ. The tree menu has to be switched to the mode "**Combinations**". We add new combinations with the help of the button "**Add**" in the tab"**1st order combination ULS**" in the bottom frame.



*The button for input of load combinations*

Combination properties are organized in the window "**New combination**". Load cases can be added into the combination with the help of the left check boxes in the column "**Consider**" of the table, which can be found in the mid part of the window. We also tick the second check box for the load case "**Q2**" in the column "**Consider**". This setting sets the load case "**Q2**" as main variable load. We insert the combination into the project with the help of the button "**Add**".




*The main variable load*

We change the setting in the second column "**Consider**" and set the load case "**Q3**" as main variable load. Again, this new combination has to be added by the button "**Add**". We close the window by the button "**Cancel**". Created combinations can be displayed in the dedicated window "**Table of combinations**". This window can be opened with the help of the button "**Table**" in the toolbar on the left side of the bottom frame.

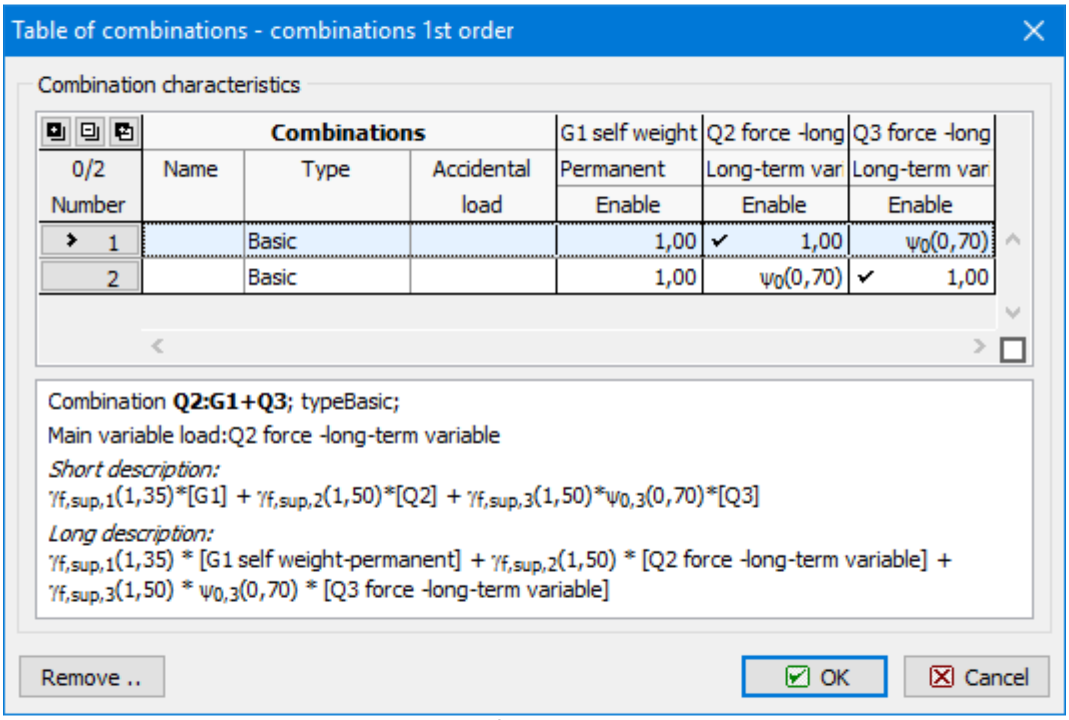
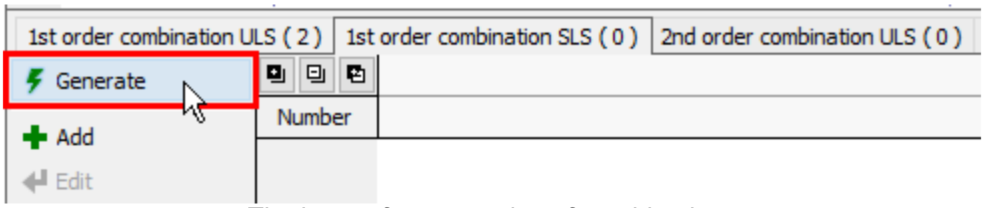



*Table of combinations*

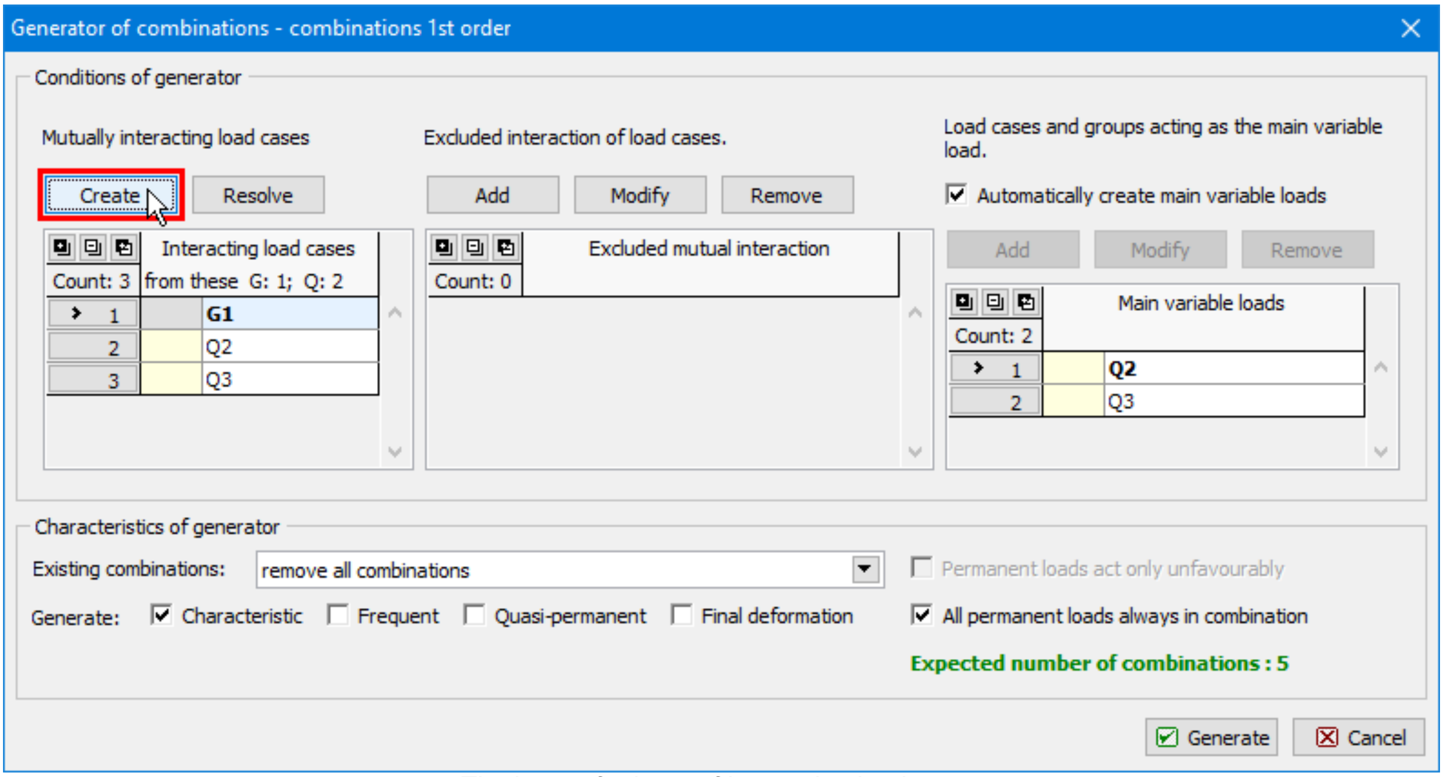
The input of combinations for serviceability limit states follows. We will use an automatic tool "**Generator of combinations**" in this case. This tool can be opened by the button "**Generate**" in the tab "**1st order combination SLS**".




*The button for generating of combinations*

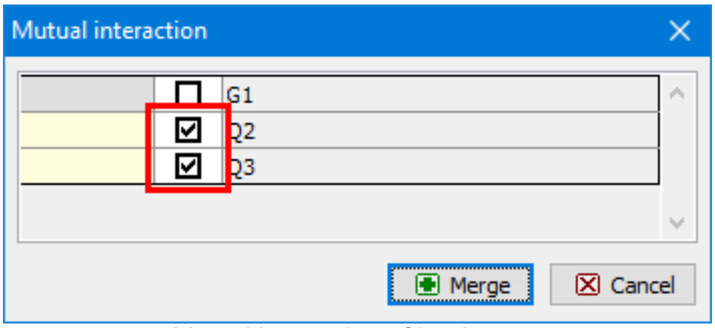
The window "**Generator of combinations**" contains rules, which affect the count of created combinations. We add a new rule which ensures, that specified variable load cases will be included in combinations together. We will use the button "**Create**" for the table "**Mutually interacting load cases**".



*The button for input of interacting load cases*

We tick load cases "**Q1**" and "**Q2**" in the window "**Mutual interaction**" and insert this rule by the button "**Merge**".

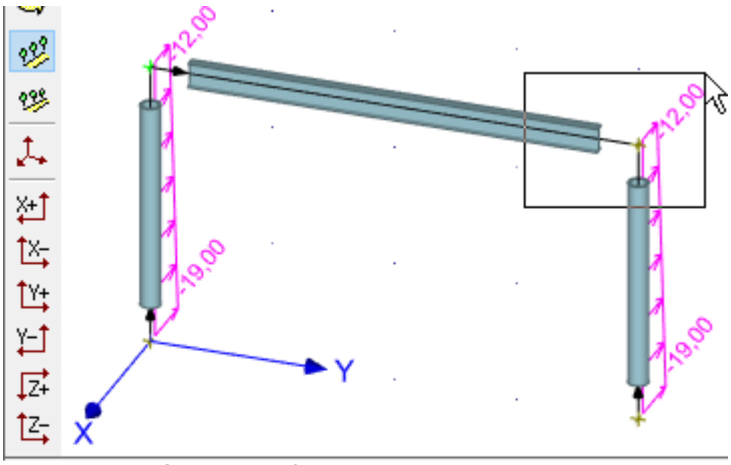


*Mutual interaction of load cases*

Other parameters of generator may stay unchanged. The tool creates a set of combinations for serviceability limit states after closing the generator window by the button "**Generate**".

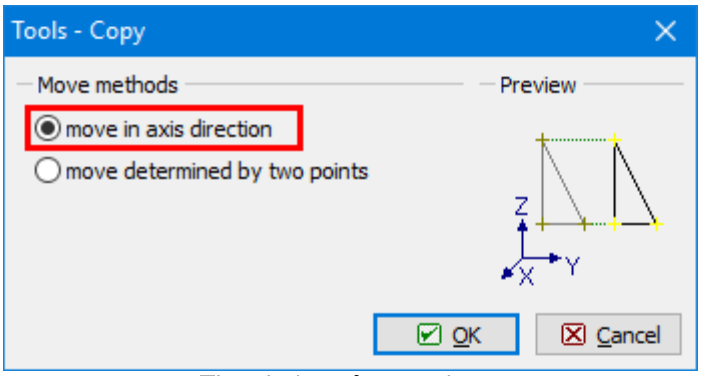
## Enlargement of structure

This chapter describes the enlargement of the structure with the help of tool for copying the elements. This tool will copy existing frame to a new position. First, we select upper joints, as transverse beams can be created automatically in the positions of selected joints.



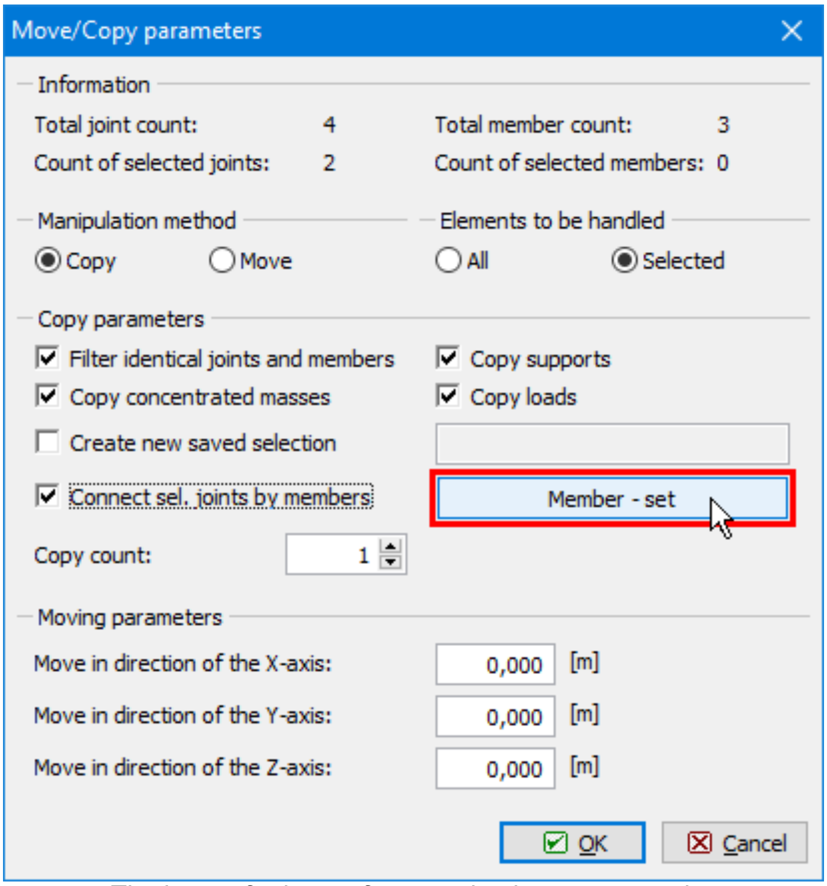
*Selection of joints in the workspace*

We use the tool "**Copy**" in the tree menu and select the mode "**Move in axis direction**".




*The choice of vector input*

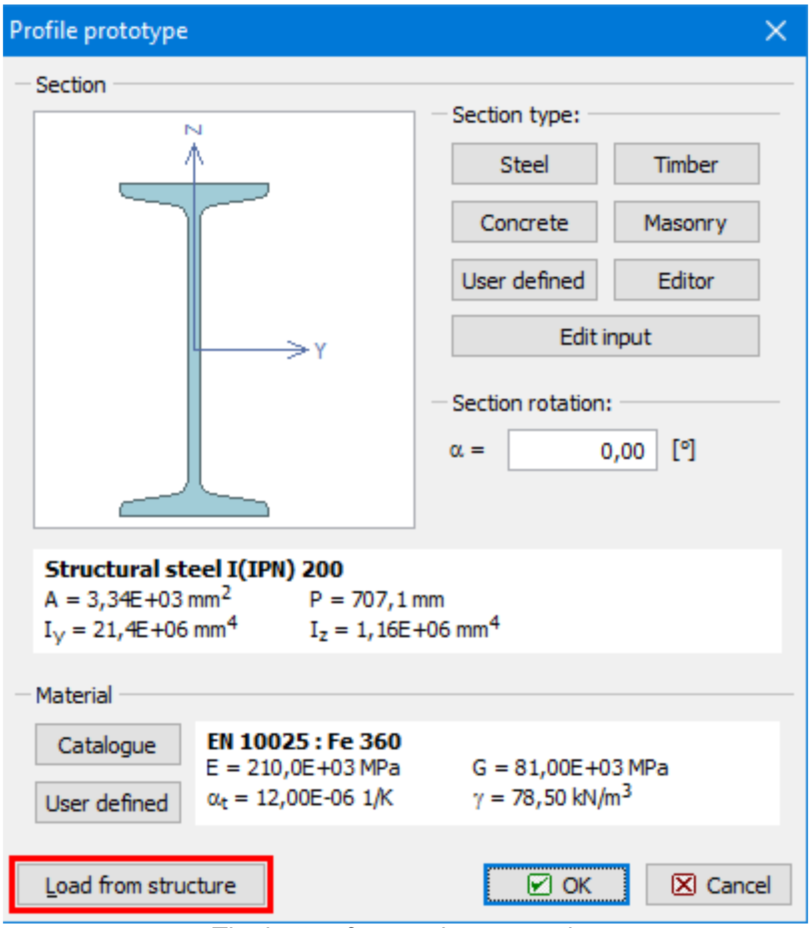
The settings in the window "**Move/Copy parameters**" should be defined according to the figure below. We use also the setting "**Connect sel. joints by members**", which creates new beams connecting old and new frames in selected joints. The cross-section of these members has to be set with the help of the button "**Member - set**".




*The button for input of connecting beams properties*

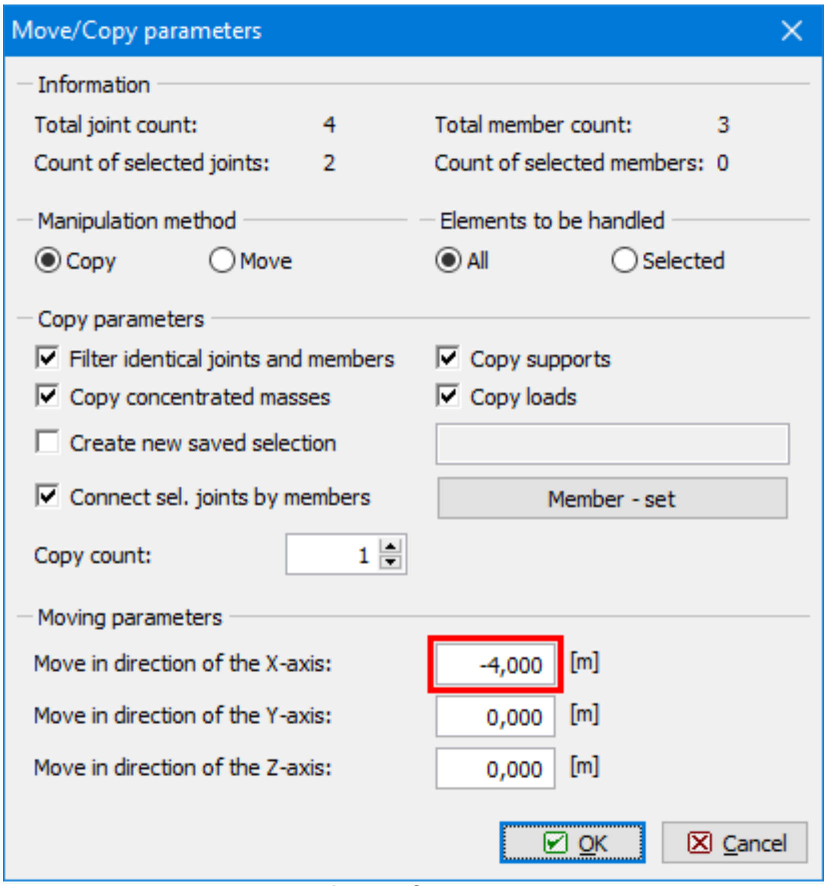
The known window "**Profile prototype**" appears. It is possible to use the button "**Load from structure**" in the left bottom corner and copy cross-sectional properties from existing member.




*The button for copying properties*

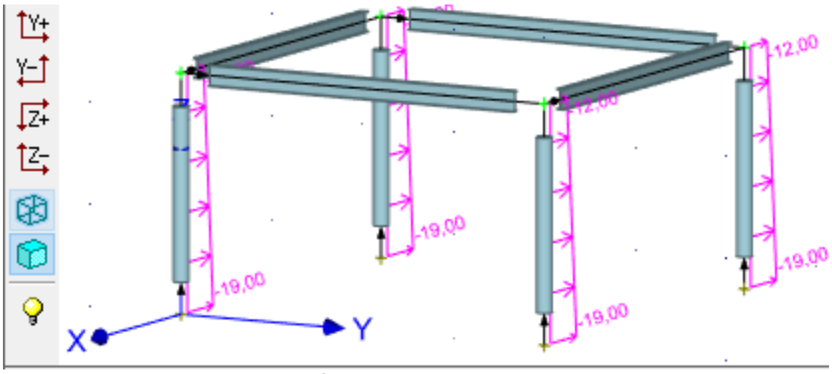
The last part of copy properties is the vector of transformation. We enter the value *-4m* into the input line "**Move in direction of the X-axis**". The parameters have to be confirmed by the button "**OK**".




*Input of vector*

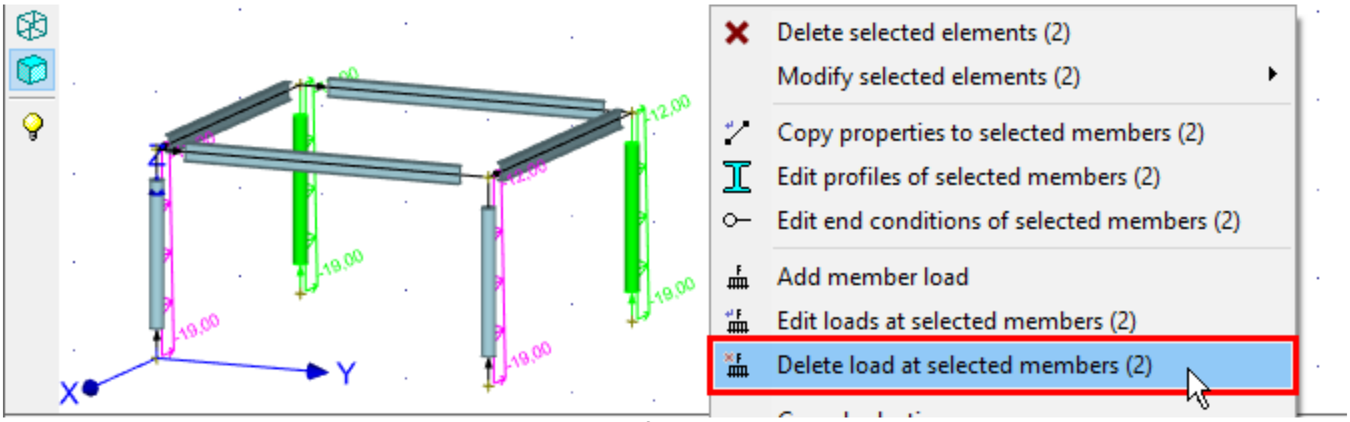
 

The tool copied the frame into a new position and also added new beams connecting old and new frame.



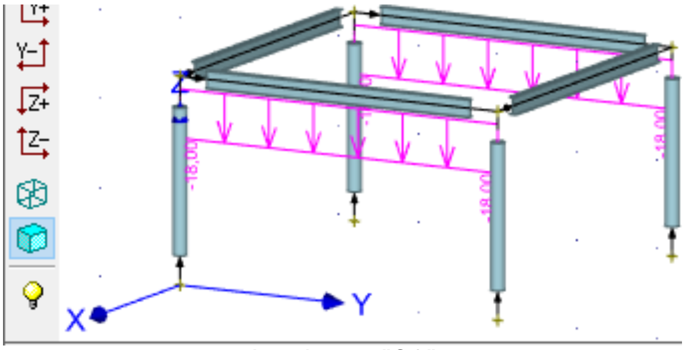
*Created structure*

Columns were copied including loading. As only two columns should be loaded, we have to delete loads on new columns. We select these columns and use the tool "**Delete load at selected members**" in the context menu.




*Deletion of selected loads*

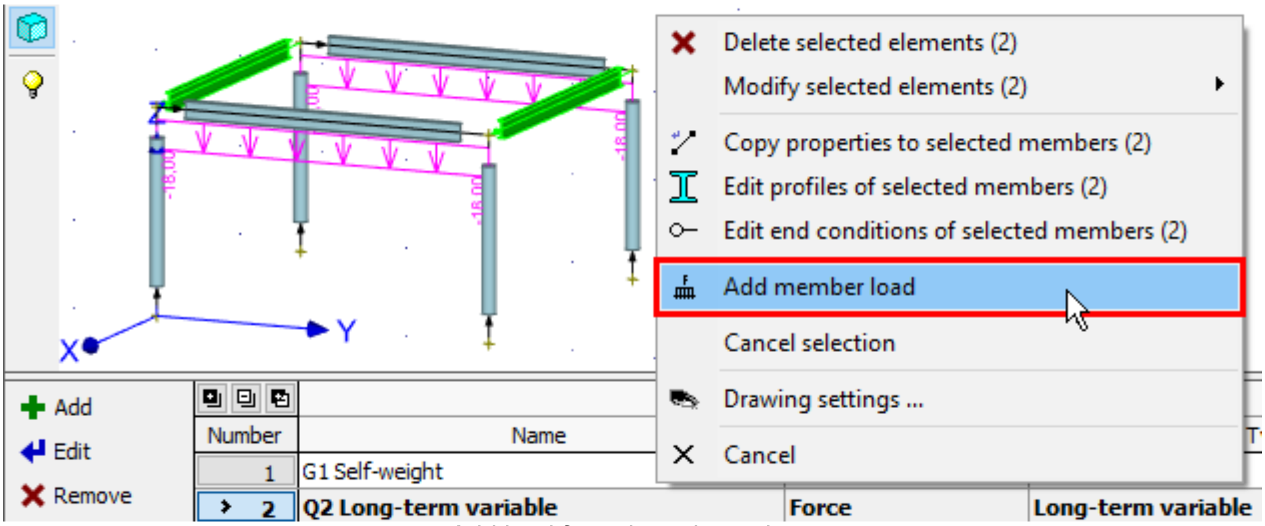
We cancel the selection by the key "**Esc**". After changing the active load case to "**Q2**" in the heading of the workspace, the workspace shows, that there is not any variable load pplied to new transverse beams.



*Load case "Q2"*

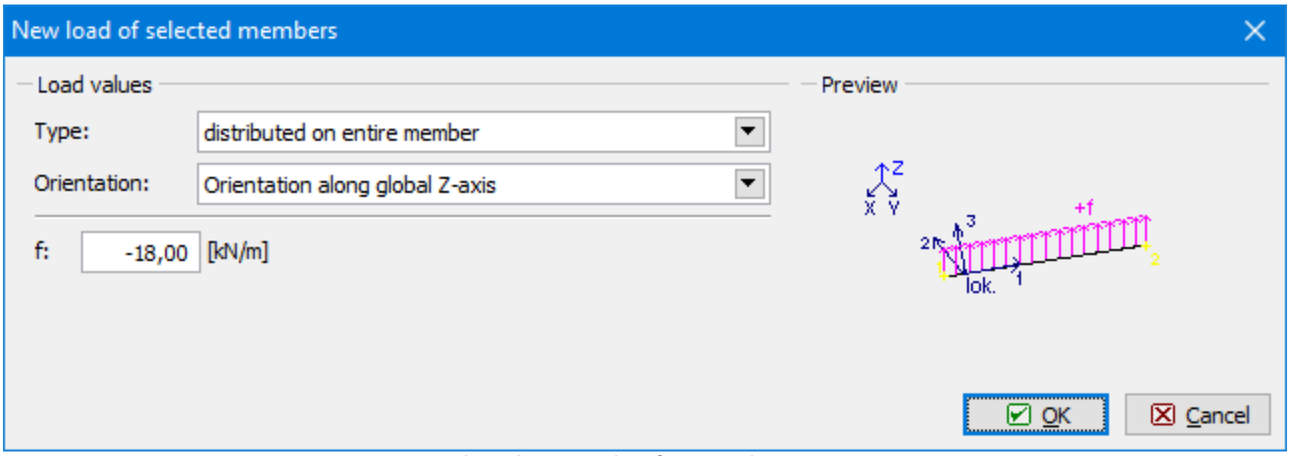
We select these beams and insert loads with the help of the tool "**Add member load**" in the context menu.



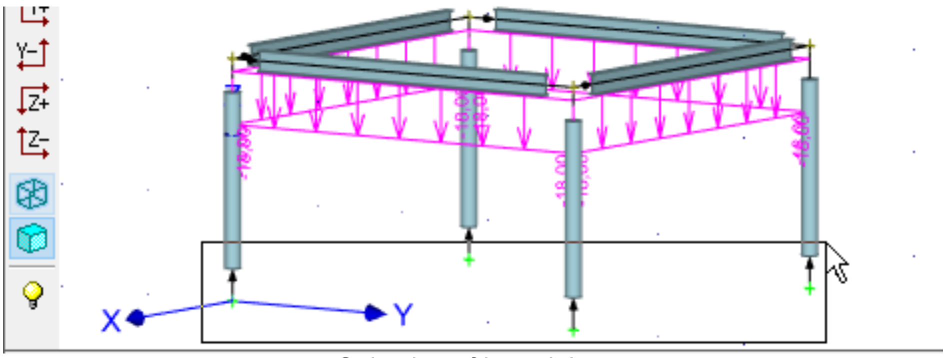

*Add load for selected members*

The new load properties are identical to existing ones.




*Load properties for new beams*

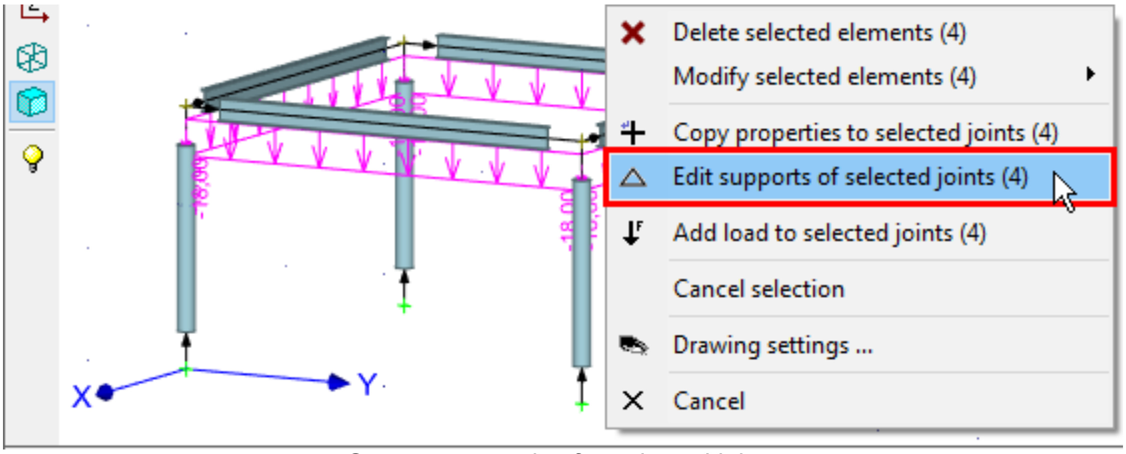
The last topology editing is the input of supports. First, we select all base joints of columns.



*Selection of base joints*

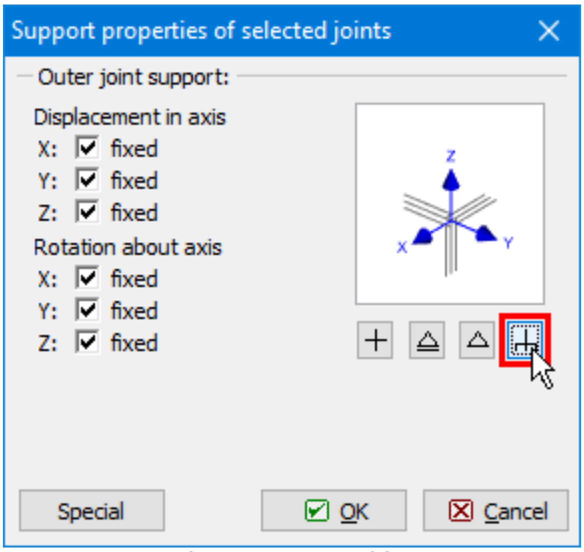
We will use the tool "**Edit supports of selected joints**" in the context menu for selected joints.



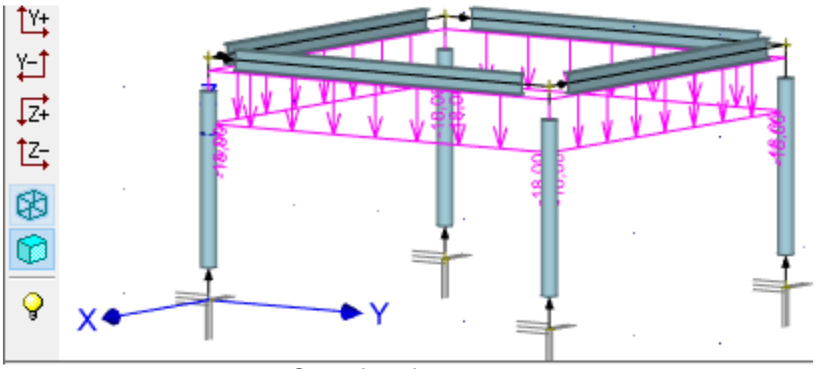

*Support properties for selected joints*

The joinst should be supported in all directions. We can use pre-defined button for fixed support in the right part of the window. The window has to be closed by the button "**OK**".




*The button for easy input of fixed support*

The structure is complete and it is possible to start the analysis.

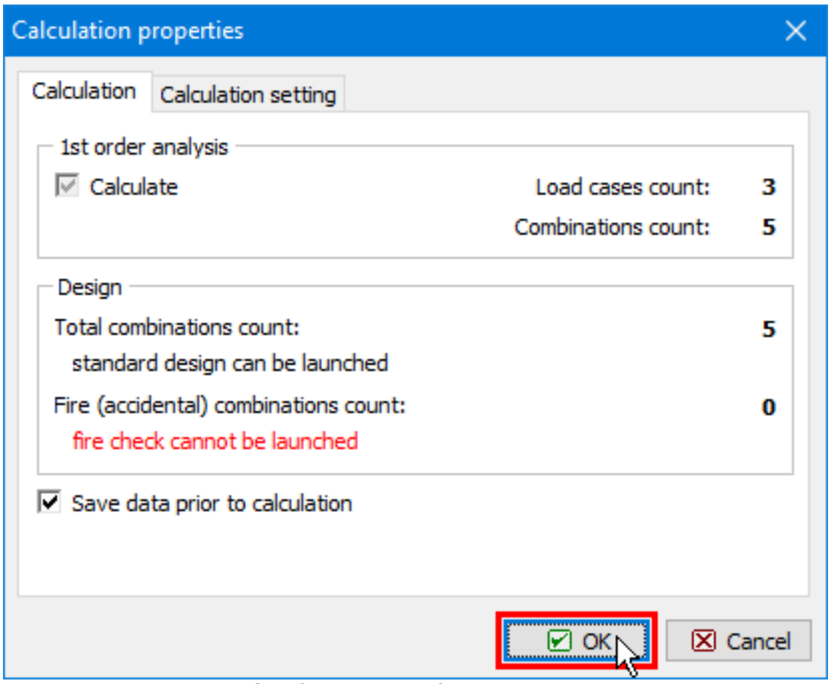



*Completed structure*

## Analysis and results

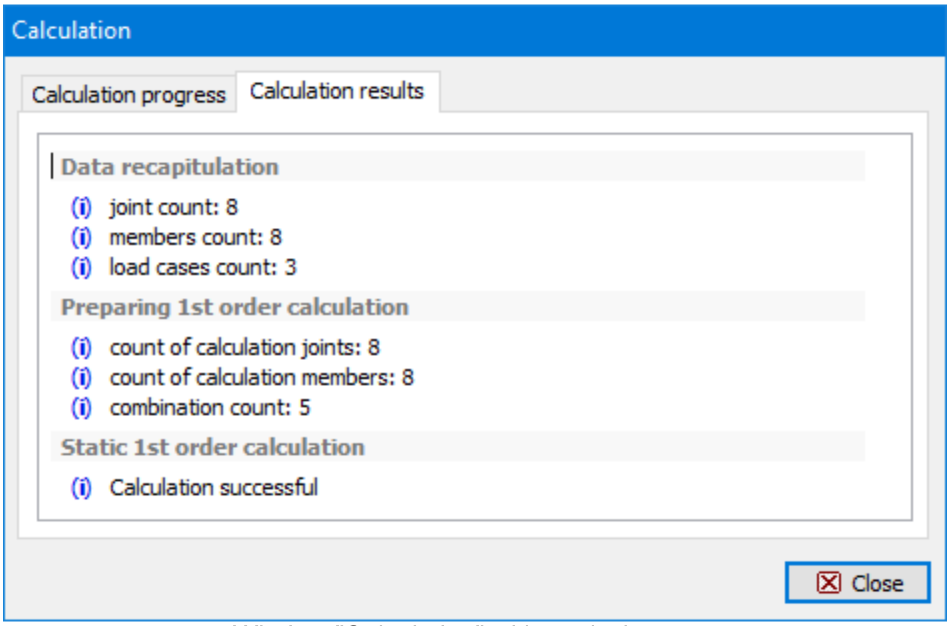
The analysis can be run using the tool "**Calculation**" in the tree menu. The window "**Calculation properties**" with analysis parameters appears before the analysis. We start the calculation by the button "**OK**".




*Confirmation before analysis*

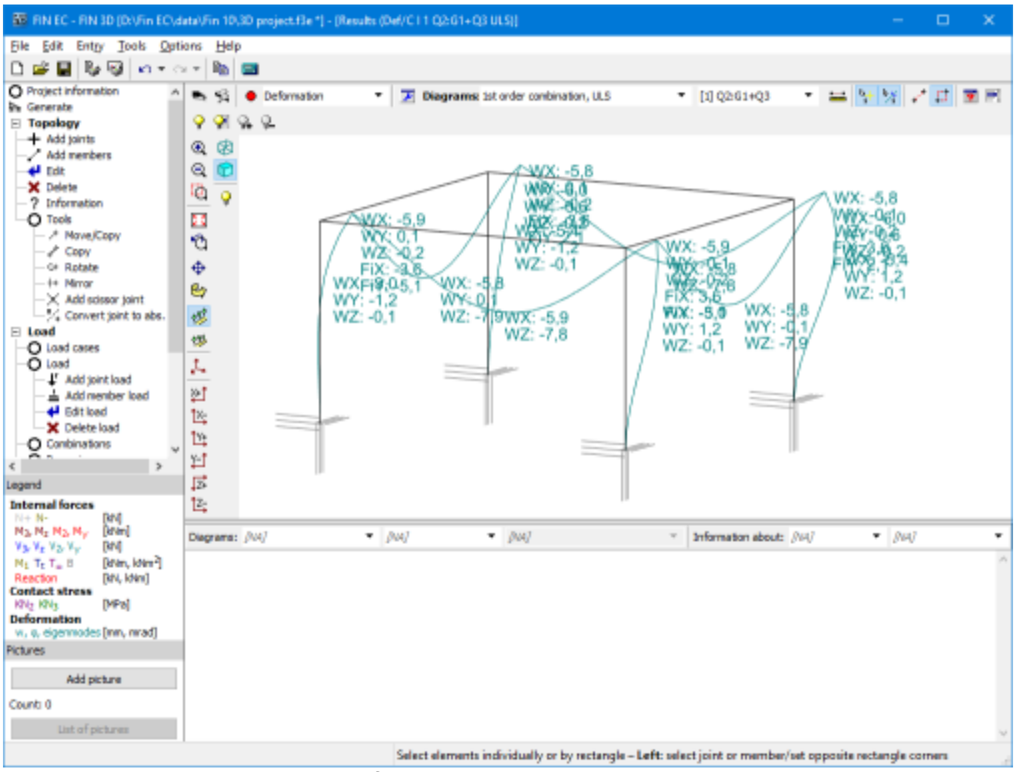
The window with analysis log is shown after the calculation is finished. We will go back to the main window with the help of the button "**Close**".




*Window "Calculation" with analysis report*

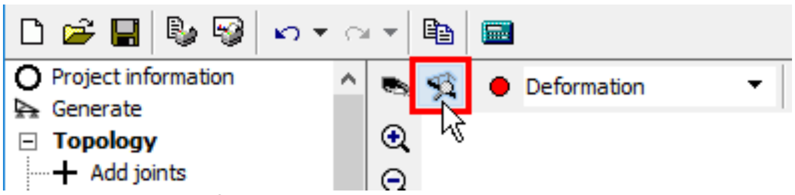
The deformed structure appears in the workspace.



*Deformations in the structure*

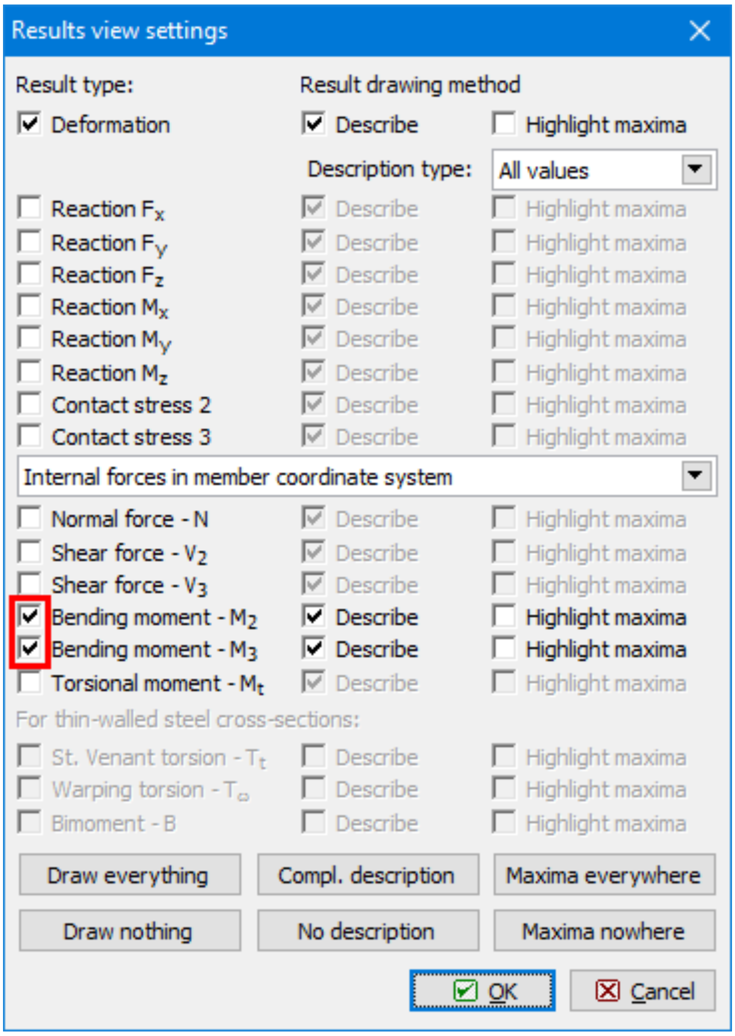
The program contains variety of tools for displaying the results. The window "**Results view settings**", where the displayed quantities can be selected, can be launched by the button "" in the toolbar above the workspace.




*The button for opening window "Results view settings"*

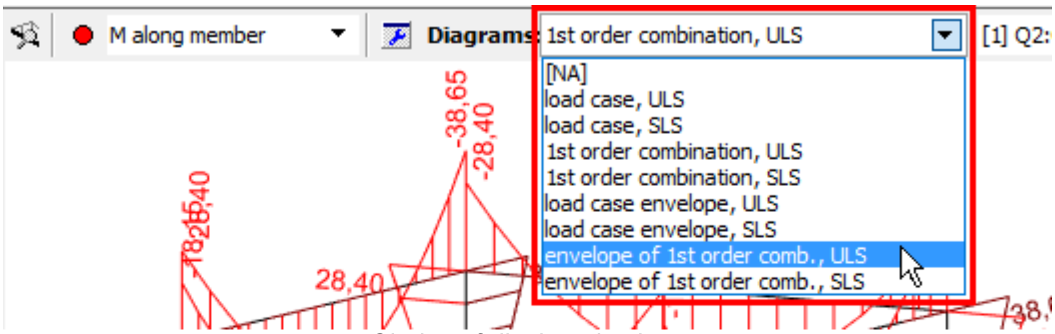
We switch on the display of bending moments.




*Bending moments*

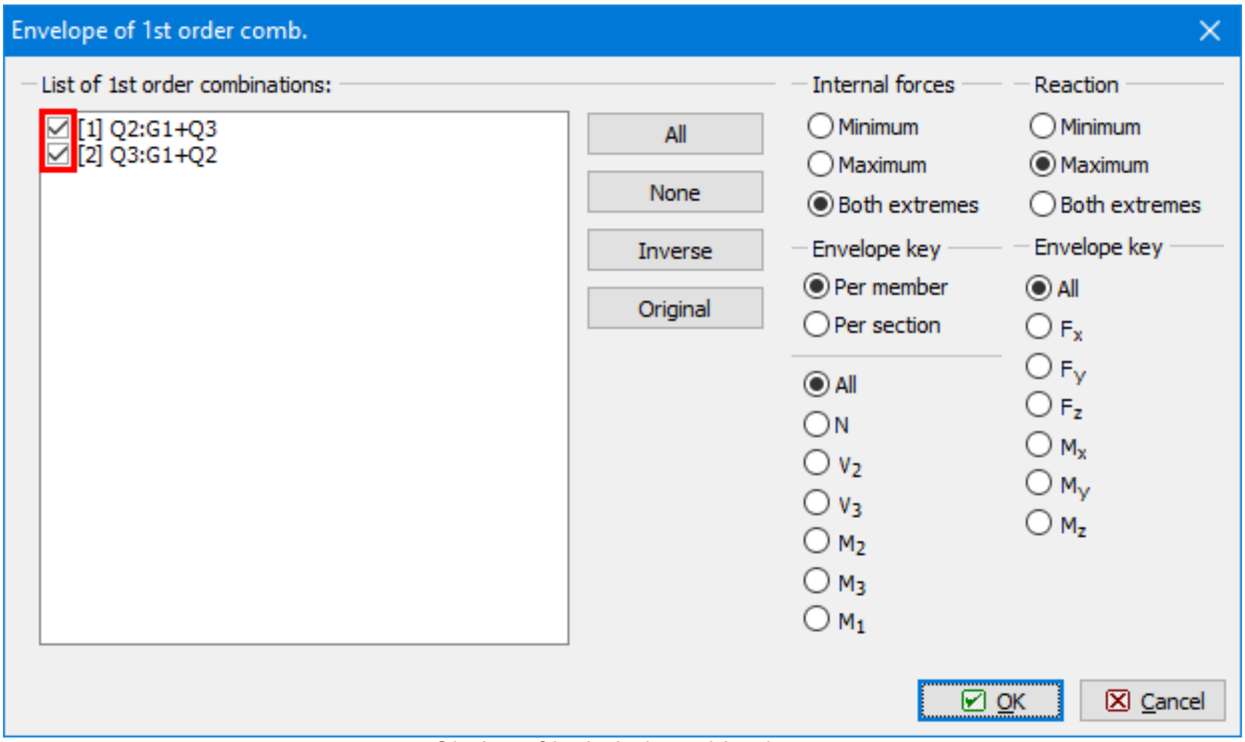
Results can be shown for given load case, combination or envelope. Envelopes show extreme values for more selected load cases or combinations. We will define the envelope of both combinations for ultimate limit states. We select the option "**Envelope of 1st order combination, ULS**" in the drop-down menu "**Diagrams**".




*Choice of displayed values*

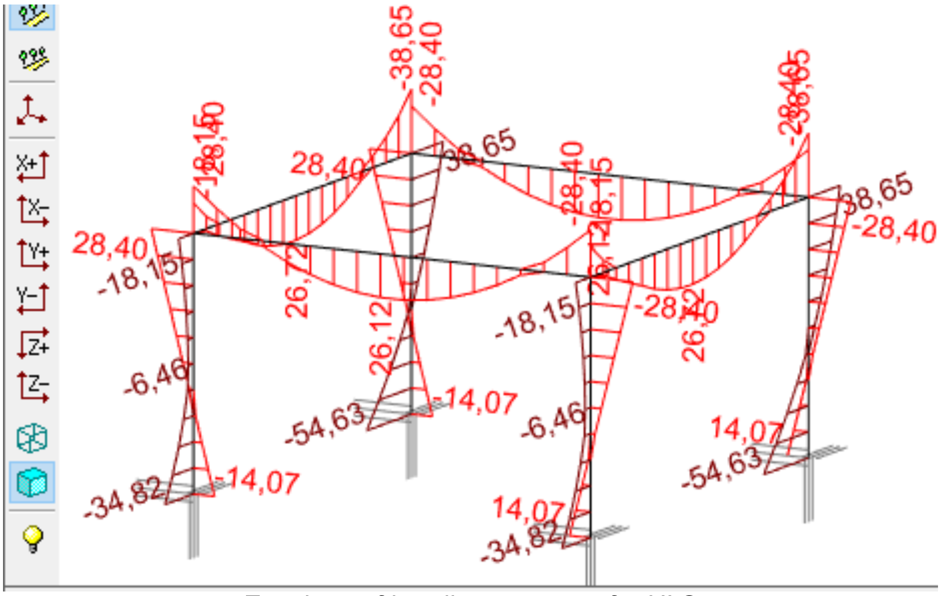
The window "**Envelope of 1st order combinations**" appears. It is necessary to tick both combinations in the left part of the window.




*Choice of included combinations*

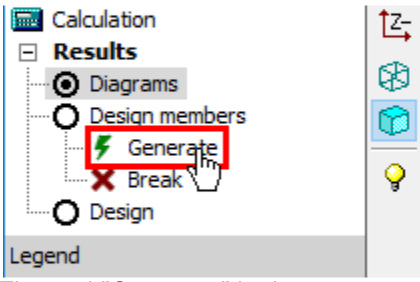
New diagrams show extreme values of bending moments in all points of the structure.




*Envelope of bending moments for ULS*

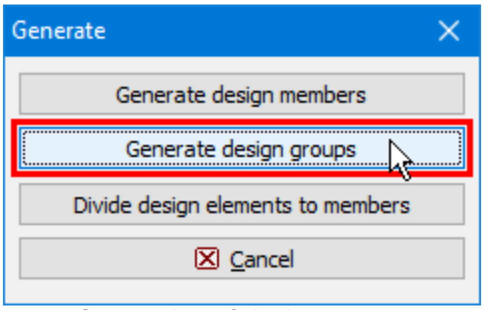
## Verification

The last part of the work is verification of members in designing modules. To reduce the necessary inputs, we merge eight members into two design groups (columns and beams). Design group is verified as one member, however, more results of analysis are considered during the verification. The design groups can be created automatically using the tool "**Design members**" "**Generate**" in the tree menu.



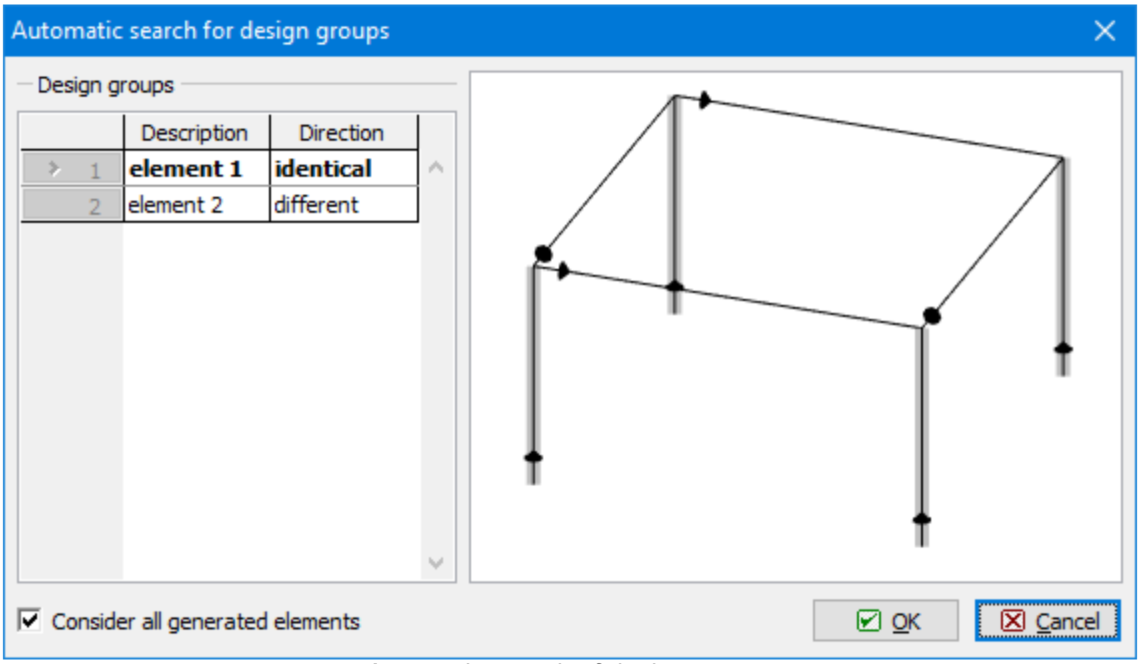

*The tool "Generate" in the tree menu*

We will use the option "**Generate design groups**" in the window "**Generate**".



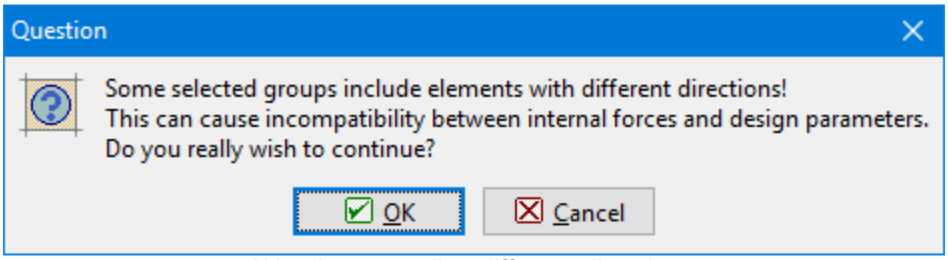

*Generation of design groups*

The window "**Automatic search for design groups**" shows, that two design groups (columns and beams) were found. The right part of the window shows the structure view with highlighted active group.



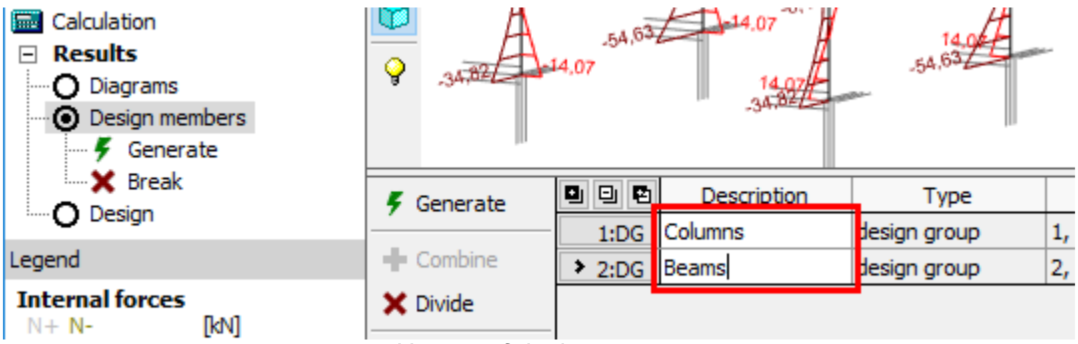

*Automatic search of design groups*

The warning regarding the different orientation of beams appears after the confirmation of design group search. As beams have symmetrical verification parameters (buckling properties etc.), we can continue using the button "**OK**".




*Warning regarding different directions*

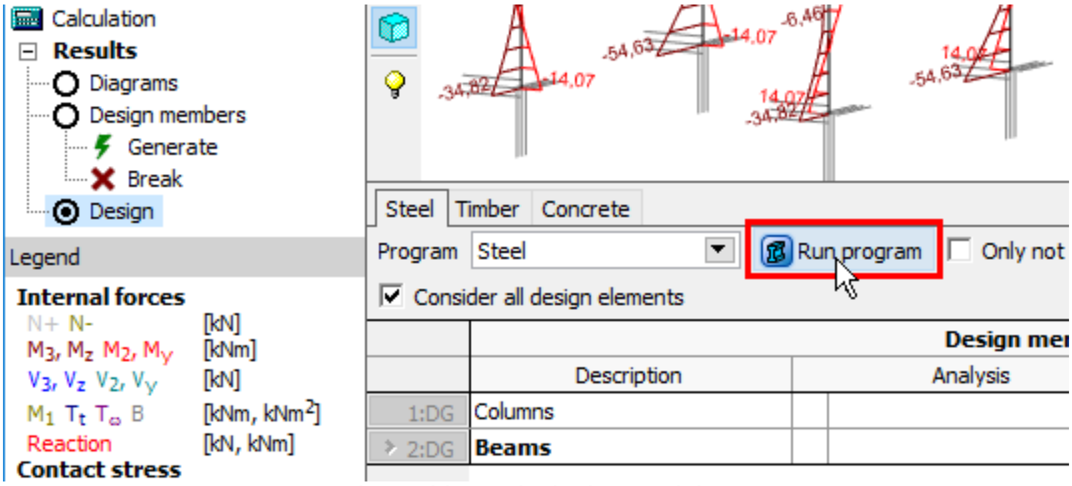
The created design groups can be described in the bottom frame for the mode "**Design members**" of the tree menu.




*Names of design groups*

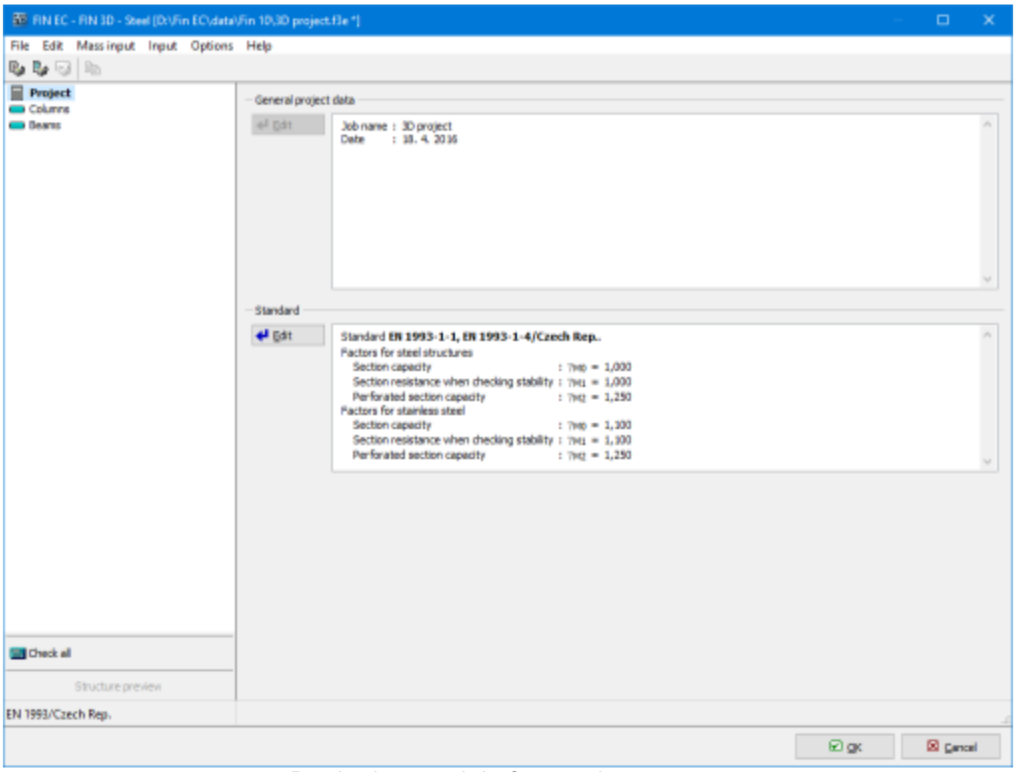
 

We switch the tree menu into the mode "**Design**" and run the design module "**Steel**" by the button "**Run program**".




*Launching a designing module*

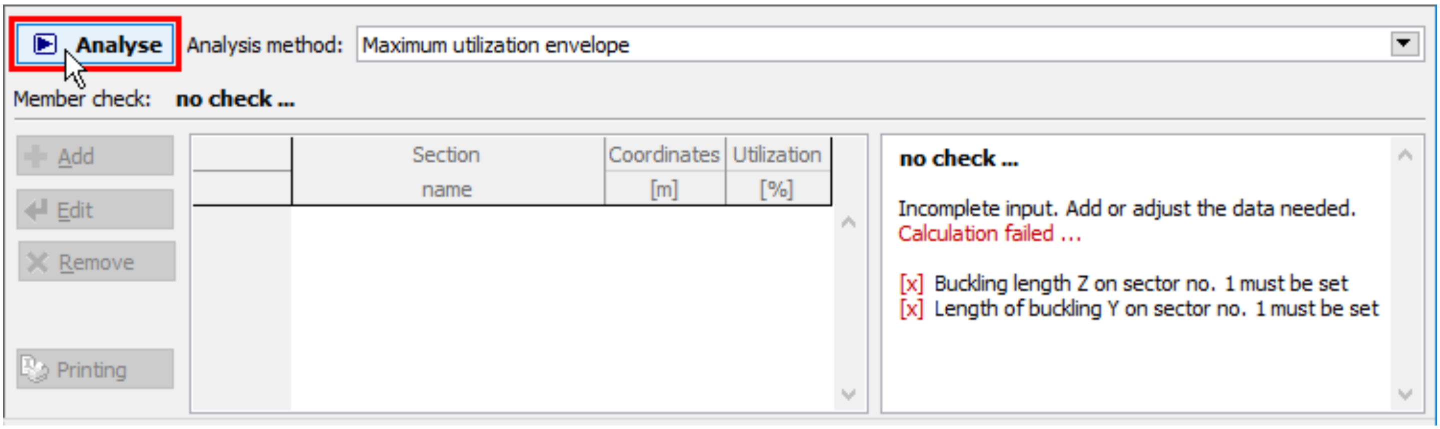
All necessary data (geometry, cross-section, material, internal forces) are transferred into the design module. The design members are organized into the tree menu in the left part of the window.



*Designing module for steel structures*

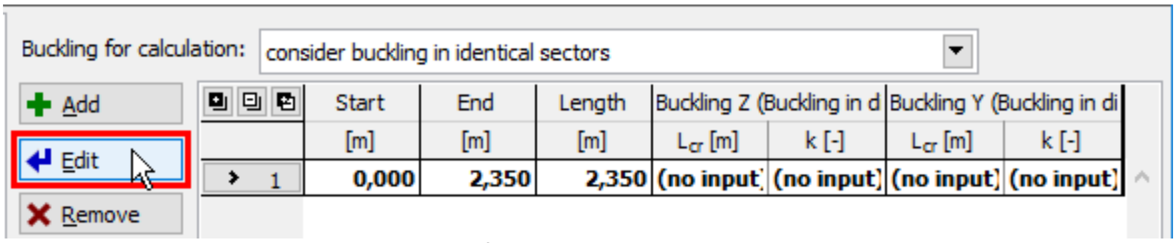
We switch to the mode "**Columns**" "**Check**" and run the verification by the button "**Analyse**". The program shows warnings, that parameters of buckling and lateral torsional buckling were not set and it is not possible to run analysis.



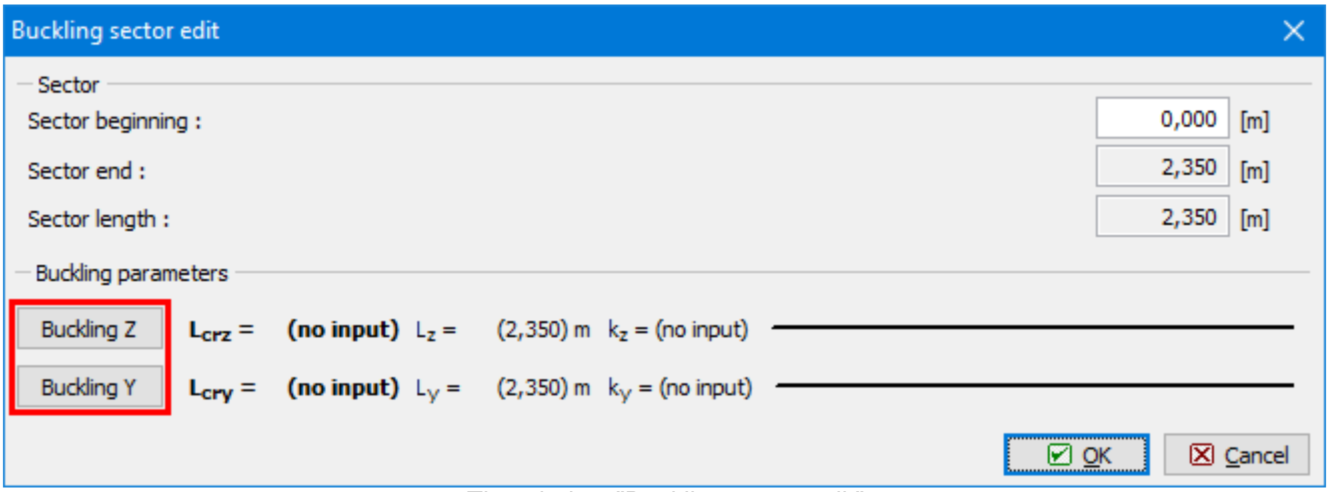

*Button for running the analysis*

We switch to the part "**Buckling**" of the tree menu. The parameters can be defined in the bottom frame. As the parameters are constant for the whole length of the column, we will use only default length sector with analysis properties. We will edit it by the button "**Edit**" in the toolbar on the left side of the bottom frame.




*The button for editing buckling properties*

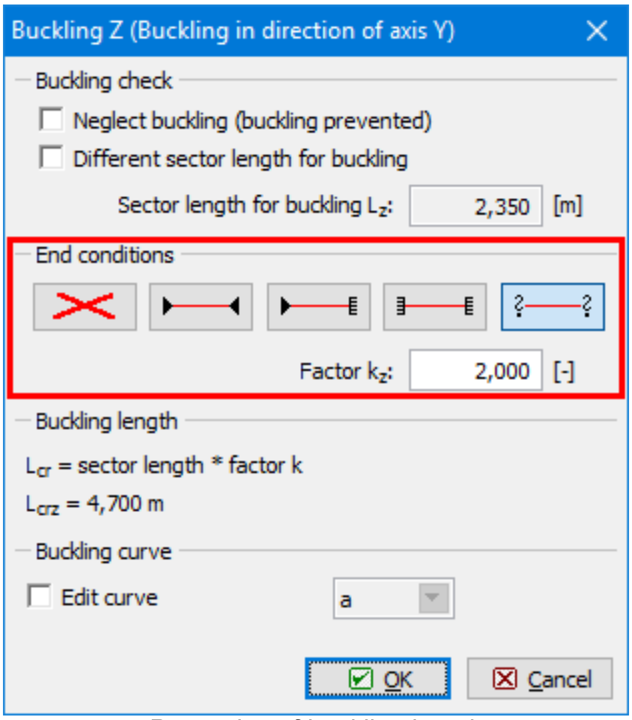
The window "**Buckling sector edit**" contains buckling parameters for directions $z$ and $y$. The buttons "**Buckling z**" and "**Buckling y**" has to be used for the input. "**Buckling z**" contains parameters for buckling in the direction perpendicular to the axis $z$, "**Buckling y**" parameters for buckling perpendicular to the axis $y$.




*The window "Buckling sector edit"*

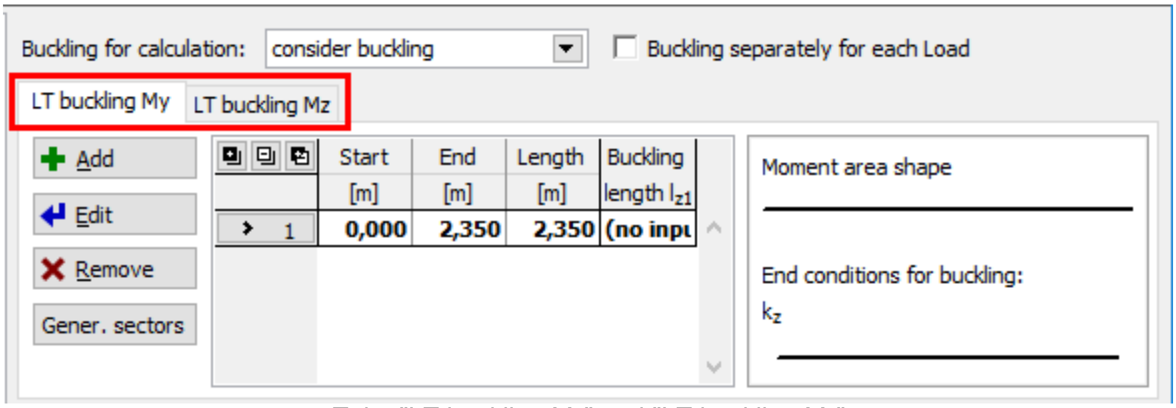
The buckling length has to be specified in the window "**Buckling**". To be on a safety side, we will select general end conditions (the button with question marks at ends) and specify value *2.0* for buckling factors "$\mathbf{k_y}$" and "$\mathbf{k_z}$".




*Properties of buckling length*

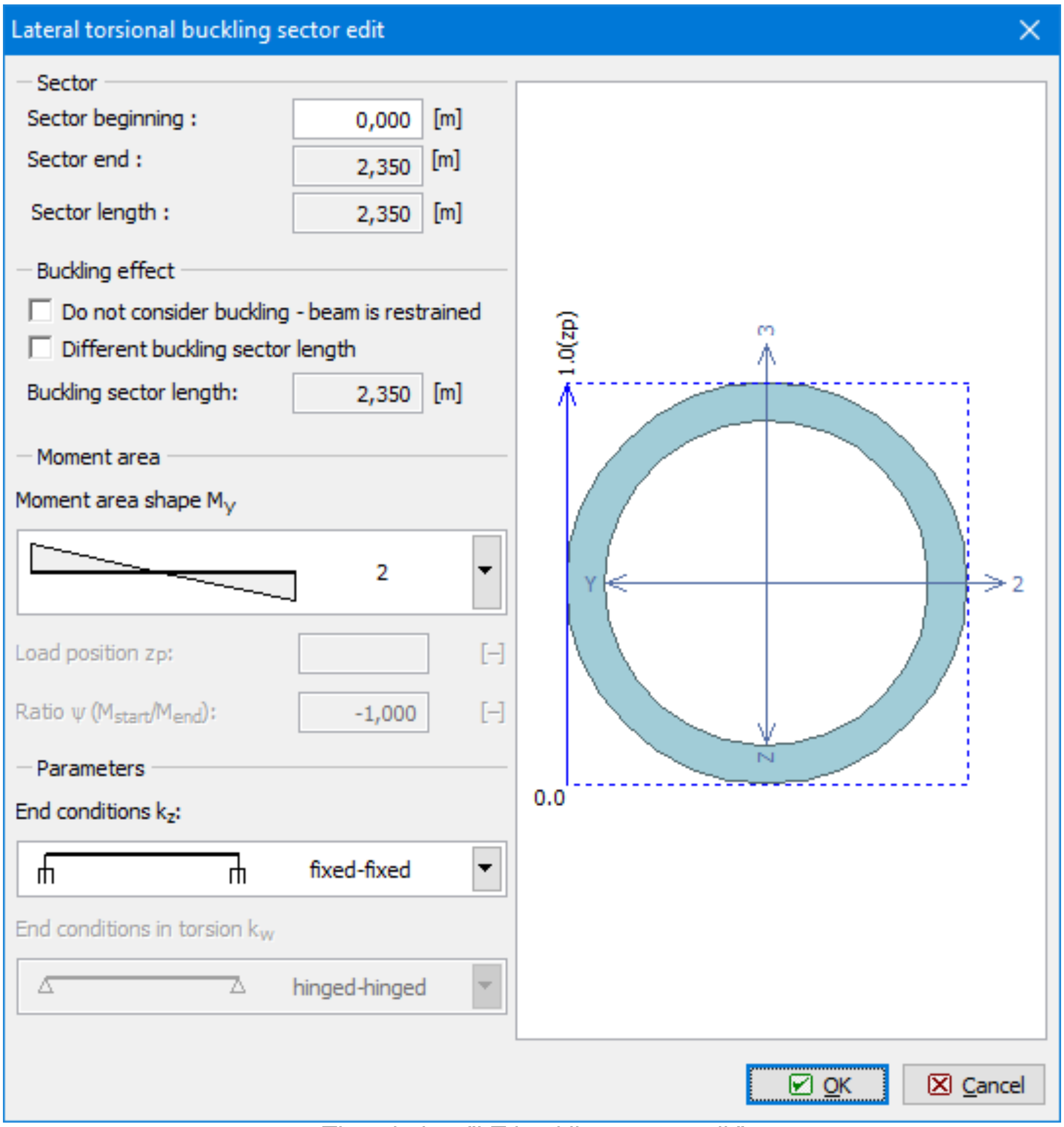
Also shape of bending moment area and end conditions have to be specified for the analysis. We switch to the part "**LT buckling**" of the tree menu. The parameters of lateral torsional buckling are organized into two tabs according to the bending moments $M_y$ and $M_z$.




*Tabs "LT buckling $M_y$" and "LT buckling $M_z$"*

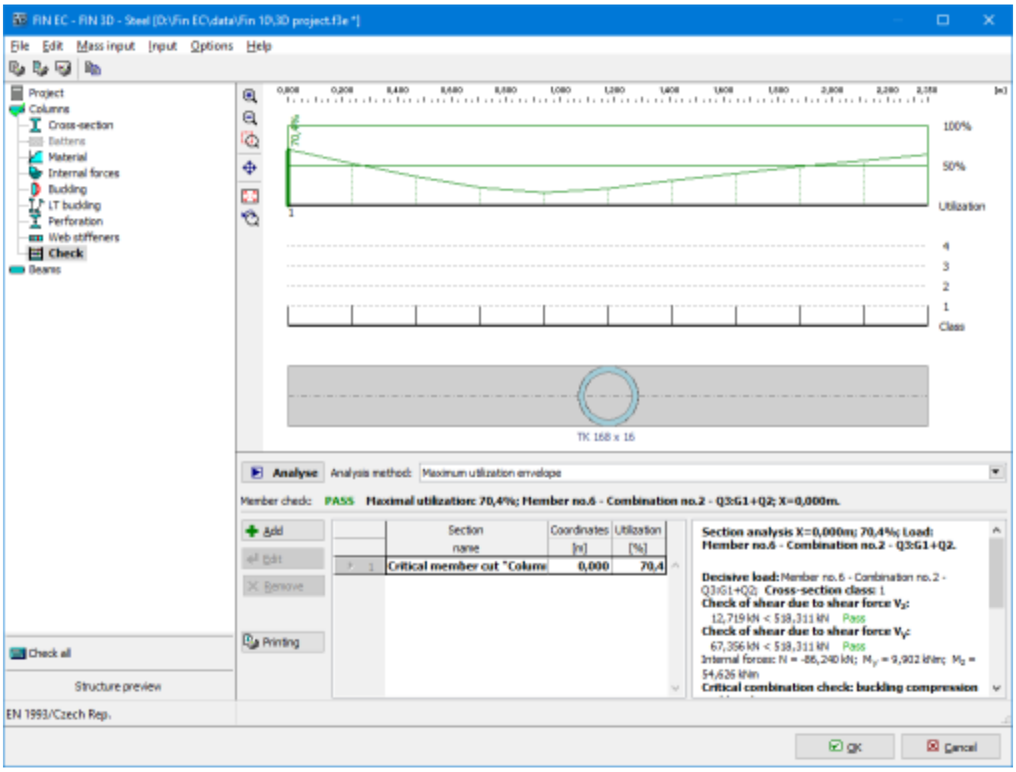
The properties can be edited in the same way as buckling parameters. We specify identical properties for both directions.



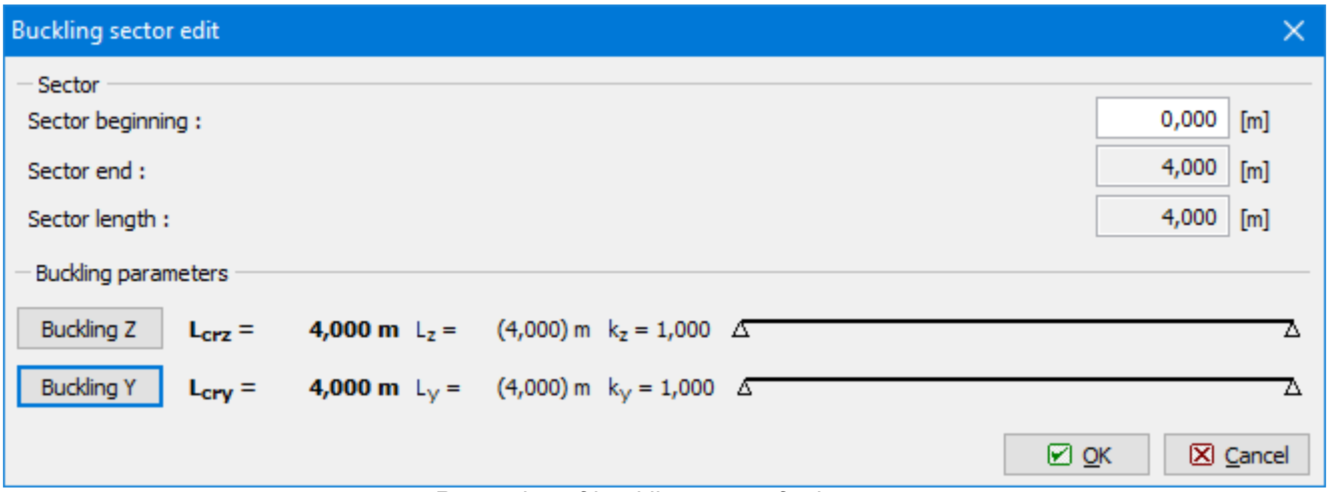
The window "LT buckling sector edit"

We switch back to the mode "**Check**" and run the analysis again. The results show that the member is OK.



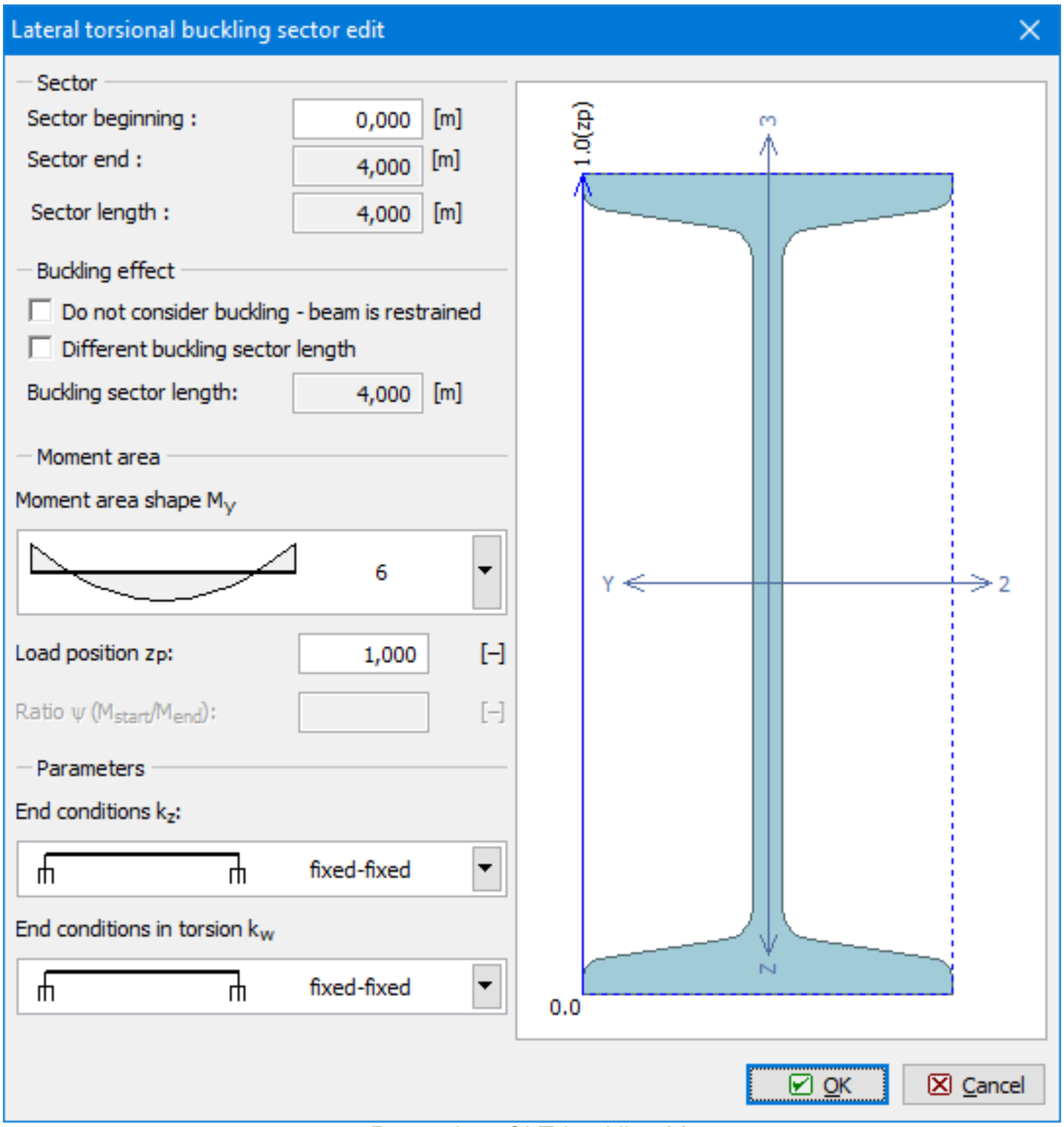
*Results for column*

The verification of beams follows. The buckling parameters will be specified in the same way as we did for column. Difference is, that we will select pinned end conditions with buckling factors "$\mathbf{k_y}$" and "$\mathbf{k_z}$" equal to *1.0*.




*Properties of buckling sector for beams*

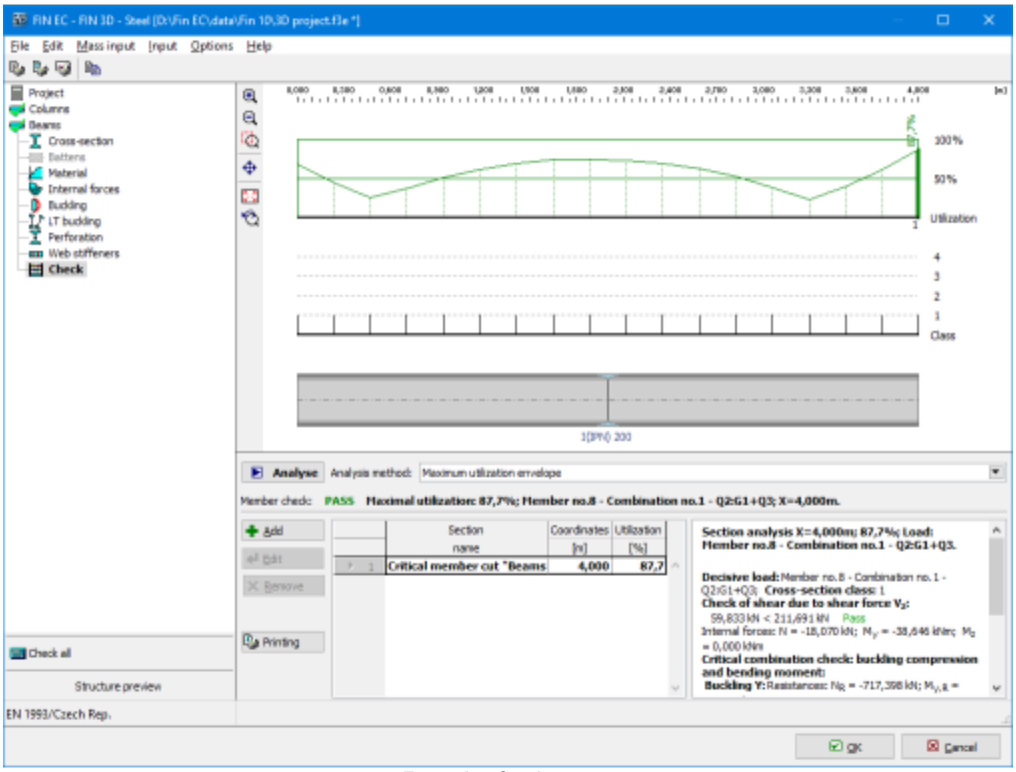
The parameters for lateral torsional buckling induced by the bending moment $M_y$ have to be also specified. The lateral torsional buckling properties are shown in the figure below.




*Properties of LT-buckling $M_y$*

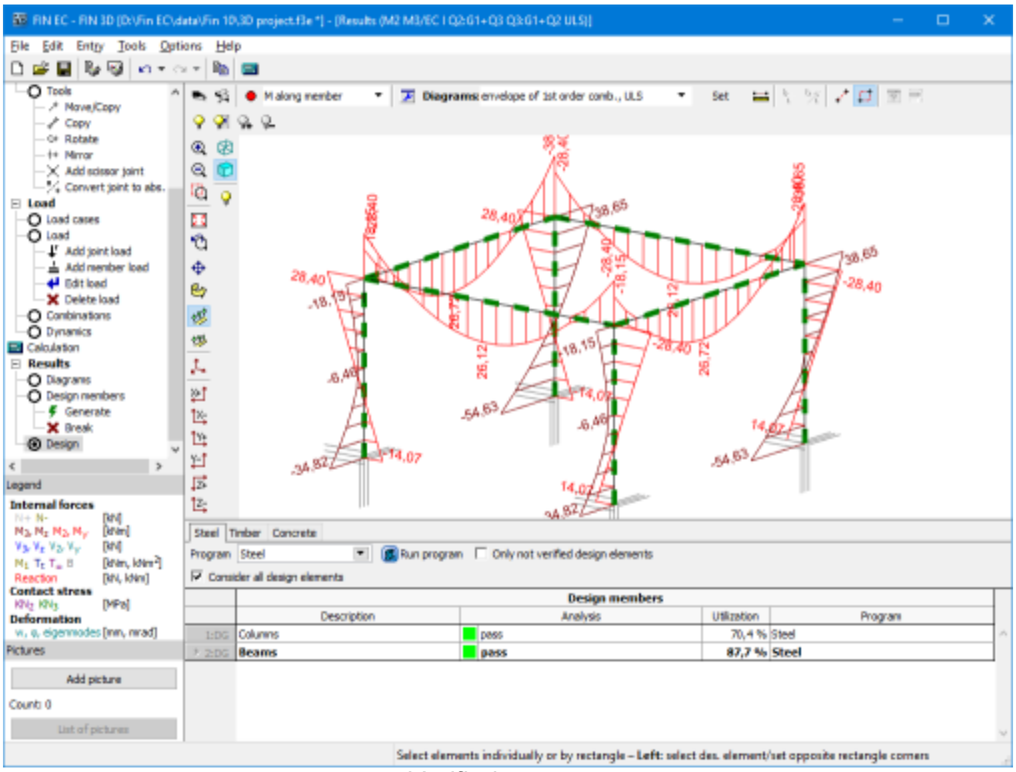
 

The analysis shows that beams have passed the verification.




*Results for beams*

Both design groups are verified. We will close the design module by the button "**OK**" in the right bottom corner and go back to the program "**Fin 3D**". The table in the mode "**Design**" shows the analysis results.




*Verified structure*

# Fire resistance of RC column

## Introduction

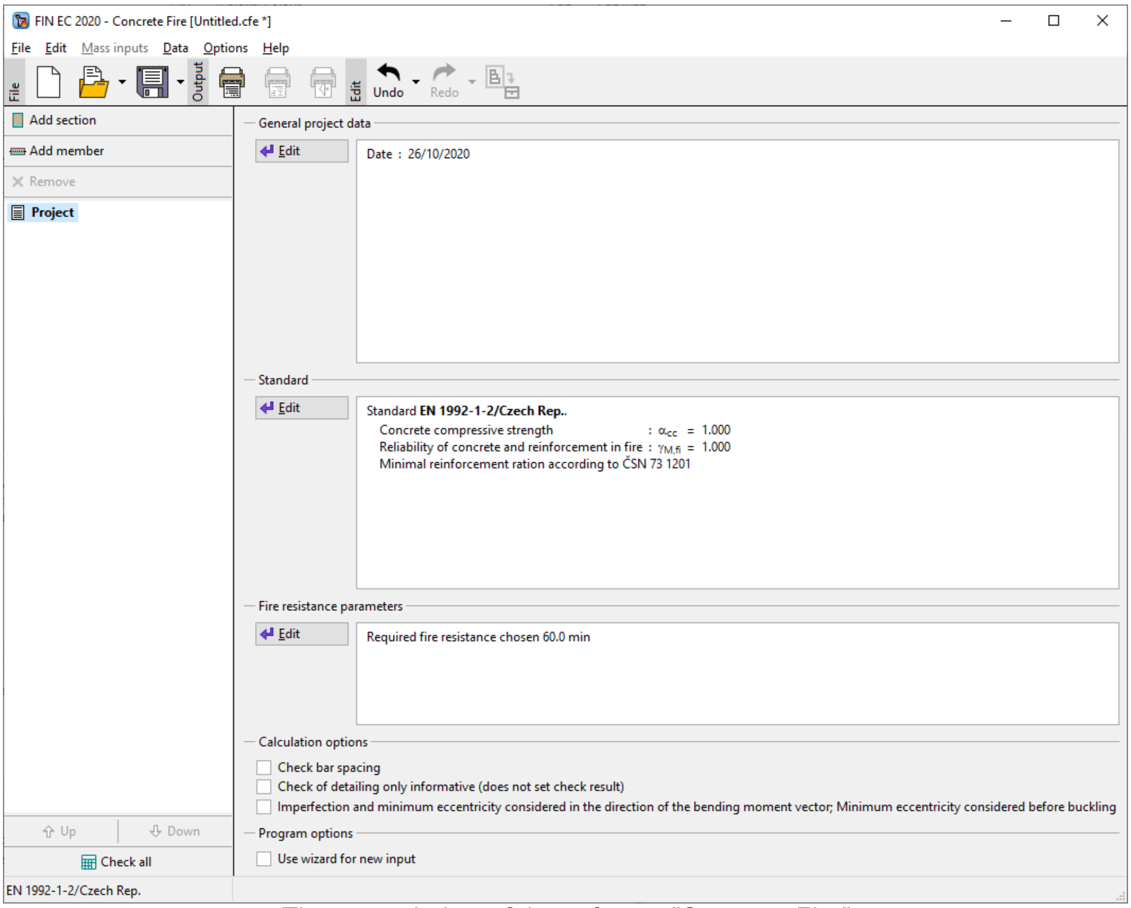
This tutorial shows a design of reinforced concrete (RC) column subjected to fire. The column has rectangular cross-section with dimensions 400x300 mm and length 3000 mm. The column is subjected to axial compressive force, biaxial bending and lateral forces from both directions. The internal forces in the Ultimate Limit State (ULS) are shown in the table below and were calculated based on the basic combination. The column is located inside of an office building and is considered to be rigid at the bottom and hinged at the top. The material of the column is concrete C35/45 and steel reinforcement B550B.

| section [m] | N [kN] | My [kNm] | Mz [kNm] | Vy [kN] | Vz [kN] | combination | η [-] |
|---|---|---|---|---|---|---|---|
| 3 | -685 | 140 | 135 | -70 | -80 | basic | 0.7 |
| 2.25 | -687.25 | 70 | 67.5 | -70 | -80 | basic | 0.7 |
| 1.5 | -689.5 | 0 | 0 | -70 | -80 | basic | 0.7 |
| 0.75 | -691.75 | -70 | -67.5 | -70 | -80 | basic | 0.7 |
| 0 | -694 | -140 | -135 | -70 | -80 | basic | 0.7 |

*The internal forces in the column*

## Starting a new project

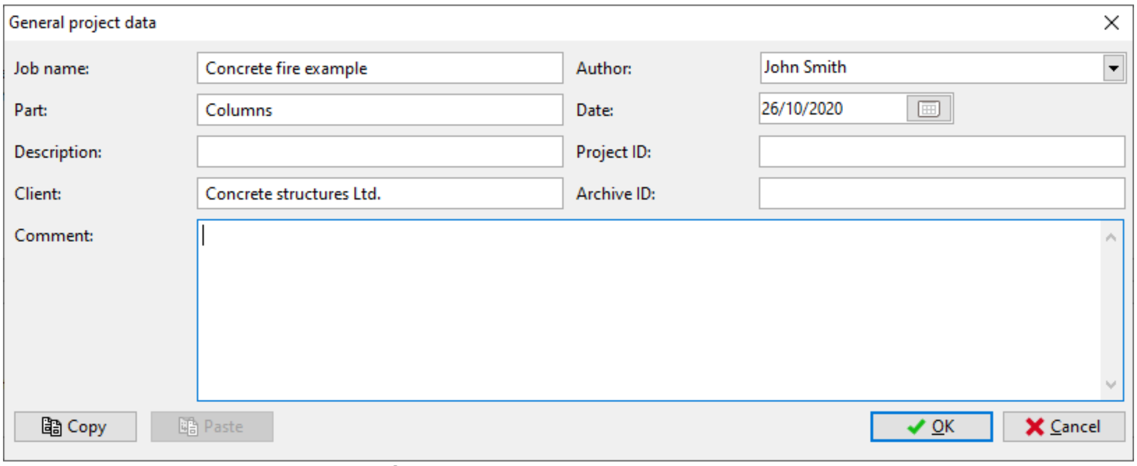
The following screen appears after running the software "**Concrete Fire**":




*The start window of the software "Concrete Fire"*

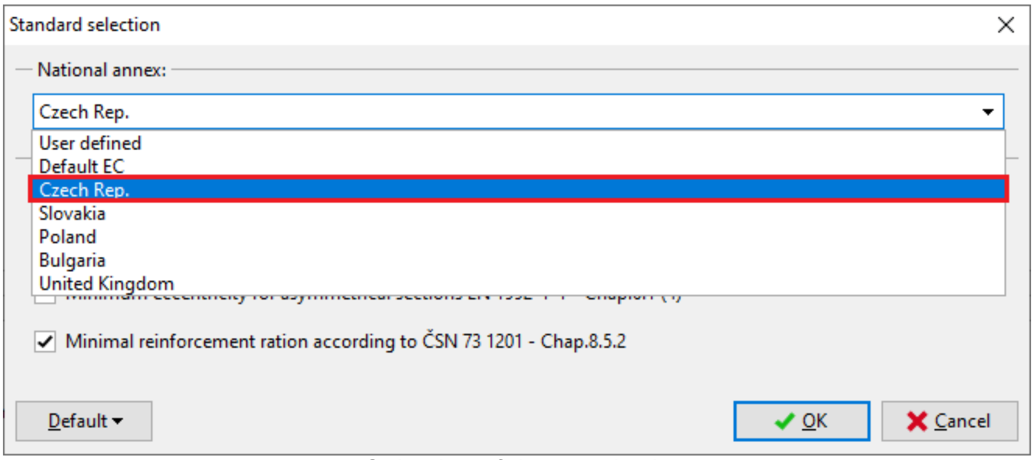
The software allows to asses unlimited number of tasks per one project. The supported tasks are, the assessment of cross section ("**Section**") or the whole member ("**Member**"). The "**Section**" may be used for simple verification of RC cross-sections subjected to fire, where the "**Member**" is usually used for the design of structures created in "**Fin 2D**" and "**Fin 3D**". This example shows the use of "**Section**" analysis. The start screen contains a part "**General project data**", where the job name, description and other project identification data can be entered. After clicking the "**Edit**" button, we first enter the job name and other project details:



*"General project data" dialog box*

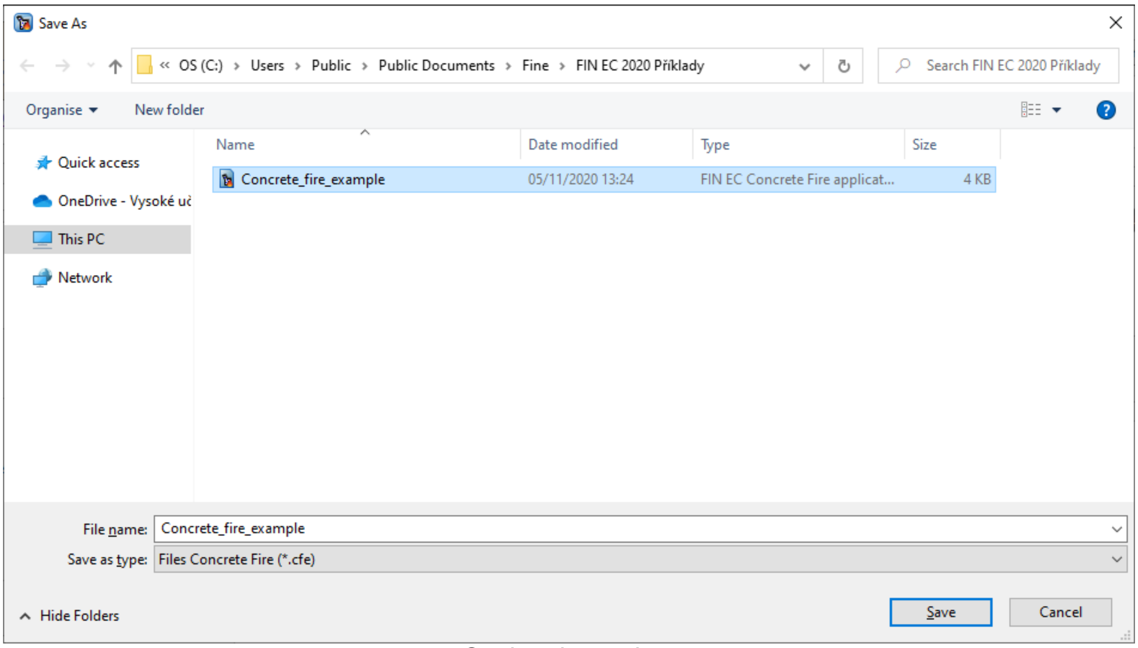
These data is displayed in the header or footer of the final documentation. Next, click the "**Edit**" button in the "**Standard**" frame and select the appropriate national annex. The supported national annexes are listed in the drop-down list. In case, the proper national annex is not listed, it is possible to choose *User defined* and set up custom coefficients.



*Selection of the standards*

Last, the fire resistance properties are set in the "**Fire resistance parameters**" part. This project data can be edited any time during the analysis.

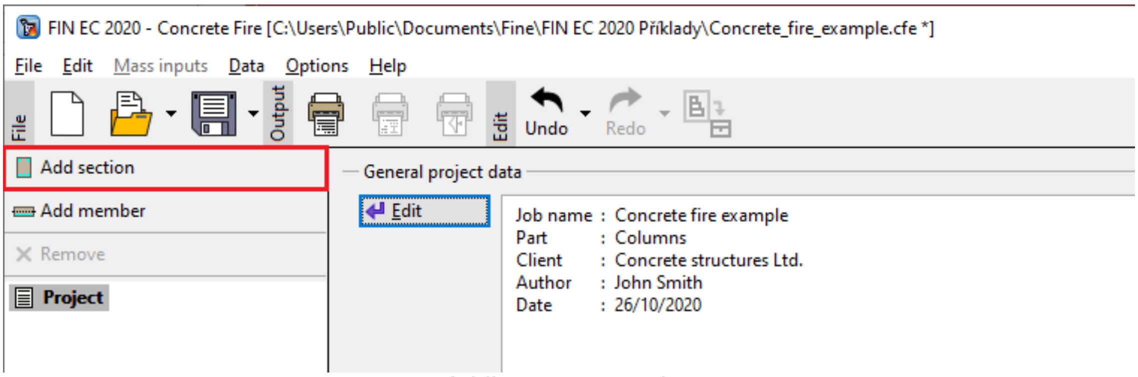
Before any data is assigned to the project, it is highly recommended to save the file. This can be done either using "" button, or in the main menu clicking on "**File**" – "**Save As**", or using the "**Ctrl+S**" shortcut.



*Saving the project*

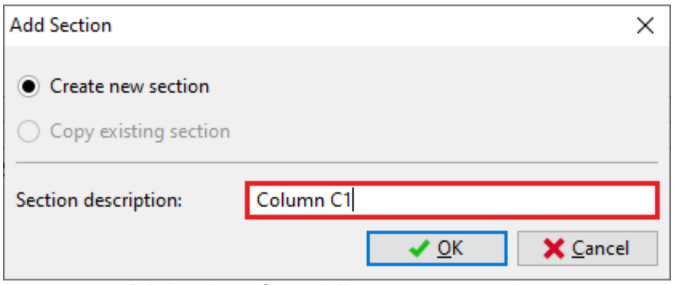
 

Now we can proceed to entering a new task by clicking the "**Add Section**" button in the upper part of the program's tree menu.



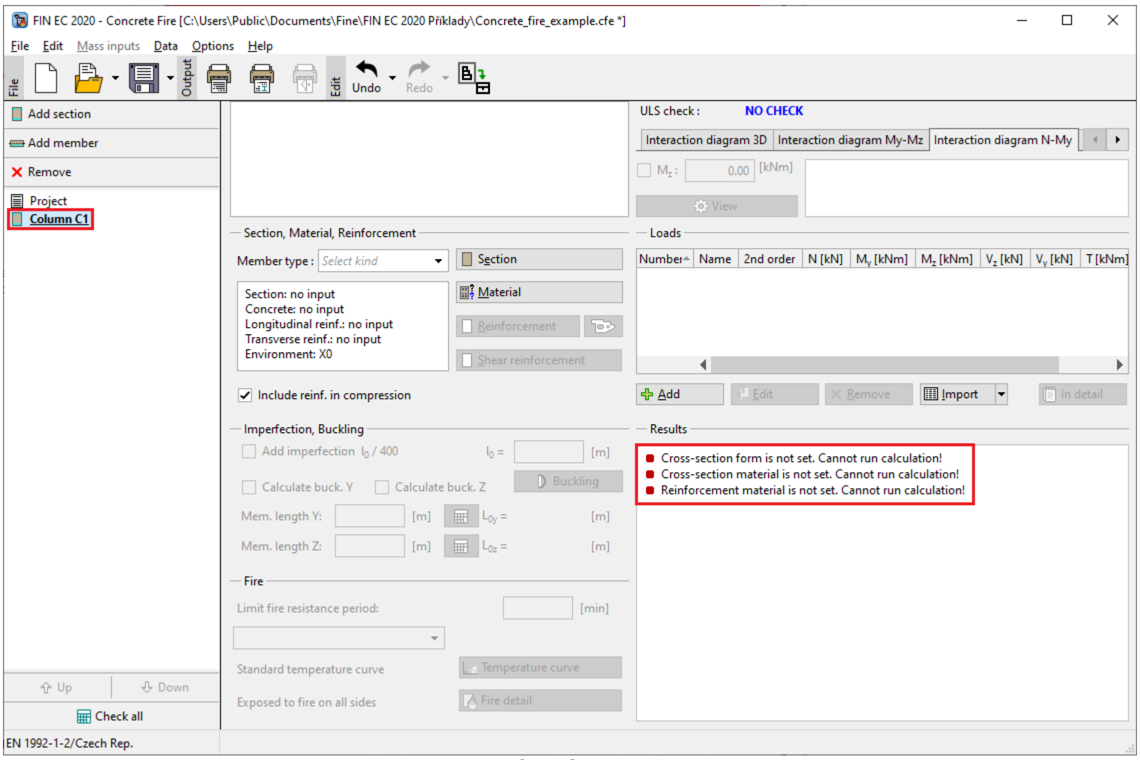

*Adding a new section*

The following dialog box appears, in which we can enter the section's name ("**Column**") into the "**Section description**" field, confirming by clicking the "**OK**" button.




*Dialog box for adding a new section*

The new task "**Column C1**" has been created in the tree menu, every time a new "**Section**" or "**Member**" is created, it will appear in the tree menu. The software has automatically selected the "**Column C1**", therefore it is possible to directly edit this task. In the right bottom corner it shows what is required in order to run the analysis.

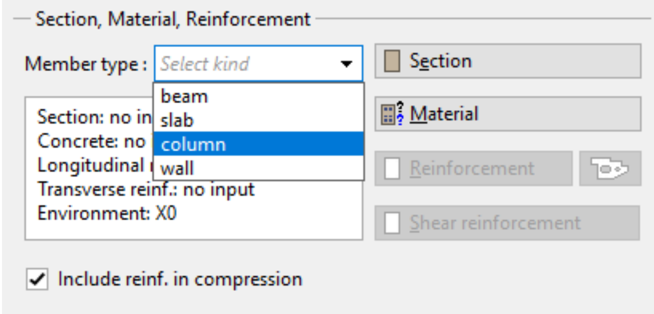



*Main screen for "Section" task type*

## Section, Material, Reinforcement

In the beginning it is necessary to enter the dimensions of the geometrical and material characteristics of the considered RC column in the "**Section**, **Material**, **Reinforcement**" frame. First select the type of the member in "**Member type**" drop-down list. Available types are "**beam**", "**slab**", "**column**" or "**wall**".

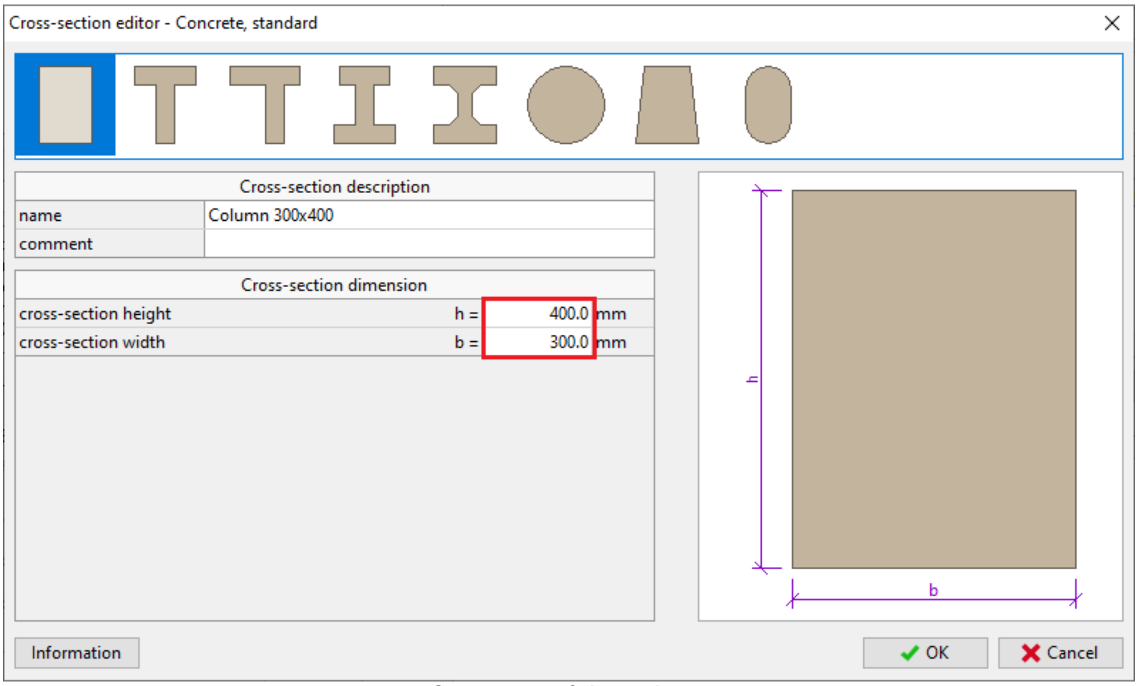
 




*Defining the member type*

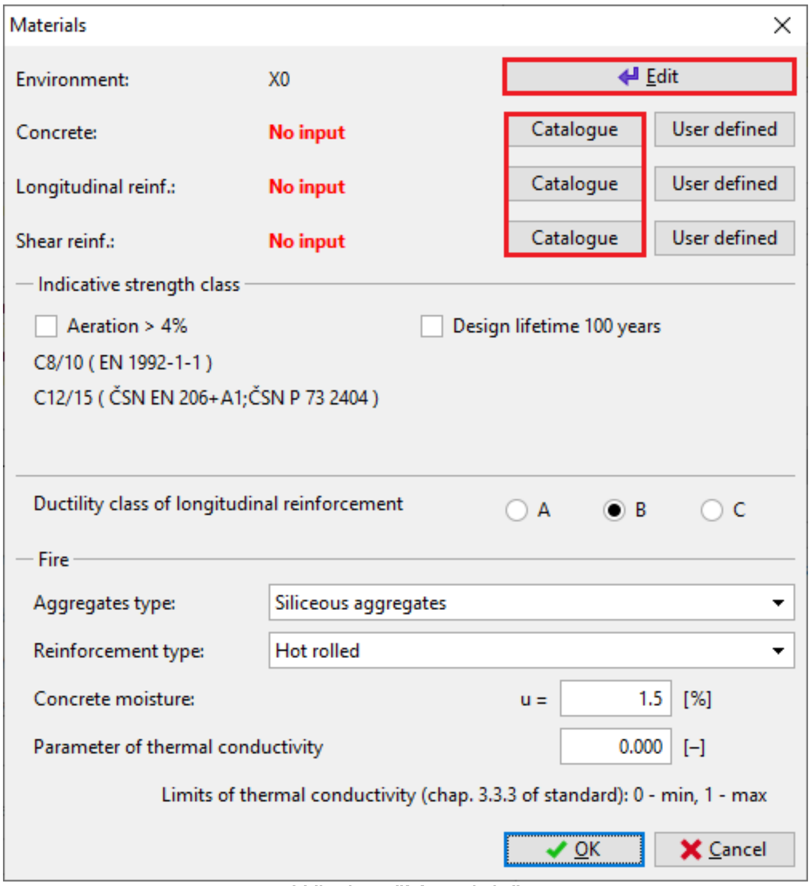
In our example we select member type "**column**". This selection affects the analysis and verifications of the reinforcement arrangement.

By clicking the "**Section**" button, the cross-section can be defined by the first suggested rectangular shape. The dimensions of the cross-section are simply defined by changing it's height and depth. The software allows to use custom user-defined shapes, however for the purpose of this example it is not needed.




*Geometry of the column*

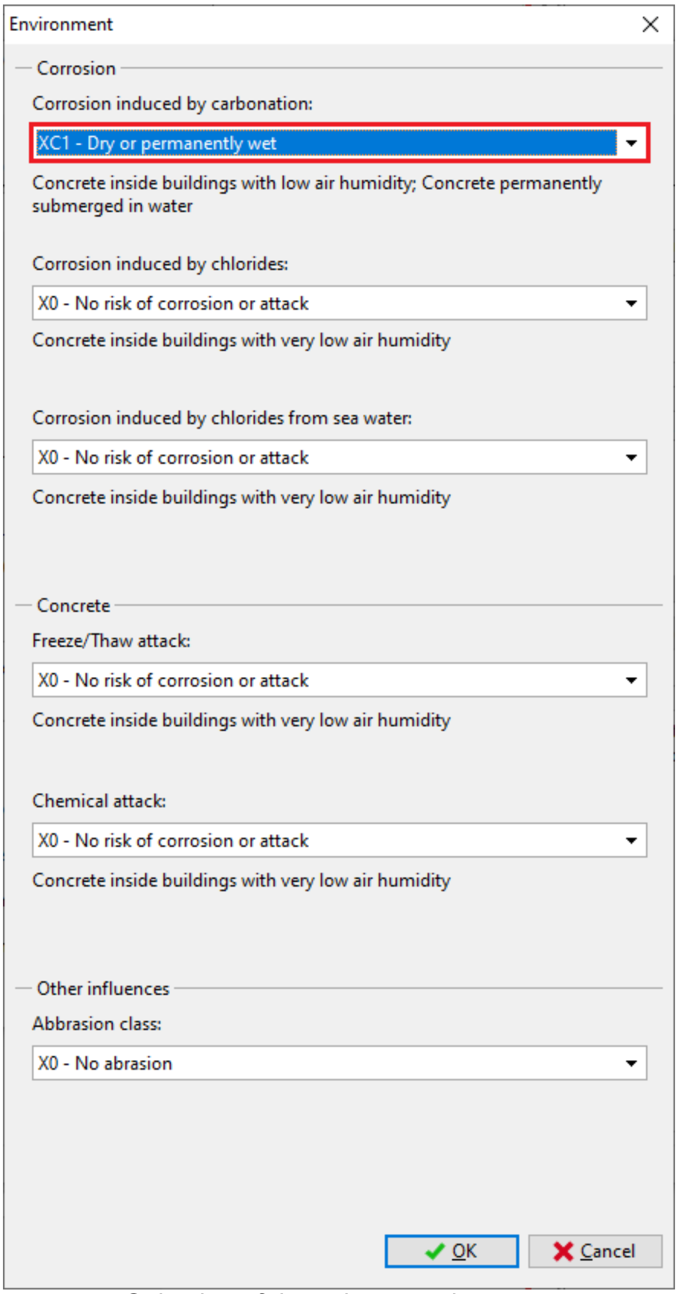
We proceed with defining the material properties in "**Materials**" dialog box which is run by clicking the "**Material**" button in the "**Section**, **Materials**, **Reinforcement**" part. Assuming the column is located inside of the structure we select "**X1**" for "**Exposure class**", as the column is not in contact with outside environment. Subsequently we define the material properties of concrete and longitudinal and transversal reinforcement. We can select standardized materials from the library of pre-defined materials by clicking the "**Catalogue**" button at relevant lines.




*Window "Materials"*

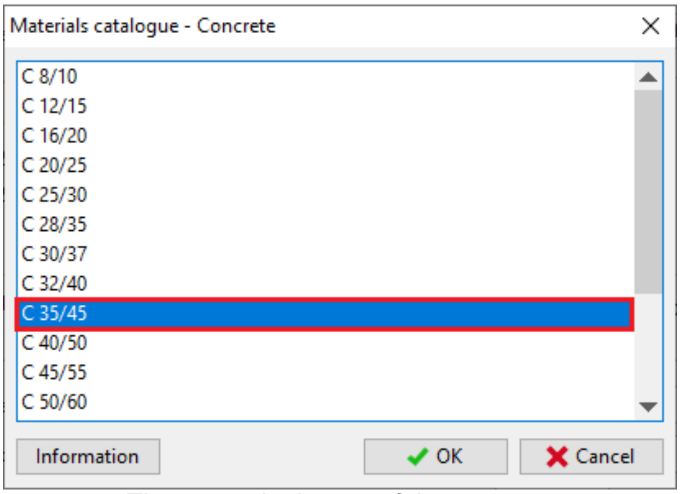
For column environment select the corrosion induced by carbonation "**XC1**" and close the dialog box by clicking "**OK**".




*Selection of the column environment*

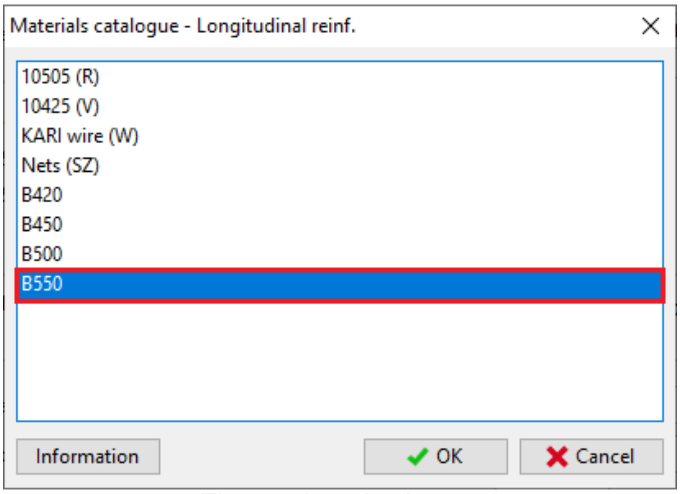
For concrete select the strength class "**C35/45**" and close the dialog box by clicking "**OK**".




*The strength classes of the concrete*

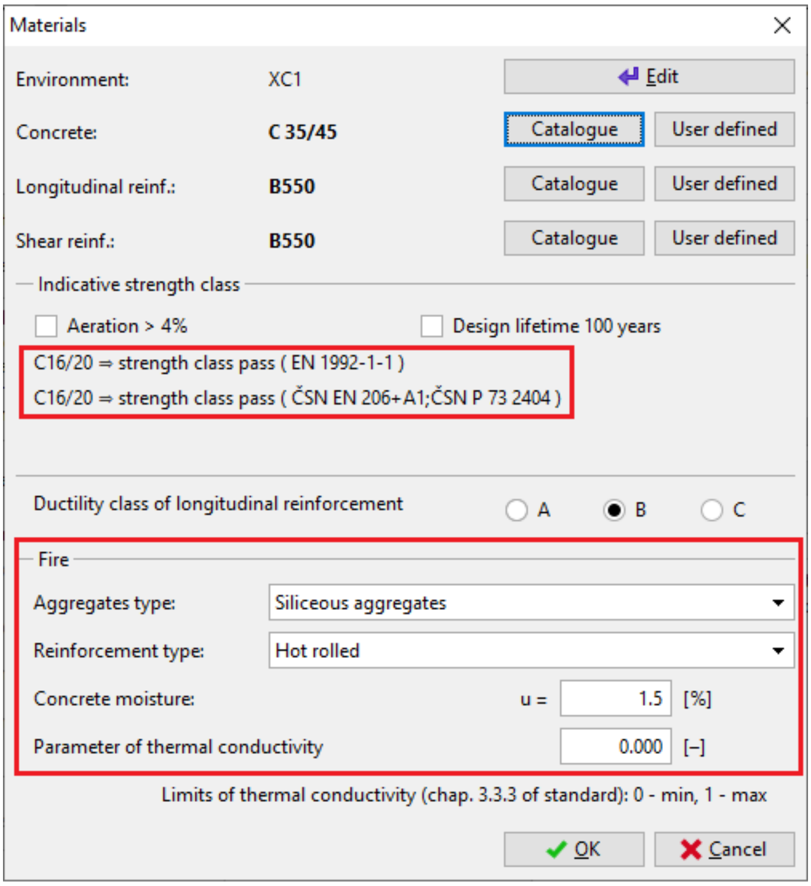
Proceeding to definition of the steel properties, select the grade "**B550**" for both longitudinal and transversal reinforcement and close the dialog box by clicking "**OK**".




*The steel grade classes*

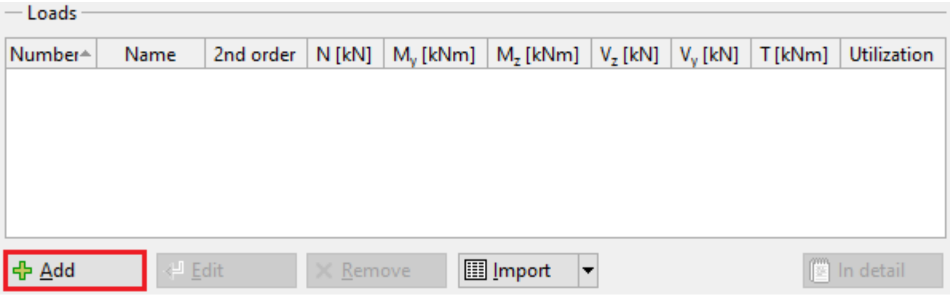
When returning to the "**Materials**" dialog box, the selected materials are checked with the predefined standard and displayed whether the material requirements on the "**Indicative strength class**" are fulfilled. At the bottom of the "**Materials**" dialog box, the material properties influencing the fire resistance properties can be changed. Finally, exit the window by clicking "**OK**".




*Indicative strength class check in "Materials" window*

## Loads

After defining the geometry of the section and material properties, either the loads or the reinforcement may be defined. In this example, the load is defined first, then the reinforcement can be assessed immediately during its definition. The load case is created by clicking the "**Add**" button located under the "**Loads**" table.




*Inserting the new load*

In the "**New load**" window select a "**Combination type**". This type should be selected according to the type of combination, which was used for determination of the forces and moments. This input affects the type of verification. The following options are available:

**Basic design (ULS)**

- Forces and bending moments have been obtained from the basic combination for persistent and transient design situations according to EN 1990, equations ***6.10*** resp. ***6.10a*** and ***6.10b*** . These loads are used for basic assessment of cross-section's capacity in the ultimate limit state.

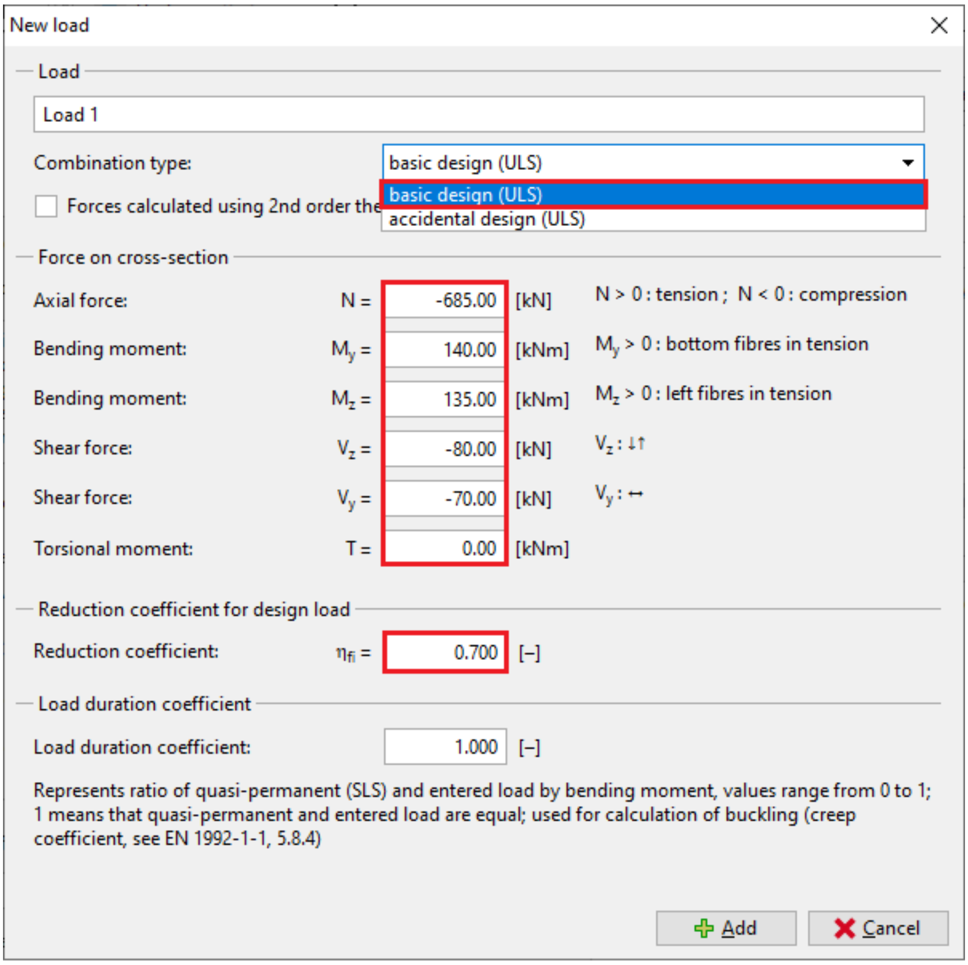
**Accidental design (ULS)**

- Forces and bending moments have been obtained from the combination for accidental design situations according to EN 1990, equation ***6.11*** . These loads are used for assessment of cross-section's capacity in accidental design situations in the ultimate limit state (partial safety and material factors for accidental design situations are used).

In this example, the internal forces were calculated based on basic design combination, therefore pick the "**Basic design (ULS)**". Now in the dialog window add the internal forces on section 3m according to the table in the introduction. The axial force is $N = -685 kN$ (negative value denotes compression), the bending moments are $M_y = 140 Nm$, $M_z = 135 kNm$ *and the lateral forces are* $V_z = -80 kN$ *and* $V_y = -70 kN$ (negative value denotes they act in the same direction as the moments). Next, define "**Reduction coefficient**", the procedure to determine this coefficient is in the chapter 2.4.2 of the standard EN 1992-1-2. As a simplification, the standard suggests to use $\eta_{fi}$ with value of *0,7*.

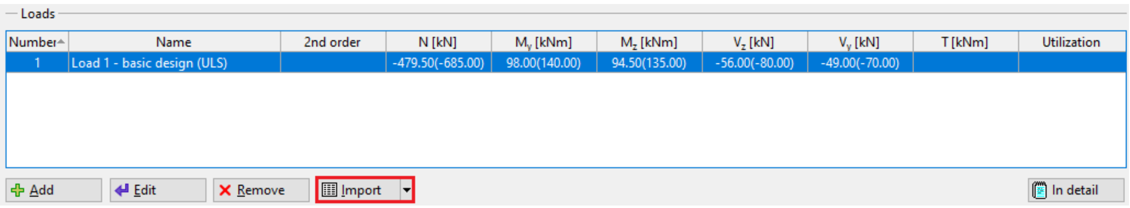
The "**Load duration coefficient**", i.e. the ratio of quasi-permanent and total loads for calculation of the creep coefficient. If the exact value is not available, the conservative value is *1.00*, which means that the total load is considered quasi-permanent in the calculations. The new load is confirmed by clicking the "**Add**" button and "**Cancel**" to exit the dialog box.




*Defining the loads*

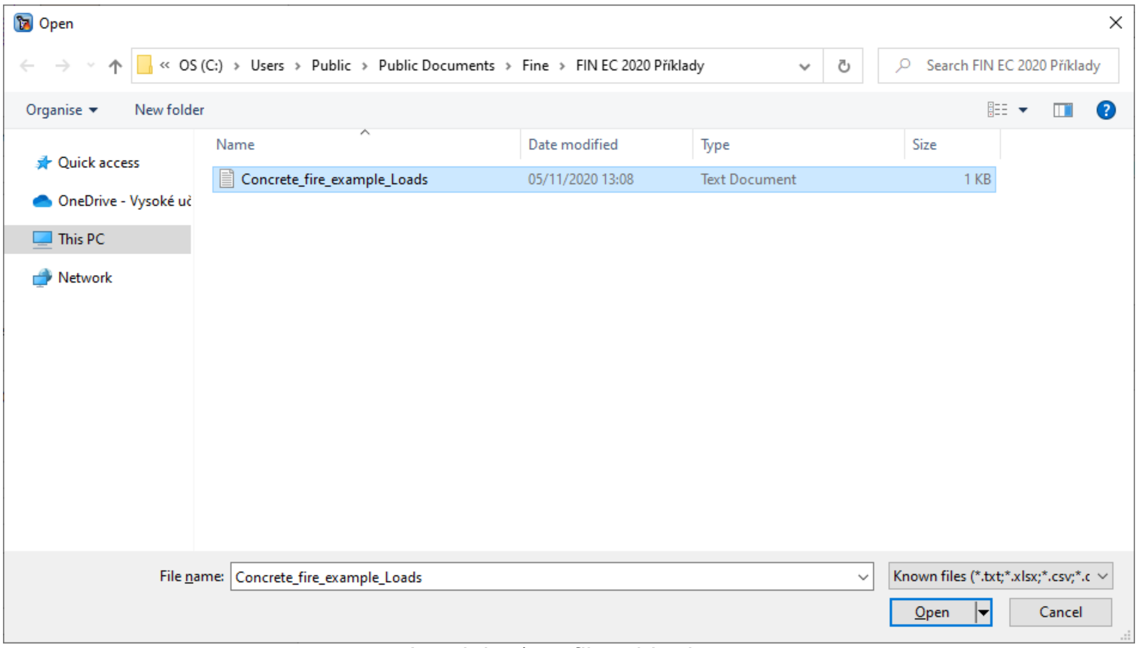
As a result, the table summarizing all defined load cases is generated in the dialog box.




*Overview of the loads*

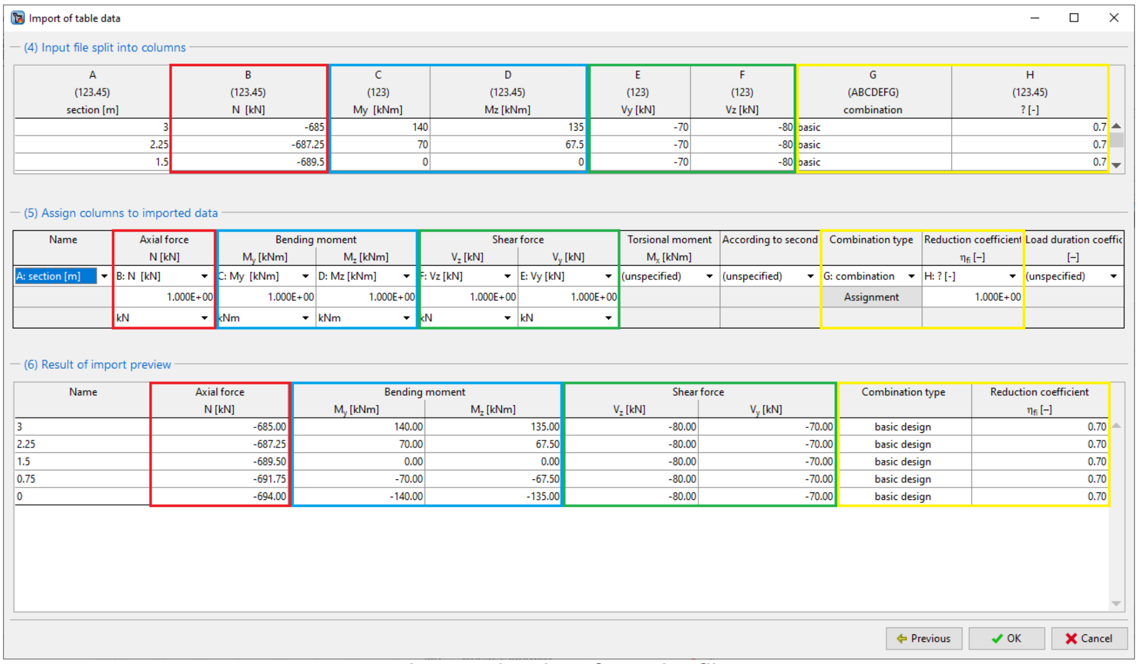
However, adding all the load cases one by one is quite time consuming, therefore the load cases can be imported in a batch using **.txt, *.xslx, *.csv* file (button "**Import**"). The number of loads is not limited in the software.



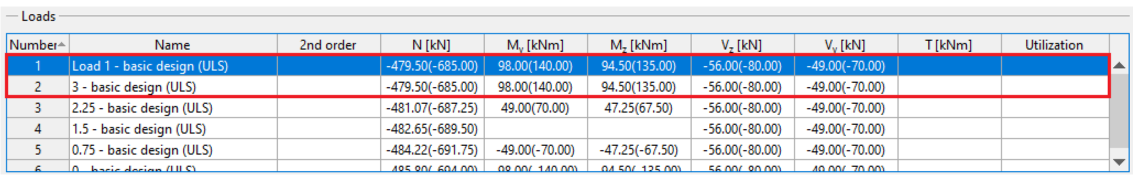

*Load the *.txt file table data*

The dialog box "**Import of table data**" will appear after loading the source file. In this dialog box it is possible to align the data from the source file to the group of data, that are recognized by the software. When the data is formatted in the same manner as in the table in the introduction chapter, the dialog box "**Import of table data**" will look similar as the picture below. Next, it is necessary to align the columns in the source file (4) with the columns of the imported data (5). Check the imported data in the result of import preview (6) and make sure the imported data are aligned with the source file.




*Import the data from the file*

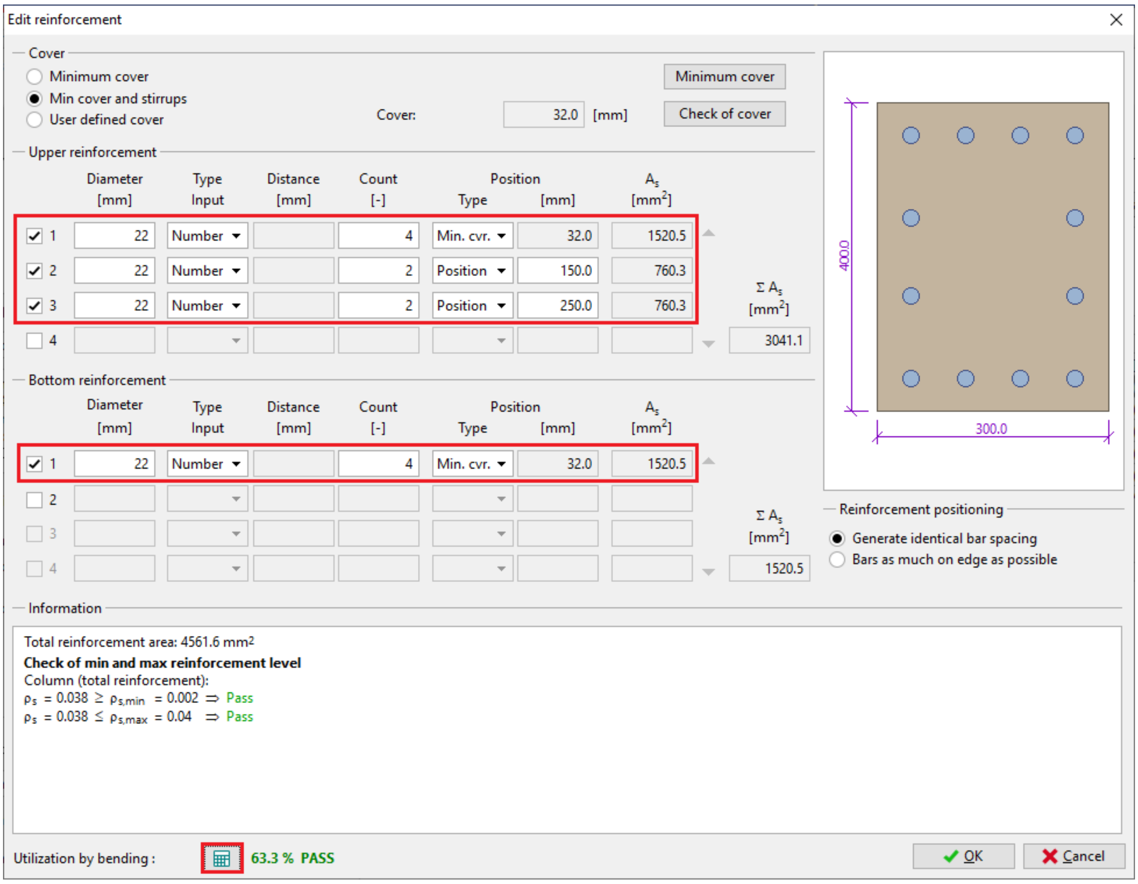
When clicking the button "**OK**" the load cases will appear in the dialog box "**Loads**". If the import of the load cases was done correctly, the first imported load case should match with the load case defined manually.




*Overview of the loads with after the data import*

## Longitudinal reinforcement

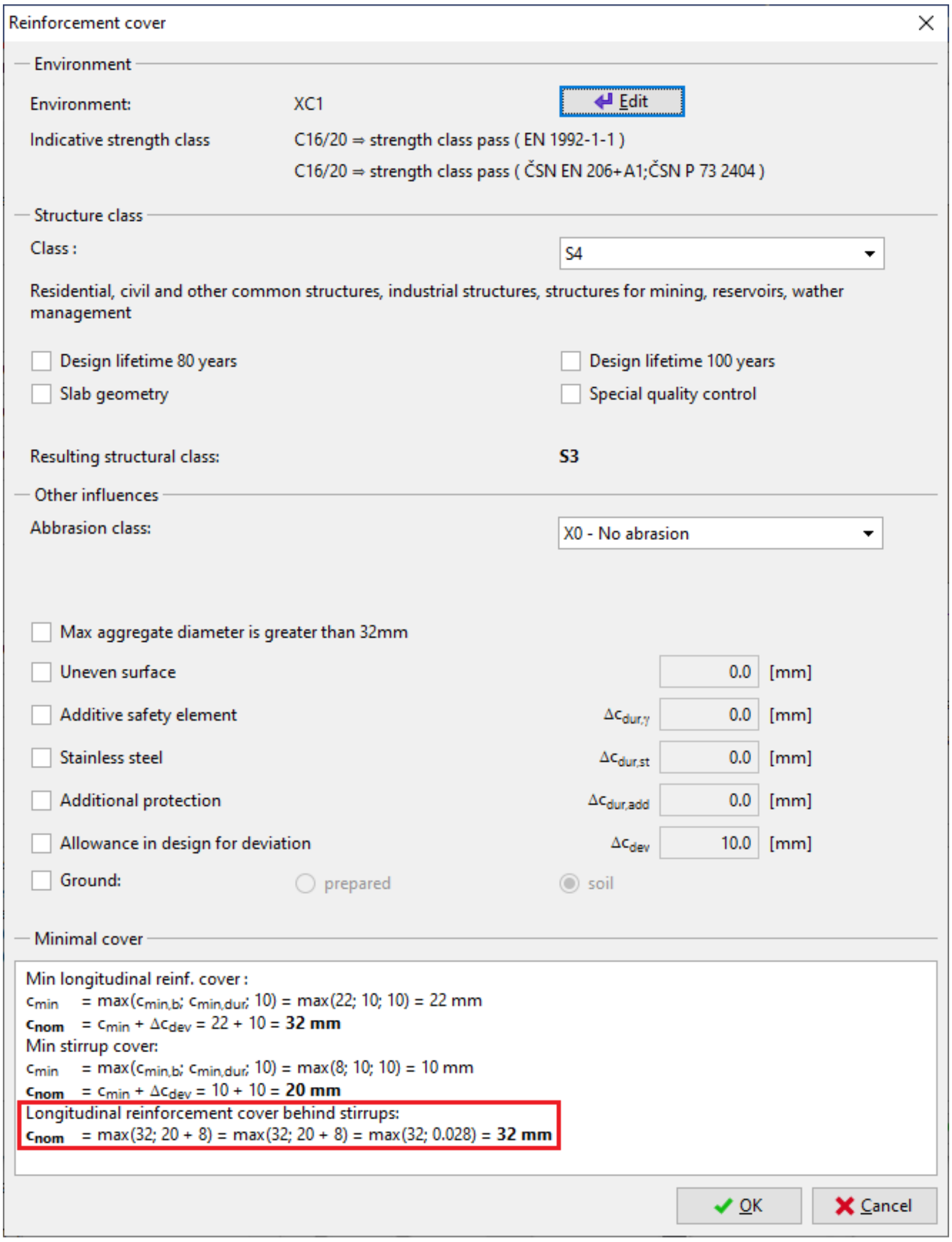
When returning to the main dialog box, the longitudinal reinforcement is defined in the "**Edit reinforcement**" dialog that is accessed by the "**Reinforcement**" button in the "**Section**, **Material**, **Reinforcement**" frame. The upper part of the window allows to select calculation method for the cover. Keep the predefined method for now.




*Defining the longitudinal reinforcement*

When the reinforcement is defined, click "  " button and the software will check whether the cross-section has sufficient capacity in bending. It shows that the defined cross-section passes the design criteria with *63.3%* utilization in bending. In the section "**Information on reinforcement**" it shows that the detailing requirements given by the code are satisfied. Next, check the minimum cover requirements by clicking the "**Minimum cover**" button:



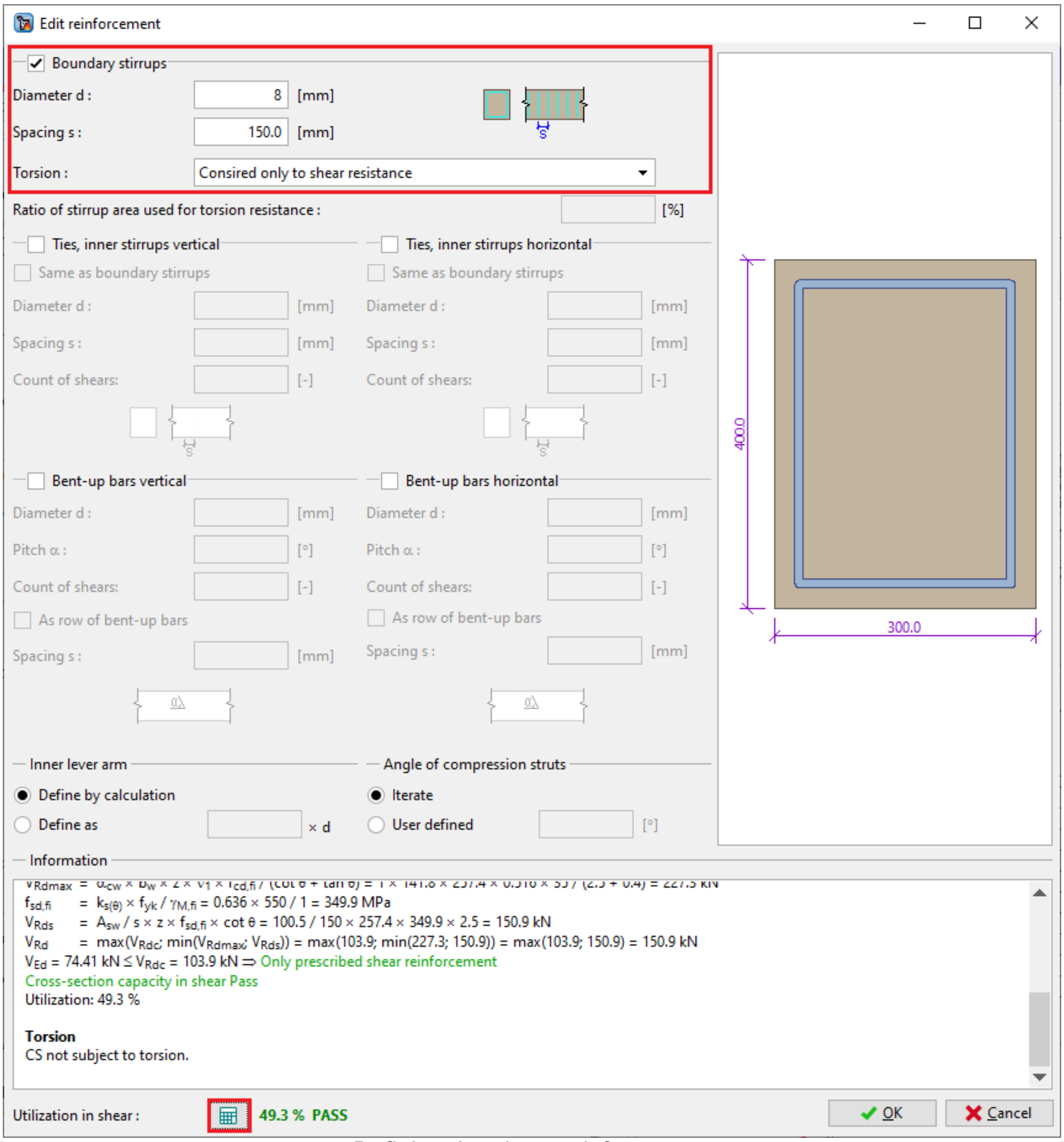

*The minimal reinforcement cover*

The minimum required cover may change, when the stirrups are defined. Therefore, you should always look into this dialog box, after the stirrups are defined.

## Shear reinforcement

The shear reinforcement is defined by clicking the "**Shear reinforcement**" button. The column is subjected to lateral forces, therefore the calculation for the shear resistance must be made. Define the shear reinforcement and click "  " button, the software will check whether the cross-section has sufficient capacity in shear. It shows that the defined cross-section passes the design criteria with *49.3%* utilization in shear.

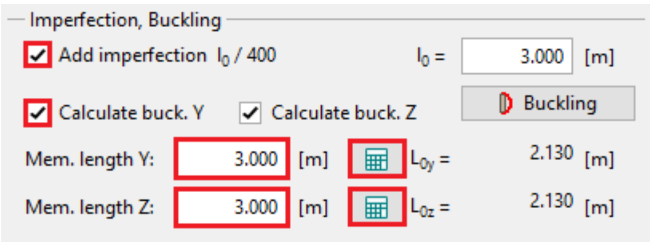
 



*Defining the shear reinforcement*

## Buckling

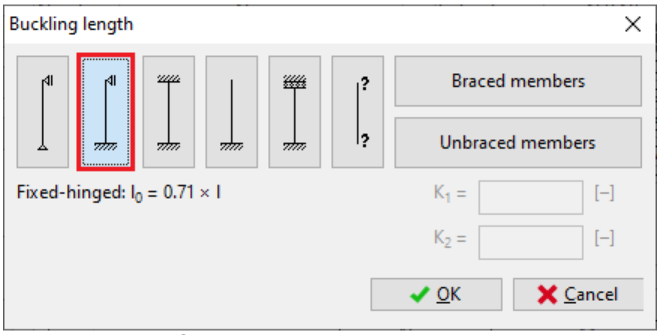
The next step is defining the buckling parameters. First tick "**Calculate buck. Y/Z**" boxes followed by entering the nominal lengths of the column for both directions, based on which the effective buckling lengths will be calculated. For a column rigid at the bottom and simply supported at the top, the effective buckling length is reduced.



*Defining buckling parameters*

Define the boundary conditions by clicking the "  " buttons for each direction.

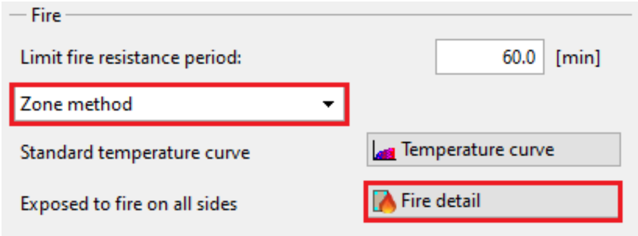
 



*Setting the buckling length*

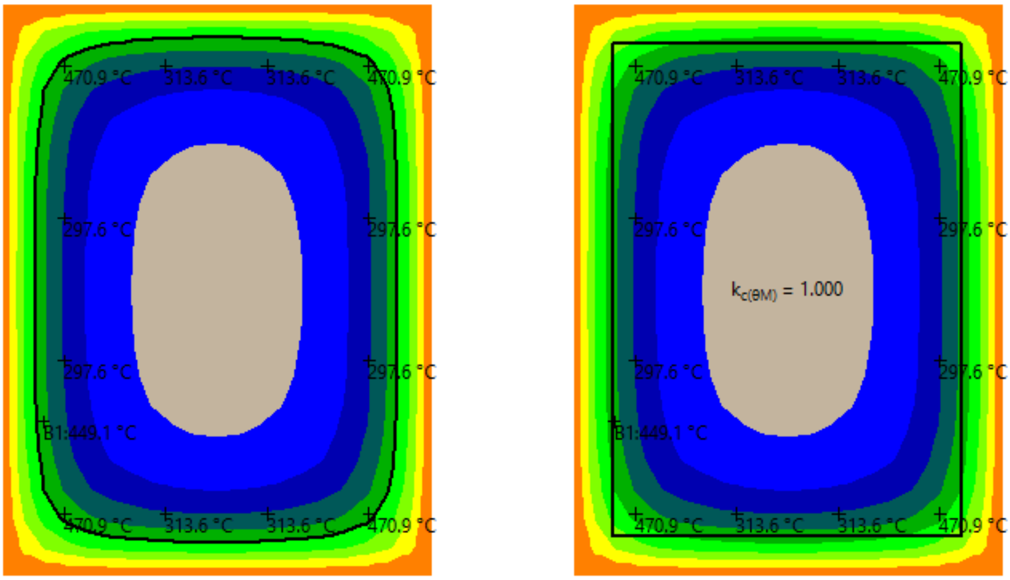
## Fire

Last, the parameters for the fire assessment are defined at the bottom of the main dialog box in the "**Fire**" frame. The software allows to do the calculations based on the "**500°C isotherm method**" or the "**Zone method**". For the column it is recommended to use the "**Zone method**".



*Fire calculation parameters*

The picture of the temperature distribution is shown at the top of the main dialog box, under the "**Temperature distribution**" bookmark.



*500°C isotherm method (left) and Zone method (right)*

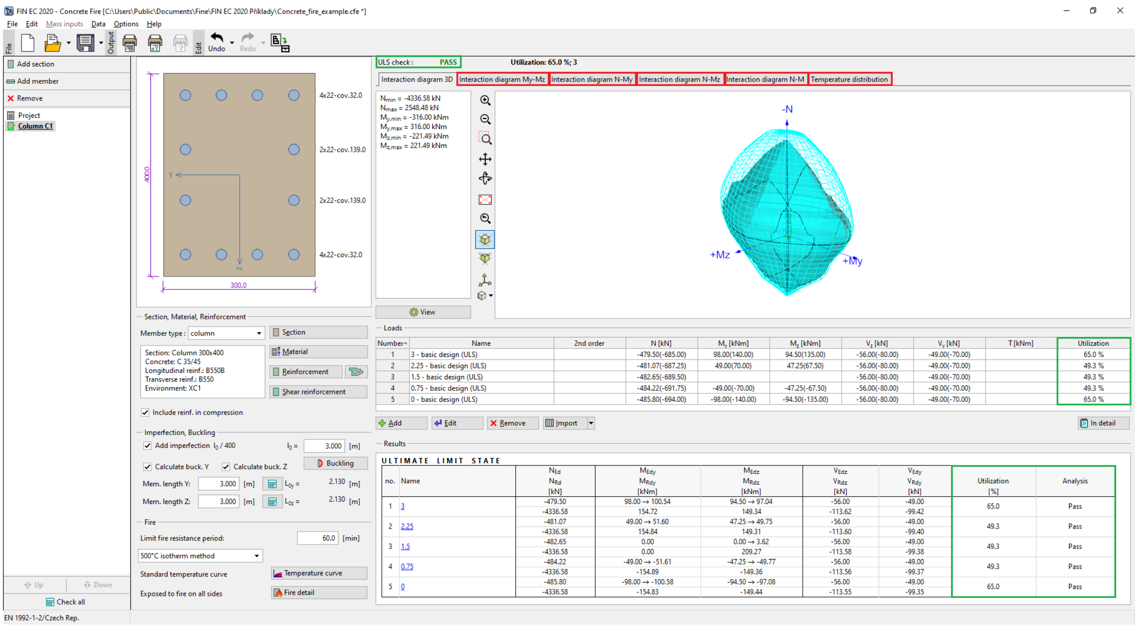
The position, where the fire is acting on the column is displayed in the "**Fire detail button**". The column is considered to stand alone, thus select the first fire detail.


*Fire detail*

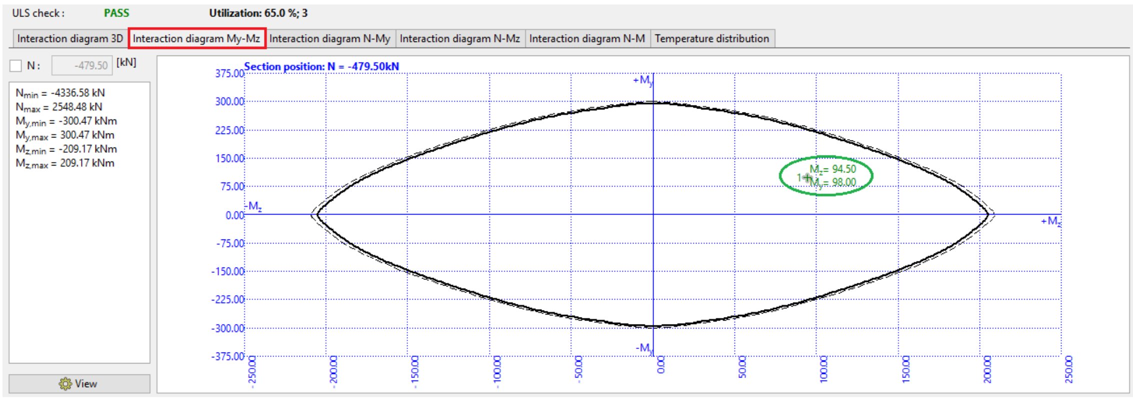
## Results

When all the input has been defined, the results are shown in the main dialog box. If the defined element satisfies the software shows "**PASS**" at the top of the main dialog box.



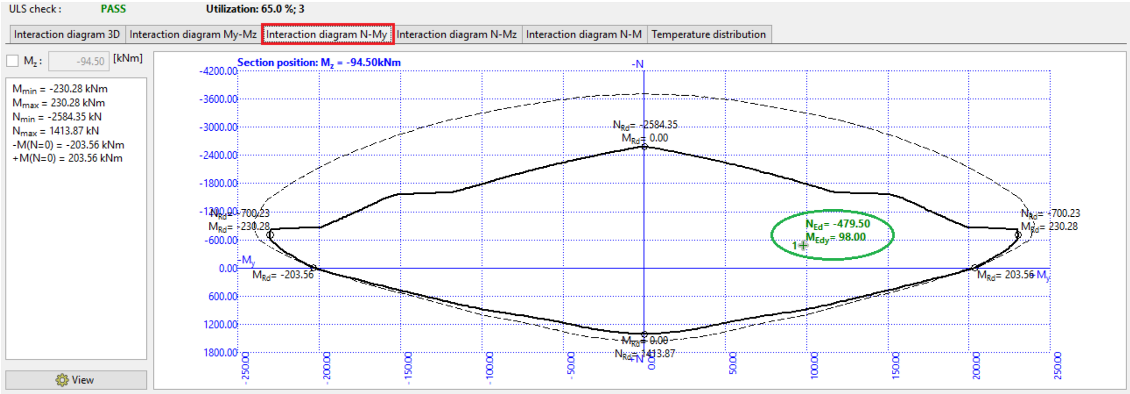
*The main dialog box with the 3D representation of the interaction diagram*

The results may be shown in terms of interaction diagram in the bookmarks at the top of the main dialogue box. The defined loads are shown as points in the interaction diagram. If the load lies inside of the solid line of the diagram, the cross-section satisfies the fire assessment. The dashed line shows the iteration diagram of the column without buckling.

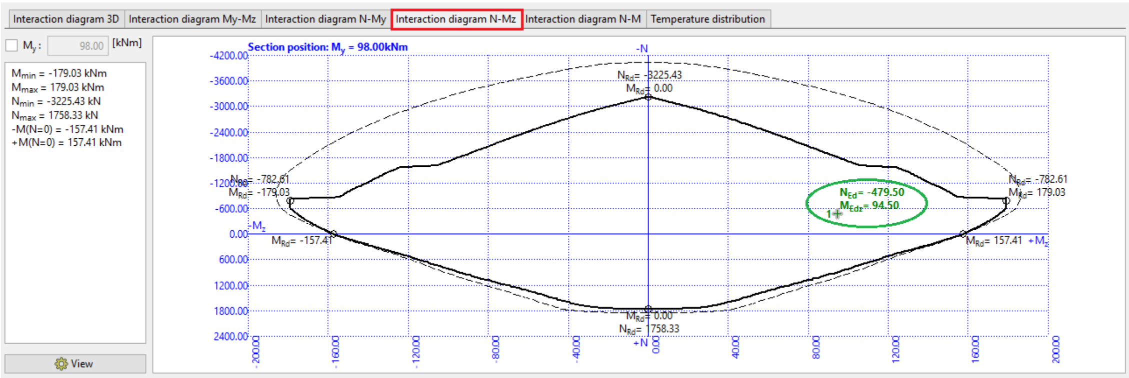



*The $M_y$ - $M_z$ interaction diagram*

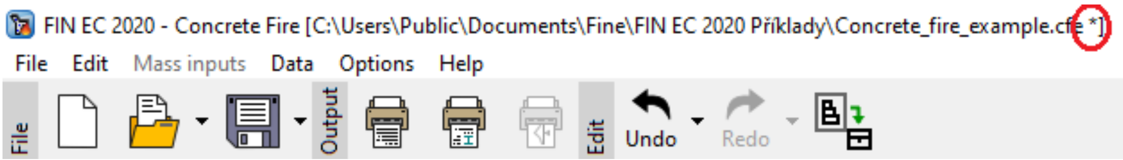



*The N - $M_y$ interaction diagram*




*The N - $M_z$ interaction diagram*

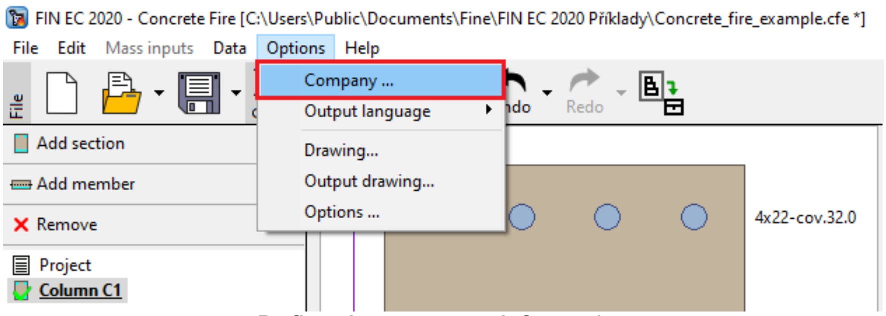
When the project is finished, it is highly recommended to save the file by clicking "" button, or in the main menu clicking on "**File**" – "**Save As**", or using the "**Ctrl+S**" shortcut. The indication of not saved state is "*" in the program window header. In such case it is advisable to save the state.




*Indication of the not saved state*

## Outputs

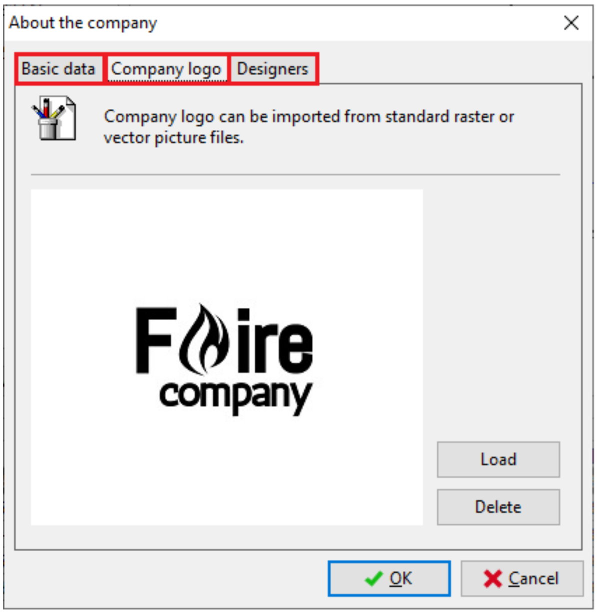
In order to show the information about the company in the output files, the information must be defined in the project. Click the button "**Company**" in the Options drop-down menu.




*Define the company information*

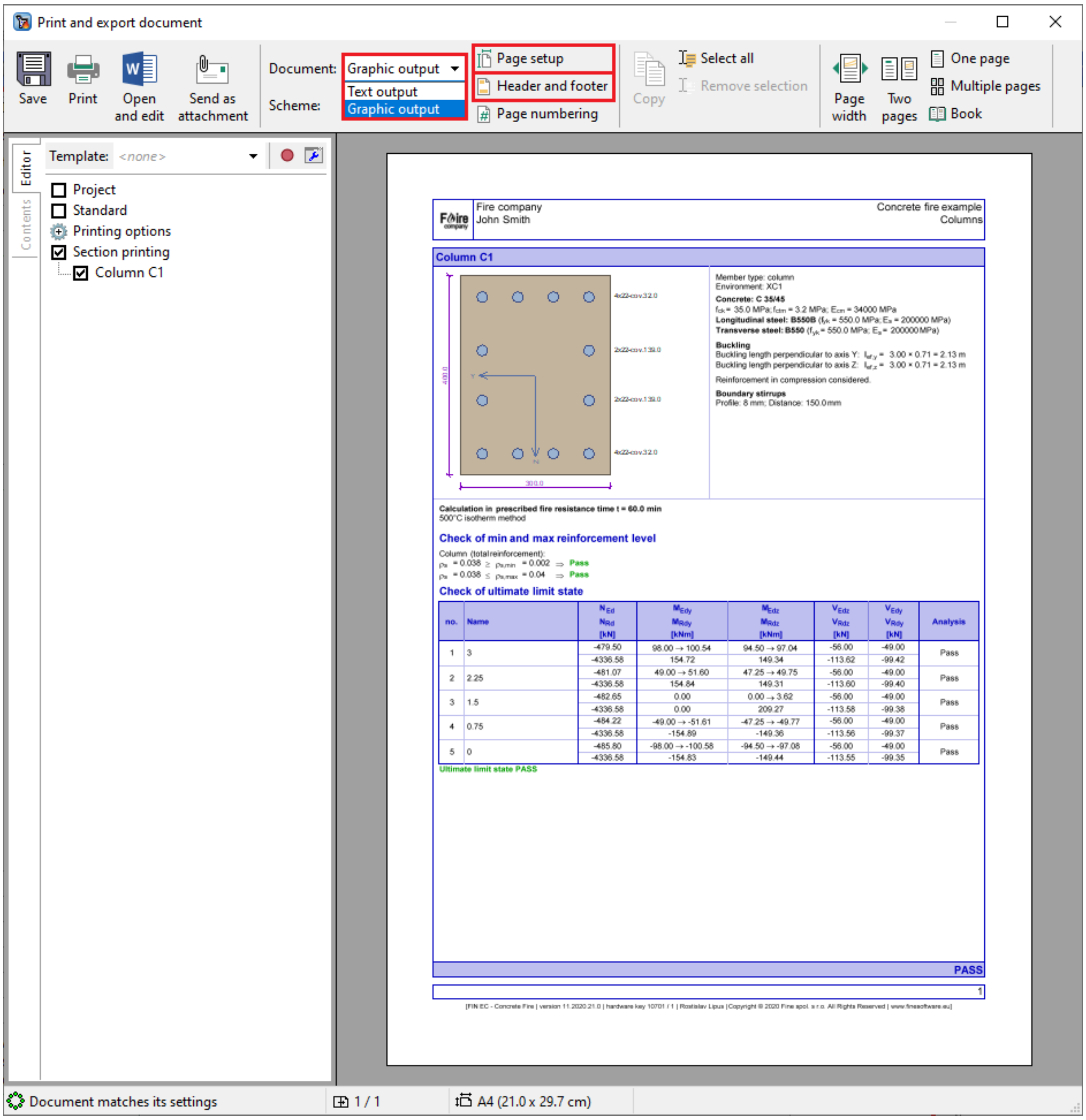
The window "**About the company**" will appear, where the information about the company can be defined. It is also possible to import the company logo, that will show at the header of the exported documentation.




*Set the company data and import the logo*

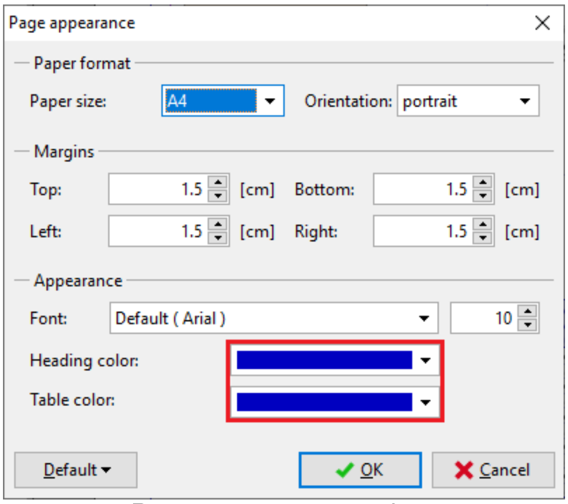
If all the settings finished and saved, the output documentation can be composed. First print out a single title page summarizing all the input data and the check results. This is done by clicking the "**File**" and "**Graphic output**" or "" in the toolbar.




*The graphic output*

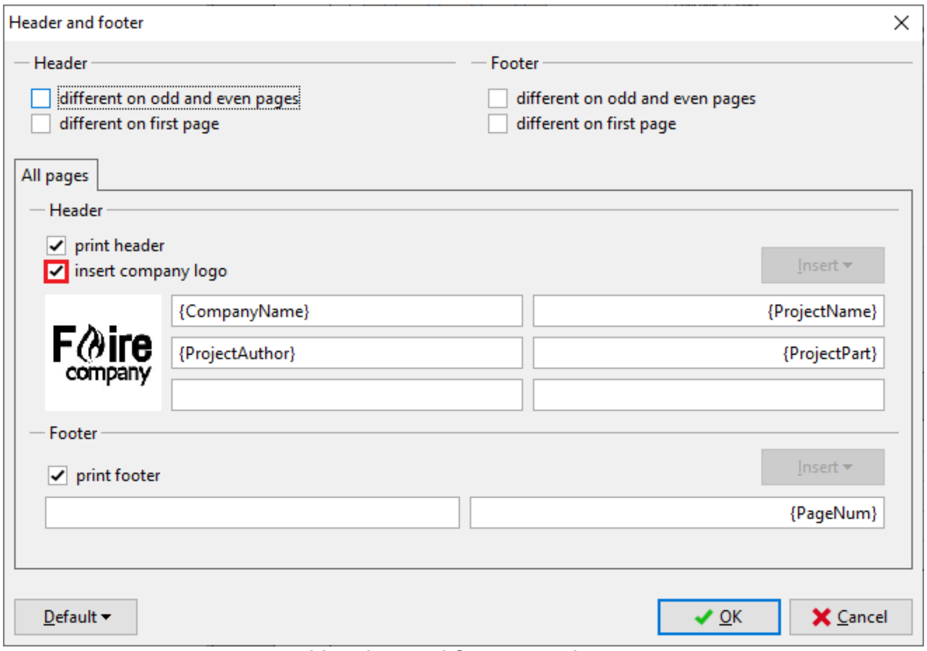
 

At the top part of this window, there are buttons "**Page setup**" and "**Header and footer**". Click the button "**Page setup**" and a window for editing the paper size, margins or a colour of the output document will appear. Pick the preferred colour based on your liking and continue.




*Page appearance settings*

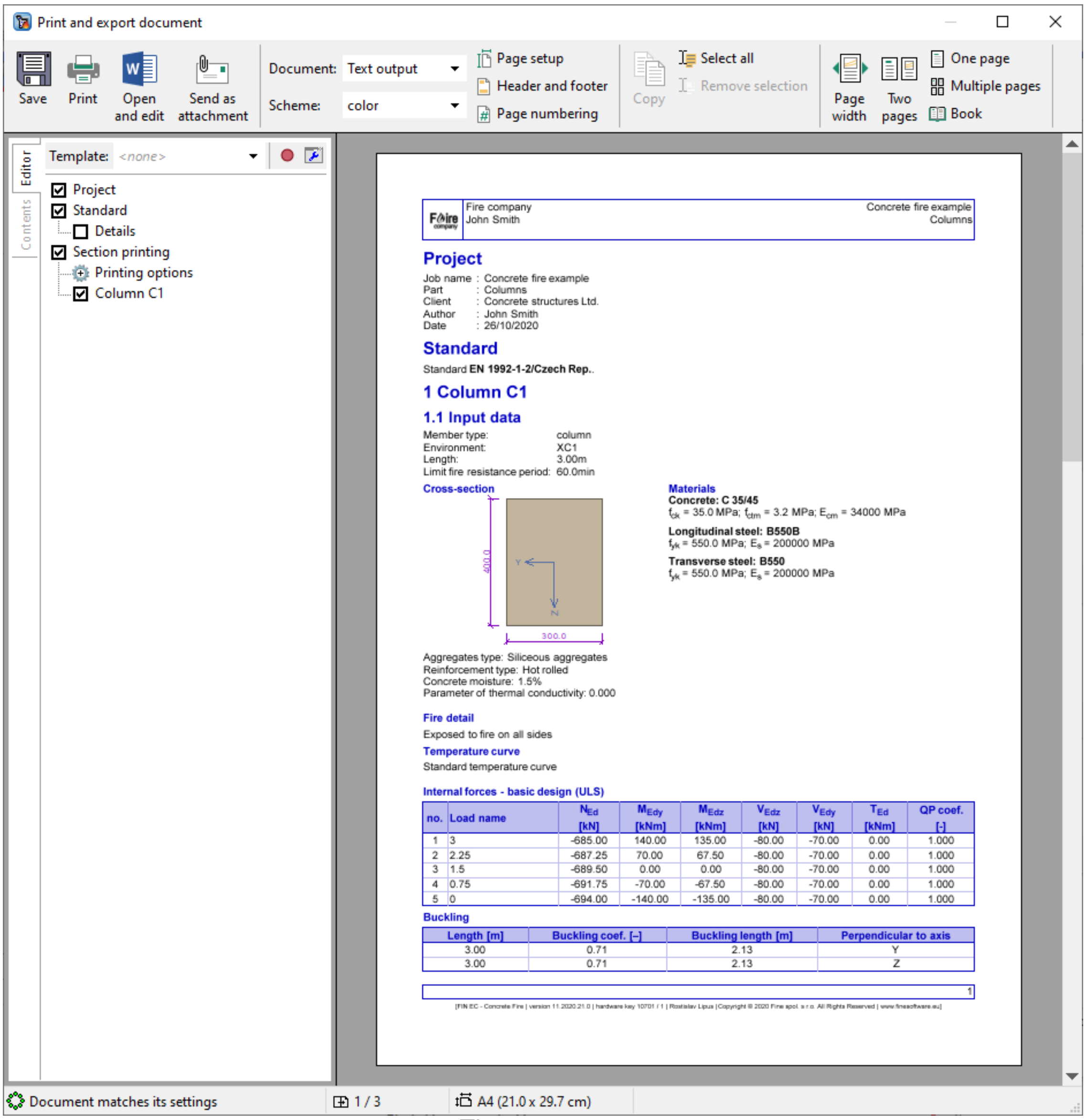
When clicking the button "**Header and footer**" the window for adding information about the project to the header or footer of the output documentation will appear. Check the "**insert company logo**" box in order to show the company logo in the header. The other information about the project can be also added to the footer or header by clicking the button "**Insert**".




*Header and footer settings*

Apart from this title document, the detail text output can be composed by clicking the "" button in the toolbar or selecting "**File**" and "**Text output**" in the main menu. If the print and export document dialog box is still opened, the document type can be changed in the toolbar's "**Document**" drop-down list.

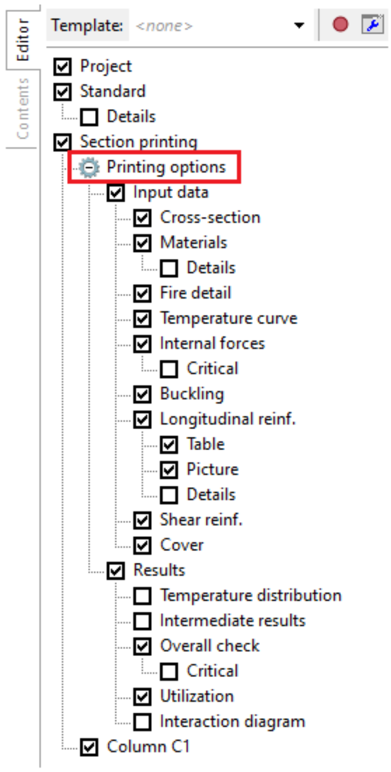
 



*The text output*

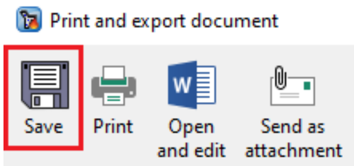
After switching to the "**Text output**" mode, the parts of the documentation can be added or excluded in the "**Editor**".



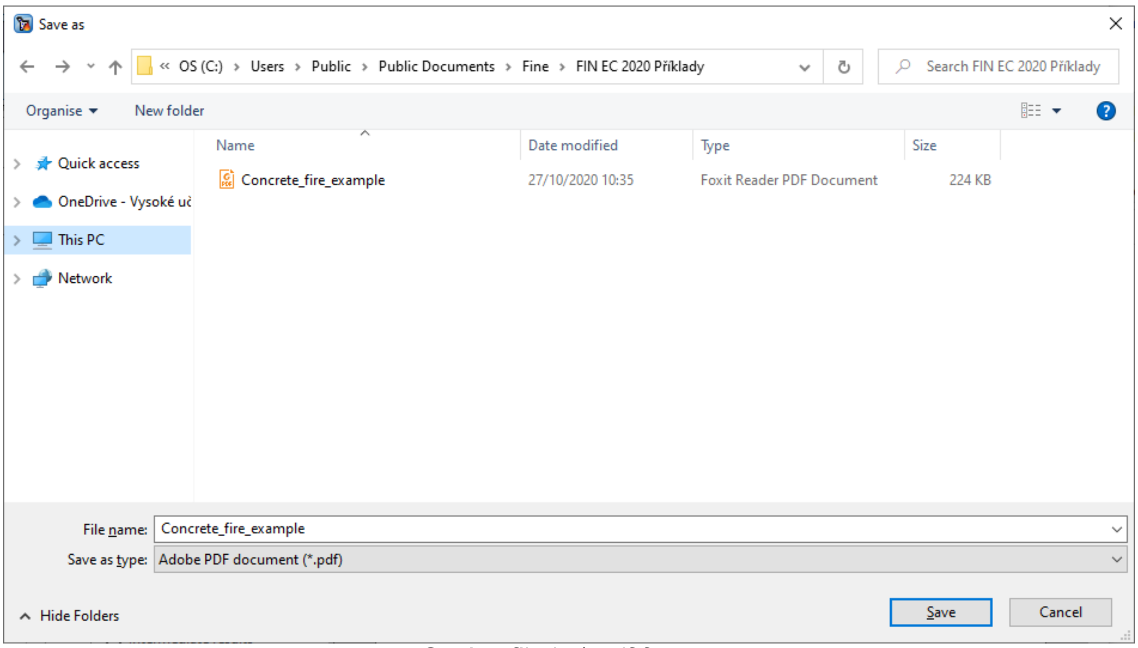

*Add or exclude the output parts*

The software will immediately re-generate the output to reflect each change made in the settings in the tree on the left-hand side. Once the output contains all required information, the document can be again saved on disk.




*Click the save button*

The document can be printed directly by clicking "" button or save it on disk as *.pdf or *.rtf file by clicking "" button.




*Saving file in *.pdf format*

Completing the outputs generation, the work is done.

# Beam to column connection with end plate

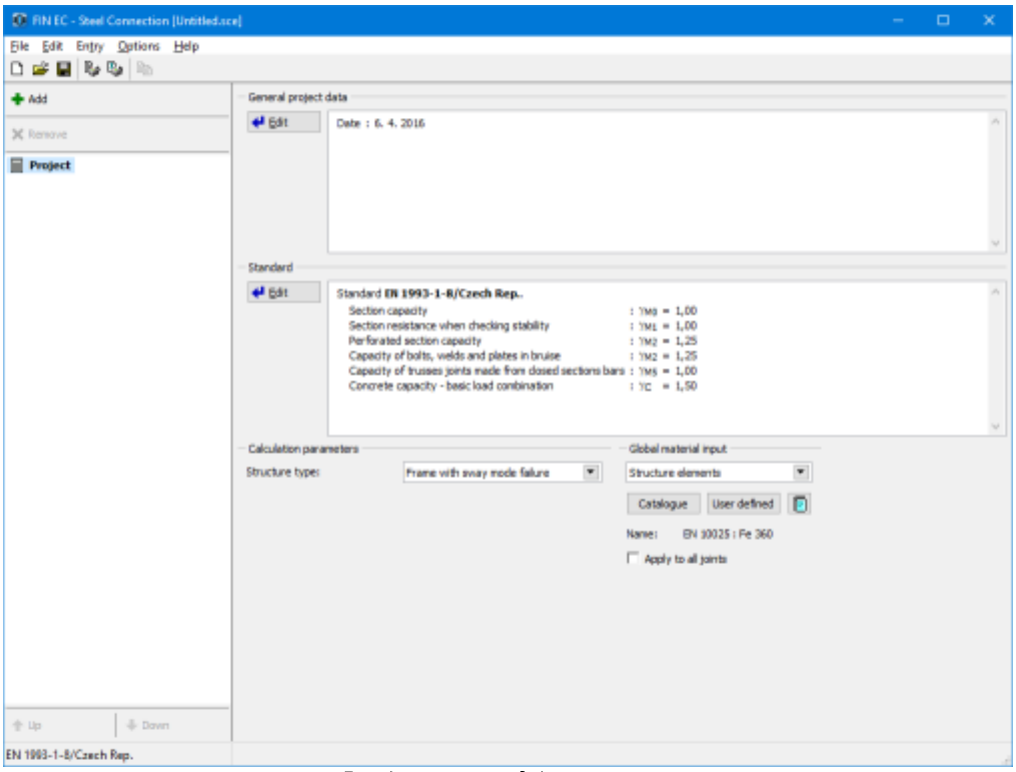
## Introduction

This guide shows the input of double-sided connection of beams to a column in the program "**Steel Connection**". The connection on the left flange of the column is designed using end plate, a beam on the right flange of the column is connected with the help of welding. The joint is a part of a structure without sway mode failure.

| | |
|---|---|
| Beam forces: | $M_{y,Sd}$= 30 kNm, $V_{z,Sd}$= 100kN |
| Axial force in column: | $N_{x,2}$ = 500kN |
| Column: | *HE 140B - EN 10025 : Fe360* |
| Beam: | *IPE 200 - EN 10025 : Fe360* |
| Welds: | $a_{w,f}$= 6mm; $a_{w,w}$= 4mm |
| End plate: | $b_p$= 120mm, $h_p$= 300 mm, $t_p$= 12mm, $a_1$= -90 mm - EN 10025: Fe360 |
| Bolts position: | $w_1$= 30mm, e= [35, 95, 120]mm |
| Bolts: | *M16 10.9* |

## Starting new project

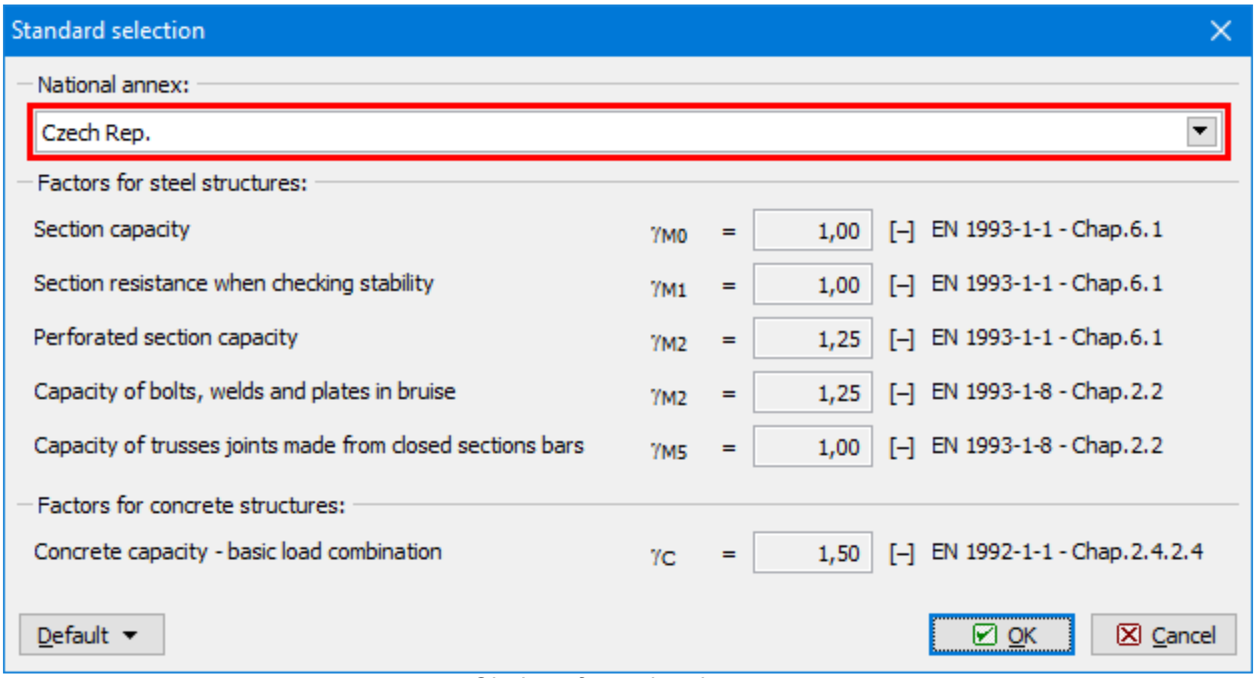
The main screen of the program contains general project information, properties of design standard and also an option to specify global material for all parts of connections.



*Basic screen of the program*

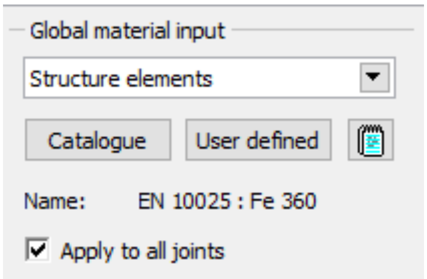
First, we select a national annex. It is necessary to click on the button "**Edit**" in the part "**Standard**" in the middle of the basic screen. A window "**Standard selection**" appears. Upper part contains a list box with available national annexes. National annex "**User defined**" contains an option to specify partial safety factor by the user in the bottom part of the window.



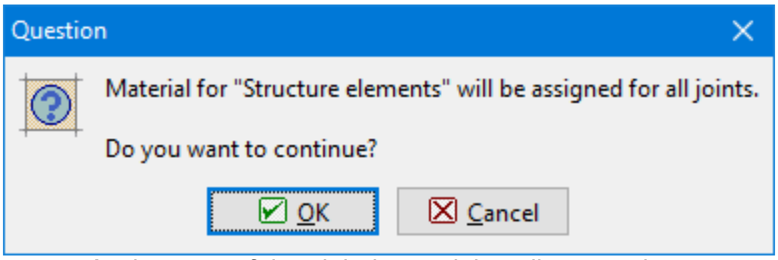

*Choice of a national annex*

As all parts of the joint have identical material, we can use a tool "**Global material input**" in the right bottom corner of the window and assign this material to all parts of the connection. We select "**Structure elements**" in the list of entities. "**Structural elements**" are all steel parts of the detail (column, beams, end plates etc.) except connectors. This material will be assigned to all structural parts after ticking the check box "**Apply to all joints**".



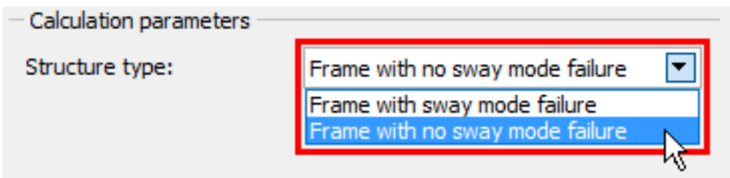

*Input of a global material*

To avoid unwanted changes in existing connections, this step has to be confirmed by a confirmation window.




*Assignment of the global material to all connections*

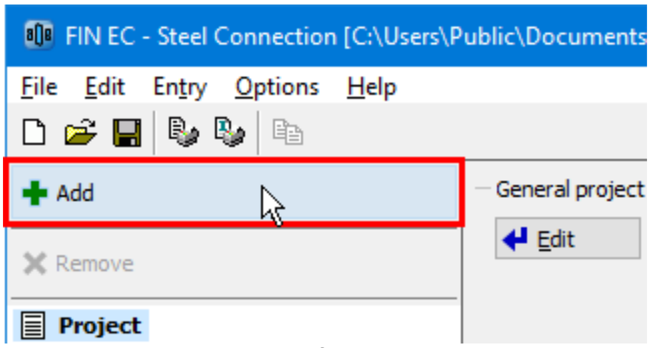
The part "**Calculation parameters**" contains an option to specify a structure type. This setting is important for a calculation of stiffness and classification of the connection (pinned, semi-rigid, rigid).




*Choice of structure type*

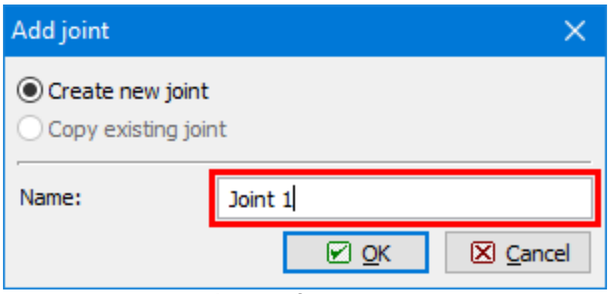
After the setting of all these parameters, it is possible to start with the input of joints. The wizard for a new joint can be launched by the button "**Add**" in the heading of the tree menu.



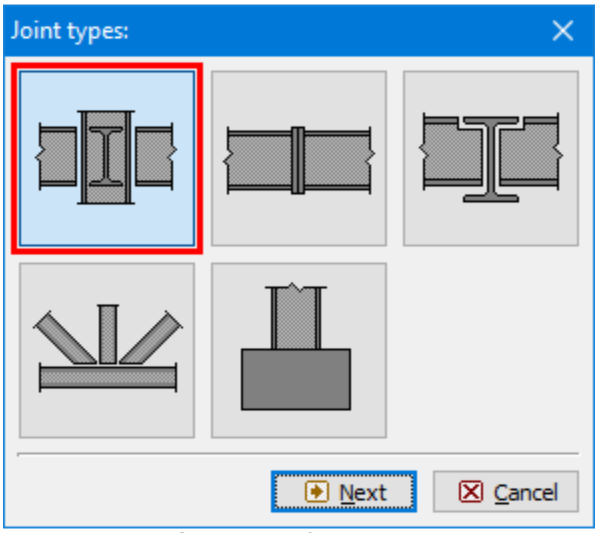
*Insertion of a new joint*

We specify a name ("**Joint 1**") in the window "**Add joint**", that appears after the clicking on that button.



*Input of a name*

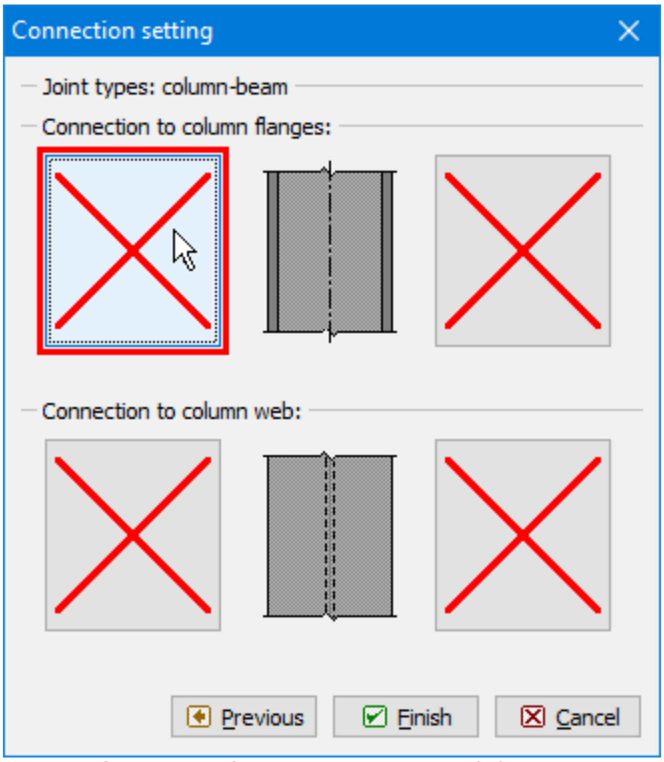
After the confirmation by the button "**OK**", the window with main joint types appears. We select beams connected to a column (left option in the upper row).



*Selection of joint type*

Following window contains an option to specify particular connections in the joint. New connection can be assigned to corresponding direction with the help of buttons with red "**X**". We have to click on the left upper button to assign a connection to the left flange.

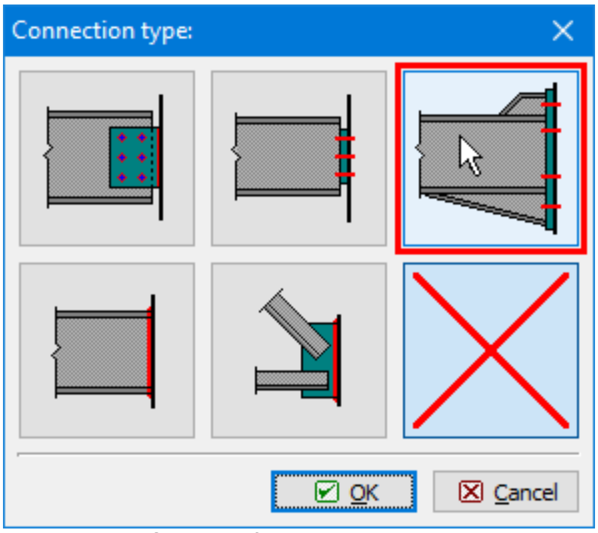



*Selection of connection to the left flange*

The new window "**Connection type**" appears after clicking on the button. Following connection types are available for this type of joint (first column is the position in the window "**Connection type**"):

| | |
|---|---|
| **Upper row, left** | • Hinged end plate |
| **Upper row, middle** | • Fin plate |
| **Upper row, right** | • Stiff end plate |
| **Bottom row, left** | • Welded connection |
| **Bottom row, middle** | • Truss connection |

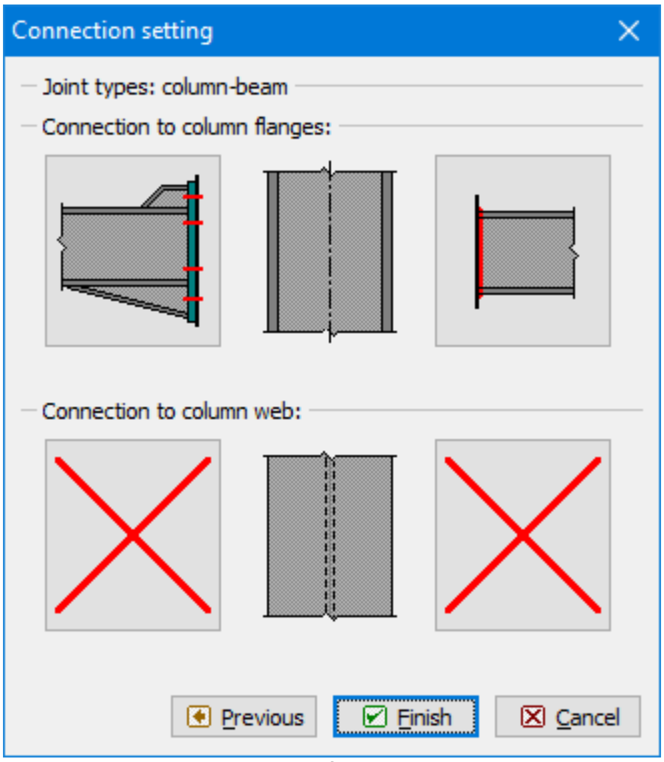
We select stiff end plate (the button in the right upper corner) and confirm the input by the button "**OK**"




*Choice of connection type*

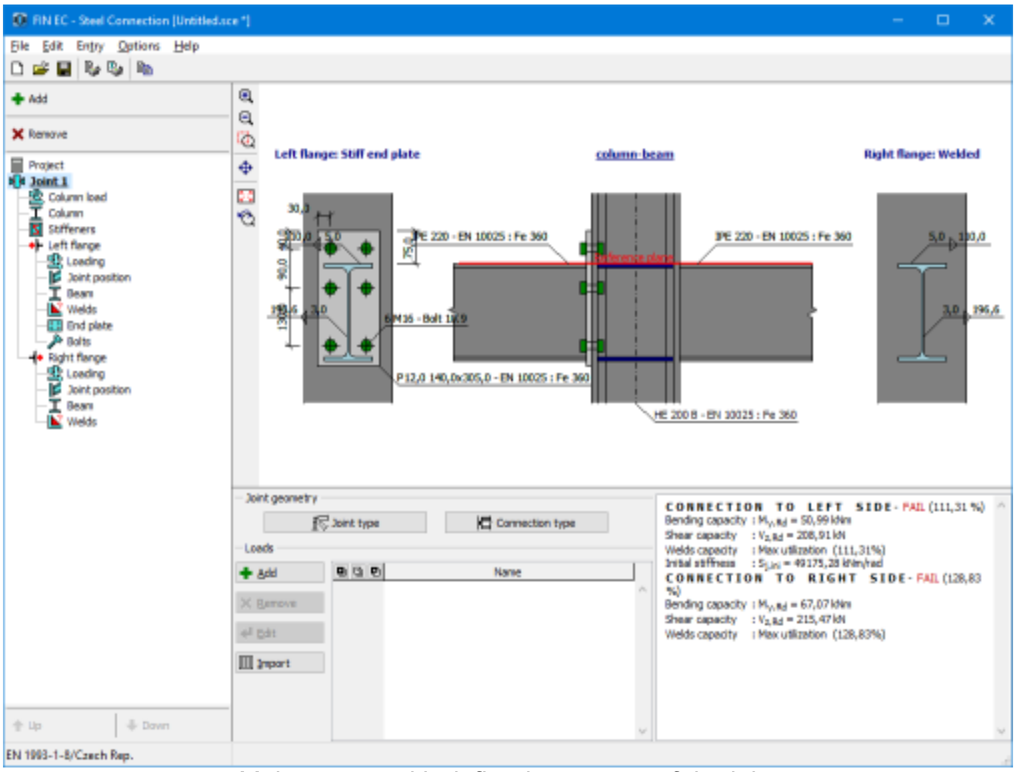
We specify also the connection to the right flange. For this connection, we select the type "**Welded connection**". Following figure shows the joint with selected connections:




*Joint with specified connections*

The initial wizard has to be confirmed by the button "**Finish**". The fundamental geometry of the joint will be created in the workspace, the structure of the tree menu corresponds to this geometry.




*Main screen with defined geometry of the joint*

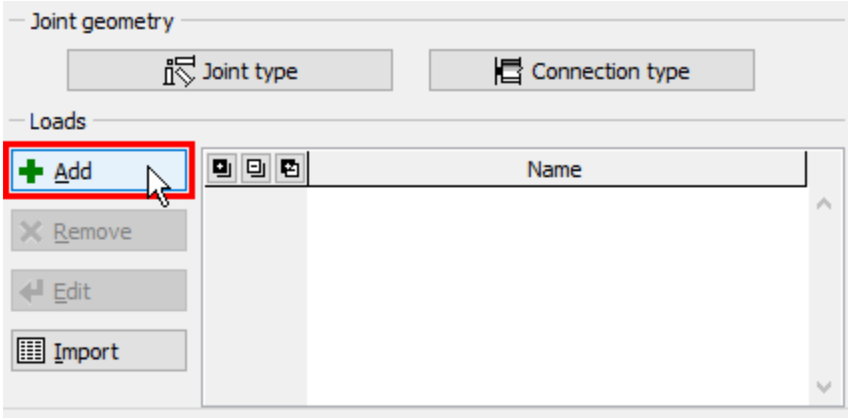
## Input of particular components

The following work is done with the help of the tree menu on the left side of the program. The structure of this menu is generated according to the specified joint geometry. We go through all parts from the top to the bottom and modify inputs. The connection to the left flange will be described in detail. The connection to the right flange would be solved in the same way.

On the main screen of the joint, it is possible to change the specified geometry of the joint (buttons "**Joint type**" and "**Connection type**") and specify list of loads. The load represents a set of internal forces that have to be defined for all members in the joint (column, beams). These internal forces should be resulting values of certain load combination.

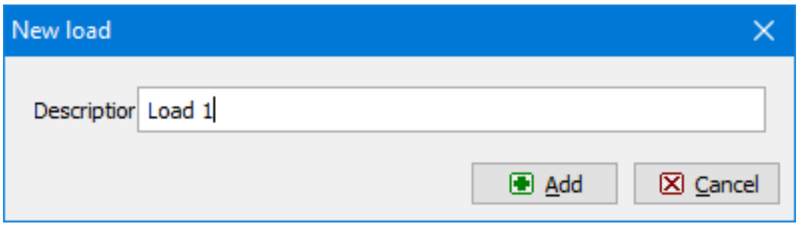
 

Therefore, they are considered as design values. Number of loads for the joint is not limited. We insert a new load with the help of the button "**Add**" in the toolbar on the left side of the loads table. The toolbar also contains buttons for editing and deletion of loads and also a tool for import of loads including internal forces from **.txt* or **.csv* file.




*Button for input of new load*

The new is specified by a name. The input has to be confirmed by the button "**Add**".

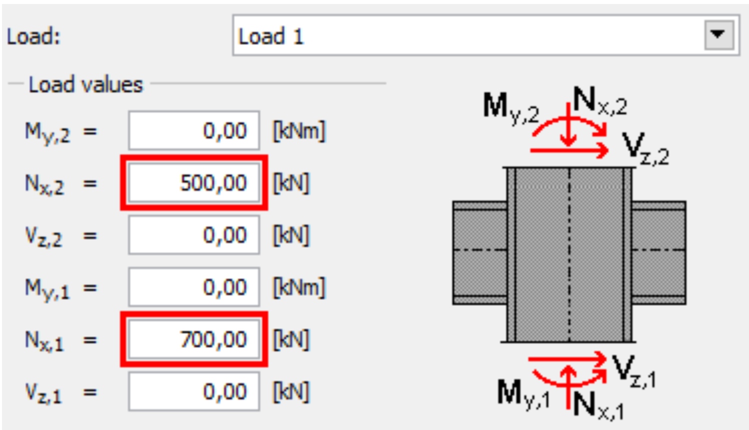



*Window "New load"*

As we want to add only one load, we close the window by the button "**Cancel**" after the input of first load.

## Column load

If at least one load is entered, it is possible to switch to the part "**Column load**" in the tree menu and specify internal forces for the column. The axial force above the joint "$\mathbf{N_{x,2}}$" should be *500kN*. As the increment due to connected beams is *2x100kN*, the axial force under the joint "$\mathbf{N_{x,1}}$" is *700kN*.



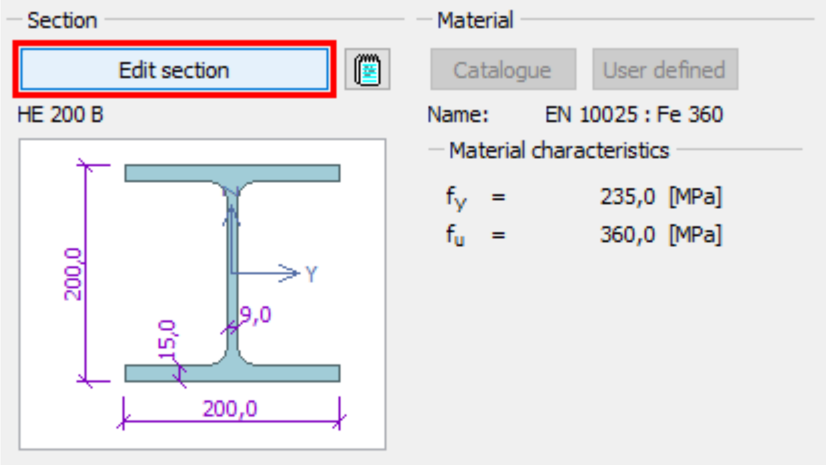

*Input of forces in the mode "Column load"*

The next step is an input of column geometry in the part "**Column**".

## Column

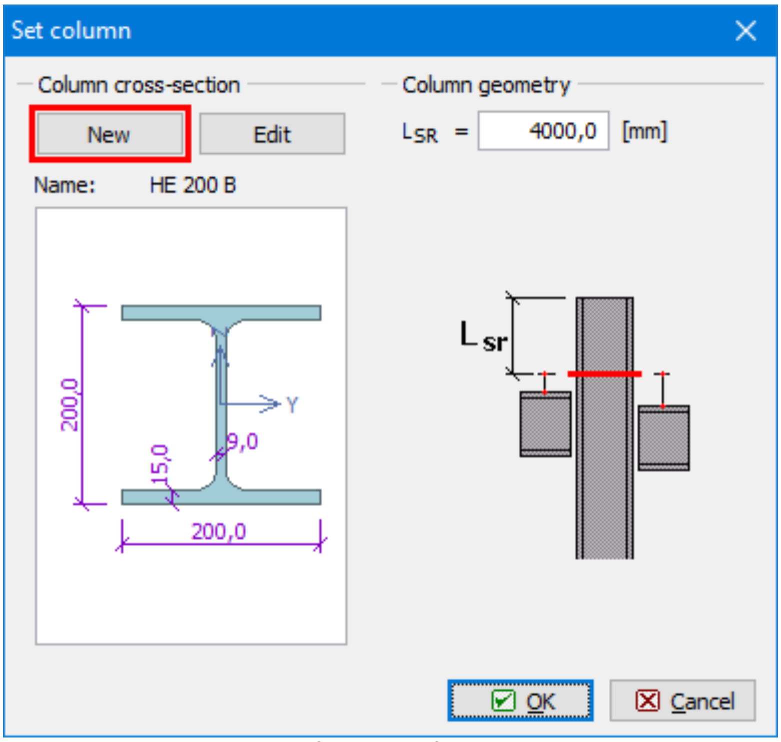
This part contains a geometry of the column (cross-section, length, material). Buttons for an input of material are disabled, as the global material specified at the beginning is used. The window for the input of column geometry can be launched by the button "**Edit section**" or by clicking on the cross-section in the bottom frame.



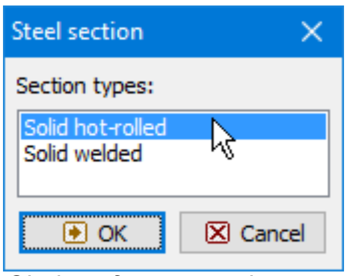

*Button for editing the column cross-section*

The window "**Set column**" is a general window for input of cross-section. The identical window is also used for input of beams etc. The distance between fundamental level of the joint and column end can be also specified here. This value is important for joints, that are affected by the proximity of column end. We change the cross-section with the help of the button "**New**".




*The button for input of cross-section*

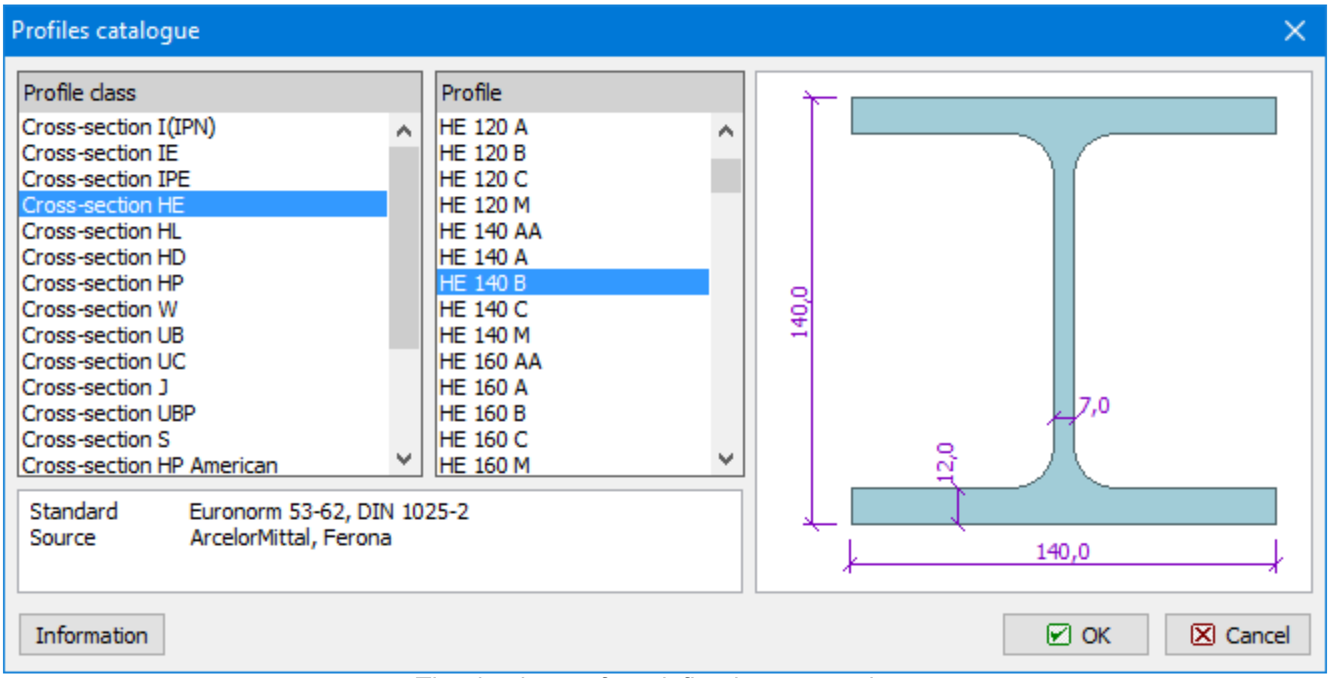
The window "**Steel section**" that appears after the clicking on the button contains an option to select type of cross-section. We select database of rolled cross-sections (the option "**Solid hot-rolled**") and open the window "**Profiles catalogue**" by pressing the button "**OK**".




*Choice of cross-section type*

We select the section type "**Cross-section HE**" in the first column of the database and the item "**HE 140B**" in the second column. The choice of cross-section has to be confirmed by the button "**OK**".

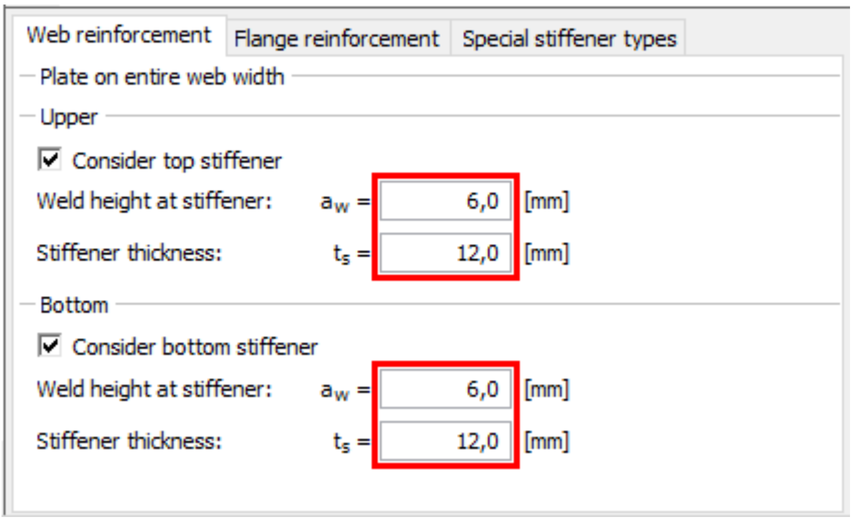


*The database of predefined cross-sections*

The part "**Stiffeners**" of the tree menu follows.

## Stiffeners

The reinforcement of the column can be added in this part. Several types of reinforcement are organized into tabs. We specify web reinforcement on the first tab "**Web reinforcement**". It is necessary to change the values "$\mathbf{a_w}$" and "$\mathbf{t_s}$" both for upper and bottom stiffeners.

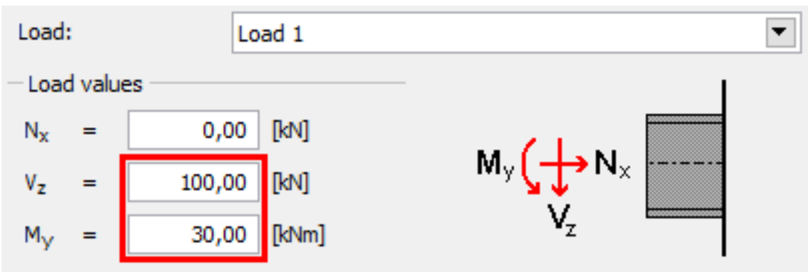


*Web stiffeners*

This is the last part of column properties. Now, we switch to part "**Left flange**".

## Load

Similarly to column, this part contains the input of internal forces for beam. The bending moment *30kN* has to be entered into the input line "$\mathbf{M_y}$" and shear force *100kN* into the input line "$\mathbf{V_z}$". Also a value for axial force "$\mathbf{N_x}$" can be specified in this part.



*Internal forces for left flange*

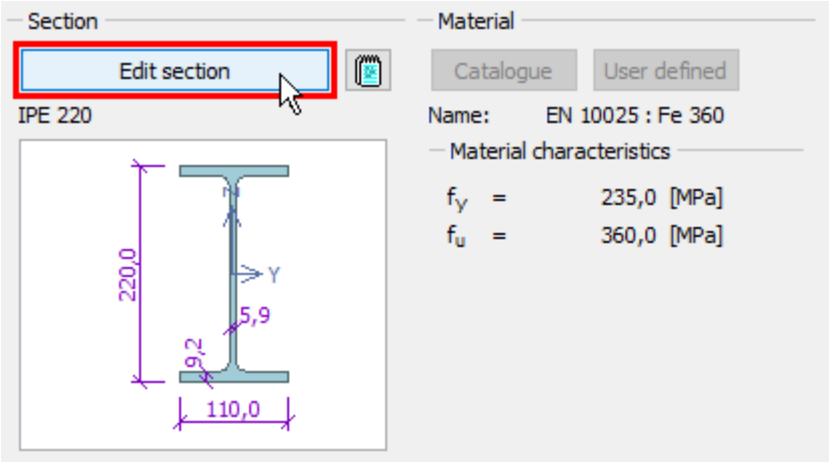
The part "**Joint position**" follows.

## Joint position

The eccentricity of connection or pitch of beam can be specified in this part. We keep default settings and switch to part "**Beam**".

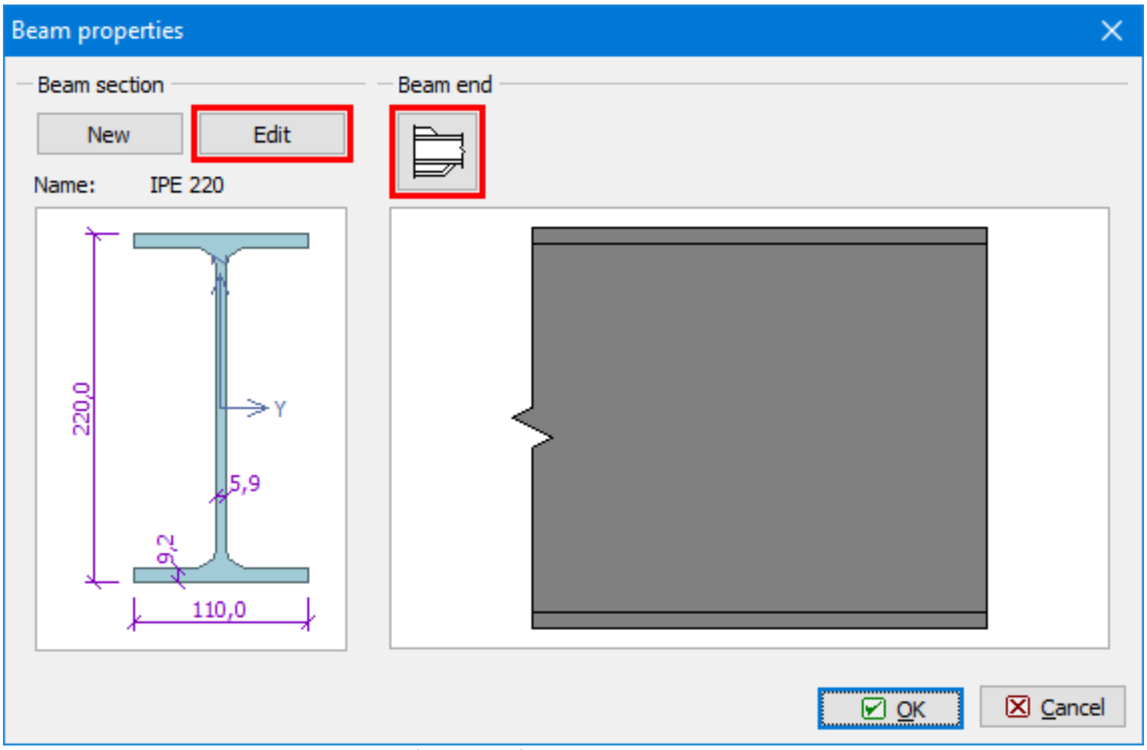
## Beam

We specify the beam geometry (cross-section, haunch) with the help of the button "**Edit section**".




*The button for edit of beam geometry*

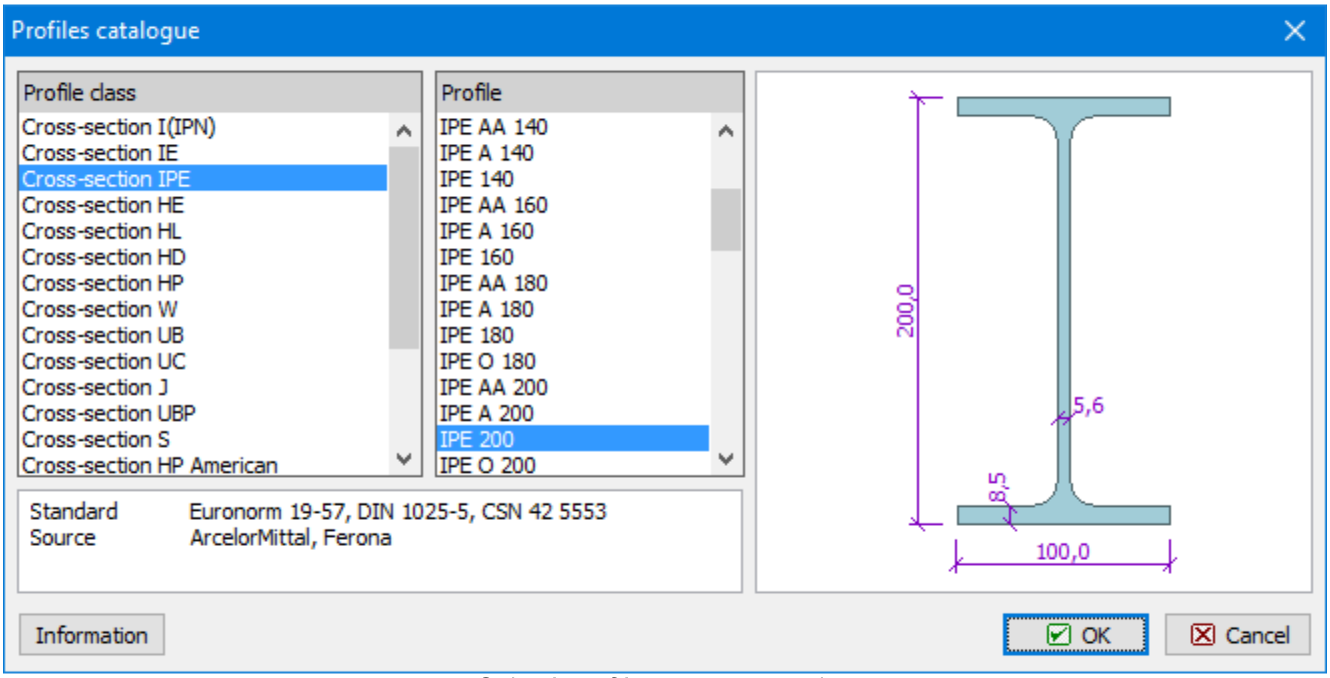
This window is similar to the window with column geometry. Left part of the window contains input of cross-section, right part input of haunches.




*Buttons for edit of cross-section and haunch*

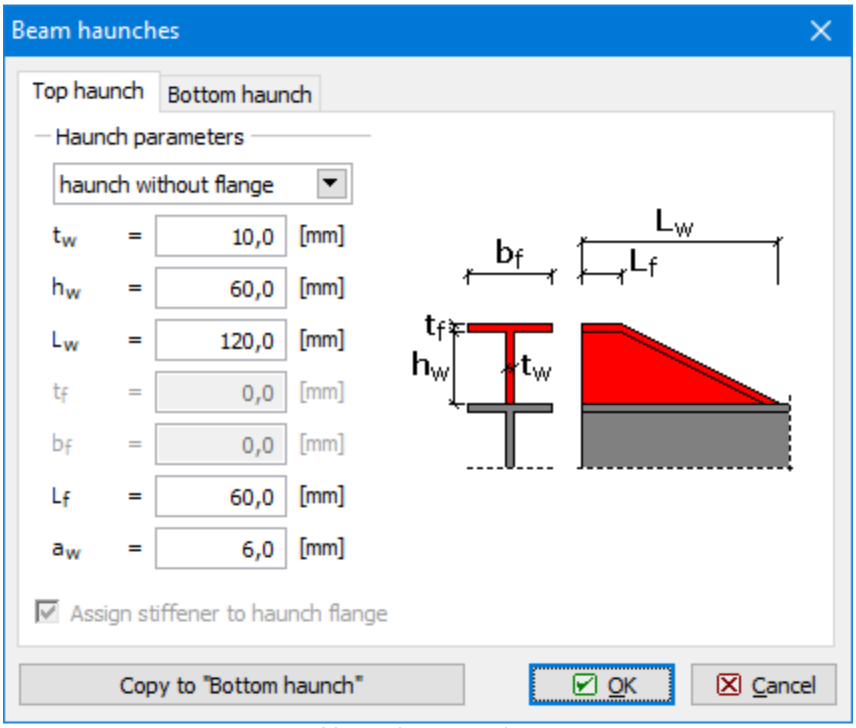
The button "**Edit**" opens the database of rolled cross-sections. We change the cross-section to "**IPE 200**".



*Selection of beam cross-section*

The button in the part "**Beam end**" opens the window "**Beam haunches**". We select an option "**Haunch without flange**" and specify dimensions according to the following figure in the tab "**Top haunch**".



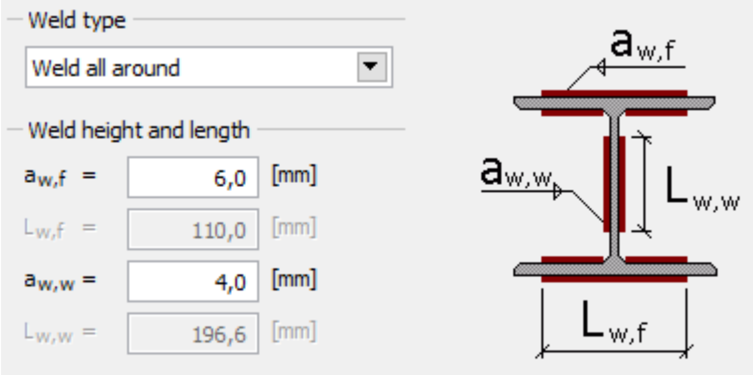
*Haunch properties*

We close windows "**Beam haunches**" and "**Beam properties**" by buttons "**OK**" and switch the tree menu into the mode "**Welds**".

## Welds

Fillet welds all round will be used for the connection of the end plate to the beam. Therefore, we select weld type "**Weld all around**" and enter the throat thickness for flanges "$a_{w,f}$" and the throat thickness for the web "$a_{w,w}$". Lengths are calculated automatically according to the geometry of the cross-section. Arbitrary lengths can be defined using weld type "**User defined weld**".

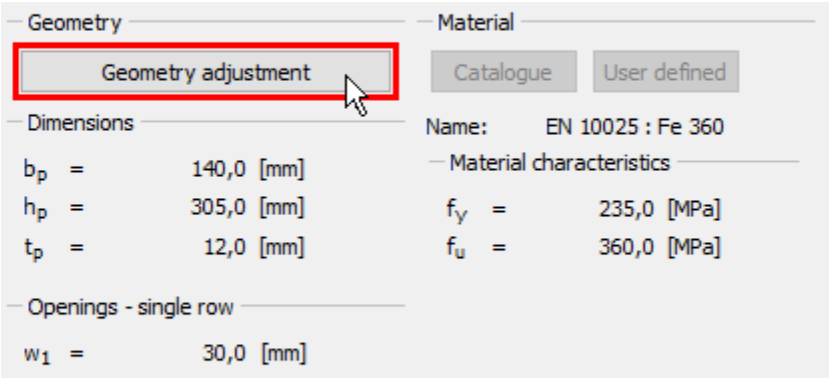
 




*Weld properties*

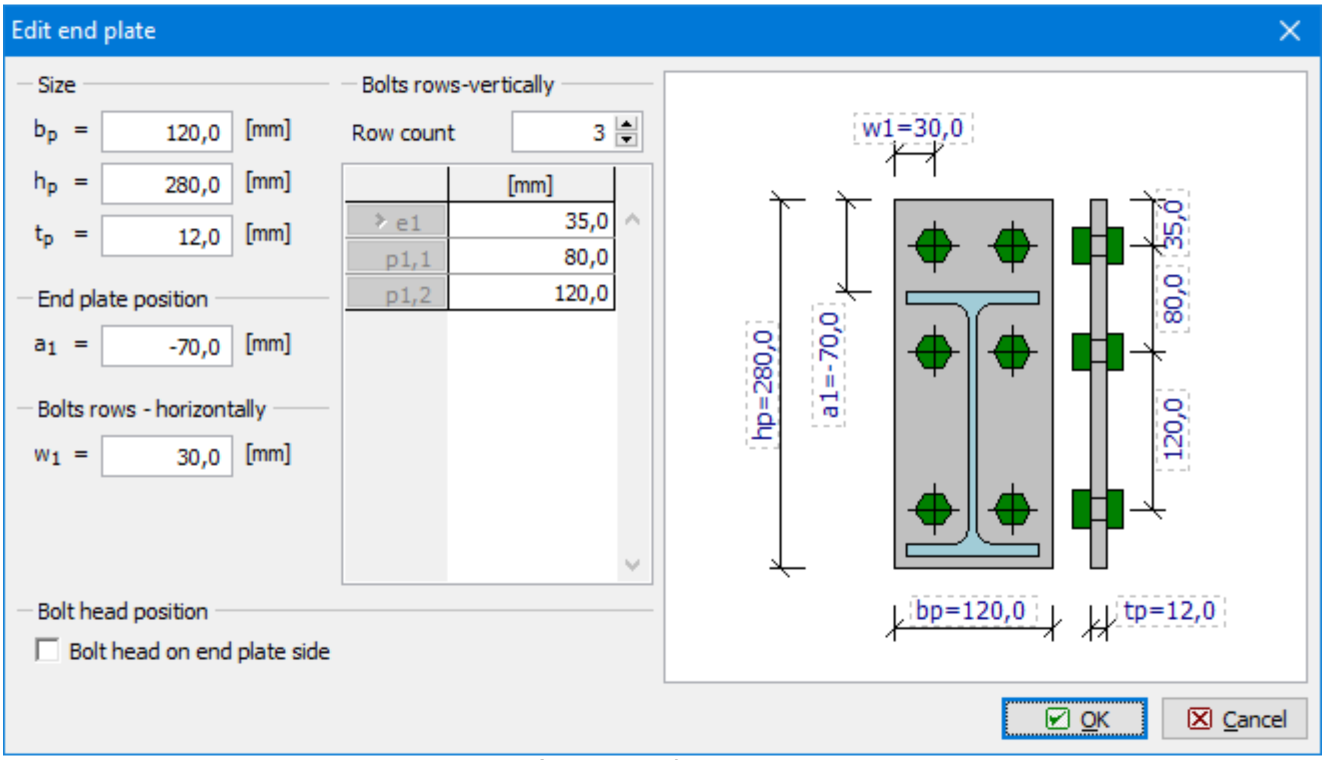
## End plate

The input of end plate geometry follows. We open an appropriate window using the button "**Geometry adjustment**" in the bottom frame.




*Button for input of end plate geometry*

We specify end plate dimensions "$\mathbf{b_p}$", "$\mathbf{h_p}$", "$\mathbf{t_p}$", position of end plate relatively to the beam edge "$\mathbf{a_1}$", horizontal position of bolts "$\mathbf{w_1}$" and vertical positions of rows. Entered values are shown in the figure below. Only two bolt columns can be specified for rigid end plate. Four columns are permitted for hinged end plate. The input can be done with the help of input lines in the left part of the window or using active dimensions in the end plate figure in the right part of the window. The input has to be confirmed by the button "**OK**".

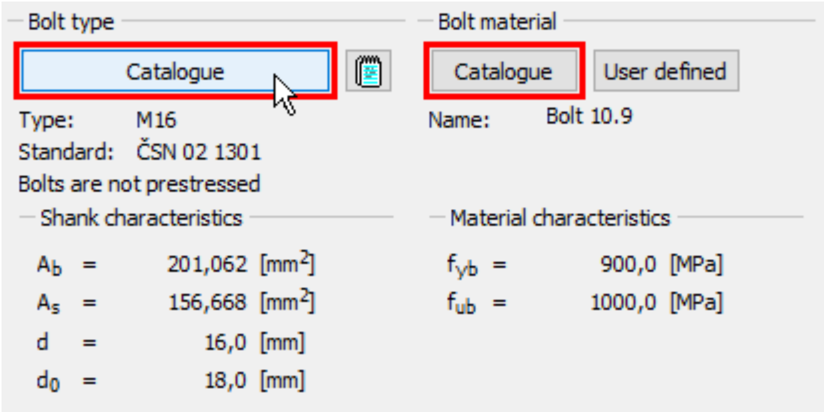



*Geometry of end plate*

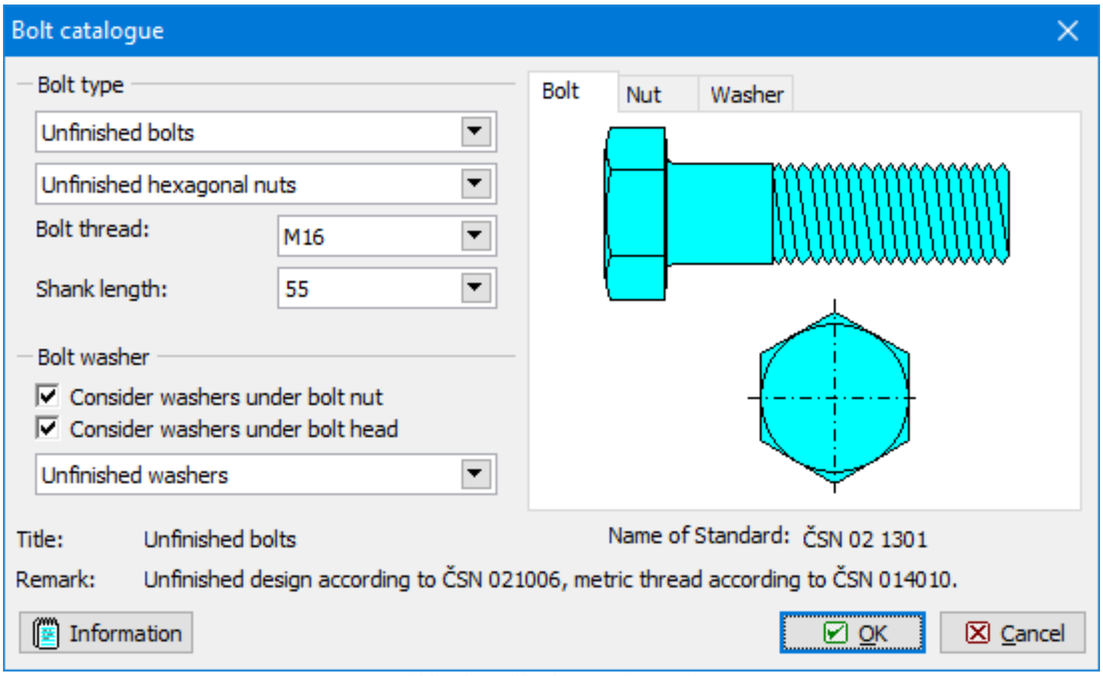
## Bolts

The part "**Bolts**" contains the input of bolt type, dimensions and material. The type and dimension can be specified with the help of the button "**Catalogue**" in the part "**Bolt type**" of the bottom frame. The material can be selected using the button "**Catalogue**" in the part "**Bolt material**".



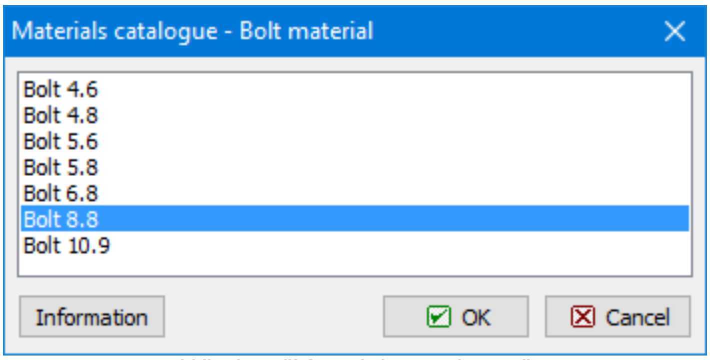

*The bottom frame with buttons for input of bolt properties*

We select bolt diameter "**M16**" and shrank length *55mm* in the window "**Bolt catalogue**". The shrank length is an important input for the determination of shear plane. We tick on also washers under the bolt head and nut. The window has to be closed by the button "**OK**".




*Window "Bolt catalogue"*

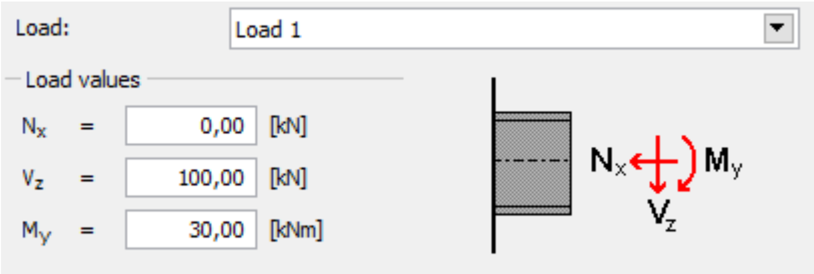
We choose the material "**Bolt 8.8**" in the window "**Materials catalogue**" and confirm the input by the button "**OK**".




*Window "Materials catalogue"*

## Right flange

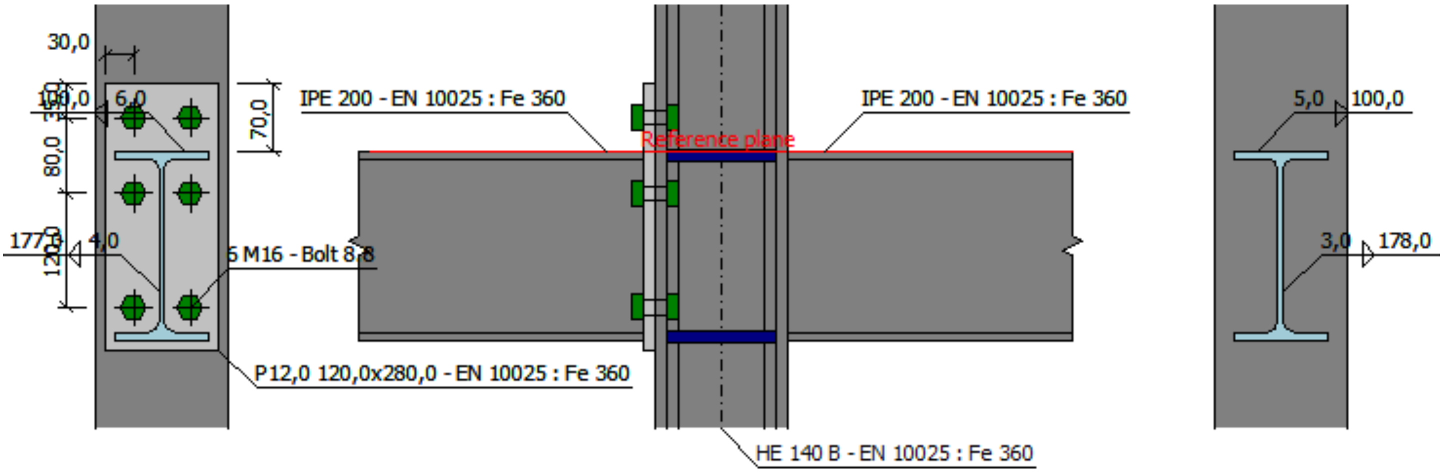
The connection of the second beam to the right flange is similar to the already entered connection. Only properties of the end plate are missing in this case. In the part "**Load**", we specify the bending moment $M_y$= *40kN* and shear force $V_z$= *100kN*.



*Load for the right connection*

Warnings regarding the unbalance of forces in joint should disappear after the input of these forces. No changes are required in the part "**Joint position**".

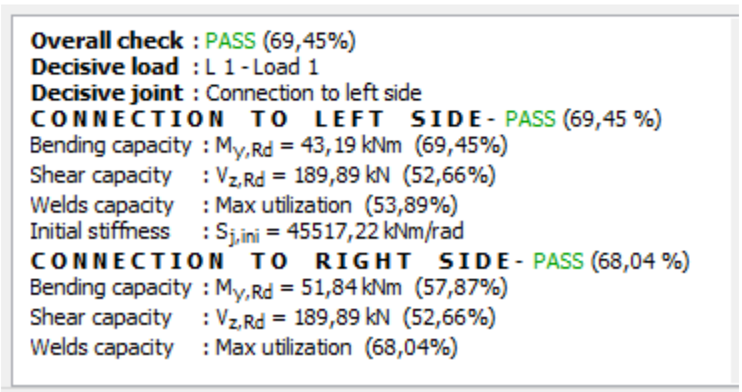
We select also the profile *IPE 200* as the beam cross-section in the part "**Beam**". Finally, the weld type in the part "**Welds**" should be "**Weld all around**" and the throat thickness for flanges "$\mathbf{a_{w,f}}$" and for the web "$\mathbf{a_{w,w}}$" should be identical to the welds in the left connection.



*Joint geometry*

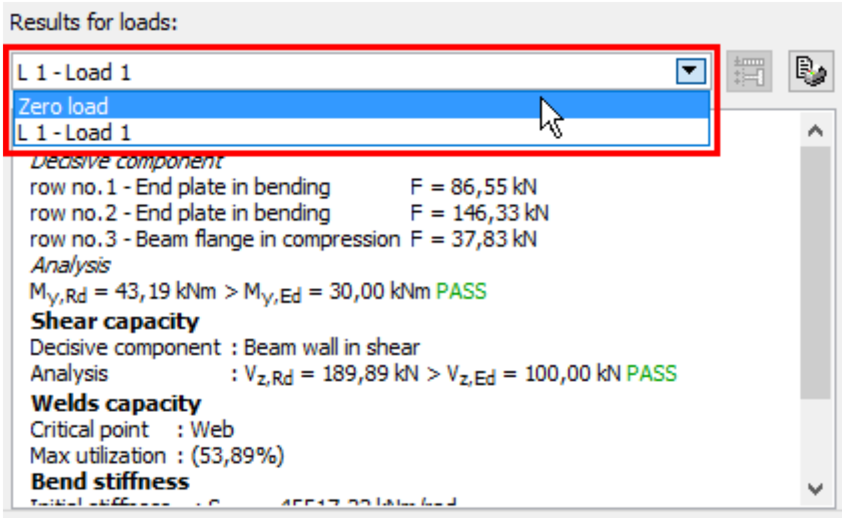
## Results

The total results are displayed in the right part of the bottom frame. They contain maximum utilization, the decisive load and connection and also brief results for all connections.



*Total results*

Th detailed results for particular components are displayed in the basic mode of corresponding component (nodes "**Left flange**" and "**Right flange**" of the tree menu). These results contain detailed bearing capacities and decisive components. The results can be displayed for all entered loads. Available is also an option "**Zero load**". In this case, maximum bearing capacities for all components are displayed.




*Choice of loads*

# Steel truss joint

## Introduction

This guide shows the input of truss joint in the program "**Steel Connection**". A web made of two L-profiles is bolted to a joint plate.

| | |
|---|---|
| Chord: | *2x L 120x12* |
| Web: | *2x L 90x8 (X= -20mm, Y= 0, D= 100mm)* |
| Plate: | $b_p$= 230mm, $h_p$= 140mm, $t_p$= 12mm, $h_{p1}$= $h_{p2}$= 15mm, $a_w$= 3mm |
| Material: | *EN 10025: Fe360* |
| Bolt type: | *M16 8.8* |
| Position of bolts: | *single row, e = [40, 70, 70] mm* |
| Welds: | $a_{w.1}$ = 3 mm, $a_{w.2}$ = 3 mm, $a_{w.3}$ = 3 mm |

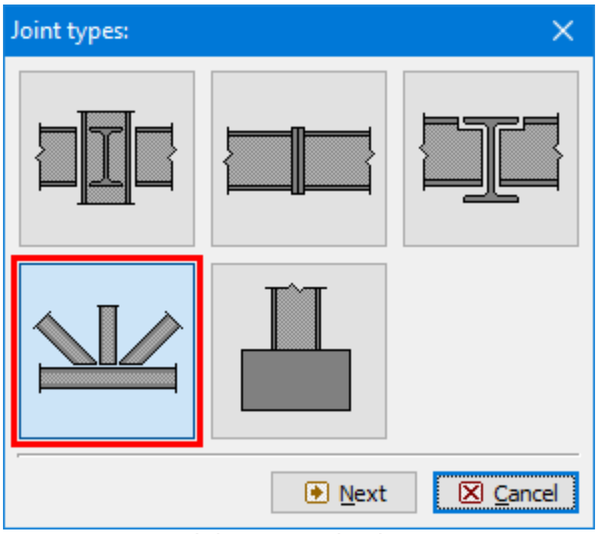
## Creation of a new joint

As general characteristics (standard, global material etc.) were already specified in the previous guide, We start directly with addition of a new joint into the existing project. We use the button "**Add**" at the top of the tree menu on the left side of the window. The window "**Add joint**" appears, it is necessary to specify a name there. The input has to be confirmed by the button "**OK**".


*Input of joint name*

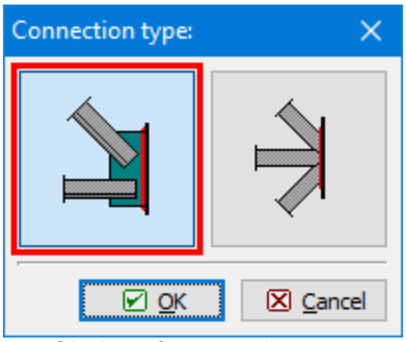
The window "**Joint types**" appears. We select a truss joint (left option in the bottom row) and continue with the help of the button "**Next**".



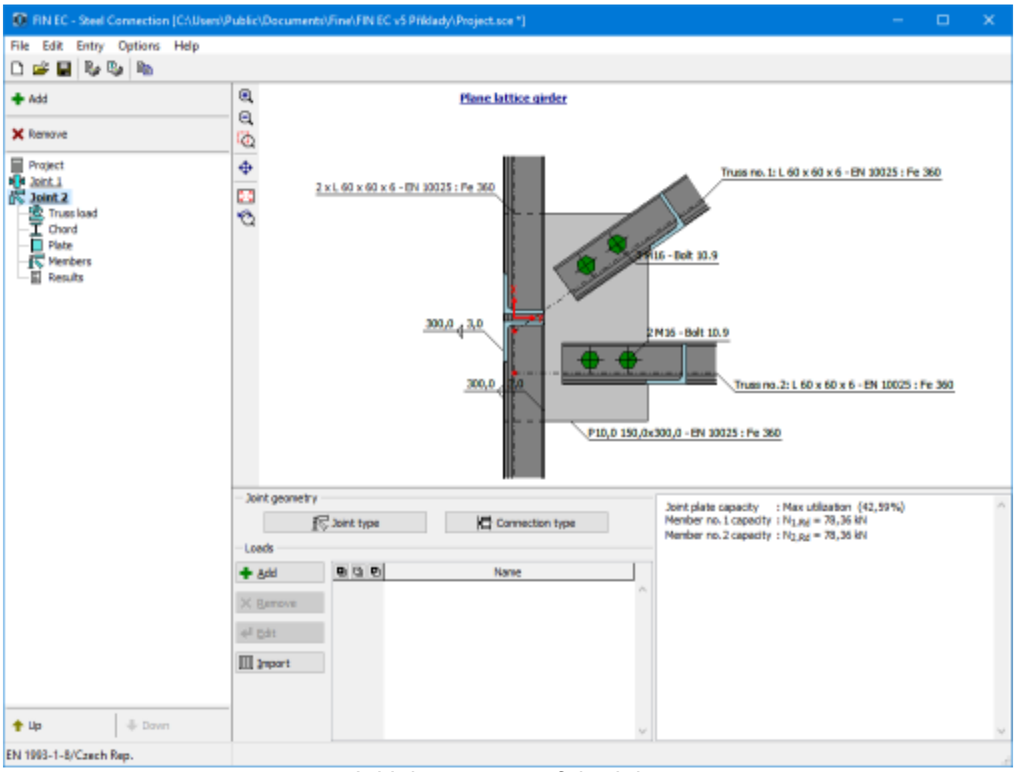

*Joint type selection*

The choice of connection follows. We select connection with fin plate and confirm the choice by the button "**OK**".




*Choice of connection type*

The initial geometry of the joint appears in the workspace.



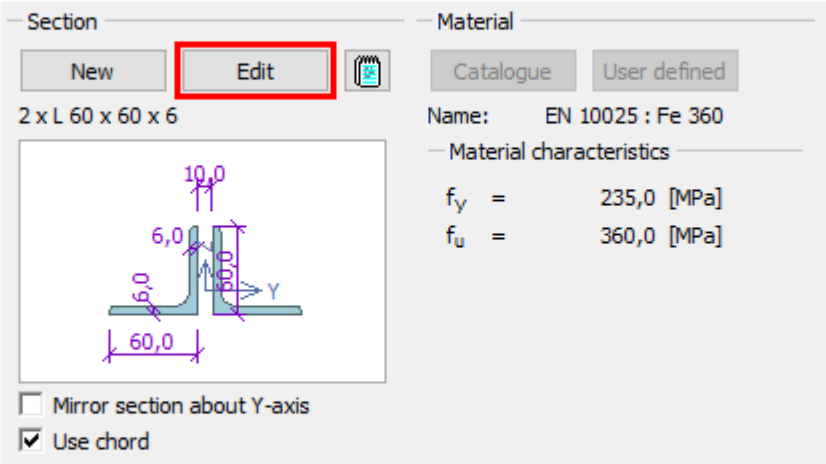

*Initial geometry of the joint*

The inputs for this joint are also organized into few parts, that are listed in the tree menu. As we are searching for the maximum resistance of the joint, we will skip parts "**Joint 2**" (list of loads) and "**Truss load**" (internal forces for connected webs).

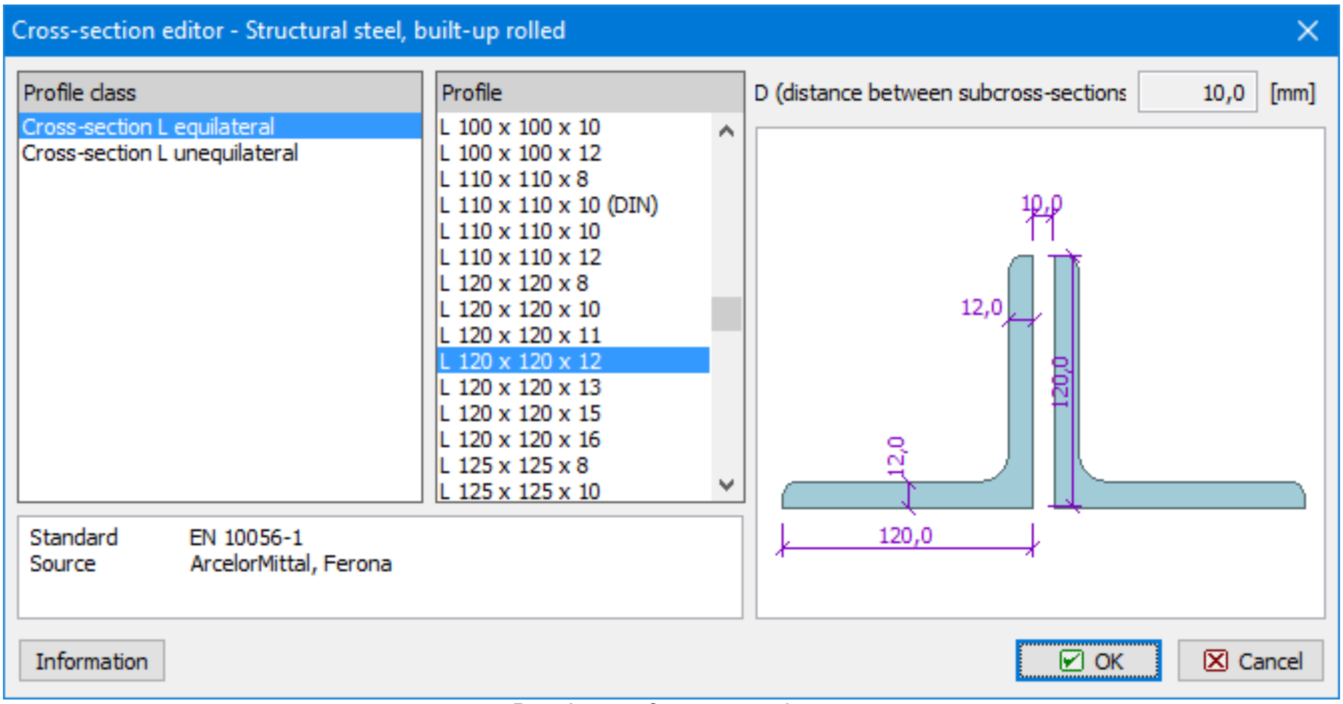
## Chord

The part "**Chord**" contains the properties of continuous member of the joint. The input and consideration of the chord can be switched off with the help of the setting "**Use chord**". Unsymmetrical L-profiles can be mirrored by using the setting "**Mirror section about Y-axis**". We use the button "**Edit**" to modify the cross-section.




*Chord properties*

We select the profile class "**Cross-section L equilateral**" in the left column of the database in the window "**Cross-section editor**". The cross-section "**L 120x120x12**" should be selected in the second column of the database. The distance "**D**" between partial cross-sections cannot be edited, as it is selected automatically according to the plate thickness. The choice has to be confirmed by the button "**OK**".




*Database of cross-sections*

## Plate

The geometry of the plate and weld thickness can be changed in this part. We use values according to the following figure. Buttons for an input of material are disabled, as the global material specified at the beginning is used.

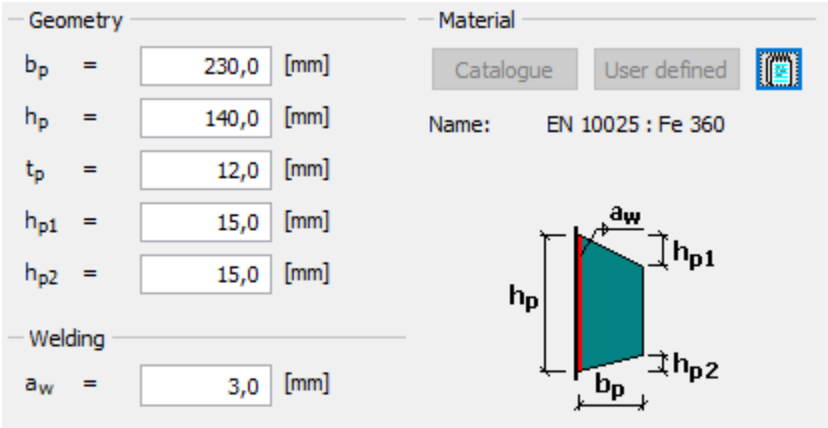
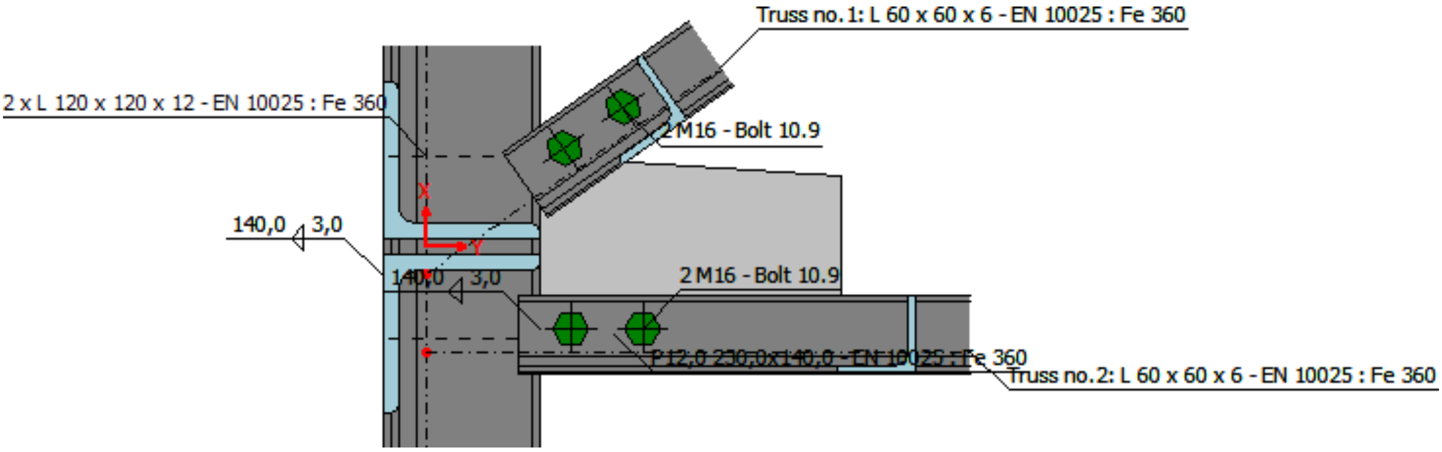


*Plate properties*

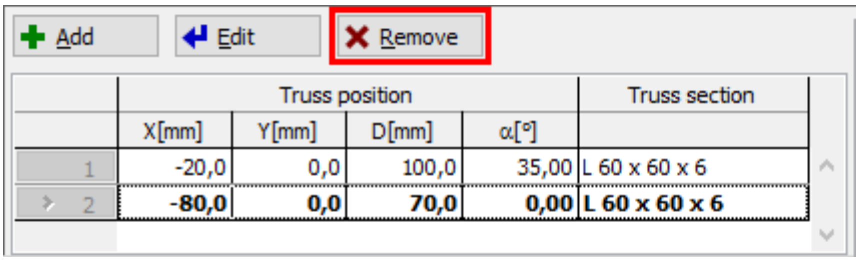
Workspace shows following geometry of the joint:



*Joint geometry*

## Members

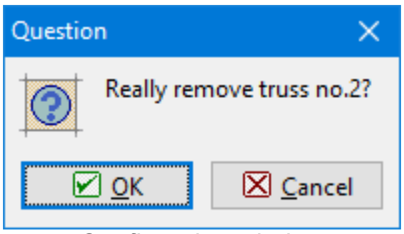
This part contains a list of connected webs. As we need a joint with one connected web, we have to delete one of existing members. This member has to be selected in the table (highlighted by a bold font) and after that it can be deleted with the help of the button "**Remove**".




| | Truss position | | | | Truss section |
|---|---|---|---|---|---|
| | X[mm] | Y[mm] | D[mm] | α[°] | |
| 1 | -20,0 | 0,0 | 100,0 | 35,00 | L 60 x 60 x 6 |
| 2 | **-80,0** | **0,0** | **70,0** | **0,00** | **L 60 x 60 x 6** |

*Deletion of the member number 2*

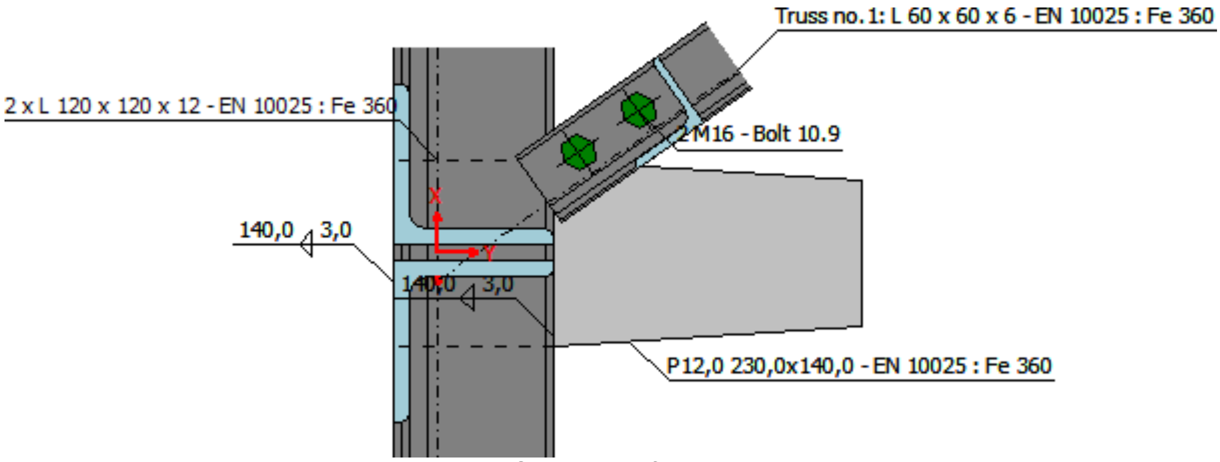
The confirmation window appears, we use the button "**OK**".



*Confirmation window*

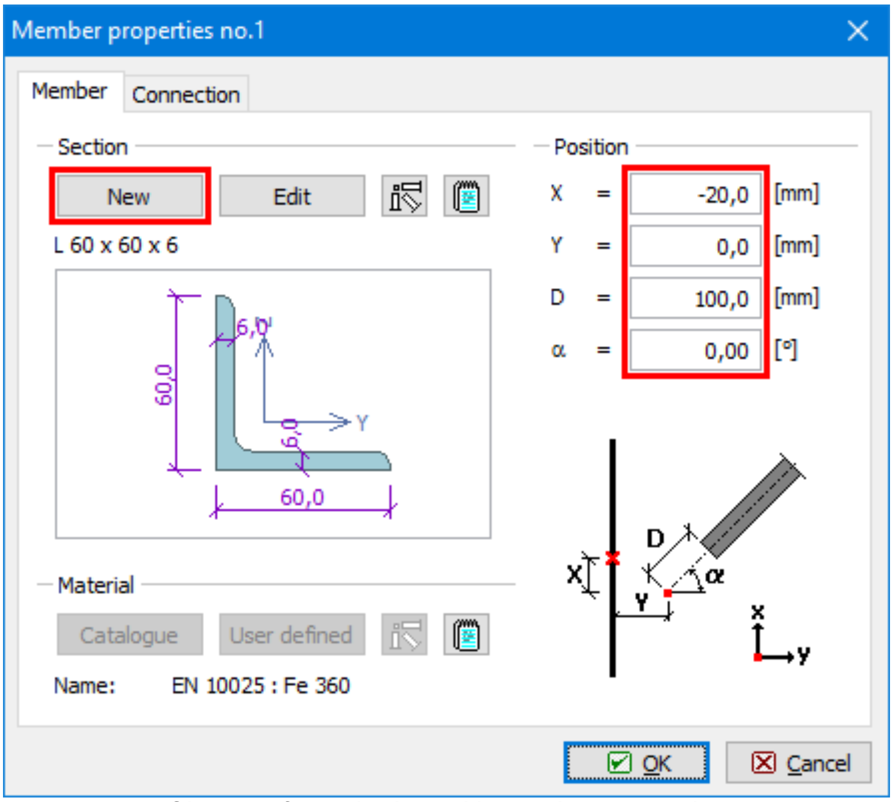
Only one connected member remains in the joint. This member has a wrong position. Therefore, we have to change its position.



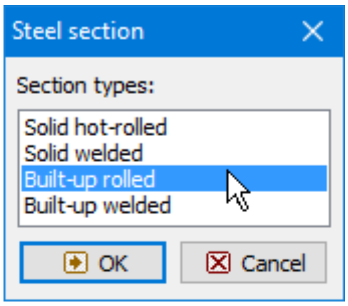

*Geometry of joint*

For editing the member, we can use the button "**Edit**" above the table of members or double-click on the corresponding row of the table. The window "**Member properties**" contains two tabs: "**Member**" and "**Connection**". We change the position of the member with the help of input fields on the right side of the tab "**Member**". We also have to change the cross-section. We use the button "**New**" in the left upper corner of this tab.



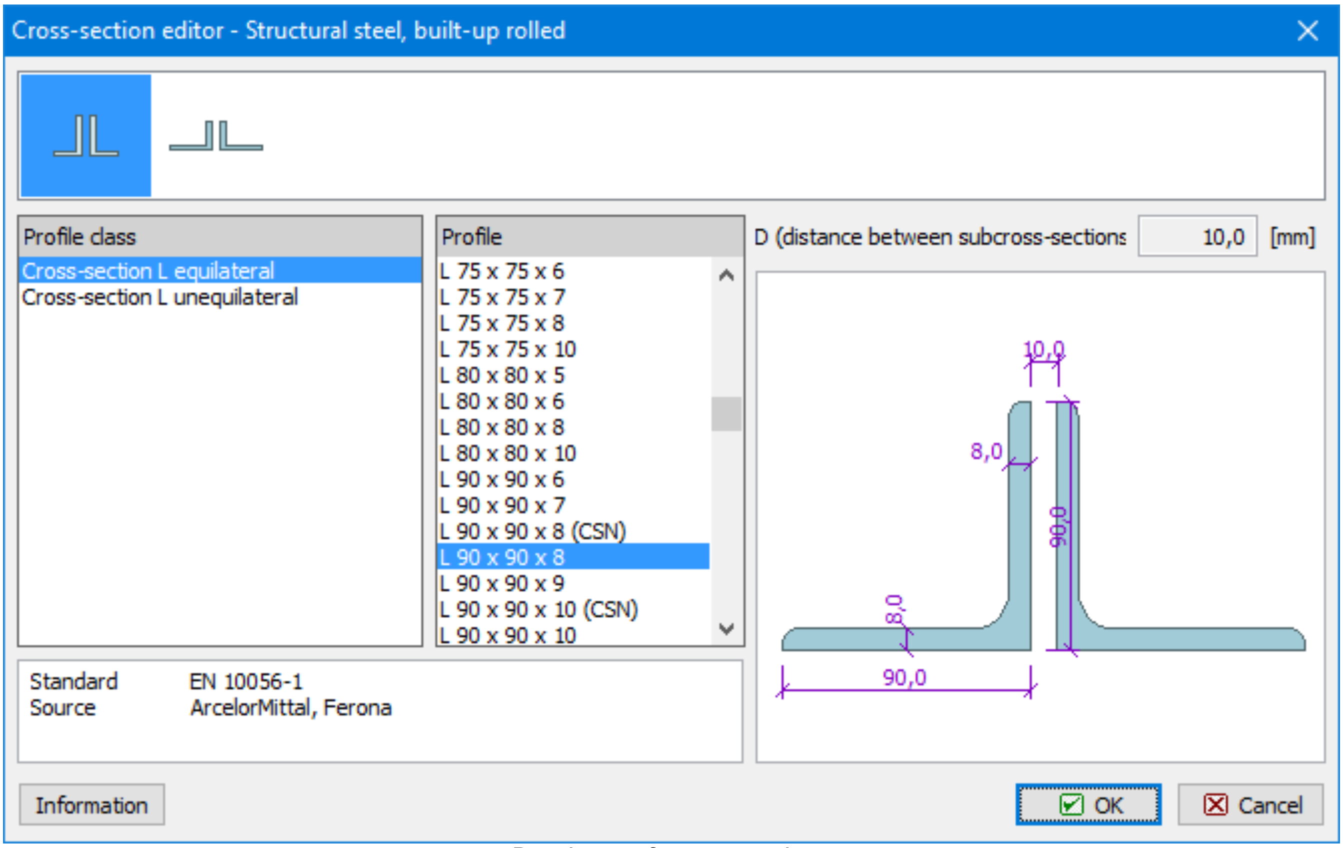

*Change of member's position and cross-section*

We choose type "**Built-up rolled**" in the next window.




The database of cross-sections opens. The type "**L 90x90x8**" has to be selected here.

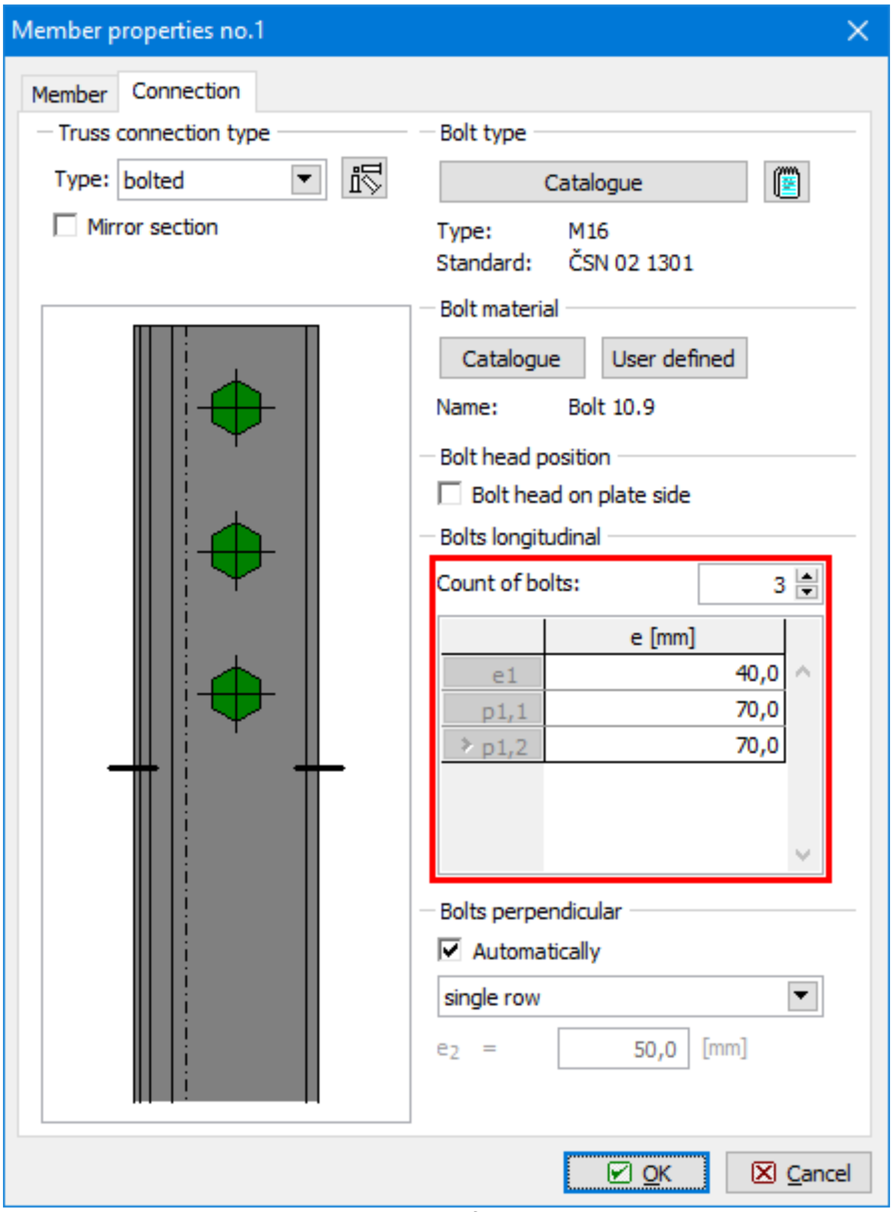



*Database of cross-sections*

We close the window using the button "**OK**".

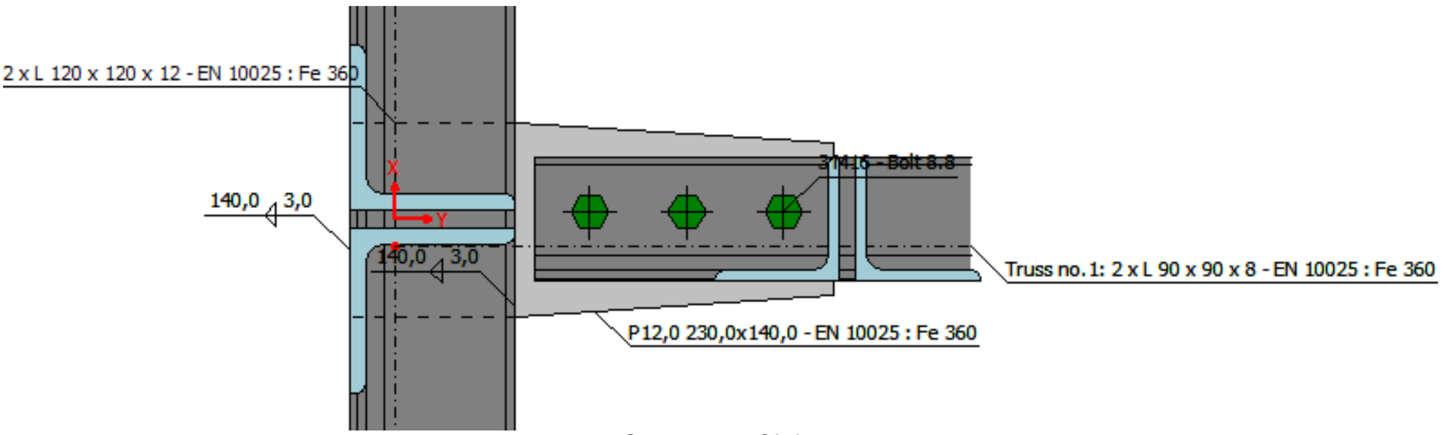
We switch to the tab "**Connection**" in the window "**Member properties**". It is necessary to increase the number of bolts to *3* and specify the distances between bolts *(40,70,70)* .




*Position of bolts*

We also change the material of bolts. In the same tab "**Connection**" of "**Member properties no.1**" we use the button "**Catalogue**" for selection of the material and select the material "**Bolt 8.8**". We close the window "**Member properties no.1**" by the button "**OK**". Final geometry of the joint can be seen on the following picture:

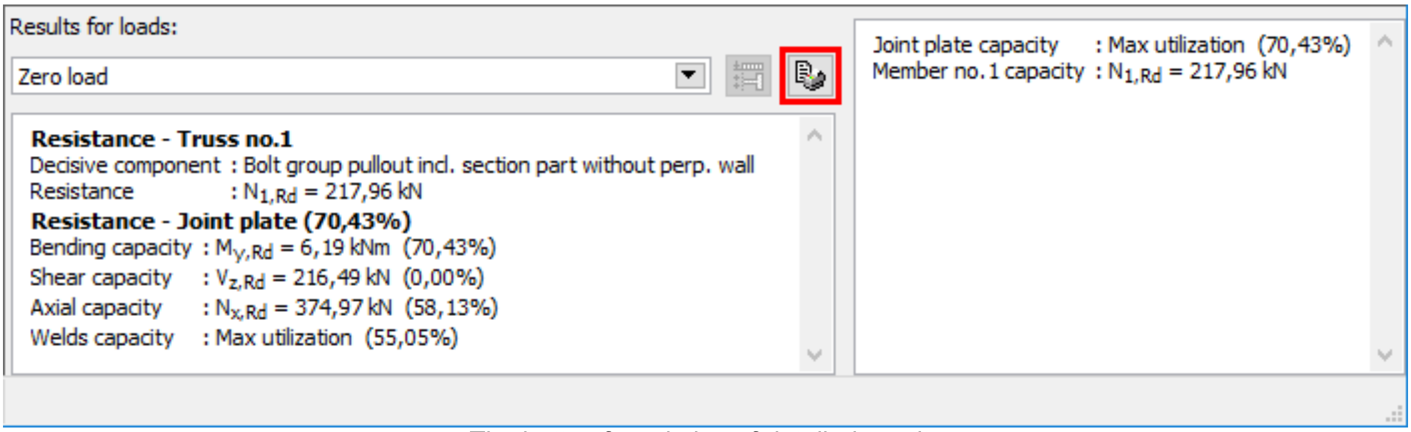



*Geometry of joint*

## Results

The results show that the maximum resistance of the connection is *217,96 kN*. The plate would have an utilization *70,43%* in this case. Detailed results are available only for "**Zero load**", as we have not specified any load. The detailed analysis can be printed easily using the button "".




*The button for printing of detailed results*

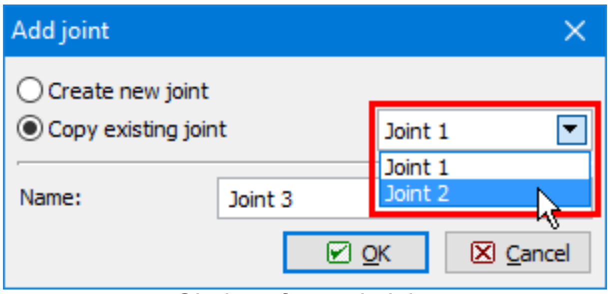
# Web to chord connection

## Introduction

This guide shows how to create a new joint using an existing one. The joint created in the previous guide "**Steel truss joint**" will be copied and will be converted into welded joint.

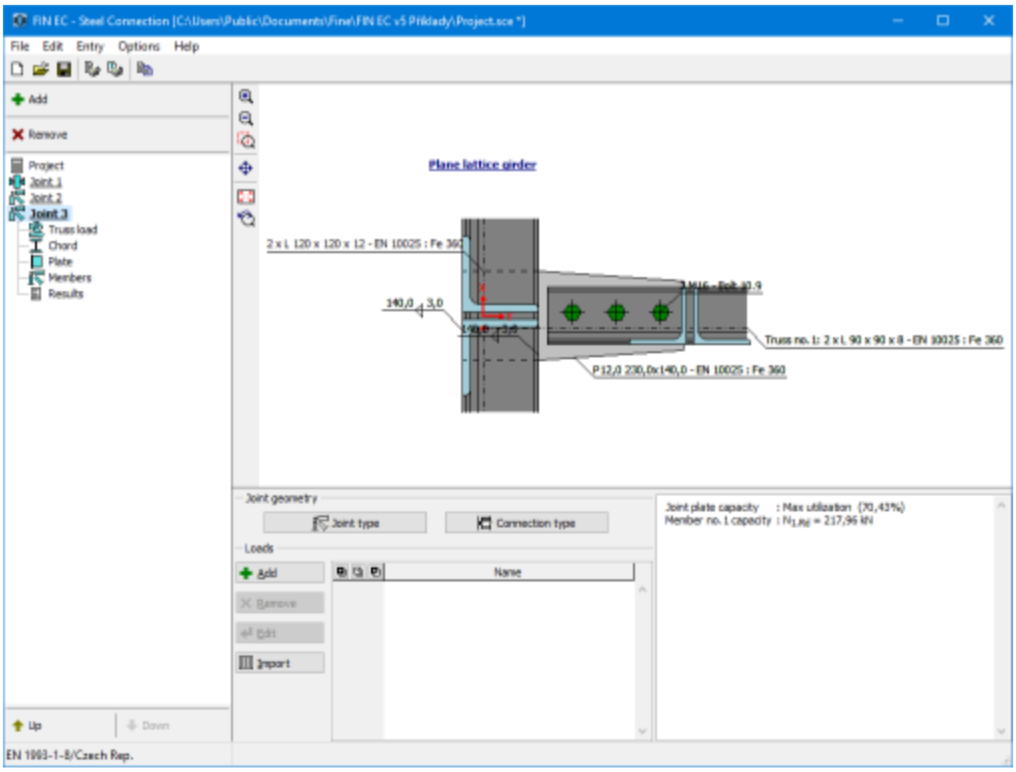
## Creation of a new joint

Similarly to previous joints, we start with insertion of a new joint with the help of the button "**Add**" in the tree menu. We select an option "**Copy existing joint**". The list of existing joints appears on the right side, we select "**Joint 2**" there. Additionally, we change the name and close the window by the button "**OK**".




*Choice of sample joint*

After the closing of the window, the new joint including appropriate data structure appears in the tree menu.

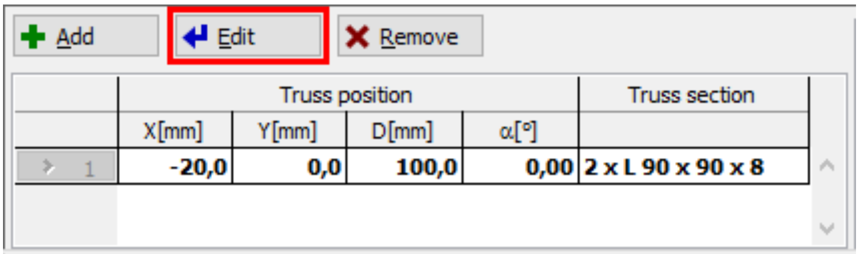


*Basic screen for the new joint*

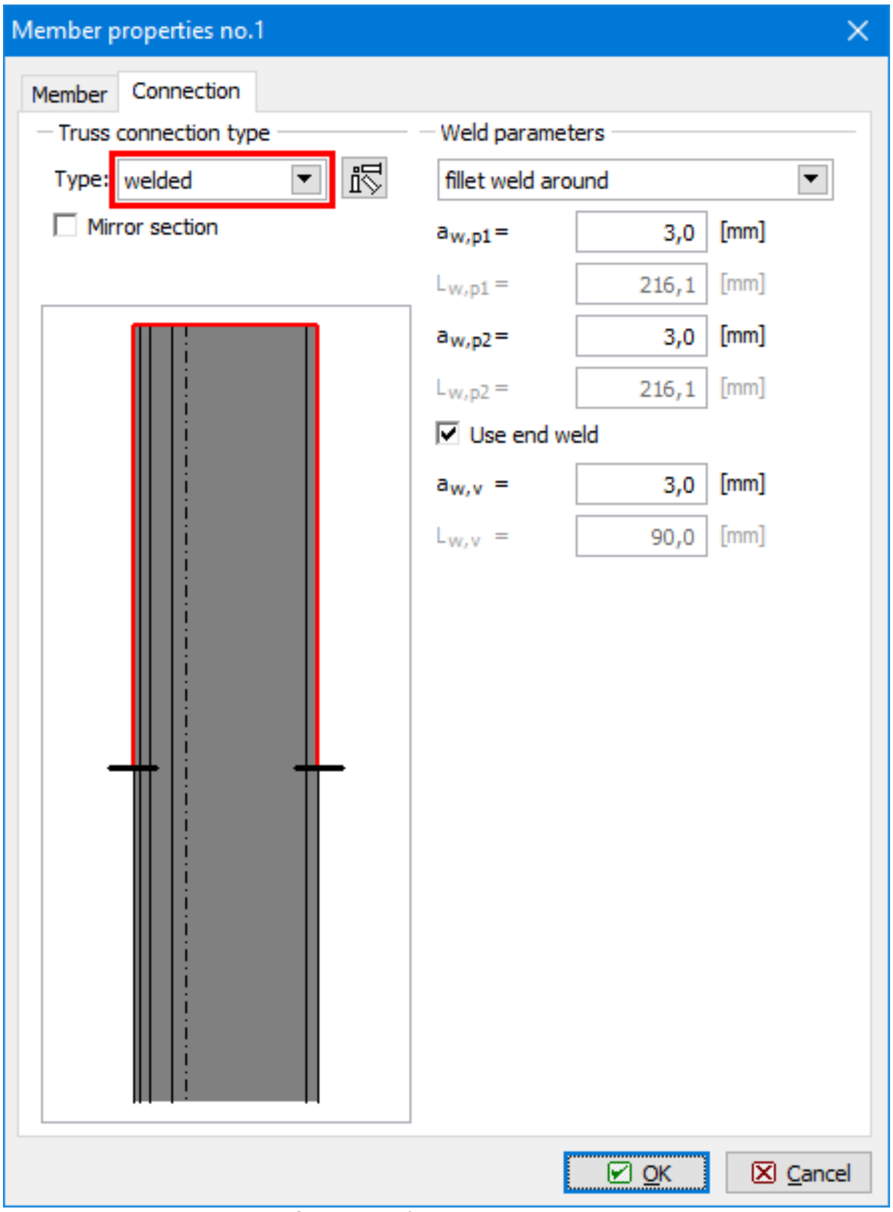
## Changes of topology

We only need to change the connection style. Therefore, we switch directly to the part "**Members**" of the tree menu and edit the connected member. We can use the button "**Edit**" above the table or double-click on the corresponding row of the table.




*Editing of connected member*

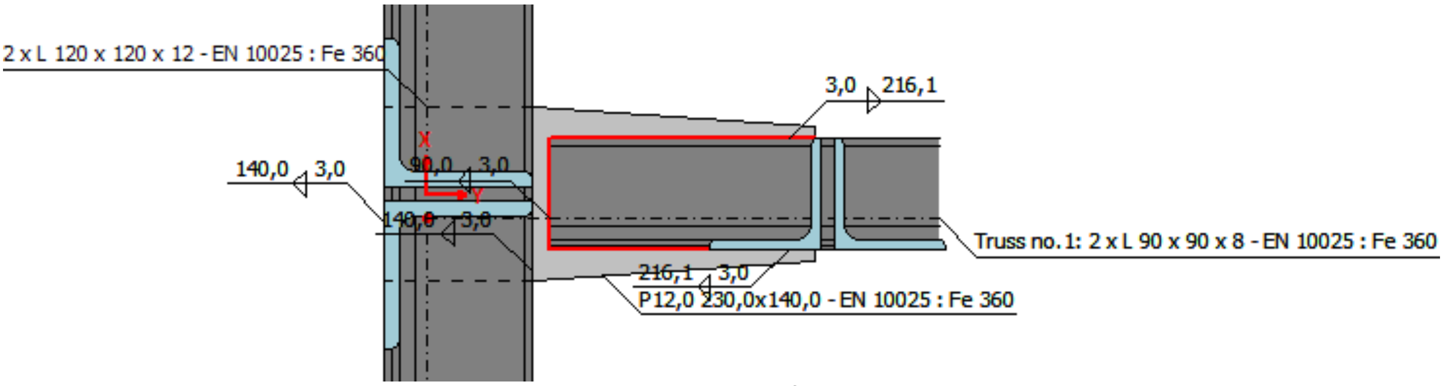
We switch to the tab "**Connection**" and change the connection type in the right upper corner to "**Welded**". The parameters of welds appears, we keep default settings. The change has to be confirmed by the button "**OK**".




*Change of connection type*

The final joint is shown on the following figure.

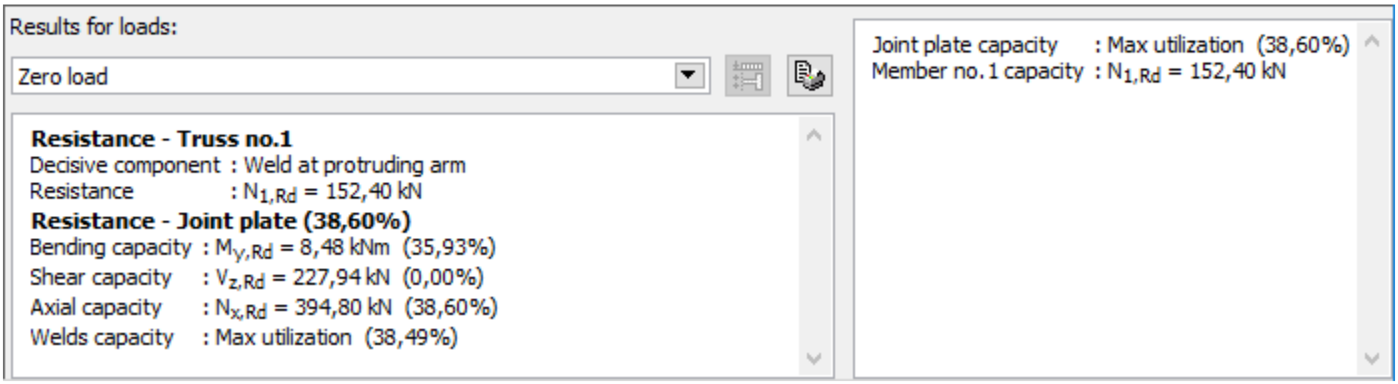
 




*Final geometry of joint*

## Results

The results show that the maximum resistance of the connection is *152.4 kN*. The plate would have a utilization *38.6%* in this case. The decisive component is welded connection.




*Results*

# Anchorage of column with end plate

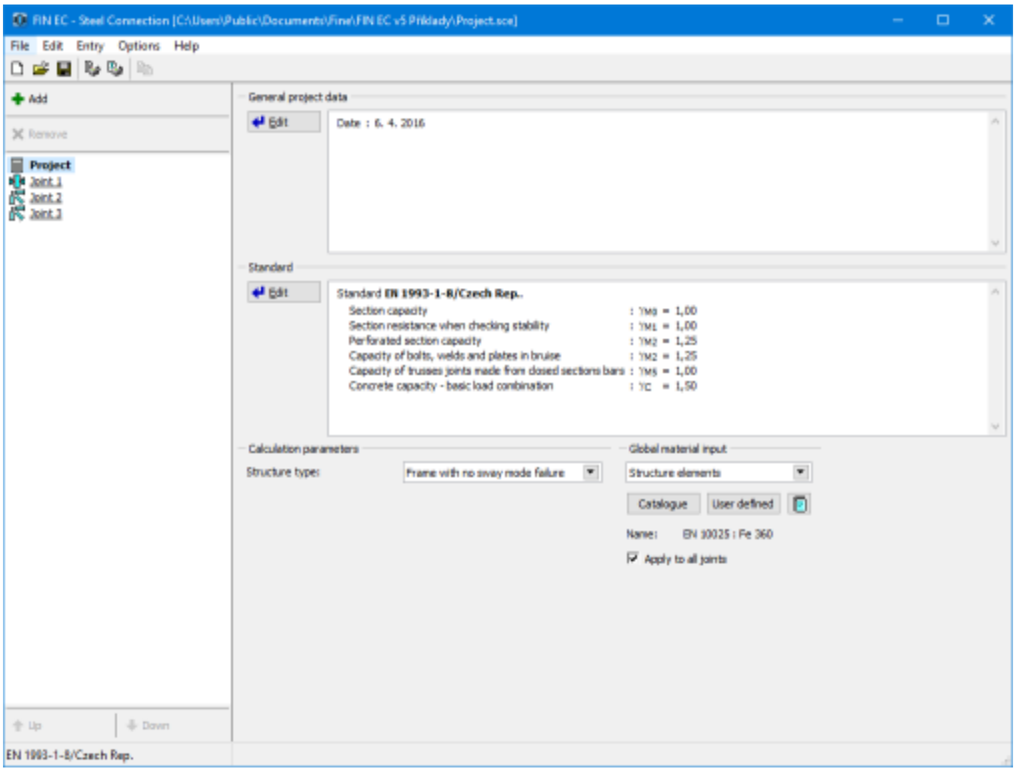
## Introduction

This guide shows how to calculate resistance of column connection to the concrete base using detail with end plate and haunches. The column is loaded by the axial force $N_x = 500kN$ and bending moment $M_y = 60kNm$. The steel class *EN 10025:Fe360* and concrete class *C20/25* are used.

| | |
|---|---|
| Column: | $b = 150mm$, $h = 300mm$, $t_w = 12mm$, $t_f = 16mm$ |
| Haunches: | $t_w = 16mm$, $h_w = 90mm$, $L_w = 300mm$, $L_f = 50mm$, $a_w = 6mm$ |
| Concrete base: | $b_b = 1600mm$, $a_b = 1600mm$, $h_b = 1000mm$, $t_g = 30mm$ |
| Welds: | $a_{w,f} = 8mm$; $a_{w,w} = 6mm$ |
| End plate: | $b_p = 180mm$, $h_p = 510mm$, $t_p = 30mm$, $a_1 = -105mm$ |
| Bolts: | *M24 10.9*, $w_1 = 45mm$, $e = [50,410]mm$ |

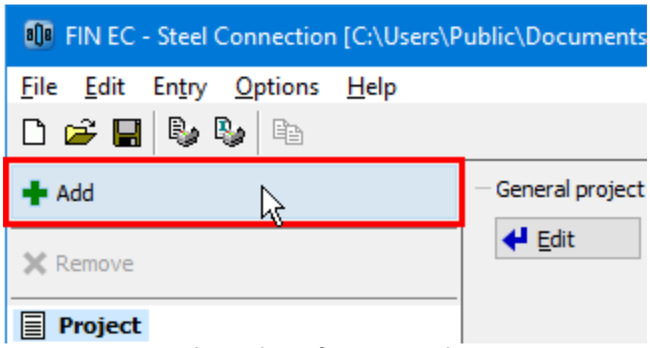
## Starting a new task

We use a project created in previous tutorials.



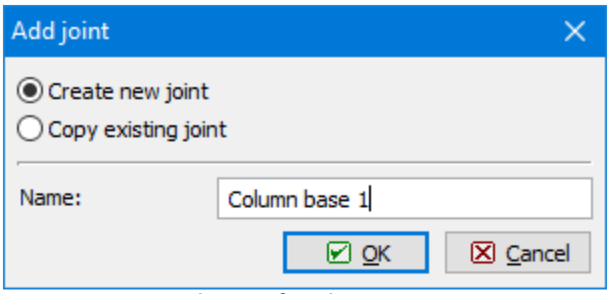
*Main screen*

We add a new task with the help of the button "**Add**" at the top of the tree menu



*Insertion of a new task*

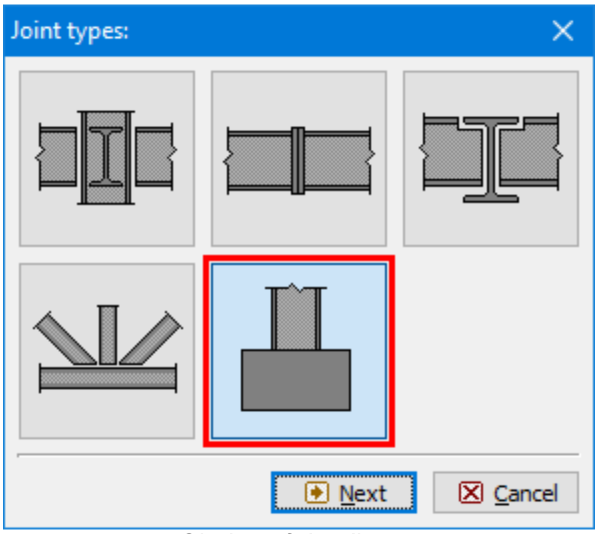
We specify a name ("**Column base 1**") in the window "**Add joint**", that appears after the clicking on that button.



*Input of task name*

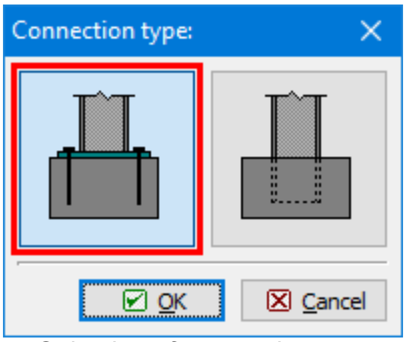
After the confirmation by the button "**OK**", the window with main joint types appears. We select column base (in the middle of the upper row).



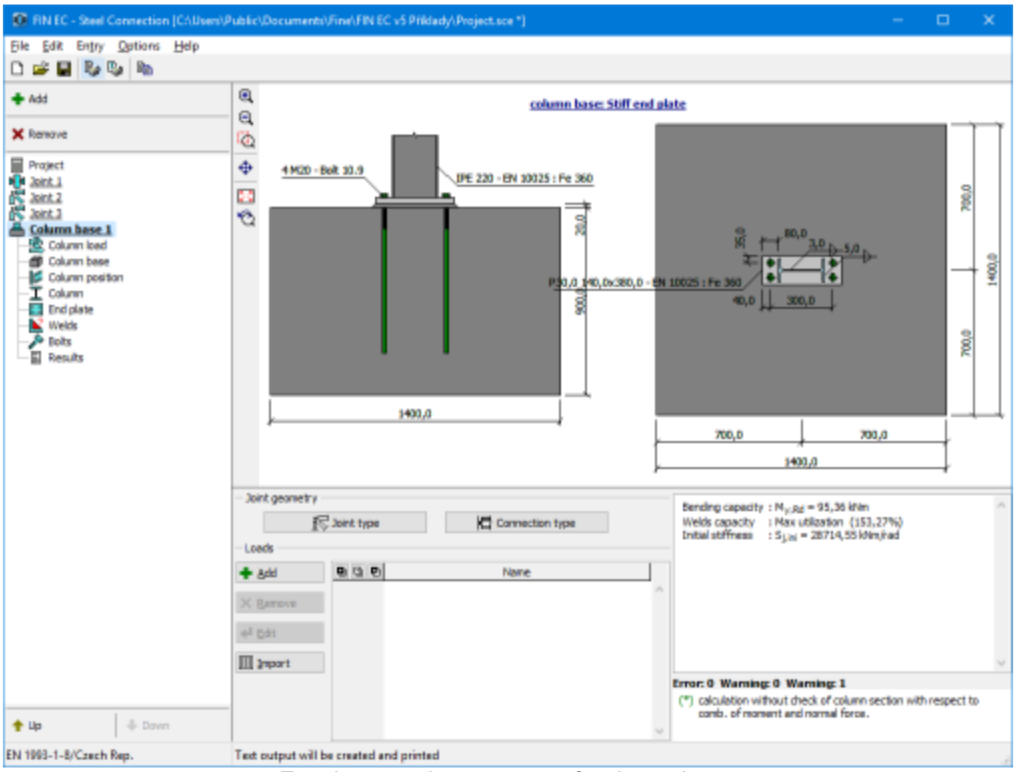
*Choice of detail type*

The choice of connection type follows. We select connection with end plate (left option) and confirm the choice by the button "**OK**".



*Selection of connection type*

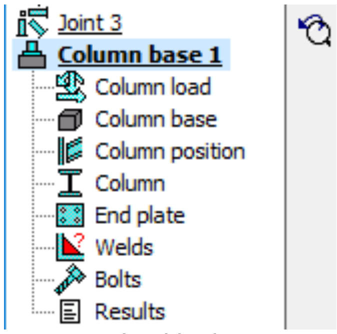
The initial geometry of the joint appears in the workspace.



*Fundamental geometry of column base*

The following work is done with the help of the tree menu on the left side of the program. The structure of this menu is generated according to the specified joint geometry.

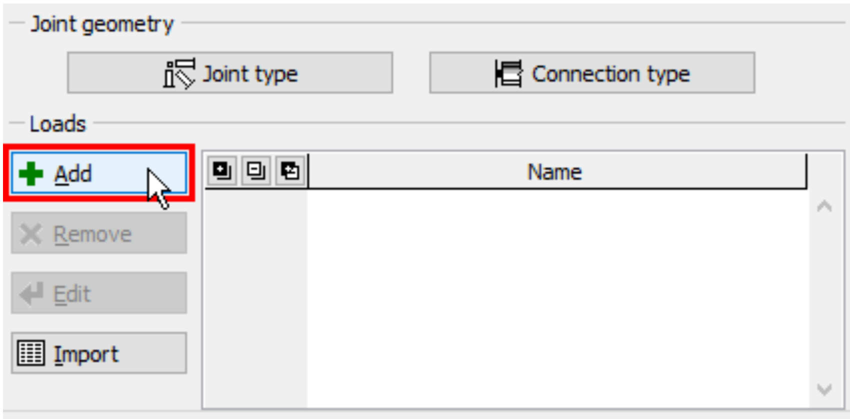
 




*Inputs organized in the tree menu*

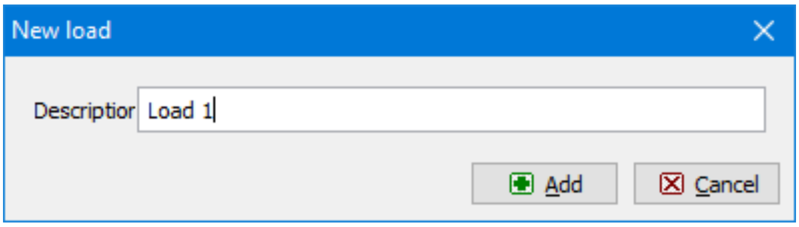
We go through all parts from the top to the bottom and modify inputs. The connection to the left flange will be described in detail. The connection to the right flange would be solved in the same way.

On the main screen of the joint, it is possible to change the specified geometry of the joint (buttons "**Joint type**" and "**Connection type**") and specify list of loads. The load represents a set of internal forces that have to be defined for all members in the joint (column, beams). These internal forces should be resulting values of certain load combination. Therefore, they are considered as design values. Number of loads for the joint is not limited. We insert a new load with the help of the button "**Add**" in the toolbar on the left side of the loads table. The toolbar also contains buttons for editing and deletion of loads and also a tool for import of loads including internal forces from **.txt* or **.csv* file.




*Button for input of new load*

The new is specified by a name. The input has to be confirmed by the button "**Add**".



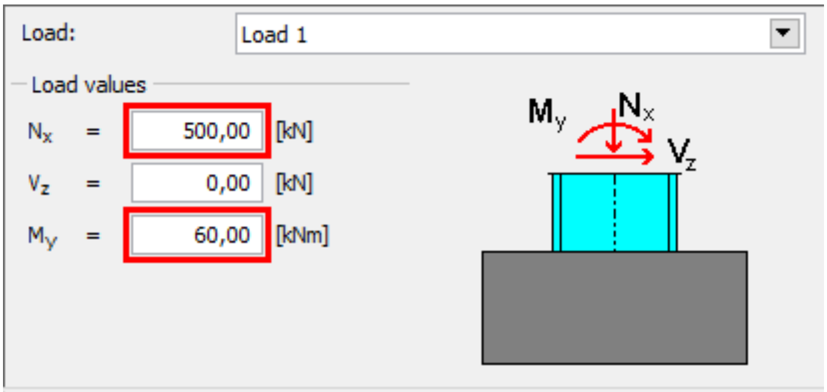

*Window "New load"*

As we want to add only one load, we close the window by the button "**Cancel**" after the input of first load.

## Column load

If at least one load is entered, it is possible to switch to the part "**Column load**" in the tree menu and specify internal forces for the column. The axial force "$\mathbf{N_x}$" should be *500kN* and the bending moment "$\mathbf{M_y}$" is *60kNm*.

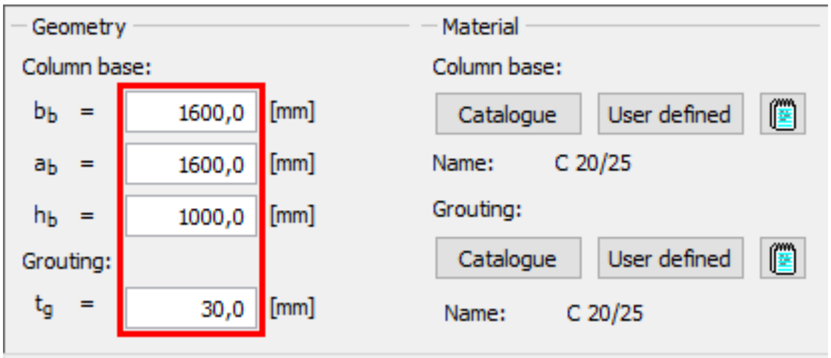
 




*Input of forces*

## Column base

This mode contains dimensions of concrete base ("**$b_b$**" - width, "**$a_b$**" - length, "**$h_b$**" - height) and thickness of grouting "**$t_g$**".



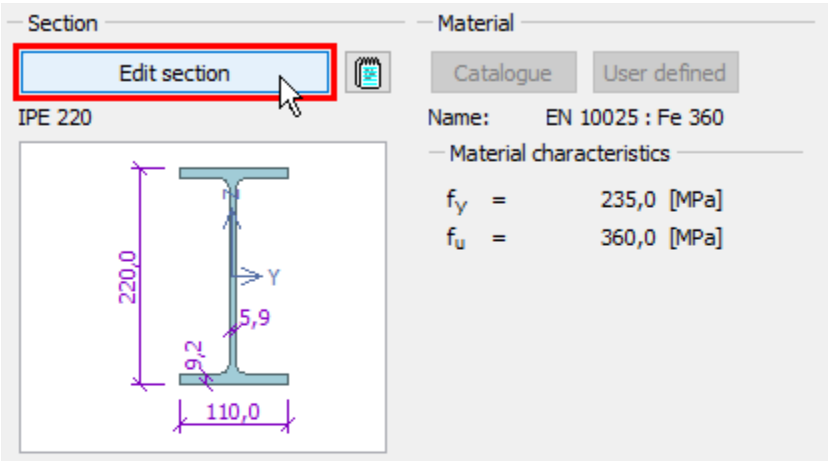

*Dimensions*

## Column position

The position of the column relatively to the centre of concrete base and the column pitch can be specified here. We do not change default values and go to another part.

## Column

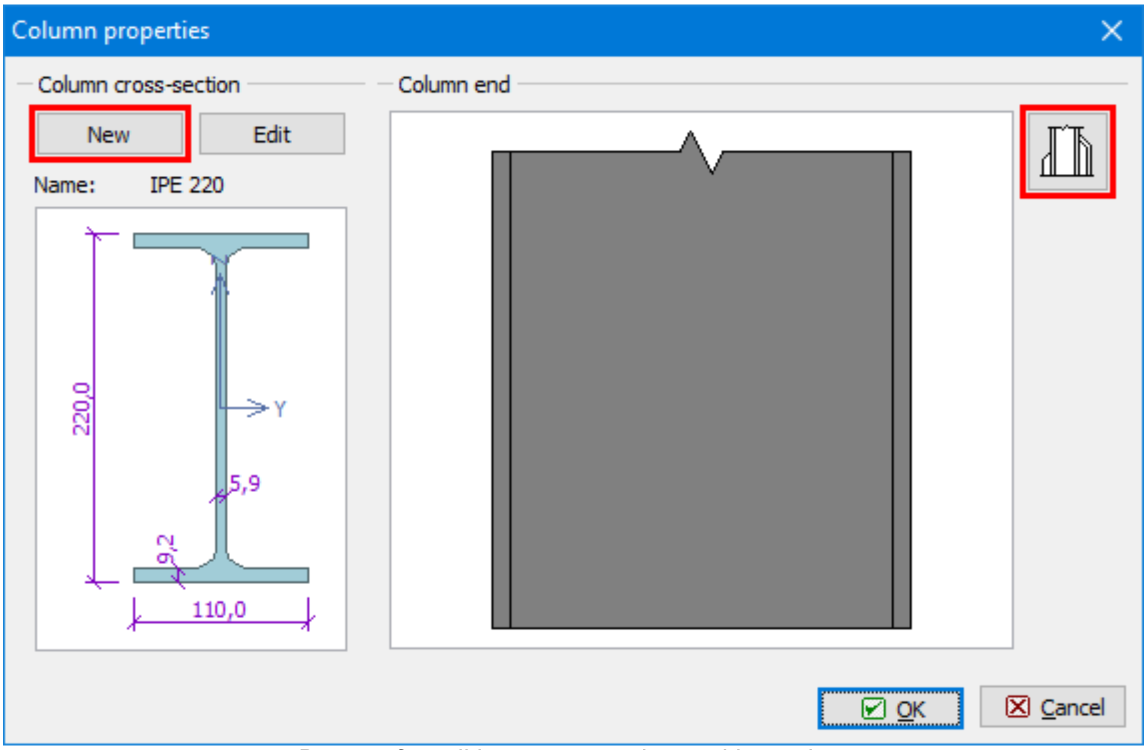
We specify the column geometry (cross-section, haunches) with the help of the button "**Edit section**".




*The button for edit of beam geometry*

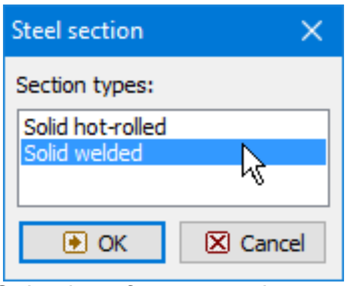
The window "**Column properties**" appears. Left part of the window contains input of cross-section, right part input of haunches. We change the cross-section with the help of the button "**New**".



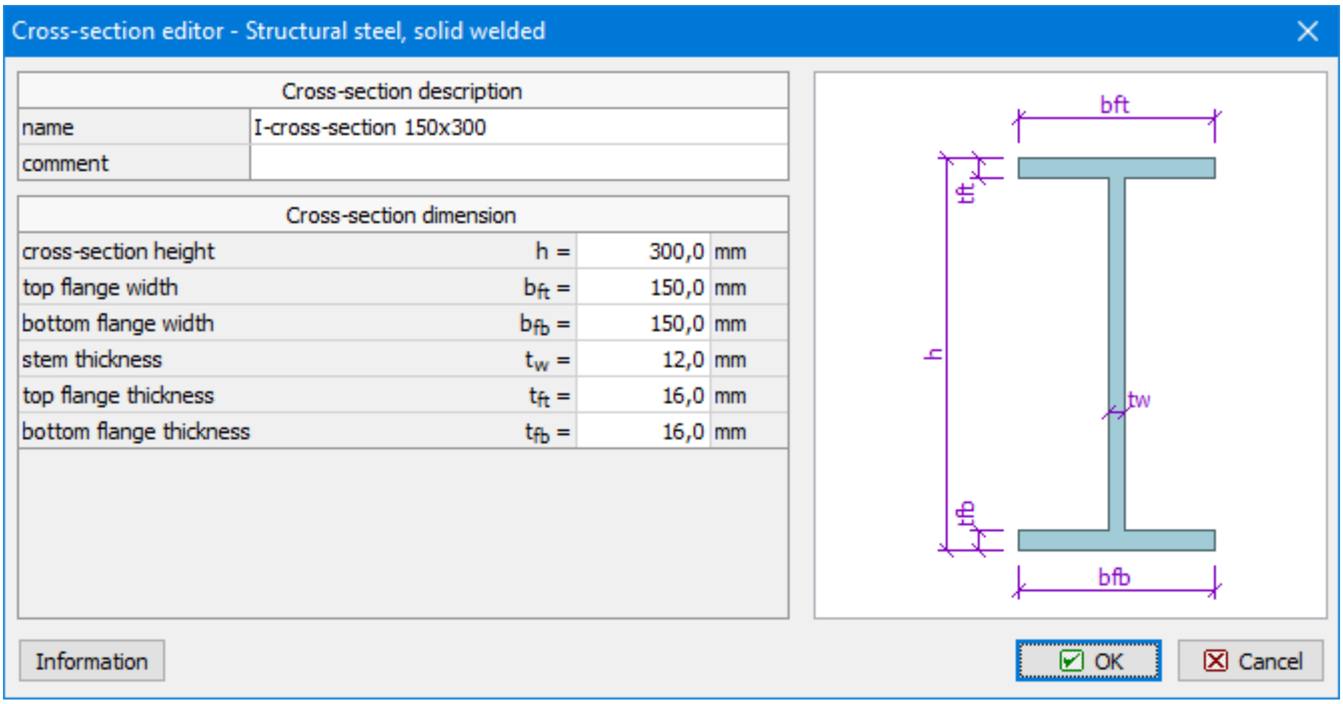

*Buttons for editing cross-section and haunches*

The window "**Steel section**" that appears after the clicking on the button contains an option to select type of cross-section. We select an option to specify arbitrary dimensions of welded cross-section (the option "**Solid welded**") and open the window "**Cross-section editor**" by pressing the button "**OK**".




*Selection of cross-section type*

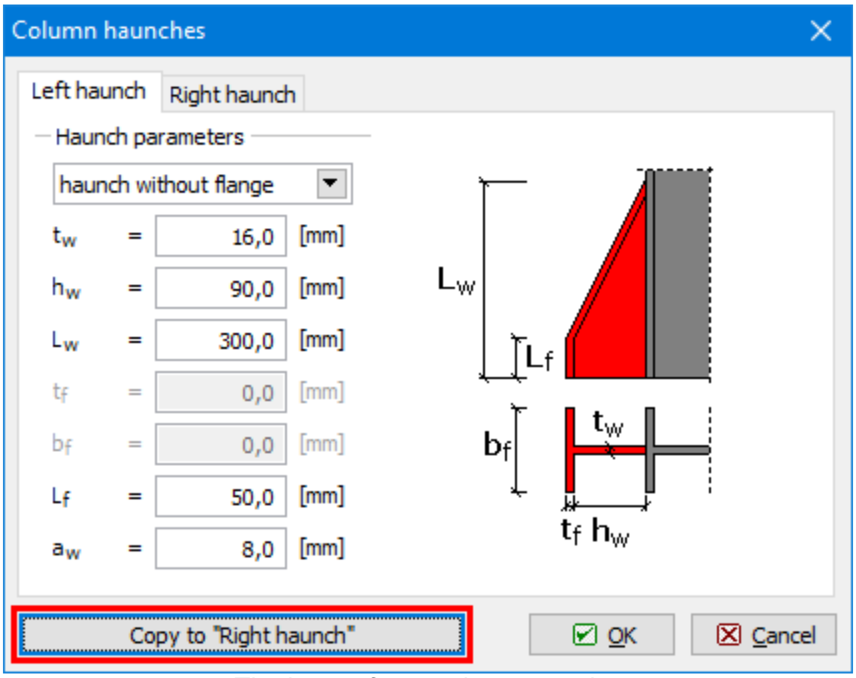
We enter the cross-sectional dimensions and confirm the input by the button "**OK**".



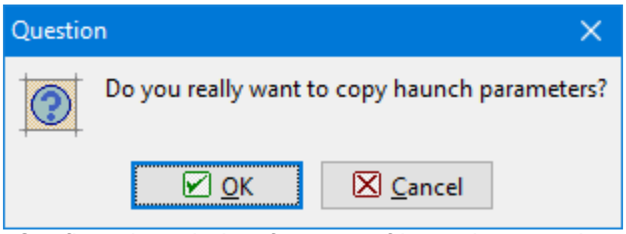
*Dimensions of welded cross-section*

The button in the part "**Column end**" opens the window "**Column haunches**". We select an option "**Haunch without flange**" and specify dimensions according to the following figure in the tab "**Left haunch**". As the column cross-section is symmetrical, we copy the haunch also to the right flange. We use the dedicated button "**Copy to "Right flange"**" in the left bottom corner for this operation.




*The button for copying properties*

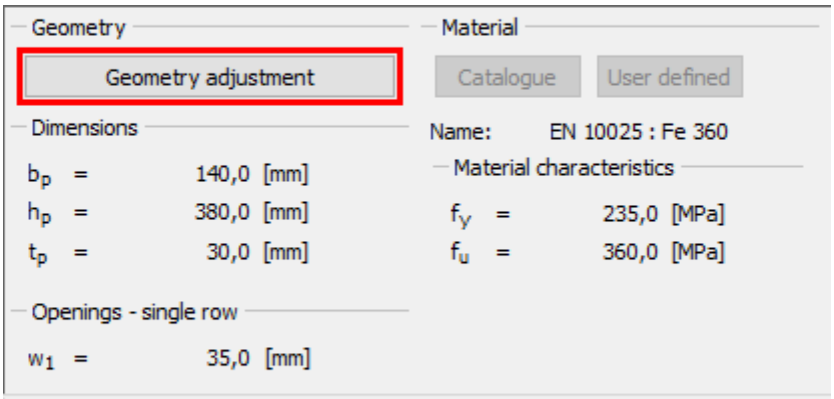
The copy of haunch properties have to be confirmed in a new window.




*Confirmation window for copy of haunch properties*

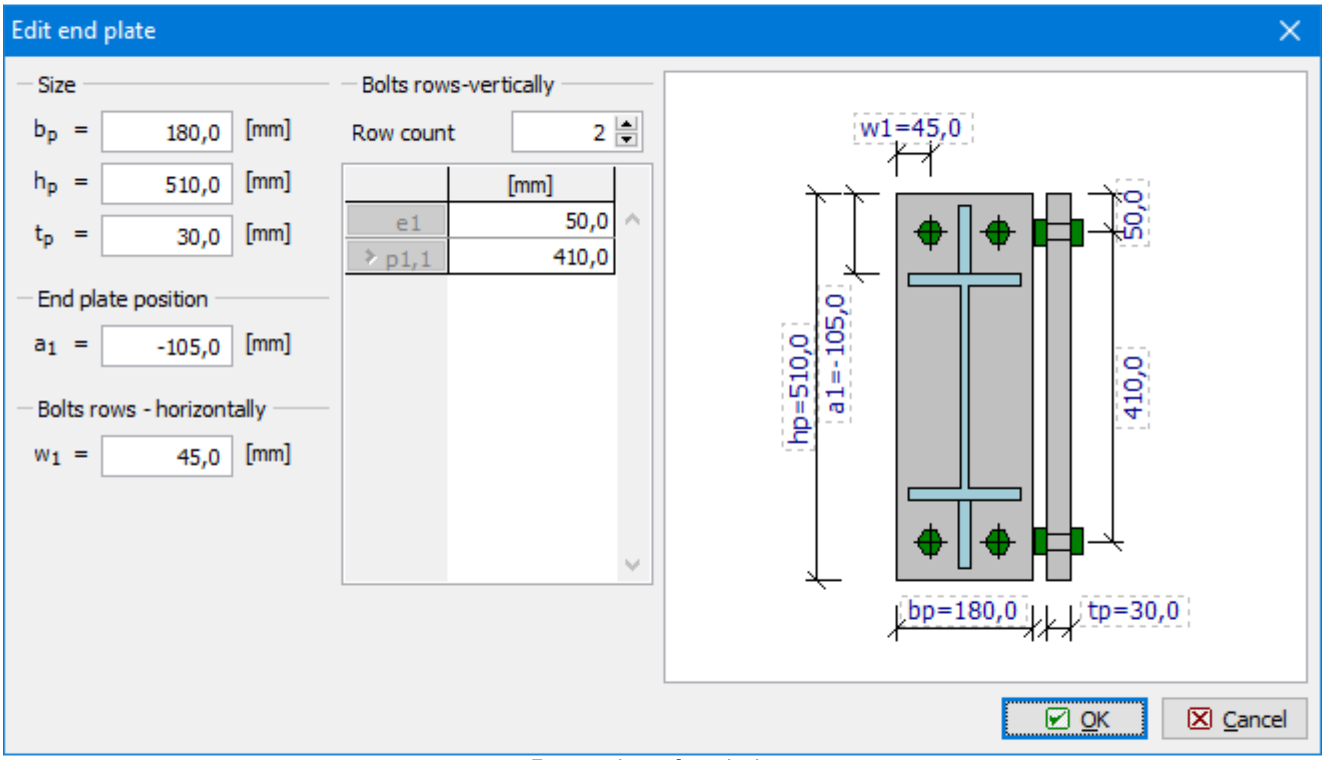
## End plate

The input of end plate geometry follows. We open an appropriate window using the button "**Geometry adjustment**" in the bottom frame.




*The button for editing plate properties*

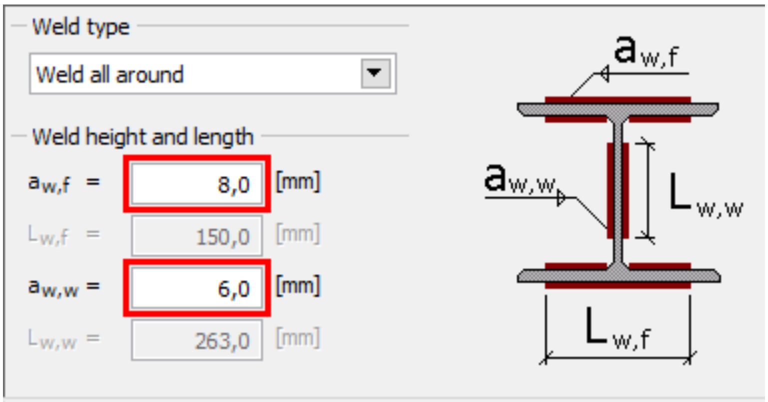
We specify end plate dimensions "$\mathbf{b_p}$", "$\mathbf{h_p}$", "$\mathbf{t_p}$", position of end plate relatively to the beam edge "$\mathbf{a_1}$", horizontal position of bolts "$\mathbf{w_1}$" and vertical positions of rows. Entered values are shown in the figure below. The input can be done with the help of input lines in the left part of the window or using active dimensions in the end plate figure in the right part of the window. The input has to be confirmed by the button "**OK**".



*Properties of end plate*

## Welds

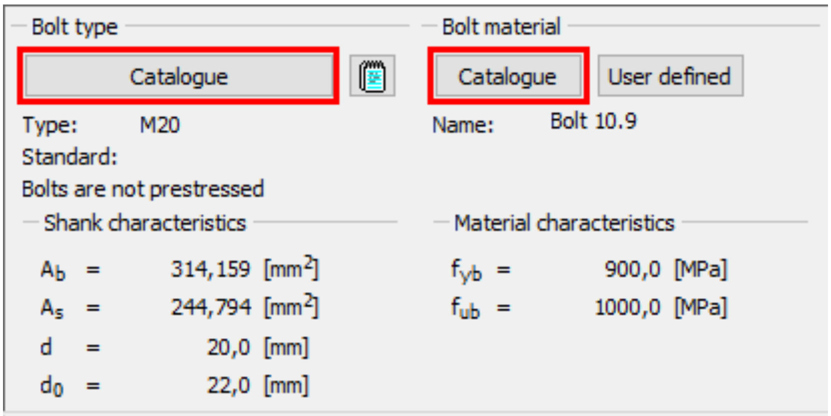
Fillet welds all round will be used for the connection of the end plate to the column. Therefore, we select weld type "**Weld all around**" and enter the throat thickness for flanges "$\mathbf{a_{w,f}}$" and the throat thickness for the web "$\mathbf{a_{w,w}}$". Lengths are calculated automatically according to the geometry of the cross-section. Arbitrary lengths can be defined using weld type "**User defined weld**".



*Welds properties*

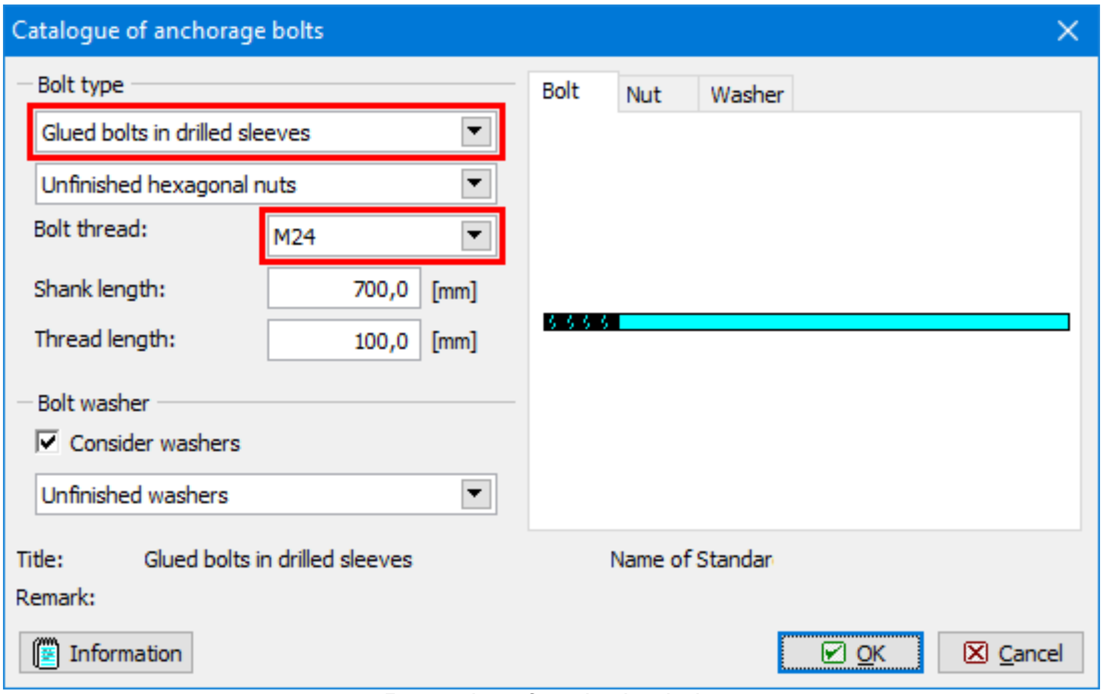
## Bolts

The part "**Bolts**" contains the input of bolt type, dimensions and material. The type and dimension can be specified with the help of the button "**Catalogue**" in the part "**Bolt type**" of the bottom frame. The material can be selected using the button "**Catalogue**" in the part "**Bolt material**".



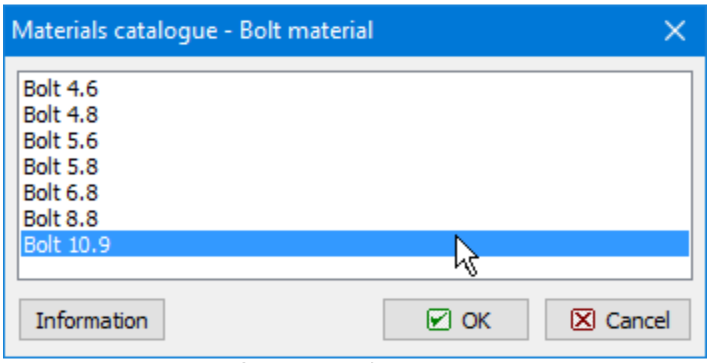

*Buttons for input of bolt type and material*

We select bolt type "**Glued bolts in drilled sleeves**" and the diameter "**M24**". The window has to be closed by the button "**OK**".




*Properties of anchoring bolts*

We choose the material "**Bolt 10.9**" in the window "**Materials catalogue**" and confirm the input by the button "**OK**".

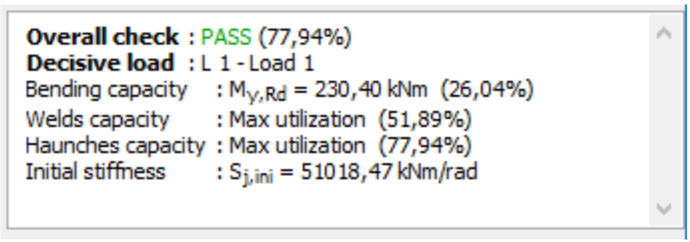



*Selection of material*

## Results

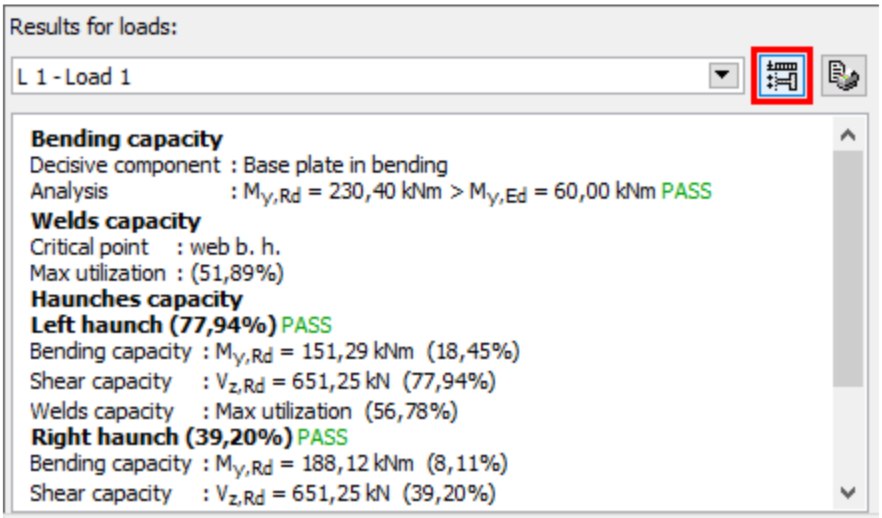
The total results are displayed in the right part of the bottom frame. They contain maximum utilization, the decisive load and connection and also brief results for decisive load.



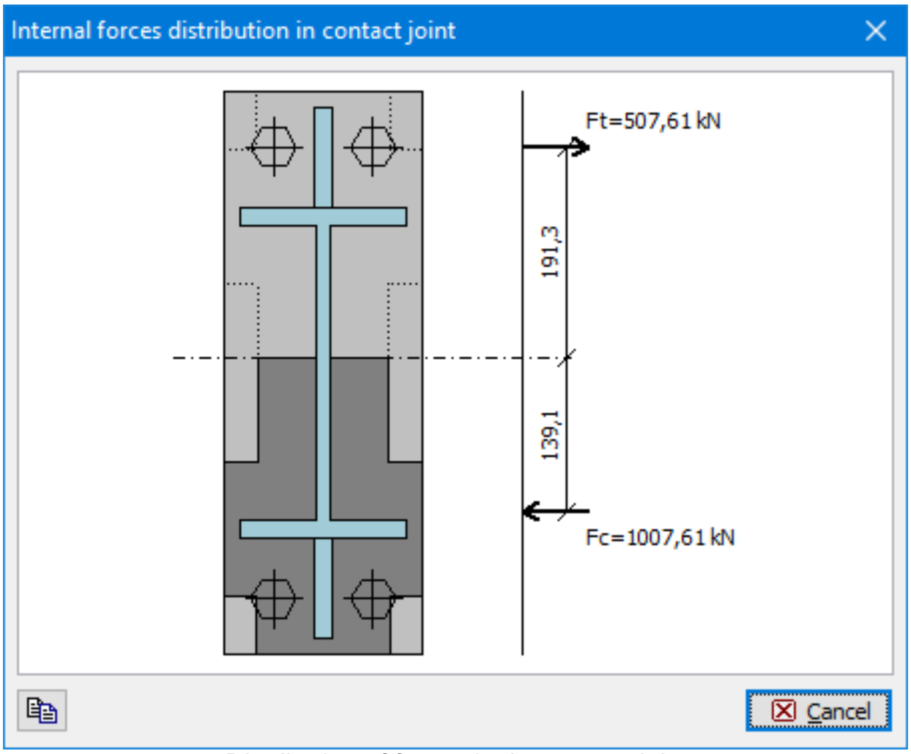

*Overall results*

Th detailed results for particular components are displayed in the mode "**Results**". These results contain detailed bearing capacities and decisive components. The results can be displayed for all entered loads. Available is also an option "**Zero load**". In this case, maximum bearing capacities for all components are displayed.




*The button for scheme of forces in contact joint*

The button on the right side of the load list (can be seen in the figure above) is able to open a scheme of contact joint with distribution of forces.




*Distribution of forces in the contact joint*

# Steel column anchored in concrete footing

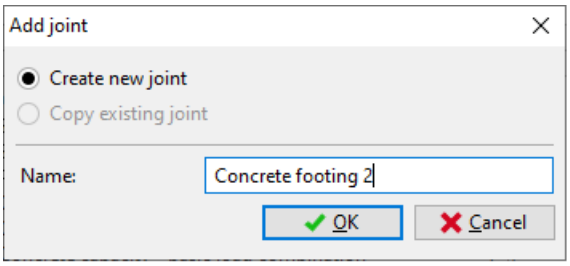
## Introduction

Calculate the capacity of steel column - concrete footing connection. The steel column is anchored in the concrete footing. The depth of the anchorage is 600mm and the column is located in the centre of the footing. The values of internal forces acting in the connection are: $N_x$ = 694kN, $M_y$ = 140kNm $V_z$ = 80kN. The material of the column is steel EN 10025: Fe360 and the footing is made of concrete grade C20/25.

| | |
|---|---|
| Column: | *HE 240B* |
| Footing: | $b_b$= 1600mm, $a_b$= 1600mm, $h_b$= 1000mm |
| End plate: | $b_p$= 280mm, $h_p$= 280mm, $t_p$= 30mm |

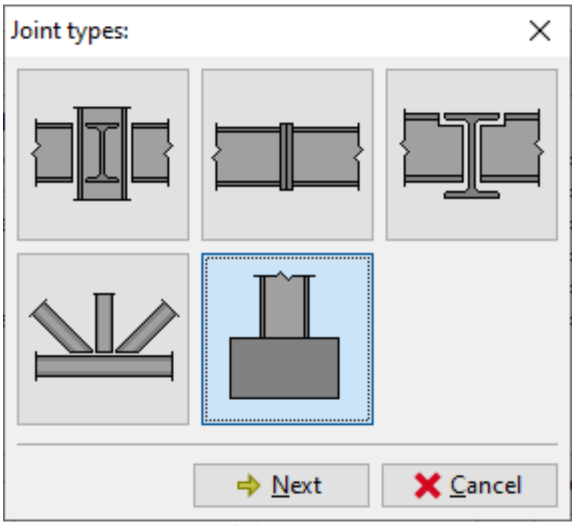
## Creating new project

New connection to the project can be added by clicking the button "**Add**" in the main dialogue window. The "**Add joint**" dialogue window will appear, change the joint name to "**Concrete footing 2**" and confirm by clicking the button "**OK**".



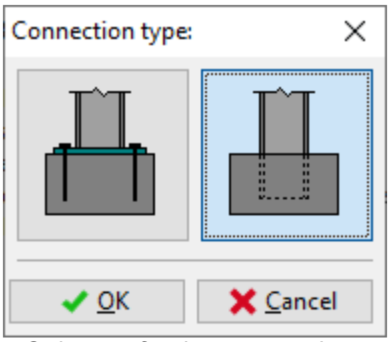
*Creating new joint*

Next, the "**Joint types**" dialogue window will open automatically, choose the preferred type of joint. In this example, it is the column - footing joint.



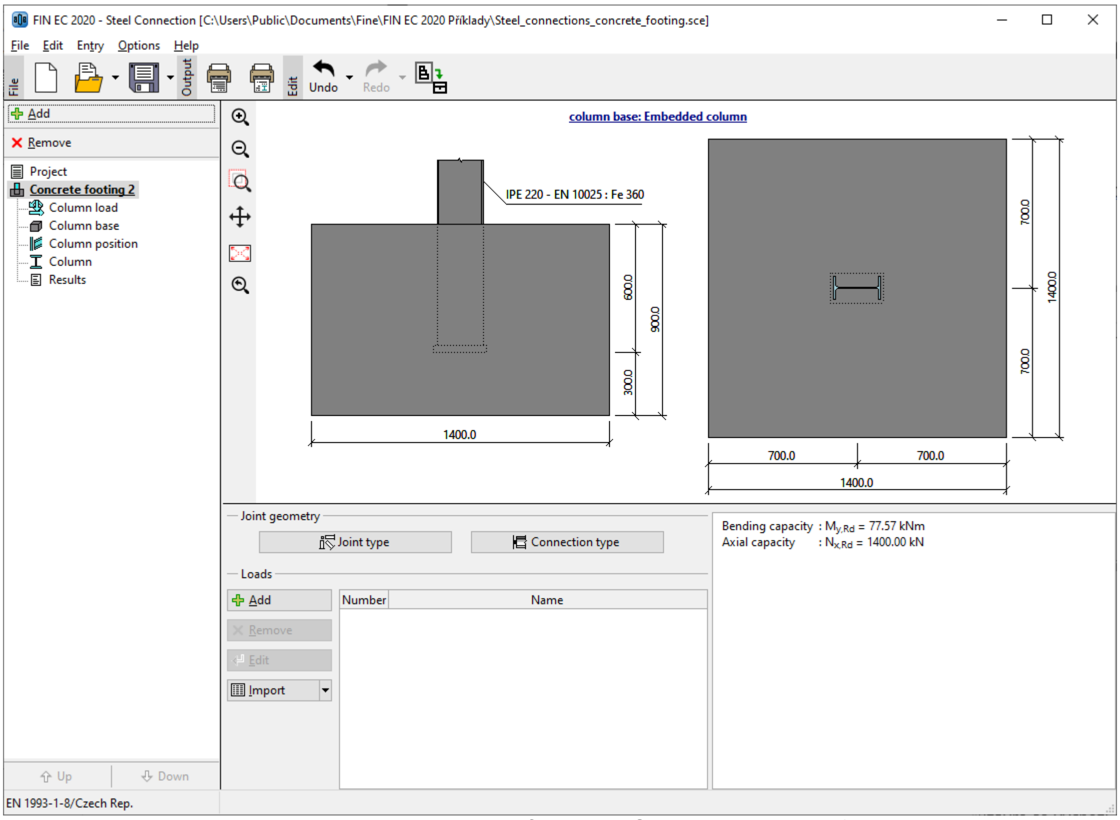
*Joint types*

Continue by clicking the button "**Next**", the dialogue window with the types of connection will appear. Pick the option, where the column is anchored in the concrete footing.



*Column - footing connections*

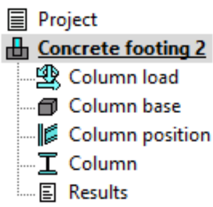
When this input is defined, the software will create new task named "**Concrete footing 2**" at the left side of the main dialogue window.




*Main dialogue window for task "Concrete column 2"*

## Defining properties of the connection

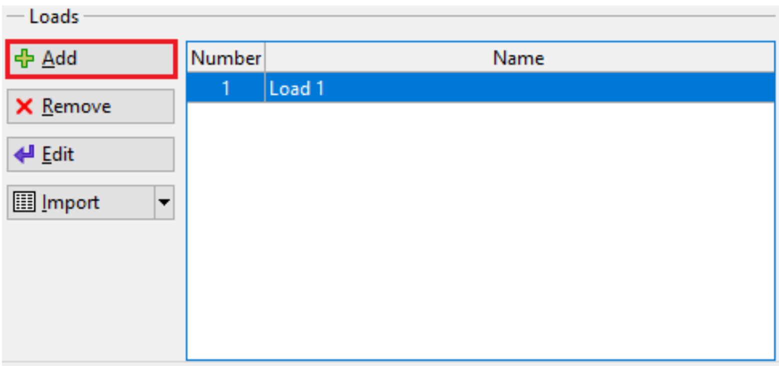
The connection properties are defined in the control tree under the task "**Concrete column 2**". Every connection has it's own control tree.




*Control tree for the selected task*

## Load cases

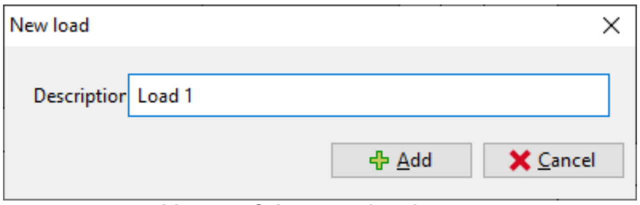
First, create new load case by clicking the button "**Add**" in the "**Loads**" section.




*Adding new load case to the task*

The dialogue window, with name of the new load case will appear. Keep the name "**Load 1**" and continue.

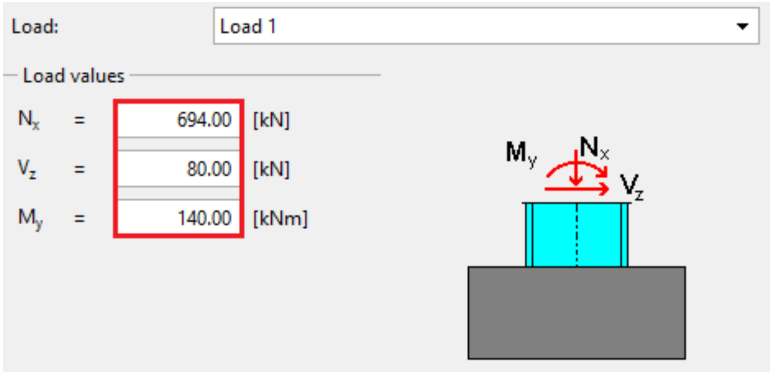
 




*Name of the new load case*

## Column load

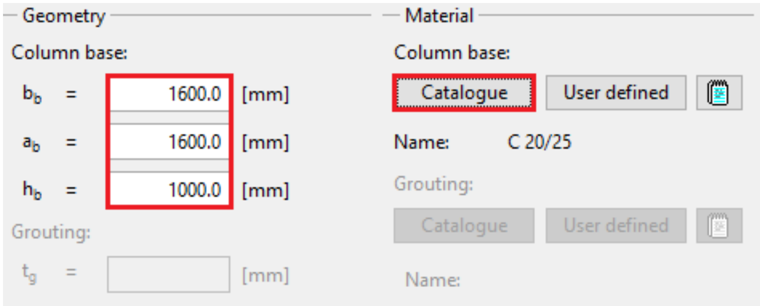
The column load values are defined in the "**Column load**" section in the control tree. The "**Load values**" are $N_x$ = *694kN*, $M_y$ = *140kNm* $V_z$ = *80kN*.




*Defining load values*

## Column base

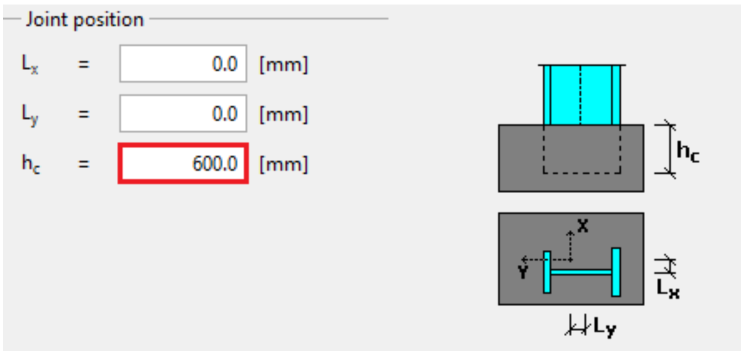
In this section, define the values of the concrete footing geometry and material. When the geometry is changed, the graphical preview of the footing will automatically re-generate its dimensions.




*Geometry and material of the footing*

## Column position

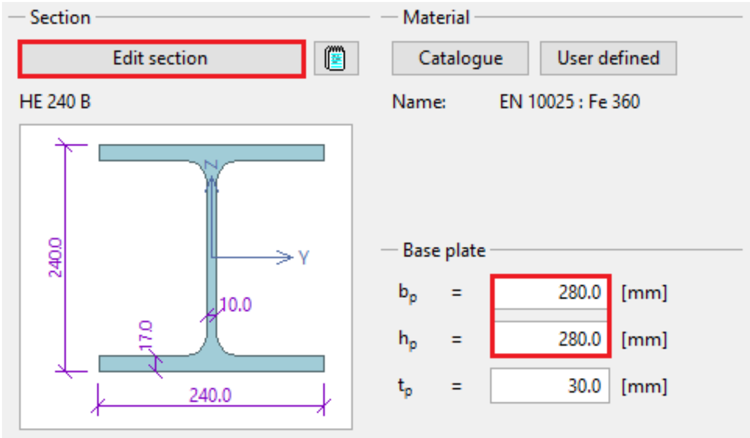
Now, insert the anchorage depth of the steel column in the concrete footing, $h_c$ = *600mm.*




*The joint position*

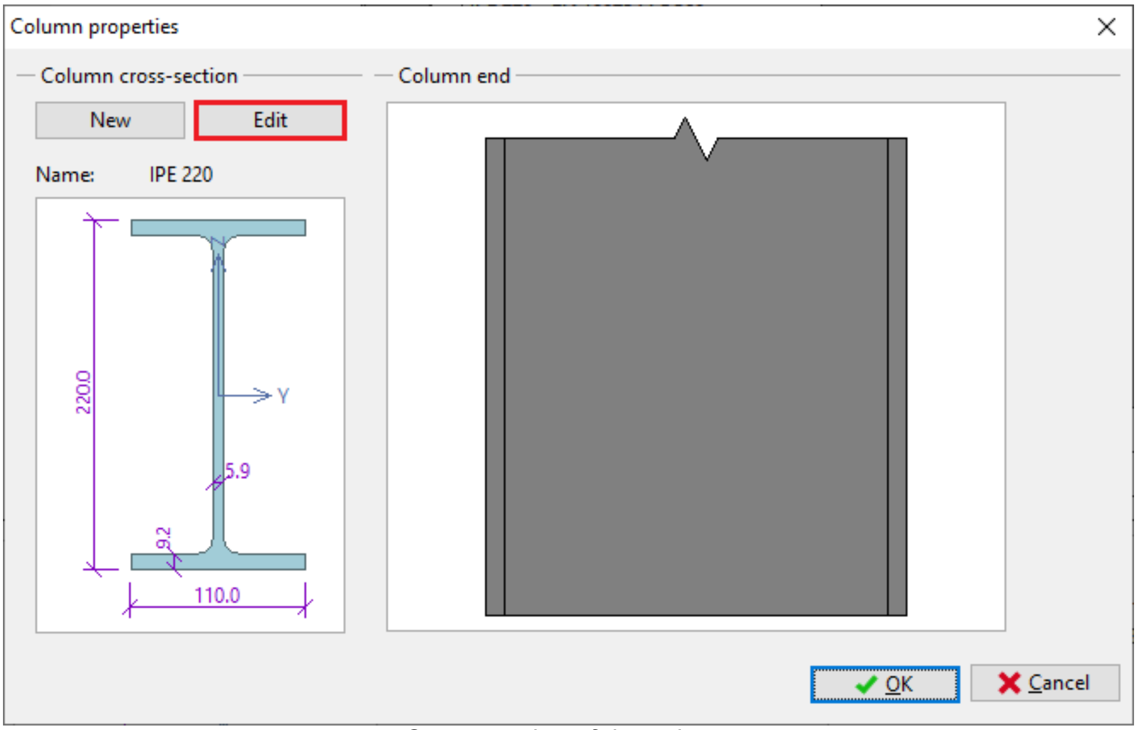
## Column

Define the cross-section of the column by clicking the button "**Edit section**" and the dialogue window "**Column properties**" will appear.




*The column and the end plate dimensions*

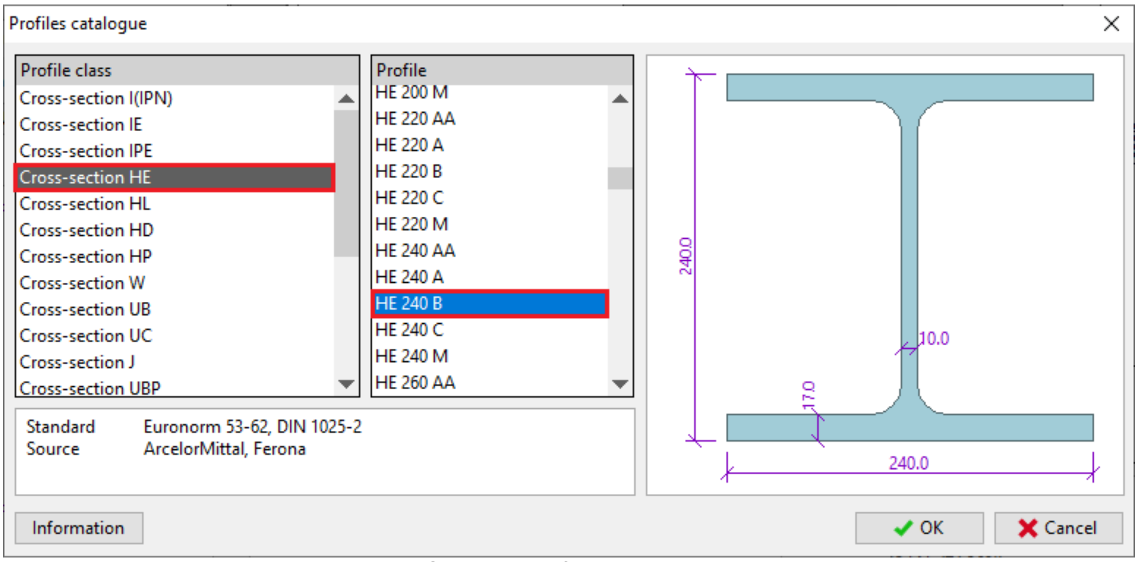
The dialogue window "**Column properties**" shows the preview of the cross-section. The supported types of steel columns are "**welded**" or "**hot-rolled**". The predefined type is "**hot-rolled**", therefore just click the "**Edit"** button and the catalogue with the cross-sections will appear.




*Cross-section of the column*

In the dialogue window "**Profile catalogue**" pick the profile class "**Cross-section HE**" and choose the profile "**HE 240 B**". Continue by clicking the "**OK**" button.

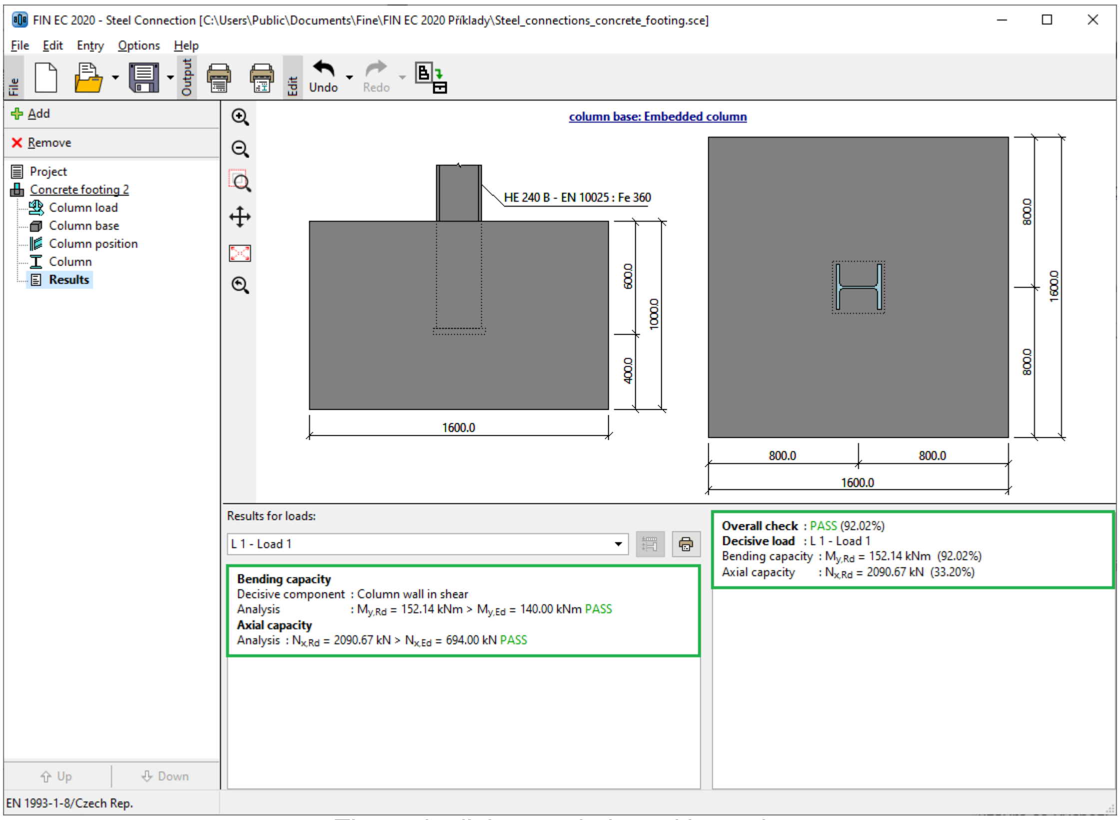
 



Column profile catalogue

## Results

The overall results can be seen at the right-bottom section of the main dialogue window. It shows the capacity of the connection for the least favourable load case. The overall results show that the moment capacity of the connection is *$M_{y,Rd}$= 152,14kNm (utilization 92.02%) and normal force capacity $N_{x,Rd}$= 2090,67kN (utilization 33.2%)*. The detailed results for every load case can be shown by selecting the section "**Results**" in the control tree.



The main dialogue window with results