StruSoft
Structural Design Software

This document gives a detailed summary of the new features and improvements of FEM-Design version 16.

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 16?

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

- New SLS combinations
- Colour schema
- Deflection check for RC, steel and timber bars
- Diaphragm
- Local-Global stability shape
- Limit design utilization can be set for steel bars and timber bars and panels
- Lateral torsional buckling calculation according to general case and k_{ij} interaction factors calculated according to Method 1
- Automatic minimum and maximum numeric value display
- Pick properties and Copy properties function
- Romanian national annex
- Lock numbering
- Alternative view rotation by snapped point
- Automatic selection of supporting structures for covers and building covers
- Leading temporary load cases
- Deviation load by load cases
- RC shell reinforcement plane automatically follows the selected result
- Additional section class dependent options at steel bar design
- Save/load steel and timber bar design parameters
- Steel joint manual load combinations
- Crack width result – cracks that exceed the limit are displayed with different colour
- Load comments
- Load import / export
- No snap to objects

`

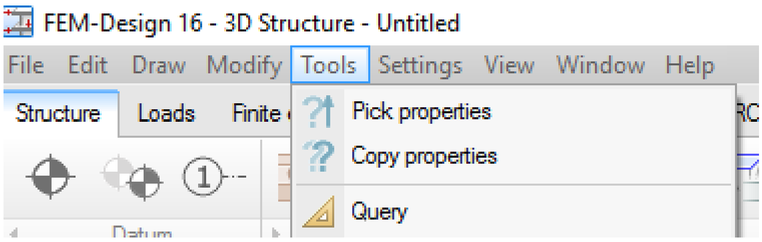
Table of contents

`

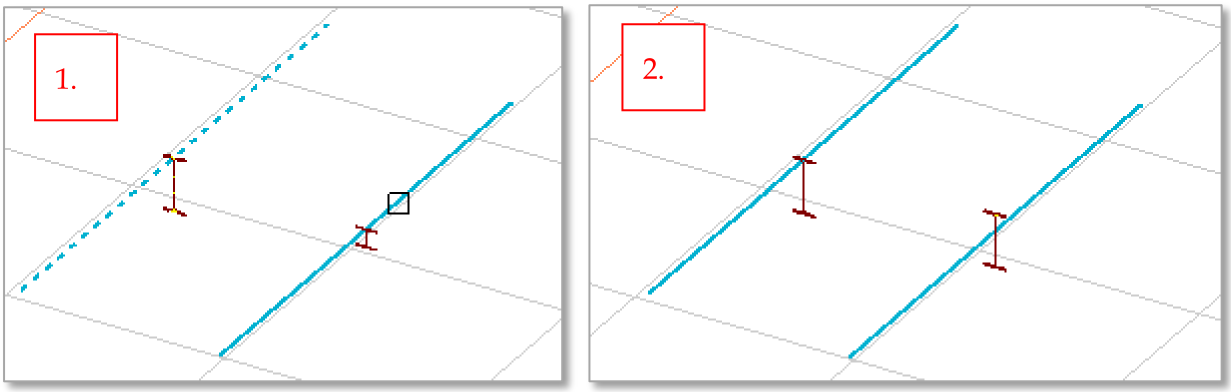
1. Tools

1.1. Pick properties and Copy properties

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

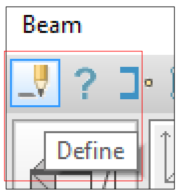


To copy properties with the *Copy properties* ( ) function, first select the source object, then one or more objects to which the properties are to be applied.



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

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

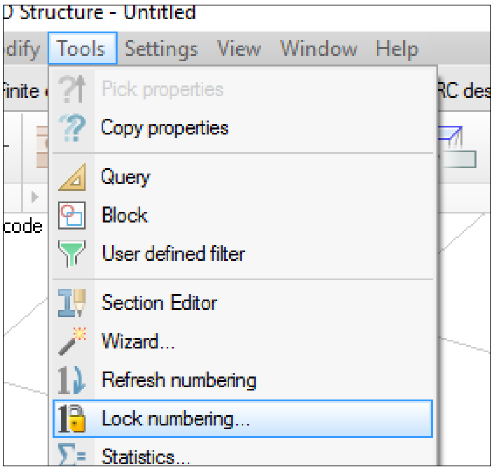


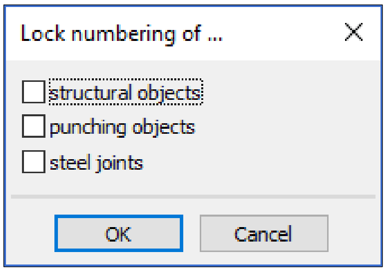
Copy and *Pick properties* work according to the following rules:

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

1.2. Lock numbering

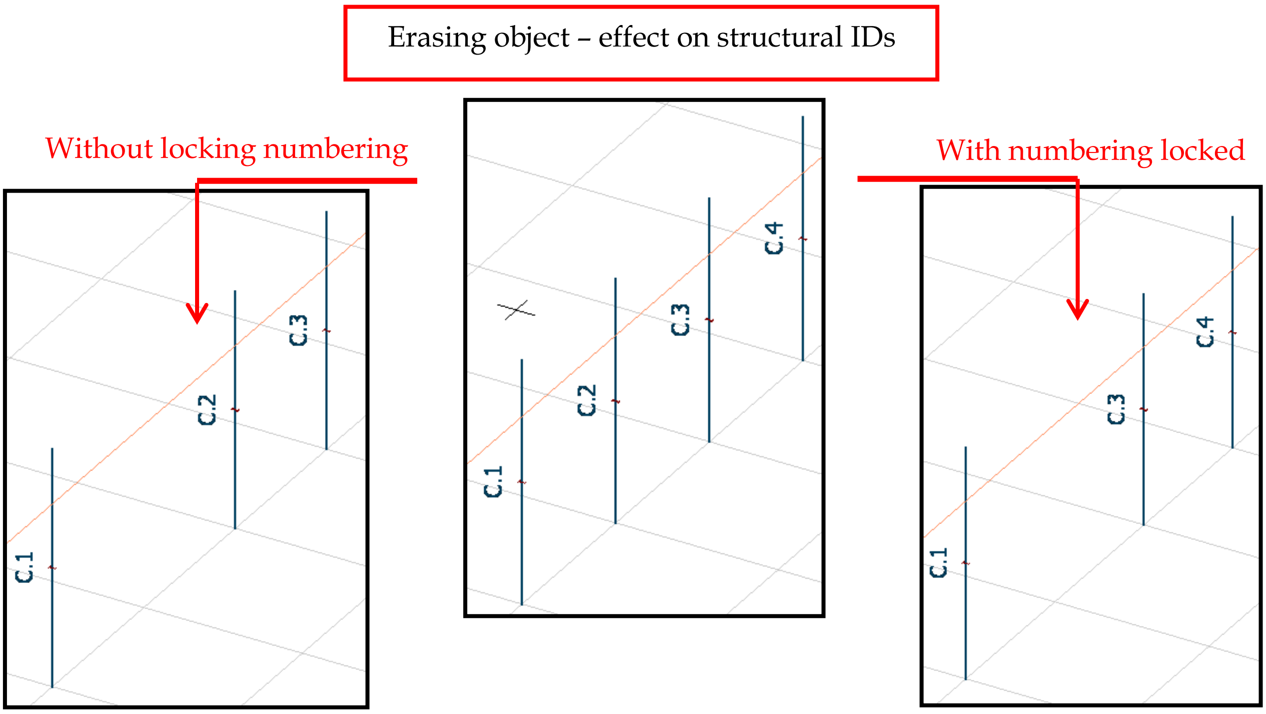
The option to lock numbering is found in the Tools menu or can be accessed by clicking its icon  in the toolbar.





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

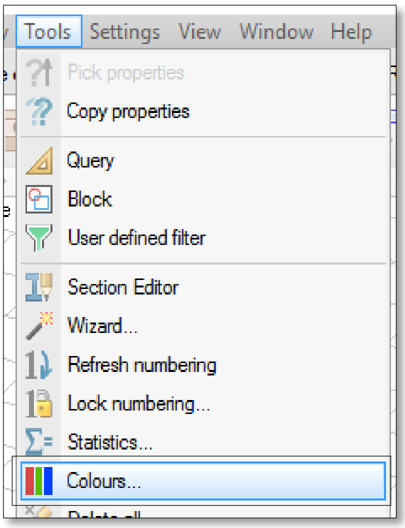
This means all the element types with their numbering locked will retain their original ID, even after refreshing the numbering (either manually, or automatically, e.g. when creating a list).

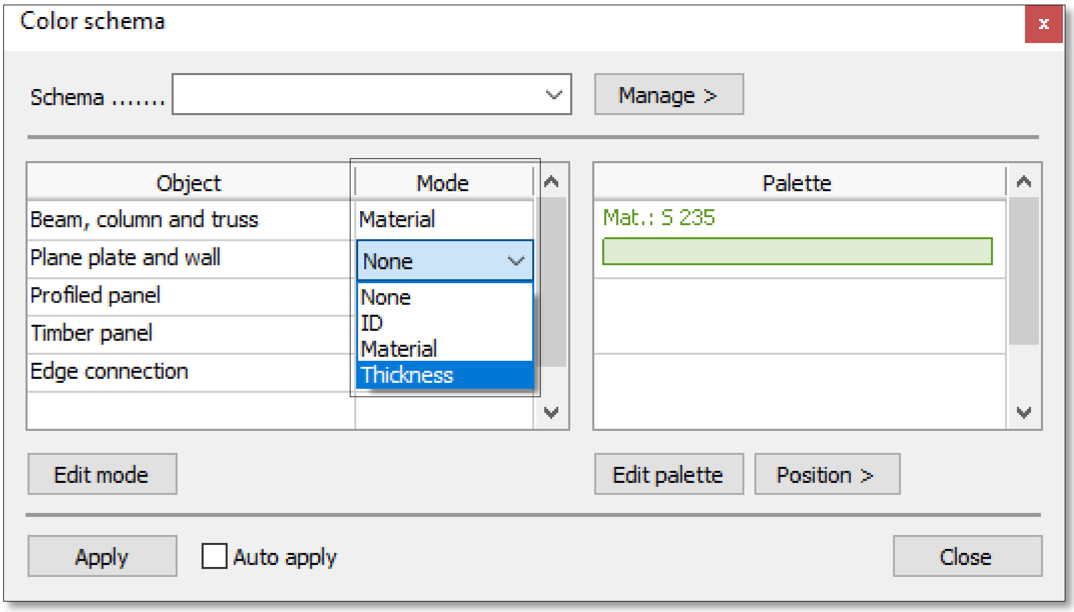


1.3. Colour schema

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

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

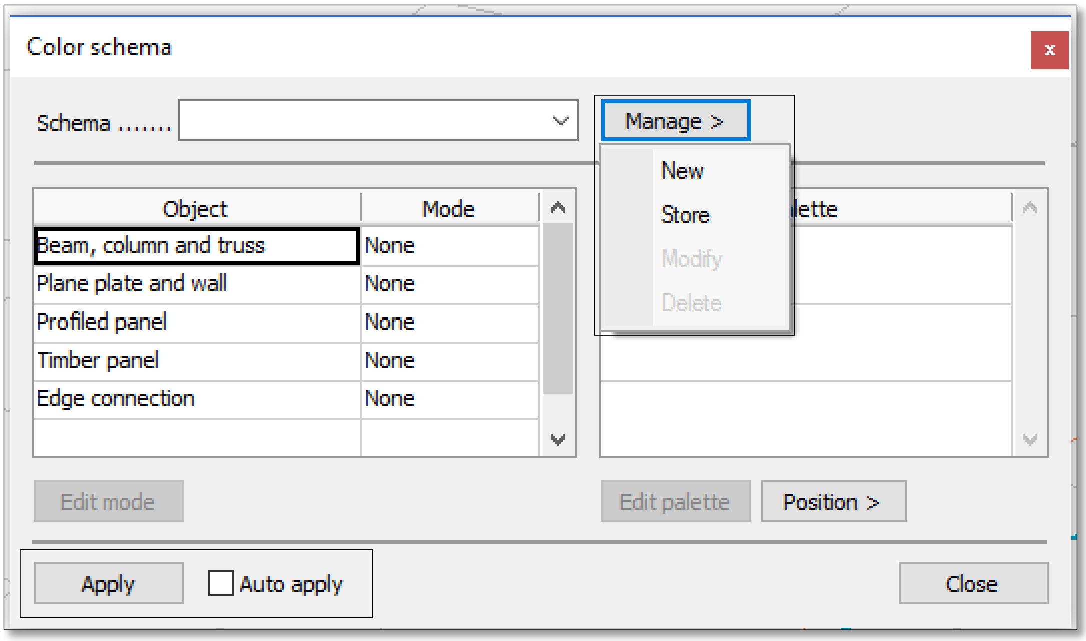




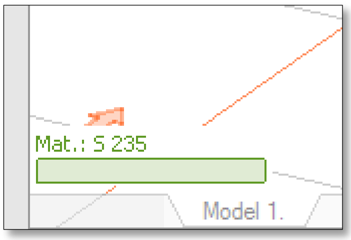
For each object type, the following attributes are available:

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

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

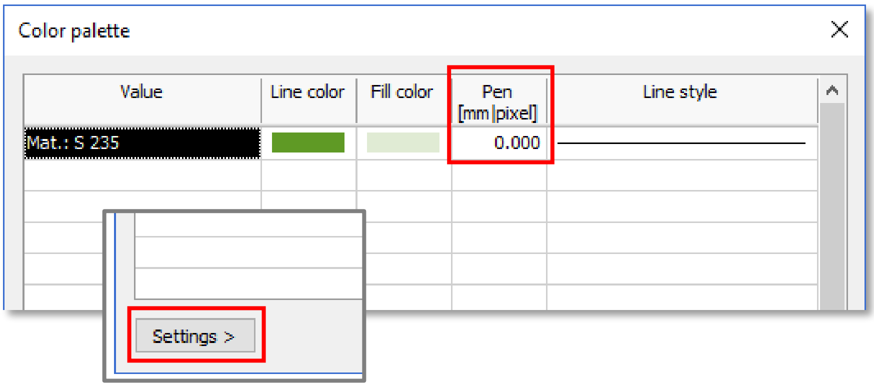


Clicking the *Apply* button will apply the color code to the model. Enabling the *Auto apply* checkbox will apply any changes automatically.



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

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



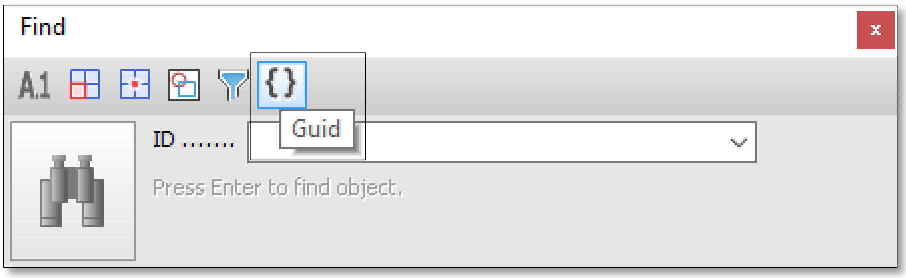
The Settings button will allow you to save your colour scheme to a file (**.cpal*) or as default, or load another schema.

When a colour schema is saved, only the colours and their order will be stored, values need to be specified each time.

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

1.4. Find by Guid

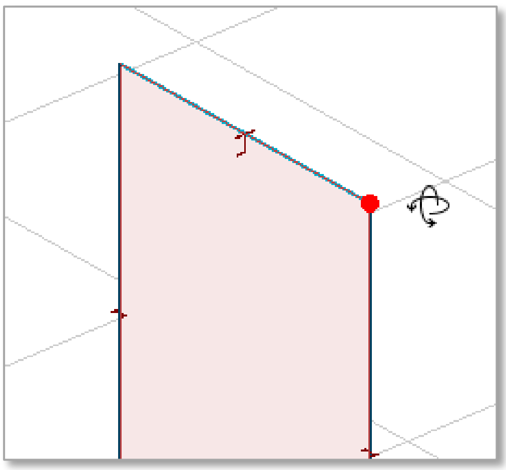
In the *Find* dialogue box it is now possible to search objects by Guid. Since structure elements imported and exported via the StruXml system will generally be defined and referred to, most importantly in error messages, by their Guid, this feature will make finding and eliminating errors easier.



2. User interface

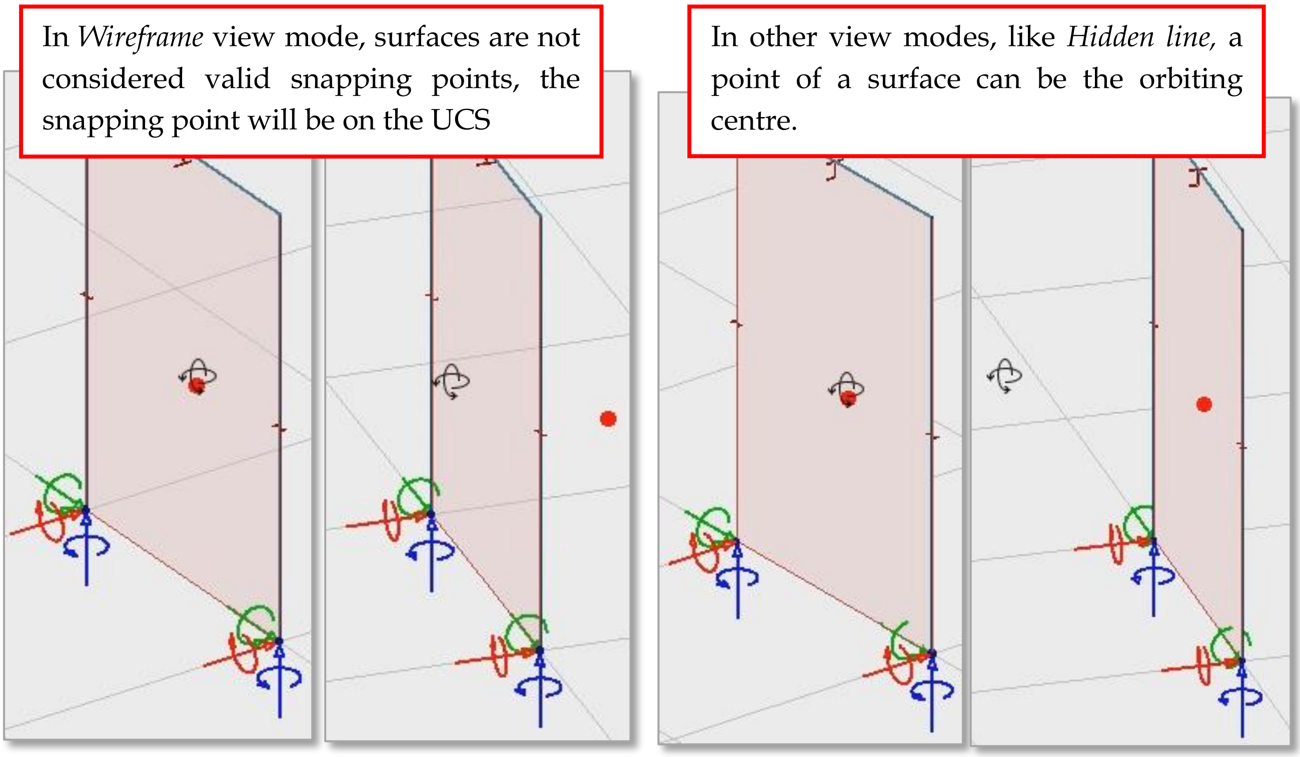
2.1. Orbit around snapped point

Orbit around snapped object is introduced in version 16. Hold down the Alt key, then click and hold the middle mouse button to orbit the model around a snapped point. The 'object' can even be the closest point of a surface. In some cases, this will result in more predictable 3D orbit behaviour. The snapped centre of rotation is indicated by a red dot.



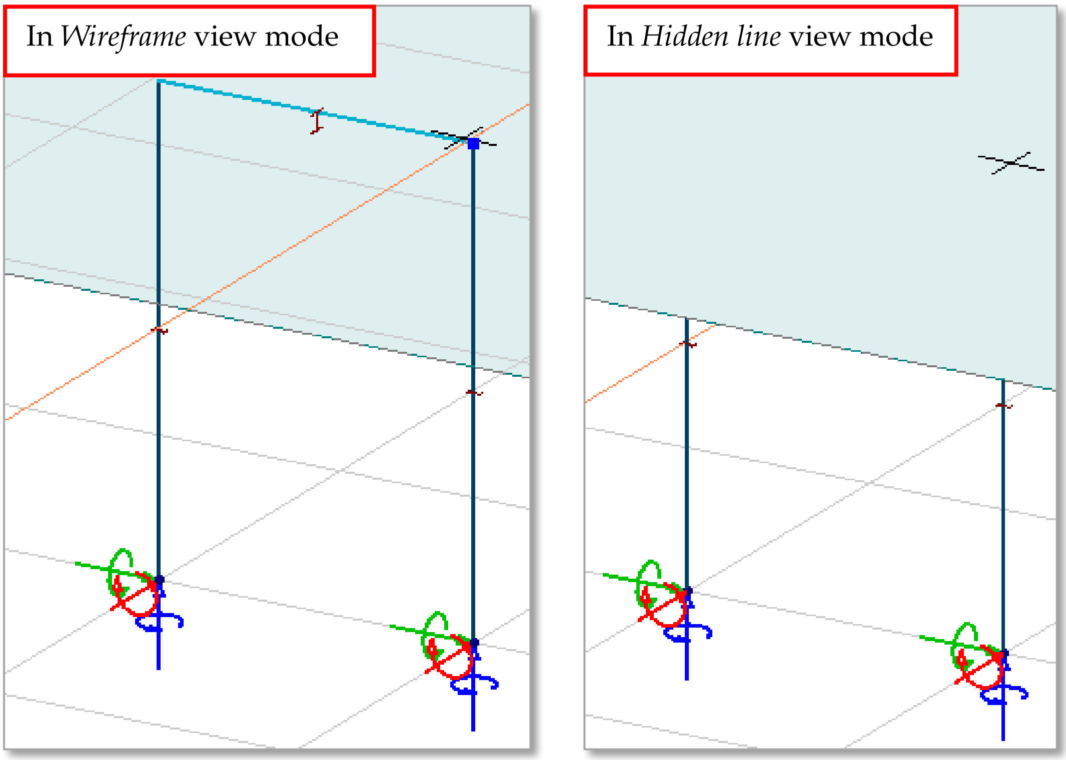
`

Snapping to surface is not enabled in *Wireframe* view mode. The point will instead snap to the User Co-ordinate System.



2.2. No snap on hidden objects

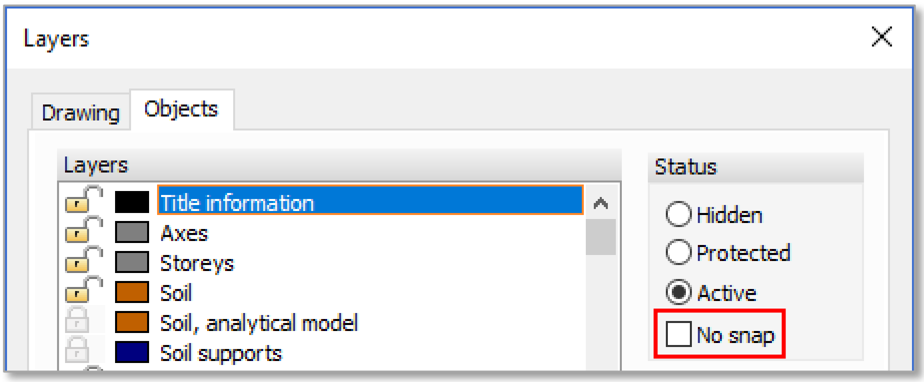
Objects not visible outside of *Wireframe* display mode will no longer be snapped to.



`

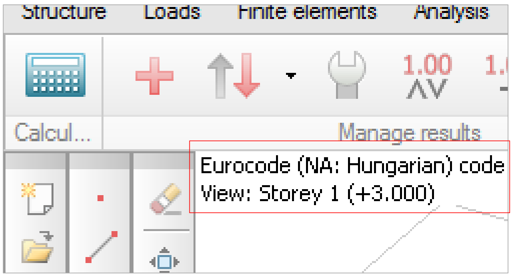
2.3. No snap on *Objects*

In Layers dialog, it is now possible to turn off snapping to objects.



2.4. View name is displayed

The name of the current view is now displayed in the upper-left-corner, under the current code.



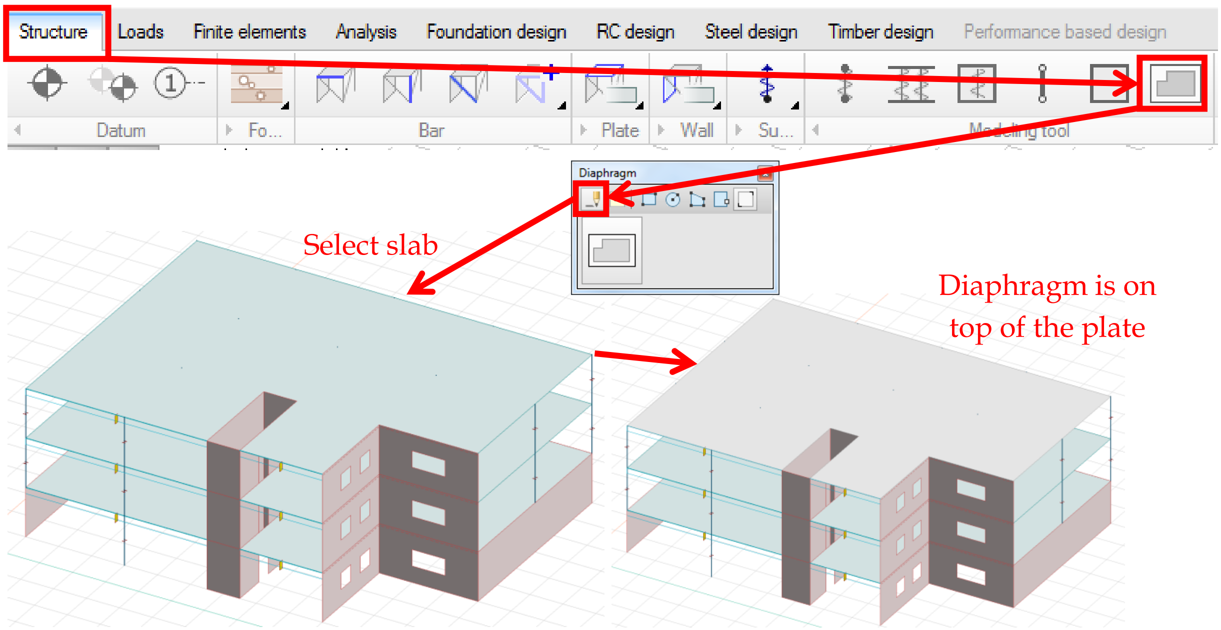
`

3. Structure

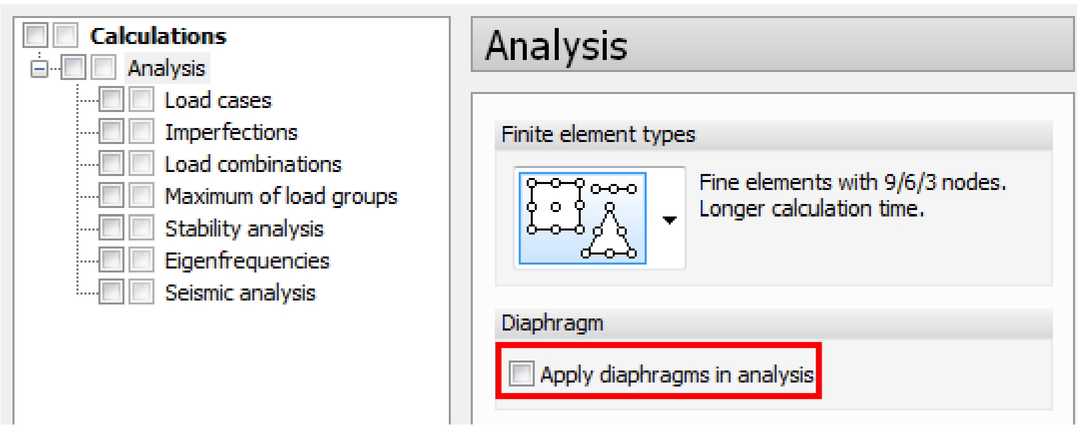
3.1. Diaphragm

Users can model horizontally infinitely rigid regions with the new Diaphragm object. The relative horizontal displacement of the finite element nodes inside the diaphragm region will be 0.

Diaphragm tool can be launched from Structure tab / Modelling tool. The user has to define a region. Finite element nodes inside this region belong to the diaphragm



To consider the diaphragmas in the calculation the User needs to check "Apply diaphragms in analysis" option under *Calculate* dialog.



Limitations:

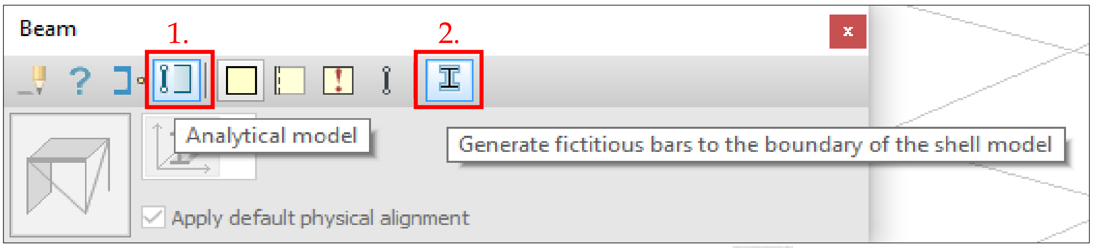
- The diaphragms have to be horizontal.
- The support elements and 3D soil elements are not able to connect to a diaphragm because according to the mechanical behaviour it does not make any sense.

- Eccentricities of plates or bars laying in plane of diaphragm will be ignored during diaphragm calculations.

For information about the theory of diaphragm, please, check the following link.

3.2. Bar-shell model option for fictitious bars in the boundary

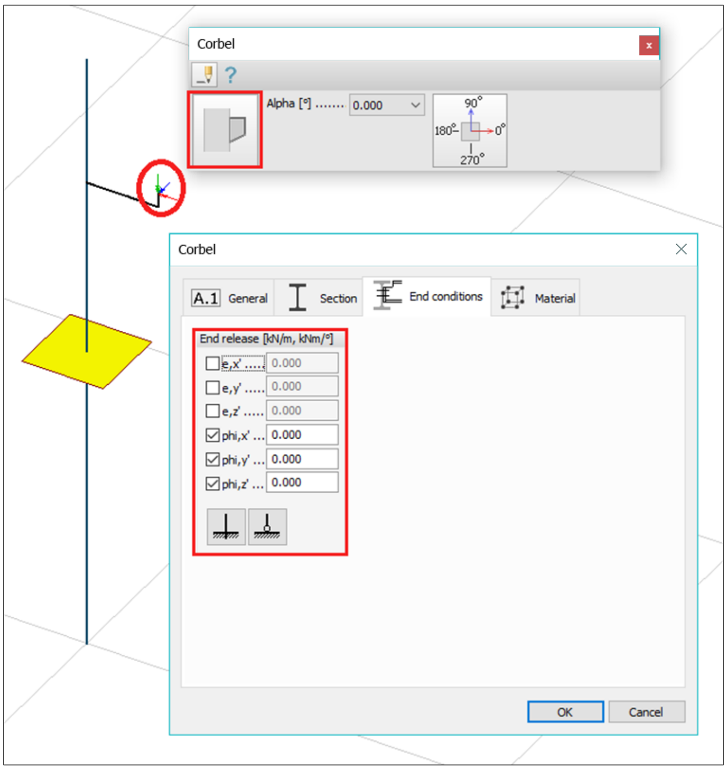
For beams and columns, it can now be specified whether to generate fictitious bars in the shell model's boundaries. This will affect stress distribution in the ends of these objects.



When creating a beam or column, (1.) select *Shell model* (), then (2) turn the option on or off as desired.

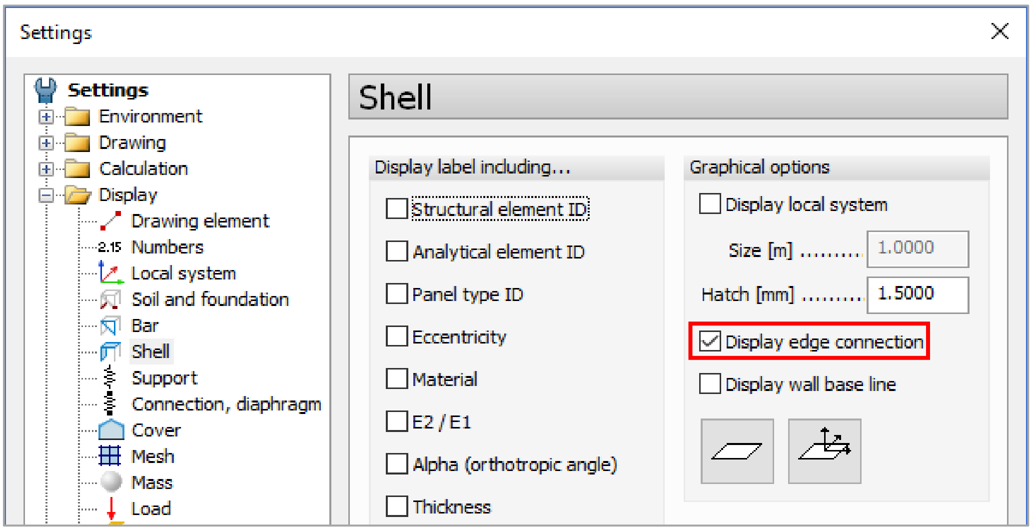
3.3. User defined end connections for corbels

In FEM-Design 16 the user can set the end connection of corbel in the new End conditions tab in Corbel dialog. The default setting is hinged.



3.4. Edge connection can be hidden

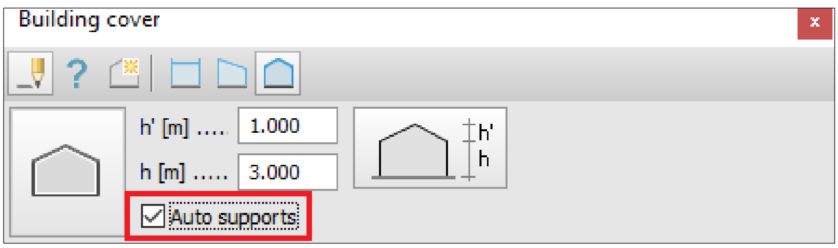
In the *Settings* dialog, an option has been added to display/hide edge connections. Hiding edge connections will reduce the time required to render large drawings.



3.5. Automatic selection of supporting structures for covers and building covers

For *Slab* and *Wall covers*, FEM-Design 16 can automatically find available supporting structures, when the option is selected.

This option is also available when creating *Building covers*.



When using *Auto supports*, the following line objects or elements will be recognised as supports if they are located in the cover's plane:

- Beams and columns (including fictitious ones)
- Line supports and line support groups
- All inner and outer edges of slabs and walls (also working with fictitious ones)

The Supporting structures and the Load bearing direction commands can be used to alter the automatically determined specifications.

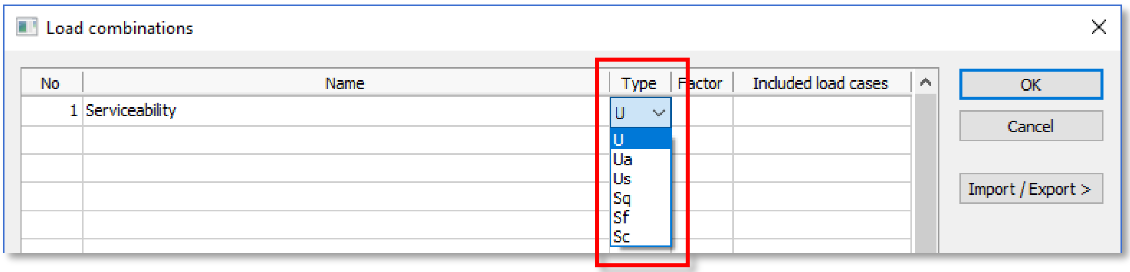
`

4. Load

4.1. New SLS combinations

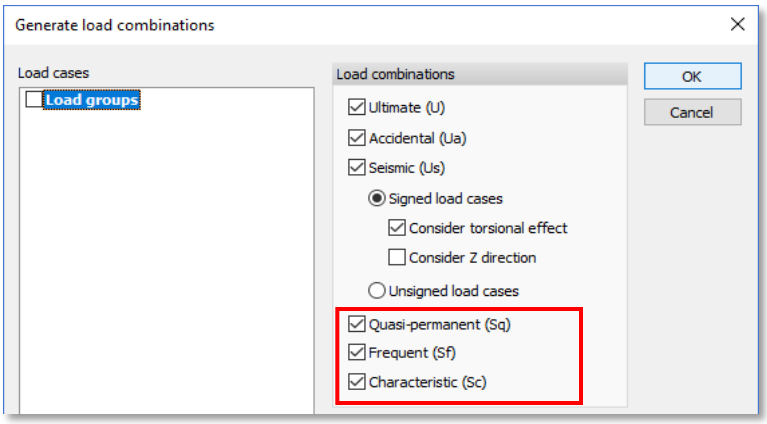
FEM-Design now handles all Serviceability Limit State load combinations defined by Eurocode:

- Quasi-permanent (S_q)
- Frequent (S_f)
- Characteristic (S_c)



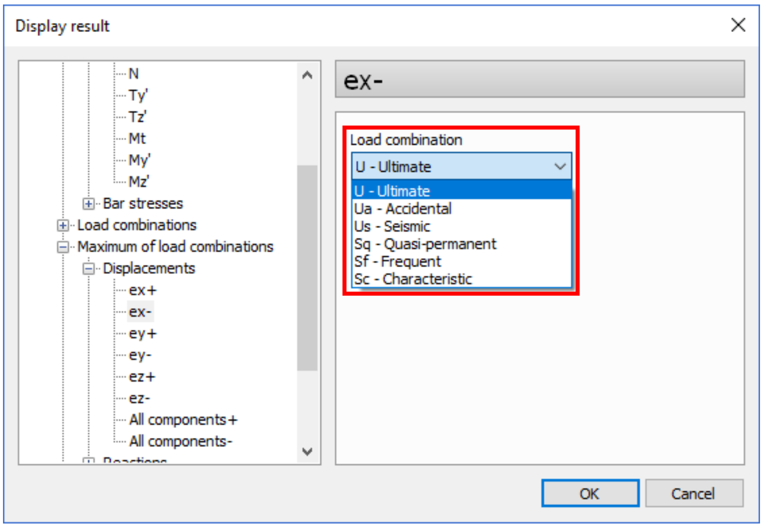
⚠ In earlier versions of FEM-Design only *Characteristic* load combination (S) existed. It means that in case opening old model, all SLS combination will be shown as "Sc" instead "S".

The desired limit state can be specified for each load combination in the Load combinations dialog. All three serviceability limit states are available for automatically generating load combinations from load groups as well.

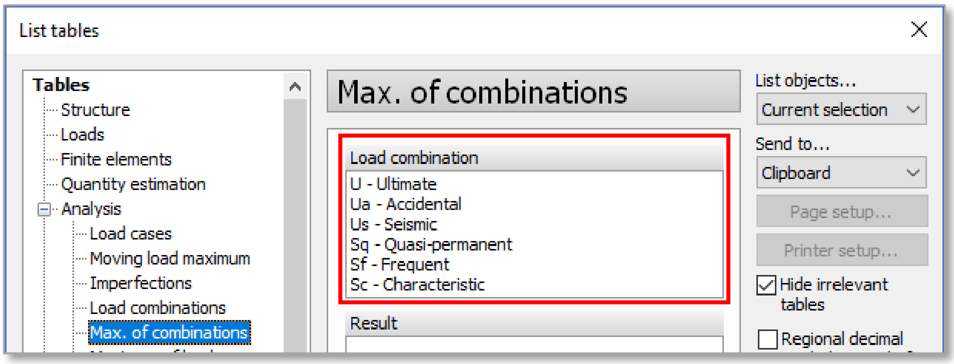


When displaying results from the *Maximum of load combinations* and *Maximum of load groups*, we can choose from the six Eurocode limit states for any type of result.

`



This choice can also be made when listing results.



For the different design calculations different SLS load combinations are used:

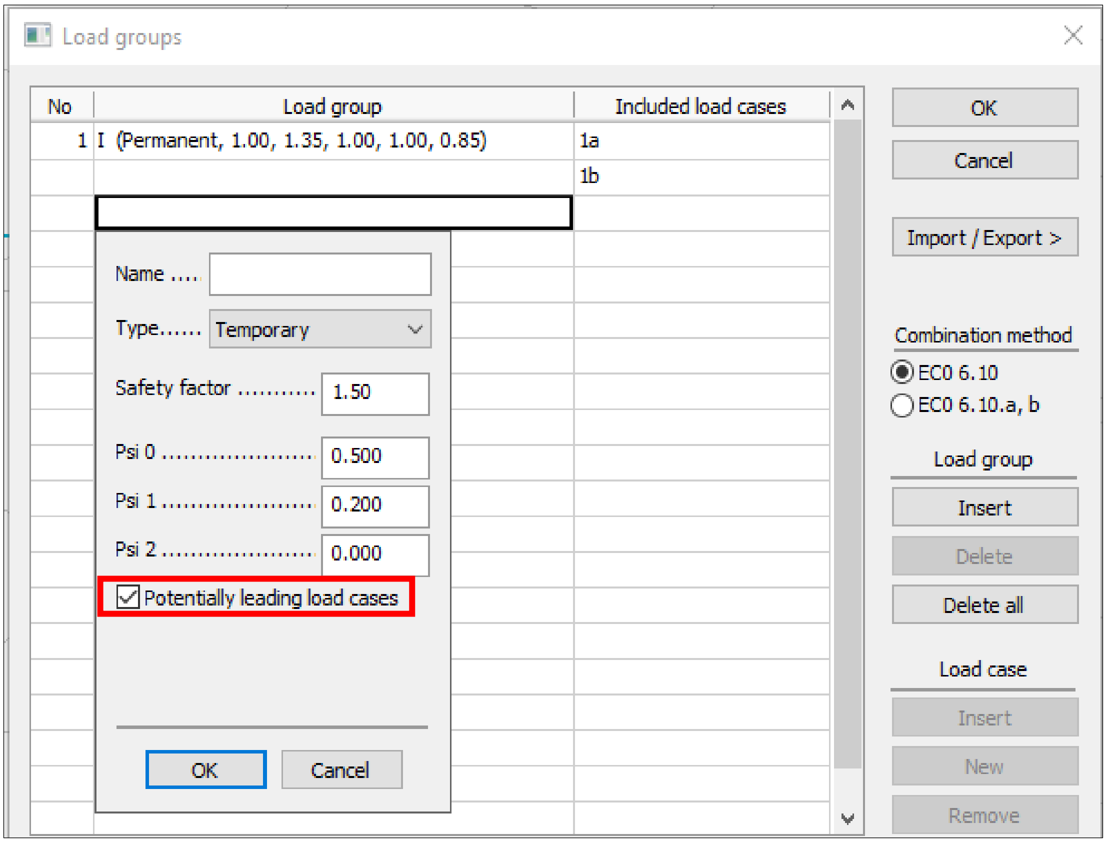
- Deflection check: user selectable (see in chapter 5.1)
- Foundation settlement: user selectable (see in chapter 6.1)
- Crack width: Quasi-permanent (S_q)

4.2. Leading temporary load cases in load group

At the definition of temporary load groups a new option is available. It defines that the load cases within the group can be leading cases or not.

By turning it off the number of automatically generated load combinations is reduced, which also affects the analysis results and object design for *Maximum of load groups.*

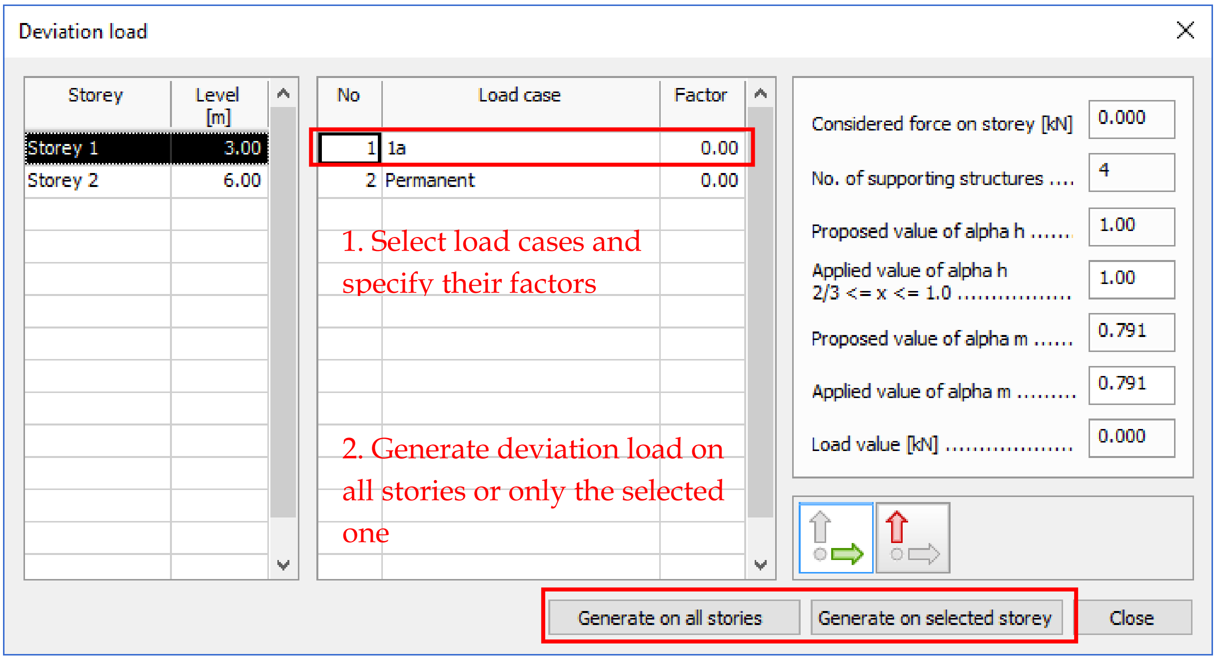
It has no effect on seismic and quasi-permanent results since these, by Eurocode definition, do not contain leading load cases.



⚠ When a model made with an earlier version of FEM-Design is opened, the load cases in the temporary load groups are automatically *Potentially leading load cases*.

4.3. Deviation load for load cases

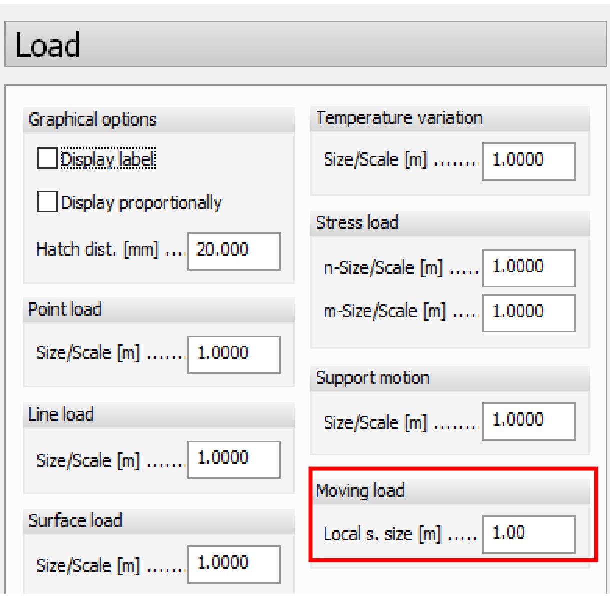
Now deviation load can be generated for any load case with given factor. Deviation load can be generated on all storeys, or on the selected one only.



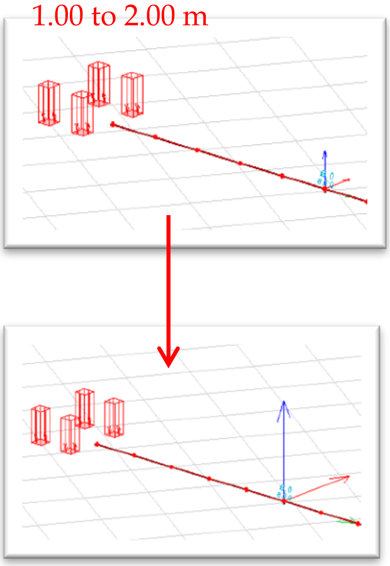
⚠ When a load case used to determine the deviation load changes, deviation load WILL need to be generated again.

4.4. Size of symbol of local co-ordinate system of moving load can be set

The size of the symbol of local co-ordinate system of moving loads can now be scaled.



The effect of changing co-ordinate system size from 1.00 to 2.00 m

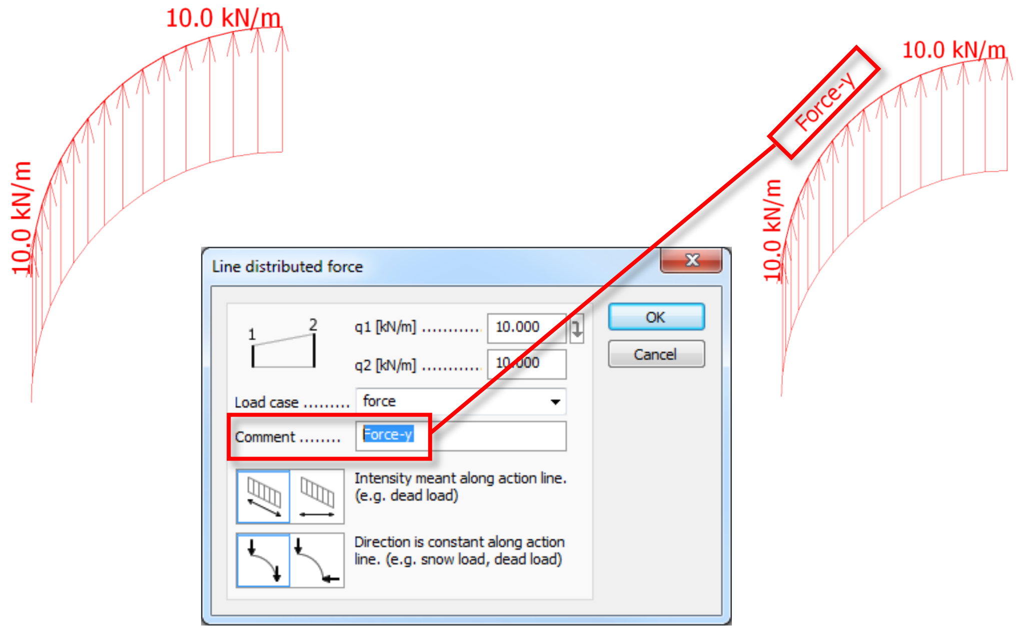


`

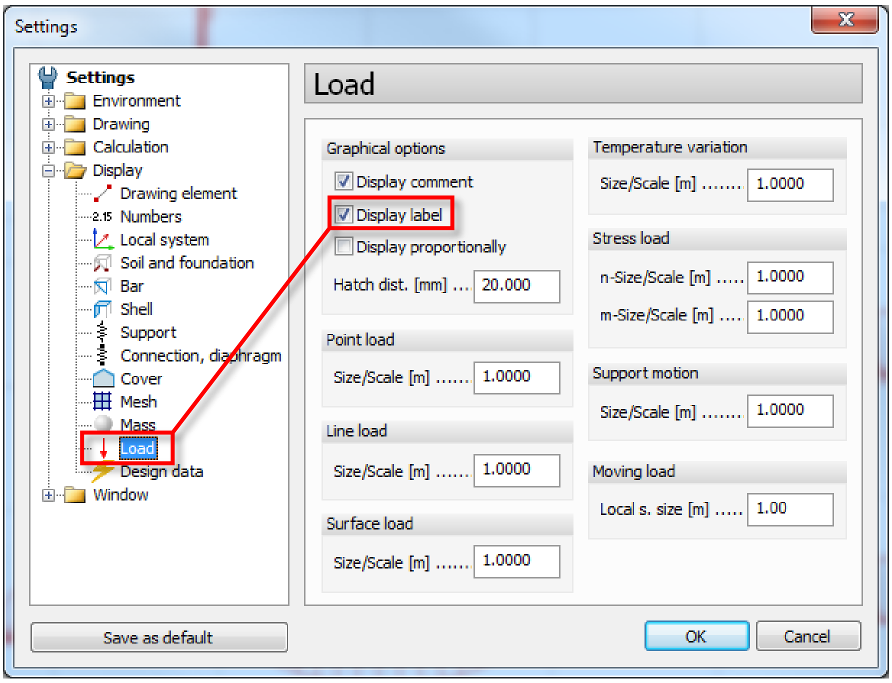
4.5. Load comments

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.

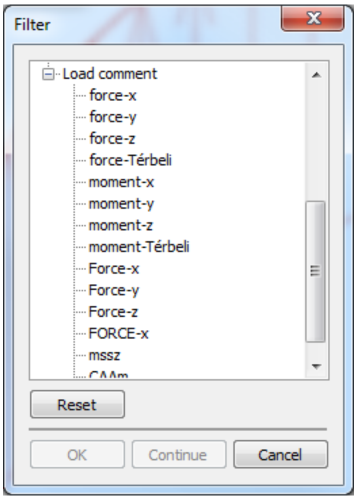


Load comments are displayed or hidden depending on the *Settings/Display/Loads/Display label* options.



Load comments appear in the Filter dialog

`



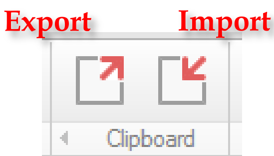
and also in the listed load tables

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

4.6. 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 informations 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 to do NOT edit other columns to avoid errors in Importing.

`

Editable columns

#Type	Guid	Case	Comment	q1 [kN/m	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 to clipboard, then in FEM-Design clicking to *Import*.

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

5. Analysis

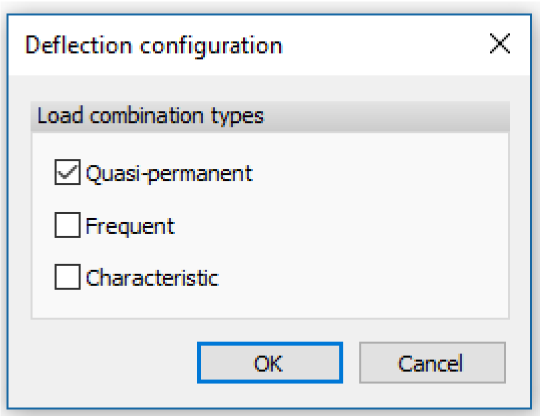
5.1. Deflection check for RC, steel and timber bars

A new checking criteria is available for reinforced concrete, steel and timber bars. Deflection utilization is calculated for *load combinations*, *Maximum of load combinations* and *Maximum of load groups* according to the user defined serviceability limit states.

This new result type is based on the displacement of the bars and the deflection limitation settings which can be defined by the so-called *deflection lengths*.

In the  *Deflection configuration* dialog, we can specify the types of load combinations/groups for which the deflection check is performed.

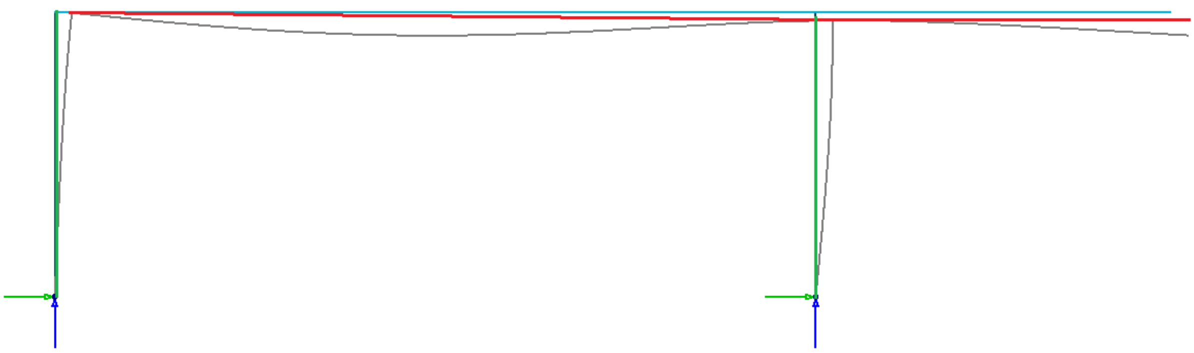
`



Deflection lengths are used to define those bar segments, where the deflection checking criteria/limitations are coincide. The "Simply supported" deflection lines are denoted with blue arcs below the bar, the "Cantilever and column" types are orange and the "not relevant" types are black. Relative and/or absolute limit can be set for each *length* individually. If both are requested the dominant one will be calculated and displayed.

The first option we can set here is the behaviour of the lengths which affects the calculation method of the deflection. If we choose not relevant for a specific length, it will be excluded from the checking process.

For the better understanding of the next two options, namely the *Simply supported* and *Cantilever* mode let us consider the following example, a cantilever frame structure.

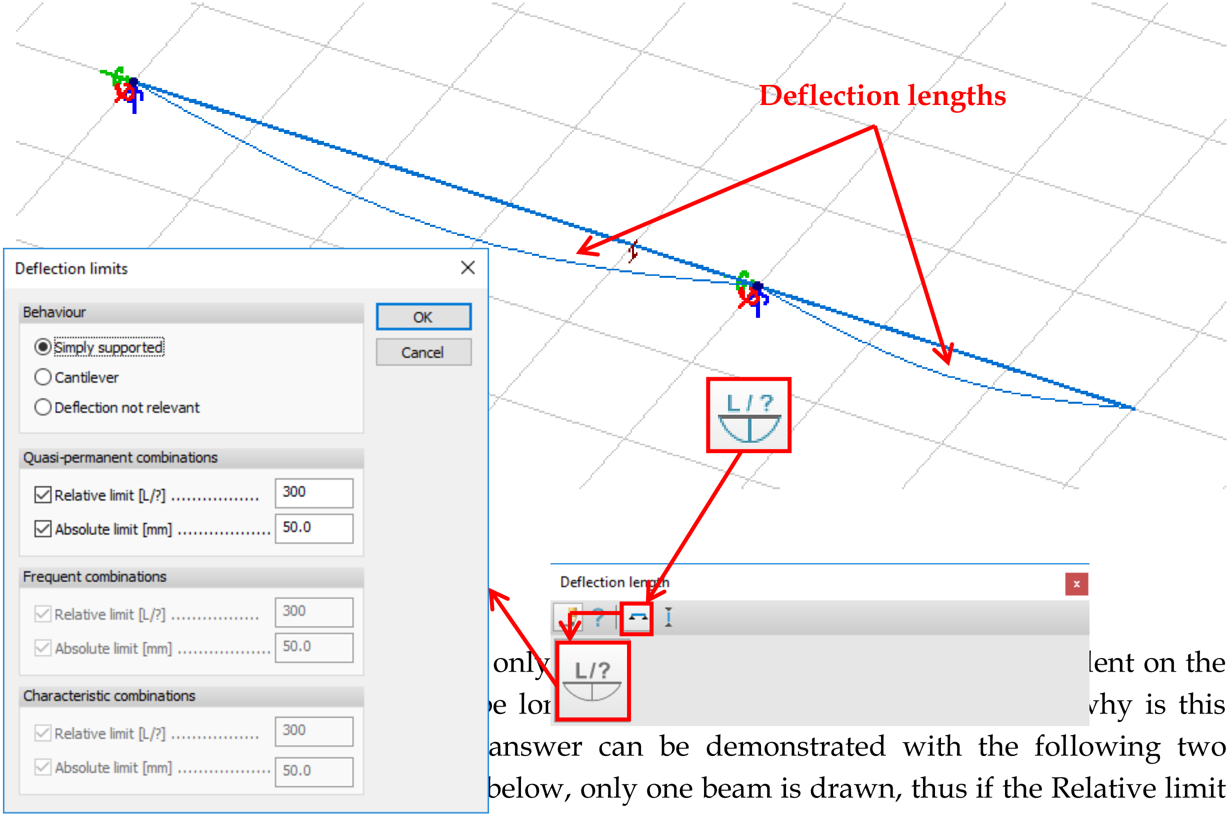


In the midspan we should use the Simply supported option, where we eliminate the rigid body motions in such a way that we connect the endpoints of the length, and measure the deflections of the middle sections from this imaginary line (red skew line on the picture above).

As a consequence of this method, the deflections of the endpoints are zero, the dominant section is usually at the middle of the length.

On the cantilever, we would like to use the cantilever mode, where the dominant value of deflection on this length will be the difference between the maximum and minimum absolute deflection (in this example the largest distance from the red horizontal line). For columns the same calculation method is used, the only difference is that the deflection is measured in the horizontal plane (from the green lines).

⚠ For columns only this (*"Cantilever and column"*) option is available.

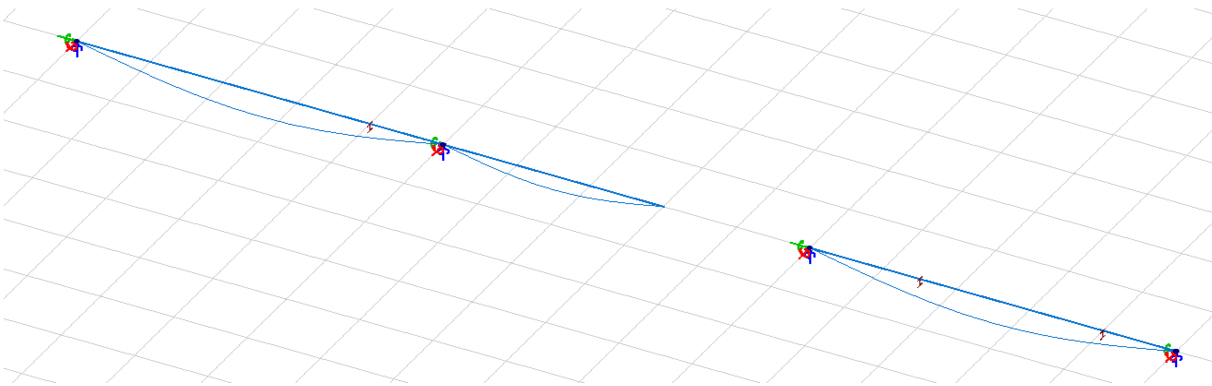


only ... lent on the
e lor ... vhy is this
answer can be demonstrated with the following two
below, only one beam is drawn, thus if the Relative limit would be calculated directly from the length of this beam, the results would be misleading.

In other words, in the *L/?* formula, instead of the length of the midspan or the cantilever, the whole length of the beam would be substituted. Therefore, we need two Deflection lengths to differentiate the *L* in the Relative limit formulae on the midspan and on the cantilever. In addition, the limit value also can differ for the two types of structure according to the National Annexes.

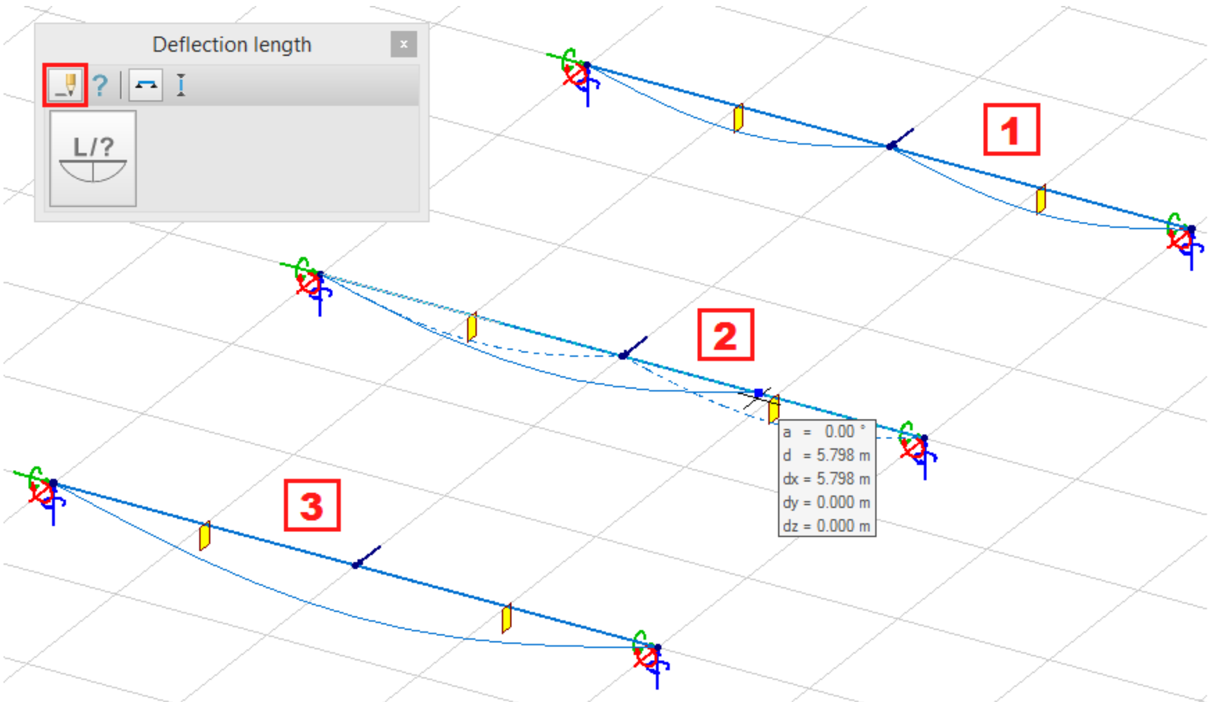
In the second case (to the right on picture below) imagine that our aim is to design a beam splice and check deflections. Two beams need to be drawn for the steel design, but of course during the deflection check we want to use the summed length of the two beams for the calculation of the relative limit value. For this purpose, we define one Deflection length over the two bars - this way we make correct calculations in both cases without any additional modification on the structure.

`



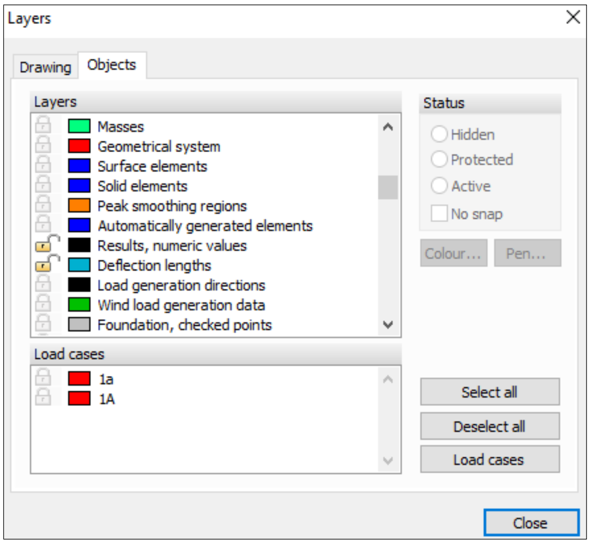
It is worth to note that in the second case we had two beams, but in contrary to the buckling lengths, definition and editing of deflection lengths can be performed on such set of beams, which are both parallel and continuous.

By default, deflection lengths are generated automatically. This procedure first search all the previously mentioned parallel and continuous beams sets, then intersect them with the edges/axes of the structural elements (beams, columns, trusses, plates, walls, line and surface supports) and point supports. In the majority of the cases deflection lengths obtained by this way are reasonable from engineering point of view, but in some cases we may want to modify them. A good example can be a structure consisting of two beams with a horizontal support, which should not be considered in the deflection checking process. The following flow diagram illustrates the modification of the two beams step by step. By default, as we can see in the upper picture, the automatically generated deflection lengths coincide with the beams because they are intersected with the horizontal point support. If we would like to have one deflection length over the two beams, we can draw it between the support groups using the Define tool, similarly to the buckling lengths. By this way, the new length substitutes the original ones!



`

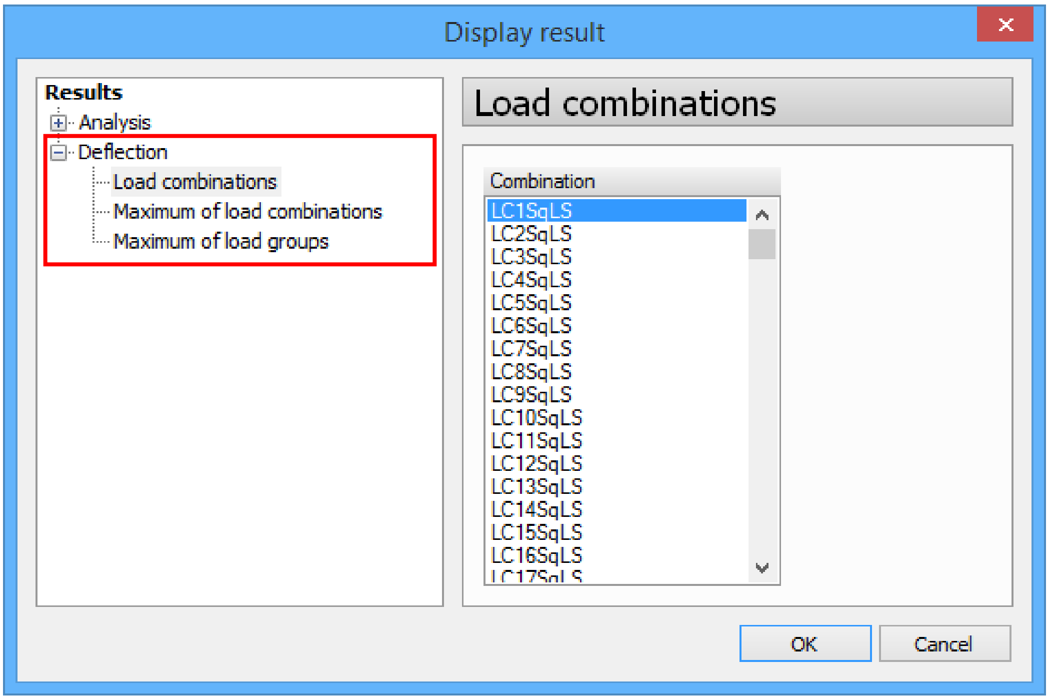
Deflection length has its own layer.



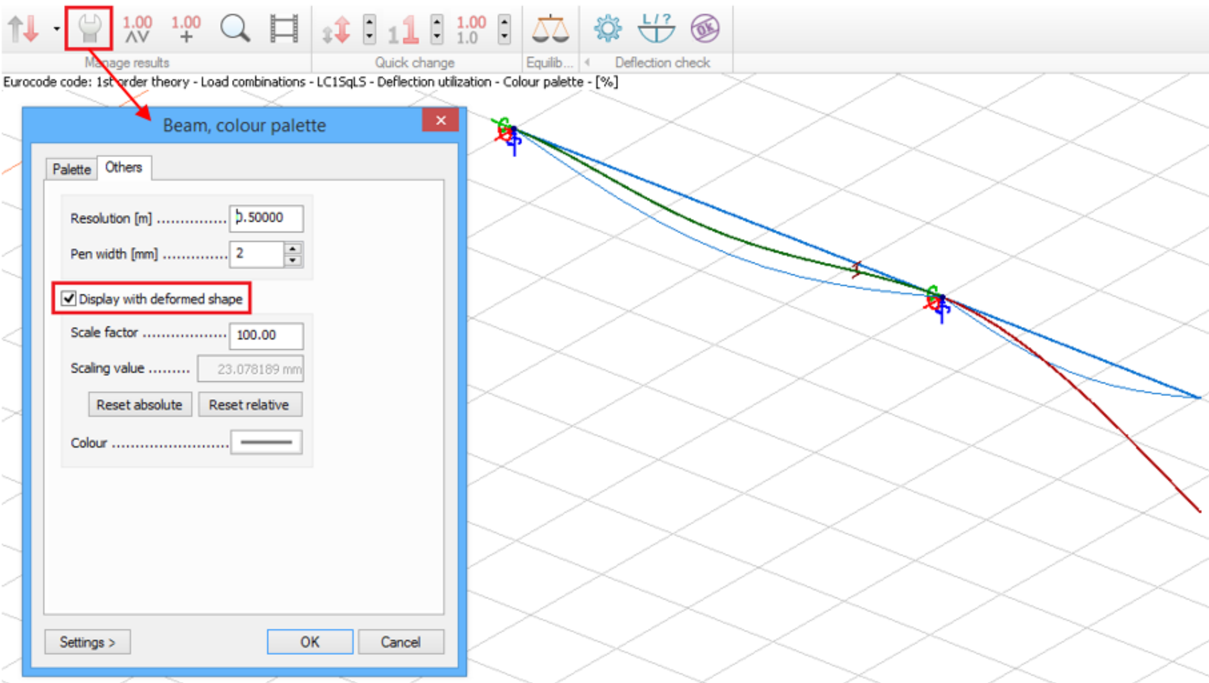
Deflection check button becomes active if Load combinations and/or Load groups are already calculated. The utilization results can be displayed from the *New result* dialog.

The Deflection checking process considers only the straight beams and columns. For beams the deflection is measured along their own local z' axis, for columns it is measured in the global horizontal x-y plane.

Results requested for a *Load combination* can be displayed both on the deformed and undeformed shape.



Due to the fact that the limit values of the calculation are controlled by the Deflection lengths, the result is constant along them. In other words we have one (dominant) utilization value for each Deflection length. Results for *Maximum of load combinations* and *Maximum of load groups* are only displayed on the undeformed shape of the structure.



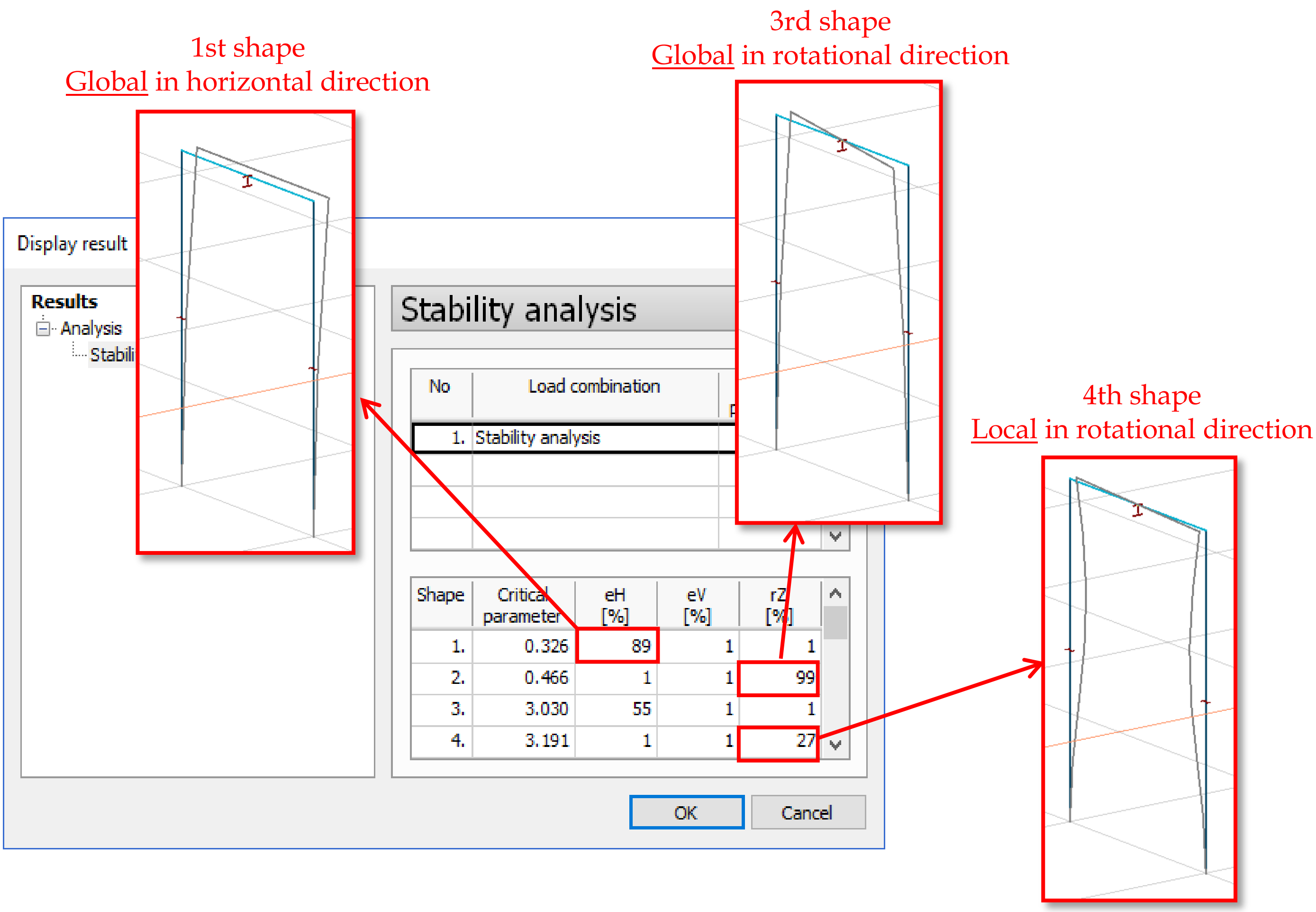
5.2. Local-Global stability shape

`

Additional information has been added to help determine if a buckling shape is global or local.

For each shape, a probability percentage is given: higher values shows high probability that the shape is global. If there are not enough shapes calculated, none might be global. Probability values are given for horizontal and vertical displacements as well as rotation around the vertical axis.

In the example below, the *eH* value of the first shape is 89%, which means it is probably a global buckling shape with horizontal displacement.

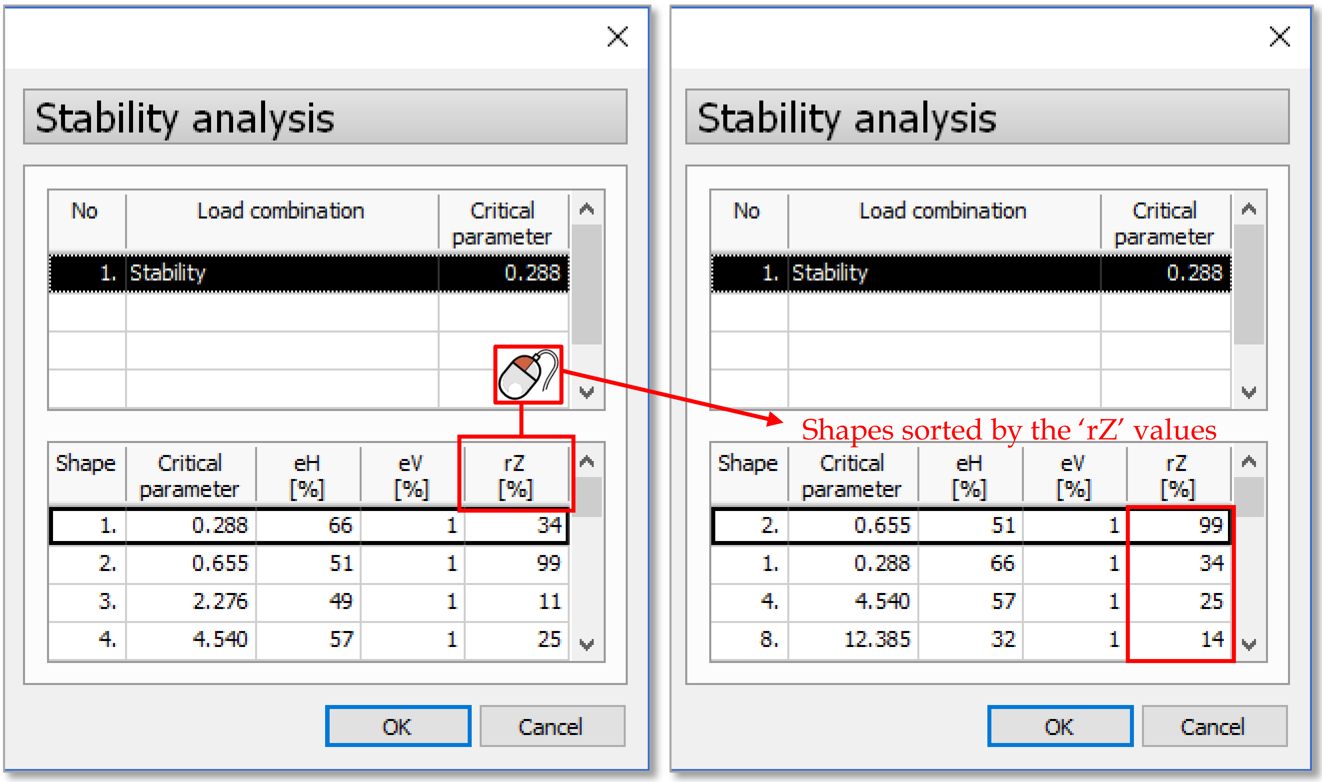


Displaying the result (see the leftmost inset above) and examining the buckling shape shows that this is indeed a case of global buckling with the horizontal displacement of the frame's top.

The same structure's second shape possesses a very high *rZ* value (99%), meaning this almost certainly is a global torsional buckling shape (shown in the middle inset).

The fourth shape's *eH, eV* and *rZ* values are significantly lower, which implies it is a local buckling shape. As the rightmost inset shows, the assumption was correct (local buckling of both columns).

Buckling shapes can be sorted according to any column of the table by left clicking on the column header.



5.3. Applying Geometric Stiffness Matrices for Truss elements

Truss elements are considered in second order calculations. The geometric stiffness matrix of a truss element is the following:

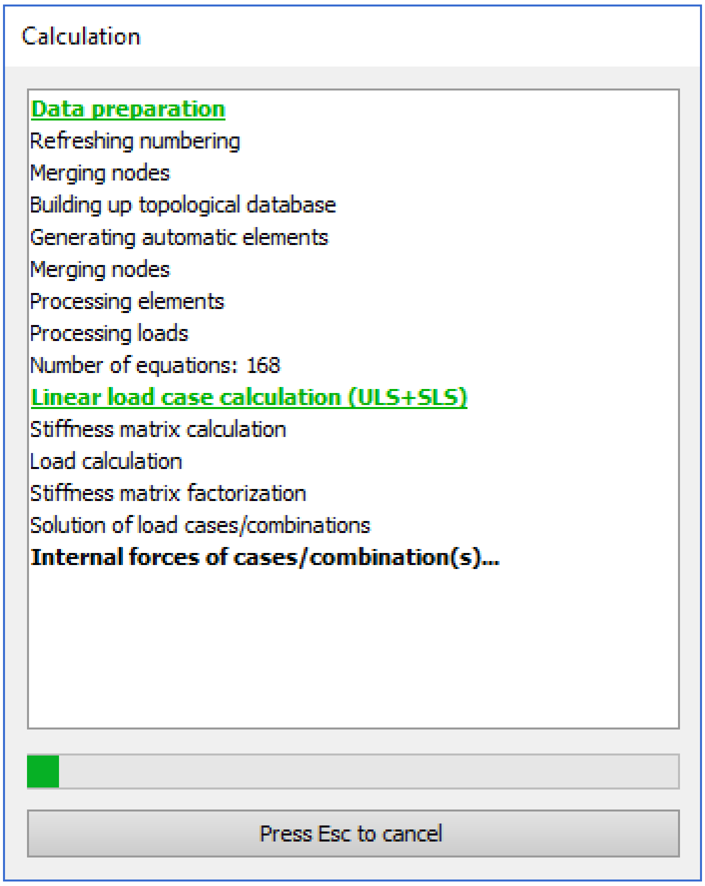
$$
\begin{bmatrix}
0 & 0 & 0 & 0 & 0 & 0 \\
0 & \frac{N}{L} & 0 & 0 & -\frac{N}{L} & 0 \\
0 & 0 & \frac{N}{L} & 0 & 0 & -\frac{N}{L} \\
0 & 0 & 0 & 0 & 0 & 0 \\
0 & -\frac{N}{L} & 0 & 0 & \frac{N}{L} & 0 \\
0 & 0 & -\frac{N}{L} & 0 & 0 & \frac{N}{L}
\end{bmatrix}
$$

where N is the normal force and L is the length of the truss.

5.4. Data preparation has been improved

The *Data preparation* sequence of calculations has been made more clear. In addition, the whole process has been made faster.

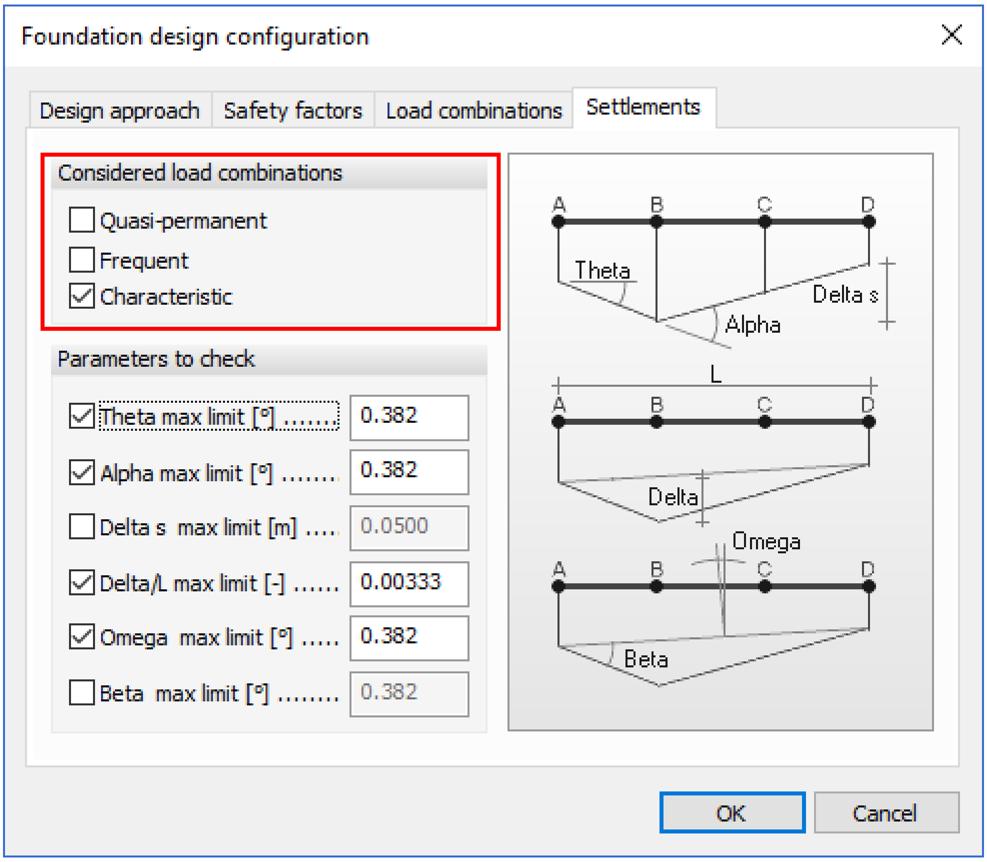
The *Calculation* window will now list the specific subprocesses of said sequence, with their own progress bar at the bottom. This helps clarify the mechanisms under the *Data preparation* step and enables users to follow more details of the calculation.



6. Foundation

6.1. Foundation settlement check – considered load combination types can be set

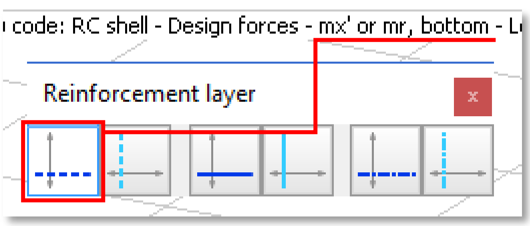
The three new SLS combinations has been added to the foundation settlement check dialog, as well.

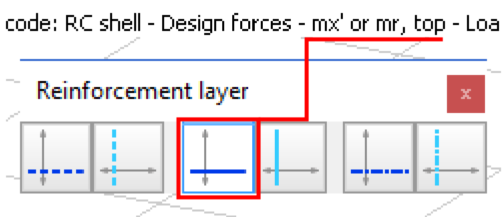


7. RC design

7.1. RC shell reinforcement layer automatically follows the selected result

In the *RC design* tab, the displayed layer of reinforced concrete panels will now automatically synchronise with the result being displayed.



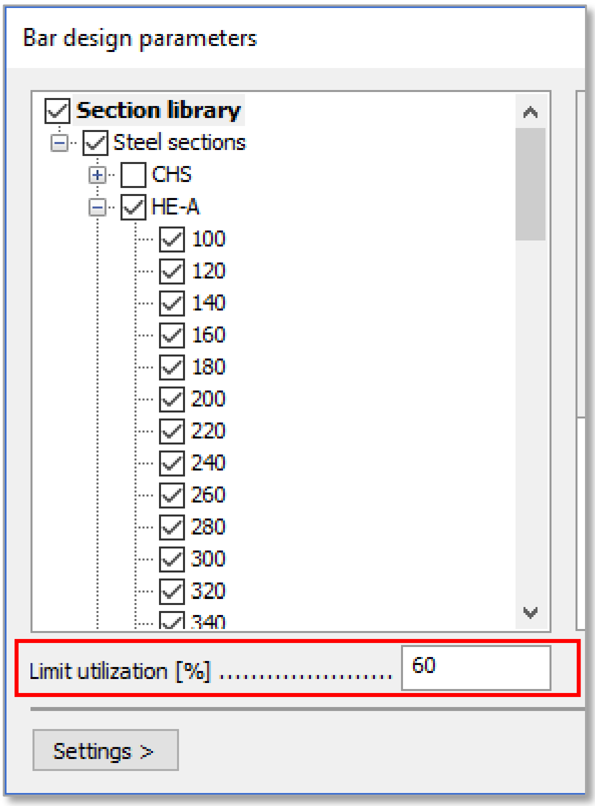


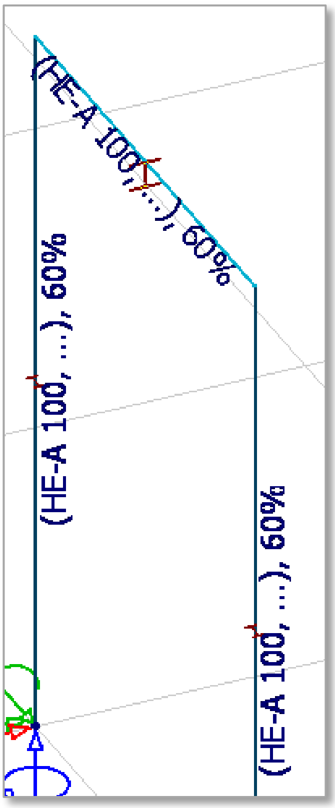
`

8. Steel and Timber design

8.1. Auto design for limited utilization steel bars, timber bars and panels

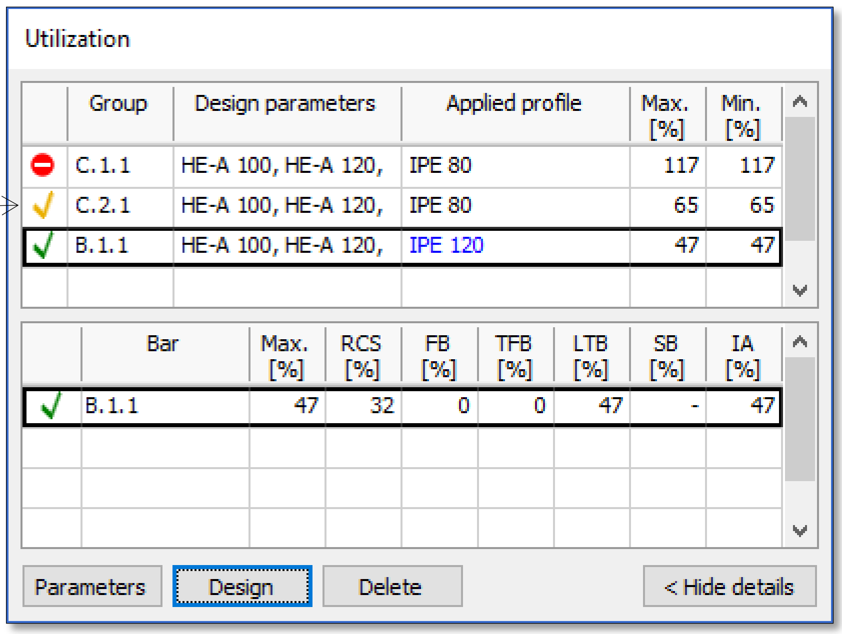
In auto design of steel bars, timber bars and panels, the utilization can be now limited and be set between 10% and 100%. The specified utilization limit is displayed as part of the object ID.





Status of bars with utilization less than 100% but more than the specified limit value will be displayed by a yellow tick in the *Utilization* dialog

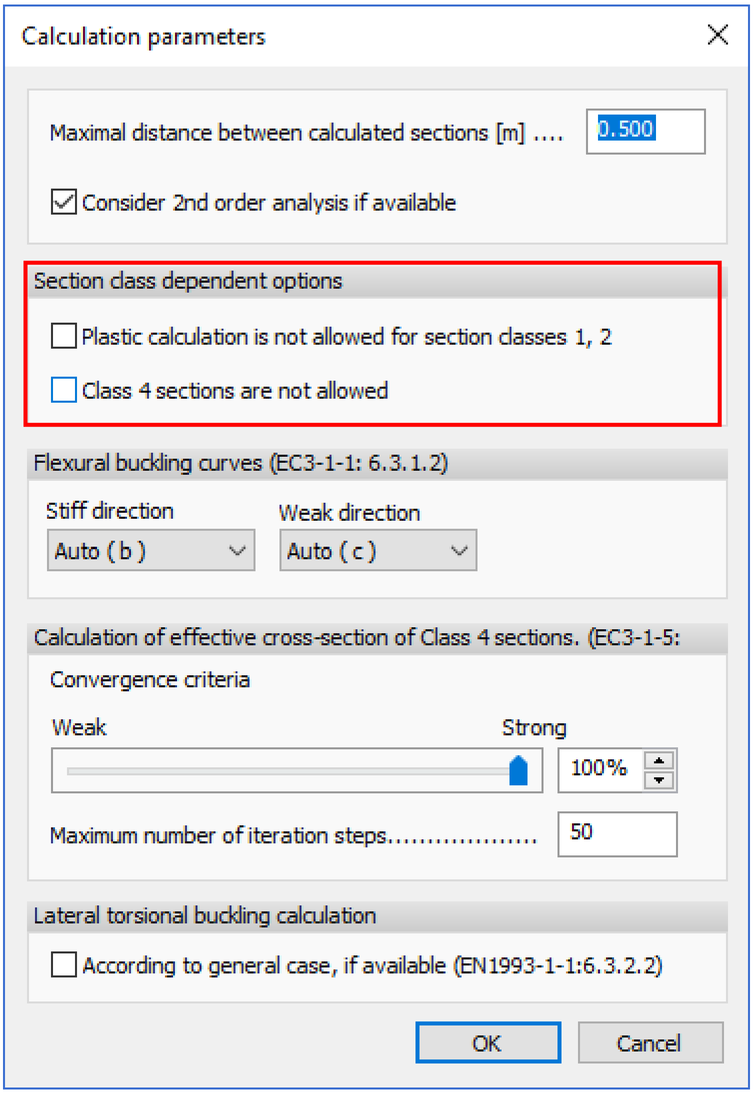
More than 60%,
but less than 100%



`

8.2. Additional section class dependent options for steel bar design

When designing steel elements, it is now possible to use elastic calculation method instead of plastic not only for Class 3 sections, but also Class 1 and 2 as well. Also, Class 4 sections can be completely ignored during the design phase.

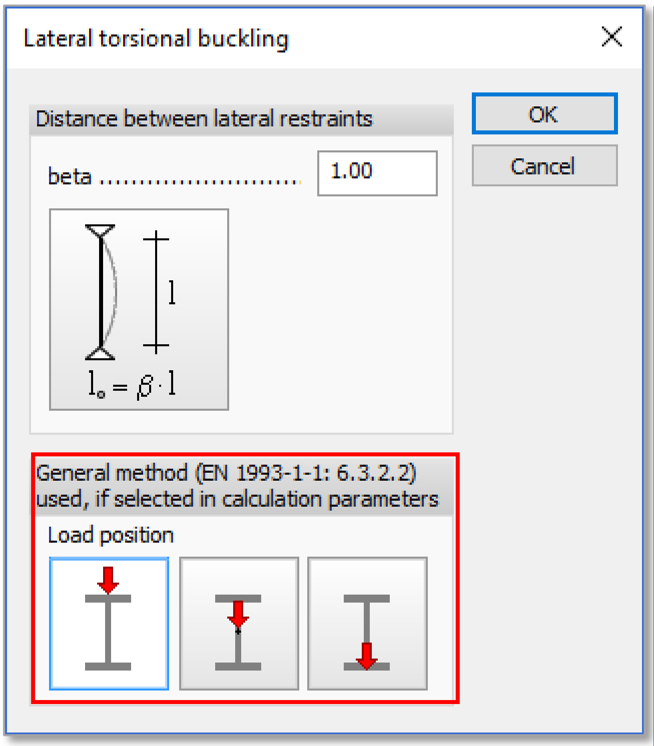


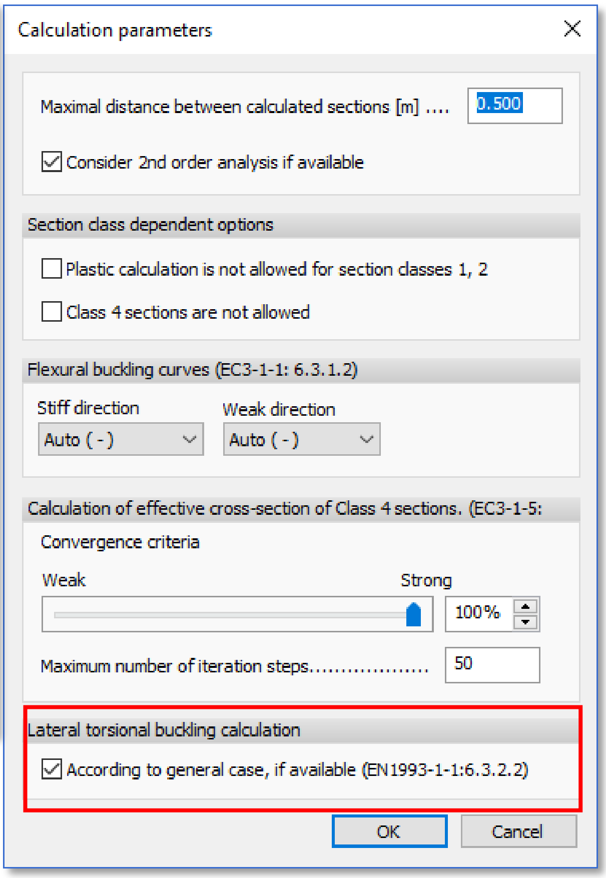
The main reason for implementing this feature is bridge design where plastic calculation is generally not permitted due to the fact that material fatigue is a significant critical condition.

8.3. Lateral torsional buckling calculation according to general case and k_{ij} interaction factors calculated according to Method 1

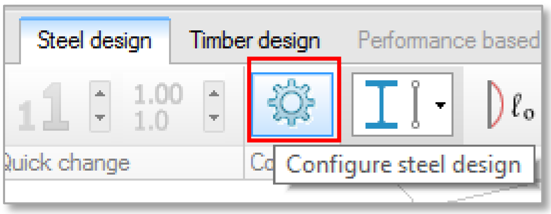
Lateral torsional buckling can now be calculated using the formulas to general case instead of using only the simplified method.

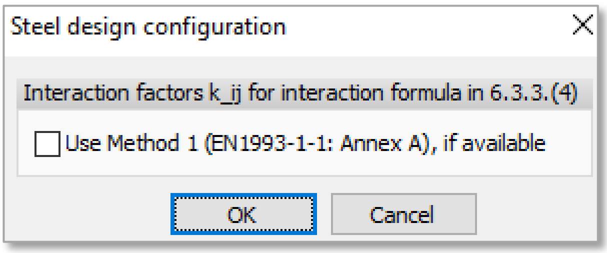
When the general method is used for lateral torsional buckling calculation, the position of the load needs to be specified as well.





Calculation method of the k_{ij} interaction factors can be specified by selecting the design configuration option from the ribbon:



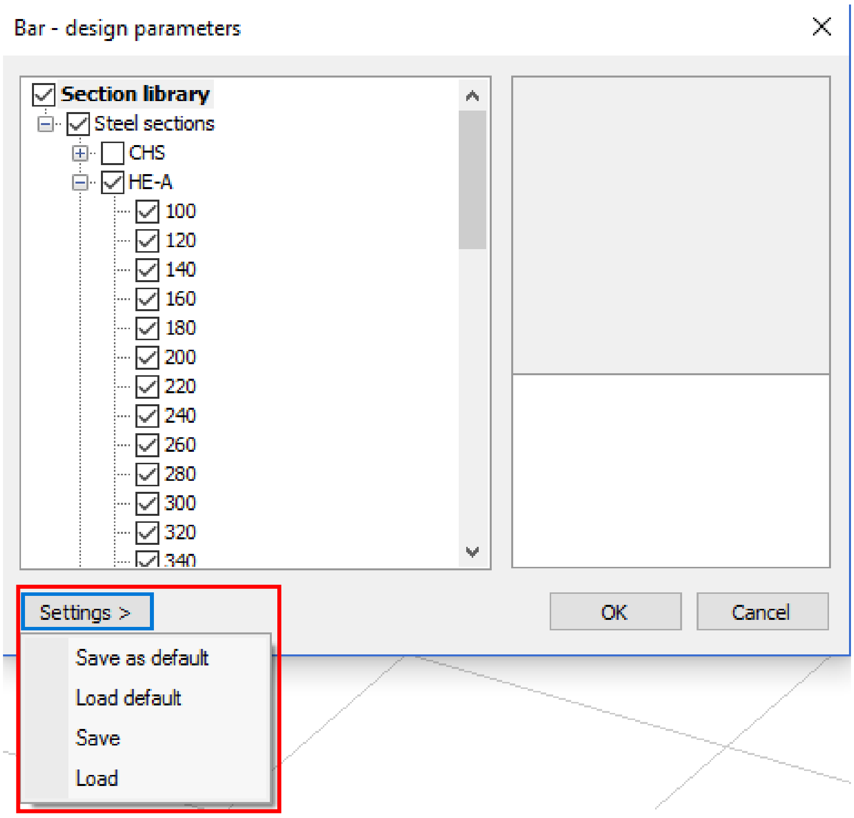


`

8.4. Save/load steel and timber bar design parameters

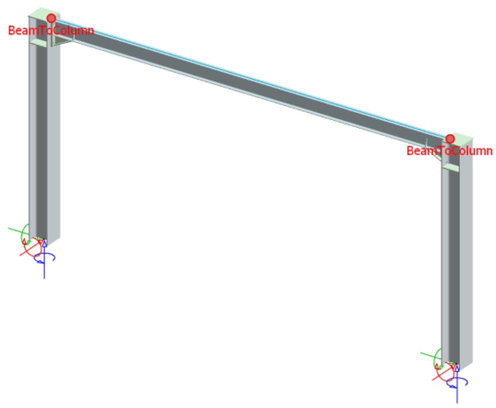
An option has been added to the *Auto design* dialogue panel to save and load the specified design parameters.

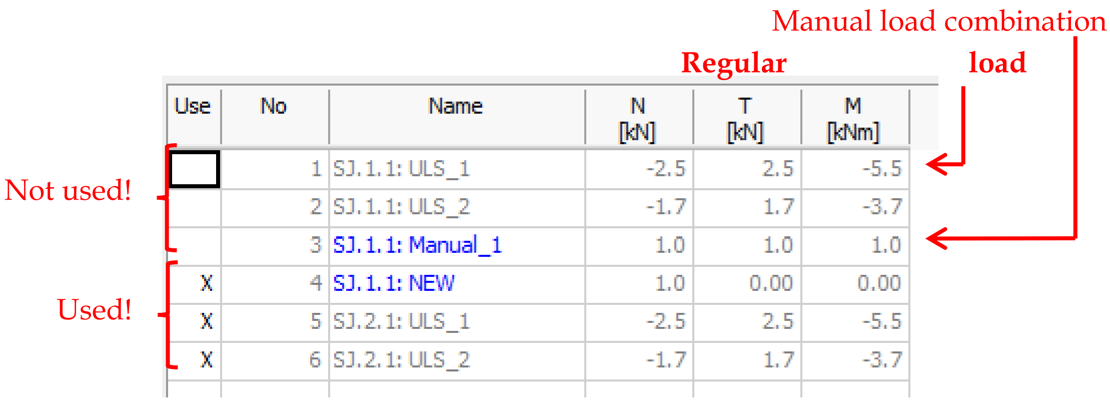
For each section type (e.g. IPE, HEA, CHS, etc.) a set of sections can be saved/loaded as default. This will only work with *one type* cross-section (e.g. only HEA, or only KKR) selected. Otherwise, with *Save* command, the user can save a set of arbitrary sections into a file, and use them later for another model by *Load* command.



8.5. Manual load combinations in Steel Joint (Built in version)

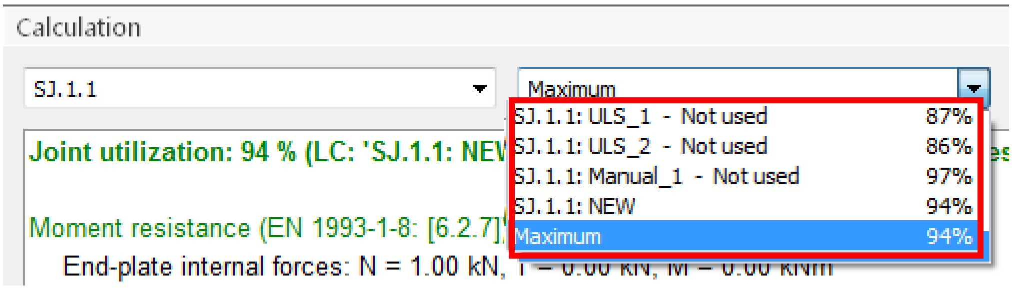
In the built-in version of Steel Joint module the User can add manual load combinations for their joints. The regular combinations - which come from the 3D model - are displayed in grey text, the manual combinations are blue. The combination name has two parts, the first part is the joint name, and the second is the combination in the 3D model or defined by the user. E.g. “SJ.1.1: ULS_1”: The SJ.1.1 joint is checked for the “ULS_1” combination.

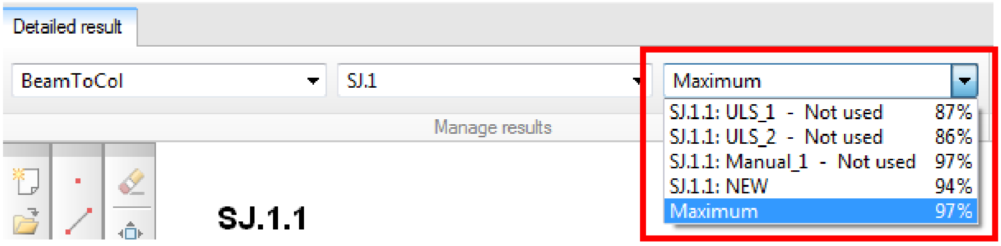




Use	No	Name	N [kN]	T [kN]	M [kNm]
	1	SJ.1.1: ULS_1	-2.5	2.5	-5.5
	2	SJ.1.1: ULS_2	-1.7	1.7	-3.7
	3	SJ.1.1: Manual_1	1.0	1.0	1.0
X	4	SJ.1.1: NEW	1.0	0.00	0.00
X	5	SJ.2.1: ULS_1	-2.5	2.5	-5.5
X	6	SJ.2.1: ULS_2	-1.7	1.7	-3.7

A 'Use' column is added to the load combination table, where the User can set which load combinations should be taken into consideration for the maximum of load combinations, these are marked with "X". The combinations that are not marked as "Use" are also calculated, but they are marked as "Not used" in the Steel Joint *Calculation* dialog and in the *Detailed result*.



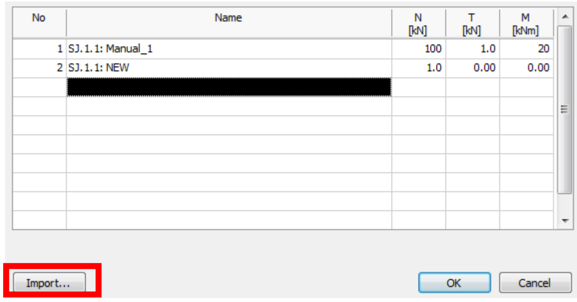


If the steel joints belong to a Design group the automatic load combinations are generated for all the joints in the group. In the example the ULS_1 and ULS_2 combinations are calculated for both SJ.1.1 and SJ.2.1 joints.

If the joints are grouped in a Design group it is possible to make a load combination for only one specific joint. As it mentioned earlier the User has to define a new Manual load combination following the template: "JOINT Name.ID.Position number: Manual combination name".

In the example there are two joints (SJ.1.1 and SJ.2.1) which are in the BeamToCol design group. In order to make a new load combination **only** for the SJ.2.1 joint the new combination name should be "**SJ.2.1**: Manual combination name"

Load combinations can be imported from csv (**semicolon** separated value) format in the Manual combination dialog.



⚠ The format of a line in these files should be "*Name; [Internal force 1], [Internal force 2], ….*", with as many internal forces as is needed for the current joint. See the example below, how Load combinations can be imported from an Excel table.

Imported Combination_1	100	13	34
Imported Combination_2	0	24	62
Imported Combination_3	80	14	0

Save as ".csv" file

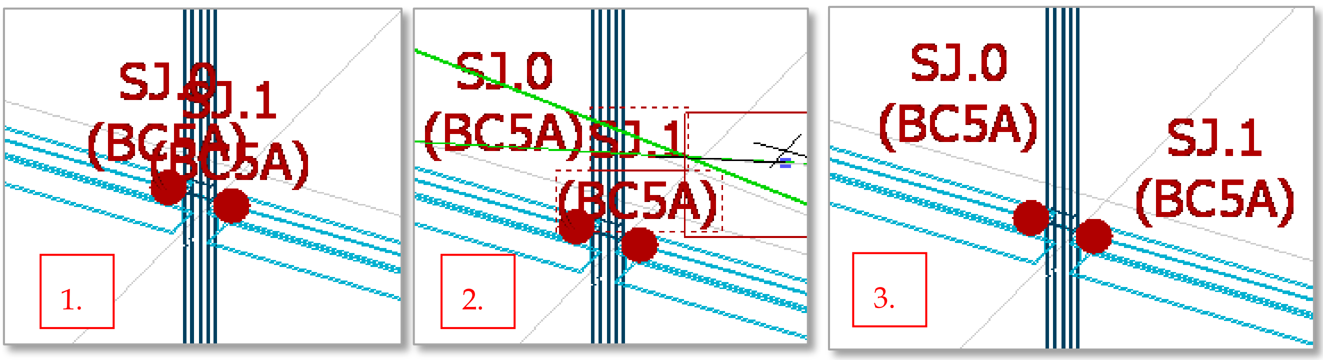
```
Imported Combination_1;100;13;34
Imported Combination_2;0;24;62
Imported Combination_3;80;14;0
```

Import the ".csv" file

No	Name	N [kN]	T [kN]	M [kNm]
1	Imported Combination_1	100	13	34
2	Imported Combination_2	0.00	24	62
3	Imported Combination_3	80	14	0.00

8.6. It is possible to move steel joint ID

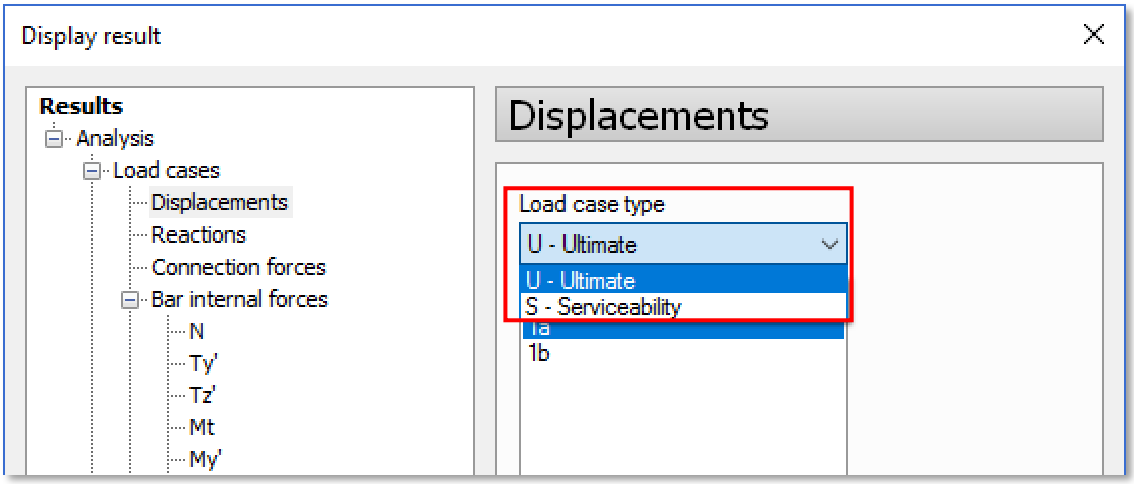
The ID's of steel joints can now be moved and aligned freely by clicking and holding the left mouse button, or selecting the joint ID with one click, then clicking it again to move.



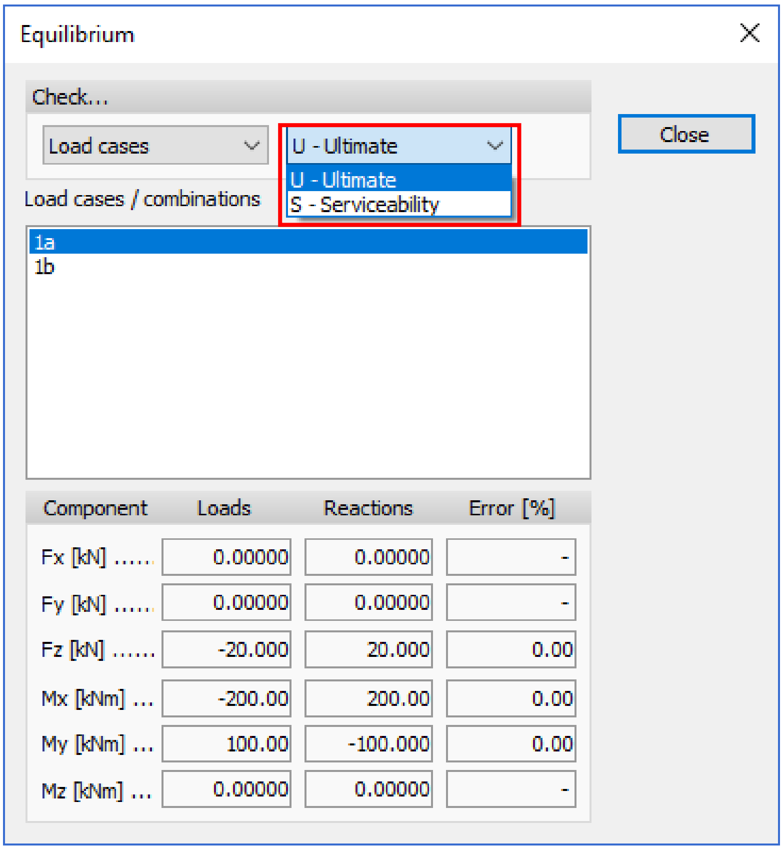
9. Results

9.1. Load case results for user-defined limit state

At the Analysis tab, at the new Load case results we can select from the results based on the ultimate and serviceability value of the material parameters, similarly to the moving load influence line and moving load maximum results.

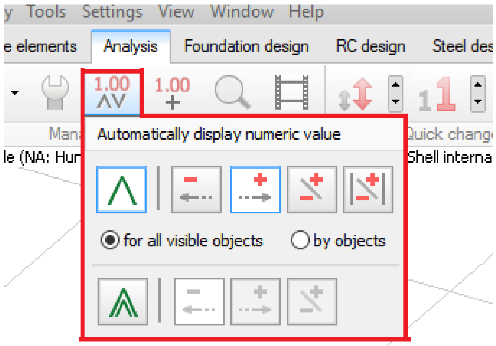


In the *Equilibrium* window, in case of load case results we can also select from the results based on the ultimate and serviceability value of the material parameters.

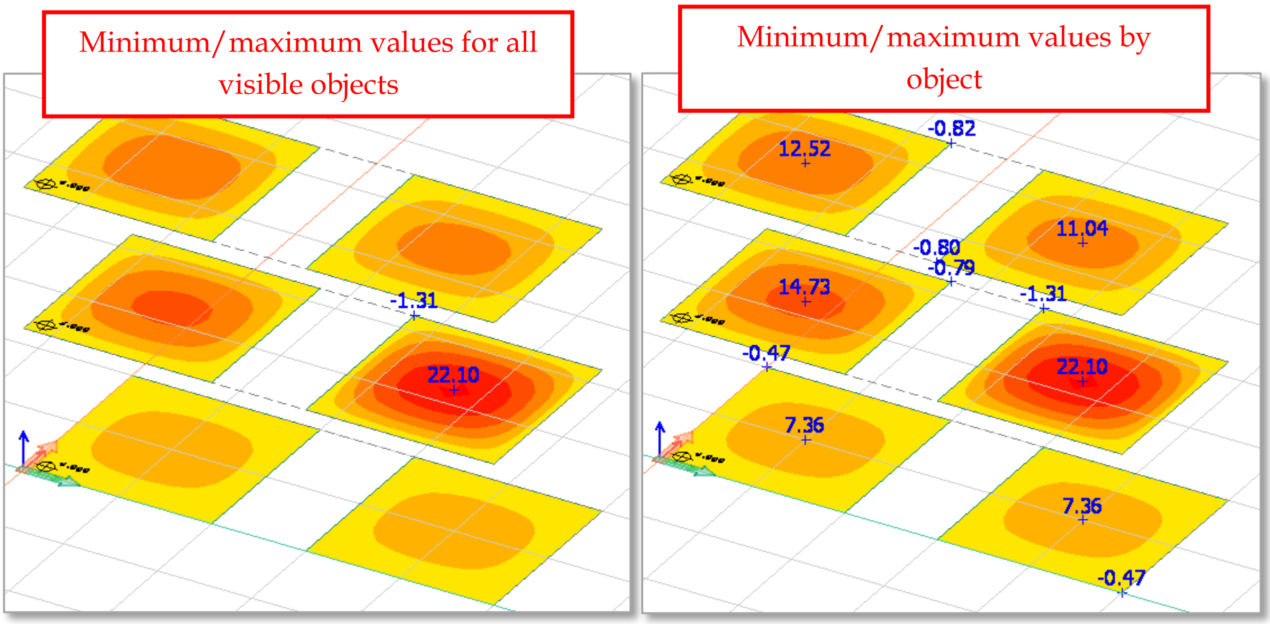


9.2. Automatic minimum and maximum numeric value display

The new function called *Automatic numeric values* displays global and local maximum, minimum or both numeric values of results.

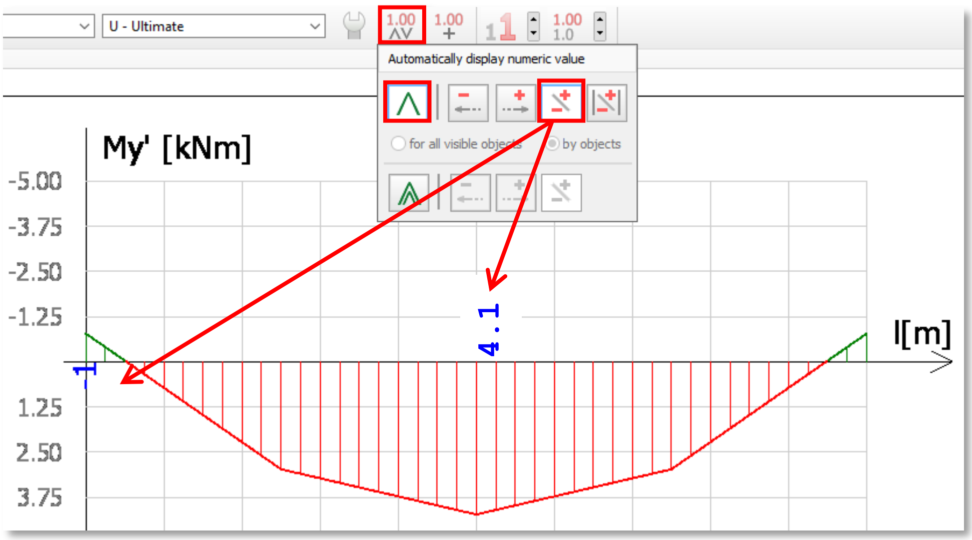


Automatically displayed values will be shown in blue, while values displayed by manual query remain black. It can be set to show the minimum and/or maximum values of all results across all visible objects (below, left), or local extreme values for visible each object (below, right).



When in storey view, 'All visible objects' means the objects on that storey

This feature is available for each result (even for analysis detailed results, as shown below) where minimum/maximum can be displayed by the *Numeric value* function.



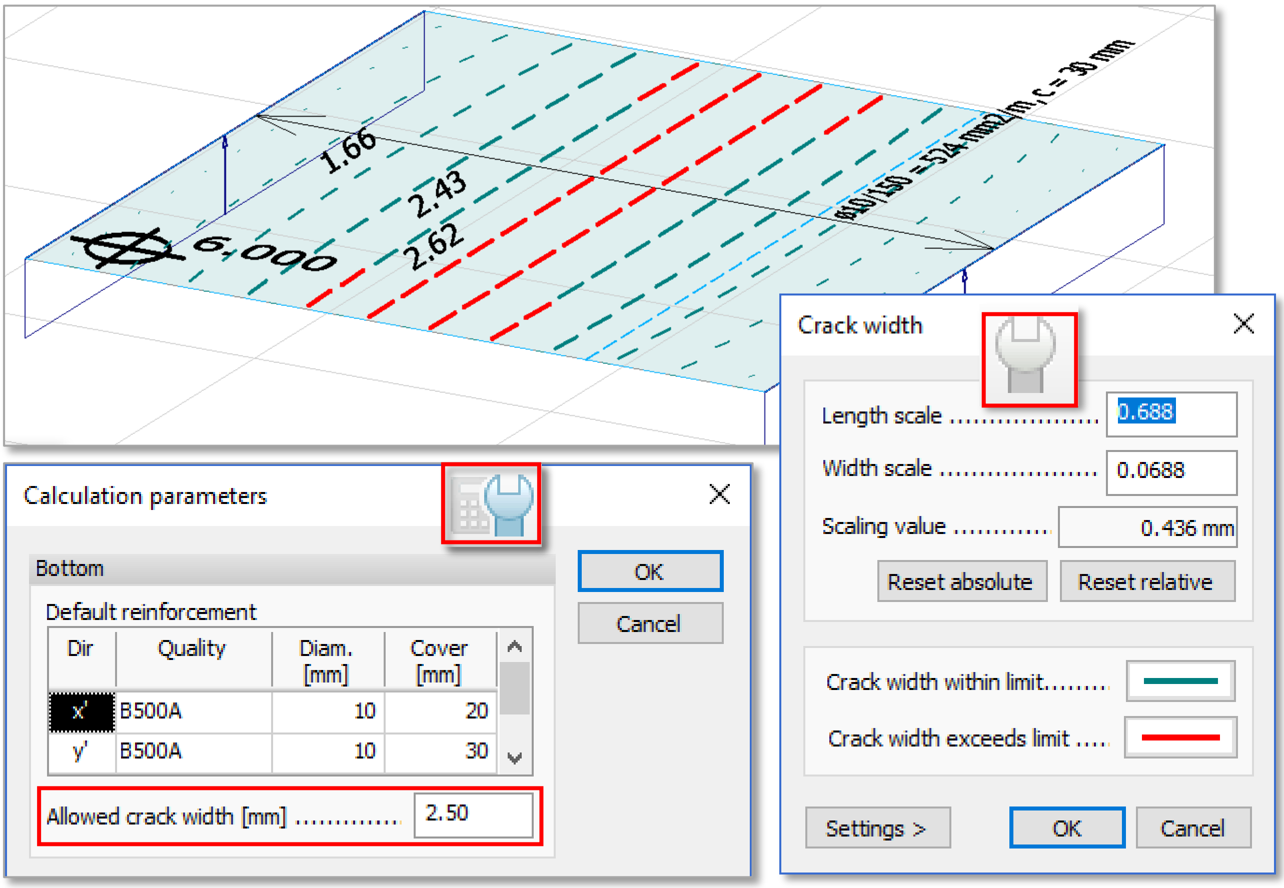
The last selected display settings for display automatic values are applied for all the new results, until they are modified by the user.

These settings are kept for the results when

- they are added to the documentation,
- they are hidden then shown again,
- the model is recalculated (naturally, only the display settings stay the same – the actual numerical values will be refreshed according to the new results).

9.3. Crack width result colour code

Crack width values that exceed the specified limit are displayed with a different colour. The maximum allowed value of crack width can be set at the *Calculation parameters* ( ). At the *Display options* ( ), the weight, scale and colour of the crack lines can be defined.



9.4. Reading result files has been sped up

Getting the data from the results files has been made more effective. This means that displaying results will be faster.

`

10. Documentation

10.1. Listing point support coordinates

From now it is possible to list the coordinates of *Point supports* and *Point support groups* when listing:

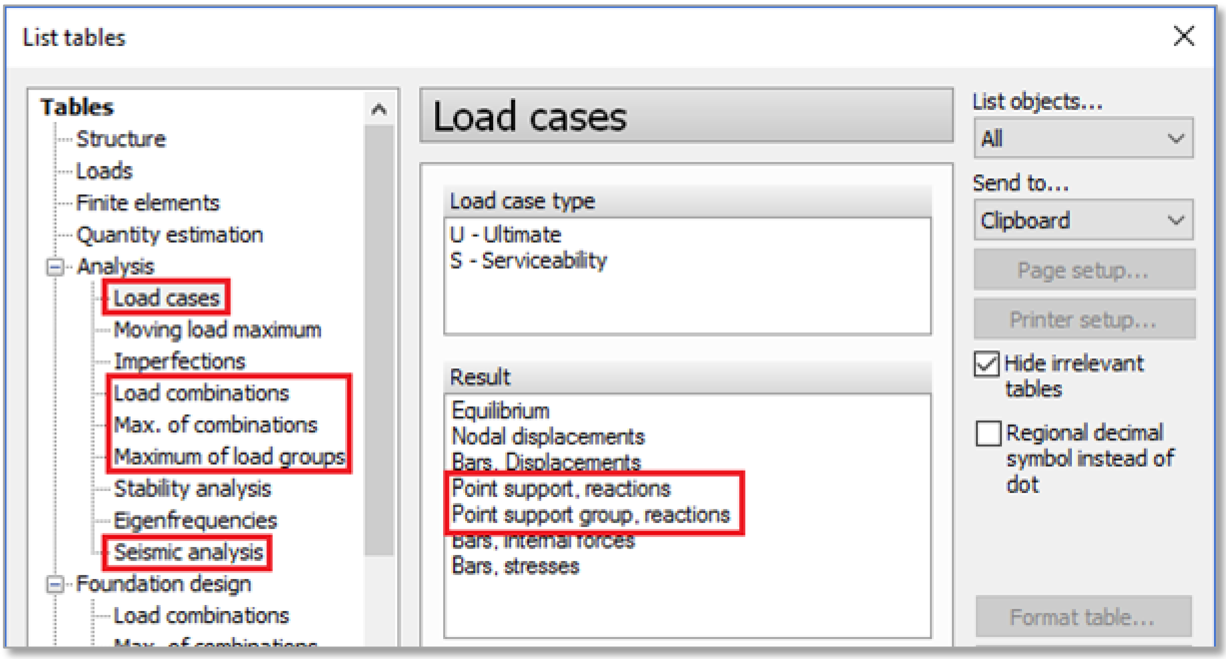
Structure

- Point supports,
- Point support groups

Analysis

- Point support, reactions
- Point support group, reactions

for *Load cases, Load combinations, Max. of combinations, Maximum of load groups*, and *Seismic analysis*.



To include the coordinates of point supports or point support groups, simply select them from the available results. The global coordinates of the supports will be displayed from column 2 to column 4 in the list (see below).

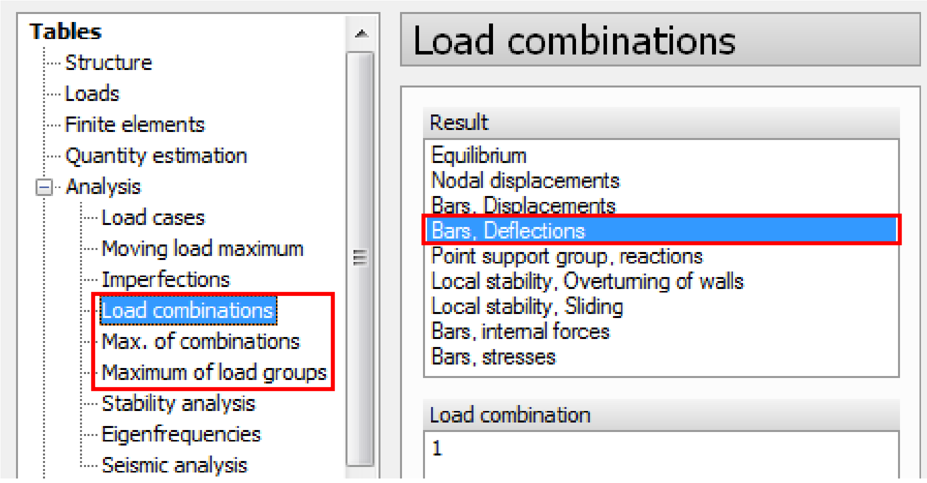
9							
10	Point support, reactions						
11	ID	x	y	z	Node	F	M
12	[-]	[m]	[m]	[m]	[-]	[kN]	[kNm]
13	S.3	5.519	11.611	0.000	3	-6.460	0.000
14							

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Point support group, reactions												
2	ID	x	y	z	Node	Fx'	Fy'	Fz'	Mx'	My'	Mz'	Fr	Mr
3	[-]	[m]	[m]	[m]	[-]	[kN]	[kN]	[kN]	[kNm]	[kNm]	[kNm]	[kN]	[kNm]
4	S.1	3.296	11.611	0.000	1	0.000	0.000	-2.118	0.000	0.709	0.000	2.118	0.709
5	S.2	7.743	11.611	0.000	5	0.000	0.000	-2.540	0.000	0.000	0.000	2.540	0.000

`

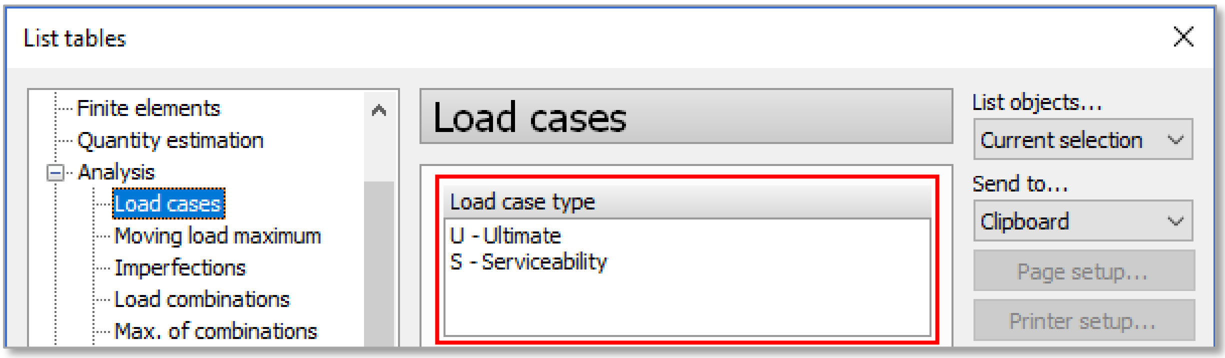
10.2. Deflection results

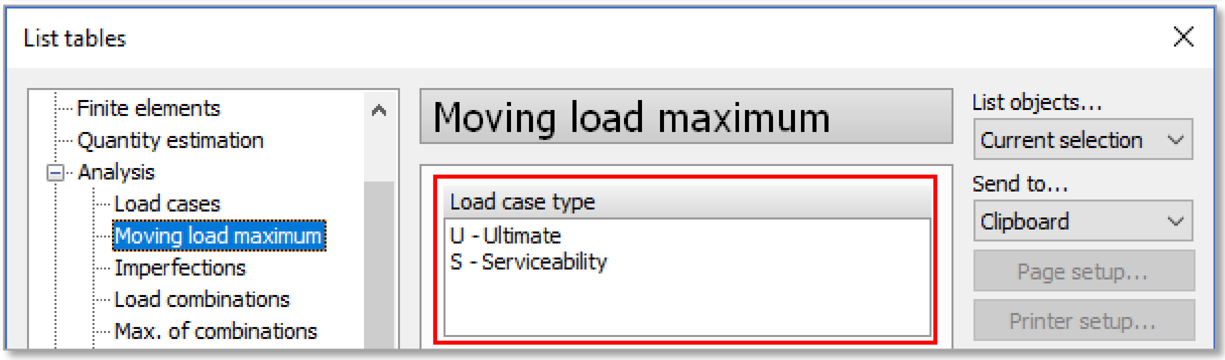
Deflection results can be listed for Load combinations, Max. of load combinations and Max. of load groups.



10.3. Modifications in listing of load case results

When listing the Analysis results at *Load case* and *Moving load maximum*, two limit states can be used: Ultimate and Serviceability.





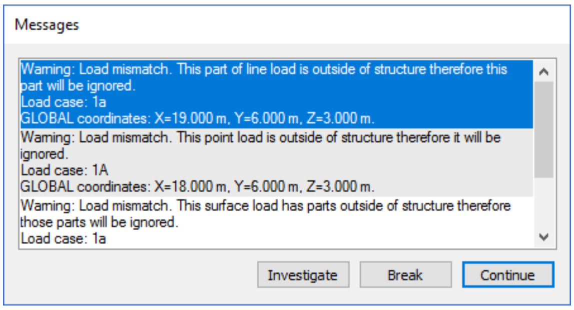
11. Others

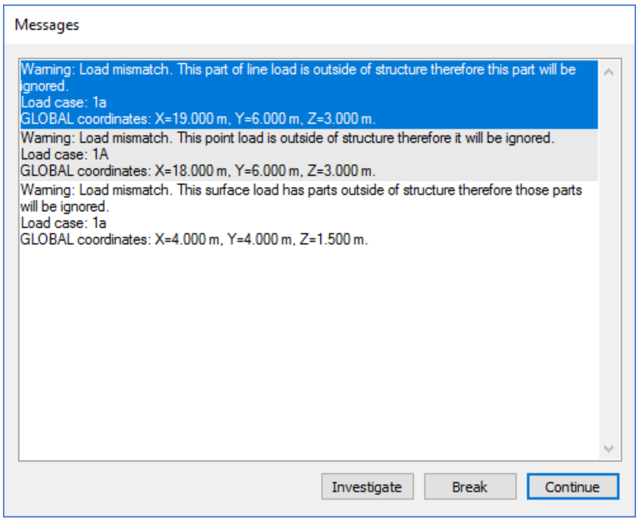
11.1. Romanian annex

The Romanian National Annex for Eurocode standards is now built into the software.

11.2. Resizable message window

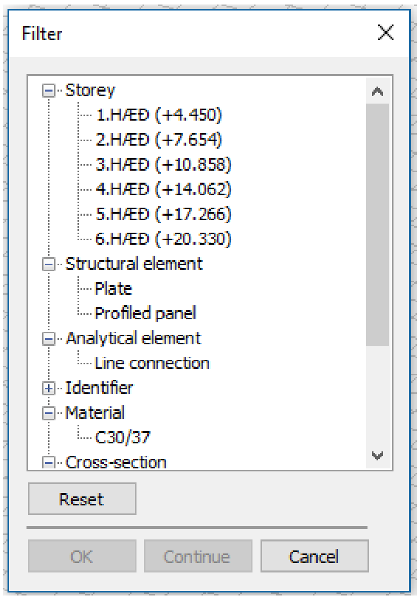
The message window can now be resized by clicking and dragging the lower right corner, as usual in Windows operating systems.

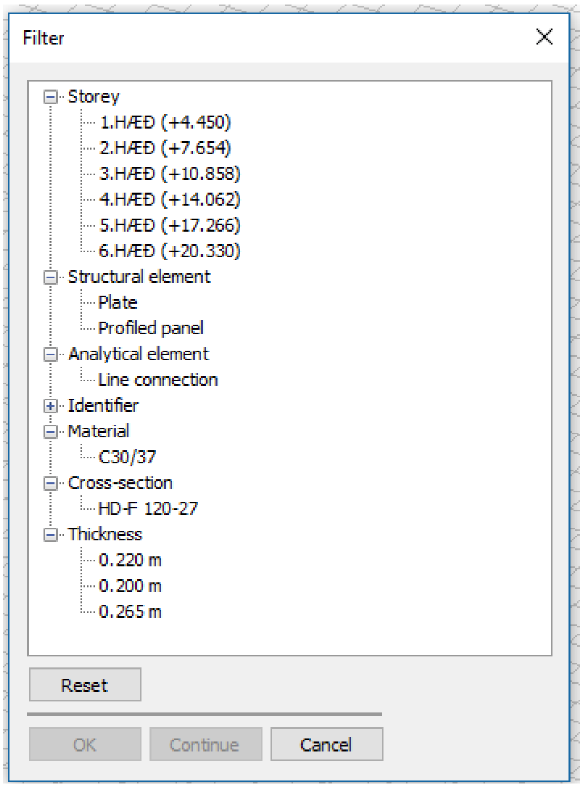




11.3. Resizable Filter dialog

The Filter dialog can now be resized by clicking and dragging the lower right corner, as usual in Windows operating systems.





`

11.4. Struxml available in 3D Frame module

In FEM-Design 16 export and import via Struxml files is available in 3D Frame module.