This document gives a detailed summary of the new features and improvements of FEM-Design version 17.

We hope you will enjoy using the program and its new tools and possibilities. We wish you success.

StruSoft,
Developer Team

Legend

 Pay attention / Note

 Useful hint

 Example

 Clicking left mouse button

 Clicking right mouse button

 Clicking middle mouse button

What is new in FEM-Design 17?

The list of all new major features and improvements is following:

- Correct model tool
- Plastic calculation of trusses, supports and connections
- Post-tensioned cable
- Fire design for steel bars
- Pile
- Composite sections for beams, columns and piles
- Wall buckling
- Properties dialogs available from the quick menu
- Joint stiffness calculation
- Displaying relevant load combination for max. of combinations results
- Fully rigid diaphragm
- Colour palette for displacement-like results in 3D modules
- New tables to list and new listing options
- Wall corbel
- Joint library and many more improvements of steel joints
- Notional load
- Manual position numbering
- Improved documentation module

What is new in FEM-Design 17.1?

- Construction stages (Phase 1)
- Improved storey management
- Run Script
- Option to hide extra information in the numeric results

Table of contents

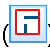
1. Tools

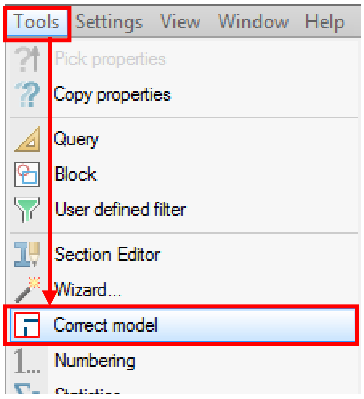
1.1. Correct model

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:

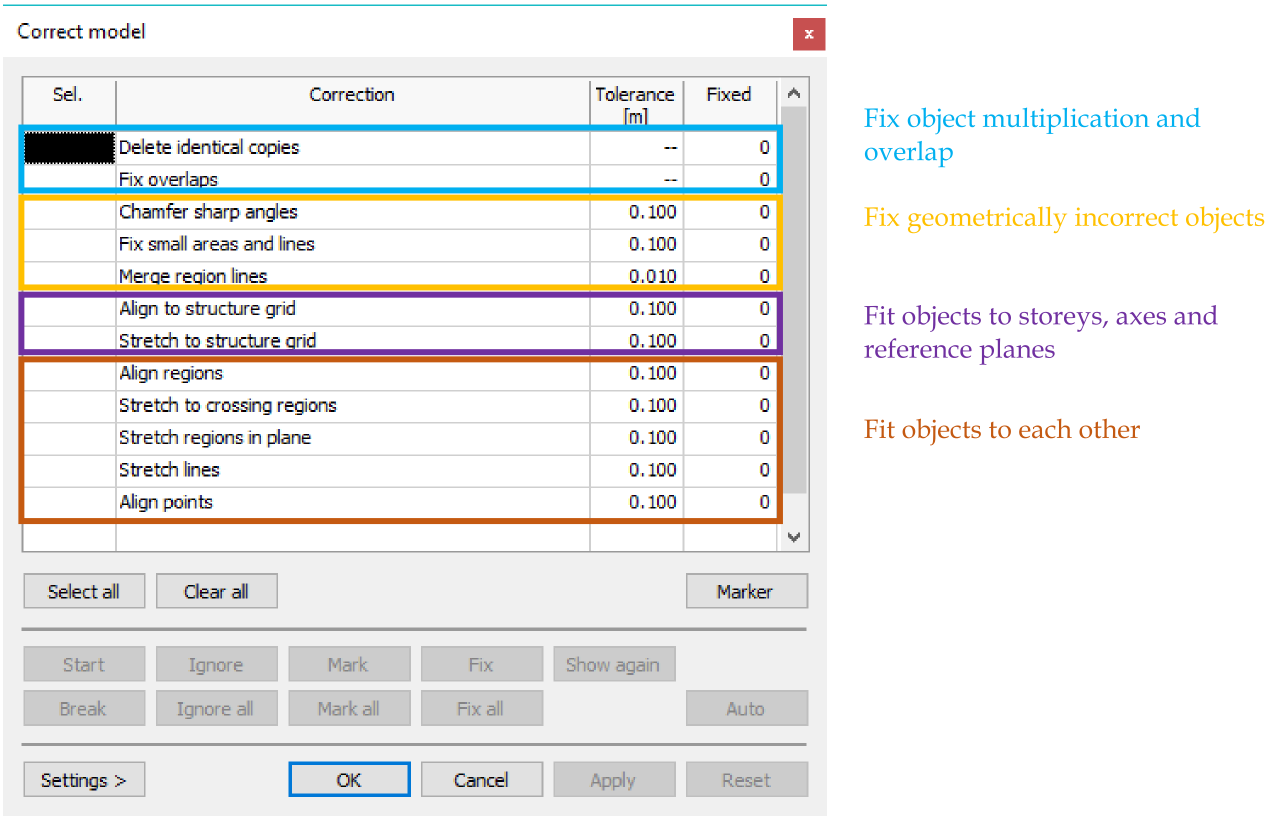
- object multiplication and overlap
- geometrically incorrect objects (very small region parts, very small angles, divided region edges, etc.)
- incorrectly positioned objects

Correct model tool is the replacement of *Object merge* (in earlier versions), which is no more available in version 17. 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 the User has to decide whether it is really an error or not, and if it is, how to fix it.

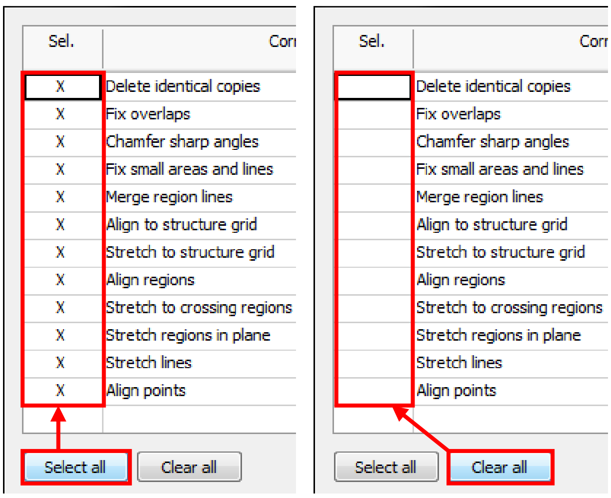
Correct model can be launched from *Tools/Correct model* or from the toolbar ().



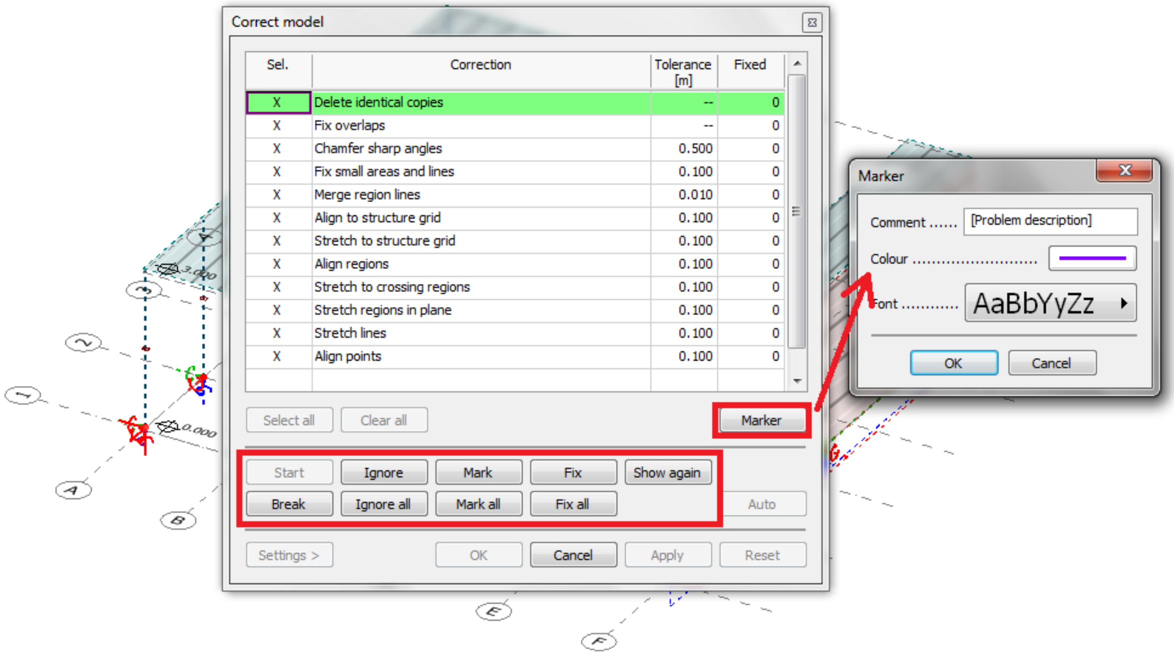
After launching *Correct model,* User needs to select the elements to be checked.



Then in the pop-up dialog, task(s) should be selected. Using the buttons *Select all* and *Clear all,* all correction tasks can be selected or cleared in one click.



After pressing *Start,* the correcting process begins. 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	

In order to decide what to do with each object, the User has the following choices:

- *Ignore*: the object will not be modified
- *Mark:* the object will be marked according to the colour and text set in the Marker dialog
- *Fix:* the object will be 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.

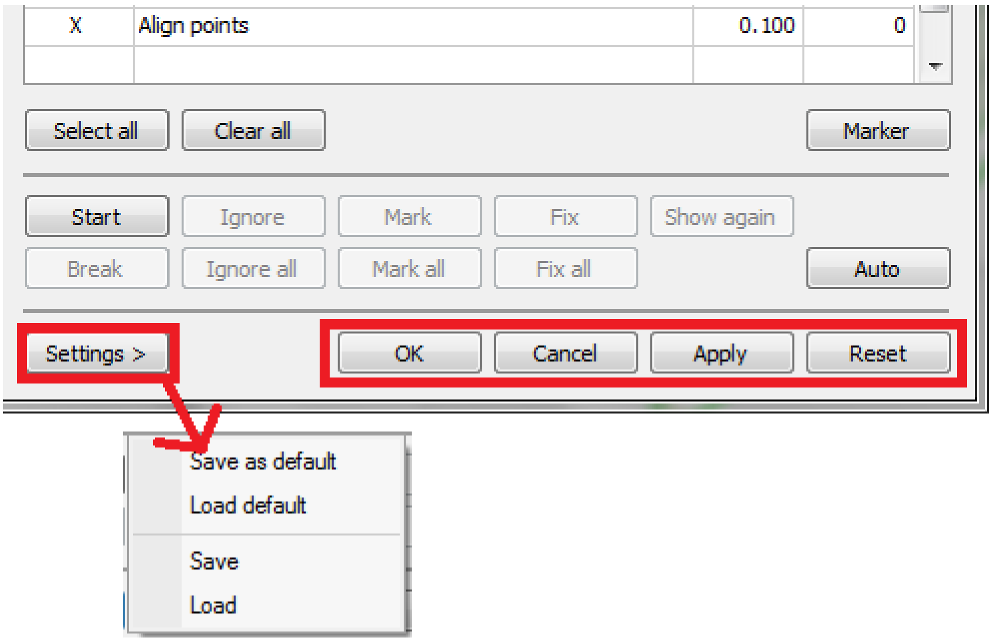
Ignore all, Mark all, Fix all acts on all further incorrect objects found by the **current correction task** (highlighted by green in the table).

After one of the abovementioned buttons is pressed, the next incorrect object – if exists – gets into focus and is highlighted

Show again focuses on the current objects. It can be useful if the User gets lost in 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 to 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.  A small but quite complex example at the end of this chapter shows in detail how to use *Correct mode*l tool.

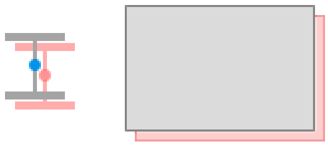
Using *Auto* button and applying its modification on the model is not recommended, because in practice in most cases there is no exact solution:

- 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 their type in the model.

See detailed description of each correction task as follows.

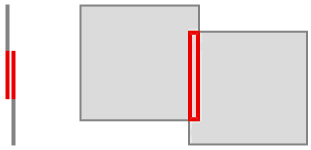
Delete identical copies



If more identical element exists in the same position, it deletes all identical copies, 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

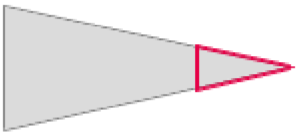


It fixes overlapping regions and lines. Existing objects can be shrunk or erased, but function will not generate new objects (e.g. a region can not fall apart).

Overlapping is allowed for the Loads.

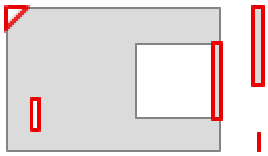
⚠ This is not working for isolated and wall foundations' regions and corbels.

Sharp angles



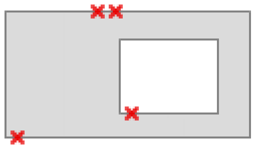
Sharp corners under 10° will be fixed according to the given tolerance. The highlighted part of object will be removed. If an element has smaller dimension than the given tolerance, the whole element will be removed.

Fix small areas and lines

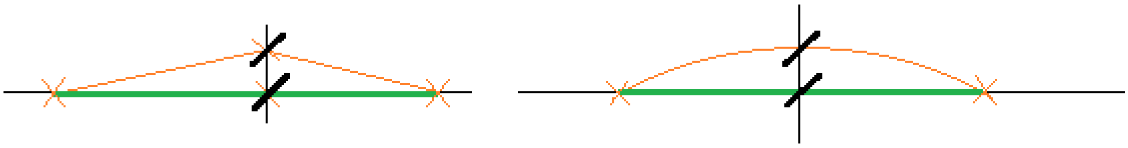


Objects within the tolerance range will be removed. It will fix small holes, long narrow areas etc.

Merge region lines

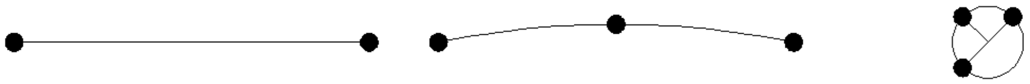


Merge region lines function merges two lines/curves of a region into one line/curve if they are in the range of tolerance. Too flat curves may be replaced by line. The figure below shows examples how the tolerance is measured.



Align to structural grid

This function aligns objects to 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:

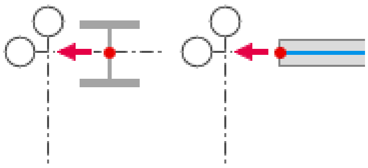


⚠ Once an object is placed on a grid by *Correct model* tool, it will not be detached from it by any next correction step.

⚠ *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

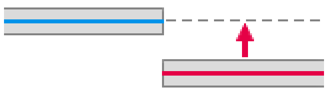
This function will stretch regions and lines to the intersection of axes within given tolerance.

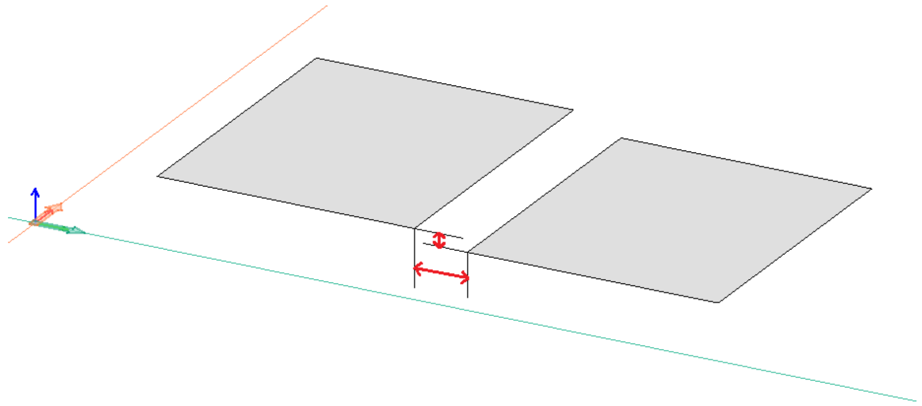


⚠ If the geometry of the object is incorrect (e.g. shell is not laying in plane) this function cannot be performed.

Align regions

It aligns a region (projection) to other region's plane that is within the given tolerance both in direction perpendicular to and parallel with the region's plane.

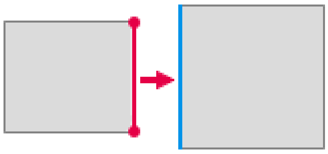




Stretch to crossing regions

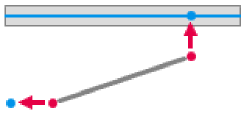
This function will stretch objects to the crossing regions within given tolerance.

Stretch regions in plane



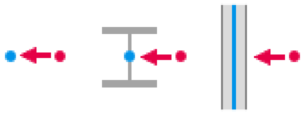
Regions laying in same plane will be stretched to each other within given tolerance.

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.

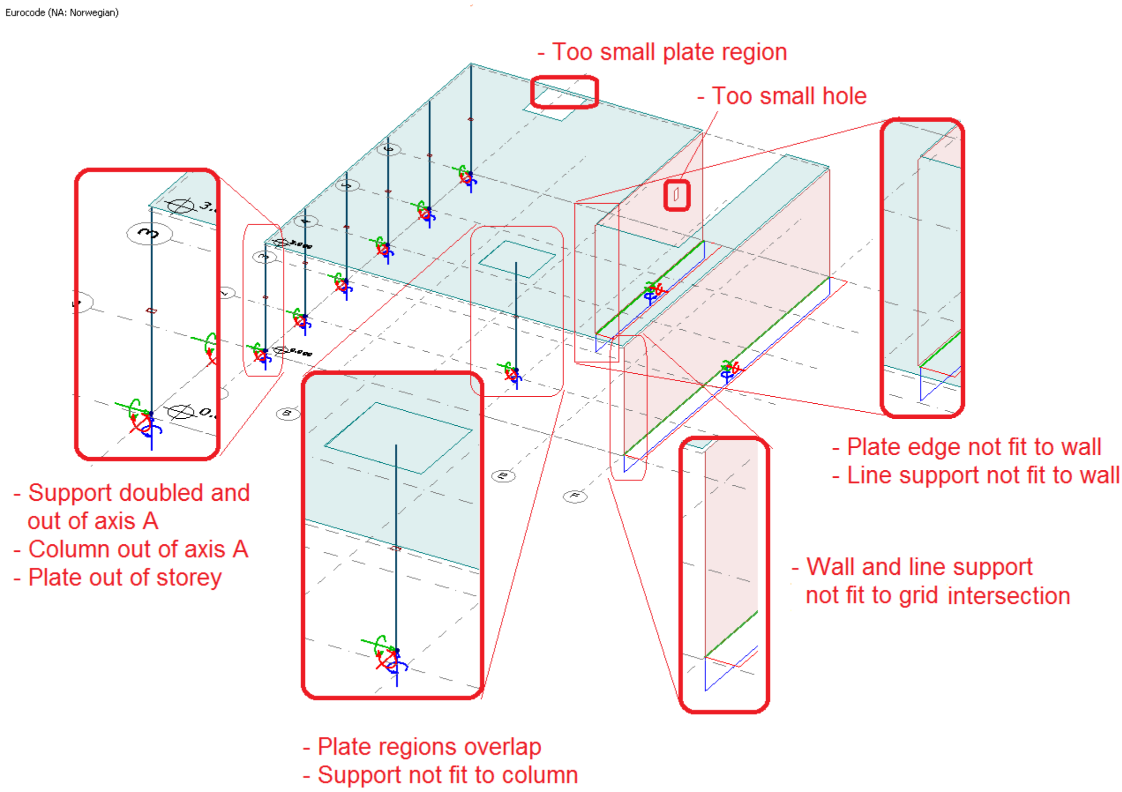
Restrictions

- *Correct model* cannot handle:
  - peak smoothing regions
  - connections (point, line, surface)
  - bar, shell components
  - building cover
  - it 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, wall 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.

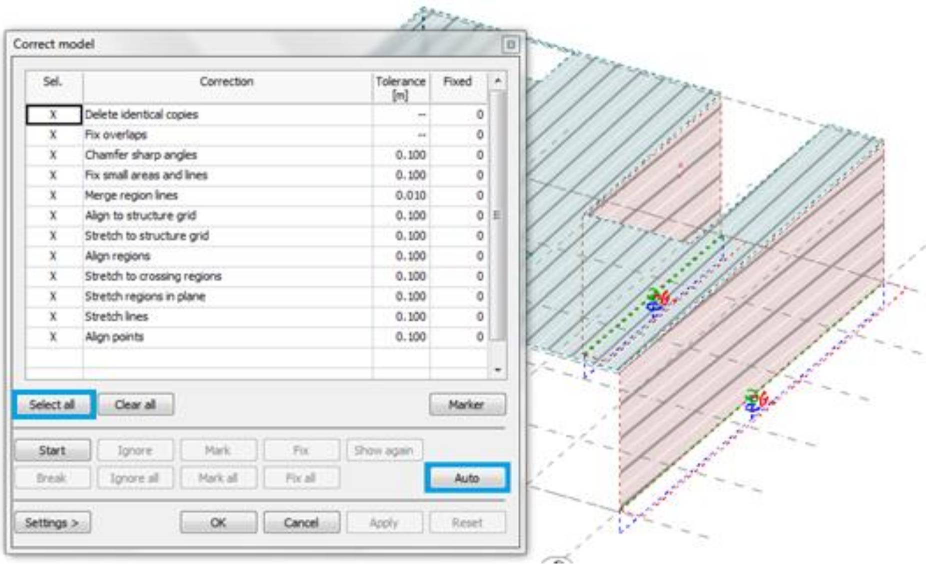
Example

The following example shows the process of fixing geometrical errors of a small model with different types of errors:



The example file can be downloaded from [here](#).

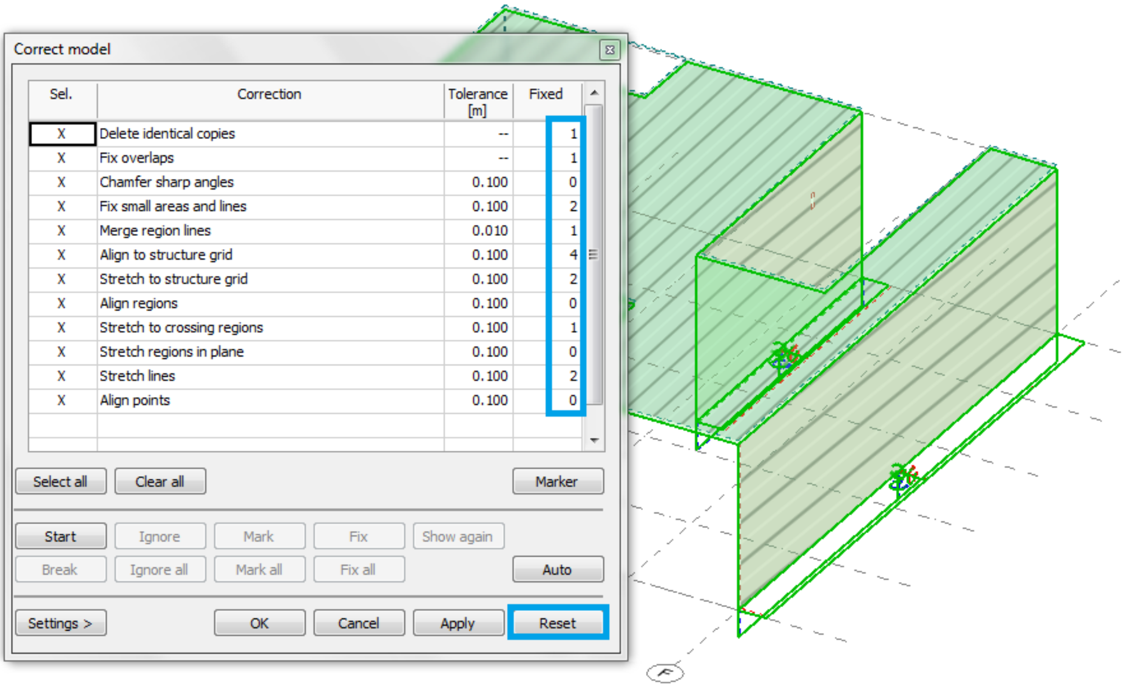
Let's launch *Correct model* tool. Select all structural elements by Ctrl+A (or box), then all correction tasks in the dialog by clicking on **Select all**. To get a quick overview of what kind of problems occur in the model, just click on **Auto** button.



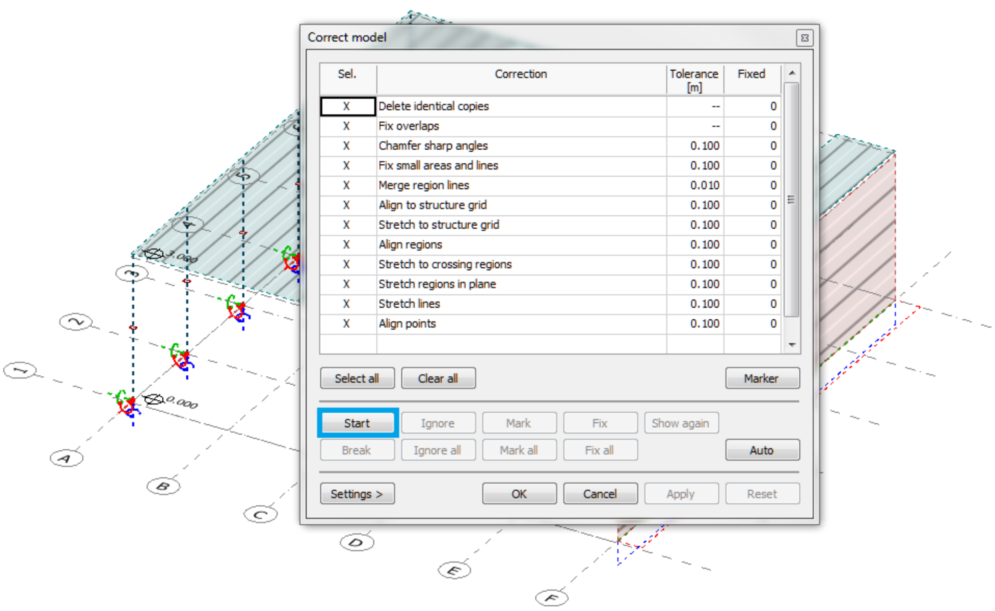
The process searches for all errors and fixes them in a way that most probably is not always the way that the user intends to fix them However, this way User can get a good estimation on the type and amount of errors (by looking at the values in the *Fixed* column).

So, in this case, all together about 15 errors should be fixed.

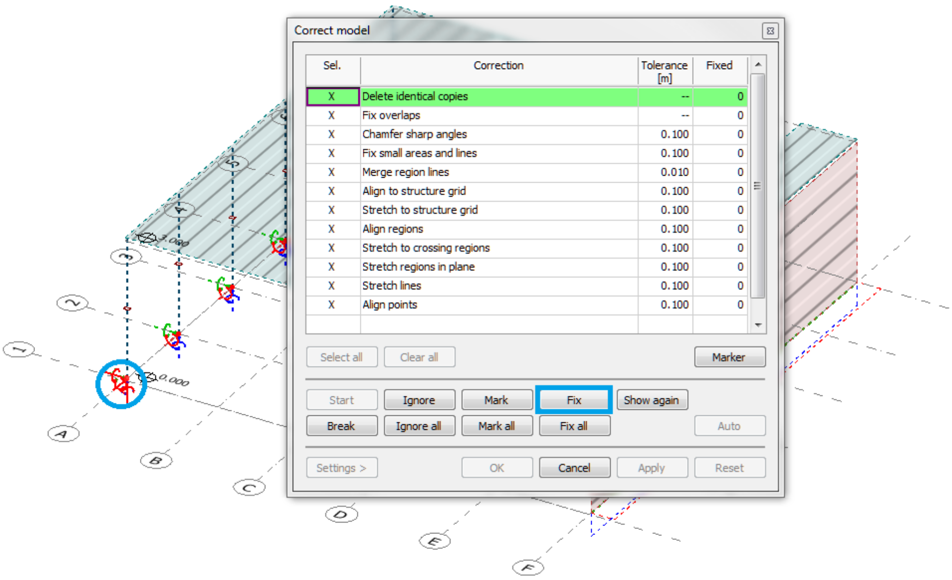
Now click on **Reset** to discard all modifications done by the automatic process.



Now click on **Start** to see and decide about all the errors one by one.

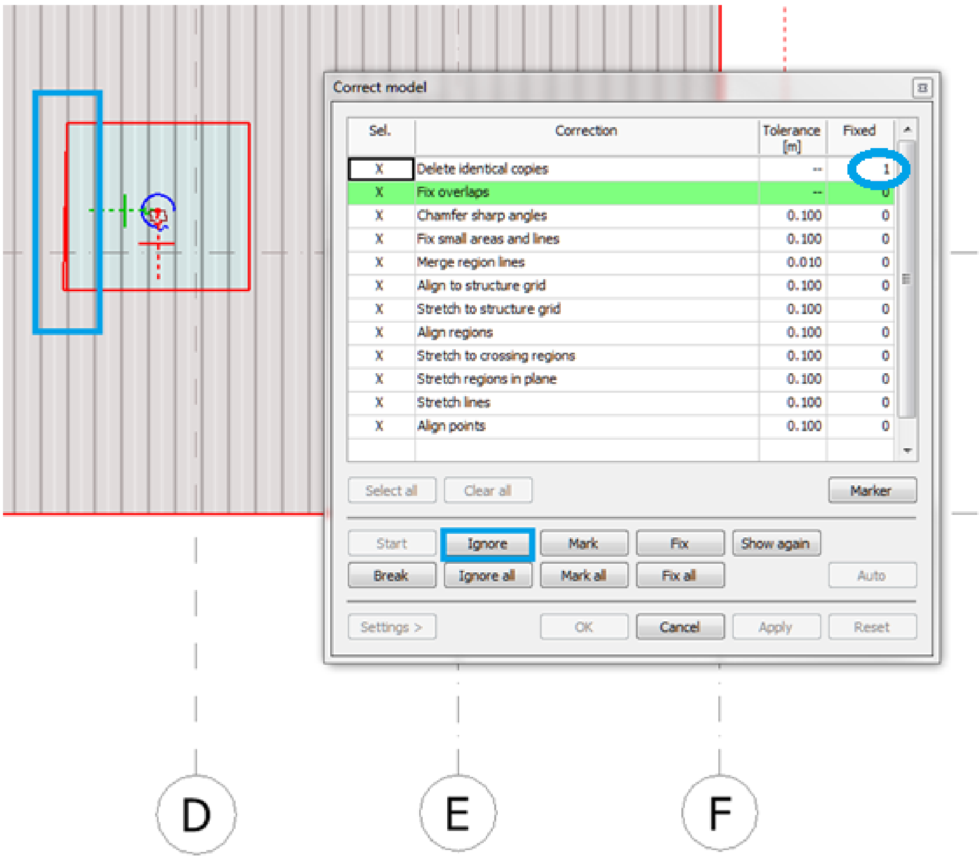


The doubled support in the bottom left corner is found first. Now click on **Fix** to eliminate it.

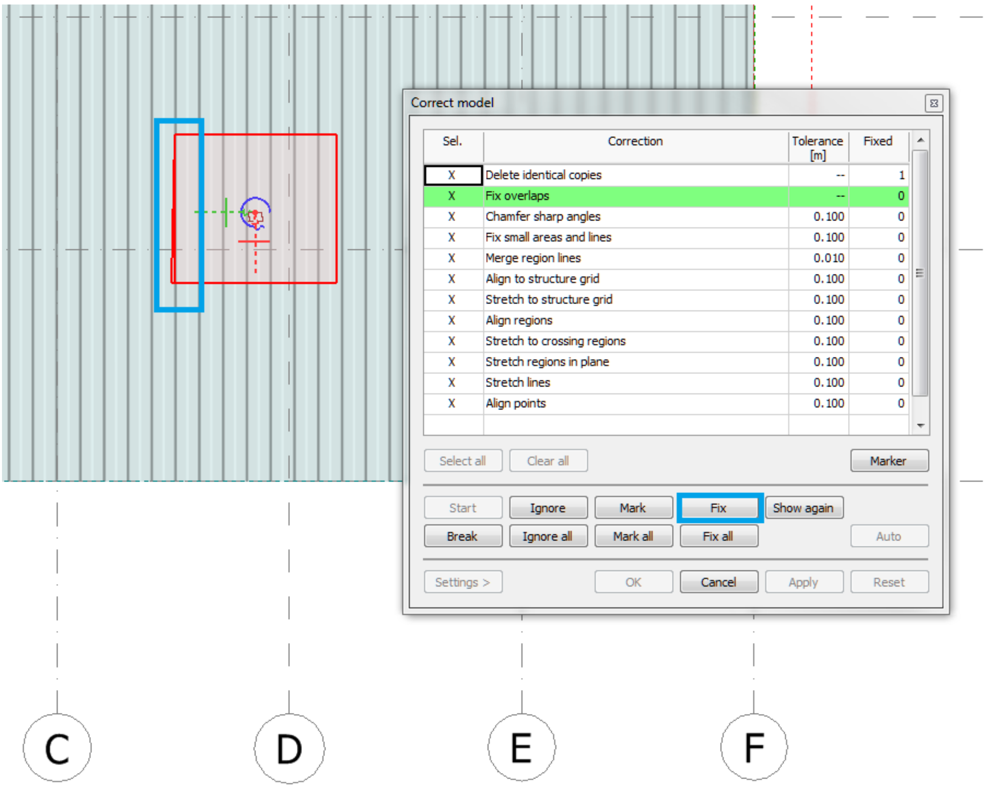


The number in the *Fixed* column of “Delete identical copies” task is set to 1.

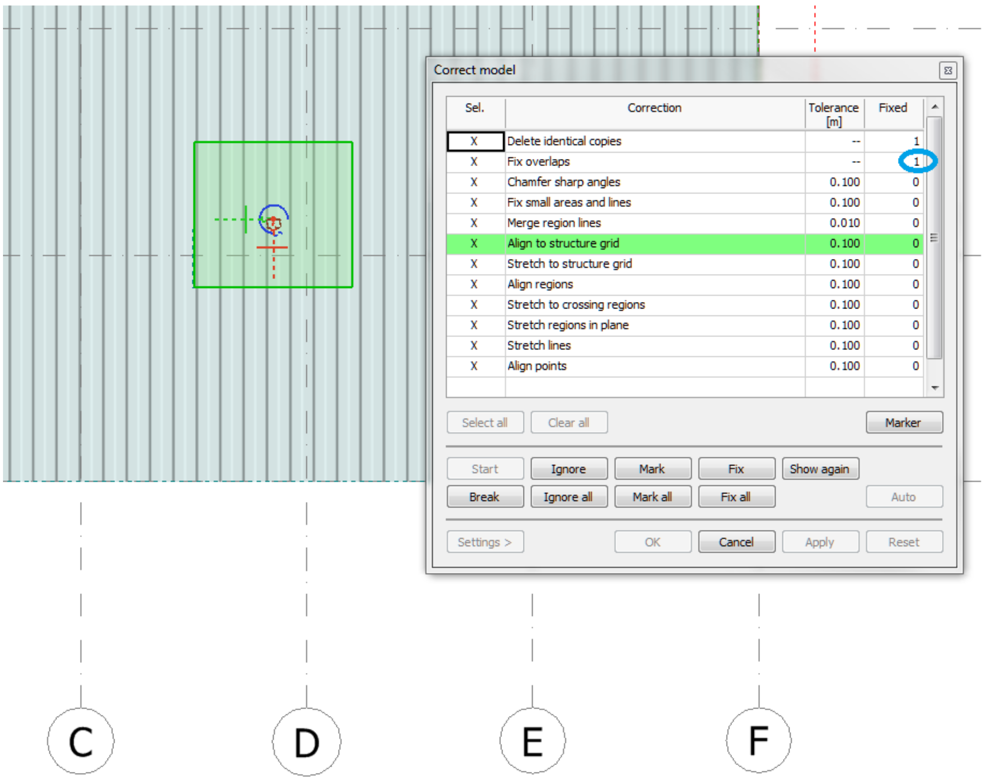
Overlap of the large plate and the small plate over the column is found as next error. The large plate is blinking, so by clicking on **Fix,** that one would be modified. However, we know that it is the small region that is incorrect, so this time let’s click on **Ignore** instead.



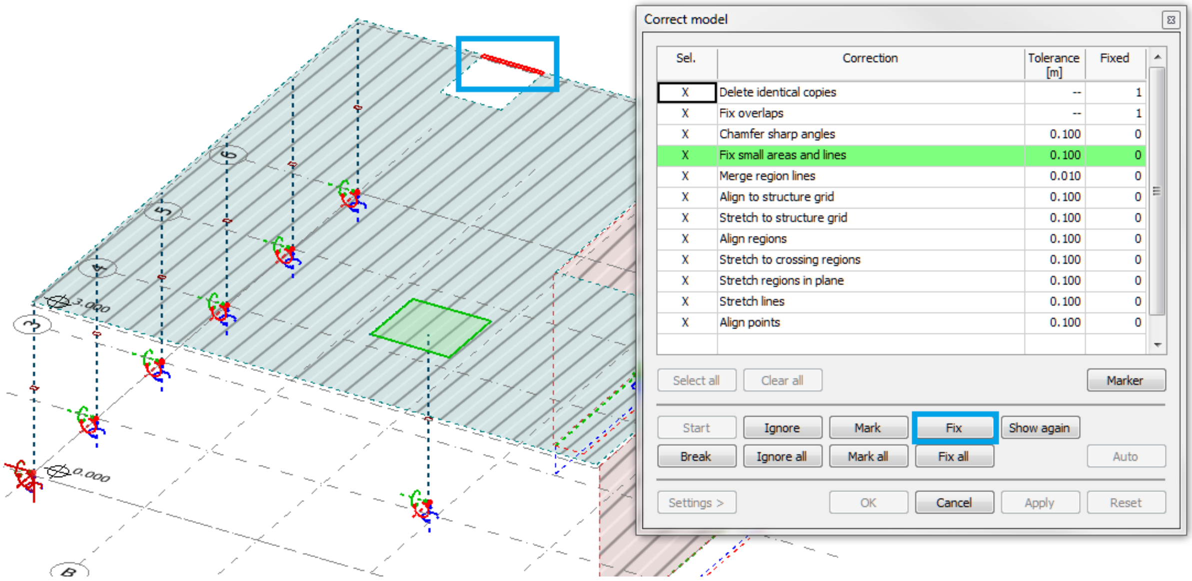
Now the small plate is blinking, so by clicking on **Fix,** it is modified and fit into the large plate.



The corrected small plate is displayed in green contour. The number in the *Fixed* column of “Fix overlaps” task is set to 1.

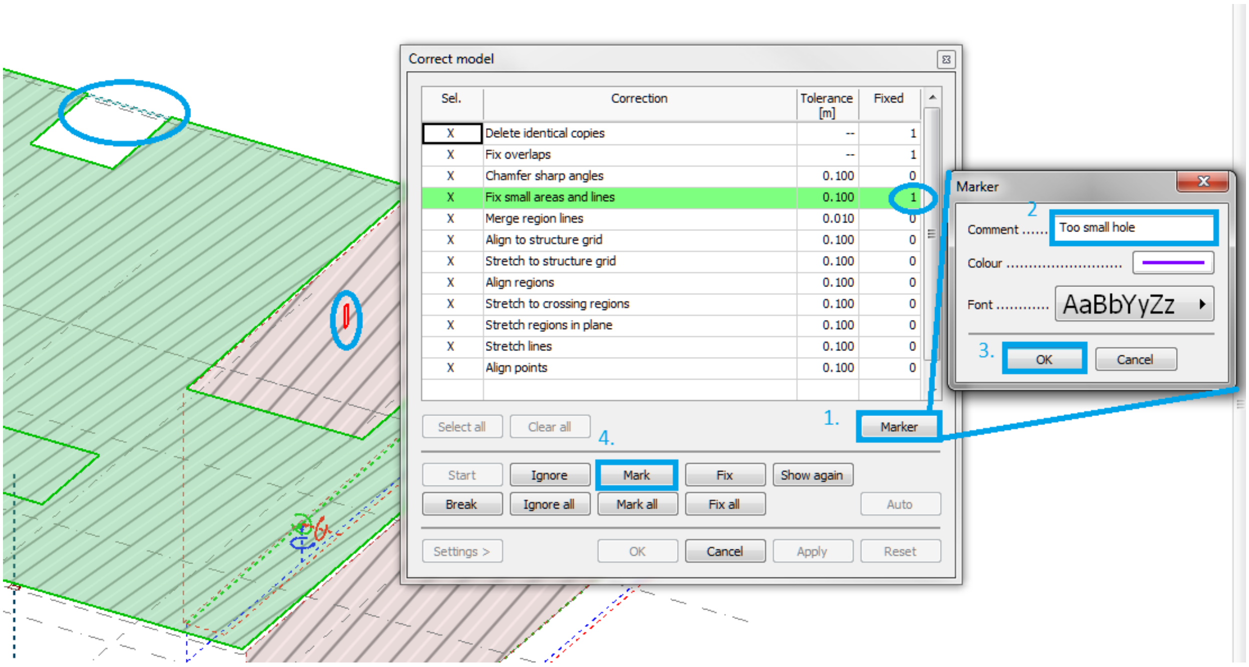


The next problem is found by “Fix small areas and lines” task: a very small region part. Let’s remove it by clicking on **Fix**.

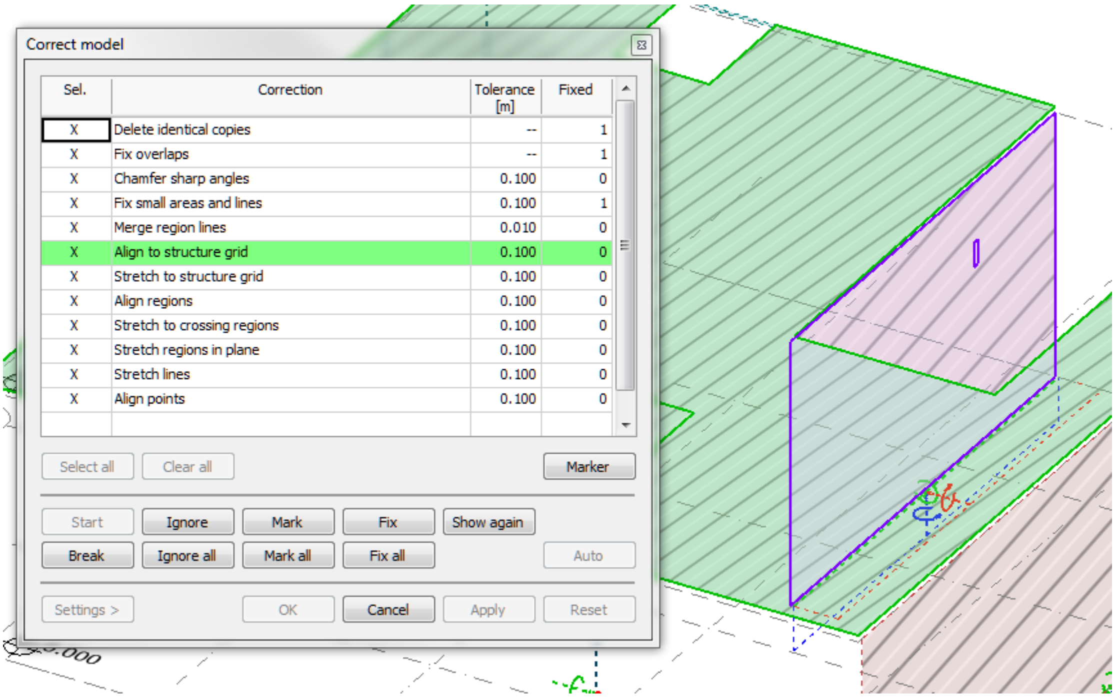


The removed small region part is displayed by dashed line, and number of fixed objects for “Fix small areas and lines” task is set to 1.

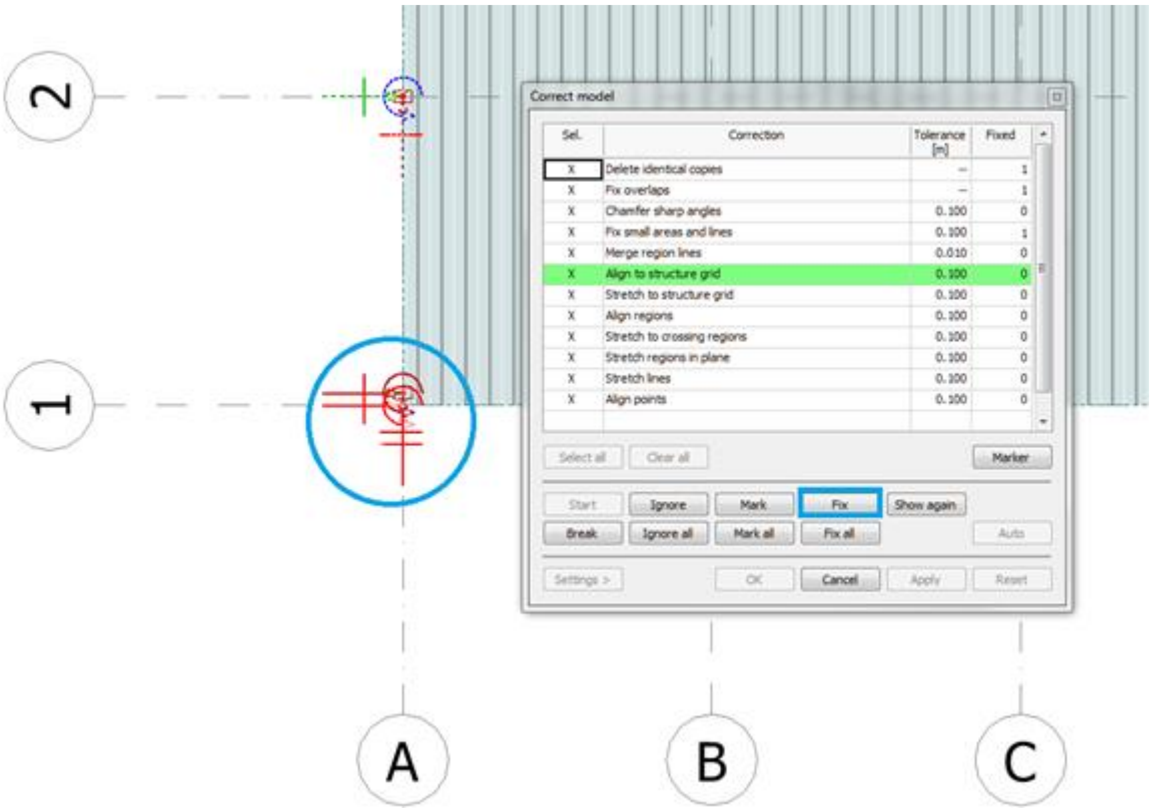
The next problem, a very small hole is highlighted. We decide not to delete it but deal with it later, so let’s mark it. First let’s click on the **Marker** and set the marking text, colour and font type. After clicking OK in the Marker dialog, let’s click on the **Mark**.



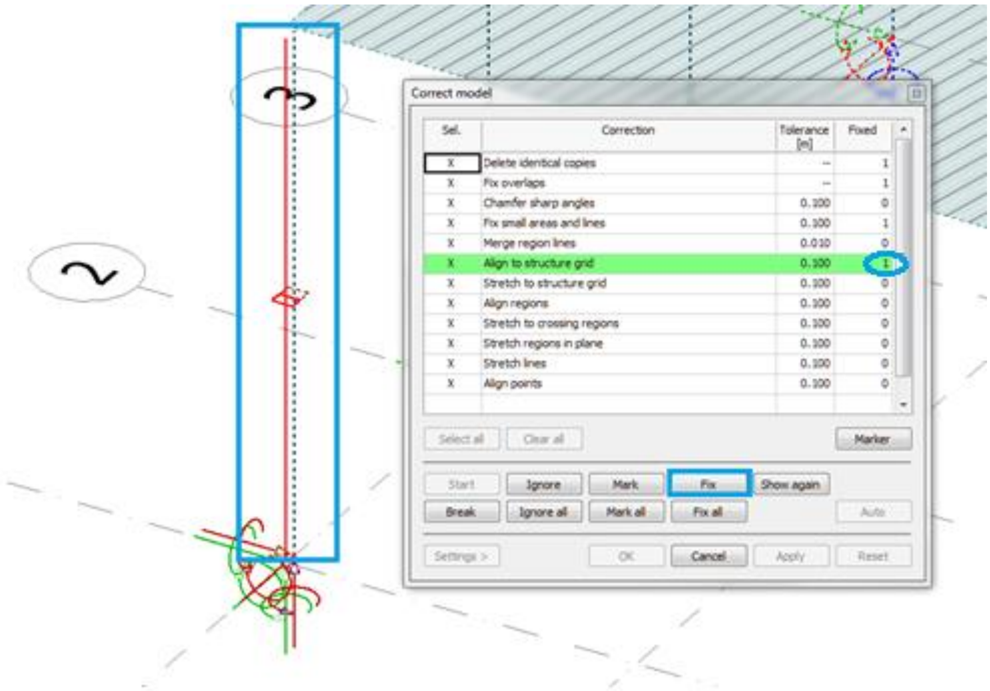
The wall with very small hole will be marked for later investigation.



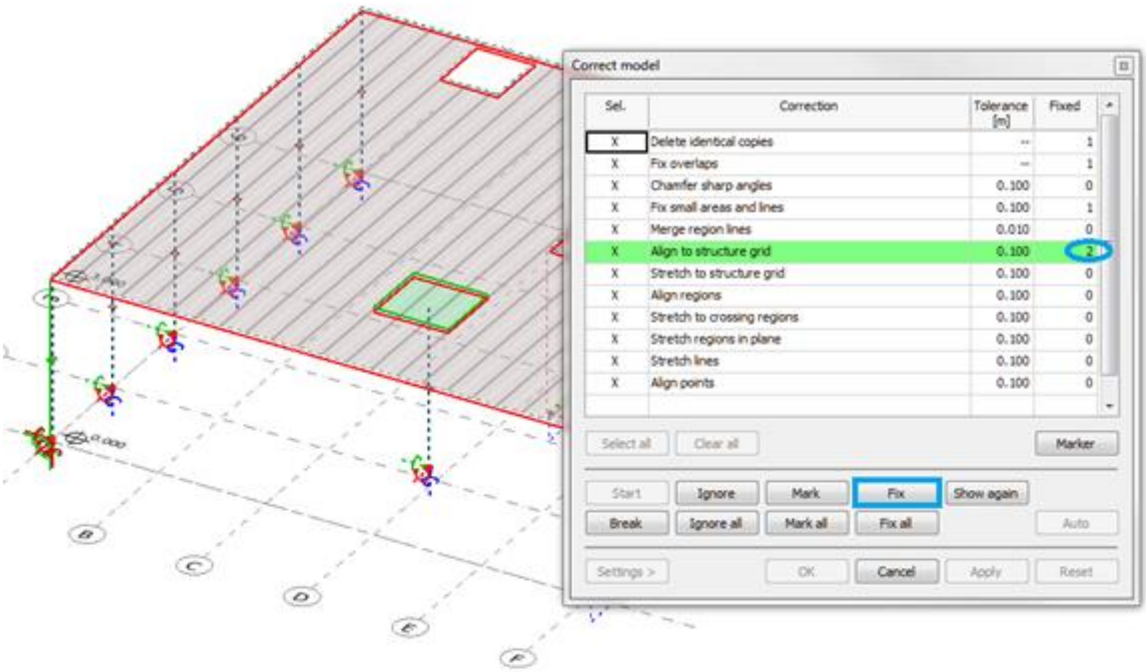
The next error is the support that is not on Axis 1, but within the given *Tolerance* (0.1 m) for “Align to structure grid”. Let’s click on **Fix,** to align it to Axis 1.



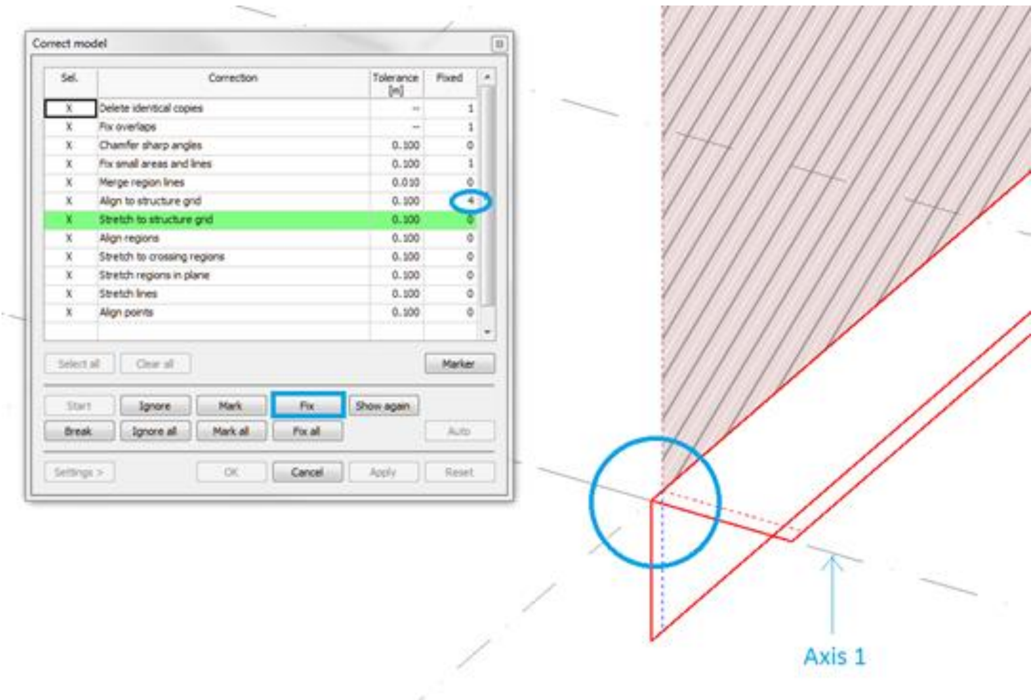
The number in the *Fixed* column of "Align to structure grid" task is set to 1. The corrected support is green on the next picture. The next misplaced object - the column, above it - is blinking. Let's fix it too.



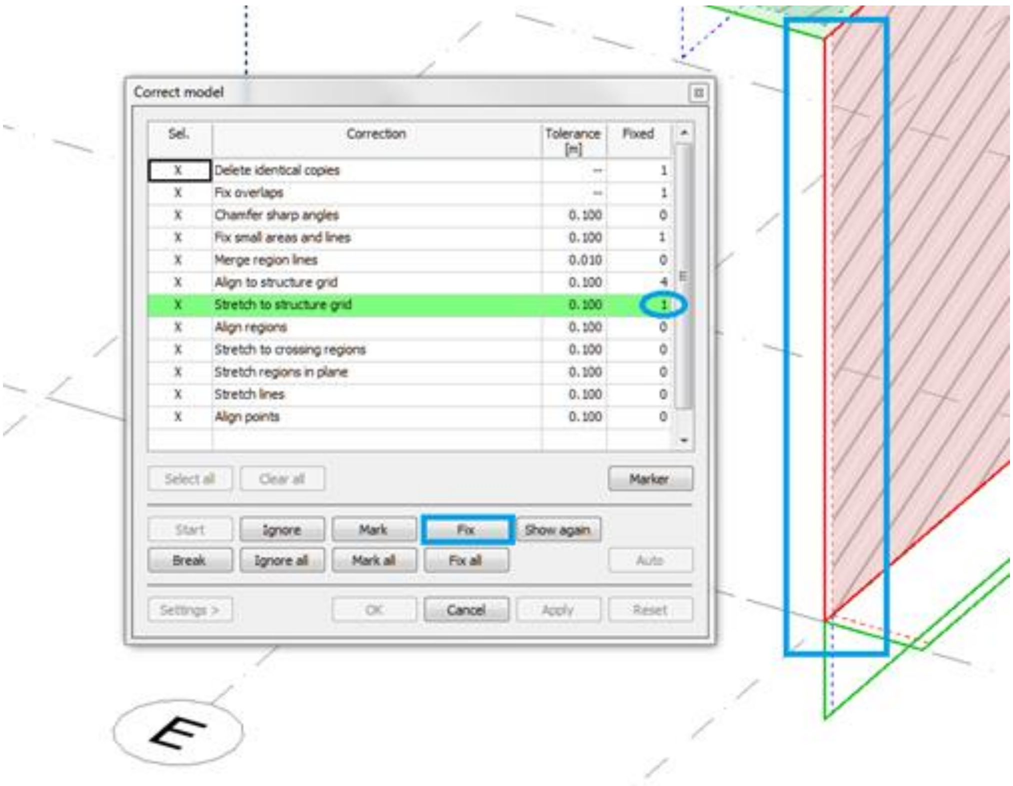
The number in the *Fixed* column of "Align to structure grid" task is increased to 2. The corrected column is green on the next picture, and the next misplaced object - the large plate - is blinking (since it is slightly above Storey 1). Let's fix it as well and do the same for the next blinking small plate.



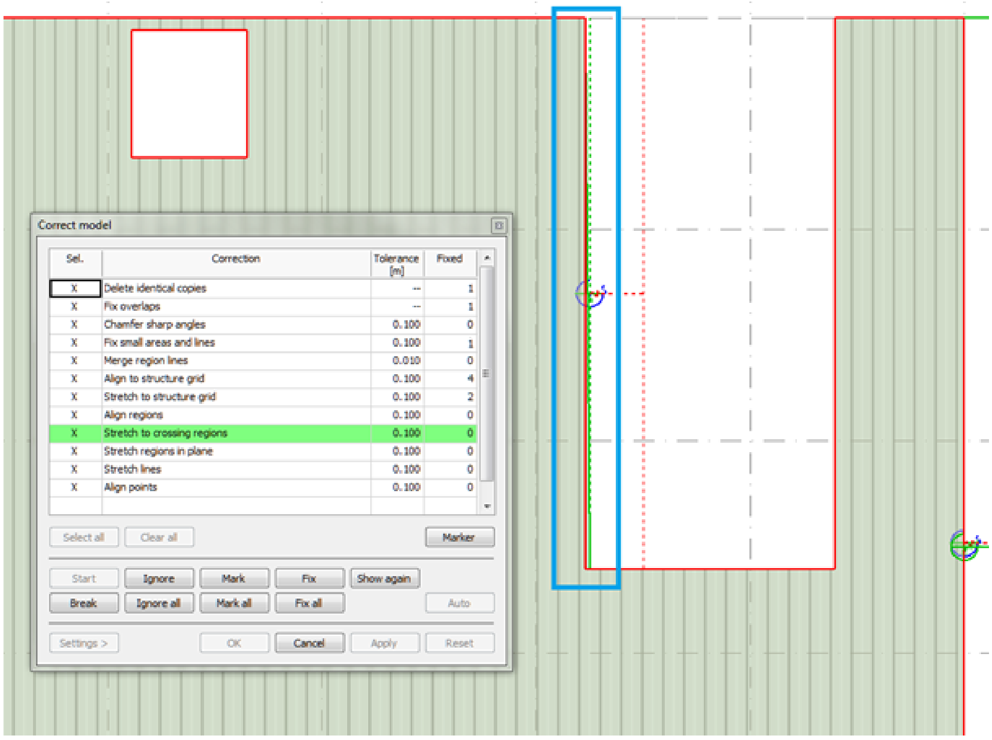
Finally, four objects are fixed by "Align to structure grid" task (one support, one column and two plates) and the next incorrect object is the line support that is just a bit short to reach to Axis 1. Let's click **Fix** to stretch it to Axis 1.



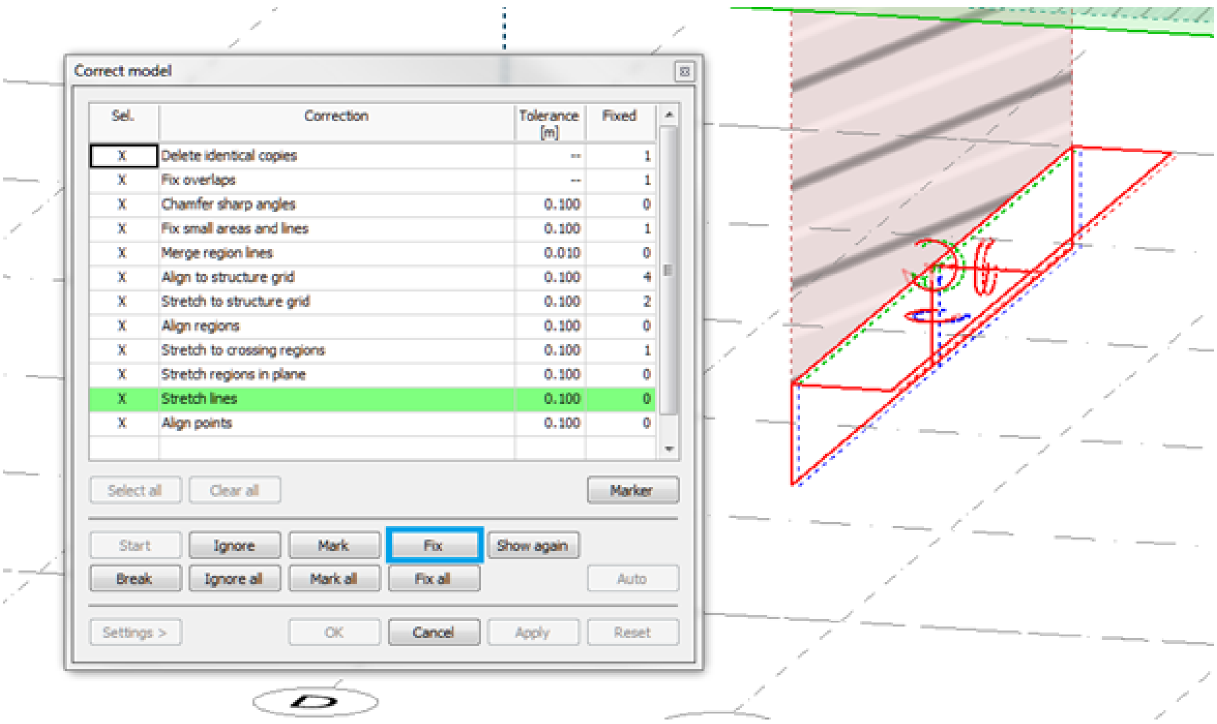
The number for the *Fixed* objects in the "Stretch to structure grid" task is set to 1. The next object to be fixed is the wall above the previously fixed line support, which is also too short to reach Axis 1. Let's click **Fix** to stretch it, too.



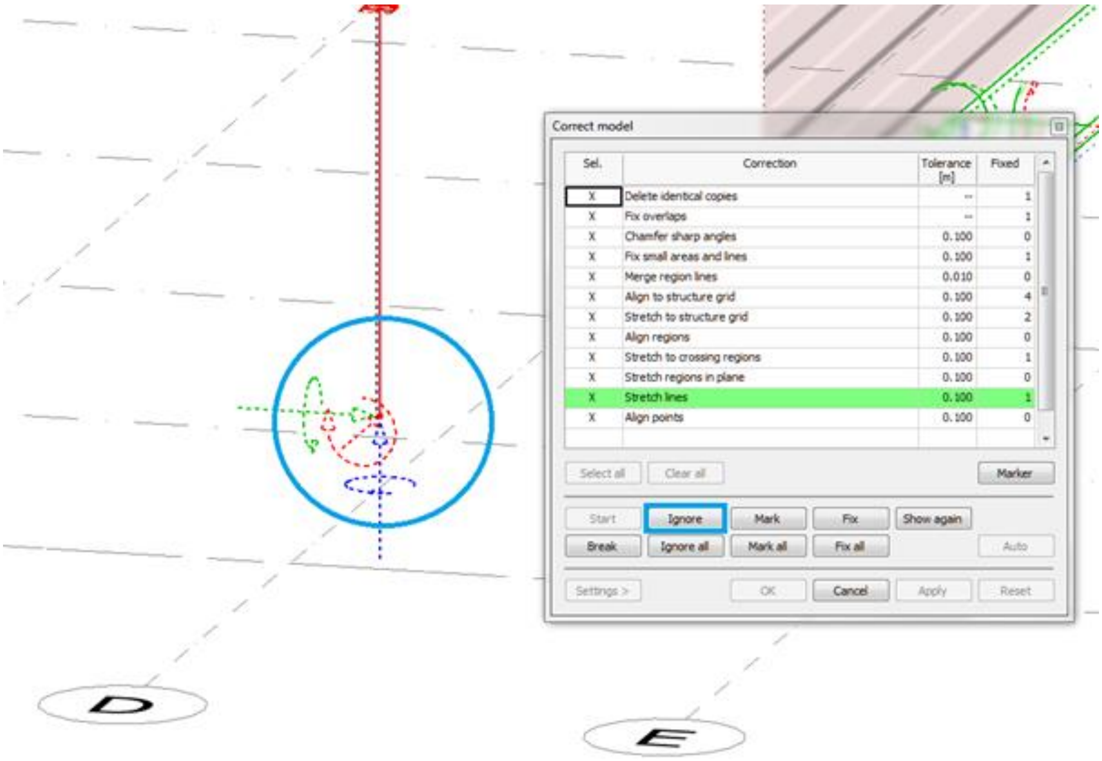
The next error is found by the "Stretch to crossing regions" task: the plate edge does not fit to the wall below.



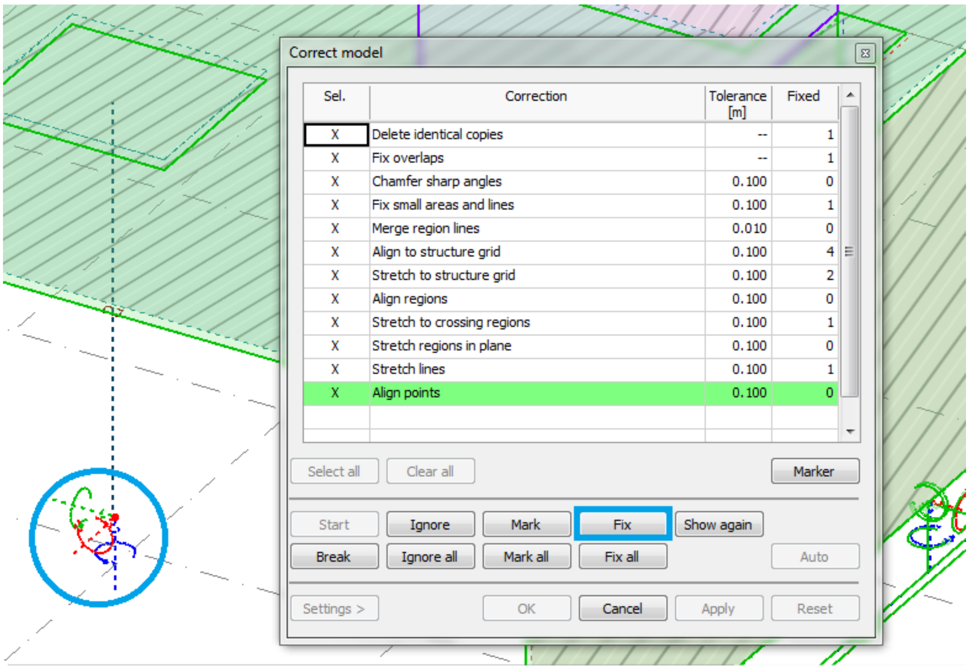
Then “Stretch lines” task finds a line support that is not connected to the wall above it. Let’s click on **Fix** to move it under the wall.



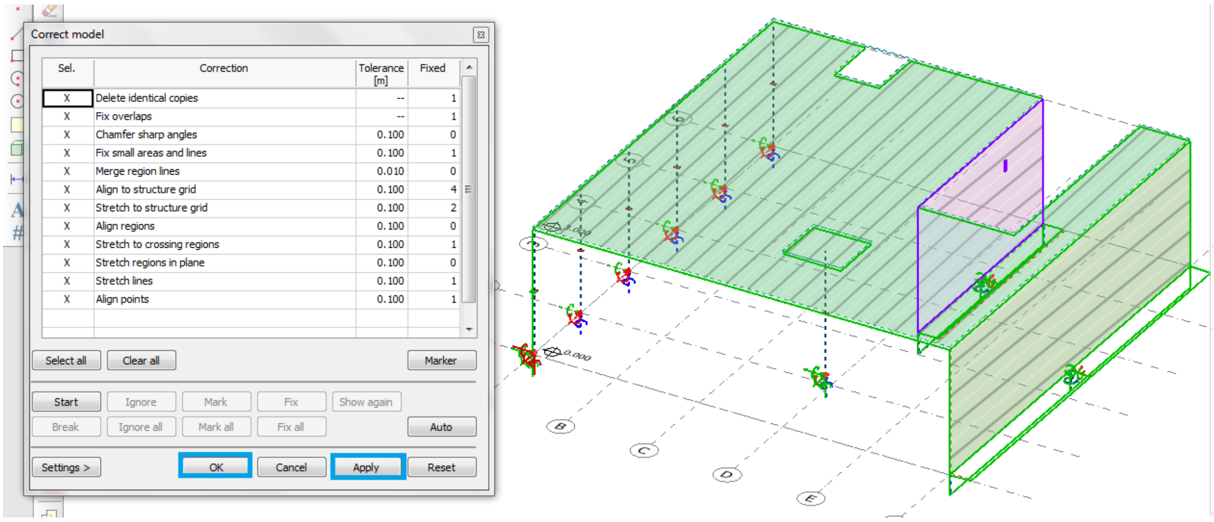
The next incorrect object is a column that is not exactly above the point support. But we decide that the column is at the right place, so click **Ignore** to skip it.



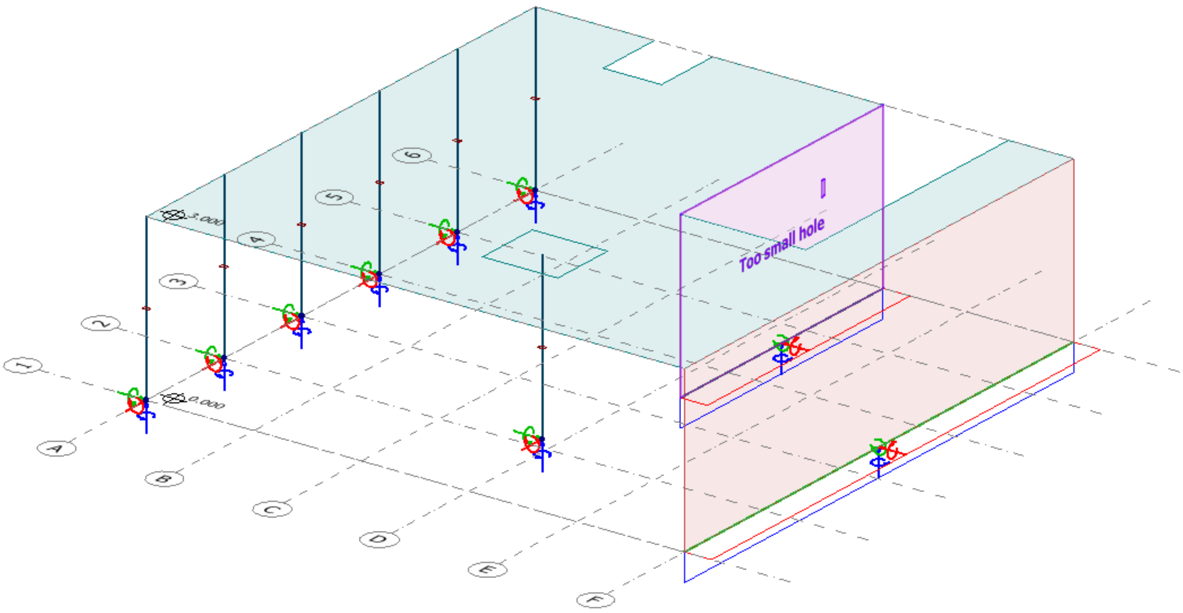
Now "Align points" task finds the point support which is almost below the column, so let's move it to the bottom of the column by clicking on **Fix**.



Since this is the last error, the green highlighting disappears from the Correction table and **OK** and **Apply** buttons become active.



By clicking on **OK,** the corrections will be applied to the model. The objects that were marked in the dialog are displayed with the marker text.



1.2. Scripting

We support a basic automation workflow through scripting. It is capable to load/save file, execute analysis calculation and create outputs as .csv list or .docx documentation. With that you can batch-analyze models created in other programs or directly in .struxml. Or execute long calculations during the night.

The script commands approximate the usual interface, as if you filled inputs on the dialog and press the OK button. So for the meaning of the parameters look at the corresponding panel in the UI.

To launch the script use the menu command Tools/Run script or start the program with the /s command line:

fd3dstruct /s c:\mydir\example.fdcsript

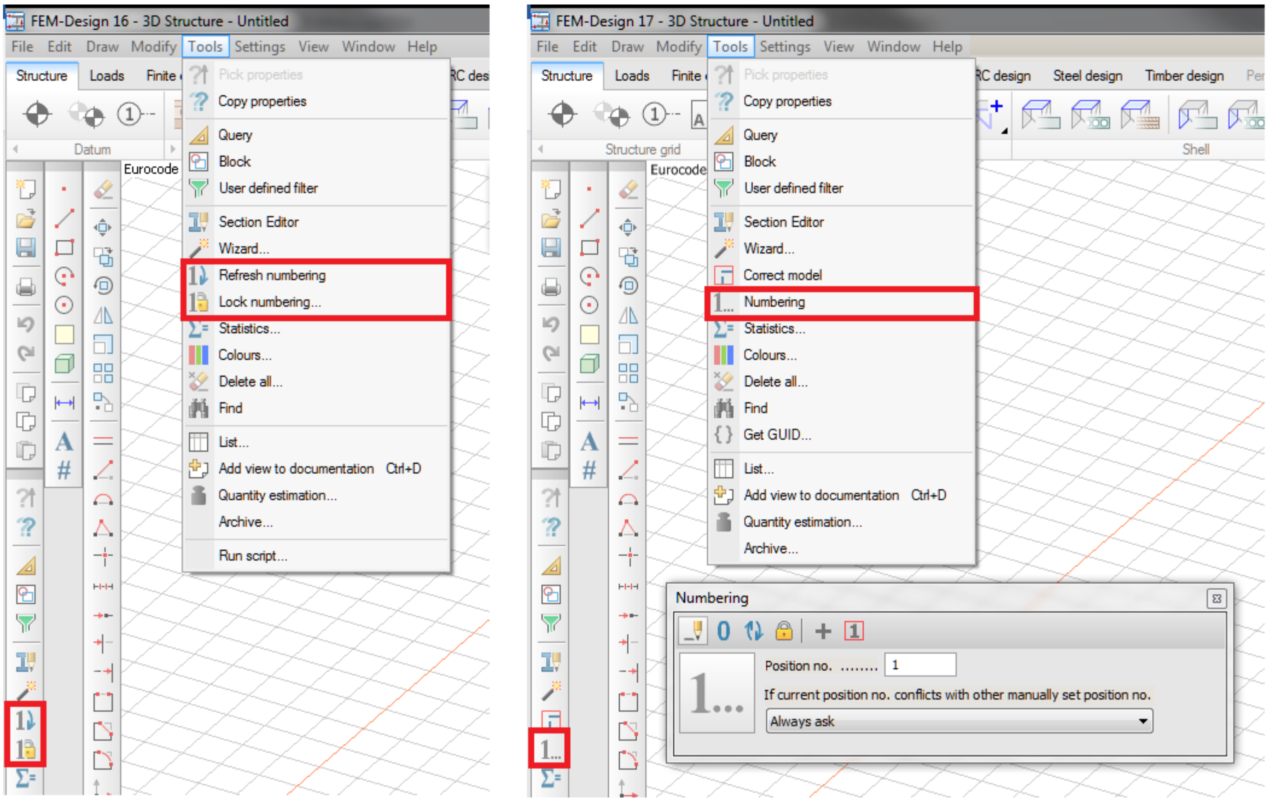
The installation of FEM-Design 17.01.001 (or later version) creates the **fdscript.xsd** and an **example.fdcsript** in the templates folder, the latter can be used as a starting point of custom scripts. It has all the commands intended for users and useful comments on how to proceed.

Script execution writes a log file (specified within the script header), you have to look into that to see what was actually executed and for any warning/error message that would normally appear on the screen. The execution stops at first serious error.

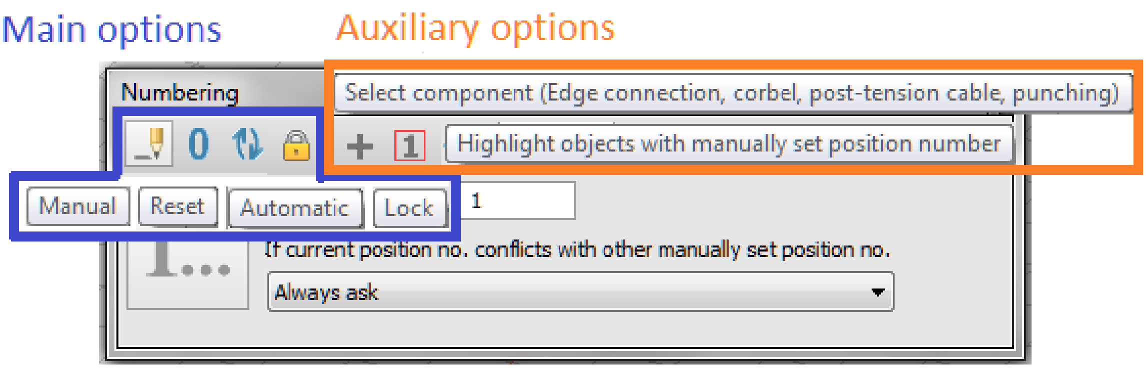
If you launch FEM-Design many times to execute script for many inputs it's a good idea to turn off all the satellite executables it may launch: update and upgrade in settings and set environment FD_NOLOGO=1 to avoid the splash screen.

1.3. Position numbering tool

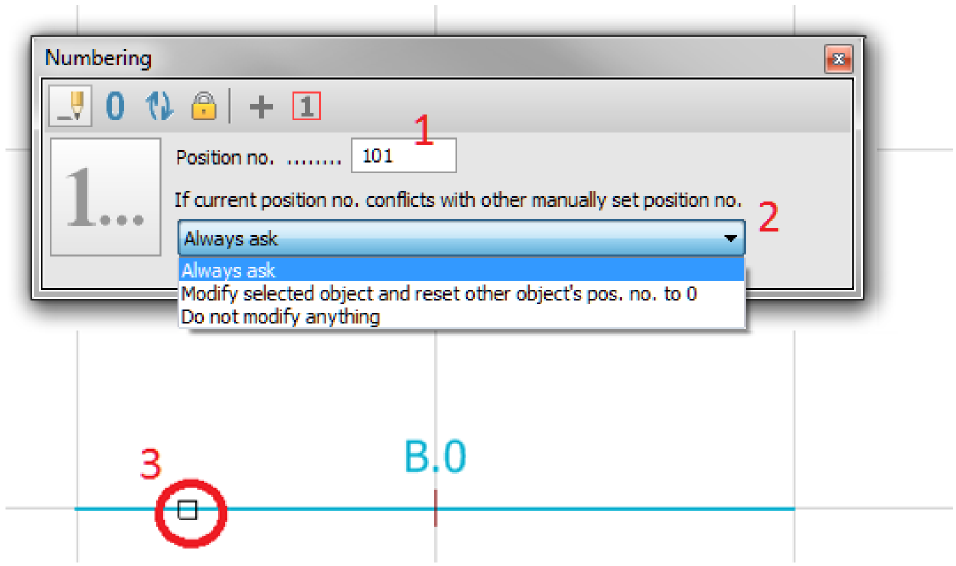
All position numbering possibilities are gathered into the new *Tools/Numbering...* tool. It lets the user manually set/reset the object's position numbers, it also contains *Refresh numbering* and *Lock numbering...* commands of FD16.



Four main and two auxiliary options are available in the tool window.



Manual position numbering

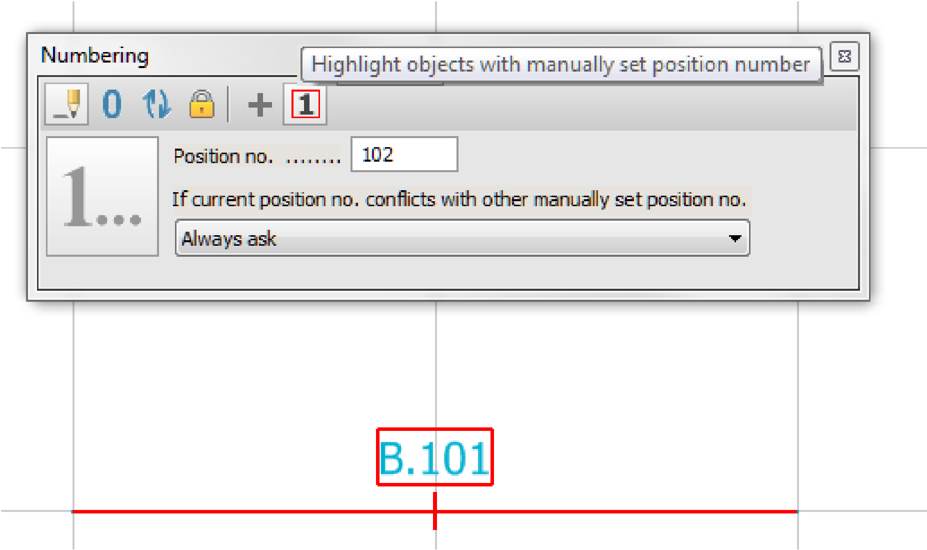


1. type required position number into the *Position no.* textbox
2. select option for handling position number conflict
3. select object(s) to set position number for

⚠ 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.



Reset position number

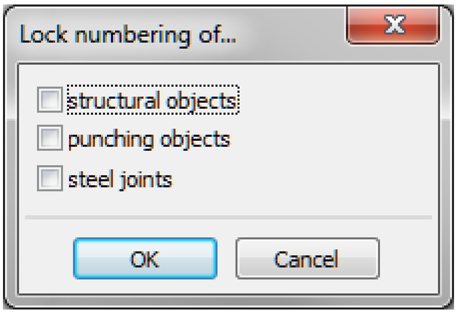
User can reset the position number to zero by choosing *Reset* option, then selecting one or more objects. *Select component* option is available here, too.

Automatic numbering

This option works the same way as *Renumbering* command in FD16. It sets position number automatically for all objects in the database except the ones with manually set position number.

Lock numbering

This option works exactly the same way as *Lock numbering...* command in FD16. If it is pressed, the following dialog pops up, where position number of different object types can be locked.

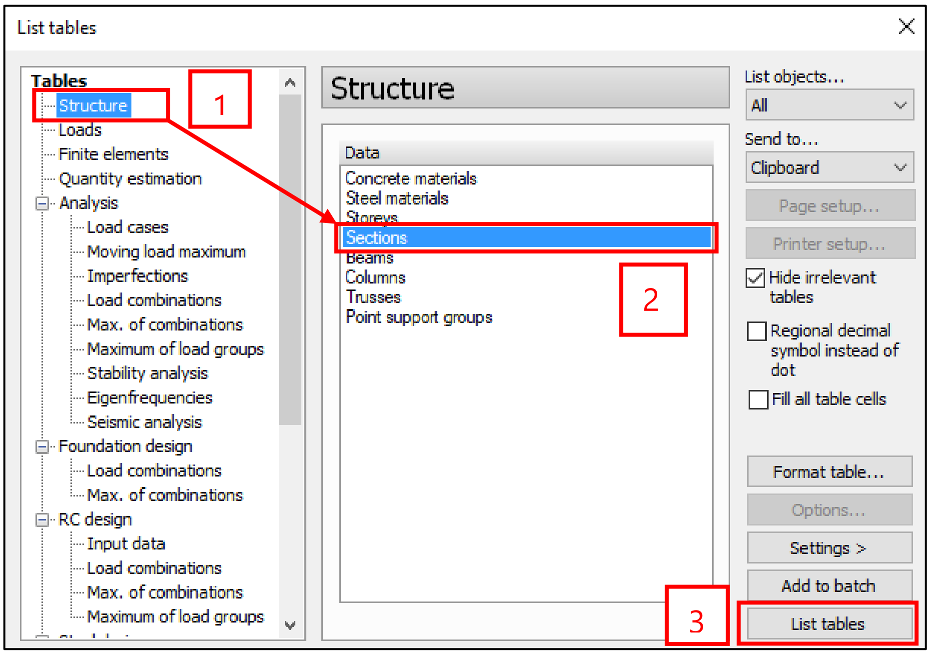


⚠ Manual numbering can not be applied on locked objects.

1.4. Section list

A new type of table – *Sections* – can be listed. It contains all sectional data (width, height, area, inertias, etc.)

In the *List tables* dialog select *Tables/ Structures/ Sections*, then click on the *List table* button.



In the generated Section table, the last column – *Other* - contains the detailed dimensions of the section. Currently it is filled only for RHS, I and composite sections.

Sections

Section	Composite	Height	Width	A	P	A/P	Yg	Zg
[-]	[-]	[mm]	[mm]	[mm2]	[mm]	[mm]	[mm]	[mm]
Steel sections IPE 300	No	300	150	5381	1160	4.6	0.000	0.000
Steel sections IPE 450	No	450	190	9882	1605	6.2	0.000	0.000
Steel sections IPE 330	No	330	160	6261	1254	5.0	0.000	0.000
Steel sections KKR 40x20x2	No	40	20	214	214	1.0	0.000	0.000
Steel sections KKR 70x50x2	No	70	50	454	454	1.0	0.000	0.000

Ys	Zs	Iy	Wy	ez max	ez min	iy	Sy	Iz	Wz
[mm]	[mm]	[mm4]	[mm3]	[mm]	[mm]	[mm]	[mm3]	[mm4]	[mm3]
0.000	0.000	83561132	557074	150	150	124.6	314179	6037784	80504
0.000	0.000	337429603	1499687	225	225	184.8	850900	16758611	176406
0.000	0.000	117669093	713146	165	165	137.1	402166	7881421	98518
0.000	0.000	40495	2025	20	20	13.8	1307	13428	1343
0.000	0.000	314755	8993	35	35	26.3	5400	187577	7503

ey max	ey min	iz	Sz	It	Wt	Iw	Iyz	z omega	alpha1
[mm]	[mm]	[mm]	[mm3]	[mm4]	[mm3]	[mm6]	[mm4]	[-]	[°]
75	75	33.5	62644	197538	11318	124256407470	-0	0	0.000
95	95	41.2	138282	660584	27370	780966316453	0	0	0.000
80	80	35.5	76882	275905	13973	196088467329	-0	0	0.000
10	10	7.9	798	34618	2192	287807	0	0	0.000
25	25	20.3	4291	375319	9641	3206691	0	0	0.000

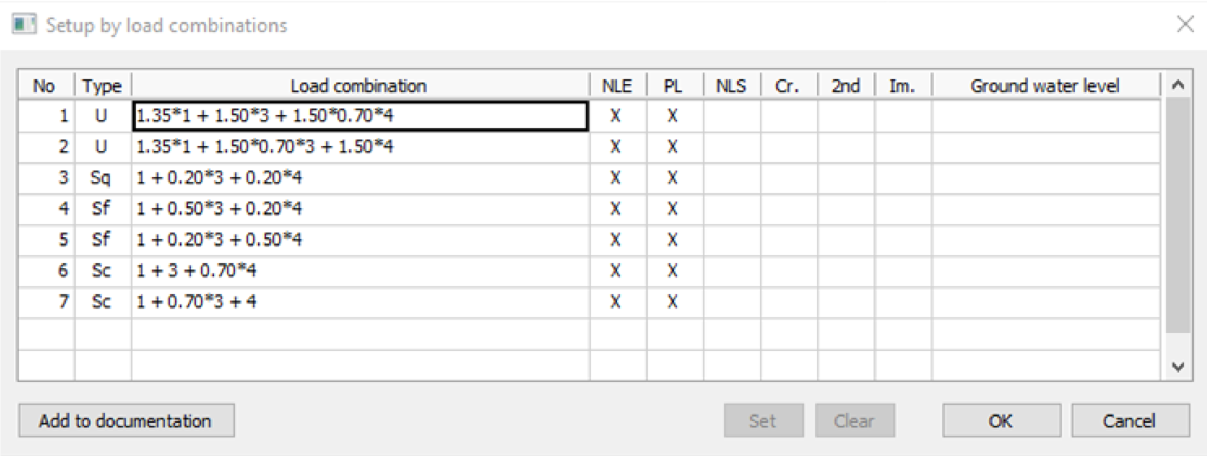
I1	W1 min	W1 max	e2 max	e2 min	i1	S1	S01	c1	Rho 1
[mm4]	[mm3]	[mm3]	[mm]	[mm]	[mm]	[mm3]	[mm3]	[-]	[-]
83561132	557074	557074	150	150	124.6	314179	314179	1.128	0.546
337429603	1499687	1499687	225	225	184.8	850900	850900	1.135	0.519
117669093	713146	713146	165	165	137.1	402166	402165	1.128	0.546
40495	2025	2025	20	20	13.8	1307	1307	1.291	0.230
314755	8993	8993	35	35	26.3	5400	5400	1.201	0.326

z2	alpha2	I2	W2 min	W2 max	e1 max	e1 min	i2	S2	S02
[mm]	[°]	[mm4]	[mm3]	[mm3]	[mm]	[mm]	[mm]	[mm3]	[mm3]
0	1.571	6037784	80504	80504	75	75	33.5	62644	62644
0	1.571	16758611	176406	176406	95	95	41.2	138282	138279
0	1.571	7881421	98518	98518	80	80	35.5	76882	76882
0	1.571	13428	1343	1343	10	10	7.9	798	798
0	1.571	187577	7503	7503	25	25	20.3	4291	4291

c2	Rho 2	z1	Other
[-]	[-]	[mm]	[-]
1.556	0.386	0	tw=7.1mm; hw=279mm; tf=10.7mm; wf=150mm; r=15.0mm
1.568	0.419	0	tw=9.4mm; hw=421mm; tf=14.6mm; wf=190mm; r=21.0mm
1.561	0.388	0	tw=7.5mm; hw=307mm; tf=11.5mm; wf=160mm; r=18.0mm
1.189	0.659	0	t=2.0mm; r=2.00mm
1.144	0.536	0	t=2.0mm; r=2.00mm

1.5. Load combination analysis setup list

In the *Calculation/Setup by load combination* dialog, a new option called *Add to documentation* is available to document the analysis setup for load combinations.

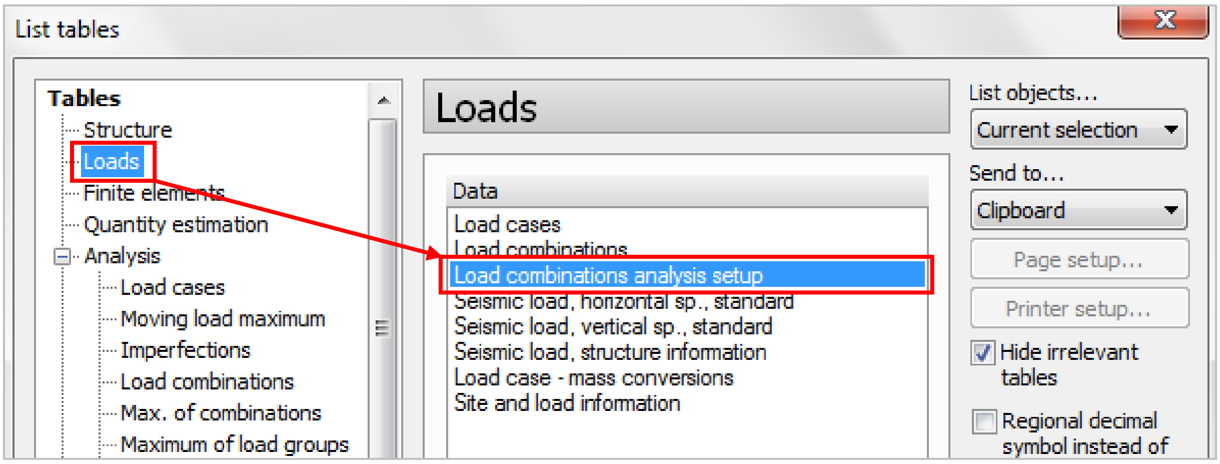


The table in the documentation:

Load combinations analysis setup

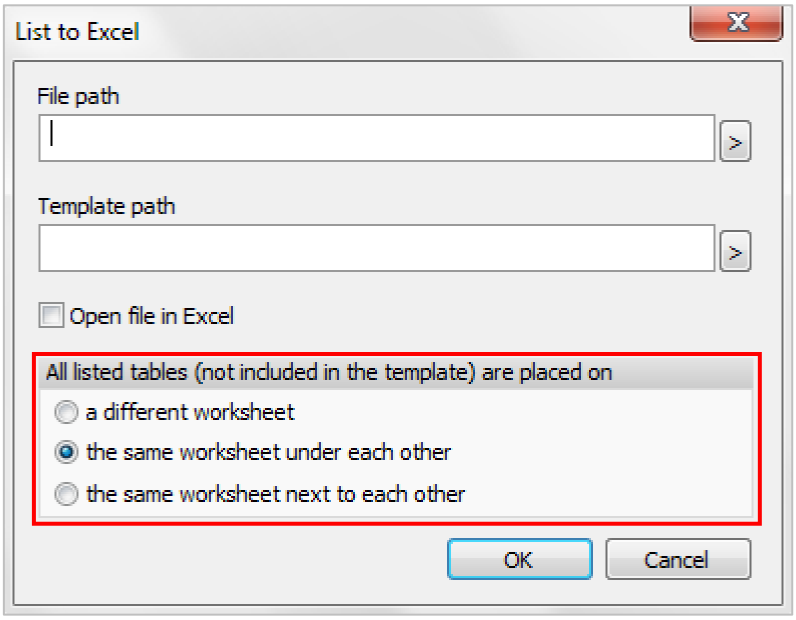
No	Type	Load combination	Non-linear elements	Plastic elements	Non-linear soil	Cracked section	2nd order	Imperfection shape	Ground water level
1	U	1.35*1 + 1.50*3 + 1....	Yes	Yes	No	No	No	No	-
2	U	...	Yes	Yes	No	No	No	No	-
3	Sq	1 + 0.20*3 + 0.20*4	Yes	Yes	No	No	No	No	-
4	Sf	1 + 0.50*3 + 0.20*4	Yes	Yes	No	No	No	No	-
5	Sf	1 + 0.20*3 + 0.50*4	Yes	Yes	No	No	No	No	-
6	Sc	1 + 3 + 0.70*4	Yes	Yes	No	No	No	No	-
7	Sc	1 + 0.70*3 + 4	Yes	Yes	No	No	No	No	-

This option can be reached from the *List tables* dialog, too. Pick *Tables/Loads/Load combinations*.



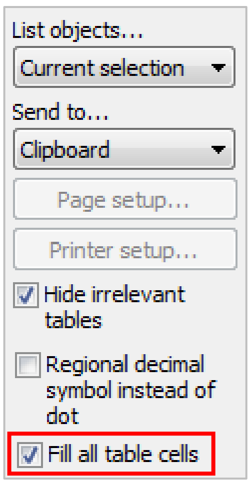
1.6. Arranging tables when listing to Excel

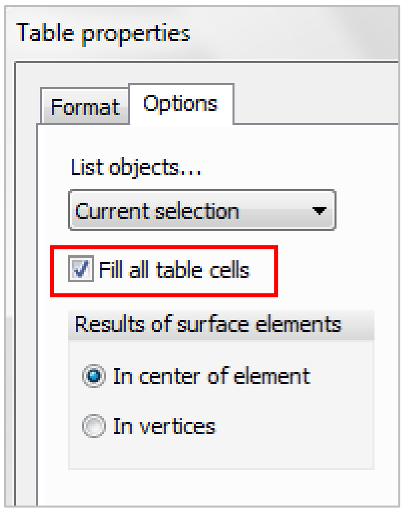
New options have been implemented into the feature *List to Excel*. From now on User can choose, if the tables should be placed into different Excel-worksheets or into the same worksheet under or next to each other.



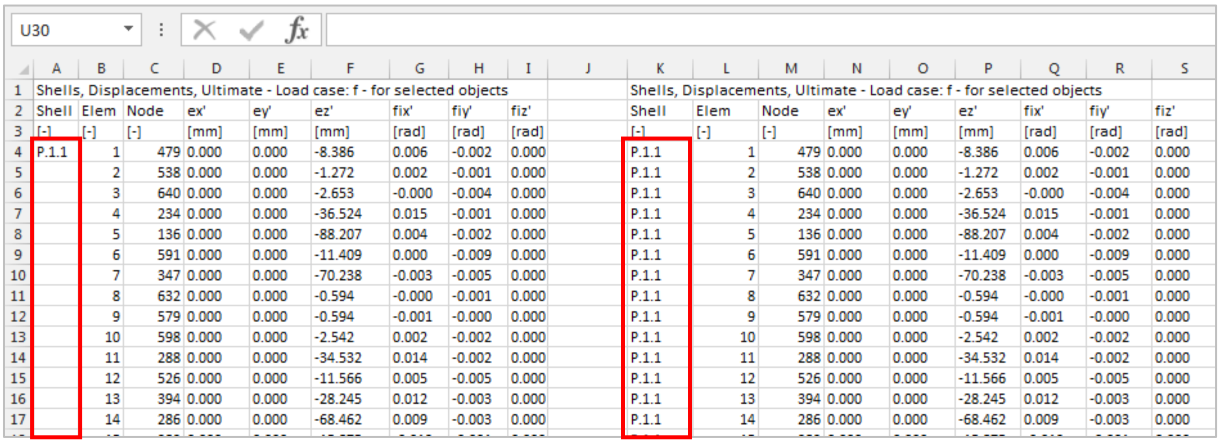
1.7. "Fill all cells" option for listing tables

In order to ease sorting data in Excel after exporting tables, User can fill the empty cells by turning on the "*Fill all table cells*" option. This option is also available in the Documentation, in the *Table properties* dialog in the *Option* tab.





The left part of table on figure below shows the result if the checkbox is off, and right part shows when it is on.

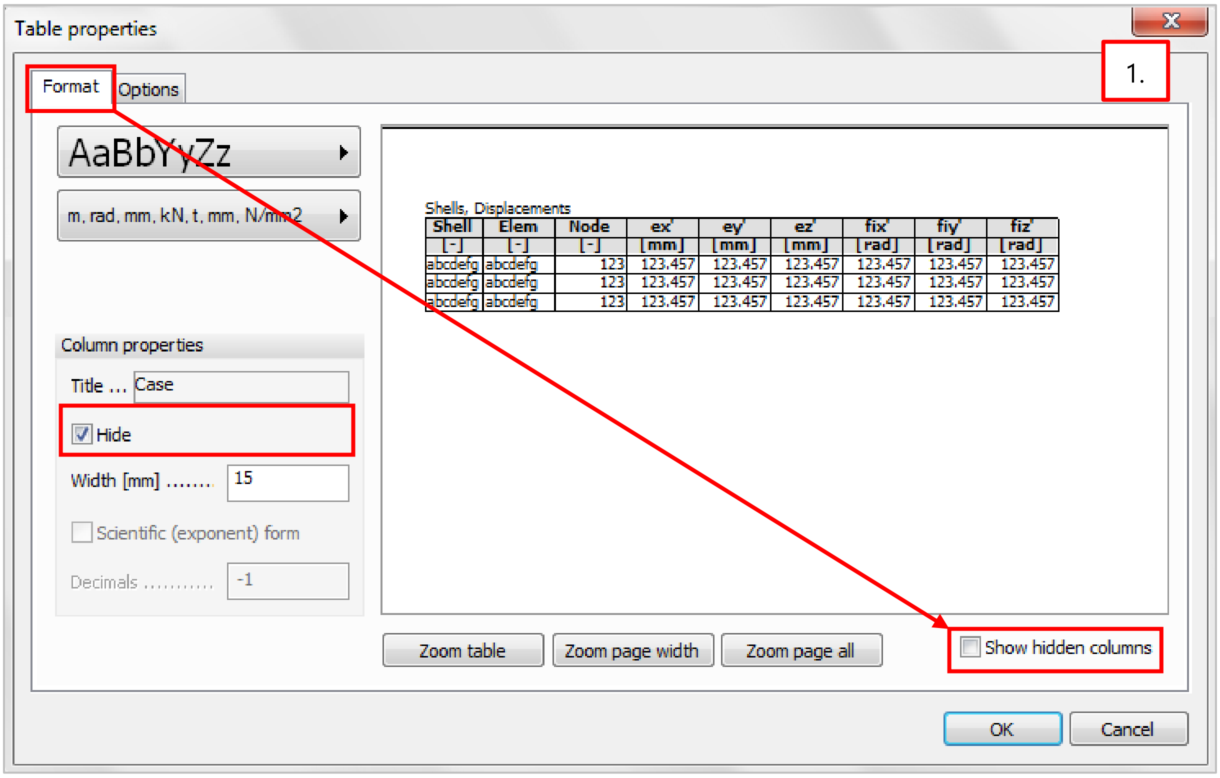


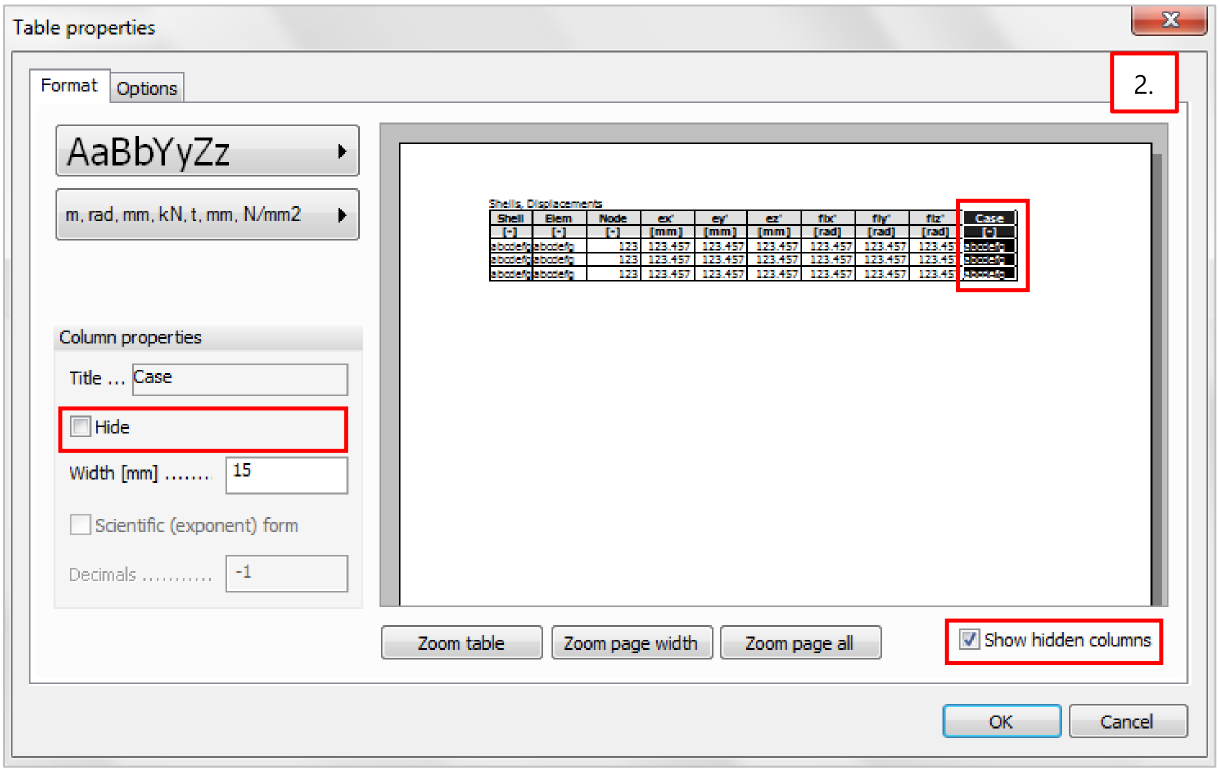
	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S
1	Shells, Displacements, Ultimate - Load case: f - for selected objects										Shells, Displacements, Ultimate - Load case: f - for selected objects								
2	Shell	Elem	Node	ex'	ey'	ez'	fix'	fiy'	fiz'		Shell	Elem	Node	ex'	ey'	ez'	fix'	fiy'	fiz'
3	[-]	[-]	[-]	[mm]	[mm]	[mm]	[rad]	[rad]	[rad]		[-]	[-]	[-]	[mm]	[mm]	[mm]	[rad]	[rad]	[rad]
4	P.1.1	1	479	0.000	0.000	-8.386	0.006	-0.002	0.000		P.1.1	1	479	0.000	0.000	-8.386	0.006	-0.002	0.000
5		2	538	0.000	0.000	-1.272	0.002	-0.001	0.000		P.1.1	2	538	0.000	0.000	-1.272	0.002	-0.001	0.000
6		3	640	0.000	0.000	-2.653	-0.000	-0.004	0.000		P.1.1	3	640	0.000	0.000	-2.653	-0.000	-0.004	0.000
7		4	234	0.000	0.000	-36.524	0.015	-0.001	0.000		P.1.1	4	234	0.000	0.000	-36.524	0.015	-0.001	0.000
8		5	136	0.000	0.000	-88.207	0.004	-0.002	0.000		P.1.1	5	136	0.000	0.000	-88.207	0.004	-0.002	0.000
9		6	591	0.000	0.000	-11.409	0.000	-0.009	0.000		P.1.1	6	591	0.000	0.000	-11.409	0.000	-0.009	0.000
10		7	347	0.000	0.000	-70.238	-0.003	-0.005	0.000		P.1.1	7	347	0.000	0.000	-70.238	-0.003	-0.005	0.000
11		8	632	0.000	0.000	-0.594	-0.000	-0.001	0.000		P.1.1	8	632	0.000	0.000	-0.594	-0.000	-0.001	0.000
12		9	579	0.000	0.000	-0.594	-0.001	-0.000	0.000		P.1.1	9	579	0.000	0.000	-0.594	-0.001	-0.000	0.000
13		10	598	0.000	0.000	-2.542	0.002	-0.002	0.000		P.1.1	10	598	0.000	0.000	-2.542	0.002	-0.002	0.000
14		11	288	0.000	0.000	-34.532	0.014	-0.002	0.000		P.1.1	11	288	0.000	0.000	-34.532	0.014	-0.002	0.000
15		12	526	0.000	0.000	-11.566	0.005	-0.005	0.000		P.1.1	12	526	0.000	0.000	-11.566	0.005	-0.005	0.000
16		13	394	0.000	0.000	-28.245	0.012	-0.003	0.000		P.1.1	13	394	0.000	0.000	-28.245	0.012	-0.003	0.000
17		14	286	0.000	0.000	-68.462	0.009	-0.003	0.000		P.1.1	14	286	0.000	0.000	-68.462	0.009	-0.003	0.000

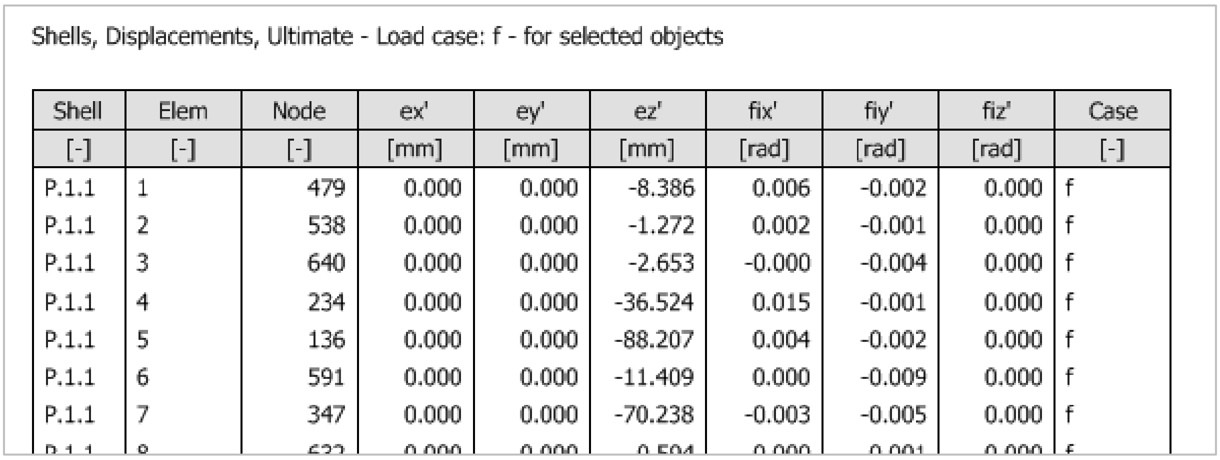
1.8. Displaying name of load case/combination for load case/combination result tables

In FD 17 the name of the load combinations and load cases can be displayed for the load case and load combination result tables.

Check *Show hidden columns* checkbox in *Table properties* dialog:







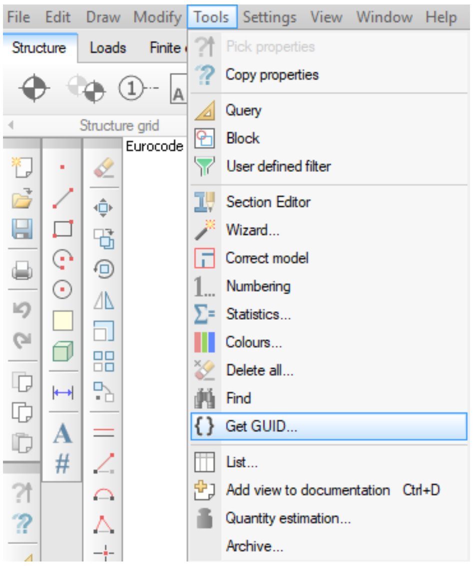
Shells, Displacements, Ultimate - Load case: f - for selected objects

Shell	Elem	Node	ex'	ey'	ez'	fix'	fiy'	fiz'	Case
[-]	[-]	[-]	[mm]	[mm]	[mm]	[rad]	[rad]	[rad]	[-]
P.1.1	1	479	0.000	0.000	-8.386	0.006	-0.002	0.000	f
P.1.1	2	538	0.000	0.000	-1.272	0.002	-0.001	0.000	f
P.1.1	3	640	0.000	0.000	-2.653	-0.000	-0.004	0.000	f
P.1.1	4	234	0.000	0.000	-36.524	0.015	-0.001	0.000	f
P.1.1	5	136	0.000	0.000	-88.207	0.004	-0.002	0.000	f
P.1.1	6	591	0.000	0.000	-11.409	0.000	-0.009	0.000	f
P.1.1	7	347	0.000	0.000	-70.238	-0.003	-0.005	0.000	f

1.9. Get GUID (Globally Unique Identifier)

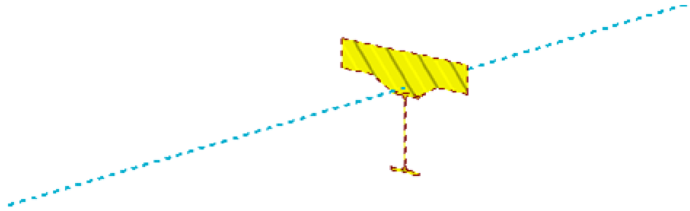
The *Get GUID* function enables the query of the GUID of elements. This can be useful for identification of structural objects imported/exported via Struxml.

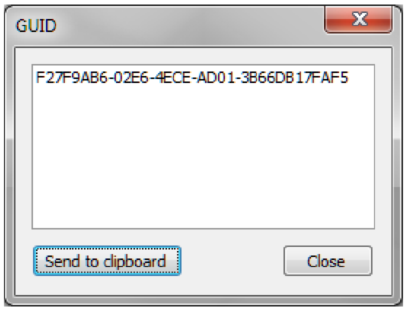
The function-button can be found in the *Tools* menu, but it is not on the *Toolbar* ({}) by default.User can put it there by using *Customize...* command which is available by right clicking on the toolbar.



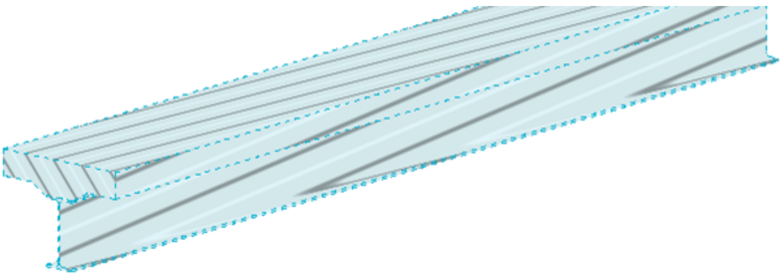
After launching *Get GUID*, pick the element(s). The pop-up window shows the Globally Unique Identifiers of selected element(s). They can be sent to the clipboard by clicking on "Send to clipboard" button. GUIDs for analytical and physical model of a bar are different.

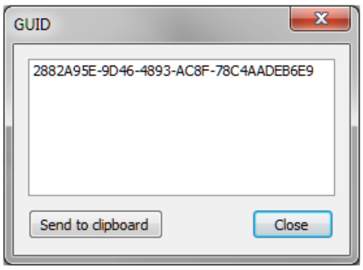
Analytical view:





Physical view:





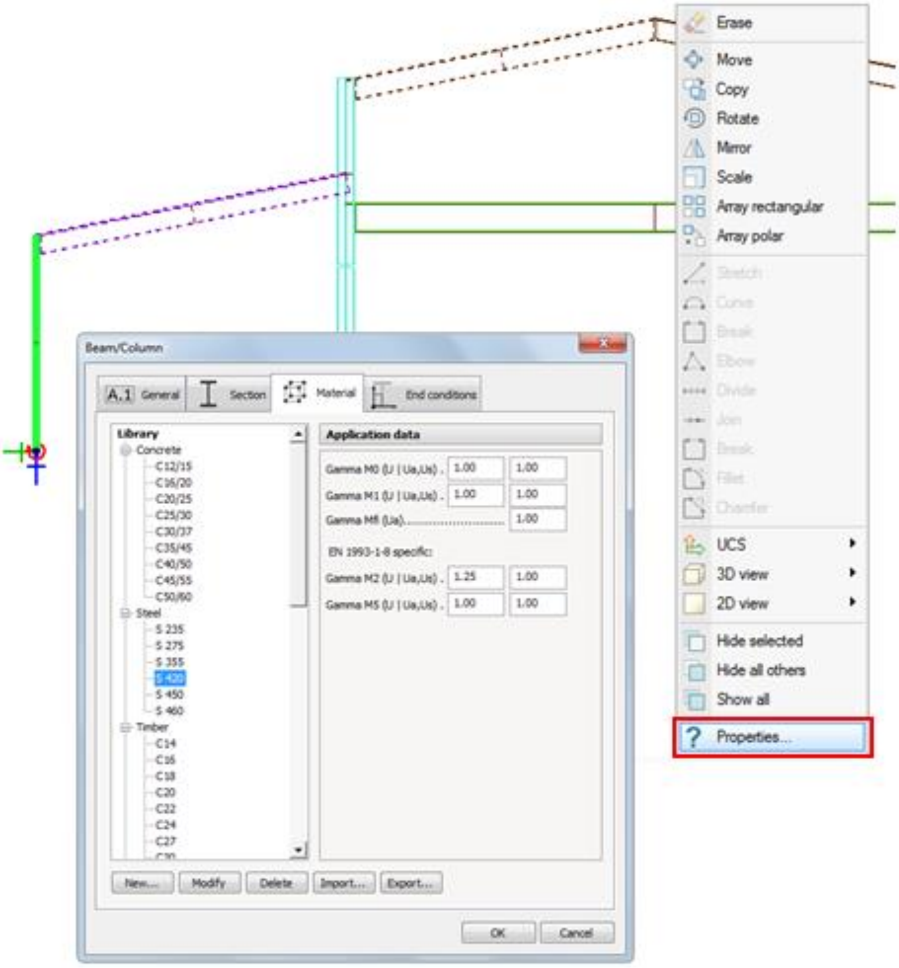
If the bar is saved in struxml format, these GUIDs can be found there as well.

```
<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>
```

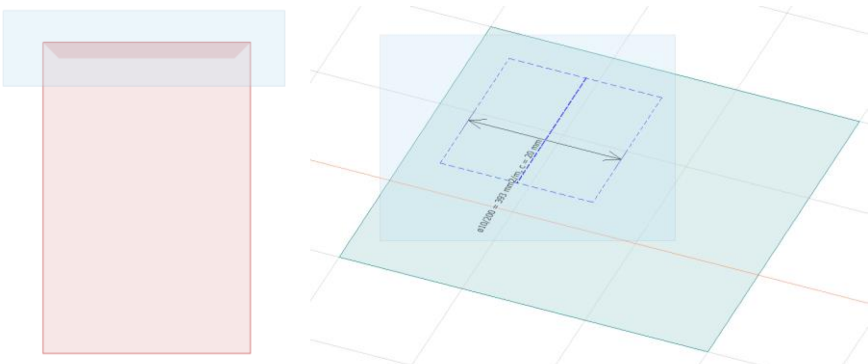
2. User interface

2.1. Properties dialog available from Quick menu for objects

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

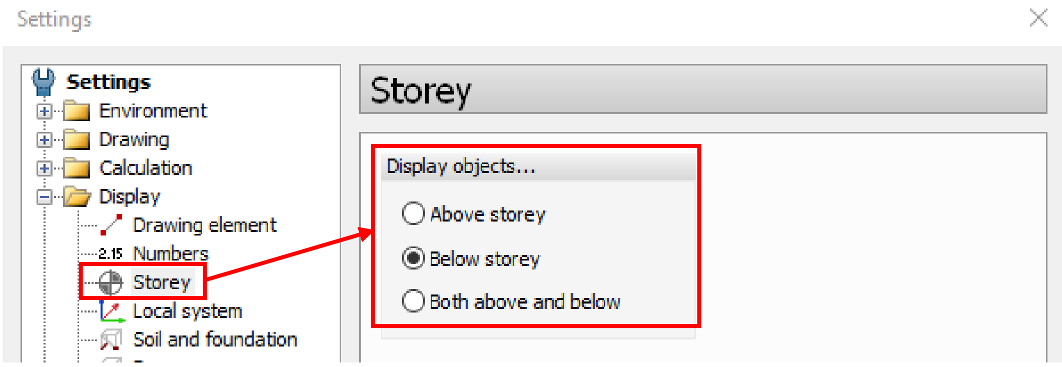


Objects that are part of other object (e.g. edge connections, corbels, reinforcement regions, etc.)usually can be selected by left-to-right (blue) box selection only, because right clicking always selects the main object (shell, bar).



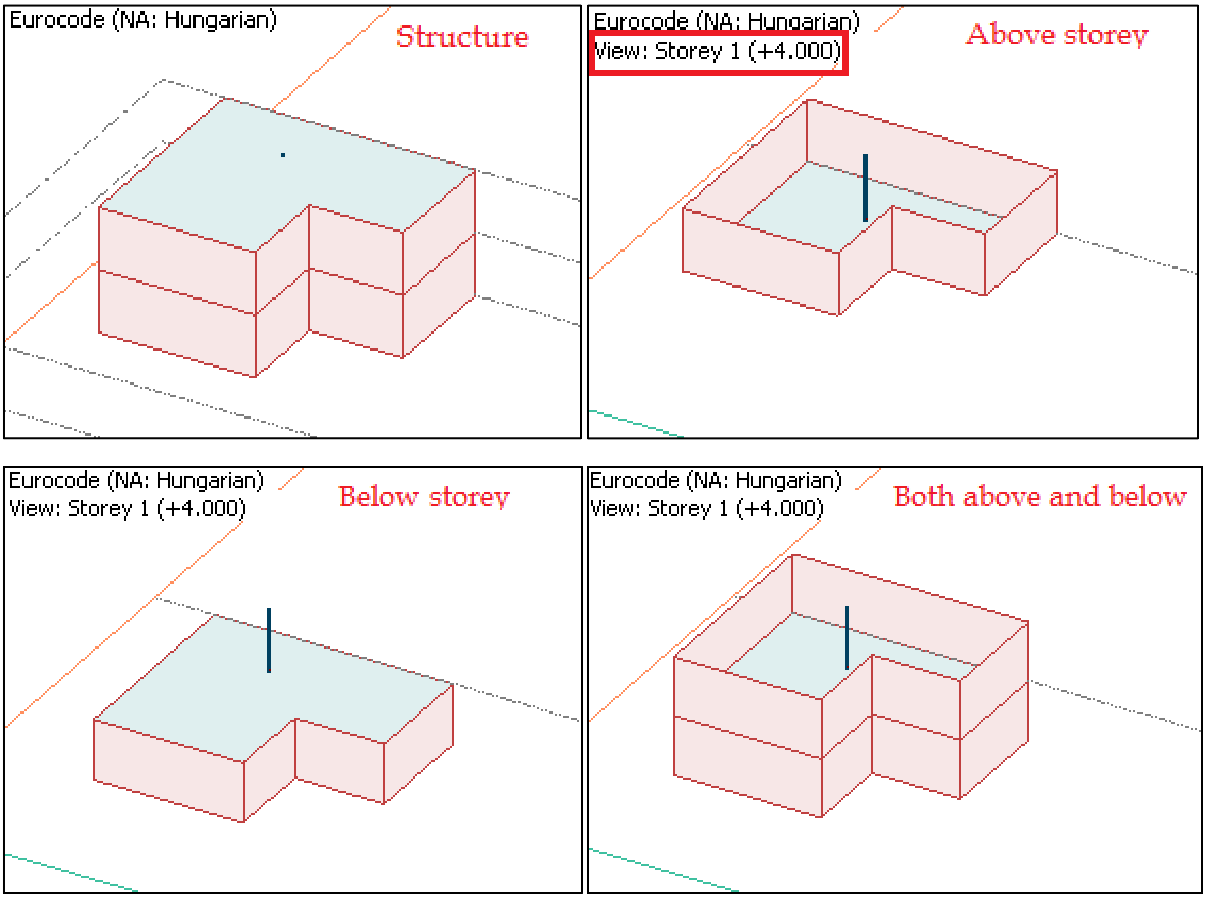
2.2. Different modes to display storeys

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



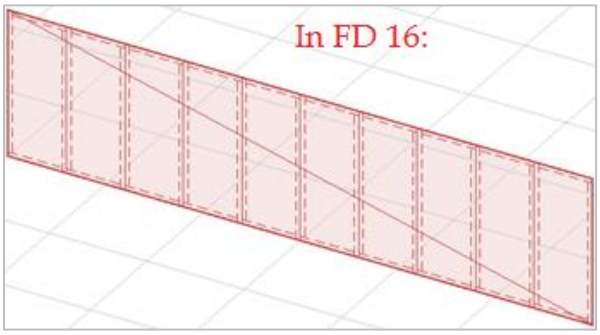
All objects that are in the plane of the storey, or crossing the storey above or below according to the selected setting, are displayed

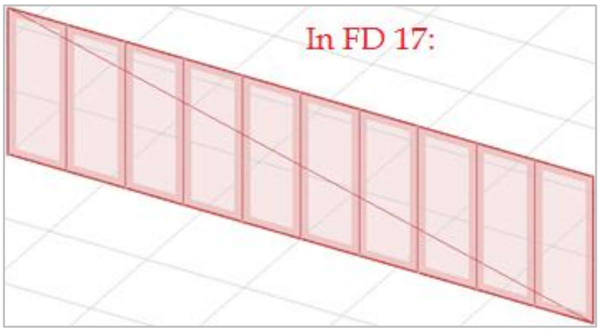
The pictures below show the whole structure and how it is displayed according to the selected option.



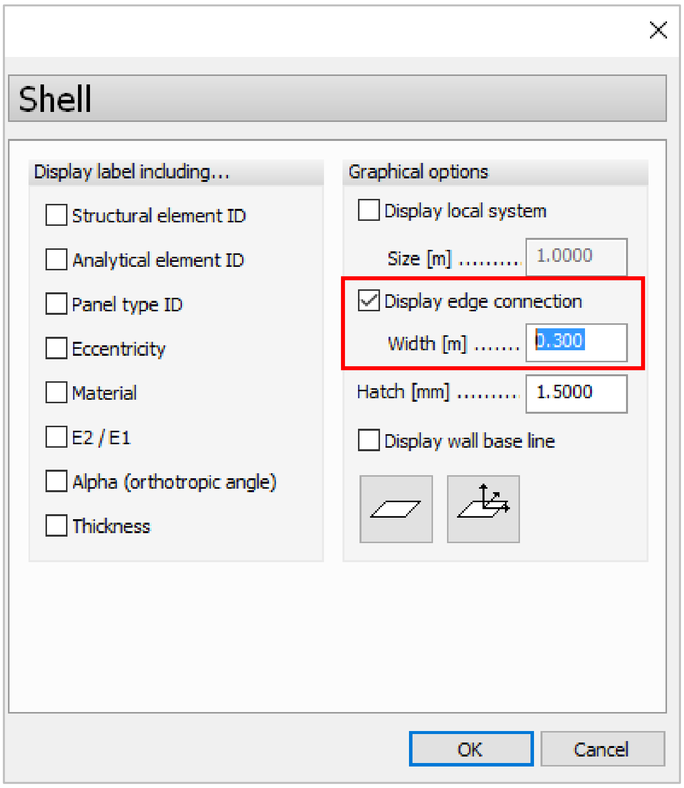
2.3. Visualisation of edge connection is improved

In the new version, *Edge connections* are displayed with customizable lanes.





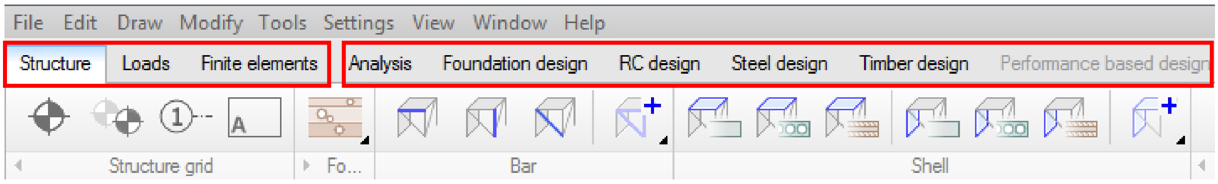
The width of the edge connection can be modified in the *Settings/Display/Shells* dialog.



2.4. Restore view returning to input

Input tabs

Analysis and design tabs

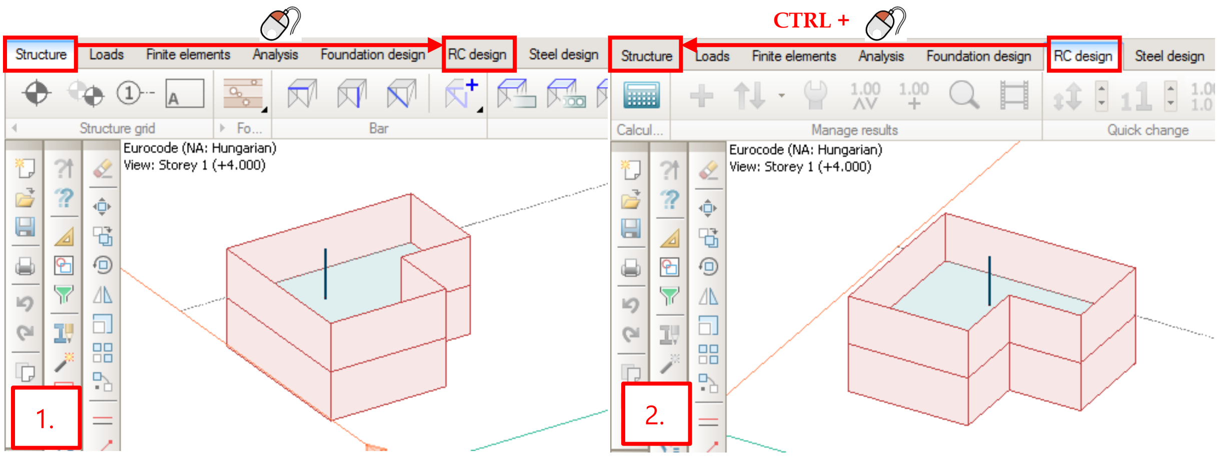


Once a view set in an input tab gets modified in the Analysis or in a design tab, User can return into an input tab while automatically restoring the original view. See an example in the following figures.

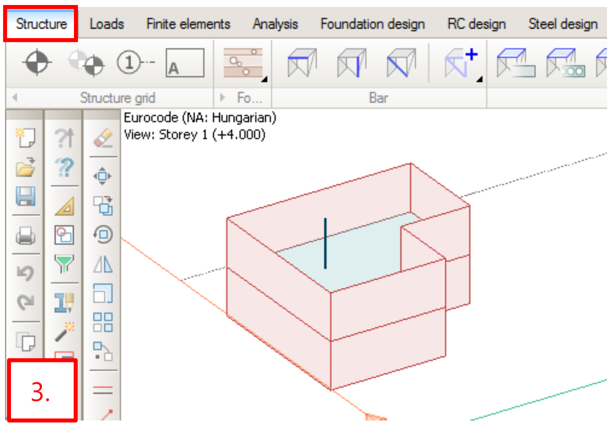
Using Ctrl:

Original view

New view in RC design



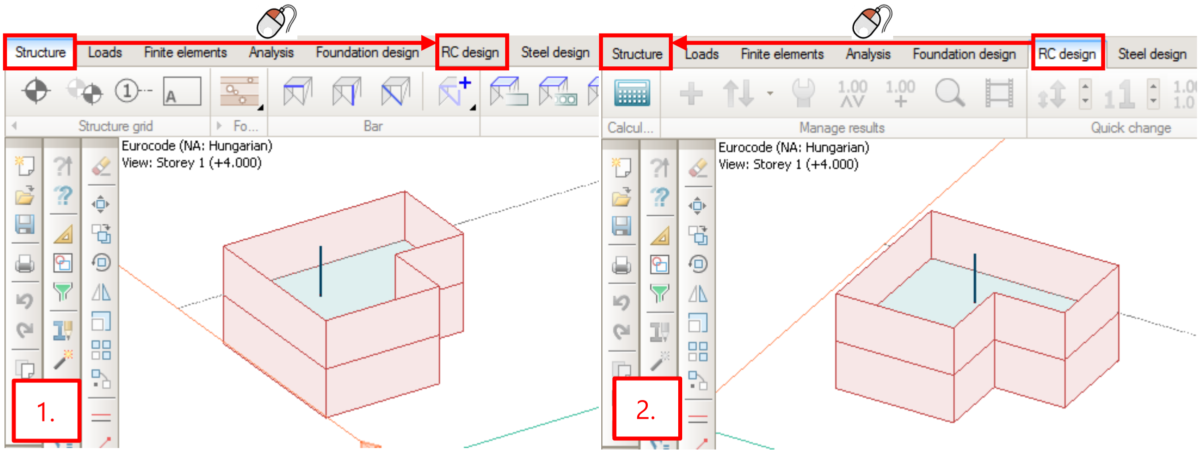
Back to original view



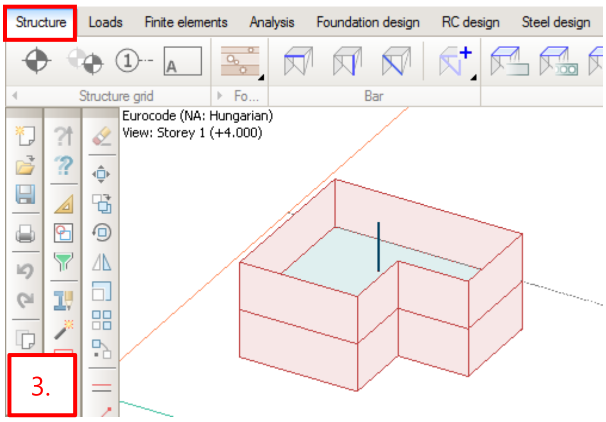
Without pressing Ctrl the view doesn't switch back:

Original view

New view in RC design

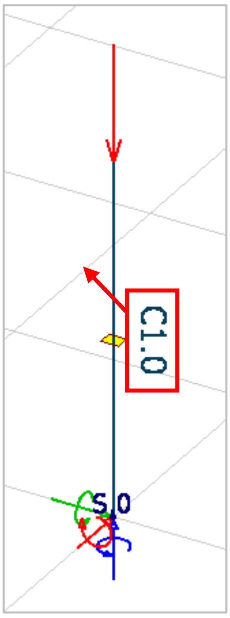


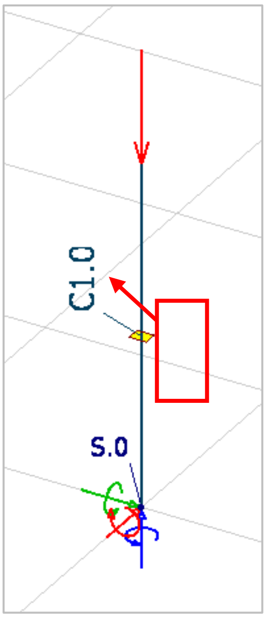
Back to original view



2.5. Leading line for numeric values and labels

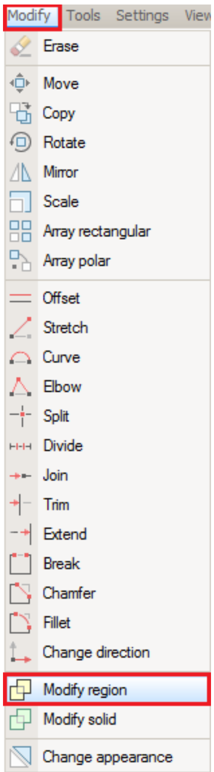
If a numeric value or a label is moved away from the referred point, a thin leading line appears.

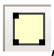

C1.0
S.0

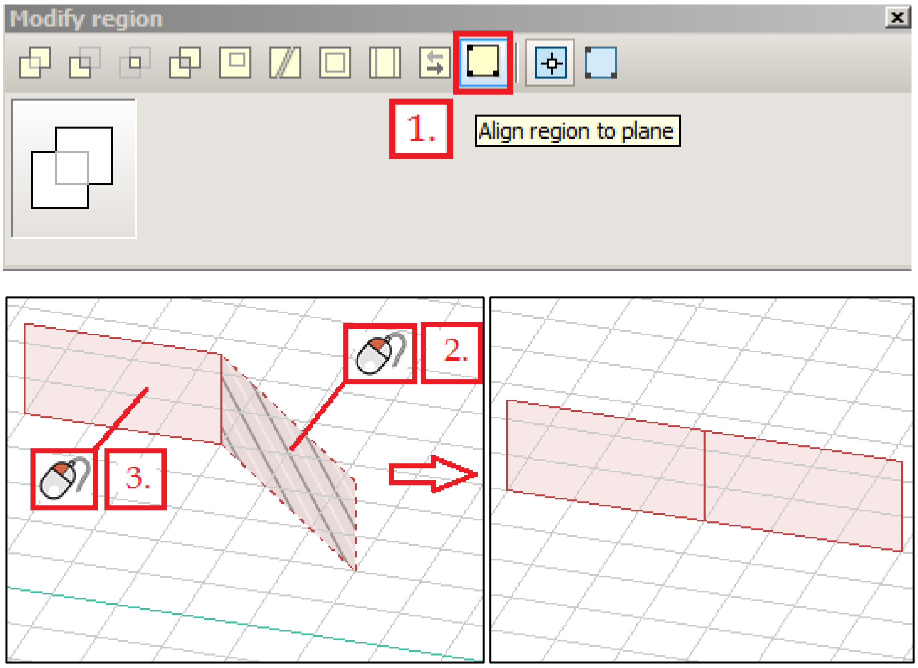

C1.0
S.0

2.6. Align region to plane

This function will align regions to selected plane in any direction with two clicks. The feature can be found in *Modify/Modify region*.



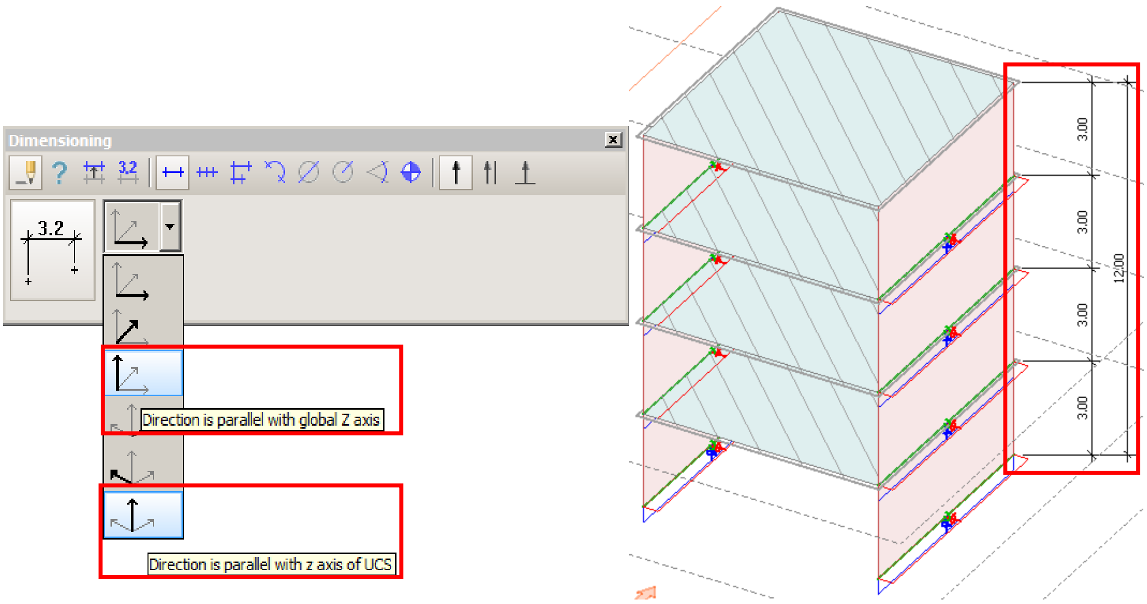
Click the icon , select regions to be aligned, and then select the reference plane. All selected object will be projected to the reference plane.



2.7. Vertical dimension lines

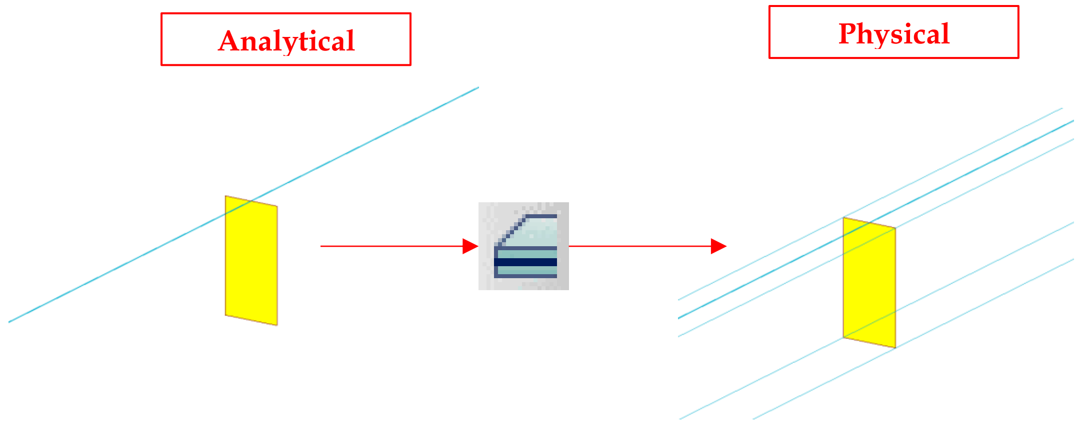
From now vertical dimension lines can be placed without modifying the user coordinate-system.

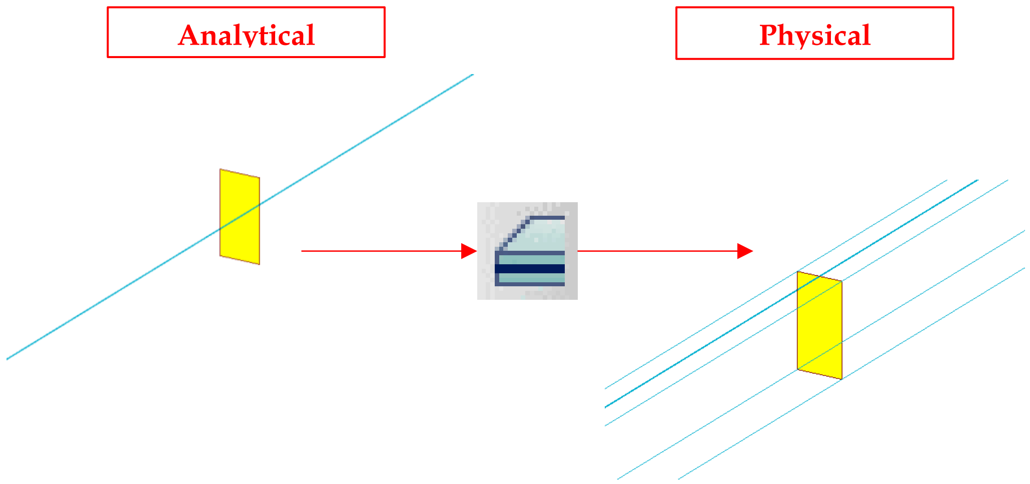
In the *Dimensioning dialog* select the parallel direction with global Z axis or with z axis of UCS.

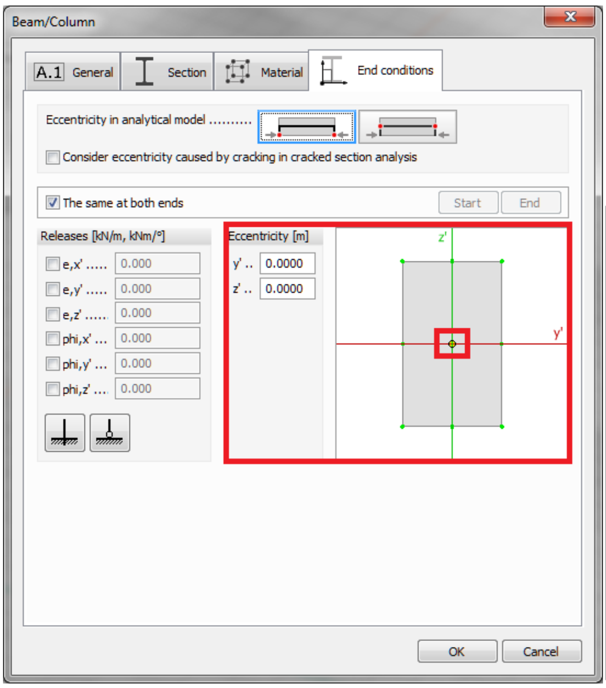


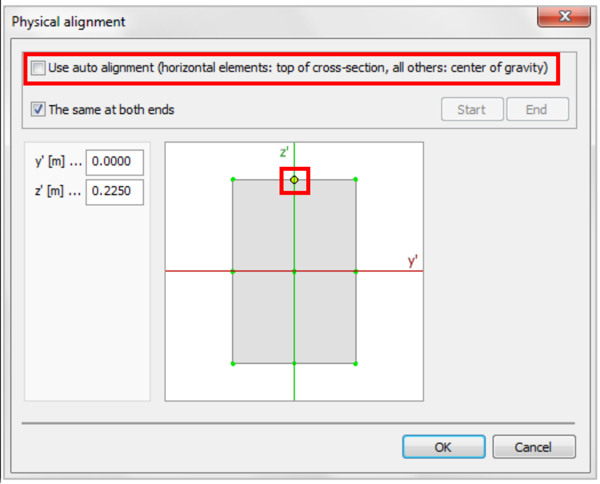
2.8. Physical view

In FEM-Design 17 the analytical and physical view of an object is clearly separated. The physical view shows the real position of the physical element, while the analytical view shows the calculation model.



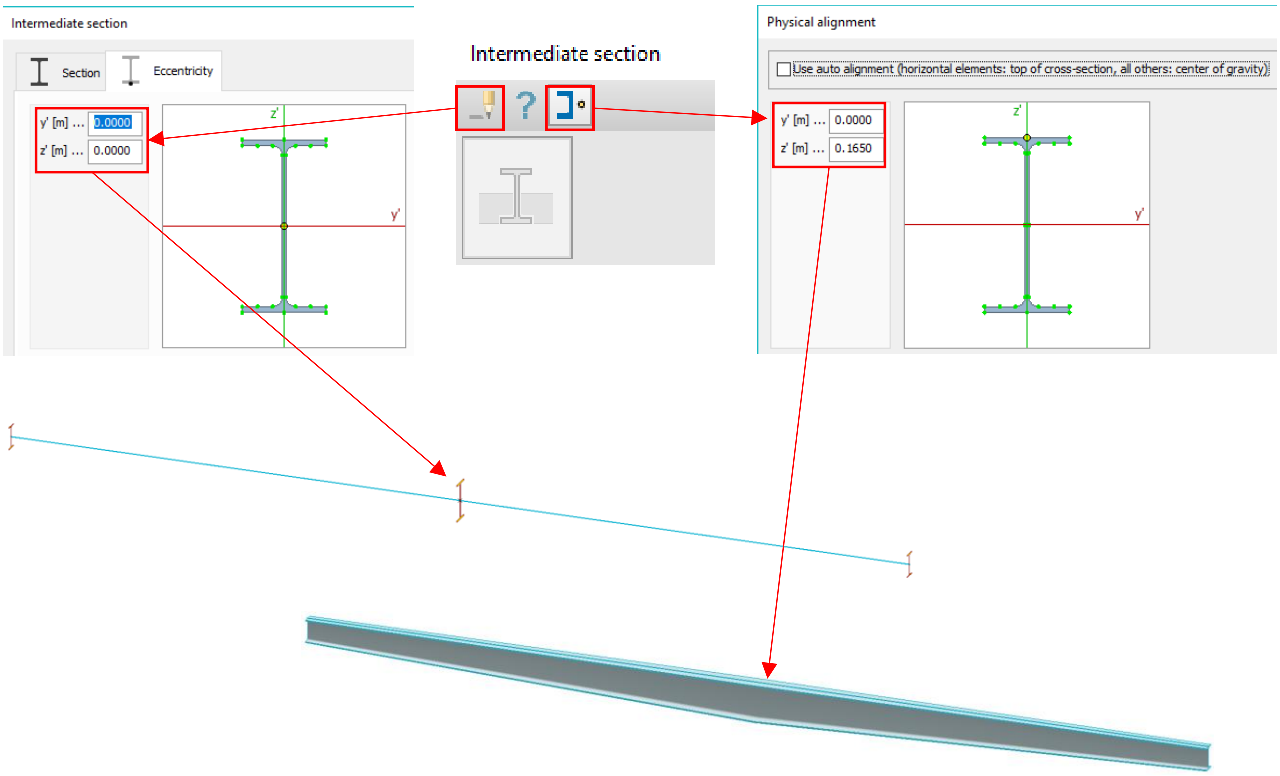






2.9. Physical alignment for intermediate section

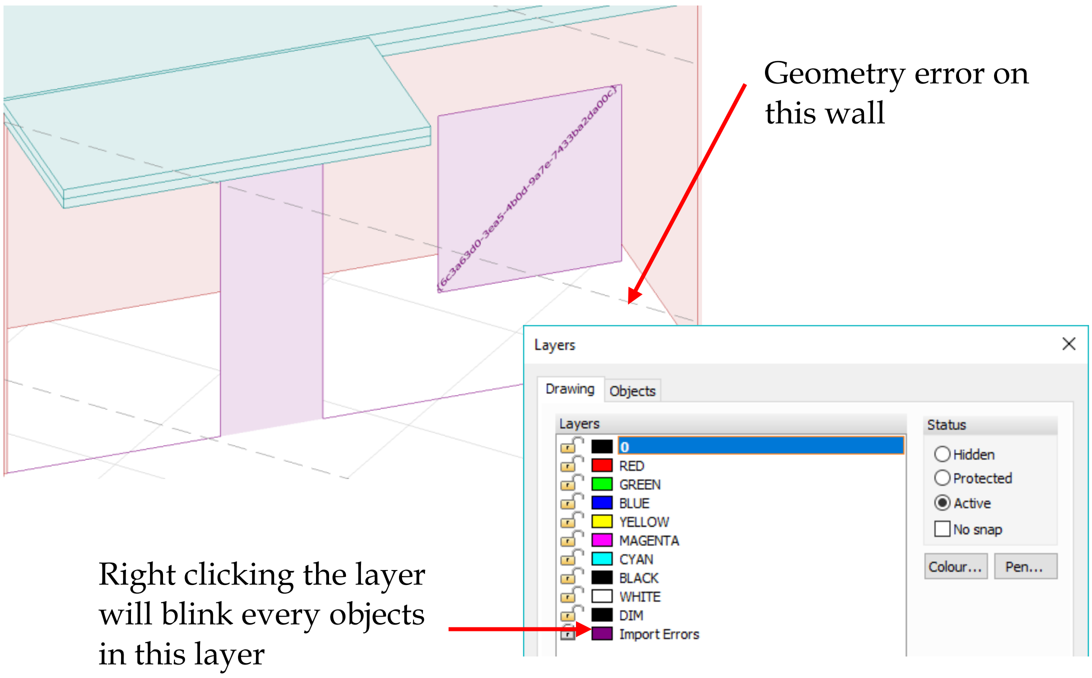
In version 17 the User has the option to set the physical and analytical eccentricity separately for intermediate section.



2.10. Blinking layers

In FEM-Design 17, upon right clicking on any Layer in the *Layers* dialog, all elements belonging to that Layer will blink. This works both for Drawing and Objects layers.

Additionally, a new Layer called Import error is added into Drawing layers. This is relevant only in case when the model was imported through struxml and contained some bad shell objects. In case those shell objects could not be imported, they will be shown (in some cases) as graphical image and placed on above mentioned layer.


Geometry error on this wall
Right clicking the layer will blink every objects in this layer
Layers
Drawing
Objects
Layers
0
RED
GREEN
BLUE
YELLOW
MAGENTA
CYAN
BLACK
WHITE
DIM
Import Errors
Status
Hidden
Protected
Active
No snap
Colour…
Pen…

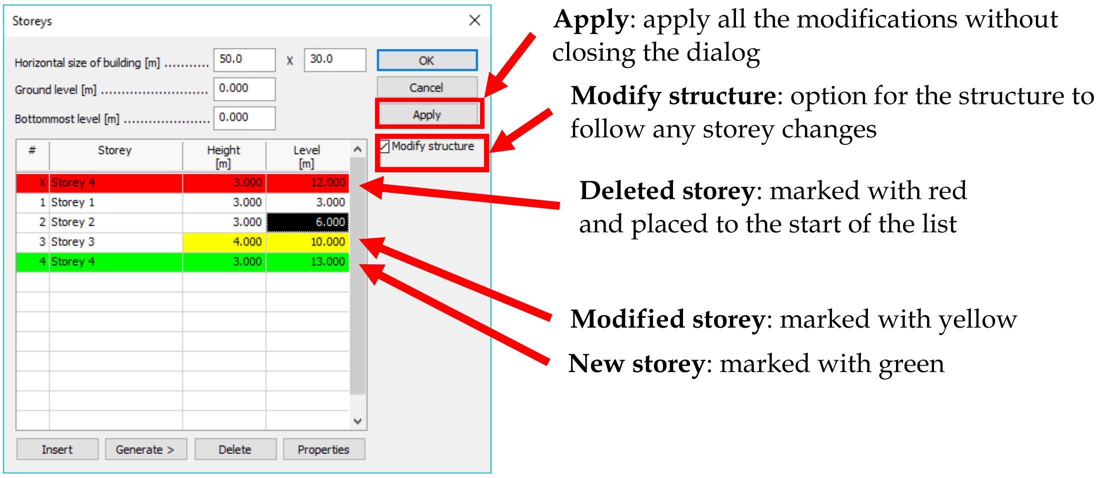
3. Structure

3.1. Improved storey dialog

If the *Modify structure* is enabled the structure follows the storey system modification, otherwise not. The modifications are displayed with three color code, if the *Modify structure* is active:

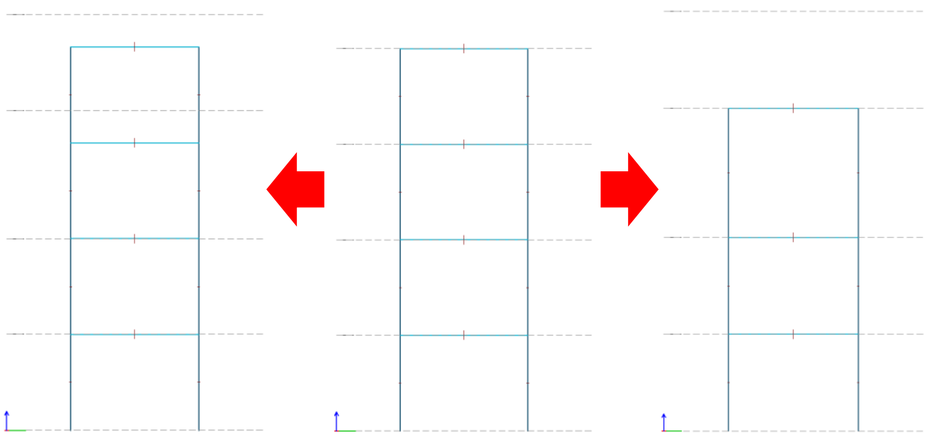
- green: new storey
- yellow: modified storey
- red: deleted storey

When the *Apply* button is clicked the modifications proceed immediately without closing the dialog. The following pictures shows this feature though a 4-storey building, where the *Storey 4* will be **deleted**, a **new storey is added** and the *Storey 3* **height is increased** to 4.0 meters.



„Modify structure" disabled

„Modify structure" enabled



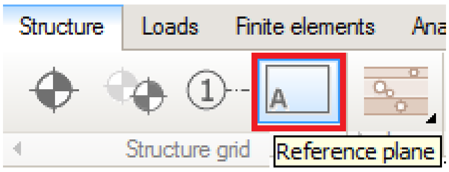
Modified storey system, original structure

Original storey system, original structure

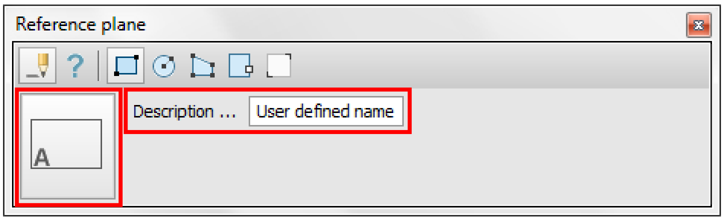
Modified storey system and original

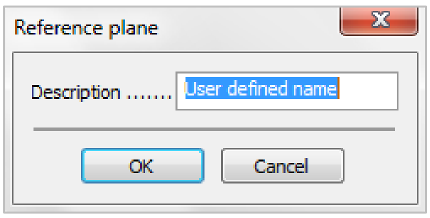
3.2. Reference plane

Reference plane is an auxiliary object, which allows User to align elements to it. It can be found in the Structure tab.

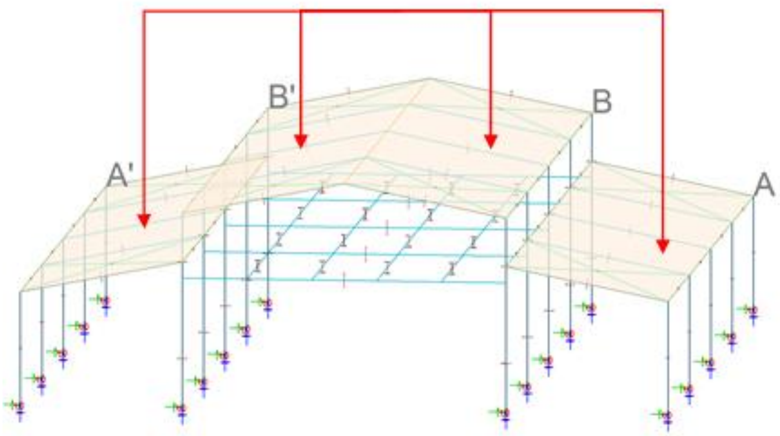


As is it shown below, *Description* of *Reference plane* can be typed in the tool palette or after clicking on the Default properties button, it can be set in the new dialog.





Reference plane can be drawn in any plane. It is displayed by the region contours and its description given by the user.

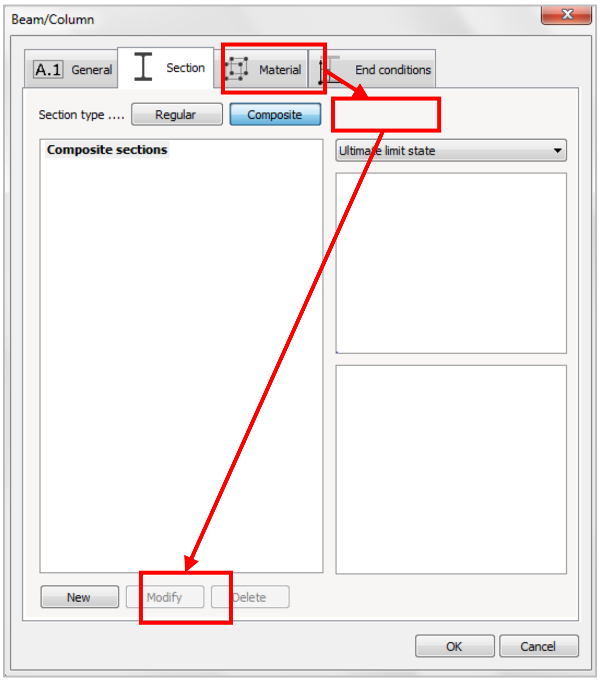


The *Reference plane* can be used with the feature *Align region to plane*. (Modify/Modify region/Align region to plane/Pick region to align/Select *Reference plane* - See chapter 2.6) and it also can be used by *Correct model* tool to fit structural objects to.

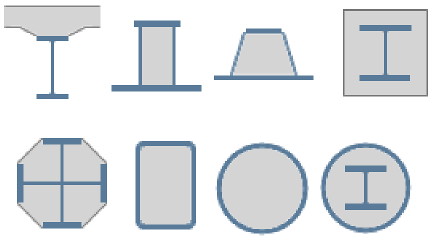
Reference planes can be exported into/imported from Autodesk Revit via struxml format.

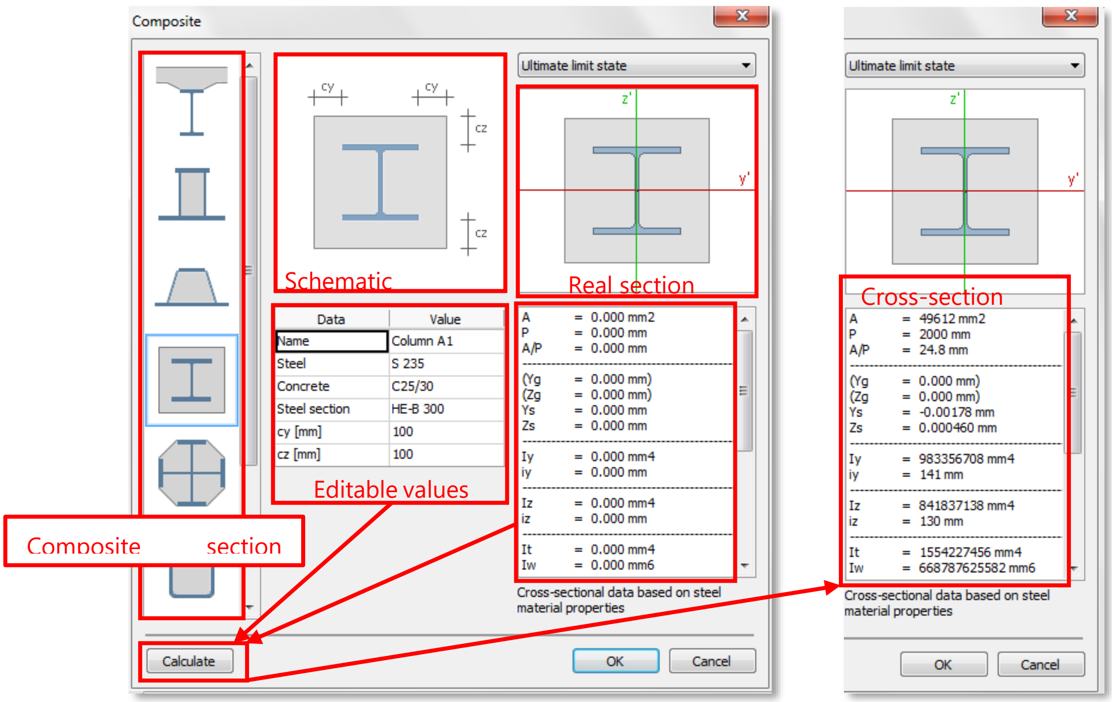
3.3. Composite sections

The composite sections below have been implemented for *Beams* and *Columns*. These sections can be reached in *Beam/Column dialog/Section tab/Composite/New*.



Available composite section types:





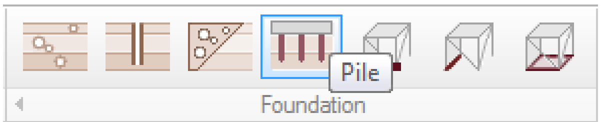
⚠ For the time being only above listed composite sections can be selected, there is no choice to create alternative section.

3.4. Pile

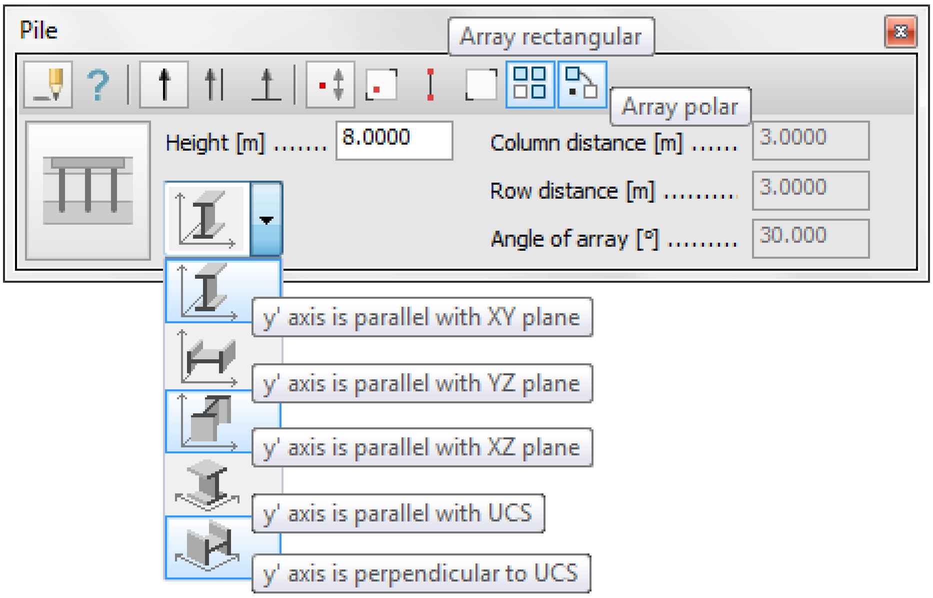
In FEM-Design 17, a new structural element is added - the pile. Its main purpose is to calculate pile internal forces in either linear or non-linear way (e.g. limited shaft friction, only compression behaviour at the tip of pile can be considered).

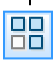
The basic concept is that the soil surrounding the pile is modelled with continuous line supports along it. These supports stand for the supporting effect of the soil together with a vertical point support at the tip of pile. The new plastic element feature offers a more precise way to calculate the internal forces by considering the nonlinear characteristic of the soil-structure interaction.

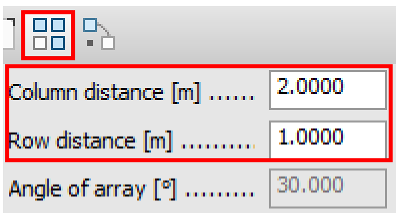
This feature can be found among the *Foundation objects* on *Structure tab*.

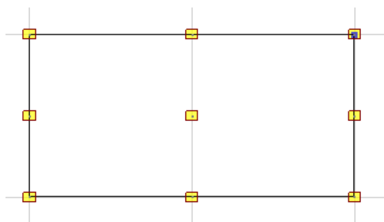


Piles can be placed either one by one, or in groups using the "Array rectangular"/ "Array polar" options. In the Tool window User can declare the *Height* of the pile and set the array options as *Column distance, Row distance* and *Angle of array*.

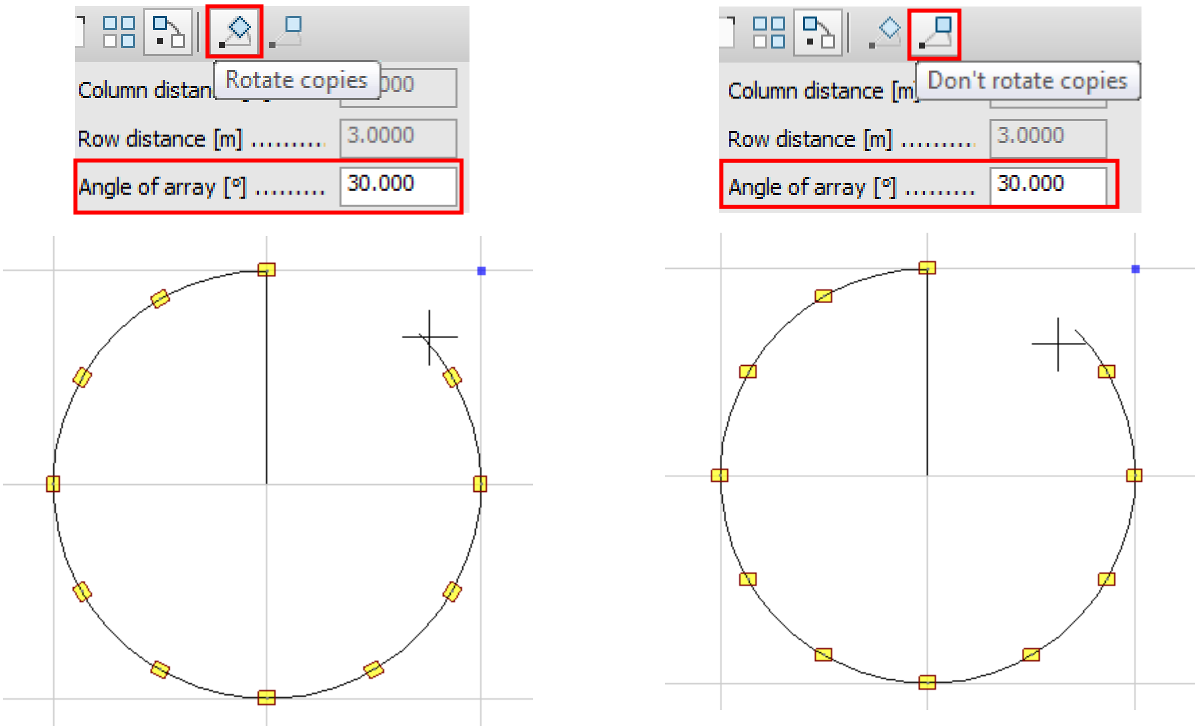


If choosing *Array rectangular* define option, at first User needs to set the *Column* and *Row distances*, before placing the piles. By clicking  these textboxes turn active.

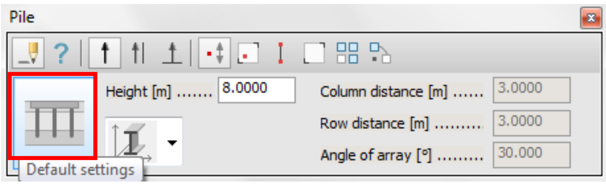




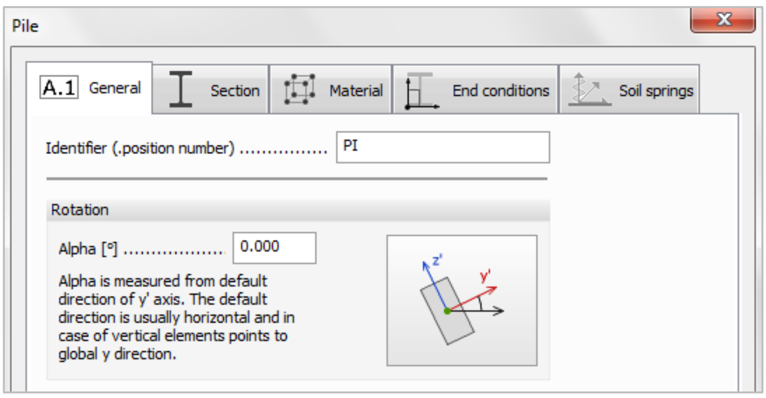
If choosing *Array polar*, two more options become available according to the following figure:



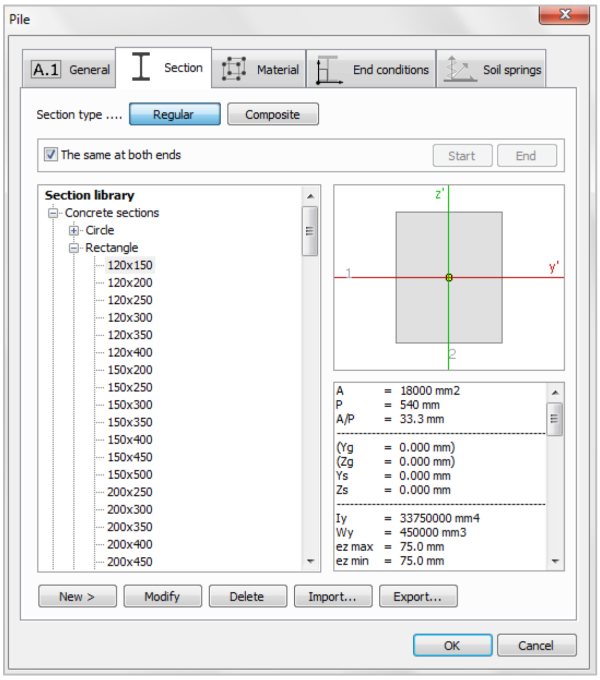
To open *Pile dialog* click *Default settings.*

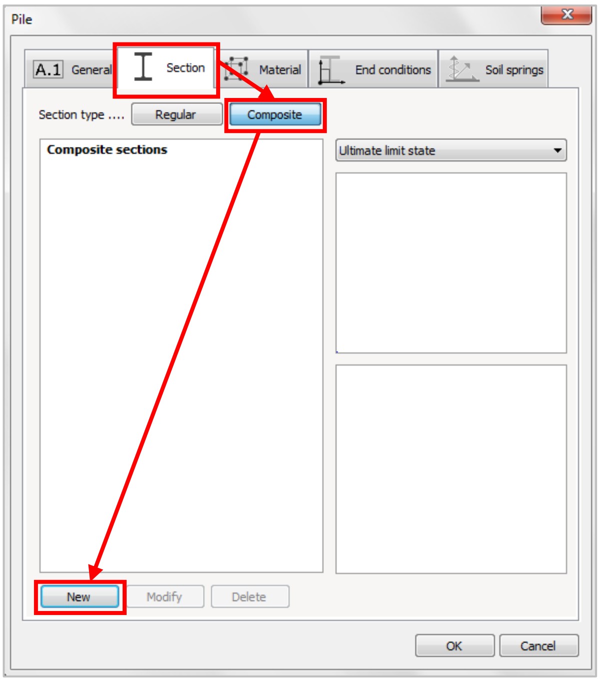


On *General tab* User can give an identifier and a rotation angle.

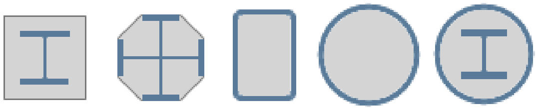


Regular and certain types of composite sections can be selected in *Section tab/Composite/ New*.

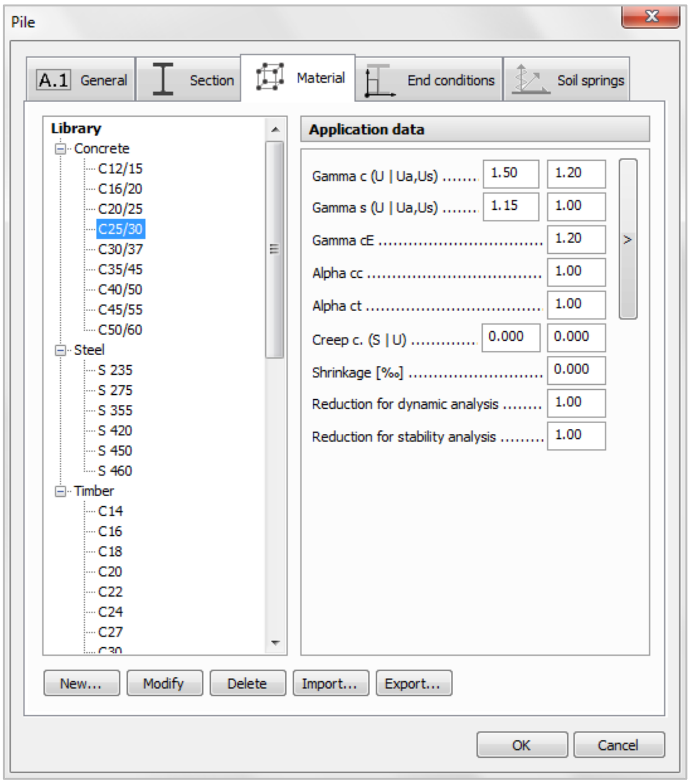




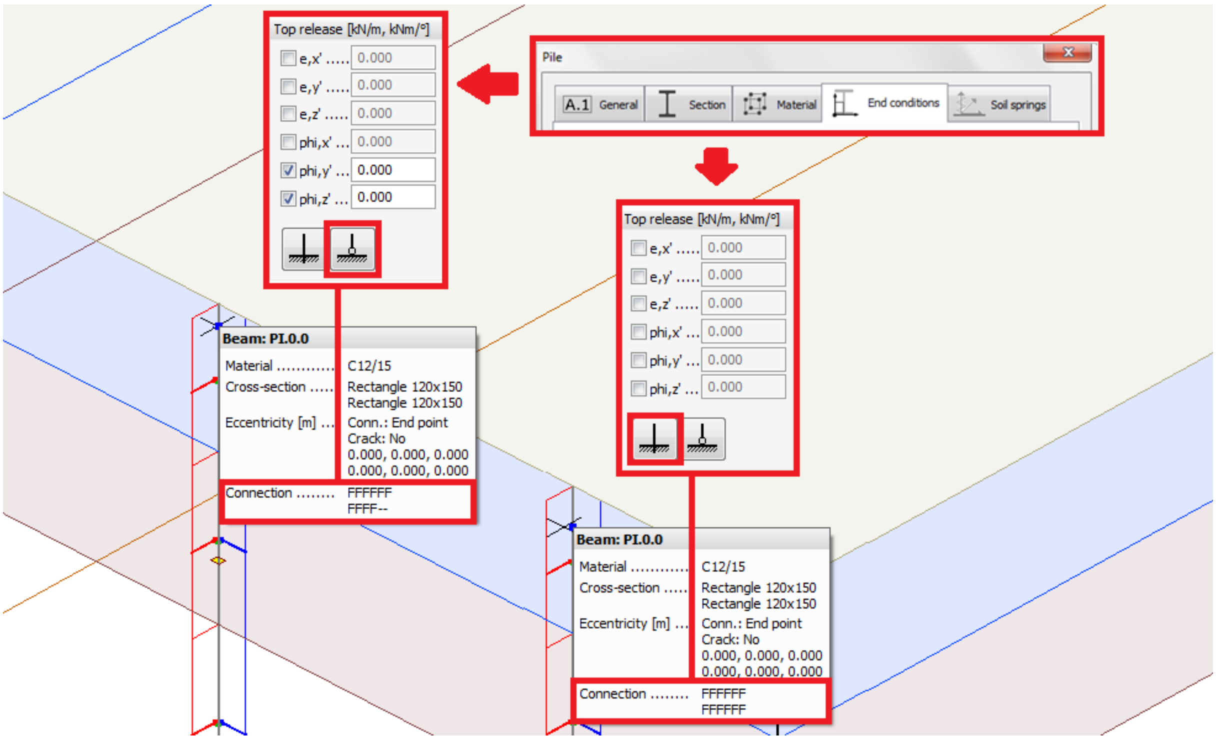
The available composite sections for simple pile are the followings:



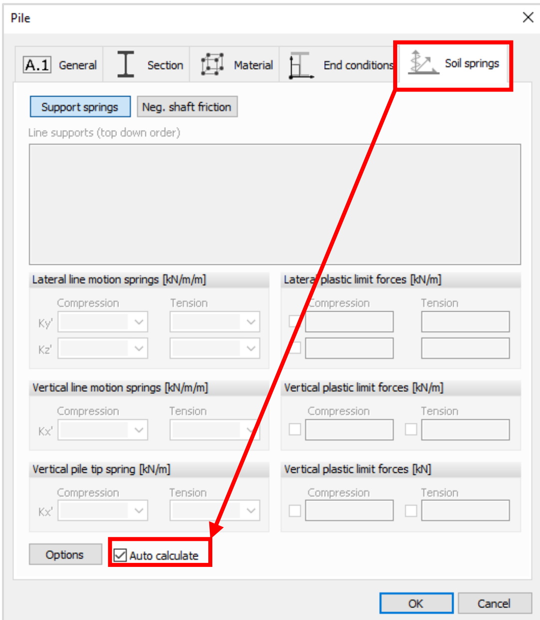
The material can be selected in the same way as for *Beams/Columns*.



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.



The program generates a set of line support groups for the pile (at least one for each stratum with an additional breakpoint at the highest water level) and one point support at the tip of the pile. If the pile is edited or some changes happen in the input data (e.g. changing of the material of a stratum), these stiffness and plastic limit values will be recalculated. The automatic calculation can be switched off in *Pile dialog/Soil springs tab/Auto calculate checkbox*.



⚠ The *Soil springs tab* will be active only after placing the pile and its properties is displayed in the dialog.

From the previously defined strata, FEM-Design calculates the stiffness and plastic limit values for both the line and point supports.

The *line support groups* along the pile have only translational stiffness, $K_{x'}$ vertical (shear) and $K_{y'}$, $K_{z'}$ horizontal stiffness [kN/m/m]. For the *horizontal spring stiffness*, it does not have any practical sense to define compression and tension stiffness, thus for the horizontal directions (y', z') one value defines each direction. The horizontal stiffness of the line support in y' direction (similar in z' direction):

$$K'_y = k_{s,y'} \cdot B \quad \left[\frac{kN}{m^2}\right]$$

where B is the width of the pile and ks,y' is the horizontal coefficient of subgrade reaction. According to *Vesic (1961),* it can be calculated by using both soil and pile properties:

$$k_{s,y'} = \frac{0.65 \cdot E_s}{B \cdot (1 - \mu_s^2)} \cdot \left[\frac{E_s \cdot B^4}{E_p \cdot I_{p,z'}}\right]^{\frac{1}{12}} \quad \left[\frac{kN}{m^3}\right]$$

where E_s and μ_s is the Young's modulus and Poisson's ratio of the soil, E_p and $I_{p,z'}$ are the Young's modulus and moment of inertia of the pile.

The vertical behaviour, thus the *vertical spring stiffness* might be different for compression and tension, so both spring stiffness are available for the User to define (for example One may want to neglect the friction forces for tensioned piles). Based on the analytical solution of *Zhang Q. et al. (2014)*, the vertical stiffness of the line support is:

$$K_{x'} = k_s \cdot P = \frac{G_s}{r_0 \cdot ln\left(\frac{r_m}{r_0}\right)} \cdot P \quad \left[\frac{kN}{m^2}\right]$$

where G_s is the shear modulus of the soil, r_0 is the pile radius (or equivalent radius for noncircular piles), r_m is the radial distance at which shear stresses in the soil become negligible and P is the perimeter of the cross section of the pile. In the different soil layers, r_m distance can be calculated as the following:

$$r_m = 2.5 \cdot L \cdot \rho_m \cdot (1 - \mu_s)$$

where L is the (total) length of the pile, ρ_m is the factor of vertical homogeneity of soil stiffness and μs is the Poisson's ratio of soil layer around the pile. The value of ρ_m can be calculated in the following way:

$$\rho_m = \frac{G_{s,middle}}{G_{s,bottom}}$$

where $G_{s,middle}$ is the shear modulus at the middle of the layer and $G_{s,bottom}$ is the shear modulus at the bottom of the layer (relevant only in case of soils with variable material properties with height).

The point support at the tip of the pile has only vertical (or in case of skew piles only x' directional) stiffness. The tension stiffness is set to zero by default as tension forces are not transmitted between the soil and the tip of the pile. *Zhang Q. et al. (2014)* suggest the following formulae to use for the compression stiffness:

$$K_{x'} = \frac{4 \cdot G_s \cdot r_0}{(1 - \mu_s)} \quad \left[\frac{kN}{m}\right]$$

Besides the spring stiffness, plastic limit values are also calculated for both line and point supports. For the vertical line support the plastic limit value is based on the shaft friction, for the point support at the tip of the pile it depends on the end bearing of the pile. In both cases the calculation is affected by the behaviour of the soil (drained/undrained), as it is summarized by *Wrana B (2015)*.

In case of *undrained soil* the limit value is based on the undrained shear strength (c_{uk}). The limit value of the *line support*:

$$P_{lim,x'} = q_s \cdot P = \alpha \cdot c_{uk} \cdot P \quad \left[\frac{kN}{m}\right]$$

where q_s is the shaft friction and α is the adhesion coefficient. The latter one is calculated according to the proposition of the *NAVFAC DM 7.2 (1984)*, the corresponding details can be found in the literature. The limit value of the *point support* (for compression):

$$P_{lim,x'} = A_{base} \cdot c_{uk} \cdot N_c \quad [kN]$$

where A_{base} is the cross-sectional area of the pile (at the tip), N_c is the bearing capacity coefficient that can be assumed equal 9.0 according to *Skempton (1959)*.

In case of *drained soil* the resistances are based on the internal friction angle (φk) and cohesion (ck) of the soil layers. The limit value of the *line support*:

$$P_{lim,x'} = q_s \cdot P = \beta \cdot \sigma_v' \cdot P \quad \left[\frac{kN}{m}\right]$$

where $\sigma_{V'}$ is the vertical effective stress and β is a friction coefficient multiplier. As the vertical stresses are increases with the depth, an average value of the current layer is used in the equation. At water level vertical stresses have a break point, thus at the generation of line supports this point is always considered in such a way that it is the start/endpoint of the neighbouring line supports.

Only the *highest water level* is considered in the calculation of plastic limit values and negative shaft friction forces.

The value of β is taken according to the *NAVFAC DM 7.2 (1984)*. The limit value of the *point support* (for compression):

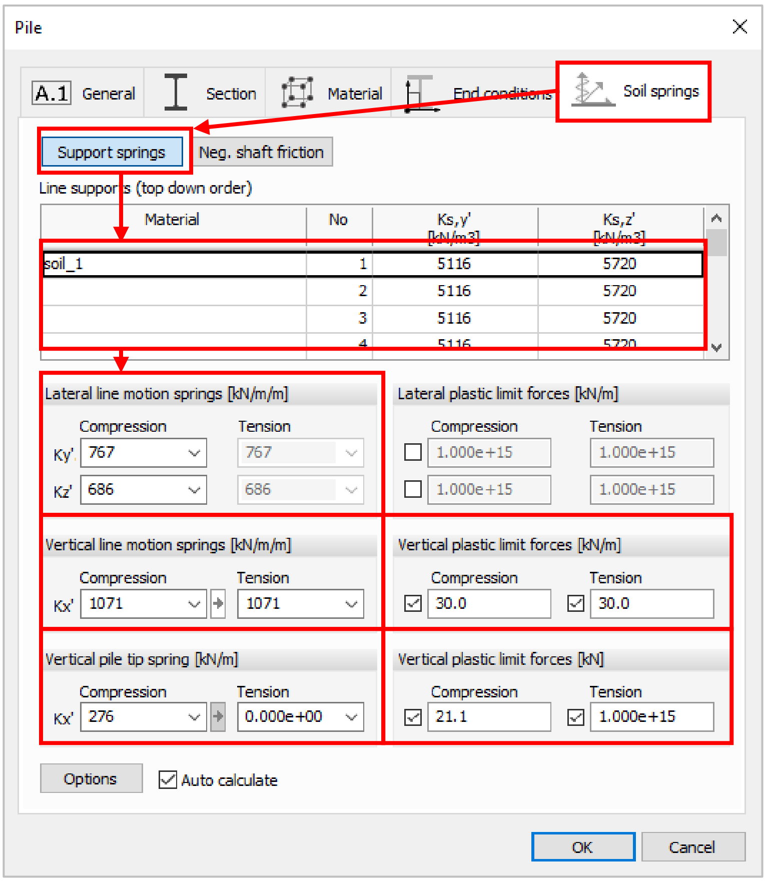
$$P_{lim,x'} = A_{base} \cdot q_b = A_{base} \cdot \left(\sigma_v' \cdot N_q + c_k \cdot N_c\right) \ [kN]$$

Where

$$N_c = \left(N_q - 1\right) \cdot \cot \phi_k$$

where Nq is the bearing capacity factor, and according to the NAVFAC DM 7.2 (1984), its value depends on the fabrication technology of the pile and the internal friction angle of the soil.

In the *Support springs* side of the *Soil springs tab* User 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*.



The internal forces of the compressed piles should be increased by the value of the negative shaft friction, as it is an additional effect coming from the soil-pile interaction. In FEM-Design it is considered with a special, automatically generated load case. This load case exits only if there is at least one pile in the model. As negative shaft friction appears only above the *neutral level* (measured from the top of the pile), by default it is set to zero, which means that there are no actual loads in this load case. After we change the neutral level, the corresponding loads are calculated and generated automatically.

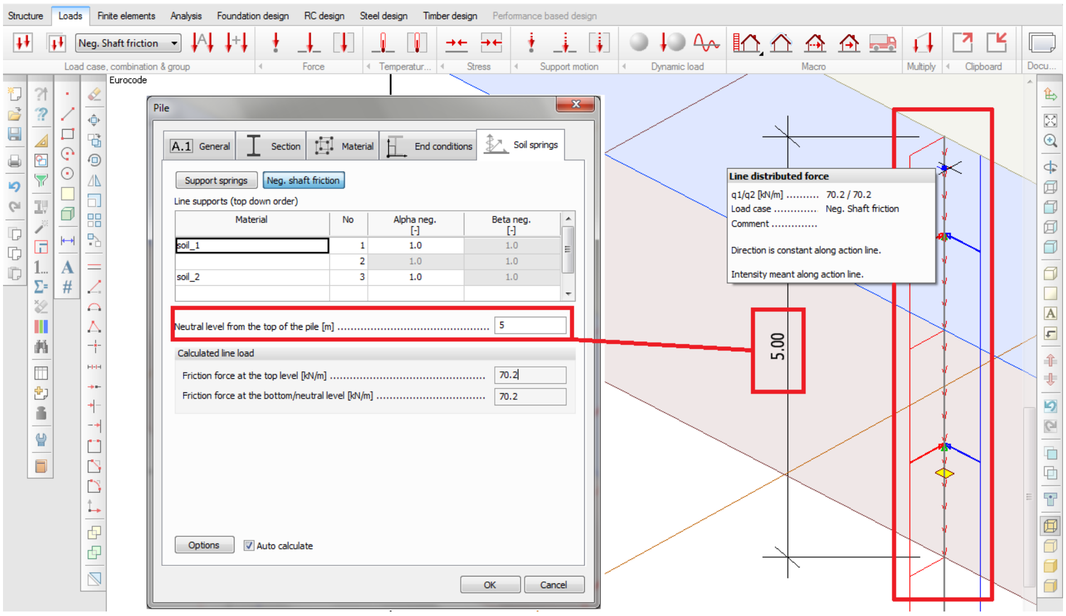
The value of the friction force in case of *undrained soil*:

$$P_s = P \cdot \alpha_{neg} \cdot c_{uk} \ \left[\frac{kN}{m}\right]$$

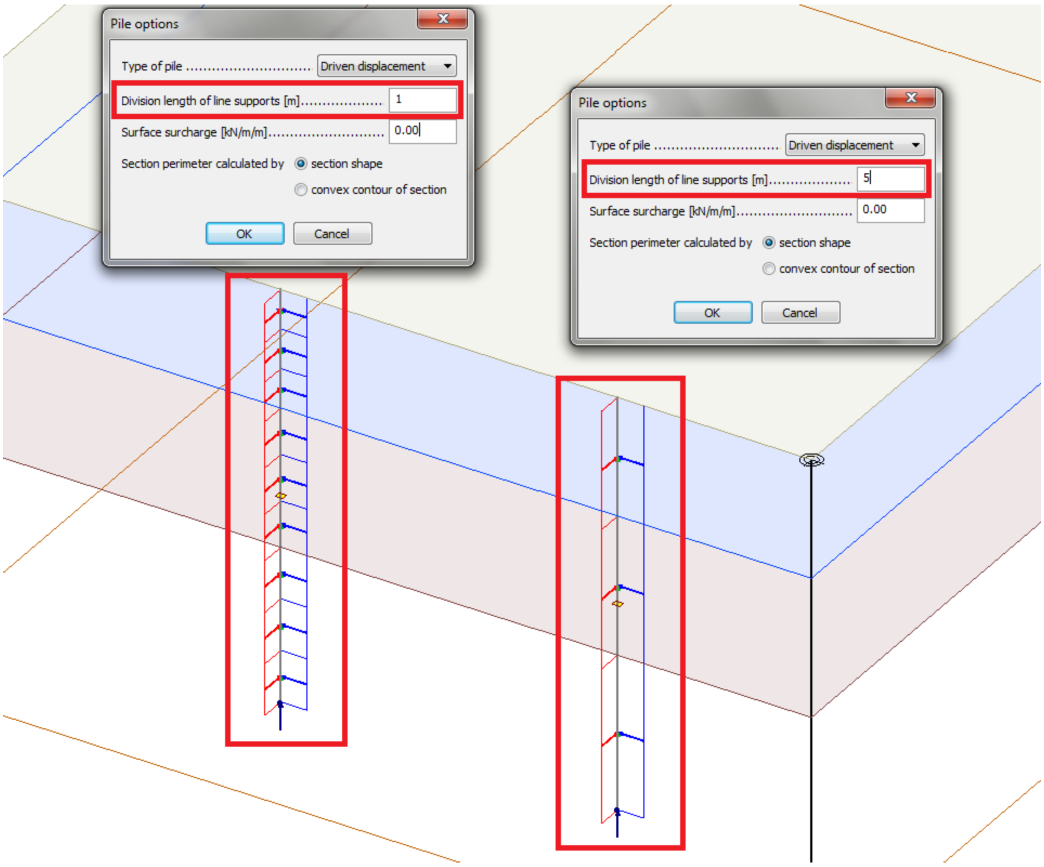
where α_{neg} is the factor for negative shaft friction for undrained soils (default value is assumed to 1.0). The friction force for *drained soil*:

$$P_s = P \cdot \beta_{neg} \cdot \sigma_v' \ \left[\frac{kN}{m}\right]$$

where 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 User by the modification of the α_{neg} and β_{neg} values (each stratum has one value) together with neutral level on the *Neg. shaft friction* side of the *Soil springs tab*:

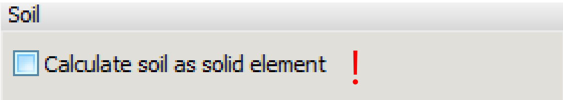


If User clicks on *Option*, the *Pile option dialog* opens. Here User can select the *Type of pile*, declare the *Surface surcharge* (affecting the vertical stresses in case of drained soils), the *division length of line supports* and can decide the method of the *Section perimeter*'s calculation. Division length becomes a very important setting in case on plastic analyses. The plastic limit forces are increasing with depth, but along a line support it is constant. Consequently, the more line supports are created along the pile (with the automatically calculated, increasing limit values) the more accurate plastic distribution of internal forces we get.

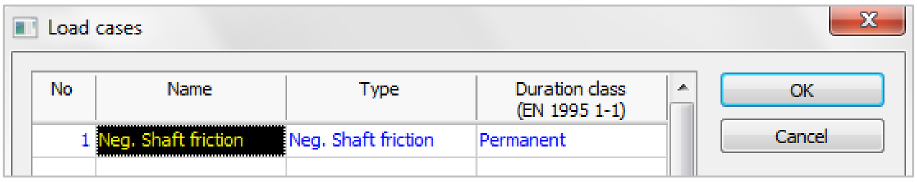


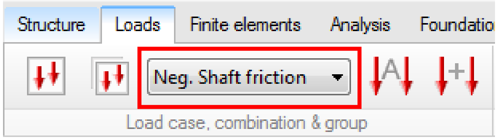
The feature has some important limitations!

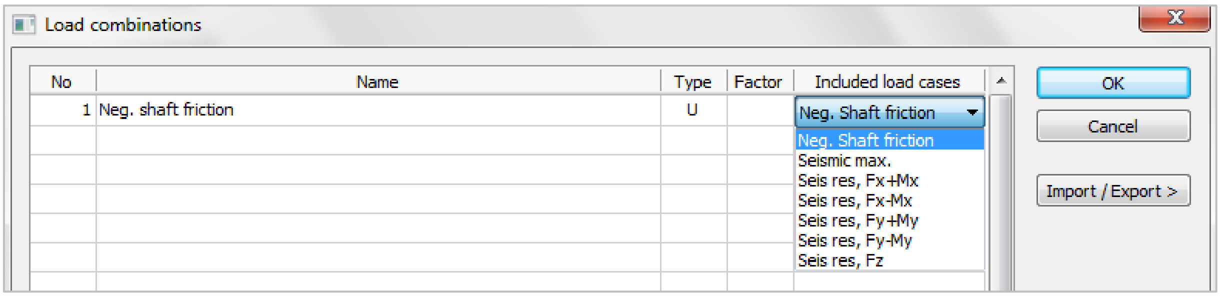
- Pile model is used only for analysis purpose (displacements, internal forces), geotechnical design calculations are not implemented yet.
- During the calculations soil must not be modelled as soil element.



- 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.







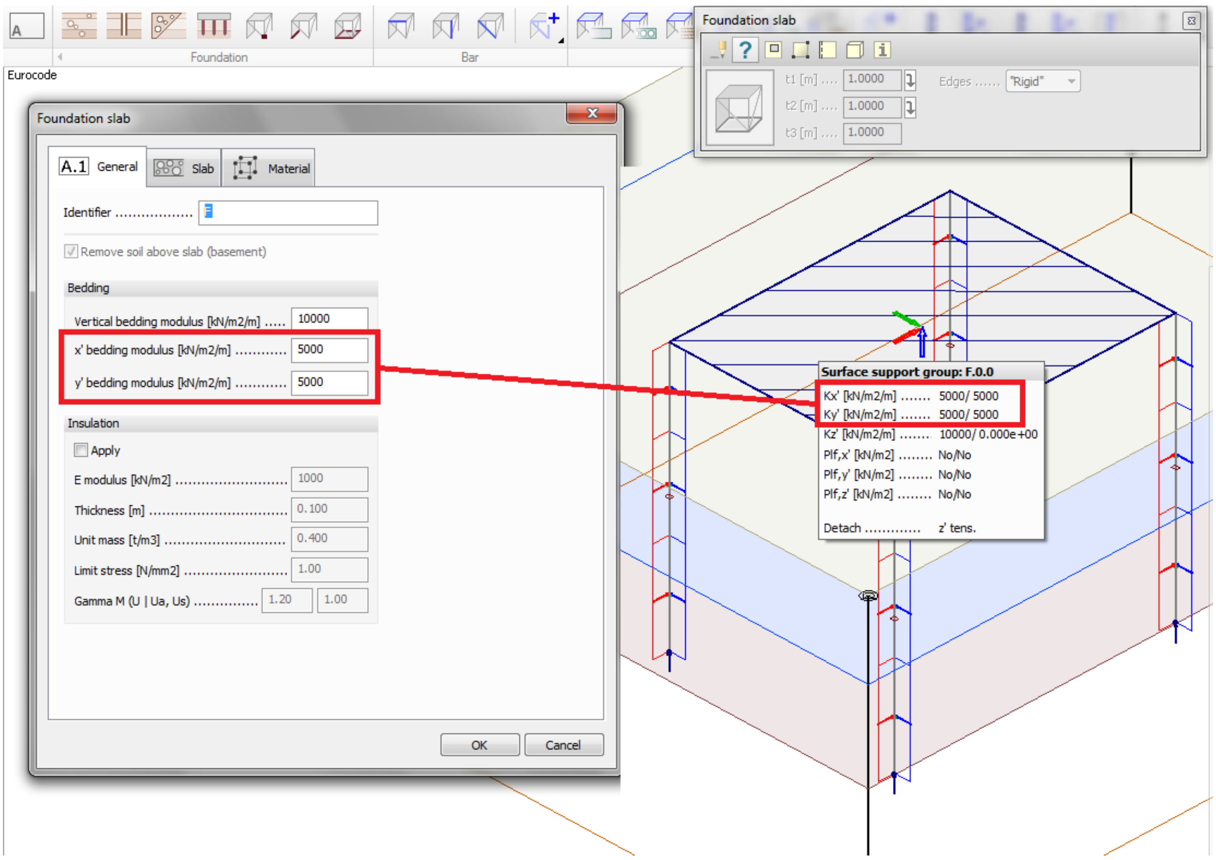
References:

- 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

3.5. Horizontal bedding moduli of foundation slab

In the properties dialog of the *Foundation slab* a new option has been added.

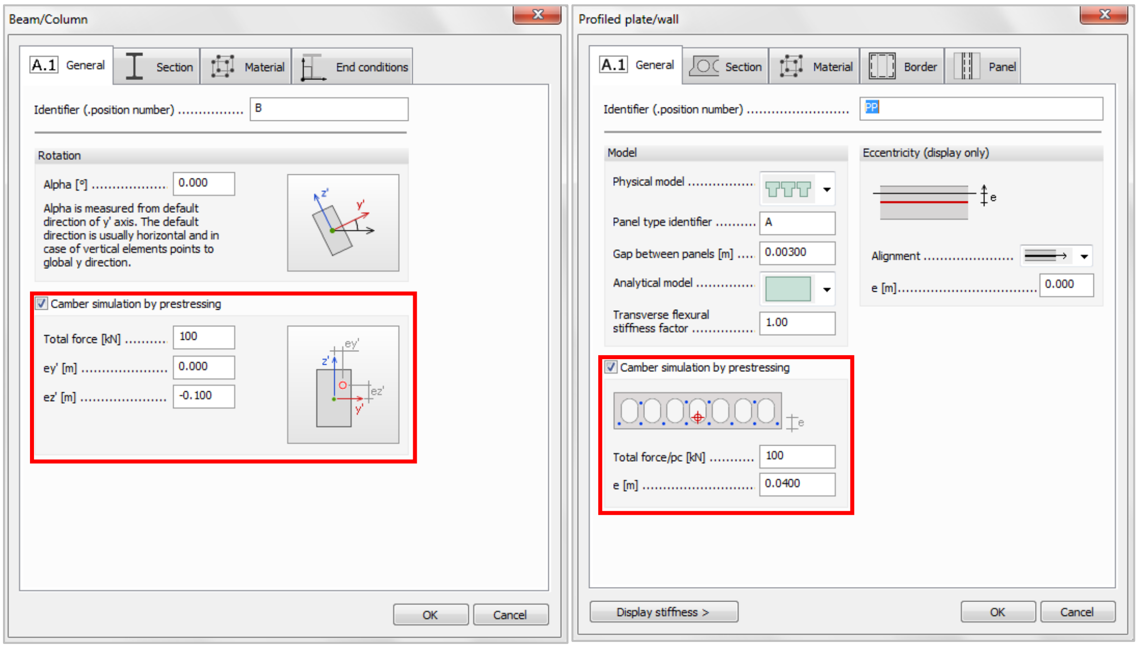
From now, besides the vertical bedding modulus, the *horizontal* one can be specified by the User, instead of making them equal to the vertical one. These three values define the motion spring stiffness of the automatically created surface support group under the foundation slab. By default, the horizontal values are the half of the vertical one.



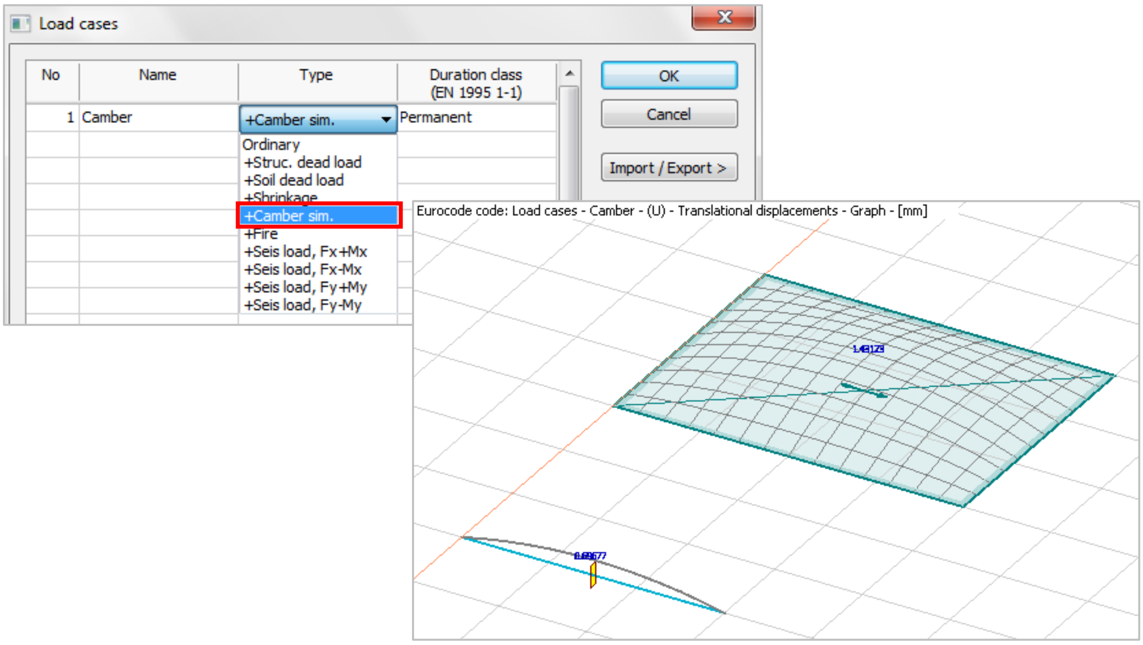
This modification has a strong connection with the recently added Pile element. If User would like to model a pile group connected to a foundation slab, the reduction of the horizontal stiffness of the foundation slab can be reasonable. As piles have horizontal supporting directly by the surrounding soil, they are able to carry either the total or a portion of the total horizontal loading.

3.6. Camber simulation option for Beams and Profiled plates

A new option has been implemented for beams and profiled plates thatmakes easy to simulate camber of the elements. It can be found in *Beam* and *Profiled plate dialog/General/Camber simulation by prestressing.*



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

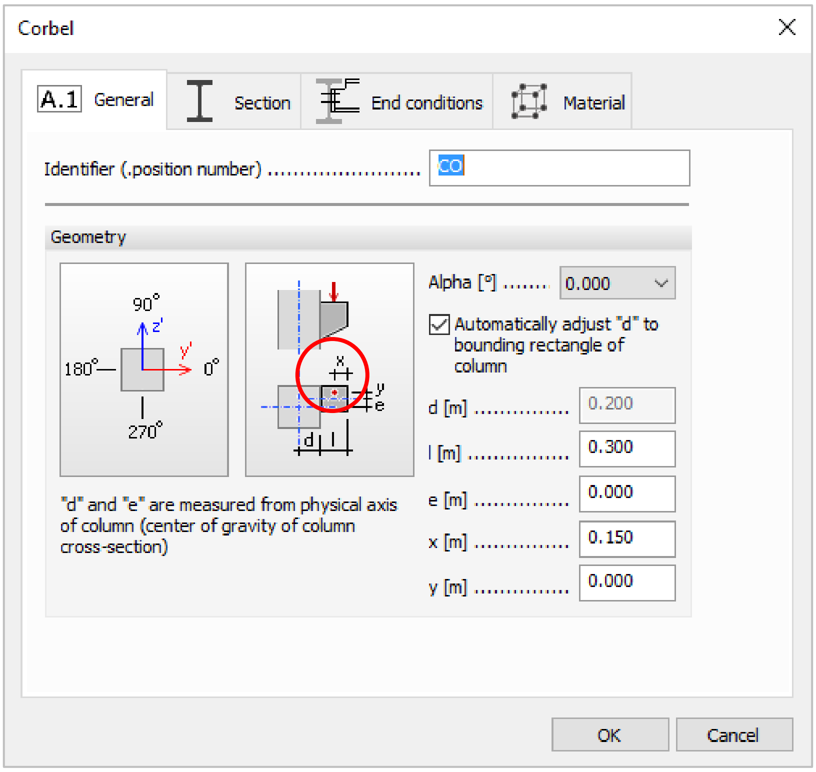


Camber calculation gives statically correct result only for non-eccentric, hinged beams and shells. This only corrects the displacement result and no internal forces will rise in the model.

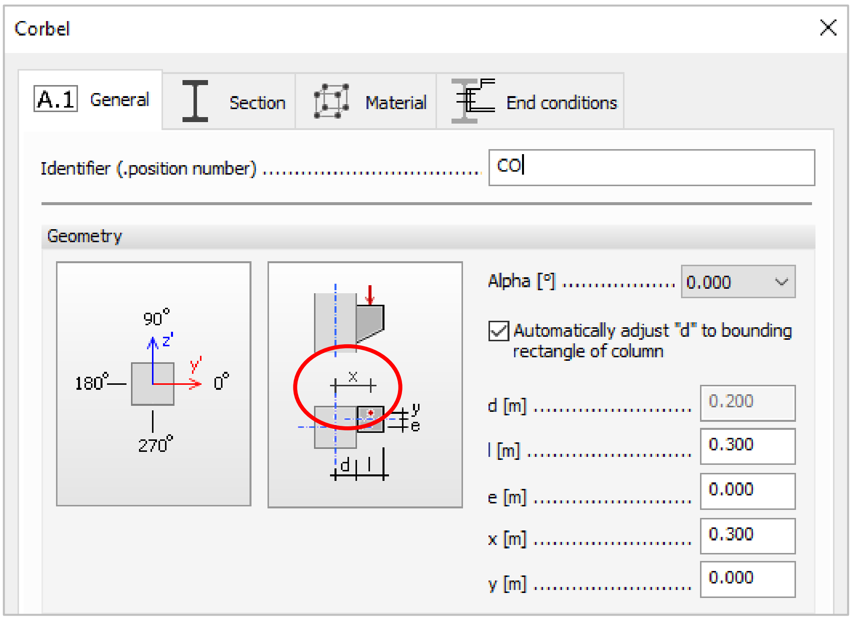
3.7. Easier definition of column corbel's load position

The column corbel's load position is measured from the column axis, instead of the edge of the corbel, which is easier to define.

In version 16:



In version 17:



3.8. Post-tensioned cable

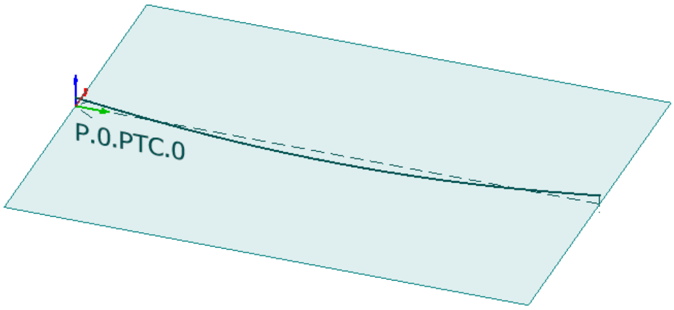
3.8.1. General

Modelling

Post-tensioned cable object (hereinafter PTC) is a structural component, modelled by equivalent load system.

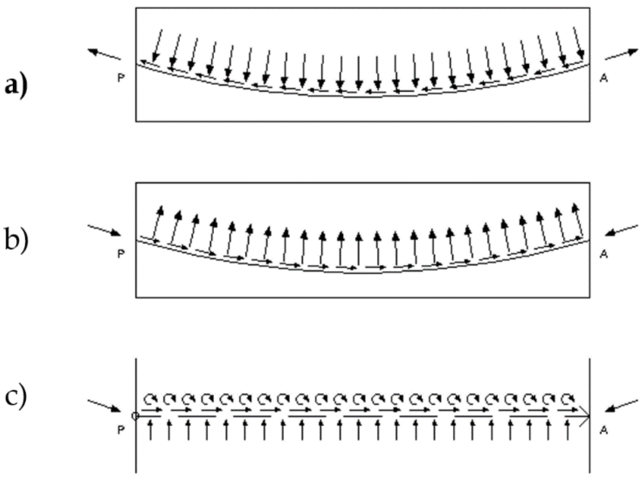
Currently the unbonded structural design modelling is available.

The object contains of the shape (continuous line), the reference line (dashed line) and an arrow marking the active end.



The following figure shows a post-tensioned beam after jacking: the actions on the cable a), the actions on the structure b) and the modelled forces acting on the reference line c).

Besides the equivalent forces in the local z direction from the angular deviation, the neglected x' component of the angular deviation, the friction force, and moment caused by the friction force - cable eccentricity (relative to the reference line) could be significant along the reference line.



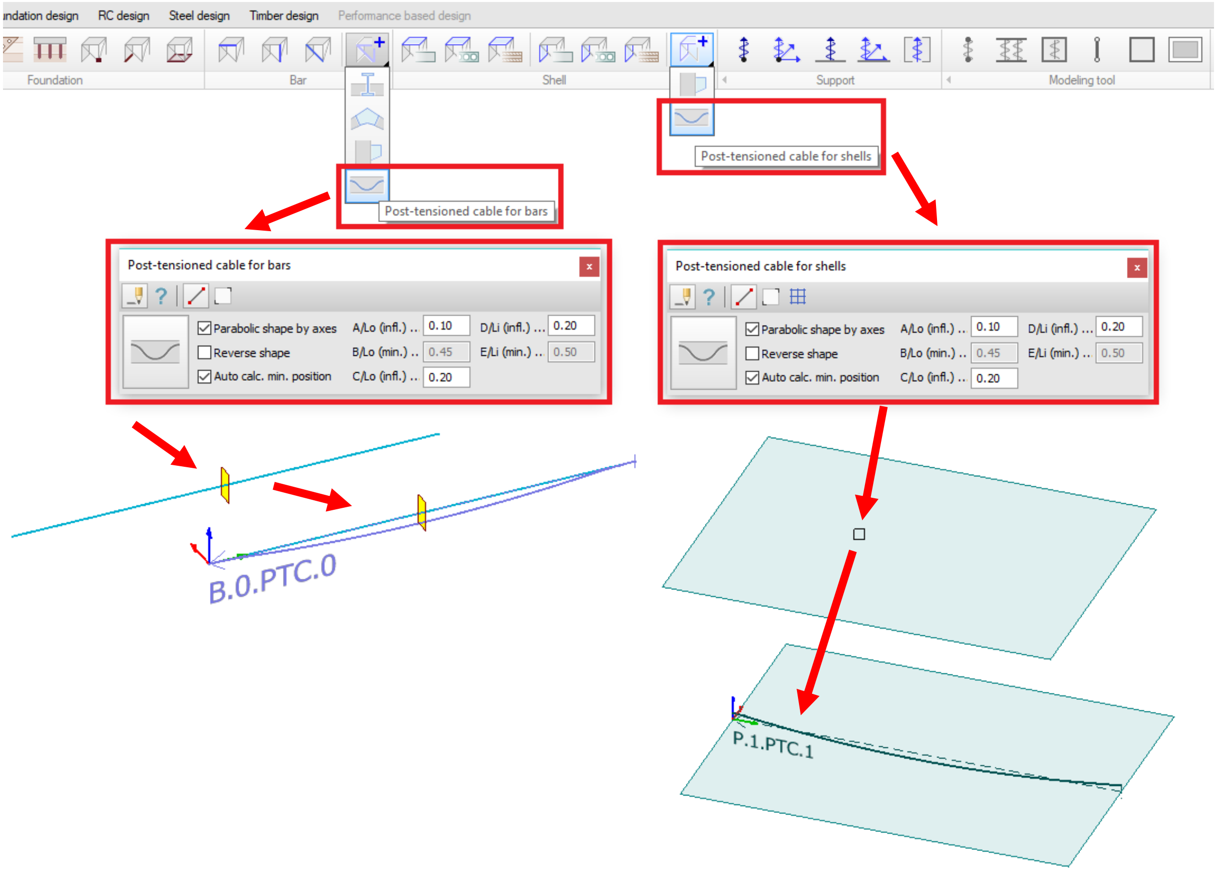
The plane of the shape can be modified by using "Change direction" or "Rotation" functions.

The definition of a PTC automatically creates two load cases:

- PTC T0: Initial stress state (after the jacking process, it contains the short term stress losses)
- PTC T8: At the end of the design lifetime (it contains the time-depenedent stress losses)

Definition process

Post-tensioned cable command can be launched from Structure/Bar component/Post-tensioned cable and Structure/Shell component /Post-tensioned cable.

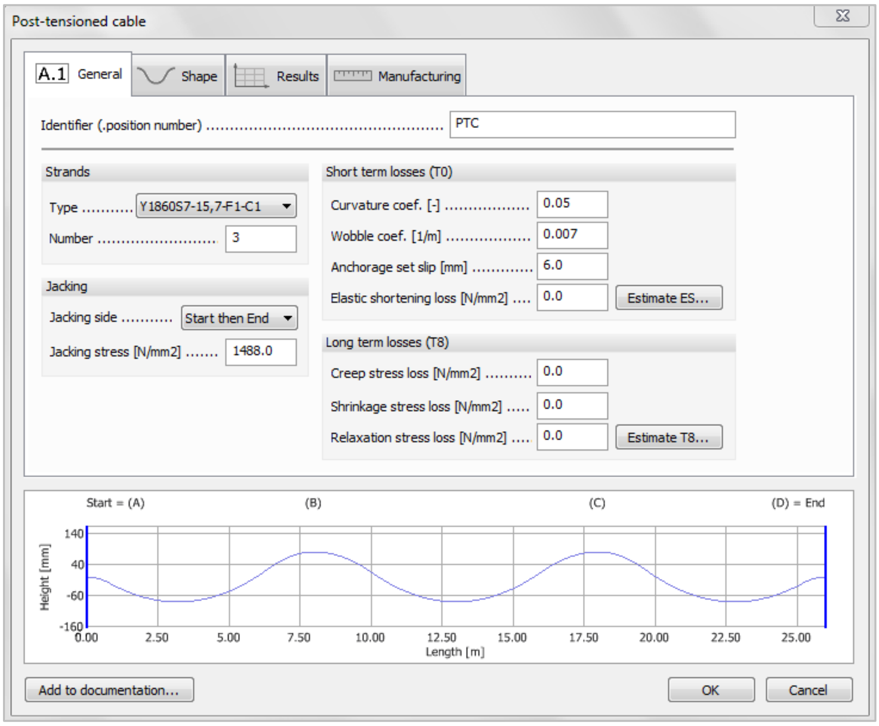


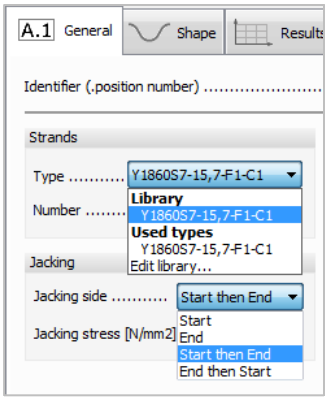
After clicking *Post-tensioned cable* the *Tool window* shows up and User needs to pick the parent object (bar or shell), then define the line of the PTC. The two *Tool window* work slightly different: after the shell selection the user can define multiple cable.

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.

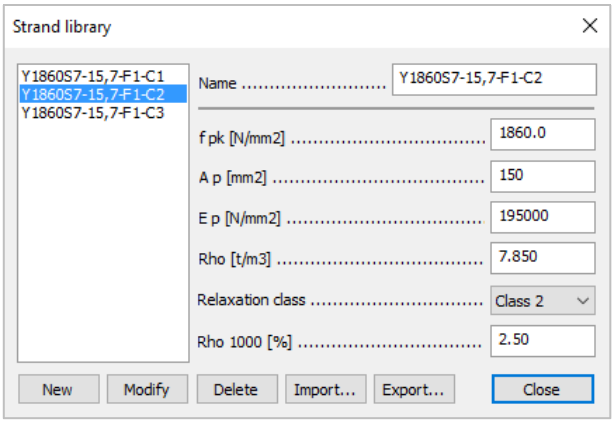
Properties dialog: General tab

By clicking Default settings or Properties the Post-tensioned cable dialog opens.





Strand type can be selected from the strand library. The strand library can be edited by the User by clicking *Strands/Type/Edit library…*



In the same dialog 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 results different stress functions using shorter cables.

Jacking stress is calculated as $0.8 * f_{pk}$ by default after the Strand type selection.

The cable force is reduced by several losses. Their settings are on the right side of the dialog:

Short term losses (initial, T0):

- 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):

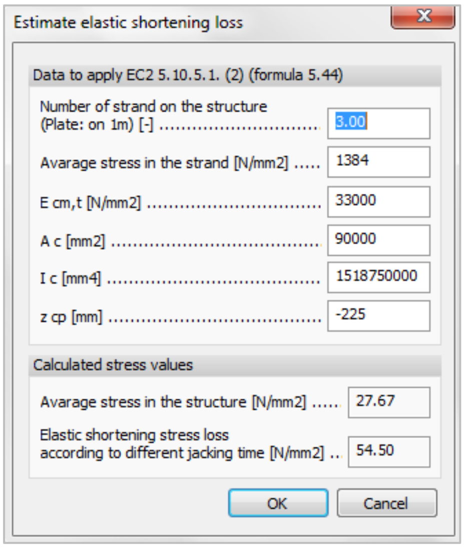
$$\Delta P_{\mu}(x) = P_{max} + (1 - e^{-\mu(\theta + kx)})$$

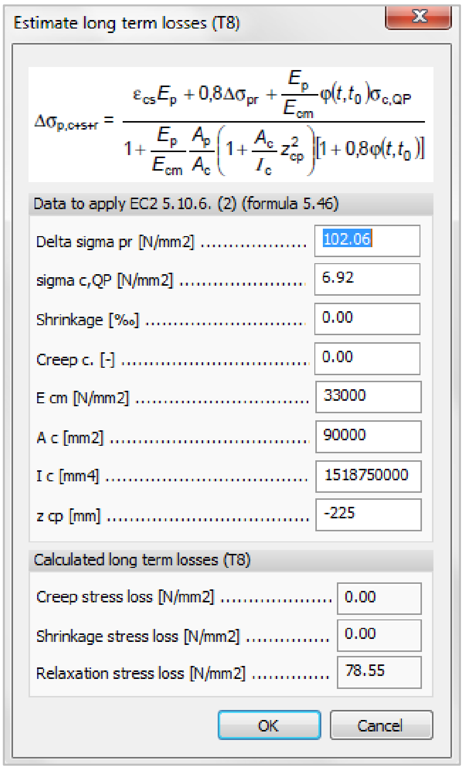
- anchorage set slip
- elastic shortening

Long term, time dependent losses (final, T8):

- shrinkage of structure
- creep of structure
- relaxation of post-tensioned cable

The *Elastic shortening loss* and *Long term losses* can be estimated by the specific dialog.





Elastic shortening loss can be estimated by *Estimate ES...* button. This dialog is automatically filled with the parent object's data. Calculate stress values are the result of the estimation and the *Elastic shortening stress loss* field is applied on the *General tab* if the User accepts. It uses modified EN 1992-1-1 5.10.5.1. (2) (formula 5.44) to handle sparsely placed cables:

$$\Delta\sigma_{el} = E_p \sum \left[\frac{j\Delta\sigma_c}{E_{cm}}\right]$$

$$if\ n \geq 2\ than\ j = \frac{n-1}{2n}$$

$$else\ j = 0.5n$$

Where n is the Number of strand.

Average stress in the structure (σ_c) is informative only.

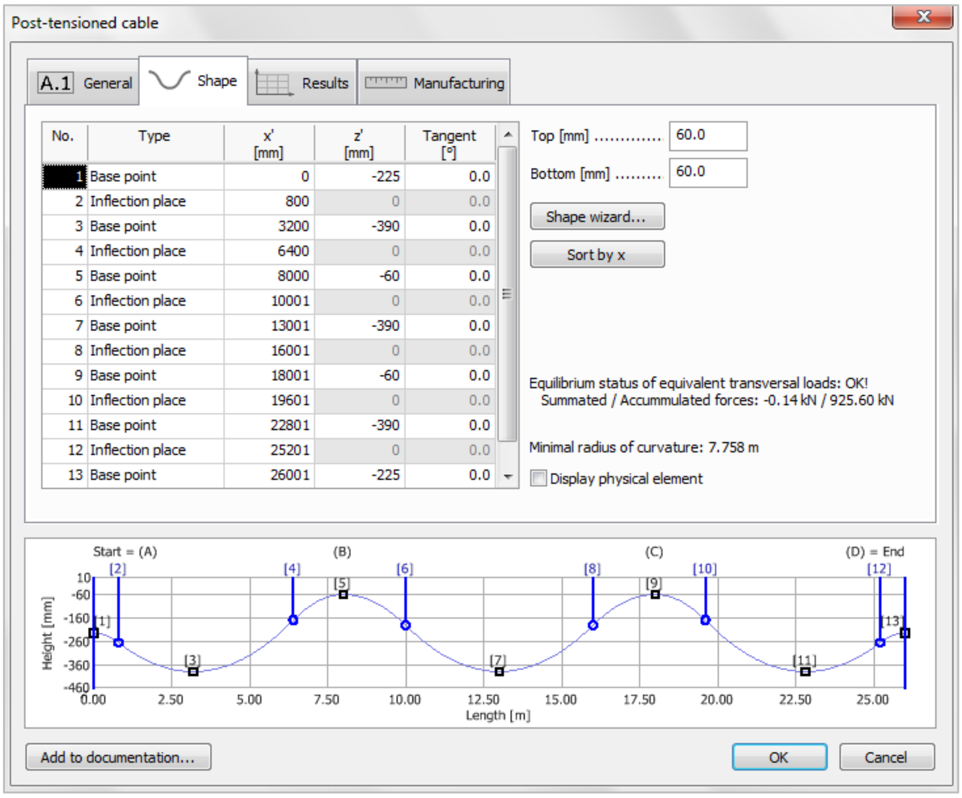
Long term losses can be estimated by the *Estimate T8*... This dialog works as the previous one: fields are filled with the parent object's data and the calculated results are at the bottom (Relaxation information is in the Strand library). It uses EN 1992-1-1 5.10.6 (2) (formula 5.46). The summation of the calculated long term losses is equal to the result of this interaction formula.

Estimation dialogs calculate cross section data for 1m wide stripes if plate object is the parent.

Both estimation dialog is available during the *Default settings* editing, edit fields are filled with zeros.

Properties dialog: Shape tab

On the *Shape tab* there are settings related to the geometry of the cable.



The Shape table can contain *Base points* and *Inflection places*: these determine whether linear or parabolic shape is applied.

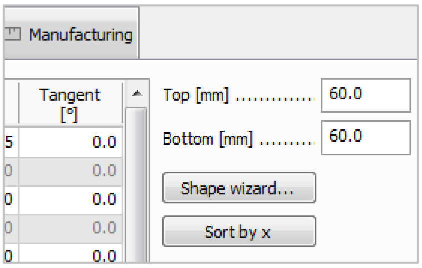
Base point: A point with exact position, user known x' – z' coordinates and the angle of the tangent. Usually minimum and maximum places and end points. (Black rectangles in the preview.)

Inflection place: A function connection place (x_{inf}), where user defines x coordinate only. Using it between Base points determines two parabolic function: f_n and f_{n+1} where C^1 continuity is fullfilled: $f_n(x_{inf}) = f_{n+1}(x_{inf})$, $f'_n(x_{inf}) = f'_{n+1}(x_{inf})$. (Blue circles in the preview.)

"*Inflection place*" is not an accurate mathematical expression in this feature: the change of function convexity is not ensured. However this term makes recognizable the underlying function.

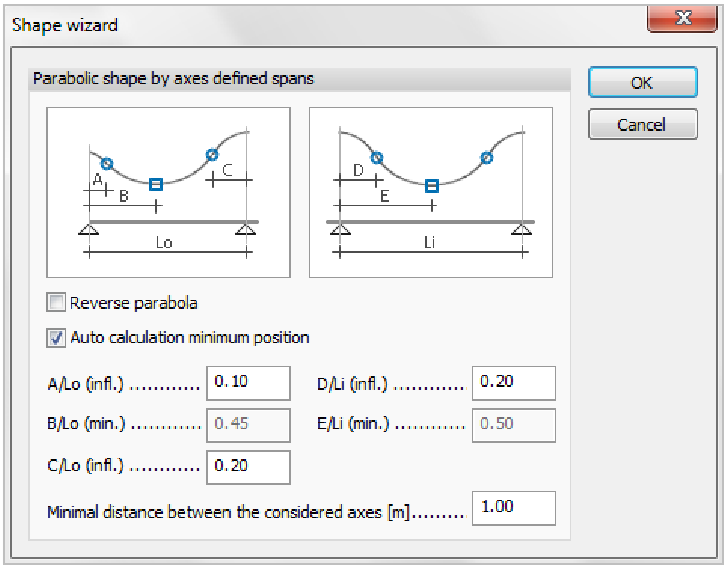
For x' and z' fields some macros are available:

- x' macro: Start, End, Axis distances
- z' macro: Top, Middle, Bottom

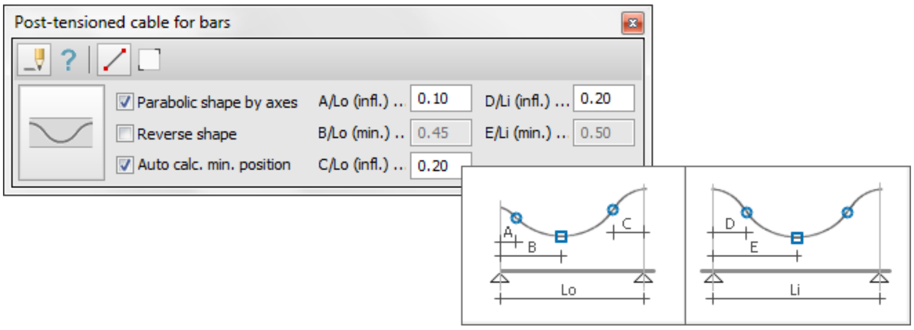


The *Top* and *Bottom* values mean the distance between the axis of the cable and the edge of the element. These values are used by the *z' macro* and *Shape wizard*.

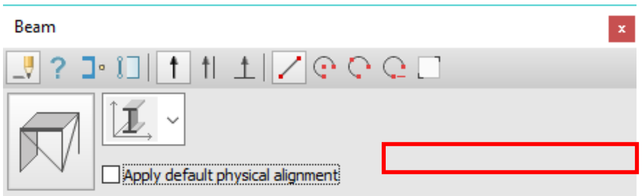
Using *Shape wizard* (*Shape wizard...* button) parabolic shapes can be easily created. This tool highly relies on previously defined axes. The *Axes* define *outer* (marked by *o*) and *inner* spans (marked by *i*) along the PTC. At the span ends *Base point* will be created with maximal z' coordinate, the minimal (*B, E*) place of the *Base points* and Inflection places (*A, C, D*) can be defined by ratio of spans (*Lo* and *Li*). *B/Lo* and *E/Li* minimum positions can be calculated automatically by checking the box next to *Auto calc. min. position*. At that time these textboxes are disabled.



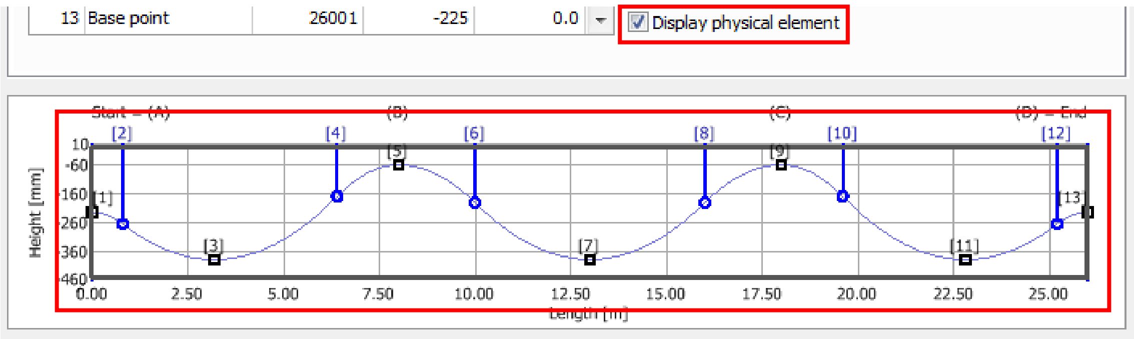
The same settings can be found in the *Tool window* as well for easier definition:



It is not recommended to use *Beam/Apply default physical alignment* option for beam containing PTC, since it can cause unnecessary eccentricity in the shape because *Shape wizard* uses the physical element.

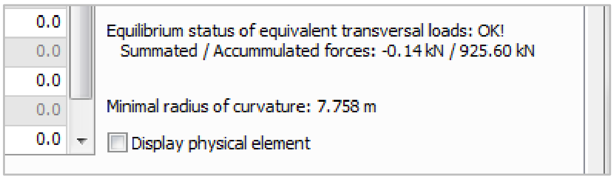


By allowing the *Display physical element* optionthe section of the cross-sectioned element is shown.



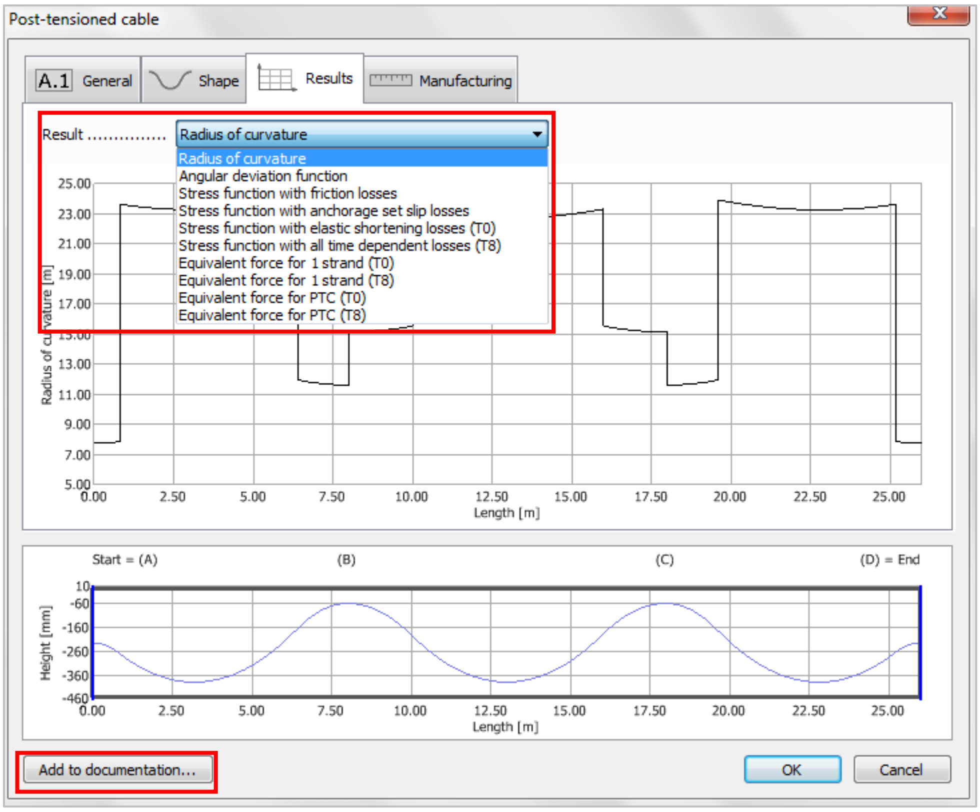
Display physical element option shows all cross-sectioned element, the parent object is signed by dark-grey, the non-parents are signed by light-grey.

The Equilibrium status and Minimal radius of curvature are also displayed.

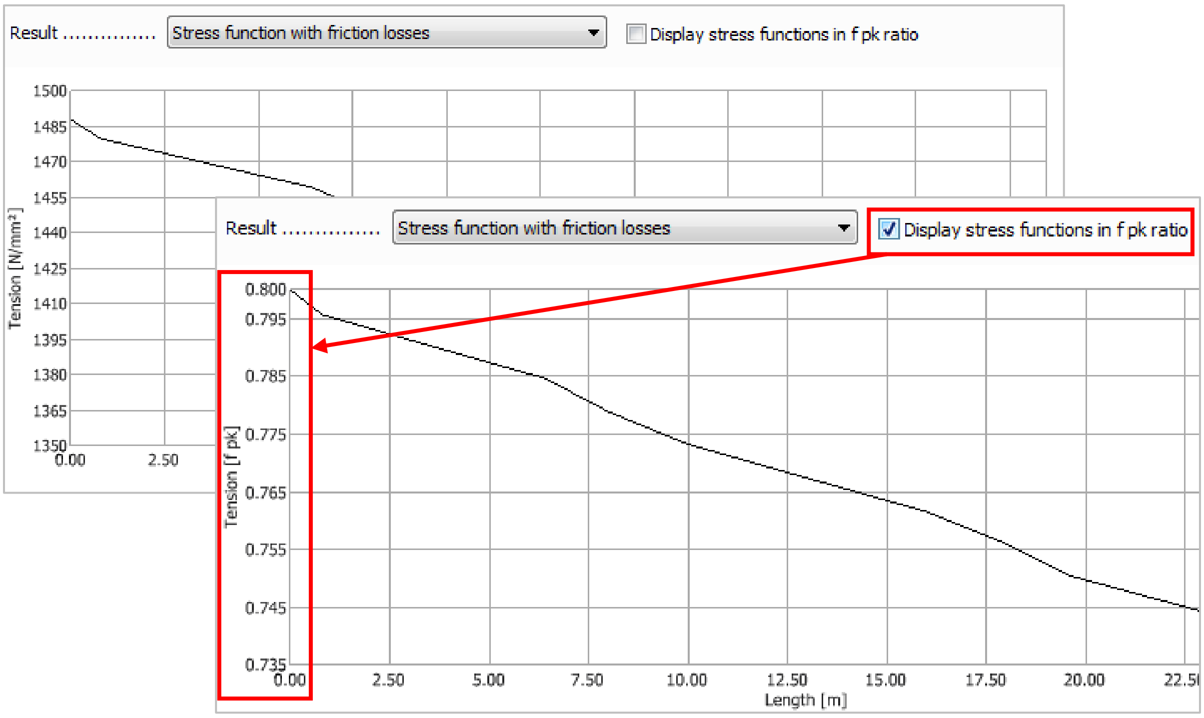


Properties dialog: Result tab

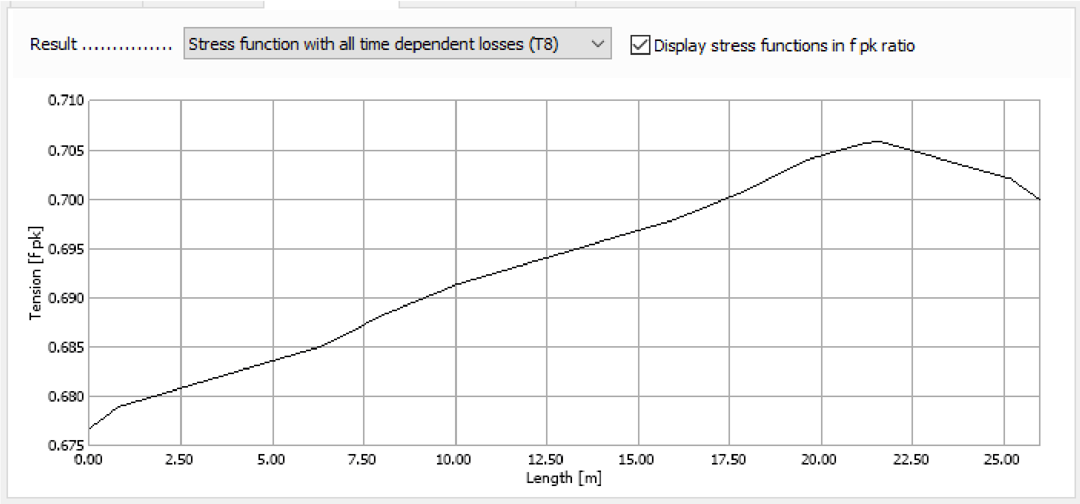
Result tab is implemented to make easier the checking process. There are several results -listed in the order of calculation - which can be chosen from *Result tab/Result drop-down menu.*

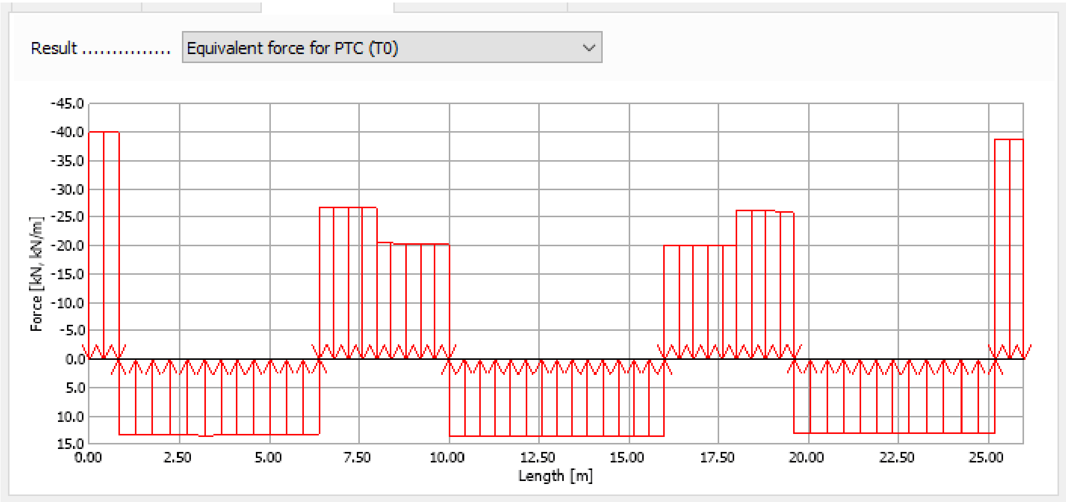


Stress functions can be displayed in f_{pk} ratio by *Display stress functions in f pk ratio* option.



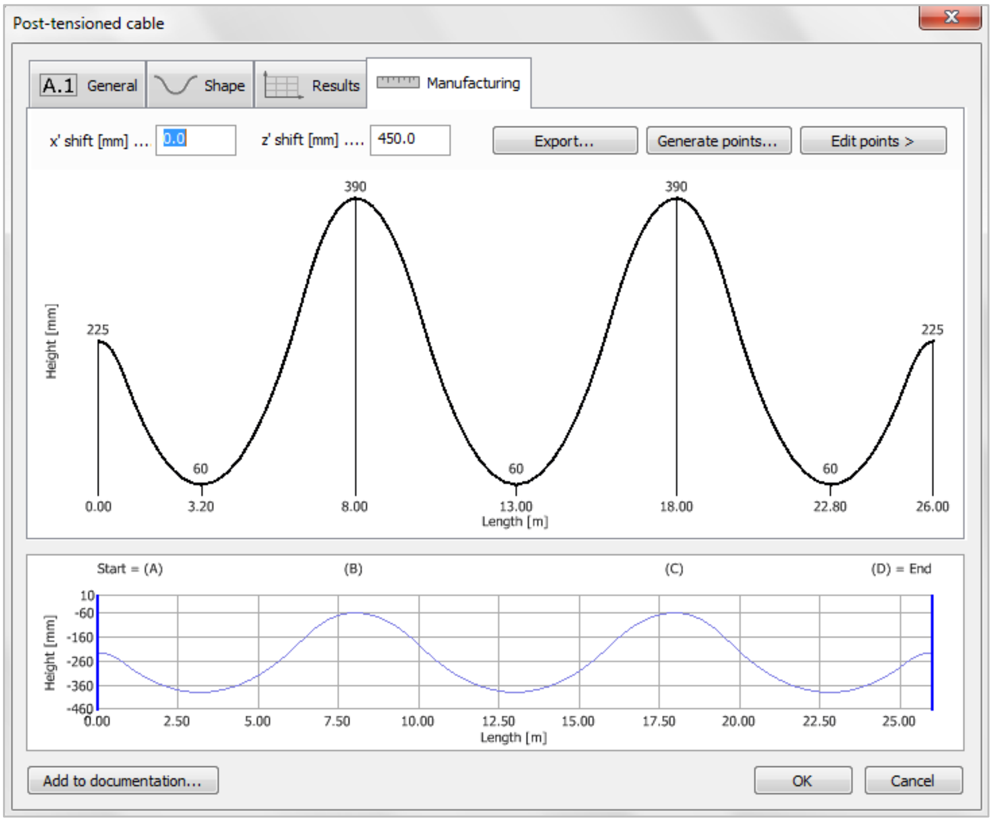
The example shows a typical Stress function in the T8 state and an Equivalent force at T0 state.



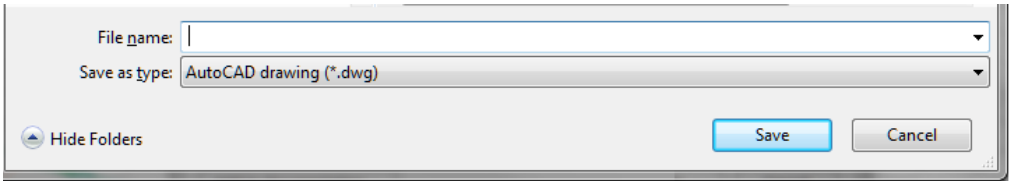


Properties dialog: Manufacturing tab

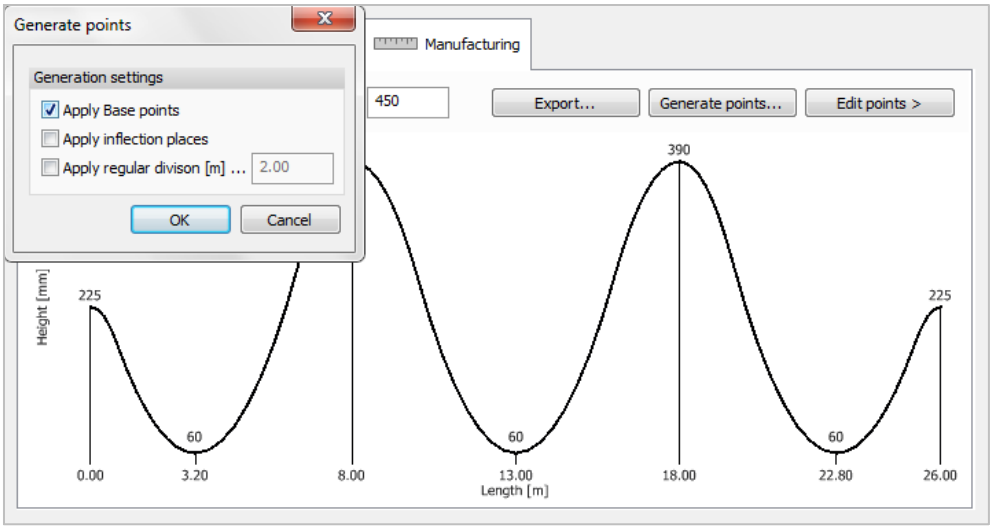
On this tab the Manufacture drawing can be set with proper x' and z' shift.

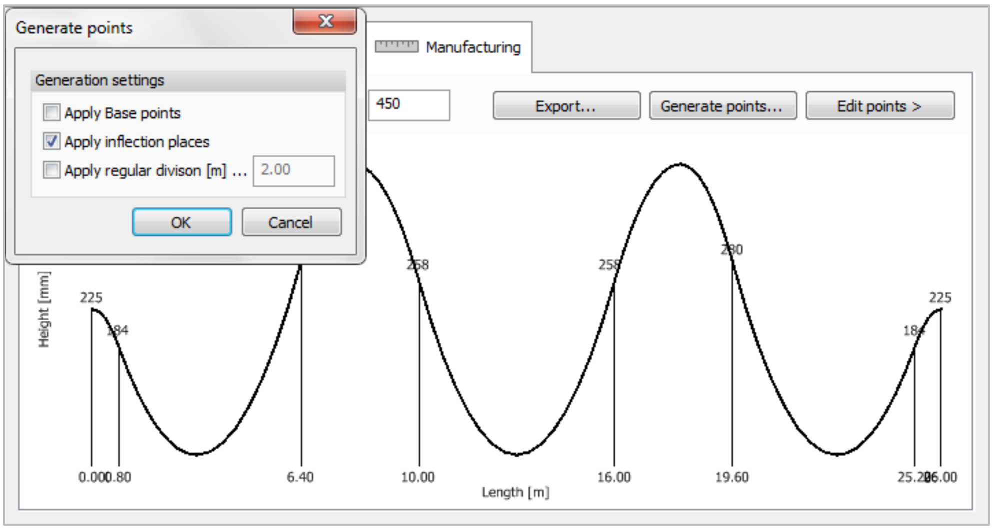


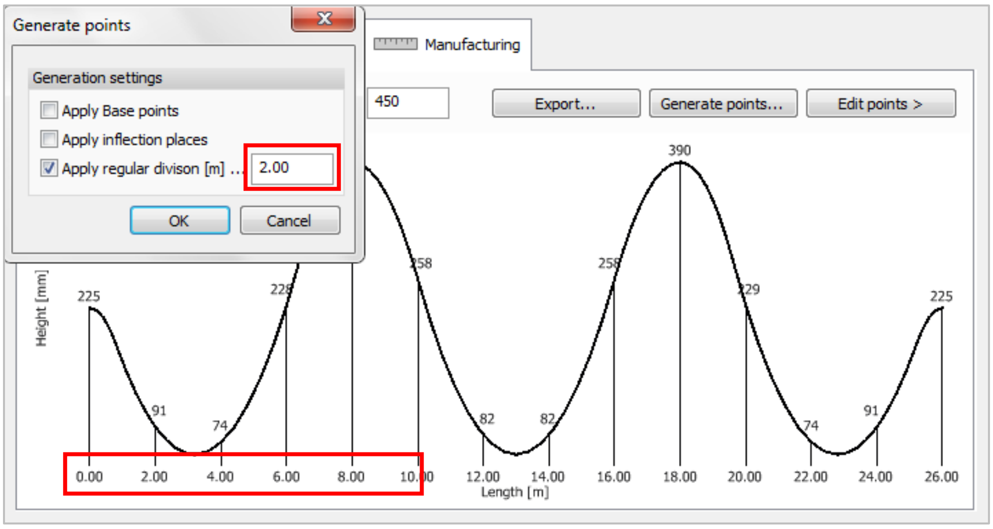
The Manufacture drawing can be exported into AutoCAD by clicking *Export...*



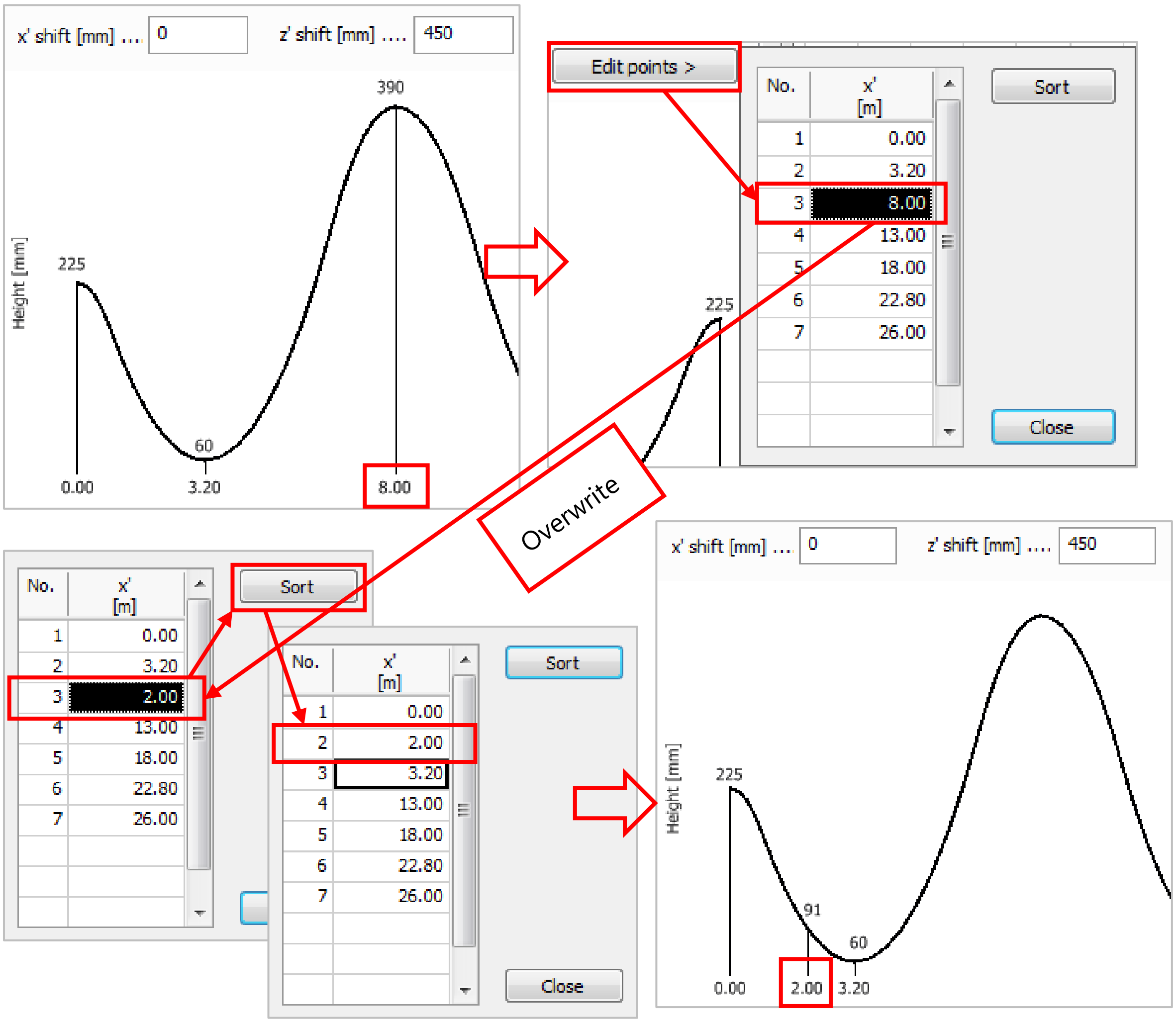
To fine the division section points can be inserted along the cable. Click *Generate points* to open the dialog. At the third option the division-distance can be set by the User.





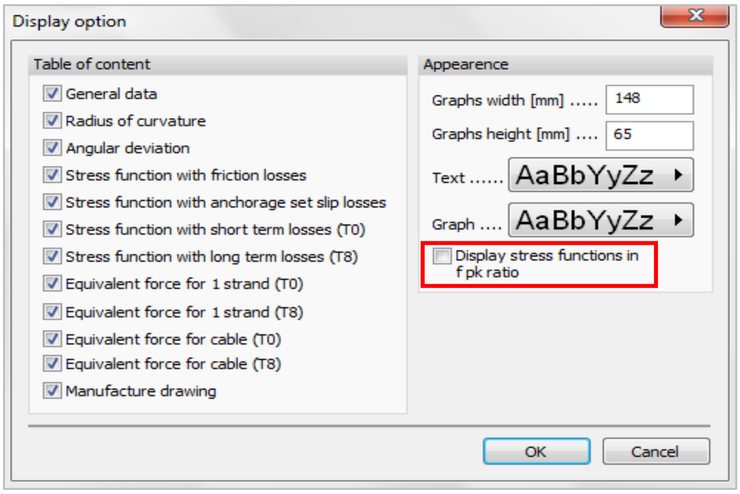


The position of each point can be edited with *Edit points* button. By clicking *Sort* button the points will be sorted by their x' value in increasing order.



Documentation

By clicking *Add to documentation…* User can decide about the content of the documentation.



The example shows the first page of a typical PTC documentation.

P.1.PTC.1
Post-tensioned cable details

General data
Strand type: Y1860S7-15,7-F1-C1

Strand No.: 3

Jacking stress: 1488.0 N/mm²; 0.800 * f pk

Jacking side: Start

Curvature c.: 0.050

Wobble c.: 0.007 1/m

Anchorage set slip: 6.0 mm

Elastic shortening loss: 0.0 N/mm²; 0.000 * f pk

Creep stress loss: 0.0 N/mm²; 0.000 * f pk

Shrinkage stress loss: 0.0 N/mm²; 0.000 * f pk

Relaxation stress loss: 0.0 N/mm²; 0.000 * f pk

Length (projected): 8.000 m

Length (real): 8.026 m

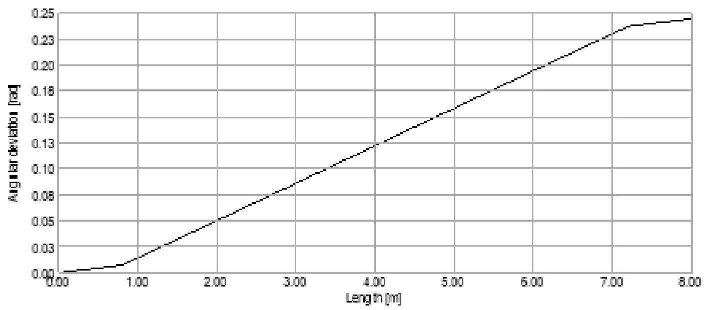
Stress T0 at Begin: 1319.5 N/mm²; 0.709 * f pk

Stress T0 at End: 1341.7 N/mm²; 0.721 * f pk

Stress T0 avg: 1330.6 N/mm²; 0.715 * f pk

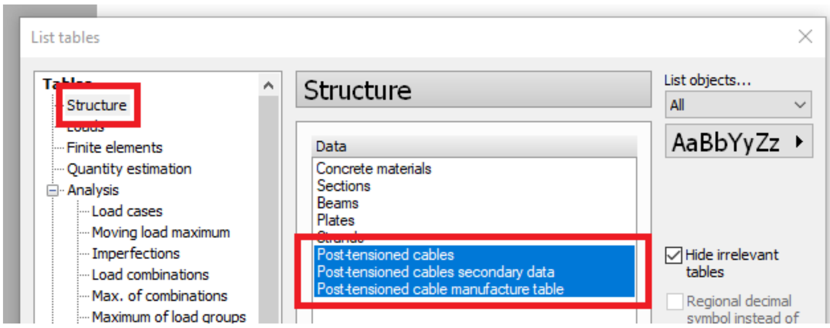
Stress T8 avg: 1330.6 N/mm²; 0.715 * f pk

Angular deviation function



There are three connected lists available:

- post-tensioned cables: Major and General tab data
- post-tensioned cables secondary data: Stress status and quantity estimation
- post-tensioned cables manufacture table: Manufacture drawing coordinates



Post-tensioned cables

ID	Strand type	Strand No.	Jacking stress	Jacking side	Curvature c.	Wobble c.
[-]	[-]	[-]	[N/mm²]	[-]	[-]	[1/m]
P.1.PTC.1	Y1860S7-15,7-F1-C1	3	1488.0	Start	0.05	0.007

Anchorage set slip	Elastic shortening loss	Creep stress loss	Shrinkage stress loss	Relaxation stress loss
[mm]	[N/mm²]	[N/mm²]	[N/mm²]	[N/mm²]
6.0	0.0	0.0	0.0	0.0

Post-tensioned cables secondary data

ID	Strand type	Strand No.	f pk	Stress T0 S	Stress T0 E	Stress T0 Avg
[-]	[-]	[-]	[N/mm²]	[N/mm²]	[N/mm²]	[N/mm²]
P.1.PTC.1	Y1860S7-15,7-F1-C1	3	1860.0	1319.5	1341.7	1330.6

Stress T8 Avg	Min. radius of Curvature	Length (projected)	Length (real)	Length (all strand)	Volume
[N/mm²]	[m]	[m]	[m]	[m]	[m3]
1330.6	27.719	8.000	8.026	24.077	0.004

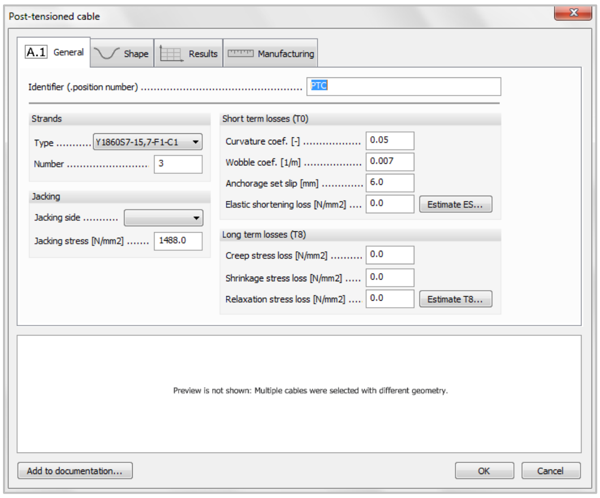
Mass
[t]
0.028

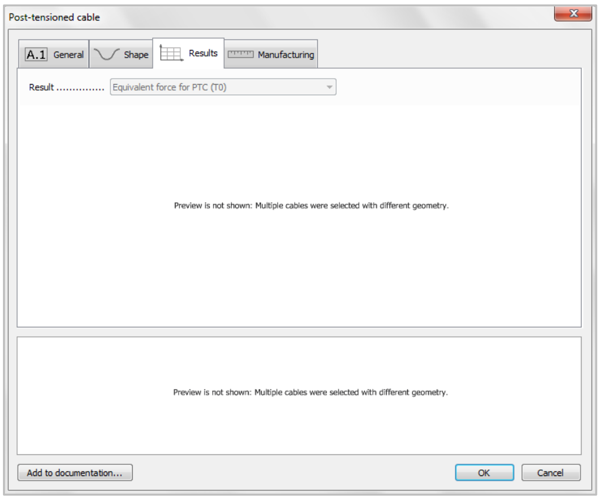
Post-tensioned cable manufacture table

ID	x' shift	z' shift	Point ID	x'	z'
[-]	[mm]	[mm]	[-]	[m]	[mm]
P.1.PTC.1	0	100	1	0.000	240
			2	1.000	122
			3	2.000	32
			4	3.000	-22
			5	4.000	-40
			6	5.000	-22
			7	6.000	32
			8	7.000	122
			9	8.000	240

Handling multiple PTCs properties

If the PTC-s have the same settings, length and shape, all features of properties dialog are available, otherwise only the common properties appear.





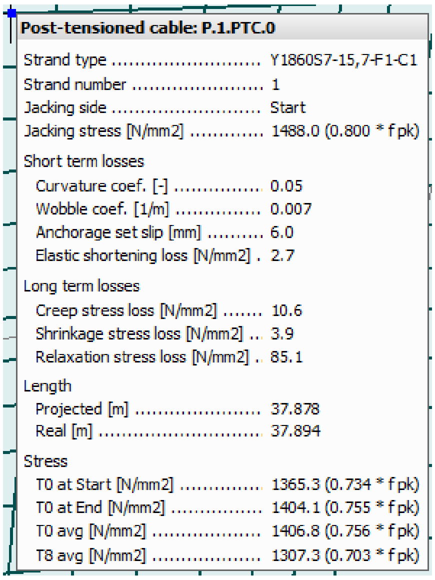
Add to documentation... and *Manufacture tab/Export...* functions are available at multiple selection and they document all selected cables at the same time.

Others

Post-tensioned cable option is added to the Colour schema. Available colour modes: ID, Strand type, Strand number, Jacking side.

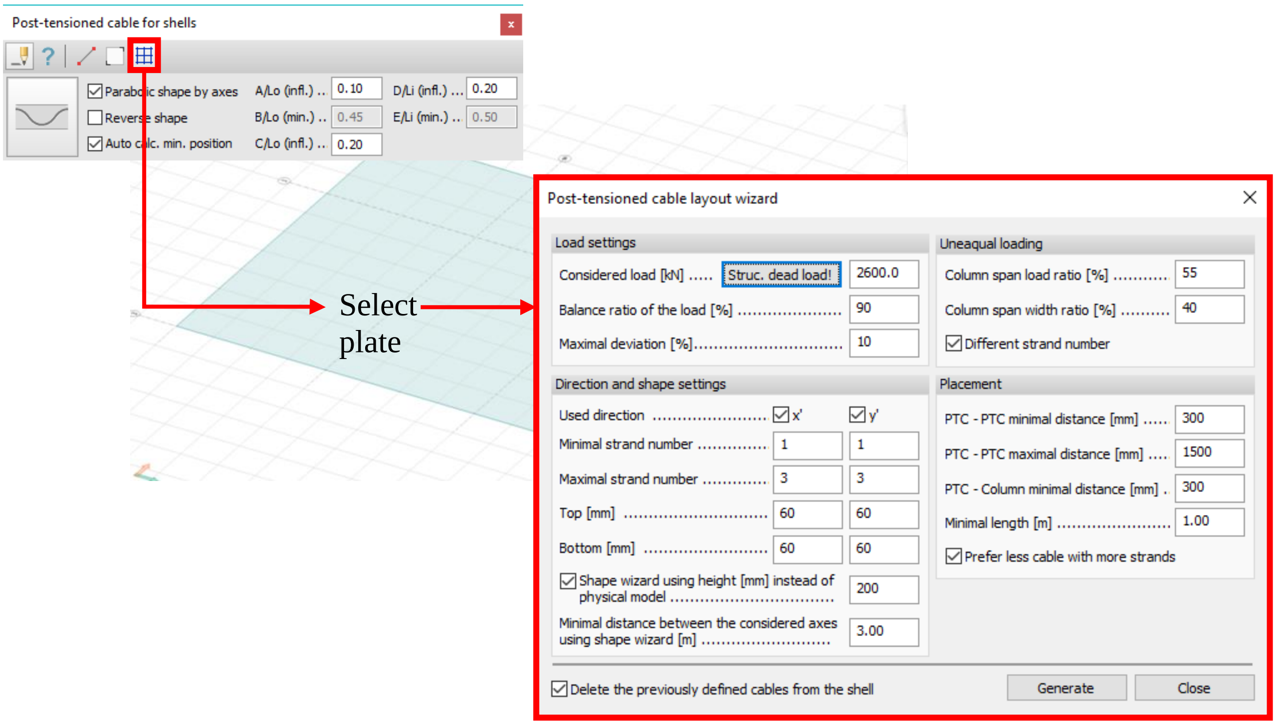
PTCs can be selected by several option of the *Filter*: *Structural element, Identifier, Strands*

PTCs have detailed tooltips supplemented by length and Stress status (connected chapters from the EN 1992-1-1: 5.10.2.1 (1) and 5.10.3 (2)).



3.8.2. Layout wizard

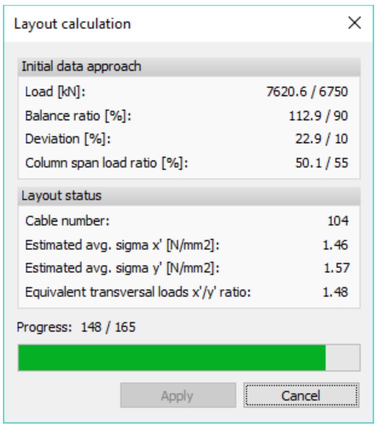
It is a parametric tool to create a set of PTCs in a specific layout at the same time. It can be launched from *Structure/Shell component/Post-tensioned cable/Layout wizard.*



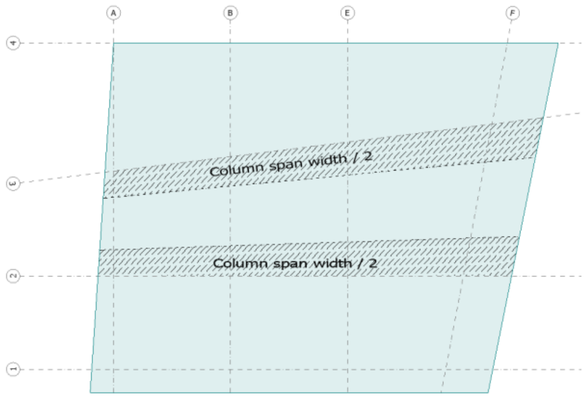
The *Layout wizard* searches the closest solution to the *Unequal loading* settings from the PTC layout variations which fulfilled the following conditions:

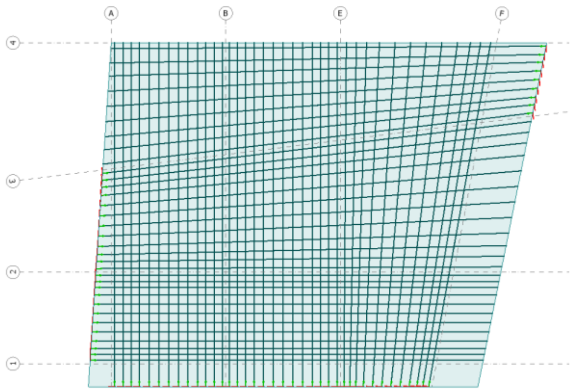
- difference between the sum of the PTC's forces acting in local z+ direction and the product of *Considered load* and *Balance ratio* is less than the *Maximal deviation*
- geometrical requirements

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.





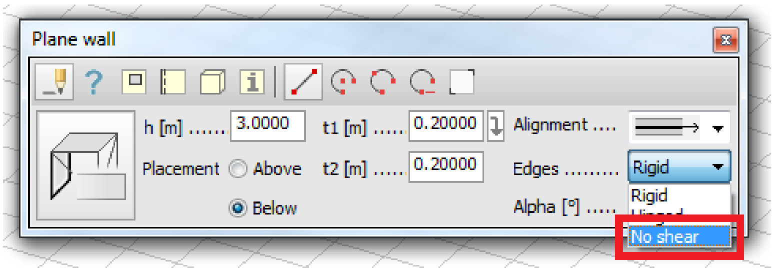
The algorythm can handle the holes of the plate regions.

At least two *Axes* crossing the plate are needed to use this function.

Checking *Shape wizard using height [mm] instead of physical model* option could highly reduce the runtime if the selected plate's thickness is constant.

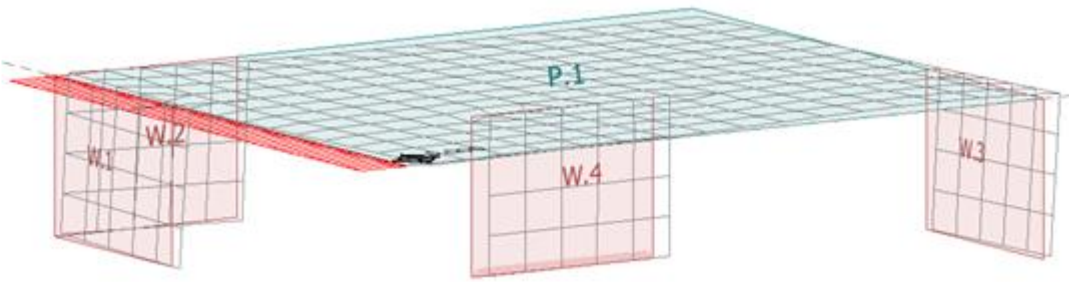
3.9. "No shear" edge macro for Plane walls

"*No shear*" macro has been added to the *Edge* list of *Plane wall* edit tools. Using it defines edge connection with predefined "*No shear*" rigidity type on the defined wall's bottom edge.

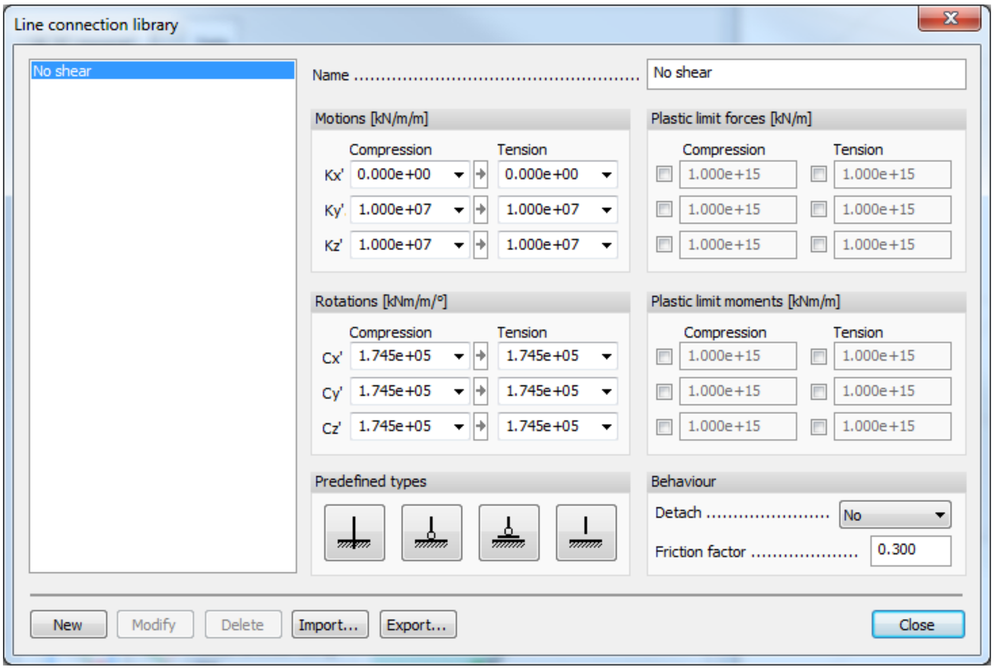


It is the recommended wall definition method to create a non-bracing system element.

The following example shows a displacement graph of a multi-storey building where W.4 wall is not part of the bracing system.

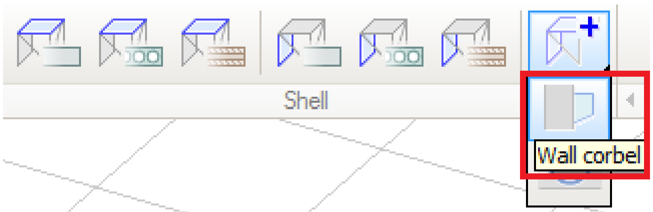


"*No shear*" rigidity type settings with free local x' motion rigidity:

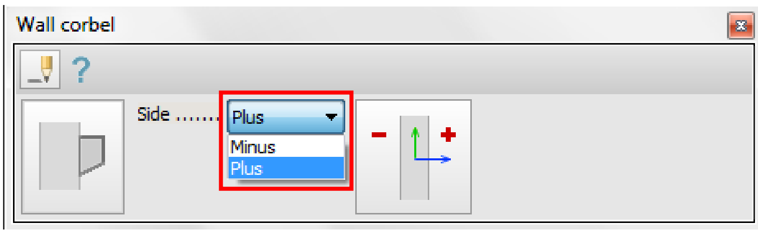


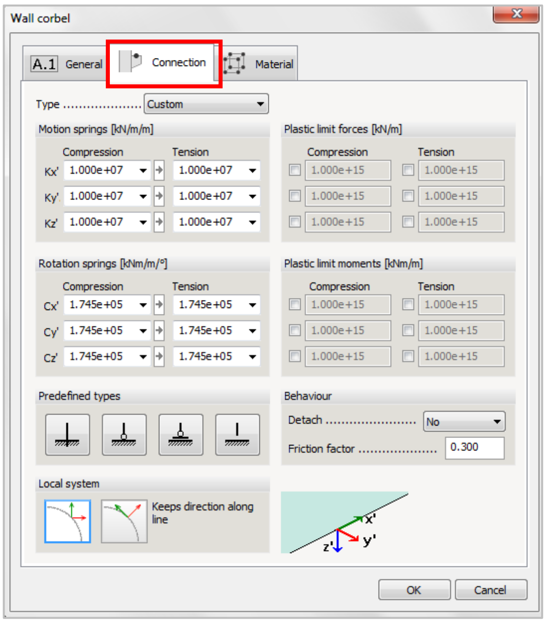
3.10. Wall corbel

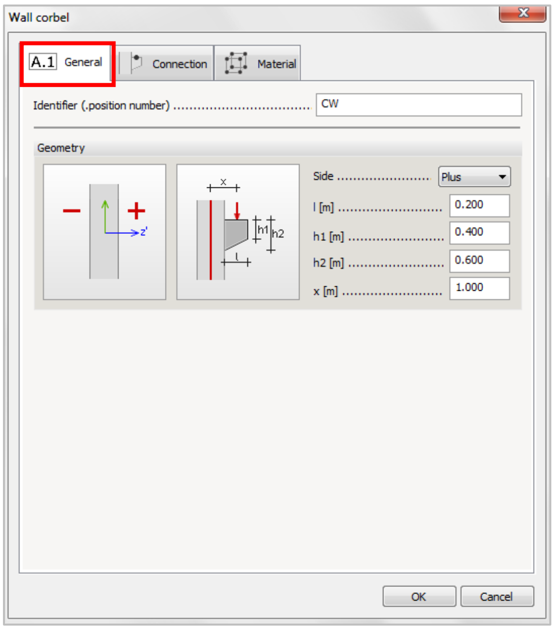
Wall corbel is a new structural element. It can be found in the Structure tab/Shell component/Wall corbel.

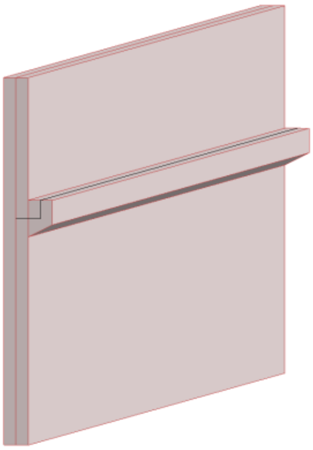


In the tool window it can be set whether the corbel should be adjusted to the positive or negative side of the wall according to its local coordinate-system.









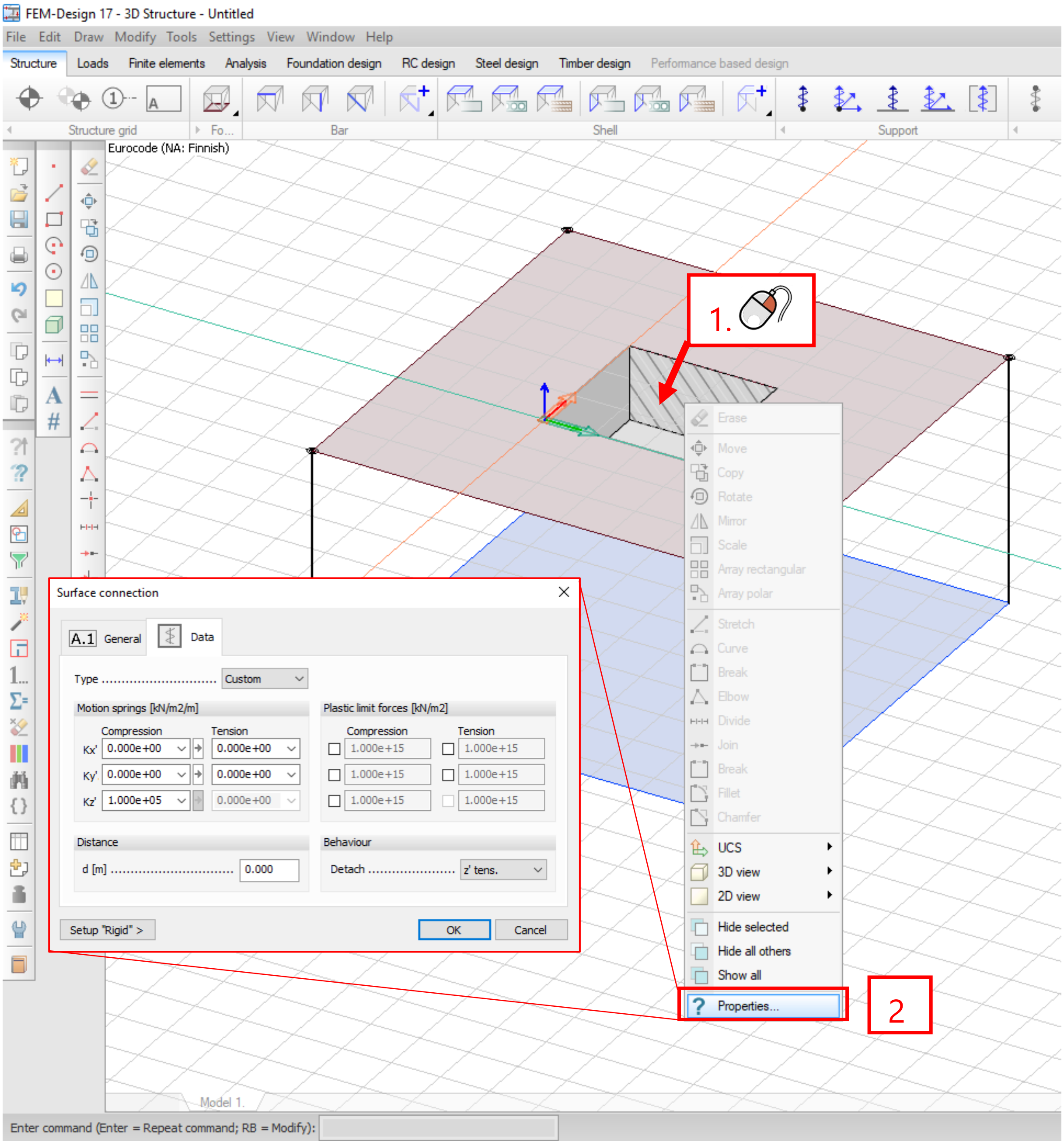
⚠ Wall corbel can be placed on **Plane wall** only.

3.11. Editable surface connection (soil)

Stiffness properties of automatically generated surface connections are editable.

The example shows a settled foundation slab into soil.

Right click on the surface connection, then select *Properties* (see chapter 2.1.).



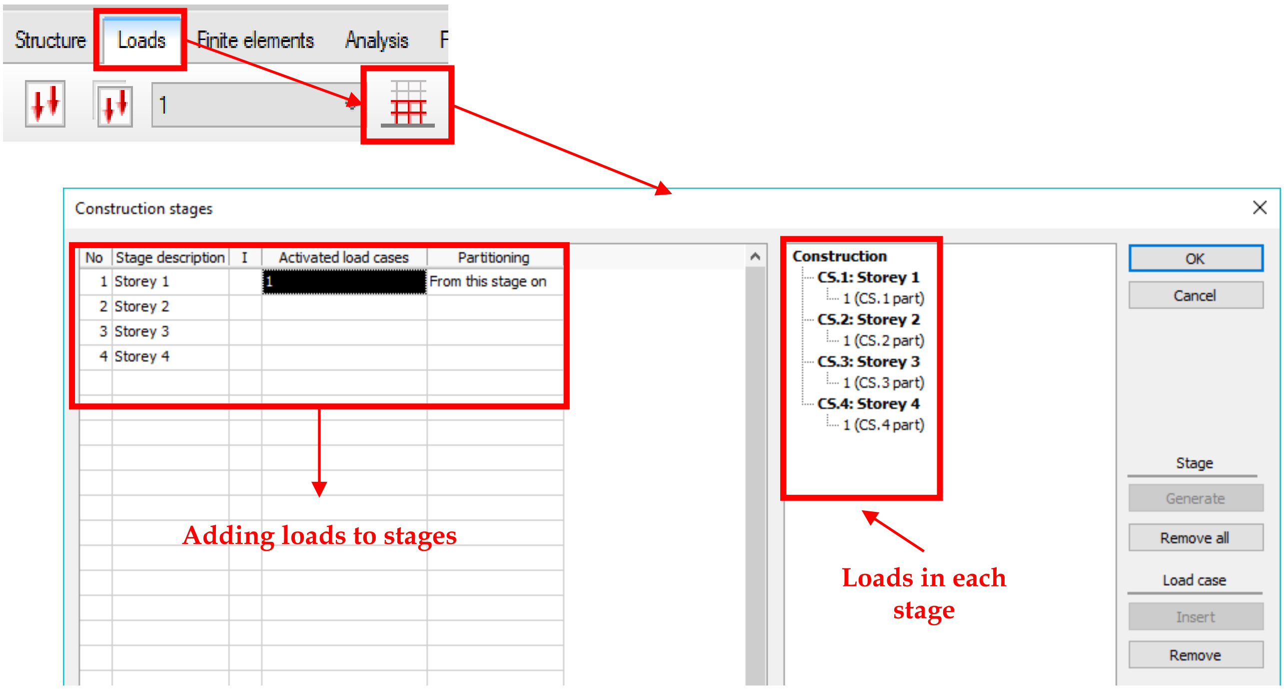
4. Load

4.1. Construction stages

The Construction stages function helps to model and design the different building phases, and it helps to determine the construction process affected displacement/internal force distribution of the finished structure. **Currently we implemented this feature for high-rise buildings, making the structural element separation by the storey system, but it will be generalized to a user-defined stage system in the near future.**

Storeys must be defined to use Construction stages, since currently one storey is considered as one Stage.

Construction stages can be defined under the load tab. Click "Generate" button to create stages with the correspondent storey name.



Columns in the table mean the following:

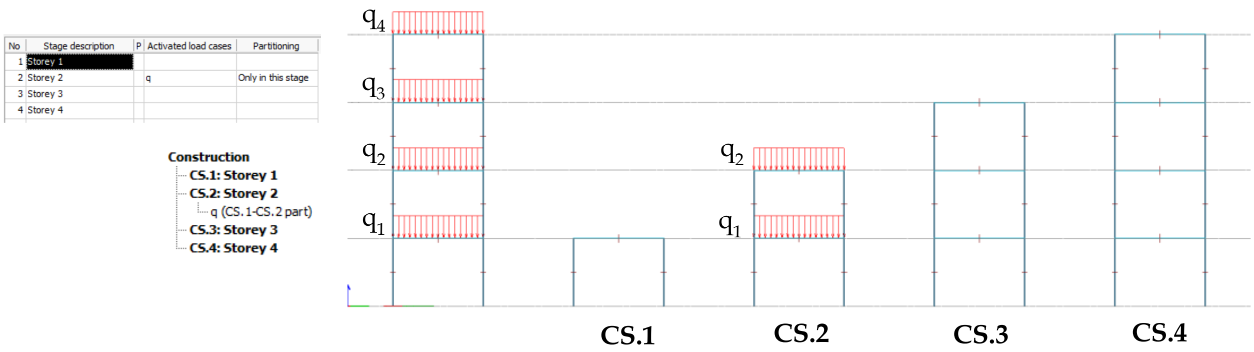
- *No*: Number of the stage
- *Stage description*: Name of storey which is built in the stage
- *I* (*Initial stress*): if checked, in calculation of the stage displacements will be reset to zero, but other results (internal forces) are accumulated
- *Activated load cases*: Activated load cases for the stage
- *Partitioning*:

  This defines for each load from the load cases that how and when it is activated during the construction stage calculation.

  - *only in this stage*: Load case is activated in the specified construction stage and acts only in this stage.

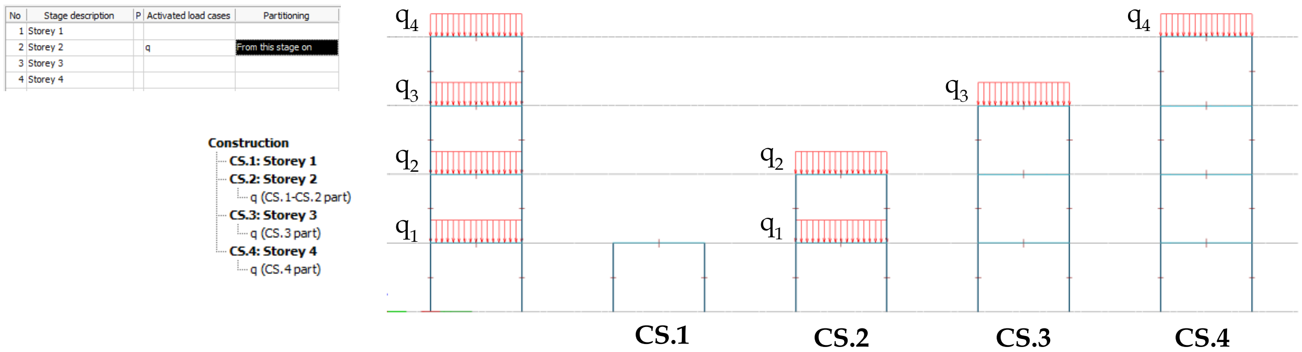
Remaining loads from these load cases will not activate in other stages.

Only in this stage (applied in 2nd stage)



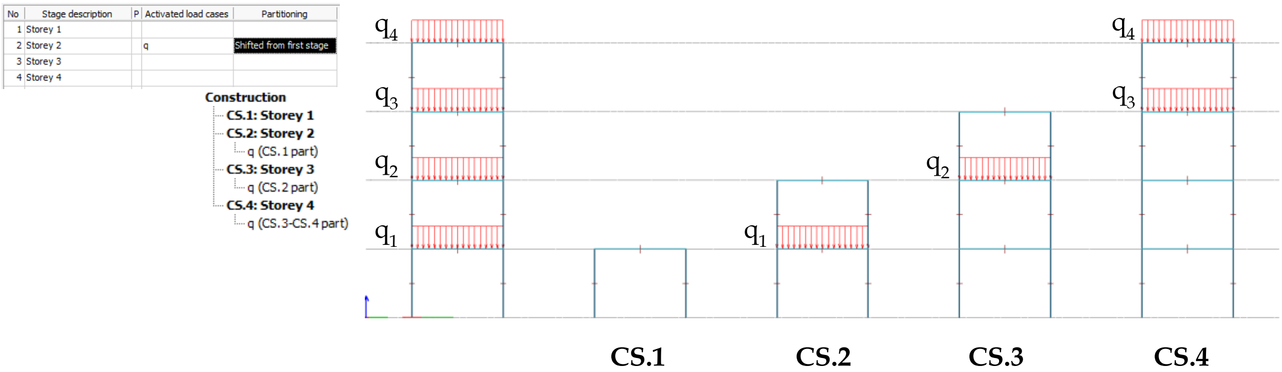
- *from this stage on*: Load case is activated in the specified construction stage and acts in this and also in the remaining stages – the parts of the loads that act on the previous storeys are also activated in the specified stage

From this stage on (applied in 2nd stage)



- *shifted from first stage*: Load case is activated in the specified construction stage and acts in this and also in the remaining stages – the parts of the loads that act on the first storey will act in this specified stage and the remaining parts of the loads that act on the second storey will act in the next stage and so on (e.g.: flooring and covers)

Shifted from first stage (applied in 2nd stage)



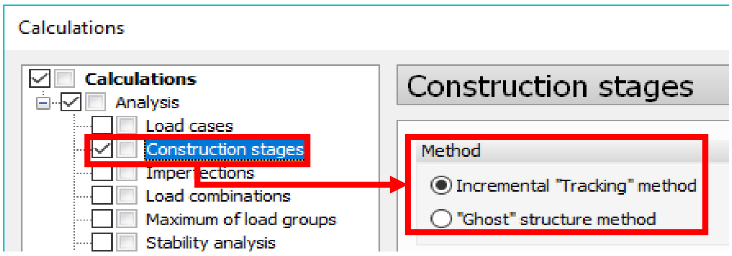
It is possible to add any construction stage to any load combination with the following limitations:

- Only one construction stage is allowed in one combination.
- Cannot combine a construction stage and a load case which is already activated in a construction stage
- The fire and/or seismic load cases can be combined with only the final construction stage

For load groups only the final construction stage can be added too.

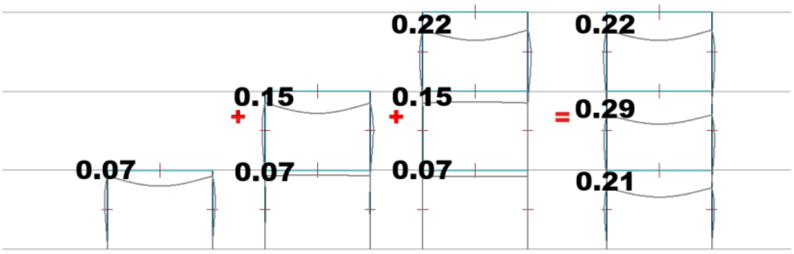
The construction stages automatically follow all storey modifications.

User can start the construction stage calculation at Analysis/Calculation/Construction stages. There are two calculation methods, so called *Incremental "Tracking" method* and *"Ghost" structure method.*

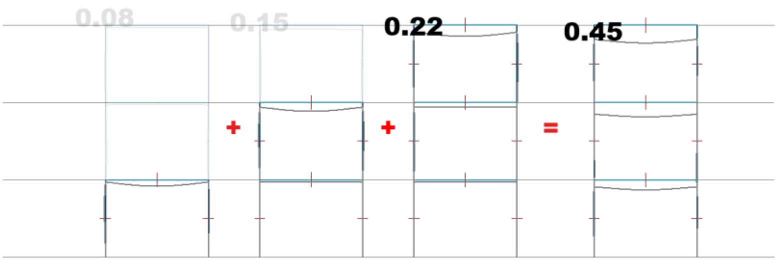


When incremental method is chosen, the model is built stage-by-stage. In case of "ghost" structure method the full structure is in the calculation, but stiffness of those structural parts which aren't in the specific stage is highly reduced.

Incremental "Tracking" method

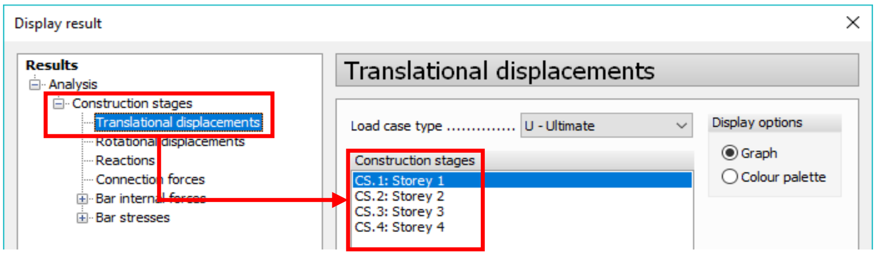


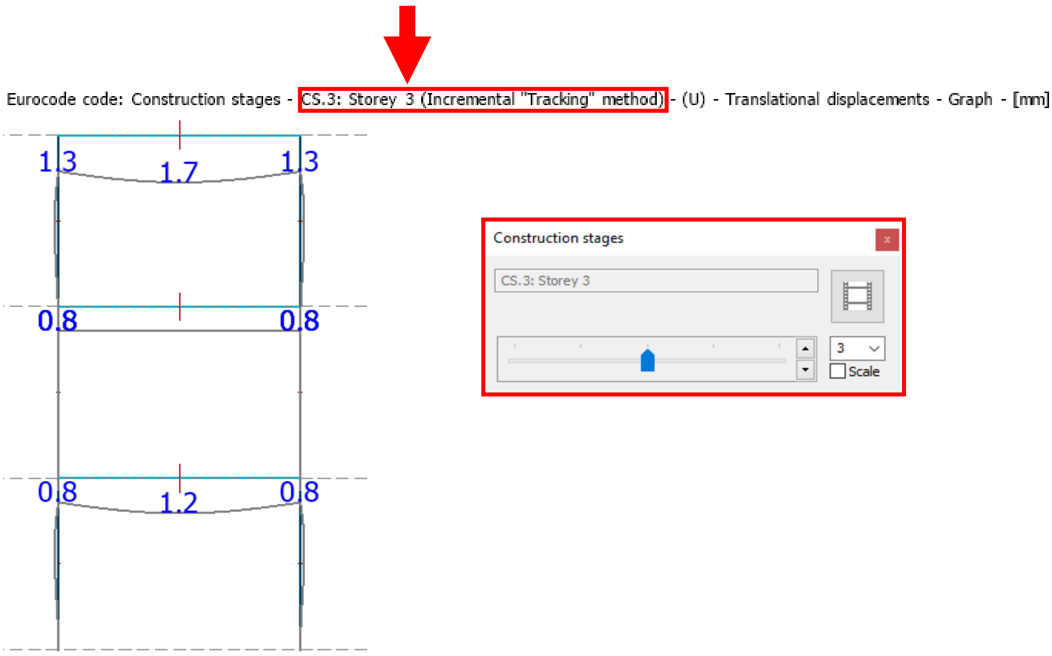
“Ghost” structure method



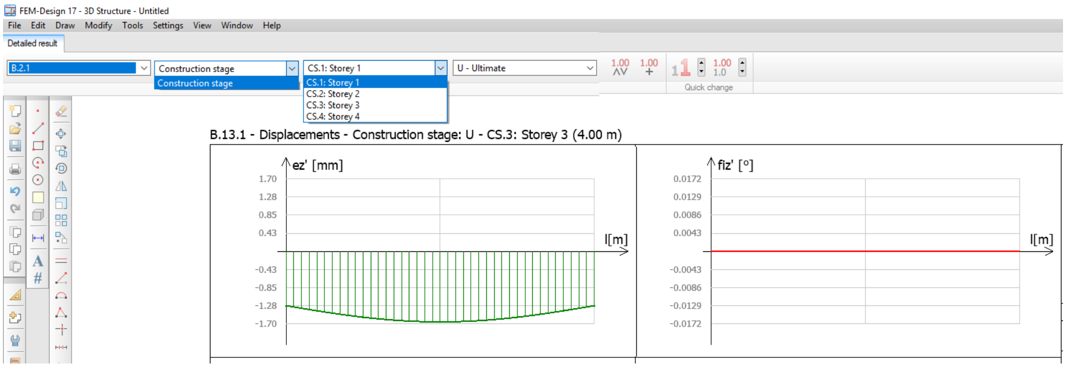
The construction stage results can be found in the New results/Analysis/Construction stages. For every stage result the method name and the displayed construction stage (e.g. CS.1 Storey 1) appears in the information panel.

Adding any new Construction stage result will open a "Construction stages" (result-display) tool to make easier the navigation between the stage results and to animate the Construction process if it is needed.

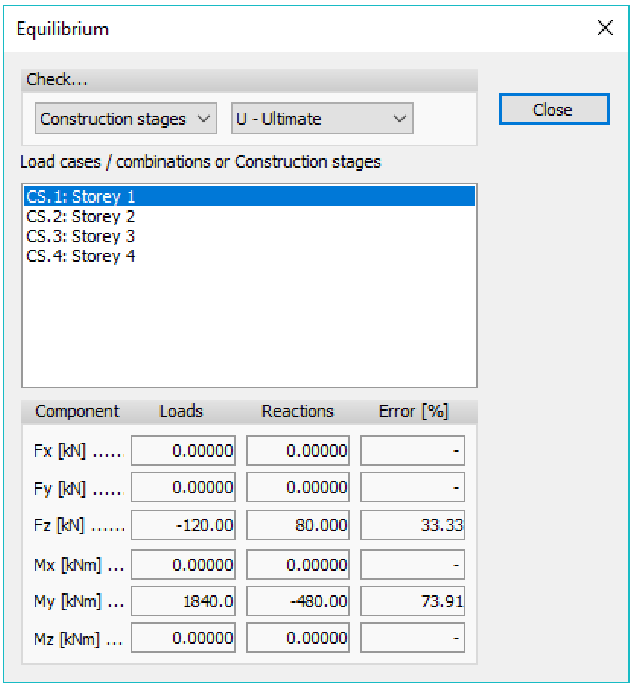




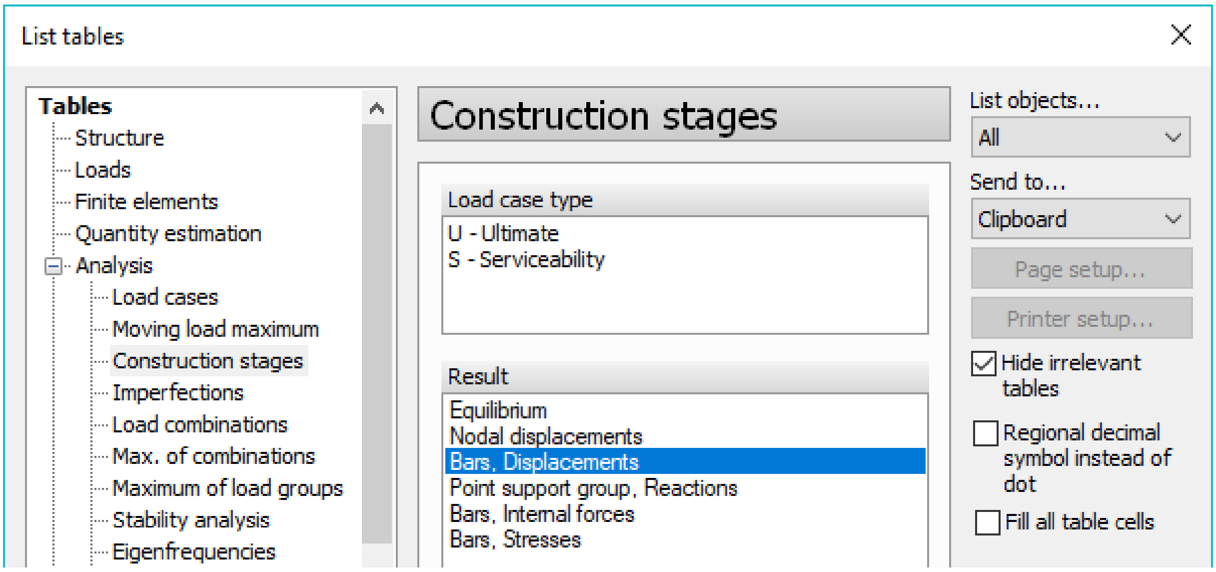
It’s also possible to choose the construction stage in detailed results.



The equilibrium dialog contains the construction stages, too.



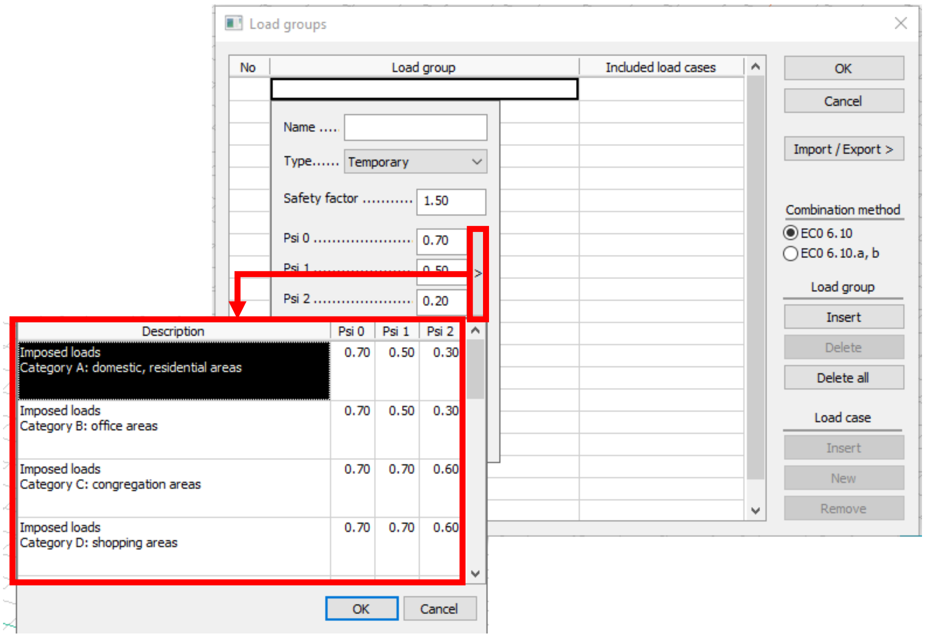
It’s possible to list the construction stages result, which can be found under Analysis/Construction stages.



List tables
Tables
Structure
Loads
Finite elements
Quantity estimation
Analysis
Load cases
Moving load maximum
Construction stages
Imperfections
Load combinations
Max. of combinations
Maximum of load groups
Stability analysis
Eigenfrequencies
Construction stages
Load case type
U - Ultimate
S - Serviceability
Result
Equilibrium
Nodal displacements
Bars, Displacements
Point support group, Reactions
Bars, Internal forces
Bars, Stresses
List objects...
All
Send to...
Clipboard
Page setup...
Printer setup...
Hide irrelevant tables
Regional decimal symbol instead of dot
Fill all table cells

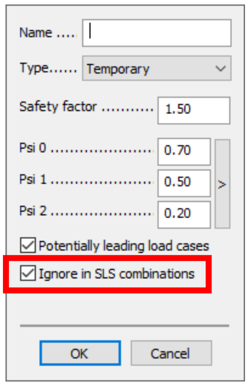
4.2. Pre defined Psi values for temporary load groups

There is an option for temporary load groups to choose predefined ψ_0, ψ_1 and ψ_2 values.



4.3. Ignored temporary load cases

Temporary load groups has an option to ignore in SLS combinations in Load group maximum results and in Generating load combinations by load groups.



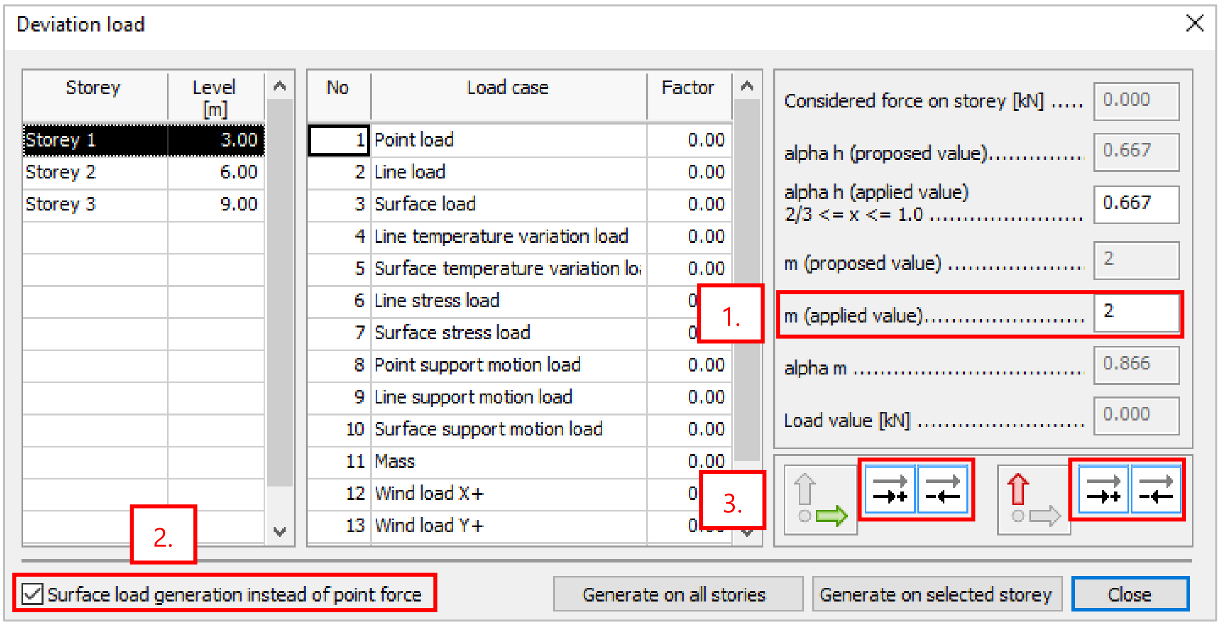
4.4. Deviation load improved

There are changes in deviation load macro.

- User can set the value of "m" instead of "alpha m", which is calculated automatically from "m". The applied values can be given for each storey, which will be stored.
- A checkbox has been implemented, which allows generation of surface load instead of point load.

- Loads can be defined in positive and/or negative direction, which can be applied with the *plus/minus buttons* next to the icons , in the bottom-right corner of the dialog.

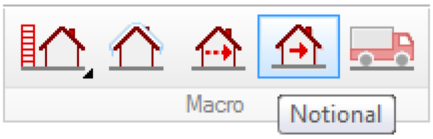
⚠ As many load cases will be generated as many buttons are checked.

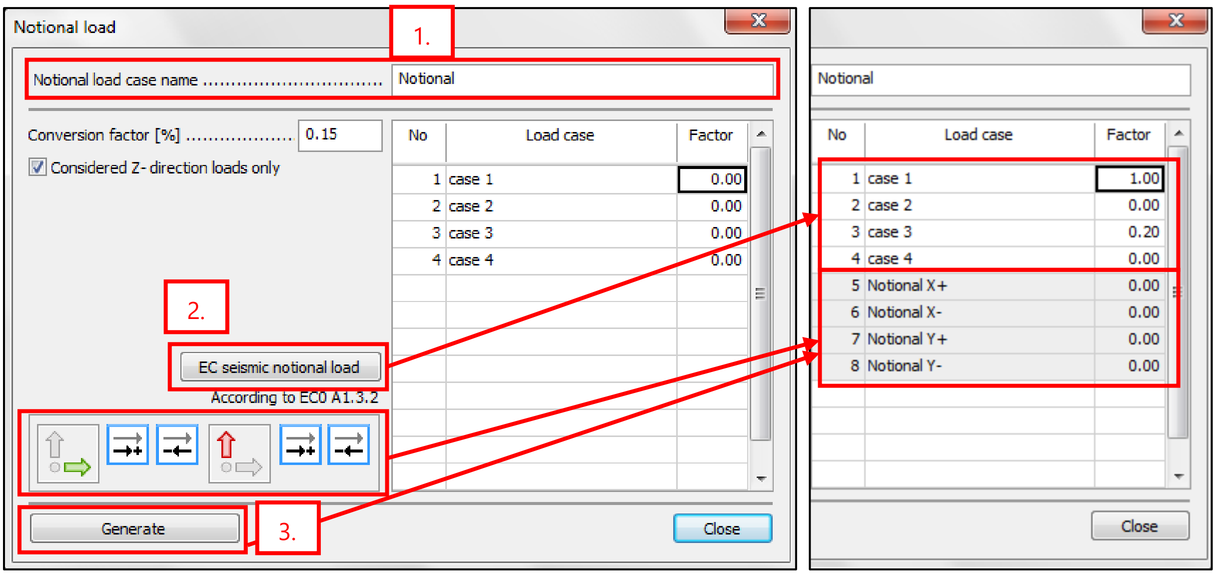


4.5. Notional load

This function generates horizontal load from non-horizontal loads using a conversion factor.

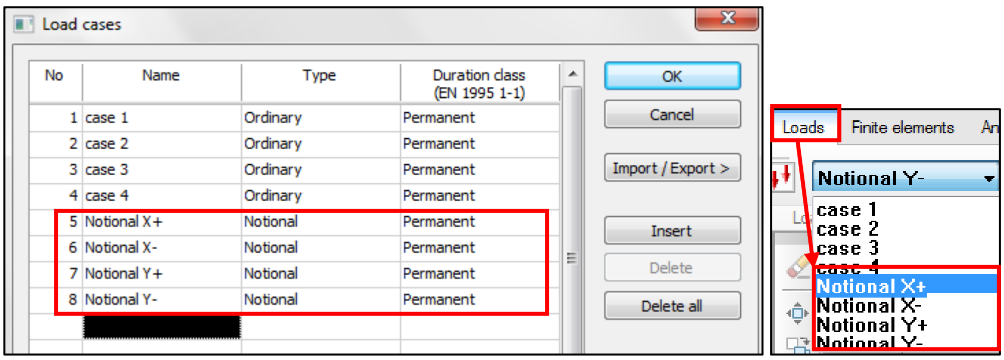
It can be reached in *Load tab/Macro/Notional load.*





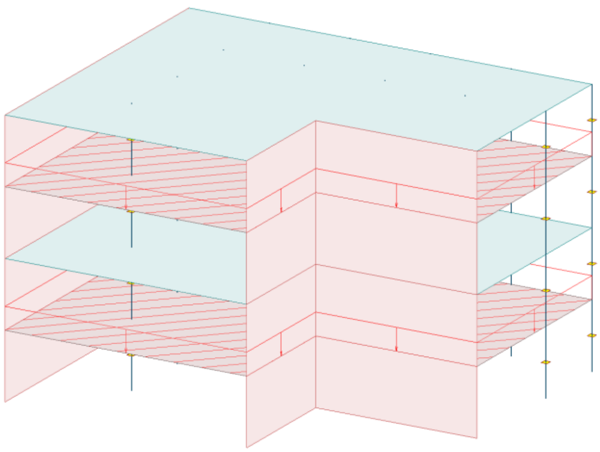
- For the notional load case a name can be given by typing it into the textbox on top of the dialog
- After clicking *EC seismic notional load* button, Load case factor will be set according to parameters of Load groups the Load case belongs to. It uses EN 1990-1-1 A1.3.2 DK NA: $A_d = 1.5\%(\sum G_{k,j}" + "\sum_{i\geq 1} \psi_{2,i} Q_{j,i})$
- After clicking on the *Generate button*, as many *Notional Load cases* will be created, as many directions are chosen above the *Generate button*

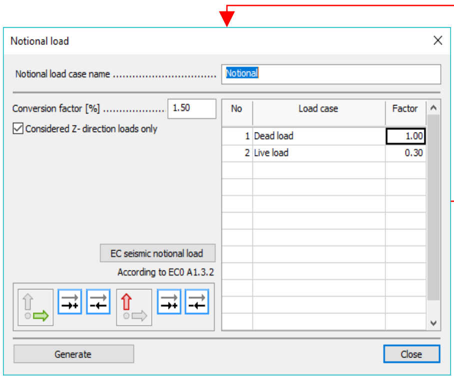
The generated *Notional Load Cases* appear in the *Load Cases dialog* and they can be selected from *Loads tab/Current load case* drop-down list.

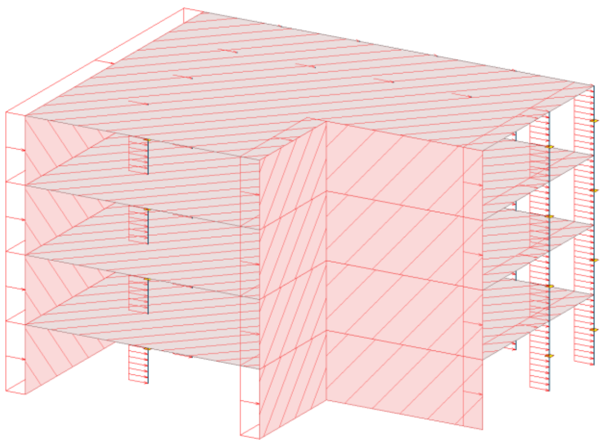


The process of the load generation by *Notional load* macro is shown on the picture below.

No	Name	Type	Duration class (EN 1995 1-1)
1	Dead load	+Struc. dead load	Permanent
2	Live load	Ordinary	Permanent



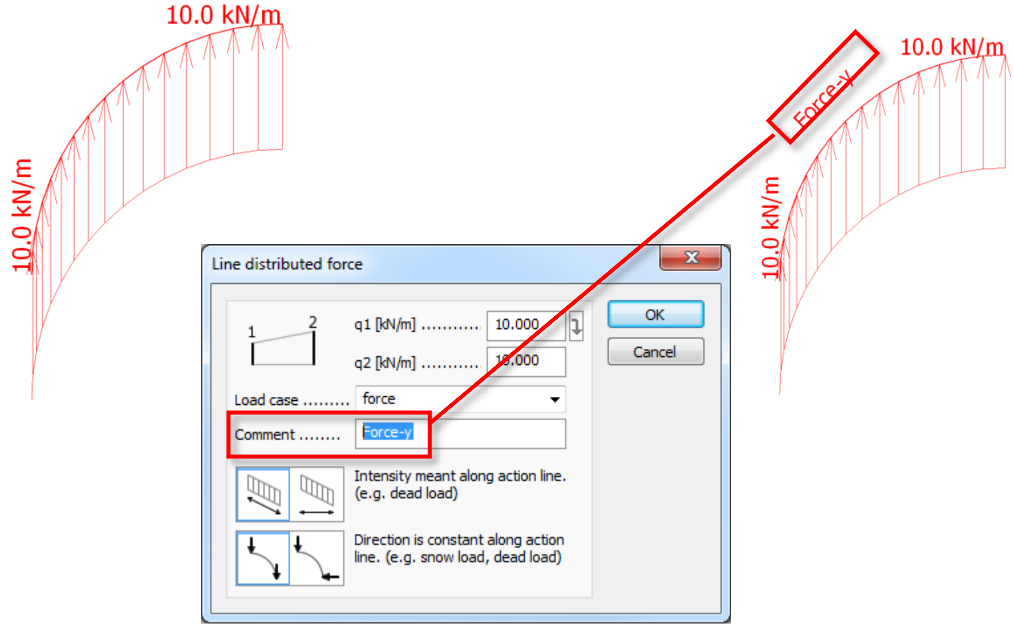




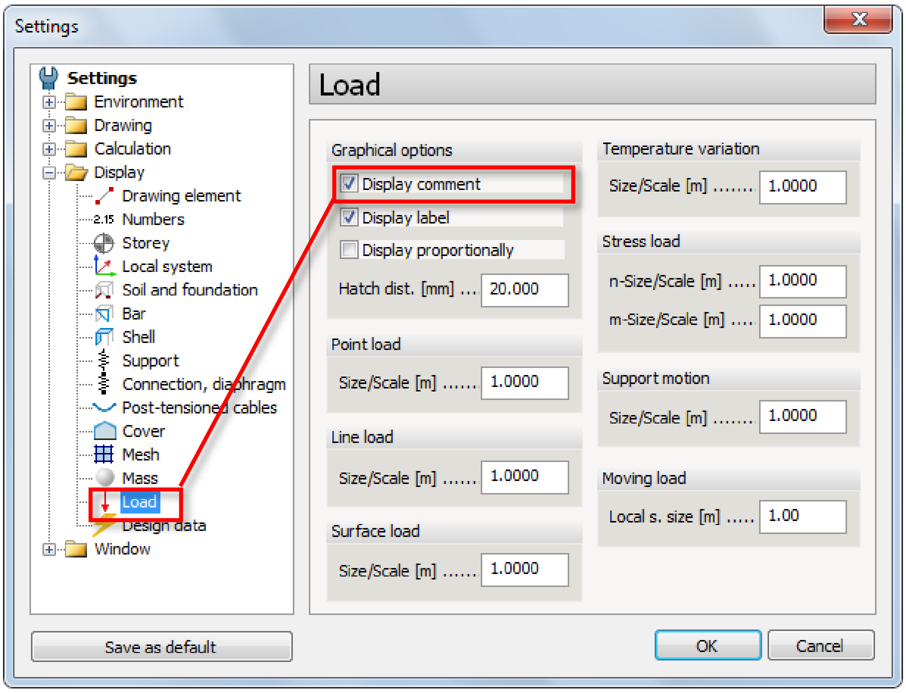
4.6. Load comments

A new property – *Comment* - has been added to all load types. With this new feature the User can label every single load in order to easily identify them.

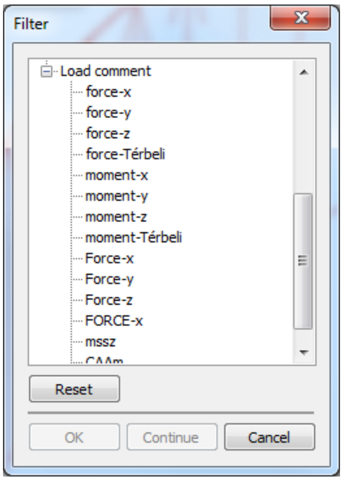
The comments can be set for every load types in the Default settings/Properties dialog.



Load comments can be turned on/off by *Settings/Display/Load* dialog's *Display comment* option.



Load comments appear in the Filter dialog and also in the listed load tables.

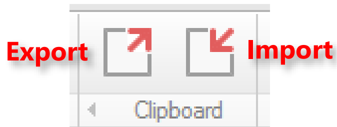


Point loads

No.	F	M	Load case	Comment
[-]	[kN]	[kNm]	[-]	[-]
1	10.000	0.000	force	force-x
2	10.000	0.000	force	force-y
3	10.000	0.000	force	force-z

4.7. Load export / import via clipboard

Load Export and *Import via clipboard* is implemented, to let 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.

Editable columns

#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*.

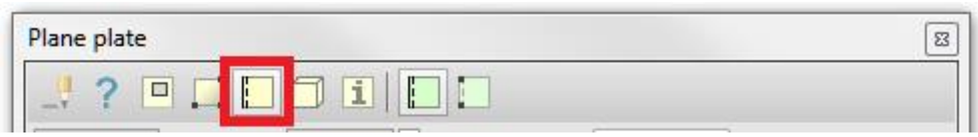
If the User exported constant surface load, only changing the first intensity value will have effect on the surface load.

5. Analysis

5.1. Plastic trusses, supports and connections

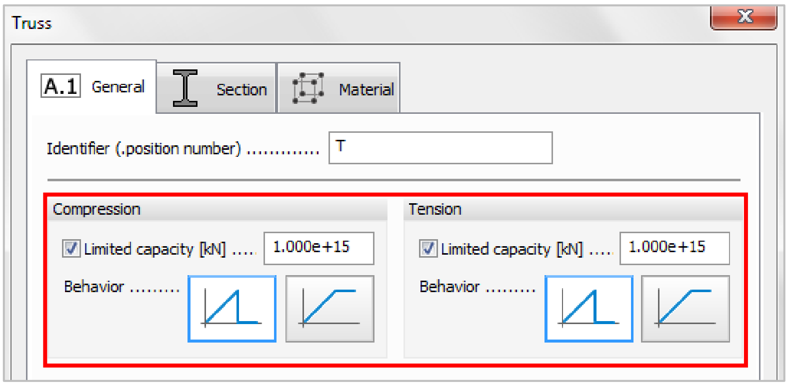
Plastic calculation has been implemented for trusses, supports and connections.

It is also available for edge connections of all shell elements (Plane plate and wall, Profiled plate and wall, Timber plate and wall, Fictitious shell)



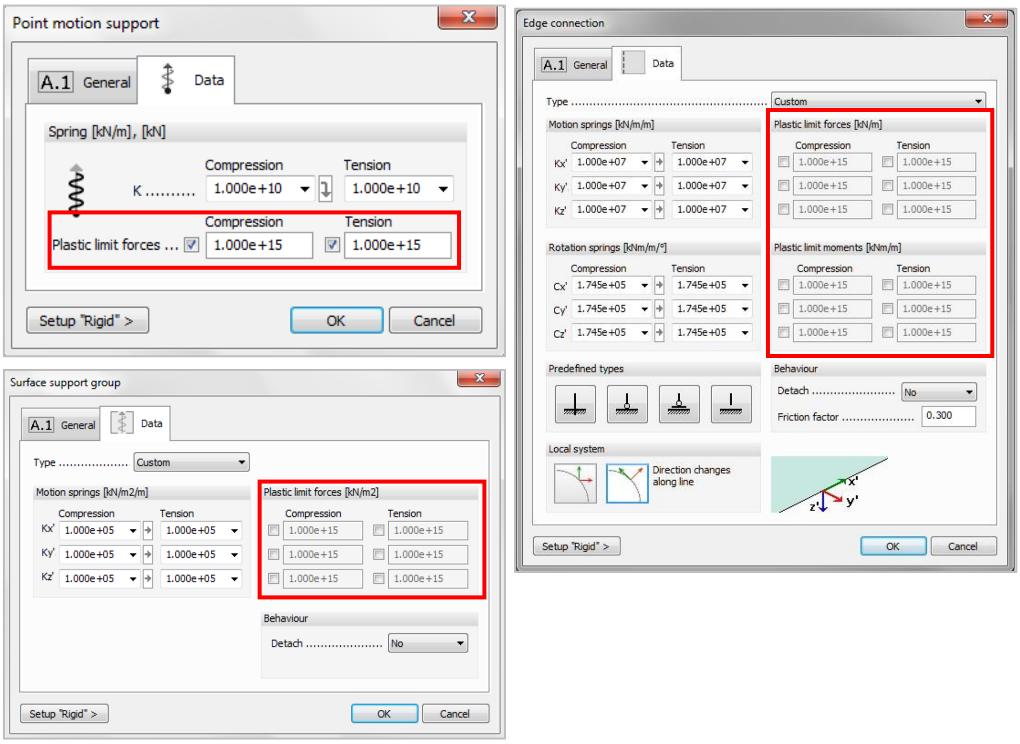
The plastic data can be set in the *Default settings/Data tab* of each above mentioned options. The figures show where the feature is placed in the three types of dialogs of these items.

Some examples with detailed calculations can be found in the verification book.



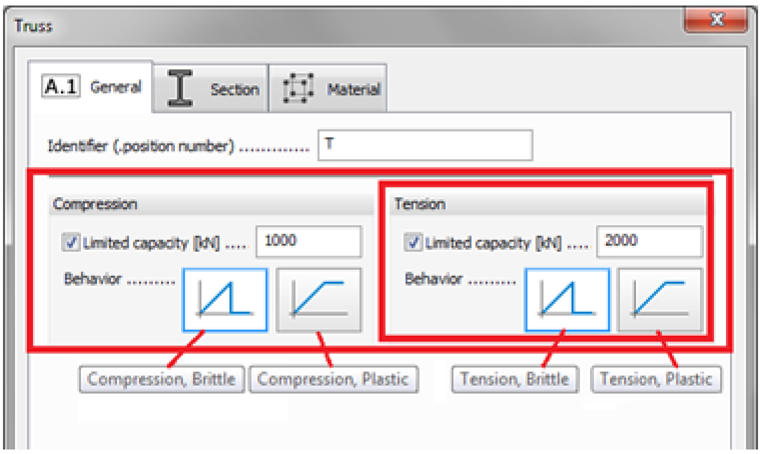
The options above are considered only for load combinations calculated as non-linear elastic. *Plastic* behaviour is considered for load combinations calculated as non-linear elastic + plastic. See more details in the next chapter.

For further information check the documentation.



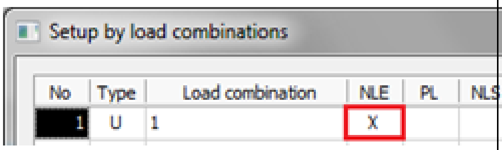
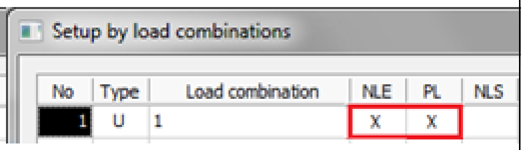
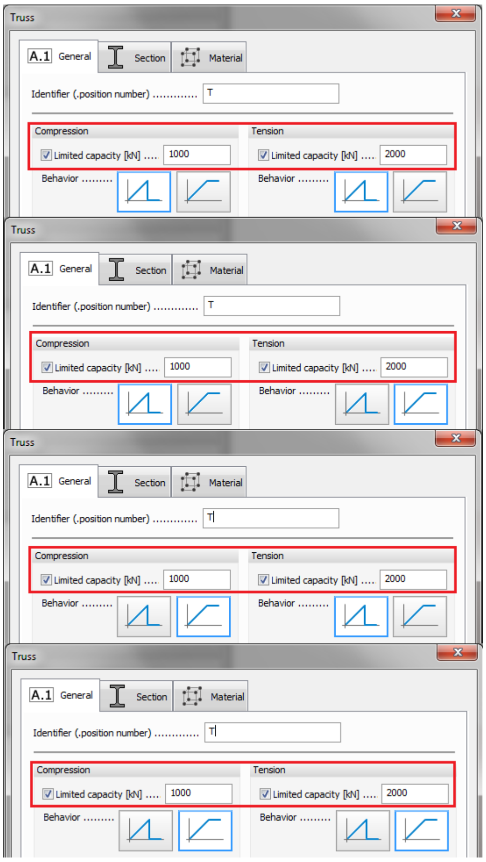
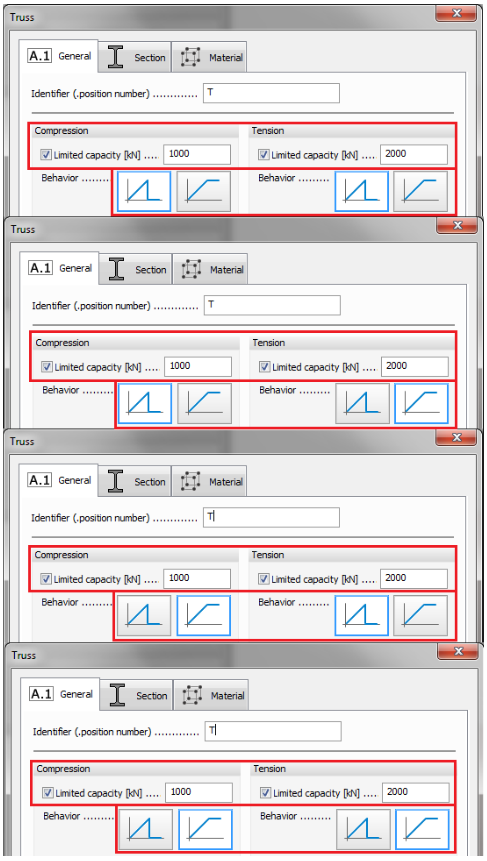
5.2. Modified behaviour of trusses in non-linear elastic (NLE) and non-linear elastic + plastic (PL) calculations

Limited tension capacity is available for trusses from now on. In the former versions only limited compression capacity was implemented.



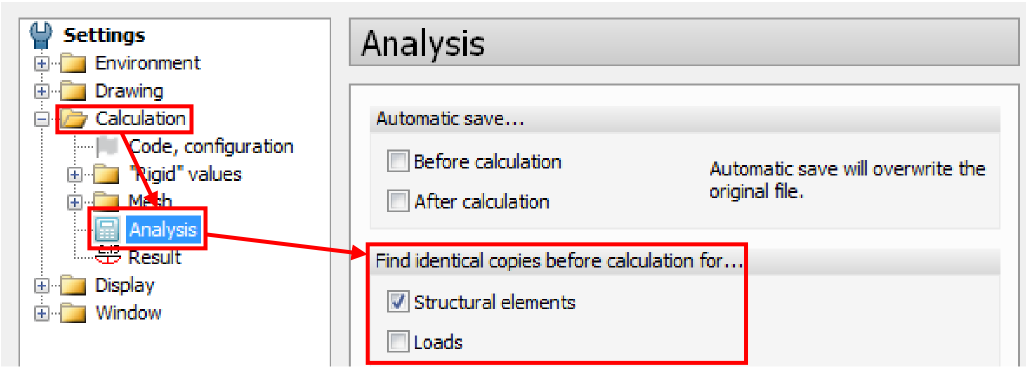
In the earlier versions of FEM-Design in case of NLE (so-called uplift) calculation, if maximal compression capacity of a truss was set to zero, and in the first iteration step compression arose in the truss due to the external loads, the calculation method did not allow tension to arise during the further iteration steps, even if it was theoretically possible. **Now it has been implemented that compression and tension are changing during the iteration in case of NLE** calculation.

However the actual effect of the two behaviour options (brittle or plastic) depends on the setup of the specific load combination.

Load combination calculation setup	NLE	NLE + PL
		
Properties dialog settings		
Behaviour	**Brittle**, independently on which behaviour is selected in the properties dialog	**Brittle or plastic**, depending on which behaviour is selected in the properties dialog

5.3. Check for identical copies of structural objects and loads

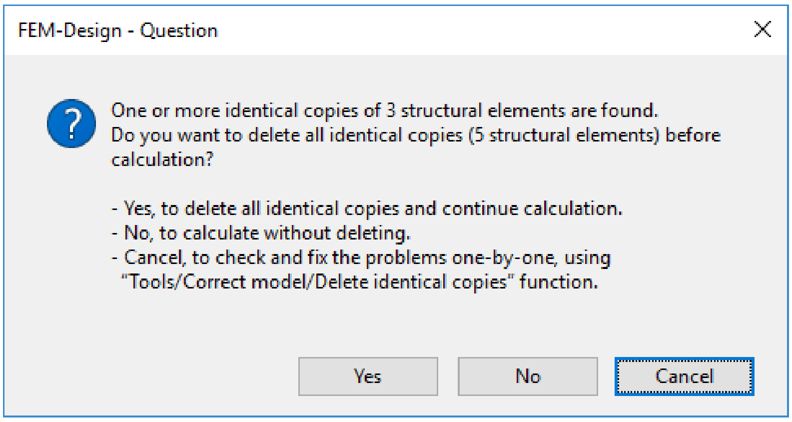
A new option is added to the *Settings/All.../Calculation/Analysis* dialog. User can set if the program should check the identical copies of structural elements and/or loads.



For example, let's take the following example for object multiplication in a model:

- *point support: 2 identical supports (1 unnecessary copy)*
- *line support: 3 identical supports (2 unnecessary copies)* → ***3 objects, 5 unnecessary copies***
- *beam: 3 identical beams (2 unnecessary copies)*

Having this option ON, during pre-processing of calculation, the following question pops up:

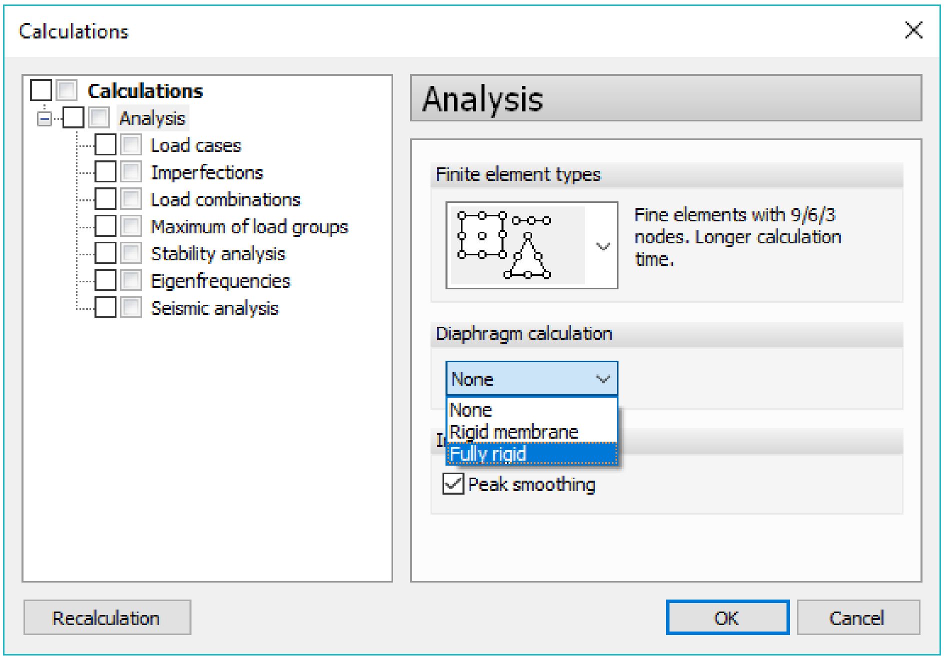


The program informs about the number of unnecessary copies. User has three options to select:

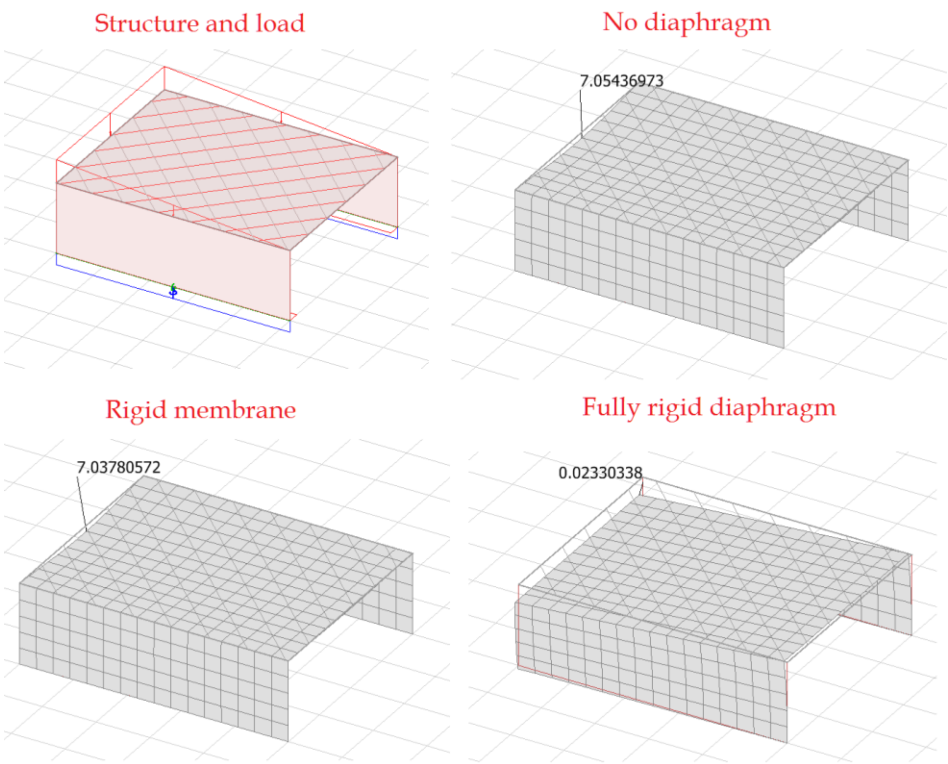
- clicking "Yes", FEM-Design will delete all unnecessary copies automatically,
- clicking "No" will ignore this check, and the calculation will runs without any modification,
- clicking "Cancel" will abort the calculation and allows User to fix the *unnecessary objects* with the new feature called *Correct model – Delete identical copies,* described in Chapter 1.1.

5.4. Fully rigid diaphragm

There is a new type of diaphragm calculation: Fully rigid.



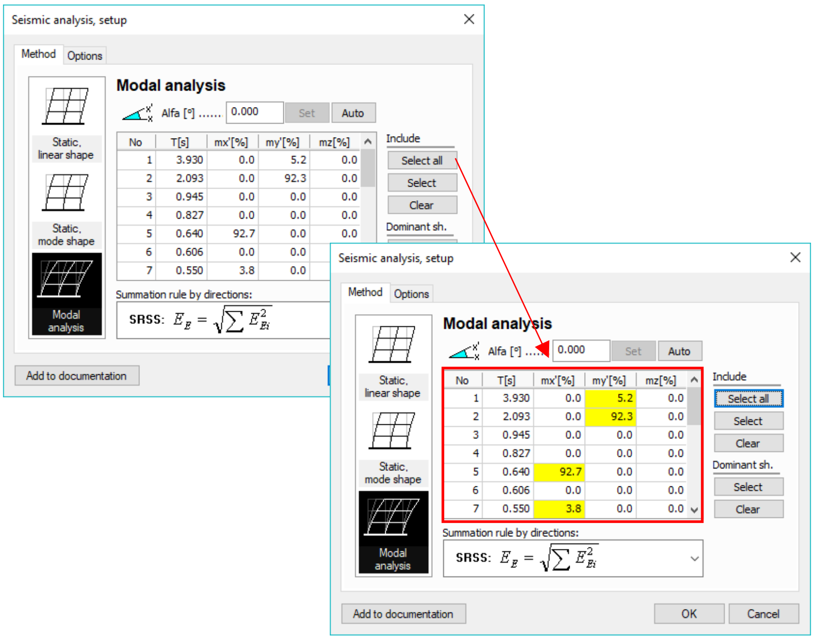
Fully rigid diaphragm will ensure the rigid movement of the all objects lying in diaphragm's region and move them all together in one plane in any direction (as rigid body). For further information check the documentation.



5.5. Selecting all relevant shapes in Modal analysis

This new feature helps to select all the relevant shapes in the Modal analysis. By clicking the "Select all" button the program selects all the shapes with non zero modal mass percentage.

⚠ This feature is not selecting the dominant shapes, it has to be selected manually.

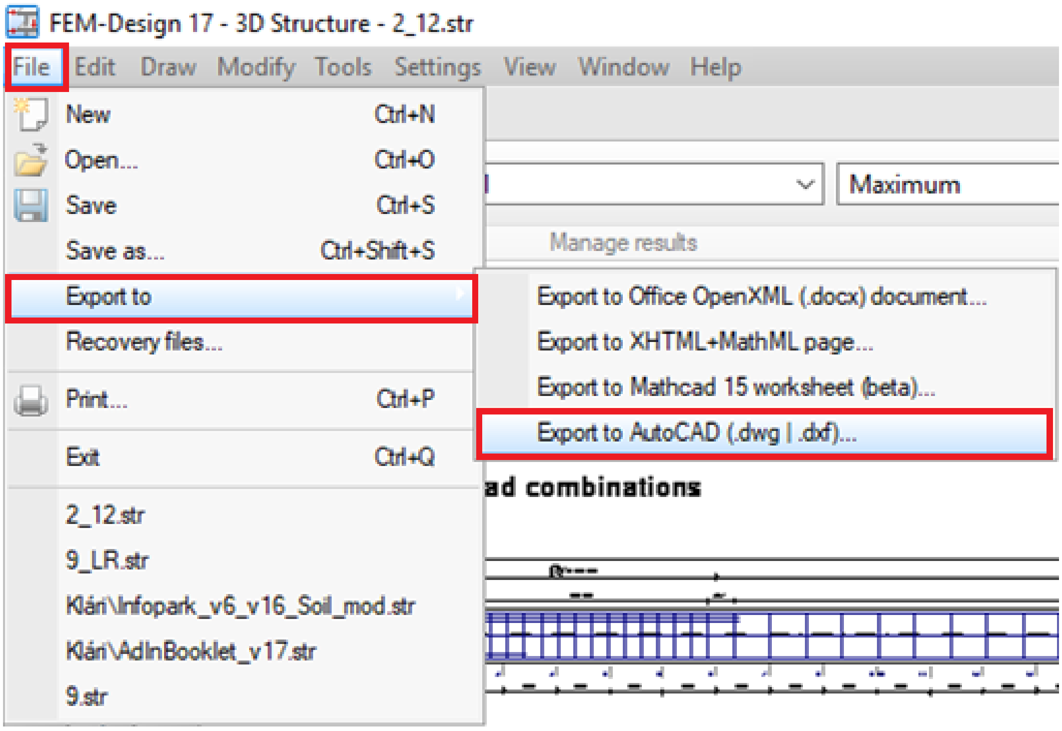


6. RC design

6.1. RC Bar detailed result reinforcement export to DWG/DXF

Now the detailed results for RC bar reinforcement can be exported into *.dwg or *.dxf file format.

This feature can be reached in detailed result window, File/Export to/Export to AutoCAD…

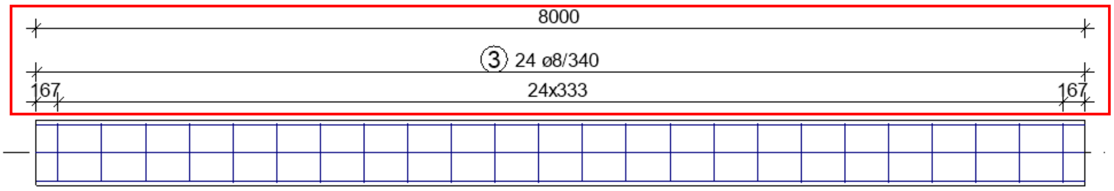


6.2. RC Bar detailed result drawing improvements

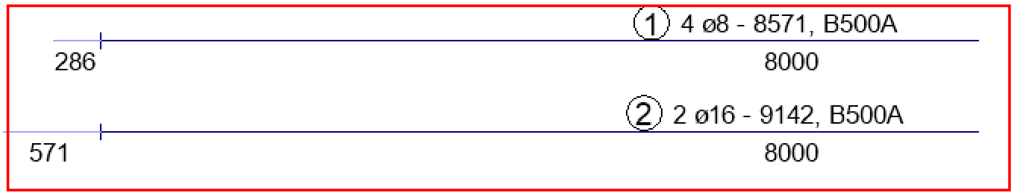
The improvements on the Reinforcement drawing are as follow:

- stirrup's dimension lines
- longitudinal bar dimension lines
- cross -section with reinforcement numbers

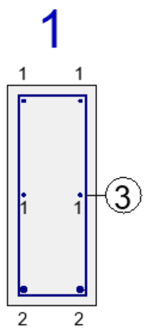
Stirrup dimension lines



Longitudinal bar dimensions



Cross-sections with reinforcement members



6.3. RC Shell buckling

For the consideration of buckling failure of walls and plates, a new checking criterion is available for RC shells, the *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.

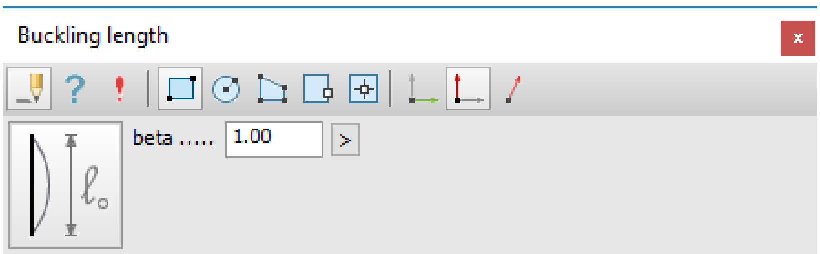
Only RC Plane plates and Plane walls with straight reinforcement and uniform thickness are suitable for shell buckling calculation.

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.

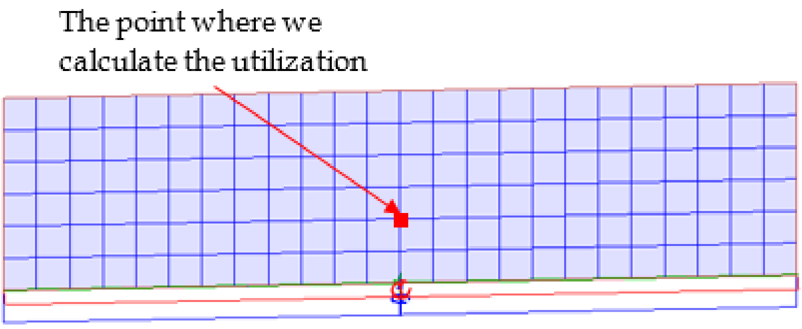
The default buckling regions can be modified by adding new regions to the shell. One shell may have more buckling regions with different beta factor and direction vector, but the shell must be completely covered by these regions.



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.

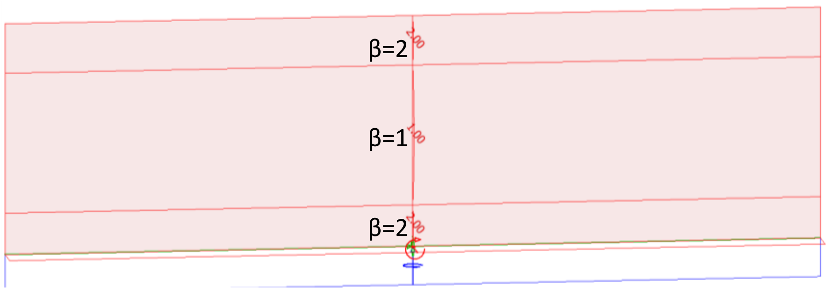
The calculation process consists of the following steps:

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).

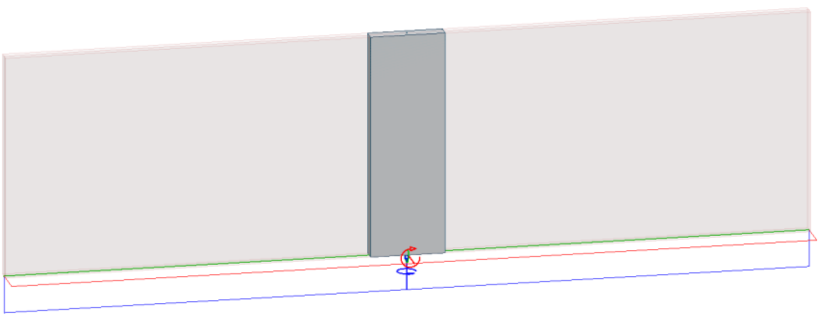


2. An equivalent bar is generated for the examined node as follows. The edges of the shell are intersected by the ray determined by the node and the direction vector of the corresponding buckling region. The two intersection points are taken as the start and the end point of the equivalent bars.

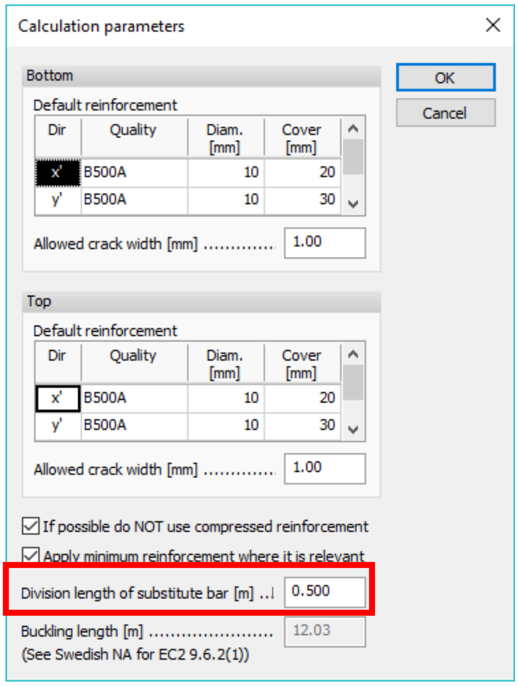
⚠ Note that this intersection is always made with the edges of the shell and not with the edges of the buckling region corresponding to the node! If a node is on the border of two or more buckling regions, it is calculated with both different beta values and direction vectors, and the higher utilization will be used.

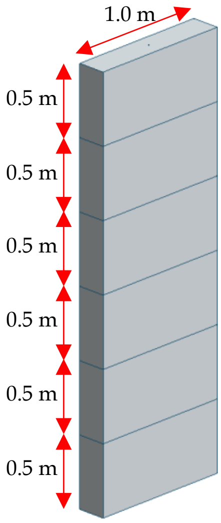


3. The cross section of the equivalent bar is 1 m wide and its height equals to the thickness of the shell. Along the bar, the applied reinforcement of the shell is transformed into the direction of the bar and placed into it.

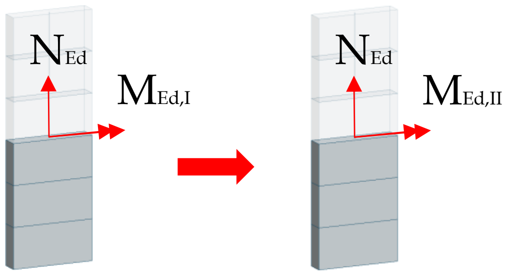


The checking process is executed section by section along the bar. The distance between these sections is given by *Division length of substitute column* parameter in Calculation parameter dialog (see the lower figure). Internal forces acting at these sections are calculated by transforming shell internal 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.



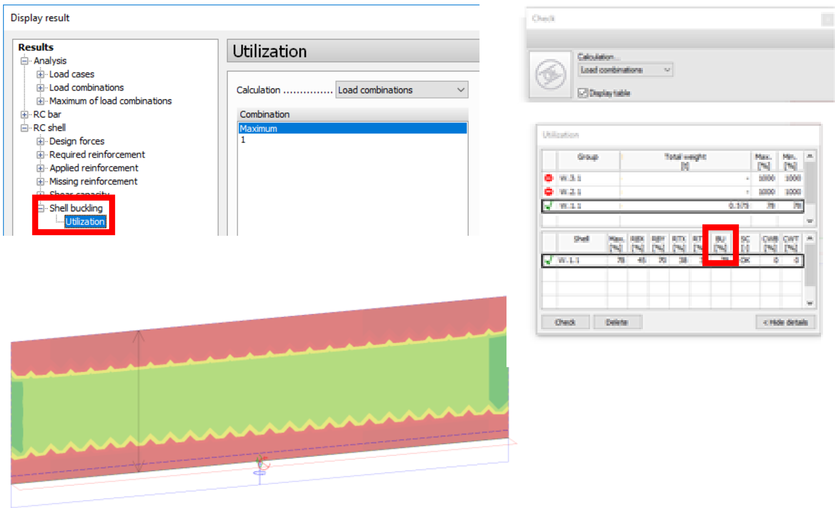


4. Once the first order internal forces are obtained in every section, the second order internal forces are calculated based on the *nominal stiffness* or *nominal curvature* method, according to the configuration settings. The only difference in the checking process of a real bar and this equivalent 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.

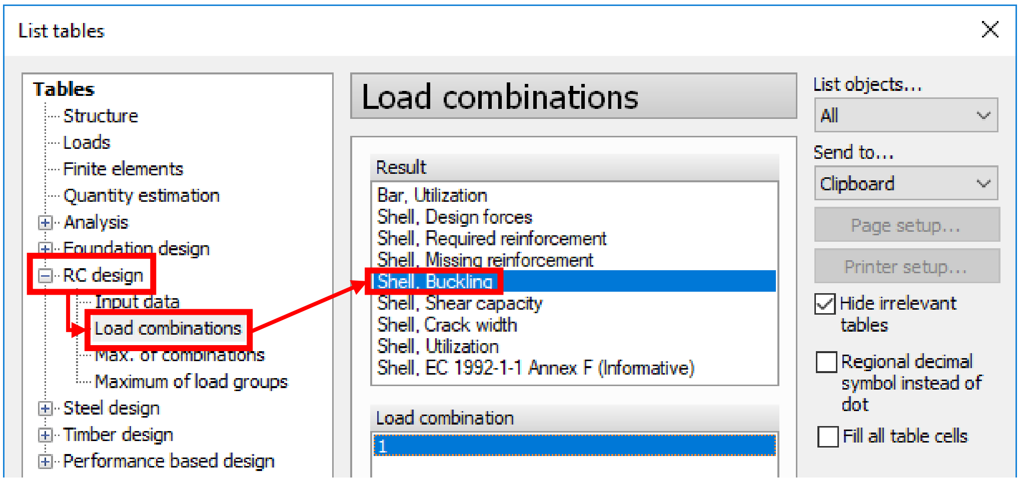


5. Finally, based on the second order internal forces, the utilization is calculated for every cross section of the equivalent bar (based on the interaction curve), and the highest section utilization is assigned to the node.

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.

6.4. RC shell – EN 1992-1-1, Annex F

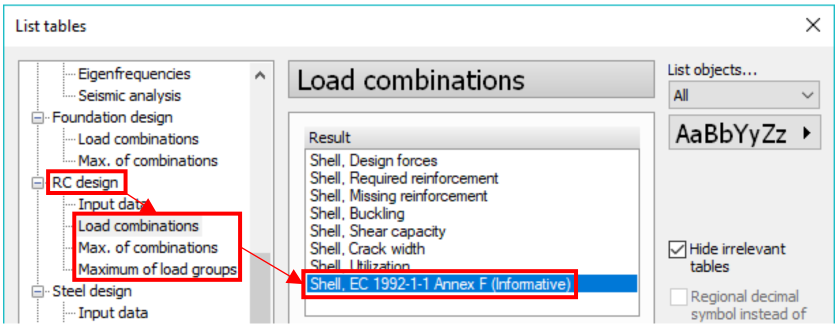
A new checking criterion is added for RC shells according to EN 1992 Annex F.

This feature is meant to inform the user whether the shell reinforcement is optimal or not according to the EN 1992-1-1, Annex F. The reinforcement is optimal, if the cracks for serviceability limit state are acceptable and the required deformation capacity for ultimate limit state is provided. This method only works for in-plane stress state. FEM-Design calculates the stresses in the reinforcement plane. The program uses the F1.(4) formulas for both the applied and required reinforcement.

Compression is positive in the formulas.

The results can be listed in the RC design for Load Combinations, Max. of combinations and Maximum of load groups.

If the Danish national annex set in code selection the calculation is based by the 1992-1-1 DK NA:2007.



Shell, EC 1992-1-1 Annex F (Informative), Load comb.: 1

ID	Thickness	Elem	Node	Face	Sigma Edx	Sigma Edy	Tau Edxy	cot(theta)	F.1 (3)
[-]	[mm]	[-]	[-]	[-]	[N/mm²]	[N/mm²]	[N/mm²]	[-]	[-]
W.1.1	200	7	0	Bottom	0.02	0.37	-0.00	106.45	Reinforcement not required.
			0	Top	-0.03	0.62	-0.00	390.16	Reinforcement required.
		8	0	Bottom	0.00	0.21	0.00	82.51	Reinforcement not required.
			0	Top	0.00	0.79	0.01	61.07	Reinforcement not required.
		9	0	Bottom	-0.01	0.04	0.00	17.89	Reinforcement required.
			0	Top	0.01	0.96	0.01	67.06	Reinforcement not required.
		10	0	Bottom	-0.00	0.21	0.00	105.78	Reinforcement required.
			0	Top	0.00	0.79	0.01	150.20	Reinforcement not required.
		11	0	Bottom	0.03	0.44	0.01	40.23	Reinforcement not required.
			0	Top	-0.05	0.56	0.01	88.38	Reinforcement required.
		12	0	Bottom	-0.12	-0.45	0.02	21.12	Reinforcement required.
			0	Top	0.18	1.46	-0.05	27.39	Reinforcement not required.
		13	0	Bottom	-0.00	0.12	-0.01	21.01	Reinforcement required.
			0	Top	0.00	0.88	0.02	35.64	Reinforcement not required.

Applied formulas	f'tdx	f'tdy	Sigma'cd	f tdx (Appl.)	f tdy (Appl.)	nu	f cd	Sigma Rd
[-]	[N/mm²]	[N/mm²]	[N/mm²]	[N/mm²]	[N/mm²]	[-]	[N/mm²]	[N/mm²]
-	-	-	-	0.96	0.96	0.562	10.7	5.99
(F5.); (F6.); (F7.)	0.03	0.00	0.62	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99
(F5.); (F6.); (F7.)	0.01	0.00	0.04	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99
(F5.); (F6.); (F7.)	0.00	0.00	0.21	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99
(F5.); (F6.); (F7.)	0.05	0.00	0.56	0.96	0.96	0.562	10.7	5.99
(F2.); (F3.); (F4.)	0.14	0.47	0.03	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99
(F5.); (F6.); (F7.)	0.00	0.00	0.12	0.96	0.96	0.562	10.7	5.99
-	-	-	-	0.96	0.96	0.562	10.7	5.99

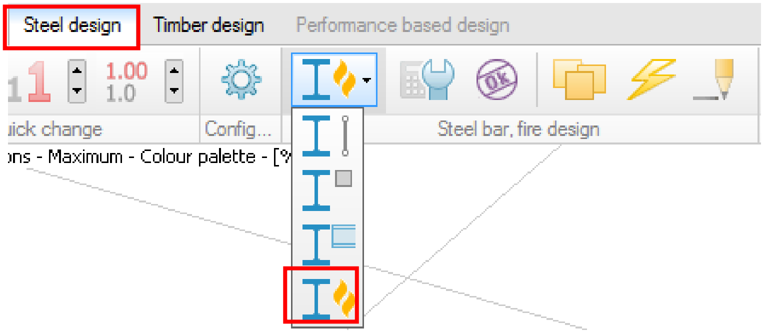
F.1 (4) Appl.	f tdx (F.8)	f tdy (F.9)	Sigma cd (F.10)	F.1 (4)
[-]	[N/mm²]	[N/mm²]	[N/mm²]	[-]
-	-	-	-	-
Not fulfilled!	0.03	0.03	0.66	Not fulfilled!
-	-	-	-	-
-	-	-	-	-
Not fulfilled!	0.01	0.01	0.05	Not fulfilled!
-	-	-	-	-
Not fulfilled!	0.00	0.00	0.21	Not fulfilled!
-	-	-	-	-
-	-	-	-	-
Not fulfilled!	0.05	0.05	0.61	Not fulfilled!
Not fulfilled!	0.45	0.45	0.34	Not fulfilled!
-	-	-	-	-
Not fulfilled!	0.00	0.00	0.13	Not fulfilled!
-	-	-	-	-

7. Steel design

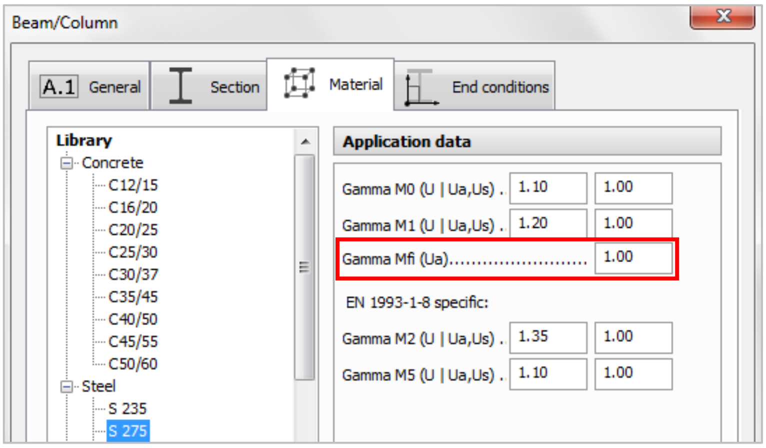
7.1. Fire design

The aim of this feature is to check and design steel bars for fire effects according to EN 1993-1-2.

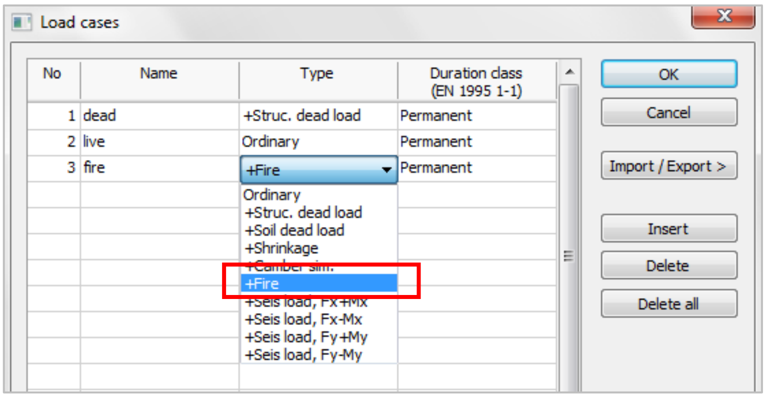
The feature can be found in *Steel design/Steel bar, fire design*. It needs some new input data of the bars, and a special load combination and/or load group must be defined.



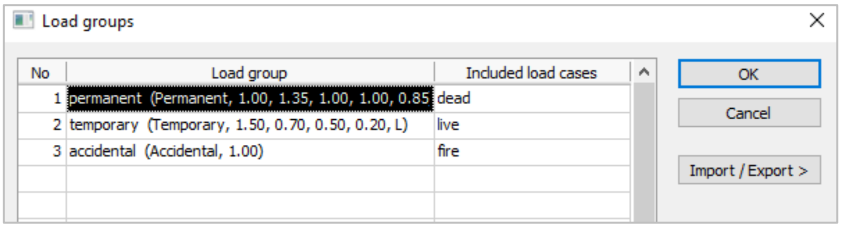
In *Beam/Column /Material/Application data* there is a new safety factor $\gamma_{M,fi.}$ required for fire design calculations.



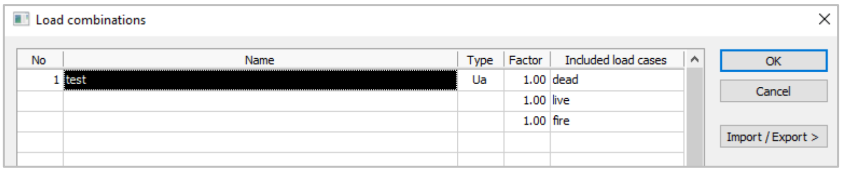
A „+Fire” type load case has to be defined in the *Load cases* dialog.



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.

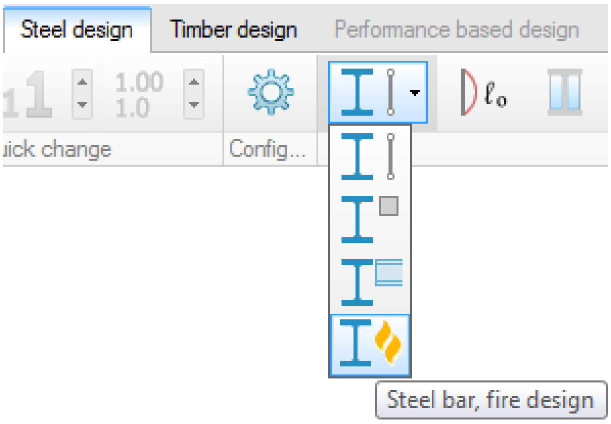


The effects (internal forces) are calculated from accidental load combinations, where fire is the accidental effect. Resistance of bar is calculated by using reduced yield strength and elasticity modulus for steel at the elevated temperature.

Steel design tab

A new design type can be found on the Steel design tab: „Steel bar, fire design”.

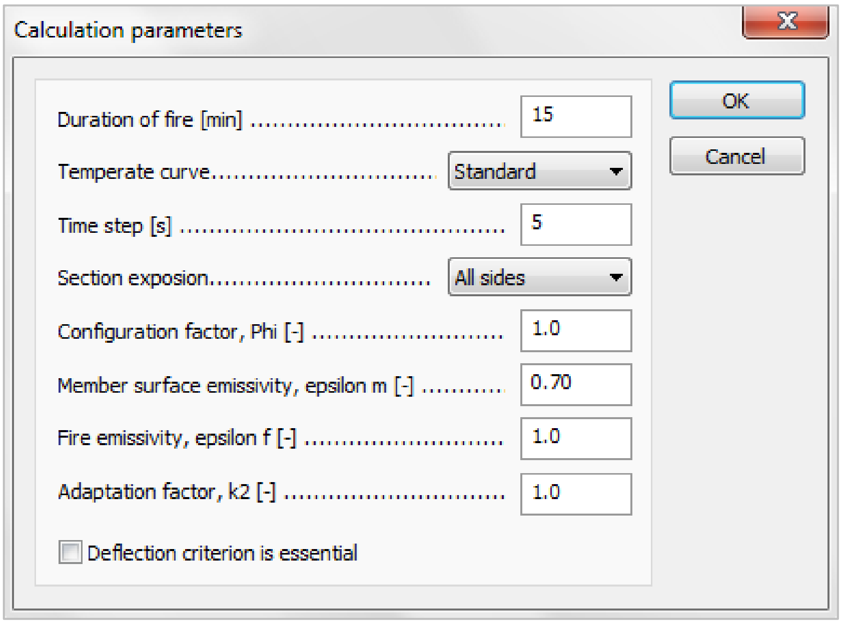
It contains *Calculation parameter, Check, Design group, Auto design* and *Manual design* commands.



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.

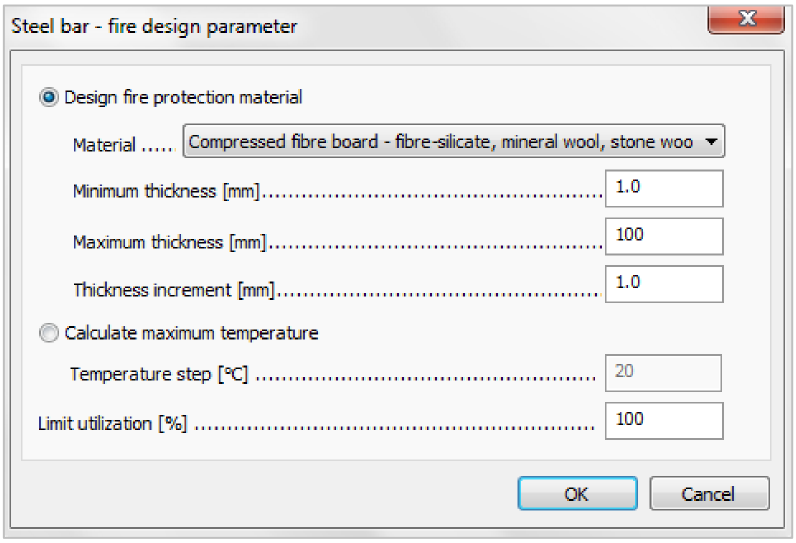
Auto design

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

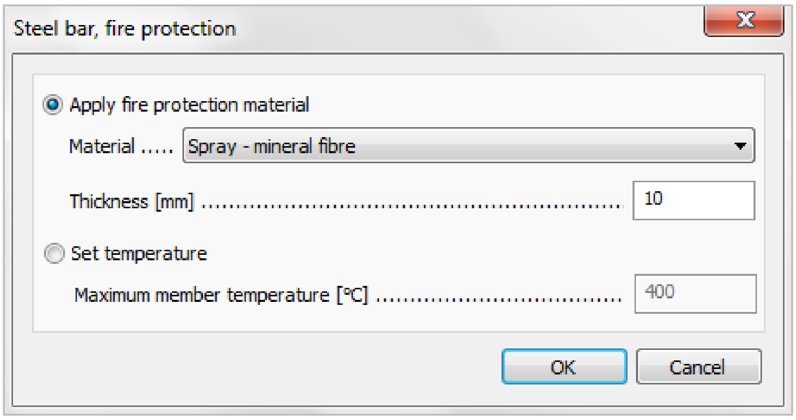
- *Calculate maximum temperature*
  The *Temperature step* for maximum temperature calculation can be defined by the User.

Limit utilization can also be set in the Steel bar - fire design parameter dialog



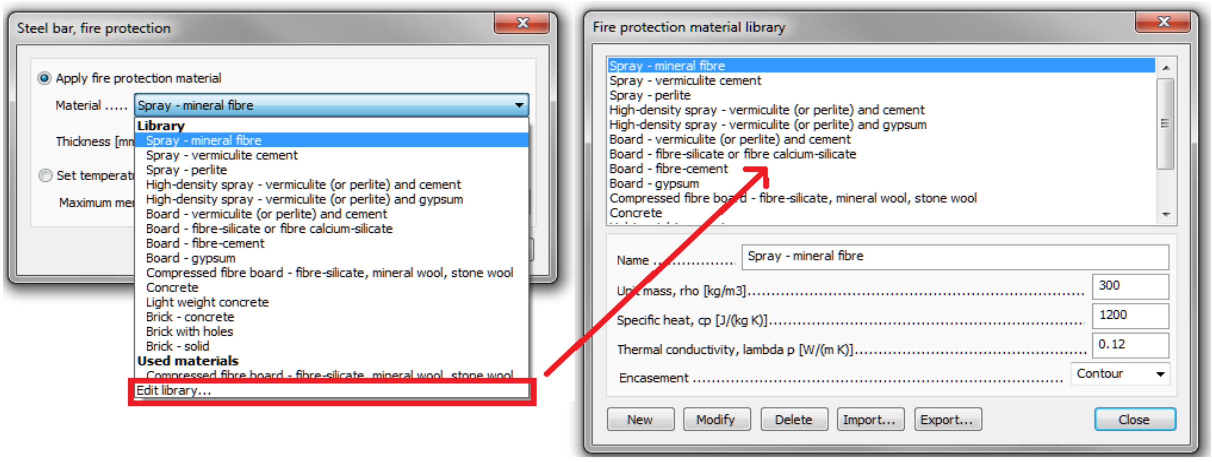
Manual design

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

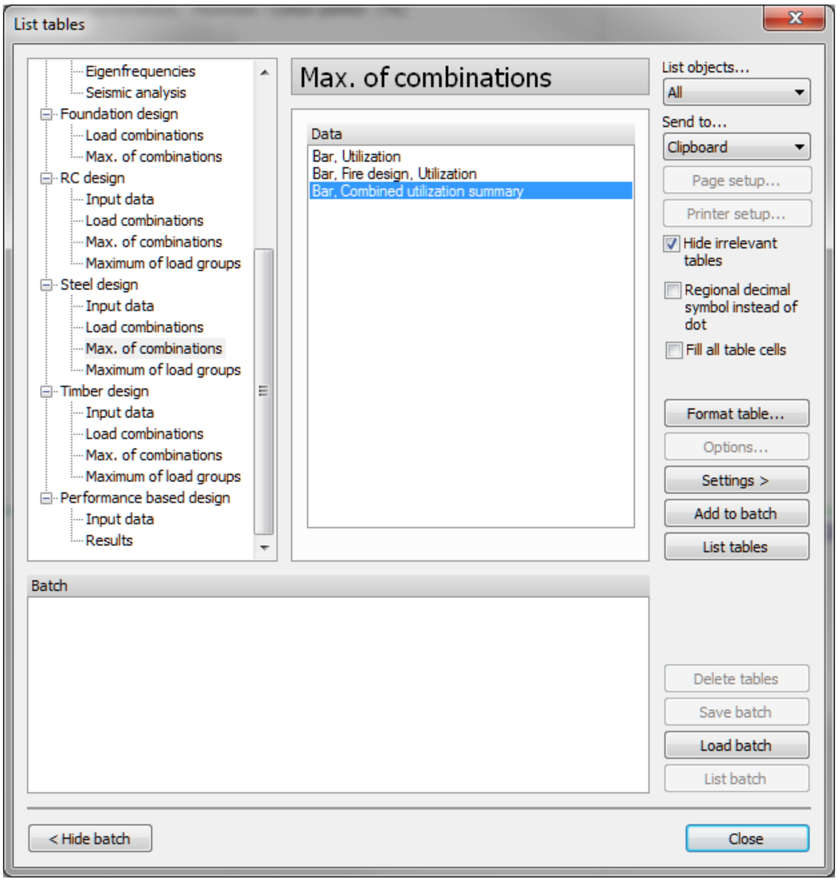
It is available by clicking on „Edit library…" item in the material list of fire protection parameter in Auto and Manual design dialogs.



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



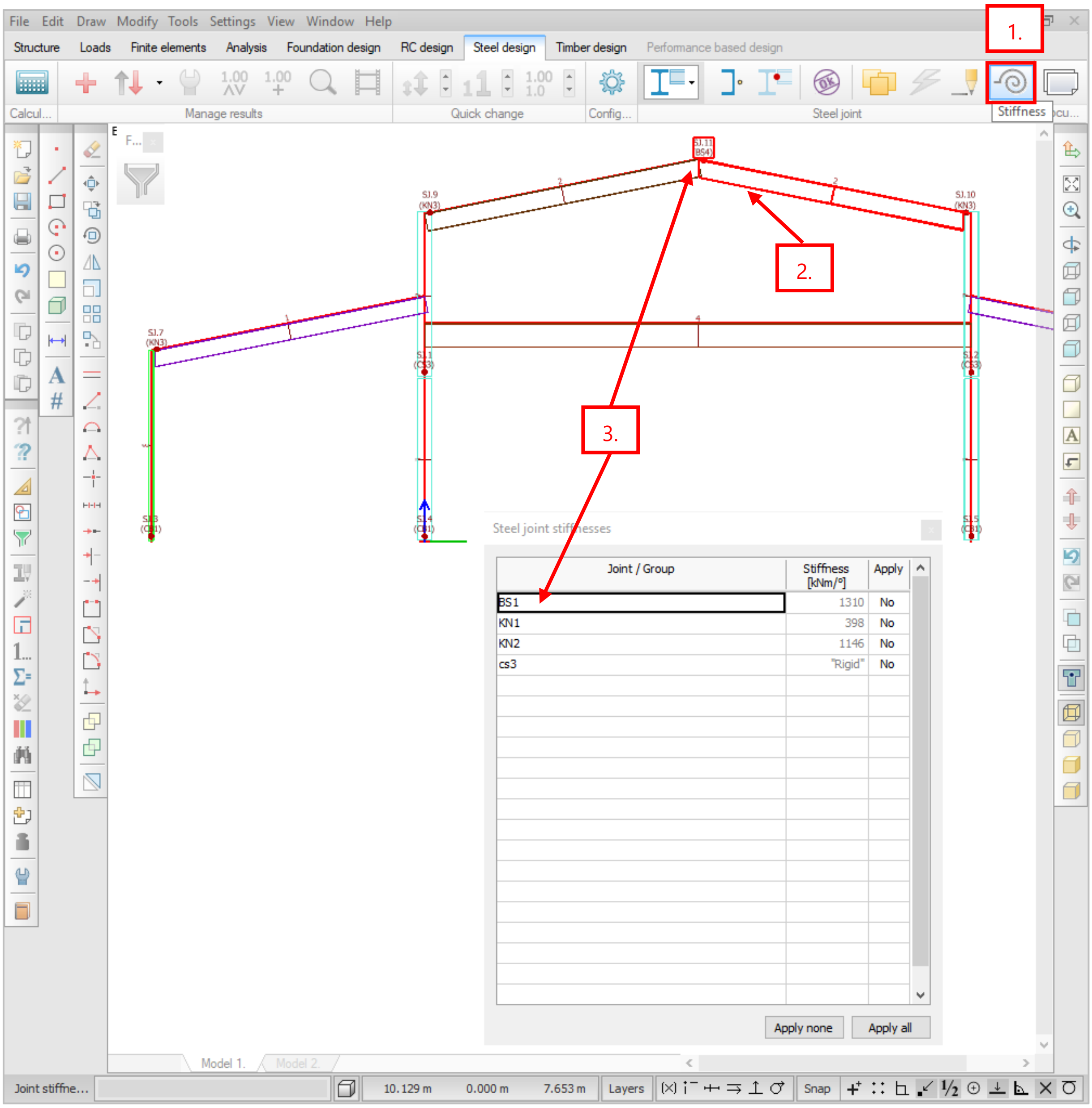
Max. of load combinations, Bar, Combined utilization summary

Member	Calculation	Maximum	Combination
[-]	[-]	[%]	[-]
B.1.1	ULS	121	Standard
	ALS (Fire)	98	Accidental

7.2. Steel joint stiffness

FEM-Design 17 gives the opportunity to set the rotation stiffness of *Steel Joints* automatically or manually, and apply it to the structure. This feature is available for *Steel Joints,* where at least one of the attached bars is connected with its end point by definition.

1. By clicking on the Steel design/Stiffness button, the *Steel joint stiffnesses* dialog appears.
2. Selecting a joint or joint group in the list will highlight the elements with thick red lines.
3. After that User can double click on the name of a joint/group in the list or just left click on the joint in the model to open the *Set joint stiffness* dialog.



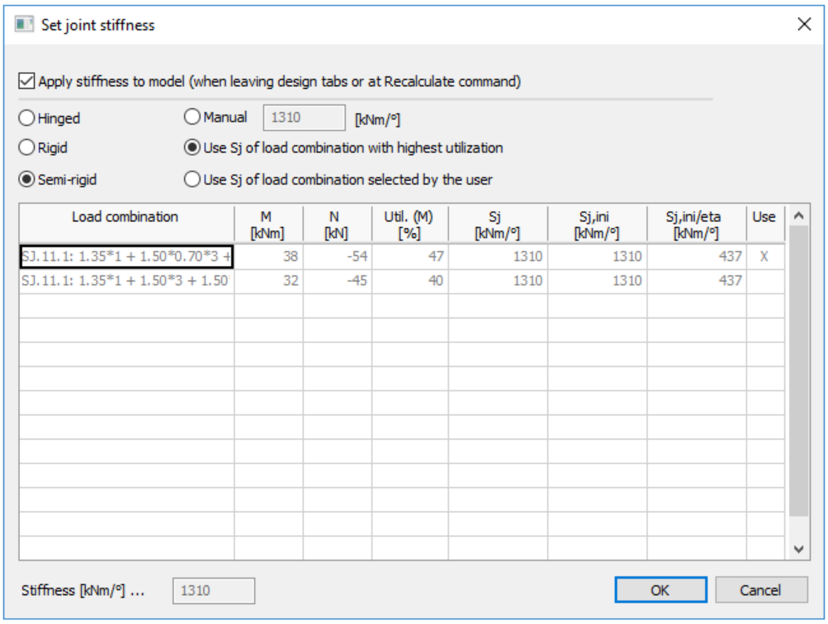
⚠ Stiffness can be set only for previously calculated *Steel Joints*!

FD 17 uses the calculation method described in the Eurocode 3-8 for calculating joint stiffnesses. This method is valid only for joints with bolted endplate and column base of I section bars.

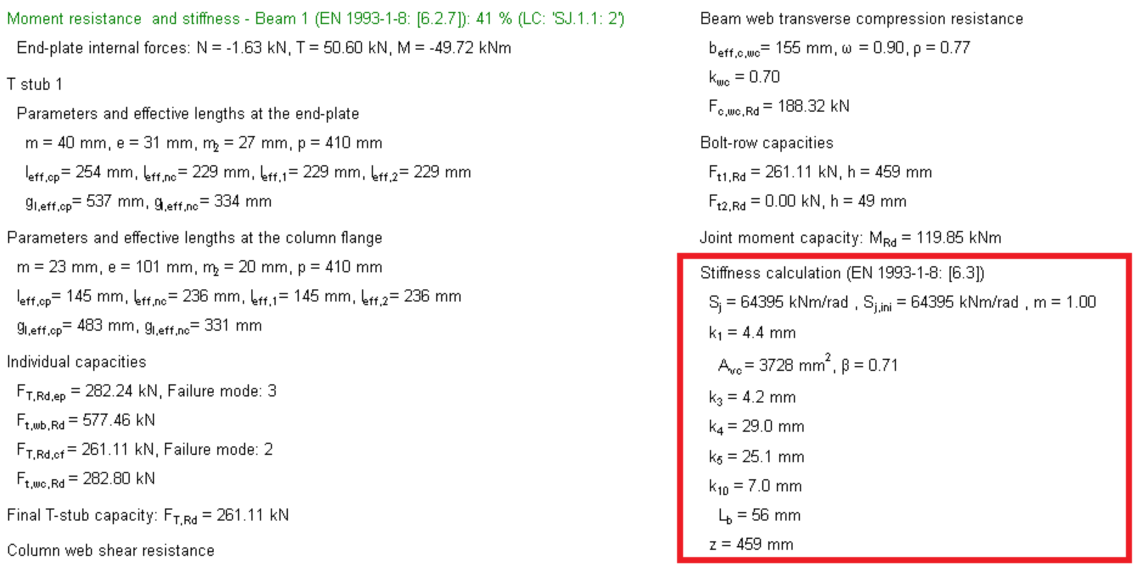
The program sets the stiffness of a joint by default as:

- hinged, if it can't take moment,
- semi-rigid with the calculated value, if it can take moment and stiffness can be calculated according to EC3-8,
- rigid, if it can take moment but the stiffness cannot be calculated according to EC3-8.

As the calculated stiffness value depends on the load of the joint, FEM-Design 17 by default uses the load combination that gives the highest utilization for moment. User however, can select another load combination, or can set stiffness manually.



The details of the calculation can be found in the *Detailed result* and in the *Manual design calculation* dialog.



Moment resistance and stiffness - Beam 1 (EN 1993-1-8: [6.2.7]): 41 % (LC: 'SJ.1.1: 2')

End-plate internal forces: N = -1.63 kN, T = 50.60 kN, M = -49.72 kNm

T stub 1

Parameters and effective lengths at the end-plate

m = 40 mm, e = 31 mm, m_2 = 27 mm, p = 410 mm

$l_{eff,cp}$ = 254 mm, $l_{eff,nc}$ = 229 mm, $l_{eff,1}$ = 229 mm, $l_{eff,2}$ = 229 mm

$g_{l,eff,cp}$ = 537 mm, $g_{l,eff,nc}$ = 334 mm

Parameters and effective lengths at the column flange

m = 23 mm, e = 101 mm, m_2 = 20 mm, p = 410 mm

$l_{eff,cp}$ = 145 mm, $l_{eff,nc}$ = 236 mm, $l_{eff,1}$ = 145 mm, $l_{eff,2}$ = 236 mm

$g_{l,eff,cp}$ = 483 mm, $g_{l,eff,nc}$ = 331 mm

Individual capacities

$F_{T,Rd,ep}$ = 282.24 kN, Failure mode: 3

$F_{t,wb,Rd}$ = 577.46 kN

$F_{T,Rd,cf}$ = 261.11 kN, Failure mode: 2

$F_{t,wc,Rd}$ = 282.80 kN

Final T-stub capacity: $F_{T,Rd}$ = 261.11 kN

Column web shear resistance

Beam web transverse compression resistance

$b_{eff,c,wc}$ = 155 mm, ω = 0.90, ρ = 0.77

k_{wc} = 0.70

$F_{c,wc,Rd}$ = 188.32 kN

Bolt-row capacities

$F_{t1,Rd}$ = 261.11 kN, h = 459 mm

$F_{t2,Rd}$ = 0.00 kN, h = 49 mm

Joint moment capacity: M_{Rd} = 119.85 kNm

Stiffness calculation (EN 1993-1-8: [6.3])

S_j = 64395 kNm/rad , $S_{j,ini}$ = 64395 kNm/rad , m = 1.00

k_1 = 4.4 mm

A_{vc} = 3728 mm^2, β = 0.71

k_3 = 4.2 mm

k_4 = 29.0 mm

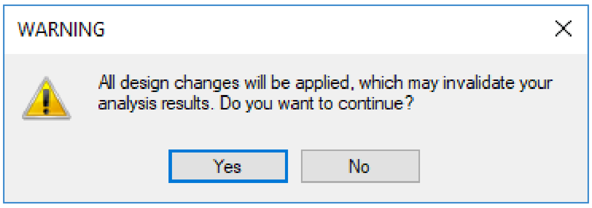
k_5 = 25.1 mm

k_{10} = 7.0 mm

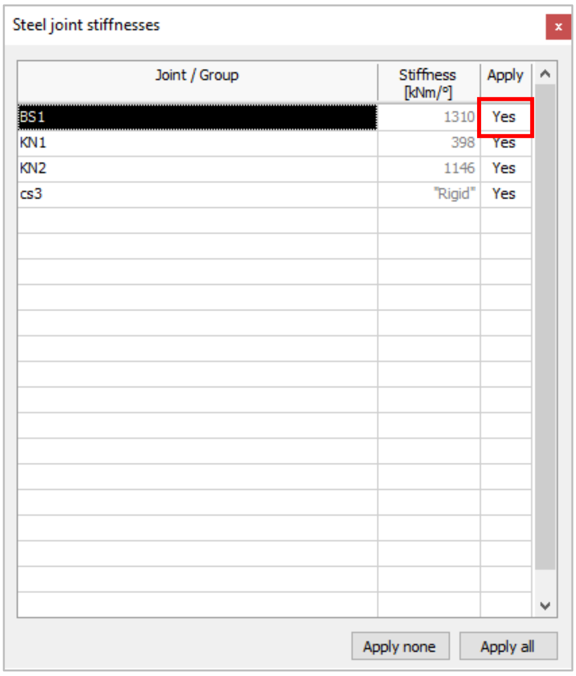
L_b = 56 mm

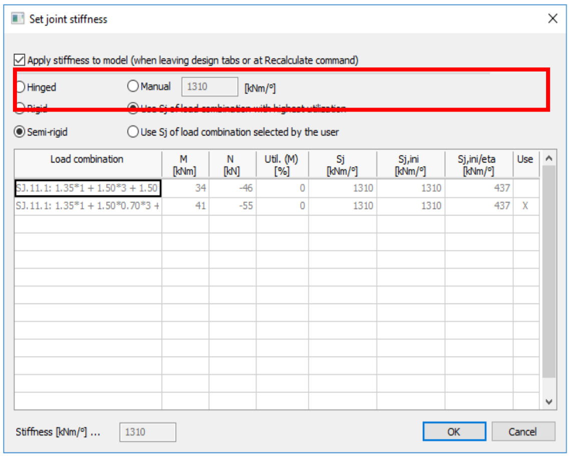
z = 459 mm

Once the stiffnesses of the joints are set, when leaving design tabs the following dialog will pop up:

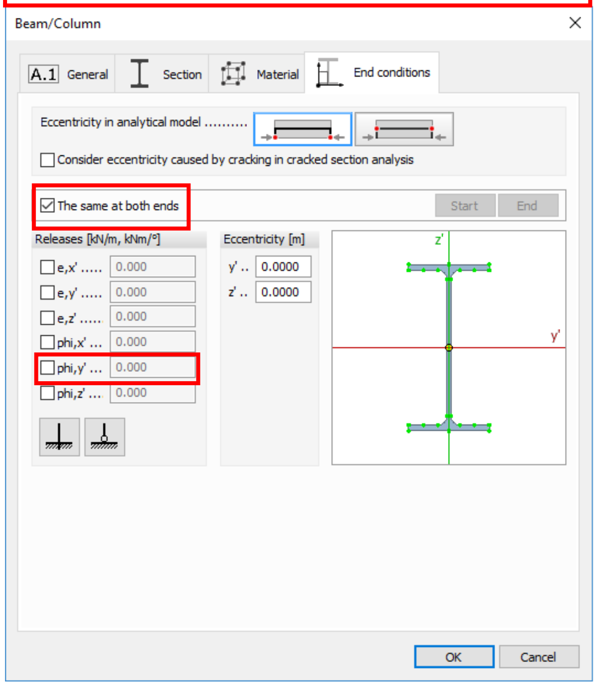


Selecting „Yes" will apply the new stiffnesses of joints to the attached bars, if „Apply stiffness to model..." box is checked in the *Set joint stiffness* dialog of the joint, or Apply column is checked in the *Steel joint stiffnesses* dialog next to the joint/group.

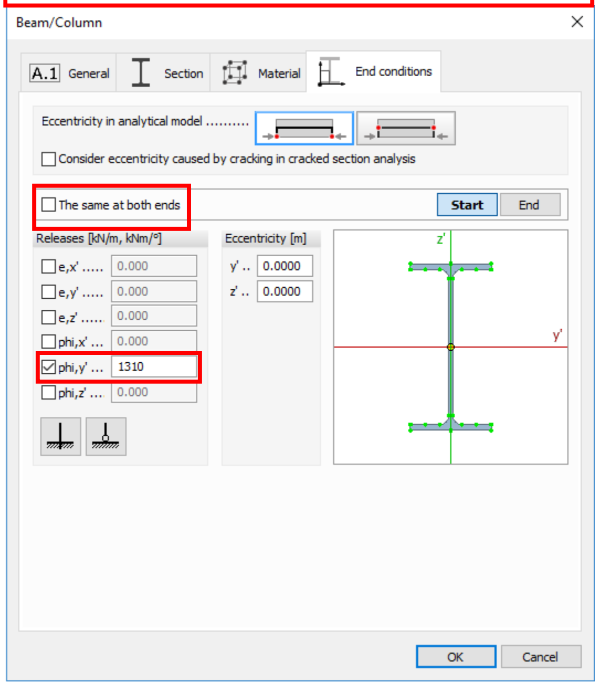




Before joint stiffness calculations



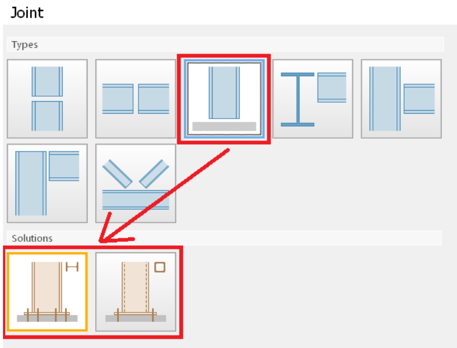
After joint stiffness applied on bar

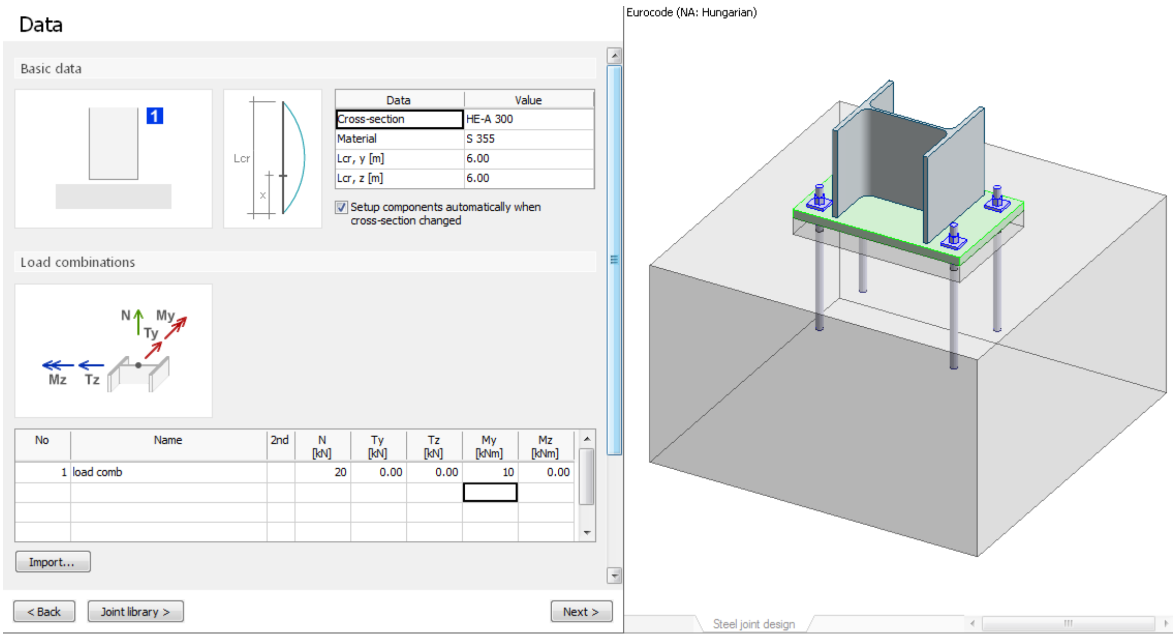


7.3. Column base joint concrete tension failures

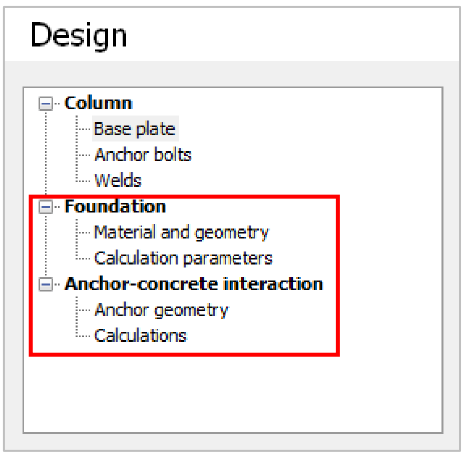
New verification is implemented for Column base joints that:

- calculates anchor forces and maximum concrete stress with nonlinear steel and concrete material behaviour,
- calculates capacities for adhesive and headed anchors for tension failure modes,
- works for both I and RHS sections.





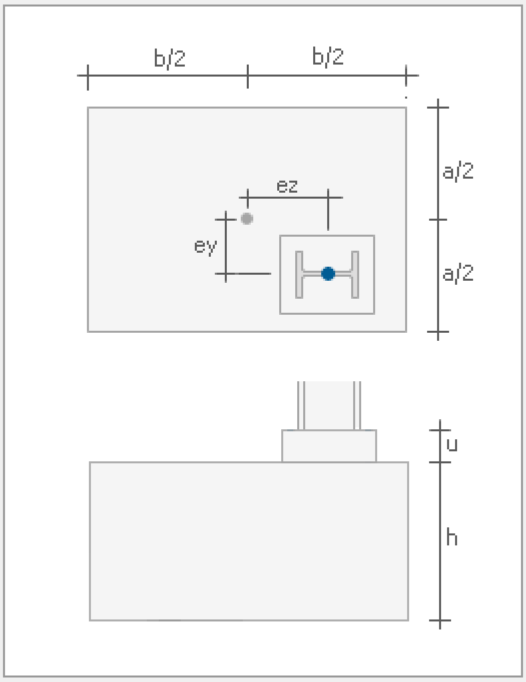
The new parameters required for the new calculations can be found in *Design tab*.



Design

- **Column**
  - Base plate
  - Anchor bolts
  - Welds
- **Foundation**
  - Material and geometry
  - Calculation parameters
- **Anchor-concrete interaction**
  - Anchor geometry
  - Calculations

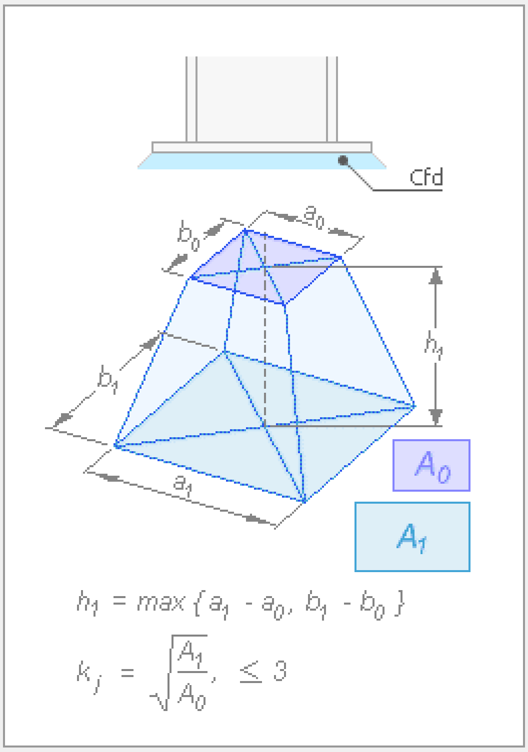
Data	Value
Material	C25/30
a [mm]	1000
b [mm]	1000
h [mm]	600
ey	0
ez	0
u [mm]	50

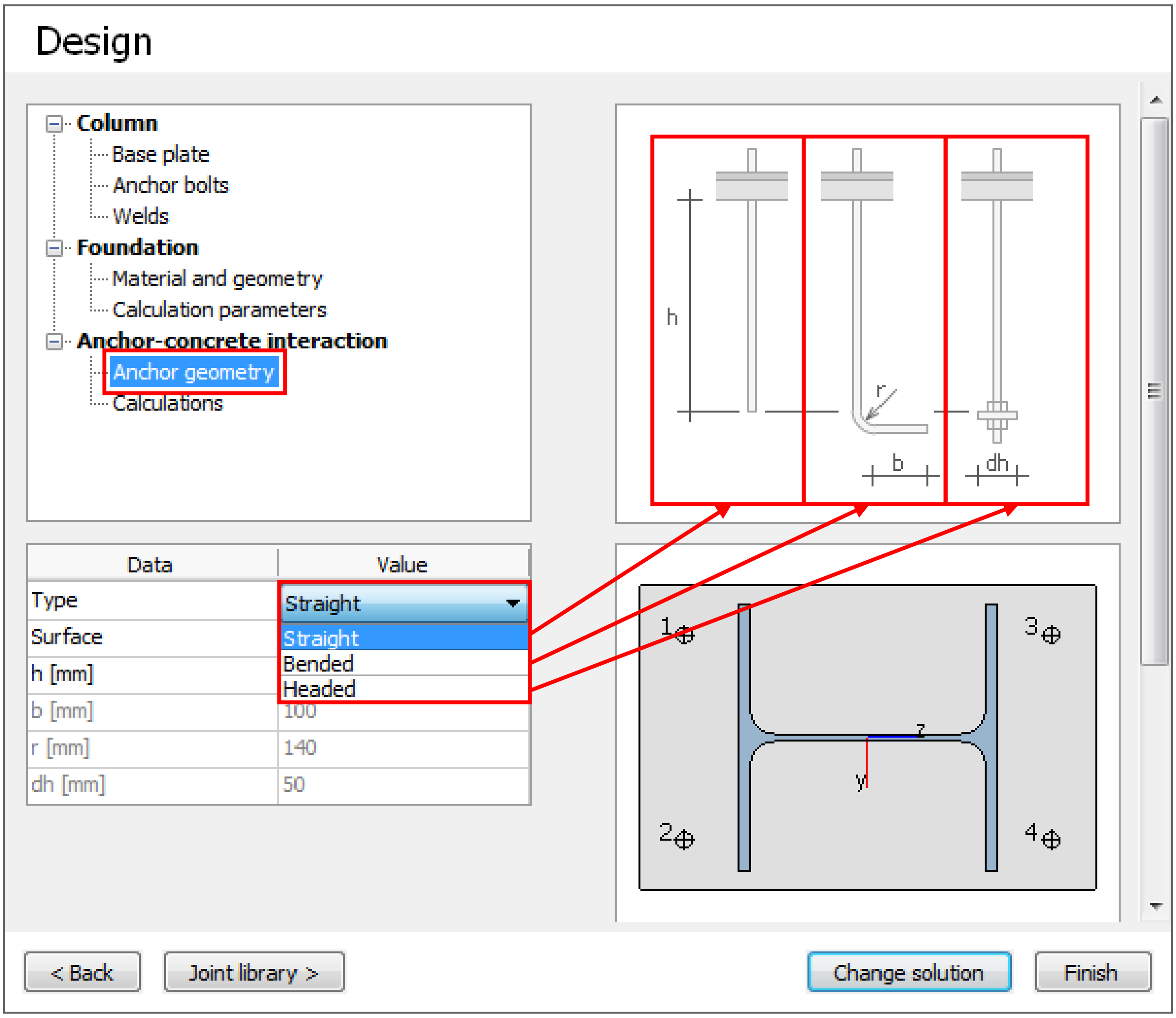


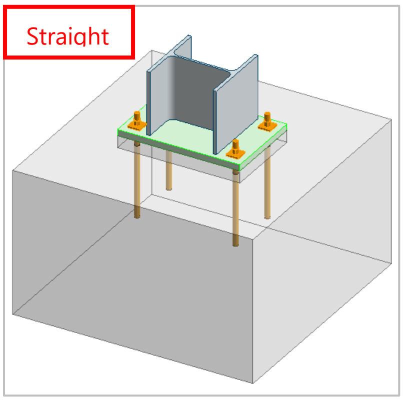
Design

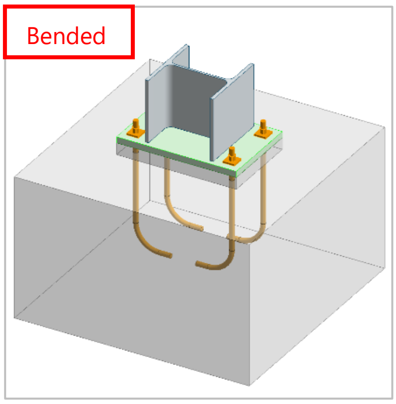
- **Column**
  - Base plate
  - Anchor bolts
  - Welds
- **Foundation**
  - Material and geometry
  - Calculation parameters
- **Anchor-concrete interaction**
  - Anchor geometry
  - Calculations

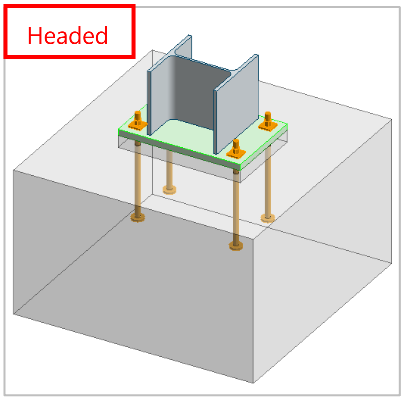
Data	Value
Friction coefficient (Cfd)	0.00
Beta j	0.67
kj	1.00




Design
Column
Base plate
Anchor bolts
Welds
Foundation
Material and geometry
Calculation parameters
Anchor-concrete interaction
Anchor geometry
Calculations
Data
Value
Type
Straight
Straight
Bended
Headed
Surface
h [mm]
b [mm]
100
r [mm]
140
dh [mm]
50
h
r
b
dh
1
2
3
4
z
y
< Back
Joint library >
Change solution
Finish

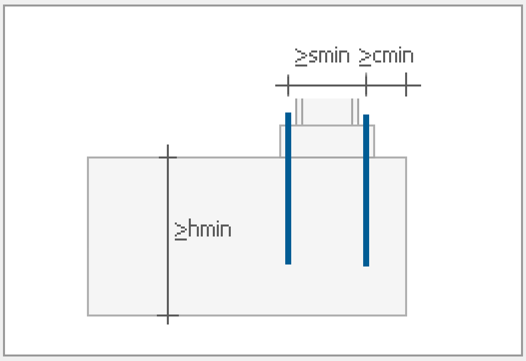

Straight


Bended


Headed

Design

- Column
  - Base plate
  - Anchor bolts
  - Welds
- Foundation
  - Material and geometry
  - Calculation parameters
- Anchor-concrete interaction
  - Anchor geometry
  - Calculations



Straight

Data	Value
Check requested	Yes
Concrete is cracked	Yes
gamma,Mp	1.50
gamma,Mc	1.50
gamma,Msp	1.50
kcr	8.50
kucr	11.90
Ignore cone-failure	No
Ignore splitting-failure	No
cmin [mm]	50
smin [mm]	80
hmin [mm]	260

Bended

Data	Value
Check requested	Yes
Concrete is cracked	Yes
gamma,Mp	1.50
gamma,Mc	1.50
gamma,Msp	1.50
kcr	8.50
kucr	11.90
Ignore cone-failure	No
Ignore splitting-failure	No
cmin [mm]	50
smin [mm]	80
hmin [mm]	260

Headed

Data	Value
Check requested	Yes
Concrete is cracked	Yes
gamma,Mp	1.50
gamma,Mc	1.50
gamma,Msp	1.50
kcr	8.50
kucr	11.90
Ignore cone-failure	No
Ignore splitting-failure	No
cmin [mm]	50
smin [mm]	80
hmin [mm]	260

< Back | Joint library >

The picture below shows the output for adhesive (straight or bended) and headed anchorage.

Adhesive

Anchorage tension resistance (Ref.8: [2.4.1]): 86 % (LC: 'aa')

Anchor normal and shear forces
- $F_{Ed,1}$= -8.23 kN, $V_{Ed,1}$= 15.81 kN
- $F_{Ed,2}$= -56.82 kN, $V_{Ed,2}$= 15.81 kN
- $F_{Ed,3}$= 62.29 kN, $V_{Ed,3}$= 15.81 kN
- $F_{Ed,4}$= 13.70 kN, $V_{Ed,4}$= 15.81 kN

Maximum concrete compression stress
- $\sigma_{c,Ed}$= -15.68 N/mm^2

Adhesive failure resistance: 86 %
- Topology-independent anchor parameters
  - l_b= 300 mm, f_{bd}= 2.70 N/mm^2
- Dominant anchor: 3
- Topology-dependent parameters for the dominant anchor
  - c_d= 105 mm, α_1= 1.00, α_2= 0.70
- Normal force and capacity for the dominant anchor
  - $F_{Ed,3}$= 62.29 kN, $F_{Rd,a}$= 72.71 kN

Headed

Anchorage tension resistance (CEN/TS 1992-4-2: [6.2]): 66 % (LC: 'aa')

Anchor normal and shear forces
- $F_{Ed,1}$= -8.23 kN, $V_{Ed,1}$= 15.81 kN
- $F_{Ed,2}$= -56.82 kN, $V_{Ed,2}$= 15.81 kN
- $F_{Ed,3}$= 62.29 kN, $V_{Ed,3}$= 15.81 kN
- $F_{Ed,4}$= 13.70 kN, $V_{Ed,4}$= 15.81 kN

Maximum concrete compression stress
- $\sigma_{c,Ed}$= -15.68 N/mm^2

Pull-out failure resistance: 31 %
- Topology-independent anchor parameters
  - A_h= 1649 mm^2, $f_{ck,cube}$= 30.00 N/mm^2, $\psi_{ucr,N}$= 1.00
- Dominant anchor: 3
- Normal force and capacity for the dominant anchor
  - $F_{Ed,3}$= 62.29 kN, $F_{Rd,p}$= 197.92 kN

Concrete cone failure resistance: 66 %
- Dominant anchor group: 3, 4
- Topology-dependent parameters for the dominant anchor group
  - h_{ef}'= 257 mm, $s_{cr,N}'$= 770 mm, $A_{c,N}^0$= 592900 mm^2, $A_{c,N}$= 680000 mm^2
  - $N_{Rk,c}^0$= 191.44 kN, $\psi_{s,N}$= 0.93, $\psi_{re,N}$= 1.00, $\psi_{ec,N}$= 0.84
- Normal force and capacity for the dominant anchor group
  - $F_{Ed,3,4}$= 75.99 kN, $F_{Rd,c}$= 114.28 kN

Splitting failure resistance: 31 %
- Topology-independent anchor parameters
  - h_{min}= 260 mm, $s_{cr,sp}$= 780 mm, $A_{c,N}^0$= 608400 mm^2
  - N_{Rk}^0= 241.91 kN, $\psi_{h,sp}$= 1.75
- Dominant anchor group: 3, 4
- Topology-dependent parameters for the dominant anchor group
  - $A_{c,N}$= 685000 mm^2, $\psi_{s,N}$= 0.93, $\psi_{re,N}$= 1.00, $\psi_{ec,N}$= 0.84
- Normal force and capacity for the dominant anchor group
  - $F_{Ed,3,4}$= 75.99 kN, $F_{Rd,sp}$= 247.29 kN

Blow-out failure resistance: 0 %
- Not relevant

The verification is carried out according to:

- The Swedish Institute of Steel Construction, Detail Handbook, Publication 183, 2.4.1 for the Straight and Bended type of anchors.
- CEN/TS 1992-4-2: [6.2] for the Headed type of anchors.

7.4. “Rotated” option for hollow sections

Hollow sections can be rotated separately in all joints where they are available.

Data	Value
Cross-section 1.	KKR 200x120x8
Material 1.	S 355
Rotated 1.	No
Cross-section 2.	KKR 200x120x8
Material 2.	S 355
Rotated 2.	No
a [mm]	0
b [mm]	1000

☑ Setup components automatically when cross-section changed

Eurocode (NA: Hungarian)

Data	Value
Cross-section 1.	KKR 200x120x8
Material 1.	S 355
Rotated 1.	Yes
Cross-section 2.	KKR 200x120x8
Material 2.	S 355
Rotated 2.	No
a [mm]	0
b [mm]	1000

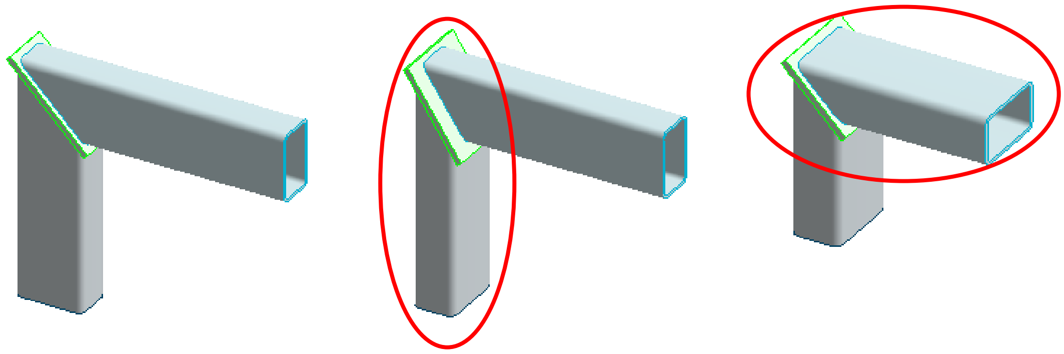
☑ Setup components automatically when cross-section changed

Eurocode (NA: Hungarian)

Data	Value
Cross-section 1.	KKR 200x120x8
Material 1.	S 355
Rotated 1.	Yes
Cross-section 2.	KKR 200x120x8
Material 2.	S 355
Rotated 2.	Yes
a [mm]	0
b [mm]	1000

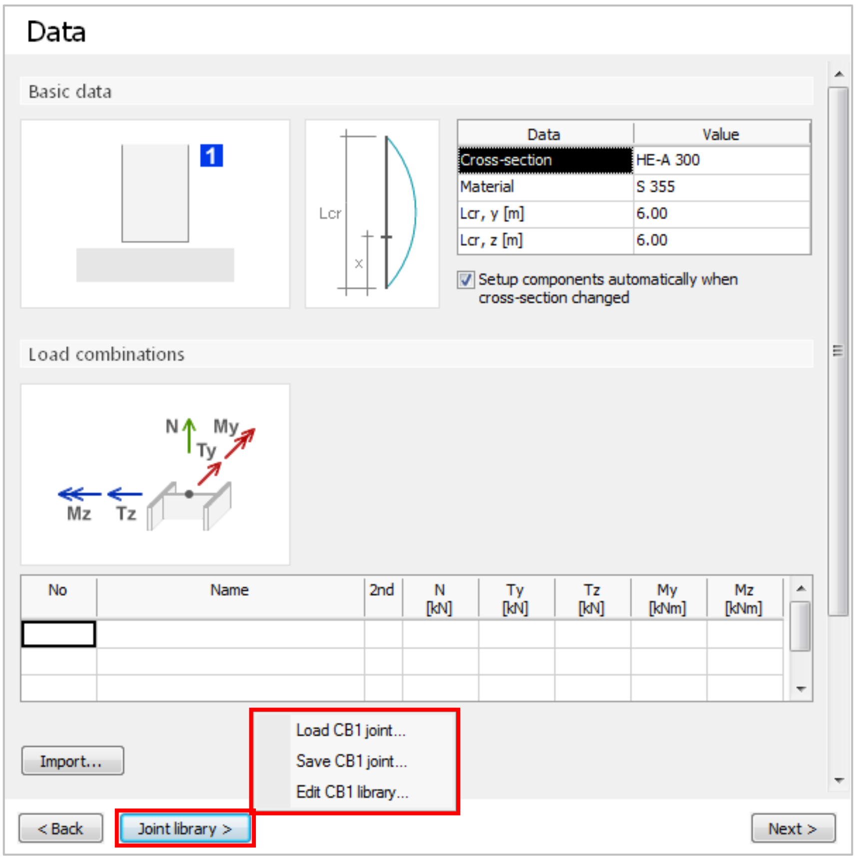
☑ Setup components automatically when cross-section changed

Eurocode (NA: Hungarian)

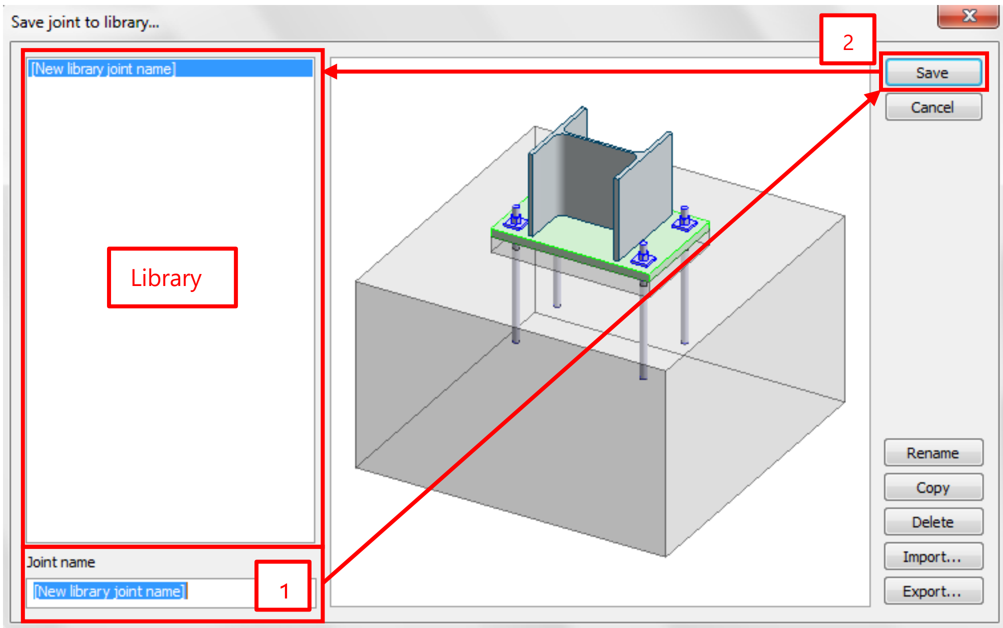


7.5. Joint library

Both in *Steel joint* module and in *Steel joint manual design* window of *3D Structure* module the customized joint types can be saved into/loaded from a library.

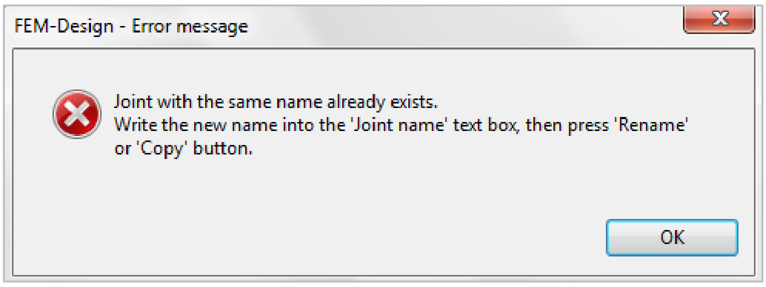


By clicking *Save joint*, the *Save joint to library...* dialog opens. In the dialog, in the bottom-left corner an arbitrary name can be typed for the joint. After clicking Save in the top-right corner, the saved joint appears in the library.

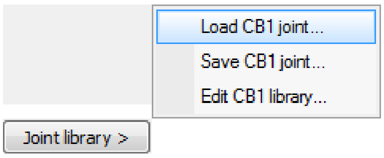


Using the buttons in the bottom-right corner, User can rename, copy, delete, import or export joints (from/to *.fdljoint format).

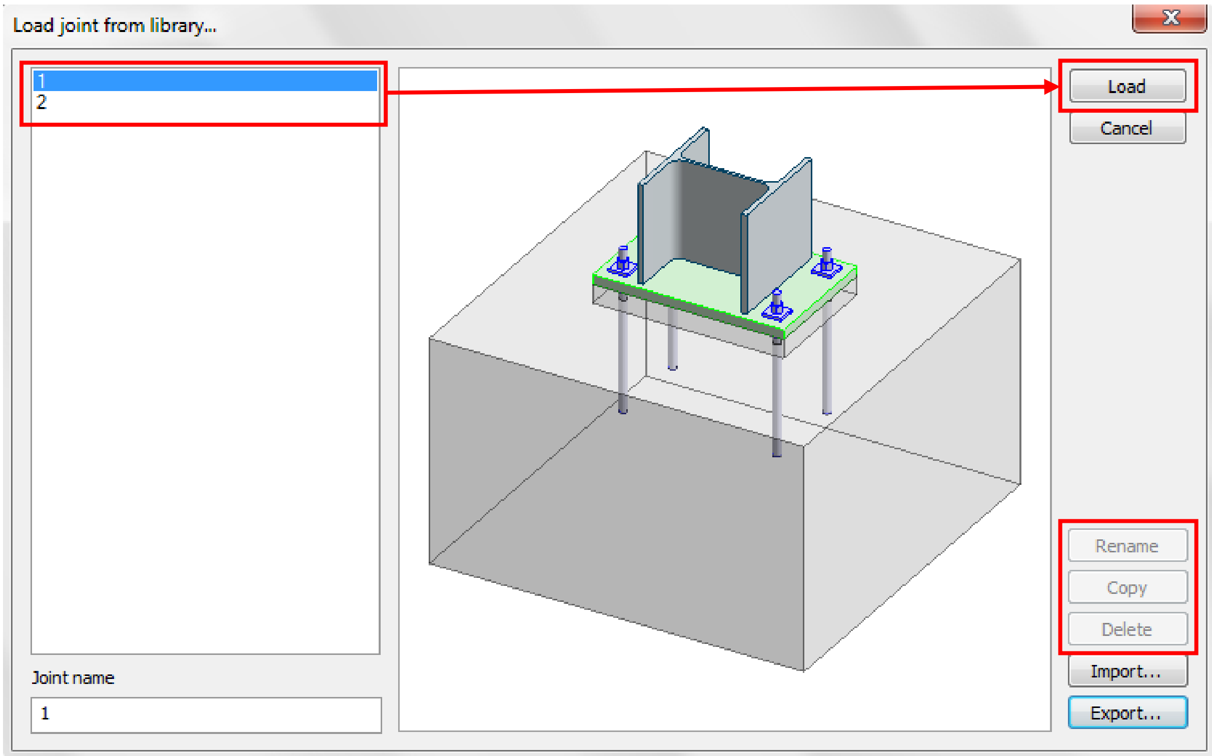
To rename or copy an existing joint, firts type the new name into the textbox under the library, then click on *Rename*.



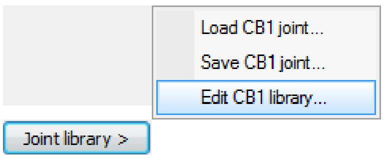
User can load previously saved joints by clicking on *Joint library/Load joint*. The *Load joint from library...* dialog opens, which is similar to *Save joint to library...* dialog, except that the *Rename, Copy* and *Delete* buttons are disabled, and instead of *Save* button, *Load* button appears.



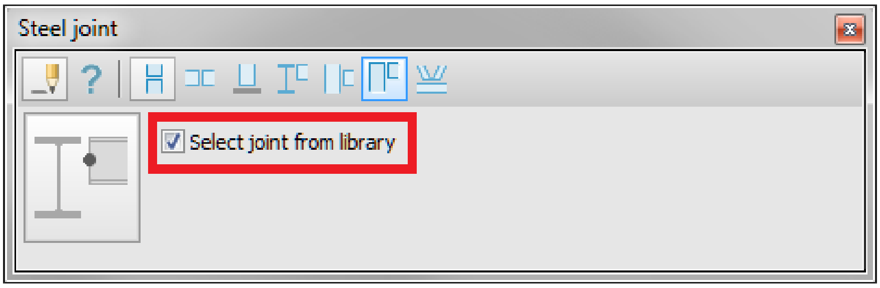
To load an existing joint, select it from the library then click *Load*.



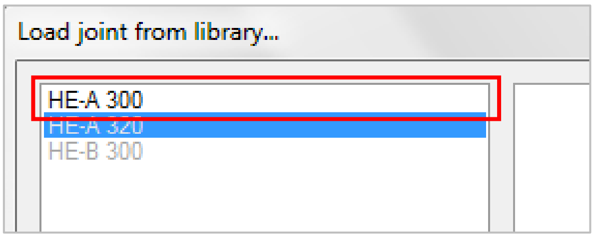
By clicking *Joint library/Edit joint* the *Edit library dialog* opens It looks the same as *Save joint to library*. Here the user can rename, copy, delete joints and import/export from/to the library.



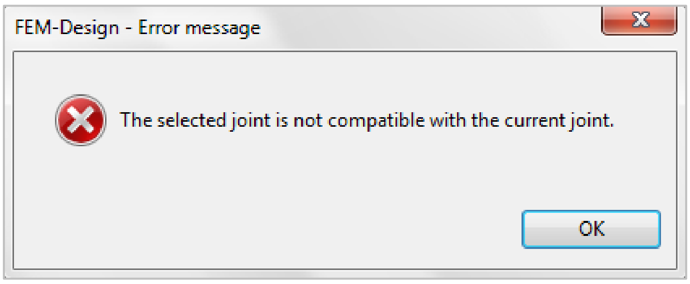
In *3D Structure* module at *Steel design tab/Steel joint/Define joint* User can select a joint from the joint library by checking the *Select joint from library* checkbox.



In the *Load joint from library* dialog, only those joints can be chosen thatare compatible with the selected bar(s). Those joints are shown active (black) in the library.



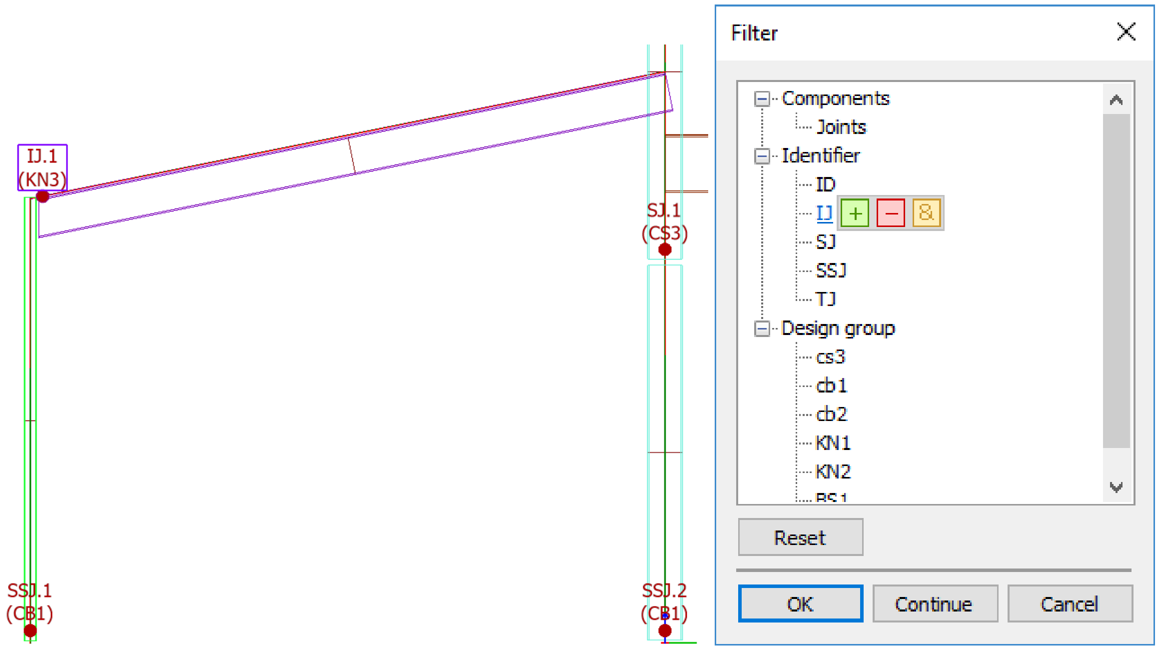
⚠ If User selects a non-appropriate joint for the selected element(s), FEM-Design sends a warning and automatically creates a joint fitting for the selected item.



⚠ Each joint solution has to be saved to its own library. There is no possibility to save all joint types and solutions into one library.

7.6. Steel joints added to Filter

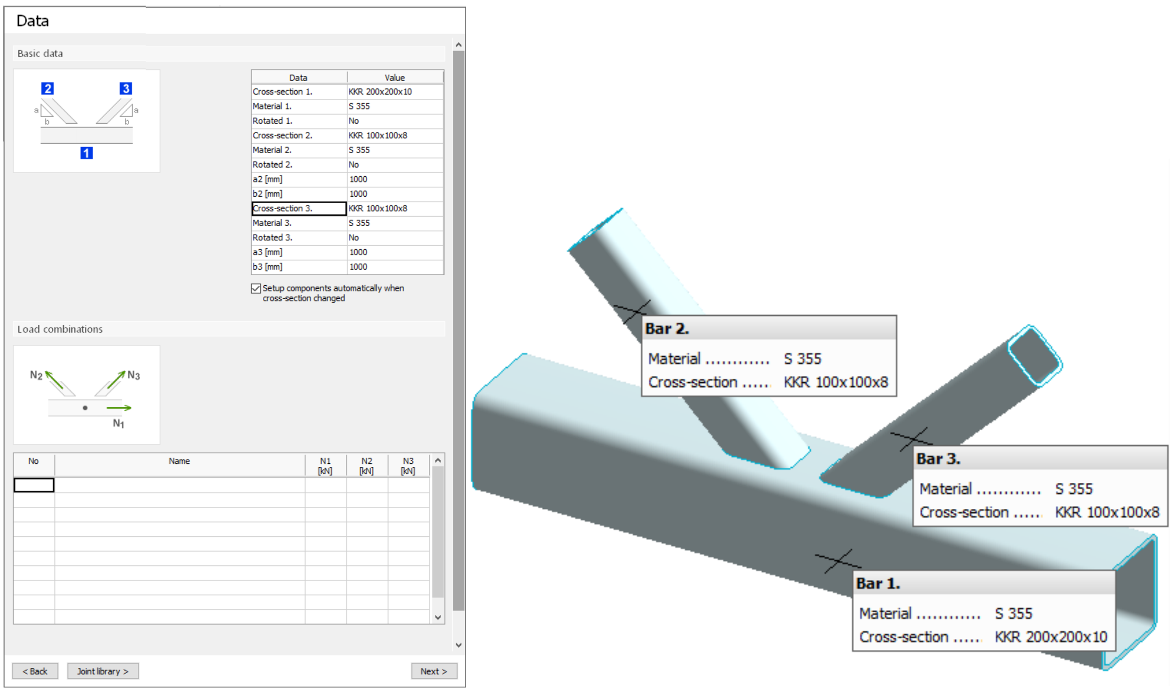
In *3D Structure* module steel joints can be filtered either as Joint component, or by their identifier.



7.7. User interface improvements in Steel joints module

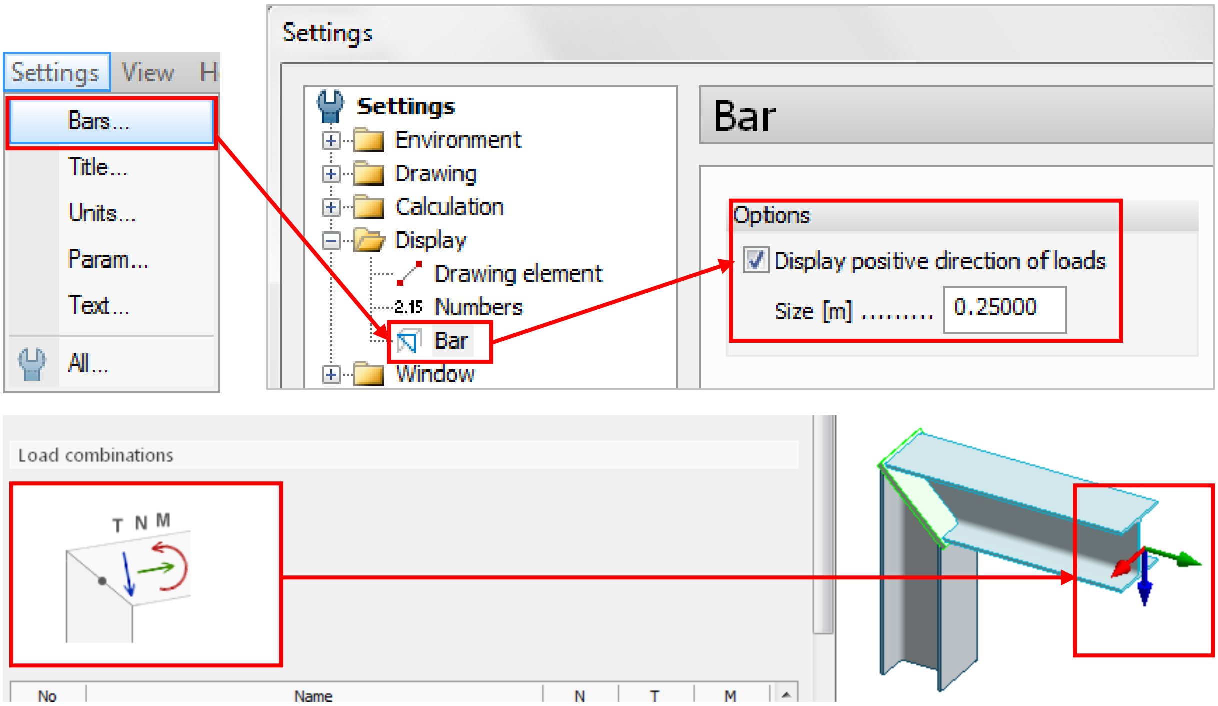
7.7.1. Tooltip for joint bars

Tooltip for joint bars has been implemented for easier identification in *Steel joint* module and *Manual design* window in *3D Structure* module.



7.7.2. Joint bar display option

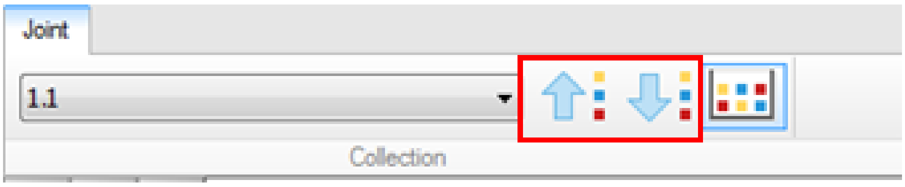
The positive directions of loads can be displayed on the joint by checking the *Display positive direction of loads* option in the *Settings/Bars.../Display/Bars* dialog in the *Steel joint* module and in the *Manual design* window in the *3D Structure* module. The size of the arrows can be set, too.



7.7.3. Navigation buttons for steel joints

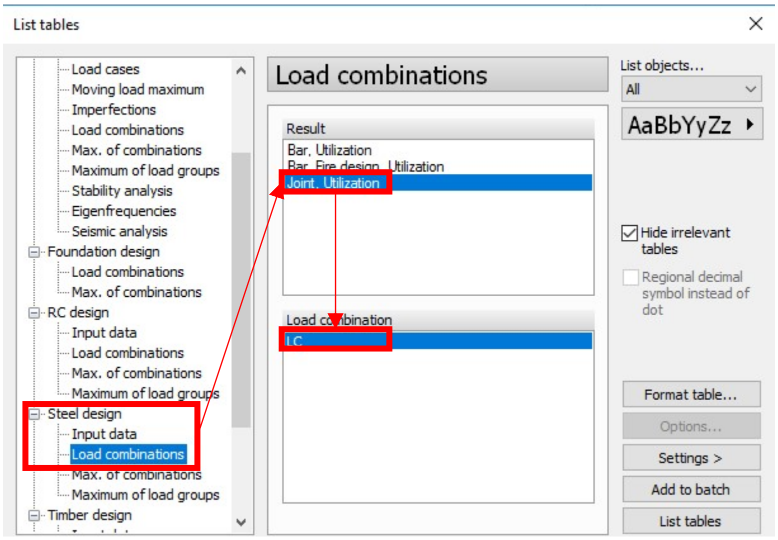
In FEM-Design 17, while using Steel joint Manual Design in *3D Structure* module, User can switch between *Previous* or *Next* joint by clicking up and down arrows next to the drop-down list.

This feature can be found in *Steel Joint* module, too.



7.8. Steel joint utilization in Documentation

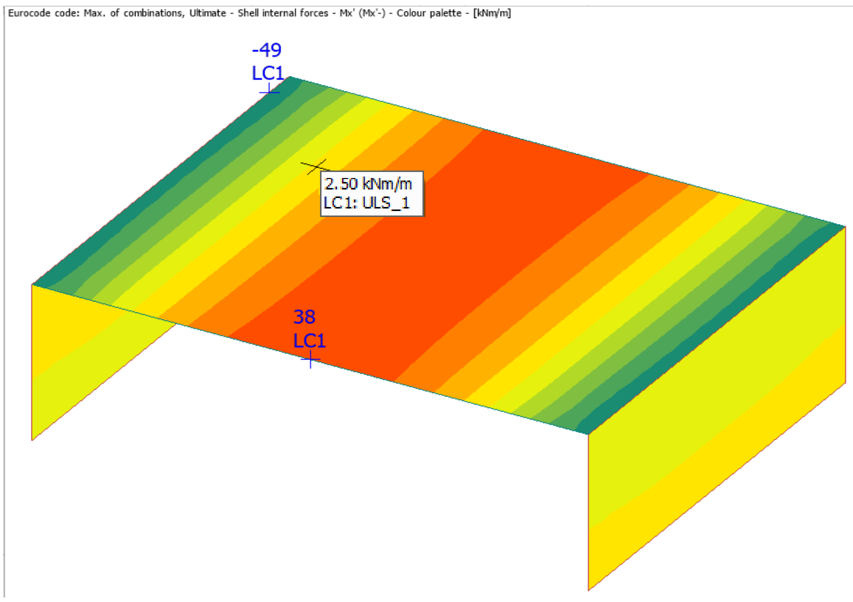
Steel joint utilization can be listed under *Steel design/Load combinations/Joint, utilization.*



8. Results

8.1. Dominant load combination shown on results for maximum of load combinations

Maximum of load combination results now shows the ordinal number of the load combination, from which the min/max value comes from. If User holds the cursor over a point, the ordinal number and the full name of the load combination are shown in a tooltip.

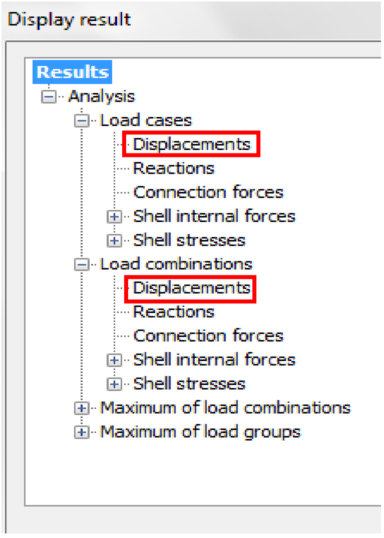


The display of number of load combination can be hidden by unchecking the 'Shoe extra information, if available' box in the Numeric Value dialog.

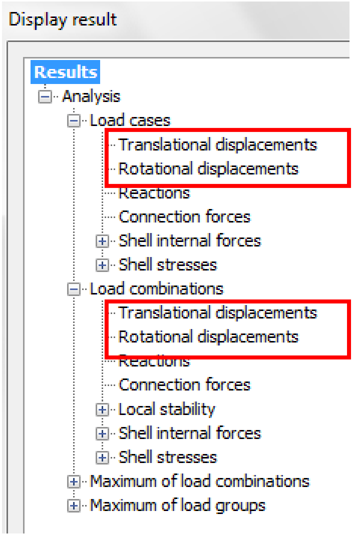
8.2. Displacement-like results improved

Displacements result's name has been modified to *Translational displacements* and a new result called *Rotational displacements* is added.

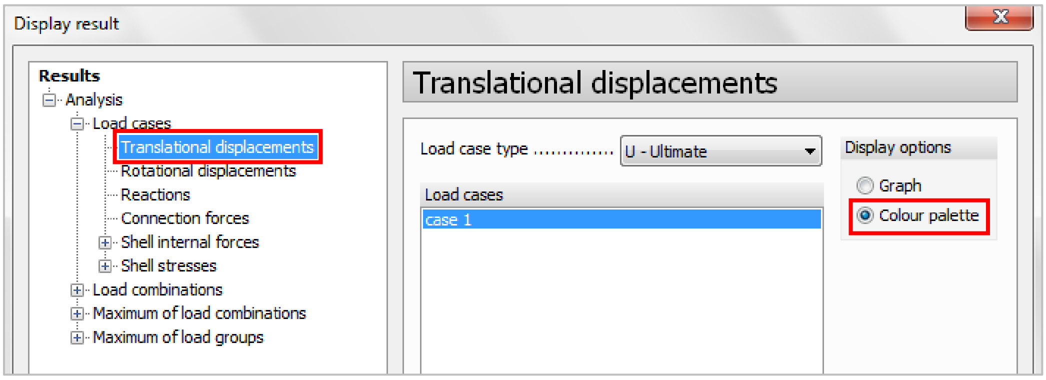
FD 16:



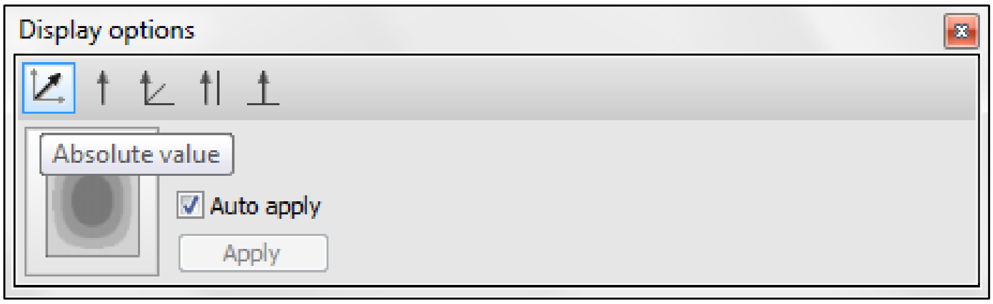
FD 17:

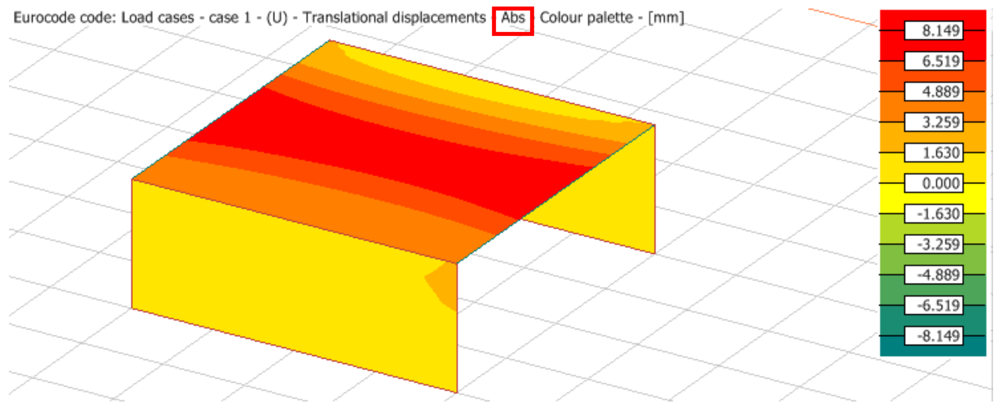


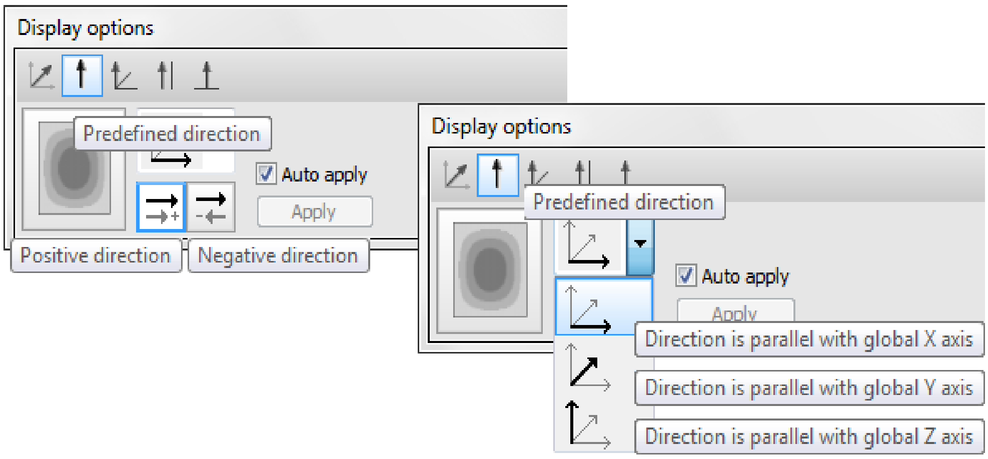
Also colour-palette option is added to *Translational displacements* results that allows for better presentation and understanding of displacements results in some cases.

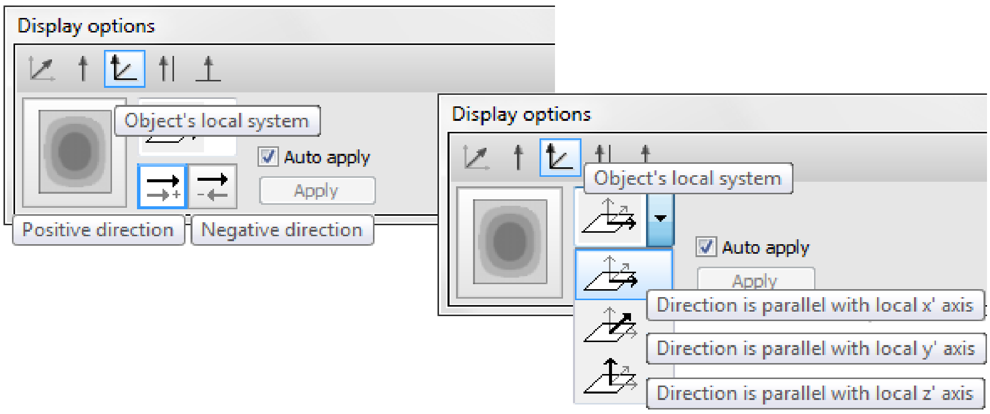


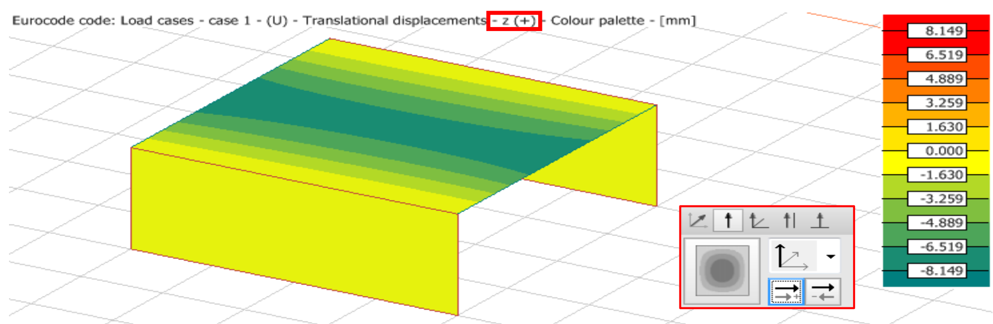
User can find a few powerful options to visualise the displacement results in *Display options* dialog. They give many possibilities to show the result in any pre- or user defined directions. The pictures below present visualisation possibilities.

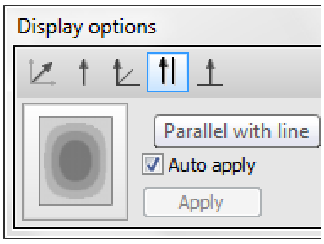


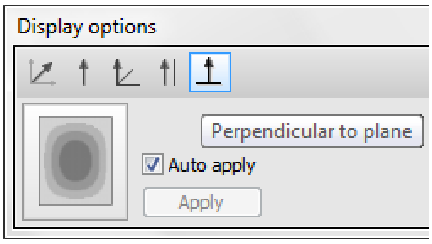

Eurocode code: Load cases - case 1 - (U) - Translational displacements Abs Colour palette - [mm]
8.149
6.519
4.889
3.259
1.630
0.000
-1.630
-3.259
-4.889
-6.519
-8.149

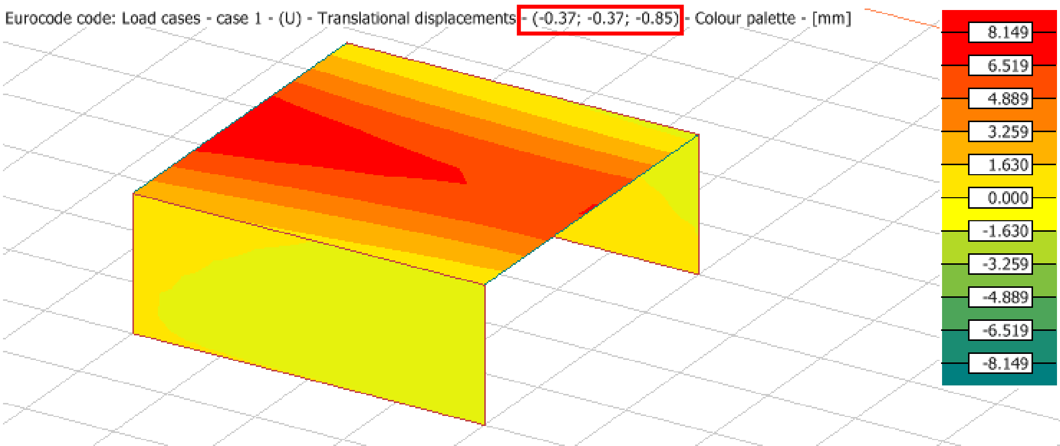

Display options
Predefined direction
Auto apply
Apply
Positive direction
Negative direction
Display options
Predefined direction
Auto apply
Apply
Direction is parallel with global X axis
Direction is parallel with global Y axis
Direction is parallel with global Z axis


Display options
Object's local system
Auto apply
Apply
Positive direction
Negative direction
Display options
Object's local system
Auto apply
Apply
Direction is parallel with local x' axis
Direction is parallel with local y' axis
Direction is parallel with local z' axis


Eurocode code: Load cases - case 1 - (U) - Translational displacements - z (+) - Colour palette - [mm]
8.149
6.519
4.889
3.259
1.630
0.000
-1.630
-3.259
-4.889
-6.519
-8.149



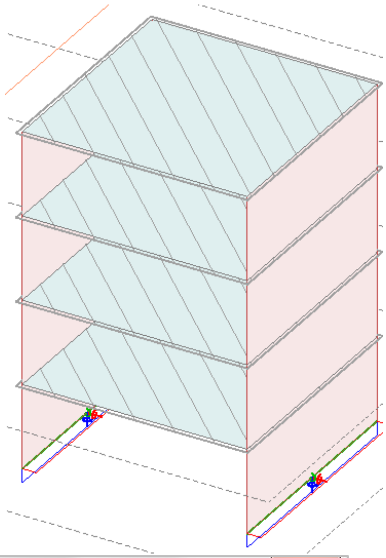




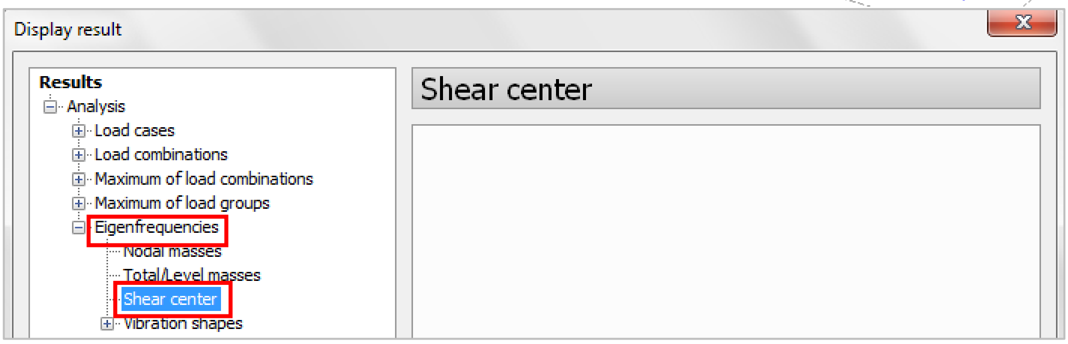
The *Auto apply* can be turned off in case of huge models, to avoid long-time refreshing.

8.3. Shear center result

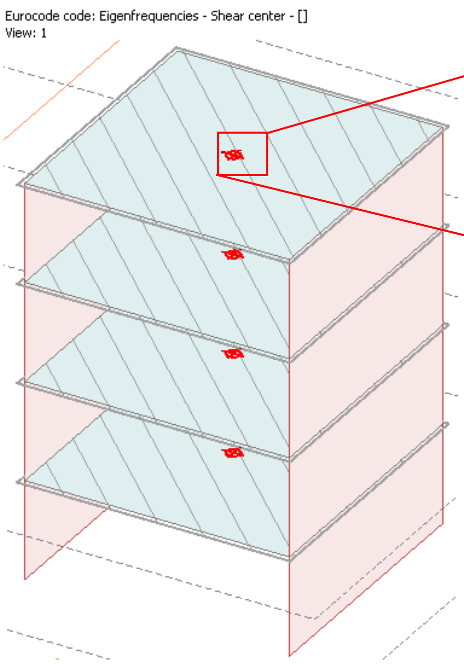
In FD 17, the shear center can be displayed for every storey.

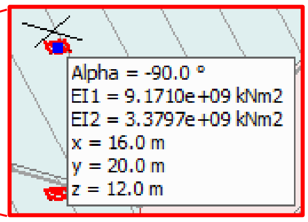


⚠ For displaying shear center, diaphragms are needed for every storey.



⚠ 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.

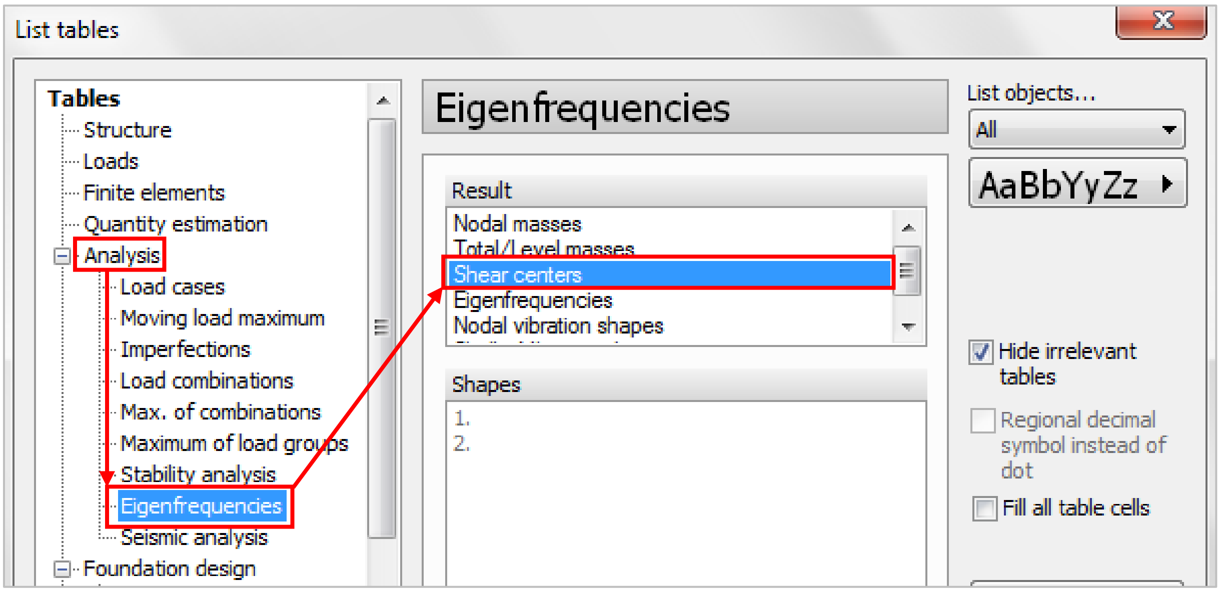




Values in the Tooltip:

- 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.*



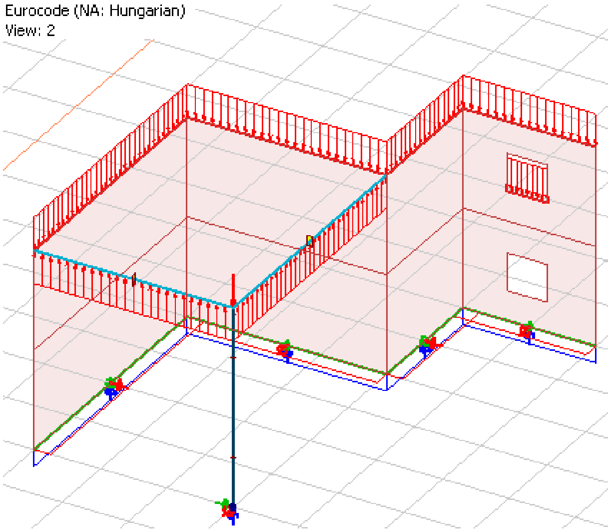
Shear centers

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

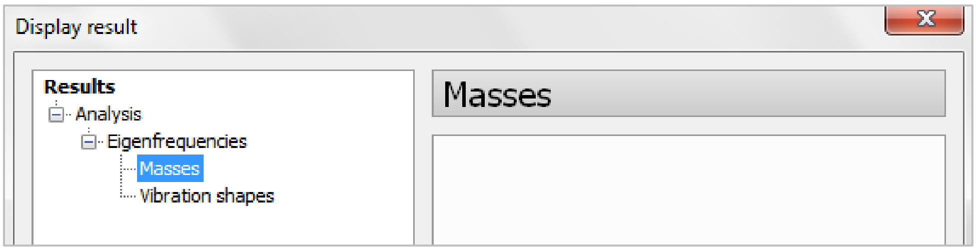
8.4. Mass result

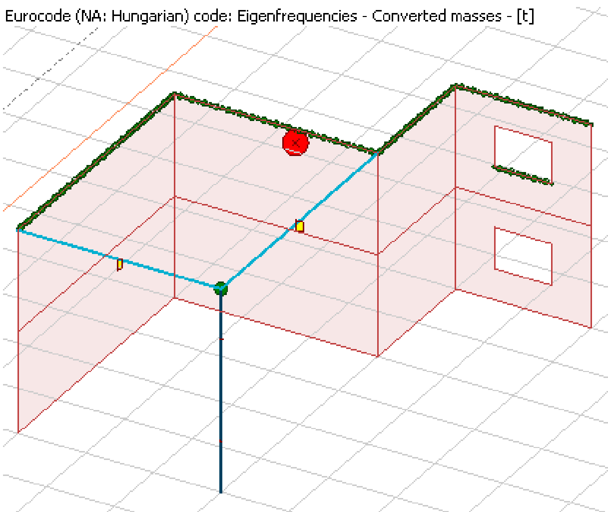
In FEM-Design 17, Masses result is renamed to Nodal masses, and new result types: Nodal masses and Total / Level masses are available,

Loads:

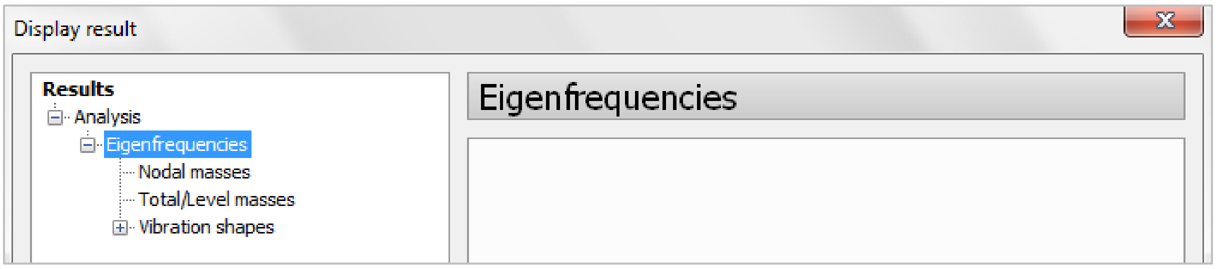


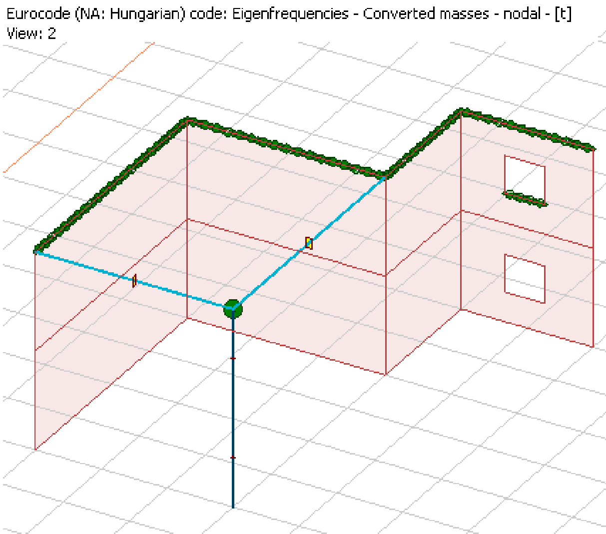
In FD 16:

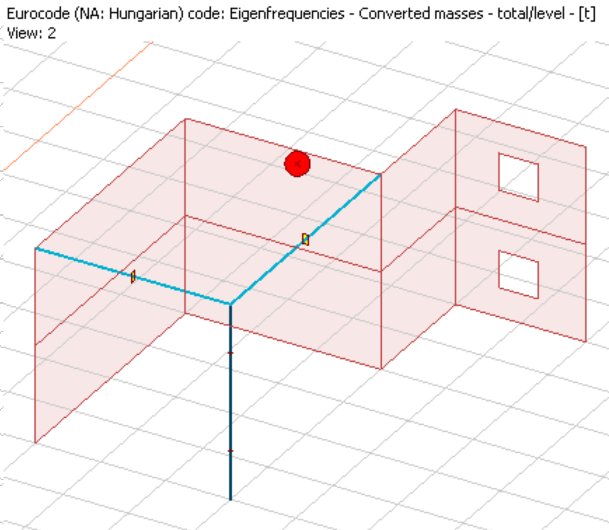




In FD 17:


Display result
Results
Analysis
Eigenfrequencies
Nodal masses
Total/Level masses
Vibration shapes
Eigenfrequencies


Eurocode (NA: Hungarian) code: Eigenfrequencies - Converted masses - nodal - [t]
View: 2

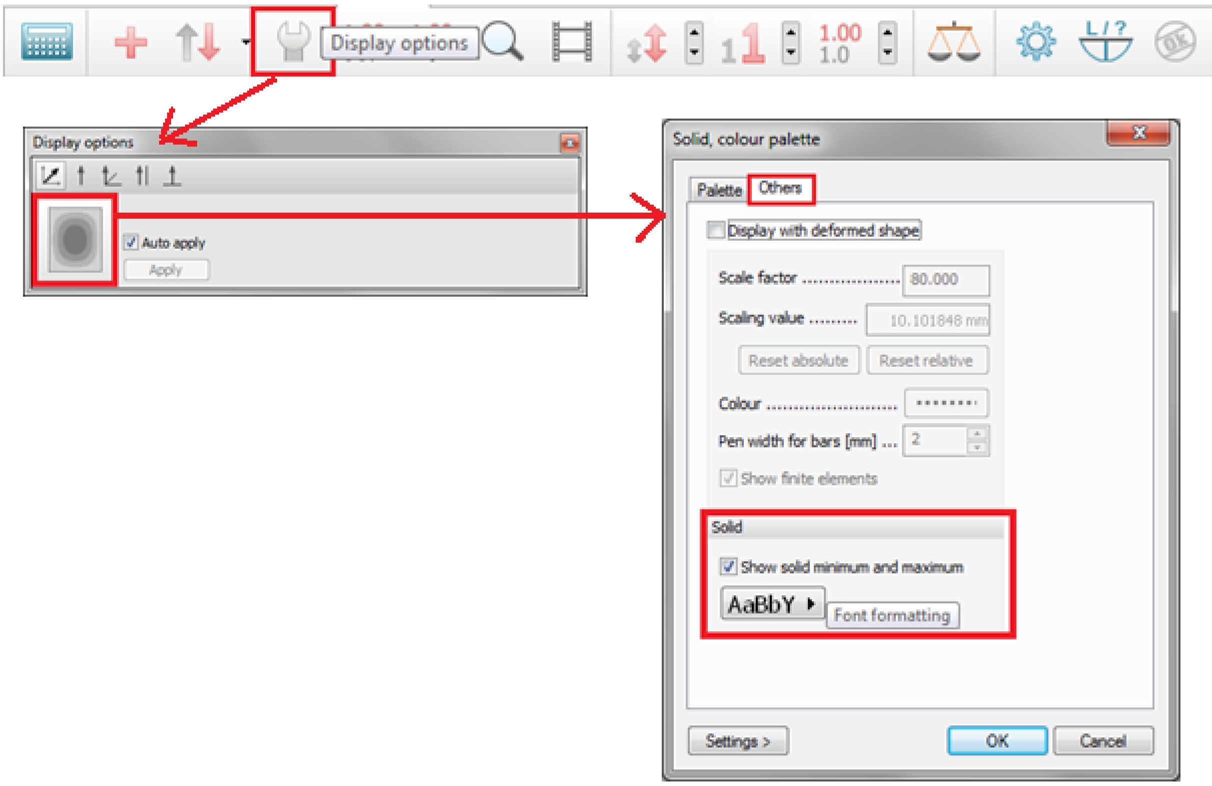

Eurocode (NA: Hungarian) code: Eigenfrequencies - Converted masses - total/level - [t]
View: 2

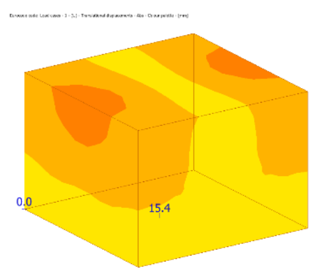
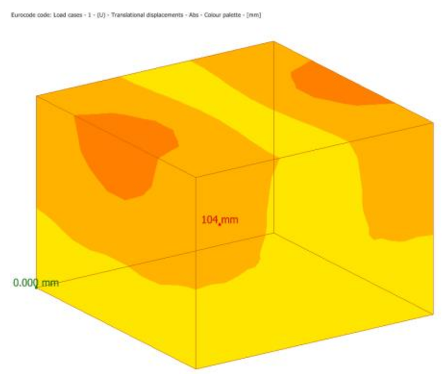
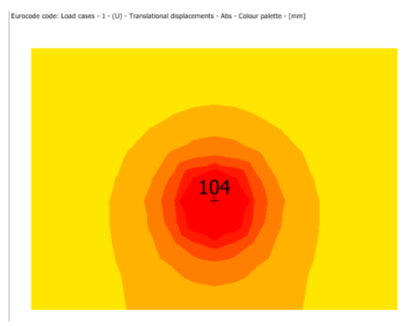
8.5. Minimum and maximum values of soil results in colour palette view

In the earlier versions, the results (translations or stresses) of solid (volume) elements in colour palette showed only the results on the sidewalls of the volume.

In some cases the **minimum/maximum values are inside the volume** (not on the surface) therefore, now the minimum/maximum **points are indicated**, and are showing the positions (coordinates) and the minimum/maximum values. Using these points: User can for example make a 2D view section where the minimum/maximum value can be visualised.

This new feature is available for *Translational displacements* or *Solid stresses* results in *Display options/ Default settings/ Others* dialog. User can set the font size and type, too.

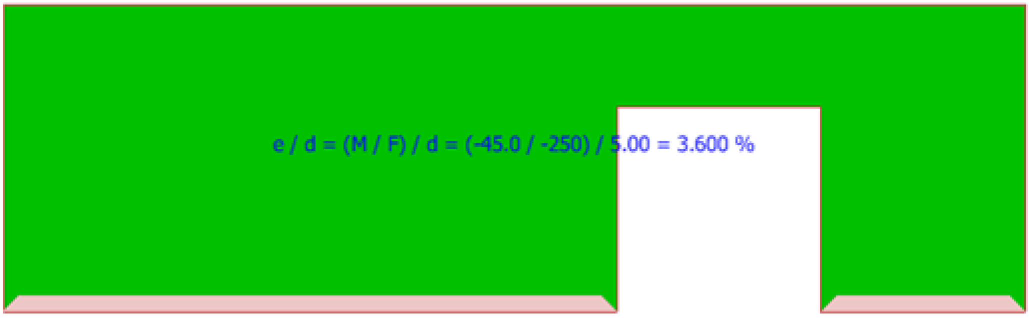


Result displaying auto min/max stresses	Result displaying auto min/max translations	2D view through the maximum point
		

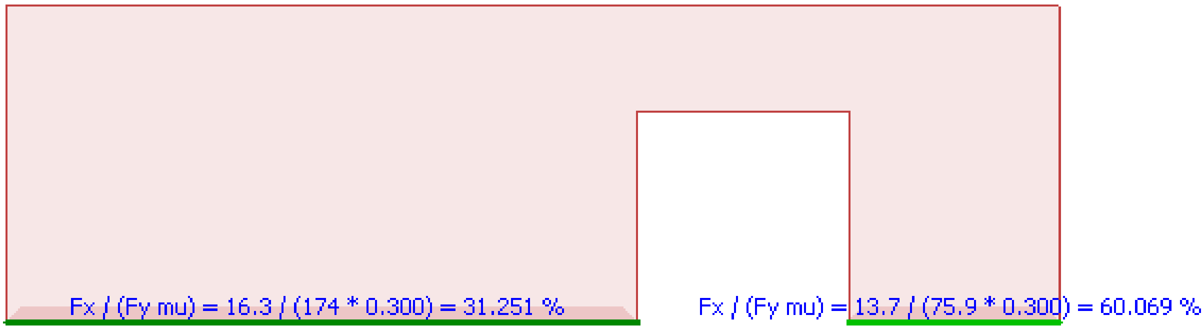
8.6. Local stability results with more details

The maximum value and detailed calculation of the result is displayed.

Eurocode (NA: Hungarian) code: 1st order theory - Load combinations - U - Local stability - Overturning of walls - [%]



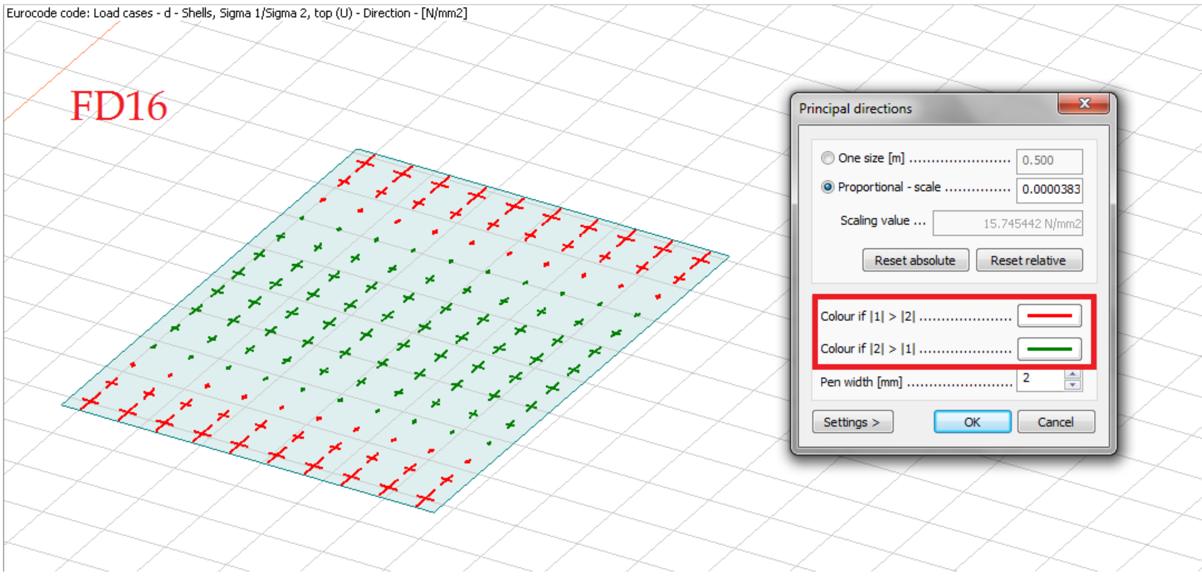
Eurocode (NA: Hungarian) code: 1st order theory - Load combinations - U - Local stability - Sliding - [%]

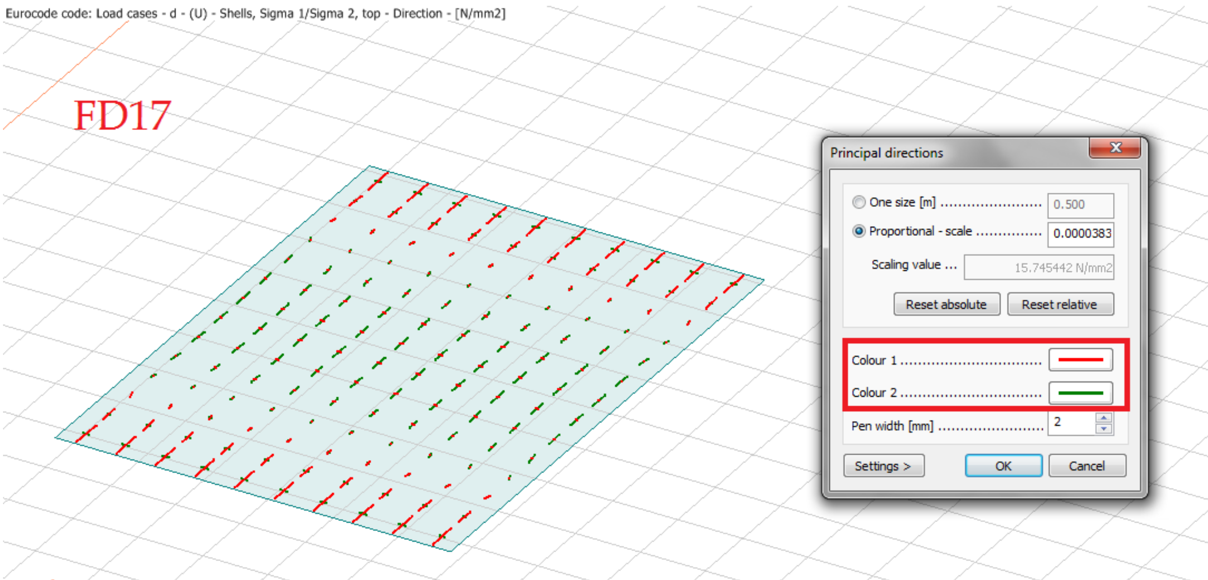


The detailed calculation can be hidden by unchecking the 'Show extra information, if available' box in the Numeric Value dialog.

8.7. Colours of principal stresses, moments and normal forces

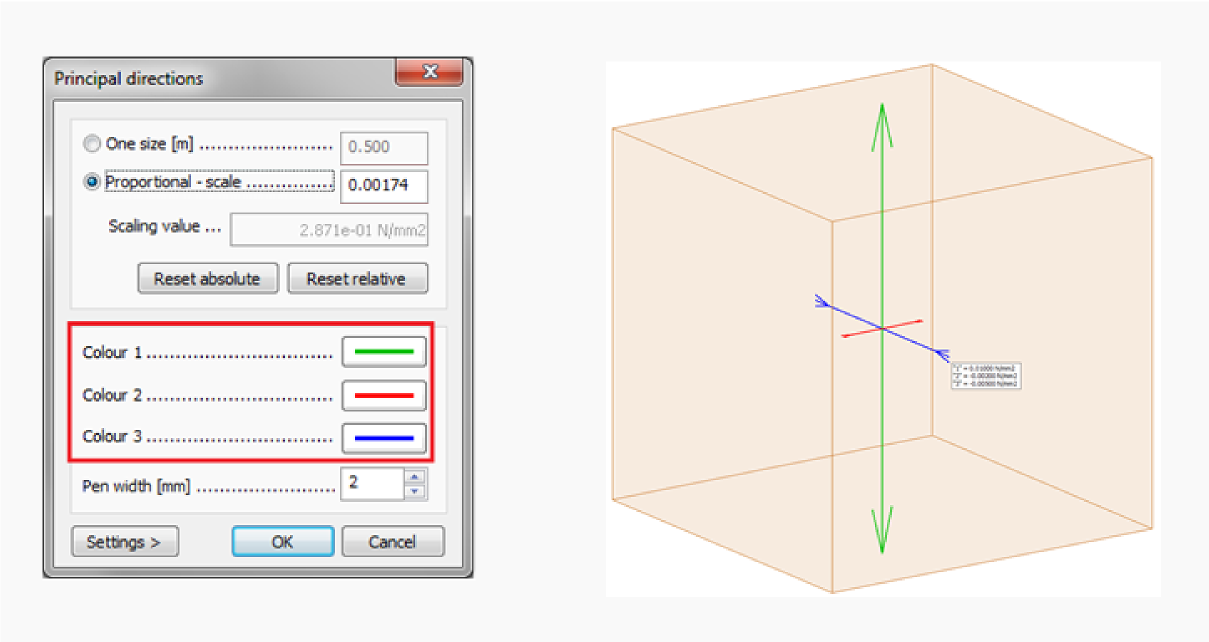
Colouring of principal stress, moment and normal force results of shells has been modified to make the result more clear. The algebraically **larger value** will be **the first principal value,** by default indicated with green colour**.** The algebraically **smaller value** will be **the second principal value**, by default indicated with red colour**.** Difference in the *Display options* dialogs and the results is shown on the picture below.





Display options dialog for principal stresses of solids has been modified in a similar way as for shells, but in this case, three colours can be set for the three principal stresses.

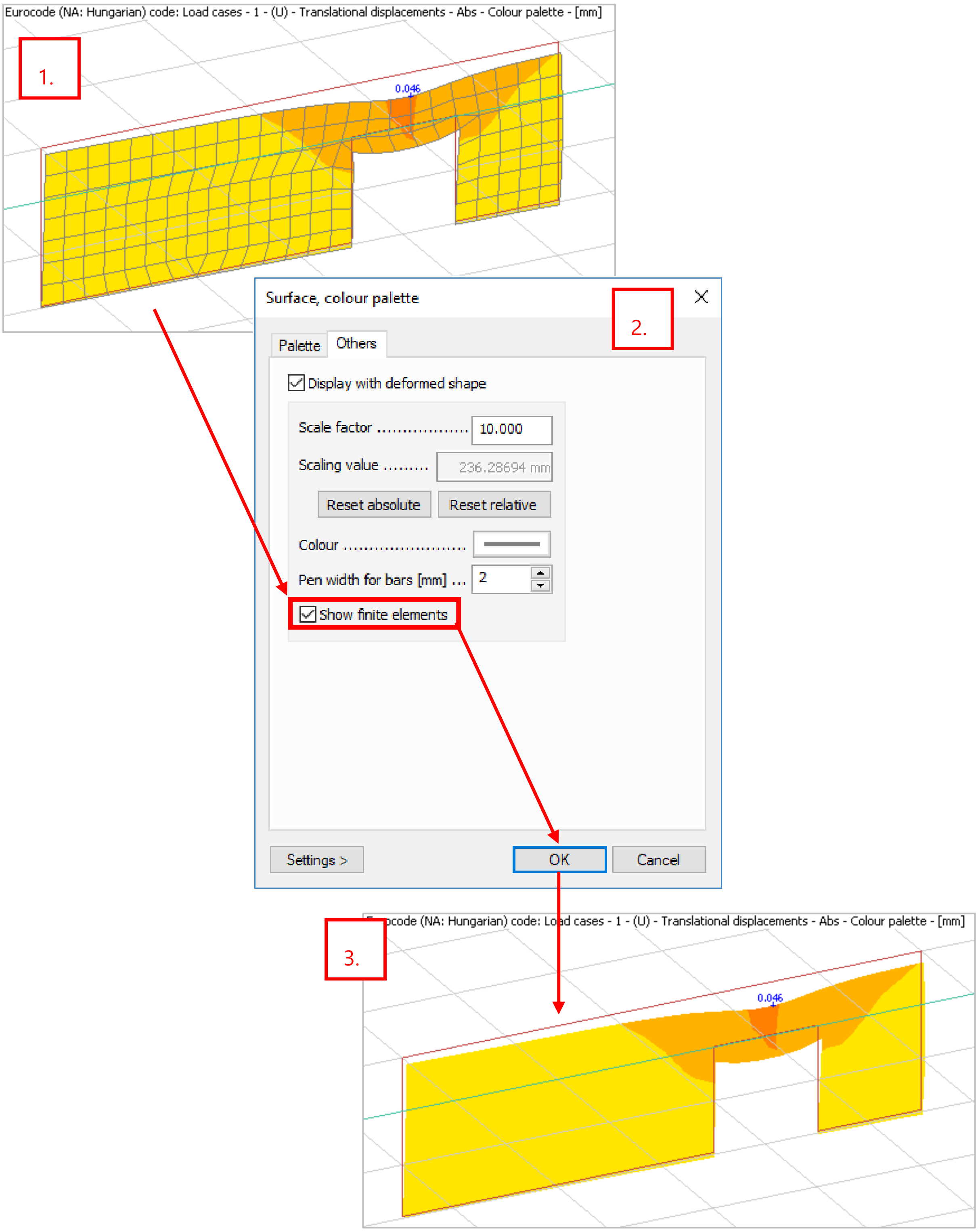
Arrows at the ends of the lines are indicating whether the principal stresses are tension (positive) or compression (negative).


Principal directions
One size [m]
0.500
Proportional - scale
0.00174
Scaling value ...
2.871e-01 N/mm2
Reset absolute
Reset relative
Colour 1
Colour 2
Colour 3
Pen width [mm]
2
Settings >
OK
Cancel

8.8. Option to hide finite elements for colour palette results

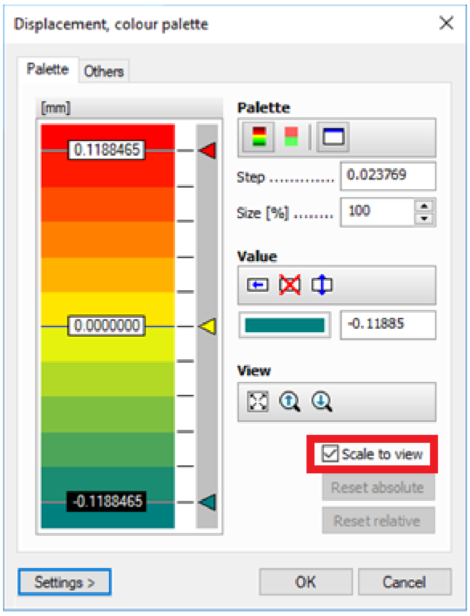
To hide the finite elements from the result, click on the  icon. In the pop-up window, *Surface, colour palette* click the *Others* tab and uncheck the box next to *Show finite elements,* then click 'OK'.

Picture No. 3. shows the result without the finite elements grids.

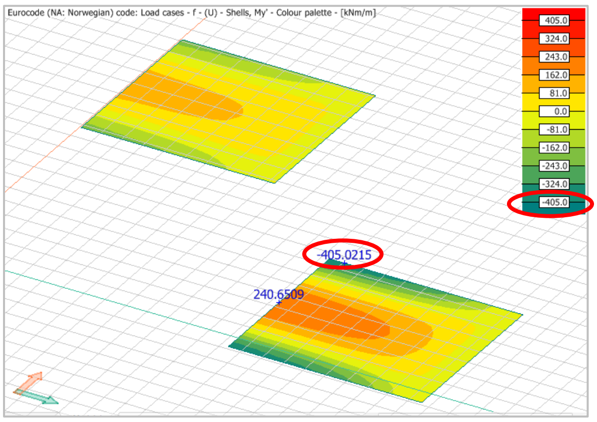


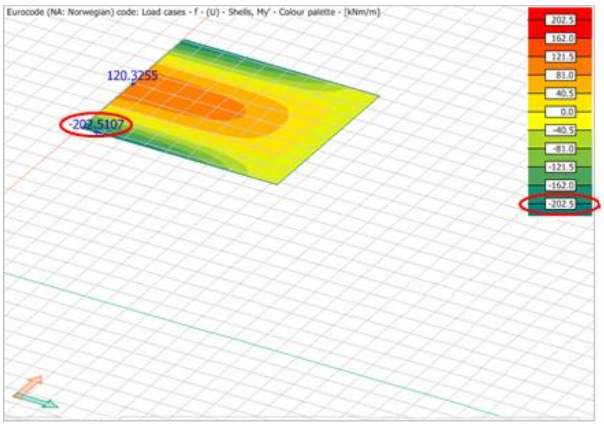
8.9. Scale to view option for colour palette results

In FEM-Design 17 for results displayed by colour palette, a new option is available in the *Display options* dialog. It automatically rescales the results considering the maximum and minimum values of the current view.



With *Scale to view* option turned ON, if User hides part of the structure, the palette will be rescaled automatically:





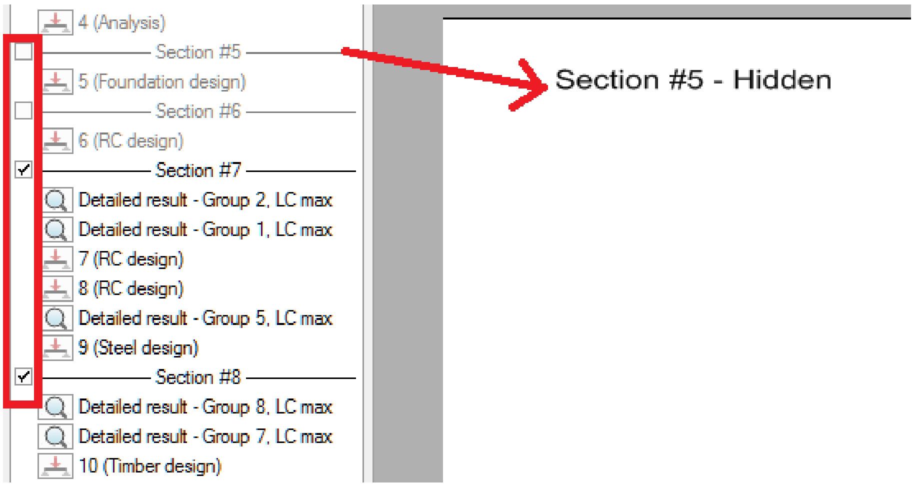
If the result to be displayed needs wider scaling than the customized has, the program overwrites the scaling to the view.

9. Documentation

9.1. Option to hide sections

In order to speed up work in Documentation, Sections can be hidden/shown by clicking the checkbox next to the section's name.

The items' titles of the hidden Sections are listed with grey color and their contents are not visible in the documentation. Only a comment on a single page shows that the Section is hidden. Consequently, if a Section gets hidden, the page numbering of the Sections following it, changes.



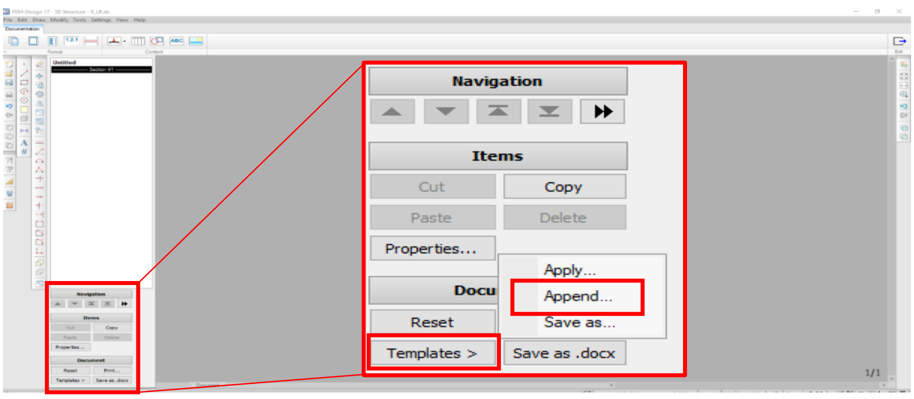
The hidden sections are not locked thus, they can be cut, copied, pasted, moved, dropped or modified. If an active section is moved into a hidden one, both sections will be hidden and vice versa.

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.

9.2. Append templates

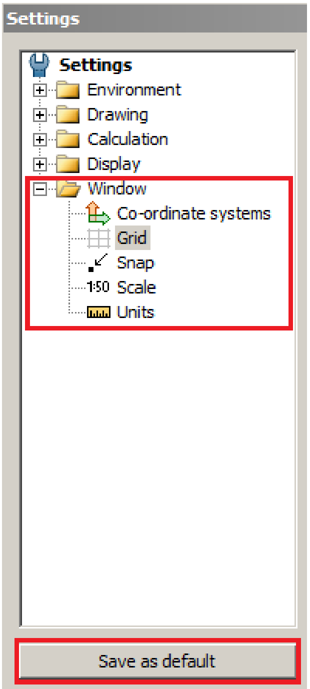
From now on, User can append previously saved Templates in Documentation. Items of the appended templates will be inserted after the last existing document item.

The *Append* function can be found at the *Templates* menu in the bottom-left corner:

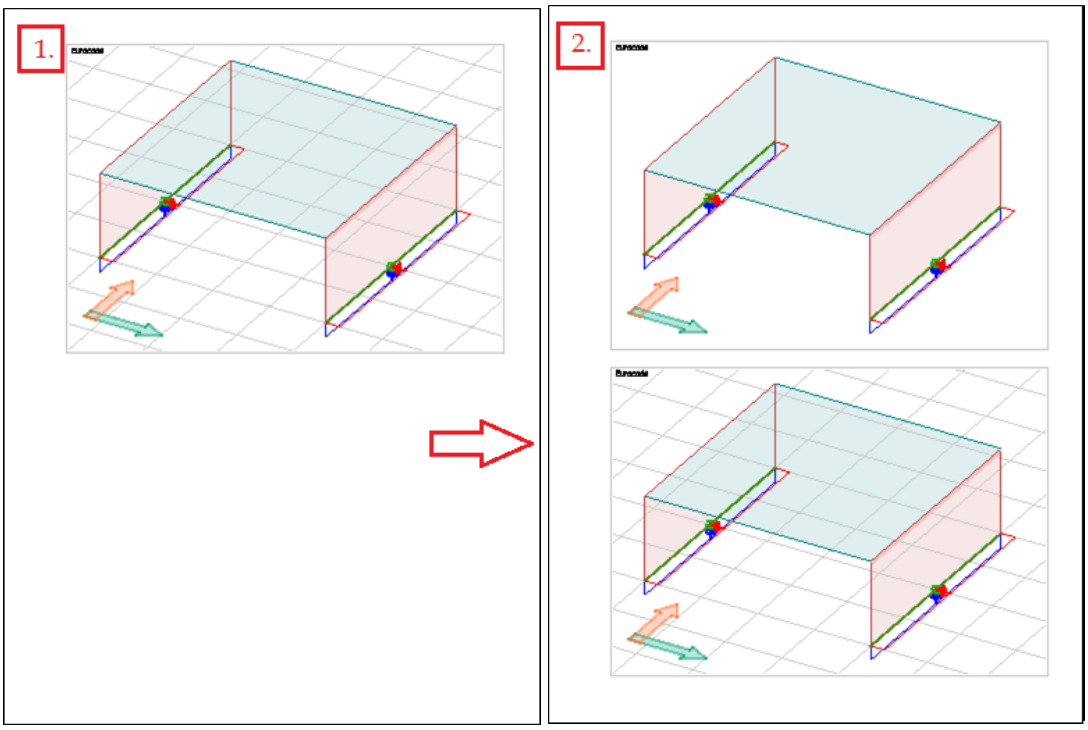


9.3. Docgraph windows are created according to the saved Window settings

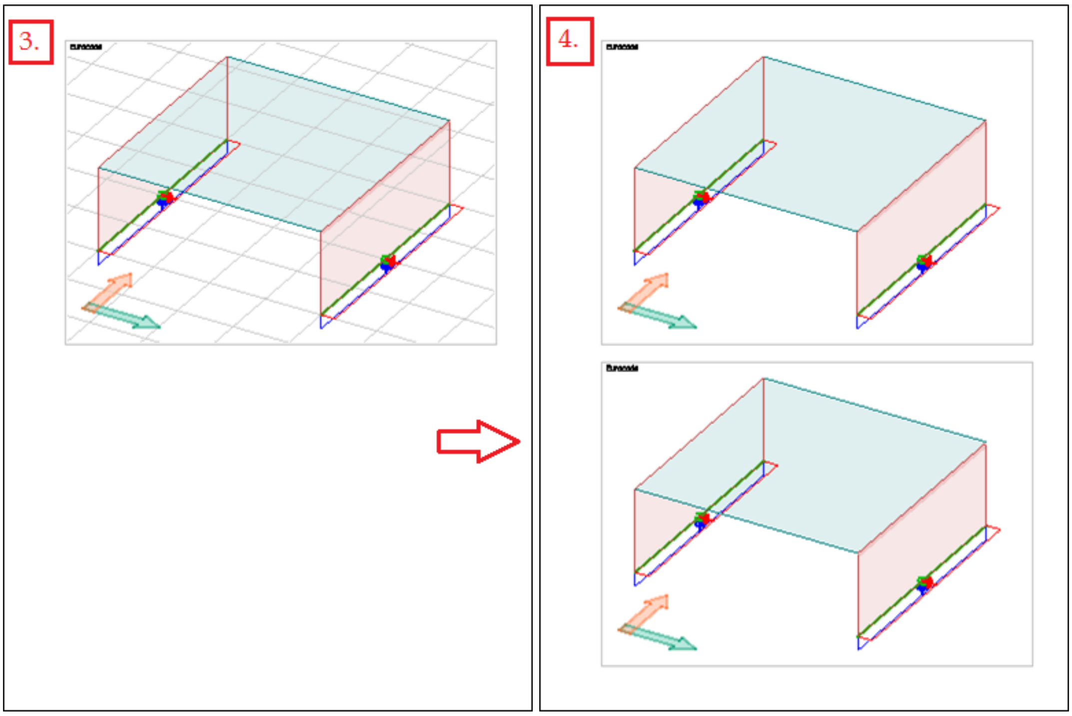
User can save the Window settings for Docgraphs as default by clicking the *Save as default* button in the *Settings* dialog.



Pictures 1 and 2 show that, if User turns off the grid but doesn't save that setting as default, the next inserted docgraph will be displayed as the first one was displayed originally.



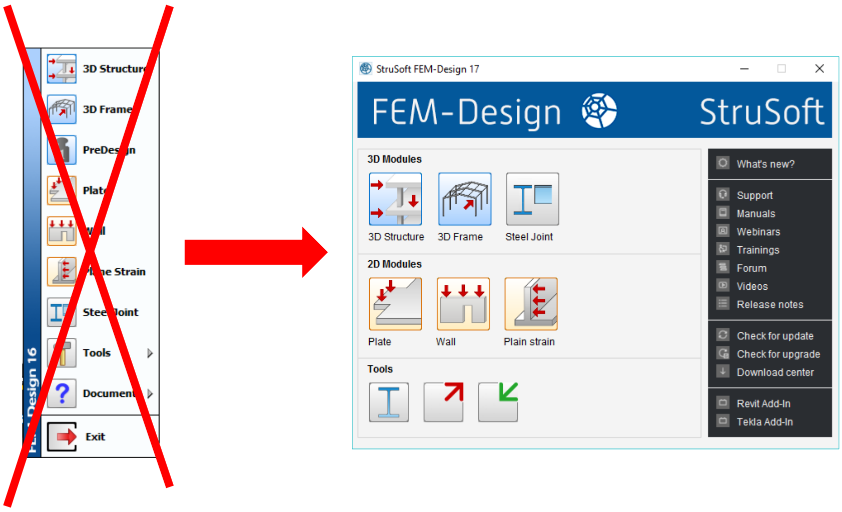
If User saves the customized settings as default, every next inserted docgraph, will be displayed according to the saved properties. (See picture 3. and 4.)


3.
4.

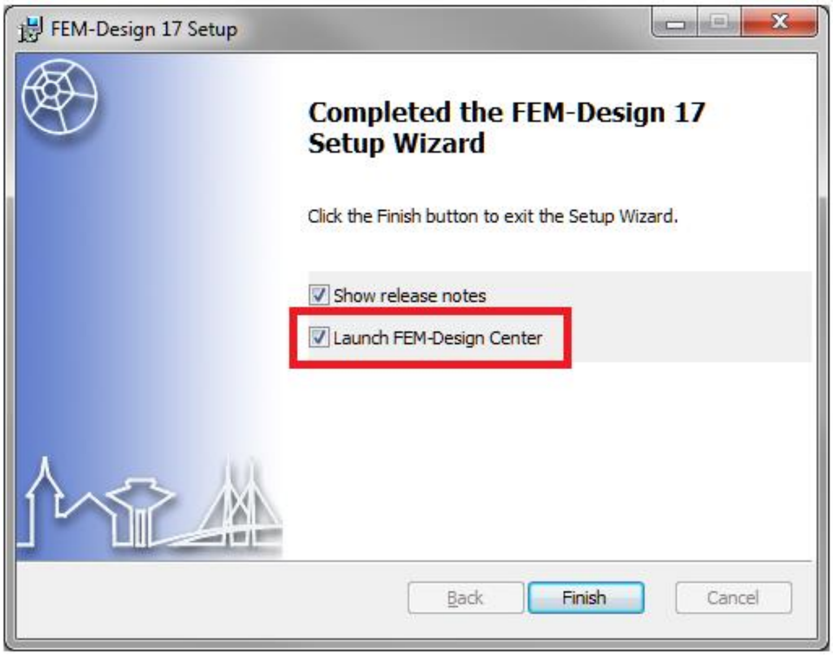
10. Others

10.1. FEM-Design Center

FEM-Design systray has been removed and replaced with a new, more informative tool called FEM-Design Center. All the modules, useful links, updates and links to the add-ins are now in one place.



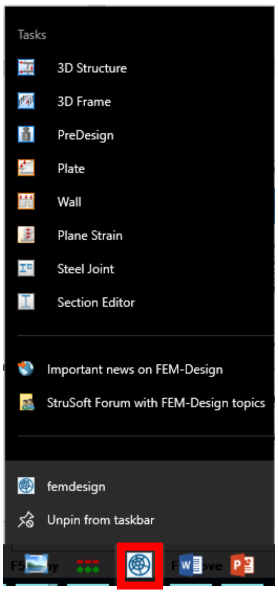
At the end of FEM-Design installation, user can select an option to launch the FEM-Design Center and pin it to the taskbar for easier access, if necessary.



CTRL+Left mouse button will open the last model.

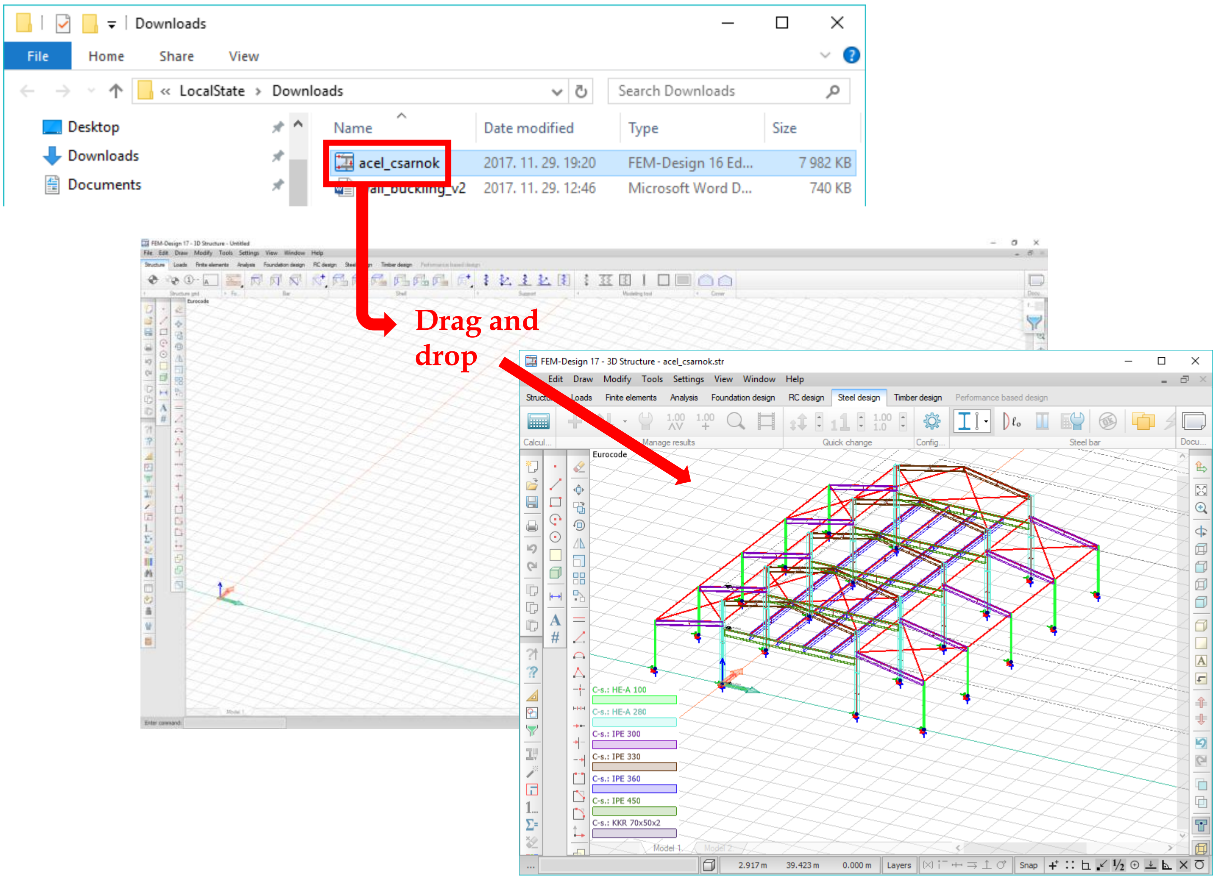
Shitf + Left mouse button will open history in a jumplist.

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



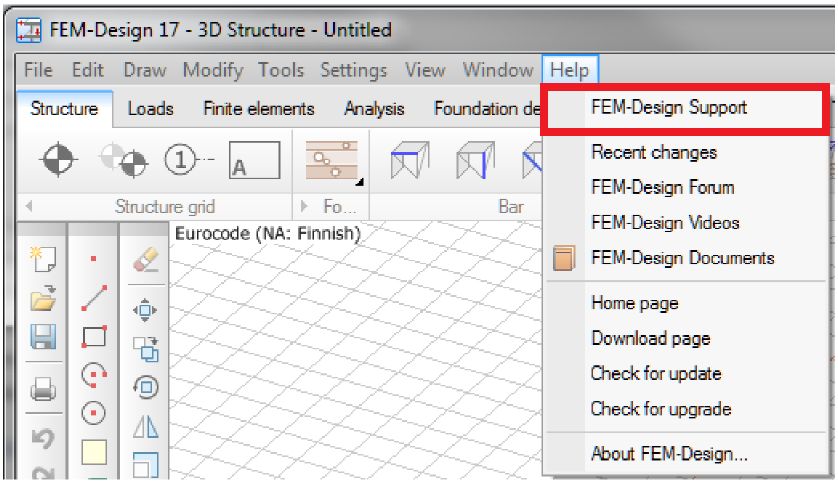
10.2. Drag and drop

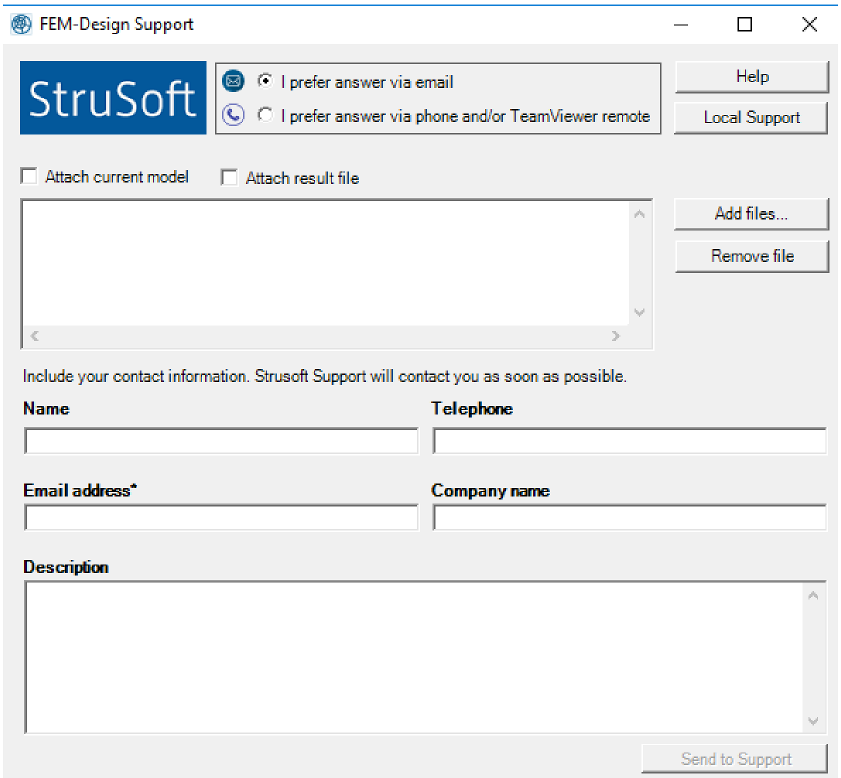
In FEM-Design 17 “drag and drop” function is implemented. Files in all supported formats can be now simply dragged and dropped to the FEM-Design.



10.3. FEM-Design Support tool

This tool helps the User to reach the Support team easily and get answers for specific model dependant questions. This can be found at *Help/FEM-Design Support* menu.





User can decide how to receive answer, via email or phone and/or TeamViewer (remote desktop

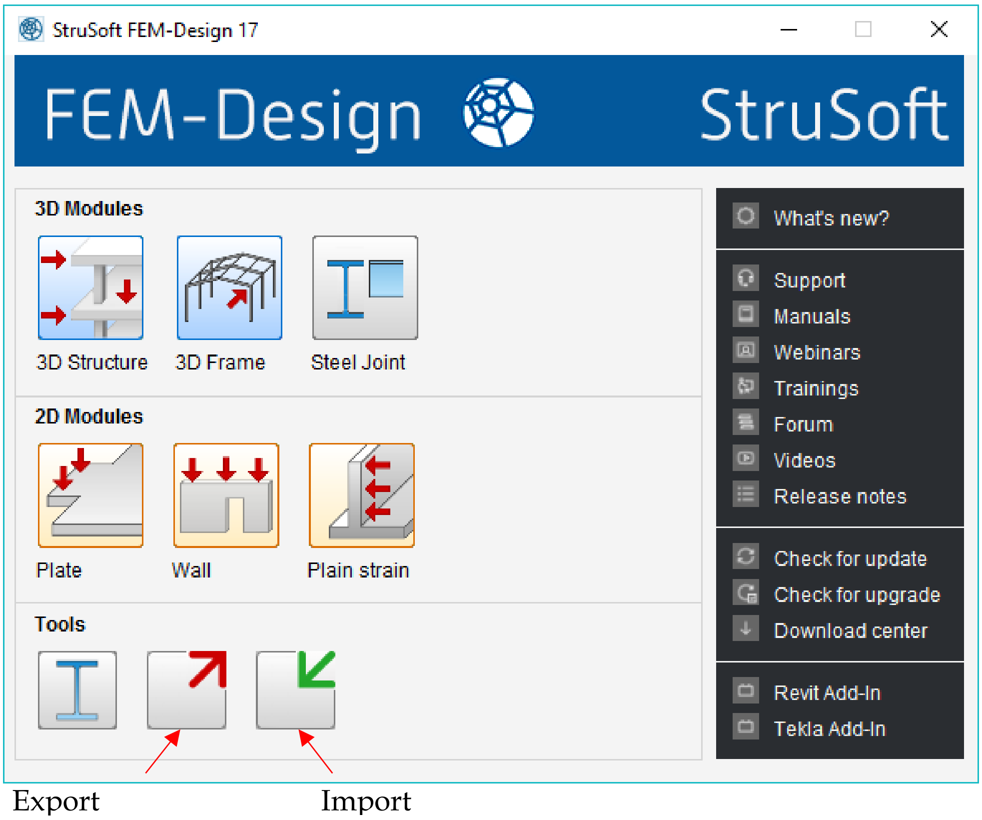
If the *Attach current model* is active, the current, saved model will be send to the support along with the report. If the model includes linked DWG files, those files will be attached as well.

Add Files allows to add more files (e.g. pictures) to the report.

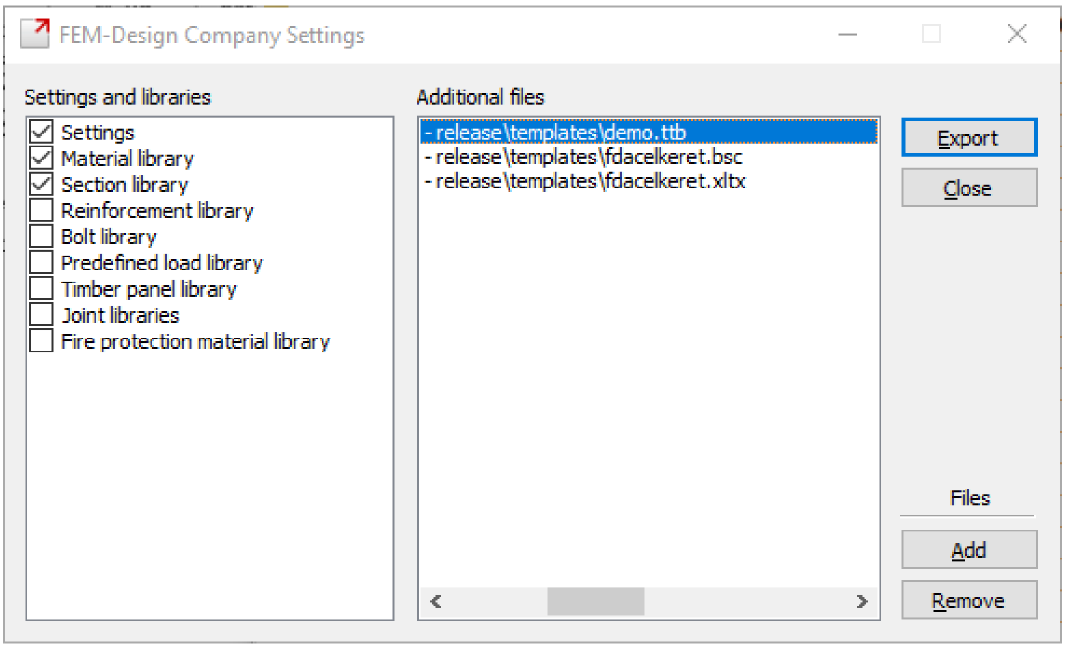
The *Help* button links to a guide on how to use this tool. There, one can find more detailed explanation of the tool functionality and its limitation.

10.4. Company settings

It is a very fast way to export all FEM-Design settings and libraries to any other computer. With this feature User can deliver their company preferences from one computer to another very easy and quickly.



Export and import settings can be found in the FEM-Design Center. The *company settings* file has **.fdcs* format.



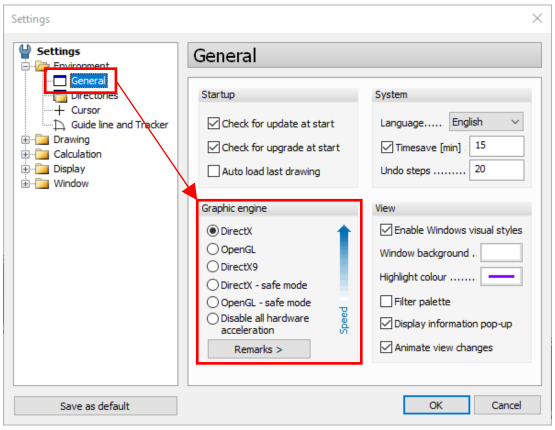
The following items can be exported:

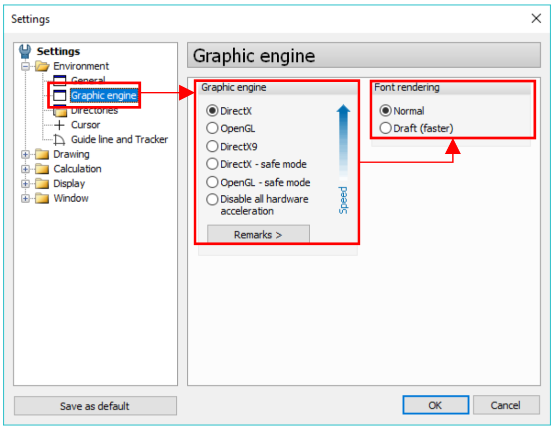
- settings
- libraries
- doc templates
- list batches
- title blocks
- list templates
- Office OpenXML templates

10.5. Graphic engine settings and Draft mode

The graphical settings moved to Environment/Graphic engine and two font rendering option is available. By using the Draft mode, drawing the texts to the screen is much faster but the quality is bit lower.

Version 16 | Version 17





10.6. Timesave

Timesave (backup) in FEM-Design 17 will save only the input file and not the result file anymore. This ensures a quite fast backup of model using less disk space. There is no option to save the result file in case of timesave. To save the result file, please *Save* the model from *File/Save* or *(Ctrl+S)*.