

BASIC CONCEPT TRAINING

SCIA Engineer 19

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2019 SCIA nv. All rights reserved.

Table of Contents

Table of Contents	3
Chapter 1: Getting started.....	7
1.1. General user interface	7
1.1.1. Main tree	7
1.1.2. Toolbars	8
1.1.3. Properties window	9
1.1.4. Command line.....	9
1.1.5. Shortcuts for view settings	10
Snap settings.....	11
1.2. Options.....	11
1.2.1. Environment.....	11
1.2.2. Other	12
1.3. Project data.....	13
1.3.1. Basic data	13
1.3.2. Functionalities	14
1.3.3. Actions	15
1.3.4. Unit set.....	15
1.3.5. Protection.....	16
Chapter 2: Modelling	17
2.1. Line grid	17
2.2. Views, activity and layers.....	18
2.2.1. Views	18
2.2.2. Activities.....	19
2.2.3. Layers	20
2.3. Selection	21
2.4. View parameters for all.....	21
2.5. Material database	22
2.6. Cross-sections	22
2.7. 1D Elements.....	24
2.8. 2D Elements.....	25
2.9. Load panels	27
2.10. Supports	29
2.11. Catalogue blocks	31
2.12. Haunch	32
2.13. Hinges	34
2.14. Beam non-linearity.....	35

2.15.	Subsoil	36
2.16.	Modify shape	38
2.17.	Connect members	39
2.18.	Check structure data	40
2.19.	Modification commands	41
2.19.1.	Copy.....	41
2.19.2.	Multiple copy.....	41
2.19.3.	Mirror.....	42
2.19.4.	Break in defined points	42
2.20.	Table input	43
Chapter 3:	Loads	44
3.1.	Load cases	44
3.2.	Load groups	44
3.2.1.	Permanent load group	44
3.2.2.	Variable load group.....	45
3.3.	Combinations	46
3.3.1.	Linear combination	46
3.3.2.	Envelope combination	47
3.3.3.	Eurocode combination	48
3.4.	Nonlinear combination	50
3.5.	Result class	51
3.6.	Point force	52
3.6.1.	Point force in node.....	52
3.6.2.	Point force on beam.....	53
3.6.3.	Free point force.....	53
3.7.	Line force	56
3.8.	Surface load	56
Chapter 4:	Calculation	57
4.1.	Mesh	57
4.2.	Calculation/solver	58
Chapter 5:	Results	59
5.1.	Test input data	59
5.2.	Requesting results	60
5.2.1.	Displacement of nodes	61
5.2.2.	3D Results	61
5.2.3.	Results per component.....	61
5.2.4.	Bill of material	62
5.2.5.	Calculation protocol	62

5.2.6.	Setting the properties menu.....	63
5.3.	Table results	68
5.4.	Results preview.....	69
5.5.	Section on 2D member	70
5.6.	Averaging strip.....	71
5.7.	Integration strip / integration member	73
Chapter 6:	Steel design	75
6.1.	Steel setup	75
6.2.	Buckling settings	76
6.2.1.	Default buckling calculation	76
6.2.2.	Assign buckling groups.....	78
6.3.	Member check data.....	80
6.4.	ULS Check	82
6.4.1.	Graphic output	82
6.4.2.	Preview	83
6.4.3.	Table output	84
6.5.	SLS Check	85
6.6.	General autodesign.....	86
6.7.	Connections	90
Chapter 7:	Concrete design	93
7.1.	Concrete setup	93
7.2.	Recalculated internal forces	93
7.3.	Provided reinforcement.....	96
7.4.	Required reinforcement	98
7.4.1.	1D members	98
7.4.2.	2D members	99
7.5.	User reinforcement	101
7.5.1.	1D Members	101
7.5.2.	2D members	106
7.6.	1D ULS & SLS checks	107
7.6.1.	Capacity response	107
7.6.2.	Capacity diagram.....	107
7.6.3.	Shear + Torsion	108
7.6.4.	Stress limitation.....	108
7.6.5.	Crack width	109
7.6.6.	Deflection	110
7.7.	2D Crack width check.....	111

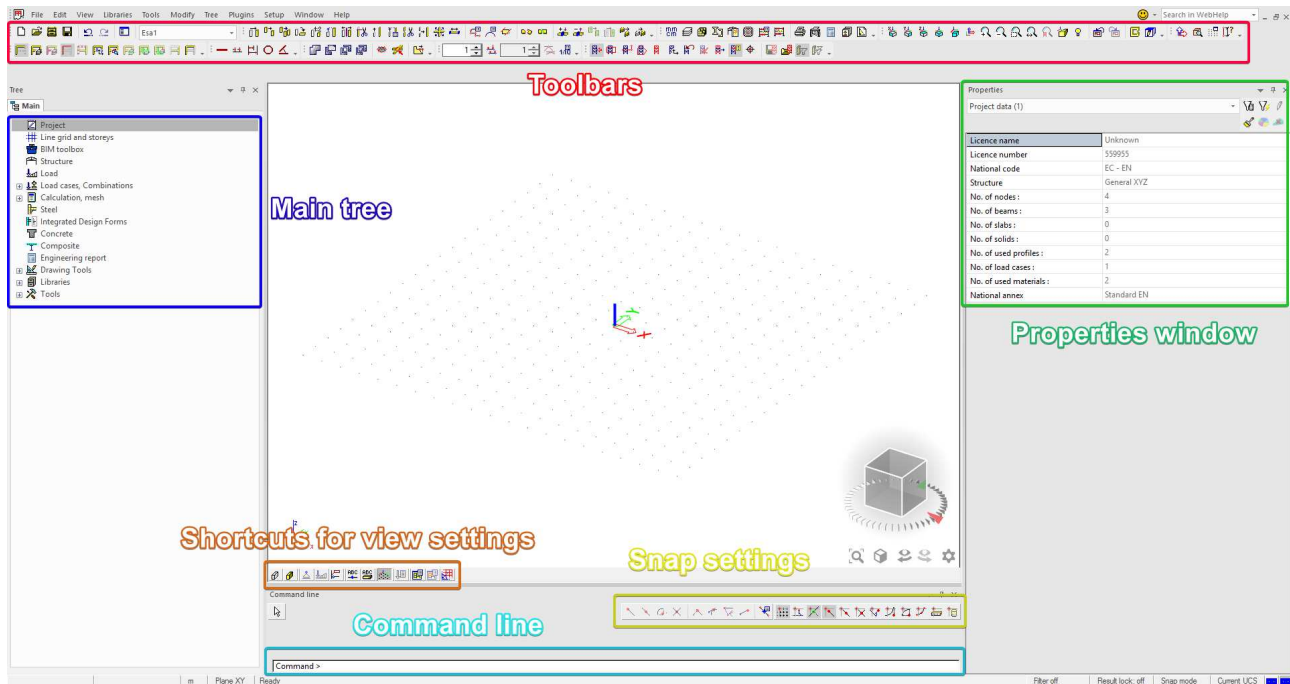
7.8. Punching check	113
7.9. Code dependent deformations	115
Chapter 8: Engineering report	116
8.1. General interface	116
8.2. General page layout	118
8.2.1. Page layout	118
8.2.2. Page format and page break	119
8.2.3. Header and footer	120
8.2.4. Chapter	120
8.2.5. Formatted text.....	121
8.3. Adding pictures	122
8.3.1. Screenshot or live pictures	122
8.3.2. Inbox	123
8.3.3. Picture gallery	123
8.3.4. Paperspace gallery	125
8.3.5. Generating result pictures.....	125
8.3.6. Report template	128
8.3.7. Export.....	130
8.3.8. Print.....	130

Chapter 1: Getting started

This chapter will provide you a general overview of SCIA Engineer. It contains the general interface, the options and the project data.

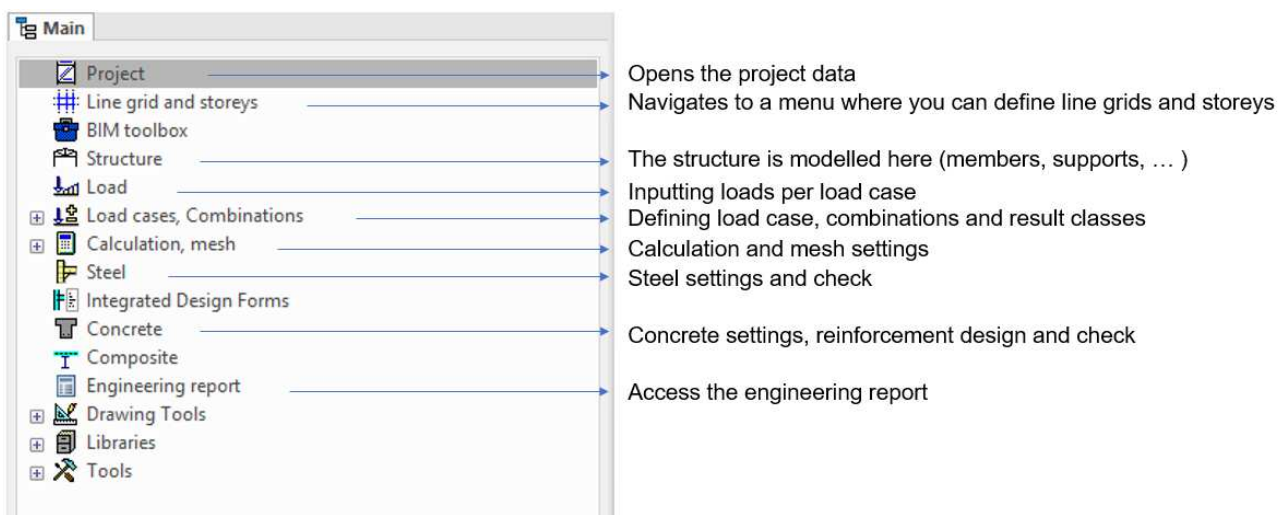
1.1. General user interface

The general user interface (GUI) of SCIA Engineer consists of the following elements: the main tree, the toolbars, the properties window, the command line, the shortcuts for view settings and the snap settings. These components will be explained in detail during this chapter.



1.1.1. Main tree

The main tree window provides an easy way to navigate between the different functions and services of SCIA Engineer. If you click on one of the items in the main tree, a new (sub)tree with more detailed functions for this service will open. The main tree is arranged in a logical order. This means that functions/services that need to be used earlier on in your project are on top and functions/services that you use later in the project are at the bottom. For example, 'Line grid' is placed above 'Structure' because the line grid usually gets defined before modelling the structure.



1.1.2. Toolbars

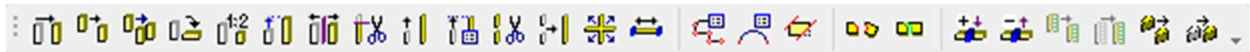
The toolbars give you quick access to the most used tools in SCIA Engineer. The tools listed are independent of the tree menu and will always stay the same. More information concerning the tools is given when you hover over the icon. Below the available toolbars are listed.

Basic



This toolbar contains the basic tools: starting a new project, opening a project, saving a project, changing between projects, ...

Geometry manipulations



This toolbar contains all geometry manipulations: copy, move, mirror, connect nodes, ...

Project



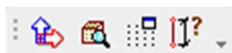
This toolbar contains the general settings and commands concerning the project: libraries (such as materials and cross-sections), units, structure check, engineering report, ...

View



This toolbar contains all the actions you can perform to change the view: view from x, y or z axis, zoom, zoom selection, structural model, ...

Tools



This toolbar contains settings for the UCS (User Coordinates System), the clipping box, settings for the dot grid and information about point coordinates.

Activity



This toolbar contains tools to adapt the activity of the model. By using this toolbar, you can easily show or hide certain elements.

Dimension tools



This toolbar contains shortcuts to start the dimension function.

Modelling tools



This toolbar contains some manipulation of general solids. (In a standard model general solids are not used).

Scales



In this toolbar you can set the scale of model data (supports, loads, ...) and results.

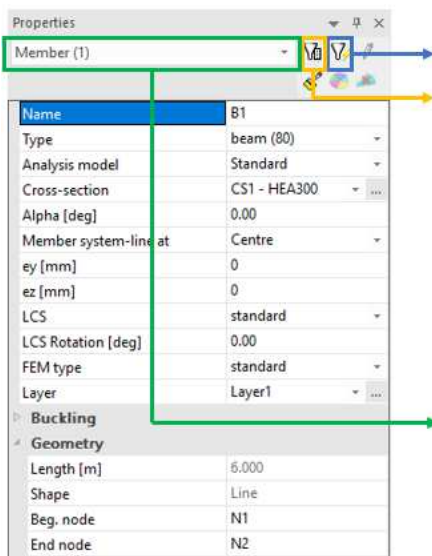
Selection of object



This toolbar contains some settings that influence selection of objects.

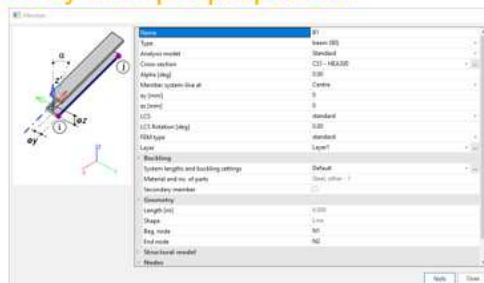
1.1.3. Properties window

The properties window displays all the properties of the selection. You can easily check or modify the properties by using this window. You can use the drop-down menu to change the entity when multiple entities are selected.

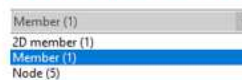


Select all members with the same property as highlighted

Filter by multiple properties:



Switch between selected entities:



1.1.4. Command line

The command line has two functions:

- The command line will mention you the necessary steps that you need to execute while performing an action.

```
Copy - Select entities to be copied (finish selection with ESC) >
```

- You can start a function or perform an action by typing the correct command.

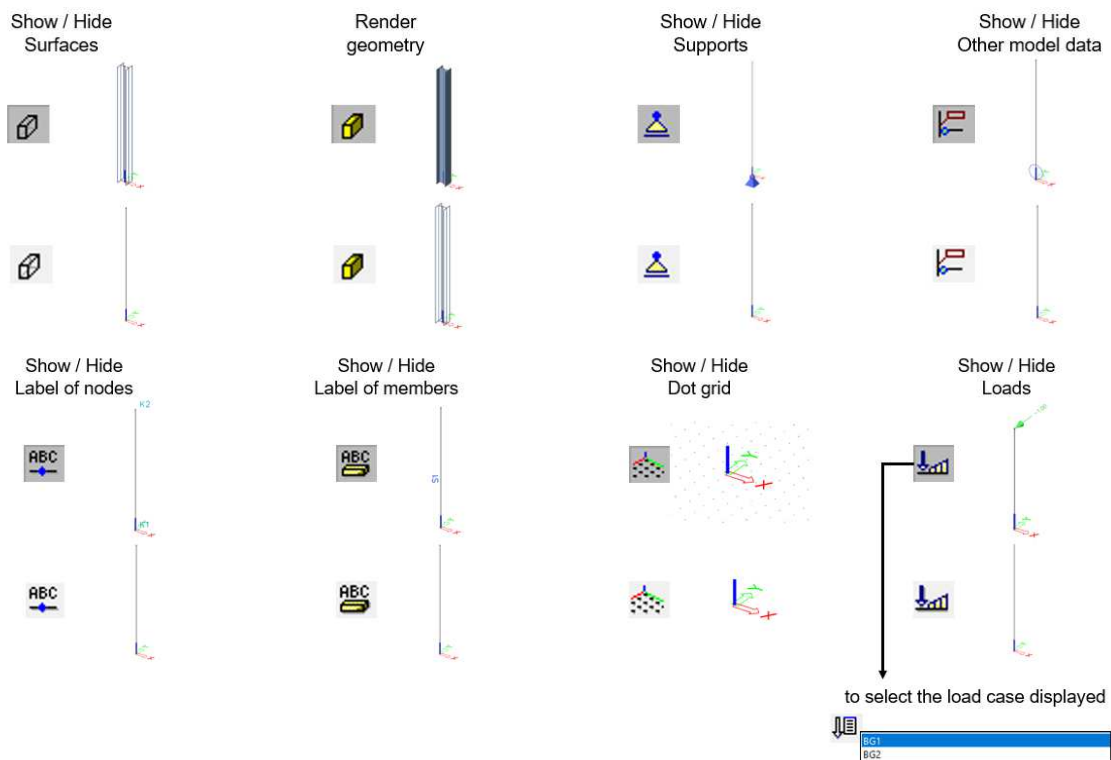
Some useful commands:

- sel → select a node/member
- selmn → select a meshnode
- x, y, z → view in X, Y or Z direction
- copy, move, rotate, mirror, ... → start geometry manipulation
- calc → start calculation

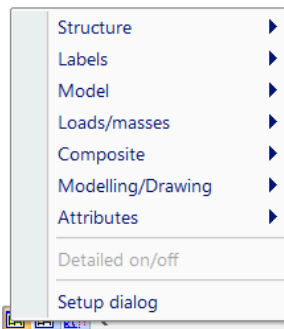
You can find the full list of commands on: https://help.scia.net/webhelplatest/en/#vit/gui/list_of_commands.htm

1.1.5. Shortcuts for view settings

The shortcuts for view settings enables the quick adjustment of the view parameters. The following pictures provide a graphical representation of all the options.

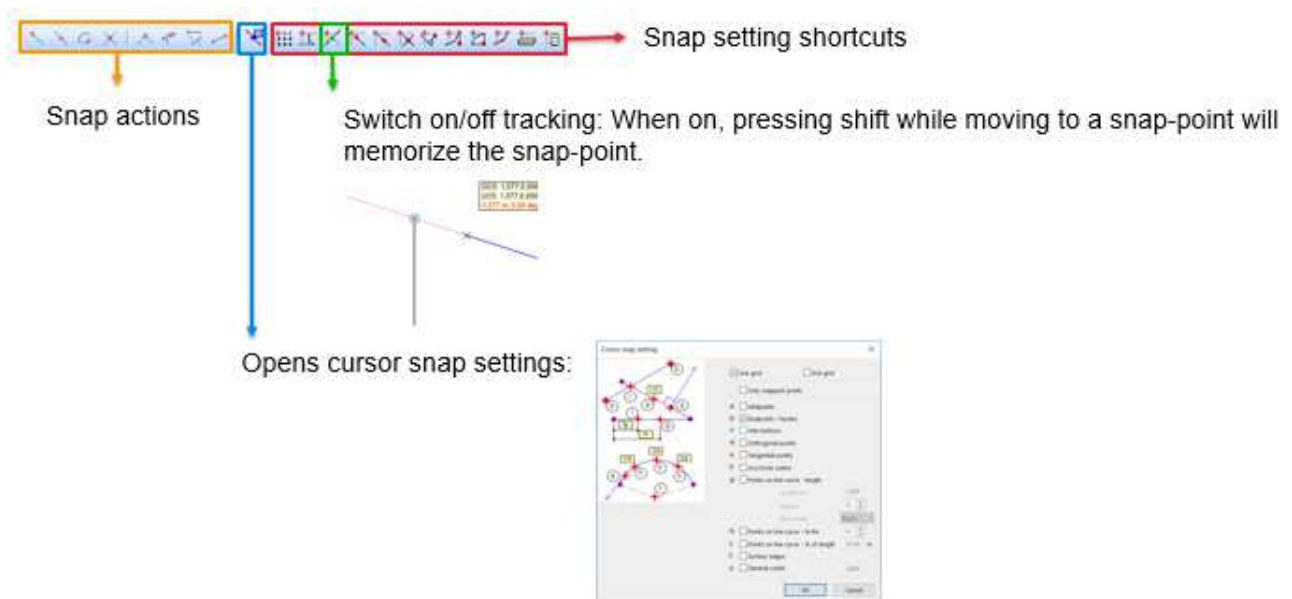


Next to the shortcuts you also have a trimmed down version of the view parameters in the toolbar. You can access it by clicking on the following icon.



Snap settings

In the toolbar 'Snap settings' you can modify the way the program snaps to points while modelling. A graphical overview of all the settings is provided below. More information concerning the tools is given when you hover over the icon.



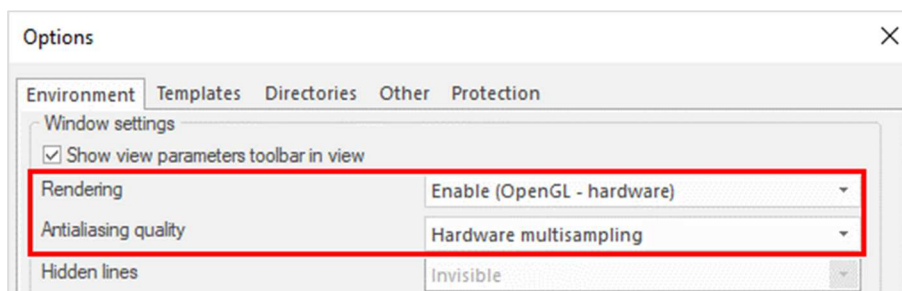
1.2. Options

In this chapter the most essential settings in the options menu are explained. You can find the options menu in the setup of the project.

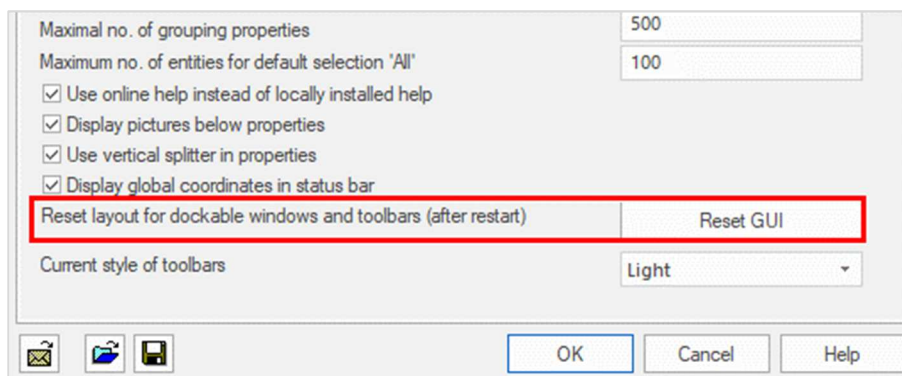
1.2.1. Environment

In the 'Environment' tab you can edit the following settings:

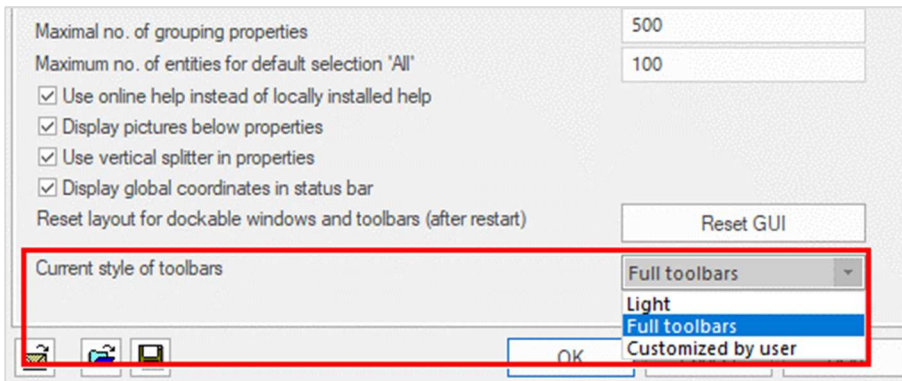
- 1) You can change the rendering and antialiasing settings. If the graphics card of your computer is having issues with rendering, the following settings have to be decreased.



- 2) You can reset the graphical user interface.

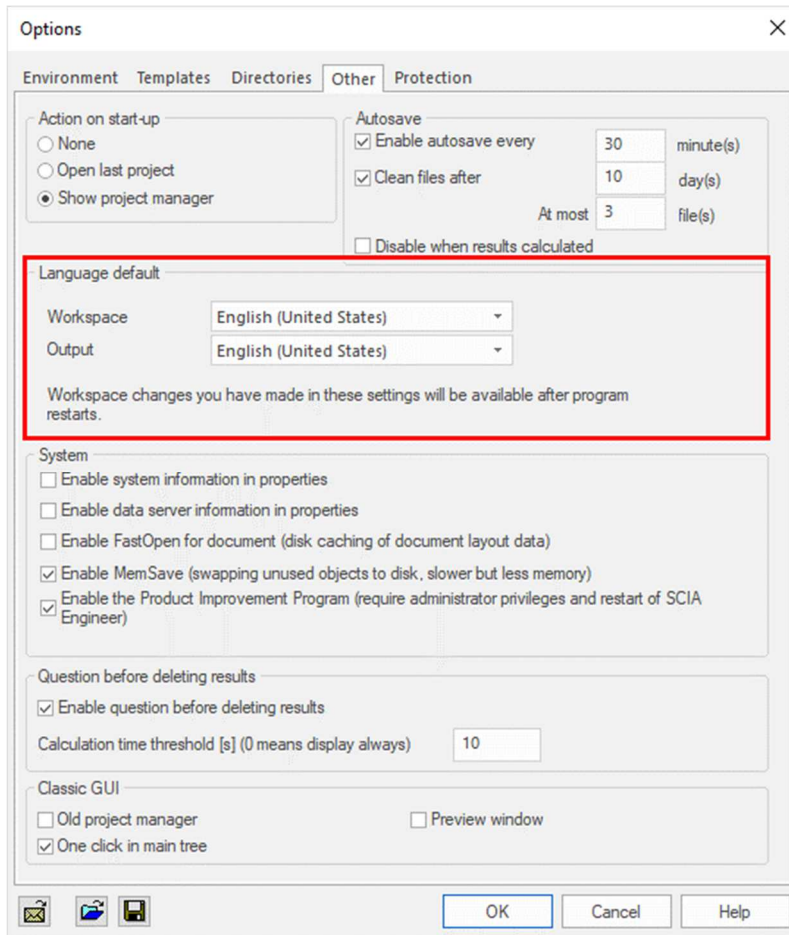


- 3) You can modify the presentation of the toolbars by changing the option 'Current style of toolbars'. In order to display all the available toolbars, the option 'Full toolbars' has to be selected.



1.2.2. Other

In the 'Other' tab you can edit the language used in SCIA Engineer. The workspace and the output can have different languages.



NOTE: only the installed languages are shown here. You should also have the language module(s) to be able to use them.

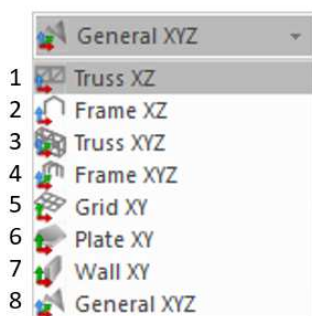
1.3. Project data

The first action that you need to perform while starting a new project, is to define the project data. The project data window consists of the following tabs: 'Basic data', 'Functionality', 'Actions', 'Unit Set' and 'Protection'.

1.3.1. Basic data

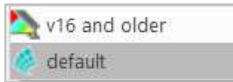
In the 'Basic data' tab you define some specific data for your project.

- **Material:** choose the materials that you want to use in your project and the default value of the quality.
- **Code:** select the code and national annex you want to apply.
- **Structure environment:**



1		Only 1D-members, 2D environment, axial force only
2		Only 1D-members, 2D environment
3		Only 1D-members, 3D environment, axial force only
4		Only 1D-members, 3D environment
5		Horizontal grate
6		1D-members, 2D-members, 2D environment (Horizontal)
7		1D-members, 2D-members, 2D environment (Vertical)
8		Everything

Post processing environment: There are 2 post processing environments: 'v16 and older' and 'default'. A post processor converts the results of the FEM calculation into useable results and a graphical representation.

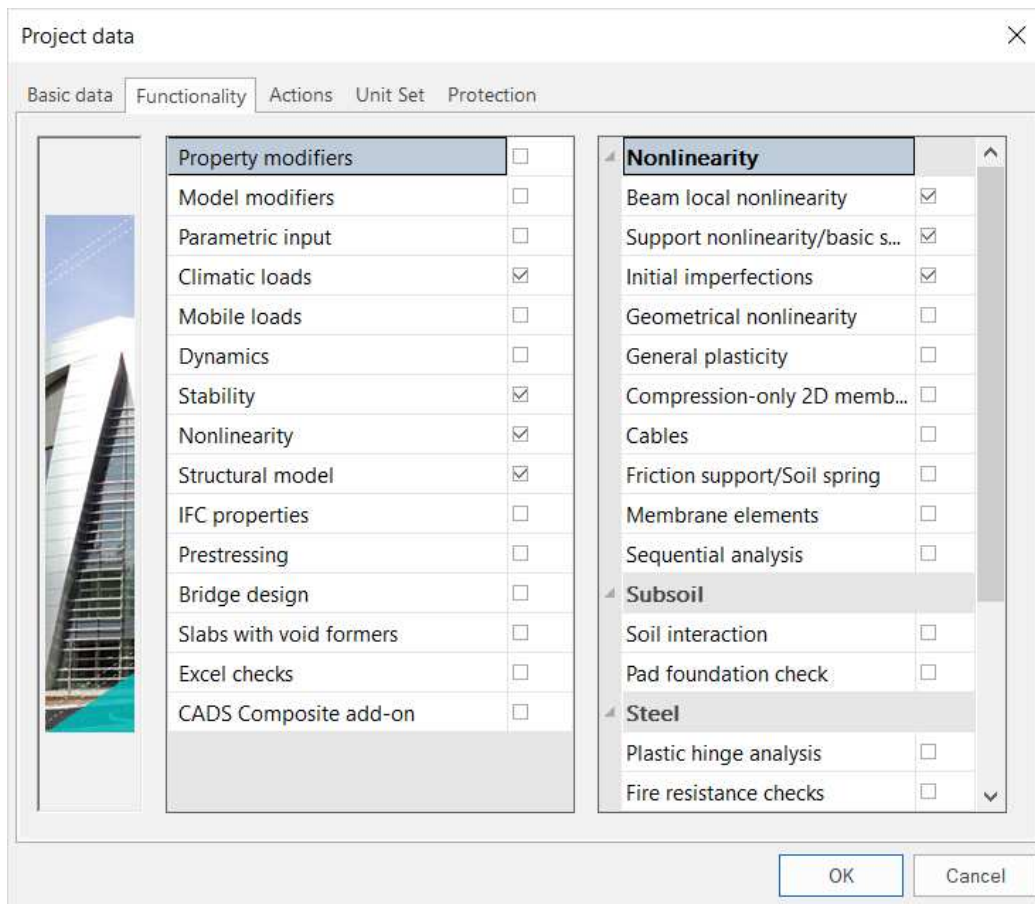


'v16 and older' post processor: this is the post processor that was used by SCIA Engineer 16 and older and contains all the old functionalities. You need to select this post processor when you need a functionality that is not (yet) developed in the 'default' post processor.

'default' post processor: from SCIA Engineer 17 on there is a new post processor. This post processor is compatible with 64 bit and has new and more polished functions. Not every function of the 'v16 and older' post processor is converted to the 'default' post processor.

1.3.2. Functionalities

In the 'Functionalities' tab you need to check the functionalities you want to use in SCIA Engineer. From SCIA Engineer 18 on the program will automatically turn on some functionalities based on the basic data set.



1.3.3. Actions

In this tab you can set the acceleration of gravity, the wind load, the snow load, the model factor for pond loads, the factor for concomitant components of a seismic combination and the automatic code combinations.

Project data

Basic data Functionality **Actions** Unit Set Protection


Acceleration of gravity: m/s²

Wind Load: ... EC 1 / 26,200m/s / 0

Snow Load: ... EC 1 / Sk=1,00kN/m² Ce=1,00 Ct=1,00

Pond Load: Model factor:

Seismic Combinations: Factor for concomitant components:

Code Combinations: Automatic 

OK Cancel

1.3.4. Unit set

In this tab you choose the unit set used in the project.

Project data

Basic data Functionality Actions **Unit Set** Protection

Type of units

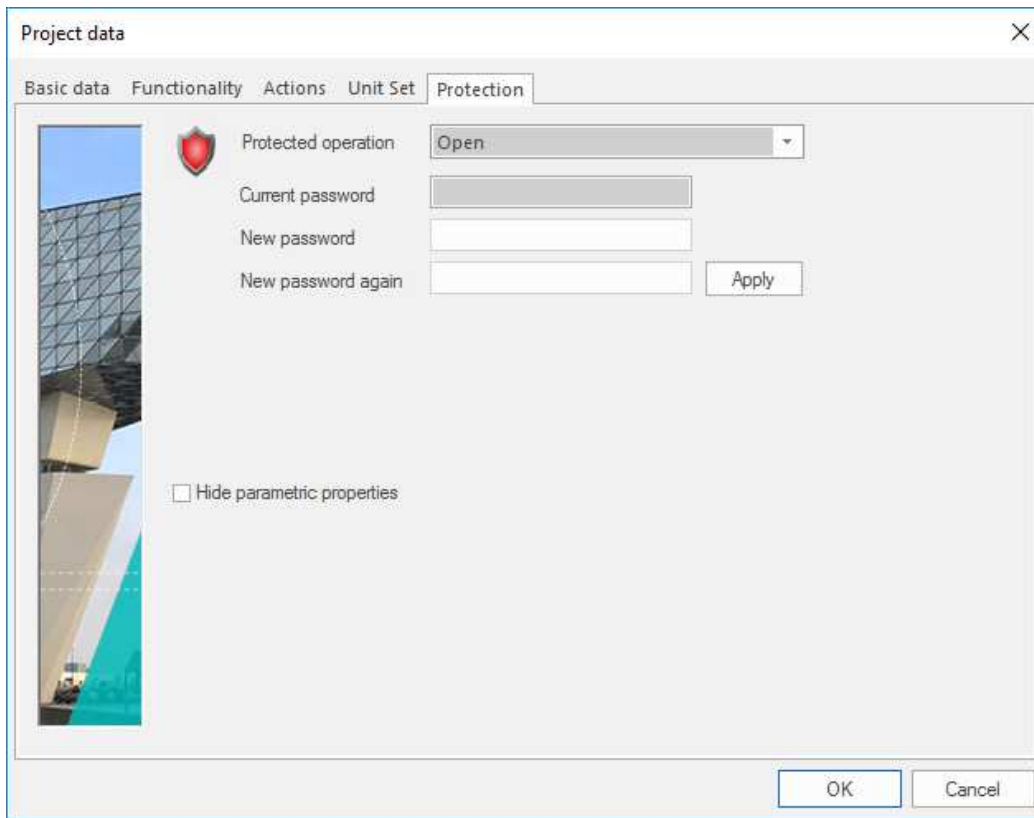
Metric format

Imperial format

OK Cancel

1.3.5. Protection

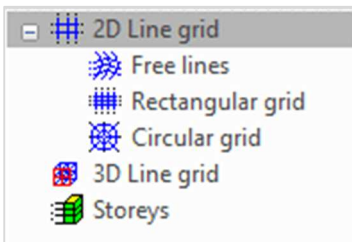
In this tab you can password protect your project.



Chapter 2: Modelling

2.1. Line grid

In order to insert a line grid and/or storeys, click on 'Line grid and storeys' in the main tree.

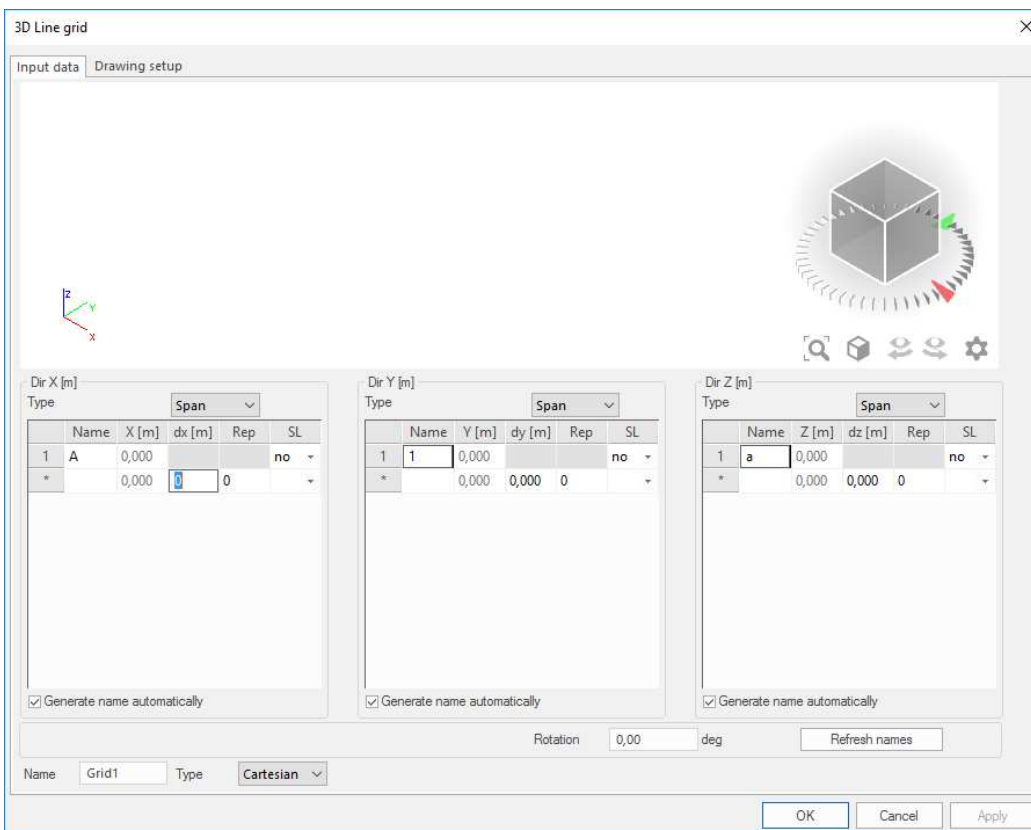


2D Line grid – Free lines

By using this option, you can draw free lines to create or to add to a line grid.

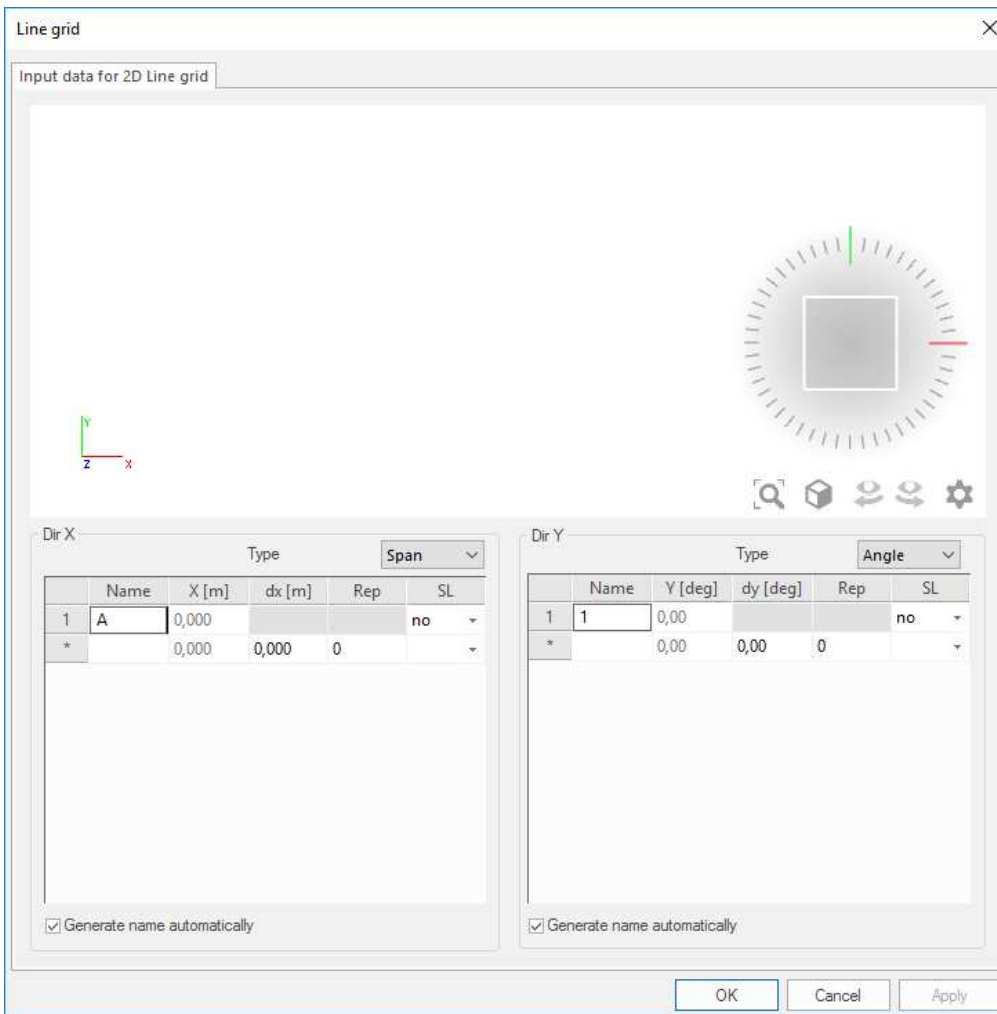
2D Line grid - rectangular grid/3D Line grid

In this menu you need to define the distance **dx,y,z** between two lines of the grid and the amount of lines (**Rep**).



2D Line grid – Circular grid

In order to insert a circular grid, you need to define a distance dx and a rotation dy.



NOTE: you can only edit the line grid (by clicking on it) in the 'Line grid and story's menu. In the other menu's the line grid won't be editable.

2.2. Views, activity and layers

2.2.1. Views

There are 3 ways to change the view in the model.

1) Using shortcuts

- CTRL + right click = rotate the model
- SHIFT + right click OR pressing the scroll wheel = move/pan the model
- SCROLL = zoom in/out

2) Using the view toolbar



In this toolbar you can choose to set the view in X, Y or Z direction or in orthogonal direction.

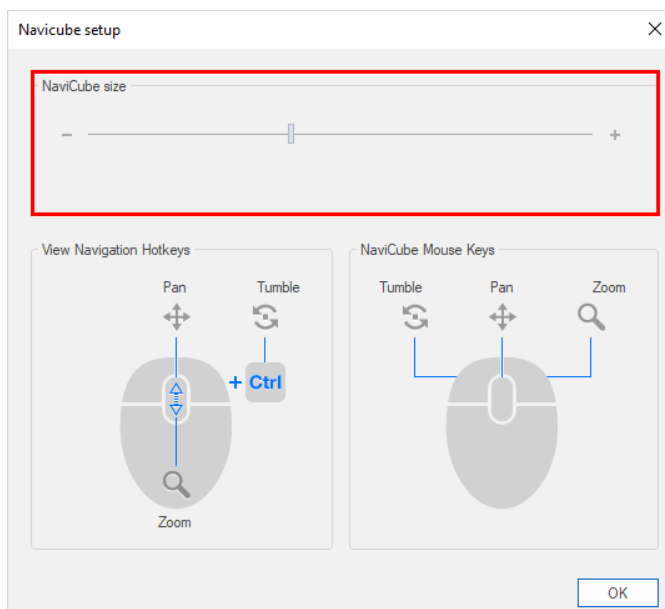
3) Using the Navicube



With the Navicube you can change the view in an easy way:

- left click = rotate;
- right click = zoom;
- clicking on a plane will give you a view perpendicular to this plane;
- clicking on a corner will give an orthogonal view (from this point).






You can change the size of the Navicube by pressing on the gear icon (you cannot fully hide the Navicube).




2.2.2. Activities

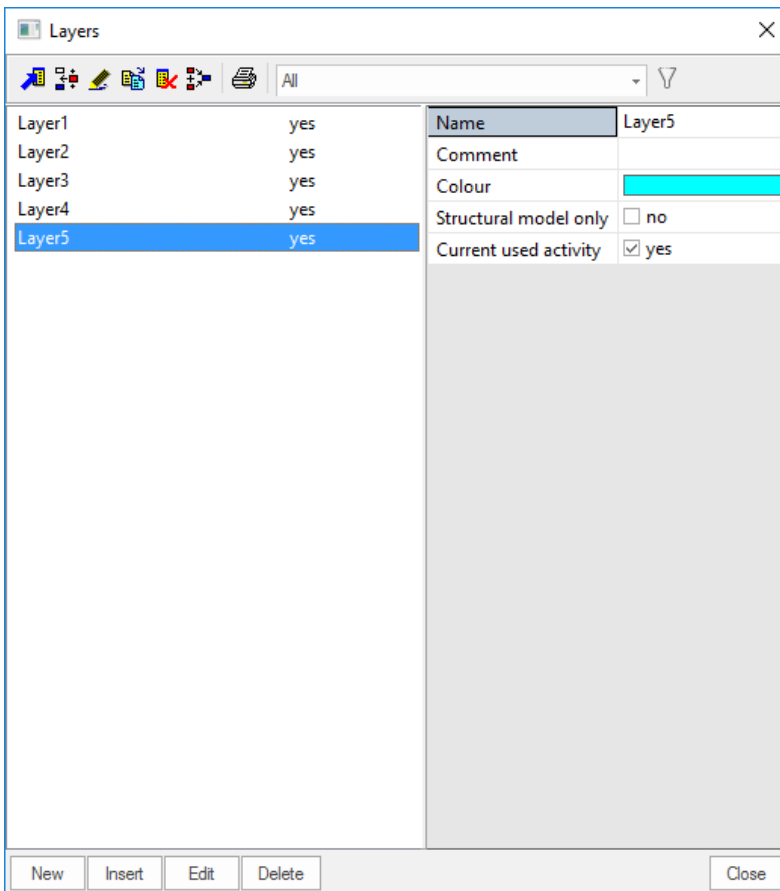
In SCIA Engineer you can use 'Activities' toolbar to hide/show elements.



-  Put activities on
-  Add activities by layer
-  Hide all elements that are not in the current selection
-  Hide the current selection
-  Show the hidden elements as a line

2.2.3. Layers

You can open the 'Layers' menu by clicking on the shortcut in the toolbar  OR go to 'Tools' → 'Layers'



You can create layers in this window. There are 2 checkboxes which you can check or uncheck.

- **Structural model only:** if this is checked the elements in this layer will not be taken into account in the calculation.
- **Current used activity:** if this is unchecked the elements in this layer will not be visible.

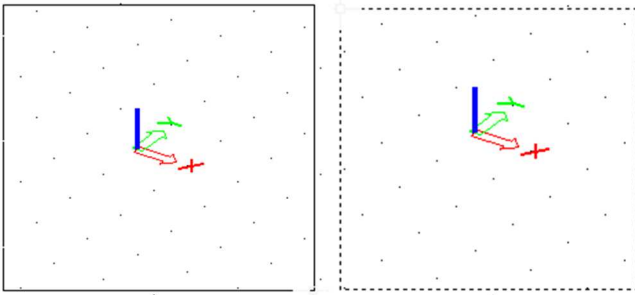
You can assign elements to a layer by selecting them and changing the layer property.

2.3. Selection

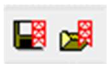
When you hold the left mouse button, you can create a selection box to select multiple elements.

Selection box with full line: this is created by dragging the mouse to the right. With this selection box, you will only select the elements that are entirely in the box.

Selection box with dashed line: this is created by dragging the mouse to the left. With this selection box, you will select all the elements that are partially in the selection box. In other words, you will select all elements fully in the box + all elements that cross the box.



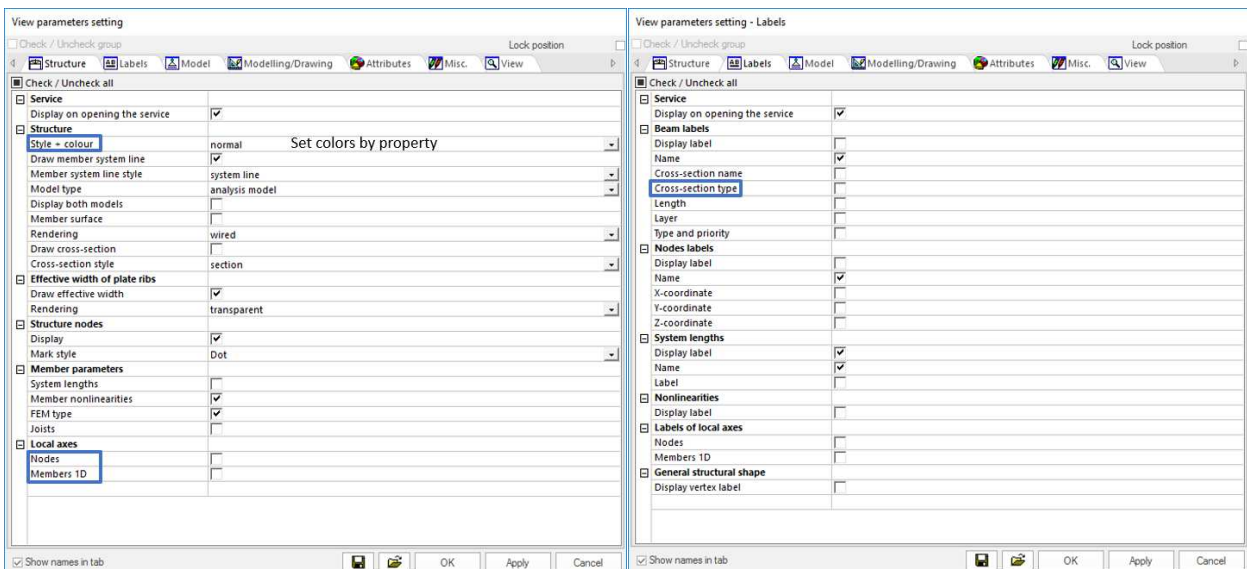
Next to selecting with the mouse and some automatic selection tools in the toolbar, you can also save a current selection. This way you can load a previous made selection easily.



You can find these two buttons in the 'Selection' toolbar or in 'Tools' → 'Selection'.

2.4. View parameters for all


You can open the window 'View parameters setting' by right clicking → 'View parameters for all'. In this menu you can edit the view settings of your project.

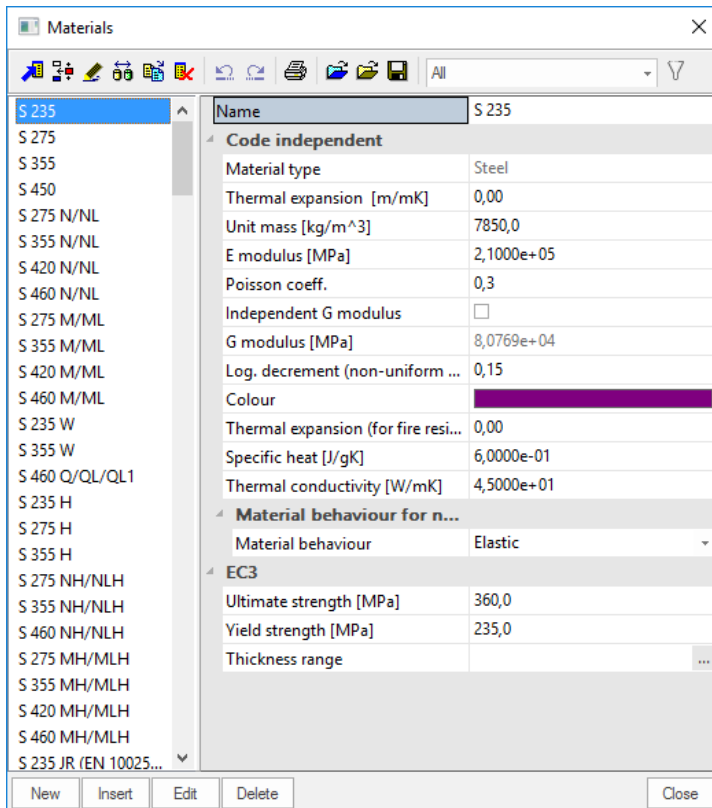


Some of the most used view parameters are highlighted:


- **Colours by property:** you can set the colour of elements in the model by element type, cross-section;
- Displaying **local axes**;
- Displaying the **cross-sections** type.

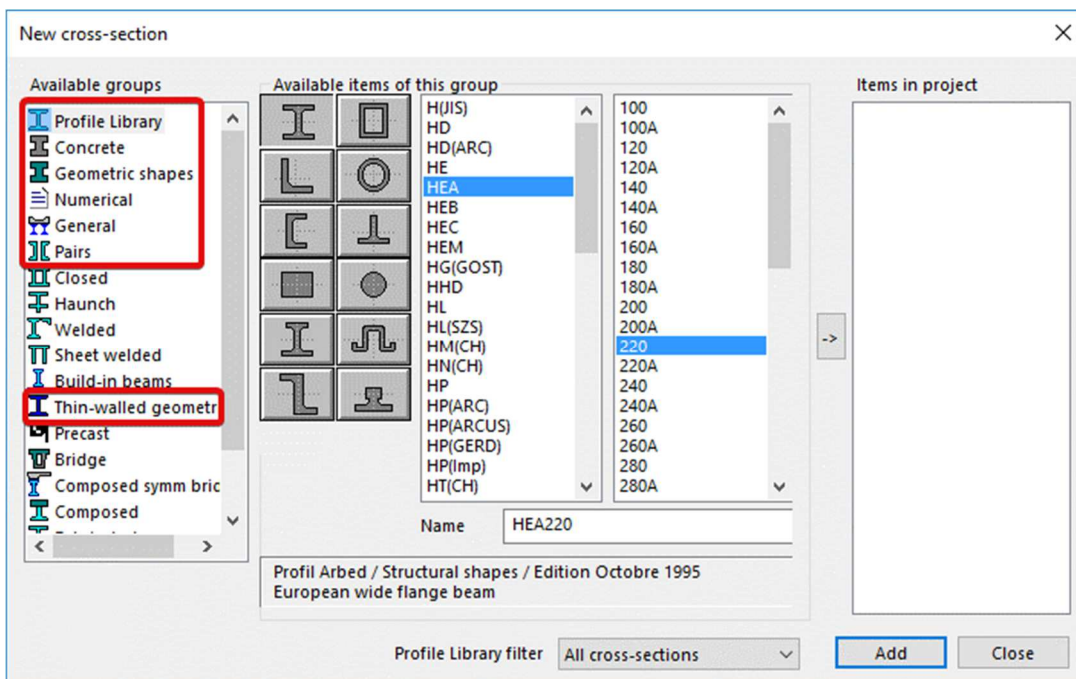
2.5. Material database

You can open the material database by going to 'Libraries' → 'Materials' or by clicking on the icon  in the toolbar 'Project'. In this window you can view all the activated materials and their different qualities. When you click on a material, you can edit its parameters.



2.6. Cross-sections

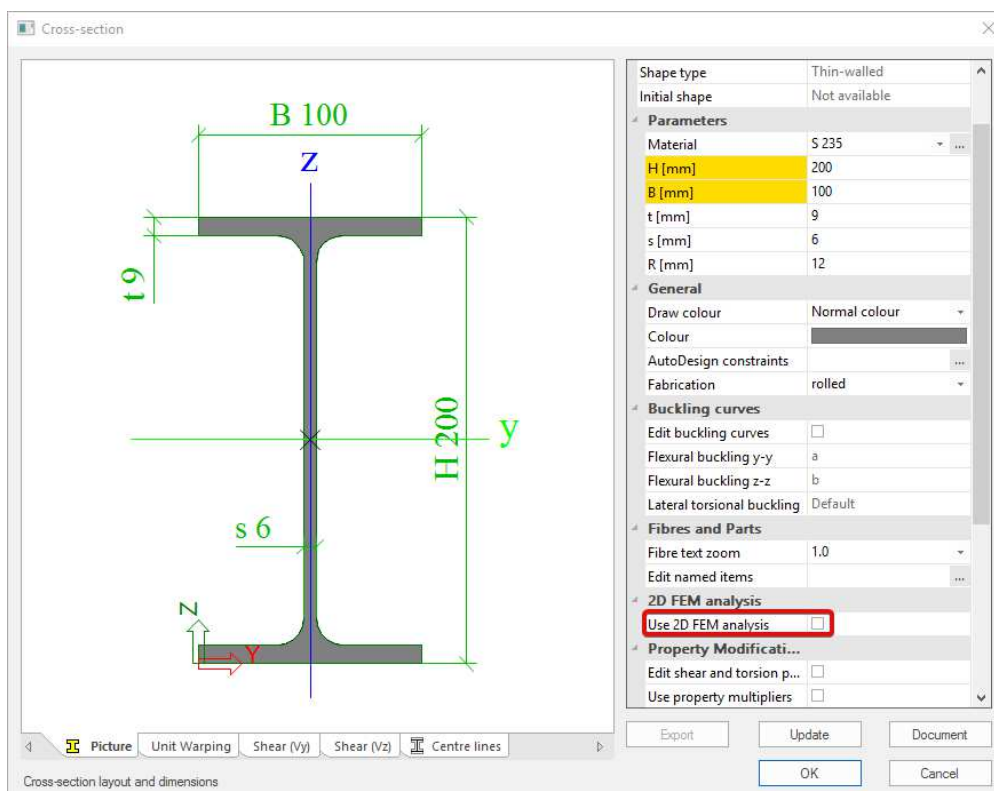
You open the cross-section database by going to 'Libraries' → 'Cross-sections' or by clicking on the shortcut  in the toolbars.



In the picture above the most used cross-sections are highlighted.

- **Profile Library:** you can find all standard (tabulated) profiles in the library.
- **Concrete:** you can use this option to create a concrete cross-section.
- **Geometric shapes:** with this option you can create a geometric cross-section. However, it is not advised to use this option. This will result in a thick-walled cross-section (buckling curves d) for steel. When you select this type of profile for concrete, you need to modify some settings. The best practice is that you use the option 'Concrete' for a concrete cross-section and the option 'Thin-walled' for a steel cross-section.
- **Numerical:** this option can you use to add a numerical cross-section. You can assign all the parameters to this cross-section without giving it a surface in the 3D-view. You can use this type for dummy elements.
- **General:** with this option you can draw a custom cross-section.
- **Pairs:** with this type you create a paired cross-section which results in a fully cooperative cross-section.
- **Thin-walled geometric shape:** with this option you create custom thin-walled (steel) cross-sections.

When you add a cross-section, the following window will open.



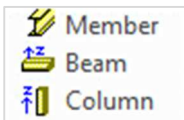
In this window you need to set the parameters. In order to view the calculated properties, you can press 'Update'. The properties of the cross-section are calculated when pressing 'OK'.

Use 2D FEM analysis: when this option is not checked, the properties are calculated with some simple formulas. If this option is checked, a mesh will be generated to calculate the properties. By checking this option A_y , A_z , I and I_w are calculated more accurate. This option is important to check – only for thick walled sections – if the cross-section is experiencing torsion.

NOTE: when you use a cross-section of the profile library, you don't need to set the settings above since the parameters and the properties are tabulated.

2.7. 1D Elements

There are three ways to draw a 1D Member:



Member: with this option you draw a 1D Member by defining two points. This element automatically has the type 'general(0)' or 'beam(80)'. You can change the shape of the member by using the toolbar below (it appears above the command line when you draw the member).



Beam: with this option you model a horizontal 1D Member by defining a point, a length, a direction and an insertion point. This element automatically has the type 'beam(80)'.

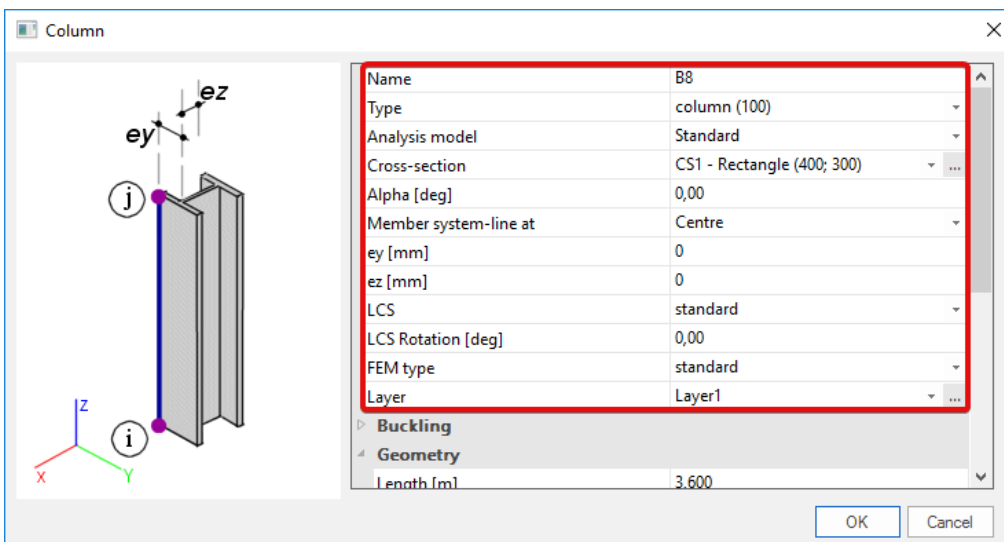
Direct	axis X
Length [m]	6,000
Insertion point	begin

Column: with this option you can draw a vertical 1D Member by defining a point, a length and an insertion point. This element automatically has the type 'column(100)'.

Length [m]	3,600
Insertion point	Bottom

NOTE: the type is important for defining connections, to perform the ULS check of steel and to determine the calculation method in the 'Concrete' menu.

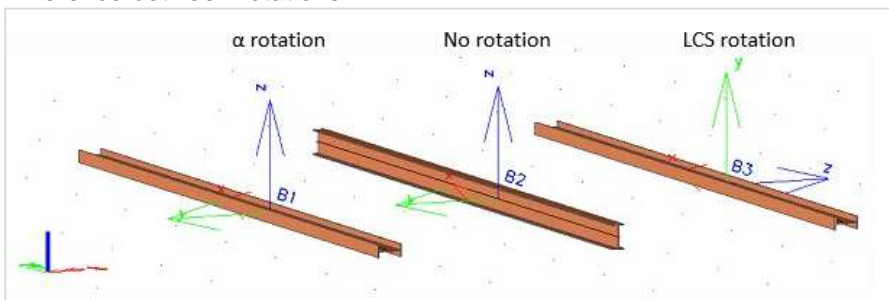
Next to the settings above you also need to set some other parameters:



- **Cross-section:** set the cross-section.
- **Alpha:** rotate the member around its own axis. With this rotation you will not rotate the LCS (Local Coordinate System). This means that the LCS is different from the strong and weak axis of the profile.
- **Member system-line at:** change the system line of the member.
- **ey, ez:** add an eccentricity
- **LCS Rotation:** rotate the member around its own axis. With this rotation you will rotate the LCS together with the member.

- **FEM type:** here you can choose for a standard FEM analysis (bending + normal force) or axial force only. This option only needs to be changed for members that are only subjected to a normal force.
- **Layer:** set the layer.

Difference between rotations:



You can add 1D-member components to the elements. The type 'Haunch' is often used and is explained further in this manual.

2.8. 2D Elements

There are three different ways to model a 2D element:



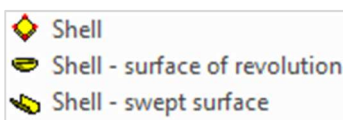
Plates: you define this 2D element by drawing the edge. You can change the shape of the edge by using the toolbar below (it appears above the command line when you draw the plate). This element automatically has the type 'plate(90)'.



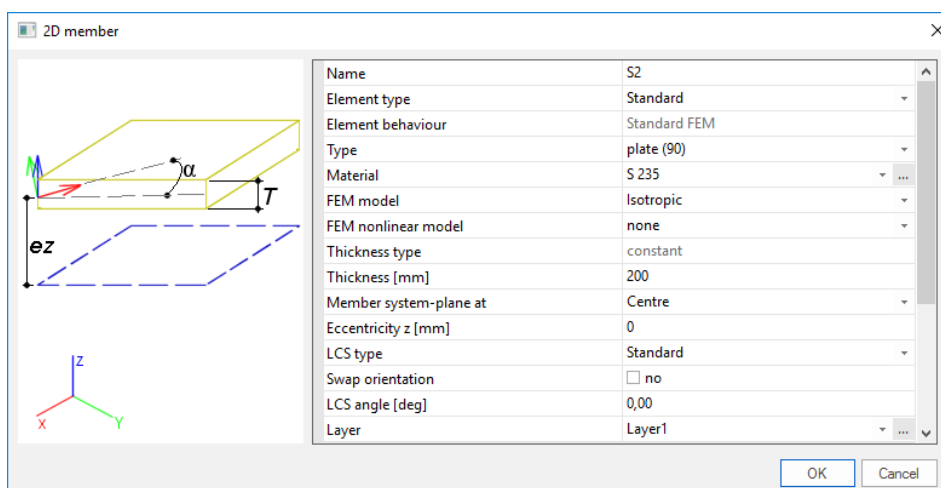
Wall: this 2D element you define by drawing the baseline (bottom or top) and by giving it a height. This element automatically has the type 'wall(80)'.

Height [m]	3,600
Insertion point	bottom

Shells: you define a shell element by drawing the edge, by a surface of revolution or by a swept surface.

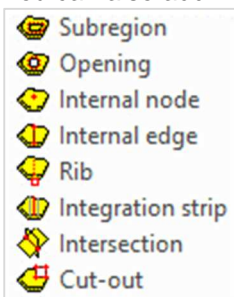


Next to the settings above you also have to set some other parameters:



- **Type:** select the type.
- **Material:** select the material.
- **FEM model:** select the FEM model: 'isotropic' or 'orthotropic'. An orthotropic plate has different properties in perpendicular direction.
- **Thickness:** set the thickness of the plate. By default, you create a plate with a constant thickness. You can change it afterwards to a variable thickness in the properties window.
- **Member system-plane at:** set the member system plane: top, bottom or middle.
- **Eccentricity:** add an eccentricity to the plate.
- **LCS angle:** change the angle of the local axis of the plate.
- **Layer:** set the layer.

You can also add '2D Member components' to a plate:



Subregion: create a subregion. This is an area of the 2D element that has another thickness and/or material quality than the rest of the 2D element.

Opening: create an opening in the 2D element.

Internal node: create a node on the 2D element.

Internal edge: create an edge on the 2D element. 1D elements parallel with the plate are only connected to the plate when there is an edge.

Rib: create a rib on the 2D element. A rib is automatically connected to the 2D element.

Integration strip: an integration strip makes it possible to view the result of a part of the 2D element as if it is a 1D member.

Intersection: create an intersection between two 2D elements. This will connect the 2 elements with each other.

NOTE: the function 'Internal node' is only applicable to insert a node within the polyline of the element. However, this function does not work when a node needs to be created on the edge of a 2D member. In order to insert a node on an edge of a 2D member, you can use the option 'Edit polyline – insert node' which can be found in the tab 'Modify' → 'Polyline edit'.

2.9. Load panels

A load panel is a 2D element that transfers the applied load to its edges or nodes. The load panel adds no stiffness to the model.

You can find Load panels in the 'Structure' menu. There are four different types of load panels:



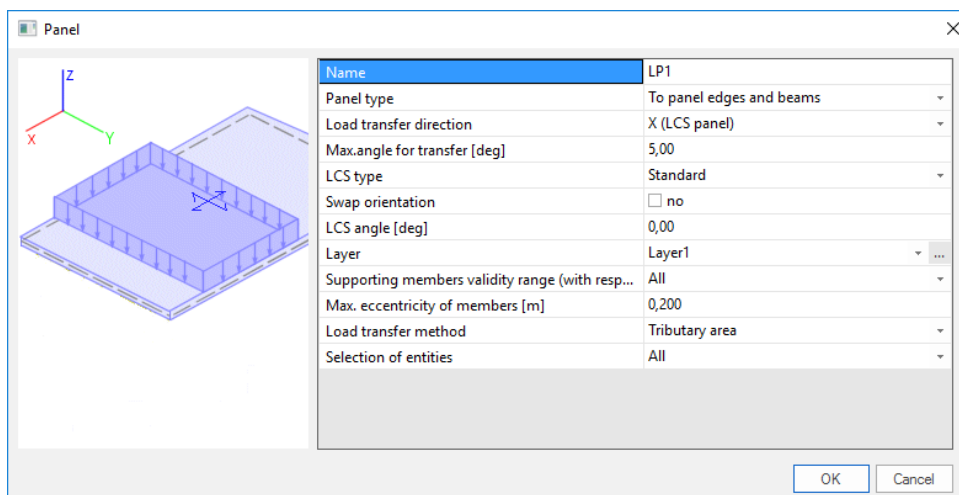
Load to panel nodes: the panel transfers the load to the nodes of the panel.

Load to panel edges: the panel transfers the load to the edges of the panel.

Load to panel edges and beams: the panel transfers the load to the edges of the panel and the beams in the plane of the panel.

Panel with parallel beams: same as the type above, however the beams will automatically be generated by using this function.

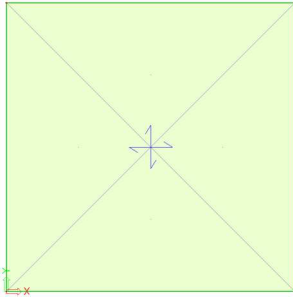
After you have chosen the type of panel, you need to set the properties. It is not necessary to set all the properties correctly before modelling the panel. You can modify the properties afterwards as well.



- **Load transfer direction:** you can choose the direction in which the loads are transferred. This can be direction X, Y or both. The X and Y direction follow the local axes of the panel.
- **Max. angle for transfer:** maximum allowed angle difference of an edge or beam, perpendicularly to the load transfer direction, in order to transfer the internal forces.
- **LCS angle:** rotate the LCS.
- **Max. eccentricity of members:** maximum allowed eccentricity of the members. If the eccentricity is higher than the set limit, the load will not be transferred.

- **Load transfer method:** there are four load transfer methods.

1) Tributary area: this method divides the surface in such a way that every separate part of the surface is 'carried' by the beam lying on its edge. This means that the load on this part of the surface is completely transferred to that beam. This method does not work for complex load panels. The advantage of this method is the quick calculation of the loads.



2) Standard: this method uses weight factors to divide the load between the beams. You can set the weight factors in the properties window.

Weight of loaded ed...	
LP1/B1	1
LP1/B2	1
LP1/B3	1
LP1/B4	1

3) Accurate(FEM), fixed link with beam: this method generates a mesh on the panel and uses a FEM calculation to determine the load distribution. For this option the beams have a fixed connection to the panel.

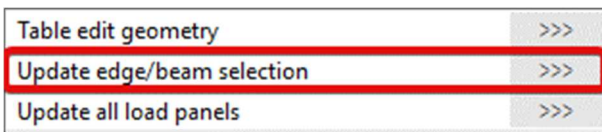
4) Accurate(FEM), hinged link with beam: this method generates a mesh on the panel and uses a FEM calculation to determine the load distribution. For this option the beams have a hinged connection to the panel.

NOTE: the FEM method allows to determine a load distribution on underlying elements based on the FEM response of an auxiliary plate with finite stiffness. The plate response is determined prior to the analysis of the 3D structure (upon pressing the 'Generate loads' action button).

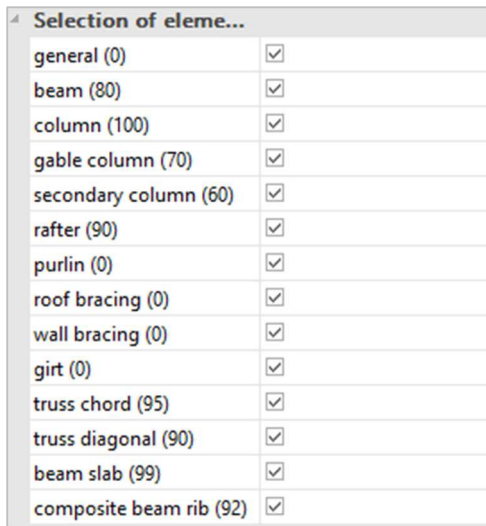
The boundary conditions of this auxiliary plate affect the load transfer from the panel to adjacent elements and nodes. You can set the connection of the panels to the underlying beams at the panel edges as rigid or as hinges. You may choose between two options: 'fixed link with beams' or 'hinged link with beams', which results in different final load distribution. The resulting reactions of the background calculation of auxiliary plates are converted to the loads generated on the actual structure. In the case of 'rigid link with beams', moment reactions are ignored, as these do not originate in the structural response.

- **Selection of entities:** the entities/beams on which the load is transferred, are automatically selected. You can overwrite the automatic selection by changing this option to 'user selection' or 'type'.

When you select 'user selection' you need to press the action button 'Update edge/beam selection' and select/deselect the members.

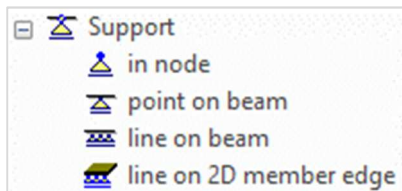


When you select 'type' you need to check/uncheck the types taken into account.



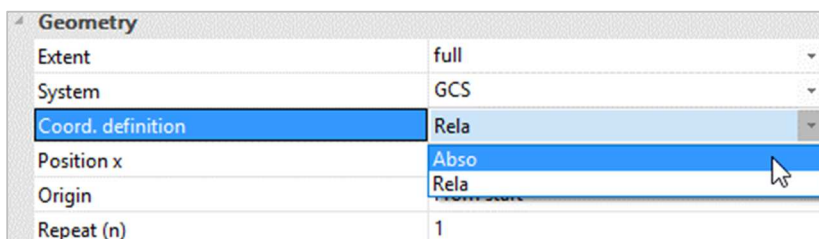
2.10. Supports

You can find supports in the 'Structure' menu under 'Model data'.

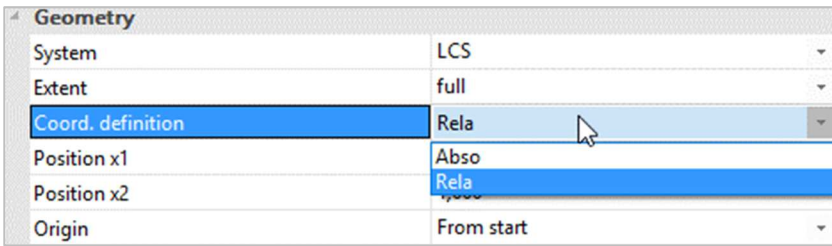


In node: you can only assign this type of support to a node.

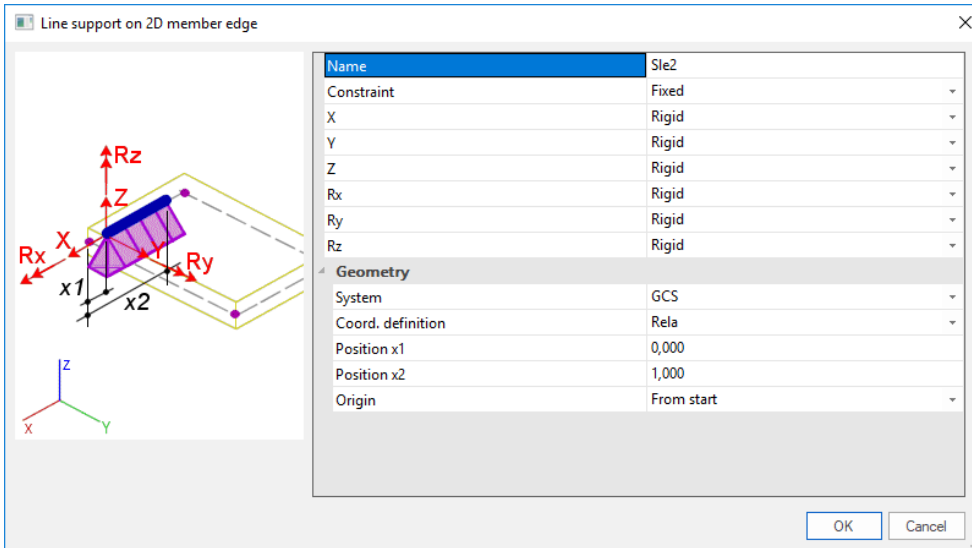
Point on beam: you can put this point support along the length of a member. You need to define the relative or absolute position of this support along the length of the member.



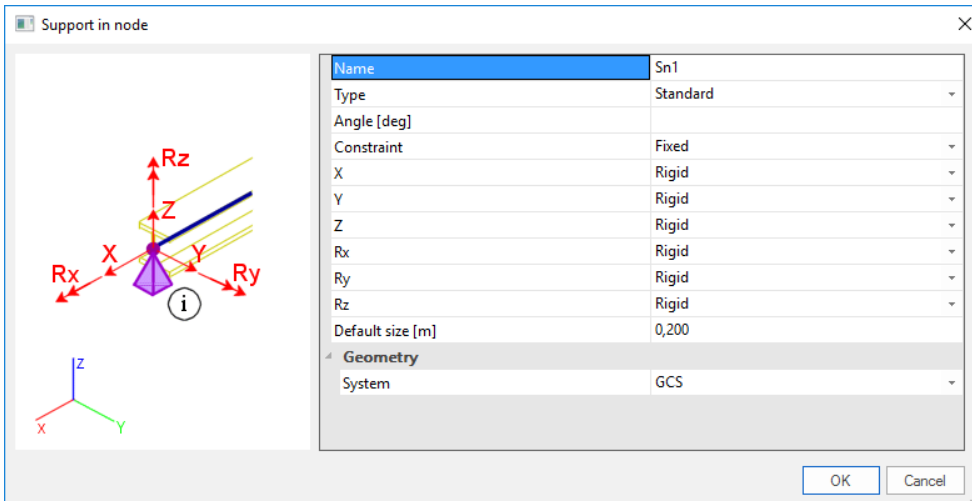
Line on beam: you can put this line support along the length of a member. You need to set the relative or absolute position and the length of this support along the length of the member.



Line on 2D member edge: this is a line support which you can define on a 2D member edge. You need to set the relative or absolute position and the length of this support along the length of the edge.



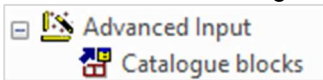
Next to the supports position you need to define the constraints (boundary conditions) as well.



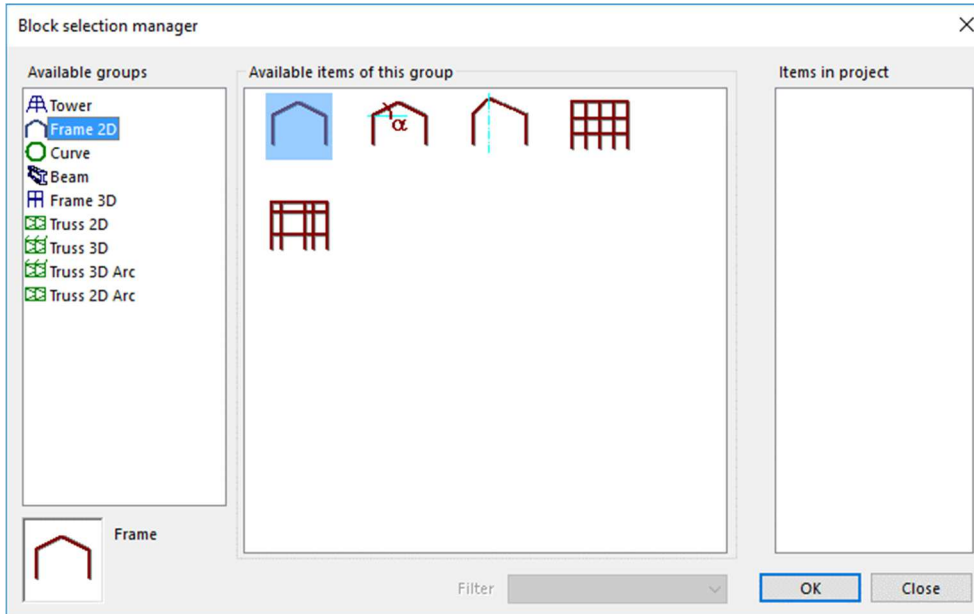
- **Angle:** at default the support follows the local axes of the node. You can give it an angle relative to the LCS by using Rx(angle),Ry(angle),Rz(angle). For example: Rx90,Ry90,Rz90.
- **Constraint:** you can choose a default constraint: 'fixed', 'hinged' or 'sliding'.
- **X, Y, Z, Rx, Ry, Rz:** you can change these values to set a non-default constraint.
- **Default size:** this option is only used for the moment reduction above a support of a continuous beam (in the 'Concrete' menu).

2.11. Catalogue blocks

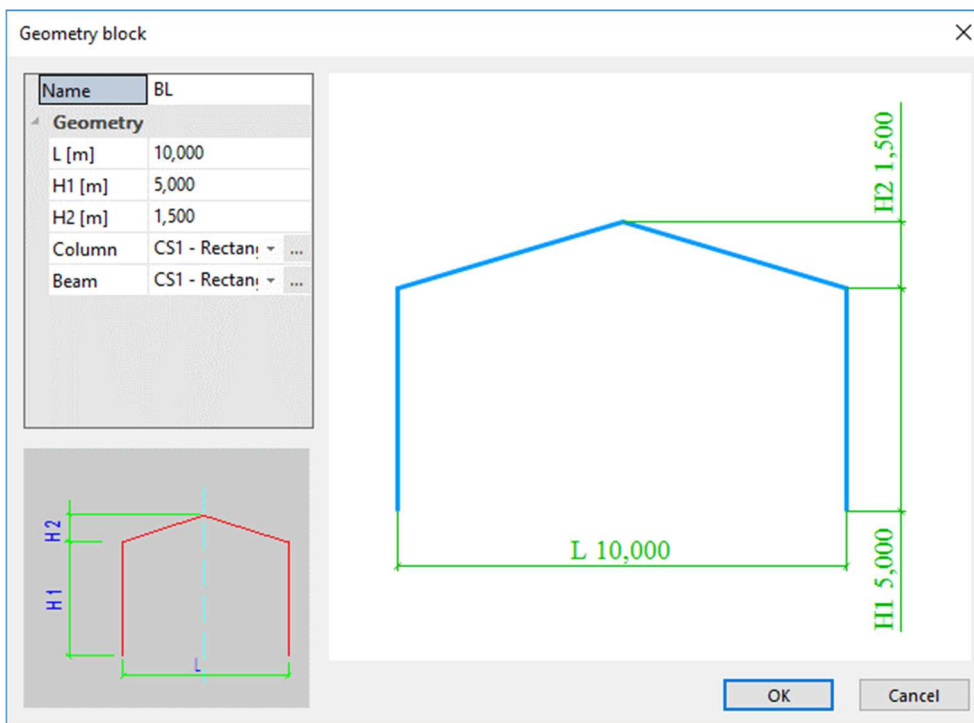
You can find the catalogue blocks in the 'Structure' menu under the option 'Advanced input':



With catalogue blocks you can import frames and trusses in an easy and fast way.



Select the predefined block you want to add to your model and set the parameters.



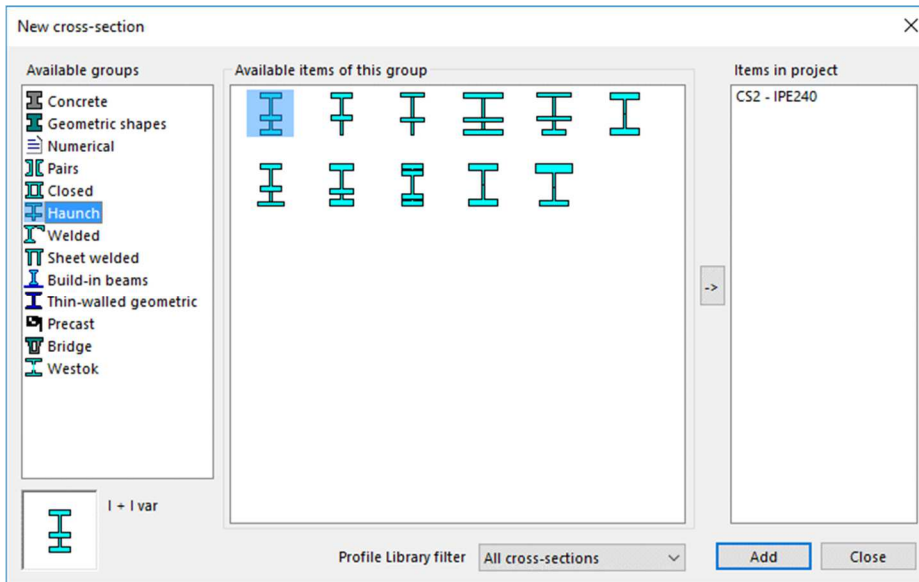
After pressing 'OK' you can add the catalogue block to your model. The catalogue blocks consist of 1D members that are fully connected to each other.

2.12. Haunch

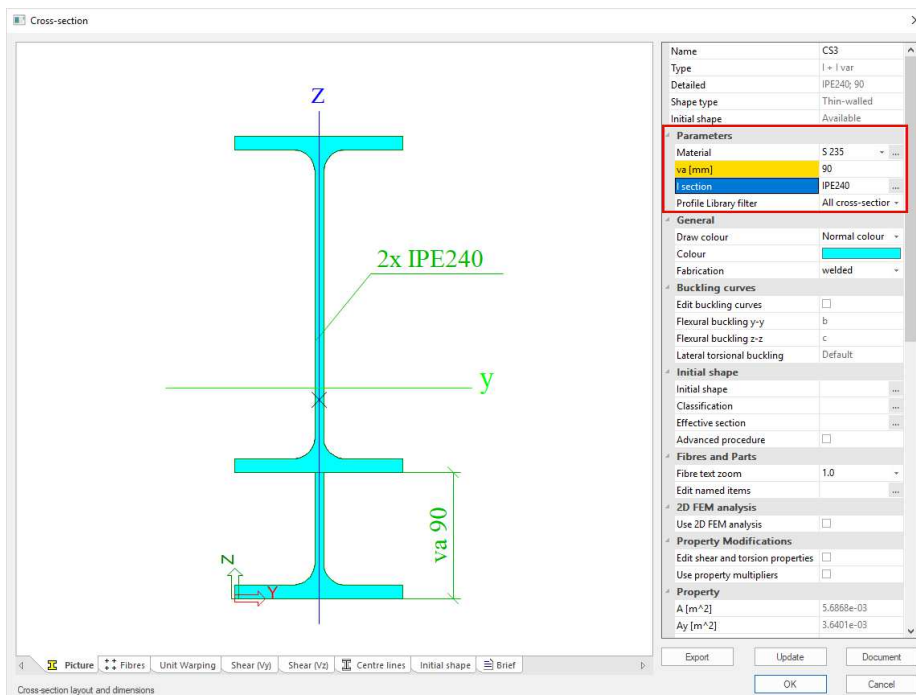
In the 'Structure' menu under '1D Members' → '1D Member components' you can find the function 'Haunch'.

Before you can define a haunch, you need to add a cross-section of the type 'Haunch'. This is the cross-section at the beginning of the haunch, more specifically the largest cross-section. You can add the cross-section in the 'Cross-section' menu OR you can click on the 'Haunch' function to automatically open this window.

First select the type of haunch and press 'Add':

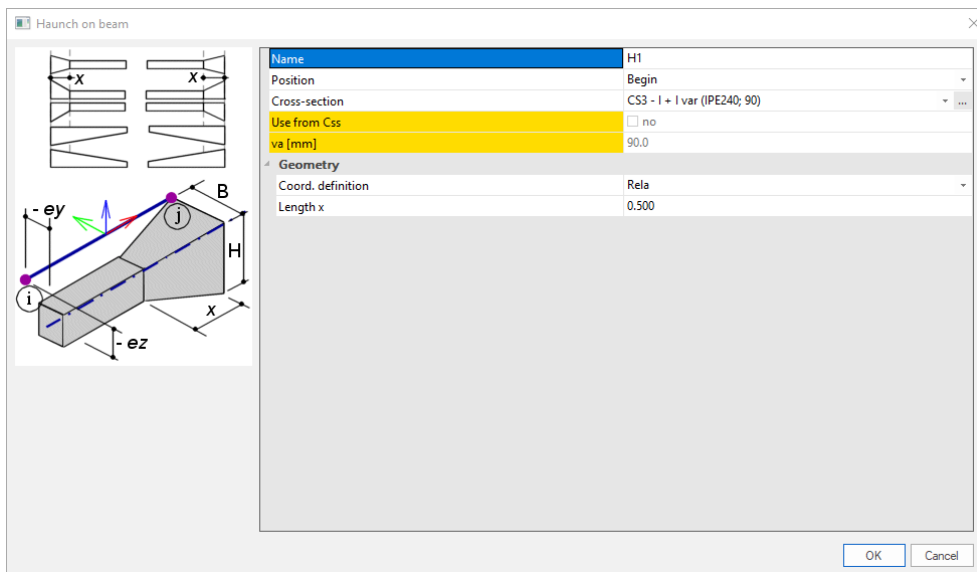


Then you need to fill in the parameters of the cross-section.

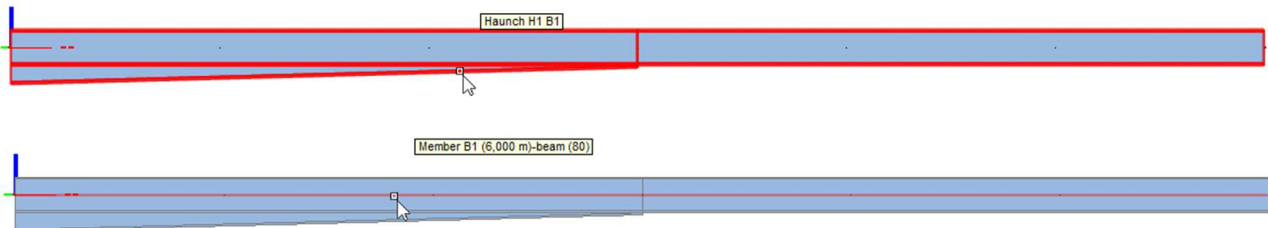


Confirm with 'OK' → 'Close' → 'OK'

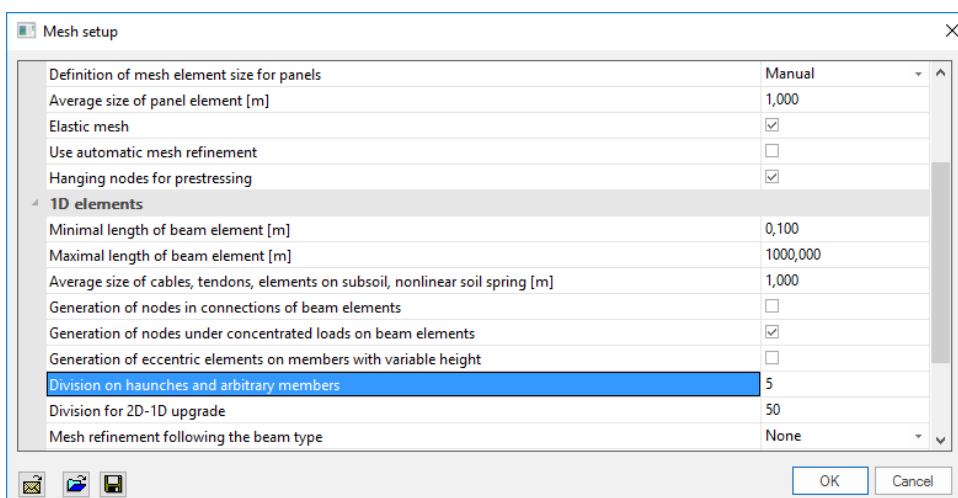
In the last step you define the position (in the beginning or at the end of the beam) and the length (relative or absolute) of the haunch.



A haunch is model data you put on a member. This means if you select the surface of the member, you are not selecting the member itself but only the additional data 'Haunch'. In order to select the member, you need to select the centreline.



A haunch is calculated by the solver in 5 segments with each segment having its own, constant cross-section. You can increase this number of segments in 'Setup' → 'Mesh'

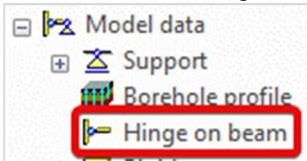


NOTE: if you want to have a different haunch at the beginning and at end of a beam you need to cut the beam in half.

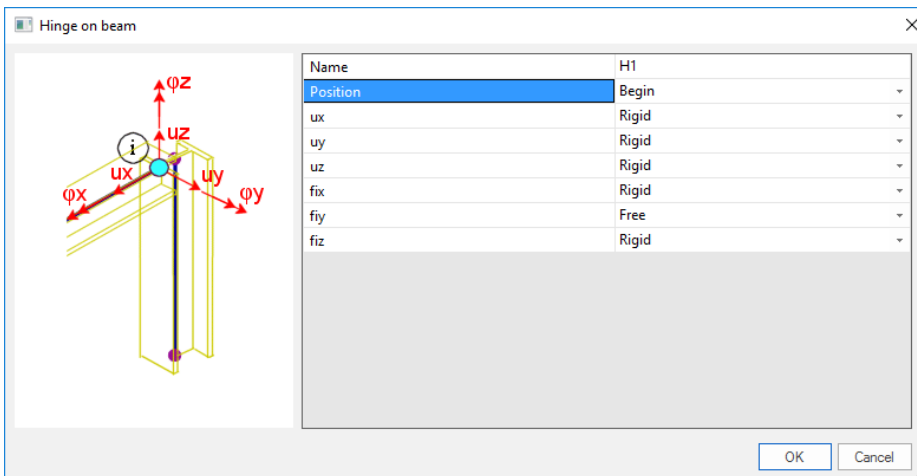
2.13. Hinges

By default, every node in your project is considered fixed, unless your project is modelled in a truss environment. In order to make a member hinged, you need to add a hinge to the member. A hinge is considered as additional data you add on a member.

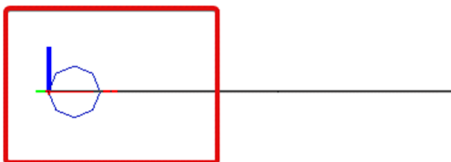
You can find the hinges in the 'Structure' menu under 'Model data':



To add a hinge you need to define the degrees of freedom and the position (begin, end or both sides of the beam).



The hinge is visualised in SCIA as in following figure:

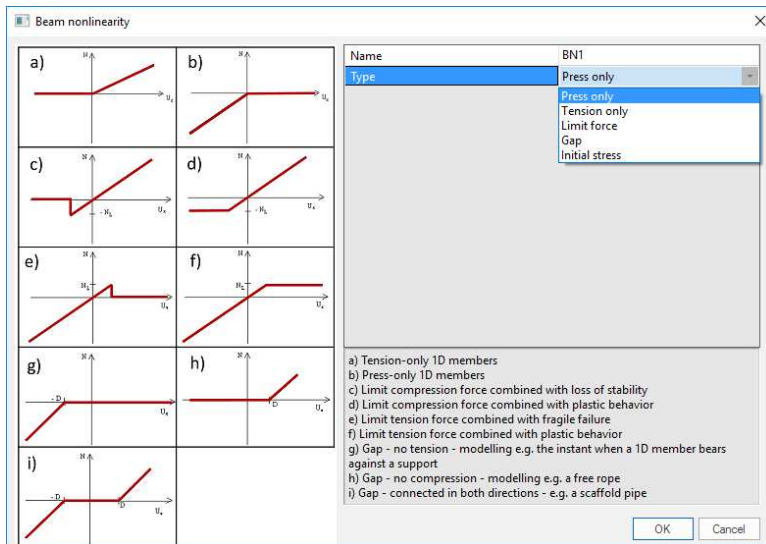


2.14. Beam non-linearity

It is possible to add non-linear behaviour to a beam. To enable these option, you need to tick following functionalities in the 'Project':

- nonlinearity;
- beam local nonlinearity.

You can find the option 'Beam – nonlinearity' in the 'Structure' menu under 'Model data'. You need to choose the nonlinearity type in the following window.

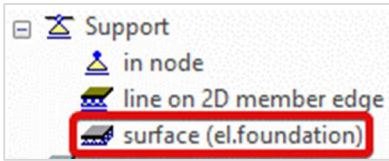


You need to perform a nonlinear calculation to consider the nonlinear behaviour. Therefore, you should create nonlinear combination(s). Chapter 3 explains the creation of loads and combinations.

NOTE: the most used nonlinearity type is 'Tension only'. It is used to model wind bracings. Best practice is to combine the 'Tension only' type with the 1D member property 'axial force only'. See the chapter about 1D members to check how this is enabled.

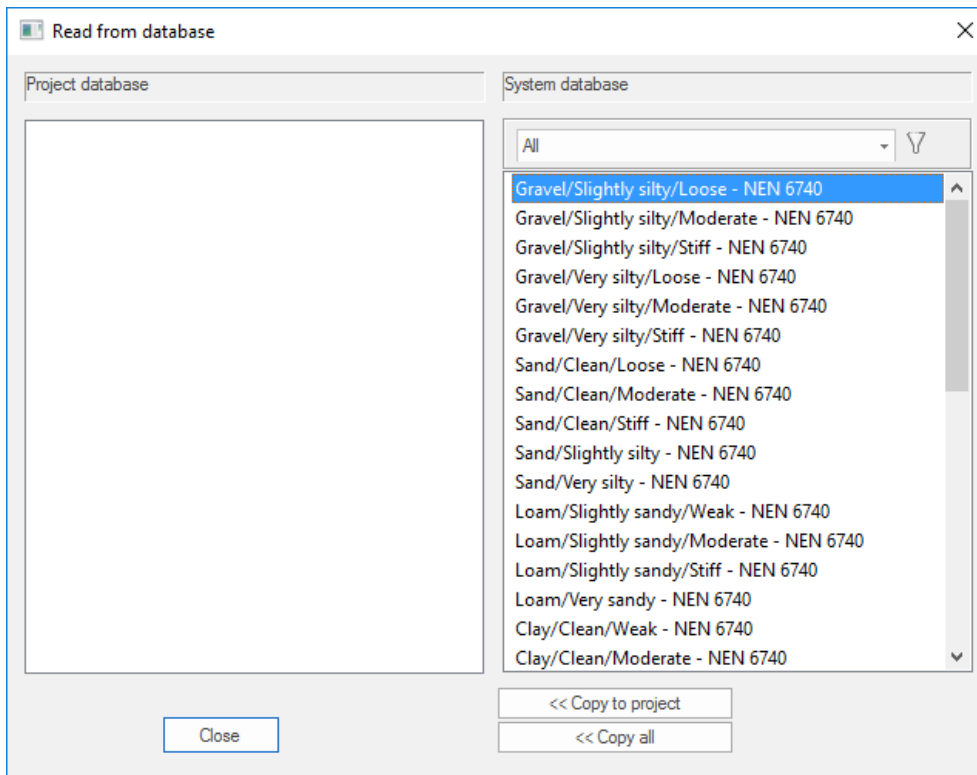
2.15. Subsoil

If there is a 2D element in your model, you can find the subsoil function in the 'Structure' menu under 'Model data' → 'Support':

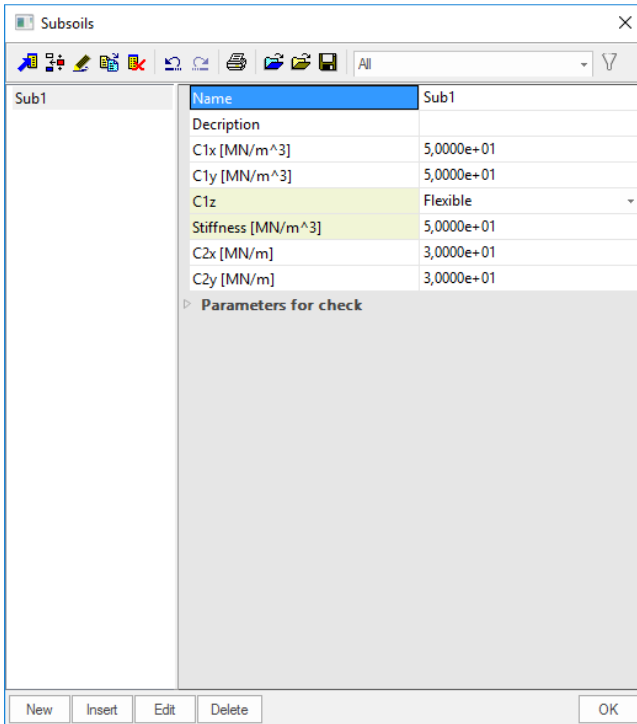


You can use the option 'surface (el.foundation)' to add a subsoil to a 2D element.

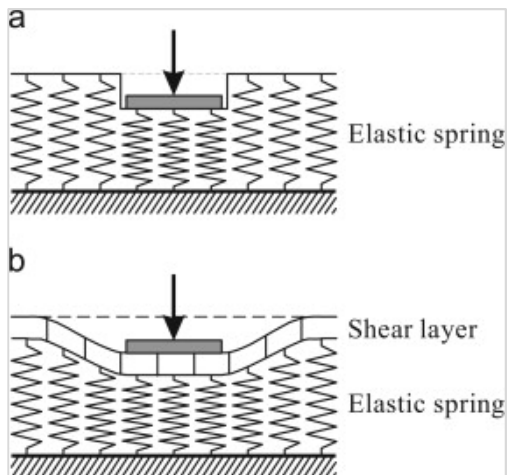
In the first window you can choose a predefined subsoil (according to NEN 6740). If you want to define your own subsoil press 'Close'.



Now you should set the following parameters before adding the subsoil to your model:



- **C1z**: stiffness of the ground in Z direction.
- **C1x, C1y**: stiffness of the ground in horizontal direction. If these values are not known 10% of the stiffness in Z direction can be taken as an approach.
- **C2x, C2y**: these values couple the deformation in Z direction between different sections of the ground. The image below shows the theory of Winkler (picture a) where these parameters are 0. Picture b illustrates the theory of Pasternak where the C2 parameters have a certain value. Usually these values are not determined, and the values are set to 0.



2.16. Modify shape

There are three ways you can modify the shape of elements.

1) With properties window

You can change the shape of an element by editing the coordinates of the nodes of the element. If a node is in the current selection, you can change the coordinates in the properties window.

Name	N2
GCS coordinate	
Coord X [m]	6,000
Coord Y [m]	0,000
Coord Z [m]	0,000
UCS coordinate	
Coord ux [m]	6,000
Coord uy [m]	0,000
Coord uz [m]	0,000
Members	
Member	B1

2) With the action 'Table edit geometry'

You can edit the shape of an element by selecting it and clicking on the action button 'Table edit geometry' on the bottom of the properties window.

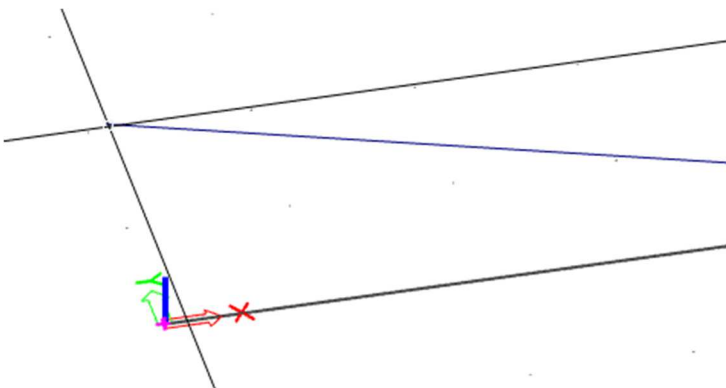


In this table you can edit the coordinates of the nodes of the element.

	Node name	X [m]	Y [m]	Z [m]	ux [m]	uy [m]	uz [m]	Linked	Shape
1	N1	0,000	0,000	0,000	0,000	0,000	0,000	<input type="checkbox"/> Rela	Line
2	N2	6,000	0,000	0,000	6,000	0,000	0,000	<input type="checkbox"/> Rela	

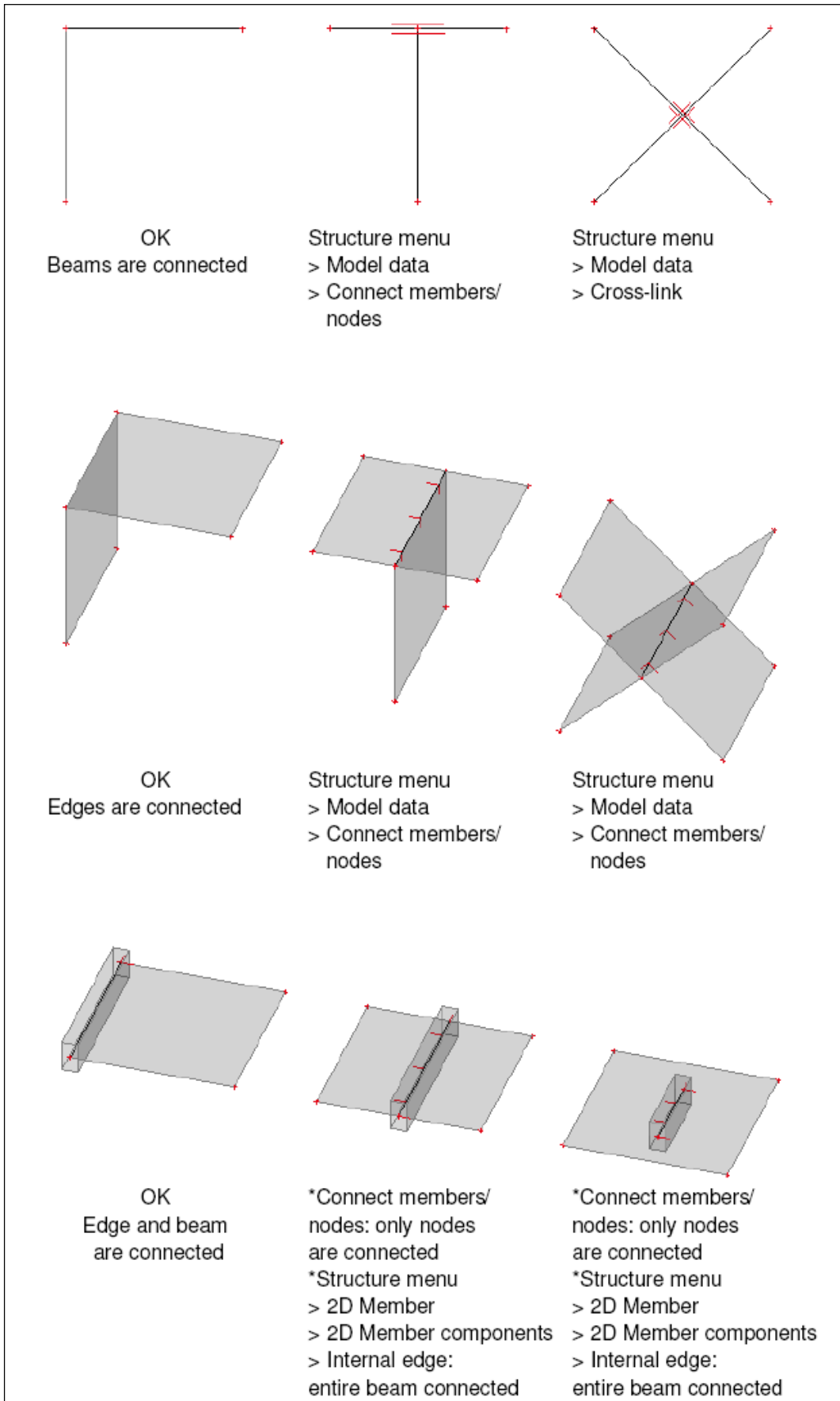
3) By clicking and dragging

When you have selected a node, an element or several nodes, you can these by holding the left mouse button on top of it, clicking and dragging the mouse towards the new location.



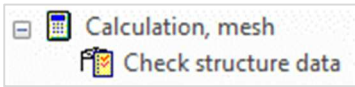
2.17. Connect members

It is important to know that not all members are connected automatically. The scheme below provides a graphical representation of all the members which you need to connect by yourself.

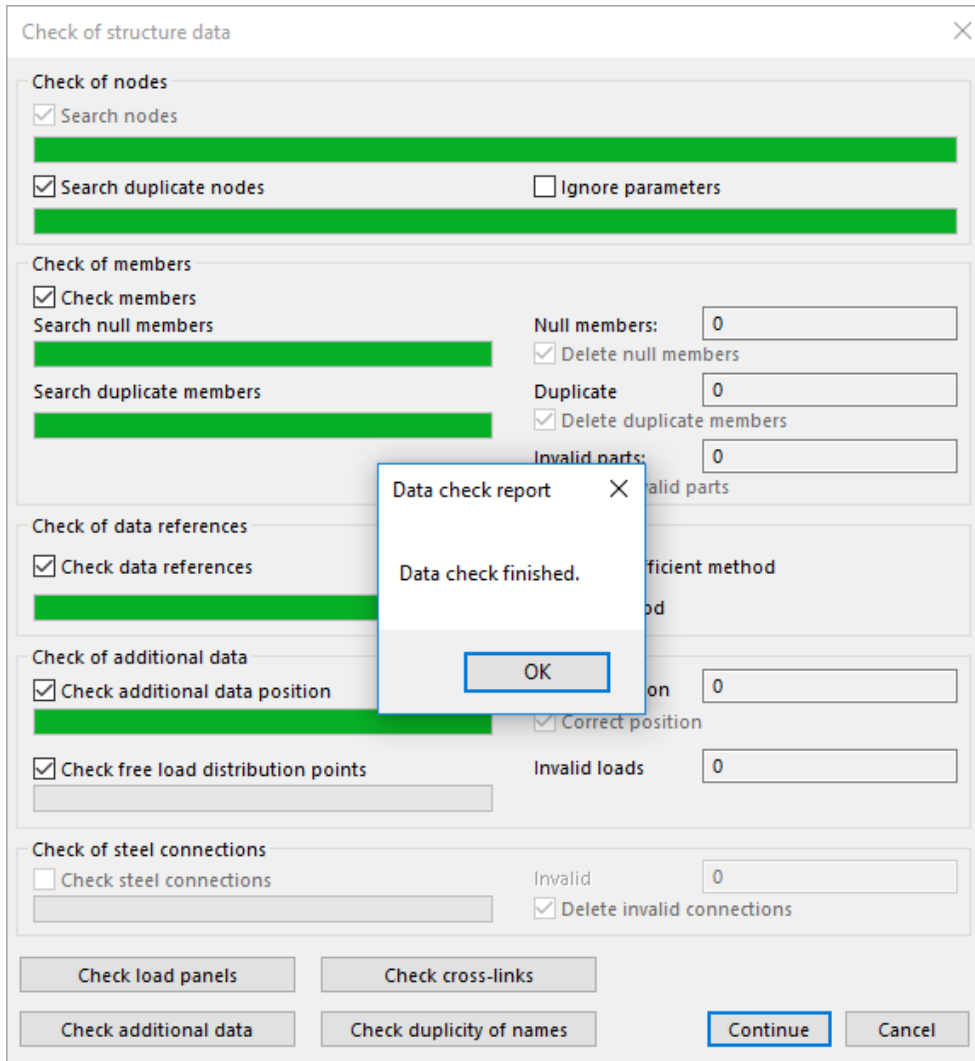


2.18. Check structure data

As soon as you have completed your model and before you run the calculation, the 'Check structure data' function should be executed. This function checks if there are errors in the project, more specifically duplicate elements, corrupt data, ...

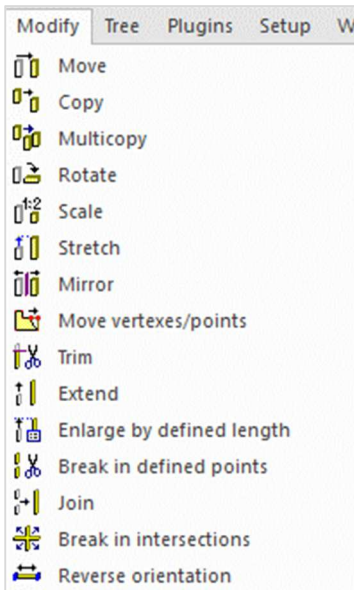


If there are no issues in the model the following window will appear.



2.19. Modification commands

In this chapter you can find some explanation about modification commands. The steps you need to follow to successfully complete a modification command are shown in the command window. You can find the modification commands in the toolbars or under 'Modify'.



2.19.1. Copy

Press the 'Copy' command or use the shortcut CTRL+C.



Below you can see the actions you should take to perform the copy command. You can find these steps in the command line.

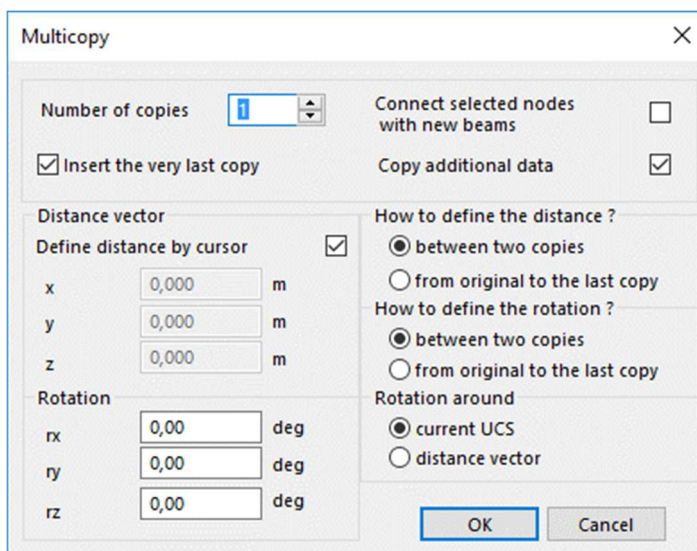
Copy - Select entities to be copied (finish selection with ESC) >

Copy - Start point >

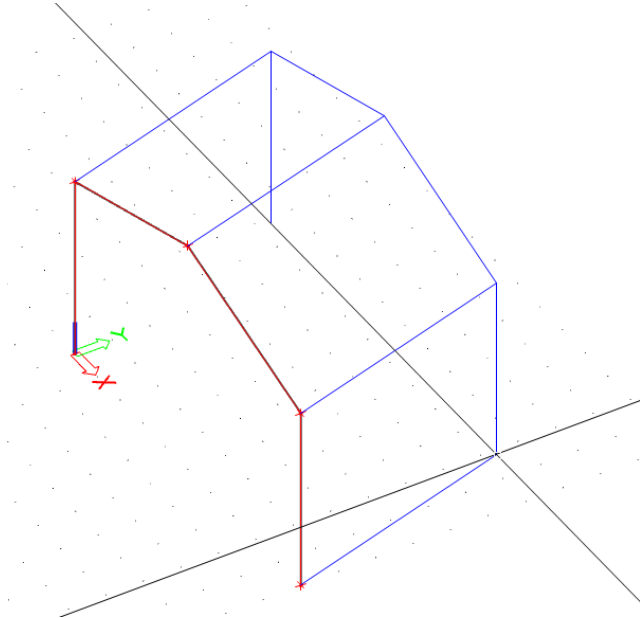
Copy - End point >

2.19.2. Multiple copy

In order to perform a more advanced copy, you can select the 'Multicopy' command.



- **Number of copies:** set the number of copies.
- **Insert the very last copy:** this option defines the number you need to fill into the 'Number of copies' field. If this option is checked and the 'Number of copies' is 7, 7 copies will be made. If the option is unchecked and the 'Number of copies' is 7, 6 copies will be made. The existing selection counts as the first copy.
- **Connect selected nodes with new beams:** if this is checked, all the nodes in the current selection will be connected with beams to the same nodes of the next copy.



- **Copy additional data:** when this is checked, all additional data of the selection (supports, haunches, ...) will be copied as well.
- **Distance vector:** setting values for this vector will define the distance/rotation between copies.

2.19.3. Mirror

Use the 'Mirror' command.



Below you can see the actions you should take to perform the 'Mirror' command. You can find these steps in the command line.

```
Mirror - Select entities to be mirrored (finish selection with ESC) >
Mirror - Mirroring plane - Start point >
Mirror - Mirroring plane - End point >
```

2.19.4. Break in defined points

Use the 'Break in defined points' command. With this function you can divide a member into multiple segments.

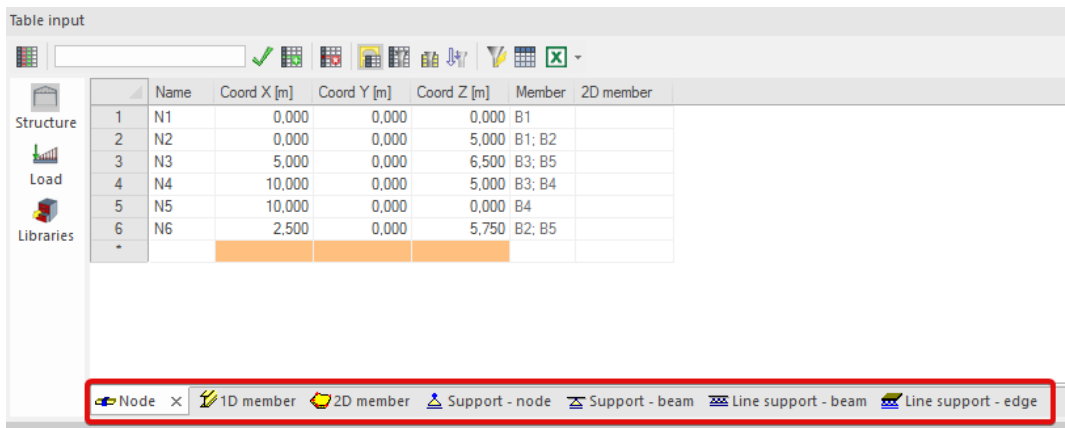


Below you can see the actions you should take to perform the 'Break in defined points' command. You can find these steps in the command line.

```
Divide - Select curves to be divided (finish selection with ESC) >
Divide - Insert division points >
```

2.20. Table input

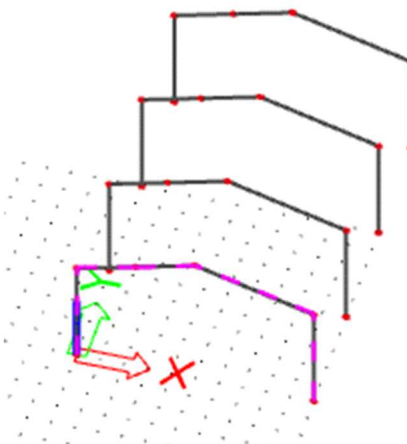
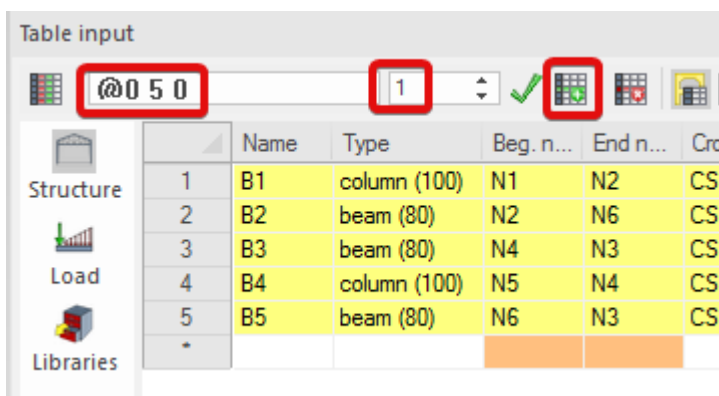
In SCIA Engineer elements are often drawn, but you add them as well via 'Table input'. The 'Table input' window has a tab per type of element (nodes, 1D elements, 2D elements, ...). The tabs are highlighted in the picture below. You need to fill in the orange boxes to add an element. If a tab is missing, you can add it by right-clicking on another tab and choosing the missing tab.



You can copy the content of the table to Excel and vice versa. In order to do this, you can use the short cut CTRL+C. There is also a function to export the table input as an Excel file:



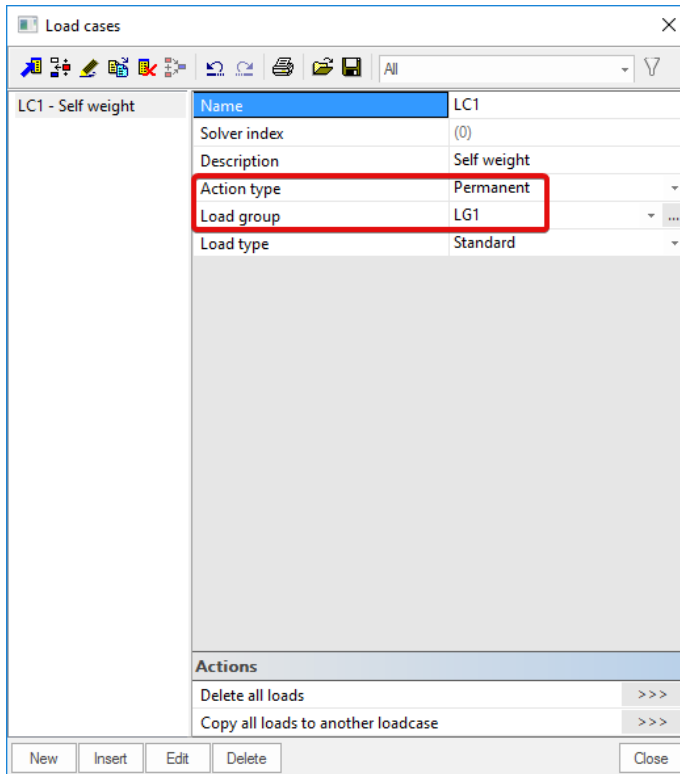
You can copy rows in the table input to perform an action similar to the 'Multicopy' function. To do this you need to select (the rows of) the elements, fill in the distance between the copies and the amount of copies and then press the 'copy row' button.



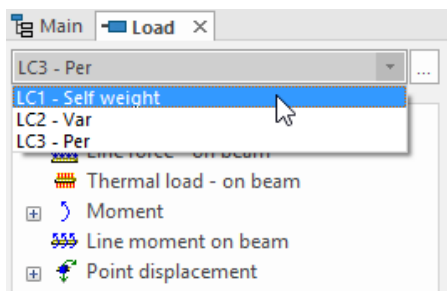
Chapter 3: Loads

3.1. Load cases

For every load you can create a load case. In the settings of the load case you determine if it is a permanent or variable load and to what load group the load case is assigned. These two settings are important to generate the correct combinations. You can open the window below in the main tree → 'Load cases, combinations' → 'Load cases'.



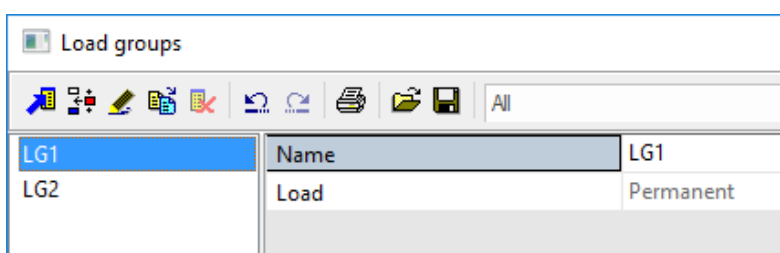
In the 'Loads' menu you can switch between the different defined load cases to draw/define the loads.



3.2. Load groups

With load groups you define the type of load and the relation between the loads in this group. Depending on the type of load case (permanent or variable) you can define the matching load group.

3.2.1. Permanent load group

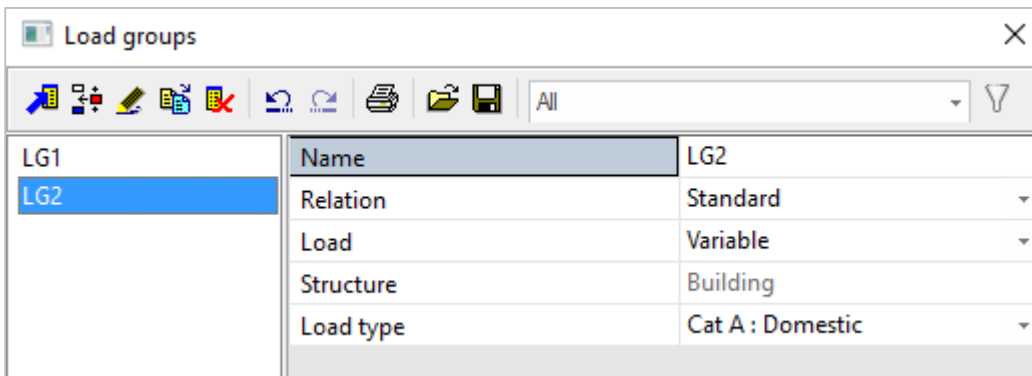


There is only one permanent load group per project, because all the permanent loads should be summed up and they do not have different categories.

$$\left\{ \sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} \psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \right. \quad (6.10a)$$

$$\left\{ \sum_{j \geq 1} \xi_j \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \right. \quad (6.10b)$$

3.2.2. Variable load group



Name	LG2
Relation	Standard
Load	Variable
Structure	Building
Load type	Cat A : Domestic

Compared to the permanent load group you can define multiple variable load groups. For every type of load (wind, snow, ...) you can create a different load group. This is because in 6.10a(b) the different type of loads should vary between the main variable group and the sub variable group. The different types of loads are defined by the option 'Load type'. With the load type you can set the category of the load. This way the program knows what psi factors to use (according to Eurocode 0).

$$\left\{ \sum_{j \geq 1} \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} \psi_{0,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \right. \quad (6.10a)$$

$$\left\{ \sum_{j \geq 1} \xi_j \gamma_{G,j} G_{k,j} + \gamma_P P + \gamma_{Q,1} Q_{k,1} + \sum_{i > 1} \gamma_{Q,i} \psi_{0,i} Q_{k,i} \right. \quad (6.10b)$$

You should also define the relation between the loads in the same load group.

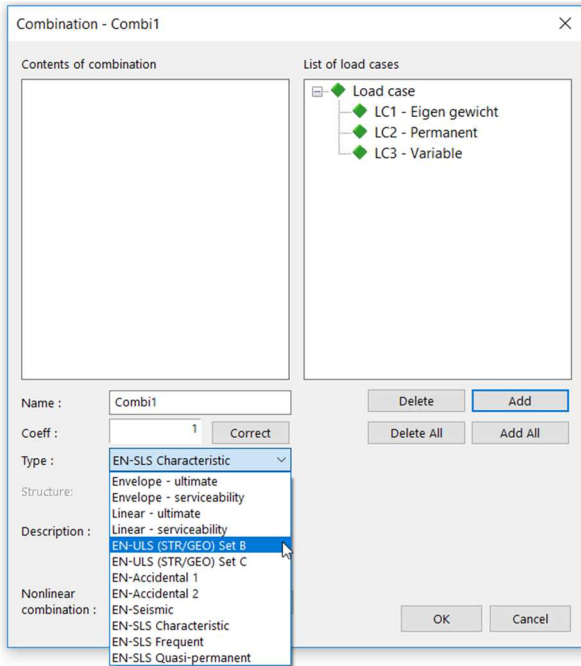
- **standard:** all combinations are possible;
- **exclusive:** only one load of every load group can be present in a combination;
- **together:** the loads can only be present together in a combination.

EXAMPLE: assume the load cases LC A and LC B which are both assigned to the same load group.

- **standard:** LC A and/or LC B
- **Exclusive:** LC A or LC B
- **together:** LC A and LC B

3.3. Combinations

You can choose between three types of combinations. The differences will be explained below. To add a combination, go to main tree → ‘Load cases, combinations’ → ‘Combinations’.

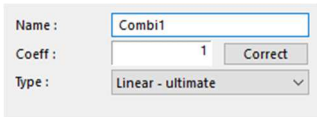


This window contains two lists with load cases. The left list contains the content of the combination you are making, the right list contains all available load cases. You can add a load case by double-clicking in the right list of with the buttons ‘Add’ and ‘Add All’. After setting the load cases you should set the type of combination.

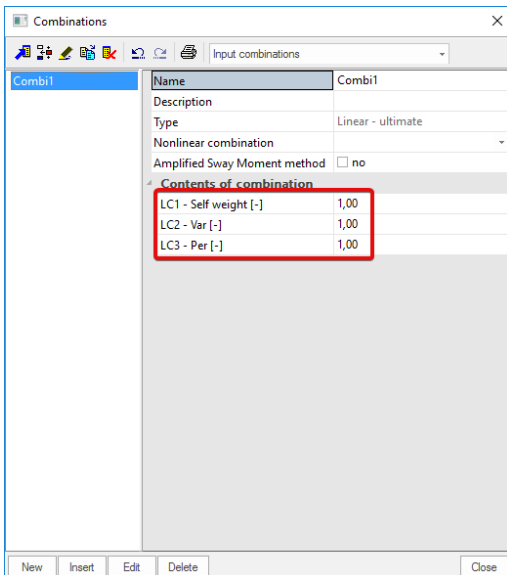
3.3.1. Linear combination

A linear combination is a combination completely defined by yourself. It is a single combination with the selected load cases in it and with coefficients that you set.

Choose the linear combination type (ultimate or serviceability):



Set the coefficients:



3.3.2. Envelope combination

An envelope combination is a group combination. All possible combinations that can be generated, taking into account the relations of the load groups, are within this one combination. You still need to set the coefficients.

Choose the combination type (ultimate or serviceability):

Name :

Coeff :

Type : ▾

Set the coefficients:

The screenshot shows the 'Combinations' dialog box with the 'Combi1' combination selected. The 'Contents of combination' table is highlighted with a red box, showing the following data:

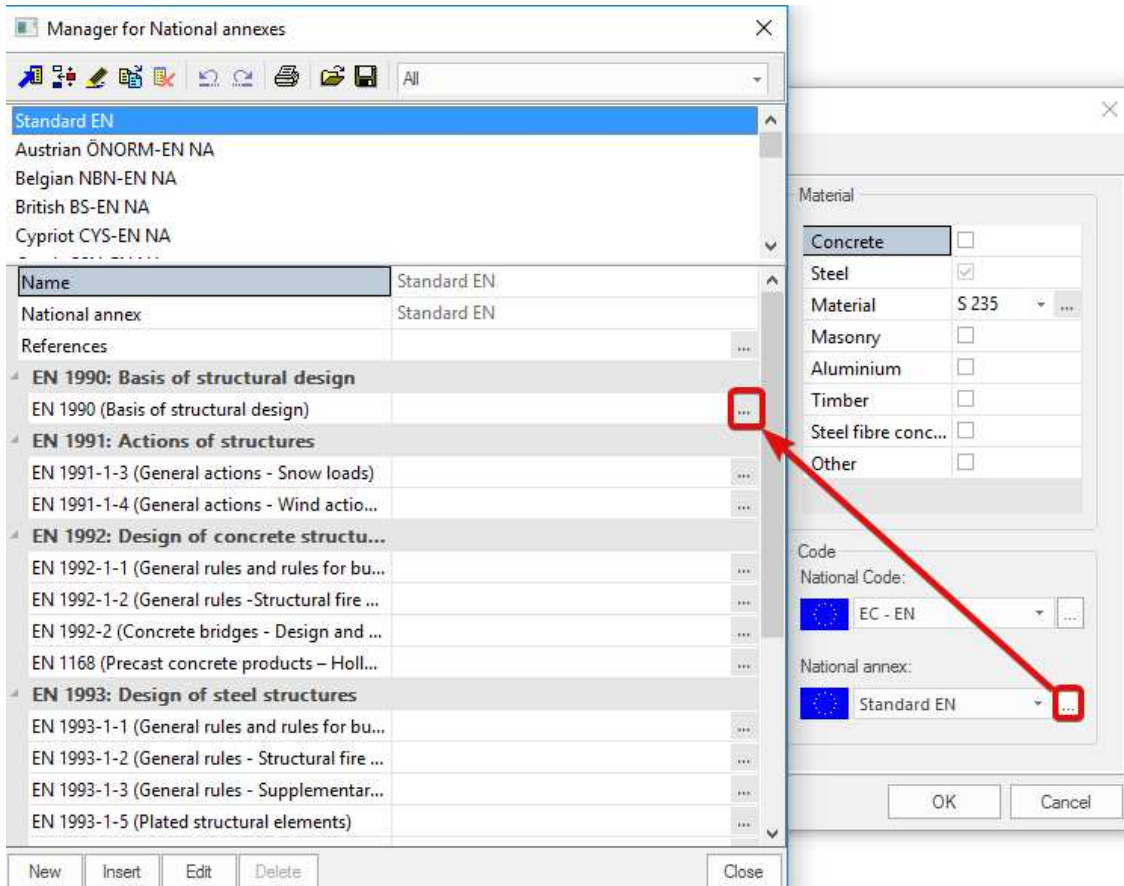
Contents of combination	Coefficient
LC1 - Self weight [-]	1,00
LC2 - Var [-]	1,00
LC3 - Per [-]	1,00

Below the table, the 'Actions' section includes an 'Explode to linear' button with a '>>>' icon. The dialog also features a toolbar at the bottom with 'New', 'Insert', 'Edit', 'Delete', and 'Close' buttons.

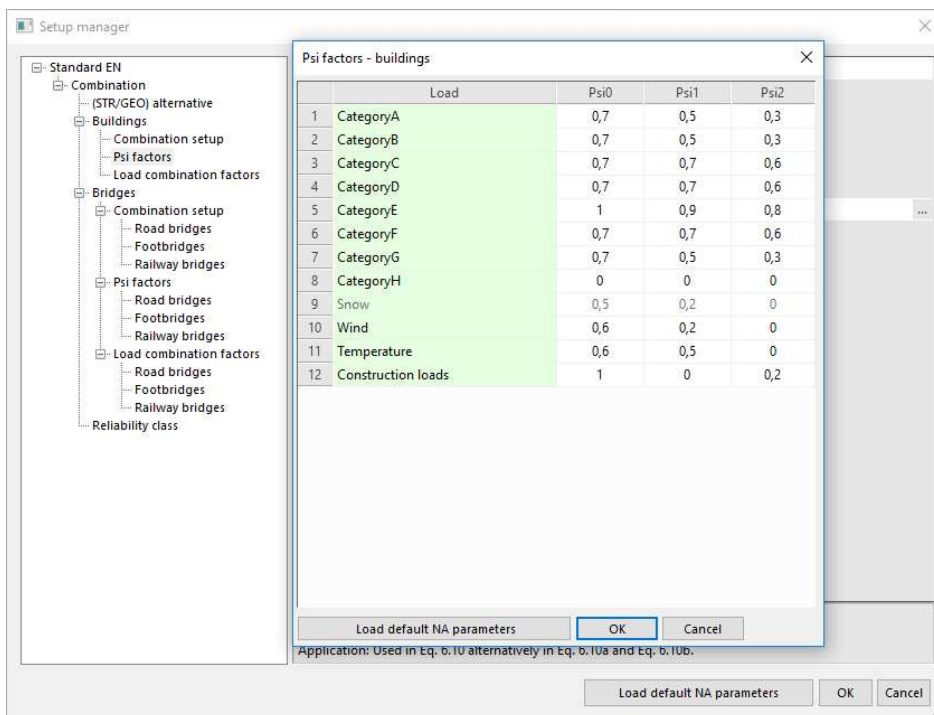
If you want to see the contents of the 'group' combination press the 'Explode to linear' button. This will split this combination in single linear combinations.

3.3.3. Eurocode combination

A Eurocode combination is a group combination. The relations between the loads and the categories are generated by using the data of the load groups. The categories define the psi factors. They can be found by going to 'Project' in the main tree menu. Then you open the library of the national annex by clicking on the 3 dots. Then you choose the relevant national annex (EN 1990).



This opens the code settings of 1990. Here you can find the table with the psi factors. You can edit the values if needed.



To create a Eurocode combination, select the type EN-ULS or EN-SLS:

Name :

Coeff :

Type :

Structure:

Description :

The coefficients cannot be edited. They will be loaded automatically to the load case in the combination.

Combinations

Input combinations

Name	Combi1
Description	
Type	EN-ULS (STR/GEO) Set B
Structure	Building
Nonlinear combination	
Active coefficients	<input type="checkbox"/>
Contents of combination	
LC1 - Self weight [-]	1,00
LC2 - Var [-]	1,00
LC3 - Per [-]	1,00

Actions

Explode to envelopes >>>

Explode to linear >>>

Show Decomposed EN combinations >>>

New Insert Edit Delete Close

If you want to see the content of the 'group' combination press the 'Explode to linear' button. This will split this combination in single linear combinations.

Combinations

Input combinations

Name	Eurocode
Description	
Type	EN-ULS (STR/GEO) Set B
Structure	Building
Nonlinear combination	
Active coefficients	<input type="checkbox"/>
Contents of combination	
LC1 - Self weight [-]	1,00
LC2 - Var [-]	1,00
LC3 - Per [-]	1,00

Actions

Explode to envelopes >>>

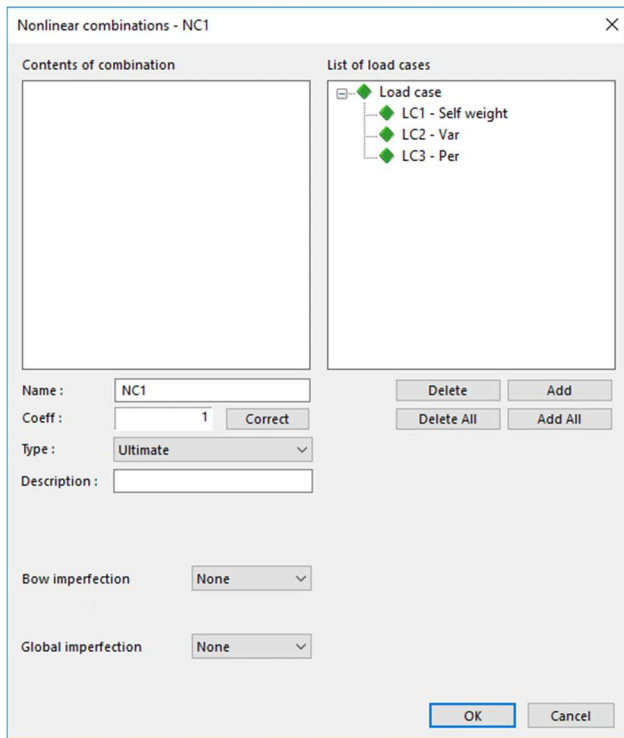
Explode to linear >>>

Show Decomposed EN combinations >>>

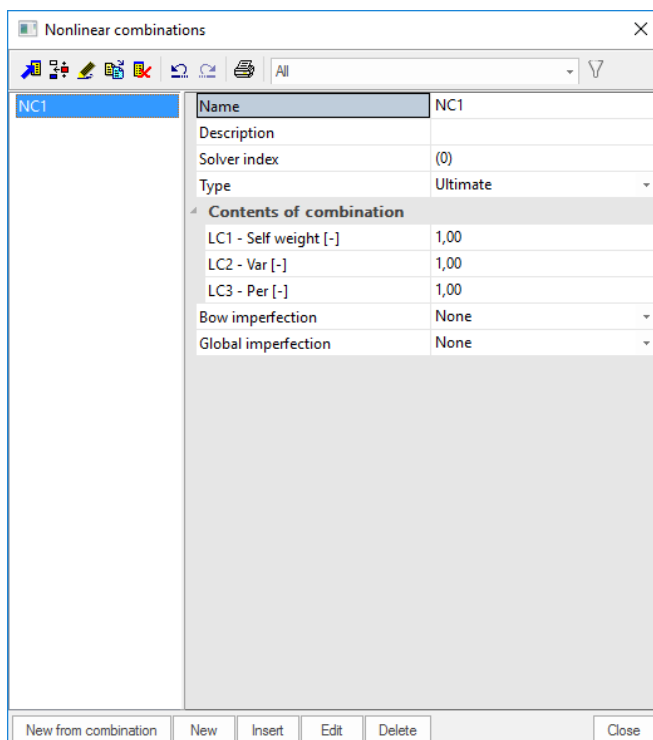
New Insert Edit Delete Close

3.4. Nonlinear combination

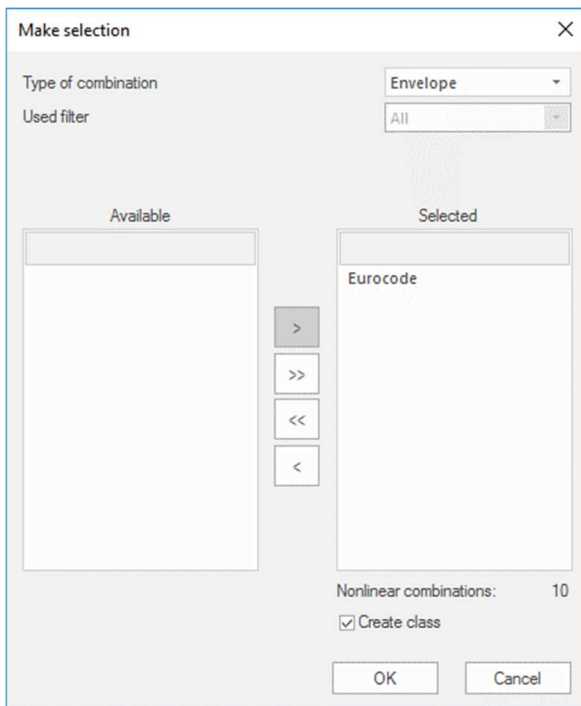
In order to perform a nonlinear calculation, nonlinear combinations should be added. You can add a nonlinear combination by going to the main tree → Load cases, combinations → Nonlinear combination.



As in the 'Combination' window, you need to select the load cases that should be assigned to the nonlinear combination. A nonlinear combination is a single combination, comparable with a combination of the type 'Linear'. You define the type of the nonlinear combination: 'Ultimate' or 'Serviceability'. At last a bow and/or global imperfection can be defined. This is only necessary when you perform a second order calculation.



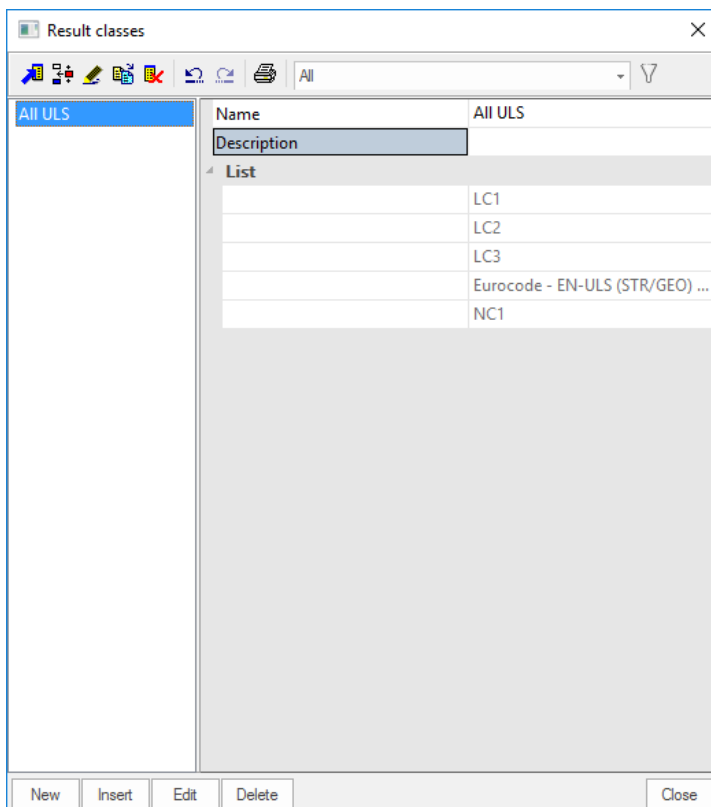
In this window you can set the coefficients and edit the imperfections. There is also a function to automatically create nonlinear combinations from the combinations menu. This can be executed with the button 'New from combination'.



Choose the type of combination (linear or envelope) and make a selection of the combinations you want to turn into nonlinear combinations.

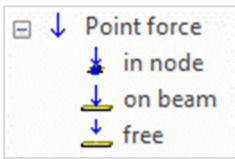
3.5. Result class

In a result class you can put multiple combinations. In the 'Result' menu you will see the envelope of these combinations when asking for the result class. Result classes can be found in the main tree → 'Load cases, combinations' → 'Result classes'.



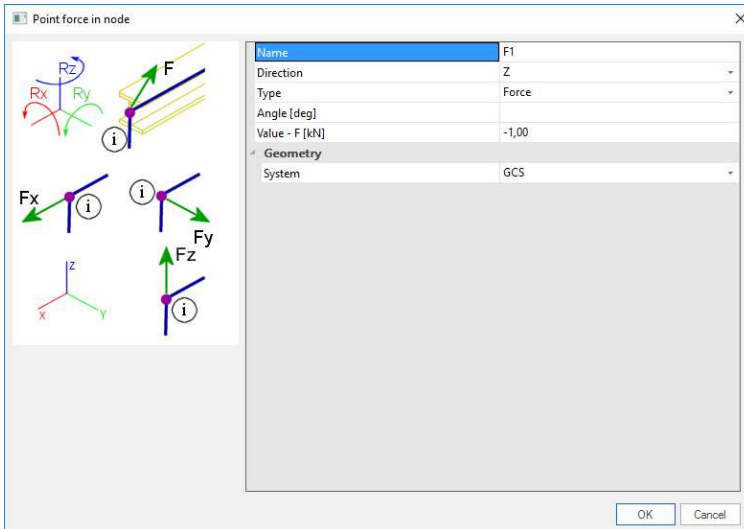
3.6. Point force

You can draw/define loads in the load menu. You can choose from three types of point loads.



3.6.1. Point force in node

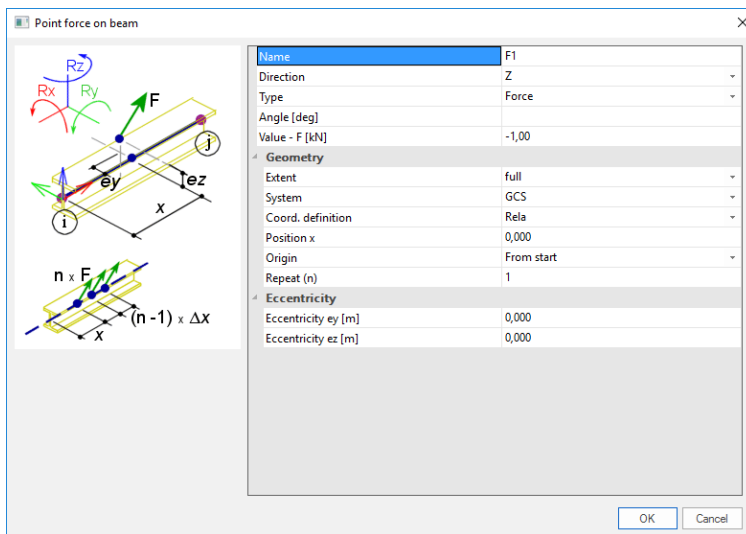
You can only place this point force in nodes. You can set following options.



- **Direction:** defines in what direction the force works.
- **Type:** the standard type is 'Force'. You can change it to 'Wind' or 'Snow'. If you change it to wind or snow, the way the option 'Value' is determined changes.
- **Angle:** the point force can be given an angle. This is done by filling in a value for Rx, Ry, Rz.
- **Angle [deg]**
- **Value:** define the magnitude of the force.
- **System:** defines if the direction is following the global axis (GCS) or the local axis (LCS).

3.6.2. Point force on beam

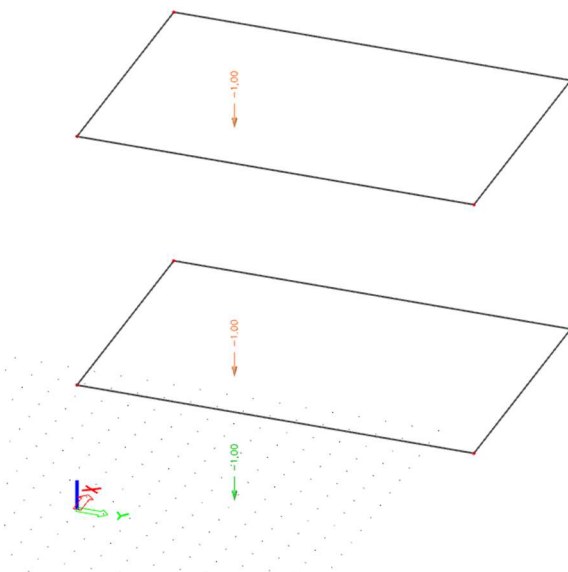
You can place this point force along the length of an element.

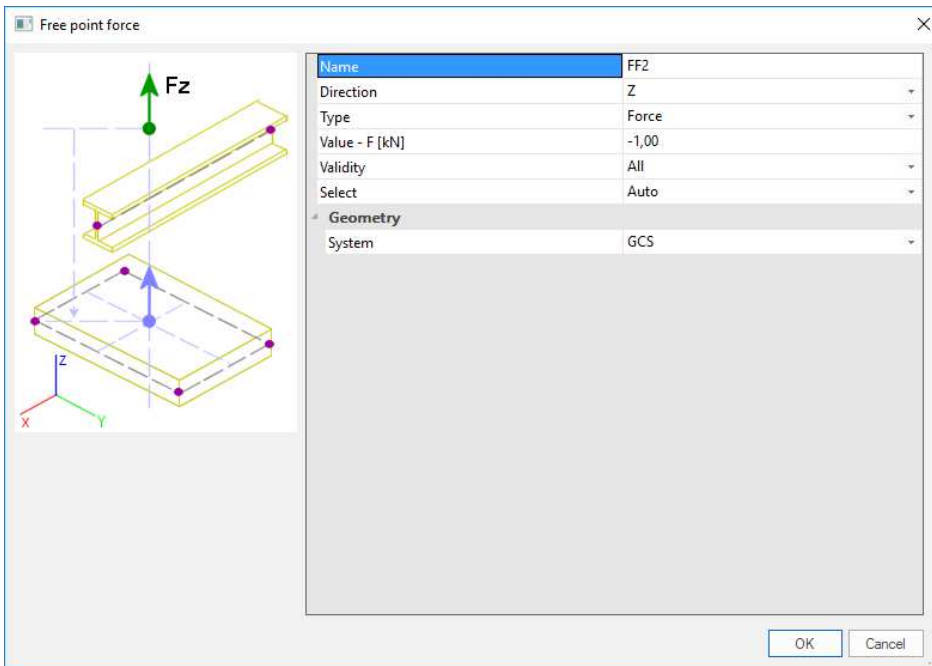


- **Extent:** definition on the full length of the beam or per span.
- **Coord. Definition:** define the position relative (value between 0 and 1) or absolute (value in m).
- **Origin:** definition from the start or the end of the element.
- **Repeat (n):** repeat the point force. If value is higher than one, you should define a distance between point forces.
- **Eccentricity:** gives the point force an eccentricity.

3.6.3. Free point force

You can only assign a free point force to a 2D member. You have to define the geometry in the XY-plane of the current UCS. When you draw the force it only has an X and Y coordinate. It will be generated on every 2D member that has a point that has this X and Y coordinate. This means that if you have a plate above this point force the force will be generated on this plate as well.



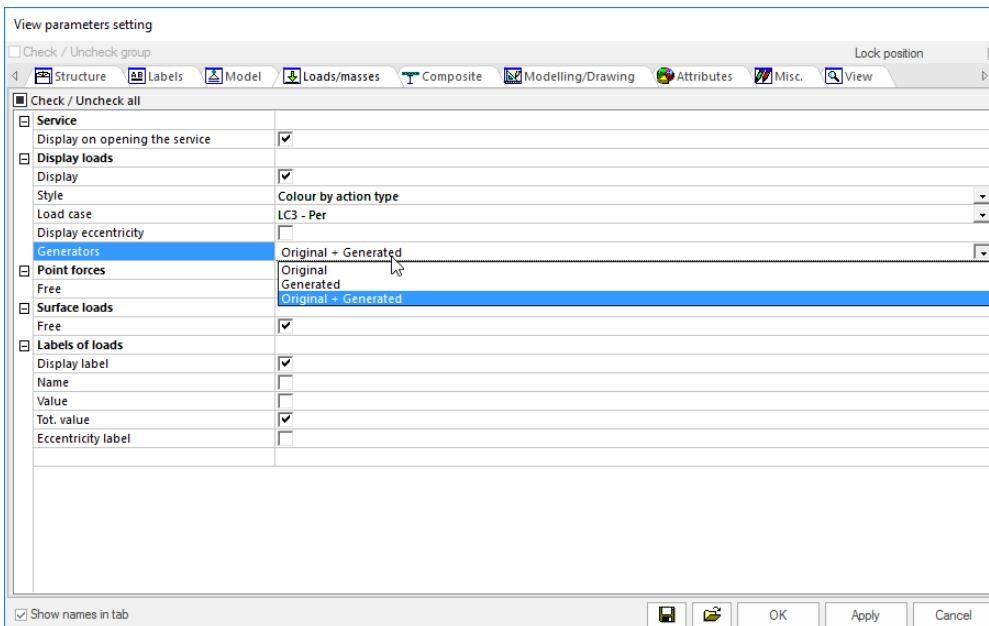


- **Validity:** this setting has an influence on the generation. You can set it to 'All', 'Z+', 'Z-' or '0'. This means that the loads will be generated on all the members or the members with a positive/negative/zero Z coordinate. When the option '**Select**' is defined for the property '**Select**', the loads will only be generated on the elements which are selected after executing the command '**Update 2D members selection**' in the tab '**Actions**'.
- **Select:** there are 2 options: 'Auto' and 'Select'. 'Auto' means the load is generated on all members. 'Select' means you should select the members to generate the load on.

To view an example of the generation you can select the load and press 'Generate loads' in the properties window.

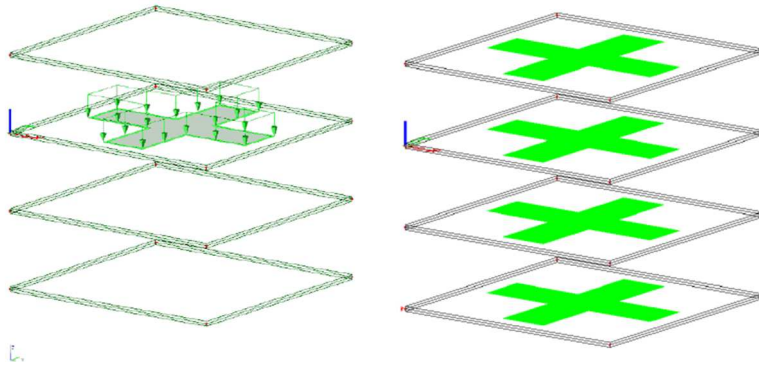


To switch the view from the generated loads to the original ones or the other way around you can delete the generated load OR right click → 'View parameters for all' → 'Loads/masses'

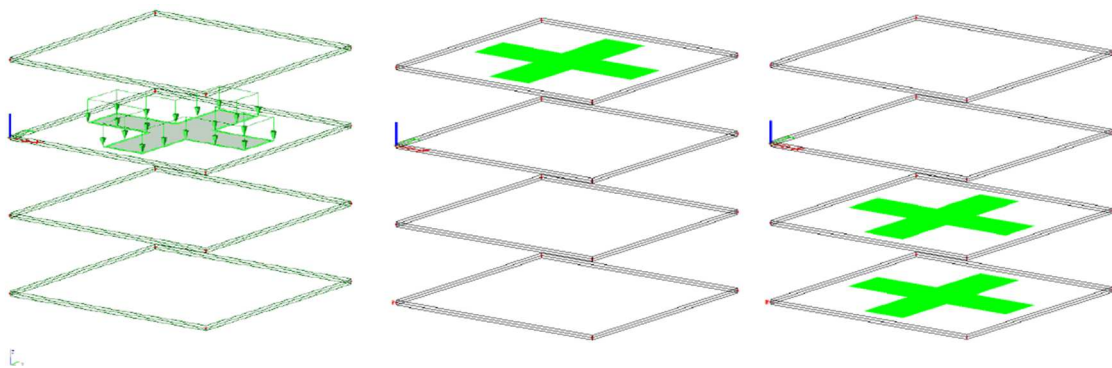


The following pictures provide a graphical representation of all the different validities which can be chosen.

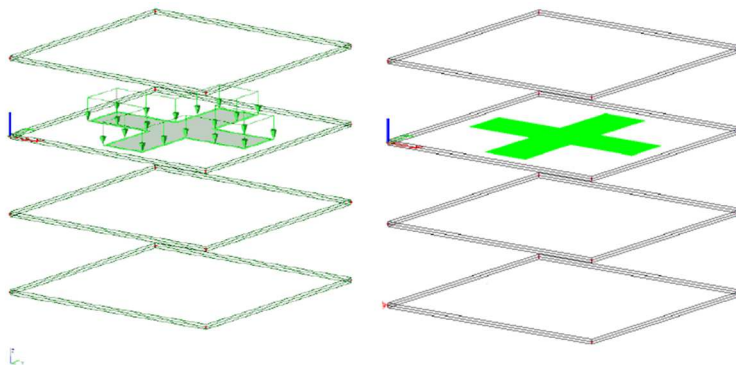
- **Select = Auto, Validity = All**



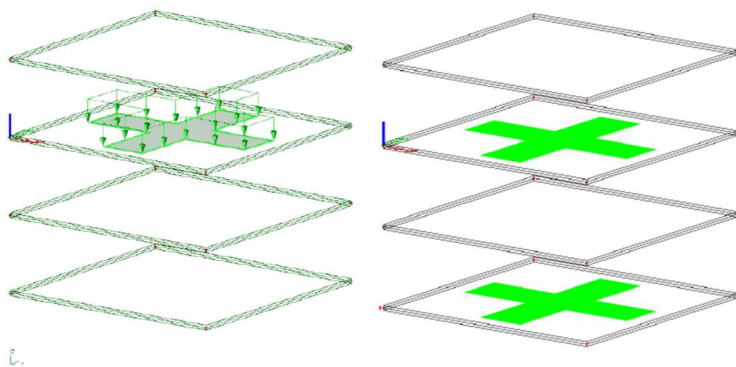
- **Select = Auto, Validity = +Z and -Z**



- **Select = Auto, Validity = Z = 0**

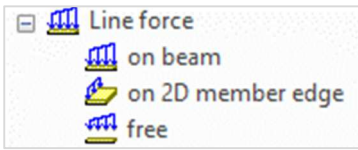


- **Select = Select, Validity = All (Plate 1 and 3 are selected)**



3.7. Line force

You can add three types of line forces: 'on beam', 'on 2D member edge' and 'free'.

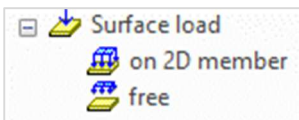


The settings for line forces are the same as for point forces. The only extra setting is the length of the line force. This can be set to relative (value between 0 and 1) or absolute.

Coord. definition	Rela
Position x1	0,000
Position x2	1,000

3.8. Surface load

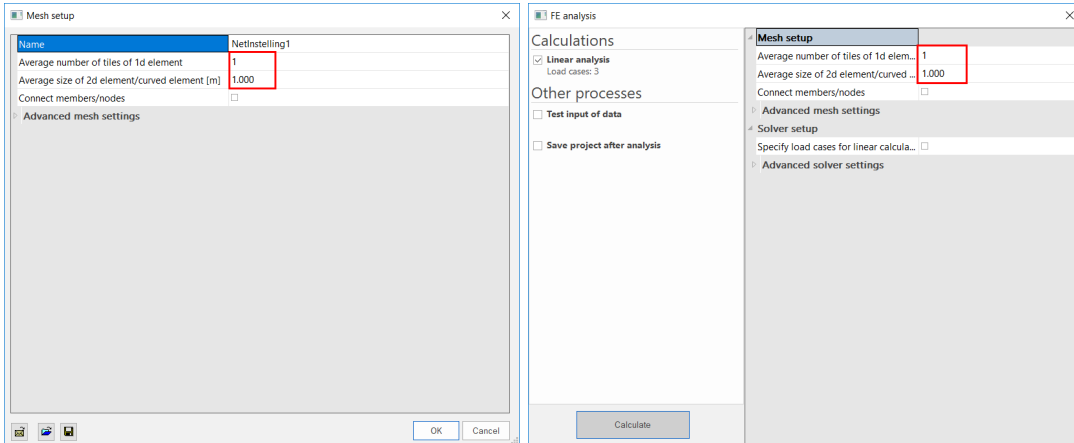
Next to point forces and line forces there are also surface loads. You can choose between two types of surface loads: 'on 2D member' and 'free'. The settings are the same as for the load types above.



Chapter 4: Calculation

4.1. Mesh

As default setting a 1D element consists of 1 mesh element and a 2D element has an average mesh element of 1m. You can change these settings in 'Setup' → 'Mesh' OR when starting the calculation.



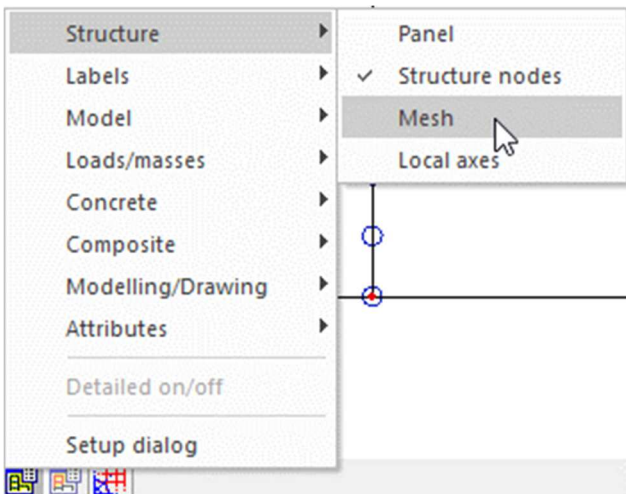
This setting has an influence on the accuracy of the results and on the speed of the calculation.

The mesh can be viewed by going to 'Set view parameters for all', via right mouse click in screen


- 'Structure' tab → 'Mesh' → Draw mesh
- 'Labels' tab → 'Mesh' → Display label

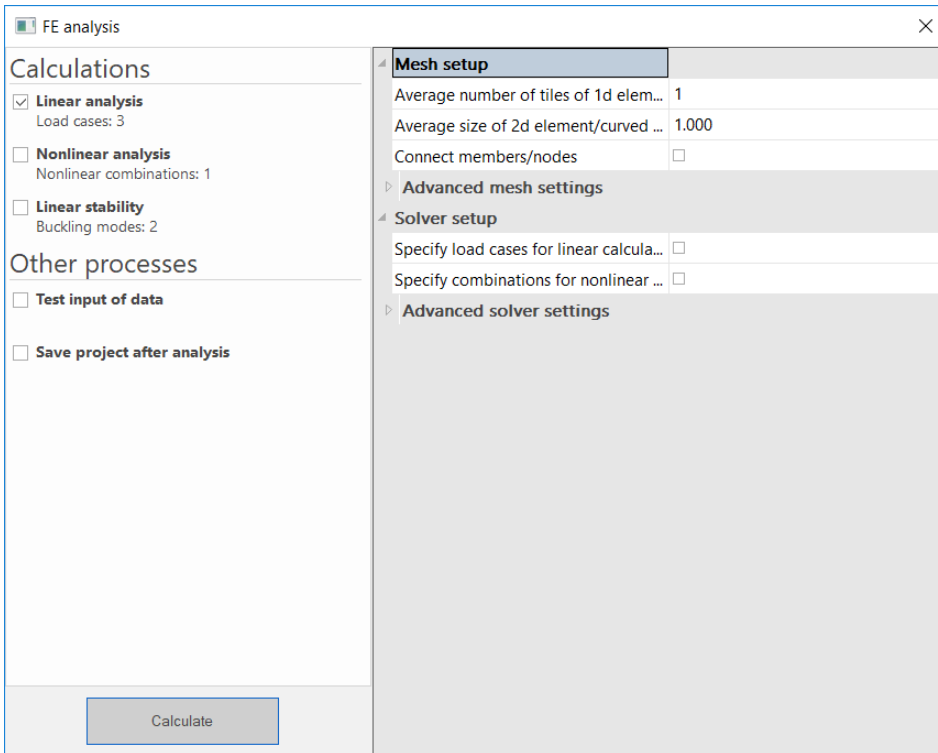
OR

Click on the 'Fast adjustment of view parameters on whole model' above the command line 



4.2. Calculation/solver

You can start the calculation can be started by clicking on the calculation shortcut in the toolbars , by going to the main tree → 'Calculation, mesh' → 'Calculation' or by typing **calc** in the command line.



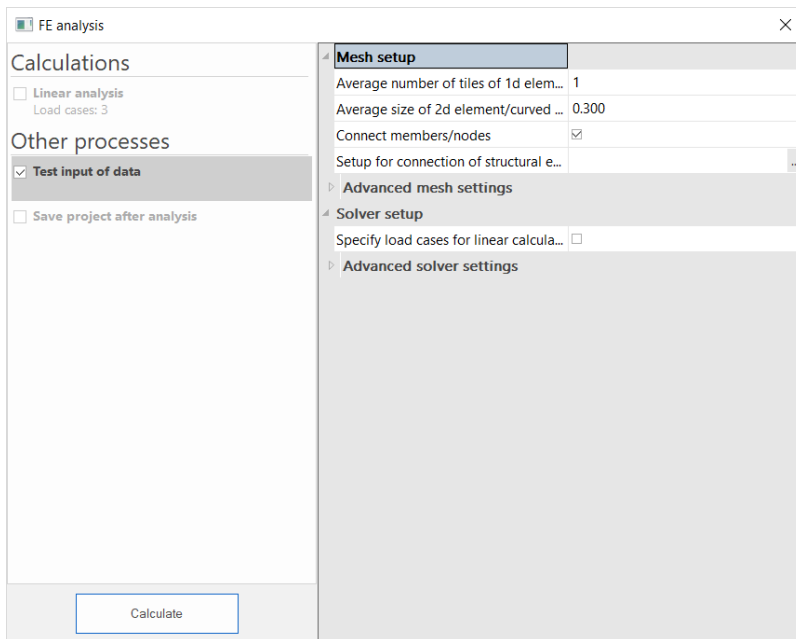
In the top left you can choose desired type(s) of analyses by checking/unchecking the boxes.

NOTE: you can turn on the option 'Connect members/nodes' so the program will automatically execute this function before starting the calculation.

Chapter 5: Results

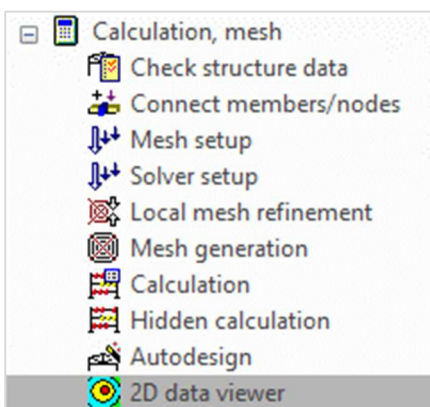
5.1. Test input data

When you start the calculation, the function 'Test input data' can be performed.

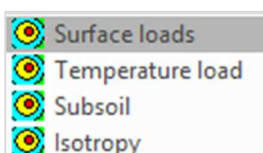


NOTE: the option 'Test input of data' is not supported in the 64 bit version.

In the main tree under 'Calculation, mesh' you will find a new option '2D data viewer'.



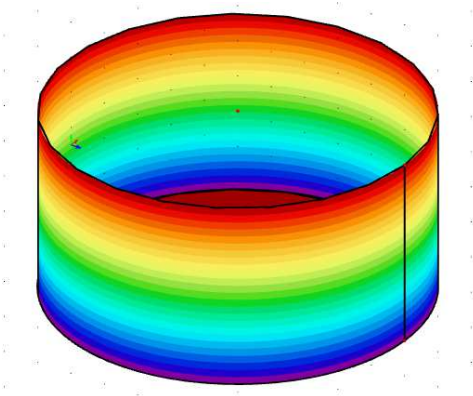
When you open this menu, you can check the input of the project. It contains: 'Surface loads', 'Temperature load', 'Subsoil' and 'Isotropy'.



Select the input to be checked and change the properties menu to view the correct input. You can find an example below for a tank with a water pressure load.

Name	Oppervlakte lasten
Selection	All
Type of loads	Load cases
Load cases	BG2 - Variabele druk
Filter	No
Values	qz
System	Local
Values	More comp
Fx	<input checked="" type="checkbox"/>
Fy	<input checked="" type="checkbox"/>
Fz	<input checked="" type="checkbox"/>
Mx	<input checked="" type="checkbox"/>
My	<input checked="" type="checkbox"/>
Mz	<input checked="" type="checkbox"/>
Drawing setup 2D	...

- **Selection:** view the input on the complete structure or on a part of it.
- **Type of loads, Load cases:** select the load case to test the input.
- **Values:** set the value to check.
- **System:** choose 'Local' or 'Global' system. The local system follows the local axis of the elements while the global system is according to the global axis. For this example, 'Local' is chosen because we look at a curved element.

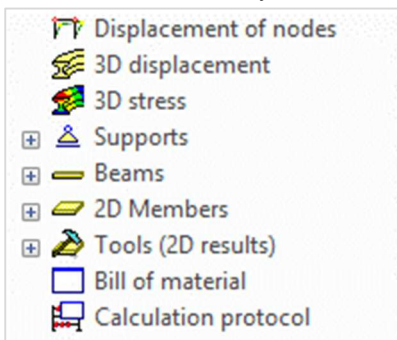


5.2. Requesting results

After you perform the calculation, the 'Results' menu will appear in the main tree.

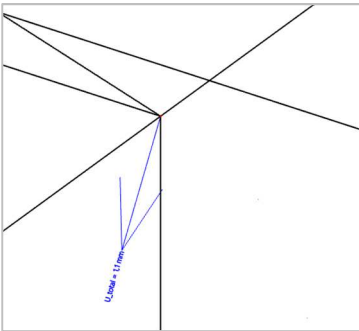


In the 'Results' menu you can ask for the following results:



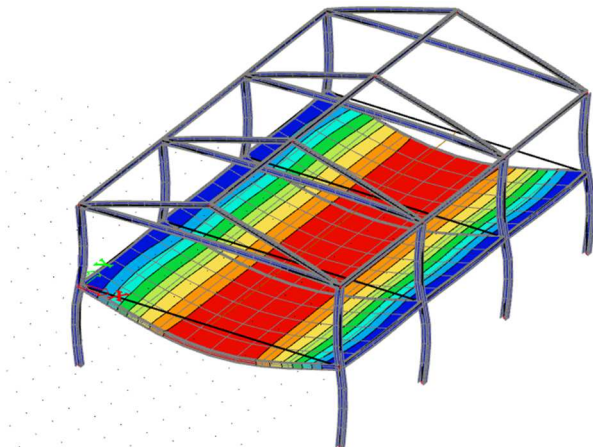
5.2.1. Displacement of nodes

This result gives you the displacement and rotation of all the nodes in the structure. This is the purest result out of the FEM analysis.



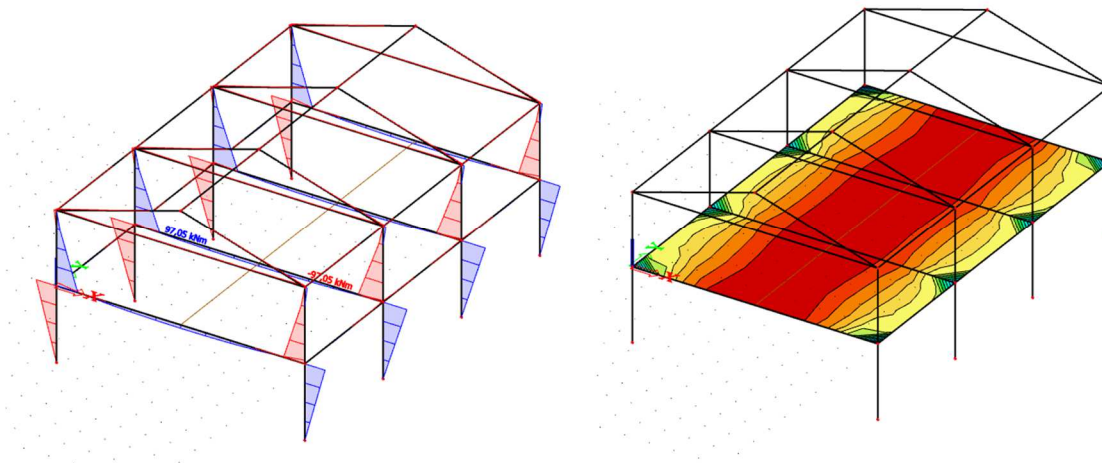
5.2.2. 3D Results

There are two types of 3D results: '3D displacement' and '3D stress'. In order to generate these results, the 1D results of 1D members and 2D results of 2D members are converted to 3D results. This is done with transformation formulas. These results are not calculated in the calculation by the solver but are generated by the post processor. Due to this transformation, the generation of results may take some time, depending on the size of the structure (and the refinement of the mesh).



5.2.3. Results per component

For each component there is an item in the result menu: **supports, beams (1D members) and 2D members**. For each of these components there are detailed results available. These results are calculated during the calculation by the solver and take no additional time to generate. The difference with the 3D results is that these results are viewed in 1D for beams and in 2D for 2D Members.



5.2.4. Bill of material

In the bill of material you will find information concerning the **mass**, **surface** and **volume** of the used materials in the project.

Bill of material

Selection: All
Type of sorting: Material

Summary

Material	Mass [kg]	Surface [m ²]	Volume [m ³]
Steel	183028,2	353,413	2,3316e+01
Total	183028,2	353,413	2,3316e+01

Note: Value 'Surface' represents for 1D members the total exposed surface area, while for 2D members it corresponds only to the surface area of the centroidal plane.

Steel (1D)

Material	Density [kg/m ³]	Mass [kg]	Surface [m ²]	Volume [m ³]
S 235	7850,0	6403,2	203,413	8,1570e-01
Total		6403,2	203,413	8,1570e-01

Steel (2D)

Material	Density [kg/m ³]	Mass [kg]	Surface [m ²]	Volume [m ³]
S 235	7850,0	176625,0	150,000	2,2500e+01
Total		176625,0	150,000	2,2500e+01

5.2.5. Calculation protocol

In the calculation protocol you can check the equilibrium between the inputted loads and the acting reactions.

Calculation protocol

Linear calculation

Number of 2D elements	150
Number of 1D elements	107
Number of mesh nodes	196
Number of equations	1176
Bending theory	Mindlin
Load cases	BG1, BG2
Start of calculation	19.11.2018 00:50
End of calculation	19.11.2018 00:50

Sum of loads and reactions

Load case	Value	X [kN]	Y [kN]	Z [kN]
BG1	loads	0,00	0,00	-1795,51
	reaction in nodes	0,00	0,00	1795,51
	reaction on lines	0,00	0,00	0,00
	contact 1D	0,00	0,00	0,00
	contact 2D	0,00	0,00	0,00
BG2	loads	0,00	0,00	-300,00
	reaction in nodes	0,00	0,00	300,00
	reaction on lines	0,00	0,00	0,00
	contact 1D	0,00	0,00	0,00
	contact 2D	0,00	0,00	0,00

5.2.6. Setting the properties menu

After you selected a result type, you need to set some parameters in the properties window. As an example the properties window of the '2D internal forces' will be taken. Other results have similar options in the properties window.

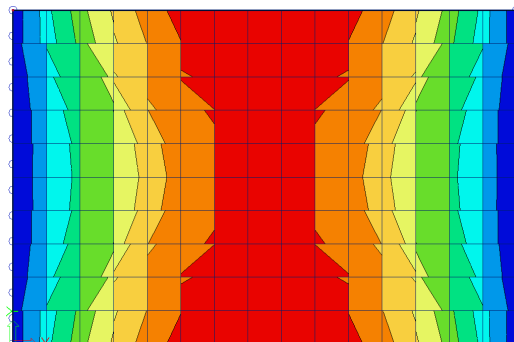
Name	2D internal forces
Selection	
Type of selection	All
Filter	No
Result case	
Type of load	Combinations
Combination	UGT-Set B (automatic)
Envelope (for 2D drawing)	Absolute extreme
Averaging of peak	<input type="checkbox"/>
Location	In nodes avg. on macro
System	LCS mesh element
Extreme	Global
Type of values	Basic magnitudes
Values	m_x
Output settings	
Print combination key	<input checked="" type="checkbox"/>
Standard result	<input checked="" type="checkbox"/>
Results on sections	<input type="checkbox"/>
Results on edges	<input type="checkbox"/>
Table setup	
Errors, warnings and not...	
Drawing setup 2D	...

- **Type of selection:** view results on the complete structure or on a selection of it.
- **Filter:** filter on material, wildcard, layer or thickness.
- **Type of load:** choose the load case, combination or result class.
- **Envelope:** show maximum, minimum or absolute extreme values. Absolute extreme show you the biggest absolute value.
- **Location:** the finite elements mesh in SCIA Engineer exists of linear 3- and/or 4-angular elements. Per mesh element 3 or 4 results are calculated, one in each node. When you ask for the results on 2D members, the option 'Location' in the properties window gives you the possibility to display these results in 4 ways.

- **In nodes, no average**

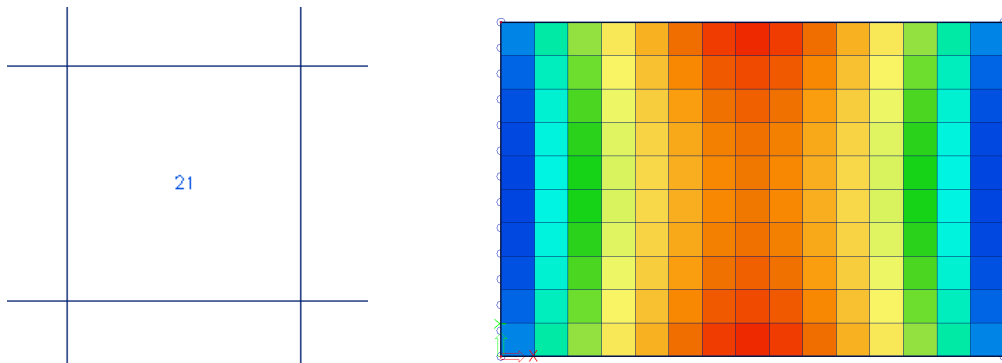
All result values are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large. This display of results therefore gives you a good idea of the discretisation error in the calculation model.

12	16	24	30
9	18	25	31
11	16	24	29
9	17	24	30



o **In centres**

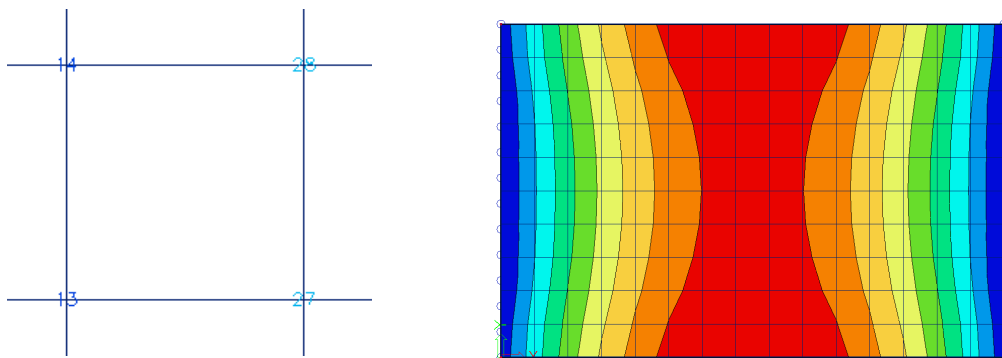
Per finite element, the mean value of the results in the nodes of that element is calculated. Since there is only 1 result per element, the display of isobands becomes a mosaic. The course over a section is a curve with a constant step per mesh element.



o **In nodes, average**

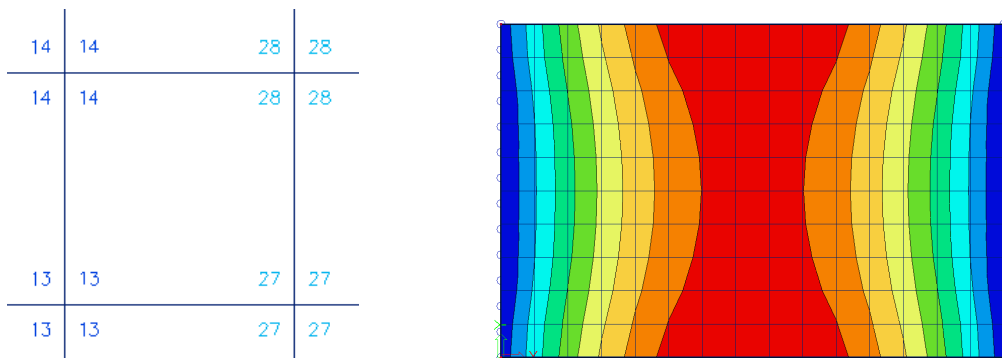
The values of the results of adjacent finite elements are averaged in the common node. Because of this, the graphical display is a smooth course of isobands. In certain cases, it is not permissible to average the values of the results in the common node:

- At the transition between 2D members (plates, walls, shells) with different local axes;
- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.



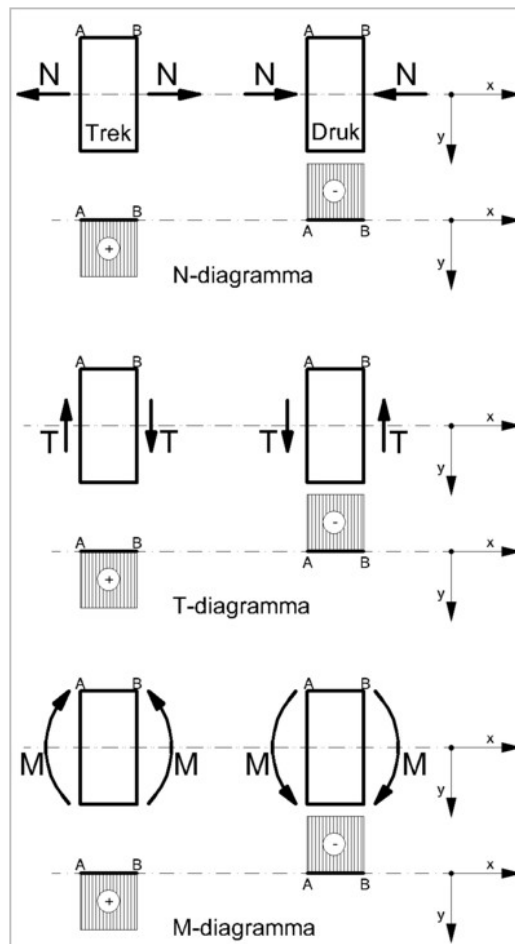
o **In nodes, average on macro**

The values of the results are averaged per node *only* over mesh elements which belong to the same 2D member and which have the same directions of their local axes. This resolves the problems mentioned at the option 'In nodes, average'.



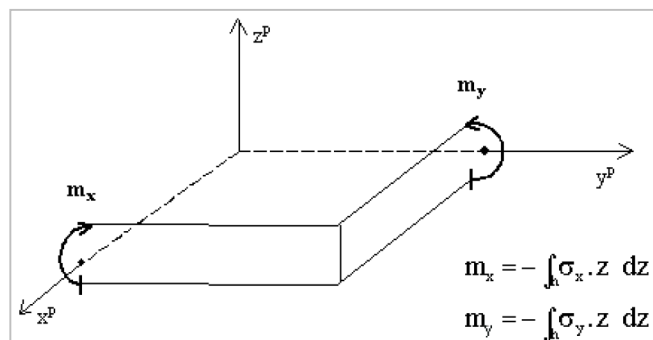
- **System:** direction according to: 'LCS of the mesh' or 'LCS of the element'.
- **Extreme:** you can set this setting on 'Global', 'Member' or 'Mesh'. 'Global' means that only the maximum value of the complete model is shown. For 'Member' the maximum per member is shown. For 'Mesh' the maximum per mesh element is shown.
- **Type of values:** there are three types of values: 'Basic magnitudes', 'Principal magnitudes' and 'Design values'.
 - **Basic magnitudes = Characteristic values**

- **Beams 1D**

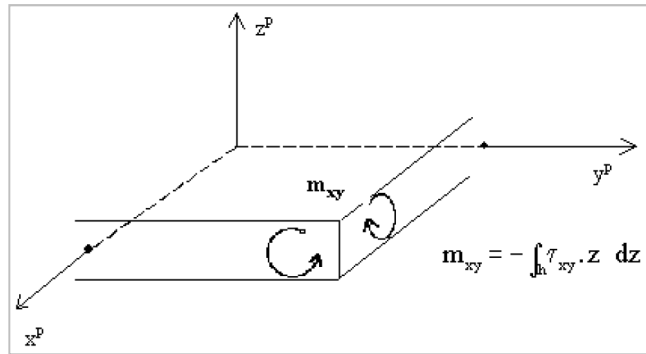


- **Bending (plates, shells)**

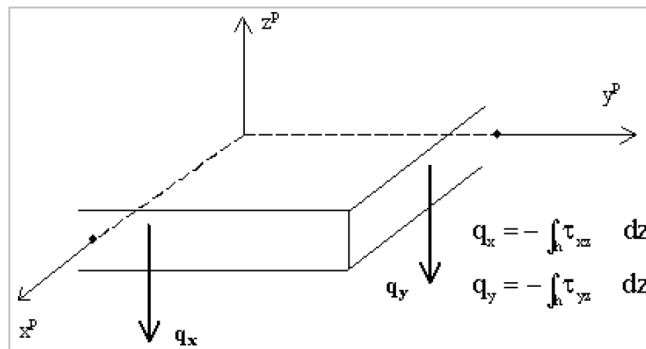
- Bending moments m_x , m_y



- Torsion moment m_{xy}

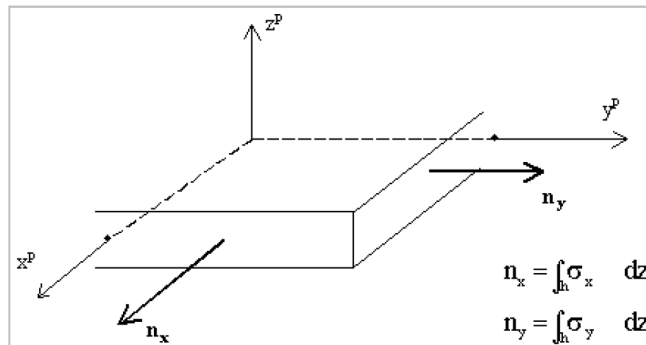


- Shear forces q_x, q_y ($=v_x, v_y$)

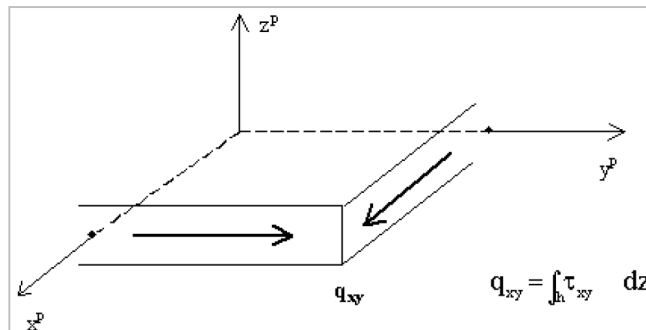


▪ **Membrane effects (walls, shells)**

- Membrane forces n_x, n_y

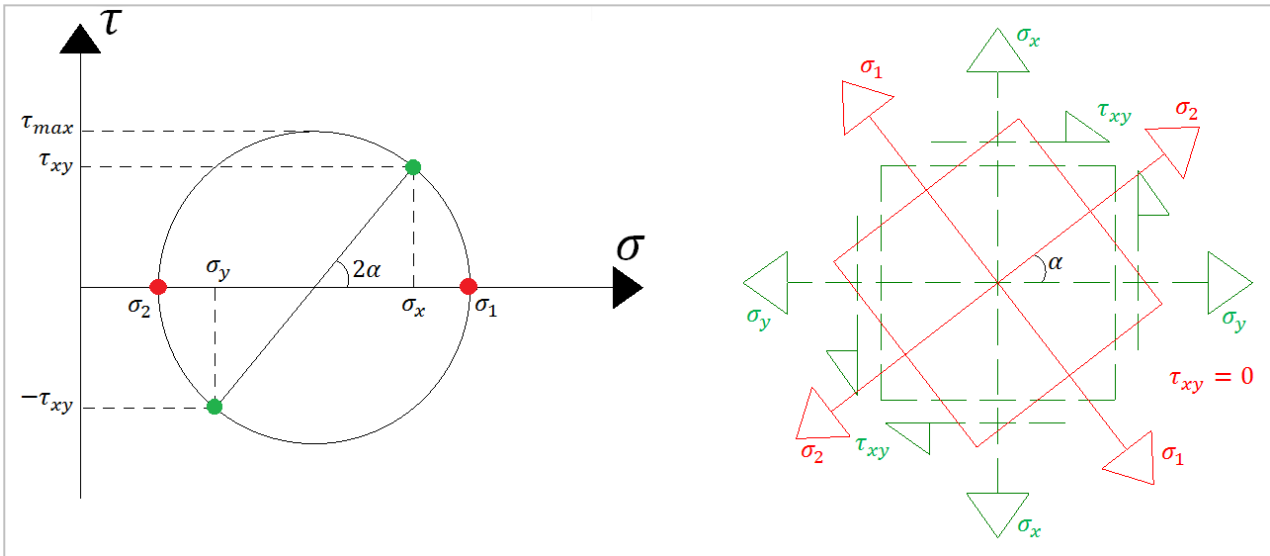


- Shear force q_{xy} ($=n_{xy}$)



o **Principal magnitudes**

The principal magnitudes give the results according to the axes of the directions of the largest stresses (principal directions). These directions are defined with the help of the Mohr's circle.



o **Elementary design magnitudes = Design values**

To derive the dimensional magnitudes from the basic magnitudes, formulas of the Eurocode are used (you can find these formulas also on our help-page https://help.scia.net/webhelplatest/en/#b/results/calculated_results_for_2d_members.htm%3FTocPath%3DResultaten%7CResultaten%2520op%2520platen%7C_____1)

https://help.scia.net/webhelplatest/en/#b/results/calculated_results_for_2d_members.htm%3FTocPath%3DResultaten%7CResultaten%2520op%2520platen%7C_____1

Contents

- SCIA Engineer Help
- Version info
- Learning Videos
- Installation
- Protection of SCIA Engineer
- Basics
- Materials
- Cross-sections
- Geometry
- Model data
- Foundation and subsoil
- Loads
- Masses
- Construction stages, TDA, prestressing
- Calculation
- Results
 - Opening the service Results
 - Selecting the beams for display
 - Selecting the load for the display of results
 - Selecting the geometry for display of result
 - Adjusting the style of result diagrams
 - Adjusting the style of result diagrams since
 - Regenerating the diagrams
 - Preventing the results after model modifica
 - Animation of results
 - Upgrade from 2D to 1D project
 - Bill of material (v17)
 - Results on beams
 - Results on slabs
 - Calculated results for 2D members
 - Displaying the deformation of nodes on

The calculation of the design forces for the plates and shells according to the EC2 algorithm uses the flow diagram in ČSN P ENV 1992-1-1 (731201), Appendix 2, par. A2.8. Following rules are valid for the indices:

$m_y \geq m_x : a = x, b = y$
 $m_y < m_x : a = y, b = x$

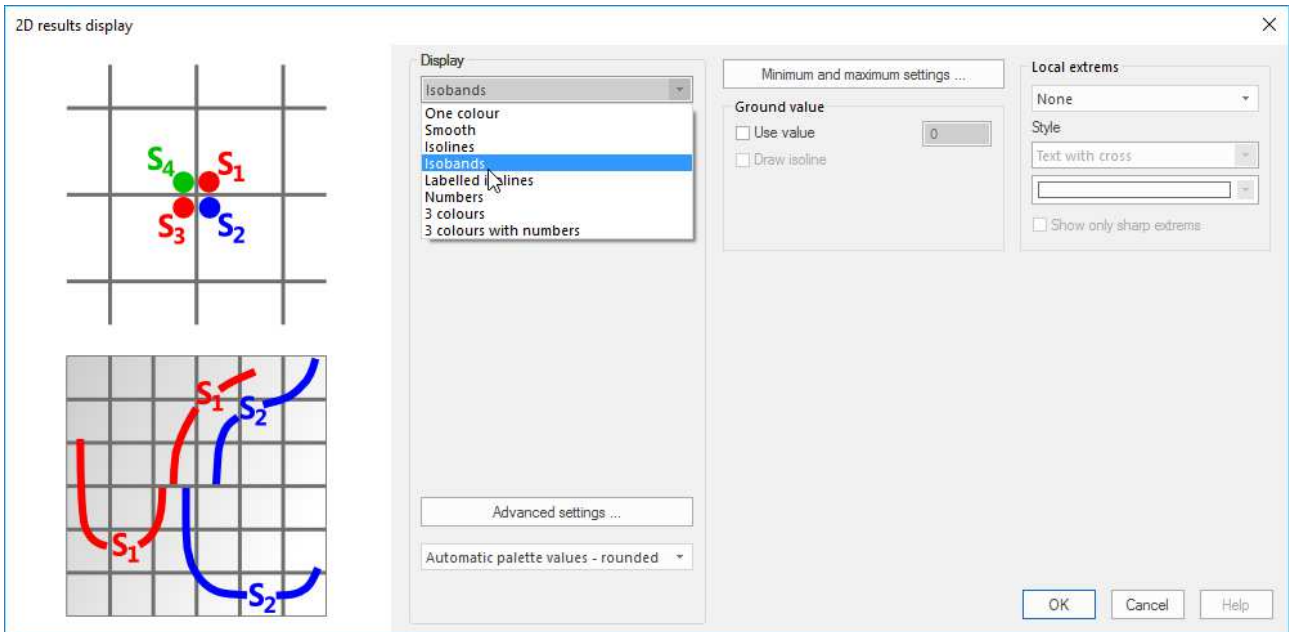
```

    graph TD
        A{m_x >= -|m_y|} -- YES --> B["m_{aD} = m_a + |m_y|  
m_{bD} = m_b + |m_y|  
m_{tD} = -2 * |m_y|"]
        A -- NO --> C["m_{aD} = 0  
m_{bD} = m_b + m_y^2 / |m_a|  
m_{tD} = -|m_a| * (1 + (m_y / m_a)^2)"]
        B --> D{m_b <= |m_y|}
        C --> D
        D -- YES --> E["m_{aD} = -m_a + |m_y|  
m_{bD} = -m_b + |m_y|  
m_{tD} = -2 * |m_y|"]
        D -- NO --> F["m_{aD} = -m_a + m_y^2 / |m_b|  
m_{bD} = 0  
m_{tD} = -|m_b| * (1 + (m_y / m_b)^2)"]
    
```

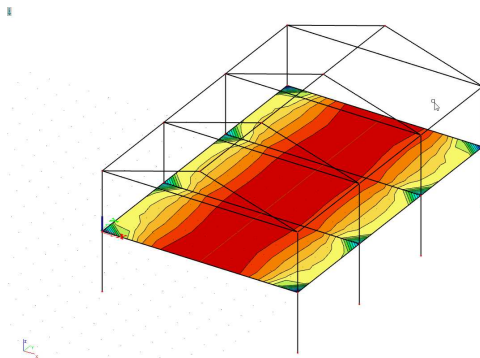
The calculation of the design forces for the walls and shells according to EC2 algorithm uses the flow diagram in ČSN P ENV 1992-1-1 (731201), Appendix 2, par. A2.9. Following rules are valid for the indices:

$n_y \geq n_x : a = x, b = y$
 $n_y < n_x : a = y, b = x$

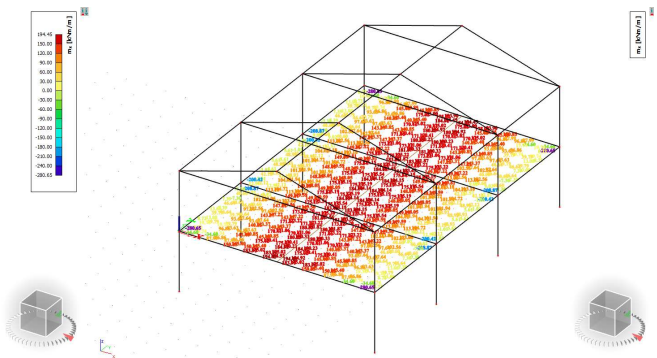
- **Drawing setup 2D:** by pressing the three dots you open the window '2D results display'. In this window you can set how you want to see the results (isobands, isolines, numbers, ...). You can also set a maximum and minimum value.



Isobands



Numbers



5.3. Table results

You can also ask for the results in a table. You can open the table results by clicking on the 'Table results' in the properties window.

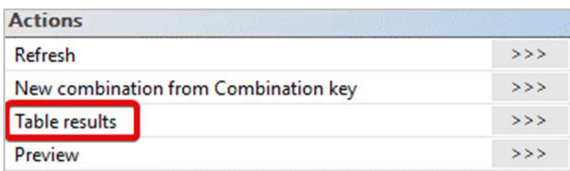


Table results

2D internal forces: Linear calculation: Combination: CO1; Extreme: Global; Selection: All; Location: In nodes avg. on macro; System: LCS mesh element

Id	Name	Mesh	x [m]	y [m]	z [m]	Case	mx [kNm/m]	my [kNm/m]	mxy [kNm/m]	vx [kN/m]	vy [kN/m]	rx [kN/m]	ry [kN/m]	rxv [kN/m]
1	E1	Element: 1; Node: 1	0,000	0,000	3,000	CO1/1	-280.65	-78.80	-40.44	682.47	164.53	11.63	2.59	4.66
2	E1	Element: 5; Node: 180	5,000	0,000	3,000	CO1/1	194.45	4.42	0,00	0,00	9.82	2,00	-0,05	0,00
3	E1	Element: 20; Node: 33	10,000	2,000	3,000	CO1/1	13,08	64.26	5,18	-50,14	24,57	-0,15	-2,21	0,01
4	E1	Element: 8; Node: 186	8,000	0,000	3,000	CO1/1	87,99	6,62	-42.57	-187,51	27,47	4,30	0,23	-0,08
5	E1	Element: 2; Node: 66	2,000	0,000	3,000	CO1/1	87,99	6,62	42.57	187,51	27,47	4,30	0,23	0,08
6	E1	Element: 41; Node: 5	0,000	5,000	3,000	CO1/1	-208,87	-194,48	39,17	545,00	-319.78	6,42	2,75	-3,45
7	E1	Element: 101; Node: 8	10,000	10,000	3,000	CO1/1	-208,87	-194.48	39,17	-545,00	319.78	6,42	2,75	-3,45
8	E1	Element: 1; Node: 57	0,000	1,000	3,000	CO1/1	8,72	42,68	-15,27	-249,11	107,95	-4.26	-3.30	-1,47
9	E1	Element: 51; Node: 6	10,000	5,000	3,000	CO1/1	-200,42	-192,12	36,89	-542,29	293,27	6,76	3.27	-3,74
10	E1	Element: 10; Node: 2	10,000	0,000	3,000	CO1/1	-280,65	-78,80	40,44	-682.47	164,53	11,63	2,59	-4.66

Basic magnitudes List of Combination keys

2D internal forces

5.4. Results preview

You can view the results as well in the preview. The preview shows you the table that is generated by the engineering report.

Actions	
Refresh	>>>
New combination from Combination key	>>>
Table results	>>>
Preview	>>>

2D internal forces

Linear calculation

Combination: CO1

Extreme: Global

Selection: All

Location: In nodes avg. on macro. System: LCS mesh element

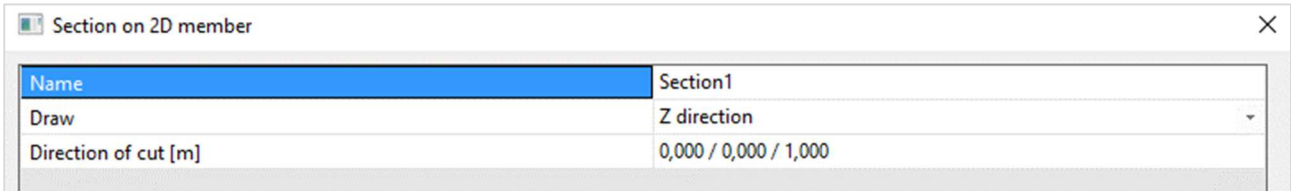
Basic magnitudes

Name	Mesh	Position [m]	Case	m_x	m_{xy}	v_x	n_x	n_{xy}
				[kNm/m] m_y [kNm/m]	[kNm/m]	[kN/m] v_y [kN/m]	[kN/m] n_y [kN/m]	[kN/m]
E1	Element: 1 Node: 1	0,000 0,000 3,000	CO1/1	-280,65 -78,80	-40,44	682,47 164,53	11,63 2,59	4,66
E1	Element: 5 Node: 180	5,000 0,000 3,000	CO1/1	194,45 4,42	0,00	0,00 9,82	2,00 -0,05	0,00
E1	Element: 20 Node: 33	10,000 2,000 3,000	CO1/1	13,08 64,26	5,18	-50,14 24,57	-0,15 -2,21	0,01
E1	Element: 8 Node: 186	8,000 0,000 3,000	CO1/1	87,99 6,62	-42,57	-187,51 27,47	4,30 0,23	-0,08
E1	Element: 2 Node: 66	2,000 0,000 3,000	CO1/1	87,99 6,62	42,57	187,51 27,47	4,30 0,23	0,08
E1	Element: 41 Node: 5	0,000 5,000 3,000	CO1/1	-208,87 -194,48	39,17	545,00 -319,78	6,42 2,75	-3,45
E1	Element: 101 Node: 8	10,000 10,000 3,000	CO1/1	-208,87 -194,48	39,17	-545,00 319,78	6,42 2,75	-3,45
E1	Element: 1 Node: 57	0,000 1,000 3,000	CO1/1	8,72 42,68	-15,27	-249,11 107,95	-4,26 -3,30	-1,47
E1	Element: 51 Node: 6	10,000 5,000 3,000	CO1/1	-200,42 -192,12	36,89	-542,29 293,27	6,76 3,27	-3,74
E1	Element: 10 Node: 2	10,000 0,000 3,000	CO1/1	-280,65 -78,80	40,44	-682,47 164,53	11,63 2,59	-4,66

Name	Combination key
CO1/1	1.35*BG1 + 1.50*BG2

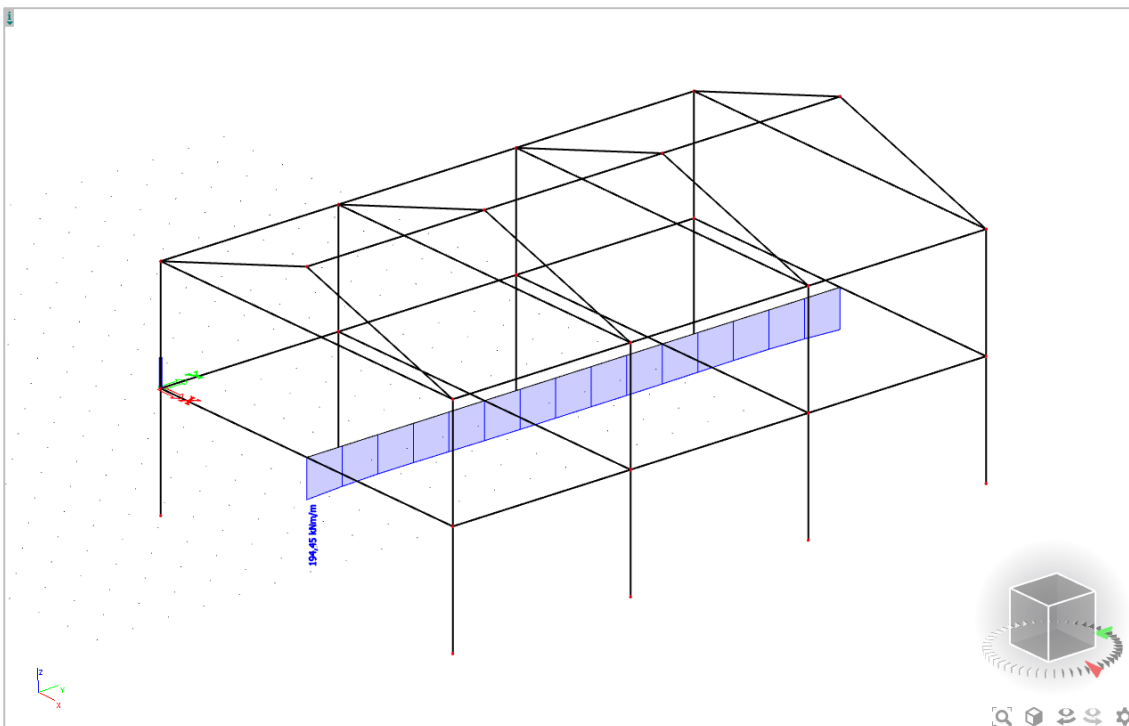
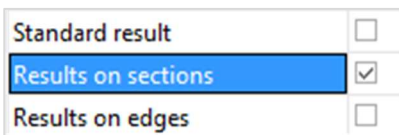
5.5. Section on 2D member

You can find the 'Section on 2D member' command in the 'Results' menu under 'Tools (2D results)'.



- **Draw:** determines the direction of the result which is drawn on the section.
- **Direction of cut:** defines the direction of the cut that will be made by a vector X/Y/Z. For example, 0/0/1 is a cut in the Z direction.

To view the results on the section you need to check the 'Results on sections' in the properties window when you ask for a result.



5.6. Averaging strip

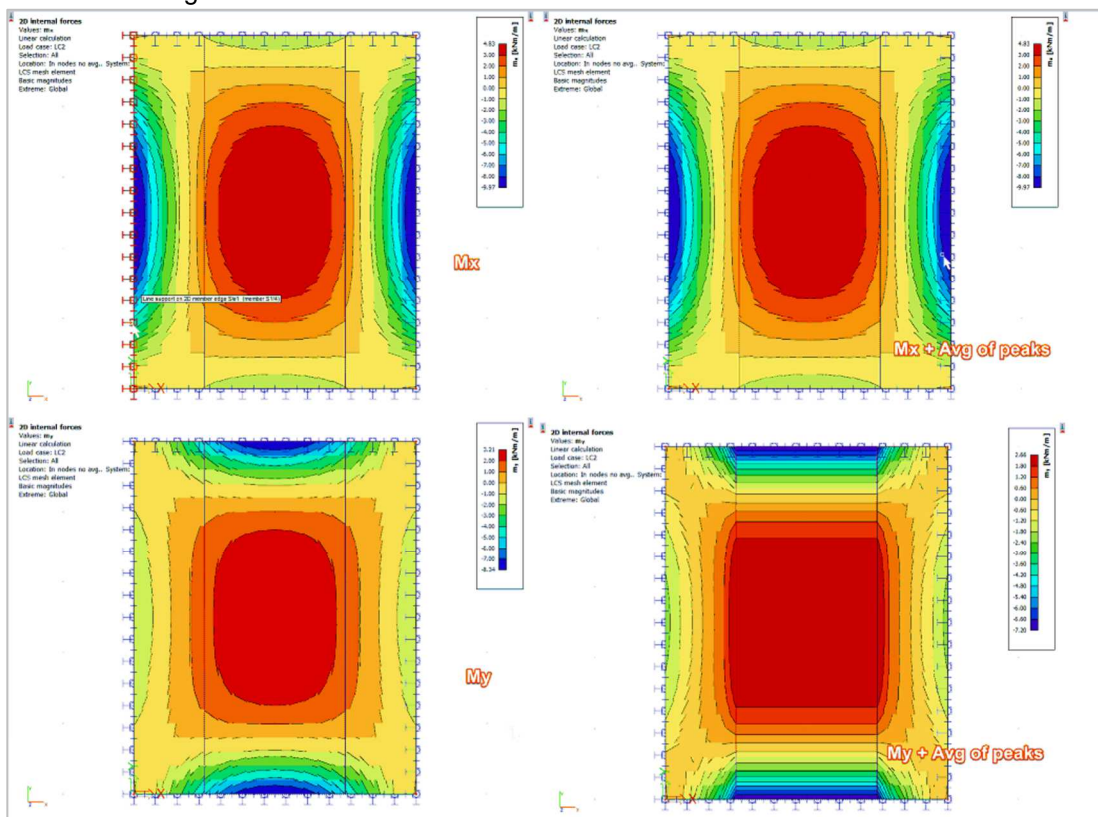
An averaging strip averages peak values over a zone. You can find the averaging strip in the 'Results' menu under 'Tools (2D results)'.



Name	RS1
Type	Strip
Width [m]	1,000
Direction	longitudinal

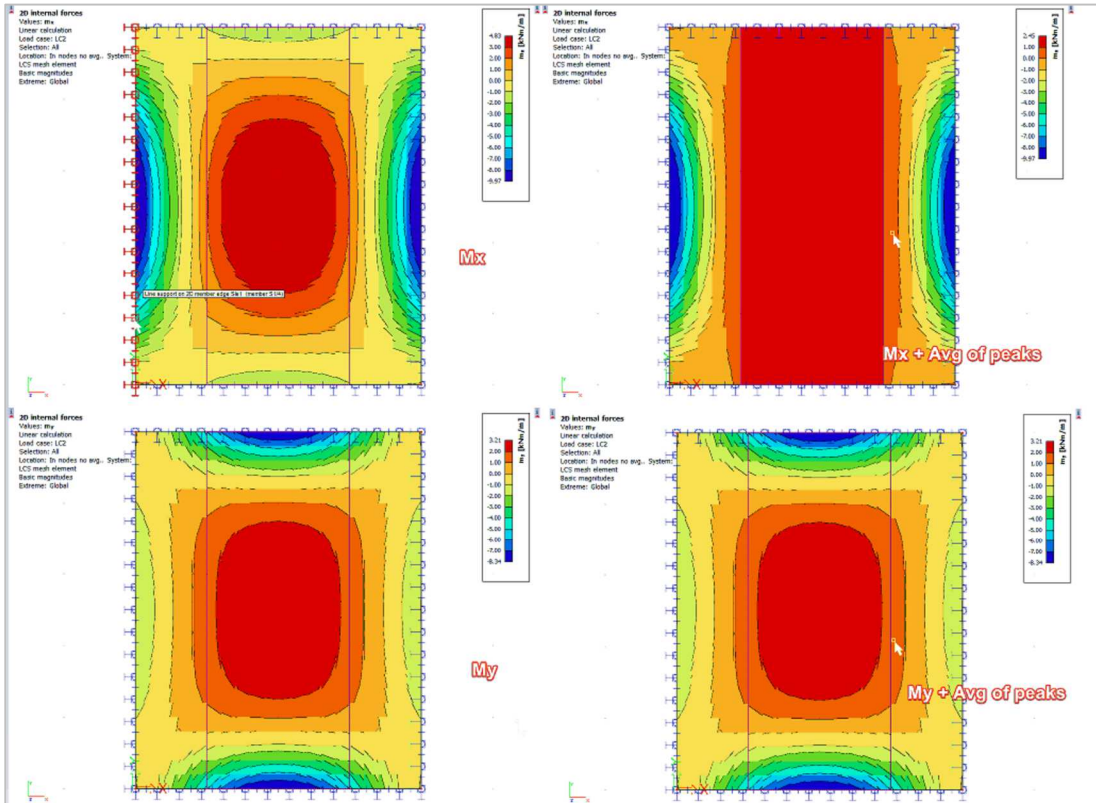
- **Type:** you can choose between a point or a strip.
- **Dimensions:** here you need to set the dimensions of the point/strip.
- **Direction:**

1) Direction = Longitudinal



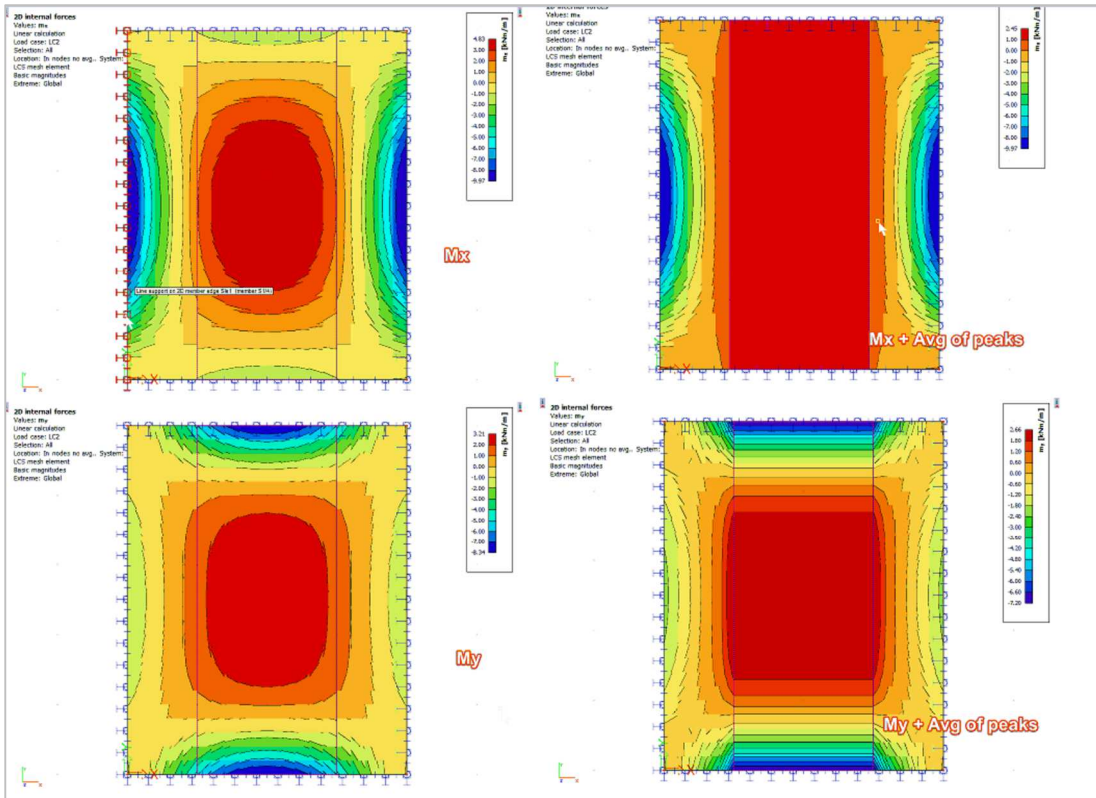
Longitudinal means that the averaging is done in the longitudinal direction of the strip. In the example above this is the y-direction. This means that the averaging is done for my. The values my are averaged in the x-direction.

2) Direction = perpendicular



Perpendicular means that the averaging is done perpendicular to the longitudinal direction of the strip. In the example above this is the x-direction. This means that the averaging is done for m_x . The values m_x are averaged in the y-direction.

3) Direction = Both



Both means that the averaging is done in both directions of the averaging strip. This means the values are averaged for m_x as well as m_y in the direction perpendicular to m_x and m_y .

To activate the averaging strip you need to check the option 'Averaging of peak' in the properties window.

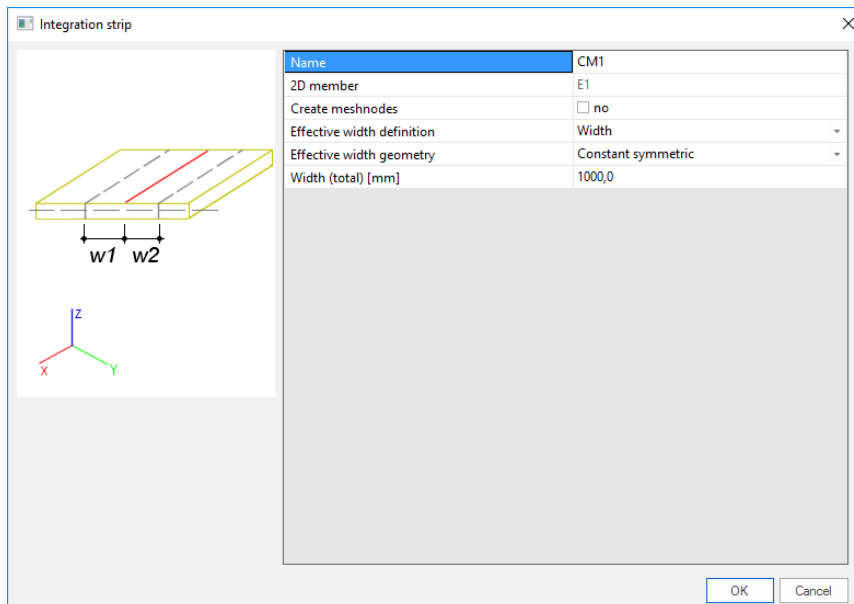
Result case	
Type of load	Combinations
Combination	CO1
Envelope (for 2D drawing)	Absolute extreme
Averaging of peak	<input checked="" type="checkbox"/>
Location	In nodes avg. on macro
System	LCS mesh element
Extreme	Global
Type of values	Basic magnitudes
Values	m_x

5.7. Integration strip / integration member

An integration strip is a strip which is defined on a 2D member. On this strip you can ask results as if it is a 1D member. An integration member works in 3D, so the results on multiple 2D members will be given on the strip as if it is a 1D member. You can find both functions in the 'Results' menu under 'Tools (2D results)'.

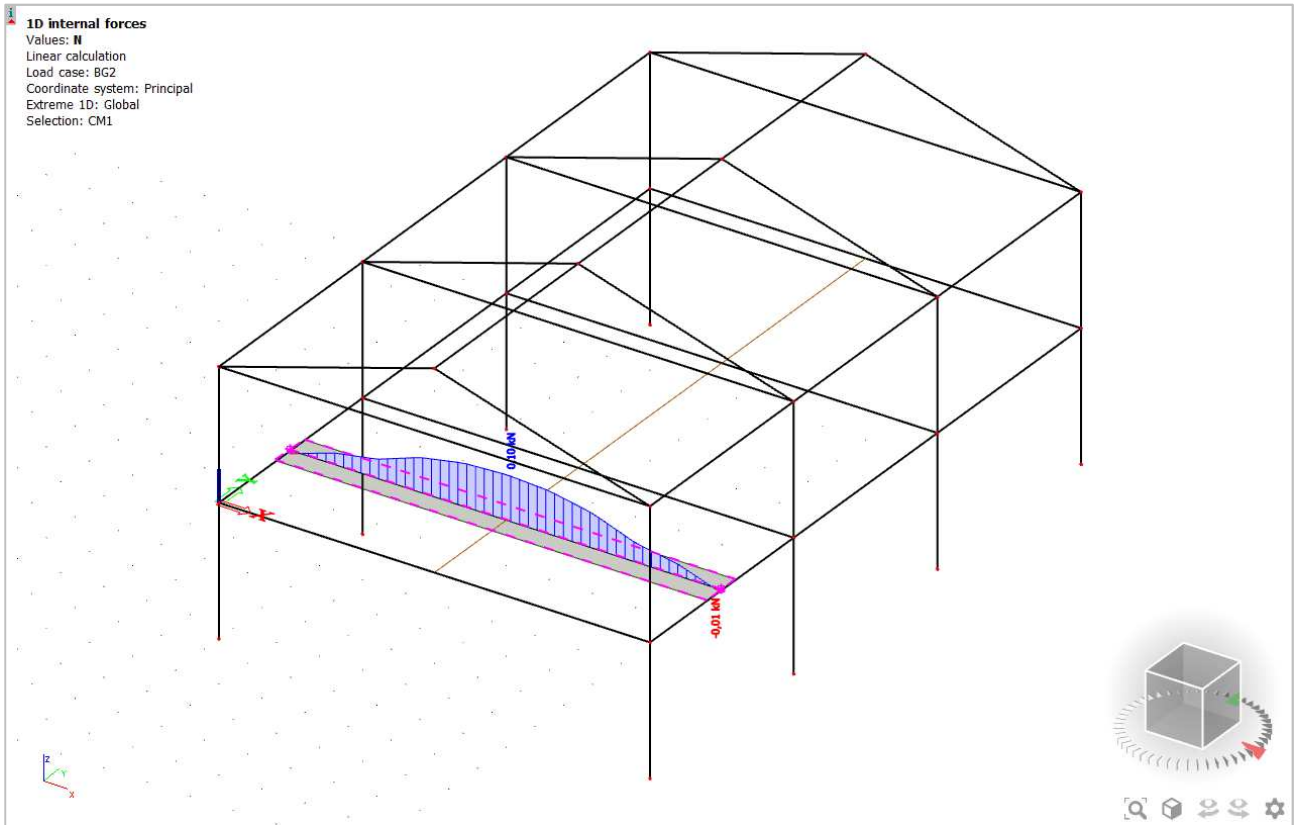


Set the width of the averaging strip.



Go to 'Beams' → 'Internal forces' → check the option 'Integration strip/member'

Result case	
Type of load	Load cases
Load case	BG2 - Use
Integration strip/member	<input checked="" type="checkbox"/>



The workflow is similar for an integration member. In the integration member window you can set the shape and dimensions of this member.

Integration member

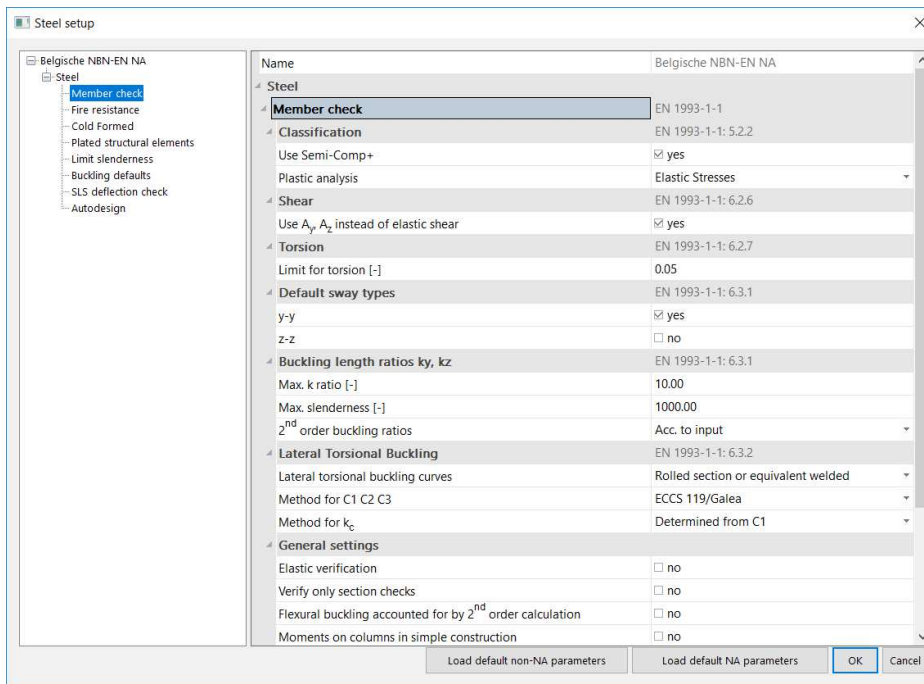
Name	IM1
Bounding box	
Shape	Rectangle
Rectangular integration shape	
Left [m]	0.500
Right [m]	0.500
Bottom [m]	0.500
Top [m]	0.500
Layer	Laag1

OK Cancel

Chapter 6: Steel design

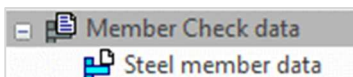
6.1. Steel setup

You can open the steel setup by going to the 'Steel' menu → 'Beams' → 'Steel setup'.



In this window you can change the general settings. These settings have an influence on the checks. By default these settings are according to the Eurocode. You can find more detailed information about this menu in the steel manual.

You can overrule the settings in the steel setup for one (or more) member(s) by assigning '**Steel member data**'. If you choose this option a window will open where you can assign these settings to one or more member(s).

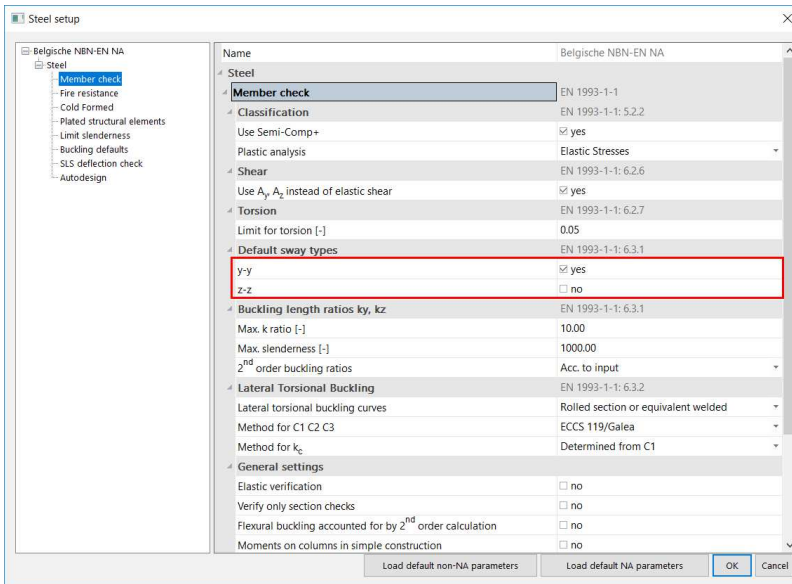


6.2. Buckling settings

You have several ways to determine the buckling factors. The first option is to leave everything by default and let SCIA calculate the buckling factors. A second option is to assign buckling groups to members to guide the determination of the buckling factor.

6.2.1. Default buckling calculation

When you use this method only one adjustment needs to be made. In the steel setup you need to set the default sway type.



With these settings it is determined if the structure is sway or not (braced or not) around the strong (y-y) and weak (z-z) axis of the profiles in the structure. Depending on this setting a different formula is used to determine the buckling factor:

- for a non-sway structure:

$$k = \frac{(\rho_1\rho_2+5\rho_1+5\rho_2+24)(\rho_1\rho_2+4\rho_1+4\rho_2+12)2}{(2\rho_1\rho_2+11\rho_1+5\rho_2+24)(2\rho_1\rho_2+5\rho_1+11\rho_2+24)}$$

- for a sway structure:

$$k = x \sqrt{\frac{\pi^2}{\rho_1 x} + 4}$$

- with
- k the buckling factor
 - L the system length
 - E the modulus of Young
 - I the moment of inertia
 - C_i the stiffness in node i
 - M_i the moment in node i
 - ϕ_i the rotation in node i

$$x = \frac{4\rho_1\rho_2 + \pi^2\rho_1}{\pi^2(\rho_1 + \rho_2) + 8\rho_1\rho_2}$$

$$\rho_i = \frac{C_i L}{EI}$$

$$C_i = \frac{M_i}{\phi_i}$$

The values for M_i and ϕ_i are approximately determined by the internal forces and the deformations, calculated by load cases which generate deformation forms, having an affinity with the buckling form. So, when performing a linear calculation, in the background 2 additional load cases are calculated, just to calculate the buckling factors for the elements.

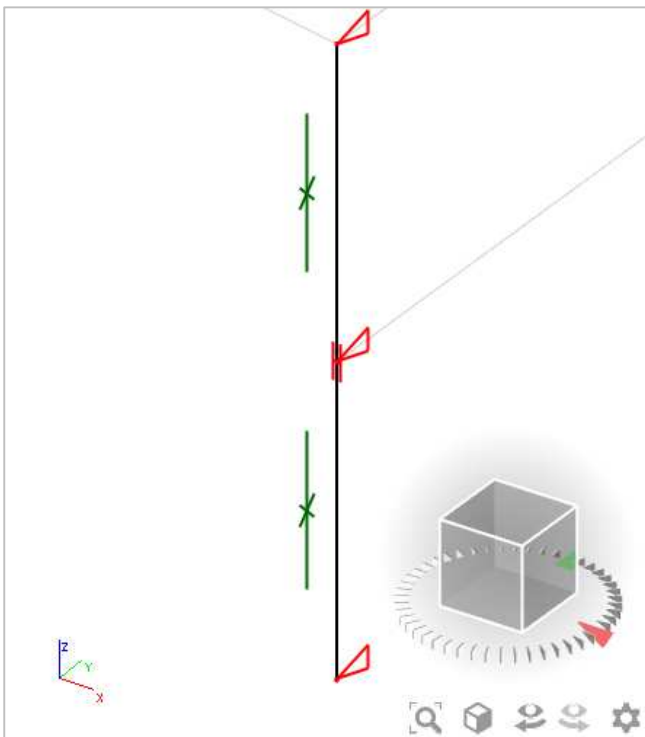
This calculation is automatically done when calculating the construction linearly. **So when calculating nonlinear, the user should also perform a linear calculation otherwise no buckling factors are calculated and no steel code check can be performed.**

The following load cases are considered in the linear calculation for the calculation of the buckling factors:

- load case 1:
 - on the beams, local distributed loads $q_y = 1 \text{ N/m}$ and $q_z = -100 \text{ N/m}$ are used;
 - on the columns, global distributed loads $Q_x = 10000 \text{ N/m}$ and $Q_y = 10000 \text{ N/m}$ are used.
- load case 2:
 - on the beams, local distributed loads $q_y = -1 \text{ N/m}$ and $q_z = -100 \text{ N/m}$ are used;
 - on the columns, global distributed loads $Q_x = -10000 \text{ N/m}$ and $Q_y = -10000 \text{ N/m}$ are used.

The used approach gives good results for frame structures with **perpendicular rigid or semi-rigid beam connections**. **For other cases, you have to evaluate the presented buckling ratios.**

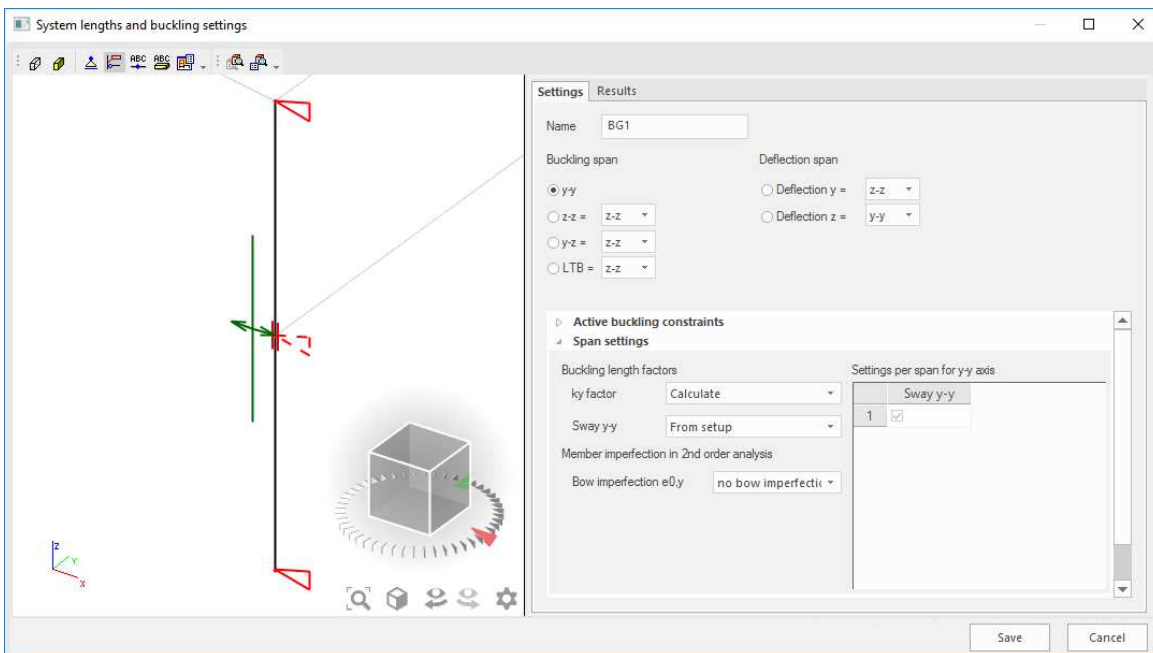
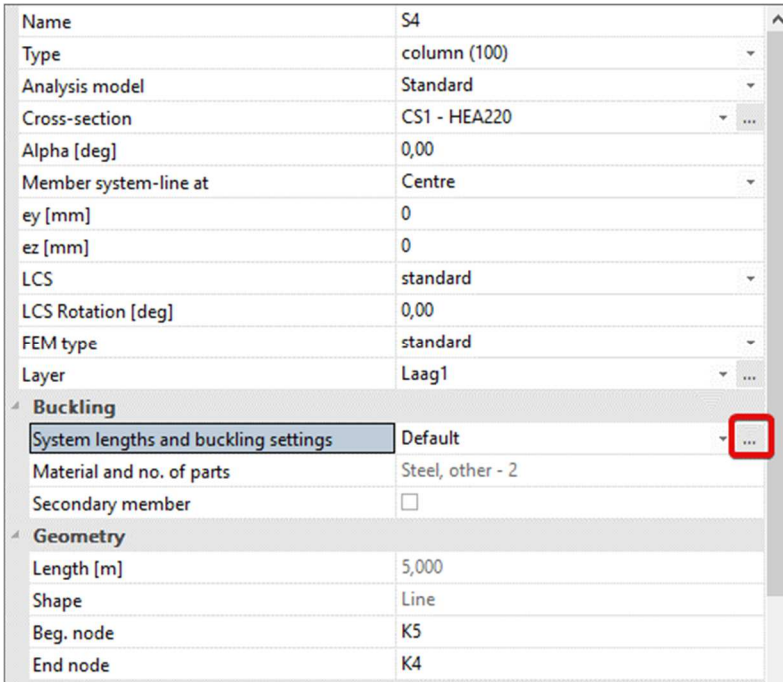
The system length of every buckling system is automatically determined by SCIA. Every element that lies in the same line is one buckling system. The buckling system can be split up in multiple systems when there is a support or a member perpendicular on the member in the direction of the strong or weak axis. The buckling system is only shortened for the relevant axis.



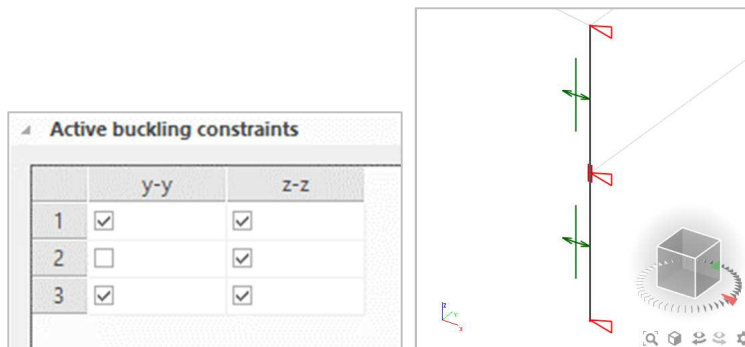
6.2.2. Assign buckling groups

To have more influence on the determination of the buckling factors, you can assign buckling groups to members. You can use the same buckling group for members that have the same buckling system (same length and buckling supports).

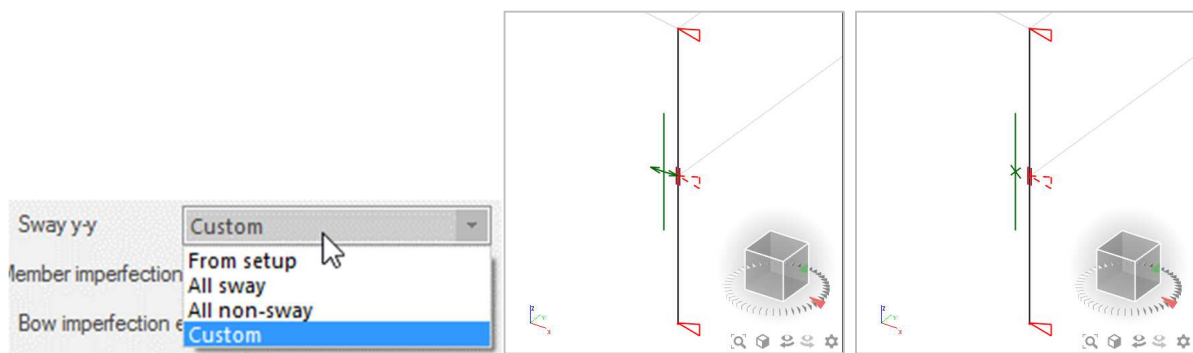
To create a buckling group for a member, select the member and press on the three dots next to the **'System lengths and buckling settings'** property in the properties window.



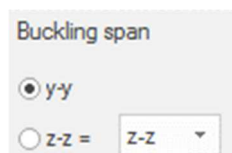
- **Buckling constraints:** you can set the buckling constraints by clicking on the triangles or by opening the active buckling constraints and checking/unchecking the boxes. Adding a constraint shortens the buckling system.



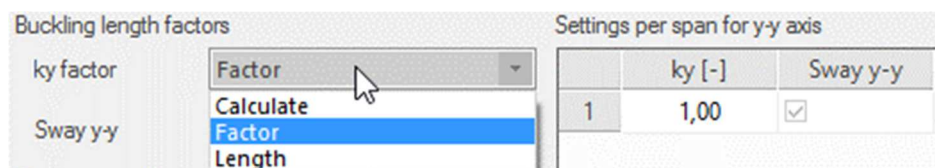
- **Sway setting:** you can determine if the structure is sway or not along the active axis by clicking on the green line or by changing the 'Sway' setting. A cross means non-sway and a double arrow means sway.



- **Buckling span:** this setting determines what axis you are changing (strong or weak).



- **k-factor:** the k-factor can be calculated by SCIA Engineer as explained above, can be set by you or the buckling length can be set by you. When you choose 'factor' or 'length' the value can be changed in the table next to the setting.



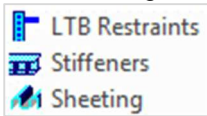
After setting the buckling group you need to press 'Save' to apply it to the member. The property now changed in the properties window. You can assign it to other members with the same buckling system. The buckling group overwrites the settings for buckling (default sway) in the steel setup.



6.3. Member check data

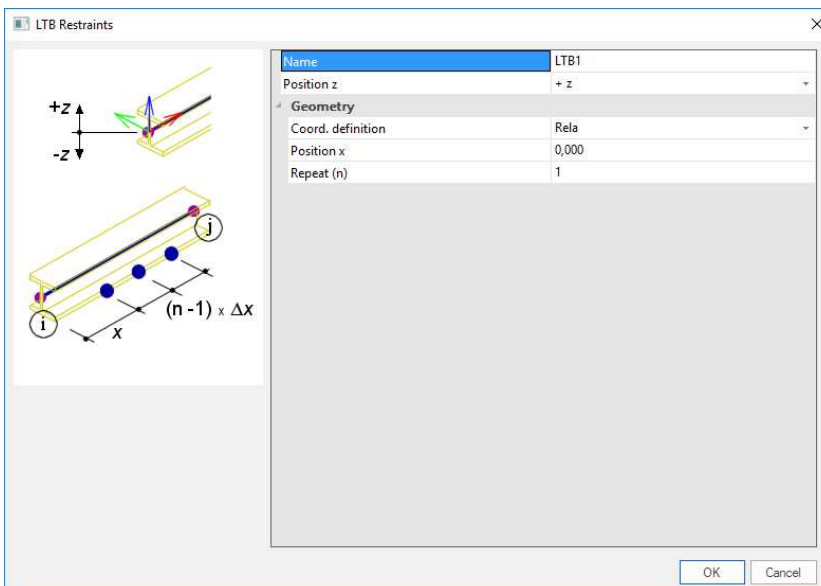
There are some effects that cannot be taken into account in the model that have an influence on the check. You can take this into account by adding 'Member check data'. This is data that has only an influence on the check. These elements are not 'physically' added to the model and have no influence on the internal forces.

The following member check data will be discussed in this course:



LTB Restraints

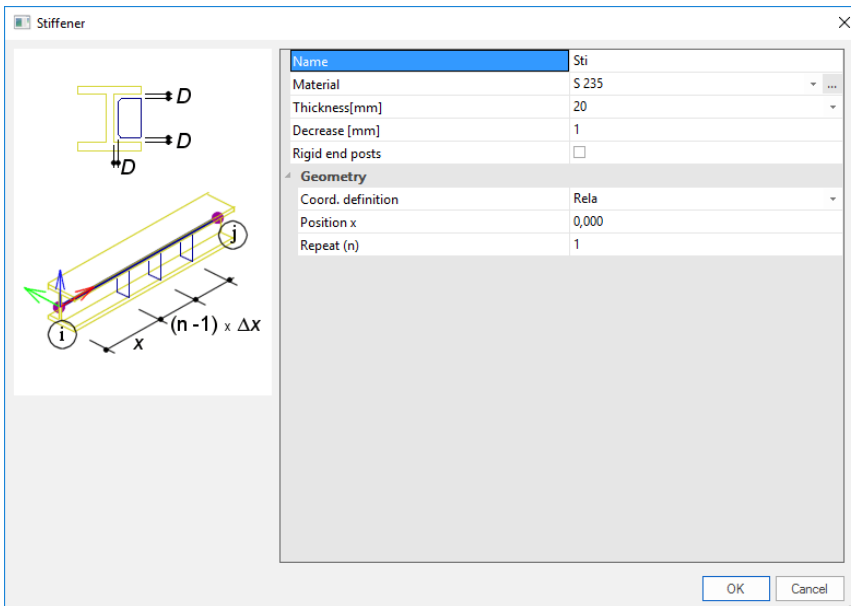
In reality, the LTB (Lateral Torsional Buckling) length can be shorter than determined by SCIA. This is because members are connected by their centre lines or because some elements are not modelled. A LTB restraint is model data added to a beam that changes the LTB length of this beam. The new LTB length is the distance between the added LTB restraints.



You need to set the position (top or bottom of the member) and the location of the restraints.

Stiffeners

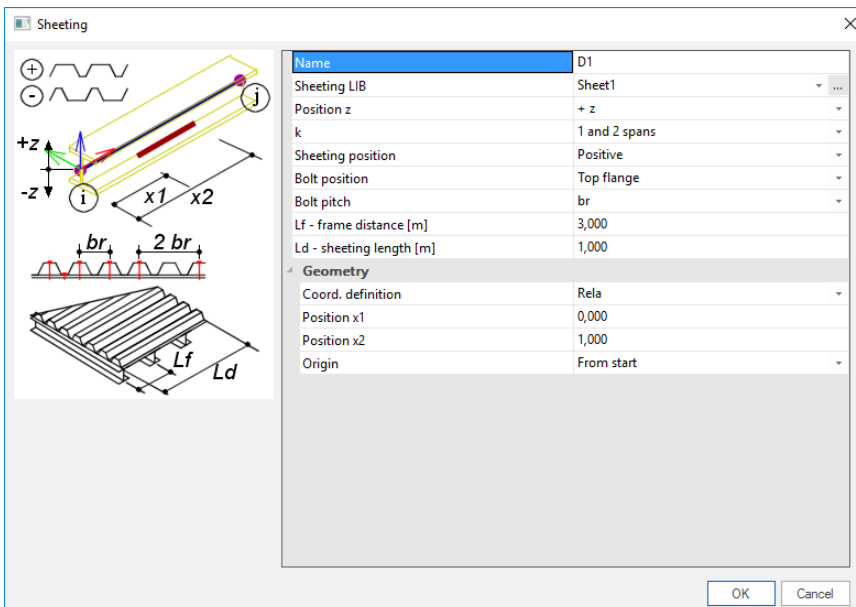
Stiffeners increase the shear area of a section. This is advantageous for the shear check. This model data is added because it is not possible to 'physically' model the stiffener. This is because a member (beam/column) is a 1D member and thus has no surface.



To add a stiffener you need to set its geometry, material and position.

Sheeting

Usually the sheeting is not modelled. This is because the assumption is made that for a steel hall the complete load is transferred to the 1D structure. To take the stiffness of the sheeting + the effect on the calculation of M_{cr} into account without changing the standard assumption, this model data can be added.



You need to set the parameters of the sheeting and its location.

NOTE: you can find more information about the member check data in the steel manual.

6.4. ULS Check

You can find the ULS check in the 'Steel' menu **after** calculating the project. Graphically it gives you the highest unity check (from all checks performed) per section of a member. You can perform a section check and a stability check. All checks are performed according to the Eurocode. Next to the graphic output you can also open the preview to see more detailed results.

6.4.1. Graphic output

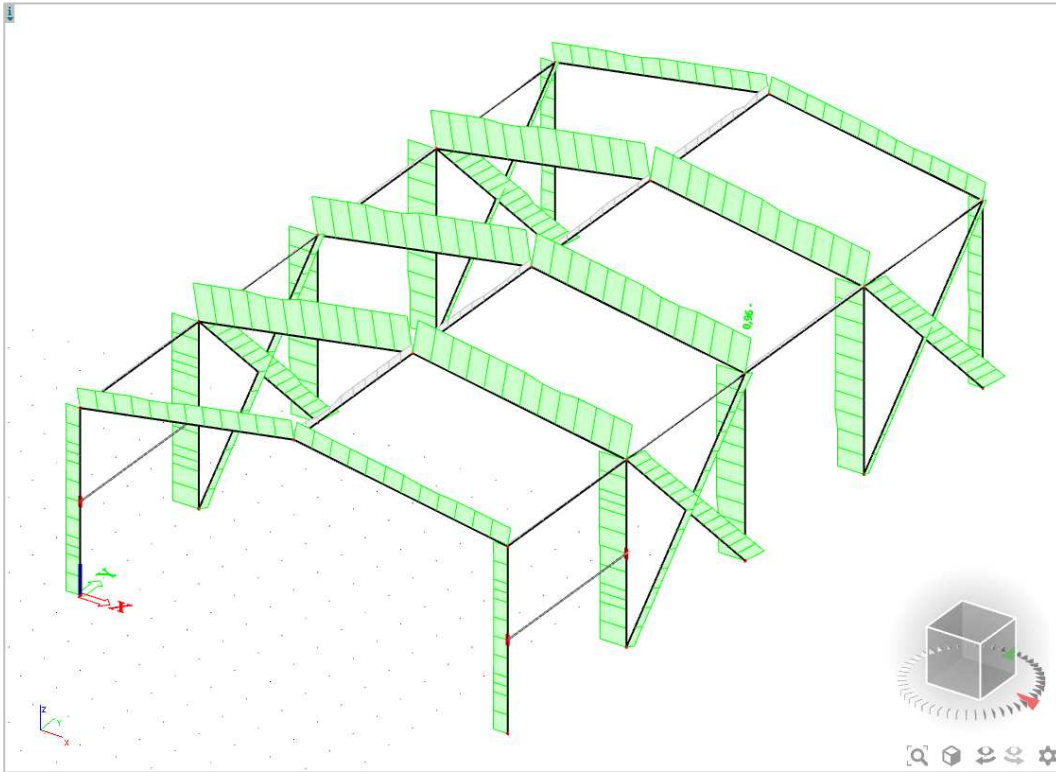
To view the graphic output you need to set the properties window correctly. Most of the settings are the same as in the 'Results' menu.

Name	EC-EN 1993 Steel check ULS
Selection	
Type of selection	All
Filter	No
Results in sections	All
Result case	
Type of load	Combinations
Combination	EN-UGT
Extreme 1D	
Extreme 1D	Global
Type of values	Overall Unity Check
Values	Overall check
Output settings	
Output	Brief
Print combination key	<input checked="" type="checkbox"/>
Drawing Setup 1D	
Errors, warnings and notes settings	

- **Type of values:** you can choose between 'overall unity check', 'section check' or 'stability check'.
- **Values:** you can view the overall check (from the setting above) or a single check.

Type of values	Section check
Values	Section Class
Output settings	
Output	Section Class
Print combination key	Tension
Drawing Setup 1D	
Errors, warnings and notes settings	
	Compression
	Bending y
	Bending z
	Shear y
	Shear z
	Torsion
	Combined Shear y Torsion
	Combined Shear z Torsion
	Web Crippling
	Bending and Reaction
	Combined Axial, Bending, Shear Torsion
	Chord as beam
	Purlin Resistance

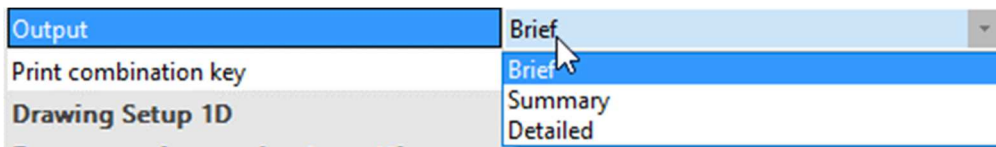
After setting the properties window, press 'Refresh' on the bottom of the properties window.



6.4.2. Preview

After setting the properties window and refresh you can press the preview button to see the output generated by the engineering report. This contains more detailed tables.

You have three output levels: '**Brief**', '**Summary**' and '**Detailed**':



When the detailed output is selected, you can set this output to view tables with the calculated values, to view the formulas or to view both.



Below you can see an example of the detailed output with tables and formulas:

Compression check
According to EN 1993-1-1 article 6.2.4 and formula (6.9)

A	5,3800e-03	m ²
N _{c,Rd}	1264,30	kN
Unity check	0,02	-

$$N_{c,Rd} = \frac{A \times f_y}{\gamma_{M0}} = \frac{5,3800 \cdot 10^{-3} [m^2] \times 235,0 [MPa]}{1,00} = 1264,30 [kN] \quad (EC3-1-1 : 6.10)$$

$$\text{Unity check} = \frac{|N_{Ed}|}{N_{c,Rd}} = \frac{|-24,17 [kN]|}{1264,30 [kN]} = 0,02 \leq 1,00 \quad (EC3-1-1 : 6.9)$$

Bending moment check for M_y
According to EN 1993-1-1 article 6.2.5 and formula (6.12),(6.13)

W _{pl,y}	5,7500e-04	m ³
M _{pl,y,Rd}	135,13	kNm
Unity check	0,61	-

$$M_{pl,y,Rd} = \frac{W_{pl,y} \times f_y}{\gamma_{M0}} = \frac{5,7500 \cdot 10^{-4} [m^3] \times 235,0 [MPa]}{1,00} = 135,13 [kNm] \quad (EC3-1-1 : 6.13)$$

$$\text{Unity check} = \frac{|M_{y,Ed}|}{M_{pl,y,Rd}} = \frac{|-82,71 [kNm]|}{135,13 [kNm]} = 0,61 \leq 1,00 \quad (EC3-1-1 : 6.12)$$

6.4.3. Table output

Lastly you can view the results in the table output. When you double click on a row in the table the preview of this row is previewed. In the top you can set the output level (summary or detailed).

Refresh	>>>
New combination from Combination key	>>>
Autodesign	>>>
Table results	>>>
Preview	>>>

Table results

Summary

EC-EN 1993 Steel check ULS; Linear calculation; Combination: EN-UGT; Coordinate system: Principal; Extreme ID: Section; Selection: All

	Name	dx [m]	Case	Cross-section	Material	UCOverall [-]	UCSec [-]	UCStab [-]
1	S1	0,000	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,03	0,38
2	S1	0,500	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,03	0,38
3	S1	1,000	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,07	0,38
4	S1	1,250-	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,08	0,38
5	S1	1,250+	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,08	0,38
6	S1	1,500	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,10	0,38
7	S1	2,000	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,13	0,38
8	S1	2,500-	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,17	0,38
9	S1	2,500+	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,17	0,38
10	S1	3,000	EN-UGT/1	CS1 - HEA220	S 235	0,38	0,20	0,38

Overall Unity Check List of Combination keys

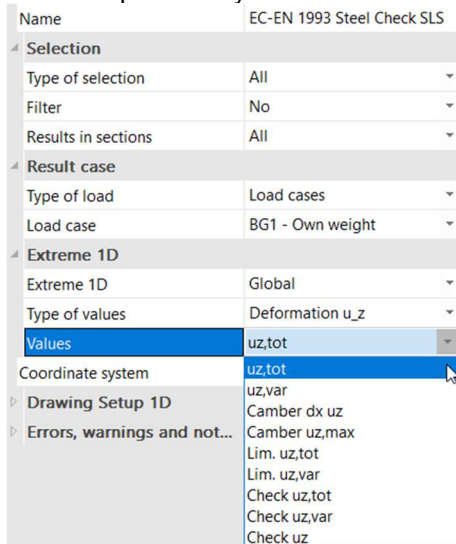
EC-EN 1993 Steel check ULS

6.5. SLS Check

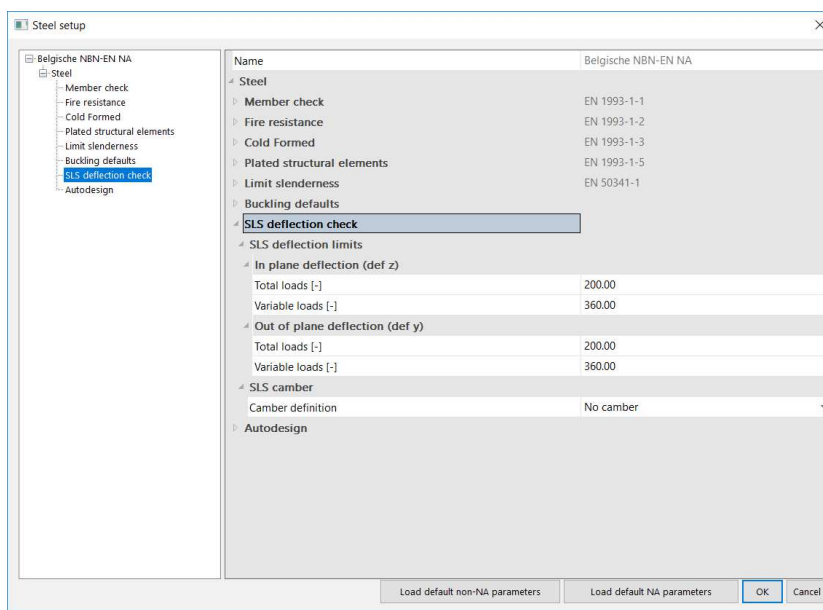
The SLS check can be found in the 'Steel' menu **after** calculating the project. You can choose between the overall unity check or a specific value for Y or Z direction.



For example when you choose for 'Deformation u_z', following values are available:



- **u,tot** or **u,var**: the relative displacement of each member for the total load or for the variable part of the load.
- **Camber**: the camber over the length of the element (dx uz) or the maximum camber over the entire element (uz,max).
- **Lim. u,tot** or **Lim u,var**: the limit of the relative displacement L/x for the total load or for the variable part of the load. You can set x-value in the steel setup or on element level in the 'system lengths and buckling settings'. You can set the value for the camber there as well.



- **Check**: the relative displacement is compared with the limit. You can check the total displacement or the variable part of the displacement or you can perform an overall check.

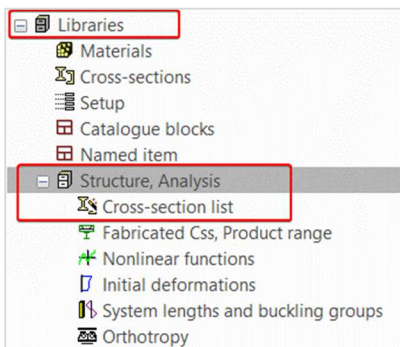
6.6. General autodesign

You can access the general 'Autodesign' function in the main tree under 'Calculation, mesh'.

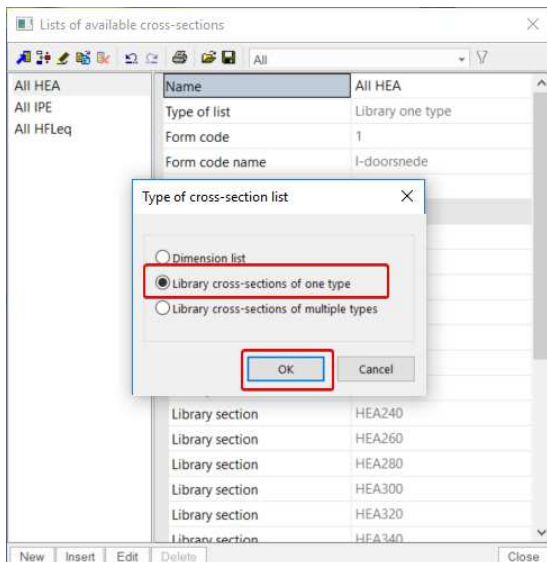


You can use the 'Autodesign' function to automatically receive an optimal unity check for the cross-sections which are used in your project. You can perform a single autodesign by following the steps below.

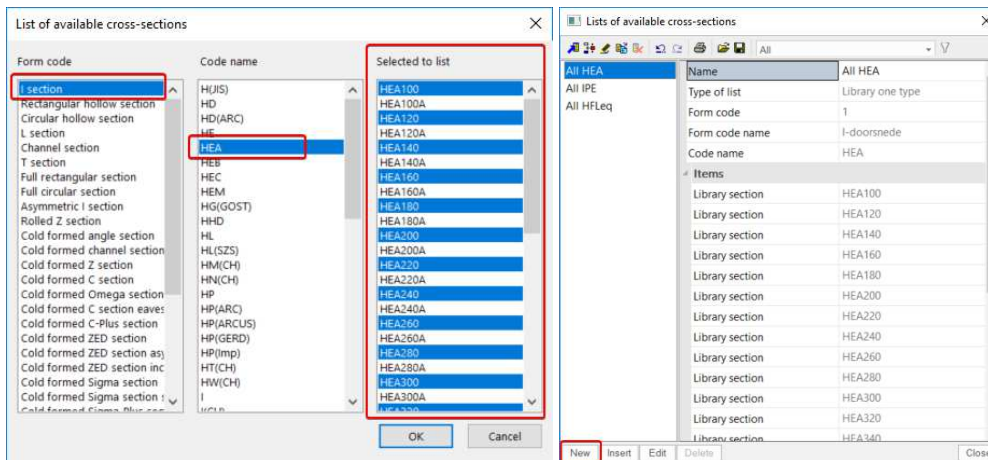
The first step of the autodesign you create multiple cross-section lists. By applying this cross-section list, the software is forced to use the steel profiles which you have added to these lists and we can avoid the use of non-standard profiles such as for example an IPE400O or a HEA260A section. You can find the function 'Cross-section list' in the main tree → 'Libraries' → 'Structure, Analysis'. You can access it as well in the toolbar.



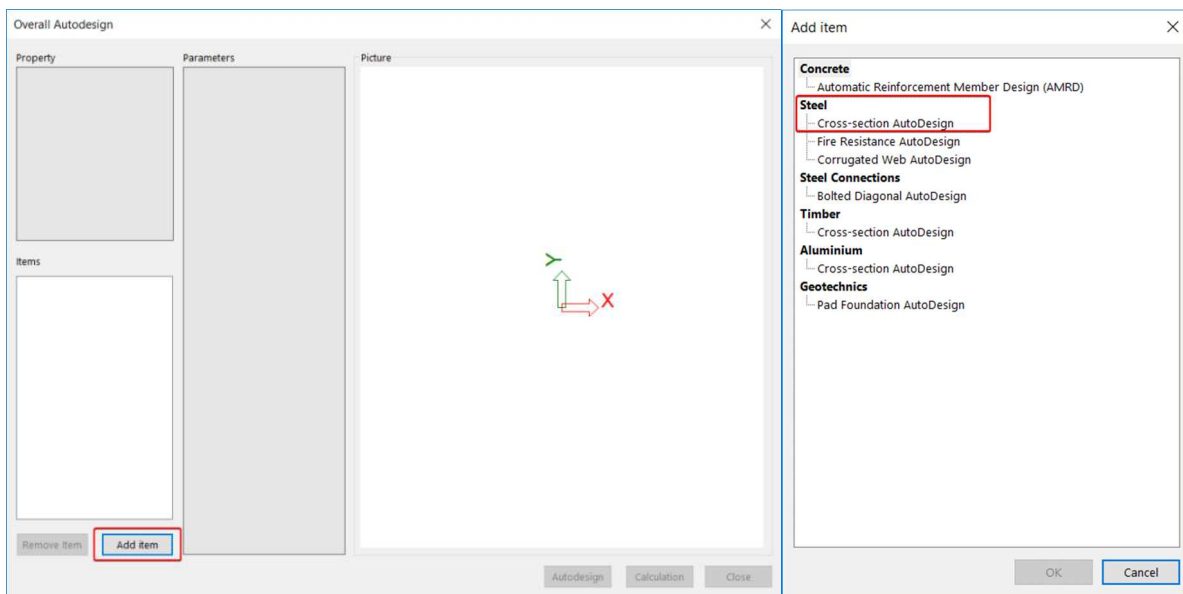
After you have selected this function, the cross-section list manager will open, and you can create a new list by selecting the button 'New'. Afterwards you can define the type of cross-section list, for example a cross-section list with only one type of steel profiles or a list with multiple types of profiles. In this example the option '**Library cross-sections of one type**' will be selected.



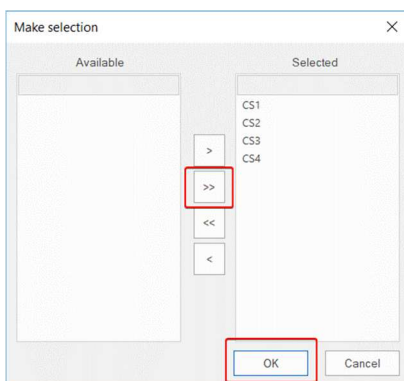
After you confirm with 'OK', the profile library will open and you can select the cross-sections which you want to add to the list. In the following picture, this will be executed for HEA-profiles. When you have defined this cross-section list, it will be added and be accessible in the cross-section list manager. You can create multiple cross-sections lists in one project.



When you select the function 'Autodesign' in the menu 'Calculation, mesh', the autodesign manager will open and you can define a new one. After you clicked on 'New' and on 'Add item', you can select the option 'Cross-section AutoDesign' in the tab 'Steel'.



In the following step you should select and confirm the cross-section(s) which should be optimized.



In the following window ‘Overall Autodesign’, you can define the parameters for the design.

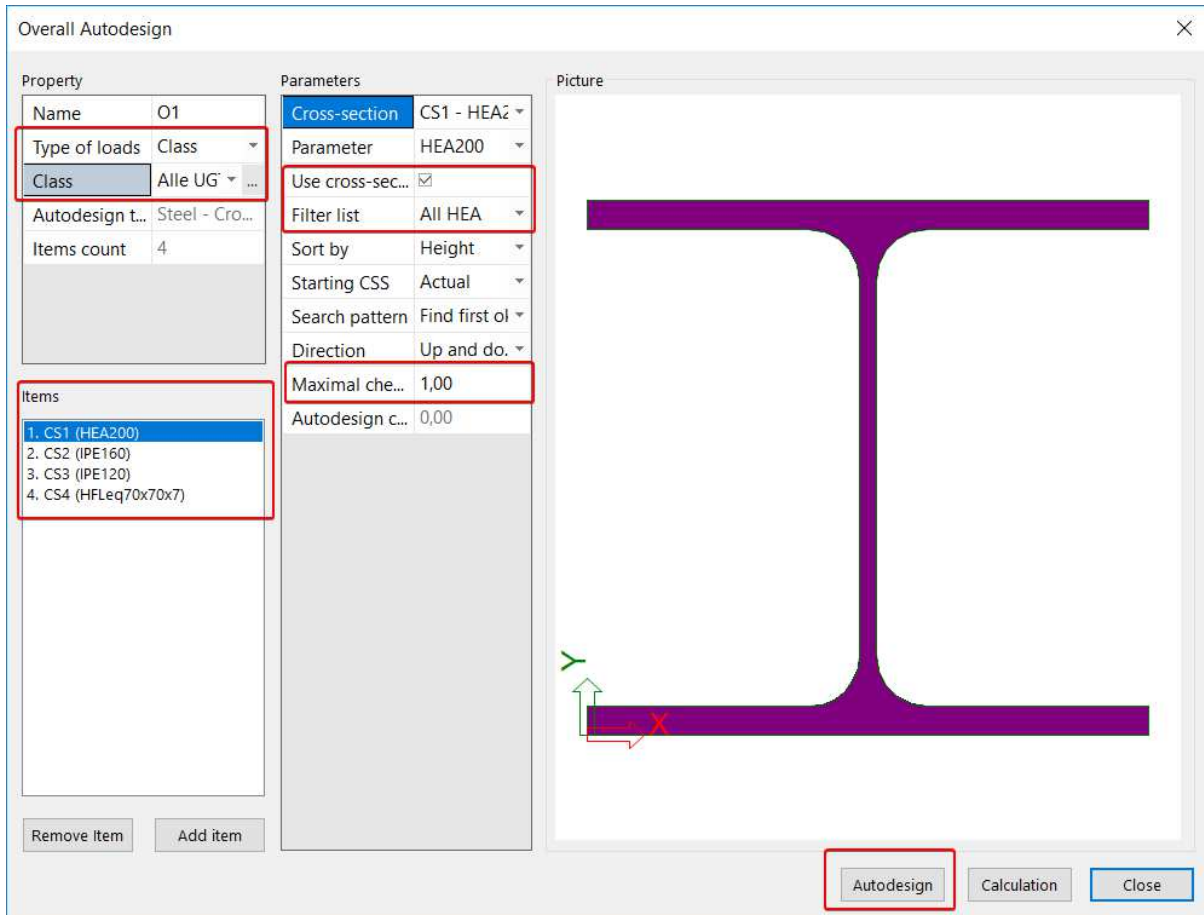
It is important that you select the correct type of loads for each cross-section in the autodesign. You can perform the autodesign for a load case, for a combination or for a result class.

You can select the defined cross-section lists as well when you activate the option ‘Use cross-section list’ and then select the list in the tab ‘Filter list’.

You should define the maximum value for the optimized unity check with the option ‘Maximal check’.

You can modify these parameters for each cross-section that is mentioned in the window ‘Items’.

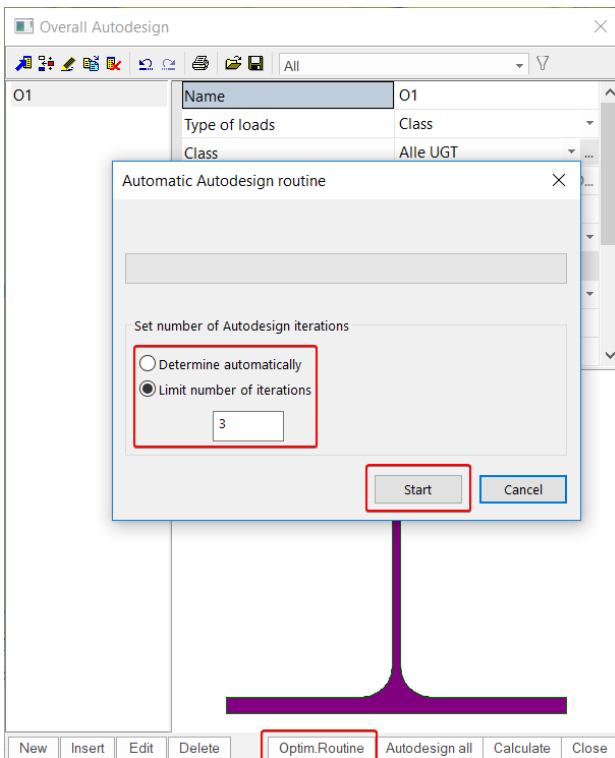
The final step in this window is that you activate the command ‘Autodesign’ at the bottom of this window.



As soon as you have executed this final command, a summary of the autodesign will be displayed. This table will provide you the optimized profiles as well as the optimized unity check.

Cross-section	Parameter	Sort by	Filter list	Original cross-section	Autodesign of cross-section	Autodesign check [-]
CS1 - HEA260	HEA260	Height	All HEA	CS1 - HEA200	CS1 - HEA260	0,94
CS2 - IPE400	IPE400	Height	All IPE	CS2 - IPE160	CS2 - IPE400	0,88
CS3 - IPE160	IPE160	Height	All IPE	CS3 - IPE120	CS3 - IPE160	0,73
CS4 - HFLeq80x80x8	HFLeq80x80x8	Height	All HFLeq	CS4 - HFLeq70x70x7	CS4 - HFLeq80x80x8	0,83

Within this command, you can perform an iterative autodesign as well. You can execute this by closing the tab **Overall Autodesign** and selecting the command **Optim. Routine** in the autodesign manager. After you have selected this option, you can define the iterations of the design by a limit number of iterations or it can be determined automatically. In this example the number of iterations will be set to 3. The autodesign of the profiles will be executed after you have pressed on **Start**.



As soon as the iterations are finished, you will get an overview of all the routine steps as well as the optimized profiles and unity checks.

1. Routine step: 1

1.1. O1

Cross-section	Parameter	Sort by	Filter list	Original cross-section	Autodesign of cross-section	Autodesign check []
CS1 - HEA220	HEA220	Height	All HEA	CS1 - HEA200	CS1 - HEA260	0,93
CS2 - IPE400	IPE400	Height	All IPE	CS2 - IPE160	CS2 - IPE400	0,87
CS3 - IPE80	IPE80	Height	All IPE	CS3 - IPE100	CS3 - IPE140	0,70
CS4 - HFLeq60x60x8	HFLeq60x60x8	Height	All HFLeq	CS4 - HFLeq70x70x7	CS4 - HFLeq80x80x8	0,88

2. Routine step: 2

Cross-section	Parameter	Sort by	Filter list	Original cross-section	Autodesign of cross-section	Autodesign check []
CS1 - HEA220	HEA220	Height	All HEA	CS1 - HEA260	CS1 - HEA240	0,74
CS2 - IPE400	IPE400	Height	All IPE	CS2 - IPE400	CS2 - IPE400	0,79
CS3 - IPE80	IPE80	Height	All IPE	CS3 - IPE140	CS3 - IPE100	0,79
CS4 - HFLeq60x60x8	HFLeq60x60x8	Height	All HFLeq	CS4 - HFLeq80x80x8	CS4 - HFLeq70x70x7	0,89

3. Routine step: 3

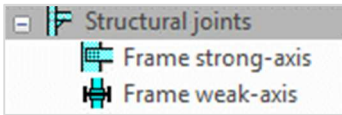
Cross-section	Parameter	Sort by	Filter list	Original cross-section	Autodesign of cross-section	Autodesign check []
CS1 - HEA220	HEA220	Height	All HEA	CS1 - HEA240	CS1 - HEA220	0,90
CS2 - IPE400	IPE400	Height	All IPE	CS2 - IPE400	CS2 - IPE400	0,86
CS3 - IPE80	IPE80	Height	All IPE	CS3 - IPE100	CS3 - IPE80	0,93
CS4 - HFLeq60x60x8	HFLeq60x60x8	Height	All HFLeq	CS4 - HFLeq70x70x7	CS4 - HFLeq60x60x8	0,92

6.7. Connections

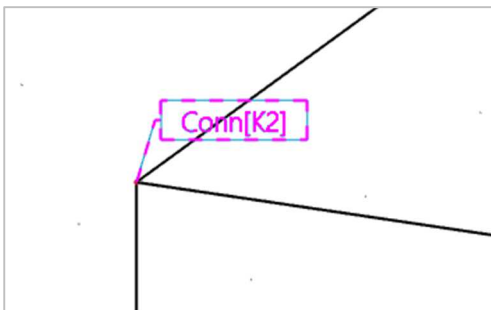
To create a connection in the 'Steel' menu you should activate the functionality 'Steel connections':



To add a connection to your model, go to the 'Steel' menu → 'Connections' → 'Structural joints'. Choose for a strong-axis connection or a weak-axis connection:



After you have selected a connection type, you should choose the node where you want to add the connection. In this example a strong-axis connection is chosen. After you have chosen the node, the following label will appear on the node.

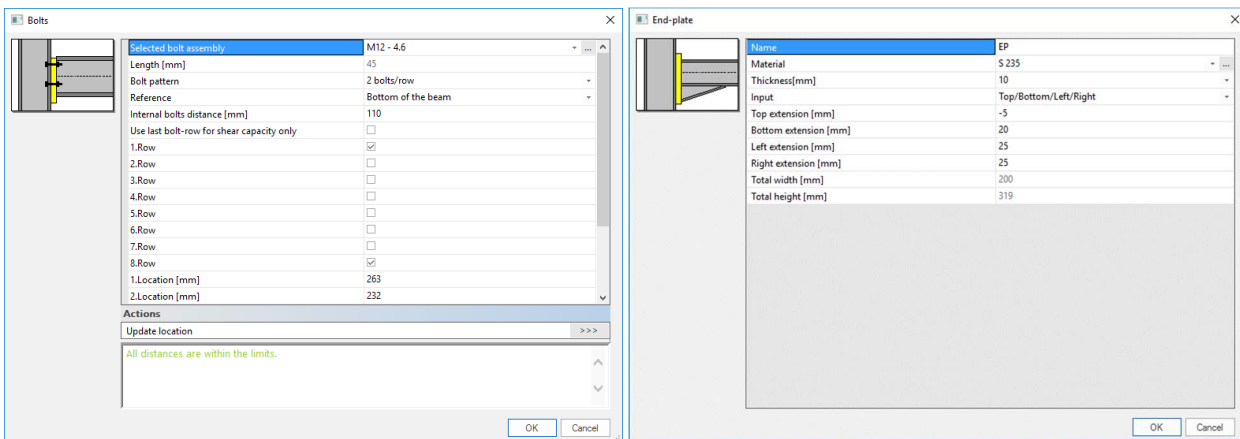


NOTE: you can create the connection only if the members have the correct type. A column should have the type 'column' and a beam the type 'beam' (see chapter 1D elements).

When you select the label, you can define the connection in the properties window.

Side -> [S2]	
Connection type	Frame bolted
End-plate	<input checked="" type="checkbox"/> ...
Backing plate	<input type="checkbox"/>
Bolts	<input checked="" type="checkbox"/> ...
Top haunch	<input type="checkbox"/>
Bottom haunch	<input type="checkbox"/>
Top stiffener	<input type="checkbox"/>
Bottom stiffener	<input type="checkbox"/>
Diagonal stiffener	<input type="checkbox"/>
Web doubler	<input type="checkbox"/>
Update stiffness	<input type="checkbox"/>
Calculation type	Internal forces
Output	Summary
Length for stiffness classification [m]	6,083
Stiffeners	
Between bolt-rows 1 2	<input type="checkbox"/>
Welds	...

To create the connection, choose the connection type and check the components of the connection. To edit the components, press on the three dots next to the component.



When you have defined the connection click on the 'Refresh' button at the bottom of the properties window. The preview window will automatically open. In the preview window all the strength checks according to the Eurocode will be performed.

.....RESULTS:.....

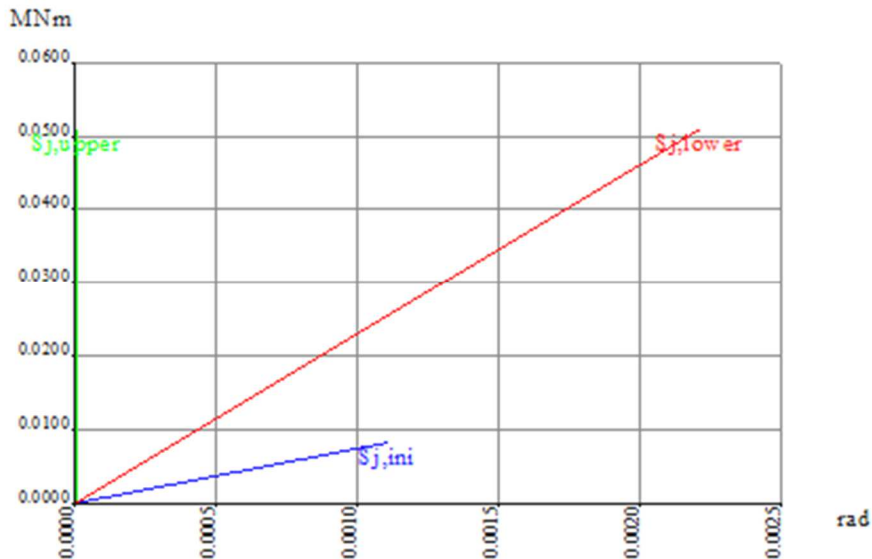
Unity checks

$M_{y,Ed}/M_{j,y,Rd}$	3.18
$M_{z,Ed}/M_{j,z,Rd}$	0.05
$N_{Ed}/N_{j,Rd}$	0.04
$V_{z,Ed}/V_{z,Rd}$	1.39
$V_{y,Ed}/V_{y,Rd}$	0.00
$V_{z,Ed}/V_{z,Rd} + V_{y,Ed}/V_{y,Rd}$	1.39
$M_{y,Ed}/M_{j,y,Rd} + M_{z,Ed}/M_{j,z,Rd}$	3.23

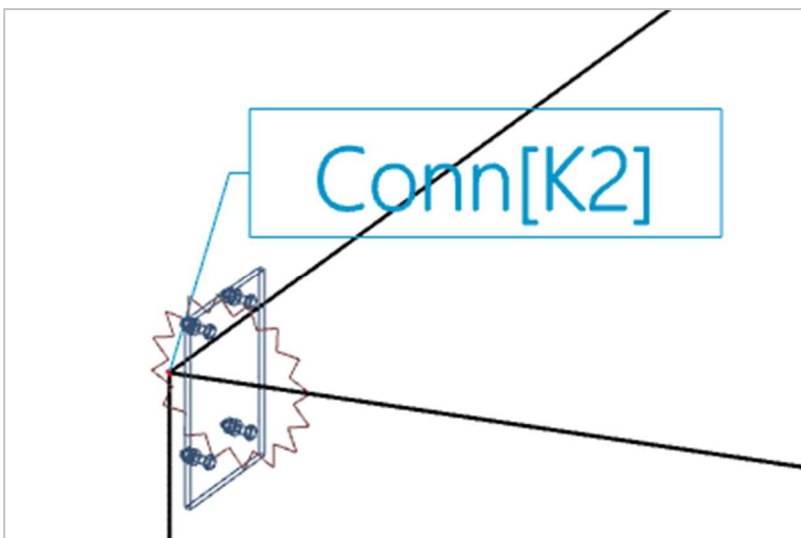
You can get a more detailed view by changing the output type in the properties window.

Output	Summary
Length for stiffness classification [m]	Brief
Stiffeners	Summary
	Detailed

Next to the strength check, a stiffness check is performed as well. In the main model a stiffness is assumed for each node. This stiffness is fully fixed or the stiffness of a hinge that is added to the node. The stiffness assumed in the project should be checked with the connection that you just have created. If the stiffness of the connection does not lay between certain boundaries (so that it can be assumed to be the same as the stiffness in the model), then the stiffness in the model should be adapted. In this example the stiffness of the connection (in blue) does not lay between the boundaries (a stiff node is assumed).



You can do this automatically by checking the option 'Update stiffness'. When you have checked this option a hinge will be added to the node after calculating the project another time. This hinge has the real stiffness of the connection.



NOTE: when the stiffness is updated the internal forces are different as well. This means that the ULS and SLS checks of the beams are different as well.

Chapter 7: Concrete design

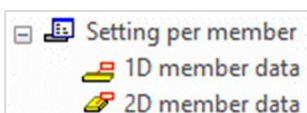
7.1. Concrete setup

You can open the concrete setup by going to the 'Concrete' menu → 'Concrete settings'.

Description	Symbol	Value	Default	Unit	Chapter	Code	Structure	Check Type
<all>	<all>	<all>	<all>	<all>	<all>	<all>	<all>	<all>
Solver setting								
General								
Limit value of unity check	Lim.check	1,0	1,0				Independent	All (Beam... Solver setti...
Creep								
Type input of creep coefficient	Type φ	Auto	Auto		Annex B.1	EN 1992-1-1	All (Beam... Solver setti...	
Default sway type								
Sway around y axis	Sway yy	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>				Independent	All (Beam... Solver setti...
Sway around z axis	Sway zz	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>				Independent	All (Beam... Solver setti...
Minimal concrete cover								
Design working life		50,00	50,00	year	4.4.1.2(5), t...	EN 1992-1-1	All (Beam... Solver setti...	
Risk of corrosion attack								
Corrosion induced by carbonation	XC3	XC3	XC3		4.4.1.2(5)	EN 1992-1-1	All (Beam... Solver setti...	
Corrosion induced by chlorides	None	None	None		4.4.1.2(5)	EN 1992-1-1	All (Beam... Solver setti...	
Corrosion induced by chlorides from sea water	None	None	None		4.4.1.2(5)	EN 1992-1-1	All (Beam... Solver setti...	
Freeze / thaw attack	None	None	None		4.4.1.2(12)	EN 1992-1-1	All (Beam... Solver setti...	
Chemical attack	None	None	None		4.4.1.2(12)	EN 1992-1-1	All (Beam... Solver setti...	
Concrete characteristics								
Type of concrete	In-situ	In-situ	In-situ		4.4.1.3(1P, 3)	EN 1992-1-1	All (Beam... Solver setti...	
Internal forces								
Absolute limit ratio for modification of internal forces	Ratio _{int,abs}	5,00	5,00	kN			Independent	1D (Bea... Solver setti...
Relative limit ratio for modification of internal forces	Ratio _{int,rel}	0,10	0,10	-			Independent	1D (Bea... Solver setti...

In this window you can set all concrete settings. This contains general concrete settings (creep, risk of corrosion attack, ...), settings for the recalculation of the internal forces and all the settings for the ULS and SLS checks. You can find a complete explanation of all these settings in the concrete manual.

You can change these settings for only one member by using the option 'Setting per member'. This option will open a window where you can change the settings and assign them to a selection.



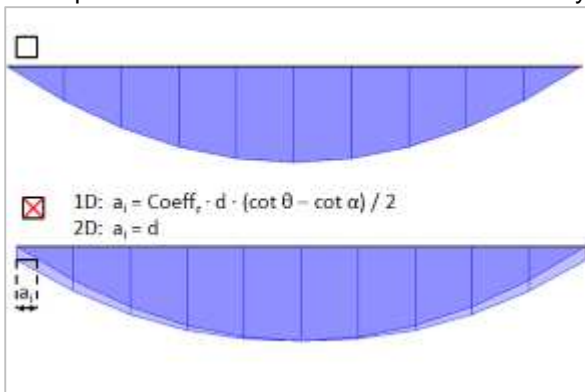
7.2. Recalculated internal forces

For the concrete design recalculated internal forces are used. Following modifications can be applied. You can find this table in the concrete settings menu.

Internal forces ULS						
Take into account additional tensile force caused by shear (shi...	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		9.2.1.3(2)	EN 1992-1-1	
Use minimum value of eccentricity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		6.1(4)	EN 1992-1-1	
Use geometric imperfection	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		5.2(5)	EN 1992-1-1	
Use second order effect	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>		5.8.8	EN 1992-1-1	
Estimation ratio of longitudinal reinforcement for recalculation in... μ_s		2,00	2,00	%	5.8.3.1	EN 1992-1-1
Shear force reduction above supports	<input type="checkbox"/>				6.2.1(8)	EN 1992-1-1
Moment reduction above supports	<input type="checkbox"/>				5.3.2.2 (4)	EN 1992-1-1

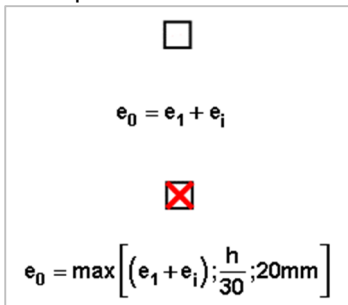
Shift rule

This option takes an extra tensile force caused by shear by shifting the moment line into account:



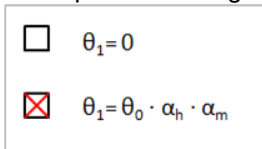
Minimum value of eccentricity

This option takes a minimum value for eccentricity into account:



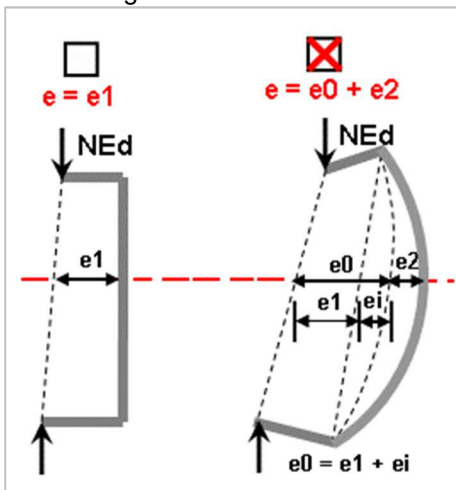
Geometric imperfection

This option adds a geometric imperfection:

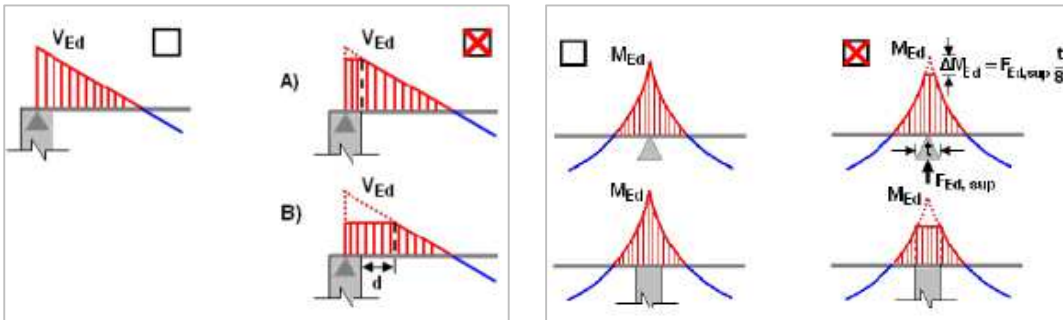


Second order effect

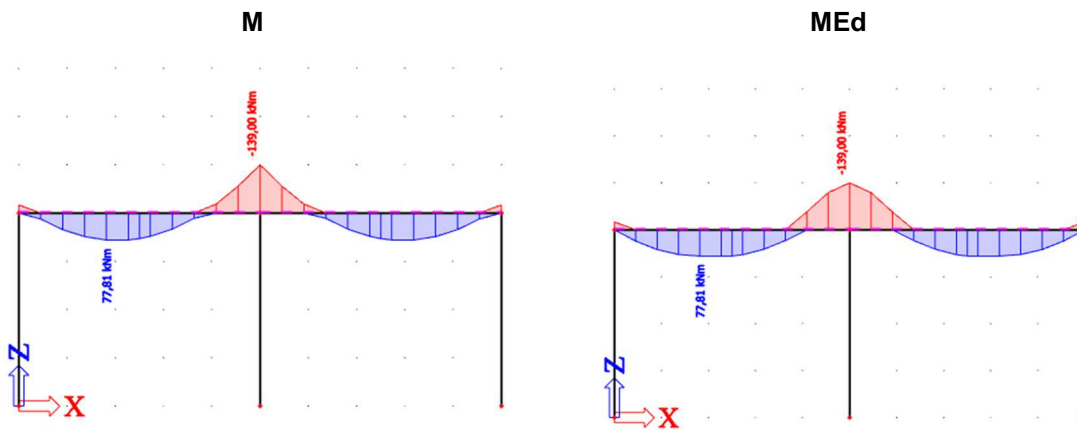
This setting takes the second order effect into account:



Shear force/Moment reduction above supports



You can view the values for the recalculated internal forces with 'Reinforcement design' → '1D Members' → 'Internal forces'. You can view the normal internal forces (N,M) as well as the recalculated internal forces (Ned, Med).



7.3. Provided reinforcement

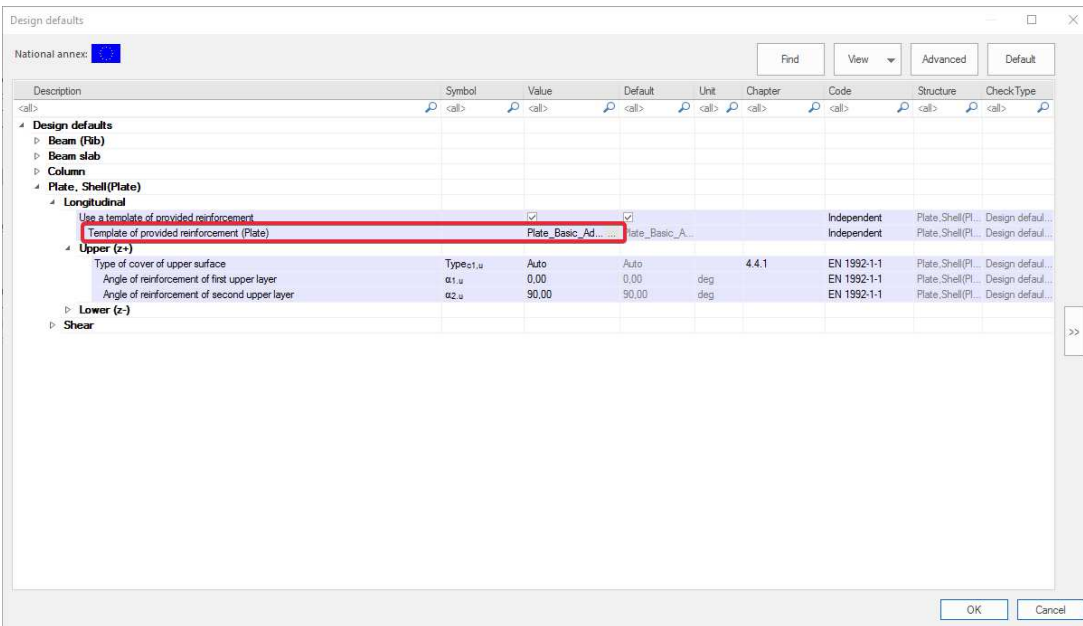
Before you calculate the theoretical reinforcement, you can add a template of reinforcement to your element(s). You can use this template to:

- compare the template with the calculated theoretical reinforcement. By doing this, you can easily see where this basic template is not sufficient;
- perform the punching design, crack width check and the code dependent deflections (**only for 2D members**).

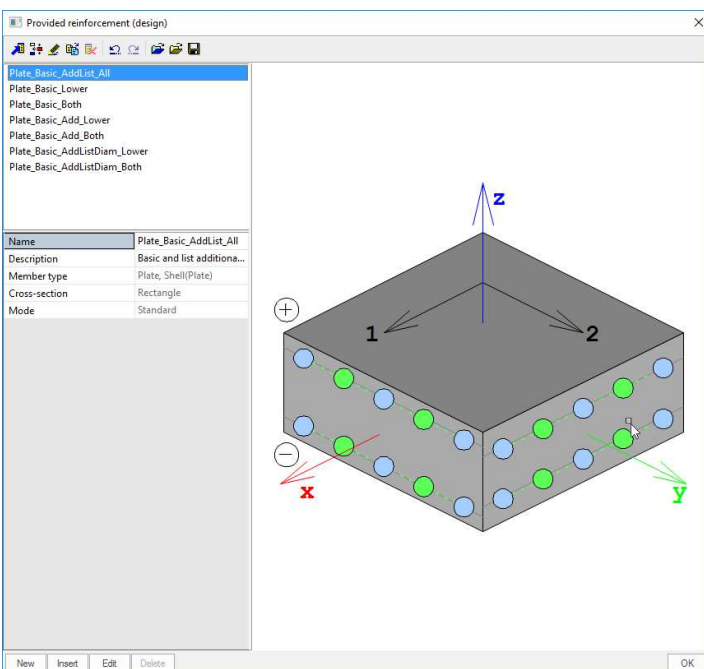
The reinforcement added by the template is called ‘**Provided reinforcement**’.

To add ‘**Provided reinforcement**’, go to ‘Concrete’ → ‘Reinforcement design’ → ‘Design defaults’

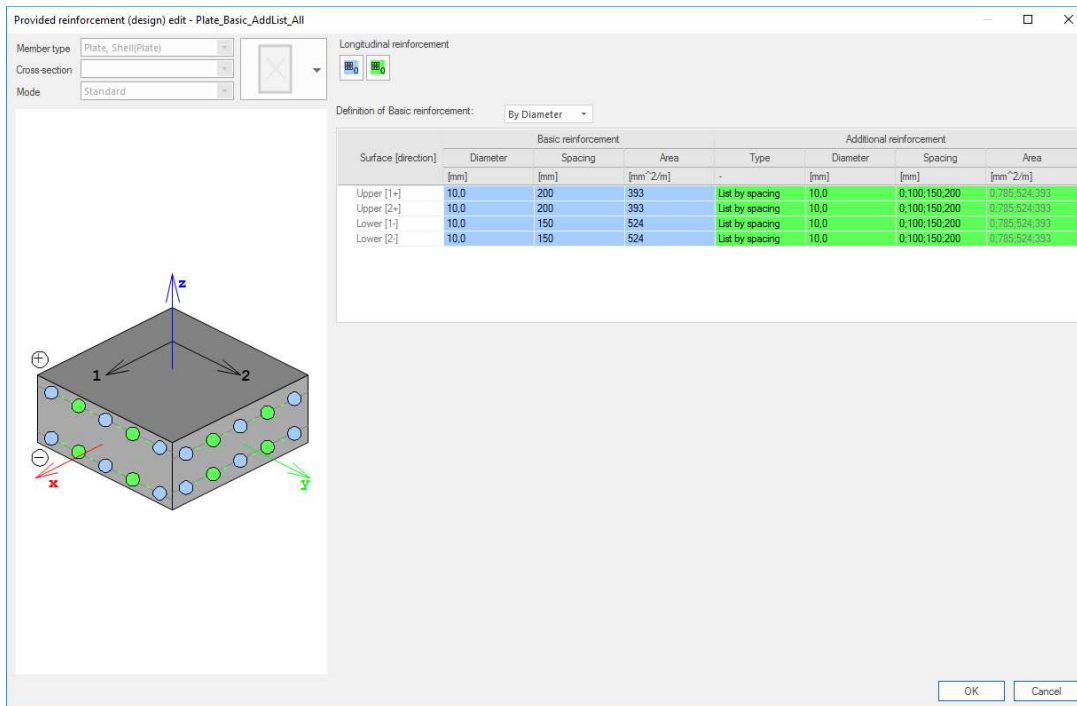
This example is for a 2D member. The workflow is the same for 1D members.



Click on the 3 dots next to the ‘Template of provided reinforcement (Plate)’. This opens a window with all the default templates.



You can select one of these templates, make a new one or edit one of the existing templates. Select the first template and click on 'Edit'.



In this window you can define the reinforcement. There are 2 types of reinforcement in templates:

- **Basic reinforcement:** this type of reinforcement is added over the entire plate.
- **Additional reinforcement:** this type of reinforcement is only added in zones where, according to the calculated theoretical reinforcement, extra reinforcement is needed. You can define a single diameter and spacing as extra reinforcement or a list of reinforcement with either various diameters or various spacings.

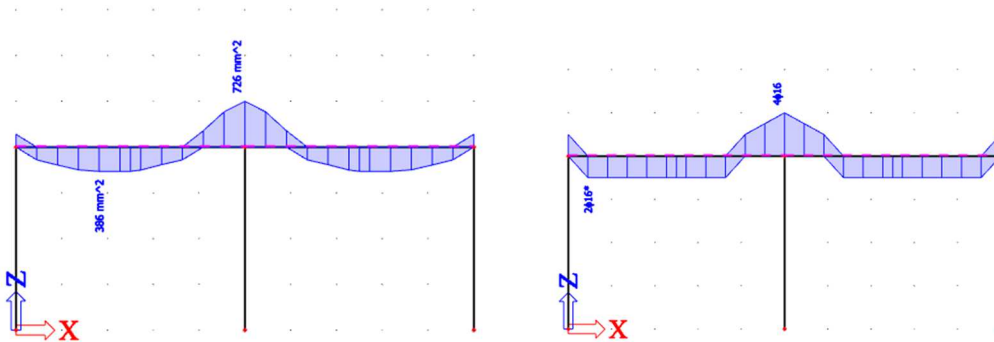
7.4. Required reinforcement

You can find the required reinforcement under 'Concrete' → 'Reinforcement design' → '1D/2D members' → 'Reinforcement design (ULS)'

7.4.1. 1D members

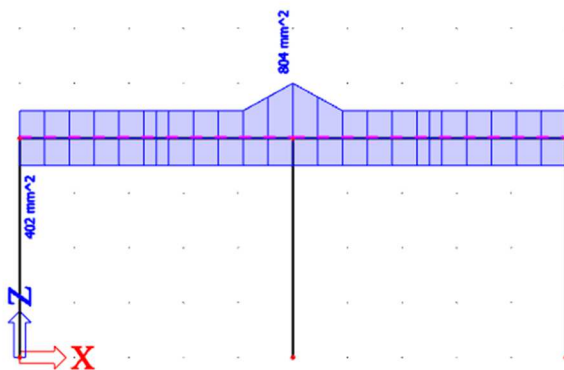
In the menu 'Reinforcement design (ULS)' you have 2 types of values:

- Required:** these values represent the theoretical reinforcement calculated by SCIA Engineer. The following required values are available: **Asreq** (amount of longitudinal reinforcement), **Aswmreq** (amount of shear reinforcement), **Asreq(ϕ)** (same as Asreq but the value is displayed in diameters) and **Aswmreq(ϕ)** (same as Aswmreq but the value is displayed in diameters).

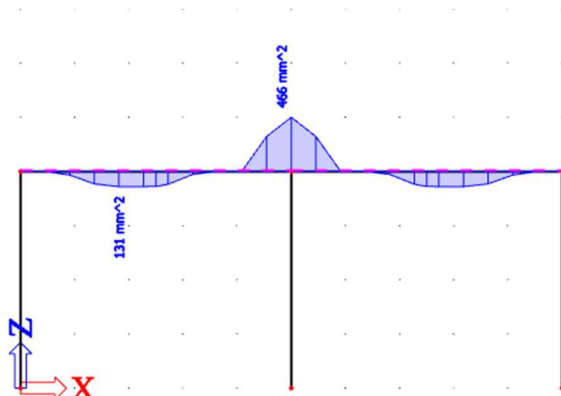


- Provided:** the provided reinforcement can be viewed in two different values as well: **As,add,req** and **As,prov**.

As,prov is the provided reinforcement defined in the template.



As,add,req is the reinforcement that is needed on top of the provided template according to the theoretical design. This means if the required value is higher than the provided value it can be seen in this view.

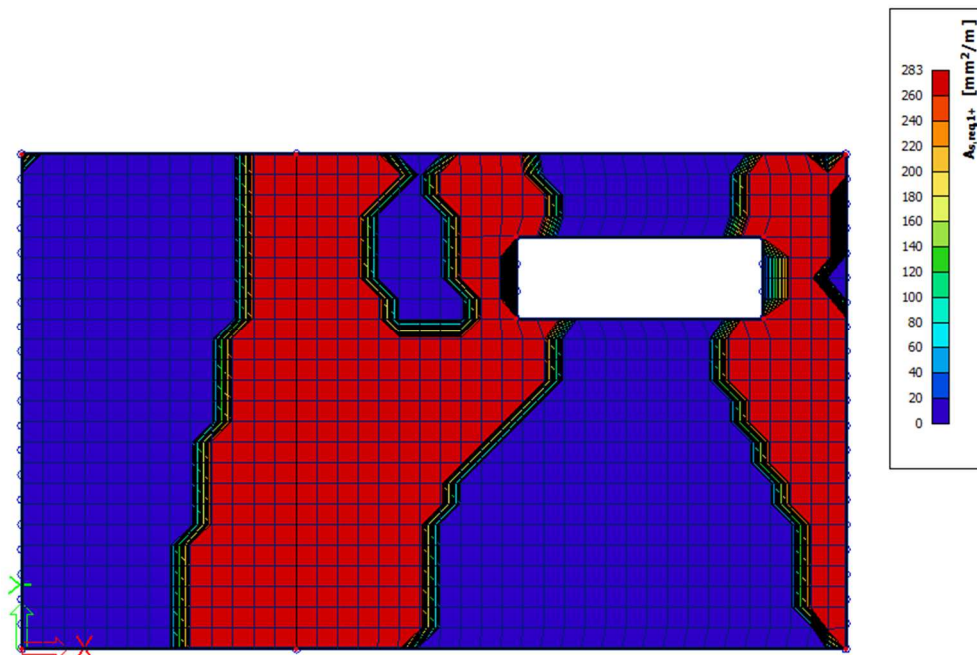


7.4.2. 2D members

In the menu 'Reinforcement design (ULS)' you have 4 types of values:

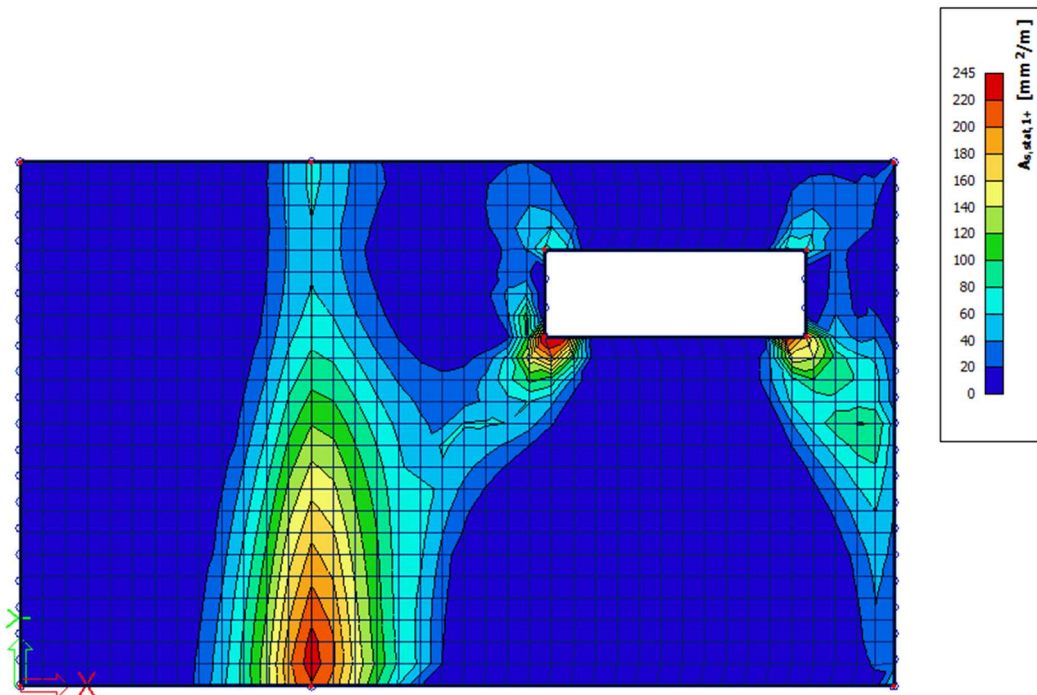
- **Required:** these values represent the theoretical reinforcement calculated by SCIA Engineer. This takes into account the detailing provisions.

▾ Plate, Shell(Plate)			<input type="checkbox"/>
▾ Longitudinal			<input type="checkbox"/>
Check min. ratio of principal reinforcement			<input checked="" type="checkbox"/>
Type of the minimum tension principal reinforcement for the upper surface			Auto
Type of the minimum tension principal reinforcement for the lower surface			Auto
Check max. ratio of principal reinforcement			<input checked="" type="checkbox"/>
Check min. transverse ratio of secondary reinforcement			<input type="checkbox"/>
Check min. bar distance			<input checked="" type="checkbox"/>
Minimal bar distance	slp.min	20	
Check max.spacing of principal longitudinal reinforcement			<input checked="" type="checkbox"/>
Check max.spacing of secondary longitudinal reinforcement			<input checked="" type="checkbox"/>
▾ Shear			<input type="checkbox"/>
Check min. ratio of shear reinforcement			<input checked="" type="checkbox"/>
Check min. thickness of member with shear reinforcement			<input checked="" type="checkbox"/>
Min. thickness of member with shear reinforcement	h _{min}	200	
Check max. spacing of shear links			<input checked="" type="checkbox"/>
Max. spacing of shear links	Coeff _{smax.p.s}	0,8	



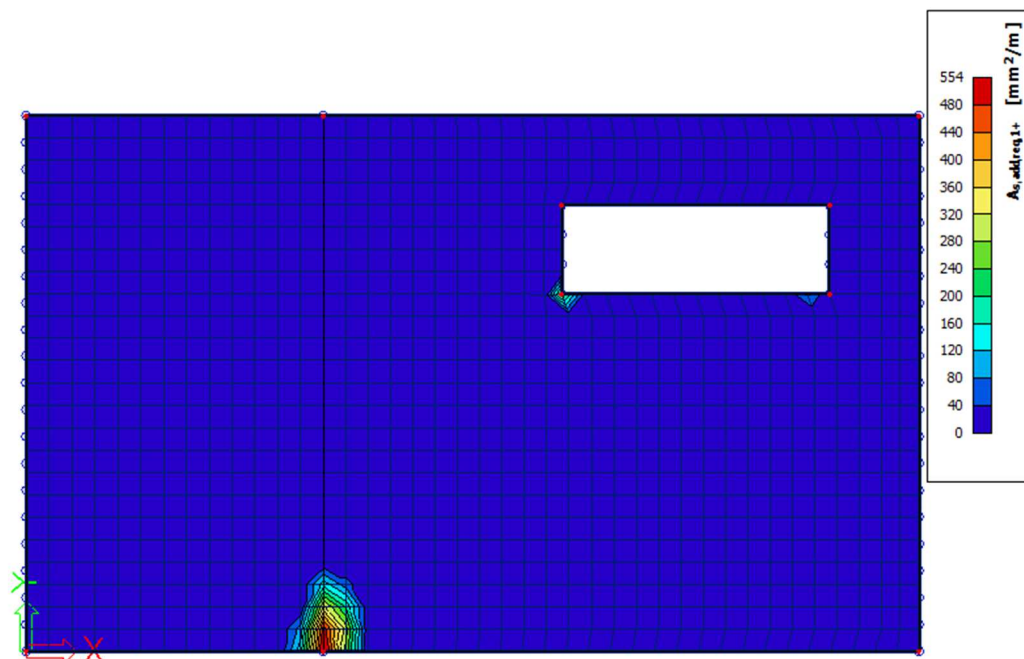
As,req1+: theoretical required reinforcement on the top side of the plate (positive Z direction) in the first reinforcement direction. Taking into account the detailing provisions.

- **Required (statically):** these values represent the theoretical reinforcement calculated by SCIA Engineer **without** the detailing provisions taken into account.



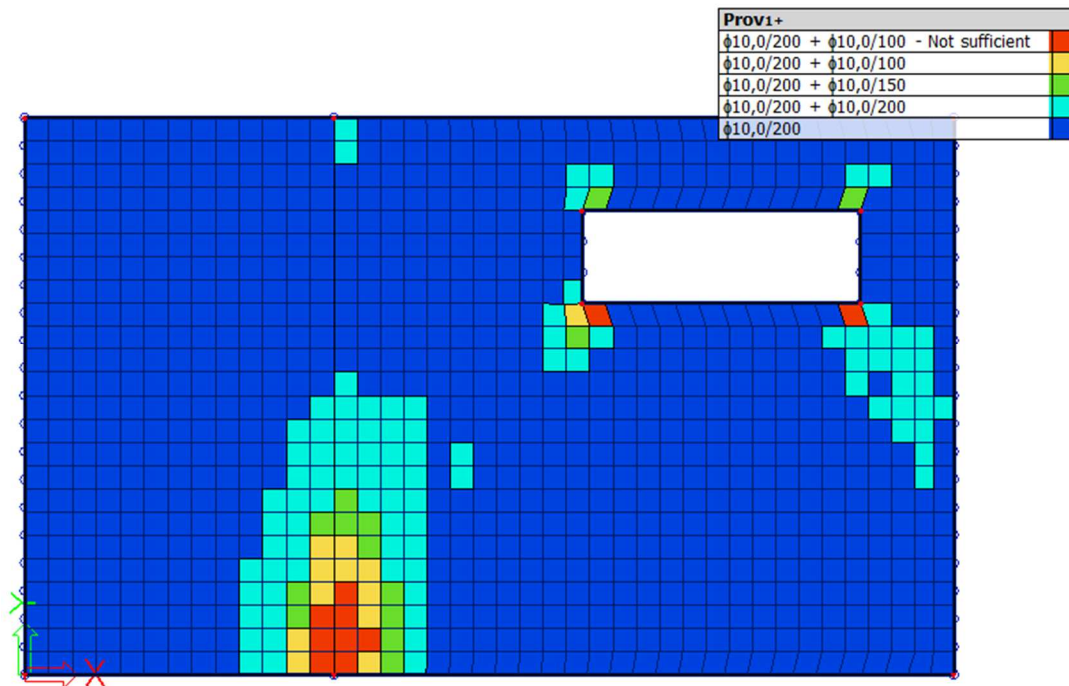
As,stat1+: theoretical required reinforcement on the top side of the plate (positive Z direction) in the first reinforcement direction. **Without** taking into account the detailing provisions.

- **Required (additional):** these values show if there is extra reinforcement needed on top of the provided reinforcement. Areas where this value is 0 are areas where no extra reinforcement is needed (compared to the provided reinforcement). Areas where these values are not 0 are areas where the provided reinforcement is not sufficient.



As,add,req1+: theoretical additional required reinforcement on top of the provided reinforcement on the top side of the plate (positive Z direction) in the first reinforcement direction.

- **Provided:** these values show you the provided reinforcement which you have defined in the templates.



As,Prov1+: provided reinforcement on the plate. If elements are red the additional reinforcement in the template is not sufficient.

7.5. User reinforcement

7.5.1. 1D Members

In the theoretical reinforcement design, we have calculated where reinforcement is needed.

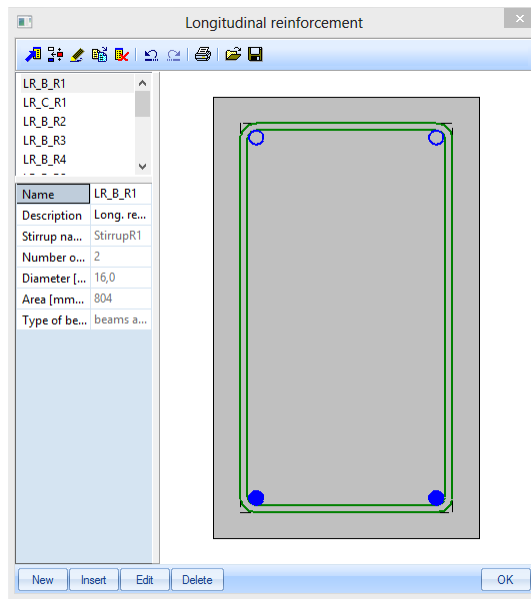
Since SCIA Engineer 19.0 it is possible to convert the provided reinforcement directly to practical reinforcement. You can choose for the action 'Conversion for real bars' when you have chosen the option 'Reinforcement design' → '1D members' → 'Reinforcement design' and if you have generated the results for 'Provided' as 'Type of values' in the property window.

Now a window appears that mentions you if it was possible to convert into real bars or not. If not, some explanation is given.

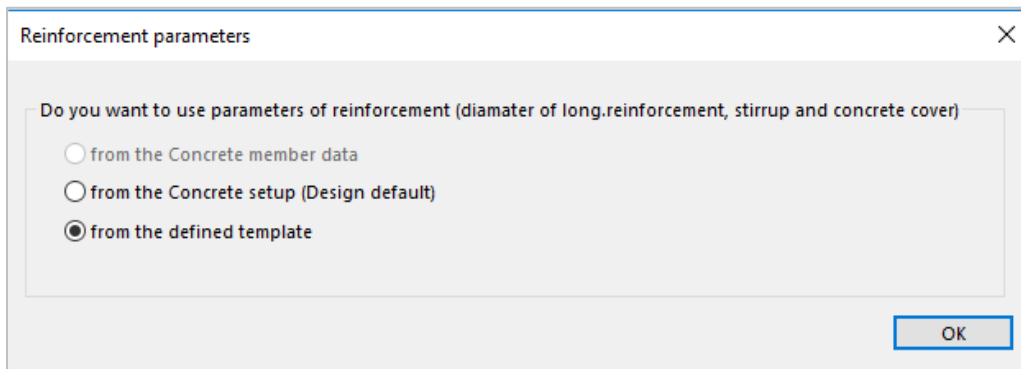
Afterwards you can edit the reinforcement by selecting the reinforcement data and choosing the action 'Edit reinforcement'.

You can also input manually the practical reinforcement by adding 'New reinforcement' for the whole length of the beam.

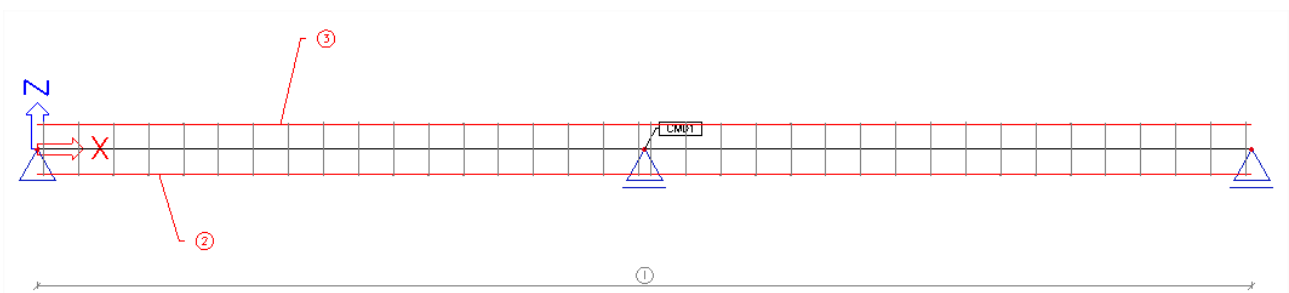
You can first select a template for the longitudinal reinforcement:



Next, you have to decide where the parameters of reinforcement are coming from:

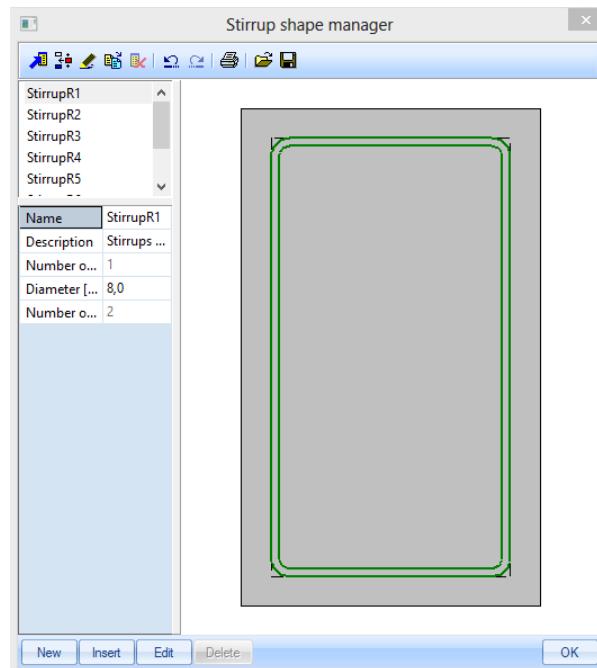


Now the practical reinforcement is shown graphically on the screen:

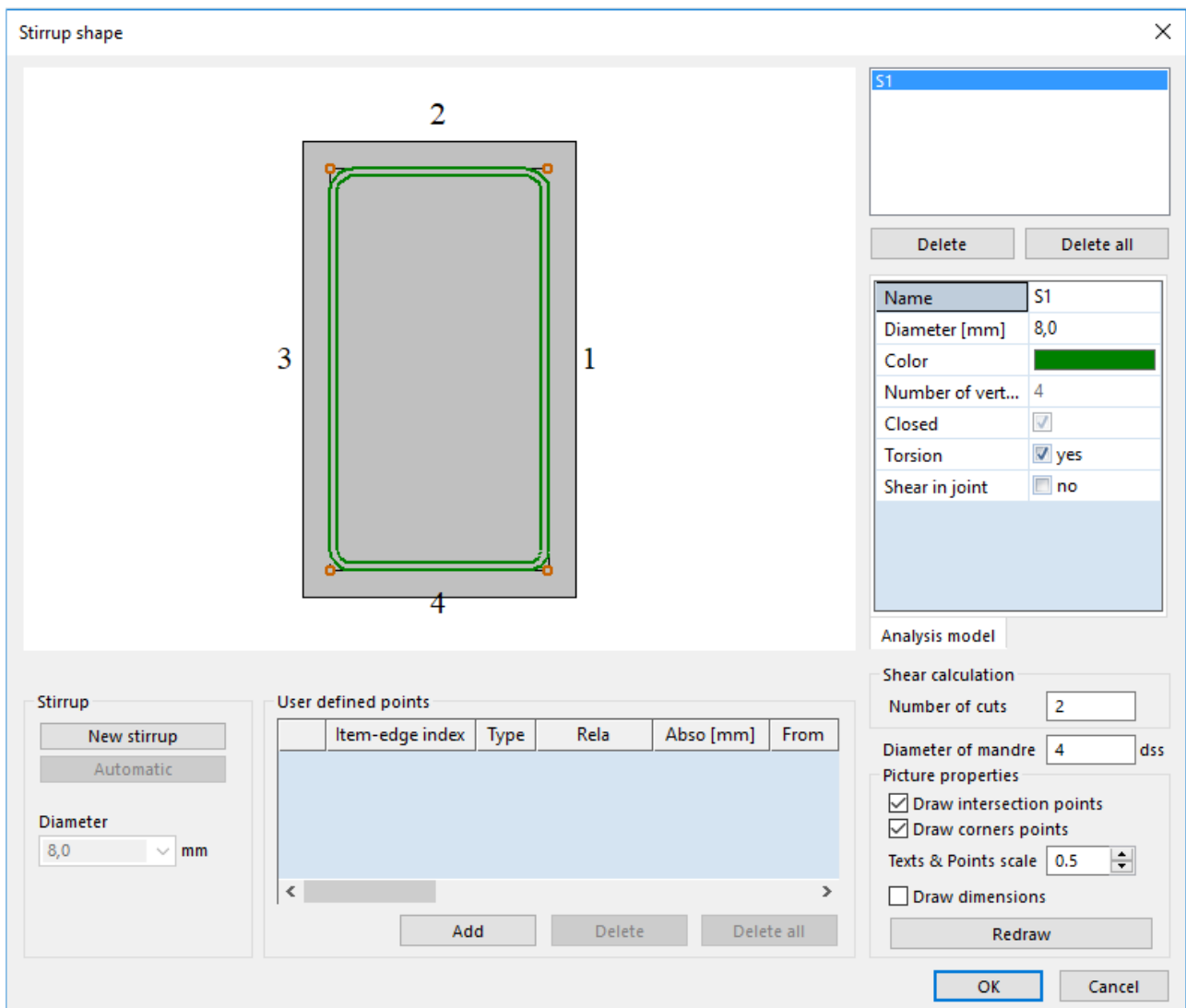


As a user, you can add locally 'New stirrups' or 'New longitudinal bars'.

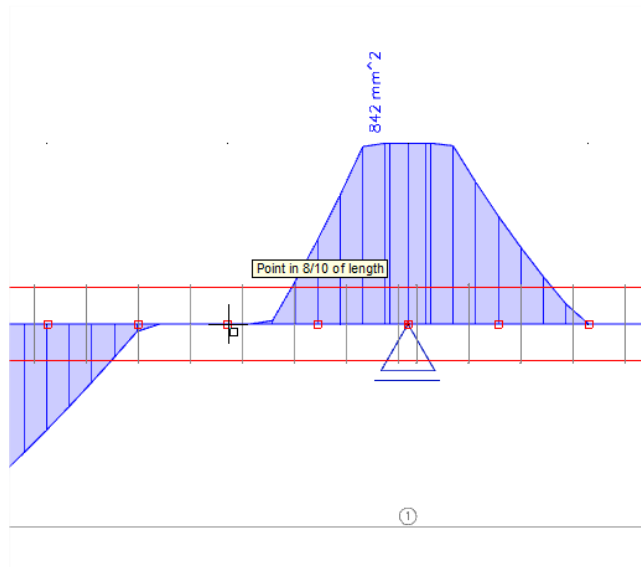
For the stirrups, you can select a certain stirrup shape:



You can edit the stirrup shape, or you can make a new one. Therefore, user points must be added.



For the longitudinal reinforcement, you can define precisely where you want to place the additional practical reinforcement (with the cursor snap settings):



The configuration for the selected zone of the member is shown:

Member S1, Zone from 0,500 m to 4,000 m(0.050 - 0.400)

Name	L2-S1E2
Position num...	3
Material	B 500A
Diameter [mm]	20,0
Number of b...	2
Area [mm ²]	628
Layer type	Uniform
Cover type	Surface to
Cover [mm]	0,0
Left bar	Before the
Right bar	Before the
Stirrup name	S1

Longitudinal reinforcement

- New layer
- Add bars to corners
- Bars positions
- Collision of bars
- Collision

New reinforcement parameters

- Number of bars: 2
- Diameter [mm]: 8,0
- Stirrup name: S1
- Edge index: 2
- Between existing bars
- Move layer

Type of beam

- beams and ribs

Stirrups

- Edit stirrups
- Edit cover
- Save to template

Reinforcement layers area

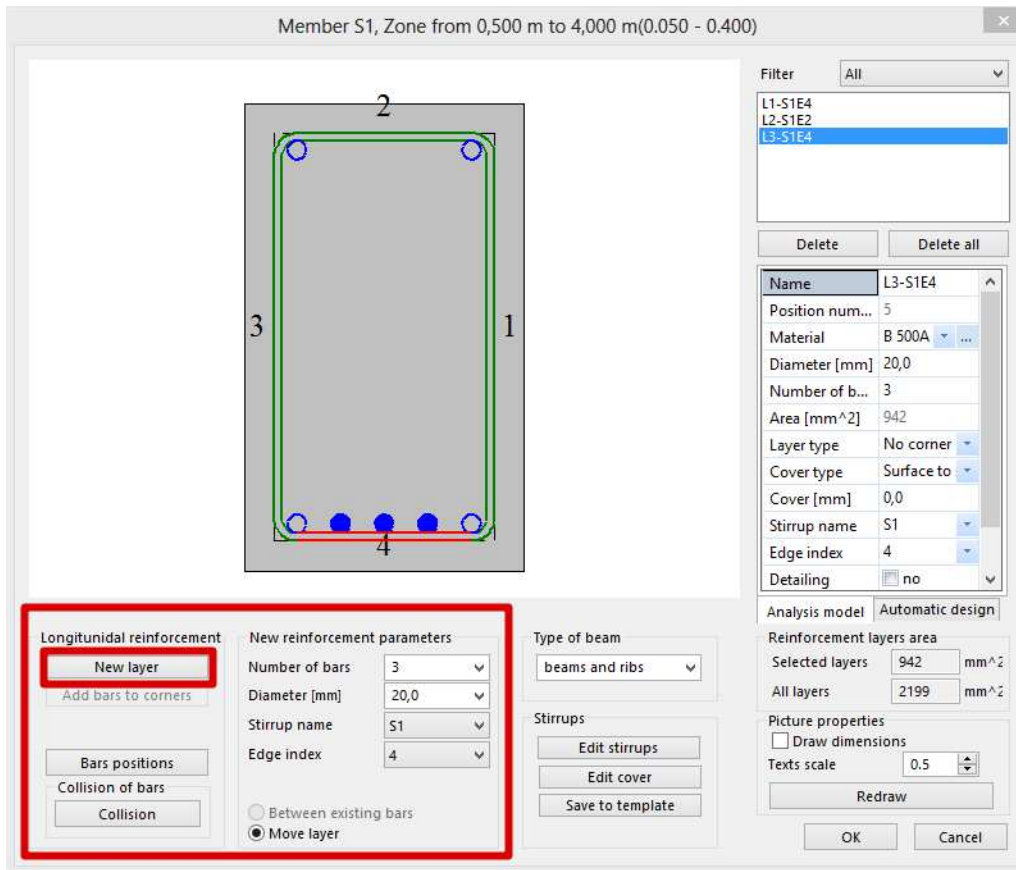
- Selected layers: 628 mm²
- All layers: 1257 mm²

Picture properties

- Draw dimensions
- Texts scale: 0,5
- Redraw

OK Cancel

Here you can set on which face extra reinforcement needs to be added:

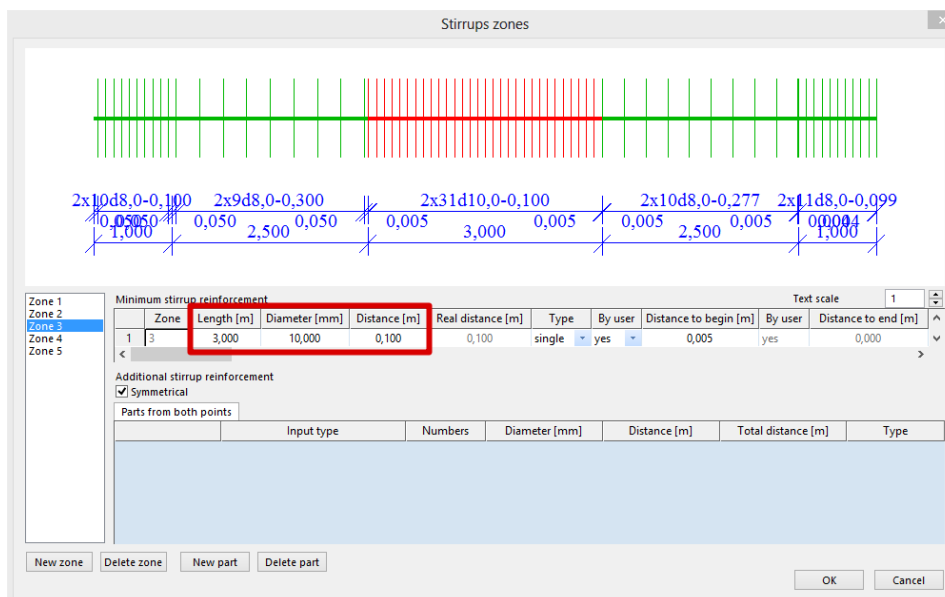


For reasons of simplicity we will add 3 bars of 20 mm that are still needed over the whole area where extra reinforcement is required. This can of course be done more detailed.

The same procedure will be repeated for the upper reinforcement (over the support).

You also need to increase the shear reinforcement in the zone above the support. You can do this by increasing the diameter of the stirrups or by decreasing the distance between the stirrups.

You can create different stirrup zones with the action 'Edit stirrup distances':

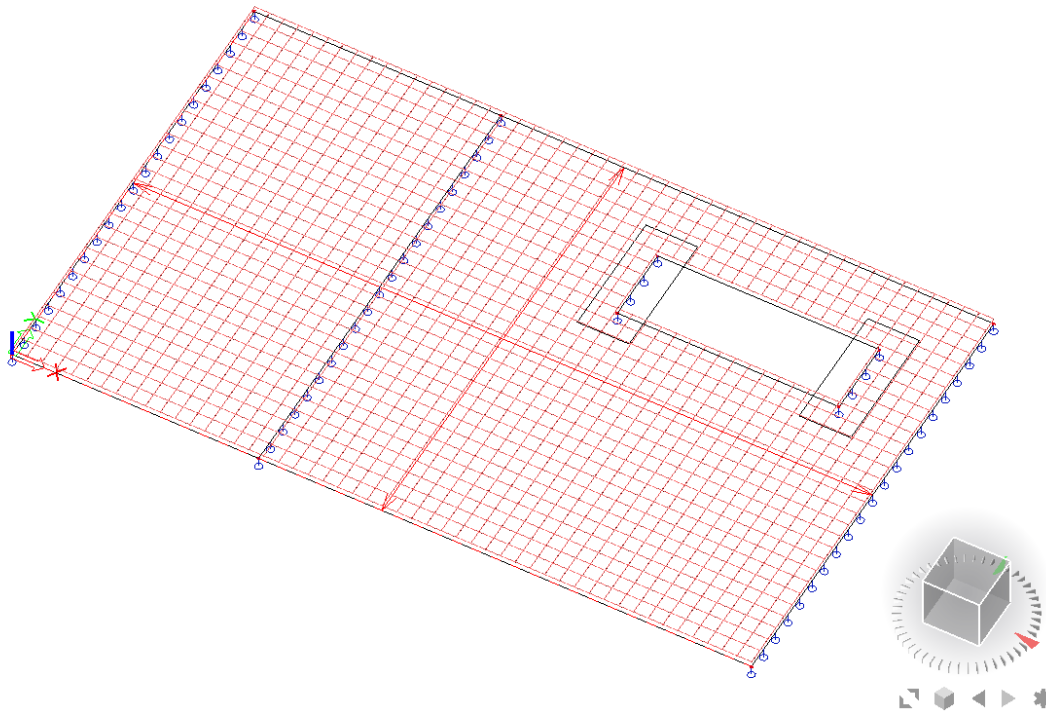


7.5.2. 2D members

Next to theoretical required and provided reinforcement you have also practical or user reinforcement. You can add this type of reinforcement to the plate via 'Concrete' → 'Reinforcement input + edit' → 'Reinforcement 2D'.

Name		RR1
2D member		Slab1
Reinforcement		
Type		Bars
Material		B 500A
Surface		Upper
Number of directions		2
Direction closest to surface		1
Angle of first direction [deg]		0,00
1		
Diameter (dl) [mm]		10,0
Concrete cover (cl,cu) [mm]		30
Bar distance (sl) [mm]		200
Offset [mm]		0
Reinf. area [mm ² /m]		393
2		
Diameter (dl) [mm]		10,0
Concrete cover (cl,cu) [mm]		40
Bar distance (sl) [mm]		200
Offset [mm]		0
Reinf. area [mm ² /m]		393
Total weight [kg]		58,26
Geometry		
Geometry defined by		Polygon
Actions		
Load from setup		>>>

You have to add this reinforcement separately at the upper and lower side (each time with two reinforcement directions).



NOTE: you can add multiple layers of practical reinforcement on the same area. The reinforcement added to this area is the sum of all these layers.

7.6. 1D ULS & SLS checks

The following checks are available for 1D Members in SCIA Engineer. Requesting output from these checks is the same as in the 'Results' and 'Steel' menus and will not be explained in this chapter.

7.6.1. Capacity response

The 'Capacity response' check is based on the calculation of strain and stress in a particular component (concrete fibre or reinforcement bar). The check consists of the comparison of those strains and stresses with the limited values according to EN 1992-1-1 requirements.

Extreme values of stress/strain in component

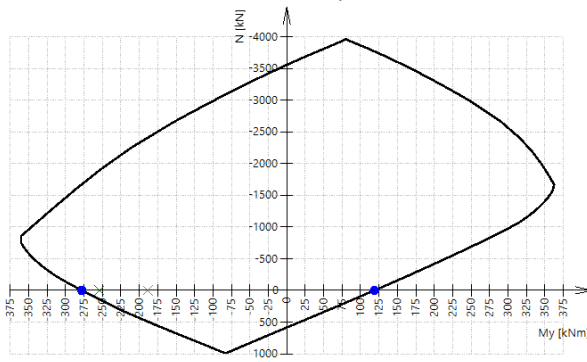
Type of component	Fibre / Bar	ϵ [‰]	ϵ_{lim} [‰]	σ [MPa]	σ_{lim} [MPa]	UC [-]	Status
Concrete - compression	1	-1.63	-3.5	-18.7	-20	0.93	OK
Concrete - tension	3	2.64	0	0	0	0.00	OK
Reinforcement - compression	3	-1.16	-22.5	-233	-454	0.51	OK
Reinforcement - tension	1	2.17	22.5	434	454	0.95	OK

7.6.2. Capacity diagram

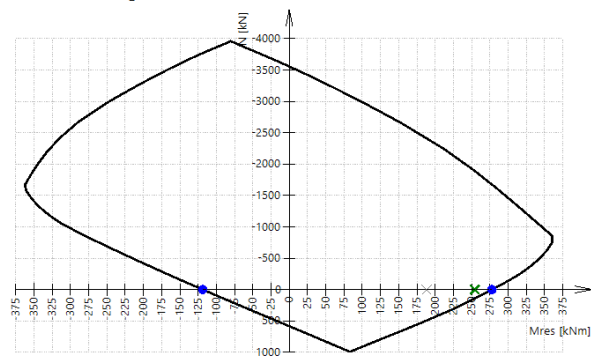
The 'Capacity diagram' service uses the creation of interaction diagram (graph presenting the capacity of a concrete member to resist a set of $N+My+Mz$).

This check calculates the extreme allowable interaction between the normal force N and bending moments My and Mz .

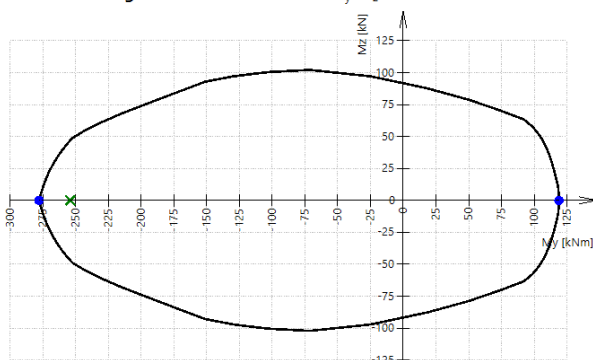
3D interaction diagram - Vertical section N-M_y



3D interaction diagram - Vertical section N-M_{res}



3D interaction diagram - Horizontal section M_y-M_z



7.6.3. Shear + Torsion

The check of the interaction between shear and torsion consists of three checks according to clause 6.1 - 6.3 in EN 1992-1-1:

- check of shear;
- check of torsion;
- check of interaction of shear and torsion.

Shear check

Check V_{Rdmax}

$$V_{Ed} = 152 \text{ kN} \leq V_{Rdmax} + V_{ccd} + V_{td} = 598 \text{ kN}$$

Note: The check satisfies for crushing of the compression strut ($V_{Ed} \leq V_{Rd,max} + V_{td} + V_{ccd}$).

Check V_{Edmax}

$$V_{Ed} = 152 \text{ kN} \leq V_{Edmax} + V_{ccd} + V_{td} = 705 \text{ kN}$$

Note: The check satisfies for shear force near the support ($V_{Ed} \leq V_{Ed,max} + V_{td} + V_{ccd}$).

Check V_{Rdc} and V_{Rds}

$$V_{Ed} = 152 \text{ kN} > V_{Rdc} = 87.8 \text{ kN} \text{ and } V_{Ed} = 152 \text{ kN} > V_{Rds} + V_{ccd} + V_{td} = 66.5 \text{ kN}$$

Error: The check does not satisfy, because of shear reinforcement ($V_{Ed} > V_{Rds} + V_{ccd} + V_{td}$). It is necessary to increase area of shear reinforcement or to increase dimensions of the cross-section or quality of shear reinforcement.

Unity check

$$UC = \frac{abs(V_{Ed})}{V_{Rd}} = \frac{abs(152 \text{ kN})}{66.5 \text{ kN}} = 2.28$$

7.6.4. Stress limitation

The stress limitation check is based on the verification of:

- **compressive stress in the concrete:** the high value of compressive stress in concrete could lead to appearance of longitudinal cracks, spreading of micro-cracks in concrete and higher values of creep (mainly nonlinear). This effect can lead to a state where the structure is unusable.
- **tensile stress in the reinforcement:** stress in reinforcement is verified due to limitation of unacceptable strain existence and thus appearance of cracks in concrete.

Verification of cracks in cross-section

Load	Type of module	E_c [MPa]	Combi.	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	σ_{ct} [MPa]	h [mm]	$f_{ct,eff}$ [MPa]	Cracks appear
Short	E_c	0	Char.	0	-188	0	12.6	500	2.9	YES

Stress limitation in concrete

Check type	Load	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	y_i [mm]	z_i [mm]	σ_c [MPa]	$\sigma_{c,lim}$ [MPa]	$\sigma_c/\sigma_{c,lim}$ [-]	Status
§7.2(2) Char.	Short	0	-188	0						OFF
§7.2(3) Q.-P.	Short	0	-188	0	0.15	-0.25	-21.2	-13.5	1.57	Not OK

Stress limitation in non-prestressed reinforcement

Check type	Load	N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	y_i [mm]	z_i [mm]	σ_s [MPa]	$\sigma_{s,lim}$ [MPa]	$\sigma_s/\sigma_{s,lim}$ [-]	Status
§7.2(5) Char.	Short	0	-188	0	0.09	0.2	300	400	0.75	OK

7.6.5. Crack width

The crack width is calculated according to clause 7.3.4 in EN 1992-1-1.

Calculation of cracking forces (uncracked section)

Maximal stress in concrete

$$\sigma_{ct} = 12.6 \text{ MPa}$$

Cracking forces

$$N_{cr} = 0 \text{ kN} \quad M_{cry} = -43.3 \text{ kNm} \quad M_{crz} = 0 \text{ kNm}$$

$$\sigma_{ct} = 12.6 \text{ MPa} > \sigma_{cr} = 2.9 \text{ MPa} \Rightarrow \text{Cracks appear}$$

Note: The crack is appeared, because maximal tensile stress is greater than cracking strength.

Maximum crack spacing

$$s_{max} = 45 \text{ mm} \leq 5 \cdot (c + 0.5 \cdot \phi_{eq}) = 275 \text{ mm} \text{ or } \rho_{p,eff} = 0, \text{ therefore:}$$

$$s_{r,max} = k_3 \cdot c + \frac{k_1 \cdot k_2 \cdot k_4 \cdot \phi_{eq}}{\rho_{p,eff}} = 3.4 \cdot 0.045 + \frac{0.8 \cdot 0.5 \cdot 0.425 \cdot 0.02}{0.0428} = 232 \text{ mm} \quad (7.11)$$

Mean strain in the reinforcement

$$\begin{aligned} \varepsilon_{sm,cm} &= \max \left(\frac{\sigma_s - k_t \cdot \left(\frac{f_{ct,eff}}{\rho_{p,eff}} \right) \cdot (1 + \alpha_E \cdot \rho_{p,eff})}{E_s}, \frac{0.6 \cdot \sigma_s}{E_s} \right) \\ &= \max \left(\frac{300 \cdot 10^6 - 0.46 \cdot \left(\frac{2.9 \cdot 10^6}{0.0428} \right) \cdot (1 + 6.06 \cdot 0.0428)}{200 \cdot 10^9}, \frac{0.6 \cdot 300 \cdot 10^6}{200 \cdot 10^9} \right) = 1.3 \text{ ‰} \end{aligned} \quad (7.9)$$

Calculated crack width

$$w = \varepsilon_{sm,cm} \cdot s_{r,max} = 1.3 \cdot 232 = 0.303 \text{ mm} \quad (7.8)$$

Limit value of crack width

$$w_{max} = 0.4 \text{ mm}$$

7.6.6. Deflection

The calculation of deflection is done according to chapter 7.4.3 from EN 1992-1-1.

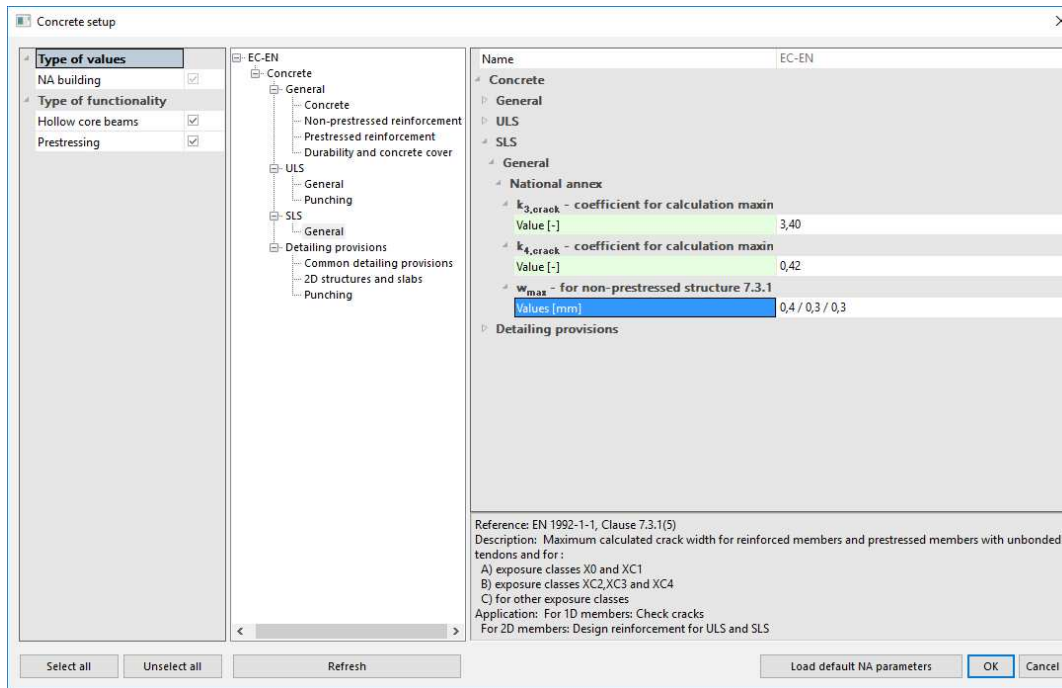
The simplified method is used where the calculation is done twice, assuming the whole member to be uncracked and fully cracked, and then interpolating formula 7.18 according to clause 7.4.3(7). This is the default used method.

Deflections	
Linear deflection	
$\delta_{lin,y}$	$= u_{y5} + u_{y1} = 0 + 0 = 0 \text{ mm}$
$\delta_{lin,z}$	$= u_{z5} + u_{z1} = 0 + -3.08 = -3.08 \text{ mm}$
Immediate deflection	
$\delta_{imm,y}$	$= u_{y1} \cdot \text{ratio}_{uy5} = 0 \cdot 2.88 = 0 \text{ mm}$
$\delta_{imm,z}$	$= u_{z1} \cdot \text{ratio}_{uz5} = -3.08 \cdot 2.5 = -7.7 \text{ mm}$
Short-term deflection	
$\delta_{short,y}$	$= u_{y5} \cdot \text{ratio}_{uy5} = 0 \cdot 2.88 = 0 \text{ mm}$
$\delta_{short,z}$	$= u_{z5} \cdot \text{ratio}_{uz5} = 0 \cdot 2.5 = 0 \text{ mm}$
Long-term + creep deflection	
$\delta_{long,creep,y}$	$= u_{y1} \cdot \text{ratio}_{uy1} = 0 \cdot 5.22 = 0 \text{ mm}$
$\delta_{long,creep,z}$	$= u_{z1} \cdot \text{ratio}_{uz1} = -3.08 \cdot 3.38 = -10.4 \text{ mm}$
Creep deflection	
$\delta_{creep,y}$	$= u_{y1} \cdot (\text{ratio}_{uy1} - \text{ratio}_{uy5}) = 0 \cdot (5.22 - 2.88) = 0 \text{ mm}$
$\delta_{creep,z}$	$= u_{z1} \cdot (\text{ratio}_{uz1} - \text{ratio}_{uz5}) = -3.08 \cdot (3.38 - 2.5) = -2.69 \text{ mm}$
Long-term deflection	
$\delta_{long,y}$	$= \delta_{long,creep,y} - \delta_{creep,y} = 0 - 0 = 0 \text{ mm}$
$\delta_{long,z}$	$= \delta_{long,creep,z} - \delta_{creep,z} = -10.4 - -2.69 = -7.7 \text{ mm}$
Additional deflection	
$\delta_{add,y}$	$= \delta_{short,y} + \delta_{long,creep,y} - \delta_{imm,y} = 0 + 0 - 0 = 0 \text{ mm}$
$\delta_{add,z}$	$= \delta_{short,z} + \delta_{long,creep,z} - \delta_{imm,z} = 0 + -10.4 - -7.7 = -2.69 \text{ mm}$
Limit additional deflection	
$\delta_{add,lim,y}$	$= 0 \text{ mm}$
$\delta_{add,lim,z}$	$= \frac{-l_z}{\text{Lim}_{add}} = \frac{-10}{500} = -20 \text{ mm}$
Total deflection	
$\delta_{tot,y}$	$= \delta_{short,y} + \delta_{long,creep,y} = 0 + 0 = 0 \text{ mm}$
$\delta_{tot,z}$	$= \delta_{short,z} + \delta_{long,creep,z} = 0 + -10.4 = -10.4 \text{ mm}$
Limit total deflection	
$\delta_{tot,lim,y}$	$= 0 \text{ mm}$
$\delta_{tot,lim,z}$	$= \frac{-l_z}{\text{Lim}_{tot}} = \frac{-10}{250} = -40 \text{ mm}$

7.7. 2D Crack width check

For 2D Members there are fewer checks: the crack width check and the punching check.

The values of the maximum crack width (w_{max}) are national determined parameters, depending on the chosen exposure class. Therefore, you can find this value in the setup for National Determined Parameters, via the main menu → 'Project data' → 'National annex [...] → 'EN 1992-1-1 [...]'.



Type of used reinforcement

You can perform the 'Crack width check' for all three types of reinforcement: required, provided and user reinforcement. The crack width check is performed on a quasi-permanent SLS combination.

If you use as type of reinforcement for the crack width either the provided or required reinforcement, you should choose a ULS combination as well. This is necessary because the required/provided reinforcement is calculated based on an ULS combination. After this reinforcement is calculated, it can be used to perform the crack width check (this is done automatically by the software).

Required/provided reinforcement

ID	NAME	CRACK WIDTH (SLS)
Selection		
Type of selection	All	
Filter	No	
Result case		
Type of load	Combinations	
Combination	EN-SLS Quasi-permanent	
Envelope (for 2D drawing)	Absolute extreme	
Type of reinforcement	Required	
Result case for required rei...		
Type of load	Combinations	
Combination	EN-ULS (STR/GEO) Set B	
Averaging of peak	<input type="checkbox"/>	

User reinforcement

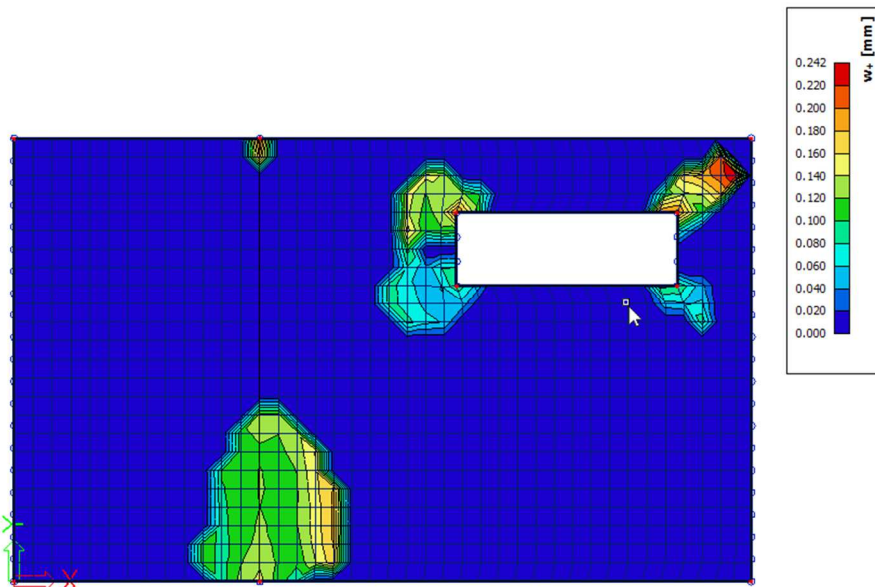
Combination used to perform the crack width check

Amount of reinforcement is already known -> no ULS combination necessary

Filter	No
Result case	
Type of load	Combinations
Combination	EN-SLS Quasi-permanent
Envelope (for 2D drawing)	Absolute extreme
Type of reinforcement	User
Averaging of peak	<input type="checkbox"/>
	In nodes avg.
	100 mesh element

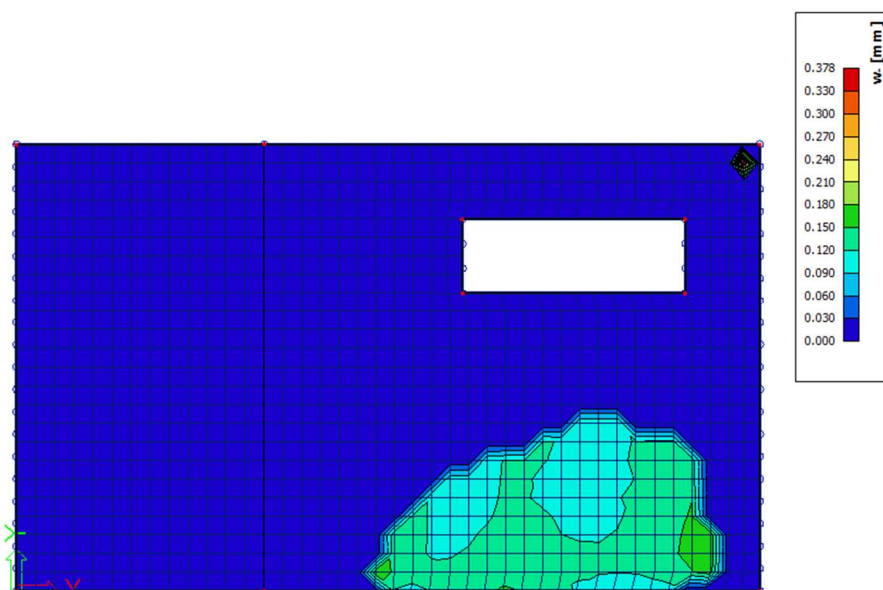
Crack width w_+

Combination = SLS; Type of used reinforcement = Required; Value = w_+



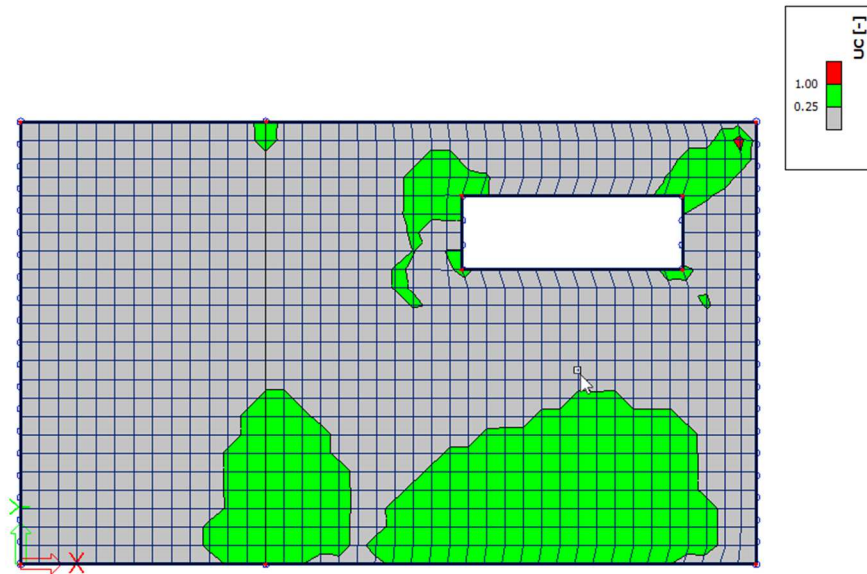
Crack width w_-

Combination = SLS; Type of used reinforcement = Required; Value = w_-



Unity check

Combination = SLS; Type of used reinforcement = Required; Value = **Check value**

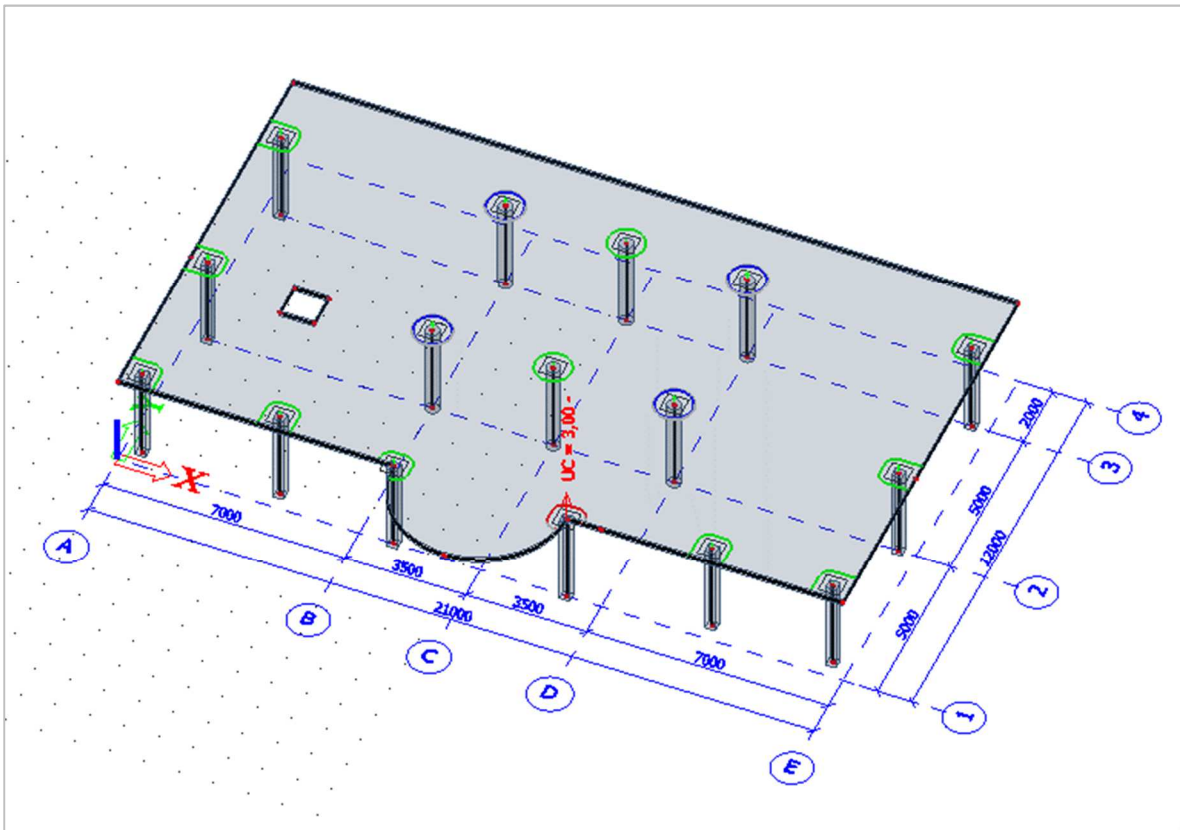


7.8. Punching check

You can find the punching design in the 'Concrete' menu → 'Reinforcement design' → '2D Members' → 'Punching design'.

The beta factor is automatically calculated, and the shape of the column is automatically recognized. This means that you don't have to define anything before performing the punching check.

Before performing the punching design, you need to set the properties window. Here you need to choose the type of reinforcement (this is the same as for the crack width check).



There are three different type of results.

- **Green visualization:** the concrete can take the concentrated load.

Punching design
 Linear calculation
 Combination: ULS
 Extreme: Global
 Selection: N61
Summary

Name	Case	Punching case	Punching shape	UC v _{Rd,max} [-]	UC v _{Rd,c} [-]	Shear reinforcement perimeters	UC v _{Rd,cs} [-]	UC A _{sw,det} [-]	UC [-] Check
N61	ULS/1	Internal column	Circle (400)	0,17	0,52	not required	-	-	0,52 OK

Concrete

Name	Case	Punching case	Punching shape	V _{Ed} [kN] β [-]	M _{Ed,y} [kNm] M _{Ed,z} [kNm]	Plate h [mm]	Material f _{cd} [MPa]	d _{eff} [mm] ρ _l [%]	u ₀ [m] u ₁ [m]	V _{Ed,u0} [MPa] V _{Ed,u1} [MPa]	V _{Rd,max} [MPa] V _{Rd,c} [MPa]	UC v _{Rd,max} [-] UC v _{Rd,c} [-]
N61	ULS/1	Internal column	Circle (400)	128,46 1,15	0,09 13,98	Ceiling 200,00	C30/37 20,00	160,00 0,18	1,257 3,267	0,73 0,28	4,22 0,55	0,17 0,52

- **Blue visualization:** the concrete cannot take the concentrated load but punching reinforcement can be designed.

Punching design
 Linear calculation
 Combination: ULS
 Extreme: Global
 Selection: N59
Summary

Name	Case	Punching case	Punching shape	UC v _{Rd,max} [-]	UC v _{Rd,c} [-]	Shear reinforcement perimeters	UC v _{Rd,cs} [-]	UC A _{sw,det} [-]	UC [-] Check
N59	ULS/1	Internal column	Circle (400)	0,36	1,06	3x 12Ø8(radial) 80+2x80=240	0,71	1,00	1,00 OK, BUT

Concrete

Name	Case	Punching case	Punching shape	V _{Ed} [kN] β [-]	M _{Ed,y} [kNm] M _{Ed,z} [kNm]	Plate h [mm]	Material f _{cd} [MPa]	d _{eff} [mm] ρ _l [%]	u ₀ [m] u ₁ [m]	V _{Ed,u0} [MPa] V _{Ed,u1} [MPa]	V _{Rd,max} [MPa] V _{Rd,c} [MPa]	UC v _{Rd,max} [-] UC v _{Rd,c} [-]
N59	ULS/1	Internal column	Circle (400)	265,21 1,15	26,85 6,10	Ceiling 200,00	C30/37 20,00	160,00 0,38	1,257 3,267	1,52 0,58	4,22 0,55	0,36 1,06

Reinforcement

Name	Case	Shear reinforcement perimeters	u _{out} [m] B _{out} [mm]	S _{t,out} [mm] S _{t,out} [mm]	Control perimeters (distance/capacity)	Material f _{ywd,or} [MPa]	A _{sw,req} [mm ²] A _{sw,min} [mm ²]	A _{sw} [mm ²] A _{sw,tot} [mm ²]	V _{Rd,cs} [MPa] k _{max} V _{Rd,c} [MPa]	UC v _{Rd,cs} [-] UC A _{sw,det} [-]
N59	ULS/1	3x 12Ø8(radial) 80+2x80=240	3,472 354	230 230	320/71%	B 500B 290,0	103 11	603 1810	1,42 0,82	0,71 1,00

- **Red visualization:** the concrete cannot take the concentrated load and punching reinforcement cannot be designed.

Punching design
 Linear calculation
 Combination: ULS
 Extreme: Global
 Selection: N20
Summary

Name	Case	Punching case	Punching shape	UC v _{Rd,max} [-]	UC v _{Rd,c} [-]	Shear reinforcement perimeters	UC v _{Rd,cs} [-]	UC A _{sw,det} [-]	UC [-] Check
N20	ULS/1	Corner column	Rectangle (300;300)	0,86	1,01	3x 9Ø8(radial) 80+2x80=240	0,68	3,00	3,00 NOT OK

Concrete

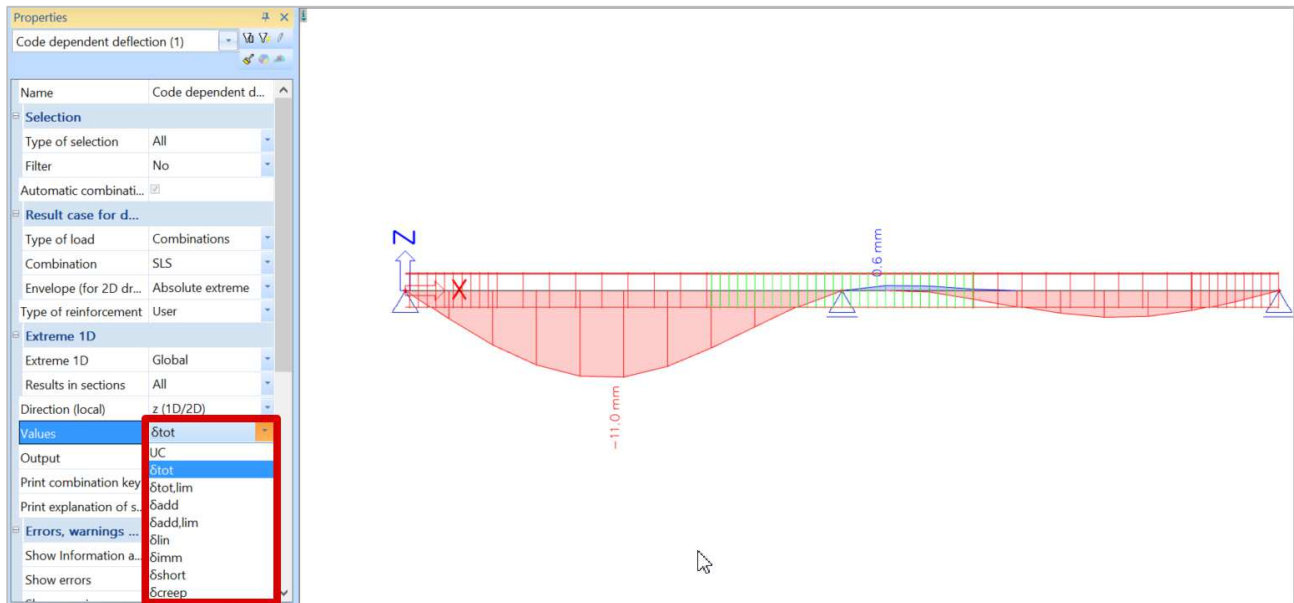
Name	Case	Punching case	Punching shape	V _{Ed} [kN] β [-]	M _{Ed,y} [kNm] M _{Ed,z} [kNm]	Plate h [mm]	Material f _{cd} [MPa]	d _{eff} [mm] ρ _l [%]	u ₀ [m] u ₁ [m]	V _{Ed,u0} [MPa] V _{Ed,u1} [MPa]	V _{Rd,max} [MPa] V _{Rd,c} [MPa]	UC v _{Rd,max} [-] UC v _{Rd,c} [-]
N20	ULS/1	Corner column	Rectangle (300;300)	139,68 1,50	19,42 6,98	Ceiling 200,00	C30/37 20,00	160,00 0,30	0,359 2,362	3,65 0,55	4,22 0,55	0,86 1,01

Reinforcement

Name	Case	Shear reinforcement perimeters	u _{out} [m] B _{out} [mm]	S _{t,out} [mm] S _{t,out} [mm]	Control perimeters (distance/capacity)	Material f _{ywd,or} [MPa]	A _{sw,req} [mm ²] A _{sw,min} [mm ²]	A _{sw} [mm ²] A _{sw,tot} [mm ²]	V _{Rd,cs} [MPa] k _{max} V _{Rd,c} [MPa]	UC v _{Rd,cs} [-] UC A _{sw,det} [-]
N20	ULS/1	3x 9Ø8(radial) 80+2x80=240	2,395 328	222 222	320/68%	B 500B 290,0	63 10	452 1357	1,45 0,82	0,68 3,00

7.9. Code dependent deformations

The 'Code dependent deformations' calculation is a bi-linear calculation. The calculation is performed, stiffnesses are reduced and the calculation is performed another time with these reduced stiffnesses. This explanation is simplified. In the concrete manual you can find the full explanation. Below you can see the values calculated by this type of calculation.



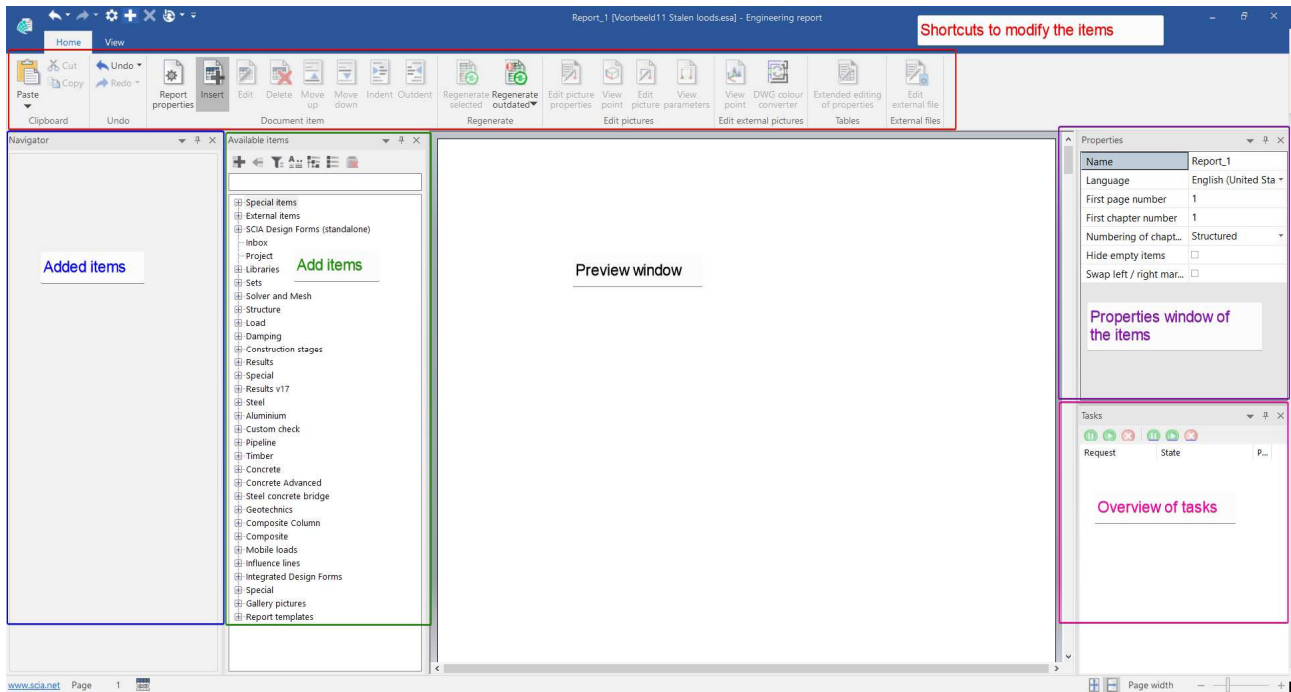
For 1D member

Name	dx [m]	Case Type of reinf.	$\delta_{in,y}$	$\delta_{imm,y}$	$\delta_{short,y}$	$\delta_{creep,y}$	$\delta_{add,y}$	$\delta_{add,lim,y}$	$\delta_{tot,y}$	$\delta_{tot,lim,y}$	UC [-] Check
			$\delta_{in,z}$ [mm]	$\delta_{imm,z}$ [mm]	$\delta_{short,z}$ [mm]	$\delta_{creep,z}$ [mm]	$\delta_{add,z}$ [mm]	$\delta_{add,lim,z}$ [mm]	$\delta_{tot,z}$ [mm]	$\delta_{tot,lim,z}$ [mm]	
S1	2.500-	SLS/1	0.0	0.0	0.0	0.0	0.0	20.0	0.0	40.0	0.55
		User	-3.4	-6.8	-8.0	-3.0	-4.2	10.0	-11.0	20.0	OK
S1	5.500-	SLS/2	0.0	0.0	0.0	0.0	0.0	20.0	0.0	40.0	0.03
		User	0.2	0.5	0.5	0.1	0.1	10.0	0.6	20.0	OK

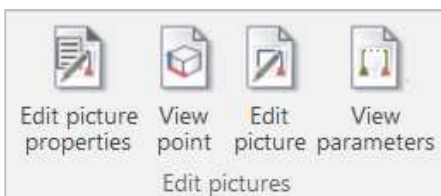
Chapter 8: Engineering report

8.1. General interface

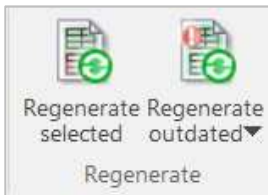
The general interface of the engineering report consists of 5 different windows. You can modify the layout of the interface to your own demands by dragging these windows to the desired position. The windows are shown in the following picture provided by an explanation of its purposes.



The engineering report works with items. These items contain all the information that is available in SCIA Engineer. In order to add an item into the report, you should double click on the item in the window **'Available items'** or type the item's name into the search box of the available items. As soon as you have added this item, it will be displayed in the tab **'Navigator'** or **'Added items'** in the previous picture. You can always change the added items by using the properties window, which is based on the same principle as in the model space of SCIA Engineer. You can also modify inserted pictures by using the shortcuts **'Edit picture properties'**, **'View point'**, ... which are shown in the following picture.



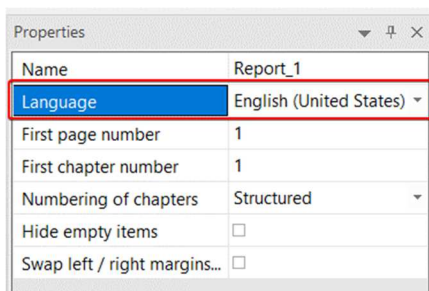
After modifying one or multiple added items, you have to regenerate the content of the report. You can execute this command for an individual item as well as for the entire report by using the shortcuts '**Regenerate selected**' or '**Regenerate outdated**'.



You can use the shortcut '**Edit**' to customize the advanced properties of the added items.



You can modify the language of the report, both in- and output, by adjusting the language in the properties window of the engineering report.

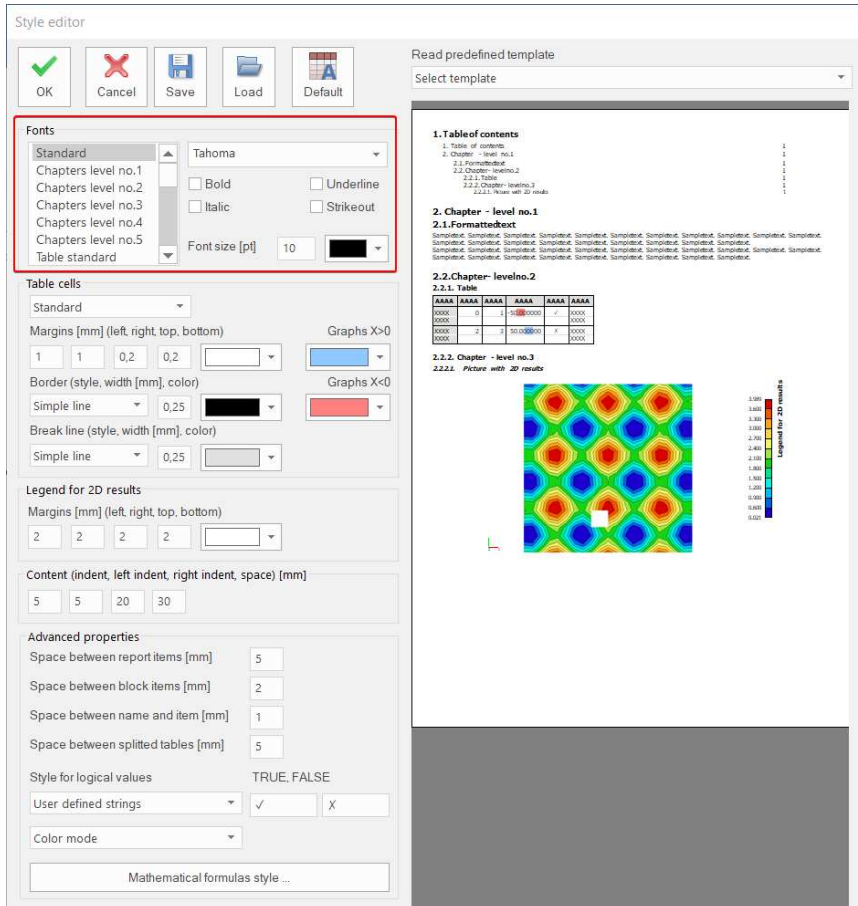


8.2. General page layout

The following chapter contains an explanation of several items concerning the general page layout of the engineering report.

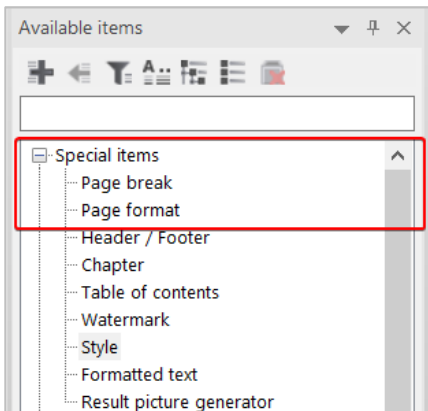
8.2.1. Page layout

You can select the subitem 'Style' to modify the page layout of the report. You can find this item as a subitem of the available item 'Special items'. When you have added this item to the report, you can select the shortcut 'Edit'. Then the style editor will open. This editor allows you to modify all the settings concerning the fonts, the spacings and the colours of the text in the report.

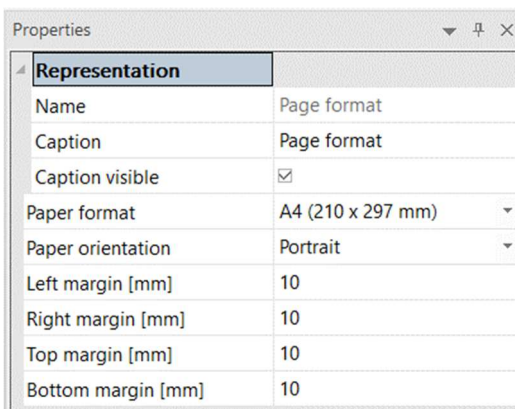


8.2.2. Page format and page break

You can adjust the page format of the report by inserting the item '**Page format**'. This item is a subcategory of the item '**Special items**'. It contains also the item '**Page break**' by which you can define the end of a page, for example at the end of a chapter.

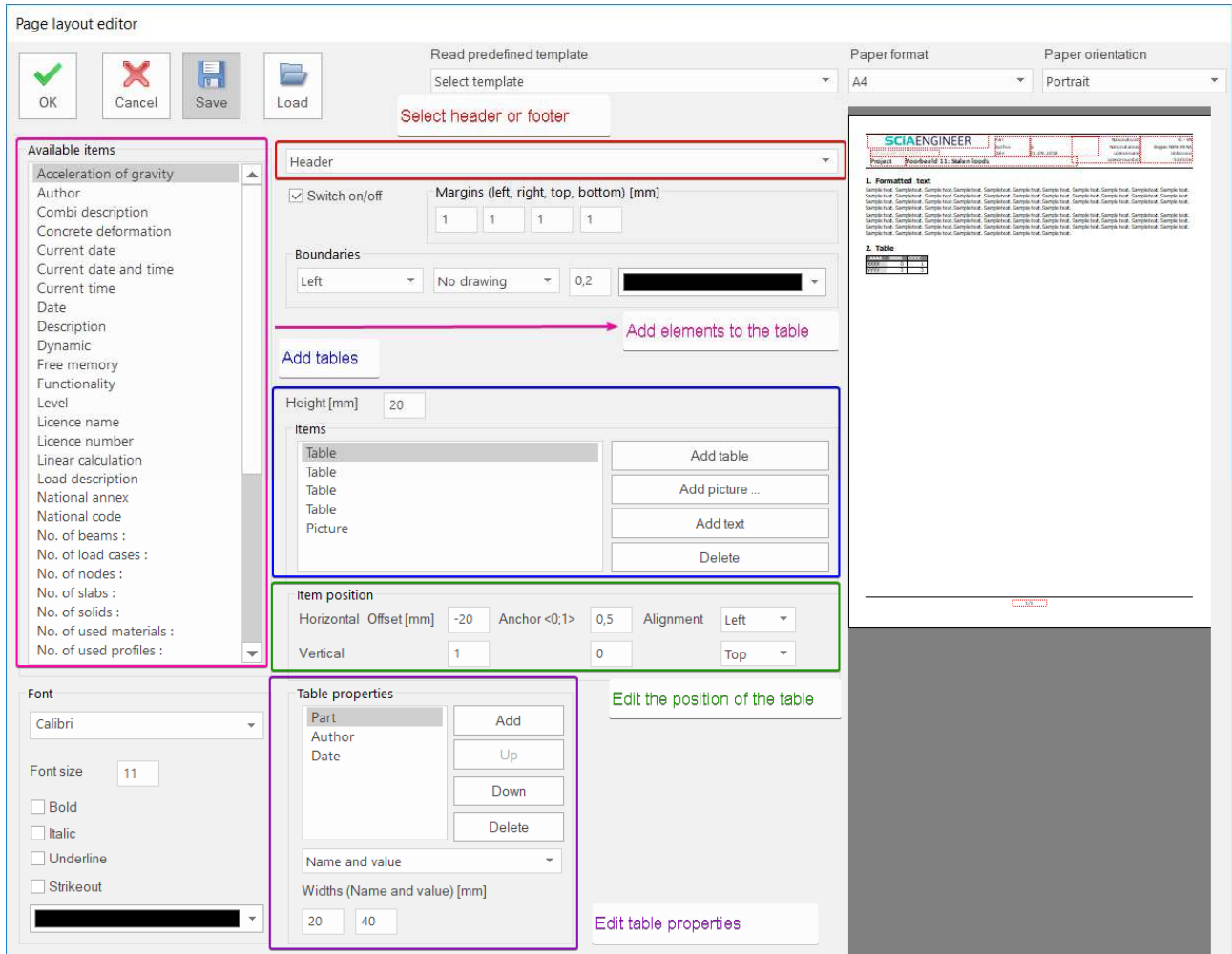


When you have inserted the item '**Page format**', you can define all its settings – the format and the margins – in the properties window.



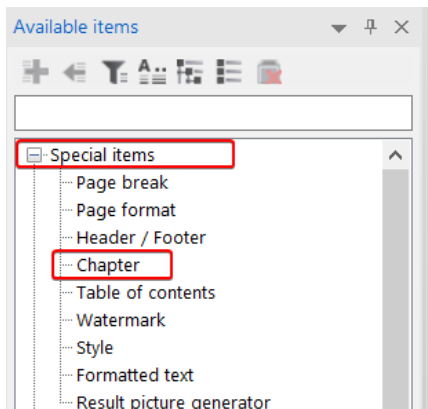
8.2.3. Header and footer

You can find the item **'Header / Footer'** in the category **'Special items'**. Activating the shortcut **'Edit'** allows you to define a completely customized header and footer for your report. You can save this item as a template and apply it in other reports as well.



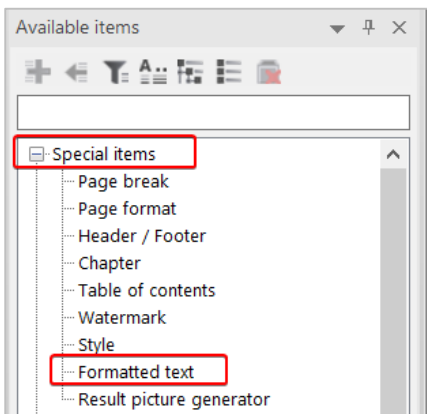
8.2.4. Chapter

You can create several chapters in the engineering report by adding the special item **'Chapter'** into the report.

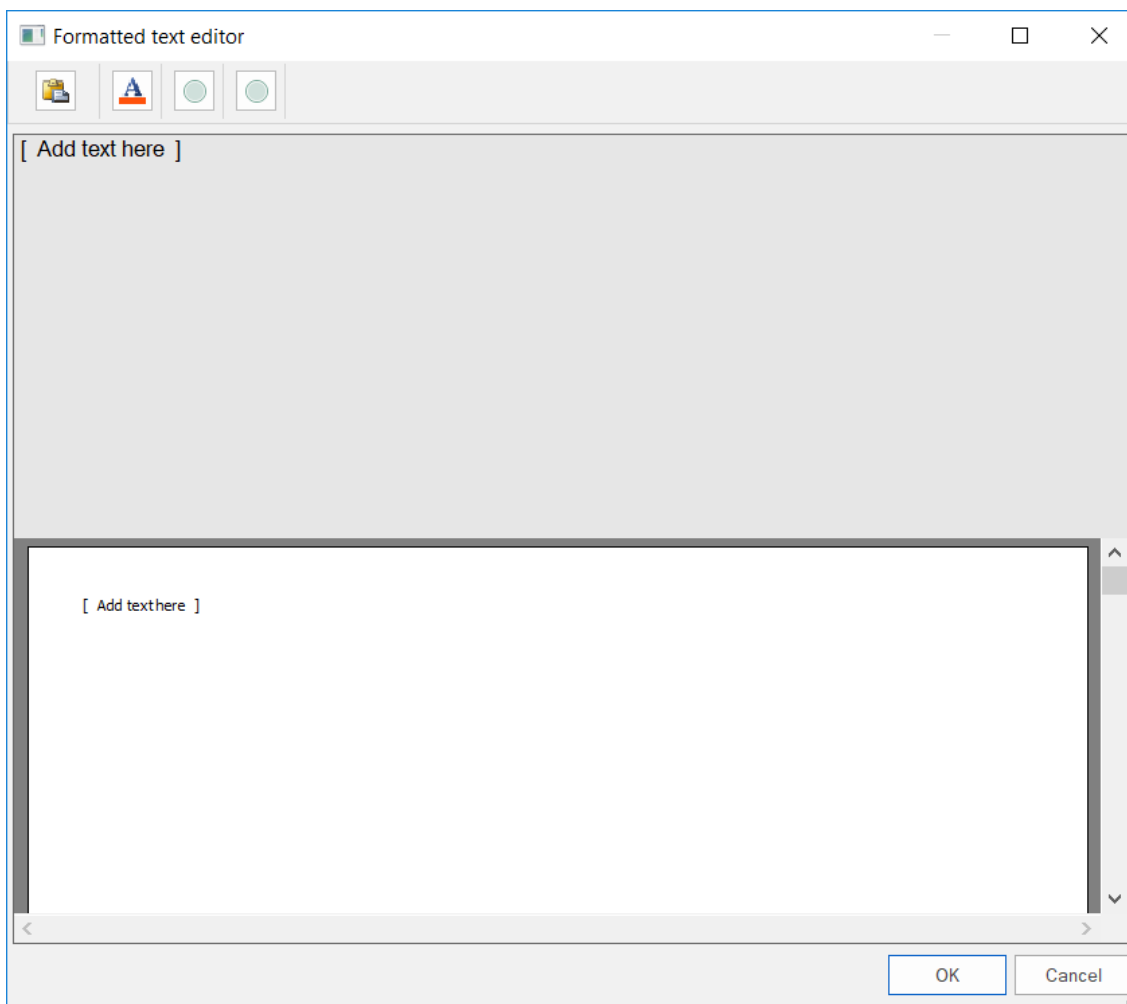


8.2.5. Formatted text

With the special item '**Formatted text**' you can input text in the engineering report manually.



As soon as you have added this item to the report and you have chosen the command '**Edit**', you can insert text in the editor window.



8.3. Adding pictures

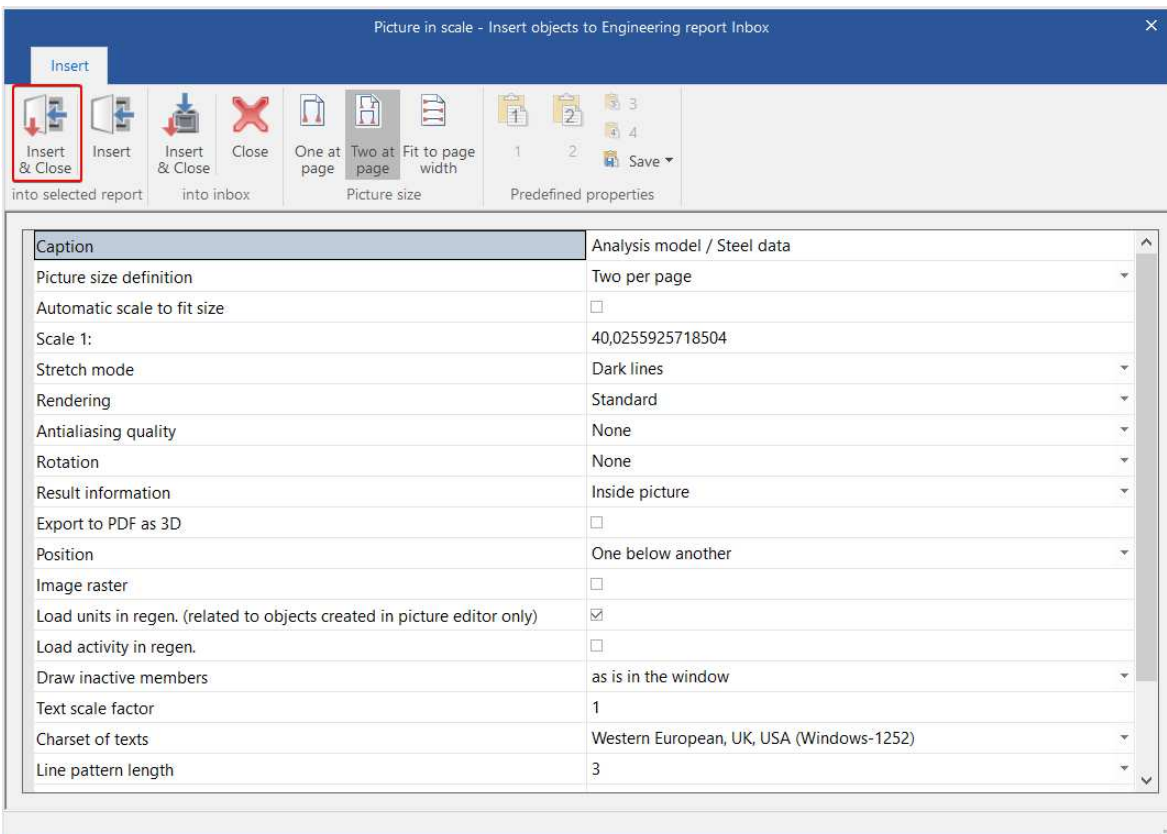
You can also add pictures to the engineering report. You have different methods in order to insert these items. We explain these methods in this chapter.

8.3.1. Screenshot or live pictures

As first option to insert pictures into the report you can use the commands **'Screenshot into Engineering report'**, **'Live picture in scale into Engineering report'** and **'Live picture into Engineering report'**. These commands allow you to send pictures of the model to the report and you can access it by right clicking in SCIA Engineer.

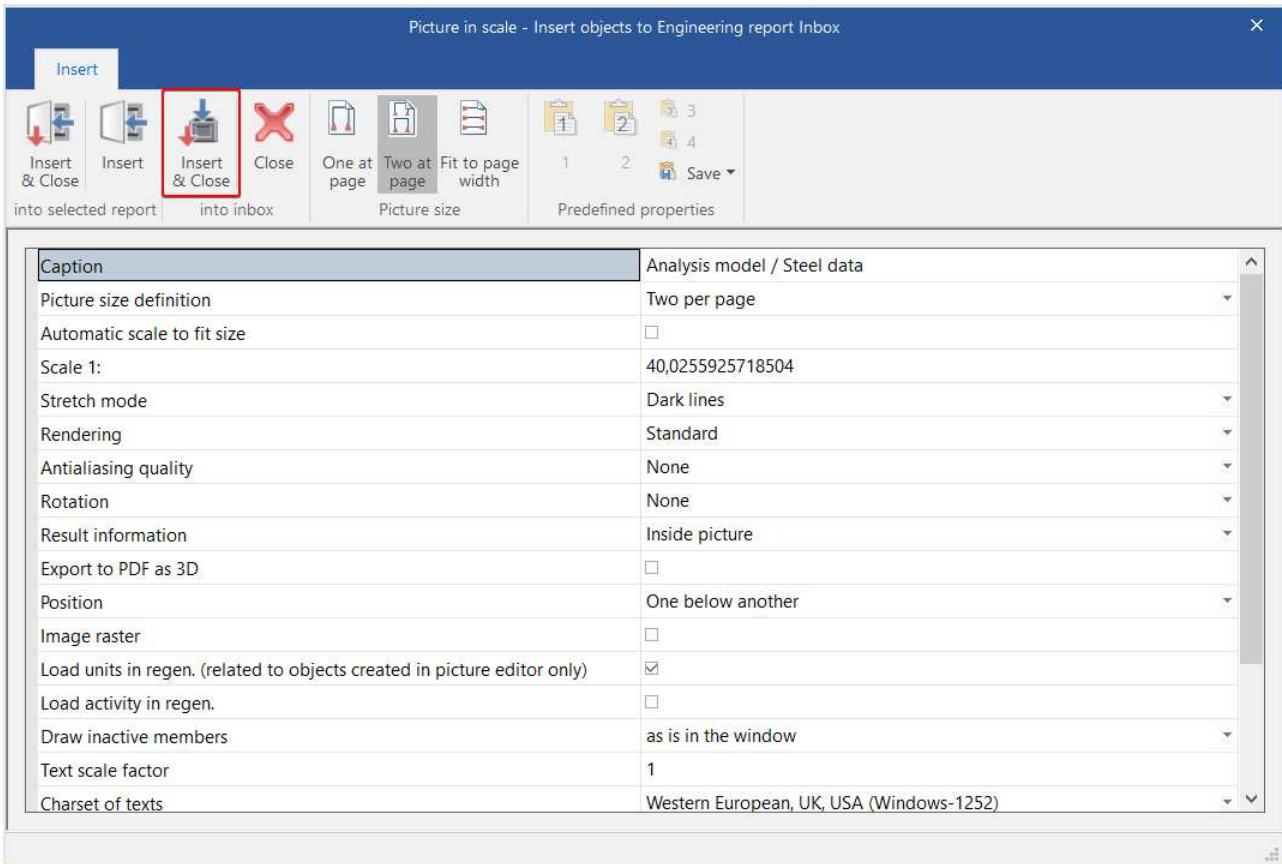


However, it is important that you are aware of the difference between both types of picture. A screenshot is a fixed shot, meaning it will **not** be updated when you modify the structure. Both types of live pictures will be updated when you modify the structure. You can add the picture into the report by executing the command **'Insert & Close'**.



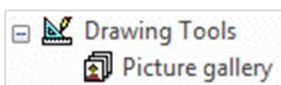
8.3.2. Inbox

You can send live pictures or screenshots to the **'Inbox'** of the engineering report instead of immediately into the report. This option allows you to insert the generated pictures into the report in a later stage. In order to send them to the inbox, you should choose the command **'Insert & Close'** in the tab **'into Inbox'**.

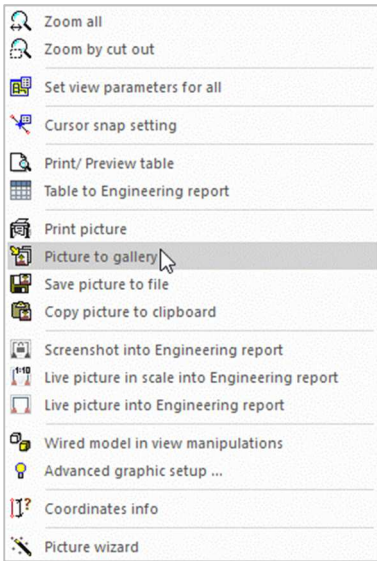


8.3.3. Picture gallery

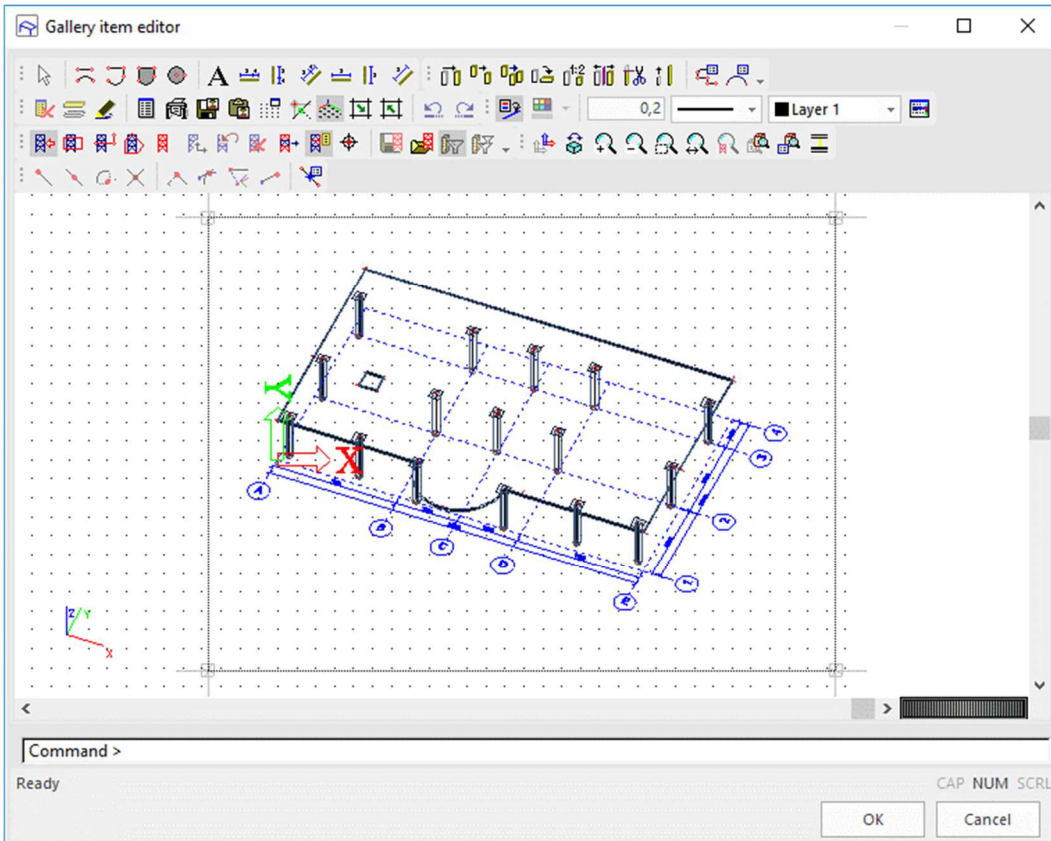
You can send pictures, generated in the model space, to the picture gallery as well. This option allows you to edit the generated picture without adding data – for example dimension lines – to the model. You can access the picture gallery by selecting it in the main tree of SCIA Engineer.



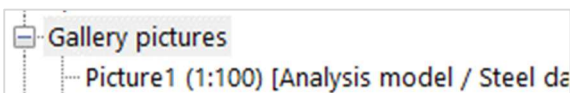
You can send a picture into the picture gallery by performing a right click in the graphical window and selecting the option **'Picture to gallery'**.



As mentioned before, you can modify the picture in the picture gallery without adding data to the model in the model space. In order to modify the picture, you should activate the option **'Edit'** in the gallery.

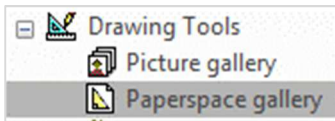


You can add pictures from the picture gallery to the engineering report by selecting the available item **'Gallery pictures'**.

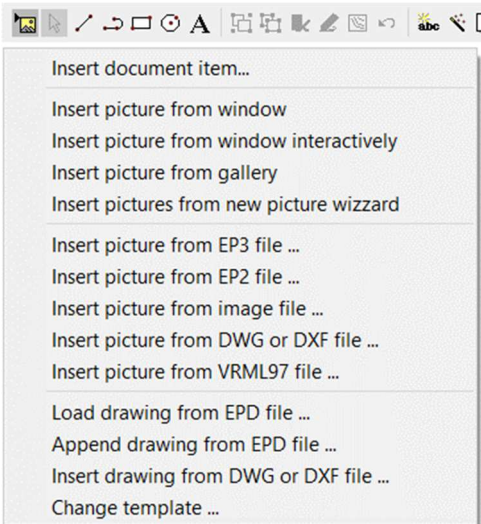


8.3.4. Paperspace gallery

You can find the paperspace gallery in the main tree under the menu '**Drawing Tools**'. The paperspace allows you to create a plan view of the model.



After you have created a new paperspace gallery picture, you can add several items by applying the following functions:

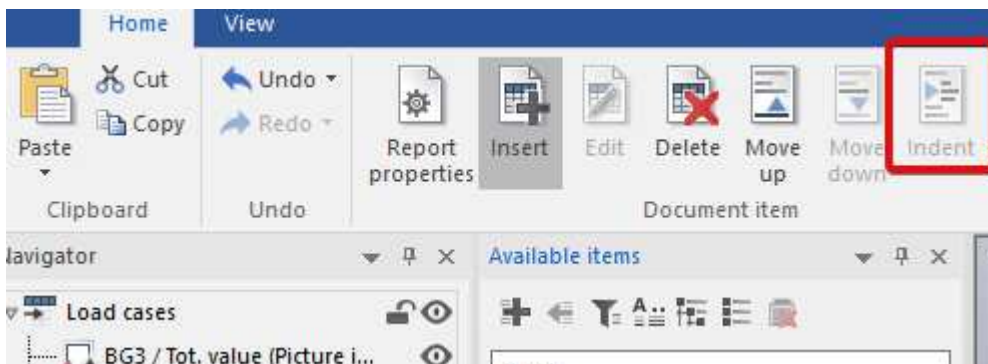


8.3.5. Generating result pictures

In order to create pictures concerning the results of the executed calculation, you have two possibilities: using the shortcut '**Indent**' or adding the '**Result picture generator**'. Both options are explained below.

Indenting

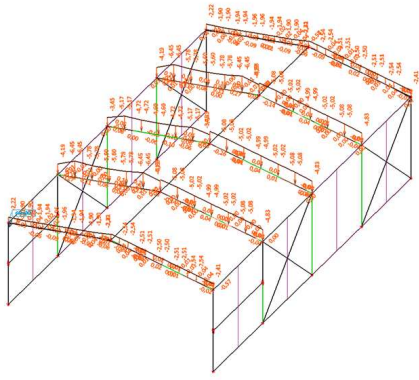
While composing an engineering report, you can execute the command '**Indent**'. By activating this command, meaning indenting a table or a picture under an item, SCIA automatically recognizes the relations between those items and generates the output according to these settings. The following pictures provide a graphical representation of an indented image under a load case.



1.2. Load cases - BG2

Name	Description	Action type	Load group
BG2	Dakbeësting	Permanent Standard	LG1

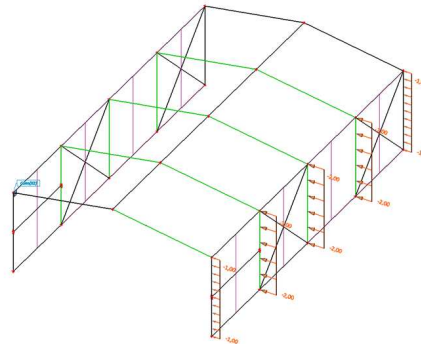
1.2.1. BG3 / Tot. value



1.3. Load cases - BG3

Name	Description	Action type	Load group	Duration	Master load case
BG3	Wind X Standard	Variable Static	LG2	Short	None

1.3.1. BG3 / Tot. value



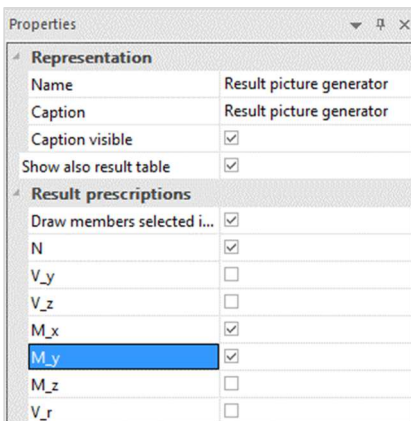
Result picture generator

The result picture generator is an item which you can find under 'Special items'. This item requires an indented result table and an indented live picture.

The result picture generator:

- takes the information about the results from the result table;
- takes the viewpoint of the image;
- views the result set in its properties.

You should define in the properties window of the generator which results need to show in a picture. In the following example, pictures for the selected results 'N', 'M_x' and 'M_y' will be generated.



1. Result picture generator

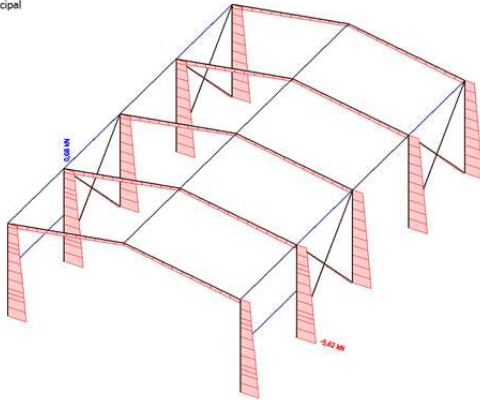
1.1. 1D internal forces

Linear calculation
Load case: BG1
Coordinate system: Principal
Extreme 1D: Global
Selection: All

Name	dx [m]	Case	N [kN]	V _y [kN]	V _z [kN]	M _x [kNm]	M _y [kNm]	M _z [kNm]
S8	0,000	BG1	-5,62	0,07	0,78	0,00	0,00	-0,20
S32	0,000	BG1	0,06	0,00	0,12	0,00	-0,10	0,00
S36	7,071	BG1	-0,45	-0,12	-0,12	0,00	-0,14	-0,15
S35	0,000	BG1	-0,42	0,12	0,12	0,00	-0,15	-0,14
S5	2,500+	BG1	-4,21	0,03	-0,78	0,00	-1,95	0,10
S3	0,000	BG1	-1,16	-0,01	2,41	0,00	-3,73	0,00
S2	0,000	BG1	-1,16	0,01	2,41	0,00	-3,73	0,00
S7	0,000	BG1	-1,21	0,00	2,49	0,00	-3,91	-0,01
S8	5,000	BG1	-2,97	0,04	0,78	0,00	3,90	0,20
S20	5,000	BG1	-2,77	-0,10	0,74	0,00	3,72	-0,31
S16	5,000	BG1	-2,95	0,08	0,78	0,00	3,90	0,21

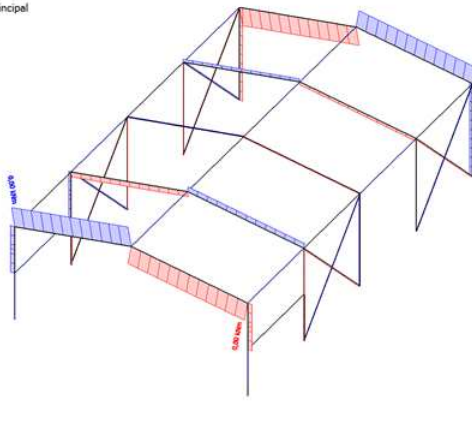
1.2. BG3 / Tot. value - N

Values: N
Linear calculation
Load case: BG1
Coordinate system: Principal
Extreme 1D: Global
Selection: All



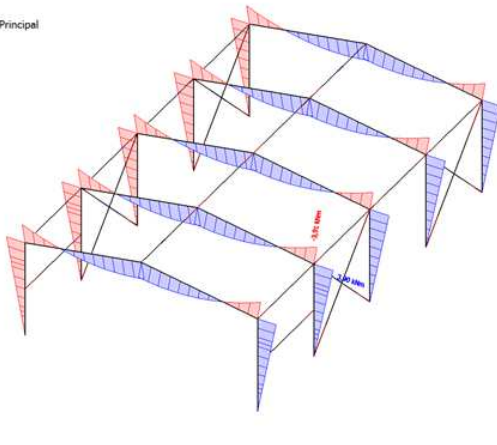
1.3. BG3 / Tot. value - M_x

Values: M_x
Linear calculation
Load case: BG1
Coordinate system: Principal
Extreme 1D: Global
Selection: All



1.4. BG3 / Tot. value - M_y

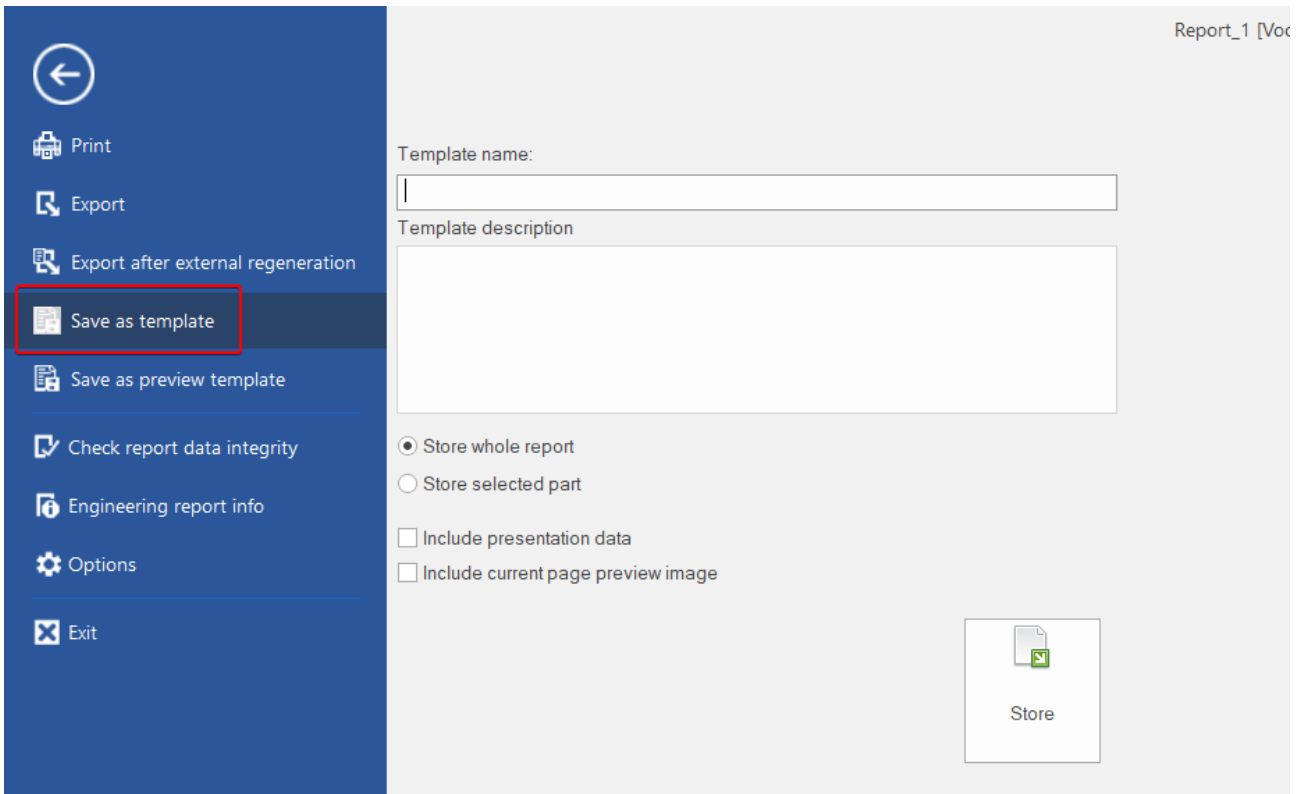
Values: M_y
Linear calculation
Load case: BG1
Coordinate system: Principal
Extreme 1D: Global
Selection: All



8.3.6. Report template

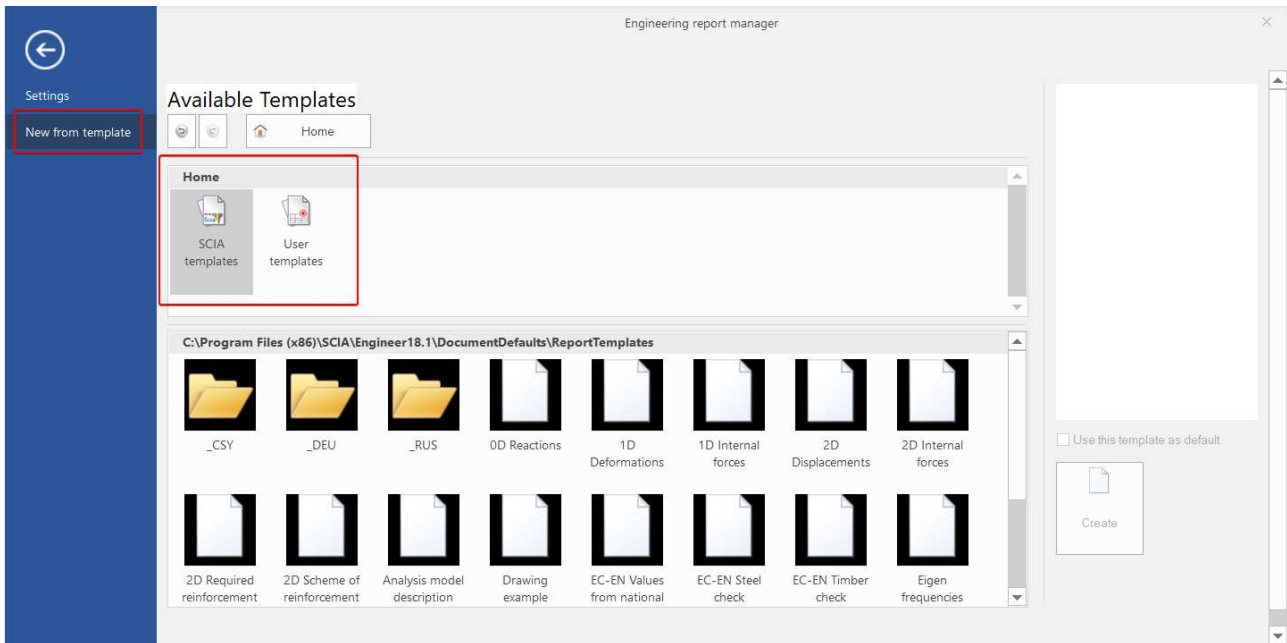
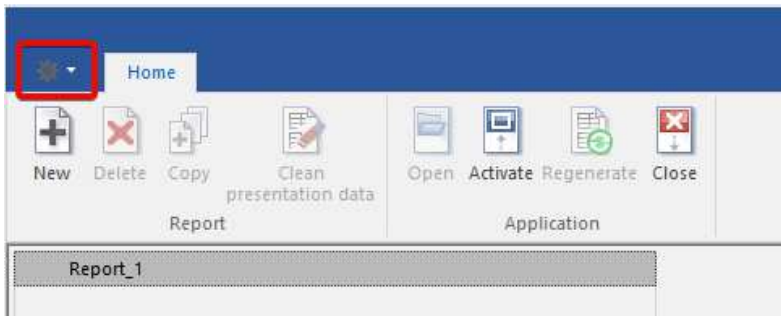
You can save a certain layout of a report as a report template. As soon as you have saved this template, you can import it in a project. The engineering report will then automatically generate all the items of the template according to the new project.

You can save the layout by navigating to the icon of SCIA Engineer and selecting the option '**Save as template**'.



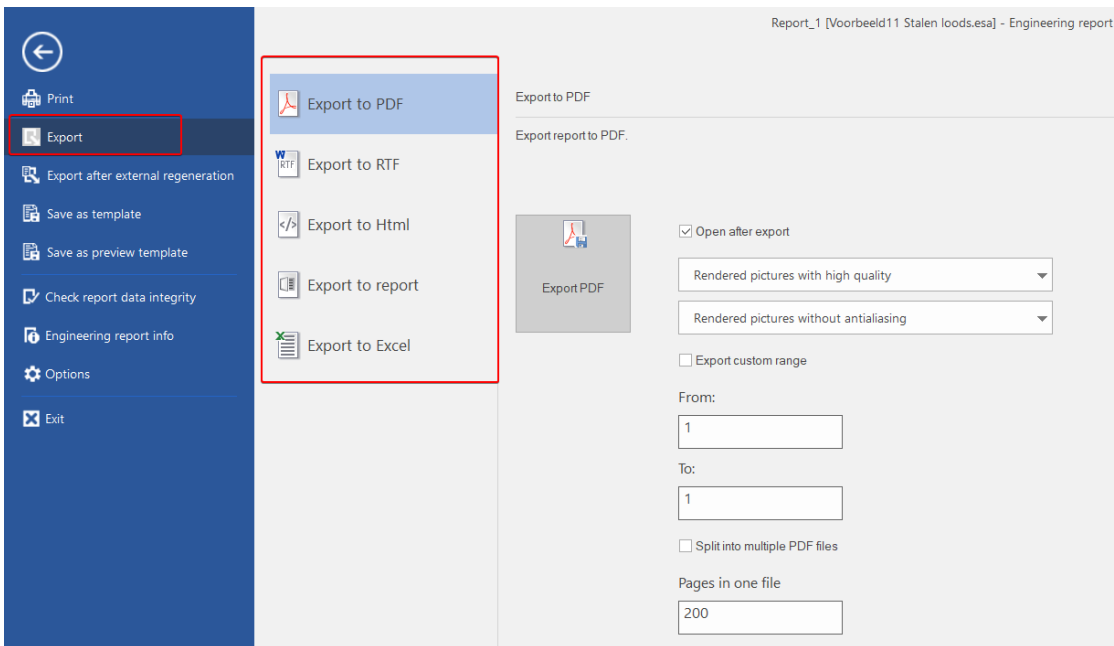
NOTE: it is recommended that you save the template in the folder suggested by SCIA. That allows the software to easily apply the template in your future projects.

The engineering report manager allows you to create a report according to a predefined template by SCIA Engineering or a user's defined template. This is shown in the following pictures.



8.3.7. Export

You can export the engineering report to several formats as well. These formats are displayed in the following picture. You can access the export settings by navigating to the icon of SCIA Engineer and selecting the option 'Export'.



8.3.8. Print

You can print the engineering report by navigating to the icon of SCIA Engineer and selecting the option 'Print'. This is shown in the following picture.

