USER MANUAL

FEM-Design

StruSoft ***Structural Software Design***

Visit the StruSoft website for company and FEM-Design information at http://www.strusoft.com .

User Manual

Trademarks

ABOUT FEM-DESIGN

FEM-Design offers a powerful, yet easy to use package that can be tailored to suite the exact analysis and design needs of the structural engineers using finite element method.

As FEM-Design is a full 64-bit software, all memory of the computer can be utilized. In previous 32-bit versions the memory usage was limited to 2 Gigabytes, which could cause out of memory error message when displaying results at large models. The analysis core was already 64-bit in earlier versions.

32-bit operating systems are not supported anymore!

FEM-Design runs on Microsoft Windows ® 8, 7, and Vista operation systems.

Single elements or a complete building, made from any number of materials and structural elements can be analyzed with ease related to Eurocode 2, Eurocode 3, Eurocode 5 and Eurocode 7.

FEM-Design Modules

The FEM-Design software system is a group of modules developed for different 2 and 3 dimensional structural problems, but it provides additional special features such as 2D drawing services and 3D solid modeling too.
The following tables summarize the design and auxiliary modules with their features and functionality.

The list of available modules depends on what license you have.

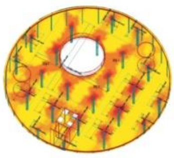
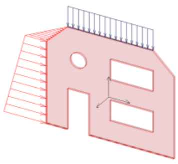
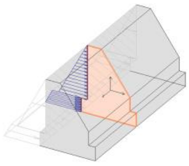
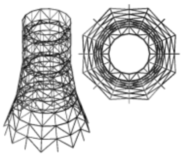
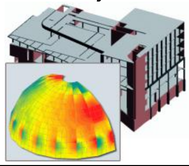
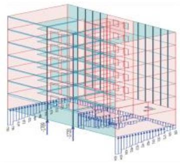
Plate	Wall	Plane Strain	3D Frame	3D Structure	PreDesign
Problem description					
Slabs, slab-systems or other 2D elements loaded perpendicularly to their planes 	Plane stress problems, shear and load bearing walls, elements loaded in their planes 	Structure with long extension and uniform section such as retaining walls, pipes etc. 	Two and three-dimensional frames and trusses with arbitrary loads 	3D structures include combinations of shell elements, slabs, walls, bars, beams, columns and foundation elements with arbitrary loads 	Pre-calculation for reactions and internal forces of load bearing walls and columns of 3D structures. 
Analysis					
Linear Non-linear Dynamic Cracked-section	Linear Non-linear Dynamic Cracked-section	Linear Non-linear Dynamic	Linear Non-linear Dynamic Imperfections Second order Stability Seismic Cracked-section	Linear Non-linear Dynamic Imperfections Second order Stability Seismic Cracked-section	Linear Non-linear Cracked-section
Foundation design					
				Isolated foundation Wall foundation Foundation slab	
RC Design					
RC slab RC beam Punching	RC wall		RC bar	RC shell RC bar Punching	
Steel Design					
Steel beam			Steel bar	Steel bar Steel bar-shell	
Timber Design					
Timber slab Timber beam	Timber panel		Timber bar	Timber panel Timber bar	
File format					
.pla	.wal	.pls	.frm	.str	.prd

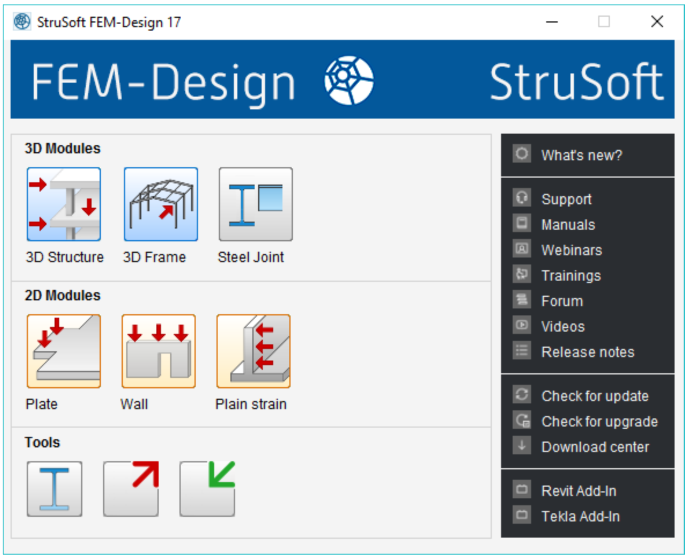
Table: The Design Modules

Starting Program

FEM-Design modules can be started from dialogs appear after choosing one of the following ways:

- Click *Start* button of Windows status bar and select *All Programs* > *FEM-Design…*
- Double click the short-cut, if it is placed on the Desktop.
- From FEM Design Center

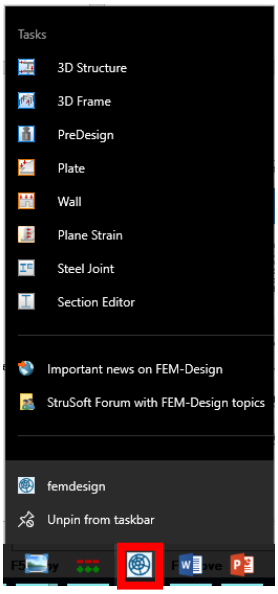
FEM-Design Center helps to reach all the modules, useful links, updates and links to the add-ins in one place and much faster.



CTRL + Left mouse button will open the last model

Shift + Left mouse button will open history in a jumplist

We suggest the User to pin the command center so right clicking the icon on the tray will open a jumplist where all the modules available.



Starting a design module (or a new project) offers codes for structure design. Code selection influences:

- the range of available modeling materials,
- automatic combination of load groups, and
- the method and result of foundation, reinforcement, steel and timber design.

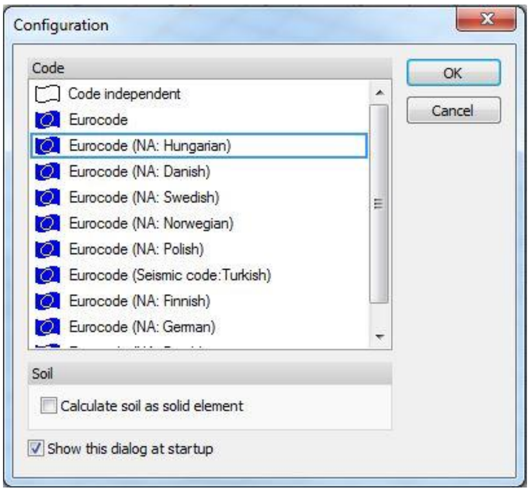


Figure: Design codes

⚠ The current code of a project can be modified any time during input process (structure and load definition) at *Settings* > *All...* > *Calculation* > *Code, configuration*. Changing a code will modify all materials and design parameters according to the selected new code by using an editable conversion table.

Basic Concepts

User Interface

Each module has a similar user interface and contains the following parts:

- Menu bar
- Toolbars
- Tabmenus
- Status bar
- Application window
- Tool palette
- Dialog

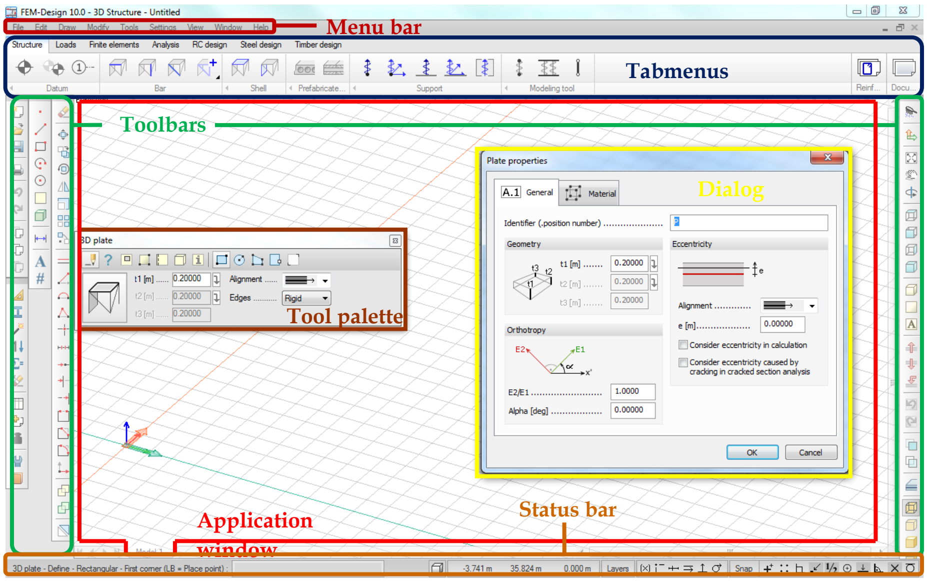


Figure: FEM-Design user interface

Menu Bar

The *Menu bar* contains file operations (*File*), drawing (*Draw*) and editing commands (*Edit*), assistant tools (*Tools*), settings (*Settings*), views (*View*), window-system (*Window*) and user-guides (*Help*).

Menu commands having icons next to their names can be organized in *Toolbars* around the *Application window*. Some commands can be executed by hotkey displayed behind the command name.

Toolbars

The *Menu bar* commands can be grouped in toolbars with their icons and placed next to *Application window*.

By default, the *Standard* and the *View* toolbars are displayed and the other toolbars are hidden. If you right-click anywhere on the menu or on one of the displayed toolbars, a list appears, where toolbars can be set visible or hidden.

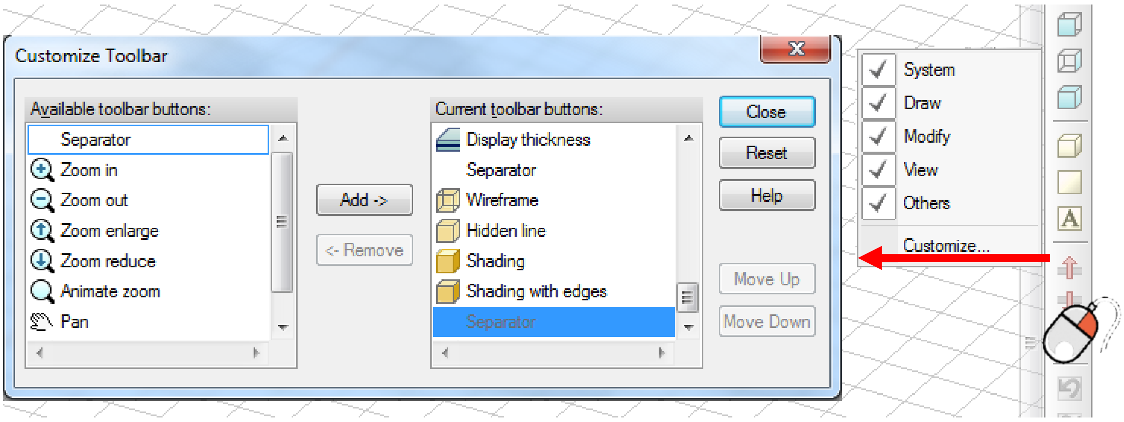


Figure: Available Toolbars and the Customize tool

Customize… allows you to edit the command content of the toolbar, on which the right-click is used.

A toolbar can be moved into new position by dragging it with its "thick vertical line" start.

Tabmenus

Tabmenus contain the main functionalities and their order displays the recommended structural design workflow.

Tabmenu	Function
Structure	Tools to define axis and storey system, structural elements, supports and connections
Loads	Tools to define load cases, loads, load combinations and load groups
Finite elements	Tools to define finite elements and finite element mesh
Analysis	Tools to run analysis for the current project and to display results
RC design	Tools to run automatic and manual design for concrete elements and to display results
Steel design	Tools to run automatic and manual design for steel bars and to display results
Timber design	Tools to run automatic and manual design for timber elements and to display results

Table: Tabmenu types

By default, *Tabmenus* have different **Object layer** settings to protect their elements from the others. That means, for example, structural elements (defined at *Structure* tab) are protected against load editing (*Load* tab) although they can selected for load definition. Of course, the available layers of tabs can be customized (Status bar > **Current layer**).

The optional module **Documentation** can be launched from the *Tabmenus* bar.

Status Bar

The *Status bar* is situated under the **Application window** and separated into the following parts.

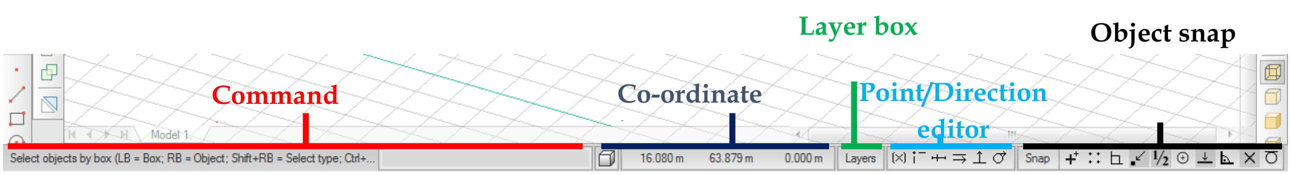


Figure: The parts of the Status bar

- **Co-ordinate box**
  It shows the exact coordinates of the crosshair cursor in the current **co-ordinate system**. Coordinates can be also given for point/direction definition in a dialog appears after clicking on the *Co-ordinate* box (see **Entering Co-ordinates**).

- **"Current layer" button**
  Clicking the button opens a dialog contains the **layer-system** of the current project. All layer operations like defining new, renaming, color-assignment, showing/hiding and deleting are available in the layer-system dialog. The "*Current layer*" button displays the name of the current **drawing layer**.

- **"Current style" button**

This button shows the line type currently available for drawing lines, arc, circles, edges etc. (*Draw* menu). Clicking the button opens a dialog, where new line type can be defined or line types can be edited.

- **Point/Direction editor**
  Editor tools provide reference points and lines by using existing lines and points during element definition, drawing and editing.

- **Object snap tools**
  Here you can set the snapping distance and turn object snaps on and off.

- **"Current color" button**
  Here you can set the drawing color independently from the color of the current drawing layer. This color is used for numeric values on result figures too.

- **"OK" and "Cancel" buttons**
  These tools approve or reject the current operation.

- **Command line**
  Here you can directly communicate with the program for example by typing coordinate values in it. In the command line coordinates can be defined in several ways (see **Entering Co-ordinates**). *Command line* also displays additional messages to show the steps of the current command.

Application Window

Two main windows types are available in FEM-Design:

- **Graphical window**
  It works as a drawing board and displays the model defined in the current project.

- **Detailed result window**
  Analysis and design results with detailed background calculation formulas (code references), figures and tables can be displayed by single elements or by design groups in separate windows. Quick navigation is powered with zooming details.

An arbitrary number of windows can be opened (*Window* menu) to show for example a model in different views at the same time. The list of the currently available windows is shown at the bottom of the windows (tabs) and in the *Window* menu. You can swap between the windows by clicking on their name tabs or by choosing the requested one in the *Window* menu. Applied windows can be arranged by *Window* > *Arrange*. Right-clicking on a window tab, the window's name can be edited.

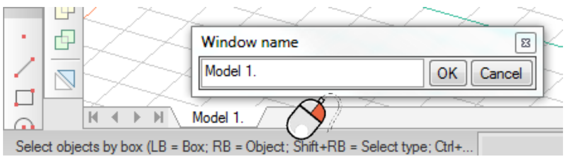


Figure: Application windows

Tool Palette and Dialog

In most cases, a command/tool has own palette (*Tool palette*) that contains its definition and setting tools.

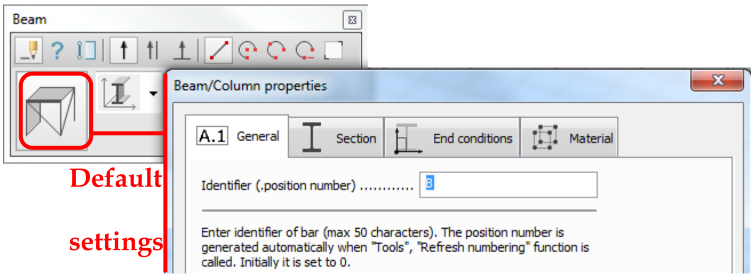


Figure: An example for tool palette (Beam command in 3D Structure module)

The parts of a tool palette are:

- **Toolbar**
  It gives the editing modes and their additional tools. The main editing modes are:

   *Define*: creates new objects according to its current settings ("*Default settings*").

  See more **Element Definition**.

   *Properties*: asks and/or changes the properties of selected object(s).

  See more **Object Properties**.

- **"Default settings"**
  If this button has 3D shape, then you can set all default parameters for the new object. Otherwise, it only shows the symbol of the function.

- **Main settings**
  The frequently changed parameters of the new object (*Define*) can be set directly in tool palette.

You can usually use a dialog to modify the settings, the properties of a command; it contains text and number fields where you can set parameter values.

The main difference between dialogs and *Tool palettes* is that you can work beside an opened tool palette, but cannot at a dialog.

There are three main switch types at a dialog or a tool palette: check boxes, radio and chain buttons.

In case chain button exists and it is "Active" (pushed in), any changes made to the related edit box will be transferred to the next edit box automatically. It gives a quick definition of defining surface elements with constant thickness, loads with constant intensities etc.

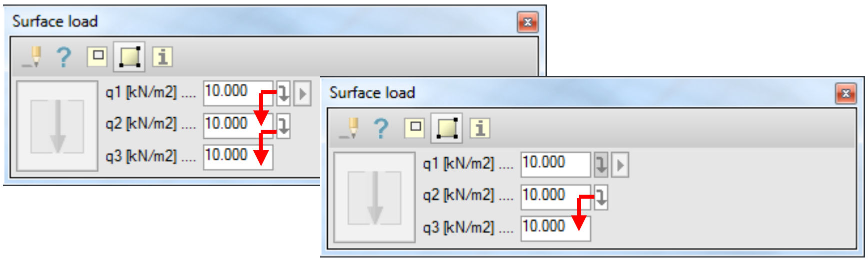


Figure: Function of chain buttons

Program Settings

Clicking *Settings* > *All* opens a dialog with all available settings of the program and the current project.

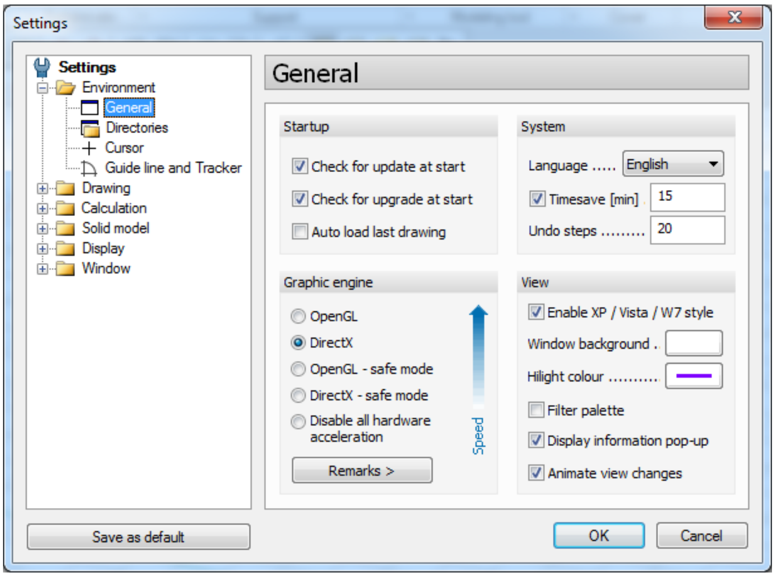


Figure: Project and program settings

The settings are valid for the current project, but they can be extended for later projects by saving them as default settings. *OK* closes the dialog and validates the settings for the current project. *Save as default* set the "selected" project settings available for next projects and new files.

⚠ "Selected" means that *Save as default* works only for the branch belongs to the list/folder title selected by the cursor. For example, apply *Save as default* for "*Environment*" to save the *General*, the *Directories* and the *Cursor* settings as default, or select "*Settings*" and click *Save as default* to store all current settings as default values.

The range of available settings depends on the currently opened **Tabmenu**. For example, all setting are available at launched "*Structure*" tabmenu, but the settings are reduced with e.g. finite element (*Mesh*) settings in case of activated "*Analysis*" tabmenu.

The program stores project and default settings in the *fem.ini* file can be found in the "*FEMData*" folder of the installed program.

New program release can use the default settings of the previous release, if you keep them after the first running of the new release in a warning message dialog.

Environment

General settings enable the user to adjust the main program settings affect on startup, system, rendering and display style.

- **Check for update at start**
  By default, the program informs the user if a new official release of the current FEM-Design version is available.

- **Check for upgrade at start**
  It sends a message if a newer commercial FEM-Design version is released. In this case, the newer version can be downloaded, but it can be run only in demo mode till renewing the current license agreement (please contact your local distributor).

- **Auto load last drawing**
  Checking this box opens the last project at program start-up.

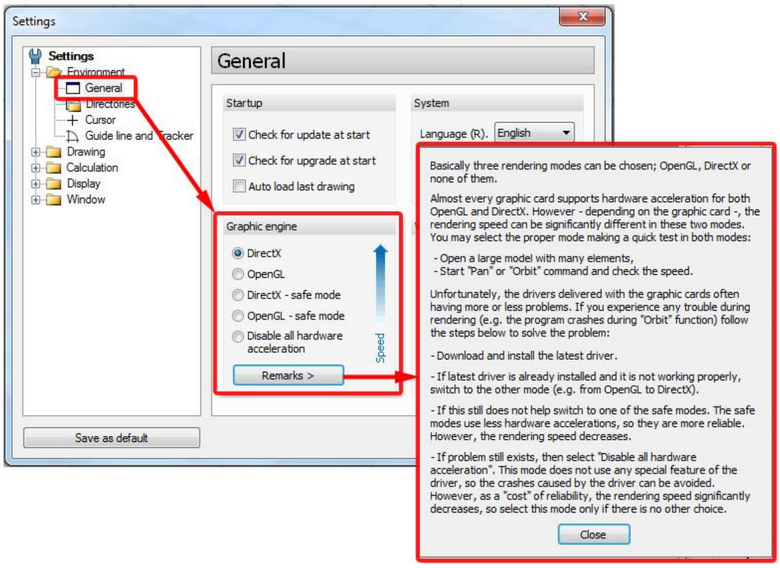
- **Language**
  By default, the user interface is displayed in the installation language. Ask your local distributor about the available languages in your country.

- **Timesave [min]**
  It defines the time period in minute for automatic project saving. Read more at **Data safety**.

- **Undo steps**
  The number of executable undo steps can be set in the field. But do not forget, that there are some complex calculation processes (analysis, mesh preparing etc.) *Undo* does not work on.

- **Graphic engine**
  The suitable graphic engine can be set here by choosing "DirectX" or "OpenGL" according to the video card.

  The DirectX graphic engine is faster than OpenGL on most of the computers. We recommend the User to check which graphic engine performs better on his computer.



For more information about the graphic engine read the Remarks.

If you have any rendering problem with the default graphic engine, apply the other one. In case of further troubles, download the latest driver of your graphic card.

"*Safe mode*" is developed to protect the program from possible video drive crashes. If no graphic problem is detected with the current driver, uncheck this box to take the full advantage of the new powerful graphic engine and to reach the maximum rendering speed.

- **Enable Win8/Win7/Vista style**
  By default, the user interface is developed in the height of Microsoft Windows 8, 7 and Vista fashion. Unchecking the box a simpler interface will be available after restarting the program.

- **Window background**
  The color of the drawing background (graphical windows) can be set here. The default and suggested color is white. Objects having the same color with the background are always displayed in inverse color.

- **Display information pop-up**
  If there is no running command, moving the cursor over an element displays an *Information pop-up* with the element's main properties. Depending on the currently used working mode (*Structure*, *Loads* etc.), pop-up is available for different elements (structural elements, loads etc.) only. For example, the pop-up displays the ID, the material, the thickness, the alignment and orthotropic features for *Plates* in *Structure* mode.

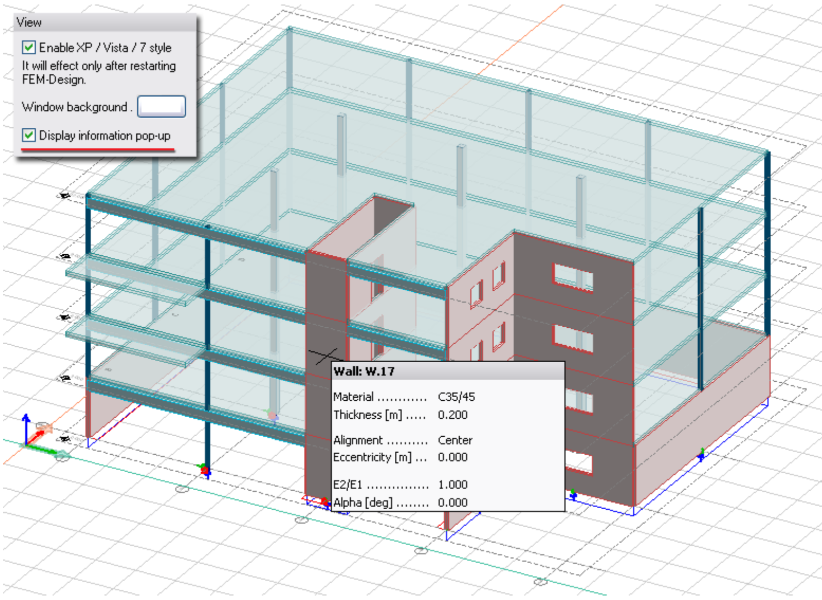


Figure: Information pop-up examples

Directories settings allow the user to specify the location of FEM-Design files.

- **Temporary**
  The program stores temporary files during calculations and mesh generations in the given folder, which will be automatically removed, if the operations are done successfully. Remained temporary files can be erased from the *Temporary* folder after closing the currently opened project. The changes made in *Temporary* directory will affect after the program is restarted.

- **Working**
  The default favorite folder can be set here for the open and save commands.

The style of cross-hair cursor can be set in *Cursor* dialog. It can be full-sized in graphical windows or custom-sized (*Limited*) by the scroll tool.

Drawing

Title dialog allows the user to specify the content of drawing title block (*Draw* > **Title information table**) and the header title of printing (*File* > *Print*). *Project*, *Description*, *Designer*, *Signature* and *Comments* attributes are autotexts. That means, modifying their content updates all titles used in the current project. These five title attributes can be used in documentation and reinforcement list templates with the help of **Field** tool (*Draw* menu).

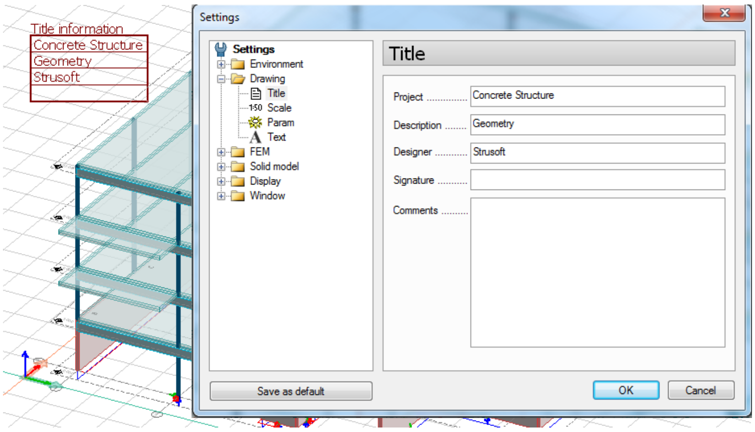


Figure: Title information table

Scale has effect in displaying texts, line types and wall hatches in graphic windows. Of course, **printing scale** can be set independently from drawing scale.

Param settings allow the user to specify the default initial settings for drawing (*Draw* menu commands) in the current or later projects (*Save as default*). Line type, pen width and drawing color can be edited independently from the default values with the **Status bar** tools "*Current style*", "*Current layer*" and "*Current color*".

Text settings enable the user to set the default font style and sizes for general text items.

	Text setting effect
Drawing Title block	yes
Structural element ID and label	yes
Design data label	yes
Load label	yes
Mass label	yes
Support ID and label	yes
Default settings of *Draw* > *Text*	yes
Default settings of numeric result values	yes
Draw > *Dimension*	no
Grid axis label (*Structure* > *Axis*)	no
Storey label (*Structure* > *Storey*)	no
Finite element and node ID (*Settings* > *Display* > *Mesh*)	no

Table: List of elements on which Text setting has effect or not

Dimension, *Axis* and *Storey* tools and the finite element display have individual text settings from the general *Text* settings.

Script solves the character problems of different languages. For example character "**ő**" appears as "**õ**" in "*Western*", but as correct "**ő**" in "*Central European*" script.

Size is defined in real values (mm), so text size varies on drawings by modifying its *Scale*.

The *Width* factor 1.0 refers to the normal character width, so factor smaller/greater than 1.0 results a condensed/extended text.

For "italic" style use *Slant* between 10 or 15 degrees.

Calculation

Calculation includes settings influence structural behavior, finite element mesh generation, analysis and result.

Code displays the current and available design code for the current project. *Code* influences:

- the range of available modeling materials,
- automatic combination of load groups, and
- the method and result of reinforcement, steel and timber design.

Changing the current code erases the materials, the design parameters and the results in the project, so these properties have to be redefined to run valid calculations later.

Read more details from other *Calculation* settings in the connected topics:

- **"Rigid" values**
  The default value of "infinite rigidity" can be set for **supports**, **connections** and **fictitious bars** by types.

- **Mesh**
  Mesh and peak smoothing settings of **automatic finite element mesh generation**.

  Perform gives additional features to **data safety**.

- **Analysis**

  In *Analysis* you can decide the way of ***Automatic save***, in *Find identical copies before calculation for* you can set if the program should check the identical copies of structural elements and/or loads and in *Warnings* it can be checked if the program should warn about using non-linear elements in a linear calculation.

- **Result**
  Options define the calculation sections of bars and affect on **detailed results** of bars.

Display

The current and default display settings of the drawing elements, numbers, **structural objects**, **loads** and **finite elements/mesh** can be set here.

Displaying structural and analytical element ID's is separated in order to avoid duplicate labels on the screen. The required ID can be set by Bar and Shell elements in Display settings dialog.

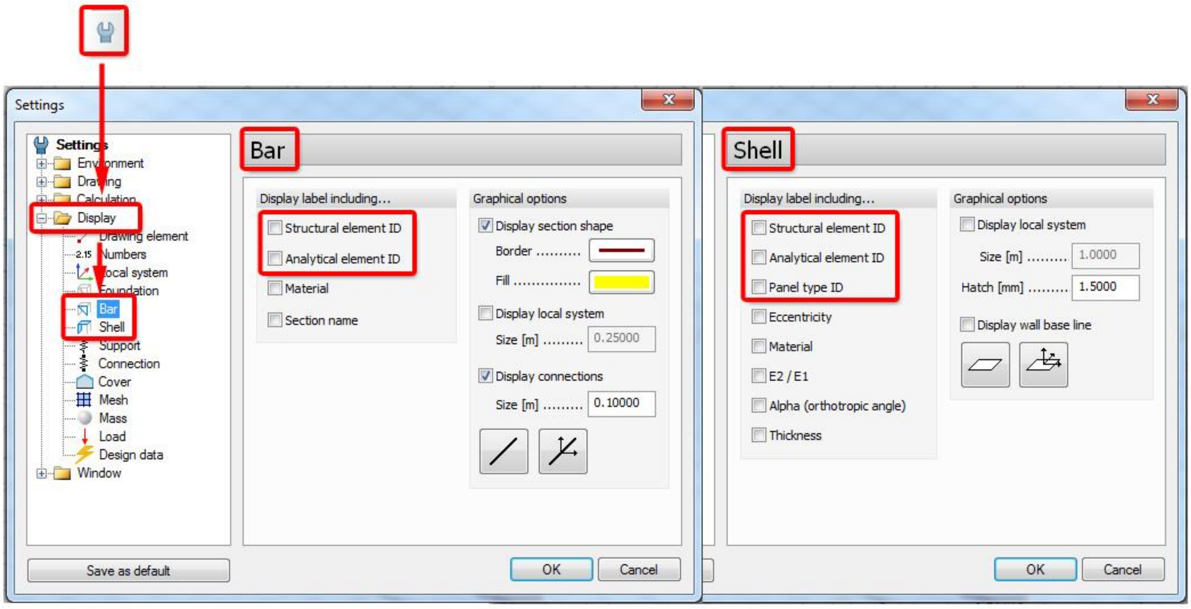


Figure: Display settings (Bar, Shell)

Window

Co-ordinate systems include the display settings of the *Global* and the *User-defined* (*UCS*) **co-ordinate system** symbols. The symbols are shown in the defined size in working windows, but they can be hidden by unchecking the *Display co-ordinate systems* box.

Grid defines the distribution settings of the visible **Construction grid**. The grid can be hidden by unchecking the *Display grid* box. The grid lines can be set to be parallel with the axis directions of the *Global* (*Ground*) or *UCS* system.

All default settings of **Object Snap tools** are grouped in *Snap* dialog.

In different windows different *scales* can be defined, which is especially useful in documentation module. By choosing an appropriate scale factor the displayed labels are easy to read. In another window the parameters can be decreased to highlight another result.

Units setting offers various unit types for modeling (length, angle, force, mass and cross-sectional data) and for results. Stress and displacement units can be set independently of the length and force units.

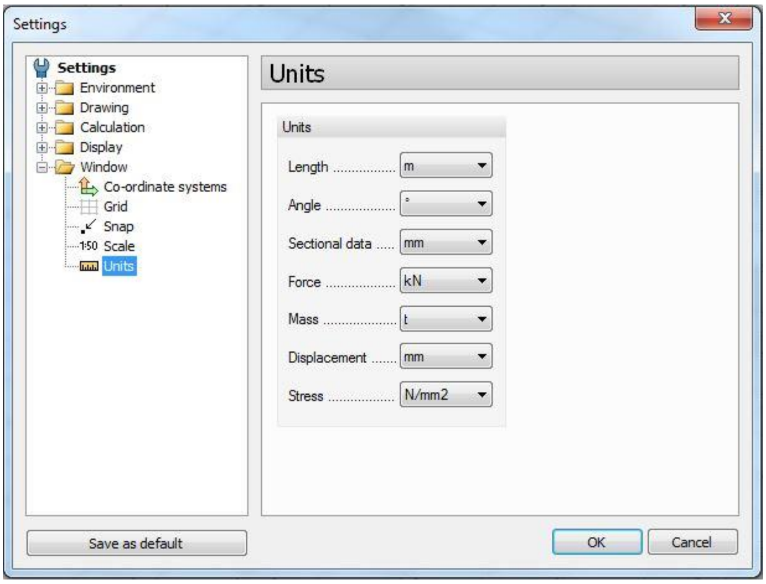


Figure: Available units

Data Safety

Time saving and protection system ensures more the data safety of working files.

Time Save

FEM-Design automatically saves copy of the working file by given time period.

The saving time period can be set in minutes at *Settings* > *All* > *Environment* > *General* > *System* > *Timesave*.

Working on a project file called e.g. "steel construction.str" in the *3D Structure* module with a 15-minute time save period, an extra file will be generated and updated by the time period (0, 15, 30min …) with special extension and in a hidden folder:

Rules of time save:

- In case of a program crash, the last saved copy opens automatically in the right module.
- If you successfully exit from FEM-Design after saving the current project, its time save copy will be erased from the recovery folder.
- The location of time save files cannot be modified for data safety.

Additional automatic savings can be set before and after calculations at *Settings* > *Calculation* > *Analysis* > *Automatic save…*, but automatic savings overwrite the original project file.

Recovery files

Recovery file command in File menu is for managing time-save files which are generated after some unexpected events. Its tool-window:

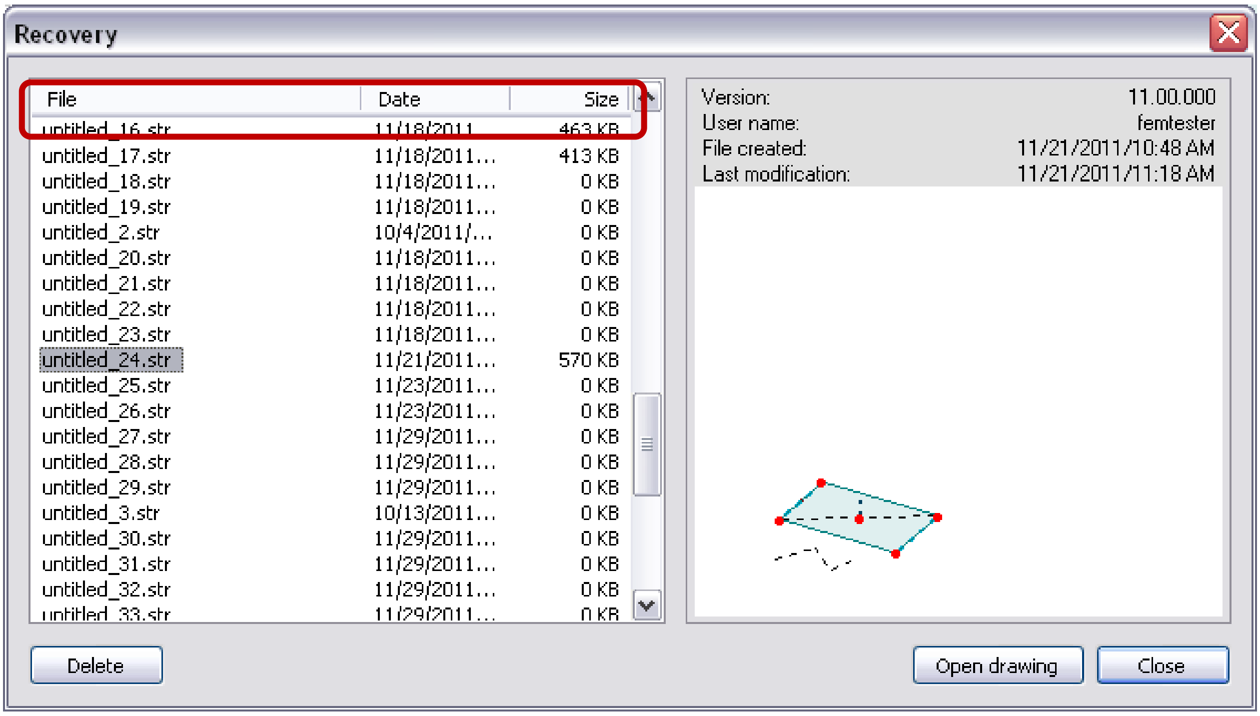


Figure: Recovery file dialog

You can open the desired version and delete the useless ones, and you can also set the sorting criterion, by clicking on File / Date / Size.

Protected Work File

All working files are locked not to be modified, overwritten by another user or process. Starting a project, a file having the project name with ***.lck*** extension is generated automatically, which guarantees the protection till closing the working file.

Opening a protected file sends the warning message: "*The file is locked by another user or process.*" You are allowed to open and modify the content of a protected file, but changes can be saved only under a new name (*File* > *Save as*).

Element Types

Two main types of elements are available in FEM-Design: *Drawing elements* and (structural) *Objects*. These element types are stored in different ***Layer-systems*** and have different kind of property and display settings.

Drawing Elements

Type	Command	Property and Display Settings
Point	Draw > **Point**	*Settings > Drawing > Param, "Current layer"*
Line	Draw > **Line**	*Settings > Display > Drawing elements,* *"Current style", "Current layer"*
Rectangle	Draw > **Rectangle**	*Settings > Display > Drawing elements,* *"Current style", "Current layer"*
Arc	Draw > **Arc**	*Settings > Display > Drawing elements,* *"Current style", "Current layer"*
Circle	Draw > **Circle**	*"Current style", "Current layer"*
Region	Draw > **Region**	*Settings > Display > Drawing elements,* *"Current style", "Current layer"*
Solid	Draw > **Solid**	*Settings > Display > Drawing elements,* *"Current style", "Current layer"*
Text	Draw > **Text**	*Text* tool > *Default settings,*
Field	Draw > **Field**	*Field* tool > *Default settings*
Dimension	Draw > **Dimension**	*Dimension* tool > *Default settings, "Current style"*
Title information table	Draw > **Title information table**	*Text* tool > *Default settings, "Current style",* *Settings > Title, "Current layer"* (! Object layer)

Table: Drawing elements

Objects

Type	Command	Property and Display Settings
Axis	Structure, **Axis**	*Axis* tool > *Default settings*, *"Current layer"*
Storey	Structure, **Storey**	*Storey* tool > *Default settings*
Soil	Structure, **Soil**	*Settings* > *Display* >*Soil and foundation*, *Soil* tool > *Default* *settings*
Borehole	Structure, **Borehole**	*Settings* > *Display* >*Soil and foundation*, *Borehole* tool > *Default* *settings*
Isolated foundation	Structure, **Isolated foundation**	*Settings* > *Display* >*Soil and foundation*, *Isolated foundation* tool > *Default settings*
Wall foundation	Structure, **Wall foundation**	*Settings* > *Display* >*Soil and foundation*, *Wall foundation* tool > *Default settings*
Foundation slab	Structure, **Foundation slab**	*Settings* > *Display* >*Soil and foundation*, *Foundation slab* tool > *Default settings*
Beam	Structure, **Beam**	*Settings* > *Display* > *Drawing elements* and *Bar* *"Current style"*, *"Current layer"*, *Beam* tool > *Default settings*
Column	Structure, **Column**	*Settings* > *Display* > *Drawing elements* and *Bar*, *"Current layer"*, *Column* tool > *Default settings*
Intermediate section	Structure, **Intermediate section**	*Intermediate section* tool > *Default settings Drawing elements* and *Bar*
Apex	Structure, **Apex**	*Apex* tool > *Default settings*

Corbel	Structure, **Corbel**	*Settings* > *Display* > *Drawing elements* and *Bar* *Corbel tool* > *Default settings*
Truss member	Structure, **Truss member**	*Settings* > *Display* > *Drawing elements* and *Bar*, *"Current style"*, *"Current layer"*, *Truss member* tool > *Default settings*
Plate	Structure, **Plate**	*Settings* > *Display* > *Drawing elements* and *Shell*, *"Current layer"*, *Plate* tool > *Default settings*
Wall	Structure, **Wall**	*Settings* > *Display* > *Drawing elements* and *Shell*, *"Current layer"*, *Wall* tool > *Default settings*
Profiled panel	Structure, **Profiled panel**	*Settings* > *Display* > *Drawing elements* and *Shell*, *"Current layer"*, *Profiled panel* tool > *Default settings*
Timber panel	Structure, **Timber panel**	*Settings* > *Display* > *Drawing elements* and *Shell*, *Timber panel* tool > *Default settings*
Support	Structure, , **Supports**	*Settings* > *Display* > *Drawing elements* and *Support*, *Settings* > *FEM* > *"Rigid" values*, Support tools > *Default settings*, *"Current layer"*
Connection	Structure, , **Connections**	*Settings* > *Display* > *Drawing elements* and *Connections*, *Settings* > *FEM* > *"Rigid" values*, Connection tools > *Default settings*, *"Current layer"*
Fictitious bar	Structure, , **Fictitious bar**	*Settings* > *Display* > *Drawing elements*, *"Current layer"* *Settings* > *FEM* > *"Rigid" values*, *Fictitious bar* tool > *Default settings*
Fictitious shell	Structure, , **Fictitious shell**	
Load	Loads, , , , , **Loads**	*Settings* > *Display* > *Load*, *"Current layer"*, Load tools > *Default settings*,
Finite element	Finite elements, **Finite element tools**	*Settings* > *Display* > *Mesh*, *"Current layer"*, *Settings* > *FEM* > *Mesh* and *Calculation*

RC bar	RC design, **Auto design** and **Manual design**	*Settings* > *Display* > *Design*, *"Current layer"*, *Auto design* and *Manual design* tools > *Default settings*
Steel bar stiffener	Steel design, , **Steel bar stiffener**	*"Current layer"*

Table: Objects

Layers

The FEM-Design layer-system helps you to work in a well-organized way in drawings and in models while constructing and documenting. The layers can be considered as transparent papers put on top of each other will seem to be one drawing. All layers can be reached in a dialog appears by pressing the **Current layer** button.

Layer Types

Regarding the **element types** there are two main layer types: "*Drawing*" and "*Objects*".

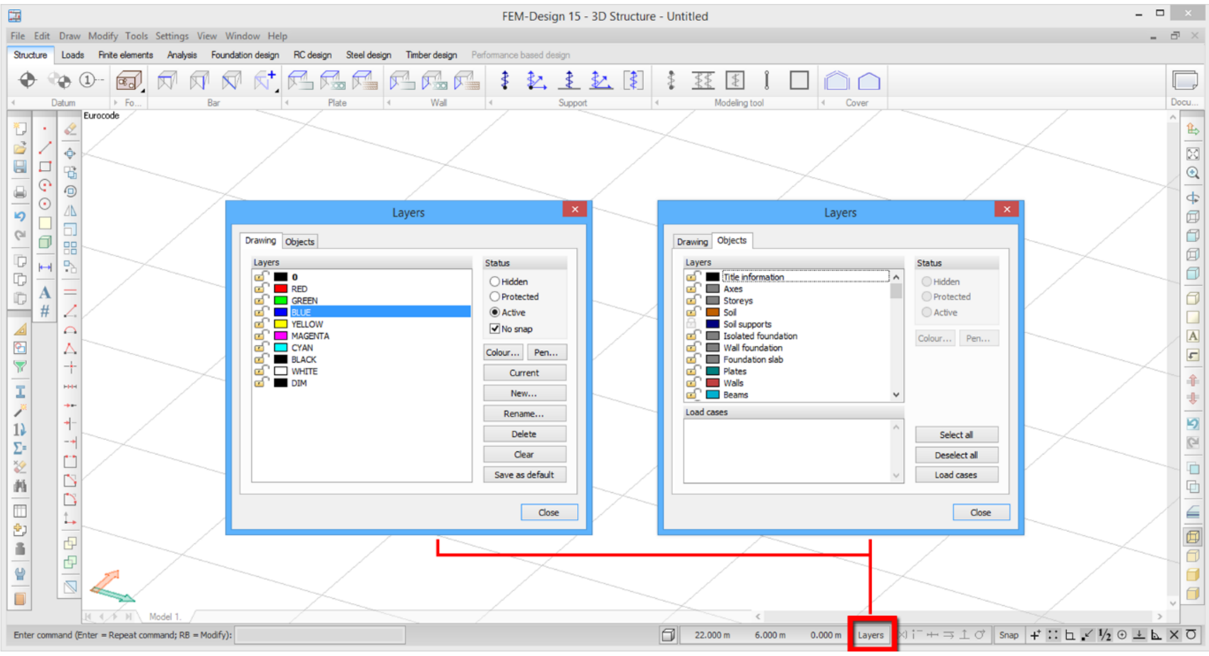


Figure: "Drawing" and "Objects" layers

"***Drawing***" layer contains all drawing elements defined by the *Draw* menu commands or generated from **DWG and DXF imports**. *Drawing* layers can be renamed and erased, and their colors and states can be edited. Only the default "*0*" layer cannot be deleted and edited. At the same time only one drawing layer called "Current" can be used for drawing. The name of the "Current" layer is appears on the *Current layer* button. The "*DIM*" layer is a special drawing layer for the dimensions. It is automatically created by using the **Dimension** command (*Draw* menu). "*Drawing*" *Layers* can be excluded from the **Object Snap** by checking the „*No snap*" option

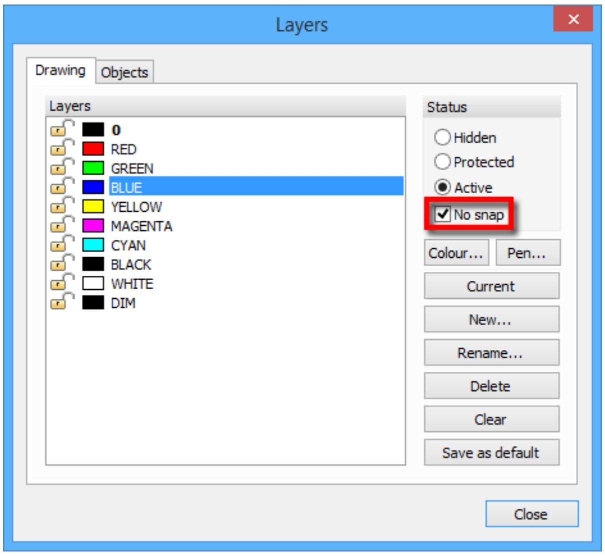


Figure: „No snap" option

"***Objects***" layers are built-in layers for objects like plate, wall, column, supports, loads etc. Each object type has an own layer (e.g. walls are displayed on the "*Wall*" object layer). *Objects* layers are not allowed to be deleted and renamed, but their states and the display colors and pen width of objects can be edited. "*Objects*" *Layers* can also be excluded from the **Object Snap** by checking the „*No snap*" option.

Layer Status

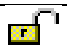
Layer Status	Symbol	Function
Hidden		The layer content is hidden in graphic windows
Protected		The layer content is visible, but protected from editing
Active		The layer content is visible and editable in graphic windows

Table: Layer status

In Layers dialog changing the layer status is real-time, which means that the selected active (hidden) drawing or object layers become hidden (active) without closing the dialog.

Editing Layers

Tool	Function	Available for layer type
Make current	Sets selected layer to be the current one	Drawing

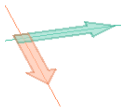
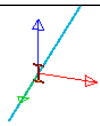
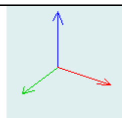
New…	Creates new layer in the available layer list	Drawing
Rename…	Modifies the name of the selected layer	Drawing
Delete	Deletes the selected layer from the layer list ⚠ Delete function cannot be undone All elements will be erased of a deleted layer	Drawing
Clear	Removes all elements of the selected layer ⚠ Clear function cannot be undone	Drawing
Color…	Modifies the current color of the selected layer Defines the current drawing color Defines the display color of Objects Color can be modified by elements with **Edit > Change appearance**	Drawing, Objects Drawing Objects
Pen…	Sets the pen width assigned to the selected layer Defines the current drawing pen width Defines the display pen width of Objects Pen width can be modified by elements with **Edit > Change appearance**	Drawing, Objects Drawing Objects
Select all	Quick selection of all layers	Objects
Deselect all	Deselecting currently selected layers	Objects
Objects	Quick selection structural object layers	Objects
Statical system	Quick selection of Supports, Modeling tools and finite element/mesh layers	Objects
Load cases	Quick selection of load (cases) layers	Objects

Table: Editing layers

The full content of an *Objects* layer can be erased with *Tools* > *Delete all*. The objects deleted with *Delete all* can be restored with *Edit* > *Undo*. *Delete all* erases the objects of protected layers too!

Co-ordinate Systems

Various co-ordinate systems are available in FEM-Design with different function, properties and display settings.

	Global co-ordinate system	User-defined co-ordinate System (UCS)	Local co-ordinate System
Function	Definition of co-ordinates Definition of directions Interpretation of results -	Definition of co-ordinates Definition of directions - -	- Definition of directions Interpretation of results Definition of profiles
Type	Right-handed Cartesian	Right-handed Cartesian	Each structural object has got Its own right-handed Cartesian co-ordinate system
Axes	X, Y, Z	X, Y, (Z)	x′, y′, z′
Symbol			 
Symbol color	Fixed 3 colors, X = green Y = red Z = blue	Fixed 2 colors, X = cyan Y = orange	User-defined 3 colors, *Settings* > *All* > *Display* > *Local systems*
Position	Fixed	User-defined: - *View* > *UCS* -  moves the origin to the position of the cross-hair cursor -  +  moves origin back to the origin of the Global system	Fixed for an object
Direction	Fixed	User-defined: - *View* > *UCS*	Changeable: *Edit* > *Change direction*

		- Shift ⇧ + insert resets the directions to the original state	
Show/Hide	*Settings* > *All* > *Window* > *Co-ordinate systems*	*Settings* > *All* > *Window* > *Co-ordinate systems*	By element types: *Settings* > *All* > *Display*
Symbol size	*Settings* > *All* > *Window* > *Co-ordinate systems*	*Settings* > *All* > *Window* > *Co-ordinate systems*	By elements types: *Settings* > *All* > *Display*

Table: Co-ordinate systems

User-defined co-ordinate system (*UCS*) is developed in order to easily define coordinates and directions in a 2D user-defined **working plane**.

Point Definition with Co-ordinates

Points can be defined with their co-ordinates in **Descartes** or **Cylindrical co-ordinate systems**.

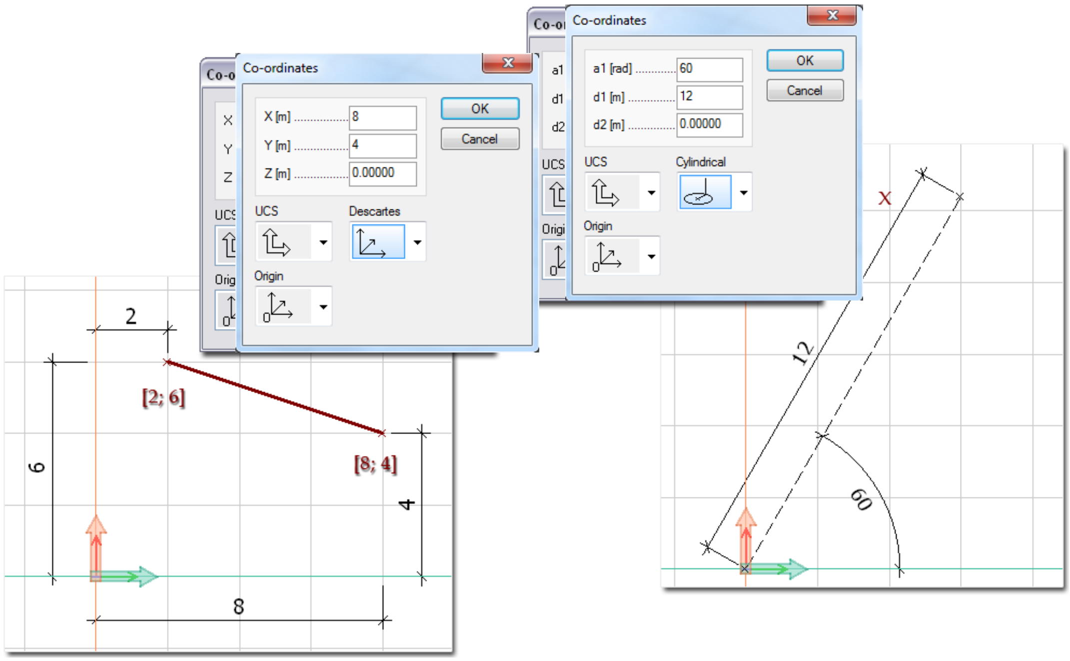


Figure: Co-ordinate system types for point definition

Point co-ordinates can be defined in various unit types available at **Settings > Units** (*Length* and *Angle*).

The crosshair cursor position can be displayed on the **Status bar** in the *Descartes* or in the *Cylindrical* co-ordinate system by clicking the  symbol of the *Co-ordinate box*.

Descartes Orthogonal Co-ordinate System
(Cartesian co-ordinate system)

The individual co-ordinates represent the distance of the point from the origin of the orthogonal co-ordinate system (***absolute***) or from a previously defined point (***relative***) measured along X, Y and Z axes. *Cartesian co-ordinate system* is available for co-ordinate definition both in the *Global co-ordinate* and the *UCS systems*.

Cylindrical Co-ordinate System
(Polar co-ordinate system)

Point is defined by three components: an angle (a1) and two distances (d1 and d2). The a1 and d1 co-ordinates define the point position in the XY plane of the *Global co-ordinate* and the *UCS systems*, d2 sets the distance from the XY plane. The angle and the distances can be given from the origin of the co-ordinate systems (***absolute***) or from a previously defined point (***relative***).

Co-ordinate Definition Modes

Point co-ordinates can be defined by typing them in the *Command line* or the *Co-ordinate box* of the **Status Bar**, or just by mouse-clicking combined with the **Object snap tools** in the drawing area or in special points.

Absolute co-ordinate definition
Co-ordinates of a point are defined as distances/positions from a co-ordinate system origin.

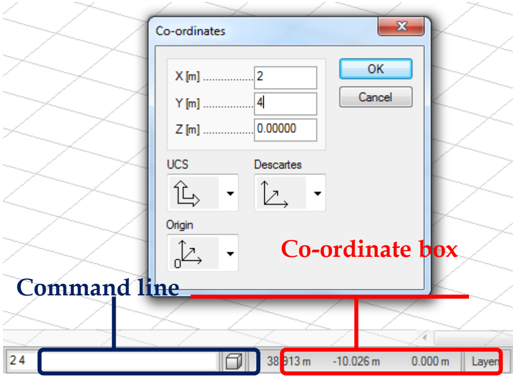
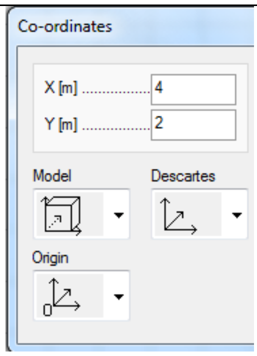
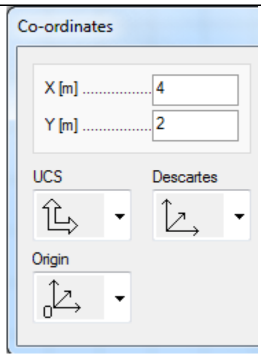
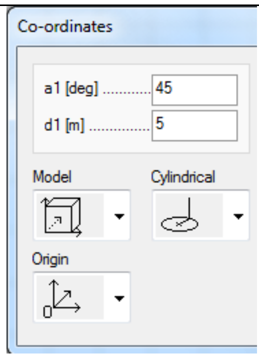
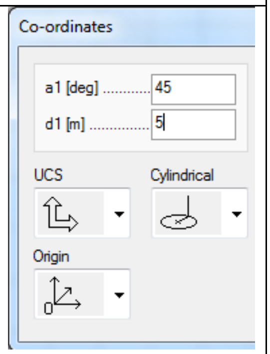


Figure: Absolute co-ordinates given in Command line and Co-ordinate box

<table>
<tr><th></th><th colspan="2">Descartes Orthogonal System</th><th colspan="2">Cylindrical co-ordinate system</th></tr>
<tr><th></th><th>Global co-ordinate system</th><th>UCS system</th><th>Global co-ordinate system</th><th>UCS system</th></tr>
<tr><td></td><td colspan="4">2D</td></tr>
<tr><td>Command line</td><td>-</td><td>X_Y or X,Y

Example:
Point (LB, RB = Place point) : 4 2
Point (LB, RB = Place point) : 4,2</td><td>-</td><td>P_a1_d1

Example:
Point (LB, RB = Place point) : P 45 5</td></tr>
<tr><td>Co-ordinate box</td><td>
</td><td>
</td><td>
</td><td>
</td></tr>
<tr><td></td><td colspan="4">3D</td></tr>
<tr><td>Command line</td><td>-</td><td>X_Y_Z or X,Y,Z

Example:
Point (LB, RB = Place point) : 4 2 5
Point (LB, RB = Place point) : 4,2,5</td><td>-</td><td>P_a1_d1_d2

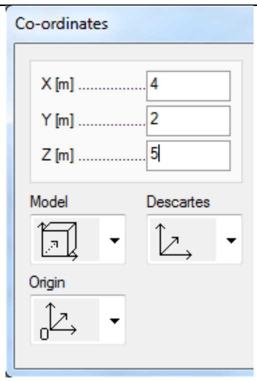
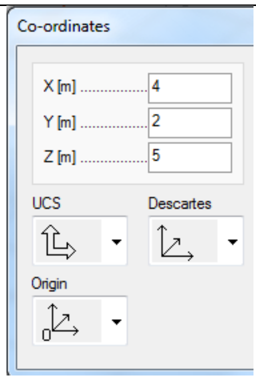
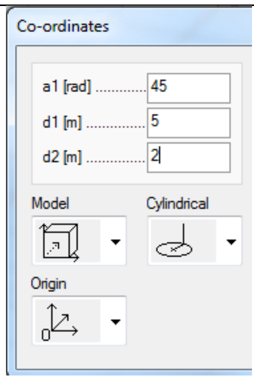
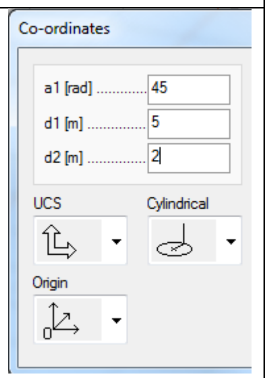
Example:
Point(LB,RB = Place point) : P 45 5 2</td></tr>
<tr><td>Co-ordinate box</td><td>
</td><td>
</td><td>
</td><td>
</td></tr>
</table>

Table: Absolute co-ordinate definition modes

Rules of co-ordinate definition:

- Co-ordinates given in *Command line* are valid only in *UCS*. But, if the *UCS* and *Global system* are in the same position (same origin and same axis directions), the defined point will be in the similar position in both two systems.
- Decimal point has to be defined as ". " and not ",".
- If you do not give Z value in the 3D modules, its value will be automatically zero.

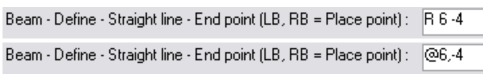
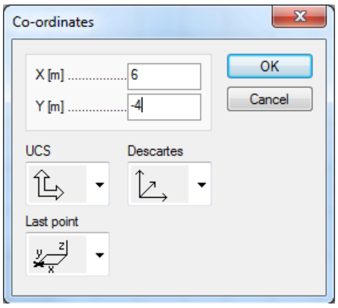
Relative co-ordinate definition

There are three ways to define a point with given distance from another point (called "relative co-ordinates").

Relative coordinates can be defined both in the *Descartes* and the *Cylindrical* co-ordinate systems and both in the *Global* and the *UCS* systems.

Relative (A): Distance from the last defined point

Distance from the last point can be defined in the *Command line* or with the *Co-ordinate box*. This mode is useful while creating line/polyline/polygon/surface with the distance from the start/previous point.

Steps	Command Line	Co-ordinate box
1	Define the start/previous point	
2	**2D**	
	Type the distance co-ordinates as: R_X_Y or @X,Y Example: 	Launch the *Co-ordinate box* dialog, change *Origin* to Last point, and give the distance co-ordinates.  Example: 
	3D	
	Type the distance co-ordinates as: R_X_Y_Z or @X,Y,Z	Launch the *Co-ordinate box* dialog, change *Origin* to Last point, and give the distance co-ordinates.

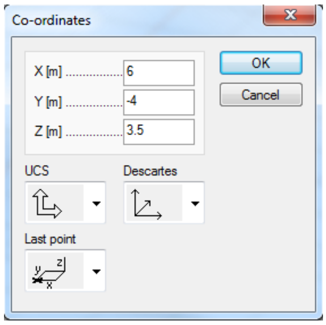
	Example: Beam - Define - Straight line - End point (LB, RB = Place point) : R 6 -4 3.5 Beam - Define - Straight line - End point (LB, RB = Place point) : @6,-4,3.5	 Example: 

Table: Relative (A) co-ordinate definition modes

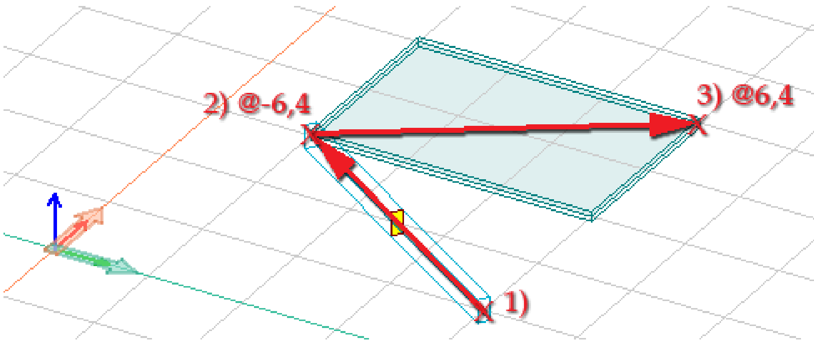


Figure: Meaning of Relative (A) way in case of beam and plate definition

Relative (B): Distance from a selected point

With the help of function key  and the *Co-ordinate dialog box*, a point can be defined from the point on which the cross-hair cursor is left. The steps of definition:

1. Leave the cursor above the required point found by one of the **Object snap tools**, which you would like to define the distance from.
2. Click  function key.
3. Set the distance co-ordinates in the *Co-ordinate dialog* according to a selected co-ordinate system.

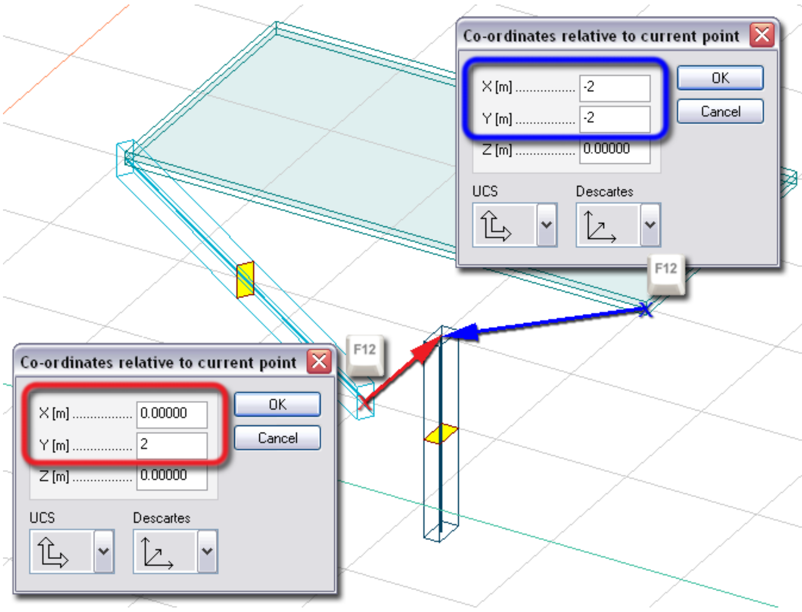


Figure: Column definition with the distance from a slab corner or from a beam endpoint (Relative (B) way)

Relative (C): Point defined on line/edge with the distance from the end point

With the help of function key F11 and the *Co-ordinate dialog box*, a point can be placed accurately on a line/edge defined with a given distance from the closer end point of the line/edge. The steps of definition:

1 Leave the cursor above the required line/edge found and next to its end point you would like to define the distance from. The **"Nearest" Object snap tool** helps you to find the line/edge.
2 Click F11 function key.
3 Set the distance (d) from the closer end-point you left the cursor in the appeared dialog box.

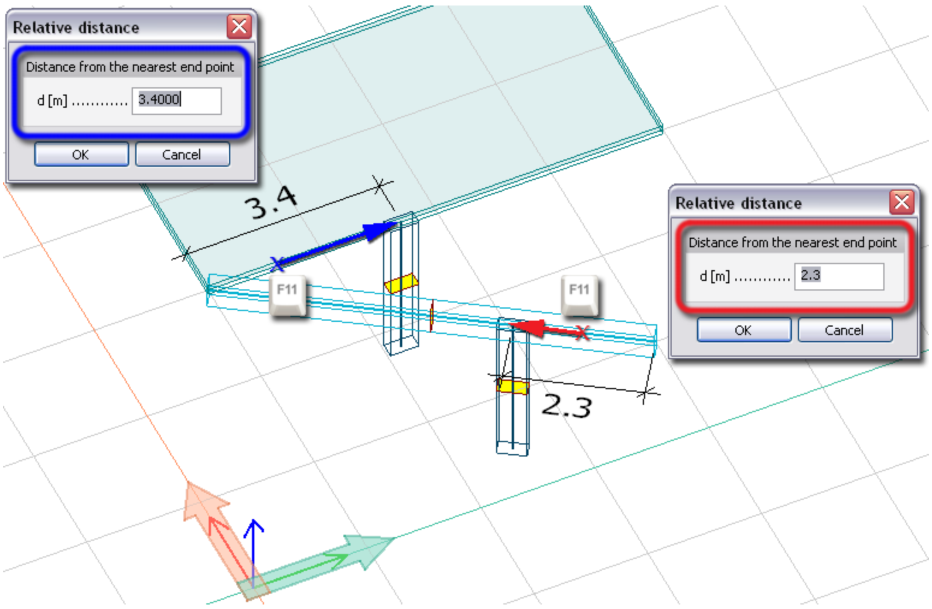


Figure: Column definition on a beam by using the Relative (C) definition mode

Calculator

If you press  while defining numeric values in a field of a dialog box, the Windows *Calculator* comes up. It is automatically filled with the content of the edit box. When you close the *Calculator*, the value calculated or typed in it will be written into the numeric field. It can be used with both *Normal* and *Scientific* views of the *Calculator*.

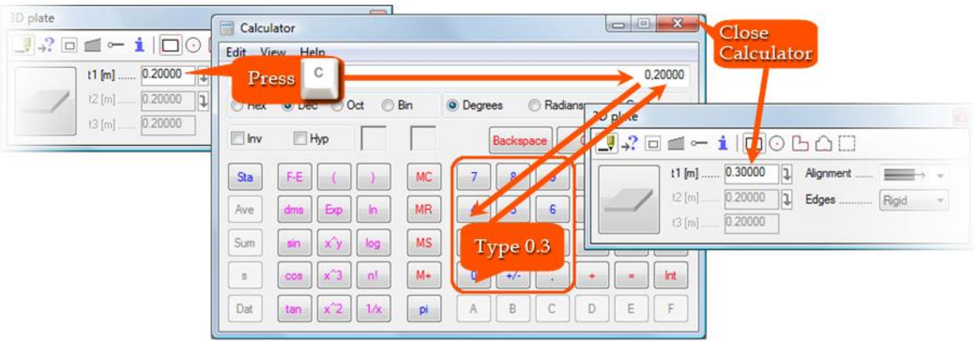


Figure: Calculator used in numeric fields of dialog boxes

Working Plane

The default working plane – where the points and coordinates are defined – is the following by different **FEM-Design modules**.

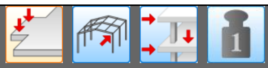
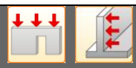
		
Design modules		**Section Editor**
Global XY plane	Global XY plane	Global YZ plane
Gravity direction		
Global Z direction	Global Y direction	

Table: The default working plane by FEM-Design Modules

As the **User-Coordinate system** defines the working plane, its custom position can be set by the UCS definition tools. The  *UCS* command (*View* menu or toolbar) gives different definition ways depending on the applied FEM-Design module.

UCS definition tools	Function	Note
Axis	Defines vertical working plane by the global Z axis and a selected **axis**	Available only in the 3D modules
Storey	Defines horizontal working plane in the plane of a selected **storey**	Available only in the 3D modules
Object plane	Defines working plane in the plane/reference plane of a selected drawing/structural region	Available in all modules, but it has not effect on the default working plane in 2D modules, where only the position of the origin changes
3 points	Defines working plane with 3 given points in their common plane	Available in all modules, but it is useful in 3D modules, where the working plane can be set in arbitrary 3D position with 3 arbitrary points
Global XY plane	Sets the working plane in the global XY plane	Available in all modules, but it has not effect on the default working plane in 2D modules, where only the position of the origin changes
Global XZ plane	Sets the working plane in the global XZ plane	Available only in the 3D modules
Global YZ plane	Sets the working plane in the global YZ plane	Available only in the 3D modules
Origin	Move the current working plane parallel with its original position into a given point	Available in all modules, but it has not effect on the default working plane in 2D modules, where only the position of the origin changes

Table: UCS – working plane definition tools (in Design modules)

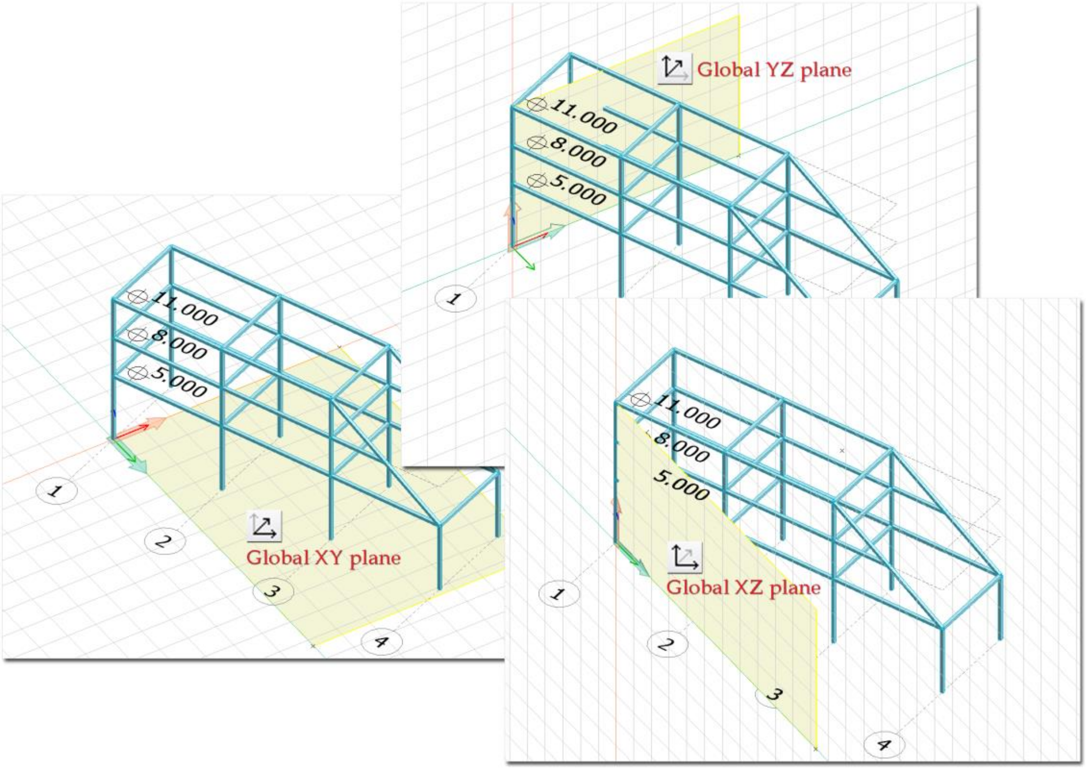


Figure: Working plane set into one of the global coordinate planes

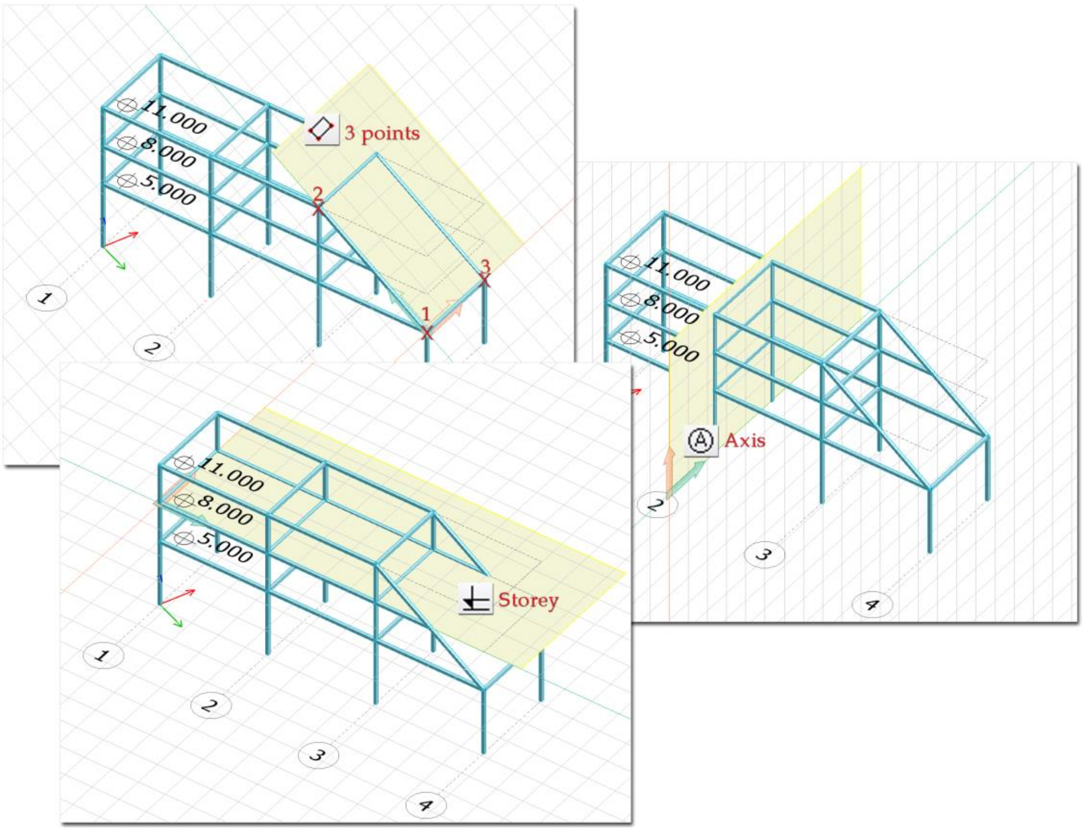


Figure: Custom-defined working planes

⚠ It is very important to set correct position (plane) for the structural model. For example, although the default working plane is the Global XY plane in the 3D design modules, place the model of a 2D frame in a vertical (Global Z) plane with the help of a vertical working plane, because the gravity direction is always the Global Z axis direction.

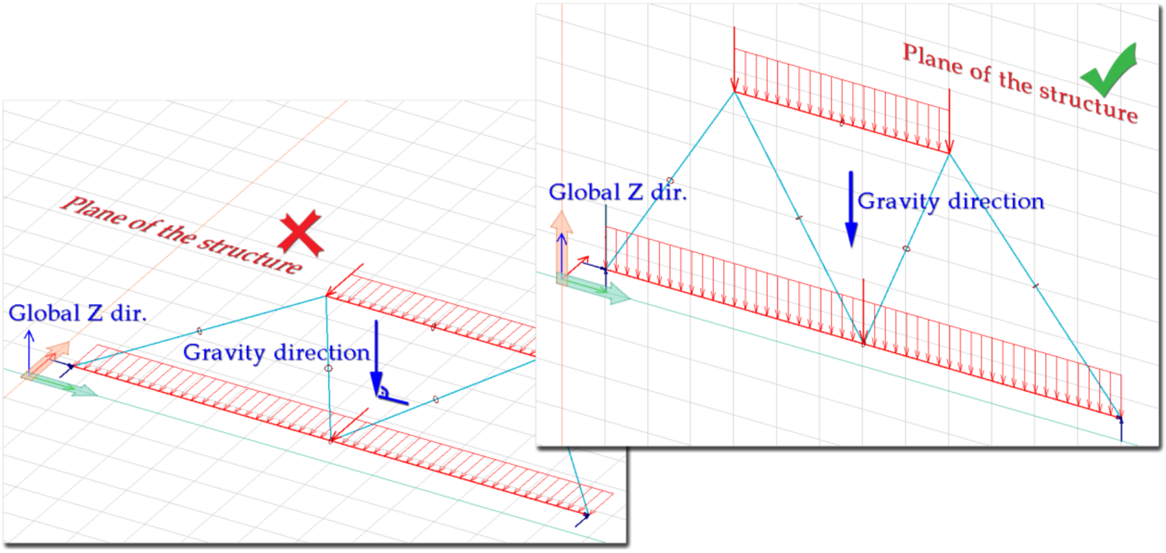


Figure: Correct model (plane) definition of a 2D frame done in the 3D Frame module

The program automatically sets the working plane into the reference plane of a planar element (plate, wall, surface support, surface load etc.), if you use the *Hole* tool to cut **hole in a structural element** or a **load**.

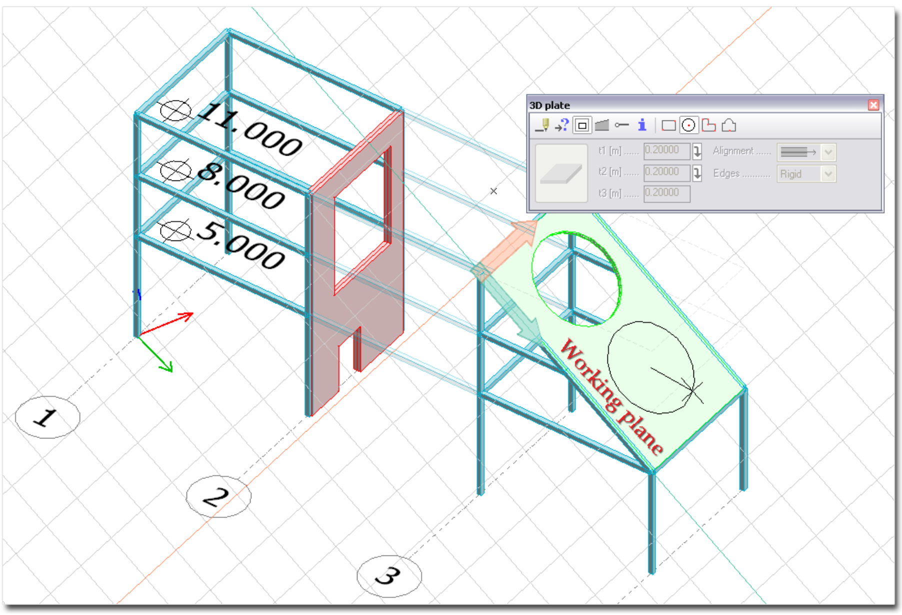


Figure: Hole definition automatically sets the working plane into the plane of the attached object

Grid Systems

The program offers a number of aid tools finding specific points and directions while drawing and editing. These are the *Grid system* and the **Object snap tools**.

Two grid systems are available in the program: *Construction grid* and *Snap grid*.

Construction grid
By default, *Construction grid* is the grid visible in the **working plane**. Of course, it can be hidden at the program settings (*Settings* > *All* > *Window* > *Grid*). That grid sets the characteristic sizes of your project.

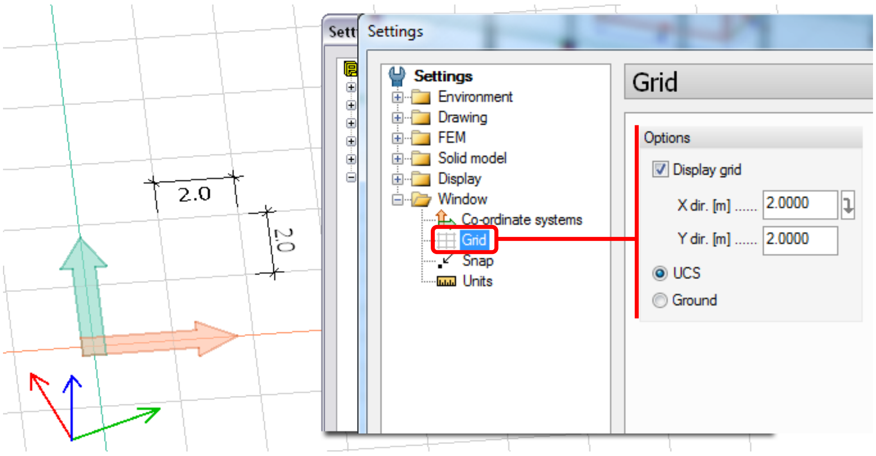


Figure: Construction grid and its settings

You can snap to the raster points of the *Construction grid* permanently by activating the **"Raster" Object snap tool** and temporarily by holding down F1 function key. By default, the grid directions are parallel with the **UCS** axis directions, so you can rotate the grid together with the *UCS*.

Snap grid
Although *Snap grid* is an invisible grid, you can join to its raster points permanently (**Raster tool**) or temporarily F1 as written before at *Construction grid*. The distribution of the grid can be set at *Object snap tools* or at *Settings* > *All* > *Window* > *Snap*. The grid distribution can be set as default for the project/program in the latter setting dialog.

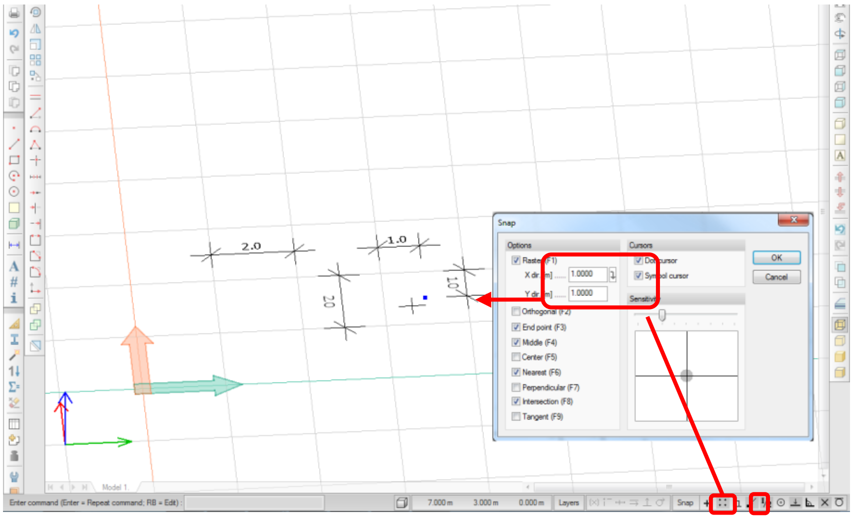


Figure: Snap grid and its settings

The X and Y directions are always parallel with the *UCS* axes.

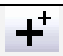
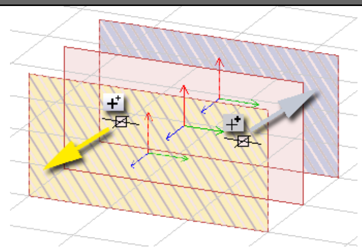
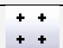
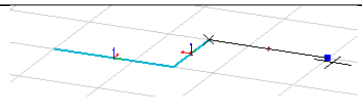
Settings affect the *Snap grid* (*Raster* dialog box):

- **Dot cursor**
  If it is active, a small black dot shows the grid points at applied *Raster* snap.

- **Sensitivity**
  It sets the sensitivity area of the raster snap for both the grids and other snapping tools. The sensitivity is constant while zooming.

Object Snap Tools

The *Object snap tools* finds and joins to special points of drawing elements and objects. The cursor form changes to the symbol of the actually used *Object snap tool*. A snap tool can be activated permanently by clicking its icon on the **Status bar** or checking its box in the *Raster* dialog box. The dialog shows the hotkeys of the snap tools activate the current tool temporarily while holding down the key.

Object snap tool	Function	Hotkey	Example
Select closest/  Farthest point 	Two states: finds the closest or the farthest point/element from the overlapped ones	-	
Raster 	Activates the Snap grip and finds its points	F1	

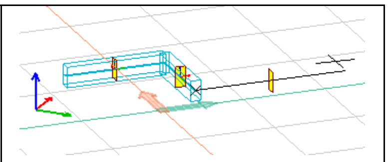
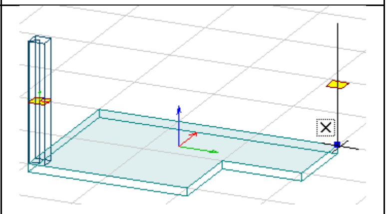
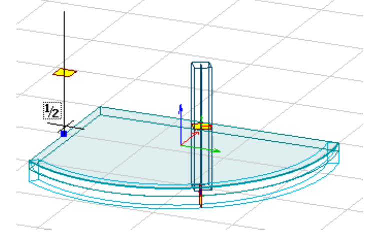
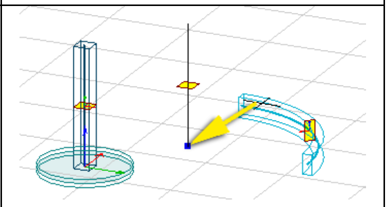
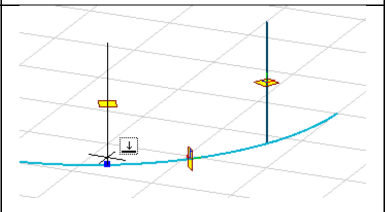
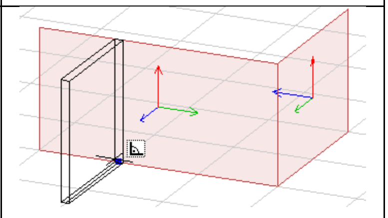
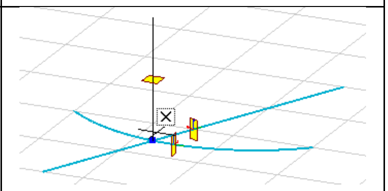
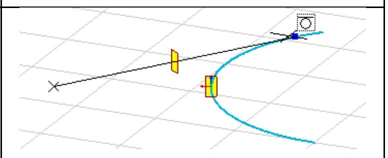
Orthogonal		Defines straight directions parallel with the UCS axes Available for line-type elements	F2	
End point		Joins to point, corner point and endpoint	F3	
Middle		Joins to line/edge midpoint	F4	
Center		Joins to the center of circle/ arc/circular surface	F5	
Nearest		Finds line/edge with its points It joins to line/edge with the closest point to the cross-hair	F6	
Perpendicular		Locates the point on an element that forms a perpendicular line to it from the last set point	F7	
Intersection		Finds the nearest intersection point of two elements	F8	
Tangent		Locates the point on a circular/ curved element that forms a tangential line from the last set point	F9	

Table: Object snap tools

Settings affect to the *Object snap tools* (*Raster* dialog box):

- **Dot cursor**
  If it is active, a small black dot shows the points found by the applied *Raster* tool.

- **Symbol cursor**
  If it is active, the symbol of the current *Object snap tool* when the cursor finds the similar snap point.

- **Sensitivity**
  It sets the sensitivity area of the raster snap for both the grids and other snapping tools. The sensitivity is constant while zooming. Too high *Sensitivity* value makes more difficult to find a point from the neighbors close to it.

Snap will not work on objects obscured by other elements (such as surfaces) outside of *Wireframe*.

User can exclude entire Drawing layers from the snapping on its objects. For more information see **„No snap" option**.

Input Devices

The following figure summarizes the main keyboard and mouse functions. Of course, these functions, which depend on the current working mode (drawing, editing, selection, documentation or no command is running), are mentioned at the related topics of this user manual.

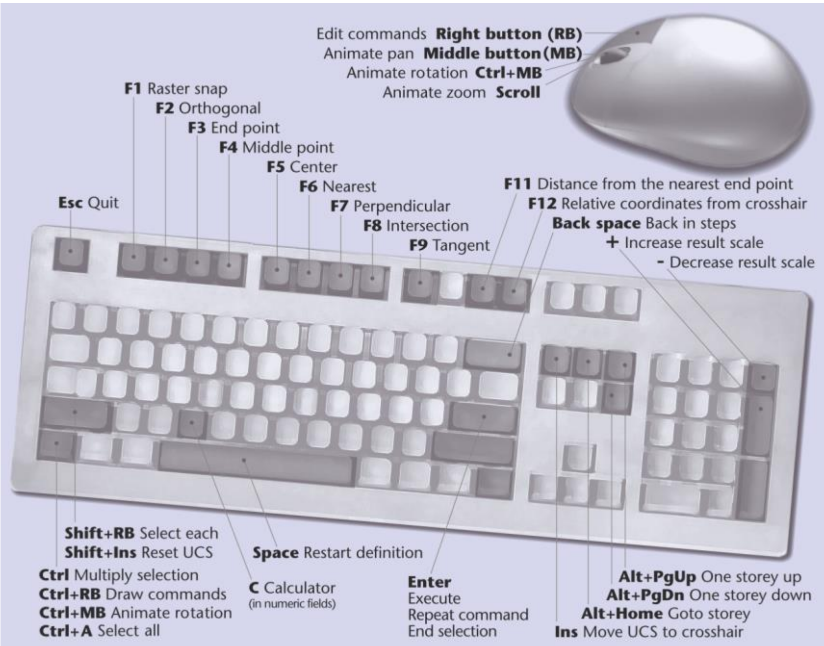


Figure: Keyboard and mouse functions

From the above displayed buttons, the following ones have special functionalities during drawing/editing:

F1 … F9 Activate temporarily the **Object snap** functions.

F11 Defines point on a line or edge by giving distance from the closest end point (**Relative (C)**).

F12 Inserts a relative point from the position of the crosshair cursor (**Relative (B)**).

Insert Moves **UCS** to the position of the crosshair cursor.

Shift + Insert Moves UCS back to the **Global co-ordinate system** position (original state).

Esc Pressing 1st Esc breaks the current command (e.g. placing a structural element).

For pressing 2nd Esc the active tool palette closes.

PgUp Opens the properties dialog (**Default settings**) of the current command.

Enter Confirms data input / repeats the last command / finishes multi-selection.

Space Restarts the steps of the current command.

Backspace Goes back to previous step in a multi-step command.

Some commands can be accessed by hotkeys. See those commands in the **Menu bar** or in the **Toolbars**.

Selections

Mouse and keyboard selection

FEM-Design offers various selection modes to select objects for requiring and modifying their properties and for editing. If selection is possible, the available selection modes appear in the **Command line**.

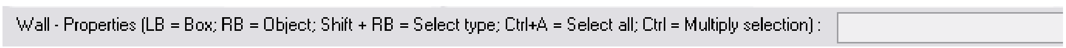


Figure: Selection modes displayed in the Command line

Use Space to restart selection, if you make mistakes while selecting.

Box selection

Rectangular selection box can be defined with two points placed with the mouse button. Depending on the box definition, the box selects elements:

- that are fully in the defined rectangular area → the end corner of the box is on the right from the start one,
- that are fully or partly (intersected) in the defined area → the end corner of the box is on the left from the start one.

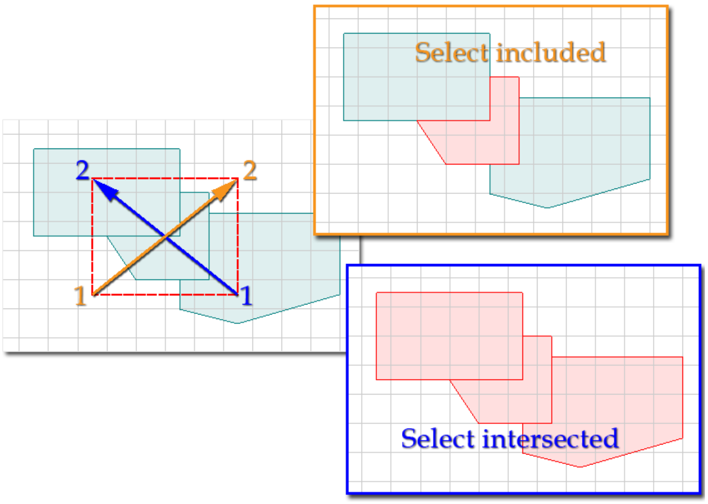


Figure: Box selection modes

Object selection

Use on an object to select it directly. If you click on a common part of more objects, the first drawn one will be selected.

Holding pressed and by using objects can be added to or subtract from the current selection.

Selection of same type elements

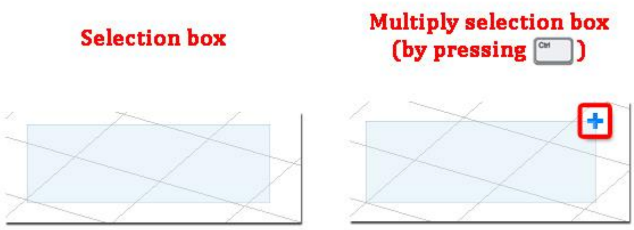
It is a quick selection to select visible elements defined with the same command (e.g. selection of all walls created by the *Wall* command). Just hold down and select on element (e.g. one wall element) with , and all same type elements will be selected (e.g. all walls) independently their properties differ or not.

Select all

Depending on the current command, all visible elements (e.g. for the *Edit* > *Move* command) or all visible same type elements (e.g. for the *Properties* tool) will be selected by using and together.

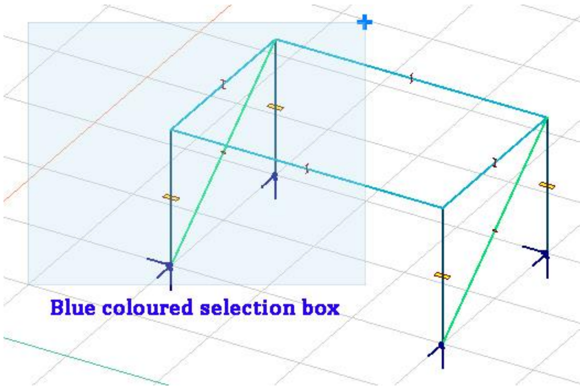
Multiple-selection

- '+' sign is drawn if multiply selection is active (by pressing)

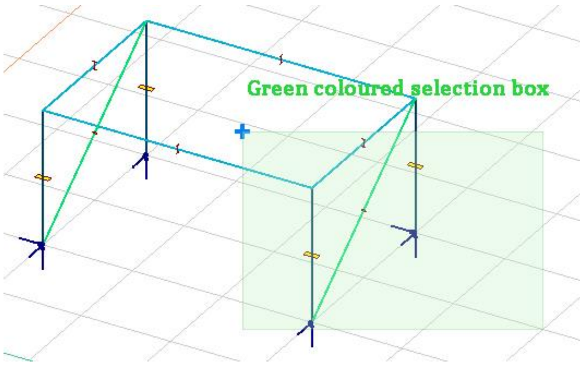


- The selection box has different colours depending on the way of the selection:

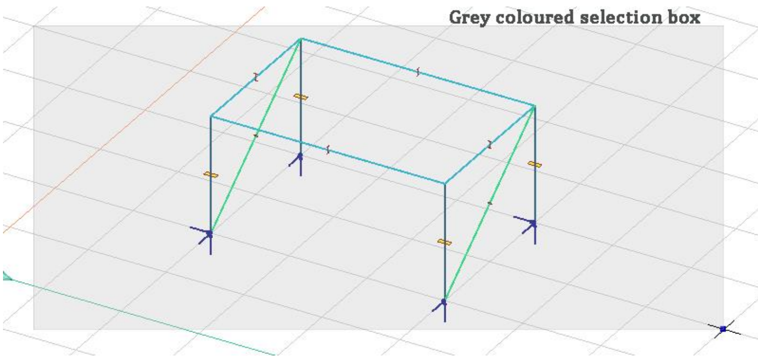
  a. By ‘left-to-right’ selection (the included objects get selected) the colour of the selection box is blue.



  b. By ‘right-to-left’ selection (all intersected objects get selected) the colour of the selection box is green.



  c. By area selection (e.g. to print the selected area) the colour of the selection box is grey.



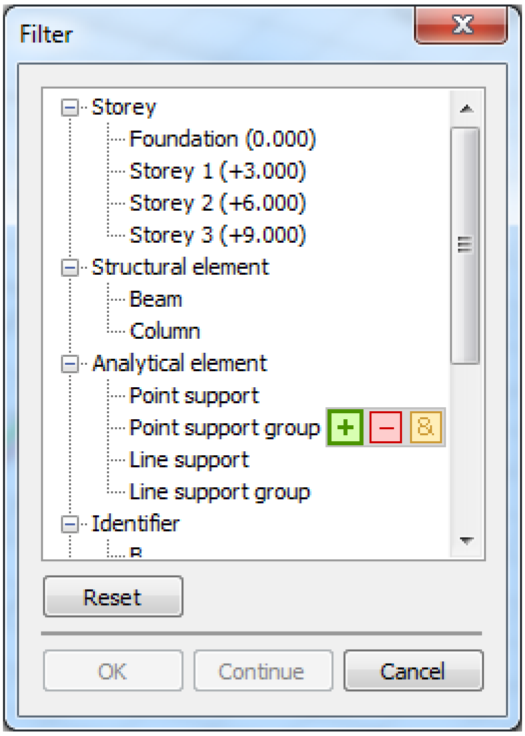
Multiple selection can be ended by  +  (it has the same effect like pressing ).

- For pressing  +  in the application window (without any active tool palette) the quick menu appears.

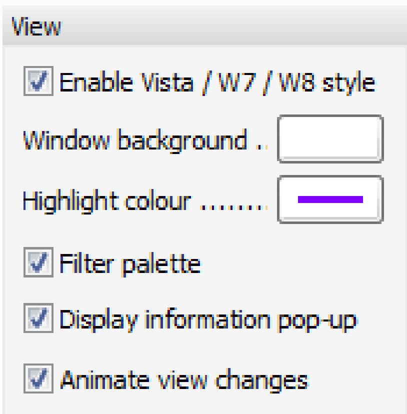
- When a tool palette (e.g. Beam tool palette or Point load tool palette) is active, for pressing  +  the properties of the items belonging to the active window appear.

Filter selection

Filter is a pop-up window in the workspace, which helps to select the requested objects easily for all kind of editing and modifying functions. When clicking on it, the following appears:



With the cursor pointing on Point support group "+", "-" and "&" buttons appear, enabling the user to add or remove point support group elements. If you want to select most, but not all of the point support groups, you should click "+", and then "Continue" for deselecting the few individual supports not needed for the selection.
The "&" button works as an "AND" boolean operator: If you want to select the columns on the second storey, you have to click "+" at column and then "&" at storey 2.
After closing the Filter palette, it can be switched on again by a right click on any toolbar, or on the main menu zone. Another way is available through Settings>All…>Environment>General>View where filter palette can switched on and off:



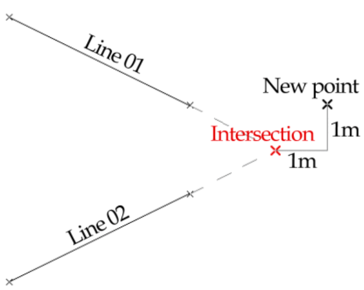
User defined filter in Tools menu is a function for saving some frequently used selections. If you define a filter of your own, it will appear at the bottom of the list. When defining, a name has to be given and then desired objects have to be selected. A window appears "Filter has been created." After this, if needed, with properties "?", the name of the block can be modified; with "+" you can add other members, and with "-", you can remove undesired ones.

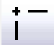
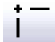
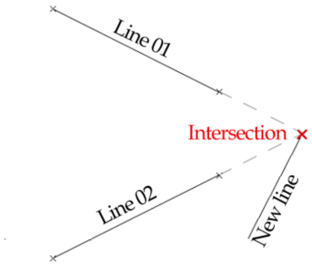
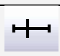
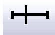
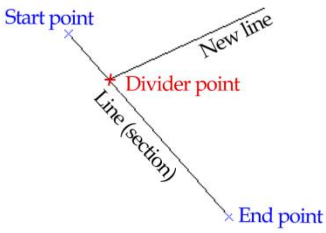
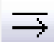
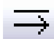
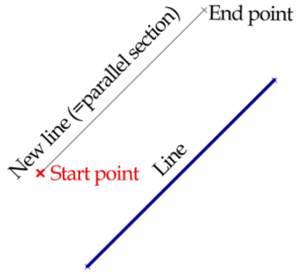
It also can be exploded with :

At first (like in the figure above) "OK" is grey, but after clicking on Explode button "OK" becomes active. Don't forget to validate the explosion with "OK", before closing this window.

Point/Direction Editors

The program offers tools for defining special points and directions while drawing and editing. You can reach them from the **Status bar**.

Object snap tool	**Function**	**Example**
Multi steps	Fixes reference points defined by the following tools	*Define a new point with X=Y=1m distances from the virtual intersection of two predefined lines:* Launch *Point* (*Draw* menu), and then activate both and . Define the virtual intersection of the two lines and place the cursor close to it. Press F12 and type 1 [m] for X and Y in the *Co-ordinate dialog box.* 

		Pressing  on *Status bar* defines the new point.
Virtual intersections of lines 	Edits the intersection of two lines crossed virtually each other	*Start a new line from the virtual intersection of two predefined lines:* Launch *Line* (*Draw* menu), then activate , and select the two lines one-by-one. The new line starts from the virtual intersection. 
Divider point 	Edits a point on a straight-line with a given ratio: ratio = the distance of the divider and start point divided by the section length	*Draw a new line from the quarter point of another line:* Launch *Line* (*Draw* menu), and then activate . Define the start and end point of the line (section), and set the ration to 0.25 (from the start point). The new line starts from the quarter point of the defined section. 
Parallel with line 	Defines direction parallel with a line	*Draw a new line parallel with another line starting from a point:* Launch *Line* (*Draw* menu), and define the start point of the new line in the point. Activate  and select the line. Click the point again and finally define the end point of the new line. 

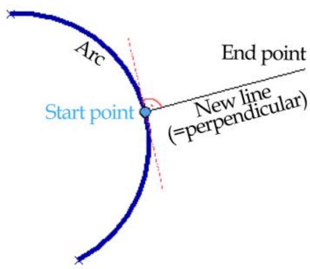
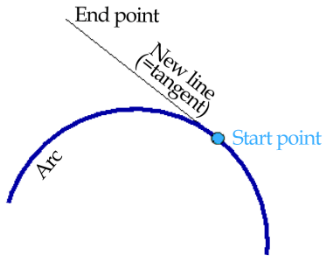
Perpendicular from line	Defines direction perpendicular from a line	*Draw a new line perpendicular from a fixed point of an arc:* Launch *Line* (*Draw* menu), and define the start point of the new line in the fixed point. Activate and select the arc. Click the fixed point again and finally define the end point of the new line.	
Tangent from line	Defines tangent direction from an arc	*Draw a new line tangent with an arc and starts from a fixed point of the arc.* Launch *Line* (*Draw* menu), and define the start point of the new line in the fixed point. Activate and select the arc. Click the fixed point again and finally define the end point of the new line.	

Table: Point and direction editors

Navigation

Thanks to the improved video engine, the advantage of all modern displaying technologies such as live pan, live rotation, live zoom, special display modes and transparency (see the next chapter) is available in FEM-Design.

The suitable graphic engine can be set at **Settings > All > Environment > General > Graphic engine**.

Pan

Panning view on the drawing area can be done in different ways. "Panning" modifies the current view only and not the position of elements.

First start *Animate pan* from the *View* menu or *View* toolbar, then press or and drag the mouse to pan the current view. If you are in panning mode, you cannot edit the model and the currently running command is also paused. Click to exit panning mode and return to editing mode.

However, while in editing mode, you can temporarily simulate *Animate pan* by pressing the to pan the view. Release the mouse button to stop panning.

With the command *Pan* (*View* menu) the current view can be moved with a displacement vector defined by two points.

Zoom

Different tools allow you to zoom on the current view.

Live zoom

First start *Animate zoom* from the *View* menu or *View* toolbar, then press or and finally move the mouse up to zoom in or move it down to zoom out. If you are in zooming mode, you cannot edit the model and the currently running command is also paused. Click to exit zooming mode and return to editing mode.

However, while in editing mode, you can temporarily simulate *Animate zoom* by scrolling forward to *zoom in* or backward to *zoom out*.

Other zooming tools of View menu

Zooming tools	Function
Zoom margin	Fits the view to all visible elements and leaves a 5% additional margin around it
Zoom in	Enlarges the view area defined with a box and fits it to the current window size
Zoom out	Displays the current view in the defined box area
Zoom enlarge	Zooms in to the 133% of the actual size

Zoom reduce		Zooms out to the 75% of the actual size

Table: Zooming tools

Orbit
View rotation can be done in different ways. "Orbit" modifies the current view only and not the position of elements.

First start *Orbit* from the *View* menu or *View* toolbar, then press or and drag the mouse to turn the model around its center point. If you are in rotation mode, you cannot edit the model and the currently running command is also paused. Click to exit rotation mode and return to editing mode.

However, while in editing mode, you can temporarily simulate *Orbit* by pressing together with to orbit the model. Release the keys to stop rotating.

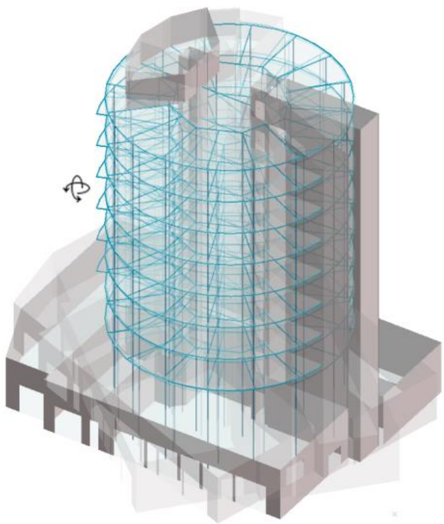
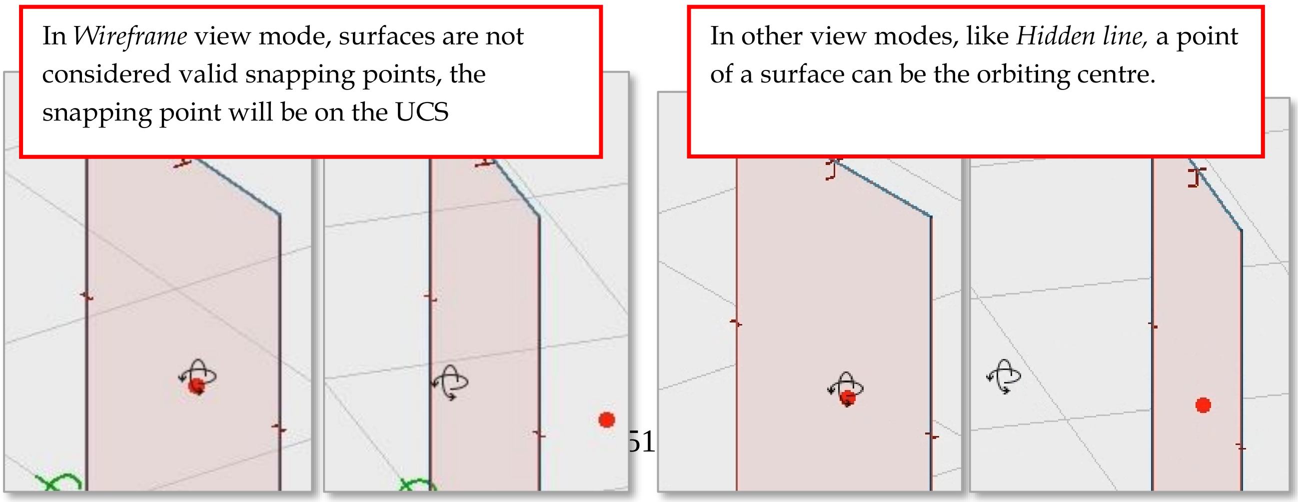


Figure: Temporary Orbit

Alternatively, you can press together with to orbit around the closest valid snapping point to the cursor; in some cases, this may result in a more predictable orbiting behaviour. Valid snapping points include points specified by the **Object Snap** tool, or outside *Wireframe* view mode, also the closest surface point to the cursor.



Rotation view can be set in a dialog box too with the command *3D Rotation* (*View* menu).

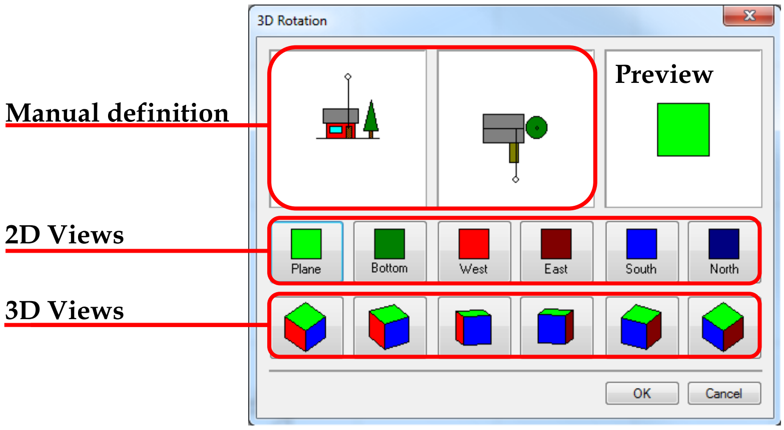
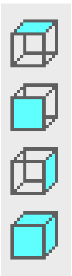


Figure: Rotation by dialog box

Views

Besides of navigation tools, specific built-in 2D and 3D views can be used to set the required project view. All views can be stored with given names (user-defined views).

Built-in views



Plane view (top view; Alt + F1)

South view (side view; Alt + F2)

East view (side view; Alt + F3)

Space view (general 3D view; Alt + F9)

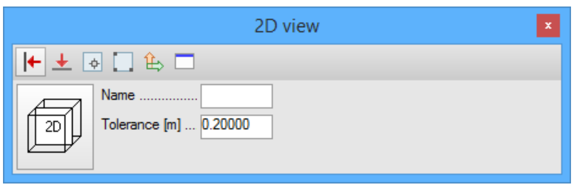
Just click one of the following view buttons to set a specific 2D view or a general 3D view:

User defined views and Home view

To return frequently to a view of the current project, you can save it as "*Home view*" (*View* > *Home view* > *Save as*). *Choose View* > *Home view* > *Return* to set this view just with one-click. *Home view* is saved with the project.

Current view can be also saved with names in a dialog appeared by clicking (*View* > *User defined views*) or using Alt + M. Define a name and click *Add* to store the view. If you have more than one user defined views, *Select* displays preview of them. To use a stored view as the current one, click *Select and exit*. The list does not contain the *Home view*. User defined views can be saved with the *2D* and *3D View* commands too.

specifies a section view, which contains objects within a given range (*Tolerance*), while defines a space view perpendicular to a given plane. The section/plane can be defined with the following tools:



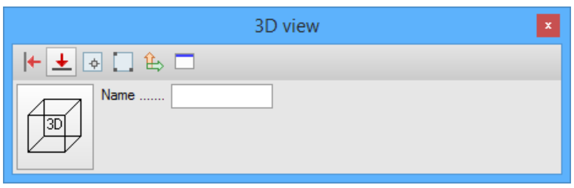


Figure: 2D and 3D View commands

Vertical plane

Creates a section/view parallel with the global Z axis by selecting two points.

Horizontal plane

Creates a section/view perpendicular to the global Z axis by selecting two points.

Object plane

The section/view plane is parallel with a selected region (*Plate*, *Wall* etc.) plane. The final direction can be set by moving the cursor at the selected storey. The view direction is always the opposite of the local Z′ direction of the selected surface object.

3 points

The section/view plane is defined by adding three points. The first point defines the origin, then the second point defines the X direction together with the first one, and finally the third point defines the Y directional extension of the plane. The last point also defines the view direction (an arrow shows it).

User Co-ordinate System

The section/view plane is set to the X-Y plane of the **UCS**. The view direction is always the opposite of the Z axis of the *UCS*.

Dialog

With this tool, you can define the position of the section/view plane in a dialog box. At *3D View*, *Dialog* opens **3D Rotation**.

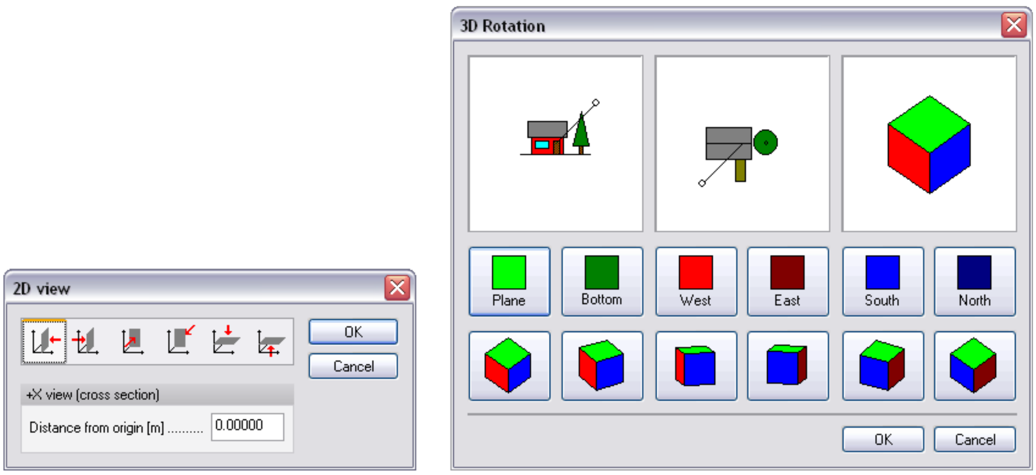


Figure: Section/view plane definitions in dialogs

Swapping among views

Moving one step back from the current view, zoom operations etc., click (*View* > *Previous view*) or use Alt + P. To go to the next view, click (*View* > *Next view*) or use Alt + N.

Select view

With this tool, you can select some specific views (Storey view, Axis view and User defined view).

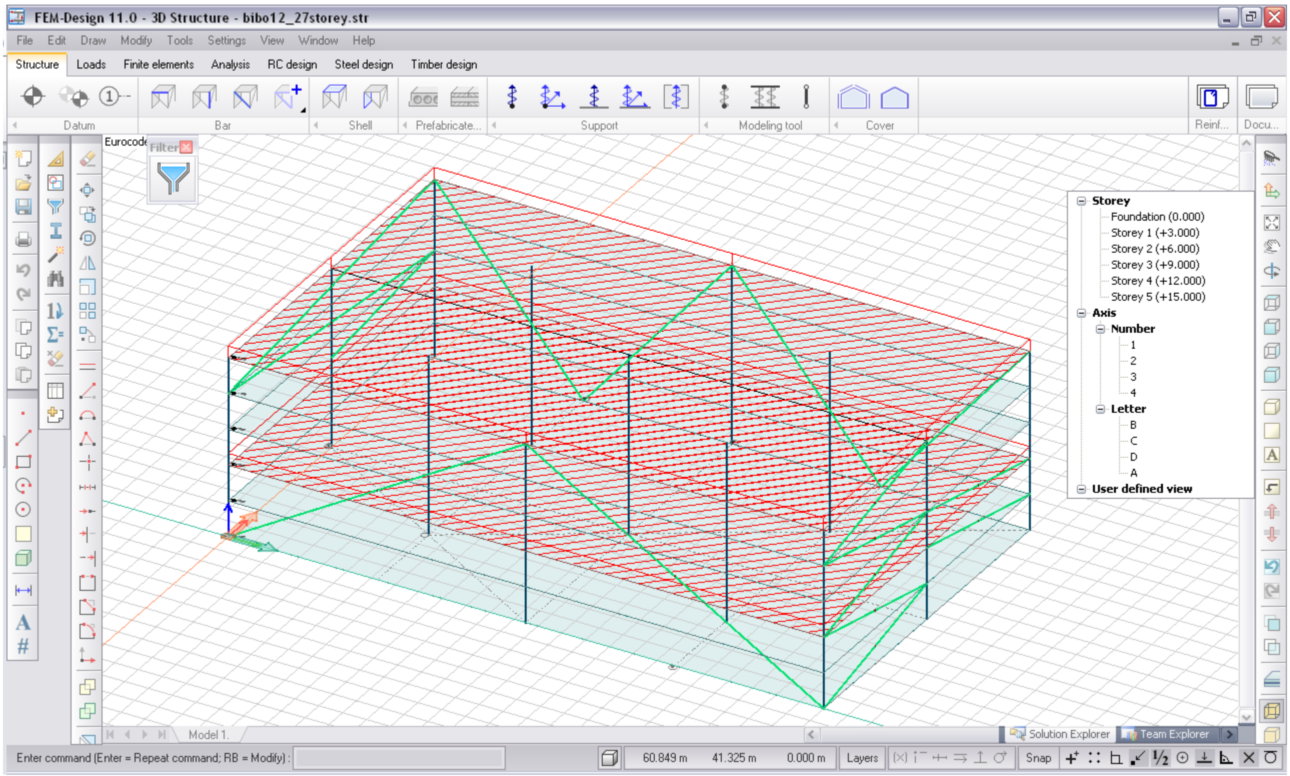


Figure: Example for using Select view command

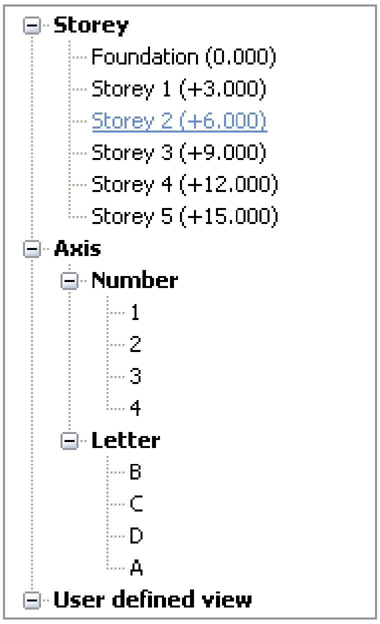


Figure: List of selectable views

After choosing Storey 2 you can see all the elements belonging to the current level, but for example, if you have high columns going through the whole building, they will belong only to the level where they have been created. You can return to previous view by clicking on the icon  again and then selecting "Return to 3D view".

You can also obtain cross-sections of the whole building. By choosing axis no. 1. (which is the global X axis) in the example above, this is what you can see:

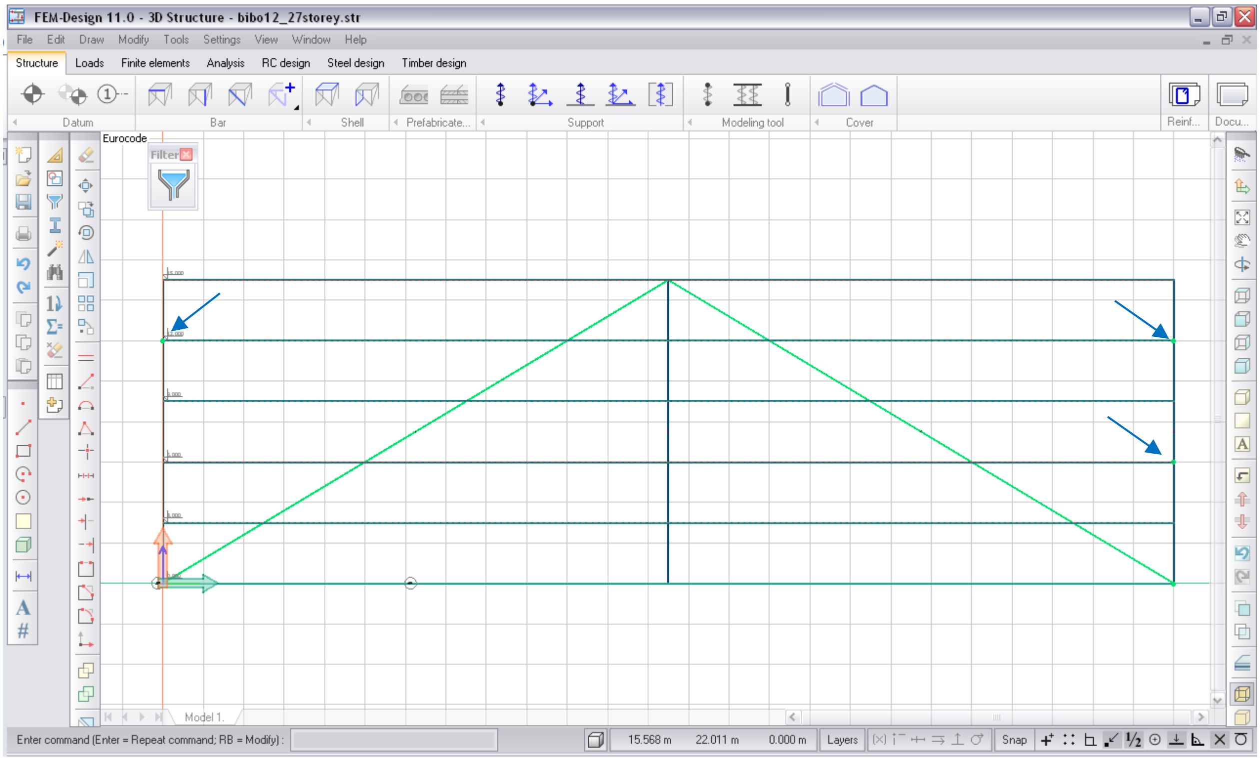
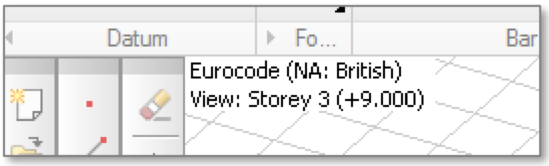


Figure: Cross-section view of the building

The green dots emphasized with the blue arrows in the figure above indicate the truss elements pointing out of the plane determined by the current axis and global Z.

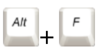
The name of the selected view is displayed in the upper left corner under the current code.



Display Modes

Thanks to the graphic engine, the following display modes (*View* menu or *View* toolbar) are available:

Wireframe

 or  displays the model with all edges and lines drawn, but with no surfaces drawn.

Hidden line

or Alt + L displays the model with all edges and lines drawn except those covered by surfaces.

Shading

or Alt + S displays the model with all surfaces shaded according to their assigned layer color. A default (non-editable) light source provides illumination for shading. All non-building elements such as drawing elements (lines, regions, texts etc.), loads, supports and the grid-system (if not hidden) are also displayed in shading mode.

Shading with edges

or Alt + E displays the non-occluded edges of the model in addition to *Shading* mode.

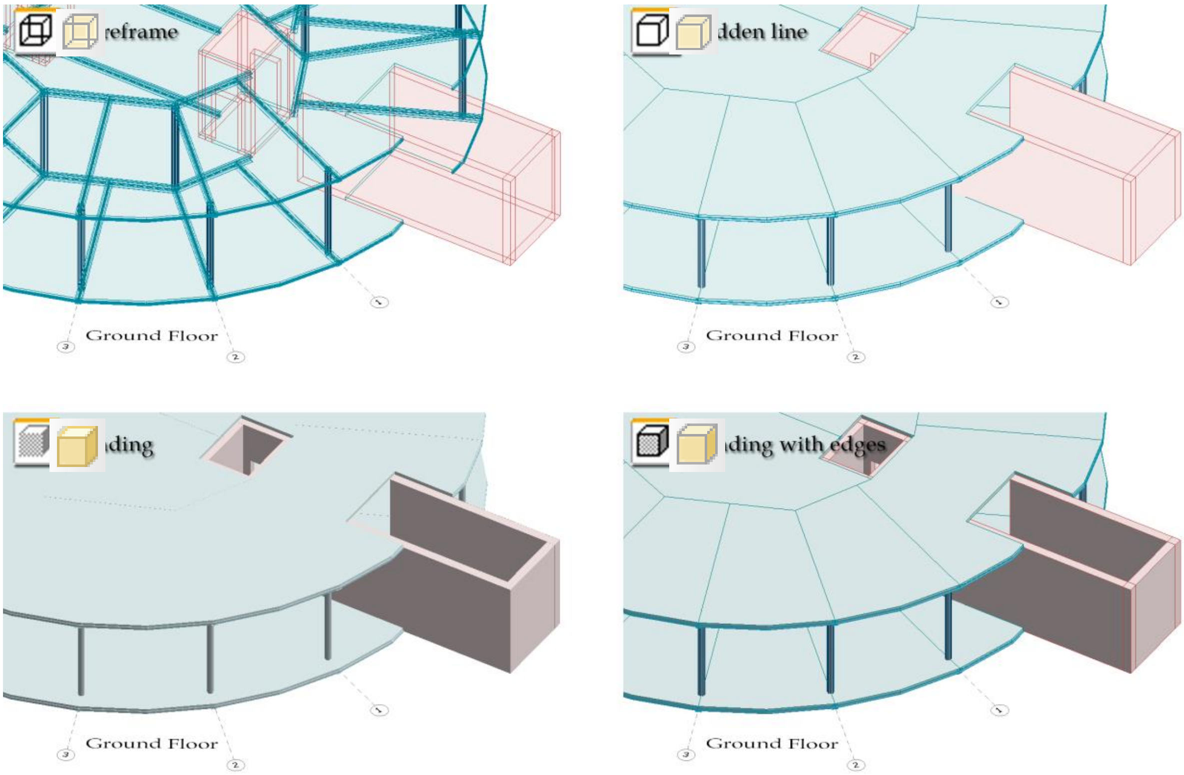


Figure: The meanings of the view modes

Display thickness

is a quick tool developed to real-time display or hide 3D solid representation of structural objects (*Plate*, *Wall* etc.). It's a switch that can be turned on/off in the *View* menu/toolbar or by using Alt + D.

	ON	OFF
Beam, Column and Truss member	3D solid	Reference line
Shell	3D solid	Reference plane

Table: The visual effects of "Display thickness" switch status on element types

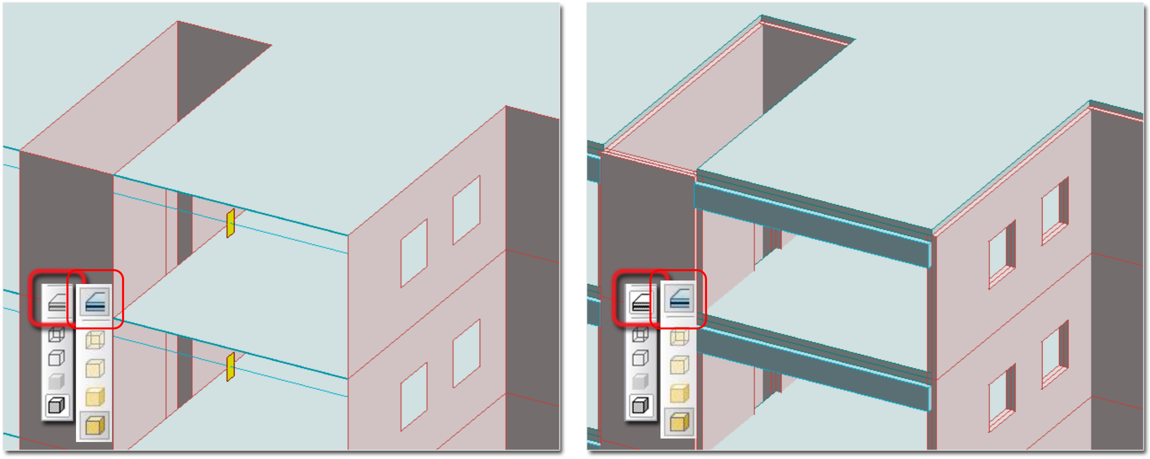


Figure: The statuses of "Display Thickness"

Hiding Element

The hide and show functions are merged in one command called *Hide/show objects*, which is available in the *View* menu/toolbar or by using Alt + H. To hide elements use the *Hide* tool for selecting them. The selected elements are displayed as transparent till closing the command and then they disappear on screen. Resuming the command, the transparent objects represent the previously hidden objects. Apply the *Show* tool to display hidden elements again by selecting them. Click *All (!)* to hide or show immediately (without any selection) every project elements depending on the *Hide* or the *Show* tool is the active.

Hide all stories of the building except the first story and the ground floor.

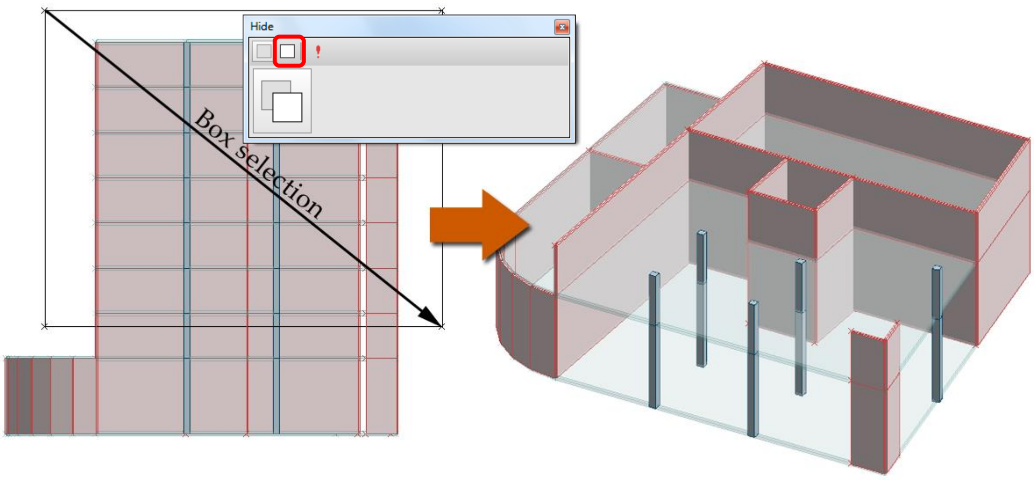


Figure: Hide tool and box selection

Unhide the second and third stories.

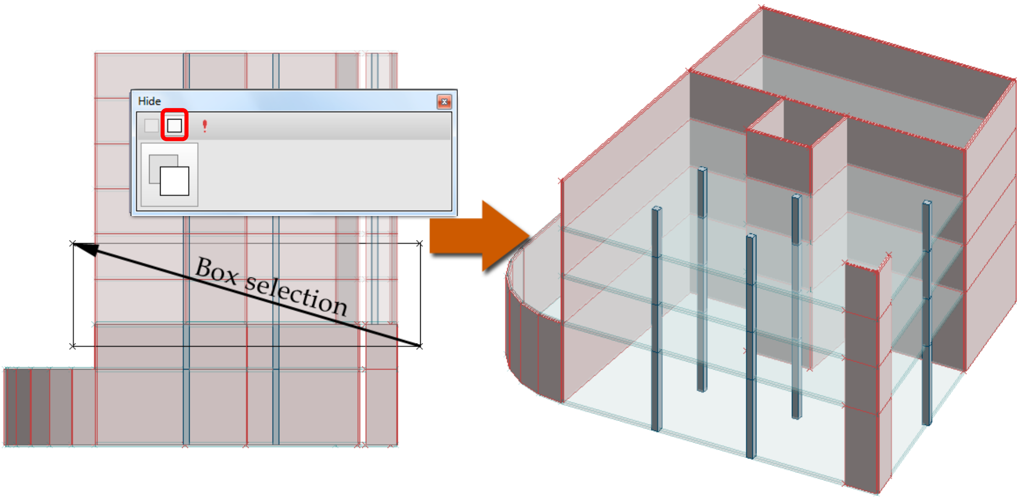


Figure: Show tool and box selection

Transparency

Transparency can be set for all element types such as structural objects, drawing elements, loads, supports etc. Start the *Transparency* command from the *View* menu or *View* toolbar or with Alt + T. Set the transparency value in the *Set transparency* dialog and then select elements you would like to display as transparent.

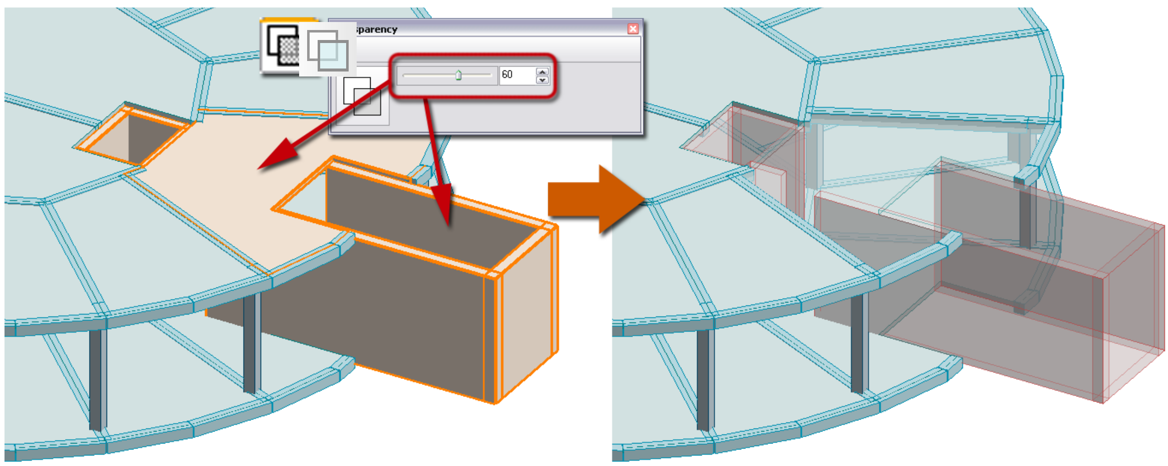


Figure: Example for Transparency

To reset transparency settings to opaque for all elements, set the transparency value to 0% and click *All* (!).

At solid (**Display thickness** switch is on) and **Hidden line/Shaded** representation of plates/walls, the finite element mesh can be displayed, if transparency is given for the host surface element.

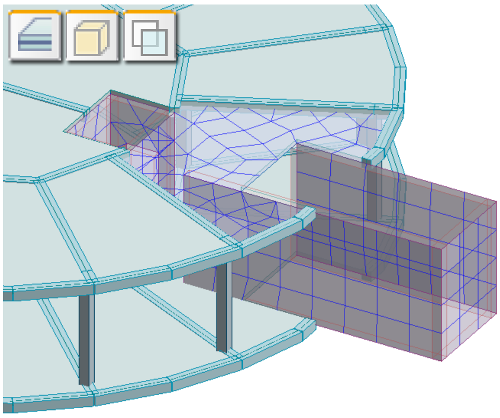


Figure: Display finite elements in solid and shaded view

To set the transparency of all walls to 50% in a project, select one wall with Shift + .

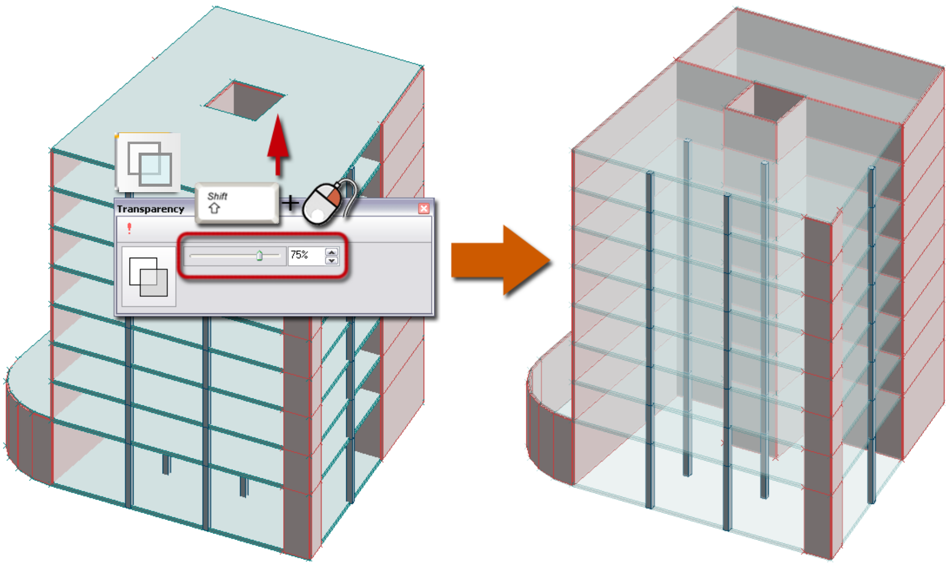


Figure: Transparency command combined with "Select type" selection

Axis

Axes (Axis objects) have multifunction in FEM-Design. They can be used to find special points and to define different kind of planes:

- **Intersections of axes**

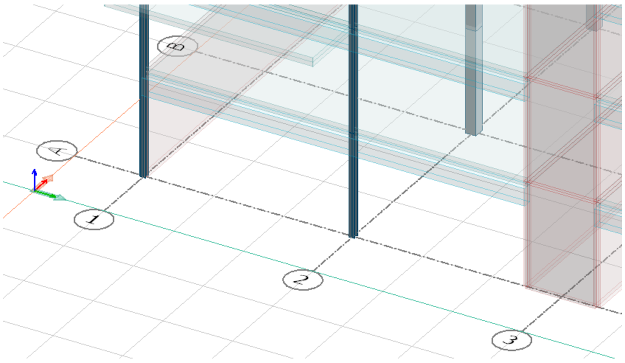


Figure: Intersections may define column base points

- **Working plane**

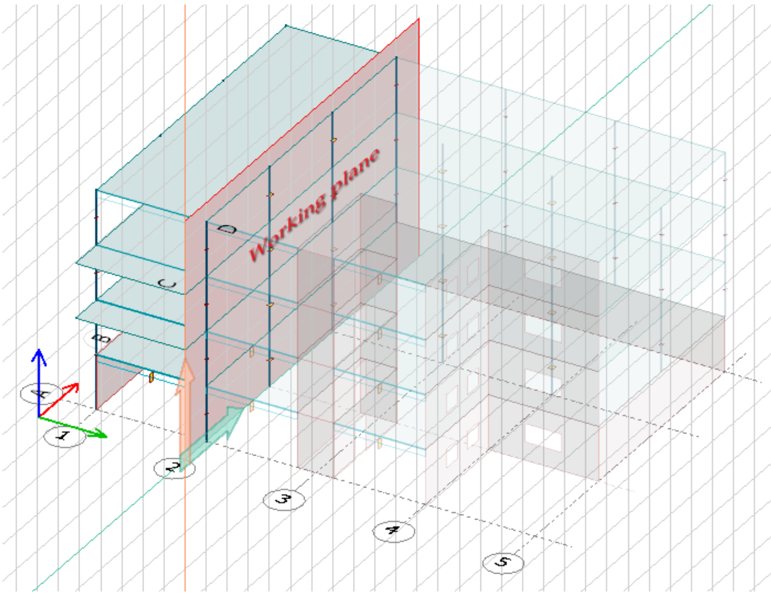


Figure: Working plane defined on an axis by UCS

- **View planes (2D View and 3D View commands)**

- **Project planes (used in the model transfer between the FEM-Design modules)**

Wall module can open an axis plane defined in the *3D Structure* model and imports the walls attached to the view.

New axis or axis-system can be defined with the *Axis* command of the Structure tabmenu. Both constant and variable distribution of axes can be defined in the *Axis* tool palette.

Definition steps

1. Set the required horizontal **working plane** (UCS), where you would like to place axis/axes.
2. Use the *Define* tool. Define the distribution in the *Multiple axes description* field. If distance is not defined, you can place one axis object as a line on the *UCS* working plane.
   Constant spacing:

   (n-1)xd — where *n* means the numbers of axes, *x* is the multiplication sign and *d* means the distances between the axes.

   Variable spacing:

   (n1-1)xd1,(n2-1)xd2,... — where *ni* means the numbers of axes per different spacing, *x* is the multiplication sign and *di* means the different distances.

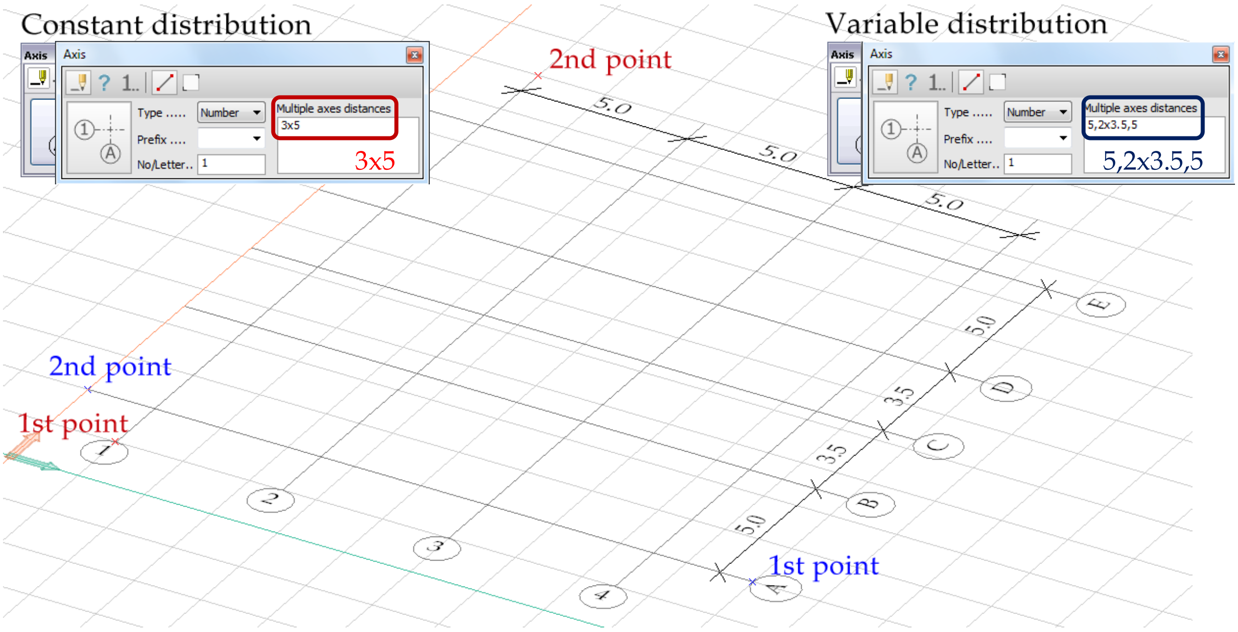


Figure: Axis-system with constant and variable spacing

3. Label type (*Number* or *Letter*), starting character (*No/Letter*) and *Prefix* can be defined for the new axis/axis-system in the *Axis* tool palette.
4. Display settings of axis symbols, colors and label can be set at *Default settings*.
5. An axis or the start axis of a system can be easily defined as a line with 2 points in the working plane. The distribution orientation can be changed by clicking before the definition of the axis end point. The program places all axes on the *Axes* **object layer**.

Optional steps:

6. The numbering of an axis or one direction of an axis-system can be modified with the *Renumbering* tool. Define new (start) numbering value in *No/Letter* field, and then select the axis you would like to renumber.
7. Display settings of the axis symbols can be modified with the *Properties* tool.

Storey

Stories (Story objects) have multifunction in the *3D Frame*, *3D Structure* and *PreDesign* modules. They can be used to define different kind of planes:

- **Working plane**

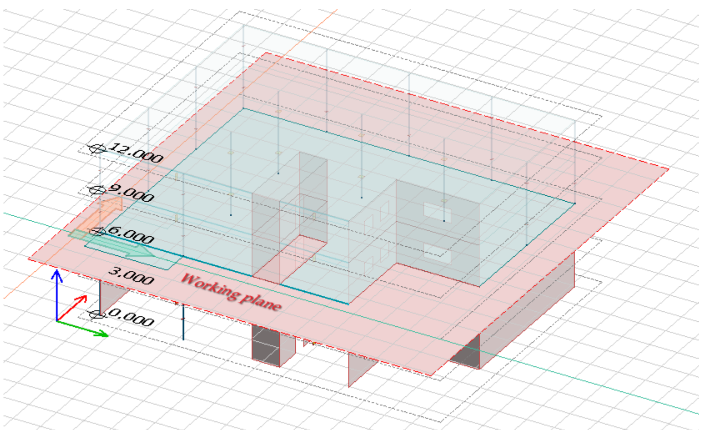


Figure: Working plane defined on a storey by UCS

- **Project planes (used in the model transfer between the FEM-Design modules)**

  Plate module can open storey defined in the *3D Structure* model and imports the slabs, the supports and the loads attached to the loaded storey.

New storey or storey-system can be defined with the *Storey* command of the Structure tabmenu. Both constant and variable distribution of stories can be defined manually in the *Storey* dialog box.

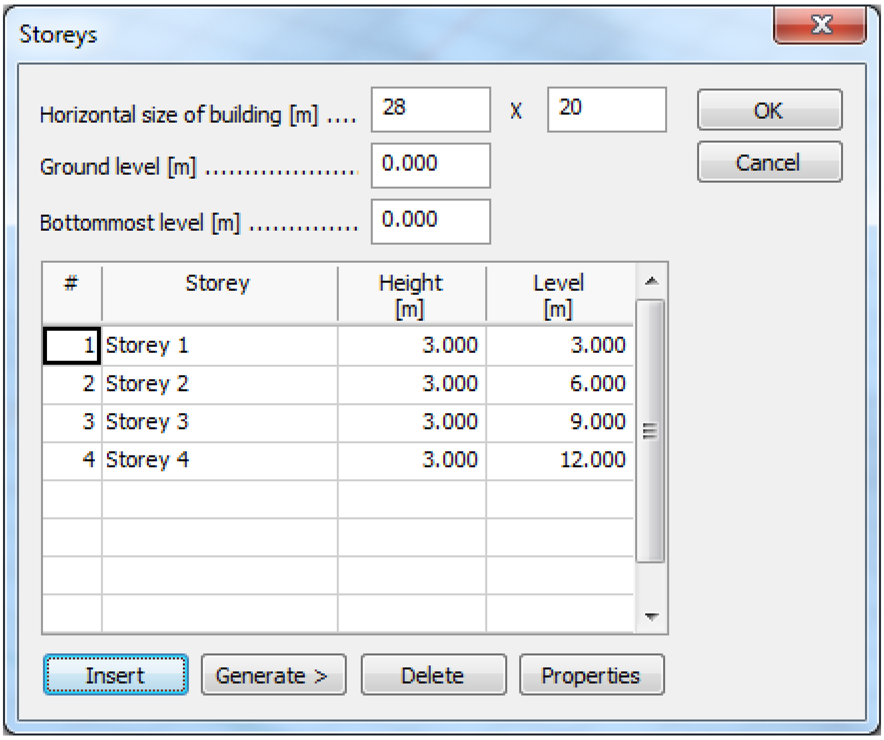


Figure: Storey dialog box

Definition steps

1. Set the *Horizontal size of the building*. This rectangular size will be the floor plane size of the storey object.

   ⚠ The horizontal size and the summa height (see later), which define the whole model size, effect the **automatic wind load** calculation.

2. Set the height of the *Ground level* and *Bottommost level*.

   ⚠ *Ground level* also effect to the **automatic wind load** calculation, because the load values depend on the distance from the foundation level.

   Bottommost level

   This is the storey's origin height, so the storeys are starting from this level.

3. To define the first storey, set a name for it in the *Storey* cell of the first row, and then give the *Height* of the storey. Then, define further required stories in the same way. You can also edit the level of a storey that automatically updates the story height.
4. Set the display settings (symbol and label size of the level dimensions) at *Properties*.

5. Clicking *OK* generates and displays the storey-system in the project. The program places all stories on the *Stories* **object layer**.

Creating a storey-system automatically generates view planes by stories. Fast navigation among those storeys can be done with the keyboard or by using **Select view**:

One storey up; Alt + Pg Up

One storey down; Alt + Pg Dn

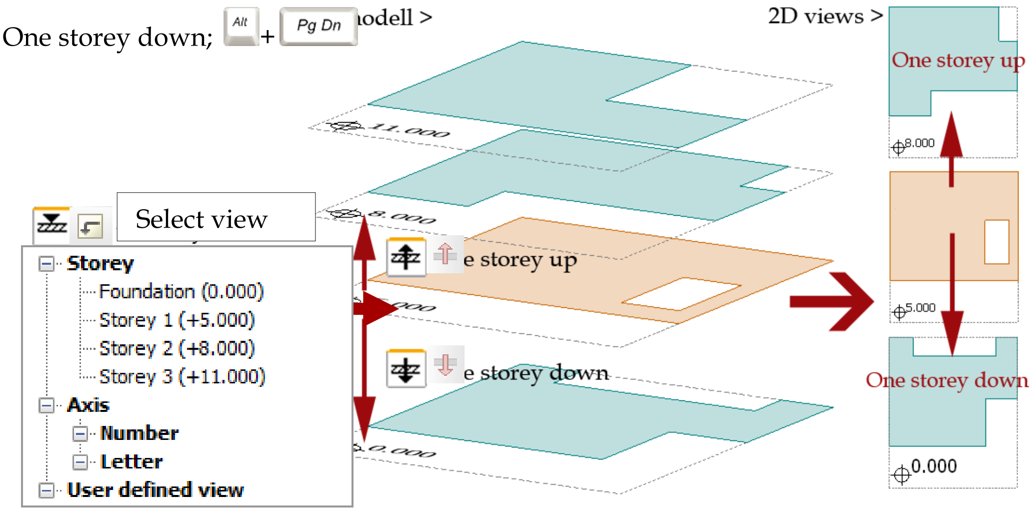
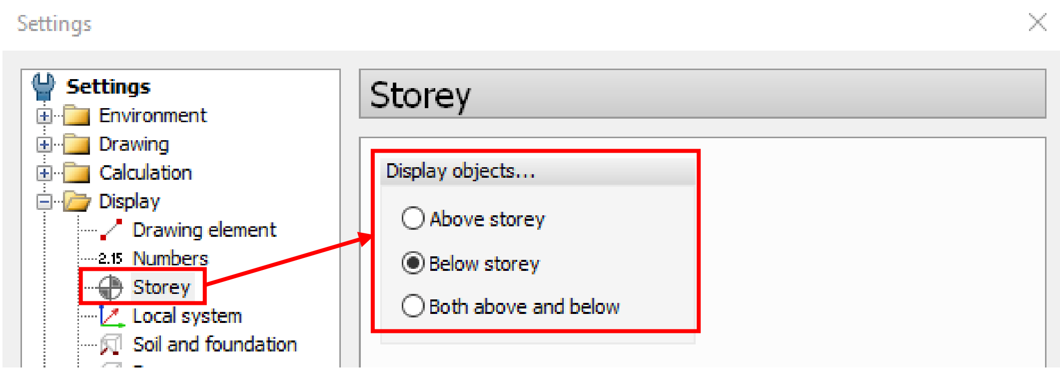


Figure: Navigation by Storey

In *Settings/Display/Storey* dialog you can choose from three options for the appearance of the given storey.

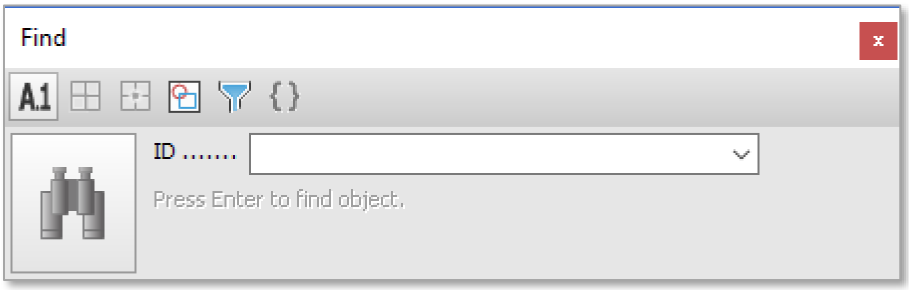


All objects are displayed which are in the plane of the storey or crossing the storey above or below according to the selected setting.

Find

"Find" can be used for finding structural and analytical elements with a given ID (A.1), some finite elements , nodes ,.blocks , user defined filters ., or global unique identifier (GUID) .

You can scroll down all the members of the current type.



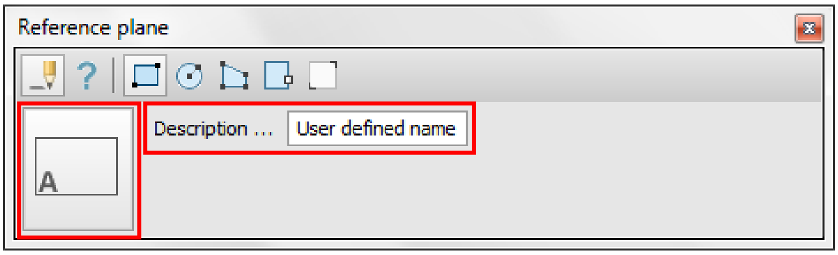
You can scroll down all the members of the current type.

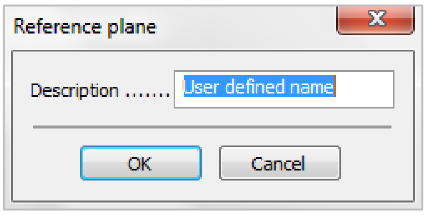
Reference plane

Reference plane is an auxiliary object, which allows User to align elements to it.

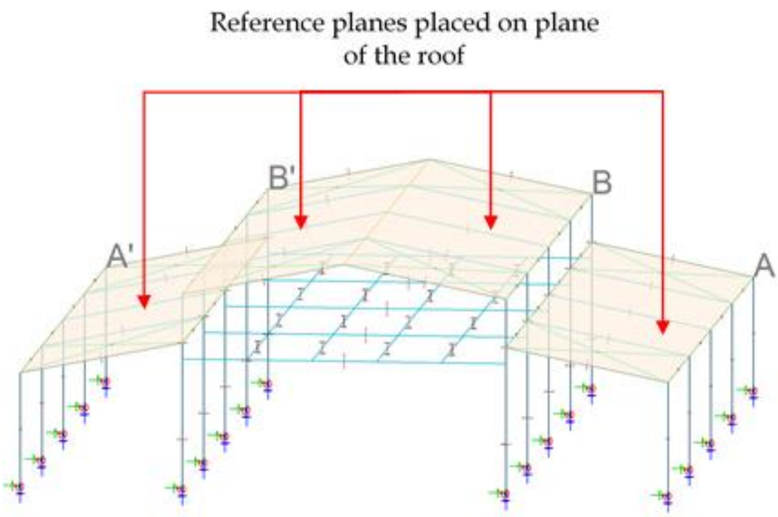
Click the *Reference plane* icon  in *Structure tab*.

You can type a *Description* for the *Reference plane* in the tool window or in the Default properties.





Reference plane can be drawn in any plane. It is displayed by the region contours with the predefined description.



The *Reference plane* can be used for align regions to it and also can be used by *Correct model* tool to fit structural objects to.

Reference planes can be exported into/imported from Autodesk Revit via struxml format.

Structure Definition

A model consists of structural objects, **loads** and **finite elements**. This chapter summarizes the definition ways, the properties and the features of the structural objects.

Depending on the current **FEM-Design Module** (license you have), the available Object types are different. Although the structural objects are real 3 dimensional objects, they are 1D members and 2D planar elements (because of the finite element method) having sectional (thickness, profile etc.) and material properties. So, they can be defined as lines or regions. Some elements like point supports can be defined as points.

Type	Modules where available	Definition mode	Position	Material	Profile	Load-bearing
Soil		Region	Horizontal	Arbitrary	-	Arbitrary
Borehole		Point	-	-	-	-
Isolated foundation		Regular shape/Solid	Horizontal	Concrete	Arbitrary	Arbitrary
Wall foundation		Line	Horizontal	Concrete	Arbitrary	Arbitrary
Foundation slab		Region	Horizontal	Concrete	Arbitrary	Arbitrary
Beam		Line	Horizontal Arbitrary	Arbitrary	Arbitrary	Arbitrary
Column		Point Point/Line	Vertical	Arbitrary	Arbitrary	(Support) Arbitrary
Truss member		Line	Arbitrary	Arbitrary	Arbitrary	Axial
Intermediate section		Point	-	Arbitrary	Arbitrary	-
Apex		Line	Vertical/ Horizontal	Timber	Rectangle	Arbitrary
Column corbel		Point	Horizontal	Arbitrary	Arbitrary	Arbitrary

Wall corbel		Line	Horizontal	Arbitrary	Arbitrary	Arbitrary
Plate		Region	Horizontal Arbitrary	Arbitrary	Constant/ variable thickness	Vertical Axial

Wall		Line Region Line	Vertical	Arbitrary	Constant/ variable thickness	(Support) Planar Arbitrary
Profiled panel		Line/Region	Vertical/Horizontal	Concrete	Constant	Arbitrary
Timber panel		Line/Region	Vertical/Horizontal	Timber	Constant	Arbitrary
Point support		Point	Vertical Arbitrary	-	-	-
Line support		Line	Vertical Arbitrary	-	-	-
Surface support (group)		Region	Vertical Arbitrary	-	-	-
Point-point connection		Line	Horizontal Arbitrary	-	-	-
Line-line connection		Lines	Horizontal Arbitrary	-	-	-
Fictitious bar		Line	Horizontal Arbitrary	-	-	-
Shell model		Regions	Arbitrary	Steel	Constant/ variable web height	Arbitrary
Fictitious shell		Regions	Arbitrary	-	-	-

Table: Structural Objects and their main properties

The commands for defining structural objects can be started from the Structure **Tabmenu**. Each command has a **Tool palette** with the customizable element properties (cross-sections, materials, stiffness values etc.) and the definition tools of the element geometry and position (direction).

Properties

Tool palette contains all customizable structural properties. The main properties can be set directly in the tool palette and all properties can be set in the dialog opens by clicking on the *Default settings* button. The settings dialog and fields keep the last set property values by element types (beams, columns, plates etc.)

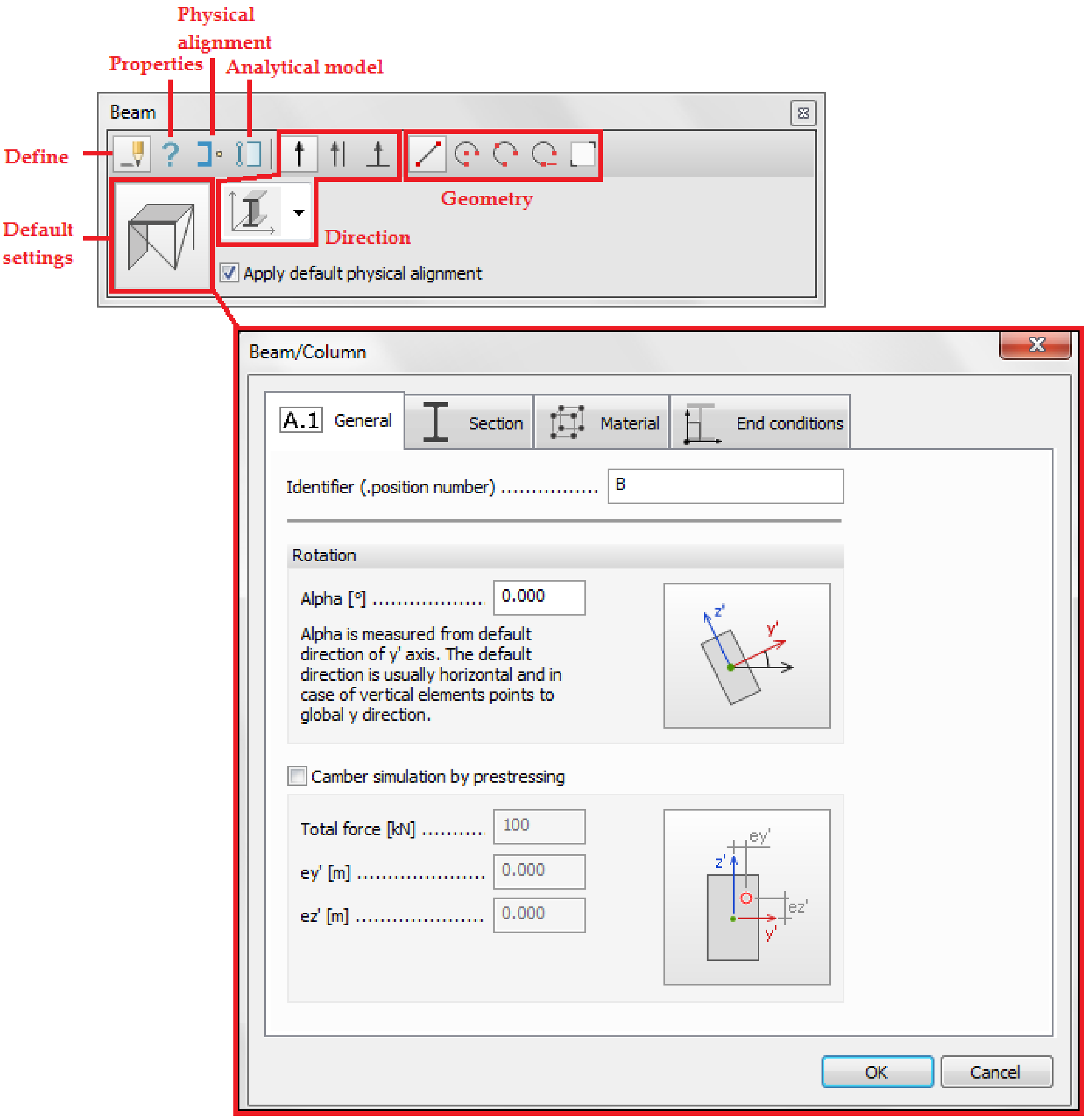


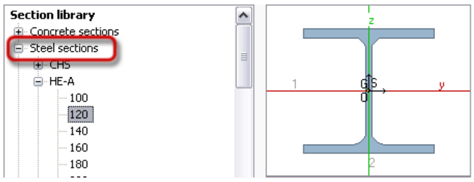
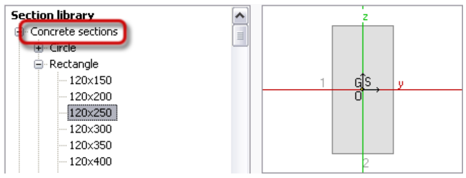
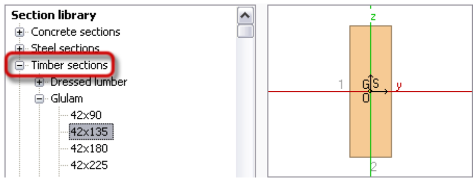
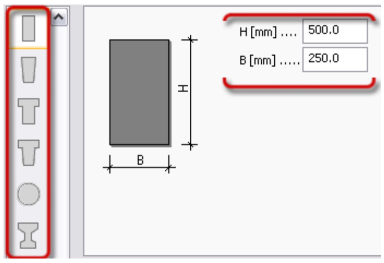
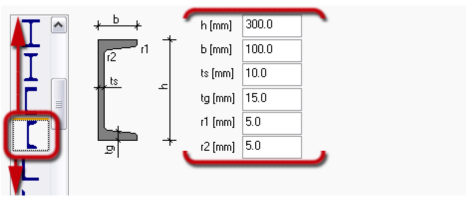
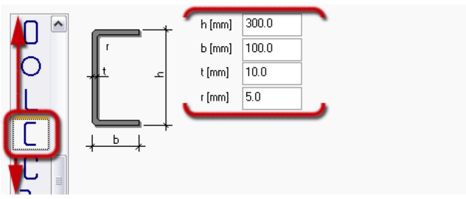
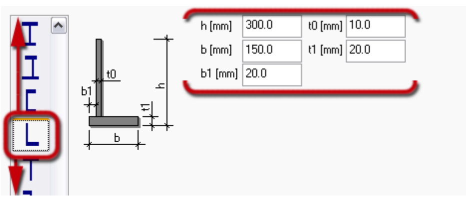
Figure: Setting part of Tool palette

Cross-sections

FEM-Design offers the possibility to add any cross-section type/shape (*Section*) to beams, columns and truss members.

⚠ To avoid design failures of concrete, steel and timber bars (section type and material do not fit), the program checks the section type - **material** compatibility while setting the properties. The program sends an error message when it finds incompatibility definition at closing the settings dialog. But, the so-called **General material** (defined by the user) can be added to each section types.

The *Section* tabpage of the settings dialog contains predefined cross-sections. Unique profiles and shapes can be define by their parameters (parametric shapes) in *New> Size* or with the **FEM-Design Section Editor**. The following table sums the available cross-section types.

Type	Description	Section library	Compatible material
Standard steel profiles	Not editable, built-in steel profiles depending on the applied national code		Steel (or General)
Often used concrete profiles	Not editable, built-in square, rectangular and circular concrete profiles		Concrete (or General)
Often used timber profiles	Not editable, built-in square timber profiles		Timber (or General)
Common parametric concrete shapes	Predefined concrete shapes to create required profiles by defining the shape parameters		Concrete (or General)
Common parametric rolled steel shapes	Predefined rolled steel shapes to create required profiles by defining the shape parameters		Steel (or General)
Common parametric cold-formed steel shapes	Predefined cold-formed steel shapes to create required profiles by defining the shape parameters		Steel (or General)
Common parametric welded steel shapes	Predefined welded steel shapes to create required profiles by defining the shape parameters		Steel (or General)

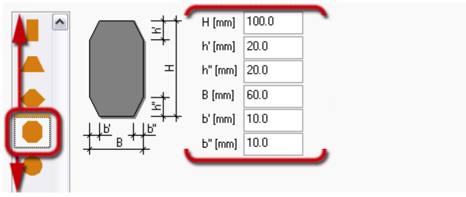
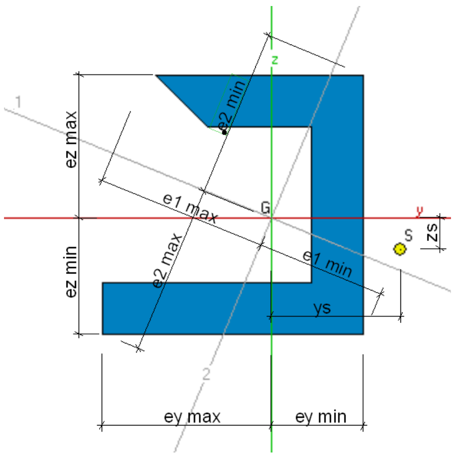
Common parametric timber shapes	Predefined timber shapes to create required profiles by defining the shape parameters		Timber (or General)

Table: Section types

Defining a new (e.g. parametric) section in the settings dialog, the program calculates automatically the parameters need for later analysis and design. Each bar element has a local coordinate-system which y and z axes define the plane of its cross-section.

Parameter	Meaning	Example
A	Area	
P	Perimeter	
A/P	Area/Perimeter	
Yg, Zg	Center of gravity	
Ys, Zs	Shear center position	
Iy, Iz	Moment of inertia	
Wy, Wz	Section modulus	
ez max, ey max	Maximum distance of extreme fiber	
ez min, ey min	Minimum distance of extreme fiber	
iy, iz	Radius of inertia	
Sy, Sz	Maximum statical moment	
It	Torsion moment of inertia	
Wt	Torsion section modulus	
Igamma	Warping parameter	
Iyz	Centroidal product of inertia	
z omega	Wagner warping parameter	
alpha1, alpha2	Angle of principal direction	
I1, I2	Principal moment of inertia	
W1 min, W2 min	Principal minimum section modulus	
W1 max, W2 max	Principal maximum section modulus	
e2 max, e1 max	Maximum distance of extreme fiber	
e2 min, e1 min	Minimum distance of extreme fiber	

i1, i2	Radius of principal inertia	
S1, S2	Principal maximum statical moment	
So1, So2	Principal statical moment of half area	
c1, c2	Plastic/elastic moment capacity	
Rho 1, Rho 2	Principal shear factor	
z2, z1	Wagner parameter	

Table: Sectional characteristics

A cross-section is stored in the following tree structure: *group* > *type* > *size*. New library items can be defined or previous ones can be edited (renamed, modified or deleted) by the following settings options.

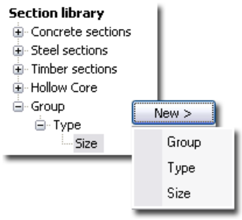


Figure: Section library structure

Cross-sections used in the project and defined as parametric profiles are grouped in *Used sections* library, but they are available only in the current project.

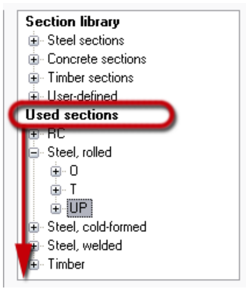


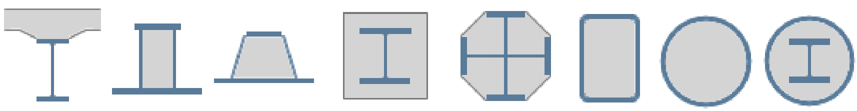
Figure: "Used sections" library

You can Import and Export the Section Library to share the content of the Library.

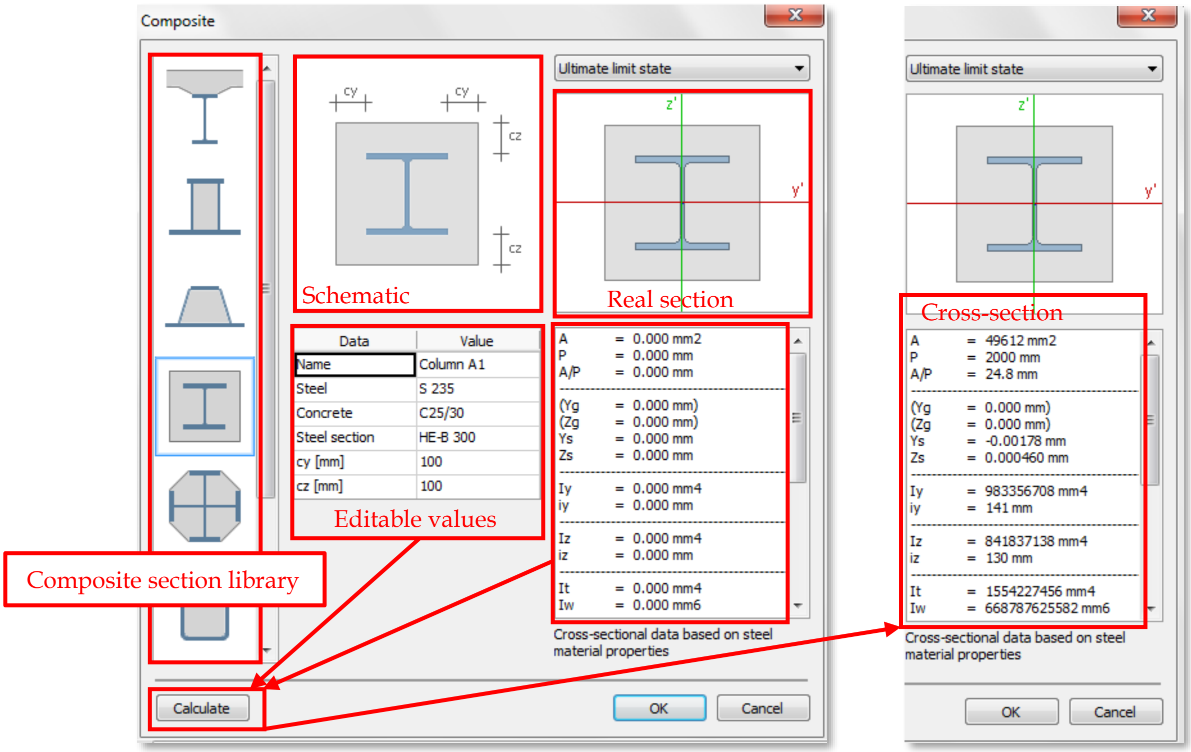
Composite sections

Composite sections can be set for beams, columns and piles.

The available composite section types:



Click *Default settings/Section tab/Composite/New* to open the settings of *Composite sections*.



For the time being only above listed composite sections can be selected, there is no choice to create alternative section.

Materials

The *Material* tabpage of the settings dialog contains predefined materials according to the current national standard.

A model may include mixture of elements with different materials, and analysis can be done for the complete model, but design can be done for elements having proper materials.

Material	Design
Concrete	**RC design**
Steel	**Steel design**
Timber	**Timber design**
General	- (only for analysis)
Bar steel	**RC design**

Table: Available and design materials

Material library stores the available materials by material type groups. Materials used in the project are grouped in *Used materials* library, but they are available only in the current project.

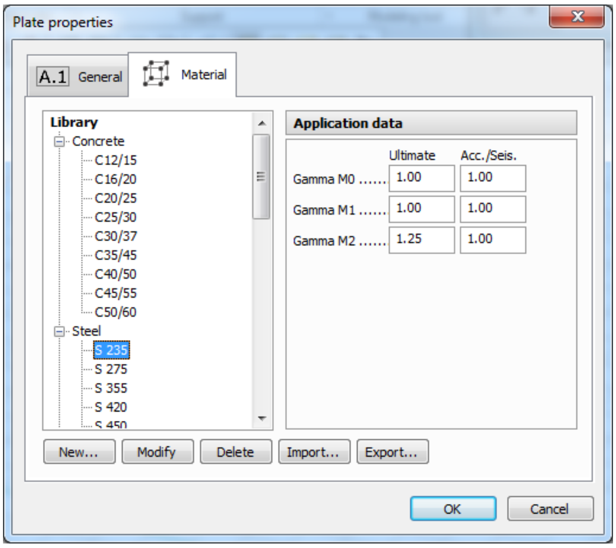


Table: Material library

Above the standard materials, user can define concrete, steel, timber and general materials. The new materials are also stored in the *Material library* in the proper material type group. To define a new (e.g. concrete) material, select the required material group name (e.g. *Concrete*), click *New* and set the required material properties starting with the material name first.

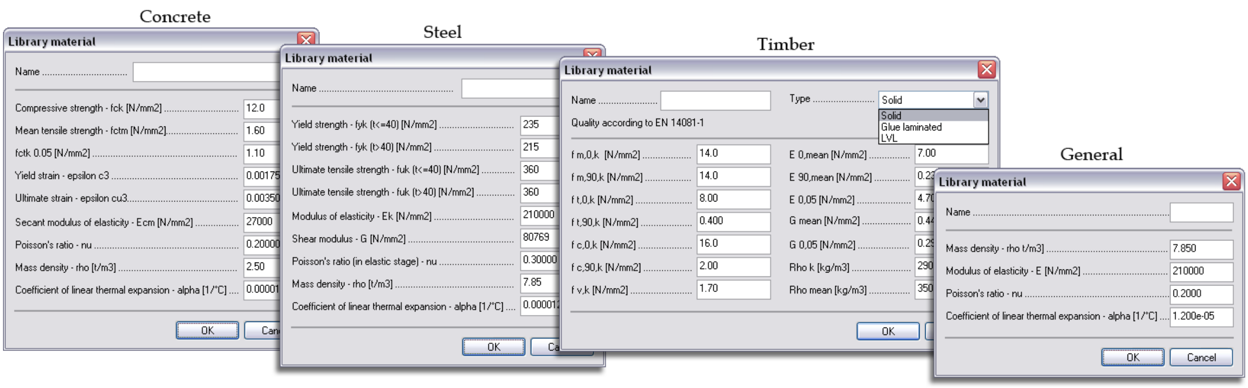


Table: Material types

Concrete

For concrete structures, the partial safety factors γ_c and γ_S can be calculated automatically from Eurocode 2 reductions. Click *Safety factor calculator* next to the safety factors (*Default settings* > *Material* > *Application data*), check the required reduction box and press *OK*.

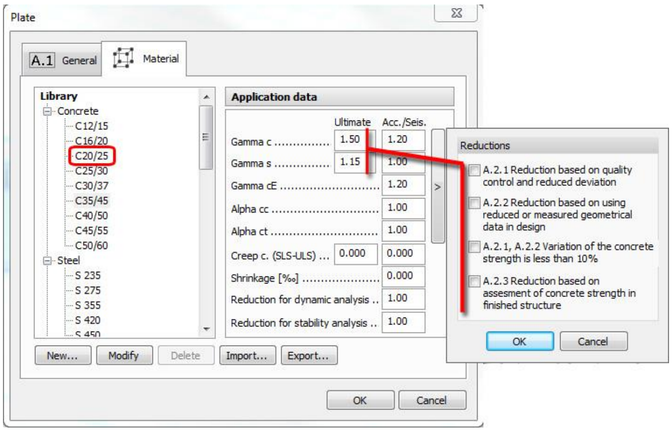


Figure: Safety factor calculator

In Material settings dialog different creep coefficients can be specified for Serviceability Limit State (SLS) and Ultimate Limit State (ULS). It has some consequences in load case calculation and results:

- All load cases are calculated twice (first with the SLS Creep coefficient, than with the ULS Creep coefficient.
- The displayed displacements are the results of the SLS calculation.
- The displayed internal forces, reactions are the results of the ULS calculation.

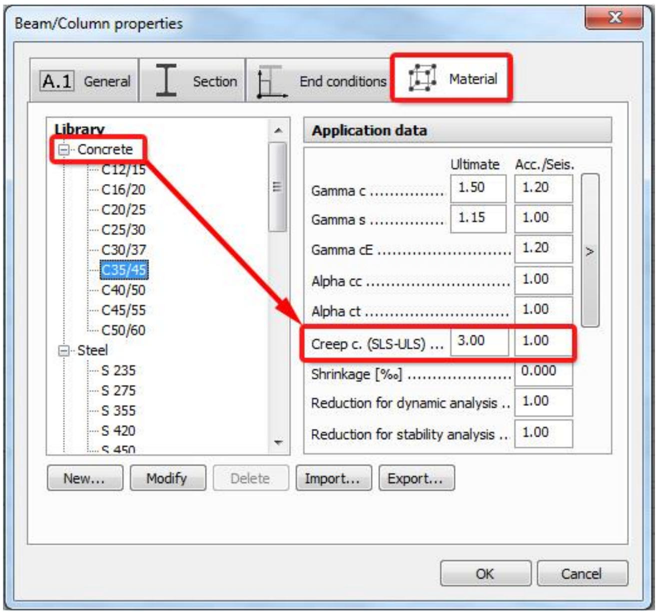


Figure: Setting of creep coefficients

For reinforced concrete structure the user has the possibility to reduce the element stiffness in order to model the cracking's effect in eigenfrequency calculation.

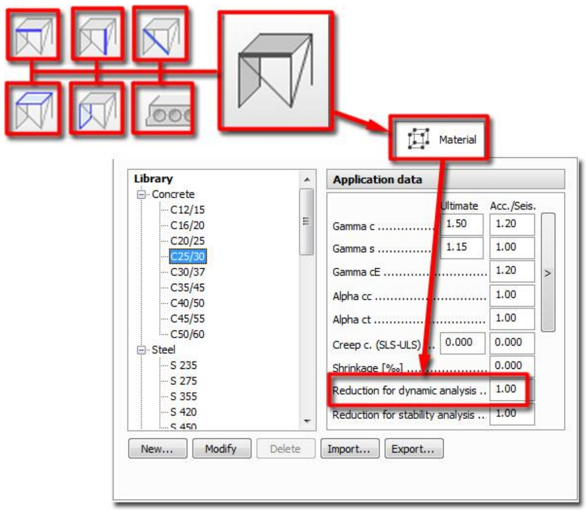


Figure: Stiffness reduction for reinforce concrete elements

In Material settings dialog the stiffness for stability analysis can be decreased. Taking a reduction factor into account is needed in those calculations where it is specified by the standards (e.g. at the cracked section analysis).

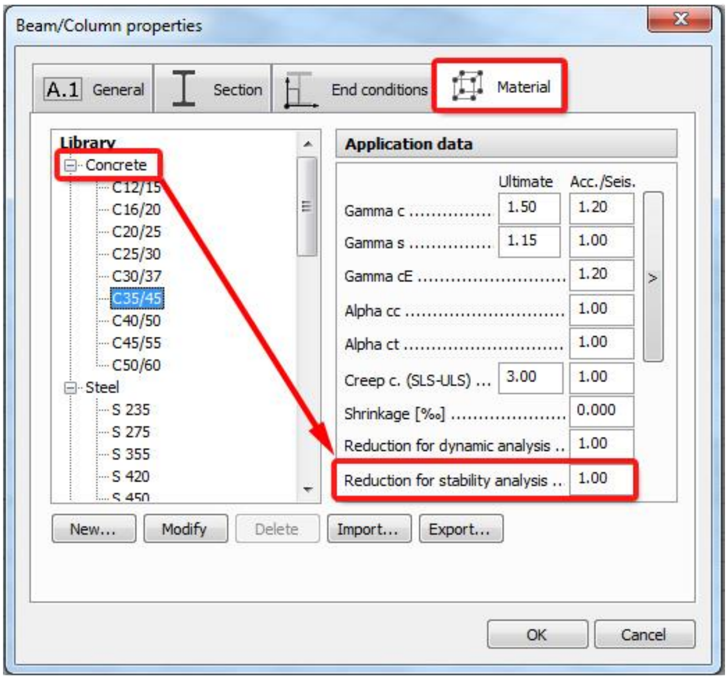


Figure: Reduction for stability analysis

Steel

For steel structures, the γ values (M0, M1 and M2) can be set.

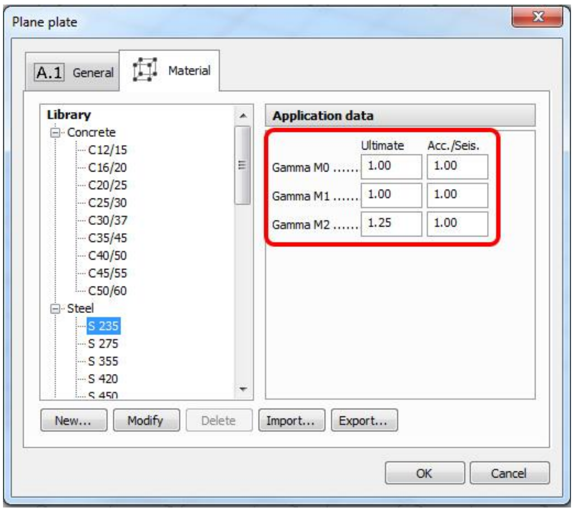


Figure: Setting γ factors

Timber

For timber structures, the γ_M, Service class, the System strength factor and k_{cr} values can be set.

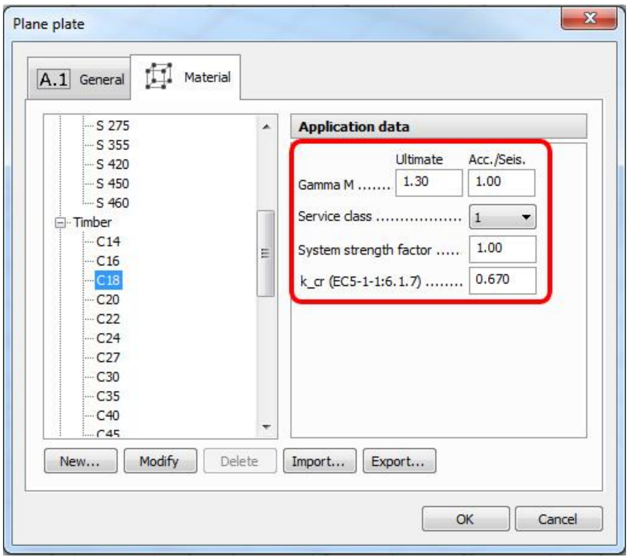


Figure: Timber material settings

The following table and figures summarize the calculation of Young moduli used in different analysis types for concrete, steel and timber materials.

Analysis type		**Concrete**	**Steel**	**Timber**
Load cases, Load group, 1st order load combination	ULS	$\frac{E_{cm}}{(1+\varphi_u)}$	E_k	$\frac{E_{o,mean}}{(1+k_{def})}$
	SLS	$\frac{E_{cm}}{(1+\varphi_s)}$		

2nd order load combination, Imperfection calculation	ULS	$\frac{E_{cm}}{(1+\varphi_u)} \cdot \frac{1}{\gamma_{cE}}$		$\frac{E_{o,mean}}{\gamma_M}$
	SLS	$\frac{E_{cm}}{(1+\varphi_s)} \cdot \frac{1}{\gamma_{cE}}$		
Stability analysis		$f_{stab} \cdot E_{cm}$		$\frac{E_{o,mean}}{(1+k_{def})}$
Eigenfrequency calculation Seismic analysis		$f_{din} \cdot E_{cm}$		$E_{o,mean}$

Table: Calculation of Young-moduli in different analysis types

The properties of the current (selected in the list) material can be edited with the *Modify* tool.

Customized material database can be shared between projects and users with the *Export* and *Import* tools. Click *Export* to save all materials of the current project in a named database file (*.fdlmat*). To load an exported material database to a project, just apply *Import* and browse for it.

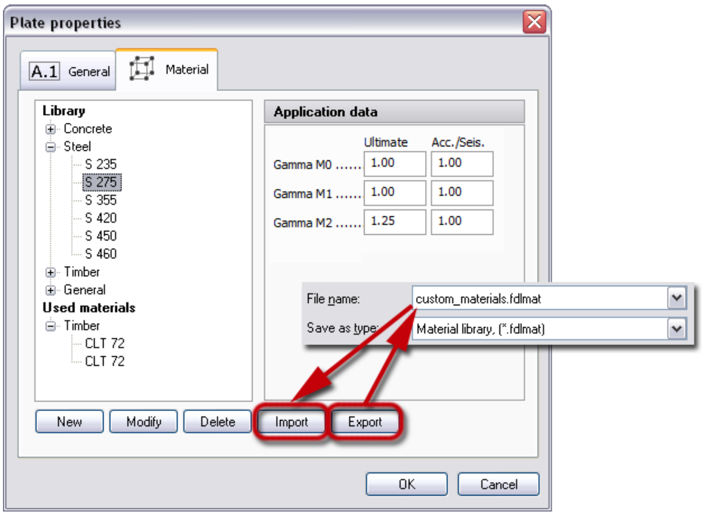


Figure: Material export/import

Just one click on **Quantity estimation** (*Tools* menu), and a fast process collects all structural elements of the current project with their applied materials, material qualities, identifiers, main geometric parameters (e.g. profiles), quantities etc.

Modifying the national standard for a model the program shows a dialog where you can convert the material property of the structural elements and the reinforcing steel, so the materials will automatically change after switching the code.

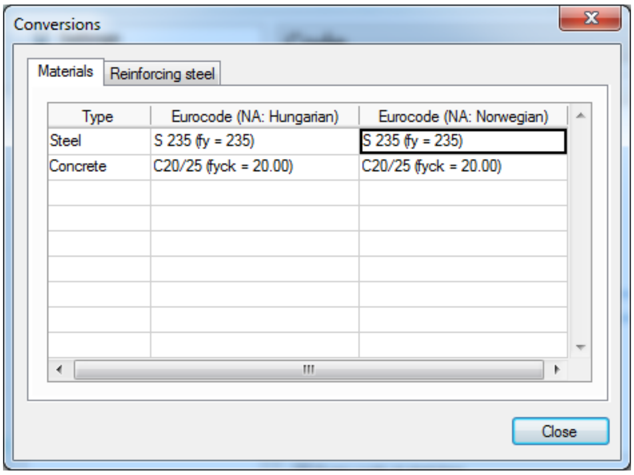


Figure: Material conversion dialog

Information Pop-up

Moving the cursor over a structural object, an ***Information pop-up*** appears with its main properties. For example, the pop-up displays the ID, the material, the thickness, the alignment and orthotropic features for *Plates*.

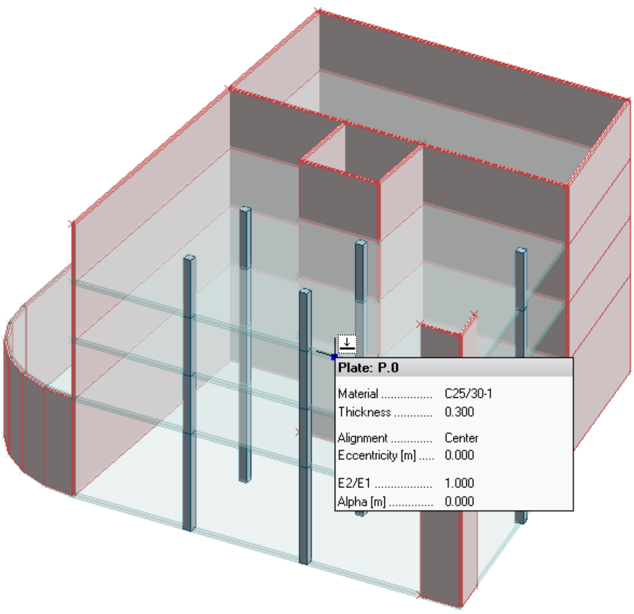


Figure: Information pop-up example

Information pop-up appears only for elements on visible layers and if there is not running command.

Pop-up is enabled by default. To unable it, uncheck the *Display information pop-up* at *Settings* > *All* > *Environment* > *General* > *View*.

"Properties" Tool

With the *Properties* tool of a tool palette, the properties of a selected object or objects can be inquired and edited in dialog format (similar to *Default settings*).

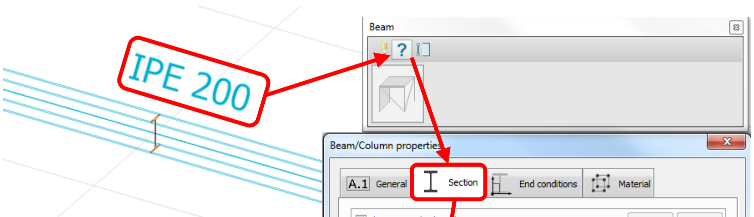
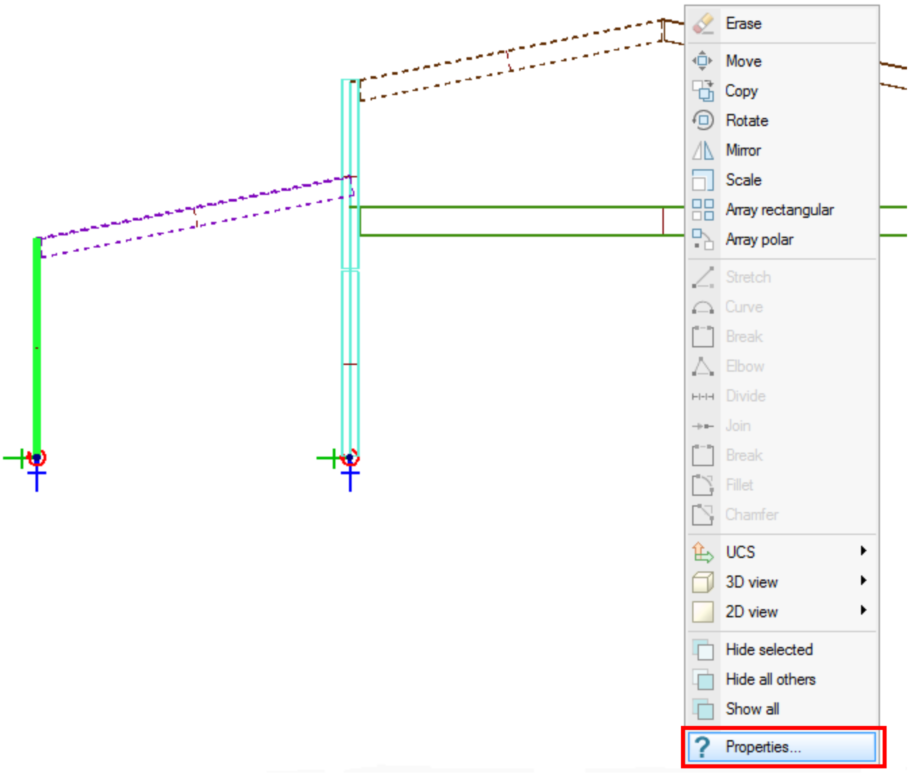
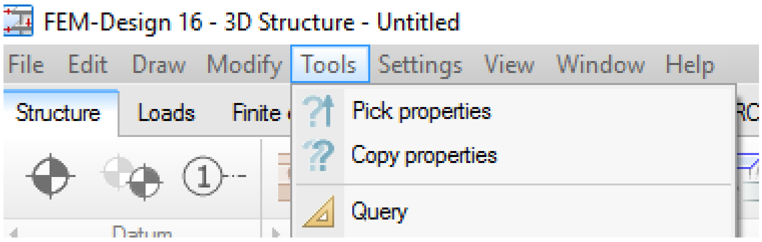


Figure: Modifying object properties (e.g. cross-section of a selected beam)

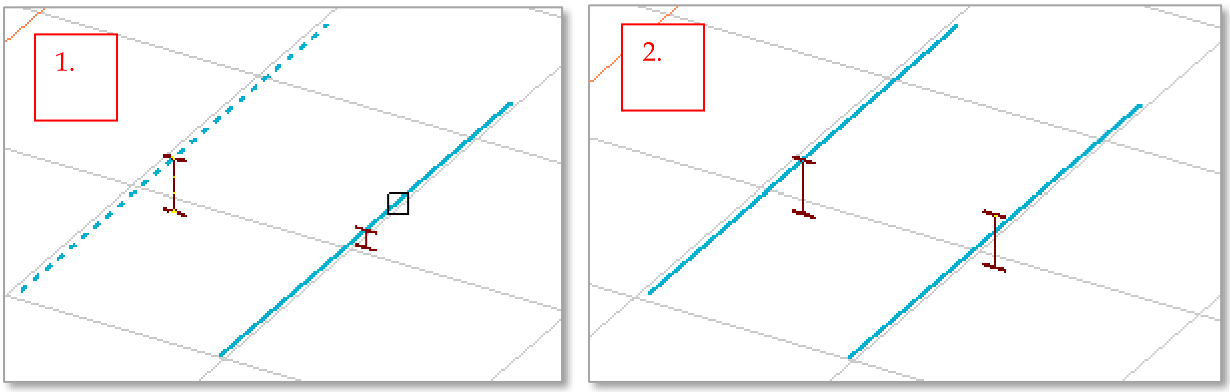
Modify properties by "Quick menu"
Alternative way of modifying object properties is to select any object by right click, or more objects of the same type by box, then click "*Properties*" in the Quick menu to check/modify its/their properties. This function works for structural objects, loads or design elements.



"Pick properties" and "Copy properties"
With these functions the properties of an object can be copied to others of the same type, or picked to use as default. Both can be found in the *Tools* menu and in the toolbar as well.

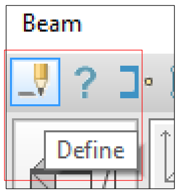


To copy properties with the *Copy properties* ( ) function, first select the source object, then one or more objects to which the properties are to be applied.



Pick properties ( ), when used on an object (of the same type as in the active editing dialogue) will update the default settings for its type.

⚠ *Pick properties* can only be used when *Define* is selected in an editing window, otherwise it is disabled.

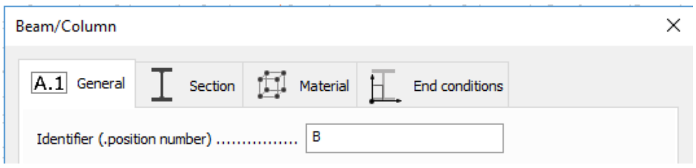


Copy and *Pick properties* work according to the following rules:

- Generally, the properties defined by the property dialogue will be picked or copied.
- They are only available in the *3D Structure* and *Plate* modules.
- ID will not be copied.
- Load case of loads will not be copied.
- Varying shell thickness and surface load value will not be copied.

Numbering

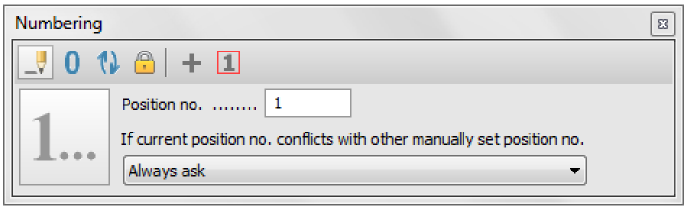
Structural elements with an 'Identifier (. position number)' will have an automatically or manually assigned position number.



The position number will be *0* upon creating the object and will be automatically set to another value in the following cases:

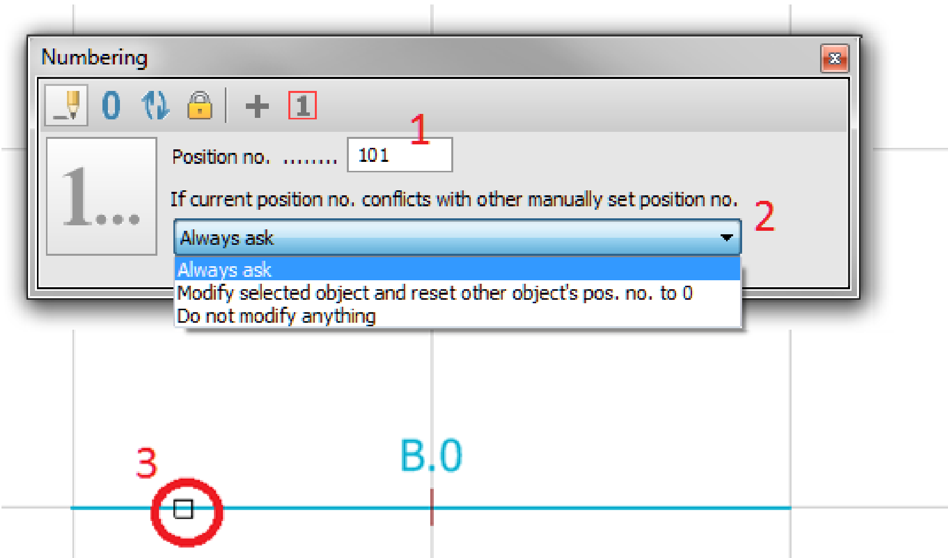
- before creating a list
- using the *Find* tool
- before calculation

User can manually set position numbers by *Tools/Numbering…*  tool.



Use  for *Manual position numbering*

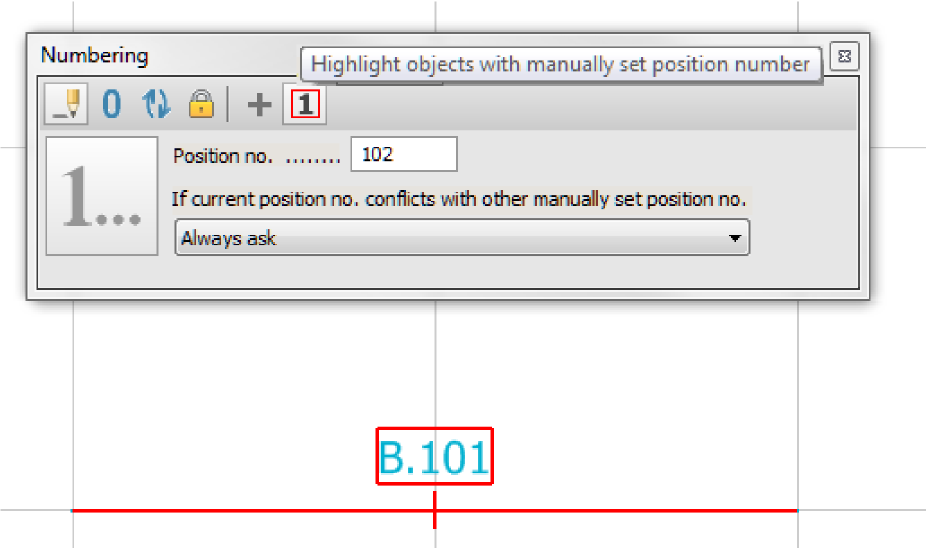
1. type required position number into the *Position no.* textbox
2. select option for handling position number conflicts
3. select object(s) to set position number for



 In case more objects are selected, the first one gets the position number typed by the user and for the next ones it is increased automatically.

To set position number of **component objects**, like edge connections, corbels, post-tensioned cables, punching regions, the *Select component (…)* auxiliary option has to be checked.

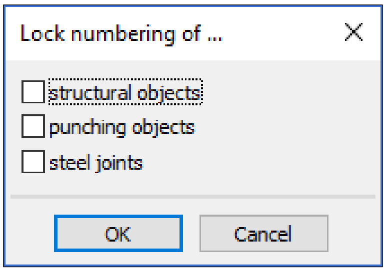
Objects with manually set position number can be **highlighted** by checking the last option of the tool window.



The position number can be reset to zero by choosing *Reset*  option then selecting one or more objects.

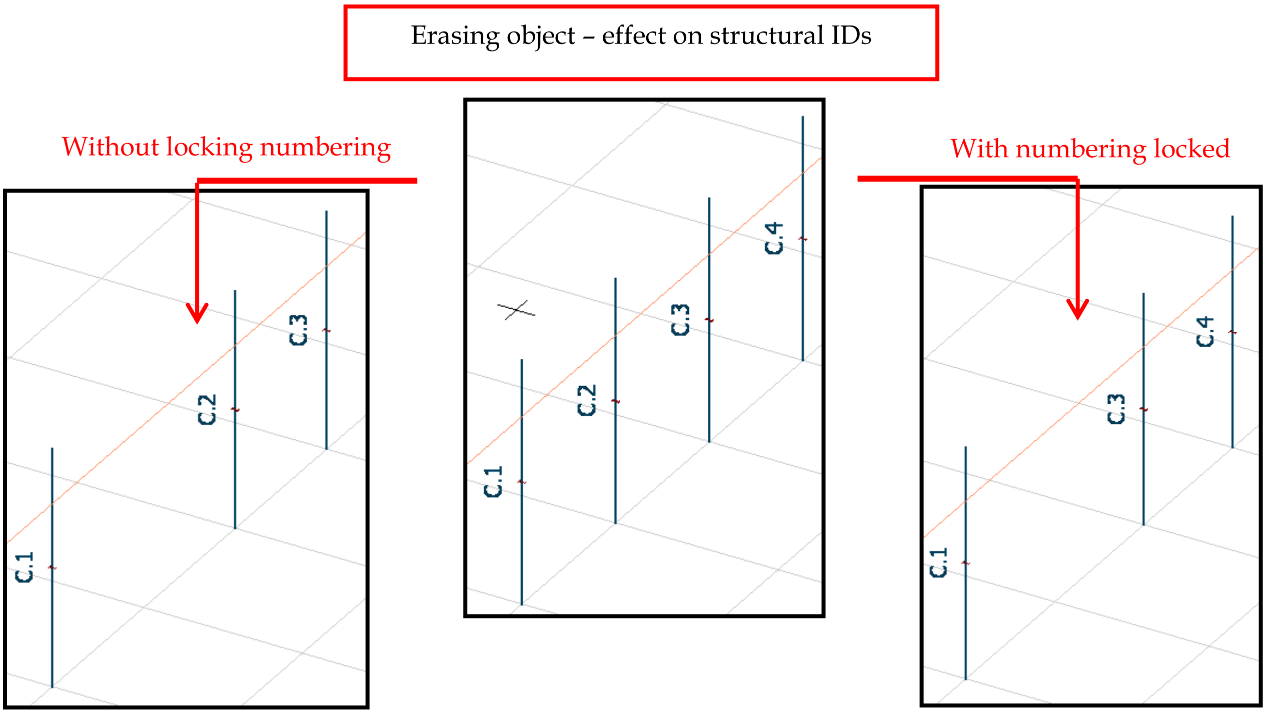
Automatic numbering  sets position number automatically for all objects in the database except the ones with manually set position number.

The option to *Lock numbering* can be accessed by clicking its icon  in the same *Tool* window.



When the numbering of a selected object type is locked, the position number of individual elements will remain the same. The maximum position number is saved for all IDs, so position numbers less than that will not be used in the future, even if some items are deleted.

This means all the element types with their numbering locked will retain their original ID, even after refreshing the numbering (either manually, or automatically).



Display Settings of Structural Elements
The display properties of the structural elements can be set at the *Settings* > *All…* > *Display*.

The available options depend on the current FEM-Design module.

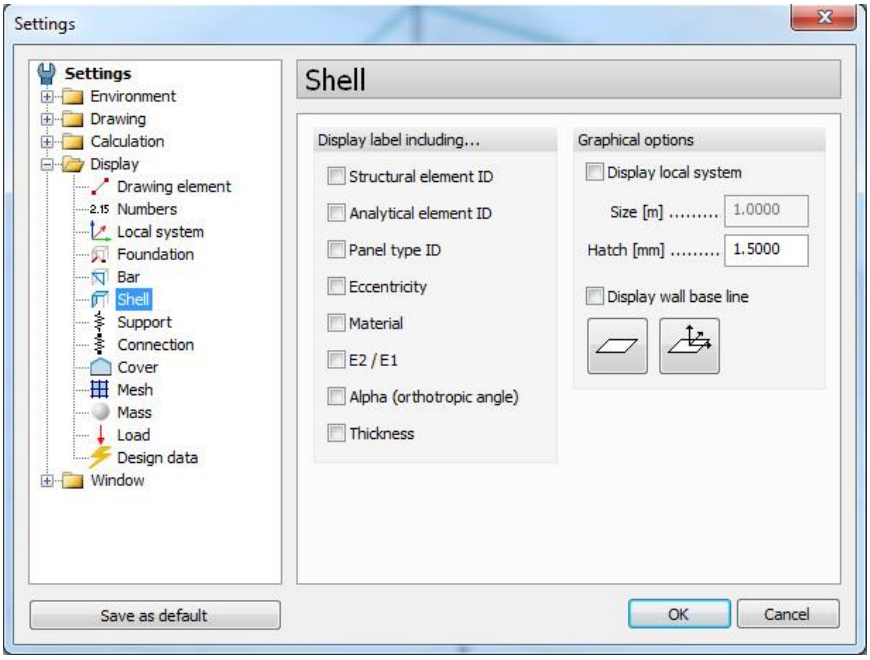


Figure: Settings options affect on the appearance of the structural elements

- **Drawing elements**
  With the *Show end points of line* option you can show (or hide) the insertion points of planar structural objects, their holes and bar objects (beams, columns etc.). These points are visible in

all **display modes**, but the **Wireframe** mode without **displaying the elements' thickness** gives the clearest appearance of them.

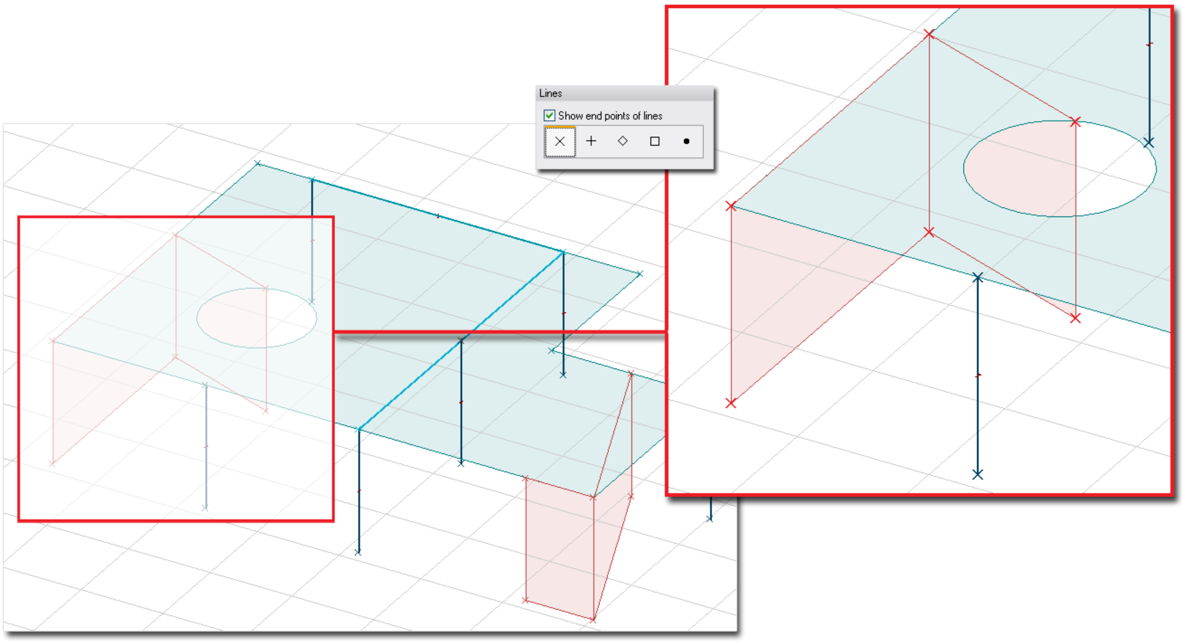


Figure: Insertion points displayed in Wireframe mode

By showing the insertion points you can also detect the unnecessary (for example accidentally) defined points, which may cause "too refined" finite element mesh in planar structural elements. The unnecessary points can be deleted with editing tools (*Edit* menu) which can be applied for region elements (such as plate, wall regions etc.). For example, stretch the unnecessary points outside the host region, and then cut the stretched region part(s) with the Split tool of the **Modify region** (*Edit* menu) or with the **Hole tool** of the structural planar object.

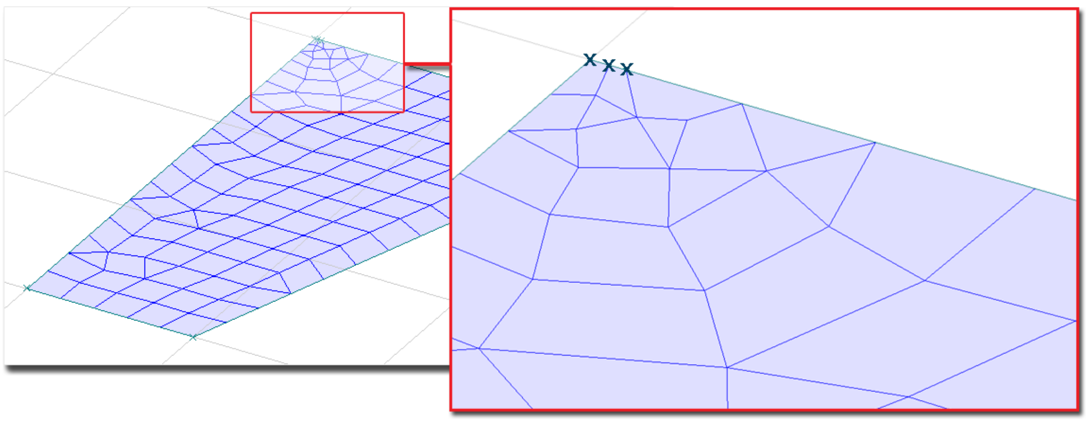


Figure: Unnecessary definition points detected visually

With the *Fill-up with color* option the planar objects (plate, wall, drawing regions) can be displayed with their reference plane as colored surface or with their contour lines only. Applying the fill colors is useful to display holes/openings clearly. The filling color of an element is the default color of the **Object layer** includes the element. It can modified at the layer settings by element types (Plate, Wall etc.) or with the *Color* option of the *Change properties* tool (*Edit* > *Properties*) by elements (independently from their types).

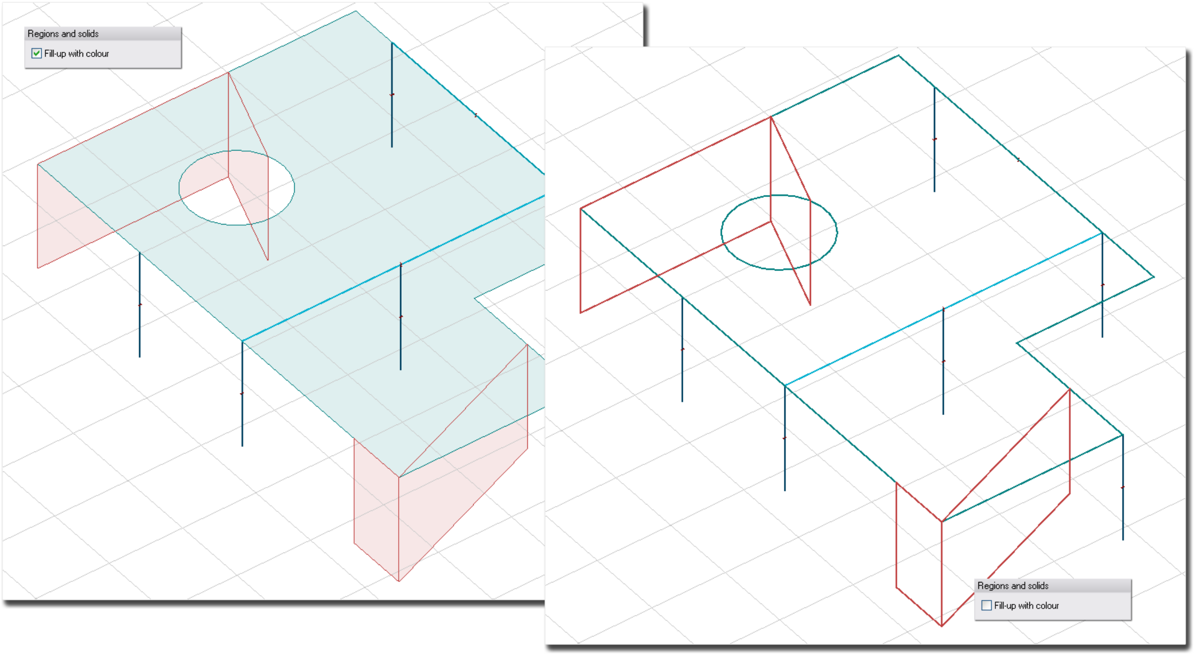


Figure: Planar objects displayed with their colored surface or without it in Wireframe mode

- **Display labels**
  Information of structural elements (such as ID, position number, material properties, section names etc.) can be displayed on the screen by element types in **Wireframe display mode**.

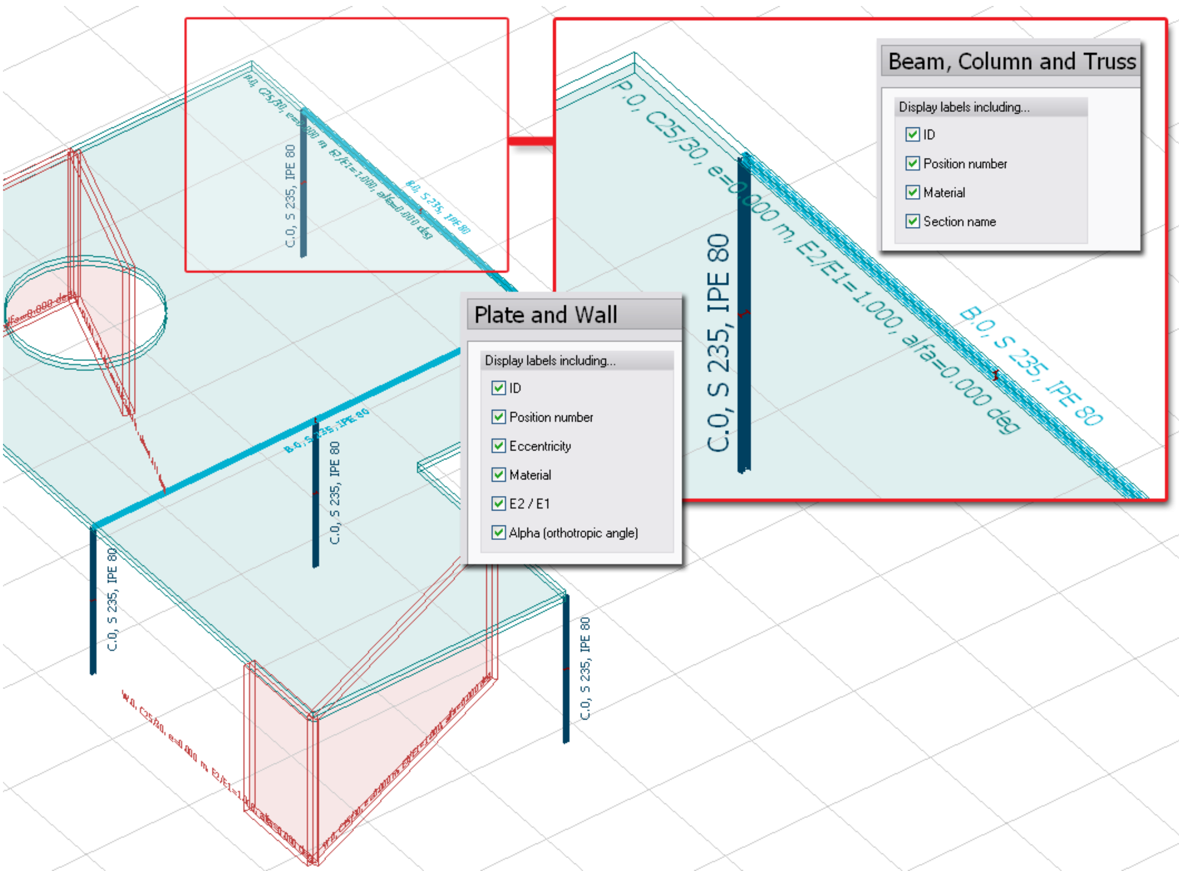


Figure: Info labels displayed on screen by element types

- **Numbers**
  This setting affects on the numeric values of the displayed labels (see the previous option).

- **Display local system**
  The **local co-ordinate system** of the structural elements can be displayed with the *Display local system* option by element types. The size of the local system symbol can be set at the *Size* option. The color of the local system axes can be set at *Local systems* setting. The default colors are: green for the local x' axis, red for the local y' axis and blue for the local z' axis.

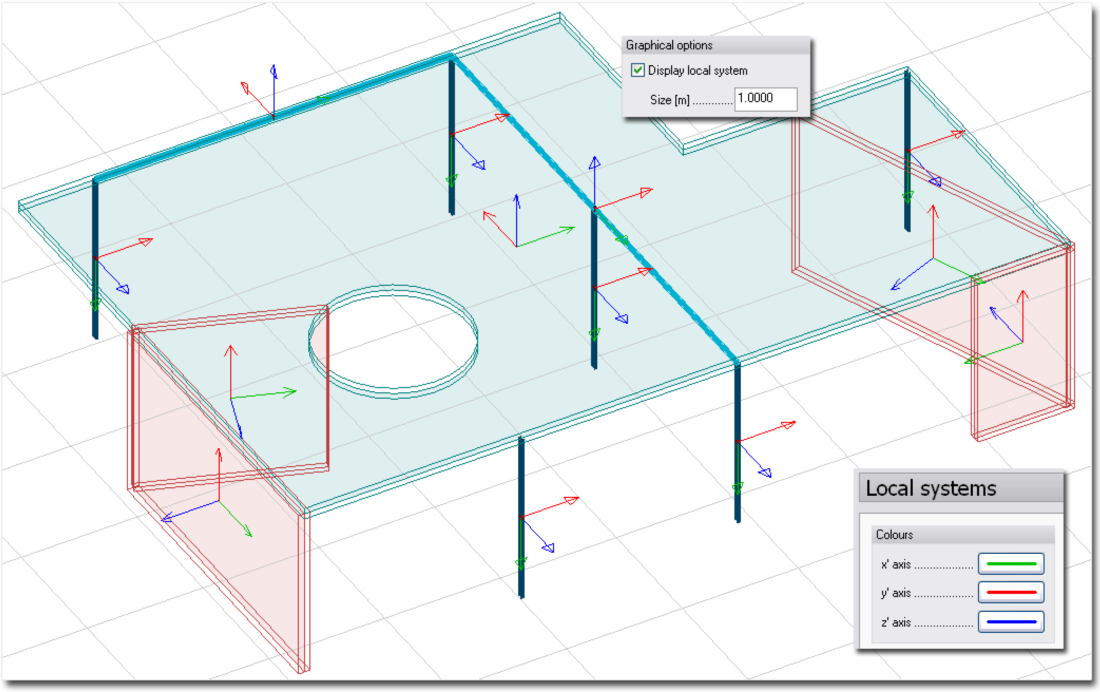


Figure: Local co-ordinate system displayed on screen by element types

- **Special display settings of walls**
  With the *Display wall base line* option (*3D Structure* and *Predesign* modules only) the bottom end of the walls can be displayed as a hatched surface.

  By inactivating the *Display wall height* option (*Plate* module only) only the reference support line (as hatched surface) can be displayed without showing the height-extension that is out from the plates' calculation 2D plane.

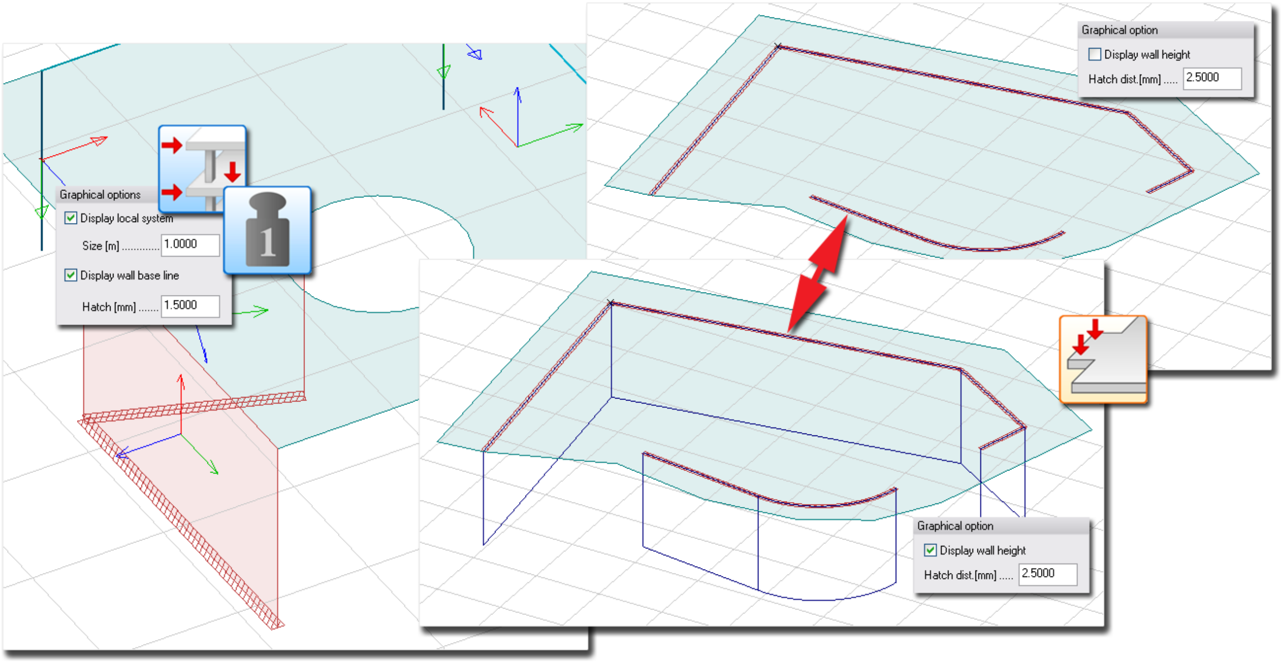


Figure: Special display options of walls

- **Special display settings of bar elements**

  The *Display section shape* option shows the applied section of the bar elements as a colored symbol in the middle of the bar reference lines. The size (*Scale*), the filling (*Fill*) and contour (*Border*) colors are customizable. Although, section shape symbols are visible in most of the different display modes, their "best" display mode is the **Wireframe** mode without **displaying the elements' thickness**.

  The *Display connections* option shows the end connection property (see **Beam**, **Column** or **Fictitious bar connection** settings) of the bar elements. Only the free end motion components can be displayed as arrows, where a simple arrow shows a motion component by its direction and a double-headed arrow shows the axis direction of a rotation component. The fix (rigid) end connection components are never displayed. The color of an end motion component equals with the color of the proper axis of the host bar element's local system (see before, **Display local system**). The size of the symbols can be set at the *Size* option.

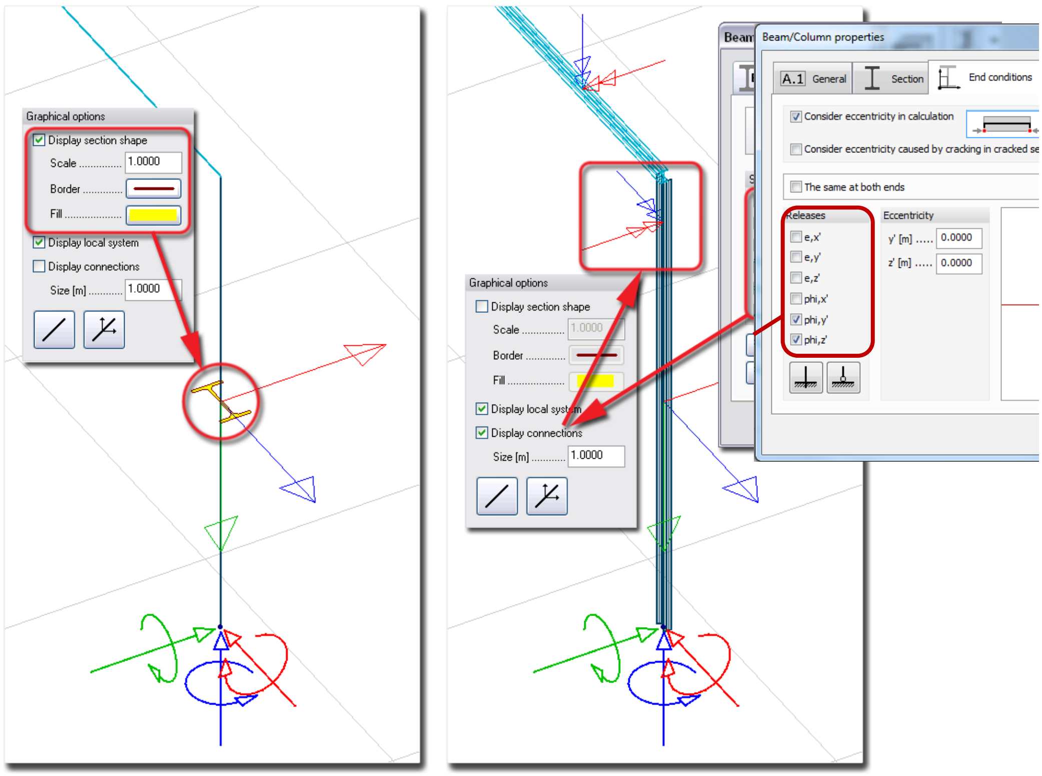


Figure: Special display options of bar elements

- **Layer, color and pen width**

  All elements are placed (and grouped) on **Object layers** according to their type. So, for example, columns are on the "Columns" layer and the walls are on the "Walls" layer. The default color and pen width of elements' contours/reference lines are represented by their host layers. For example, by default, walls are red, if the color of the "Walls" layer is also red.

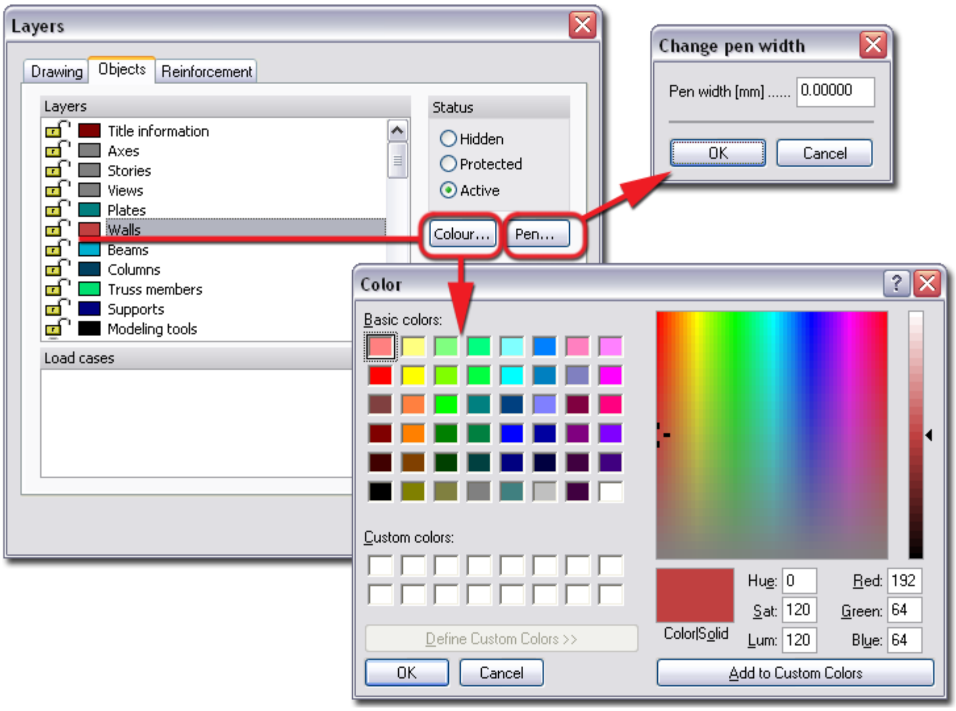
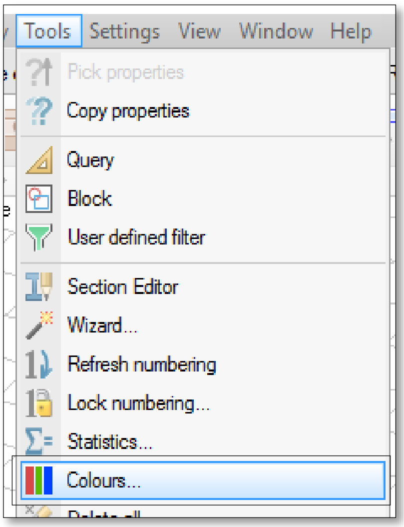


Figure: Layer-system of structural objects

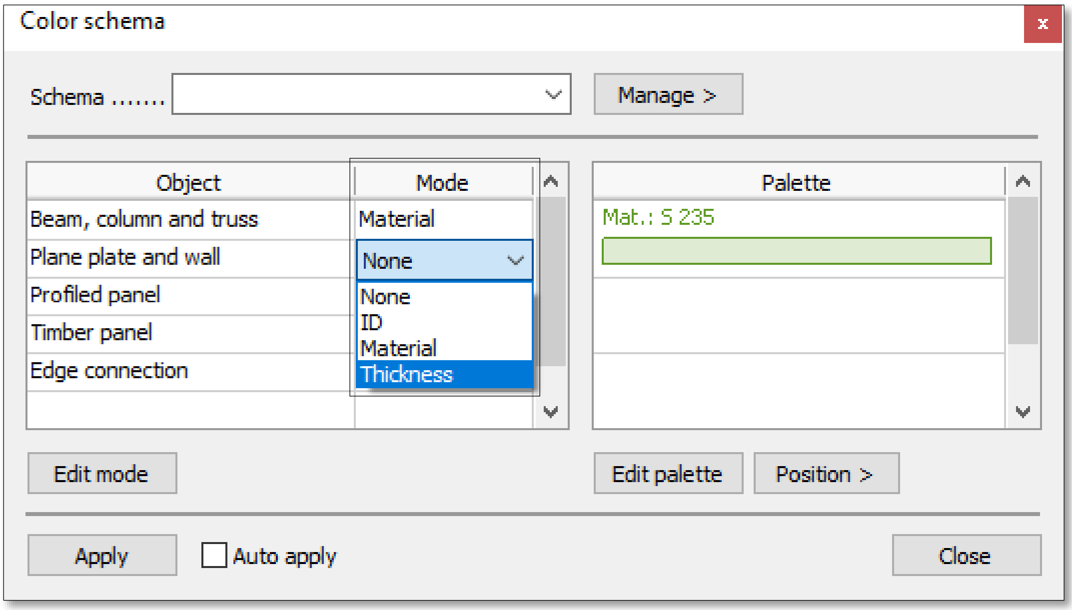
The default color of an object can be modified independently from their types with the *Color* option of the *Change properties* tool (*Edit* > *Properties*).

Color schema

This feature lets the users to navigate a model easier, by allowing them to fully customize element colors. These options can be accessed through the *Tools* → *Colors* menu command, or by clicking on their icon ( ) in the toolbar.



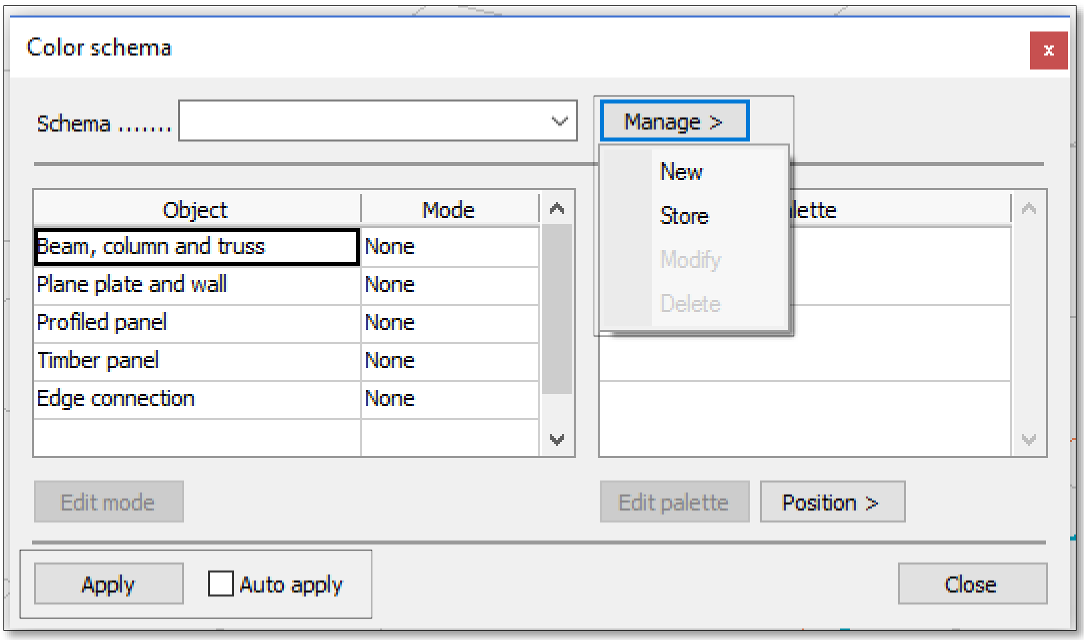
Mode determines the attribute on which the color coding is based. When *None* is selected, the default colors for object types will be used.



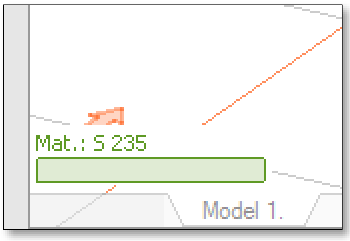
For each object type, the following attributes are available:

Beam, column and truss	ID, Material, Cross-section
Plane plate and wall	ID, Material, Thickness
Profiled plate and wall	ID, Material, Cross-section
Timber panel	ID, Material
Edge connection	ID, Rigidity type

The *Manage* button can be used to save a specified color coding system. These can be selected afterwards from the drop-down list to the left from the *Manage* button. Schema color is saved only for a given project.

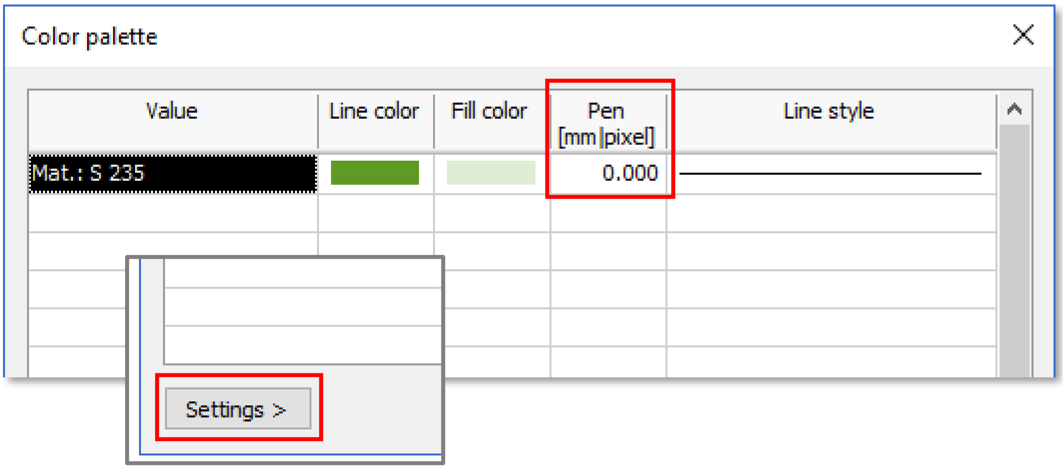


Clicking the *Apply* button will apply the color code to the model. Enabling the *Auto apply* checkbox will apply any changes automatically.



When a custom color scheme is in use, a legend window will be displayed. With the *Position* button, the placement of this window can be specified; it can even be hidden. By default, the legend is displayed in the bottom left corner of the screen.

Colors, borders, and their line types (with thickness) can be modified by clicking on the *Edit palette* button. In the dialog box, all parameters can be set for every value of the specified attribute (*ID*, *Material* or *Thickness*). It is possible to save these settings separately to be used in another color scheme.



The Settings button will allow you to save your color scheme to a file (**.cpal*) or as default, or load another schema.

When a color scheme is saved, only the colors and their order will be stored, values need to be specified each time.

Pen (line weight) can use pixels as well as millimeters. A positive number means millimeters, while a negative one will give the line width in pixels. A line weight given in millimeters will remain the same in the documentation. When pen size is given in pixels, the line weight will remain the same on-screen, regardless of zoom level.

Geometry

The definition modes and the available shapes of structural elements' reference line or reference plane depend on:

- the structural type: 1D Member or Planar Object, and
- the current FEM-Design module.

The Tool palette of a structural element contains only the available modes. The next table summarizes the geometry possibilities by structural elements.

Type	Modules where available	Definition mode	Geometry
Soil		Reference region	Rectangular Circular Polygonal Pick lines Pick existing region
Borehole		Reference point	- (Insertion point)
Isolated foundation		Regular shape	Rectangular Circular Polygonal Pick lines Pick existing region
		Reference solid	Pick existing solid
Wall foundation		Line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Foundation slab		Region	Rectangular Circular Polygonal Pick lines Pick existing region
Column		Reference point	- (Insertion point)
		Reference line	Line by insertion point Vertical line Select axes *Select point* Line by selection
		Reference line	Straight line

Truss member			Line by selection
Intermediate section		Reference point	- (Insertion point)
Apex		Reference beams	-
Column corbel		Reference point on a column	- (Insertion point)
Wall corbel		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Plate		Reference region	Rectangular Circular Polygonal Pick lines Pick existing region
Wall		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
		Reference region	Rectangular Circular Polygonal Pick lines Pick existing region
		Reference region	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Profiled plate		Reference region	Rectangular Circular

			Polygonal Pick lines Pick existing region
Profiled wall		Reference region	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Timber panel		Reference region	**Use as plate:** Rectangular Circular Polygonal Pick lines Pick existing region
		Reference region	**Use as wall:** Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Point support		Reference point	- (Insertion point)
Line support		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Surface support		Reference region	Rectangular Circular Polygonal Pick lines Pick existing region
Point-point connection		Reference points	- (Insertion points)

Line-line connection		Reference lines	**Straight line** **Arc by center, start and end points** **Arc by 3 points** **Arc by start, end point and tangent** **Line by selection**
Fictitious bar		Reference line	**Straight line** **Arc by center, start and end points** **Arc by 3 points** **Arc by start, end point and tangent** **Line by selection**
Cover Wall type		Reference line	**Straight line** **Arc by center, start and end points** **Arc by 3 points** **Arc by start, end point and tangent** **Line by selection**
Cover Slab or roof type		Reference region	**Rectangular** **Circular** **Polygonal** **Pick lines** **Pick existing region**
Building cover		Reference plane shape	-

Table: Structural Objects and their geometry definition

Straight line

The steps of a straight line definition:

1. Define the start point of the line by giving coordinates or mouse-clicking.
2. Define the end point of the line by giving coordinates or mouse-clicking.

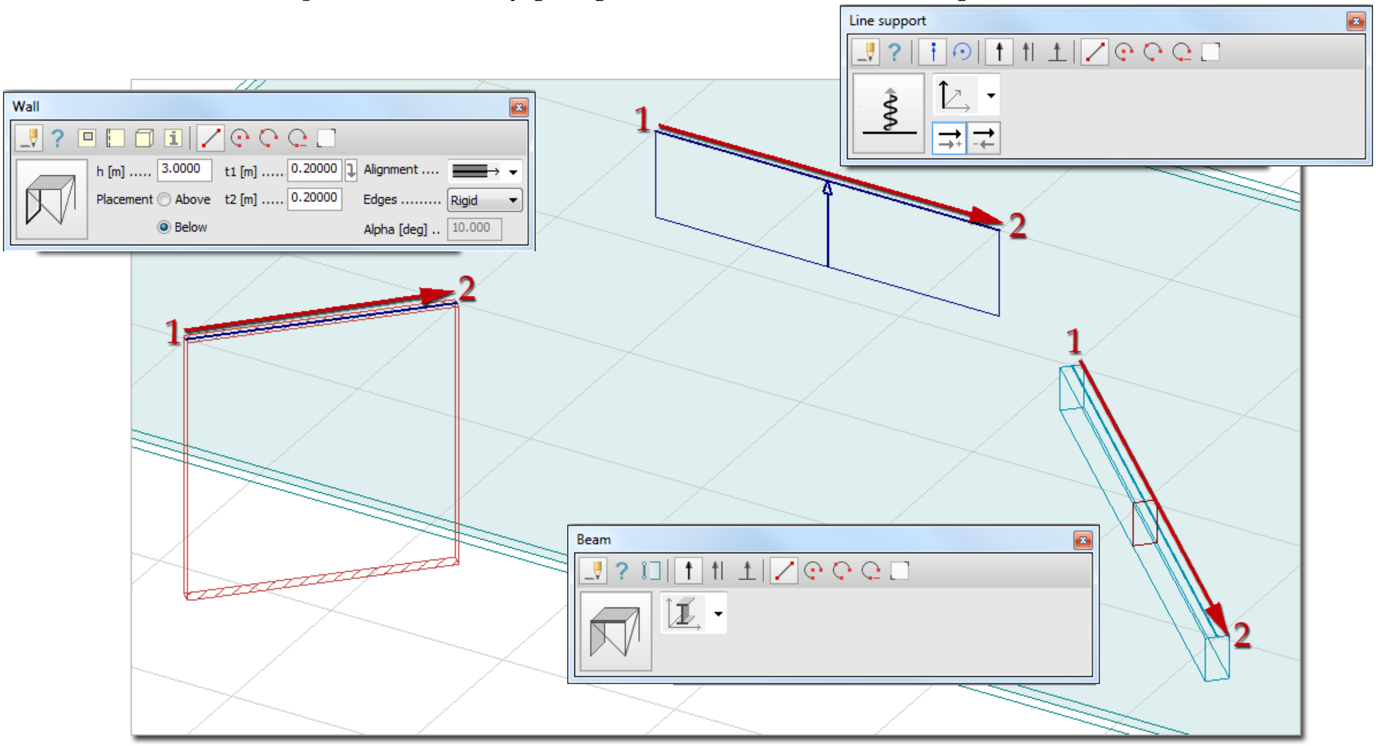


Figure: Some examples for defining structural objects with Straight line

Although Walls are planar objects with reference line, they are vertical and defined by their base reference line only in the **FEM-Design Plate** and **3D Structure** Modules. The final geometry of the reference region is set by the wall height. The next figure shows the differences of height measuring between the *Plate* and *3D Structure* Modules. Of course, the height defines the position of reference region of curved Walls too.

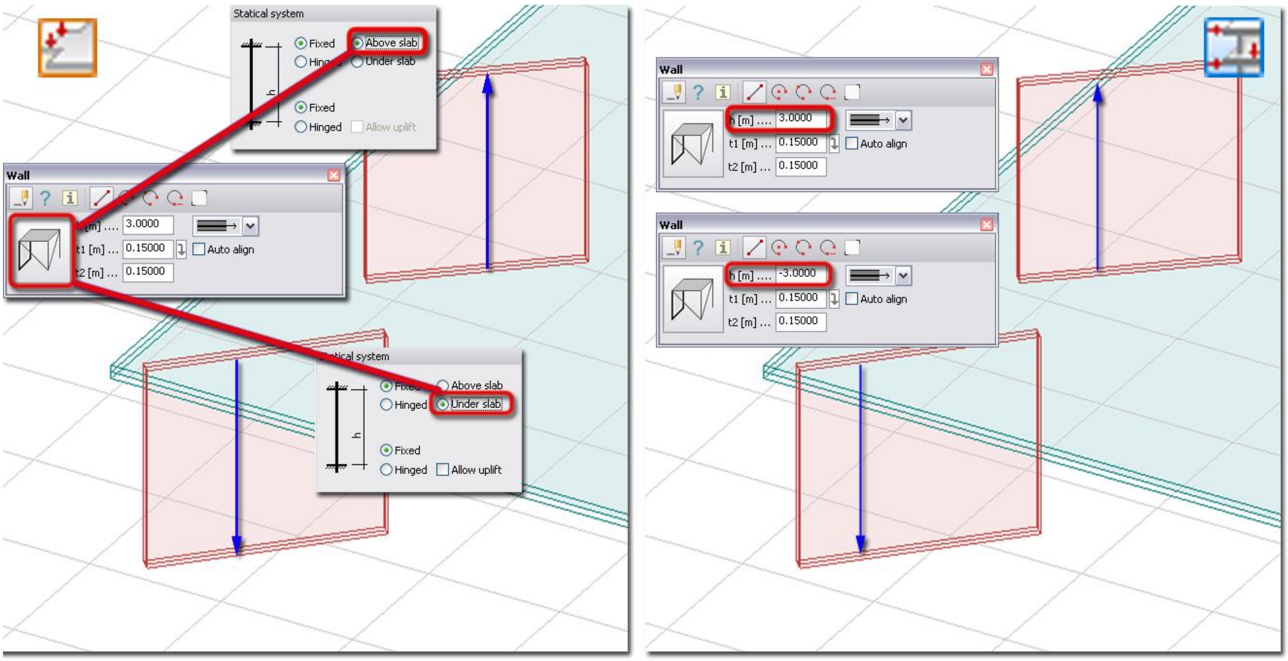


Figure: Height direction of Walls in Plate and 3D Structure Modules

In 3D Structure Module, the rectangle shape of the wall can be edited by the **Modify region > Split region** tool and other editing tools (*Edit* menu). Also the **Hole** tool of Wall tool palette can be used to edit the reference regions.

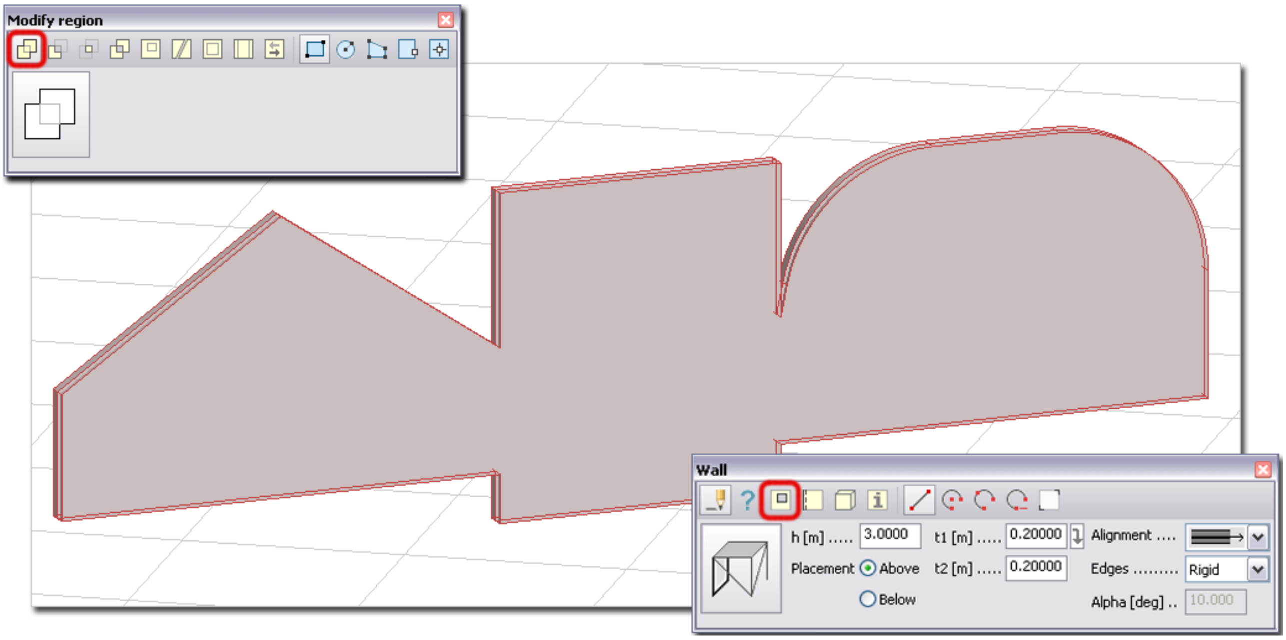


Figure: Some examples for additional editing of Wall region

Arc by center, start and end points

The steps of an arc definition with its center, start and end points:

1. Define the center point of the arc by giving coordinates or mouse-clicking.
2. Define the start point of the arc by giving coordinates or mouse-clicking.
3. Set the drawing direction (clockwise or counterclockwise) with mouse-clicking. Define the end point of the arc by giving coordinates or mouse-clicking, or set the central angle (4.) by giving its value. Circle can be defined by angle 360°.

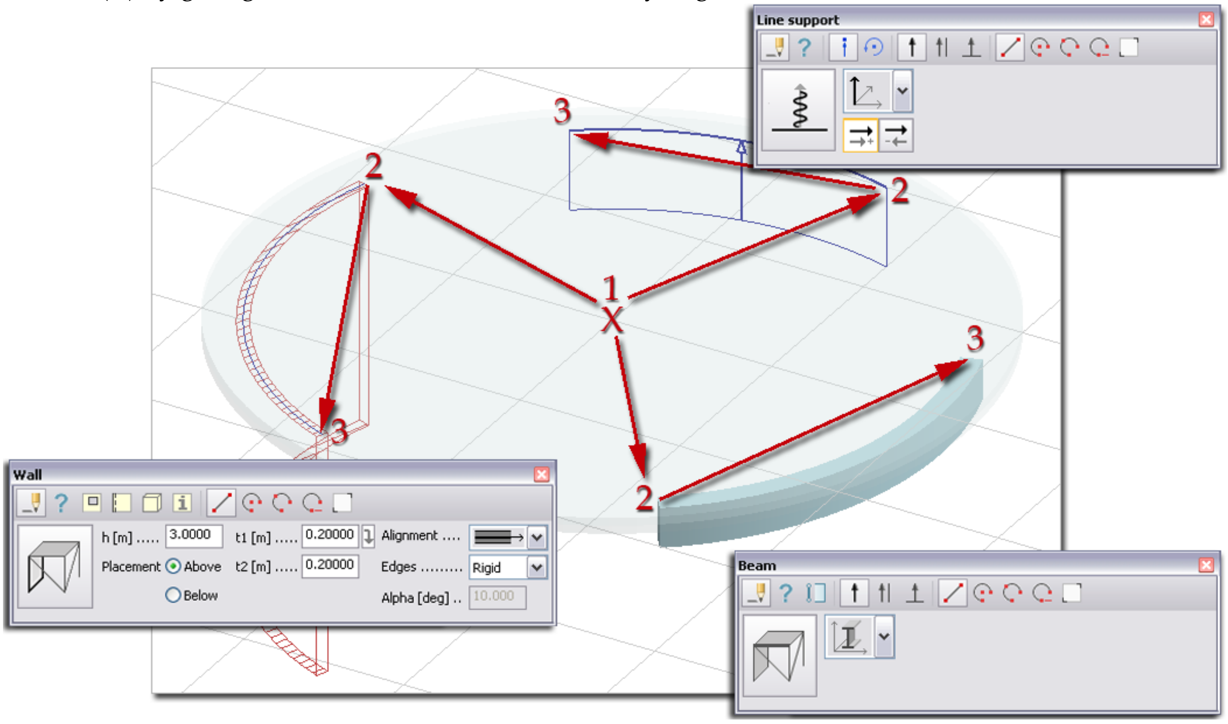


Figure: Some examples for defining structural objects with Arc by center, start and end points

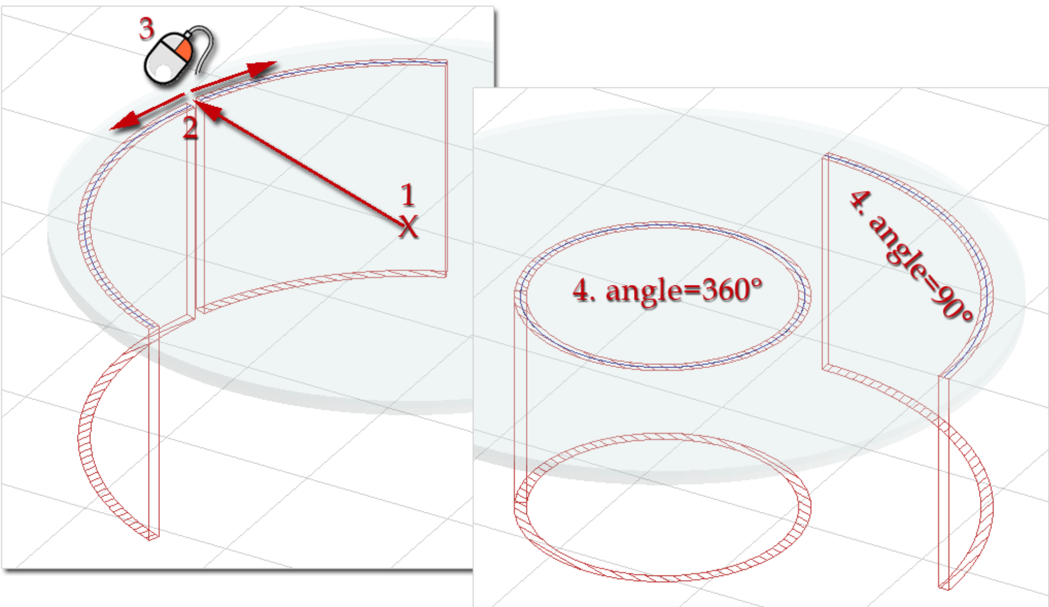


Figure: Drawing direction and angle definition

Arc by 3 points

The steps of an arc definition with its three points:

1. Define the start point of the arc by giving coordinates or mouse-clicking.
2. Define the end point of the arc by giving coordinates or mouse-clicking.
3. Define the third, peripheral point of the arc by giving coordinates or mouse-clicking.

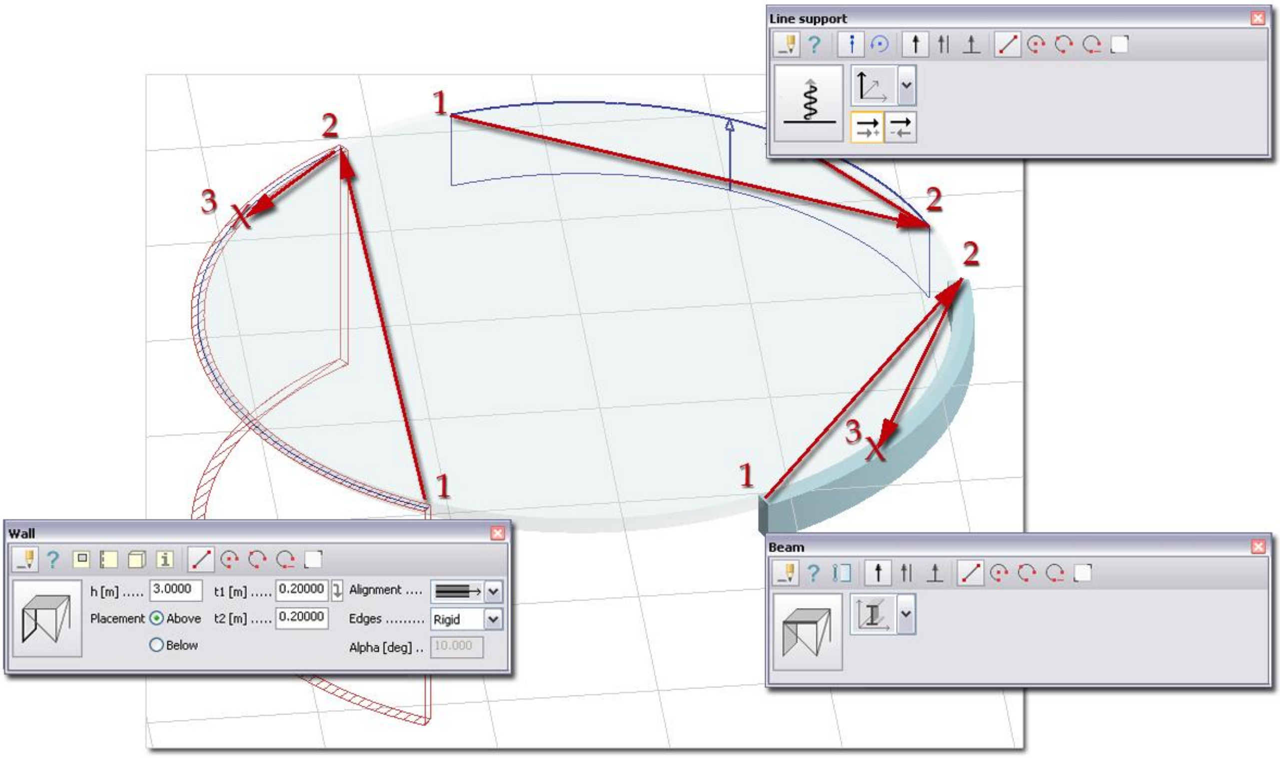


Figure: Some examples for defining structural objects with Arc by 3 points

Arc by start, end point and tangent

The steps of an arc definition with its start, end point and tangent:

1. Define the start point of the arc by giving coordinates or mouse-clicking.
2. Define the end point of the arc by giving coordinates or mouse-clicking.
3. Set the tangent side with mouse-clicking. Define the tangent direction from the start point with a third point (e.g. a point on a tangentially connected line) by giving coordinates or mouse-clicking.

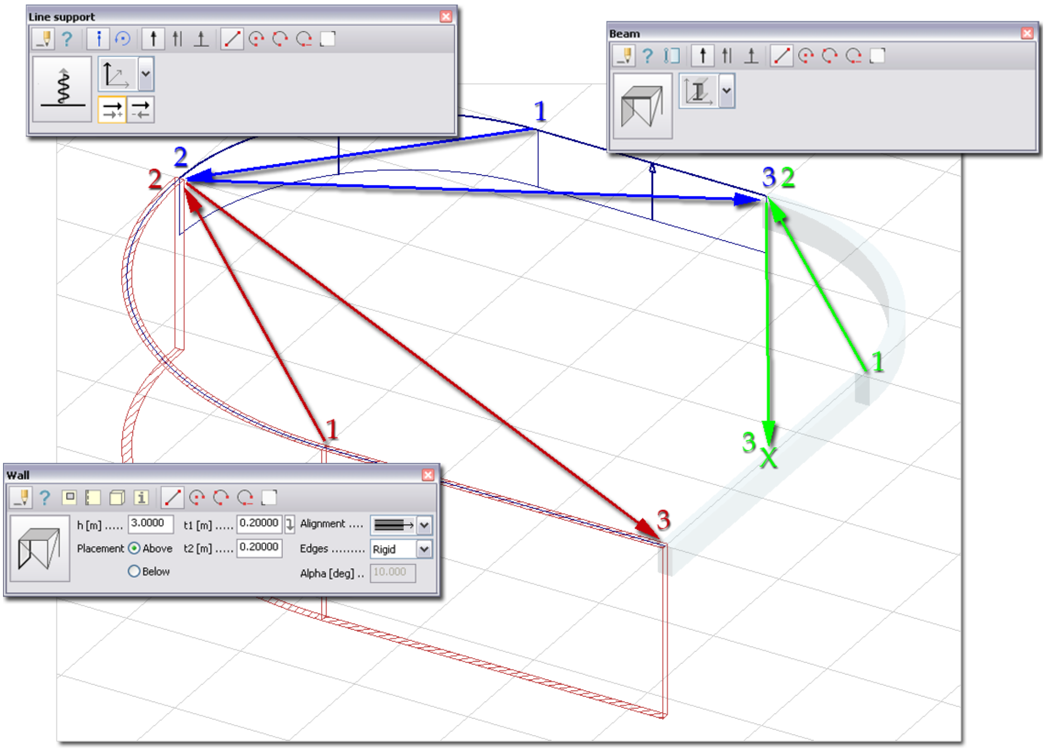


Figure: Some examples for defining structural objects with Arc by 3 points

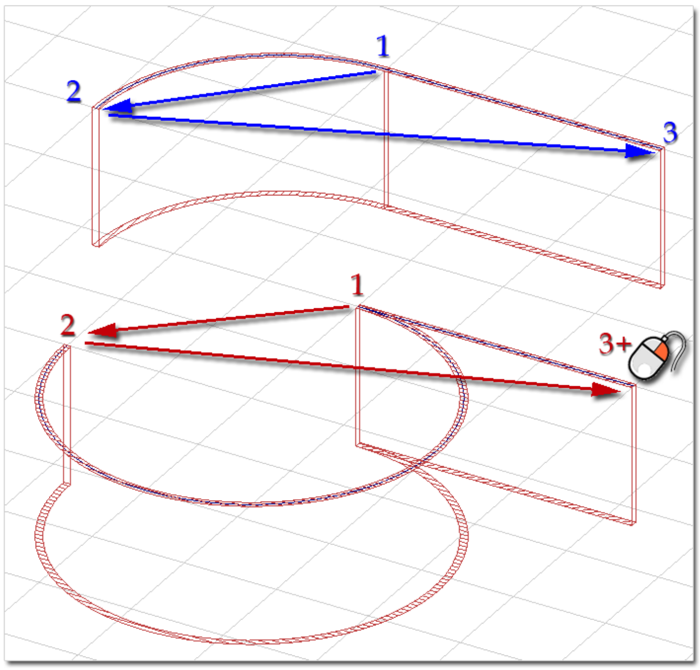


Figure: Although same definition points are defined, the tangent side is different

Line by selection

The step of a reference line (straight or arc) definition by selecting a previously defined line:

Select lines or region (drawing or structural object) edges define the requested shape of the reference line with one of the **selection modes**.

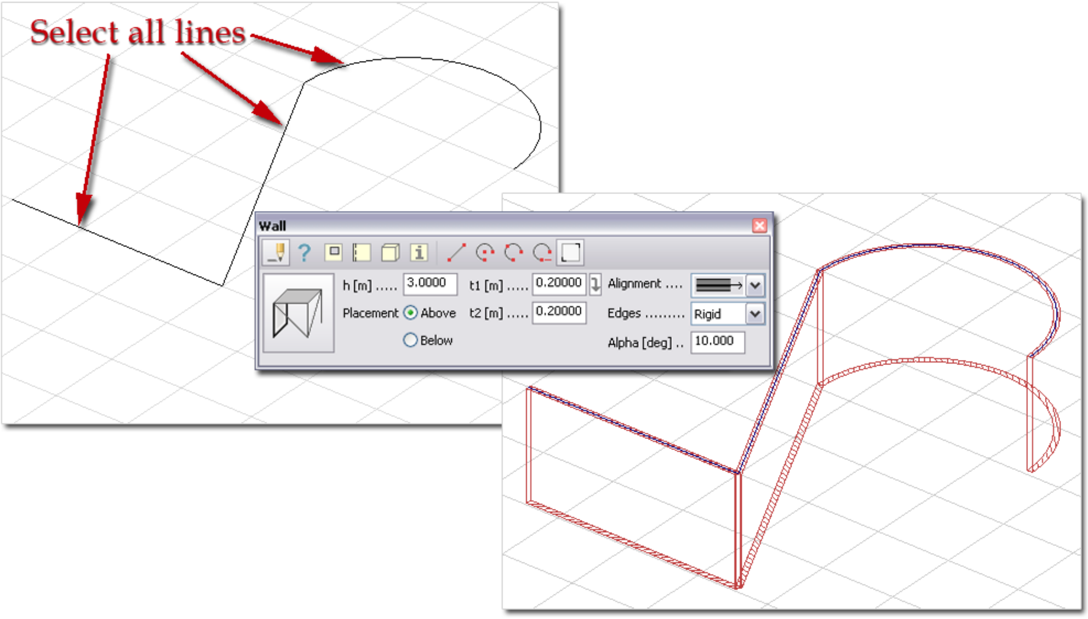


Figure: An example for defining Wall by selecting all lines

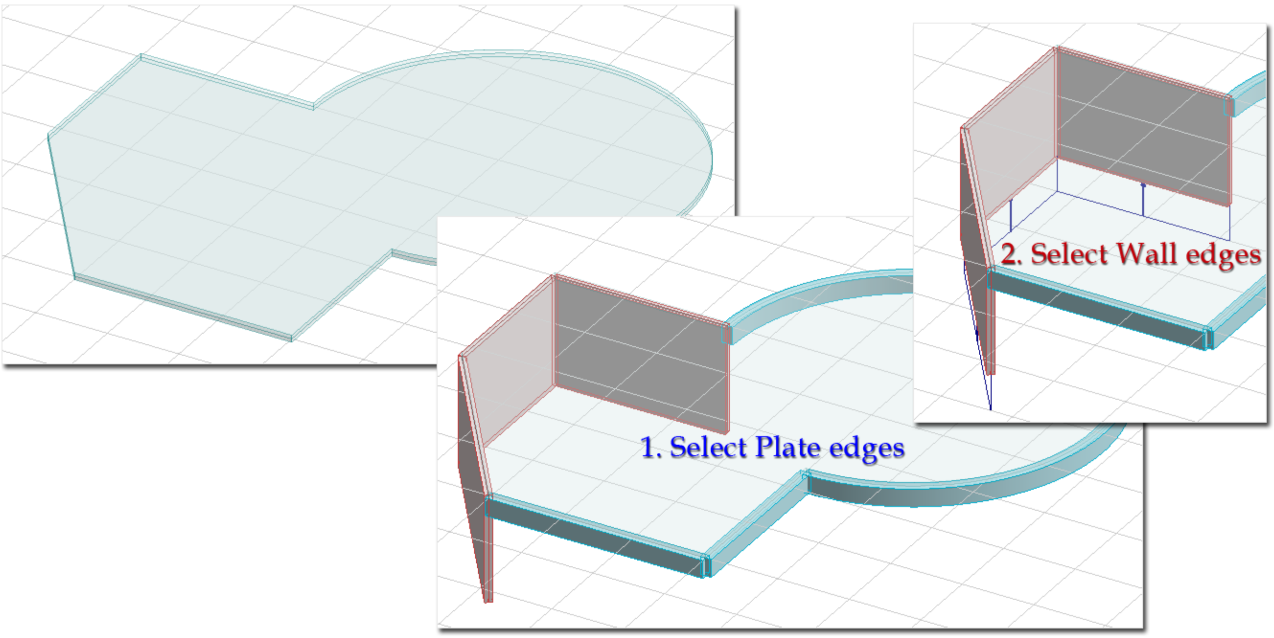


Figure: Defining Beams and Walls on Plate edges (1.) and Line supports on Wall edges (2.)

⚠ For *Columns* in FEM-Design 3D Modules, only vertical lines or edges can be selected.

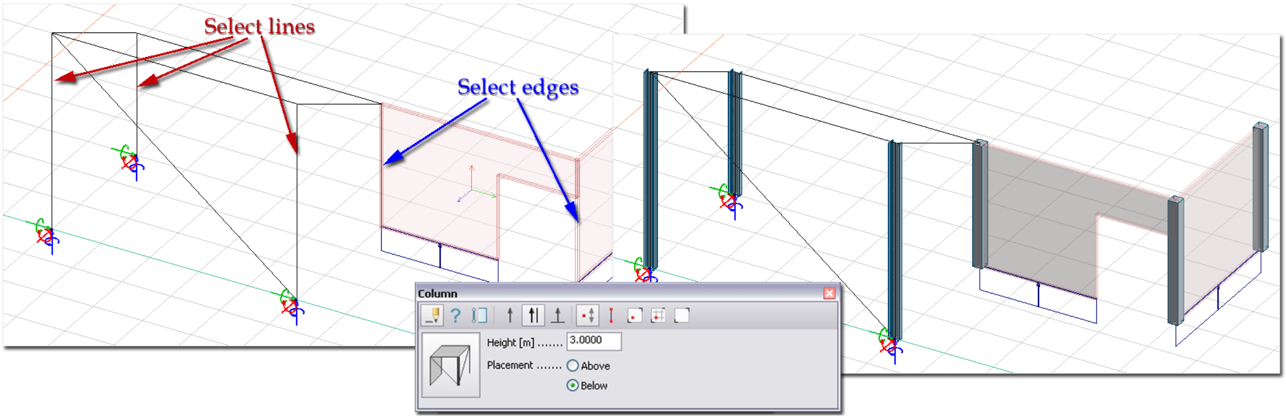


Figure: Column definitions by selecting vertical lines/edges

Line by insertion point and height

As a simple definition of Columns in **3D Structure**, **3D Frame** and **PreDesign** modules define the position of the Column with its insertion point and height. The positive or negative value of the Height sets the measuring direction of the height.

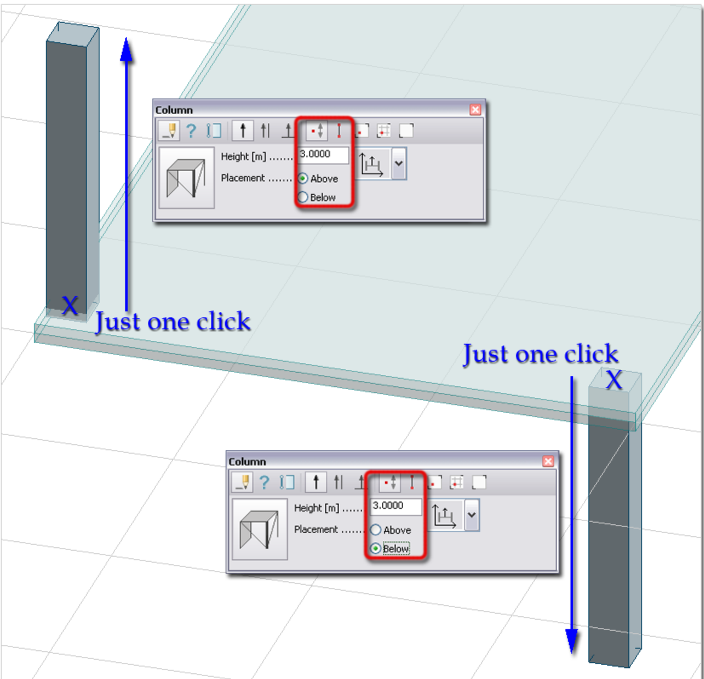


Figure: Column definition with one-click in 3D modules

Select axes

This definition mode is for defining **Columns** in the intersections of selected axes.

Select axes with one of the **selection modes**.

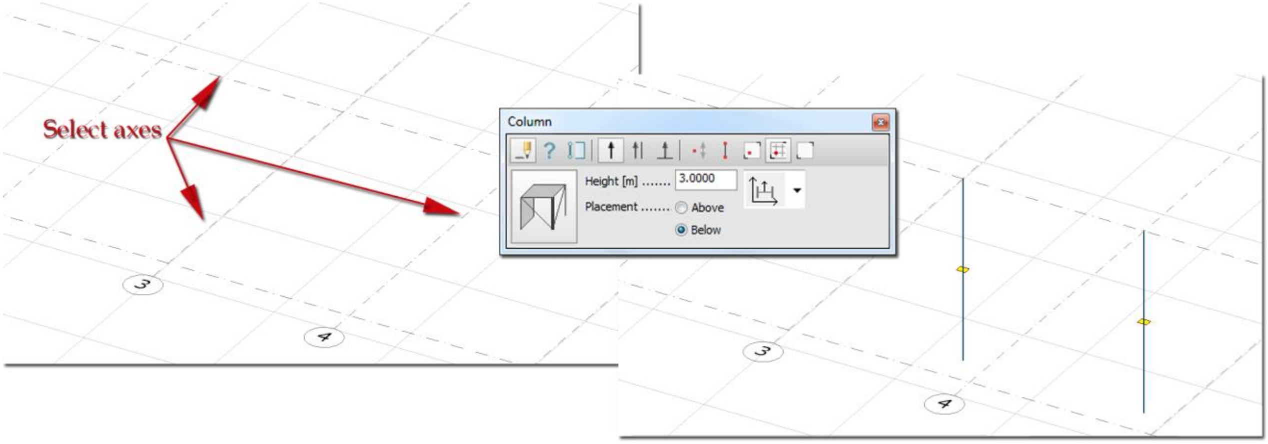


Figure: An example for defining columns with selecting axes

Vertical line

This is a special tool for defining Column with two points. FEM-Design Columns can be only vertical, so the second point is not needed to be on the column axis, because its vertical distance from the start (first) point defines the height of the column. This Column definition tool is useful, when you do not know the height value of the new column, but points and lines defines the column's vertical extension (height) can be easily found. Use **Object Snap** tools to find point defines indirectly the column's end point.

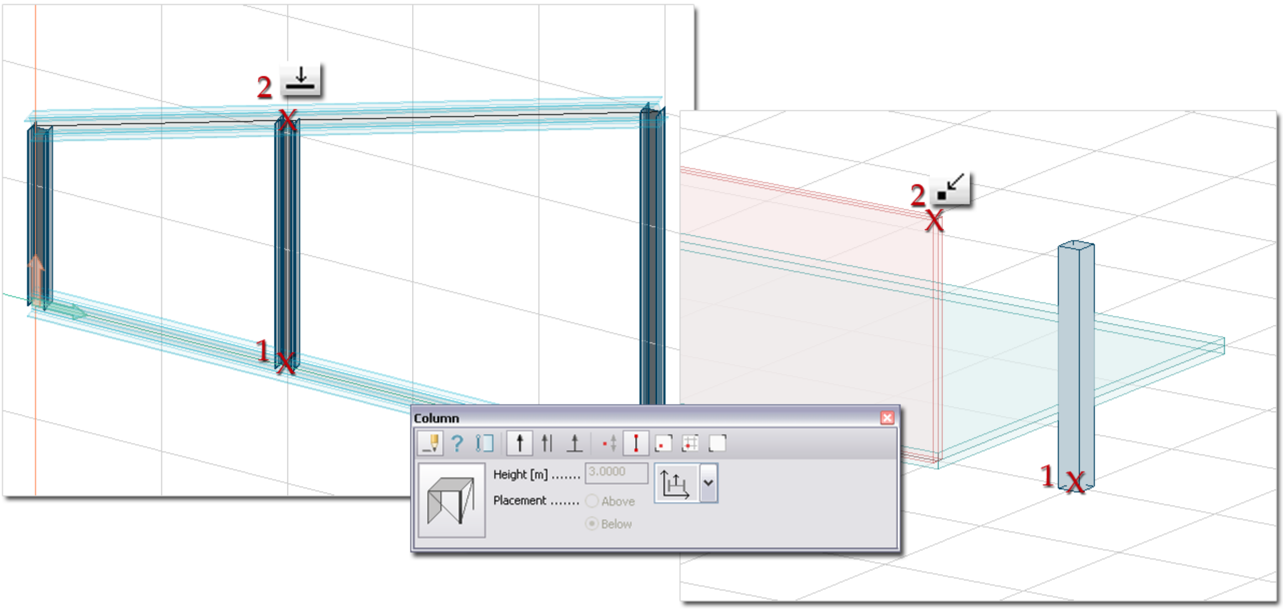


Figure: Examples for defining vertical columns with two points

**Rectangular**

The steps of a rectangular region definition:

I. Rectangle's edges parallel with the UCS:

1. Define the point of the first corner by giving coordinates or mouse-clicking.
2. Define the point of the end corner by giving coordinates or mouse-clicking.

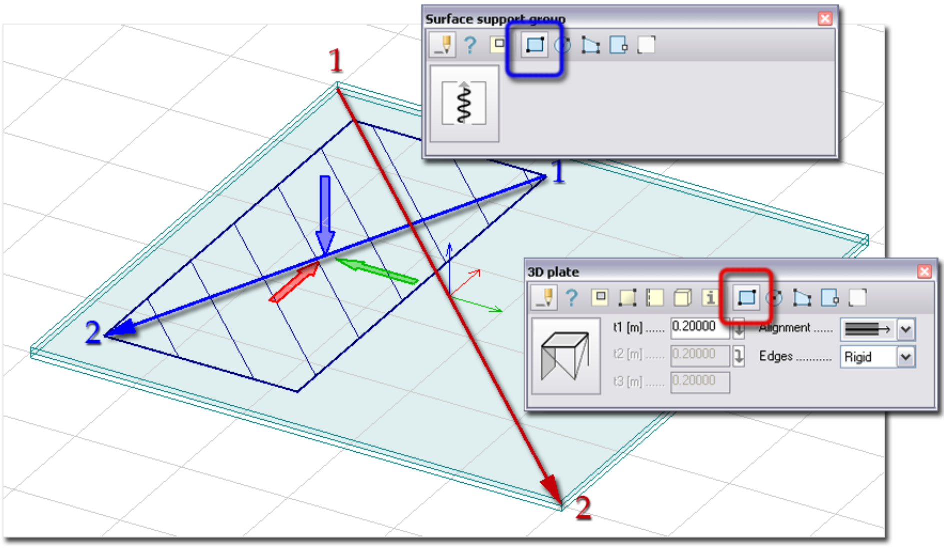


Figure: Defining rectangular Plate and Surface support

II. Rectangle's edges not parallel with UCS:

1. Click 

one or two times to define the rectangle's first edge's direction

   a. If you click 

one time, the direction is can be defined from the global coordinate system's origin,

   b. If you click 

one times, the direction is defined in an arbitrary direction.

2. Define the arbitrary direction of the rectangle's one line.
3. Define the point of the first corner by giving coordinates or mouse-clicking.
4. Define the point of the end corner by giving coordinates or mouse-clicking.

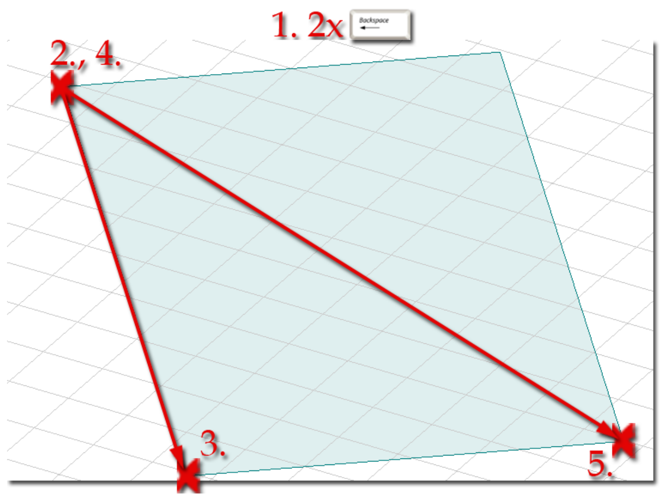


Figure: Defining rectangular plate

The geometry of rectangular regions as well as other (later mentioned) region shapes can be edited by the **Modify region > Split region** tool and other editing tools (*Edit* menu). Also the **Hole** tool of planar objects' definition command can be used to edit the reference regions.

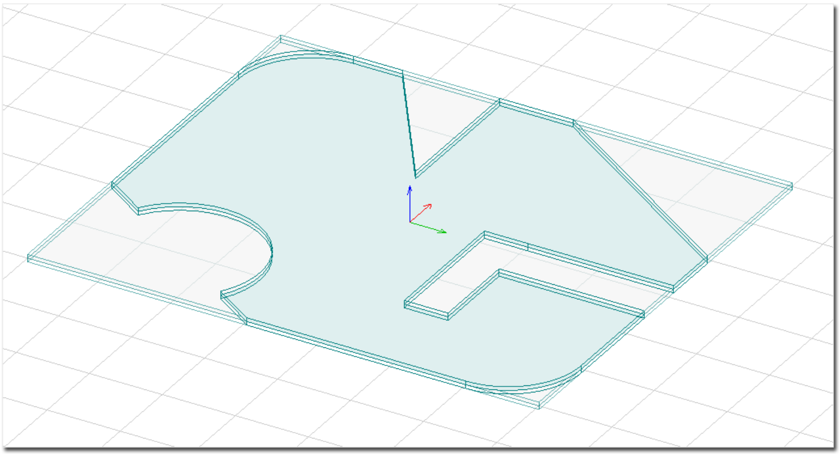


Figure: An edited rectangular Plate region

Circular

The steps of a circular region definition:

1. Define the center point by giving coordinates or mouse-clicking.
2. Define the radius by giving its value or a point on the circle (with coordinates or mouse-clicking).

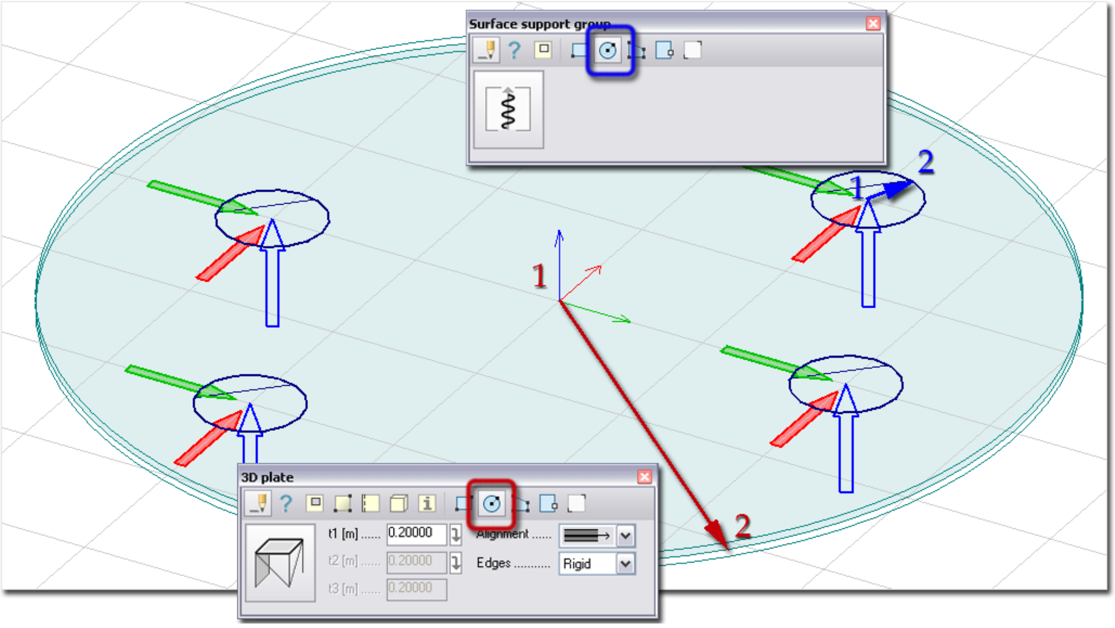
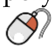


Figure: Defining circular Plate and Surface supports

 Polygonal

The steps of a polygonal region definition:

1. Define the points of the polygon vertexes by giving coordinates or  mouse-clicking.
2. Close the polygon with  mouse-clicking or  key.

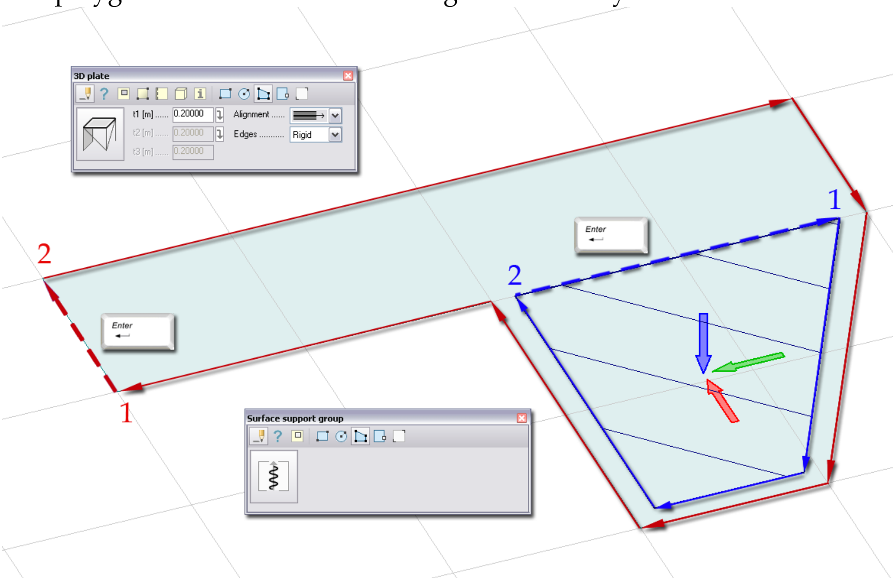


Figure: Defining polygonal Plate and Surface support

Pick lines

With this method, previously defined or imported (DWG/DXF) drawing elements can be used as sketches of structural region shapes. The step of definition:

Select a closed line defines the requested shape of the reference region with mouse-clicking.

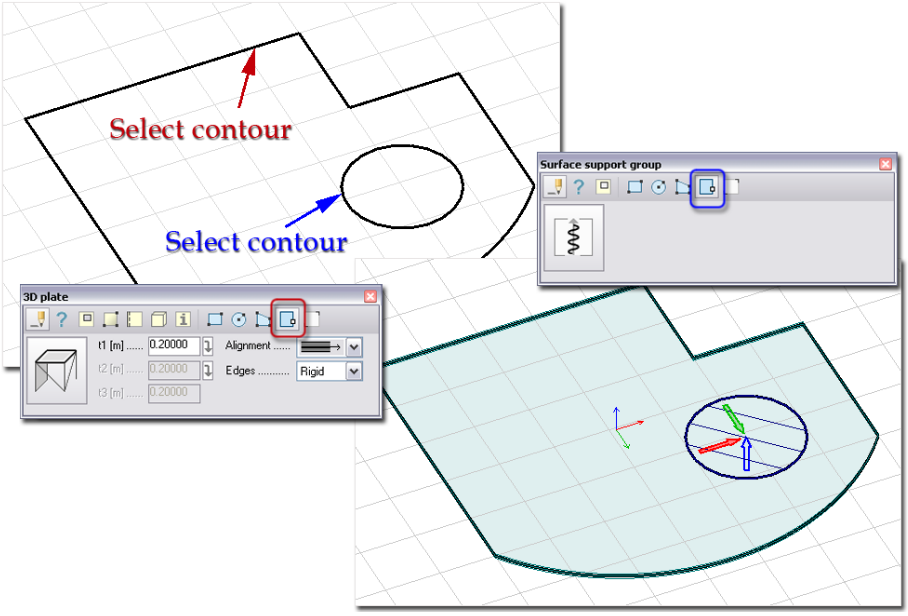


Figure: Defining Plate and Surface support by using close contours

In case of line junctions, more than one line has to be selected to make clear the continuity of the requested closed contour.

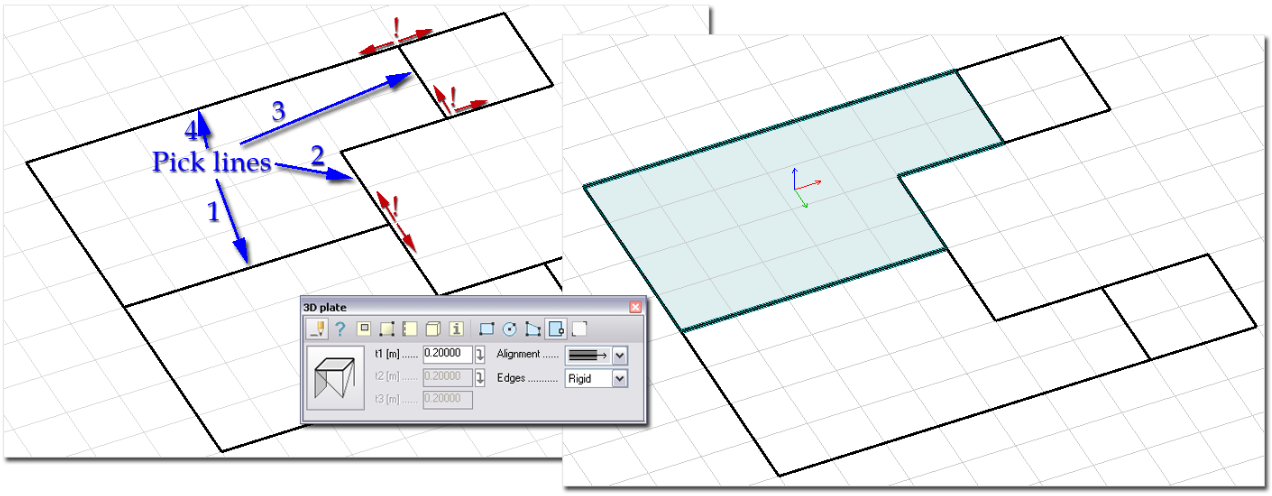


Figure: Selection of more lines to define the right path for the closed shape

Pick existing region

The step of a reference region definition by selecting a previously defined object or drawing region:

select the region(s) defines the requested shape of the reference region(s) with one of the **selection modes**.

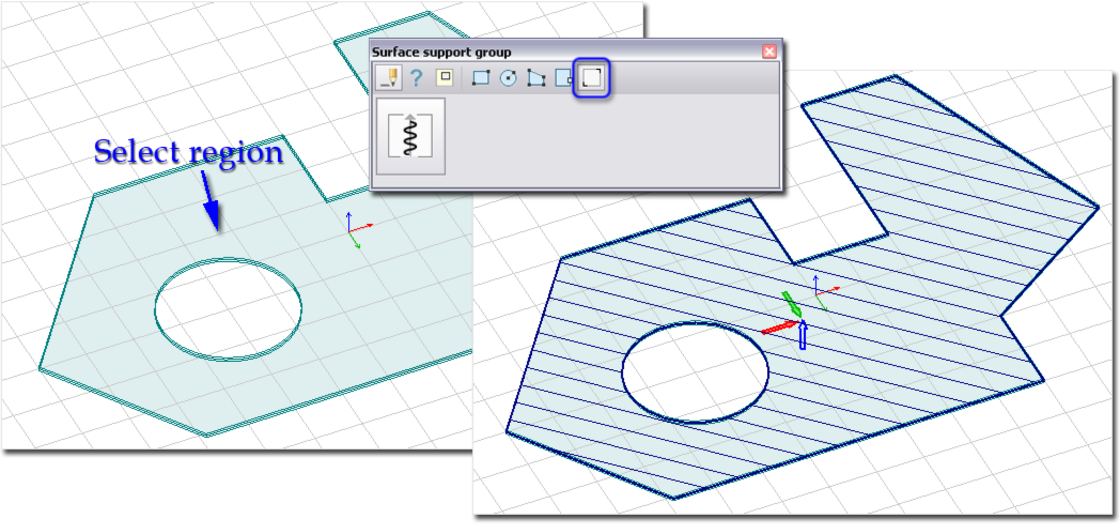


Figure: An example for placing Surface support under a Plate by selecting its region

With *Pick existing region*, shell (*Plate*) elements can be easily place on the entire surface or some surface components of a **Solid** body (*Draw* > *Solid*).

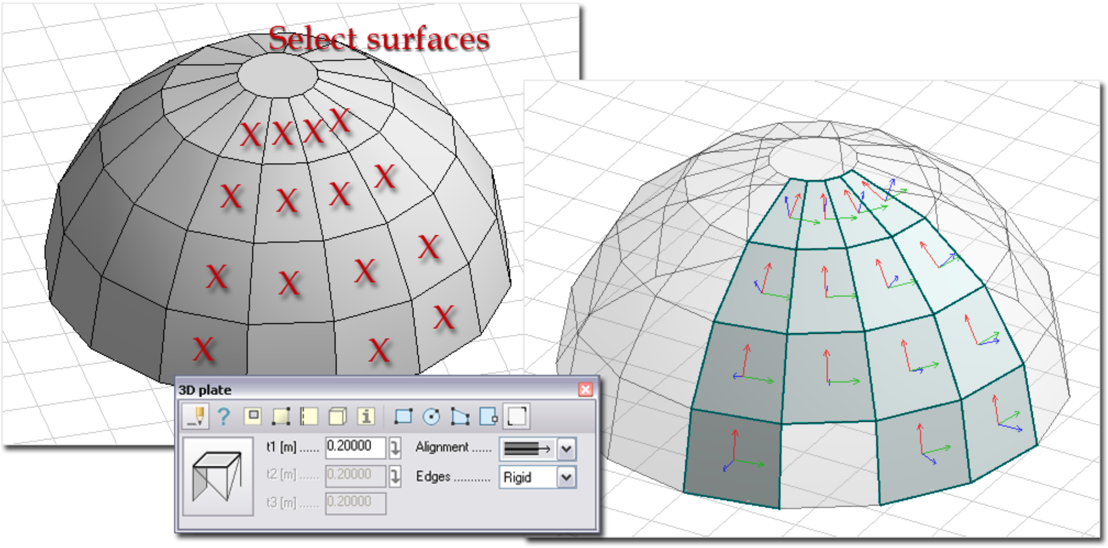


Figure: Defining shell (Plate) regions by picking the surface regions of a Solid body

Hole

Holes, openings and cuttings can be added to reference regions (Plate, Wall and Surface support) with the *Hole* tool. The following geometries can be used for holes:

The steps of a hole definition:

1. Select the host region with mouse-clicking. Clicking a region places the ***UCS*** into the region plane, so giving hole coordinates needs only X and Y values from the UCS origin.
2. Define the geometry of the hole with one of the following geometry modes:

   Rectangular

   Circular

 ***Polygonal***

 ***Pick lines***

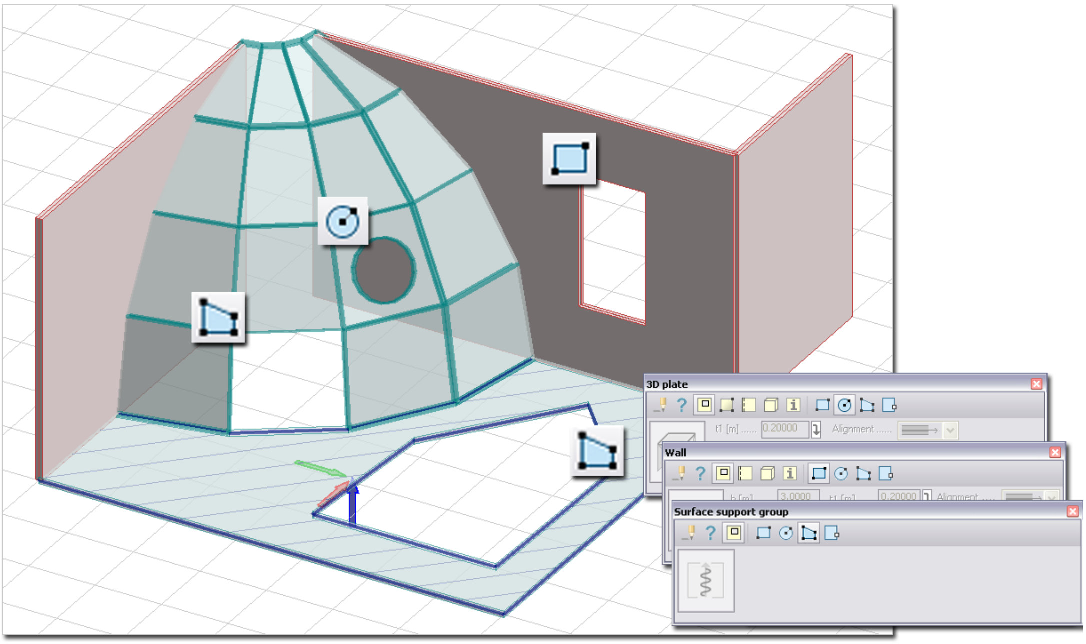


Figure: Examples for holes

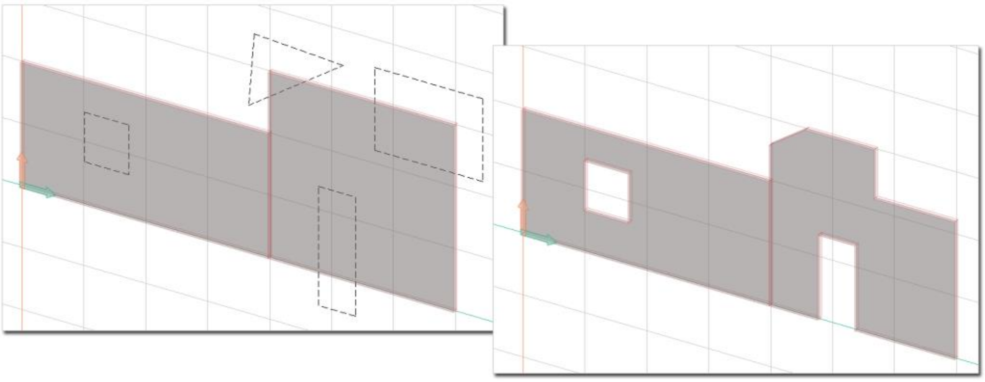


Figure: Hole tool can be used to edit region geometries as cuttings

Holes can be easily copy inside a region or among regions with same reference plane position with the **Copy** ***command*** (*Edit* menu). It is recommended to display only the regions' reference plane (inactive **Display thickness** option) to easily select the contour of the hole you would like to copy. To set the distances/new positions, the **UCS** has to be in the plane of the host region(s).

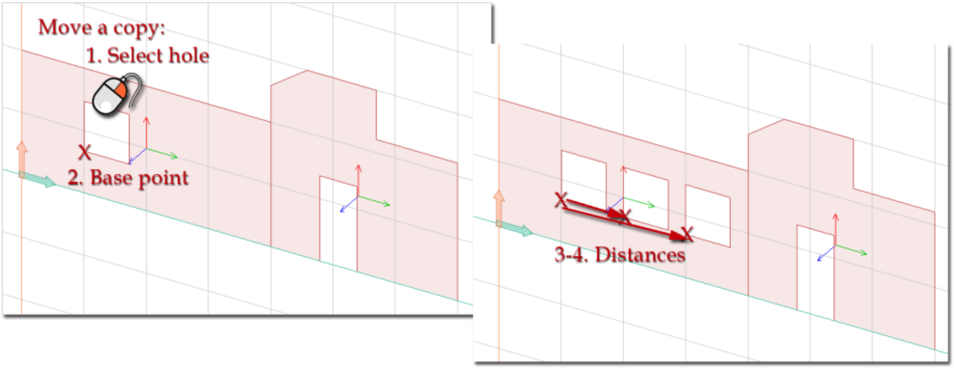
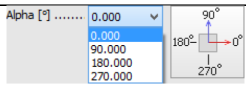


Figure: Copying holes in a Wall

Direction

Numerous objects need direction settings for their positioning (bar elements) or their components definition (supports). The next table summarizes the direction possibilities by structural elements.

Type	Modules where available	Direction for	Direction Modes
Soil		-	-
Borehole		-	-
Isolated foundation		Solid position: Direction of local x' axis	**Predefined direction** **Local system**
Wall foundation		-	-
Foundation slab		-	-
Beam		Cross-section position: Direction of local y' axis	**Predefined direction** **Parallel with line** **Perpendicular to plane**
Column		Cross-section position: Direction of local y' axis	**Predefined direction** **Parallel with line** **Perpendicular to plane**

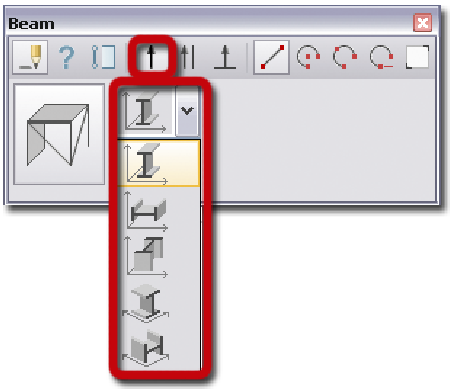
Truss member		Cross-section position: Direction of local y′ axis	**Predefined direction** **Parallel with line** **Perpendicular to plane**
Column corbel		Horizontal axis position: Angle between the horizontal axis of the corbel and the local y′ axis of the column	 **Alpha[°]**
Point support		Rotation direction	**Predefined direction** **Parallel with line** **Perpendicular to plane**
		Motion component direction	
		Component direction	
Point support group		Component direction	- (**Predefined direction**)
		Component direction (reference system)	**Predefined direction** **Local system**
Line support		Rotation direction	**Predefined direction** **Parallel with line** **Perpendicular to plane**
		Motion component direction	
		Component direction	
Line support group		Component direction	- (**Predefined direction**)
		Component direction (reference system)	**Predefined direction** **Parallel with line** **Perpendicular to plane**

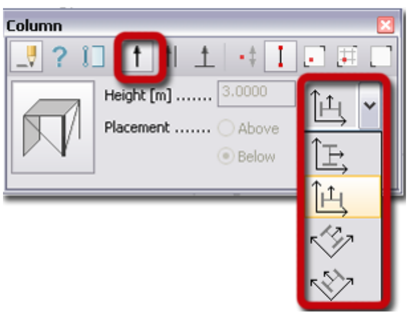
Point-point connection		Component direction	**Predefined direction** **Local system**
Line-line connection		Component direction	**Predefined direction** **Local system**

Table: Structural Objects and their direction settings

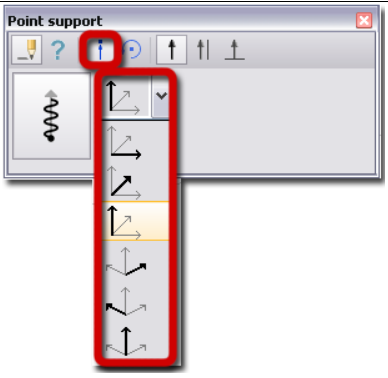
Predefined direction

With this option an axis/a plane of the **Global** or the **User-defined (UCS)** co-ordinate system can be set for the required direction (e.g. the local y' axis of a beam profile). The direction can be chosen from the drop-down list attached to the *Predefined direction* option. The available directions depend on the applied FEM-Design Module (e.g. *Plate, 3D Structure* etc.).





Symbol	Meaning of direction	System
	Parallel with XY plane	Global
	Parallel with YZ plane	Global
	Parallel with XZ plane	Global
	Parallel with UCS (XY plane)	UCS
	Perpendicular to UCS (XY plane)	UCS
	Parallel with global X axis	Global
	Parallel with global Y axis	Global



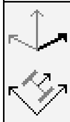
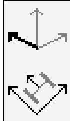
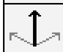
	Parallel with global Z axis	Global
	Parallel with X axis of UCS	UCS
	Parallel with Y axis of UCS	UCS
	Parallel with Z axis of UCS	UCS

Table: The available axis directions to set the new direction

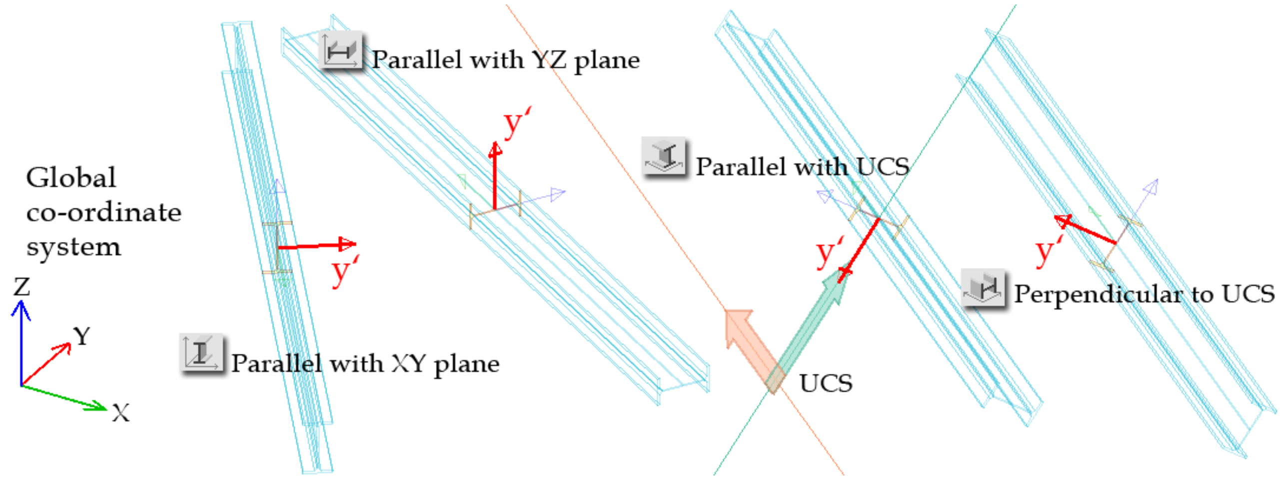


Figure: Examples for Beam profile positions

At supports, additional direction setting can be chosen from the definition tool palette:

 "Positive direction": The orientation is the same with the selected axis orientation;

 "Negative direction": The orientation is the opposite of the selected axis orientation.

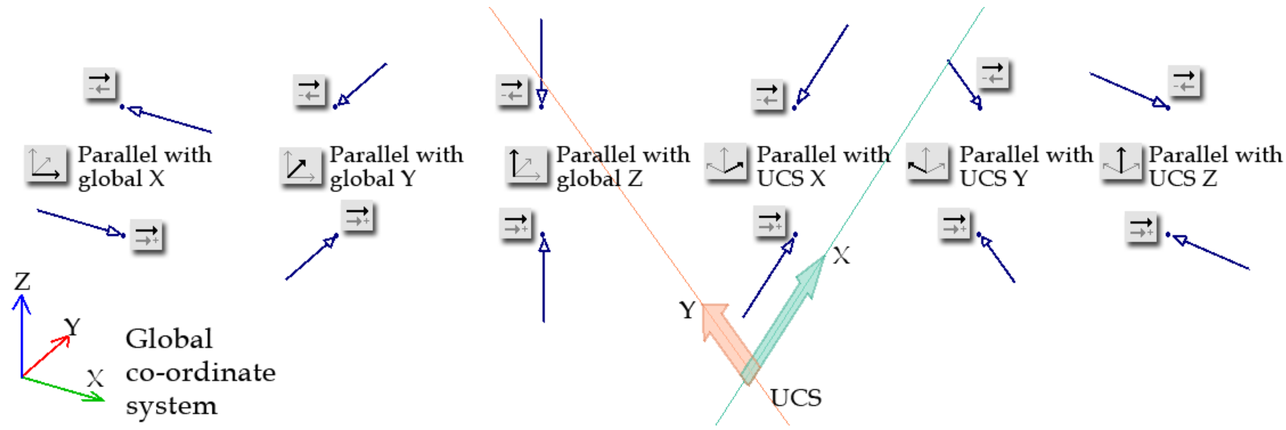


Figure: Examples of Point support position

For some support and connection types, only the reference system can be selected as "predefined direction". That means the direction of the components (x', y' and z') are valid in the selected co-ordinate system (Global or UCS).

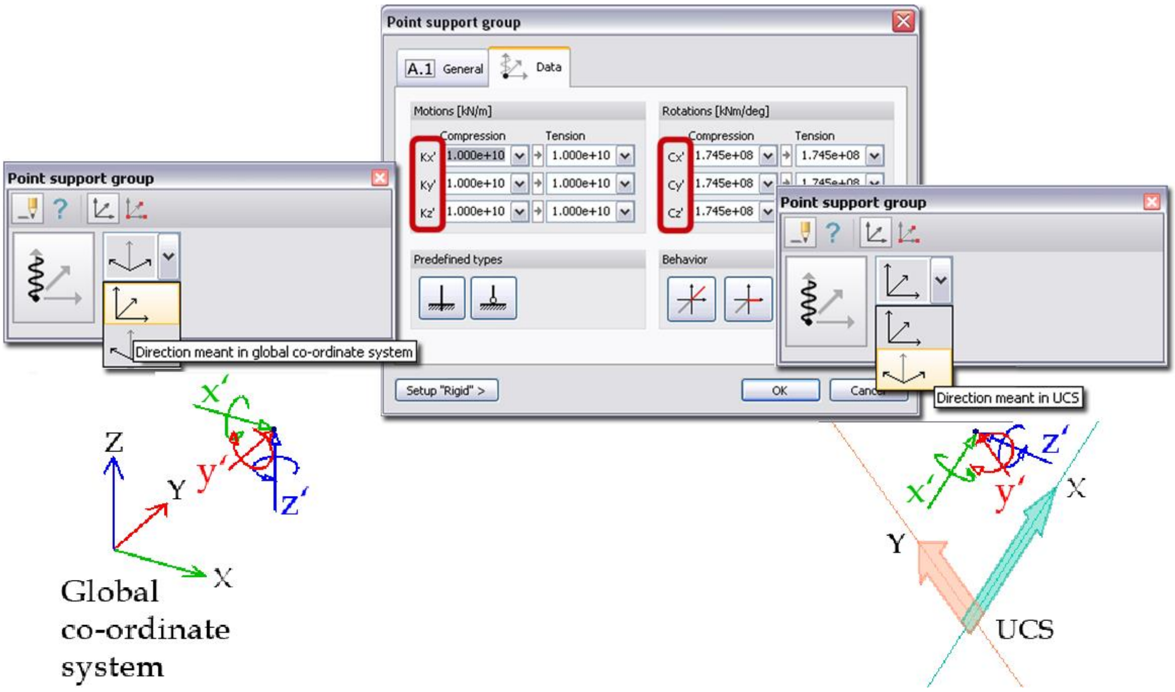


Figure: Support components in different co-ordinate systems

Parallel with line

With this option, the required direction (e.g. the local y' axis of a column profile) can be defined manually with its start and end points. The new direction is set for all objects till [Space] stops the repetition of object insertion and restarts with another direction definition.

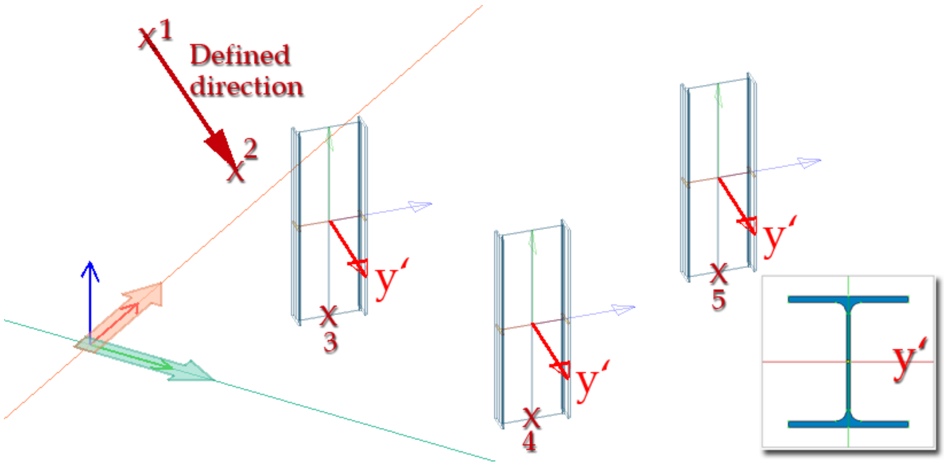


Figure: The local y' axis of column profiles is parallel with the defined direction

Perpendicular to plane/line

With this option, the required direction (e.g. the local y' axis of a beam/column profile) will be perpendicular to a defined plane/line. The plane can be given with three points and the line with two points (start and end points). In case of the perpendicular plane, the third point defines the final orthogonal direction, which the new direction will be parallel with. The new direction is set for all objects till  stops the repetition of object insertion and restarts with another direction definition.

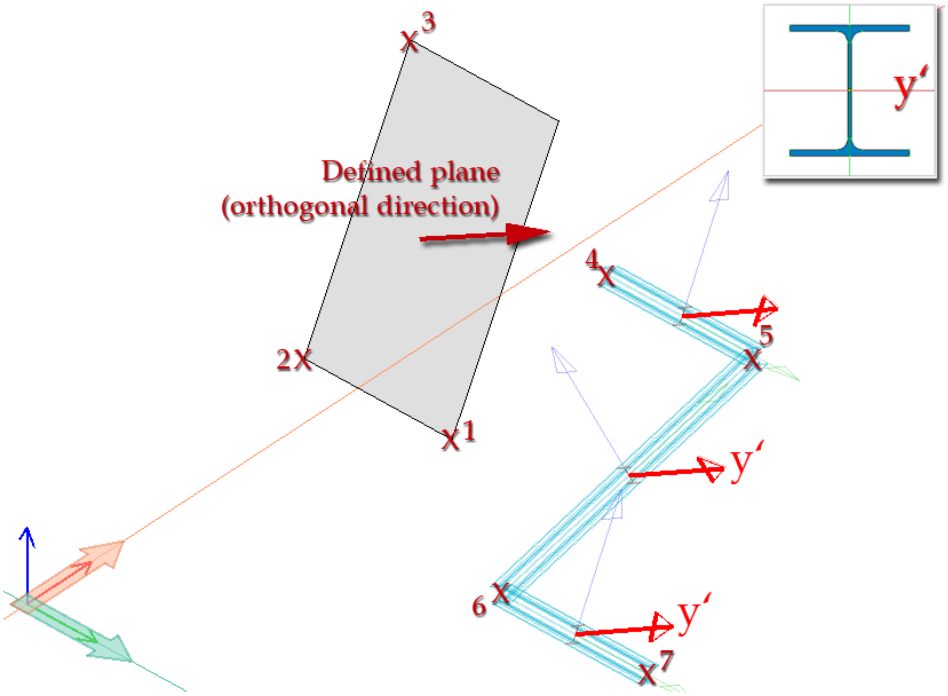


Figure: The local y' axis of beam profiles is perpendicular to the defined plane

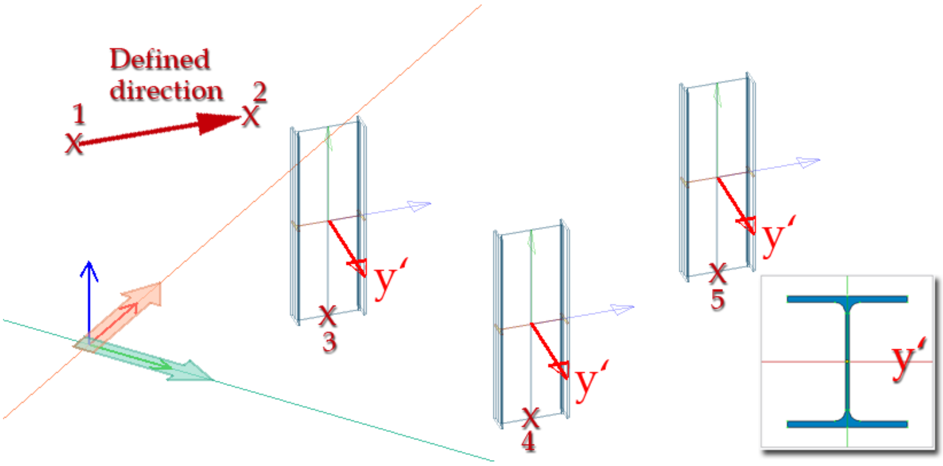


Figure: The local y' axis of column profiles is perpendicular to the defined line

Local system

When the **Global** or **User-defined (UCS)** co-ordinate system is not enough as a predefined system for component directions (**Point support**, **Point-point connection** and **Line-line connection**), a custom local system can be defined. Tree points declare the system and the 4th point places the new object:

1st point: system origin,

2nd point: defines the x' axis direction, 3rd point: defines the y′ axis direction.

The z′ axis of the system always perpendicular to the x′y′ plane.

The new system directions are set for all objects till 

stops the repetition of object insertion and restarts with another system definition.

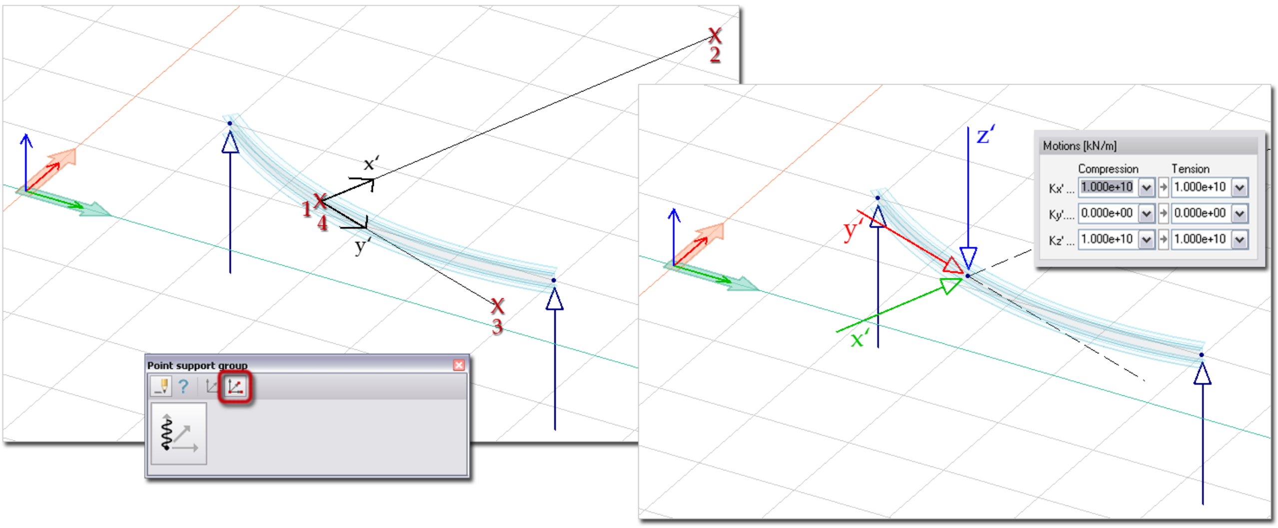


Figure: Point support group set by a special local system

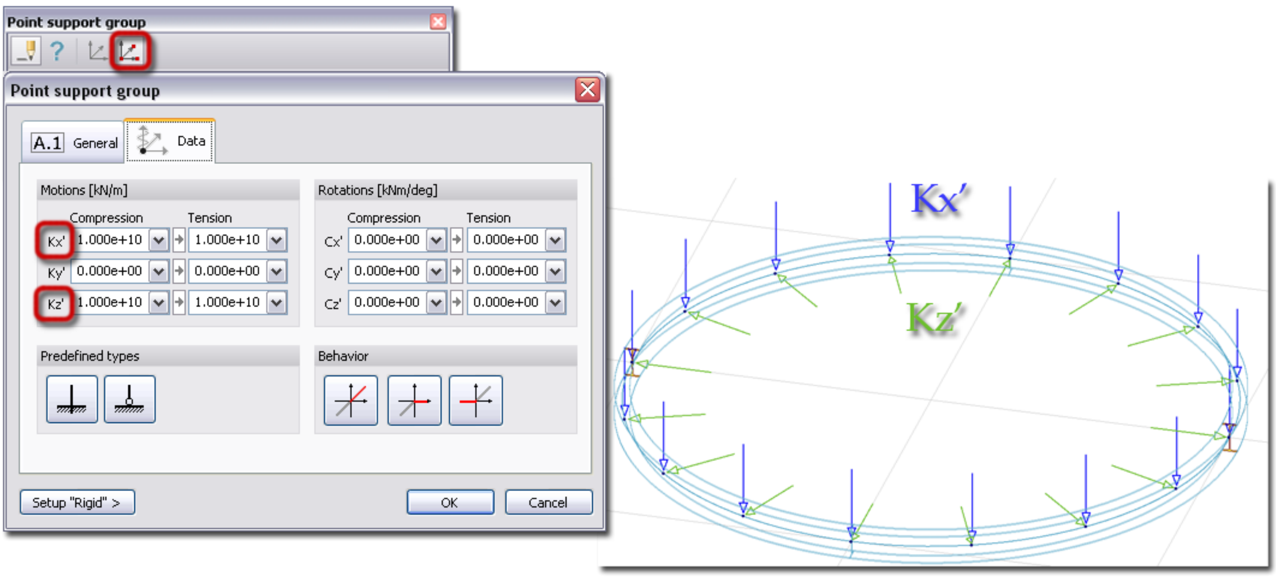


Figure: Typical example for special direction (Point support group)

Change Direction

Any previously set direction can be modified by the editing commands (*Edit* menu): **Change direction** and **Rotate**.

Change direction uses the **Predefined direction**, **Parallel with line** and **Perpendicular to plane** direction definition tools.

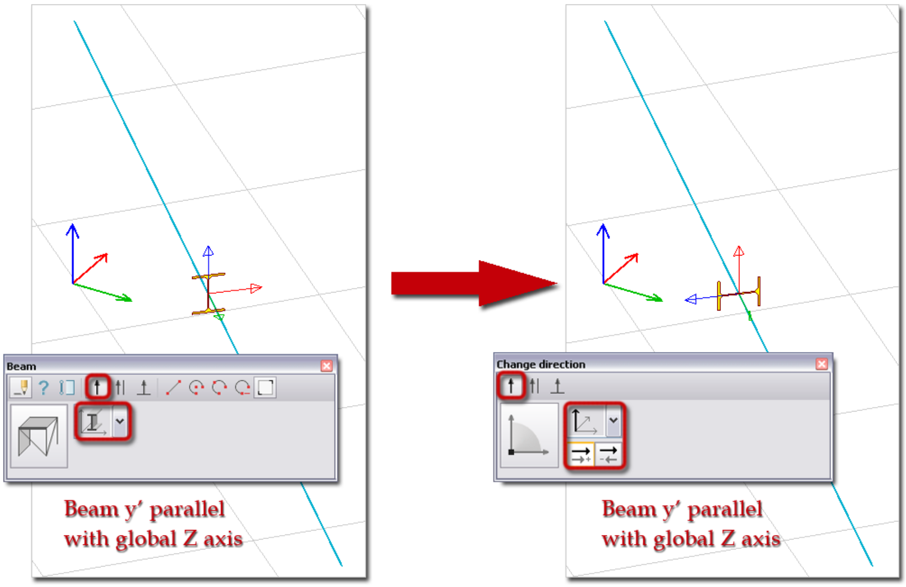


Figure: Beam position (local y′ axis) is modified with Change direction

Rotate edits a selected direction or the main direction of a selected system with giving new direction points or the rotation angle. Rotation works around a given point or an axis.

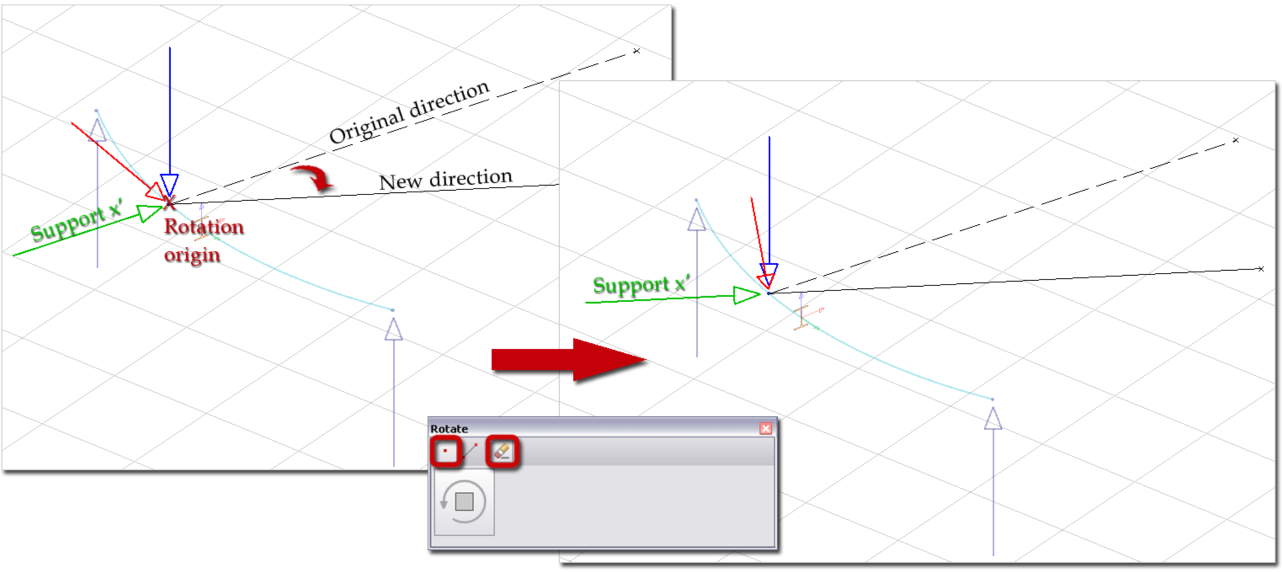


Figure: Rotate modifies the main direction (x′) of the Point support group

1D Members

This chapter summarizes all features and properties of a 1D member such as **Borehole**, **Isolated foundation**, **Wall foundation**, **Beam**, **Column**, **Truss member**, **Fictitious bar** and **Corbel**.

Borehole

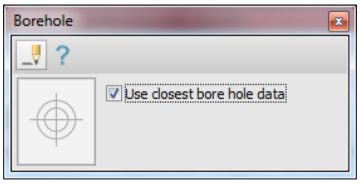
Borehole property	Description
Modules where available	
Geometry	Straight

Table: Borehole properties

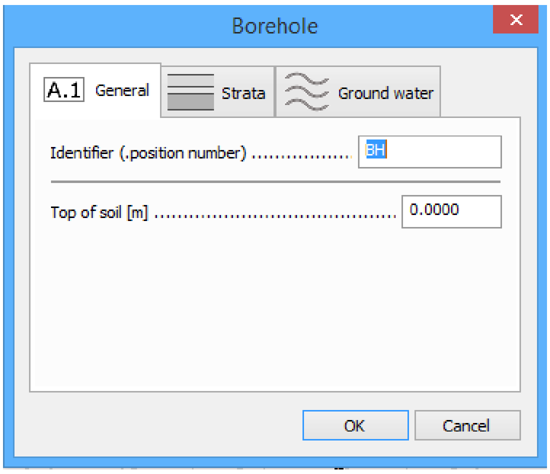
Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**. In the 3D design modules, respect that the gravity direction is always the Global Z axis direction.
2. Start *Borehole* command from Structure tabmenu and choose *Define*.
3. Place a borehole. There are two way to place a borehole:

   - If "Use closest bore hole data" is checked, the borehole properties will be the same as the nearest borehole.

   - If "Use closest bore hole data" is unchecked properties of the new *Borehole* can be set at *Default settings*:



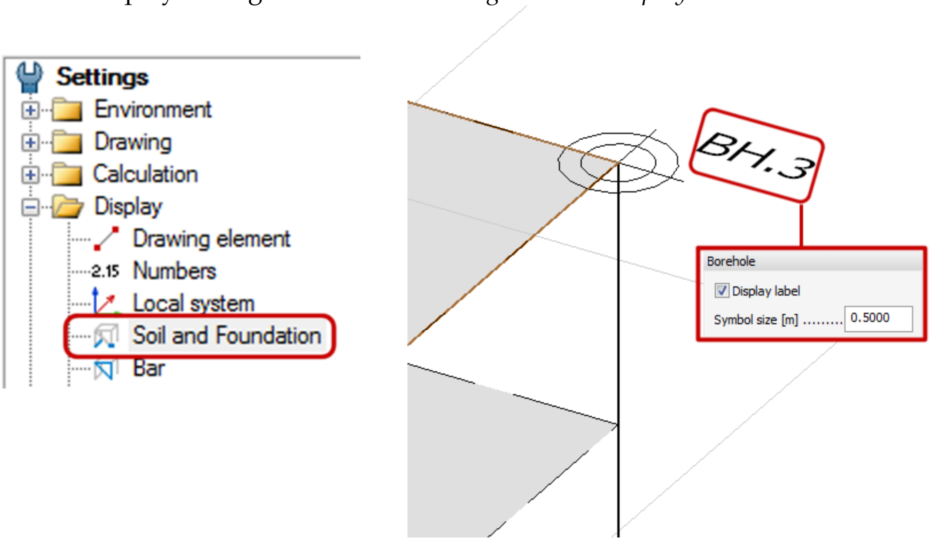
- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Groung Water Level* (Ground water)
  Water level has effect to the design calculations. Soil and foundations which are under the water level, FD calculates with modified density.

- Strata

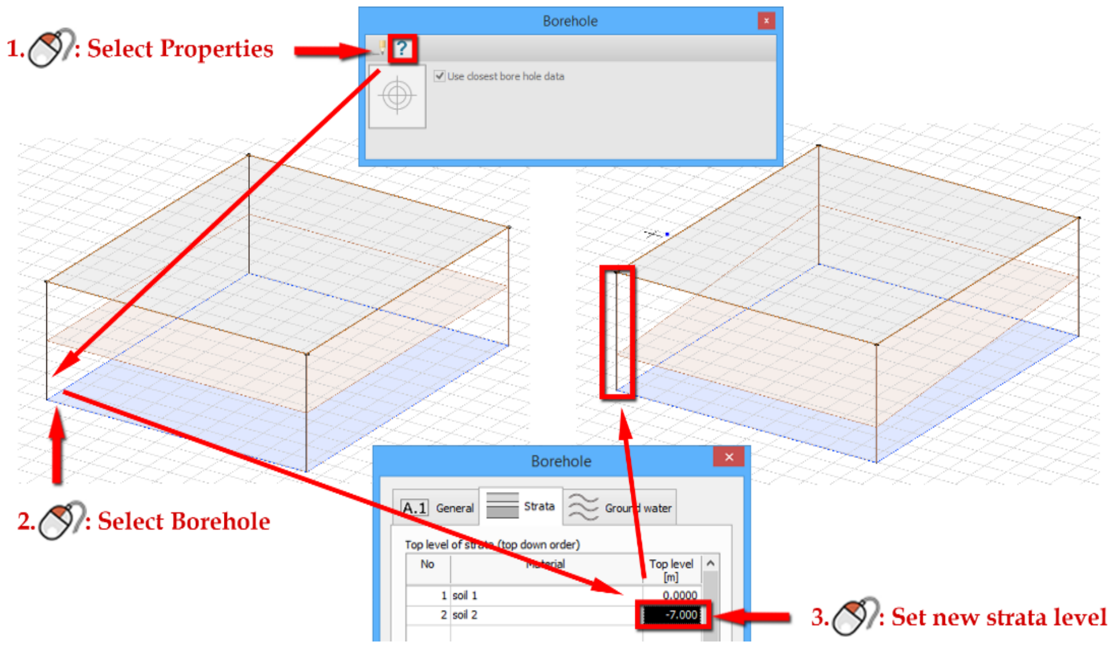
Optional steps:

1. Modify the borehole properties with the  *Properties* tool of the *Borehole* tool palette.
2. Set the display settings of beams at *Settings* > *All* > *Display* > *Soil and Foundation*.



3. The boreholes are stored on “Soil” **Object layers**. At layer settings, the default color and pen width can be set for all beams.

With boreholes the user can define inclined level strata by setting the strata levels.



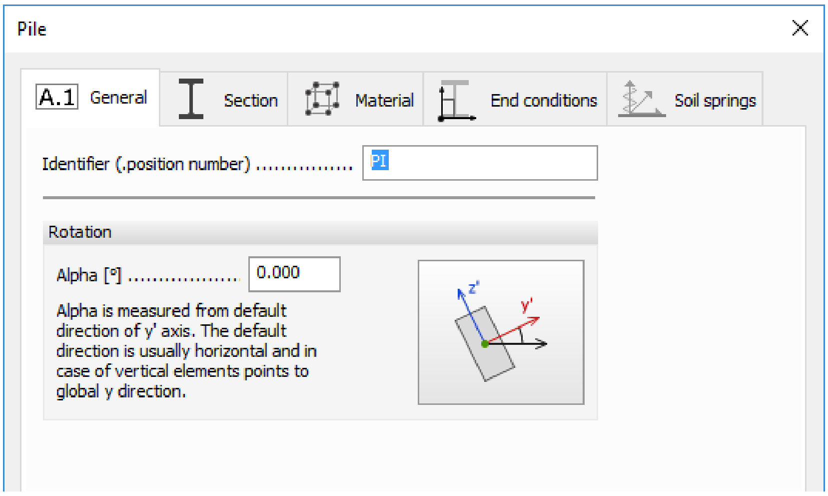
Pile

Pile	Description
Modules where available	
Geometry	Straight, placed in any arbitrary angle but 90° to vertical
Cross-section	Arbitrary concrete, steel, timber, and certain types of composite profiles
Material	Concrete, steel, timber, composite
End condition	Rigid and hinged, or custom defined by **Point-point connection**
Soil springs	Support springs, Negative shaft friction
Load direction	Arbitrary in
Load type	Point/line load (force and moment), Line support motion
Available analysis results	Displacement, stresses, stability and vibration shape in
Available design	-

Table: Pile properties

Definition steps

1. You need to define the soil first.
2. Start *Pile* command from Structure /Foundation tabmenu and choose *Define*.
3. Set the properties of the new *Pile* in *Default settings*:



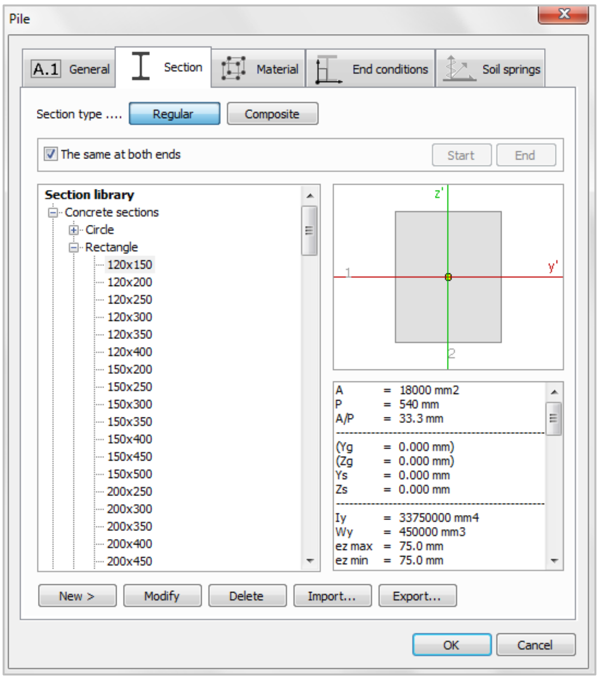
- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (**Display settings**).

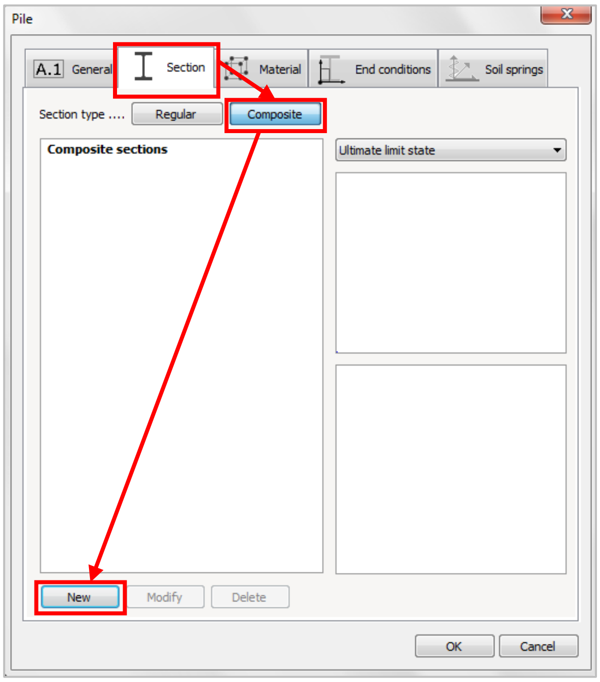
- *Rotation* (General)

  You can set the direction of the local y axis.

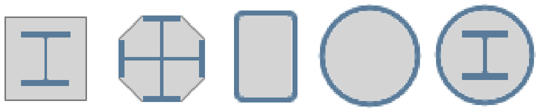
- Cross-section (Section)

The section of the pile can be selected from the *Section library* or you can set a *Composite* one for it. (**Cross-sections**)





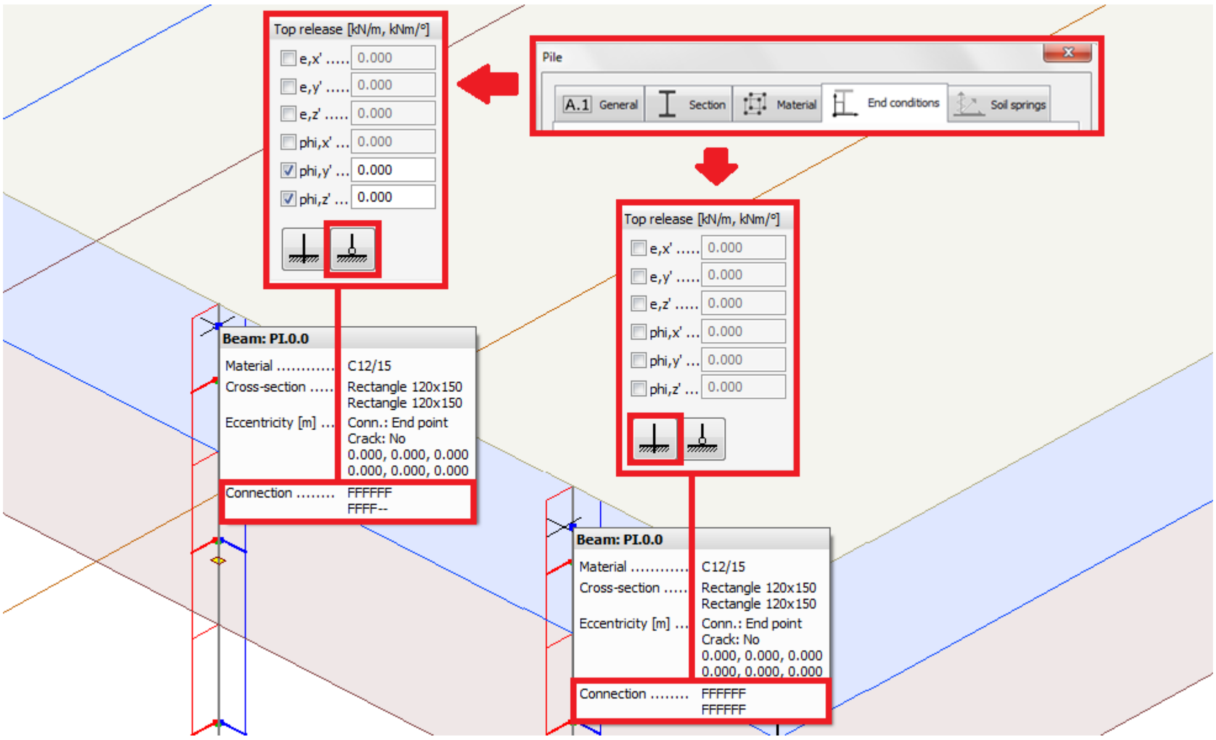
The available composite sections for simple pile are the followings:



- **Material**
  Concrete, steel, timber or composites can be set as material for pile *Analysis*.

- End conditions

  On the *End conditions* tab User can set the top release of the pile, which can be useful in case piles are connected to a foundation slab. The connection can be either fixed or hinged.

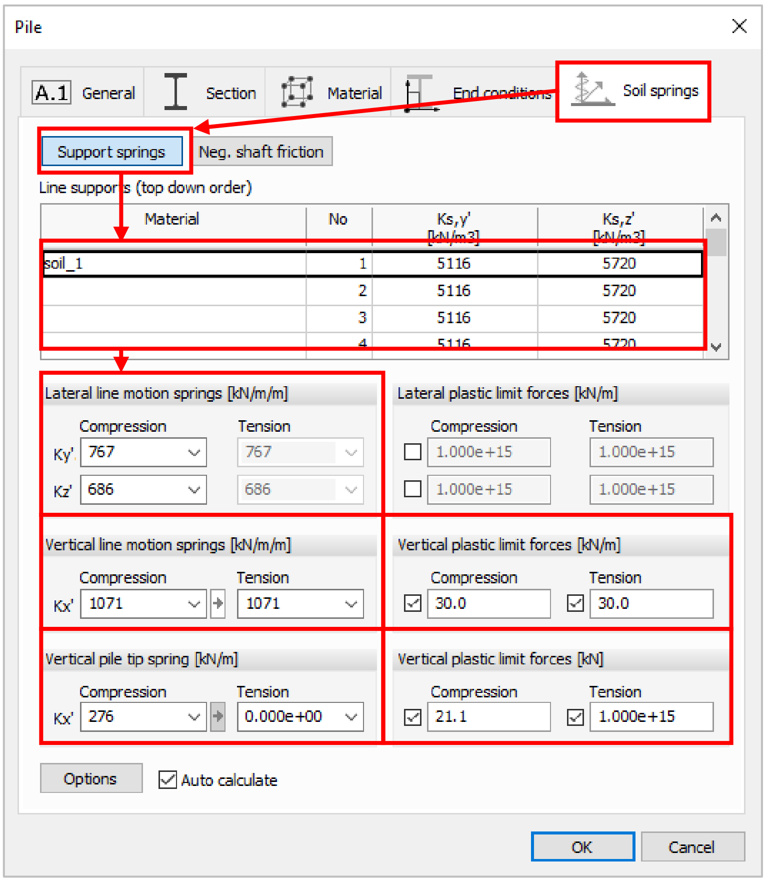


- Support springs (Soil springs)

  In the *Soil springs tab* you can overwrite the automatically calculated values of any support, such as *Lateral/ Vertical line motion springs*, *Vertical pile tip springs* or *Vertical plastic limit forces*.

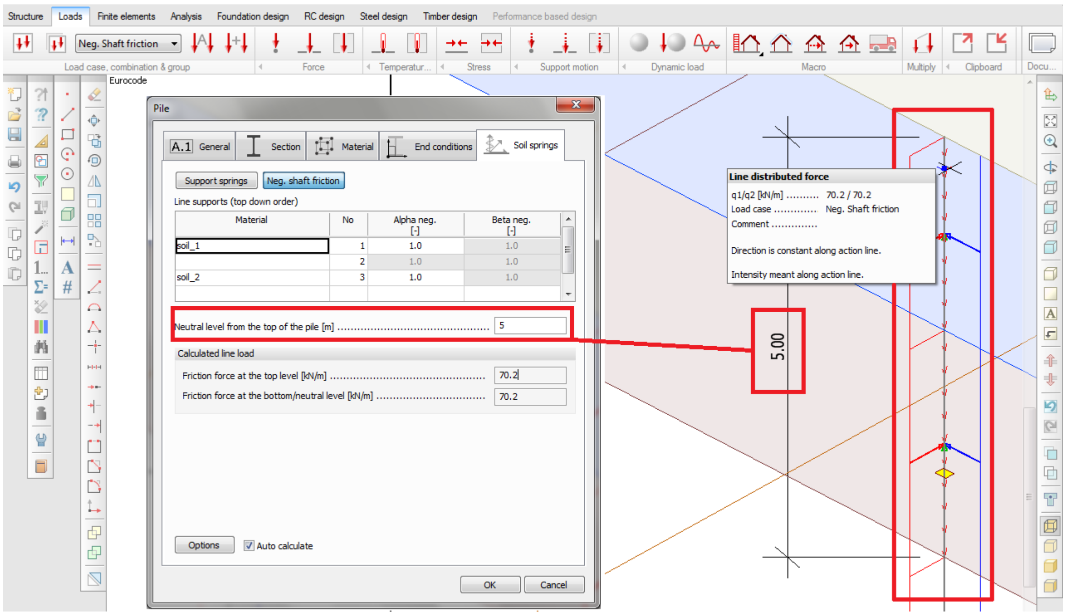
References:

- Bogumił Wrana (2015) Pile load capacity – calculation methods. Studia Geotechnica et Mechanica, Vol. 37, No. 4, pp. 83-93
- NAVFAC DM 7.2 (1984): Foundation and Earth Structures, U.S. Department of the Navy
- Skempton A.W. (1959), Cast-in-situ bored piles in London clay, Geotechnique, Vol. 9, No. 4, pp. 153–173
- Qian-qing Zhang, Shu-cai Li, Fa-yun Liang, Min Yang, Qian Zhang (2014) Simplified method for settlement prediction of single pile and pile group using a hyperbolic model. International Journal of Civil Engineering Vol. 12, No. 2, Transaction B: Geotechnical Engineering, pp. 146-159
- Vesic, A.B. (1963) Beams on Elastic Subgrade and the Winkler's Hypothesis. Proceedings of the 5th International Conference of Soil Mechanics, pp. 845-850

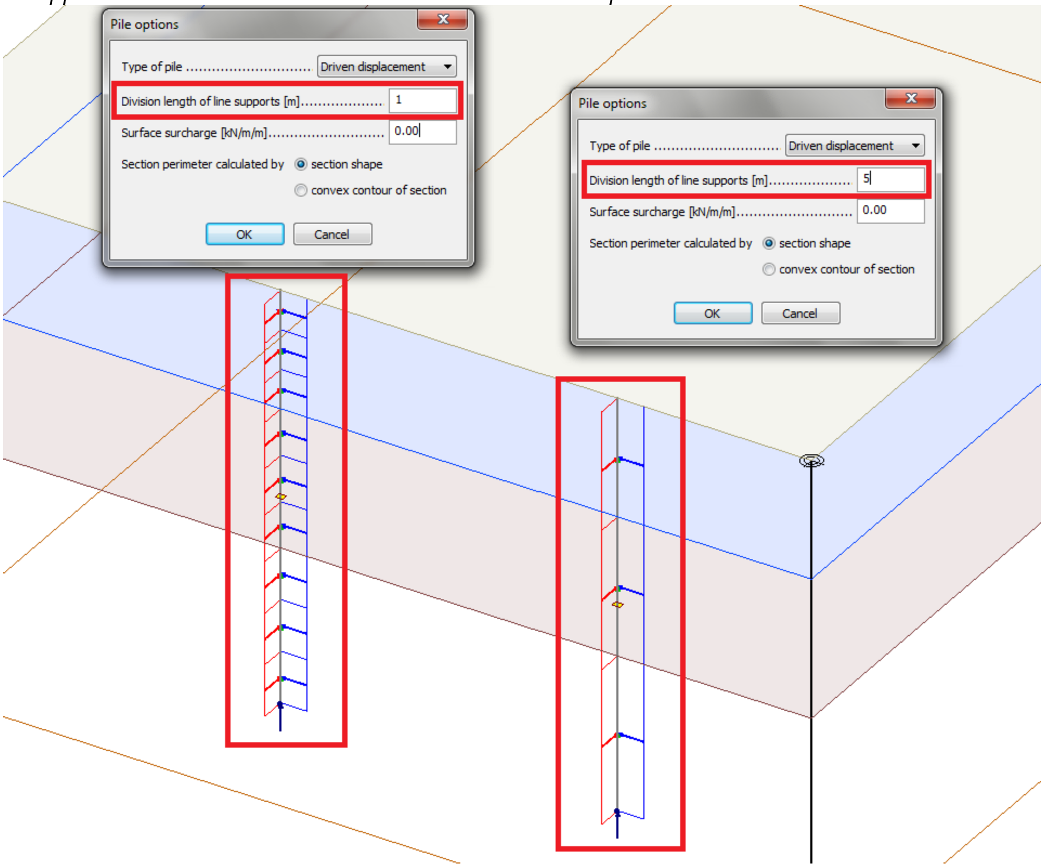


- Negative shaft friction (Soil springs)

  β_{neg} is the factor for negative shaft friction for drained soils (default value is also assumed to 1.0). This load is applied as a linearly variable line load along the pile, and its value can be changed by the modification of the α_{neg} and β_{neg} values (each stratum has one value) together with neutral level:

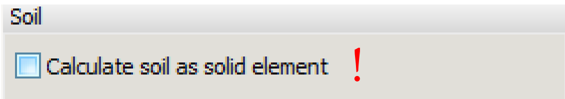


4. Click *Option* to open *Pile option dialog*. Here you can declare the *Type of pile* or the *Surface surcharge* (affecting the vertical stresses in case of drained soils), the *division length of line supports* and can decide the method of the *Section perimeter*'s calculation.

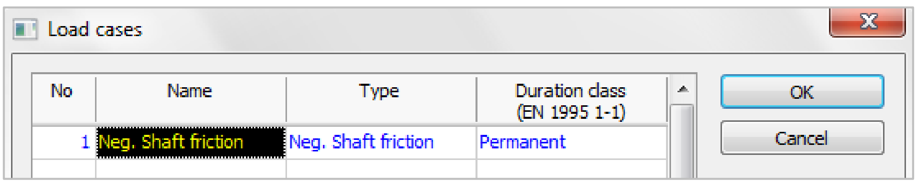


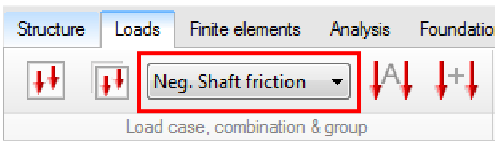
The feature has some important limitations!

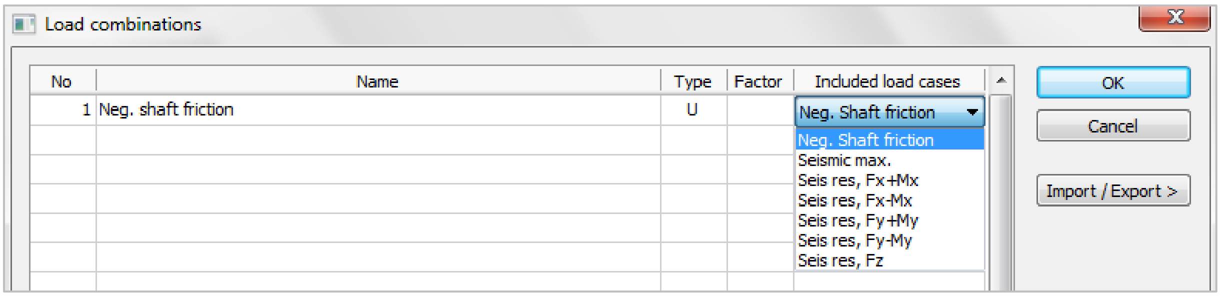
- Pile model is used only for analysis purpose (displacements, internal forces), geotechnical design calculations are not implemented yet.
- During the calculations soil must not be modelled as soil element.



- Negative shaft friction is considered in a load combination only if it includes "Neg. shaft friction" load case, which is generated automatically when a pile is created in the model with non-zero neutral level.







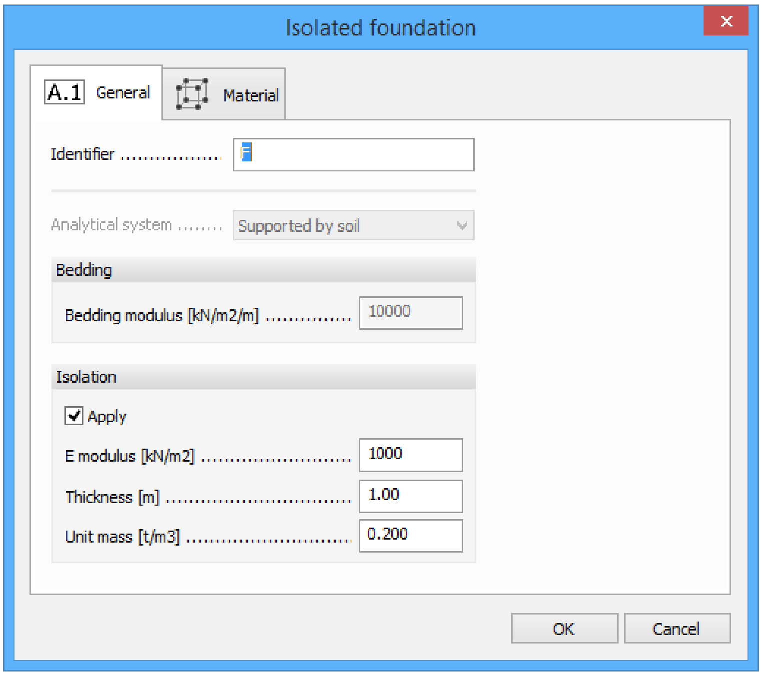
Isolated foundation

Isolated foundation property	Description
Modules where available	
Geometry	Regular shape, Solid
End connection	Rigid and hinged, or custom defined by **Point-point connection**
Material	Concrete
Load direction	Arbitrary in
Load type	Point load (force and moment), Point support motion
Available analysis results	Displacement, stresses, stability and vibration shape in
Available design	**Foundation design**

Table: Isolated foundation properties

Definition steps

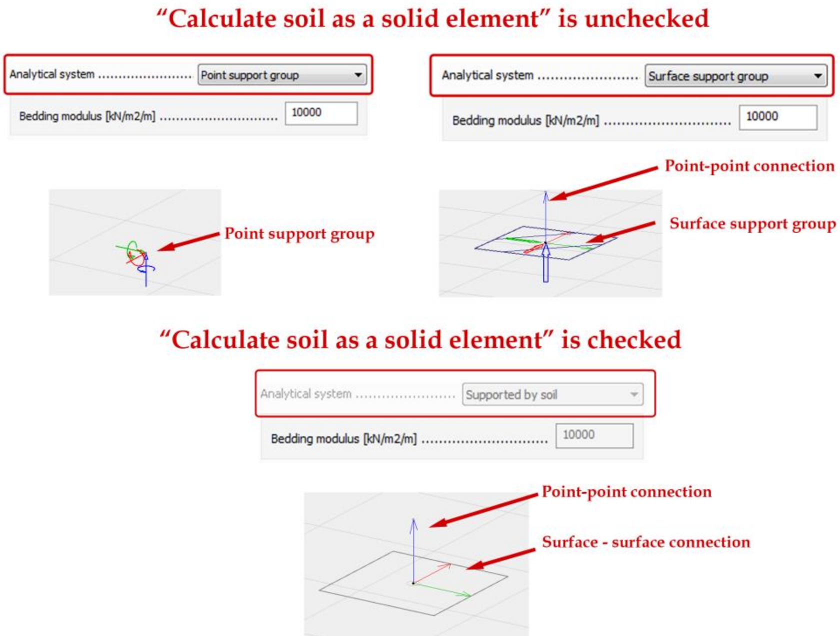
1. If needed, set a proper position for the **working plane**.
2. Start *Isolated foundation* command from Structure tabmenu and choose *Define*.
3. Set the properties of the new *Isolated foundation* at *Default settings*:



- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (**Display settings**).

- *Analytical system* (General)

  Here the user can choose how to model the *Isolated foundation* depends on if *"Calculate soil as a solid element"* is chose or not. If *"Calculate soil as a solid element"* is not chosen in *"Code, Configuration"* dialog, user can choose if he/she wants to model the *Isolated foundation* by line support group or by surface support group. In this case bedding modulus can be set in *Default settings / General* dialog.



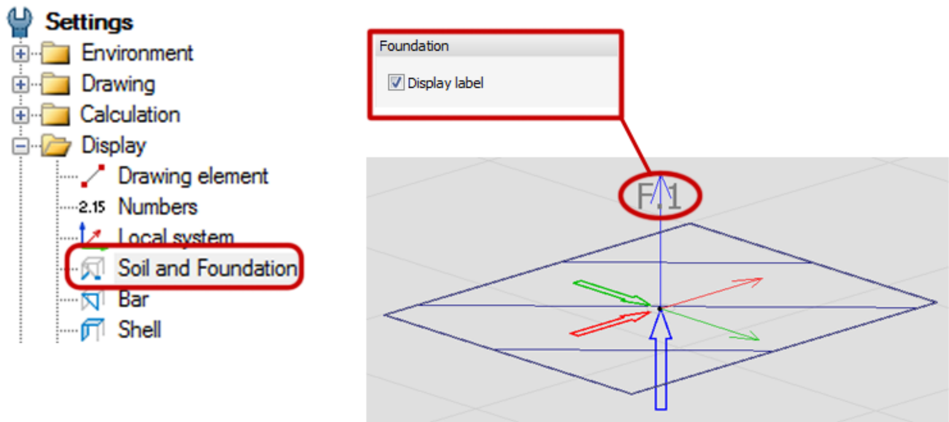
- *Insulation* (General)
  User can apply insulations attached to the bottom of the isolated foundation. If **Calculate soil as a solid element** is activated, Insulations are calculated as part of the Foundation object with the user-specified parameters. Otherwise, they are ignored.

- **Material**
  Only concrete material can be set for isolated foundation *Analysis*.

4. Choose a **geometry** definition method.
5. Define the Wall foundation in the model view based on the chosen geometry method.

Optional steps:

6. Modify the isolated foundation properties with the *Properties* tool of the *Isoltated foundation* tool palette.
7. Set the display settings of isolated foundations at *Settings* > *All* > *Display* > *Beam, Column and Truss*.



8. The Isolated foundations are stored on "Foundation" and if they modeled with *Surface support group* their supports will be stored at the "Support" **Object layers**. The color and pen width settings by selected beam elements can be modified by *Edit* > **Properties** > *Change appearance*.

Wall foundation

Wall foundation property	Description
Modules where available	
Axis position	Arbitrary (horizontal, vertical and skew)
Geometry	Straight and curved
Cross-section	Arbitrary profile
Eccentricity	Available
End connection	Rigid and hinged, or custom defined by **Point-point connection**
Material	Concrete

Load direction	Arbitrary in 
Load type	Point load (force and moment), Line load (force and moment), Line temperature variation load, Line stress load, Line support motion
Available analysis results	Displacement, stresses, stability and vibration shape in 
Available design	**Foundation design**

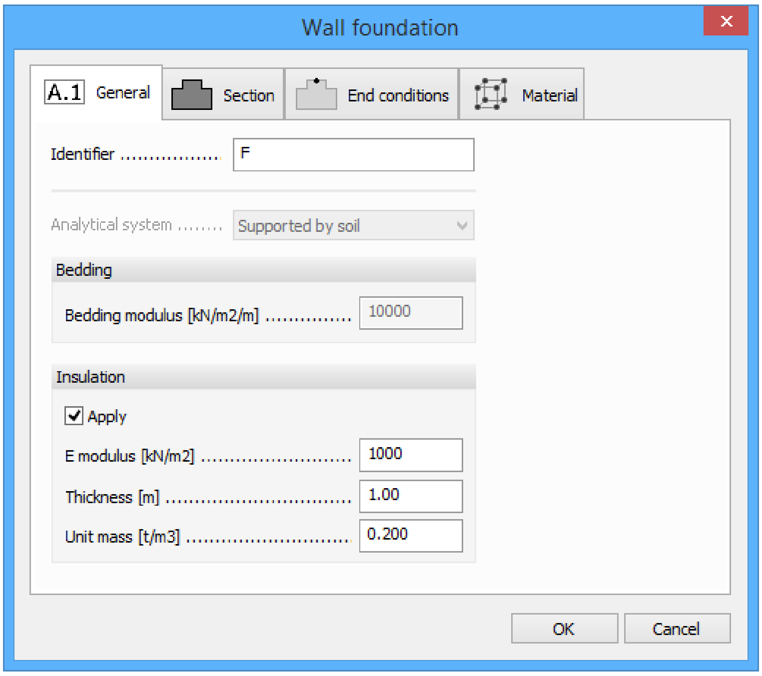
Table: Wall foundation properties

Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**.

2. Start *Wall foundation* command from Structure tabmenu and choose *Define*.

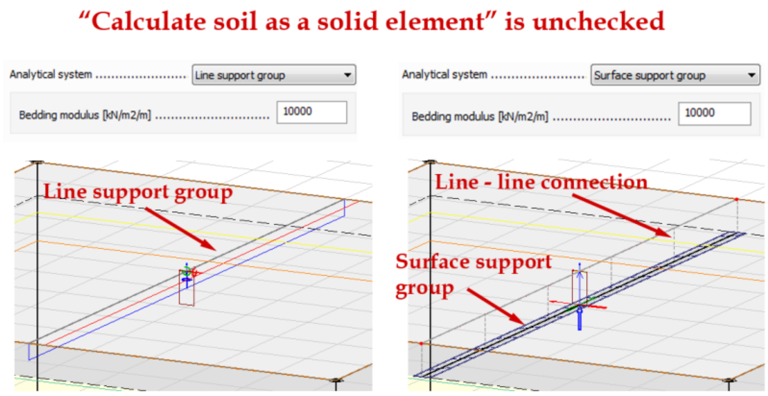
3. Set the properties of the new *Wall foundation* at *Default settings*:

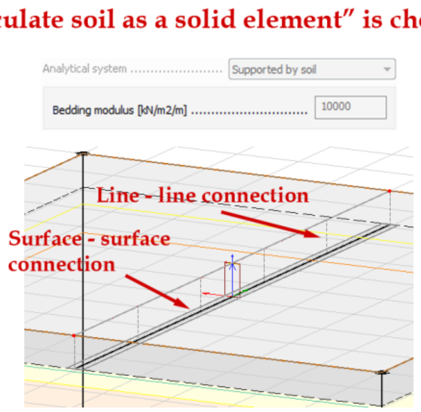


- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (**Display settings**).

- *Analytical system* (General)

  Here the user can choose how to model the *Wall foundation* depends on if *"Calculate soil as a solid element"* is chose or not. If *"Calculate soil as a solid element"* is not chosen in *"Code, Configuration"* dialog, user can choose if he/she wants to model the Wall foundation by line support group or by surface support group. In this case bedding modulus can be set in *Default settings / General* dialog.





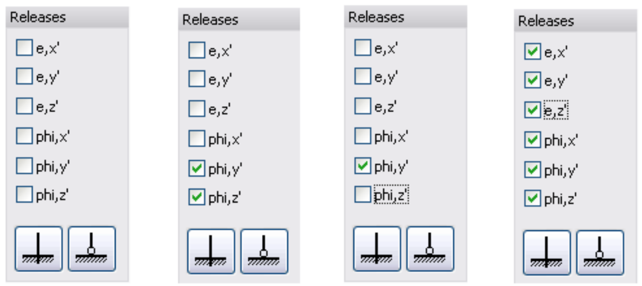
- *Insulation* (General)
  User can apply insulations attached to the bottom of the wall foundation. If **Calculate soil as a solid element** is activated, Insulations are calculated as part of the Foundation object with the user-specified parameters. Otherwise, they are ignored.

- **Section**
  Choose a profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (**Display settings**).

- *End conditions*
  The end connections can be set under the *End conditions* tab similarly to **Beams**. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**). Rigid connections can be customized to hinged or semi-rigid (spring) with additional **Point-point connection**.

  The end connections (in the beam start and end) with joined objects can be set to rigid or hinged by motion and rotation components. The component directions are valid in the beam's **local co-ordinate system**.

  Inactive component means a fixed motion/rotation, while a checked component behaves as free motion/rotation in the assigned direction.

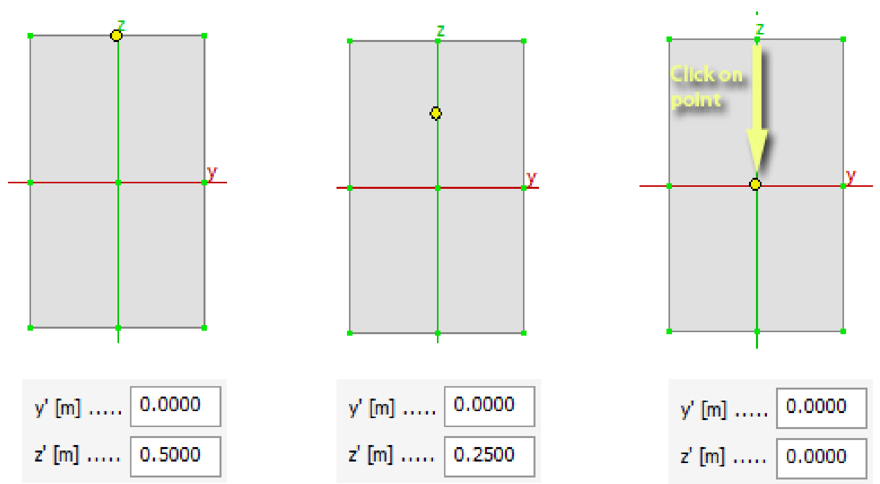


Fully rigid Hinged in 3D Hinged in 2D Free end

Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**).

- *Eccentricity*
  Eccentricity has effect in calculation and it can be turn on/off with *Consider eccentricity in calculation* option. The default position (center of gravity) of the beam axis (local x' axis) can be set in the aspect of analysis/design calculation or accurate display.

The axis position (symbolized with yellow point in the section) can be set with its coordinates in the local y'z' plane, or can be placed into special section points (e.g. plate corners) by dragging the yellow point into the requested position.



Default: Eccentricity at top Position modified by distance Position modified by mouse

Two type of eccentricity option can be chosen:

First type ("End releases applied at the ends of the theoretical axis") is typically used for modeling RC bars which work together with concrete slabs (slab normal forces transfers to the bar).

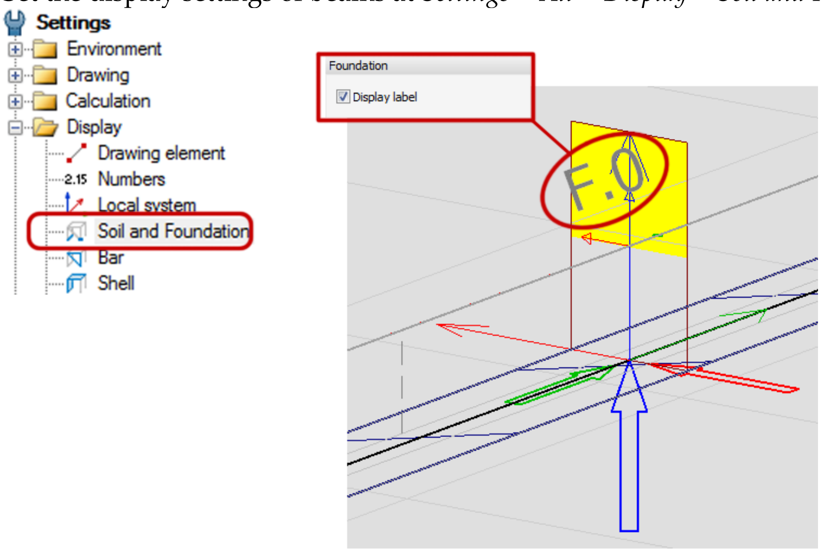
The second type ("End releases applied at the ends of the gravity (physical axis)") is for modeling single plate and bars which are not working together.

- **Material**
  Only concrete material can be set for isolated foundation *Analysis* and *Design*.

4. Choose a **geometry** definition method for the wall foundation reference line (axis).
5. Define the Wall foundation in the model view based on the chosen geometry method.

Optional steps:

6. Modify the geometry with the *Edit* menu commands valid for line elements.
7. Modify the bar properties with the *Properties* tool of the *Wall foundation* tool palette.
8. Set the display settings of beams at *Settings* > *All* > *Display* > *Soil and Foundation*.



9. The Wall foundations are stored on "Foundation" and if they modeled with *Surface support group* their supports will be stored at the "Support" **Object layers**. The color and pen width settings by selected beam elements can be modified by *Edit* > **Properties** > *Change appearance*.

Beam

Beam property	Description
Modules where available	
Axis position	Horizontal in
	Arbitrary (horizontal, vertical and skew) in
Geometry	Straight and curved
Cross-section	Arbitrary profile
Eccentricity	Available
End connection	Arbitrary
Material	Steel, concrete, timber and general
Load direction	Vertical in
	Arbitrary in

Load type	All point and line load
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in
Available design	**Steel design** in
	RC design in
	Timber design in
Alternative	Steel beam can be modeled/designed as the set of shells (**Shell model**)

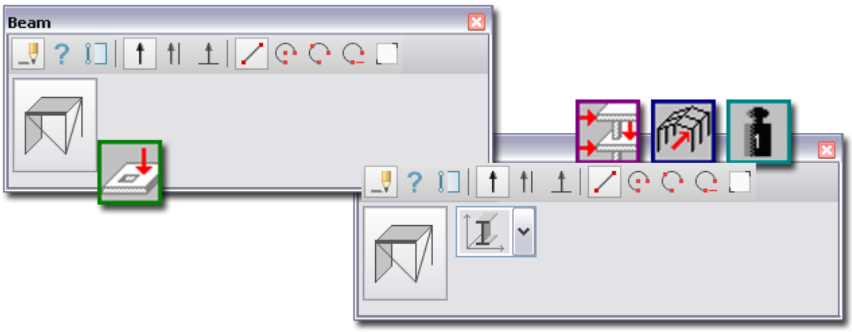
Table: Beam properties

Typical 2D and 3D frame structures build by beams (only) can be fast defined with the so-called **Wizard** tool. Fast means, that in a dialog you can set the beam sizes and positions by parameters. Besides the geometry, Wizard adds loads and supports to the beam-system.

Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**. In the 3D design modules, respect that the gravity direction is always the Global Z axis direction.

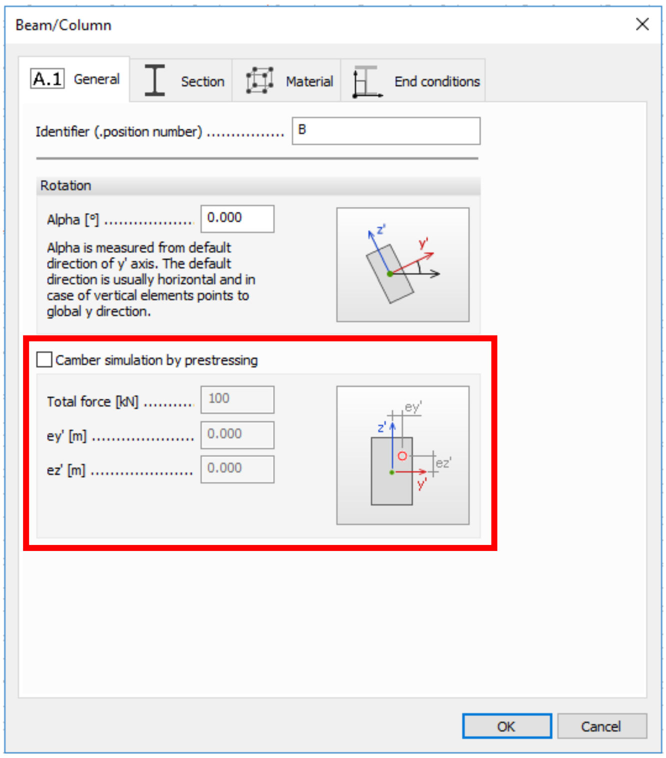
2. Start *Beam* command from Structure tabmenu and choose *Define*.



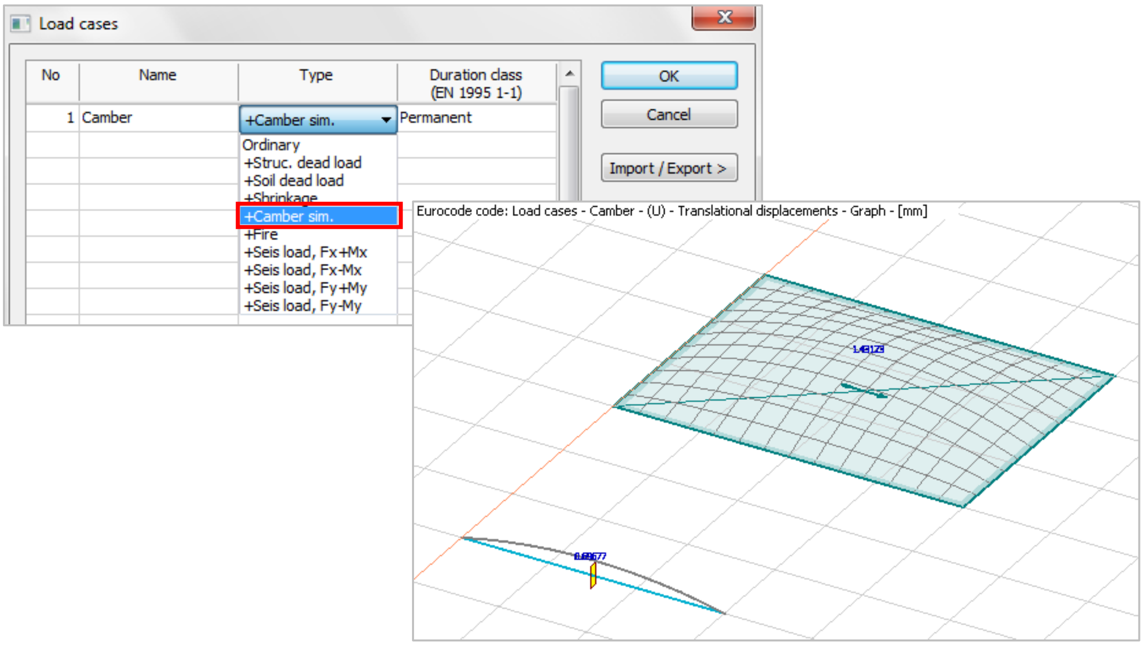
3. Set the properties of the new beam at *Default settings*:

- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Camber simulation by prestressing* (General)
  To simulate camber of the elements check the box next to *Camber simulation by prestressing* in the *General tab.*



To calculate camber a camber-type load case needs to be defined. The effect is calculated as a kinematic load.



⚠ Camber calculation gives statically correct result only for non-eccentric, hinged beams and shells. This only corrects the displacement result and no internal forces will rise in the model.

- **Section**
  Choose a beam profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (**Display settings**).

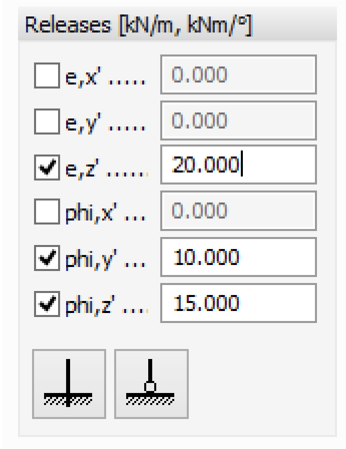
- *End conditions*

In , the end connections are fully rigid.

In 3D modules, the end connections can be set under the *End conditions* tab. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**). End connections can be fully customised for bar ends by setting the rigidity of the displacement components.

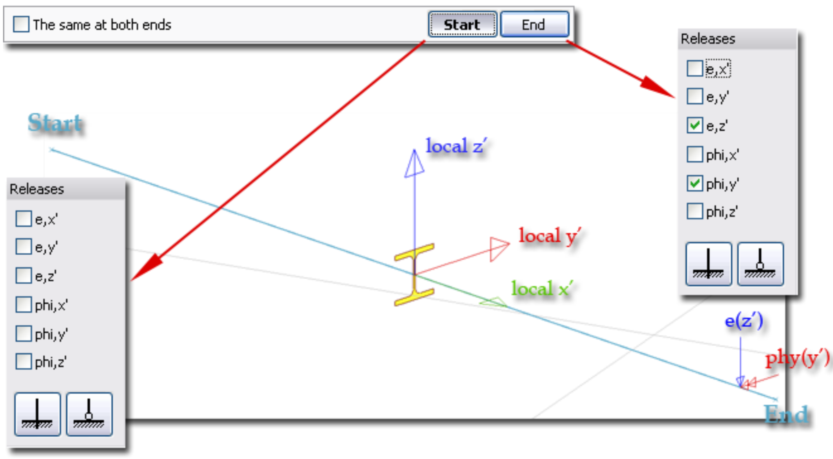
The component directions are valid in the beam's **local co-ordinate system**.

Inactive component means a fixed motion/rotation, while a checked component behaves according to the user-specified rigidity value.



With the „Rigid" and the „Hinged" buttons pre-defined end condition values can be applyed to the bar ends.

The local x' axis always points the beam end, so the start and end point position can be followed from the beam's local co-ordinate system.



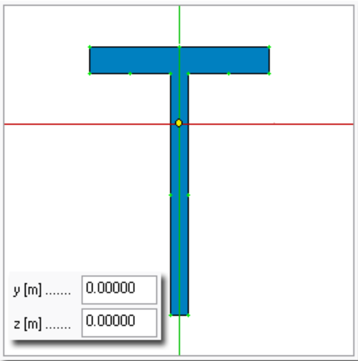
Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**).

The connection settings is not available in , the end connections are always fully rigid.

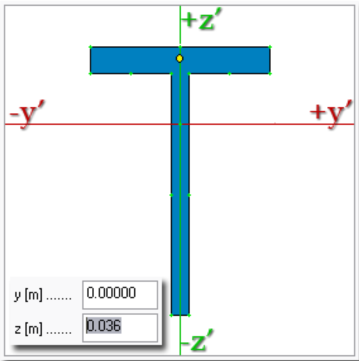
Eccentricity in analytical model (End conditions)

Eccentricity has effect in calculation. The default position (center of gravity) of the beam axis (local x′ axis) can be set in the aspect of analysis/design calculation.

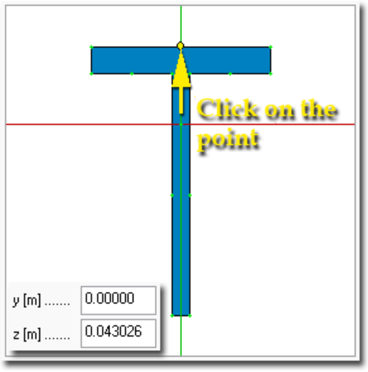
The axis position (symbolized with yellow point in the section) can be set with its coordinates in the local y′z′ plane, or can be placed into special section points (e.g. plate corners) by dragging the yellow point into the requested position.



Default: "center of gravity"



Position modified by distances



Position modified by the mouse

Two type of eccentricity option can be chosen:

First type ("End releases applied at the ends of the theoretical axis") is typically used for modeling RC bars which work together with concrete slabs (slab normal forces transfers to the bar).

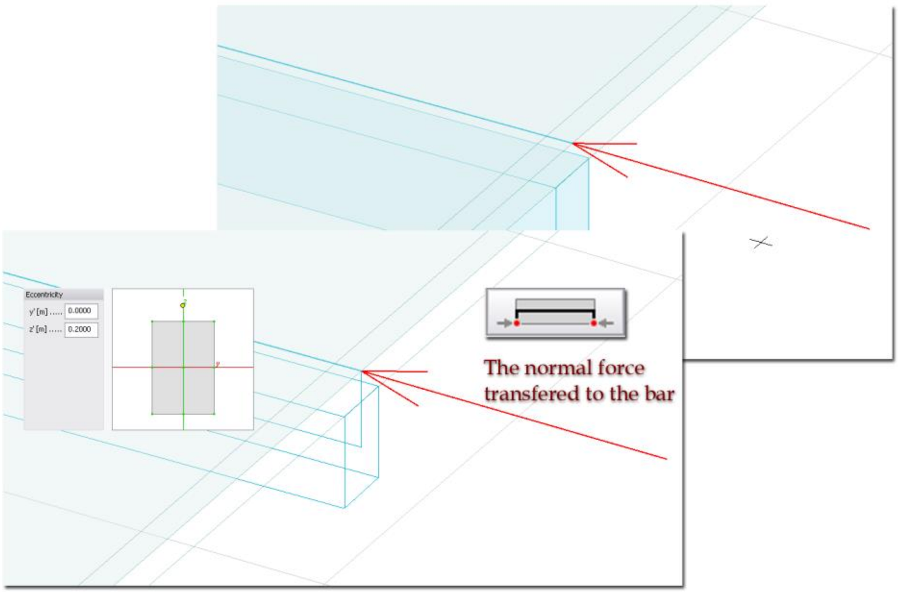


Figure: Modeling of a "ribbed concrete slab"

The second type ("End releases applied at the ends of the gravity (physical axis)") is for modeling single plate and bars which are not working together.

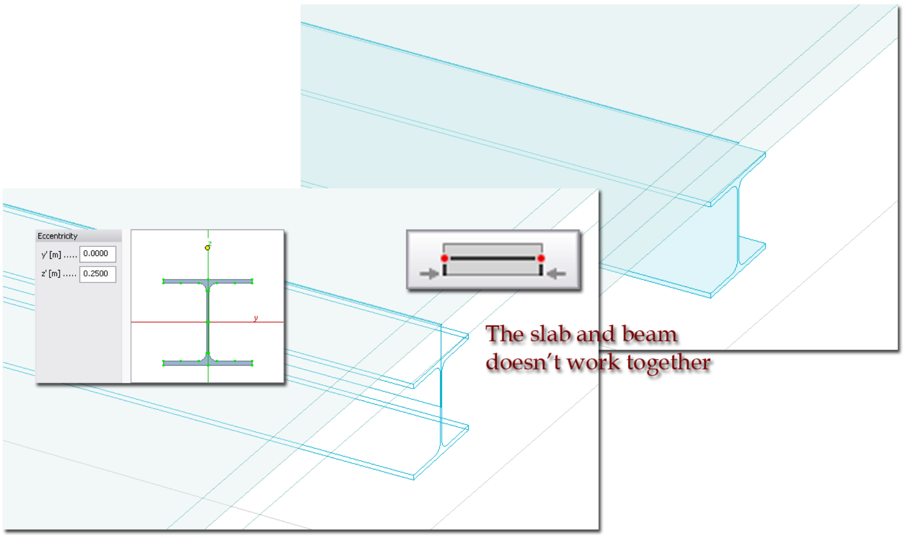


Figure: Modeling of a steel beam under slab in 3D Structure module

Physical model

Eccentricity of the physical model can be set in the Physical model tab, in a similar way as the **Eccentricity of analytical model**. It has no effect on the calculation.

- **Material**
  Any type of materials can be set for beam *Analysis*, but design can be run for concrete, steel and timber beams only. The applied material name can be displayed in model view (**Display settings**).

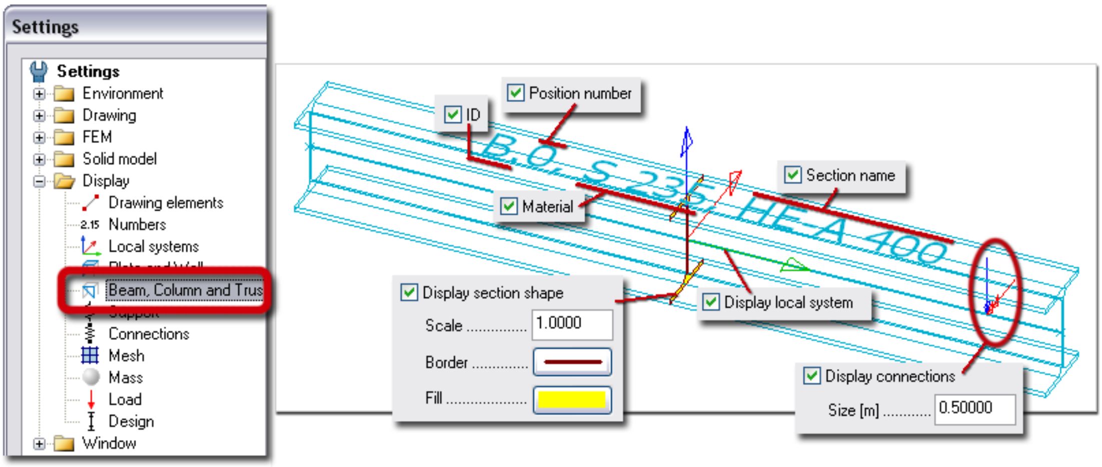
4. Define the **direction** of the cross-section y’ axis (only in 3D modules).

The y’ axis cannot be modified in the module, so it is always parallel with the Global X-Y (working).

5. Choose a **geometry** definition method for the beam reference line (axis).
6. Define the beam in the model view based on the chosen geometry method.

Optional steps:

7. Modify the geometry with the *Edit* menu commands valid for line elements.
8. Modify the bar properties with the *Properties* tool of the *Beam* tool palette.
9. Set the display settings of beams at *Settings* > *All* > *Display* > *Beam, Column and Truss*.



10. The beams are stored on “Beams” **Object layers**. At layer settings, the default color and pen width can be set for all beams. The color and pen width settings by selected beam elements can be modified by *Edit* > **Properties** > *Change appearance*.

Column

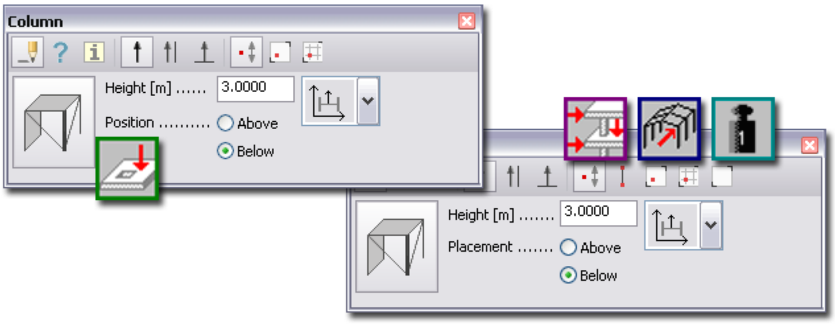
Column property	Description
Modules where available	
Function	Point support in
	Column in
Position	Always vertical
Geometry	Straight
Cross-section	Arbitrary profile
Eccentricity	Available
End connection	Arbitrary
Material	Steel, concrete, timber and general
Load direction	Arbitrary in
Load type	All point and line load in
Available analysis results	Reaction forces in
	Displacement, internal forces, stresses, stability and vibration shape in
Available design	**Steel design** in
	RC design in (Column punching is available in)

	Timber design in
Alternative	Steel column can be modeled/designed as the set of shells (**Shell model**)

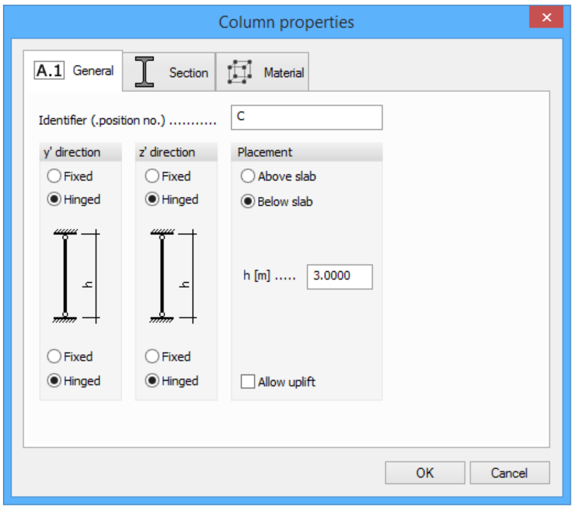
Table: Column properties

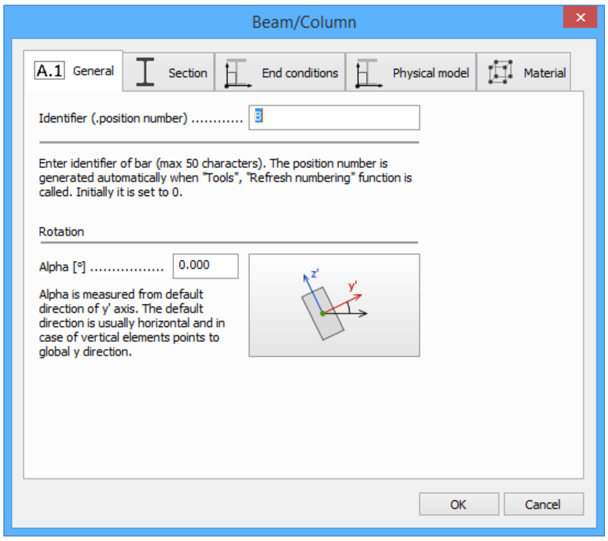
Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**.
2. Start *Column* command from Structure tabmenu and choose *Define*.



3. Set the properties of the new column at *Default settings*:





- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Allow uplift* (General)
  Available in only. "Uplift" behavior can be modeled by activating this option. It works in case of tensional reaction force.

  In 3D modules, "uplift" can be modeled with non-linear **Point support** settings.

- **Section**

Choose a column profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (**Display settings**).

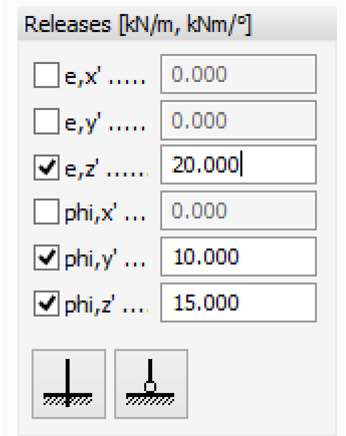
- **End conditions**

In , the support conditions (*General*) of the top and bottom ends can be set to fully rigid or fully hinged. Because *Columns* are point supports further point support is not needed to define at *Column* end.

In 3D modules, the end connections can be set under the *End conditions* tab. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**). End connections can be fully customised for bar ends by setting the rigidity of the displacement components.

The component directions are valid in the beam's **local co-ordinate system**.

Inactive component means a fixed motion/rotation, while a checked component behaves according to the user-specified rigidity value.



-

With the „Rigid" and the „Hinged" buttons pre-defined end condition values can be applied to the bar ends.

Eccentricity of analytical model (End conditions)

In 3D modules, eccentricity of the analytical model can be set in the *End conditions* tab similarly to **Beams**. The default position (center of gravity) of the column axis (local x' axis) can be set in the aspect of analysis/design calculation.

- **Physical model**

Eccentricity of the physical model can be set in the Physical model tab, in a similar way as the **Eccentricity of analytical model** for **Beams**. It has no effect on the calculation.

- **Material**

Any type of materials can be set for column *Analysis*, but design can be run for concrete, steel and timber columns only. The applied material name can be displayed in model view (**Display settings**).

4. Define the ***direction*** of the cross-section y' axis.
5. Choose a method for insertion point definition (**geometry**).
6. Place the column in the model view based on the chosen geometry method.

Optional steps:

7. Modify the bar properties with the  *Properties* tool of the *Column* tool palette.
8. Set the **display settings** of columns at *Settings* > *All* > *Display* > *Beam, Column and Truss* similarly to *Beams*.

9. The columns are stored on “Columns” **Object layers**. At layer settings, the default color and pen width can be set for all columns. The color and pen width settings by selected column elements can be modified by *Edit* > **Properties** > *Change appearance*.

10. In *Plate* module, the *Info* tool displays the support stiffness (motion and rotation) values of the column ends valid in the column **local co-ordinate system**.

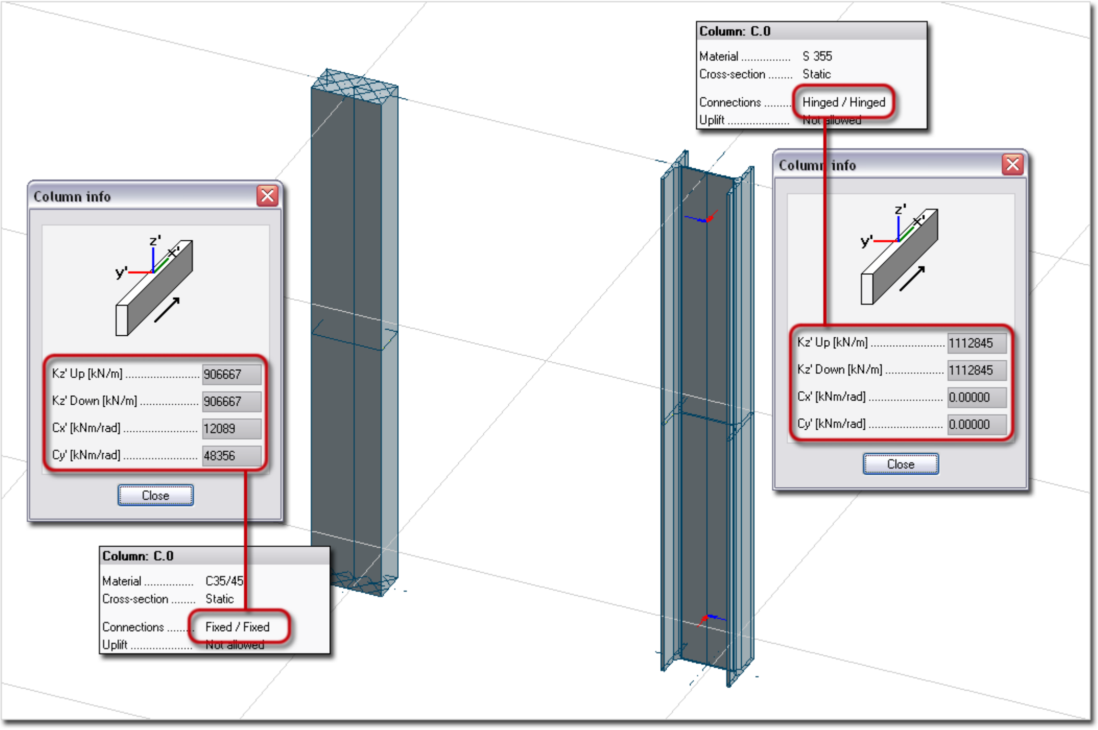
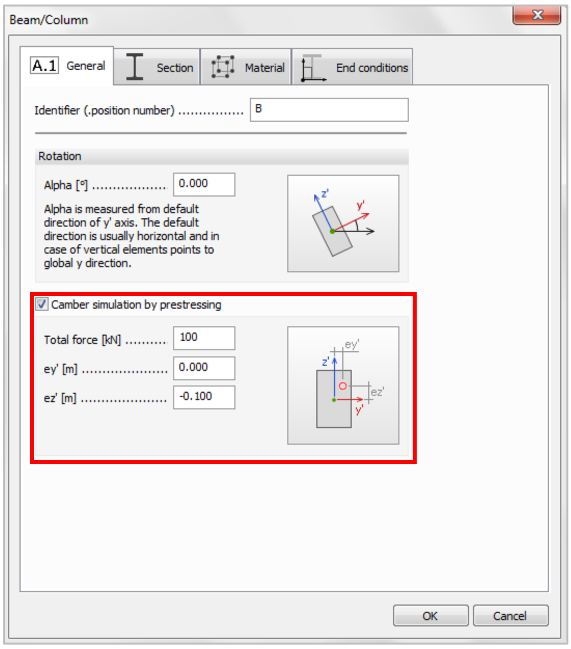


Figure: Support stiffness of Columns in Plate module

11. To simulate camber of the elements check the box next to *Camber simulation by prestressing* in the *Default setting/General tab.*



Truss Member

Truss member property	Description
Modules where available	
Function	It models bars with limited compression and tension-only bars like cables, pull rods, diagonal braces etc.
Axis position	Arbitrary (horizontal, vertical and skew)
Geometry	Straight
Cross-section	Arbitrary profile
Connection	- Always hinged (cannot be modified) - It can be connected to other elements only at its ends - It does not intersect other elements cross it visually
Material	Steel, concrete, timber and general
Load direction	Axial
Load type	- Dead load (will be distributed to the end nodes) - Uniform line temperature variation - Shrinkage - Line stress load - Point mass (will be distributed to the end nodes)
Analysis	Displacement, normal force and stress
Available design	**Steel design** in
	RC design in
	Timber design in
Alternative	Steel truss member can be modeled/designed as the set of shells (**Shell model**)

Table: Truss member properties

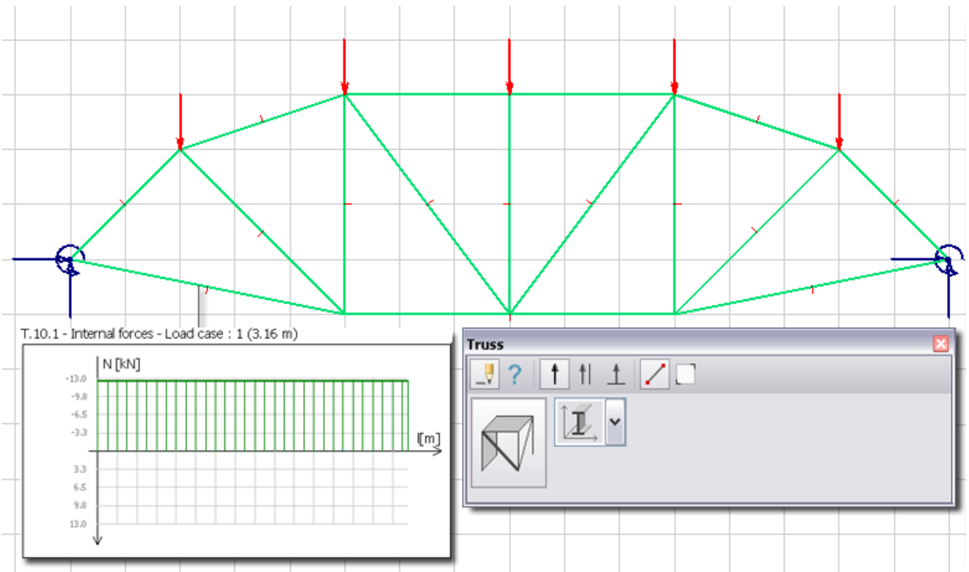
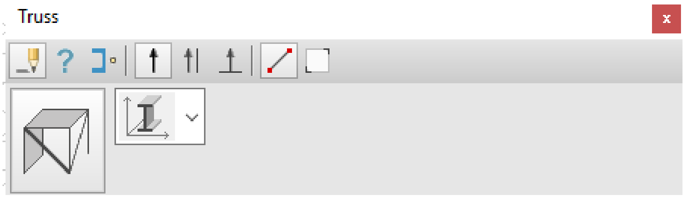


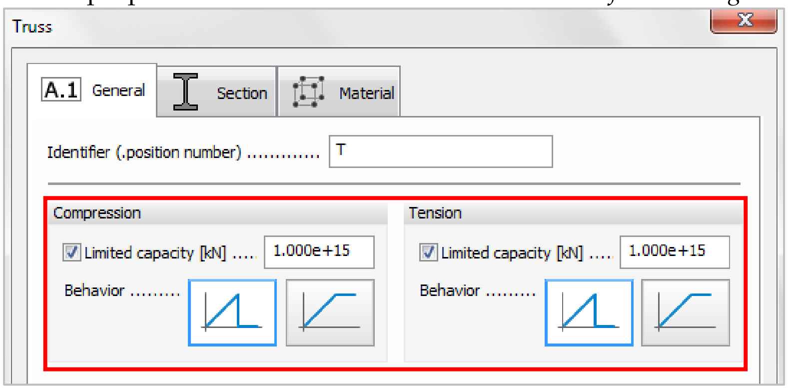
Figure: Modeling a truss

Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**. In the 3D design modules, respect that the gravity direction is always the Global Z axis direction.
2. Start *Truss member* command from Structure tabmenu and choose *Define*.



3. Set the properties of the new truss element at *Default settings*:



- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Non-linear behaviour* (General)
  Limited capacity and *Brittle/Plastic* can be set for both compression and tension. These options are considered only for load combinations calculated as non-linear elastic. *Plastic* behaviour is only considered for load combinations calculated as non-linear elastic + plastic.

- **Section**
  Choose a profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (**Display settings**).

- **Material**
  Any type of materials can be set for beam *Analysis*, but design can be run for concrete, steel and timber elements only. The applied material name can be displayed in model view (**Display settings**).

4. Define the **direction** of the cross-section y' axis.
5. Choose a **geometry** definition method for the truss element reference line (axis).
6. Define the element in the model view based on the chosen geometry method.

Optional steps:

7. Modify the bar properties with the *Properties* tool of the *Truss member* tool palette.

8. Set the **display settings** of truss elements at *Settings* > *All* > *Display* > *Beam, Column and Truss* similarly to *Beams*.
9. The truss elements are stored on "Truss members" **Object layers**. At layer settings, the default color and pen width can be set for all truss elements. The color and pen width settings by selected truss elements can be modified by *Edit* > **Properties** > *Change appearance*.

Fictitious Bar

Fictitious bar property	Description
Modules where available	
Function	Bar elements with given stiffness values
Axis position	Horizontal in
	Arbitrary (horizontal, vertical and skew) in
Geometry	Straight and curved
End connection	Rigid and hinged, or custom defined by **Point-point connection**
Stiffness	Editable (but, material and cross-section definition is not available)
Load direction	Vertical in
	Arbitrary in
Load type	All point and line load
Available analysis results	Displacement and internal forces

Table: Fictitious bar properties

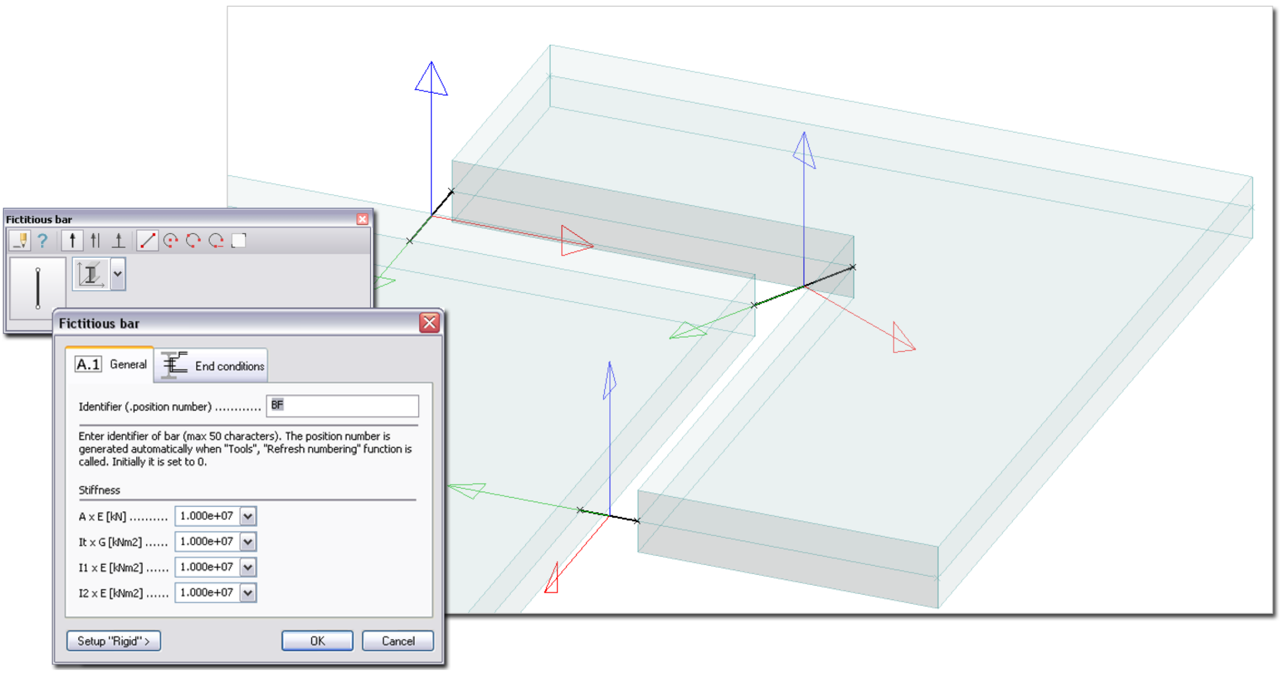
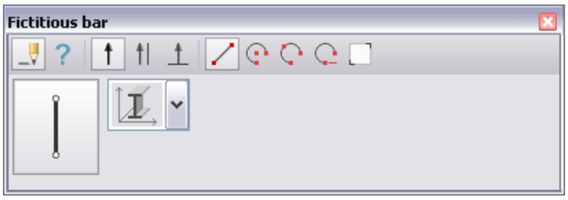


Figure: Fictitious bars connect plate elements

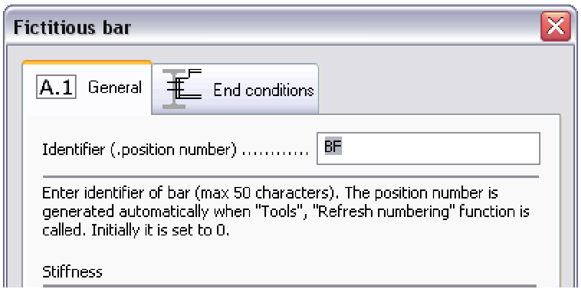
The program automatically generates "rigid" fictitious bars in the end points of "bars" modeled as a shell model (end edges of shell regions; see **Shell model**).

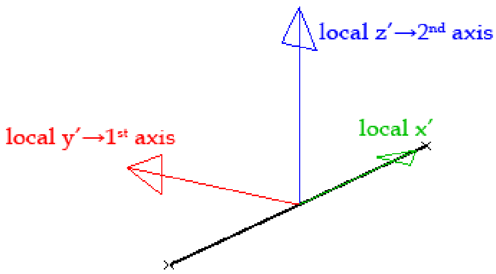
Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**.
2. Start *Fictitious bar* command from Structure Tabmenu and choose *Define*.



3. Set the properties of the new bar at *Default settings*:





- *Stiffness* (General)
  EA, GI_t, EI_1 and EI_2 stiffness values can be assigned to bars. No definition of material and cross-section is needed. The local y' axis represents the 1st principal axis, and the local z' axis defines the 2nd principal axis.

  The value of rigid stiffness ("infinite" rigidity) can be defined and set as project default at *Settings* > *Calculation*> *"Rigid" values* > *Fictitious bar*.

- *Connections*
  In , the end connections are fully rigid.

  In 3D modules, the end connections can be set under the *Connections* tab similarly to **Beams**. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**).

4. Define the **direction** of the cross-section 1st principal axis (local y' axis; only in 3D modules).
   The y' axis cannot be modified in the module, so it is always parallel with the Global X-Y (working).

5. Choose a **geometry** definition method for the bar reference line (axis).
6. Define the bar in the model view based on the chosen geometry method.

Optional steps:

7. Modify the geometry with the *Edit* menu commands valid for line elements.

8. Modify the bar properties with the *Properties* tool of the *Fictitious bar* tool palette.
9. The fictitious bars are stored on "Modeling tools" **Object layers**. At layer settings, the default color and pen width can be set for all fictitious bars.

Planar Objects

This chapter summarizes all features and properties of planar objects: **Foundation slab**, **Plate**, **Wall**, **Shell model elements**, **Fictitious shell**, **Timber panel** and **Profiled plate/wall**.

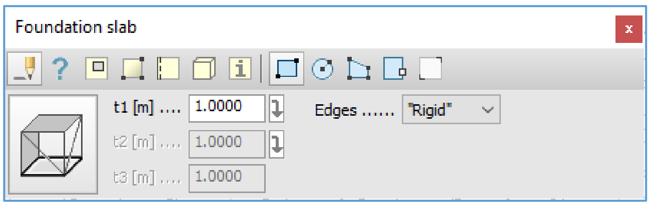
Foundation slab

Foundation slab property	Description
Modules where available	
Reference plane position	Horizontal
Geometry	Any shape
Thickness	Constant or variable
Hole	Available
Eccentricity	Available for display and calculation
Edge connection	Rigid elastic (spring) linear or non-linear
Material	Concrete
Orthotropic feature	Available
Load direction	Vertical
Load type	Point load (force and moment), Line load (force and moment), Surface load, Surface temperature variation load, Surface stress load, Surface support motion
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape
Available design	**Foundation design**

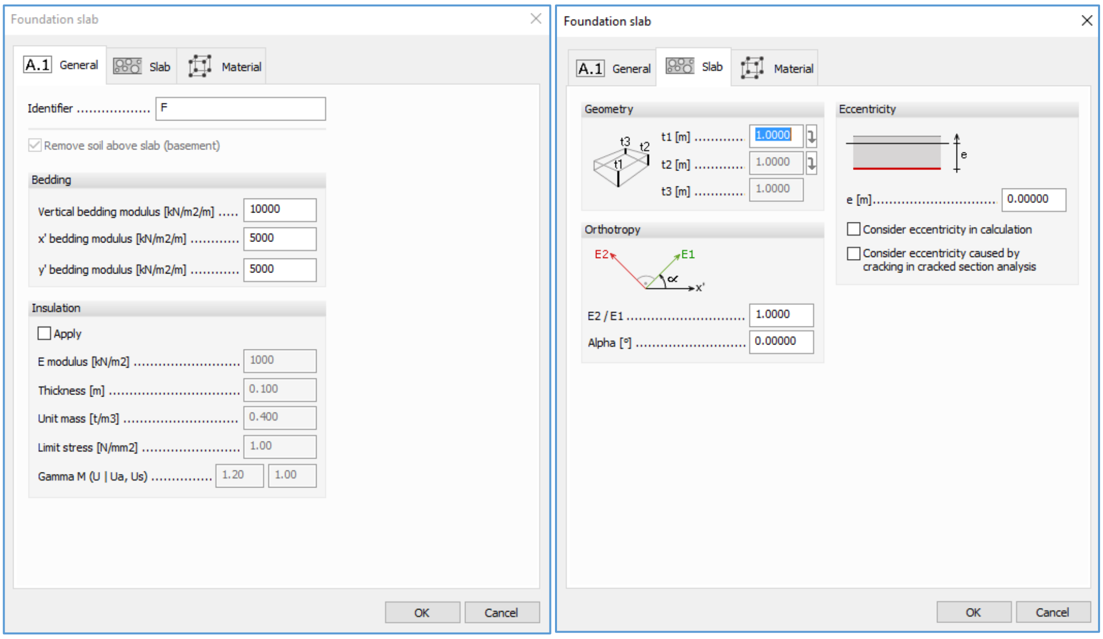
Table: Foundation slab properties

Definition steps

1. If needed, set a proper position for the **working plane**.
2. Start *Foundation slab* command from Structure tabmenu and choose *Define*.



3. Set the properties of the new plate at *Default settings*:



- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID) number can be displayed in model view (**Display settings**).

- *Bedding* (General)
  The horizontal one can be specified by the User. These three values define the motion spring stiffness of the automatically created surface support group under the foundation slab. By default, the horizontal values are the half of the vertical one.

- *Insulation* (General)
  User can apply insulations attached to the bottom of the foundation slab. If **Calculate soil as a solid element** is activated, Insulations are calculated as part of the Foundation object with the user-specified parameters. Otherwise, they are ignored.

- *Geometry* (General)
  Variable thicknes can be set for the foundation slab with the same method as for the **Plate** objects.

- *Eccentricity* (General)
  Eccentricity (e) adds "virtual" distance between the reference plane and the calculation (middle!) plane if*"Consider eccentricity in calculation"* option is unchecked, otherwise the eccentricity has effects in the calculation. This option is useful for displaying regions with different thickness values.

⚠ Eccentricity have no effect in calculations if *"Consider eccentricity in calculation* is checked off. They are only to display plate "solid" with its defined position.

Eccentricity value can be displayed in model view (**Display settings**).

The *"Consider eccentricity caused by cracking in cracked section analysis"* option is for Reinforced concrete slabs. With this option modeling the eccentricity changes after cracking is available. The following figure shows the eccentricity changes.

Uncracked analysis *Cracked analysis*

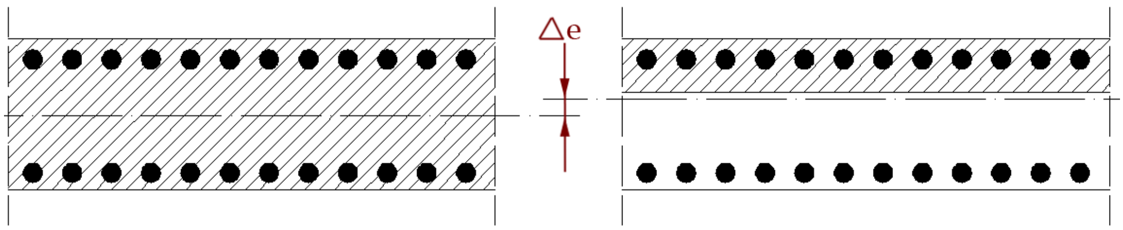


Figure: Explaining cracked and uncracked analysis

- *Orthotropy* (General)
  The ratio between the E1 and E2 elasticity modules in the main directions can be set here. So, if the value is 1, the new plate will be isotropic, otherwise it is orthotropic. The orthotropy direction can be set with the angle between E1 direction and the plate's local x' axis (*Alpha*).

- **Material**
  Only concrete material can be set for *Analysis* and *Design*.

4. Define the plate region in the model view based on the chosen geometry method.

Optional steps:

5. Modify the default edge connections with . Just select the plane and then its edge(s) to set connection conditions, and finally set the requested motion and rotation settings.
   Motion-Rotation / Compression-Tension

   The x', y' and z' are valid in the connection's local system displayed in the preview and as a symbol in the middle of the edges in the model view. Choose "Rigid" or "Free" from the drop-down lists, or type stiffness values (spring connection).

   Active link button assigns the stiffness value typed in the *Compression* field to the *Tension* field. Inactive icon lets to define stiffness values separately.

   Predefined types

   Click to set all motion and rotation components to "rigid". The result will be a "totally rigid" connection.

   Click to set all motion components to "rigid" and all rotation components to free (allow rotation around all directions). The result will be a hinged connection.

   Click to set all components to free. This tool gives the possibility to virtually connect independently moving elements.

   Behavior

Click to assign the stiffness value defined in the *Compression* field to the *Tension* field by components.

Click not to allow tension for all connection components.

Click not to allow compression for all connection components.

Setup "Rigid"

With this option the value of **"infinite" rigidity** can be defined by motion and/or rotation for selected connections.

For all edge connections, the value of "infinite" rigidity can be defined and set as project default at *Settings* > *Calculation*> *"Rigid" values* > *Connections*.

6. Add **hole** with tool.
7. Modify the reference plane geometry with the *Edit* menu commands valid for region elements.
8. Modify the *Foundation slab* properties with the *Properties* tool of the *Foundation slab* tool palette.
9. Set the display settings of plates at *Settings* > *Display* > *Soil and Foundation*.
10. The Foundation slabs are stored on "Foundation" and if they modeled with *Surface support group* their supports will be stored at the "Support" **Object layers**. The color and pen width settings by selected plate elements can be modified by *Edit* > **Properties** > *Change appearance*.

Plane plate

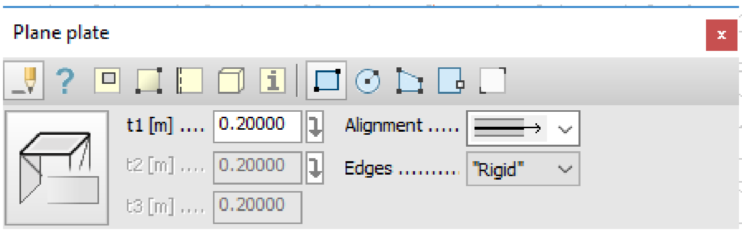
Plane plate property	Description
Modules where available	
Reference plane position	Horizontal in
	Arbitrary (horizontal, vertical and skew) in
Geometry	Any shape
Thickness	Constant or variable
Hole	Available
Eccentricity	Only for display
Edge connection	Rigid, hinged, elastic (spring) linear or non-linear
Material	Steel, concrete, timber and general
Orthotropic feature	Available
Load direction	Vertical in
	Arbitrary in
Load type	All point, line and surface load

Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in
Available design	**RC design** in
	Timber design in

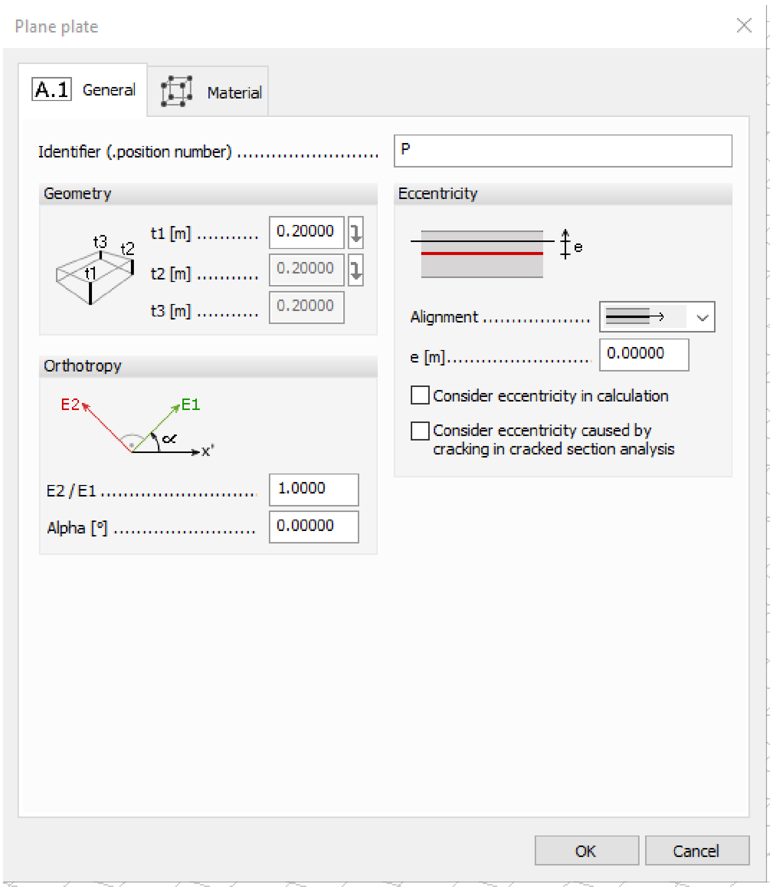
Table: Plate properties

Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**.
2. Start *Plane plate* command from Structure tabmenu and choose *Define*.



3. Set the properties of the new plate at *Default settings*:



- *Identifier* (General)

The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Alignment and Eccentricity* (General)
  With alignment the position of the reference (definition) plane can be set to top, center or bottom.

  Eccentricity (e) adds "virtual" distance between the reference plane and the calculation (middle!) plane if *"Consider eccentricity in calculation"* option is unchecked, otherwise the eccentricity has effects in the calculation. This option is useful for displaying regions with different thickness values.

  ⚠ Alignment and eccentricity have no effect in calculations if *"Consider eccentricity in calculation* is checked off. They are only to display plate "solid" with its defined position.

  Eccentricity value can be displayed in model view (**Display settings**).

  The *"Consider eccentricity caused by cracking in cracked section analysis"* option is for Reinforced concrete slabs. With this option modeling the eccentricity changes after cracking is available. The following figure shows the eccentricity changes.

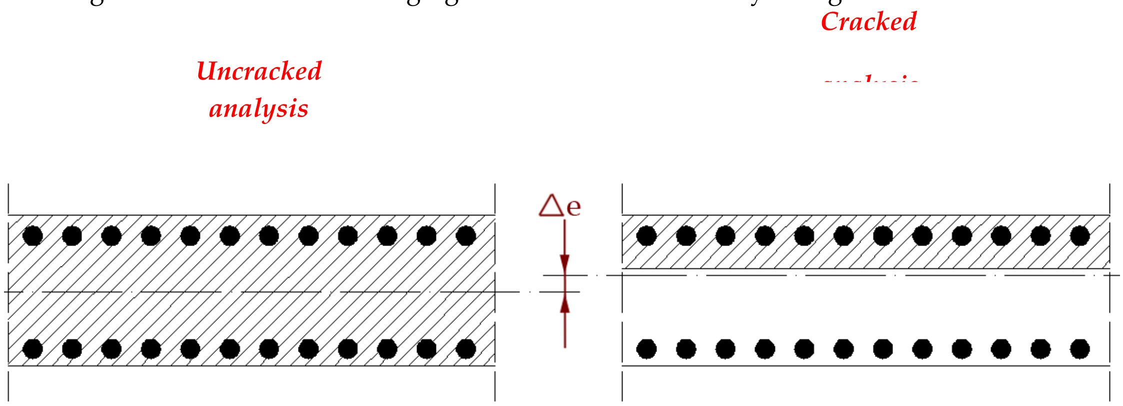


Figure: Explaining cracked and uncracked analysis

- *Thickness* (General)
  By default, only plate with constant thickness (t1=t2=t3) can be set at *Default settings*. Use the **Variable thickness** tool to define different t values that declare the top/bottom surface as a linearly variable plane.

- *Orthotropic features* (General)
  The ratio between the E1 and E2 elasticity modules in the main directions can be set here. So, if the value is 1, the new plate will be isotropic, otherwise it is orthotropic. The orthotropy direction can be set with the angle between E1 direction and the plate's local x' axis (*Alpha*).

  ⚠ In *Plate* module the local x' axis is always parallel with the global X axis.

- **Material**
  Any type of materials can be set for plate *Analysis*, but design can be run for concrete only. The applied material name can be displayed in model view (**Display settings**).

4. Set the default connections for all plate edges to rigid or hinged with the tool palette's *Edges* option.

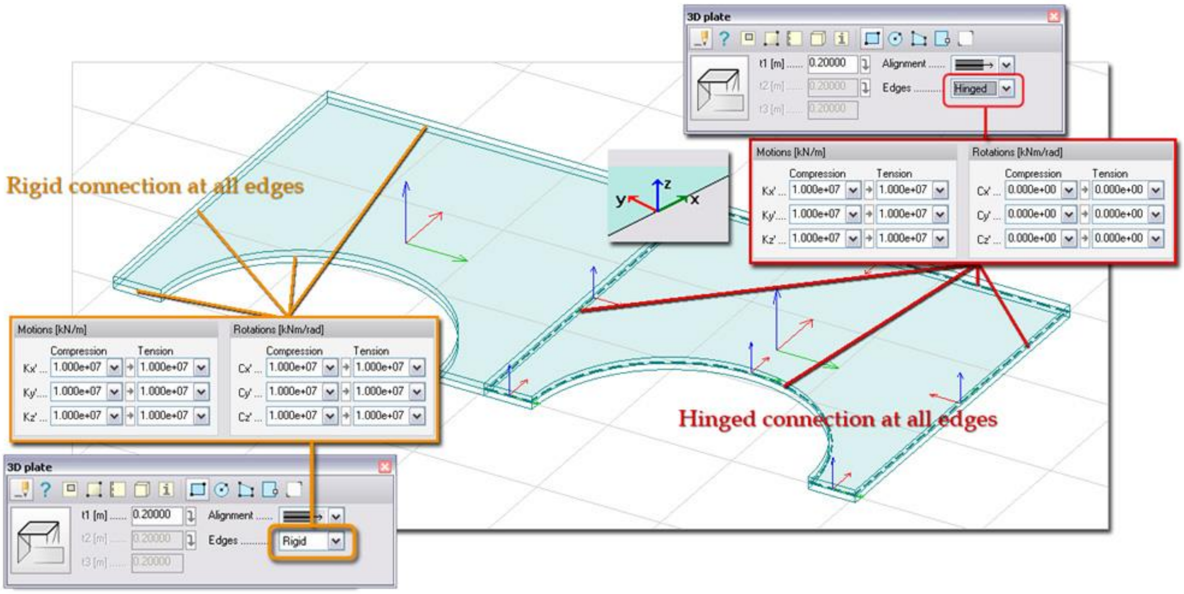
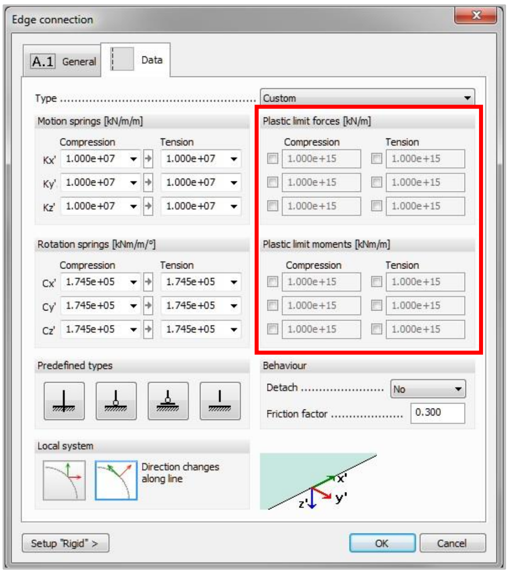


Figure: Default edge connections

Use the **Edge connection** tool to modify the default connections by the selected edges of a predefined plate. Hinged, rigid, semi-rigid (spring), free connections can be set with linear and non-linear behavior.

The edge connections can be calculated as plastic behavior, which shows the figure below. The values can be customized.



5. Choose a **geometry** definition method for the plate reference plane (set by *Alignment* and *Eccentricity*).
6. Define the plate region in the model view based on the chosen geometry method. In module the slab plane can be horizontal only, while in 3D modules any skew position can be set. All

region definition points have to be in the same plane. UCS may help to find the requested position of the new plate region.

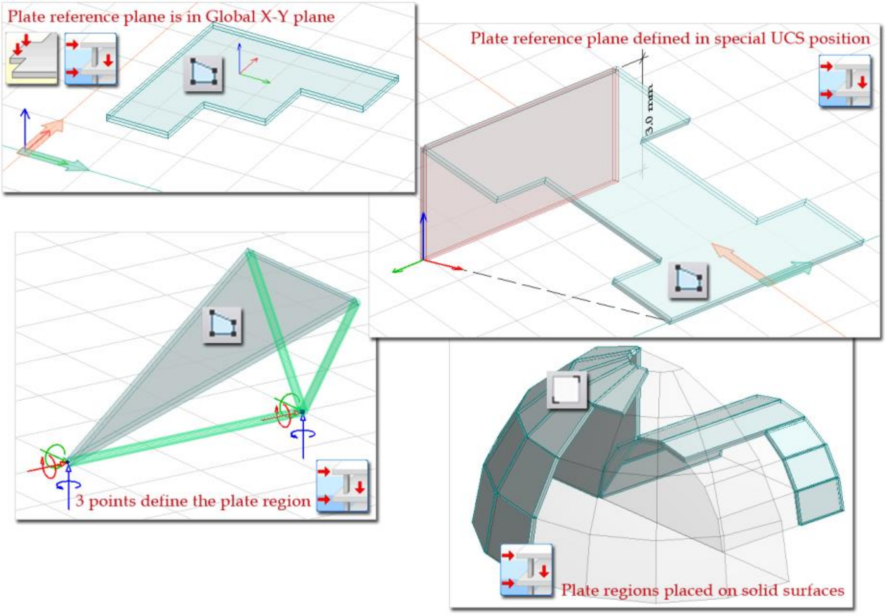


Figure: Different working planes for plate definition

Optional steps:

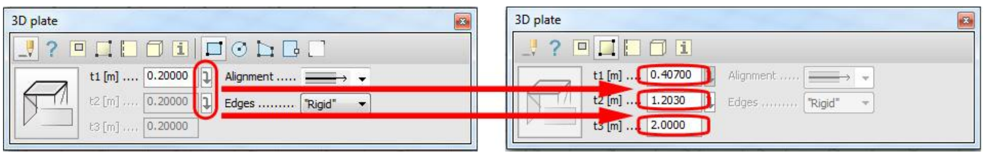
7. Modify default thickness (Variable thickness)

   Constant slab thickness can be fast modified with the *Properties* tool. Just select the plate(s) and set a new value under General Tab.

   Constant thickness can be changed to variable, or reverse, with the Variable thickness tool.

   Steps of defining plate with variable thickness:

   i. Inactivate the link buttons to allow giving different values in the *t* fields.
   ii. Type the *t1*, *t2* and *t3* according to the current length unit.



   iii. Select the plate you would like to modify.
   iv. Define three points in the plane of the selected plate's reference plane, where *t1*, *t2* and *t3* will be measured. The measurement of the thickness values depends on the position of the reference plane (*Alignment* and *Eccentricity*).

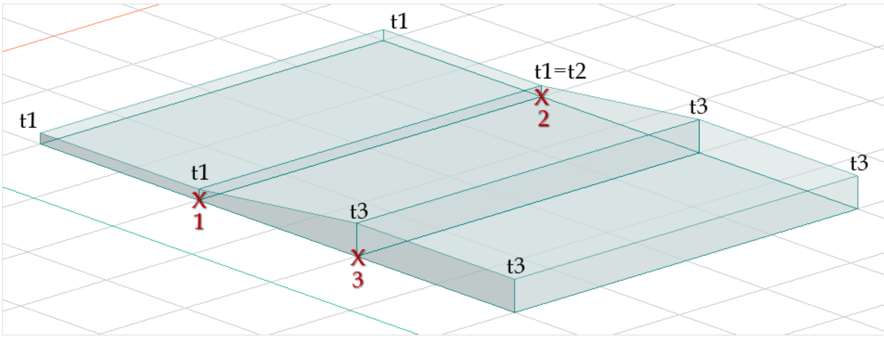


Figure: Align the thickness of the middle slab to its neighbors

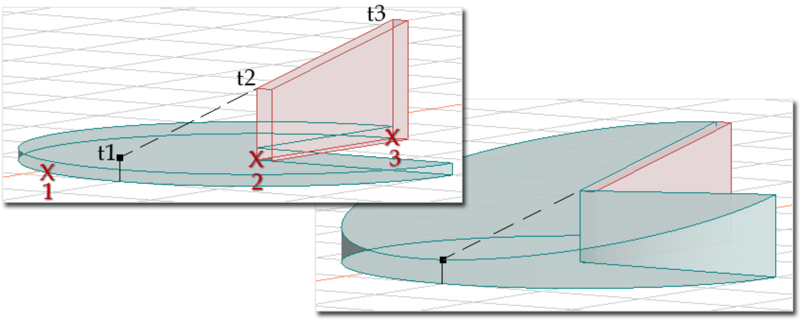
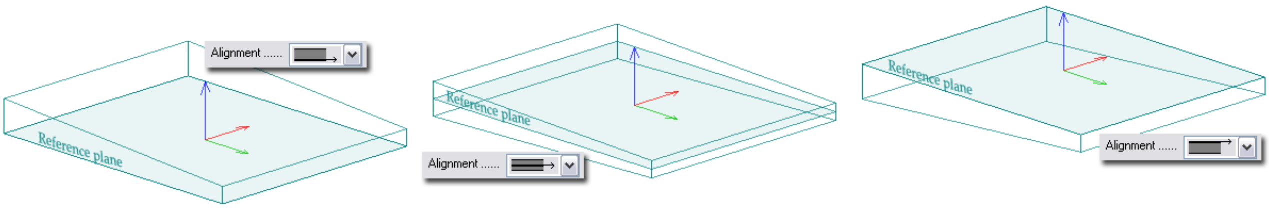


Figure: Align circular slab thickness to a variable height wall

The measurement of the thickness values depends on the position of the reference plane (*Alignment* and *Eccentricity*).



You are allowed to place the measuring points outside the region, too.

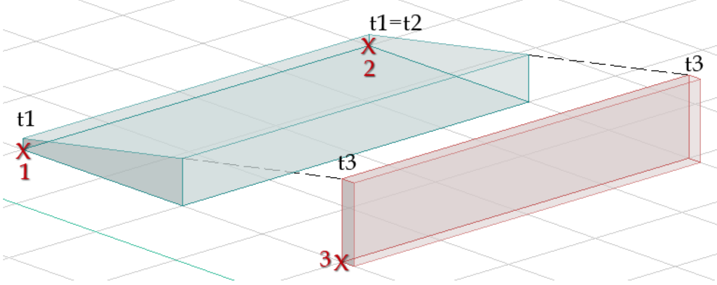


Figure: Edit thickness by using other objects' height

8. Modify the default edge connections with . Just select the plane and then its edge(s) to set connection conditions, and finally set the requested motion and rotation settings.

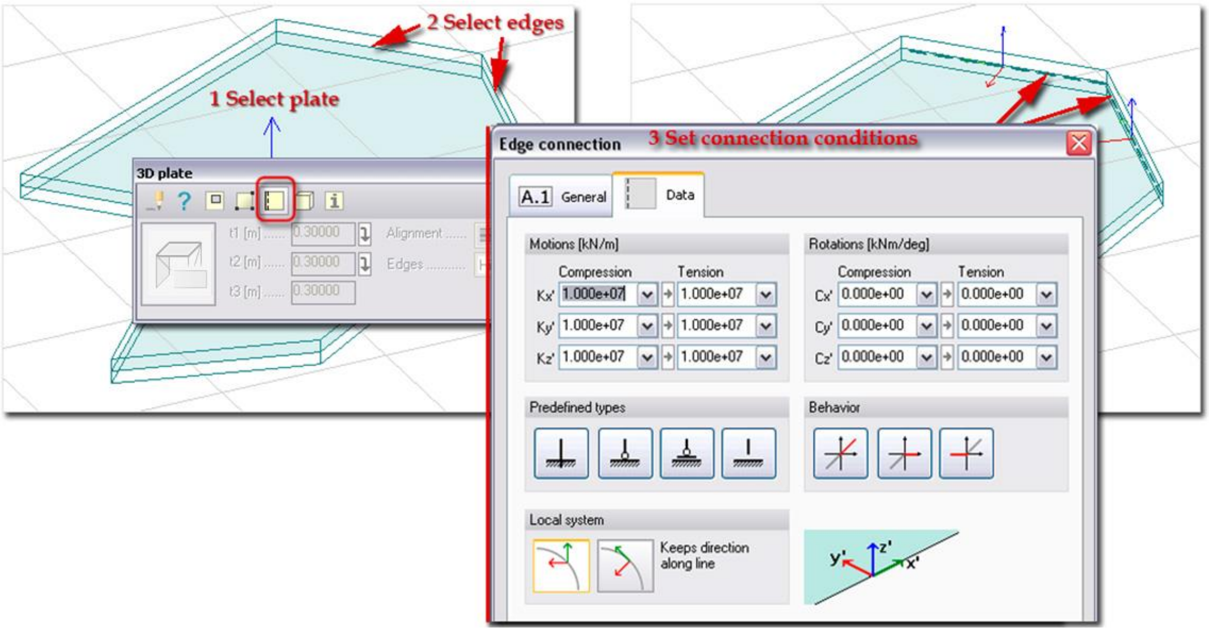


Figure: Modification of two slab edges

Motion-Rotation / Compression-Tension

The x′, y′ and z′ are valid in the connection’s local system displayed in the preview and as a symbol in the middle of the edges in the model view. Choose “Rigid” or “Free” from the drop-down lists, or type stiffness values (spring connection).

Active link button assigns the stiffness value typed in the *Compression* field to the *Tension* field. Inactive icon lets to define stiffness values separately.

Predefined types

Click to set all motion and rotation components to “rigid”. The result will be a “totally rigid” connection.

Click to set all motion components to “rigid” and all rotation components to free (allow rotation around all directions). The result will be a hinged connection.

Click to set all components to free. This tool gives the possibility to virtually connect independently moving elements.

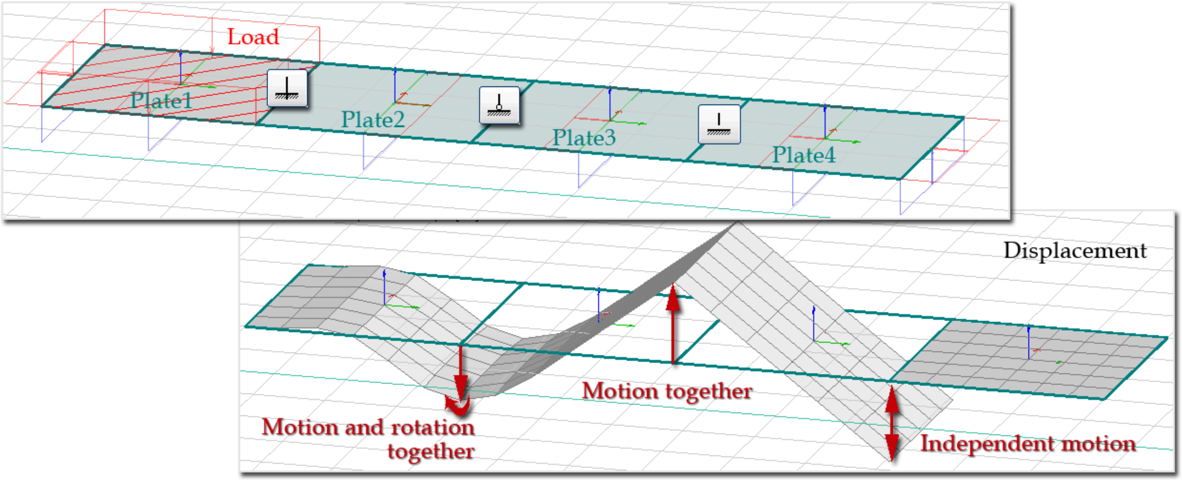


Figure: Edge connection types and their displacement effect

Behavior

Click  to assign the stiffness value defined in the *Compression* field to the *Tension* field by components.

Click  not to allow tension for all connection components.

Click  not to allow compression for all connection components.

Setup "Rigid"

With this option the value of **"infinite" rigidity** can be defined by motion and/or rotation for selected connections.

For all edge connections, the value of "infinite" rigidity can be defined and set as project default at *Settings* > *Calculation*> *"Rigid" values* > *Connections*.

As a real example, *Edge connection* gives the possibility to model prefab concrete elements too. Define the objects with the *Plate/3D plate* commands, set the thickness value of the regions to substitute the real cross-section with the holes, and finally add "hinged" edge connection (C_x = *0*) to one of the region at the common edge with the other one (see the next figure).

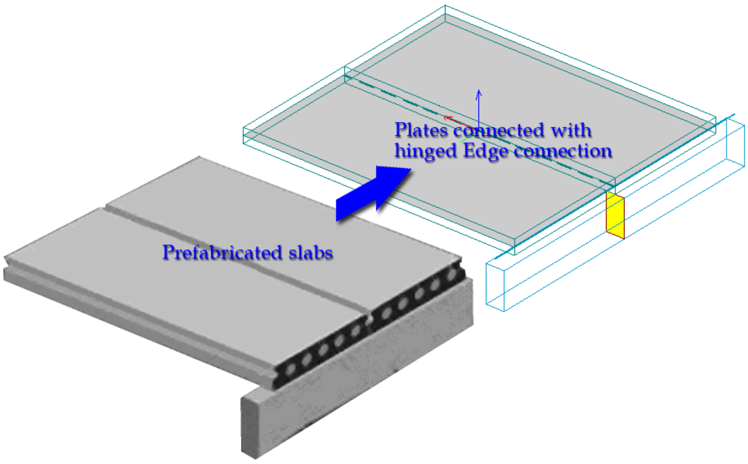


Figure: Modeling slab elements connected to each other (hollow-core prefab slabs)

9. Add **hole** with tool.
   In 3D modules, select a plate region and define hole in it or cut it by using a geometry mode. Selecting a plate displays the **UCS** in the reference plane of the plate.

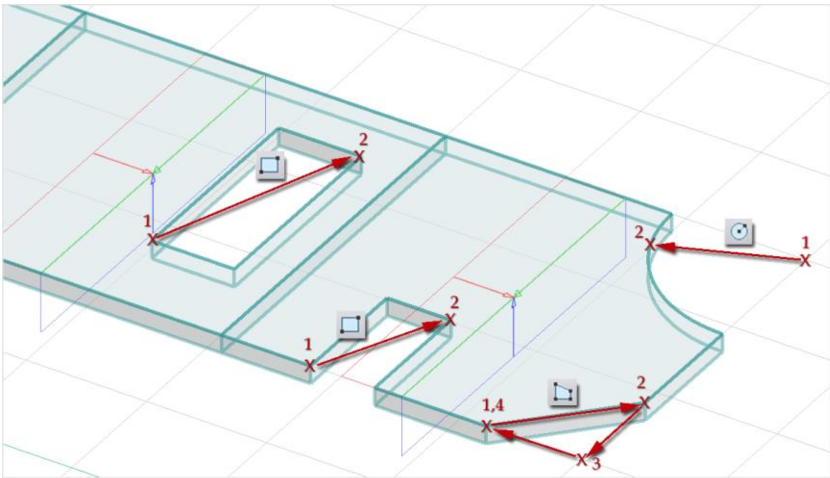
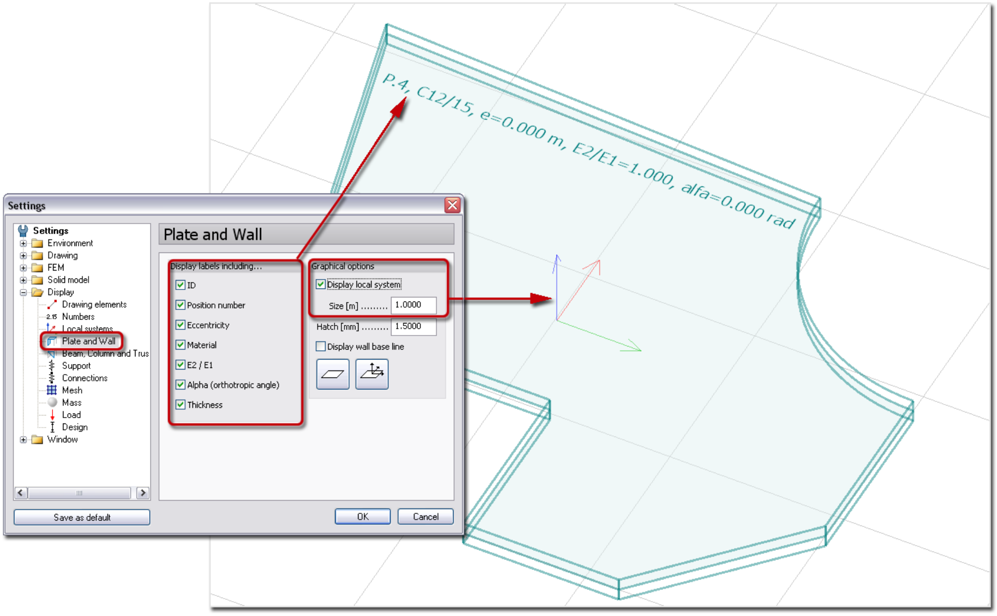


Figure: Hole and cuttings

   In *Plate* module, just define hole or cutting by using a geometry mode in the required plate reference plane. Selection of the host plate is needed, if the hole region intersects more than one regions.

10. Modify the reference plane geometry with the *Edit* menu commands valid for region elements.

11. Modify the plate properties with the *Properties* tool of the *Plate* tool palette.
12. Information (the volume of the plate solid and the position of its centre of gravity) about a selected plate can be inquired with the *Info* tool. A drawing point can be placed in the plate's centre of gravity, if needed.
13. Set the display settings of plates at *Settings* > *All* > *Display* > *Shell*.



14. The plates are stored on “Plates” **Object layers**. At layer settings, the default color and pen width can be set for all plate regions. The color and pen width settings by selected plate elements can be modified by *Edit* > **Properties** > *Change appearance*.

Plane wall

Plane wall property	Description
Modules where available	
Function	Line support in
	Shear or load-bearing wall region
	Unit section of long-extensional object (tunnels, dams, retaining walls et.)
	Structural wall in
Reference plane position	Vertical in
	Global X-Y plane
Geometry	Any shape
Thickness	Constant or variable
Hole	Available
Eccentricity	Only for display
Edge connection	Rigid, hinged, elastic (spring) linear or non-linear
Material	Steel, concrete, timber and general
Orthotropic feature	Available in

Load direction	Arbitrary in
	Planar, global X-Y plane
Load type	All point, line load and surface load in
Available analysis results	Reaction forces in
	Displacement, internal forces and stresses
	Displacement, internal forces, stresses, stability and vibration shape in
Available design	**RC design** in
	Timber design in

Table: Wall properties

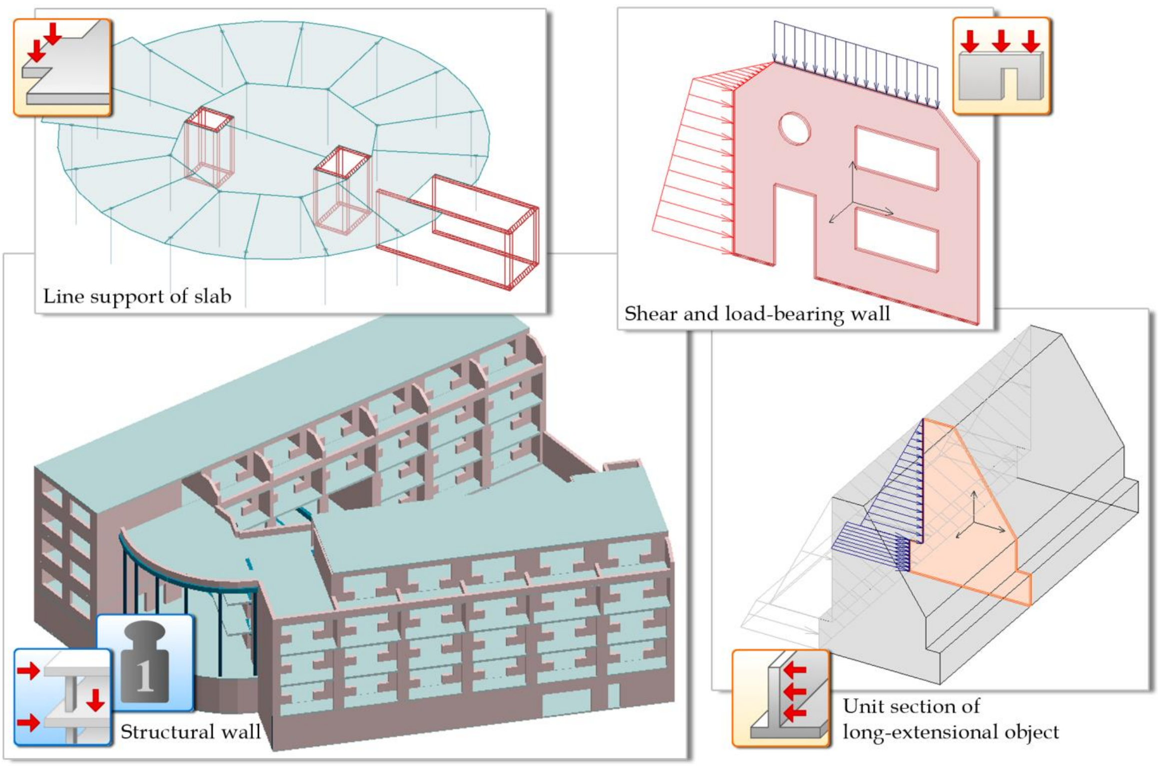


Figure: Wall function by FEM-Design modules

Definition steps

The wall definition depends on what module is used for modeling:

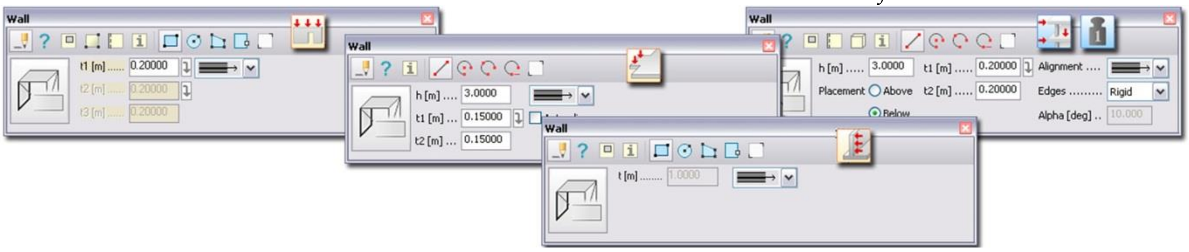
Wall is a line support, so its base line has to be defined with a given height and constant/variable thickness as a straight or curved line.

The only modeling object is the Wall, which is a region with constant or variable thickness, so its reference plane has to be defined as a rectangle, circular, polygonal or arbitrary shaped region. Holes can be added to it.

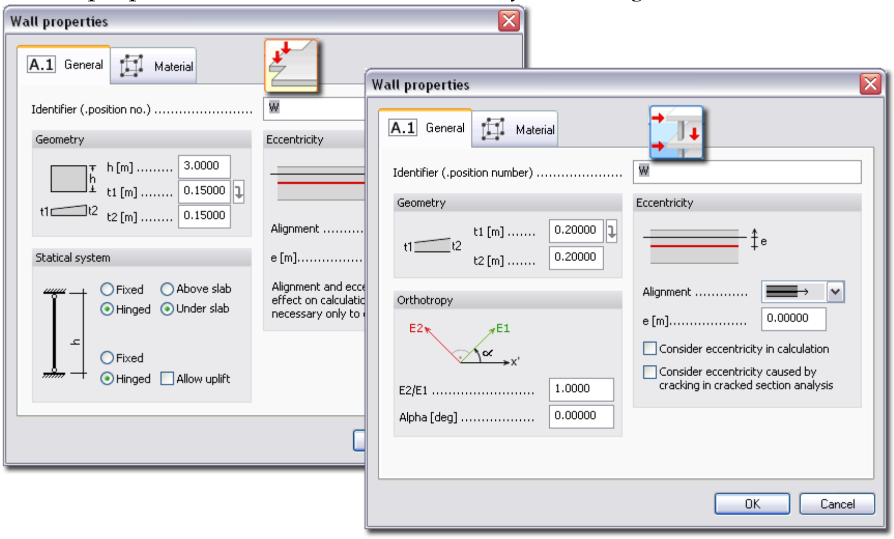
The only modeling object is the Wall, which is a region with unit (=1) thickness, so its reference plane has to be defined as a rectangle, circular, polygonal or arbitrary shaped region. Holes can be added to it.

Wall is a region in vertical position, so the base line has to be defined with given height and constant/variable thickness as a straight or curved line. Holes can be added to it.

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**.
2. Start *Wall* command from Structure tabmenu and choose *Define*.



3. Set the properties of the new wall at *Default settings*:



- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Height (h)* (General)
  In the height can be set in the dialog, but in 3D modules only in the Wall command's tool palette.

- In 3D modules, the sign (positive or negative) of the Height value defines the location of the Wall (**Straight line**).

- In , the position of the wall support can be set according to slab position: “Above slab” or “Under slab”. It also defines the location of the base line.

- *Alignment and Eccentricity* (General)
  With alignment the position of the reference base line or the reference region can be set to left/front, center or right/back.

  Eccentricity (e) is handled the same was as in **3D Plate** .

  Eccentricity value can be displayed in model view (**Display settings**).

  While modeling, think about the reference plane/reference line connections with each other. Connecting edged “perfectly” results correct and “nice” finite element mesh. If they intersect each other with small distances, the outcome of mesh generation may be very small finite elements or unexpected mesh regions. The next figure shows an example for possible (recommended or not) wall connection to slab edge.

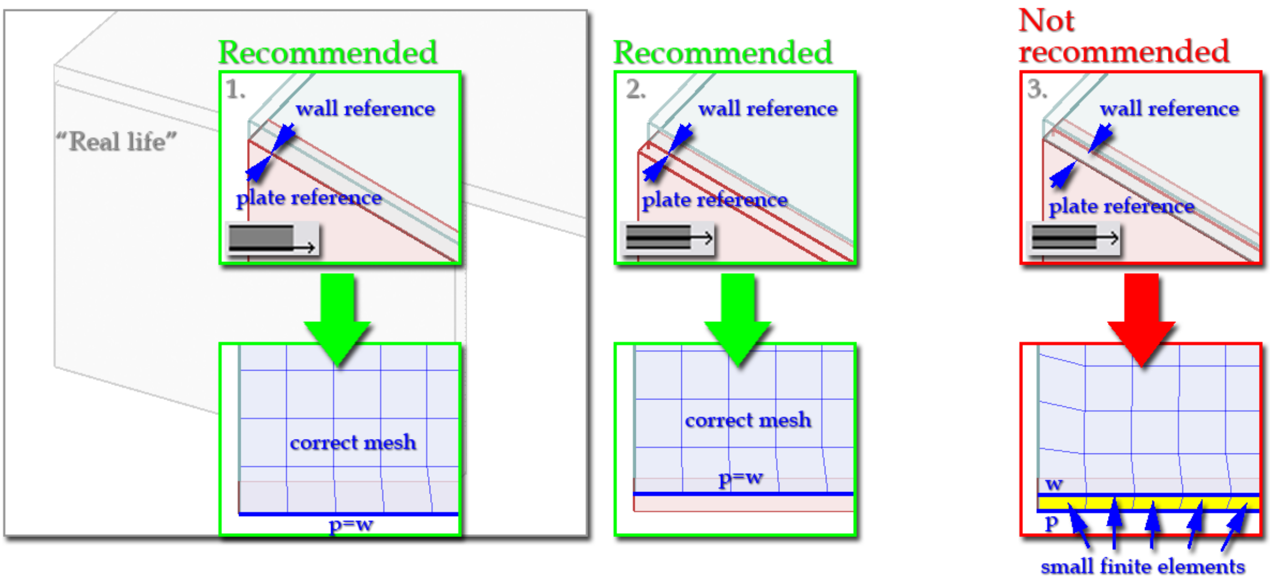


Figure: Recommended modeling of plate-wall edge connection (no calculation differences in case 1. and 2.)

- *Thickness* (General)
  In , the thickness values (t1 in the start point and t2 in the end point) can be set directly in the settings dialog. The measurement of the thickness depends on the **Alignment and Eccentricity** settings.

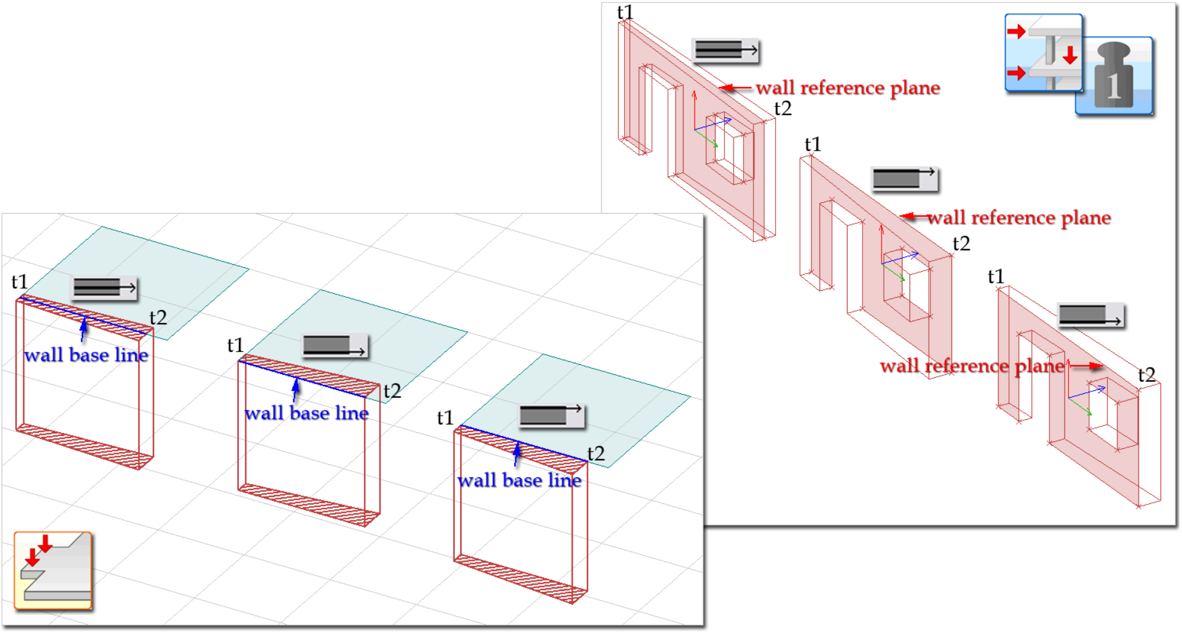


Figure: Variable thickness and eccentricity

In , the thickness value is fixed. It is always 1m, so only region(s) with unit thickness (section) can be defined.

In , the thickness value can be constant or variable like plate regions. Because walls are regions in the *Wall* module like plates in *Plate* module, the definition way of constant and variable thickness is the same. In the dialog, only constant thickness (t1=t2=t3) can be set, but with Variable thickness tool (same with Plate's **Variable thickness** tool) t values can be modified separately to define wall regions with variable thickness.

- *Orthotropic features* (General)

  This option is available in   modules. The ratio between the E1 and E2 elasticity modules in the main directions can be set here. So, if the value is 1, the new wall will be isotropic, otherwise it is orthotropic. The orthotropy direction can be set with the angle between E1 direction and the wall's local x' axis (*Alpha*).

  ⚠ In   modules the local x' axis is always parallel with the global X axis.

- *Statical System (Connections)* (General)

  In , the support conditions (*General*) of the top and bottom wall ends can be set to fully rigid or fully hinged. Because, *Walls* are line supports further line/point support is not needed to define at *Wall* end.

- *Allow uplift* (General)

  Available in only. "Uplift" behavior can be modeled by activating this option. It works in case of tensional reaction force.

  In 3D modules, "uplift" can be modeled with non-linear **Line support** settings.

- **Material**
  Any type of materials can be set for wall *Analysis*, but design can be run for concrete and timber walls only depending on the current FEM-Design module. The applied material name can be displayed in model view (**Display settings**).

3. In  modules, the default connections for all wall edges can be set to rigid, hinged or "No shear" with the tool palette's *Edges* option.

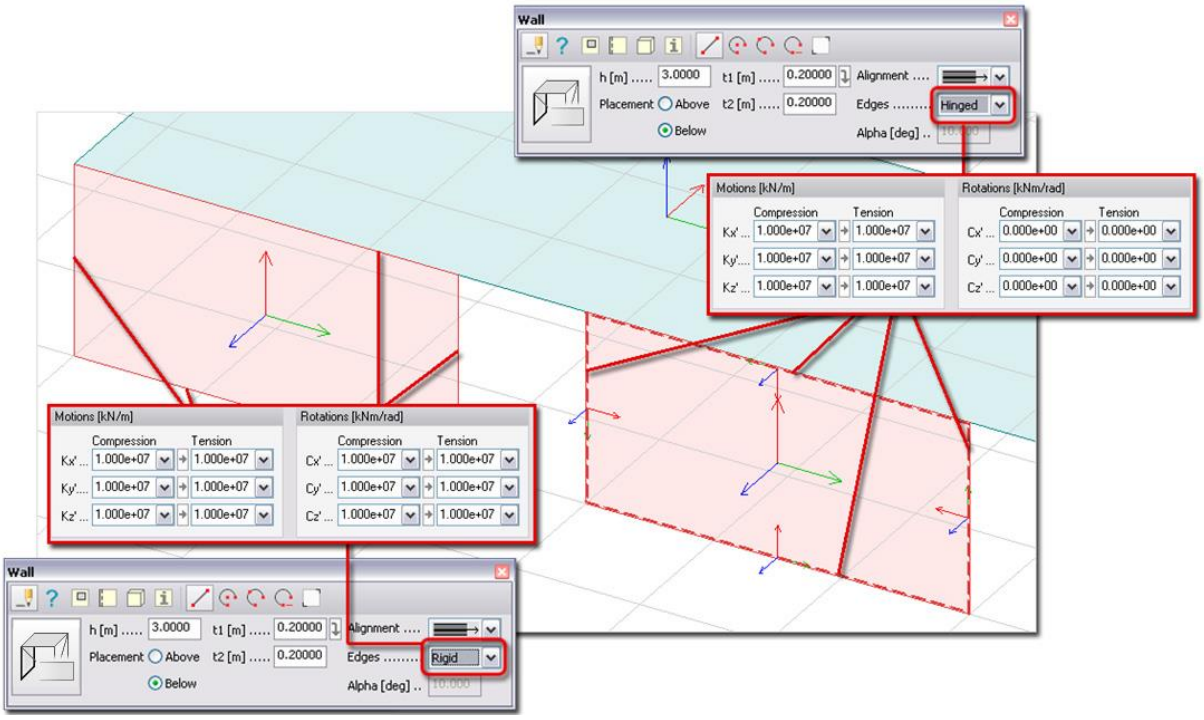
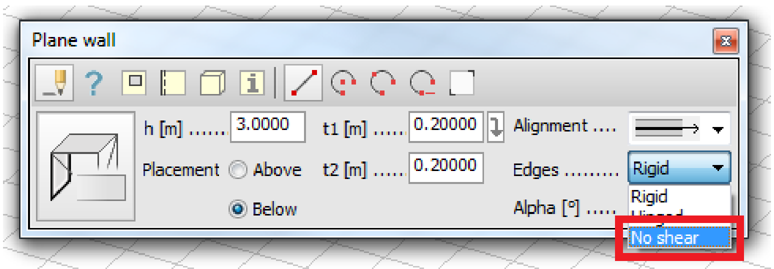


Figure: Default edge connections

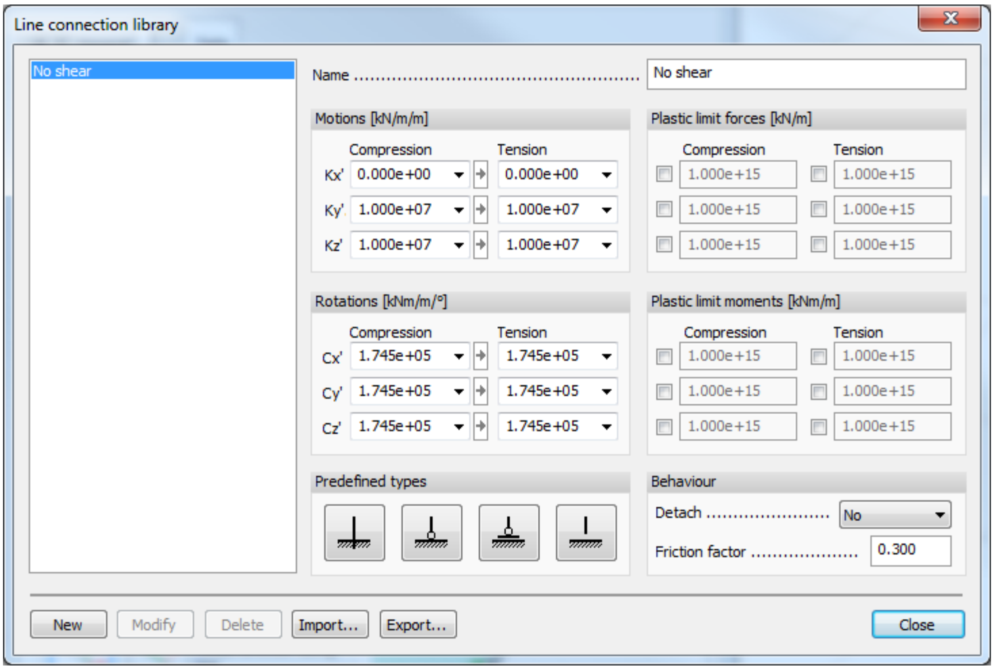
Use the **Edge connection** tool to modify the default connections by the selected edges of a predefined wall. Hinged, rigid, semi-rigid (spring), free connections can be set with linear and non-linear behavior.

"*No shear*" macro defines edge connection with predefined "*No shear*" rigidity type on the defined wall's bottom edge.



It is the recommended wall definition method to create a non-bracing system element.

“*No shear*” rigidity type settings with free local x’ motion rigidity:



⚠ If a wall connect to slab at a common edge and you would like to define hinged connection between them, only one edge connection (for plate or wall) has to be defined hinged and the other has to be kept as rigid. So, applying hinged edge connection for both plate and wall at their linked edge is not allowed to avoid instability in finite element modeling. The figure shows the possibilities to model the same hinged connection in two ways.

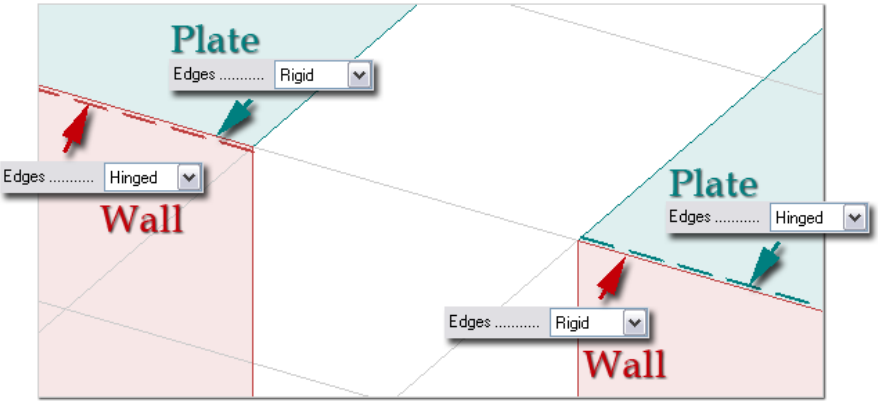


Figure: Modeling the same hinged wall-plate edge connection in two ways

The previous rule is also true for wall support in  module: if, you would like to model hinged wall support at a plate edge, define hinged statical connection for wall (top or bottom depending on the wall position) and set the plate edge connection to rigid, or, define rigid wall support and hinged edge for plate.

4. Choose a **geometry** definition method for the wall’s reference line/plane (set by *Alignment* and *Eccentricity*) and create the wall with co-ordinates or by picking points in the drawing area.

In modules, the walls are regions in the Global XY working plane.

In these wall modules the dead-load is orientated in Global Y direction, although the dead-load is orientation in Global Z direction in all other modules.

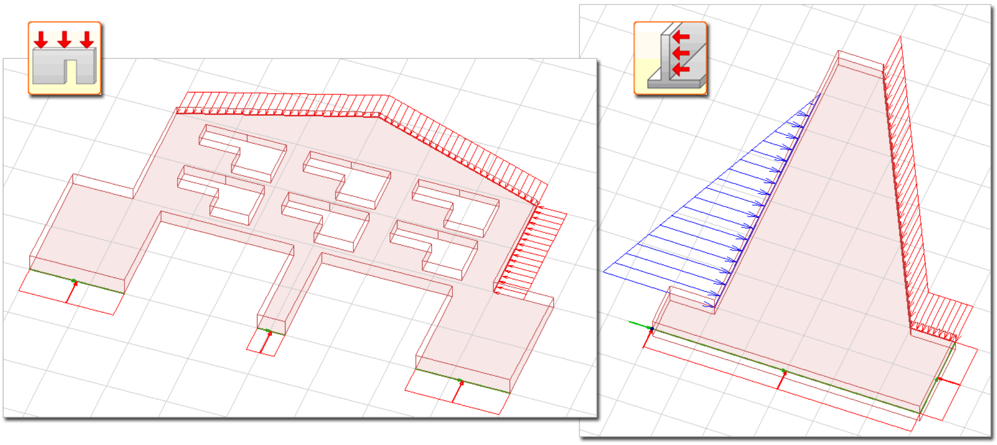


Figure: Wall region with holes, loads and supports in Wall and Plane Strain modules

In module, wall is a line support, so it has to be defined with its base/reference line. Although walls are planar objects in modules, they can be defined with their reference lines, because of the fixed vertical position. The lines can be straight or “curved”. The “curved” means real arc and circle in module, while it means the combination of straight lines (wall sections) in the 3D modules. In the second case the “curved walls” are built up by planar region members (curved finite elements are not available in FEM-Design), and the resolution of the curve approximation can be set by the peripheral angle of the sections (*Alpha*).

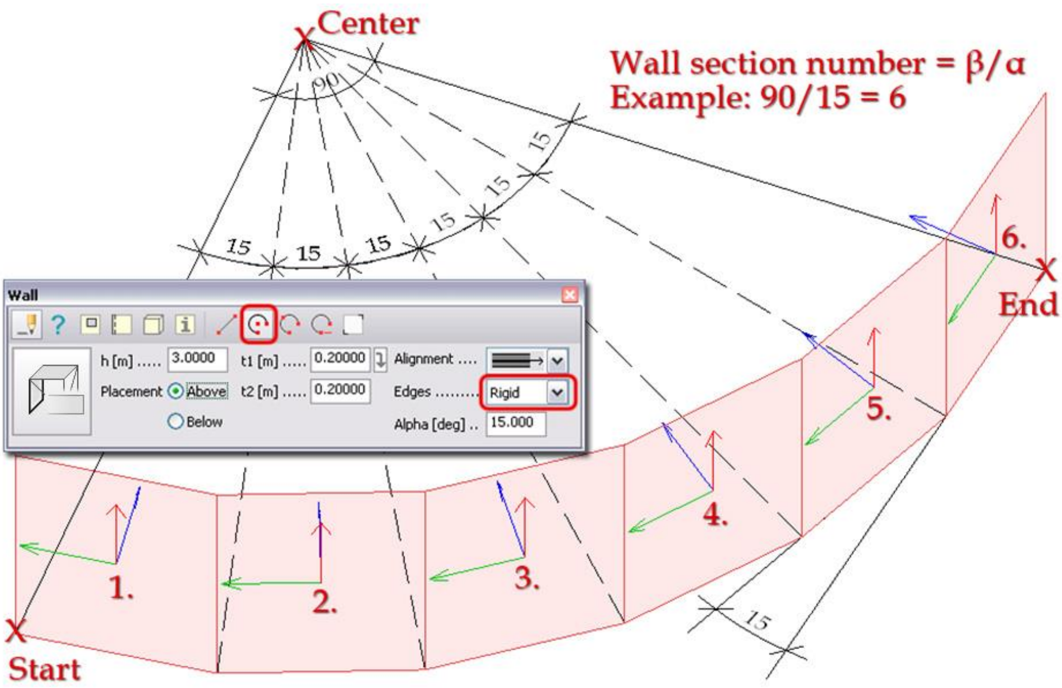


Figure: "Curved" wall definition in 3D modules

In the 3D modules, if you would like to connect a plate to "curved" walls at the edges, define polygonal slab instead of curved-edge slab. In case of curved plate, small regions are cut by the connected wall, and that results finite elements with very small angle.

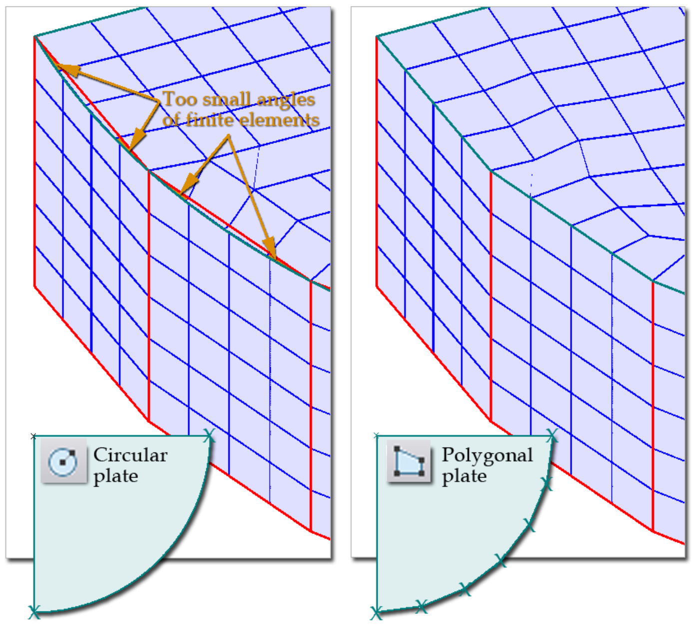


Figure: "Curved" wall definition in 3D modules

Optional steps:

5. Modify default thickness (Variable thickness)

Constant wall thickness can be fast modified with the *Properties* tool. Just select the wall(s) and set a new value under General Tab. In modules, you can define variable thickness with t1 and t2 values too.

In modules, constant thickness can be changed to variable (linear distribution), or reverse, with the Variable thickness tool.

Defining wall with **variable thickness** is similar as done in case of a Plate.

6. Modify the default edge connections with . Just select the wall and then its edge(s) to set connection conditions (rigid, hinged, free or semi-rigid (spring)), and finally set the requested motion and rotation settings. **See the definition steps and possibilities at Plate.**
7. Add **hole** with tool (**see hole definition in plates**).
   Except of module where wall is support, holes and cuttings can be defined in walls.

   In 3D modules, select a wall region and define hole in it or cut it by using a geometry mode. Selecting a wall displays the **UCS** in the reference plane of the wall.

   In modules, just define hole or cutting by using a geometry mode in the required wall reference plane. Selection of the host wall is needed, if the hole region intersects more than one regions.

8. Modify the reference line/plane geometry with the *Edit* menu commands valid for lines/region elements.
9. Modify the wall properties with the *Properties* tool of the *Wall* tool palette.
10. In *Plate* module, the *Info* tool displays the support stiffness (motion and rotation) values of the wall ends (start and end point) valid in the wall **local co-ordinate system**.
    In the other design modules, information (the volume of the wall solid and the position of its centre of gravity) about a selected wall can be inquired with the *Info* tool. A drawing point can be placed in the wall's centre of gravity, if needed.

11. Set the display settings of walls at *Settings* > *All* > *Display* > *Shell* (**see Plate display settings**).
12. The walls are stored on "Walls" **Object layers**. At layer settings, the default color and pen width can be set for all plate regions. The color and pen width settings by selected plate elements can be modified by *Edit* > **Properties** > *Change appearance*.

Shell Model

In *3D Structure* module, straight and arc bars (**Beams** and **Columns**) can be modeled more accurately for stability, utilization check (against flexural, torsional, lateral torsional and local buckling) and design according to their EC by modeling them with planar shell members.

Definition steps

1. Press *Analytical model / Shell model* option of *Beam* or *Column* toolwindow and choose the bar. By default, selecting a bar element generates a complex 3D model built by a group of steel plates derived from the geometry of the selected bar's profile. All converted plates inherit the original bar element's steel material.

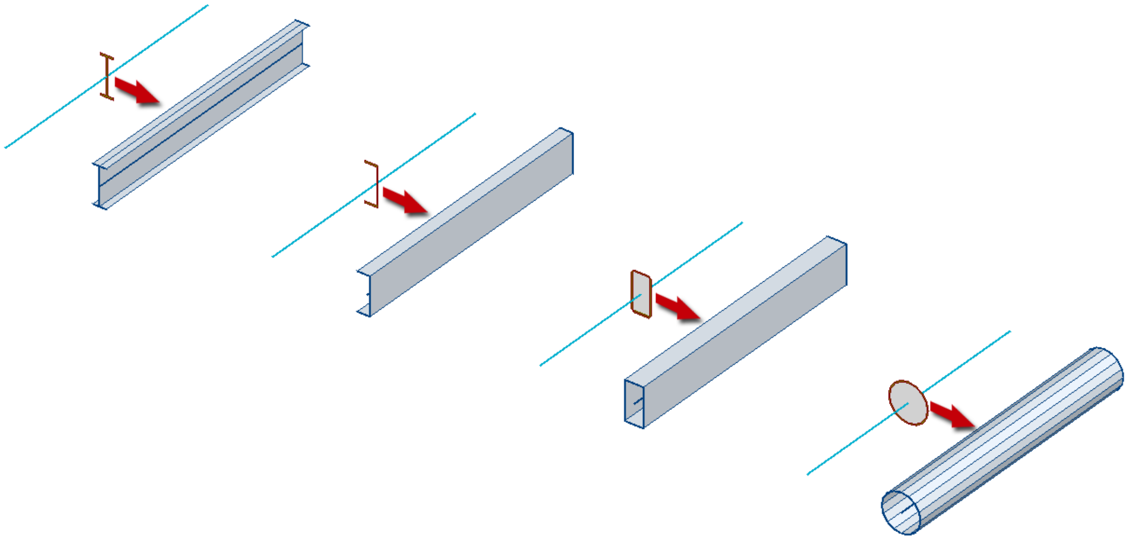
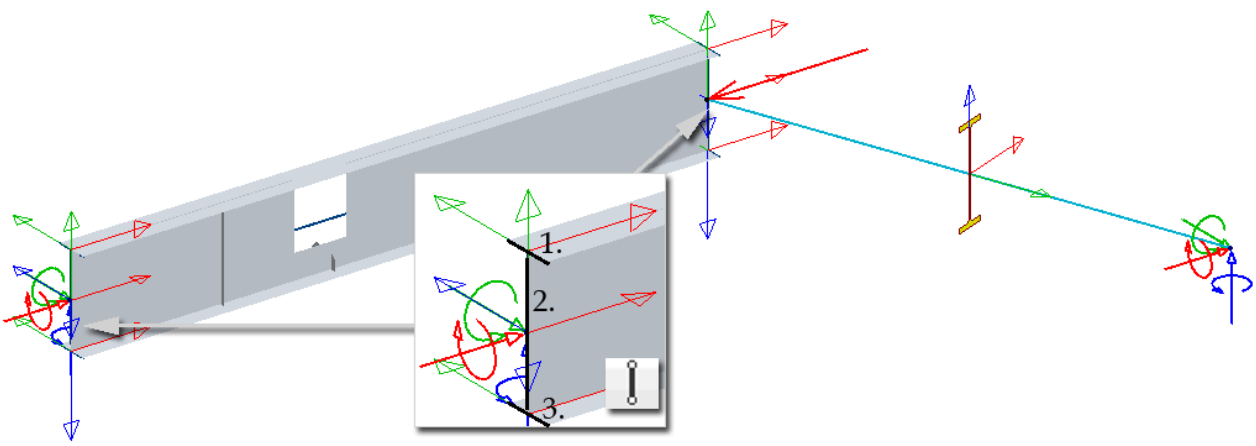


Figure: Straight bars with constant cross-section

After converting a bar to shell model, the program automatically defines "rigid" **Fictitious bar** objects at the ends of the shell model to guarantee continuous connection to other objects line 1D members, point supports and point loads. The stiffness of these fictive elements can be modified with the *Properties* tool of *Fictitious bar* command. Alternatively, this function can be turned off by clicking after selecting *Analytical model* in the *Beam* or *Column* toolwindow.



Depending on, *Beam*, *Column* or *Truss member* was converted to shell model, the model can be hidden together with the *Beam*, *Column* or *Truss member* object layer.

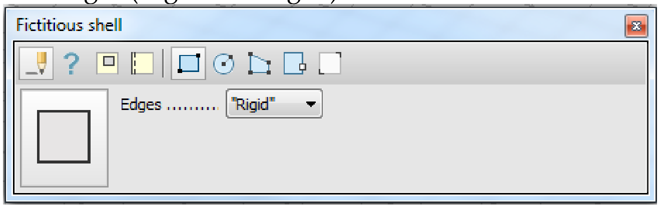
Fictitious shell

In *3D Structure* module a fictitious shell can be defined by clicking on . Fictitious shell is ideal to model a section with composite material, where the designer may calculate appropriate stiffness and fill the D, K and H stiffness values. Fictitious shell is also a powerful tool by the calculation of a section with complex geometry where the parameters are specified by the manufacturer of the structural element.

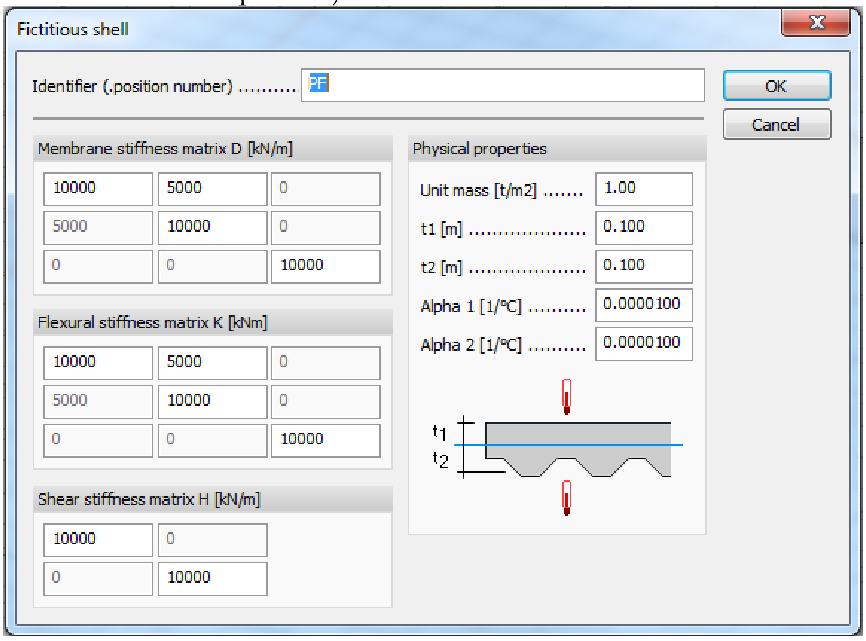
Definition steps

1. If needed, set a proper position for the **working plane**.

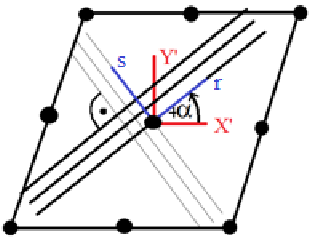
2. Start *Fictitious shell* command from Structure tabmenu and choose *Define*. In the tool-window the type of the edges (Rigid or Hinged) can be chosen.



3. Set the properties of the new plate at *Default settings*. The User can define the stiffness matrices (Membrane stiffness matrix, Flexural stiffness matrix and Shear stiffness matrix) and physical properties (Unit mass, distances of the planes where thermal load acts from the center plane, coefficient of thermal expansion).



Stiffness of orthotropic shell element for fictitious shell can be calculated as follow:



D **K** **H**

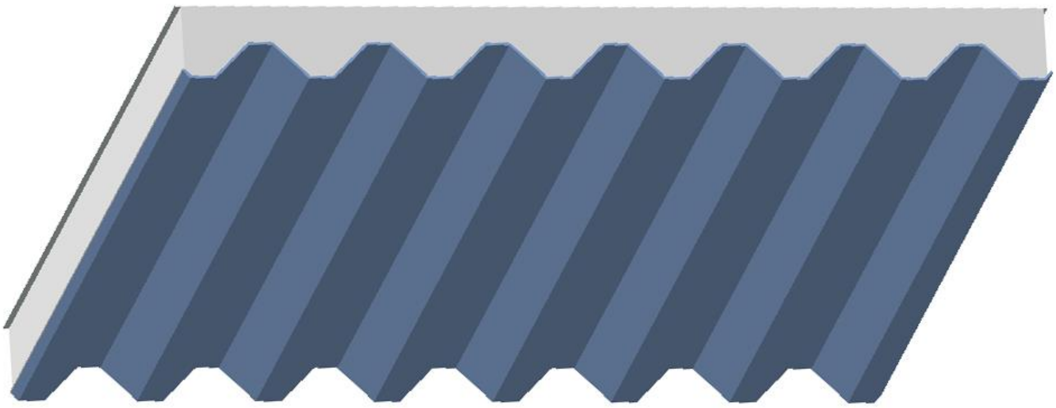
$$
\begin{bmatrix}
\frac{E_r h}{1-\nu_{rs}\nu_{sr}} & \frac{\nu_{sr} E_r h}{1-\nu_{rs}\nu_{sr}} & 0 & 0 & 0 & 0 & 0 & 0 \\
\frac{\nu_{rs} E_s h}{1-\nu_{rs}\nu_{sr}} & \frac{E_s h}{1-\nu_{rs}\nu_{sr}} & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & G_{rs} h & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{E_r h^3}{12(1-\nu_{rs}\nu_{sr})} & \frac{\nu_{sr} E_r h^3}{12(1-\nu_{rs}\nu_{sr})} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{\nu_{rs} E_s h^3}{12(1-\nu_{rs}\nu_{sr})} & \frac{E_s h^3}{12(1-\nu_{rs}\nu_{sr})} & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{G_{rs} h^3}{12} & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & \frac{G_{rt} h}{1.2} & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{G_{st} h}{1.2}
\end{bmatrix}
$$

⚠ These matrices belong to an orthotropic, homogenous, constant thickness shells according to Mindlin theory.

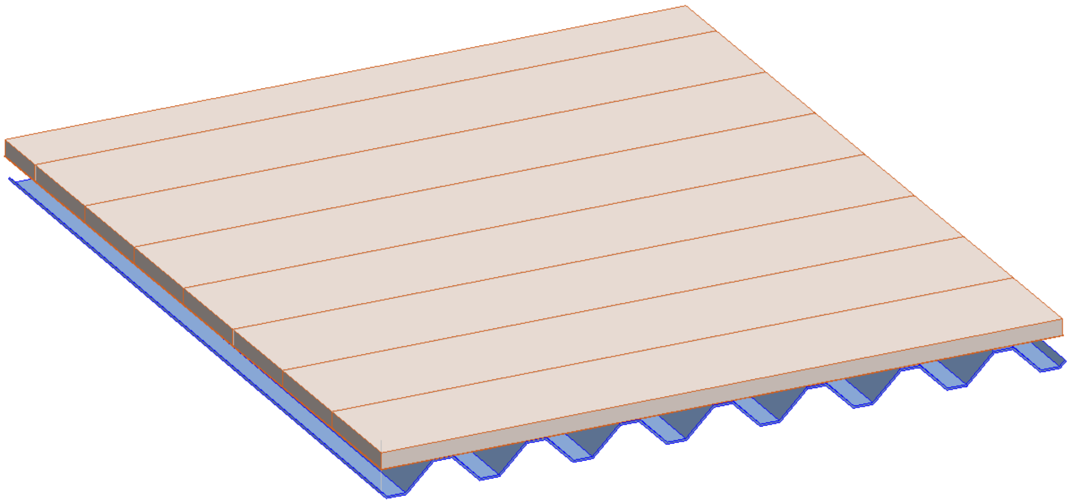
⚠ The H matrix is calculated as ρ*G*h (see in Profiled plate), where the ρ is the shear modification factor, the G is the shear modulus and the h is the thickness of the fictitious shell. For constant thickness the shear modification value is 5/6 = 1/1.2, this gives the equivalency of the energy of the shear forces and shear stresses.

As it is mentioned above, if the D, K and H stiffness values are calculated and filled in, then it is possible to model the sections with complex geometry and composite materials with new fictitious shell object. A few sections are shown below which can be handled by the help of fictitious shell.

Corrugated steel + Concrete floor



Corrugated steel + Timber floor



Timber Panel

Quick tool to model timber slabs and wall panels is optionally available for later analysis in *3D Structure* and *PreDesign* modules and timber design in .

Definition steps

1. If needed, set a proper position for the **working plane**.
2. Choose or according to the panel function to define a new timber panel: *Use as plate* or *Use as wall*.
3. Set the panel settings under (plate) or (wall) *Default settings*.

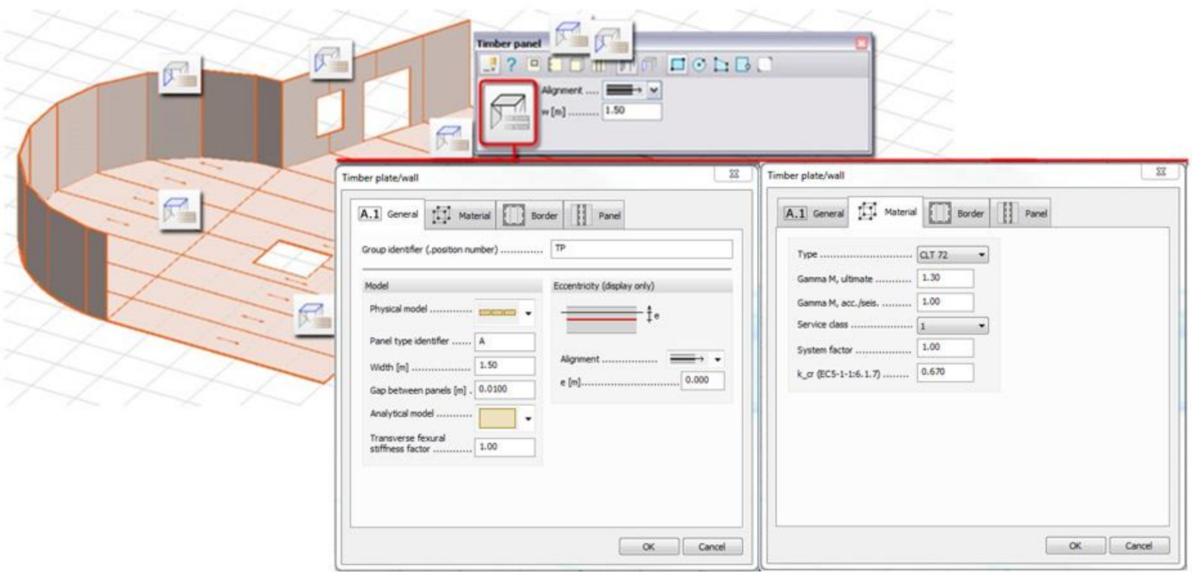


Figure: Timber command and settings

- *Group identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Panel type identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Width* (General)
  The width of the elements can be set if Prefabricated physical model is chosen.

- *Gap between panels* (General)
  The gap between panels can be set if Prefabricated physical model is chosen.

- *Alignment and Eccentricity* (General)
  With *Alignment* the position of the reference (definition) plane can be set to *Top/Left*, *Center* or *Bottom/Right*.

  Eccentricity adds "virtual" distance between the reference plane and the calculation (middle!) plane. This option is useful for displaying regions with different thickness values.

  Eccentricity (e) is handled the same was as in **3D Plate** .

  Eccentricity value can be displayed in model view (**Display settings**).

- *Type* (Material)
  Predefined panel types are available, but it can be expanded manually (*Edit library*) with new laminated, fiber boards, plywood etc. Design factors, stiffness and strength values can be edited for predefined and new panel types. Panel types can be imported or exported between users like the FEM-Design **materials**.

- *Physical model and Analytical model* (General)
  In Timber plate/wall dialog there is opportunity to select the physical model (in-situ or prefabricated), then analytical system of shell, if applicable. The analytical system can be Continuous or Panel by panel. This option is available in profiled plate and profiled wall.

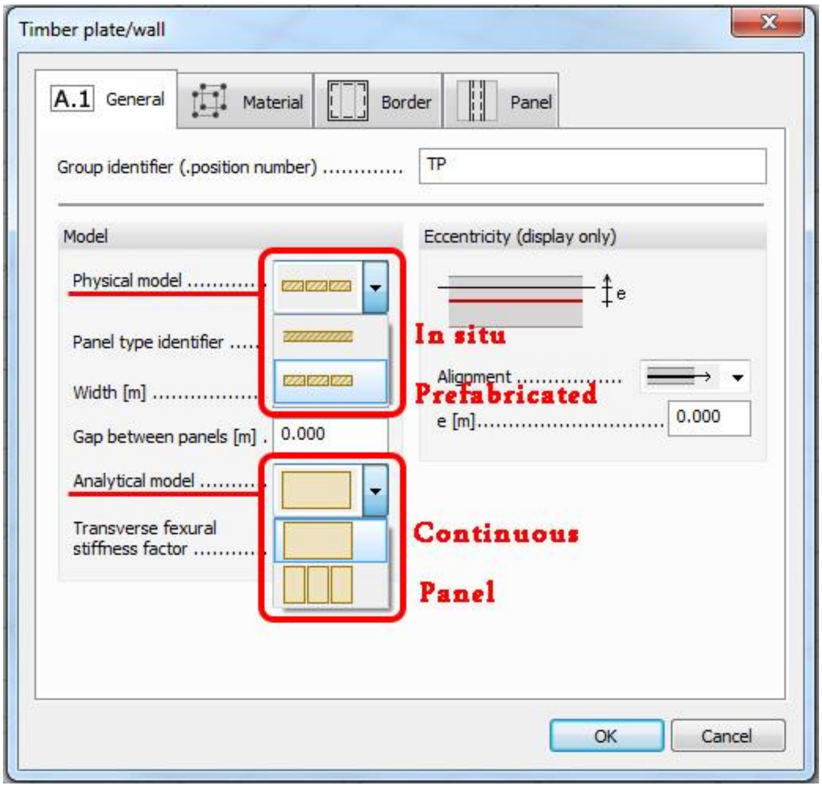


Figure: Physical model and Analytical system of a timber panel

Choosing continuous analytical system can be useful when modelling the whole structure e.g. in the preliminary design phase. The memory usage of calculation with continuous analytical system is less than panel by panel system due to the reduced number of edge connections.

The use of panel by panel analytical system is reasonable in the phase of detailed design. In this case the results of the calculation are more accurate.

In case of continuous analytical system, effect of the connections between the panels can be considered by the transverse flexural stiffness factor.

4. Set the width (w) of the selected panel type on the tool palette.
5. Set the default edge connection to hinged or rigid at *Edges* of the tool palette.
6. Define the timber plate/wall based on the **geometry** of the reference plane/base line.
   For plate panels, the stiff direction has to be defined first. (For walls, the stiff direction is always perpendicular to the wall base line.) The distribution of panels is described with an anchor point.

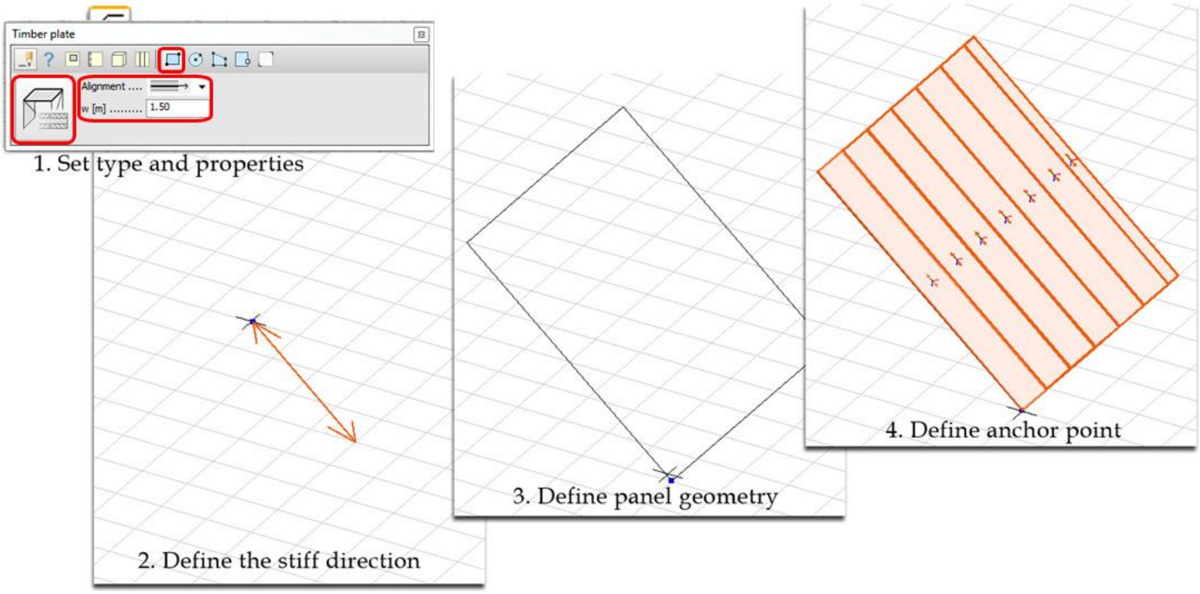


Figure: Timber plate definition

Optional steps:

7. The panel type of selected timber panels can be modified subsequently with the *Properties* tool.
8. Modify the default edge connections of the unique panel or panel group with . Just select the plate/wall panel and then its edge(s) to set connection conditions at Border (DOF of the panel group) or Panel tab (rigid, hinged, free or semi-rigid (spring)), and finally set the requested motion and rotation settings. **See the definition steps and possibilities at Plate.**

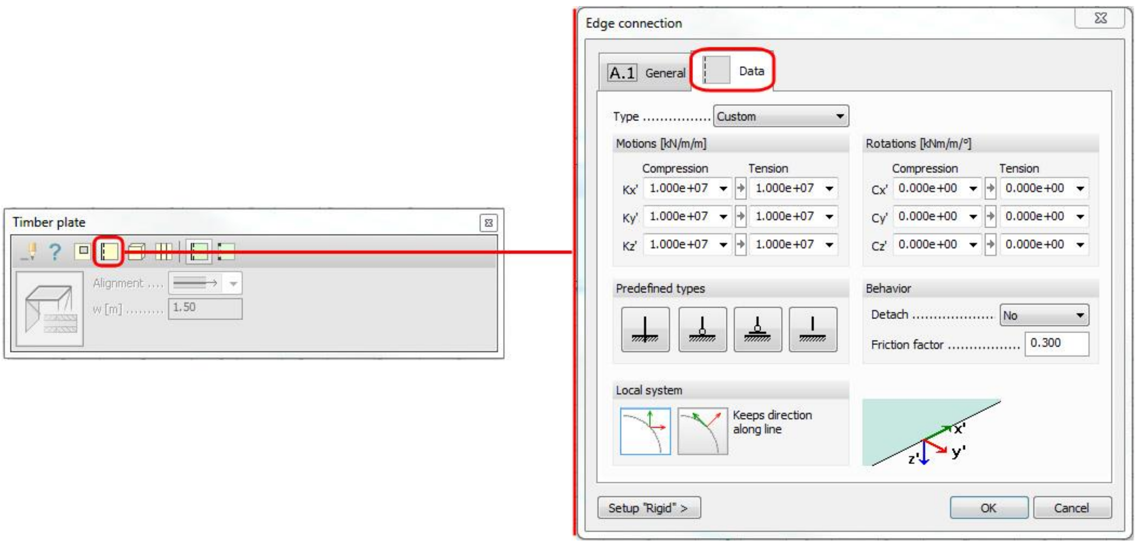


Figure: Panel edge connections

9. Openings and cuttings can be added to timber panel elements with *Edit* > *Region operations* > **Split regions**.
10. Set the display settings of timber panels at *Settings* > *All* > *Display* > *Shell* (**see Plate display settings**).

11. The panels are stored on "Plates" **Object layers**. At layer settings, the default color and pen width can be set for all panel regions. The color and pen width settings by selected panel elements can be modified by *Edit* > **Properties** > *Change appearance*.

Profiled plate/wall

Quick tool to model prefabricated slabs and wall panels is optionally available for later analysis in *3D Structure* module and the designing in .

Definition steps

1. If needed, set a proper position for the **working plane**.
2. Choose or according to the panel function to define a new profiled panel: *Use as plate* or *Use as wall*.
3. Set the panel settings under (plate) or (wall) *Default settings*.
   - *Identifier* (General)
     The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

   - *Alignment and Eccentricity* (General)
     With *Alignment* the position of the reference (definition) plane can be set to *Top/Left*, *Center* or *Bottom/Right*.

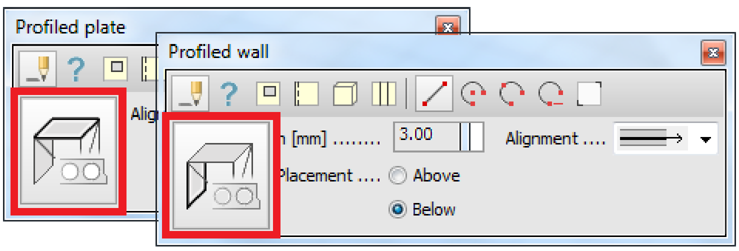
     Eccentricity (e) is handled the same was as in **3D Plate** .

     Eccentricity value can be displayed in model view (***Display settings***).

   - *Section* (Section)
     Predefined panel types are available in Section library in *Hollow Core*. Cross-section types can be imported or exported between users like the FEM-Design ***materials***.

   - *Physical and analytical model* (General)
     In Profiled plate/wall dialog there is opportunity to select the physical model (in-situ or prefabricated), then analytical system of shell, if applicable. The analytical system can be Continuous or Panel by panel. This option is available in profiled plate and profiled wall.

   - *Camber simulation by prestressing* (General)
     See details at **Beam**.



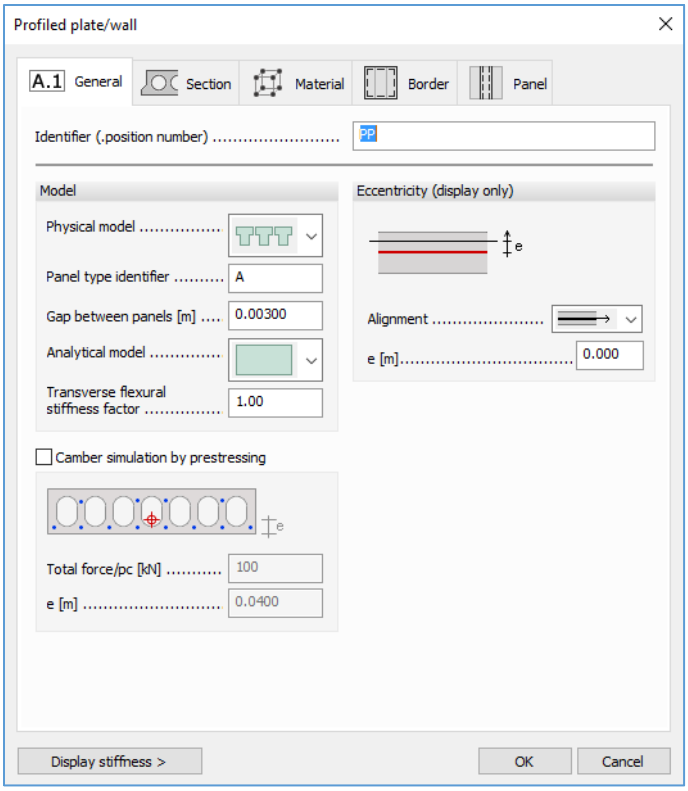


Figure: Physical model and Analytical system of a prefabricated panel

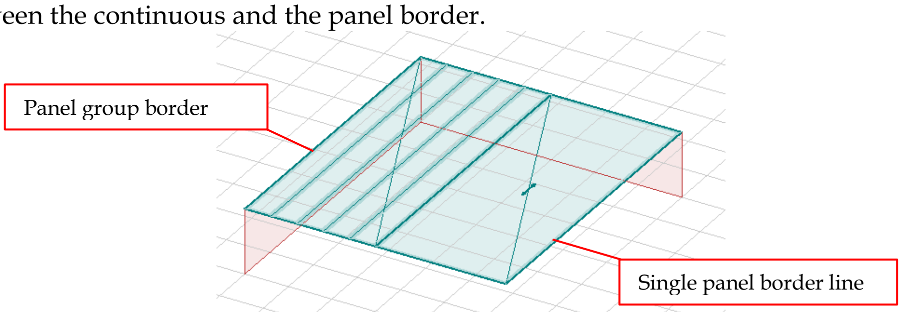
Choosing continuous analytical system can be useful when modelling the whole structure e.g. in the preliminary design phase. The memory usage of calculation with continuous analytical system is less than panel by panel system due to the reduced number of edge connections.

The use of panel by panel analytical system is reasonable in the phase of detailed design. In this case the results of the calculation are more accurate.

In case of continuous analytical system, effect of the connections between the panels can be considered by the transverse flexural stiffness factor.

4. Set the panel group's edge connection in *Border* tab.

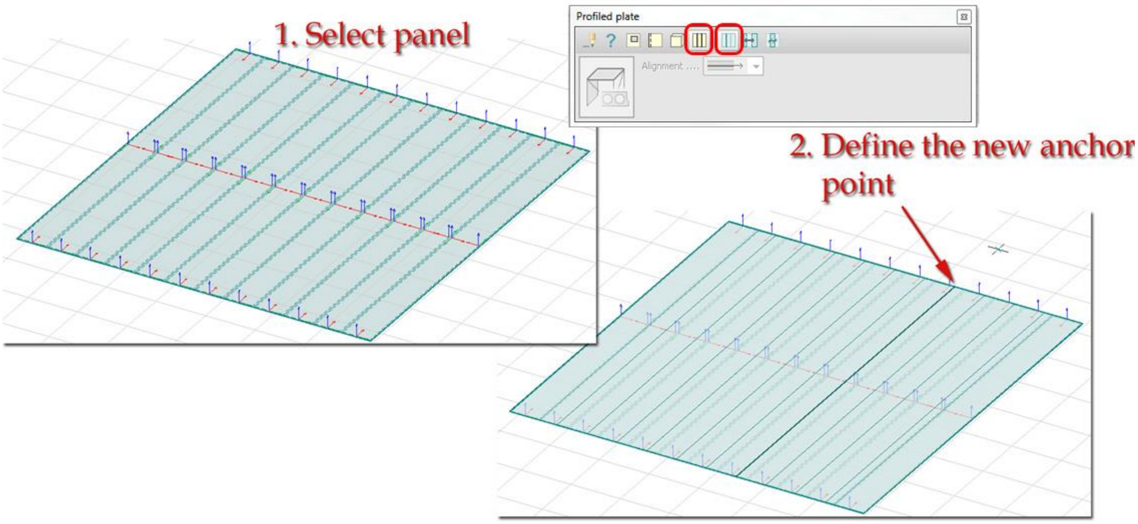
5. Set the unique panel's edge connection in *Panel* tab. The following figure shows the difference between the continuous and the panel border.



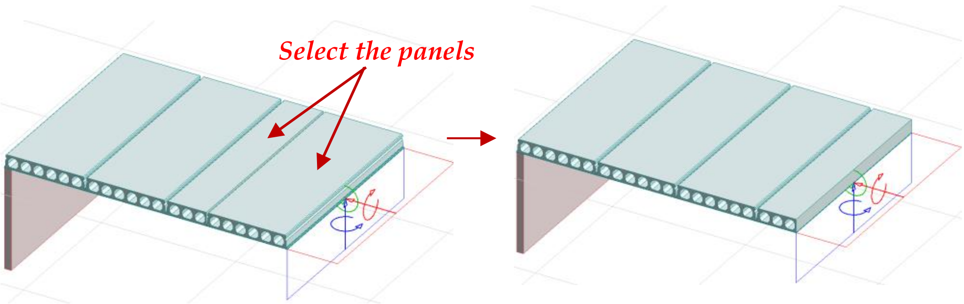
6. Define the profiled plate/wall based on the **geometry** of the reference plane/base line.
For plate panels, the stiff direction has to be defined first. (For walls, the stiff direction is always perpendicular to the wall base line.) The distribution of panels is described with an anchor point. Defining is similar to **Timber panel definition.**

Optional steps:

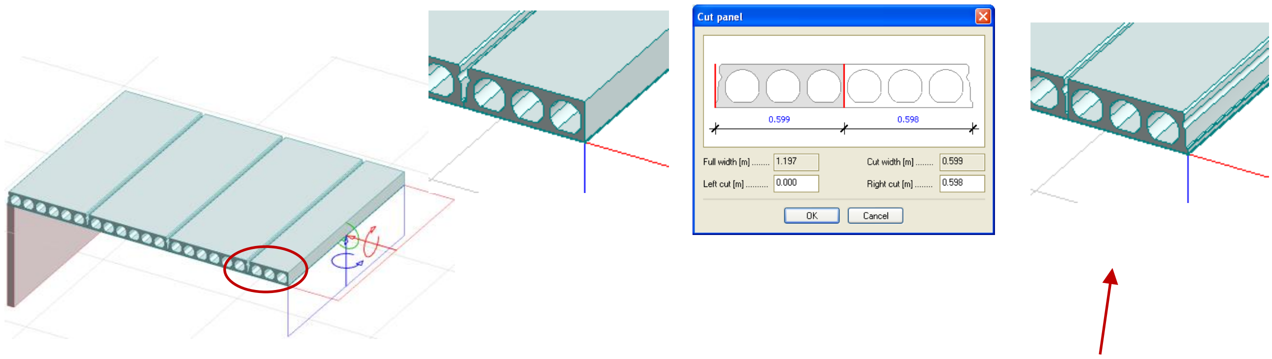
7. The panel type of selected profiled panels can be modified subsequently with the *Properties* tool.
8. Modify the default edge connections of the unique panel or panel group with . Just select the plate/wall panel and then its edge(s) to set connection conditions at Border (DOF of the panel group) or Panel tab (rigid, hinged, free or semi-rigid (spring)), and finally set the requested motion and rotation settings. **See the definition steps and possibilities at Plate.**
9. Openings and cuttings can be added to timber panel elements with *Edit* > *Region operations* > **Split regions**.
10. Set the display settings of timber panels at *Settings* > *All* > *Display* > *Shell* (**see Plate display settings**).
11. The panels are stored on "Plates" **Object layers**. At layer settings, the default color and pen width can be set for all panel regions. The color and pen width settings by selected panel elements can be modified by *Edit* > **Properties** > *Change appearance*.
12. Modifying the anchor point it's possible with *Base line* command:



13. With *Swap panel* two panel placement can be change which one panel is a truncated panel.



14. *Cut panel* will change the placement of the truncated panel.

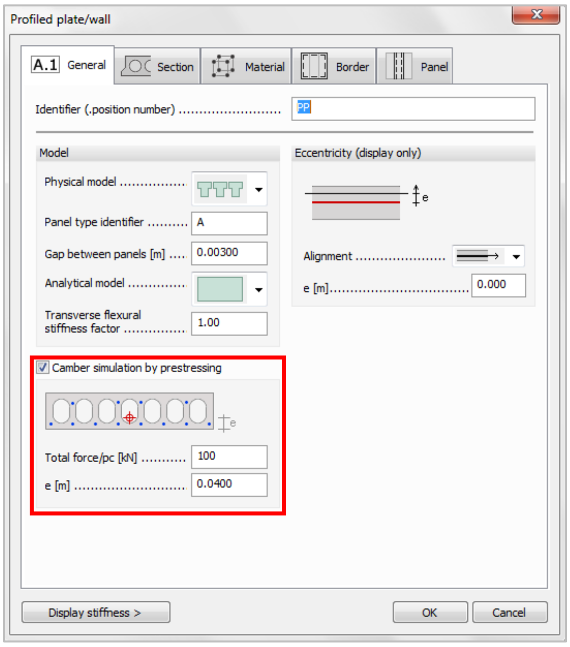


1. Select the truncated

2. Define the cut placement

New cutted panel

15. To simulate camber of the elements check the box next to *Camber simulation by prestressing* in the *Default setting/General tab.*



Calculation of Profiled shell

Profiled shells are calculated as **Fictitious shells** (the stiffness matrices are calculated automatically, according to the tables below).

$$\mathbf{D} = \begin{bmatrix} \frac{Eh_x}{1-\nu^2} & \nu\sqrt{\frac{Eh_x}{1-\nu^2}\cdot\frac{Eh_y}{1-\nu^2}} & \\ \nu\sqrt{\frac{Eh_x}{1-\nu^2}\cdot\frac{Eh_y}{1-\nu^2}} & \frac{Eh_y}{1-\nu^2} & \\ & & G\frac{h_x+h_y}{2} \end{bmatrix}$$

$$\mathbf{K} = \begin{bmatrix} \frac{EI_x}{1-\nu^2} & \nu\sqrt{\frac{EI_x}{1-\nu^2}\cdot q\frac{EI_y}{1-\nu^2}} & \\ \nu\sqrt{\frac{EI_x}{1-\nu^2}\cdot q\frac{EI_y}{1-\nu^2}} & q\frac{EI_y}{1-\nu^2} & \\ & & G\frac{\left(\frac{h_x+qh_y}{2}\right)^3}{12} \end{bmatrix}$$

$$\mathbf{H} = \begin{bmatrix} \rho_x G h_x & \\ & \rho_y G h_y \end{bmatrix}$$

Section data:

h_x, h_y : equivalent thickness in x and y direction

I_x, I_y: equivalent inertias in x and y direction

ρ_x, ρ_y: equivalent shear factor in x and y direction

x direction: strong axis of the panel cross-section

y direction: is perpendicular to the x direction in the plane of the panel

Material data:

E: Young modulus

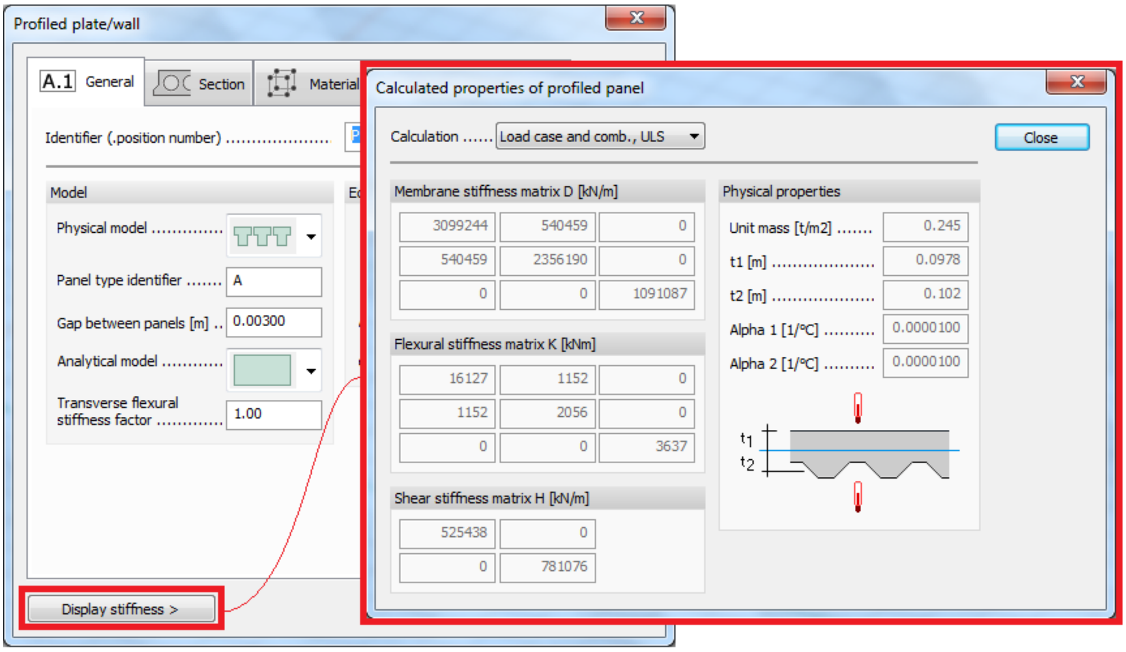
G: Shear modulus

ν: Poisson's ratio

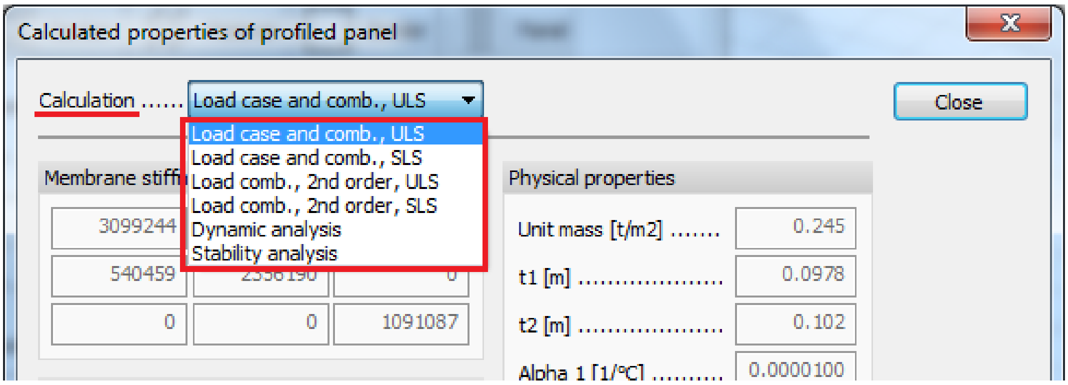
Panel data:

q: transverse flexural stiffness factor (applicable only in Continuous analytical system)

Modifying any above mentioned data will affect the stiffness of shell. The stiffness of shell is calculated automatically. The value of the stiffness (derived from properties of the shell) can be checked by clicking on the "*Display stiffness*" button.



Different types of Young moduli (except for steel material) can be set in different types of calculations. It has a direct effect on the values of stiffness matrices in different types of calculations. The stiffness values can be easily checked in "*Display stiffness*" dialog.



This feature gives the possibility to automatically calculate stiffness of concrete shell having ***complex geometry***. Some typical examples are shown below (sections are made in section-editor).

Components

This chapter introduces the component objects which can not exist bz themselves only as part of an other object (bar, shell)

Intermediate section

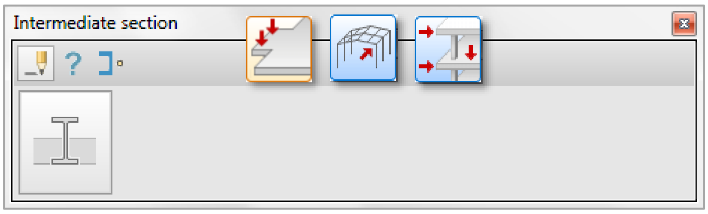
Intermediate section property	Description
Modules where available	
Axis position	Horizontal in
	Arbitrary (horizontal, vertical and skew) in
Cross-section	Arbitary
Eccentricity	Available

Material	Steel, concrete, timber and general
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in

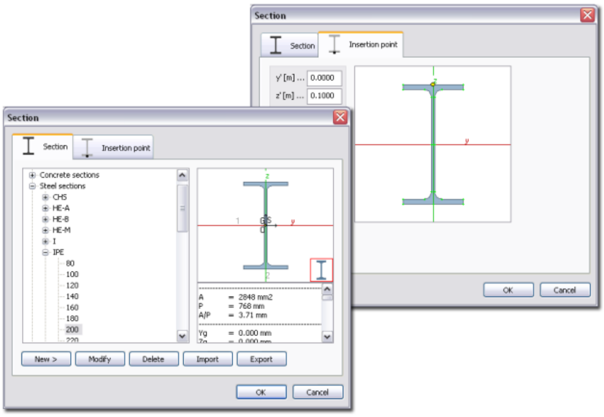
Definition steps

1. If needed (and available in the current FEM-Design module), set a proper position for **working plane**. In all modules, respect that the gravity direction is always the Global Z axis direction.

2. Start *Intermediate section* from Structure tabmenu and choose *Define*.



3. Set the properties of the new Intermediate section at *Default setting*:



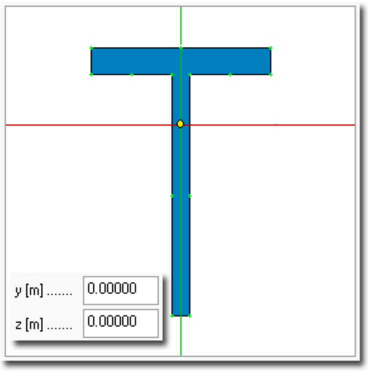
- **Section**

Choose a beam profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (**Display settings**).

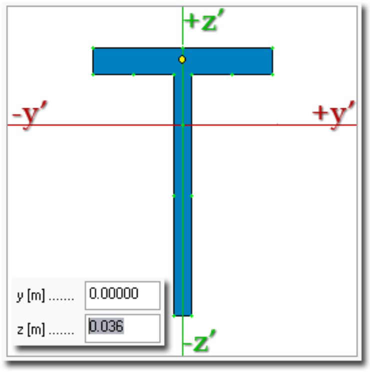
⚠ Intermediate section can be used with the same section shape and fabrication technology.

- *Insertion point*

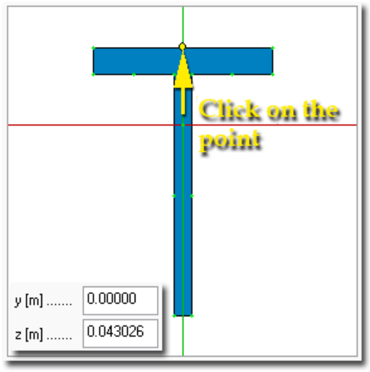
The axis default position is on the gravity center (symbolized with yellow point in the section), but it can be set with its coordinate in the local y'z' plane, or can be placed into special section points (e.g. plate corners) by clicking the yellow point into the requested position.



Default: "center of gravity"

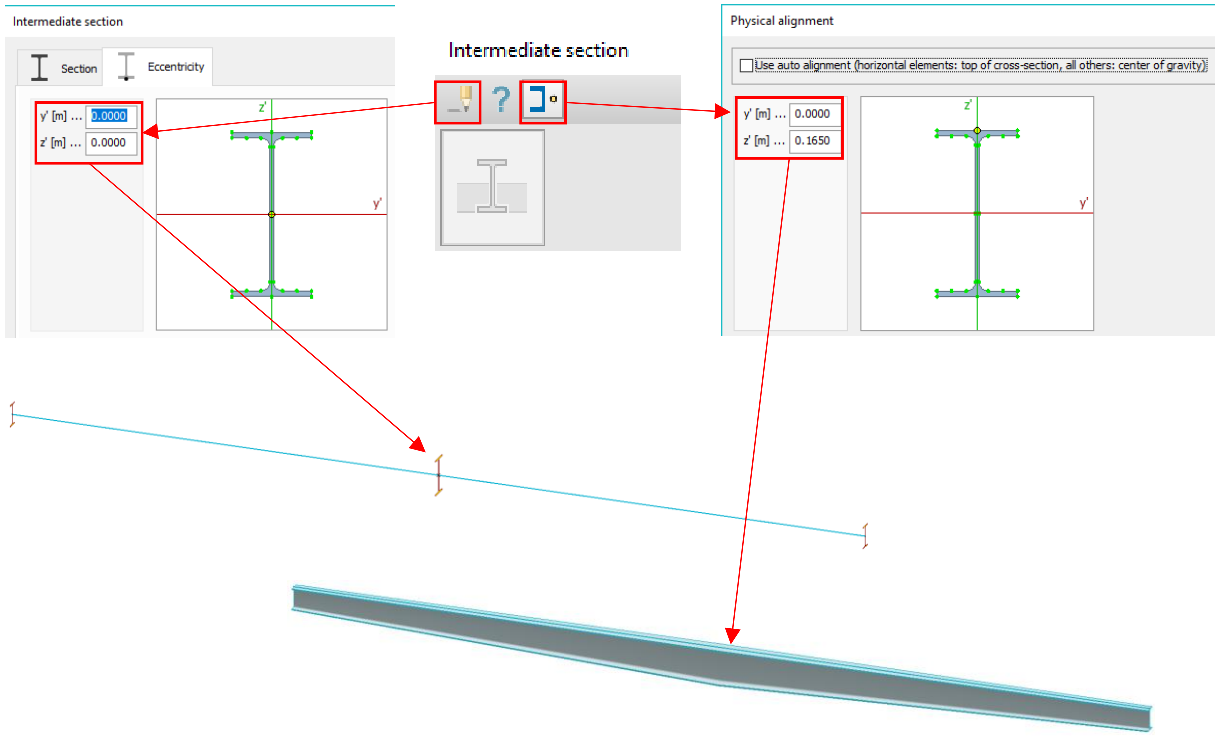


Position modified by distances



Position modified by the mouse

You can manage the physical and analytical eccentricity separately for intermediate sections:



4. Define the direction of the cross-section y' axis (only in 3D modules).

   The y' axis cannot be modified in the  module, so it is always parallel with the Global X-Y (working) plane.

5. Define the *Intermediate section* placement.

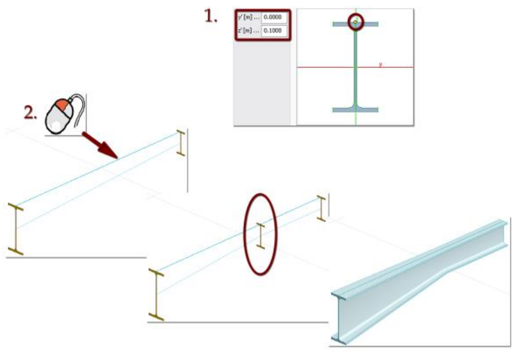


Figure: Defining an Intermediate section

Optional steps:

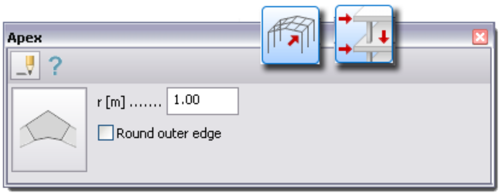
6. Modify the *Intermediate section* with the *Properties*.
7. The *Intermediate section* is part of the bar element so its layer and setting can be found like in *Beam, Column or Truss*.

Apex

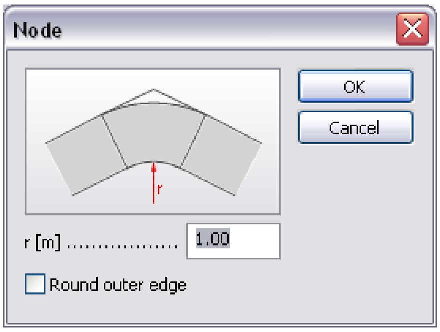
Apex property	Description
Modules where available	
Axis position	Arbitrary (horizontal, vertical and skew) in
Cross-section	Rectangular
Eccentricity	Disable
Material	Steel, concrete, timber and general
Load direction	Arbitrary in
Load type	All point and line load
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in
Available design	**Timber design** in

Definition steps

- If needed (and available in the current FEM-Design module), set a proper position for **working plane**. In all modules, respect that the gravity direction is always the Global Z axis direction.
- Start *Intermediate section* from Structure tabmenu and choose *Define*.



- Set the properties of the new Intermediate section at  *Default setting*:



- Rounding radius (r)

  The rounding radius can be invidually set. The default value is 1.00 m and rounding the inner edge. For rounding the outer edge check the *Round outer edge* option on. The following figure shows the difference between rounding the inner or both edges.

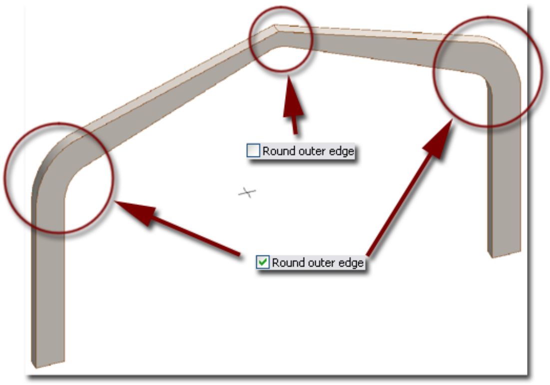


Figure: Rounding inner and both edge

- Select the neighboring bar element.

  Defining the correct way an apex, the following criteria has to meet:

  - The neighboring bars cross-section has to be **Rectangular**
  - Their Y axes are parallel and point into the same direction
  - Their material has to be the same

- Apex zones with their connected bars are only one physical element but it has more analytical element.

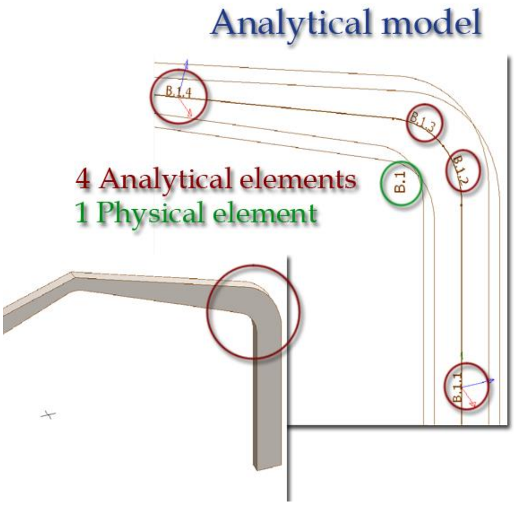


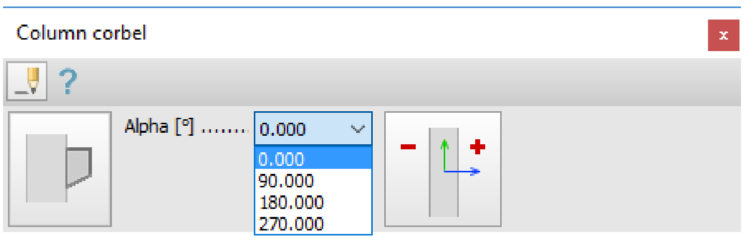
Figure: Describing analytical and physical elements

Column corbel

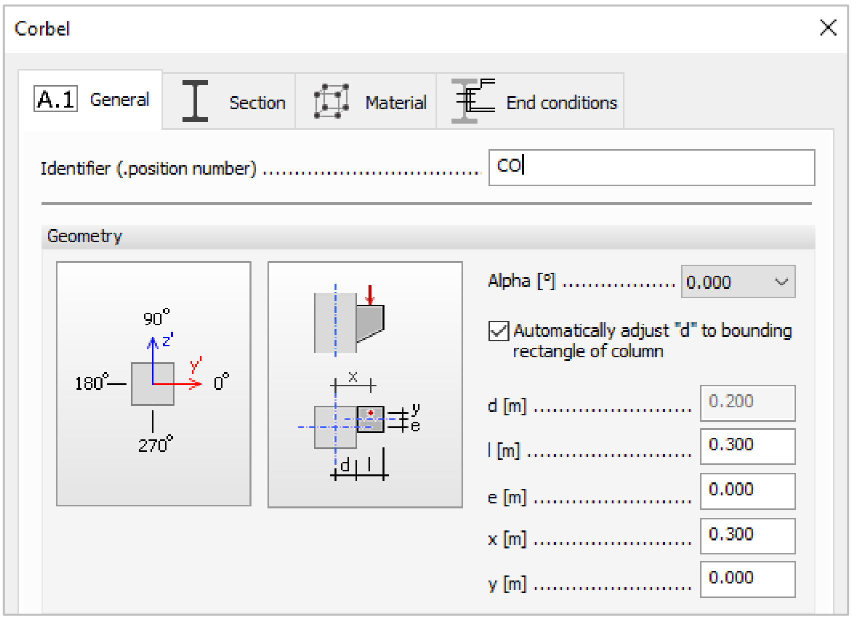
Column corbel property	Description
Modules where available	
Function	It models corbels attached to columns.
Position	Horizontal, perpendicular to the local y′ or z′ axis of the column it is attached to.
Geometry	Straight
Cross-section	Arbitrary profile
End connection	- Always rigid at the column it is attached to (Start). - Arbitrary at the end point, set by the object attached to the corbel.
Material	Concrete, steel, timber
Load direction	Arbitrary in
Load type	All point and line load in
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in
Alternative	Can be considered as eccentricity of the load acces point on the column, or can be modelled as a short beam with the suitable end conditions and eccentricity.

Definition steps

1. Start *Column corbel* command from Bar component menu in Structure tab
2. Set the direction of the Column corbel. A Column corbel is always perpendicular to the local x′ axis and parallel with the local y′ or z′ axis of the column.



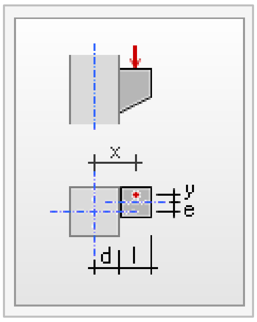
3. Set the properties of the new column corbel at *Default settings*



- *Identifier* (General)
  The program automatically generates it, but User can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Geometry* (General)
  User can specify the followings:

  - direction of the Corbel
  - relative position of the Corbel compared to the column (d, e)
  - length of the Corbel (l)
  - position of the load application point (x, e), marked with red dot in the figure below
  - whether to calculate the “d” value automatically in accordance with the Section of the column or not



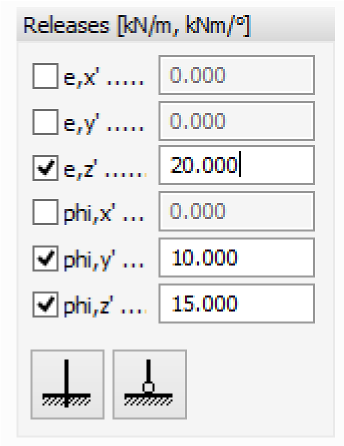
- **Section**
  User can set a profile from the Section library or define a new cross-section by parameters. The applied section name can be displayed in model view (**Display settings**).

- ***End conditions***

In 3D modules, the end connections can be set under the *End conditions* tab. Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**). End connections can be fully customised for bar ends by setting the rigidity of the displacement components.

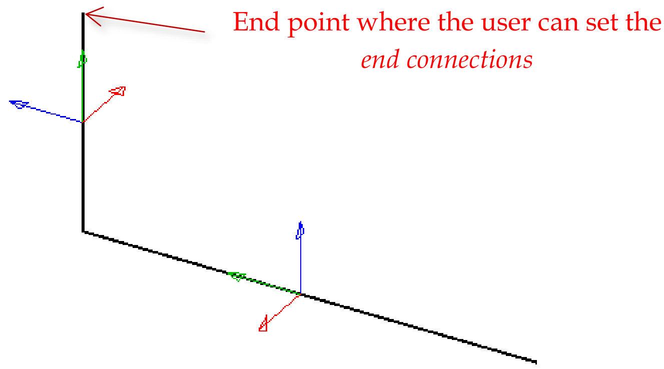
The component directions are valid in the beam's **local co-ordinate system**.

Inactive component means a fixed motion/rotation, while a checked component behaves according to the user-specified rigidity value.



With the „Rigid" and the „Hinged" buttons pre-defined end condition values can be applyed to the bar ends.

The local x' axis always points the vertical fictitious bar end, so the start and end point position can be followed from the bar's local co-ordinate system.



Only the released motion (arrow) and rotation (two-headed arrow) components are displayed in the model view (**Display settings**).

- **Material**
  Any type of materials can be set for corbel *Analysis*. The applied material name can be displayed in model view (**Display settings**).

4. Place the corbel along a column by left-clicking.

Optional steps:

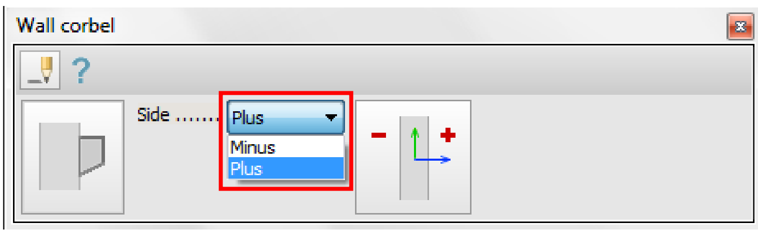
5. Modify the Corbel properties with the *Properties* tool of the *Corbel* tool palette.
6. Set the **display settings** of Corbel at *Settings* > *All* > *Display* > *Bar* similarly to *Beams*.
7. Corbels are stored on “Columns” **Object layers**. At layer settings, the default color and pen width can be set for all columns and corbels.

Wall corbel

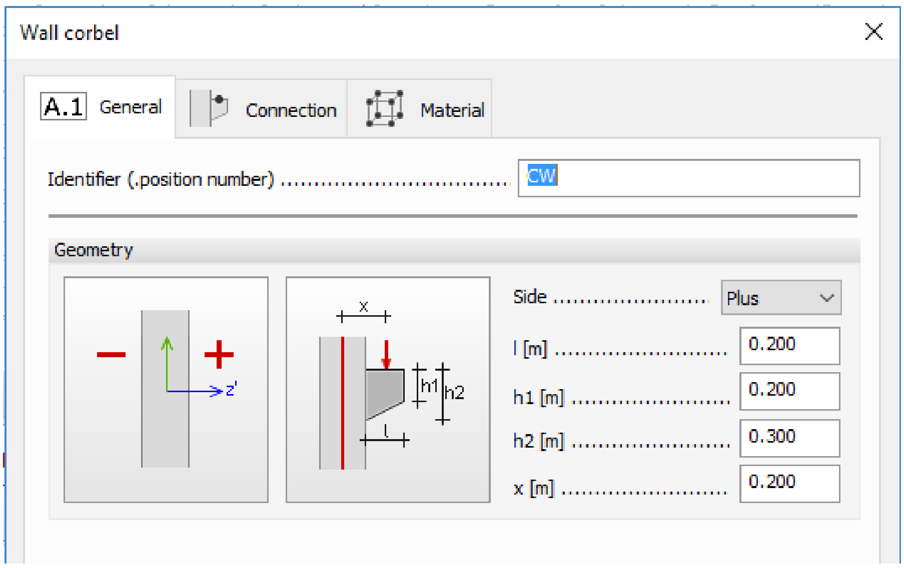
Column corbel property	Description
Modules where available	
Function	It models corbels attached to plane walls.
Position	Horizontal, attached to wall.
Geometry	Straight
Connection	- Always rigid at the wall it is attached to. - User-defined at the point where other object is attached to it.
Material	Concrete
Load direction	Arbitrary in
Load type	All point and line load in
Available analysis results	Displacement, internal forces, stresses, stability and vibration shape in

Definition steps

1. Start *Wall corbel* command from Shell component menu in Structure tab
2. Set the direction of the Wall corbel, whether the corbel should be adjusted to the positive or negative side of the wall according to its local coordinate-system.



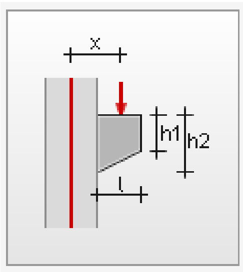
3. Set the properties of the new wall corbel at *Default settings*



- *Identifier* (General)
  The program automatically generates it, but User can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

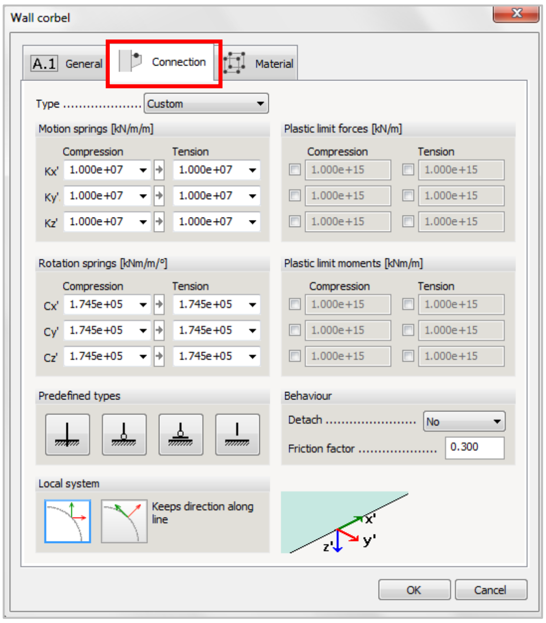
- *Geometry* (General)
  User can specify the followings:

  - the side of the wall
  - relative position of the Corbel compared to the wall (l, x)
  - length of the Corbel (l)
  - position of the load application (x), marked with red arrow in the figure below
  - the heights of the corbel at the wall and at the end of corbel



- ***Connection***
  In *Connection tab* the *Motion springs*, *Rotation springs*, *Plastic limit forces* and *Plastic limit moments* can be managed.

  With the „Rigid” and the „Hinged” buttons pre-defined end condition values can be applied to the corbel edge.



- **Material**
  Only concrete material can be set for wall corbel.

4. Place the corbel along a wall by drawing a line.

Optional steps:

5. Modify the Corbel properties with the *Properties* tool of the *Wall corbel* tool palette.
6. Set the **display settings** of Wall corbel at *Settings* > *All* > *Display* > *Bar* similarly to *Beams*.
7. Wall corbels are stored on "Walls" **Object layers**. At layer settings, the default color and pen width can be set for all walls and wall corbels.

Post-tensioned cable (PTC)

Post-tensioned cable property	Description
Modules where available	
Function	It is a structural component, modelled by equivalent load system.
Axis position	Arbitrary (horizontal, vertical and skew)
Geometry	Straight
Shape	Wavy with *Base points* and *Inflection places*
Result	- Radius of curvature - Angular deviation - Stress function with friction losses - Stress function with anchorage set slip losses - Stress function with elastic shortening losses - Stress function with all time dependent losses - Equivalent force for 1 strand - Equivalent force for PTC

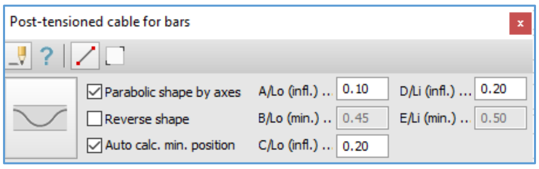
Manufacturing	the Manufacture drawing can be set with proper x′ and z′ shift

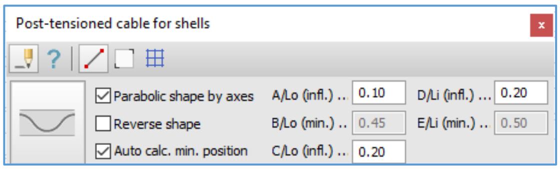
Table: Post-tensioned cable properties

Definition steps

1. Start *Post-tensioned cable for bars/Post-tensioned cable for shells* command from Structure tabmenu and pick the parent object (a bar or a shell), then define the line of the PTC.

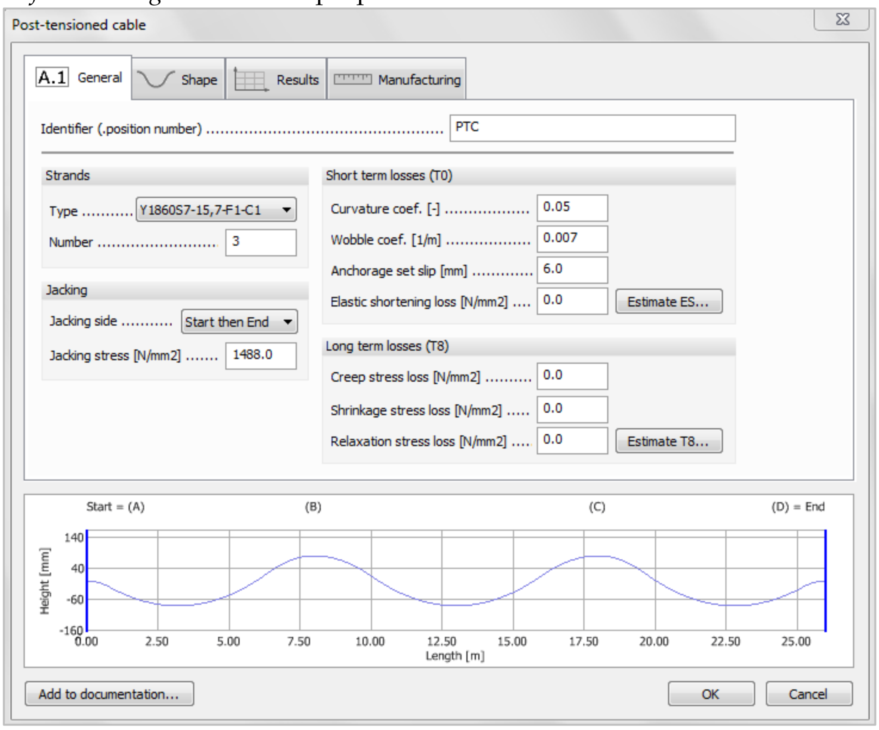
   The two *Tool window* work in slightly different ways: for shells, the user can define more cables by using *Line by selection* option, but for bars only one cable can be defined.





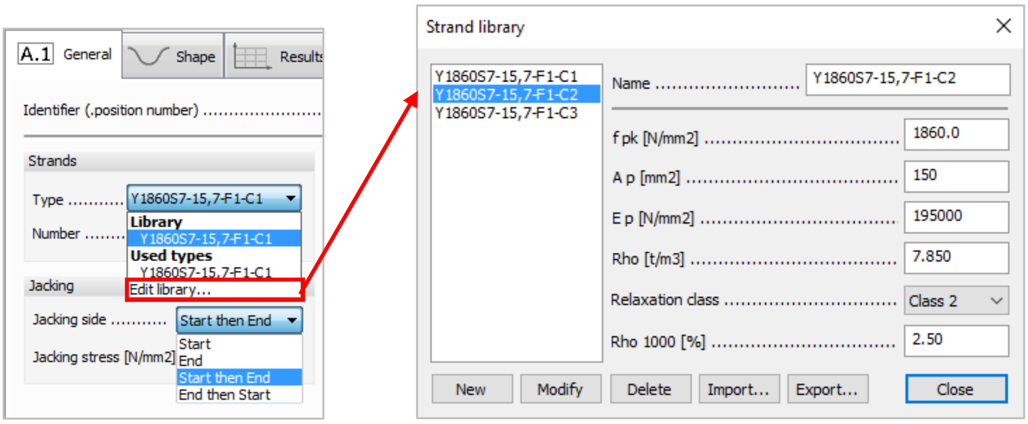
It is strongly recommended to use *Axes*: all aid functions (*Macros*, *Shape wizard*, *Layout wizard*) highly rely on them. It could greatly speed up the definition process.

2. Click *Default settings* and set the properties of the PTC.



- *Identifier* (General)
  The program automatically generates it, but you can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Strands* (General)
  User can set the type from the strand library and the number of the strands by typing.



- *Jacking* (General)
  User can select the Jacking side in the drop-down menu next to *Jacking side*. Jacking side options: Start, End, Start then End, End then Start. The last two are *Both-sided* jacking with same stress, but it was necessary to distinct, because the effect of draw-in could result in different stress functions using shorter cables.

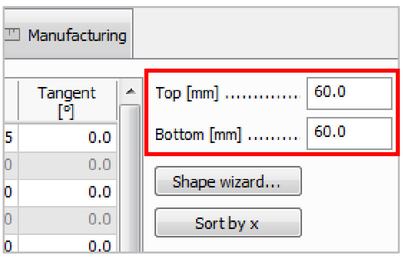
  Jacking stress is calculated as $0.8 * f_{pk}$ by default after the Strand type selection.

- *Short term losses* (General)
  - friction: It is estimated by EN 1992-1-1 5.10.5.2 (1) (formula 5.45) using the Wobble (k) and Curvature coefficients (μ):
  - $\Delta P_{\mu}(x) = P_{max}(1 - e^{-\mu(\theta + kx)})$
  - anchorage set slip
  - elastic shortening

- *Long term losses* (General)
  - shrinkage of structure
  - creep of structure
  - relaxation of post-tensioned cable

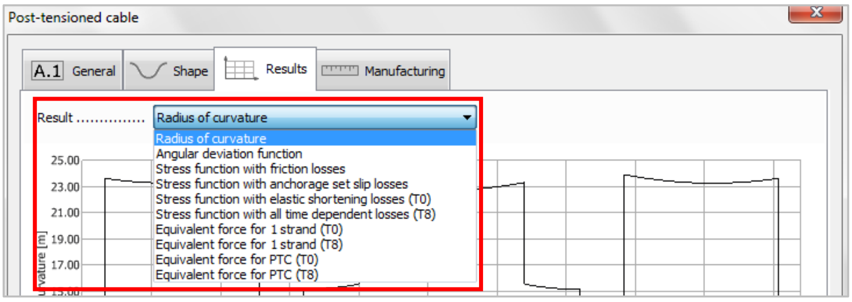
- Shape
  There are settings related to the geometry of the cable (**Display settings**) here. The Shape table can contain *Base points* and *Inflection places*: these determine whether linear or parabolic shape is applied.

  The *Top* and *Bottom* values mean the distance between the axis of the cable and the edge of the element.

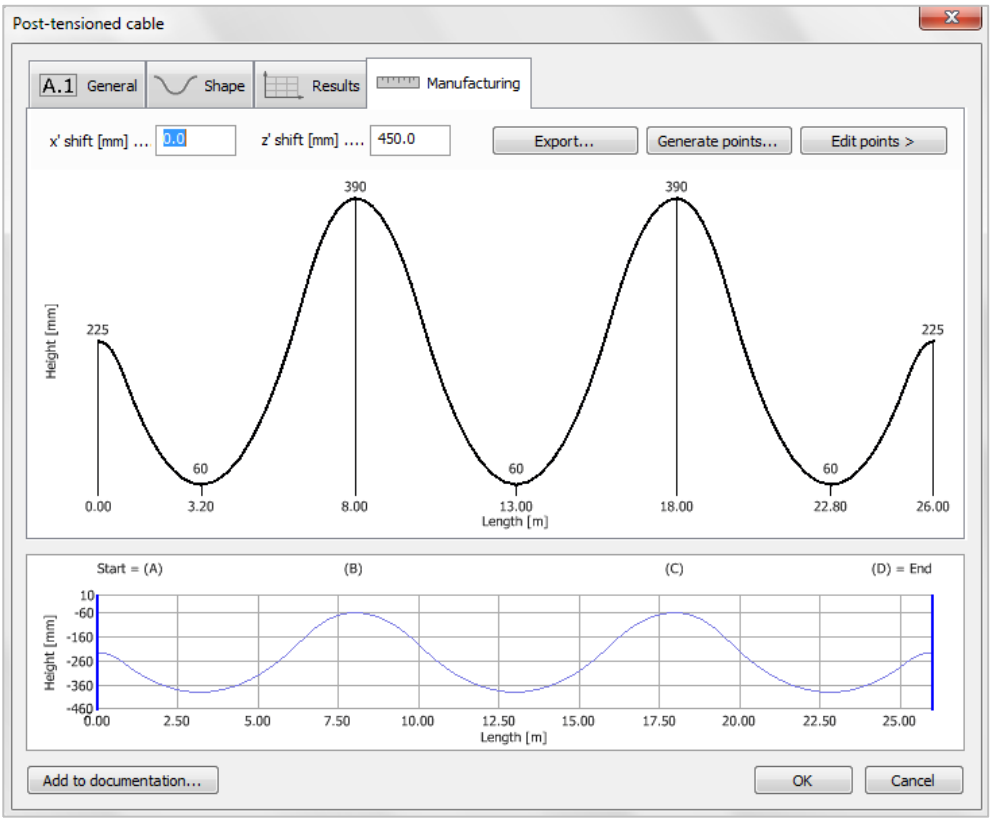


Using *Shape wizard* (*Shape wizard…* button) parabolic shapes can be easily created. This tool highly relies on previously defined axes.

- Results
  There are several results -listed in the order of calculation - which can be chosen from *Result tab/Result drop-down menu.*

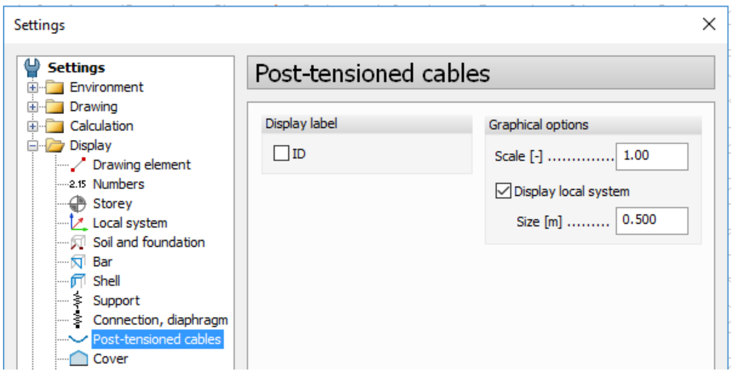


- Manufacturing
  On this tab the Manufacture drawing can be set with proper x′ and z′ shift.



Optional steps:

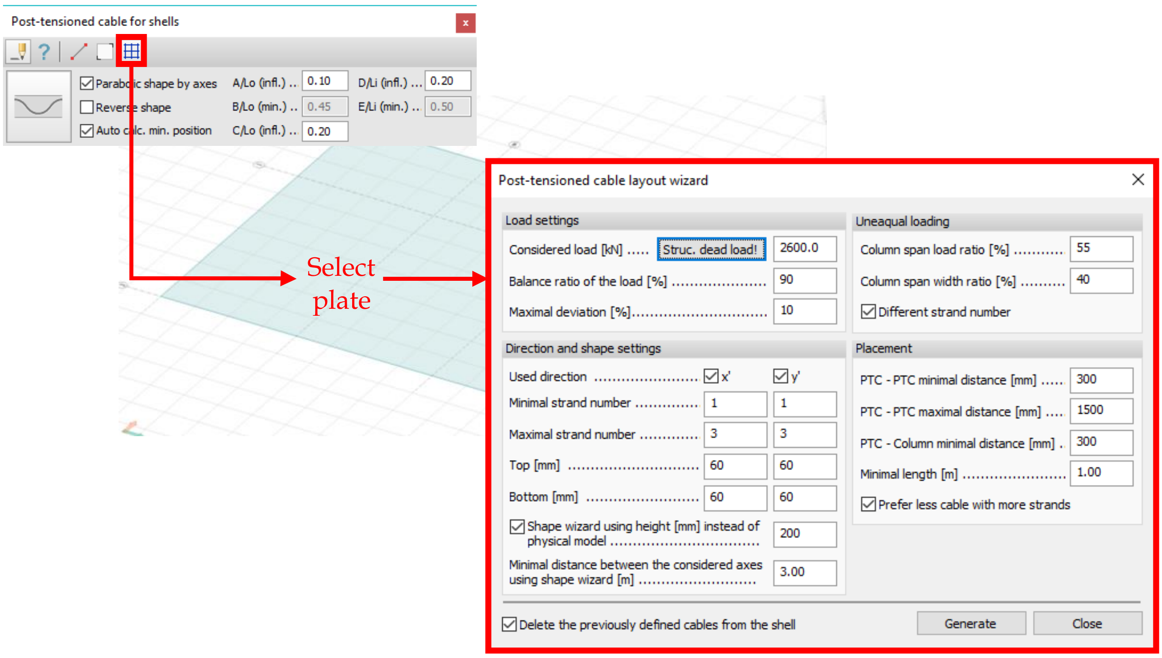
3. Modify the PTC properties with the  *Properties* tool of the *Post-tensioned cable bar/shell* tool palette.
4. Set the **display settings** of PTC elements at *Settings* > *All* > *Display* > *Post-tensioned cables*.



5. The PTCs are stored on "Post-tensioned cable" **Object layers**. At layer settings, the default color and pen width can be set for all PTC elements. The color and pen width settings by selected PTC elements can be modified by *Edit* > **Properties** > *Change appearance*.

- Layout wizard

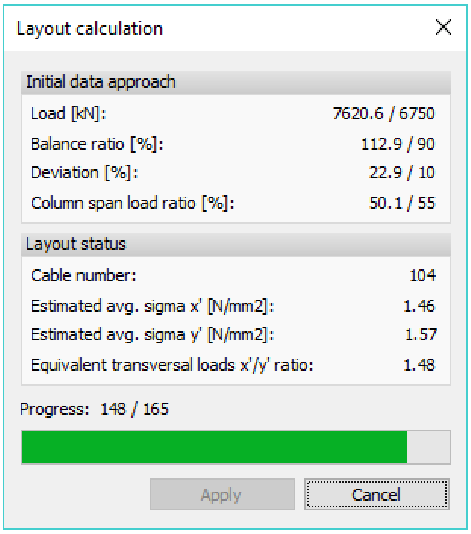
It is a tool to create a set of PTCs in a specific layout. It can be launched from *Structure/ Shell component/Post-tensioned cable/Layout wizard.*



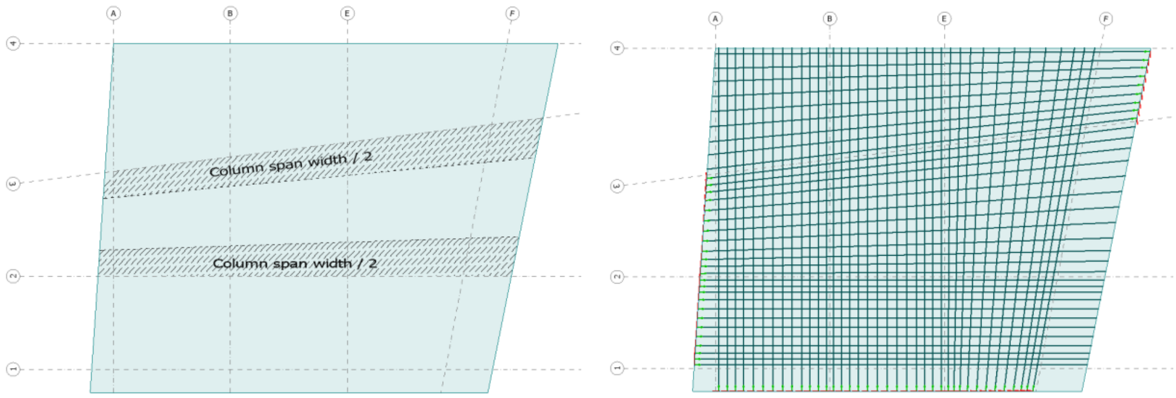
The *Layout wizard* searches for the closest solution to the *Unequal loading* settings from the PTC layout variations which fulfilled the following conditions:

- difference between the sum of the PTC's forces acting in local z+ direction and the product of *Considered load* and *Balance ratio* is less than the *Maximal deviation*
- geometrical requirements

The Layout calculation process shows the currently calculated layout parameters and a few indicator values. After the calculation it shows the parameters of the best found solution.



The function uses *Axes* to divide the selected structural element to stripes (Mid span and Column span) assuming columns in the axes cross points.



The algorythm can handle the holes of the plate regions.

At least two *Axes* crossing the plate are needed to use this tool.

Checking *Shape wizard using height [mm] instead of physical model* option can highly reduce the runtime if the selected plate's thickness is constant.

Supports and Connections

This chapter summarizes the possibilities and properties of the available supports and connections.

The connection with the ground (model space) can be modeled with *Supports*, while the link among structural elements can be defined with *Connections*. All support/connection components can be set to:

- "infinite" rigid: blocked motion/rotation,
- "free": released motion/rotation, or
- semi-rigid: given stiffness value (spring) against motion/rotation.

Non-linear behaviors, independent compression and tension, elastic and plastic behaviour settings are also available for all motion and rotation components.

Edge connection is a special tool of structural object definitions. Although it was described at planar objects (like **Plate**), the following connection settings and functions are also valid for *Edge connections*.

Support motion loads have to be assigned to supports, so supports are requested input parameters for motion-type loads.

Always try to create stable structural model by defining correct support and connection conditions. From unstable (kinematically indeterminate) structure equilibrium error can be resulted. In that case the program sends a warning message at the end of the calculations, and you can determine the location of the problems by checking the equilibrium (*Analysis* > **Equilibrium**), the displacement/buckling and/vibration shape results of the structure. It may also happen that the model is so incorrect that calculations stops with error messages and without results.

"Infinite" Rigidity

The value of "infinite" rigidity can be defined and set as project default by support/connection types and components at *Settings* > *All* > *FEM*. Go through the *Motion* and *Rotation* components by support and connections types and set the requested stiffness value that model blocked absolute/relative displacement.

The "rigid" values can be set to default for further projects, if select *"Rigid" values* in the *Settings* tree and click *Save as default*.

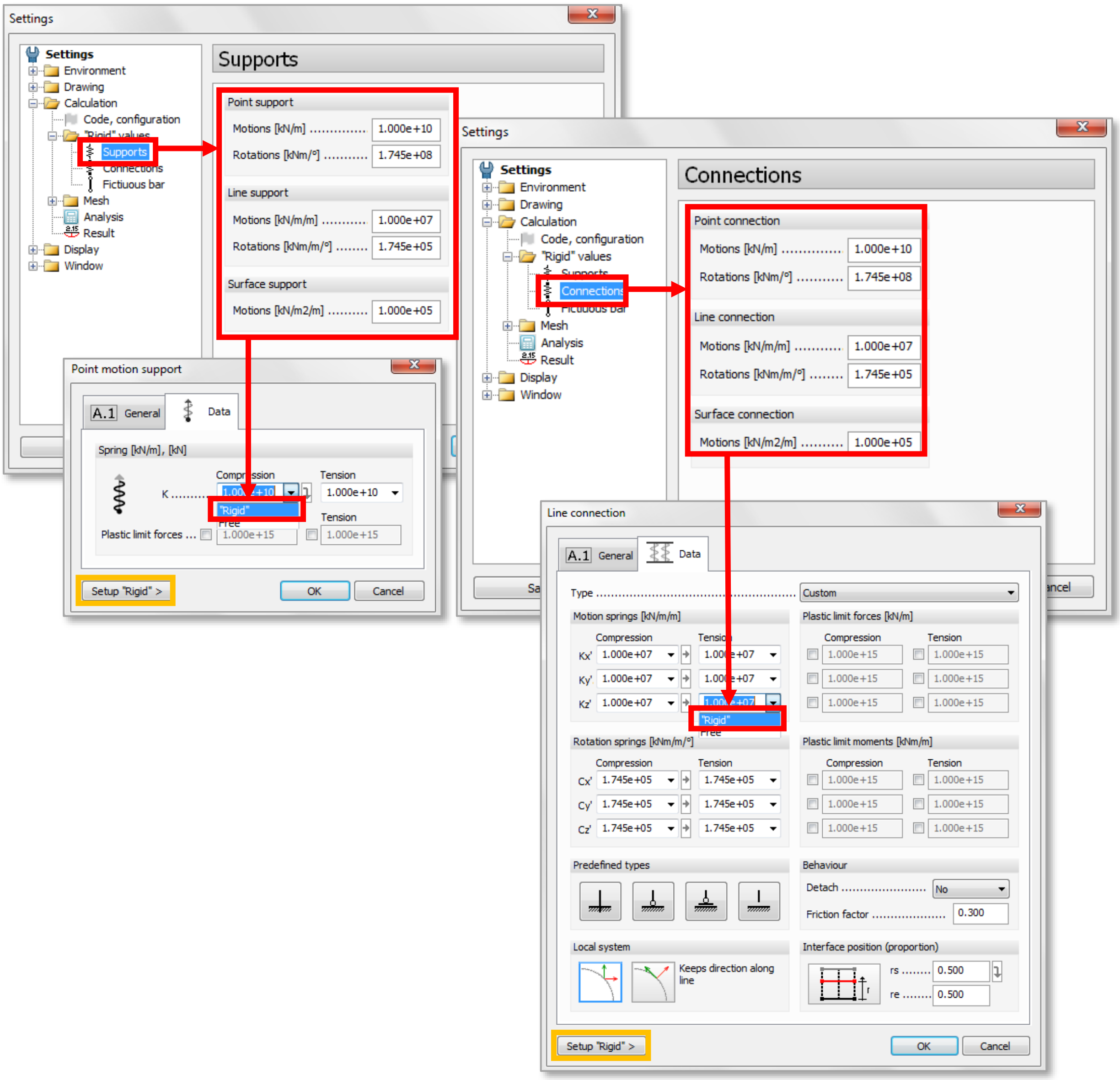


Figure: “Rigid” values

The “rigid” value can be set by support/connection/fictitious bar element inside its settings dialog (*Setup “Rigid”*) independently of the project default.

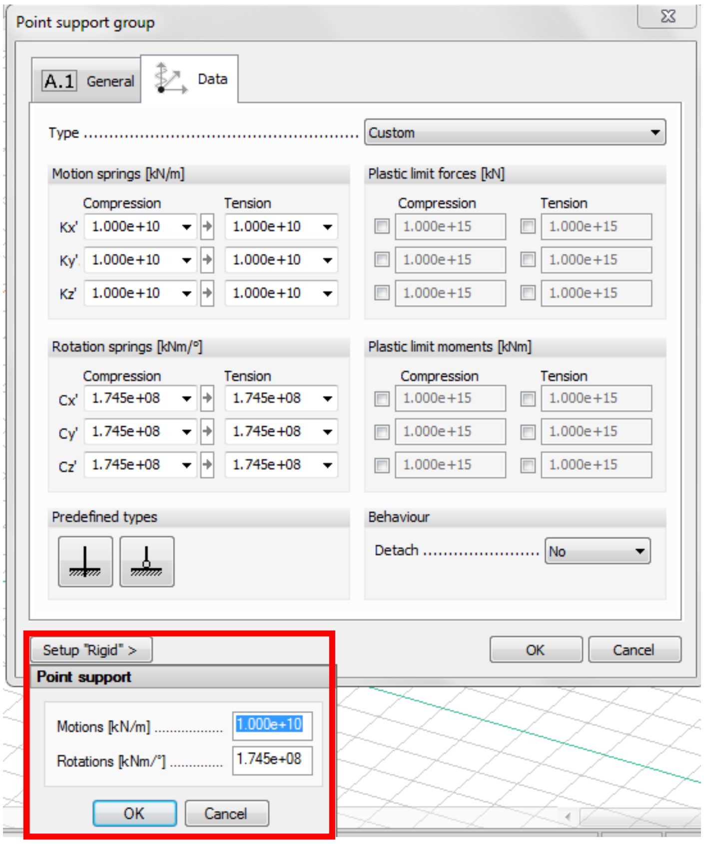


Figure: "Rigid" value set by support objects

Properties (Non-Linear Behaviors)

Compression, tension, elastic and plastic behavior of supports and connections can be set separately and by components.

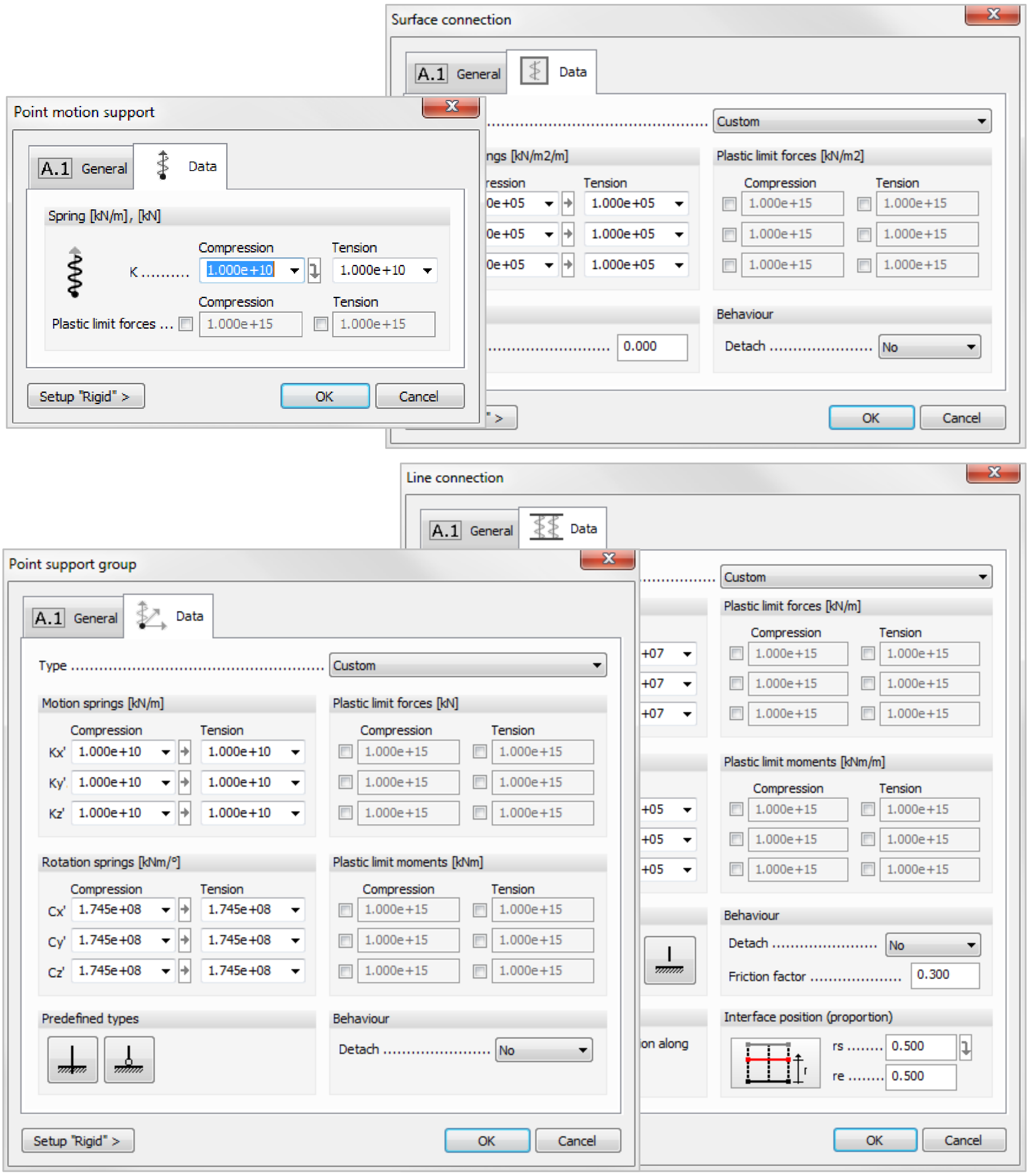


Figure: The parts of typical support/connection settings dialogs

- **Stiffness fields by compression and tension**
  "Rigid" or *Free* can be chosen from the pop-ups, or stiffness value (spring support/connection) can be typed in the fields.

Active icon assigns the stiffness value typed in the *Compression* field to the *Tension* field. Inactive icon lets to define the stiffness values separately.

- **Predefined types**

Click to set all motion and rotation components to “rigid”. The result will be a totally rigid support/connection.

Click to set all rotation components to “free” (allow rotation around all directions). The result will be a hinged support/connection.

Click to set all components to free. This tool gives the possibility to virtually connect independently moving elements.

“Uplift” can be modeled both in 2D and 3D design modules by defining compression-only *support / connection* (tension = 0 (free)) and by checking the **Consider non-linear behavior of supports, trusses and connections** box at *Calculate> Analysis*.

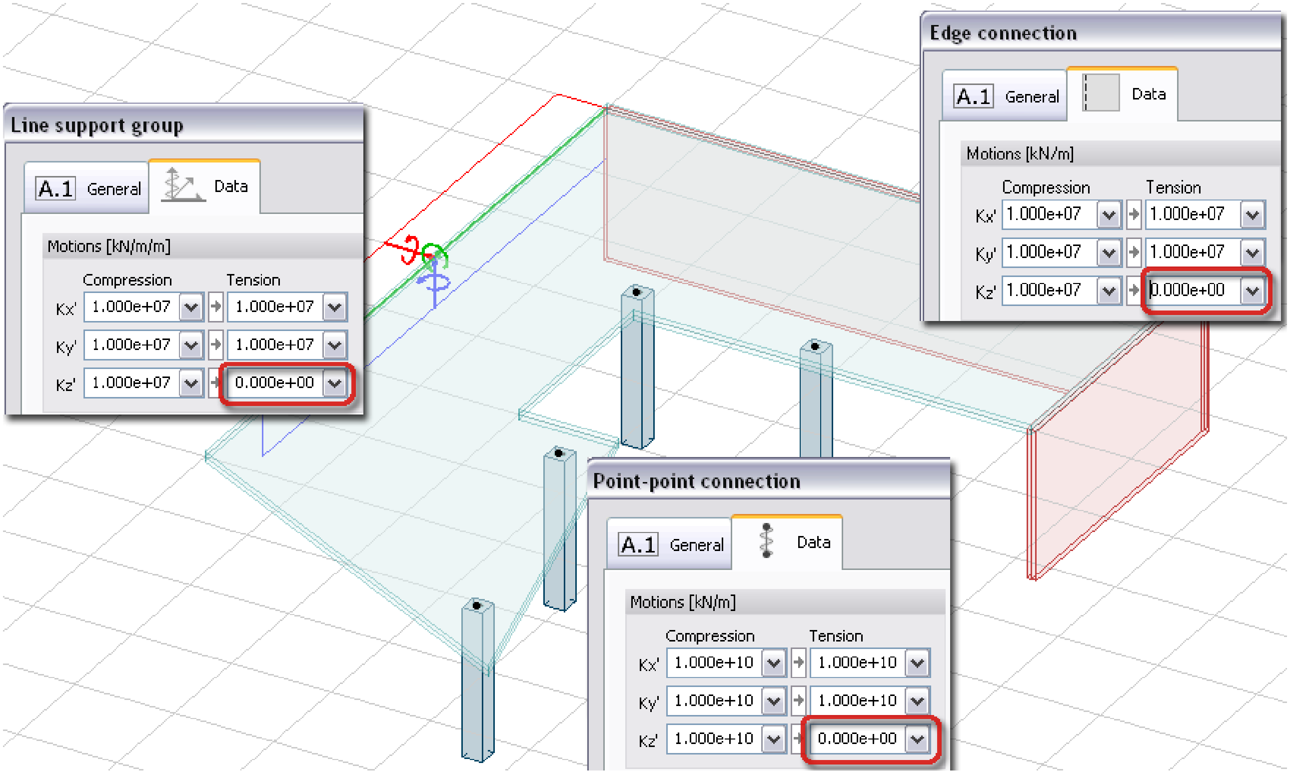
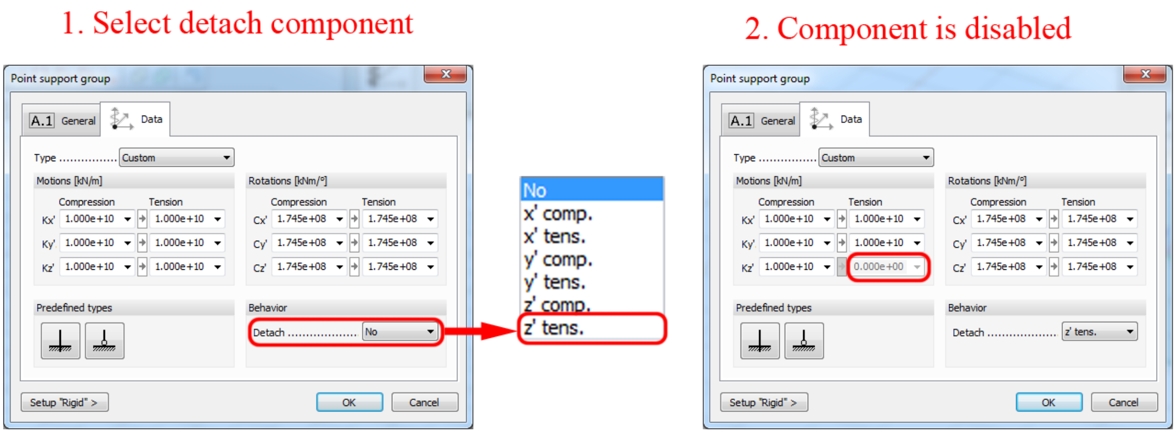


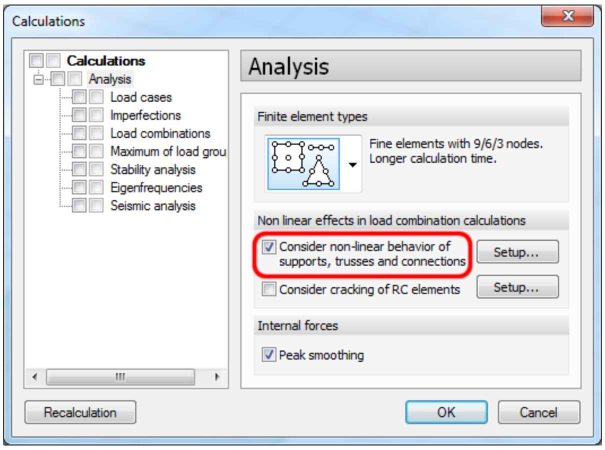
Figure: Allow uplift in Plate and 3D Structure modules

- **Using “Detach” option**

The nonlinear behavior of supports and connections (*Point support*, *Point support group*, *Line support*, *Line support group*, *Surface support*, *3D* Plates and Walls *Edge connection*, *border* and *panel* connection of *Timber panels* and *Profiled panels*) can be controlled by one signed component. It means, if the force in the connection / support happen to act in this selected direction, all spring constants will be set to 0, so called it is detached. This option can be set in the dialog, where the spring coefficients are set. If in the *Detach* list anything is selected (e.g. z’ tension) the corresponding spring constant will be automatically set to 0.



If in Calculations dialog the "*Consider non-linear behavior of supports, trusses and connections*" option is checked the calculation method is the following: If in the connection where the detach behavior is defined (e.g. z' tension) the connection force is tension in the given direction (z') in an iteration step all of the other spring constant will be set to 0. If in any later iteration step the connection force will be compression, the spring constants will be set to the previously defined values.



There is a big difference between setting any of the components to zero or selecting a detach behavior. If the user sets manually a component to zero, in the analysis the connection or support will not have rigidity in that direction.

Supports
Support types

The following table summarizes the available support types and their main features in the different FEM-Design Modules.

⚠ **Column** and **Plane wall** are developed to model single point and line support in *Plate* module. These types of support are not mentioned in the *Support* chapter.

Point support property	Description
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in
	Arbitrary direction in
Geometry	- (insertion point)
Available analysis results	**Reaction forces and moments**
Load can be assigned	**Point support motion load**
Point support group property	
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in
	Arbitrary direction in
Geometry	- (insertion point)
Available analysis results	**Reaction forces and moments**

Load can be assigned	**Point support motion load**
Line support property	
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in
	Arbitrary direction in
Geometry	Straight or curved reference line
Available analysis results	**Reaction forces and moments**
Load can be assigned	**Line support motion load**
Line support group property	
Modules where available	
Motion component	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in
Rotation component	Arbitrary direction in Global XY plane in
	Not available in
	Arbitrary direction in
Geometry	Straight or curved reference line
Available analysis results	**Reaction forces and moments**
Load can be assigned	**Line support motion load**
Surface support group property	

Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in
Rotation component direction	Not available
Geometry	Arbitrary region shape; holes can be placed into it
Available analysis results	*Reaction forces*
Load can be assigned	**Surface support motion load**

Table: Properties by Support types

Single Support versus Support group

Single Support defines only one component in one step, whilst Support group places a group of more components in one step. Although single Supports can be combined and work as a group support, it is recommended to define multi-support conditions with Support group, because all support components can be set or modified later (with the *Properties* tool) inside one settings dialog.

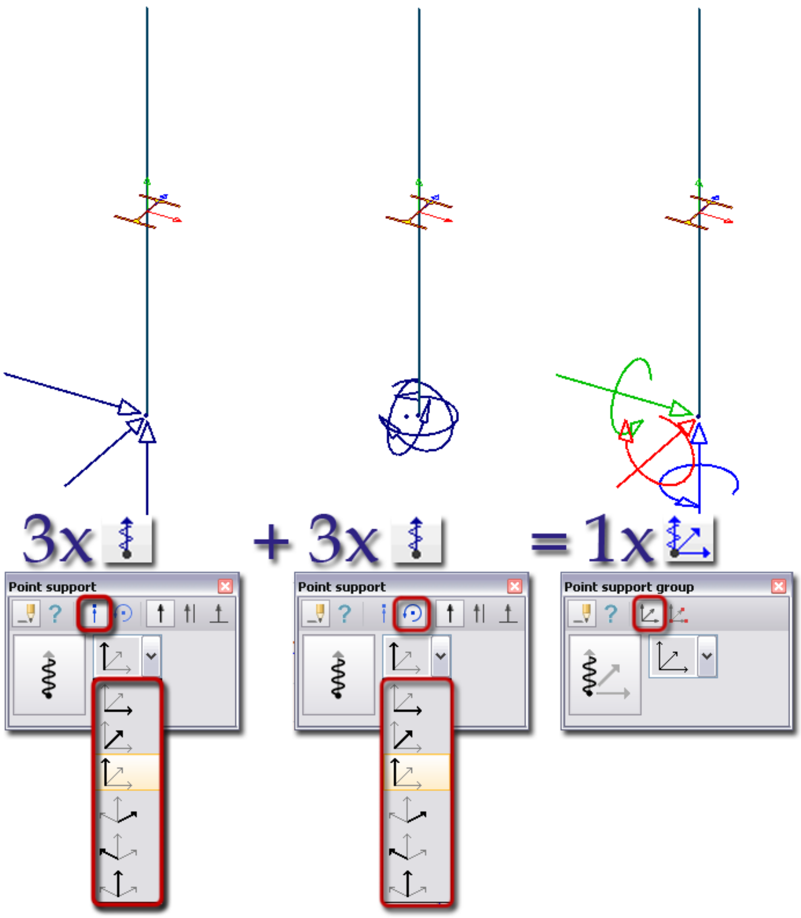


Figure: A fully rigid support modeled by six single Support components or one Support group

Special color-system differentiates *single Supports* from the *Support group* components. Whilst a single *Support* uses the color of the "Supports" **Object layer**, the components of a *Support group* are displayed in the colors equivalent with the support's *Local system* (*Settings > All > Display > Support* and *Local systems*).

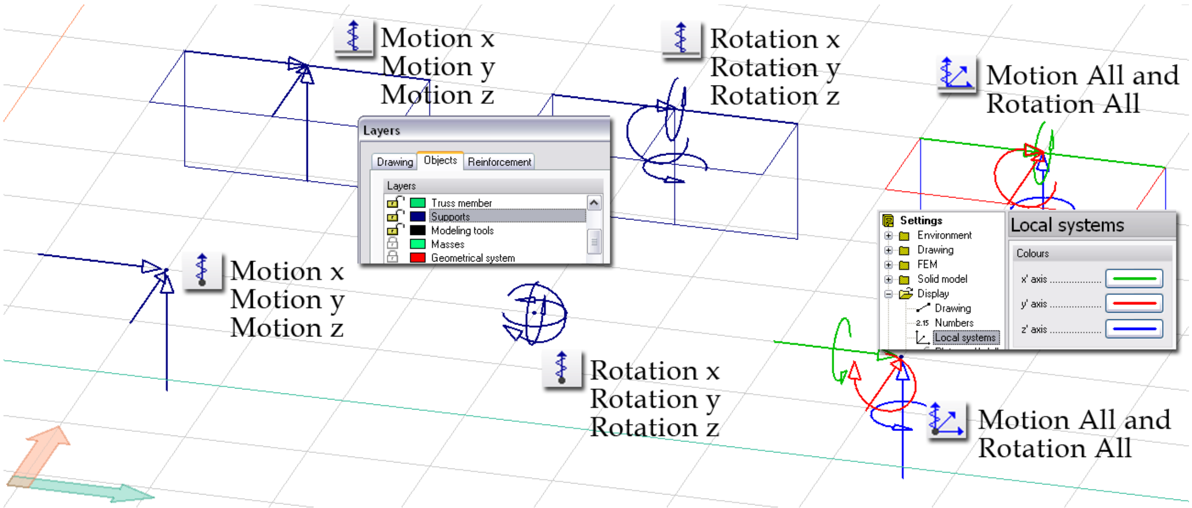


Figure: Color settings of support types

Definition steps

⚠ While support definition, keep in mind not to create unstable situation! For example in 3D modules, model a hinged support under a wall with hinged line support added to a wall at its rigid edge or with rigid line support added to a wall at its hinged wall. In case of unstable situation the program sends a warning message during calculation.

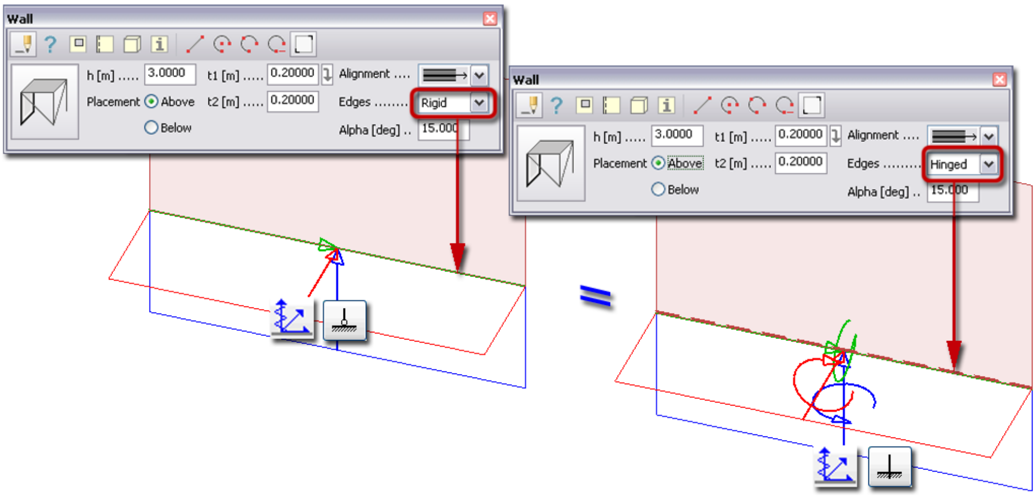


Figure: Modeling same wall support condition in two ways

1. If needed (and available in the current FEM-Design module), set a proper position for the **working plane**.
2. Start the required support command from Structure tabmenu and choose *Define*.
3. In case of single *Support*, choose the displacement type you would like to be define:

-  *Motion*
-  *Rotation*

4. Set the support **properties** as mentioned before in the applied support's *Default settings* dialog. See the settings possibilities and options at **Properties (Non-Linear Behaviors)**.
   In the settings dialog of a curved line support, the support direction has to set fixed or variable along the support's reference line.

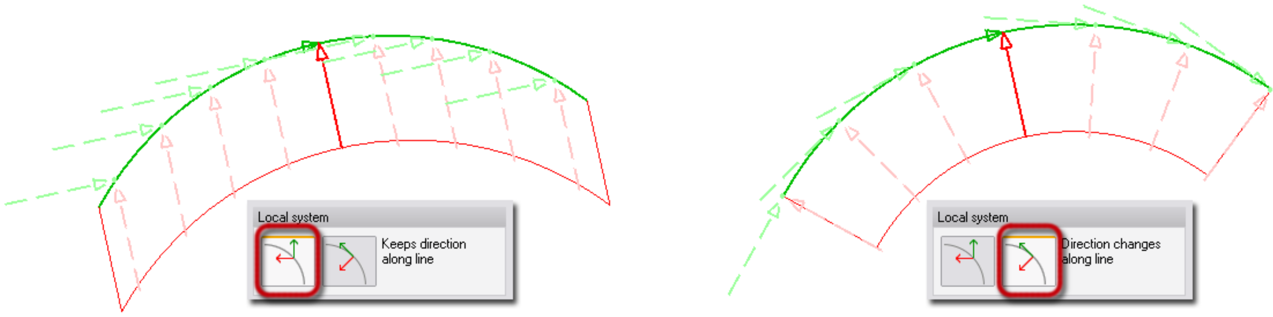
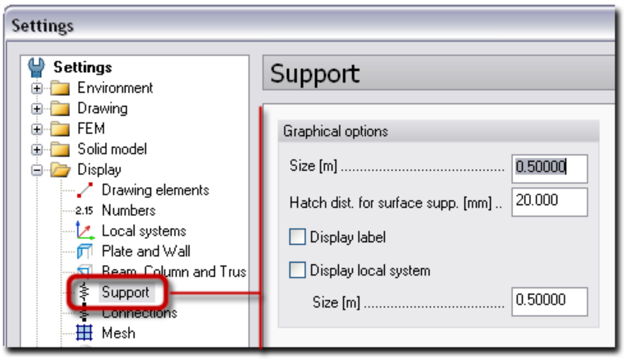


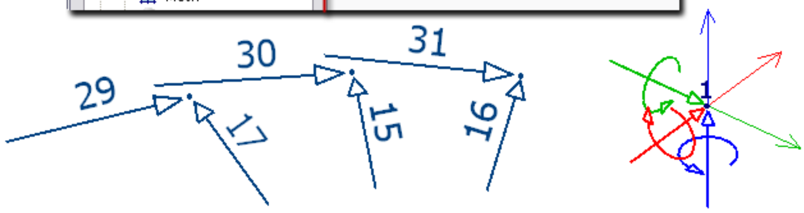
Figure: Possible support directions in case of curved reference line

5. Define the required support component **direction** for the single *Support* or set the reference system for the *Support group*.
6. Choose a **geometry** definition method for the support's reference point/line/region.
7. Define the support in the model view based on the chosen geometry method.

Optional steps:

8. Modify the geometry with the *Edit* menu commands valid for line/region elements.
9. Modify the support properties with the  *Properties* tool of the support's tool palette.
10. Set the display settings of supports at *Settings* > *All* > *Display* > *Support*.





 "*Label*" represents the supports' ID number generated by analysis/design calculation or by clicking *Refresh numbering* (*Tools* menu).

Rotation support is displayed with symbol strongly different from motion representation.

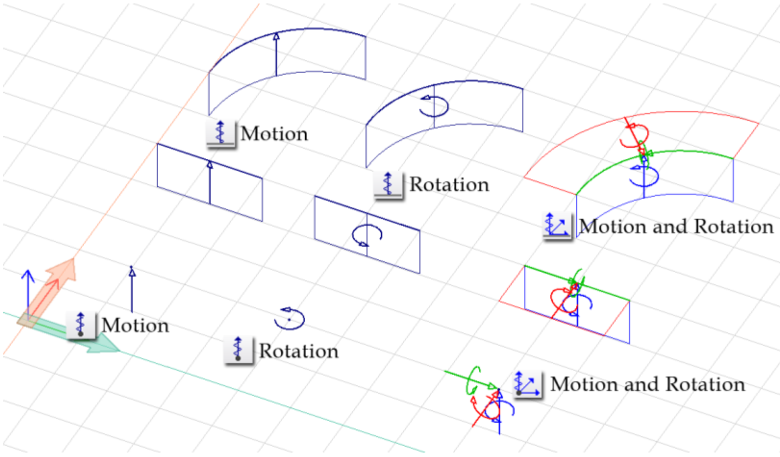


Figure: Typical symbols by support types

Support groups can also be separated from the *single Supports* by displaying their *Local system*.

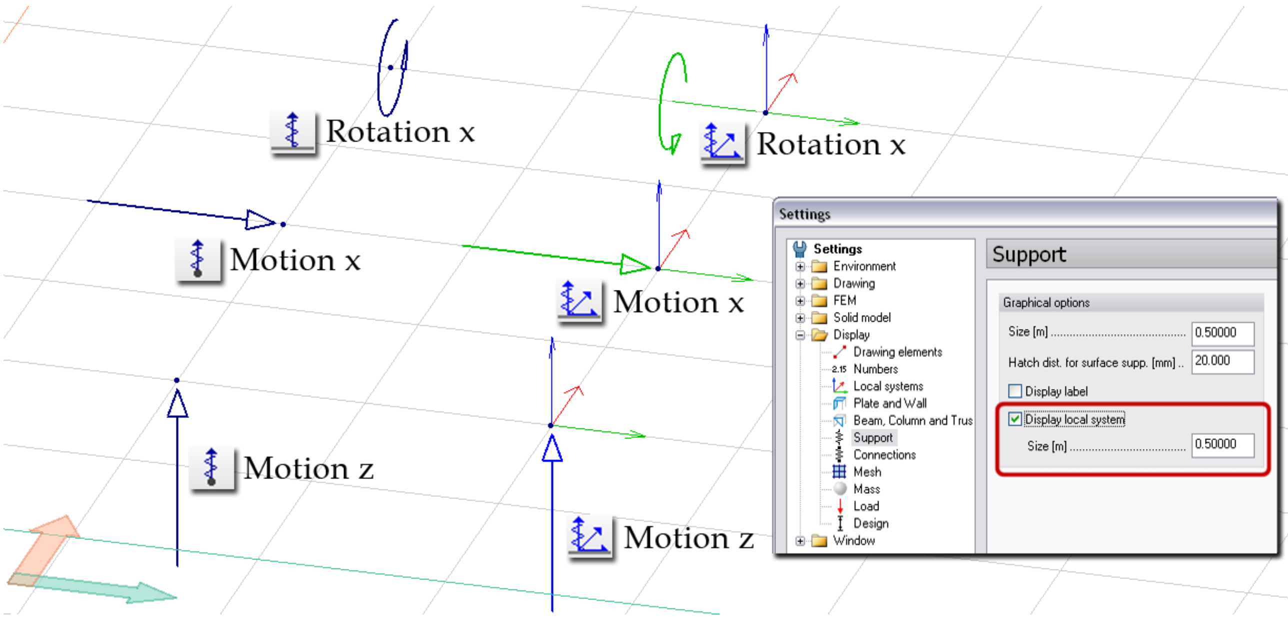


Figure: Local system separates a one-component Support group from a same directional single Support

11. The supports are stored on "Supports" **Object layers**. At layer settings, the default color of the single *Supports* and pen width of all support types can be set. The color and the symbol line type (e.g. dotted) of selected supports can be modified by *Edit* > **Properties** > *Change appearance*.

Connection Objects
Connection types

The following table summarizes the available "connection model objects" and their main features in the different FEM-Design Modules.

⚠ **Edge connection** is not a "connection object". It is a definition tool for boundary connection conditions, which is available for **Plate**/**Wall**, **Profiled panel** and **Timber Panel** elements. But, the **non-linear behavior and component settings** are valid for edge connections introduced for the supports and below mentioned connection elements. See the definition modes and possibilities of an **Edge connection at Plate**.

The end connection property of the bar elements are also not "connection objects". You can read more about them at the **Beam**, **Column** and **Fictitious bar connection** settings.

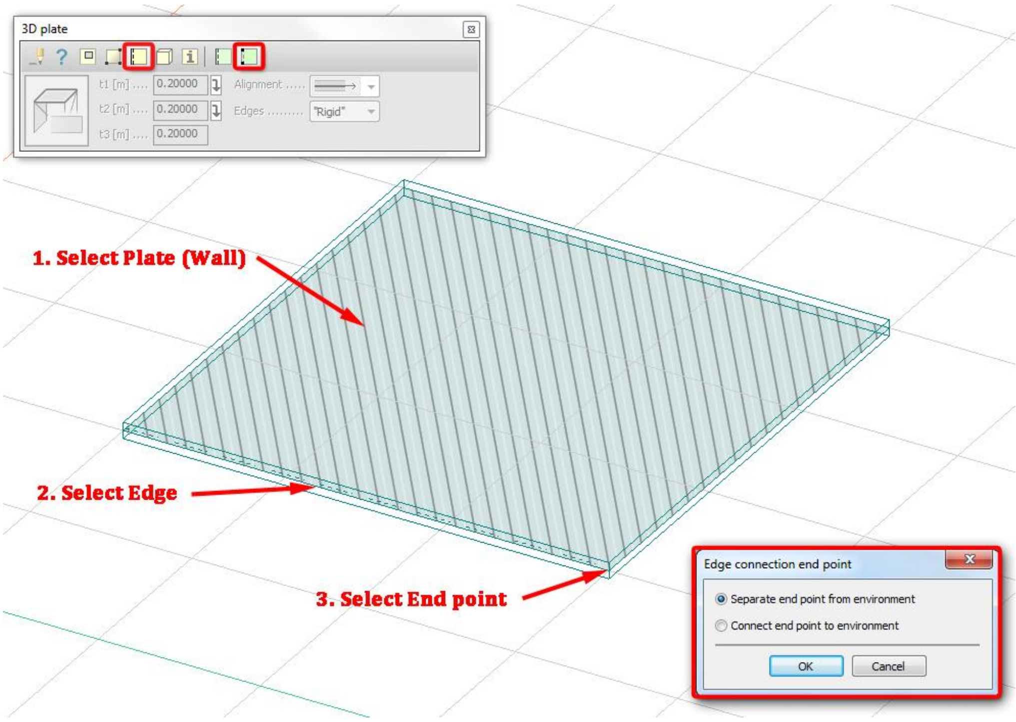
Point-point connection property	**Description**
Modules where available	
Motion component direction	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in
Rotation component direction	Arbitrary direction in Global XY plane in
	Not available in
	Arbitrary direction in
Geometry	- (insertion points)
Available analysis results	**Connection forces and moments**
Line-line connection property	
Modules where available	
Motion component	Parallel with Global Z axis in
	Arbitrary direction in Global XY plane in
	Arbitrary direction in
Rotation component	Arbitrary direction in Global XY plane in
	Not available in
	Arbitrary direction in
Geometry	Straight or curved reference lines
Available analysis results	**Connection forces and moments**

Table: Properties by Connection types

For the end points of the edge connections two options are available:

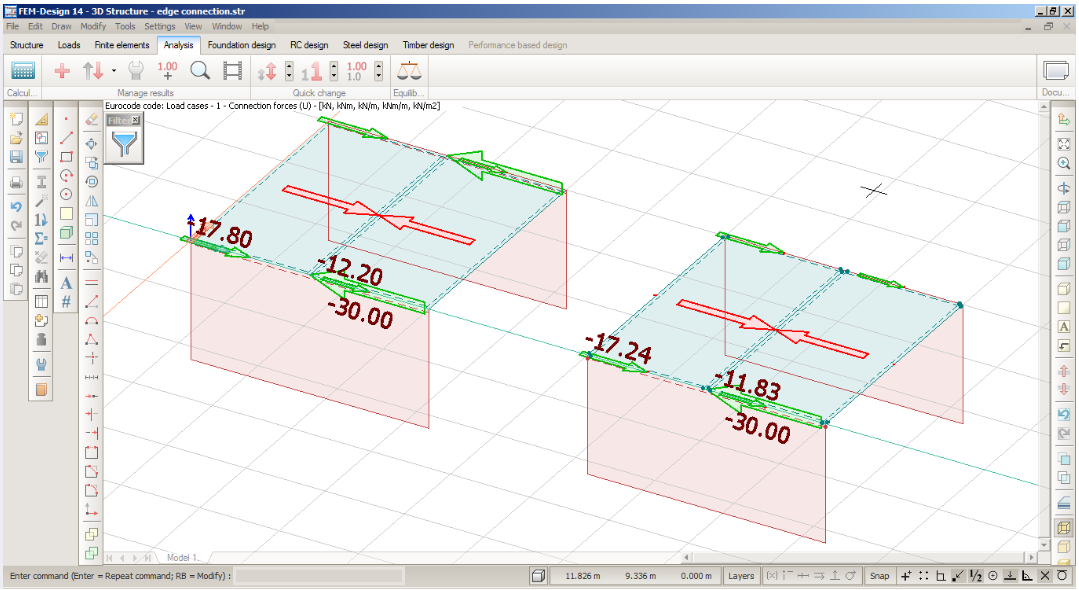
- Separate end points from environment,
- Connect end point to environment.

In all Shell Toolwindows this setting is available by choosing 'Edge connection' then clicking on 'End point behaviour' button. The end point can be chosen in three steps by selecting the Plate or Wall, then the edge and finally the end point.



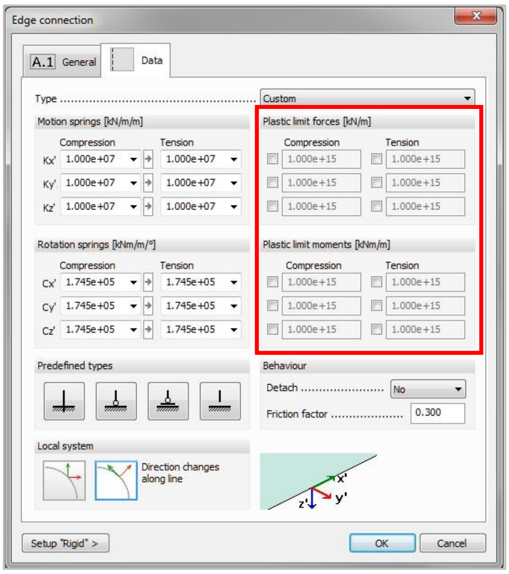
Using this function can solve problems like the one on the picture below:

On the right side structure all edge connection end points are connected to the environment which cases difference between the shear force on the wall and the sum of shear forces on the plate panels, which should be in balance according to common sense and this is exactly the case on the left side structure where edge connection end points are separated from the environment.



Plastic calculation

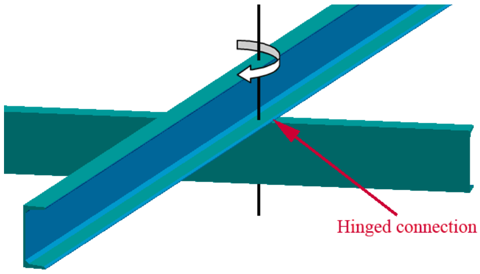
The plastic data can be set in the *Default settings/Data tab* of each above mentioned options. The figure shows where the feature is placed in the dialog.



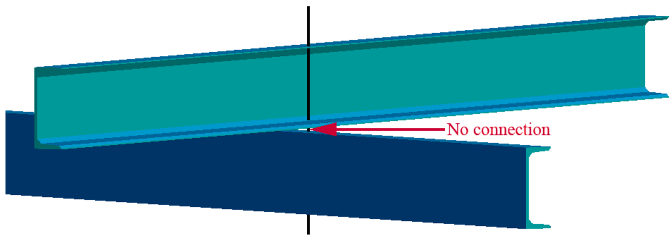
Point-point connection

With *Point-point connection*, different connections can be defined between crossed or connected bars. It is very useful in 3D modules. Some typical examples:

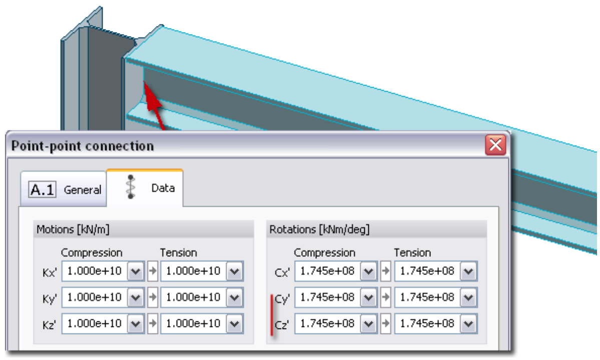
- **Bracing (called St Andrew's cross)**
  A hinged connection between two channels intersecting each other: free rotation around the axis perpendicular to the plane defined by the two channels. (It is not necessary to define the eccentricity; the next figure is schematic.)



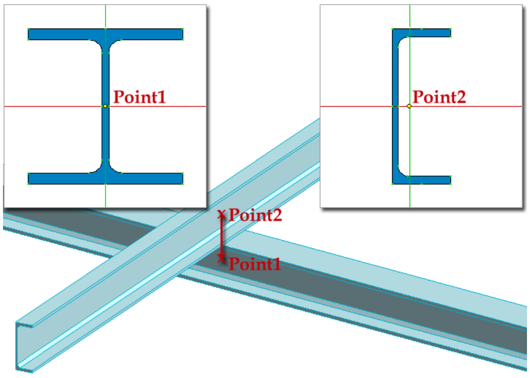
- **Independent bars**
  Defining two bars crossed each other with their structural lines but having no connection in real state is a typical problem of modeling. Now, it can be solved easily by creating point-point connection in the bars' intersection with all components set to “free”.



- **Semi-rigid connection between members**
  At the **Connections** settings of members (*Bar* > *Default settings*) only fully rigid or different type of hinged end connections can be defined. But, *Point-point connection* command gives the possibility to model any other connection types like semi-rigid connection with given stiffness values for rotation. The stiffness values can be set according to the applied standard instructions.



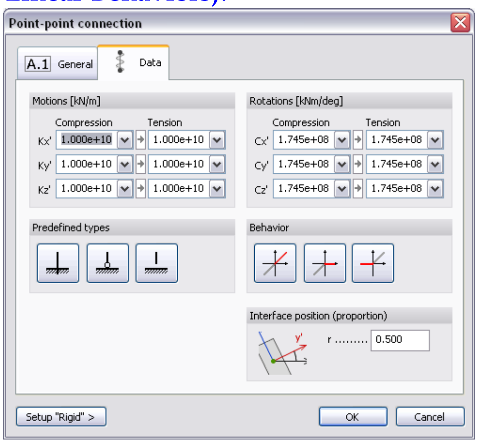
- **“Real state” modeling**
  Real eccentric position (e.g. bars connected with their web plates) without eccentricity settings can be also modeled with *Point-point connection*.



Definition steps

1. Start *Point-point connection* command from Structure tabmenu and choose *Define*.

2. Set the properties of the new connection at *Default settings*. See the settings possibilities and options at **Properties (Non-Linear Behaviors)**.



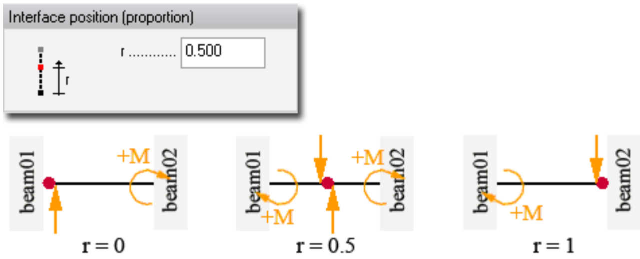
Applying a connection different from rigid state, the *Interface position* (*r*) is very important.

$(0 \leq r \leq 1\ .)$

r = distance between connection position and start point of definition /

distance between start and end points of connection

The next figure introduces the meaning of the interface position (*r*) (it has no effect if the connection points are in the same position!):

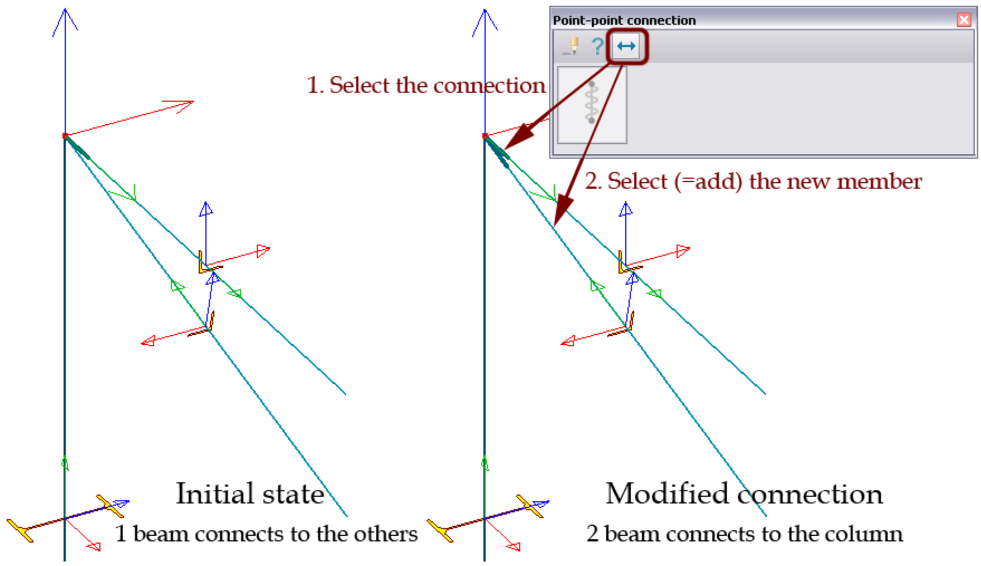


3. Set the system for the ***direction*** of the connection components.
4. Select the first object you would like to connect.
5. Define the point of the first object you would like to start the connection.
6. Define the point of the second object you would like to end the connection. If the connection points between the two objects (crossed each other) are in the same position, just click to define the connection end point.

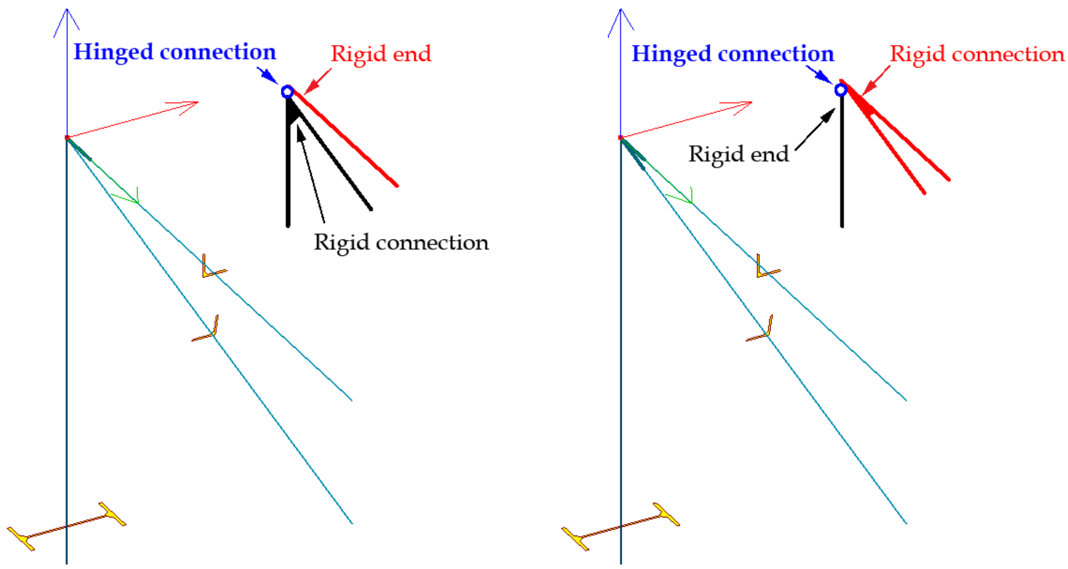
Optional steps:

7. Modify the connection properties with the *Properties* tool of the tool palette.
8. Add members to a selected connection or remove members from it with *Edit connected objects*.

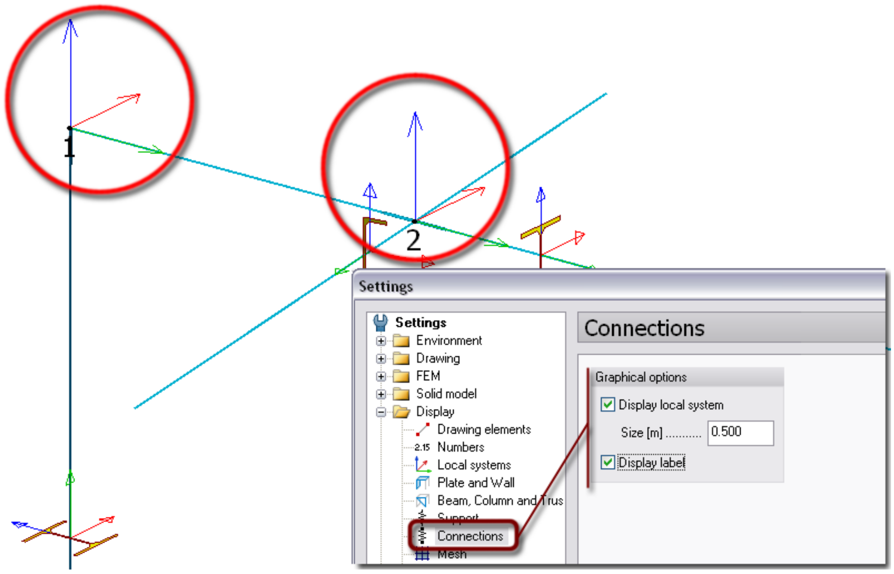
The next figure shows the steps of adding a new member (beam) to a predefined point-point connection. First, select the connection you would like to modify and then choose the new object you would like to add to the selected connection. To remove a member of a connection, do the same steps with .



The next figure shows you the difference between the static-systems of the initial and modified states at all rigid bar end connections and defined hinged point-point connection.



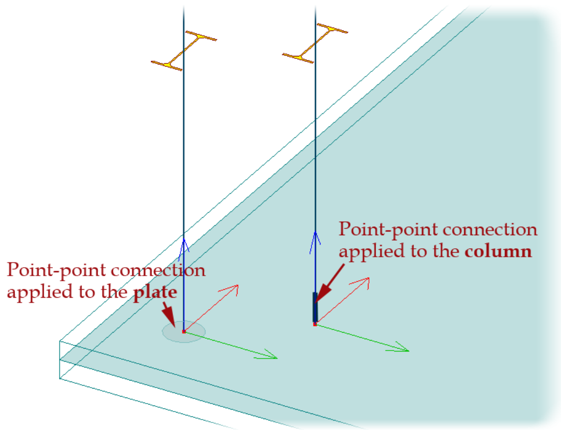
9. Set the display settings of connections at *Settings* > *All* > *Display* > *Connections*.



⚠ "*Label*" represents the connections' ID number generated by analysis/design calculation or by clicking *Refresh numbering* (*Tools* menu).

10. The connections are stored on "Modeling tools" **Object layers**. At layer settings, the default color defines the connection label color.
Each connection element has its own local system. The connection symbol (darker colors, thicker lines etc.) shows the objects having one of the associated above mentioned connection types.

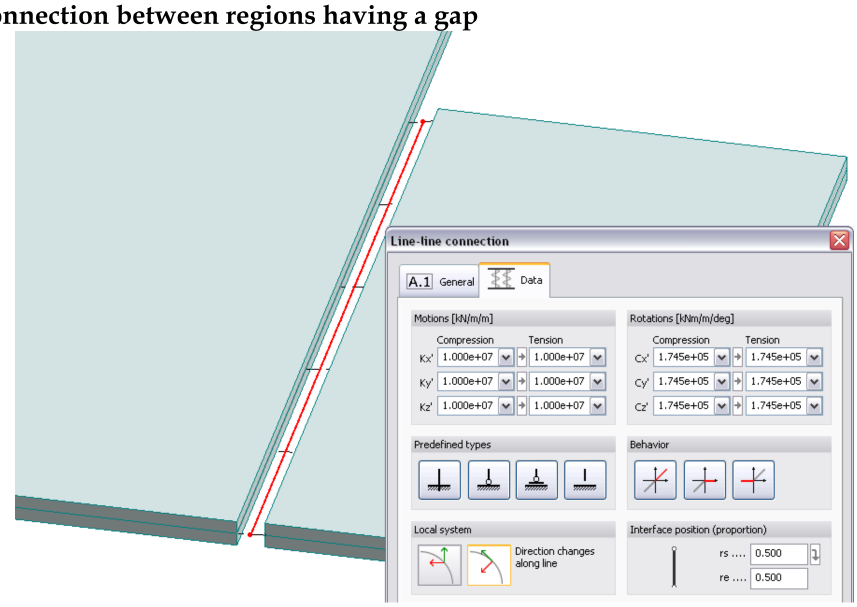
The following figure shows an example how the program makes different symbols depending on what object was selected during the selection of the connected points. But note, that although the definition ways are differ at the column-plate connections, if the properties (directions, components and stiffness values) are the same at both point-point connections (here in the example, hinged connection = all rotation components are free), the static-systems are also the same.



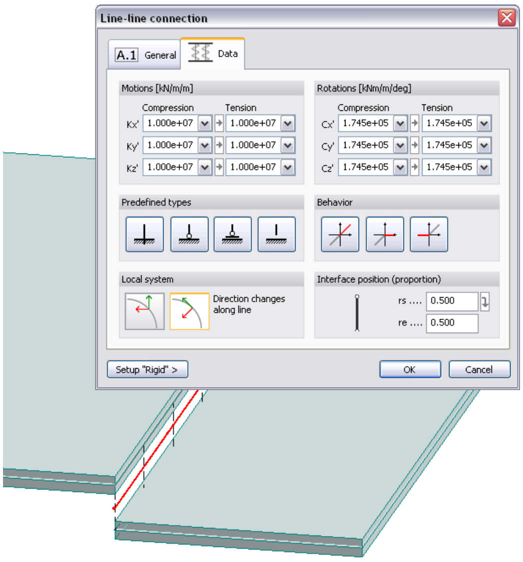
Line-line connection

With the *Line-line connection* command, different connections can be defined between two regions having different positions in 3D (so not having common parts in the geometry model). Some typical examples:

- **Hinged connection between regions having a gap**

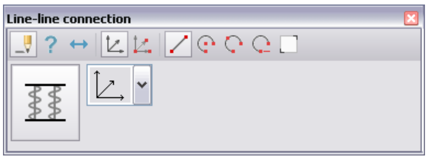


- **Slabs on different level**
  Instead of connecting two slabs with a *Wall* element, the connection can be modeled with the *Line-line connection* tool (if the internal stress and other results are not required in the wall).

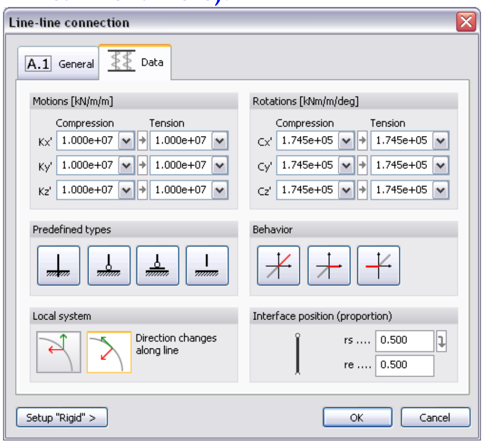


Definition steps

1. Start *Line-line connection* command from Structure tabmenu and choose *Define*.



2. Set the properties of the new connection at *Default settings*. See the settings possibilities and options at **Properties (Non-Linear Behaviors)**.



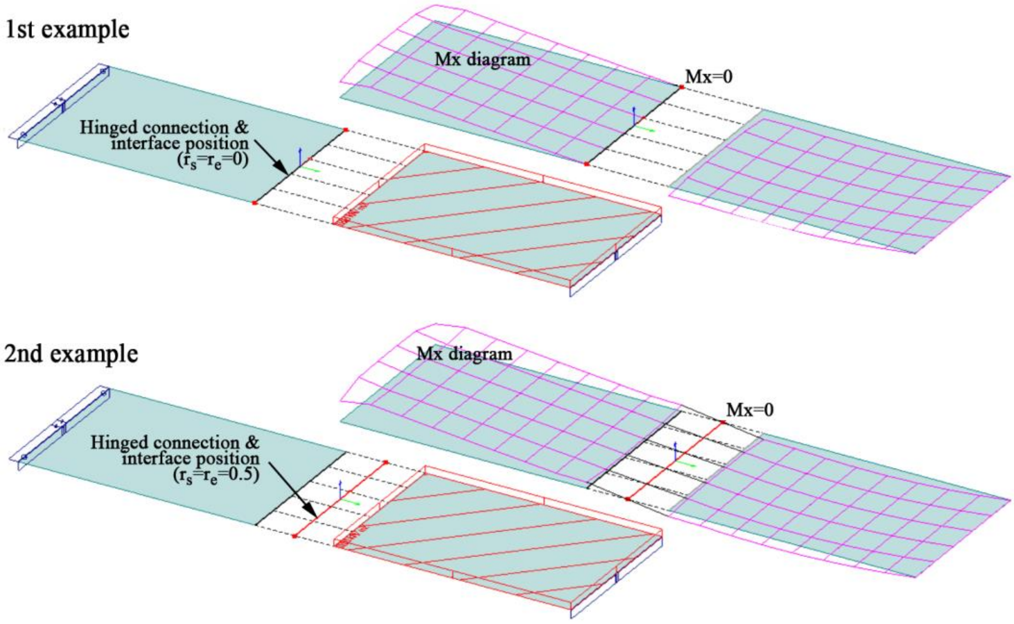
Applying a connection different from rigid state, the *Interface position* (r) is very important.

($0 \le r \le 1$.).

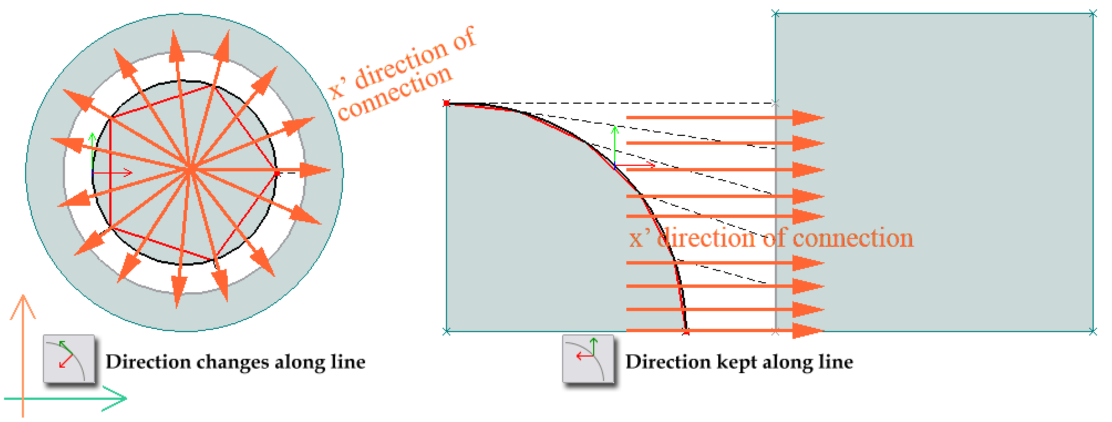
r = distance between connection position and start point of definition /

distance between start and end points of connection

The figure shows two different interface positions at hinged line-line connection between two plate regions: in the first example, the hinged line (the place of the interface) is on the edge of the left plate (r_s=r_e=0), and in the second example the interface position is in the middle (r_s=r_e=0.5). In both examples, the left plate is supported rigidly and the right one has simple support against to Z-directional motion. The M_x moment diagrams show zero moments where the interface positions are placed.



In the settings dialog of a curved line support, the support direction has to set fixed or variable along the support's reference line.



3. Set the system for the **direction** of the connection components.
4. Choose a **geometry** definition method for the first connection object's reference line (axis).
5. Select the first object you would like to connect.
6. Define the first connection line object in the model view based on the chosen geometry method.
7. Choose a **geometry** definition method for the second connection object's reference line (axis).
8. Define the second connection line object in the model view based on the chosen geometry method.

Optional steps:

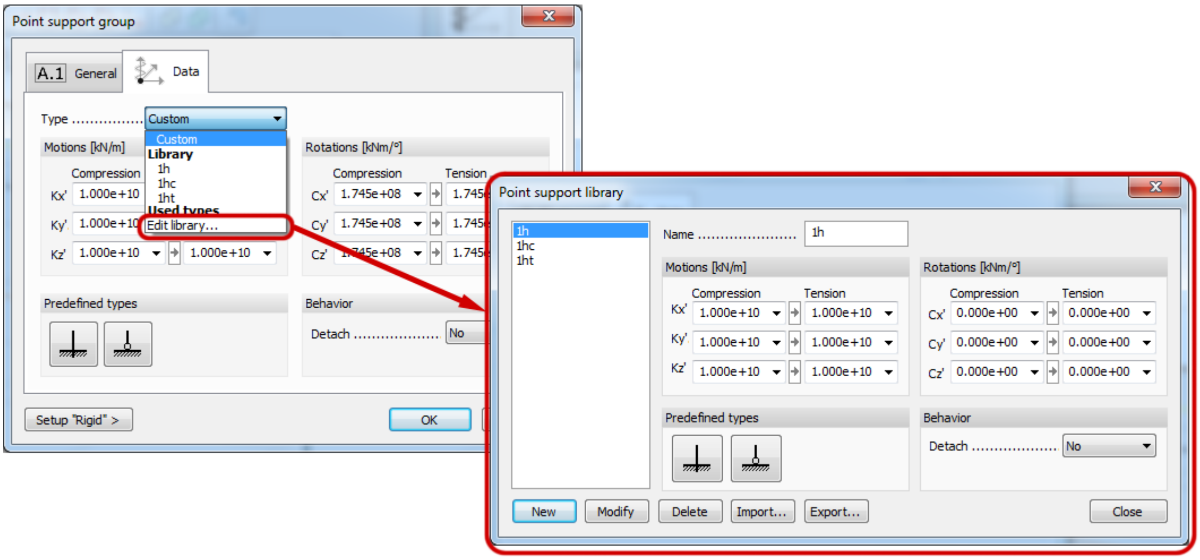
9. Modify the connection properties with the *Properties* tool of the tool palette.

10. Add members to a selected connection or remove members from it with **Edit connected objects**.
11. Set the display settings of connections at *Settings* > *All* > *Display* > *Connections*. See the possibilities at **Point-point connection**.
12. The connections are stored on “Modeling tools” **Object layers**. At layer settings, the default color defines the connection label color.

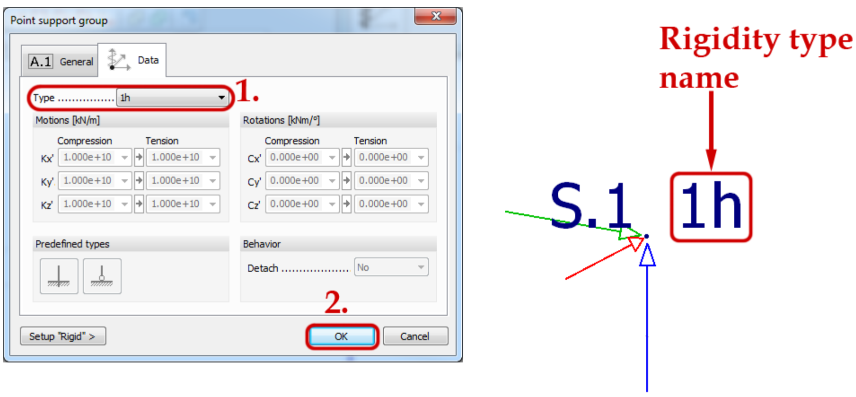
Rigidity type

This option is used by **Point support group, Line support group, Surface support group**, **Edge connection** in **3D Plate** and **Wall** structural objects, Point-point connection, Line-line connection, border and *panel* connection of **Timber** and **Profiled panels**. With rigidity type the user can define and load specific predefined spring constants.

If the rigidity type is *Custom* the user can define the spring constants as in previous versions. To define a type the user has to choose the *Edit library…* option.



After defining rigidity types, the defined spring constants will be registered in the Library and it can be loaded in the future.



There are some objects that use the same rigidity type library:

- *Line – line connection, Edge connection (3D Plate, Wall), border* and *panel* connection of *Timber panels* and *Profiled panels*

Cover

Cover

Cover property	Description
Modules where available	
Geometry	Any shape
Load direction	Arbitrary in
Load type	Surface load
Available analysis results	-

Table: Cover properties

Definition steps

1. If needed, set a proper position for the **working plane**.
2. Start *Cover* command from Structure tabmenu and choose *Define*.

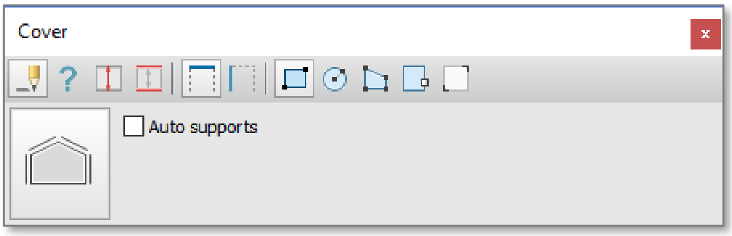


Figure: Cover toolwindow

3. Set the cover type if it is slab/roof type or wall type

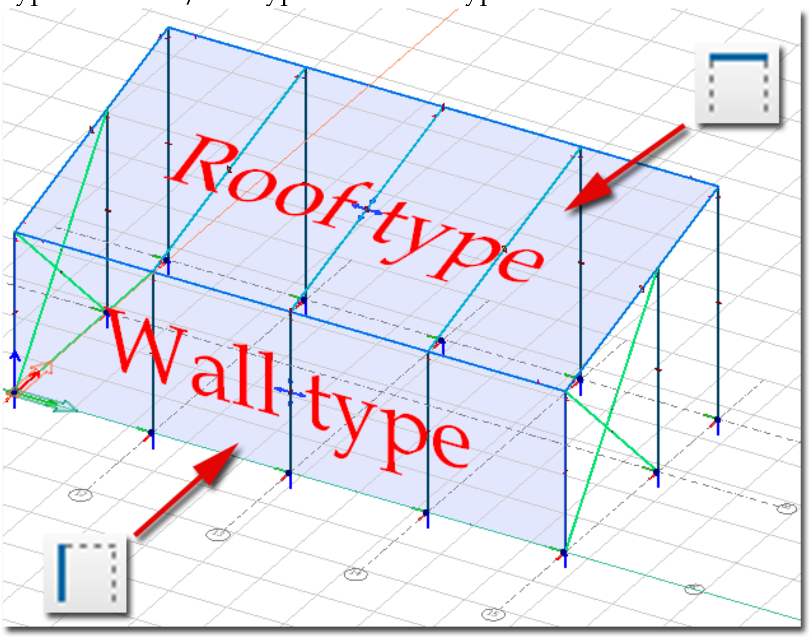
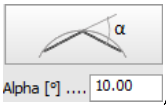


Figure: Cover types

4. Choose a **geometry** definition method for the cover's plane.

Note: In case of arc geometry in wall covers, you must set the maximum angle between neighboring covers at 

, since arced walls are replaced with straight sections.

5. Set the load bearing directions with .

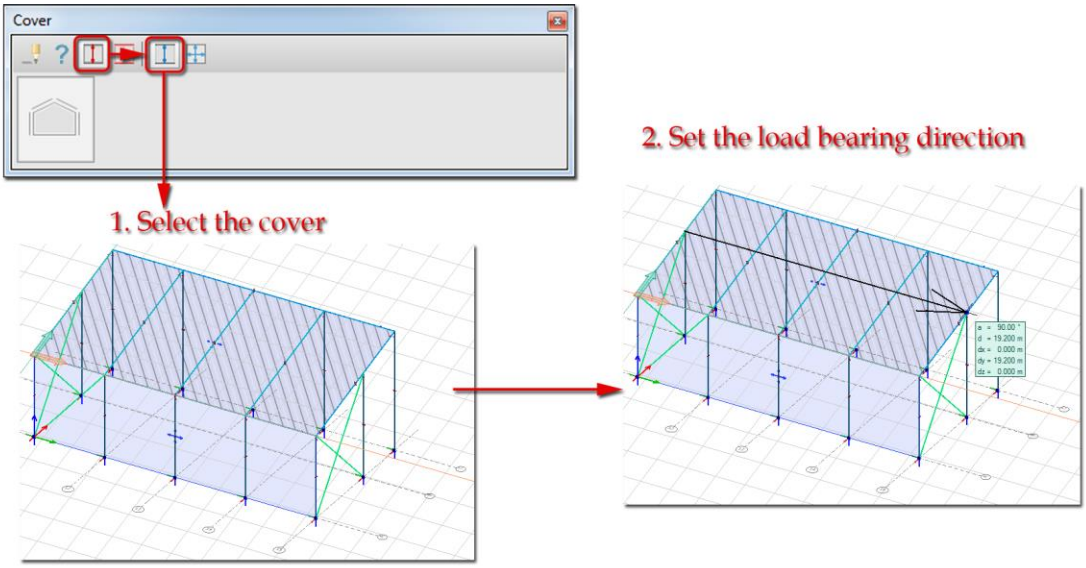


Figure: Setting the load direction (monodirection)

6. Defining the supporting structures with .

   After defining the supporting structures, the equidistant lines will appear with the load distribution zones.

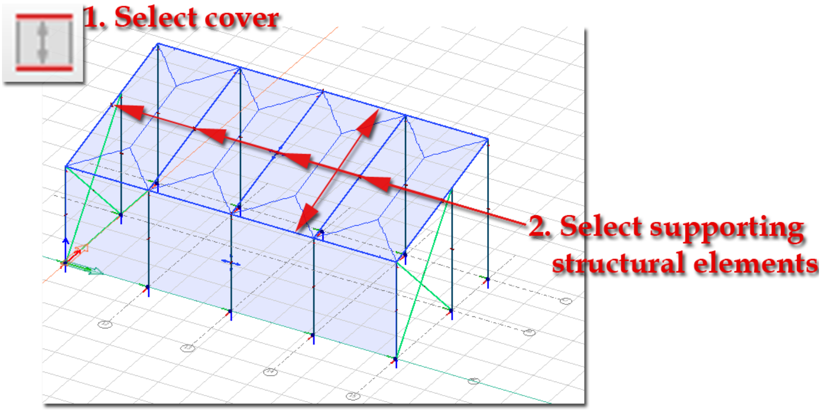


Figure: Defining cover's supporting structures

Activating the *Auto supports* checkbox will automatically determine the supporting structures for the cover. These can be modified later.

When using *Auto supports*, the following objects or elements will be recognized as supports if they are located in the cover's plane:

- Beams and columns (including fictitious ones)
- Line supports and line support groups

- All inner and outer edges of slabs and walls (also working with fictitious ones)
- all point elements (such as point supports)

The Supporting structures and the Load bearing direction commands can be used to alter the automatically determined specifications.

Building cover

With this command frequently required shapes such as flat roof, ridge roof, or inclined roof can be very quickly defined.

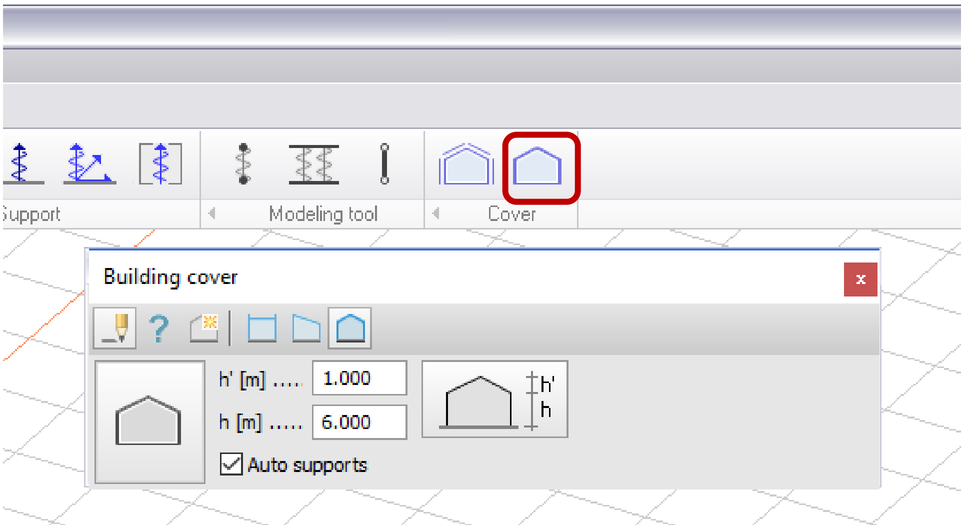
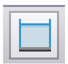
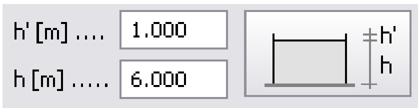
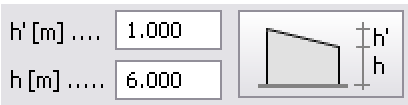


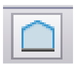
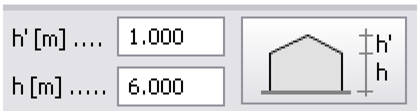
Figure: Starting Building cover command

Flat roof:  and its height parameters: 

No negative value is accepted for either h or h’.

Lean-to:  and its height parameters: 

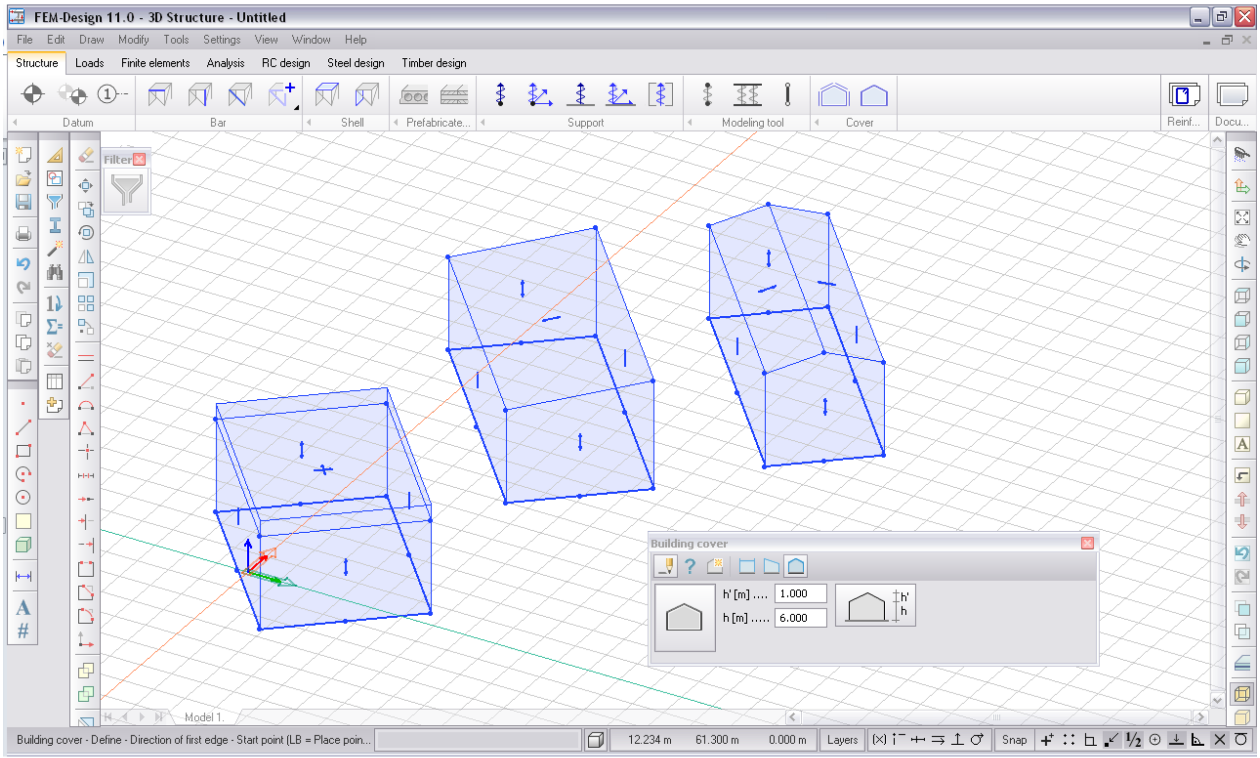
No negative value is accepted for either h or h’.

Ridge-roof:  and its height parameters: 

Negative values is accepted for h’.

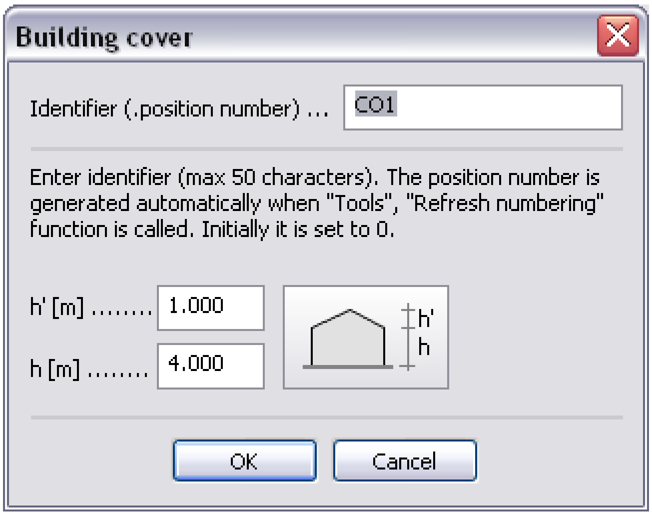
Keep in mind that in EC, the total height is signed by h which is h+h’ in the program’s notation. This is important in wind calculation according to EC which requires the reference height.

After setting the height parameters you can create the layout rectangle. In the picture below you can see the three types with their assumed stiff direction.



All building covers are monodirectional by default except the upper surface of flat-roof. (They can be changed one by one after definition.)

By clicking on  (Properties) you can change the ID and the height parameters of the building cover:



You can explode the building cover with , and then edit the covers one-by one.

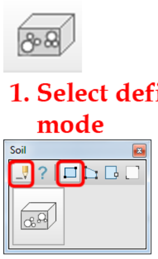
The *Auto supports* function is available for building covers as well.

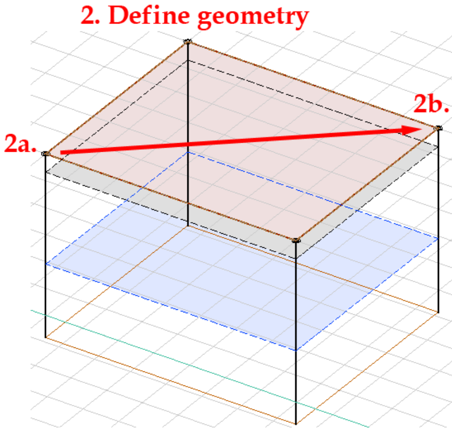
3D Members

This chapter summarizes all features and properties of a 3D member such as Soil !!! (Link).

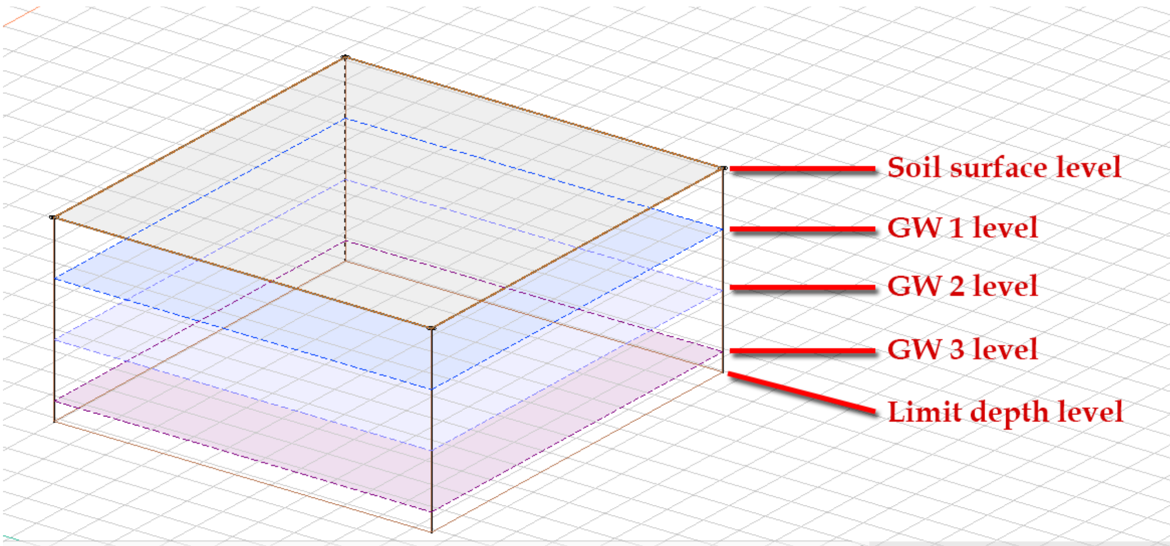
Soil

 Soil property	Description
Modules where available	
Geometry	Regular shape, Solid
End connection	-
Material	Soil
Load direction	Arbitrary in 
Load type	*Point load*, *Line load*, *Surface load*. Only forces!

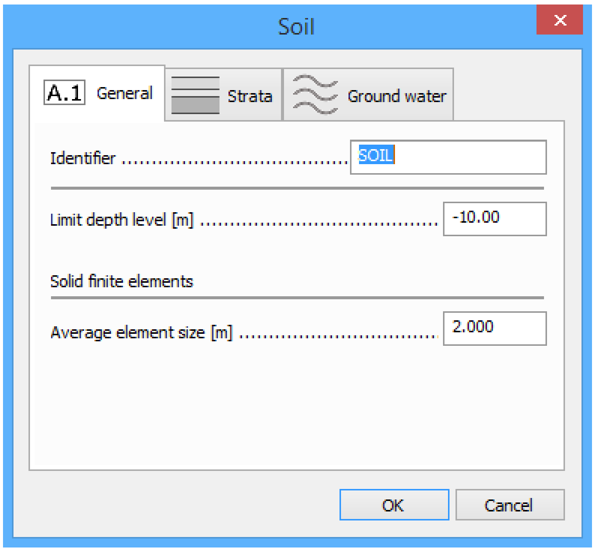




Soil's *ground level*, *water level*, and strata top level can be different in each **Boreholes** belonging to the soil. Boreholes are created in the corners of the soil region by default, but User can create more if necessary. Soil's limit depth is constant.



The soil details can be set in *Default Settings*:

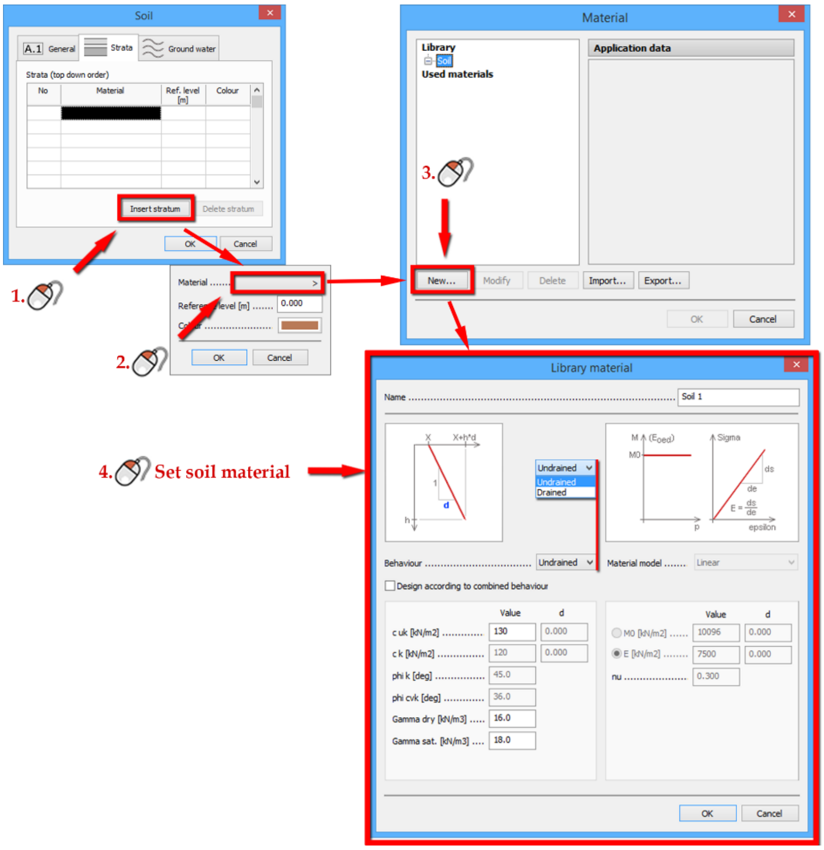


- *Identifier* (General)
  The program automatically generates it, but User can define custom value. Identifier (ID and Position) number can be displayed in model view (**Display settings**).

- *Limit deph level* (General)
  The level of the bottom boundary plane of the soil

- *Average element size* (General)
  If **Calculate soil as a solid element** is activated, Fem Design tryes to define the solid elements of the soil according to the given value

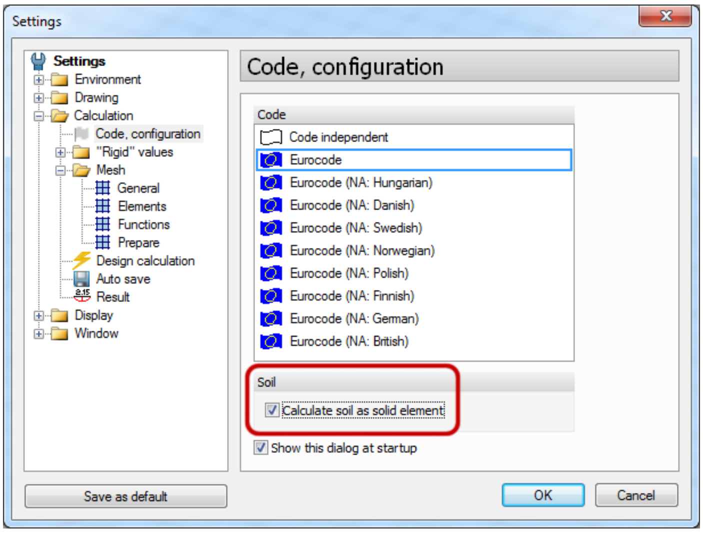
The soil material details can be set individually:



- *Name*
  The name of the soil material

- *Behaviour*
  Wether the analysis is processed with drained or undrained soil conditions. If *"Design according to combined behaviour"* is active, both conditions are considered

User can choose if he/she wishes to calculate soil as solid elements in analysis or just wants to consider it in foundation design calculation. In the Settings/Calculation/Code, Configuration dialog this option can be set.



⚠ The soil can be loaded only with *Point load*, *Line load*, *Surface load* (not with moments).

⚠ Concentrated mass cannot be defined in the soil.

Section Editor

FEM-Design *Section Editor* gives the possibility to define, edit and store profiles for the bar elements (beam, column and truss element).

The functions of the *Section Editor*:

- *drawing new sections*,
- *definition of new sections by parameters*,
- *definition of complex sections* by mixing drawing and standard profiles,
- editing cross-sections,
- *calculation of sectional data*, and
- *profile registration and organizing*.

Section editor can be started directly from design modules that work with bar elements from the *Tools* menu.

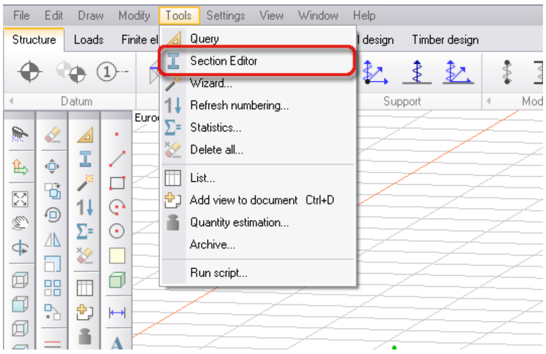


Figure: Section Editor as a built-in tool of design modules

Section Editor can be started as an independent FEM-Design application by clicking the *Section Editor* icon in the "*FEM-Design tools*" folder of

- *Start > All Programs > FEM-Design…*,
- *Windows Taskbar > FEM-Design System Tray* (if installed),
- short-cut of *Desktop* (if installed).

Section Editor module has only one *tabmenu* called Cross-section that contains all the section definition tools, but the FEM-Design drawing, editing, displaying etc. commands are also available on the *Section Editor* user-interface.

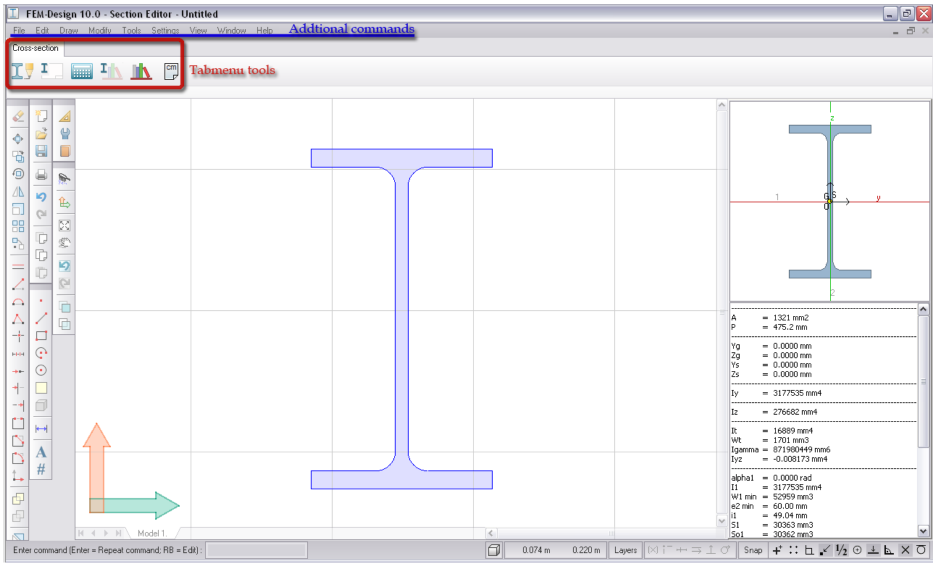


Figure: Section Editor user-interface

Sections are two-dimensional region elements defined in the Y-Z plane of the *Global* (and *UCS*) coordinate-system. Section Editor uses the Y-Z system for defining and calculating profiles, because the sections are perpendicular to the x' axis (parallel with the y'-z' plane) of the *local coordinate system* of bar elements.

Cross-Section Definition

New cross-section can be defined by *drawing section* regions with arbitrary geometry, or by giving parameter values of a *parametric section* available in the built-in *Section library*, or by combining the two methods to build *complex sections*.

Drawing a Section

To draw a section use the *Draw section* tool and apply *Define* for the required region definition (geometry) mode *Rectangular*, *Circular* or *Polygonal*.

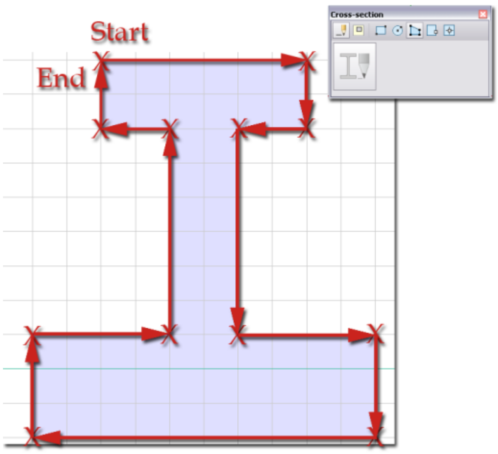


Figure: Section defined as polygon

With *Pick lines* tool, section can be generated by one-clicking drawing lines and arcs that define closed region. Drawing elements can be defined previously with the *Draw* menu commands or they can be imported from *DWG/DXF files*. With *Pick existing region* tool, section can be generated by one-clicking drawing region defined by the *Region* command of the *Draw* menu.

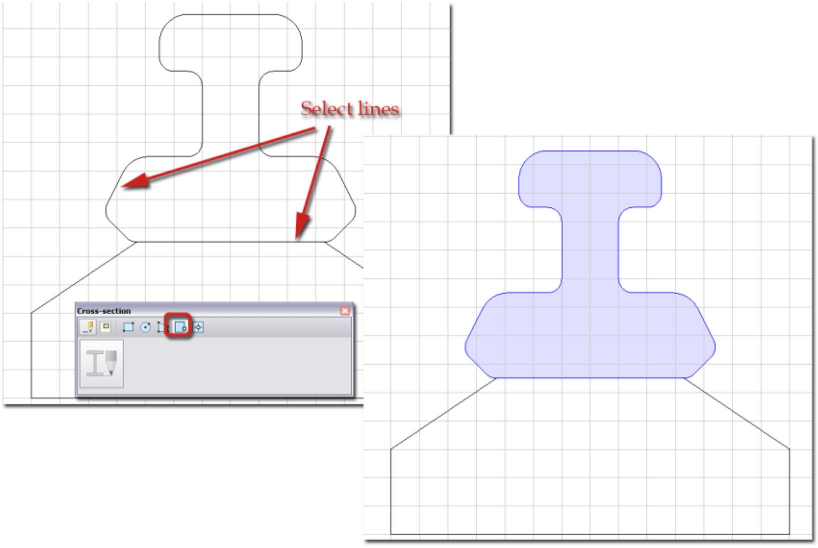


Figure: Section created by selecting closed contour

With the *Hole* tool, openings and cuttings can be inserted to section regions with any shape defined by the previously listed region definition modes.

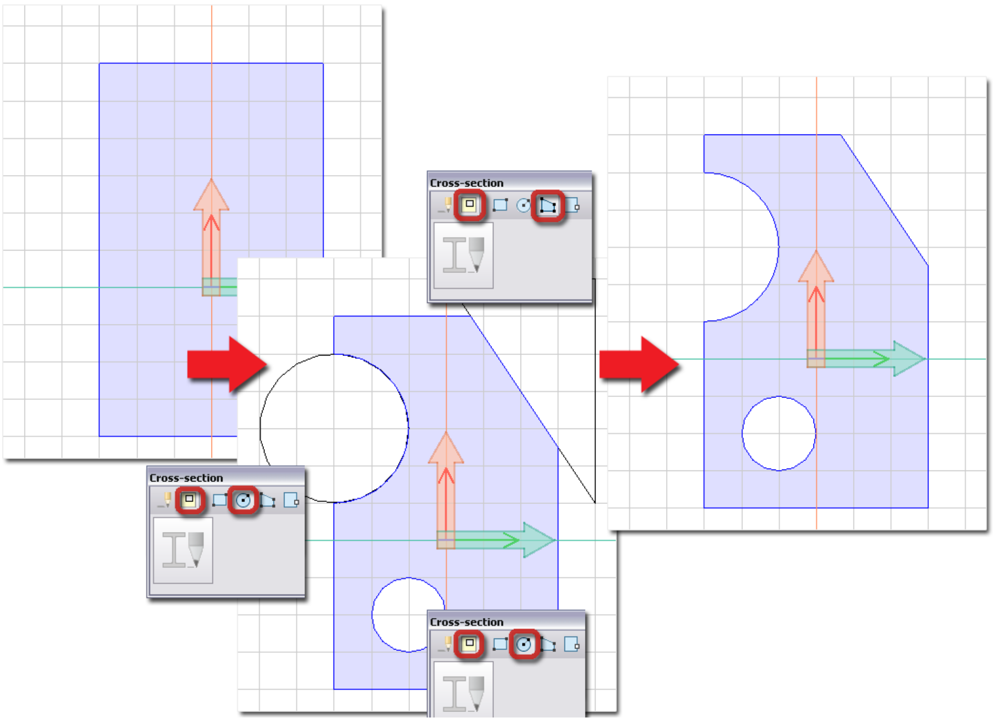


Figure: Section edited by Hole

You can read about the definition steps and functions of different region and hole geometries at the similar tools ("blue color-linked") of *Structure definition*.

Section region shapes, corner points and side lines can be modified with the *Modify* menu tools.

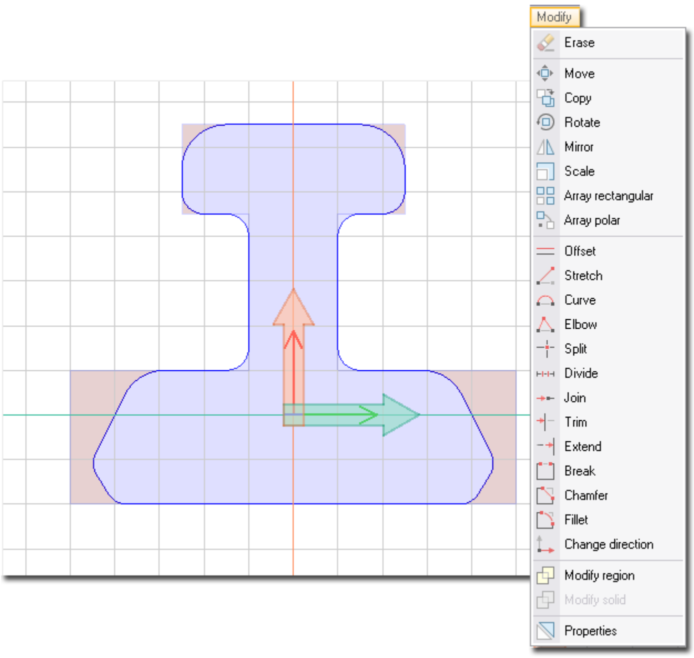


Figure: Section modified with Edit tools

Parametric Section

Although, quick definition of parametric profiles available in the settings dialogs (*Cross-sections*) of *Column*, *Beam* and *Truss member*, parametric sections can be inserted for editing and using them to create *complex sections*.

Use  *Insert section into drawing* command to browse from parametric sections as well as standard and user-defined profiles, and to insert one or more of the required section with chosen geometry and sizes into the drawing area.

⚠ Although, you may choose for example a timber profile for editing and register as another fabrication type (for example steel, concrete etc.), we suggest you to choose profile type that will suit for later design. But, always you decide the available design type of the section defined by *Section Editor* when *registering* it with its *Fabrication technology* into *Section library*.

Definition steps

1. Choose a parametric section shape in *Section library* or select a predefined (standard or registered user-defined) profile from *New >/Size…* option.

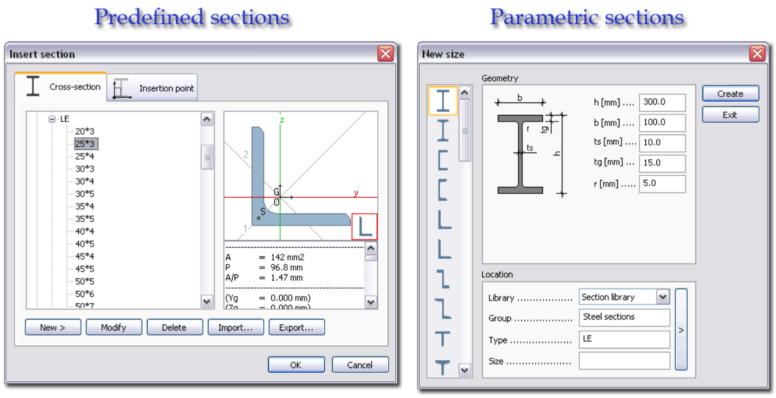


Figure: Dialog for defining parametric sections or for choosing predefined sections

2. In case of parametric profiles, set the required geometric parameters. Each shape has a preview with the meaning of the parameters.
3. Set the insertion point at *Insertion point* tab, where the placement can be defined. Default setting for placement is the center of gravity, but it can be changed by clicking on the specific points (green points) or set by y' and z' coordinates.
4. Click *OK* to start inserting the defined or selected profile into the drawing area.

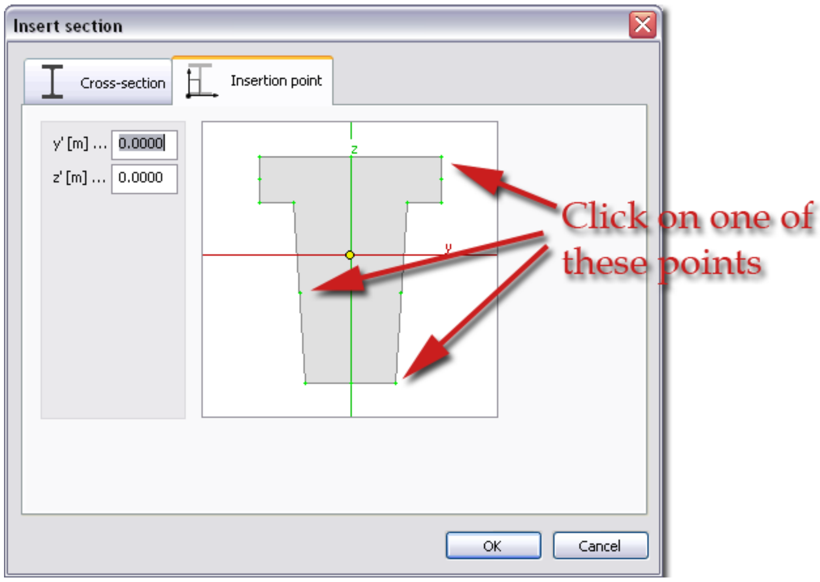


Figure: Definition of insertion point position

5. Place the section with its insertion point on the drawing area. Click  or  to insert the section in the origin of the global Y-Z system.
6. Further rotation can be set with the direction of the section's z axis. Accurate angle can be typed in the Command line or click  or  to use the direction set in the *Reference point…* dialog.

Complex Section

⚠ Only homogenic complex profiles can be defined in the *Section Editor*, because one material can be assigned to it later in the design modules when defining beams, columns and bars.

At complex sections, intersecting or overlapping among section regions are not allowed. If there is any problem, the program stops the *sectional data calculation* with an error message.

As read in the previous chapter, quick definition of complex profile can be done by the repetition of section insertion with different positions and directions.

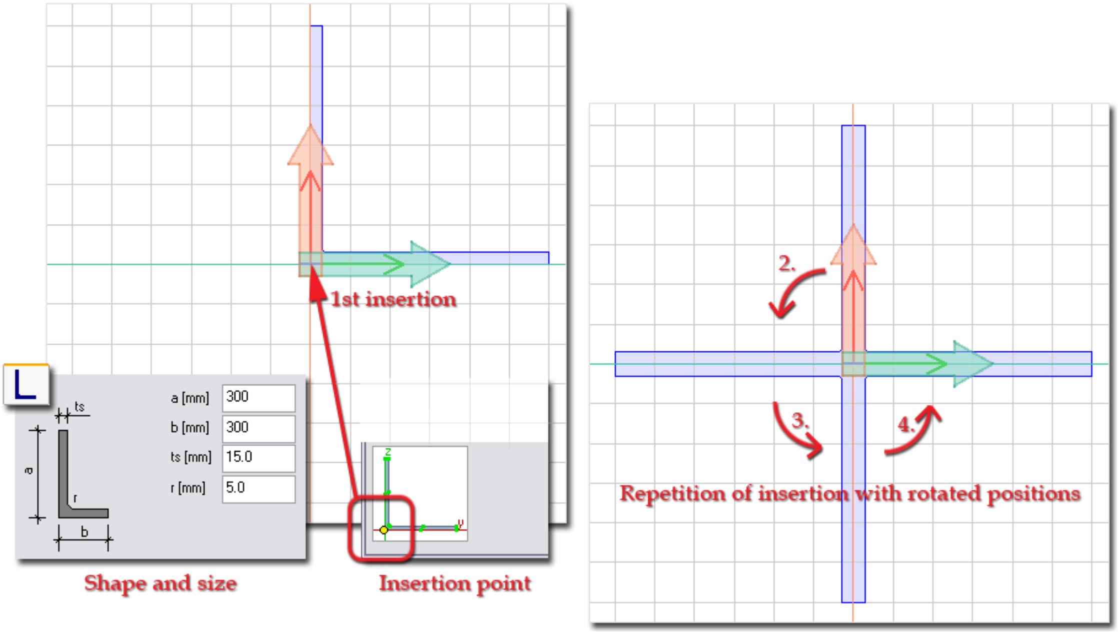


Figure: Complex profile built by parametric profiles

Of course, combining the section drawing and insertion modes various complex cross-sections can be defined.

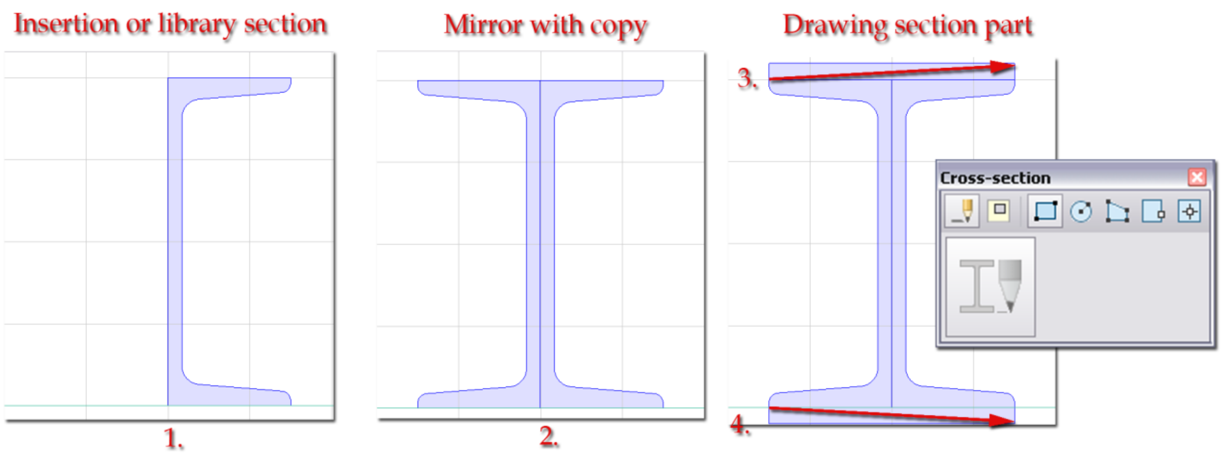


Figure: Mixture of definition modes

Cross-Sectional Data

Before calculating the data of the defined section, it is recommended to place the section's centre of gravity to the global system origin. With the *Query* command of *Tool* menu, you can easily find the center of gravity point of the defined section region and insert a drawing point.

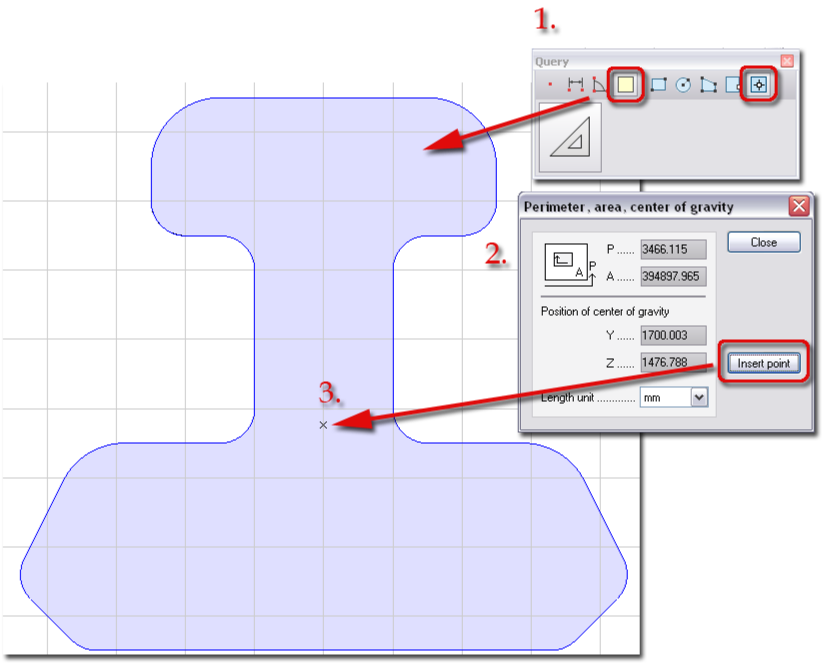


Figure: Section moved into Origin with its center of gravity

Query finds the center of gravity point of complex sections, if you merge the connected region parts with the *Union* tool of the *Modify region* command (*Modify* menu).

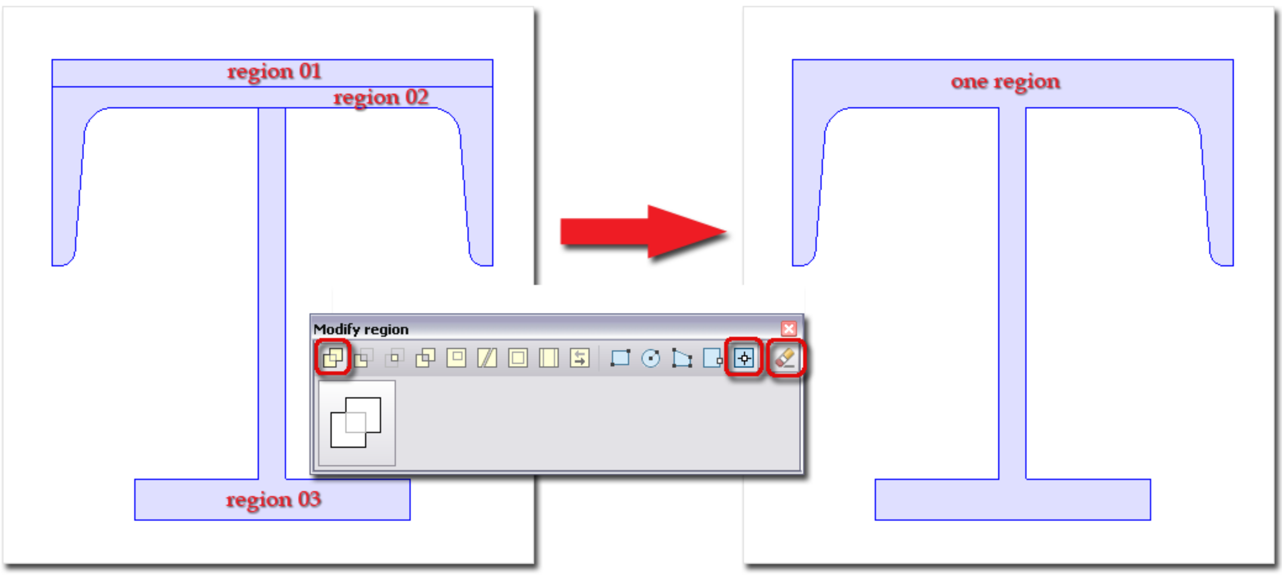


Figure: One complex region after merging region parts

The following settings (*Settings* menu) effect on sectional data calculation:

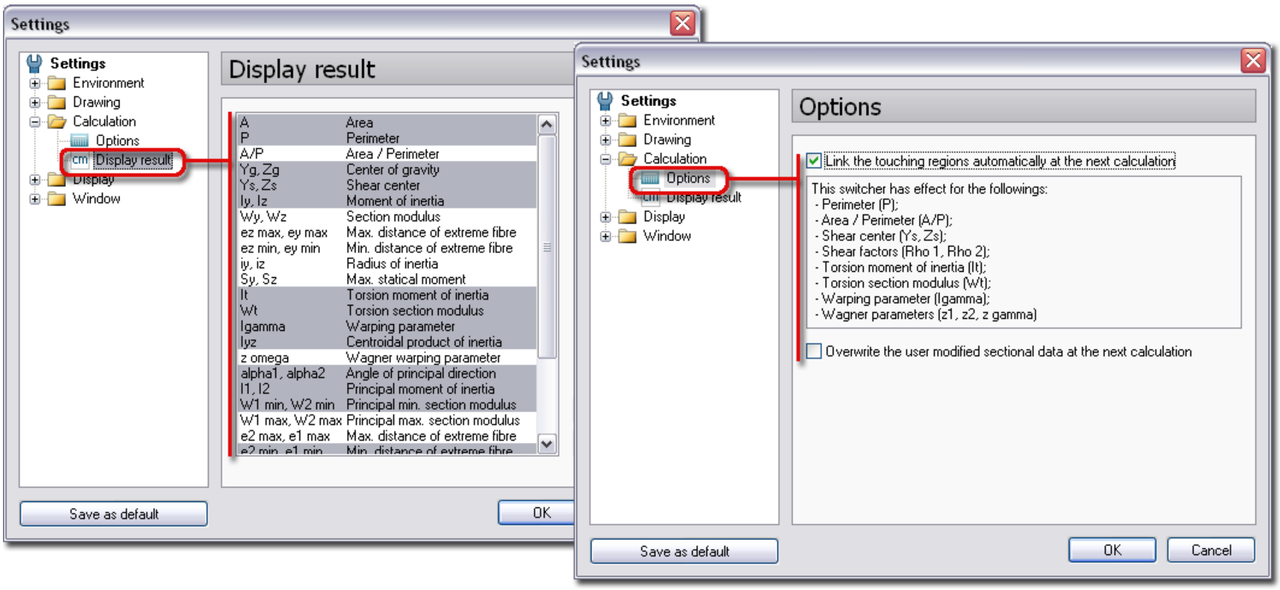


Figure: Calculation settings

- **Display result**

  Set data types you want to display after calculation in the *Sectional data window*. The data list contains all available data on which multiply-selection such as [Shift] and [Ctrl]. (See the meanings of the sectional data at *Sections*.) All data can be selected by the *Select all* button.

- **Calculation options**

  The *Link the touching regions automatically at the next calculation* option handles the connected regions as one complex region in calculations. The option influences the value of the following parameters: perimeter (*P*), area/perimeter (*A/P*), shearing center (*Ys, Zs*), shearing factors (*Rho y, Rho z, Rho yz, Rho 1, Rho 2*), torsion moment of inertia (*It*), torsion section modulus (*Wt*), warping parameter (*Igamma*) and Wagner parameters (*z1, z2, z gamma*).

To calculate the sectional data set at *Monitored data* click *Calculate* on the Cross-section tabmenu. The results are automatically displayed in the *Sectional data window*.

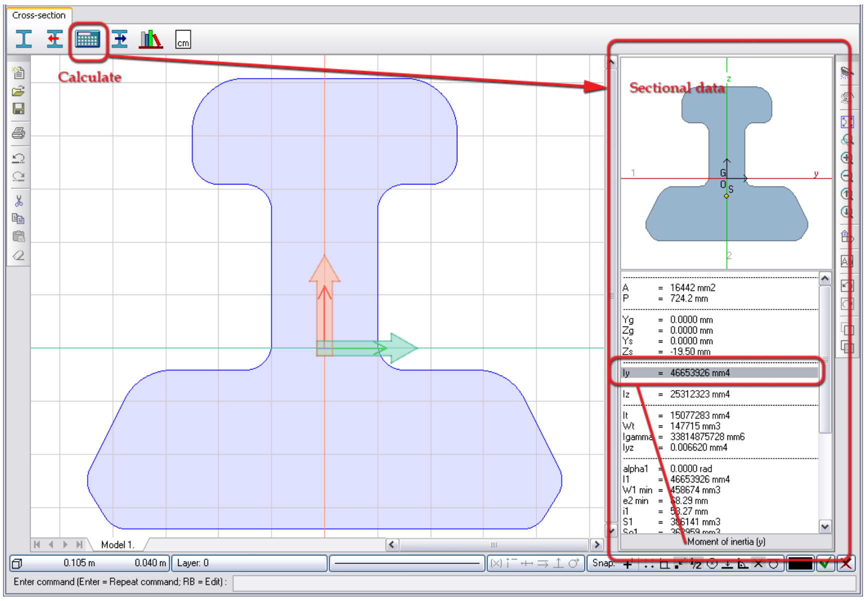


Figure: Calculated sectional data

If you make modifications on the section geometry, you have to rerun the sectional data calculation with *Calculate*.

As a modeling feature, a calculated cross-sectional data can be manually modified/overwritten manually with arbitrary given value. With this possibility the user can create virtual cross-section with known data. Double clicking on a data, custom value can be set for it in the *Modify sectional data* dialog. Modified data is signed with ":" instead of "=" in the *Sectional data window*. The original value can be reset for the current data with the dialog's *Calculate* option, or you can reset all data with the *Overwrite the user modified sectional data at the next calculation option* (*Settings* > *Calculation options*).

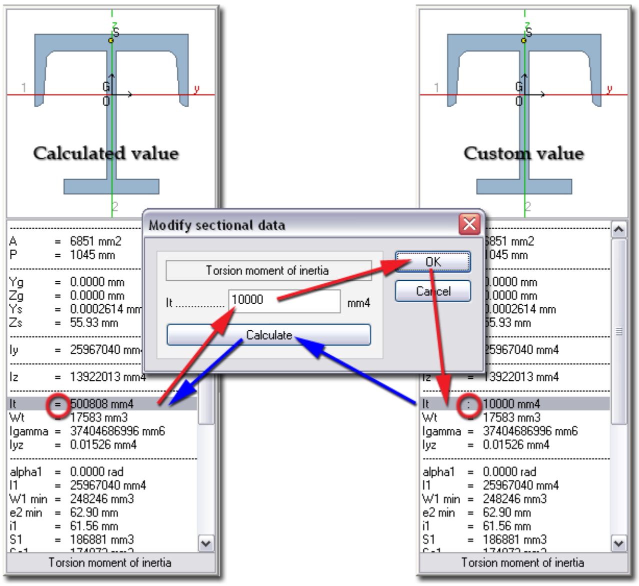


Figure: Customizable sectional data

Documentation of Cross-Section

With the *Insert report* command, the table of the calculated sectional data can be placed on the drawing area based on the *Display result* setting.

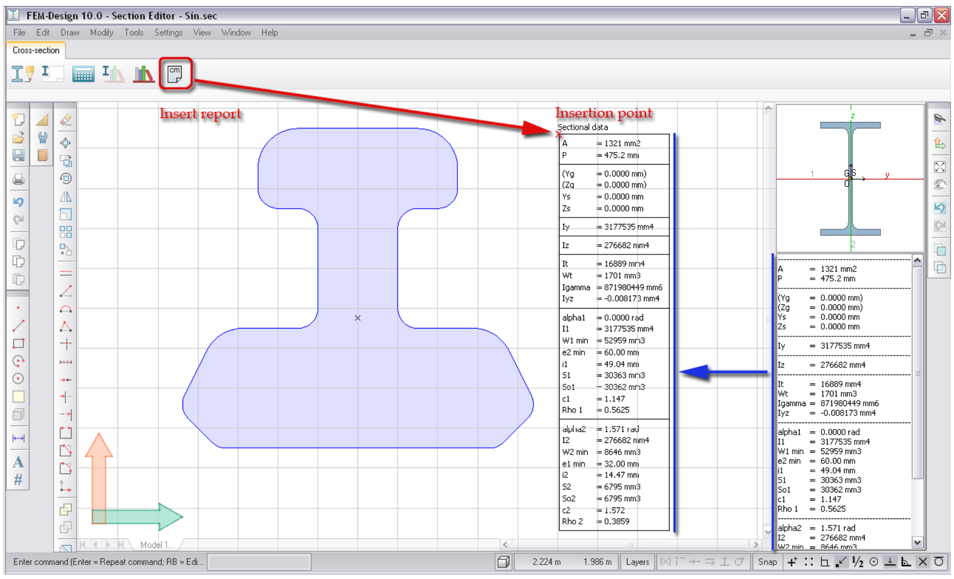


Figure: Sectional data inserted to drawing area

If you make modifications on the section geometry, you have to rerun the sectional data calculation with *Calculate* and the content of the table refreshes automatically.

The table is stored on the *Sectional data* object layer. The color of the table can be modified together with the layer's color. More display changes such as the modification of the text and border style can be done with the *Properties* > *Change properties* tool (*Edit* menu).

The defined section is lying on the *Cross-section* object layer, so the default blue color of section regions can be modified together with the layer's color.

Insert section into library – Section Library

To add a new section having valid calculations to the *Section library*, click *Register* and set the required settings such as the name of the storing folder (*Library* name) and the *Type* and *Size* IDs. Also set the fabrication technology effect on the later available design calculations. To store the section press *OK*.

⚠ Before registering the current section click *Calculate*.

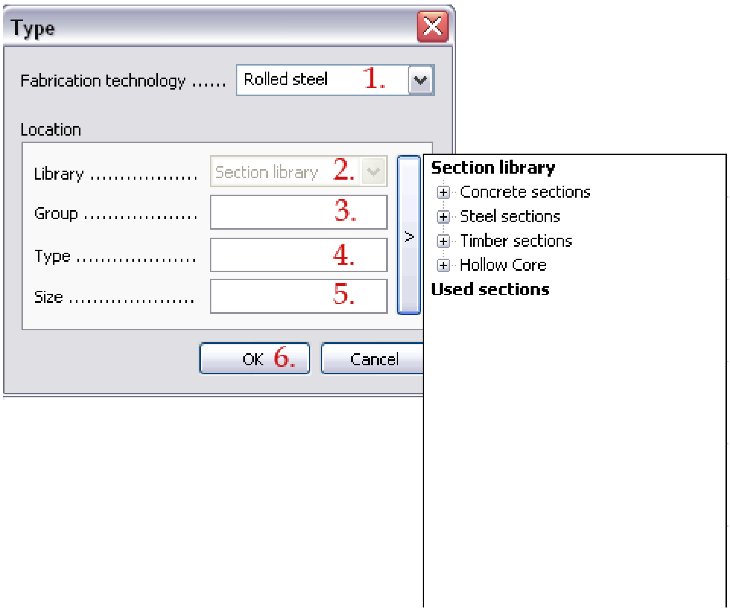
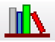


Figure: New section registered to Section library

Fabrication technology	Design
Rolled steel	*Steel bar design*
Cold-formed steel	*Steel bar design*
Weld steel	*Steel bar design*
Concrete	*RC bar design*
Timber	*Timber bar design*
Generic	- (only for analysis)

Table: Fabrication technology and the assigned design type

The program automatically detects the shape type of the sections and shows it as a symbol. It is important for the later design calculations.

The content of the *Section library* can be edited (renaming, deleting etc.) with the  *Edit library* command.

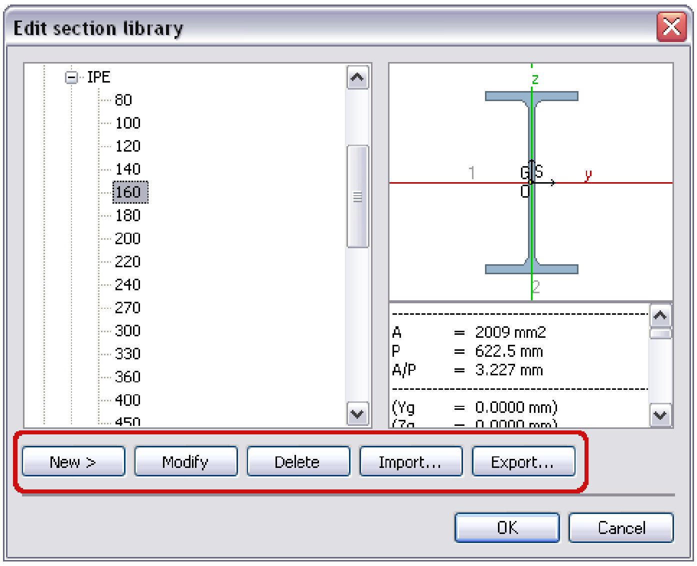


Figure: Library and content editing

A registered section can be added to beam and column elements by choosing it in the *Section* tabpage of the element settings dialog (*Default settings*).

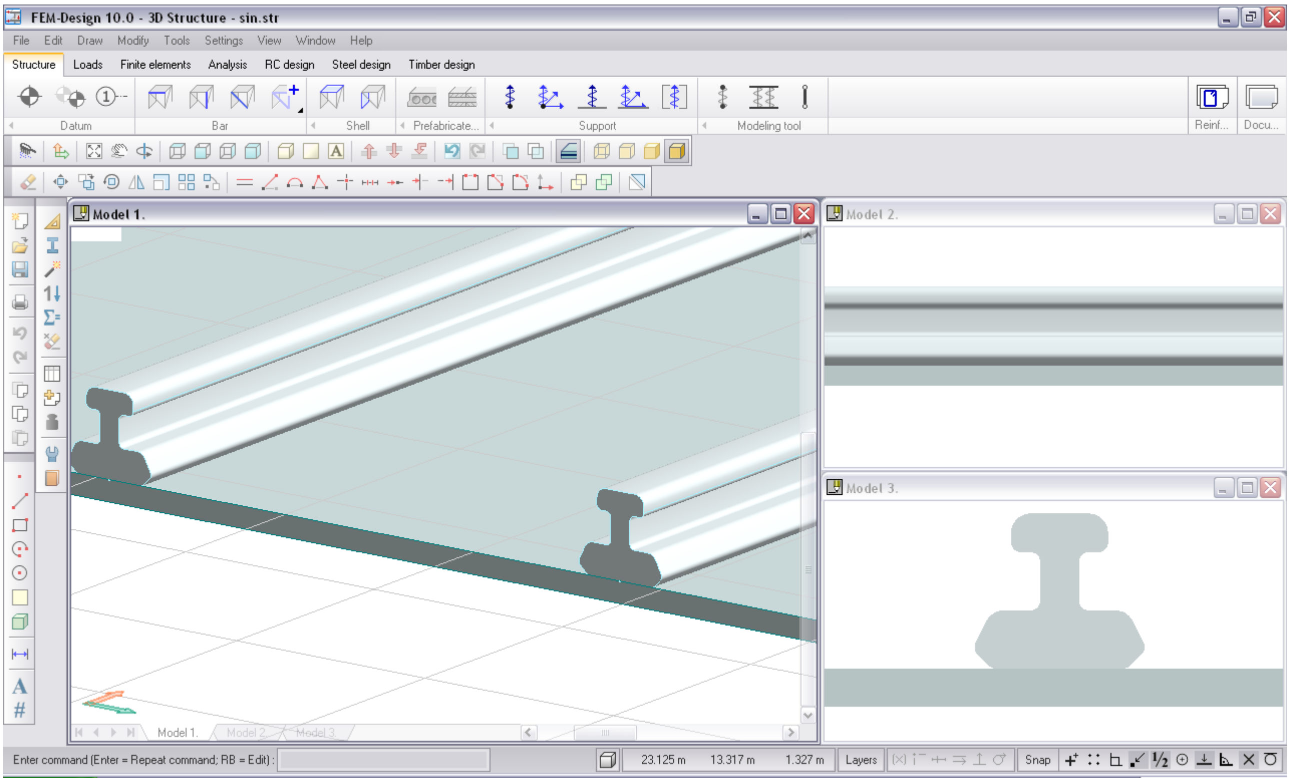


Figure: User-defined section used for bar elements (beams)

LOADS

After defining the **structure** and **support** elements, the next step is adding loads to the statical system. This chapter sums all load possibilities, load definition (directions and geometry) and combination modes. **Mass** definition required for dynamic calculation is also introduced in this chapter. Let's click on Loads tabmenu to reach all load definition commands.

Load Types

Depending on the current **FEM-Design Module** (license you have), the available type of load commands is different. Loads can be defined with their insertion point, action line or action region.

Type	Modules where available	Definition mode	Direction	Intensity
Structural dead load		(Automatic)	Global Z-axis	-
Soil dead load		(Automatic)	Global Z-axis	-
Point load (force and/or moment)		Point	Horizontal Arbitrary	Constant
Line load (force and/or moment)		Line	Horizontal Arbitrary	Constant/Variable
Surface load (force)		Region	Horizontal Arbitrary	Constant/Variable
Line temperature variation load		Line	Fixed Arbitrary	Constant/Linearly variable Constant/Variable
Surface temperature variation load		Region	Fixed Arbitrary Arbitrary	Constant/Linearly variable Constant/Linearly variable Constant/Variable
Line stress load		Line	Fixed Arbitrary	Constant/Variable

Surface stress load		Region	Fixed Arbitrary Arbitrary	Constant/Variable
Point support motion (motion and/or rotation)		Point	Support direction	Constant
Line support motion (motion and/or rotation)		Line	Support direction	Constant Constant/Variable
Surface support motion (motion)		Region	Horizontal Support direction	Constant/Variable
Shrinkage		(Automatic)	-	-
Seismic load		(Automatic)	-	-
Wind		(Automatic)	Horizontal	Constant
Snow		(Automatic)	Vertical	Constant
Deviation		(Automatic)	Horizontal	Constant
Moving Load		Special	Arbitrary	Special

Table: Load types and their main properties

Load Direction

Most of the load objects need direction settings. The next table summarizes only the editable direction possibilities by load types. Other load types have fixed direction (for example in *Plate* module, *Force* direction is always perpendicular to the calculation plane, so it is parallel with the global Z direction).

Type	Modules where available	Direction for	Direction Modes
Point load		Moment direction	Predefined direction Parallel with line Perpendicular to plane
		Force direction	
		Force/Moment direction	
Line load		Moment direction	Predefined direction Parallel with line
		Force direction	

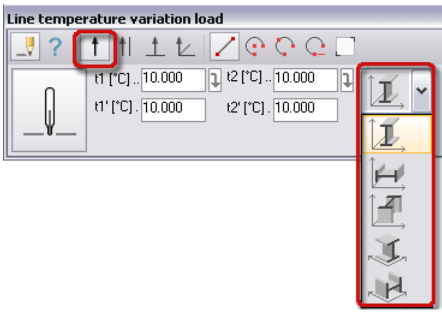
		Force/Moment direction	Perpendicular to plane Object's local system
Surface load		Force direction	Predefined direction Parallel with line Perpendicular to plane Object's local system
Line temperature variation load		Direction	Predefined direction Parallel with line Perpendicular to plane Object's local system
Line stress load		Direction	Predefined direction Parallel with line Perpendicular to plane Object's local system
Surface stress load		Direction	Predefined direction Parallel with line Perpendicular to plane Object's local system
Point support motion load		Rotation direction Motion direction Motion/Rotation direction	Predefined direction Parallel with line Perpendicular to plane Object's local system
Line support motion load		Rotation direction Motion direction	Predefined direction Parallel with line

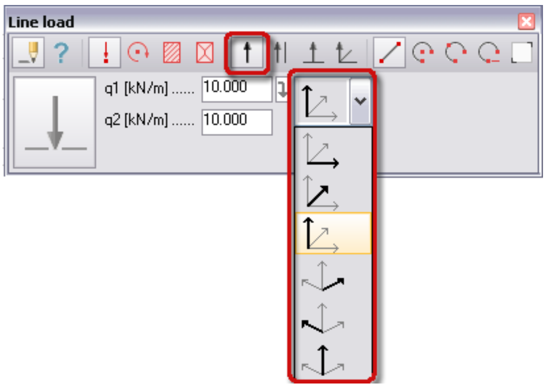
		Motion/Rotation direction	**Perpendicular to plane** **Object's local system**
Surface support motion load		Motion direction	**Predefined direction** **Parallel with line** **Perpendicular to plane** **Object's local system**

Table: Load types and their directions

Predefined direction

With this option an axis/a plane of the **Global** or the **User-defined (UCS)** co-ordinate system can be set for the required load direction. The direction can be chosen from the drop-down list attached to the *Predefined direction* option. The available directions depend on the applied FEM-Design Module (e.g. *Plate, 3D Structure* etc.).





Symbol	Meaning of direction	System
	Parallel with XY plane	Global
	Parallel with YZ plane	Global
	Parallel with XZ plane	Global
	Parallel with UCS (XY plane)	UCS
	Perpendicular to UCS (XY plane)	UCS
	Parallel with global X axis	Global
	Parallel with global Y axis	Global
	Parallel with global Z axis	Global
	Parallel with X axis of UCS	UCS
	Parallel with Y axis of UCS	UCS

	Parallel with Z axis of UCS	UCS

Table: The available directions to set the new load direction

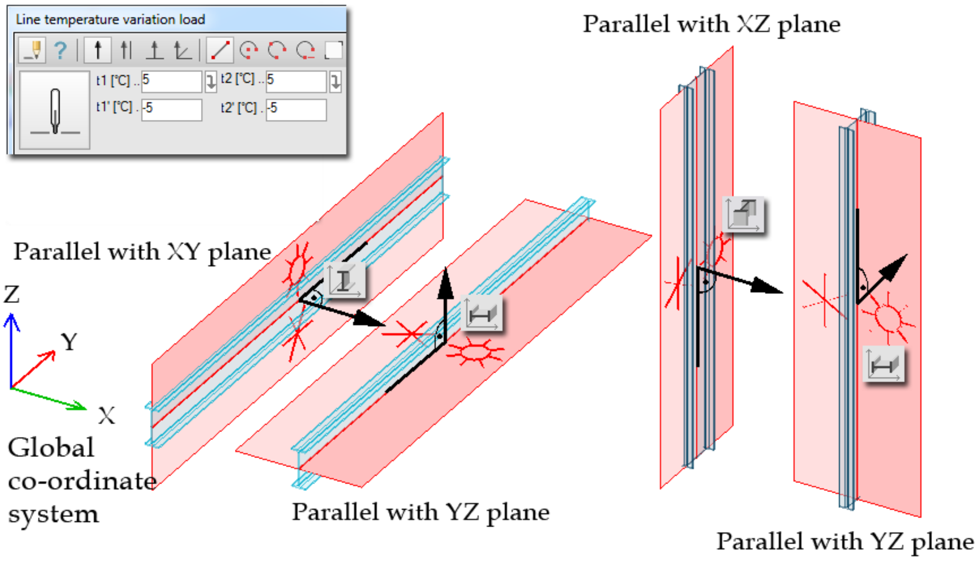


Figure: Examples of temperature loads placed on bar elements

In some cases, additional direction setting can be chosen from the definition tool palette:

"Positive direction": The orientation is the same with the selected axis orientation;

"Negative direction": The orientation is the opposite of the selected axis orientation.

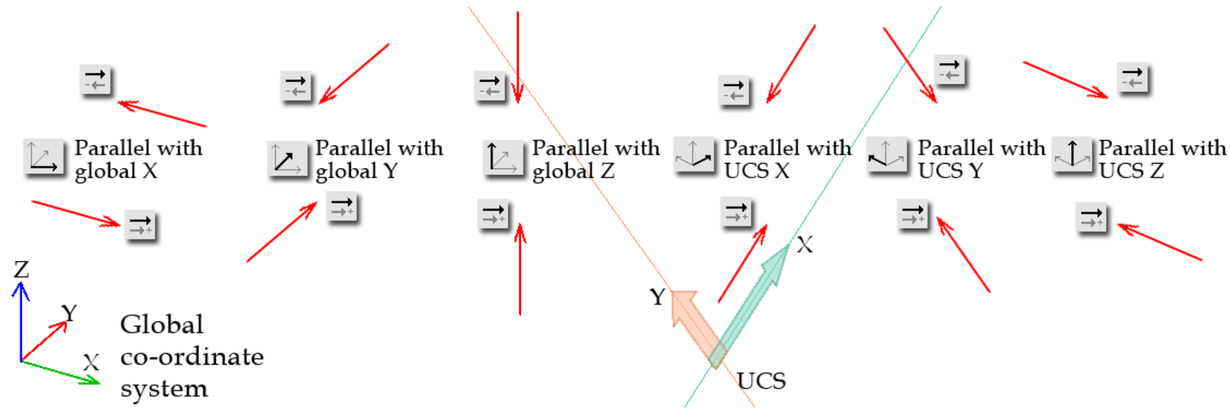


Figure: Examples of Point load direction

Parallel with line

With this option, the required load direction can be defined manually with its start and end points.

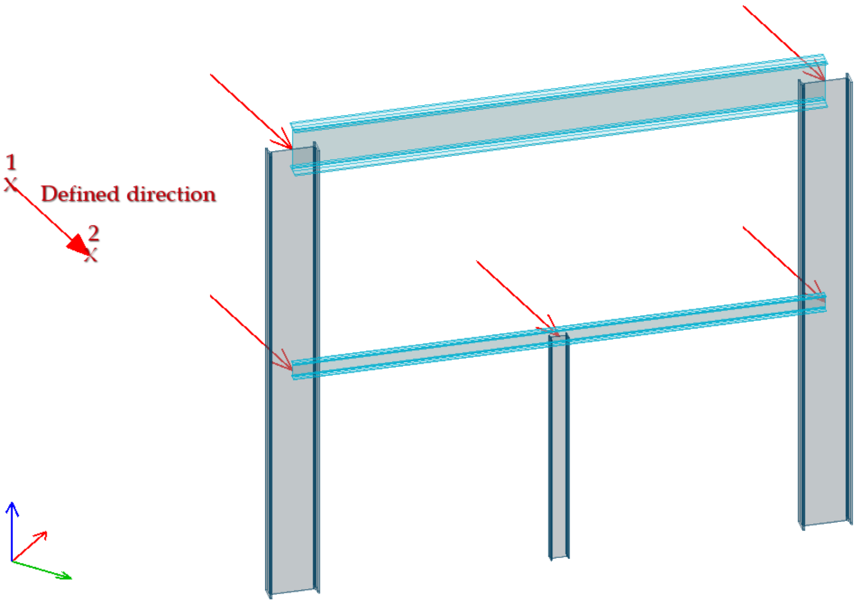


Figure: The load direction is parallel with the defined direction

Perpendicular to plane/line

With this option, the required load direction will be perpendicular to a defined plane/line. The plane can be given with three points and the line with two points (start and end points). In case of the perpendicular plane, the third point defines the final orthogonal direction, which the new direction will be parallel with.

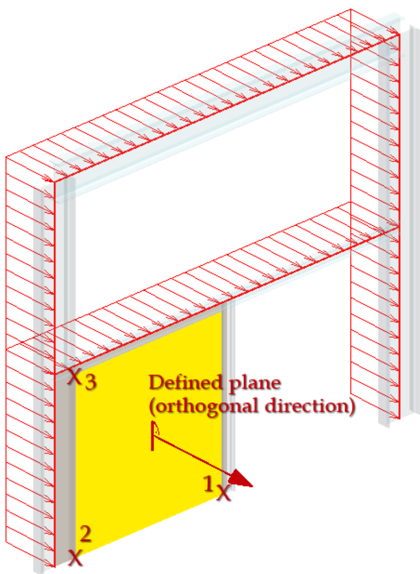


Figure: The line load direction is perpendicular to the defined plane

Object's local system

If the load direction has to be set in the local co-ordinate system of the "assigned" structural object (beam, column, plate, wall and support), the fastest definition of load direction is to use the *Objects' local* *system* option. This feature is available for *Line*, *Surface*, *Line temperature variation*, *Line stress*, *Surface stress*, *Point support motion*, *Line support motion* and *Surface support motion load* in the 3D modules.

Using *Object's local system*, the geometry definition of the line or surface load is skipped, because one click on the assigned structural object is enough after setting the requested local system axis direction.

⚠ In case of surface loads, only load with constant intensity can be defined with the tool *Select objects to load in local system*. But, you can change intensity values (linear distribution) with the **Variable intensity** tool of the applied load command as a next step.

One-click definition of constant surface loads perpendicular (e.g. wind) to planar objects (e.g. shells).

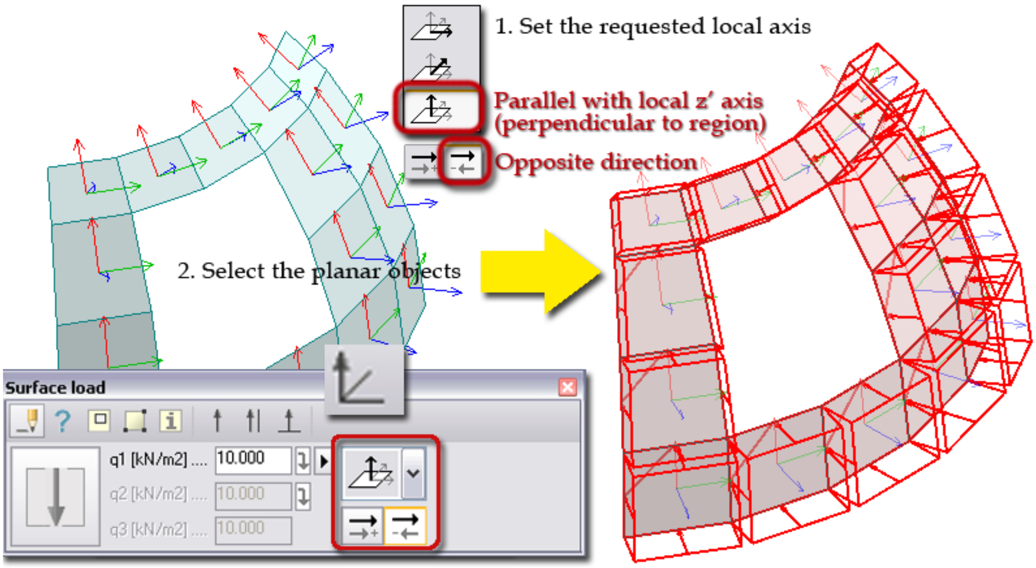


Figure: Fast definition of surface load by using object's local system

One-click definition of line load (constant or variable) perpendicular (e.g. wind load) to a beam reference line.

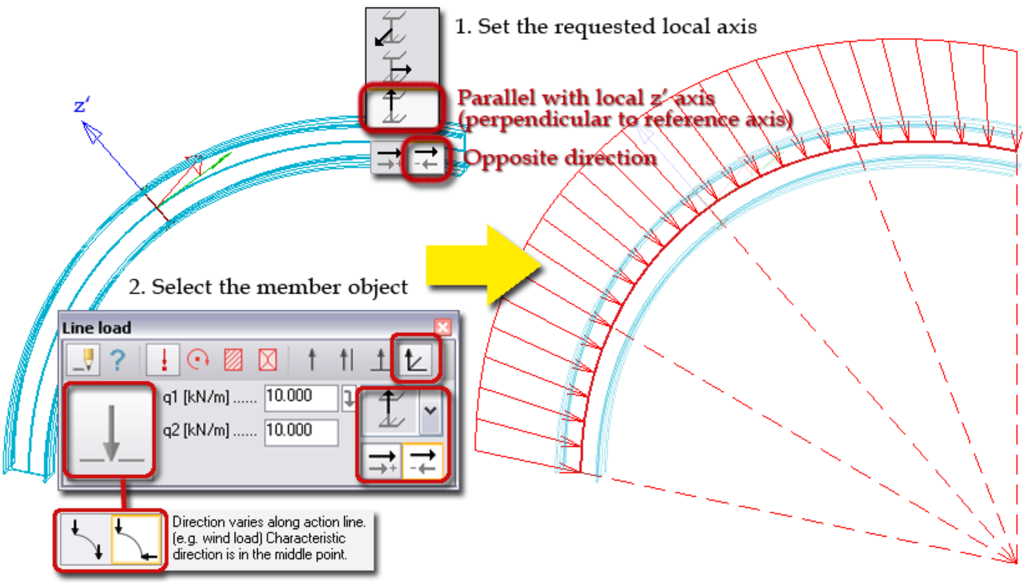


Figure: Fast definition of line load by using object's local system

One-click definition of point support motion parallel with a component direction of a point support.

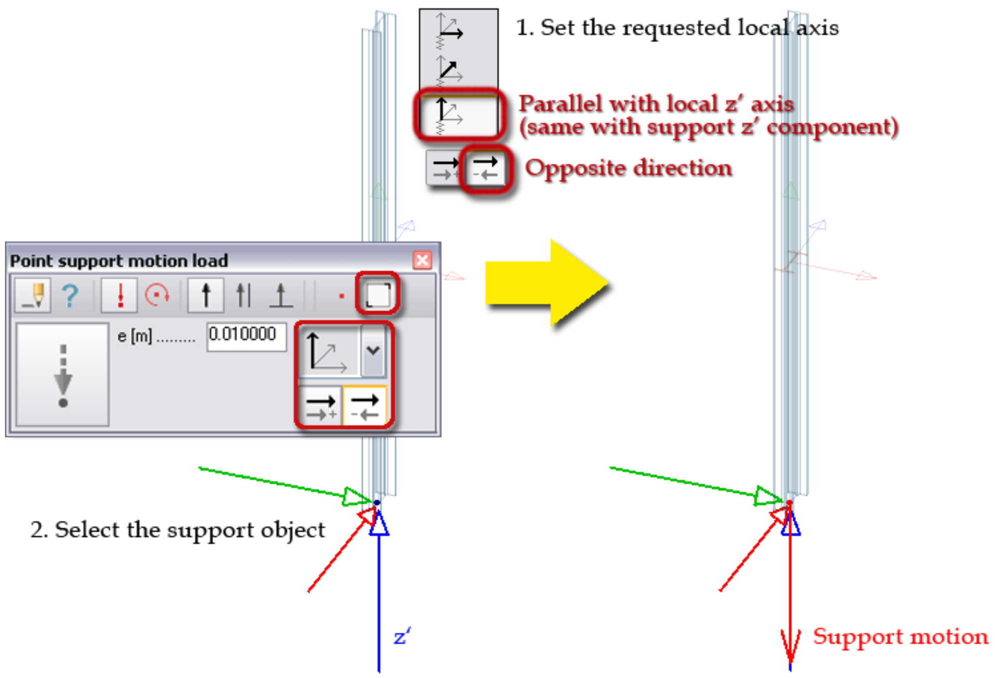


Figure: Support motion added to a point support component

Only the same type support can be selected for a support motion:

Support motion load type	Support type

Point support motion load	Point support
Line support motion load	Line support
Surface support motion load	Surface support

Table: Support motion load types and their proper support types

Change Direction

Any previously set direction can be modified by the modifying commands (*Modify* menu): **Change direction** and **Rotate**.

Change direction uses the **Predefined direction**, **Parallel with line** and **Perpendicular to plane** direction definition tools.

Rotate edits a selected direction or the main direction of a selected system with giving new direction points or the rotation angle. Rotation works around a given point or an axis.

⚠ Naturally, you cannot modify fix directions. For example in the *Plate module* the point, line and surface forces are always vertical (perpendicular to the calculation plane of the plates).

Load Geometry

The definition modes and the available shape of the loads' action line/plane depend on:

- the load type: point, linear and surface load, and
- the current FEM-Design module.

The Tool palette of a load command contains only the available modes. The next table summarizes the geometry possibilities by the load types.

Type	Modules where available	Definition mode	Geometry
Point load		Reference point	Edit point (give an insertion point) **Select point**
Line load		Reference line	**Straight line** **Arc by center, start and end points** **Arc by 3 points** **Arc by start, end point and tangent** **Line by selection**
Surface load		Reference region	**Rectangular** **Circular**

			Polygonal Pick lines Pick existing region
Line temperature variation load		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Surface temperature variation load		Reference region	Rectangular Circular Polygonal Pick lines Pick existing region
Line stress load		Reference line	Straight line Arc by center, start and end points Arc by 3 points Arc by start, end point and tangent Line by selection
Surface stress load		Reference region	Rectangular Circular Polygonal Pick lines Pick existing region
Point support motion load		Reference point	Edit point (give an insertion point) Select point

Line support motion load		Reference line	**Straight line** **Arc by center, start and end points** **Arc by 3 points** **Arc by start, end point and tangent** **Line by selection**
Surface support motion load		Reference region	**Rectangular** **Circular** **Polygonal** **Pick lines** **Pick existing region**

Table: Loads and their geometry definition

If you define a load that does not acting on a structural element, a warning message appears during the calculation. Continuing the calculation does not take the "unconnected" load into consideration.

It follows from the previous fact, that if the documentation does not need "perfect geometry" for loads in some cases, you can spare time with neglecting some editing steps.

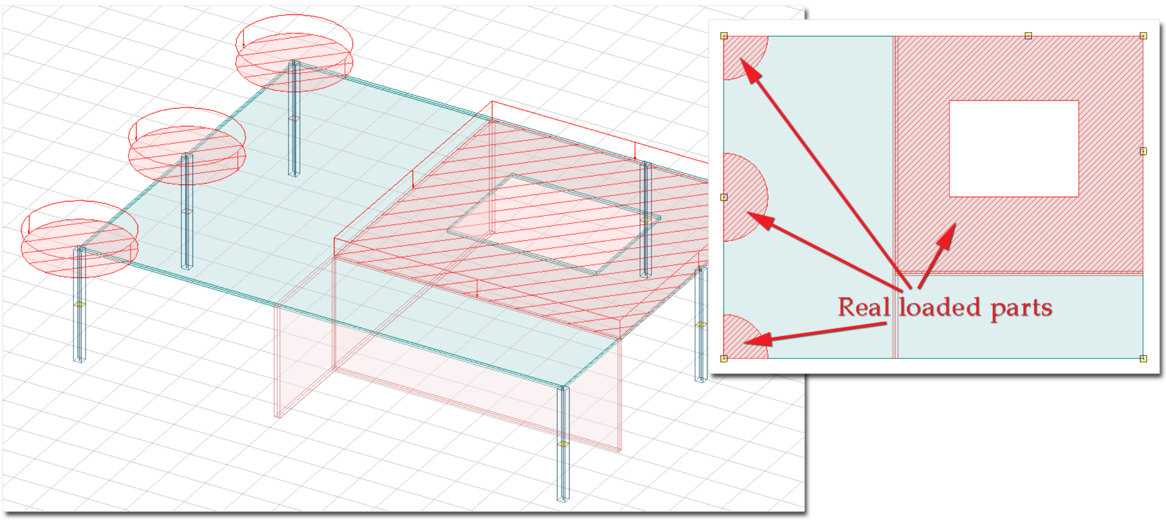


Table: Uncut, but unnecessary load parts

The units of the loads can be set at the **Settings > Units**.

Straight line

The steps of a straight line definition:

3. Define the start point of the line by giving coordinates or mouse-clicking.
4. Define the end point of the line by giving coordinates or mouse-clicking.

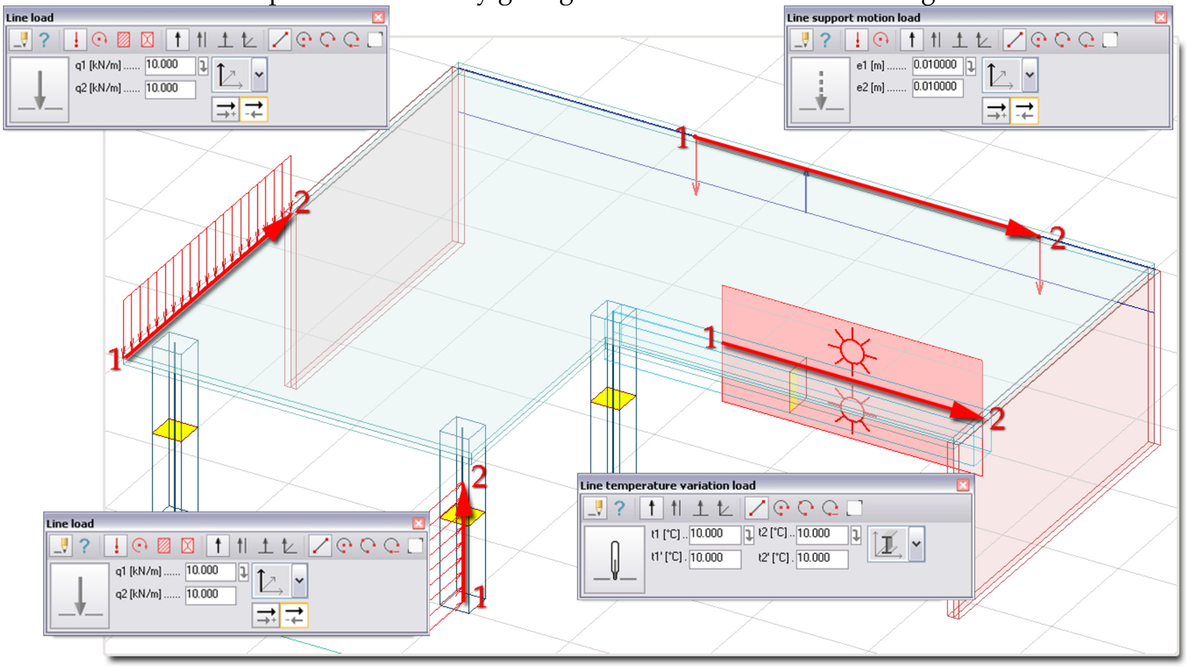


Figure: Some examples for defining line-type loads with Straight line

In 3D modules, the "curved" walls are modeled as planar shells instead of curved ones, so their base lines are straight segments and not real arcs. So, if you would like to place line loads on the reference contour of 3D walls, use the *Straight* or the **Line by selection** tool instead of one of the arc definition tools (see later). Otherwise, the misplaced load will not be taken into consideration in calculations.

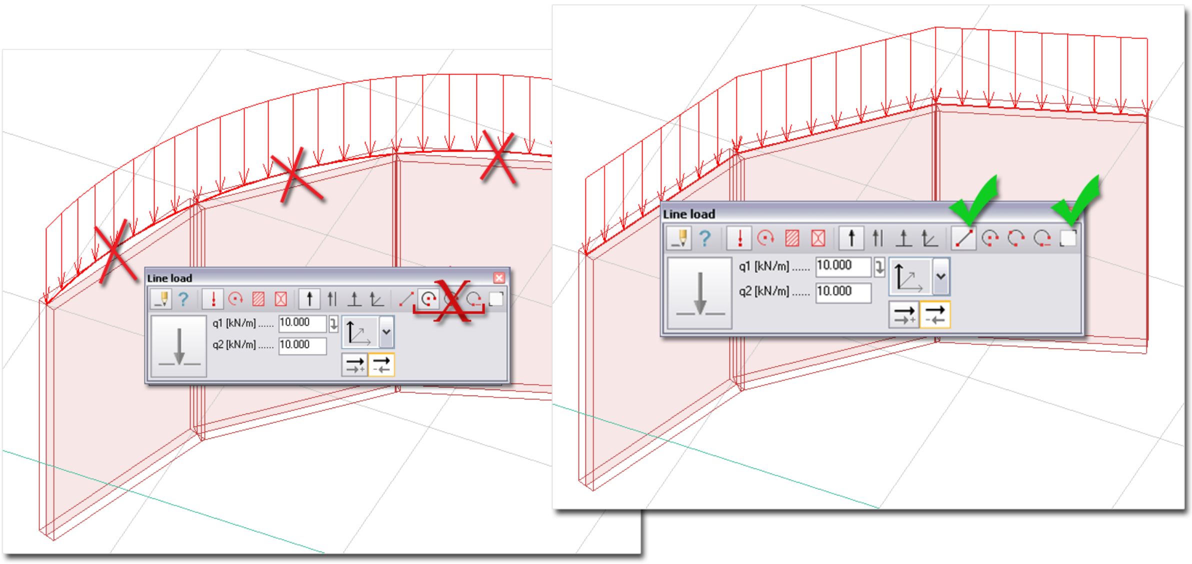


Figure: Incorrect and correct ways to define line-type loads on "curved" wall edges in the 3D modules

Arc by center, start and end points

The steps of an arc definition with its center, start and end points:

4. Define the center point of the arc by giving coordinates or mouse-clicking.

   Use the **Center Object Snap** tool, if you would like to define the center point of the line load in another center point of an arc. (See the next figure, where the (1*) step means that the center point of the curved slab edge is selected for the load's center point.)

5. Define the start point of the arc by giving coordinates or mouse-clicking.
6. Set the drawing direction (clockwise or counterclockwise) with mouse-clicking. Define the end point of the arc by giving coordinates or mouse-clicking, or set the central angle (4.) by giving its value. Circle can be defined by angle 360°.

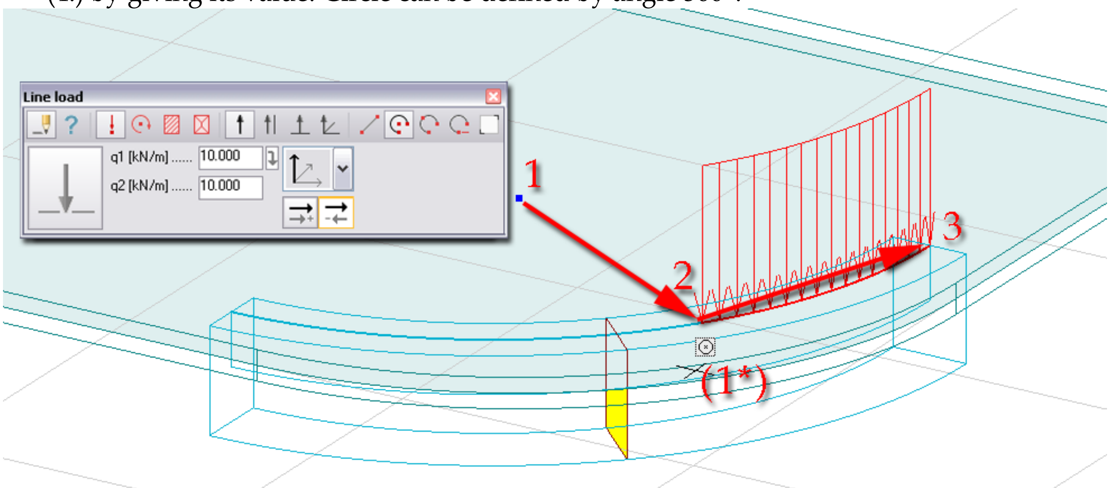


Figure: An example for defining line-type loads with Arc by center, start and end points

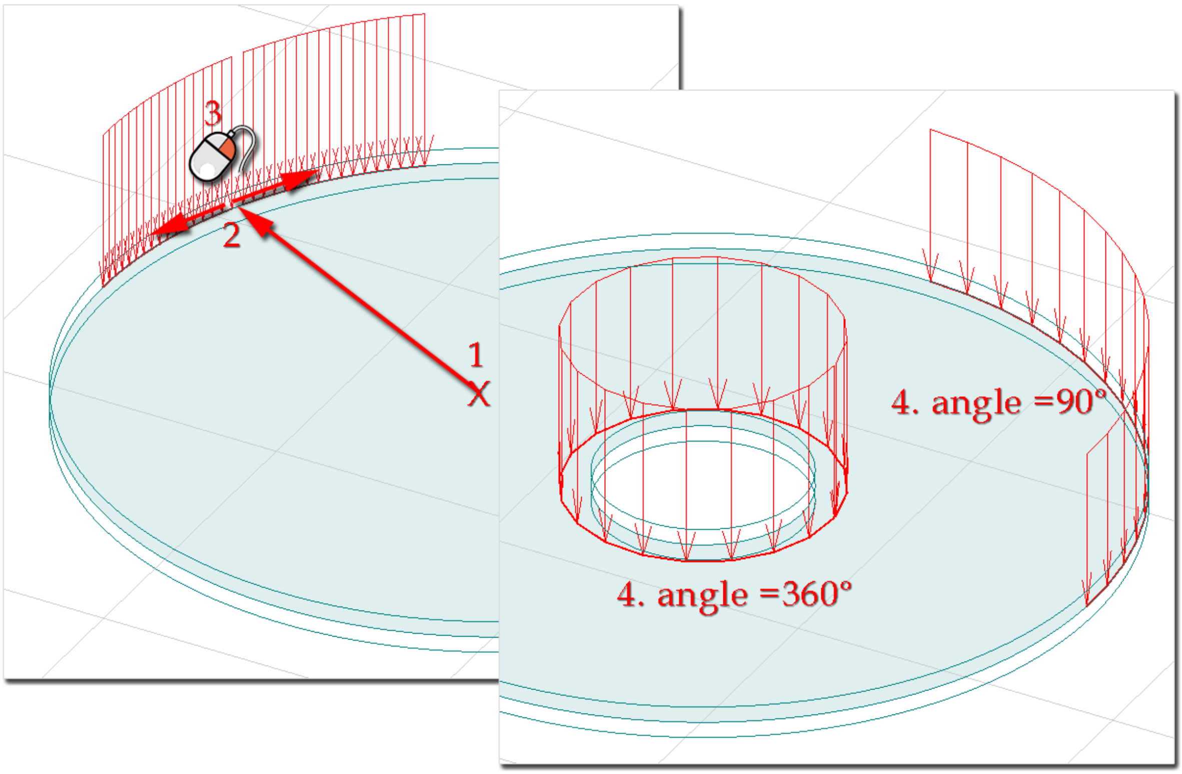


Figure: Drawing direction and angle definition

Arc by 3 points

The steps of an arc definition with its three points:

4. Define the start point of the arc by giving coordinates or mouse-clicking.
5. Define the end point of the arc by giving coordinates or mouse-clicking.
6. Define the third, peripheral point of the arc by giving coordinates or mouse-clicking.

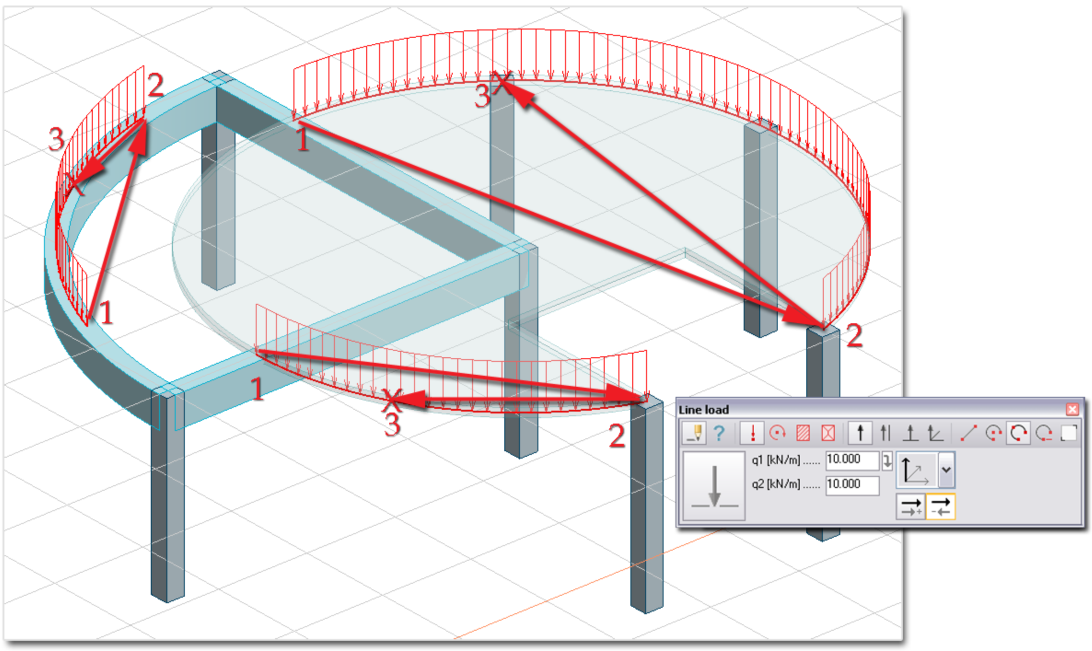


Figure: Some examples for defining curved line load with Arc by 3 points

Arc by start, end point and tangent

The steps of an arc definition with its start, end point and tangent:

4. Define the start point of the arc by giving coordinates or mouse-clicking.
5. Define the end point of the arc by giving coordinates or mouse-clicking.
6. Set the tangent side with mouse-clicking. Define the tangent direction from the start point with a third point (e.g. a point on a tangentially connected line) by giving coordinates or clicking.

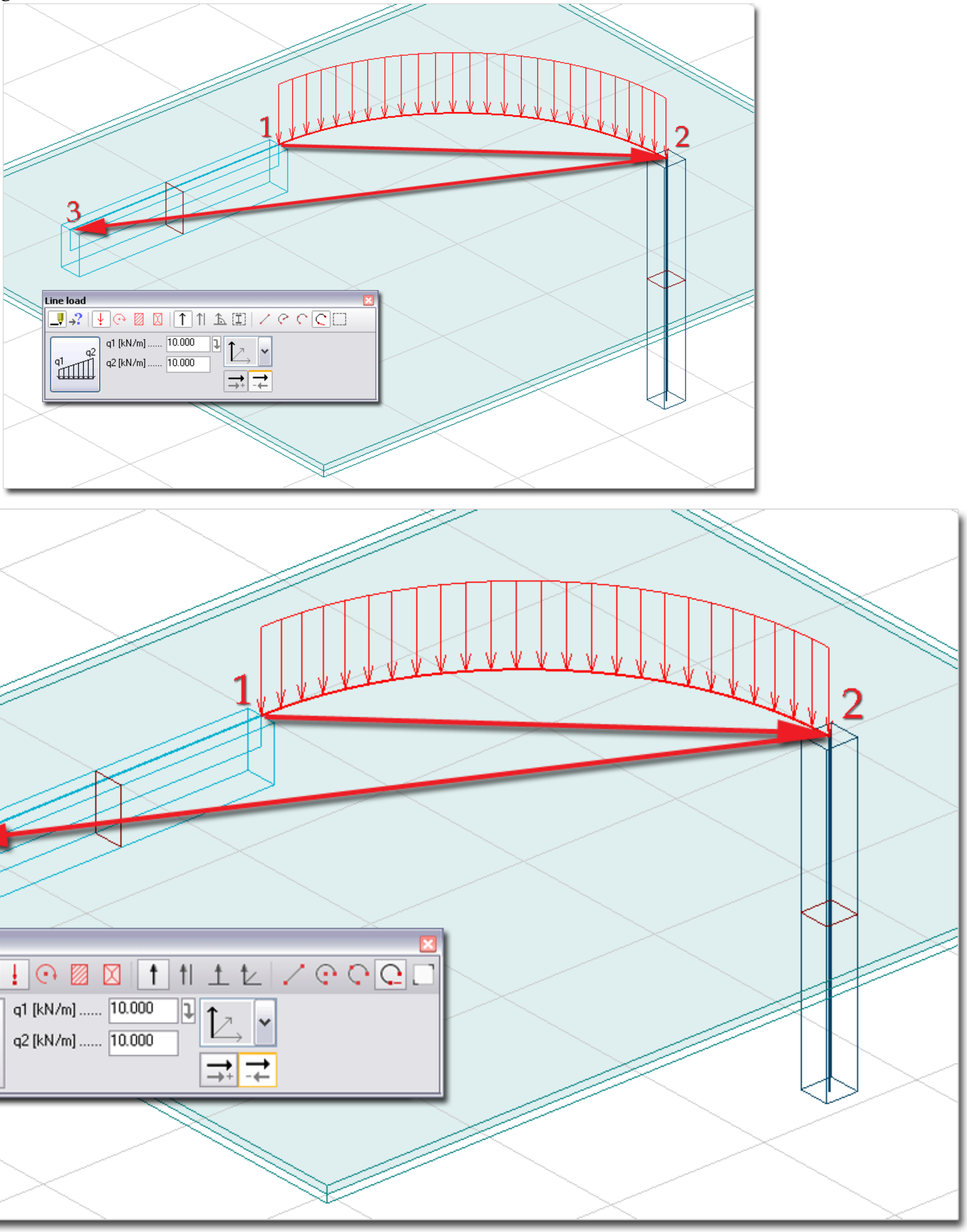


Figure: Definition of a curved line load tangent to a beam

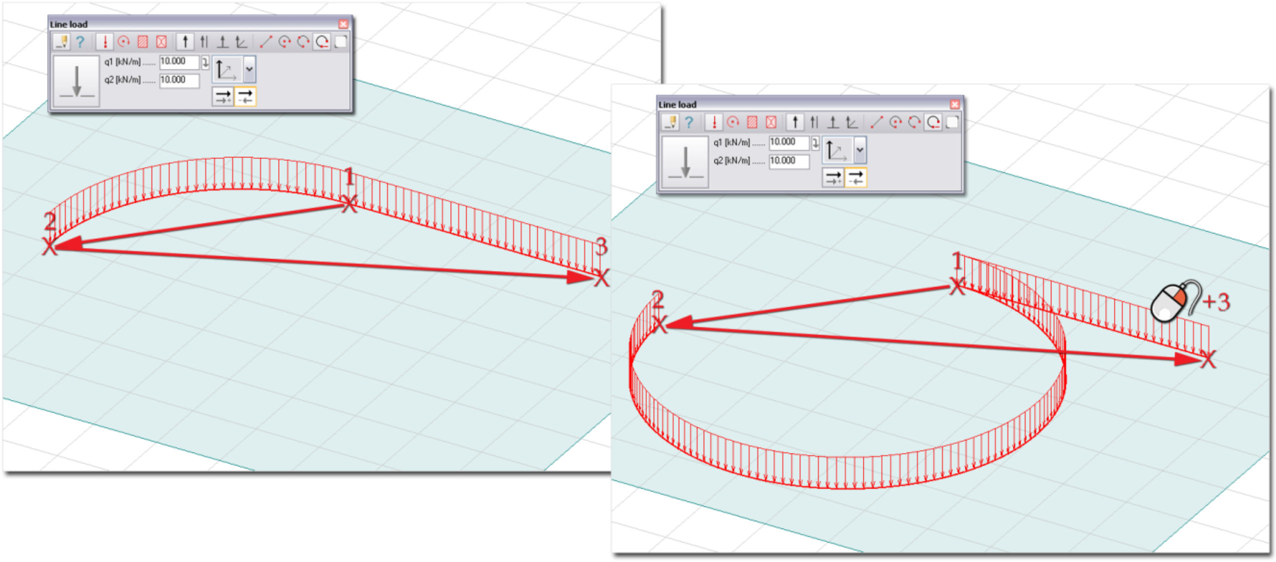


Figure: Although same definition points are defined, the tangent side is different

Select point / Line by selection / Pick existing region

By selecting previously defined points, lines and regions, loads can be inserted very fast, in one step:

- **Point-type loads**: select drawing points or **point supports** (in case of *Point support motion load*) with one of the **selection modes**.
- **Line-type loads**: select drawing lines, reference line of **1D member structural elements**, region (drawing or **structural object**) edges or line supports (in case of *Line support motion load*) with one of the **selection modes**. The length of the loads' action lines will be equal to the selected line elements.
- **Surface-type loads**: select drawing regions, drawing solid surfaces, **Planar objects** or surface supports (in case of *Surface support motion load*) with one of the **selection modes**. The size of the loads' action surface will be equal to the selected region elements.

Because only the loads/load parts located on structural elements will be considered in calculations, these "definition by selection" modes are the easiest technique to define loads with perfect accuracy.

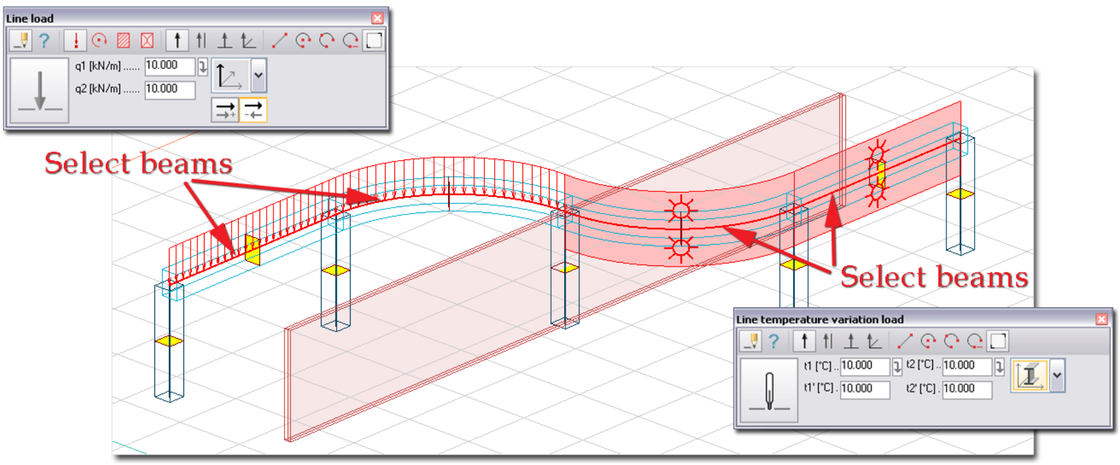


Figure: Examples for defining line-type loads by selecting beams

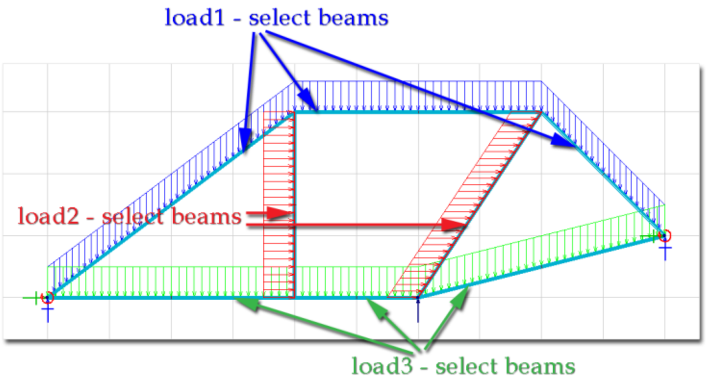


Figure: Adding line loads to the members (beams) of a frame structure

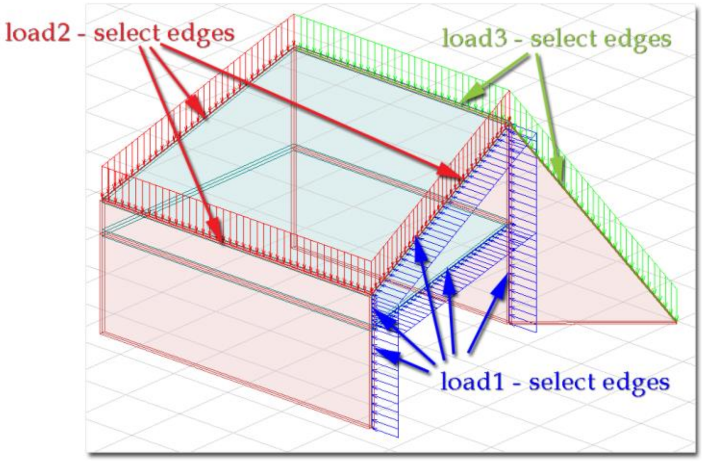


Figure: Defining line loads on structural element edges

In 3D modules, the "curved" walls are modeled as planar shells instead of curved ones, so their base lines are straight segments and not real arcs. So, if you would like to place line loads on the reference contour of 3D walls, use the *Line by selection* or the **Straight** tool instead of one of the arc definition tools. Otherwise, the misplaced load will not be taken into consideration in calculations.

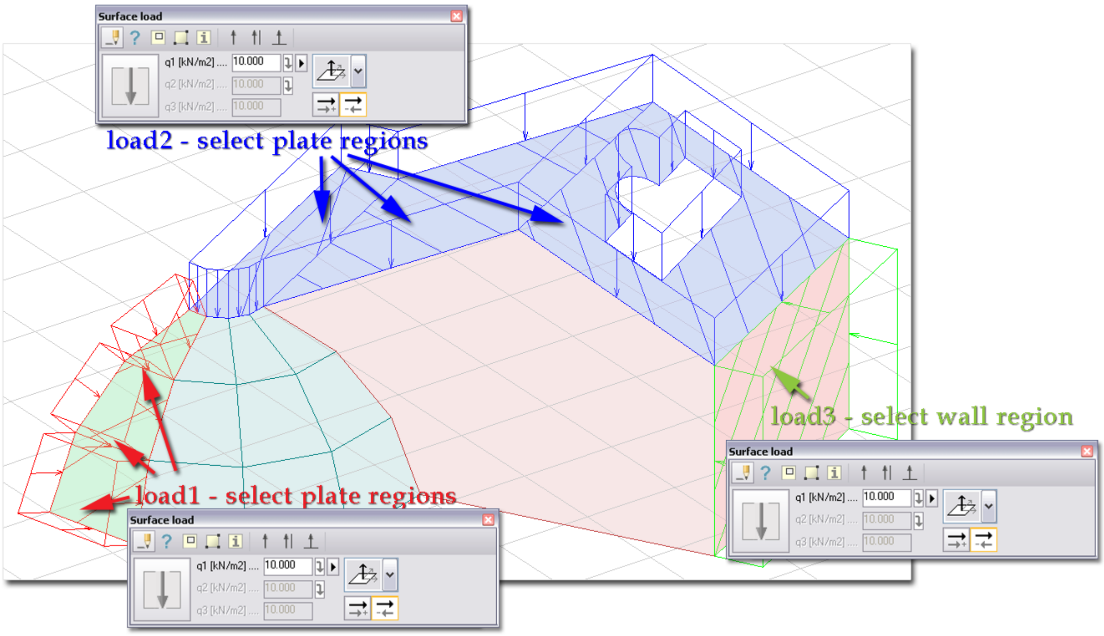


Figure: Defining line loads on structural element edges

With *Pick existing region*, surface loads can be easily place on 3D shell elements (plates and walls) or drawing solid surfaces (defined by the *Draw* > *Solid* command).

Point/Line/Surface support motion loads will be considered in the calculations, if they are placed into supports, so it is recommended to select the reference point/line/surface of previously defined supports in case of motion load definition.

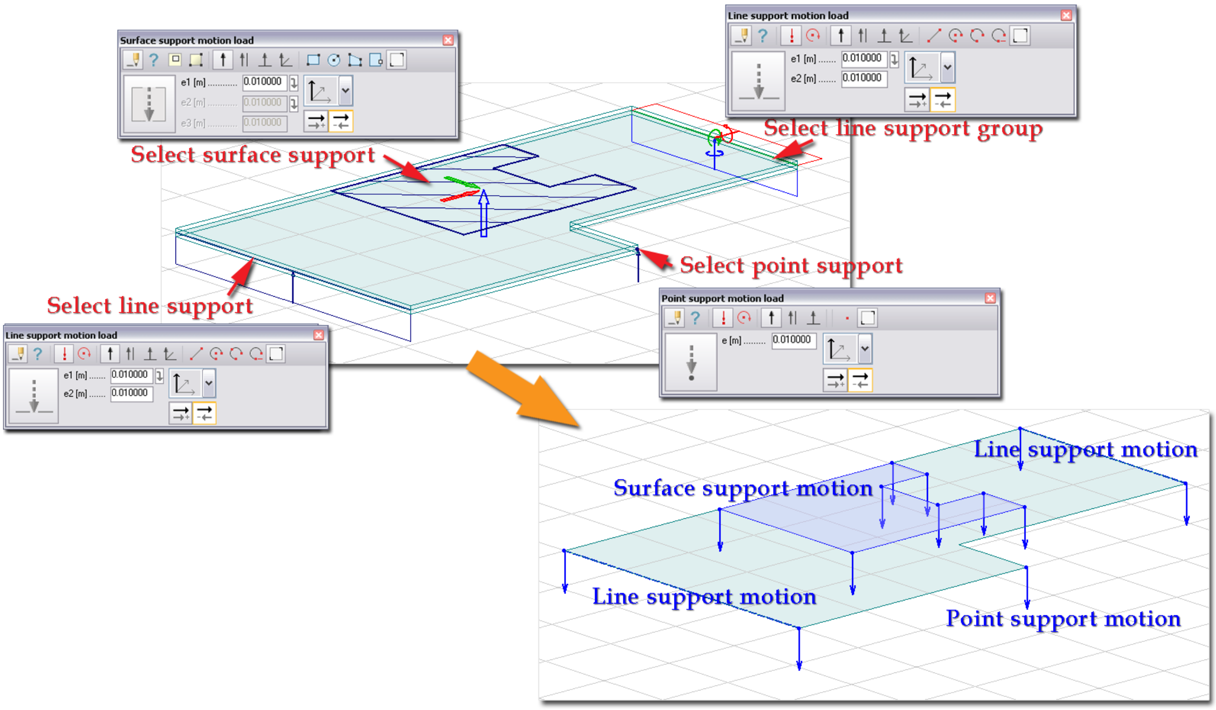


Figure: Defining motion loads in supports

Rectangular

The steps of defining a rectangular surface load:

5. Define the point of the first corner by giving coordinates or mouse-clicking.
6. Define the point of the end corner by giving coordinates or mouse-clicking.

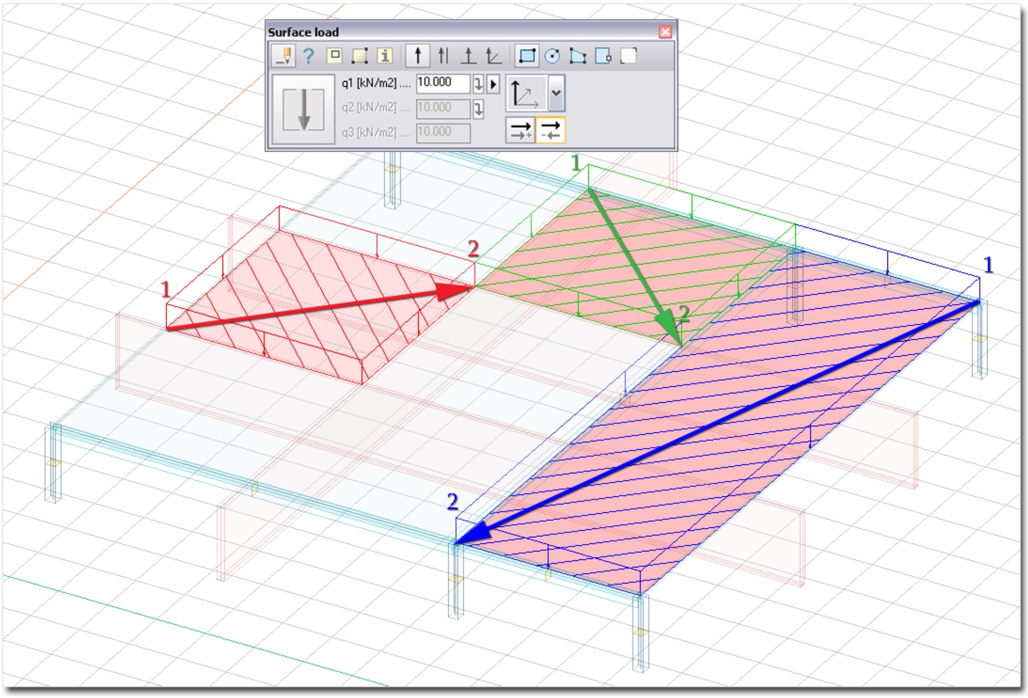


Figure: Defining a rectangular surface-type load

The geometry of rectangular regions as well as other (later mentioned) region shapes can be edited by the **Modify region > Split region** tool and other editing tools (*Edit* menu). Also the **Hole** tool of surface loads' definition command can be used to edit the reference regions.

Circular

The steps of a circular region definition:

1. Define the center point by giving coordinates or mouse-clicking.
2. Define the radius by giving its value or a point on the circle (with coordinates or mouse-clicking).

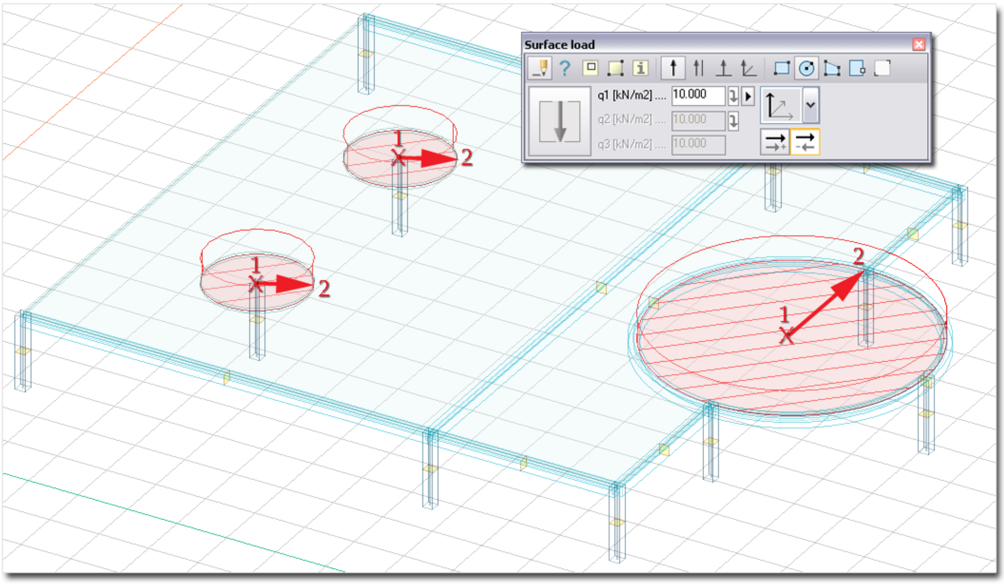


Figure: Defining circular surface loads on a plate above columns and terrace part

Polygonal

The steps of a polygonal region definition:

3. Define the points of the polygon vertexes by giving coordinates or mouse-clicking.
4. Close the polygon with mouse-clicking or key.

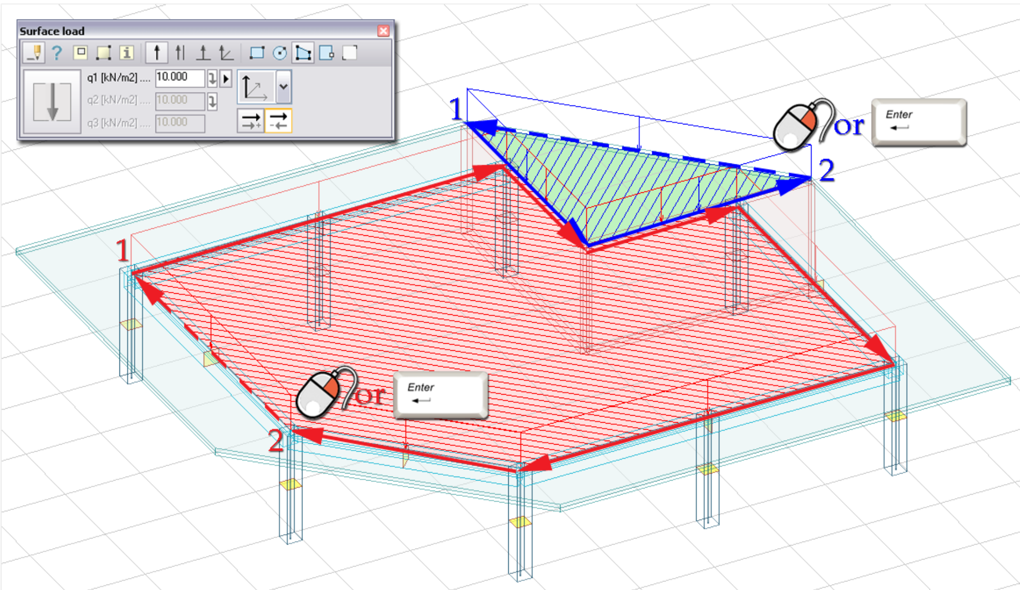


Figure: Defining polygonal surface load

Pick lines

With this method, surface-type loads can be placed on **closed contours** defined by

- previously defined drawing lines,
- the edges of a previous defined drawing region, a plate, a wall (only in 3D modules) or surface support, or

- imported (DWG/DXF) drawing lines that can be used as sketches of surface load shapes.

It is a one-click definition mode: select a closed contour defines the requested shape of the surface load with mouse-clicking.

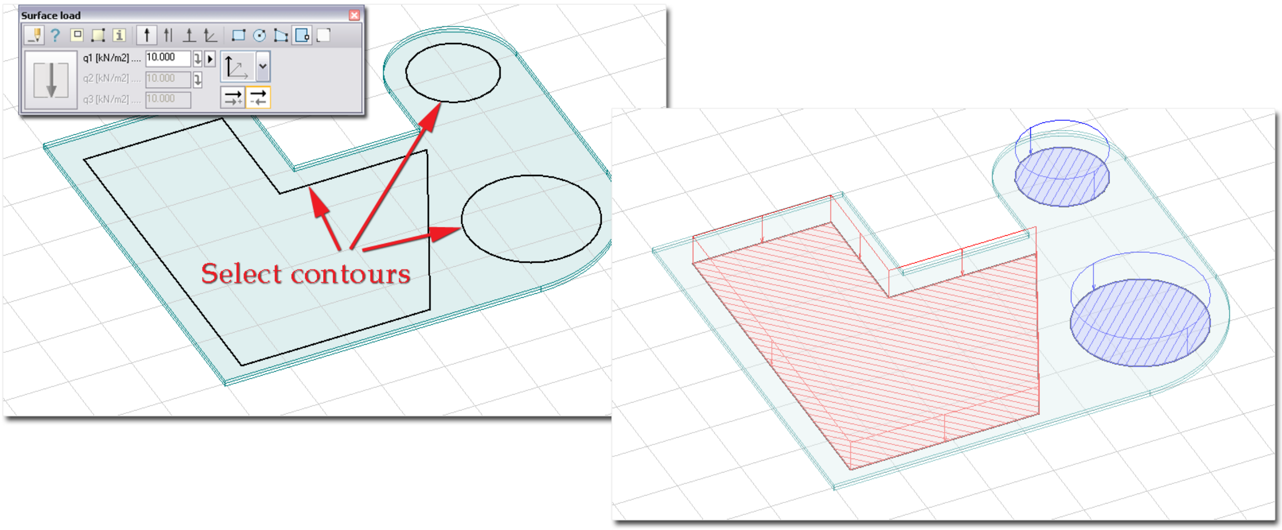


Figure: Defining surface loads by using close contours

In case of line junctions, more than one line/edge has to be selected to make clear the continuity of the requested closed contour.

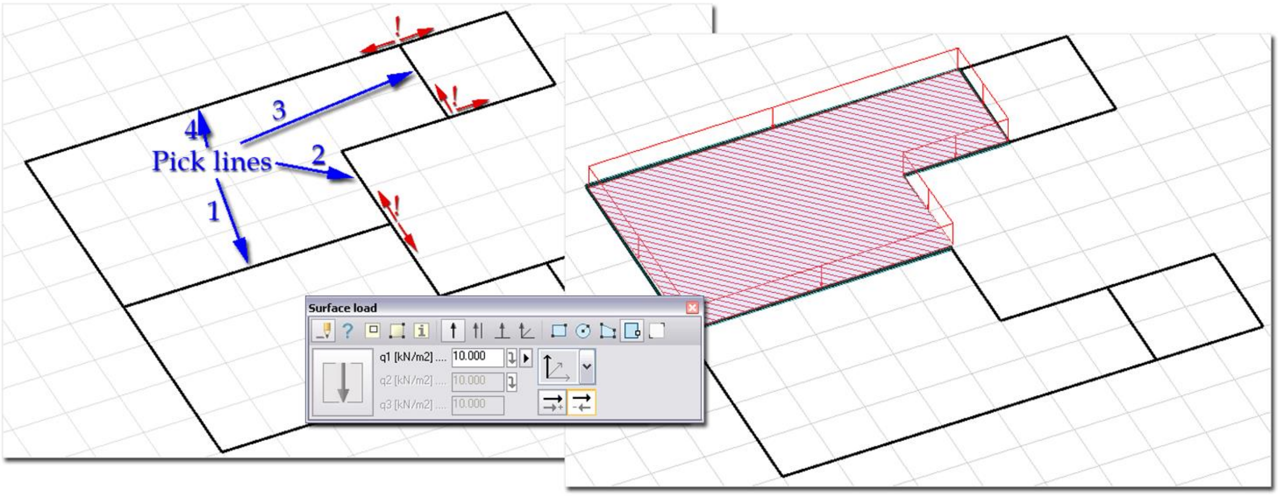


Figure: Selection of more lines to define the right path for the closed shape

"Holes" in Surface Loads

Holes and cuttings can be added to surface loads with the *Hole* tool. The following geometries can be used for holes:

The steps of a hole definition:

3. Select the surface load with mouse-clicking. Clicking a region places the **UCS** into the region plane, so giving hole coordinates needs only X and Y values from the UCS origin.
4. Define the geometry of the hole or cutting with one of the following geometry modes:

- Rectangular
- Circular
- Polygonal
- Pick lines

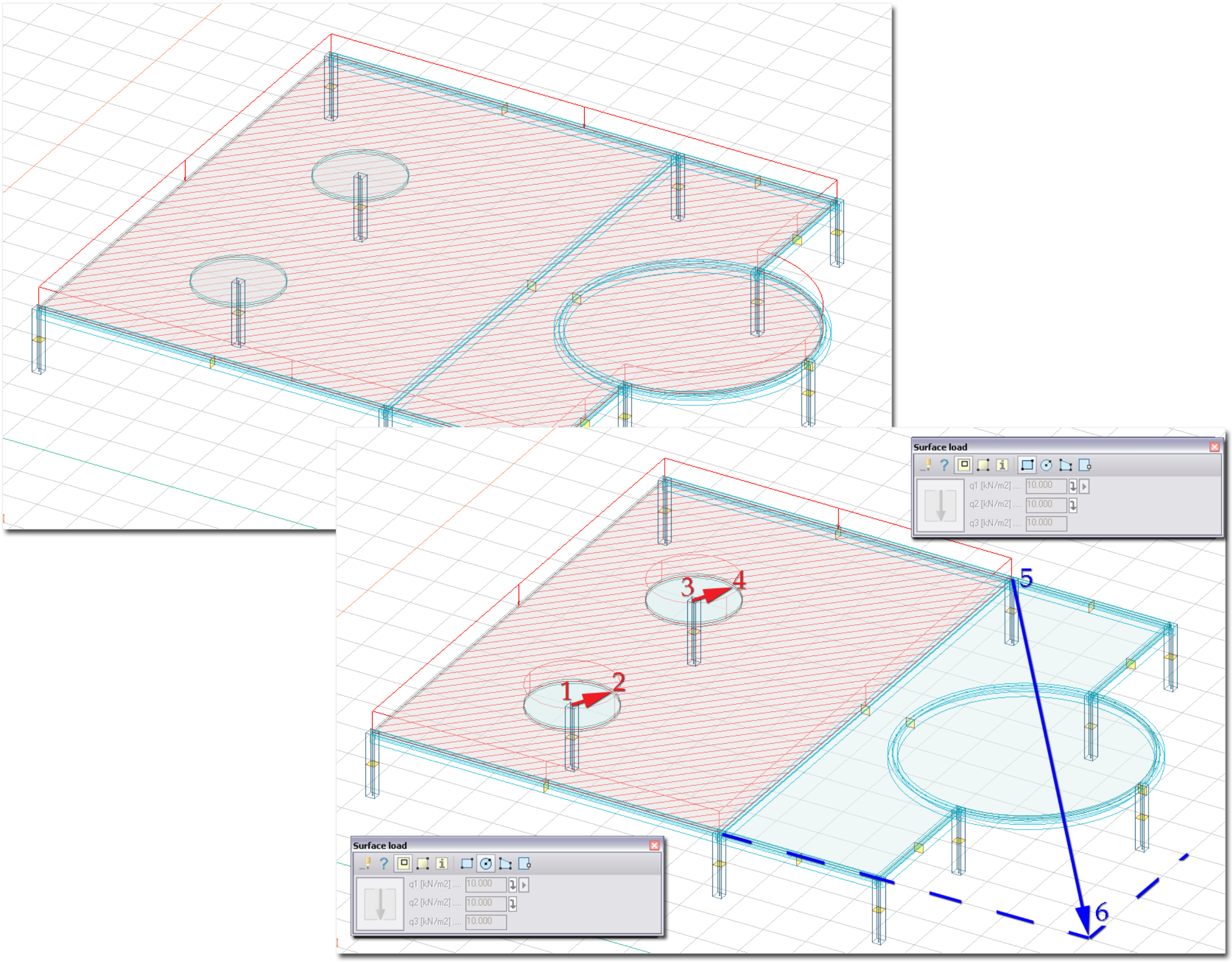


Figure: Editing a surface load (previously defined by Pick existing region) with the Hole tools

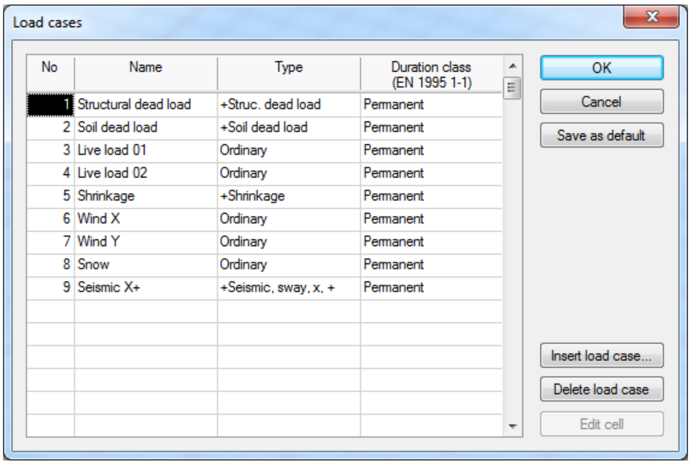
Holes can be easily copy inside a surface load with the **Copy command** (*Modify* menu). To set the distances/new positions, the **UCS** has to be in the plane of the host region(s).

Load Cases

Loads in FEM-Design are represented with *Load cases*. A Load case has a name and physically contains one or more load objects.

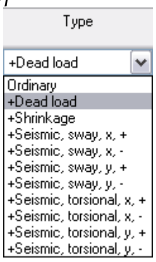
Special loads (invisible loads) like dead load, shrinkage and seismic effect can only defined with Load cases. For timber elements **duration classes** can be set with the *Load cases* command.

Although a Load case can be assigned later to loads, the first recommended step of load definition is the load case classification (load case list).



Definition steps

1. Start *Load cases* command from Loads tabmenu.
2. Define Load case name in the *Name* column.
3. Set the *Type* of the new load case:



Type	Modules where available
Ordinary	
+Dead load	
+Shrinkage	
+Seismic…	

Table: Load case types by FEM-Design modules

- "*Ordinary*" means that no additional effect will be assigned to the load case
- "*+Structural Dead load*" means that the weight of all structural elements, "*+Soil Dead load*" means the weight of soil, which is calculated automatically, will be assigned to the load case as an invisible load. So, a "Dead load"-type load case contains automatic dead-load and can contain further manually defined loads (e.g. "dead load" of the non-load-bearing (non-core) parts of a composite slab).

  Because "+ Structural Dead load" and "+Soil Dead load" type includes the dead load (calculated from the geometry and material) of all structural elements, define only one dead load type load case in one project. If you define more than one "+ Structural Dead load" and "+Soil Dead load" type load cases never group them in the same **Load combination**!

  In all modules automatic dead load works in the global Z direction, except for the *Wall* and *Plane Strain* modules, where the dead load direction is parallel with the global Y.

- "*+Shrinkage*" means that free **shrinkage strain** behavior will be considered as a load effect in concrete design. The shrinkage strain value can be set at the material properties of concrete structural elements.

- "*+Seismic…*" means that sway and torsional effect components calculated from **Seismic calculations** will be considered as a load effect in analysis and design calculations.

4. In case of *Timber design* only, set a load-duration class for a load case according to the regulations of Eurocode 5 (EN 1995-1-1:2004). The load-duration classes are characterized by the effect of constant load acting for a certain period of time in the life of the structure. For a variable action the appropriate class shall be determined on the basis of an estimate of the typical variation of the load with time.
   Actions shall be assigned to one of the load-duration classes given for strength and stiffness calculations.

Load-duration class	Order of accumulated duration of characteristic load	Examples of loading
Permanent	More than 10 years	Dead load
Long-term	6 months – 10 years	Storage
Medium-term	1 week – 6 months	Imposed floor load, snow
Short-term	less than one week	Snow, wind
Instantaneous		Wind, accidental load

Table: Load-duration classes and examples of load-duration assignment (EN 1995-1-1:2004)

⚠ Since climatic loads (snow, wind) vary between countries, the assignment of load-duration classes may be specified in the National annex.

Optional steps:

5. New load case can be inserted to the Load case list in the required row position with *Insert load case*. Just click in the *Name* field you would like to insert the new load case and press the *Insert load case* button. In the pop-up dialog, set the name, the type and the duration class of the new load case.
6. Load cases can be removed from the Load case list with *Delete load case*. Just click a field of the load case you would like to delete and press the *Delete load case* button.
7. The load case list can be set as default for next project by clicking the *Save as default* button.

After finishing the load case definition, a load case can be assigned to a load in two modes: by choosing a case directly from the Loads tabmenu (drop-down list) or in the *Default settings* dialog of the current load command.

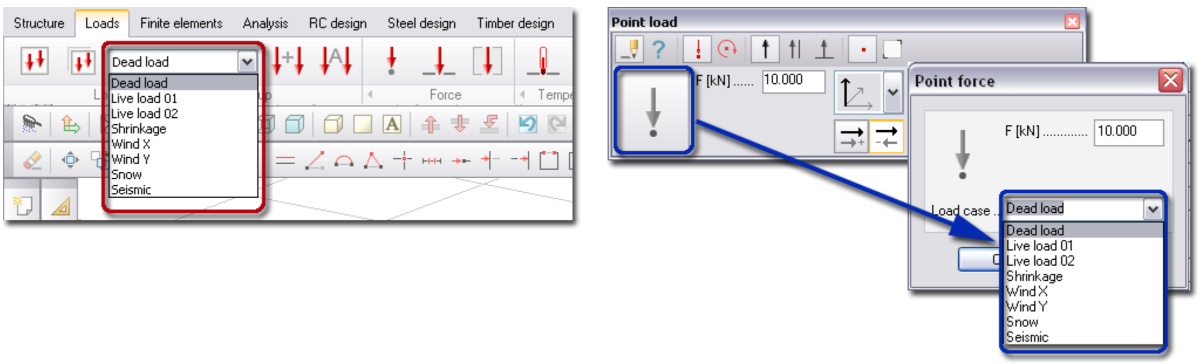


Figure: Load case selection for load objects

A color is assigned to a load case after its definition. That color represents the color appearance of the loads included in a load case. The default load case color is red, but you can set different colors by load cases at the **layer settings**.

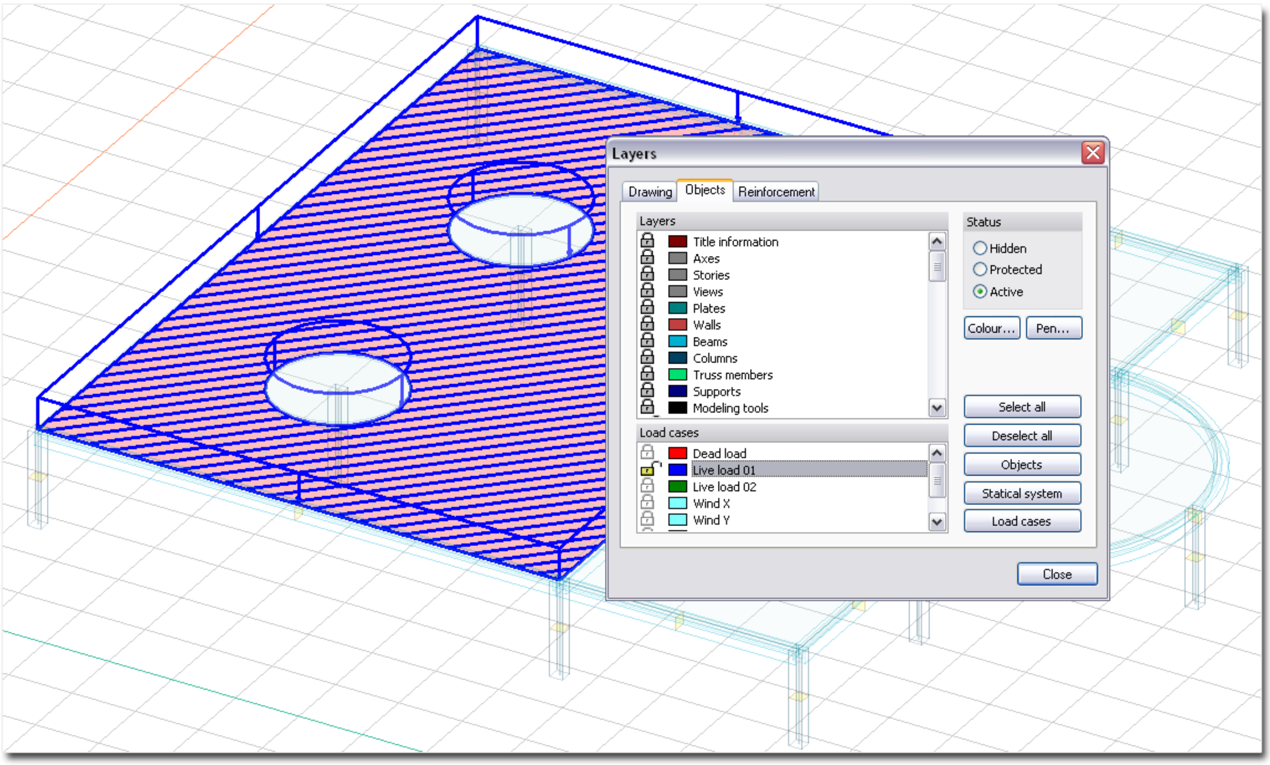


Figure: The color-system of Load cases represents the load appearance

Moving load cases

With defining a **Moving load** special load cases are created, which are displayed with blue text and can not be deleted in the Load cases dialog.

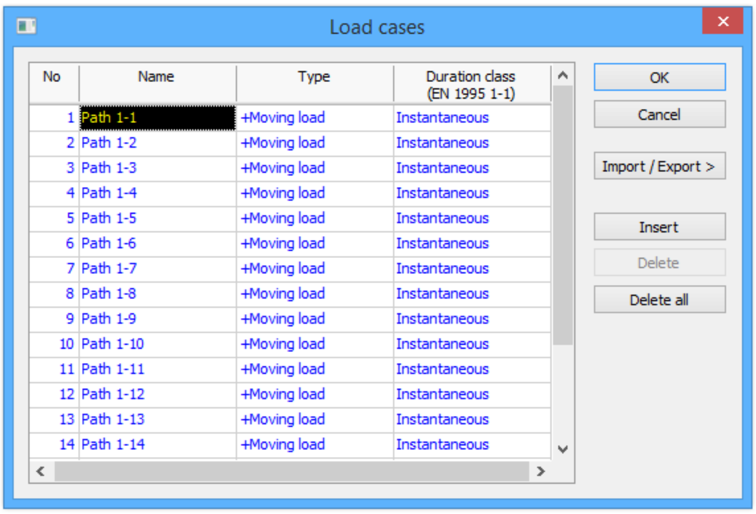


Figure: Moving load cases

Load Definition by Types

The commands for defining loads can be started from the Loads **Tabmenu**.

To show the mostly used load commands and to hide the rarely used ones; you can organize the icons in expanded or compact form.

Expanded mode	Compact mode
	

In compact mode, you see the icon of the last used command. If you click on the command's symbol, the program opens its tool palette. You can reach the other same type load by clicking on the ◢ symbol.

Each command has a **Tool palette** with the customizable load properties (load value/intensity, host load case etc.) and the definition tools of the load geometry and position (direction).

Structural dead load
The weight of the load-bearing structural elements are calculated automatically by the program and is considered in the calculations, if **"+Structural dead load" type load case** is defined and it is added to the load combinations or load groups. So, the so-called "auto dead-load" is calculated from the geometry and material-dependant density value.

The direction of the dead-load is vertical, but the vertical direction differs in the different FEM-Design modules.

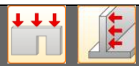
	
Structural Dead-load (gravity) direction	
Global Z direction	Global Y direction

Table: The structural dead-load direction by FEM-Design Modules

Definition steps

1. Start the **Load case command**, and define a "+Structural dead load" type load-case with a name (such as "Dead load"). So, the "auto dead-load" is allowed on load-case level.
2. Define loads (**Point-**, **Line-** and **Surface loads**) under the load-case defined in the previous, 1st step, if additional dead-loads coming from invisible not-load-bearing parts (non-load-bearing layers of composite structures) are requested.
3. Add the load case defined in the 1st step to load combinations/groups depending on the later design.

The auto dead-load is always invisible. Only the additional, manually defined dead-loads can be displayed (as point/line/surface load) by activating the layer of the host load-case.

Soil dead load
The weight of the soil is calculated automatically by the program and is considered in the calculations, if it is added to a **"+Soil dead load" type load case** and then load combinations or load groups. So, the so-called "auto dead-load" is calculated from the geometry and material-dependant density value.

The direction of the dead-load is vertical, but the vertical direction differs in the different FEM-Design modules.

Dead-load (gravity) direction
Global Z direction

Table: The soil dead-load direction by FEM-Design Modules

Definition steps

1. Start the **Load case command**, and define a "+Soil dead load" type load-case with a name (such as "Dead load"). So, the "auto dead-load" is allowed on load-case level.
2. Define loads under the load-case defined in the previous, 1st step.

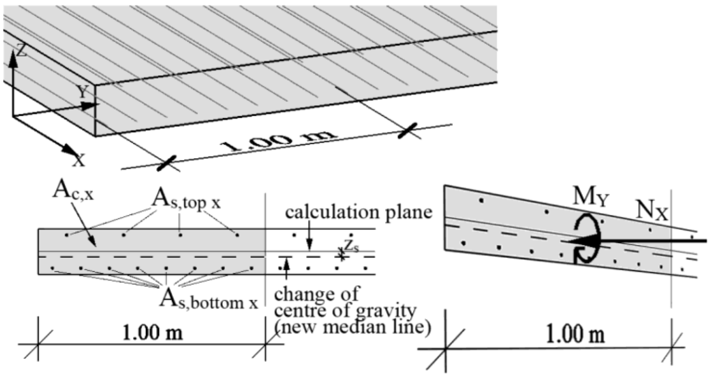
   ⚠ The soil can be loaded only with **Point load**, **Line load**, **Surface load** (not with moments).
3. Add the load case defined in the 1st step to load combinations/groups depending on the later design.

The auto dead-load is always invisible. Only the additional, manually defined dead-loads can be displayed (as point/line/surface load) by activating the layer of the host load-case.

Shrinkage Strain

Free shrinkage strain behavior can be modeled as a load effect in reinforced concrete design and check.

Shrinkage effect is calculated as a specific rotation. The following formulas give the example of calculation for reinforced concrete slabs:



$$N_X = E_c A_c \varepsilon_{cs}$$

The specific normal force causing the given shrinkage value (ε_{cs} [‰]) in the concrete zone of the section is (here in X direction).

$$z_s = \frac{n \cdot S_s}{A_c + n \cdot A_s}$$

The position change of centre of gravity considering reinforcement bars (here X-direction; see dashed line), where $n = \frac{E_s}{E_c}$ and S_s is the statical moment of (here) X-directional bars around the Y axis of the calculation plane.

$$M_Y = N_X \cdot z_s$$

The moment around the Y axis of the calculation plane from N_x because of the position change of centre of gravity.

$\kappa_Y = \dfrac{M_Y}{E_c I_Y}$ The specific rotation (curvature) from M_y for 1 meter wide section, where $I_Y = I_{c,Y} + n \cdot I_{s,Y} - z_s^2 (A_c + n \cdot A_s)$.

Definition steps

1. Set the shrinkage strain (ε_{cs}) value at the material properties of concrete structural elements.

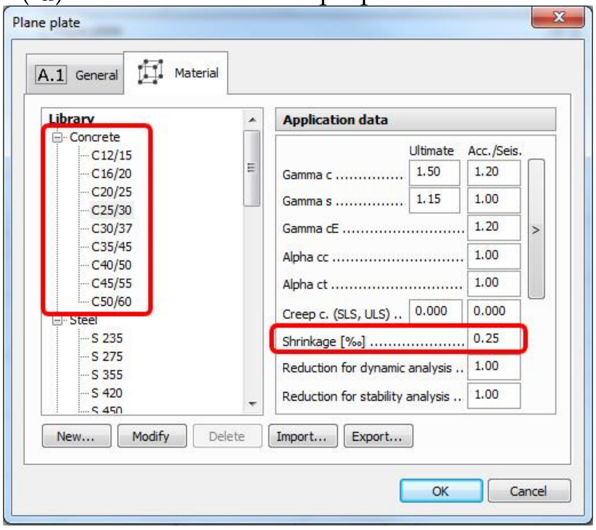


Figure: Shrinkage value of concrete materials

⚠ Shrinkage effect is taken into consideration in RC design and check of concrete elements, which have **applied reinforcement**.

2. Start the **Load case command**, and define one "+Shrinkage" type load-case with a name (such as "Shrinkage effect"). That means, the automatically calculated shrinkage effect is allowed on load-case level.

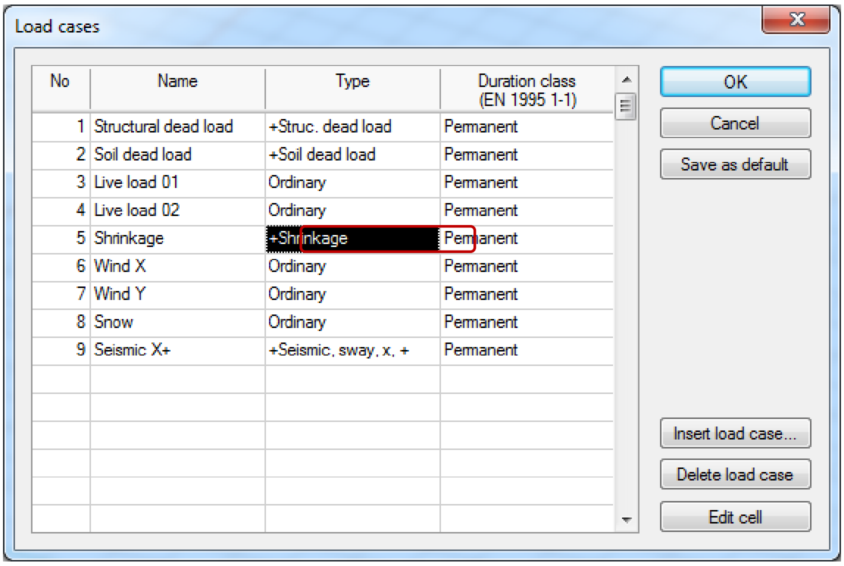


Figure: Shrinkage effect added to a load case

3. Add the load case defined in the 2nd step to load combinations/groups depending on the later RC design.

Point Load

Depending on the current FEM-Design module, point force and/or moment can be defined easily by giving just the load insertion point and direction. The load properties and definition tools are available on the tool palette of the *Point load*.

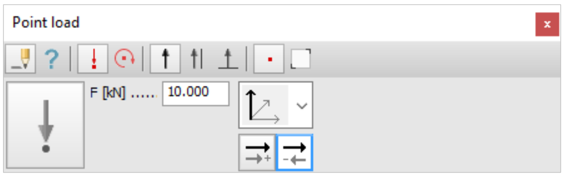


Figure: Tool palette of Point load

Definition steps

1. Select the load case from the load case drop-down list from Loads tabmenu, which you would like to add the new point load(s) to.

   If you have not defined a load case yet, you can also define and set it in the *Default settings dialog* by giving a load case name. Here you can type any comment, which will be displayed next to the force.

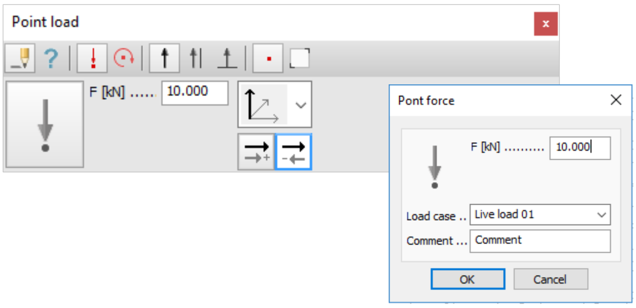


Figure: Load case definition

   The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Choose the load type: *Force* or *Moment* (FEM-Design module dependent).
3. Set the load value according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*.
4. Define the **direction** of the force or moment axis (FEM-Design module dependent).
5. Choose a method for the insertion point definition (**geometry**).
6. Place the point load in the model view based on the chosen geometry method (5th step).

Optional steps:

7. Modify the load properties (the host load case, load value) with the *Properties* tool of the *Point load* tool palette.
8. Modify the load direction with the editing tools (**Editing Loads**).
9. Set the display settings of the load (**Load Display Settings**).

Deviation load (Automatic)

Deviation load (as point loads) can be placed automatically in the centre of gravity points of plates located on stories.

⚠ Automatic deviation load generation needs **storey-system** in the project!

Definition steps

1. Entering to "Load" mode, the program displays the so-called "main directions" of the current project. These main X' and Y' load directions are used by the automatic deviation and **wind load** definition. By default, the main X and Y load directions are parallel with the ***Global*** X and Y axis directions. Set the required load directions for the deviation (and wind) loads with the **Change direction** command (*Modify* menu).

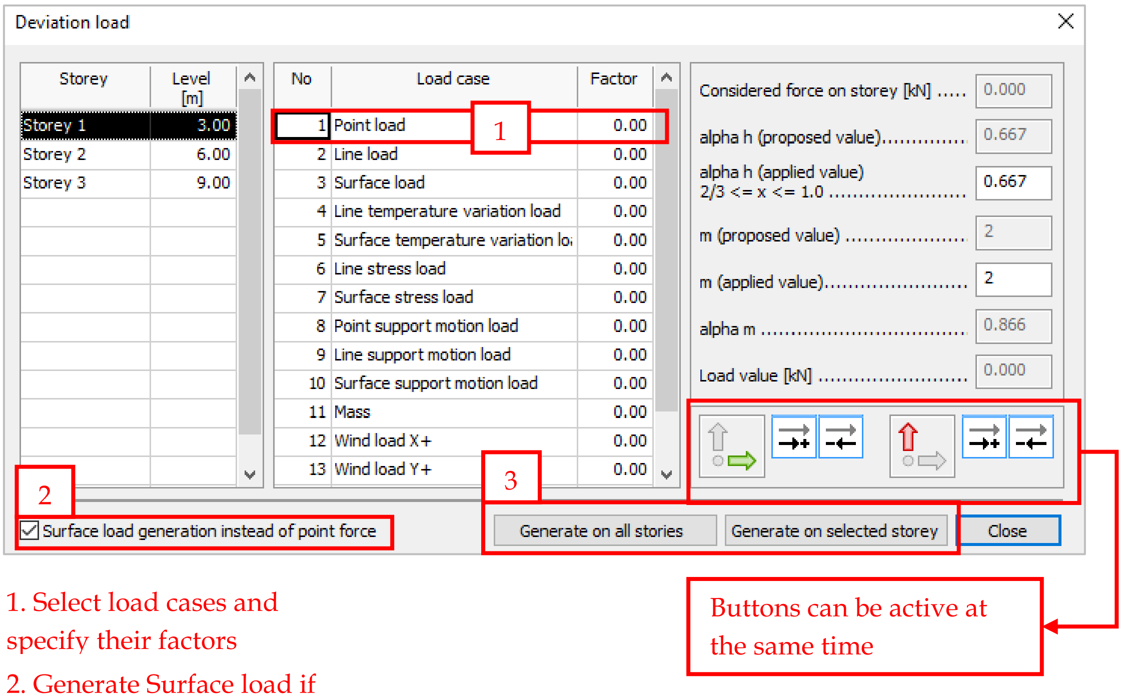
   ⚠ The symbol of the main load directions are stored on "*Main directions*" **Object layers**.

2. Start the *Deviation load* command from *Loads* tabmenu.

   ⚠ The generated point loads will be inserted into load cases called "*Deviation X (Generated by [Factor x Load case1] + [Factor x Load case2])*" and "*Deviation Y (Generated by [Factor x Load case1] + [Factor x Load case2])*". If there are no load cases with the same name in the project, the program automatically defines them for the automatic deviation load components.

3. Select the story you would like to add deviation load to. Choose the required direction X and/or Y and positive and/or negative based on the main directions set in the 1st step. Define the required *alpha h* and *m* parameters according to the current standard. The program calculates the point deviation load value in the selected storey from the selected load cases, the storey height and the number of vertical members contributing to the horizontal force. Clicking *Generate* creates deviation load in the centre of gravity of the selected storey slab with the given load direction. Checking the *Surface load generation instead of point force* option generates surface load instead of point force.
4. Repeat the previous step to generate deviation load in the other storey slabs. Click *Close* to exit from the dialog box after finishing the load definitions.

   For faster and more comfortable load definition deviation loads can be generated simultaneously for X and Y direction. Furthermore, with the *"Generate All"* button it is possible to create deviation loads for all storeys.



1. Select load cases and specify their factors

2. Generate Surface load if it's needed

3. Generate deviation load on all stories or only the selected one

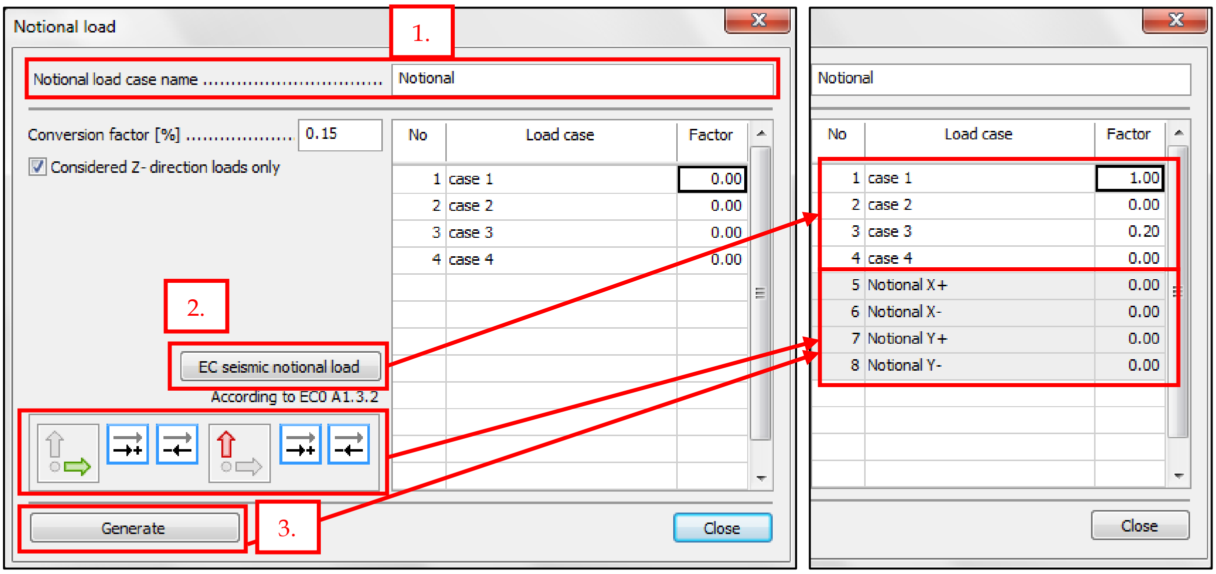
⚠ Generate buttons re-defines all existing Deviation load with the default properties. That may delete previously defined Deviation Load cases.

Notional load
Notional load function generates horizontal load from non-horizontal loads using a conversion factor.

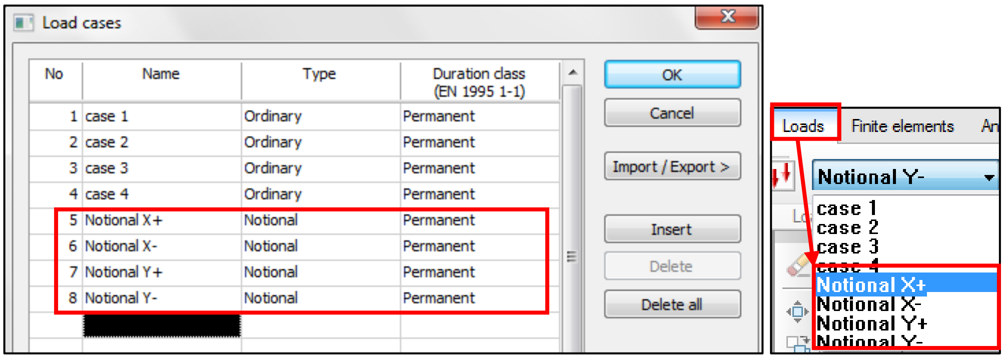
⚠ Notional load generation needs user-defined **Load Cases** in the project!

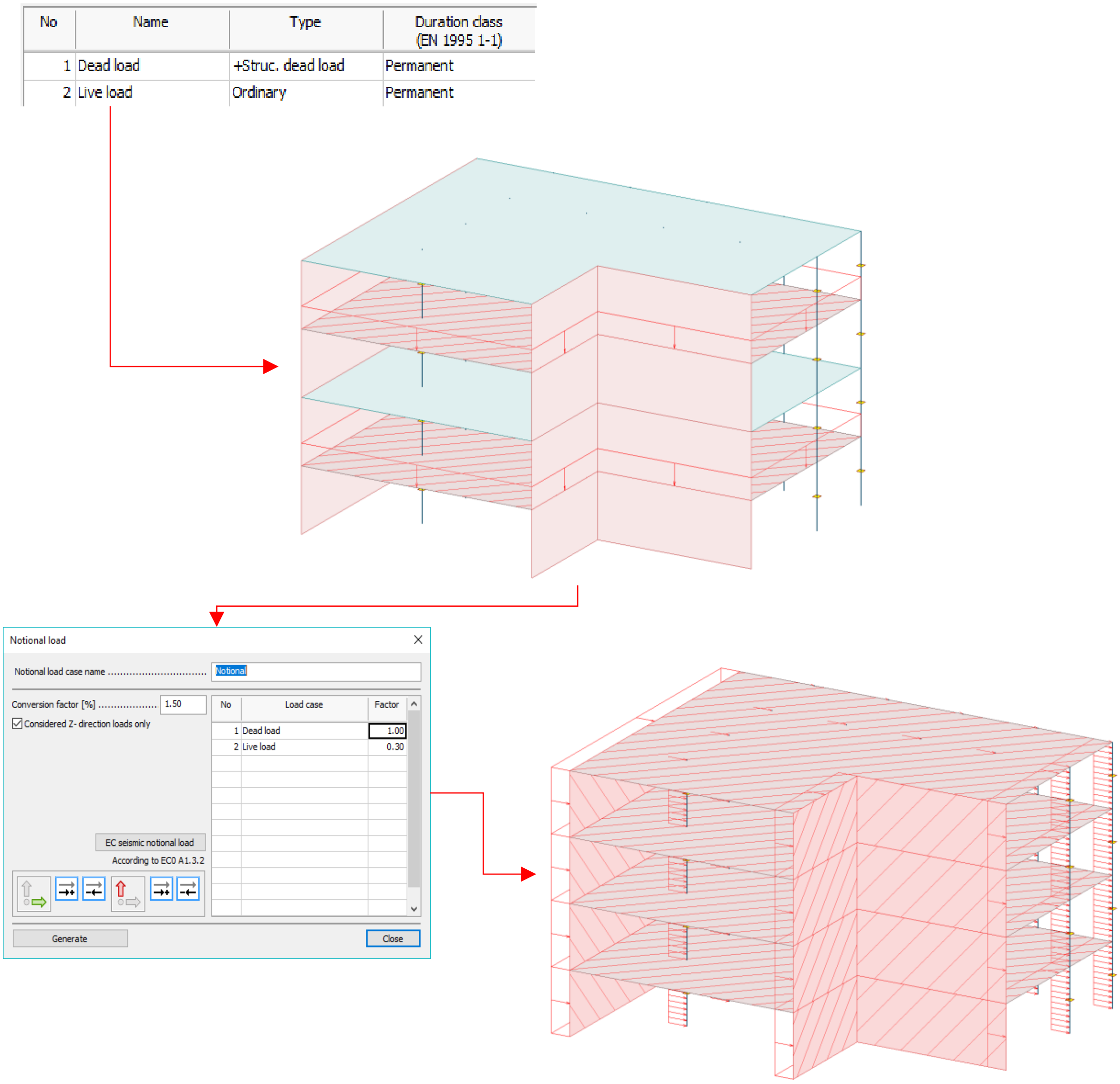
Definition steps

1. Start the  *Notional load* command from *Loads/Macro* tabmenu.
2. In the top of the dialog there is a textbox containing an automatically generated name for the notional load case, which can be changed arbitrarily.
3. Click *EC seismic notional load* button, it indicates that Load case factor will be set according to parameters of Load groups the Load case belongs to. It uses EN 1990-1-1 A1.3.2 DK NA: $A_d = 1.5\%(\sum G_{k,j}" + "\sum_{i\geq 1}\psi_{2,i}Q_{j,i})$
4. Click *Generate* to create as many *Notional Load cases*, as many directions are chosen above the *Generate* button.



The generated *Notional Load Cases* appear in the *Load Cases dialog* and they can be selected from *Loads tab/Current load case* drop-down list.





Line Load

Depending on the current FEM-Design module, constant or variable line force/moment can be defined easily by giving the load shape, direction and position. The load properties and definition tools are available on the tool palette of the *Line load*.

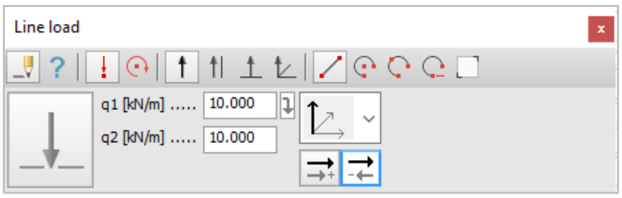


Figure: Tool palette of Line load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new line load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name. Here you can type any comment, which will be displayed next to the force.

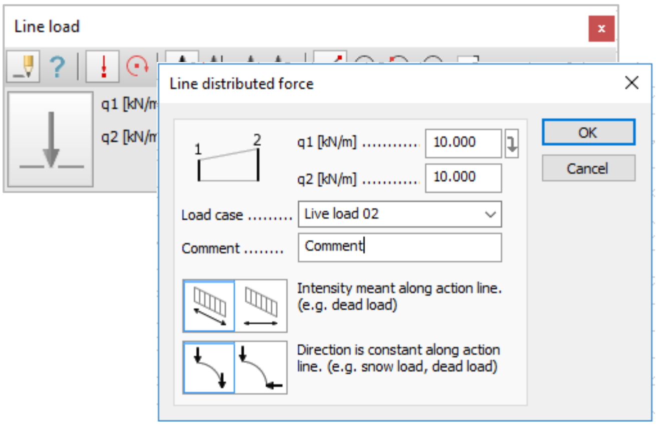


Figure: Load case definition

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Choose the load type: *Force* or *Moment* (FEM-Design module dependent).
3. Set the intensity values in the start (*q1*) and end point (*q2*) of the new load according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*. If you inactivate the arrow ( ) next to the *q1* field, you can type intensity value in *q2* different from *q1*.
4. Define the **direction** of the force or moment axis (FEM-Design module dependent).
5. Choose a method for the action line definition (**geometry**).
6. Set the "intensity mode", that means the connection between applied intensity and the action line at *Default settings*, if the new load will be in skew position (not vertical or horizontal):
   - **"Intensity meant along action line"**
     The intensity will be meant along the action line, so the resultant will be calculated from the intensity and the length of the action line. Apply this option for dead load-type loads.

   - **"Intensity meant perpendicular to direction of load"**
     The intensity will be meant perpendicular to the load direction, so the resultant will be calculated from the intensity and the length of the horizontal projection of the action line. Apply this option for example to model snow load.

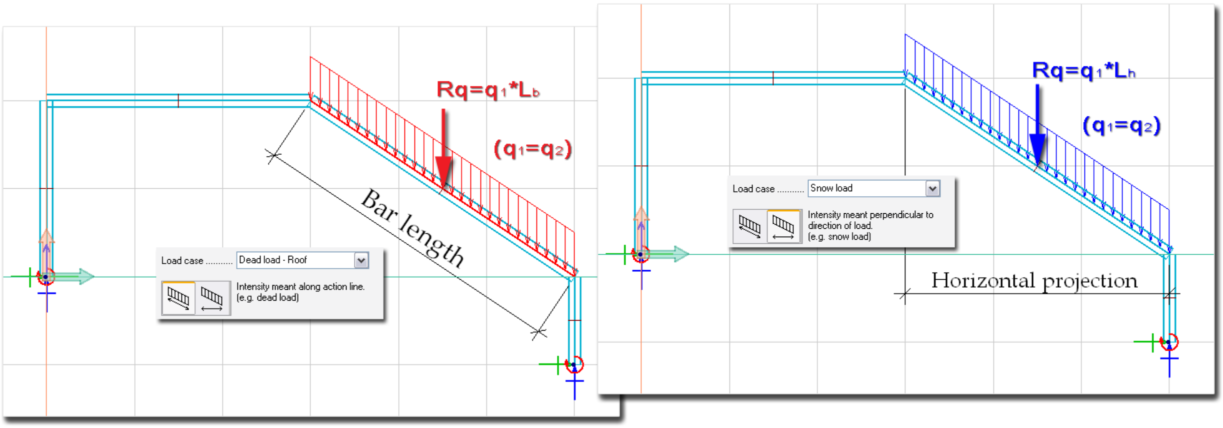


Figure: Intensity difference at skew bars

7. Set the "direction mode", that means the connection between applied load direction and the action line at *Default settings*, if the new load will be a curved line load:
   - **"Direction is constant along action line"**
     The load direction set in the 4^{th} step will be constant along the action line.

   - **"Direction varies along action line"**
     The load direction will vary along the action line, so the characteristic direction set in the 4^{th} step will be taken into consideration in the middle point of the curved action line.

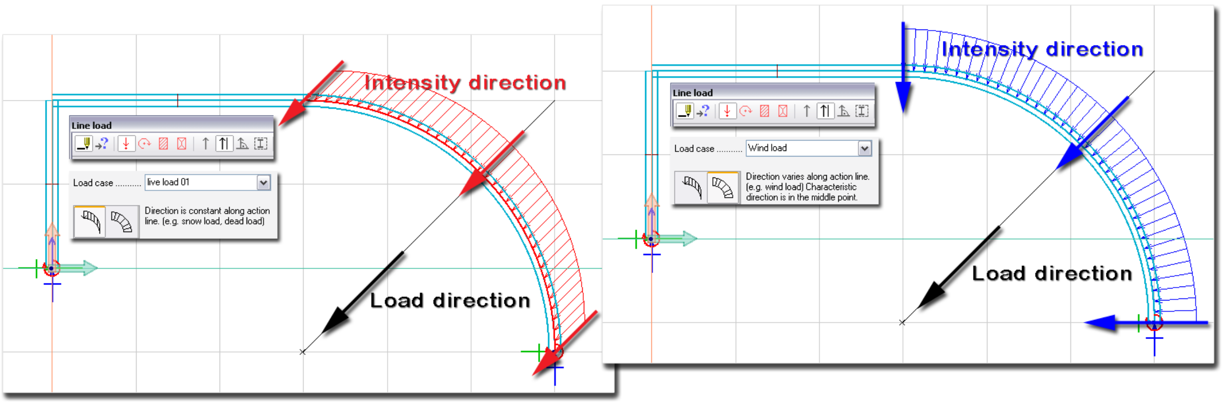


Figure: Intensity difference at skew bars

8. Place the line load in the model view based on the chosen geometry method (5^{th} step). With the special tool called *Object's local system* ( ) you can add line load directly to a selected bar element in a direction set parallel with the one of the local system axis of the bar element. So, with this tool, you can merge 4^{th}, 5^{th} and 8^{th} steps to one step, if the required load direction is equal to the direction of the assigned bar element's local axis.

Optional steps:

9. Modify the load properties (the host load case, intensity values, "intensity" and "direction mode") with the *Properties* tool of the *Line load* tool palette.
10. Modify the load direction with the editing tools (**Editing Loads**).
11. Set the display settings of the load (**Load Display Settings**).

Wind Load (Automatic) – Regular building

Two types of wind load can be generate in FEM-Design, **Regular building** and **Generic building** wind load.

Regular building wind load generation needs **storey-system** in the project!

Definition steps

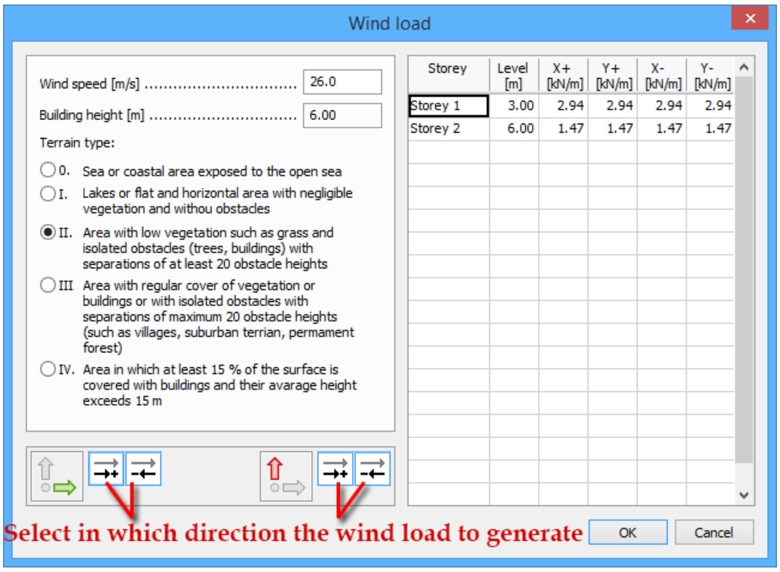
1. Entering to "Load" mode, the program displays the so-called "main directions" of the current project. These main X' and Y' load directions are used by the automatic wind and **deviation load** definition. By default, the main X and Y load directions are parallel with the ***Global*** X and Y axis directions. Set the required load directions for the wind (and deviation) loads with the **Change direction** command (*Modify* menu).

   The symbol of the main load directions are stored on "*Main directions*" ***Object layers***.

2. Start the *Wind load* command from *Loads* tabmenu.

   The generated wind line loads will be inserted in a load cases called "*Wind load X*" and "*Wind load Y*". If there are no same named load cases in the project, the program automatically defines them for the automatic wind load components.

3. Set the *wind speed* and *terrain type* in the settings dialog according to the building environment. The program calculates the applied wind load values by the levels of the current project's storey-system from the given parameters and the model size set at the **Storey** command (horizontal building size and story level heights). The calculated intensity values are displayed in table format (right part of the dialog).
4. Set the wind load X and Y directions same (+) or converse (-) with/to the **main X and Y direction system**.
5. Clicking *OK* generates line loads in the external edges of plates that are located on stories surfaces/region parts.



While generating Wind Loads all existing Wind Loads are redefined! That may delete previously defined Wind Load cases.

Surface Load

In all FEM-Design module constant or variable *Surface load* (force) can be defined in the oridinary way by giving the load region, direction and load intensity positions, if necessary. In the 3D modules, there is an additional option to define Soil/hydrostatic pressure-like load by giving the load region, direction, the soil/water level, the intensity at the soil/water level and the increment of the intensity.

The load properties and definition tools are available on the tool palette of the Surface load.

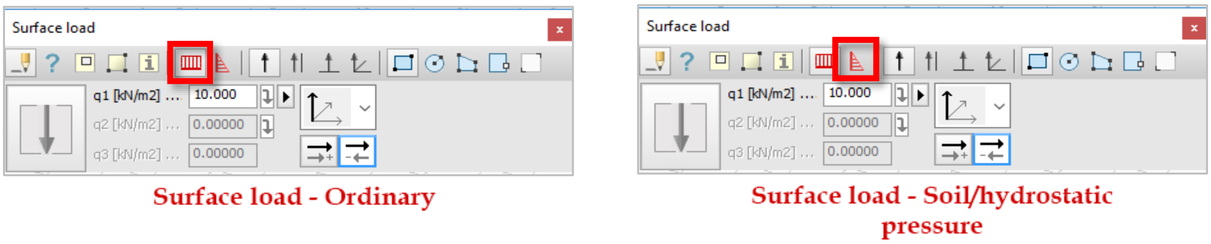


Figure: Tool palette of Surface load

Ordinary, Constant Surface Load

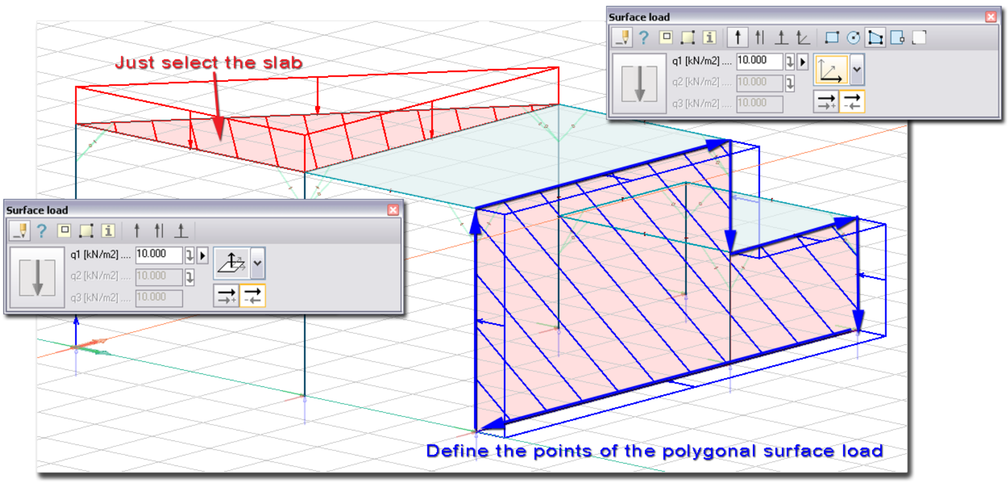


Figure: Constant surface load on a slab and on a wall

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new surface load(s) to.

   If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name. Here you can type any comment, which will be displayed next to the force.

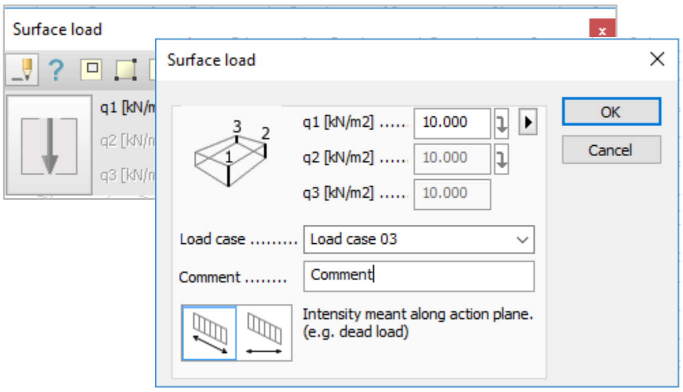


Figure: Load case definition

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Set the intensity value in the *q1* field according to the current unit (that can be set at *Settings > Units*) in the tool palette or at *Default settings*. You may also choose intensity from the **predefined load values** from the list appears after clicking on the button.
3. Define the load **direction** (FEM-Design module dependent).
4. Choose a method for the action surface definition (**geometry**).
5. In the 3D modules only, you can set the "intensity mode", that means the connection between applied intensity and the action plane at *Default settings*, if the new load will be in skew position (not vertical or horizontal):
   - **"Intensity meant along action plane"**
     The intensity will be meant along the action plane, so the resultant will be calculated from the intensity and the area of the action surface. Apply this option for dead load-type loads.

   - **"Intensity meant perpendicular to direction of load"**
     The intensity will be meant perpendicular to the load direction, so the resultant will be calculated from the intensity and the area of the horizontal projection of the action surface. Apply this option for example to model snow load.

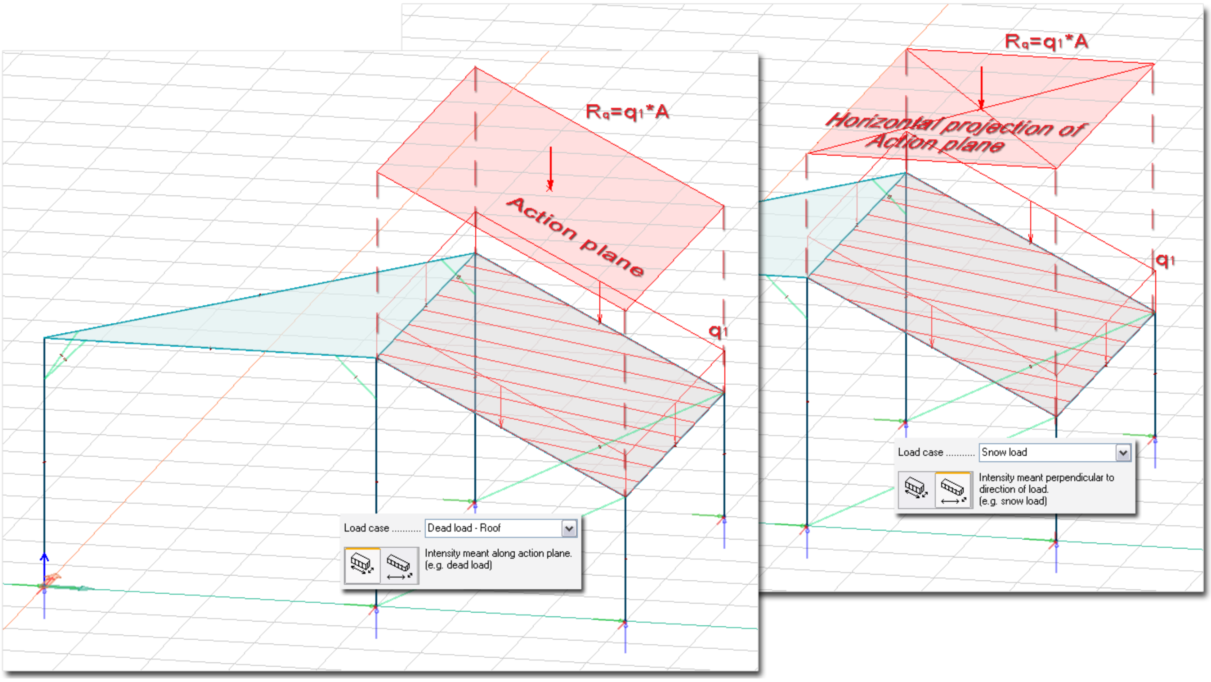


Figure: Intensity difference at skew bars

6. Place the surface load in the model view based on the chosen geometry method (4th step).
   With the special tool called *Object's local system* ( ) you can add surface load directly to a selected planar object in a direction set parallel with the one of the local system axis of the planar object. So, with this tool, you can merge 3rd, 4th and 6th steps to one step, if the required load direction is equal to the direction of the assigned planar object's local axis.

Ordinary, Variable Surface Load

Linearly variable surface loads can be defined with *Surface load* command. Three intensity values define the linearly variable load as a "skew plane".

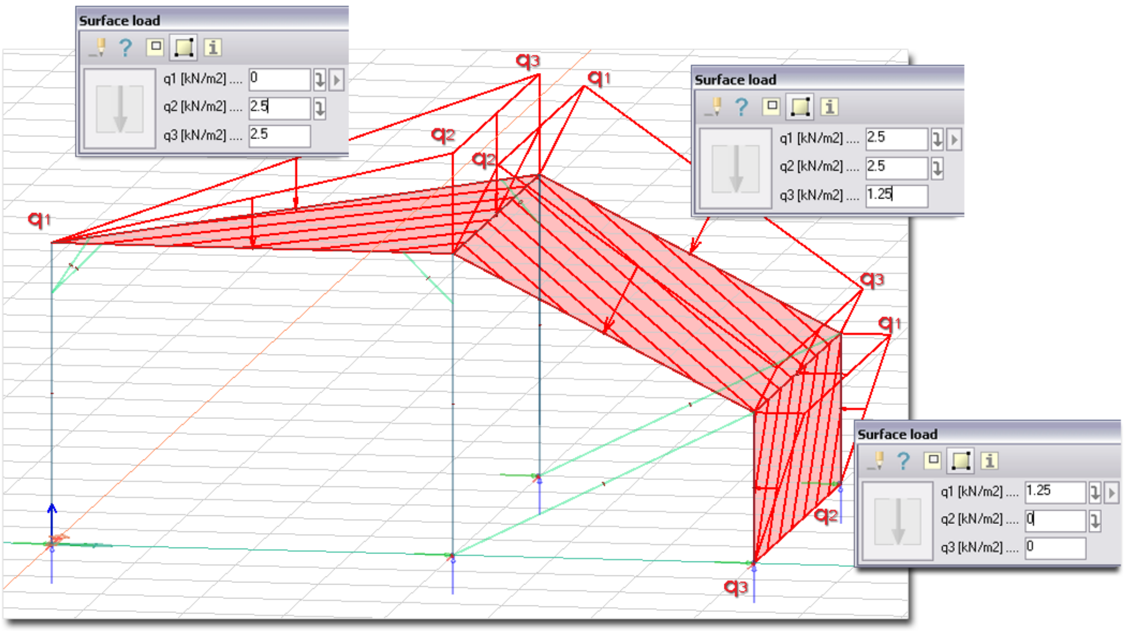


Figure: Variable surface loads

Definition steps

1. Define first a constant surface load with the required direction and geometry as written at **Constant Surface Load**.
2. Use the *Variable intensity* tool of the *Surface load* command.
3. Set the new intensity values (*q1*, *q2* and *q3*). If you inactivate the arrows ( ) next to the value fields, you can type different values by the *q* fields.
4. Select the constant surface load that you would like to modify being variable.
5. Give the position of the intensity values. You may click outside the surface load's action plane too.

Optional steps for constant/variable surface loads

1. Place a hole/opening in a surface load. Use the **Hole** tool of the *Surface load*.
2. Modify the load properties (the host load case, intensity values, "intensity mode") with the *Properties* tool of the *Surface load* tool palette.
3. Modify the load direction with the editing tools (**Editing Loads**).
4. Information (the resultant force value and the position of its action point) about a selected surface load can be inquired with the *Info* tool. A drawing point can be placed in the resultant's action point, if needed.

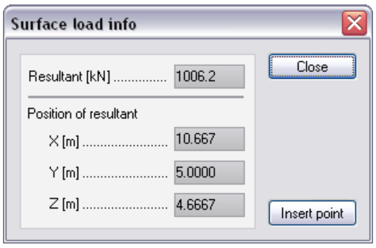
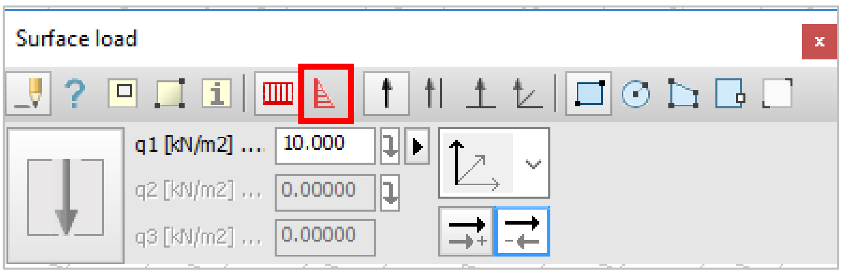


Figure: The resultant force value and position calculated by Info tool

5. Set the display settings of the load (**Load Display Settings**).

Soil/hydrostatic pressure

Soil/Hydrostatic pressure-like load can be defined in a convenient way by choosing "*Soil/Hydrostatic pressure*" option.



The load intensities are automatically calculated based on three user-defined values:

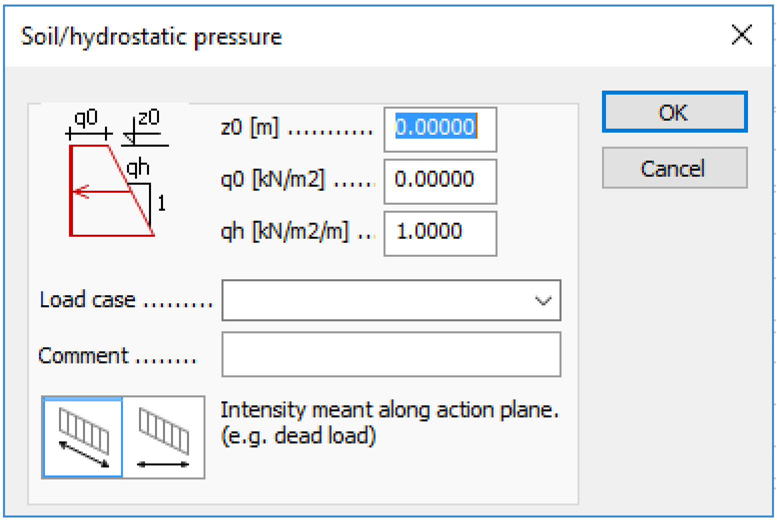
- z0: the surface level of soil/water (on the global Z axis)
- q0: the intensity of load at the surface level
- qh: the increment of load intensity (along the global Z axis)

The intensity of the load at a certain point of an element is calculated with the formula below:

$$q = q0 + (z0 - z) \cdot qh \qquad (2.1.)$$

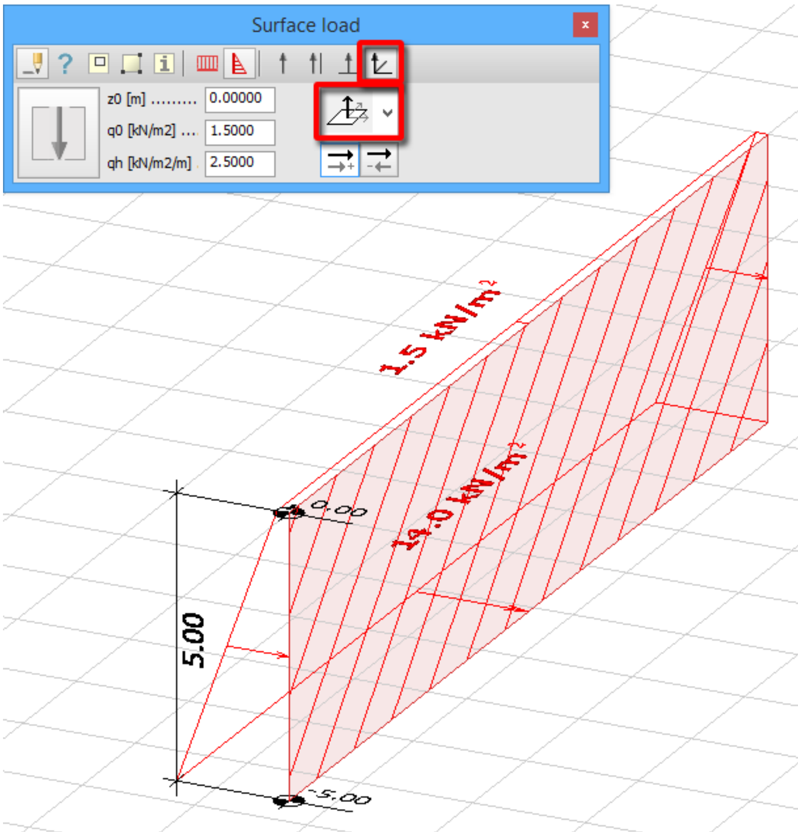
q: the intensity of the load at the particular point

z: the Z coordinate of the point

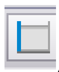


The direction of the soil/hydrostatic pressure load can be determined either in the same way as in case of the ordinary surface load.

For the fastest and easiest load definition it is recommended to use the **Object's local system** option with the "*Direction is parallel with the local z' axis*" mode.



Wind load (Automatic) – Generic building
Generic building's wind load (as surface loads) can be placed automatically on covers.

1. ***Set the type*** of the cover according to EC: External wall , Flat roof , Lean-to  or Ridge-roof :

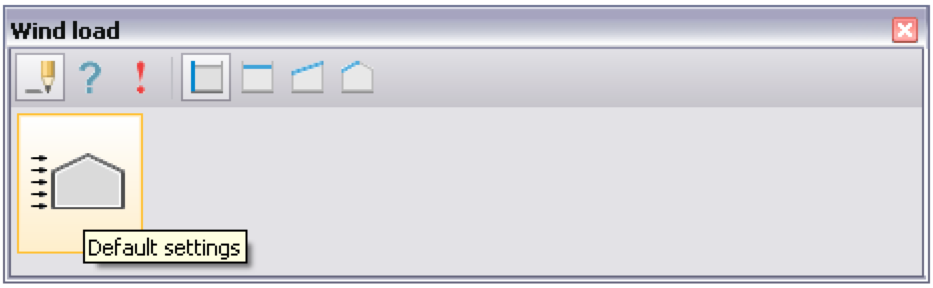


Figure: Generic building wind load dialog

In all four cases a **green arrow plays a crucial role in correct load generation**. In Lean-to and Ridge-roof the arrows are generated automatically by FD, but for flat roof and external wall, the user has to define it. Details can be found in the sections below.

The importance of the green arrows can be understood at the bottom-left figure in the four settings windows of the four cover type, showing the wind direction terminolgy of EC. In the standard, if the deviation of wind speed is in the range of +45° and -45°, it is taken as 0°. In the range of 45°-135°, it's taken to be 90°, between 135°-225°, to be 180°, and in the last interval to be 270°. By this assumption, no other direction can be imagined as the four cases in the window below, which shows the Default Settings (the button shown in the figure above) for external wall first, and the others next.

External wall

After having chosen , this is what you get in Default settings:

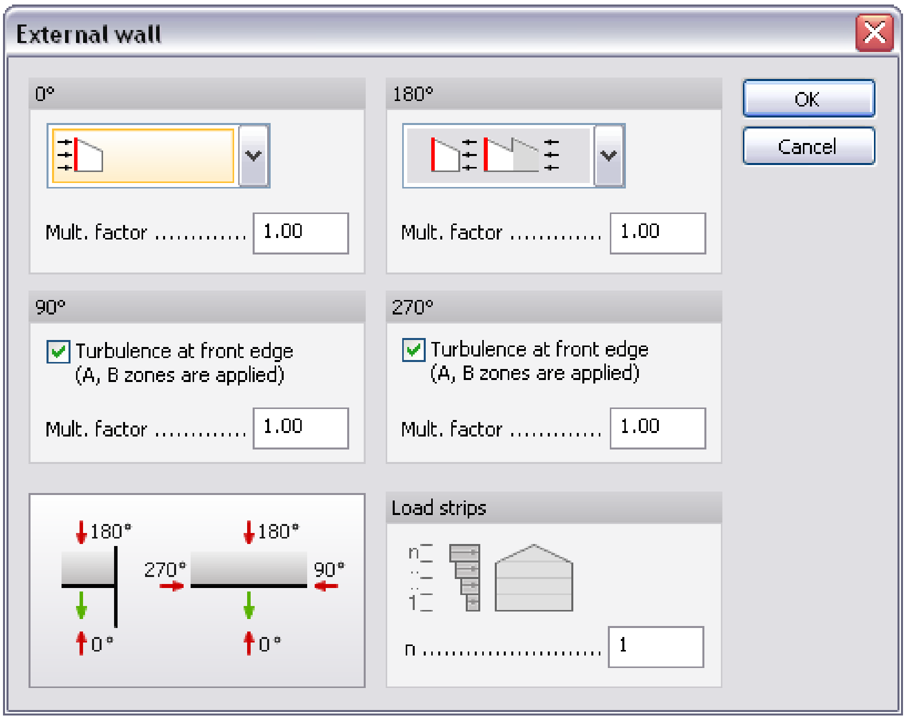
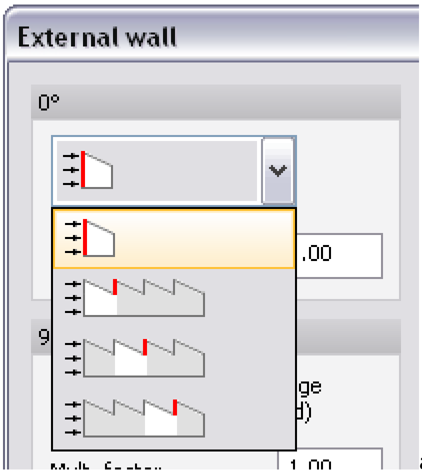


Figure: External wall dialog

The figure at bottom-left shows the direction terminology. For 0°, you can scroll down the typical cases mentioned by EC:



and for 180°

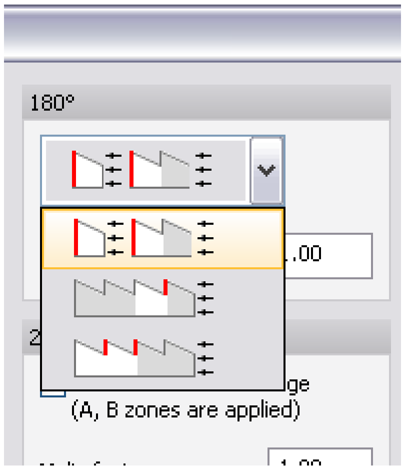


Figure: External wall – setting multiplication factor for predefined wall positions

By choosing the one valid for the current cover, the multiplication factor will be adjusted to the right value. For more sophisticated cases you have the possibility to define the multiplication factor manually.

If the wind direction is in the plane of the wall, some more highly loaded zones (A and B, see EC 1-4 for details) are applied by EC at the leading edge of the wall for taking the effect of turbulence into account. Followed by zone C, a less loaded domain. If no turbulence can be imagined at the leading edge, you can cancel out the creation of zone A and B (The legend in bottom left helps in understanding the directions.):

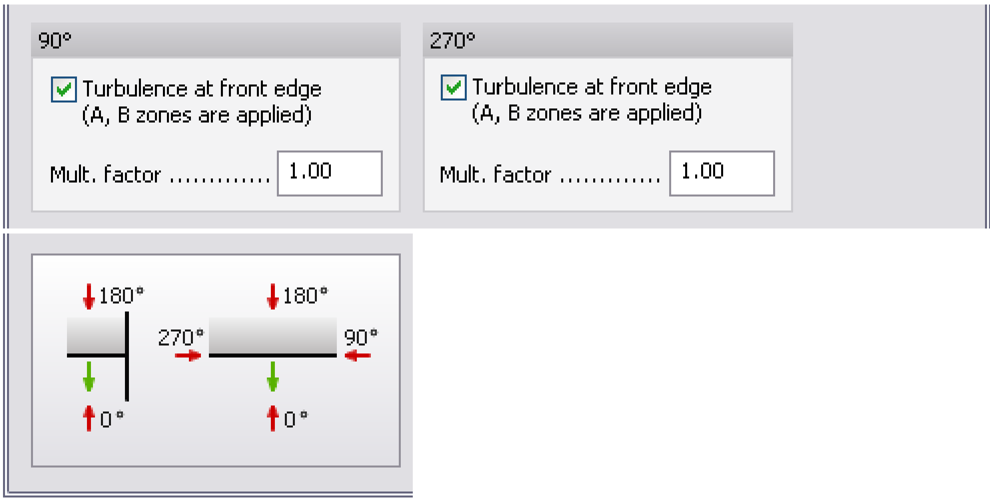


Figure: Wind load direction terminology

EC allows the division of the load domain into horizontal stripes, so you may take into account the height dependence of wind velocity by the application of several horizontal stripes:

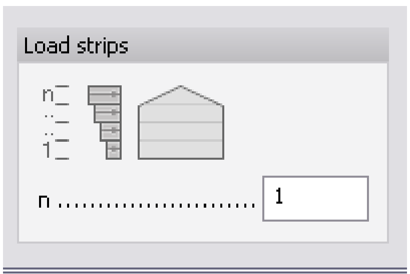


Figure: Wind load vertical distribution

In case of external wall, you have to select the cover by clicking on it, and than select a point at the external side of the building. The green label, and also the arrow will appear, but the arrow will be perpendicular to the plane of the cover. If you have a composed building like in the case below, you can unify several wall covers by pushing Ctrl, and selecting the members:

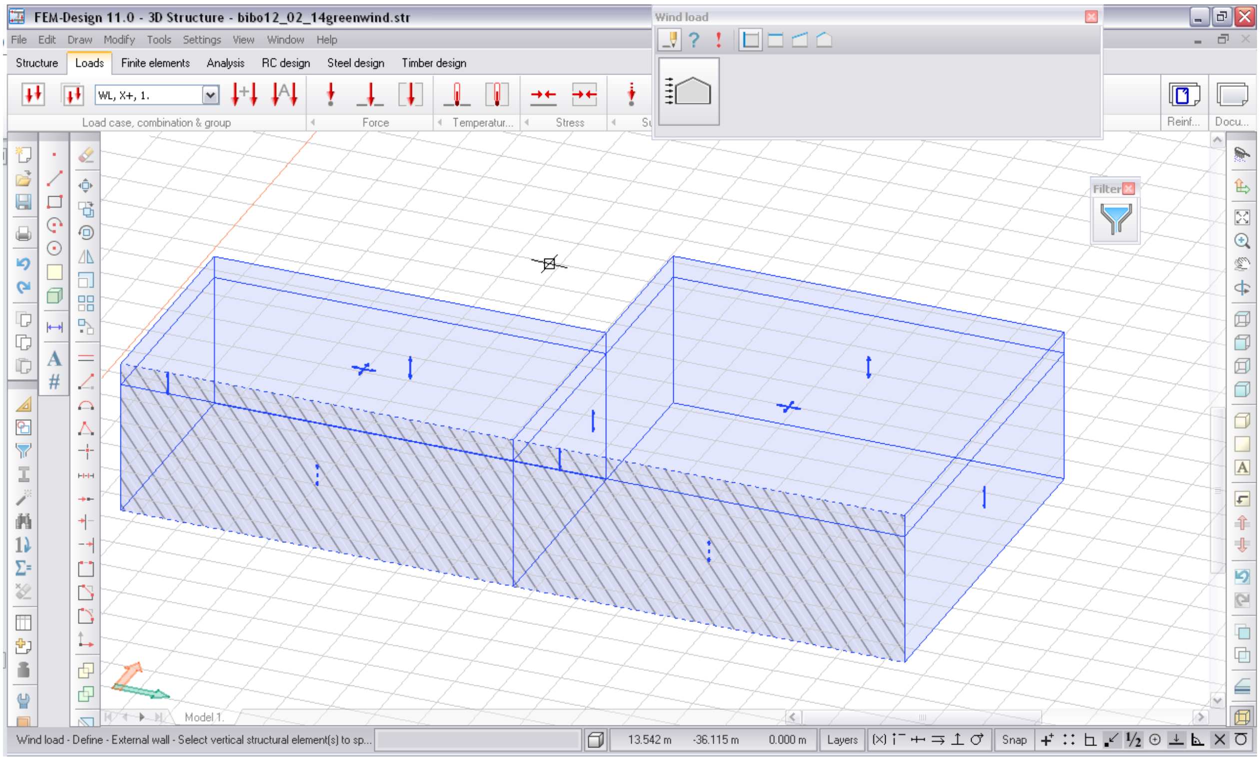


Figure: Selecting External wall

If you have selected one element, you can immediately define the green arrow by picking an external point, in case of more elements you have to end the selection by 'Enter', and then point an external point outside the walls:

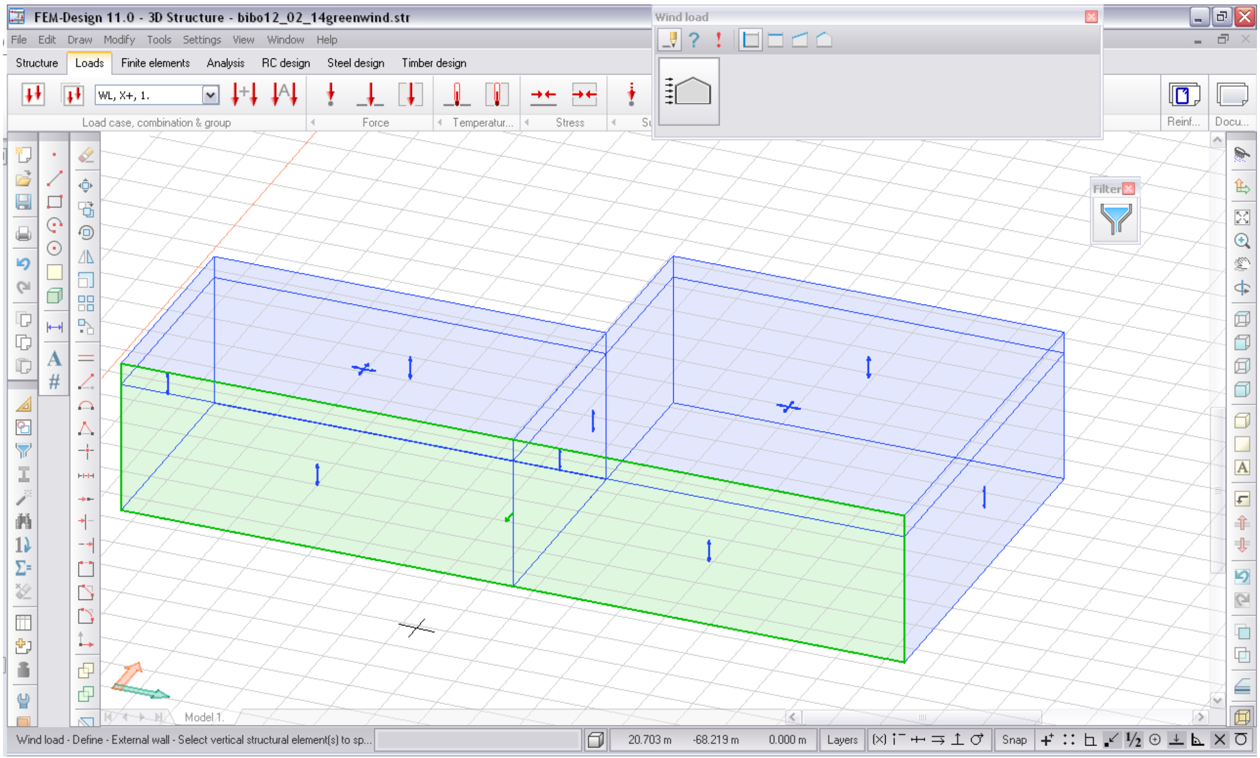


Figure: Defining wall's external point

The wind load generated according to EC will consider the unified green rectangle as one surface.

⚠ **After classified all the walls, verify all arrows' pointing out from the building in order to obtain well directed wind loads on the walls.**

Flat roof

In case of Flat roof , the following appears:

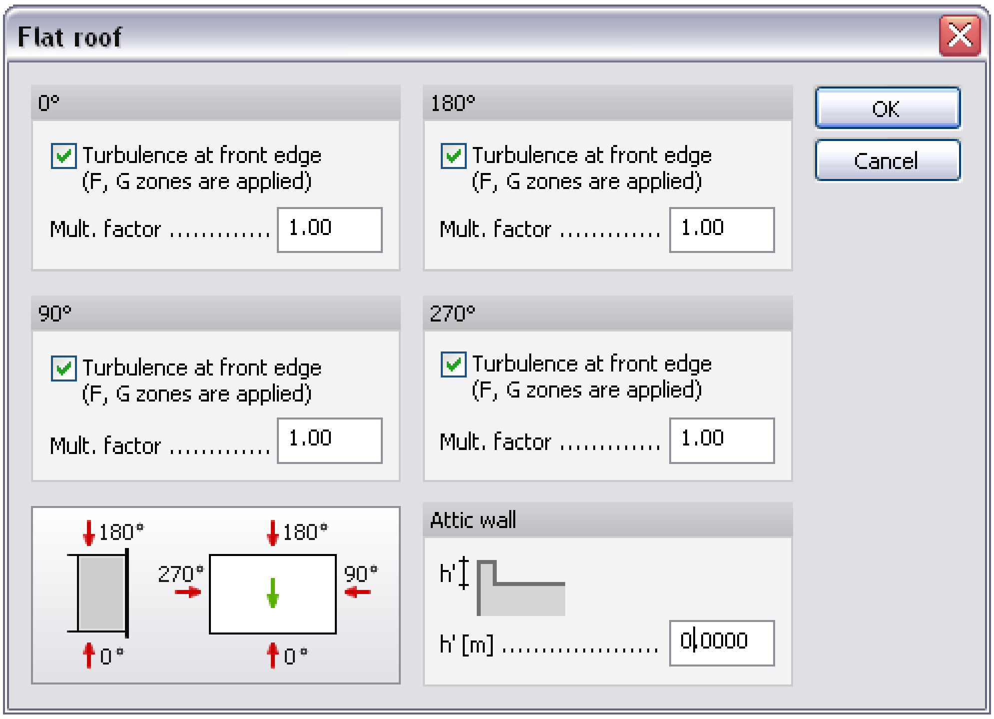


Figure: Flat roof dialog

In the bottom left, the figure shows the notation of the directions with respect to the green arrow. The user can switch on or off the creation of the turbulence zones like in the case of **External wall**, and also can change the multiplication factors manually.

If an attic wall is at the edge of the cover, the height is required, as it has influence on the values of the load.

After choosing the settings, you have to click on the cover, and then, you have to give the main direction for wind generation:

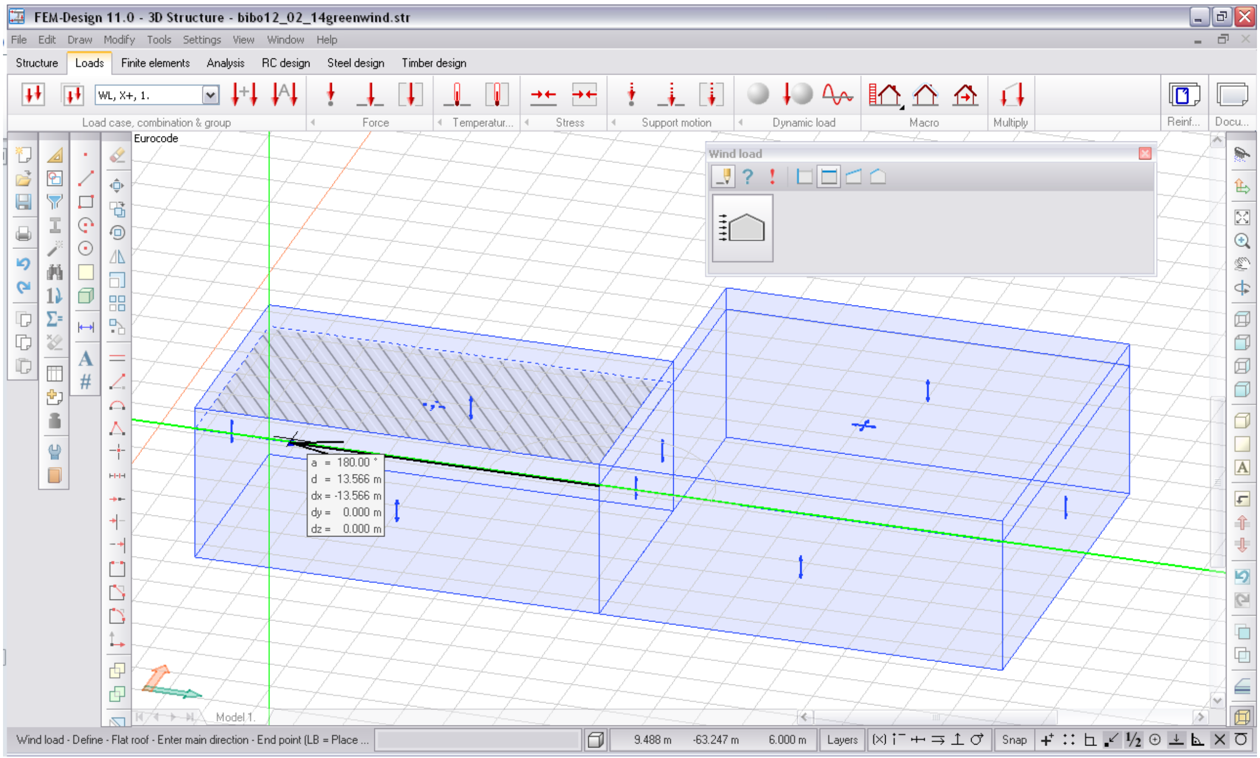


Figure: Defining the main direction for wind generation

After the definition of the direction a green label will appear with an arrow in the middle of the cover. If possible, you should choose a main direction of the building that coincides with axis X, for obtaining the correct distribution in the different wind load cases.

⚠ **An incorrectly defined direction can lead to false wind load distribution.**

You can modify its direction in Modify>Change direction. (If the label or the arrow is missing after creation, try another graphic engine in Settings>All>Environment>General>Graphic Engine)

Lean-to

In case of Lean-to , the dialog window is as follows:

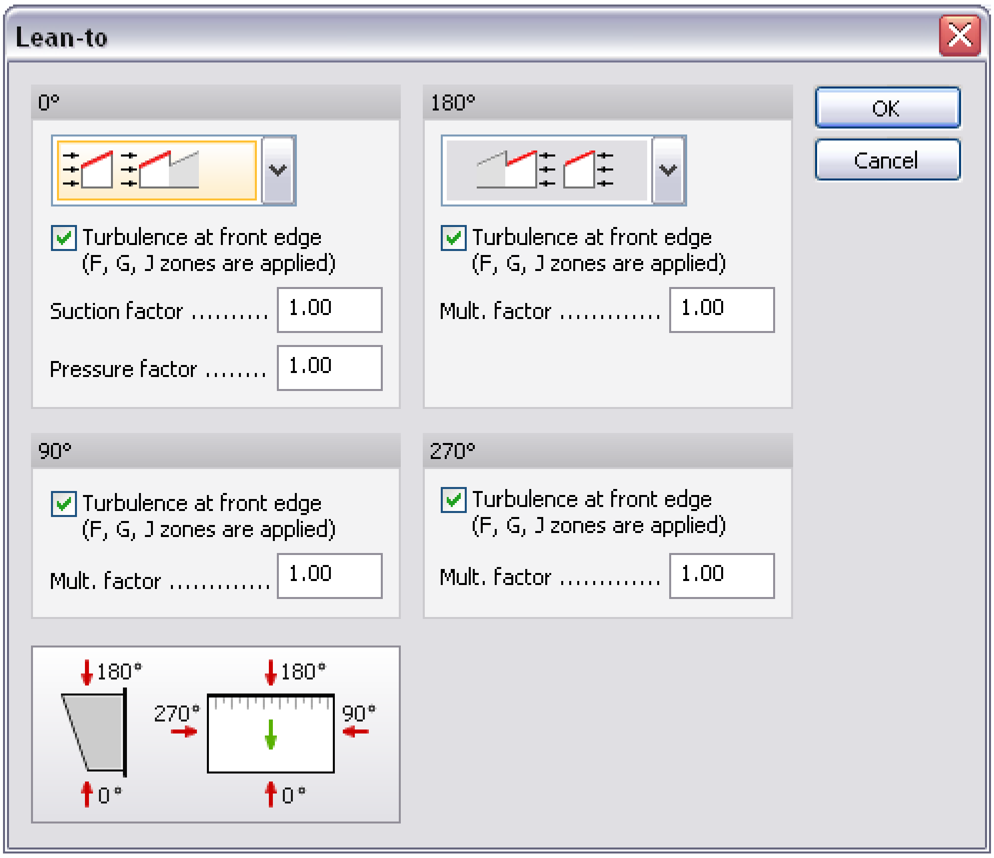
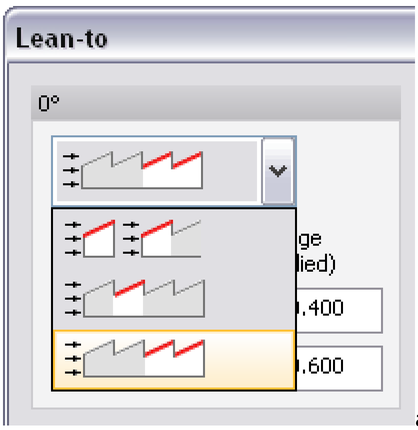


Figure: Lean-to dialog

Likely to former cases, the notation of EC appears at bottom-left corner. If you have a cover that is not parallel with the load coordinate-system, the wind load directions will fit the cover's direction. The different factors can be set eiter by scrolling down the symbol:



and

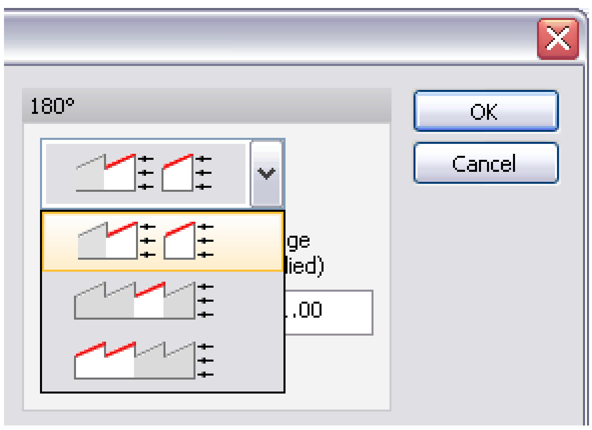


Figure: Lean-to – setting multiplication factor for predefined wall positions

or, by setting them manually.

If required, turbulence zones can be switched off:

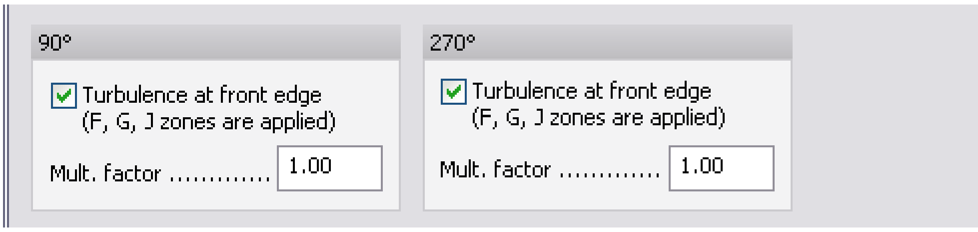


Figure: Turbulence zone application

Ridge roof

The pop up window in case of Ridge roof:

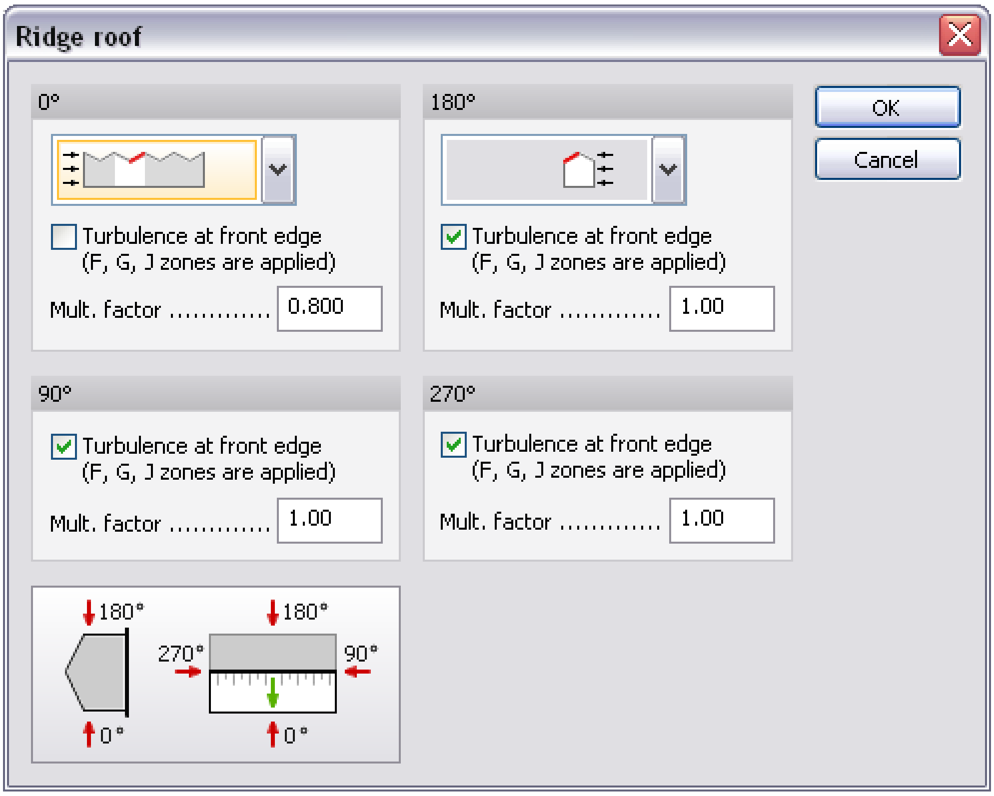
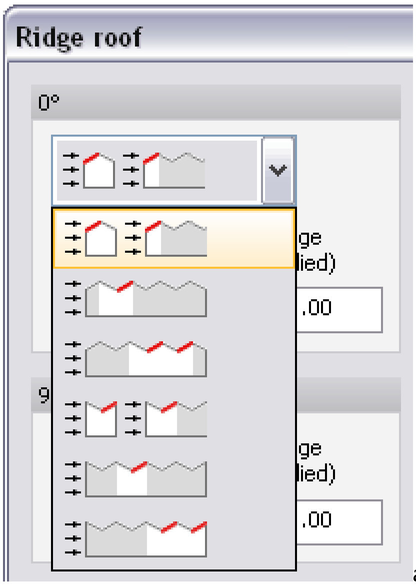


Figure: Ridge roof dialog

Likely to former cases, the notation of EC appears at bottom-left corner. If you have a cover that is not parallel with the load coordinate-system, the wind load directions will fit the cover's direction. The different factors can be set eiter by srcolling down the symbol:



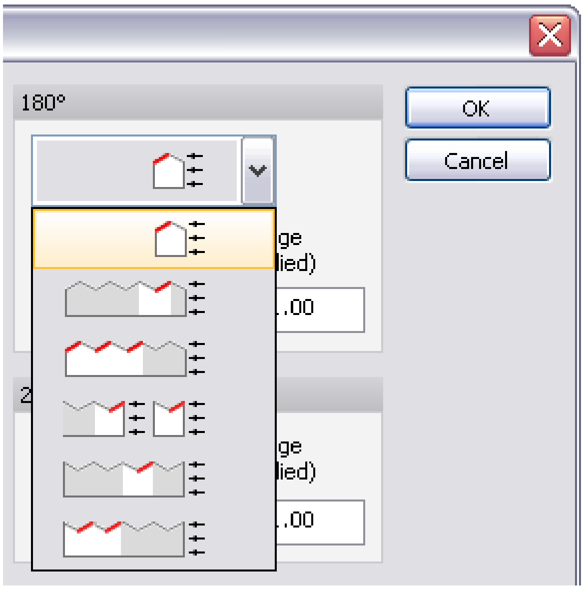
and


Figure: Ridge roof – setting multiplication factor for predefined wall positions

or, by setting them manually.

If required, turbulence zones can be switched off.

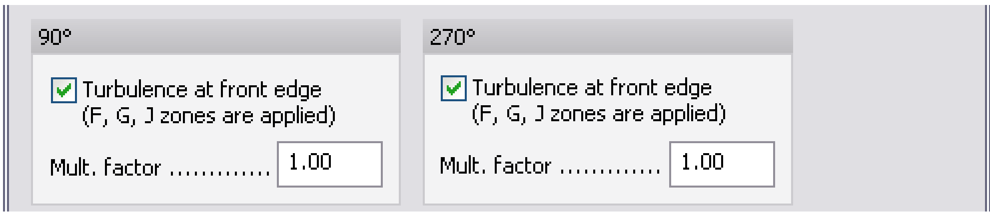


Figure: Turbulence zone application

2. ***Generate*** the wind load

When clicking on the symbol , the wind generation tool window appears:

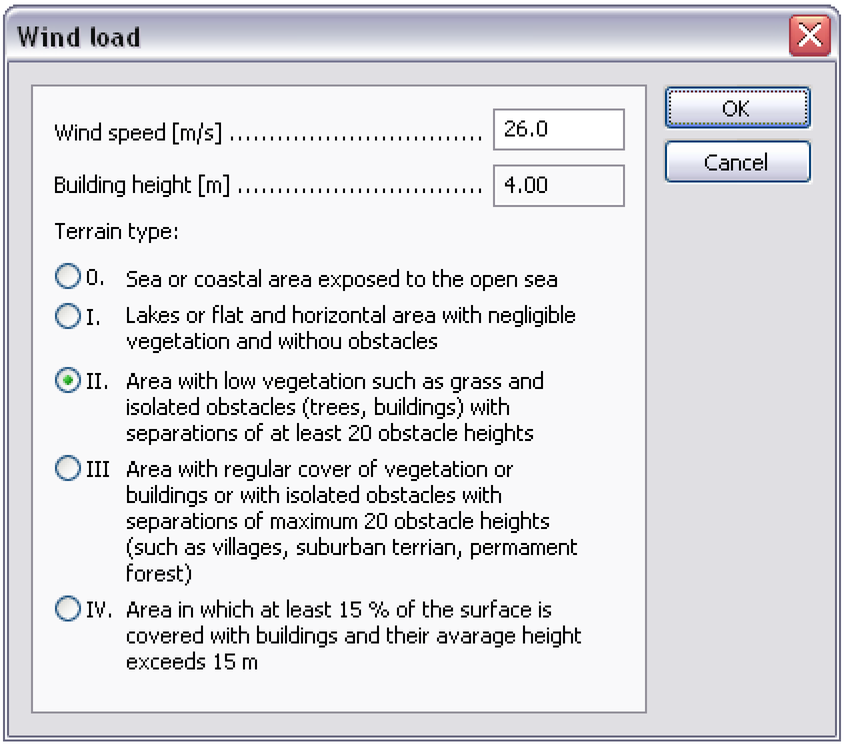


Figure: Wind load generation dialog

In this window you have to classify the building according to EC, and give the basic wind velocity for load generation.

After clicking OK, the wind loads are generated in 16 different load cases:

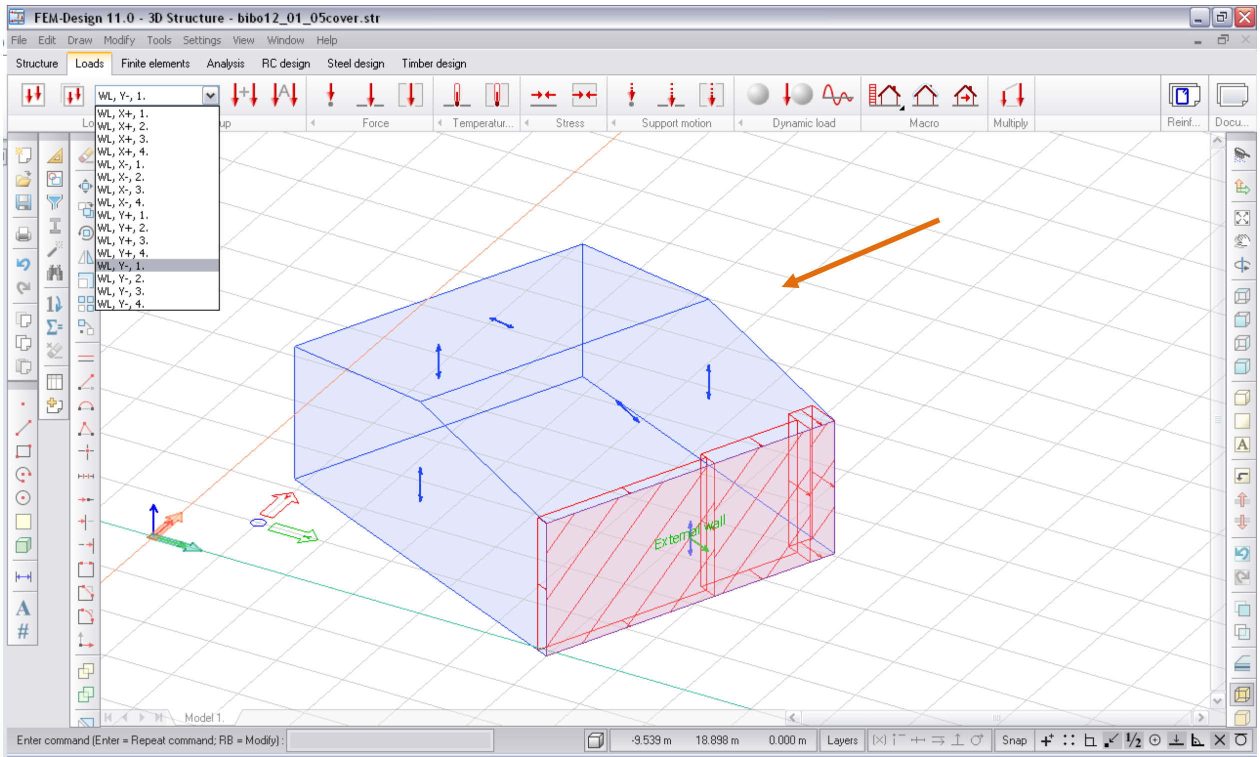


Figure: Automatically generated wind load (WL,Y-,1)

(For proportional displaying of loads click: Settings>All>Display>Load>Display proportionally) In the example above, WL Y-, 1. is active which corresponds to a wind marked by the orange arrow.

It is parallel with the loaded wall, and as it is in the range of 45°-135°, according to the load's coordinate system, it is stored in WL, Y-, 1. For each (four) directions, generally 4 different cases can be imagined in EC (because of variation of suction and pressure), but they not necessarily vary: in the case above, all four versions of WL, Y-, are the same for the loaded external wall.

⚠ **According to EC, protuding parts of roofs should be loaded at underside with the pressure applied on the wall below. This is not taken into account by the wind loading module of FD 11.0.**

If the building have a frequently used shape, the program generates the different load domains and the values will be applied on them automatically.

Snow Load (Automatic)
Snow load (as surface loads) can be placed automatically on the open, external plates and plate parts.

Definition steps

1. Start the Snow load command from *Loads* tabmenu.

   ⚠ The generated snow surface loads will be inserted in a load case called "*Snow load*". If there is no "Snow load"-named load case in the project, the program automatically defines it for the automatic snow load.

2. Set the characteristic snow load value in the settings dialog according to the current code rules. The program calculates the applied snow load value (intensity) from the characteristic value and the built-in form, exponation and thermal factors of the Eurocode.

3. Clicking *OK* generates surface loads on the open plate surfaces/region parts.

Thermal Load
Depending on the current FEM-Design module, uniform or non-uniform temperature variation can be added to the structural elements as loads.

FEM-Design Module	Temperature Variation	Load Command
	Non-uniform temperature in Beams	Line temperature variation
	Non-uniform temperature in Plates	Surface temperature variation
	Uniform temperature in Walls	Surface temperature variation
	Uniform/Non-uniform temperature in bar elements	Line temperature variation
	Uniform/Non-uniform temperature in planar elements	Surface temperature variation
	Uniform/Non-uniform temperature in bar elements	Line temperature variation

Table: Temperature load types by FEM-Design modules

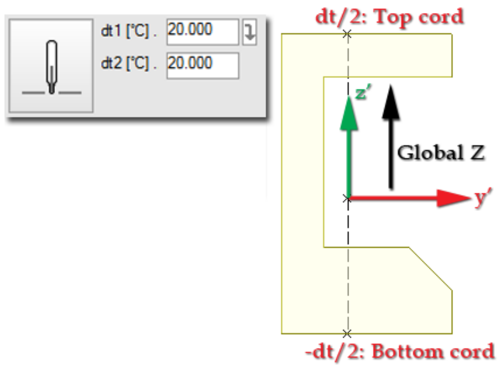
Line Temperature variation

- **Non-Uniform Temperature Variation in Beams**
  In *Plate* module, non-uniform temperature variation can be defined along beam action line or a part of it. The absolute value of the temperature has to be the same in the top and bottom cords:

  $|dt_{top}| = |dt_{bottom}|$

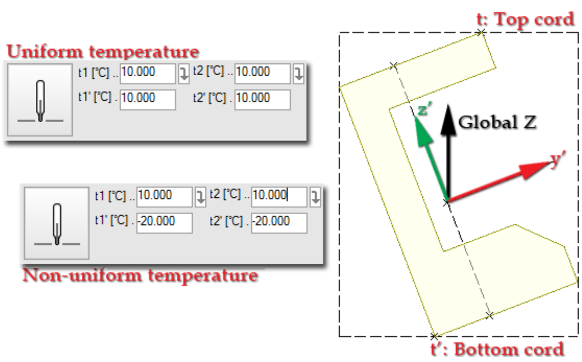
  So the meaning of the temperature variation is the following:

  $|dt| = \pm|dt_{top}-dt_{bottom}| = \pm 2 \times |dt_{top}|$



  The temperature variation can be linearly variable along the action line, so it can be different in the start and end point of the action line.

- **Uniform/Non-Uniform Temperature Variation in Bar Elements**
  In *3D Structure* and *3D Frame* modules, uniform or non-uniform temperature variation can be defined along beam and column action lines or a part of them. The temperature values can be different in the top (*t*) and bottom (*t'*) cords.



  The temperature variation can be linearly variable along the action line, so it can be different in the start and end point of the action line.

Depending on the current FEM-Design module, temperature load can be easily added to bar elements. The load properties and definition tools are available on the tool palette of the *Line temperature variation load*.

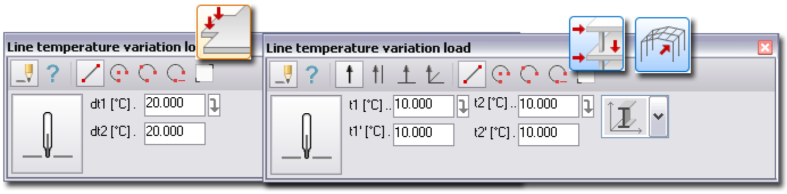


Figure: Tool palettes of Line temperature variation load

Definition steps

1. Select the load case from the load case drop-down list from Loads tabmenu, which you would like to add the new load(s) to.

   If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

   The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Set the temperature values at the start and end point of the "load" action line to the current temperature unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*. If you inactivate the arrow ( ) next to the temperature fields, you can type different temperature values in the "1" and "2" fields.

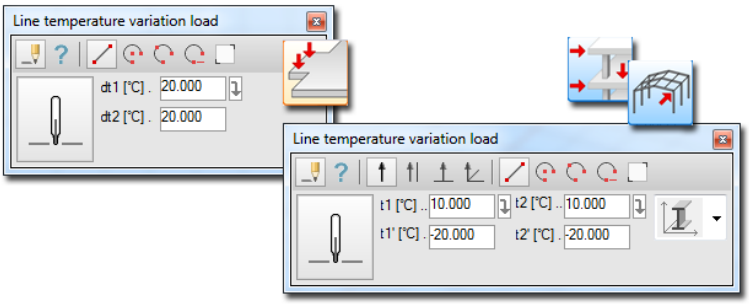


Figure: Settings dialogs (the meaning of the "t" values)

3. Define the **direction** of the plane you would like to place the thermal load (only in 3D modules).

4. Add the thermal to its belonging bar element by selecting it with .
   Or, choose a method for the action line definition (**geometry**) and define the thermal load. Use this method, if you would like to place the load on a part of the bar element.

   With the special tool called *Object's local system* (, only in 3D modules) you can set the load's plane direction directly to a selected axis of the bar element's **local system** in the tool palette and by the clicking on the bar element. So, with this tool, you can merge 3rd and 4th steps to one step, if the direction of the thermal load plane is equal to a required axis direction of the assigned bar element's local system.

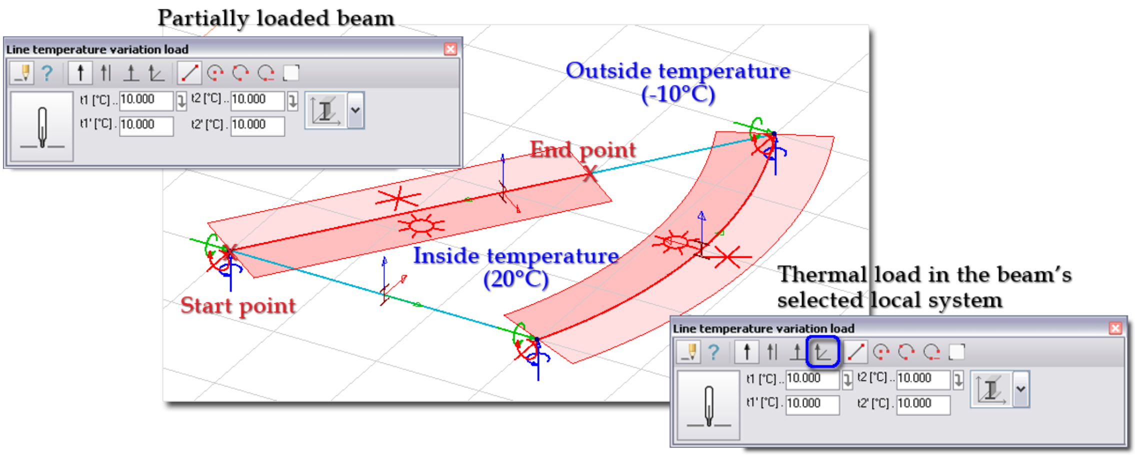


Figure: Different ways of thermal load definition

Optional steps:

5. Modify the load properties (the host load case and temperature values with the *Properties* tool of the *Line temperature variation load* tool palette.
6. Modify the direction of the thermal load plane with the editing tools (**Editing Loads**).
7. Set the display settings of the thermal load (**Load Display Settings**).

Surface Temperature variation

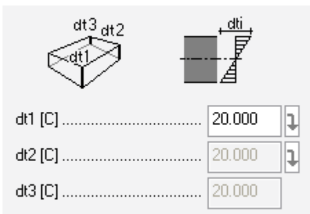
- **Non-Uniform Temperature Variation in Plates**

  In *Plate* module, non-uniform temperature variation can be defined in plate region or a part of it. The absolute value of the temperature has to be the same in the top and bottom cords:

  $|dt_{top}| = |dt_{bottom}|$

  So the meaning of the temperature variation is the following:

  $|dt| = \pm|dt_{top}-dt_{bottom}| = \pm 2 \times |dt_{top}|$

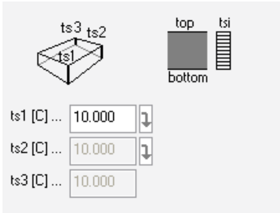


  The temperature variation can be linearly variable along the action plane. Temperature value in 3 region points defines the linearly variable temperature in a plate object.

- **Uniform Temperature Variation in Walls**

  In *Wall* module and *Plane Strain* modules, uniform temperature load can be defined in wall region or a part of it:

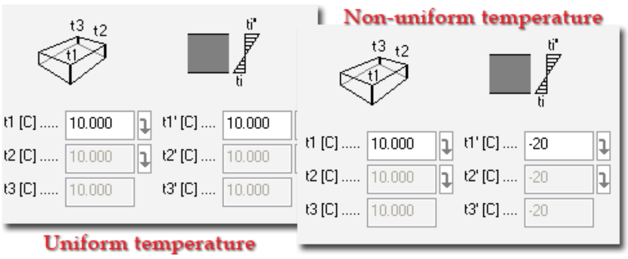
  $t_{si} = t_{top} = t_{bottom}$



The temperature variation can be linearly variable along the action plane. Temperature value in 3 region points defines the linearly variable temperature in a wall object.

- **Uniform/Non-Uniform Temperature Variation in Planar Elements**

  In *3D Structure* module, uniform or non-uniform temperature variation can be defined in plate, wall and shell regions or a part of them. The temperature values can be different in the top (*t'*) and bottom (*t*) cords.



The temperature variation can be linearly variable along the action plane. Temperature value in 3 region points defines the linearly variable temperature in a plate, wall or shell object.

Depending on the current FEM-Design module, temperature load can be easily added to planar objects. The load properties and definition tools are available on the tool palette of the *Surface support motion load*.

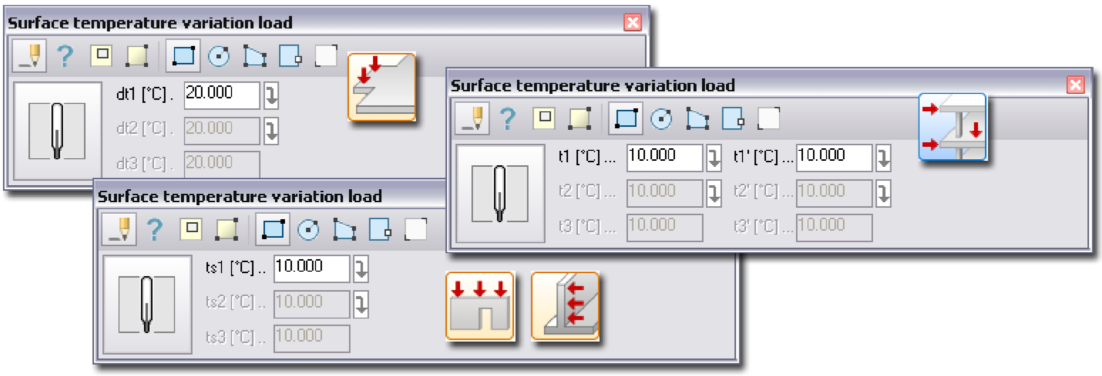


Figure: Tool palettes of Surface temperature variation load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

   If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. First, define temperature variation constant along the load action plane. Set the motion value in the "t1" fields according to the current temperature unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*.

⚠ The local z' axis of the surface element (plate, wall) points to the "top" surface temperature (t') in *3D Structure* module.

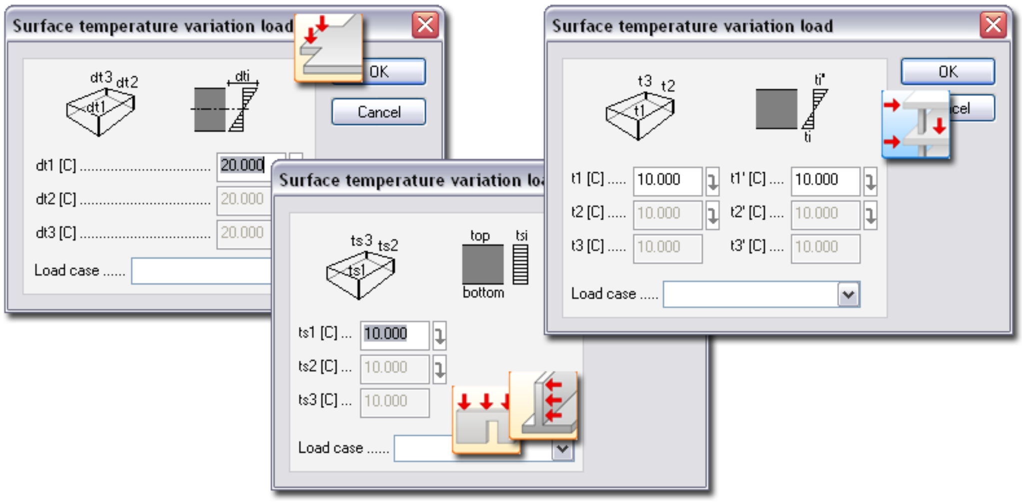


Figure: Settings dialogs (the meaning of the "t" values)

3. Add the thermal load to its belonging surface element by selecting it with .
Or, choose a method for the action surface definition (**geometry**) and define the surface thermal load. Use this method, if you would like to place the load on a part of the surface element. In this case set the **working plane** into the element's plane.

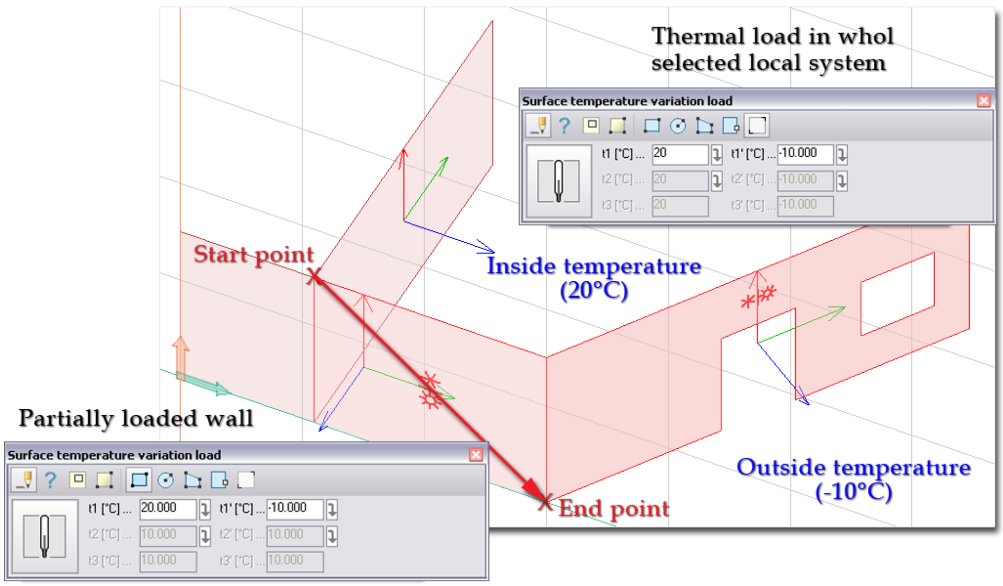


Figure: Different ways of surface thermal load definition

Optional steps:

4. Use the *Variable intensity* tool, if you would like to define variable surface thermal load. Set the new temperature values (*dt1/t1/ts1*, *dt2/t2/ts2* and *dt3/t3/ts3*). If you inactivate the arrows ( ) next to the value fields, you can type different values by the *t* fields. Select the constant surface temperature load that you would like to modify being variable. Give the position of the three temperature values. You may click outside the surface load's action plane too.
5. Place a hole/opening in a surface thermal load. Use the **Hole** tool of the *Surface temperature variation load*.
6. Modify the load properties (the host load case and temperature values with the *Properties* tool of the *Surface support motion load* tool palette.
7. Set the display settings of the thermal load (**Load Display Settings**).

Initial Internal Load

Post-stress can be added to bars (*Initial internal line load*) and structural regions (*Initial internal surface load*) as load. Constant moment and normal force values in a given direction (e.g. only in one direction) may be defined by points of line and planar structural objects.

The initial internal load commands are useful for defining post-tensioning loads in concrete bars and slabs. Post-stress is also good for steel bars at welded connections. The definition steps are the same as introduced at **Line** or **Surface temperature variation load** commands.

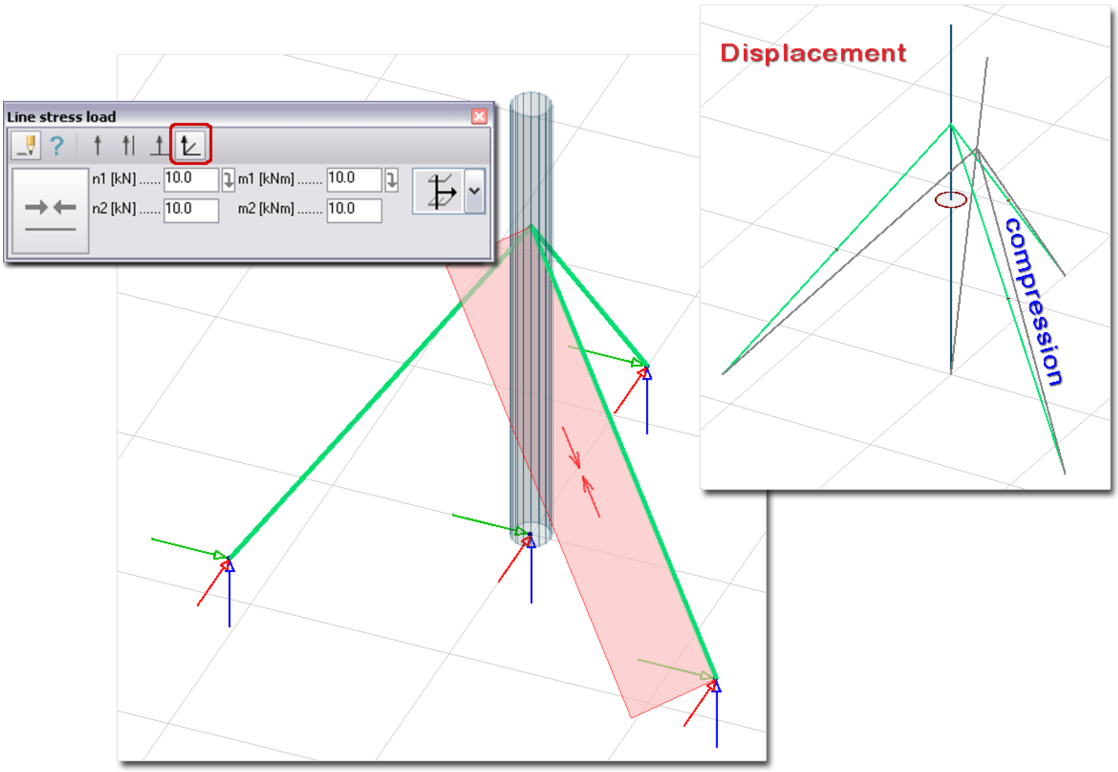


Figure: Initial internal force (compression) in a stiffener truss member

Support Motion
Predefined displacement (motion and/or rotation depending on the current FEM-Design module) can be assigned to point, line and surface supports.

Only the same type support motion can be assigned to a support.

Support motion load type	Support type
Point support motion load	**Point support**
Line support motion load	**Line support**
Surface support motion load	**Surface support**

Table: Support motion load types and their proper support types

If a support motion load is not belonging to a support, the program sends you a warning message during calculation, that that load will be ignored in the calculations.

In *Plate module* the *Column* and *Wall* elements are supports, so *Point support motion load* can be assigned to *Columns* and *Line support motion load* to *Walls*.

Point Support Motion

Depending on the current FEM-Design module, predefined point motion/rotation can be easily added to point supports by giving displacement direction and position. The load properties and definition tools are available on the tool palette of the *Point support motion load*.

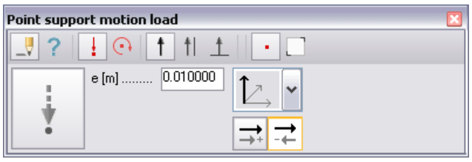


Figure: Tool palette of Point support motion load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

   If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

   The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Choose the motion type: *Motion* or *Rotation* (FEM-Design module dependent).
3. Set the displacement values in the *e* or *phi* of the new load according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*.
4. Define the **direction** of the motion or the rotation axis (FEM-Design module dependent).
5. Add the point motion to its belonging point support by clicking the support's insertion point ( ) or by selecting the point support ( ).

Optional steps:

6. Modify the load properties (the host load case and displacement values with the *Properties* tool of the *Point support motion load* tool palette.
7. Modify the displacement direction with the editing tools (**Editing Loads**).
8. Set the display settings of the load (**Load Display Settings**).

Line Support Motion

Depending on the current FEM-Design module, predefined motion/rotation can be easily added to line supports by giving displacement direction and position. The load properties and definition tools are available on the tool palette of the *Line support motion load*.

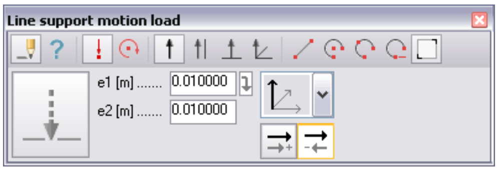


Figure: Tool palette of Line support motion load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

   If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

   The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. Choose the motion type: *Motion* or *Rotation* (FEM-Design module dependent).
3. Set the displacement values at the start and end point of the "load" action line: *e1* and *e2* or *phi1* and *phi2* according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*. If you inactivate the arrow ( ) next to the *e1* or *phi1* field, you can type intensity value in *e2* or *phi2* different from *e1* or *phi1*.
4. Define the **direction** of the motion or the rotation axis (FEM-Design module dependent).
5. Set the "direction mode", that means the connection between applied load direction and the action line at *Default settings*, if the new load will be a curved line load:
   - **"Direction is constant along action line"**
     The load direction set in the 4th step will be constant along the action line.

   - **"Direction varies along action line"**
     The load direction will vary along the action line, so the characteristic direction set in the 4th step will be taken into consideration in the middle point of the curved action line.

6. Add the motion to its belonging line support by selecting the line support ( ).
   Or, choose a method for the action line definition (**geometry**) and define the line load. Use this method, if you would like to place the load on a part of the support's action line.

With the special tool called *Object's local system* ( ) you can add motion load directly to a selected component of a support/more supports just by setting the required component direction (an axis of the support's local coordination system) in the tool palette and by the clicking on the support(s). So, with this tool, you can merge 4th and 6th steps to one step, if the required displacement direction is equal to the direction of the assigned support component.

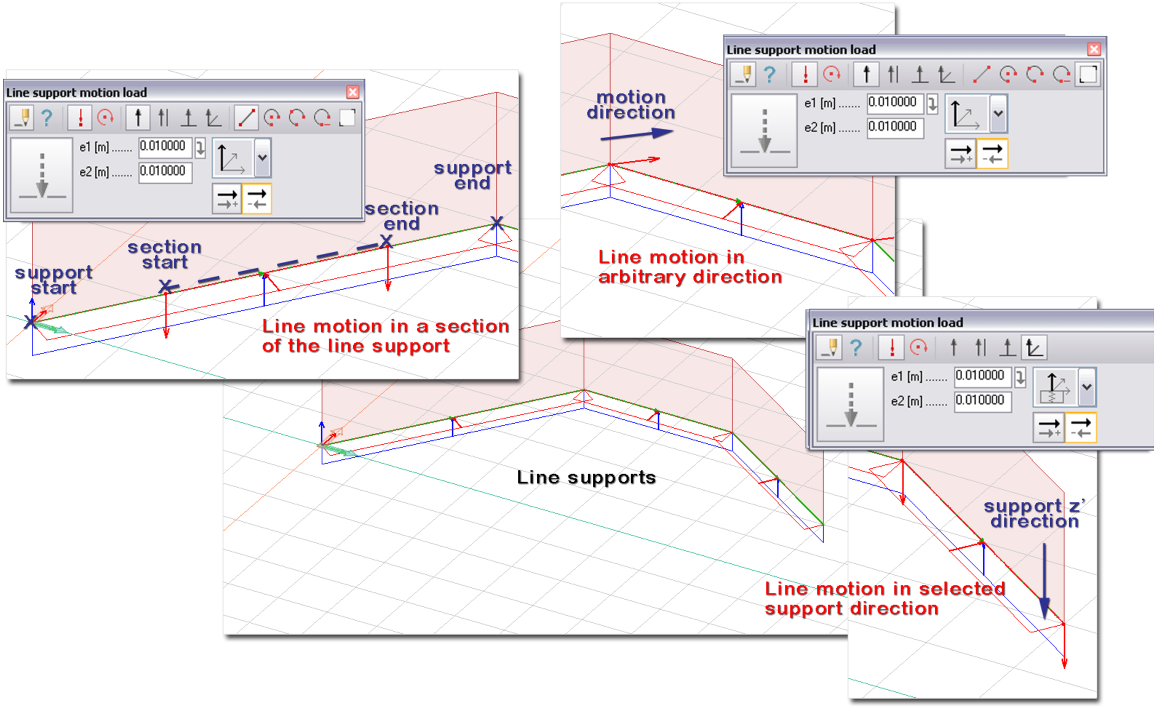


Figure: Different ways of line support motion load definition

Optional steps:

7. Modify the load properties (the host load case and displacement values with the *Properties* tool of the *Point support motion load* tool palette.
8. Modify the displacement direction with the editing tools (**Editing Loads**).
9. Set the display settings of the load (**Load Display Settings**).

Surface Support Motion

Depending on the current FEM-Design module, predefined motion can easily be added to surface support by giving displacement direction and position. The load properties and definition tools are available on the tool palette of the *Surface support motion load*.

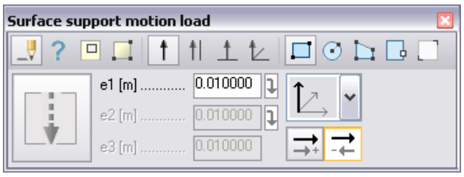


Figure: Tool palette of Surface support motion load

Definition steps

1. Select the load case from the load case drop-down list from *Loads* tabmenu, which you would like to add the new load(s) to.

If you have not defined a load case yet, you can also define and set at the *Default settings* by giving a load case name.

The load will be displayed on the layer assigned to the selected load case and in the layer's color.

2. First, define a constant predefined displacement. Set the motion value in the *e1* field according to the current unit (that can be set at *Settings* > *Units*) in the tool palette or at *Default settings*.
3. Define the **direction** of the motion (FEM-Design module dependent).
4. Add the motion to its belonging surface support by selecting the surface support ( ).
Or, choose a method for the action surface definition (**geometry**) and define the surface load. Use this method, if you would like to place the load on a part of the support's action surface.

With the special tool called *Object's local system* ( ) you can add motion load directly to a selected component of a support/more supports just by setting the required component direction (an axis of the support's local coordination system) in the tool palette and by the clicking on the support(s). So, with this tool, you can merge 3rd and 4th steps to one step, if the required displacement direction is equal to the direction of the assigned support component.

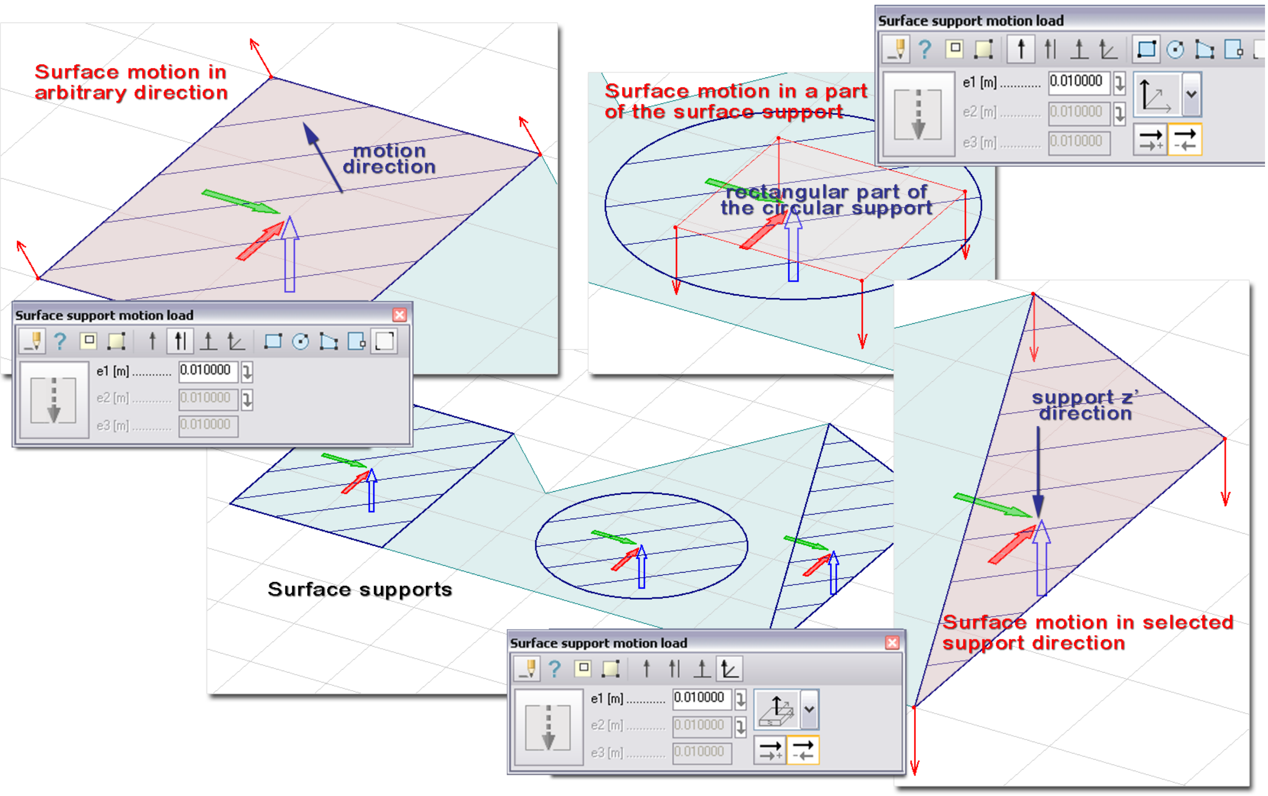


Figure: Different ways of surface support motion load definition

Optional steps:

5. Use the *Variable intensity* tool, if you would like to define variable surface motion assign to a surface support or a part of it. Set the new displacement values (*e1*, *e2* and *e3*). If you inactivate the arrows ( ) next to the value fields, you can type different values by the *e* fields. Select the

constant surface load that you would like to modify being variable. Give the position of the three motion values. You may click outside the surface load's action plane too.

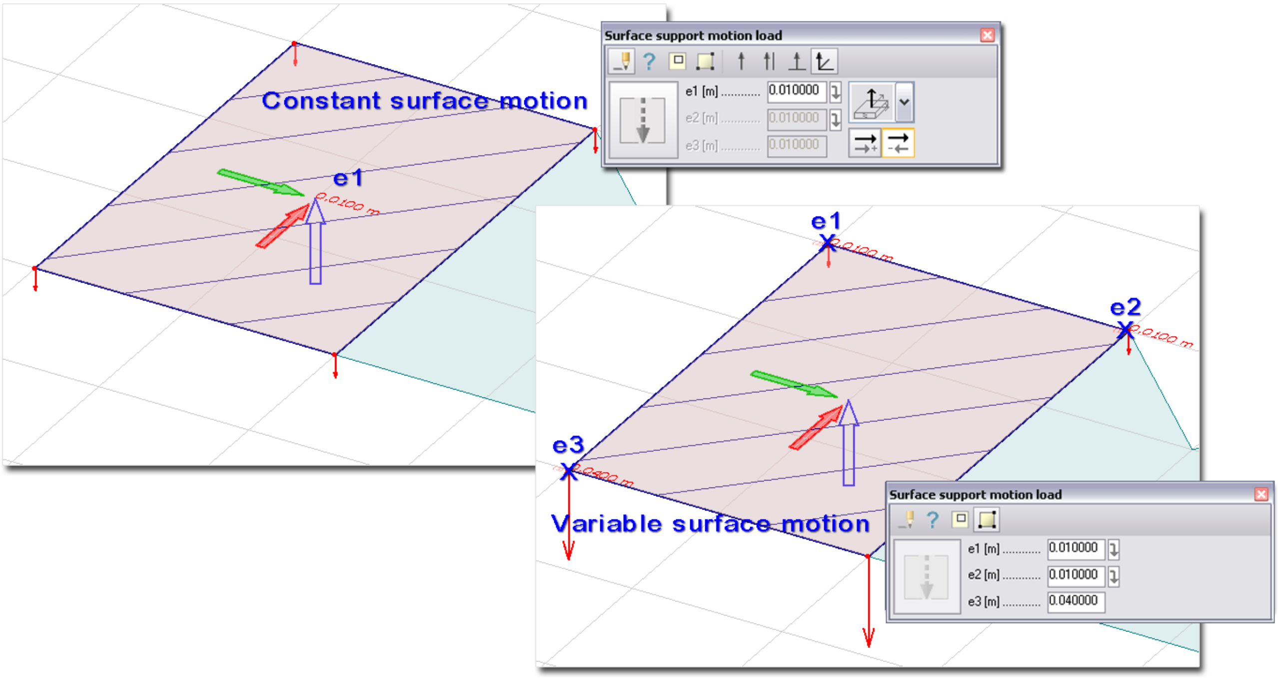


Figure: Variable surface support motion

6. Place a hole/opening in a surface load. Use the **Hole** tool of the *Surface support motion load*.
7. Modify the load properties (the host load case and displacement values with the *Properties* tool of the *Surface support motion load* tool palette.
8. Modify the displacement direction with the editing tools (**Editing Loads**).
9. Set the display settings of the load (**Load Display Settings**).

Mass

For dynamic, vibration shape and seismic effect calculations (especially for frame structures), masses can be defined optionally as point loads or automatic conversion of load cases to mass.

⚠ Masses are not participants of **load cases**, **load combinations** and **load groups**.

Concentrated Mass

Definition steps

1. Start the *Mass* command from Loads tabmenu.
2. Set the mass value according to the current unit (that can be set at *Settings* > *Units*).
3. Place the mass just by clicking in the required point.
   The mass will be displayed on the *Masses* object layer assigned in green color by default.

Optional steps:

4. Modify the mass value with the *Properties* tool of the *Mass* tool palette.
5. Set the display settings of masses (**Load Display Settings**).

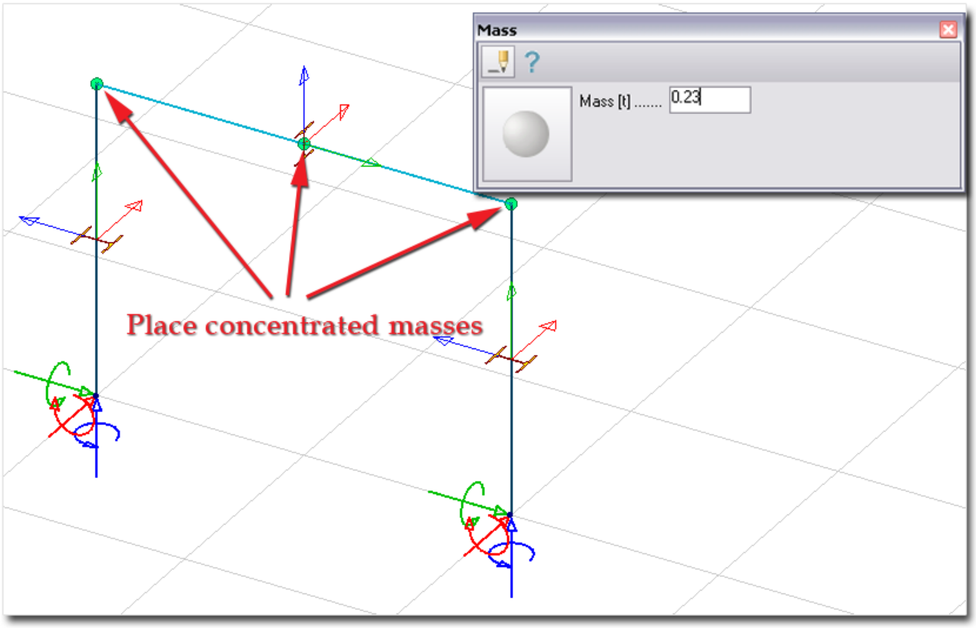


Figure: Definition of concentrated mass

Load Case-Mass Conversion

With the *Load case-mass conversion* command of *Loads* tabmenu, you may convert load cases to masses by assigning factor values to them. All conversions are done just before dynamic calculation.

Typing *1* into the *Factor* field means that the loads (of the load case) with their original value will be converted to mass. Typing *0* value removes predefined conversion of a load case (as a result you will get the "--" symbol again).

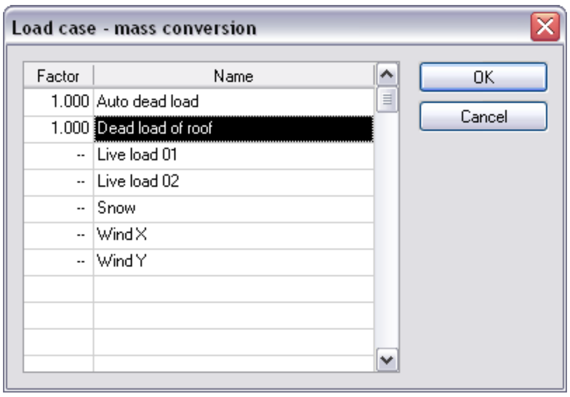


Figure: Mass definition by conversion of loads (load cases)

The following point, line and surface force types can be converted to masses:

Plate module: force with global -Z direction: mz = -Fz/g

Wall and *Plain Strain modules*: force with global -Y direction: mx = my = -Fy/g

3D Frame and *3D Structure modules*: force with global -Z direction: mx = my = mz = -Fz/g

Moments, support motions, thermal loads cannot be converted to masses.

Seismic Load
Seismic loads are taken into account according to the Response Spectrum Analysis method of Eurocode 8 or Turkish seismic code.

Only the response spectrum and some additional parameters have to be defined as Seismic load.

Required spectrums can be defined with the *Seismic load* by using standard spectra (automatic) or by manual definition (unique).

The applied/defined spectra and its properties can be added to documentation as figures and tables by clicking on *Add to documentation*.

Automatic Design Spectrum Definition ("Standard")

Clicking on *Standard* generates the horizontal and vertical spectra based on settings and parameters required by Eurocode 8.

The horizontal spectrum is always necessary. The vertical spectrum is necessary when the vertical affect taken into account.

- **Structure information**
  <u>Eurocode code:</u>

  Structure type:

  - Building structure
  - Non-building structure

  ksi: is the viscous damping ratio (generally 5%), which is used in **Modal analysis** when the summation of the effect of the same direction vibration shapes is carried out by the CQC (Complete Quadratic Combination).

  qd: behavior factor. The displacements from seismic analysis is increased with this factor.

  <u>Turkish seismic code:</u>

  Structure type:

  - New building
  - Existing building (Limited, Medium or Comprehensive *Information level):* The *Information level* affects the cross-sectional resistances in Performance Based Design calculations.
  - Non-building structure.

  Beta: This coefficient is for checking if the ratio of base shear force calculated with *Mode superposition method* and *Equivalent seismic load* is greater than this value.

⚠ Storeys definition is necessary if the user selects the *Structure information* as *building* (in Turkish seismic code *New* or *Existing Building)*

For *Non-building* structures the following result are calculated after *Seismic analysis:*

- Equivalent loads
- Displacements
- Reactions
- Connection forces
- Bar / Shell internal forces

For building structures additional result are calculated which can be listed after *Seismic analysis:*

- Second order effect (P-Delta) coefficients

- **Horizontal spectra**

Eurocode **Turkish seismic code**

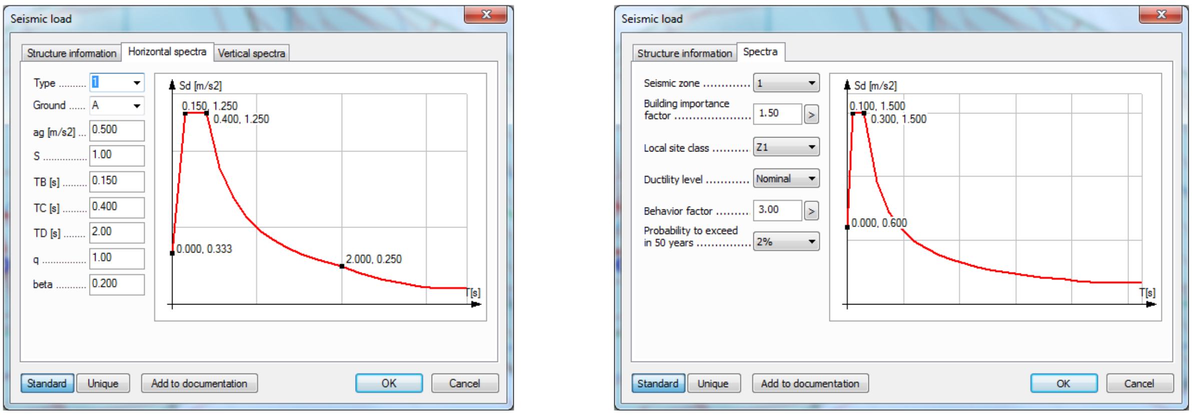


Figure: Parameters and automatic generation of horizontal spectra

Data of horizontal design spectra (based on Eurocode 8):

- *Type* (of spectra: 1 and 2) and *Ground* type (A, B, C, D and E) specify the values of *S*, *TB*, *TC* and *TD*;
- *ag* is the design ground acceleration;
- *S* is the soil factor;
- *q* is the behavior factor that depends on material and type of the structure;
- *beta* is the lower bound factor for the horizontal design spectrum.

Data of horizontal design spectra (based on Turkish seismic code):

- *Seismic zone* (1, 2, 3 or 4), *Local site class* (Z1, Z2, Z3 or Z4), *Building importance factor* and *Probability to exceed in 50 years* specify the values of S, TA, TB;
- Ductility level:
- Behaviour factor: depends on material and type of the structure;

Vertical spectra

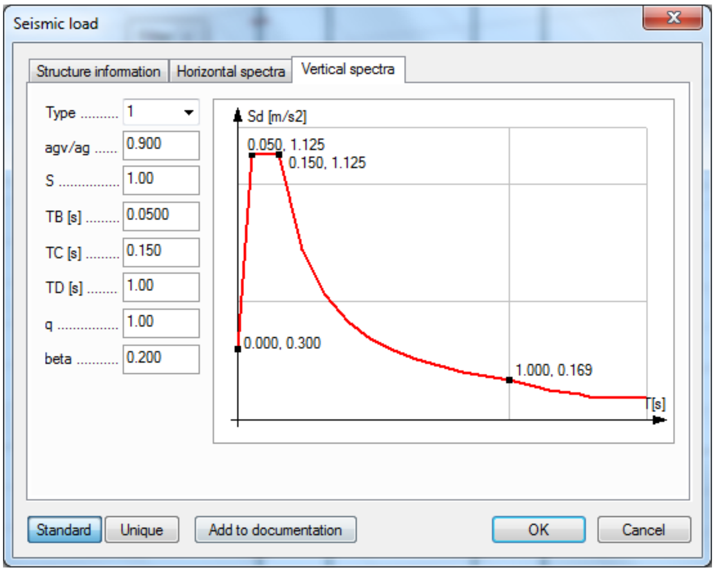


Figure: Parameters and automatic generation of vertical spectra

The vertical design spectrum is derived from the horizontal spectrum using the *agv* (vertical design ground acceleration)/ ag multiplicator based on Eurocode 8.

⚠ The vertical spectra is available only in Eurocode and not available in Turkish seismic code.

Custom Design Spectrum Definition ("Unique")

User can define custom horizontal and vertical spectra in table format or by editing spectra graphically. Switch to *Unique* mode, and then add *T* and *Sd* values to the table cells or place points in the Sd-T(s) curve by clicking in the required points of the figure.

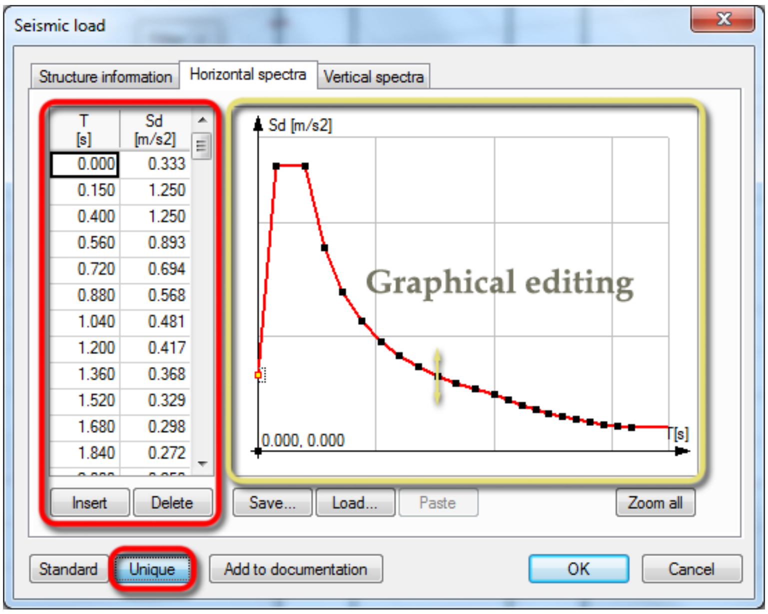
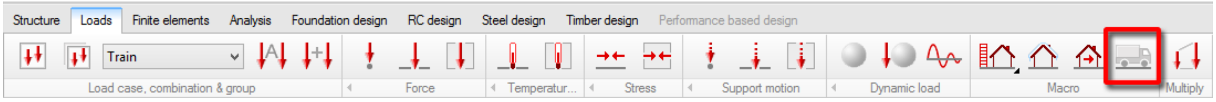


Figure: Custom spectra definition

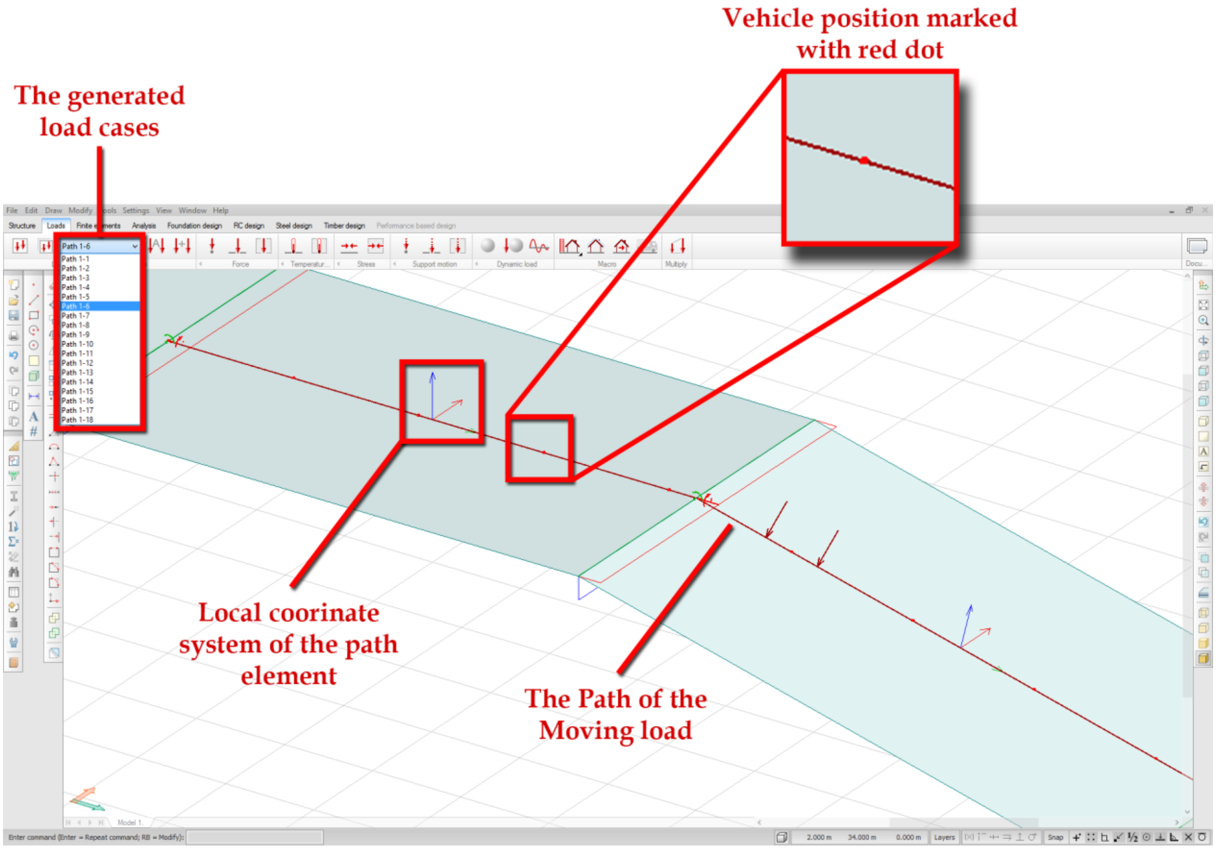
Unique spectra can be stored in files and exported with *Save*. Spectra can be imported with *Load*.

Moving load



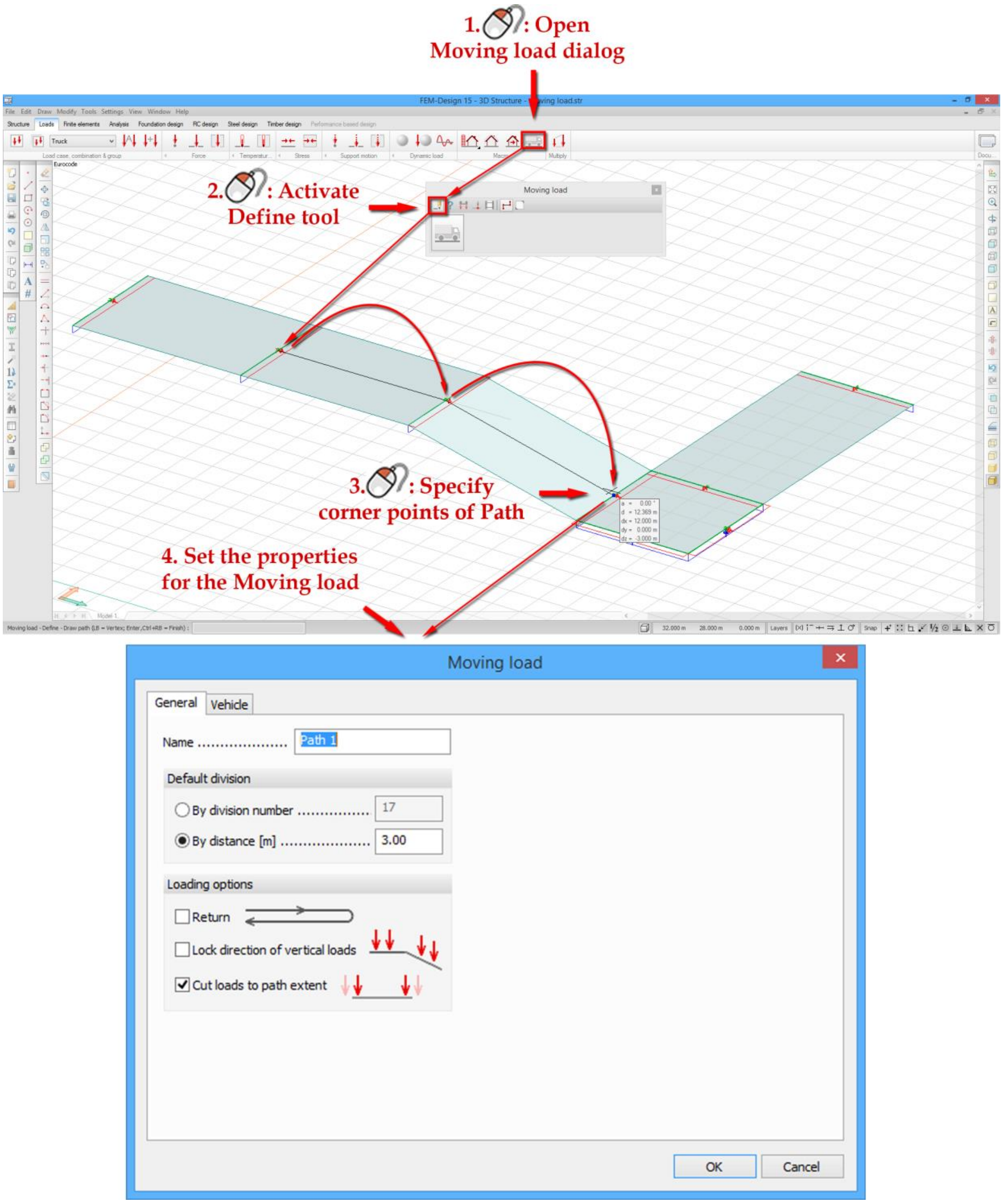
The Moving load macro applies a pre-defined set of loads (Vehicle) along a user-specified path. Its working method is the following:

1. User specifies the path and the preferences of the Moving load.
2. The macro defines Vehicle positions along the path according to the preferences.
3. The pre-defined Vehicle is added to each Vehicle position, producing singular Load cases.
4. A Load group is created, containing the Load cases of the Moving load.



Defining a Moving load

The following figure shows the definition method of a Moving load:



The options on the General tab are described below.

Default division: the method how the path of the Moving load is divided into segments. The Vehicle positions will be defined in the points of division and at the start and the end point. The division can be defined

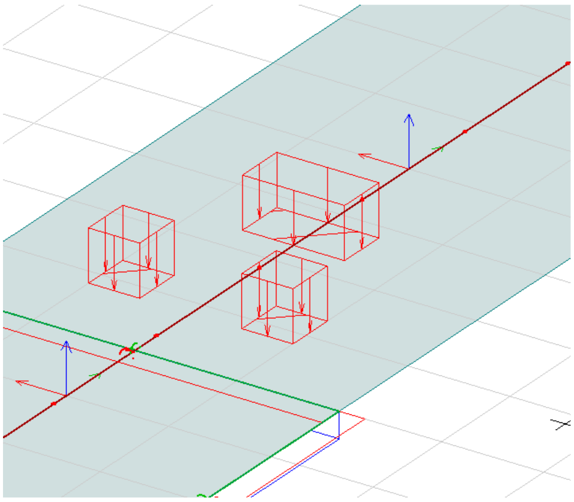
- *By division number*: the number of parts, that the macro divides the Path into, will be equal with the given value. Accordingly, the result is one more Vehicle position than the user-specified value. The distance between the points of division (and therefore between the Vehicle positions) are equal.

- *By distance*: the software defines as many segments from the start of the Path, with length equal to the specified value, as possible. The last segment may be shorter than the others.

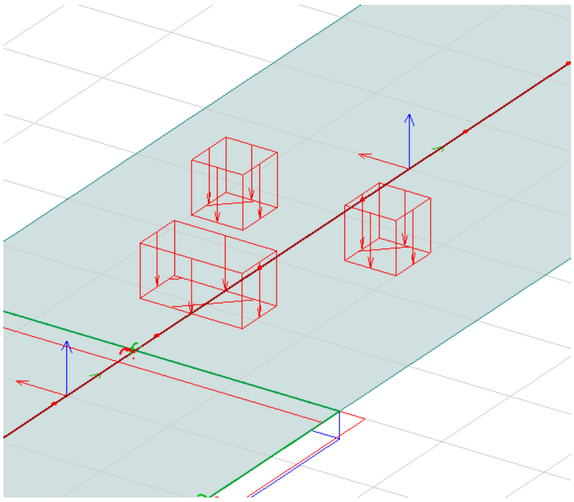
Loading options: the preferences of the way the macro defines the individual load cases along the path.

-  *Return*: if checked, two load cases are defined in each Vehicle position. The first one has the same X axis direction as the Local coordinate system of the path element its Vehicle position is located on. The second one has the opposite X axis direction.

Vehicle placed with the same X axis direction as the path element (LCS)

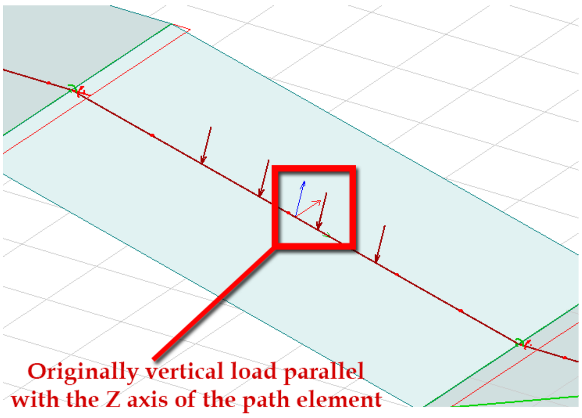


Vehicle placed with X axis direction opposite to the path element (LCS)

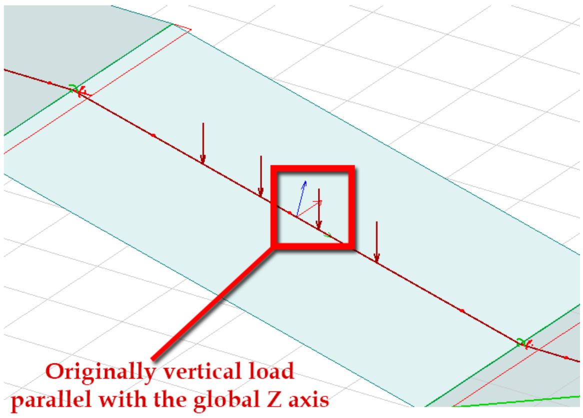


-  *Lock direction of vertical load*: if checked, the vertical loads of the selected Vehicle are forced to be parallel with the global Z axis. Otherwise, they are parallel with the Z axis of the Local coordinate system of the path element which they are located on.

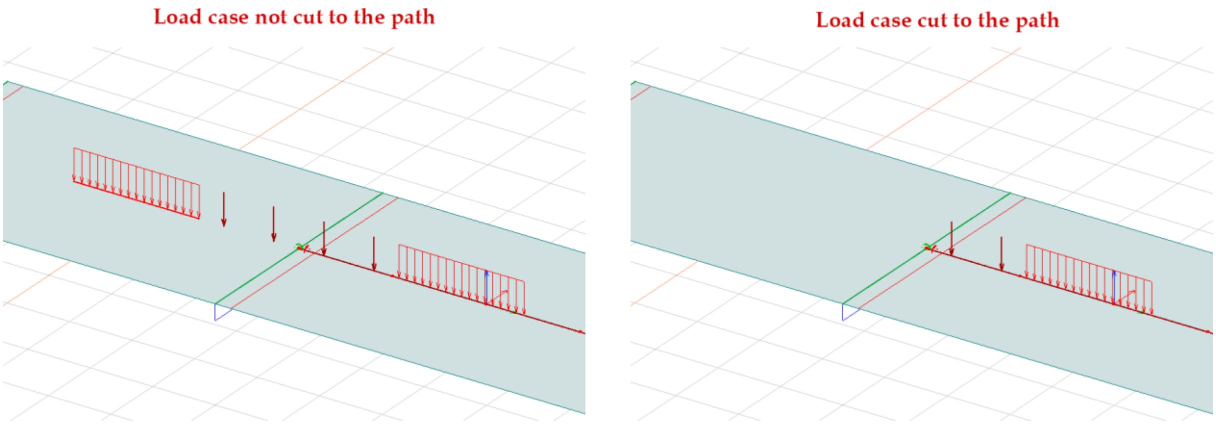
Load case without locked direction of vertical load



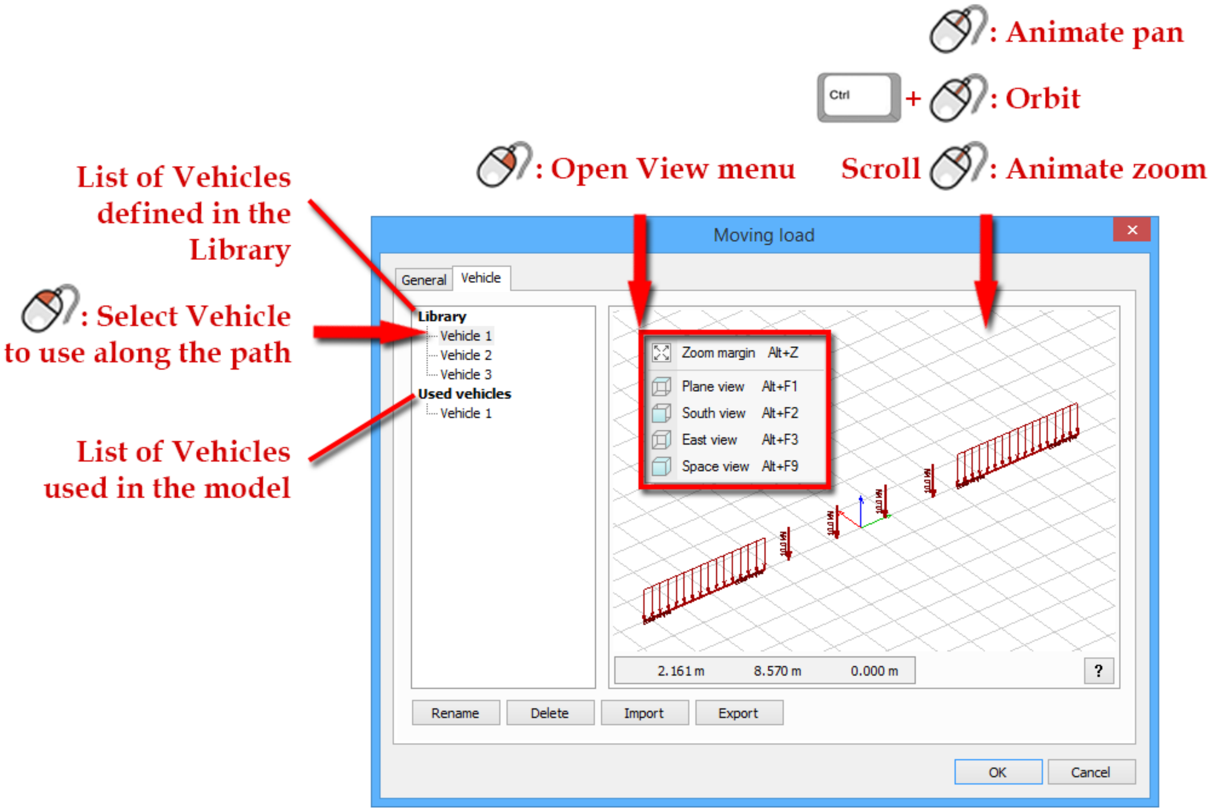
Load case with locked direction of vertical load



- *Cut load to path extent*: if checked, every load case of the Moving load gets analysed. If it extends over the path, then the software trims off the overhanging part.



User can select the set of loads to apply in the Vehicle tab.

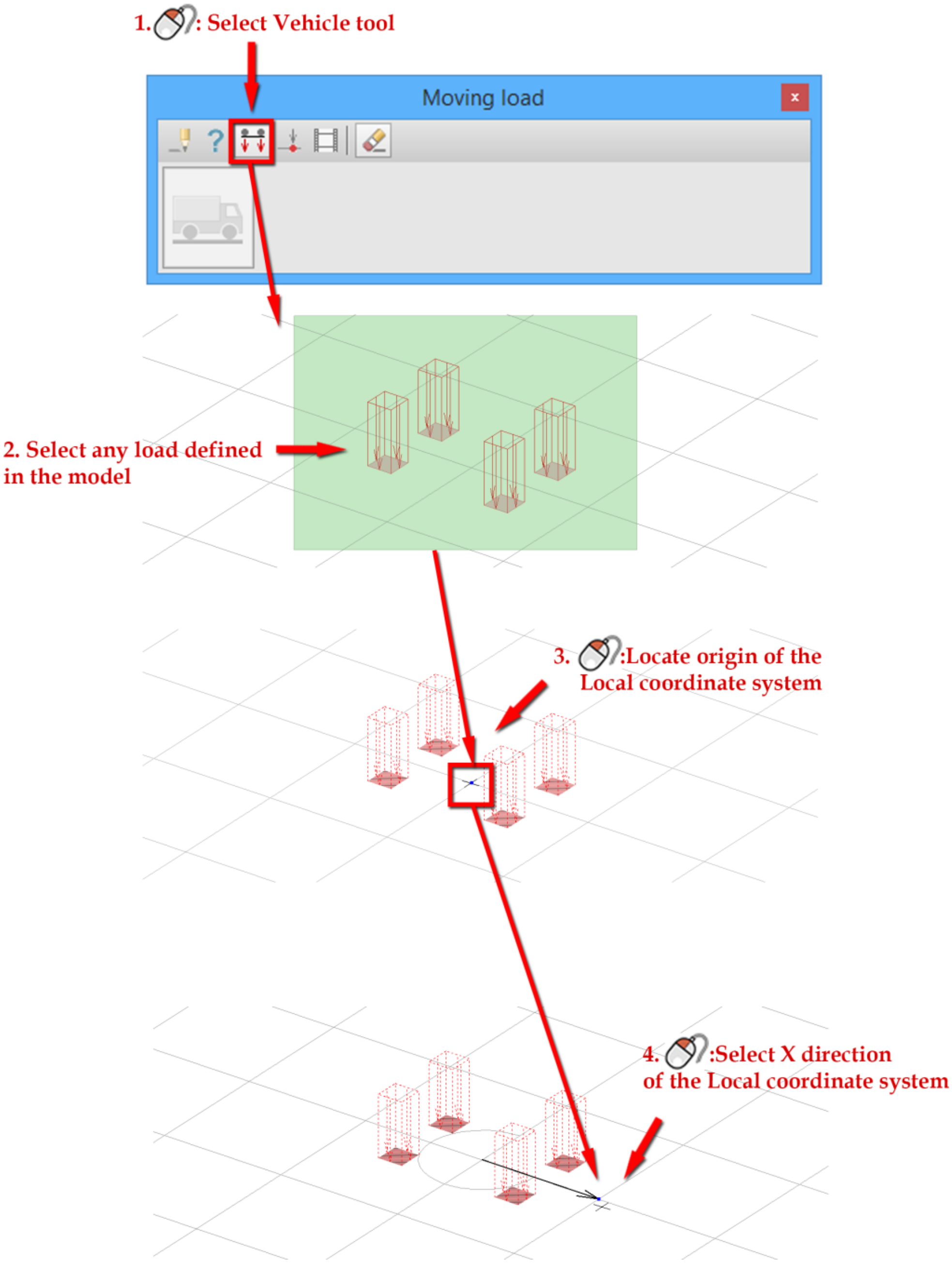


The functions of the buttons at the bottom of the dialog:

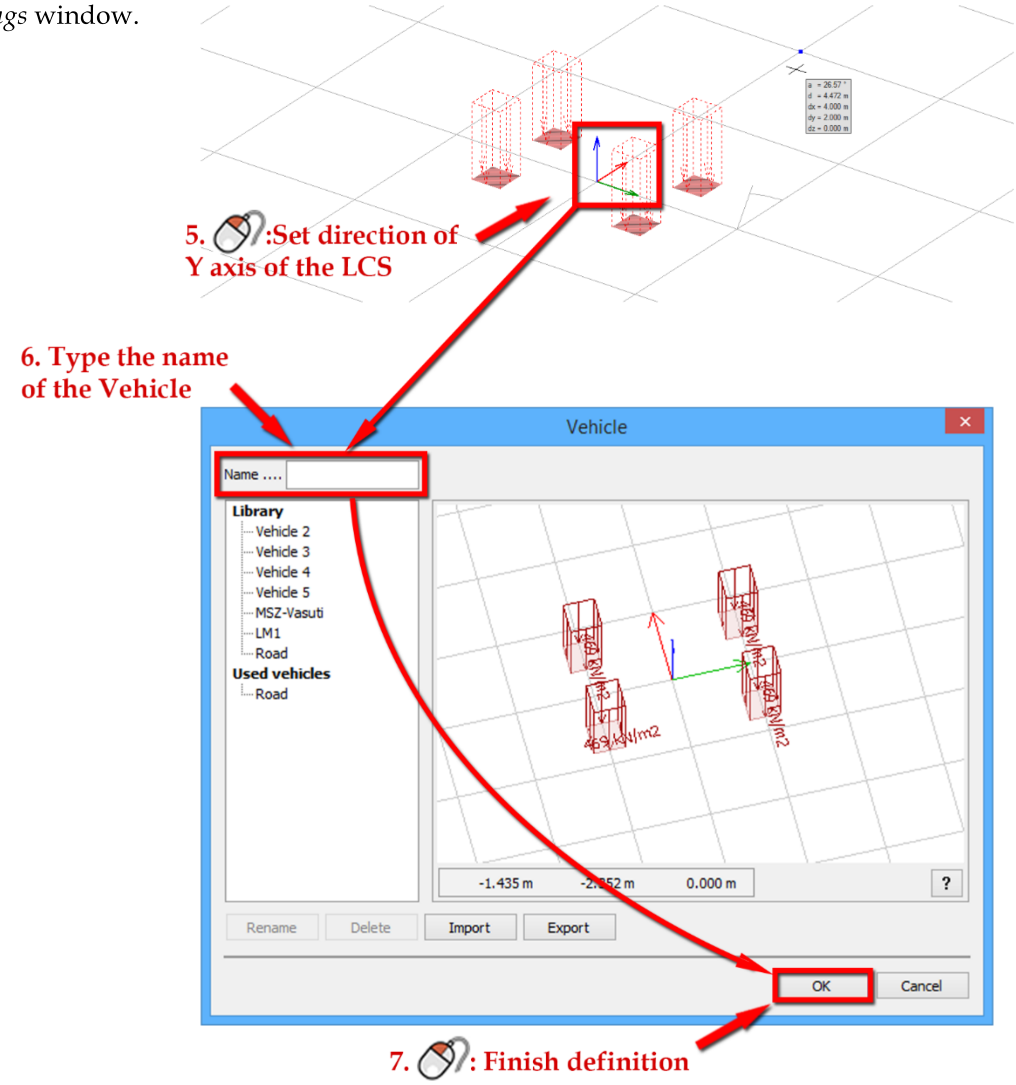
- Rename : Rename selected Vehicle
- Delete : Delete selected Vehicle
- Import : Import Vehicle library from existing (*.vehicle) file
- Export : Export Vehicle library to file

Defining Vehicles

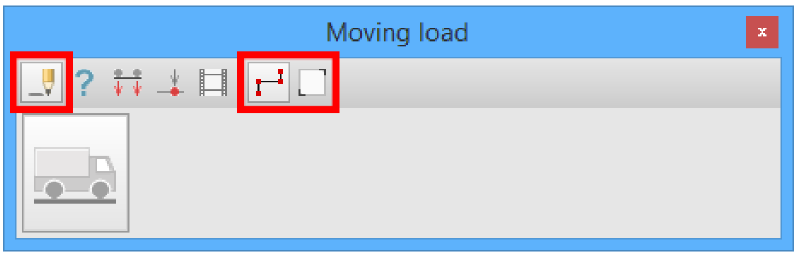
The steps of Vehicle definition:

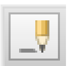


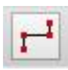
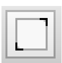
The size of the local co-ordinate system symbol can be adjusted on the *Display/Load* tab in the *Settings* window.



Summary of Moving load functions



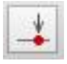
 Define: the regular **Define** tool. It has two options:

-  Draw path: selecting it allows User to define Path by its corner point.
-  Select path: when selected, path can be defined from existing linear objects, such as drawing lines, bars, edges of plates or walls.

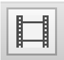
 Properties: the regular **Properties** tool.

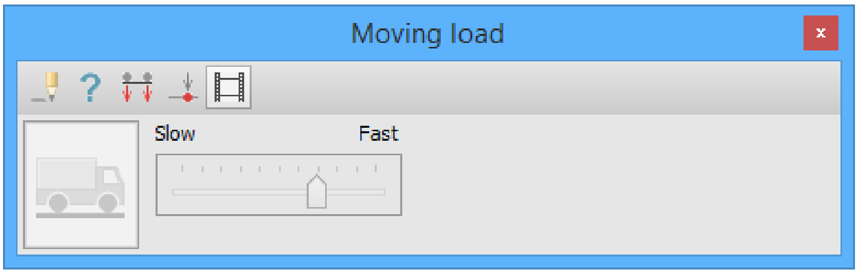
 Vehicle: User can define Vehicles with it. This tool has the option:

-  Delete original load: if activated, loads used for the definition of Vehicle will be deleted

 Vehicle position: allows the modification of Vehicle positions.

-  Add: defines a Vehicle position in a selected point along the path
-  Delete: removes the selected Vehicle position
-  Move: changes the position of the selected Vehicle position along the path
-  Reset: redefines all Vehicle positions on the selected path according to the Preferences of the Moving load. It removes all former modification of Vehicle positions.

 Animate motion: Plays an animation that shows each load case of the Moving load, one after the other. The speed of the animation can be set with the slider.



Predefined Load Values

You can browse from predefined intensity values when clicking on the  button of the Surface load command's tool palette. Just select a value from the drop-down list and it will be added in the proper *q* field.

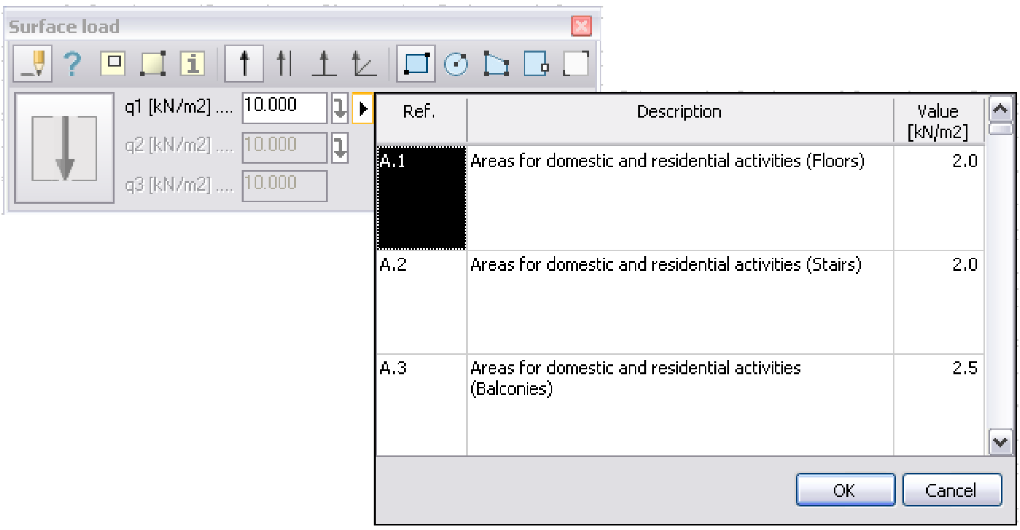


Figure: Predefined intensity value added to q field (Surface load)

Load Display Settings

The display properties of the loads can be set at the *Settings* > *All…* > *Display* > *Load*.

The available options depend on the current FEM-Design module.

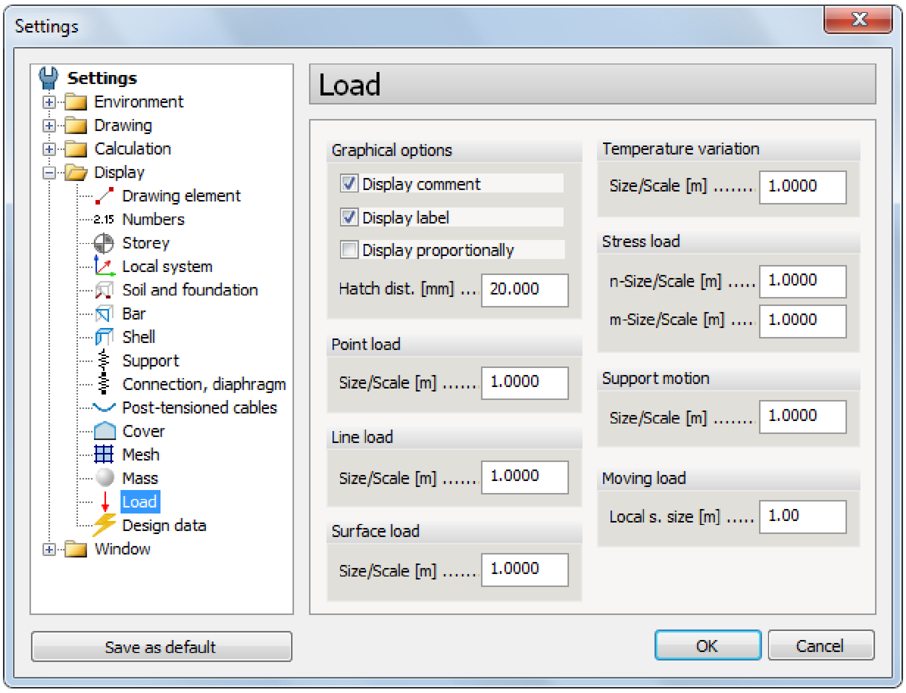


Figure: Settings options affect on the appearance of the loads

- **Display label**
  The load values can be displayed on the screen in **Wireframe display mode**. The default font size and style can be set at *Settings* > *Text* settings.

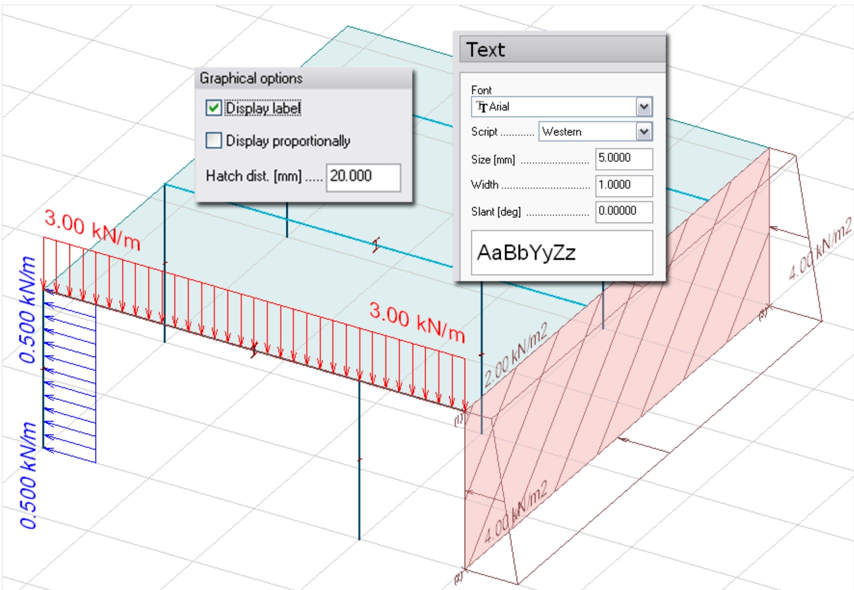


Figure: Load labels displayed according to Text settings

The position of the load labels can be modified with *Edit* > *Move*.

- **Display proportionally**
  By default, this option is inactive. That means all loads are displayed according the *Size* [m] set by load types. So, for example, point loads having different force values are displayed with same size arrow symbols with the height set by *Size*.

  Activating the *Display proportionally* option the loads will be displayed with their values multiplied with the *Scale* value set by load types. So, if the *Scale* value is 1.0 for all load types, the loads are displayed with their real values converted to meter units. For example, 5kN point force is displayed 5m-height symbol in case of 1.0 *Scale* value; but modifying the Scale value to 2.0 displays the 5kN force with 10m-height symbol.

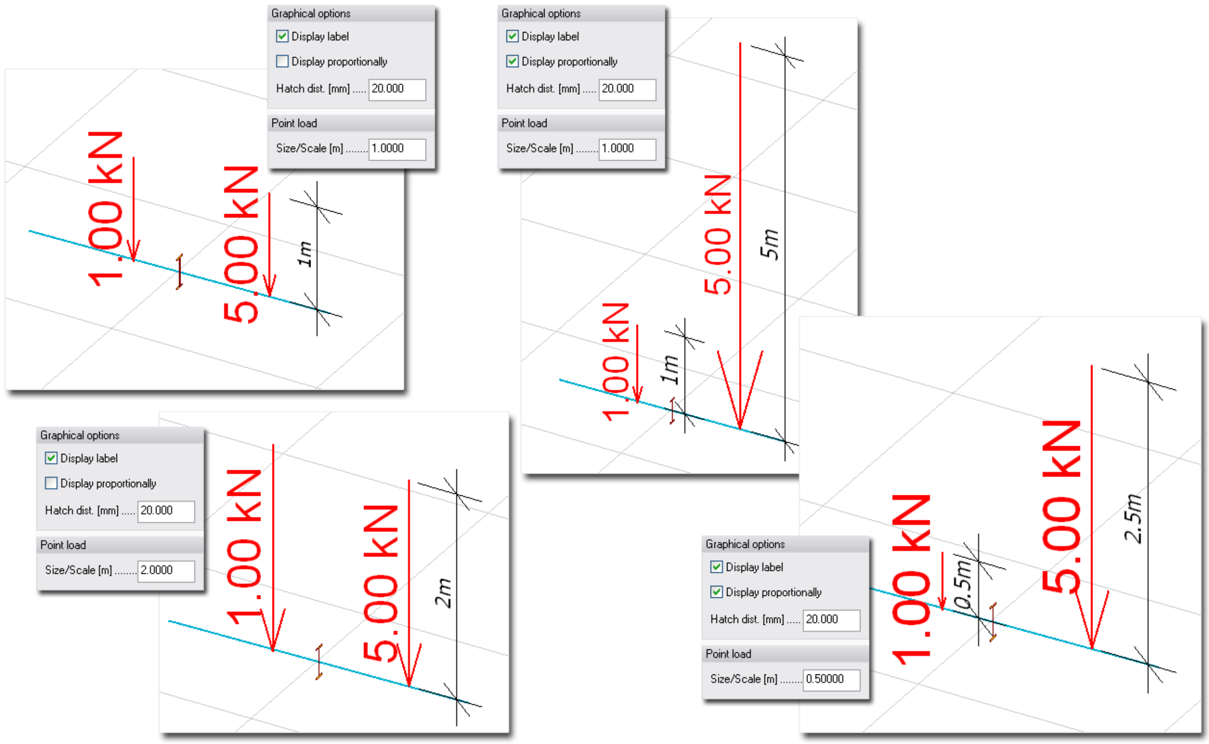


Figure: Examples for different Size/Scale values and Display proportionally option

- **Hatch distance**
  Hatch distance sets the density of the hatches of the surface loads' action plane.

Layer, color and pen width

All loads are placed (and grouped) on **Object layers** according to their host load case. Color and pen width are assigned to each load case. After defining a load case the program automatically set red color for it by default. The default color together with the pen width of the load symbol contours can be modified with the *Color* and *Pen* tools.

By default, the layer of the current load case is active and the others are hidden. Of course, you can activate, hide or protect load case layers as you want.

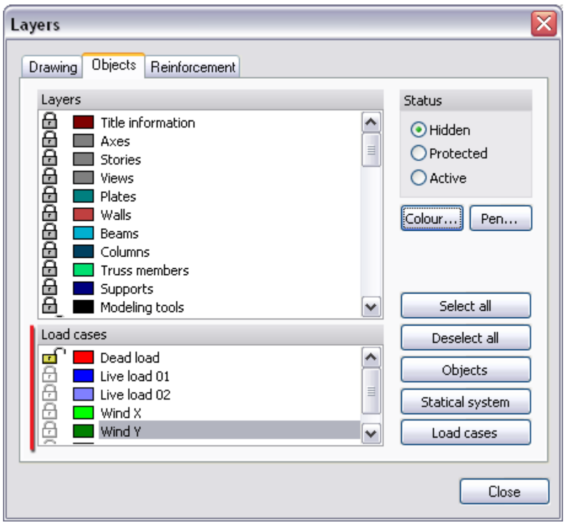


Figure: Layer-system of loads (load cases)

Editing Loads

Copying Loads

The *Copy load case* command (*Loads* menu) gives an easy way to copy all defined loads of a load case (*Source*) into another load case (*Destination*). With the *Multiplication factor* the values of copied loads can be increased or decreased proportionately.

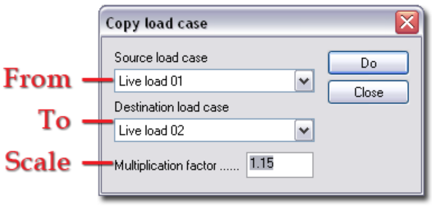


Figure: Copying loads of a load case to another one

Loads can be copied inside a load case with the **Copy** command (*Modify* menu).

Modifying Load Values

Load properties (such as the load values) can be easily modified with the *Properties* tool of the proper load definition command.

With the *Multiply load* command, you can modify values of selected loads with a given multiple factor.

Modifying Load Directions

Depending on the current FEM-Design module, the direction of predefined loads can be modified with the **Change direction**, **Mirror** and **Rotate** command of the *Edit* menu.

Modifying Load Positions

The position of Loads can be modified with the **Move** command (*Modify* menu).

Modifying the Geometries of Action Lines and Surfaces

The editing tools valid for region elements can be used to modify the geometry of surface loads' action plane. These *Edit* menu commands are for example **Modify region**, **Stretch**, **Curve**, **Elbow**, **Scale**, **Chamfer**, **Fillet** etc. The **Hole** tool of surface load commands can also be used to cut parts from a surface load.

The editing tools valid for lines and arcs can be used to modify the geometry of line loads' action line. These *Edit* menu commands are for example **Stretch**, **Curve**, **Elbow**, **Split**, **Trim**, **Extend**, **Break** etc.

Combination of Loads/Load cases

Load cases (and so their load contents) can be combined manually with given load factor multipliers (*Load Combination*) or the program finds the most unfavorable combinations of the load cases grouped in different types (permanent, temporary, accidental etc.).

Load Group

Load cases can be grouped (*Load groups* command) by their action type (permanent, temporary, accidental etc.). The program calculates (if required) the critical values of analysis and design results from the most unfavorable combinations of "grouped" load cases.

Non-linear calculations such as 2^{nd} order analysis, stability and cracked-section analysis cannot be done for load groups.

There is an option for temporary load groups to choose predefined ψ_0, ψ_1 and ψ_2 values.

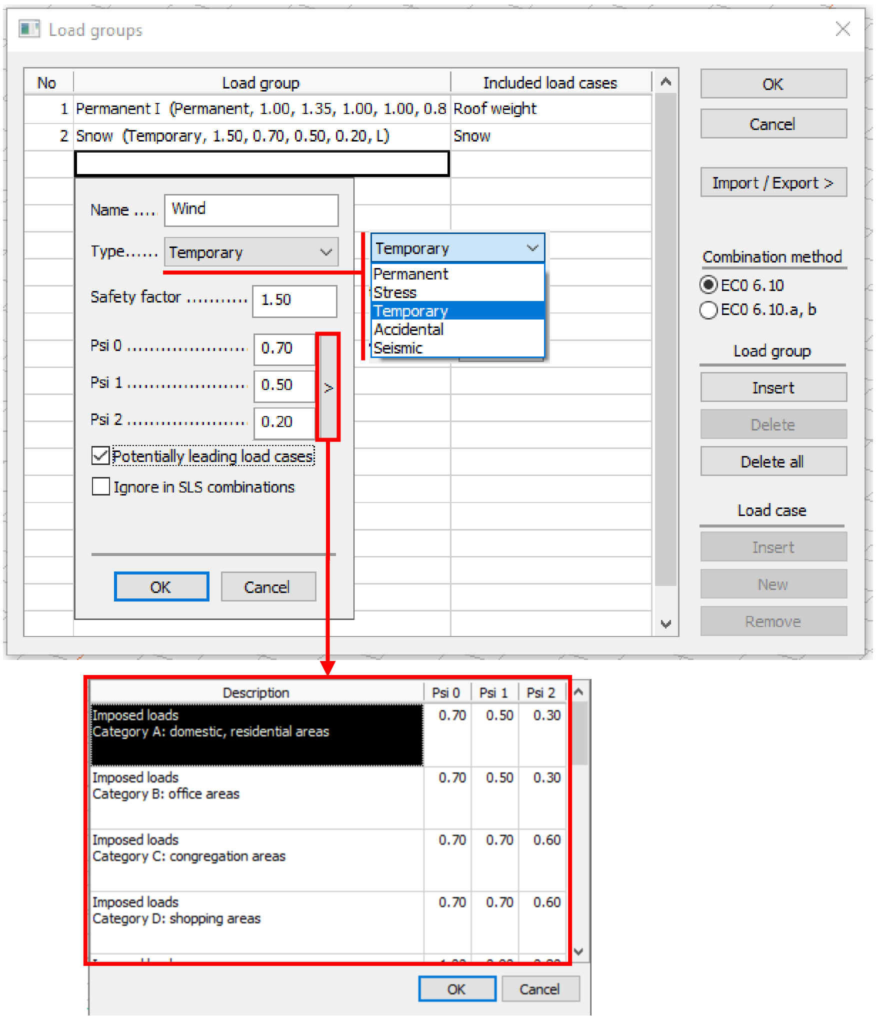
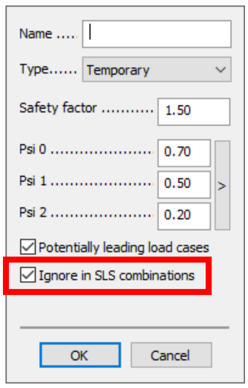
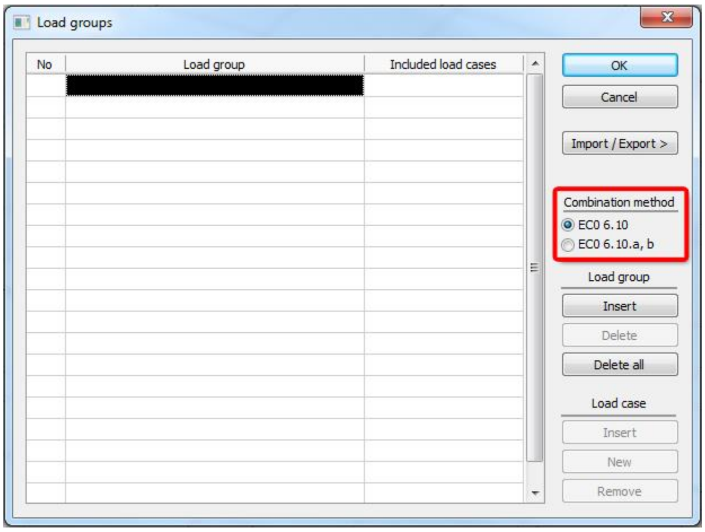


Figure: Load group definition

Temporary load groups has an option to ignore in SLS combinations in Load group maximum results and in Generating load combinations by load groups.



In Load groups dialog the User has the opportunity to choose one of the combination methods offered by Eurocode 0. Two methods of determining the combination of actions are allowed for the STR Ultimate Limit States.



The first approach is to use expression 6.10.

$$\sum_{j\geq1} \gamma_{G,j} G_{k,j} \text{" + "} \gamma_p P \text{" + "} \gamma_{Q,1} Q_{k,1} \text{" + "} \sum_{i\geq1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \qquad (6.10.)$$

The second approach is to use the more onerous of expressions 6.10.a and 6.10.b.

$$\sum_{j\geq1} \gamma_{G,j} G_{k,j} \text{" + "} \gamma_p P \text{" + "} \gamma_{Q,1} \psi_{0,1} Q_{k,1} \text{" + "} \sum_{i\geq1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \qquad (6.10.a)$$

$$\sum_{j\geq1} \xi_j \gamma_{G,j} G_{k,j} \text{" + "} \gamma_p P \text{" + "} \gamma_{Q,1} Q_{k,1} \text{" + "} \sum_{i\geq1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \qquad (6.10.b)$$

The subtle attraction of this pair of expression derives from two important changes from 6.10.:

The application of the ψ_0 factor to the leading variable action in expression 6.10.a (not applied in 6.10.)

The introduction of a reduction factor ξ applied to the permanent actions in expression 6.10.b (not applied in 6.10.)

It is possible to deactivate the *Potentially leading load cases* check box for a *Temporary* load group. This way the number of generated load combinations can be reduced.

The whole expressions appear on the screen as the cursor is moved on the name of the combination method.

Load group and Load case definition functions are shown on the right side of the *Load groups* dialog box.

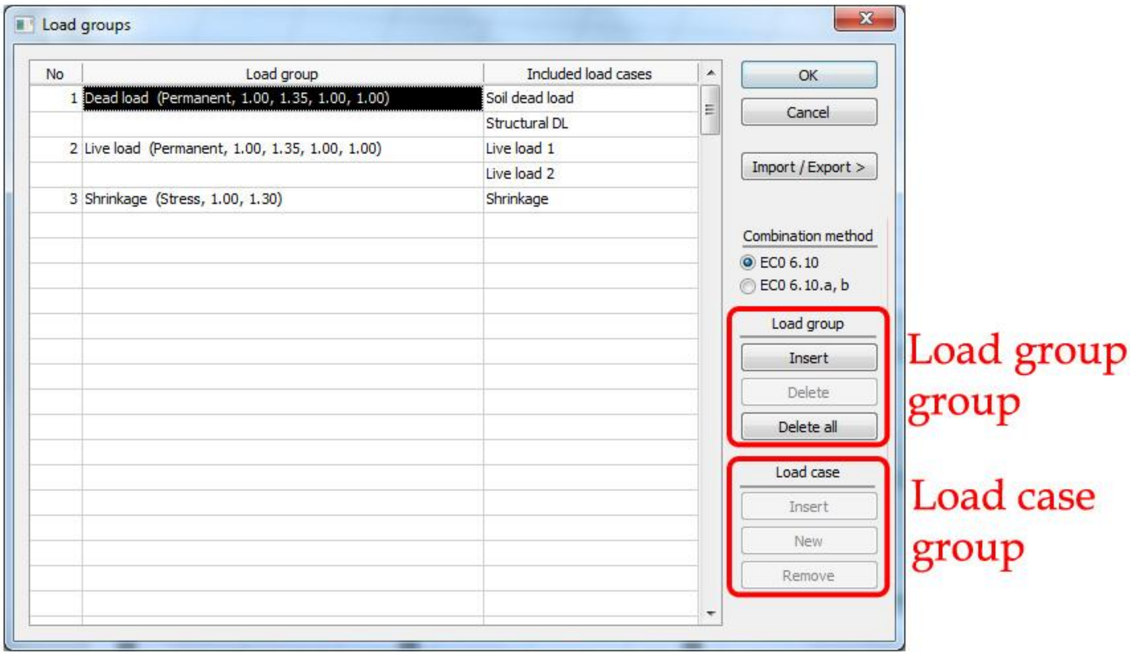


Figure: Load group definition functions

Under *Load group* group, there are three functions:

- **Insert**

  With this function the user can add a new *Load group*. After clicking on Insert button, the parameters can be set in the dialog.

- **Delete**

  This function deletes a *Load group*.

- **Delete all**

  This function deletes all defined load groups.

Under *Load case* group, there are three functions:

- **Insert**

  An existing load case can be inserted to a load group.

- **New**

  The user has the opportunity to define load cases in *Load groups* dialog. After defining a load case in this dialog, the new load case will be added to *Load case* list (the user can see it in Load cases dialog).

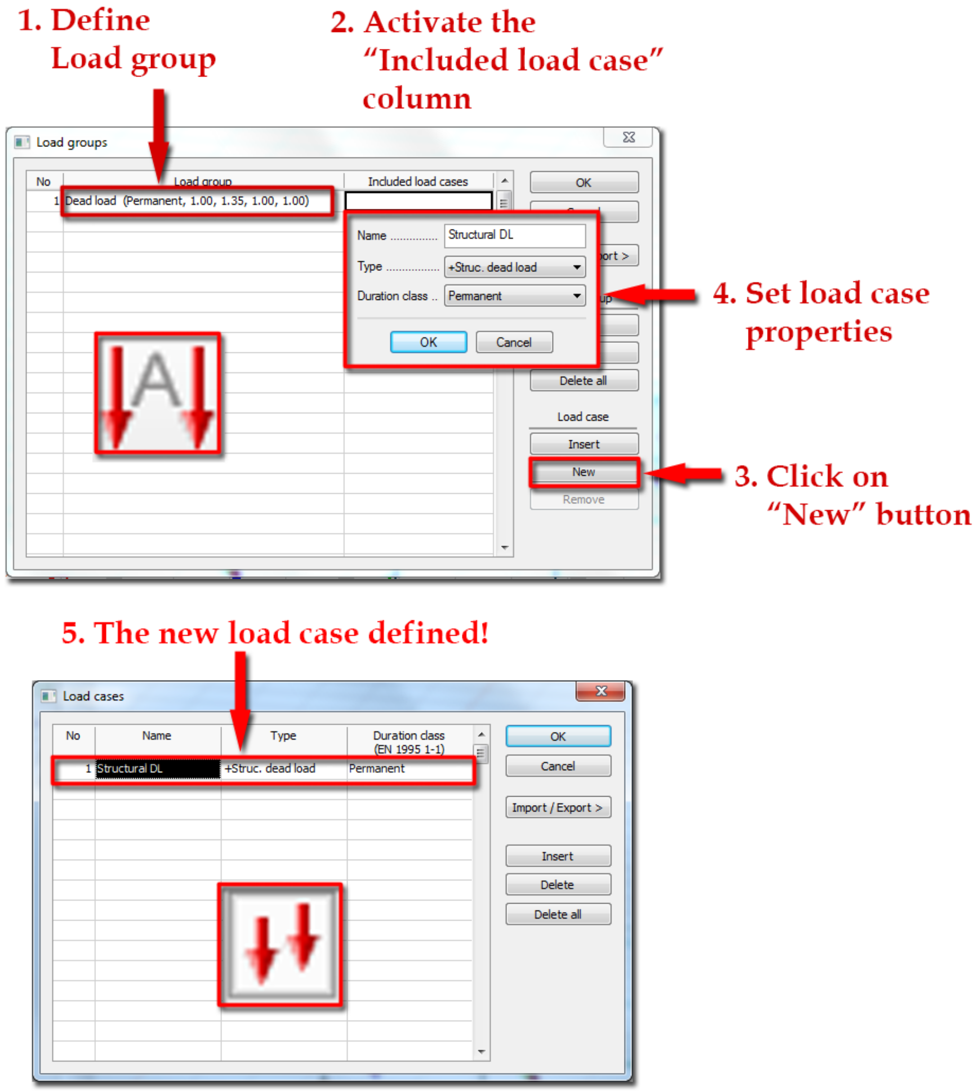


Figure: Load case definition in Load groups dialog

- **Remove**

  This function removes the selected load case from the current load group.

Definition steps of new load groups

1. Click on an empty *Load group* cell. Define the name of the new load group and set its type. Set the required factors of the group according to the selected type and the applied standard.
2. Add a load case to the group by selecting it from the *Included load cases* drop-down list. To add more than one cases to a group, use the *Insert case(s)* option.

   ⚠ "*Permanent*"-type load groups defined will be present in all load combinations.

   If more than one load cases are assigned to a load group, they will never be simultaneously present! So, it is recommended to set permanent-type load cases (e.g. automatic dead-load, roof weight etc.) in own "*Permanent*"-type groups to avoid the loss one of them.

3. Define the next load group by repeating the previous steps.

Optional steps

4. A load group can be edited by double clicking on its *Load group* cell.
5. The load group list can be set as the default group set for the next and later projects by clicking on *Save as default* option.

Let's see an example, how partial loading of slab can be modeled with load groups. Just place the load cases represented the loading statuses into one "*Temporary*"-type load group.

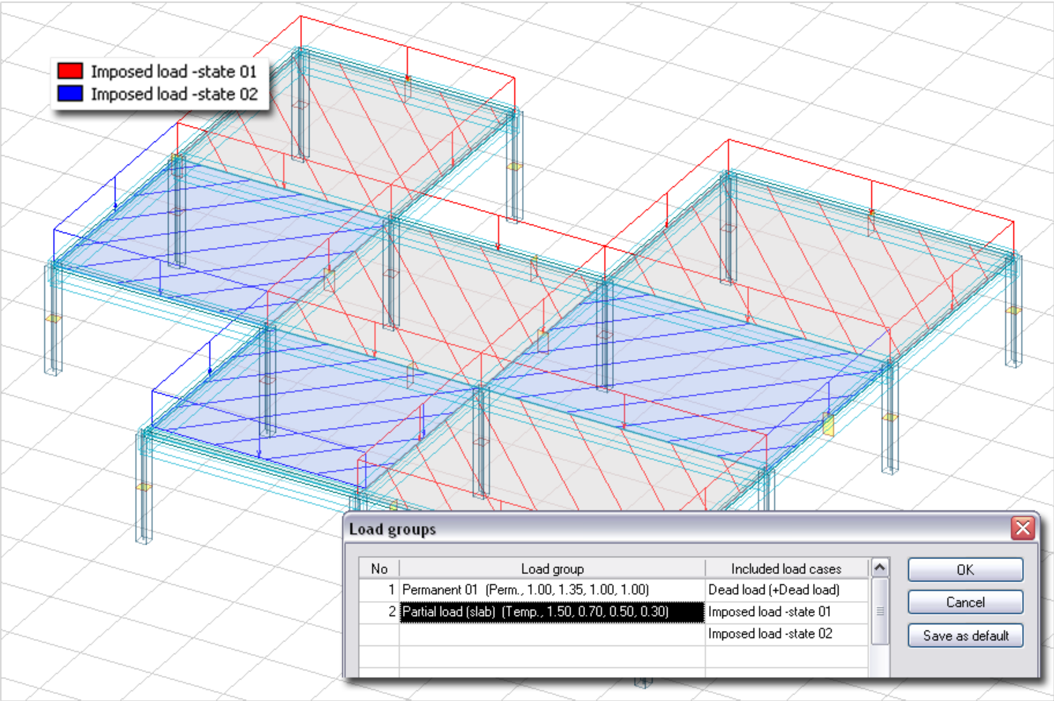


Figure: Partial loading of a slab

Load Combination

Load combination lets you combine load cases by multiplying them with given load factors.

Two main types of load combinations can be defined: load combinations for **ultimate (U)** and for **serviceability (S) limit state**. Although **Analysis** calculations can be done for both types, the recommended functions of the types are the followings:

- **Ultimate limit state-type load combination (U)**
  Define U-type load combinations for strength and stability calculations. All design calculations (except for crack width of RC Design) are done only for U combinations.

- **Ultimate limit state-type (accidental) load combination (Ua)**
  This type of load combination is like Ultimate limit state, but the difference is in the safety factors.

- **Ultimate limit state-type (seismic) load combination (Us)**
  This type of load combination is like Ultimate limit state, but the difference is in the safety factors.

- **Serviceability limit state-type load combination, characteristic (Sc)**
  Define Sc-type load combinations to calculate displacement.

- **Serviceability limit state-type load combination, frequent (Sf)**

This type of load combination is like Serviceability (characteristic) limit state, but the difference is in the safety factors.

- **Serviceability limit state-type load combination, quasi permanent (Sq)**
  Define Sq-type load combinations to calculate RC bars and slab crack width calculation.

For the different design calculations different SLS load combinations are used:

- Deflection check: user selectable
- Foundation settlement: user selectable
- Crack width: Quasi-permanent (S_q)

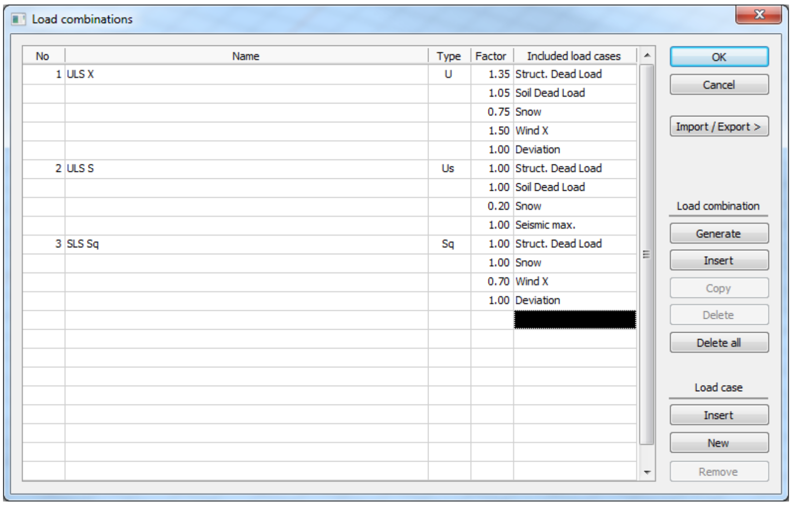


Figure: Load combination definition

Definition steps of new load combinations

1. Type the name of the new combination in the *Name* cell.
2. Set the load combination type by choosing Ultimate (U, Ua, Us) or Serviceability (Sc, Sq, Sf) from the *Type* drop-down list.
3. Select a load case that you would like to add to the combination from the *Included load cases* drop-down list, which contains all load cases predefined in the current project.
4. Type a load factor for the load case chosen in the previous step in the *Factor* cell.
5. Repeat the 3rd and 4th step in the following rows, if you would like to add more than one load cases to the combination.

   If you would like to add more than one load cases to the current load combination with the same factor in one step, use the *Insert* option of the *Load combinations* group.

6. Define the next load combination by repeating the previous steps.

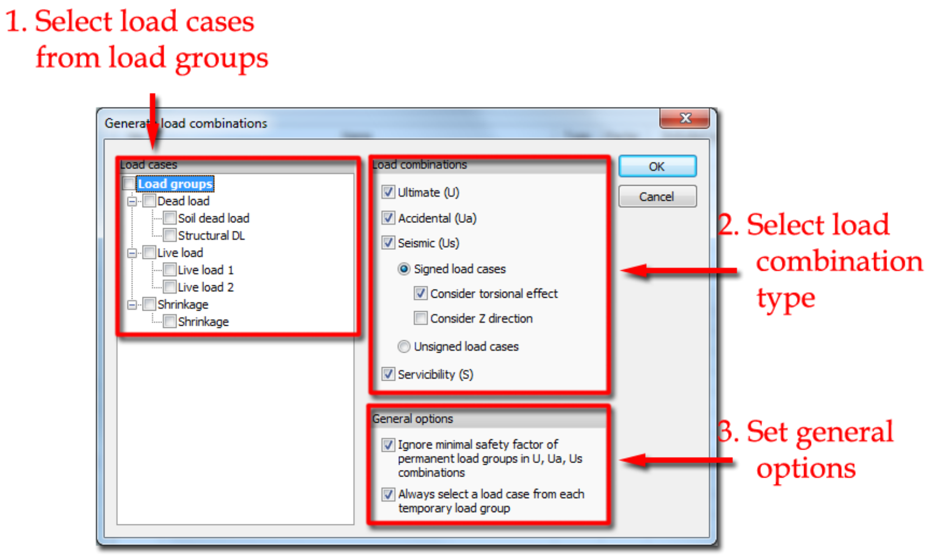
   If you would like to define a load combination with a similar content (load cases with factors) of another one, apply the *Copy comb.* option for the source load combination and define the destination combination by defining a new load combination. *Copy comb.* option can also add a load combination content to another predefined one.

Optional steps

7. Load combination can be renamed by typing a new name in its proper *Name* cell.

8. Load combinations can automatically be generated from the Load cases assigned to the Load groups, according to Eurocode 1990-Chapter 6.4.3.

- In Load combination dialog click Generate button.



- Select load cases from the Load groups.
- Select Load combination type to generate and their properties (if there is any).

⚠ The selected Load combinations will be generated if there is at least one load group of that type. E.g. if the user wants to generate Accidental load combination, but there is no Accidental Load group, or in that Load group there is no Load case, the program will not create any accidental Load combination.

- Set the General options parameters.

9. Another way to manage Load combination(s) and their Load case(s) are listed below:

Load combinations:

- **Insert**
  With this function the user can add a new Load combination. After clicking on Insert button, the parameters can be set in the dialog.

- **Copy**
  With this function the user can copy an existing combination with load cases and factors.

- **Delete**
  The user can delete a selected load combination.

- **Delete all**
  All the load cases can be deleted with Delete all function.

Load cases:

- **Insert:**
  An existing load case can be inserted to a load group with Insert function.

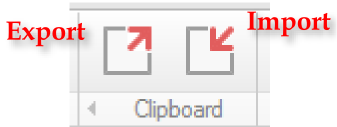
- **New:**

The user has the opportunity to define load cases in Load combinations dialog. After defining a load case in this dialog, the new load case will be added to Load case list (the user can see in Load cases dialog).

- **Remove:**
  This function removes the selected load case from the current load group.

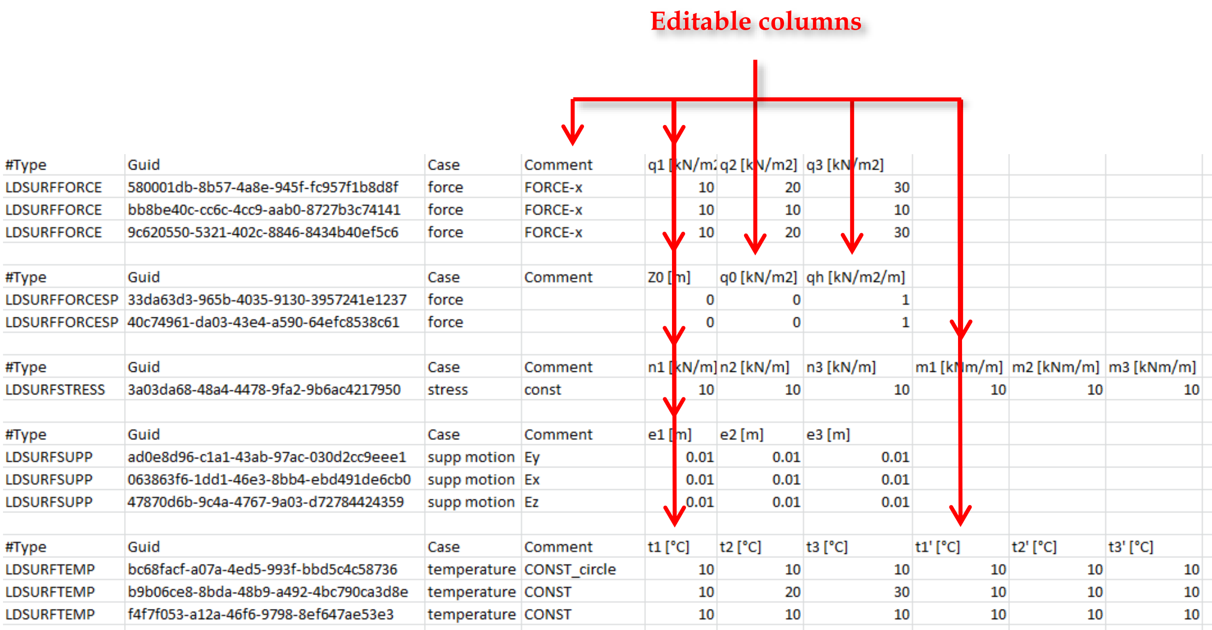
Load Export/Import via clipboard

Load Export and *Import via clipboard* lets the user easily and quickly modify loads.



In order to export loads, User has to click to *Export* to send the load information to the clipboard. Then User can paste to Excel or any editor program and modify them.

⚠ Only comments and the load intensities can be modified. We suggest NOT to edit other columns to avoid errors in Importing.



#Type	Guid	Case	Comment	q1 [kN/m2]	q2 [kN/m2]	q3 [kN/m2]			
LDSURFFORCE	580001db-8b57-4a8e-945f-fc957f1b8d8f	force	FORCE-x	10	20	30			
LDSURFFORCE	bb8be40c-cc6c-4cc9-aab0-8727b3c74141	force	FORCE-x	10	10	10			
LDSURFFORCE	9c620550-5321-402c-8846-8434b40ef5c6	force	FORCE-x	10	20	30			
#Type	Guid	Case	Comment	Z0 [m]	q0 [kN/m2]	qh [kN/m2/m]			
LDSURFFORCESP	33da63d3-965b-4035-9130-3957241e1237	force		0	0	1			
LDSURFFORCESP	40c74961-da03-43e4-a590-64efc8538c61	force		0	0	1			
#Type	Guid	Case	Comment	n1 [kN/m]	n2 [kN/m]	n3 [kN/m]	m1 [kNm/m]	m2 [kNm/m]	m3 [kNm/m]
LDSURFSTRESS	3a03da68-48a4-4478-9fa2-9b6ac4217950	stress	const	10	10	10	10	10	10
#Type	Guid	Case	Comment	e1 [m]	e2 [m]	e3 [m]			
LDSURFSUPP	ad0e8d96-c1a1-43ab-97ac-030d2cc9eee1	supp motion	Ey	0.01	0.01	0.01			
LDSURFSUPP	063863f6-1dd1-46e3-8bb4-ebd491de6cb0	supp motion	Ex	0.01	0.01	0.01			
LDSURFSUPP	47870d6b-9c4a-4767-9a03-d72784424359	supp motion	Ez	0.01	0.01	0.01			
#Type	Guid	Case	Comment	t1 [°C]	t2 [°C]	t3 [°C]	t1' [°C]	t2' [°C]	t3' [°C]
LDSURFTEMP	bc68facf-a07a-4ed5-993f-bbd5c4c58736	temperature	CONST_circle	10	10	10	10	10	10
LDSURFTEMP	b9b06ce8-8bda-48b9-a492-4bc790ca3d8e	temperature	CONST	10	20	30	10	10	10
LDSURFTEMP	f4f7f053-a12a-46f6-9798-8ef647ae53e3	temperature	CONST	10	10	10	10	10	10

After changing attributes, User can choose whether to import some. or all of the loads by selecting the desired rows and copying them to clipboard, then in FEM-Design clicking on *Import*.

⚠ If the User exported constant surface load, only changing the first intensity value will have effect on the surface load.

Model and Data Connections

There are numerous ways to get draft and initial model geometry for later design in FEM-Design.

With the so-called *Wizard* tool, typical frame and roof structures can be modeled quickly by giving the parameters of the required geometry, support and load, and the program automatically generates the real 3D model.

Models can be shared between the 2D and the 3D modeling of *FEM-Design modules*. For example, a slab of a multi-storey building (modeled in *3D Structure* module) can be design separately from the other slabs in the *Plate* module.

FEM-Design supports the major CAD drawing exchange formats (*DWG/DXF*), so imported drawings can be used as referenced draft for building up the structural models.

FEM-Design is able to import 3D models from other structural and architectural programs in two ways:

- FEM-Design supports the import of the most popular 3D model exchange data format called *IFC* (*Industry Foundation Classes*). The model-based data import gives the possibility to use the models designed by other disciplines (such as architectural) as an initial model or to show them as a reference.
- StruSoft develops *direct 3D model exchange links* with ArchiCAD (architectural discipline) and the major structural applications: Tekla Structures and Revit Structure. Add-ons build in these applications export native FEM-Design elements generated from 3D models.

Frame Wizard

Wizard helps you to model typical frame and roof structures with *beam* elements, *point supports* and *line loads*. With geometry parameters, you can define one frame/roof stand (planar model) or a multi-frame/roof system with 3D extension. Line loads such as automatic and additional dead-loads, snow, wind and live load can be added to the beam components.

 *Wizard* is available in the  *3D Structure* and the  *3D Frame* modules.

Definition steps

1. Start the *Wizard* command from the *Tool* menu.

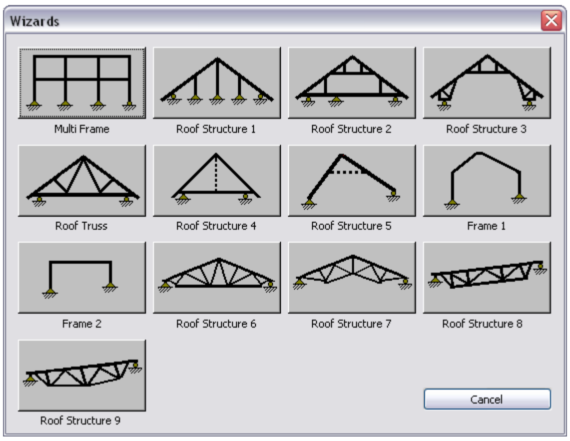


Figure: Typical frame and roof structures as the range of Wizard

2. Choose the required frame/root type in the *Wizards* dialog box.
3. According to the selected structure type, set the geometry parameters. Click *Preview* to check the defined geometry.

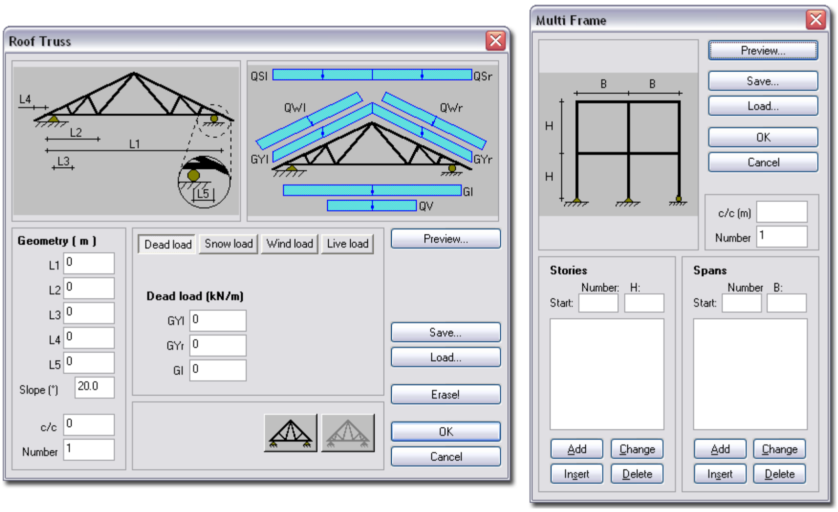


Figure: Settings dialogs

Meanings of some main parameters:

Number: Number of the frame standings

c/c: Distance between the frame standings

4. If load definition is available for the selected type, define the additional dead load (roof weight), snow, wind and live load based on the figure shows the meaning and position of line loads (intensity).

Save option (of *Wizard*) lets you save all parameters of the current wizard template. The exported wizard settings can be imported for later project with the *Wizard*'s *Load* tool.

Defining "Dead load" (additional, roof weight) line load generates the *automatic dead load* too as a separate *load case*.

5. Clicking *OK* finishes the *Wizard* and generates the 3D model with loads and supports (depending on the selected type).

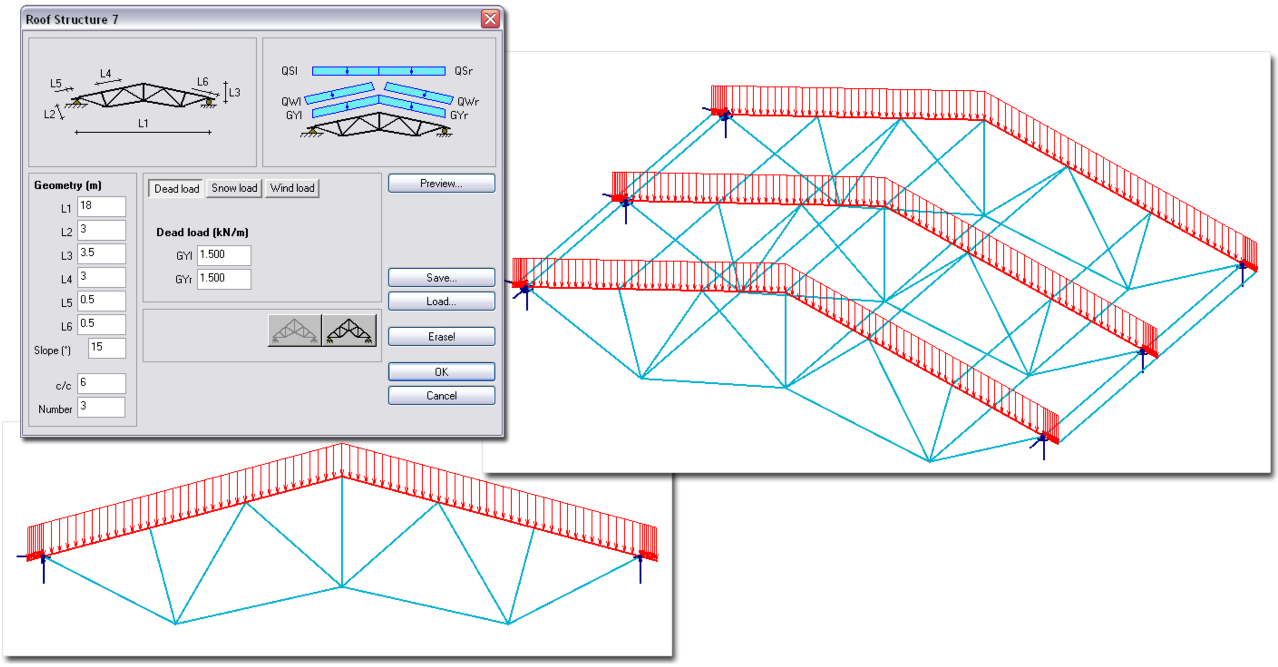


Figure: Roof structure example

6. The generated beams have no profiles and materials. So, *Wizard* generates only beam axes without properties. Add properties to beams with the *Beam* command's *Properties* tool.
7. Wizard sets up hinged and rigid supports (as displayed by wizard settings) from single support components. So, for example a fully rigid point support is modeled by 3 motion-type and 3 rotation-type single point supports. You can modify the support conditions by the *Point support* command's *Properties* tool.

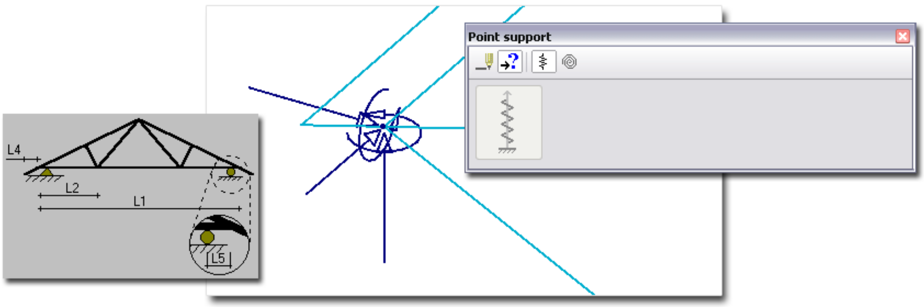


Figure: Complex (rigid) support modeled by single Point supports

Optional steps:

8. Add more *structural elements* (plate, wall, beam, column etc.) with the commands of the Structure tabmenu.
9. Edit the generated loads or add new ones with the commands of the Loads tabmenu.

Model Exchange between FEM-Design Modules

Different FEM-Design modules are able to share models between each other. The next table shows the possible connections between modules.

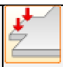
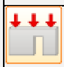
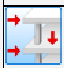
		Source Module (.file extension)				
		2D models		3D models		
		(.pla)	(.wal)	(.frm)	(.str)	(.prd)
Destination Module		✓			✓ (1)	✓ (1)
			✓		✓ (2)	✓ (2)
				✓	✓ (4)	
		✓ (3.1)	✓ (3.2)	✓ (5)	✓	
		✓ (3.1)	✓ (3.2)	✓ (5)	✓	✓

Table: Model connections between FEM-Design modules

The main featured connections from the above possibilities:

- **Slab-system imported from 3D Structure/PreDesign to Plate module (1)**

  Slabs or slab-system of a building (created by *3D Structure* or *PreDesign*) *storey* together with connected walls, columns, beams and loads can be imported into *Plate* module.

  ⚠ Only the loads directly placed on the slab elements can be imported to *Plate*. The loads arrive from other stories or connected elements have to be added manually to the slab regions as *Point*/*Line*/*Surface loads*.

  Steps of import:

  1. Click *Open* (*File* menu) in the *Plate* module. Select the *.str* (*3D Structure*) or .prd (*PreDesign*) which contents you would like to import to *Plate*.
  2. Select the *Storey*, which slabs (plates) you would like to load as a new project.

     ⚠ The import requires *storey-system* from the file selected by *Open*.

  3. Allow or forbid the import of columns and walls connected above/below the slabs of the selected storey. Beams and loads connected to related slabs will be automatically imported.
  4. Set the end conditions (fixed or hinged) of the walls (=wall supports) which you would like to import.
  5. Click *OK* to start the import.

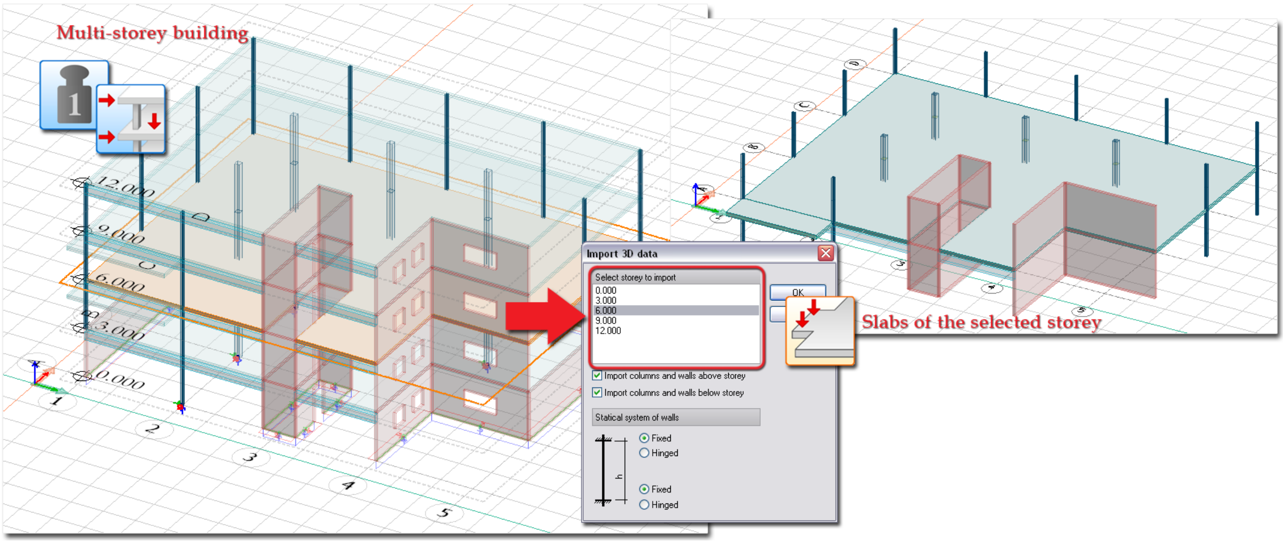


Figure: Import of slabs with connected elements into a Plate project

- **Wall-system imported from 3D Structure/PreDesign to Wall module (2)**

  Wall regions situate on a vertical plane defined by an *axis* of a building (created by *3D Structure* or *PreDesign*) can be imported into *Wall* module together with connected supports and loads.

  ⚠ Only the loads directly placed on the wall elements can be imported to *Wall*. The loads arrive from other connected elements have to be added manually to the wall regions as *Point*/*Line loads*.

  Steps of import:

  1. Click *Open* (*File* menu) in the *Wall* module. Select the *.str* (*3D Structure*) or .prd (*PreDesign)* which contents you would like to import to *Wall*.
  2. Select the *Axis*, which wall regions you would like to load as a new project.

     ⚠ The import requires *axis-system* from the file selected by *Open*.

  3. Click *OK* to start the import.

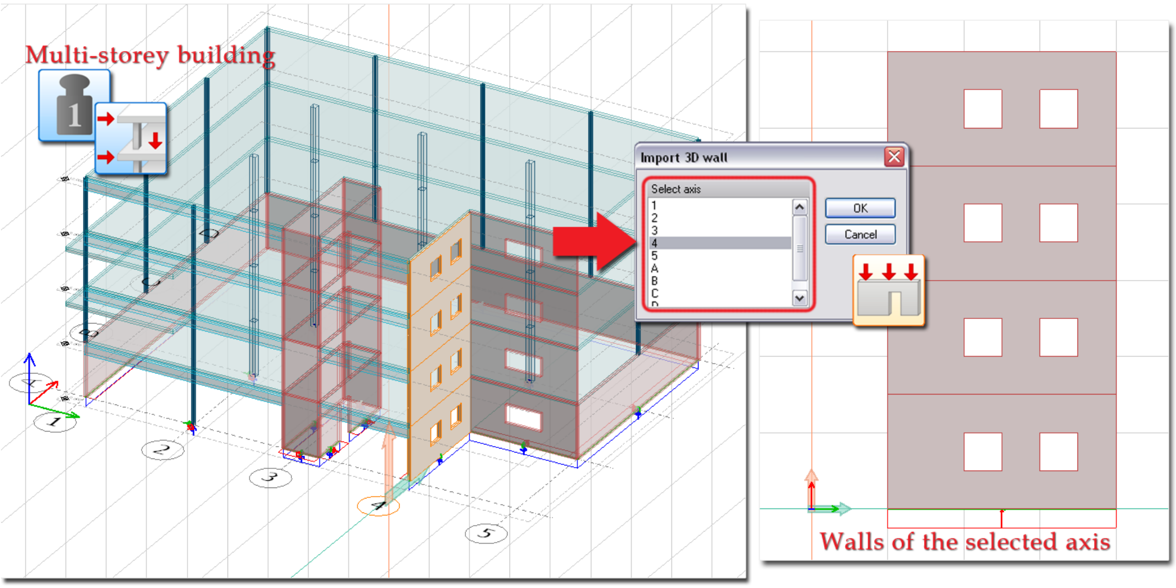


Figure: Import of walls with connected elements into a Wall project

- **Single plate/wall import to 3D Structure (3.1 and 3.2)**

  Slabs/Walls with their supports and loads created and designed in the *Plate* / *Wall* module can be imported into the *3D Structure* module. Multi-storey building can be defined by copying the opened elements, and run global stability analysis and design for the final model.

  Steps of import:

  1. Click *Open* (*File* menu) in the *3D Structure* module. Select the *.pla* (*Plate*) or *.wal* (*Wall*) which contents you would like to import to *Plate*.
  2. In case of curved walls, set their conversion to planar wall regions. Curved wall element is not available in the 3D structure modules. The value of the approximation can be set in the *tmax* field. If the "originally curved" walls are connected to curved plate edges, the program also modifies the curved plate edges to polyline.
  3. Click *OK* to start the import.

  Optional steps:

  4. Copy (*Modify* > *Copy*) plate regions with their wall/column supports and loads, if required.
  5. Only one *Plate/Wall* file can be imported (with *Open*) in a 3D project. But, you can open different files in separate 3D projects, and later you can merge their contents into one with the *Copy* and *Paste* commands and set their real position with the *Move* command (*Edit* menu).
  6. In *Plate* module, the *Wall* and *Column* elements are supports. So, do not forget to add supports to the bottom end of the columns and walls of the lowest storey.

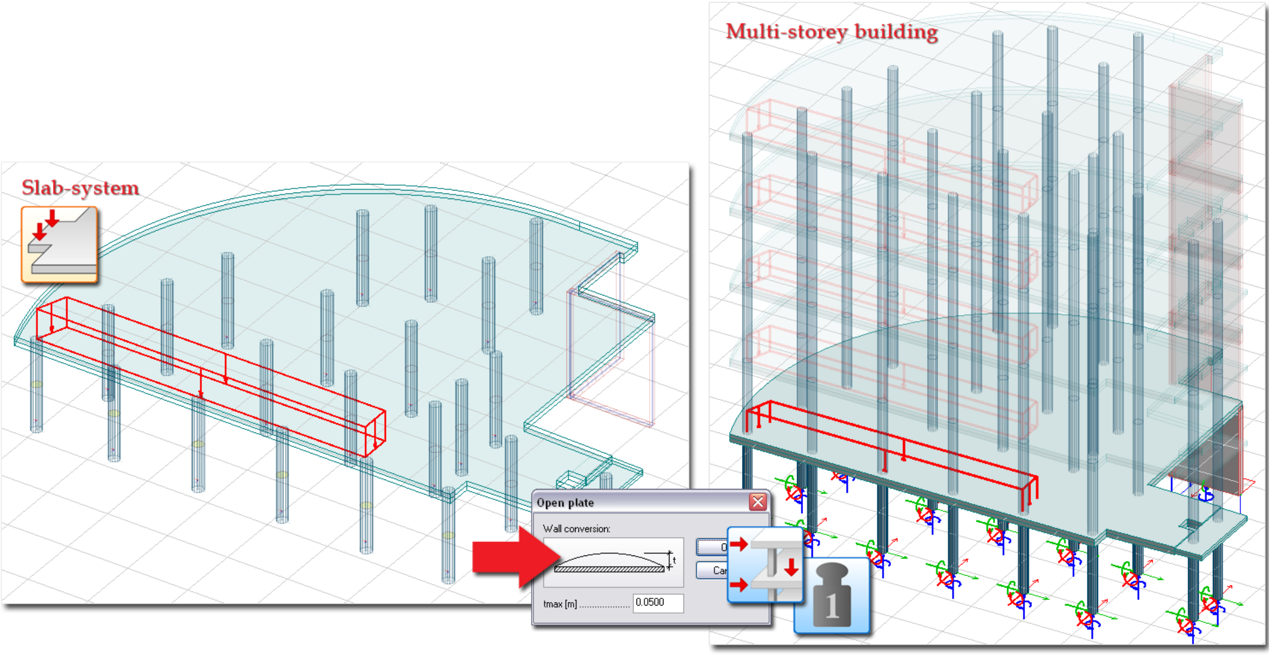


Figure: Import of a slab (system) into a 3D structure project

- **Frame part exported from 3D Structure to 3D Frame (4)**

  Frame parts (beams, columns and truss members) of a multi-storey building (created in *3D Structure*) can be imported into the *3D Frame* module by filtering out the shell, plate, wall elements and their related surface supports and loads. The import process is very simple: just apply *Open* (*File* menu) for the *.str* (*3D Structure*) files in *3D Frame*.

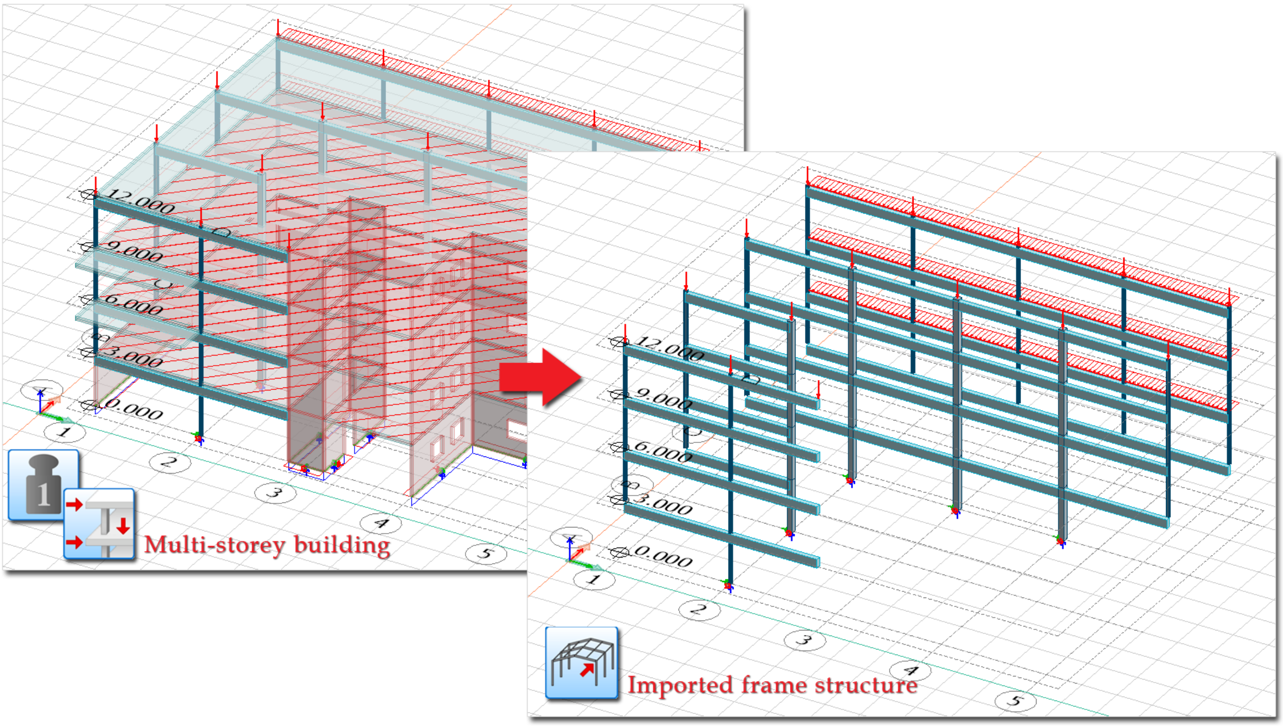


Figure: Import of frame parts into the 3D Frame module

- **Frame import to 3D Structure (5)**

  Initial analysis and design can be done for only the load-bearing frame parts in the *3D Frame* module, then the bar elements can be imported into the *3D Structure* modules to complete them with planar element such as plates, walls and their supports and loads. The import process is very simple: just apply *Open* (*File* menu) for the *.frm* (*3D Frame*) files in *PreDesign* or *3D Structure*.

⚠ Opening PreDesign model in 3D Structure and 3D Frame is not allowed since version 12.0, as PreDesigh has totally different database and not possible to keep compatibility with other 3D modules.

CAD Drawing Import/Export

A DWG or DXF file generally contains drawing objects (points, lines, dimensions, texts etc.), so it can be used as a reference draft displaying a floor plan, a section or an elevation. Each *FEM-Design module* is able to import (and also export) DWG and DXF.

All elements, so both structural model and drawing elements, can be converted and exported to DWG/DXF format. The objects are automatically converted to lines with their 3D position and placed onto separate layers by object types. Also, the symbols and info labels are broken to text and line parts.

Import

The steps of opening a DWG/DXF file:

1. Open the DWG/DXF file with *File* > *Open*.
2. Set the import settings in the upcoming dialog:

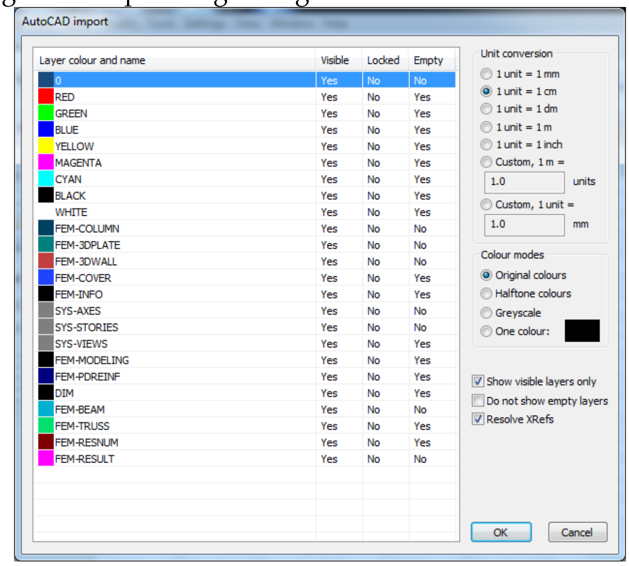


Figure: DWG import settings

- *Unit*: Set the same drawing length unit with the unit-system saved in the file by the exporting application. Most of the DWG-compatible applications use "mm" while defining and exporting drawings.
- *Scale*: Set the drawing scale of the opened file by typing the required value. Scale affects the display of texts, line types and hatches.
- *Ignore points*: Check this box to filter out drawing points, so not to allow their import.
- *Set to 2D*: Check this box, if you would like to merge all drawing elements (independently their level position) in the XY plane of the *Global co-ordinate system*. It is recommended only,

if the points of the imported drawing place in the same or near the same (small geometry inaccuracy in Z direction) plane.

- *Ignore empty layers*: Check this box not to import empty layers that do not contain drawing elements. Unchecked box merges all layers stored in the DWG/DXF file.

  The imported layers will be converted to *Drawing layers*.

- *Merge points*: Checking this box corrects the inaccuracies of the imported drawing in the given merging distance and deletes the point duplications.

3. Hide the unnecessary drawing layers to make clear the points and lines you would like to use for later model definition.

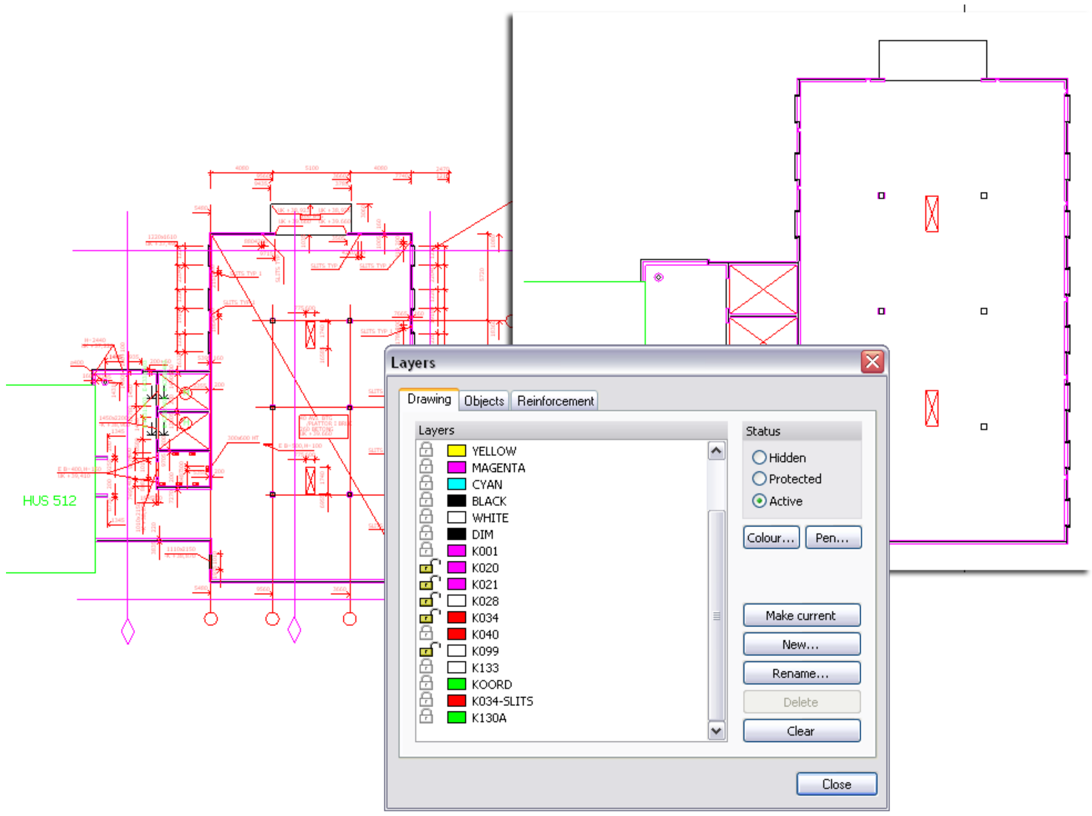


Figure: Active imported drawing layers

With *Modify* > *Change appearance* you can query the host drawing layer of a selected element.

4. Create the structural model based on the referenced (opened) drawing.

Apply the proper *Object Snap Tools* to find the required main points, endpoints, intersections, lines etc. for the input (reference points and lines) of the *model elements*.

Set the size of the elements according to the drawing elements after inquiring/measuring positions, directions and sizes with *Tools* > *Query*, if it is needed.

Pay attention to pick proper/correct lines when defining reference lines and insertion points for the structural objects. For example, think on the wall position while connecting it to a plate edge. Not always following the real state (defined by the DWG drawing) is the optimized way to model elements.

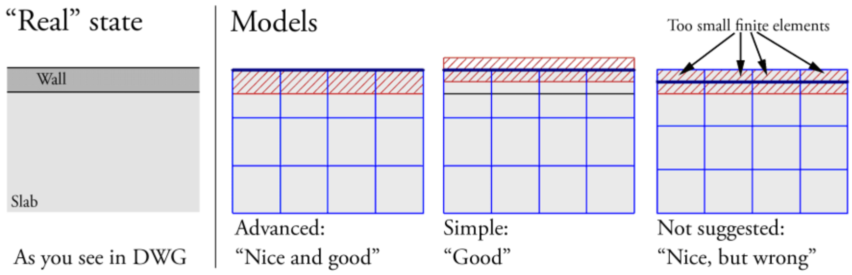


Figure: Reference line (and finite element mesh) definition based on the drawing

Rotate the position of the drawing, if it is needed. For example set a section/elevation drawing to the *Global XZ* or *YZ* plane in 3D modules.

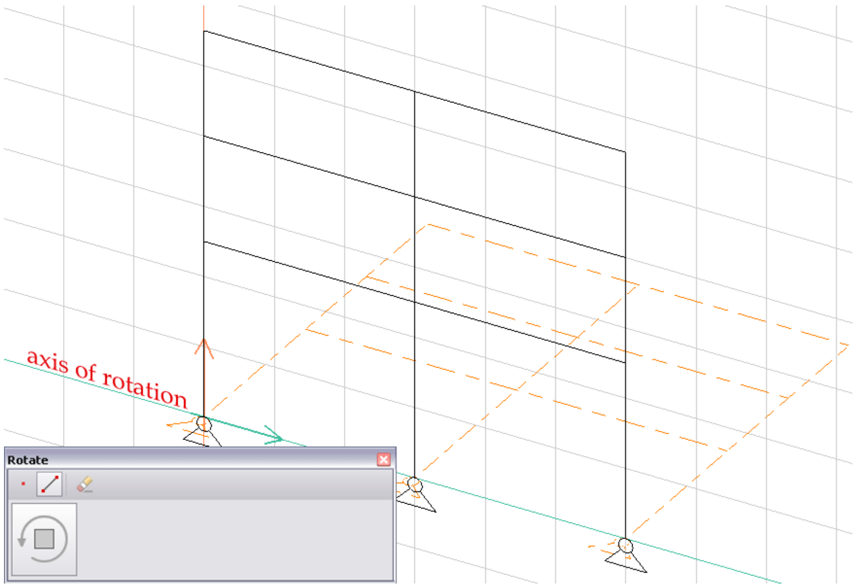


Figure: Rotation of a drawing (definition of an elevation)

In most cases the drawing elements of the merged DWG/DXF file are placed far from the origin of the *global co-ordinate system*, so click *View* > *Zoom margin* to show the entire drawing after opening it.

Only one DWG/DXF file can be opened at a time. But, you can merge more than one drawing (e.g. floor plans of different stories) into a project as the following hint:

1. Open the first drawing as a new FEM-Design project.
2. Open the second/other drawing as a separate FEM-Design project. (More than one copies of a FEM-Design module can run in the same time.
3. Apply *Edit* > *Copy* for the required elements of the second/other drawing to send them to Clipboard.
4. Use *Edit* > *Paste* to merge the drawing elements (from the Clipboard) into the first project. Set the real position of the drawing insertion (point), so for example, place the 2nd floor elements in the correct 2nd level position.

Export

The steps of saving a DWG/DXF file:

1. Click *File* > *Save as*, and choose the proper DWG/DXF file.

2. Set the export settings in the upcoming dialog:

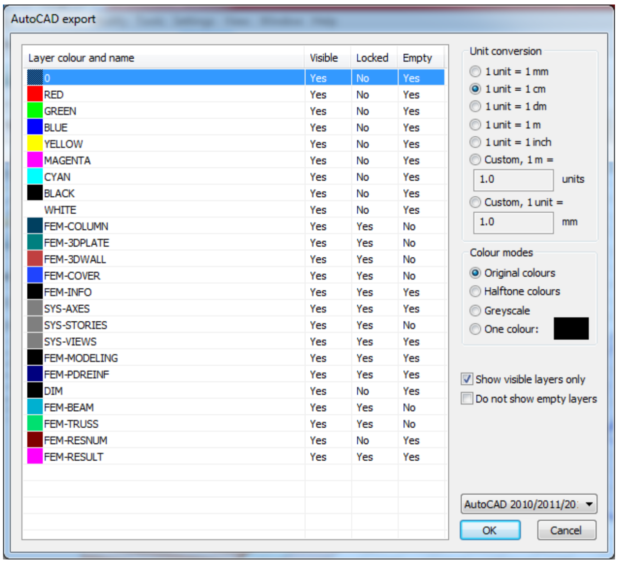


Figure: DWG export settings

- *Unit*: Set the drawing length unit of the exported elements to the type required by the other application which wants to open the DWG/DXF file.
- *Arc resolution*: Curved elements, arcs and circles will be exported as connected straight lines (polyline). The accuracy of the line-conversion approximation can be set here. Arc resolution is equal the angle closed by the tangent of the connected lines. This setting effects the number of the generated lines too.
- *Convert text*: The option converts the special Unicode characters (such as ő, ű etc.) to readable ANSI-compatible characters (such as o, u etc.). So, this option is recommended to be used in case of text, labels etc. Non-used option may cause unreadable symbols in the application opens the exported DWG/DXF file.

All elements (beams, columns… supports, loads, drawing elements, labels, etc.) will be generated to lines and texts, and they will be placed onto separate drawing layers by their types. All DWG/DXF-compatible applications are able to filter elements by layers during or after the import process.

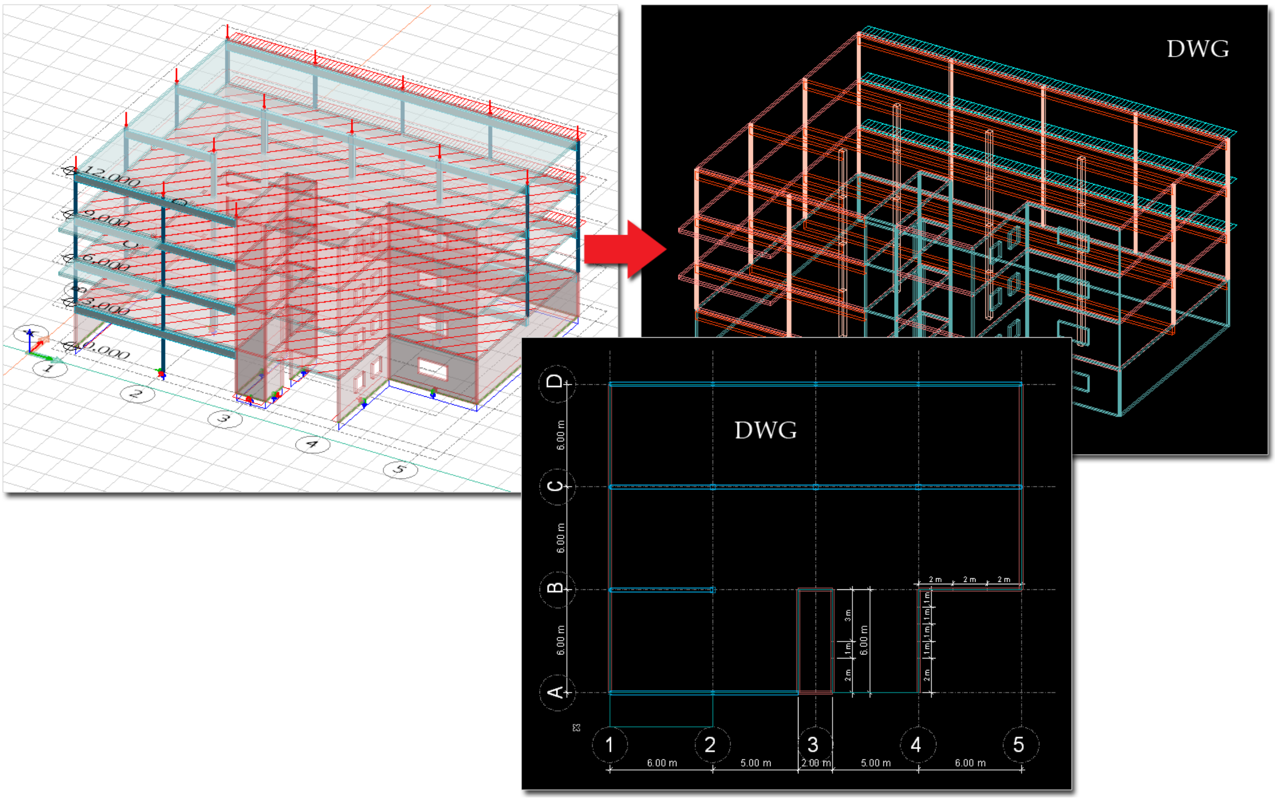
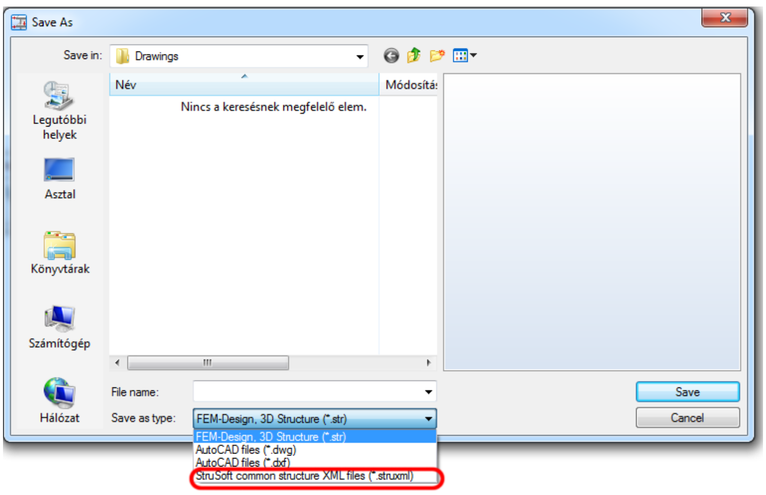


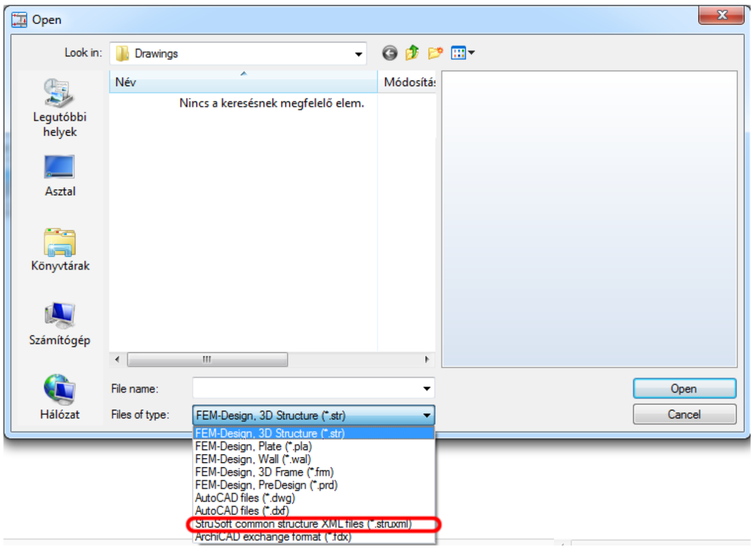
Figure: Exported DWG in 2D and 3D views

Strusoft XML

Models can be imported or exported between structural engineering programs (e.g. Tekla, Revit) in a special Strusoft XML format (struxml). Exporting from and importing to FEM-Design 3D Structure module by struxml format can be done by *Save as* or *Open* commands respectively.

⚠ Only basic structural elements and loads can be exported from and imported to FEM-Design in struxml format.





IFC Import

FEM-Design gives the possibility to import 3D models saved in IFC files from other architectural and structural programs. Some of the most important features of the FEM-Design's IFC import:

- Compatible import formats: IFC2x version 2 and 3
- Material conversion table and style
- Object-filtering
- Connections of misplaced objects (point-point and line-line connection)

An IFC model contains elements according to the mapping rule of model elements to IFC element types. For example, a column is represented as *IfcColumn* in the IFC scheme by default. Not only the geometry, but numerous properties like material, profiles etc. are assigned to one element.

Opening an IFC file generates native FEM-Design elements based on the model structure or solid body model ("reference model") depending on the chosen import method:

- **Architectural View**
  The static system is converted from the architectural/structural model elements (IFC architectural representation), so *FEM-Design* creates native structural objects such as *Plates*, *Walls*, *Beams* and *Columns*.

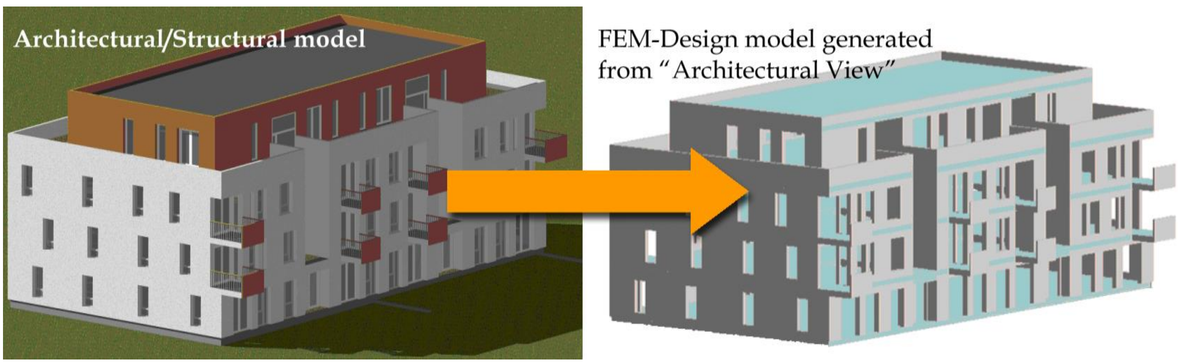


Figure: Generation of real FEM-Design objects

- **Reference Geometry View**

A solid body model is generated from the IFC model as a reference. Elements are not supported by the current FEM-Design can be also display because of the solid representation. For example, solids generated from slabs and walls can be shown as reference in the *3D Frame* module, although only bar elements can be defined in it.

After the import, FEM-Design can find and select points, edges and surfaces of the solid body model elements, so the user can define *Plates*, *Walls*, *Beams*, *Columns*and *Truss members* manually according to the referenced model. For example, to define a wall object in FEM-Design 3D modules, just pick on the requested vertical solid surface.

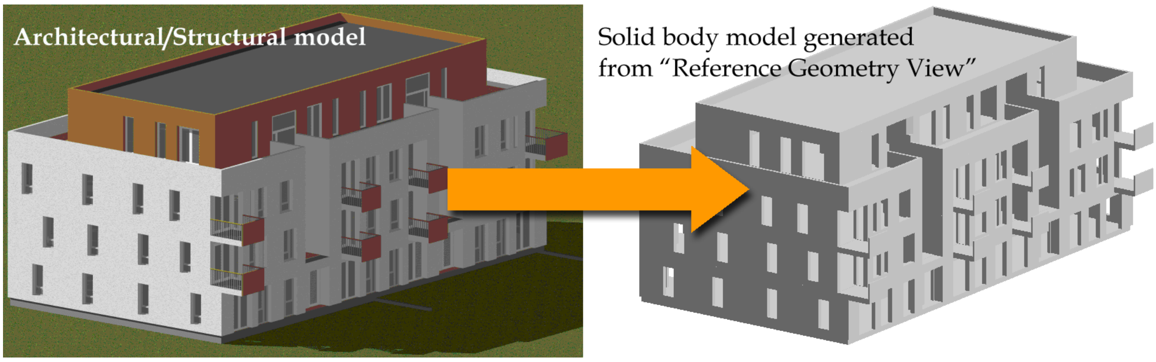


Figure: Generation of 3D solid bodies (drawing)

- **Structural Analysis View**
  Some structural application (e.g. Tekla Structures) is able to create analysis model and export it via IFC. Opening an analysis type IFC model, which contains region and line representation of structural elements generates native FEM-Design elements very fast.

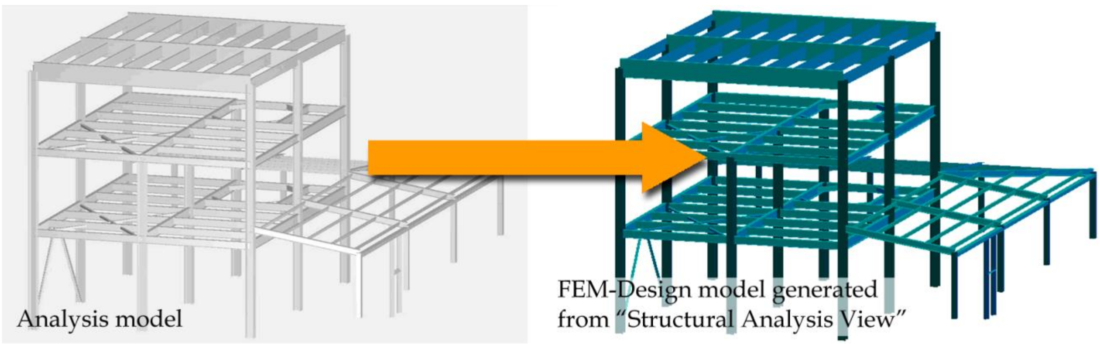


Figure: Generation of real FEM-Design objects from an imported analysis model

Of course, the result of element import depends on the FEM-Design module where the process is done:

Module	Architectural View	Reference Geometry View	Structural Analysis View
Plate	One storey only: - Plates and Beams - Walls and Columns as supports	Entire model and all building elements as Solid objects	-
3D Frame	Beams and Columns only	Entire model and all building elements as Solid objects	Beams and Columns only

3D Structure PreDesign	Plates, Walls, Beams and Columns	Entire model and all building elements as Solid objects	Plates, Walls, Beams and Columns

Table: Imported IFC elements by "views" and FEM-Design modules

The steps of opening an IFC file:

1. Load the IFC file with *File> Open*.
2. Choose the required import method: *Architectural View*, *Reference Geometry View* or *Structural Analysis View*. The first two methods are always available, but the last one depends on the received IFC model type ("Analysis-type").

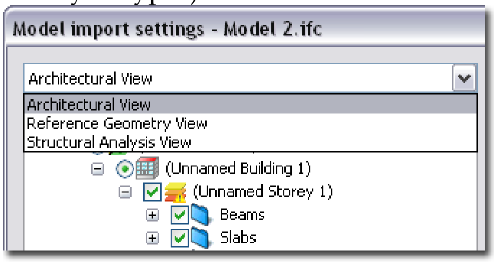


Figure: Selection of import mode (view)

3. A tree lists the IFC model content by buildings and their storeys. In the *Architectural* and *Reference Geometry* views, objects can be selected for the import.
   In *Architectural View* mode, "Irregular" label represents the element types/geometries, which cannot be converted to FEM-Design structural elements. However these elements can be displayed in the *Reference Geometry View*.

Plate module only imports plates situate on the same storey together with supports under them. Most of the architectural software places the "support elements" (like walls, columns etc.) on one storey below the plates' host storey, so the requested elements can be selected from separate storeys. In case of *3D Structure* models, the entire model can be imported independently from the storey system.

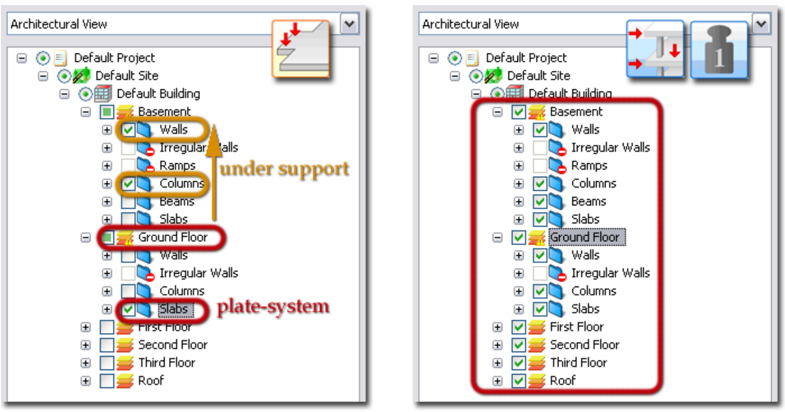


Plate-system with under supports — Entire 3D model

Figure: Differences between modules used for the model import

4. Set the main properties of the selected elements (only in *Architectural* and *Structural Analysis views*).

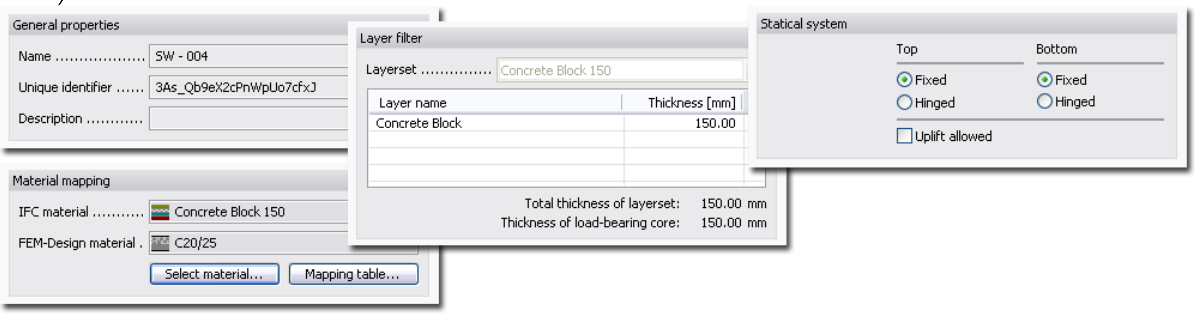


Figure: Main properties of elements

- *General properties*: it displays the ID name of the elements and the IFC global unique identifier (GUID). Description may contain the profile name of the elements depending on the exporter application.
- *Material mapping*: it displays the material of the selected element (IFC material) and the default FEM-Design material (available in the current standard) assigned to the imported IFC material. Of course, you can modify and set the required material that will model the imported one. Material can be set with *Select material* by elements or based on the material *Mapping table.* Of course, the material can be modified later with the element's *Properties* tool.

  The conversion can be done on different model levels, so for the entire structure, by storeys (e.g. 1st floor), by object type (e.g. all slabs), and for the type (e.g. slabs) of the selected objects.

  - **Select material**

    A dialog appears with the built-in (available in the current standard) and user-defined concrete, steel and timber materials of FEM-Design. New concrete, steel and general materials can be created in the dialog too.

  - **Mapping table**

    You can associate standard materials (as the applied code) to one or more materials of the imported model (= "FC Material") in dialogue format. It works for the complete model, so there is no difference that you select an object or a storey or a building. There are two independent conversion maps, one for the architectural and one for the structural model.

    Select one (or more) material in the "IFC Material" column, then press *Assign material* and finally choose proper FEM-Design material available in the current code. Here, new materials can also be created (*New*). Clicking *OK* returns to the material conversion table that shows the selected conversion: the selected material appears in the "FEM-Design Material" column. Material mapping table can be saved and exported for others with the *Export* option, or previously saved conversion rules can be loaded with *Import*.

- *Layer filter*: the core of the composite layered materials can be set here.
- *Statical system*: in *Plate* module, the end connections (fixed or hinged) of selected columns and walls (supports) can be set as a default value. Of course, the end connections can be modified later with the element's *Properties* tool.

5. Clicking *OK* loads the *IFC* model. The loading process may take a long time depending on the complexity and the size of the imported model.

If you open an IFC model in *Architectural view* mode with the 3D modules, the program generates rigid *connection objects* (*Point-point* and/or *Line-line connection*) between the misplaced elements, which solid surfaces are touching each other, but the generated calculation planes are not. The method depends on the types of the "connected" objects:

- If a bar element connects to an object, the program applies *Point-point connection*.
- If only planar elements connect to each other, *Line-line connections* are assigned to them.

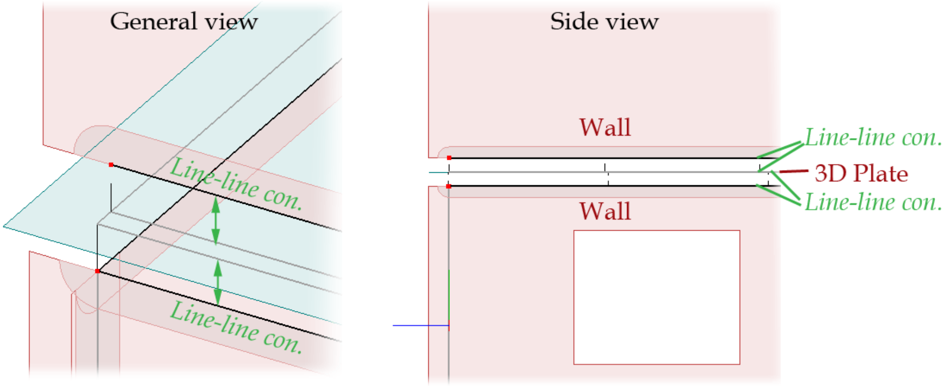


Figure: Line-line connection generated between the plate and the touching wall elements

You can modify the connection stiffness with the *Properties* tool of the *Point-point* and *Line-line connection* commands.

Direct Data Links

StruSoft develops direct links with the architectural software ArchiCAD and the structural applications Tekla Structures and Revit Structure. So, thanks to external tools, ArchiCAD can exports the entire architectural or a simplified structural model or its part, and the two structural management/editor applications can export the analysis model generated from the structural model.

Direct link means that add-ons embedded in the previous applications generates automatically FEM-Design *Plate*, *Wall*, *Column* and *Beam* elements from their native 3D elements, so further conversion is not needed inside FEM-Design while opening the model files.

The external StruSoft applications together with their user manuals are available on the ***www.fem-design.com*** (*Downloads* menu) by the versions of ArchiCAD, Tekla Structures and Revit Structure.

The ArchiCAD Add-On exports *.fdx* files compatible with the FEM-Design *3D Structure* or *Plate*.

The Revit Structure Add-In exports *.r2f* files compatible with the FEM-Design *3D Structure*.

Tekla Structures sends analysis model (*Analysis* > *Analysis & Design Models*) directly to FEM-Design *3D Structure*, so both FEM-Design and Tekla Structures licenses can be installed on the same computer.

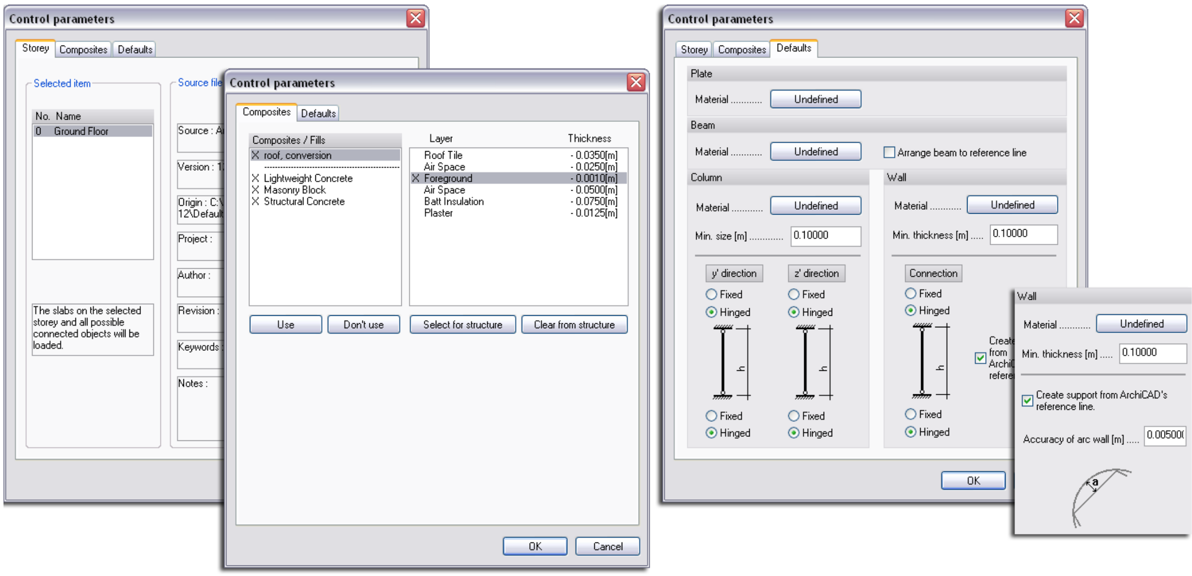
The next table summarizes the FEM-Design elements matched to the elements of the related applications by the external tools in the special exchange file formats.

FEM-Design	ArchiCAD	Revit Structure	Tekla Structures
Beam	Beam	- Beam - Truss - Brace - Beam System	- Concrete Beam - Concrete Polybeam - Steel Beam - Steel Polybeam - Steel Curved Beam
Column	Column	Structural Column (straight only)	- Concrete Column - Steel Column
Plate	- Slab - Roof	- Structural Floor - Isolated Foundation - Wall Foundation - Foundation Slab	- Concrete Slab - Steel Contour Plate
Wall	Wall (Straight, Curved, Trapezoid)	Structural Wall (straight only)	Concrete Panel

Table: Elements by applications can be imported as FEM-Design elements

Material and profile mapping is recommended at all direct link connection:

- In case of Revit Structure, the Add-In gives the possibility to map the Revit Structure profiles and materials with the FEM-Design ones.
- In case of Tekla Structures, the material and profile mappings can be done in editable (e.g. with Notepad) text files. The factory default templates can be found in the FEM-Design's "FORMATS" library: "TeklaFEM-DesignMaterialMapping.cnv" and "TeklaFEM-DesignProfileMapping.cnv".
- In case of ArchiCAD, opening the *.fdx* file in the *3D Structure* or *Plate* module, a setting dialog comes up with the following settings:



Storey (*Plate* module)

In ArchiCAD, users work by storeys. The Plate module's "Selected item" lists all available storeys. Choose the required one, and the program will import the plates and its connected objects like beams and column/wall supports. In case of *3D Structure*, storeys can be not selected, the module opens the entire model.

Composites

"Composites/Fills" lists the materials (ArchiCAD Cut Fills) together with the multi-layered composite structures defined in ArchiCAD. For composites, the core part (that thickness will be considered in FEM-Design) can be set in this dialog tab.

Defaults

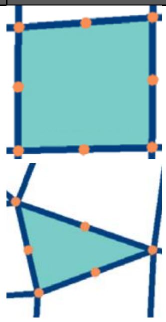
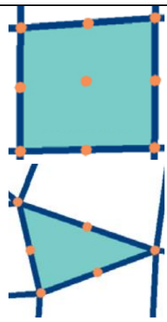
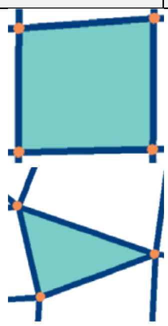
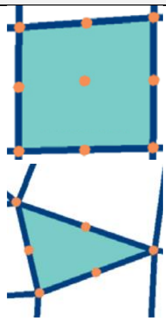
- Here you can set the "Material" mapping by element types.
- Minimum size/thickness value gives the limit of the columns/walls can be imported. For example, giving 0.1m min. thickness for walls means that the partition walls with smaller thickness (e.g. 9cm) will be not loaded while import.
- Fixed and hinged end "Connections" for columns and walls (Plate module only) can be set with the radio buttons.
- Unchecked "Arrange beam to reference line" option will be place beams in their original position as defined in ArchiCAD; but the calculation line will be their (middle) axis. The checked state derives same position of the beam's calculation line with its reference line in ArchiCAD.
- "Create support from ArchiCAD's reference line" has the same function with the previous beam setting, but it effects on walls.
- "Accuracy of arc wall" generates straight wall segments from a curved ArchiCAD wall in the 3D Structure module (because curved wall can be defined in 3D modules). The maximum deviation (a) from the arc can be set in meter unit.

FINITE ELEMENT MESH

Finite Element Method is used in calculation engine as it is also visible in the name of program (Finite Element Method = FEM). This chapter introduces the main concepts, features and functions of the built-in finite element method.

Element Types

Depending on the applied FEM-Design module, the engine uses the following line and 2D (rectangular and triangular) finite elements.

FEM-Design Module	Finite Elements: Line element "Standard"	Line element "Accurate"	2D element "Standard"	2D element "Accurate"	3D element "Standard"	3D element "Accurate"
				 8-NODE 6-NODE		
		 BEAM		 9-NODE 6-NODE		
	 TRUSS	 BEAM				

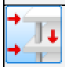
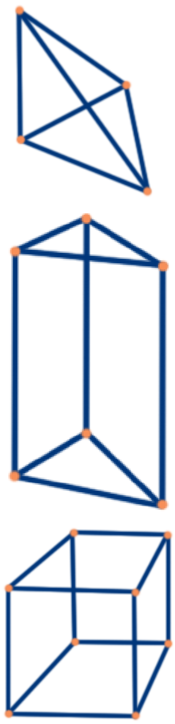
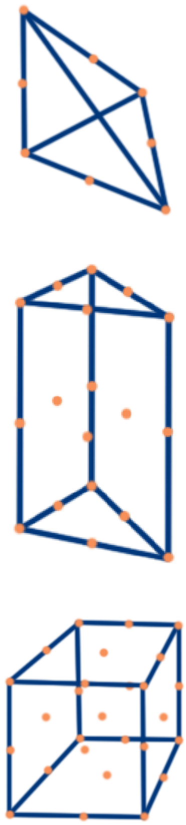
				4-NODE 3-NODE	9-NODE 6-NODE		
						 4-, 6-, 8 - nodes	 10-, 17-, 27 nodes

Table: Finite element types by FEM-Design modules

In the 3D modules, you can choose between "standard" and "accurate" 2D element types. With standard elements you can run 4-times faster but less accurate analysis than with the fine elements. In case of bar elements, the program assigns 2-node line elements to each beam and column element by default:

- 1 piece when choosing "standard" elements, and
- 2 pieces when choosing "accurate" mode.

⚠ The program divides curved beam objects by several line elements in number depends on the central angle of the arc.

The element type "standard" or "accurate" can be set in the calculation dialog (*Analysis* > *Calculate* > *Analysis* > *Finite element types*).

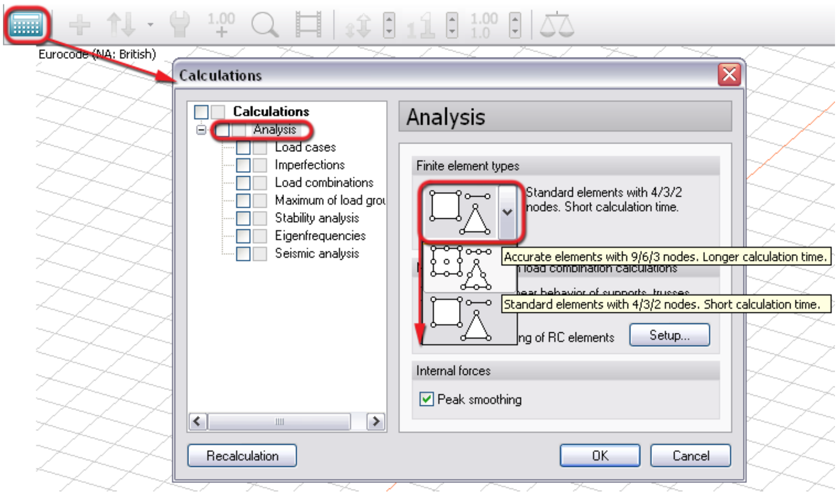


Figure: Finite element types by FEM-Design modules

⚠ Modification on the geometry of a structural object causes the deletion of its finite elements.

Mesh Generation

FEM-Design offers a fully automatic finite element mesh generation by using optimized (factory default) or custom mesh settings. Of course, the generated mesh can be modified with special easy-to-use **edit and modify functions**.

Fully automatism means that the program generates the mesh with elements having average element size optimized for the structure and its environment (supports and loads). The process can contain automatic element refinement and **peak smoothing** algorithm according to the settings.

Automatic mesh generation can be done according to the **mesh settings**:

- **Before calculations**

  Click *Prepare* in the Finite elements tabmenu. You can see and check the finite element suggested. The mesh will be visible by activating the *Surface elements* layer automatically. If you do not edit the mesh and modify the structural model, the later calculations will use the mesh generated by *Prepare*, so no further mesh generation will be done.

- **As the result of calculations**

  If the program does not find previously generated finite element mesh, running calculations (analysis or design) by *Calculate* generates it automatically. The mesh can be displayed by activating the *Surface elements* layer in the current structural or result view mode or just by returning to Finite elements mode.

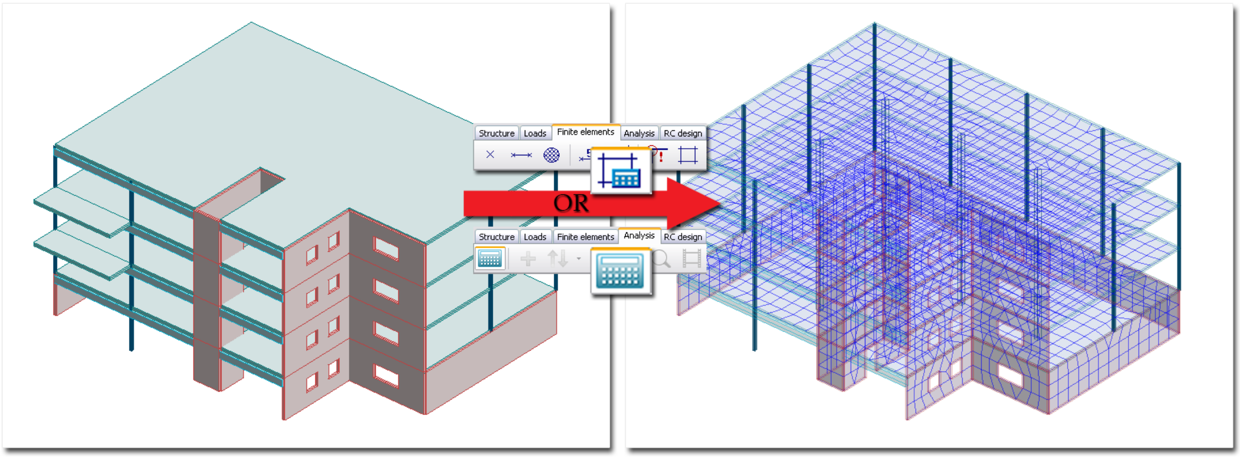


Figure: Automatic finite element mesh generation

Mesh Settings

The settings of the automatic mesh generation are available only in Finite elements mode and at *Settings*> *All...*> *FEM* > *Mesh* and *Calculation*.

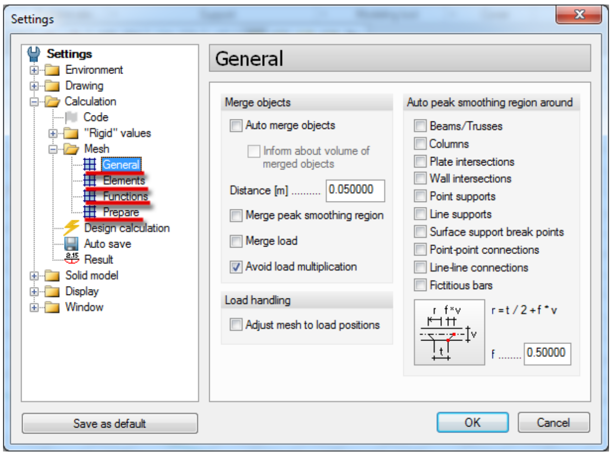
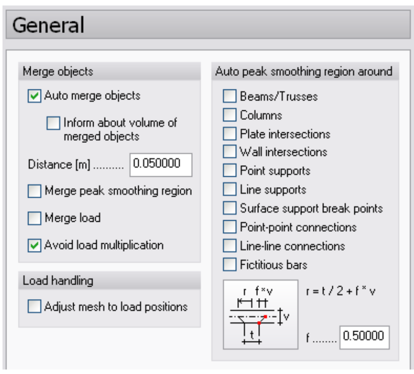


Figure: Settings affect automatic mesh generation (Prepare)

"General" settings



- **Merge objects**

  The program merges **fixed points**, **fixed lines**, **supports**, **beams**, **columns** and **walls** (only in the *Plate* module) to **plate** and/or **wall** regions (the border of the regions). It is decided randomly which objects will be let in their place or removed. The objects shorter than a merging *distance* will be deleted. The objects being at the same place (covering) and having same properties will be deleted except one.

The program also merges columns and/or beams together. It is decided randomly which line elements will be let in their place or removed. The bars shorter than the merging distance will be deleted. After that, the supports fit to the bars. The objects being at the same place (covering) and having same properties will be deleted except one.

The program merges the loads to the geometry created in the first step. Those loads that have not been merged by the previous way will be merged together. It is decided randomly which ones will be let in their place or removed. Line loads shorter than the merging distance will be deleted. If the *Avoid load multiplication* option (see later) is active, the loads having the same position and same properties will be deleted except one.

⚠ The merge process may modify the original shape of the objects to a simpler geometry; but a simple figure cannot be changed for a more complicated one. For example the program does not fit a line load having straight action line to a curved edge although merging distance requires that.

This version of object merge cannot merge the plate or wall regions to themselves or to each other. The user has to pay attention to the correctness of these objects.

Using *Auto merge objects* (recommended) corrects structural object misplacements. If the option is inactive, geometric anomalies cause too long mesh generation process or generation failure.

Using *Inform about volume of merged objects* option together with object merge sends information about the quantity of corrections.

Distance sets the maximal investigation zone between elements, so if the objects are closer than the defined distance, they will be merged. The suggested distance value is 3 to 5cm for engineering problems.

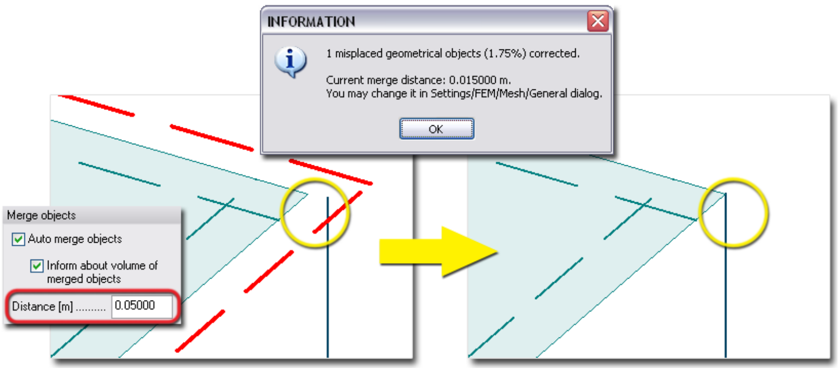


Figure: Correction of misplaced elements (Object merge)

Activating *Merge peak smoothing region* and *Merge load* allows the object merge to work for peak smoothing regions and loads.

Using *Avoid load multiplication* deletes the loads having the same position, geometry and properties (load case host and value) by keeping only one copy.

- **Load handling**

By default, load positions are independent from the finite element mesh; so for example, it is not necessary to place point loads into finite element nodes and vice versa. Although it is recommended to place loads (especially the concentrated loads with high value) into nodes, it is not necessary. *Adjust mesh to load positions* automatically places finite element nodes in the action points, on the action line and region border of the loads depending on their types (point, line and surface load), so the mesh follows the load position and geometry.

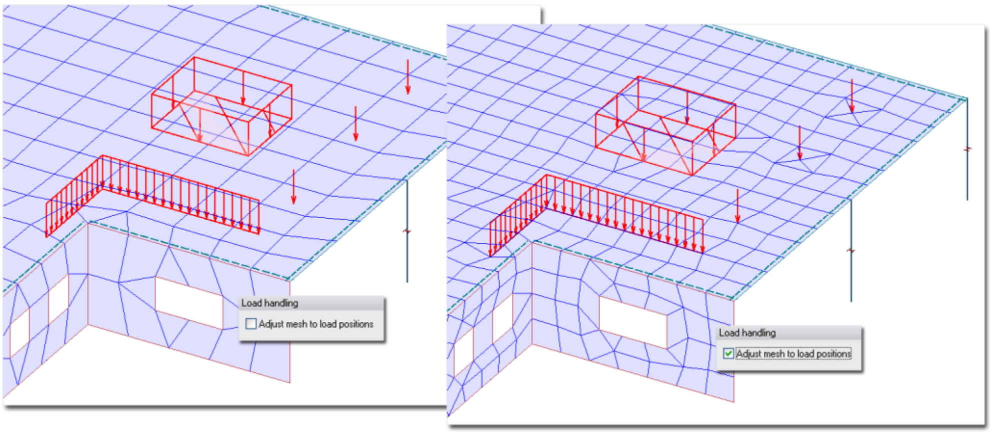
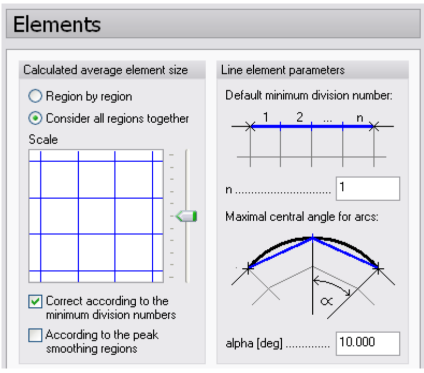


Figure: Load handling in mesh generation

- **Auto peak smoothing region around...**

  To solve the result **singularity problems** above supports and other critical points, the program may run **peak smoothing algorithm** around the listed elements. Activating an element in the list, the program automatically creates **peak smoothing region** around it.

"Elements" settings



- **Calculated average element size**

  By default, the program automatically calculates the optimal average size of the 2D finite elements considering the size, the geometry, the environment etc. of the structural elements. So, you do not need to give an initial value for it. The automatic calculation and the element size depend on the following settings options.

  ⚠ Element sizes can be set manually for all model regions or by regions with the **Average element size** command. If you modify the default "Automatic" value for a planar structural element (wall or plate) to a given value, the automatic element size calculation will be skipped for that region, and the given size will be used for that.

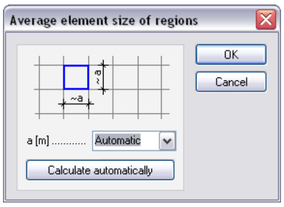


Figure: The Average element size command and the “Automatic” option

Using *Region by region*, the program optimizes the element size by model regions. In this case, the regions will contain meshes generated by different average element sizes. This option is recommended to use in case regions (e.g. having openings and holes) need to be refined (more dense mesh).

Using *Consider all regions together*, the one optimal average element size will be determined for all model regions having “**Automatic**” size setting. This option is suggested for structural models contain regions with nearly same geometry and size parameters.

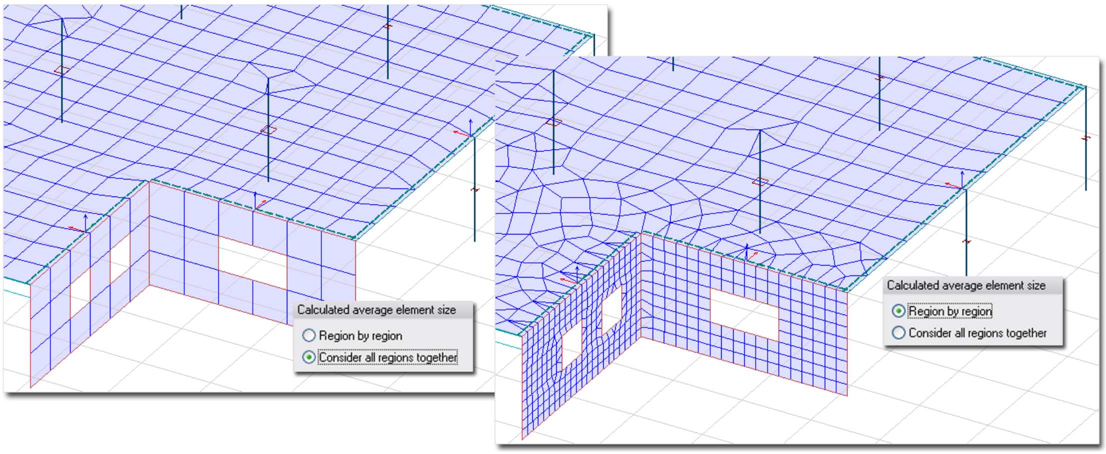


Figure: Calculation modes of average size of 2D elements

In the *Scale* figure, the optimal average element size can be reduced or increased with a given ratio set by the scroll button. The grey mesh shows the recommended optimal size, whilst the blue one shows the modified custom size. Double clicking the *Scale* figure resets the element size to the optimal average element size (1:1 ratio).

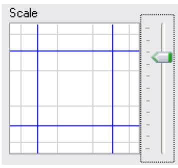


Figure: The average element size will be four times bigger than the optimal size

The *Correct according to the minimum division numbers* option modifies the average element size of the 2D elements, if the minimum division number (**automatic** or **custom**) of the boundary lines and edges requires that. This option is recommended for generating uniform finite element meshes. Skipping this option causes dense mesh near edges where the minimum division numbers are predefined.

Using the *According to the peak smoothing regions* option considers the **peak smoothing** settings of model elements in the calculation of the average element size.

- **Line element parameters**

The default minimum number of the line elements can be set here for the bar elements. The meaning of the default division number depends on the applied element type: **standard or accurate**. For example n=2 value sets (minimum) 2 finite elements for a whole bar (if neighboring elements connect to it only in its endpoints) or a continuous part of it (in case of joined or intersected neighbors) in case *standard* element type and 4 elements at *accurate* element type.

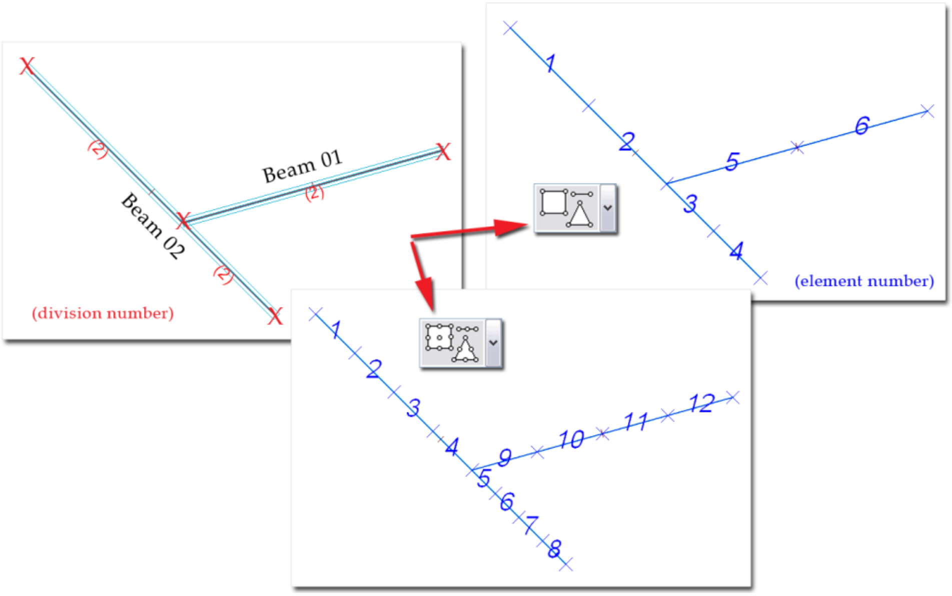


Figure: Meaning of Division number in case of Standard and Accurate element types

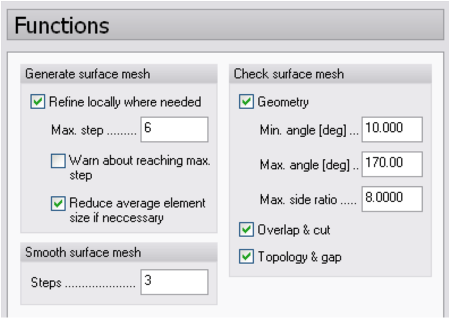
At curved beams, alpha parameter sets the minimum division number:

minimum division number = central angle of the curved beam / alpha .

Division number can be set manually for all bar elements or by line elements with the **Division number** command.

For imperfection (of steel bars), stability and dynamic calculations, it is suggested to set the default *n* value to more than 1; 4-5 division number is the recommended minimum for *n*.

“Functions” settings



- **Automatic refinement in surface mesh (Generate surface mesh)**

  The *Refine locally where needed* option – as an iteration process - eliminates distorted elements, which may normally be derived from accidental geometric errors (if **object merge** is not used). So, the option makes the finite element mesh denser at the locations where needed. (Deactivation of this option can be used in case of searching geometric errors.) The *Max. step* (recommended value is 6) defines the number of the iteration step of refining. The iteration will end when generated elements have the required side-ratio or the steps of the iteration reach their maximal value. If *Max. step* is not enough for the optimal refining, a warning message informs you the number of iteration steps is not enough and there are critical geometry errors (if the *Warn about reaching max. step* option is active).

  The automatic refinement may cause too dense mesh at special geometries (e.g. at highly depressed regions), so in that case, it would be more practical to reduce the average element size with the *Reduce average element size if necessary* option.

- **Smooth surface mesh**

  Smooth procedure calculates the optimal coordinates of the corner nodes of elements. It is recommended after splitting or merging elements. The best mesh can be achieved with the iterative use of the **Rebuild** and the **Smooth** commands. Smoothing of a mesh is executed with iteration technique: the procedure places the nodes of the triangle elements in such a way, that the area of the triangles will be balanced. The number of the smoothing steps can be set in the *Steps* field.

- **Check surface mesh**

  The *Check surface mesh* option lets the program to check the geometry of the mesh after automatic mesh generation. The mesh can be controlled with respect to unsuitable geometry, overlaps and topology. This means that mesh errors, produced by an automatic or manual mesh generation can be easily found. If the program finds defective geometric elements in the model, it sends a warning message (error list) and displays the position of the mesh errors.

  The *Geometry* tool checks the geometry of finite elements, such as the angles of the elements and the ratio of the largest and smallest sides (*Max. side ratio*).

  The *Overlap & cut* tool checks overlapping and intersecting finite elements, which can be caused for example by copying or moving regions together with their finite element meshes.

  The *Topology & gap* tool checks the topology of the finite elements and finds possible gaps. A typical topological error, when for example a corner node of an element lies on a side edge of another element. This problem can easily arise manually by using the **Split** command incorrectly.

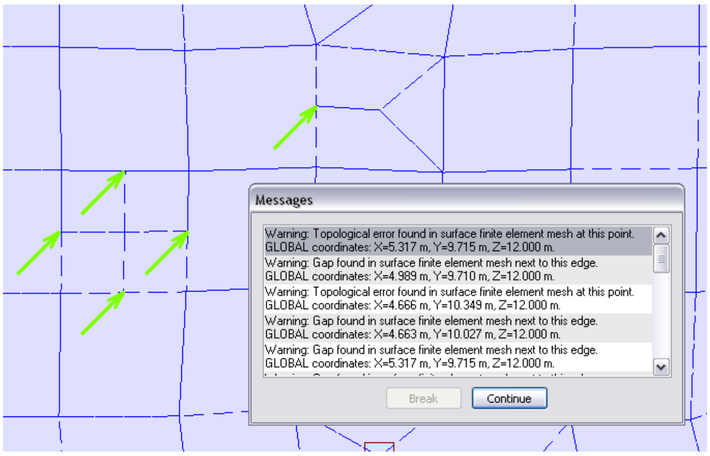
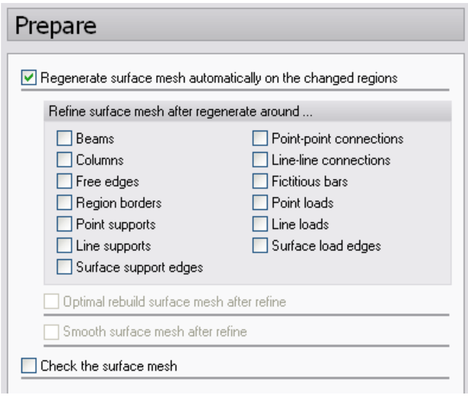


Figure: Topologic errors (unconnected nodes)

Topological errors can be easily solved by the **Rebuild** and **Smooth** commands and quick algorithms.

"Prepare" settings



- **Regenerate surface mesh automatically on the changed regions**

  Using this option, the program will regenerate the mesh at any geometrical changes of region elements and will generate mesh on the regions having no mesh.

  If you switch off the *Regenerate surface mesh automatically*... option, the program will not generate mesh on the regions modified geometrically and will send an error message. In this case you cannot start calculation until you generate mesh on those regions. And, if you would like to generate mesh manually with the **edit functions**, also inactivate this option.

  From the element list, you can choose element types for automatic mesh refinement around them.

  The *Optimal rebuild surface mesh after refine* option rebuilds the mesh automatically after refine. It is recommended to use this option, because elements having non-optimal side-ratio (after refining) will be removed. This procedure will also create more optimal quadrates from the triangles.

If you have objects to refine, the *Smooth surface mesh after refine* option will smooth the mesh after generation, refine and rebuild procedures.

- **Check the surface mesh**

  If this option is activated, the program will automatically check the mesh on regions having valid mesh and are not necessary to be regenerated. The mesh of a region is valid, if the geometry of the region has not been modified since the last generation.

Peak Smoothing
Singularity problem

As an effect of the mesh refinement the calculated results are converging to the theoretical solution. The problem is that at certain places we get infinite inner forces according to the theory, so the inner forces increase each time by refining the mesh. These places could be: point supports, end points of edge supports,

vertices of surface supports, end points of beams and columns, end points of intersection lines of adjoining surfaces, point loads, end points of line loads, vertices of surface loads etc.

In practice, usually, the singularity problem occurs at supports because they heavily influence the inner forces (e.g. negative moments) in ratio.

Possible solutions

There are three known possibilities to solve the above-mentioned problem:

- **Choosing optimal finite element size at singularity places**

  FEM-Design aids that with several built-in tools such as automatic element size adjustment, automations in mesh generation, automatic local densification etc. It is evident, that choosing optimal element size cannot be perfect, because we should have to know the appropriate values to which we adjust the average element size in advance. The functions used today for automatic element size calculation and generation are providing values with adequate precision in most cases, but it is obvious that they cannot guarantee that in any case.

- **More realistic and precise model definition**

  Point and line loads/supports with action surface (only action points and lines) do not exist in real life. So, if you model all point/line loads and supports as surface loads/supports, then you can cease the problem derived from the singularity. This opportunity is available in FEM-Design, because the user can apply **surface supports** and **loads** with any directions and any geometry of action surface.

- **Peak smoothing**

  Singularities always cause only local disturbance in the inner forces, they do not influence the inner forces at adequately short distance from the location of singularities. The "adequately short distance" is defined by the national standards. In the zones causing substantial changes three solutions can occur according to the codes: the peak can be cut (1), or it can be approached with a linear function (2), or a constant value may be set above the substantial area (3). In the last solution, the capacity of the inner force figure above the area may become equal with the capacity of the original figure. The last solution is the safest one, so it is accepted by every standard.

The peak smoothing algorithm is available (**for internal forces and stresses in planar elements**) in every FEM-Design modules work with planar objects.

Peak smoothing region

The program defines peak smoothing regions to solve the possible singularity problems. Basically, these regions are the active zones in the environment of the singularity, where the inner forces change substantially as a result of mesh refinement.

Peak smoothing regions can be generated automatically by the mesh generator or calculation processes. Automatic generation always results circular peak smoothing regions with centre points placed in the location of the singularity. The radius of a circular smoothing region depends on the geometry of singularity locations.

Peak smoothing regions with any arbitrary shape can be defined manually with the **Peak smoothing region** command. That command is able to edit predefined (automatically or manually) peak smoothing regions.

Automatic generation of peak smoothing regions can be set and controlled at the **General** settings of mesh generation (*Settings> All…> FEM> Mesh> General*). At *Peak smoothing region around...* option you can set the places (depending on the current module) where you want the program to create circular peak smoothing regions. The radius of the circular regions is calculated from the following formula:

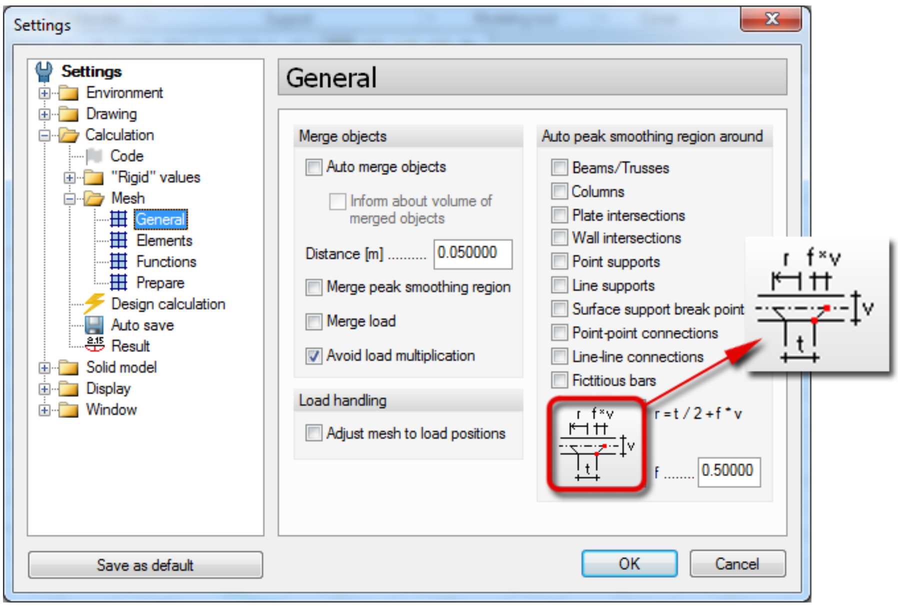


Figure: Settings of automatic peak smoothing generation

$r = t / 2 + f * v$,

where:

t is the characteristic geometric parameter of the object that causes singularity:

- 0 value in case of supports (point, line and surface),
- the diameter of circle circumscribed of a cross-section, if bar elements connect to planar elements,

- the thickness of the plate or wall, if the peak smoothing region generated in plate/wall connection;

v is the thickness of the planar element (plate, wall) in the considered place;

f is a factor can be set manually. The default value is 0.5, which means 45 degrees angle of projection starts from the connection (singularity cause) and ends in the calculation plane of the related planar element (see the figure before).

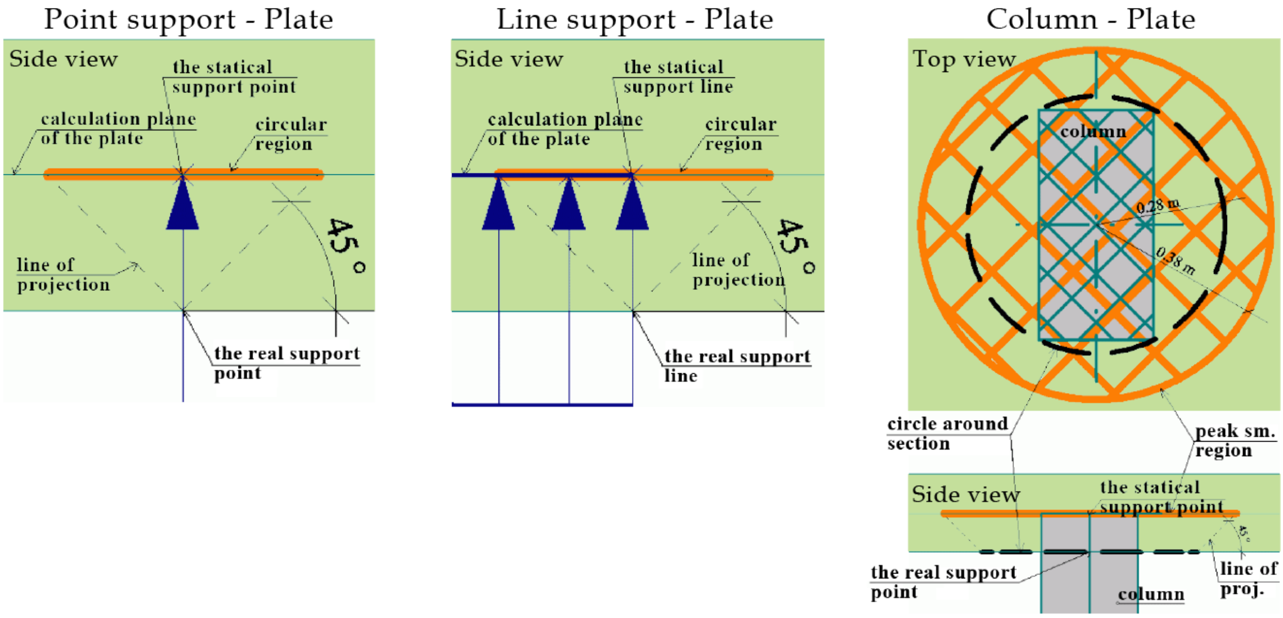


Figure: Examples for peak smoothing regions by different element-plate connection

Peak smoothing algorithm

The steps of the peak smoothing algorithm are the followings during calculations (inner forces):

1. The program creates peak smoothing regions and/or checks the predefined active zones.

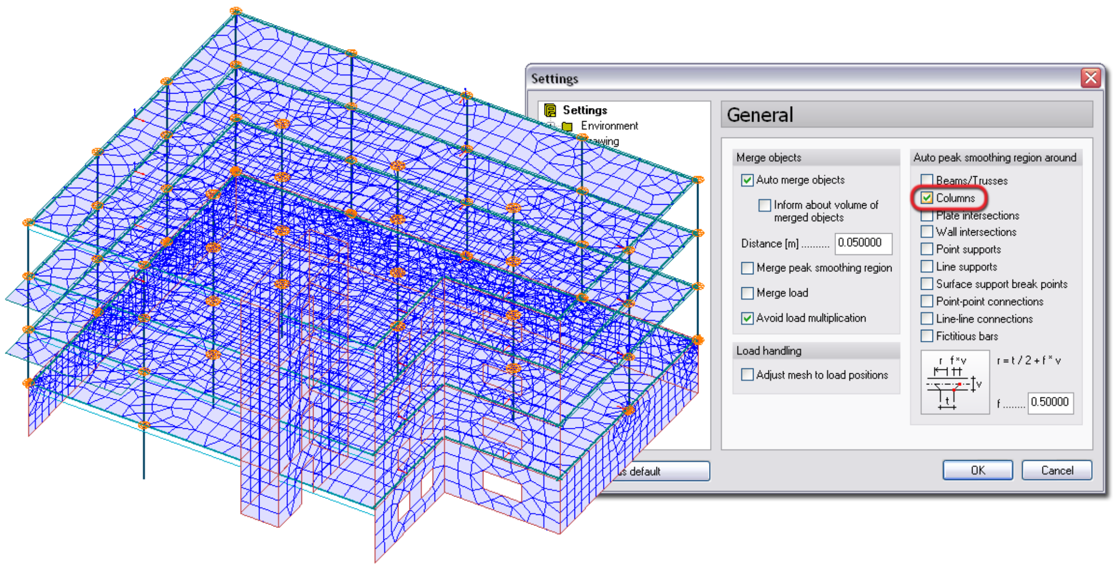


Figure: Generation of peak smoothing region at column-plate/wall connections

2. Allow peak smoothing algorithm for internal force and stress calculations. It is not enough to generate peak smoothing regions, so you have to confirm the smoothing process in the *Calculate* dialog before starting any analysis (and design) calculations.

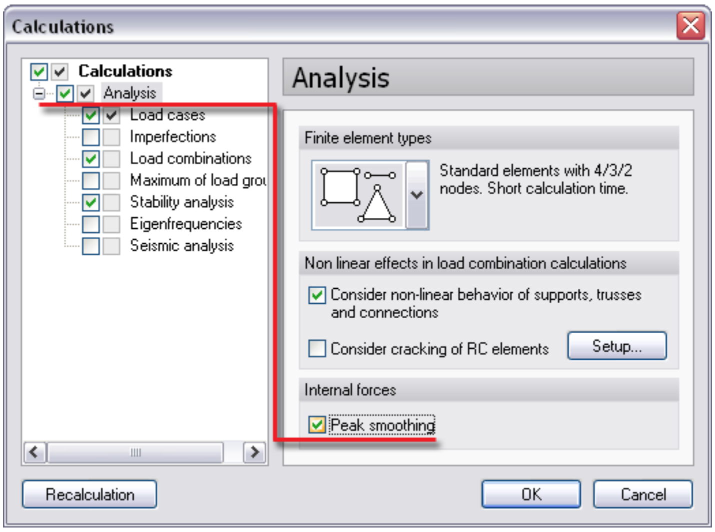


Figure: Peak smoothing algorithm set for analysis calculations

3. The program calculates a constant value for cutting the peaks according to volume calculations of inner diagrams above the peak smoothing regions. That means, the volume at the final constant result value (*Volume (smooth)*) is equal with the volume derived from the peak (singularity) value (*Volume (peak)*) above the same peak smoothing region. Let's see the next figure.

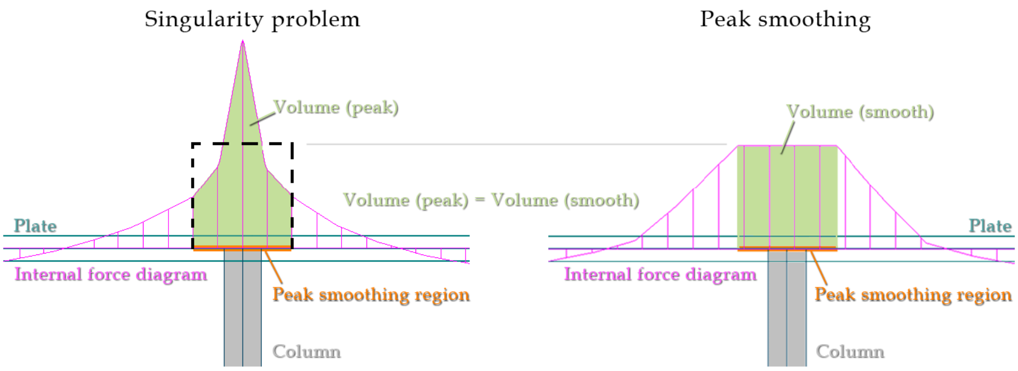


Figure: Peak smoothing algorithm (modified inner force diagram)

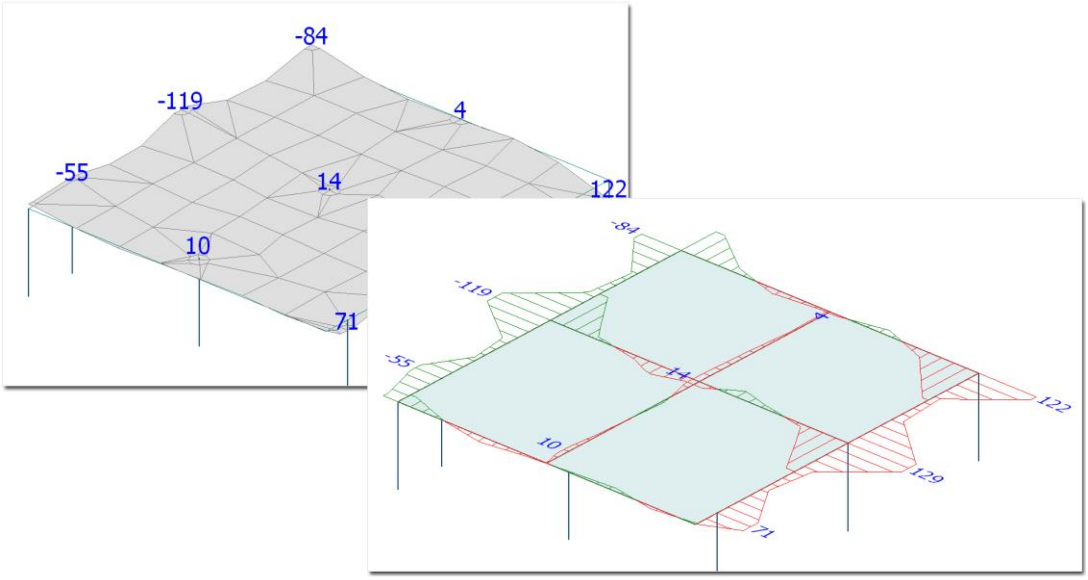


Figure: Internal force "graph" and "section" diagrams after using peak smoothing algorithm

Although peak smoothing is available for internal and stress calculations of planar elements, you can solve the singularity peak problem of line reactions and line connection forces. The program calculates the average value of the reaction and connection forces by finite element. That means, line reactions and connection forces can be displayed with constant (average) value by element (**Distribution > Constant by element**). In this case, you can easily place **numeric values** onto the steps of a figure (*Numeric value*> *Find all local maximum/minimum*).

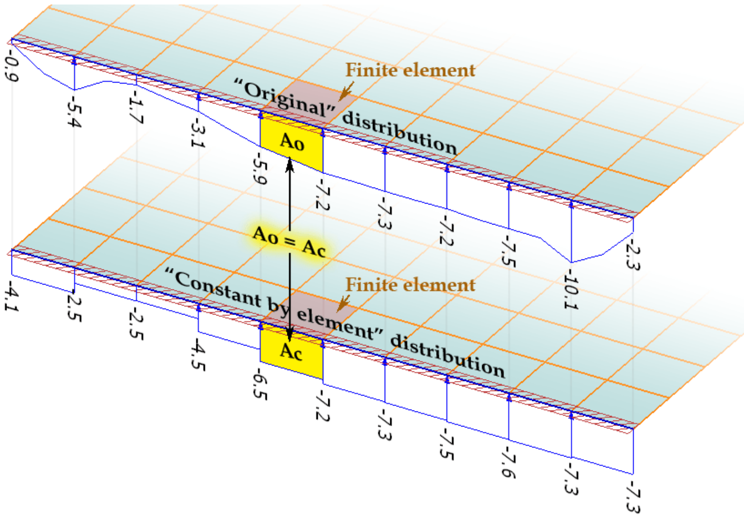


Figure: Singularity of line reaction force solved by simple display technique

Edit Functions

There are numerous tools in the Finite elements tabmenu allow you to edit the finite element mesh generated automatically.

A short summary of the edit functions:

- adding additional nodes to the mesh (**Fixed point**),
- fixing lines inside the mesh (**Fixed line**),

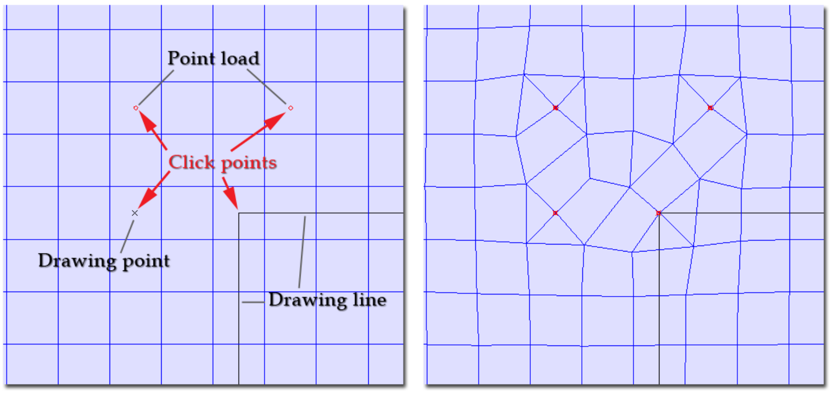
- editing peak smoothing regions or definition of new ones (**Peak smoothing region**),
- fixing the node numbers on fixed lines (**Division number**),
- modifying the element number on bar elements (**Division number**),
- modifying the average element size of 2D finite elements (**Average element size**),
- refining mesh manually (**Refine**),
- splitting elements to modify finite element geometries (**Split**),
- modifying node positions (**Move node**),
- deletion of mesh regions (**Delete**).

After editing finite element mesh, you can do automatic mesh **object merge**, **rebuild**, **smooth**, **check** etc. processes with special tools of the Finite elements tabmenu. Mesh generation can be done by planar regions with the **Generate** tool.

Adding nodes to mesh

With *Fixed point*, nodes can be manually added to a predefined mesh in given points.

Click *Prepare* to update the mesh according to the new nodes. Depending on **mesh settings**, the program automatically does refine, smooth and check processes to create an optimal mesh, if it is possible.



Definition of Fixed points — New mesh after Prepare

Figure: Adding nodes to a mesh

Fixed points defined in the model can be displayed with the *Geometrical system* object layer.

Fixing lines of mesh

With *Fixed line*, lines (straight and curved) can be fixed in a mesh to generate nodes on it.

Click *Prepare* to update the mesh according to the new nodes. The program defines nodes in the line endpoints and some points on the lines. Depending on **mesh settings**, the program automatically does refine, smooth and check processes to create an optimal mesh, if it is possible. Further node distribution of the fixed line element can be influenced by the **Division number** command.

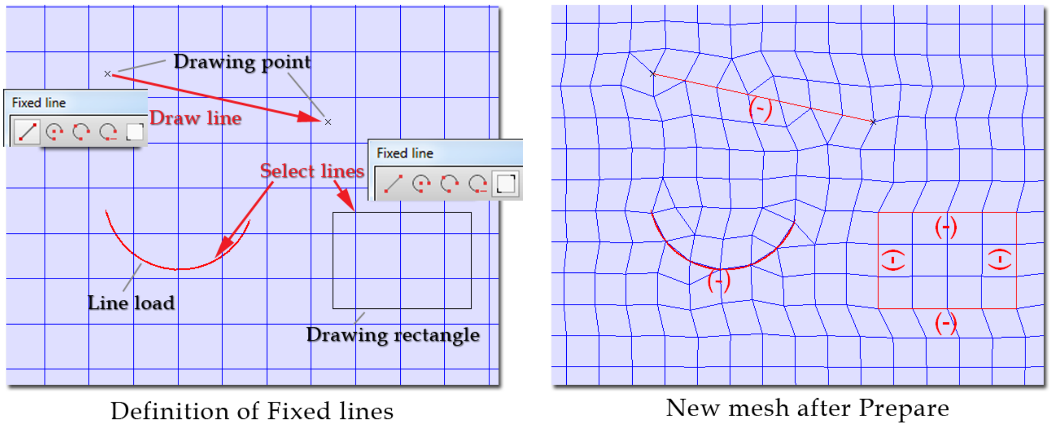


Figure: Adding lines to a mesh

Fixed points defined in the model can be displayed with the *Geometrical system* object layer. In brackets, the number of line elements can be seen defined by the **Division number** command. If "-" is displayed, there is no restriction on the element number.

Fixed lines display the intersection lines of structural regions (e.g. intersections of slabs and walls) too. This is very useful for defining holes/openings that connect accurately to finite element meshes (see the next figure).

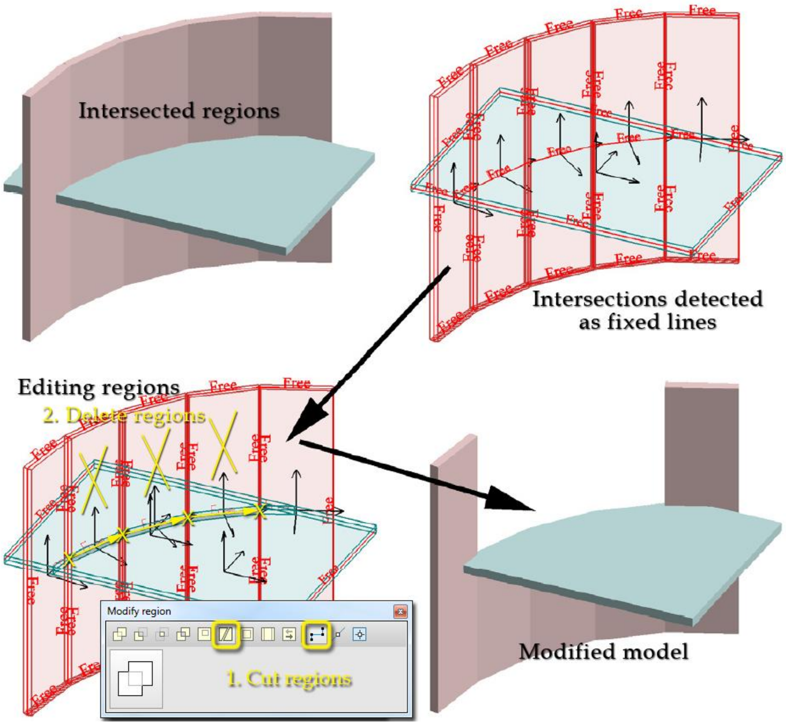


Figure: Fixed lines used for editing regions

Editing peak smoothing regions

With *Peak smoothing region*, the predefined peak smoothing regions can be edited (status, shape and size modification), or new ones can be defined manually.

The following functions can be done with the *Peak smoothing region* command:

- **Modifying the geometry of predefined peak smoothing region**

  Automatic generation of peak smoothing region always results circular geometry of regions. If you would like to create regions with custom shape, draw a new one with the *Define* tool and the different shape tools. If the new region covers an automatic one (and it is active, see later), the program will use the new region instead of the automatic one for solving the singularity.

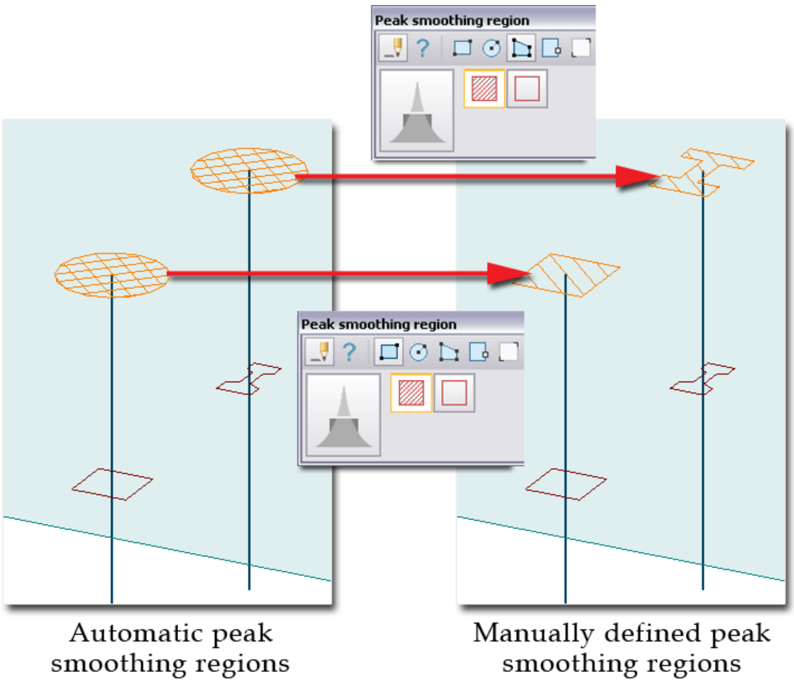


Figure: Manually drawn peak smoothing regions with custom shape

  Sometimes, you need to correct the geometry of automatic regions to avoid the creation of incorrect or too dense finite element mesh (nodes are generated in the intersections and on the edges of peak smoothing regions).

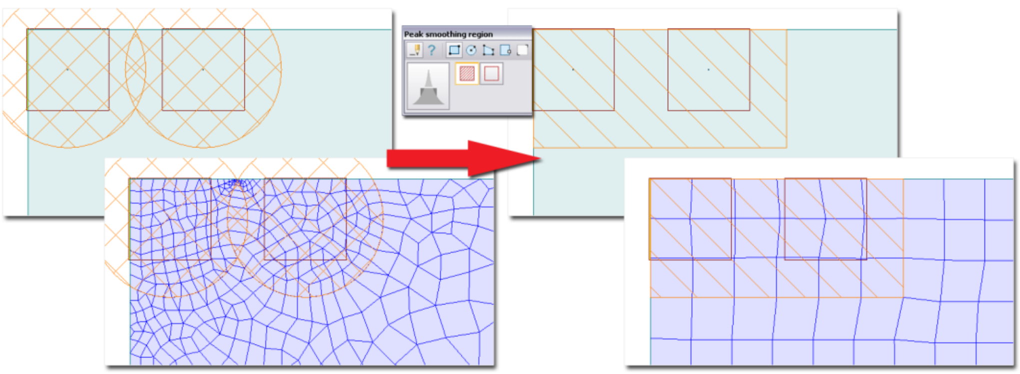


Figure: Geometry modification effect on finite element mesh generation

- **Creation of new peak smoothing regions manually**

  With the *Define* tool, totally new peak smoothing regions can be drawn with custom shapes, too.

- **Modifying the status of a predefined peak smoothing region**

  Apply the *Properties* tool and the *Inactive* option for peak smoothing regions to modify their status from active to inactive. Inactive region will not be considered in mesh generations and in calculations too. To reset an inactive status of a region to an active one, apply the *Properties* tool together with the *Active* option for the region.

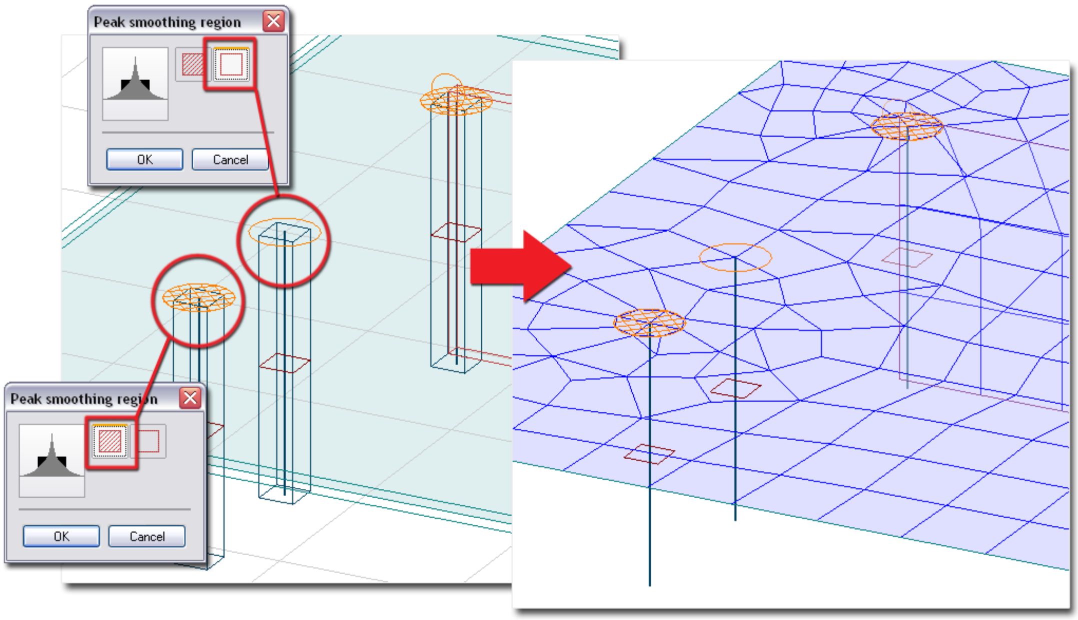


Figure: Difference between active and inactive peak smoothing regions

Peak smoothing regions defined in the model are stored in the *Peak smoothing regions* object layer.

Click *Prepare* to update the mesh according to the new and modified peak smoothing regions. Depending on **mesh settings**, the program automatically does refine, smooth and check processes to create an optimal mesh, if it is possible.

⚠ It is not enough to generate/define peak smoothing regions, so you have to confirm the smoothing process in the *Calculate* dialog before starting any analysis (and design) calculations.

Division number

Division number sets:

- the number of line finite elements of bar elements (columns and beams),
- the number of 2D element-edges on **fixed lines** and structural region edges.

In case of bar elements, the *Division number* command can modify their default element number (value in brackets) set at **"Elements" setting**.

⚠ In 3D modules, the meaning of division number depends on the element type sets at calculations. For example division number 2 sets 2 line elements for *standard* element type and 4 line elements for *accurate* element type. The line elements together with their node numbers can only be displayed after analysis (or design) calculations (see **display settings**).

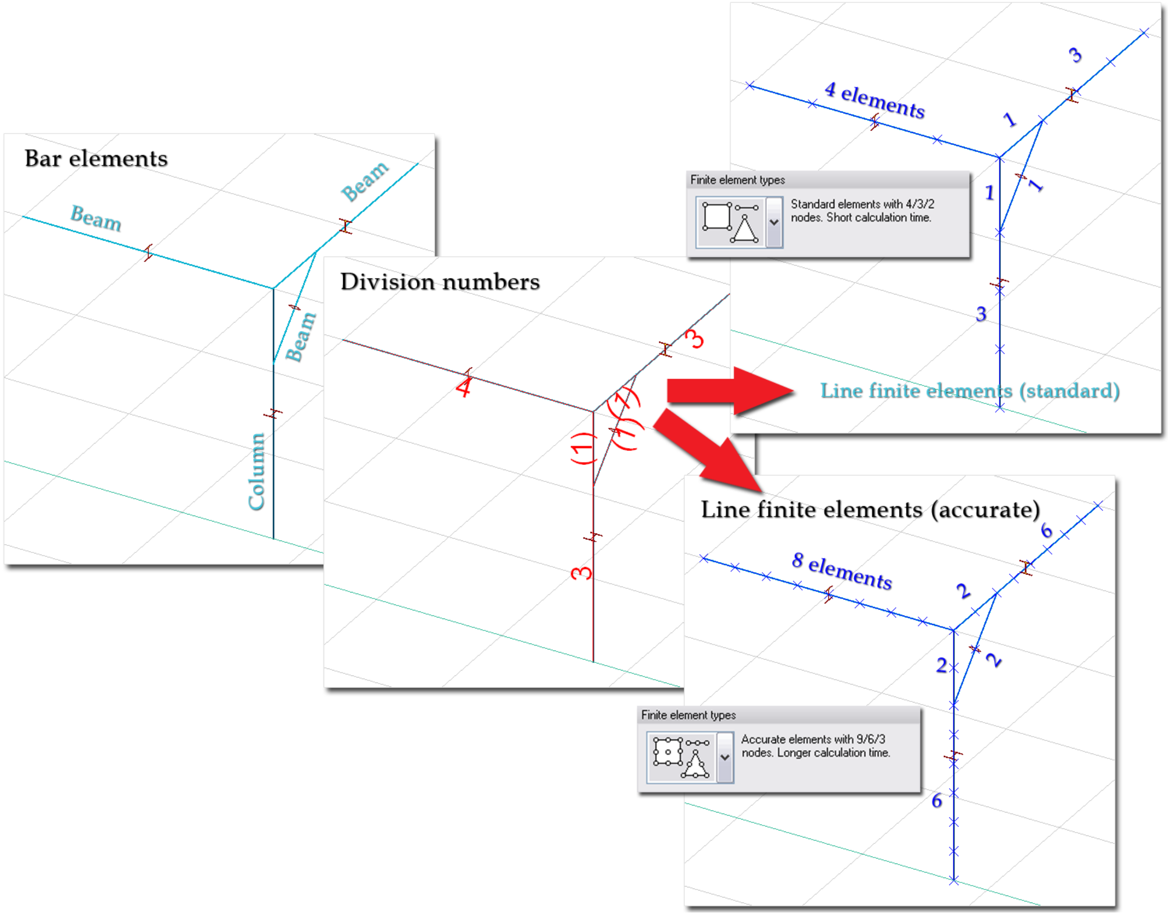


Figure: The meaning of division number for bar elements

In case of fixed lines and structural region edges, the *Division number* command can fix the **minimum number** of 2D elements (sides) will be connected to the lines/edges by the mesh generator. "-" label in brackets shows no number-restriction for lines/edges.

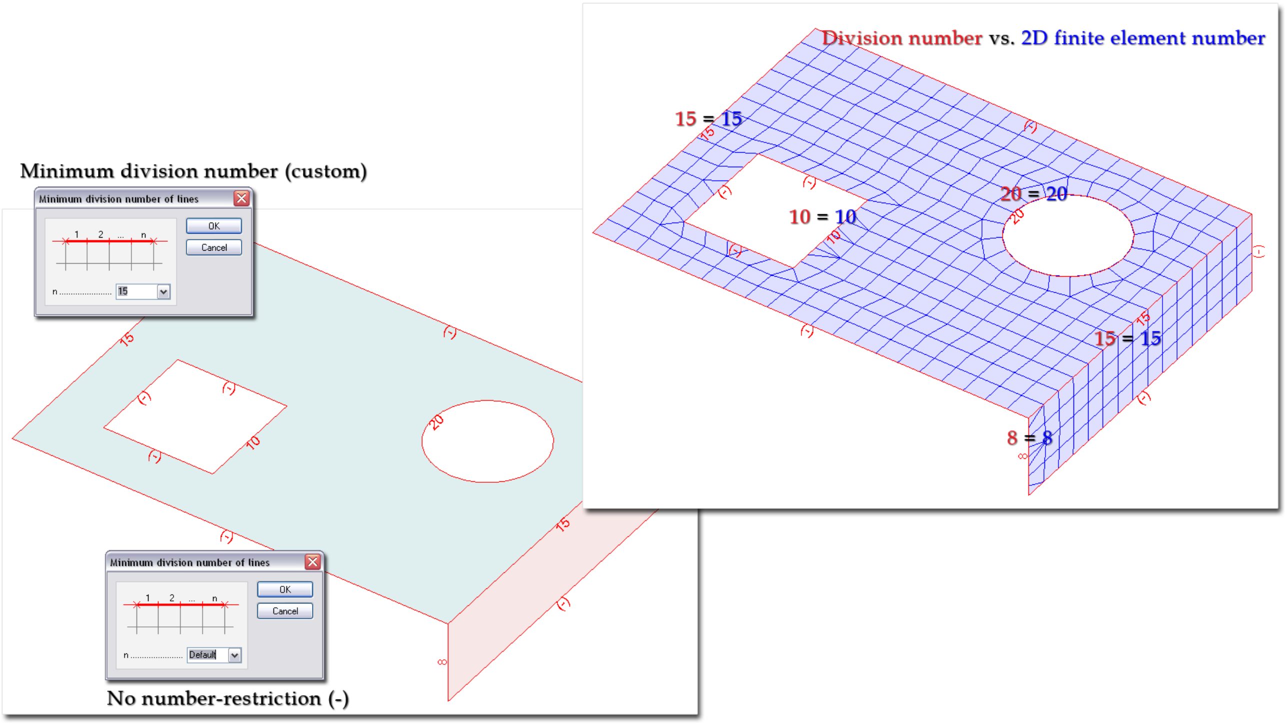


Figure: The meaning of division number for fixed lines and edges

Defined division numbers together with fixed lines, edges and structural lines of bar elements are displayed with the *Geometrical system* object layer (red by default).

Modifying average element size

The  *Average element size* command modifies the optimized element size(s) of 2D elements calculated by the program according to **mesh settings**.

Applying the command for a structural region (wall, plate etc.), a dialog shows with "*Automatic*" label, that the optimal element size calculation is the active by default for the region. Typing an arbitrary element size (in *a* field) overwrites the usage of the optimal size. Of course, the final average element size generates by the mixture of the value set by *Average element size* and the effect of **mesh settings**. Inactivating all mesh generation automatism (refine, smooth and check processes together with "*Elements*" settings), the *a* value will be the average element size for the related structural regions.

Clicking *Calculate automatically* button shows the optimized average element size used when "*Automatic*" is set for the related object region(s).

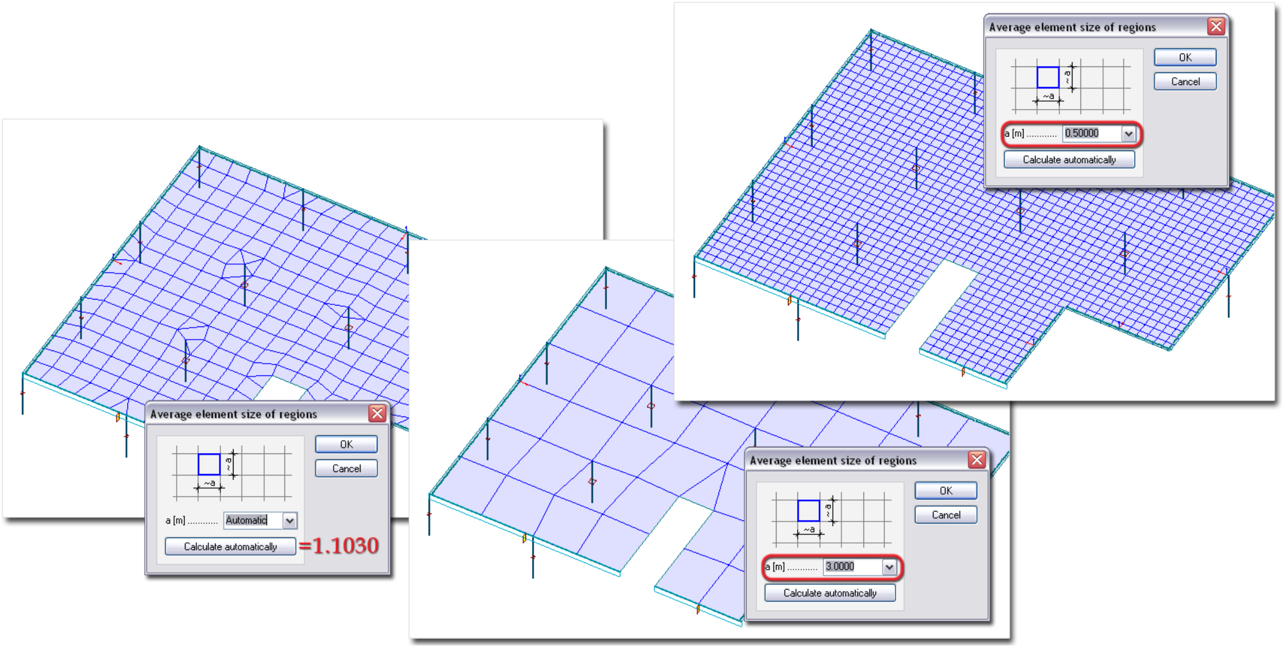


Figure: "Automatic" and custom average element size

Click *Prepare* or *Generate* (for the region only) to update the mesh according to the average element size setting.

Refine mesh manually

A predefined finite element mesh can be refined manually. Apply the *Refine* command to refine mesh parts by picking structural objects connect to the meshed region or by clicking directly finite elements.

- **Refine mesh around objects**

  Activate first the *Mark element mode*. Pick the checkbox of object types (e.g. *Region border*), which you would like to allow refining the mesh around. Select objects (e.g. Walls) assigned to allowed object types on the drawing area with *Pick object*, or click *Mark elements around all objects* to refine mesh around all objects belongs to the checked object types. Click to start the refine process.

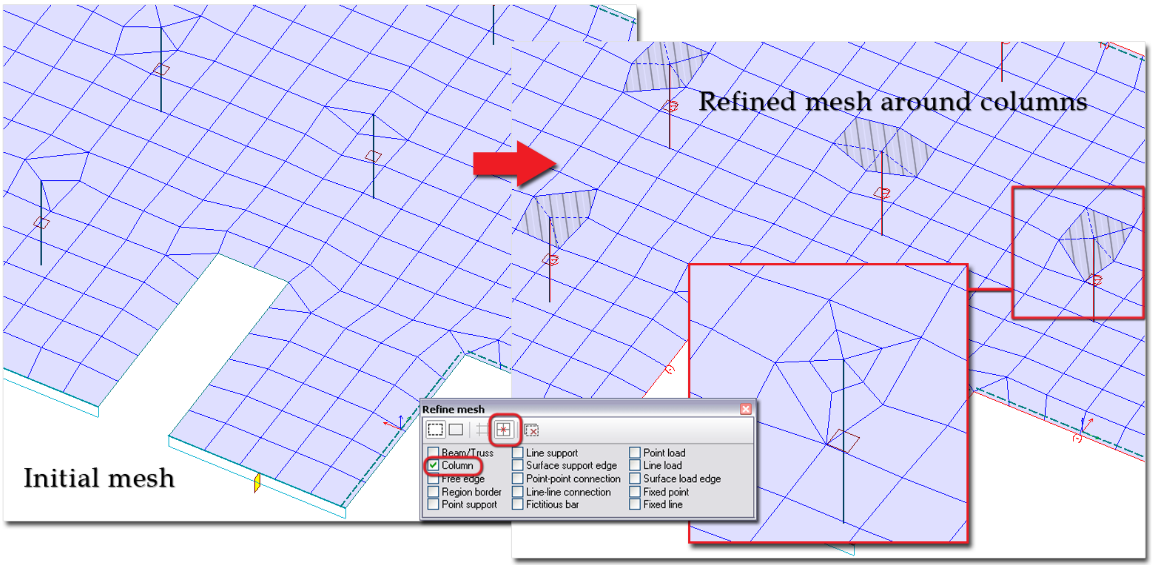


Figure: Refined finite element mesh around columns

- **Refine 2D elements**

  Activate first the *Mark element mode*. Select elements of a mesh on the drawing area with *Pick element*, or click *Mark all elements* to refine all finite elements of the project. Click Enter to start the refine process.

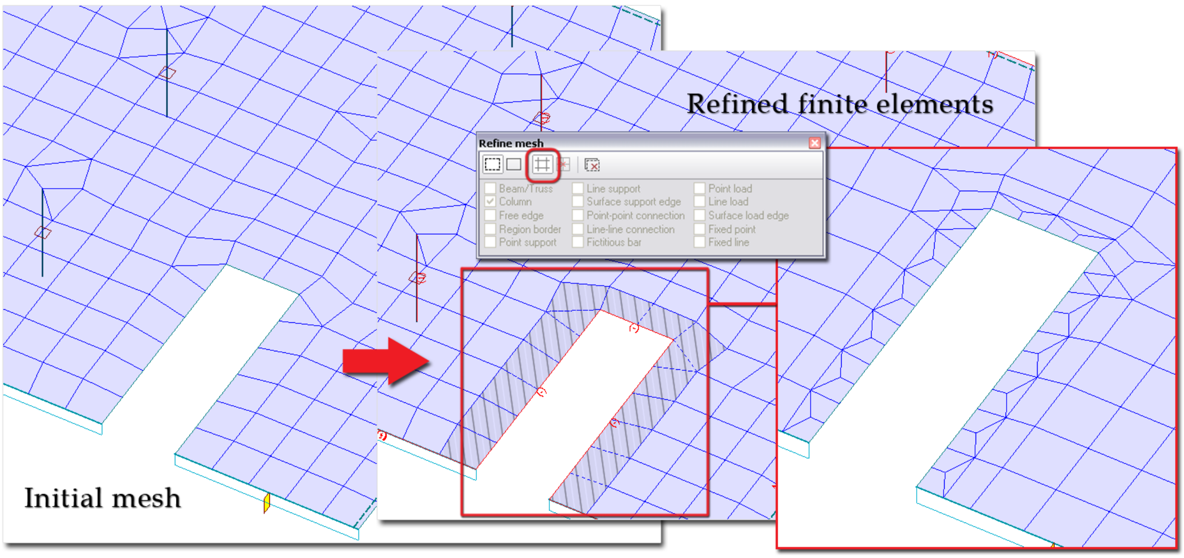


Figure: Refined finite elements (element-selection)

To deselect previously selected elements (before clicking Enter to start refine), switch to *Unmark element mode* and do the same steps mentioned for selection.

Splitting elements

2D finite elements can be refined according to splitting patterns, too. Start the *Split* command, then chose a cut pattern, and finally select an element which you would like to apply the selected cut mode. Each pattern has a short description about its usage for rectangular and triangular 2D finite elements.

At some splitting patterns, red "+" shows the click position in an element to get the requested splitting shapes.

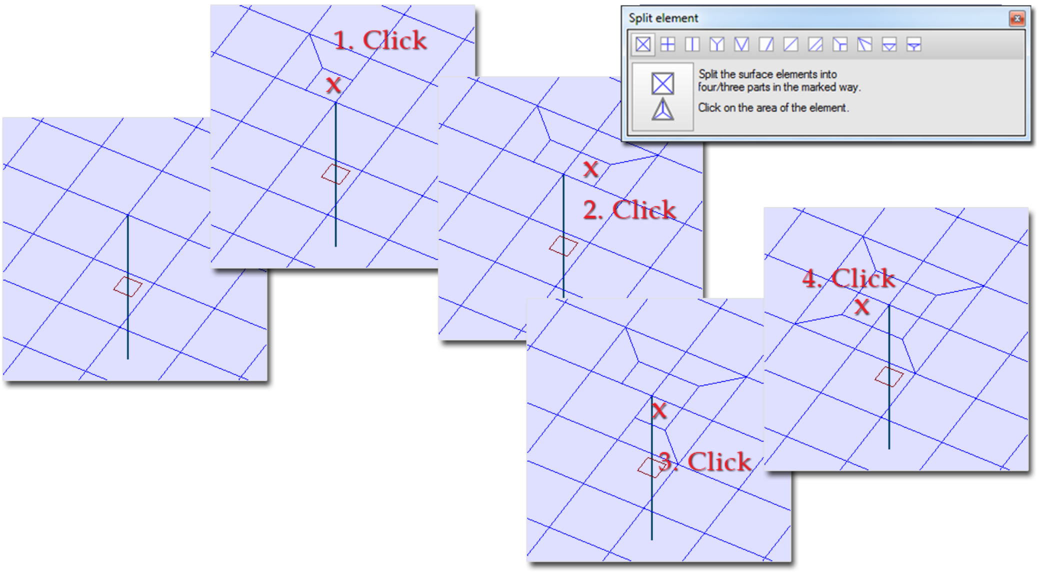


Figure: Element refinement manually with Split

Modifying node positions

The position of mesh nodes can be modified with the  *Move node* command. Select node(s) you would like to move, and define the displacement vector with a start- and endpoint. The program sends an error message, if it finds nodes being out of legal moving range.

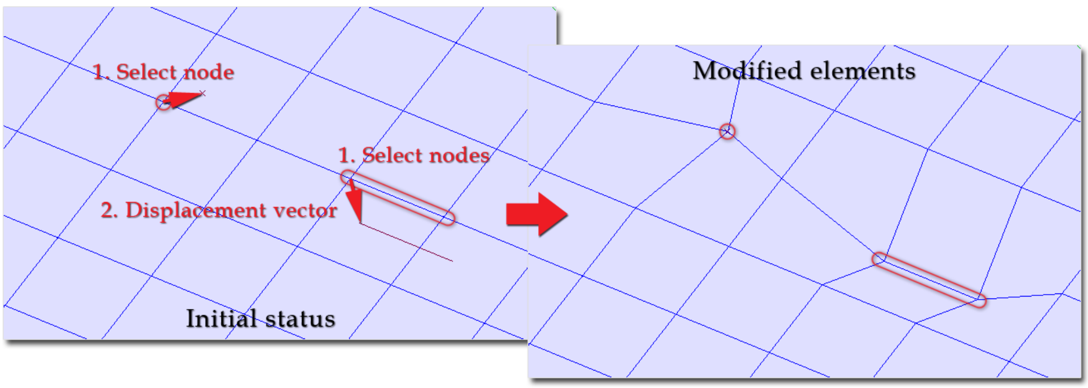


Figure: Modifying node positions

Deletion of mesh regions

With the  *Delete* command, predefined mesh of selected planar object (regions) can be deleted. Just select the required region(s) and the program erases the mesh.


If you modify the geometry of an object region (e.g. by inserting a hole, by stretching one of region corners etc.), mesh assigned to it will be automatically deleted.

Object merge

The **object merge process** can be run any time you want by clicking the *Object merge* command button.

Rebuild

After using manual tools (mentioned before) to edit the mesh generated automatically before, it is recommended to rebuild the mesh. The process rebuilds the current mesh of selected object regions according to the global optimum without the movement of mesh nodes.

The steps of the rebuild process are:

1. The program builds a mesh from triangular elements using the principles of Delaunay triangular technique.
2. It converts these triangles to quadrates corresponding to the global optimum of the mesh.

If the *Rebuild* command results drastic changes in the mesh, **smoothing** algorithm is also recommended. The best, the most optimal finite element meshes can be achieved with the iterative usage of the two commands.

Smooth

Smooth process calculates the optimal coordinates of the corner element nodes. Smoothing the mesh is highly recommended after **splitting** or **merging** elements. If the mesh is modified drastically after **rebuild**, use smoothing for the model. The best, the most optimal finite element meshes can be achieved with the iterative use of the ***rebuild*** and the smoothing.

Just apply the *Smooth* command for selected mesh regions. Mesh smoothing is executed with iteration technique: the procedure places the nodes of the triangular elements in such a way, that the area of the triangles will be balanced. The number of the smoothing iteration steps can be set at **"Functions > Smooth surface mesh" settings**.

Undo cannot restore the mesh state before using *Smooth*.

Check

After using manual tools (mentioned before) to edit the mesh generated automatically before, it is recommended to check the mesh. It can be done according to **"Functions > Check surface mesh" settings** any time you want by clicking the *Check* command button.

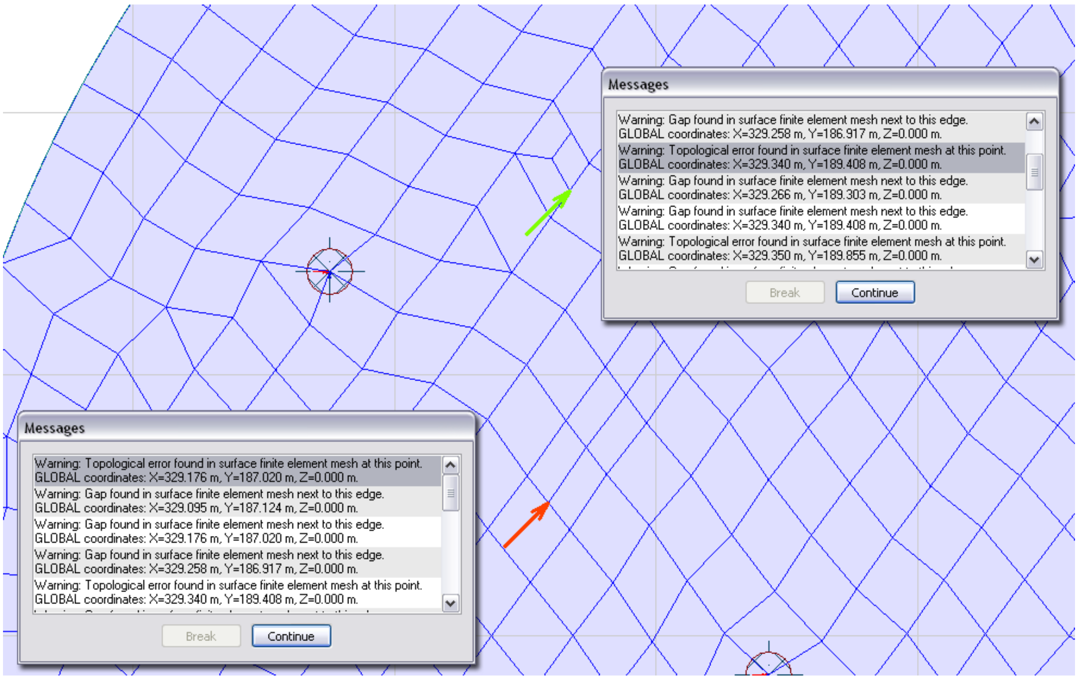


Figure: Topological errors found by Check

Generate (vs. Prepare)

Automatic mesh generation can be done by planar object region with the  *Generate* command. (**Prepare** runs automatic mesh generation for the entire structural model.) *Generate* takes into consideration the manual editing functions (such as **Fixed point**, **Fixed line**, **Peak smoothing**, **Division number** and **Average element size**) and the **Mesh settings** excluding the **Check**, the **Smooth** and the **Prepare** settings. *Generate* is also recommended to find defective geometric finite elements by checking (generating mesh) regions by regions.

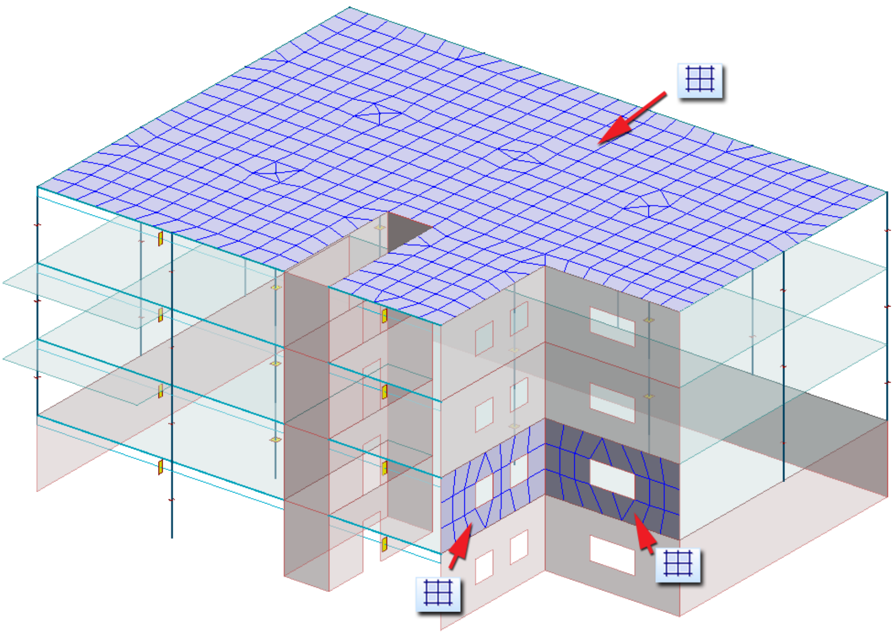


Figure: Mesh generation by regions (Generate)

Mesh generation is an iteration process, of which goal is to generate optimal and eurhythmic mesh by the given or automatically calculated average element size. The main and general steps of the generation process are the following:

1. Defining the node positions of elements.
2. Creating triangle elements by nodes.
3. Converting triangle elements to mixed quadrate mesh where it is possible.
4. Optimization of node coordinates (e.g. smoothing the mesh).
5. Definition of middle points on element sides.

Renumbering and Display Settings

Renumbering

Mesh generator automatically add numbers for finite elements and nodes. If you edit the mesh (e.g. adding new nodes and elements, mesh refinement etc.), you can rerun the renumbering process with the *Refresh numbering* command of the *Tools* menu.

Display settings

The display style of node symbols and **peak smoothing regions**, the numbering of nodes and finite elements can be set at *Settings* > *All...* > *Display* > *Mesh*.

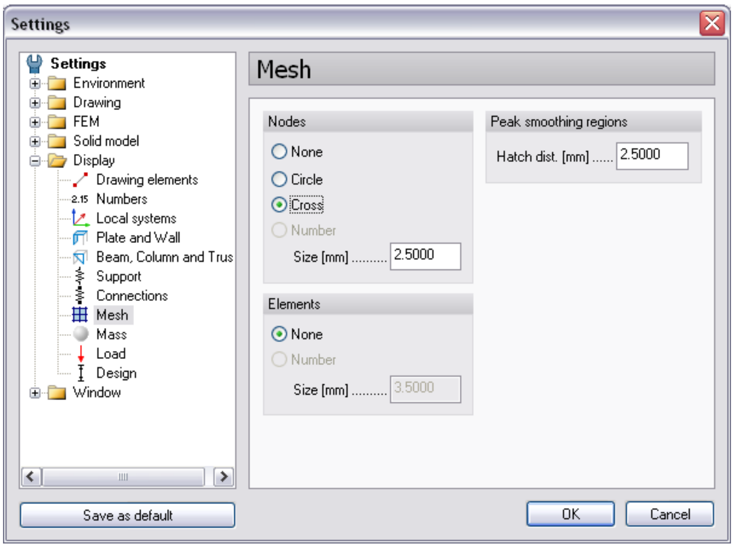


Figure: Display settings of mesh items

- **2D finite elements**

  Although 2D finite elements are displayed automatically after mesh generation (if the *Surface elements* object layer is active), the element numbers (*Number*) can be displayed only after **analysis** or **design** calculations. The nodes can be displayed with circle or cross symbols any time, but with numbers (*Number*) after analysis and design calculations.

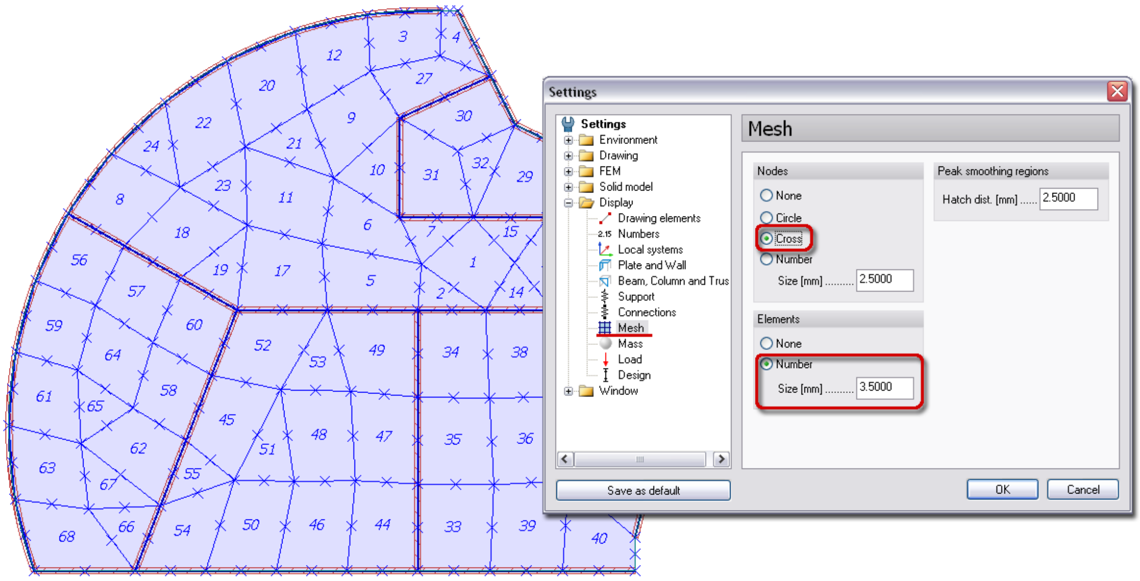


Figure: Finite elements displayed with numbers and nodes

- **Line finite elements**

  Line finite elements together with their numbers (*Number*) can be displayed only after **analysis** or **design** calculations, if the *Line elements* object layer is active. The nodes can be displayed with circle, cross symbols or with numbers (*Number*).

The color of the elements, nodes and peak smoothing regions depends on the color of their own object layer.

Error Handling

Warning and error messages assist you when problems appear in mesh creation functions, during analysis or design calculations. But, how you can find the position of these cautions to solve them later?

The program points the geometrical and mesh errors, the load misplacements and any other problems in the model, and it collects their coordinates (in the *Global coordinate-system*) in an error/warning dialog.

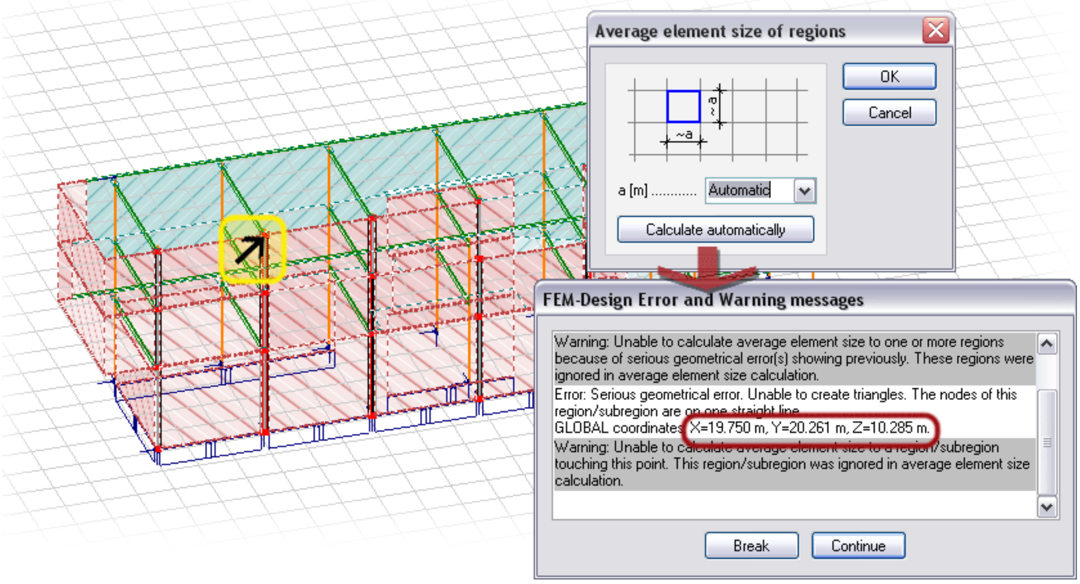


Figure: Geometrical errors detected by the average element size calculation

Analysis

Depending on the current FEM-Design module you can do different calculations: displacement, internal forces, stresses, stability, imperfections, stability analysis, eigenfrequencies and/or seismic analysis. Some extra settings such as cracked-section analysis, non-linear behaviour etc. are also available for certain modules.

Analysis type/settings						
Analysis for load cases	✓	✓	✓	✓	✓	✓
Analysis for load combinations	✓	✓	✓	✓	✓	✓
Analysis for maximum of load groups	✓	✓	✓	✓	✓	✓
Imperfections				✓	✓	
Second order analysis				✓	✓	
Stability analysis				✓	✓	
Eigenfrequencies	✓	✓	✓	✓	✓	
Seismic analysis				✓	✓	
Non-linear behavior	✓	✓	✓	✓	✓	✓
Cracked-section analysis	✓	✓	✓	✓	✓	✓
Peak-***smoothing algorithm***	✓	✓	✓	✓	✓	✓

Table: Analysis features by FEM-Design Modules

Analysis can be done independently from any design calculations by entering to Analysis tabmenu and clicking *Calculate* command, or together with designs (RC, Steel or Timber) with the same command.

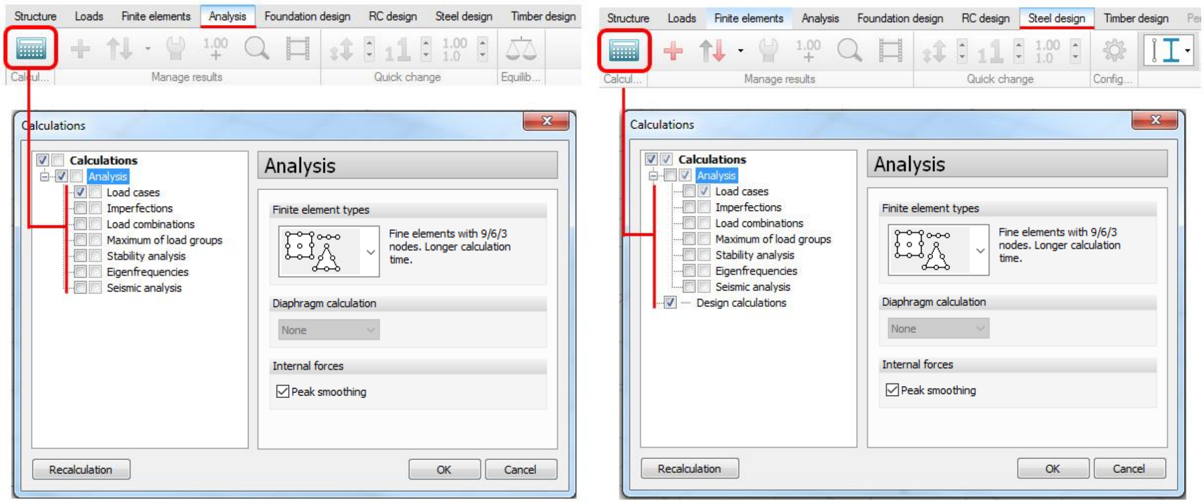


Figure: Analysis calculations

Analysis settings contain general and calculation-dependent settings. This chapter summarizes these settings and their effect on the result. Clicking *OK* runs Analysis according to the settings and selected calculation types. Other chapters introduce the display of results and their documentation (such as listing results in tables).

General Analysis Settings

Finite Element Types

In the 3D modules, you can choose between "standard" and "accurate" 2D element types. With standard elements you can run 4-times faster but less accurate analysis than with the fine elements.

Peak Smoothing

To solve singularity problem in analysis results (internal forces), it is not enough to create peak smoothing regions in the finite element mesh. The use of the peak smoothing algorithm in the calculations have to be allowed. Without that permission, peak smoothing regions cause only mesh refinements (densifications) around objects.

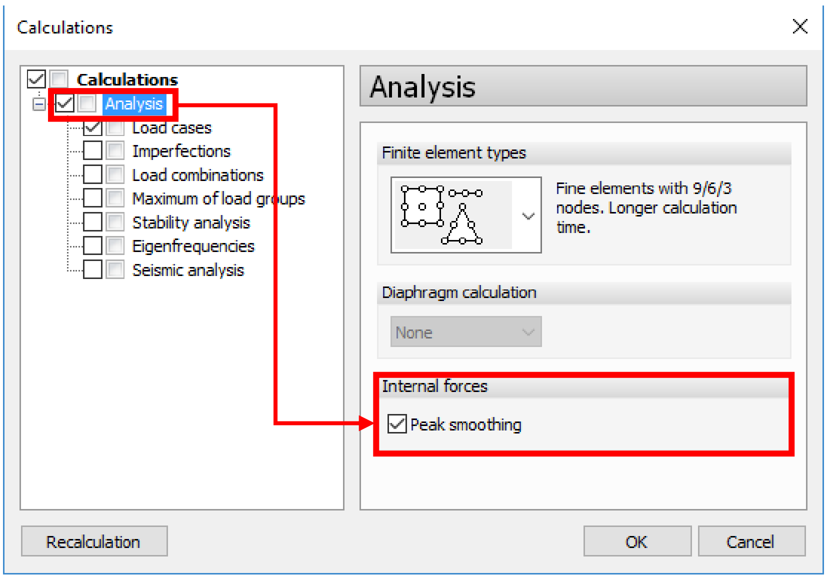
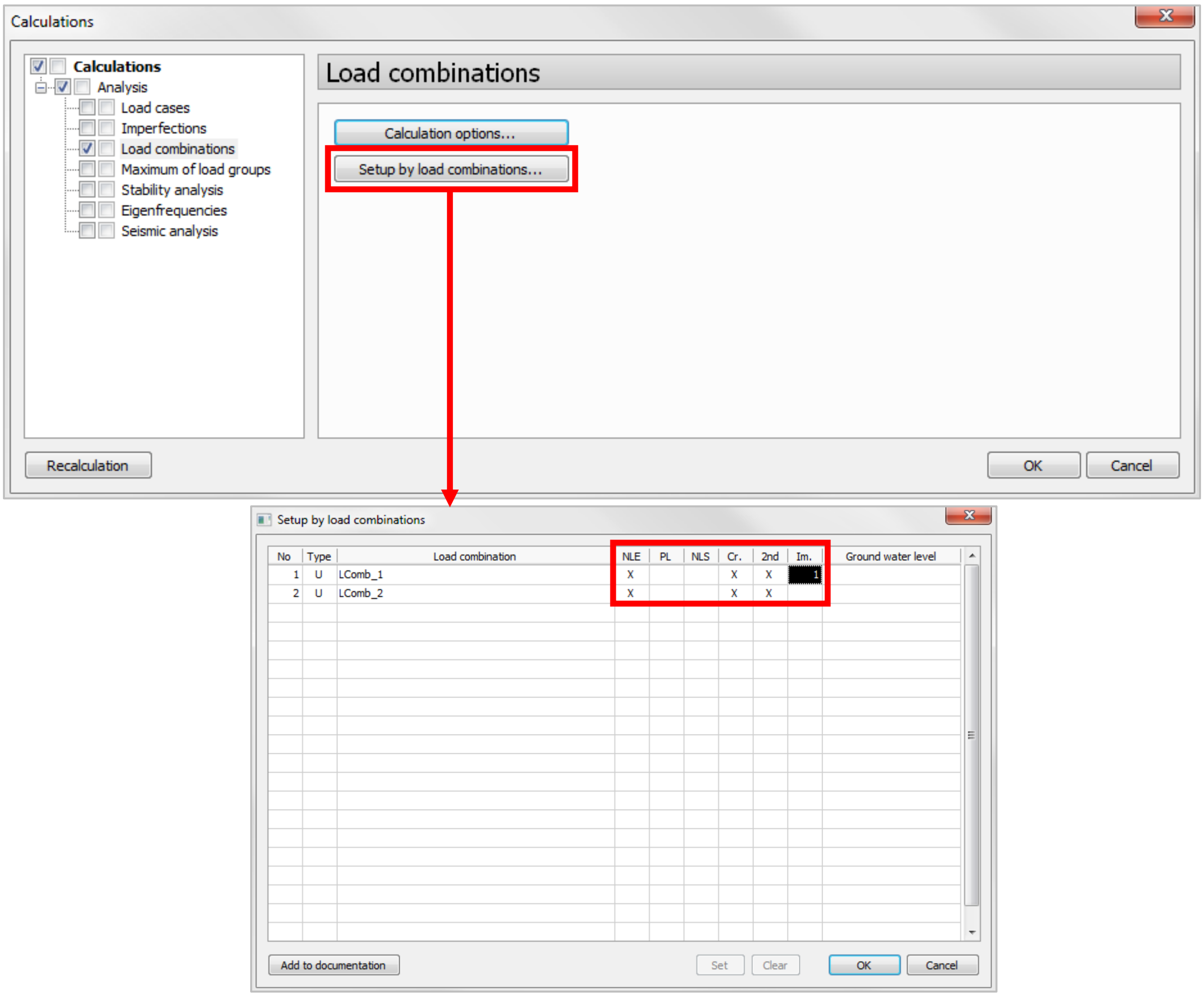


Figure: Peak smoothing algorithm for Analysis

Setup calculation by load combinations

The calculation of the load combinations can be run with different options. They can be set in Calculations dialog by selecting the Load combination items and clicking on *Setup by load combinations*.



The User has the opportunity to choose

- Non-linear elastic calculation (NLE),
- Plastic analysis (PL),
- Non-linear soil (NLS),
- Cracked-section analysis (Cr.),
- Second order analysis (2nd),
- Imperfection calculation (Im., the selected shape will be taken into account in Second order analysis)

for each Load combinations.

For example, in practice it can be useful to set 2^{nd} order analysis only for the ULS and Cracked-section analysis only for the SLS combinations.

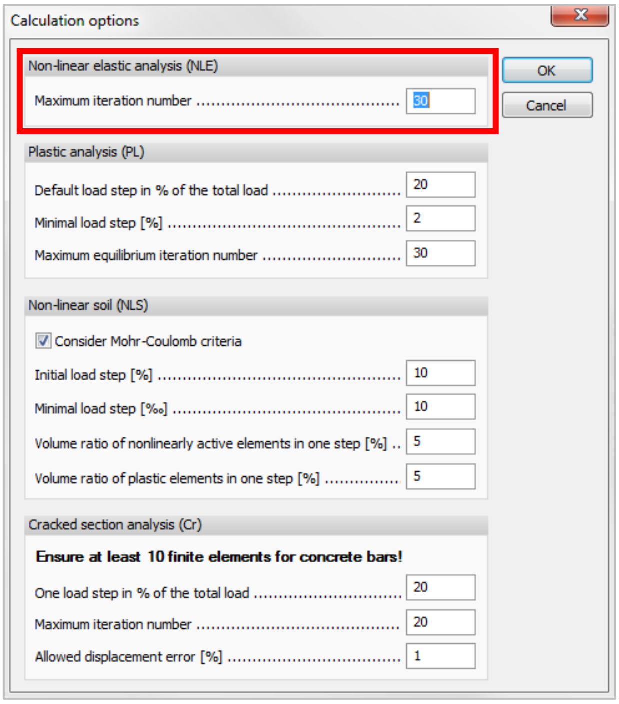
The above mentioned calculation types are described in details below.

Non-Linear Behavior

Non-linear behavior of supports (e.g. uplift), connections and truss members (e.g. tension-only) can be considered in analysis calculations (for load-combinations, imperfections and stability) by ticking *NL* checkbox at *Calculations* > *Analysis* > *Load combinations* > *Setup load combinations*.

"Uplift" can be modeled both in 2D and 3D design modules by defining compression-only *support/connection* (tension = 0 (free)) and by selecting non-linear calculation for a load combination.

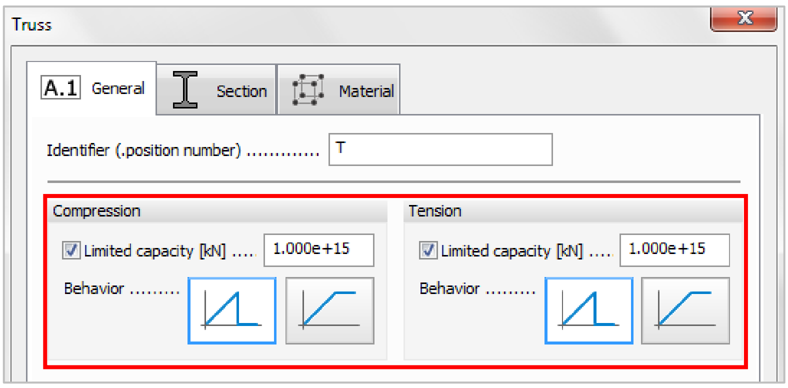
There is a possibility for the user to set the maximum iteration number of nonlinear calculation in *Non-linear calculations* tab in *Setup load combination calculation* dialog.



Plastic Analysis

In FEM-Design 3D Structure there is a plastic calculation option by the setup of load combinations.

Plastic calculation is available for trusses, supports and connections and edge connections of all shell elements (Plane plate and wall, Profiled plate and wall, Timber plate and wall, Fictitious shell).



The options above are considered only for load combinations calculated as non-linear elastic. *Plastic* behaviour is considered for load combinations calculated as non-linear elastic + plastic. See more details in the next chapter.

For further information check the documentation.

Cracked-Section Analysis

Cracked-section analysis means that the displacement of RC plates, walls, columns and beams can be calculated based on their cracked state and designed reinforcement.

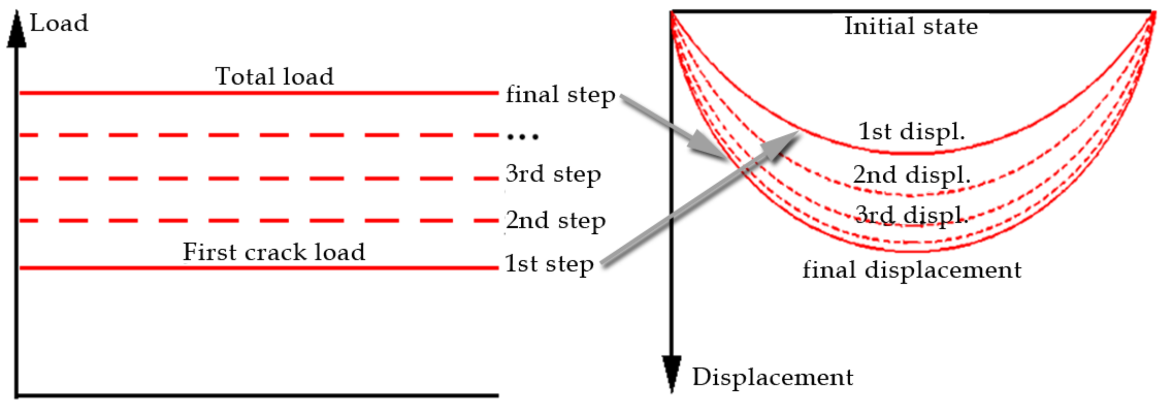
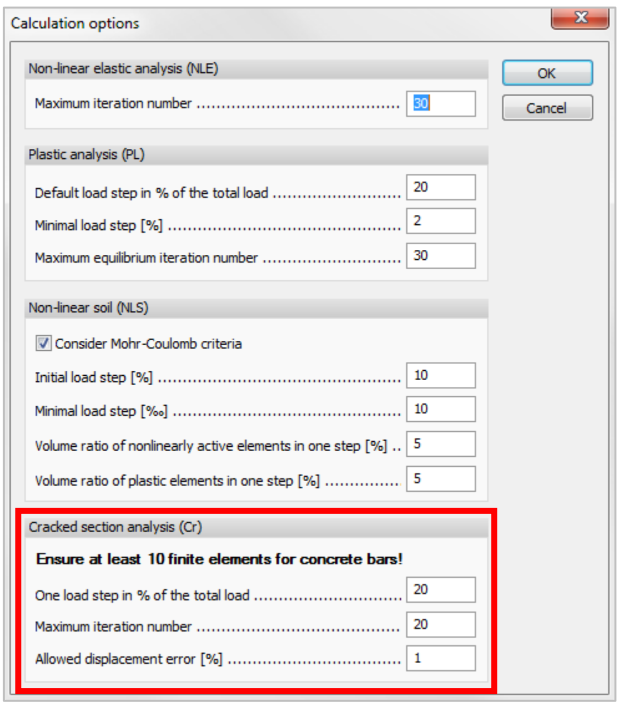


Figure: Iteration steps of cracked-section analysis

Cracked section analysis for load combinations is available by ticking the *Cr.* checkbox at *Analysis* > *Calculations* > *Load combinations* > *Setup load combinations* > *By load combinations*

The iteration process settings available at *Analysis* > *Calculations* > *Load combinations* > *Non-linear calculations*:



- **One load step in % of the total load** (= number of load steps):
  For example, 20% means 5 load steps (= 100/20[%]). Less percentage generates more steps and more running time.

- **Maximum iteration number:**
  The value must be in range 1 and 100.

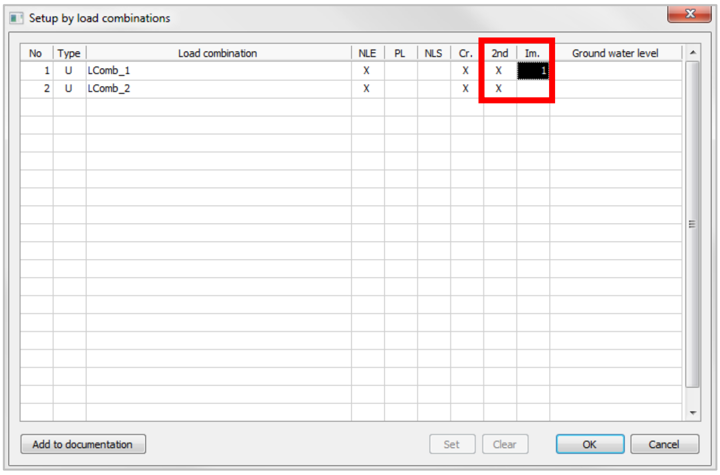
- **Allowed displacement error [%]:**
  Iteration ends, when the relative displacement error becomes less than the allowed value.

Second Order Analysis

In the and modules, 2nd order theory can be applied for load combination calculations of 3D structures. The 2nd order analysis considers the placement of the loads that changes together with the displacement, so it results additional moments derived from the new load positions.

To allow the 2nd order analysis for load combinations, just tick the *2ND* checkbox at *Calculations* > *Analysis* > *Load combinations* > *Setup load combinations* > *By load combinations*

The 2nd order analysis is recommended to be done together with imperfection calculation. In *Setup load combinations* dialog, choose load combinations which you would like to apply the 2nd order theory for, and give the number of imperfection shape (simultaneous or previous calculation for imperfection is needed) you would like to consider for the 2nd order analysis.

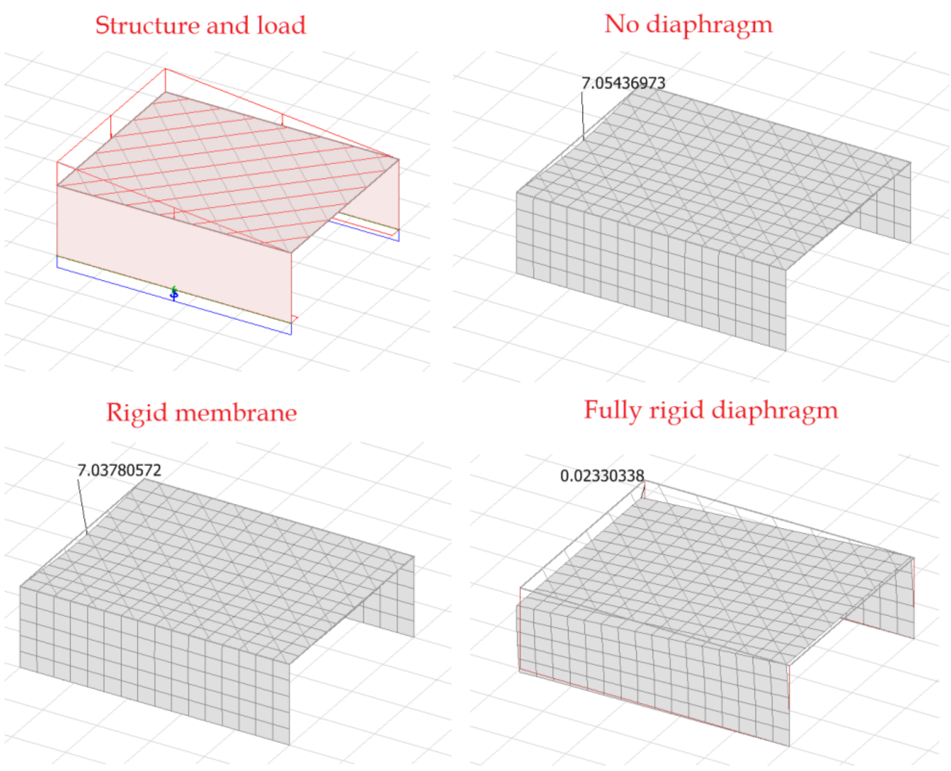


If the loads are too large when using second order analysis, the program stops the calculations with an error message.

Diaphragm calculation
If at least onw diaphragm is defined int he stucture, User can choose one of the following diaphragm calculation options:

- None
- Rigid membrane
- Fully rigid

The difference between them is demonstrated by the following pictures. See details in the documentation.



Analysis for Load Cases and Combinations

Analysis calculations can be done by load case and/or load combination. The next table summarizes the results available for load cases and load combinations by FEM-Design modules.

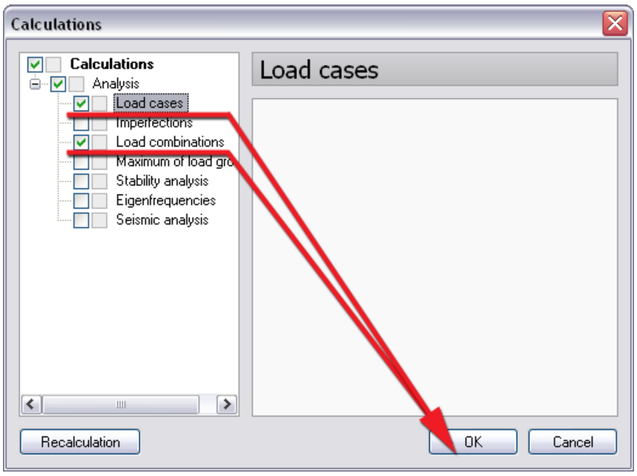
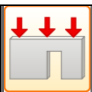
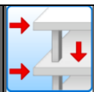


Figure: Starting analysis for load cases and/or load combinations

Analysis result						
Translational displacements	✔ (Plate/Beam)	✔ (Wall)	✔ (Wall)	✔	✔	✔ (Wall/Column)
Rotational displacements	✔ (Plate/Beam)	✔ (Wall)	✔ (Wall)	✔	✔	✔ (Wall/Column)

Reactions	✓	✓	✓	✓	✓	✓
Connection forces	✓	✓	✓	✓	✓	✓
Bar internal forces	✓ (Beam)			✓	✓	✓ (Column)
Shell internal forces	✓ (Plate)	✓ (Wall)	✓ (Wall)	✓	✓	✓ (Wall)
Bar stresses	✓ (Beam)			✓	✓	✓ (Column)
Shell stresses	✓ (Plate)	✓ (Wall)	✓ (Wall)	✓	✓	✓ (Wall)

Table: Basic analysis results by FEM-Design Modules

Displacements

Depending on the current FEM-Design module, the program calculates and displays the model displacement from linear or non-linear (for RC elements: **cracked-section analysis**) analysis. There are two types of displacement results: translational or rotational. For bar elements, the motion and rotation components can be displayed separately (**Detailed result**) by direction (**local axis**).

In *Plate*, the displacements are calculated for the plate regions and beam elements, and the motion is parallel with the global Z direction, so perpendicular to the plate regions. Only reactions can be asked for columns (point reaction) and walls (line reaction).

In *Wall* and *Plane Strain*, the motion is parallel with the calculation plane of the wall regions.

In *PreDesign*, although the 3D model contains all types of elements, displacements are calculated for the vertical elements such as columns and walls.

Displacement results are recommended to be asked for **serviceability load combinations**.

Reactions

Depending on the support types, the program calculates the reaction forces and/or moments in the **supports** by direction component, their resultants and the resultant at the support's center of gravity of line and surface supports.

The *Plate* module calculates reactions in columns and walls too above the point/line and surface supports.

The available result components:

Fx' / Fy' / Fz' - reaction force in the local x'/y'/z' axis of the support (**group**);

Fr - resultant of the reaction force components (*support group*);

F - reaction force of the **single support**;

Mx′ / My′ / Mz′ - reaction moment around the local x′/y′/z′ axis of the support (*group*);

Mr - resultant of the reaction moment components (*support group*);

M - reaction moment of the *single support*.

Connection Forces

Similarly to reactions, the program calculates the forces and/or moments in the connection objects (**Edge connection**, **Point-point connection** and/or **Line-line connection**) by direction component and their resultants.

The available result components:

Fx′ / Fy′ / Fz′ - connection force in the local x′/y′/z′ axis of the connection;

F - resultant of the connection force components;

Mx′ / My′ / Mz′ - connection moment around the local x′/y′/z′ axis of the connection;

M - resultant of the connection moment components.

The figure shows an example for displaying connection forces at the three connection types. The *Fy*' is displayed at the line (line-line and edge) connections, and the *Fx*' and *Fz*' for the point-point connection. (The color of a result component (e.g. *Fy*') is the same with the color of the local axis (e.g. *y*') associated to the component direction) . On the next figure shows the resultants.

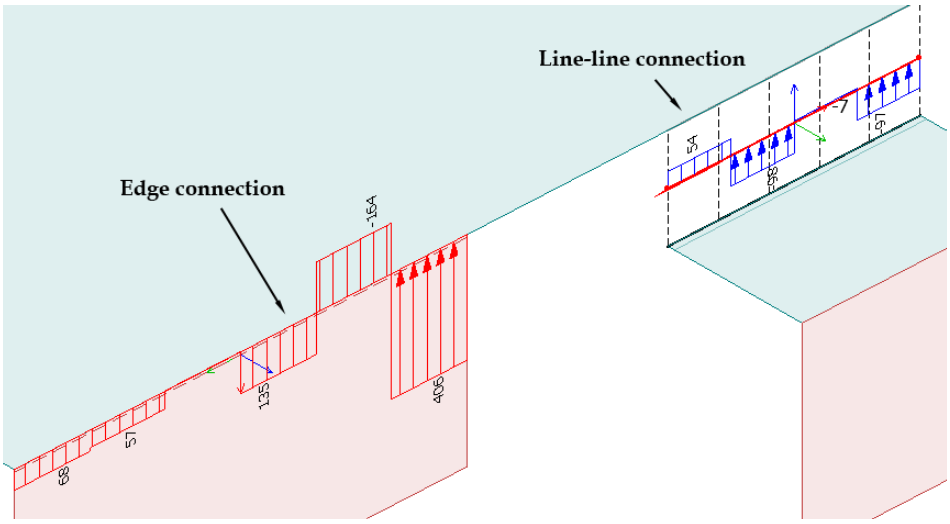


Figure: Connection forces by connection type

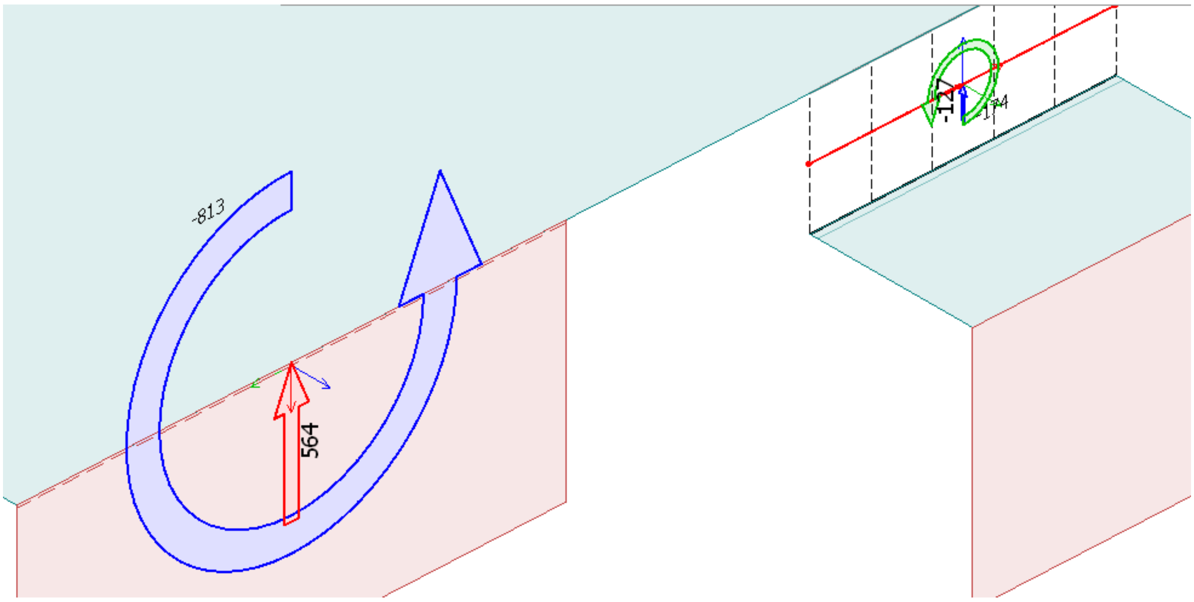


Figure: The resultants (force and moment) for edge connection and line-line connection forces

Local stability results
After calculating the load combinations the Local stability results (Overturning of walls and Sliding) are available in Display results dialog.

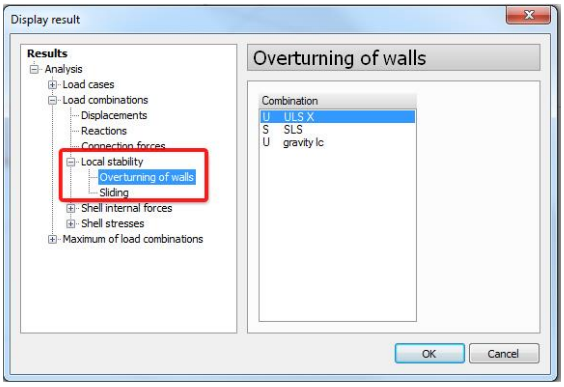


Figure: Display local stability results

- *Overturning of walls*

Only those walls can be calculated which have at least one horizontal edge in the bottom and edge connection is defined for it. The result is expressed as a percentage:

- 0% belongs to the case when the vertical force acts at the centre of bottom edges,
- 100% belongs to the case when the vertical force acts at one of the corners,
- 1000% belongs to the case when the resultant is outside the wall edge.

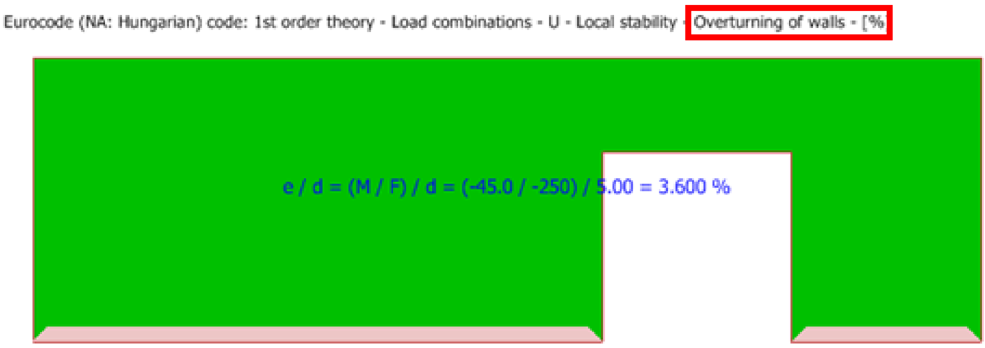


Figure: Overturning of walls

⚠ Overturning of walls results are informative. Without accurate modelling it may lead to incorrect results!

- *Sliding of edge connections*

The result is the ratio of the design force and the friction capacity. The friction factor can be set in the edge connection dialog.

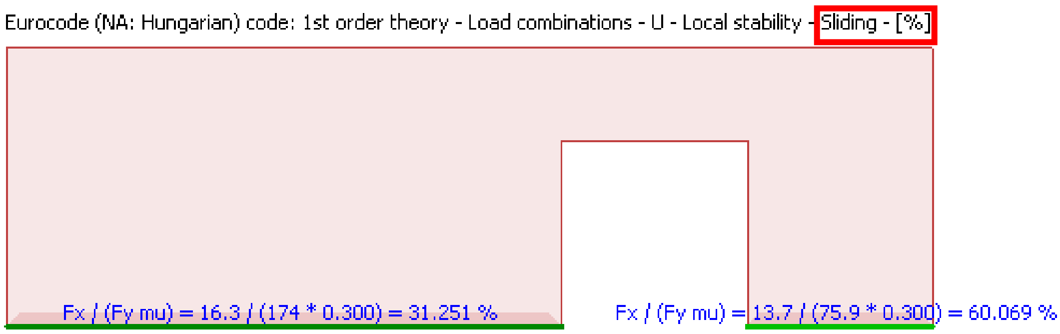
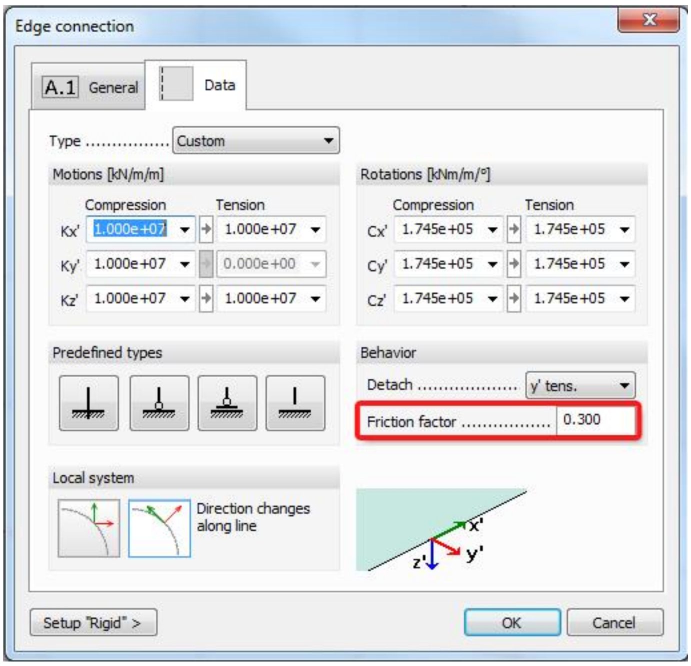
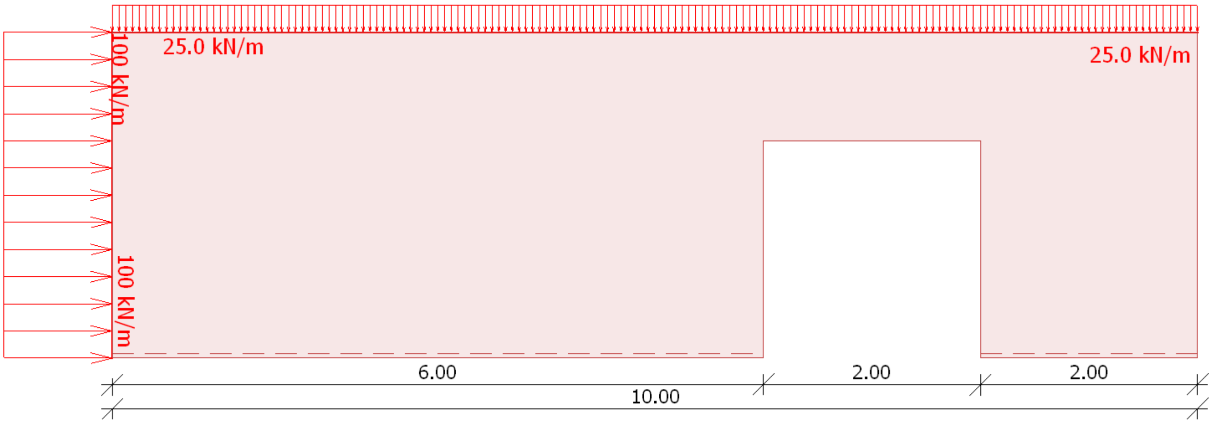


Figure: Sliding of a wall

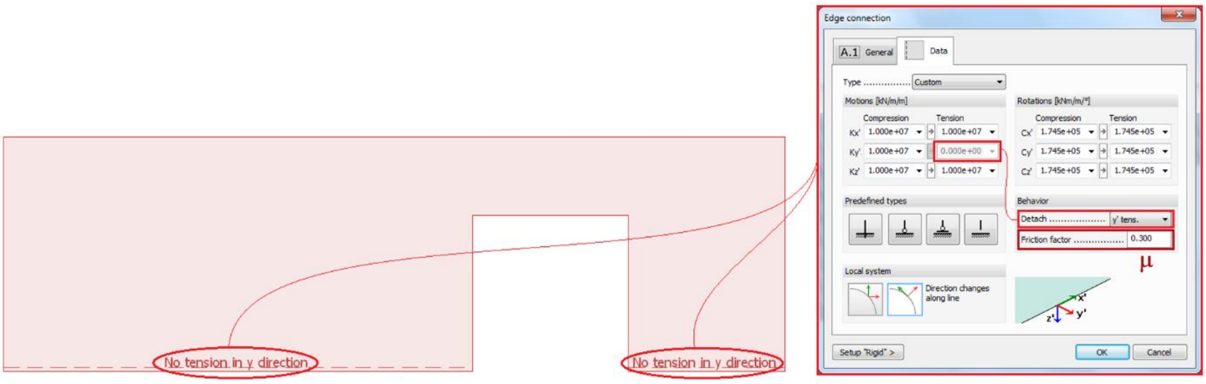


Numerical example below will illustrate the *Local Stability*.

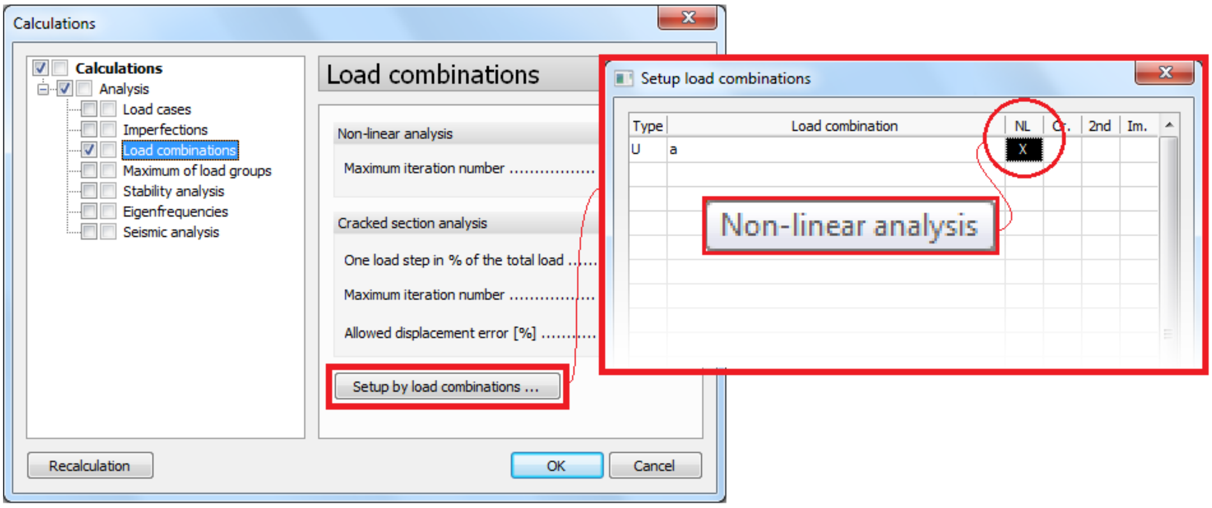
Geometry and Loads



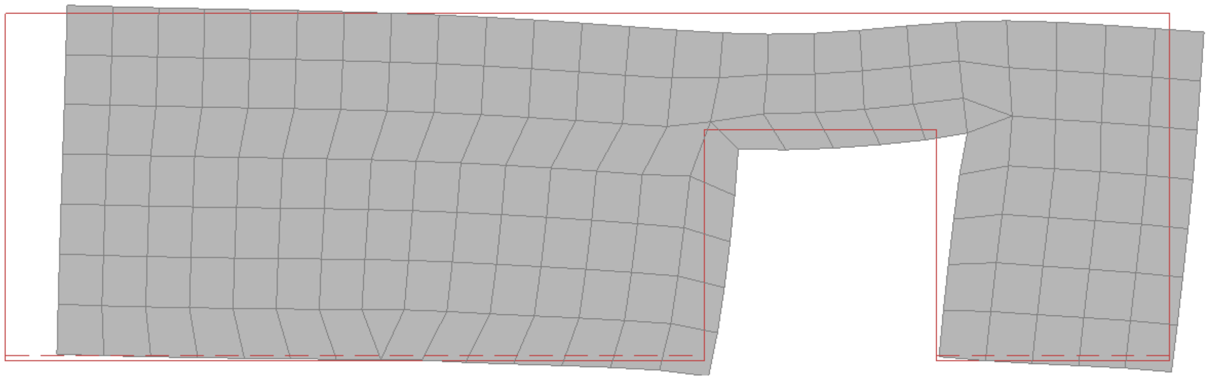
Properties of edge connections



Non-linear calculation (which allows uplift) is recommended to get correct result for local stability.

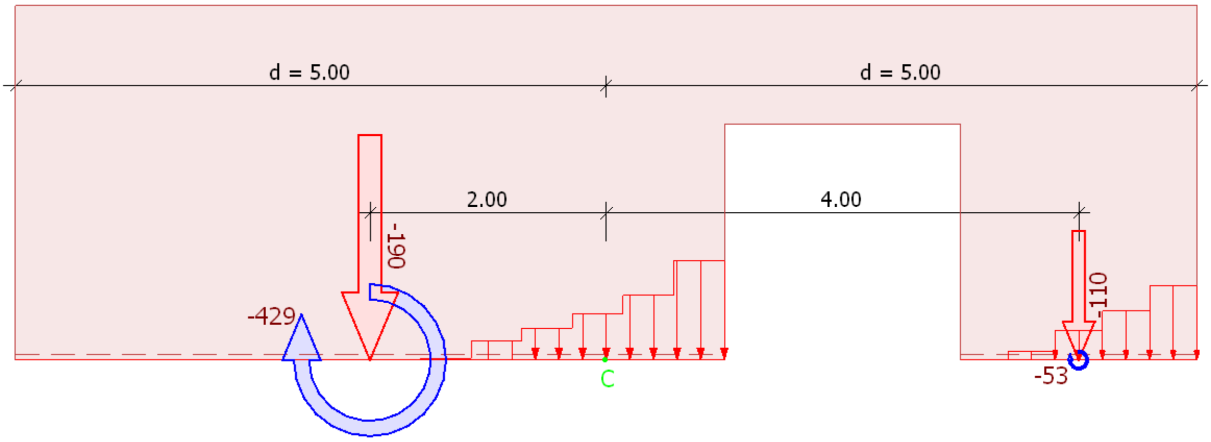


Displacement graph (as well as connection force) is the easiest way to check the uplift.



Overturning of wall

With the help of resultants of edge connections, wall's overturning can be examined as below.

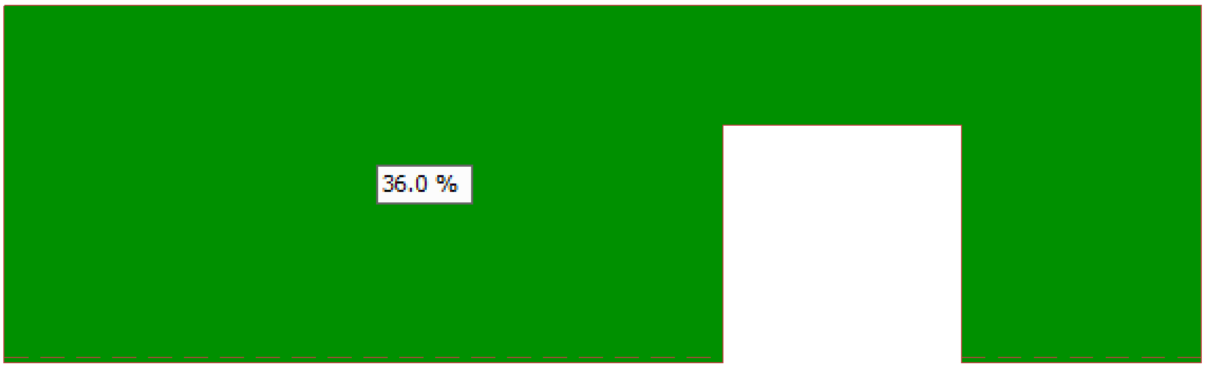


$$F = \sum_{i=1}^{n} F_i = -190 - 110 = -300\ kN$$

$$M^{(C)} = \sum_{i=1}^{n} M_i + F_i * v_i = -429 + 190 * 2 - 53 - 110 * 4 = -542\ kNm$$

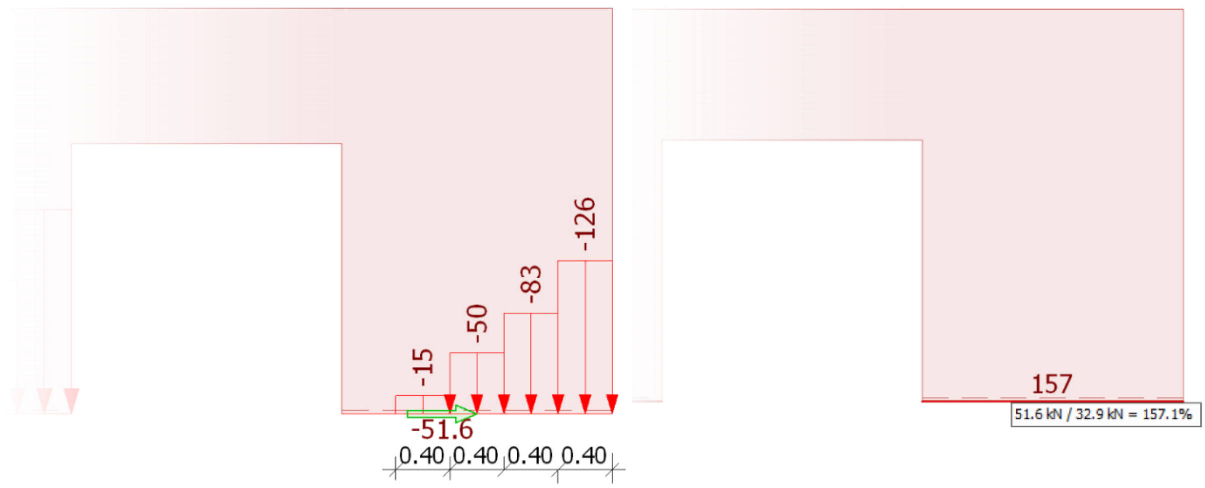
$$e = \frac{M^{(C)}}{F} = \frac{-542}{-300} = 1.81\ m$$

$$Utilization = \frac{e}{d} * 100\ \% = \frac{1.81}{5.00} * 100 = 36\%$$



Sliding of edge connections

Edge connection's sliding is calculated in each edge connection *separately* by comparing the x' component of the connection force as design force, and the limit force calculated by the y' components of the connection forces and the friction coefficient of the edge connection.



$$F_{lim} = \mu * \sum_{i=1}^{n} f_i * dx_i = 0.3 * (15 * 0.4 + 50 * 0.4 + 83 * 0.4 + 126 * 0.4) = 32.9\ kN$$

$$Utilization = \frac{F}{F_{lim}} * 100\ \% = \frac{51.6}{32.9} * 100 = 157.1\%$$

Bar Internal Forces

The program calculates internal forces and/or moments in the bar elements depending on the applied FEM-Design module.

The available result components:

N - normal (axial) force (local x' axis of the bar element);

Ty' / Tz' - shear force in the local y'/z' axis direction of the bar element);

Mt - torsion moment (around the local x' axis of the bar element);

My' / Mz' - bending moment around the local y'/z' axis of the bar element.

⚠ **Truss members** bear only normal forces (N).

The *Plate* module calculates internal forces only for beams. Columns are point supports.

In *PreDesign*, although the 3D model contains all types of elements, internal forces are calculated in columns.

Shell Internal Forces

Depending on the current FEM-Design module, the program calculates internal forces and/or moments in the planar structural elements

The *Plate* module calculates internal forces in the **plate** regions and in the **Global coordinate system**:

Mx'/ My' - bending moment around the **global Y / X axis**;

Mx'y' - torsion moment;

Tx' / Ty' - shear force for the global X / Y normal and in the Z direction;

M1 / M2 - principal moments;

M1/M2 - principal moment directions.

Although *Analysis* calculations give results for the **global Descartes system**, internal forces can be asked and displayed in arbitrary (reinforcement) directions by checking **design forces** in case of **RC design**.

The *Wall* module calculates internal forces in the **wall** regions and in the **Global coordinate system**:

Nx' / Ny' - normal force in the global X / Y direction;

Nx'y' - shear force in the global X-Y plane;

N1 / N2 - principal normal forces;

N1/N2 - principal normal directions.

The *Plane Strain* module calculates only the **shear stresses** in the **wall** regions and in the **Global coordinate system**.

The *3D Structure* module calculates internal forces and moments in the planar object regions (plate and wall) in their local coordinate system:

Mx' / My' - bending moment around the **local y' / x' axis** of the region element;

Mx'y' - torsion moment;

Nx' / Ny' - normal force in the local x' / y' axis of the region element;

Nx'y' - membrane shear force;

Vx' / Vy' - shear force for the local x' / y' normal and in z' direction;

M1 / M2 - principal moments;

M1/M2 - principal moment directions;

N1 / N2 - principal normal force;

N1/N2 - principal normal directions.

The *PreDesign* module calculates the previous internal forces and moments but only walls.

Bar Stresses

FEM-Design calculates the normal stress in bar elements (beams, columns and/or truss members) with the following meaning:

Sigma x'(max) - maximal normal stress (tension);

Sigma x'(min) - minimal normal stress (compression).

The *Plate* module calculates stresses only in beams. Columns are point supports.

In *PreDesign*, although the 3D model contains all types of elements, stresses are calculated in columns.

Shell Stresses

The program calculates stresses in the top, bottom and middle (so called "membrane") planes of the planar elements. The meaning of top and bottom side depends on the position (*Plate* module) or the **local coordinate system** (3D modules) of a region element.

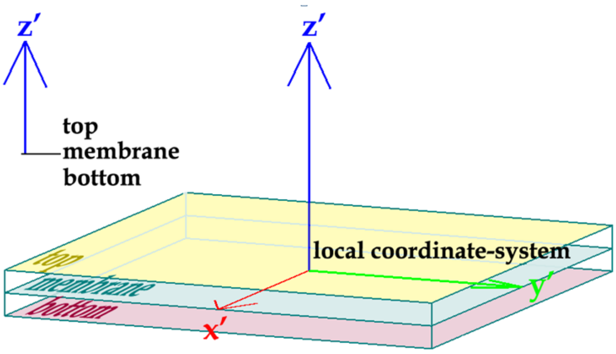


Figure: The meaning of planes depending on region position

So, depending on the current FEM-Design module, we get results from the following:

Sigma x', top / membrane / bottom - normal stress from *Nx'* in top/membrane/bottom plane;

Sigma y', top / membrane / bottom - normal stress from *Ny'* in top/membrane/bottom plane;

Tau x'y', top / membrane / bottom - shear stress from *Nx'y'* in top/membrane/bottom plane;

Tau x'z' - shear stress (x' normal and z' direction);

Tau y'z' - shear stress (y' normal and z' direction);

Sigma vm, top / membrane / bottom - von Mises stress in top/membrane/bottom plane;

Sigma 1/Sigma 2, top / membrane / bottom - principal stresses and directions in top/membrane/ bottom plane.

The x', y' and z' directions are valid in the global coordinate system at *Plate* and in the local coordinate system of planar elements in the 3D modules.

In *PreDesign*, although the 3D model contains all types of elements, stresses are calculated in walls.

In *Wall* and *Plane Strain*, stresses are calculated only in the membrane plane.

Equilibrium Check

The program automatically checks the equilibrium of the analysis calculations. Statical equation is written to the origin [0; 0; 0] of the **Global Coordinate System**. It compares the sum of the reactions and the sum of applied loads. Equilibriums can be asked by load case and load combination.

Just click the *Equilibrium* icon (in Analysis or **Design** mode), choose a load case or load combination to see the equilibrium check results.

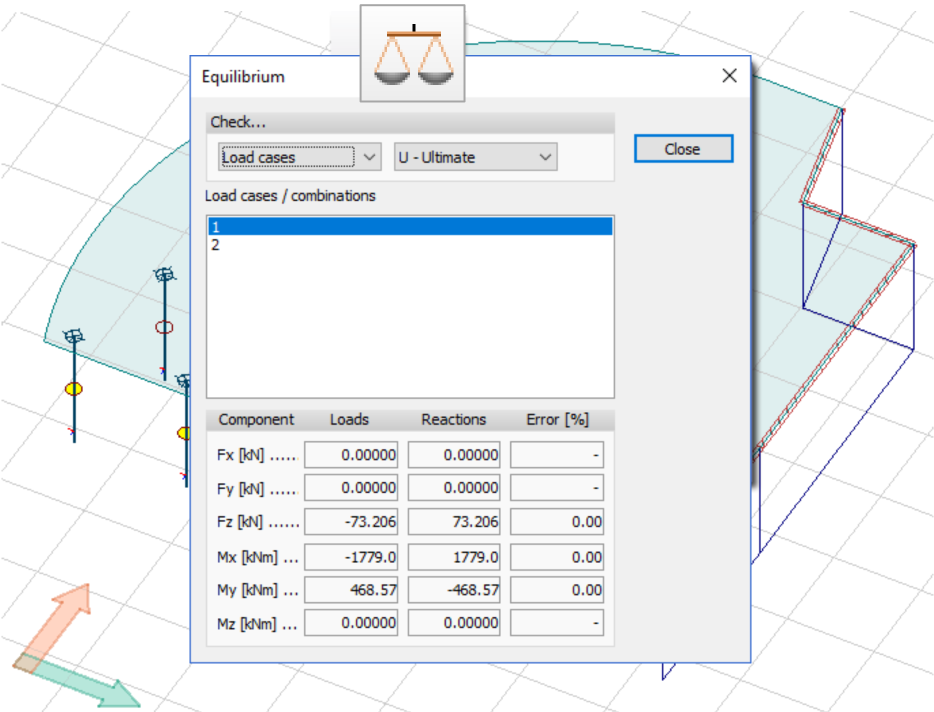


Figure: Equilibrium check of Analysis calculations

If equilibrium error derives from an analysis calculation, the error will be appeared in percentage in the Error column by equation types (force (F) and moment (M)) and directions (x, y and z directions of the global coordinate system). "Error" shows the differences between the resultants of the queried loads and the calculated reactions.

Analysis for Maximum of Load Combinations and Groups

Choosing *Load combinations* for Analysis automatically generates results for the maximum of load combinations too.

If you define **Load groups** and choose *Maximum of load groups* for Analysis, FEM-Design calculates maximum/minimum results (in all finite element nodes) from the most unfavorable combinations of the load groups according to the applied code.

So, maximum and simultaneous results of **displacements**, **reactions**, **connection forces**, internal forces (**bar** and/or **shell**) and stresses (**bar** and/or **shell**) can be calculated for maximum of load combinations and groups.

The symbol "+" and "–" sign the direction of the maximal value in the valid systems: local or global coordinate systems (depend on the current FEM-Design module). Some examples for the meaning of "+" and "-":

Displacement:

ez'(+) - maximal uplift in global z' direction in *Plate*,

- maximum motion in the positive direction of element's local system in 3D modules;

ez'(-) - maximal depression in global z' direction in *Plate*,

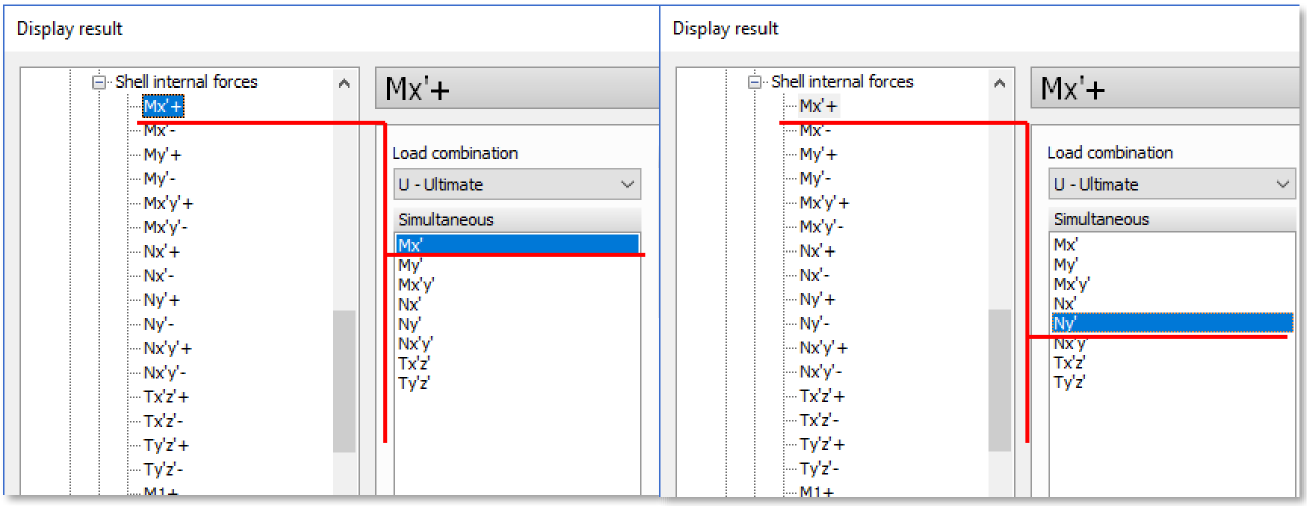
- maximum motion in the negative direction of element's local system in 3D modules;

Internal forces:

Mx'(+) - maximal bending moment around the y' axis (global/local) in positive direction (= same direction with the axis direction).

Mx' (-) - maximal bending moment around the y' axis (global/local) in negative direction (= opposite direction to the axis direction).

The next figure shows the meaning of simultaneous results.



Maximal Mx' Nx' belongs to maximal Mx'

Figure: The meaning of simultaneous results belong to a maximal value

Combination of *Load cases* that gives the maximum analysis results in *Maximum of load groups* can be listed in tables. Just use the *List* command (*Tools* menu) for the *Maximum of load groups* result data.

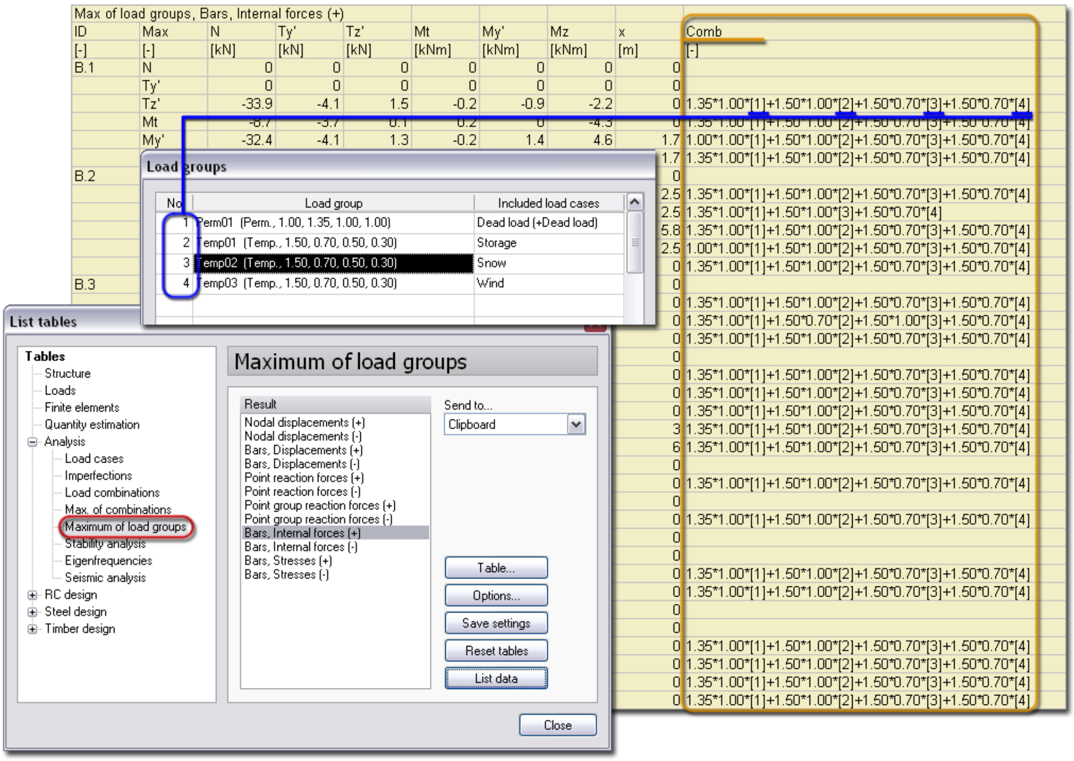


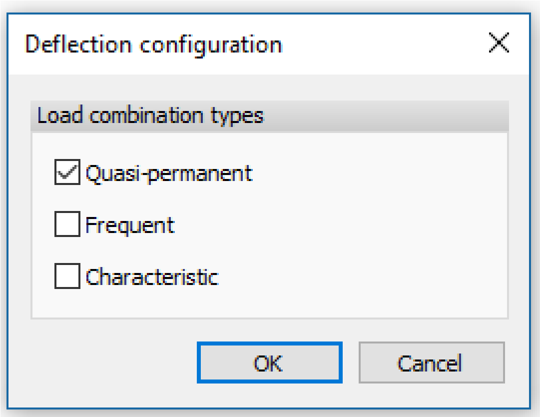
Figure: Combination of load cases for maximum of load groups results

Deflection check for RC, steel and timber bars

A new checking criteria is available for reinforced concrete, steel and timber bars. Deflection utilization is calculated for *load combinations*, *Maximum of load combinations* and *Maximum of load groups* according to the user defined serviceability limit states.

This new result type is based on the displacement of the bars and the deflection limitation settings which can be defined by the so-called *deflection lengths*.

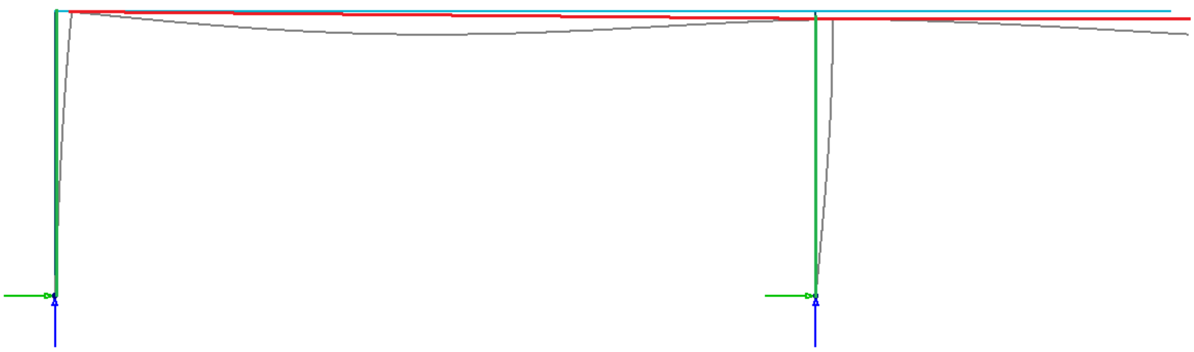
In the  *Deflection configuration* dialog, we can specify the types of load combinations/groups for which the deflection check is performed.



Deflection lengths are used to define those bar segments, where the deflection checking criteria/limitations are coincide. The "Simply supported" deflection lines are denoted with blue arcs below the bar, the "Cantilever and column" types are orange and the "not relevant" types are black. Relative and/or absolute limit can be set for each *length* individually. If both are requested the dominant one will be calculated and displayed.

The first option we can set here is the behaviour of the lengths which affects the calculation method of the deflection. If we choose not relevant for a specific length, it will be excluded from the checking process.

For the better understanding of the next two options, namely the *Simply supported* and *Cantilever* mode let us consider the following example, a cantilever frame structure.

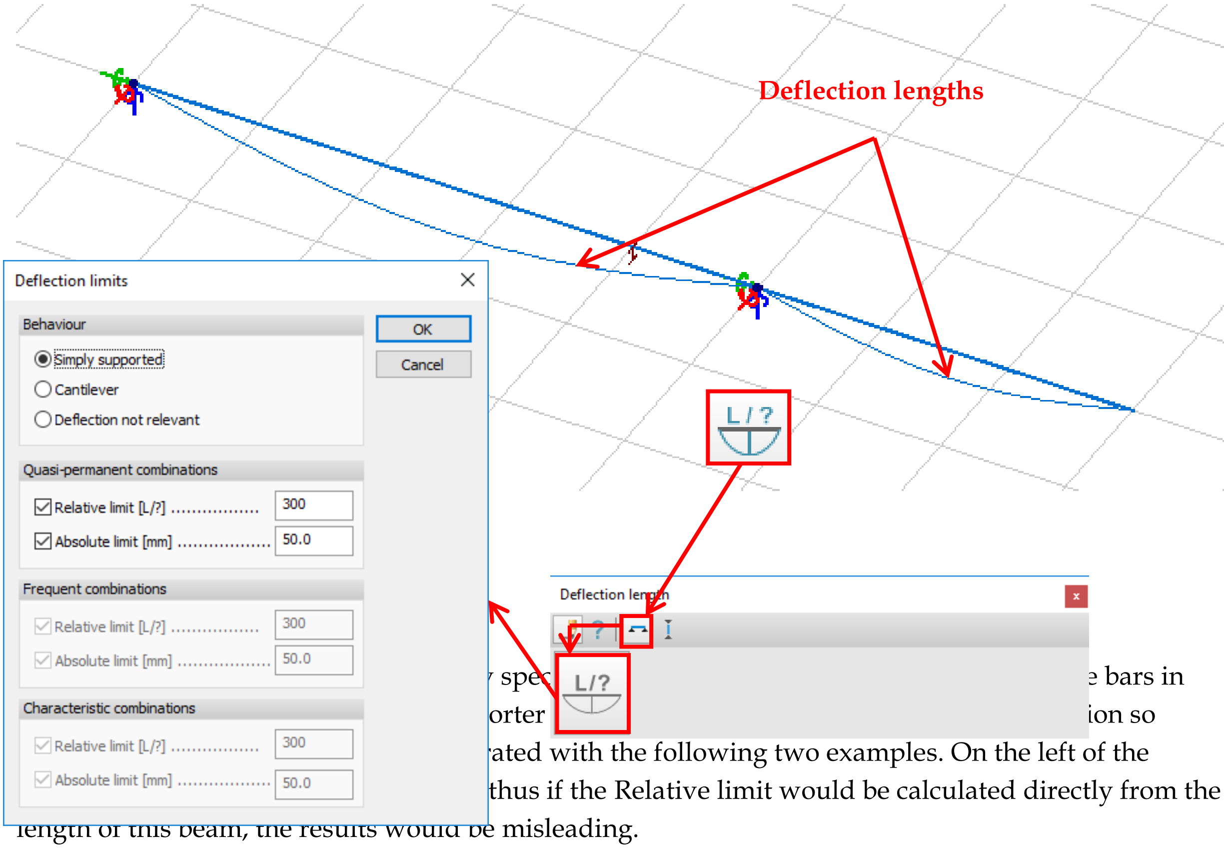


In the midspan we should use the Simply supported option, where we eliminate the rigid body motions in such a way that we connect the endpoints of the length, and measure the deflections of the middle sections from this imaginary line (red skew line on the picture above).

As a consequence of this method, the deflections of the endpoints are zero, the dominant section is usually at the middle of the length.

On the cantilever, we would like to use the cantilever mode, where the dominant value of deflection on this length will be the difference between the maximum and minimum absolute deflection (in this example the largest distance from the red horizontal line). For columns the same calculation method is used, the only difference is that the deflection is measured in the horizontal plane (from the green lines).

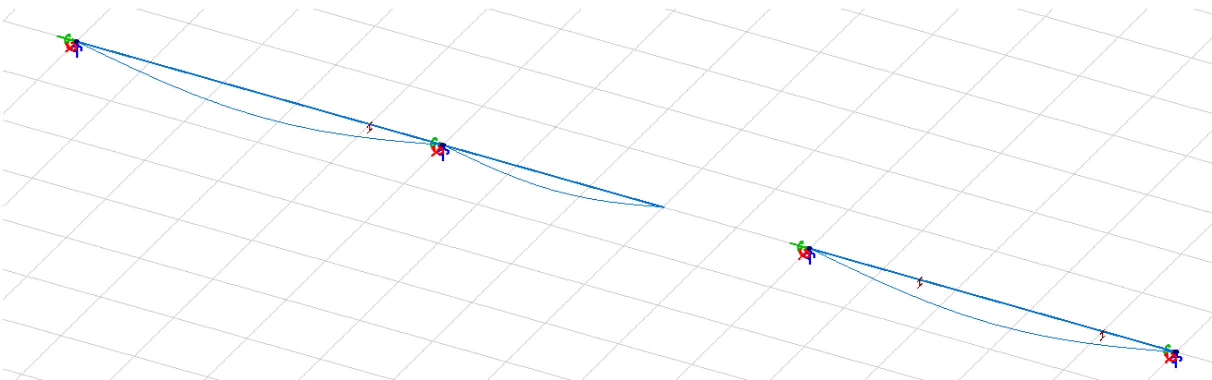
For columns only this (*"Cantilever and column"*) option is available.



… spec … e bars in … orter … ion so … ated with the following two examples. On the left of the … thus if the Relative limit would be calculated directly from the length of this beam, the results would be misleading.

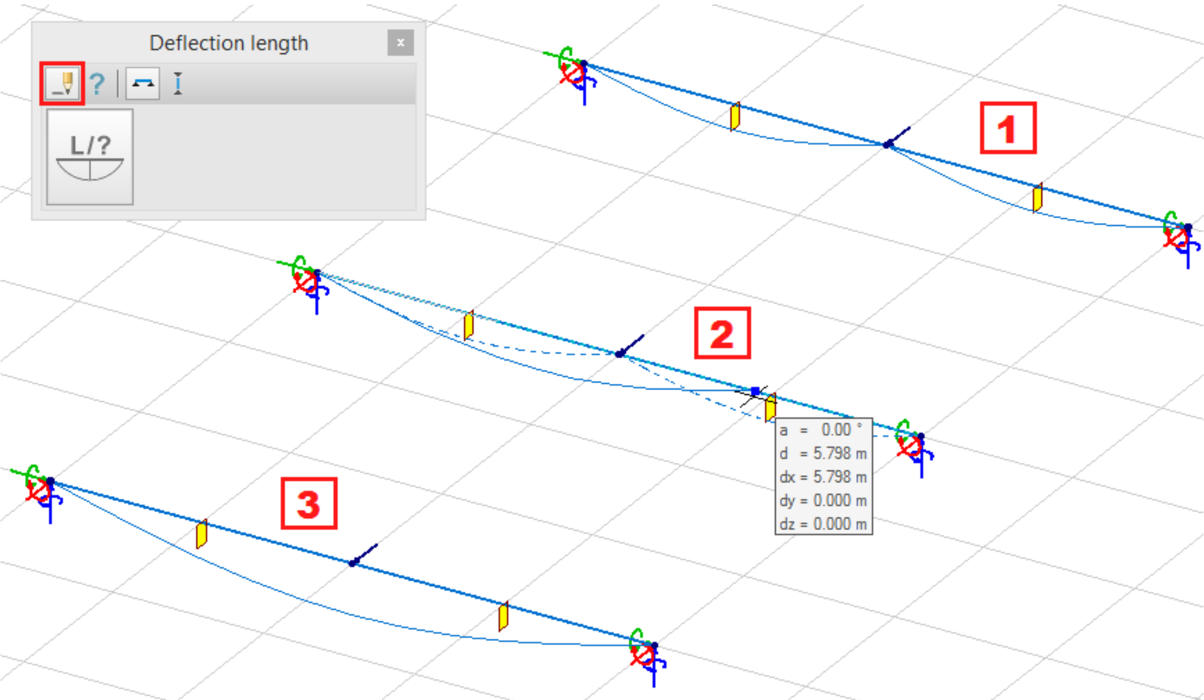
In other words, in the *L/?* formula, instead of the length of the midspan or the cantilever, the whole length of the beam would be substituted. Therefore, we need two Deflection lengths to differentiate the *L* in the Relative limit formulae on the midspan and on the cantilever. In addition, the limit value also can differ for the two types of structure according to the National Annexes.

In the second case (to the right on picture below) imagine that our aim is to design a beam splice and check deflections. Two beams need to be drawn for the steel design, but of course during the deflection check we want to use the summed length of the two beams for the calculation of the relative limit value. For this purpose, we define one Deflection length over the two bars - this way we make correct calculations in both cases without any additional modification on the structure.

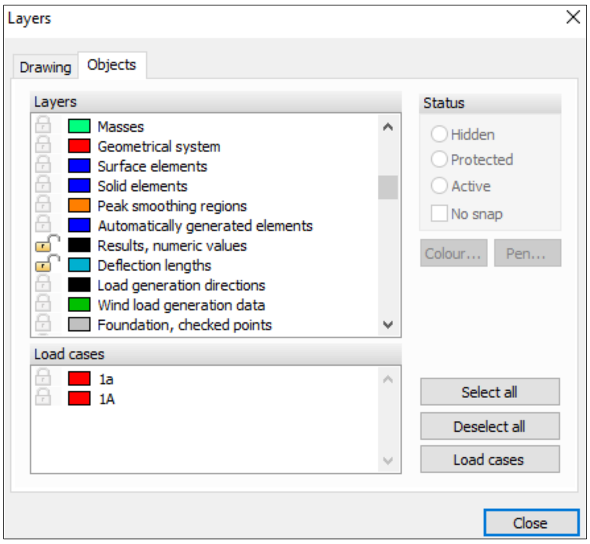


It is worth to note that in the second case we had two beams, but in contrary to the buckling lengths, definition and editing of deflection lengths can be performed on such set of beams, which are both parallel and continuous.

By default, deflection lengths are generated automatically. This procedure first search all the previously mentioned parallel and continuous beams sets, then intersect them with the edges/axes of the structural elements (beams, columns, trusses, plates, walls, line and surface supports) and point supports. In the majority of the cases deflection lengths obtained by this way are reasonable from engineering point of view, but in some cases we may want to modify them. A good example can be a structure consisting of two beams with a horizontal support, which should not be considered in the deflection checking process. The following flow diagram illustrates the modification of the two beams step by step. By default, as we can see in the upper picture, the automatically generated deflection lengths coincide with the beams because they are intersected with the horizontal point support. If we would like to have one deflection length over the two beams, we can draw it between the support groups using the Define tool, similarly to the buckling lengths. By this way, the new length substitutes the original ones!



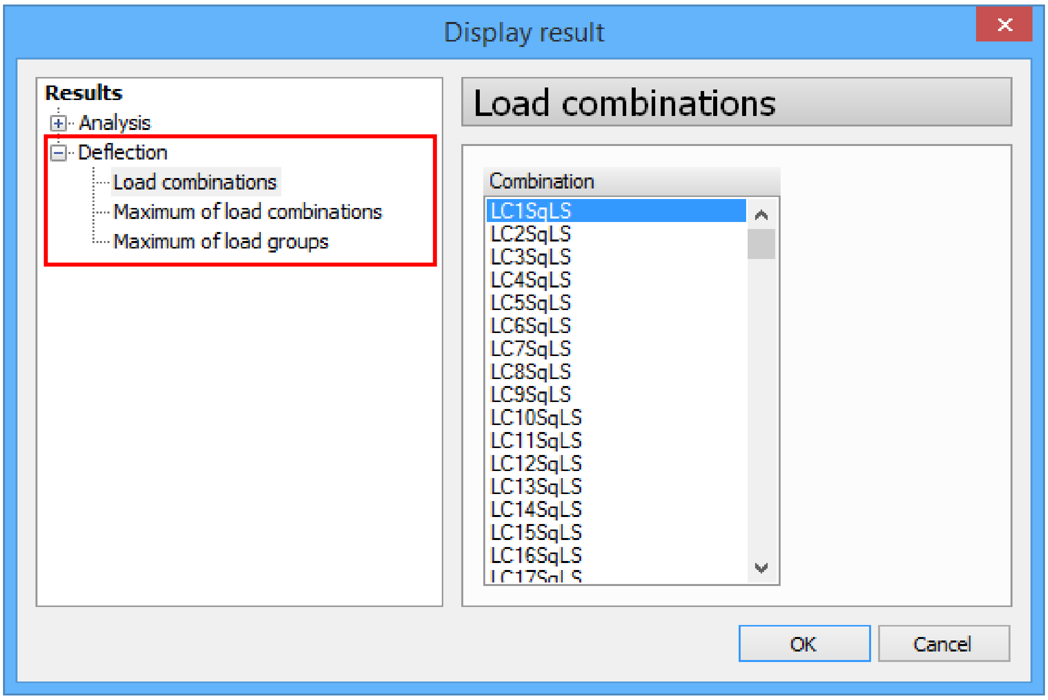
⚠ Deflection length has its own layer.



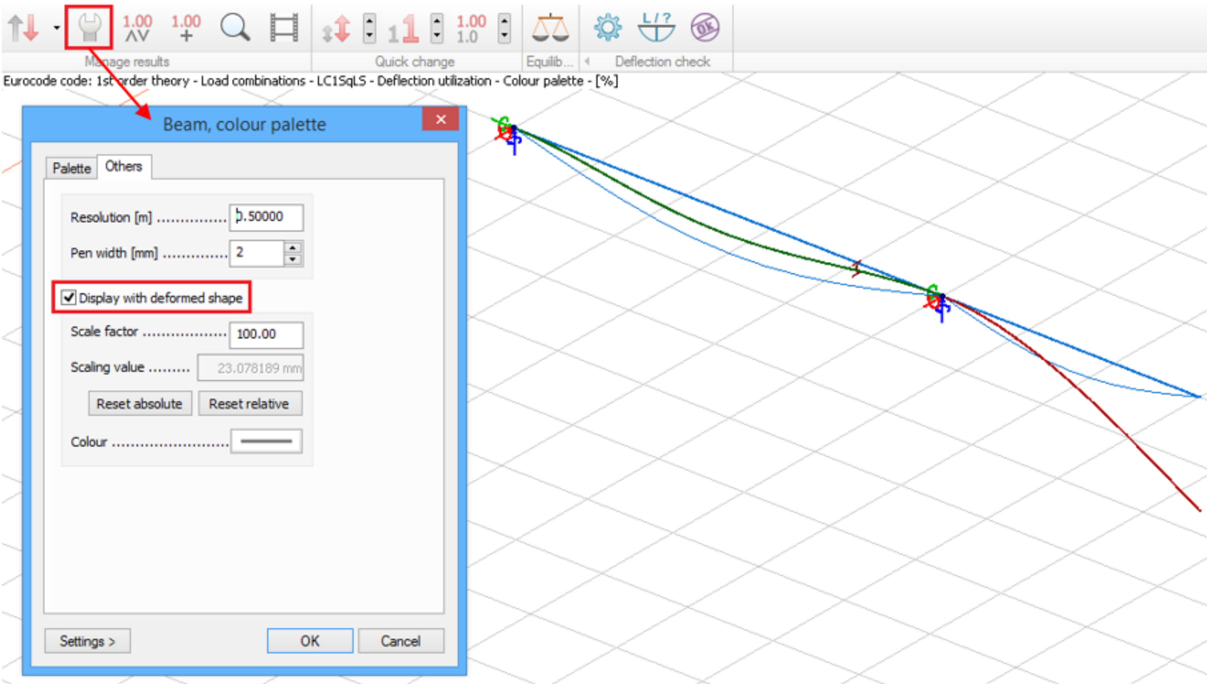
Deflection check button becomes active if Load combinations and/or Load groups are already calculated. The utilization results can be displayed from the *New result* dialog.

⚠ The Deflection checking process considers only the straight beams and columns. For beams the deflection is measured along their own local z' axis, for columns it is measured in the global horizontal x-y plane.

Results requested for a *Load combination* can be displayed both on the deformed and undeformed shape.



Due to the fact that the limit values of the calculation are controlled by the Deflection lengths, the result is constant along them. In other words we have one (dominant) utilization value for each Deflection length. Results for *Maximum of load combinations* and *Maximum of load groups* are only displayed on the undeformed shape of the structure.



Imperfections

Imperfection calculation is run only for steel bar elements of the structure. Users can add imperfections to a structure in two ways:

- **Imperfection modeled by defining loads (manual)**

  Place for example horizontal point and line loads on a multi-storey building to model imperfection manually.

- **Imperfection calculation according to the formula EC3: 1-1 (automatic)**

  For load combinations, the program can calculates the probable imperfect shapes in real dimensions from the mode shapes (get from **stability analysis**) according to Eurocode. **Second order analysis** must be run by using imperfection. To do automatic imperfection calculations, activate *Imperfections* and set the required number of the imperfect shapes (*Rqd.* cell) for the load combination which you would like to run imperfection for.

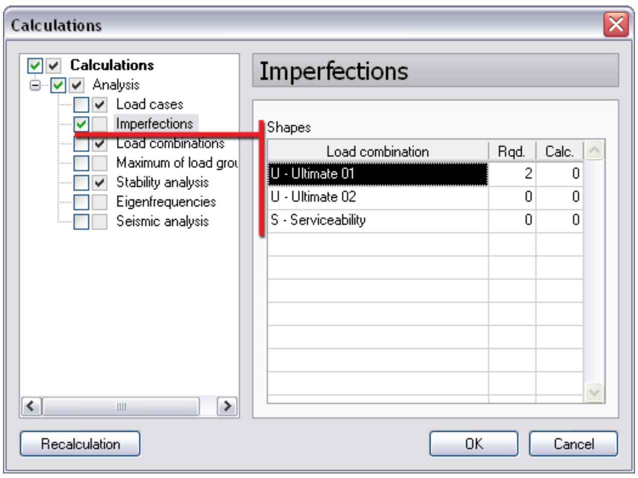


Figure: Imperfection calculation by load combination

For the automatic imperfection calculation you got the buckling shape of the structure with real size in real dimension. Critical parameter assigned to a buckling shape is also available with the following meaning:

critical parameter = critical buckling force/actual load

or in other words:

if the critical parameter is bigger than 1, the structure or a part of it is sufficient to perform the stability analysis; if it is smaller it is not.

If the critical parameters differ a lot between the buckling lengths, the first buckling shape is the critical. If the critical parameter values are close to each other, it is your decision what structural part you check by its shape.

The factor defines the real imperfect shape, so:

*imperfect shape in real dimension = factor * buckling shape*

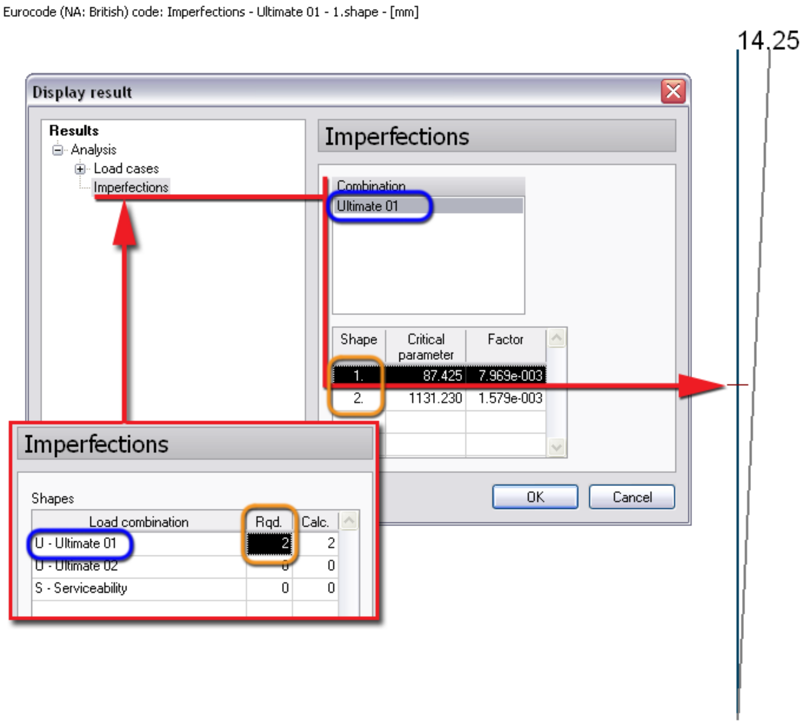


Figure: Automatic imperfect shape calculation

Before imperfection calculation, it is recommended to set minimum 4-5 ***division numbers*** (finite elements) for bars.

Stability Analysis

In 3D modules, global stability of the structure can be analyzed automatically if it is requested. Similarly to **Imperfections**, the program calculates buckling shapes together with their critical parameters for selected load combinations.

To do stability analysis, activate *Stability analysis* and set the required number of the buckling shapes (*Rqd.* cell) for the load combination which you would ask stability results for.

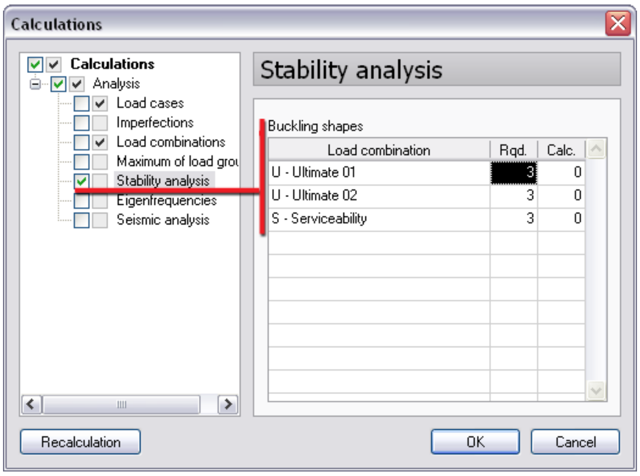


Figure: Stability analysis by load combination

As result, you got the buckling shape(s) of the structure with unit dimension. The greatest displacement value of the buckling shape is 1 and the others are the ratio of that.

Critical parameter assigned to a buckling shape is also available with the following meaning:

critical parameter = critical buckling force/actual load

or in other words:

if the critical parameter is bigger than 1, the structure or a part of it is sufficient to perform the stability analysis; if it is smaller it is not.

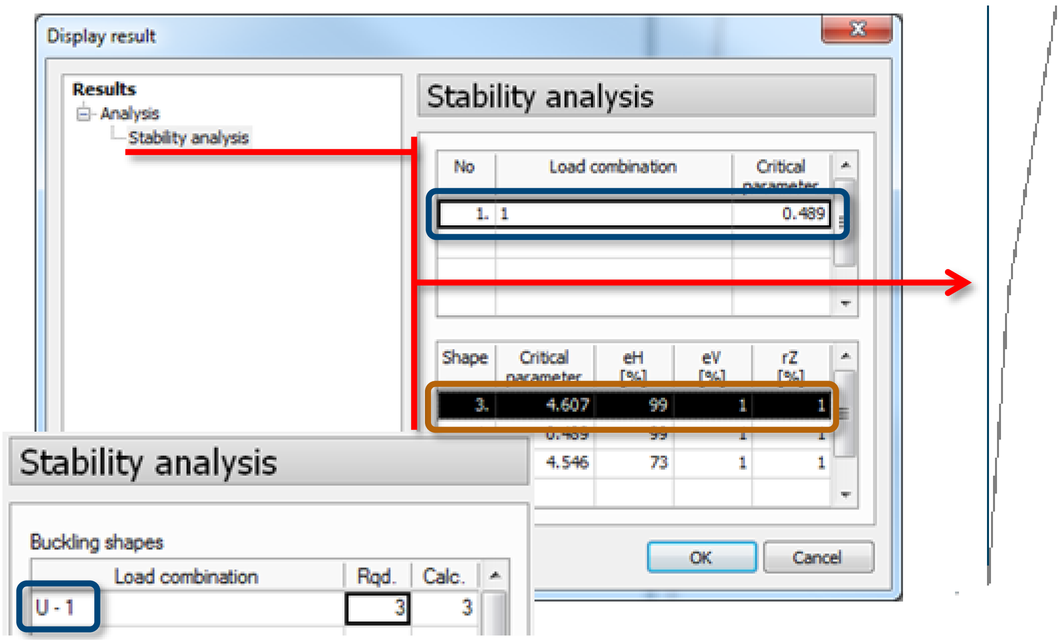
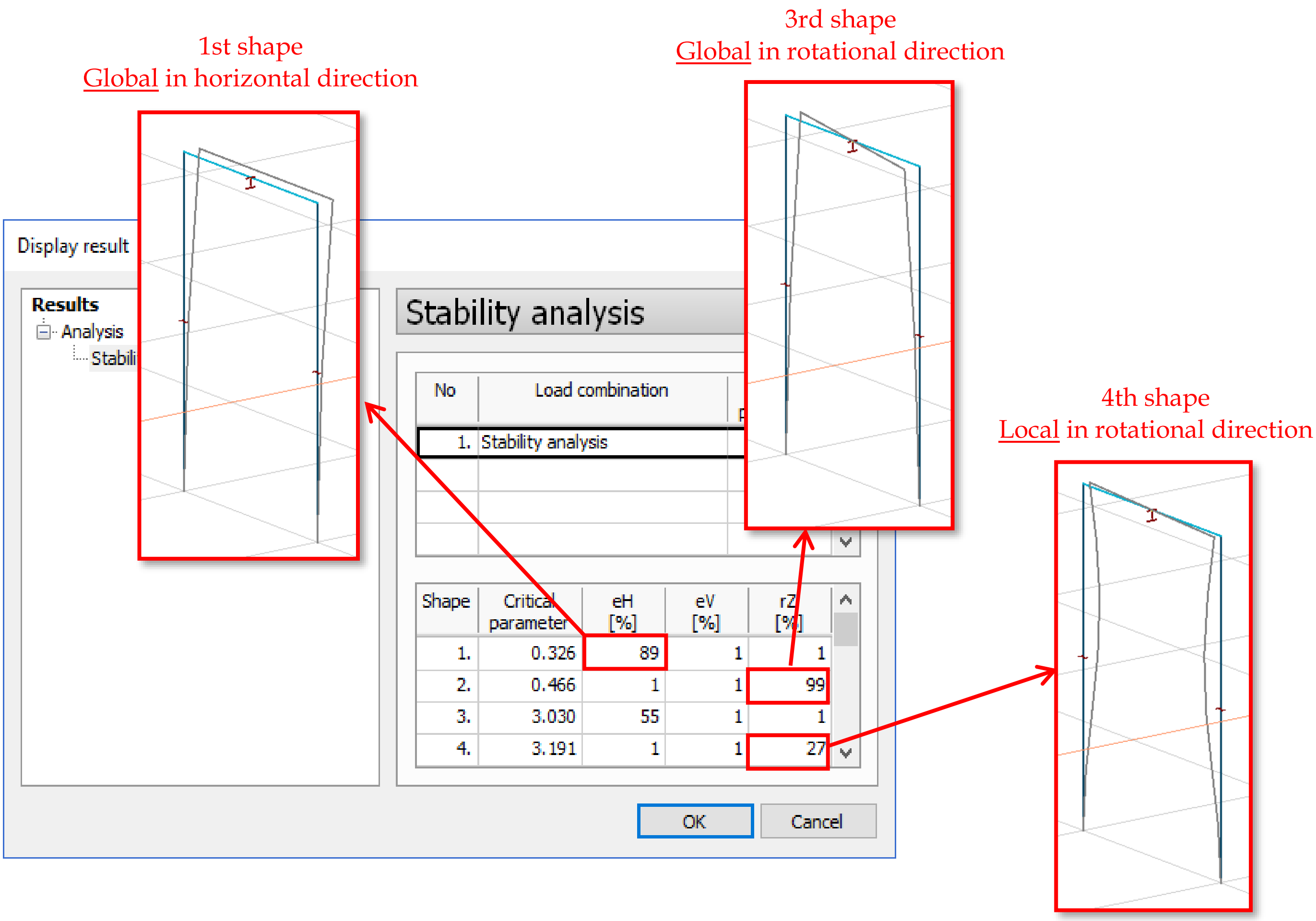


Figure: Buckling shape calculation

The last three columns shows the probability of the buckling shapes are global or local, where *eH* meant for horizontal displacement, *eV* for vertical displacement (global Z direction) and *rZ* for rotaion around global Z axis.

In the example below, the *eH* value of the first shape is 89%, which means it is probably a global buckling shape with horizontal displacement.



Displaying the result (see the leftmost inset above) and examining the buckling shape shows that this is indeed a case of global buckling with the horizontal displacement of the frame's top.

The same structure's second shape possesses a very high *rZ* value (99%), meaning this almost certainly is a global torsional buckling shape (shown in the middle inset).

The fourth shape's *eH*, *eV* and *rZ* values are significantly lower, which implies it is a local buckling shape. As the rightmost inset shows, the assumption was correct (local buckling of both columns).

⚠ Higher probability values shows high probability that the shape is global. If there are not enough shapes calculated, none might be global.

Before stability analysis, it is recommended to set minimum 4-5 **division numbers** (finite elements) for bars.

Eigenfrequencies

Mass/Vibration shape

FEM-Design can do dynamic analysis by calculating vibration shapes of the structural model and the belonging eigenfrequencies and free vibration time values (periodic time).

To do dynamic calculation, activate *Eigenfrequencies* and just set the required number of the vibration shapes (*Number of shapes* cell), the active masses in X, Y or Z direction and the *Top of substructure*.

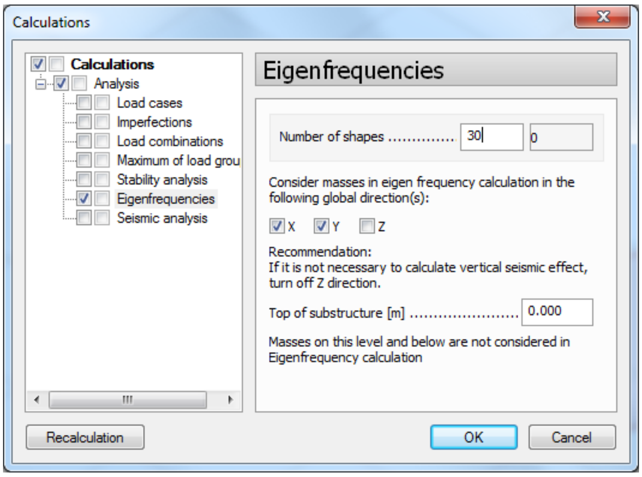
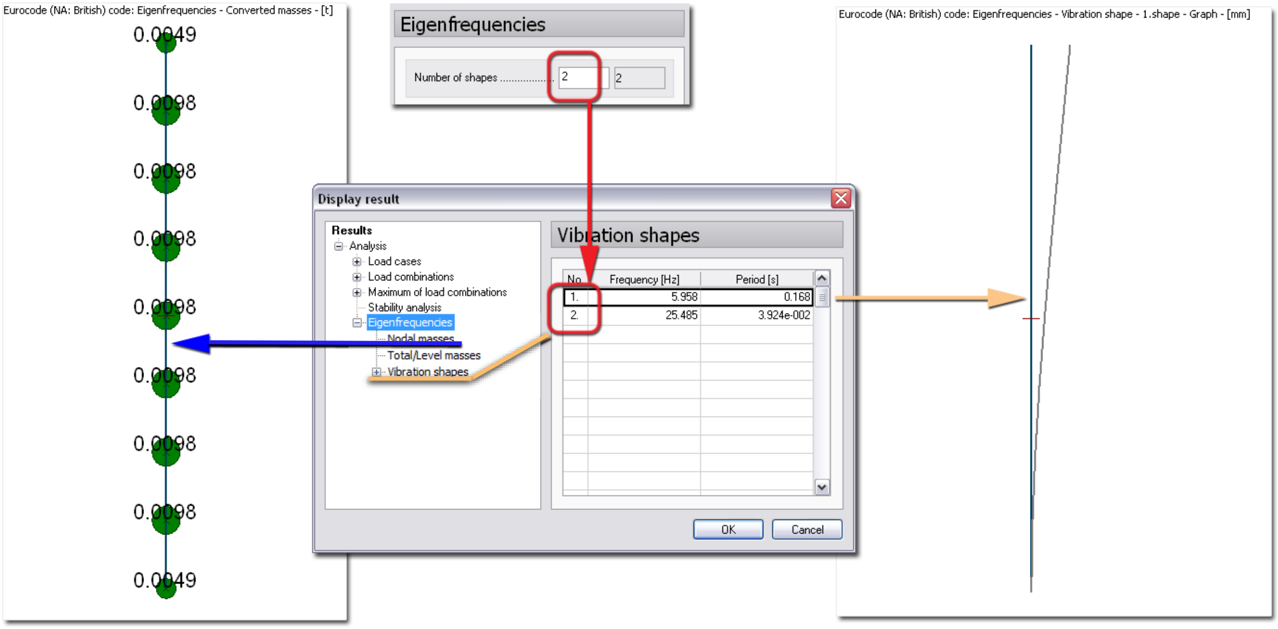


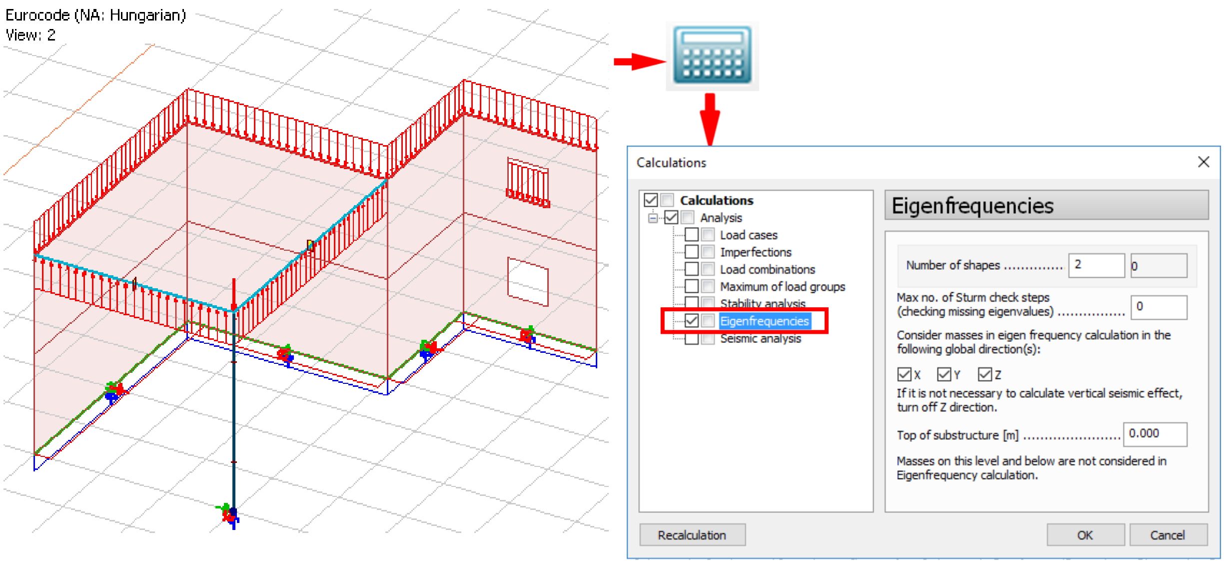
Figure: Dynamic calculation

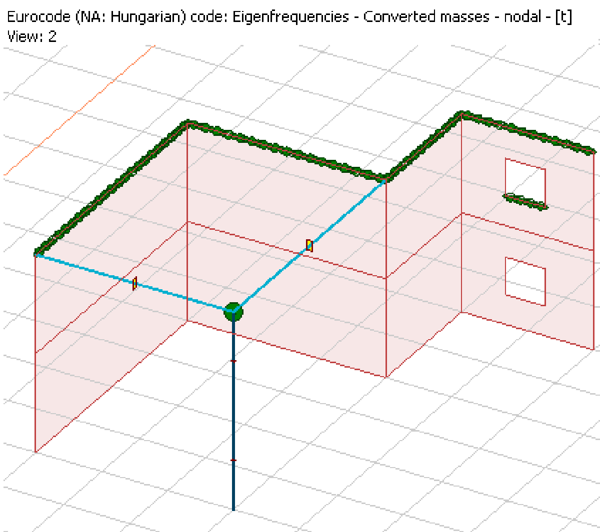
Dynamic calculation requires **masses** to be predefined.

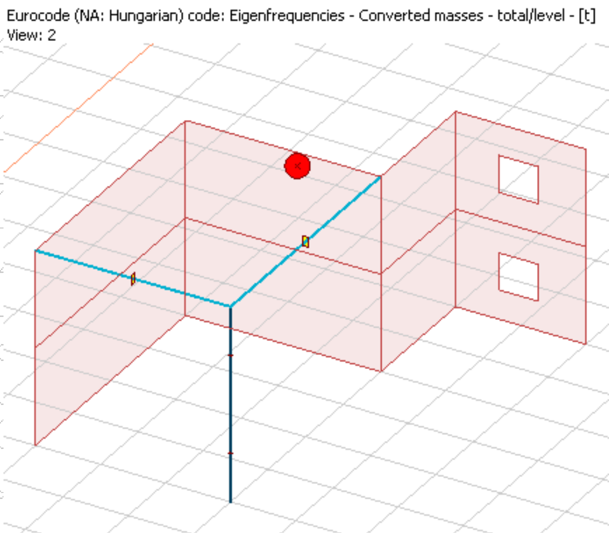
Seismic analysis needs the eigenfrequencies calculations.

In Calculation / Eigenfrequencies dialog the user can set the level of top of the substructure. The masses will be neglected <u>at</u> and <u>under</u> this level.


Eurocode (NA: British) code: Eigenfrequencies - Converted masses - [t]
0.0049
0.0098
0.0098
0.0098
0.0098
0.0098
0.0098
0.0098
0.0049
Eigenfrequencies
Number of shapes
2
2
Display result
Results
Analysis
Load cases
Load combinations
Maximum of load combinations
Stability analysis
Eigenfrequencies
Nodal masses
Total/Level masses
Vibration shapes
Vibration shapes
No.
Frequency [Hz]
Period [s]
1.
5.958
0.168
2.
25.485
3.924e-002
OK
Cancel
Eurocode (NA: British) code: Eigenfrequencies - Vibration shape - 1.shape - Graph - [mm]


Eurocode (NA: Hungarian)
View: 2
Calculations
Calculations
Analysis
Load cases
Imperfections
Load combinations
Maximum of load groups
Stability analysis
Eigenfrequencies
Seismic analysis
Eigenfrequencies
Number of shapes
2
0
Max no. of Sturm check steps
(checking missing eigenvalues)
0
Consider masses in eigen frequency calculation in the following global direction(s):
X
Y
Z
If it is not necessary to calculate vertical seismic effect, turn off Z direction.
Top of substructure [m]
0.000
Masses on this level and below are not considered in Eigenfrequency calculation.
Recalculation
OK
Cancel


Eurocode (NA: Hungarian) code: Eigenfrequencies - Converted masses - nodal - [t]
View: 2


Eurocode (NA: Hungarian) code: Eigenfrequencies - Converted masses - total/level - [t]
View: 2

In the mass centre of the masses the total mass is displayed with red circle.

To get the whole structure's mass centre position set the level of the Top of the substructure a bit under the structure.

This function is useful to neglect the foundation mass in the eigenfrequency calculation so the total mass contribution in Modal analysis can reach >=90%.

Results of Eigienfrequencies calculation:

Masses - mass matrix of **point masses** and/or **masses calculated from load cases** converted into finite element nodes;

Vibration shape - vibration shape and associated eigeinfrequency (*Frequency*) and periodic time (*Period*).

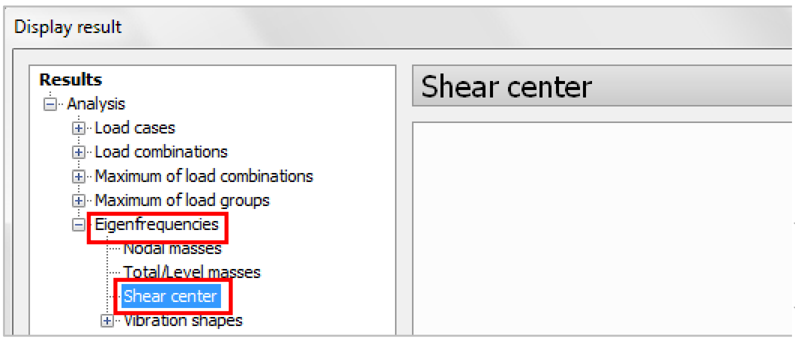
Figure: Results of dynamic calculations

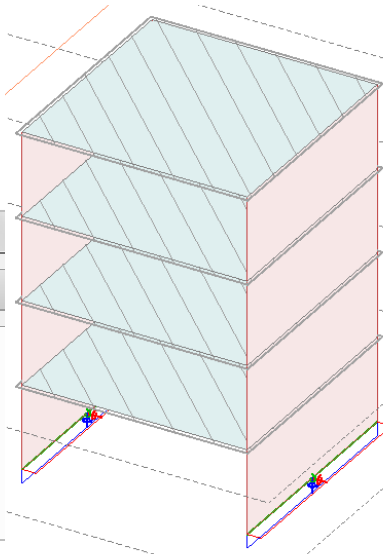
Before dynamic analysis, it is recommended to set minimum 4-5 **division numbers** (finite elements) for bars.

Shear center result

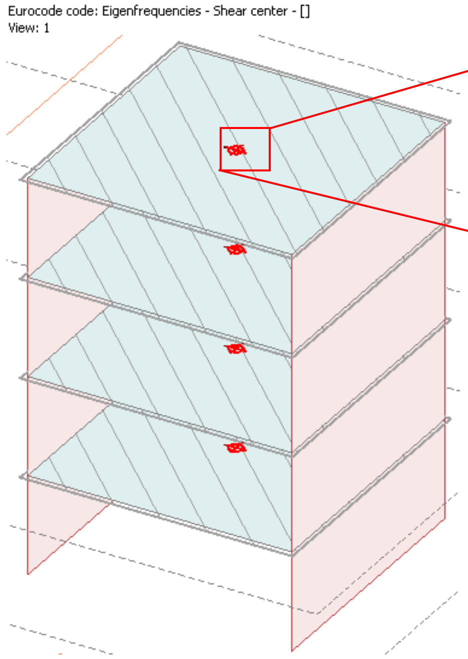
FEM-Design can calculate *Shear centers* for each storey of a building. The figures below show a shear center result of an Eigenfrequency calculation.

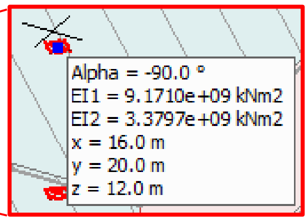
For displaying shear center, diaphragms are needed for every storey.





Each displayed shear center represents the result of a calculation based on the storeys below that storey. For example, the calculation of the center displayed on "Storey 2" takes also "Storey 1" and "Foundation" into account.

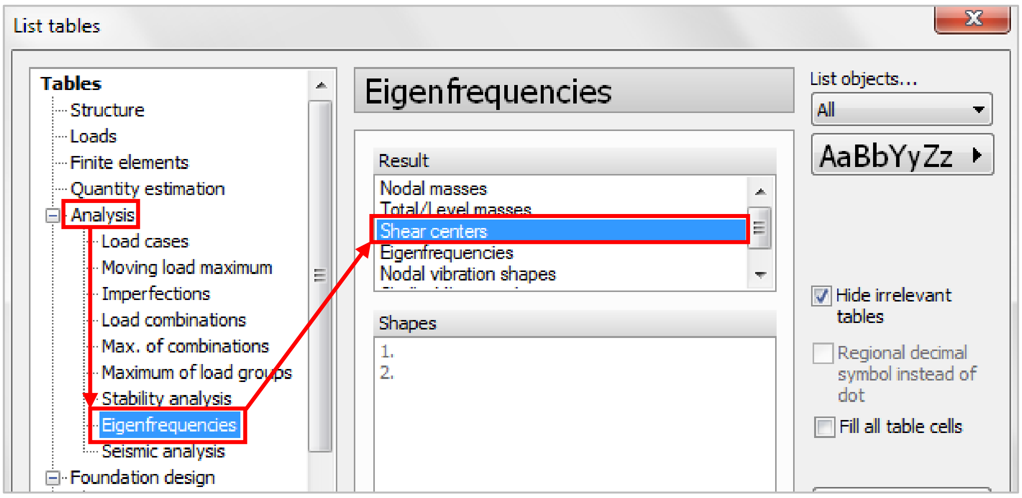




Values in the Tooltip:

- alpha: the angle between the X axis of the global coordinate system and stiffness directions,
- EI1, EI2: stiffnesses in principal directions of strains,
- x, y, z: the global coordinates of the shear center.

Shear center results can be listed in *List tables dialog/Analysis/Eigenfrequencies/Shear center.*



Shear centers

Alpha	EI1	EI2	x	y	z
[rad]	[kNm2]	[kNm2]	[m]	[m]	[m]
-1.571	885272790.059	540718616.178	16.000	20.000	3.000
-1.571	3449176940.867	1528795003.672	16.000	20.000	6.000
-1.571	6326495388.797	2475347677.958	16.000	20.000	9.000
-1.571	9170982655.342	3379707268.708	16.000	20.000	12.000

Seismic Analysis

Methods

In the and modules, seismic calculation offers the following methods to the users according to Eurocode 8.

- **Modal response spectrum analysis ("Modal analysis")**
- **Lateral force method / Equivalent static load method**

  This method can be used to calculate the seismic effect in horizontal plan, x' and/or y' direction. The main point is to calculate "base shear force" taking into account the base vibration period and design spectrum in x' or y' direction, which is distributed into those nodes of the structure where there are nodal masses. The "base shear force" formula is taken from *EC-8 4.3.3.2.2(1)P*. The "base shear force" is nothing else than the total seismic force of inertia that acts between the ground and the structure, and it can be distributed in two ways:

  - **Linear shape method (Static, linear shape)**

    The distribution of the "base shear force" happens according to a simplified fundamental mode shape, which is approximated by horizontal displacements that increased linearly along the height.

  - **Mode shape method (Static, mode shape)**

See the detailed description and the applied theory of all calculation methods in the *Theory book*. This guide introduces only the user interface and the steps of seismic analysis.

Steps of Seismic Calculation

The suggested steps of seismic calculation are the followings:

1. **Mass definition**

   To calculate the seismic effect, it is necessary to know the vibration shapes and corresponding periods (except the *Static, linear shape* method). To perform dynamic calculations, it is necessary to define mass distribution which can be defined as **concentrated mass** or **load case-mass conversion**.

2. **Design spectrum definition**

   The program contains predefined **design spectra** according to *EC8*, but you can also define your own spectra. Use the command **Seismic load** (*Loads* menu).

3. **Dynamic calculation**

   Dynamic calculation should be done before performing seismic calculation, which gives sufficient vibration shapes of the structure. Although setup for the seismic calculation can be done at any time, but the seismic calculation could be performed only after *Eigenfrequency* calculation. Run dynamic calculation under *Analysis* by setting the required number of vibration shapes.

   It is suggested to set the finite element number bigger than 1 at bars (*Finite elements/* **Division number**).

4. **Settings of seismic calculation**

   A national code always provides which seismic calculation method has to be performed for different structure, where and when it should be performed and what other effects to be considered (e.g. torsional effect, P-Δ effect). *FEM-Design* provides three types of calculation methods (depending on the applied code):

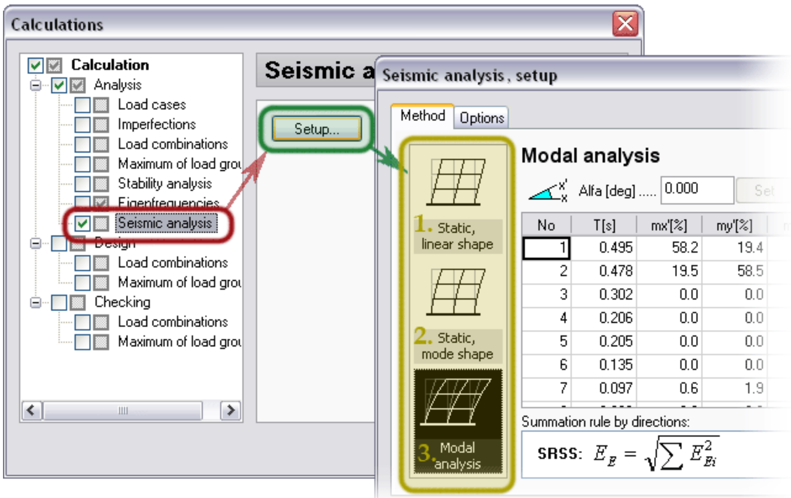
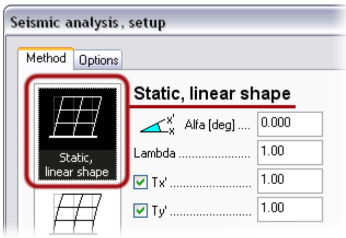


Figure: Settings of seismic analysis

- **Static, linear shape**

  As a matter of fact, eigenfrequency calculation is not necessary for this method, because giving the base period time in *x′* and *y′* directions (*Tx′* and *Ty′*) is enough for the calculation. Practically, eigenfrequency calculation performs before setting this data, but these data can be defined using experimental formulas as well. Investigation can be done in *x′* or *y′* direction, or both together.

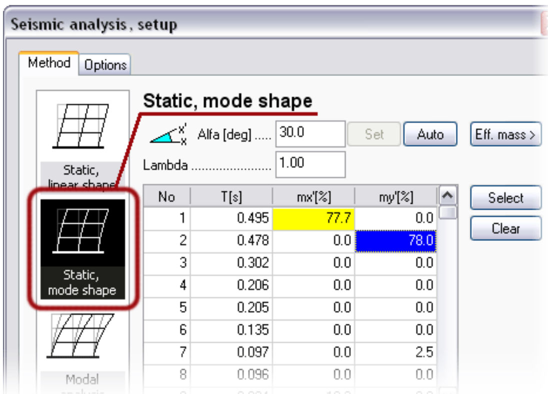


You may set the calculation direction to be performed by selecting the desired direction. To set the desired *x′*-*y′* direction, you should give *Alfa* (α is the angle between the global *X* and *x′*; see **Direction of the horizontal effect**). α =0.0 means *x′*-*y′* directions coincide with global *X*-*Y* directions.

⚠ This method is unusable, if the whole foundation is not in same plane or the horizontal foundation is elastic. In these cases, *Static, mode shape* or *Modal analysis* should be used.

- **Static, mode shape**

  In this method the distribution of "base shear force" happens according to fundamental mode shapes (base vibration shapes).



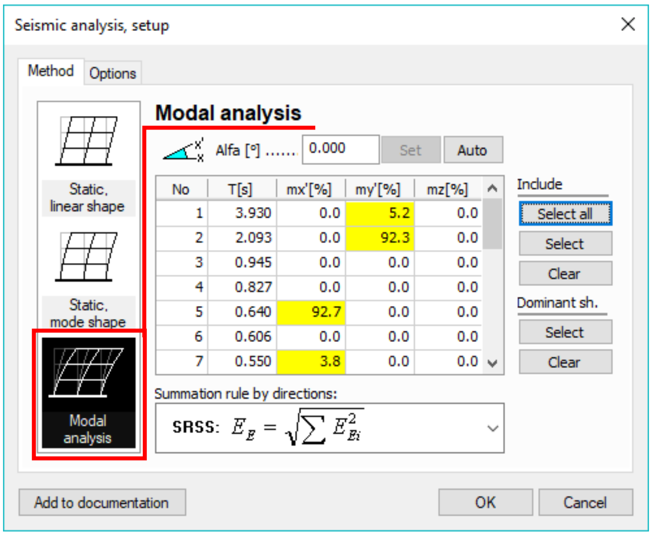
The table shows how to select the base vibration shapes. It contains all mode shapes (*No*), the vibration time (*T(s)*) and effective masses of the mode shapes in *x′* and *y′* directions (*mx′(%)* and *my′(%)*). The effective masses are given in a relative form to the total or reduced mass of the structure. The reduced mass means the total mass above the foundation or above the rigid basement. The value of the effective mass refers to how the mode shape responds to a ground motion direction, so the effective mass shows the participation weight of the mode shape.

Select (or double-click on it) one mode shape in x' or/and y' direction(s) (mx' /my'). (Yellow field color shows the activation.)

It is recommended to select that mode shape which gives the largest effective mass as the fundamental mode shape.

The calculation of "base shear force" is performed according to the total mass of the structure and not to the effective mass.

- **Modal analysis**



The essence of the method is the calculation of the structural response for different ground motions by the sufficient summation of more vibration shapes. Method gives possibility to take into account full x', y' and z (=global Z) direction investigation.

In the table, more vibration mode shape could be selected in x', y' and z directions if necessary. The last row (orange cells) of the table shows that how large is the sum of the considered effective masses compared to the total or reduced mass of the structure in a given ground motion direction.

According to *EC8*, sum of the effective mass of the choosen mode shapes (at least in horizontal direction) should reach 90% of total mass. Additionally every mode shape has to be taken into account where effective mass is larger than 5%.

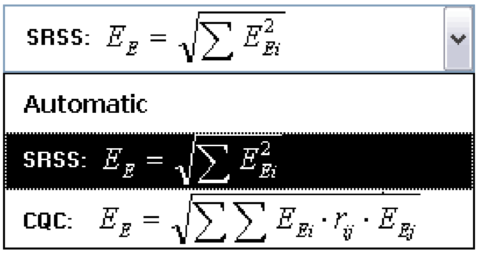
If the sum of the effective mass is much smaller than 90%, eigenfrequency calculation should be done for more shapes in order to reach 90%.

Lots of mode shapes should be ensured to reach the 90% of total mass in vertical direction. It is highly recommended to check the national code, whether it is necessary to examine the vertical effect or it is not.

The mode shapes which have small effective mass may be neglected, because their effect in result is very small, but calculation time increases.

Summation rule by directions

According to the *EC8*, the summation rule in the individual directions can be selected. In all other codes always the *SRSS* rule is used for summation (there is no choice). Read more about *SRSS* and *CQC* summation rules in *Theory book*. If the *Automatic* is selected, the rule selection procedure is as follow:



- Always three directions are investigated (if more than one mode shape is selected in a column), where all mode shape is independent from each other or not.
- If at least one dependent situation exists in a direction, the program automatically uses the *CQC* rule for all mode shape in that direction, otherwise *SRSS* rule is used.

Direction of the horizontal effect

Codes generally speak about seismic calculation in *X-Y* directions. These directions give the maximum effect, if the mass and elastic properties of the structure ensure that the calculated mode shapes lay in *X-Z* or *Y-Z* plane. But it is not always achieved in practice.

To achieve the unfavorable direction, where the results of ground motion are maximum, the program gives the possibility to set *x'-y'* direction for the seismic horizontal effect (*Alpha*). The program suggests the *Alfa* value, if you click on *Auto* button. It finds the most unfavorable direction, where any of the *mx'* and *my'* is zero and the other is maximum in the same row (same shape). But, there is a rule: the direction can be ensured only for one mode shape, so the program selects the row where the effective mass is the maximum. If manually definition is chosen, give an angle for *Alfa* and press the button *Set*.

On the left hand side figure you can see a badly adjusted *x'-y'* direction (*Alpha* = *0*). Appling *Auto* button, the program arranges the direction for the 58.5% effective mass *my'* and correct it to 78%.

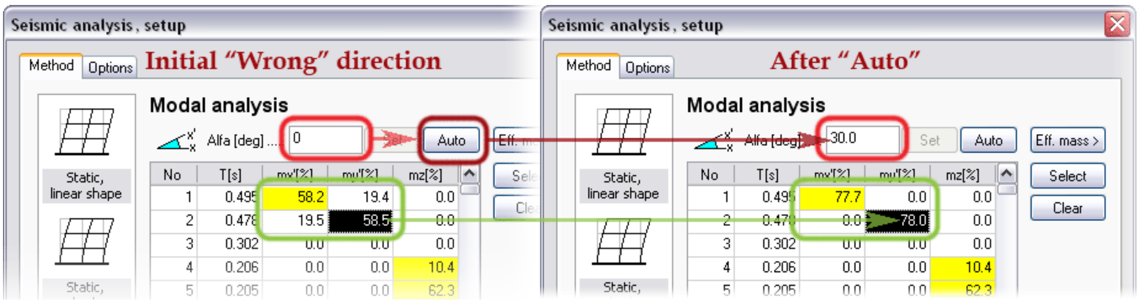
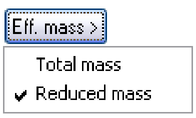


Figure: Settings of Alpha

Effective mass

The modal effective masses can be compared to the total mass or reduced mass at *Eff. mass*:

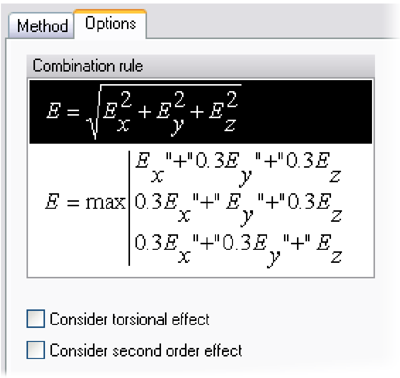


In *FEM-Design* "*Reduced mass*" means the difference between the total mass of the structure and the basement mass. The basement mass is the sum of all masses which lay on the foundation level (set at *Seismic load/ Others*).

EC8 defines the total mass without basement (*Reduced mass*). The effective masses are generally compared to the *Reduced mass*, but this is not valid for the massive basement with elastic foundation. If the above mentioned situation is the case, it might happen that the sum of the effective masses of a column is larger than the 100%.

It is uninterested in the calculation point of view, that effective masses are compared to the total or the reduced mass, because these values are given in percentage and only gives information, that which mode shape is the fundamental or which shapes are dominant in a given direction.

At *Options*, more calculation properties can be set:



Combination rule

The combination rule of effects in the x', y' and maybe z directions can be set here. You can choose from two possibilities.

Consider torsional effect / Consider second order effect

Additional effects can be taken into consideration during seismic calculation. See the detailed description of these effects in *Theory book*.

The calculation of both effects needs the definition of **storeys**.

5. **Seismic calculation**

After choosing a calculation method and setting its properties, activate first *Seismic analysis* under *Analysis* and then press *OK*.

The Results

Besides **displacements**, **reactions**, **connection forces** and **internal forces**, the program calculates the *Equivalent loads* and the "*Base shear force*". Results can be displayed by vibration shape (selected at calculation settings), from torsional effect, from sums by direction and from the total sum (*Seismic max*). If equivalent loads are displayed, also the "base shear force" appears on screen (in grey color). Torsional moment effect on the whole structure can also be displayed, if torsional effect was taken into consideration during calculation.

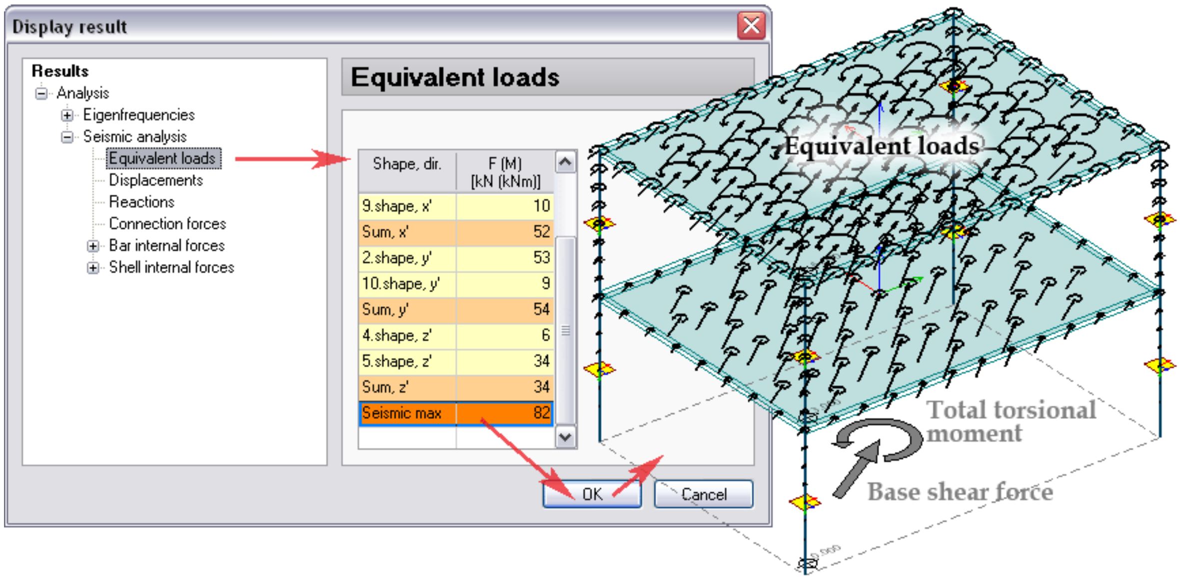


Figure: Results of Seismic analysis

⚠ Because of the square combination rule, the results summed by direction (*Sum, x'*, *Sum, y'*, *Sum, z*) and the total sum (*Seismic max*) give only positive values, so absolute maximums. Also, because of combination rule, the displacement components and the internal forces in one point are not simultaneous results.

Summary of Static and Seismic Effects

Seismic effect can be combined with static loads in two ways:

- By defining new **load cases** contain equivalent seismic loads to take them into consideration at analysis or design calculations as real static loads,

- By adding the maximum seismic effect to load combinations or load groups.

Seismic loads as load cases

The x' and y' directional loads (also torsional moments) equivalent to the horizontal ground motion can be converted to load cases. **"Seismic,..."-type load cases** behave as static loads: they can be combined, they can be added to groups, and they can be taken into consideration at stability, imperfection and design calculations. As you see in the list of load case types, the seismic effects can be considered with positive and/or negative sign.

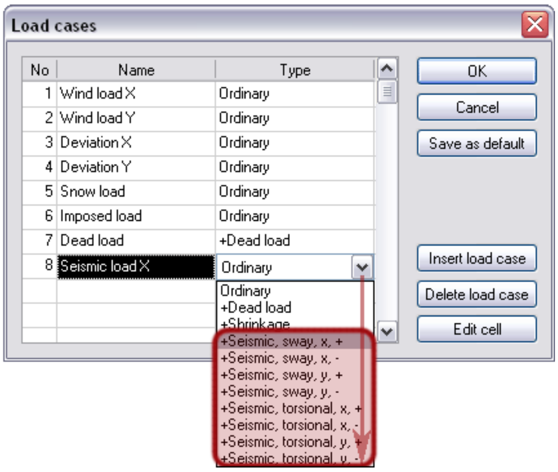


Figure: Seismic effect added as load case

Maximum seismic effect in load combinations

The total, the maximum seismic effect (see *Seismic max* at *Equivalent loads*) can be added to load combinations. Start the command *Load combinations* (*Loads* menu). Apply *Insert case(s)* on a predefined or new load combination, choose "*(Seismic max)*", define a load factor and press *OK*.

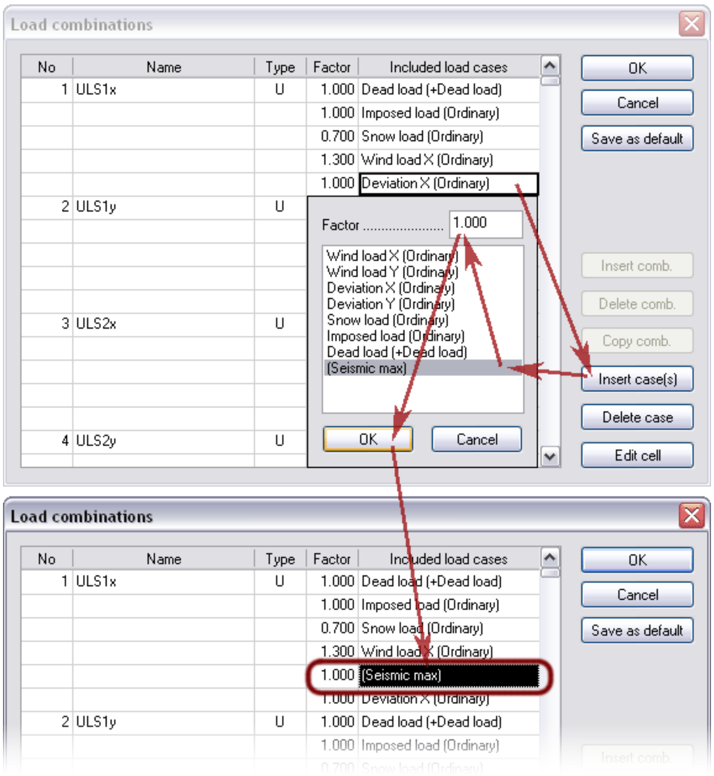


Figure: Maximum seismic effect added to load combination

Maximum seismic effect as load group

The maximum seismic effect (*Seismic max*) can also be added to groups in all codes. Define a group as "*Seismic*". The program automatically takes the "*(Seismic max)*" into consideration with +/- values in the generation of the most unfavorable results.

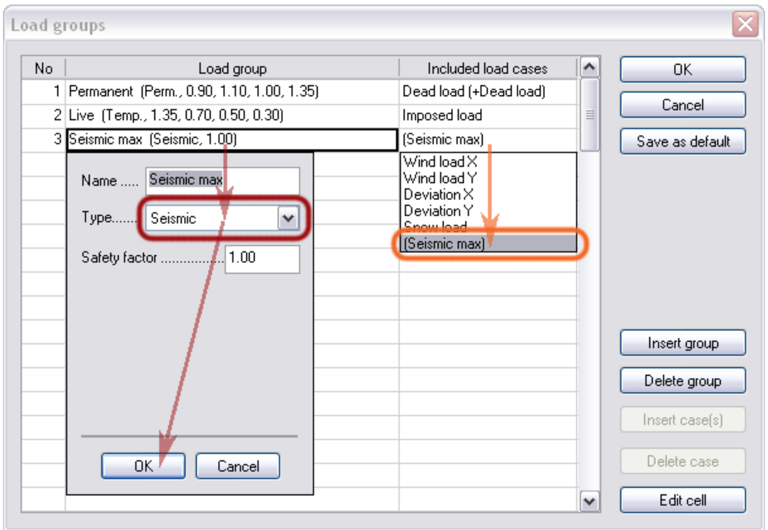
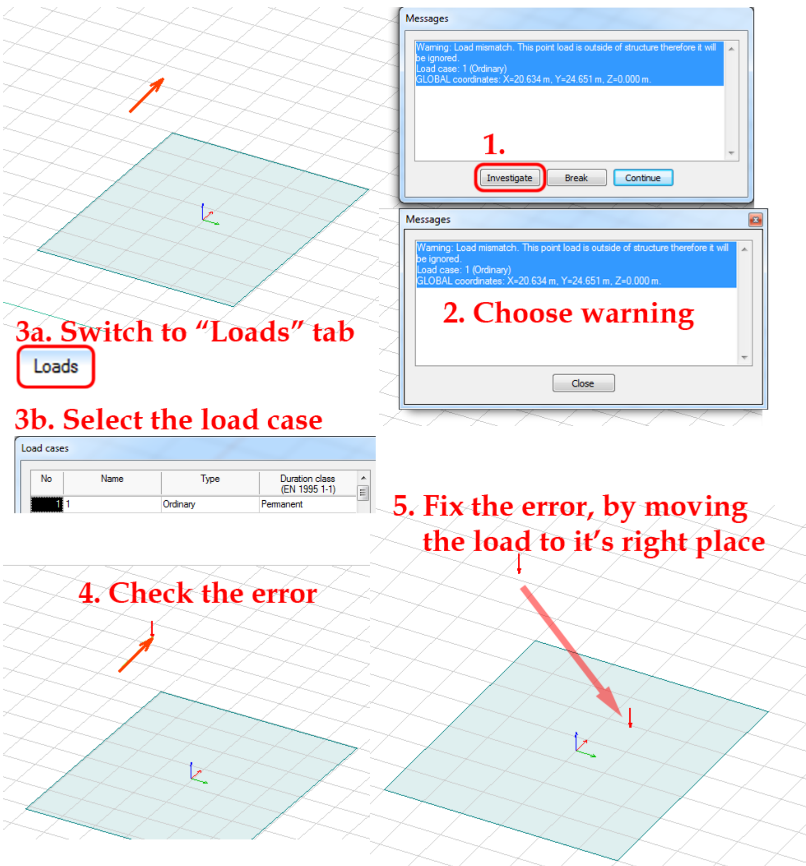


Figure: Maximum seismic effect defined as load group

Investigate

If a warning message appears during calculation (e.g. Load mismatch or Finite element mesh problem) there is a possibility to check and fix the error by navigating in the model with *Investigate*.

The following pictures show a badly defined load and how can the user check and fix the error with *Investigate* function.



Design

Depending on the current FEM-Design module you can do different design calculations for concrete, steel and timber model elements. This chapter summarizes the design possibilities and results by design type.

Design Load

All design calculation works with load combinations and load groups.

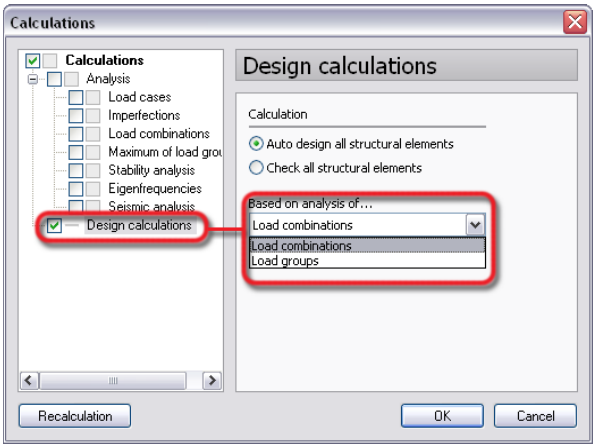


Figure: Design calculation settings

⚠ Although analysis can be run for load combinations and load groups in the same time, design calculation results exist only either for the load combinations or the maximum of load groups.

The *Shell design* can be done only for load combination, because it based on stability results.

Serviceability load combination is considered only in RC bar and shell design, other design is done just for ultimate load combinations.

Buckling Length Factors

Buckling length is claimed input data of all bar element design (RC/steel/timber columns, beams). The buckling length is determined from the bar length and a factor depends on bar end connections. For typical end conditions (hinged, fixed, cantilever etc.), proposed *beta* factors are available for flexural buckling.

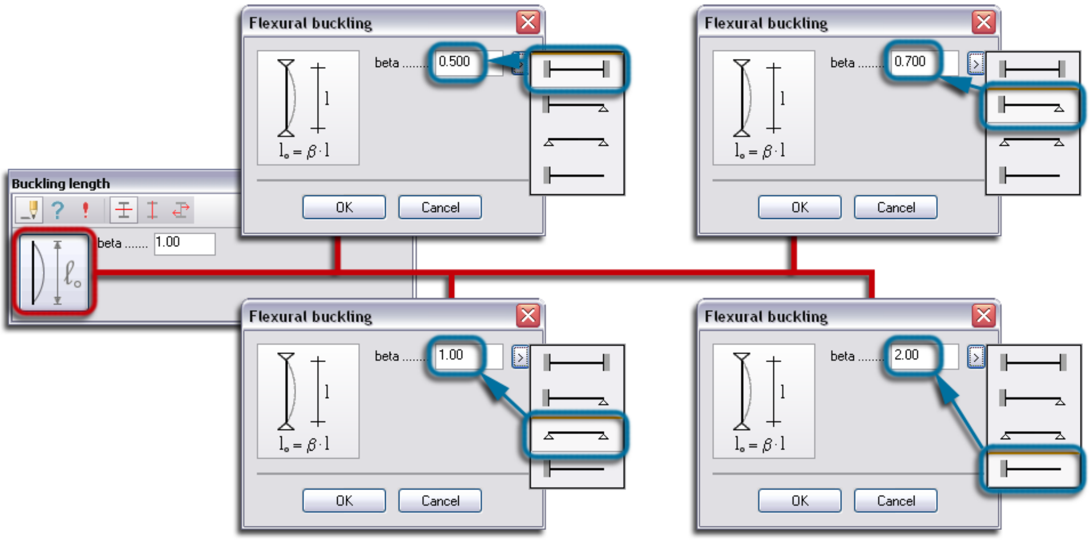


Figure: Proposed beta factors for flexural buckling

⚠ Buckling length of a *Truss member* element is equal to its length, so buckling length of trusses cannot be modified.

The default *beta* factor is 1.0 for both flexural and torsional buckling.

The defined buckling length fundamentally influences the design results.

Design Groups

Elements having certain common properties can be assigned to a *design group*, and the members in one group will be designed in the same way. For example, uniform reinforcement (longitudinal bars and stirrups) will be calculated for RC bars assigned to one design group, or the same profile will be designed for each steel bar of a design group, etc.

General terms to be a design group member:

- Same design element type (beam, column, truss member, plate, wall or timber panel)
- Same material
- Same geometry (sizes, profiles etc.)
- Same design parameters (e.g. base reinforcement)

⚠ Depending on specific cases/element types, additional conditions have to be materialized:

- Same support conditions
- Same end/edge connections
- Same load, etc.

In design mode, click  to group elements under a name and color.

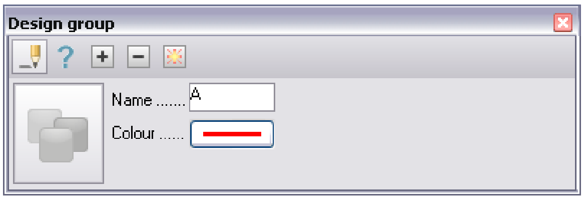


Figure: Design group tools

Design group tool provides the following group operations:

- creates a new group,
- edits the properties (name and color) of a selected group,
- adds selected elements to the current group,
- removes selected element from the group,
- is available only in *Surface* and *Punching reinforcement* design. A group member has to be set as the "*Master*" that carries visually the representative applied reinforcement designed uniformly for the group. Only the "Master's" applied reinforcement is editable; symbolic reinforcement is displayed for the other group members.
- explodes/deletes a selected group.

The next figures give examples of grouping same type steel bars, concrete slabs and beams.

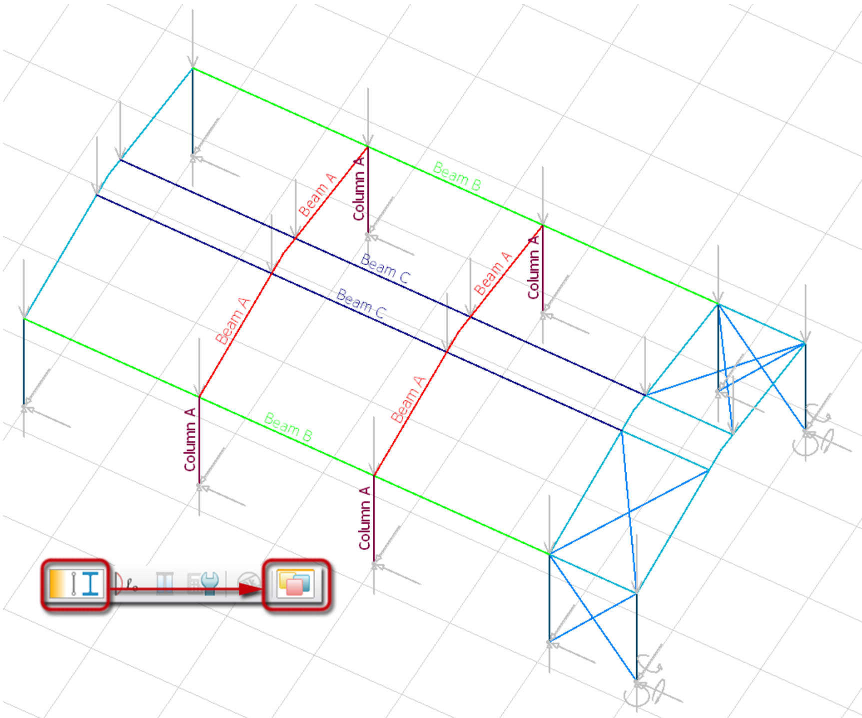


Figure: Some design groups of a steel frame

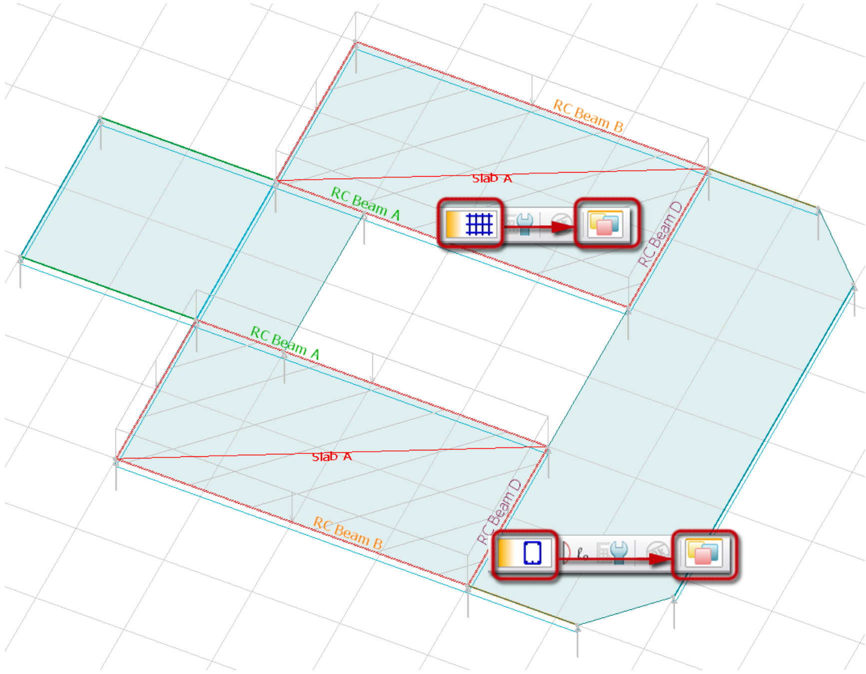


Figure: Design groups of concrete slabs and beams

From Auto Design till Final Design

FEM-Design covers the whole RC, steel and timber design process by starting with automatic predesign (so-called "*Auto design*") for the entire structure and/or by elements/*design groups*, then by continuing with detailed *manual design* of the domain structural components and by ending with the final design (so-called "Check") of the entire structure.

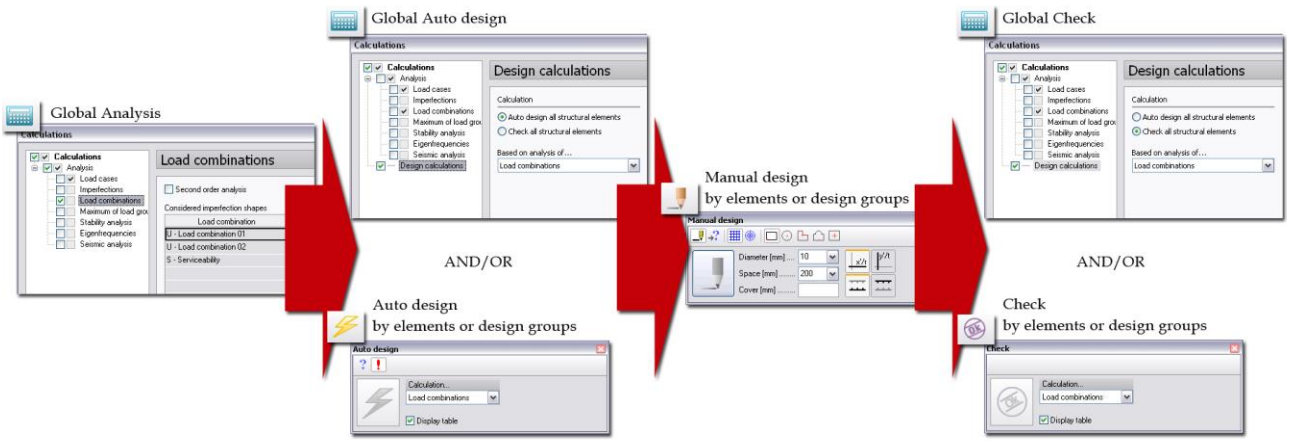
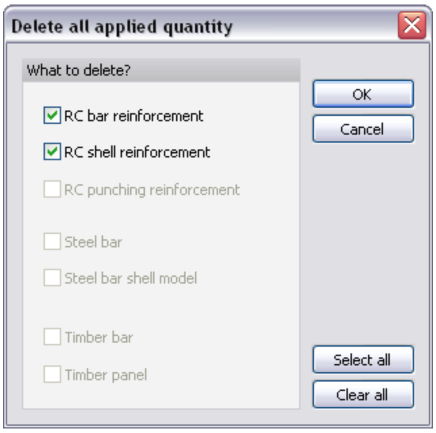


Figure: From predesign of components till final global design

Although analysis can be done in one step for the entire structure independently of the element types (concrete, steel and timber), design calculations (and so the design modules) can be run separately by the element types (e.g. one after the other).

Apply design changes and recalculate: one-click validation of design changes in the current entire model together with analysis and design recalculation according to the new states. But, leaving Design mode, the design changes are always updated in the current model.

Delete all applied quantity: restores the model state previous to the applied design according to following settings dialog:



Auto Design

Based on internal forces, utilization check and initial design settings, FEM-Design finds quickly and automatically the most suitable applied quantity depending on the design category and element type. The so-called "*Auto design*" makes a *proposal* for reinforcement distribution, for the applied profile, for the panel type, etc.

Auto design can be done for the whole project with the *Calculate* > *Design calculations* > *Auto design all structural elements*. The fast algorithm uses the default calculation and design parameters of the elements and search for the most suitable applied quantity according to the utilization. *Auto design* automatically runs *Analysis* calculations (for *Load combinations* or *Load groups*), for *Stability* etc. depending on the design type: e.g. *Steel bar design* or *Shell design*.

Auto design can be applied for single elements and/or design groups only (without running design for the complete model).

The next table summarizes the initial design parameters and the results of *Auto design* by design categories and element types.

Category	**Element type**	**Design parameter**	**Auto design**
RC design	*Bar reinforcement*	Steel quality, bar diameter and profile, concrete cover	Bar and stirrup distribution (numbers of bars, spacing)
	Surface reinforcement	Steel quality, bar spacing, shape settings of the reinforcement regions, possible bar diameters	Bar distribution, applied reinforcement regions and area, and bar diameter
	Punching reinforcement	Steel quality, range of diameters, distribution shape (bended bar, circular stirrup, open stirrup)	Bar and stirrup sizes and distribution
Steel design	*Steel bar*	Range of cross-sections	Suitable cross-section

	Shell model	Range of plate thicknesses	Suitable plate thickness
Timber design	*Timber bar*	Range of cross-sections	Suitable cross-section
	Timber panel	Range of panel types	Suitable panel type

Table: Input parameters and the results of element-based Auto design

Auto design gives summary tables, which display both the initial design parameters and the recommended (applied) quantities optimized to maximum utilization. All utilization details done by the Eurocode 2, 3 or 5 regulations can be also displayed.

Fast redesign can be started from the table by modifying the design parameters.

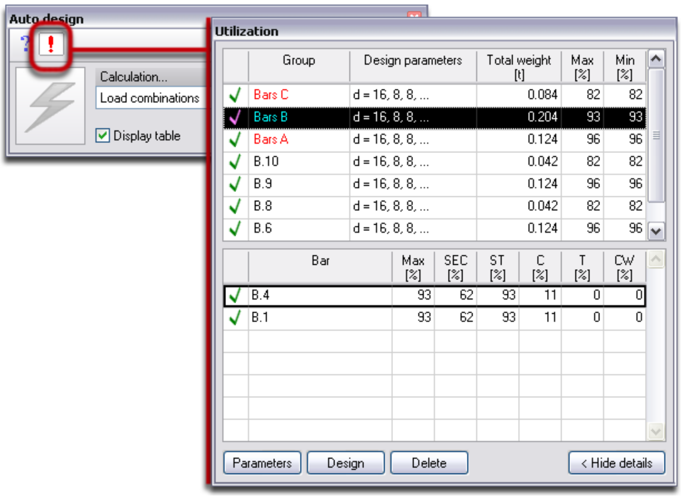


Figure: Summary table of Auto design (Bar reinforcement)

If the program is not able to find suitable parameters from the given/available design parameter range, it sends a warning message and marks the problematic elements/groups.

Leaving *Design* mode applies all design changes by updating the current model in order to be considered in a next recalculation. It also invalidates previous analysis results.

In RC modules, *Auto design* results applied reinforcement that is considered in case of cracked-section analysis.

Manual Design

Following *Auto design*, "on-the-fly" fine tuning of applied quantities can be done by elements and design groups. In RC design, *Manual design* completes the applied reinforcement editing task for both concrete surface and bar elements.

The manual design can be done without calculated analysis results in the following categories:

- RC design
- Steel design
- Timber design

This function is useful especially at the analysis of an existing building.

Category	Element type	Possible initial data	Manual design
RC design	*Bar reinforcement*	Reinforcement come from Auto design	Applied reinforcement (longitudinal bars and stirrups)
	Surface reinforcement	Reinforcement come from Auto design	Applied reinforcement (top/bottom/middle RC)
	Punching reinforcement	Reinforcement come from Auto design	Applied reinforcement (bended bar, circular stirrup, open stirrup)
Steel design	*Steel bar*		Applied cross-section
	Shell model		Applied plate thickness
Timber design	*Timber bar*		Applied cross-section
	Timber panel		Applied panel type

Table: Input parameters and the results of Manual design

The next figure gives an example for the reinforcement design of a concrete beam from starting with ***Auto design*** and finishing with detailing (*Manual design*).

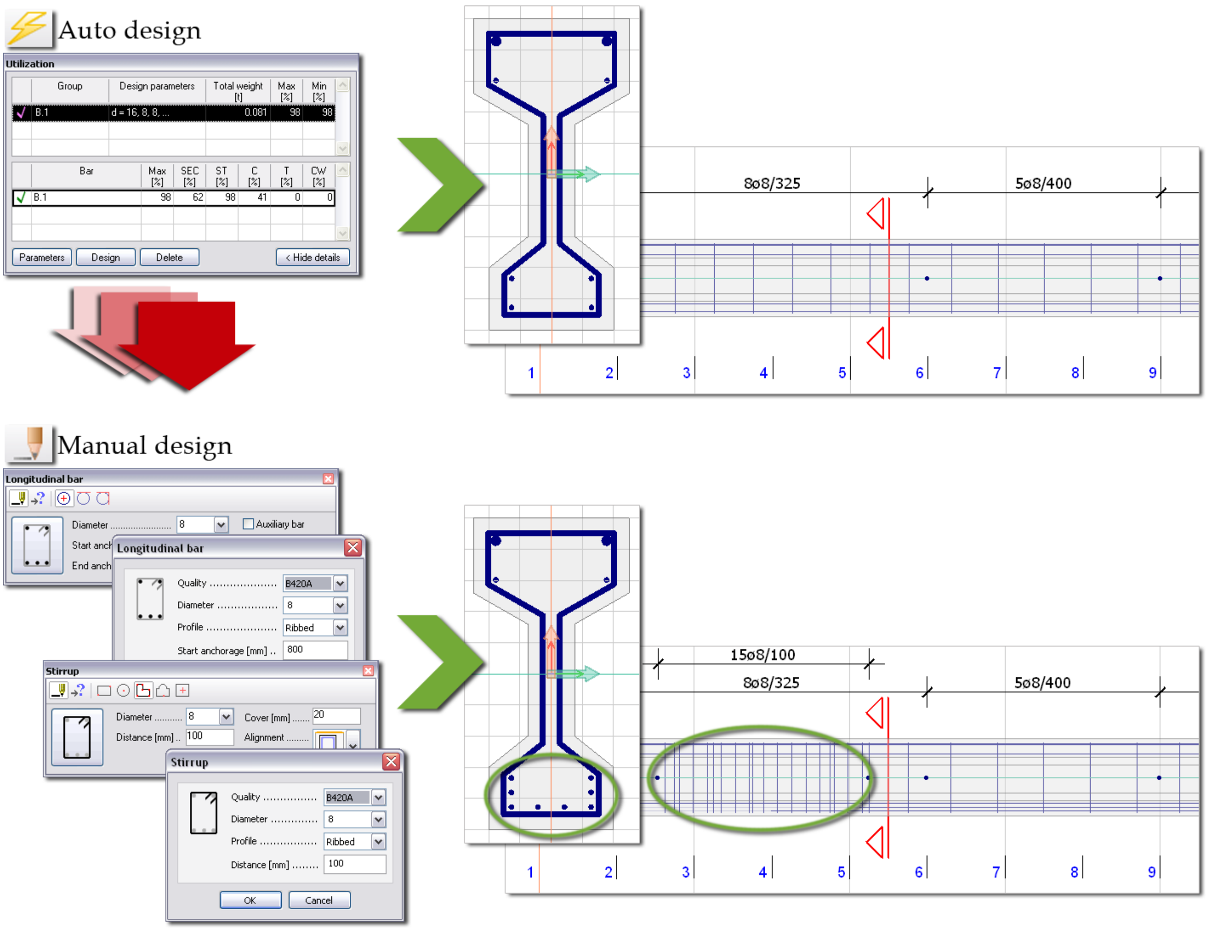


Figure: Combined RC design

Leaving *Design* mode applies all design changes by updating the current model in order to be considered in a next recalculation. It also invalidates previous analysis results.

In RC modules, *Manual design* results applied reinforcement that is considered in case of cracked-section analysis.

Detailed Result

Utilization results with detailed background calculation formulas (together with Eurocode references), figures and tables can be displayed by single elements or by design groups. Quick navigation is powered with zooming details.

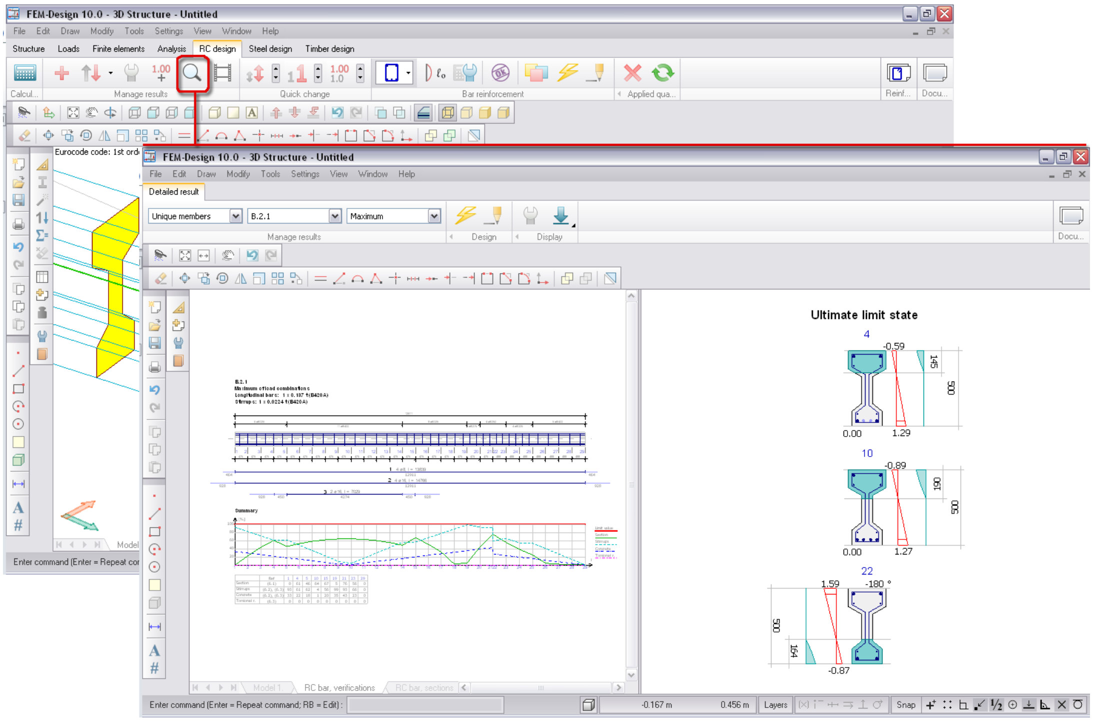


Figure: Detailed result

Of course, *Detailed result* is also available for the analysis (displacement, internal forces, and stresses) of bar elements.

RC Design

Following the analysis calculation, the applied reinforcement (longitudinal bars, stirrups etc.) can be designed automatically and refined manually for concrete beams, columns, truss members, slabs, walls and shell elements. The applied reinforcement can be considered in cracked-section analysis.

The table summarizes the available RC design features and its analysis-related effect (cracked-section analysis) by FEM-Design module.

Design element type	Design feature				
Bar reinforcement	**Auto design**	✓		✓	✓
	Manual design	✓		✓	✓
	Cracked-section analysis	✓		✓	✓
Surface reinforcement	**Auto design**	✓	✓		✓
	Manual design	✓	✓		✓
	Cracked-section analysis	✓	✓		✓
Punching reinforcement	**Auto design**	✓			✓
	Manual design	✓			✓
	Cracked-section analysis	✓			✓

Table: RC design features by FEM-Design module

Global *Auto (RC) design* (*Calculate* > *Design calculations* > *Auto design all structural elements*) gives applied reinforcement for all concrete elements based on their initial design calculation parameters, *Auto design* > *Parameters* and optimized to their internal forces and detailed utilization calculations. With *Manual design* you can fine-tune the reinforcement by elements and/or design groups. You can do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global RC design	Element-based RC design	Combined RC design
1	*Design calculation parameters*	Global *Analysis*	*Design calculation parameters*
2	**Design group**	*Design calculation parameters*	**Design group**
3	? *Auto design > Parameters*	**Design group**	? *Auto design > Parameters*
4	Global *Auto design*	? *Auto design > Parameters*	Global *Auto design*
5	*Manual design* by elements	! *Auto design* by elements	? *Auto design > Parameters*
6	*Apply design changes*	*Manual design* by elements	! *Auto design* by elements
7	Global *Check*	*Apply design changes*	*Manual design* by elements
8	Cracked-section analysis	Global *Check*	*Apply design changes*
9	*Documentation*	Cracked-section analysis	Global *Check*
10		*Documentation*	Cracked-section analysis
11			*Documentation*

Table: Recommended steps by design alternatives

Bar Reinforcement

Bar reinforcement design needs internal forces from **Analysis** calculations applied for *Load combinations* or *Load-groups*, **Buckling length** and initial design settings defined by *Design calculation parameters* and 2nd order calculation method by *Configuration.*

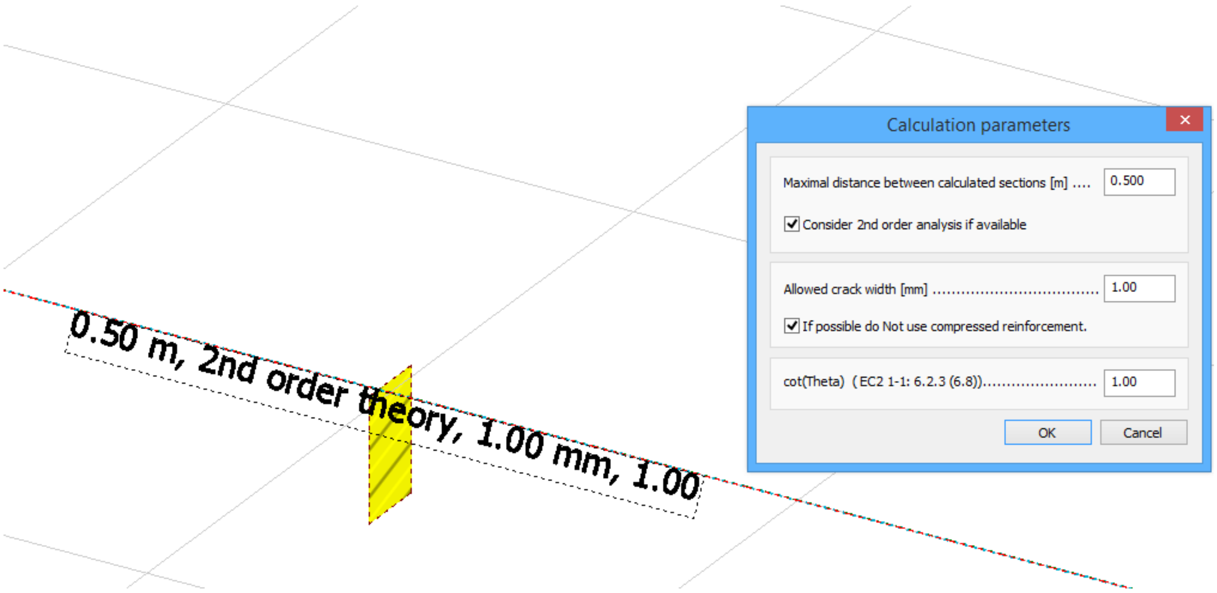


Figure: Design calculation parameters

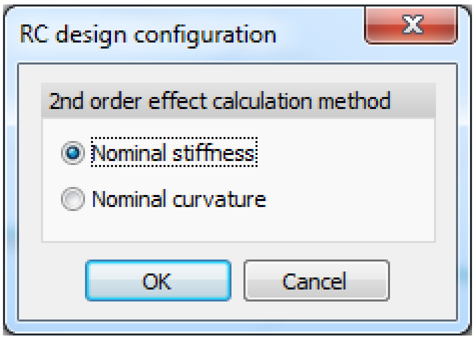


Figure: Design calculation parameters

The maximal section distance defines the approximate position of the design sections and so the available sections for detailed results.

Auto Design

Global *Auto design* gives utilization results and suitable reinforcement distribution for all concrete bars of the current project.

The applied design parameters can be displayed on screen by showing the "*RC bar, design parameters*" object layer, or click *Design* tool of the *Auto design* to show the parameters together with a detailed utilization table. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *RC bar* > *Utilization*, and clicking the *Numeric value* tool summarizes the list of maximum utilizations by elements in a dialog.

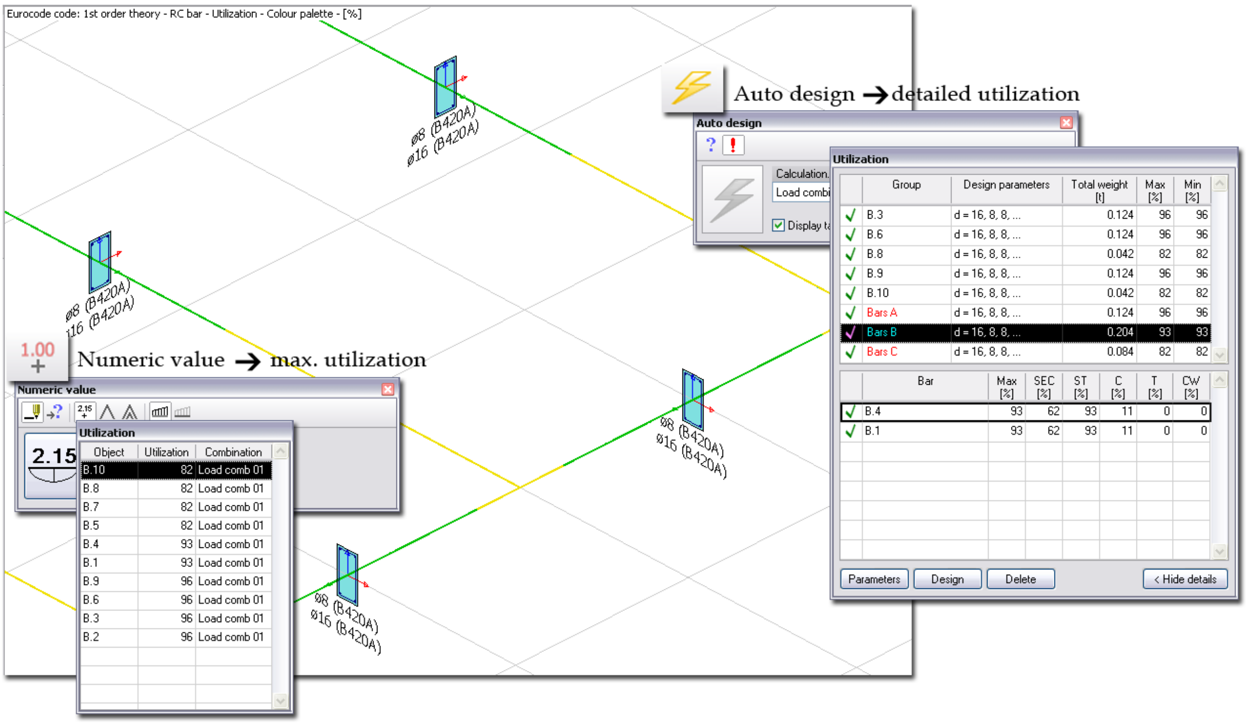


Figure: Global Auto design and utilization results

Element-based *Auto design* finds the most suitable position and distribution of longitudinal bars and stirrups for selected unique or grouped members only according to their user-defined design parameters. Initial reinforcement (steel quality, diameter, profile, concrete cover) and design (aggregate, vibration) settings can be set for the concrete members/groups with the *Parameters* tool of *Auto design*.

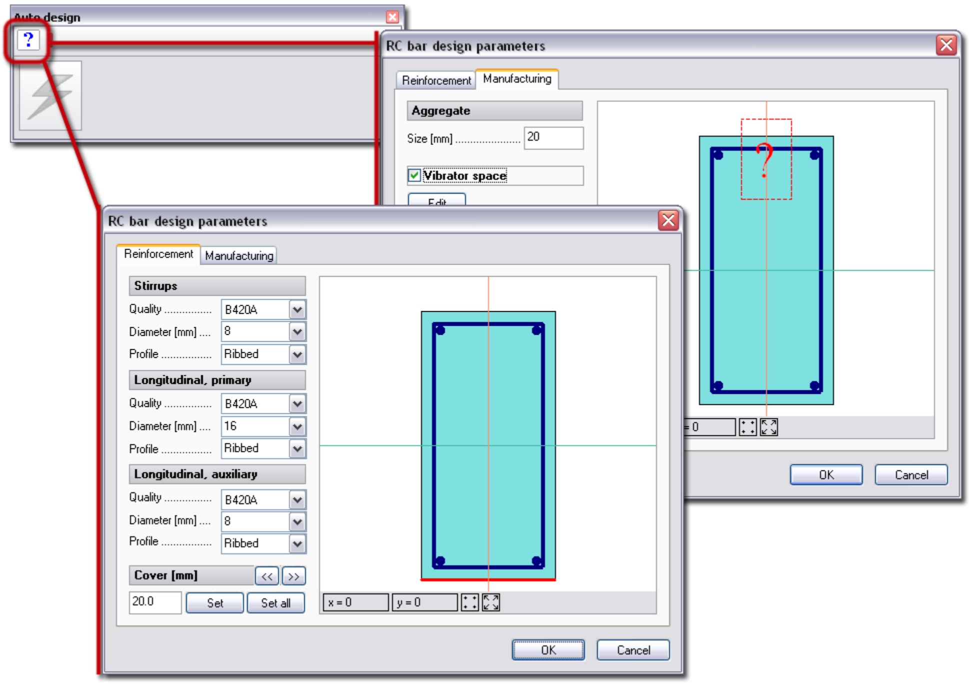


Figure: Design parameters

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results applied reinforcement and their checked utilization. Check the *Display table* box to have a look at the overall design results.

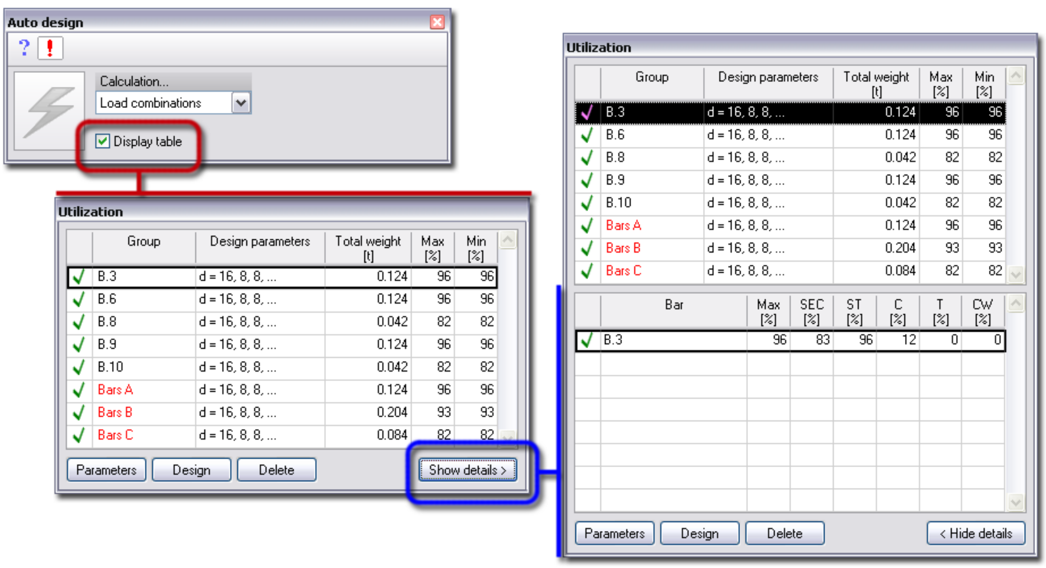


Figure: Quick summary of Auto design results

The upper table shows the design efficiency and the maximal utilization of the designed single members and groups based on the given design parameters. The bottom table ("*Show details*") displays the utilization details of the bar or the members of the group selected in the upper table.

	Meaning	Note
	Suitable reinforcement is available	
	Suitable reinforcement is not available	Modify bar profile, material or RC design parameters
Group	ID of a single bar or a group name	
Design parameters	Applied design parameters	
Total weight	Total weight of applied reinforcement	
Max	Max. utilization of a single bar or the significant member of a group	
Min	Max. utilization of the less significant group member	
Bar	ID of a single bar or a group member	
SEC	Section utilization	According to Eurocode 2: 6.1
ST	Stirrup utilization	According to Eurocode 2: 6.2 and 6.3
C	Concrete utilization	According to Eurocode 2: 6.2 and 6.3
T	Utilization of torsional reinforcement	According to Eurocode 2: 6.3
CW	Utilization for crack width	According to Eurocode 2: 7.3

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

1. Select a bar or a design group in the upper table.
2. Modify the design parameters of the select element under *Parameters*.
3. Click *Design*.

Applied reinforcement generated by *Auto design* can be displayed with:

- **Detailed result** applied for the displayed utilization result (*New result* > *RC bar* > *Utilization*) of a single bar or a group member.
- *Manual design* applied for a single bar or a group member to edit the design reinforcement and/or add further longitudinal bars and stirrups.

Manual Design

Manual design opens a new window in the current project and gives tools to define new (applied) reinforcement in concrete beams, columns and bars, or to modify/edit the reinforcement generated by *Auto design*. The drawing area is divided into two view windows:

- **Cross-section**
  It shows the cross-section of the current concrete bar. The definition of new longitudinal bars (sectional position) and stirrups (shape) starts in this window. The position of the cross-section (section view) can be set by moving the section marker in *3D view*.

- **3D view**
  It shows the side view of the concrete bar by default. But, any 2D and 3D view can be set with the *View* menu commands (e.g. general 3D view with *View* > *Space view*). The start and end pont (and so the length) of longitudinal bars and the position (the distribution) of the stirrups can be defined here.

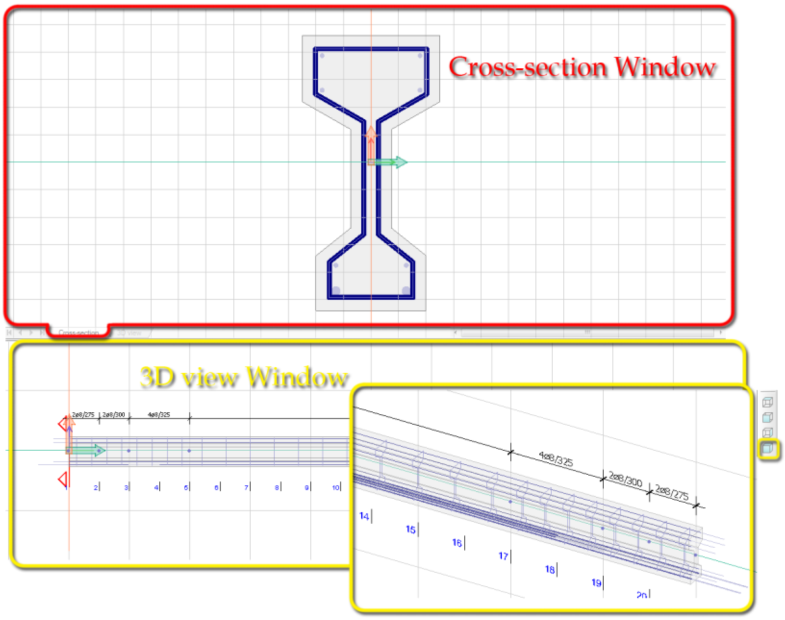


Figure: Working windows of Manual design

You can choose the required window by clicking inside it or its title.

The *Longitudinal bar* tool defines new bars in given insertion points. Set the main properties of the new bar on the tool palette or all properties under *Default settings*.

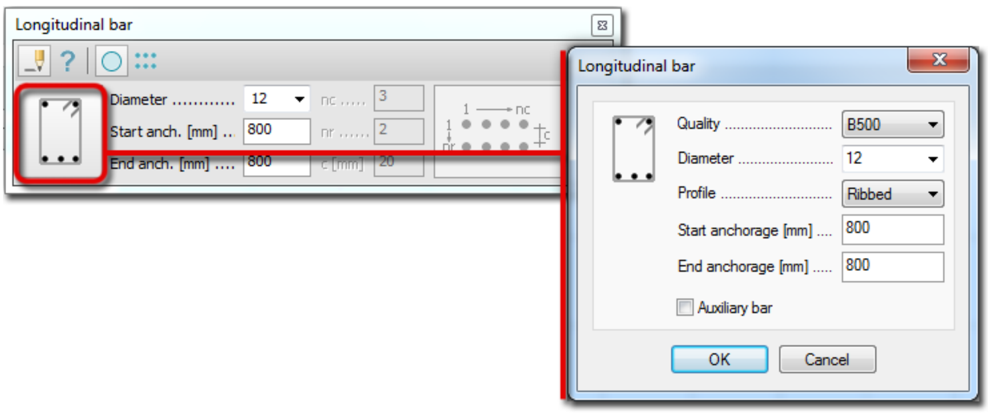


Figure: Definition tools and settings of Longitudinal bar

Use one of the following tools to place the new bar in the *Cross-section* view:

Clicking in *Cross-section*, the new bar will be placed with its center point.

Align the new bar to a line/edge by select one in *Cross-section*. Move the mouse to set the bar's relative position to the selected line/edge: the center point/upper/bottom/left/right surface will be on the line/edge.

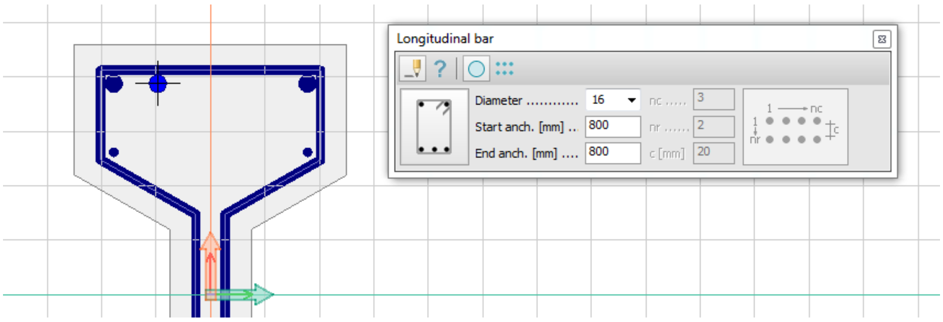


Figure: Placement of a bar aligned to an inner edge of a stirrup

Align the new bar to a corner defined by two lines/edges. The bar will be tangent for the first and then the second given line.

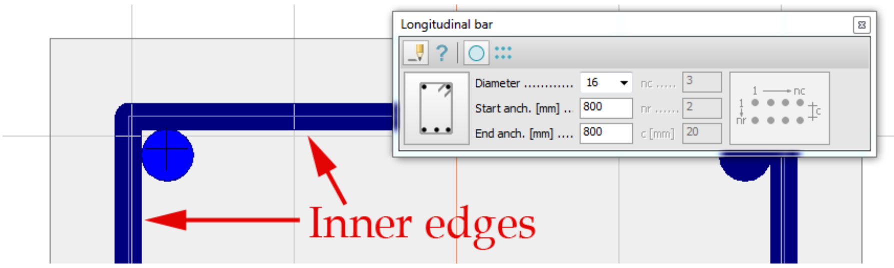


Figure: Placement of a bar aligned to a corner of a stirrup

1. Define the group of longitudinal bars by set the number of the horizontal (nc), vertical (nr) bar numbers and the distance between the rows (c)

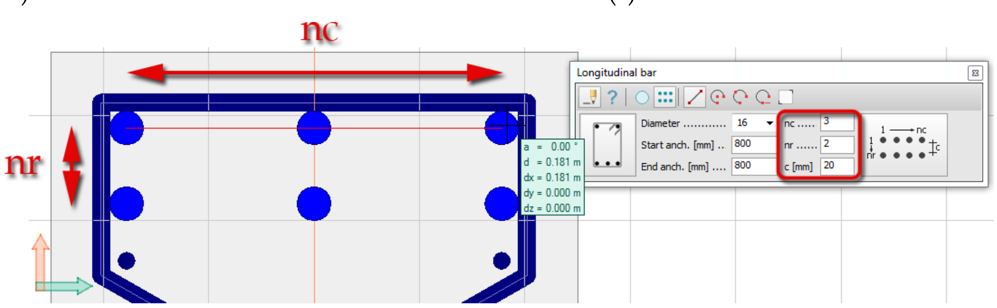


Figure: Placement of bar group

The steel bar length can be defined manually by giving the bar's start and end point in *3D view*.

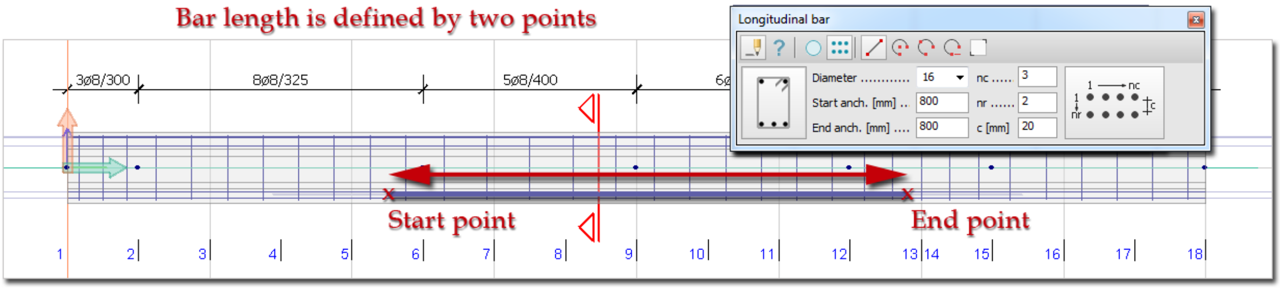


Figure: Steel bar length definition


The *Stirrup* tool defines new stirrups with given shapes. Set the main properties of the new stirrup bars on the tool palette or all properties under *Default settings*.

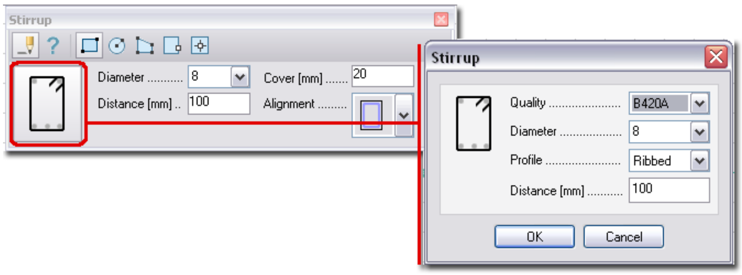


Figure: Definition tools and settings of Stirrup

Choose the contour geometry of the new bar and define the bar's relative position to the contour with *Alignment*, which also defines the final stirrup shape with the *Cover* value in the *Cross-section* view. In the final step, the distribution of the stirrups based on the *Distance* value is defined with a start and an end point in the *3D view*.

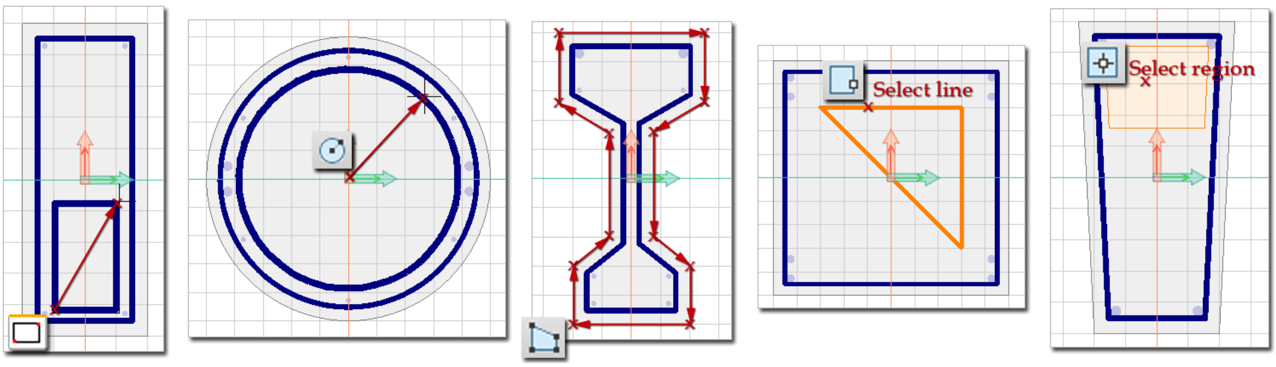


Figure: Stirrup geometries

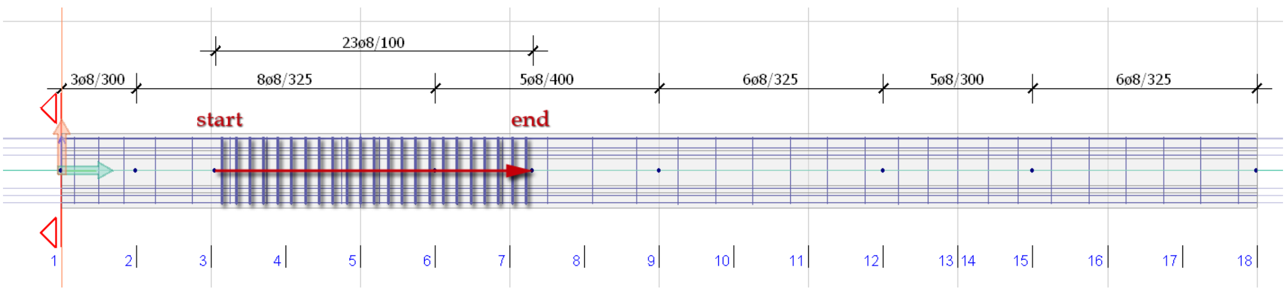


Figure: Placement of stirrups

The contour defines the stirrup shape, if the *Cover* value is set to 0.

You can modify the properties (quality, diameter, profile etc.) of previously defined bar/stirrup reinforcement(s) with the *Properties* tool of *Longitudinal bar* and *Stirrup*.

To exit from *Manual design* with validating the new and modified reinforcement bars and stirrups, click *OK*.

To exit from *Manual design* without accepting the defined and modified reinforcement, click *Cancel*.

Detailed Result
Utilization of RC bars can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all concrete bars calculated from the suggested applied reinforcement.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected elements.
- After global *Check* done for all bar elements having final applied reinforcement.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	*Calculate>Design calculation>Auto design all structural elements*	*Auto design*	*Auto design* and/or *Manual design*	*Auto design* and/or *Manual design*
2	*New result>RC bar*	*New result>RC bar*	*Check*	Apply changes
3			*New result>RC bar*	*Calculate>Design calculation>Auto design all structural elements*
4				*New result>RC bar*

Table: Steps of displaying RC bar utilization by different design cases

Utilization displayed with *New result* appears for all designed bars. The utilization components for a bar/design group can be displayed with *Detailed result*.

Detailed result opens a new windows in the current project after selecting a bar/group member, which display:

- **Applied reinforcement**
  The figure gives the distribution of the applied reinforcement.

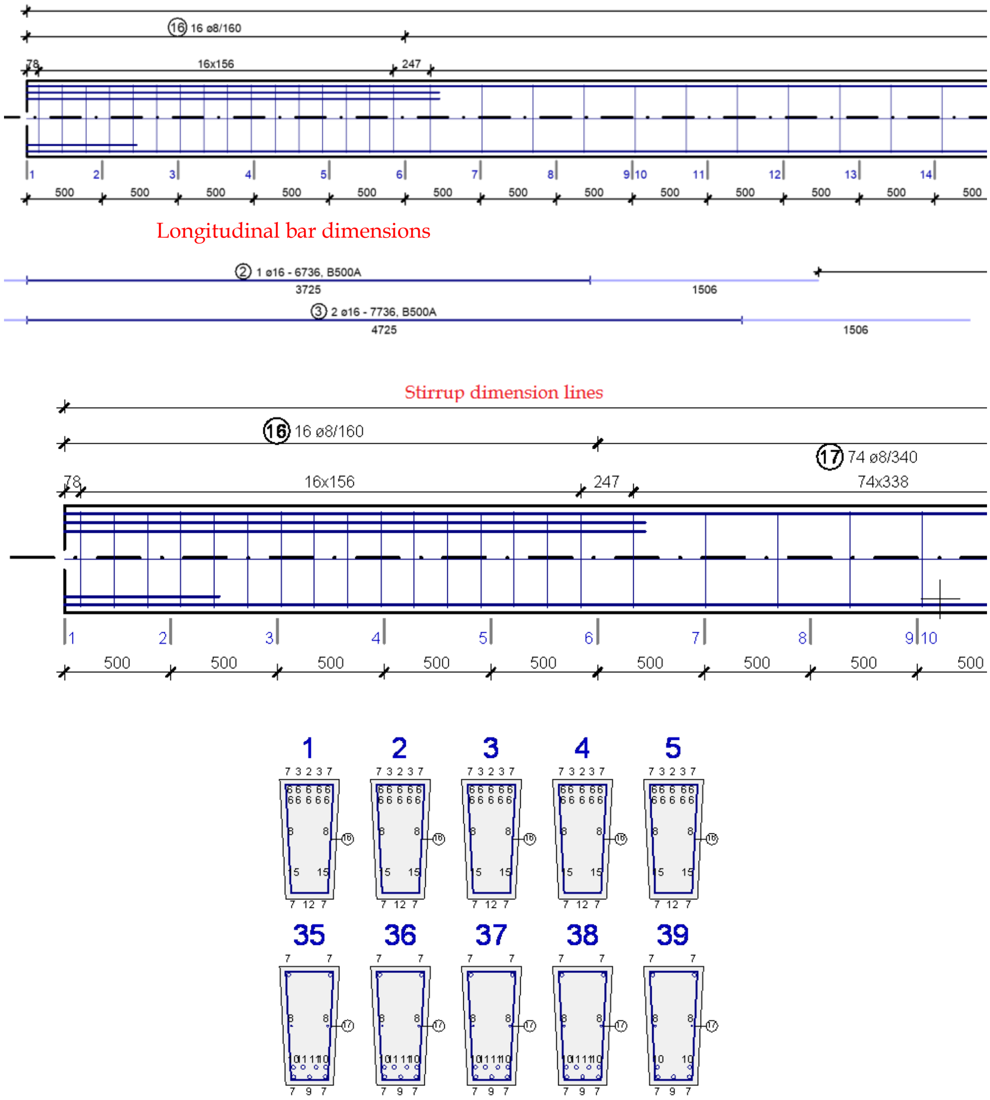


Figure: Longitudinal bar and stirrups data

- **Cross-section data**

The figure gives the cross-section datas: height, width, area, moment of inertias.

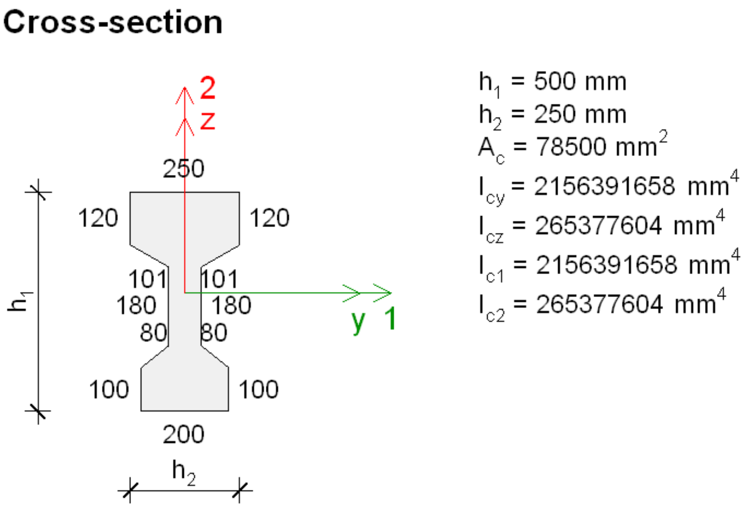


Figure: Cross-section datas

- **Material properties**

  In Materials section, the program shows the material properties of the applied concrete's and reinforcement, e.g. compression strength (f_{ck}), the mean tensile strength (f_{ctm}), mean Young modulus (E_{cm}).

Materials

C25/30				B420A			
f_{ck}	=	25.00	N/mm^2	$f_{yd} = f_{ywd}$	=	365.22	N/mm^2
f_{ctm}	=	2.60	N/mm^2	E_s	=	200000.00	N/mm^2
$f_{ctk,0.05}$	=	1.80	N/mm^2	$\varepsilon_{yd} = f_{yd}/E_s$	=	0.00183	N/mm^2
E_{cm}	=	31000.00	N/mm^2	ε_{ud}	=	0.02250	
α_{cc}	=	1.00	N/mm^2				
α_{ct}	=	1.00	N/mm^2				
γ_c	=	1.50					
γ_s	=	1.15					
φ_{ef}	=	0.00					
$f_{cd} = \alpha_{cc} f_{ck}/\gamma_c$	=	16.67	N/mm^2				
$f_{ctd} = \alpha_{ct} f_{ctk}/\gamma_c$	=	1.20	N/mm^2				
$E_{cd} = E_{cm}/\gamma_c$	=	20666.67	N/mm^2				
ε_{c2}	=	0.00200	(Table 3.1)				
ε_{cu2}	=	0.00350	(Table 3.1)				

Figure: Material properties

- **Calculation formulas**

  This contains the EC-3 formulas for checking RC bar elements, the substitutions in the formulas and the calculation table for each cross section. In the table, the amount of cross-section calculations can be set in the Display options.

Concrete utilization for shear and torsion (Part 1.1: 6.2, 6.3)

$\Theta = 45°$

$\sigma_{cp} = \min\left(\frac{N_{Ed}}{A_c}, 0.2 f_{cd}\right)$

$\alpha_{cw} = 1.0 + \frac{\sigma_{cp}}{f_{cd}}$, if $\sigma_{cp} < 0.25 f_{cd}$ (6.11.aN)

$\alpha_{cw} = 1.25$, if $0.25 \le \sigma_{cp} < 0.5 f_{cd}$ (6.11.bN)

$\alpha_{cw} = 2.5\left(1.0 + \frac{\sigma_{cp}}{f_{cd}}\right)$, if $\sigma_{cp} \ge 0.5 f_{cd}$ (6.11.cN)

$\nu_1 = 0.6$, if $f_{ck} < 60$ MPa

$\nu_1 = \max(0.9 - f_{ck}/200, 0.5)$, if $f_{ck} \ge 60$ MPa

$V_{Rd,max} = \frac{\alpha_{cw} b_w (0.9d) \nu_1 f_{cd}}{\cot(\Theta) + \tan(\Theta)}$ (6.9)

$\nu = 0.6\left(1 - \frac{f_{ck}}{250}\right)$ (6.6.N)

$T_{Rd,max} = 2\nu\alpha_{cw} f_{cd} A_k t_{ef} \sin(\Theta)\cos(\Theta)$ (6.30)

Utilization: $\max\left(\frac{T_{Ed}}{T_{Rd,max}} + \frac{V_{Ed,y}}{V_{Rd,max,y}}, \frac{T_{Ed}}{T_{Rd,max}} + \frac{V_{Ed,z}}{V_{Rd,max,z}}\right)$ (6.29)

Figure: Formulae

- **Detailed calculation tables**

  Calculation details and final values are collected in tables sorted by checking types and under utilization graphs by default. Column number depends on the number of calculation sections or the table settings defined with ***Display options***.

Sections	5	19	23
LC	a	a	a
N_{Ed} [kN]	0.00	0.00	0.00
$V_{Ed,y}$[kN]	0.00	0.00	0.00
$V_{Ed,z}$ [kN]	23.58	23.58	37.06
T_{Ed} [kNm]	0.00	0.00	0.00
σ_{cp} [N/mm^2]	0.00	0.00	0.00
A_{sl} [mm^2]	402	402	1206
d_y [mm]	84	84	84
k_y [-]	2.00	2.00	2.00
$b_{w,y}$ [mm]	350	350	350
$\rho_{1,y}$ [-]	0.01368	0.01368	0.02000
$v_{min,y}$[N/mm^2]	0.49	0.49	0.49
$V_{Rd,c,y}$[kN]	22.90	22.90	25.99
$(A_{sw,y}/s)f_{ywd}$ [kN/mm]	91.79	91.79	163.18
z_y [mm]	76	76	76
$V_{Rd,s,y}$ [kN]	22.90	22.90	25.99
$(V_{Ed,y}/V_{Rd,s,y})$ [-]	0.00	0.00	0.00
d_z [mm]	314	314	314
k_z [-]	1.80	1.80	1.80
$b_{w,z}$ [mm]	120	120	120
$\rho_{1,z}$ [-]	0.01067	0.01067	0.02000
$v_{min,z}$[N/mm^2]	0.42	0.42	0.42

Figure: Detailed calculation table

- **Stress-strain graphs**
  Different colors display the strain (red) and the concrete stress (cyan) curves grouped by *Section utilization* (ultimate limit states) and *Crack width*(serviceability limit states). Also the compressed concrete zones are shown with cyan fills in the calculation sections. The number of displayed sections can be set with **Display options**.

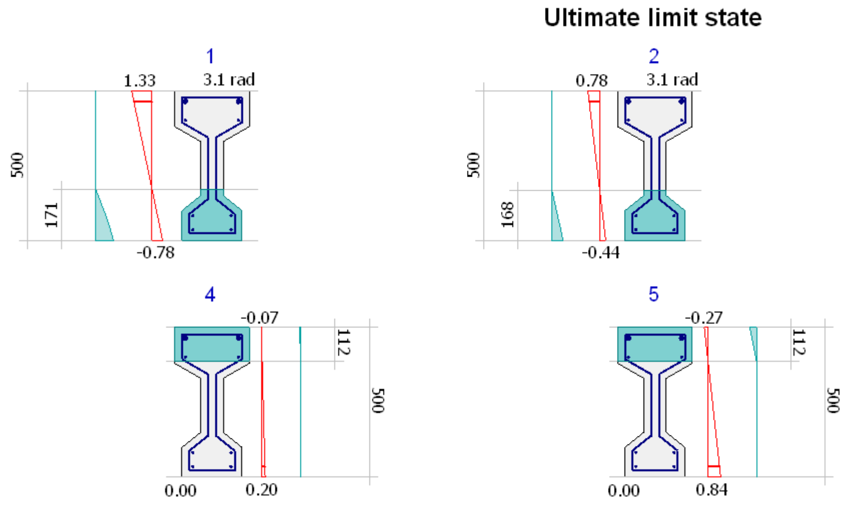


Figure: Stress-strain graphs by sections

- **Utilization graphs**
  Section (Eurocode2: 6.1), *Stirrup* (6.2, 6.3), *Concrete* (6.2, 6.3), *Torsional reinforcement* (6.3) and *Crack width* (7.3) utilization graphs together with a *Summary* graph are displayed with legends by default. Numeric values can be inquired in the calculation sections (***Design calculation parameters*** sets the maximum distance of sections).

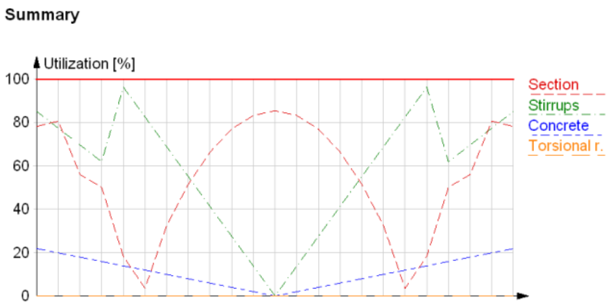


Figure: Utilization graph

Tabmenu contains the following tools and settings:

- **Selection of element to display**
  You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

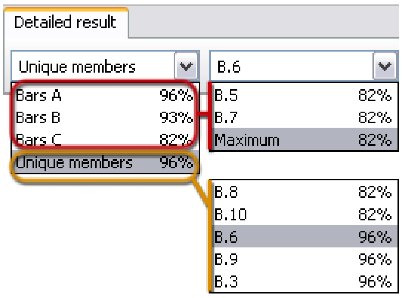


Figure: Selection of a unique or a group member

- **Selection of design load**
  Depending on RC design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means the significant load combination or component of load groups.

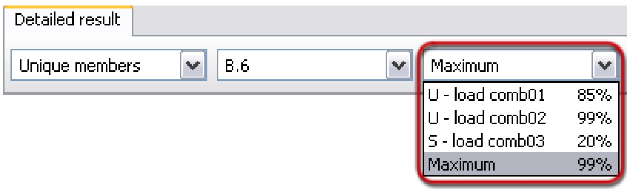


Figure: Selection from design loads

-  **Auto design**
  Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts RC design that updates all detailed result figures and tables.

- **Manual design**
  Manual design can be launched directly for the currently displayed unique/group member. Returning from reinforcement editing updates all detailed result figures and tables.

- **Display options**
  The content and the appearance of the detailed result can be set with *Display options*. For tables or stress-strain graphs, you can choose all, maximum and characteristic calculation sections to display.

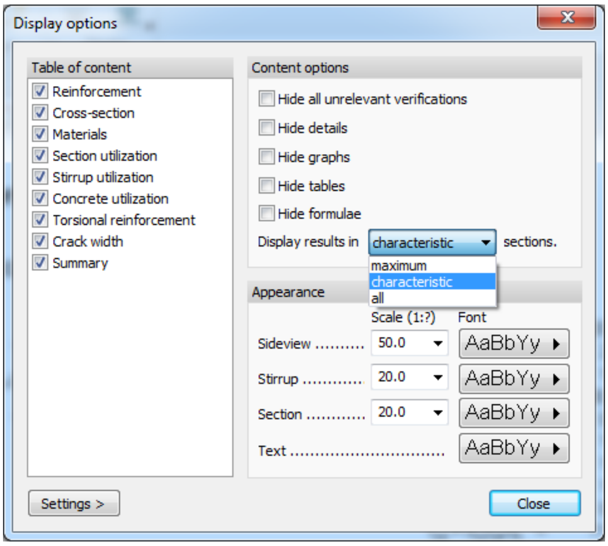


Figure: Display options of Detailed result

- **Go to**
  Navigate in the *RC bar detailed result window* by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

Click *Tools > Add view to document* to place all figures and tables or specified details only into *Documentation*.

Export RC bar reinforcement into *.dwg or *.dxf file format by clicking File/Export to/Export to AutoCAD…

Surface Reinforcement

Surface reinforcement design needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups* and initial reinforcement properties (direction, shape, steel quality, diameter and concrete cover) defined by *Design calculation parameters*.

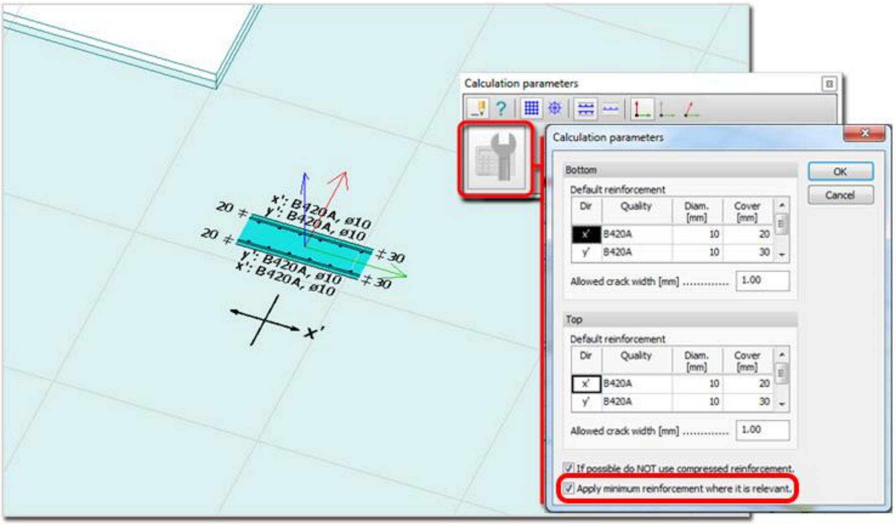


Figure: Initial reinforcement settings

The reinforcement shape (*Straight/Centric*) and the bar directions (*x'/y'/r/t*) set by *Design calculation parameters* will be fixed parameters in *Auto* and *Manual design*, so only they can be edited/modified only with *Design calculation parameters*.

Although all design results can be calculated for all reinforcement types, *Auto design* does not work for the *Centric* reinforcement! Only *Manual design* can be used to define and edit the required centric reinforcement area.

A Plate or Wall can be specified as single layer reinforced by defining "Single layer reinforcement" Calculation parameter for it.

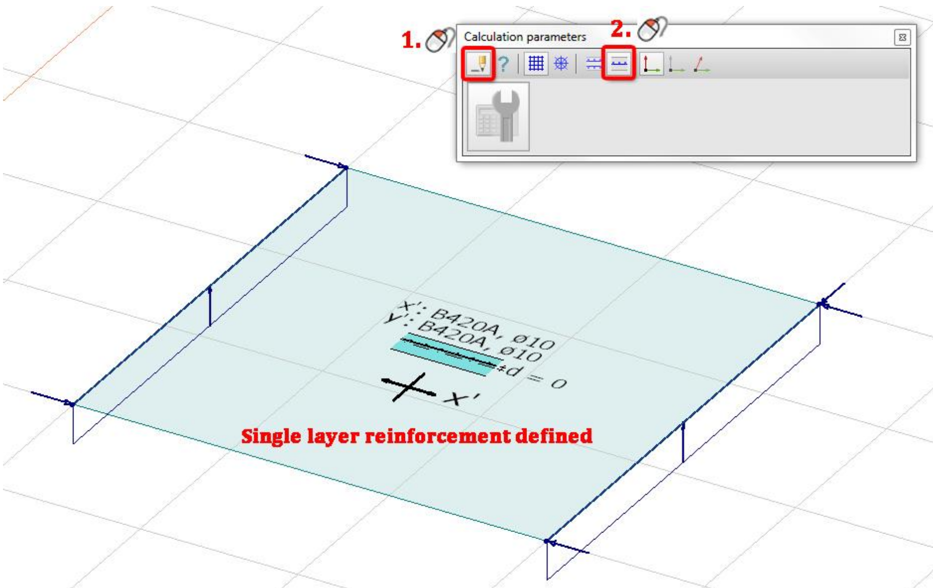


Figure: Single layer reinforcement definition

In the Calculation parameters dialog the User can define the followings:

- the quality and diameter of the reinforcement for both directions,
- the direction of the bottom layer,

- the distance of the reinforcement from the centreline,
- the allowed crack width on the bottom and on the top of the structure.

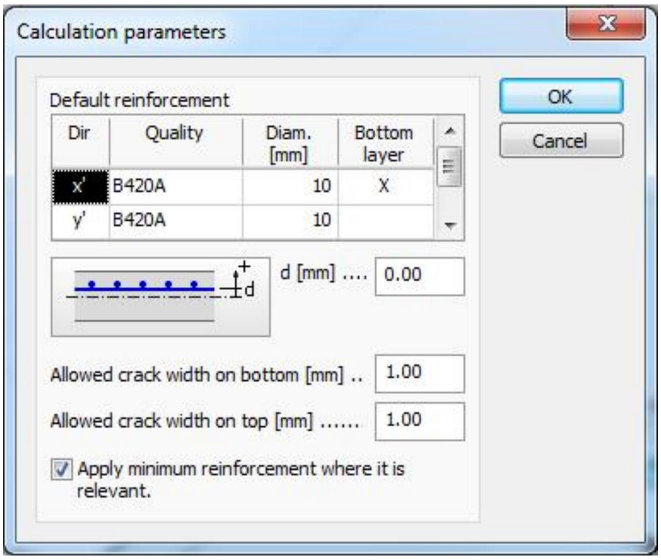
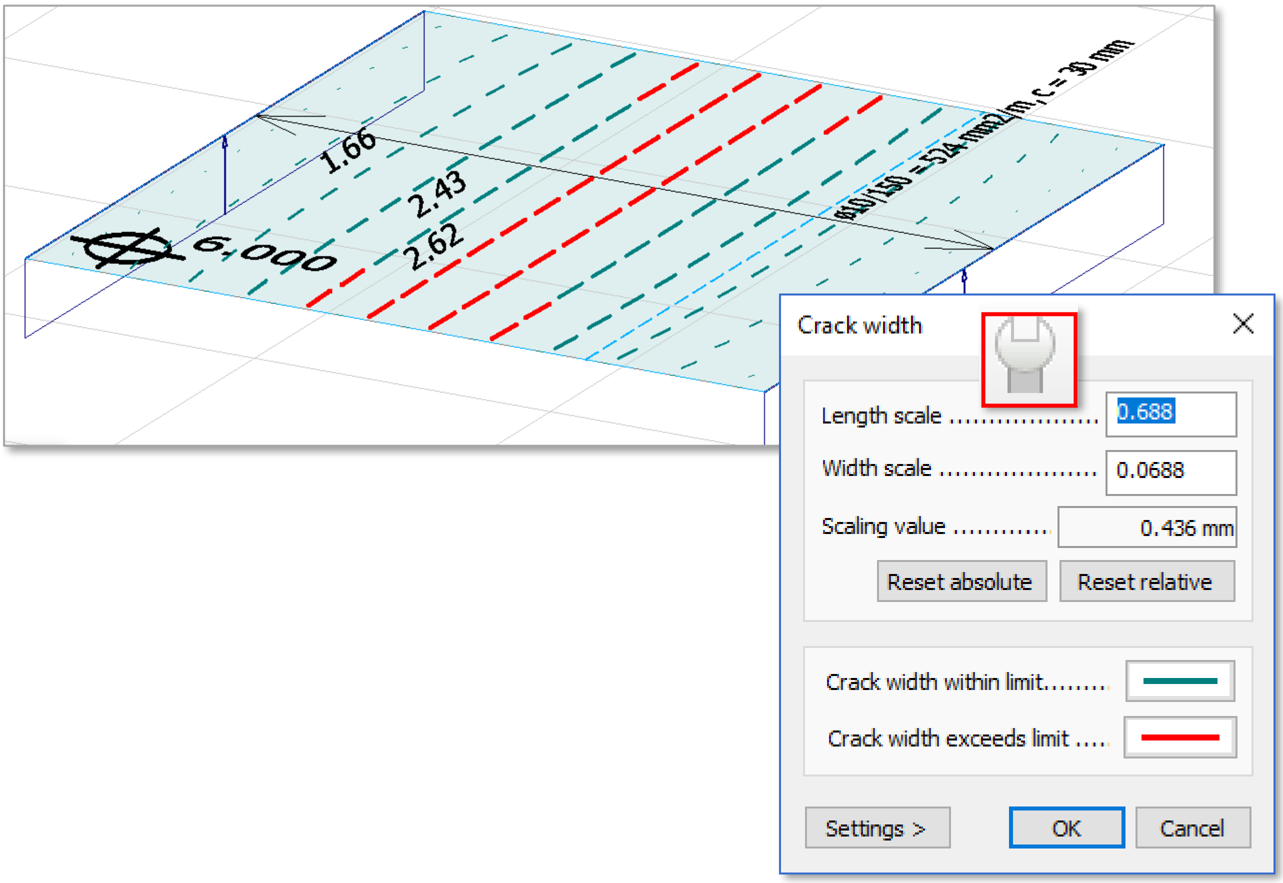


Figure: Calculation parameters for single layer reinfrocement

Crack width values that exceed the specified limit are displayed with a different colour. The maximum allowed value of crack width can be set at the *Calculation parameters* ( ). At the *Display options* ( ), the weight, scale and colour of the crack lines can be defined.



If a shell has “Single layer reinforcement” Calculation parameter, its Design parameter can be modified only if “Single layer reinforcement” option is selected in Auto design/Parameters.

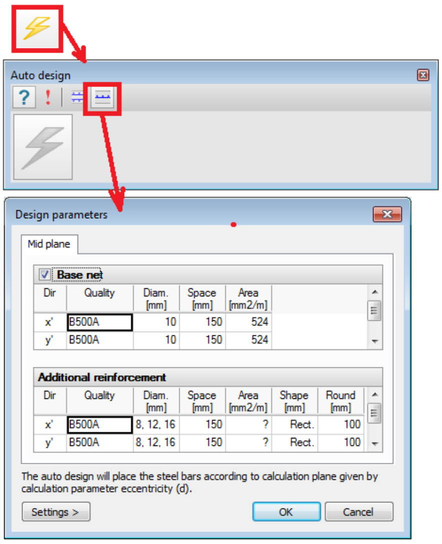
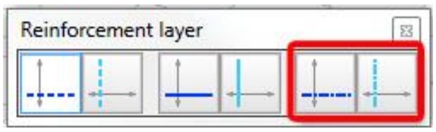


Figure: Auto design parameters of single layer reinforcement

In case of Manual design, single layer reinforcement can be placed only on „Mid, x'/r" and „Mid, y'/t" layers.



⚠ Single and double layer reinforcements cannot be used in the same Plate or Wall element.

Auto Design

Global *Auto design* gives design force and required reinforcement results and suitable applied reinforcement for all concrete surfaces of the current project. Furthermore, *Auto design* calculates missing reinforcement and crack width based on the determined applied reinforcement.

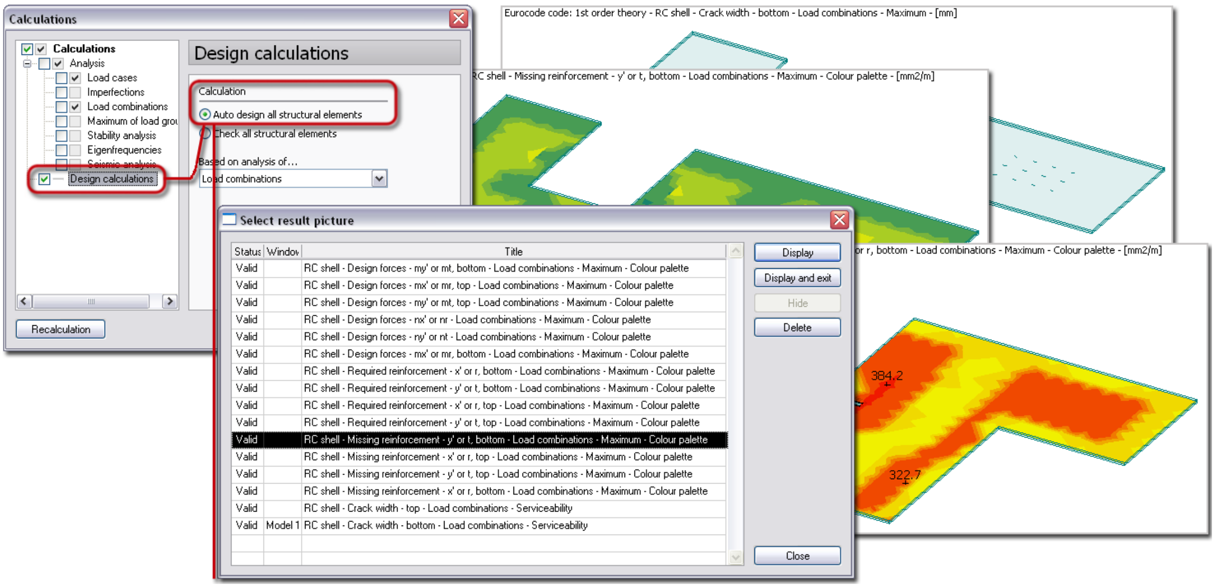


Figure: Global Auto design and its result

Element-based *Auto design* finds the most suitable top and bottom (or mid) reinforcement for selected concrete plate, wall and shell elements or element groups only according to their internal forces and design parameters. Initial settings of the base net and additional reinforcement by positions (*Bottom face/Top face/Mid face*) and directions (*x'/y'/r/t*) can be set with the *Parameters* tool of *Auto design*.

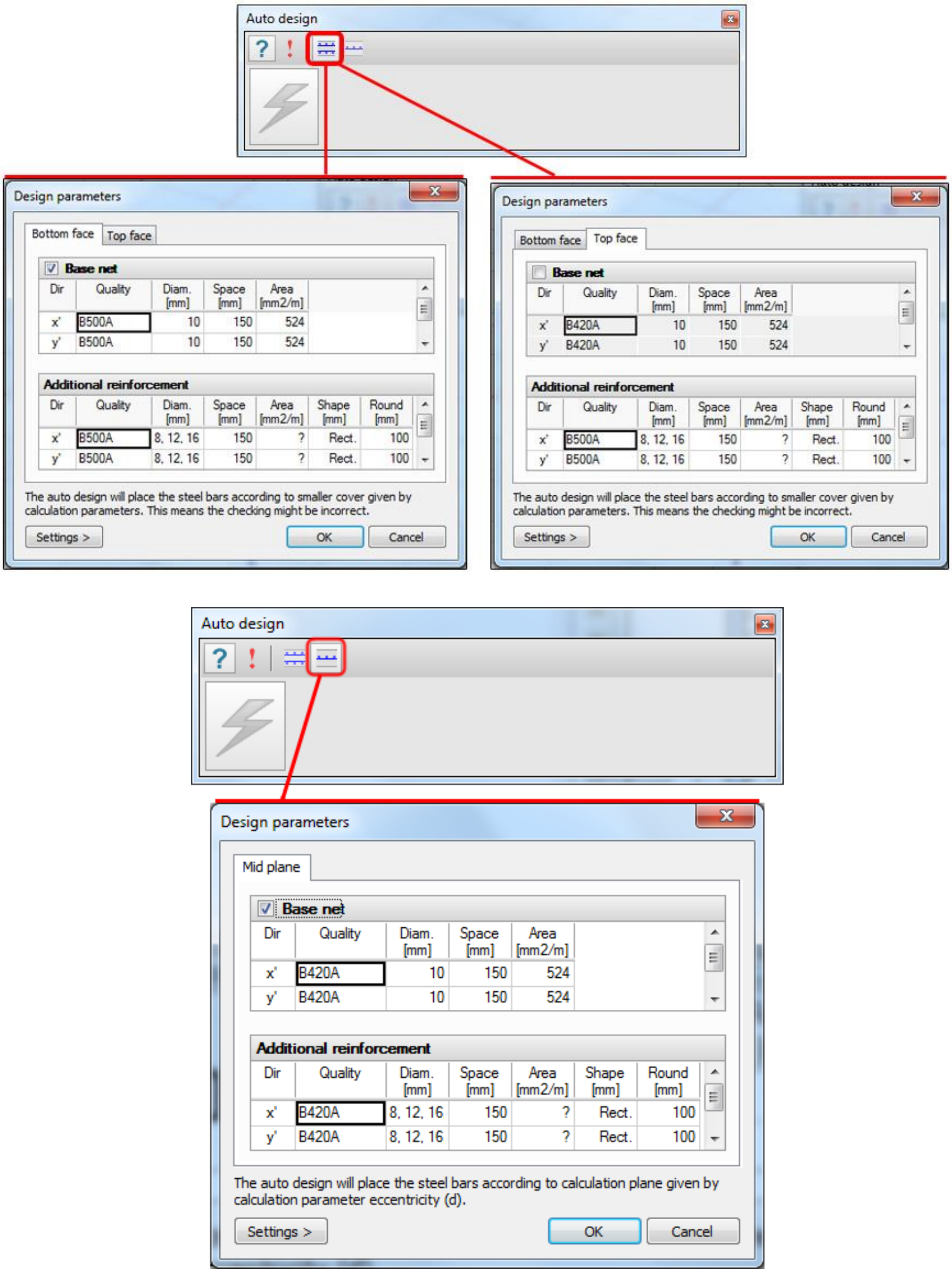


Figure: Design parameters

The minimum concrete cover together with bar directions is derived from *Design calculation parameters* settings.

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process finds the suitable bar diameter for the additional reinforcement from the defined diameter range, calculates the bars' utilization and distributes top and bottom reinforcement where required.

The placed bars can be displayed by their directions and positions with the navigator icons of the *Reinforcement layer* tool palette.

The generated applied reinforcement is also visible in Manual design, where additional reinforcement can be defined or the current state can be edited.

⚠ In case of design groups, the applied reinforcement appears only at the Master group member.

Check the tool palette's *Display table* box to have a look at the overall utilization results given for the designed surface elements or design groups.

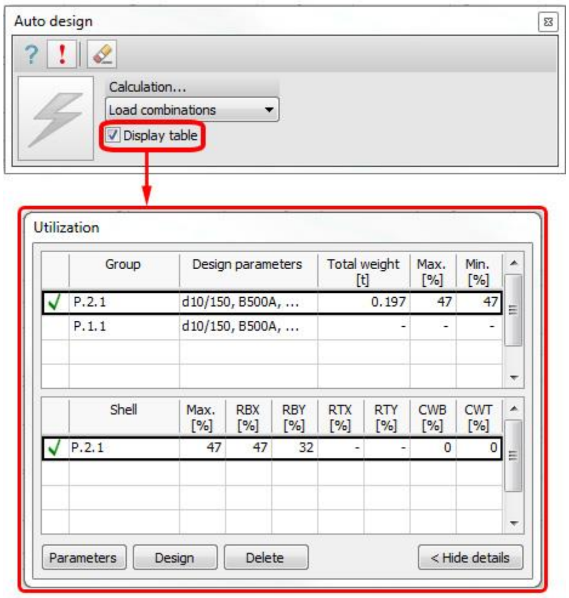


Figure: Quick summary of Auto design results

The upper table shows the design efficiency and the maximal utilization of the designed single elements and groups based on the given design parameters. The bottom table displays the utilization details of the surface element or the elements of the group selected in the upper table.

	Meaning
	Suitable reinforcement is available
	Suitable reinforcement is not available Note: Modify the element thickness, material or RC design parameters.
Group	ID of a single element or a group name
Design parameters	Main applied design parameters

Total weight	Total weight of applied reinforcement
Max	Max. utilization of a single element or the significant member of a group
Min	Max. utilization of the less significant group member
Shell	ID of a single element or a group member
RBX	Utilization of bottom x'/r reinforcement
RBY	Utilization of bottom y'/t reinforcement
RTX	Utilization of top x'/r reinforcement
RTY	Utilization of top y'/t reinforcement
CWB	Utilization for crack width on the bottom face
CWT	Utilization for crack width on the top face

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

1 Select a surface element or a design group in the upper table.

2 Modify the design parameters of the current elements under *Parameters*.

3 If the selected surface elements already contain applied reinforcement defined by *Manual design* or earlier *Auto design*, you can delete it by activating  on the *Auto design* tool palette.

4 Click *Design*.

Manual Design

 *Manual design* gives tools to define new (applied) reinforcement in concrete plates, walls and shells, or to modify/redefine the reinforcement generated by *Auto design*.

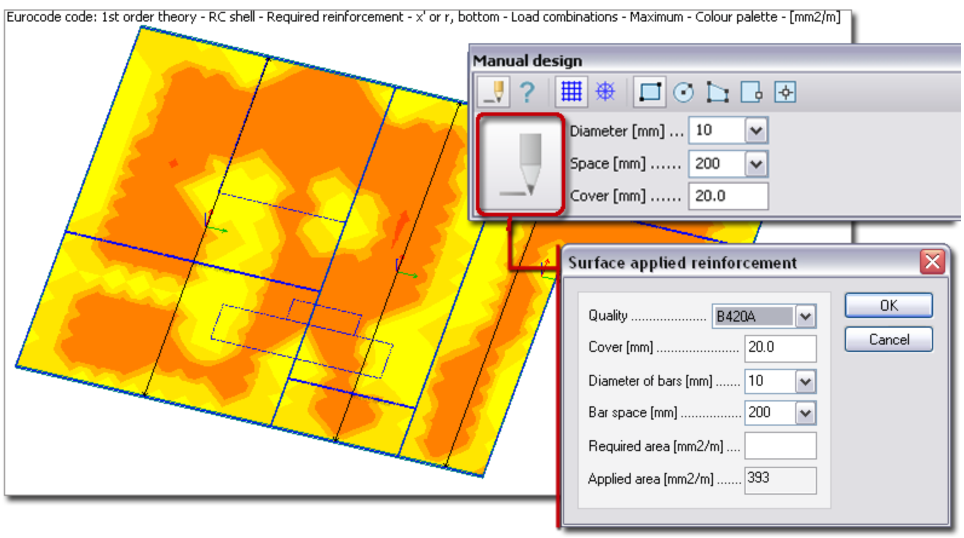
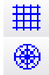


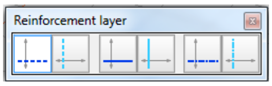
Figure: Manual design tools

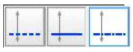
If needed, applied reinforcement defined earlier by *Auto design* or earlier *Manual design* can be deleted with *Edit* > *Erase*. The geometry of the applied reinforcement regions can be edited with the region-related *Edit* commands such as *Region operations*.



To define new reinforcement in a surface element, first choose the same reinforcement shape (*Straight*/*Centric*) set earlier for the surface element with Design calculation parameters.

Choose the geometry of the new surface reinforcement.

Choose the reinforcement layer with 



Set the position (*Bottom face*/*Top face*/*Mid face*) of the new surface reinforcement.

Set the direction (*x'*/*r*/*y'*/*t*) of the new surface reinforcement.

All parameter of the new reinforcement bars can be set under *Default settings* or the main parameters on the *Manual design* tool palette.

According to the geometry place the surface reinforcement with its required points. The new surface reinforcement will be situated parallel with the plane of the host element with the defined concrete cover thickness.

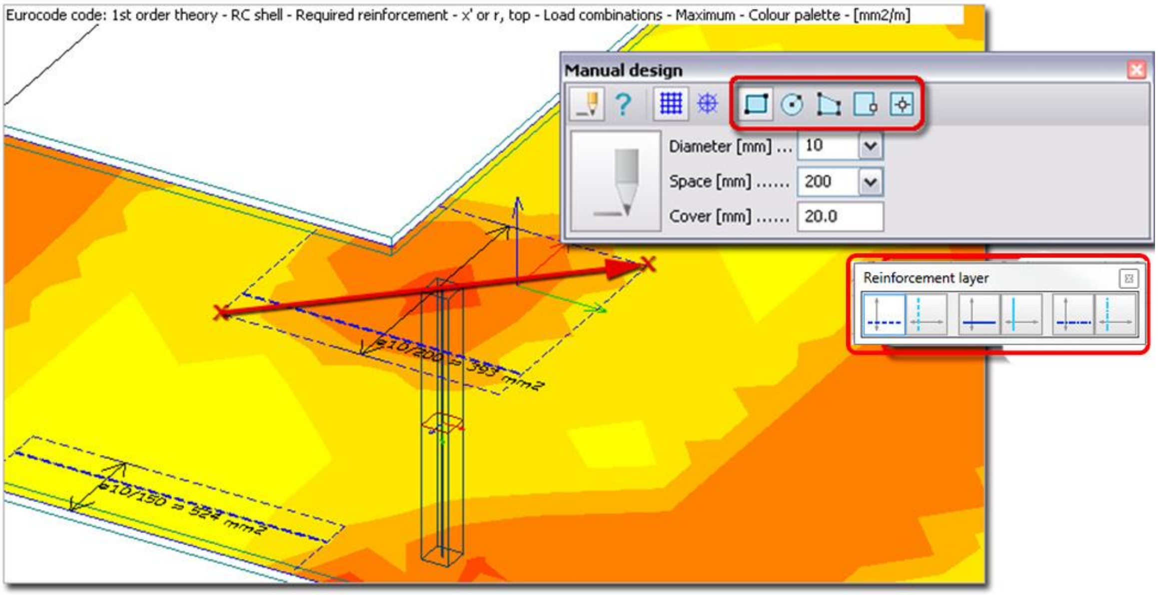


Figure: Additional reinforcement defined with Manual design

In case of design groups, the applied reinforcement of the group Master is editable only.

The active layer will automatically follow the result displayed: e.g., when showing stresses ont he top surface of a slab, the top layer of reinforcement will be activated.

You can modify the properties (quality, diameter, spacing, cover) of previously defined surface reinforcement(s) with the *Properties* tool of *Manual design*.

Running global (*Calculate* > *Design calculations* > *Check*) or element-based *Check* the program gives *Applied reinforcement* result, so the applied area of surface elements can be displayed with *Color palette*, *Contour lines*, *Graph* or *Sections mode*.

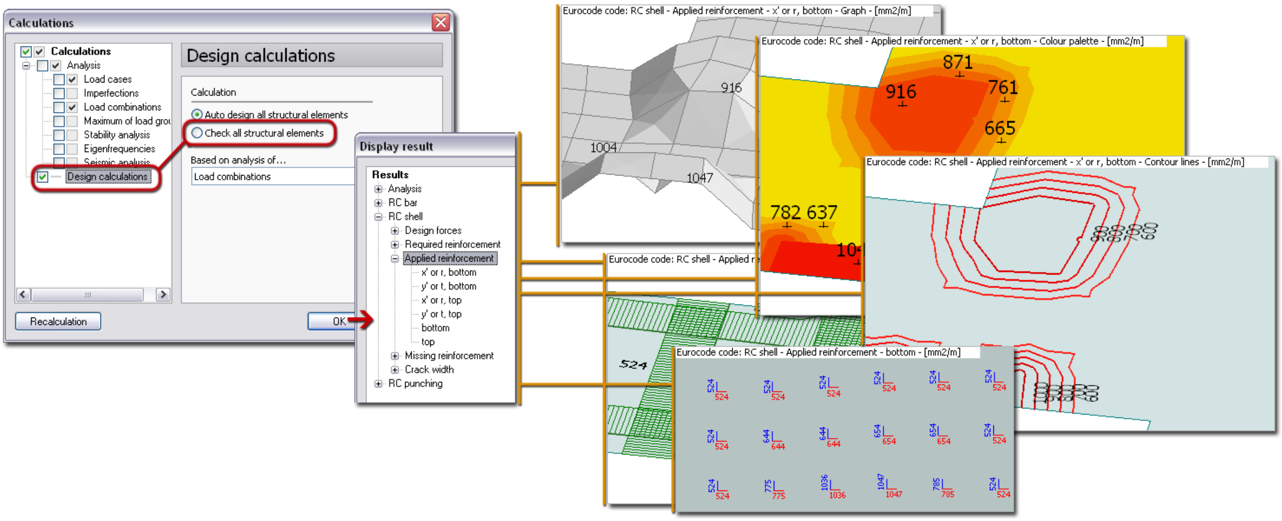
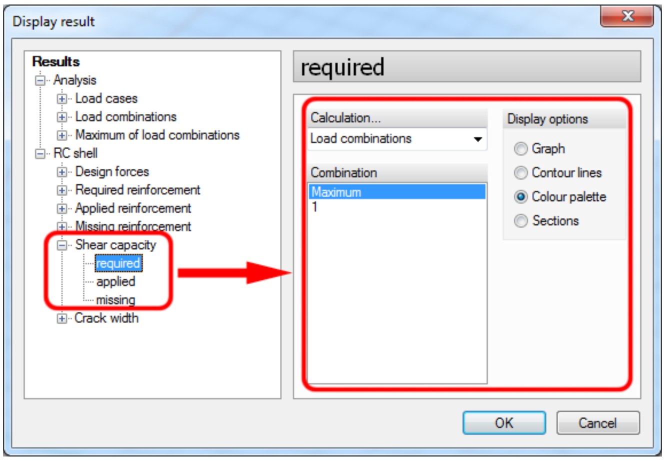


Figure: Applied reinforcement area displayed with Graph, Color palette, Contour lines, Sections and Bi-direction mode

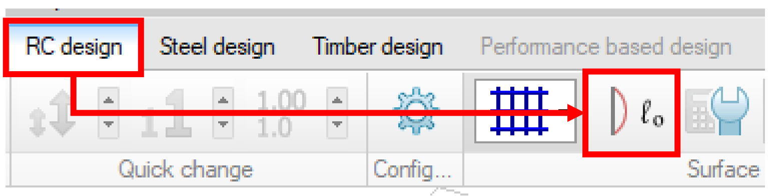
Shear capacity
FD calculates RC shells (*3D Plate, Wall*) shear capacity and their results can be seen from *New result / RC Design / Shear capacity*. These results can be displayed in *Graph, Contour Line, Color palette* and *Section* format.



Shell buckling
The buckling problem of the shell is transformed to the buckling of equivalent columns made from the shell, on which the second order resistance and utilization is calculated.

Only RC Plane plates and Plane walls with straight reinforcement and uniform thickness are suitable for shell buckling calculation.

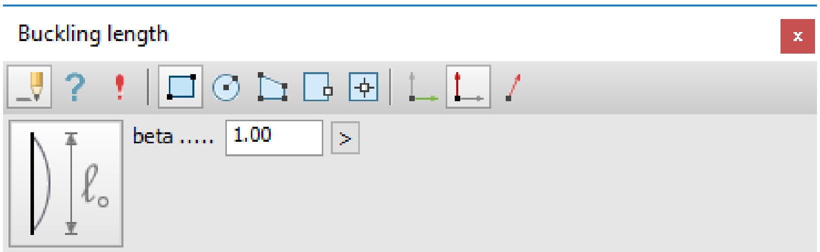
The calculation process is based on so-called buckling regions, which can be defined at *RC design/Surface reinforcement/Buckling length.*



Each buckling region on the shell has a corresponding buckling factor (beta) and a direction vector in the plane of the shell. The former will be used to calculate the buckling length of the equivalent column, while the latter one specifies the x' longitudinal axis of this column. By default, FEM-Design generates one buckling region on each RC wall and plate. Default buckling direction is vertical on walls, and parallel with the local x axis on plates. **Buckling factor is set to 0.0 on all shells in order to let the User decide whether this calculation is needed or not, since it is quite time consuming**.

Shells with zero buckling factor will not be considered for shell buckling calculation, but zero utilization is set for them.

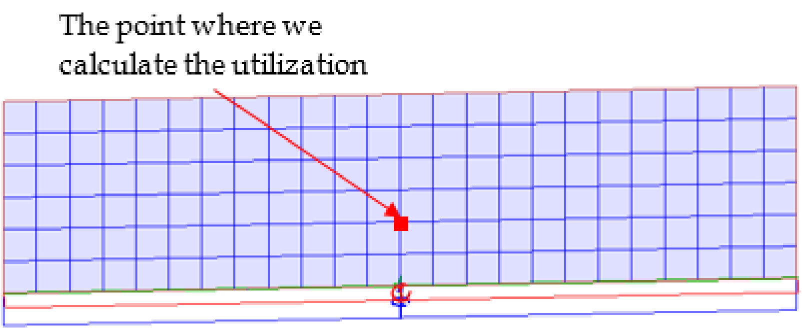
The default buckling regions can be modified by adding new regions to the shell. One shell may have more buckling regions with different beta factor and direction vector, but the shell must be completely covered by these regions.



During the checking process, the program generates equivalent bar(s) from the shell based on its material, thickness and reinforcement. This bar is checked as an RC bar: Its utilization is calculated by determining its second order internal forces and resistance.

The calculation process consists of the following steps:

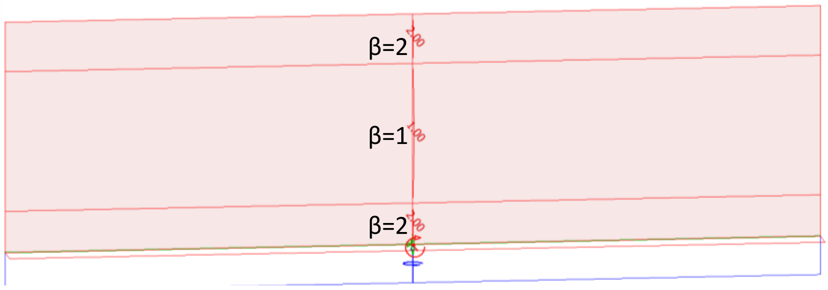
1. As other shell design calculations, the shell buckling is also calculated in every node of the shell (only where there is a buckling region with non-zero beta value).



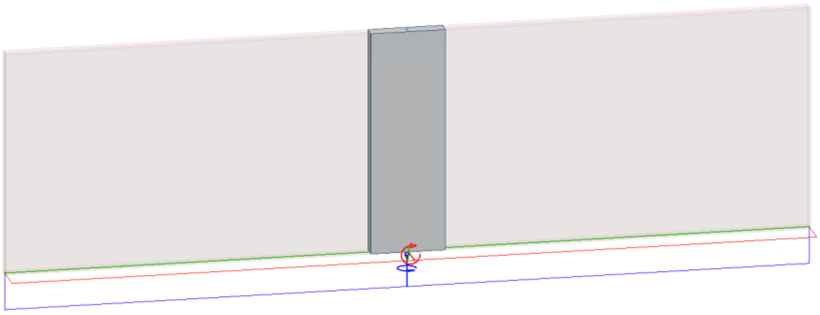
2. An equivalent bar is generated for the examined node as follows. The edges of the shell are intersected by the ray determined by the node and the direction vector of the corresponding

buckling region. The two intersection points are taken as the start and the end point of the equivalent bars.

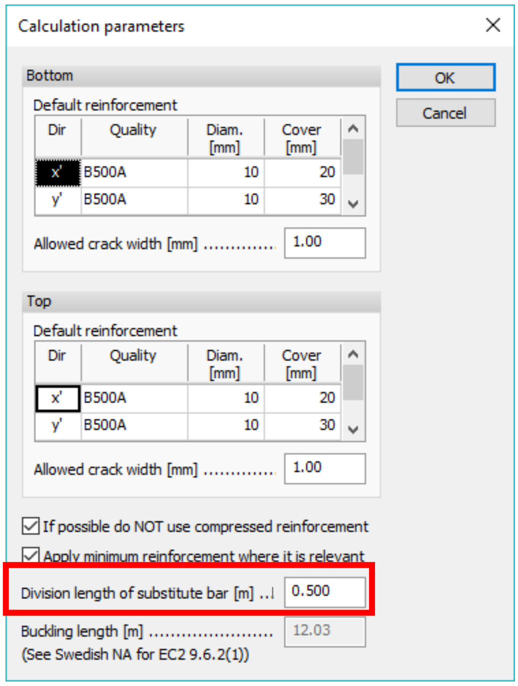
⚠ Note that this intersection is always made with the edges of the shell and not with the edges of the buckling region corresponding to the node! If a node is on the border of two or more buckling regions, it is calculated with both different beta values and direction vectors, and the higher utilization will be used.

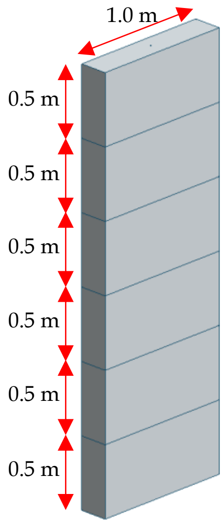


3. The cross section of the equivalent bar is 1 m wide and its height equals to the thickness of the shell. Along the bar, the applied reinforcement of the shell is transformed into the direction of the bar and placed into it.

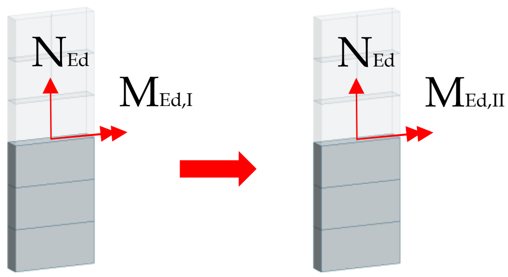


The checking process is executed section by section along the bar. The distance between these sections is given by *Division length of substitute column* parameter in Calculation parameter dialog (see the lower figure). Internal forces acting at these sections are calculated by transforming shell internal forces at the section point into the coordinate system of the column. As the buckling direction of shells is perpendicular to its plane, we need the equivalent bar's normal force and moment vector in the plane of the shell for the calculation.



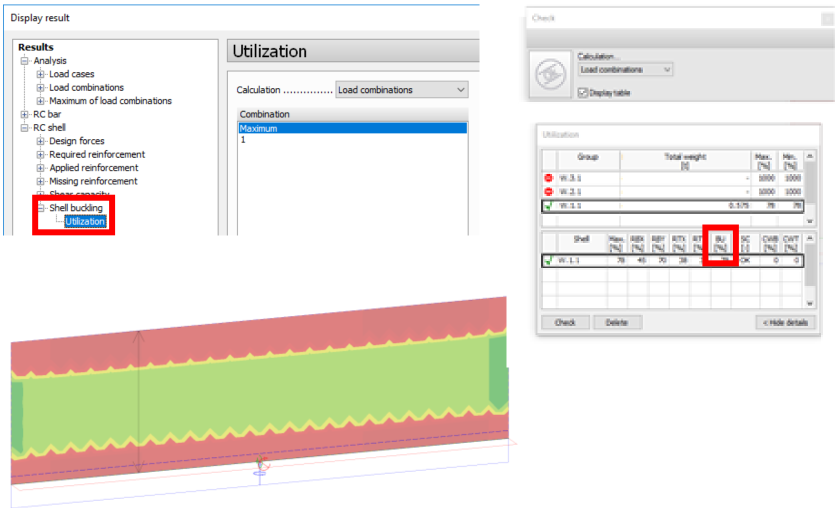


4. Once the first order internal forces are obtained in every section, the second order internal forces are calculated based on the *nominal stiffness* or *nominal curvature* method, according to the configuration settings. The only difference in the checking process of a real bar and this equivalent bar is that now the eccentricity coming from the second order effects are applied only perpendicularly to the plane of the shell. In other words, the out-of-plane normal force has eccentricity only along the z' axis of the shell. This modification is in harmony with the fact that the buckling direction of the shell is perpendicular to the plane.

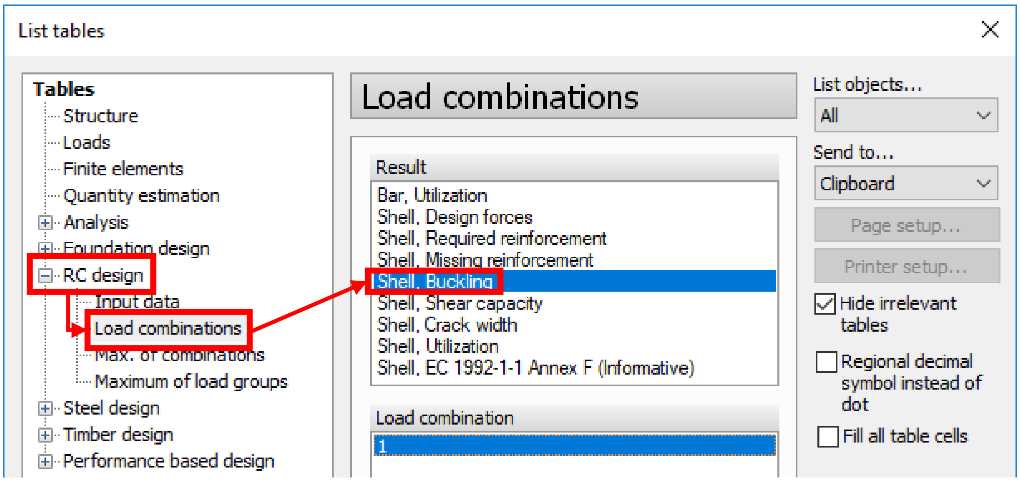


5. Finally, based on the second order internal forces, the utilization is calculated for every cross section of the equivalent bar (based on the interaction curve), and the highest section utilization is assigned to the node.

Shell buckling calculations are available for *Load combinations*, *Maximum of load combinations* and *Maximum of load groups*. The utilization results can be displayed in the *New result/RC shell/Shell buckling/Utilization*



Some details of the calculation can be obtained by listing *RC design/Load combinations/Shell, buckling* table. Also, wall buckling utilization appears in the *Shell, Utilization* list.



Shell, Buckling, Load comb.: 1

ID	Utilization	x	y	z	As, top	As, bot	As, mid	N_Ed
[-]	[%]	[m]	[m]	[m]	[mm2/m]	[mm2/m]	[mm2/m]	[kN/m]
W.1.1	28	25.519	33.973	-3.000	523.599	523.599	0.000	-98.771
W.2.1	144	43.200	26.320	-3.000	523.599	523.599	0.000	-166.218

M_Ed	M2_Ed	N_Rd	M_Rd	Bar length	Beta
[kNm/m]	[kNm/m]	[kN/m]	[kNm/m]	[m]	[-]
9.146	14.733	-353.254	52.693	3.000	2.000
31.101	57.126	-115.298	39.626	3.000	2.000

Every plate and wall has one result, containing the coordinates of the dominant section, the corresponding reinforcement, first and second order internal forces together with the capacity and buckling factor.

Punching Reinforcement

Punching check and design can be done according to Eurocode 2 both in FEM-Design *Plate* and *3D Structure* modules. It is recommended to perform after **Surface reinforcement design**, because surface reinforcement influences punching calculations.

The punching regions are automatically defined by the program at the plate – coumn intersections and you can define them manually.

Punching design/check can be done at any point where a punching region is exists. Design and check can be done by these punching zones or by their **design groups**.

The program does not generate punching objects for columns that are connected to more than one slab at the same level.

Initial properties are needed to be set for punching zones. Use *Calculation parameters* to define the followings:

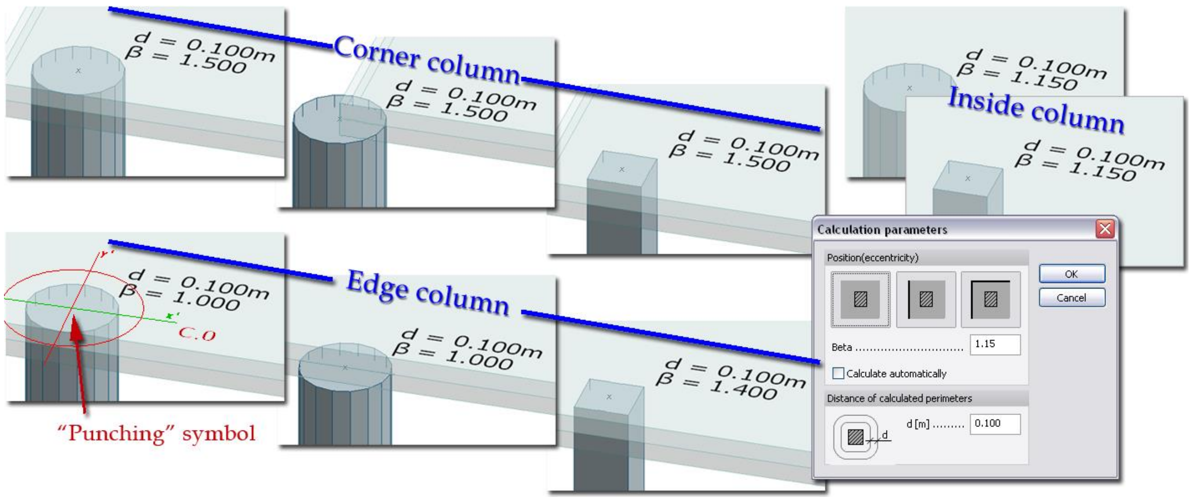
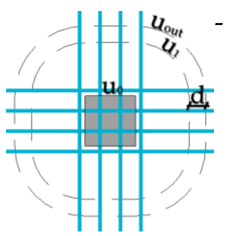


Figure:Setting β values for punching objects

- **Beta (β)**
  According to Eurocode 2, *β* coefficient is taken into account as the effect of any eccentricity of loading. Its value depends on column position. The program offers default standard values for column positions: for corner (β=1.5), for edge (β=1.4) and inside columns (β=1.15). These standard values have to de added manually by selected punching zones or by their groups. If you neglect *β* values, choosing *Calculate automatically* defines them during design/check process and gives accurate results for *β* coefficients (see **Detailed results**).



- **Distance of calculated perimeters (d)**
  This value is used in slightly different ways in checking and design process:

  Checking: perimeters to check between u_1 and $u_{out(,ef)}$ are generated with d distance from each other.

Design: searching for $u_{out(,ef)}$ perimeter starts from u_1 and distance of checked perimeters from the column is increased by the d distance until $u_{out(,ef)}$ is found.

Calculation parameters are displayed on the model view by the *Punching, calculation parameters* object layer.

Auto Design

Global *Auto design* gives utilization results for punching. Auto design applies the **initial punching reinforcement settings**.

Punching design always deletes the existing reinforcement and generates new one.

Defining punching regions

When clicking on the , the dialog appears:

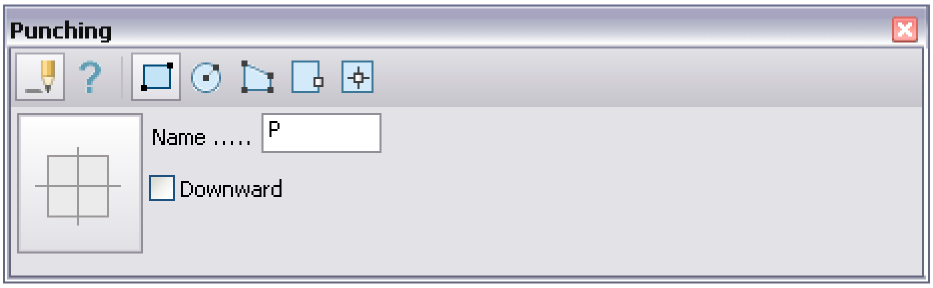


Figure: Punching region toolwindow

First the origin of the local system should be specified, which is also the calculation point for punching. The punching force is taken from the finite element corner node closest to the local system's origin.

Before analysis, the user should place a fixed point in Finite element tab, where a punching calculation is planned to be performed, because in this way it will be surely a corner node, leading to higher precision in punching calculation.

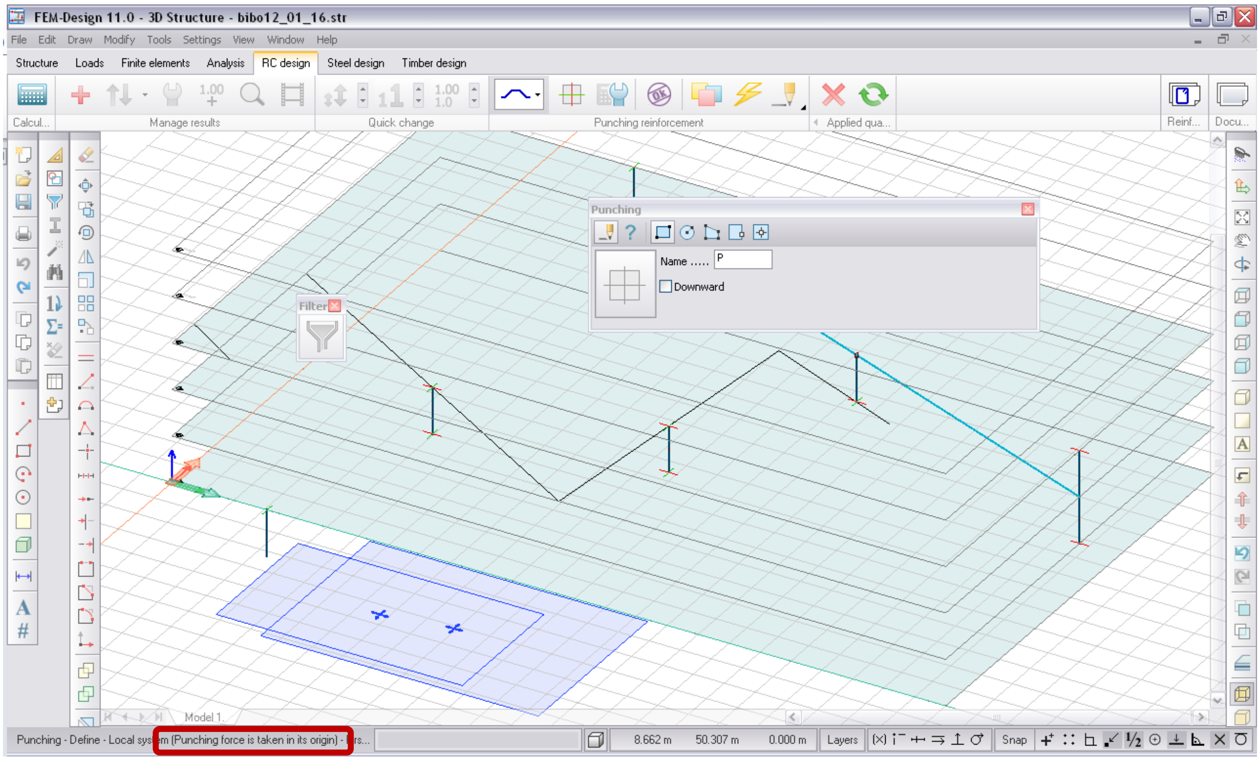
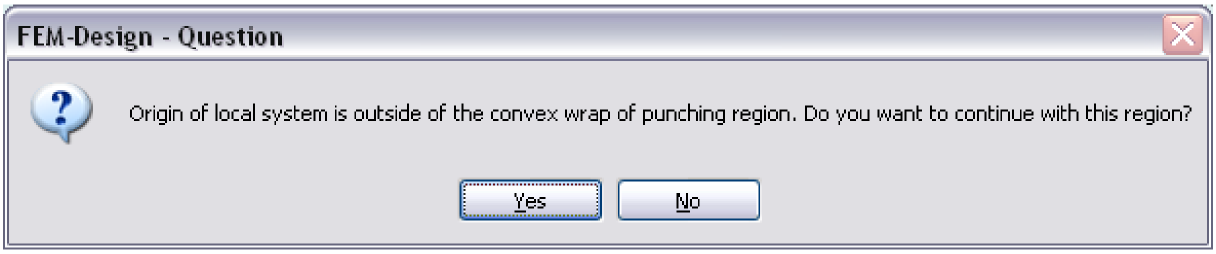


Figure: Punching region example

If you pick a punching force calculation point away from the region the following appears:



Punching utilization results can be displayed for the entire model with *New result* > *RC punching* > *Utilization*. Maximum utilization in table format can be also shown with *Numeric value*. Detailed utilization results can be asked by unique or grouped punching zones with  (see later). Different colors display the adequacy of the checked punching zones.

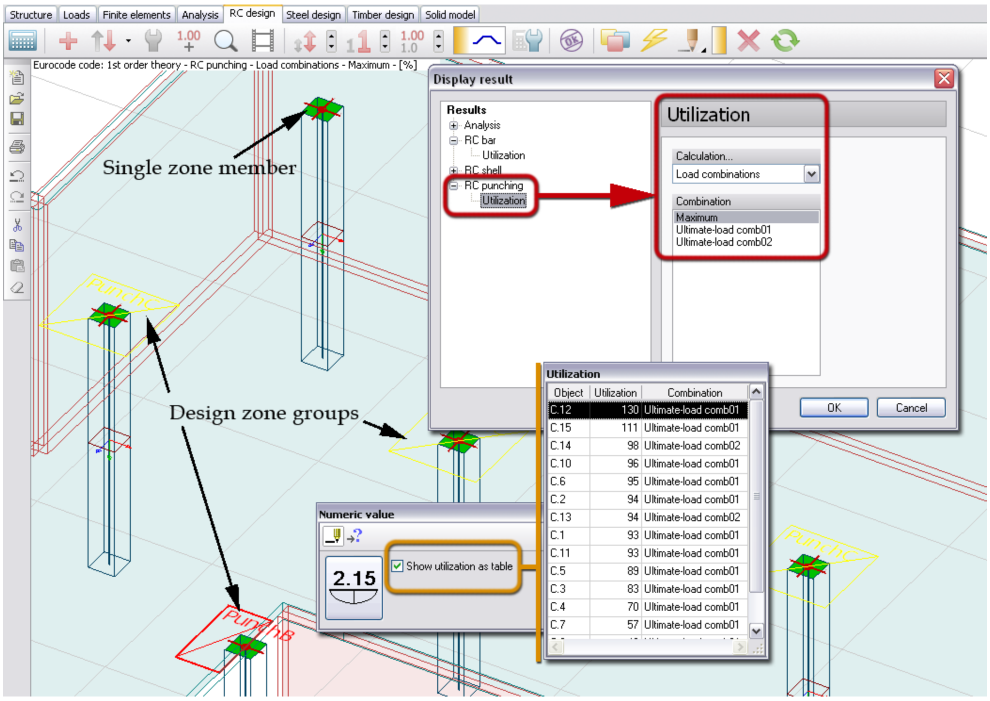


Figure: Global Auto design and its result

Element-based *Auto design* checks the utilizations against punching and finds required reinforcement for selected zones and/or design groups of zones. Initial settings can be set for required reinforcement by bended bar, circularly placed stirrup and open stirrup types with the ? *Parameters* tool of *Auto design*.

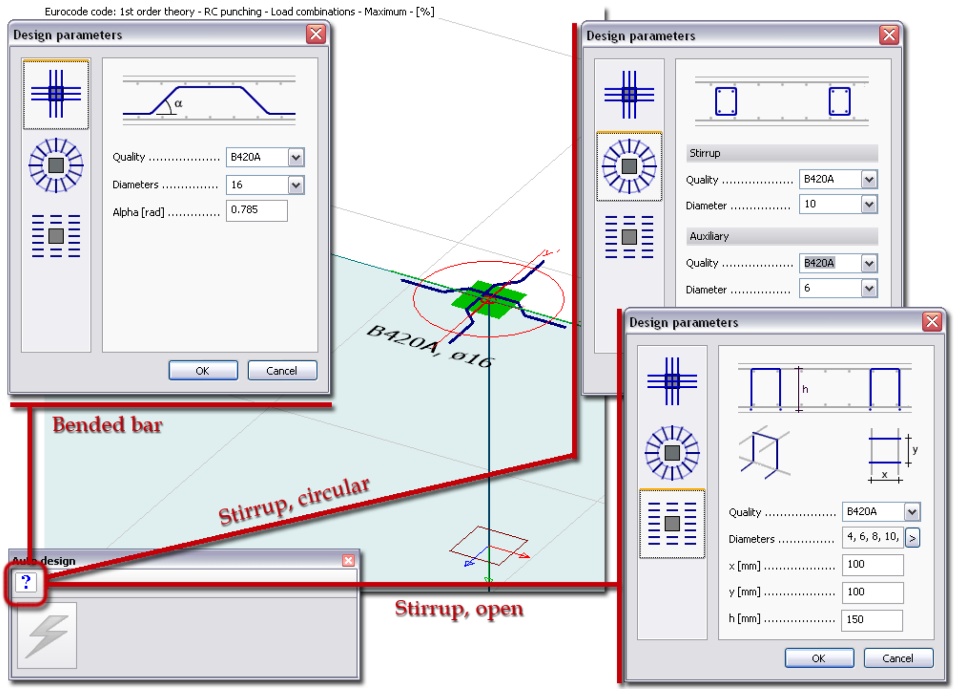


Figure: Design parameters by punching reinforcement type

The symbols of the design parameters are stored on the *Punching, design parameters* object layer.

To run element-based design for the load combinations or the maximum of load groups, select the required zones and/or groups with the *Auto design* command and click *Design* tool. The quick process runs detailed utilization and finds the suitable bar diameters for the additional reinforcement.

Check the tool palette's *Display table* box to have a look at the overall utilization results.

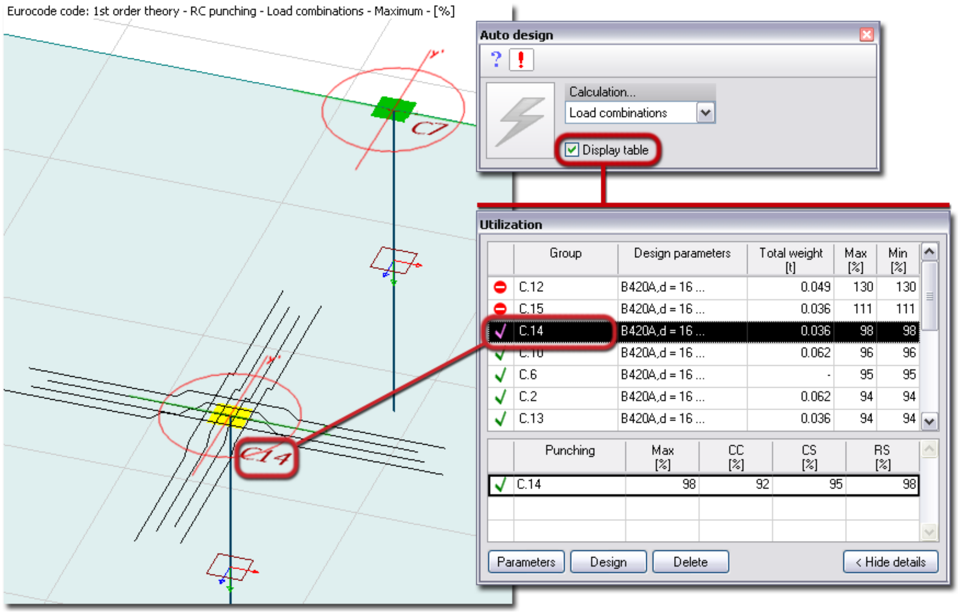


Figure: Quick summary of Auto design results

The ID of design groups can be shown with the *Punching, design groups* object layer.

The upper table shows the design efficiency and the maximal utilization of the designed punching zones based on the given design parameters. The bottom table displays the utilization details of elements or design members.

	Meaning
	RC column is suitable for punching without or with shear reinforcement
	Suitable shear reinforcement is not available Note: Modify the slabs' surface reinforcement or the design parameters.
Group	ID of a unique punching zone or design group
Design parameters	The suitable bar diameter and other design parameters as the reinforcement type
Total weight	- The "-" symbol displays that no shear reinforcement is applied - Total weight of applied punching reinforcement
Max	Max. utilization of a unique zone or the significant zone of a group
Min	Max. utilization of the less significant group member
Punching	ID of a unique column or a design group member

CC	Utilization for concrete compress
CS	Utilization for concrete shear
RS	Utilization for reinforcement shear

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

1 Select a punching zone or a design group in the upper table.

2 Modify the design parameters of the current elements under *Parameters*.

3 Click *Design*.

Detailed result summarizes the applied formulas by the design modes and display the applied shear reinforcement (if needed) drawing too.

Manual Design

Manual design gives tools to define new (applied) punching reinforcement in concrete plates, or to modify/redefine the reinforcement generated by *Auto design*.

The rules of new reinforcement definition, if predefined reinforcement already exists from *Auto design*:

- new same-type reinforcement will be added to applied reinforcement,
- new different type reinforcement always overwrites the previously defined one, and
- new "open stirrup"-type reinforcement always overwrites the previously defined reinforcement even if it was "open stirrup".

First choose the type of the new reinforcement. If you would like to modify/refine a previously defined (generated by *Auto design* or defined by *Manual design* in an earlier phase) reinforcement, you also need to set the required type from the drop-down list of *Manual design*. The *Properties* tool of *Manual design* can be used for modifying actions.

The tools of *Manual design* depends on the selected reinforcement type.

Bended bar

First set the reinforcement properties under *Default settings*, and choose the bar direction according to the local system directions of the related column. Then select the punching zone you would like to be reinforced (with totally new or additional reinforcement) and place the new bars.

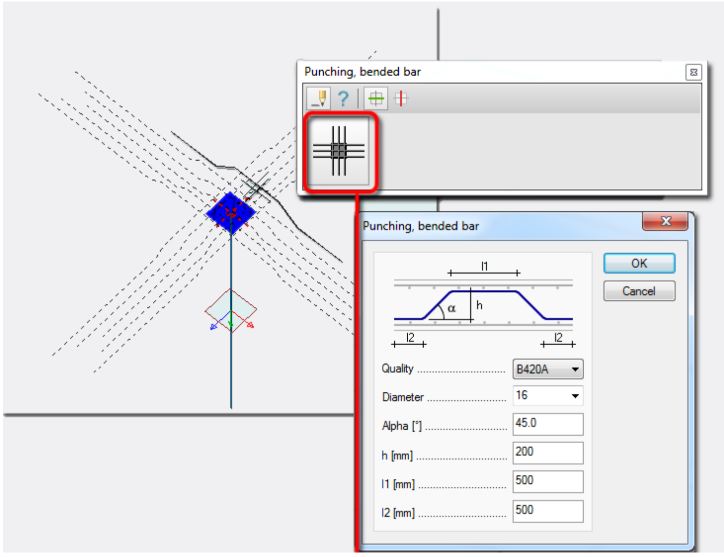


Figure: New bended bar

Stirrup, circular

First set the reinforcement properties under *Default settings*, then select the punching zone you would like to be reinforced, and finally define the circular form with its inner radius.

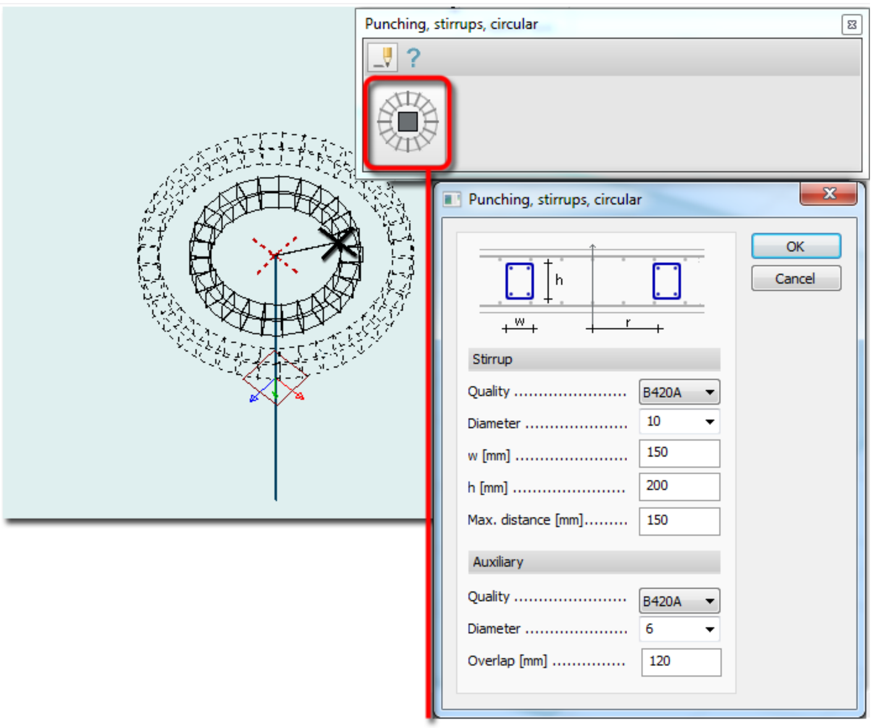


Figure: New stirrups in circular shape

Stirrup, open

First set the reinforcement properties under *Default settings*, then select a geometrical shape (e.g. rectangular, circular, polygonal) for the stirrup position. Select the punching zone you would like to be reinforced, and finally define the shape in the model view.

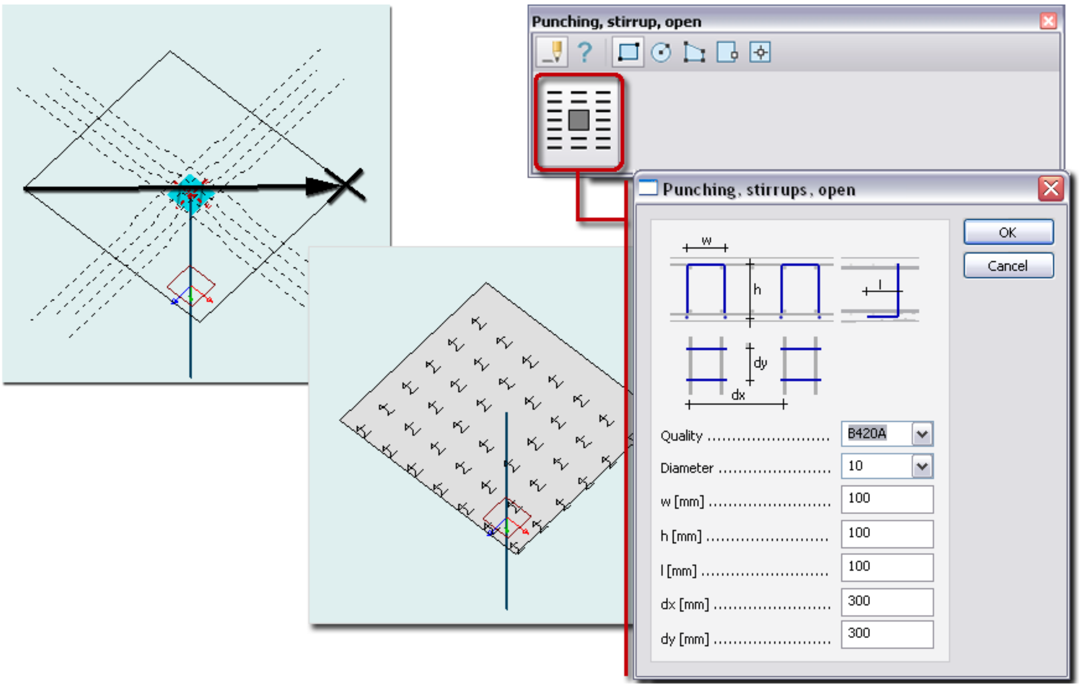


Figure: New open stirrups

⚠ In case of design groups, the applied reinforcement of the **group Master** is editable only. Other group members have symbolic reinforcement figure.

Running global (*Calculate* > *Design calculations* > *Check*) or element-based *Check* the program gives utilization result for the applied punching reinforcement.

Detailed Result
Utilization of punching reinforcement can be displayed in the following cases:

- After global *Auto design*, you can display utilization calculated from the suggested applied reinforcement.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected zones.
- Global *Check* done for final applied reinforcement.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	*Calculate>Design calculation>Auto design all structural elements*	*Auto design*	*Auto design* and/or *Manual design*	*Auto design* and/or *Manual design*
2	*New result >* *RC punching*	*New result >* *RC punching*	*Check*	Apply changes
3			*New result >* *RC punching*	*Calculate>Design calculation>Check all structural elements*
4				*New result >*

				RC punching

Table: Steps of displaying punching utilization by different design cases

Utilization displayed with *New result* appears for all designed punching zones. The utilization components with calculation formulas and the applied reinforcement (if needed) can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a unique zone or a member zone of a design group, which display:

- **Design parameters and applied reinforcement**
  The figure displays the calculation perimeters, the design parameters and the applied reinforcement calculated by *Auto design* (if required) or defined in *Manual design*.

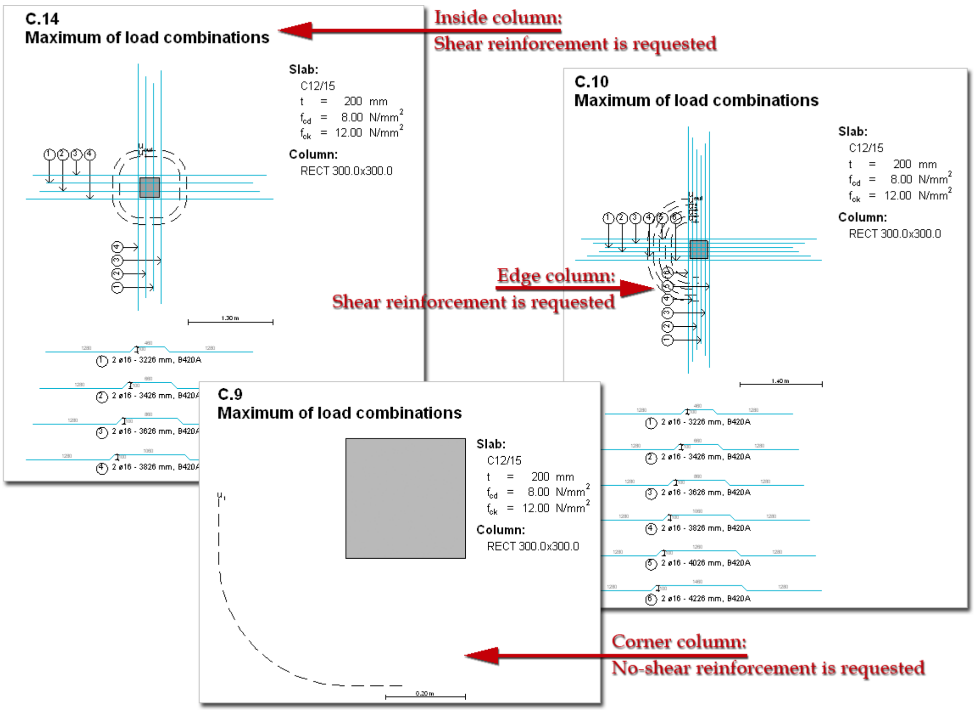
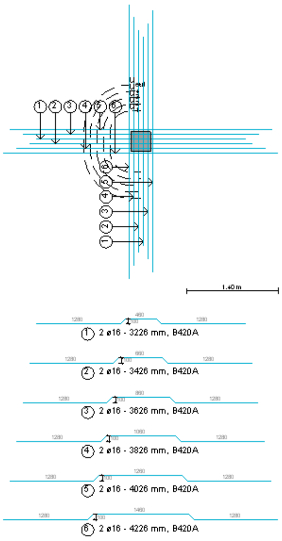


Figure: Calculation perimeters, design parameters and applied reinforcement

- **Detailed calculation formulas**
  Calculation details and final values are collected by checking types: *Concrete compression resistance* (Eurocode2: Part 1.1: 6.4.3), *Shear reinforcement resistance* (Part 1.1: 6.4.3) and *Concrete shear resistance* (Part 1.1: 6.4.3). The proper results are displayed in green, while the red result warnings you to repeat design. The content of the utilization checks depends on **Display options**. Not relevant checks can also be hidden.

C.10
Maximum of load combinations



Slab:
C12/15
t = 200 mm
f_{cd} = 8.00 N/mm^2
f_{ck} = 12.00 N/mm^2

Column:
RECT 300.0x300.0

Shear reinforcement resistance - Part 1.1: 6.4.3

$$v_{Ed} = \frac{\beta V_{Ed}}{u_1 d} \quad (6.38)$$

$$v_{Rd,sw} = 1.5 \frac{d}{s_r} A_{sw} f_{yws,ef} \frac{1}{u_1 d} \sin\alpha$$

$$v_{Rd,cs} = 0.75\, v_{Rd,c} + v_{Rd,sw} \quad (6.52)$$

Perimeter index	1	2	3	4
Dist [mm]	318	418	518	618
Load combination	Ultimate-load comb01	Ultimate-load comb01	Ultimate-load comb01	Ultimate-load comb02
V_{Ed}[kN]	152.53	152.53	152.53	130.74
β	1.00	1.00	1.00	1.00
d[mm]	159	159	159	159
u[m]	1.60	1.91	2.22	2.54
v_{Ed}[N/mm^2]	0.60	0.50	0.43	0.32
$v_{Rd,c}$[N/mm^2]	0.35	0.35	0.35	0.35
$v_{Rd,sw}$[N/mm^2]	83.06	52.11	29.88	13.15
$v_{Rd,cs}$[N/mm^2]	83.33	52.37	30.15	13.41
Utilization[%]	1	1	1	93
Check	OK	OK	OK	OK

Concrete shear resistance - Part 1.1: 6.4.3
LC: Ultimate-load comb01

$$v_{Ed} = \frac{\beta V_{Ed}}{u_{out} d} = \frac{1.00 \cdot 152535}{2852 \cdot 159} = 0.34 \text{ N/mm}^2 \quad (6.38)$$

$$v_{Rd,c} = \max\left(C_{Rd,c} k (100 \rho_l f_{ck})^{1/3} + k_1 \sigma_{cp},\, v_{min} + k_1 \sigma_{cp}\right)$$
$$= \max\left(0.12 \cdot 2.00 (100 \cdot 0.00 \cdot 12.00)^{1/3} + 0.15 \cdot 44.04,\, 0.34 + 0.15 \cdot 44.04\right) = 0.35 \text{ N/mm}^2 \quad (6.47)$$

$v_{Ed} = 0.34$ N/mm^2 $\leq v_{Rd,c} = 0.35$ N/mm^2 - OK

Concrete compression resistance - Part 1.1: 6.4.3
LC: Ultimate-load comb01

$$v_{Ed} = \frac{\beta V_{Ed,0}}{u_0 d} = \frac{1.00 \cdot 152535}{600 \cdot 159.00} = 1.60 \text{ N/mm}^2 \quad (6.38)$$

$$v_{Rd,max} = 0.5 \nu f_{cd} = 0.5 \cdot 0.57 \cdot 8.00 = 2.28 \text{ N/mm}^2$$

$v_{Ed} = 1.60$ N/mm^2 $\leq v_{Rd,max} = 2.28$ N/mm^2 (6.53) - OK

Figure: Utilization formulas and tables

- **Summary graph**
  Summary graph is displayed with legend by default.

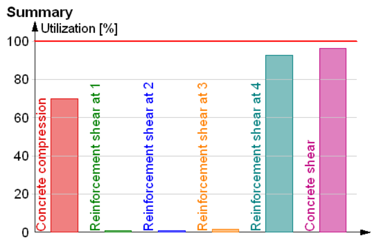


Figure: Utilization summary

Detailed result *Tabmenu* contains the following tools and settings:

- **Selection of zones checked for punching**
  You can choose a unique zone or a design group zone member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, “Maximum” means the significant member having the maximum utilization.

- **Selection of design load**
  Depending on punching design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. “Maximum” means the significant load combination or component of load groups.

Figure: Selection from checked elements and design loads

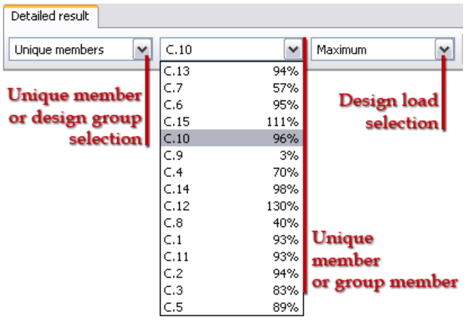


Figure: Selection from checked elements and design loads

-  **Auto design**
  Quick *Auto design* can be done for the currently displayed unique/grouped zone. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts punching design that updates all detailed result figures and tables.

-  **Display options**
  The content and the appearance of the detailed result can be set with *Display options*.

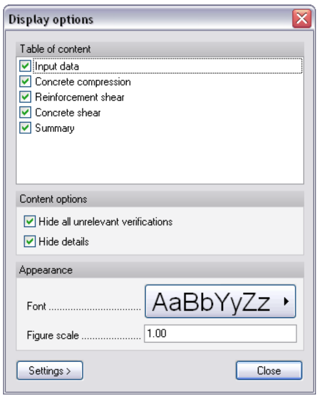


Figure: Display options of Detailed result

-  **Go to**
  Navigate in the *Detailed result* window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

 Click *Tools* > *Add view to document* to place all figures and tables or specified details only into *Documentation*.

STEEL DESIGN

Fast auto design and check are available to find the most suitable steel bar profiles. All section shapes and classes - including class 4 profiles (slender sections) - can be checked for utilization. When modeling steel bars with 3D steel shell components, an auto design finds the proper thickness of the shell elements.

The table summarizes the available steel design features by FEM-Design module.

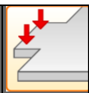
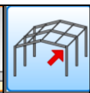
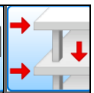
Design element type	Design feature			
Steel Bar	*Auto design*	✔	✔	✔
	Manual design	✔	✔	✔
Shell Model	*Auto design*			✔
	Manual design			✔

Table: Steel design features by FEM-Design module

Steel Bar

Global *Auto (steel bar) design* (*Calculate* > *Design calculations* > *Auto design all structural elements*) finds the most suitable cross-section (from the profile range set at *Auto design* > *Parameters*) for all steel bars (columns, beams and truss members) based on their buckling length, stiffeners, calculation parameters, internal forces and detailed utilization calculations. With *Manual design* you can run quick utilization check for given profiles by bar elements and/or design groups. You can also do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global steel bar design	Element-based steel design	Combined steel design
1	Calculation parameters	Global *Analysis*	Calculation parameters
2	*Design group*	Calculation parameters	*Design group*
3	? *Auto design* > *Parameters*	*Design group*	? *Auto design* > *Parameters*
4	Global *Auto design*	? *Auto design* > *Parameters*	Global *Auto design*
5	*Documentation*	! *Auto design* by elements	? *Auto design* > *Parameters*

6		*Manual design* by elements	*Auto design* by elements
7		*Apply design changes*	*Manual design* by elements
8		Global *Check*	*Apply design changes*
9		*Documentation*	Global *Check*
10			*Documentation*

Table: Recommended steps by design alternatives

Initial Calculation and Design Parameters

All bar design calculations needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups*, *Buckling length* and initial design settings defined by *Design calculation parameters*.

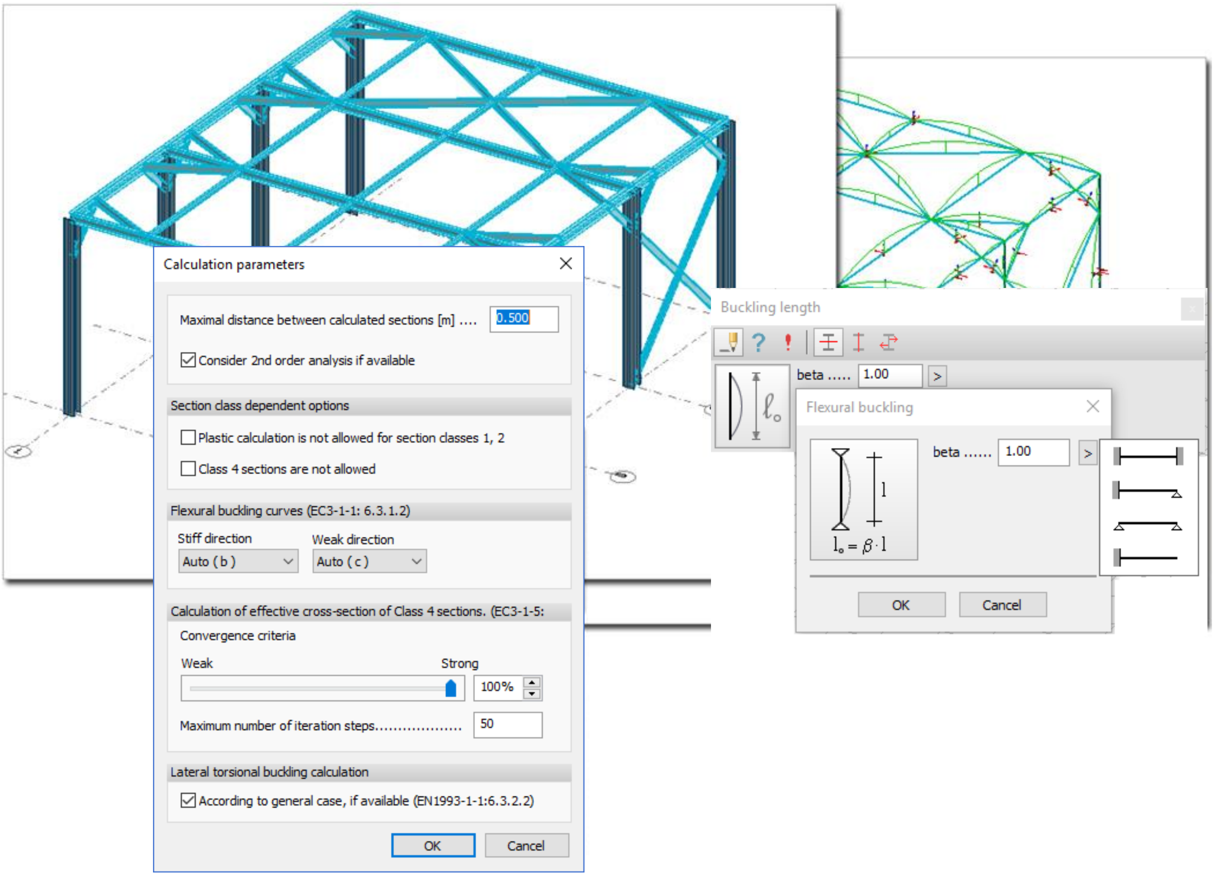


Figure: Design calculation parameters

The *flexural buckling curves* (EC3-1-1: 6.3.1.2) can be specified for each steel bar in Calculation parameters dialog or the user can let the program to calculate it as in the previous versions by selecting "Auto" option.

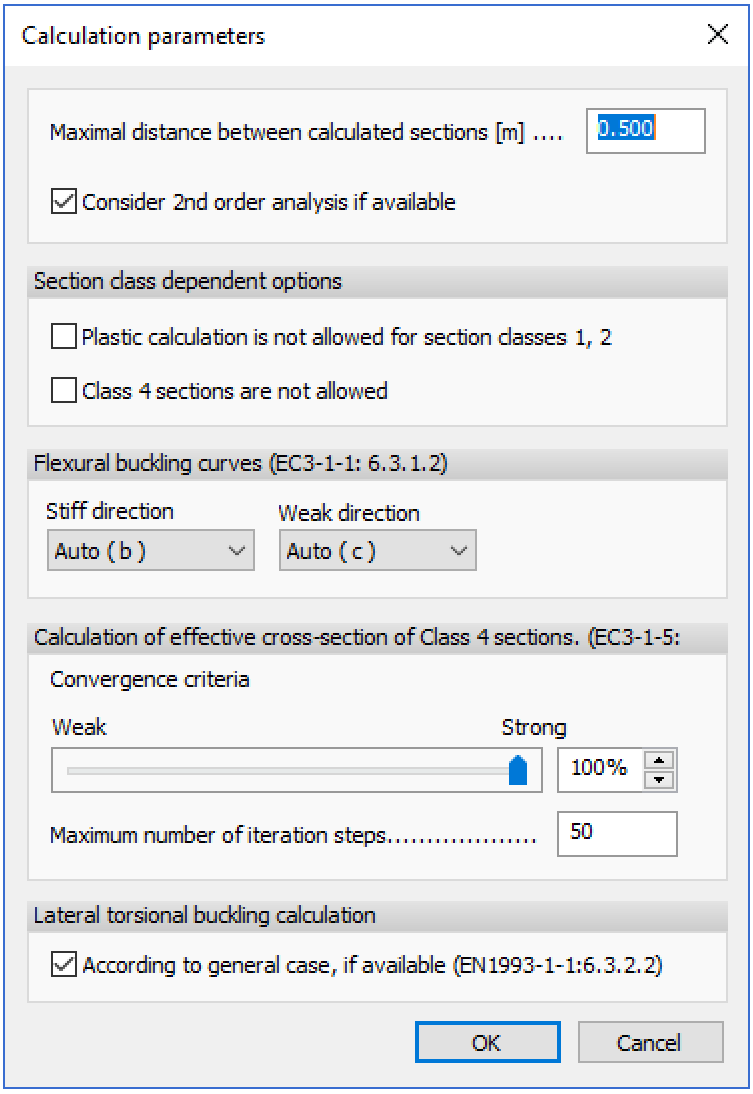


Figure: Setting calculation parameters

For steel bars with varying section the "Auto" option cannot display the automatically calculated curve, since it is determined during the design calculation.

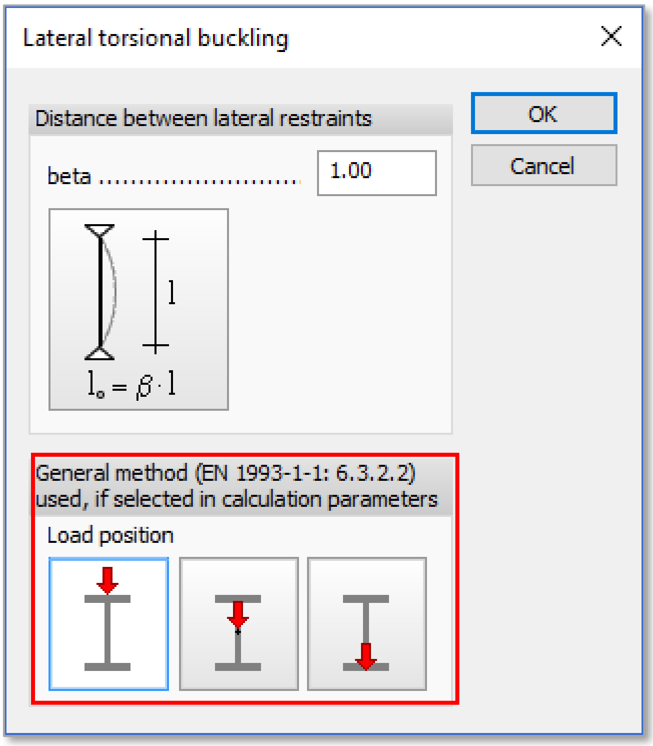
When the section of a steel bar is modified, buckling curve options of the calculation parameter is reset to "Auto"

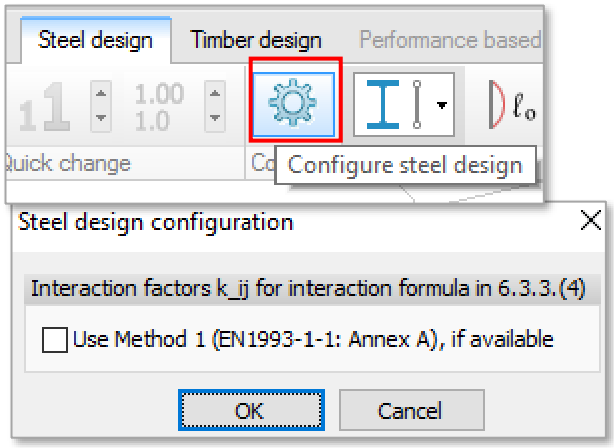
When buckling curve is calculated automatically, applied section is considered, if it exist.

Convergence criteria and the *maximum number of iteration steps* can be set for effective cross-section calculation of Class 4 steel bar section in Calculation parameters dialog.

In some cases the iteration for effective cross-section fails because of the too strong convergence criteria. In this case reducing its factor or increasing the number of the iteration steps may solve the problem.

Lateral torsional buckling can be calculated using the formulas to general case instead of using the simplified method.When the general method is used for lateral torsional buckling calculation, the position of the load needs to be specified as well. Calculation method of the k_{ij} interaction factors can be specified by selecting the design configuration option from the ribbon.





Stiffeners can be added manually to steel bars at any time during design (e.g. before *Auto design*, or between *Auto design* and final *Check*). Stiffeners can be defined element by element, but they can be copied among bar elements. Stiffener definition tools are:

- creates given number of evenly distributed stiffeners on a defined length section,
- creates stiffeners by given distance on a defined length section (in case the multiple of the distance does not fit in the section, the program distributes stiffeners with equal parts at the ends),
- defines stiffeners one by one in given position,
- copies stiffeners of a selected bar to other bars with the same length.

You can increase the number of stiffeners in more steps, if you inactivate (*Do not delete the original*).

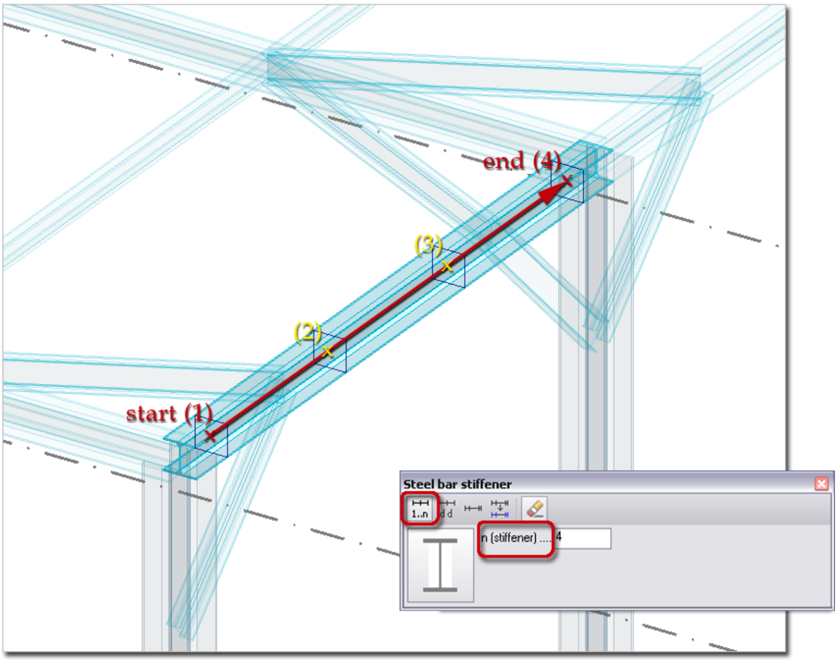


Figure: Stiffeners

Auto Design

Global *Auto design* gives utilization results and suitable profiles for all steel bars of the current project.

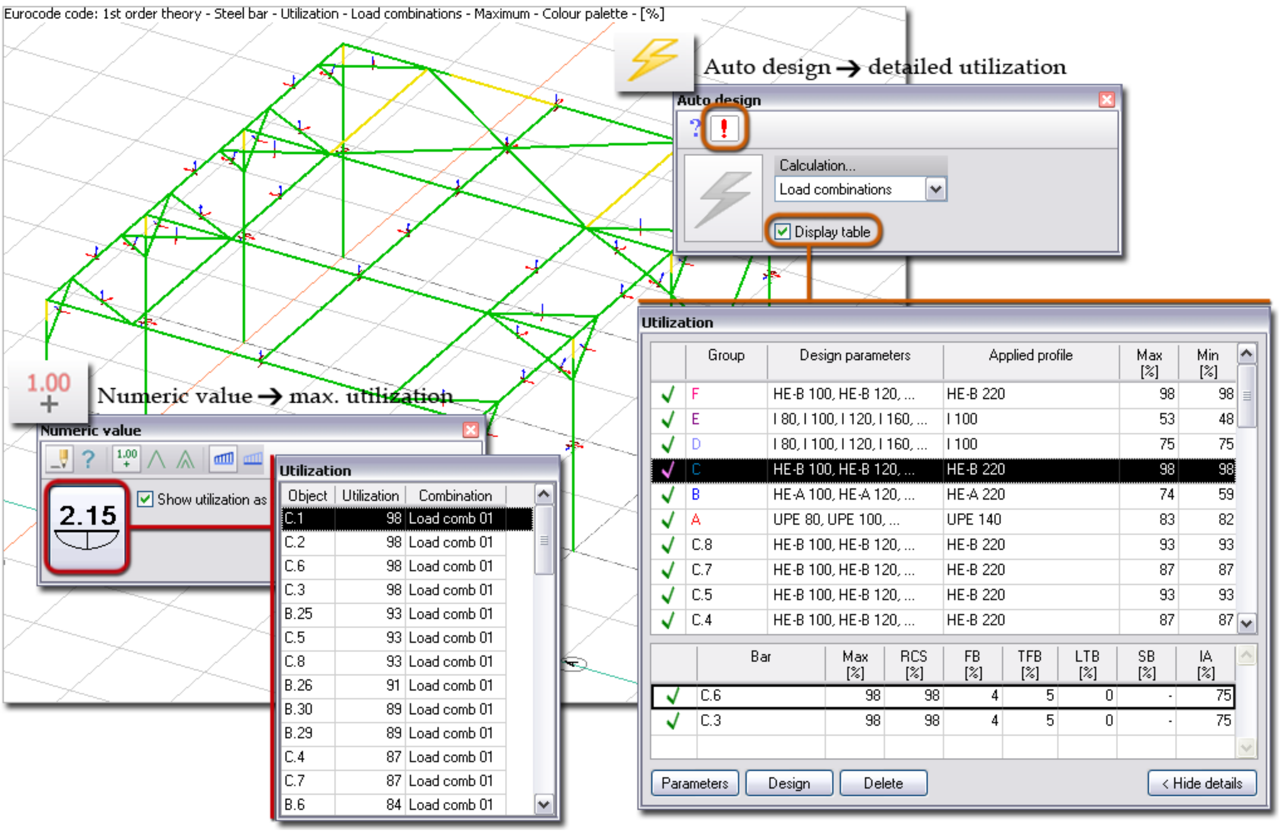


Figure: Global Auto design and utilization result

The recommended profile names can be displayed on screen by showing the "*Steel bar, applied quantity*" object layer, or click *Design* tool of the *Auto design* and the parameters together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Steel bar* > *Utilization*.

Variable cross-section steel bars can't be designed, just checked.

Applied profiles are displayed in blue in the *Utilization* table, if they are assigned to the steel bars during design, otherwise black color represents the original/initial profiles.

Element-based *Auto design* finds the most suitable profile of steel columns, beams and bars for selected unique or grouped members only from a range of available profiles defined by *Parameters*. The design utilization can be limited between 10% and 100% with *Limit utilization*.

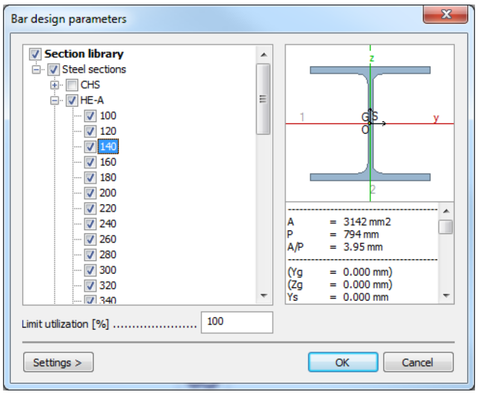


Figure: Range of available profiles for design

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results recommended profiles and their utilization. Check the *Display table* box to have a look at the overall design results (see the figure before).

The upper table shows the design efficiency and the maximal utilization of the designed single members and groups based on the given design parameters. The bottom table displays the utilization details of the bar or the members of the group selected in the upper table.

	Meaning	Note
	Suitable profile is available	
	Suitable profile is not available	Modify the range of available profiles or steel materials
	Suitable profile is available, but the utilization is over the *Limit utilization*	
Group	ID of a single bar or a group name	

Design parameters	The defined range of available profiles	
Applied profile	Profile currently assigned to the bar	
Max	Max. utilization of a single bar or the significant member of a group	
Min	Max. utilization of the less significant group member	
Bar	ID of a single bar or a group member	
RCS	Resistance of cross-section: the maximum utilization from all strength calculations	According to Eurocode 3: 6.2.3 - 6.2.10
FB	Utilization for flexural buckling	According to Eurocode 3: 6.3.1
TFB	Utilization for torsional-flexural buckling	According to Eurocode 3: 6.3.1
LTB	Utilization for lateral torsional buckling	According to Eurocode 3: 6.3.2.4
SB	Utilization for shear buckling	According to Eurocode 3: 1-5: 5
IA	Interaction (between normal force and bending)	According to Eurocode 3: 6.3.3

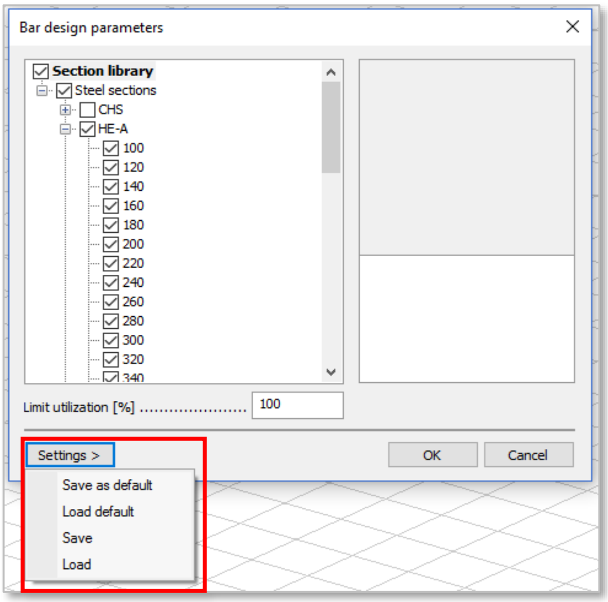
Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

1. Select a bar or a design group in the upper table.
2. Modify the range of available profiles for the select element under *Parameters*.
3. Click *Design*.

Save /load default sections

For each section type (e.g. IPE, HEA, CHS, etc.) a set of sections can be saved/loaded as default.



This will only work with *one type* cross-section (e.g. only HEA, or only KKR) selected. Otherwise, with *Save* command, the user can save a set of arbitrary sections into a file, and use them later for another model by *Load* command.

Manual Design

With *Manual design* quick utilization check can be done for given steel profile and for selected steel beam, column, bar or design group only. Just, choose the load type (load combination or load group) and a profile name from the drop-down lists and select a bar, bars or group, and program displays detailed utilization results in table format.

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*. The program use the chosen profile for all selected bar elements.

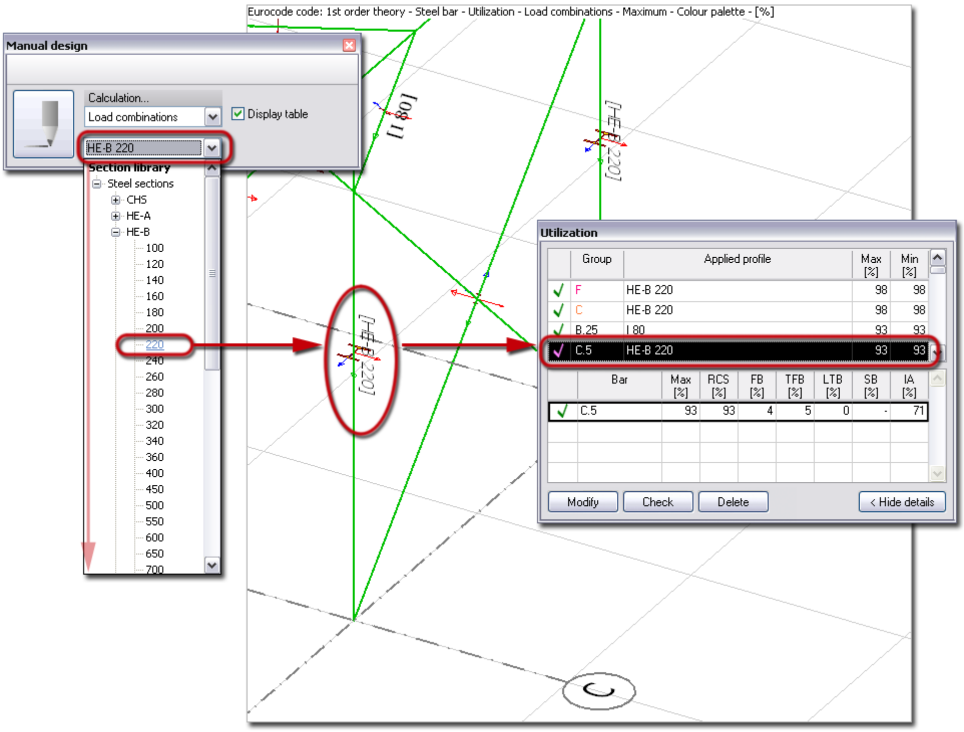


Figure: Quick check by Manual design

Detailed Result

Utilization of steel bars can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all steel bars checked for the recommended profiles.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected elements.
- After global *Check* done for all bar elements having final cross-section.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	*Calculate>Design calculation>Auto design all structural elements*	*Auto design*	*Auto design* and/or *Manual design*	*Auto design* and/or *Manual design*
2	*New result> Steel bar*	*New result>Steel bar*	*Check*	Apply changes
3			*New result>Steel bar*	*Calculate>Design calculation>Auto design all structural elements*
4				*New result> Steel bar*

Table: Steps of displaying steel bar utilization by different design cases

Utilization displayed with *New result* appears for all designed bars. The utilization components for a bar/design group can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a bar/group member, which displays:

- **Input data**
  The figure displays the applied steel cross-section with its main calculation and material parameters.

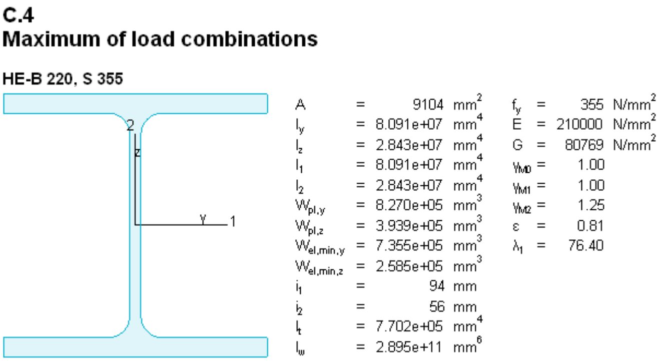


Figure: Applied cross-section

- **Detailed calculation formulas**
  Calculation details and final values are collected by checking types: *Shear resistance* (Eurocode3: 1-1: 6.2.6, 6.2.8), *Torsional resistance* (1-1: 6.2.7), *Shear stress* (1-1: 6.2.6), *Normal stress* (1-1: 6.2.1), *Normal capacity* (1-1: 6.2.1), *Flexural buckling* (1-1: 6.3.1), *Torsional-flexural buckling* (1-1: 6.3.1), *Lateral-torsional buckling* (1-1: 6.3.2.4), *Interaction between normal force and bending* (1-1: 6.3.3.) and *Shear buckling* (1-5: 5). The proper results are displayed in green, while the red result warnings you to repeat design with new bar properties. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.

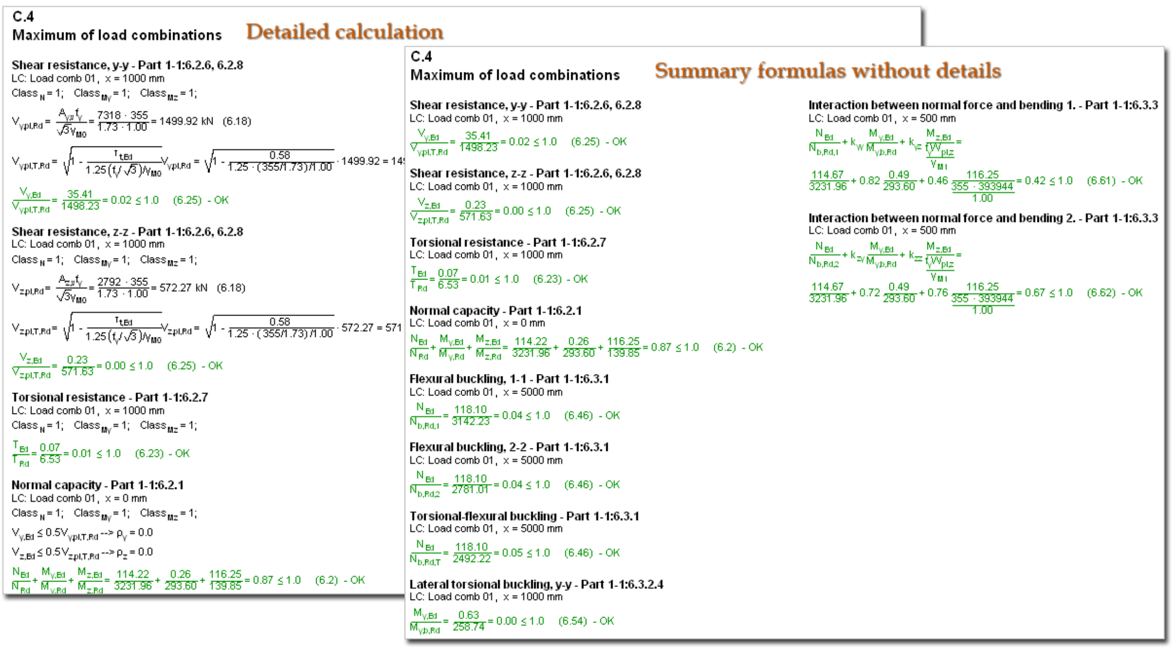


Figure: Utilization checks and formulas

- **Summary graph**
  Summary graph is displayed with legend by default. Numeric values can be inquired in the calculation sections (set by *Design calculation parameters*).

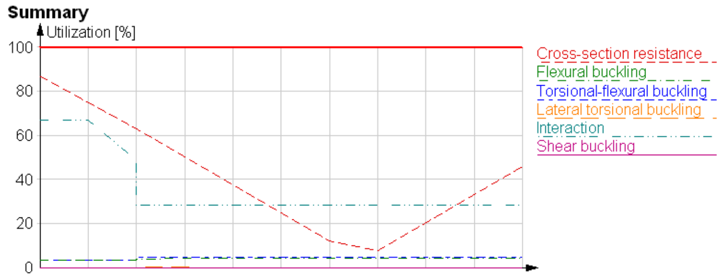


Figure: Utilization summary

Tabmenu contains the following tools and settings:

- **Selection of element to display**
  You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means each check is displayed for the significant member having the maximum utilization for that check.

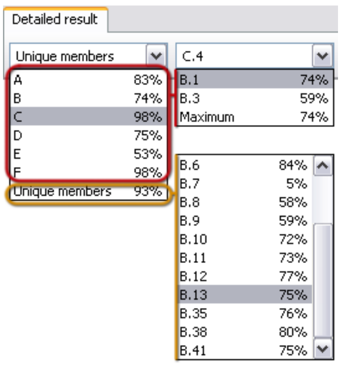


Figure: Selection of a unique or a group member

- **Selection of design load**
  Depending on steel design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means each check is displayed for the significant load combination or component of load groups having the maximum utilization for that check.

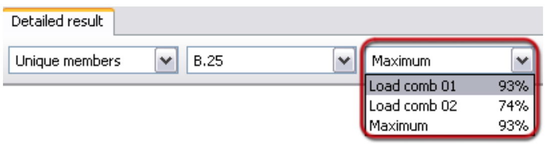


Figure: Selection from design loads

-  **Auto design**
  Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts steel design that updates all detailed result figures and formulas.

- **Manual design**
  Manual design can be launched directly for the currently displayed unique member /group.

- **Display options**
  The content and the appearance of the detailed result can be set with *Display options*. You can show only the final equation without details of the different checks (*Hide details*).

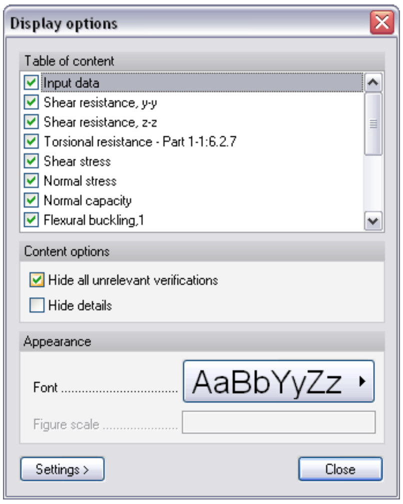


Figure: Display options of Detailed result

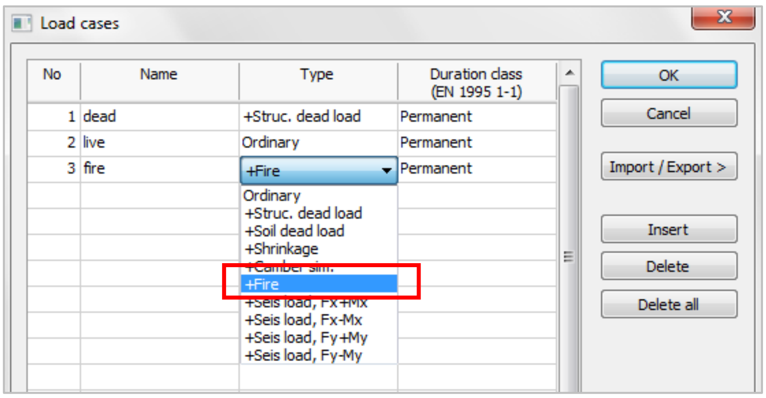
- **Go to**
  Navigate in the detailed result window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

Click *Tools* > *Add view to document* to place all figures, fomulas and summary table or specified details only into *Documentation*.

Fire design for steel bars
Fire design gives the opportunity to check and design steel bars for fire effects according to EN 1993-1-2.

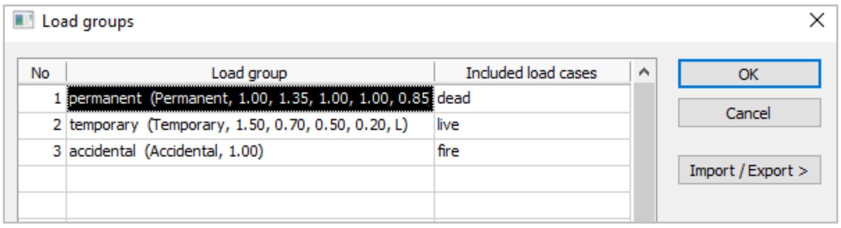
Launch *Steel design/ Steel bar, fire design* . To start *Fire design* it needs some new input data of the bars, and a special load combination and/or load group must be defined.

- A „+Fire" type load case has to be defined in the *Load cases* dialog.

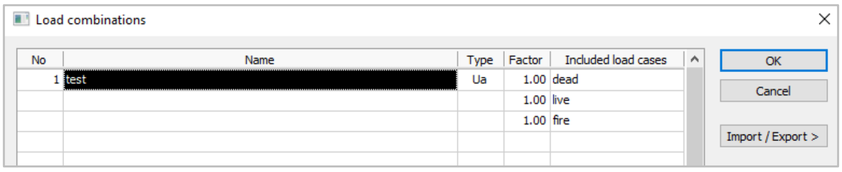


-

- For maximum of load groups calculations, an accidental load group must be defined that contains the „+Fire" type load case.



- For maximum of load combinations calculations, accidental load combinations must be defined that contains the „+Fire" type load case.



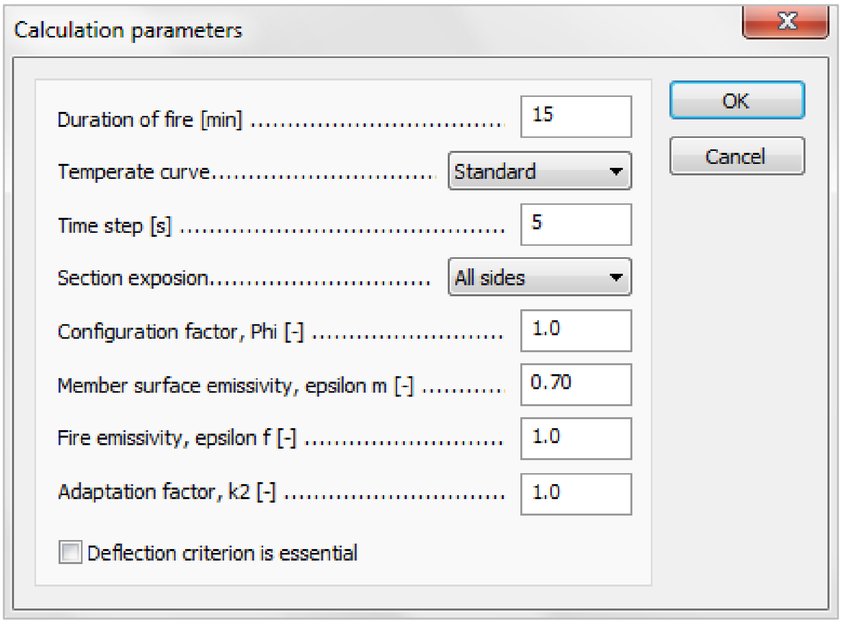
- The effects (internal forces) are calculated from accidental load combinations, where fire is the accidental effect. Resistance of bar is calculated by using reduced yield strength and elasticity modulus for steel at the elevated temperature.

Fire design contains *Calculation parameter, Check, Design group, Auto design* and *Manual design* commands.

Calculation parameter

Explanation of data in calculation parameters are in EN 1993-1-2:3, 4 and EN 1992-1-2:3.

„Deflection criterion is essential" option is available for Danish national annex only.



Check

It works in exactly the same way as in case of steel bar design.

Design group

It works in exactly the same way as in case of normal steel bar design, except that fire design parameter and fire design calculation parameter of two bars must match to be placed into the same design group.

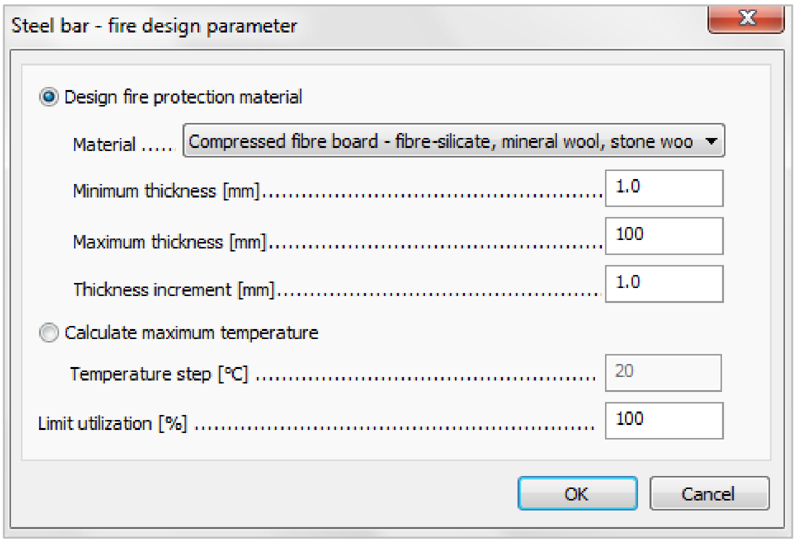
Auto design

There are two design options:

- *Design for fire protection material*
  The design parameter contains the fire protection material, which can be selected from a library (see later). Its *minimal and maximal thickness* and an increment value, which is used by the automatic design procedure to find the minimal necessary thickness of the protection material, can be given by the User

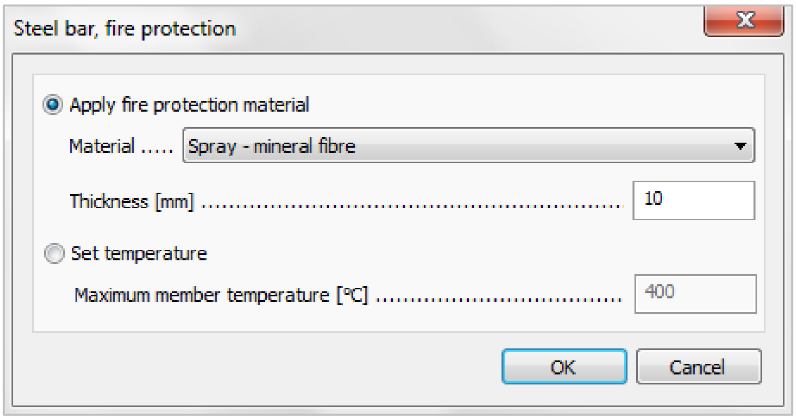
- *Calculate maximum temperature*
  The *Temperature step* for maximum temperature calculation can be defined by the User.

Limit utilization can also be set in the Steel bar - fire design parameter dialog



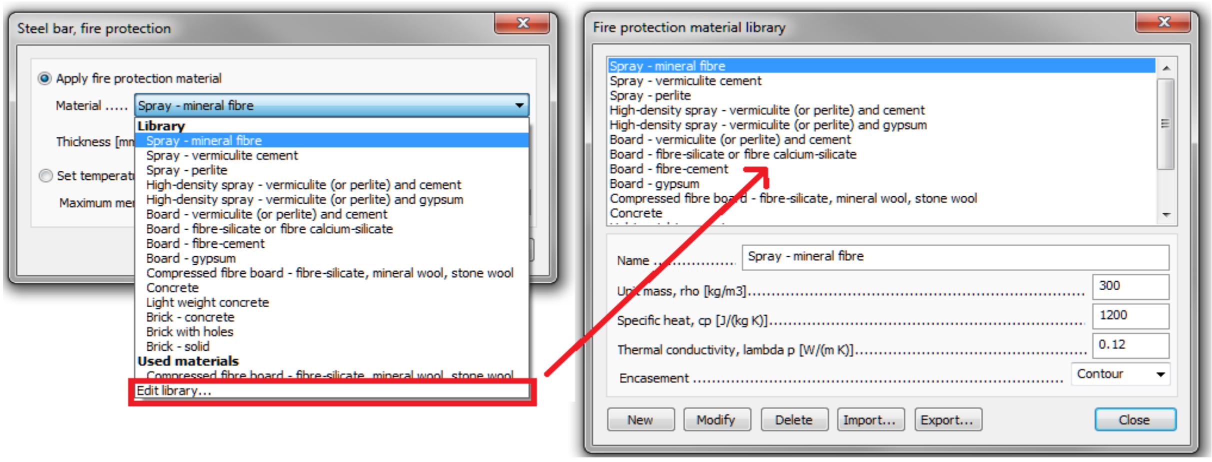
Manual design

Fire protection material (from a library, see later), its thickness, or the *Maximum member temperature* can be selected in Steel bar, fire protection dialog for Manual Design..



Fire protection material library

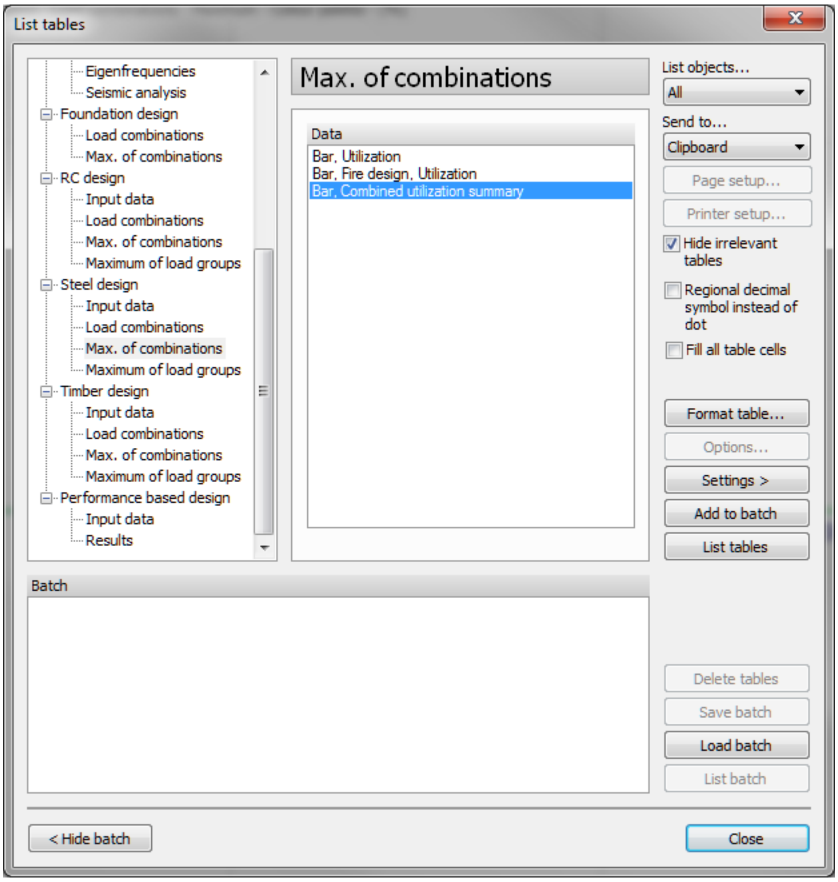
It is available by clicking on „Edit library…" item in the material list of fire protection parameter in Auto and Manual design dialogs.



Results

Utilization results are available to display on the model and to list.

A new result “Max. of combinations/Bar, Combined utilization summary” is available, where the maximum utilization of steel bars for both normal and fire check is displayed



Max. of load combinations, Bar, Combined utilization summary

Member	Calculation	Maximum	Combination
[-]	[-]	[%]	[-]
B.1.1	ULS	121	Standard
	ALS (Fire)	98	Accidental

Shell Model

In FEM-Design, a steel bar can be modeled as a real 3D element defined from steel plates.

To convert a bar element to 3D shells apply the *Steel bar, shell model* tool of the Structure *Tabmenu*.

Global *Auto (Shell model) design* (*Calculate* > *Design calculations* > *Auto design all structural elements*) finds the most suitable thickness (from a thickness range set at *Auto design* > *Parameters*) for all steel plates of the shell model based on internal forces, stability check and detailed utilization calculations. With *Manual design* you can run quick utilization check of given thickness values for selected shell models or their design groups. You can also do quick *Auto design* by shell elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global steel bar design	Element-based steel design	Combined steel design
1	*Structure>Steel bar,shell model*	*Structure>Steel bar,shell model*	*Structure>Steel bar,shell model*
2	*Design group*	Global *Stability analysis*	*Design group*
3	? *Auto design > Parameters*	*Design group*	? Design *Parameters*
4	Global *Auto design*	? *Auto design > Parameters*	Global *Auto design*
5	*Documentation*	! *Auto design* by elements	? *Auto design > Parameters*
6		*Manual design* by elements	! *Auto design* by elements
7		*Apply design changes*	*Manual design* by elements
8		Global *Check*	*Apply design changes*
9		*Documentation*	Global *Check*
10			*Documentation*

Table: Recommended steps by design alternatives

Stability analysis is required to get utilization check of steel bar-shell models. Global *Auto design* automatically runs stability analysis for the entire structure.

Auto Design

Global *Auto design* gives utilization results and suitable thickness for all steel plates of shell models.

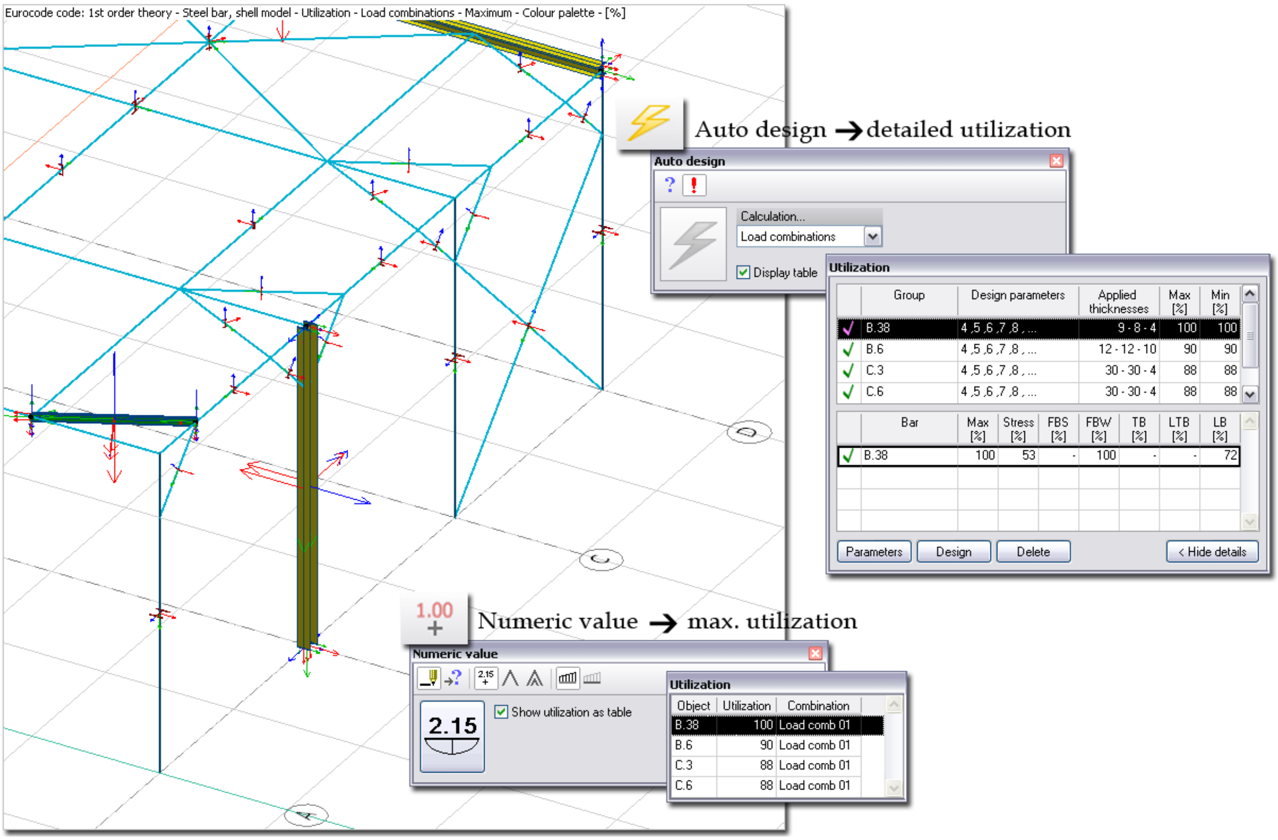


Figure: Global Auto design and utilization result

The recommended thickness values can be displayed on screen by showing the "*Steel bar, shell model, applied quantity*" object layer, or click *Design* tool of the *Auto design* and the thickness values together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Steel bar, shell model* > *Utilization*.

Applied thickness values are displayed in blue in the *Utilization* table, if they are assigned to the steel shell parts during design, otherwise black color represents the original/initial thickness.

Element-based *Auto design* finds the most suitable thickness of steel plates for selected unique or grouped shell models only from a range of available thicknesses defined by *Parameters*.

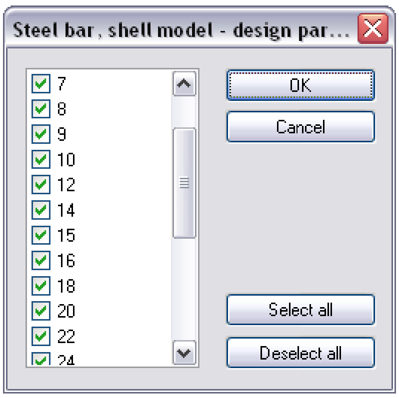


Figure: Range of available thickness values for design

To run element-based design for the load combinations, select the required shell element and/or element group with the *Auto design* command and click *Design* tool. The quick process results recommended thickness values and the utilization of the shell model elements. Check the *Display table* box to have a look at the overall design results (see before).

The upper table shows the design efficiency and the maximal utilization of the designed single element and groups based on the found thickness values. The bottom table displays the utilization details of the shell model element or the members of a group selected in the upper table.

	Meaning
✓	Suitable thickness is available
⛔	Suitable thickness is not available (Modify the range of available thicknesses or steel materials.)
Group	ID of a single element or a group name
Design parameters	The defined range of available thicknesses
Applied thicknesses	Thickness currently assigned to the shells
Max	Max. utilization of a model element or the significant member of a group
Min	Max. utilization of the less significant group member
Bar	ID of the designed single bar or a group member
Stress	Stress utilization
FBS	Utilization for flexural buckling around the stiff axis
FBW	Utilization for flexural buckling around the weak axis
TB	Utilization for torsional buckling
LTB	Utilization for lateral torsional buckling
LB	Utilization for local buckling

Table: The meaning of symbols and utilization results

Quick redesign can be done inside the *Utilization* table:

1 Select a bar or a design group in the upper table.

2 Modify the range of available plate thicknesses for the select element under *Parameters*.

3 Click *Design*.

Manual Design

With *Manual design* quick utilization check can be done for custom (also different) thickness values of selected steel plates (or their groups). Just select steel plate(s) and set a thickness value in the appeared dialog. Run *Check* calculation to run and display utilization check for the modified plates according to their new thickness values.

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*.

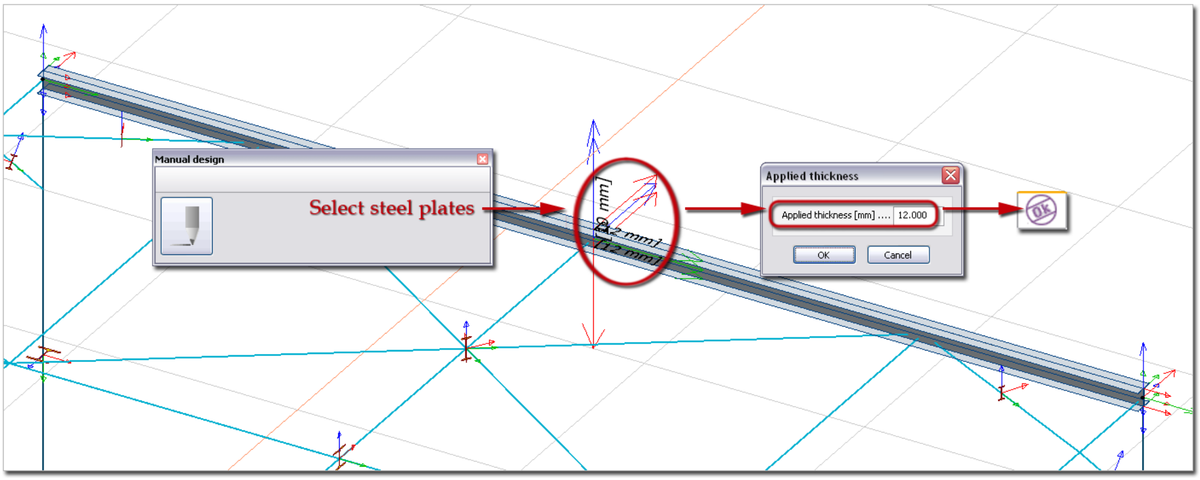


Figure: Quick check by Manual design

Detailed Result

Utilization of steel bar-shell models can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all steel shell models.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected elements.
- After global *Check* done for all bar-shell elements having final plate thicknesses.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	*Calculate>Design calculation>Auto design all structural elements*	*Auto design*	*Auto design* and/or *Manual design*	*Auto design* and/or *Manual design*
2	*New result>* *Steel bar, shell model*	*New result>* *Steel bar, shell model*	*Check*	Apply changes
3			*New result>* *Steel bar, shell model*	*Calculate>Design calculation>Auto design all structural elements*
4				*New result>* *Steel bar, shell model*

Table: Steps of displaying steel bar-shell model utilization by different design cases

Utilization displayed with *New result* appears for all designed bar-shell models. The utilization components for an element/design group can be displayed with *Detailed result*.

Detailed result opens two new windows in the current project after selecting a shell model or group member, which display:

- **Applied shell thicknesses** (*Detailed result* window)
  A list displays the applied thicknesses by the steel plate components.

- **Detailed calculation formulas** (*Detailed result* window)
  Calculation details and final values are collected by checking types: *Stresses*, *Flexural buckling, stiff/weak direction*, *Torsional buckling*, *Lateral torsional buckling* and *Local buckling*. The proper results are displayed in green, while the red result warnings you to repeat design with new thickness values. The content of the utilization checks depends on *Display options* and *Buckling mode* (see later). Not relevant checks can also be hidden.

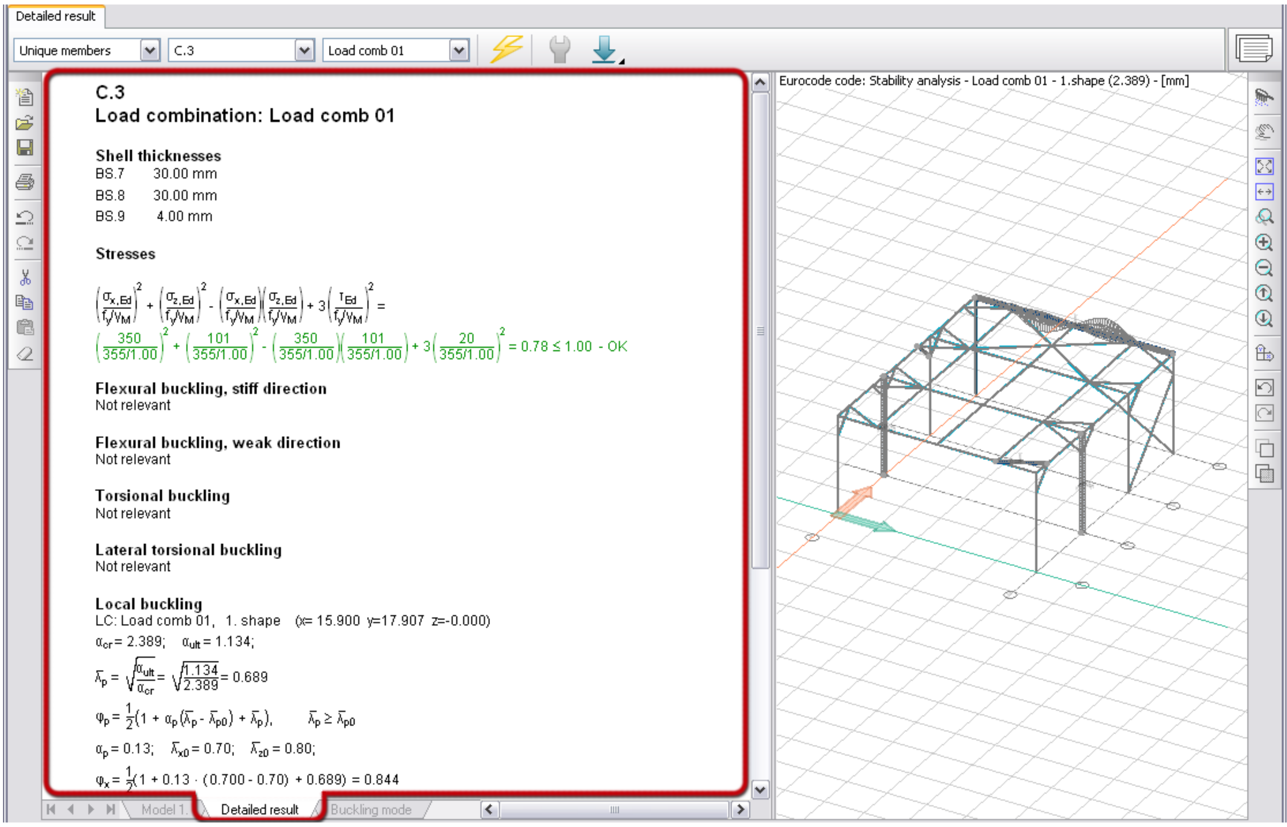


Figure: Utilization checks and formulas

- **Summary graph** (*Detailed result* window)
  Summary graph is displayed with legend by default.

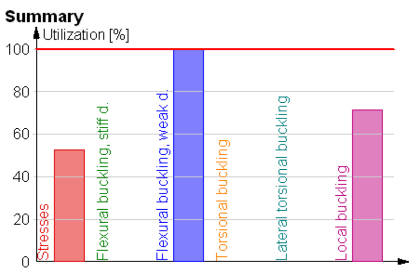


Figure: Utilization summary

- **Buckling mode** (*Buckling mode* window)
  Based on stability analysis, the calculated shapes of the bar-shell model can be displayed in 3D view. Just select the required shape from the *Shape* drop-down list of the navigator panel and the program automatically shows the deformed shape in the current *display mode*. *Buckling mode* can be also chosen for the selected shape that affects on the design check content of the *Detailed result* window. You can add numeric values to the deformed shape or run buckling animation.

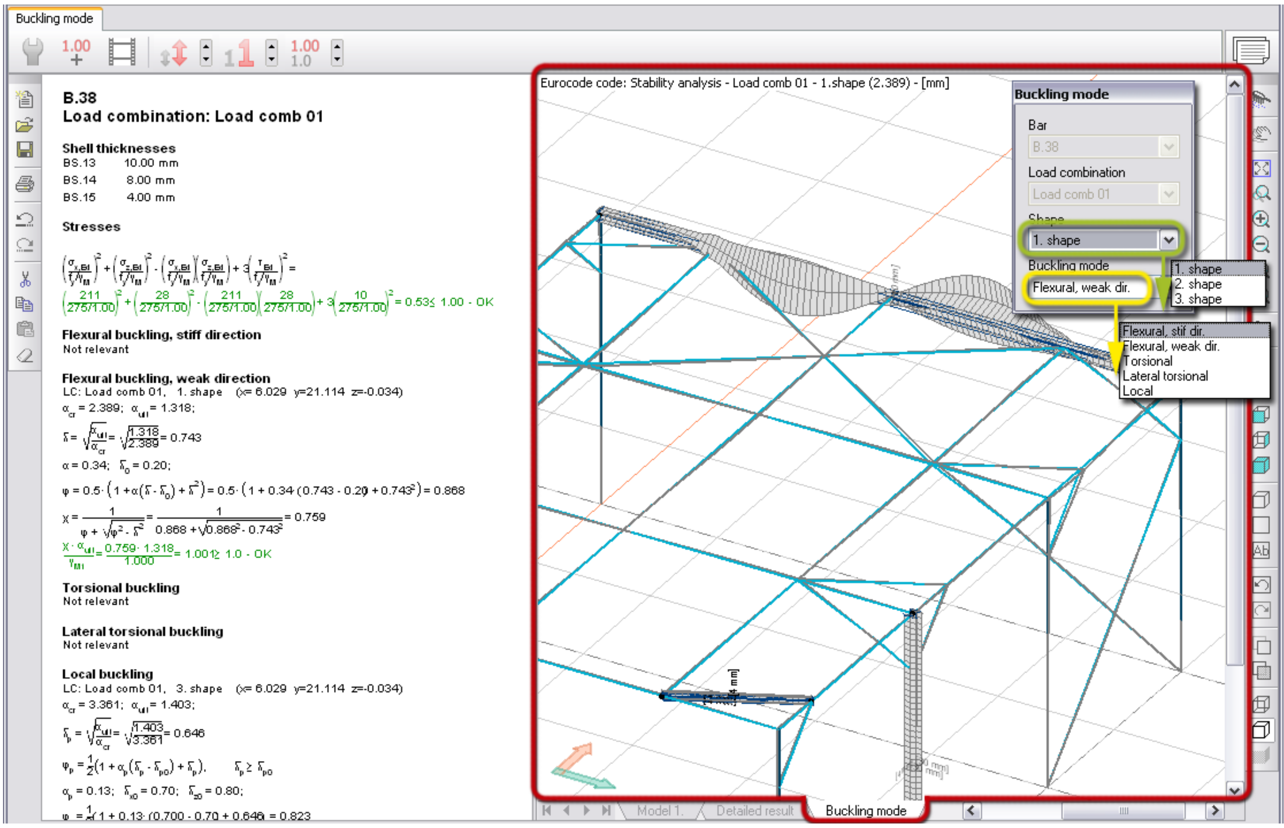


Figure: Buckling shape

Tabmenu contains the following tools and settings for the *Detailed result* window:

- **Selection of element to display**
  You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

- **Selection of design load**
  A load combination can be selected for detailed results. Each row displays the name of the available load combination and its utilization effect. "Maximum" means the significant load combination.

-  **Auto design**
  Quick *Auto design* can be done for the currently displayed unique/group member. Set the thickness of the steel plate components, and then click *OK* to start design that updates all detailed result figures and tables.

- **Display options**
  The content and the appearance of the detailed result can be set with *Display options*.

- **Go to**
  Navigate in the *Detailed result* window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

Click *Tools* > *Add view to document* to place all calculation and check formulas into *Documentation*.

Timber Design

Fast auto design and check are available to find the most suitable timber bar cross-sections and panel types.

The table summarizes the available timber design features by FEM-Design module.

Design element type	Design feature			
Timber Bar	*Auto design*	✓	✓	✓
	Manual design	✓	✓	✓
Timber Panel	*Auto design*			✓
	Manual design			✓

Table: Timber design features by FEM-Design module

Load-Duration Classes

Based on Eurocode 5 (EN 1995-1-1:2004), load-duration classes can be taken into account in the *Timber design*.

The load-duration classes are characterized by the effect of constant load acting for a certain period of time in the life of the structure. For a variable action the appropriate class shall be determined on the basis of an estimate of the typical variation of the load with time.

Actions shall be assigned to one of the load-duration classes given for strength and stiffness calculations.

Load-duration class	Order of accumulated duration of characteristic load	Examples of loading
Permanent	More than 10 years	Dead load
Long-term	6 months – 10 years	Storage
Medium-term	1 week – 6 months	Imposed floor load, snow
Short-term	less than one week	Snow, wind
Instantaneous		Wind, accidental load

Table: Load-duration classes and examples of load-duration assignment (EN 1995-1-1:2004)

⚠ Since climatic loads (snow, wind) vary between countries, the assignment of load-duration classes may be specified in the National annex.

Loads can be assigned to load-duration classes with the *Load cases* command in the Loads *Tabmenu*.

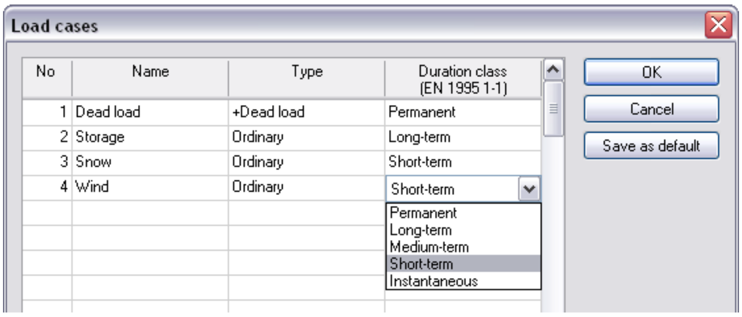


Figure: Load cases and load-duration classes

Timber Bar

Global *Auto (timber bar) design* (*Calculate* > *Design calculations* > *Auto design all structural elements*) finds the most suitable cross-section (from the profile range set at *Auto design* > *Parameters*) for all timber bars (columns, beams and truss members) based on their initial calculation parameters, internal forces and detailed utilization calculations. With *Manual design* you can run quick utilization check for given profiles by bar elements and/or design groups. You can also do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

No.	Global timber bar design	Element-based timber design	Combined timber design
1	Calculation parameters	Global *Analysis*	Calculation parameters
2	*Design group*	Calculation parameters	*Design group*
3	*Auto design* > *Parameters*	*Design group*	Design *Parameters*
4	Global *Auto design*	*Auto design* > *Parameters*	Global *Auto design*
5	*Documentation*	*Auto design* by elements	*Auto design* > *Parameters*
6		*Manual design* by elements	*Auto design* by elements
7		*Apply design changes*	*Manual design* by elements
8		Global *Check*	*Apply design changes*
9		*Documentation*	Global *Check*
10			*Documentation*

Table: Recommended steps by design alternatives

Initial Calculation and Design Parameters

All bar design calculations needs internal forces from *Analysis* calculations applied for *Load combinations* or *Load-groups*, *Buckling length* and initial design settings defined by *Design calculation parameters*.

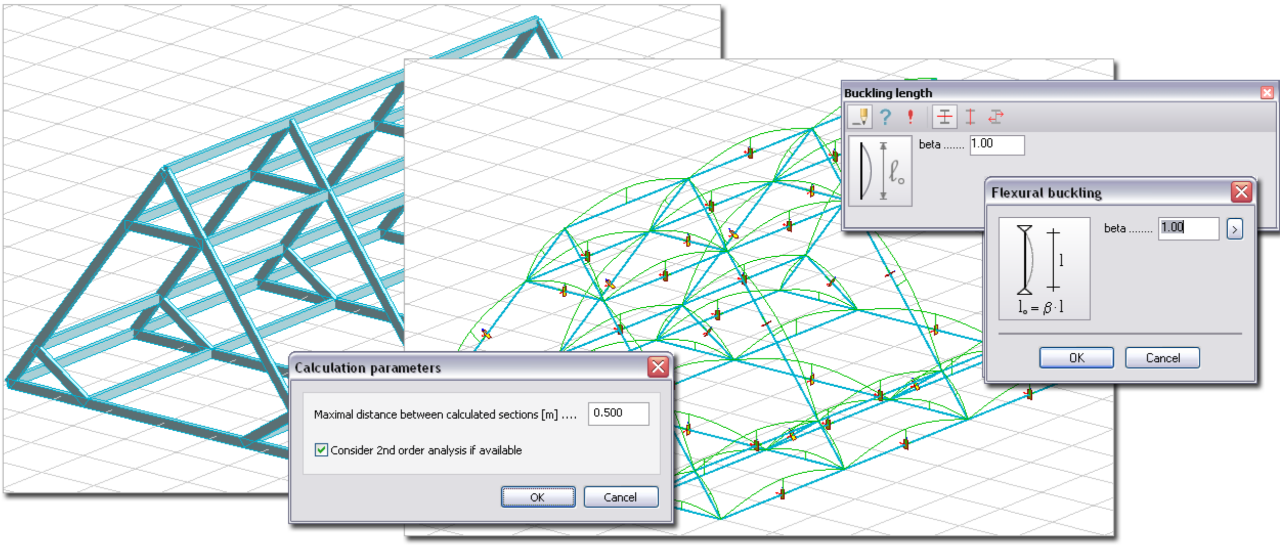


Figure: Design calculation parameters

Auto Design

Global *Auto design* gives utilization results and suitable profiles for all timber bars of the current project from a defined range of cross-sections.

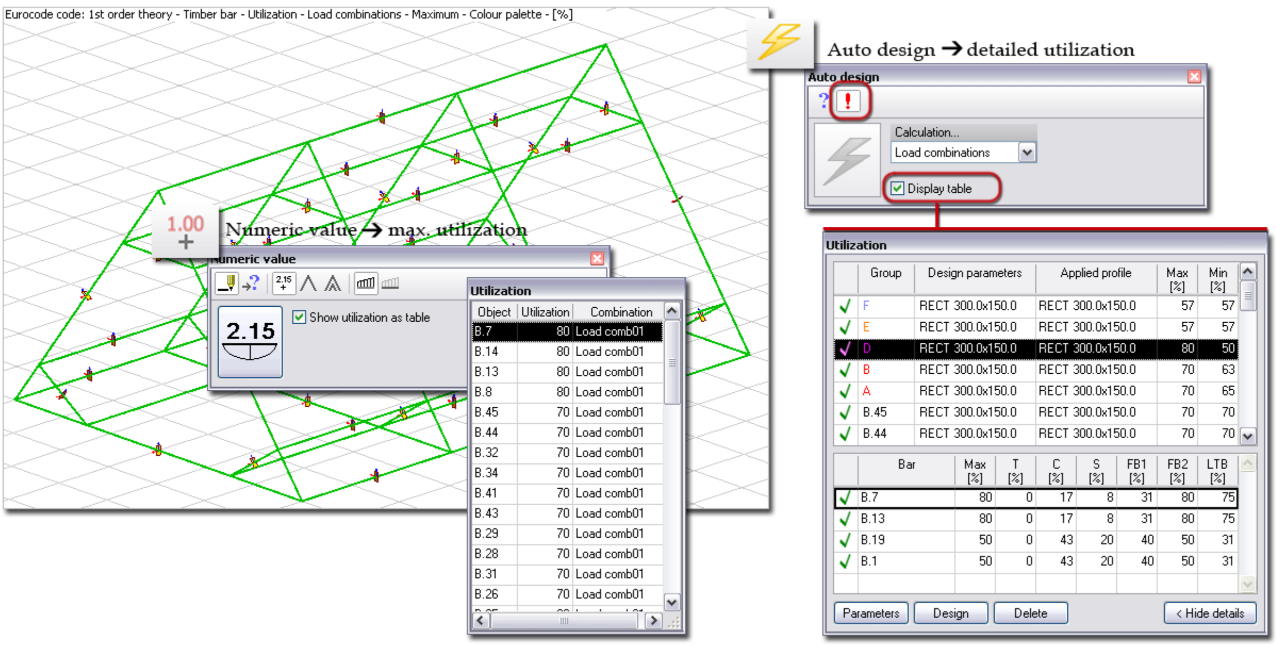


Figure: Global Auto design and utilization result

The recommended profile names can be displayed on screen by showing the "*Timber bar, applied quantity*" object layer, or click *Design* tool of the *Auto design* and the parameters together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Timber bar* > *Utilization*.

Applied profiles are displayed in blue in the *Utilization* table, if they are assigned to the timber bars during design, otherwise black color represents the original/initial profiles.

Element-based *Auto design* finds the most suitable profile of timber columns, beams and bars for selected unique or grouped members only from a range of available profiles defined by *Parameters*. The design utilization can be limited between 10% and 100% with *Limit utilization*.

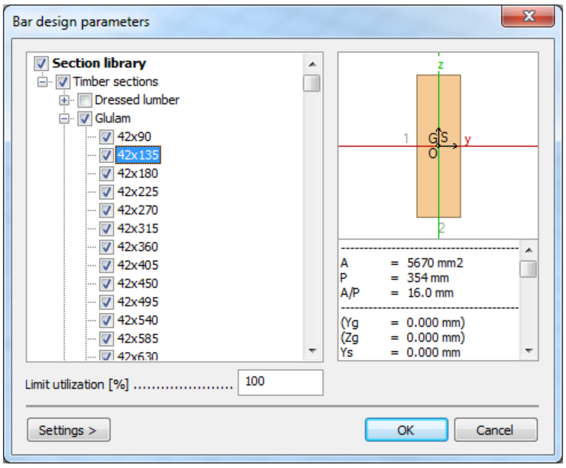


Figure: Range of available profiles for design

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results recommended profiles and their utilization. Check the *Display table* box to have a look at the overall design results (see the figure before).

The upper table shows the design efficiency and the maximal utilization of the designed single members and groups based on the given design parameters. The bottom table displays the utilization details of the bar or the members of the group selected in the upper table.

	Meaning	Note
	Suitable profile is available	
	Suitable profile is not available	Modify the range of available profiles or timber materials
Group	ID of a single bar or a group name	
Design parameters	The defined range of available profiles	
Applied profile	Profile currently assigned to the bar	
Max	Max. utilization of a single bar or the significant member of a group	
Min	Max. utilization of the less significant group member	
Bar	ID of a single bar or a group member	
T	Utilization for combined bending and axial tension	According to Eurocode 5: 6.2.3

C	Utilization for combined bending and axial compression	According to Eurocode 5: 6.1.4, 6.2.4
S	Utilization for combined shear and torsion	According to Eurocode 5: 6.1.7, 6.1.8
FB1	Utilization for flexural buckling around axis 1	According to Eurocode 5: 6.3.2
FB2	Utilization for flexural buckling around axis 2	According to Eurocode 5: 6.3.2
LTB	Utilization for lateral torsional buckling	According to Eurocode 5: 6.3.3

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

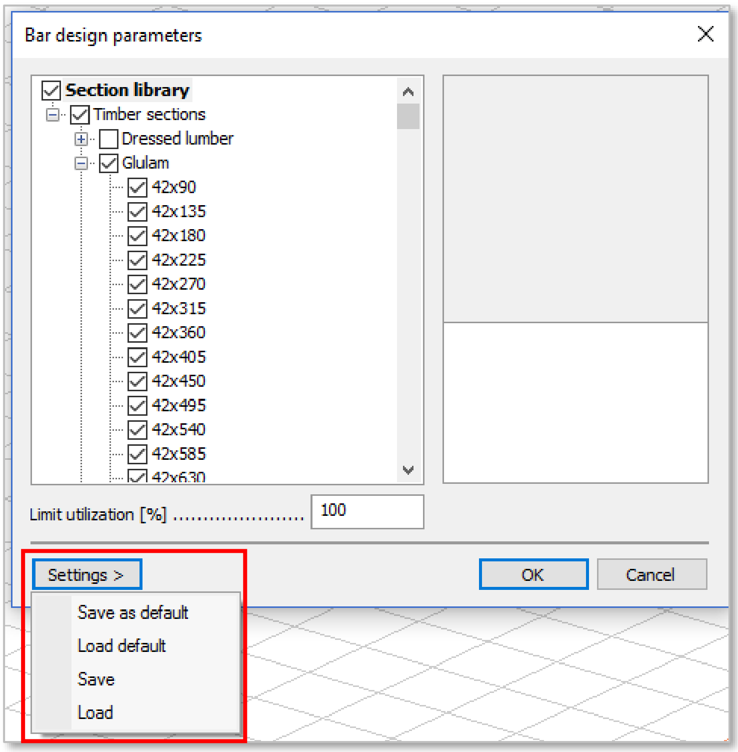
1 Select a bar or a design group in the upper table.

2 Modify the range of available profiles for the select element under *Parameters*.

3 Click *Design*.

Save /load default sections

For each section type (e.g. Dressed lumber, Glulam, etc.) a set of sections can be saved/loaded as default.



This will only work with *one type* cross-section (e.g. only Glulam, or only Sawn lumber) selected. Otherwise, with *Save* command, the user can save a set of arbitrary sections into a file, and use them later for another model by *Load* command.

Manual Design

With *Manual design* quick utilization check can be done for given timber profile and for selected timber beam, column, bar or design group only. Just, choose the load type (load combination or load group) and a profile name from the drop-down lists and select a bar, bars or group, and program displays detailed utilization results in table format.

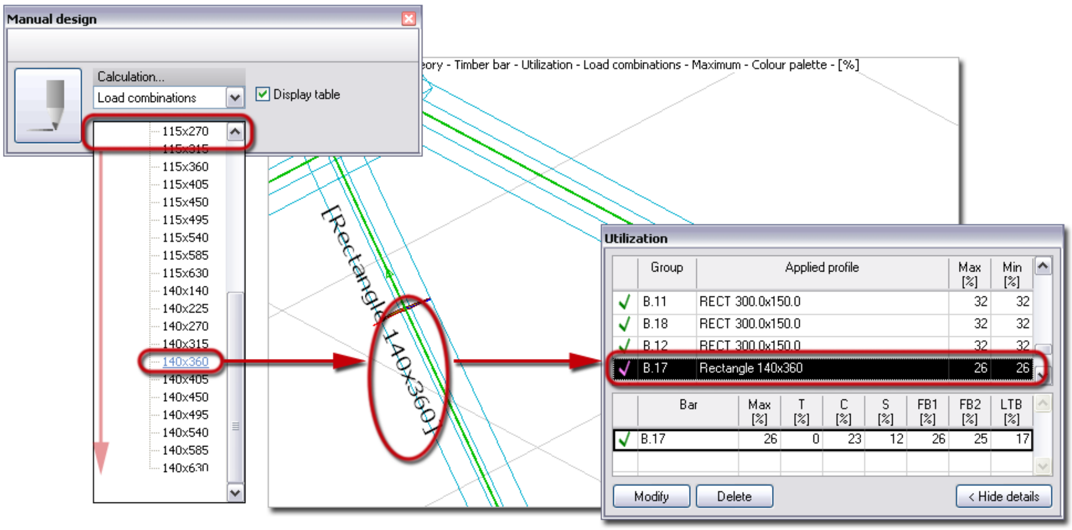


Figure: Quick check by Manual design

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*. The program use the chosen profile for all selected bar elements.

Detailed Result

Utilization of timber bars can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all timber bars checked for the recommended profiles.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected elements.
- After global *Check* done for all bar elements having final cross-section.

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	*Calculate>Design calculation>Auto design all structural elements*	*Auto design*	*Auto design* and/or *Manual design*	*Auto design* and/or *Manual design*
2	*New result> Timber bar*	*New result>Timber bar*	*Check*	Apply changes
3			*New result> Timber bar*	*Calculate>Design calculation>Auto design all structural elements*
4				*New result> Timber bar*

Table: Steps of displaying timber bar utilization by different design cases

Utilization displayed with *New result* appears for all designed bars. The utilization components for a bar/design group can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a bar/group member, which displays:

- **Applied cross-section**
  The figure displays the applied timber cross-section with its main calculation parameters.

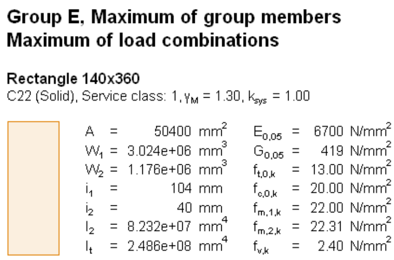


Figure: Applied cross-section

- **Detailed calculation formulas**
  Calculation details and final values are collected by checking types: *Combined bending and axial tension* (Eurocode5: 6.2.3), *Combined bending and axial compression* (6.1.4, 6.2.4), *Combined shear and torsion* (6.1.7, 6.1.8), *Flexural buckling around axis 1* and *2* (6.3.2) and *Lateral torsional buckling* (6.3.3). The proper results are displayed in green, while the red result warnings you to repeat

design with new bar properties. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.

- **Summary graph**
  Summary graph is displayed with legend by default. Numeric values can be inquired in the calculation sections (set by *Design calculation parameters*).

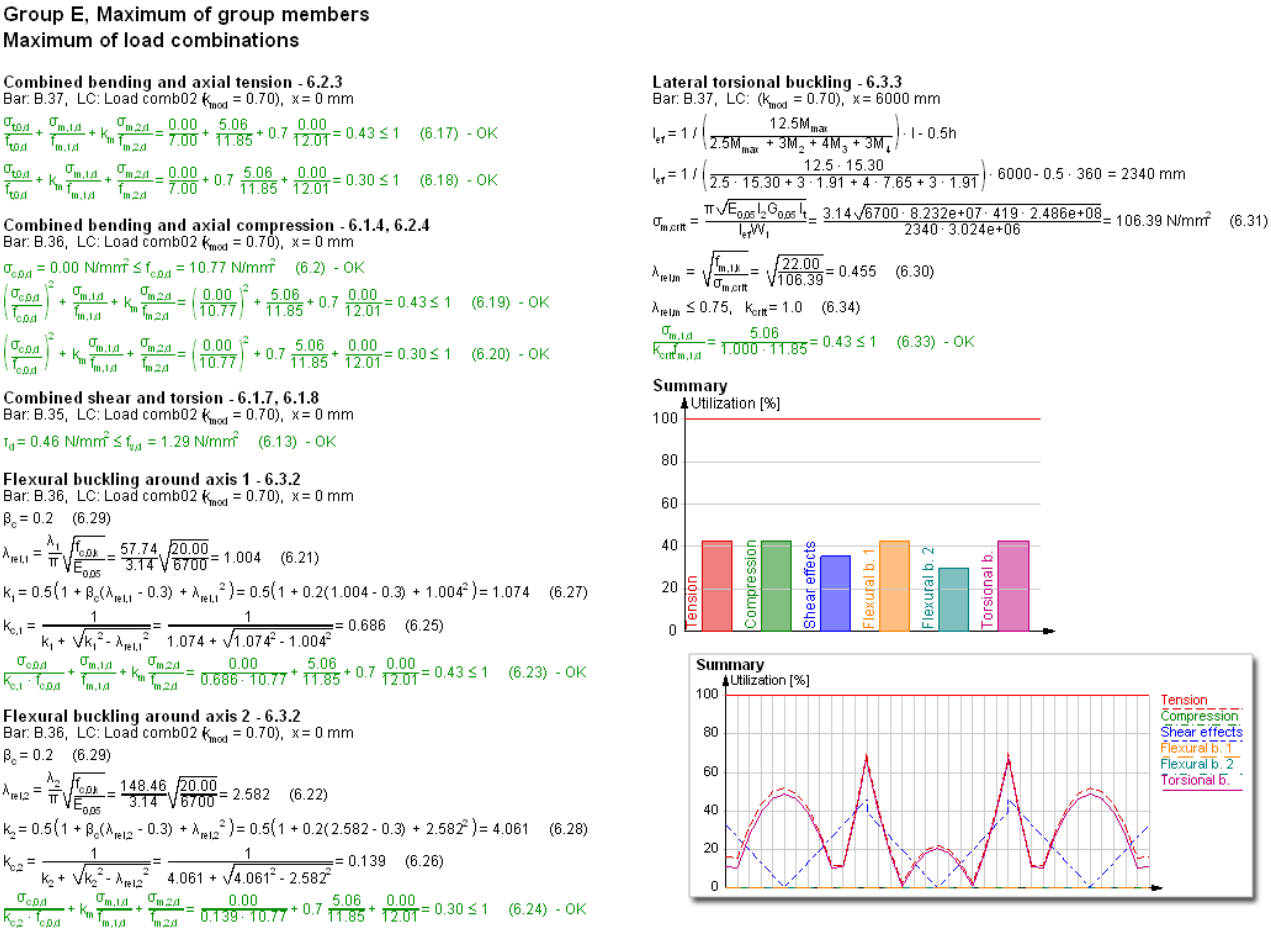


Figure: Utilization checks, formulas and summary

Detailed result *Tabmenu* contains the following tools and settings:

- **Selection of element to display**
  You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, "Maximum" means the significant member having the maximum utilization.

- **Selection of design load**
  Depending on timber design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. "Maximum" means the significant load combination or component of load groups.

- **Auto design**
  Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts timber design that updates all detailed result figures and formulas.

- **Manual design**
  Manual design can be launched directly for the currently displayed unique/group member.

- **Display options**
  The content and the appearance of the detailed result can be set with *Display options*. You can show only the final equation without details of the different checks (*Hide details*).

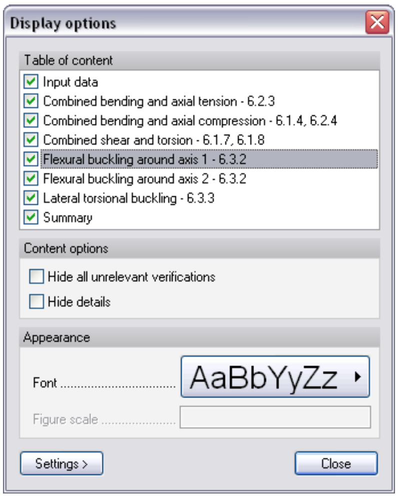


Figure: Display options of Detailed result

- **Go to**
  Navigate in the detailed result window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

Click *Tools* > *Add view to document* to place all figures, fomulas and summary table or specified details only into *Documentation*.

Timber Panel

Global *Auto (timber panel) design* (*Calculate* > *Design calculations* > *Auto design all structural elements*) finds the most suitable type (from panel type range set at *Auto design* > *Parameters*) for all timber panels (plates and walls) based on internal forces and detailed utilization calculations. With *Manual design* you can run quick utilization check for given panel types by elements and/or design groups. You can also do quick *Auto design* by elements and design groups only instead of global design. Of course, any number of design cycles is executable, so the global *Auto design* can be combined with both previous and additional element-based *Auto designs*.

⚠ To define a timber panel (slab or wall) read more at *Timber Panel Element*.

No.	Global timber panel design	Element-based timber design	Combined timber design
1	*Design group*	Global *Analysis*	*Design group*
2	? *Auto design* > *Parameters*	*Design group*	? Design *Parameters*
3	Global *Auto design*	? *Auto design* > *Parameters*	Global *Auto design*
4	*Documentation*	! *Auto design* by elements	? *Auto design* > *Parameters*
5		*Manual design* by elements	! *Auto design* by elements
6		*Apply design changes*	*Manual design* by elements
7		Global *Check*	*Apply design changes*
8		*Documentation*	Global *Check*
9			*Documentation*

Table: Recommended steps by design alternatives

Auto Design

Global *Auto design* gives utilization results and suitable panel type for all timber panels of the current project from a defined range of panel types.

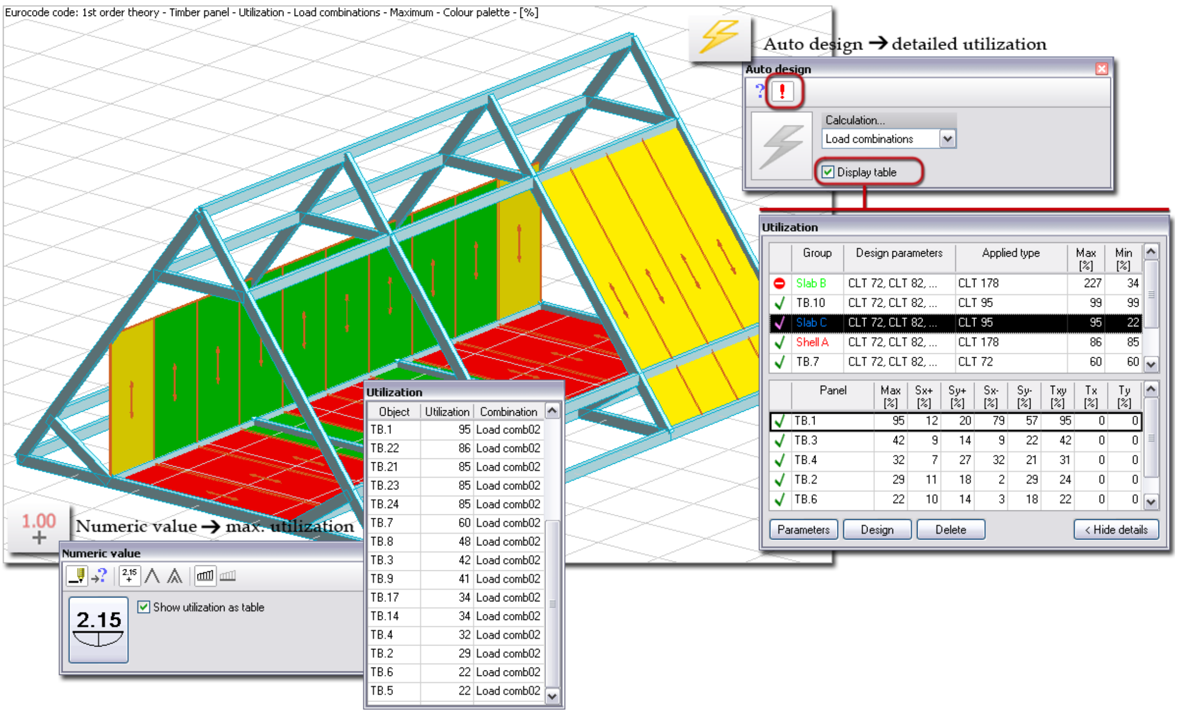


Figure: Global Auto design and utilization result

The recommended panel type names can be displayed on screen by showing the "*Timber panel, applied quantity*" object layer, or click ! *Design* tool of the *Auto design* and the parameters together with utilization results are available in table format. Utilization as colored figure (color palette) can be displayed by selecting *New result* > *Timber panel* > *Utilization*.

Applied panel types are displayed in blue in the *Utilization* table, if they are assigned to the timber panels during design, otherwise black color represents the original/initial types.

Element-based *Auto design* finds the most suitable type of timber panels for selected unique or grouped members only from a range of available types defined by *Parameters*.

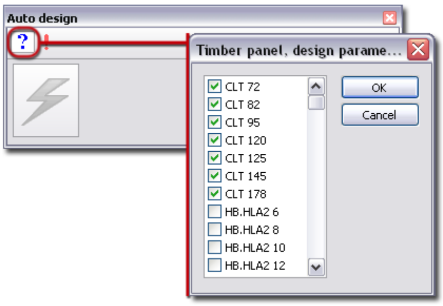


Figure: Range of available timber panels for design

To run element-based design for the load combinations or the maximum of load groups, select the required members and/or group with the *Auto design* command and click *Design* tool. The quick process results recommended panel types and their utilization. Check the *Display table* box to have a look at the overall design results (see before).

The upper table shows the design efficiency and the maximal utilization of the designed single panel element and groups based on internal forces. The bottom table displays the utilization details of the panel or the panel members of the group selected in the upper table.

	Meaning	Note
	Suitable panel is available	
	Suitable panel is not available	Modify the range of available panels or timber materials
Group	ID of a single panel or a group name	
Design parameters	The defined range of available panels	
Applied type	Panel type currently assigned to the panel	
Max	Max. utilization of a single panel or the significant member of a group	
Min	Max. utilization of the less significant group member	
Panel	ID of a single panel or a group member	

Sx+ / Sy+	Utilization for x/y-directional tension	According to Eurocode 5: 6.2.3
Sx- / Sy-	Utilization for x/y-directional compression	According to Eurocode 5: 6.1.4, 6.2.4
Txy	Utilization for shear, xy	According to Eurocode 5: 6.1.7
Tx / Ty	Utilization for shear, xz /yz	According to Eurocode 5: 6.1.7

Table: The meaning of symbols, design parameters and utilization results

Quick redesign can be done inside the *Utilization* table:

1 Select a panel or a design group in the upper table.

2 Modify the range of available panel types for the select element under *Parameters*.

3 Click *Design*.

Manual Design

With *Manual design* quick utilization check can be done for given types of selected timber panels or their design groups only. Just, choose the load type (load combination or load group) and a type name from the drop-down lists and select a panel or panel group, and program displays detailed utilization results in table format.

The meaning of the utilization components, the table content and features are the same as written before at *Auto design*. The program use the chosen type for all selected panels.

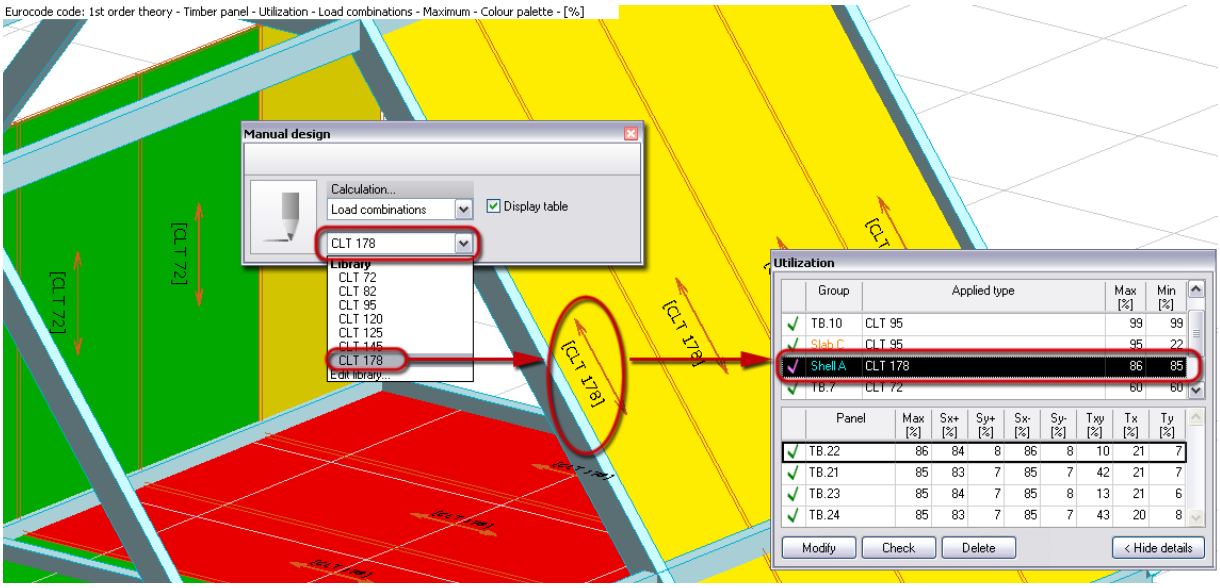


Figure: Quick check by Manual design

Detailed Result

Utilization of timber panels can be displayed in the following cases:

- After global *Auto design*, you can display utilization of all timber panels checked for the recommended panel types.
- When running element-based *Auto design*, utilization can be displayed by designed elements.
- After *Manual design*, element-based *Check* displays utilization for selected elements.

- After global *Check* done for all panel elements having final thickness (type).

No.	Global Auto design	Element-based Auto des.	Element-based Check	Global Check
1	*Calculate>Design calculation>Auto design all structural elements*	*Auto design*	*Auto design* and/or *Manual design*	*Auto design* and/or *Manual design*
2	*New result>* *Timber panel*	*New result>* *Timber panel*	*Check*	Apply changes
3			*New result>* *Timber panel*	*Calculate>Design calculation>Auto design all structural elements*
4				*New result>* *Timber panel*

Table: Steps of displaying timber panel utilization by different design cases

Utilization displayed with *New result* appears for all designed panels. The utilization components for a panel/design group can be displayed with *Detailed result*.

Detailed result opens a new window in the current project after selecting a panel/group member, which displays:

- **Applied panel type**
  A list displays the applied timber panel type with its main calculation parameters.

Group Shell A, Maximum of group members
Maximum of load combinations

CLT 178
Service class: 1, γ_M = 1.30, k_{sys} = 1.00

t	=	178	mm	$f_{m,0,k}$	=	14.00	N/mm^2
$E_{0,mean}$	=	5500.00	N/mm^2	$f_{m,90,k}$	=	10.30	N/mm^2
$E_{90,mean}$	=	4100.00	N/mm^2	$f_{t,0,k}$	=	9.80	N/mm^2
$E_{0,t}$	=	4100.00	N/mm^2	$f_{t,90,k}$	=	13.20	N/mm^2
$E_{90,t}$	=	5500.00	N/mm^2	$f_{c,0,k}$	=	9.80	N/mm^2
$E_{0,c}$	=	4100.00	N/mm^2	$f_{c,90,k}$	=	13.20	N/mm^2
$E_{90,c}$	=	5500.00	N/mm^2	$f_{v,0,k}$	=	2.00	N/mm^2
G_0	=	110.00	N/mm^2	$f_{v,90,k}$	=	2.00	N/mm^2
G_{90}	=	110.00	N/mm^2	$f_{r,k}$	=	2.00	N/mm^2

Figure: Applied timber panel type

- **Detailed calculation formulas**
  Calculation details and final values are collected by checking types: *Tension and bending* (Eurocode5: 6.2.3), *Compression and bending* (6.1.4, 6.2.4), *Shear, xy* (6.1.7) and *Shear, xz* and *yz* (6.1.7). The proper results are displayed in green, while the red result warnings you to repeat design with new panel types. The content of the utilization checks depends on *Display options*. Not relevant checks can also be hidden.

- **Summary graph**
  Summary graph is displayed with legend by default.

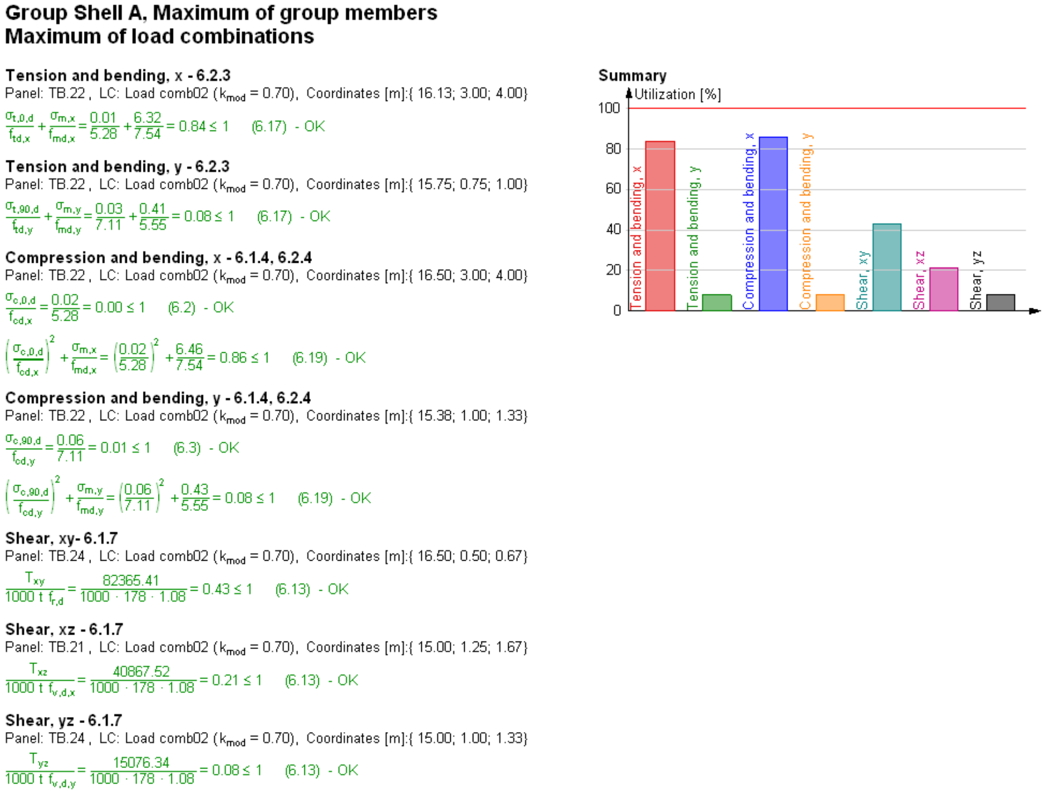


Figure: Utilization checks, formulas and summary

Detailed result *Tabmenu* contains the following tools and settings:

- **Selection of element to display**
  You can choose a unique or a design group member from the drop-down lists to display its detailed results mentioned before. Each row displays the ID and the maximum utilization of a member. In case of a design group, “Maximum” means the significant member having the maximum utilization.

- **Selection of design load**
  Depending on timber design was done for load combinations or load groups, a load combination or the maximum or a significant component of load groups can be selected for detailed results. Each row displays the name of the load combination/load group component and its utilization effect. “Maximum” means the significant load combination or component of load groups.

- **Auto design**
  Quick *Auto design* can be done for the currently displayed unique/group member. Its design parameters can be set/modified in the appearing dialog, and then clicking *OK* starts timber design that updates all detailed result figures and formulas.

- **Manual design**
  Manual design can be launched directly for the currently displayed unique/group member.

- **Display options**
  The content and the appearance of the detailed result can be set with *Display options*. You can show only the final equation without details of the different checks (*Hide details*).

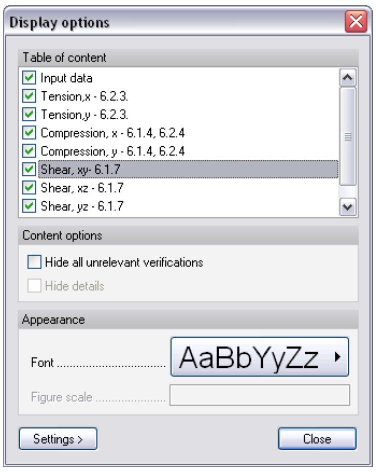


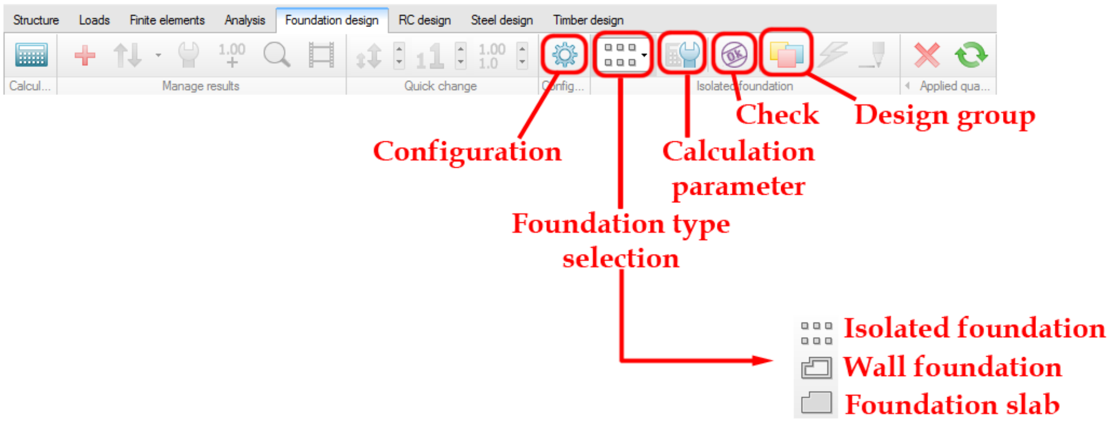
Figure: Display options of Detailed result

- **Go to**
  Navigate in the detailed result window by selecting the required design type in the drop-down list. It is useful when you are in zoomed view.

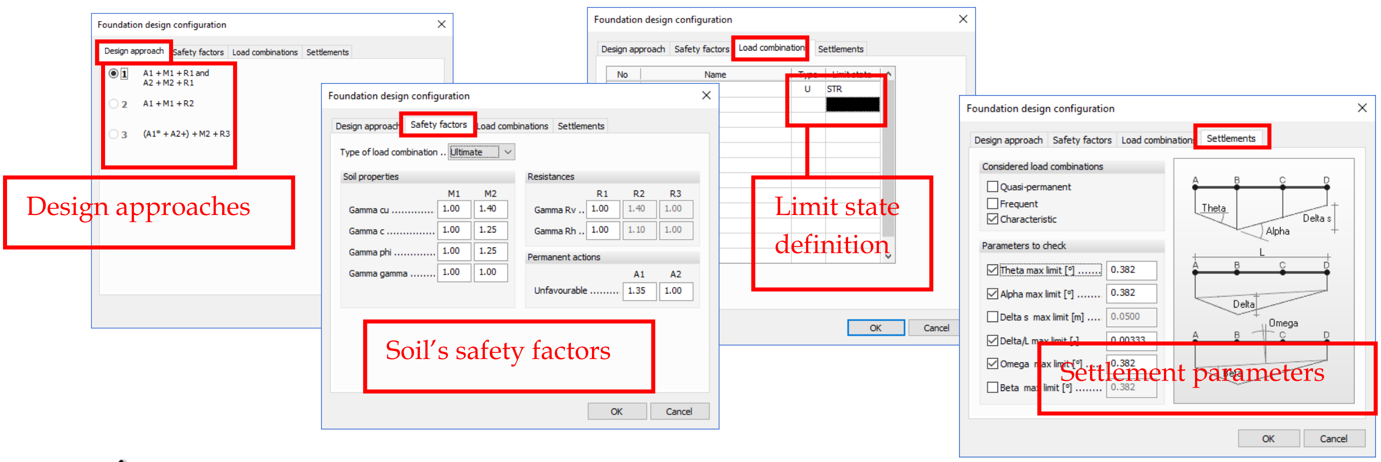
: *Tools* > *Add view to document* to place all figures, fomulas and summary table or specified details only into *Documentation*.

Foundation Design

Fast check are available to find the most suitable foundation.



In *Configuration* dialog, the user can set design options, like *Design Approach*, soil's *Safety Factors*, the load combination's *Limit state* and *Settlements parameters*.



In Foundation design only *Check* is available, *Manual design* or *Auto design* not.

In *Design approach* tab, the user can select the design approach which influences the combination of effects. For more details, see Eurocode 1997-1:2006 Chapter 2.4.7.3.4.

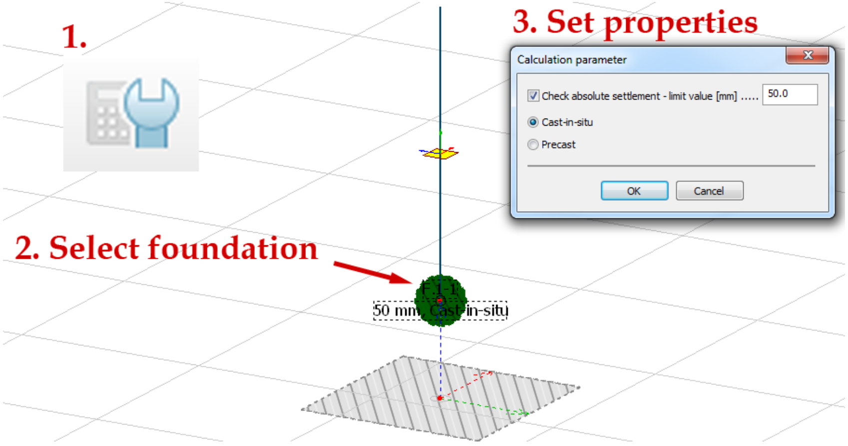
In *Safety factor* tab, the user can set the safety factors of soil properties, resistance and permanent actions.

In the *Load combination* tab, the user can set the limit states for each load combinations according to the selected *Design approach*.

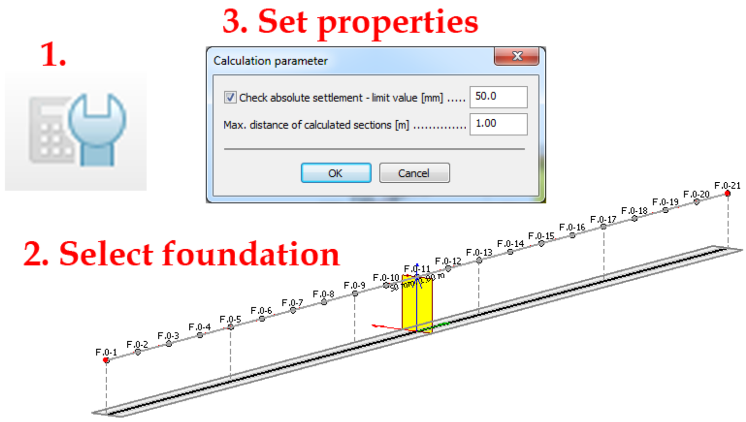
In the *Settlements* tab, the user can set for which types of load combinations (quasi-permanent, frequent, characteristic) he wants the settlement check to be performed and define the limit values of the relative displacements and rotations. For more details, see Eurocode 1997 H Annex.

By  *Design calculation parameter* command the foundation's calculation parameters can be set:

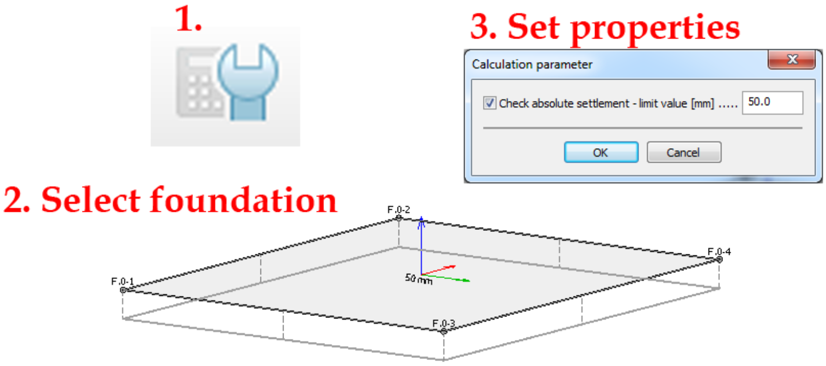
1. Isolated foundation:
    - settlement limit value
    - preparation method (Cast in-situ or Precast)



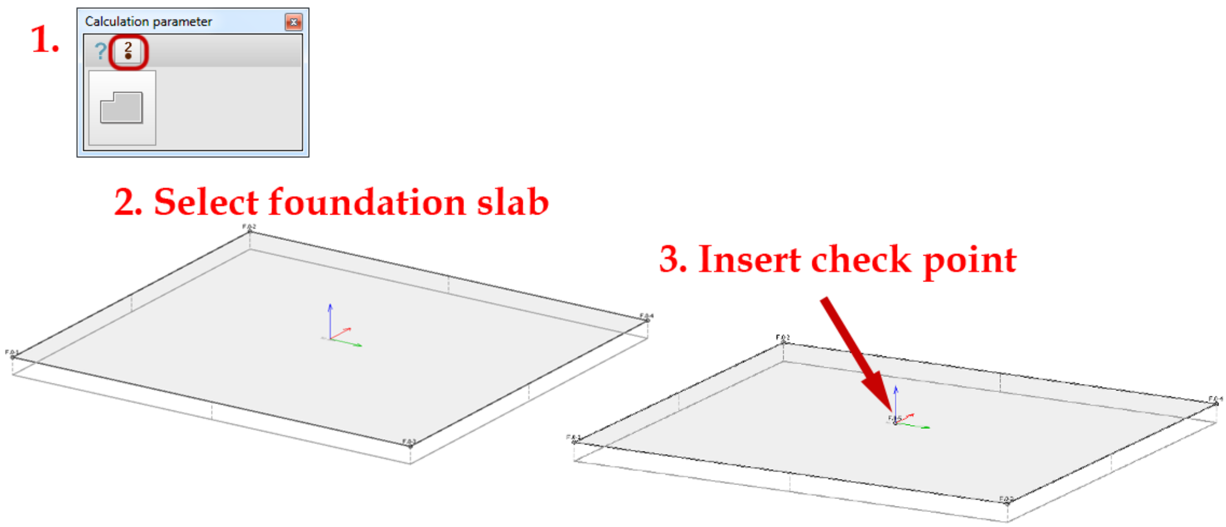
2. Wall foundation:
    - settlement limit value
    - maximum distance of calculated sections



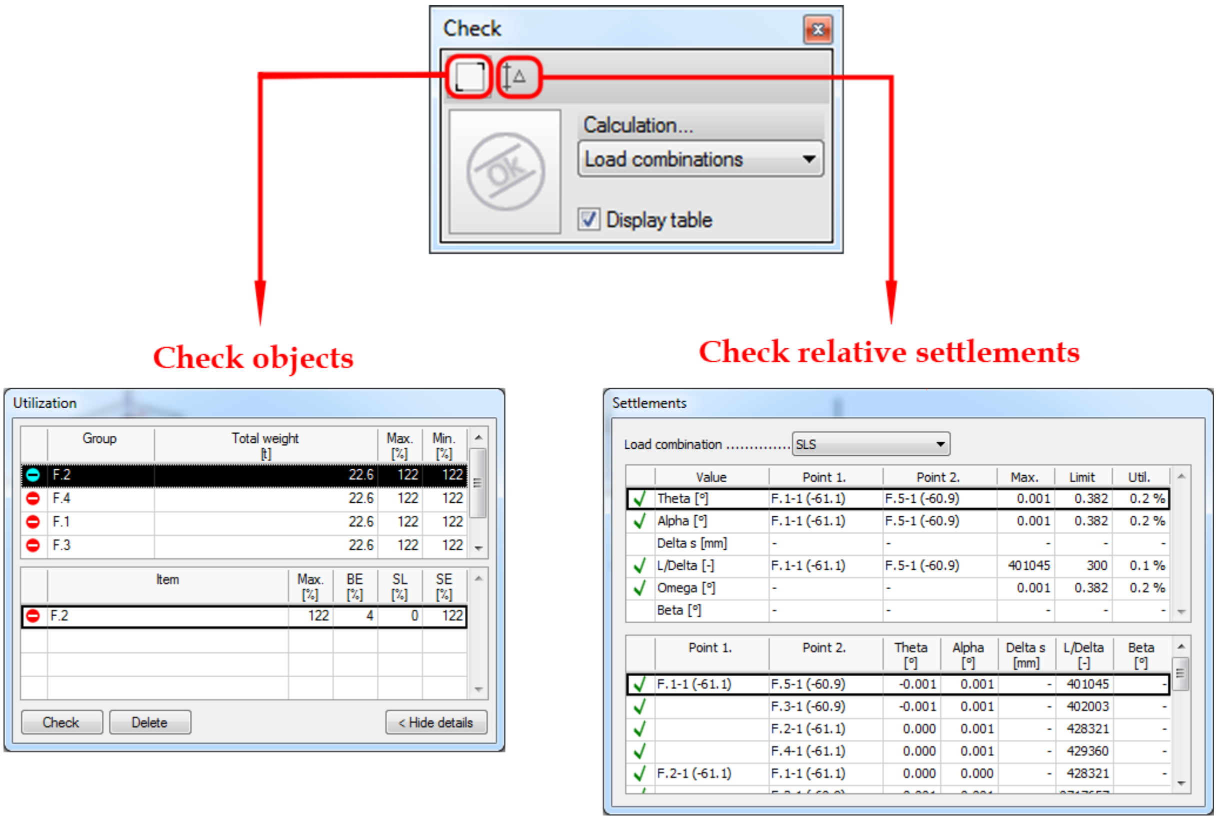
3. Foundation slab
    - settlement limit value



- Put additional check points for settlement calculation. These points automatically generated at the slab region's vertices and at the joint of columns and foundation slab.

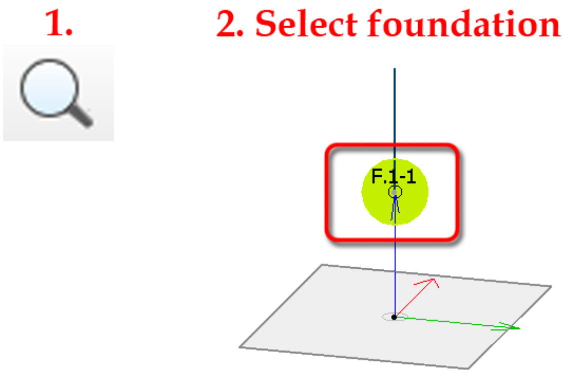


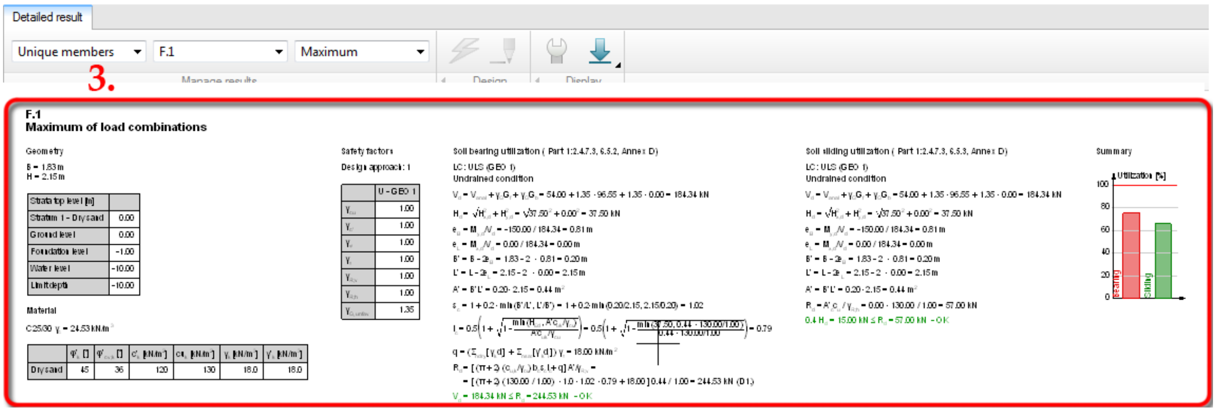
Design calculations can be done by *Check* command or in *Calculation dialog*. Under *Check* command the user can check either the objects or display the relative settlements.



For foundation slabs only relative settlements can be calculated with SLS combination. Due to this the design calculations for foundation slabs will not run the ULS combinations.

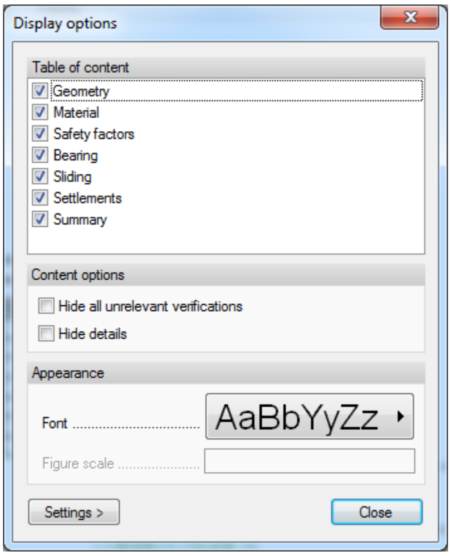
After displaying the *Foundation utilization* results, foundation design calculations can be seen by *Detailed results* command.

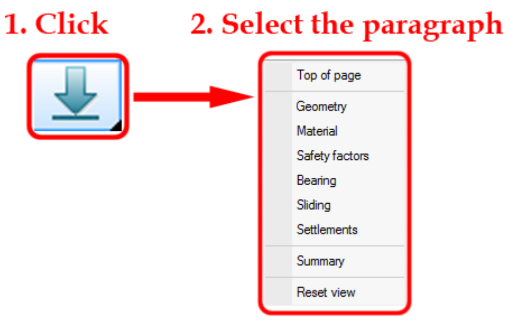




When choosing *Display options*  the following dialog appears.

Go to…  command helps the user to navigate in detailed results.





Performance based design

Model definition and calculation

Performance Based Design is implemented in FD 13 according to Turkish code. This way the performance level of existing RC buildings can be calculated from linear analysis. To calculate Performance level of an existing reinforced concrete building, follow these steps:

1. Set the code to "Eurocode (Seismic code: Turkish)".

2. Build the model in Structure tab.

   ⚠ The columns and walls have to be defined from storey to storey.

3. Add the loads in Loads tab.
   a. Define Gravity load combination.
   b. Define masses from loads in Load case – mass conversion dialog.
   c. Define the Seismic Load properties:
      i. Set the Structure type to Existing building, and also the Information level in Structure information tab.
      ii. Define the design spectra in Spectra tab.

      There are Standard and Unique spectra. Standard spectra are calculated according to the current seismic code. In order to define non-standard spectra the use the Unique option.

4. Run analysis in Analysis tab
   a. Calculate Load combinations in Calculation dialog.
   b. Calculate Eigenfrequencies in Calculation dialog.
   c. Calculate Seismic analysis (*Equivalent seismic load*) in Calculation dialog.
5. Define reinforcement for RC elements in RC Design tab.

6. In Performance based design tab

   a. Set the Calculation parameters for beams, columns and walls:

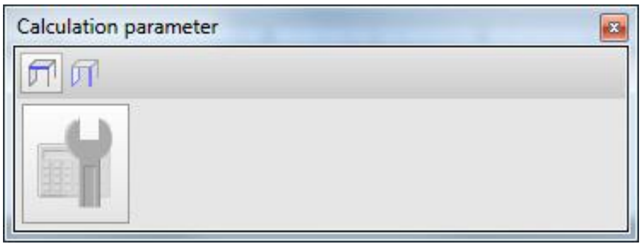


Figure: Setting the calculation parameters

- Beams: both ends are **Confined** and/or **Shear strengthened** and/or **Secondary beams**

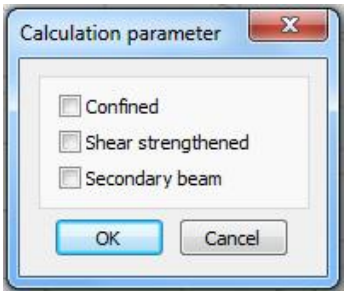


Figure: Calculation parameters for beams

- Columns / Walls: both ends are **Confined** and/or **Shear strengthened**

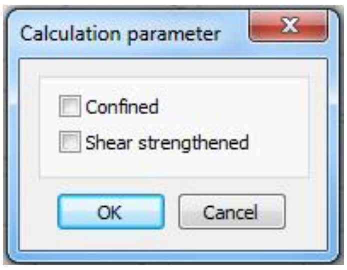


Figure: Calculation parameters for columns and walls

b. Start the Performance based design calculation by selecting the *Design calculations* under *Calculations* dialog.

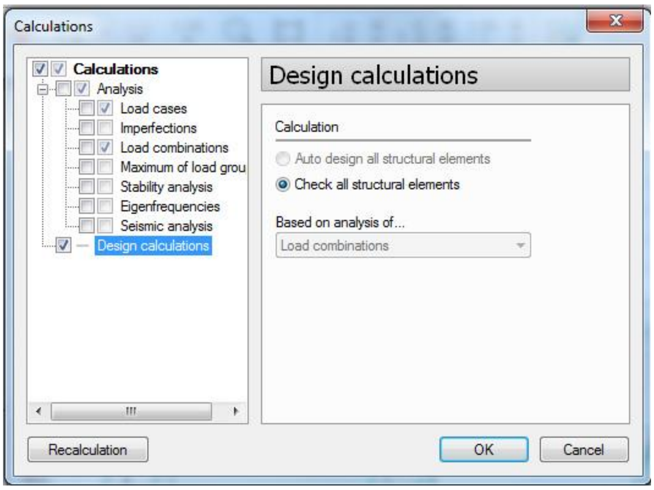


Figure: Design calculations

Results

For Performance based design the following results are available to display and list:

Results for display:

- Damage mode/zone:
  - By member
  - By member and joints
  - By member and storey drift

- By all

- Damage factor
- Relative storey drift ratio
- Performance level

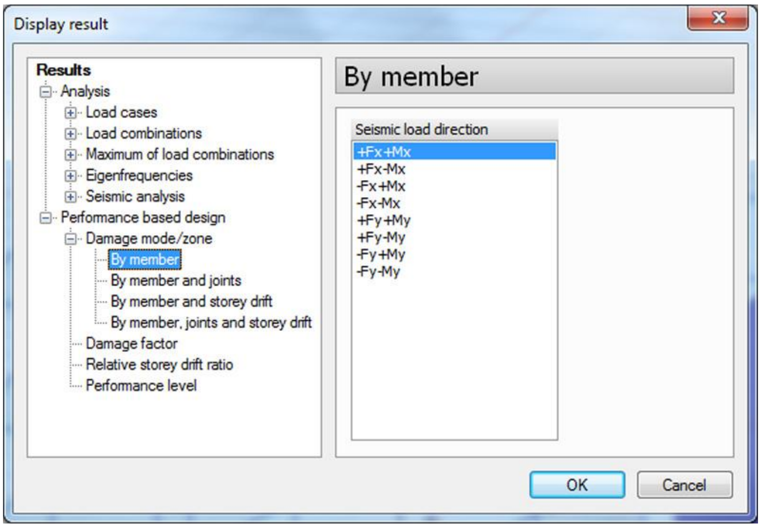


Figure: Display results

Results to list:

- Input data
  - Beam longitudinal reinforcement
  - Column longitudinal reinforcement
  - Wall longitudinal reinforcement
  - Beam strirrup reinforcement
  - Column stirrup reinforcement
  - Wall stirrup reinforcement
- Results
  - Beams, fracture mode
  - Beams, damage zone by member
  - Beams, damage zone, summary
  - Strong column check
  - Columns, fracture mode
  - Columns, damage zone by damage factor
  - Columns, damage zone by drift
  - Column-beam joint shear check
  - Columns, damage zone
  - Wall fracture mode
  - Walls, damage zone

- Building performance (Fx +Mx, Fx – Mx, -Fx + Mx, -Fx - Mx, Fy+My, Fy-My, -Fy+My, -Fy-My, summary)

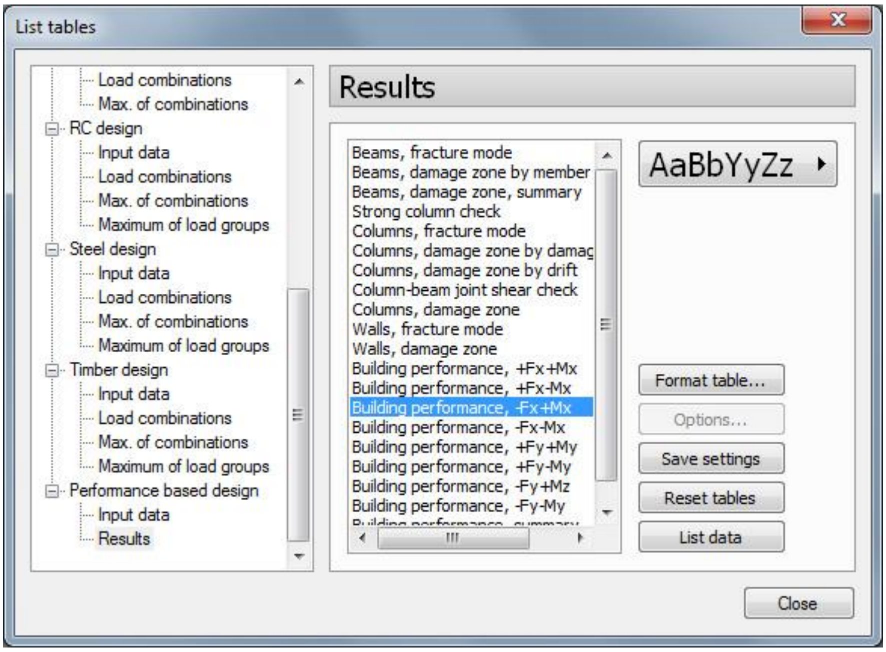


Figure: List results

Display Result

Different modes can be used for displaying Analysis and Design results. This chapter summarizes the skills and settings of display results on the screen. Of course the result figures can be printed out or can be inserted to documents (see Documentation).

Display Techniques

Finishing analysis and design calculations, the results are available in the list of the  *New* result command. Just click the command icon and browse from the results available from previously done calculations.

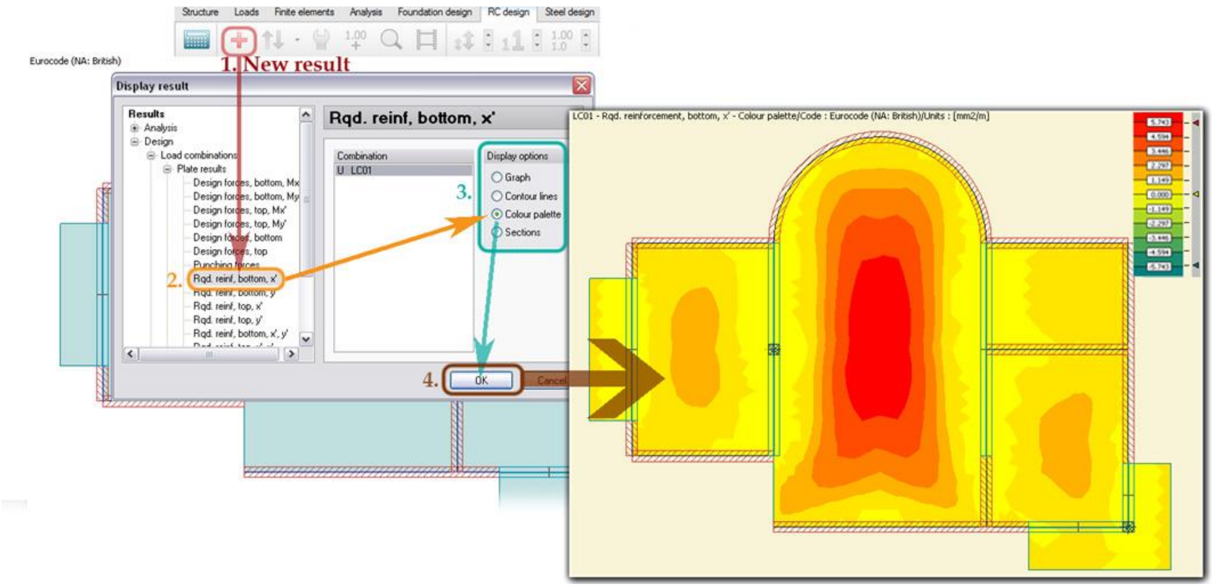


Figure: Quick way for displaying results

Select a result data, then a display technique (*Display options*) for it, and finally, clicking *OK* displays the result figure based on the default or last-used settings of the applied display technique.

	Available for	
Display option	*Bar elements*	*Planar elements*
Graph	✔	✔
Contour Lines		✔
Color Palette	✔	✔
Sections		✔

Table: General display options

If display options are not available for the selected result data, it will be displayed in one way. Some result types have special display modes.

	Available for	
Display option	*Bar elements*	*Planar elements*
Reactions/Connections	All type of supports and connections	
Bi-directional results		Principal internal forces, required reinforcement shown simultaneously in all directions
Principal directions		Directions of principal internal forces
Crack width		Result of crack width calculation (RC Design)
Punching results		Punching check results (RC Design)

Table: Special display options

For bar elements, the currently displayed result (e.g. displacement, internal forces, utilization for RC/steel/timber design etc.) can be detailed by selected bar with the Detailed result command.

The scale / color distribution of the current figure can be modified fast with Quick editing tools or with the Display option.

Graph

By choosing *Graph* display option, the result values (e.g. displacement, internal forces etc.) will measured and displayed in their calculation point (finite element nodes).

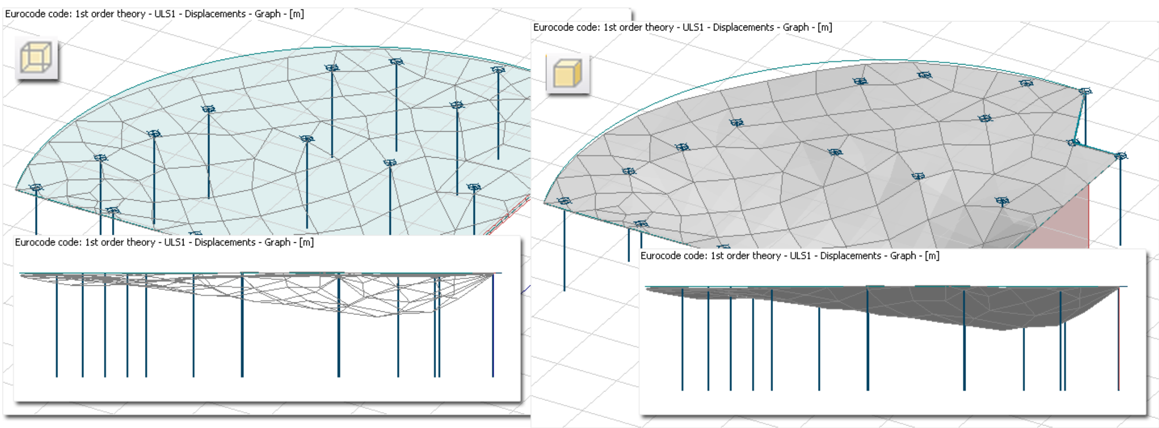


Figure: Displacement of a slab in different views and display modes

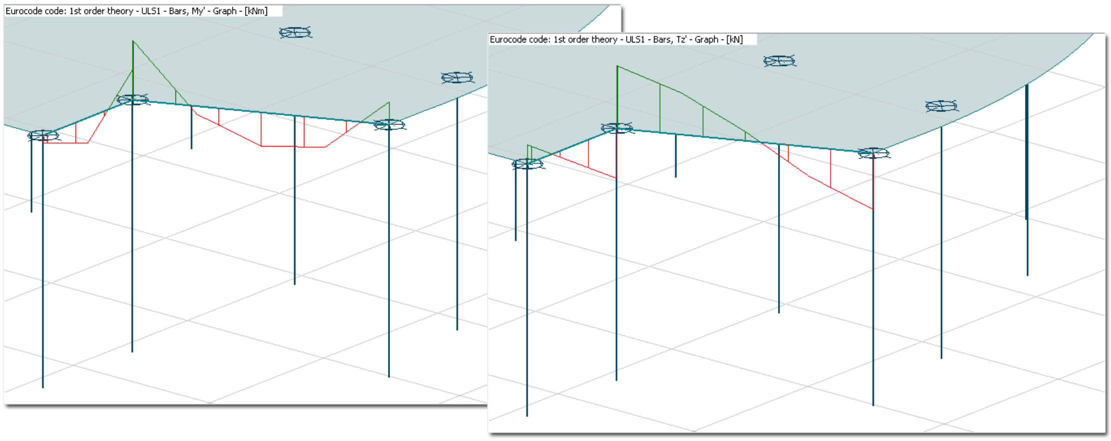


Figure: Moment and shear diagrams of beam elements

The settings of the currently displayed graph can be modified with the  *Display options* tool.

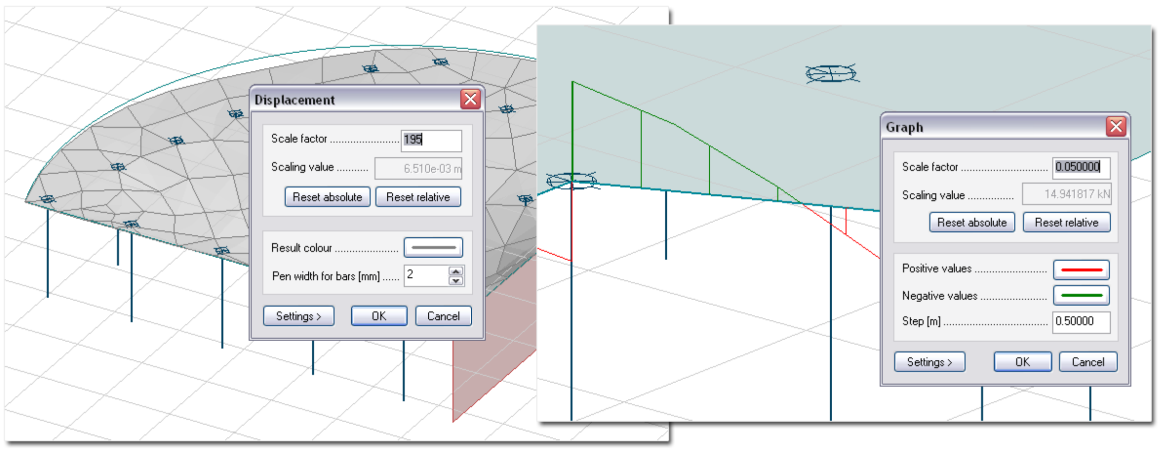


Figure: Display settings of graphs (planar and linear)

The following options can be modified and set for a graph:

- **Scale factor**

  It sets the display scale of the graph (enlarge (>1) or reduce (<1)). *Scaling value*, which is the absolute or relative maximum of the current result type, helps you in setting the scale factor.

- **Reset absolute vs. Reset relative**

  Pressing the buttons fits the *Scale factor* to the absolute or relative maximum of the same type results on the current structure.

- **Graph color**

  The color of the result and the positive/negative values (available for bar elements only) can be set by clicking the “colored line” button and by browsing requested colors from a palette. The button shows the actual color set for the displayed graph.

- **Step** (available for bar elements only)

  It is the distances between the points in which you can ask and display numeric values.

Contour Lines

Contour lines are a set of color lines. Each line connects points having the same result value and displays them with one color. It is similar to the contours of a topographical map which connect contiguous points of the same altitude.

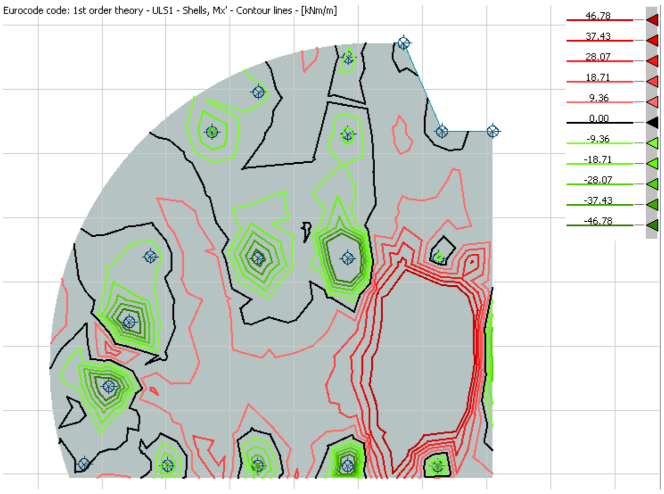


Figure: Contour lines

Contour lines display technique is developed to display results of plates, walls, shells etc. Two types of contour lines are available: "*Continuous*" and "*Discrete*".

"*Continuous*"-type contour lines contain colored line by equal step, while lines can be defined in arbitrary distribution with the "*Discrete*". You can define as many colors as you want for the contour lines.

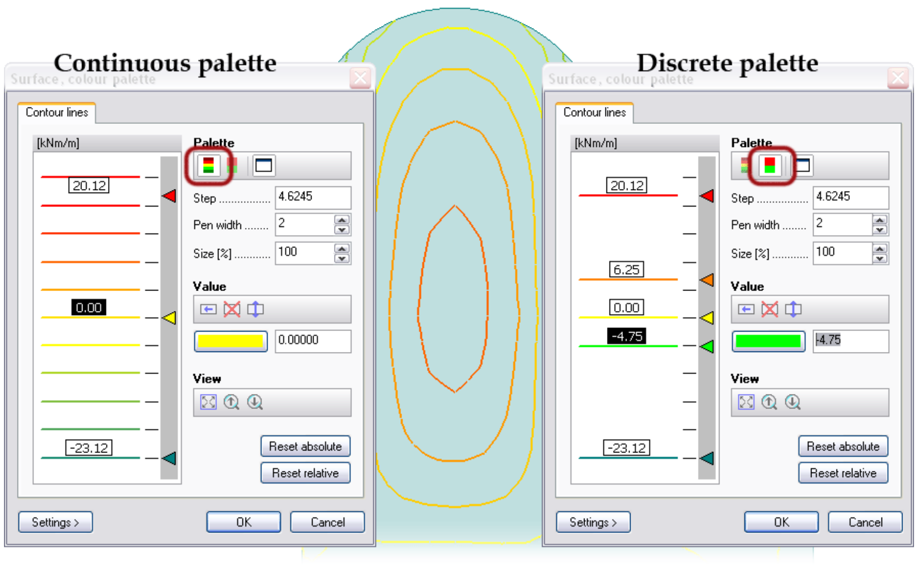
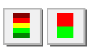


Figure: Continuous and discrete contour lines

The distribution and color settings of the currently displayed contour lines can be modified with the  *Display options* tool:

- **"Palette" settings**

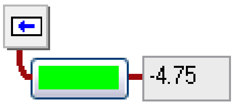
   Choose the required contour palette type: continuous or discrete.

The active option displays the meanings of the color (palette symbol) on the screen next to the result. *Size (%)* defined the size of the palette symbol, if it is displayed on screen.

In case of "Continuous" palette, *Step* sets the distance between the neighboring contour lines and so the color distribution.

Pen width sets the thickness of the contour lines on the screen.

- **"Value" settings**



Defining a color and a value in the available numeric field, the *Insert* option inserts a new line into the palette with the given color line and at the given value. This option is mainly developed to define the "*Discrete*" palette.

Delete option can be used to remove the unnecessary color line and its value currently selected in the palette with its ◄ symbol. It is equivalent with the "drag and drop out" of the line with its ◄ symbol.

Modify option moves the contour line, which is currently selected in the palette, to the given value and/or changes its color with a color set under the option.

- **"View" settings**

Zoom functions to enlarge or reduce the range of the palette. The last status will be used for the palette symbol, if it is displayed by .

- **Reset absolute vs. Reset relative**

Pressing the buttons fits the top and bottom limits of the palette adjust to the absolute or relative maximum of the current result.

Color Palette
Color palette is a set of colored regions. Each colored regions are represent values in a given interval. It is similar to *Contour line*, but not only the region borders but their inner areas are colored.

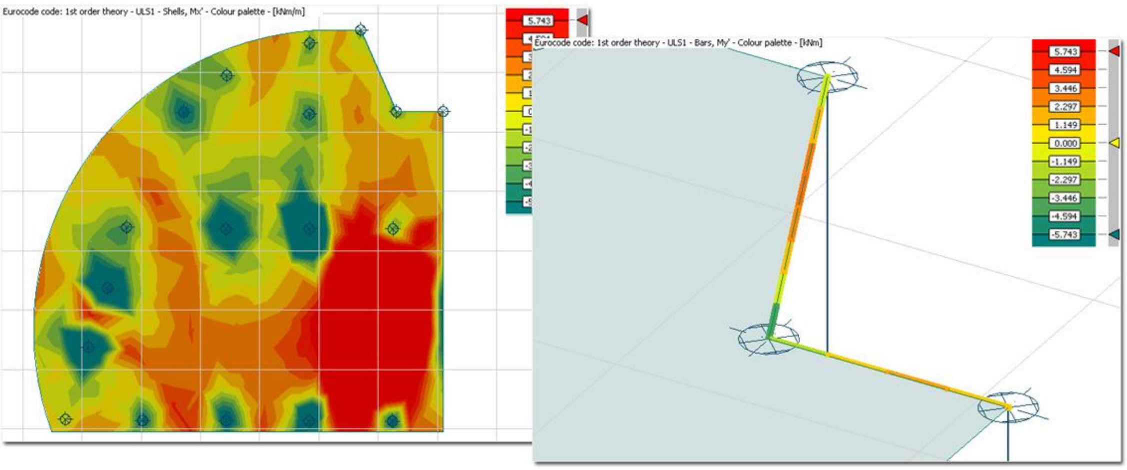


Figure: Color palette used for displaying results for planar and bar elements

Color palette display technique is developed to display results of both planar and bar elements. Two types of color palettes are available: "*Continuous*" and "*Discrete*".

"*Continuous*"-type contains colored zones by equal step, while the interval of the colored regions can be defined in arbitrary distribution with the "*Discrete*". You can define as many colors as you want for the color palettes.

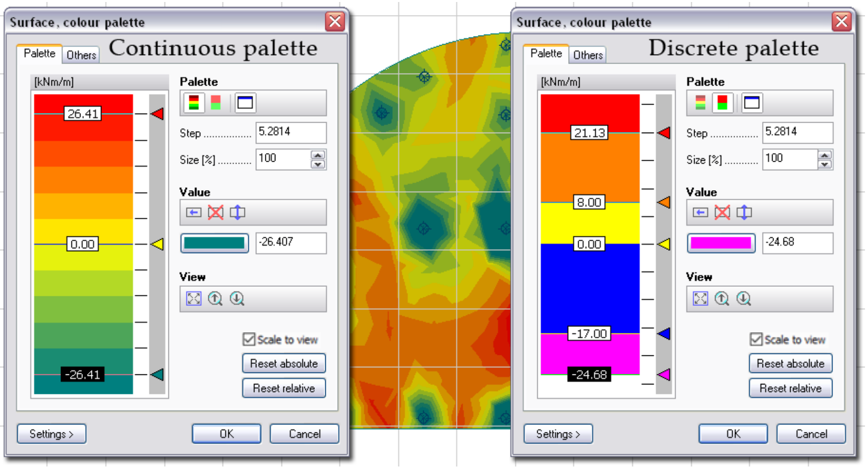
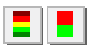


Figure: Continuous and discrete color palette for planar objects

The distribution and color settings of the currently displayed color palette can be modified with the  *Display options* tool:

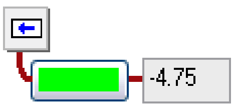
- **"Palette" settings**

 Choose the required color palette type: continuous or discrete.

 The active option displays the meanings of the color (palette symbol) on the screen next to the result. *Size (%)* defined the size of the palette symbol, if it is displayed on screen.

In case of "Continuous" palette, *Step* sets the interval of the colored zones and so the color distribution.

- **"Value" settings**



Defining a color and a value in the available numeric field, the *Insert* option inserts a new colored zone into the palette with the given color line and at the given bottom value. This option is mainly developed to define the "*Discrete*" palette.

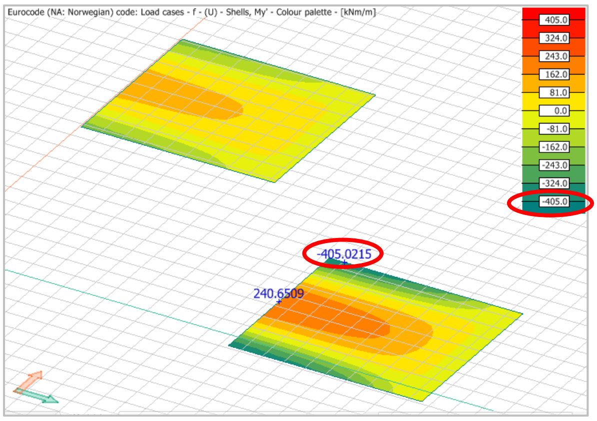
 *Delete* option can be used to remove the unnecessary colored zone and its value currently selected in the palette with its ◄ symbol of the bottom value. It is equivalent with the "drag and drop out" of the zone with the ◄ symbol.

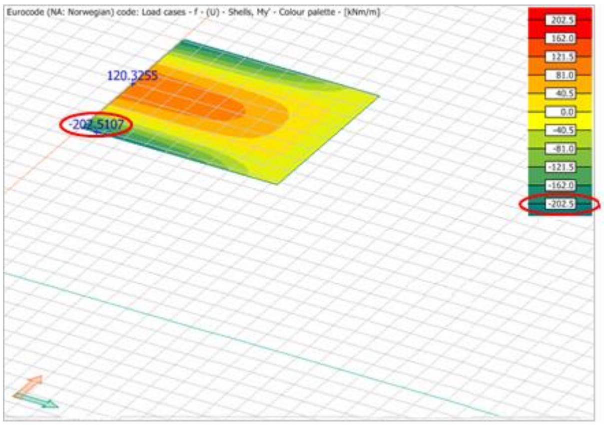
Modify option moves the colored zone, which is currently selected in the palette, to the position defined by the given bottom value and/or changes its color with a color set under the option.

- **"View" settings**

Zoom functions to enlarge or reduce the range of the palette. The last status will be used for the palette symbol, if it is displayed by .

With *Scale to view* option turned ON, if User hides part of the structure, the palette will be rescaled automatically:





If the result to be displayed needs wider scaling than the customized has, the program overwrites the scaling to the view.

- **Reset absolute vs. Reset relative**

Pressing the buttons fits the top and bottom limits of the palette adjust to the *absolute or relative maximum* of the current result.

Color palette display mode also gives the possibility to show displacement (=deformed shape) of surface/bar elements simultaneously the displayed result shown by color palette.

Apply *Deformed shape* for the color palette of *Mx'* plate result (*Analysis/ Plate internal forces*). First display *Mx'* with *Color palette*, and then open the palette's setting with the  *Display options* tool. Open the *Others* tab of the color palette dialog, and mark the *Display with the deformed shape* checkbox.

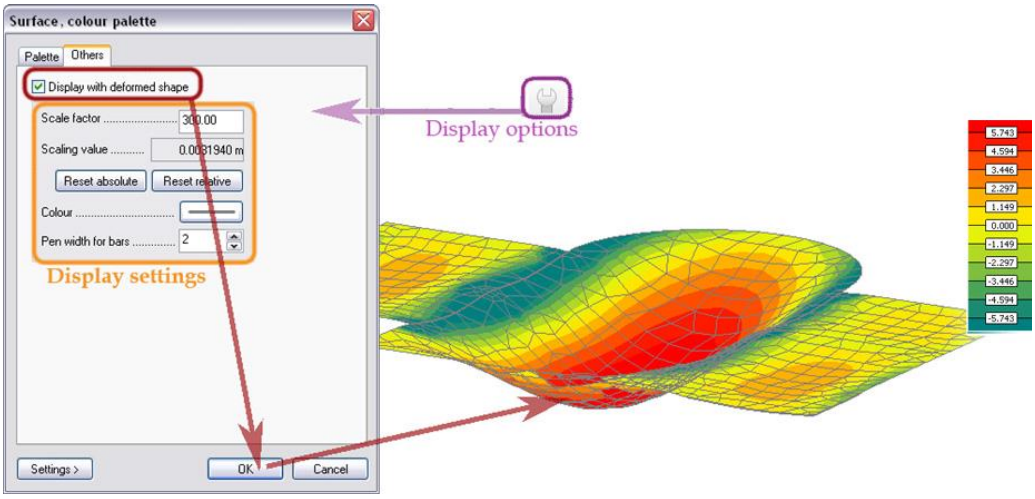


Figure: Color palette result displayed with deformed shape (planar elements)

Deformed shape can be also shown together with a color palette result (My') of bars.

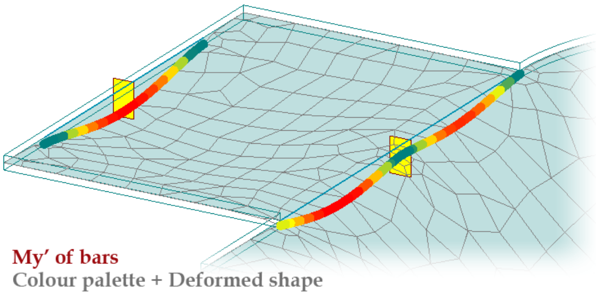
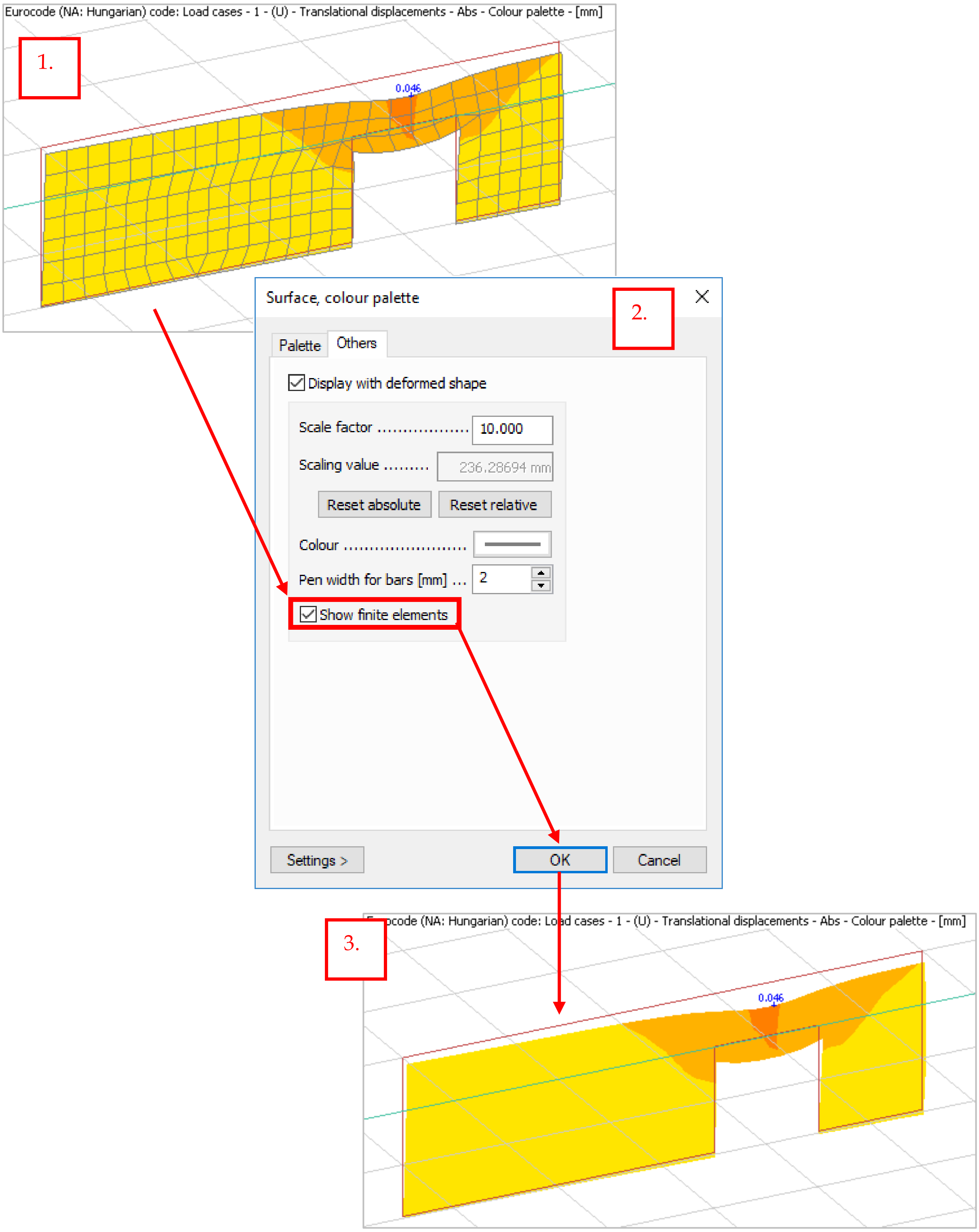


Figure: Color palette result displayed with deformed shape (bar elements)

Option to hide finite elements for colour palette results

To hide the finite elements from the result, click on the icon. In the pop-up window, *Surface, colour palette* click the *Others* tab and uncheck the box next to *Show finite elements,* then click 'OK'.

Picture No. 3. shows the result without the finite elements grids.



Sections

Results of planar objects, which are able to be shown with *Graph* mode, can be displayed in given section lines.

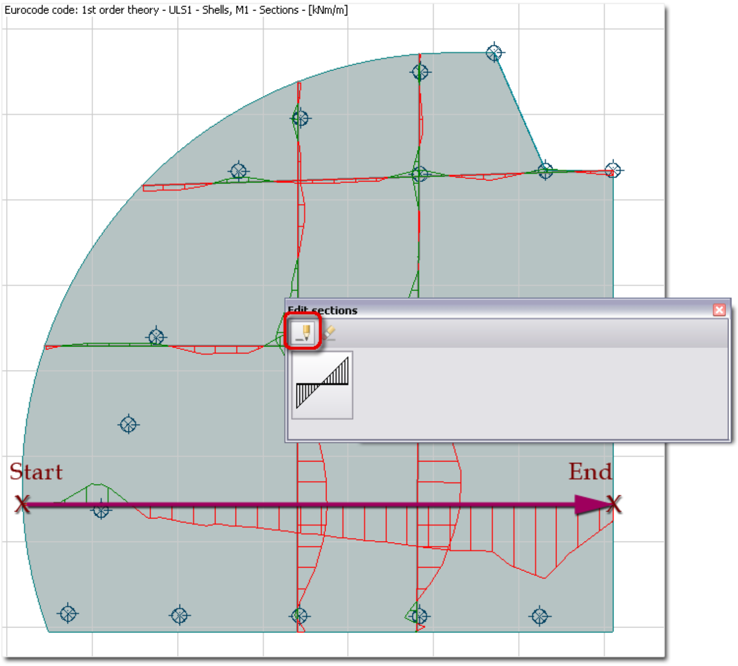


Figure: Result displayed in sections

Use the *Define* tool to the position of a section line. Just give the start and end point of the new section. You can continuously define next points with for further sections, such as you define a polyline; or stop the definition with the mouse button or the key.

After defining sections, the program fixes the section positions, so the next result - selected for displaying with *Sections* - will be displayed in the same section lines. For example, you can easily and fast check the same parts of a slab, wall etc. for different internal forces, loads or load combinations.

Display an analysis result (e.g. *Mx'*) and one of its derivatives in design (e.g. *Required reinforcement, bottom, x*') in the same section positions. Select the first result type with "*Sections*" display option, and define sections with their start and end points. Choosing the next result (e.g. *Rqd. Reinforcement, bottom, x'*) with will be displayed in the same positions.

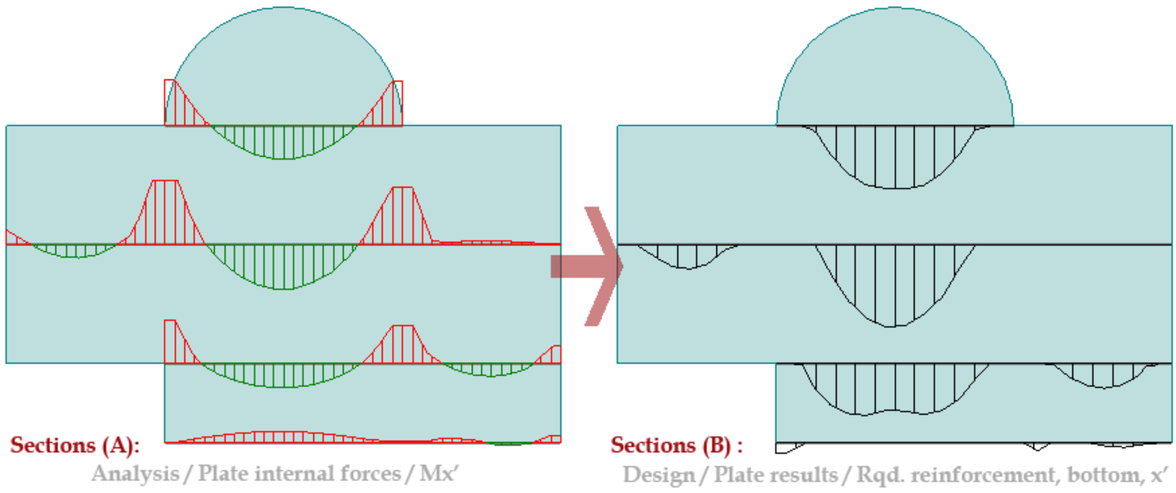


Figure: Section positions are stored for next results

To modify the color and hatch settings of the displayed sections, click *Display options* and open the settings dialog with the tool of the Section option.

Additional sections with the current settings can be added to the view with the *Define* tool. Unnecessary sections can be removed with the *Delete section* tool by just their selection.

- **Scale factor**

  It sets the display scale of the sections (enlarge (>1) or reduce (<1)). *Scaling value*, which is the absolute or relative maximum of the current result type, helps you in setting the scale factor.

- **Reset absolute vs. Reset relative**

  Pressing the buttons fits the *Scale factor* to the *absolute or relative maximum* of the same type results on the current structure.

- **Positive/Negative values**

  The color of the positive/negative values can be set by clicking the "colored line" button and by browsing requested colors from a palette. The buttons show the actual color.

- **Step**

  It is the distances between the points in which you can ask and display *numeric values* and the displayed hatch lines.

- **Distribution**

  The distribution of the displayed section results can be chosen. Besides the calculated distribution it can be linear or constant.

  This function can be useful at checking the average in-plane shear force between profiled panels.

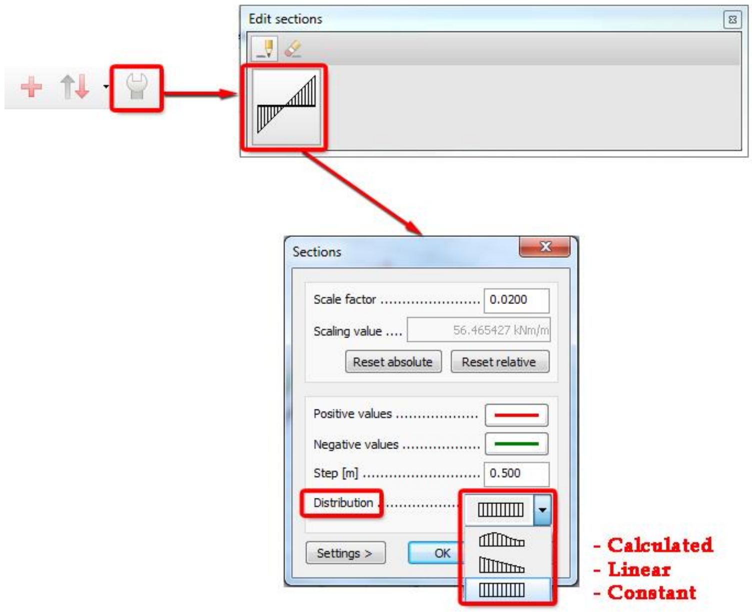
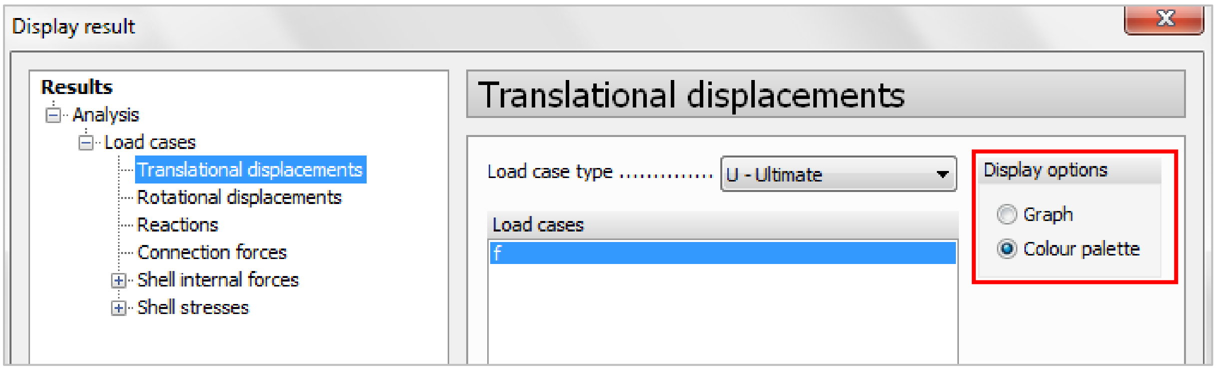


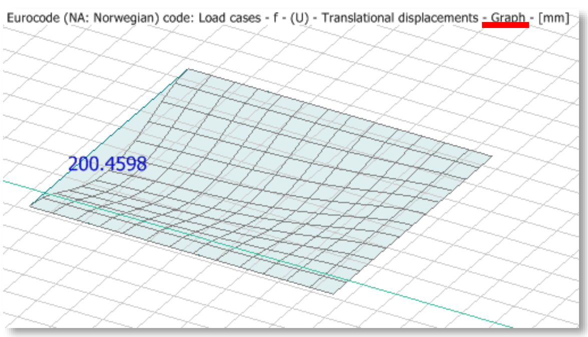
Figure: Section settings

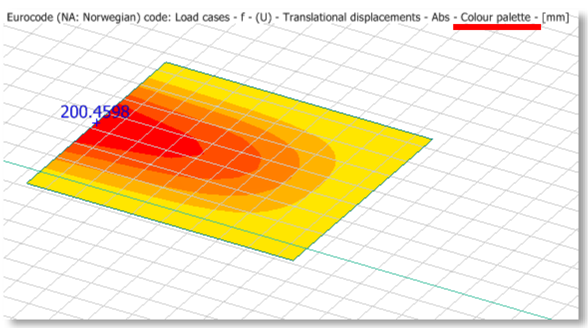
Display displacements
Displacements can be displayed as graph, contour lines, colour palette or sections.

1. Select the type of displacement result (translational, rotational) you wish to be displayed.
2. Select the load case type and the load case.
3. Click in the display-style of the result under *Display options.*



The two result types:





Display Reaction and Connection Forces

Reaction and connection forces can be displayed in supports and connection objects by type, by component, with group resultants, with the combination of them or the resultants at the middle of the support/connection.

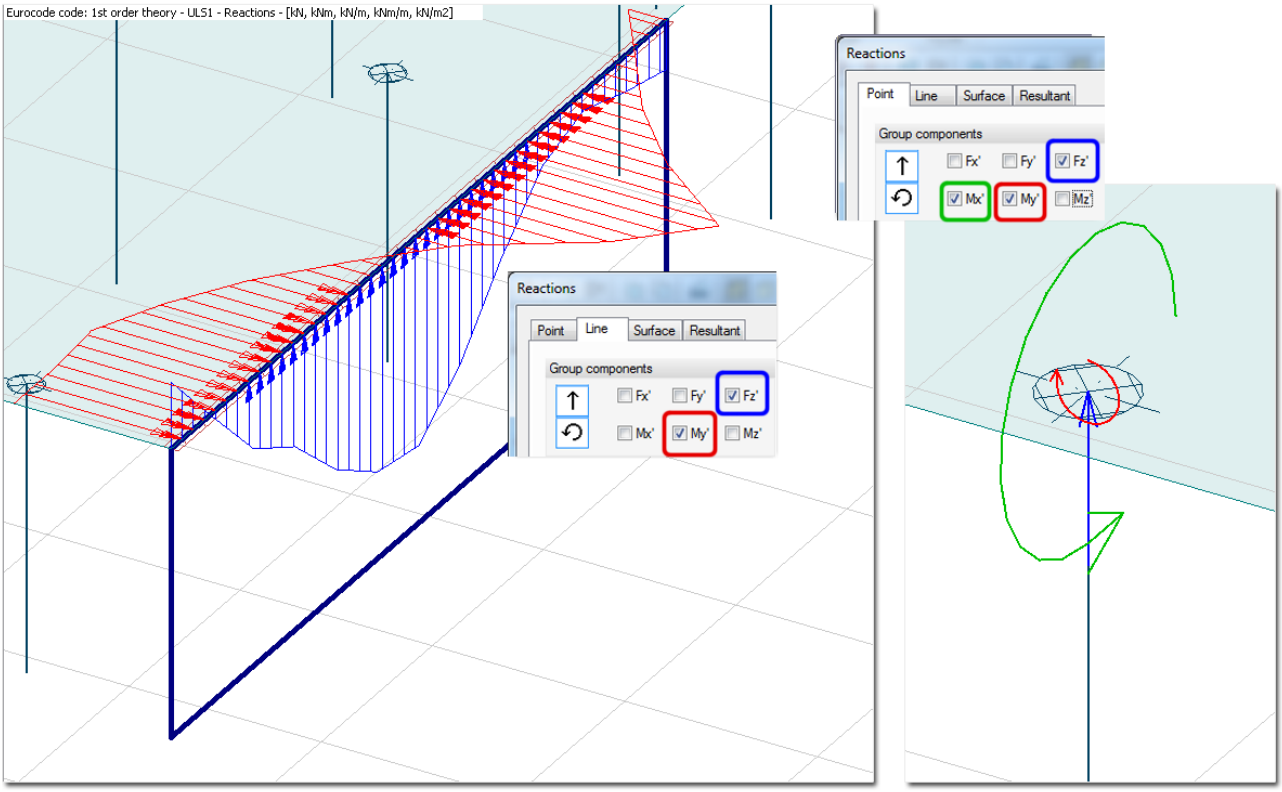


Figure: Reaction forces and moments in a Wall and Column support (Plate module)

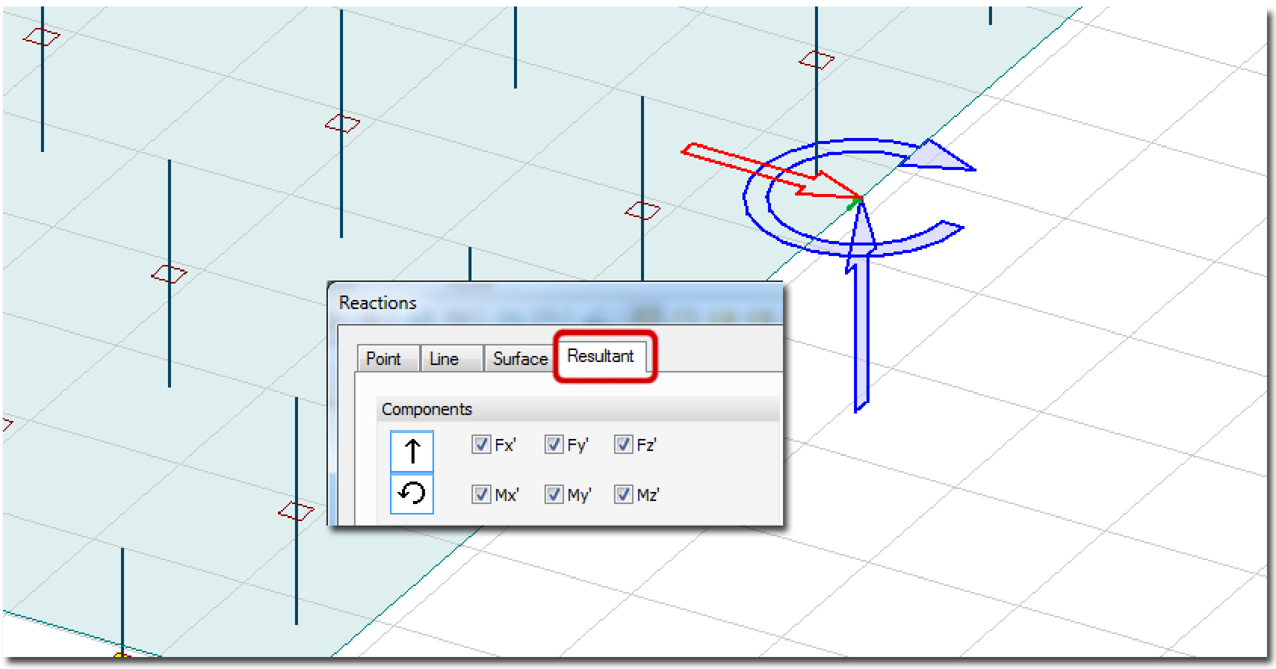


Figure: Reultants of the line support group

The color of a reaction/connection force component is represented by the proper direction component (x′, y′ or z′) of the support's/connection's local coordinate system. The color of the local system axes (directions) can be set at *Settings* > *All* > *Display* > *Local systems*.

To check and modify the settings of the displayed reaction/connection results, click *Display options*.

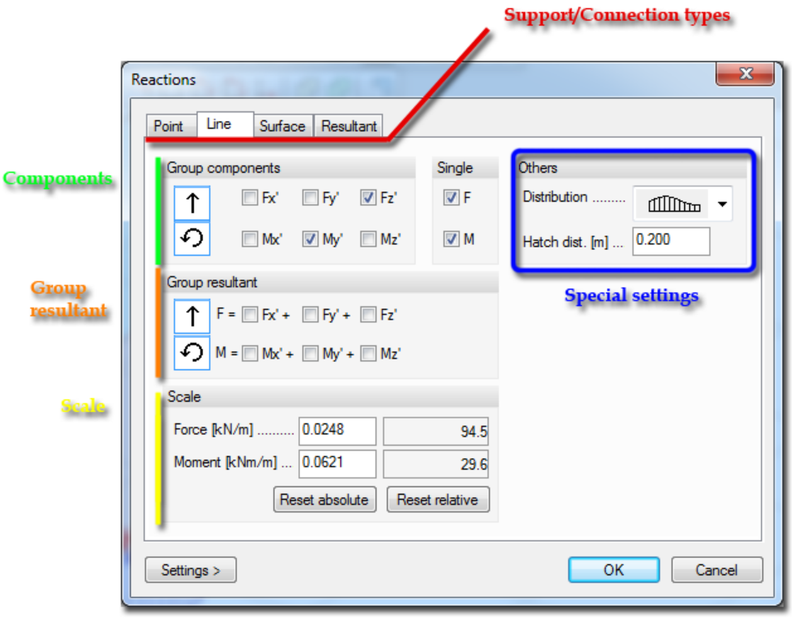


Figure: Display options of reactions and connection forces

Support/Connection types

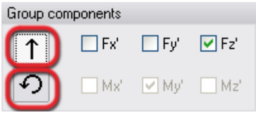
First, choose from the available support/connection types (point/line/ surface) you would like to display. Of course, the results of all types can be displayed simultaneously.

Component

Choose component(s) (force/moment and directions) you would like to display in case of single/group support or connection.

Resultant

The force (*F*) and/or moment (*M*) resultant of group support reactions or connection forces can be displayed. Resultants appear with dashed lines on the screen



Quick selection buttons let you hide all or show selected components on the screen.

Scale

It sets the display scale of the result diagrams (enlarge (>1) or reduce (<1)). The scaling value (shown in the grey fields), which is the absolute or relative maximum of the current result type, helps you in setting the scale factor. Pressing the *Reset absolute* or *Reset relative* buttons fits the *Scale factor* to the absolute or relative maximum of the same type results on the current project.

Distribution

Different displaying modes can be chosen for line reactions/connection forces:

- **Original**

  It displays the calculated result values.

- **Linear**

  It displays linear mean value of the result by support/connection element, which is calculated by the area-weight method.

- **Constant**

  It displays constant mean value of the result by support/connection element, which is calculated by the area-weight method.

- **Constant by element**

  To solve singularity peaks at line reactions and connections, the program calculates the average value (constant mean value) of the reaction and connection forces by element.

In this case, you can easily insert numeric values by the constant section (*Numeric value > Find all local maximum/minimum*).

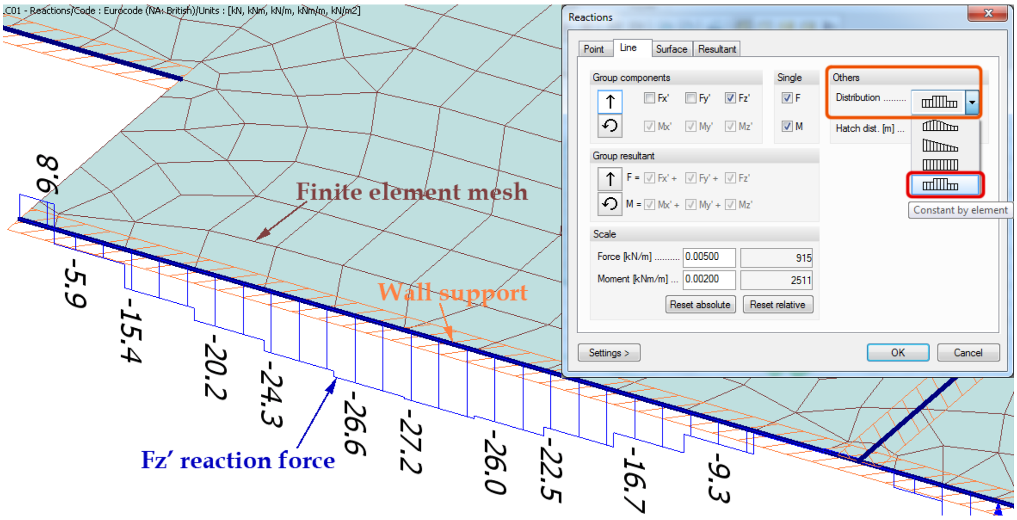


Figure: Line reactions displayed with "Constant by element"

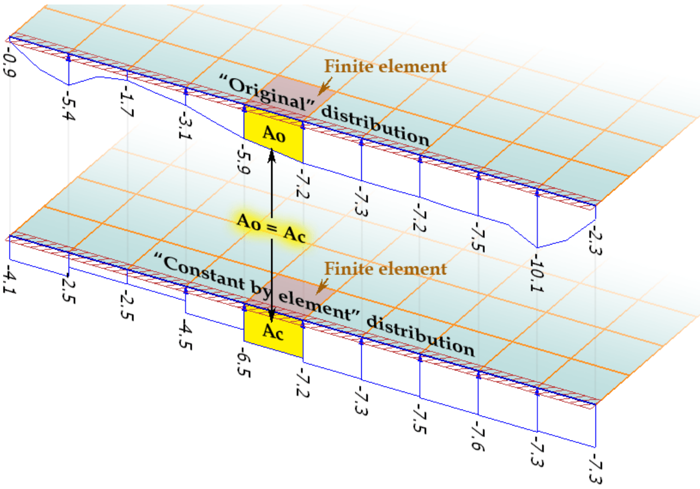


Figure: The theory of the average calculation by element (Constant by element)

The "uplifted" part of the support/connection is taken into account by displaying the distribution of line support reactions and connection forces.

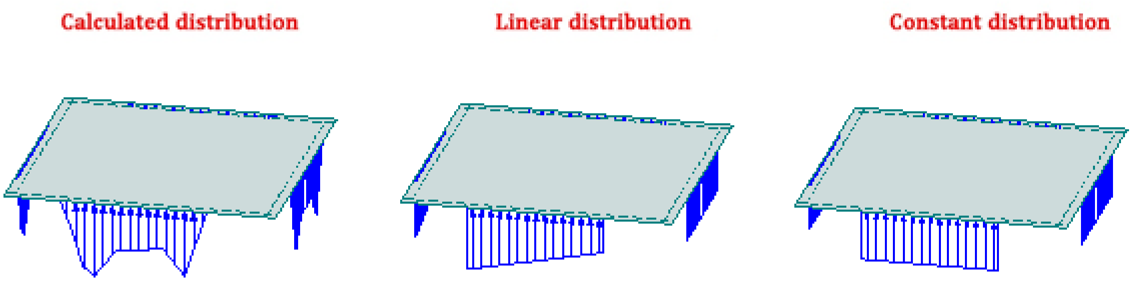


Figure: Reaction distribution on "Uplifted" connection

Bi-Directional Results

In some cases (e.g. initial forces, reinforcement results etc.) two result components can be displayed together, in one diagram. At these results, numerical values are also displayed besides the directions.

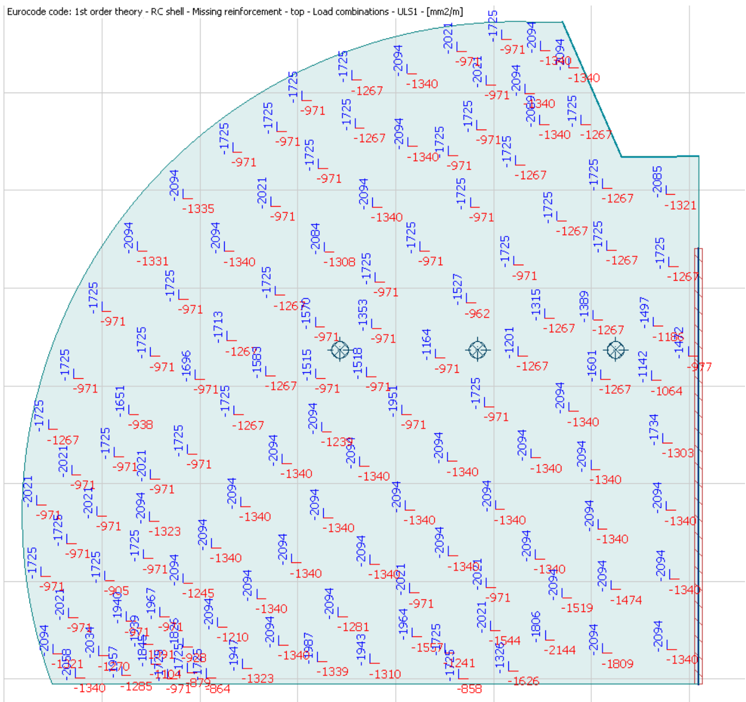


Figure: Required bottom reinforcement displayed with directions and numeric values

The color and text settings of the currently displayed bi-directional result can be modified with the  *Display options* tool.

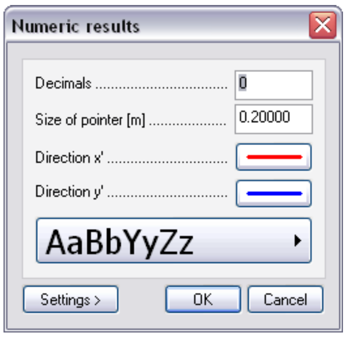


Figure: Display settings of bi-directional results

Principal Directions

The direction of principal internal forces and stresses can be displayed in two ways:

- **"One size"**

  The symbols of the principal directions are shown with uniform-length lines. The symbol line of the first principal direction is longer than the second ones.

- **"Proportional – scale"**

  The length of a symbol line represents the size of the principal force/stress. The display scale factor can be customized according to the absolute/relative maximum of the result.

  The algebraically **larger value** will be **the first principal value,** by default indicated with green colour**.** The algebraically **smaller value** will be **the second principal value**, by default indicated with red colour**.**

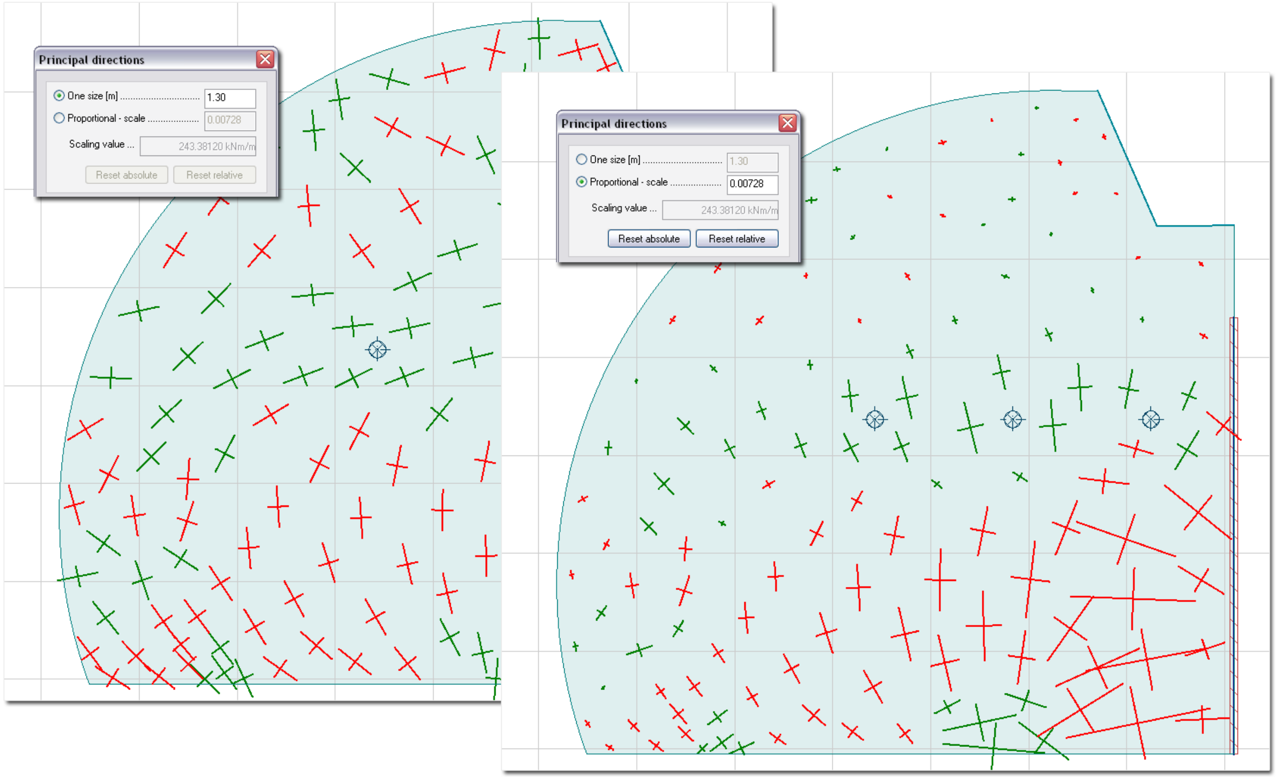
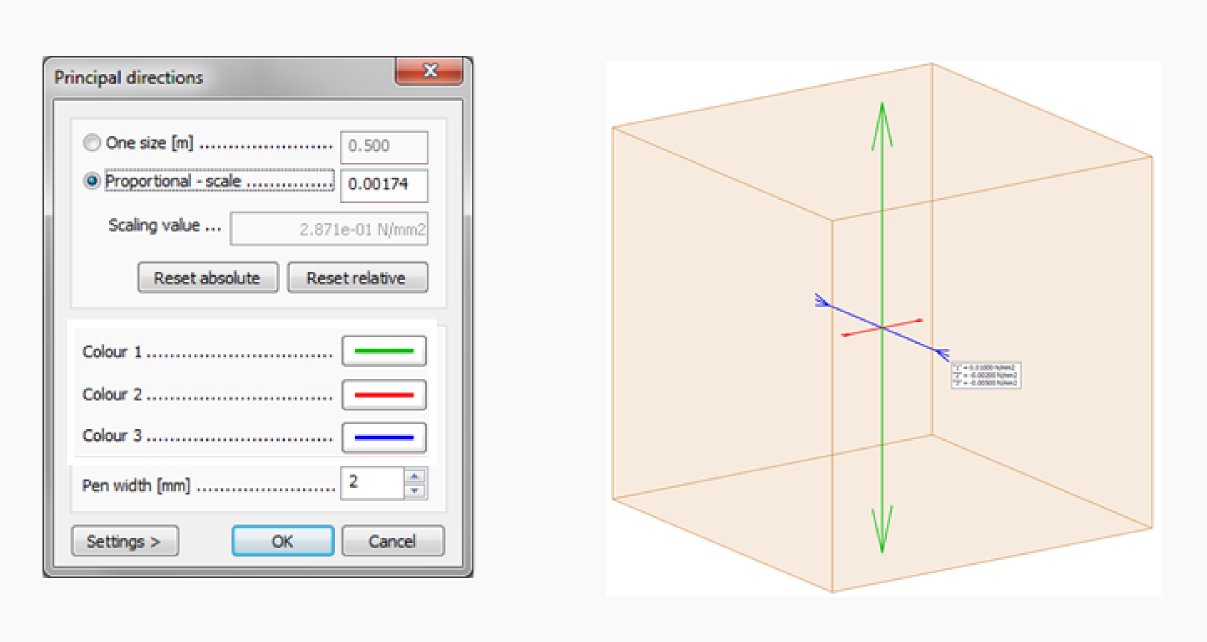


Figure: One-sized or proportional-scaled principal results

The display type and the colors of the principal directions can be set and modified with the *Display options* tool.

Arrows at the ends of the lines are indicating whether the principal stresses are tension (positive) or compression (negative).



Crack Width
Crack width results of RC slabs and bars can be displayed with crack lines. The lines represent the direction of the cracks, their pen width the size of the crack width and the color represents if the crack width exceeds the *allowed crack width*.

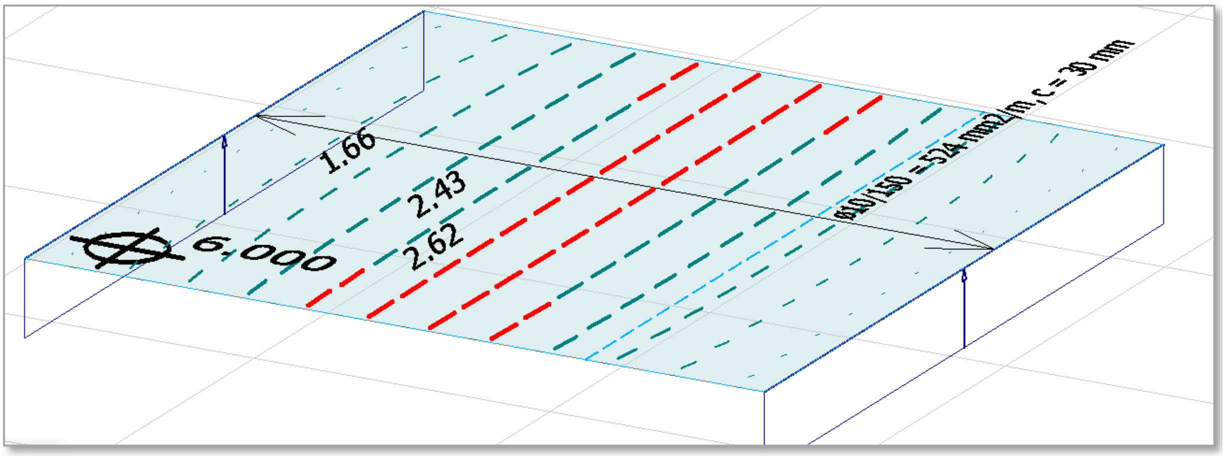


Figure: Crack width of RC slab

The length, width and color of the crack lines can be modified with the *Display options* tool and the *allowed crack width* limit can be set in the Calculation parameters under RC design tab.

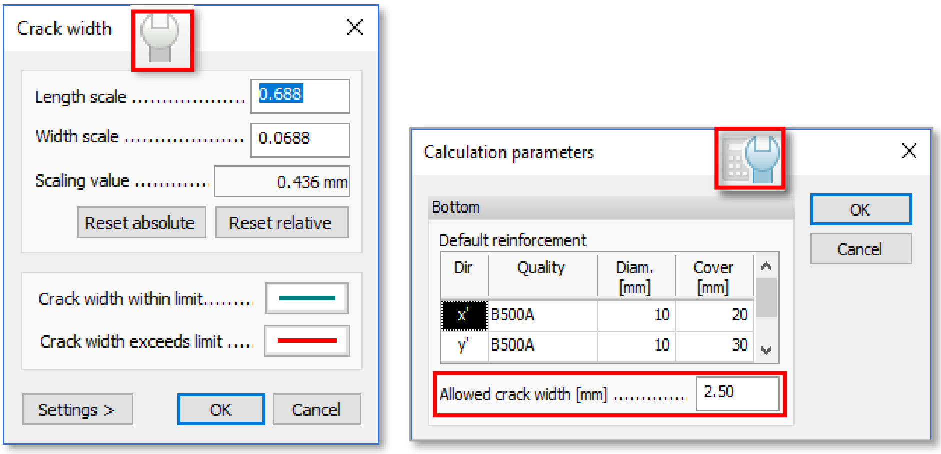


Figure: Display settings of crack width result

Design Utilization Results

Global *Auto design* and *Check* calculations give utilization results for planar and bar elements by the applied design type.

Utilization results are colored figures (color palette), and summarize the list of maximum utilizations by elements in a table dialog with the help of the Numeric value tool.

Read more about the utilization results by design type: RC design, Steel design and Timber design.

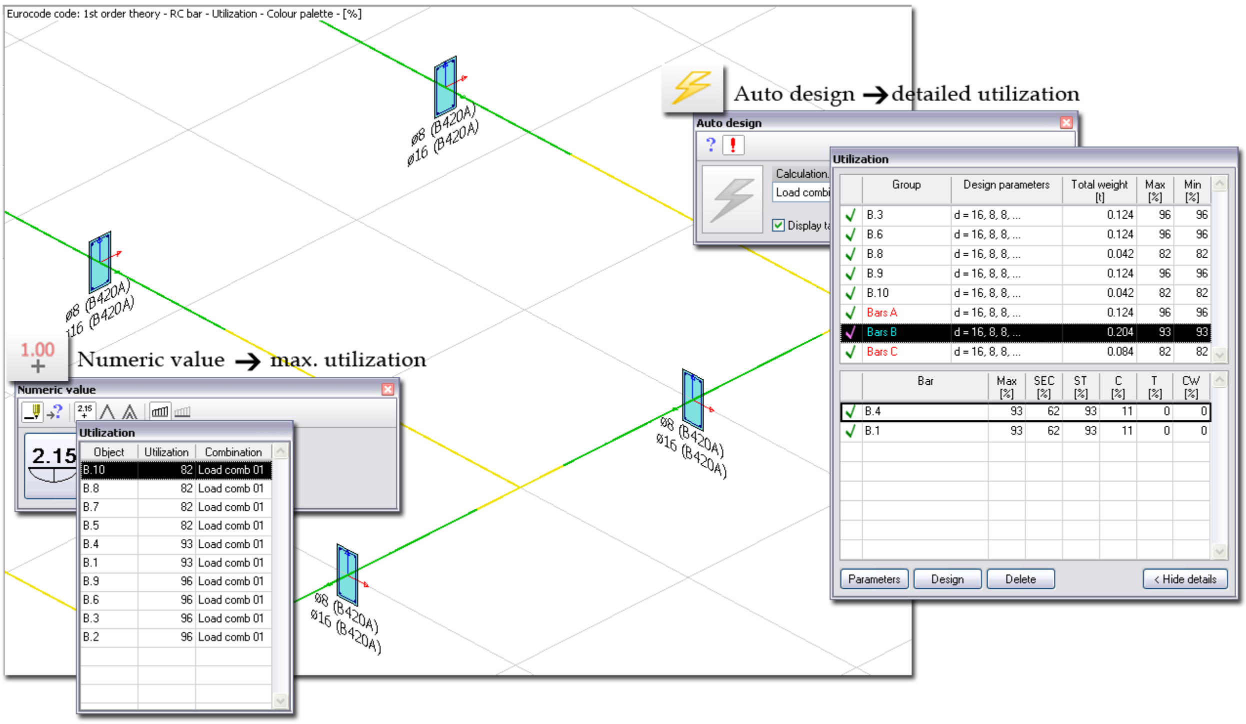


Figure: RC bar design and utilization result

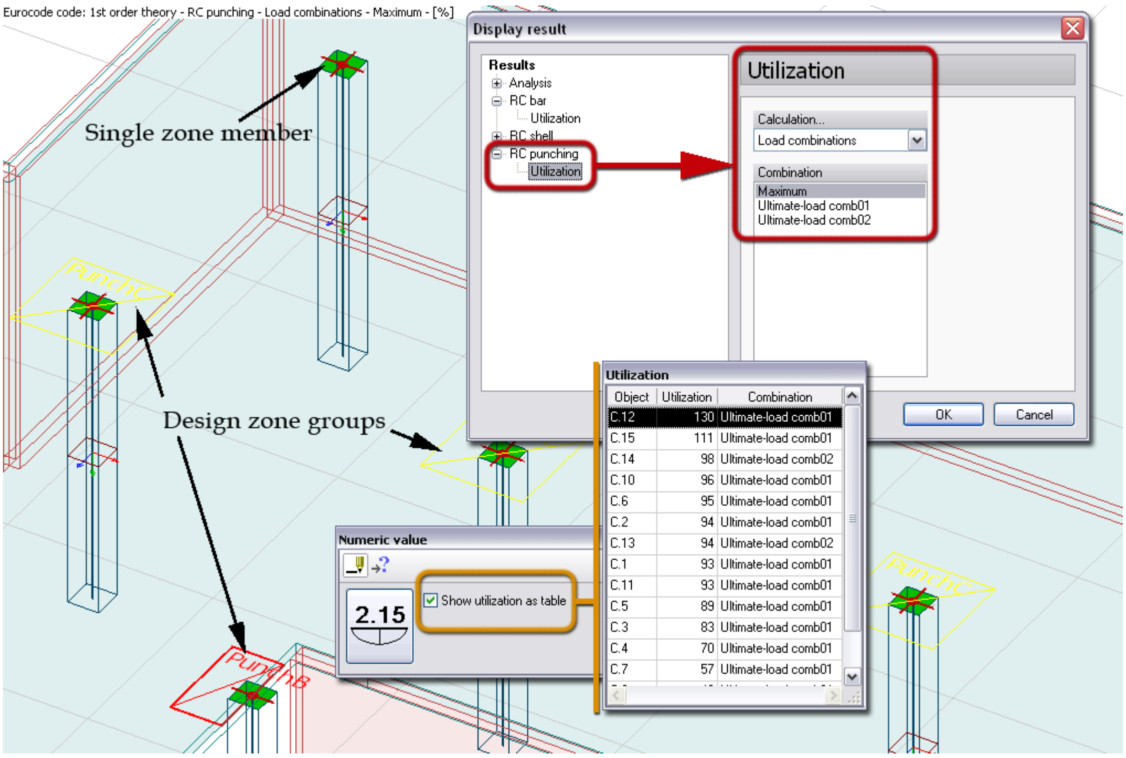


Figure: RC punching design and utilization result

Detailed 1D Object Result

Detailed analytical results are available for one-dimensional analytical objects (e.g. bars, line support groups and connections) after clicking on 🔍 *Detailed result* tool and by selecting the object.

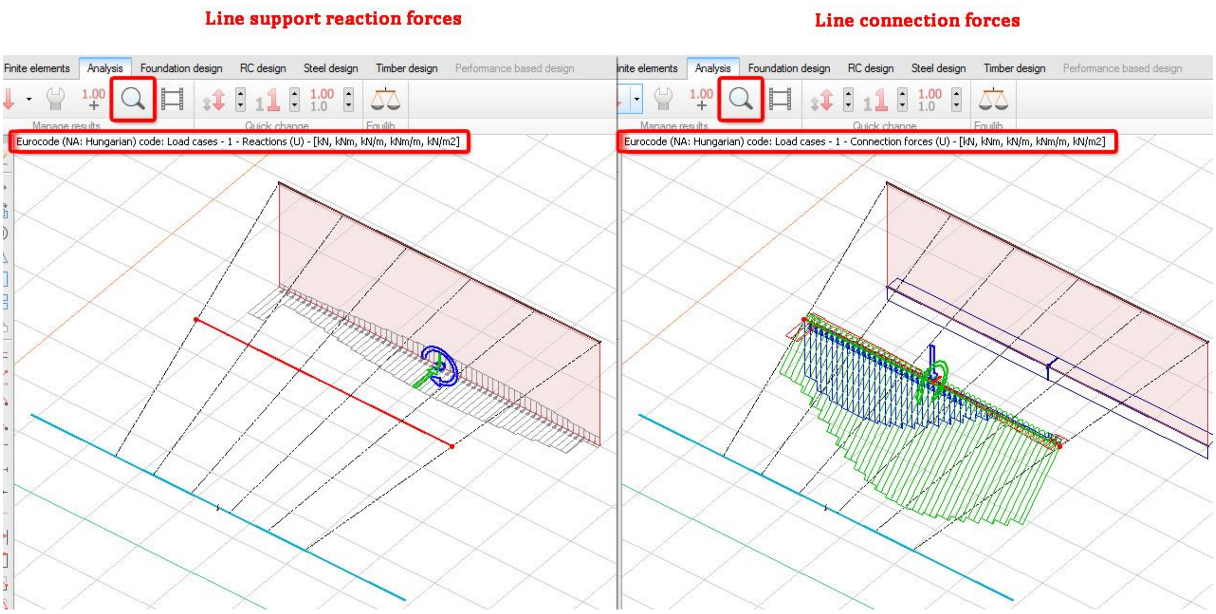


Figure: Detailed results on a line support and line connection

In Detailed result window the User can manage the results by selecting the object, the calculation and other – calculation dependent – options.

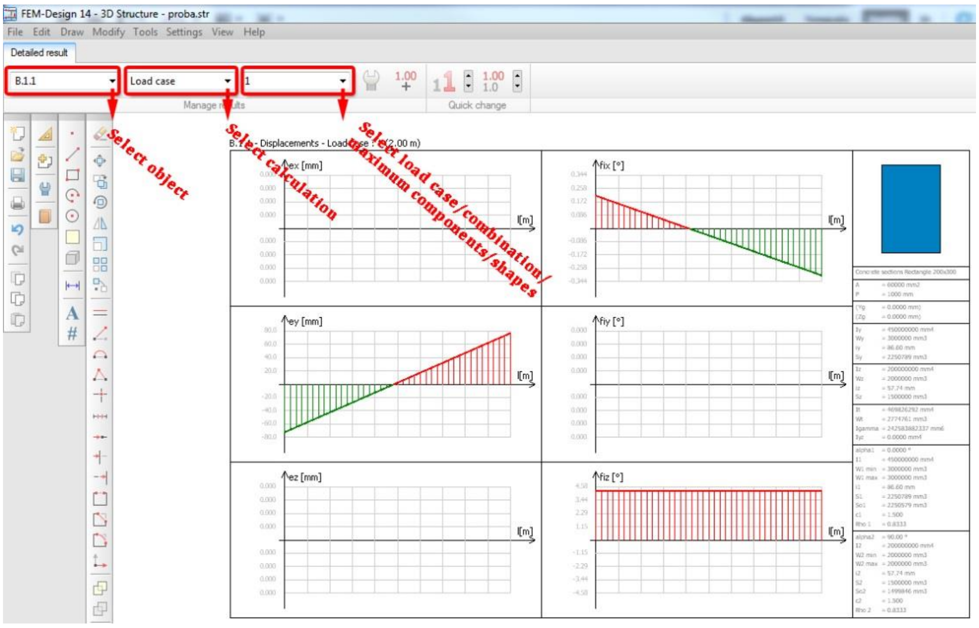


Figure: Detailed result window

The display type, colors and distribution mode of the detailed results can be set and modified with the *Display options* tool.

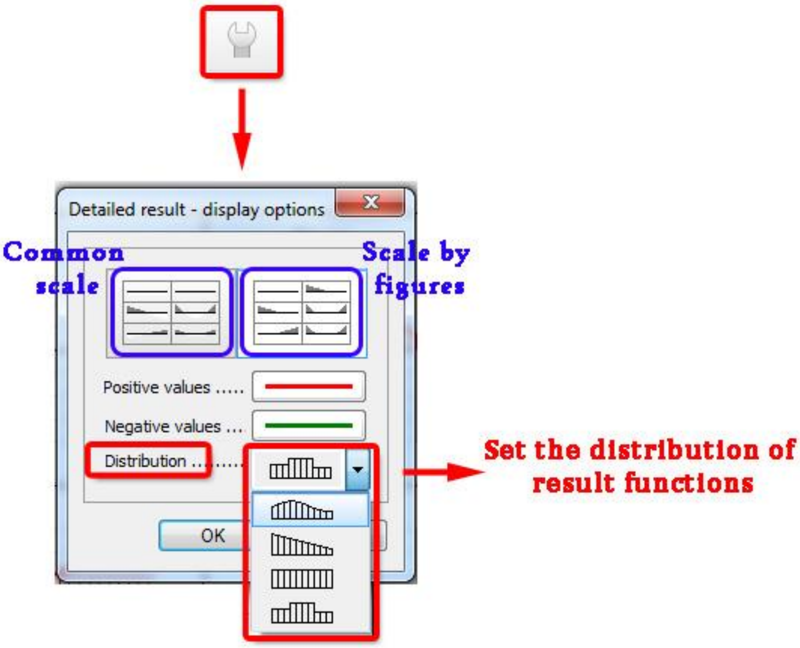


Figure: Display result – display options

- **Diagram type**

  Common scale displays the same-type result diagrams (e.g. My, Mz and Mt are moment diagrams) in common scale. The program automatically sets the scale value to the *absolute maximum* of the current result.

  Scale by figures displays diagrams in their best scale. The scale values, the upper and bottom limits will be set by figure according to its own *absolute maximum* value (e.g. the scales of My and Mz differ from each other).

- **Positive/Negative values**

  The color of the positive/negative values can be set by clicking the “colored line” button and by browsing requested colors from a palette. The buttons show the actual color.

- **Distribution**

  The distribution of the selected result functions can be chosen (calculated, linear, constant or constant by element) for line support groups and connections.

The hatch line dense by the figures is the same with the “display steps” of the original bar result, on which the *Detailed result* tool was used.

Utilization results with detailed background calculation formulas (together with Eurocode references), figures and tables can be displayed by single elements or by design groups. Quick navigation is powered with zooming details. Read more about *Detailed results* by design type: *RC design*, *Steel design*, *Timber design* and Foundation design.

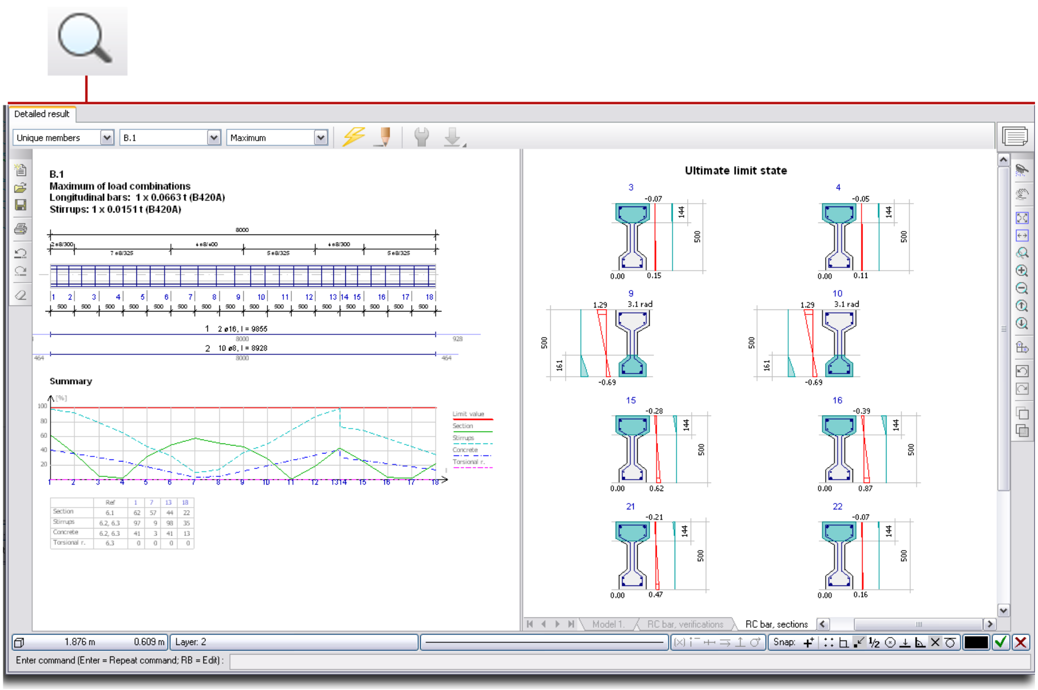


Figure: Detailed design result (RC bar design)

“Absolute” vs. “Relative” Maximum Value

The *Absolute maximum* and *Relative maximum* options of the display settings search the maximum values of a result component in all diagrams independently the negative and positive signs. *Relative maximum* finds the maximum value of the current diagram with its absolute value for the current load case/combination/group. *Absolute maximum* finds the maximum value of the same type results (e.g. shear force diagrams) with its absolute value for all load cases/combinations/groups.

Let’s see an example to understand the meaning of “absolute” and “relative” maximum values.

Let’s display the Tz shear force diagram of a frame structure in case of two different load combinations. The next table summarizes the positive and negative maximum values of both diagrams.

	Maximum values			
Display option	Positive	Negative	Relative	Absolute
Load combination 01	9.37	**-9.42**	**9.42**	**10.20**
Load combination 02	**10.20**	-8.67	**10.20**	**10.20**

Table: Maximum values of the same-type result diagrams (Tz)

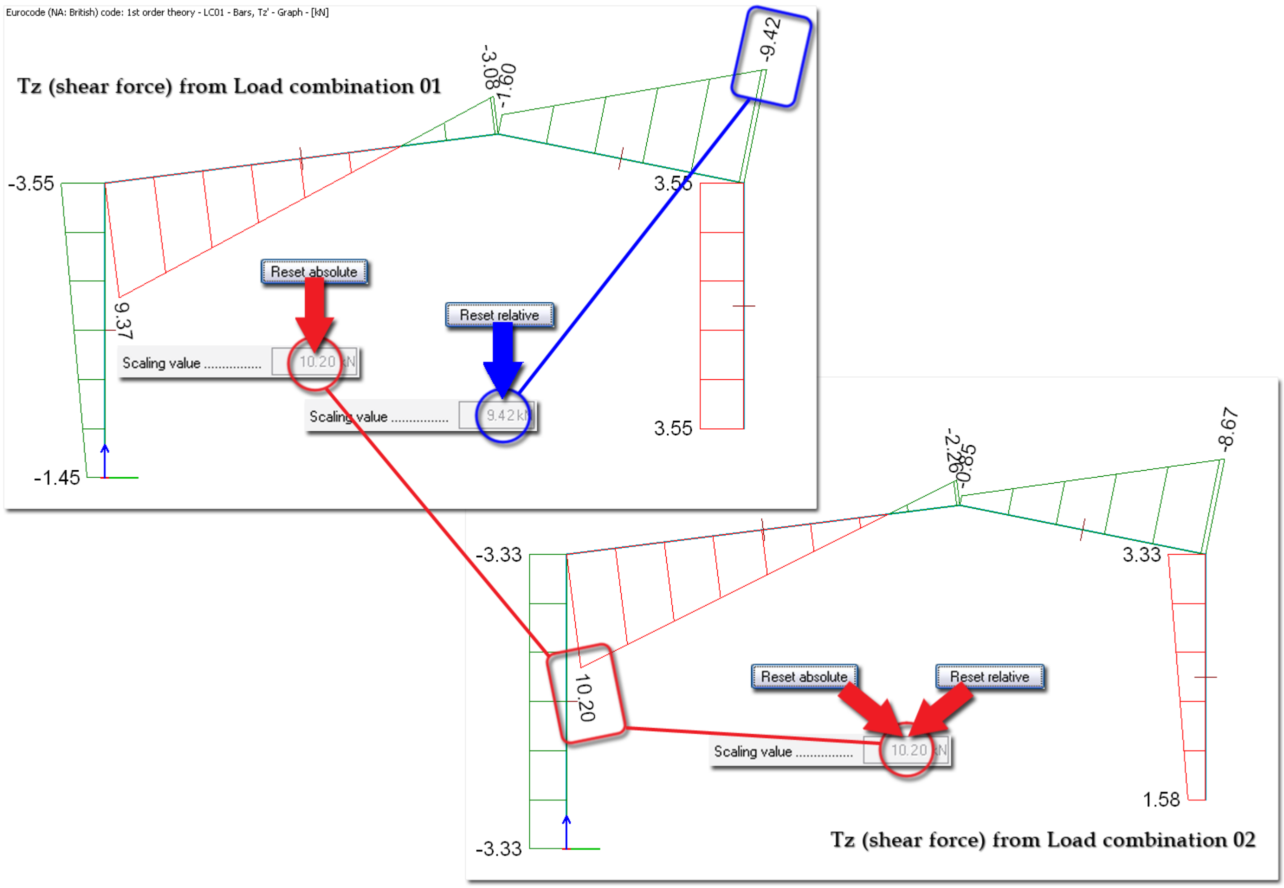
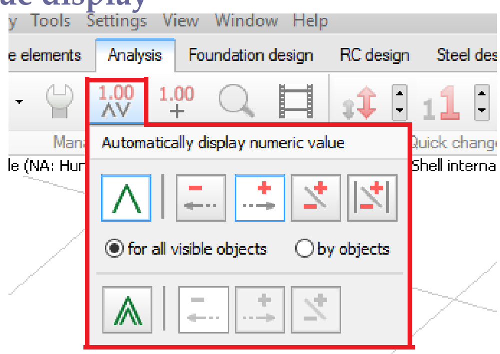


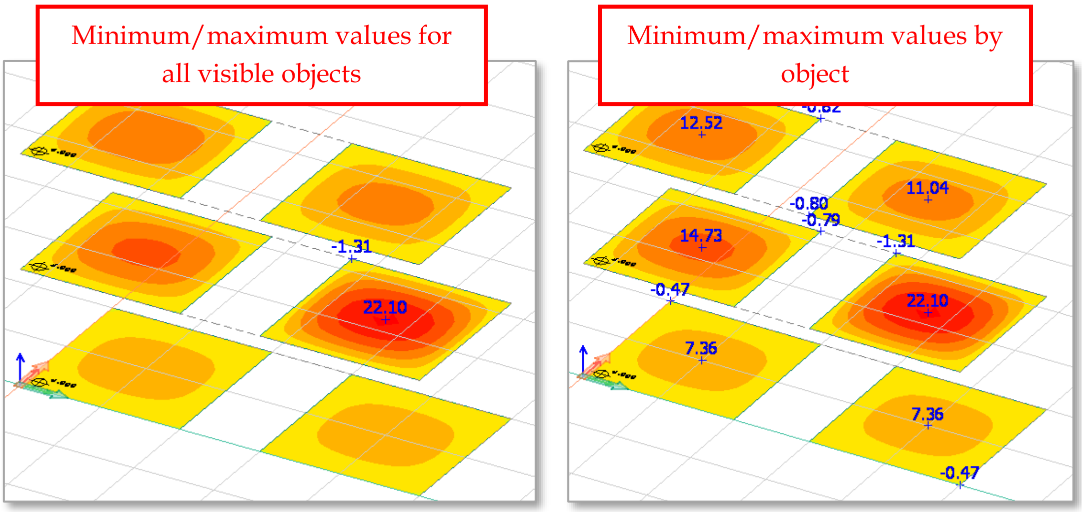
Figure: The values found by Reset absolute/relative

Automatic minimum and maximum numeric value display

The program is capable to show automatically the local or global minimum and/or maximum values of the displayed result. Automatically displayed values will be shown in blue, while values displayed by manual query remain black. It can be set to show the minimum and/or maximum values of all results across all visible objects, or local extreme values for visible each object.

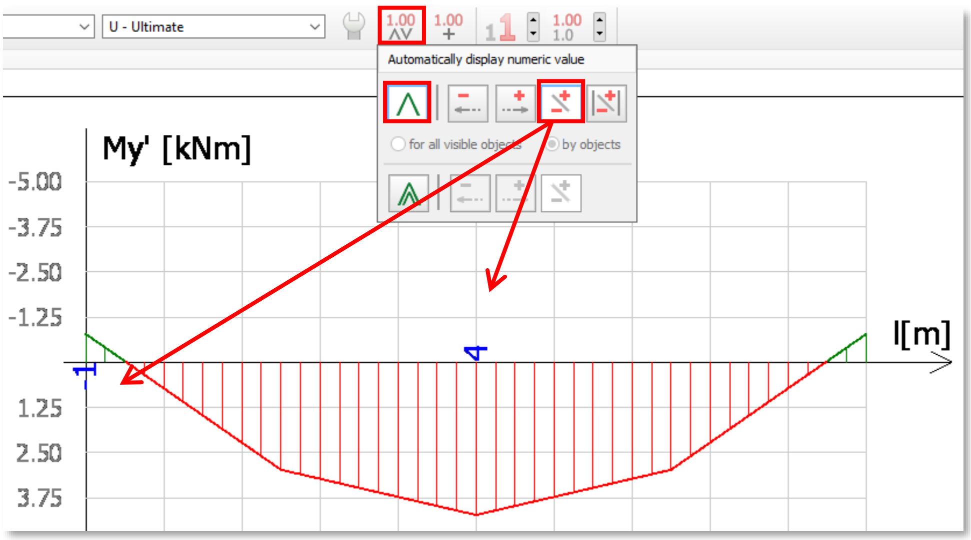


Let's display the my' moment results of a 3 storey building. The next figures shows the difference between showing the automatic results (minumum and maximum) for *all visible objects* and *values by object*. On the right figure the „by object" option choosed so the the program shows the results of the minimum and maximum values of each plates. On the left figure the „for all visible objects" option choosed, the displayed results will be the extreme maximum and extreme minimum values of all 6 plates.



When in storey view, 'All visible objects' means the objects on that storey

This is also available for each result (even for analysis detailed results, as shown below) where minimum/maximum can be displayed by the *Numeric value* function.



The last selected display settings for display automatic values are applied for all the new results, until they are modified by the user.

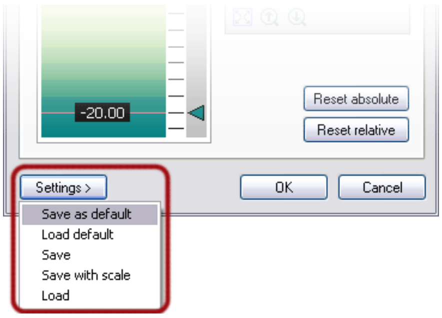
These settings are kept for the results when

- they are added to the documentation,

- they are hidden then shown again,
- the model is recalculated (naturally, only the display settings stay the same – the actual numerical values will be refreshed according to the new results).

Style Templates

Display settings (scale, colors, steps etc.) can be stored in files (so-called styles) by display technique or by figure. The saving and the loading facilities can be reached at the *Display options* tool. The styles are stored in files with different file extensions by display technique. The files can be swapped between users, so companies can create standard display setting for their own documentation style.



- **Save as default**

  It saves the current display settings as the default (without file name!) settings of the applied display technique. If you do not use a style (see later), the program always offer the last saved default settings when you apply the same technique inside one example or in further ones.

- **Load default**

  It restores the default display settings after any change.

- **Save**

  The color settings of the current dialog can be saved as a style in a template file. The file extension depends on the current display technique.

- **Save with scale**

  All display settings (scale, steps, colors etc.) of the current dialog can be saved as a named style in a template file.

- **Load**

  User-defined styles can be loaded from a file browser.

The following example helps in understanding the difference between *Save* and *Save with scale* after loading styles created by them. The color palette settings of an *Mx* internal force diagram was saved with *Save* and *Save with scale*. The figure shows the settings of *My* also displayed with color palette after loading (*Load*) the two saved styles. After loading "*Save*"-style, the palette contains the same colors but the scale is optimized for *My* values. Loading "Save with scale"-style results the same scale together with the same color information with *Mx*.

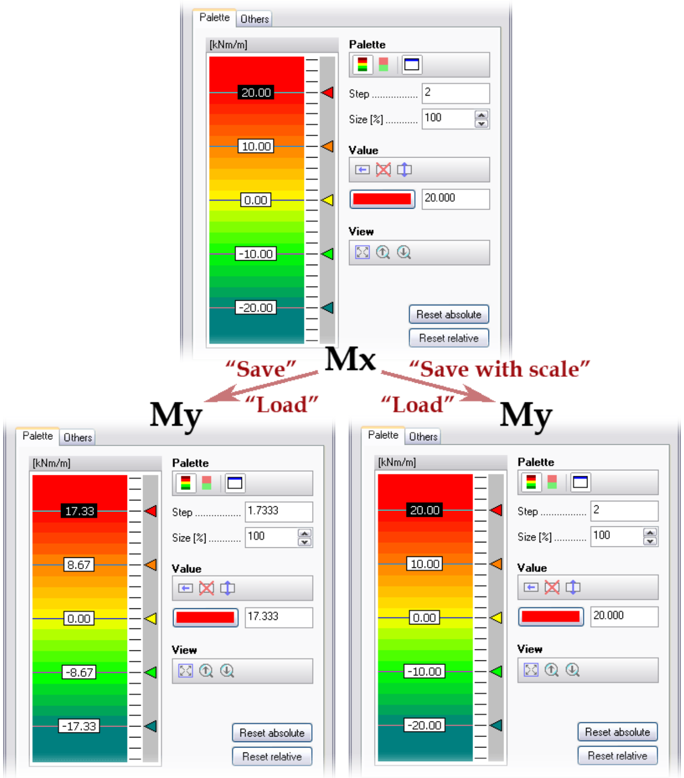


Figure: Save vs. Save with scale

Numeric Values

Tool Tip

Rolling your mouse over a result picture will display the numeric value in the current position and in the proper metrical unit as a tool tip. Tool tip works for all types of display techniques.

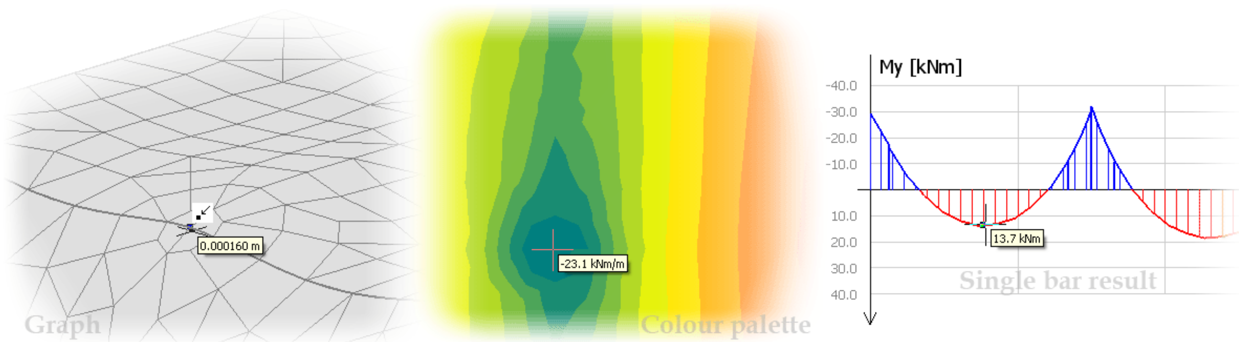


Figure: Numeric value displayed by tool tip

Value Labeling

Result diagrams can be completed by displaying the calculated numeric values. Use the  *Numeric value* tool to add labels to the currently displayed result figure.

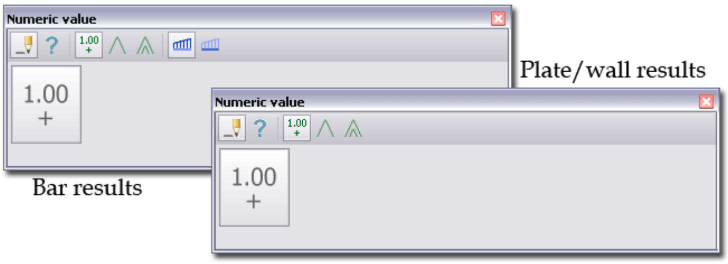


Figure: Tool palettes of Numeric value

Depending on the currently displayed result figure and the object type – you would like to ask numeric values – the tool palette of *Numeric value* contains different definition tools. With the *Define* tool, new value labels can be placed on the current figure.

The color of the numeric values can be set with the “Current color” button of the Status bar. Of course, you can place values with different colors on a result figure. For example the maximum values can be displayed in red while the others with black.

Set the text settings of the numeric values at 1.00 + *Default settings*.

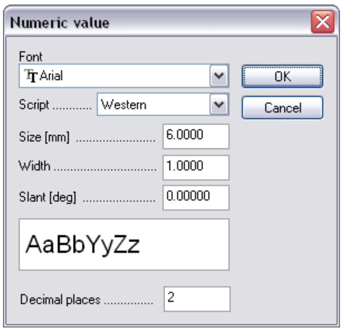


Figure: Numeric value settings

The next figures give examples for the tools of *Numeric value*.

Use 2.15 + *Select* to place numeric values in clicking points of the displayed result. With tool, result maximum, minimum and absolute maximum/minimum can be found for selected object(s). With tool, all result limits (maximum and/or minimum) can be displayed for selected object(s).

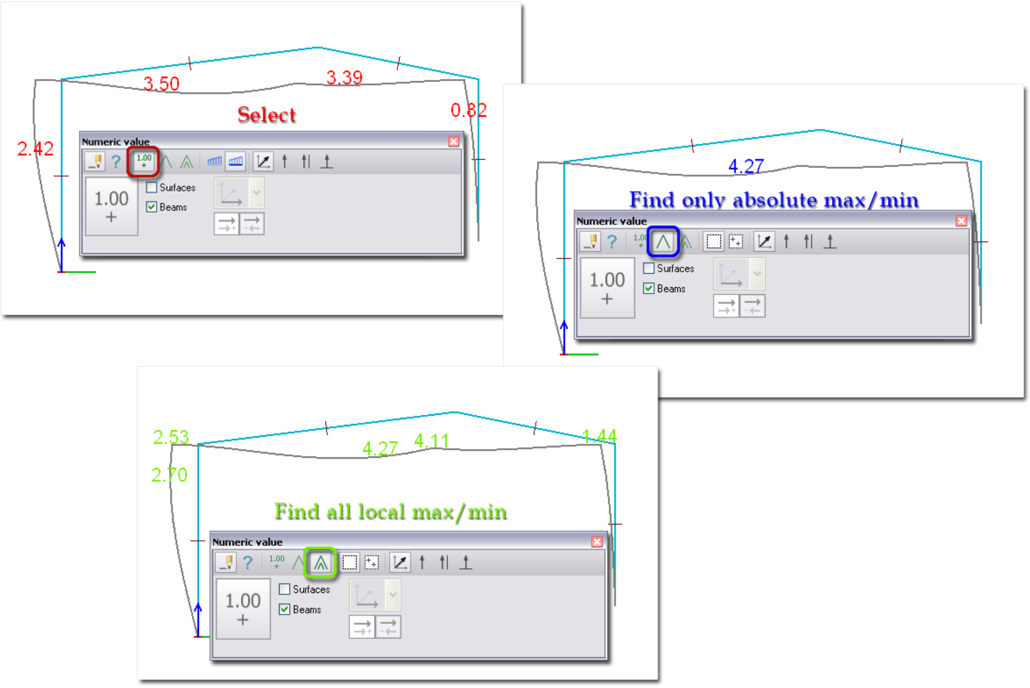


Figure: Arbitrary and maximum values (bar displacement)

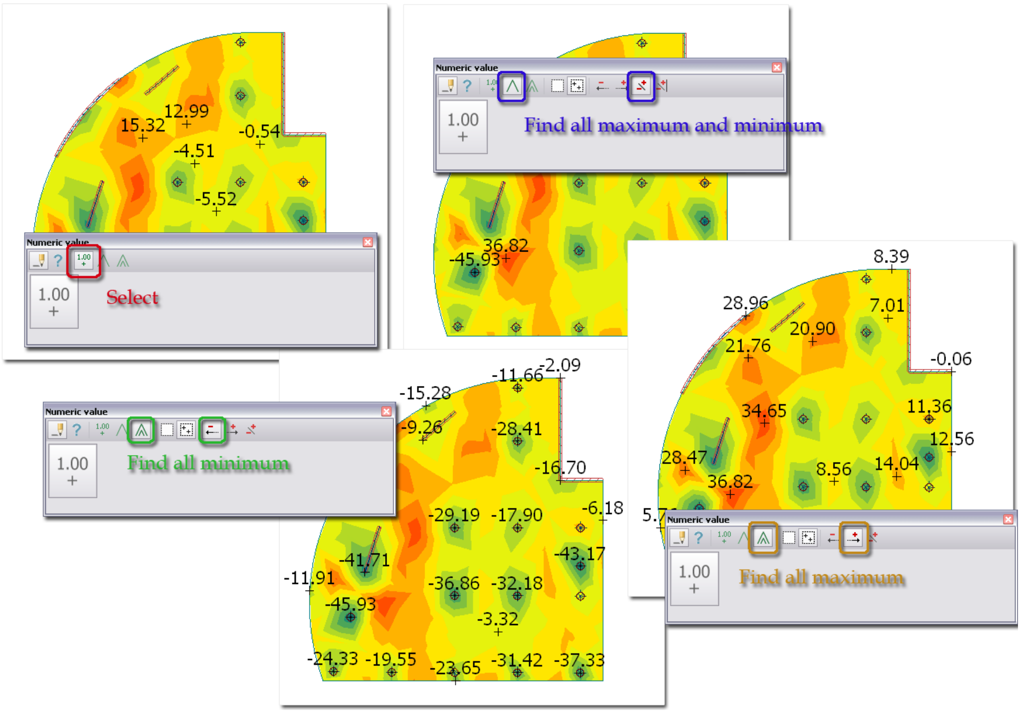


Figure: Arbitrary and maximum values (plate internal forces)

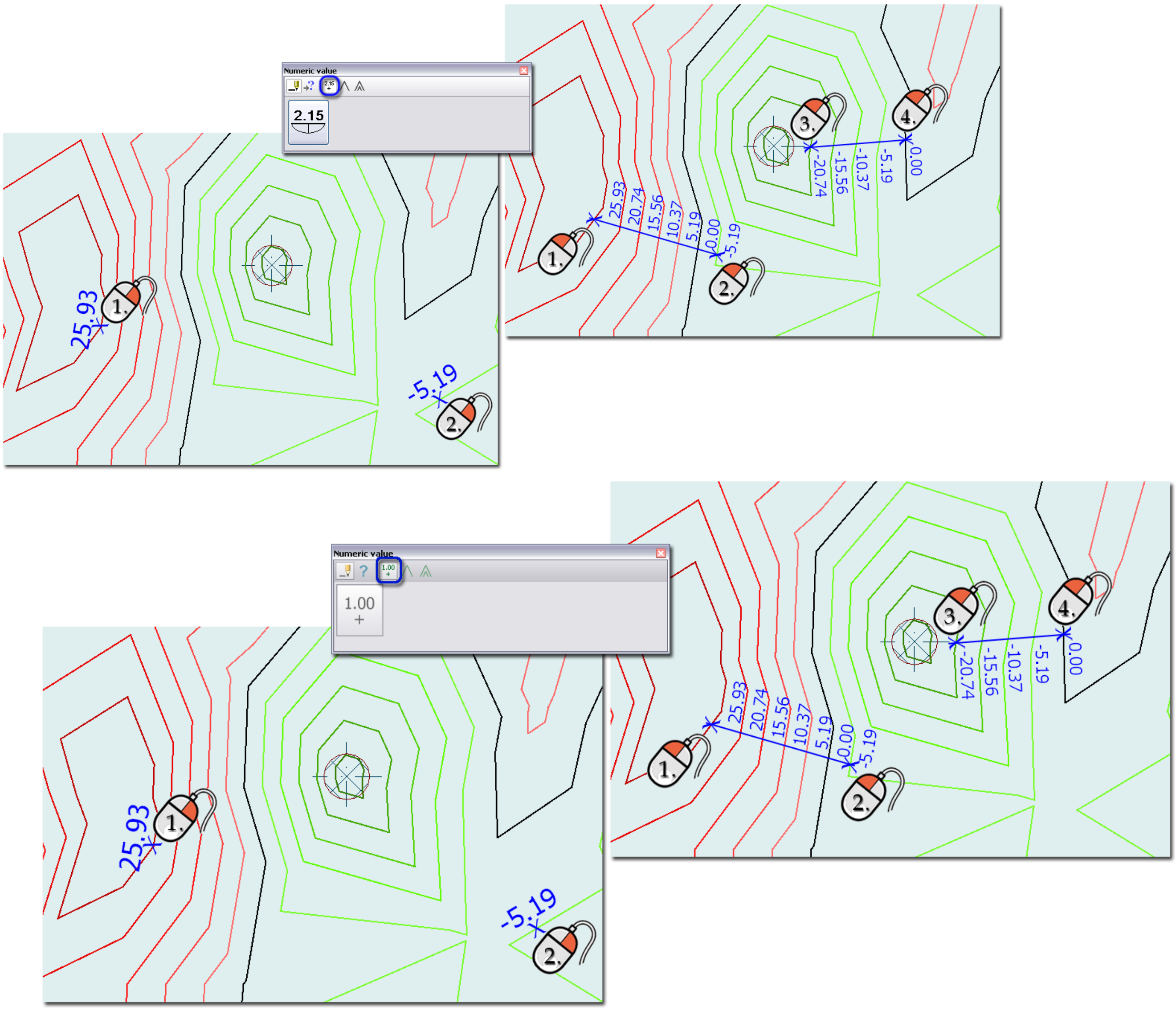


Figure: Display numeric values (with Select) on a Contour palette result

In case of bar displacement, points can be selected for displaying numeric values on bars (*Original shape*) or on the *deformed shape* of the bars.

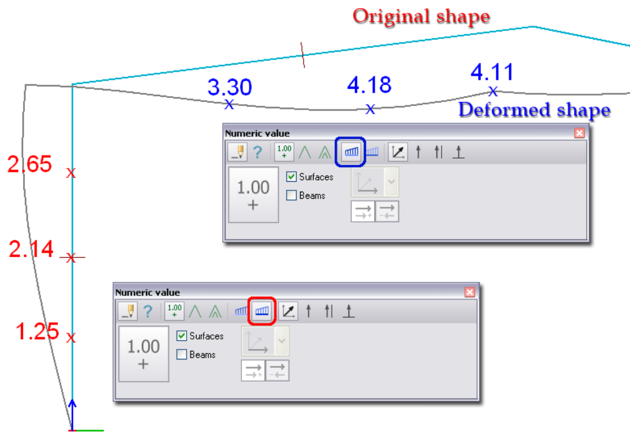


Figure: Numeric values displayed on the original and deformed shapes

Bar displacement can be asked with its real size (*Vector size*) or by its component (parallel with the global/local x′/y′ or a given direction).

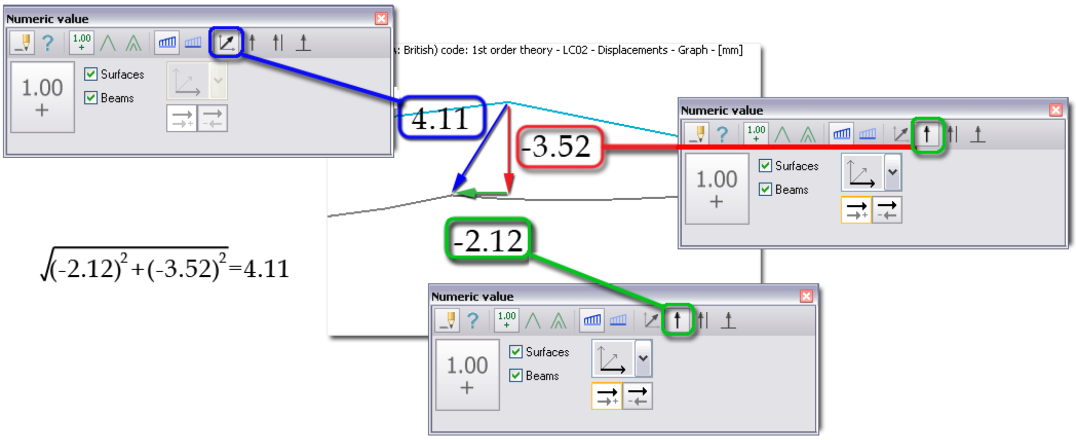


Figure: Numeric values by displacement component

The display settings of predefined numeric values can be modified by the  *Properties* tool.

The position of the text labels can be modified independently from their insertion (clicking) point with *Move* (*Edit* menu).

Editing Results

The settings of the currently displayed result can be modified with the *Display options* tool any time. The content of the setting possibilities (colors, scale, palettes etc.) depends on the display technique used for the visible result.

Above *Display options*, quick change tools give fast and real-time modification in scale/color distribution (step) of the current result figure and in size and decimal numbers of displayed numeric values. Quick change tools work for all type of display techniques. The next figures show some examples.

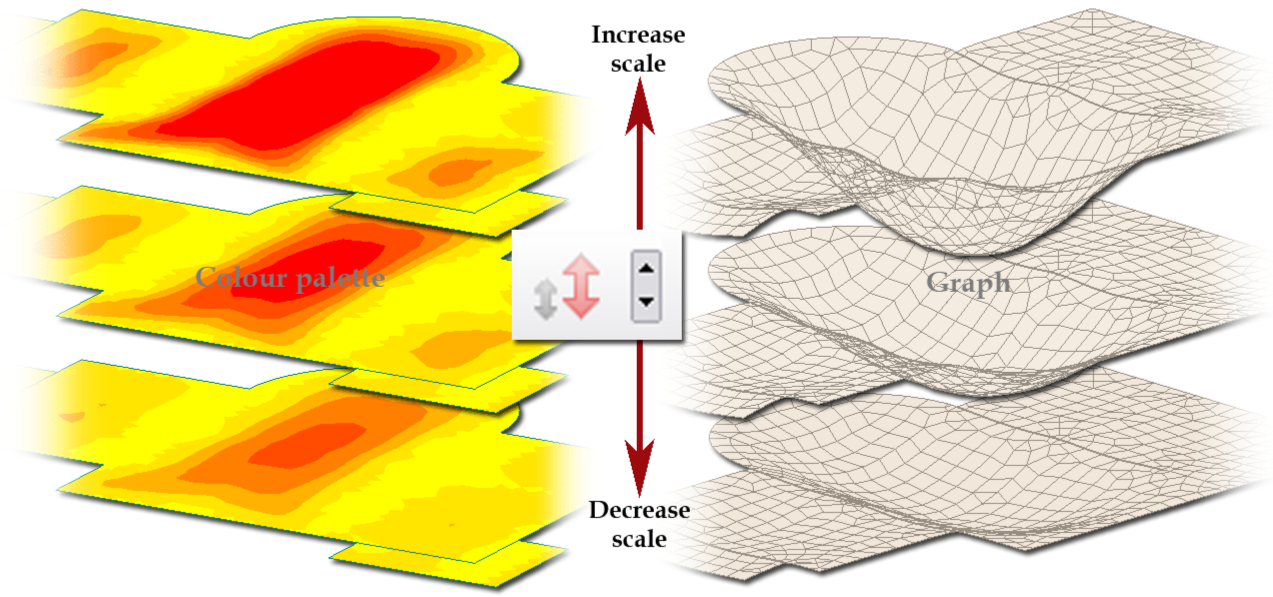


Figure: Modifying diagram scale

Scale increases/decreases the scale (e.g. graph, section) or the step distribution (e.g. contour lines, color palette) of the currently displayed result figure with 10% percent. Use the "+" or "–" key of the keyboard's numeric pad as the fast key of *Increase scale* or *Reduce scale*.

The numeric values are refreshed automatically when the display options of a result are modified.

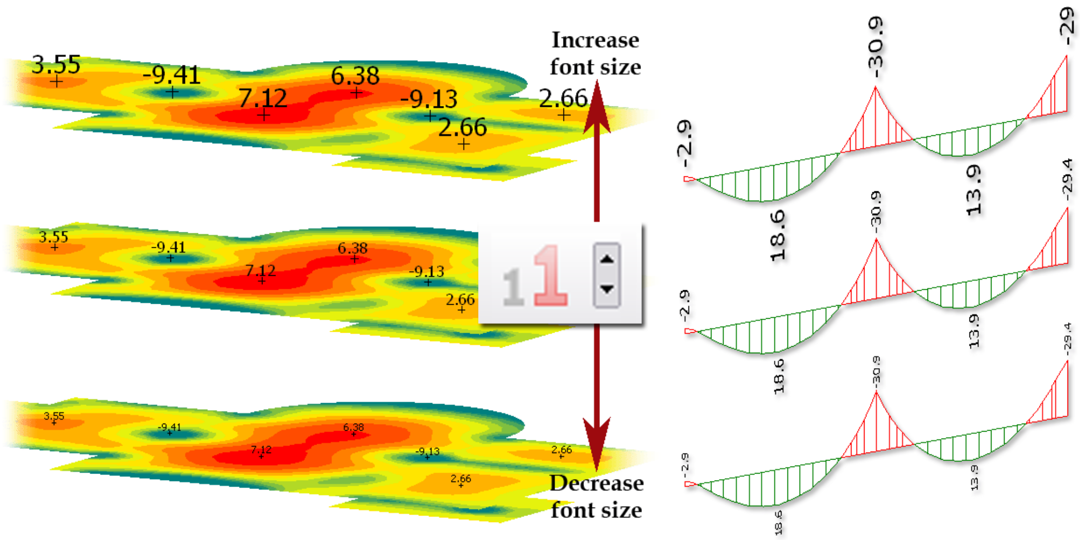


Figure: Modifying the text size of numeric values

Font size modifies the font size of the numeric values with 10% of the initial size.

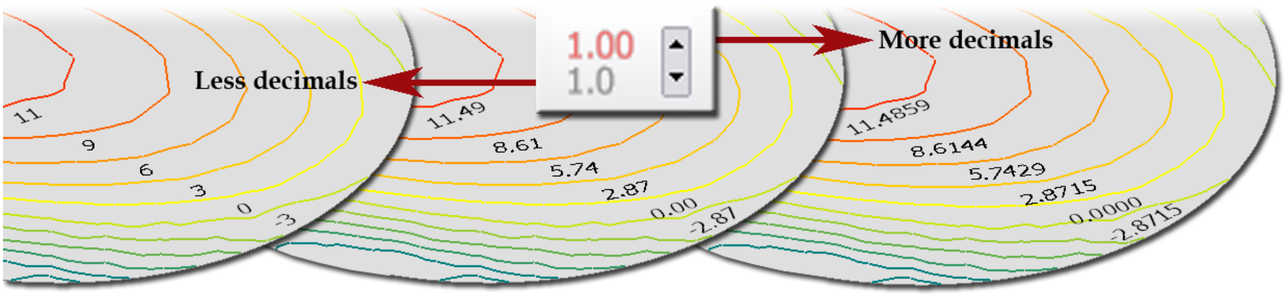


Figure: Modifying the decimal numbers of numeric values

Browsing Results

A one-click browser (*Quick selection*) is developed for changing between result figures displayed earlier on the screen. By clicking on "▼" symbol next to  *Select result*, a list appears with titles of results displayed so far. Just click on the result you wish to display again. The content of the list can be organized with *Select result*: hide current figure or delete items.

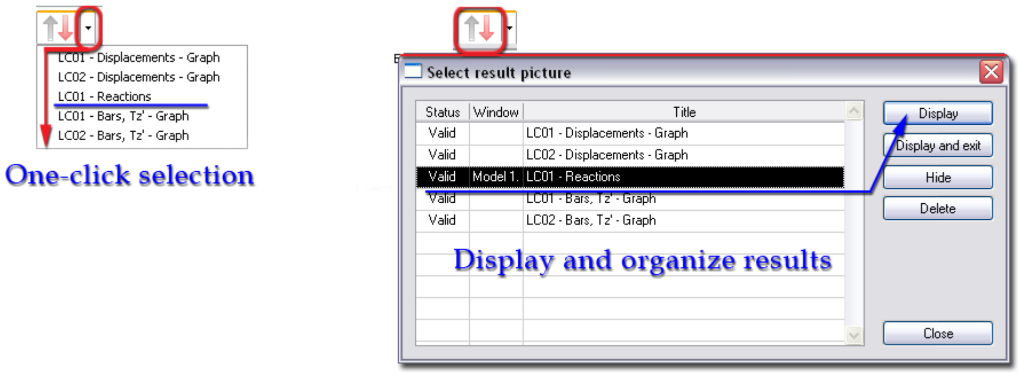


Figure: Quick or dialog browsing among previously displayed results

Animation

The currently displayed results can be animated with the *Animate result* tool. Not only for bar results (such as dynamic and stability results, displacement, internal force/stress diagrams etc.), but all result display types of planar elements (such as displacement, internal forces, RC results, utilizations etc.) can be animated. That means graph points are moving, the colors and color intervals are pulsing etc. Clicking *Animate result* starts the animation after some seconds (depending on the model size) by continuously displaying the linear interpolation statuses between the initial and the final states of the current result figure. Press any key or click by mouse the stop animation.

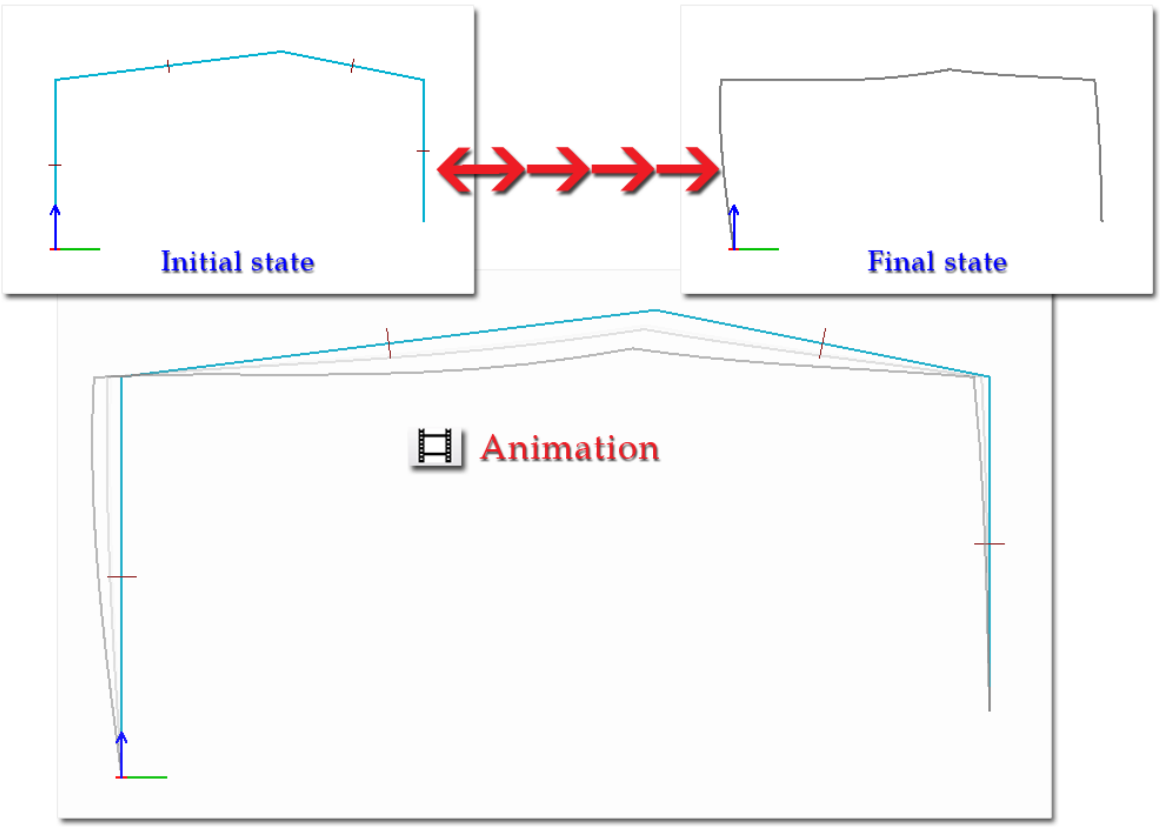


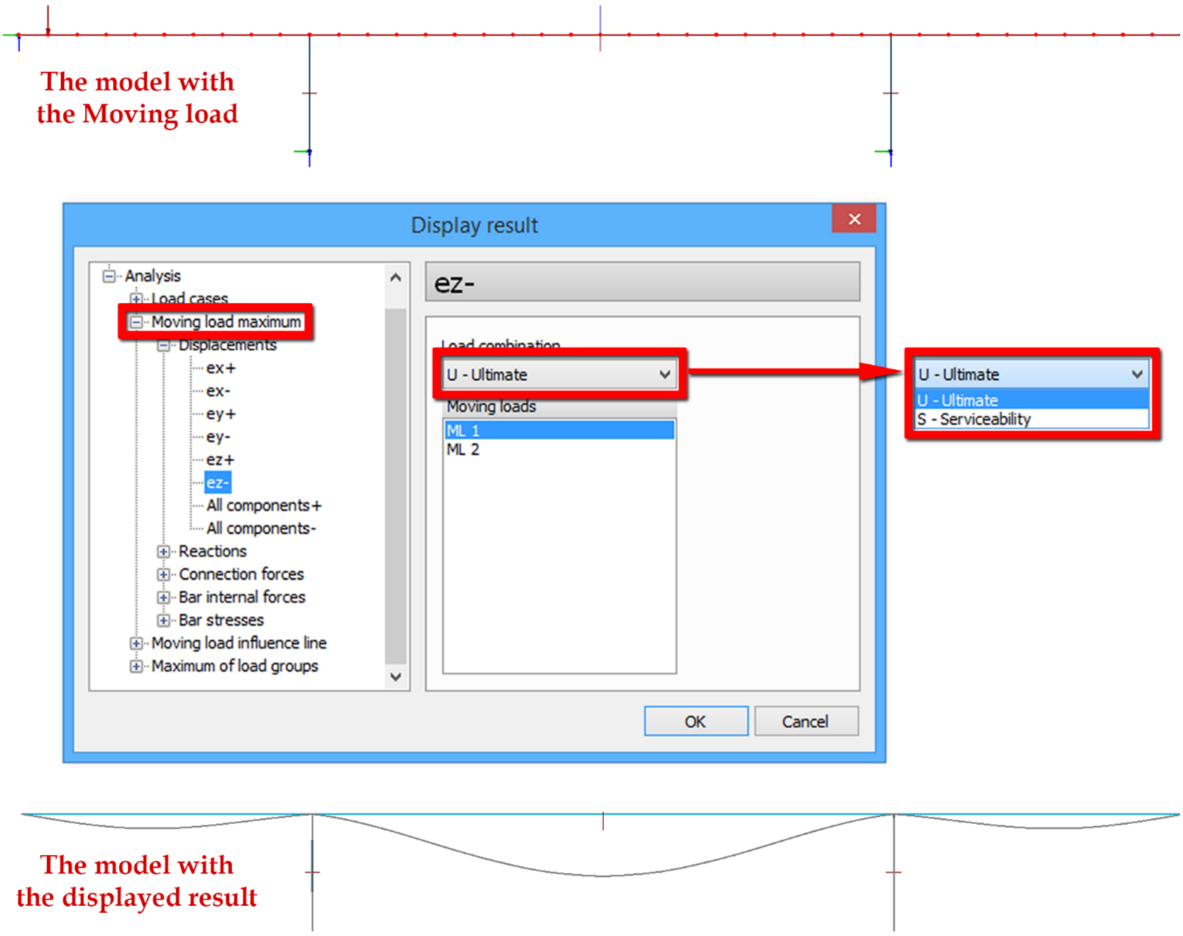
Figure: Animation of a frame displacement

Moving load results

Special results can be displayed for Moving loads, described below.

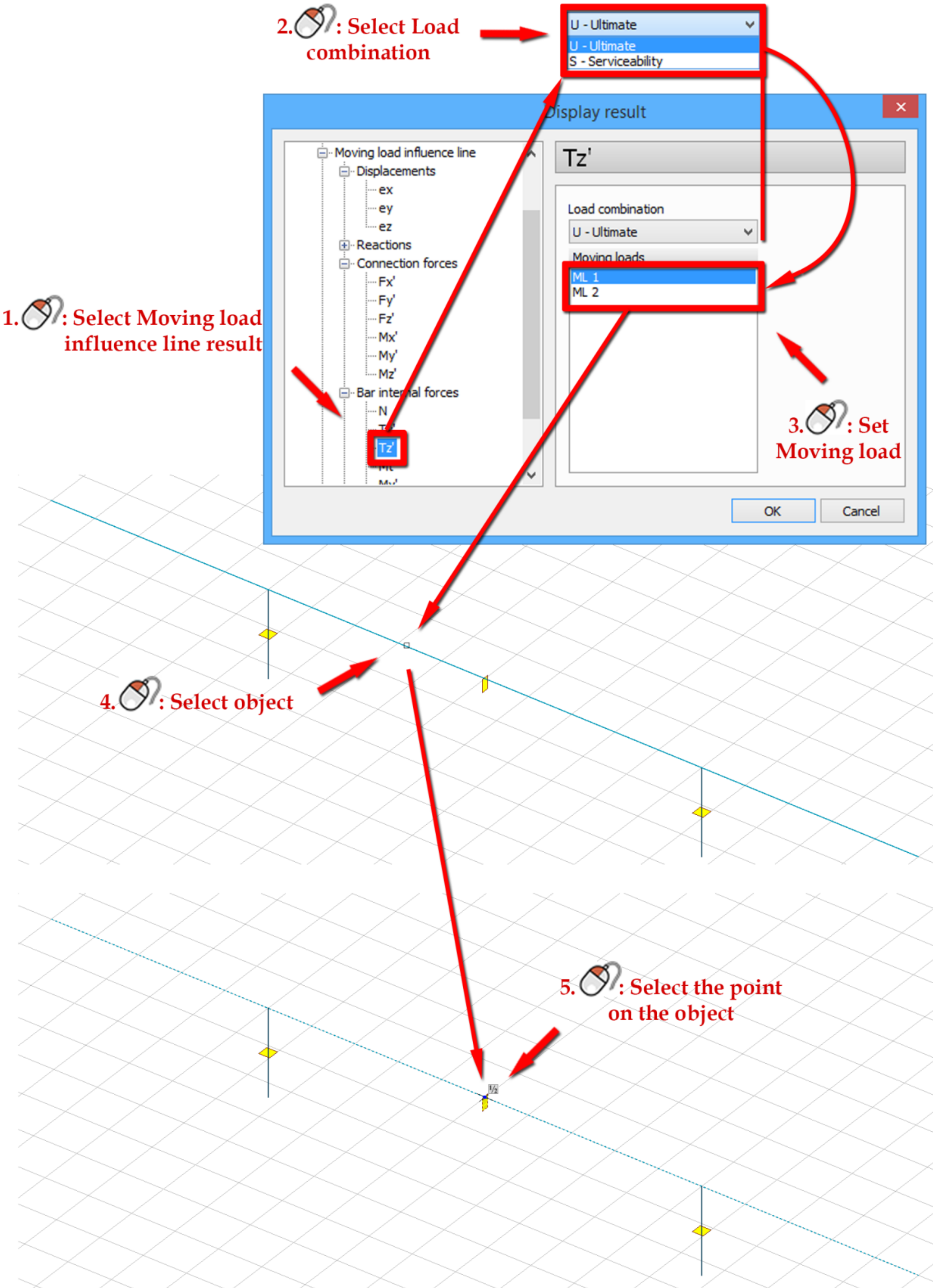
Moving load maximum

Moving load maximum is similar to Maximum of load combinations. It shows the maximum values of the selected result in all points of the model, under the effects of a Moving Load. User can change the Load combination and the Moving load of which results he or she wants to display.

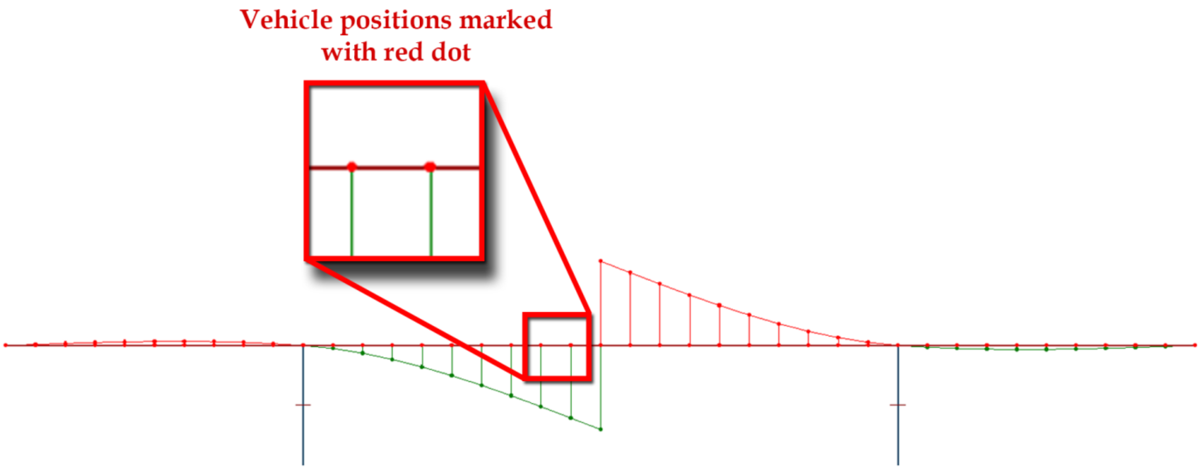


Moving load influence line
The Moving load influence line shows how the value of the selected result changes in a user-defined point of an object, while the Vehicle moves along the path.

User can define it with the following steps:



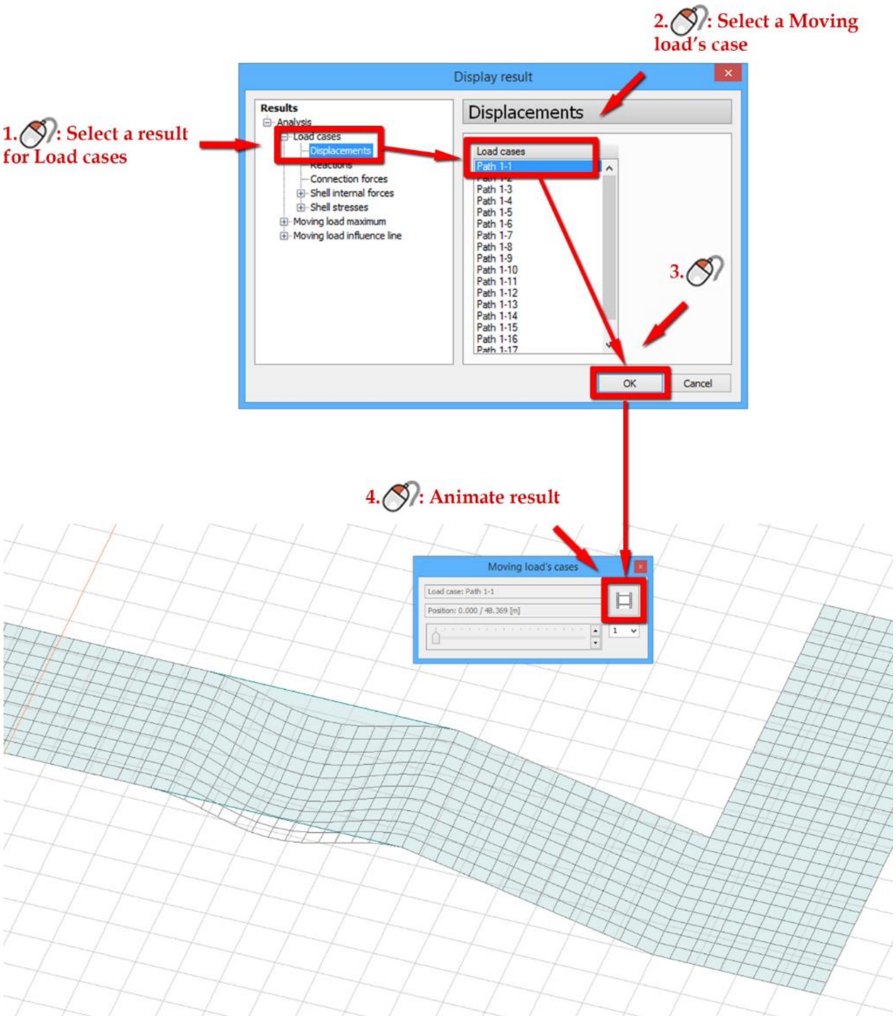
The results are displayed similarly to the one in the figure below:



⚠ Since Moving loads are based on singular load cases, these result values are exact only in the Vehicle positions. The other points of the curve show approximate values. Accuracy can be increased by defining more Vehicle positions, however, it increases calculation time, too.

Moving load's cases

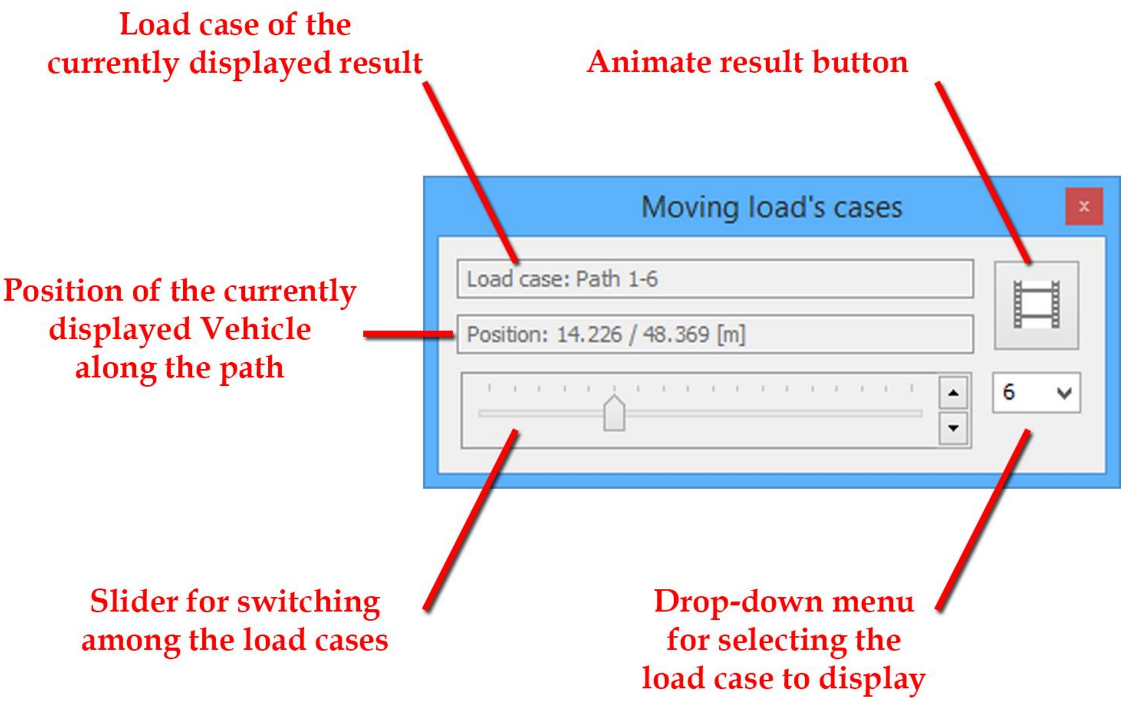
Although Moving loads are based on singular load cases, they are managed differently than regular load cases. It is possible to create an animated result for a Moving load. The figure below shows the method:



The animation displays the changing of the selected result while the Vehicle moves along the path.

The next figure describes the new Moving load's cases dialog:

: Switch among Moving load's cases with mouse scroll while the dialog is active



⚠ Animate result button in the Moving load's cases dialog has a different effect than the regular Animate result button in the Analysis and the Design tabs.

Documentation

This chapter summarizes the documentation possibilities of FEM-Design projects, models and results. It introduces the printing and *listing* (summary tables) functions and the automatic documentation based on templates (*Documentation module*).

Printing

The full content or a selection part of the screen (*Drawing area*) can be printed out with the *Plot* command (*File* menu). Just click with to send the entire screen to print preview, or define a selection box with to set it content as the plotting input.

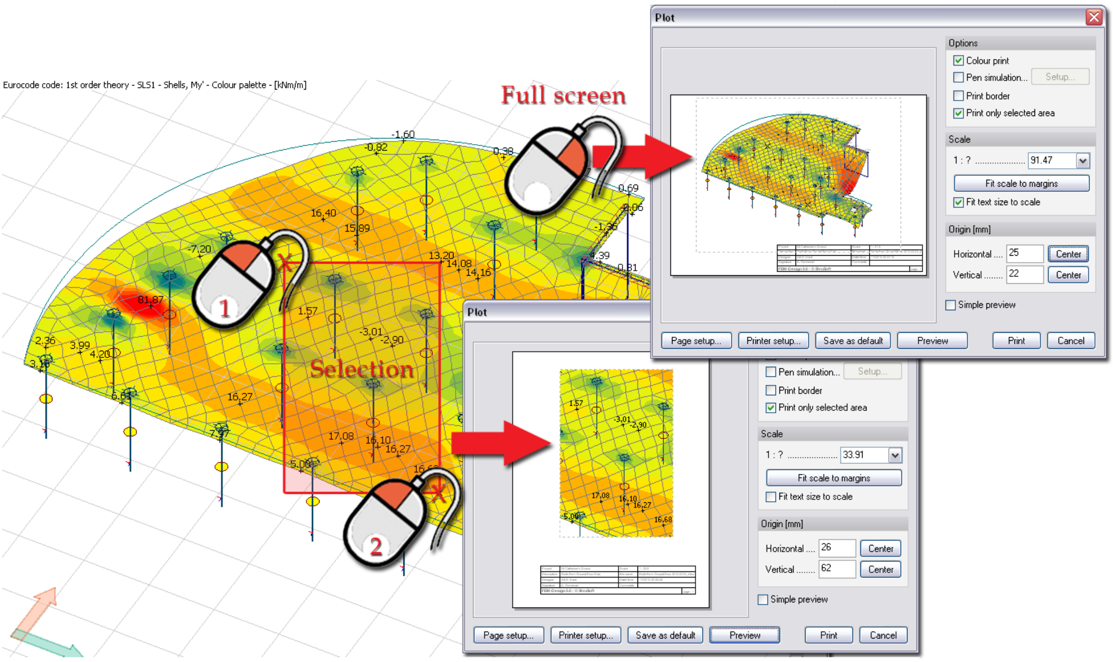


Figure: Plotting screen (entire or selected part)

The printing settings are the following:

- **Settings dialog style**

  With *Dialog* and *Preview* buttons you can switch between the dialog box and the full screen mode of the plotting settings.

- **Printer setup**

  The goal printer can be chosen from the available (installed or network) printers and plotters. PDF creator (not included by default) can be also set here to save the screen content as PDF. The printer or PDF settings are available at *Properties*.

- **Page setup**

The size of the paper and the margins together with the paper orientation can be set here in a separate dialog. Title label can be added to the print area. Check the box "*Info label*" to add title label to the bottom right corner of the print area. Set the page number and set the text size and the width of the label at *Setup*. The label contains custom text set by *Settings* > *Title* and autotext such as the file name and date modified.

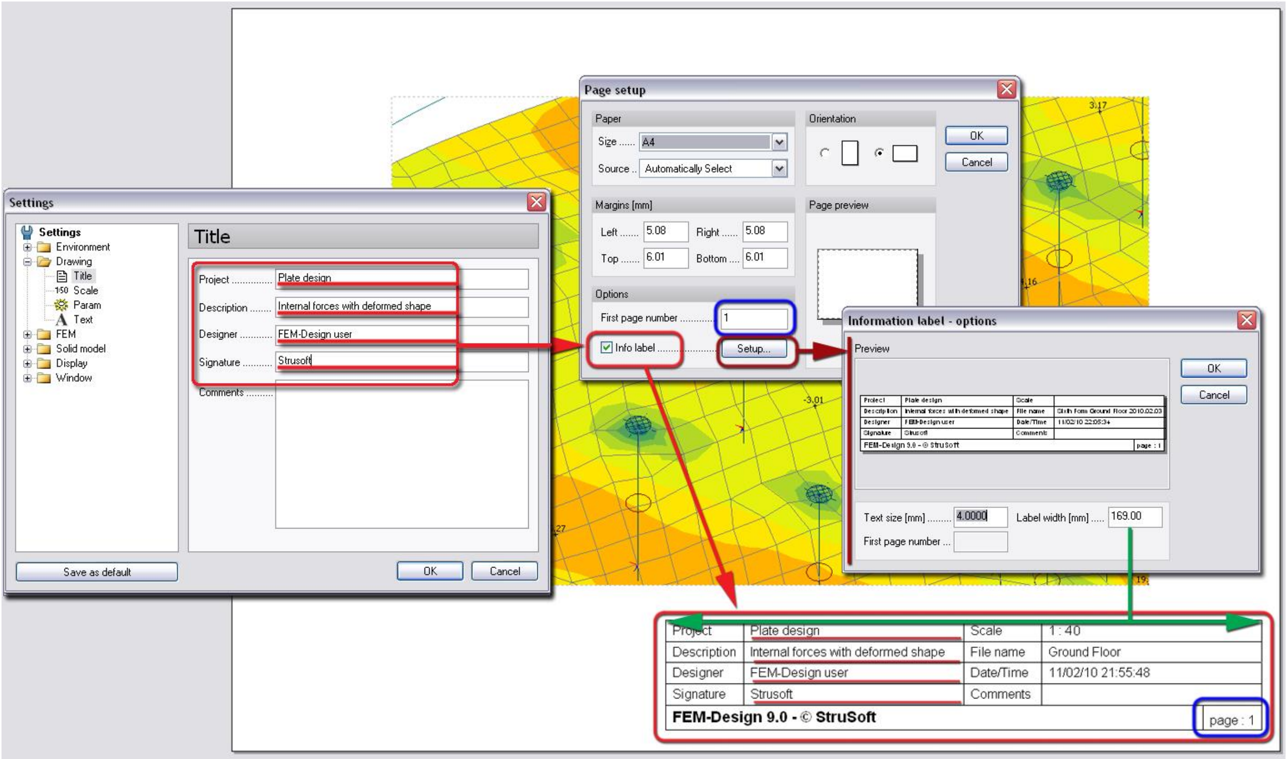


Figure: Title label as header

If you would like to print out only the content of the *Title*, place it on the drawing area with *Draw* > *Title information table*, add it to the print area selection and inactivate the *Info label* option at *Page setup*.

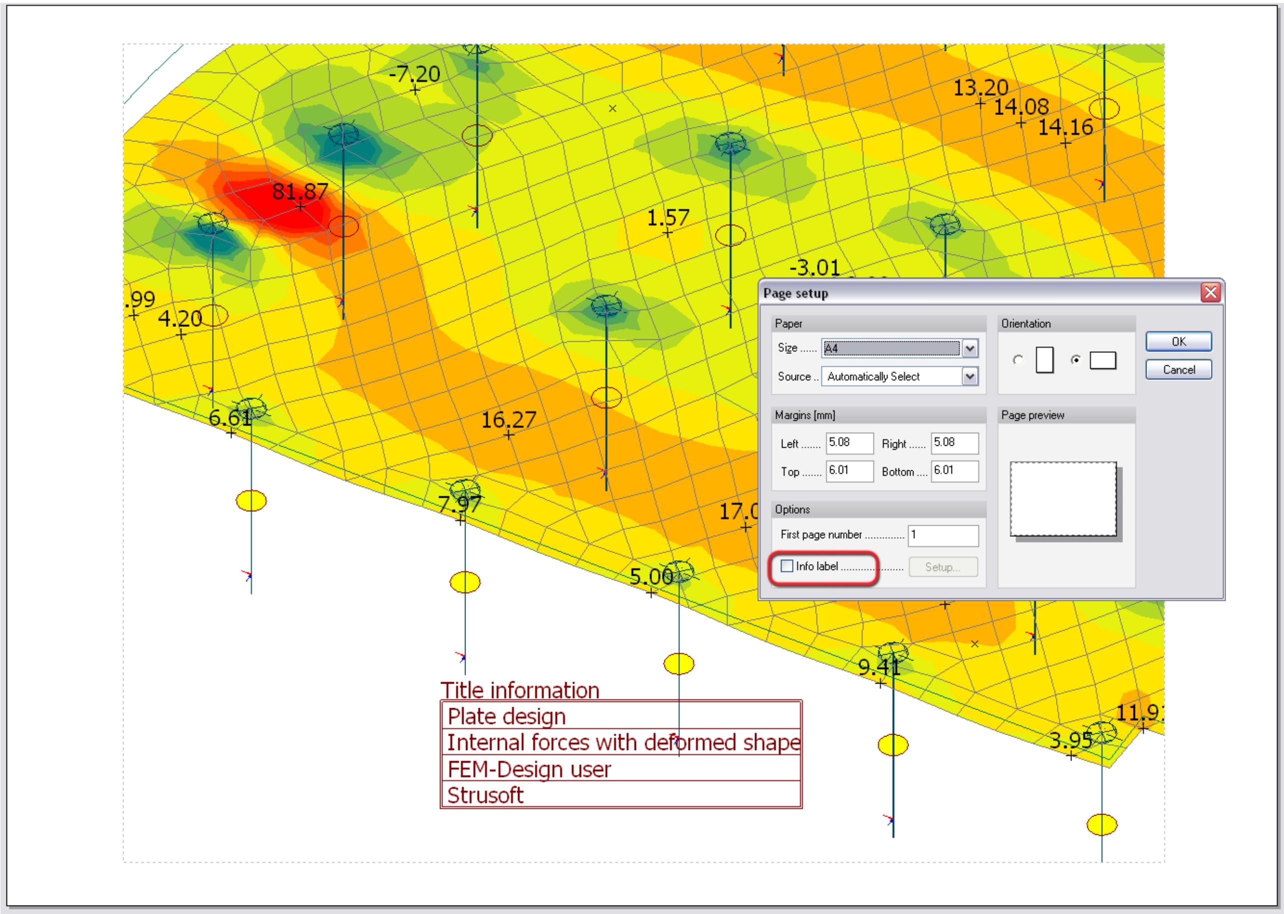


Figure: Title label as drawing element

- **Color print**

  Switch between grayscale and color (seen on screen) printing.

- **Pen simulation**

  Different pen width values can be assigned to colors at *Setup*.

- **Print border**

  Rectangle border fit to paper margins (*Page setup*) can be shown and set for printing out.

- **Print only selected area**

  If this check box is active, only the selected area (in case of selection) will be printed otherwise the its environment (the whole content of the drawing area).

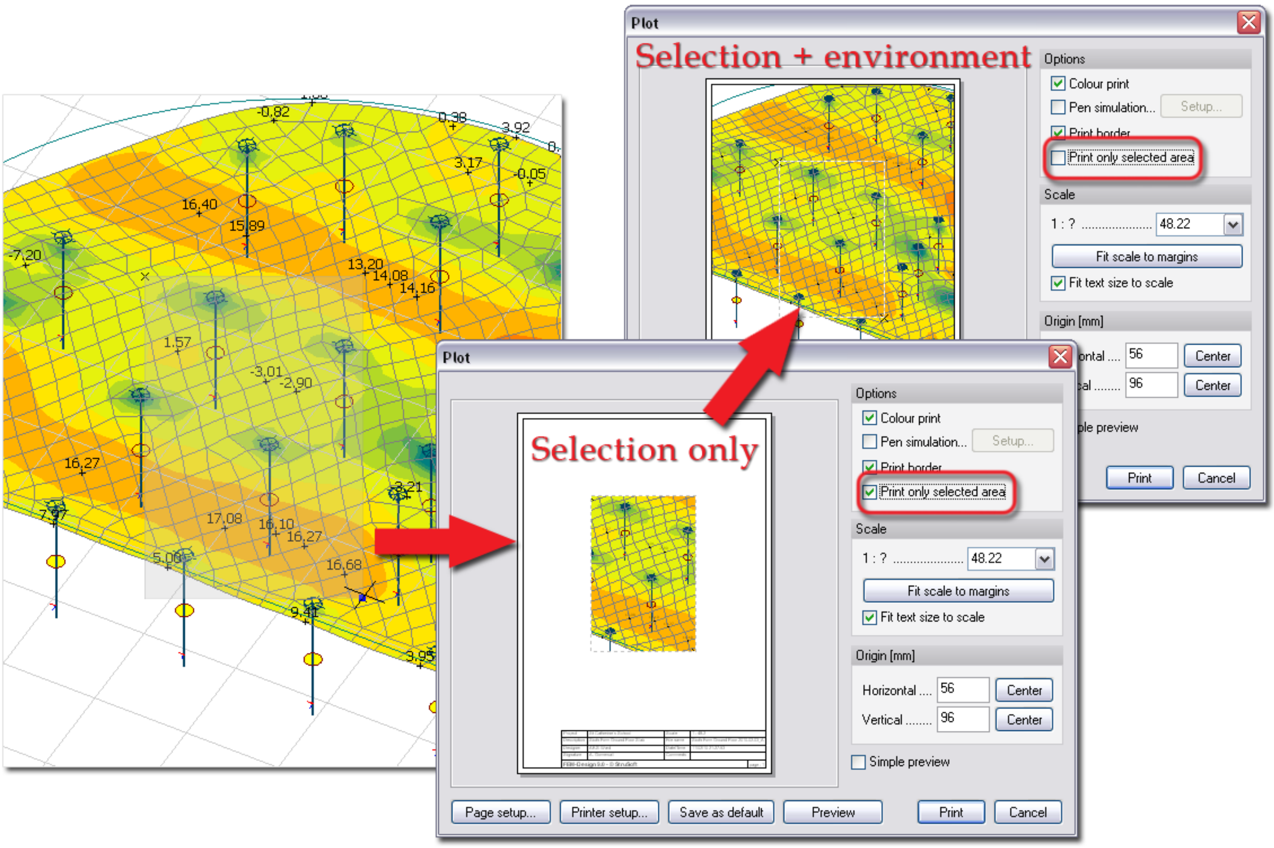


Figure: Printing selection

- **Scale**

  Set the printing scale manually or select one from the list. Dashed border marks objects (selection size) to print and helps in setting the scale. *Fit scale to margins* option sets the biggest scale of the selected print area according to the paper size, margins and the origin (see later). *Fit text size to scale* option sets the scale of the texts according to the drawing scale (*1:?*) or keeps the original size of the text in case of its inactive status.

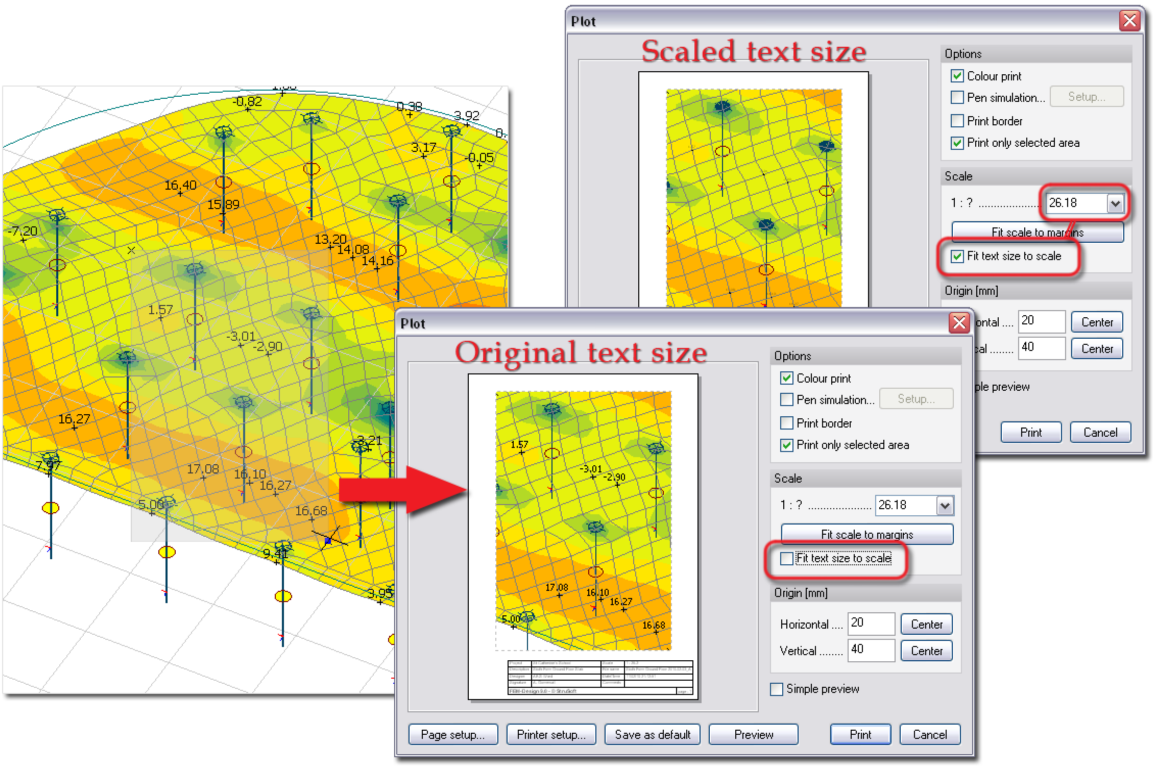


Figure: Printing text size

- **Origin**

  The position of the selected printing area can be set with its distance from the origin. *Horizontal* defines the distance of the bottom-left corner of the printing area from the left margin and *Vertical* sets the distance of the bottom-left of the printing area from the bottom margin. *Center* buttons fit the printing area to the middle of the page area bordered by the margins. The printing area can be dragged by the mouse also on the symbolic white page.

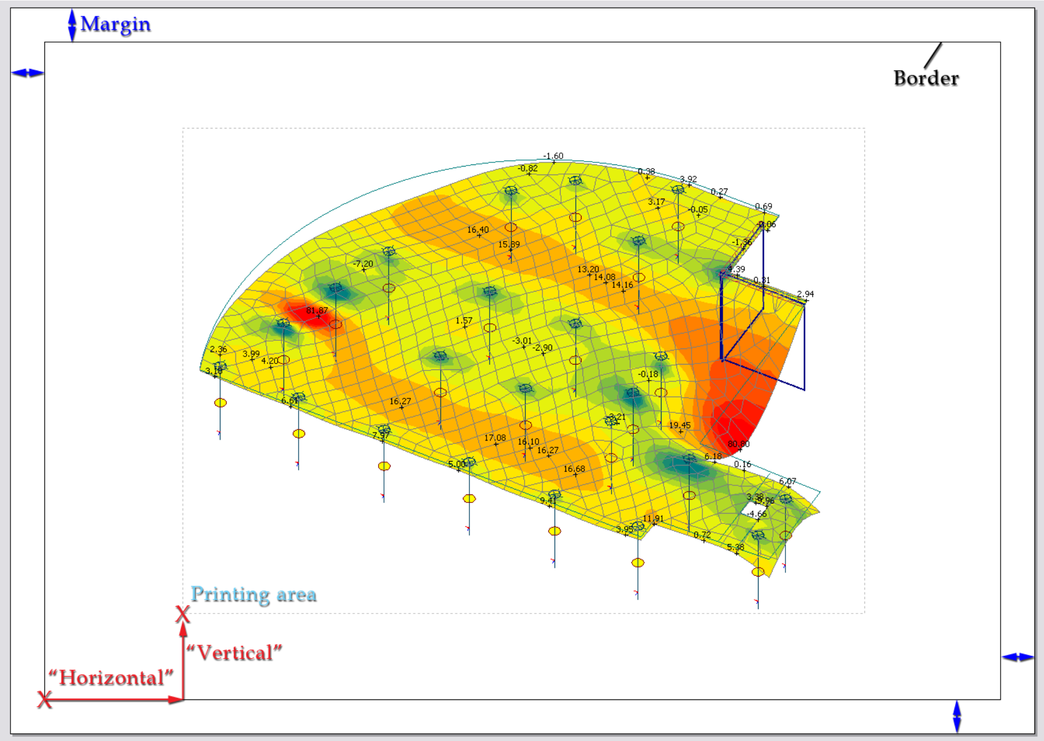


Figure: Printing area position

- **Simple preview**

  It is generally a problem to plot big models; setting the final position of drawings on the paper takes time. By activating *Simple preview*, the program shows only the frame of the selected printing area to reduce the generation time of the drawing when setting its position (*Origin*) and *Scale*.

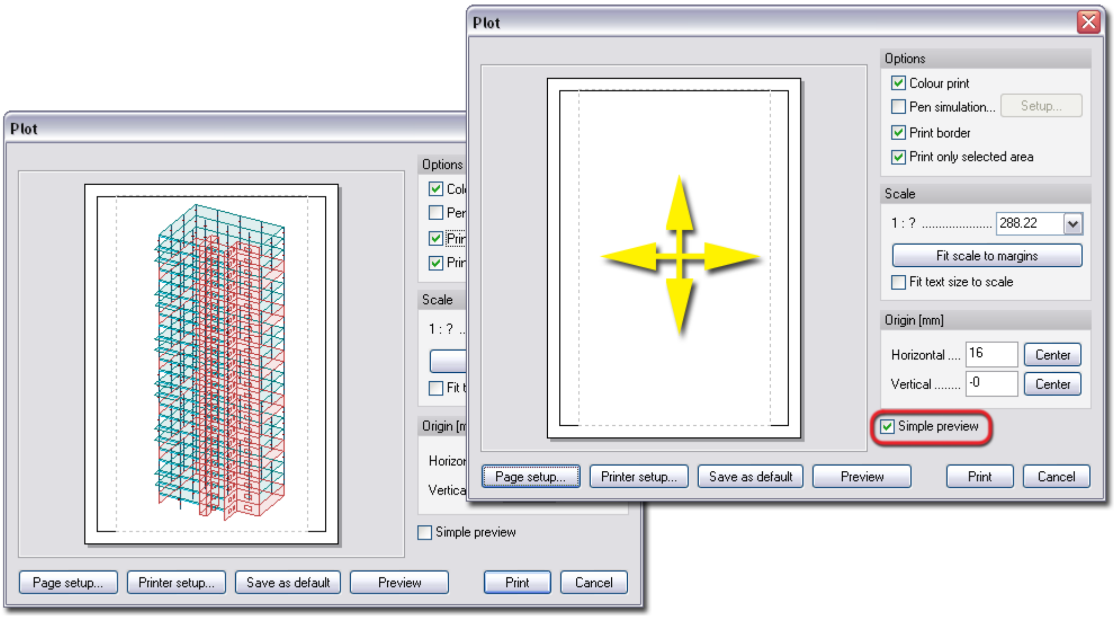


Figure: Simple preview of the printing area

- **Save as default**

It saves the paper and color printing settings as the default plotting settings for later use and next projects.

Click *Print* to start printing. A small dialog appears, in which you may cancel/stop printing.

Quantity Estimation

Just one click on *Quantity estimation* (*Tools* menu), and a fast process collects all structural elements of the current project and lists their materials, identifiers, main geometric parameters (e.g. profiles/thickness), quantities, reinforcement diameter etc.

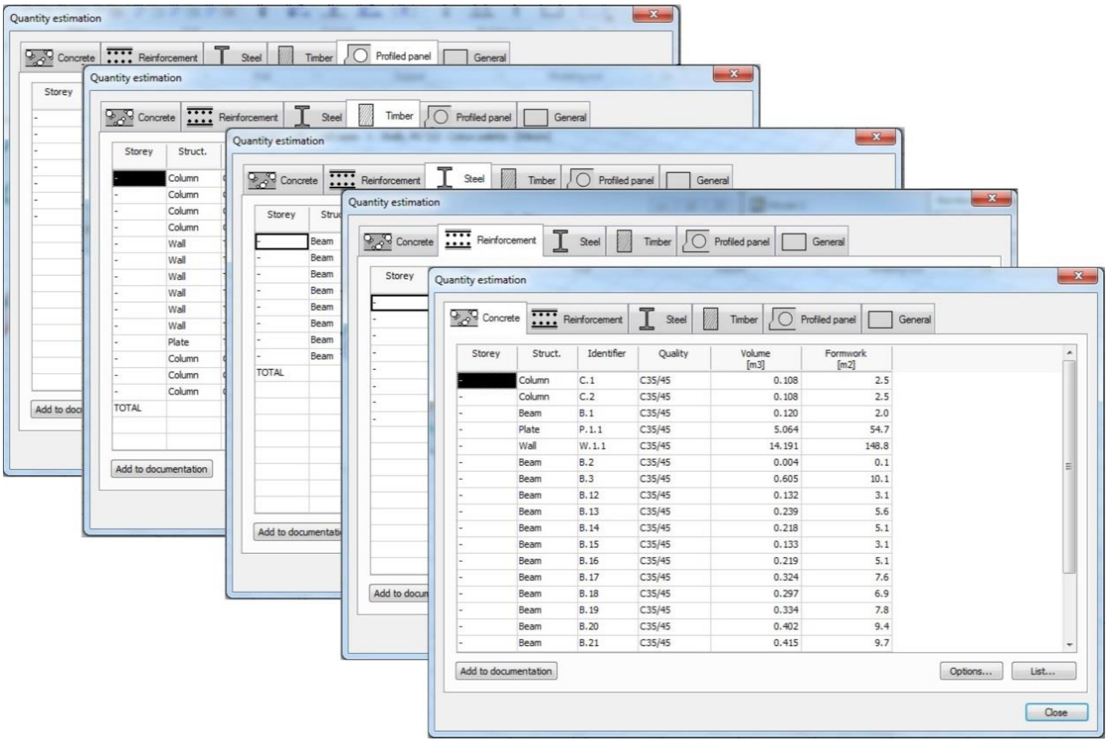


Figure: Quantity estimation

⚠ Reinforcement quantities are displayed after *RC design* only.

Categorization by priorities and summary by certain properties are also available in the quantity estimation tables (*Options*).

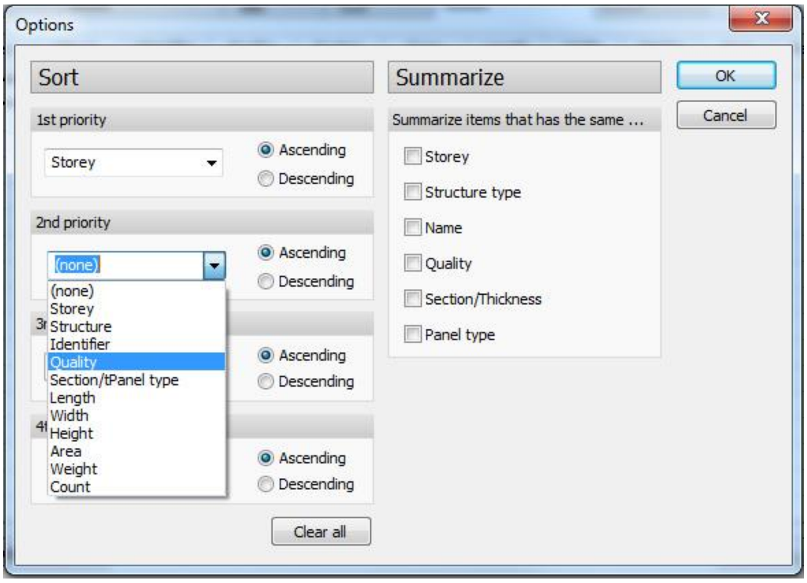


Figure: Categorization and summary features

The currently displayed table can be attached to *Documentation* by clicking *Add to documentation*.

The results of quantity estimation can be printed out directly as tables with the *List* option. (Other summary tables such as structural, load, finite element input and analysis/design result outputs of the current project can be also listed here. You may also use the *List* command of the *Tools* menu any time.)

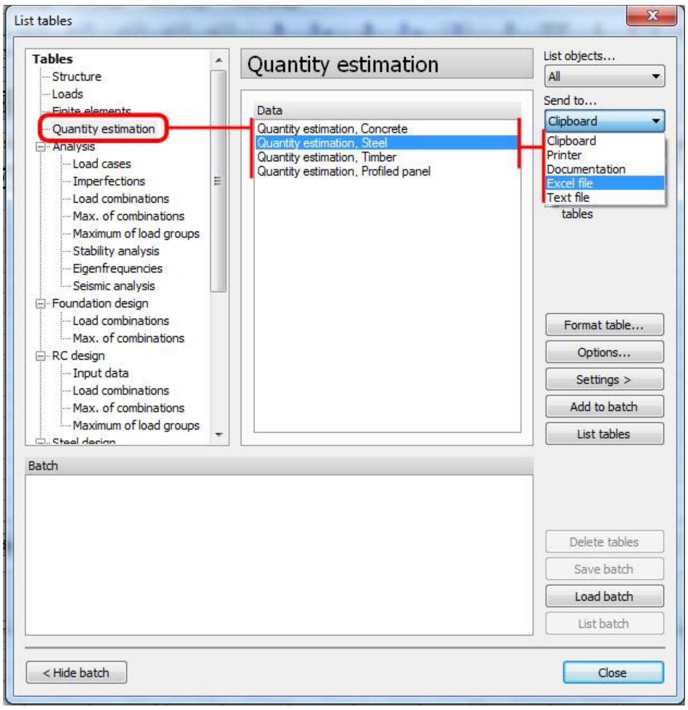


Figure: Listing quantity estimation results

Listing

All tables of input and output data can be collected and printed out.

After launching the List  command, the User can select the **required objects** – using Filter and/or common graphical selection methods - before the List tables dialog appears. The whole database can be listed by pressing 

button. Selection can be carried out in the List tables dialog as well using the drop-down menu in the top right corner. The User can choose the followings:

- All (all objects of the model),
- Current selection (objects selected before List tables appeared; this option is not available when all objects were selected (by pressing Enter) after launching the List command),
- Visible objects (objects visible in the Application window),
- User defined filter (objects belonging to pre-defined filters).

Listed data depend on the selected objects. By ticking 'Hide irrelevant tables' check box those tables which are irrelevant by the current selection disappear (otherwise they are shown in grey).

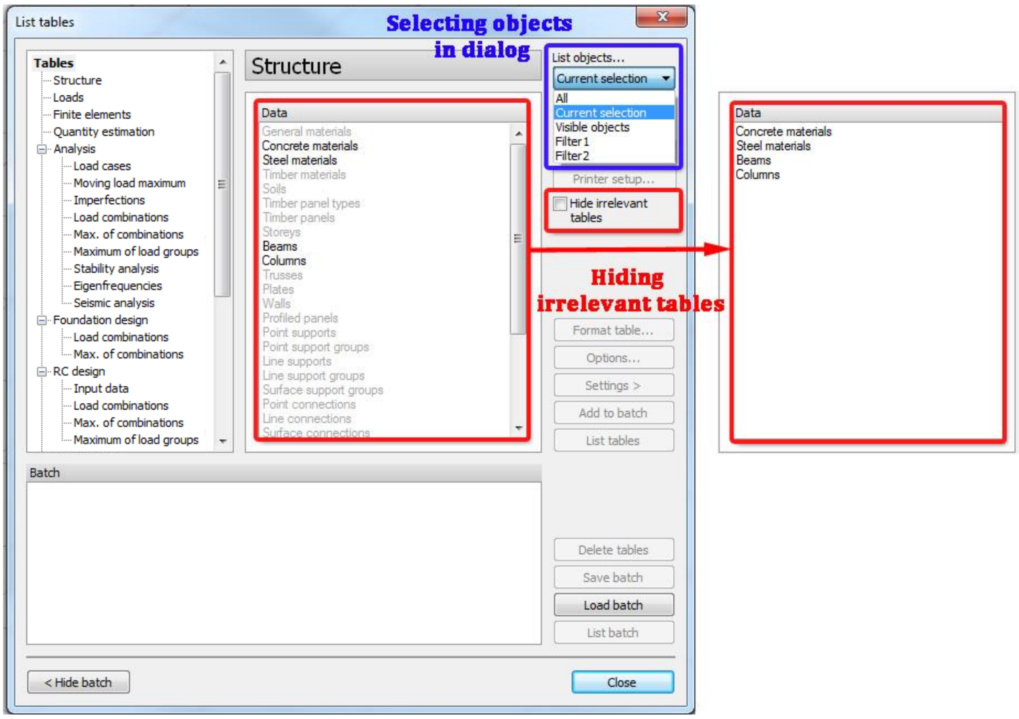
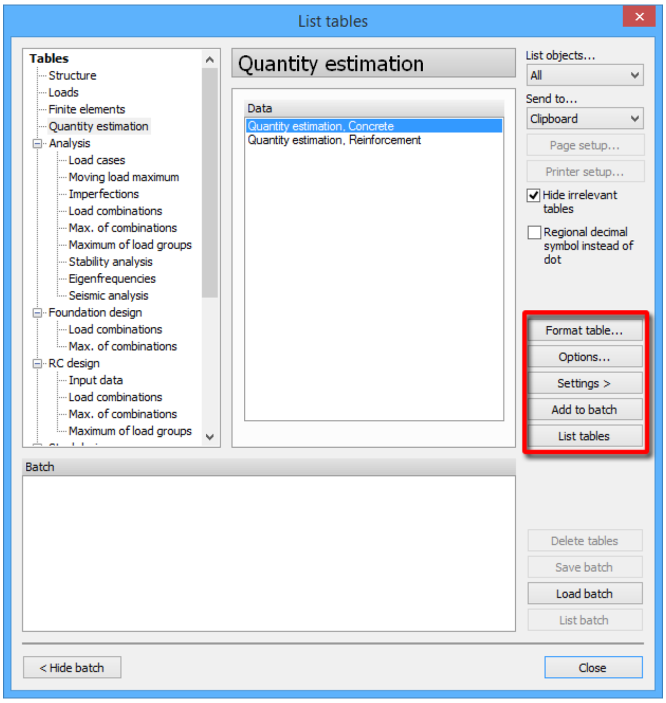


Figure: List tables dialog

Further options are available in *List tables* dialog:



- ***Format table…***: The table settings can be set for all export modes at Format Table…:

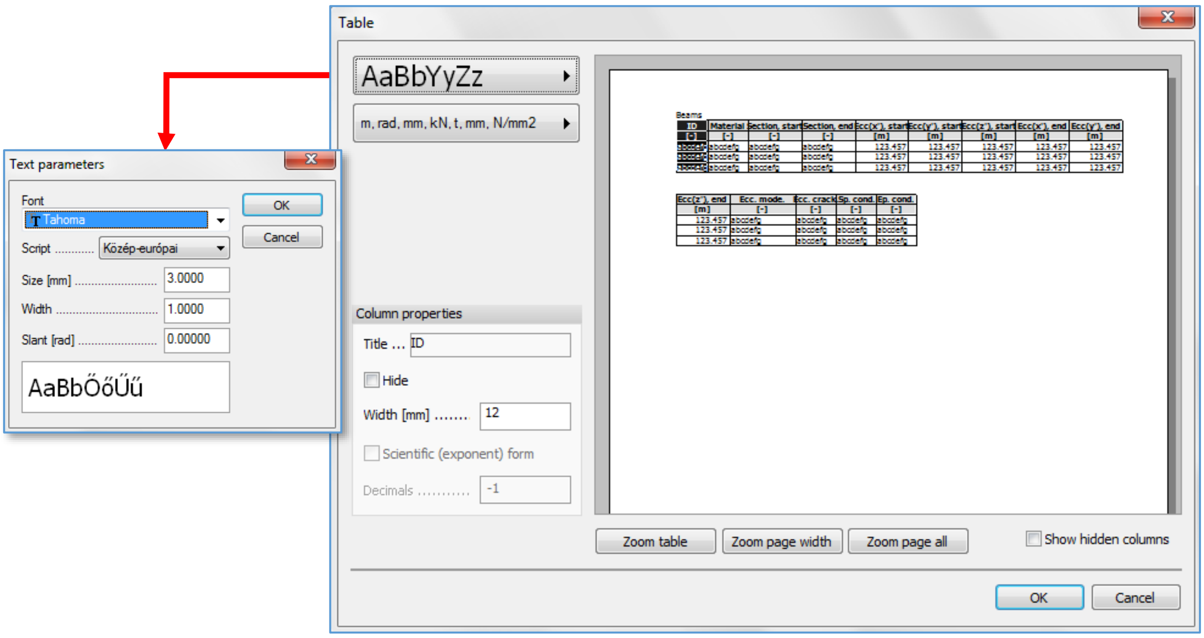


Figure: Customizable table settings

The following properties can be set:

- a common text style (font type and size) for table titles, text and values;
- the unit of the values by element type (*Length*, *Force*, *Displacement* etc.);
- the column width that can be defined by the Width option or by stretching the right side of the selected column with mouse;

- decimal number of numeric data; and

- scientific form of numeric data.

- At ***Options*** (depending on the data type), the calculation points (nodes) can be set where you would like to ask and list (result) values.
- *Settings*> stores the current list settings as the default settings for later use and next projects or previously saved defaults can be loaded.
- With *Add to batch* option the selected data can be added to *Batch list* (see later).

In 'Send to…' drop-down menu the destination of the listed tables can be set. These are the followings:

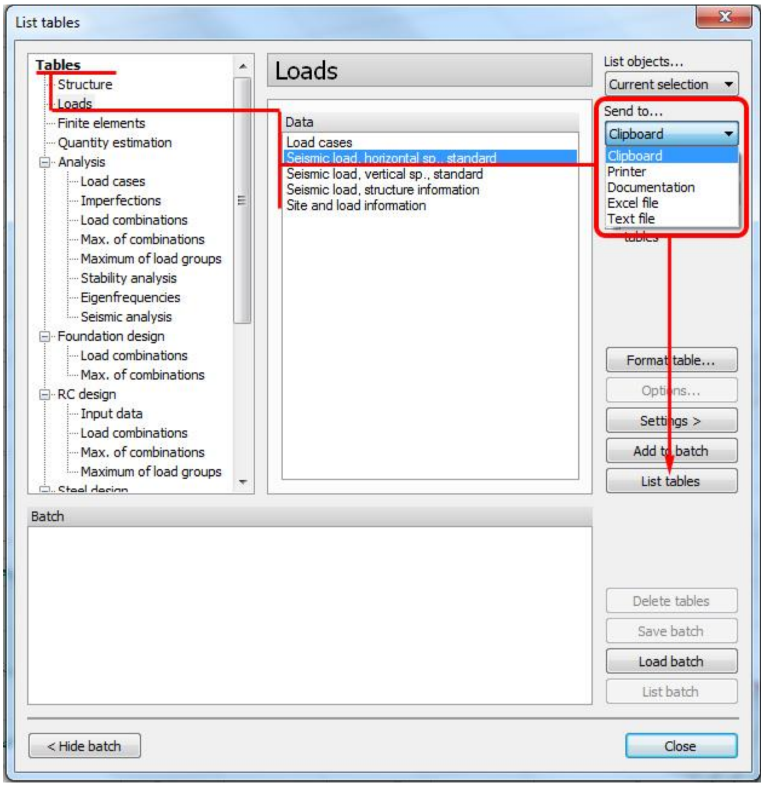
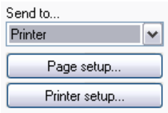


Figure: Listing procedure and modes

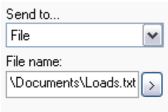
- **Clipboard**

  Clicking *List data* sends the table of the selected item to the *Microsoft Office Clipboard*, which allows you to copy the table and text items from FEM-Design and paste them into another document such as Microsoft Excel, Word etc.

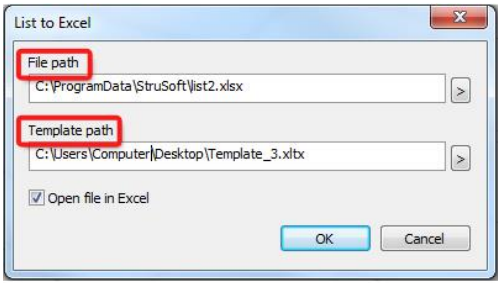
- **Printer**



List data sends the selected data table directly to the printer according to *Page* and *Printer setup*. *Page setup* offers the same possibilities mentioned at *Printing*.



- **Documentation** (listed tables appear in the documentation),
- **Excel file** (listed tables can be exported to an Excel file)

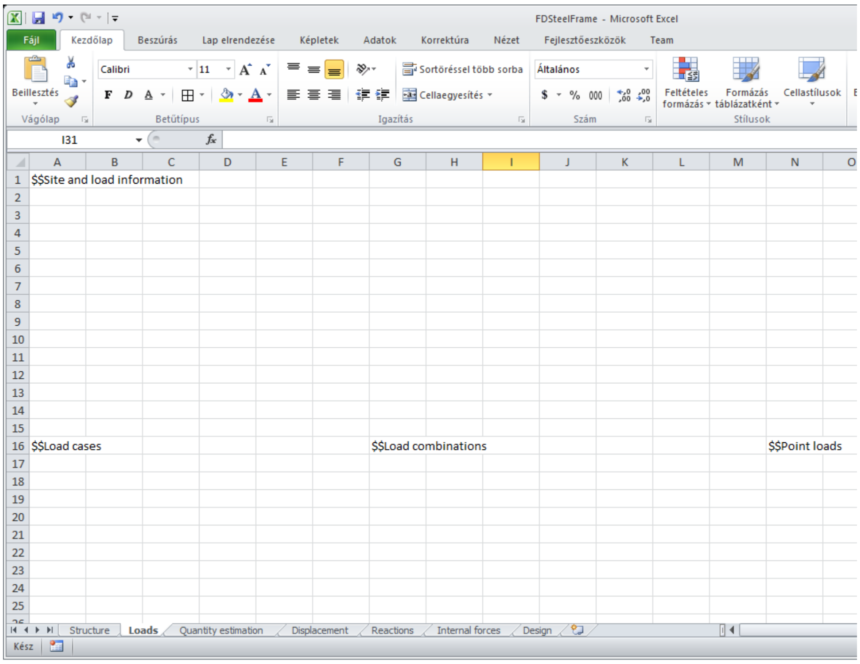


In List to Excel dialog besides the File path also a Template path can be set if the User has a pre-defined Excel-template (with .xltx, .xlt, or .xltm extension). The created Excel file can be opened immediately by ticking 'Open file in Excel' checkbox.

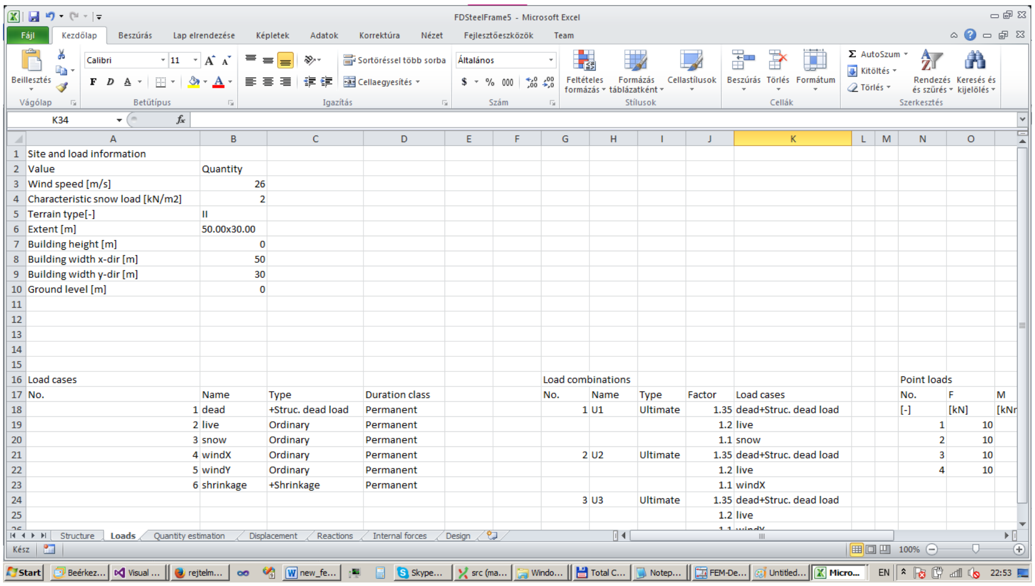
- Listing without a template file: creates an Excel file in which all the tables are located on separate spreadsheets.

- Listing with a template file: the User can create a template in which the locations of the required tables in the spreadsheet are marked with '$$' followed by the exact title of the table. Using a template allows the User to gain exactly the required data.

Template file:



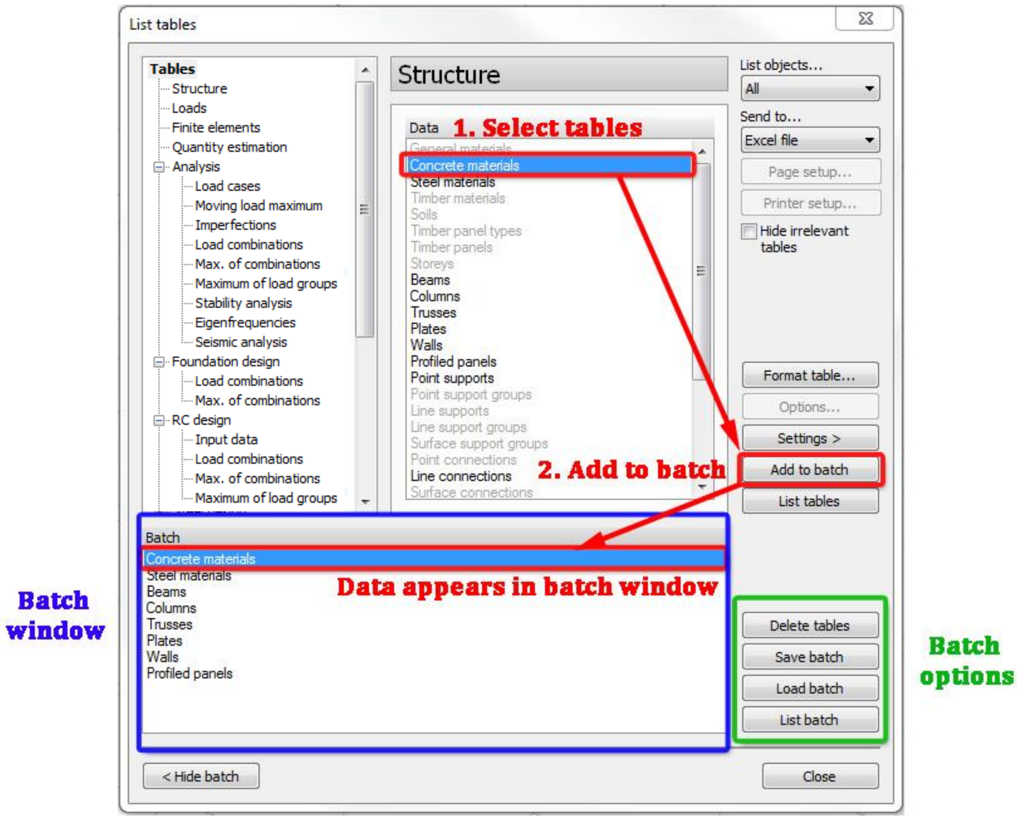
Template-based exported file:



After listing to Excel without any template and examining the number of columns belonging to a table, it is easier to create a template.

2 batch files + 2 templates are delivered with the program (FDSteelFrame, FDRCStruct).

To avoid the time-consuming procedure of selecting the tables for listing one by one **Batch** is implemented for the tables which can be saved and reloaded, even in different models. This way all the required tables can be listed by a few click. Tables can be added to the **Batch** by selecting the tables in the "Data" window and clicking on *Add to batch* button.



The following options can be done with the batch:

- Delete tables (selected data will be deleted),
- Save batch (the batch can be saved to a batch script file with .bsc extension),
- Load batch (the saved batches can be loaded and used in other models; those tables which are not relevant in other models are displayed in grey),
- List batch (all tables in batch window will be listed in accordance with the settings of 'List objects…' and 'Send to…' drop-down menus.

The batch window and options can be hidden by clicking on <*Hide batch* button.

In batch window more data can be selected by holding pressed or with the help of and buttons.

In a listed batch the order of the tables is the same as in the batch window. The order of the tables in batch window can be rearranged with pressed and using .

A saved batch file with load cases (or load combinations, stability analysis etc.) can be used in other models when the names of the load cases (or load combinations, stability analysis etc.) are the same due to that the identification is based on the name. If all load cases (or load combinations, stability analysis etc.) are selected in the batch window and the batch file is loaded in another model, all the load cases (or load combinations, stability analysis etc.) of the other model will be taken into account.

Documentation Module

Although the *Plot* and *List* commands give quick report data possibilities, you can create multi-page documentation with the *Documentation* module. Titles, figures, texts, tables, images, headers, footers, table of content etc. can be placed in custom style into the complete documentation. *Documentation* module can be run in all FEM-Design modules (depending on your license).

Click  to enter *Documentation* mode.

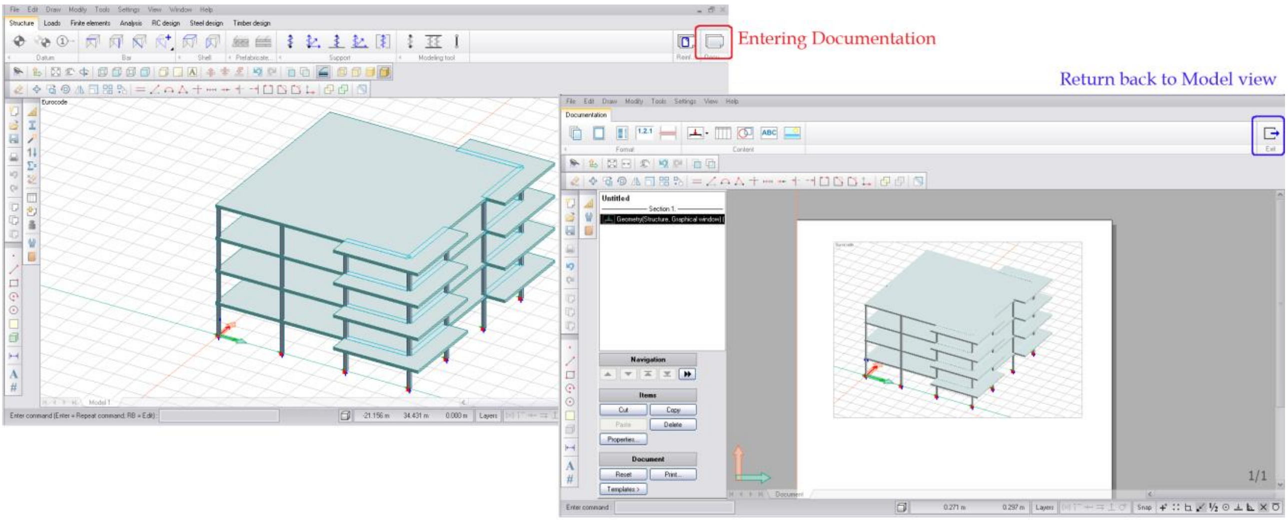


Figure: Starting Document module

The next chapters introduce the possibilities and the proposed steps for defining complete set of design documentation.

Main Commands

The commands and their functions:

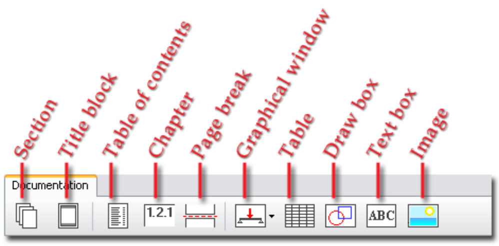


Figure: Documentation tools

- **Section**

 *Section* defines documentation parts having different properties (="sections") such as page size, orientation, margin sizes and associated printer.

Define a section for the cover page with a color image as *A4* page size and with a color associated printer, and another section for the other pages as *A3* size and with grey-scale printer for texts and figures with black lines.

- **Title block**

This is a command for creating and registering new labels, headers, footers and frames, and for inserting stored title templates into the document from the built-in or user-defined title block library. You can apply drawing and auto text tools to design your private label, title and frame style and then register it with *Title block* command for current and future use.

- **Table of contents**

It inserts a table of contents into the current document. It contains the list of chapters (see the next command) and their page numbers. If you define a new chapter or make modifications, the program refreshes this list.

- **Chapter**

Titles of main and sub-chapters can be inserted into the document with item numbers and names.

- **Page break**

Defining a page break generates a new page with the same properties of the current section.

- **Graphical window**

The command lets you to insert (working) windows in which you can display different figures (geometry, loads, mesh, analysis and design results, etc.) into the current document.

A window object defined by *Graphical window* is not a static figure with constant and fixed content. It is a working window, where you can use the layer systems, display settings and view commands (zoom and rotate operations, plane, side and space views, shade and wireframe display modes, etc.) to set the required appearance of the model, the loads, the supports, the results, etc.

- **Table**

It inserts tables of input and output data (e.g. list of load cases and load combinations). The function and the table settings are similar with the possibilities of the *List* command.

- **Draw box**

The tool inserts a working window into the document where you can create your own drawing with the commands of the *Draw* and *Edit* menus. Drawing layer-system is also available in a *Draw box*.

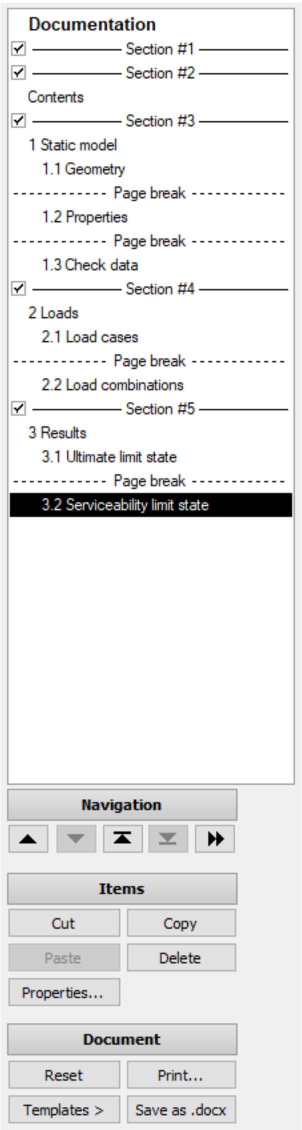
- **Text box**

It is a simple word processor for creating texts, notes into the document. The size of the text box, the font styles and text alignment can be set manually.

- **Image**

Image files (e.g. jpg, bmp etc.) can be inserted to the document with this command.

Control Panel
The control panel enables you to easy move between different parts of the current document, to set the properties of the inserted objects, and to print and create templates for further documentation.



(Documentation map)

The list contains the defined title boxes, sections, graphic boxes, images, drawings and texts (hereafter called "objects"). You can activate the selected list object by double clicking on its name.

In order to speed up work in Documentation, Sections can be hidden/shown by clicking the checkbox next to the section's name.

The hidden sections are not locked thus, they can be cut, copied, pasted, moved, dropped or modified. If an active section is moved into a hidden one, both sections will be hidden and vice versa.

It is possible to select more sections to hide at one go. Uncheck an active checkbox, all the selected sections will turn off. Check an inactive one, they will all be shown.

Navigation

You can navigate between pages by *Page up*, *Page down*, *Home* and *End* buttons. Clicking *Go to* switches to the object selected in the list (the same as double clicking the selected object in the *List*).

Titles added to documentation elements provide easy navigation through defined *text boxes* and figures (*Graphical window*, *Image* and *Draw box*).

Items

With *Cut*, *Copy* and *Paste* tools you can modify the position of selected list items (multi-selection also works by using the mouse together with Shift and/or Ctrl keys). You can move a selected list object into a new position by holding down the Alt key and moving the mouse into the required row of the list. To erase one or more selected objects click on *Delete*. With *Properties* the settings (sizes, styles, contents, etc.) of the selected object can be modified. The variability of objects depends on the type of object (drawing, text, image, section, etc.).

- **Document**

With *Reset*, you can delete all contents of the current documentation and "open" a new, empty document with the default settings. At *Templates* you can save the current document structure and settings under a name as a template. You can also load a predefined one to create fast documentation of your current project (see later *Templates*). The *Print* command lets you print out the whole document or a part of it. With *Save as .docx* button the whole documentation can be exported to an Office OpenXML (.docx) file.

Steps to Create Your Own Documentation

This chapter shows the proposed steps of document creation via an easy example (in *Plate* module).

For a faster documentation process, we recommend that you start documentation after finishing model generation and calculations and not before them.

1. **Document title**

   Set the main properties of the document by selecting *Untitled* in the list and click on *Properties* at *Items*. Give a name for your document (in *Title*) and set the font settings by chapter orders (levels). Under *Utilities* you can save the settings as default or as a named style, so you can then load styles for future projects.

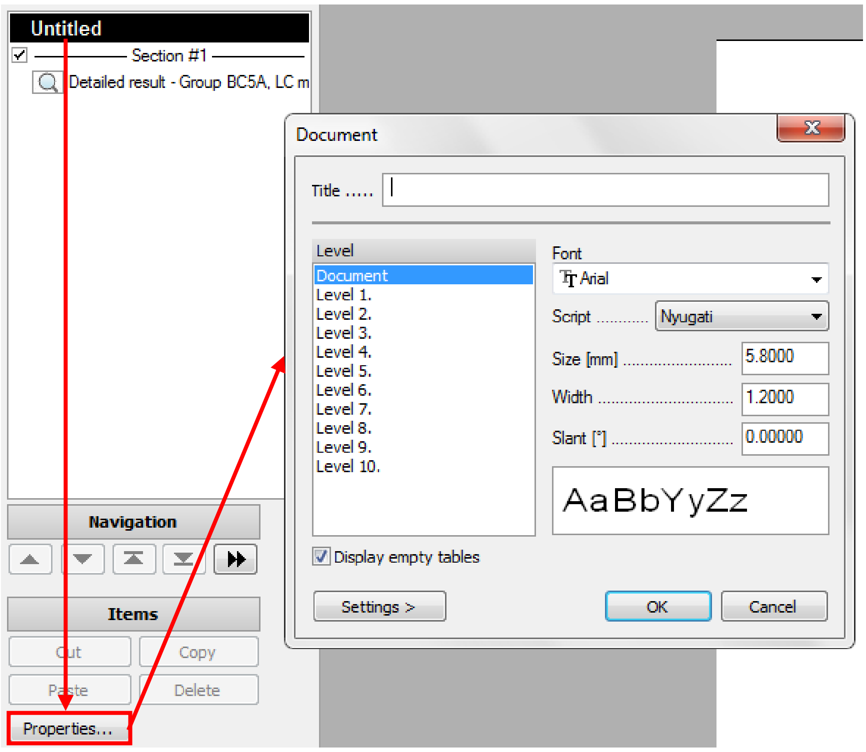


Figure: Document title

2. **Cover page (**Text box and Image - Section 1.**)**

   Set the properties of *Section 1.* by activating it in the list and by clicking on *Properties*.

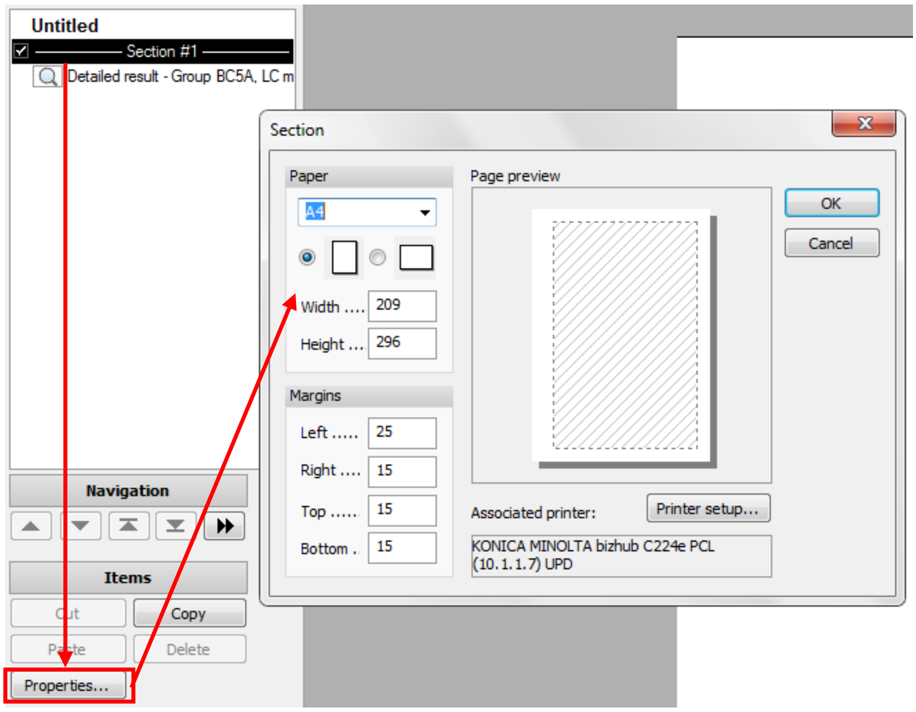


Figure: Cover page

- Set the paper size and orientation (e.g. *A4* and portrait).
- Choose a printer for the current section. You can set different printers, plotters and PDF creators per sections. It is very typical that the text parts are wanted to print as A4 and the figures etc. as A3, etc.
- Set the margin, from which the texts, figures, images, etc. will start. For example, the main title will start 60mm from the top of the page.

Write the title of the document on the first page with ABC *Text box*. The text will start from the upper margin.

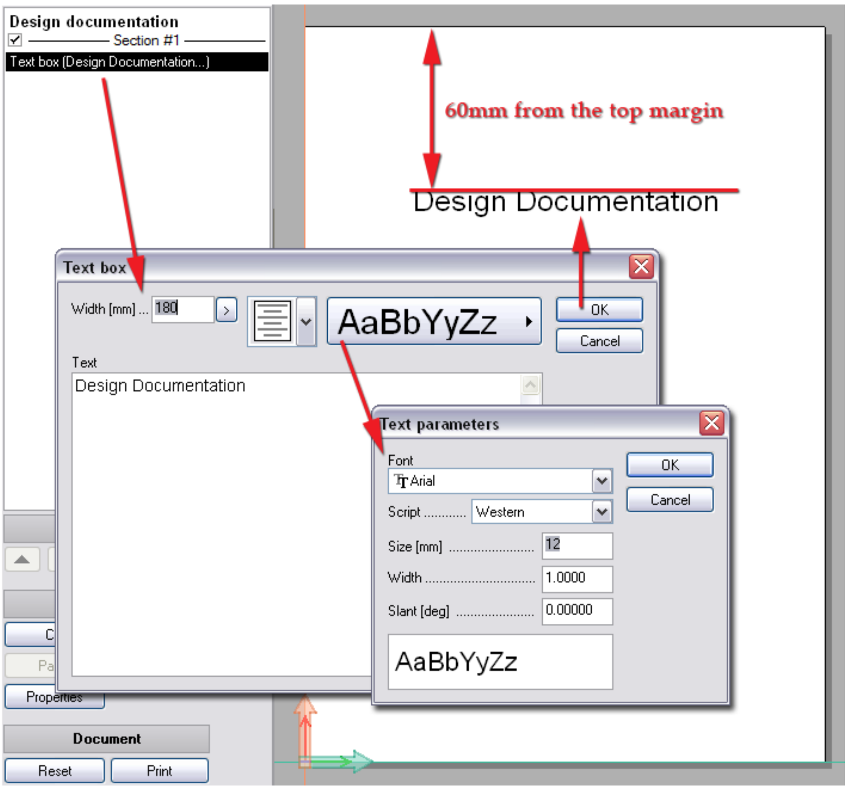


Figure: Cover page with title

Also a subtitle can be created by re-running *Text box*.

Empty rows and new lines can be defined with the + key-combination.

Place an image file (e.g. a picture about the model) with the *Image* command. To open an image file, browse for it at > option of *File name*. Set the position of the picture from the side margins at *Alignment*. At *Size* you can define arbitrary size or special size (*Page width*/ *Full page*) by choosing from the list under > symbol. Apply *Preserve ratio* to avoid figures from distortion during size definition.

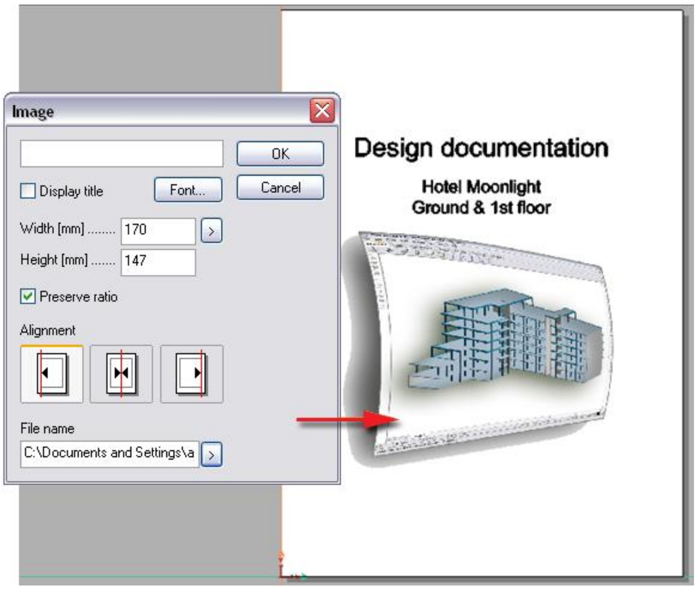


Figure: Cover image

The settings for an inserted object (text box, image, etc.) may be modified by selecting it in the list and by pressing on the *Properties* option. It works the same as with the initial dialog of the object.

3. **Table of contents** (Section 2.)

   Create a new section (leave your selection on the last item (here the image)) with *Section* and set the value of the top margin, which the table of content will start from (e.g. 30mm). Click on *Table of contents* command and an empty "table" appears on the *Section 2* page. The table will be refilled automatically when defining *chapters*.

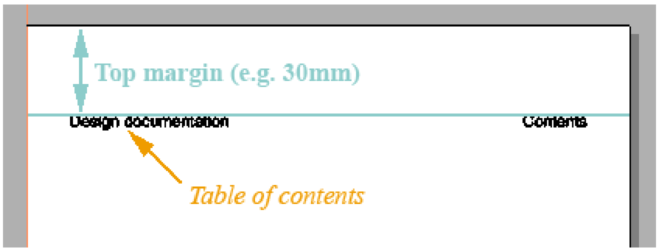


Figure: Inserted (empty) table of contents

4. **Header and footer** (Title block - Section 3.)

   Create a new section (leave your selection on the last item (here "Table of contents")) with *Section* and set top and bottom margin values that define the available area for further text, figures and tables (30mm in this example). Use the commands from the *Draw* menu to prepare the appearance (frames and texts) of headers and footers. Apply e.g. *Line* and *Field* commands (*Auto-text*).

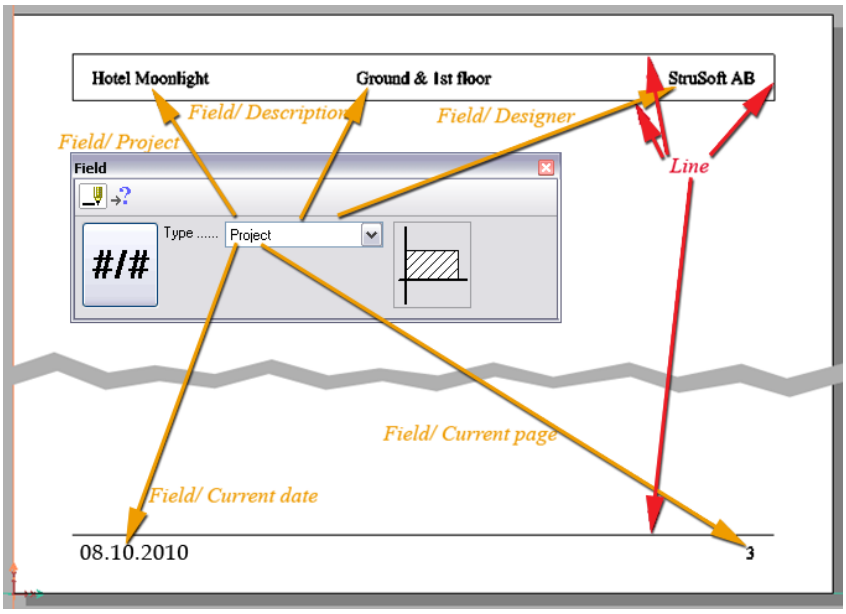


Figure: Header/footer defined by Line and Field commands

Where you see "?" symbol at an auto-text, the content can be defined at *Settings* > *Title*.

After finishing drawing, use the *Register* tool of *Title block* to save your "header and footer" as built-in label parts. Type an identifier ("header&footer" in this example) for the title in *Name* field, select the lines and texts by window selection and give an insertion point (bottom left corner of the page in this example).

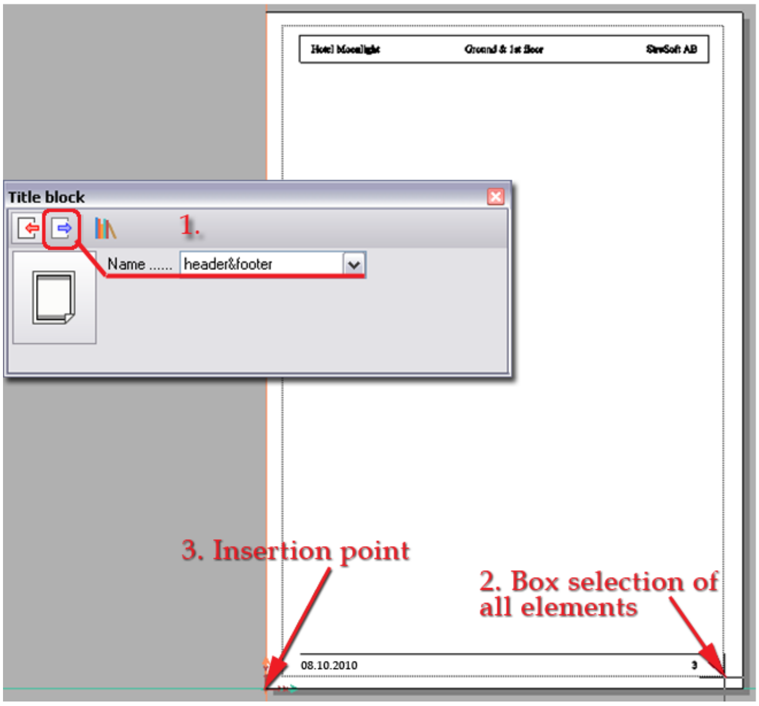


Figure: Header/footer defined by Line and Field commands

After registration, delete the drawings and text from the document by *Erase* (*Edit* menu)!

Insert the defined and registered title block into the document with the *Insert* tool of *Title block*. Choose the saved "header&footer" from the *Name* list, and then place the insertion point of the selected title block on the bottom left corner of the page. As result, the "header&footer" will be repeated on every page in the current section (*Section 3.*).

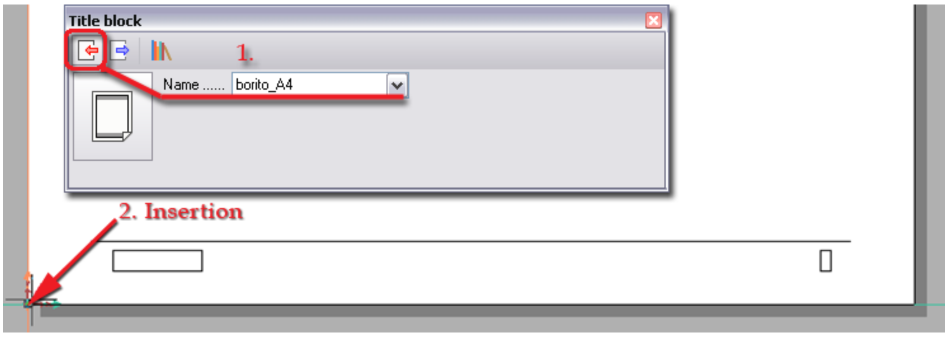


Figure: Insertion of title block (header & footer)

5. **Chapters (**Section 3.**)**

Place your first chapter with the 1.2.1 *Chapter* command. Give a name in *Title* field for the current chapter ("Geometry" in this example).

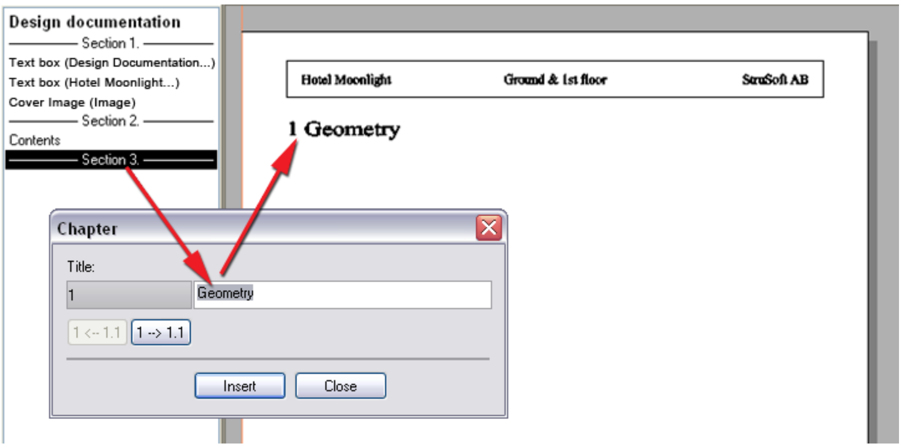


Figure: Insertion of chapter

For further chapters the program offers alternative numbering systems for the next main chapter ("2") or subchapter ("1.1"). It depends on which button is active: 1 --> 1.1 *Increase indent* or 1 <-- 1.1 *Decrease indent*.

6. **Figures (**Graphical window - Section 3.**)**

In a chapter you may insert text (see *Text box*), images (see *Image*) and figures of the model geometry, loads, results, etc. In this example, first place a figure about the model geometry (plates and supports). Click *Graphical window*, choose *Structure*-type and set the display parameters of the image in the appeared dialogue box. You may give a name to the figure for displaying it (*Display title*, "Plane view"). A "1m scale bar" can be displayed inside the figure with *Display scale*. The size of the window can be set with a width and a height value or by choosing predefined sizes under > (*Page width*/*Full page*). The required figure position can be defined by activating one of the buttons for *Alignment* ("centre" in this example). The position of the title is the same as the activated *Alignment* of the window. Clicking *OK* inserts the new figure into the document.

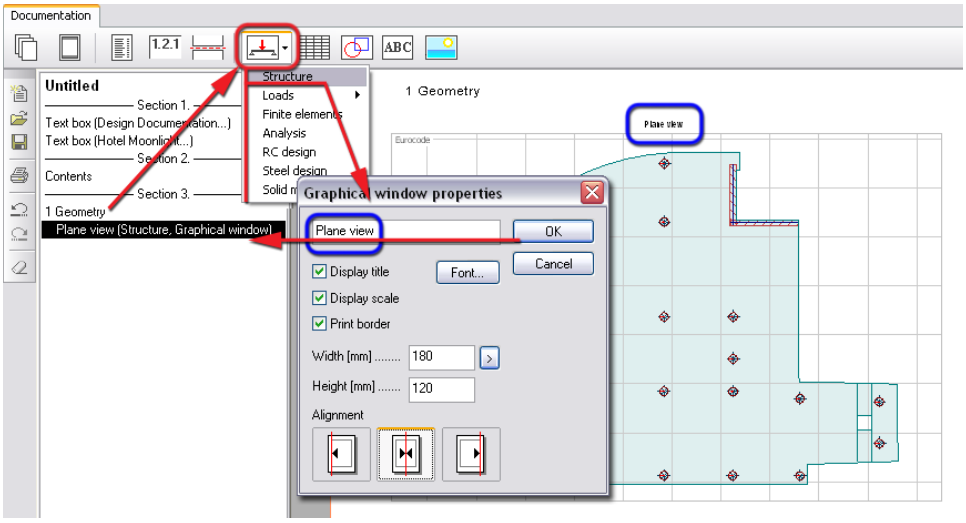


Figure: Insertion of figure

Other graphical windows can be defined by repeating the previous step. For example, place a side view of the structure, etc.

Clicking the window area activates it as a working window, so you can modify the view and position of the window content. Black thick frame shows what graphical window is the active.

For example, clicking the *Structure*-type graphical window, the display settings (*Settings* menu), the view commands (*View* menu) and the *layer system* (*Current layer*) may be used; but at an *Analysis* (result)-type graphical window, you can browse from results (*New result* and *Select result*), displaying numeric values (*Numeric value*).

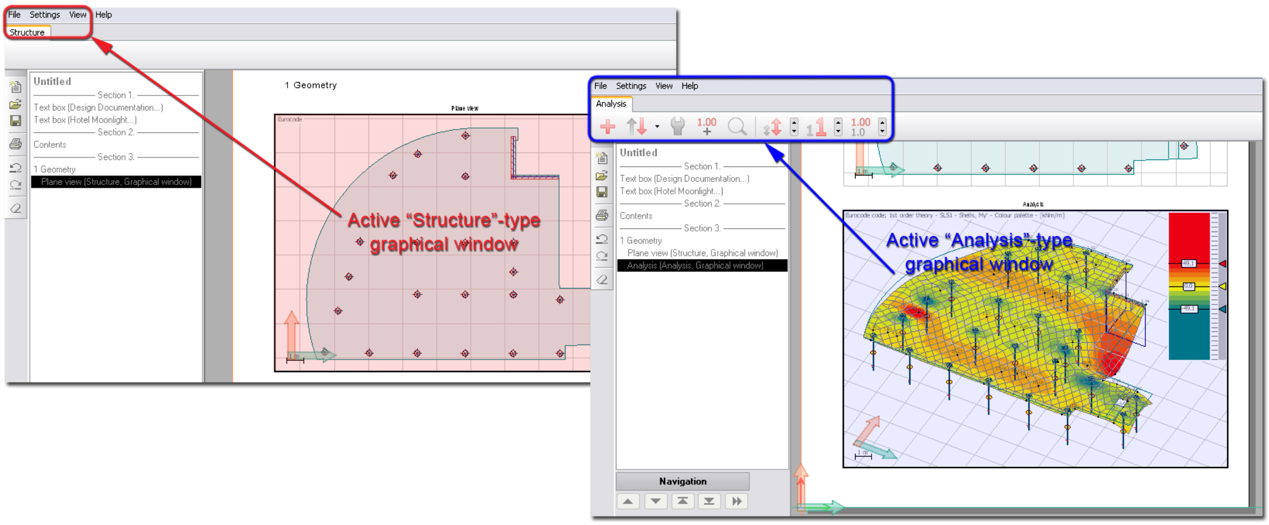


Figure: Available tools depend on the graphical window type

Each graphical window has its own layer system, main and display settings. *FEM-Design* also gives the possibility to define different units for displaying geometry and various results.

Click the white paper (outside from the active graphical window) to return back documentation.

Repeat the previous steps to define more *chapters* (2. Supports, 3. Loads, 3.1 Load cases, 3.2 Load combinations, etc.), and to place new figures (such as loads with a *Loads-type graphical window*).

There is no facility to set distances between texts, chapter title rows and figures, but with ABC *Text box* command you can insert empty lines between objects into the document.

Another way to insert figures is, that the current model view can be placed onto the last page of documentation with the *Add view to document* command (*Tool* menu). Of course, the settings of the new graphical window can be modified later with the *Properties* tool of *Documentation*.

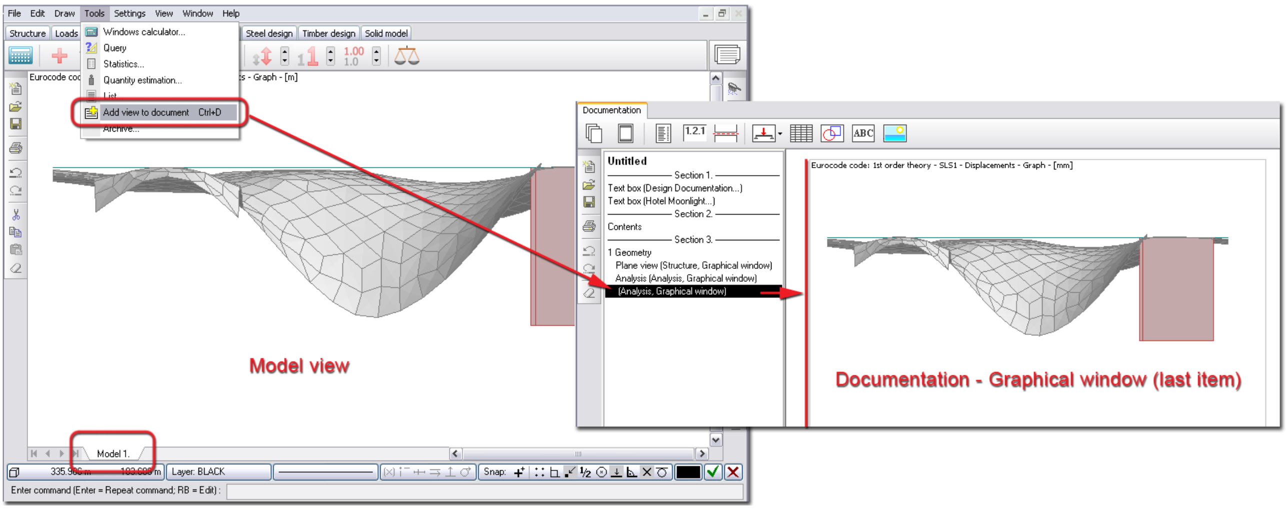
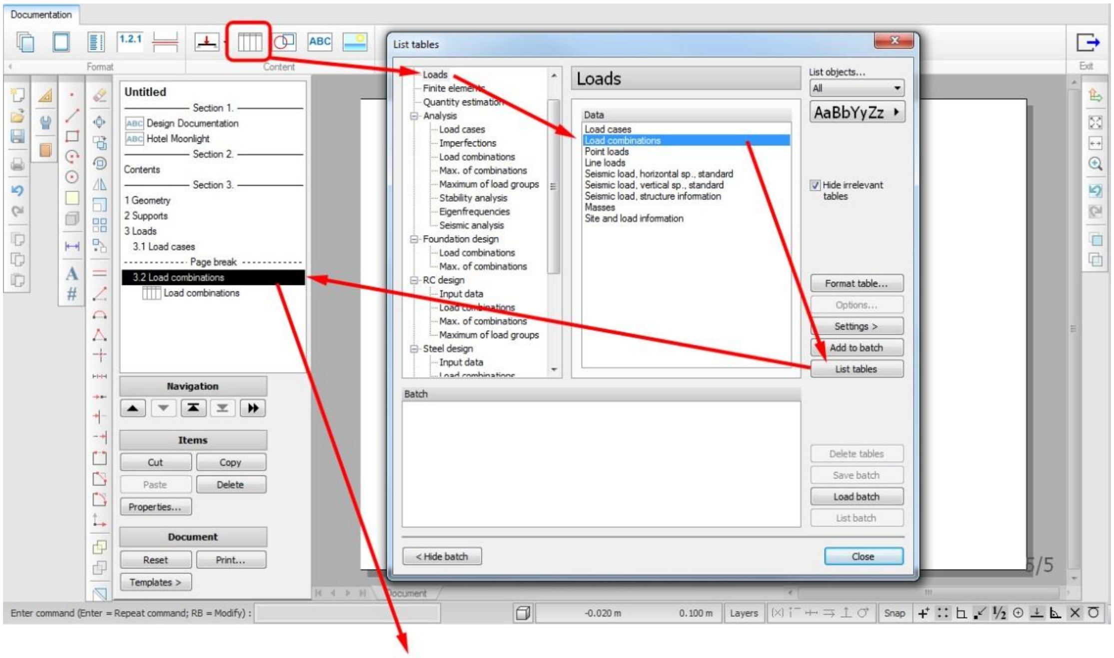


Figure: Quick export of current view as a figure into documentation

7. **Tables** (Section 3.)

Input and output data as tables can be added to the document with the *Table* command. The functions and tools of *Table* are same as described at ***List***. Choose data to print out as a table from the available *Tables*, *Data* and *Result* lists. In this example, place a summarization table of load combinations (under chapter 3.2: Load combinations). First select *Load combinations* in *Data* list (*Tables/ Loads*), second set the table settings at *Table* and *Options*, and finally click on *List data*. As a result the selected table appears after the current documentation object.



Load combinations

No.	Name	Type	Factor	Load cases
1	1st c...	Ultimate	1.000	LC 1
2	2nd c...	Ultimate	1.000	LC2
3	3rd c...	Service...	1.000	LC2

Figure: Insert table to documentation

8. **Result figures (**Graphical window - Section 3.**)**

   To document result figures, use *Graphical window* and select *Analysis-*, *RC design-* , *Steel design-* or *Timber design* (depending on the available calculation type and what result you would like to display). After insertion, click inside the graphical window and choose result types with *New result* and/or *Select result*.

Templates and Automatic Documentation

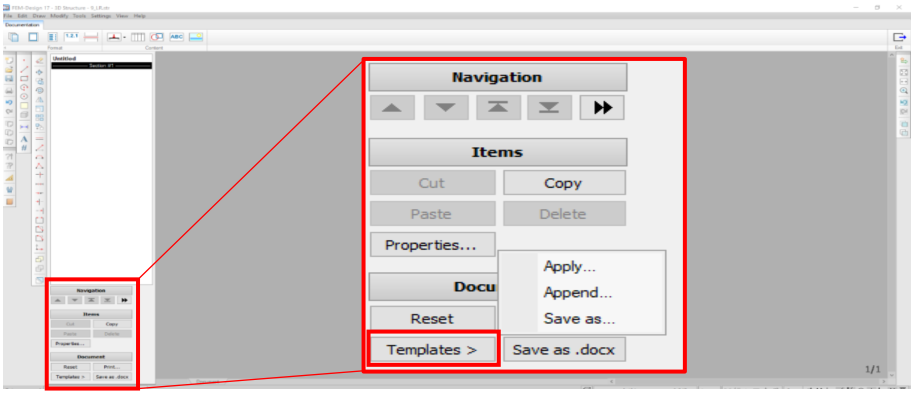
The documentation structure (cover, chapters, text and window positions) defined in *Documentation* module can be saved under a file name (*.DSC*) as a template for further projects. Click *Control Panel's Templates* > *Save as* tool.

The program automatically generates your complete documentation, if you apply a previously defined or built-in template. Use *Templates* > *Apply* when entering to *Documentation* mode after modeling and calculate a new project, and the program automatically generates complete documentation by refreshing based on the applied template (it updates the title label, table of content, page numbers, figures and tables etc. of the template).

A pictures inserted by *Image* are deleted in a new template!

If you load a template into a document having no calculated results, the *Analysis*, *RC design* or *Steel design*-type graphical windows contain only the structure, but they will be refreshed any time you have calculations.

The *Templates* can be appended into the Documentation by *Templates >/Append…* command. Items of the appended templates will be inserted after the last existing document item.



If an applied template generates documentation with *tables* having no valid values, the empty tables can be hidden. Select the title of the document (first row) in the *Control panel*, then click *Properties* and inactivate the "*Display empty tables*" option in the dialogue that pops up.

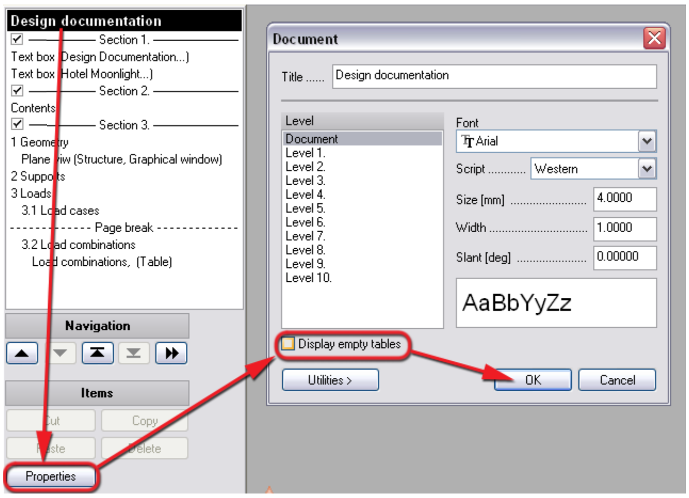


Figure: Empty tables hidden by Properties

The documentation can be exported to Office OpenXML (docx). For further information check the following link: http://download.strusoft.com/FEM-Design/inst150x/savetodocxfaq.pdf

Printing

The completed document or only some parts of it can be printed out with *Control Panel's Print* tool or *File* > *Print document*.

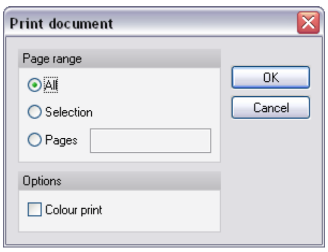


Figure: Printing setup

Set the *Page rage* as:

- **All**

  The whole document (all pages from start to end) will be printed out.

- **Selection**

  The objects selected in the *Control panel* (multi-selection also works) and their pages will be printed out.

- **Pages**

  This provides the possibility for printing by defined page. Enter page number and/or page ranges by separating with commas and by using "-" (an example: 1,10,15-20,25, etc.).

The printing process requires associated printer(s) by the sections of the current documentation. If a section has not got a defined printer (*Undefined*), you can add one by selecting the section name in the document list and by using the *Properties* tool (*Change* option).

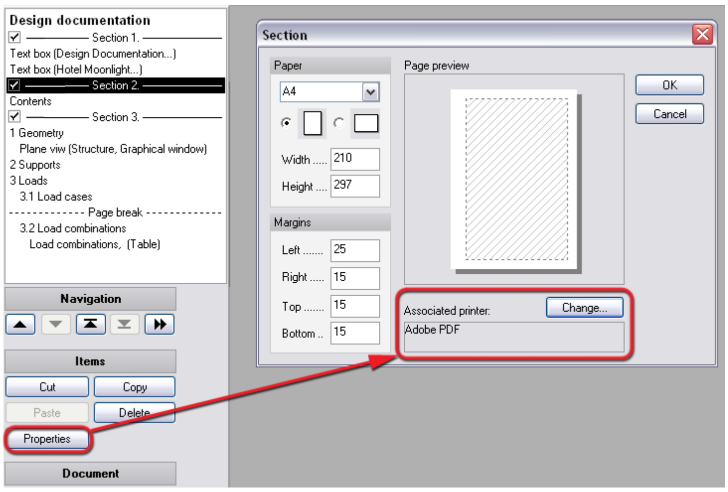


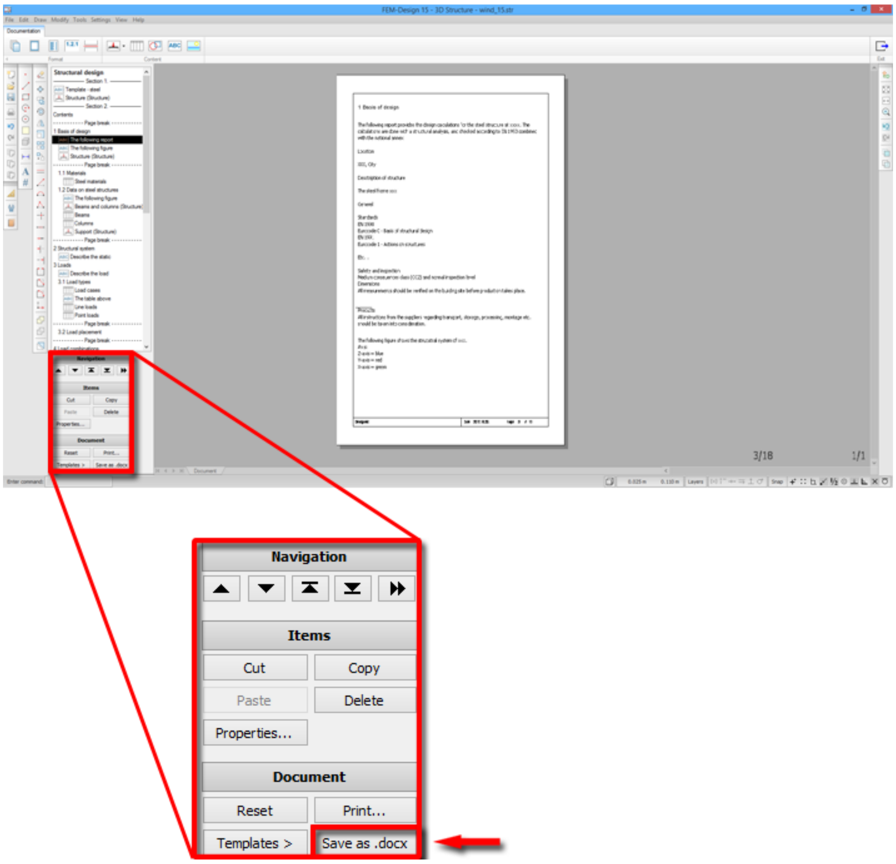
Figure: Printer assigned to Section

If you assign a PDF creator program to all sections, you can save your documentation as a *PDF* file.

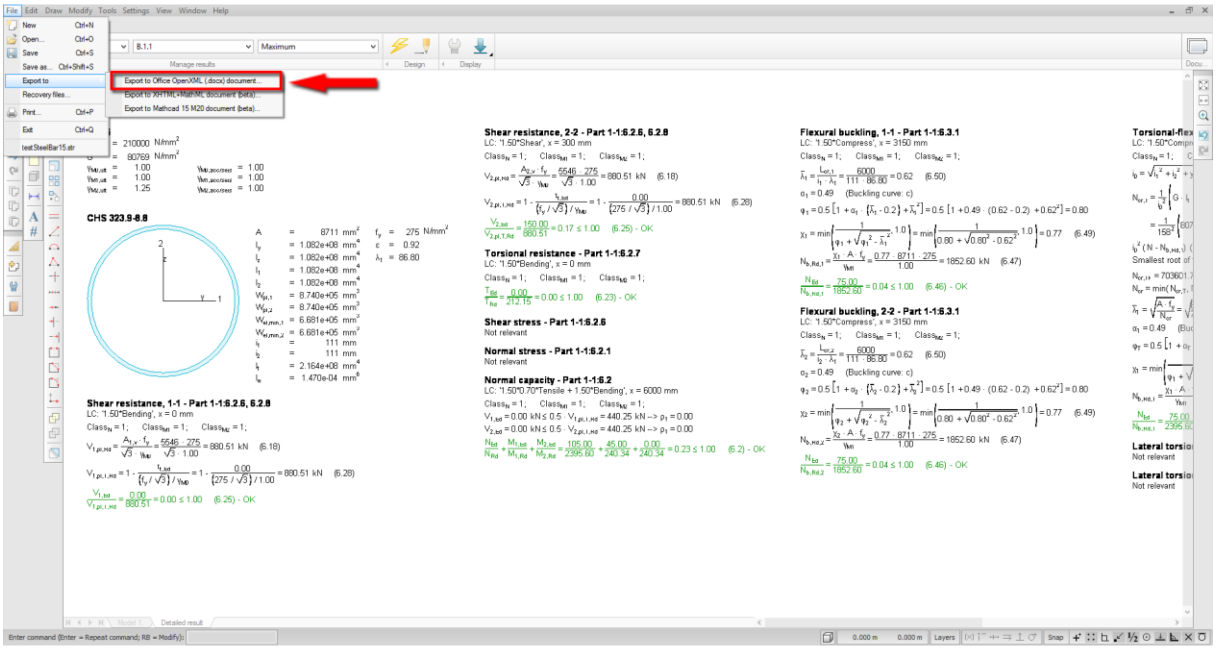
Export to .docx format

It is possible to export the Documentation and any Detailed result to Office OpenXML (.docx) file.

In the Documentation module this feature can be accessed by the "*Save as .docx*" button at the bottom right corner of the Control panel.


Navigation
Items
Cut
Copy
Paste
Delete
Properties...
Document
Reset
Print...
Templates >
Save as .docx

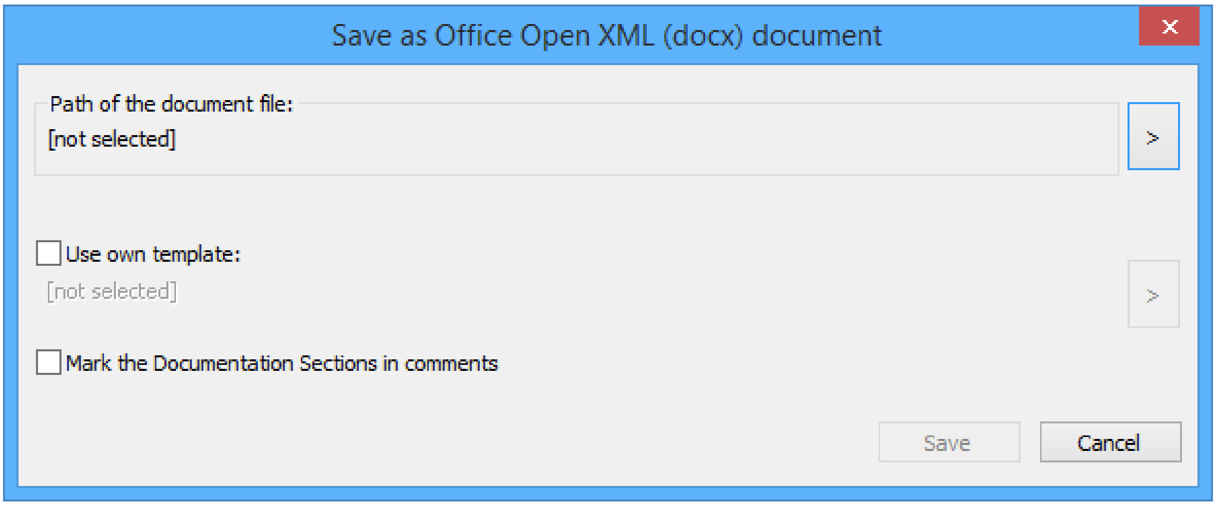
To export data from the Detailed result window use the *File* > *Export to* > *Export to Office OpenXML (.docx) document* command or directly the *File* > *Export to Office OpenXML (.docx) document* command where it is available.



In both cases the "Save as Office open XML (docx) document" dialog appears, where User can decide the followings:

- where to save the result file
- which MS Office Word template file to use for the convention (optional)
- whether to mark the Documentation Sections in comments or not (only in Documentation module)

⚠ The used template files must be based on the original template file *My Documents\FEM-Design document templates\template for fem-design document templates.dotx*, as this file contains the style required for the exportation.



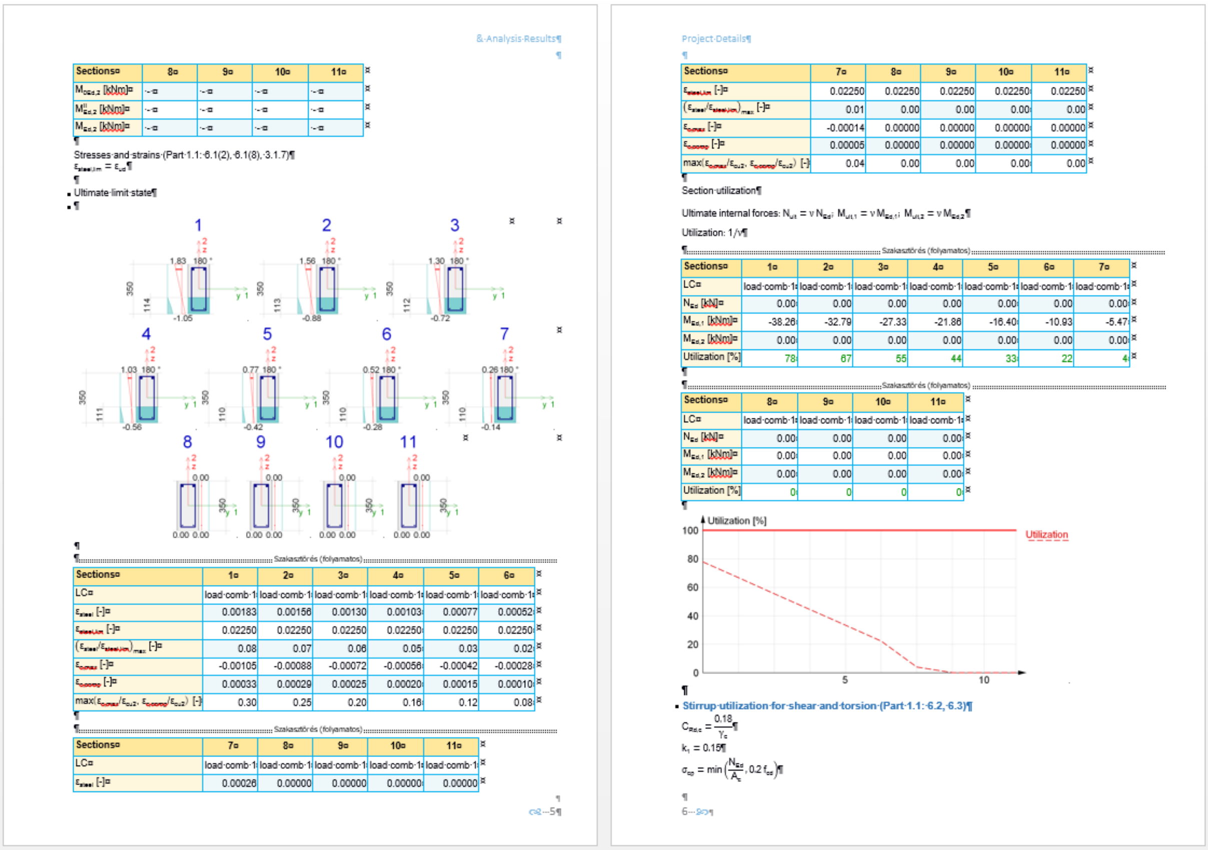
⚠ MS Office Word 2007 or newer is required for opening files with .docx distribution

It is highly recommended to mark the documentation sections in the exported file, since the page rendering in MS Word and in FEM-Design are different. Furthermore, the exported document has A4 format pages. Therefore, if there are pages in the FD Documentation with different size, it is easier to find them and change their size in the result .docx file with section marks.

Using own Word templates is useful when the FD Documentation contains Title block, since it is not possible to export it to .docx. As an alternative, User can create a Word template with custom headers and footers, and use it for the exportation.

Since MS Word has more sophisticated solutions for creating the table of contents, FEM-Design does not export it to the .docx document. However, chapters are exported, therefore table of contents can be created in MS Word easily.

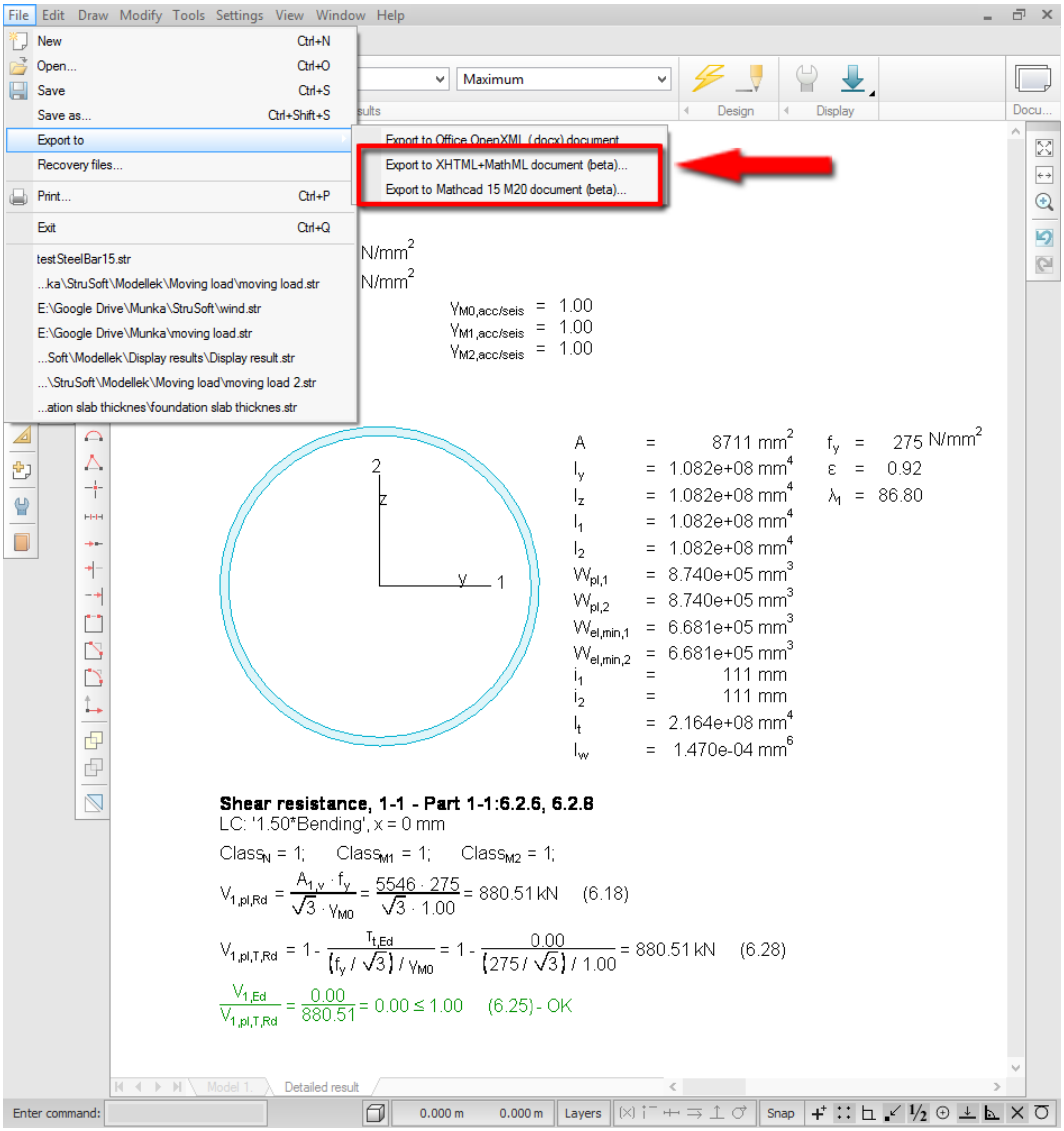
An example of the exported Detailed result can be seen in the following figure:



More information and useful hint can be found in the Docx FAQ.

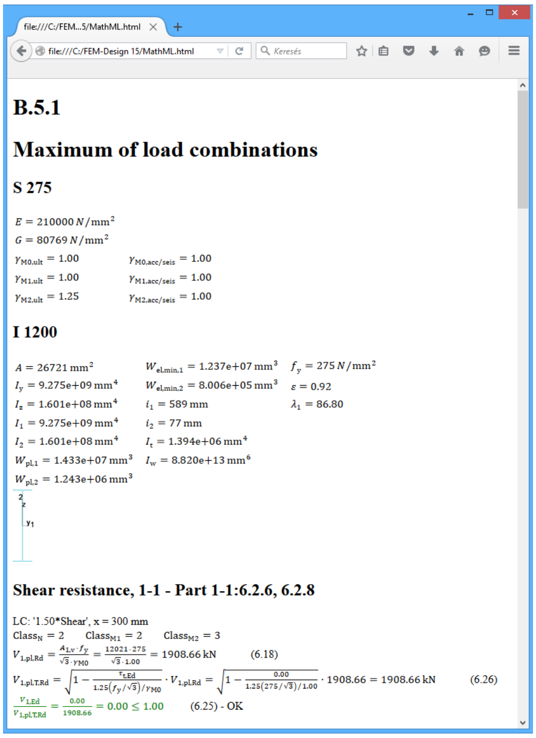
Export to Mathcad and to XHTML

User can export Detailed results to .xmcd (Mathcad 15) and to .html (XHTML with MathML). This opportunities can be accessed in the Detailed Result window's *File* > *Export to* menu. The .html exportation is available for the detailed results of all Designs, while the .xmcd is for Steel and Timber design.

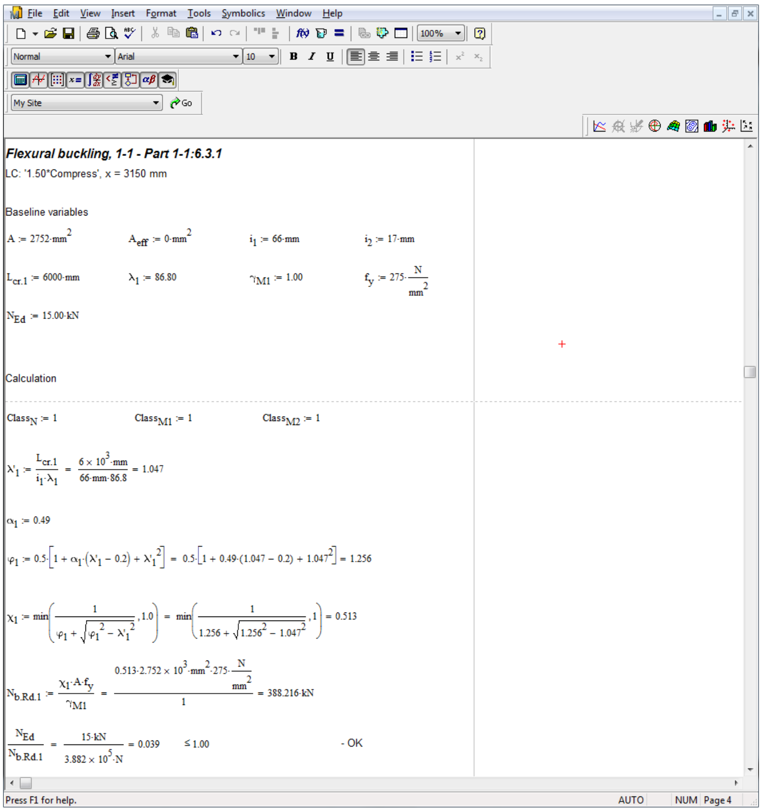


⚠ The exported .html file can be displayed properly only in web browsers supporting MathML. Currently: Firefox 38 or higher, Safari 8 and 9, iOS Safari 8.4 or higher.

The result of the XHTML exportation is a .html file and also a folder, which contains elements of the html page.



The result of the exportation to Mathcad is a single .xmcd file, which is fully editable in Matchcad 15.



⚠ Mathcad 15 M20 version is required to open the .xmcd file.

DRAWING AND EDITING TOOLS

Although FEM-Design is an analysis and design application, CAD drawing can be also done with its built-in drawing tools. Editing tools also available to edit, modify and multiply both structural and drawing elements. Drawings can be the draft of the later design model, but you may also export them in the popular **DWG and DXF** formats.

Drawing tools are grouped in the *Draw* menu, but they are directly available from the list appears when clicking  key together with .

Editing tools are grouped in the *Edit* menu, but they are directly available from the list appears when clicking .

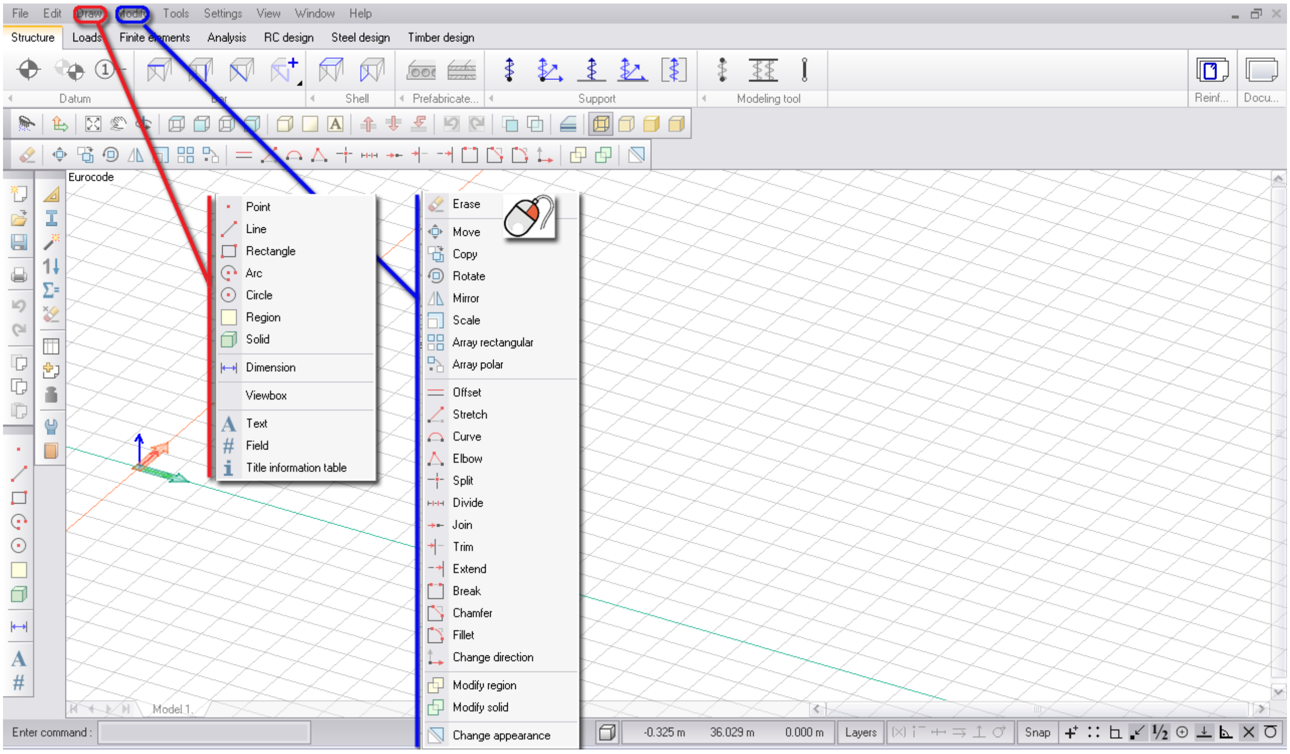


Figure: Drawing and editing tools

Draw Menu Commands

The following rules are valid for the drawing commands and elements:

- Drawing elements are placed and stored in **Drawing layers**.
- The color of the current drawing layer is the default drawing color. Under **"Current color"** button of **Status bar**, you can customize the drawing color independently from the host layer's color.
- The style of the drawing lines and edges can be set with **"Current style"**.
- **Drawing and text styles** together with **display settings** of drawing elements are available at *Settings*.

- Although, you may draw in 3D, in most cases, it is recommended to set the ***user coordinate-system*** in the drawing plane.
- To define drawing points, type coordinate and angle values in the **Command line** and/or **Co-ordinate box** or use the **Object snap tools** to pick requested special points.
- **Command line** guides you with the steps of drawing element definition.

The next table summarizes the available drawing tools, their functions and definition steps.

Command	Description	Definition modes
Point	Defines a point based on the current drawing settings.	-
Line	Defines a line or connected lines (open, closed and/or polyline) based on the current drawing settings.	Definition of start- and endpoints
Rectangle	Defines a rectangle (closed contour) based on the current drawing settings.	Definition of two opposite corners
Arc	Defines a circular arc based on the current drawing settings.	Arc by center, radius and angle Arc by 3 points Arc by start, end point and tangent
Circle	Defines a circle arc based on the current drawing settings.	Circle by center and radius Circle by diameter Circle by 3 points
Region	Defines a drawing region with arbitrary shape. Also holes can be added to the region.	Rectangular Circular Polygonal Pick lines (close contour)
Solid	Defines prismatic, centric, sphere, regular and general 3D solid surface. Generic solid can be generated with the help of arbitrary shapes and directrix. Solids can be also cutted in their heights.	Prismatic - Rectangular base (brick/cube) - Circular base (cylinder) - Polygonal base (prism) - Pick lines (arbitrary base) - Pick existing region (arbitrary base)

		Centric - Rectangular base (pyramid) - Circular base (cone) - Polygonal base - Pick lines (arbitrary base) - Pick existing region (arbitrary base) Sphere - Sphere - Segment of sphere Regular - Tetrahedron - Cube - Octahedron - Dodecahedron - Icosahedron General (arbitrary shape with directrix)
Text	Defines arbitrary text in custom style and any direction.	-
Field	Places autotext in custom style and any direction.	-
Dimension	Defines length, angle and level dimensions in custom style.	Linear (Length) Arc Diameter Radius Angle Level dimension
Title information table	Places a table that displays title Information of the current project. The content of the table can be set by *Settings* > *Title*. The width of the information table is defined by	-

	the text size and style (*Settings*> *Text*). Table is placed in the Title information object layer. Layer color defines the table color too.	

Table: Draw menu tools

The following chapters give more details for some drawing items.

Solids

This chapter shows some example for available solid geometries and introduces how general solids can be defined with the help of shape contour and directrix.

⚠ There are no real curved surfaces in 3D modeling, so the program approximates the curved surfaces with polygonal planes. The value of approximation can be set by the *Refracting angle* option (*Primary*/ *Secondary*). The next figure shows the meaning of refracting angle values for a double-curved surface (defined by *General*).

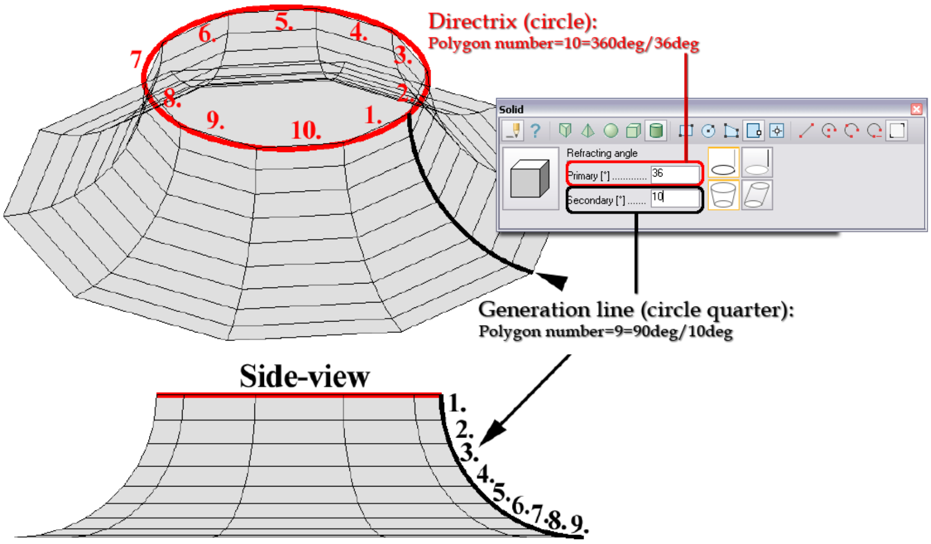


Figure: Curved surface approximation

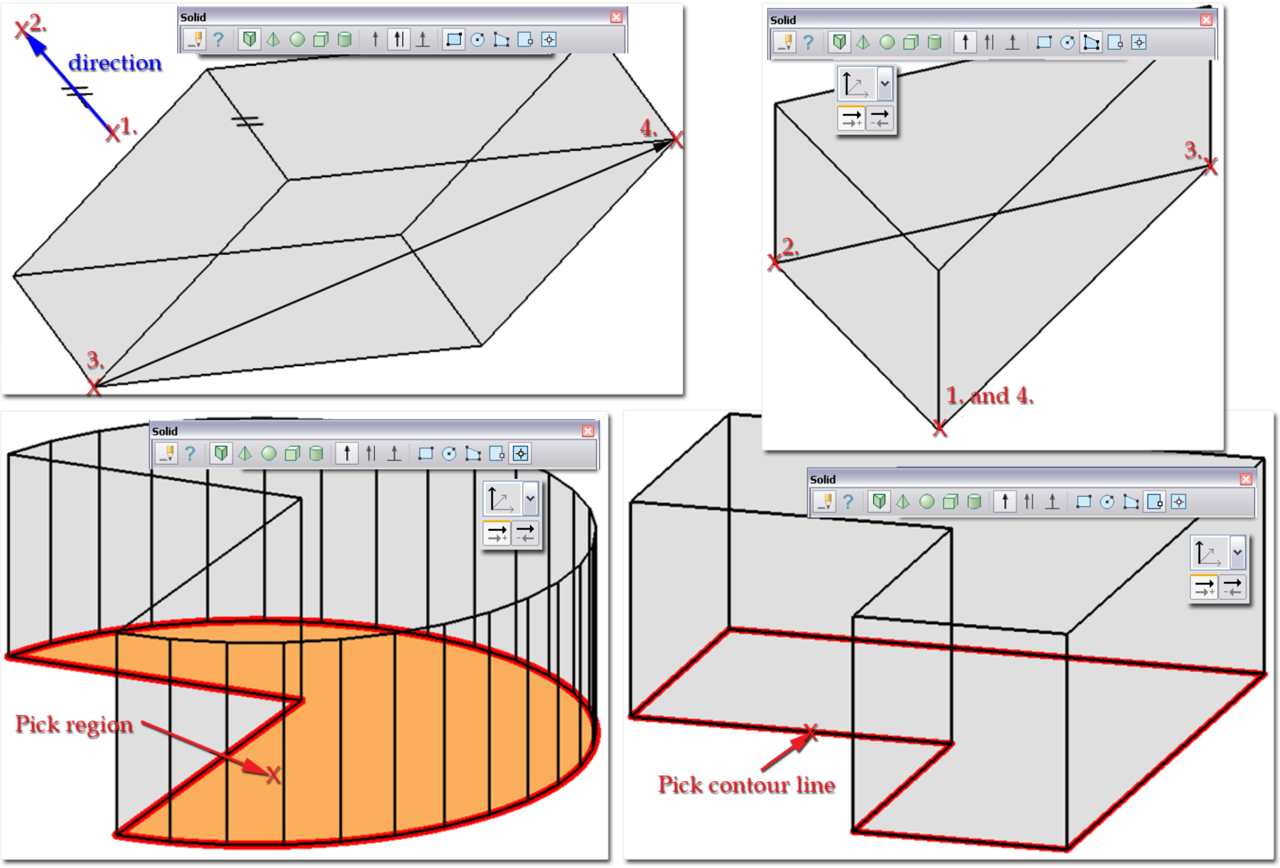


Figure: Some examples for prismatic solids with definition steps

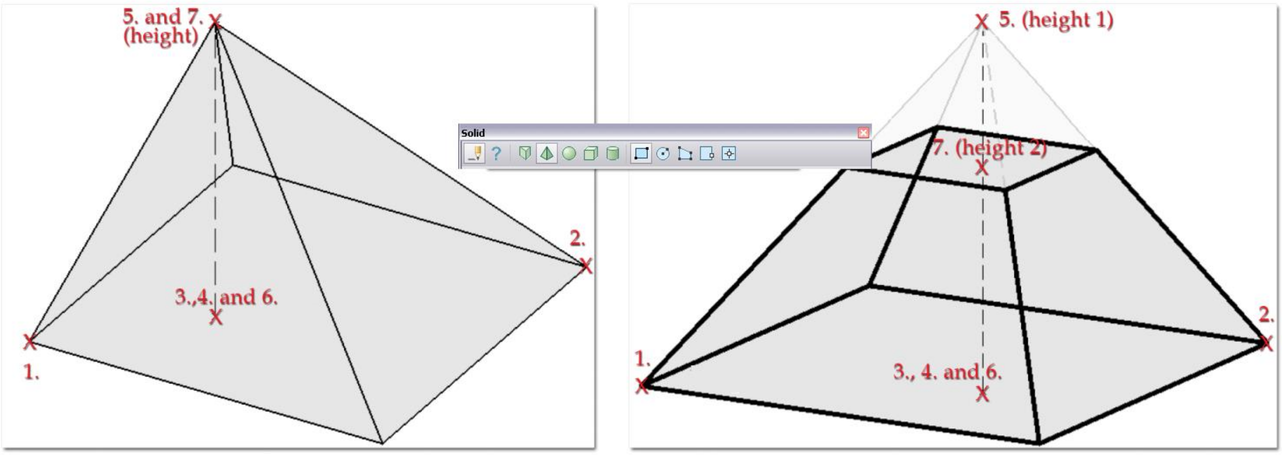


Figure: Full or truncated pyramid (Centric solid)

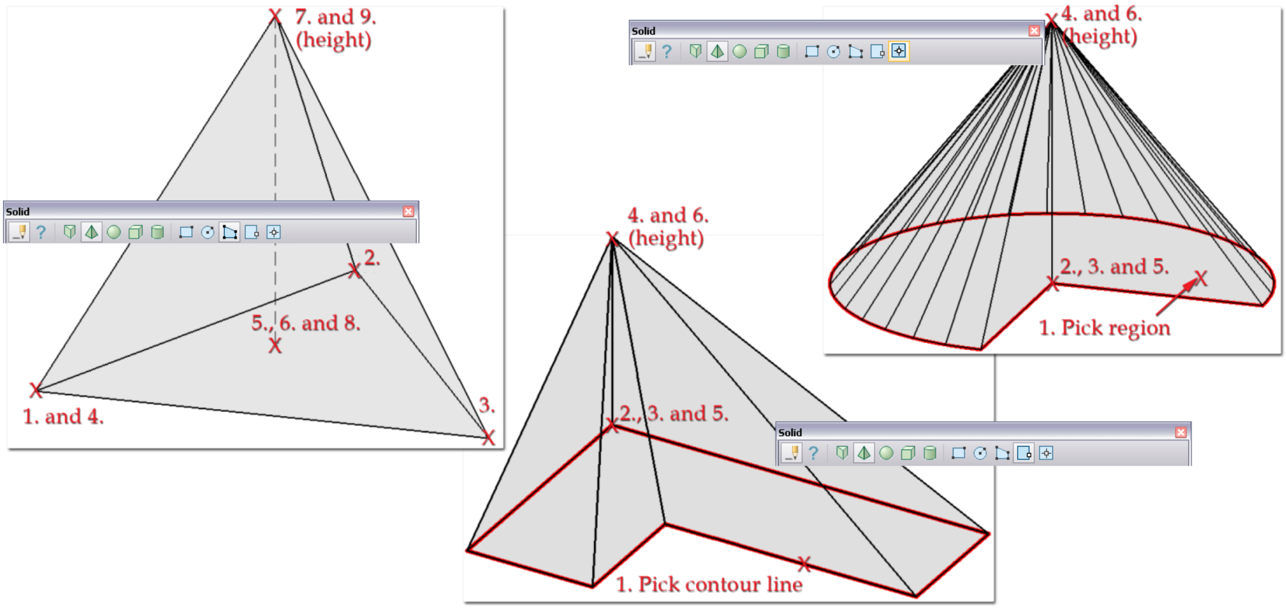


Figure: Some examples for centric solids with definition steps

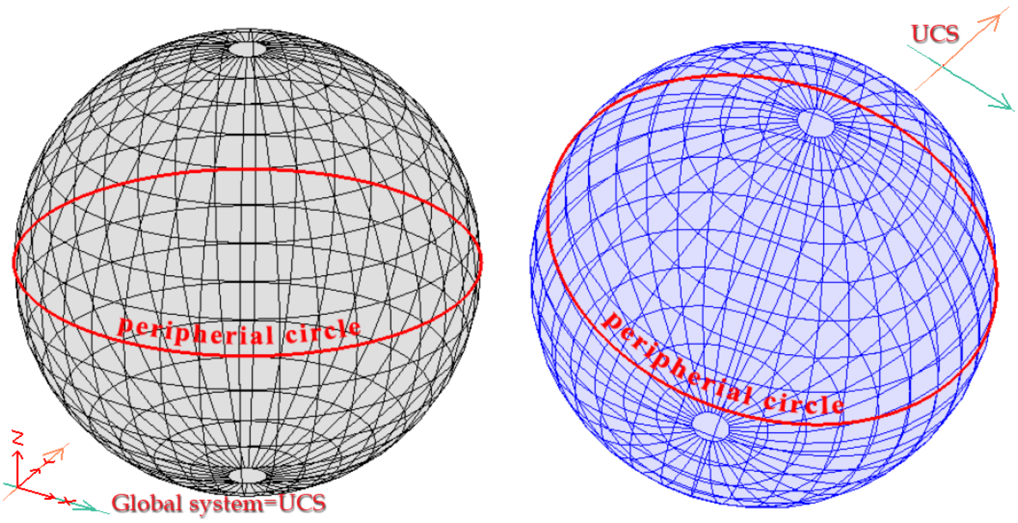


Figure: Spheres defined in different coordinate-systems

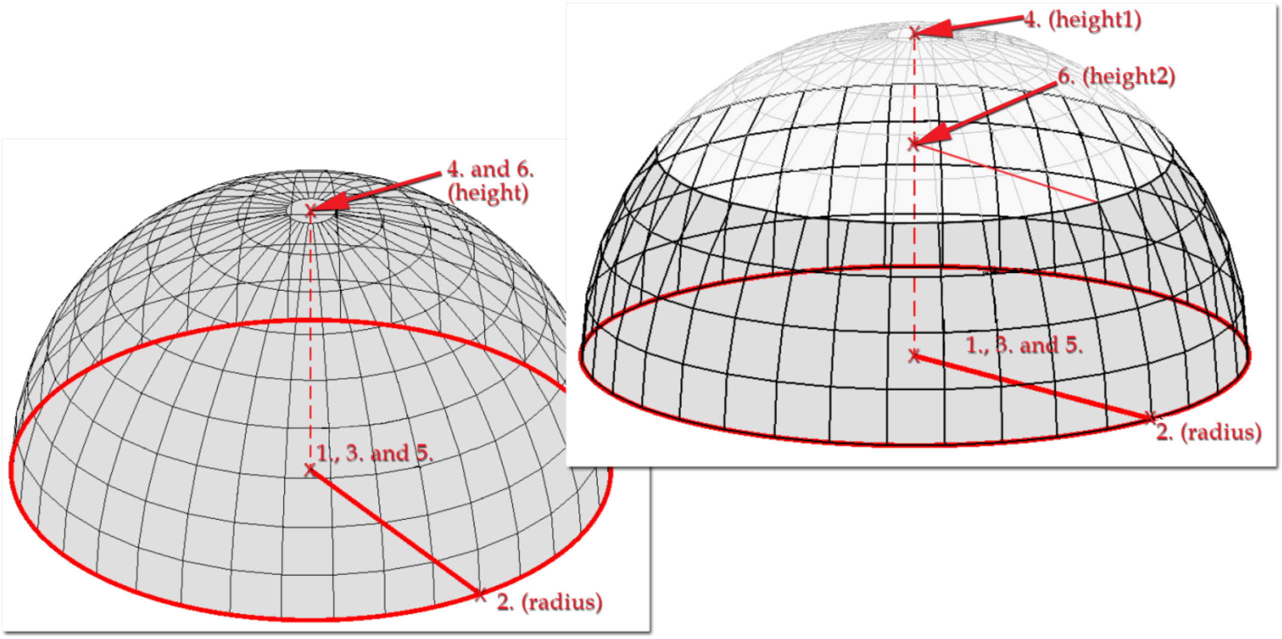


Figure: Half and truncated spheres

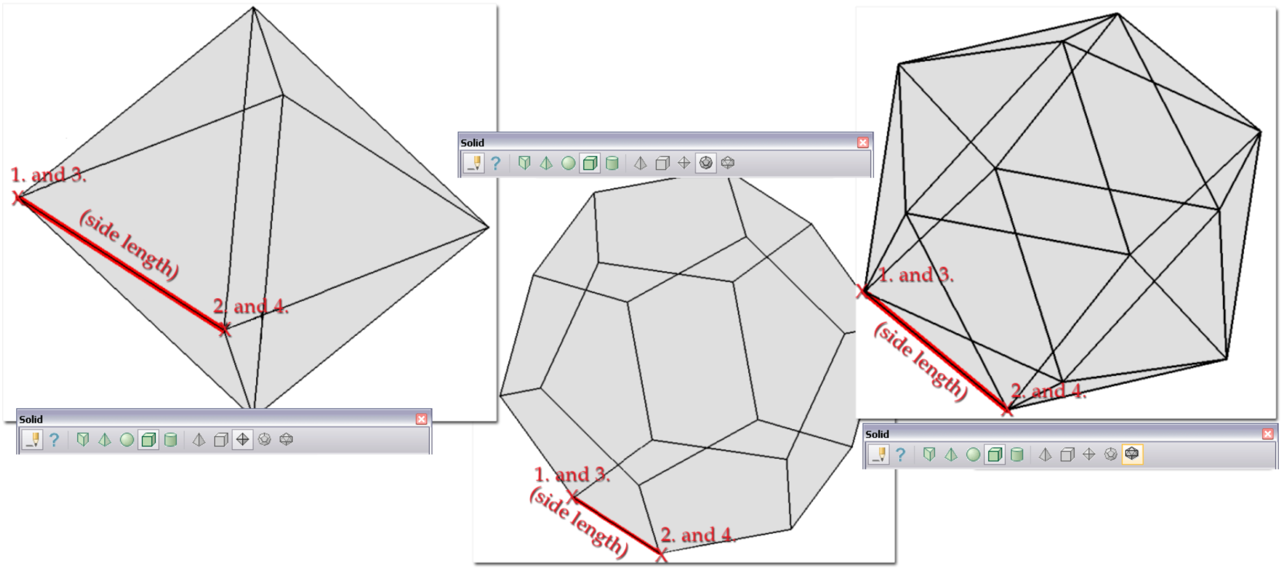


Figure: Octa-, dodeca- and icosahedron

With *General* tool, general 3D solids can be defined with a directrix and a generation line (builds up the surface of the solid). Both of them can be a closed contour line or an arbitrary line (it can be also closed), but one by one. The only rule is that they must touch each other. Five tools can be used for defining the contour geometry and other five ones for defining the line geometry. The function of the contour line or the arbitrary line can be set with the *Directrix is contour line* (and the generation line is the arbitrary line) or the *Directrix is line* (and the generation line is the contour line) tools. The final shape of the solid depends on the *Perpendicular* and *Constant* options that define how (in what angle) the generation line will be transformed around the directrix:

- *Perpendicular*: the solid sections having the same shape with the generation line will be perpendicular to the directrix in its all points.
- *Constant*: the solid sections (inner surfaces) will be parallel with the generation line along the directrix.

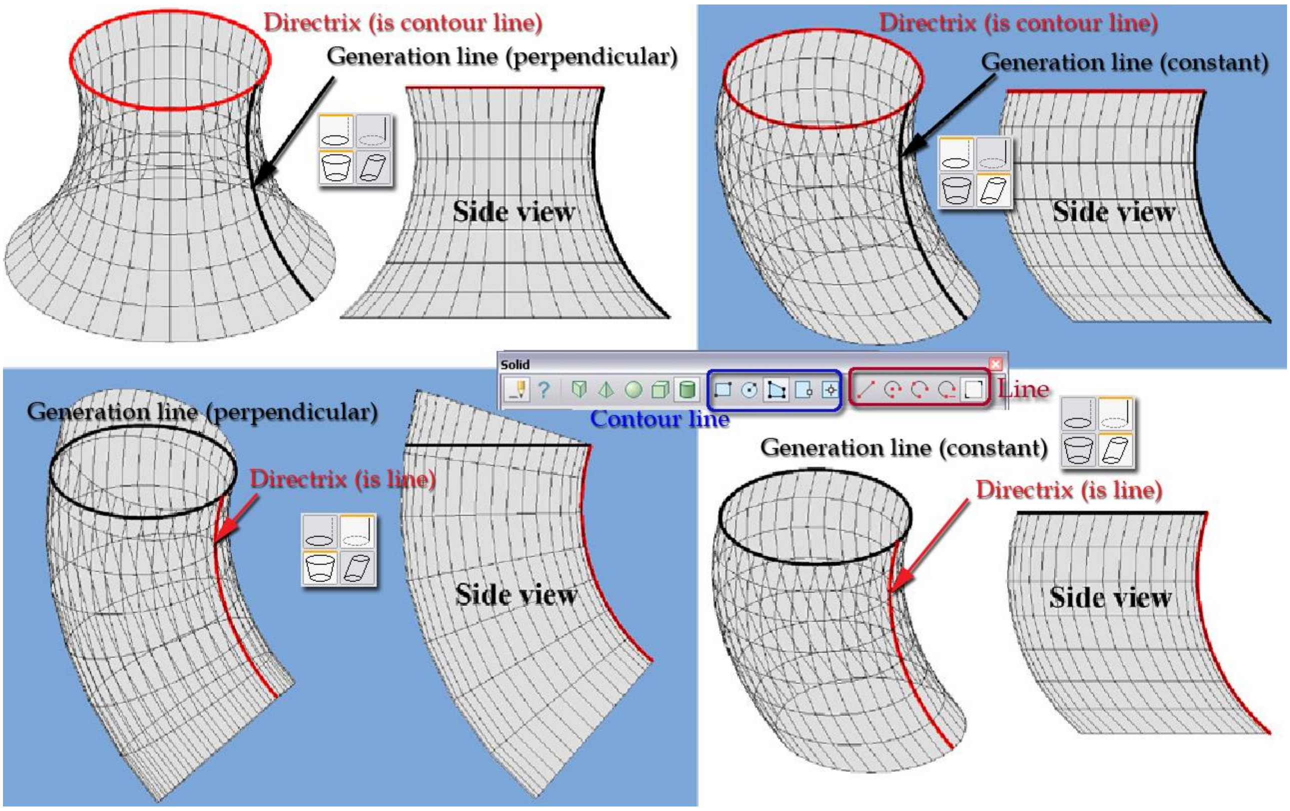


Figure: Different combinations of directrix and generation line

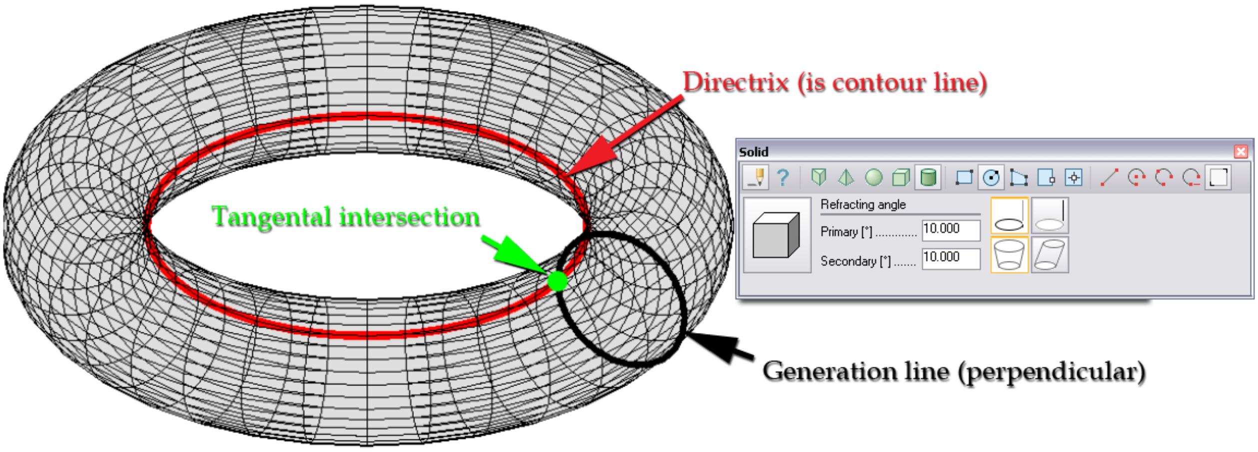


Figure: Both directrix and generation line are closed contours

Text and Field

Custom (*Text* command) and automatic (*Field* command) texts can be placed on the drawing area in arbitrary position. They can also be edited any time.

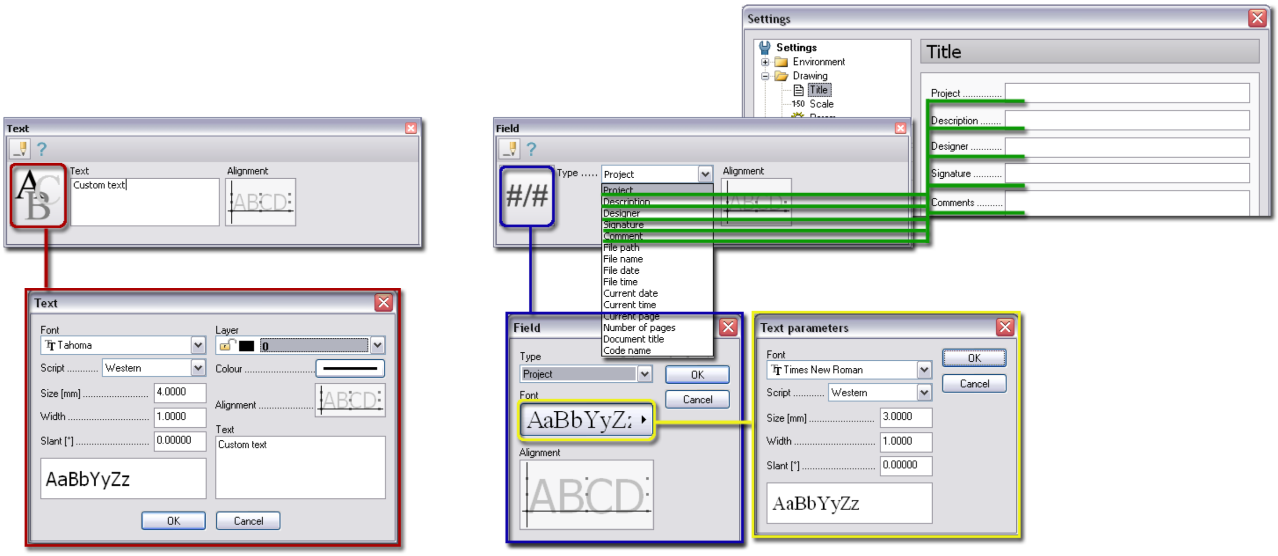


Figure: Text and Field commands with their settings

At *Text*, the custom text can be defined in the *Text* field. At Field, the content of autotext can be set at *Type*. Some autotext types (such as *Project* title, *Description*, *Designer*'s name, *Signature* and *Comments*) are defined by the user at *Settings* > *Title*, and the other ones automatically set by the current project settings (such as *File name*, *File date*, *Code name* etc.). Field is especially useful in **Documentation**, as you can see it from the range of autotext types.

The following text properties can be set for both text and field items:

- **Font**

  Choose the required font type from available MS Windows offered types.

- **Script**

  Script solves the character problems at different languages. An example for the Times New Roman font: character "ő" appears as "õ" in *Western*, but as correct "ő" in *Central European* script.

- **Size**

  It is the height of the font in millimeters.

- **Width**

  It is the width of the font. Its value that is lower than 1 defines narrower letters. Its value that is bigger than 1 results thicker letters.

- **Slant**

  You can set the slant of the letters in degrees that closes angle with the vertical direction. The angle is measured clockwise. The standard italic font has about 10-15 degree slant.

- **Alignment**

  In the *Alignment* figure you can set the position of the text insertion point. The hatched rectangle is the border of the text and the insertion point is in the intersection of the two black lines. Moving the black lines with the mouse offers nine possible positions for the insertion point.

To place the text/field on the drawing area, give the coordinates of the insertion point, and then define its direction. The mouse button sets the direction parallel ("horizontal") with the X direction of **UCS**.

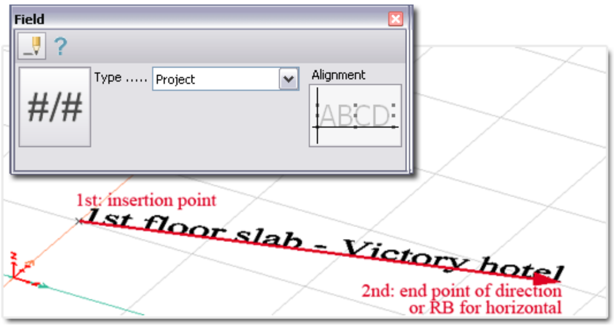


Figure: Text position

Special characters can be defined in *Text* field by typing "%" before letters and numbers. These are the followings:

- %g = ° (degree symbol)
- %d = Ø (diameter symbol)
- %p = ± (plus/minus symbol)
- %% = %
- %2 = ² (square sign)
- %3 = ³ (cube sign)

Dimension

With *Dimension*, length, angle and level dimensions can be defined in custom style. The style of dimensions can be set by *Default settings*.

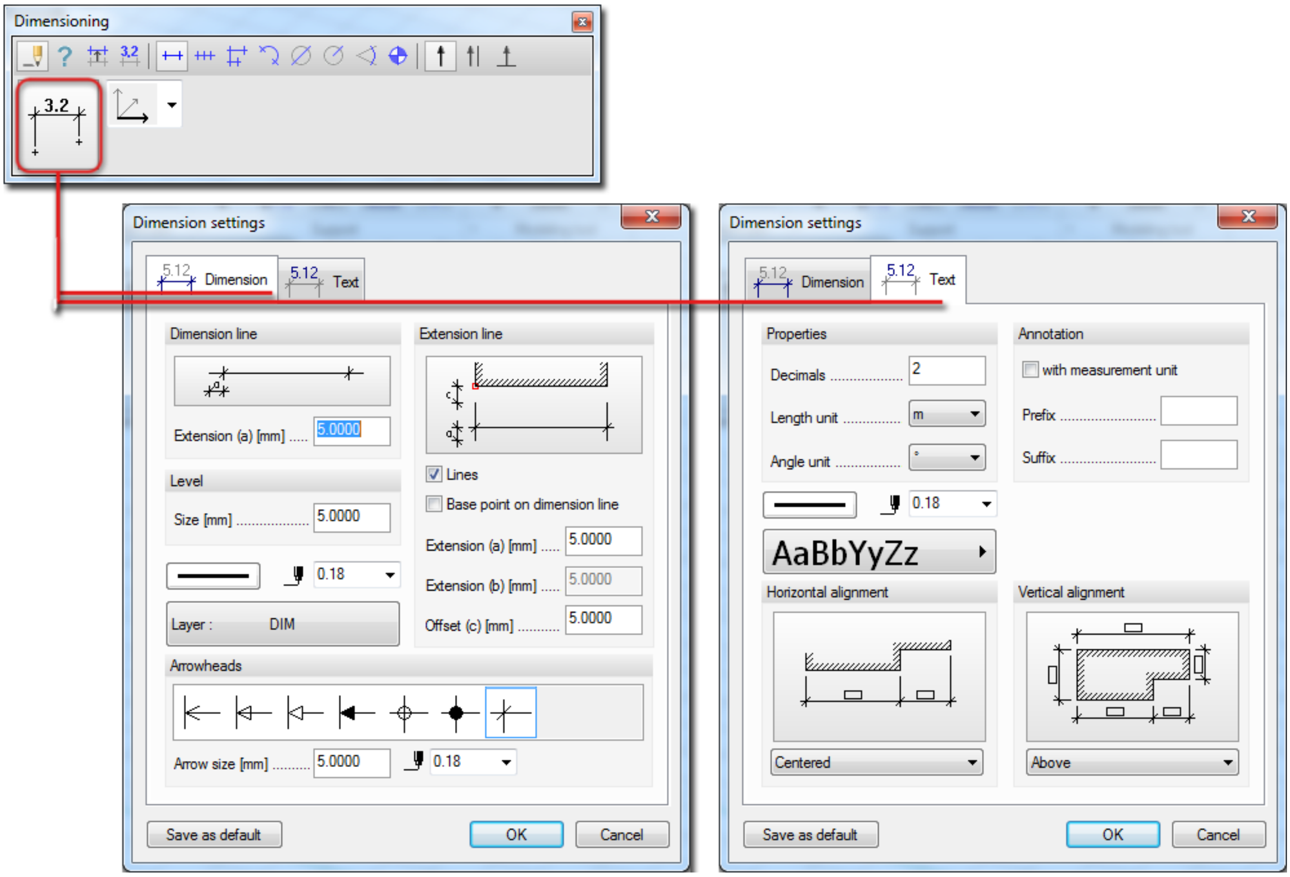


Figure: Dimension settings

- **Dimension line extension**

  Dimension line can be extended with *a* value.

- **Level**

  The size of the  *Level dimension* symbol can be set with *Size* value.

- **Color and Pen width**

  You can choose a color (black by default) for all dimension items, and set a pen width for the lines.

- **Layer**

  Drawing layer ("DIM" by default) can be chosen for the new dimension items. Of course, you can define and use different layers by position (external/internal), by direction (horizontal/vertical), by type (length/level dimension) for example.

- **Extension line**

  Sizes, positions and display settings can be set for extension lines.

- **Arrowheads**

  The style, the size and the pen width of the arrowheads can be set here.

- **Text display properties**

The text style, the decimal numbers and the measurement unit type (independently from the project unit settings) of the numeric values can be set here. Additional text can be inserted before and after the measured values with *Prefix* and *Suffix*. Also **special characters** can be used as dimension text.

- **Horizontal / Vertical alignment**

  The alignment of text to dimension lines can be set by direction.

Dimension types

From the *Dimension* tool palette chose the required type of the new dimension.

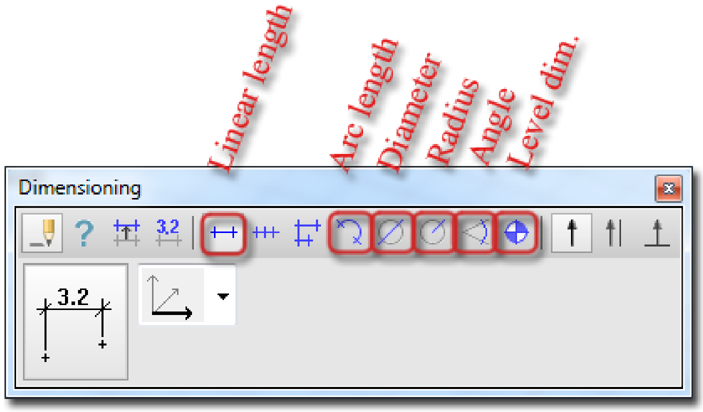


Figure: Dimension types

Different directions can be set for  *Linear* type dimensions:

- **Predefined direction**

  The direction of dimension lines can be set parallel with one of the axis directions of the **UCS** or the **global system**.

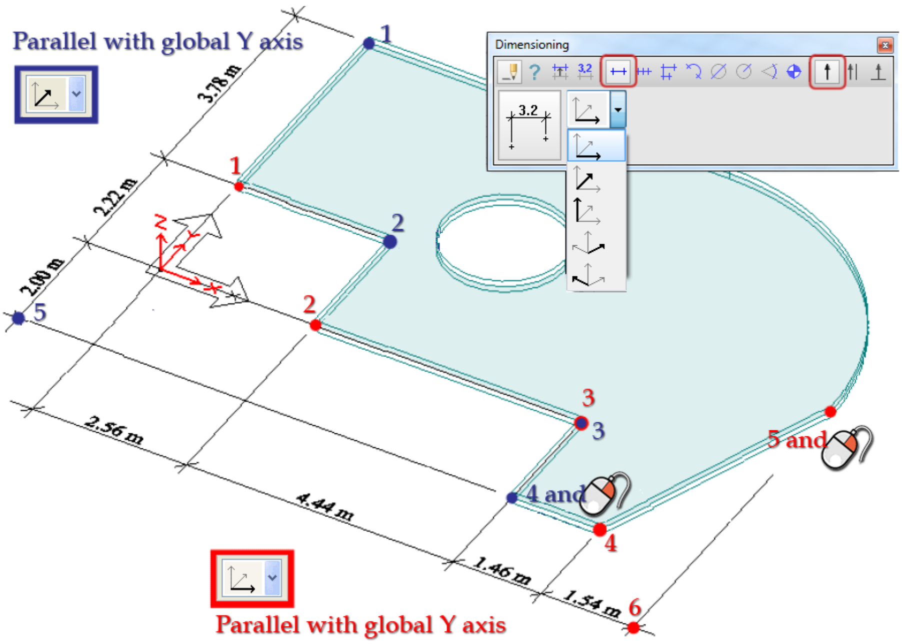


Figure: Dimensions parallel with global axes

- **Parallel with line**

  The direction of dimension lines can be set parallel with a line defined by two points.

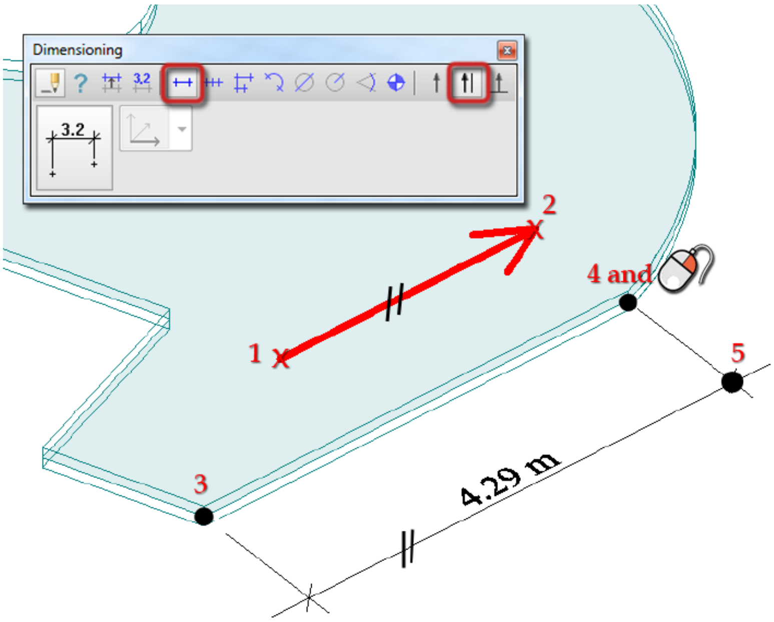


Figure: Dimensions parallel with a given line

- **Perpendicular to plane**

  The direction of dimension lines can be set perpendicular to a given plane (e.g. the plane of a wall).

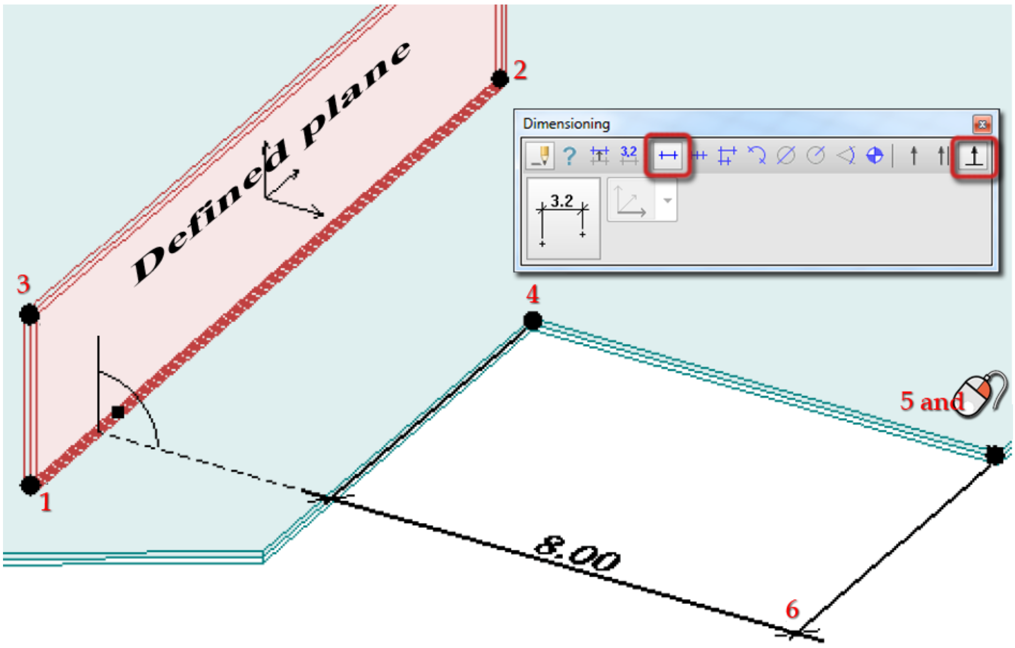


Figure: Dimensions perpendicular to a given plane

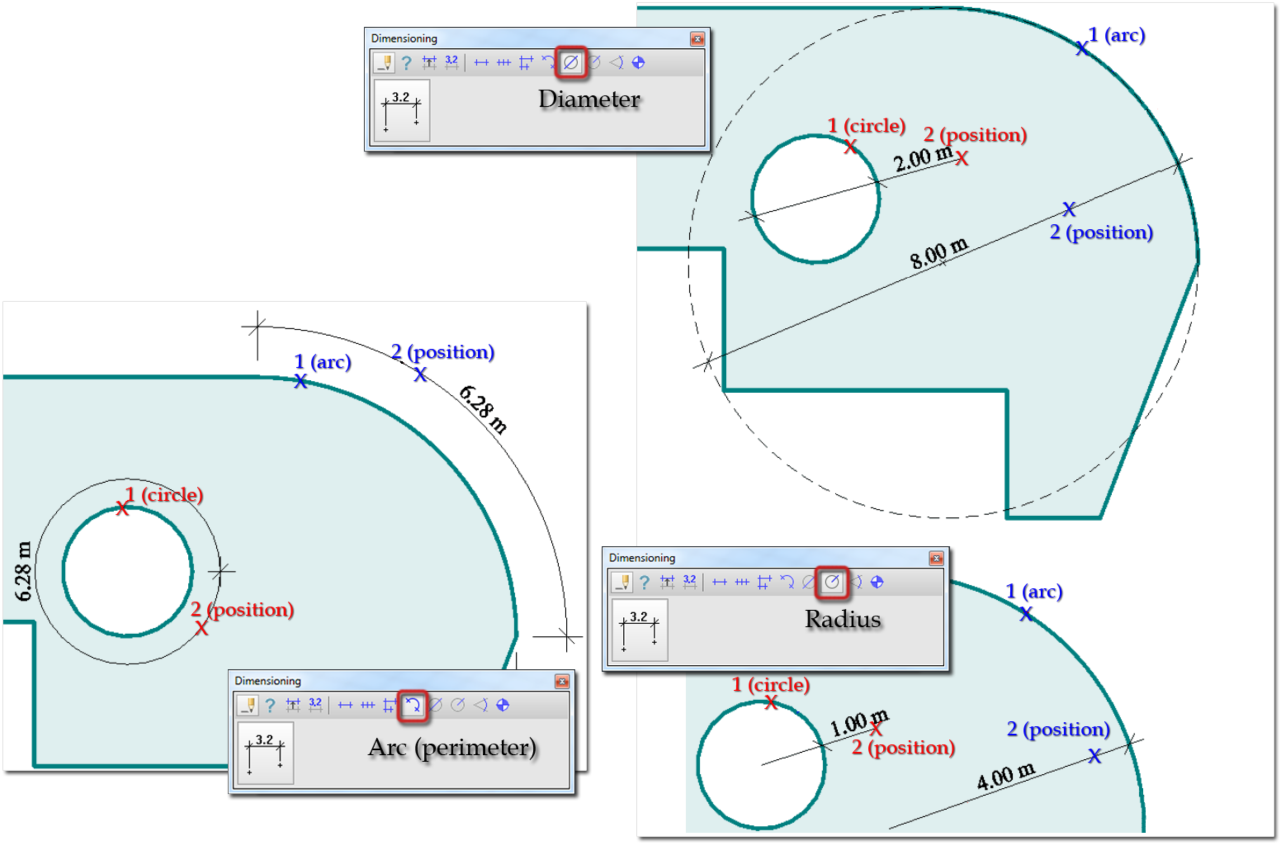


Figure: Dimensions of arcs and circles

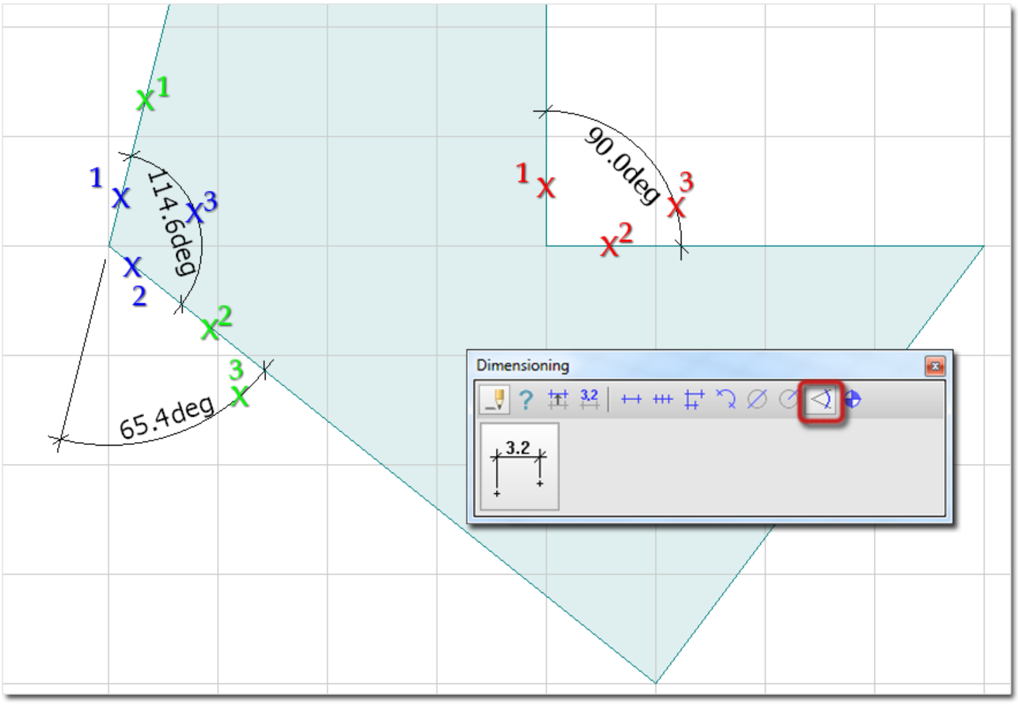


Figure: Dimensions of angles

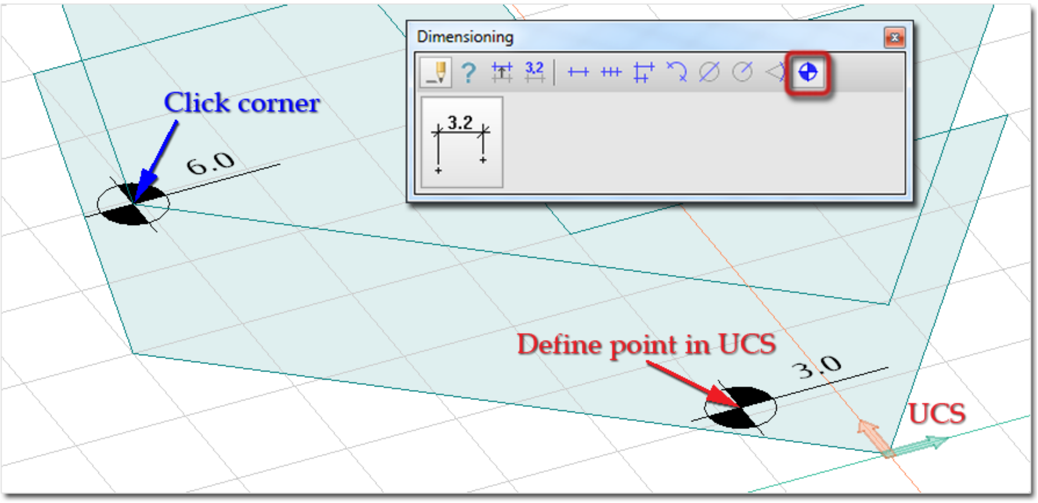


Figure: Level dimension

Modification in dimension

- **Modify line dimension**

  With this tool you can add new (reference) points to a selected predefined linear dimension line. First select the required dimension line, and then place the new point.

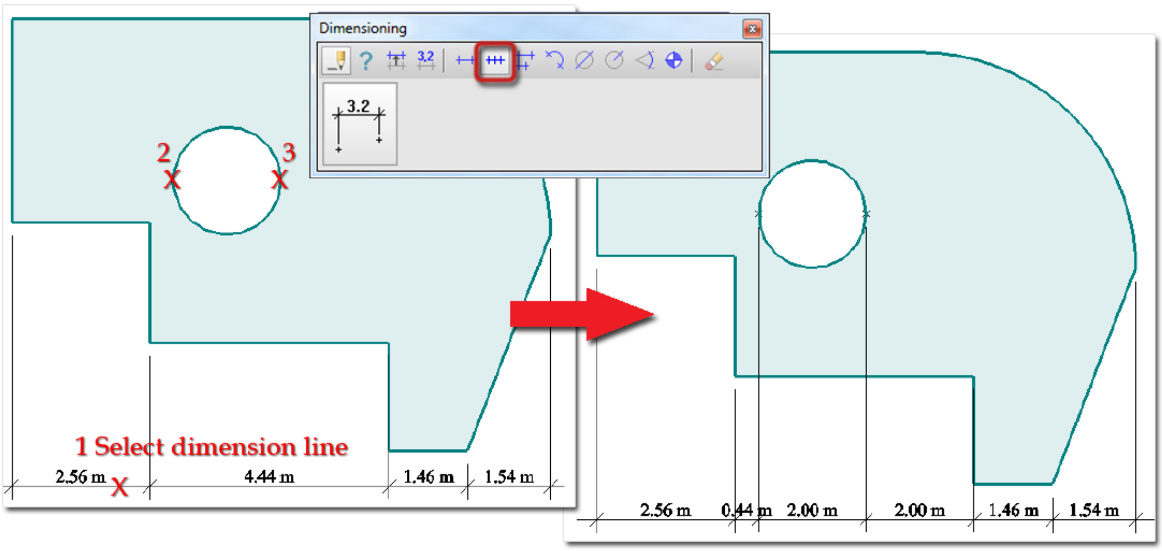


Figure: New points added to dimension line

- **Add dimension to base line**

  New dimension lines can be added to a baseline of an existing linear dimension line. This means, that the starting point of the new lines is at the selected baseline of the dimension line. First select the dimension line, then choose the side or baseline, and finally add the new points. The new lines will inherit all properties from the selected one. The location of the selected dimension line determines that the new lines will be above it or below it.

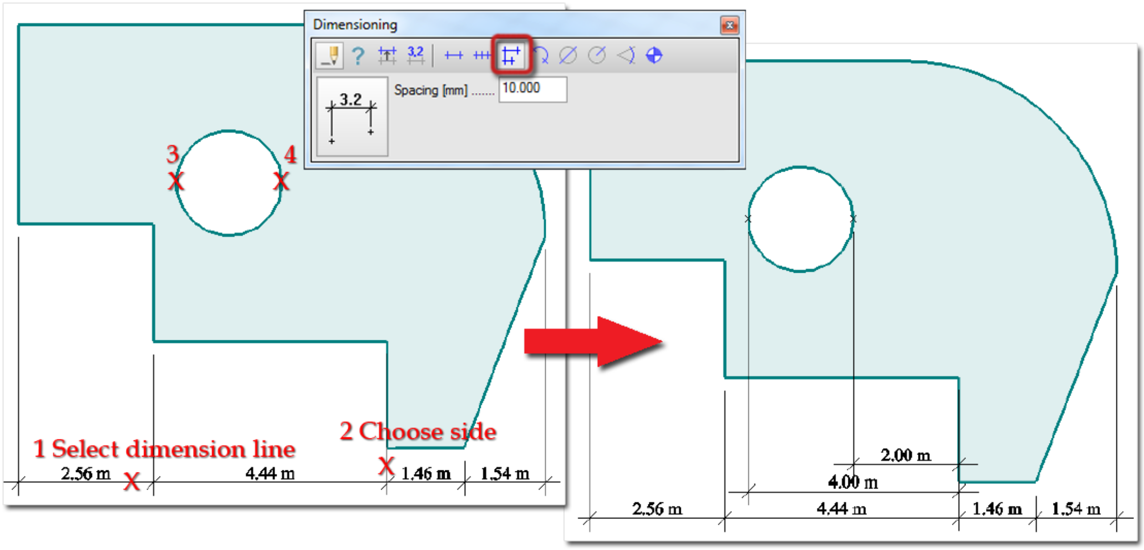


Figure: New dimension line assigned to a base line of an existing one

- **Dimension line position**

  The position of a predefined dimension lines can be modified easily. Just select the required dimension line, and then define its new position.

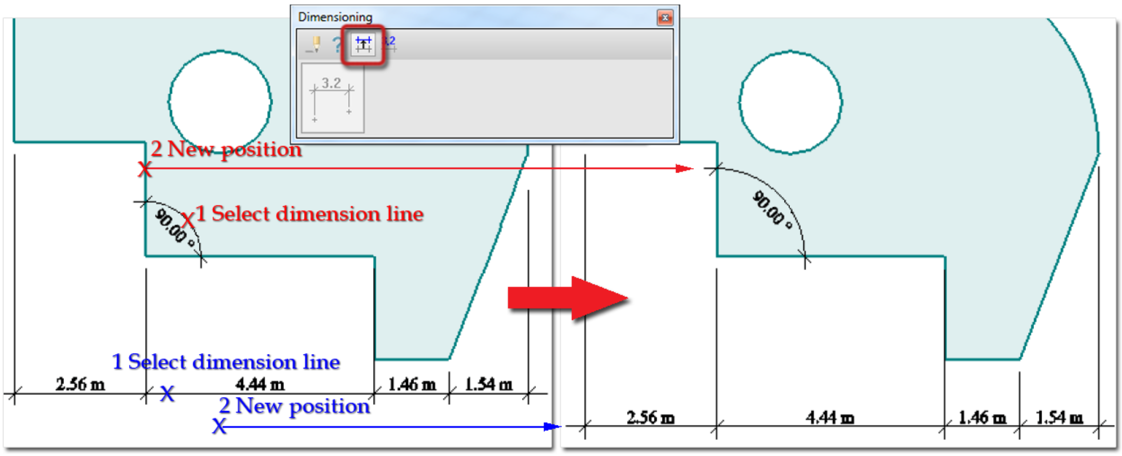
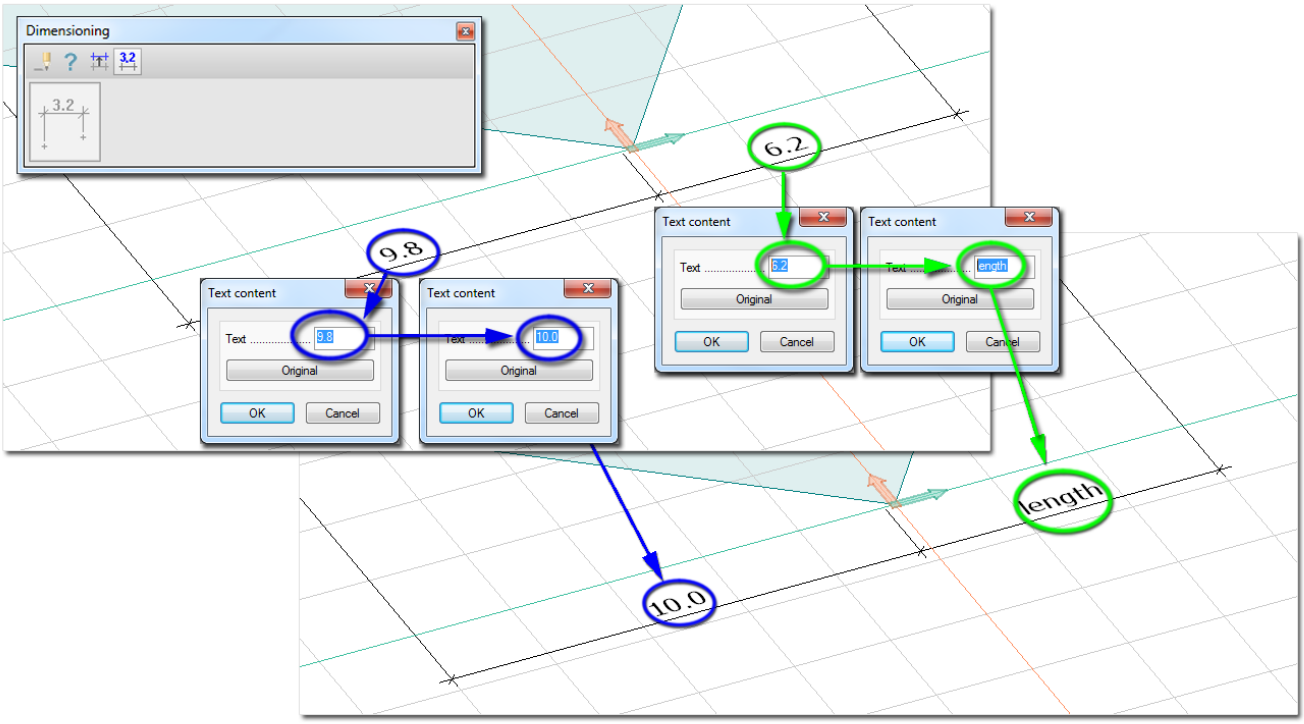


Figure: Modification in dimension line positions

- **Text content**

  The real measured values can be substitute with any custom text. Not only numbers but letters can be added as text content. Do not forget to hide unit system, if you want to display only the new text. First select the text value, and then type the required text in the *Text content* dialog. Clicking the *Original* button restores the measured dimension value and overwrites the custom, the user-modified text.

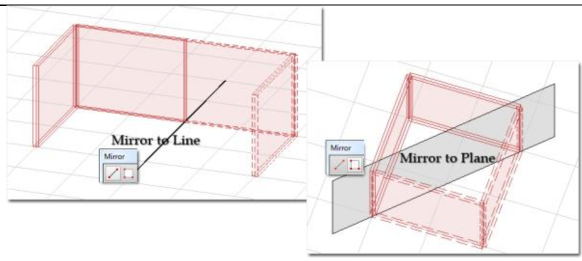
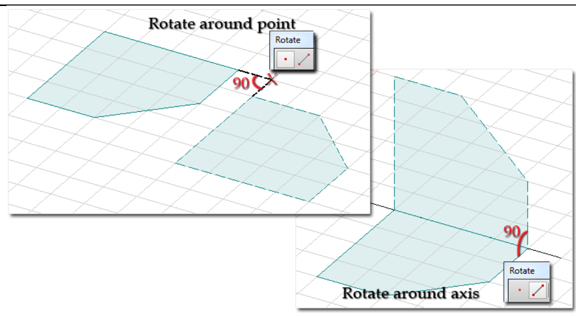
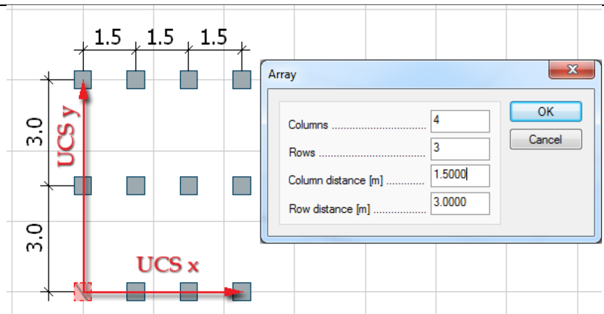


Edit and Modify Menu Commands

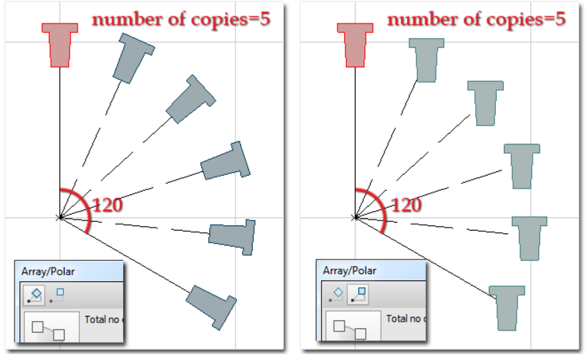
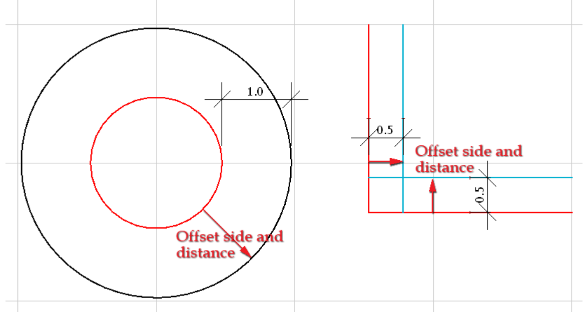
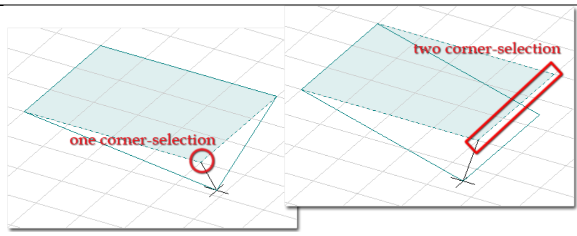
The *Edit* menu contains tools for editing and modifying drawing elements, structural elements, loads, reinforcement objects, result data, etc. The following table summarizes the editing tools, their function and examples.

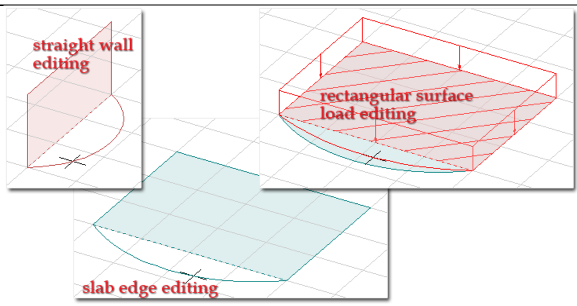
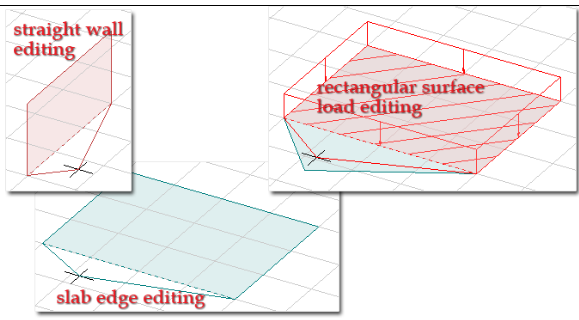
Edit Command	Function	Example
Repeat command	Reruns the last used command.	
Undo	Undoes the last operation such as the editing operations summarized in this list. The maximum numbers of undoable operations can be set at *Settings* > *Environment* > *General* > *System* > *Undo steps*. Calculation operations and previous commands cannot be undone. Undo command does not close the current action window only resets it.	
Redo	Executes the previously undone command. Redo command does not close the current action window only resets it.	

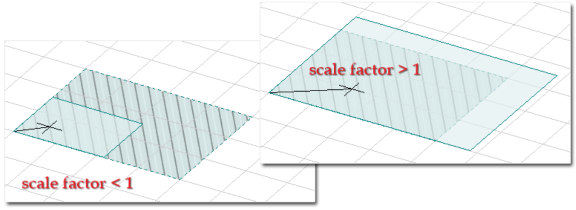
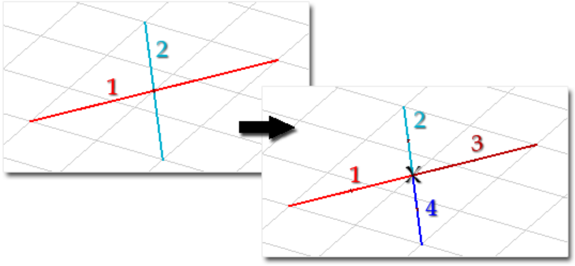
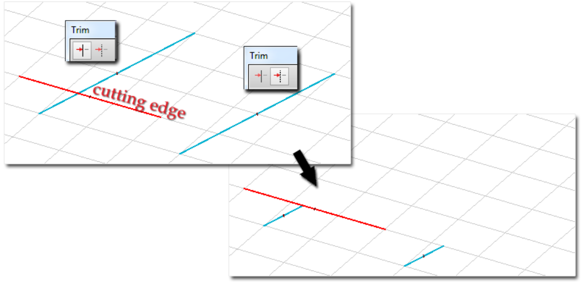
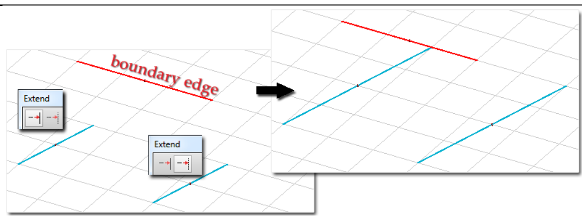
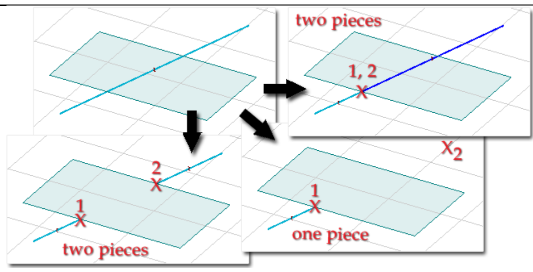
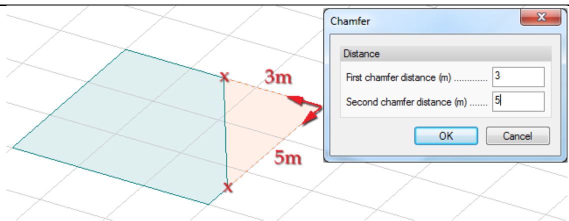
Cut	Erases selected elements of the project and places it onto the *Clipboard* in order to *Paste* them later to the drawing area. An insertion point is required for the later insertion (paste).	
Copy	Places selected items on the *Clipboard*. It does not delete the original objects. An insertion point is required for the later insertion (paste).	
Paste	Inserts the FEM-Design-type drawing contents of the *Clipboard* into the current project's drawing area. The elements are placed with the coordinates of the insertion point defined by *Cut* and *Copy*. Paste together with Cut and Copy is working between more than one opened projects too.	
Erase	Deletes selected objects from the current project and so from the drawing area. Some special objects can be deleted with their own eraser tool: - Holes of planar regional elements (plates, walls etc.) can be deleted with the *Remove hole* tool of **Modify region** command. - Finite elements of a mesh can be deleted with the *Delete* tool of the *Finite elements* tabmenu.	
Copy bitmap	It exports the selected part of the drawing area to the *Clipboard* as an image, and it can be inserted to other programs such as photo editors, MS Word, MS Excel, etc.	
Paste file	Inserts the whole geometrical content (drawing and structural elements, loads, finite element mesh) of a FEM-Design file into the current project. The insertion point of the copied content is the global system origin of the pasted file.	-

	The loaded file must be compatible with the current **FEM-Design module**.	
External reference	It imports DWG/DXF drawings as an image in the background	
Modify commands	**Function**	**Example**
Move	Moves objects or their copies with a given distance (displacement vector) in the drawing area.	
Copy	Makes more than one copies of selected objects. Each copies can be moved with different distances. The base point of each copies are the same point of the selected elements.	
Mirror	Mirrors selected objects or make a mirrored copy of them. Elements or their copies can be mirrored to a given line or plane.	
Rotate	Rotates selected objects or make a rotated copy of them. Elements or their copies can be rotated around a given point or line with an accurate angle or a defined new position.	
Array > Rectangular	Multiplies selected objects as row (UCS y direction) and column (UCS x direction) distribution. The number of copies and the distance between the copies by direction are required.	

<table>
<tr><td>Array > Polar</td><td>Multiplies selected objects by rotating them around a centre point.

Required data: direction of rotation (Clockwise/Counter-clockwise), the origin of the rotation, the rotation angle and the number of copies.</td><td>
</td></tr>
<tr><td>Offset</td><td>Makes a copy of line-type objects (line, arc, region contour, beam, column etc.) that will be placed parallel with the original object with a given distance and on the defined side.</td><td>
</td></tr>
<tr><td>Stretch</td><td>Stretches line-type elements or the selected corners of closed polygons and region-type elements.</td><td>
</td></tr>
<tr><td>Curve</td><td>Curve a line-type element or an edge of region-type elements.

Clicking Enter on a curved line straigntens that.</td><td>
</td></tr>
<tr><td>Elbow</td><td>Breaks a selected line or region edge, or in other word, it adds new point to lines and edges in new position.</td><td>
</td></tr>
</table>

Scale	Scales objects, blocks, even the entire structure. A base point (will be the only point that will remain in its original place after scaling) and the scale factor are required for scaling	
Split	Automatically splits selected line-type elements in their common intersections.	
Trim	Cuts line-type objects along an edge defined by another object. The cutting edge may have a real or virtual intersection with the selected line wanted to be trimmed.	
Extend	Extends line-type objects to an edge defined by another object. The boundary edge may have a real or virtual intersection with the selected line wanted to be extended.	
Break	Cuts a section from line-type object and breaks it into two independent object. *Break* can be used to cut a line into two parts by double clicking in the same line point (zero section length).	
Chamfer	1. Cuts the corner of a closed polygon and region. 2. Connects two line-type objects with a skew line defined by the chamfer distances.	

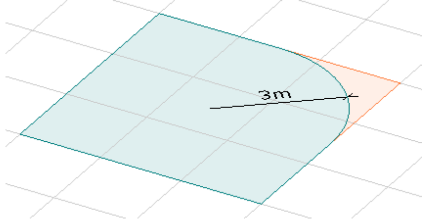
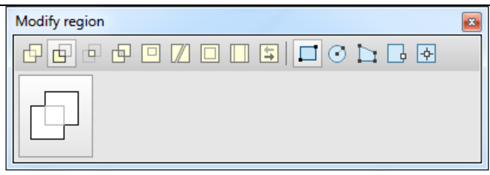
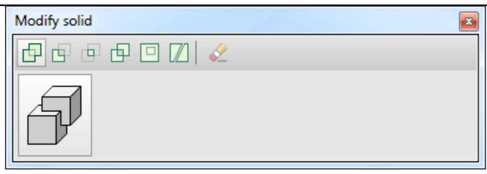
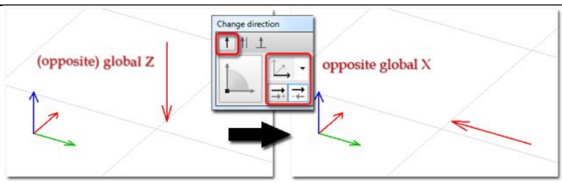
Fillet	1. Rounds the corner of a closed polygon and region. 2. Connects two line-type objects with an arc defined by the fillet radius.	
Modify region	1. Removes hole from region-type object. 2. Splits region by cutting edge defined manually or by the intersection of another region. 3. Reduces/increases region area with parallel offset of region edges. 4. Swaps faces of wall/plate regions (mirrors local system). 5. Does logical operations (union, substarct, intercection and excluded "OR") between regions.	
Modify solid	1. Removes hole from drawing solid. 2. Splits solid by cutting region/plane defined manually or by the intersection of another solid. 3. Does logical operations (union, substarct, intercection and excluded "OR") between solids.	
Change direction	Modifies the direction of the following object types: - column and beam section position, - support, - load, - parametric reinforcement, - applied reinforcement bar.	
Change appearance	1. Changes display properties of drawing and structural objects such as color, line type, pen width, text style etc. 2. Matches properties of an object with others.	

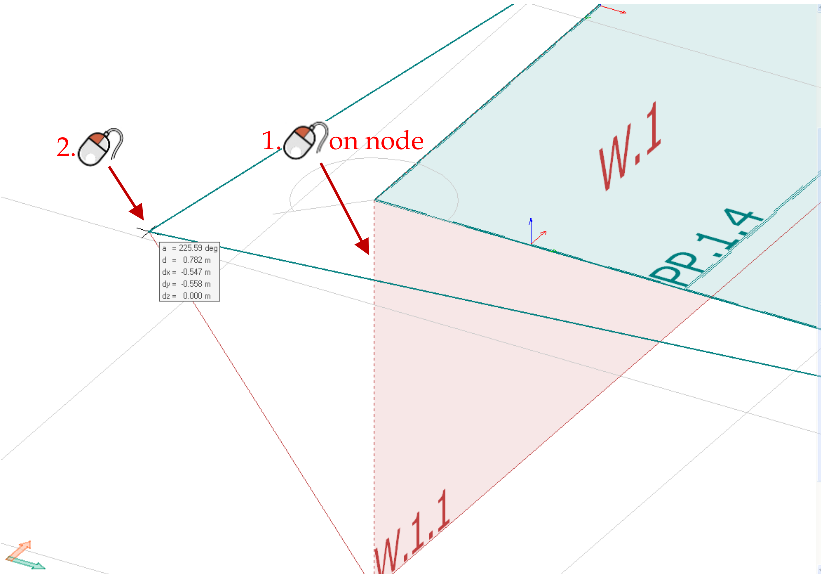
Table: Edit menu tools

Quick modifications

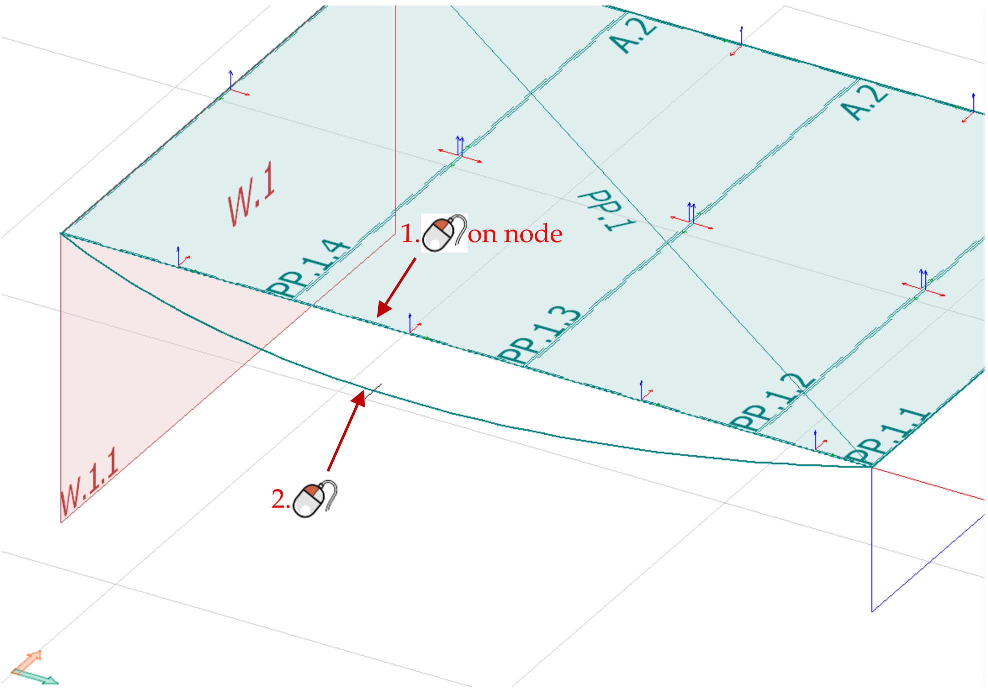
By clicking right mouse button the quick menu appears. After clicking on an arbitrary element all the commands of quick menu are visible but those that are not available for the chosen element(s) are disabled.

	Click LB (default)	**Click + hold LB**	**CTRL + Hold LB**
Line's end point **Node on surface edge**	Stretch	Stretch	Drag a copy
Along point along line	Curve	Move	Drag a copy
Any point inside Surface	Move	Move	Drag a copy
Any point on screen, where there is nothing	Box selection		

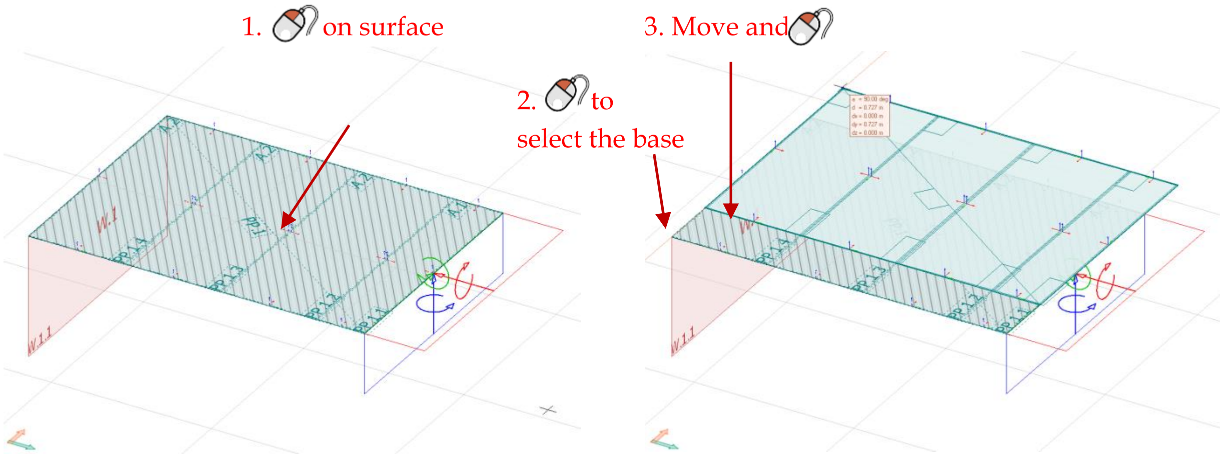
- At  user can select another command in the Modify menu which is also available by .
-  on line's end point or node on surface edge starts 'Stretch' command or the last used command selected by the user for this case



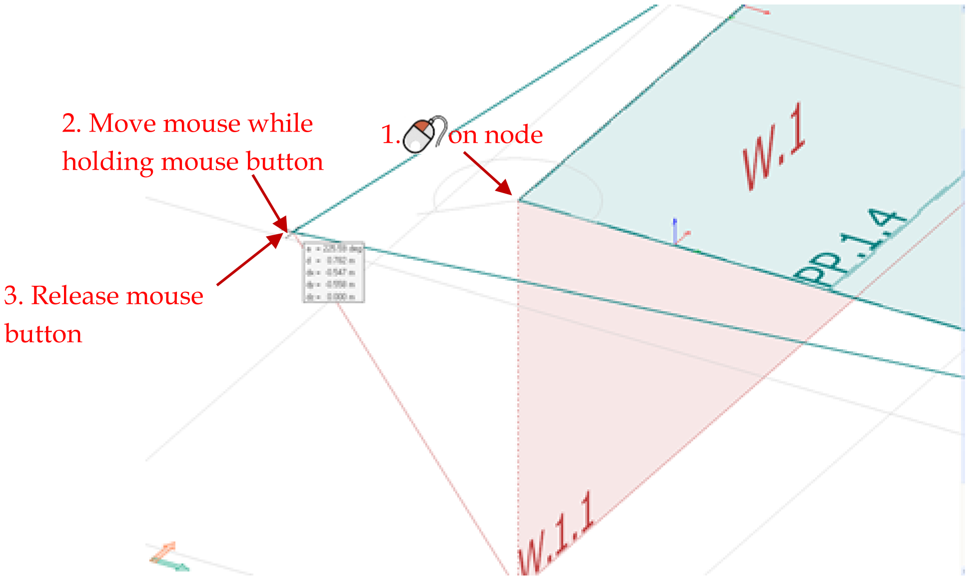
- on any point along the line starts 'Curve' command or the last command



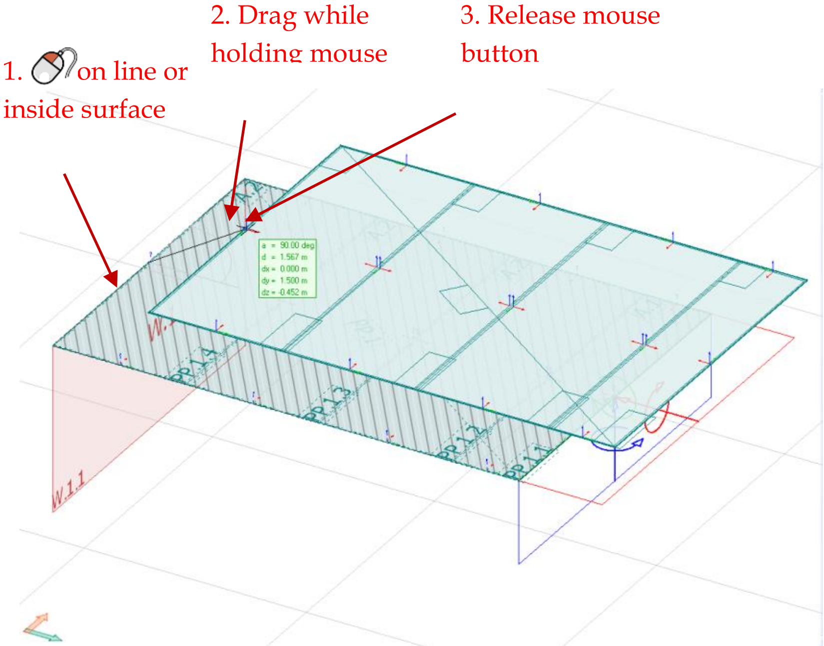
- on any point inside on surface starts ‘Move’ command



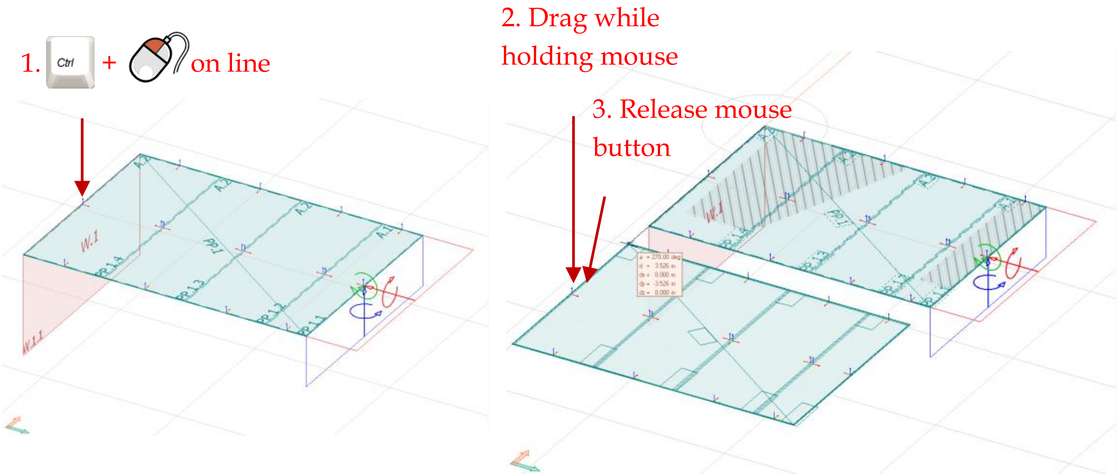
- and holding at line’s end point or surface edge’s end node starts ‘Stretch’ command



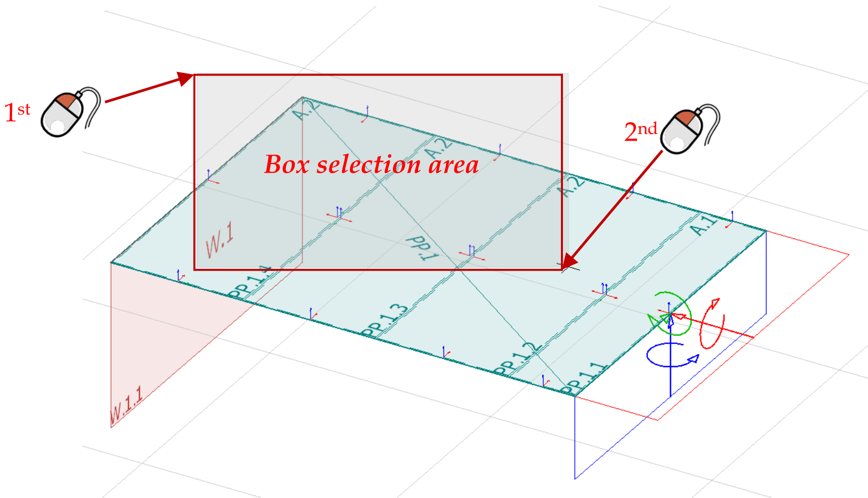
-  and holding at any point along line or any point inside surface starts 'Move' command



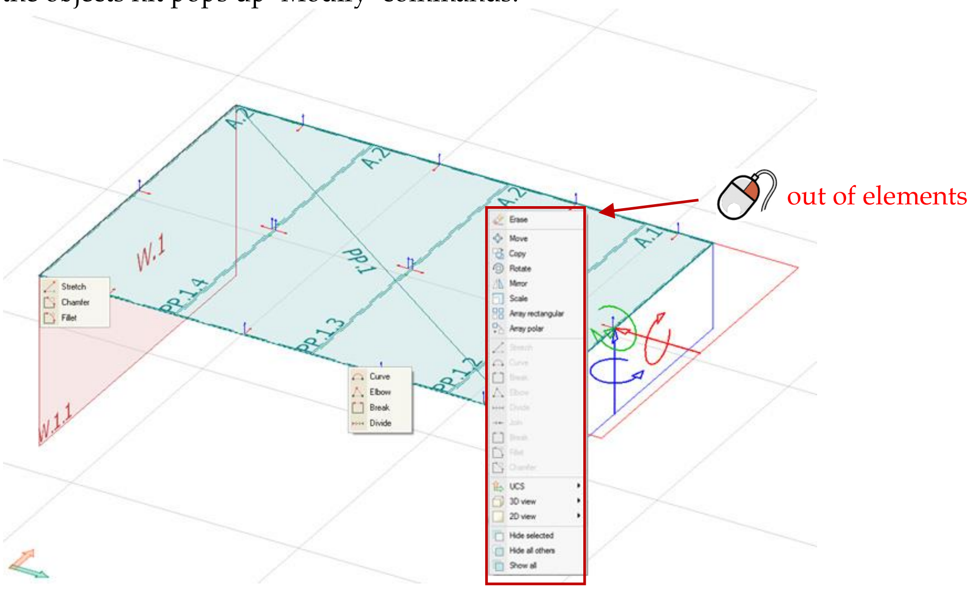
-  +holding  at any point in a structural or drawing element starts to drag a copy of the object.



- none of the objects starts 'Box selection'

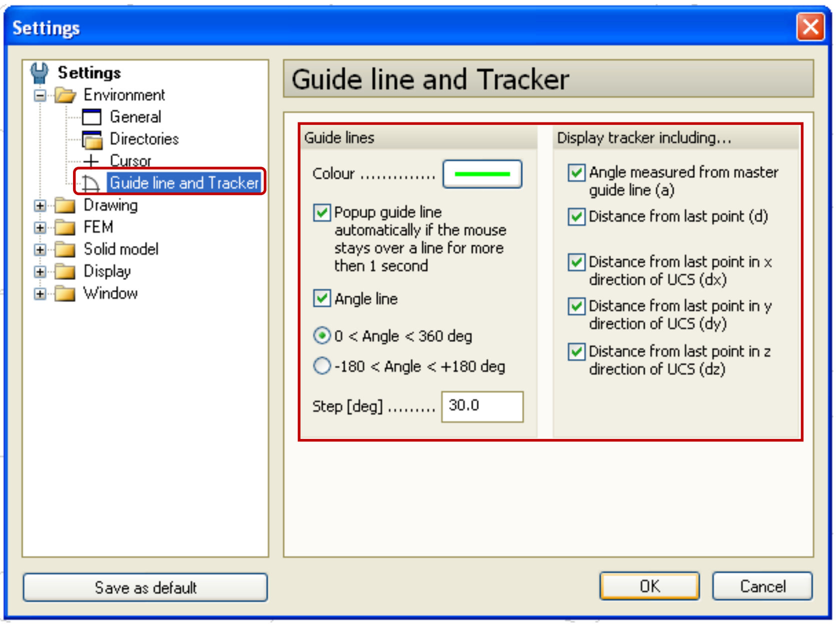


- none of the objects hit pops up 'Modify' commands.

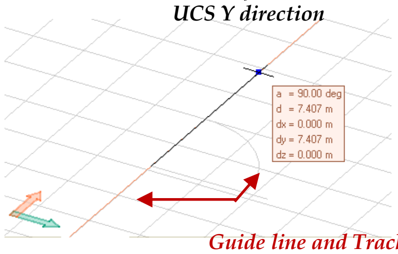


Guide lines and Tracker

The Guide lines and Tracker appears when any commands started and assist the user in modeling and modifying elements in a much more effective way. Guide lines are displayed temporarily during modification or drawing to assist user to find desired point in a fast and easy way. There are two kinds of Guide lines **Straight** and **Circular**. The Color and Angle step can be set in the *Settings/Environment/Guide line and Tracker*. All features of Guide lines can be turned OFF/ON in its Setting dialog. There are three colors of Guide lines in FEM-Desig. Dark green line will be drawn if the Guide line is parallel with the X axis of UCS, orange line will be drawn if it is parallel with the Y axis of UCS and light green (as default color) line will be drawn for any other guide line.

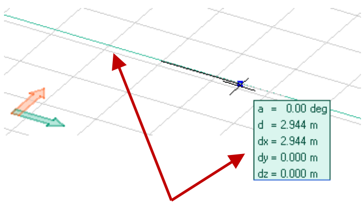


1. Guide line parallel with UCS Y direction



Guide line and Tracker are orange

2. Guide line parallel with UCS



Guide line and Tracker are dark green

	Stand	**Click RB**
Along line or arc	Parallel guide line	Parallel guide line
Line's middle or end point	Perpendicular guide line	Perpendicular guide line
Guide line	-	Remove that guide line

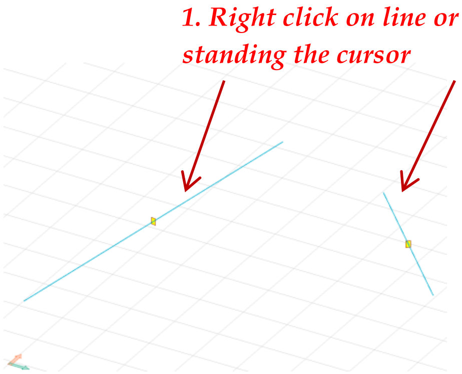
Guide lines can be defined on the following ways:

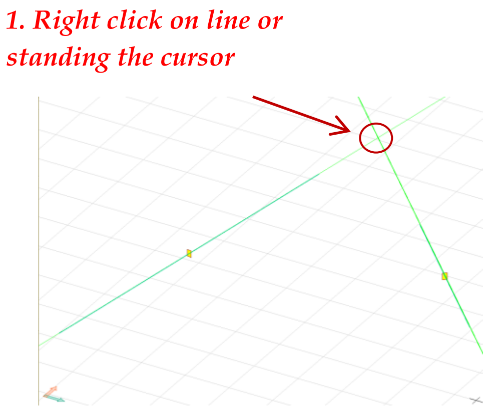
- It appears automatically after 1 second, if mouse stands over a line

This function can be turned off in its Settings by unchecking „Pop up guide line automatically if the mouse stays over a line for more than 1 second"

-  on a line (straight, arc, circle or any object having these kind of geometry)

Virtual intersections can be found easily with Guide lines

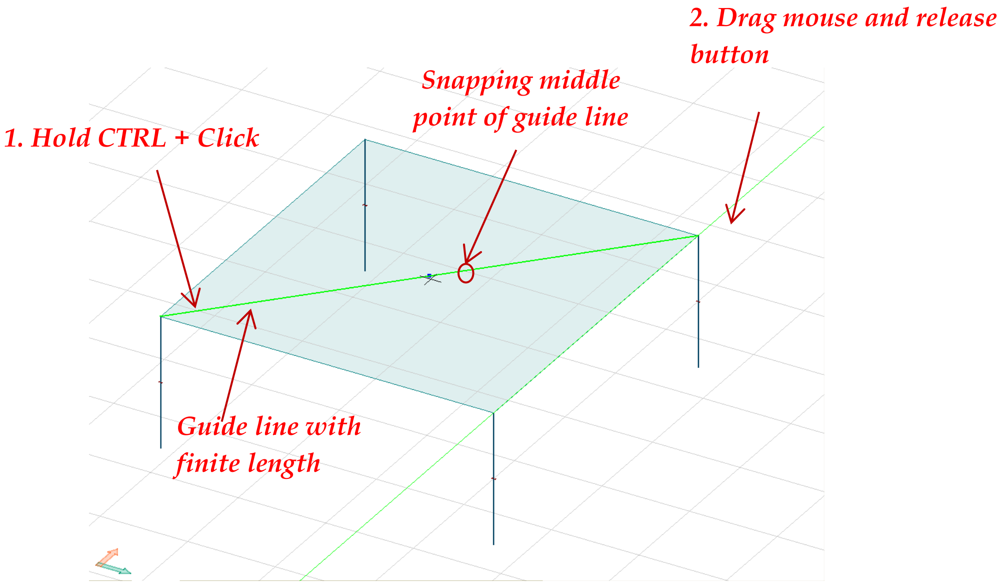




-  and drag mouse to define arbitrary guide line. If 

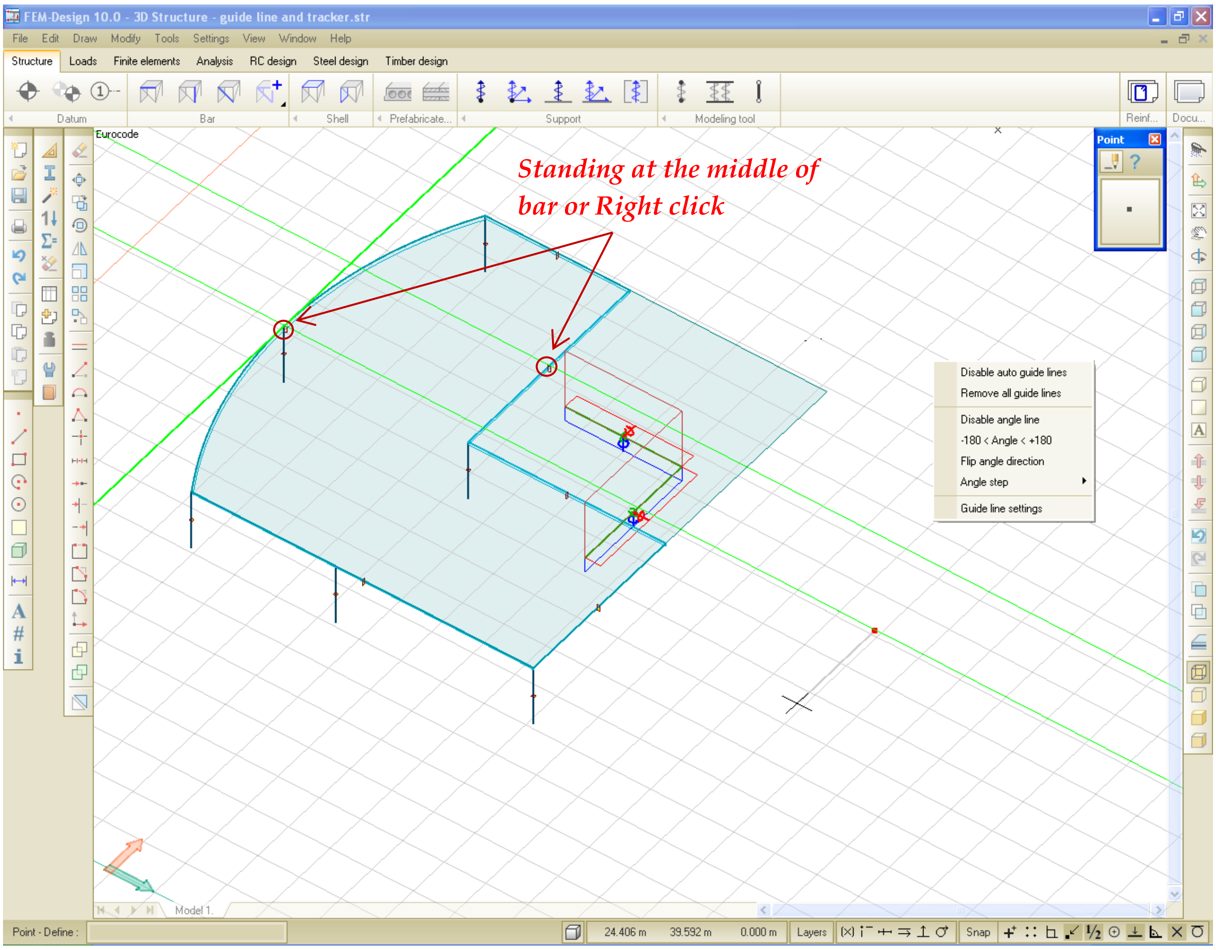
is pressed, the line is created with finite length (from start point to end point), otherwise it will have infinite length.

  Middle point of a rectangle can be found easily with Guide lines

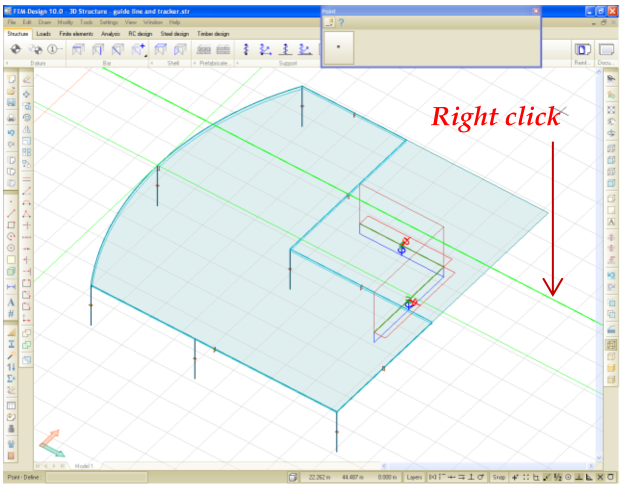


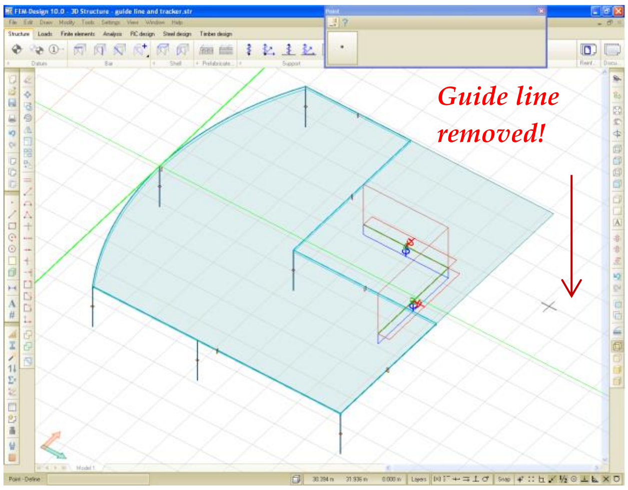
Guide lines are drawn in **3D**, but Master Line (thick line) from which angle is measured, can only be placed in UCS or in a plane parallel to UCS.

- **Perpendicular** guide line appears if mouse cursor stands over the end or middle point of a line, otherwise **Parallel** guide line appears.

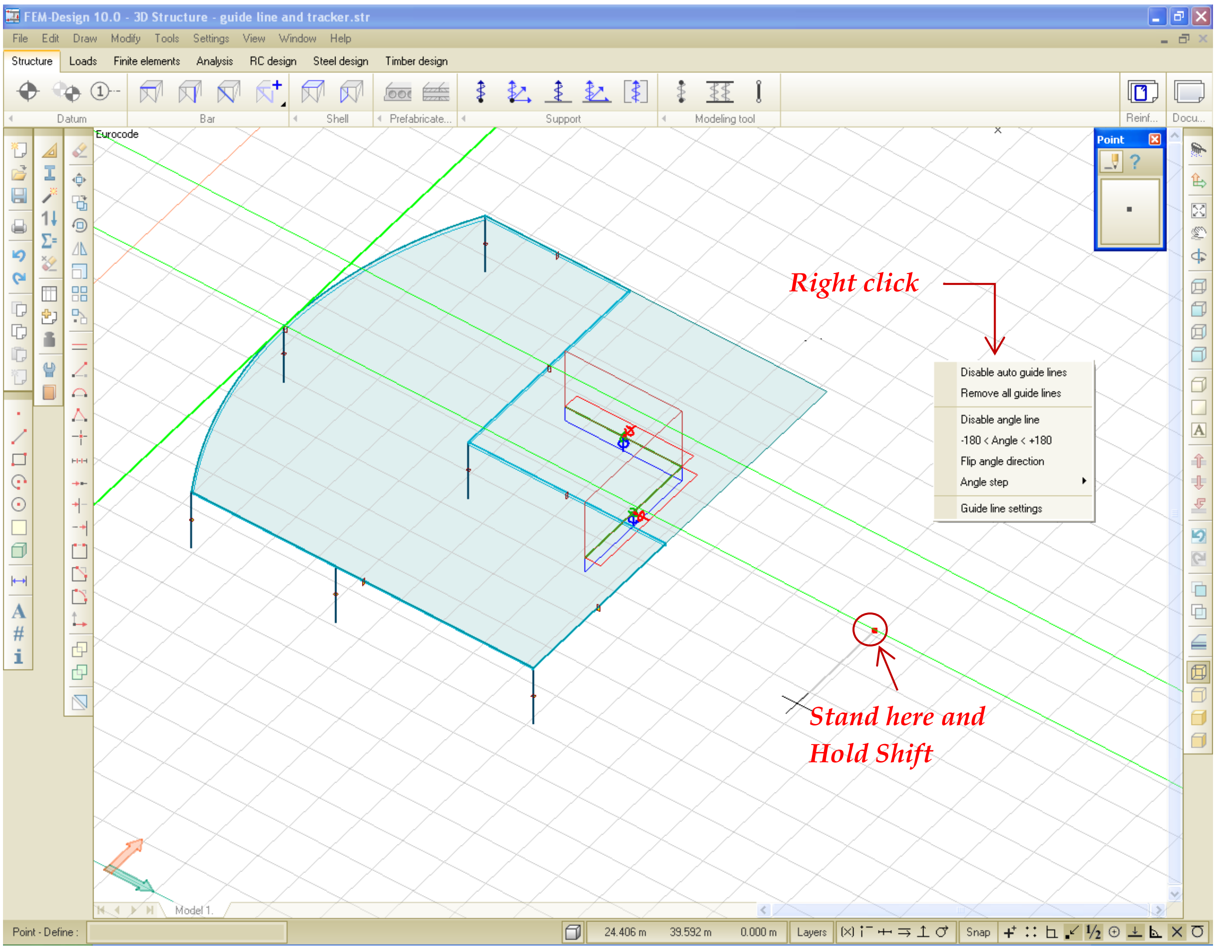


-  over guide line will remove that guide line.

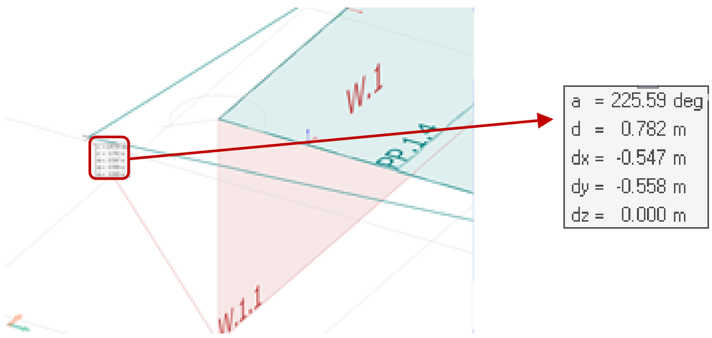




- By  over snap-free places, guide line settings will pop up.
- Snapping on a Guide line +  will stick the pointer to that guide line.



The **Tracker tooltip** shows **Angle, Distance, dx, dy** and **dz** from the last point**.** It pop up next to the cursor while drawing and the tracker will be the same color as the Guide line.



Modify region

Modify region command includes editing functions of drawing regions and region-type structural objects (plate, wall, surface load, surface support, surface reinforcement etc.).

- **Remove hole**
- **Split region**

  It divides regions into several pieces by cutting edge that can be defined manually as custom polyline or by selecting predefined lines or by selecting a region intersects the original one.

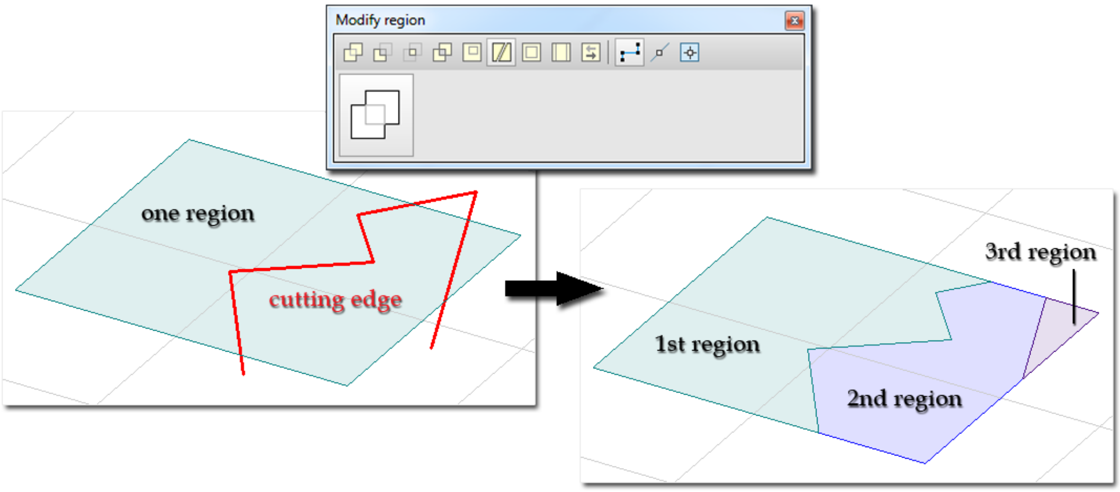


Figure: Splitting region

- **Perpendicular region offset**

  It increases or decreases region area parallel with region edges. Offset can be done for all region edges, for all external contour, or for the hole edges.

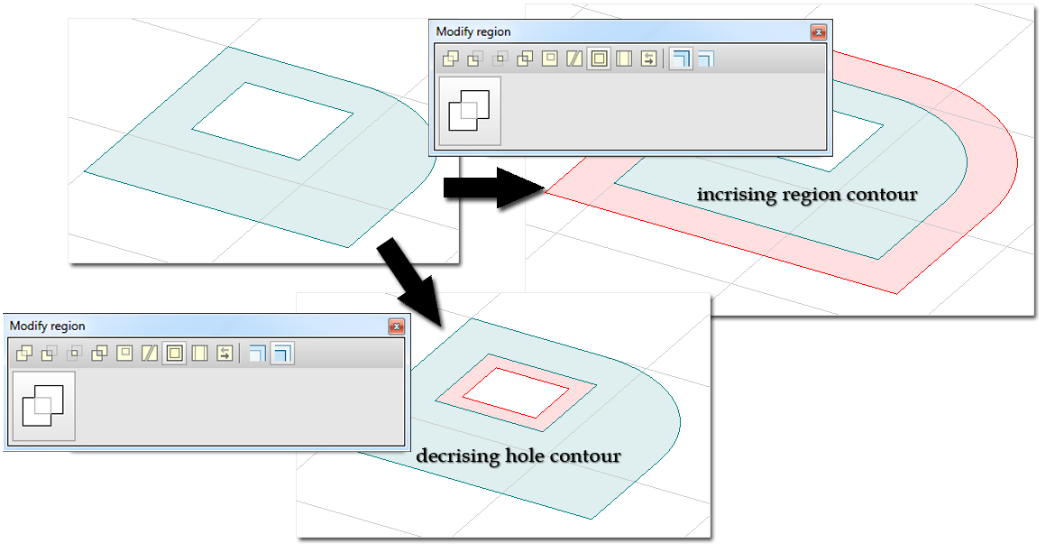


Figure: Perpendicular region offset

- **Direction region offset**

  It increases or decreases region area along a defined direction. Offset can be done for all region edges, for all external contour, or for the hole edges.

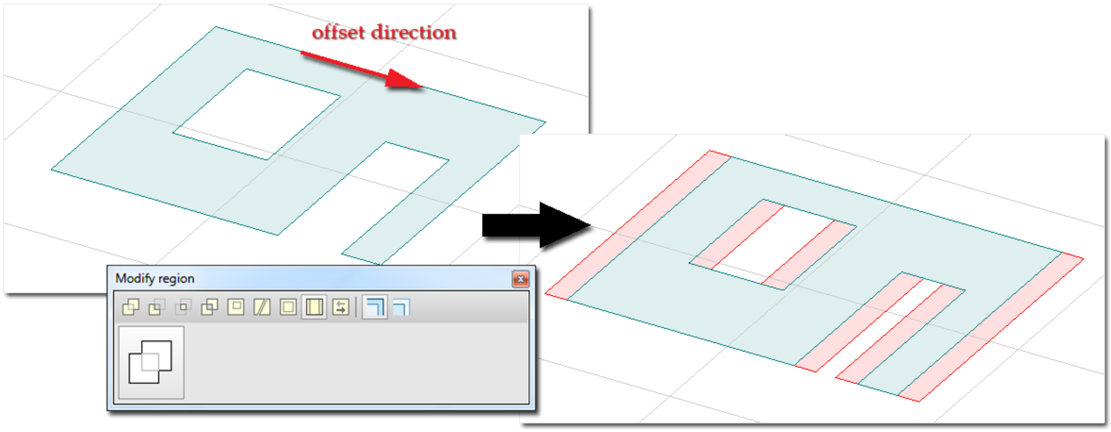


Figure: Direction region offset

- **Swap top and bottom faces**

  It exchanges the upper and lower surface of plates and the front and back sides of walls. It also mirrors objects with variable thicknesses.

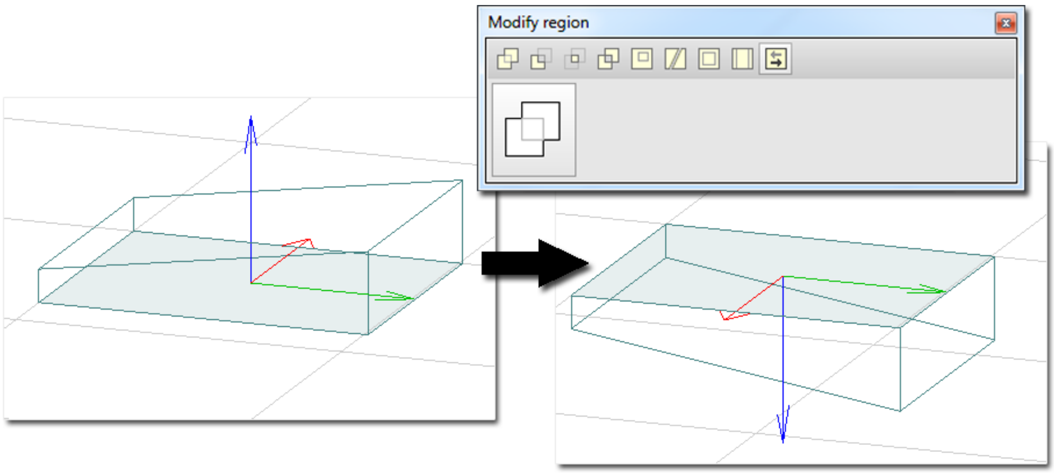


Figure: Swap plate region surface (variable thickness)

- **Logical operation: Union**

  It adds user-defined area to a selected region or unifies two predefined regions. The result region always inherits the properties of the goal region selected first.

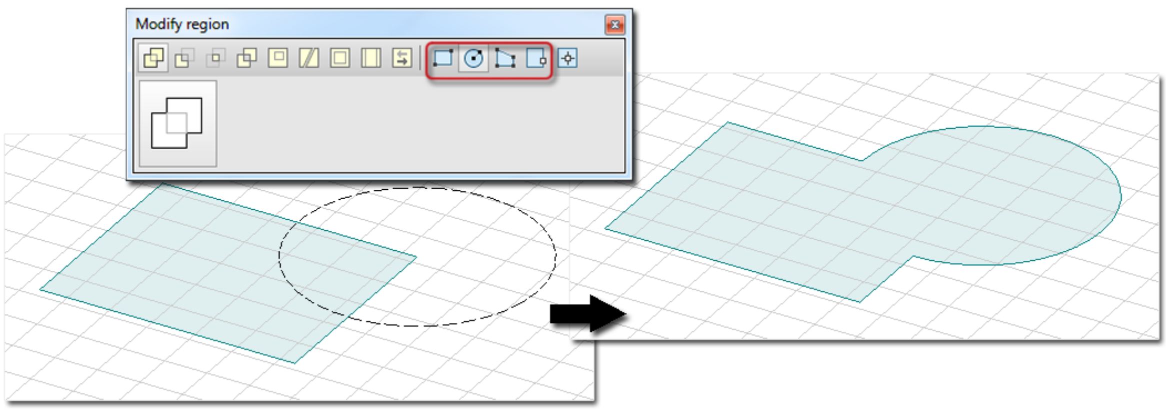


Figure: Custom drawn part added to a (plate) region

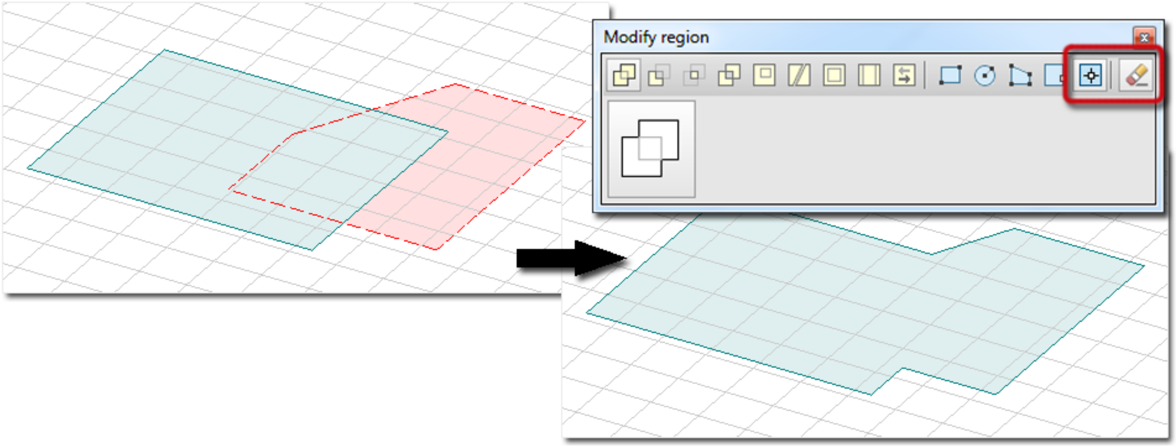


Figure: Unifying two regions into one

- **Logical operation: Substract**

  It cuts out a part of a region, which will be determined by another region intersects the original one. The result region always inherits the properties of the goal region selected first.

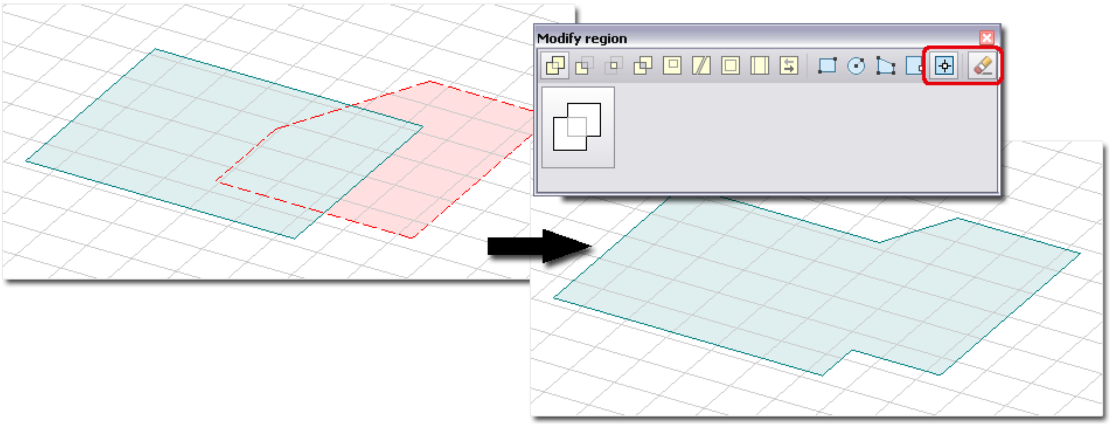


Figure: Substract of two intersected regions

- **Logical operation: Intersection**

  It creates a new region from the intersection of two regions. The result region always inherits the properties of the goal region selected first.

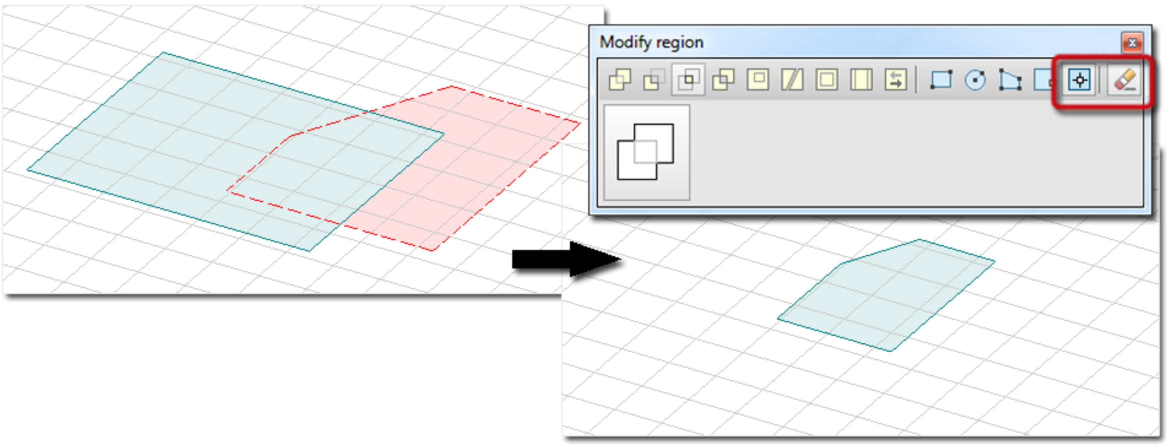


Figure: Intersection of two regions

- **Logical operation: Excluded "OR" (XOR)**

  It performs an algebral XOR operation on an existing region and a shape drawn to intersect it or two regions having common part. The common part will be deleted. The result region always inherits the properties of the goal region selected first.

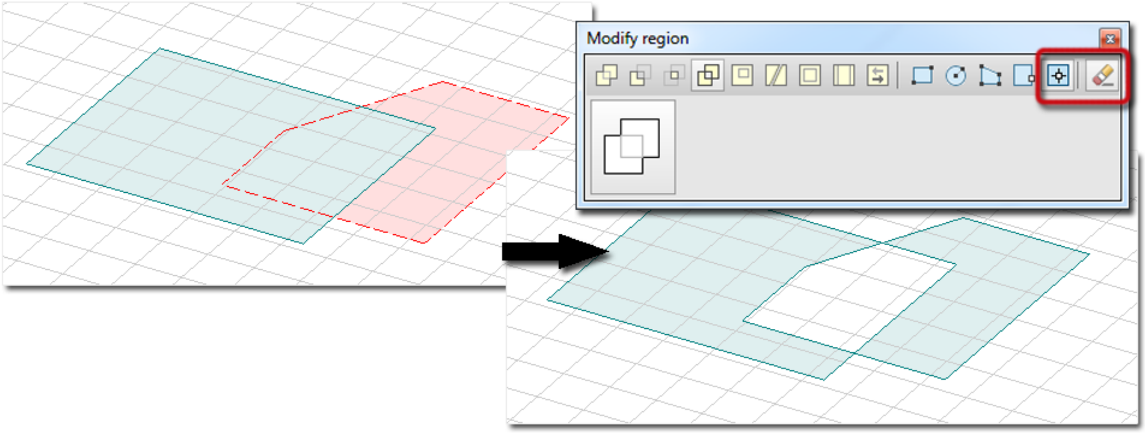


Figure: Deleted intersection part of two regions

Modify solid

Solid operations command includes editing functions of drawing **solids**.

- **Remove hole**
- **Split solid**

  It divides solids into several pieces by cutting plane that can be defined manually as custom plane or by selecting existing regions or by selecting a solid intersects the original one.

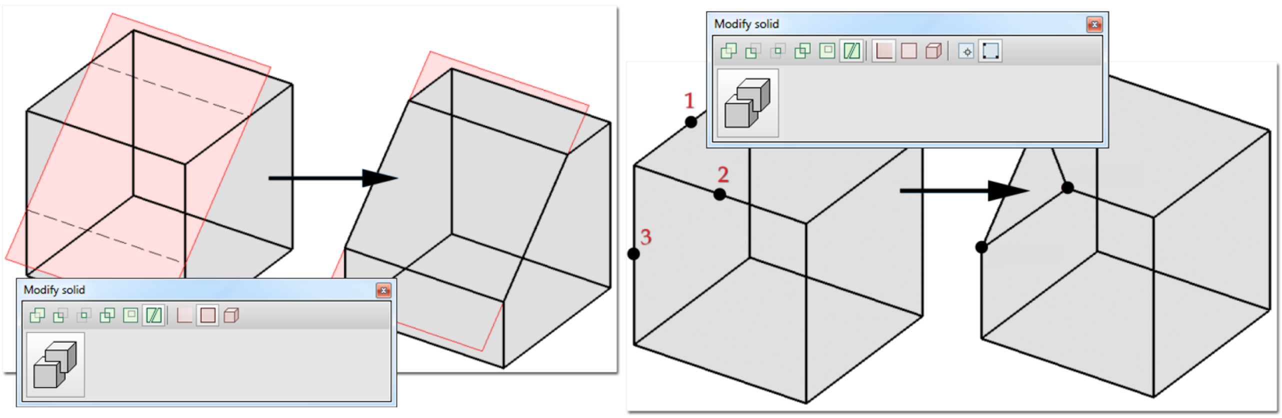


Figure: Splitting solid with predefined region and custom defined plane

- **Logical operation: Union**

  It unifies one solid with another one(s). The result solid always inherits the properties of the goal solid selected first.

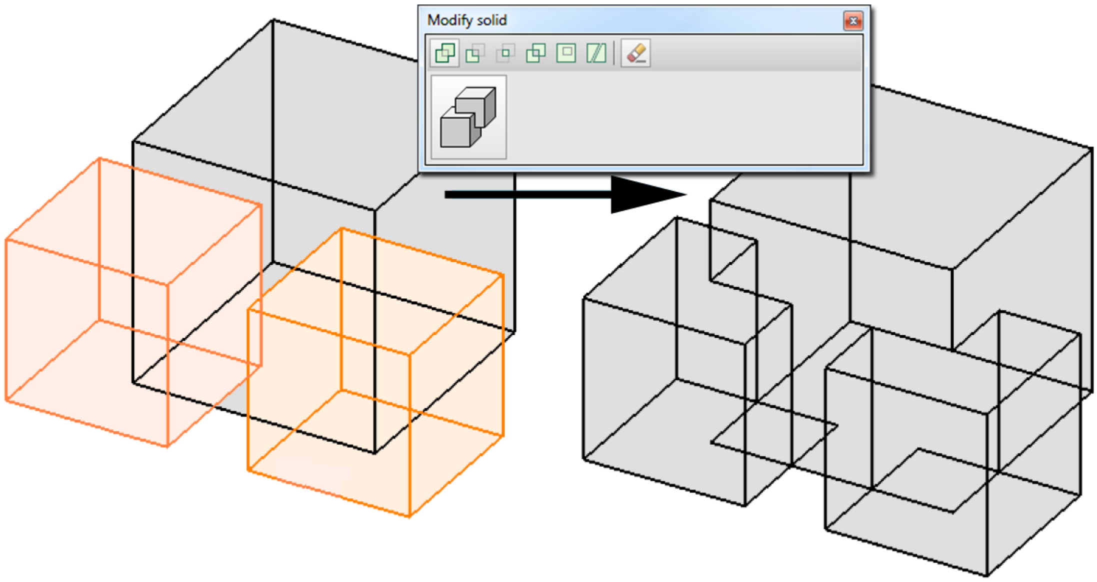


Figure: Unifying three solids into one

- **Logical operation: Substract**

  It cuts out a part of a solid, which will be determined by another solid intersects the original one. The result solid always inherits the properties of the goal solid selected first.

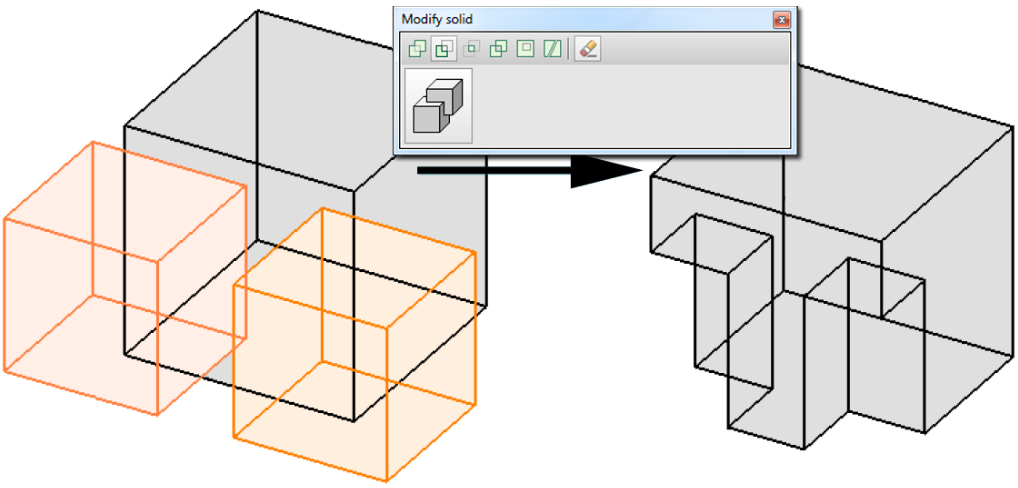


Figure: Substract of solids

- **Logical operation: Intersection**

  It creates a new solid from the common part of two solids. The result solid always inherits the properties of the goal solid selected first.

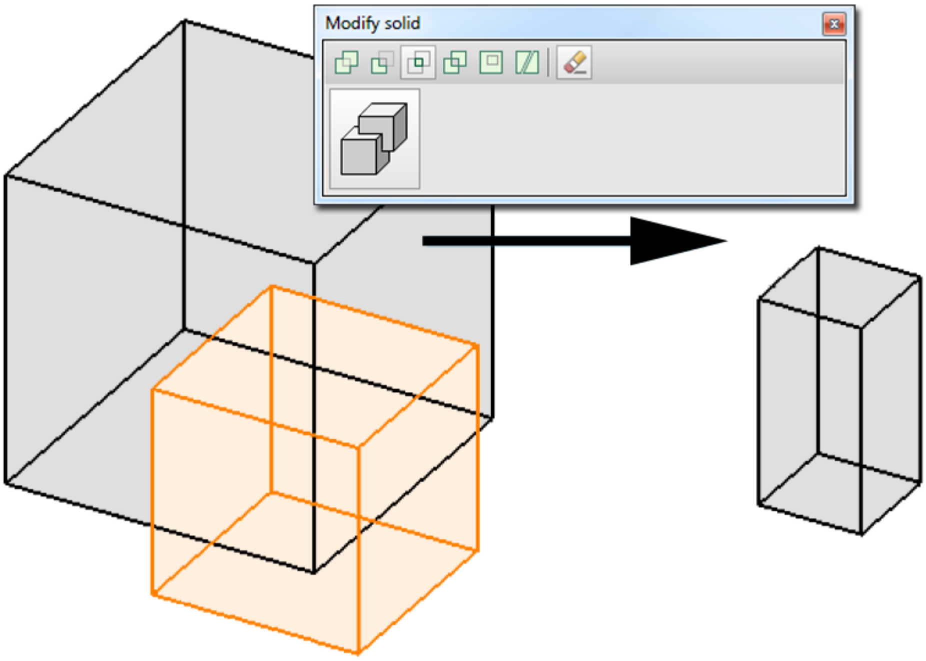


Figure: Intersection of two solids

- **Logical operation: Excluded "OR" (XOR)**

  It performs an algebral XOR operation on two selected solids. The common part will be deleted. The result solid always inherits the properties of the goal solid selected first.

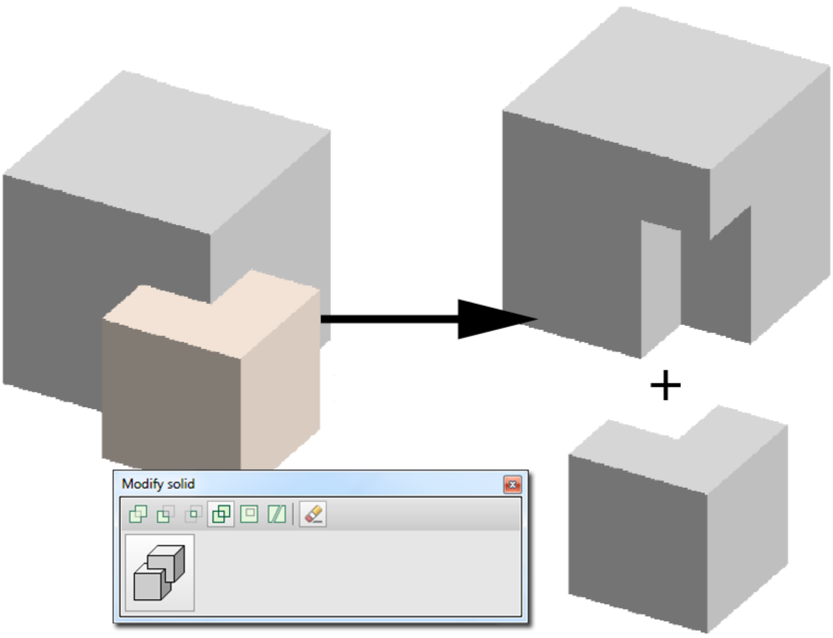


Figure: Deleted intersection part of two solids

Change appearance
The *Change appearance* command changes several properties of objects according to user modifications.

- **Change properties**

  The display properties of objects can be modified in one settings dialog:

- layer of drawing objects,
- line type of lines, edges and axes,
- text style of texts, tables, dimensions etc.
- color of selected objects (different color can be set from the color of the layer assigned to the object),
- pen width of objects.

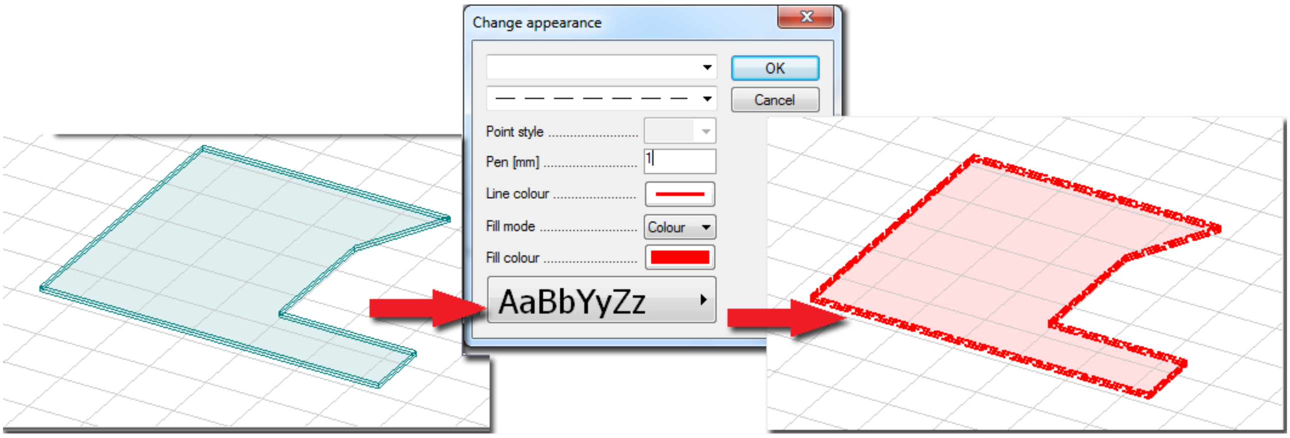


Figure: Appearance modification

Accessories

Correct model tool

Correct model tool is a revolutionary function to fix the models having geometrical errors, in semi-automatic way, with supervision of the user. This tool helps to fix the following types of errors in the model:

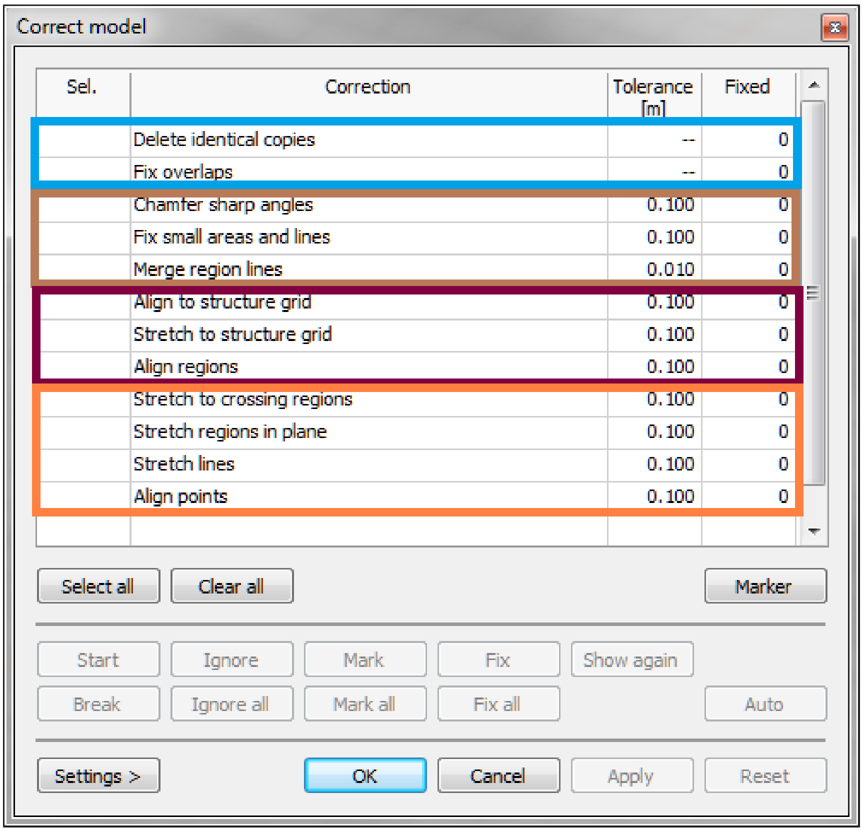
- object multiplication and overlap
- geometrically incorrect objects (very small region parts, very small angles, divided region edges, etc.)
- incorrectly positioned objects

Correct model tool is a **semi-automatic system** which necessarily **needs user-interaction**. It draws attention to the possible errors of the model and in most cases also offers solution(s), but you have to decide whether it is really an error or not and if it is, how to fix it.

Pick *Correct model* from *Tools/Correct model* or from the toolbar ().

Select the elements to be checked.

Then in the pop-up dialog, select the task(s) to be run.



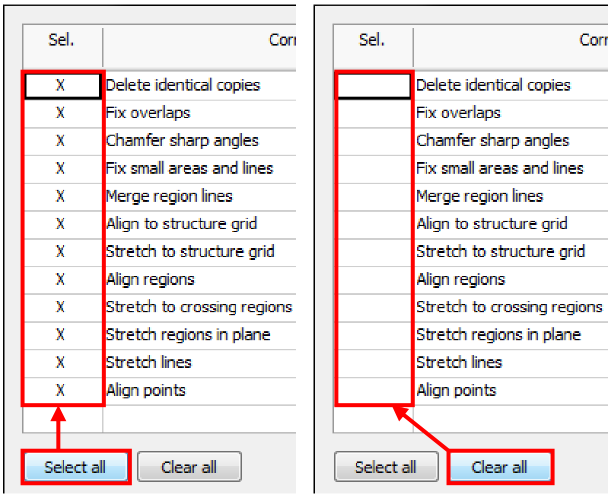
Fix object multiplication and overlap

Fix geometrically incorrect objects

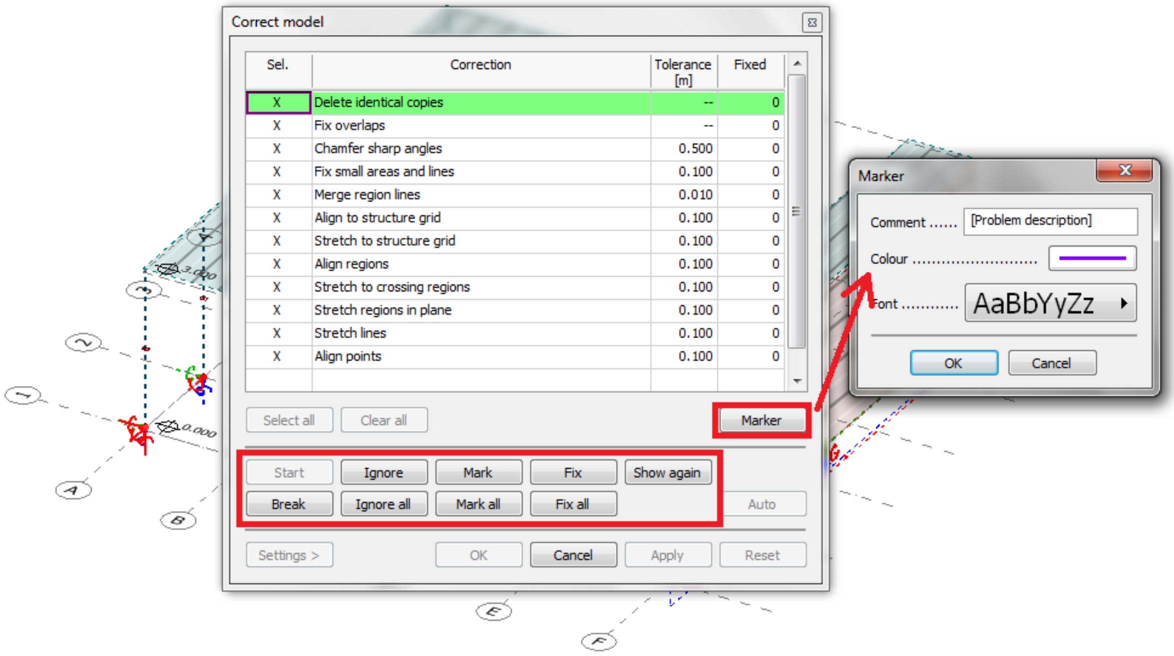
Fit objects to storeys, axes and reference planes

Fit objects to each other

All correction tasks can be selected or cleared in one click. Use the *Select all* and *Clear all* buttons for it.



Press *Start* to start the correcting process. It runs row by row and the current step turns green. The view is focused on the incorrect object (part).



The table below contains the meaning of the flashing red object depending on the current correction task:

Correction task (type)	What is flashing?
Delete identical copies	object to be deleted
Fix overlap	current object and its modified version are both flashing
Chamfer sharp angles	the part of the region to be removed

Fix small areas and lines	the part of the line/region to be removed
Merge region lines	the region line to be merged
Fixing geometrically incorrect objects	the object to be fixed
Fit objects to storeys, axes and reference planes	the suggested new position of the incorrectly placed object
Fit objects to each other	

You can choose from the following choices:

- *Ignore*: the object is not modified
- *Mark:* the object is marked according to the colour and text set in the Marker dialog
- *Fix:* the object is deleted/modified according to the program's suggestion. The number in the table in *Fixed* cell of the current correction task's row is increased by one. The fixed object turns into green

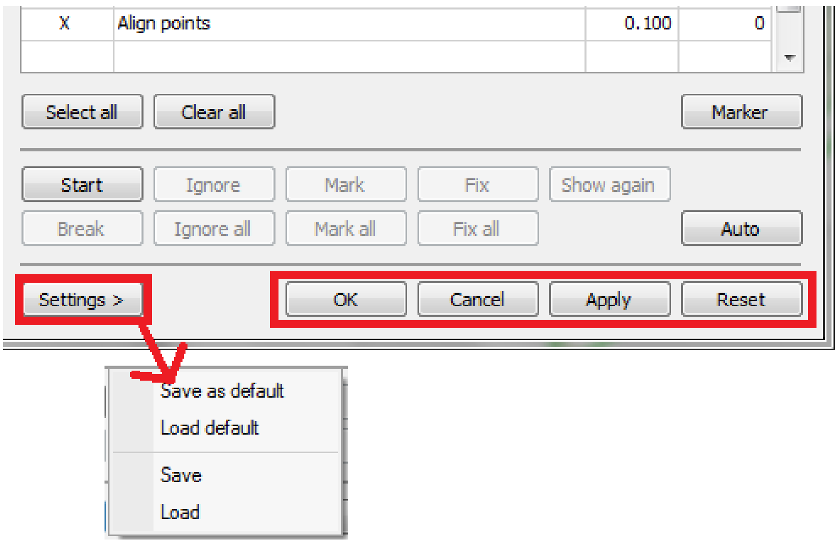
Ignore all, Mark all, Fix all acts on all further incorrect objects found by the **current correction task** (highlighted by green in the table).

After one of the abovementioned button is pressed, the next incorrect object – if exists – gets into focus and highlighted

Show again focuses on the current objects. It can be useful at a large model.

Break cancels the correction process but the previously performed modifications are kept.

Once the correction process is finished (or cancelled) the modifications can be applied on the model by pressing either *Apply* or *OK.* In case of OK the dialog closes. All modifications can be discarded by pressing *Reset* if they were not yet applied on the model.



Settings commands let the user save/load the selected correction tasks and their tolerances.

Using *Auto* button and applying its modification on the model is not recommended, because in practice in most cases there is no exact solution:

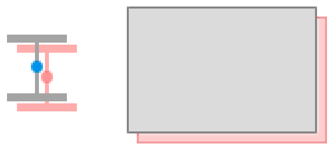
- fixing an error may generate or eliminate other problems
- fixing the same error in different ways may generate/eliminate different problems
- consequently, the correcting process can be iterative.

However, *Auto* is useful to get an estimation of errors and type of them in the model.

See detailed description of each correcction task as follows.

Delete identical copies

If more identical element exist in the same position, it deletes all multiplication, and counts as one correction in the *Fixed* column on the *Correct model* dialog.

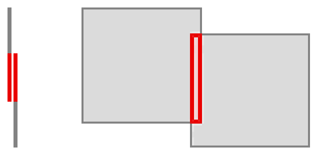


This is not working for intermediate sections, post-tensioned cables and corbels.

Fix overlaps

It fixes the overlapping regions and lines. The existing objects can be shrunk or erased, but function will not generate new objects (e.g. a region can not fall apart)..

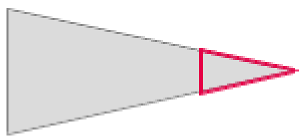
Overlapping is allowed for the Loads.



This is not working for isolated and wall foundations'region and corbels.

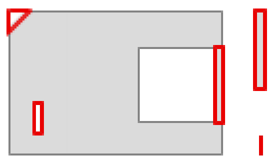
Sharp angles

Sharp corners under 10° will be fixed according to the given tolerance. The highlighted part of object will be removed. If the element has smaller dimension than the given tolerance, the whole element will be removed.

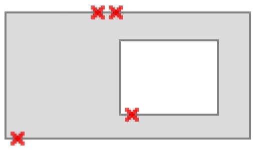


Fix small areas and lines

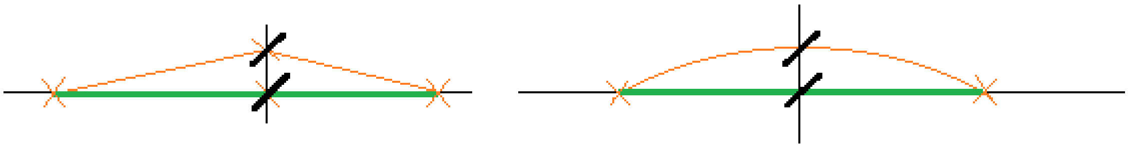
Objects within the tolerance range will be removed. It will fix small holes, long narrow areas etc.



Merge region lines

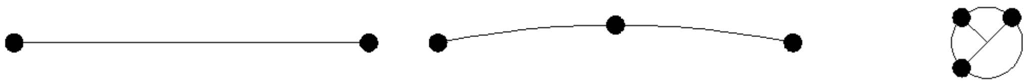


Merge region lines function merges two lines/curves of a region into one line/curve if they are in the range of tolerance. Too flat curves may be replaced by line. The figure below shows examples how the tolerance is measured.



Align to structural grid

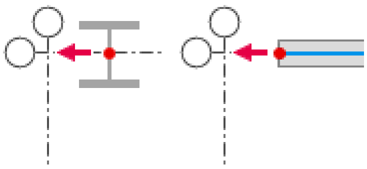
This function aligns objects to the structural grids (*axes*, *storeys* or *reference planes*). Alignment is done by orthogonal projection of object's checkpoints to the structural grid within the tolerance. The check-points are shown below for line, arc and circle:



⚠ Once an object is placed on a grid by *Correct model* tool, it will not be detached from it by any next correction step.

⚠ *Correct model* tool does not check structural grid for possible errors, like axes too close to each other or not perfectly parallel.

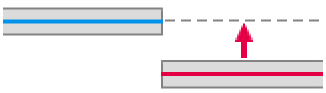
Stretch to structural grid



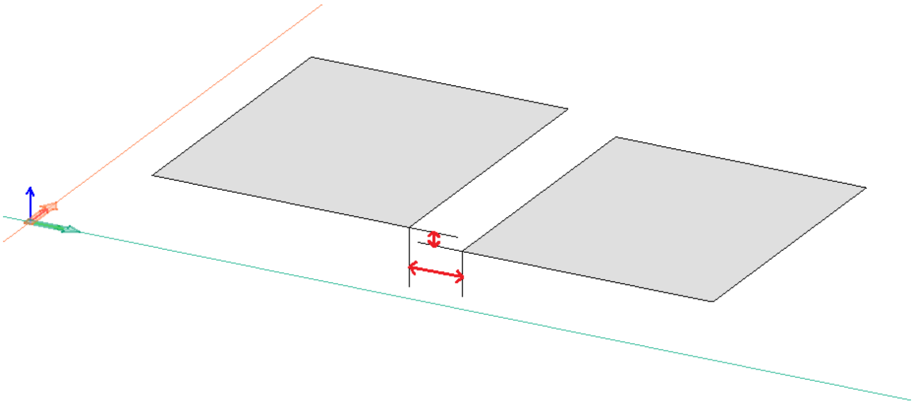
This function will stretch the regions and lines to the intersection of axis within given tolerance.

⚠ If the geometry of the object is incorrect (e.g. shell is not laying in plane) this function cannot be performed.

Align regions



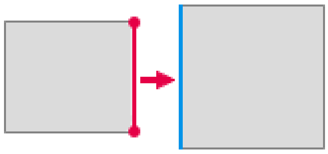
It aligns the regions (projection) to an other region's plane which is within the given tolerance both in direction perpendicular to and parallel with the region's plane.



Stretch to crossing regions

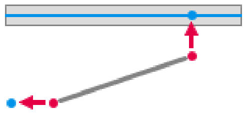
This function will stretch the objects to the crossing regions within given tolerance.

Stretch regions in plane



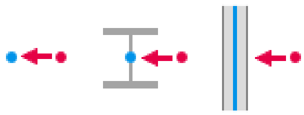
Regions laying in same plane will be stretched to each other within given tolerance.

Stretch regions in plane



Lines laying within given tolerance to other objects will be stretched.

Align points



Points (objects) will be aligned to closest object within tolerance.

Restrictions

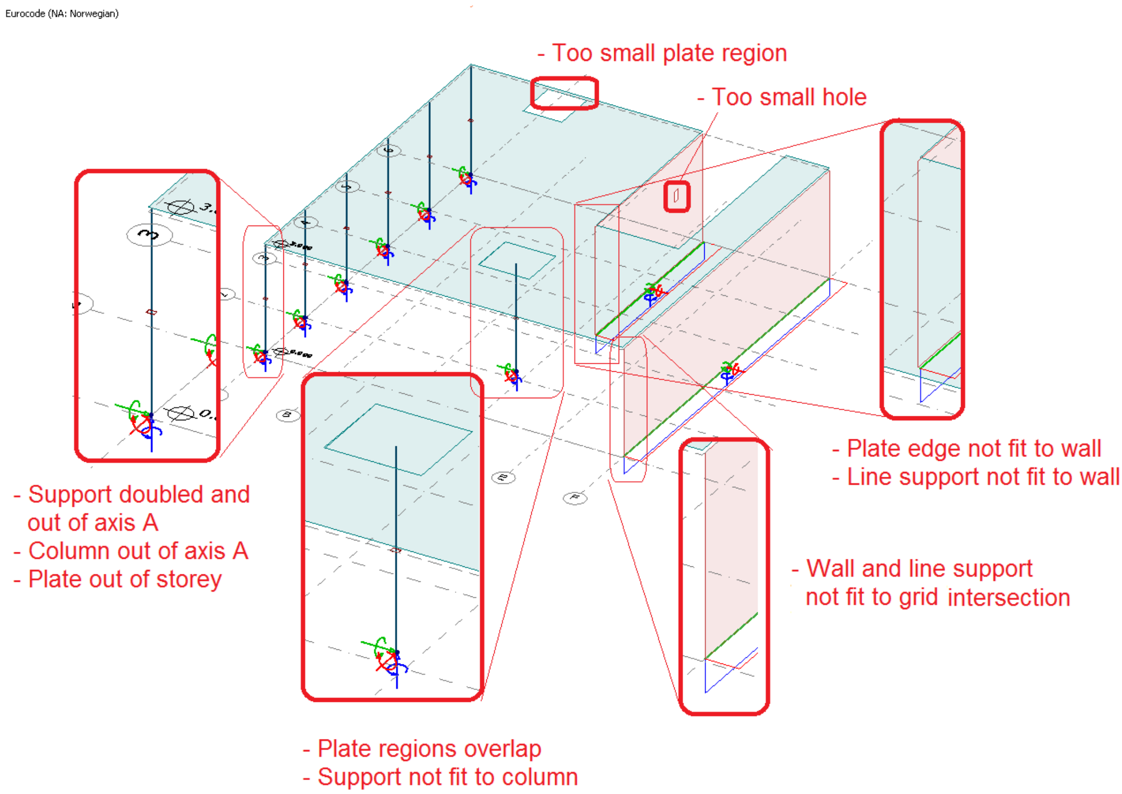
- *Correct model* cannot handle:
    - peak smoothing regions
    - connections (point, line, surface)
    - bar, shell components
    - building cover and
    - doesn't work in *Analysis* and *Design tabs*
- only the visible objects can be modified
- columns and walls need to be vertical and will stay vertical after the modifications as well
- isolated, shell foundation and foundation slab need to be and remain horizontal
- pile can be placed in any direction, but cannot be horizontal

- corbel is not handled by *Correct model* and cannot be used as an object to fit other objects to

Correct model works in multiple-window mode as well.

 *Example*

The following example shows the process of fixing geometrical errors of a small model with different types of errors:



Query

Starting  in Tools menu helps to get geometrical informations in the model. You can get the point coordinates, length between two point, angle, region and solid parameters.

Point coordinates

Choosing a point with shows the point coordinates in orthogonal and cylindrical *Global co-ordinate system*.

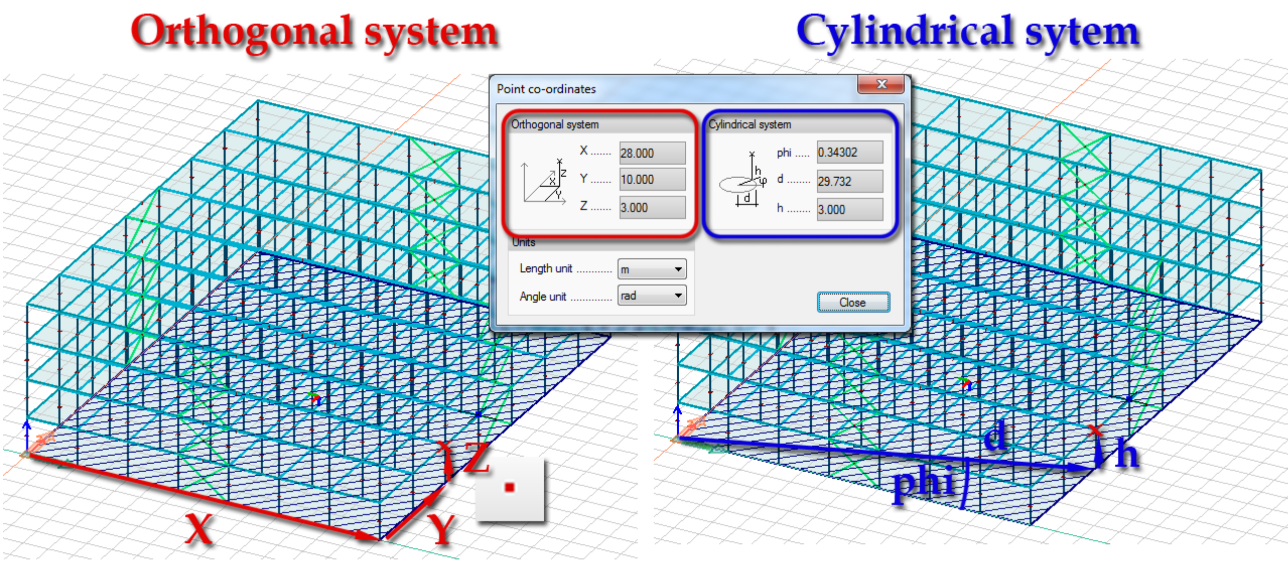
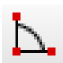


Figure: Point coordinate

Length

Choosing two points with  shows distance between the selected points and their projection to the co-ordinate axes.

Angle

Choosing two lines with  shows the angle between the lines.

Region

Choosing or drawing a region with  shows area, perimeter, center of gravity (co-ordinates).

Solid

Choosing a solid with  shows volume, perimeter, area and center of gravity (co-ordinates).

Block

Blocks can contain only drawing elements (*point*, *line*, *arc*, *circle*, *region*, *solid* and *text*) for editing these elements in one step.

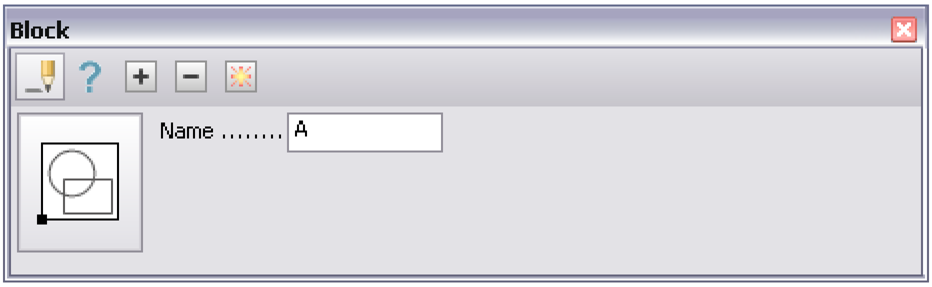


Figure: Block toolwindow

Similarly to filter, for definition, you have to type the name first, then select the desired objects. If required, members can be added with the “+”, and removed with the “-“ buttons, and exploded with the explosion symbol at the right hand side. With properties “?”, you can also change the name of the block.

For grouping structural elements or loads *User defined filter* function can be used.

Archive

This command deletes the result files for save free space in the hard drive. To see the result, you have to run calculation again.

External reference

With this command User can display a drawing of dwg or dxf format as an image in the background. The main advantage of external reference is that modifcations made in the source file will be executed in the FEM-Design file each time it is reopened, or Refresh function called.

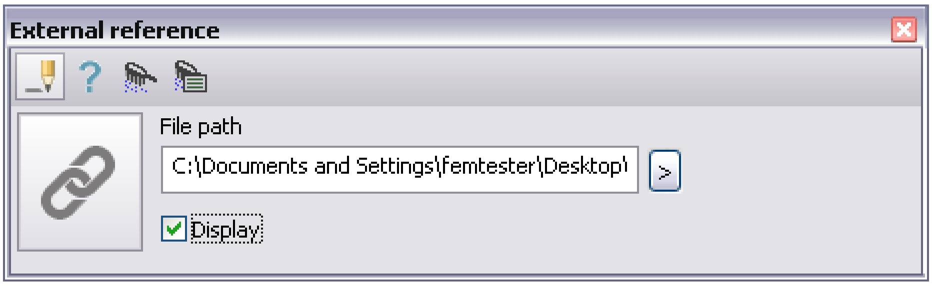


Figure: External reference toolwindow

After Clicking on Define you can specify the insertion coordinate system:

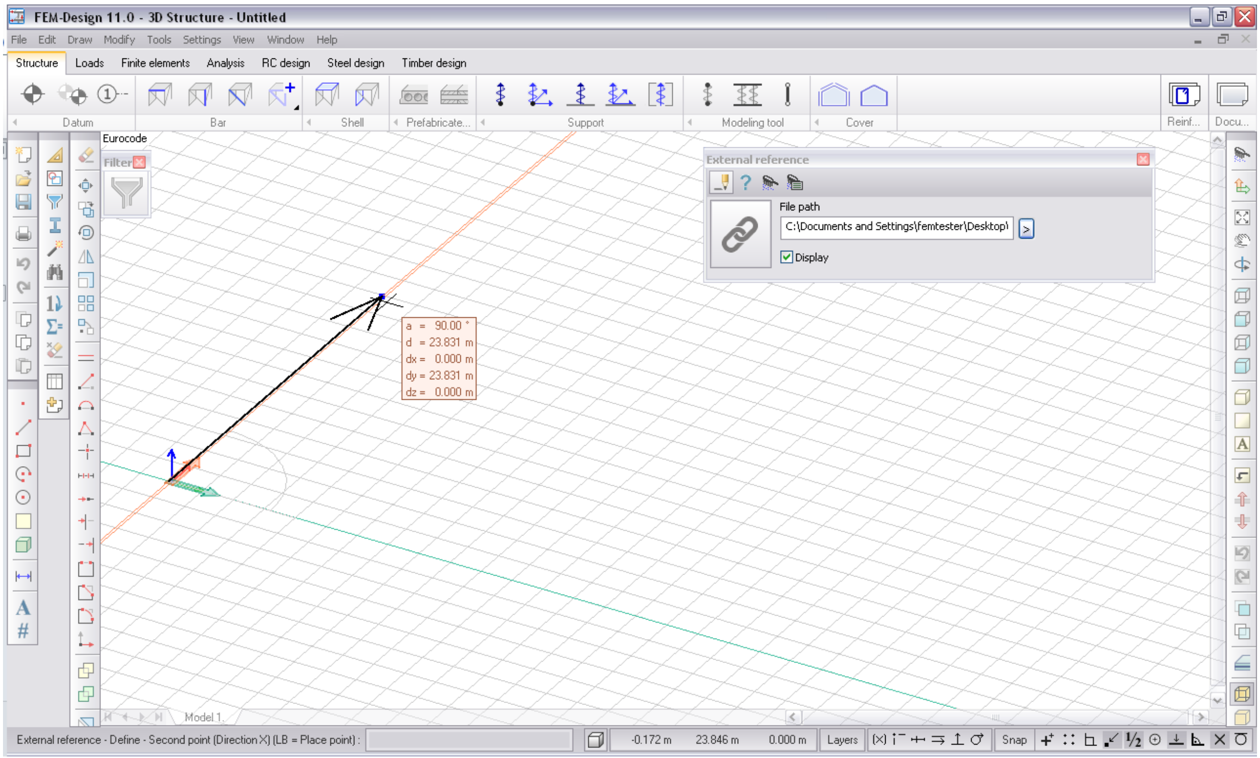


Figure: Insertion coordinate system of DWG/DXF drawing

With the third click, you specify the coordinate system’s second axis. The second axis can define reverse polarity compared to the polarity of the referenced image and thus you can get a reverse image (with wrongly directed texts, etc.).

If the origin in the dxf or dwg file is far away from the drawing, you may not find the image in FD's workspace. Click on „Zoom margin" in this case:

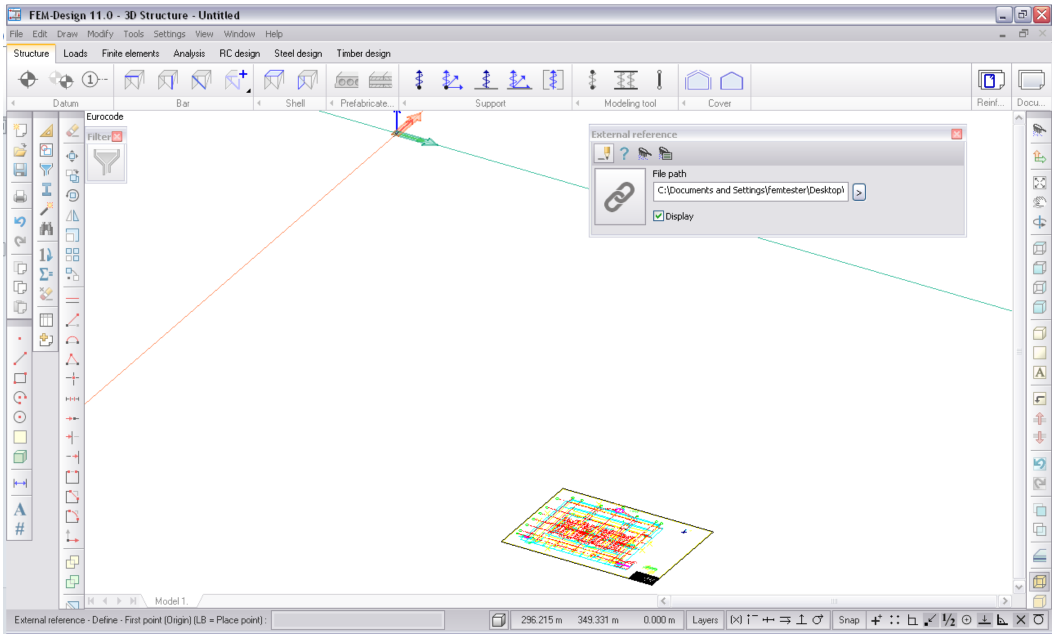


Figure: Inserted DWG drawing

In the import window, you can set the conversion scale, select the desired layers, with different colour representations:

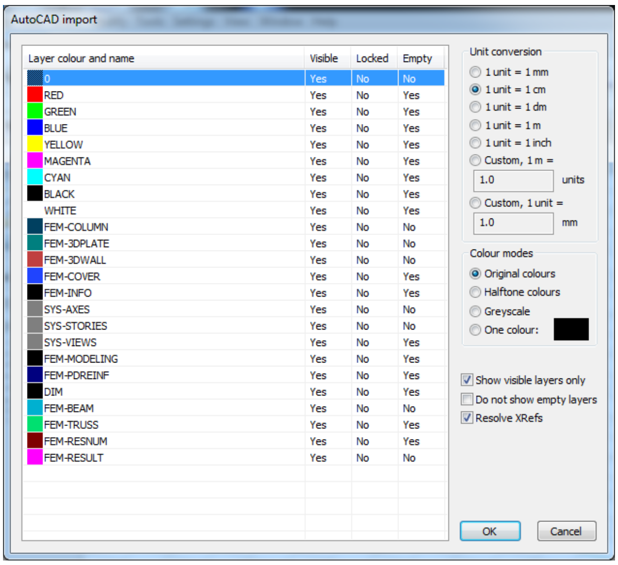


Figure: Import dialog

With the display option in the red box, you can turn the picture on and off.

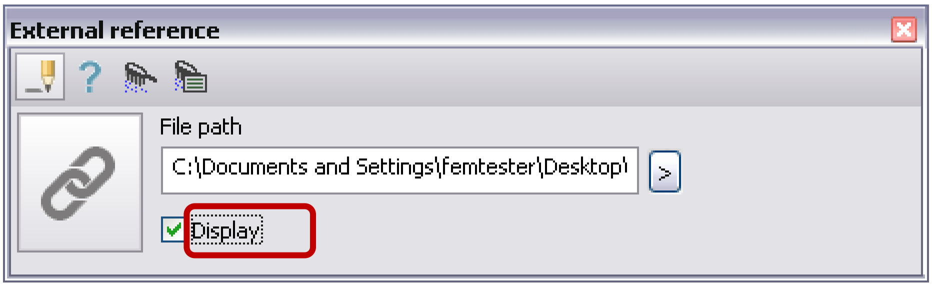
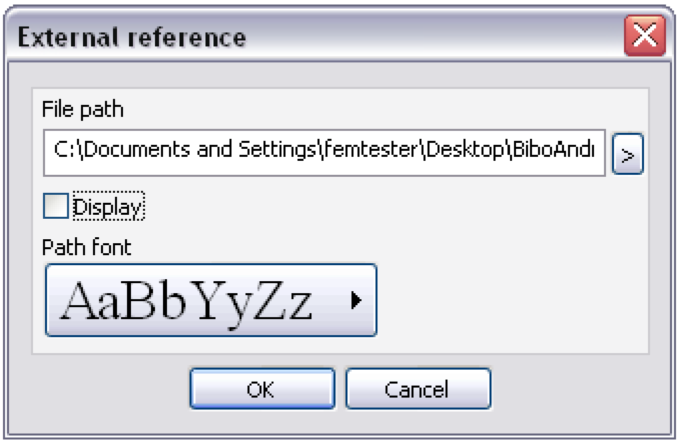


Figure: Turning on the Display option

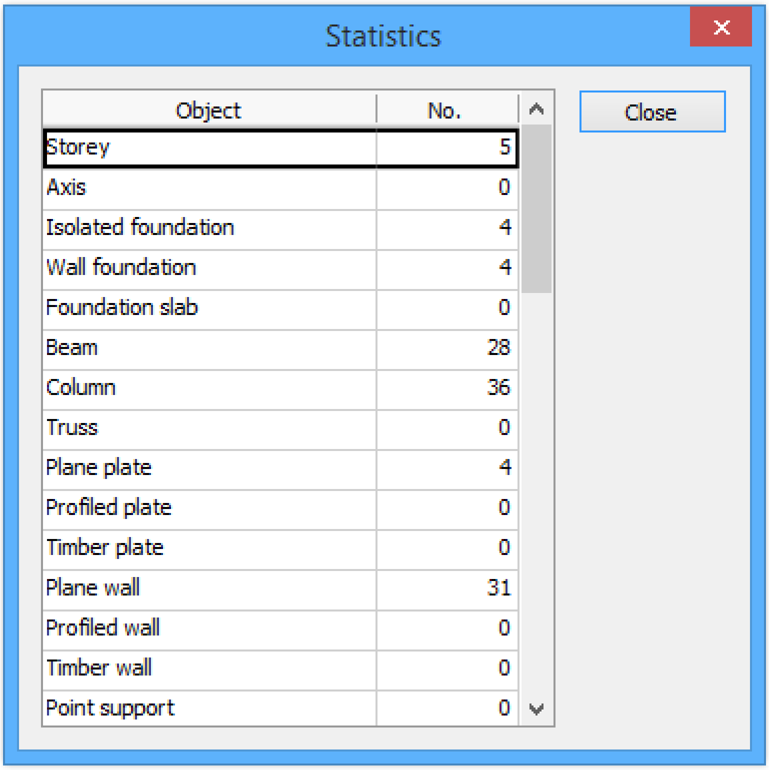
In "off" state, the file path is printed at the insertion point, and you can set its font parameters in the window below, which opens in default settings during definition , or when clicking on an existing external reference with  active:



If the source file has changed, you can refresh  it, or refresh it with different parameters .

Statistics

Starting  in *Tools* menu opens the Statistics dialog, where User can find information about the quantities of different objects in the model.



Object	No.
Storey	5
Axis	0
Isolated foundation	4
Wall foundation	4
Foundation slab	0
Beam	28
Column	36
Truss	0
Plane plate	4
Profiled plate	0
Timber plate	0
Plane wall	31
Profiled wall	0
Timber wall	0
Point support	0

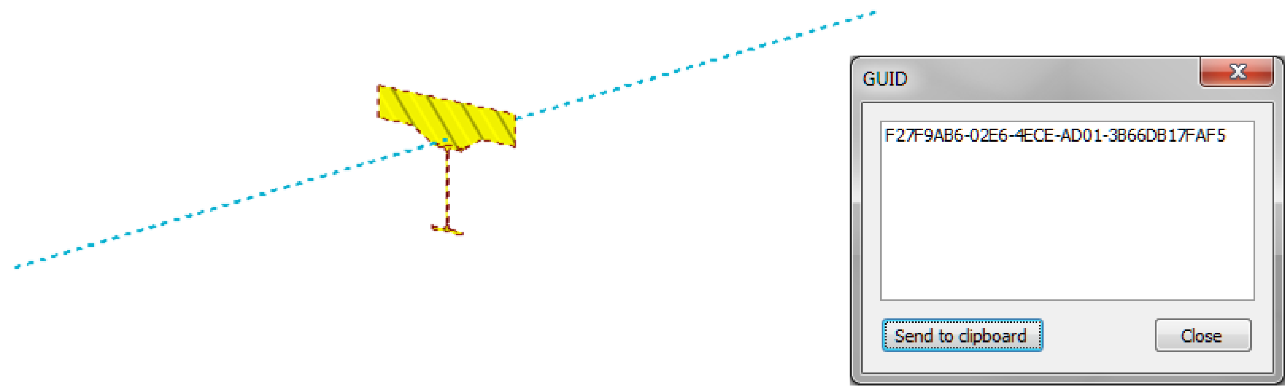
Get GUID

The *Get GUID* function enables the query of the GUID of elements. This can be useful for identification of structural objects imported/exported via Struxml.

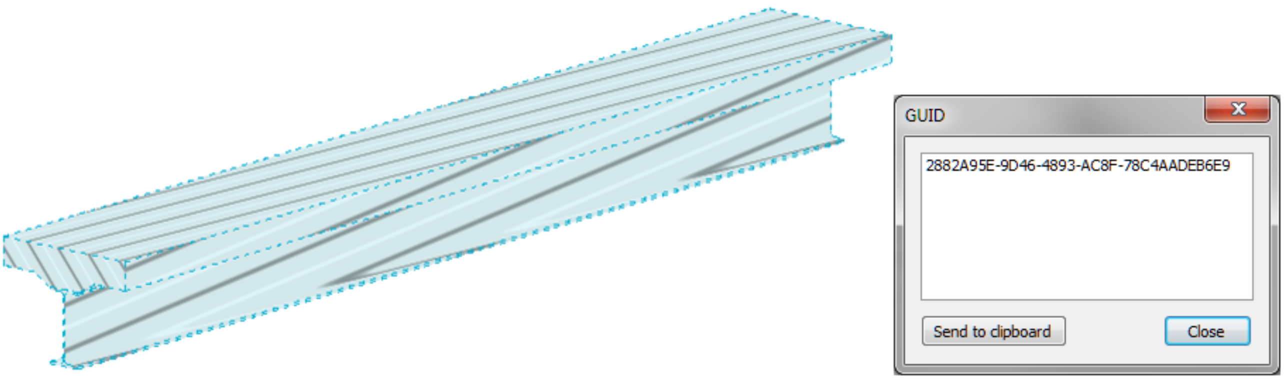
Picking {} in *Tools* menu launches *Get GUID*. You needs to select teűhe elements, of which GUID you wish to know.

The pop-up window shows the General Unique IDs which can be sent to the clipboard by clicking on "Send to clipboard" button. GUIDs for analytical and physical model of a bar are different.

Analytical view:



Physical view:

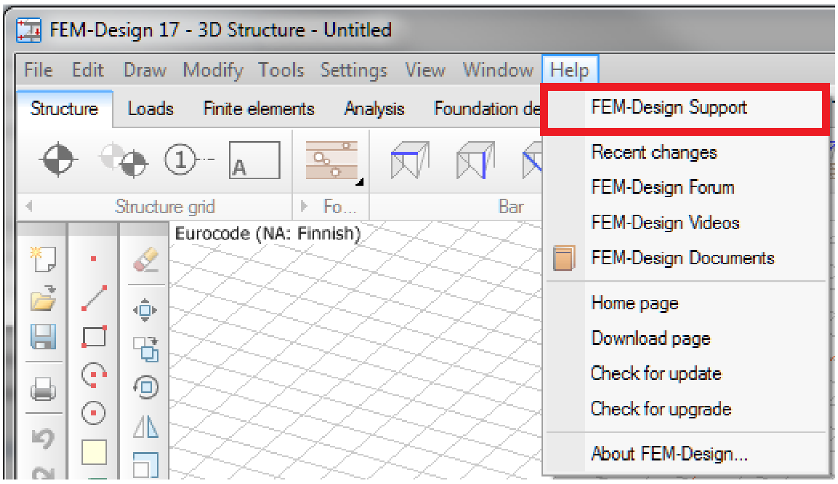


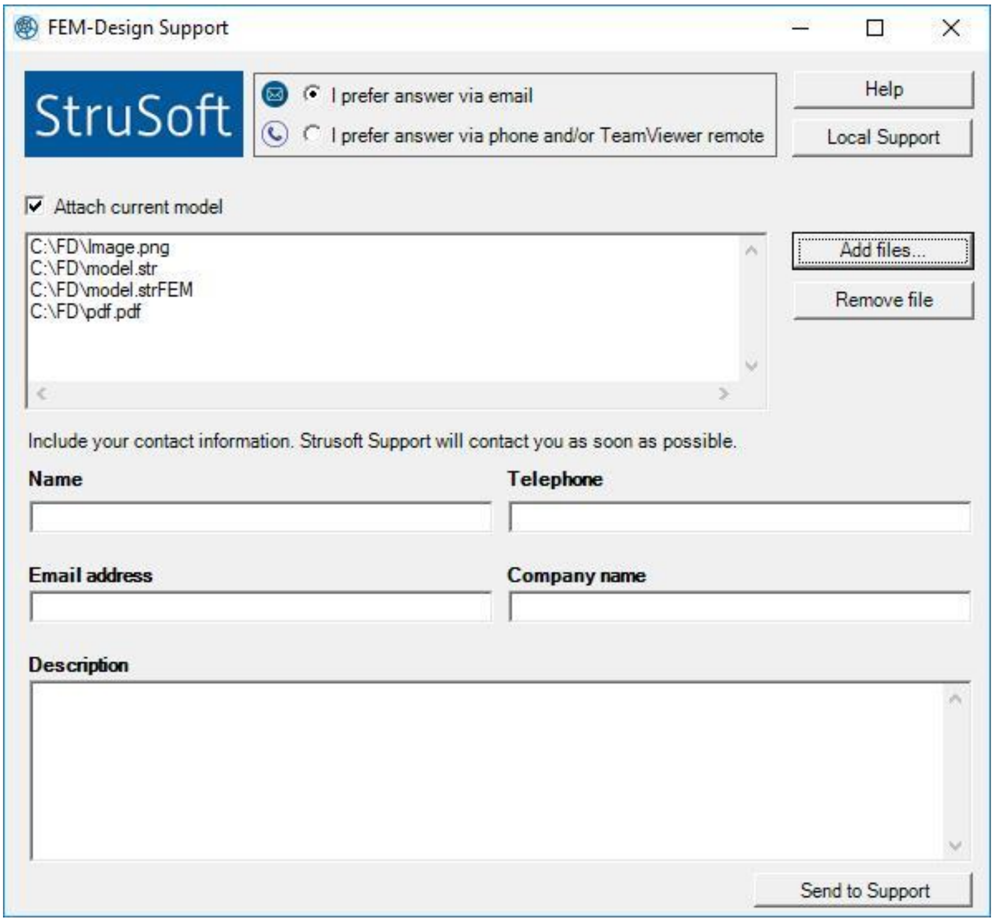
If the bar is saved in struxml format, these GUIDs can be found there as well.

```
   <bar type="beam" guid="2882a95e-9d46-4893-ac8f-78c4aadeb6e9" last_change='
     <bar_part guid="f27f9ab6-02e6-4ece-ad01-3b66db17faf5" last_change="2017-
'2a2219bf-21bf-49f6-ad1a-c884602e5eb1" complex_section="da02d4ce-6c0c-4467-9cl
       <curve type="line">
         <point x="8.71881755251908" y="33.04560763543" z="0"></point>
         <point x="8.71881755251908" y="46.1308053379253" z="0"></point>
```

Feedback mail

In *Help/FEM-Design Support* you are able to reach the support easily and get answers for specific model dependant questions.





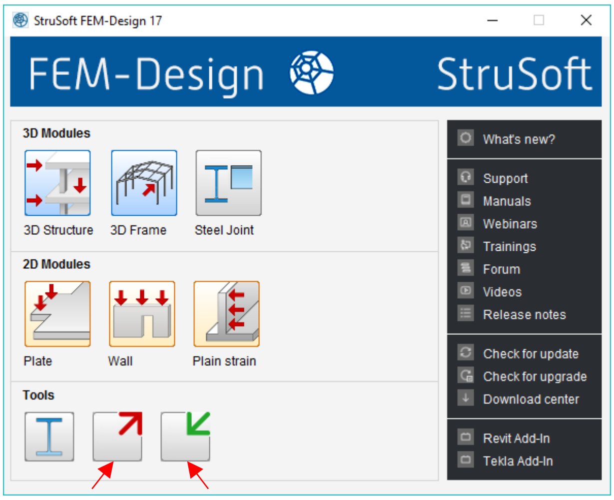
In the dialog you can decide how to receive answer, via email or phone and/or Teamviewer (remote desktop). The *Help* button links to a guide, how to write a request.

Local support button opens StruSoft's support website, where the local contacts are listed.
If the *Attach current model* is active, the last saved model will be send to the support. If the model includes linked DWG files, those files will be attached as well.

There is an option to add more files (e.g. pictures) to the report.

Company settings

It is a very fast way to export all FEM-Design settings and libraries to any other computer. You are able to deliver your company preferences from one computer to another very easy and quickly.



Export Import

Pick export or import settings in the FEM-Design Center and select the items, which wanted to be forwarded. The *company settings* file has **.fdcs* format.

The following items can be exported:

- settings
- libraries
- doc templates
- list batches
- title blocks
- list templates
- Office OpenXML templates

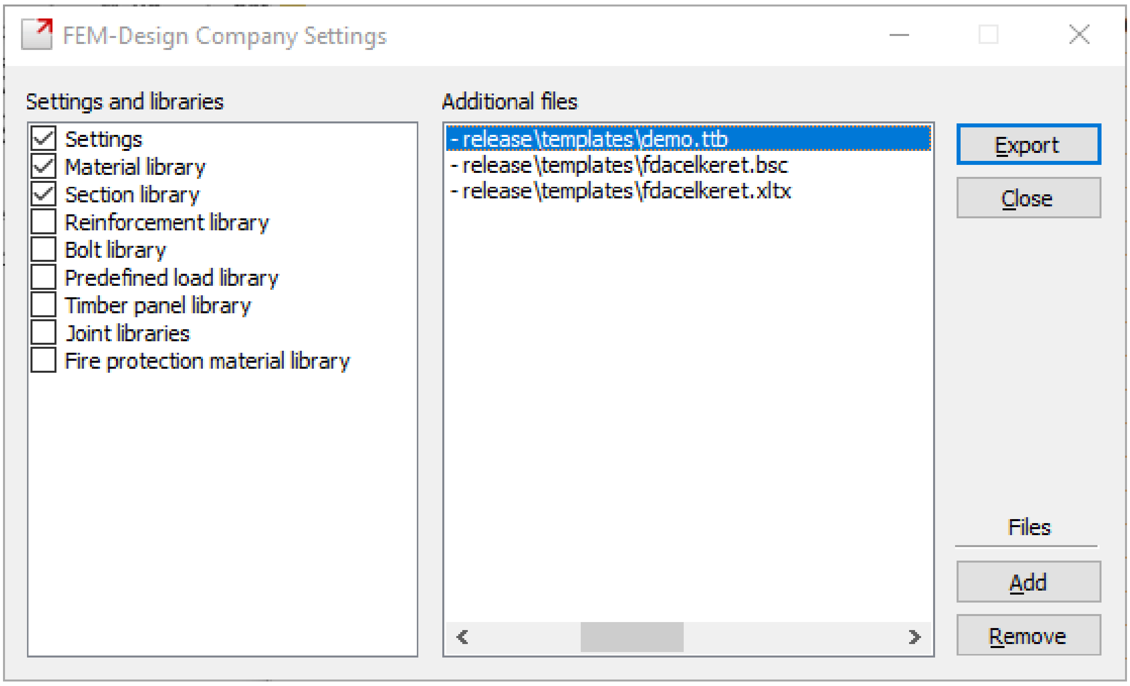

FEM-Design Company Settings
Settings and libraries
Settings
Material library
Section library
Reinforcement library
Bolt library
Predefined load library
Timber panel library
Joint libraries
Fire protection material library
Additional files
- release\templates\demo.ttb
- release\templates\fdacelkeret.bsc
- release\templates\fdacelkeret.xltx
Export
Close
Files
Add
Remove

TABLE OF CONTENTS

Tartalom