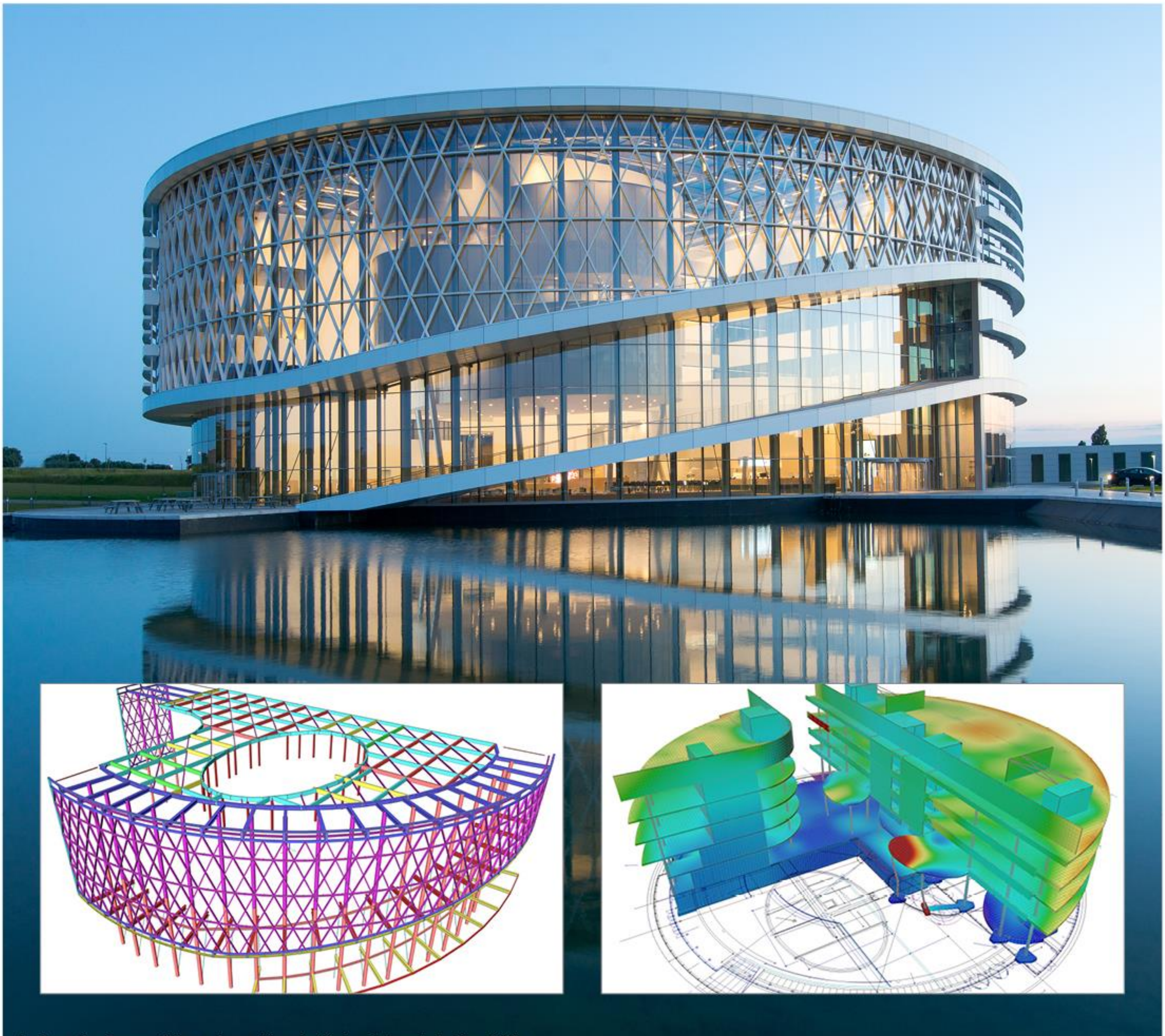


Barco One Campus - © Image Jaspers-Eyers Architects – photography by Marc Detiffe

Basic Concept Training

SCIA Engineer 17

Table of contents

General Information

Modules

Most of the functionalities presented in this course are available in SCIA Engineer Concept Edition.

Other functionalities are not included in this edition and require specific modules. When a section of this course deals with one of these modules, additional information is given.

SCIA Engineer Support service

You can contact the SCIA Engineer support service

- by e-mail
  Send an e-mail to support@scia.net with a description of the problem and the concerning *.esa file, and mention the number of the SCIA Engineer version you are currently working with.
- by telephone
  From Belgium : +32 13 550990
  From the Netherlands : +31 26 3201230
- via the SCIA Support website
  http://www.scia.net/en/company/news/scia-customer-portal

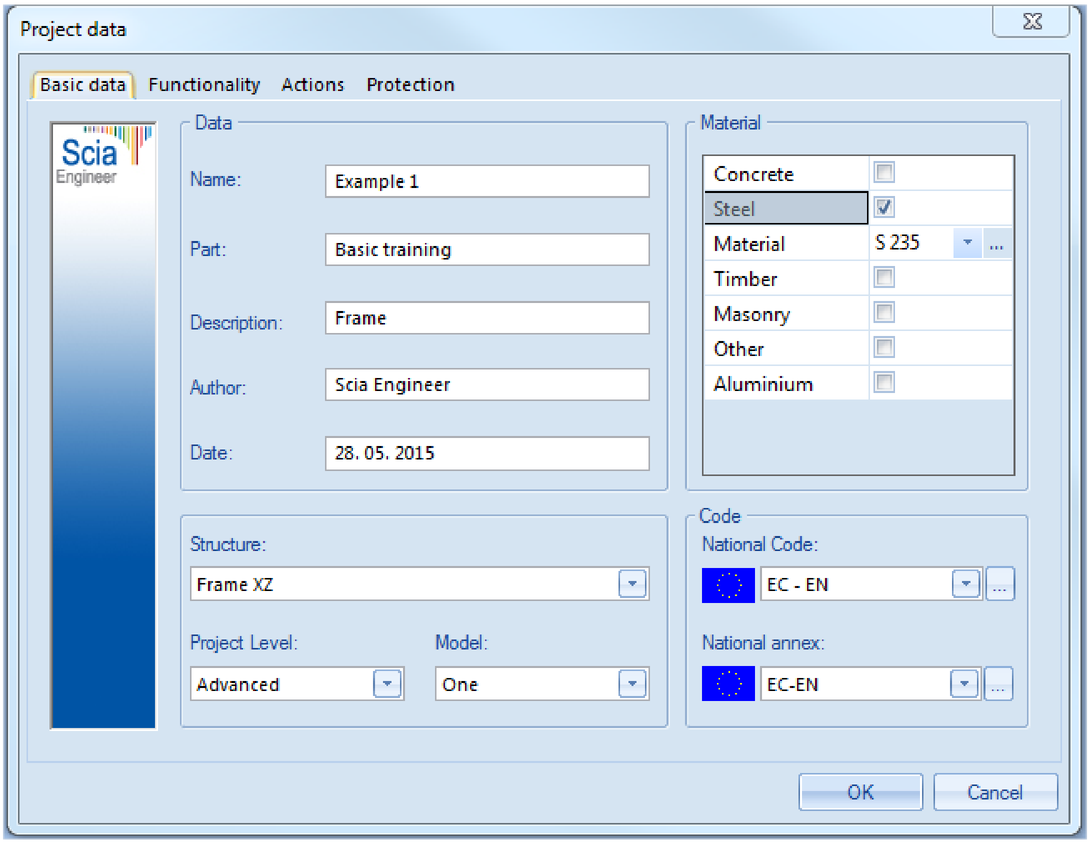
Website

www.scia.net

- Link to eLearning
  http://elearning.scia.net/
- Link to manuals & tutorials
  http://www.scia.net > Support & Downloads > Free Downloads > input your e-mail address > SCIA Engineer > SCIA Engineer Manuals & Tutorials
- Link to the latest SCIA Engineer patch
  http://update.scia.net/

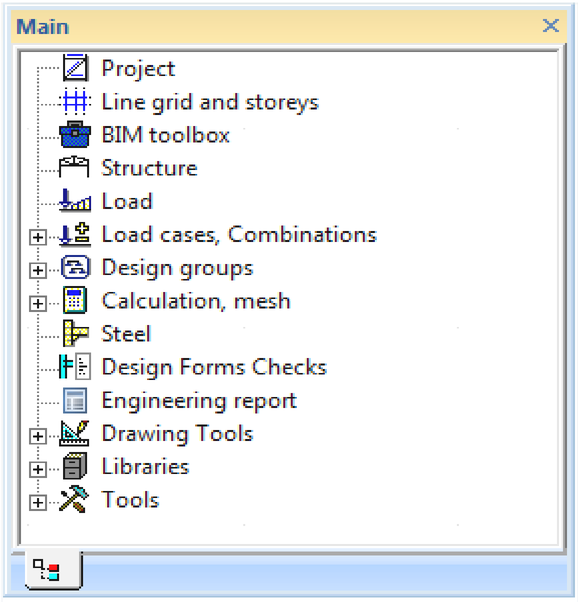
SCIA Engineer – General environment

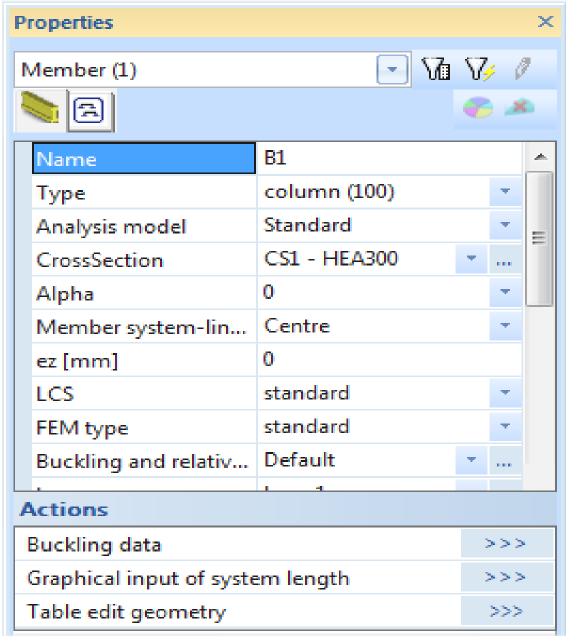
Setup > Options
Help > Contents > Reference guide
File > New > Project data


Project data
Basic data
Functionality
Actions
Protection
Scia
Engineer
Data
Name:
Example 1
Part:
Basic training
Description:
Frame
Author:
Scia Engineer
Date:
28. 05. 2015
Material
Concrete
Steel
Material
S 235
Timber
Masonry
Other
Aluminium
Structure:
Frame XZ
Project Level:
Advanced
Model:
One
Code
National Code:
EC - EN
National annex:
EC-EN
OK
Cancel

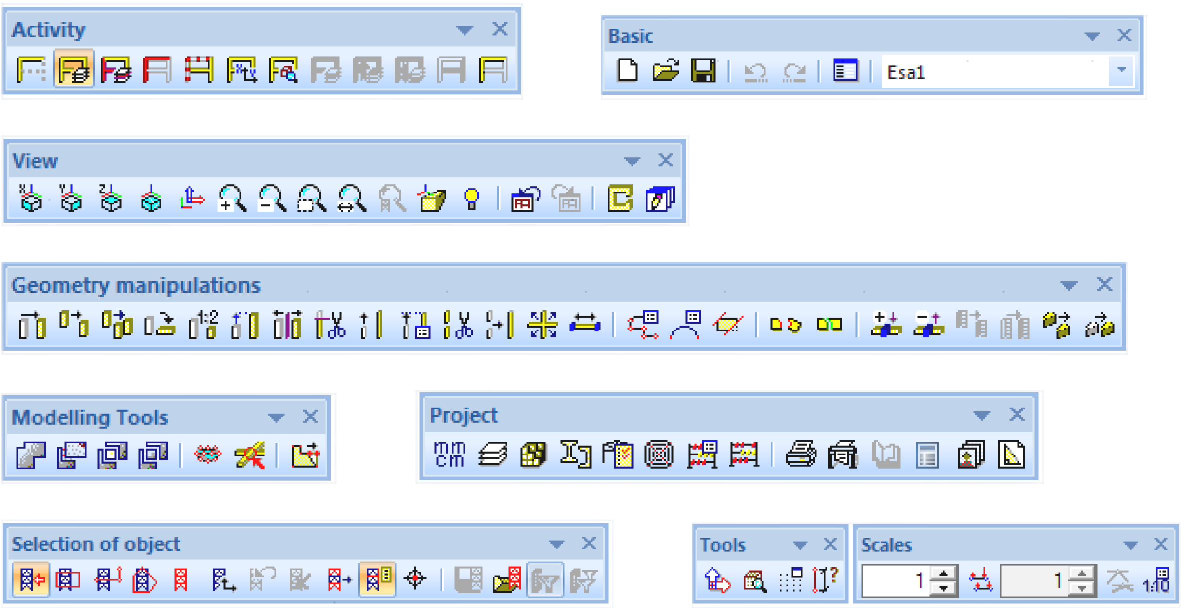
Overview of the menus

Main menu & Properties menu + Actions

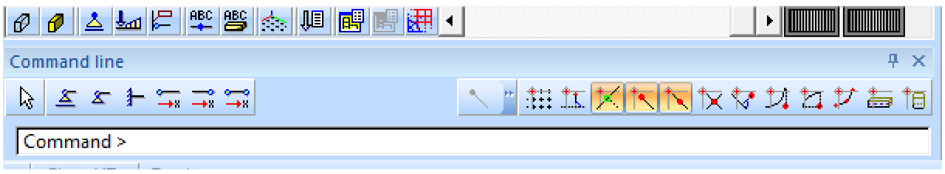




Overview of the toolbars



Command line



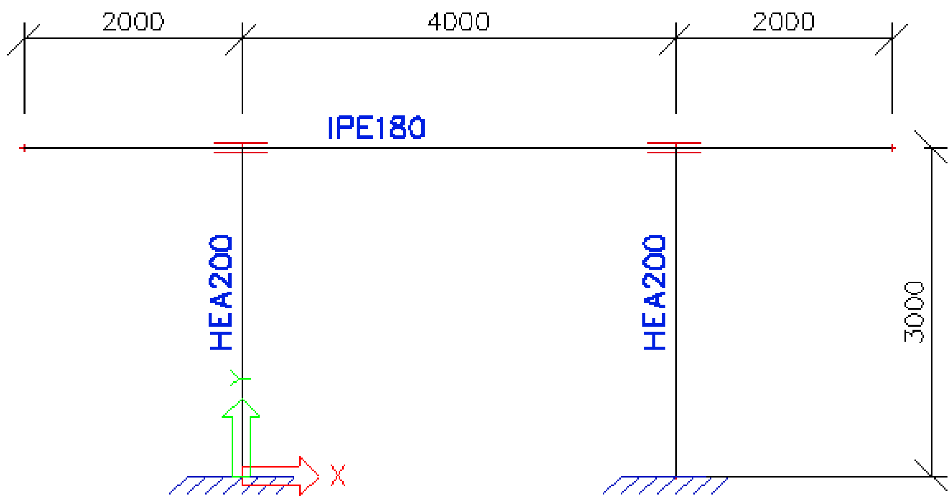
REMARK: If a menu or toolbar has been (accidentally) removed from the project, it can be re-activated via the View menu > Toolbars.

Part 1 – Input of Structural entities

Example 1: Frame

1_Input of geometry

*Project data: Frame XZ – Steel S235



*Adding cross sections which will be used in the project

Libraries > Cross sections > New cross section, or ‘Project’ toolbar 

Adding materials which will be used in the project

Libraries > Materials, or ‘Project’ toolbar 

*Input of members: Structure menu > 1D Member
Define nodes via
-Command line Absolute co-ordinates 0 0 of 0;0
Relative co-ordinates @

-Raster points Dot grid, on ‘Tools’ toolbar 

Line grid, on ‘Tools’ toolbar 

Snap to the raster points by means of Cursor snap settings, on Command line toolbar 

After input, you can adapt the geometry of a selected entity via Actions > Table edit geometry & adapt the properties via Properties menu

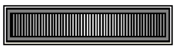
*Input of supports: Structure menu > Model data > Support
Select one or more existing nodes

REMARK: Instructions are being shown on the Command line!

2_Display on screen

*Manipulations

-‘View’ toolbar 

-Scroll bars, at the right bottom of the work area 
-Hotkeys SHIFT + right mouse button > Move
CTRL + right mouse button > Rotate
SHIFT + CTRL + right mouse button > Zoom

*Selection of entities

-'Selection of object' toolbar

-Left mouse button — Frame from left to right > All entities which are located entirely in the frame, are selected
Frame from right to left > All entities which are located entirely in the frame or are intersected by the frame, are selected

-At the top of the Properties window

Select elements by property

Select elements by more properties

-Command line > type 'SEL' commando + name of entity (e.g. SEL K1)

*Deselection of entities
-Deselect all, using ESC key
-Deselect one entity at a time, via CTRL key + click on entity with left mouse button

*Display of structure

-Limited, via Command line toolbar: Rendering of structure , Display of supports , Display of names of nodes & beams

-Detailed, via Command line toolbar: Set view parameters for all/for selection , or via right mouse click in screen

3_Actions AFTER input of geometry

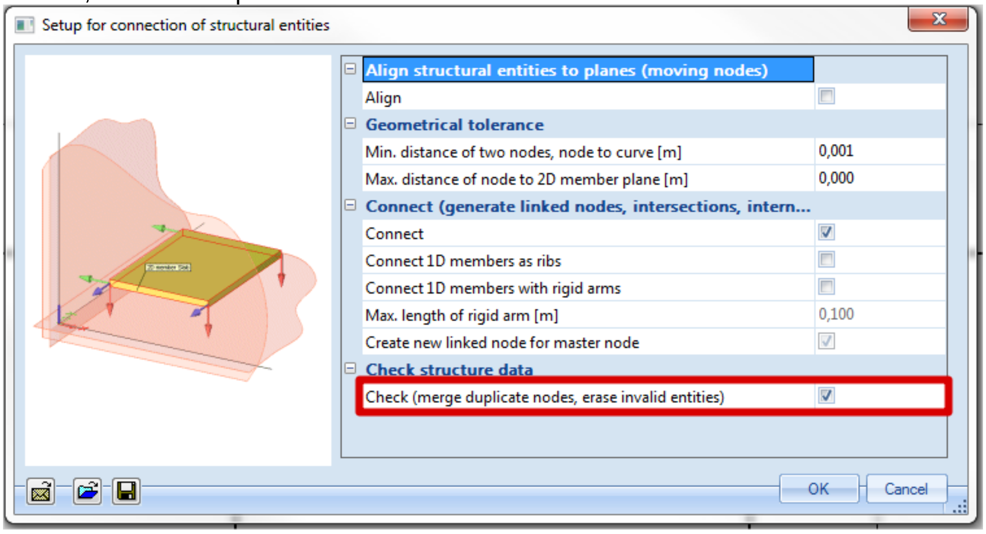
Two actions always have to be performed after input of the geometry, to avoid problems during calculation:

*Structure menu > Check structure data, or 'Project' toolbar
Duplicate nodes and beams, and incorrect entities are detected and removed. Also the additional data are being checked.

*Structure menu > Model data > Connect members/nodes, or 'Geometry manipulations' toolbar
Nodes which coincide with beams, and edges (of 2D members) which intersect with beams, are connected to the concerned beams. See also Annex 1.
Attention: Previous to this action everything has to be deselected, only then the entire structure is connected. In the other case, SCIA Engineer looks for connections only in the selection.

In this example the end nodes of the columns are connected to the beam, see the double red lines around the connecting nodes. To show/hide these lines on the screen, see Command line toolbar

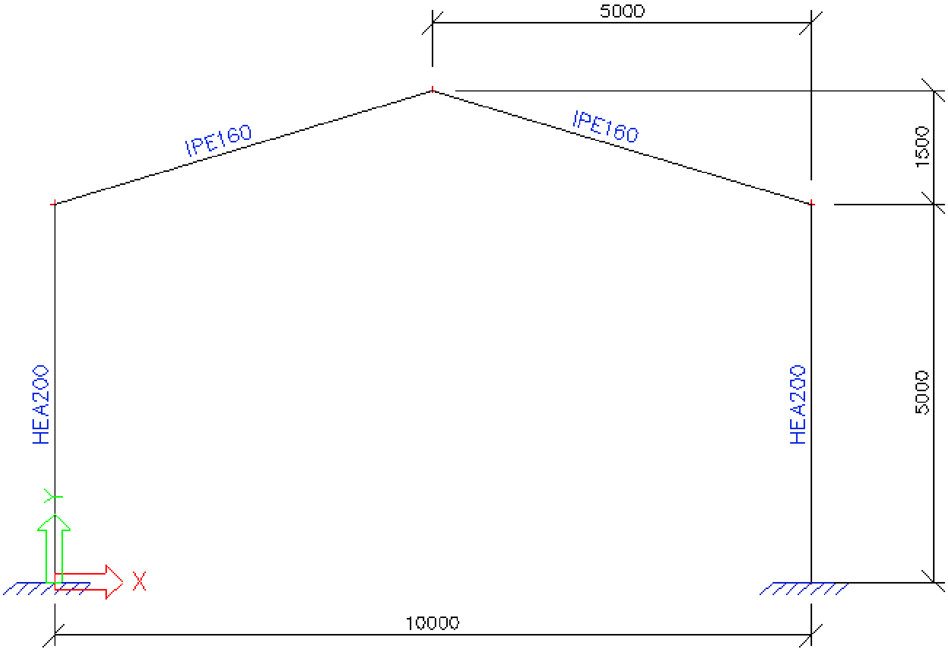
*OR: It is possible to execute both actions at the same time > In the window Connection of structural entities, select the option Check structure data



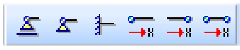
Example 2: Frame

1_Input of geometry

*Project data: Frame XZ – Steel S235



*Input of members
-Left part of frame, via Structure menu > 1D Member; afterwards Mirror option via 'Geometry manipulations' toolbar 
-Complete frame, via Structure menu > Advanced Input > Catalogue blocks; choose for Frame 2D

*Input of supports
-Structure menu > Model data > Support
-Fast input of supports (and hinges) via Command line toolbar 

2_Manipulations

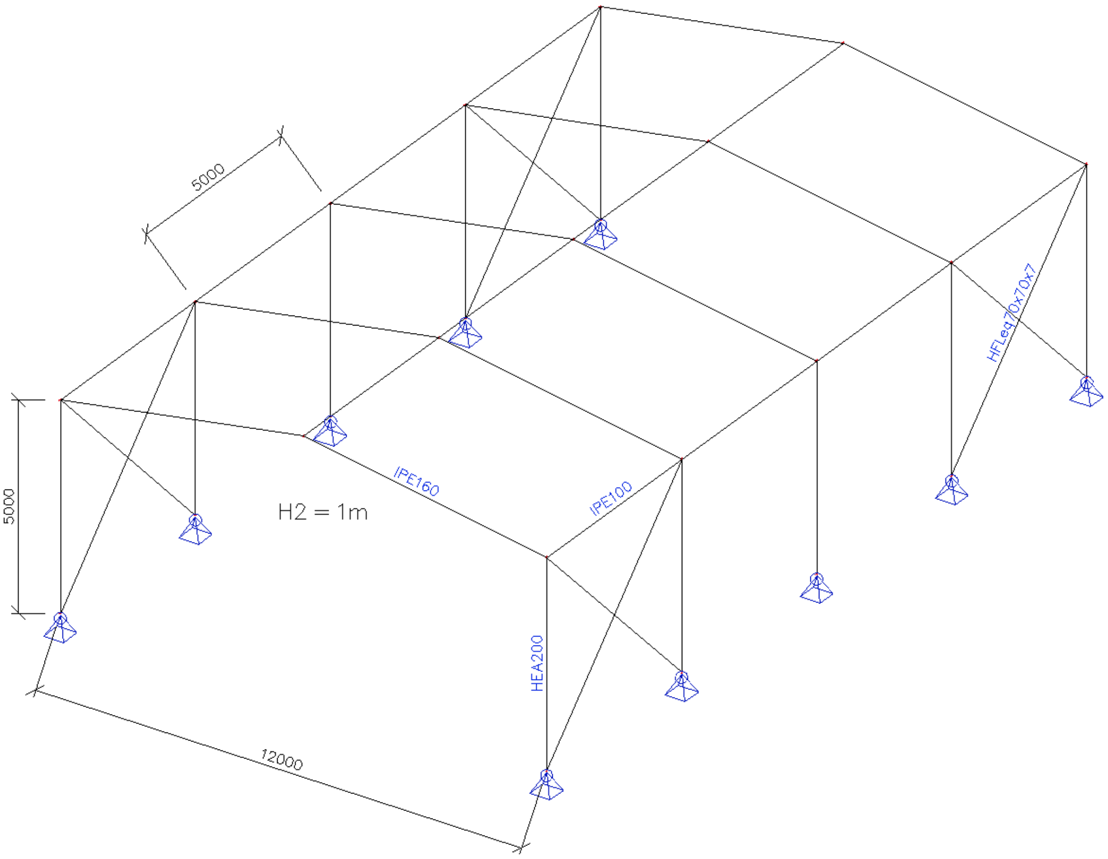
To move nodes:
First select node, afterwards
-Drag node with left mouse button
-Change co-ordinates of the node in the Properties menu
-Move node, via 'Geometry manipulations' toolbar , or via right mouse click in the screen

3_Actions after input

*Check structure data 
*Connect members/nodes  (Attention: connect the entire structure!)

Example 3a: Steel hall

1_Input of geometry

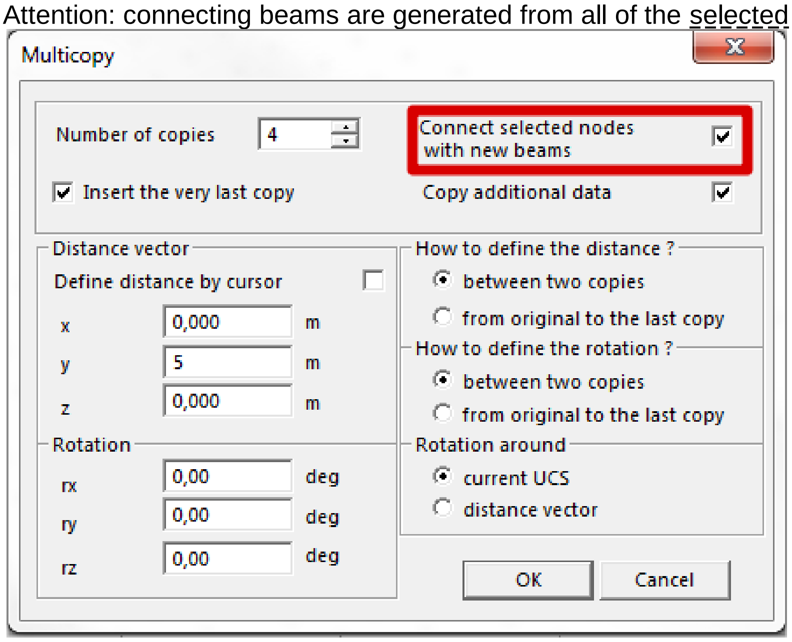


*Input of first frame: analogous to Example 2

*Copy the first frame: via 'Geometry manipulations' toolbar

-Copy ; input afterwards manually the connecting beams

-Multicopy ; generate the connecting beams automatically
Attention: connecting beams are generated from all of the selected nodes.



*Input of supports
Select first the nodes at the bottom of the columns, afterwards

-Filter Properties window > Selection of all nodes with co-ordinates Z = 0

-Select by working plane, see 'Selection of object' toolbar > Selection of all elements which lie exactly in the Active working plane, see Plane XY at the bottom of the Command line

2_Actions after input

*Check structure data

*Connect members/nodes (Attention: connect the entire structure!)

3_Structural model

*Main menu > Project > Functionality: Structural model

-Activate Rendering, see Command line toolbar

-Generate structural model, see 'View' toolbar
-Alter priorities via Setup > Beam types (structural)

*Alternating between Structural model and Analysis model via Select view parameters for all > Structure > Model type, or via View > Set view parameters

Structural model = Presentation model, but also necessary for the input of steel connections, anchoring of reinforcement, …
Attention: modifications in the Structural model (e.g. eccentricities) are not taken into account for the calculation!

4_Display of screen

-Set view parameters for all/for selection, via right mouse click in screen

-Fast adjustment of view parameters, see Command line toolbar

For example: Check if the correct cross-sections have been inputted
Set view parameters for all > Structure > Style & colour = Colour by cross-section

-Alter colours, fonts, background colour etc.
via Setup > Colours/Lines > Palette settings; the settings for Screen, Document and Graphic output are done via separate tabs

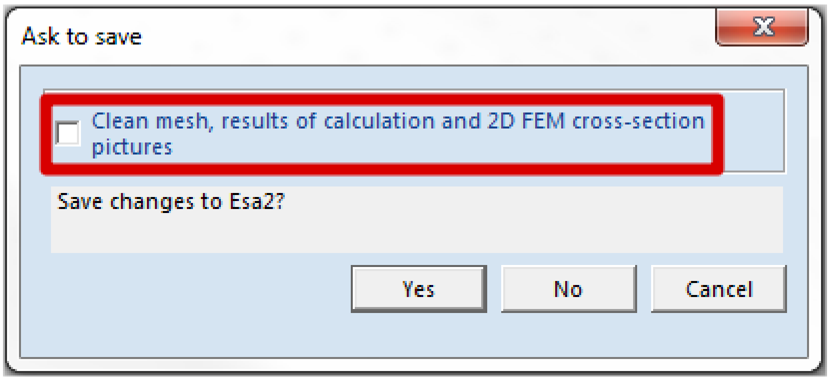
5_Activity & Visibility

*Define layers, via 'Project' toolbar
-Current used activity: defines if the layer is visible or not
-Structural model only: when set to 'yes' the layer is NOT taken into account for the calculation

*'Activity' toolbar > The entire Analysis model is taken into account for the calculation, but only part of it may be visible

6_Saving a file

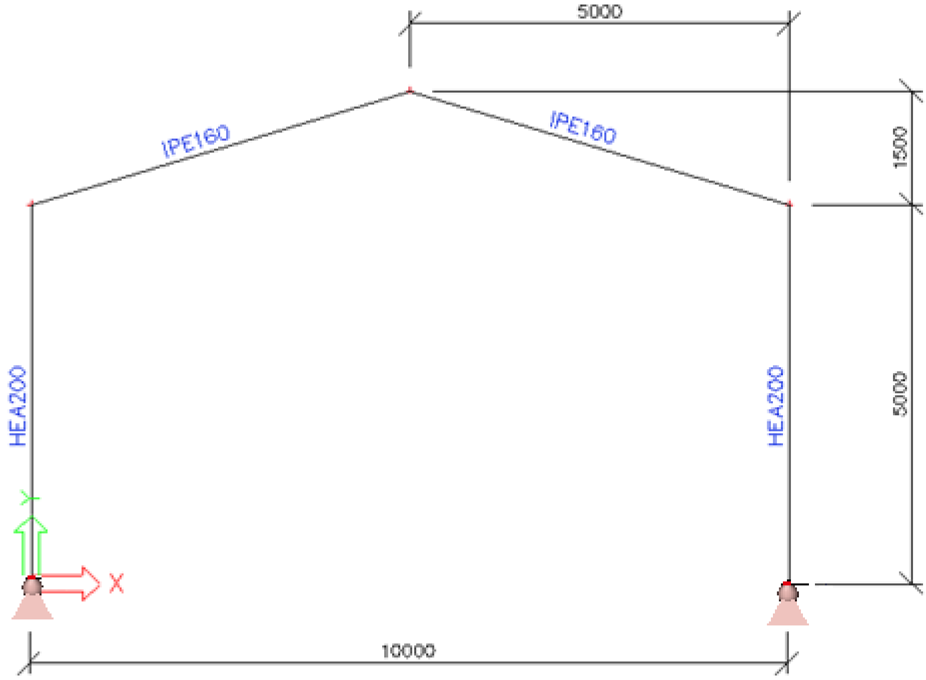
Select the option 'Clean mesh, results of calculation' if you want to remove these parts from the saved file. The size of the file is in this way considerably reduced, but when the file is reopened in SCIA Engineer it is necessary to calculate again to view the results.



Example 3b: Steel hall

1_Input of frame geometry

*Project data: Frame XYZ – Steel S235 – Purlins IPE 100 – H2 = 1,5m

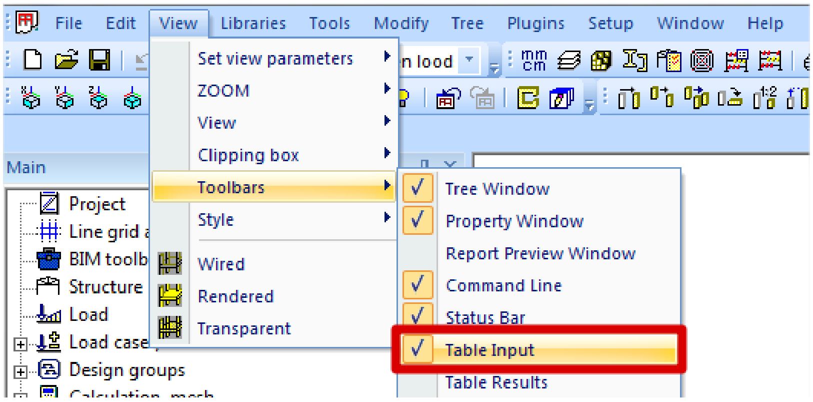


*Input of first frame: analogous to Example 2

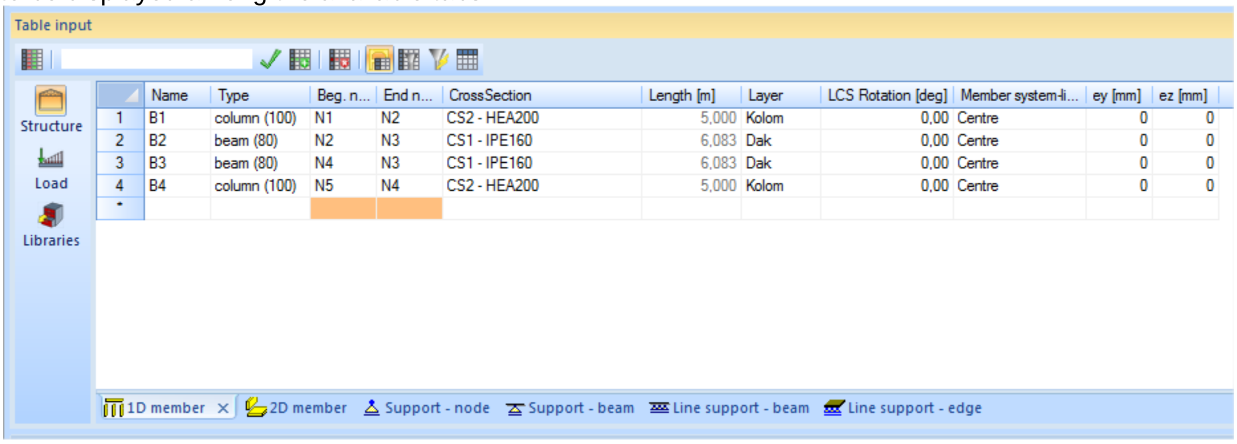
2_Table input

Table input is a functionality introduced since SCIA Engineer 2011. It enables the user to numerically introduce or edit project data. Numerical data can also be handled simply by a Copy/Paste from SCIA Engineer into Excel and vice versa.

To be able to use the Table input, you have to display it through View > Toolbars > Table input



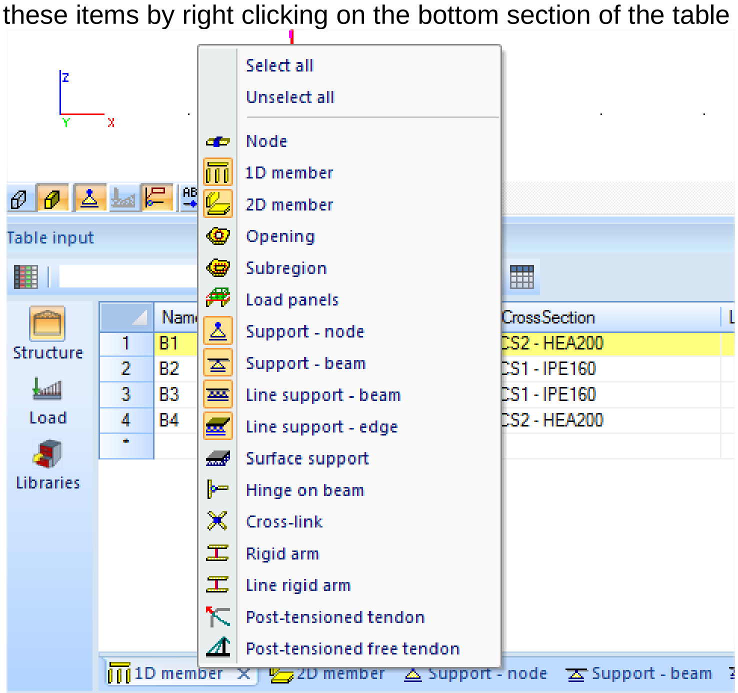
The menu is displayed under the command line but can be dropped into any other position like this is already possible for other menu windows (main menu, properties menu…). You can open the different tables using the tabs that are at the bottom of the Table editor. You can choose the data table that has to be displayed among the available tabs.



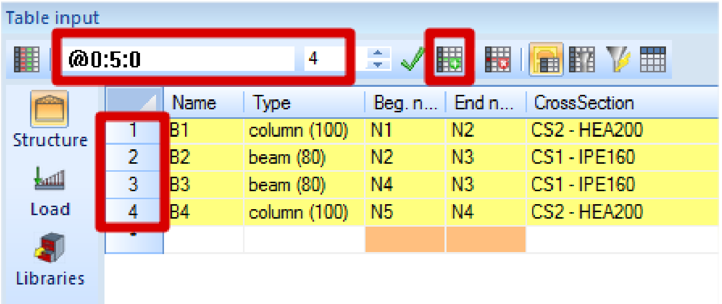
	Name	Type	Beg. n...	End n...	CrossSection	Length [m]	Layer	LCS Rotation [deg]	Member system-li...	ey [mm]	ez [mm]
1	B1	column (100)	N1	N2	CS2 - HEA200	5,000	Kolom	0,00	Centre	0	0
2	B2	beam (80)	N2	N3	CS1 - IPE160	6,083	Dak	0,00	Centre	0	0
3	B3	beam (80)	N4	N3	CS1 - IPE160	6,083	Dak	0,00	Centre	0	0
4	B4	column (100)	N5	N4	CS2 - HEA200	5,000	Kolom	0,00	Centre	0	0

In the above table you can see the inputted frame with its properties in a table. These properties can be edited in this table. The user can also input new elements via this table input. This will be done for the input of the rest of the structure.

Notice the three possibilities that you can edit in the table input (Structure, Load, Libraries). At the bottom you can find for instance in the structure section: 1D member, support node,... you can expand these items by right clicking on the bottom section of the table input window.



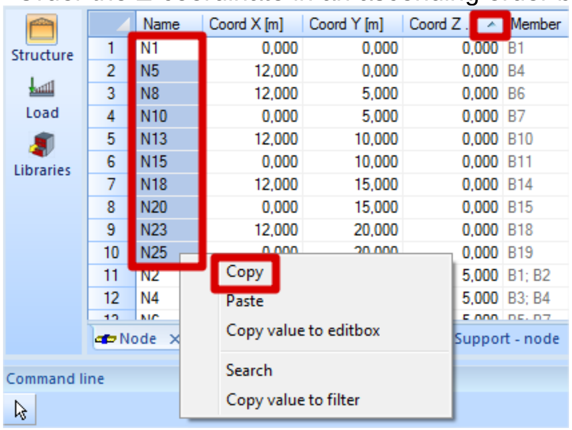
3_Multicopy with table input



- Select the 4 beams by selecting the 4 rows of the 1D-members.
- Fill in the text box of the 1D-members the following relative coordinate: @ 0 ; 5 ; 0
- And set the repetition on 4 (= 4 copies)
- Start the multicopy by clicking on

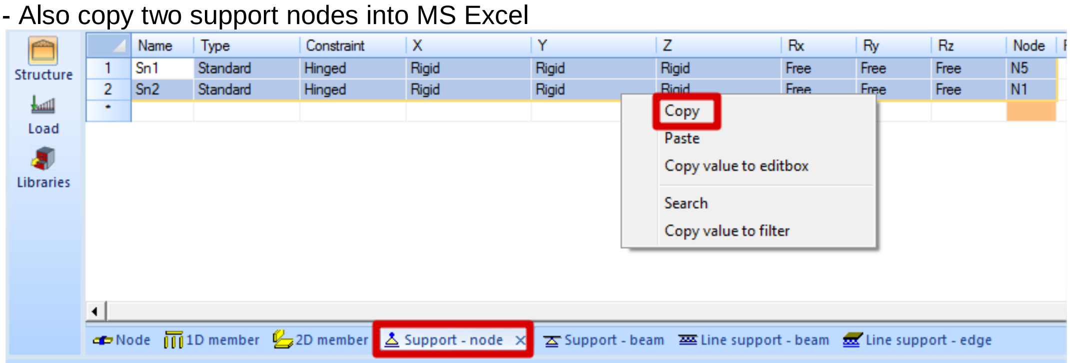
4_Inputting structure elements with table input

The nodal supports of the columns need to be added.
- Right mouse click at the bottom of the table input window and add the nodes tab.
- Order the Z-coordinate in an ascending order by clicking on Coord Z (arrow facing upwards).



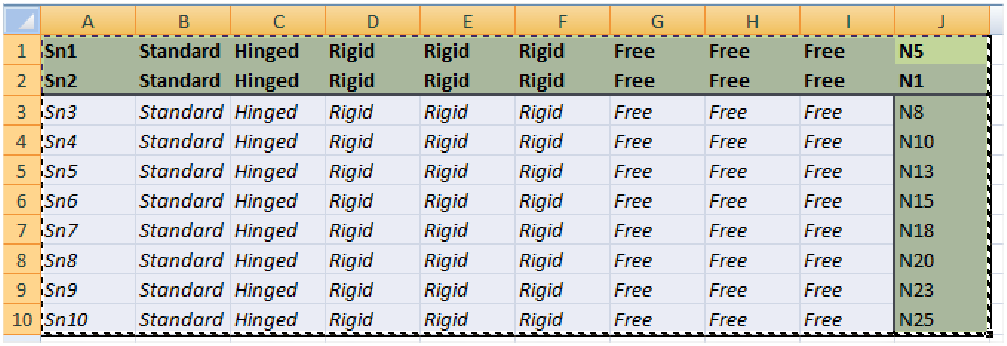
Copy the names of the nodes with a Z=0m and paste it in MS Excel

- Also copy two support nodes into MS Excel



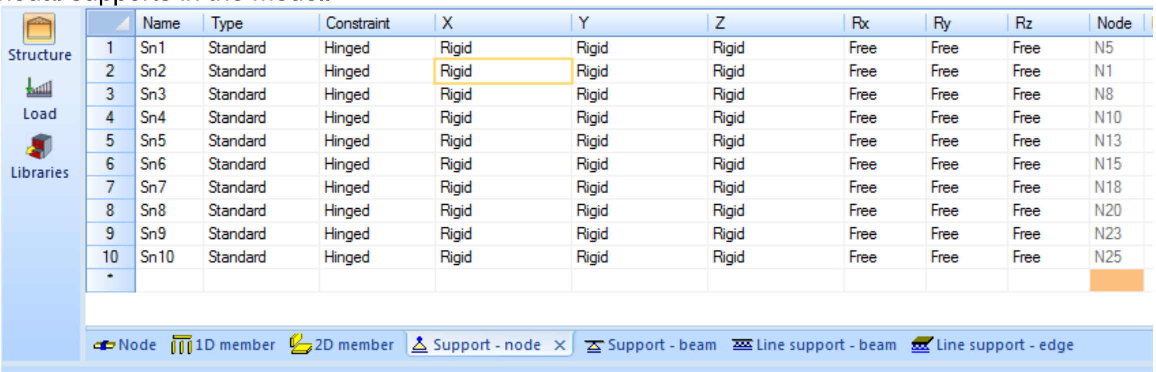
- Now try to compose a table with 10 identical nodal supports.

	A	B	C	D	E	F	G	H	I	J	K	L
1	**Sn1**	**Standard**	**Hinged**	**Rigid**	**Rigid**	**Rigid**	**Free**	**Free**	**Free**	**N5**		N1
2	**Sn2**	**Standard**	**Hinged**	**Rigid**	**Rigid**	**Rigid**	**Free**	**Free**	**Free**	**N1**		N5
3	*Sn3*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N8
4	*Sn4*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N10
5	*Sn5*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N13
6	*Sn6*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N15
7	*Sn7*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N18
8	*Sn8*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N20
9	*Sn9*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N23
10	*Sn10*	*Standard*	*Hinged*	*Rigid*	*Rigid*	*Rigid*	*Free*	*Free*	*Free*			N25



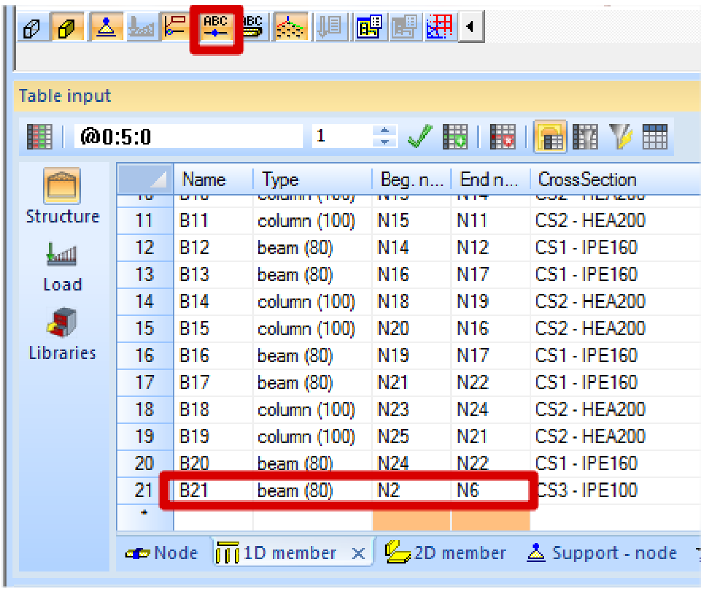
	A	B	C	D	E	F	G	H	I	J
1	Sn1	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N5
2	Sn2	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N1
3	Sn3	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N8
4	Sn4	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N10
5	Sn5	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N13
6	Sn6	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N15
7	Sn7	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N18
8	Sn8	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N20
9	Sn9	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N23
10	Sn10	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N25

- Next copy this table from MS Excel to the nodal support section of the table input in order to insert the nodal supports in the model.



	Name	Type	Constraint	X	Y	Z	Rx	Ry	Rz	Node
1	Sn1	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N5
2	Sn2	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N1
3	Sn3	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N8
4	Sn4	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N10
5	Sn5	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N13
6	Sn6	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N15
7	Sn7	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N18
8	Sn8	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N20
9	Sn9	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N23
10	Sn10	Standard	Hinged	Rigid	Rigid	Rigid	Free	Free	Free	N25

- Go to the 1D-members and try inputting manually one purlin (IPE100) by only entering the name, the type, beg node & end node. SCIA Engineer will then insert this beam into the model with a random cross section. This cross section can then be changed to the correct cross section.



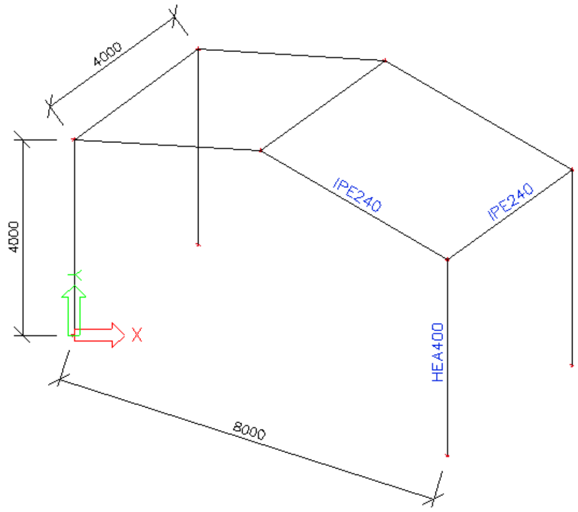
	Name	Type	Beg. n...	End n...	CrossSection
11	B11	column (100)	N15	N11	CS2 - HEA200
12	B12	beam (80)	N14	N12	CS1 - IPE160
13	B13	beam (80)	N16	N17	CS1 - IPE160
14	B14	column (100)	N18	N19	CS2 - HEA200
15	B15	column (100)	N20	N16	CS2 - HEA200
16	B16	beam (80)	N19	N17	CS1 - IPE160
17	B17	beam (80)	N21	N22	CS1 - IPE160
18	B18	column (100)	N23	N24	CS2 - HEA200
19	B19	column (100)	N25	N21	CS2 - HEA200
20	B20	beam (80)	N24	N22	CS1 - IPE160
21	B21	beam (80)	N2	N6	CS3 - IPE100

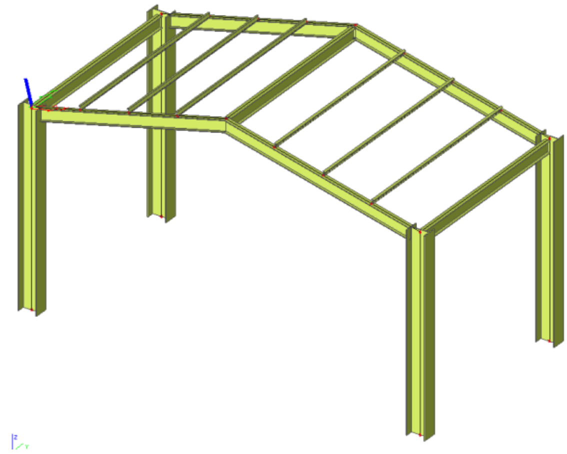
Now try inserting the remaining 1D-elements into the model.
Extra functionalities : see annex 5

Example 4: Purlins

1_Input of geometry

*Project data: Frame XYZ – Steel S235 – Purlins IPE 100 – H2 = 1m





*Input of purlins_Method 1
-Input of purlins at the left side

by means of Cursor snap settings > Select option h: divide into 4 parts

-Rotate the purlins, to get them perpendicular to the roof plane
This means: local z-axis of purlins = local z-axis of main girder

Relocation of UCS to roof plane, via 'Tools' toolbar
Select the purlins > Properties menu: LCS = z from UCS

-Input of eccentricity

Select the purlins > Properties menu:
Member system line at: relative eccentricity, move member with regard to system line
ey, ez: absolute eccentricity

-Copy the purlins to the right side

Copy and Rotate , via 'Geometry manipulations' toolbar
Or use Mirror , via 'Geometry manipulations' toolbar
> Plane of symmetry: 1st direction of plane = Z from UCS; 2nd direction = to define by user

*Input of purlins_Method 2
Structure menu > Panel – Load to beams
Result: model of a fictive 2D member and real 1D members

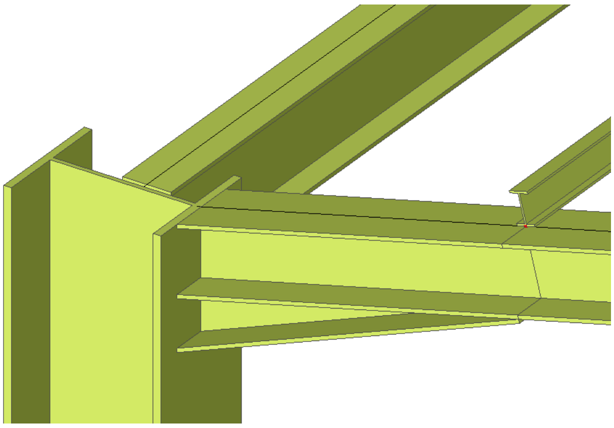
2_Activate Structural model

Main menu > Project > Functionality: Structural model

Generate structural model, see 'View' toolbar
Or via View > Set view parameters > Generate structural model

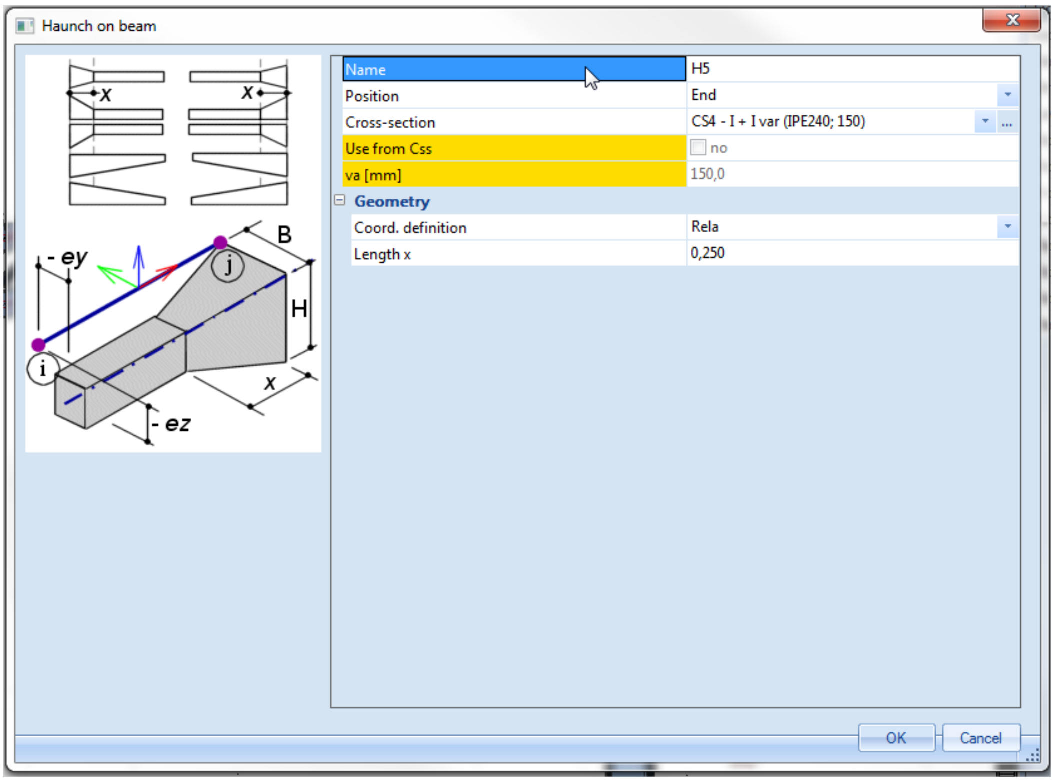
Attention: Eccentricities in Analysis model and Structural model have to be inputted separately in the Properties menu.

REMARK: It might be necessary to generate the Structural model again after certain actions or adjustments to the model.



3_Input of Haunch

Structure menu > 1D Member > 1D member components > Haunch
Add new cross-section first, type I + Ivar
Afterwards define the haunch as follows:



Haunch = additional data to an entity (just like supports, charges, …)
It is possible to copy additional data

-via ‘Geometry manipulations’ toolbar 
-via right mouse click in screen, choose option Copy add data

Extra possibility: Input of Arbitrary profile
Structure menu > 1D Member > 1D member components > Arbitrary profile
Divide member into a number of sections with different cross-sections / different geometrical properties
e.g. Haunch with different dimensions at the beginning and the end of the beam

4_Actions after input

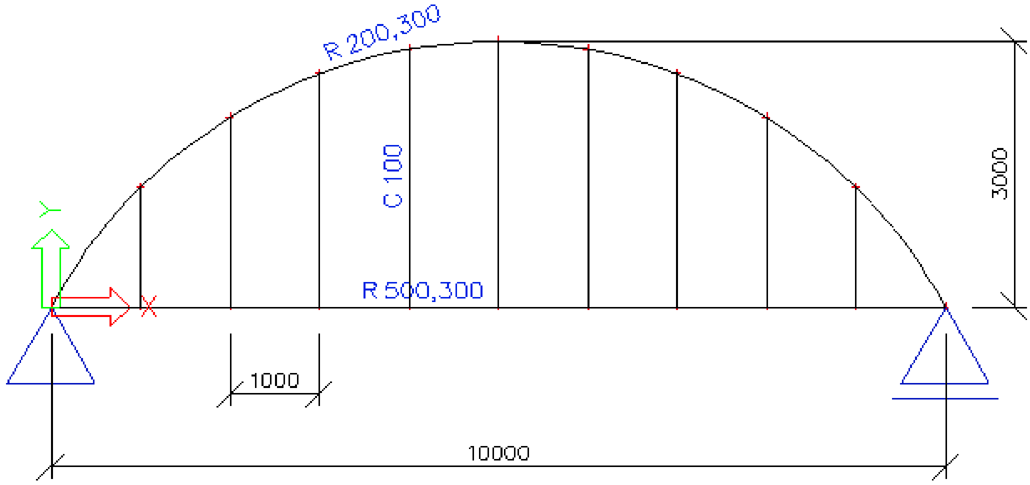
*Check structure data 

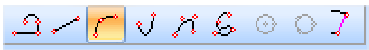
*Connect members/nodes  (Attention: connect the entire structure!)

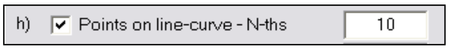
Example 5: Bridge

1_Input of geometry

*Project data: Frame XZ – Concrete & Steel



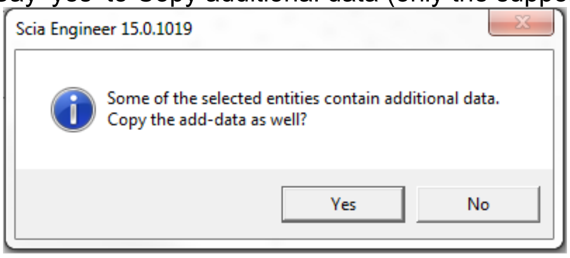
*Input of curved beam
Structure menu > 1D Member > Member
New circular arc, via Command line toolbar 

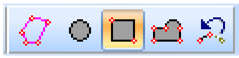
*Input of steel tension only members
-Cursor snap settings  > Select option h 

In this way it is possible to snap to each 10th part of a member.
-Structure > 1D Member > Column; length of all columns = 3m
-Cut columns at the height of the arc > Trim, see ‘Geometry manipulations’ toolbar 

2_From 2D to 3D

Main menu > Project > Project data: change Structure type
*Frame XYZ
Copy arc: Copy ; spacing is 4m in Y direction (@0;4;0)
Say ‘yes’ to Copy additional data (only the supports in this case)



*General XYZ
Add concrete roadway: Structure > 2D Member > Plate
New rectangle, via Command line toolbar 

REMARK: It is only possible to switch to a ‘higher’ Structure type!

3_Actions after input

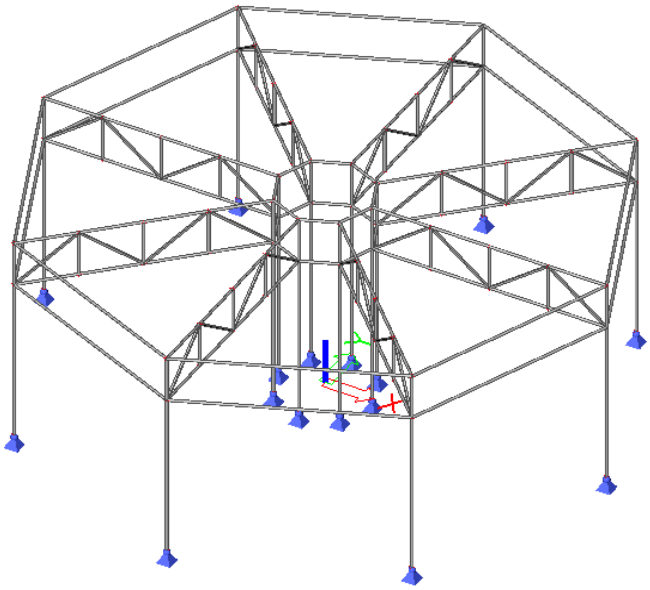
*Check structure data 

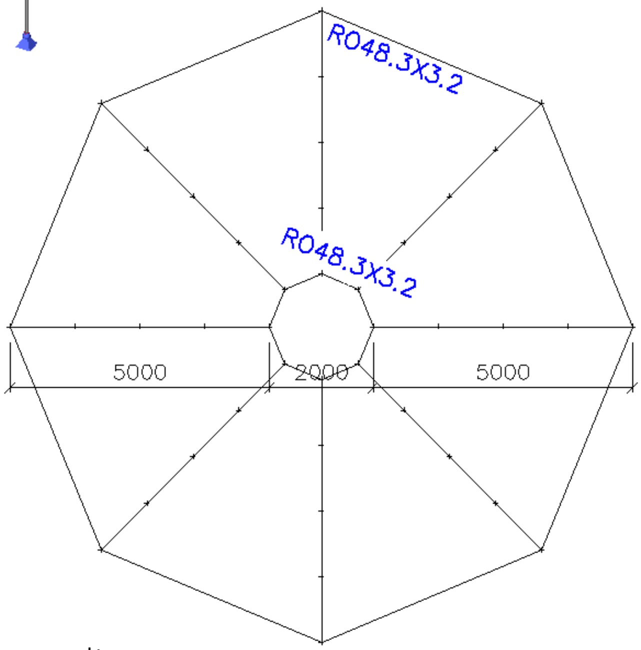
*Connect members/nodes  (Attention: connect the entire structure!)

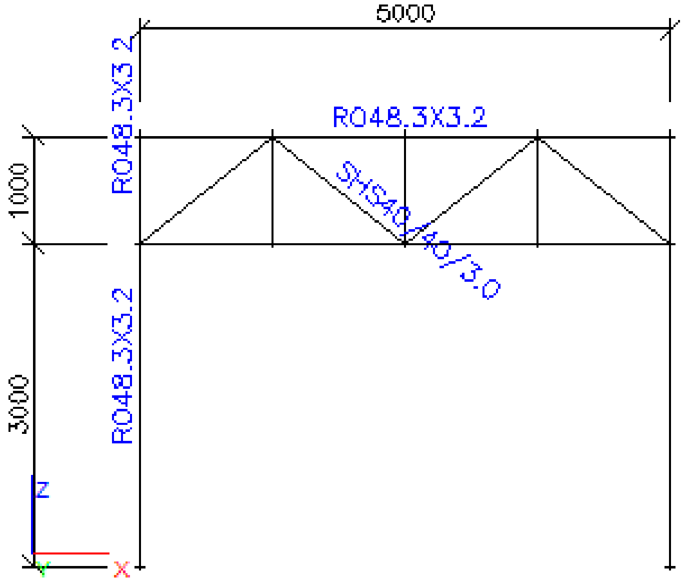
Example 6: Carrousel

1_Input of geometry

*Project data: Frame XYZ – Steel S235







*Input of one frame

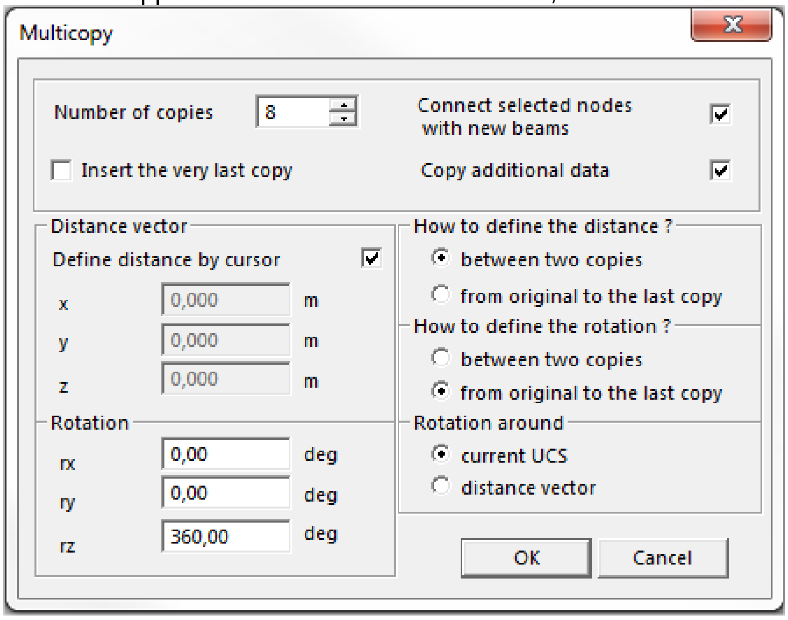
Structure menu > 1D Member > Column
Structure menu > Advanced Input > Catalogue blocks: Frame 2D

Move the frame so the bottom node of the left column coincides with co-ordinate 1;0;0

Or move UCS, see ‘Tools’ toolbar 

*Multicopy, via ‘Geometry manipulations’ toolbar 
Copy + Rotation at same time: around current UCS

-Let generate connecting beams automatically
Attention: connecting beams are being generated from all of the selected nodes.
-Copy additional data
In this case only supports; if loads, hinges etc. are added to the original frame, those are copied to the new frames as well.
Attention: supports are additional data to nodes, not to members.

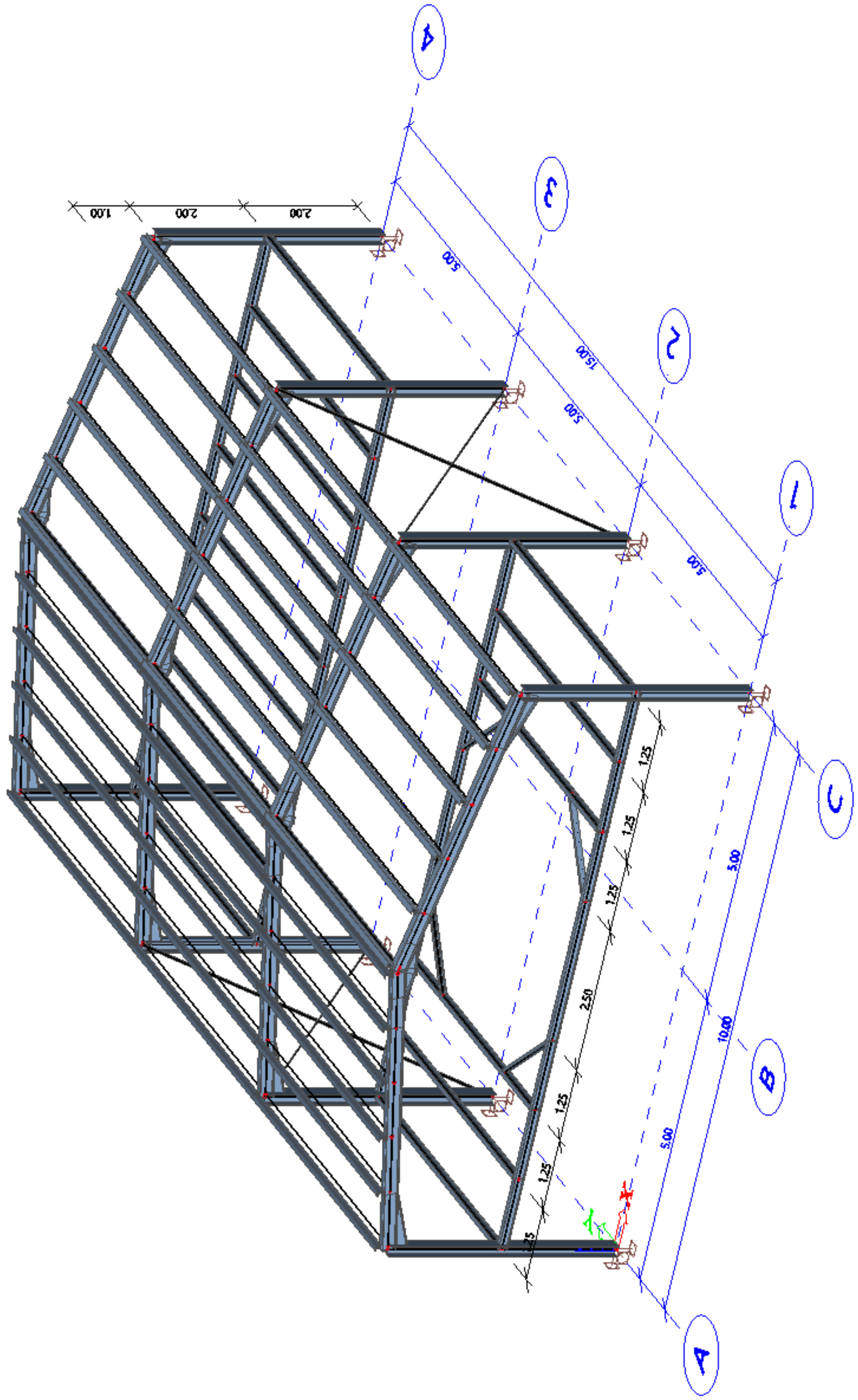


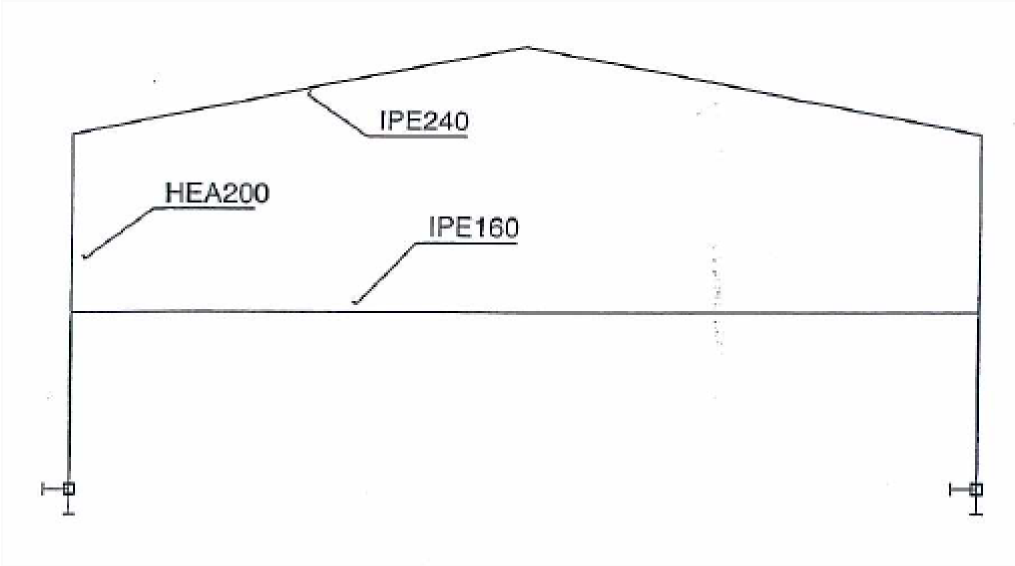
2_Actions after input

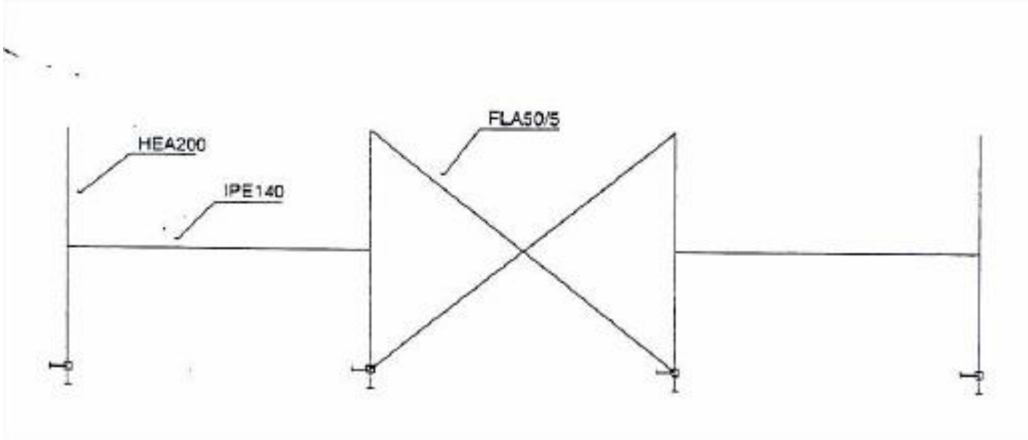
*Check structure data 

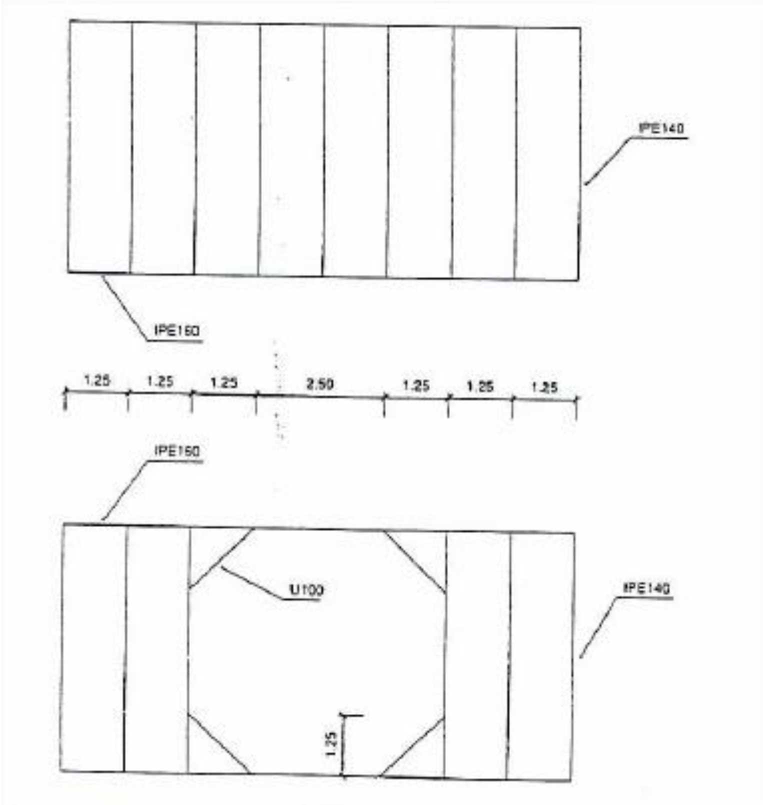
*Connect members/nodes  (Attention: connect the entire structure!)

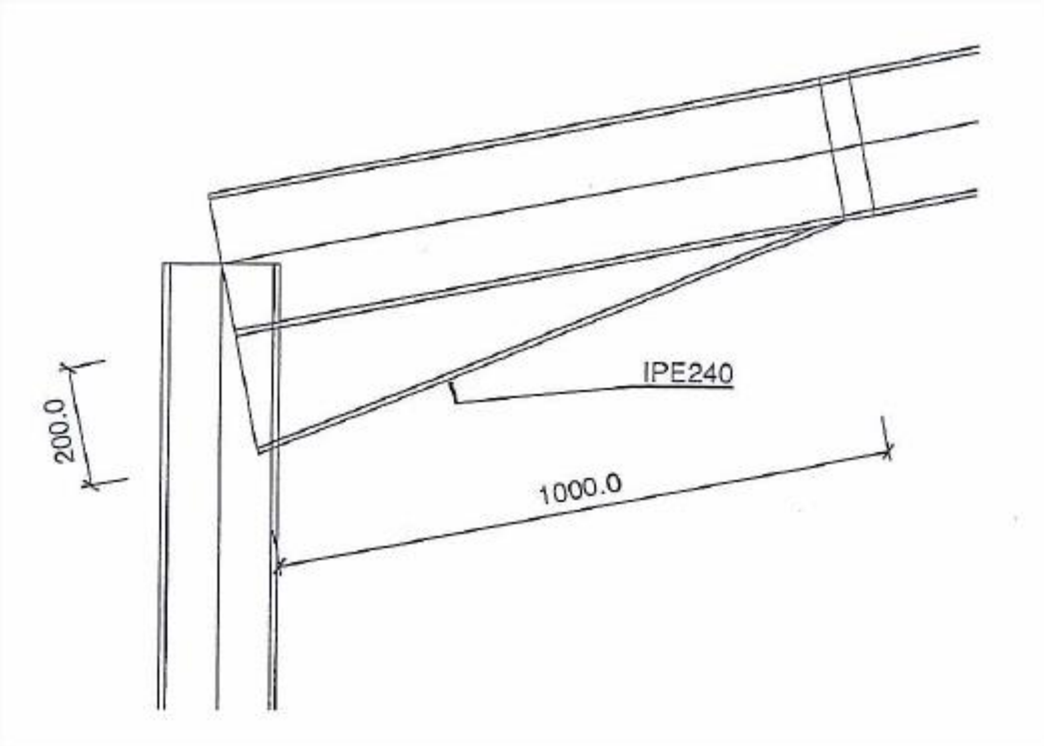
Extra example: 3D Hall

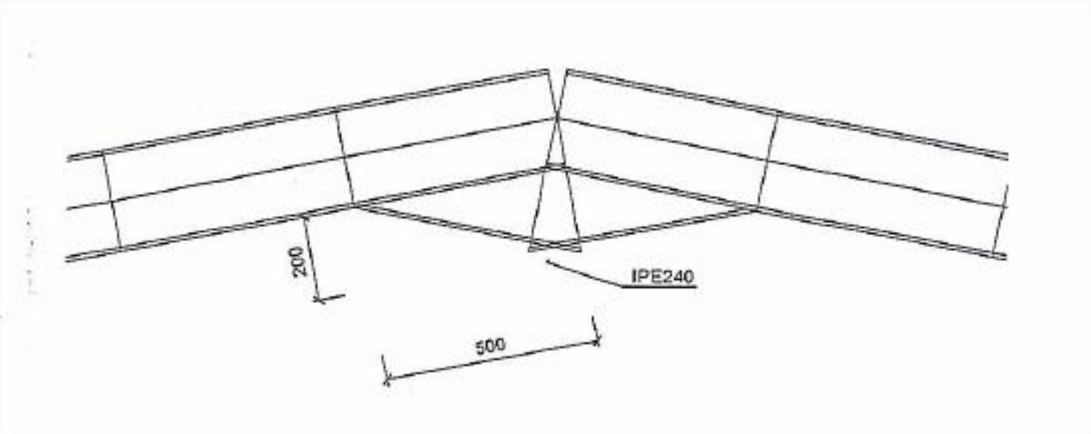


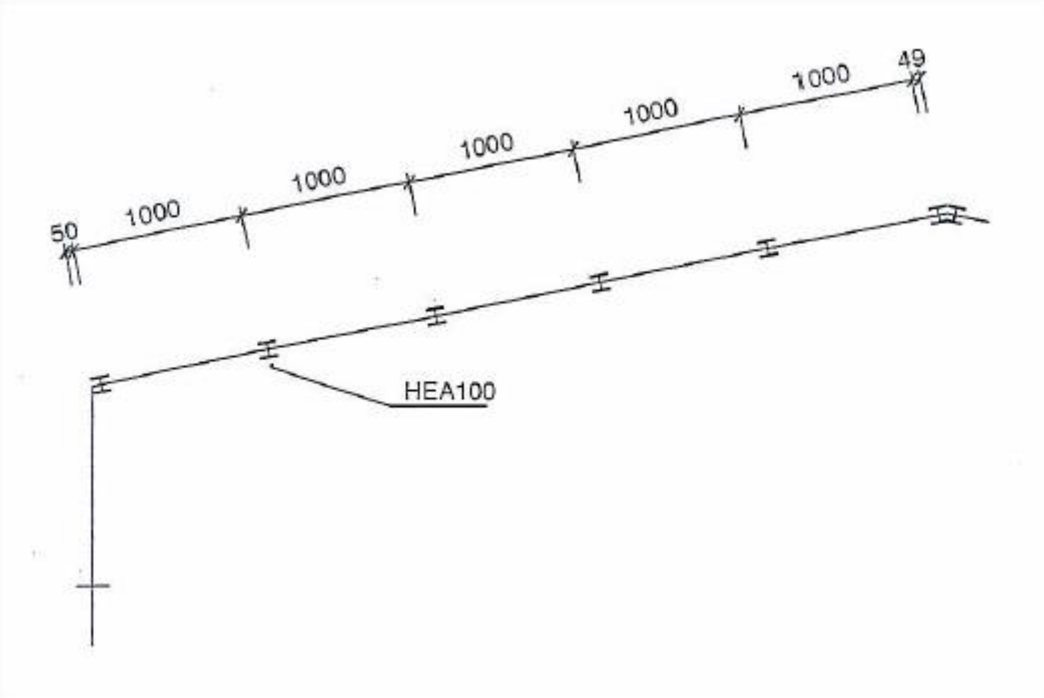

IPE240
HEA200
IPE160


FLA50/5
HEA200
IPE140


IPE140
IPE160
1.25
1.25
1.25
2.50
1.25
1.25
1.25
IPE160
U100
IPE140
1.25


IPE240
200.0
1000.0

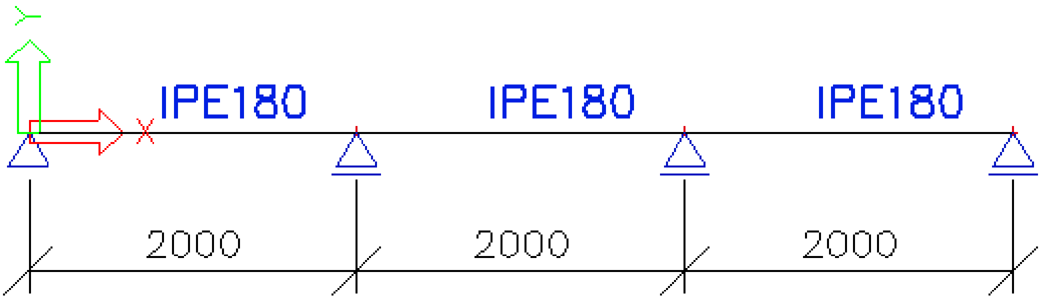

200
IPE240
500


50
1000
1000
1000
1000
1000
49
HEA100

Part 2 – Loads, Load combinations, Calculation and Results

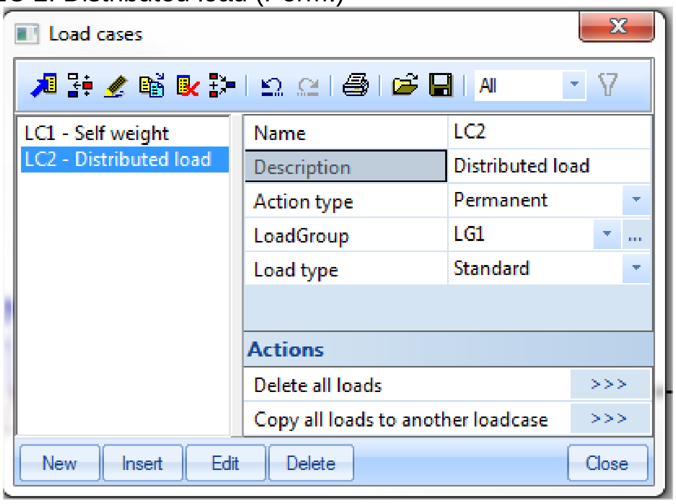
Example 7: Beam with 3 spans

1_Input of geometry



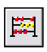
2_Loads

*Definition of load cases
Main menu > Load cases, Combinations > Load cases
LC 1: Self weight
LC 2: Distributed load (Perm.)



*Input of loads
Main menu > Loads
LC 1: Self weight > Calculated by SCIA Engineer
LC 2: Distributed load (Perm.) > Line force on beam 10 kN/m

3_Calculation

Main menu > Calculation, mesh > Calculation  or Hidden calculation , see also ‘Project’ toolbar
Difference: When performing a Hidden calculation the windows with the status of the calculation are suppressed, as a result of which the calculation cannot be interrupted prematurely.

4_Results

After calculation: Main menu > Results

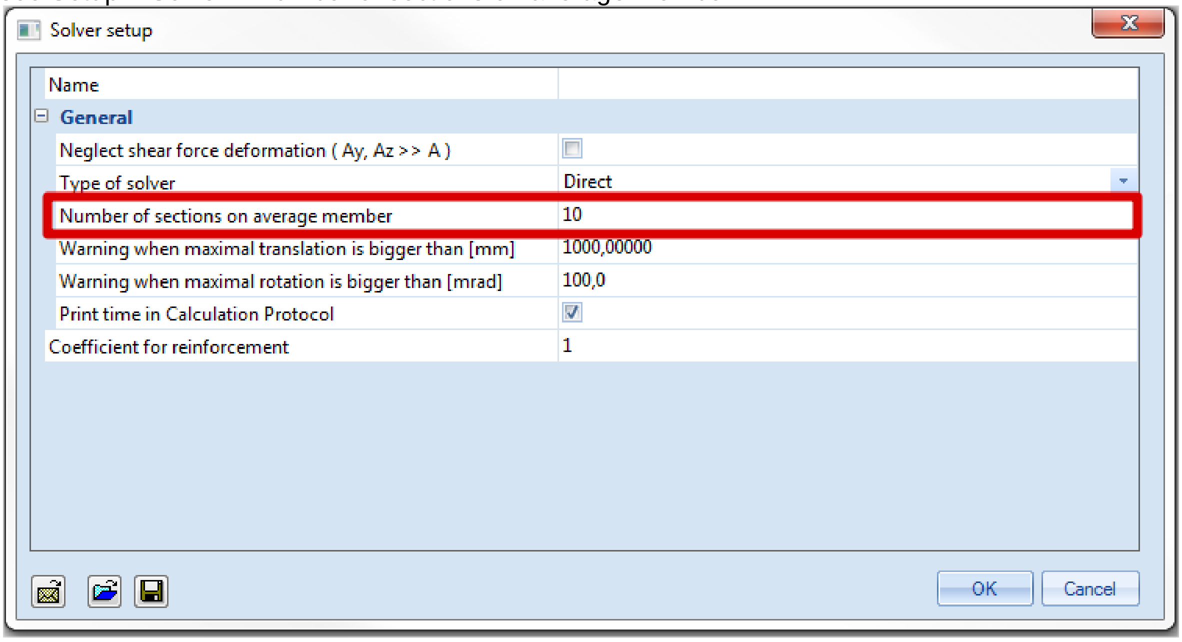
*Graphical display of results
Results > Supports > Reactions
Results > Beams > Internal forces on beam
Results > Beams > Deformations on beam

Specify the desired result in the Properties menu
-Selection: All > result on all of the members; Current > result on the selected members
-Extreme: Place(s) where the result values are displayed numerically
-Drawing setup: click on ... > Change the display of the diagrams, display the units, …
After any modification, choose for Actions > Refresh

To change unities and number of decimals: go to Setup > Units, or 'Project' toolbar

*Numerical display of results
At the bottom of the Properties menu: Actions > Preview

The exact values are calculated in (by default) 10 sections per beam,
see Setup > Solver > Number of sections on average member

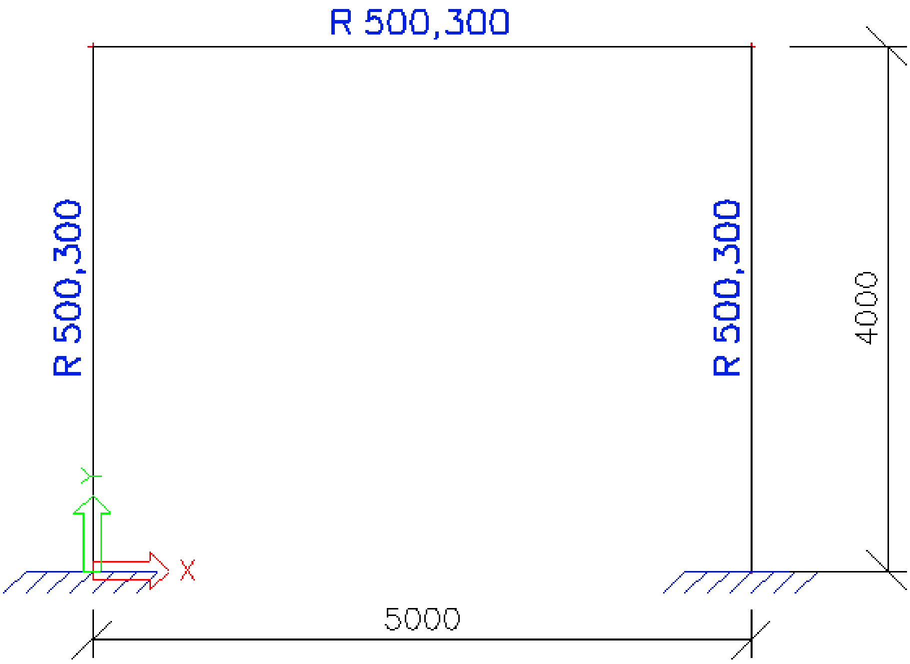


*Result on a specific location
Structure > Model data > Section on beam; afterwards it is necessary to calculate again

*Extra information
-Main > Results > Bill of material
Ask for the Mass and Surface of a specific Cross-section or Material type
-Main > Results > Calculation protocol
Consult the Data of calculation, and Sum of loads and reactions

Example 8: Concrete frame

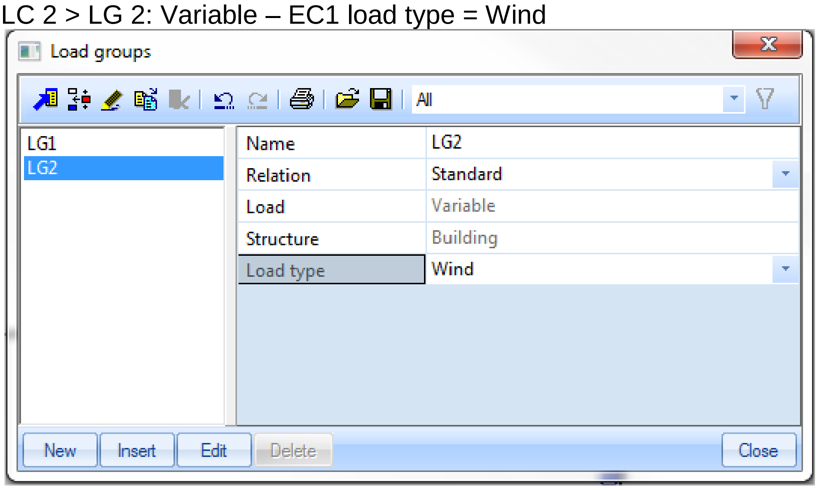
1_Input of geometry



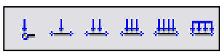
2_Loads

*Load cases
Main menu > Load cases, Combinations > Load cases
LC 1: Self weight
LC 2: Wind in direction X (Var.) > Line force on beam 5 kN/m

*Load groups
Main menu > Load cases, Combinations > Load groups
LC 1 > LG 1: Permanent
LC 2 > LG 2: Variable – EC1 load type = Wind

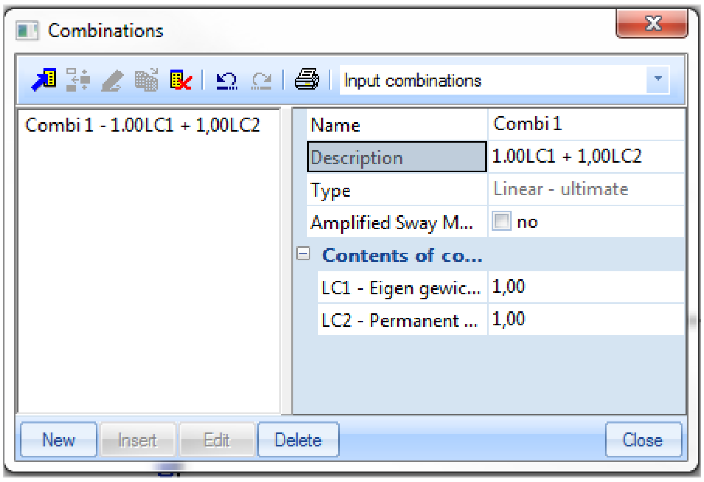


*Input of loads
Main menu > Load

Fast input of loads via Command line toolbar ; modification of properties via Properties menu

*Load combinations

Main menu > Load cases, Combinations > Combinations
Linear combination: 1,00.LC 1 + 1,00.LC 2



*Graphical display of results

-Loads, via Command line toolbar  and 

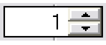
-Values of loads, via Command line toolbar  > Loads/Masses > Labels of loads

3_Results

*Ask for Results
Main menu > Results

Fast displaying of results via Command line toolbar  ; automatic adaptation in the Properties menu

*Scaling of Results

Via 'Tools' toolbar  and 

4_Table results

Activate the table results functionality via view > toolbars > table results

*Ask for Results
Main menu > Results > Beams > Internal forces on beam

Next click on the green refresh button in order to load the results into the table.

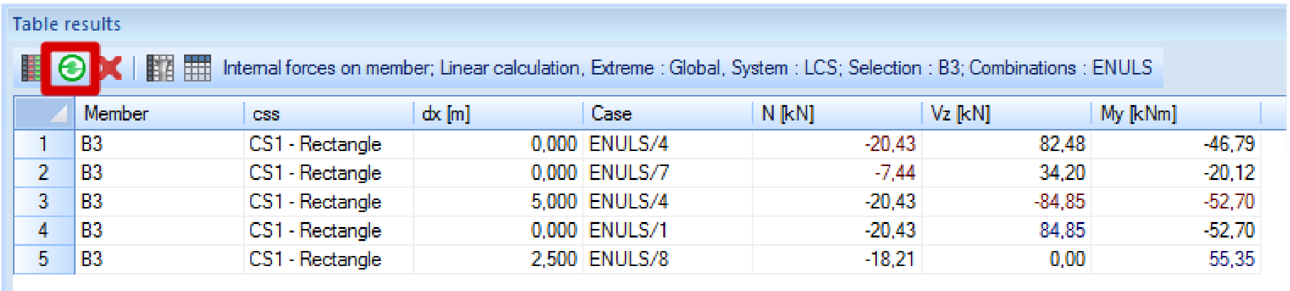


Table results

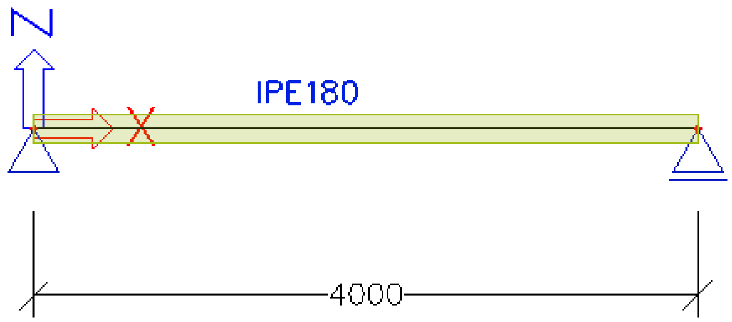
Internal forces on member; Linear calculation, Extreme : Global, System : LCS; Selection : B3; Combinations : ENULS

	Member	css	dx [m]	Case	N [kN]	Vz [kN]	My [kNm]
1	B3	CS1 - Rectangle	0,000	ENULS/4	-20,43	82,48	-46,79
2	B3	CS1 - Rectangle	0,000	ENULS/7	-7,44	34,20	-20,12
3	B3	CS1 - Rectangle	5,000	ENULS/4	-20,43	-84,85	-52,70
4	B3	CS1 - Rectangle	0,000	ENULS/1	-20,43	84,85	-52,70
5	B3	CS1 - Rectangle	2,500	ENULS/8	-18,21	0,00	55,35

These table results can then be copied to a spreadsheet application such as MS Excel.

Example 9a: Beam on 2 supports

1_Input of geometry



*Configuration of the example
Suppose this is the section of a narrow roadway, with a footpath and a traffic lane over which only one car at a time can drive.

2_Loads

*Load cases & Load groups

Load Case	Type	Load Group	Type
Self Weight	P	LG 1	
LC 1: Permanent	P	LG 1	
LC 2: Pedestrian	V	LG 2	Standard
LC 3: Car Left	V	LG 3	Exclusive
LC 4: Car Right	V	LG 3	Exclusive

Relationships Variable LG	
1. Standard	A AND/OR B
2. Exclusive	A OR B
3. Together	A AND B
4. Master/Slave	A; A AND B
A B	

By placing *Car Left* and *Car Right* in the same load group with type 'exclusive', we define that both load cases can never appear together in a load combination.

*Input of loads
Input all loads as point loads of 1kN. Only the *Self Weight* is not taken into account for this example.

3_Load combinations

Suppose a combination with content & coefficients as follows:

LC 1	1,35
LC 2	1,20
LC 3	0,50
LC 4	1,50

*Type = Linear combination:
Only one combination is generated
Relationships of the load groups are NOT taken into account + Coefficients as inputted by the user

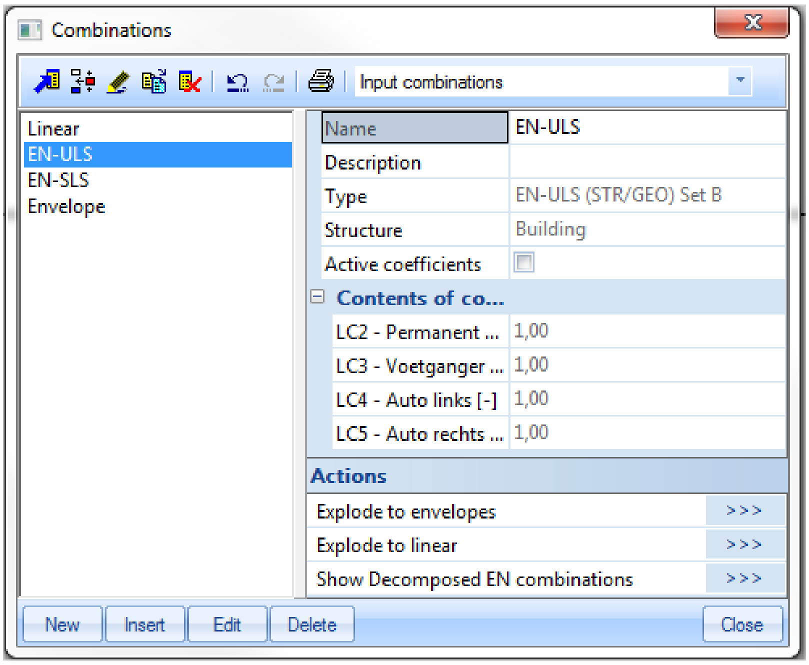
1,35.LC 1 + 1,20.LC 2 + 0,50.LC 3 + 1,50.LC 4

*Type = Eurocode combination at ULS or SLS:
All possible *linear* combinations according to the relationships of the load groups are generated
Safety factors according to the Eurocode + Psi-factors according to the Eurocode (see content of the load groups) + ~~Coefficients as inputted by the user~~

1,35.~~1,35~~.LC 1
1,35.~~1,35~~.LC 1 + 1,50.~~1,20~~.LC 2
1,35.~~1,35~~.LC 1 + 1,50.~~1,20~~.LC 2 + 1,05.~~0,50~~.LC 3
1,35.~~1,35~~.LC 1 + 1,50.~~1,20~~.LC 2 + 1,05.~~1,50~~.LC 4
…

*Type = Envelope combination:
All possible *linear* combinations according to the relationships of the load groups are generated
Coefficients as inputted by the user

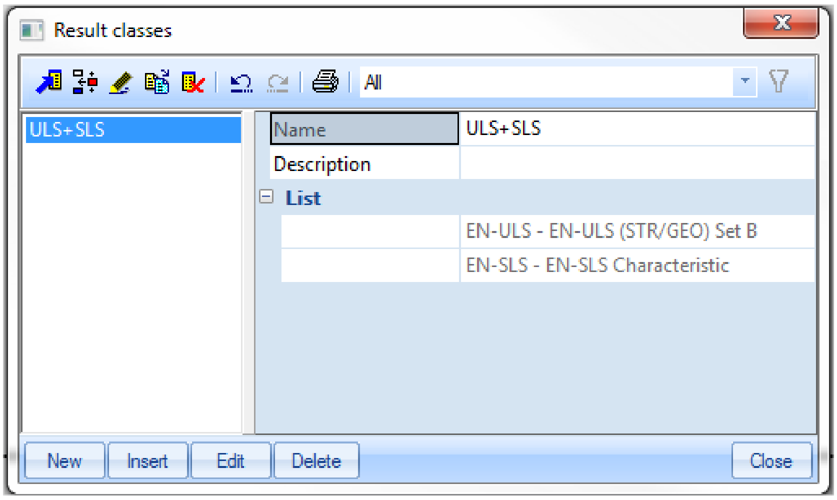
1,35.LC 1
1,35.LC 1 + 1,20.LC 2
1,35.LC 1 + 1,20.LC 2 + 0,50.LC3
1,35.LC 1 + 1,20.LC 2 + 1,50.LC4
…



'Black Box': As well for combinations according to the Eurocode as for Envelope combinations, the generated *linear* combinations are not shown.
If the user wants to know the content of such combinations, the Action 'Explode to linear' has to be executed.

4_Result classes

Main menu > Load cases, Combinations > Result classes
A Result class makes it possible to make an Envelope combination of an arbitrary amount of Combinations and/or Load cases.
RC 1: ULS + SLS



5_Results

*Results of EN-ULS / EN-SLS / Envelope combination

Only the envelope of the results is shown → On every section of the structure you will find the most positive & most negative result.
It is only possible to ask the results of the (in the background generated) linear combinations separately, if the Action ‘Explode to linear’ has been executed.

*Governing linear combinations
See Actions > Print preview: ULS/1, ULS/2, et cetera
The numbers after the combination name refer to the Combination key, where the governing linear combinations are fully displayed. This Combination key can only be asked for in the Document.

Part 3 – Engineering report and Images

Example 9b: Beam on 2 supports

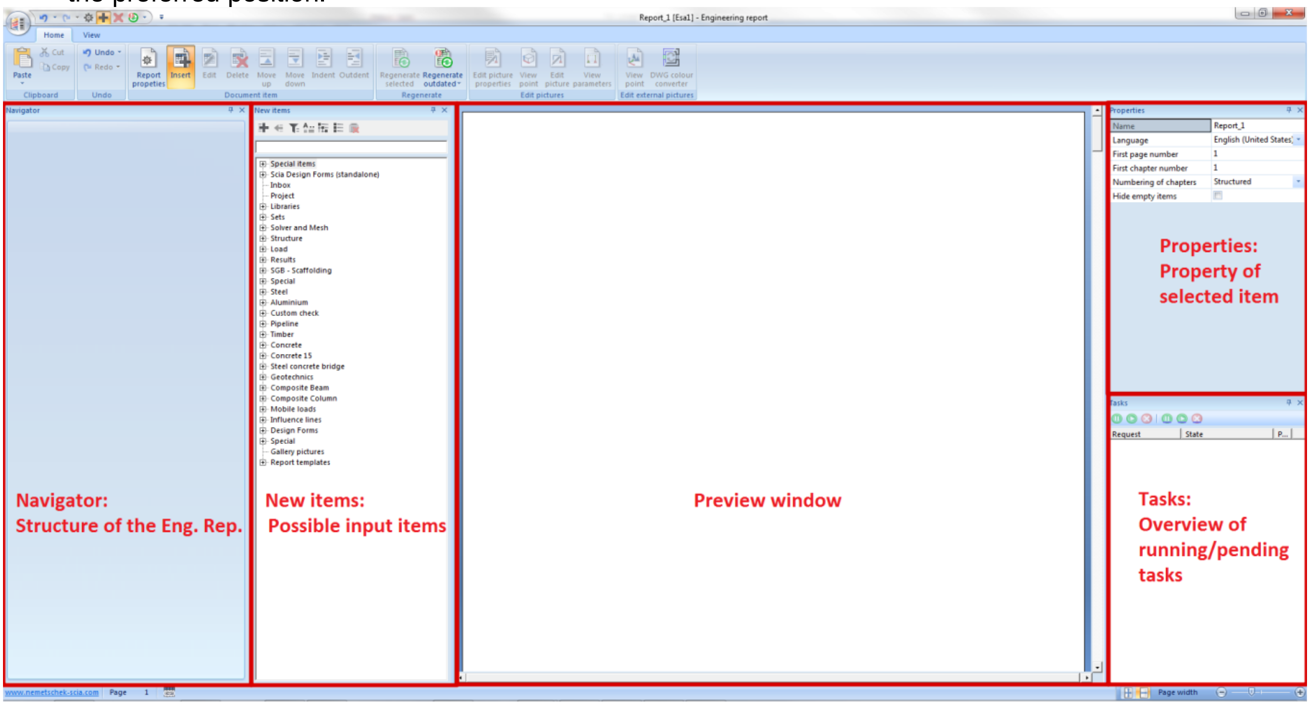
1_Input of geometry

See Example 9a

2_Engineering report

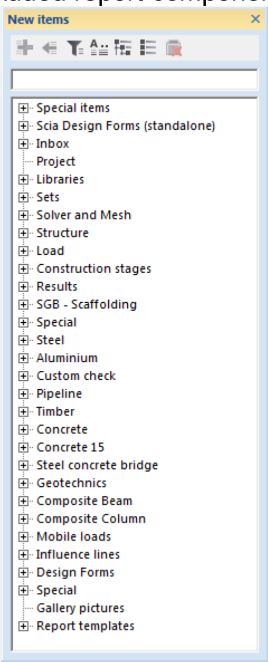
Main menu > Engineering report, or 'Project' toolbar 

In the Engineering report you can find 5 windows which are described in the following picture. Besides the preview window the user can choose to move the other 4 windows by dragging these to the preferred position.



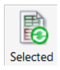
*Content of engineering report
Via button Insert at the top of the Start menu > Show/hide to be added report components

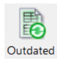
Added report components > Navigator



The above tables can be displaced on the screen.
The components to be inserted can be filtered and it is possible to hide or lock the added parts.

*Refresh of engineering report
After adjustments of data in the project > some components of the engineering report must be regenerated.

-Refresh of selection, see Engineering Report Toolbar 

-Refresh of entire Engineering Report, see Engineering Report Toolbar 

With this option, it is possible to not regenerate some components by hiding or locking them.

*Properties of the different components
After selecting an item in the Navigator, some of its properties can be accessed and modified in the Property menu.

The advanced properties can be customized > see Engineering Report Toolbar 

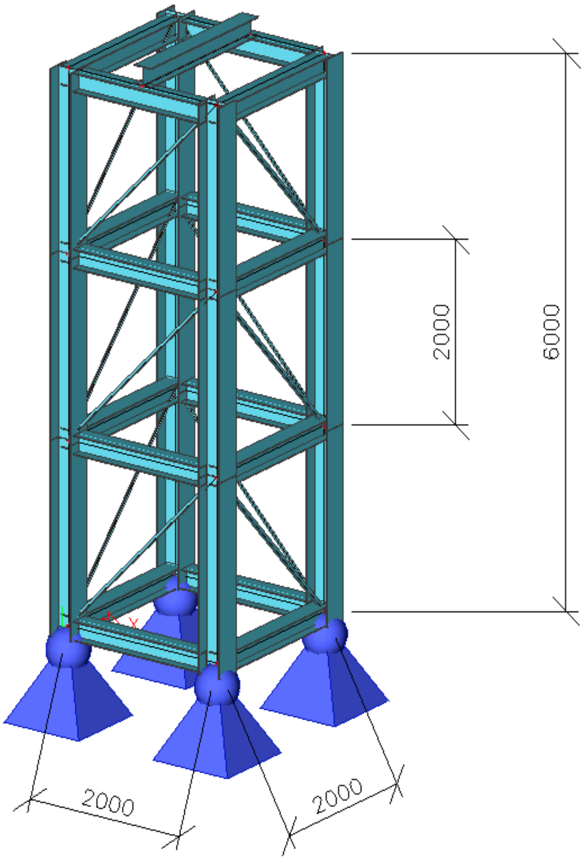
*Combination key: display of governing linear combinations
New Engineering Report item > Sets > Combination key
Example: Take a look at the Internal forces on beam, according to Combinations = ULS; Deformations on beam, according to Combinations = SLS. In these tables with results is referred to ULS/1 etc., and SLS/2 etc. The numbers after the combination names refer to the Combination key, where the governing linear combinations are written out.

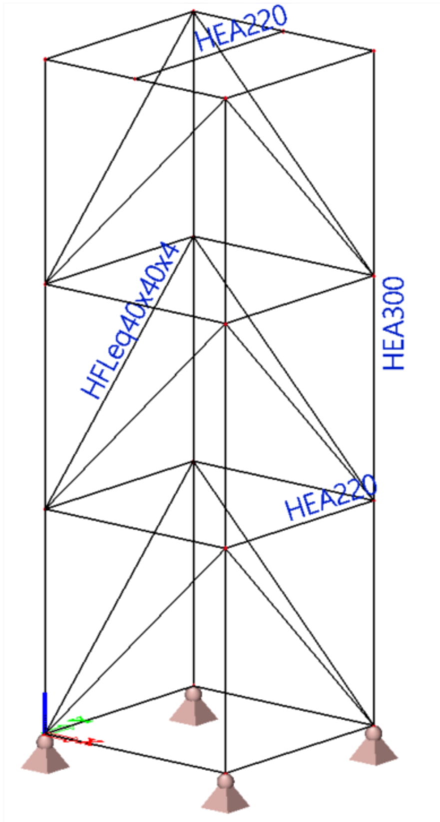
*Language Engineering report
The language of both in-and output of the Engineering report can be changed using the Properties menu.

Example 10: Bearing frame

1_Input of geometry

*Project data: General XYZ – Steel S235





*Building up a Line grid, see 'Tools' toolbar 
-As a help to input the structure
-Necessary to generate Overview drawings

*Actions after input!

2_Loads

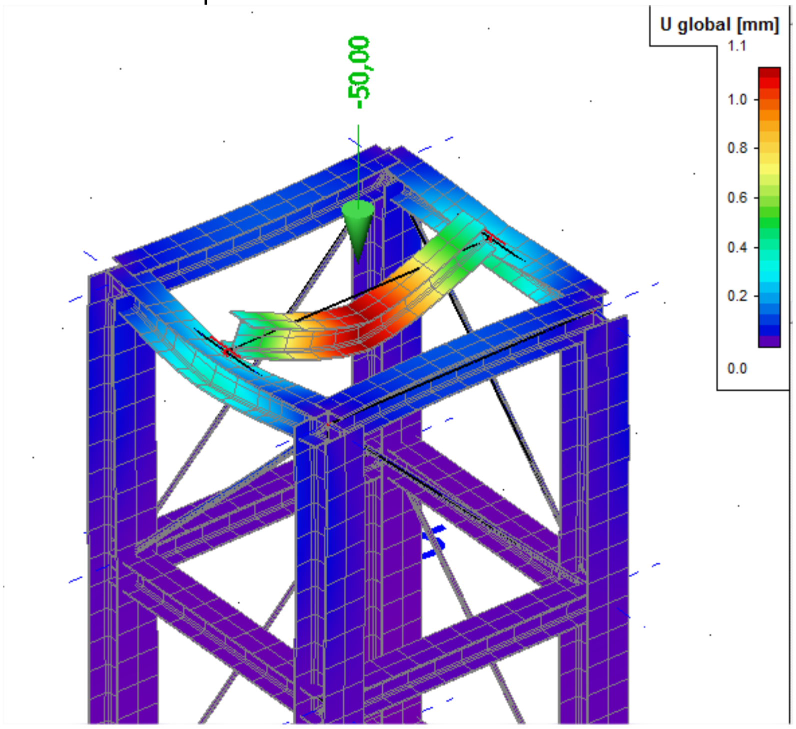
LC 1: Self weight
LC 2: Vertical load (Var.) > Point force 50 kN
LC 3: Horizontal load (Var.) > Point force 20 kN

3_3D displacement & 3D stress

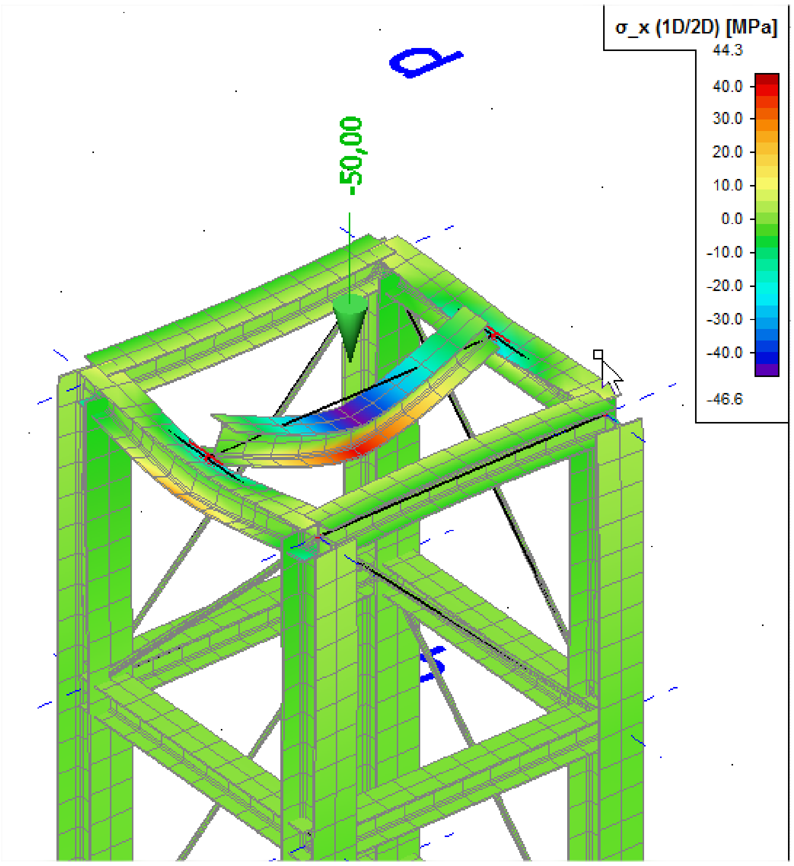
It is possible to view the displacement & stresses on surfaces of 1D members.

-Run the calculation

-Results > 3D displacements: use loadcase LC2 and tick on ‘deformed structure’; click on refresh.



-Results > 3D stress



4_Pictures

Following actions are accessible via Main menu > Drawing tools, or ‘Project’ toolbar, or right mouse click in screen

-Print data

Claim the Preview of a certain table, or send Table to document

-Print picture
Print the picture on the display, after choice of printer, choice of template file and possible editing

-Picture to document
Send the picture on the display directly to the document

-Picture to gallery
Send the picture on the display to the Picture gallery, where it can be edited before saving it or adding it to the document

-Picture gallery
Edit pictures by means of the Gallery editor; e.g. text and dimension lines can be added

-Paperspace gallery
Choose/make a template file for printing + input and arrange the picture(s) to be printed

5_Overview drawings

Main menu > Project > Functionality: Overview drawings

Picture wizard, via Picture gallery > New by wizard , or right mouse click in screen ; choose Sections by planes of line grid

6_Engineering report

Main menu > Engineering report, or 'Project' toolbar

*Content of engineering report
Via button Insert at the top of the Home menu > Show/hide to be added report components
Added report components > Navigator

*Add picture to Engineering report

-Add picture directly into Engineering report
-Add picture into the Inbox in the Item menu

-Picture to report as a printscreen via Screenshot into Engineering report
-Picture to report as a dynamic image via Live picture into Engineering report

Images can first be edited using Picture to gallery

*Add text to Engineering report
Special items > Formatted text

Also special symbols can be entered, example 'σ' = 'σ'

It is possible to insert the contents of a table of for instance Excel via copy-paste.

*Chapter Maker
Indented tables: Each 2 tables which have a logical relationship, can be linked to each other, e.g. the tables Nodes and Displacement of nodes. Select the item Displacement of nodes > choose the option Indent

Indented pictures: Also a picture can be linked to a table, e.g. picture of the structure with a particular load displayed, and table Load cases. Select the picture > choose the option Indent

*Add header and footer
Special items > Header / Footer

Adapt the header and footer properties > see Engineering Report Toolbar Edit

Both text and images can be inserted.
The header and footer can be saved as a template, a .HFX file.

*Adapt layout of Engineering report

-Change page orientation > Special items > Page format

-Adapt the general layout of the report > Special items > Style
Note: The SCIA logo can be removed in here.

-Adapt tables:
Empty cells may be hidden.
Multiple properties displayed in one row > Template Header

Select the table > Edit: adjustments can be saved and copied to the other tables in the engineering report. These templates are saved as .TLX files.

*Export and print of Engineering report

-Print: Engineering report setup  > Print

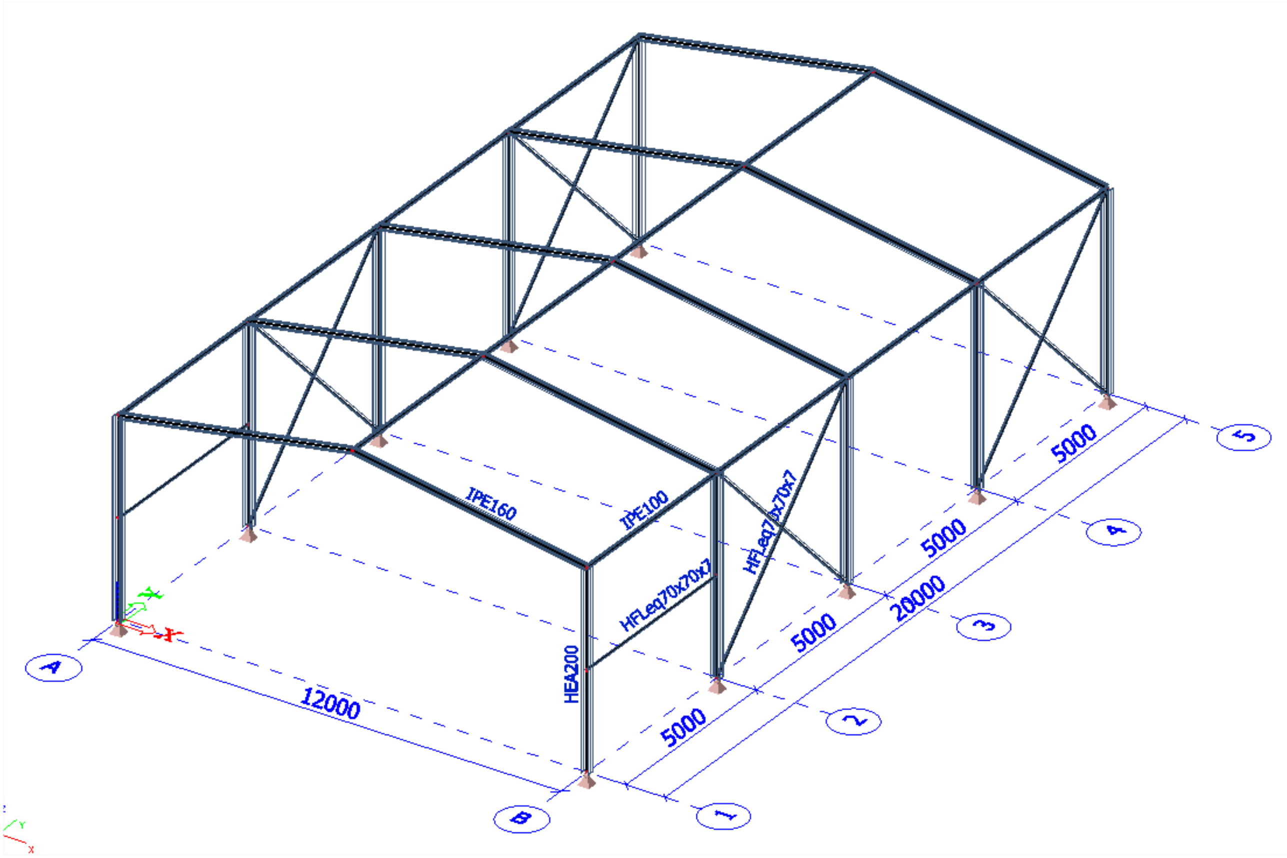
-Export: Engineering report setup  > Export

Part 4 – Introduction to Steel and Concrete code checks

Example 11: Steel hall

1_Input of geometry

*H1 = 5m
*H2 = 1m



*Modification of the geometry: see picture above

*Actions after input! These are necessary to connect the newly added beams.

2_Loads

LC 1: Self weight
LC 2: Roof load (Perm.) > Line force 5 kN/m
LC 3: Wind load X direction (Var., Exclusive)> Line force 2 kN/m
LC 4: Wind load –X direction (Var., Exclusive) > Line force 2 kN/m

3_Load Combinations

CO1: EN-ULS (STR/GEO) Set B
CO2: EN-SLS Characteristic

4_Steel Setup

Main menu > Steel

a) General settings

All of the input in the section Steel > Beams > Setup is valid for the whole project.
-Setup > Member check

A steel structure is by default sway for buckling around the Y-Y axis, and non-sway for buckling around the Z-Z axis.
-Setup > Relative deformation
The user can per beam type impose a limit for permissible relative deformation.
-Setup > Buckling defaults
The ky and kz factors are by default calculated by SCIA Engineer. Attention, this is only valid for simple structures! In other cases: input buckling factor or buckling length yourself.

b) Specific settings

-It is possible to overwrite a number of general settings per member, by means of the option Steel > Beams > Steel member data.

-To overwrite buckling data: Select a beam, and click on ... behind Buckling and relative lengths in the Properties menu. Buckling data BC1 are created and can be edited.

5_Steel Checks

a) ULS check

Steel > Beams > Check
Combinations = ULS; Values = Section check, Stability check, Unity check (=maximum of both previous checks); Extreme = Member

Actions > Preview: Summarizing overview
Selection = Current: select 1 column; Output = Detailed

Actions > Single Check: Detailed information per member

Actions > Autodesign: Optimize one cross-section group at a time, to obtain "1" as maximal value for the unity check
Attention: After optimization the structure has to be recalculated!

b) SLS check

Steel > Relative deformation
Combinations = SLS; Values = Check uz (= unity check with regard to the inputted values in Steel > Beams > Setup > Relative deformation)

6_Steel Connections

*Input of steel connection

➔ This option is not included in the Concept Edition. You need the module esa.18 that is also available with Professional or Expert Editions

Main menu > Project > Functionality: Steel – Frame rigid connections
The functionality Structural model is automatically activated.

-Generate structural model, see 'View' toolbar
-Main menu > Steel > Connections > Frame bolted/welded – strong axis; select connecting node and beams
-Input properties of the connection in the Properties menu
-Display label of steel connection, via Set view parameters for all > Connections > Steel connections label > Display label + Name

*Check of steel connection

➔ This option is not included in the Concept Edition. You need the module esasd.02 that is also available with Professional or Expert Editions

Actions > Results; verify if the unity checks satisfy

*Transfer stiffness of connection to analysis model
-In the Properties menu of the steel connection, select the option Update stiffness

-Recalculate the structure
-Display analysis model, via Set view parameters for all > Structure > Model type; Show model data, via Command line toolbar : Hinge with adapted stiffness has been added to the connecting node

Since connections and hinges are additional data, it is possible to copy these, via 'Geometry manipulations' toolbar or via right mouse click in screen.

7_Steel connection monodrawings

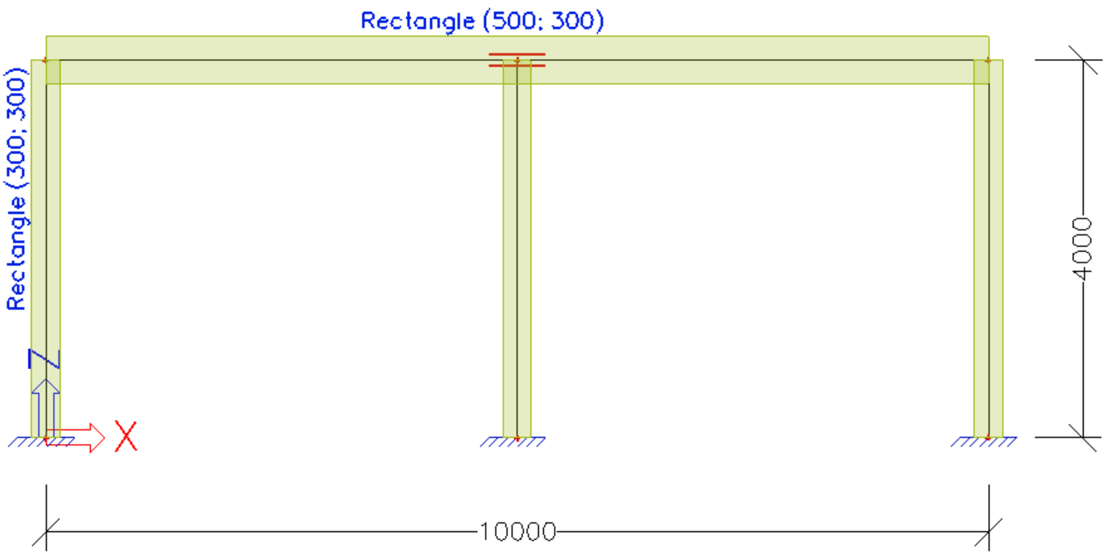
➔ **This option is not included in the Concept Edition. You need the module esadt.02 that is also available with Professional or Expert Editions**

Main menu > Project > Functionality: Steel – Connection monodrawings
Picture wizard, via Picture gallery > New by wizard , or right mouse click in screen

Example 12: Concrete frame

1_Input of geometry

*Project data: Frame XZ – Concrete C30/37 – Reinforcement steel B500A



2_Actions after input

*Check structure data

*Connect members/nodes (Attention: connect the entire structure!)

3_Loads

*Load cases
LC 1: Self weight
LC 2: Roof load (Perm.) > Line force 33 kN/m

*Load combinations
CO 1: EN-ULS (STR/GEO) Set B
CO 2: EN-SLS Quasi-Permanent

4_Concrete Settings new Concrete 15

Main menu > Concrete 15

Concrete settings
All of the input in the section Concrete settings is valid for the whole project.
-Concrete settings > Design defaults > Default sway type
Concrete beams and columns are by default sway for buckling around both the Y-Y and Z-Z axis.
-Concrete settings > Design defaults > Beams
Choose for upper and lower reinforcement: diameter 16mm

Specific settings
It is possible to overwrite a number of general settings per member, by means of the option Concrete 15 > Setting per member > 1D member data.
A label is displayed on each member with 1D member data, e.g. CMD1. This label can be selected to view or edit the settings in the Properties menu. Since 1D member data are attributes, it is possible to copy these to other beams, via 'Geometry manipulations' toolbar or via right mouse click in screen > copy attributes.

5_Reinforcement design of beam new Concrete 15

Theoretical reinforcement

Internal forces

Concrete 15 > Reinforcement design – 1D members > Internal forces; view for Class = All ULS (created by SCIA Engineer) the Values = M_y and M_Edy (you might have to increase the number of sections to be able to see the capping of the momentline; e.g. 20 via setup > solver > number of sections on average member)

*The user can set the capping of the momentline on/off in the Concrete settings > Solver setting > Internal forces ULS > Take into account additional tensile force caused by shear force (shift rules).

Slenderness

Concrete 15 > Reinforcement design – 1D members > Slenderness; Using of second order effect in calculation depends on the check of slenderness, because if the check is slenderness is greater than limit slenderness, the second order effect has to be taken into account for column calculation.

Conditions	Calculation of second order effect
$\lambda_y > \lambda_{imy}$ or $\lambda_z > \lambda_{imz}$	YES
$\lambda_y \leq \lambda_{imy}$ and $\lambda_z \leq \lambda_{imz}$	NO

*By default the above check is executed automatically. This can be modified in Concrete settings > Solver settings > Internal forces > Internal forces ULS > Use second order effect

Theoretically required reinforcement

Concrete 15 > Reinforcement design - 1D member > Reinforcement design ; select the beam and view for Class = All ULS the Value = A_sz_req+ & A_sz_req-

Actions > Preview: Summarizing overview (output = brief), Normal output (output = standard), Detailed output (output = detailed).

For rectangular sections the following definitions are applicable:

-A_sz_req+ = theoretically needed reinforcement placed on the edges at the positive direction of the z-axis (LCS)

-A_sz_req- = theoretically needed reinforcement placed on the edges at the negative direction of the z-axis (LCS)

-A_sy_req+ = theoretically needed reinforcement placed on the edges at the positive direction of the y-axis (LCS)

-A_sy_req- = theoretically needed reinforcement placed on the edges at the negative direction of the y-axis (LCS)

- A_swm_req = theoretically needed shear reinforcement.

- A_sz_prov+ = is the provided reinforcement placed on the edges at the positive direction of the z-axis (LCS) in order to satisfy A_sz_req+.

The used diameters are defined in the concrete settings (Concrete settings > Design defaults > Beam) or can be set per member via 1D member data (Setting per member > 1D member data)

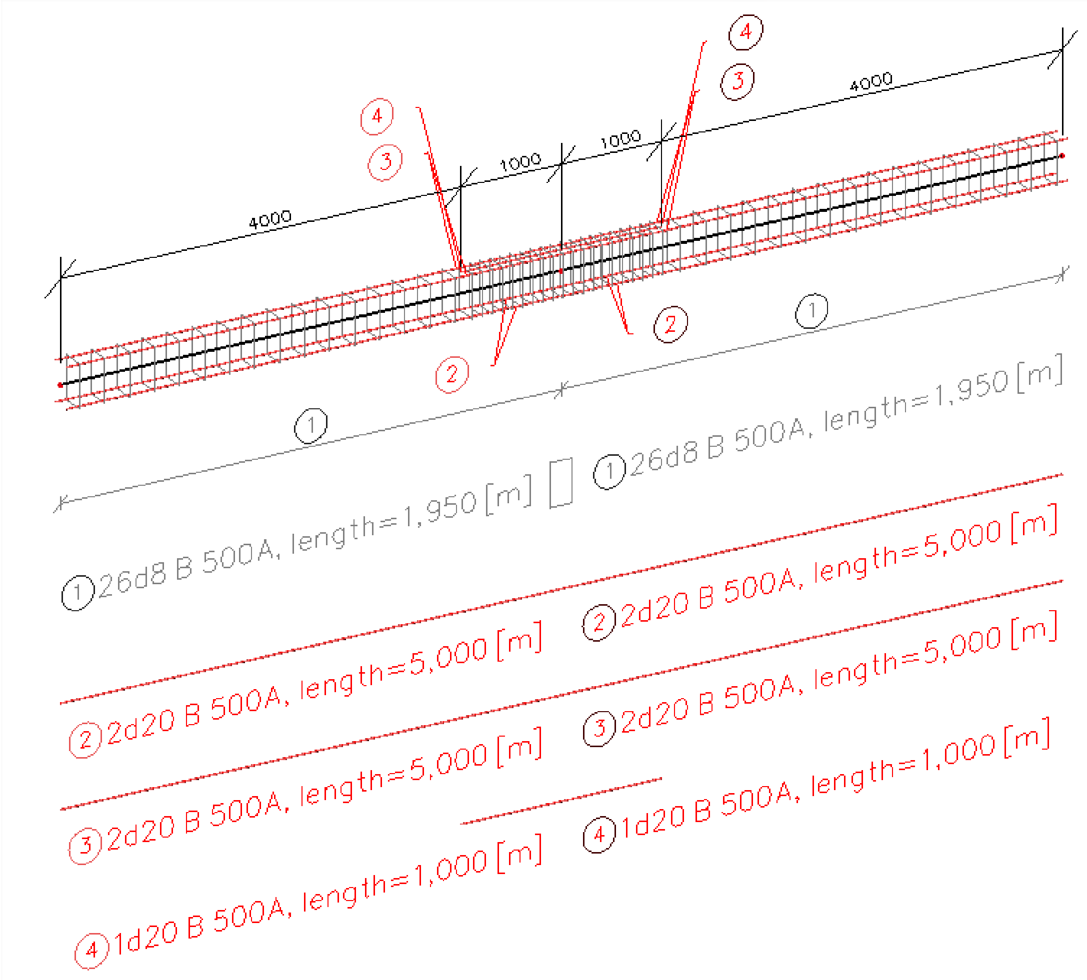
The same methodology applies for the other remaining results (A_sz_prov-, A_sy_prov+, A_sy_prov-, A_swm_prov)

When you choose for a detailed output (output > detailed) and then generate the preview you will get an explanation of errors/warnings and notes at the bottom page.

Practical reinforcement

Adding visible reinforcement into the model.

* Concrete 15 > Reinforcement input + edit > 1D members > New reinforcement: select the beam in which you want to add reinforcement. After that you'll need to define the start- and endpoint of the reinforcement. Try inserting the following practical reinforcement for the beam.



*Longitudinal reinforcement window:
- Click on edit and try adding in each corner Ø20mm longitudinal reinforcement.
- In the upper right corner: you can find the reinforcement layers. New layers can be added via the button 'new layer' in the bottom left corner.

*Stirrups are split in zones (4m – 1m – 1m – 4m) in which two different stirrup distances are defined (250mm & 100mm).

*Stirrup zones can be defined by clicking on a stirrup and then click on 'Edit stirrup distances' in the bottom right corner.

Checks

Concrete 15 > Reinforcement check (ULS+SLS)

*Stiffnesses: Stiffness presentation command is used for presentation of calculated stiffness.

*Capacity - response (ULS): is based on the calculation of strain and stress in particular component (concrete fibre, reinforcement bar) and comparison with limited values with respect of EN 1992-1-1 requirements

*Capacity – diagram (ULS): Capacity - diagram services uses creation of interaction diagram which is a graph illustrating the capacity of concrete member to resist a set of combinations of axial force and bending moment.

*Shear + torsion (ULS): this check consists of three checks: shear check, torsion check, interaction of shear and torsion check.

*Stress limitation (SLS): is based on the calculation of stresses in particular component (concrete fibre, reinforcement bar) and comparison with limited values with respect of EN 1992-1-1 requirements.

*Crack width (SLS): is calculated according to clause 7.3.4 in EN 1992-1-1

*Detailing provision: applies the rules from the Eurocode for a proper design respecting safety, serviceability and durability of the structure.

REMARK: the user can let the program automatically design the needed reinforcement that satisfies the checks in the old concrete menu. This can be done via Concrete > 1D member > Automatic member reinforcement design.

6_Concrete Settings old Concrete

Main menu > Concrete

General settings
All of the input in the section Concrete > 1D member > Setup is valid for the whole project.
-Setup > Design defaults
Concrete beams and columns are by default sway for buckling around both the Y-Y and Z-Z axis.
-Setup > Design defaults > Tab Beams
Choose for upper and lower reinforcement: diameter 16mm

Specific settings
It is possible to overwrite a number of general settings per member, by means of the option Concrete > 1D member > Member data.
A label is displayed on each member with Member data, e.g. DC1. This label can be selected to view or edit the settings in the Properties menu. Since Member data are additional data, it is possible to copy these to other beams, via 'Geometry manipulations' toolbar  or via right mouse click in screen.

7_Reinforcement design of beam old Concrete

Theoretical reinforcement
Internal forces
Concrete > 1D member > Internal forces

*1D member > Setup > General > Calculation > Tab Beams; select the options Moment capping & Shear force capping at supports

*1D member > Internal forces; view for Class = All ULS (created by SCIA Engineer) the Values = My and My,recalc

Theoretically required reinforcement
Concrete > 1D member > Member design - Design; select the beam and view for Class = All ULS the Value = As,total req

Actions > Preview: Summarizing overview
-As,req = theoretically needed reinforcement
-Reinf. (no.) = suggested by SCIA Engineer as practical reinforcement, taking into account the diameter inputted in Concrete > 1D member > Setup > Design defaults > Tab Beams (upper and lower reinforcement: diameter 16mm)

Actions > Calculation info: Description of errors and warnings
When asking results for Member design – Design, the option Print explanation of errors and warnings can be selected in the Properties menu. In that case the explanation is shown when opening the Preview.

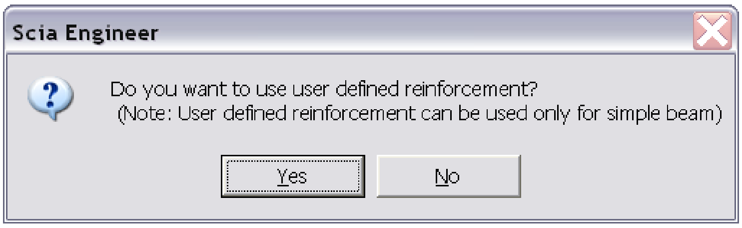
Actions > Single check: Detailed information per member; select a member and then the Single cross-section window is opened. Choose an extreme internal force and click on the Calculation button at the left.

Adding basic reinforcement = along the length of the beam
*1D member > Member data; select the beam and set Upper reinforcement to 2x diameter 14mm, Lower reinforcement to 2x diameter 12mm. SCIA Engineer is then forced to use at least this amount of reinforcement.

*1D member > Member design - Design; Class = All ULS
Actions > Preview
-As,user = specified basic reinforcement in the Member data
-As,req = As,additional req = what is needed supplementary (on top of As,user) to obtain the theoretically needed reinforcement
In this case: extra reinforcement is needed above the middle column
-Reinf. (no.) = what is specified in the Member data as basic reinforcement + what SCIA Engineer proposes as supplementary reinforcement to obtain the theoretically needed reinforcement

Practical reinforcement
Adding additional reinforcement = on specific location(s) on the beam
*1D member > Redes (without As) > New reinforcement: add supplementary reinforcement where necessary (both stirrups and longitudinal reinforcement)
In this case: select the span over the middle column, where extra reinforcement is needed.

*Adopt the user basic reinforcement: Yes > The basic (theoretical) reinforcement of 2x 14mm (Upper reinforcement) and 2x 12mm (Lower reinforcement) is now transferred to practical reinforcement.



*Stirrup shape manager: choose predefined stirrup shape

*Longitudinal reinforcement window:
-In the upper right corner: already defined layers, sc. L1 and L2. This is the transferred basic reinforcement, respectively at the top and bottom of the beam.
-Add additional reinforcement: via “New reinforcement parameters”; set Number of bars to 1, Profile to 14mm, Stirrup name to S1, Edge index to 2. After a click on [New layer], layer L3 is added.

*1D member > Member design - Design; select the beam and view for Class = All ULS the Value = As,add req

Checks
Concrete > 1D member > Member check – Check of non-prestressed concrete

*Crack control: for Class = All SLS
Possible for both theoretical and practical reinforcement, see Concrete > 1D member > Setup > General > Calculation > For stiffness, allowable stress, punching and crack-proof calculation use reinforcement

*Check response: for Class = All ULS
Only possible for practical reinforcement, because for this check the exact location and diameter of each reinforcement bar has to be known

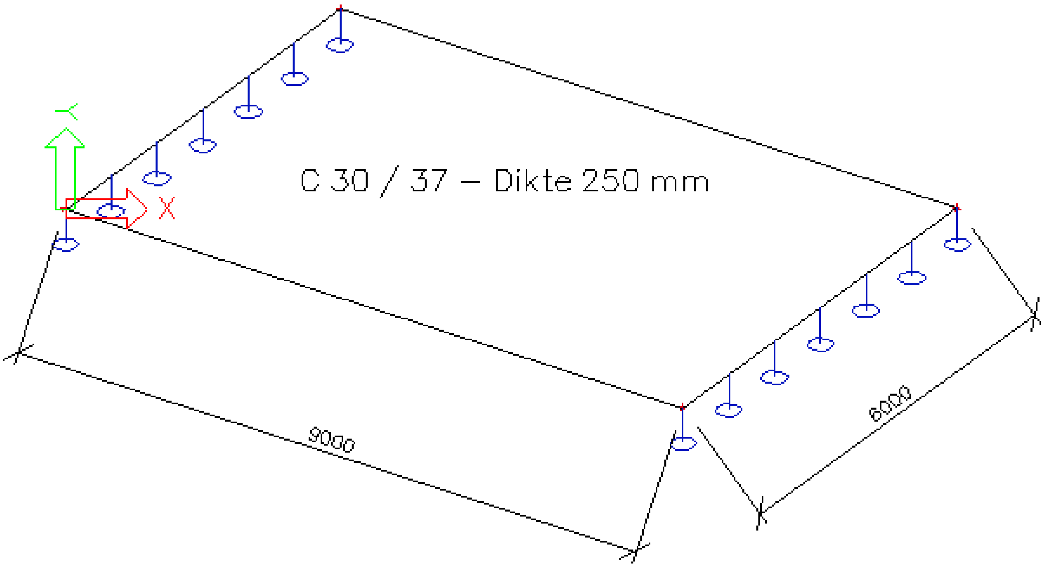
*Check capacity: for Class = All ULS
Only possible for practical reinforcement, because for this check the exact location and diameter of each reinforcement bar has to be known

Part 5 – Plates, Walls and Shells

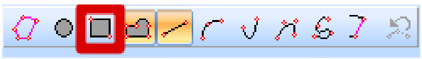
Example 13: Rectangular plate

1_Input of geometry

*Project data: Plate XY – Project level Advanced



*Input plate: Structure menu > 2D Member > Plate

New rectangle, via Command line toolbar  ; define the 2 nodes on a diagonal of the rectangle

After input, you can adapt the geometry of a selected entity via Actions > Table edit geometry & adapt the properties via Properties menu

*Input supports: Structure menu > Model data > Support > Line on 2D member edge

2_Load cases

LC 1: Self weight
LC 2: Walls on long edges (Perm.)> Line force 10 kN/m
LC 3: Service load (Var.) > Surface load 2 kN/m²

3_Finite elements mesh

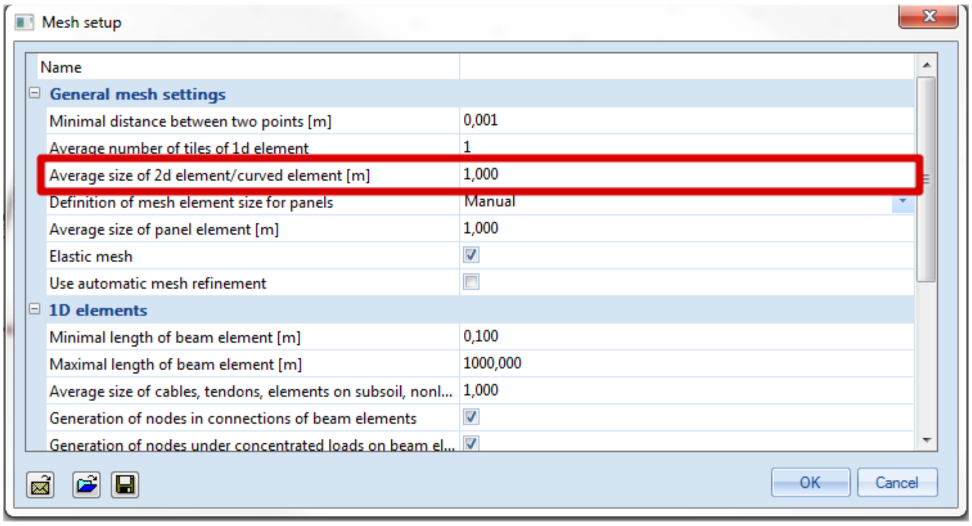
*Mesh generation

Main menu > Calculation, Mesh > Mesh generation, or 'Project' toolbar 

*Graphical display of mesh

Set view parameters for all, via right mouse click or Command line toolbar 
> Structure > Mesh > Draw mesh
> Labels > Mesh > Display label

*Mesh refinement
Main menu > Calculation, Mesh > Mesh setup, or Setup > Mesh
Average size of 2D elements, by default = 1m



4_Check of input data

*Main menu > Calculation, Mesh > Calculation; option Test of input data is sufficient. With this function, the applied loads are redistributed to the mesh elements and mesh nodes.

*Main menu > Calculation, Mesh > 2D data viewer
Surface loads: Values = qz, System = Global
LC 1 & 3: Uniform distribution over the mesh elements
LC 2: Line forces are redistributed to point forces in the mesh nodes

5_Results

*Results on the plate
Main menu > Results > 2D Members > Displacement of nodes
Main menu > Results > 2D Members > Internal forces
Main menu > Results > 2D Members > Stresses

Specify the desired results in the Properties menu
-System
Local: according to the local axes of the mesh elements
LCS – Member 2D: according to the axes of the LCS of the 2D member
Attention when using curved shell elements!
-Location: 4 ways to ask for the results, see Annex 3
-Type forces: Basic, Principal or Dimensional magnitudes, see Annex 2
-Drawing setup: Click on ... > Adapt display of 2D results, Minimum and maximum settings, …

After adaptations, always perform Actions > Refresh

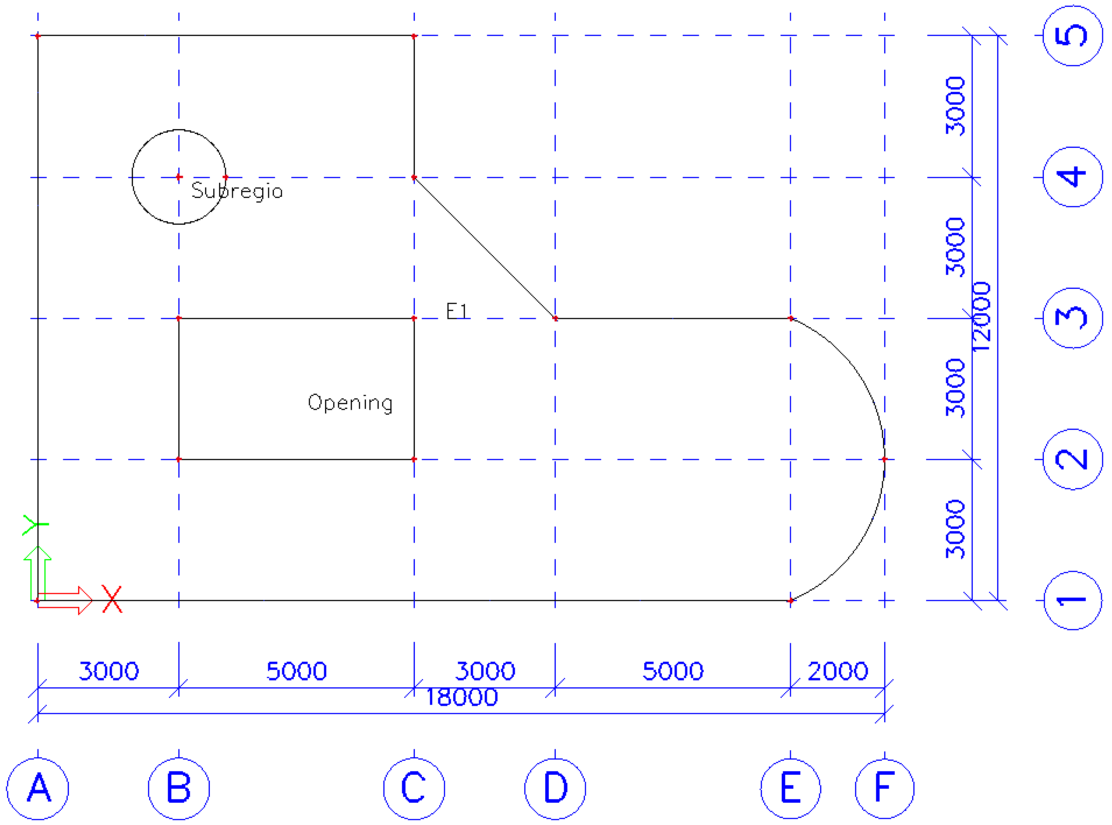
*Accuracy of the results
If the results at the 4 locations differ a lot, then the results are inaccurate and the mesh has to be refined.
Basic rule for size of mesh elements = 1 to 2 times the thickness of the plate

*Reactions in the line support
Results > Supports > Intensity

Example 14: Slab on elastic foundation (subsoil)

1_Input of geometry

*Project data: Concrete C20/25 – Plate thickness 200mm



*Input plate

Input by means of a Line grid, see ‘Tools’ toolbar 

Snap to the points of the line grid by means of the Cursor snap settings, see Command line toolbar 

Structure > 2D Member > Plate

New polygon, via Command line toolbar  ; choose options New straight line & New circular arc

*Input extra parts
Structure > 2D Member > 2D member components > Opening
New rectangle
Structure > 2D Member > 2D member components > Subregion
New circle (centre - radius) with radius = 1m; define centre point + point on circle @1;0;0

REMARK: Instructions are being shown on the Command line!

*Input supports
Main menu > Project > Functionality: Subsoil
Structure > Model data > Support > Surface (elas. foundation)

2_Loads

*Load cases
LC 1: Self weight
LC 2: Walls on the outer edges (Perm.) > Line force 10 kN/m
LC 3: Freestanding walls (Perm.) > Line force 6,5 kN/m
LC 4: Service load (Var.) > Surface load 2 kN/m²

LC 5: Service load on subregion (Var.) > Surface load 1,5 kN/m²

*Load combinations
CO 1: EN-ULS (STR/GEO) Set B
CO 2: EN-SLS Quasi-Permanent

3_Finite elements mesh

*Mesh generation

Main menu > Calculation, Mesh > Mesh generation, or 'Project' toolbar 

*Mesh refinement
Main menu > Calculation, Mesh > Mesh setup;
Average size of 2D elements = 1 to 2 times the thickness of the plate

4_Check of input data

*Main menu > Calculation, Mesh > Calculation; option Test of input data

*Main menu > Calculation, Mesh > 2D data viewer

5_Results

*Results on the plate
Results > 2D Members > Internal forces

*Result on specific place
Results > 2D Members > Section on 2D member; it is not necessary to calculate again
Attention: Properties of a section
-Draw = direction for the graphical display of the results on the section
-Direction of cut = 2nd co-ordinate of a direction vector which defines the direction of the section (1st co-ordinate is the origin)

*Elastic foundation
Results > 2D Members > Contact stresses

REMARK: Convention for soil stresses: positive value = compressive stress, negative value = tensile stress.

6_Eliminate tension in subsoil

➔ This option is not included in the Concept Edition. This is the module esas.08 that is available with Professional or Expert Editions

*Main menu > Project > Functionality: Nonlinearity + Support nonlinearity/Soil spring

*Main menu > Load cases, Combinations > Nonlinear combinations

*Main menu > Calculation, Mesh > Calculation; option Nonlinear calculation

*Take a look at the new results > Contact stresses: tension has been eliminated

7_Concrete Settings

For the reinforcement design the user should use at the moment the old concrete menu since 2D reinforcement design isn't yet supported in the new concrete menu (=Concrete 15).
Main menu > Concrete

a) General settings

All of the input in the section Concrete > 2D member > Setup is valid for the whole project.
2D member > Setup > Design defaults > 2D structures and slabs

Choose for upper and lower reinforcement: diameter 10mm

b) Specific settings

It is possible to overwrite a number of general settings per 2D member, by means of the option Concrete > 2D member > Member data.
A label is displayed on each 2D member with Member data, e.g. DSC1. This label can be selected to view or edit the settings in the Properties menu. Since Member data are additional data, it is possible to copy these to other 2D members, via 'Geometry manipulations' toolbar  or via right mouse click in screen.

8_ Reinforcement design of plate

a) Theoretical reinforcement

Internal forces
see Main menu > Results

Theoretically required reinforcement
Concrete > 2D member > Member design – Design – ULS; view for Class = All ULS the Reinforcement = Required reinforcement, with Value = As

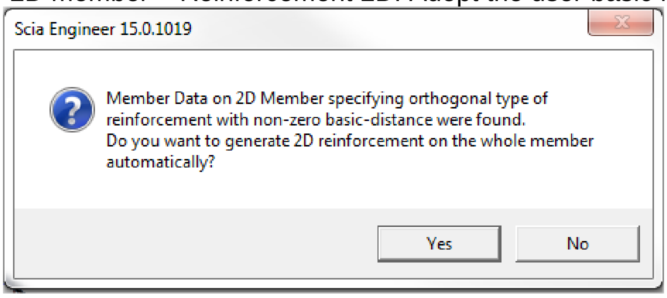
Actions > Preview: Summarizing overview
-As_up = theoretically needed upper reinforcement, As_lo = theoretically needed lower reinforcement
-direction 1 is by default = x direction of LCS of the plate, direction 2 is by default = y direction of LCS of the plate

Adding basic reinforcement = on the whole plate
*2D member > Member data; select the plate, choose under Basic data for the option User reinforcement, and fill in diameter and basic distance for directions 1 and 2

*2D member > Member design – Design – ULS; view for Class = All ULS the Reinforcement = User reinforcement/Additional reinforcement, with Value = As

b) Practical reinforcement

*2D member > Reinforcement 2D: Adopt the user basic reinforcement as practical reinforcement: Yes



Adding additional reinforcement = on specific location(s) on the plate
*2D member > Reinforcement 2D: Where necessary, add extra reinforcement layers – the layout of the geometry can be chosen by the user

*2D member > Member design – Design – ULS; view for Class = All ULS the Reinforcement = User reinforcement/Additional reinforcement, with Value = As

c) Checks

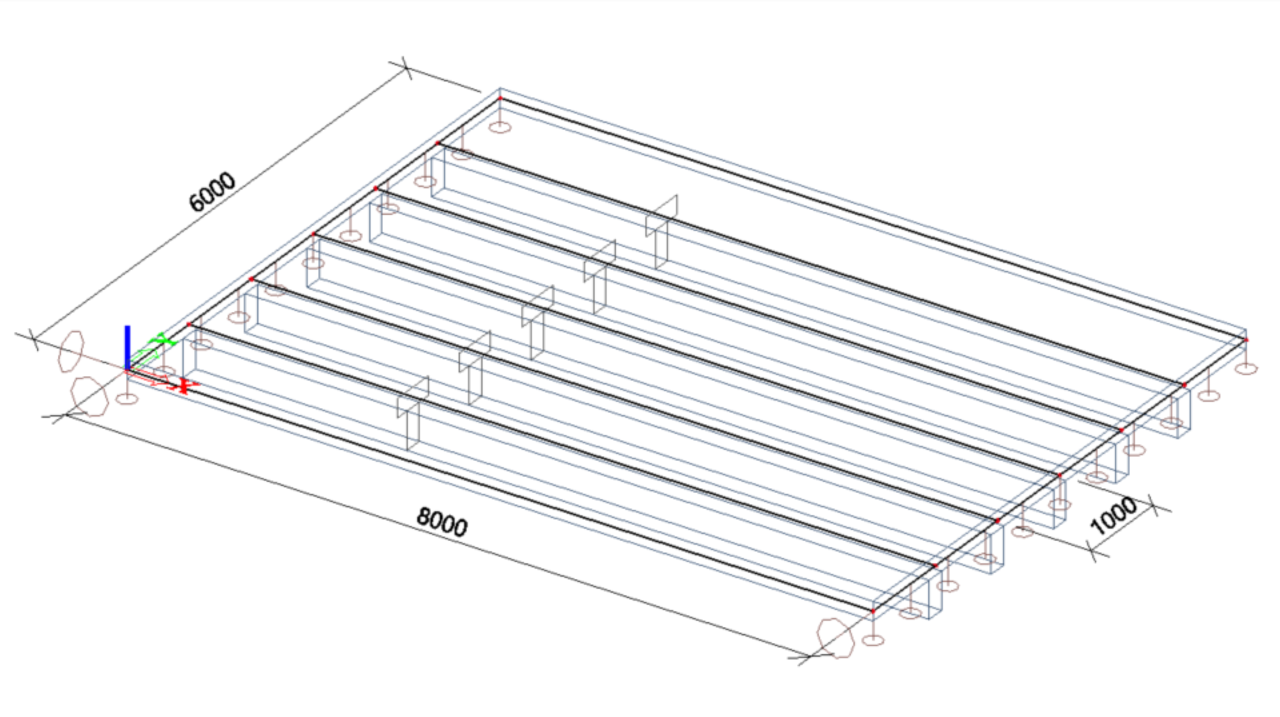
*Concrete > 2D member > Member check – Design – Crack width: for Class = All ULS+SLS, Type values = Required areas/Maximal diameters/Maximal distances/Shear stresses
Possible for both theoretical and practical reinforcement, see Concrete > 2D member > Setup > General > Calculation > For stiffness, allowable stress, punching and crack-proof calculation, use reinforcement

*Concrete > Punching > Punching check: for Class = All ULS
Possible for both theoretical and practical reinforcement, see Concrete > 2D member > Setup > General > Calculation > For stiffness, allowable stress, punching and crack-proof calculation, use reinforcement

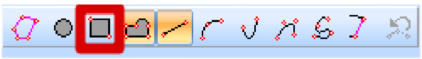
Example 15: Slab with ribs

1_Input of geometry

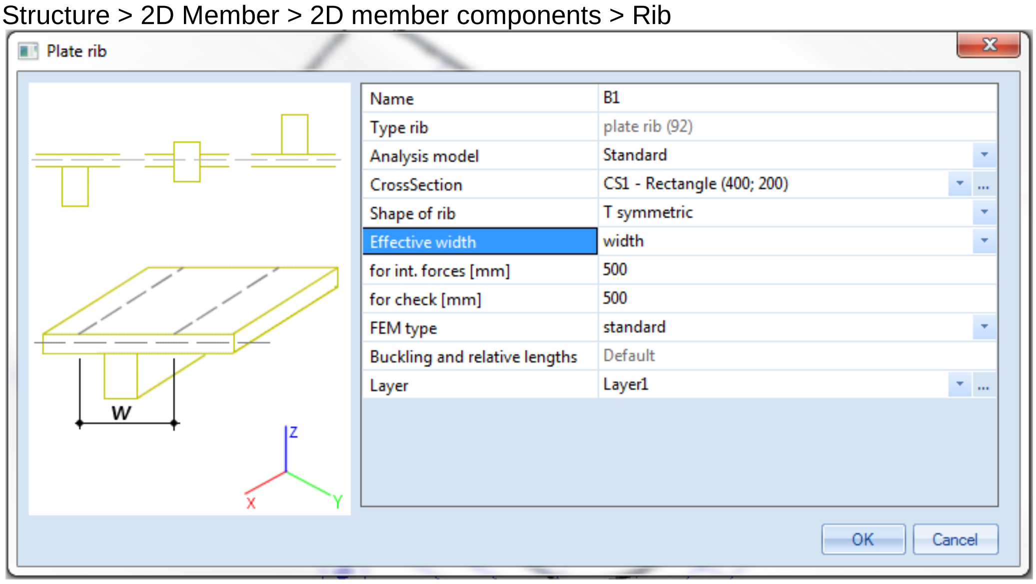
*Project data: General XYZ > necessary because of eccentricity of the ribs
Concrete C20/25 – Plate thickness 200mm – Ribs R 200mm x 400mm



*Input plate + ribs (Method 1):
Structure > 2D Member > Plate

New rectangle, via Command line toolbar 

Structure > 2D Member > 2D member components > Rib



Effective width = Default, Number of plate thickness, or Width in mm
Default: see Setup > Solver > Number of thicknesses of plate rib

Graphical display of effective width (T-section ribs)
via Set view parameters for all  > Structure > Draw cross-section

* Input plate + ribs (Method 2):
Structure > 2D Member > Ribbed slab

*Input supports: hinged
Structure > Model data > Support > Line on 2D member edge

2_Load cases

LC 1: Self weight
LC 2: Service load (Var.) > Surface load 5 kN/m²

3_Finite elements mesh

Refine mesh via Main menu > Calculation, Mesh > Mesh setup; size of 2D mesh elements = 0,25m

4_Results

*Results > Beams > Internal forces on beam; Values = N
Option Rib off: Results on the rectangular sections
Option Rib on: Results on the T-sections

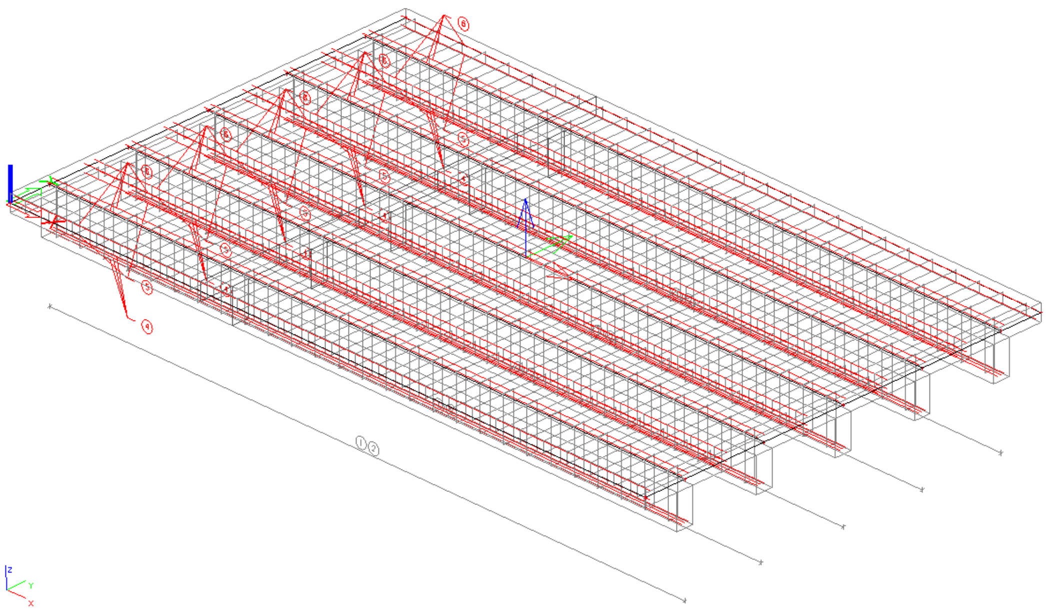
*Results > 2D members > Internal forces; Values = nx
Option Rib off: Results on the entire plate
Option Rib on: Results on the pieces of plate between the T-sections

5_Reinforcement in T-sections

The effective width is an approximation from the norm, where the connection beam-plate is replaced by a T-beam for the design of the reinforcement. By selecting the option Rib, the internal forces in the beam are adapted. These adapted forces represent the forces in the T-section, so they can be used to design the reinforcement in the T-beam.

Suppose: effective width = distance between the ribs

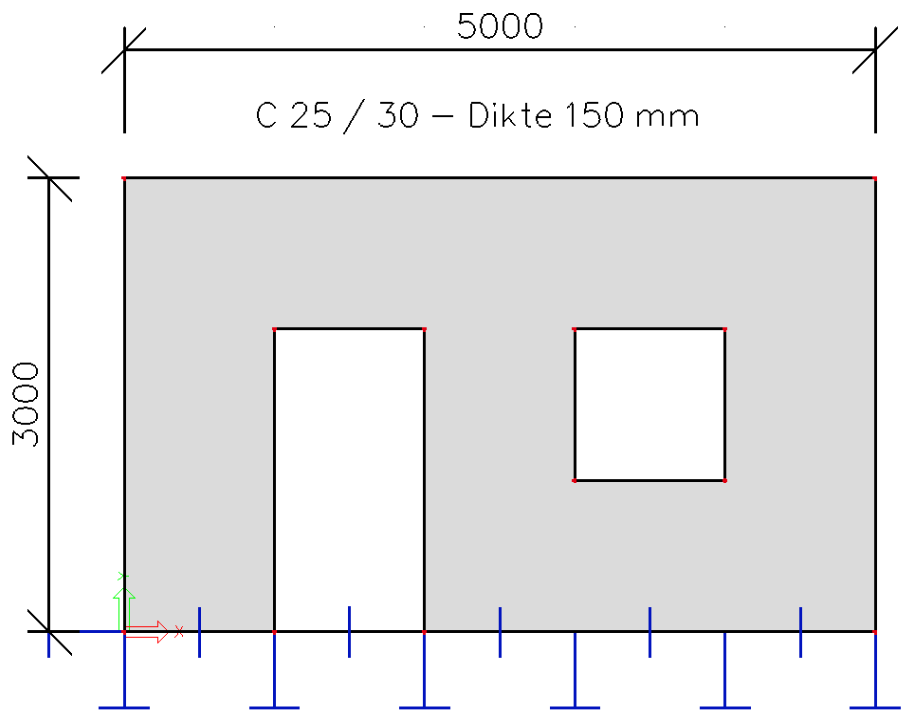
Define reinforcement:
Main menu > Concrete > 1D Member > Redes (without As) > New reinforcement



Example 16: Prefab wall

1_Input of geometry

*Project data: Wall XY – Project level Advanced



*Input wall
Structure > 2D Member > Plate
Structure > 2D Member > 2D member components > Opening

2_Load cases

LC 1: Self weight
LC 2: Prefab plates (Perm.) > Line force 13,2 kN/m

3_Finite Elements Mesh

*Global mesh = 0,3m

Set view parameters for all  > Structure > Mesh > Draw mesh

*Mesh refinement around the openings
Main menu > Calculation, Mesh > Local mesh refinement > 2D member edge mesh refinement; Size = 0,1m

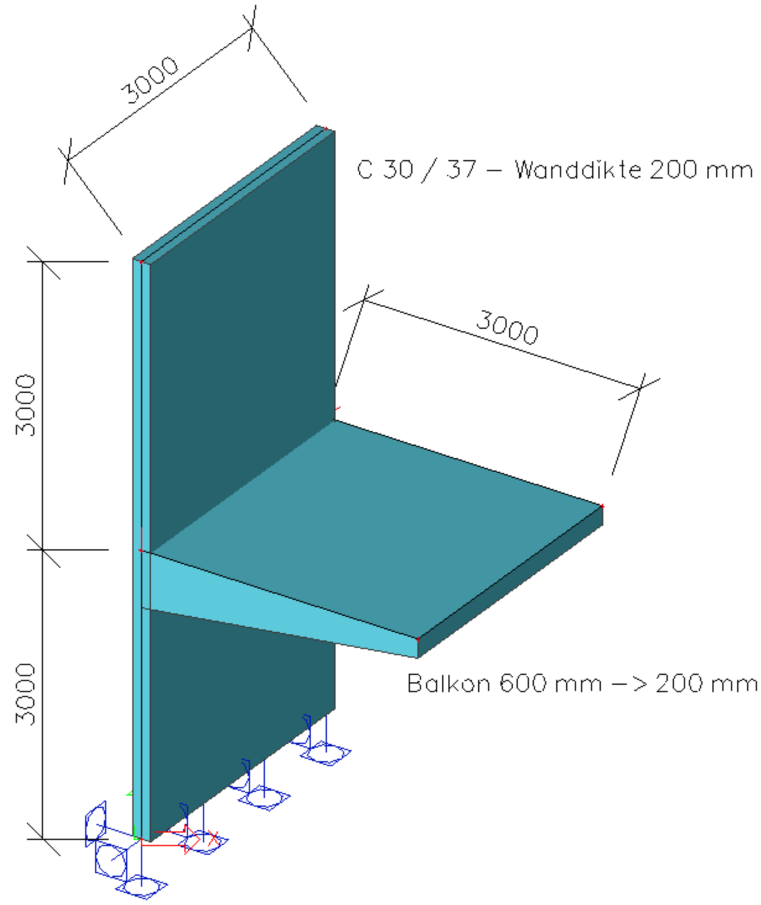
4_Results

Display the direction of the principal stresses as follows:
Results > 2D Members > Stresses
For LC 2: Type forces = Principal magnitudes, Values = sig1 or sig2, Drawing = Trajectories

Example 17: Balcony

1_Input of geometry

* Project data: General XYZ – Project level Advanced



*Input balcony
Structure > 2D Member > Wall
Structure > 2D Member > Plate; Thickness type = Variable, Member system plane at = Top

2_Actions after input

*Check structure data 

*Connect members/nodes  (Attention: connect the entire structure!)
This action is necessary to connect 2D members to each other, see Annex 1

3_Load cases

LC 1: Balustrade (Perm.) > Line force 10 kN/m

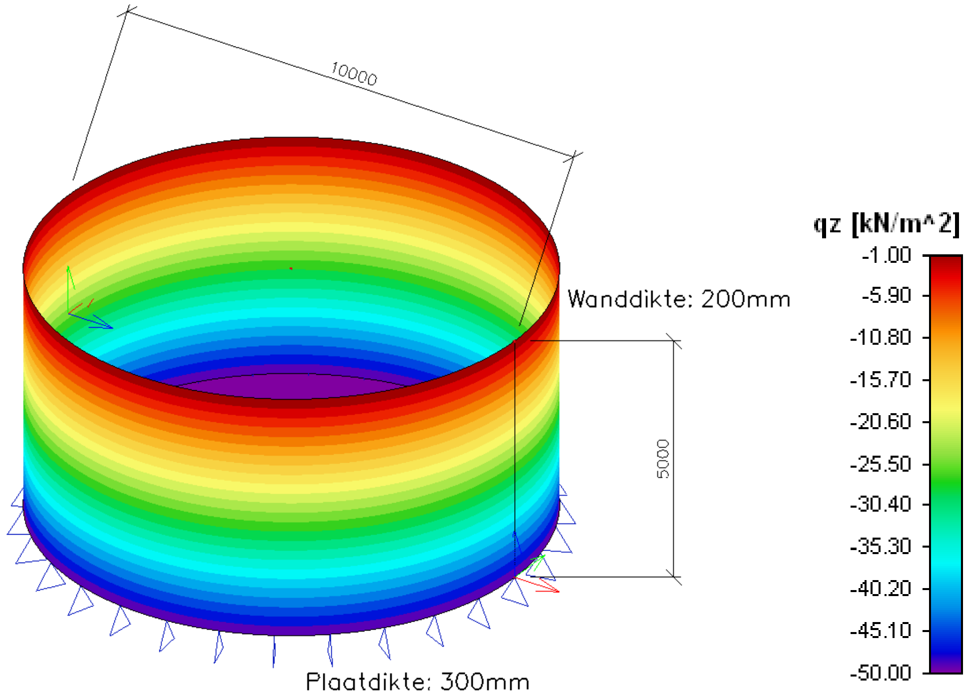
4_Results

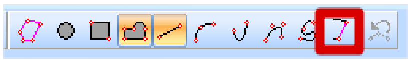
Check as follows if the structure is completely connected:
Results > 2D Members > Displacement of nodes
For LC 1: Structure = Initial, Values = Deformed mesh

Example 18: Tank

1_Input of geometry

*Project data: General XYZ – Project level Advanced



*Input tank
Base plate: Structure > 2D Member > Plate
New circle (centre – radius) 
Wall: Structure > 2D Member > Wall
Select line  ; select edge of base plate

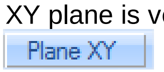
Display local axes of the 2D members, via Set view parameters for all > Structure > Local axes > Members 2D

*Input supports
Main menu > Project > Functionality: Subsoil
Structure > Model data > Support > Surface (elast. foundation)

2_Loads

*Load cases
LC 1: Self weight
LC 2: Varied pressure (Var.) > Surface load 0 to 50 kN/m²

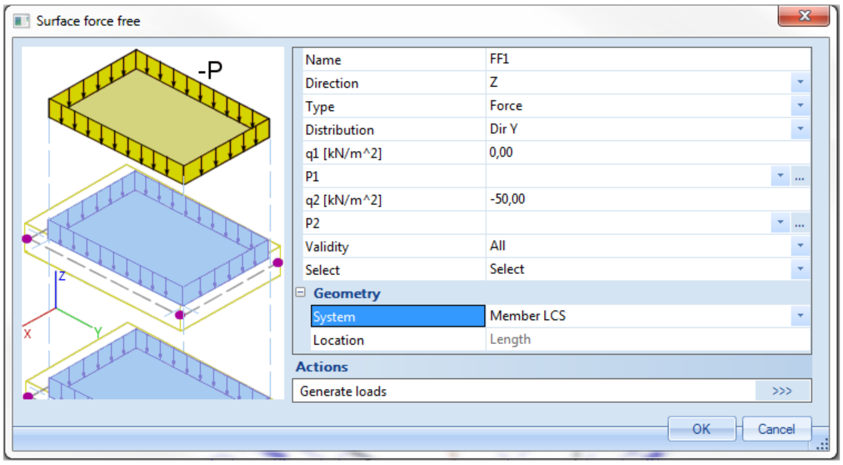
*Free surface load
Input of varied pressure as a free surface load

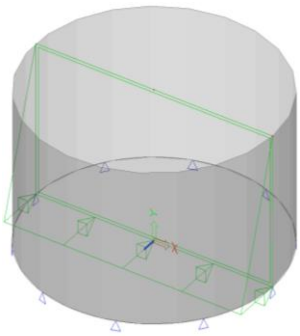
a)The geometry of a free load always has to be inputted in the XY plane of the current UCS > Define UCS at first, via ‘Tools’ toolbar , so the XY plane is vertical and e.g. the Y axis is pointing upwards
Set Plane XY = Active working plane, see 

at the bottom of the Command line

b)Surface load > Free
  -Surface load acts in the direction of the local z axis of the 2D members
  Direction = Z, System = Member LCS

-Linear variation of the load over the height
Distribution = Direction Y





Input the geometry of the free load as a New rectangle in the XY plane

After input: change positions P1 and P2 in the Properties menu if necessary; since these are dependent of the way of inputting the geometry

- Select yourself the members on which the free load has to act
Select = Select
Actions > Update 2D members selection > Select 2D members

See also Annex 4: Free loads

3_Finite Elements Mesh

Refine mesh; size of mesh elements = 0,2m

4_Check of input data

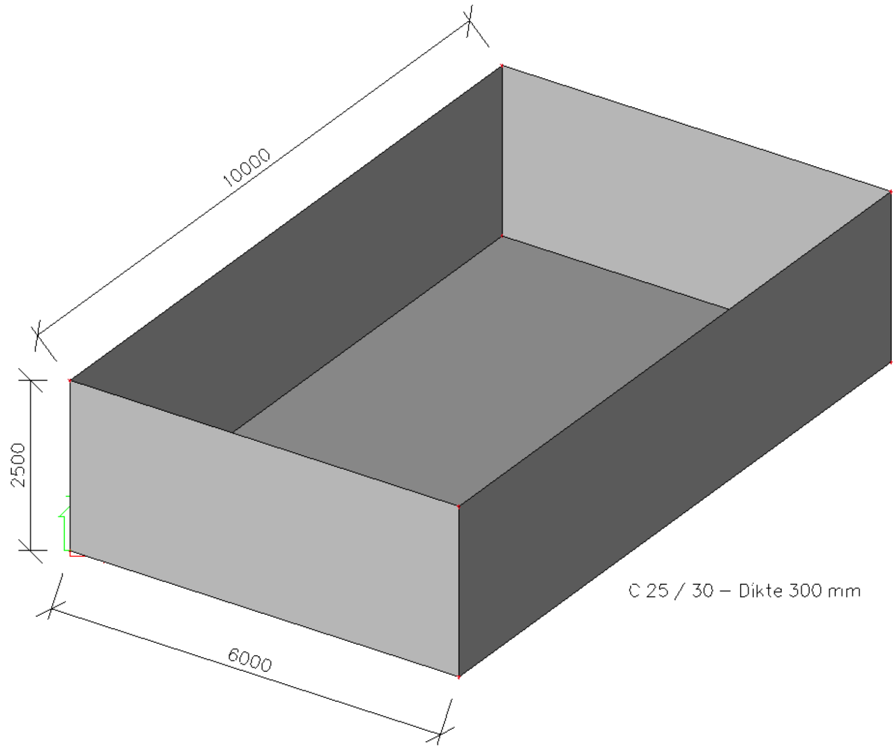
*Main menu > Calculation, Mesh > Calculation; Test of input data

*Main menu > Calculation, Mesh > 2D data viewer > Surface loads
LC 1: Values = qz, System = Global
LC 2: Values = qz, System = Local

Example 19: Swimming pool

1_Input of geometry

*Project data: General XYZ – Project level Advanced



*Input swimming pool
Structure > 2D Member > Plate

Structure > 2D Member > Wall; Select line 

*Input supports
Main menu > Project > Functionality: Subsoil
Structure > Model data > Support > Surface (elast. foundation); select plate and walls
Structure > Model data > Support > Line on 2D member edge; select edges of ground plate

Select 2D members

-All members at once: via 'Selection of object' toolbar 

-Specific element: Simple (single) selection of member, via 'Selection of object' toolbar 

A subsoil is always added at the negative side of the local z axis of the 2D member.

-Check orientation of the local z axes, via Set view parameters for all  > Structure > Local axes > Members 2D
-Select the elements for which the local z axis is not pointed at the middle of the swimming pool > in Properties menu: select option Swap orientation

2_Loads

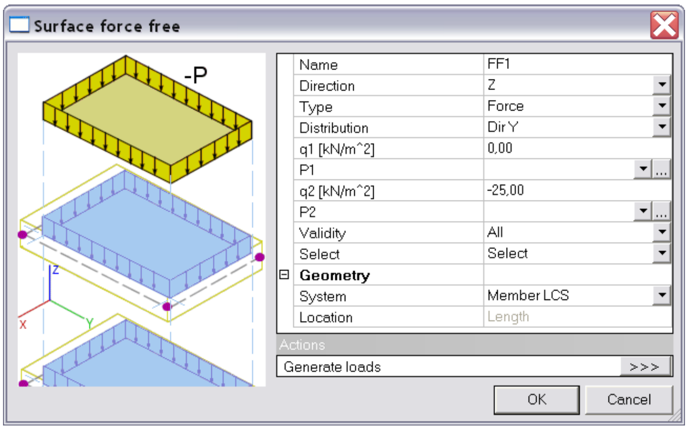
*Load cases
LC 1: Self weight
LC 2: Water pressure (Var.) > Surface load 0 to 25 kN/m²

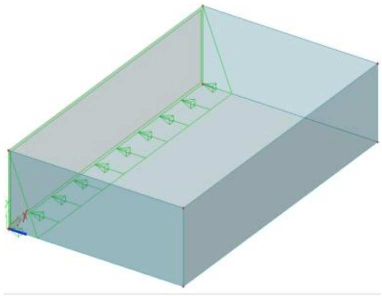
*Free surface load
Input of water pressure as a free surface load

a)The geometry of a free load always has to be inputted in the XY plane > Define UCS at first, via 'Tools' toolbar , so the XY plane is vertical and for instance the Y axis is pointing upwards
Set Plane XY = Active working plane, see Plane XY at the bottom of the Command line

b)Surface load > Free
-Surface load acts in the direction of the local z axis of the 2D members
Direction = Z, System = Member LCS

-Linear variation of the load over the height
Distribution = Direction Y





Input the geometry of the free load as a New rectangle in the XY plane

After input: change positions P1 and P2 in the Properties menu if necessary; since these are dependent of the way of inputting the geometry

-Select yourself the members on which the free load has to act
Select = Select
Actions > Update 2D members selection > Select 2D members

3_Finite elements mesh

Refine mesh; size of mesh elements = 0,3m

4_Check of input data

*Main menu > Calculation, Mesh > Calculation; Test of input data

*Main menu > Calculation, Mesh > 2D data viewer > Surface loads
LC 1: Values = qz, System = Global
LC 2: Values = qz, System = Local

5_Results

Section on wall: Results > 2D Members > Section on 2D member

Direction of cut = 1;0;0 (for section in X direction) or 0;1;0 (for section in Y direction) = 2nd co-ordinate of a direction vector which defines the direction of the section (1st co-ordinate is the origin)

Example 20: Cooling tower

1_Input of geometry

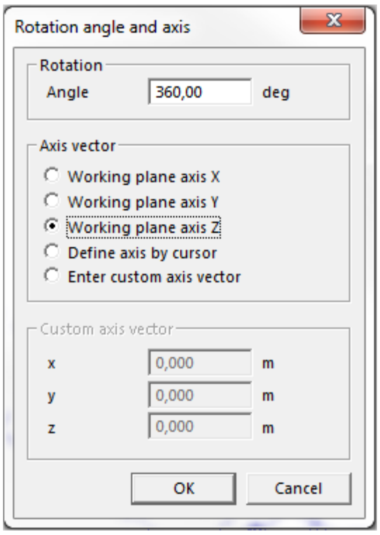
*Project data: General XYZ – Project level Advanced
Concrete C30/37 – Shell thickness 200mm – Height of pillars 5m – Height of tower 35m
Radius base plate 15m – Radius tower Bottom 13,5m / Top 9m – V-pillars CIRC (500)

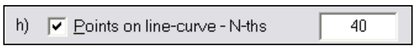
*Input of base plate
Structure > 2D Member > Plate; New circle with radius 15m

*Input of tower
Structure > 2D Member > Shell – surface of revolution

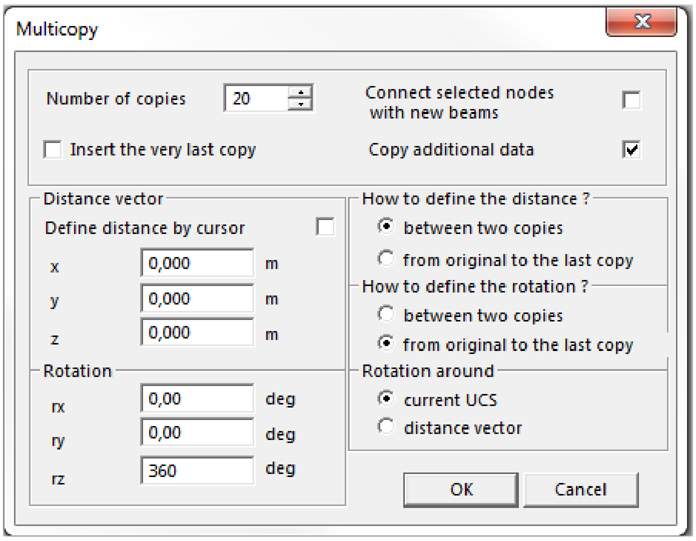
Define line of revolution:
New parabolic arc,
see Command line toolbar

Start point 13,5;0;5
Intermediate point 8;0;25
End point 9;0;40



*Input of 20 V-pillars
Cursor snap settings  > Select option h 

Structure > 1D Member > Beam; input 2 bars to form a V-shape
Multicopy, via ‘Geometry manipulations’ toolbar 



*Input of support
Structure > Model data > Support > Line on 2D member edge

2_Actions after input

*Check structure data

*Connect members/nodes (Attention: connect the entire structure!)

3_Loads

*Load cases
LC 1: Self weight
LC 2: Temperature load (Var.) > Thermal on 2D member, Delta = 40 K
LC 3: Wind load (Var.) > Surface load 0 to 1,4 kN/m²

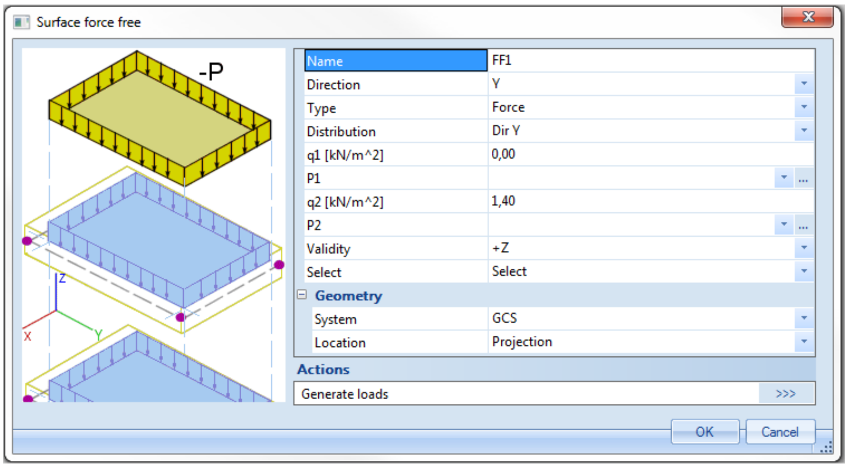
*Load groups
LG 1: Permanent
LG 2: Variable, EC1 Load type = Temperature
LG 3: Variable, EC1 Load type = Wind

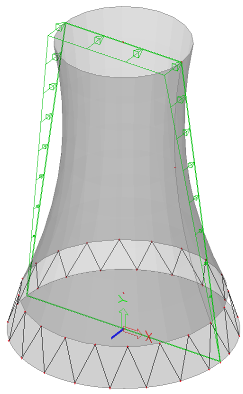
*Free surface load
Input of wind load as a free surface load

a)The geometry of a free load always has to be inputted in the XY plane > Define UCS at first, via 'Tools' toolbar , so the XY plane is vertical and for instance the Y axis is pointing upwards
Set Plane XY = Active working plane, see Plane XY at the bottom of the Command line

b)Surface load > Free
-Surface load acts in the direction of the Y axis of the GCS
Direction = Y, System = GCS

-Linear variation of the load over the height
Distribution = Direction Y





Input the geometry of the free load as a New polygon in the XY plane

-Select yourself the members on which the free load has to act
Select = Select
Actions > Update 2D members selection > Select 2D members

-Only one side of the cooling tower is loaded by the wind
Validity = +Z

-Location = Projection

4_Check of applied loads

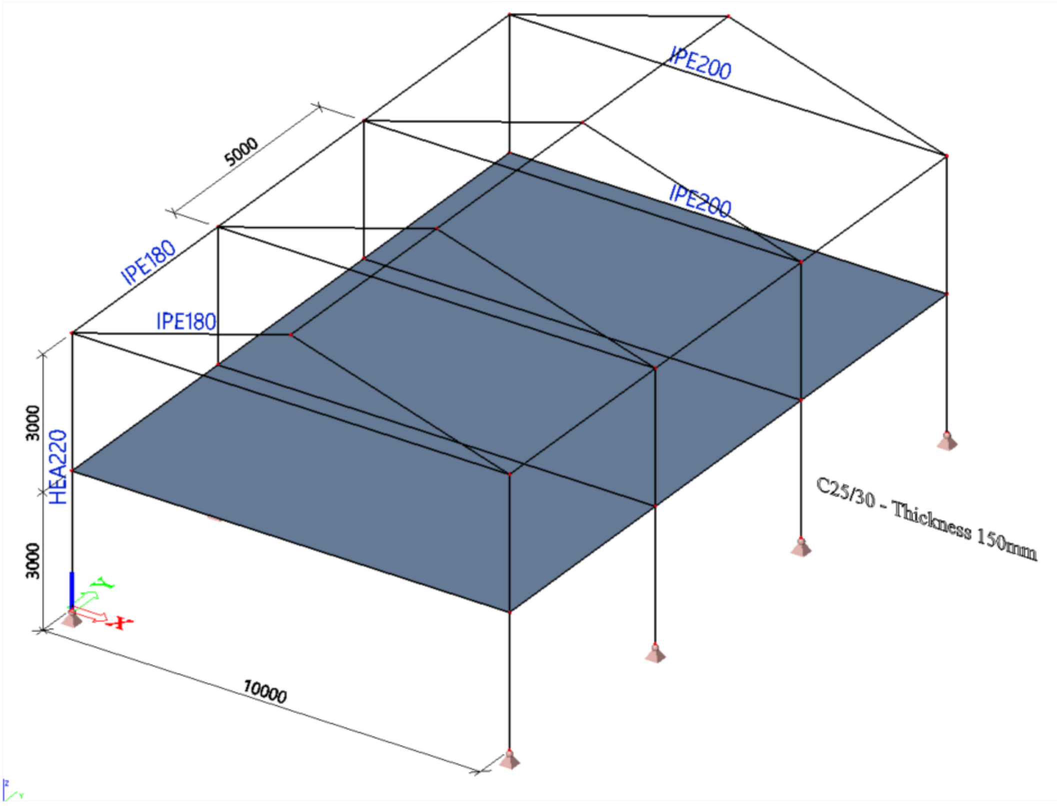
*Main menu > Calculation, Mesh > Calculation; Test of input data
Main menu > Calculation, Mesh > 2D data viewer > Surface loads
LC 1: Values = qz, System = Global
LC 3: Values = qy, System = Global

*Main menu > Calculation, Mesh > Calculation; Linear calculation
Main menu > Calculation, Mesh > 2D data viewer > Temperature load
LC 2: Values = epsilon

Example 21: Steel hall with concrete plate

1_Input of geometry

*Project data: General XYZ – Project level Advanced – Concrete & Steel
*H2 = 1,5m



*Input hall
-First frame, via Structure > Advanced input > Catalogue blocks; choose Frame 2D

-Multicopy, via 'Geometry manipulations' toolbar  > Automatic generation of connecting bars from the selected nodes

*Input slab
Structure > 2D Member > Plate
-New rectangle: only possible to input this geometry in the Active working plane

Move GCS at first to the first story, via 'Tools' toolbar  + Set Plane XY = Active working plane
-New polygon: input of this geometry is independent of the Active working plane
Input of the geometry line by line

2_Connections between entities

*Connection of the whole structure

Connect members/nodes, via 'Geometry manipulations' toolbar 
+ Select option Check structure data

*Connection beam – plate
Concerning a beam which does not coincide with the edge of a plate, the connection beam – plate has to be created manually by means of an internal edge. See also Annex 1
Structure > 2D Member > 2D Member components > Internal edge

REMARK: When a beam has been inputted as a plate rib, it is by default connected rigidly to the plate. The use of an internal edge is in that case superfluous, see also Ex. 15

3_Load cases

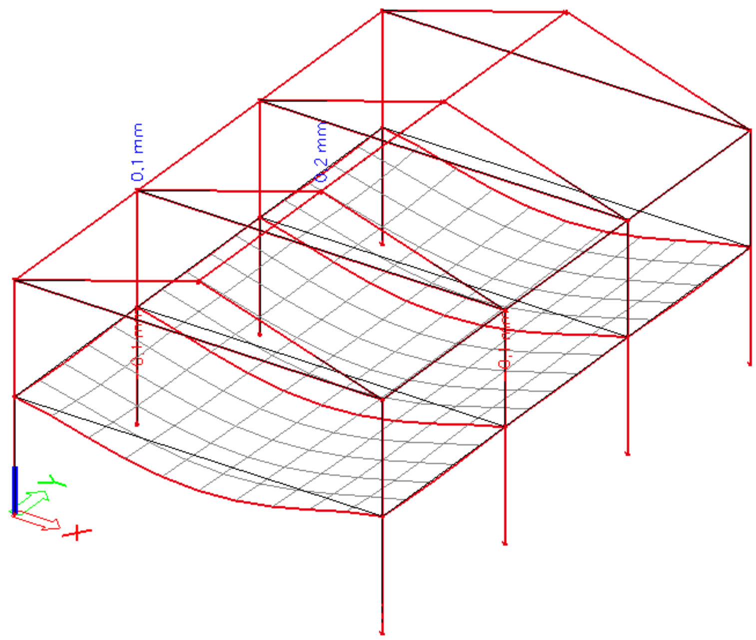
LC 1: Self weight
LC 2: Service load (Var.) > Surface load 2 kN/m²

4_Check connections

After calculation, check as follows if the construction has been completely connected:

*Compare deformation Uz of beams & plate
-Results > Beams > Deformations on beam
-Results > 2D Members > Displacement of nodes
Make sections on the plate at the connections with the beams: Results > 2D Members > Section on 2D member

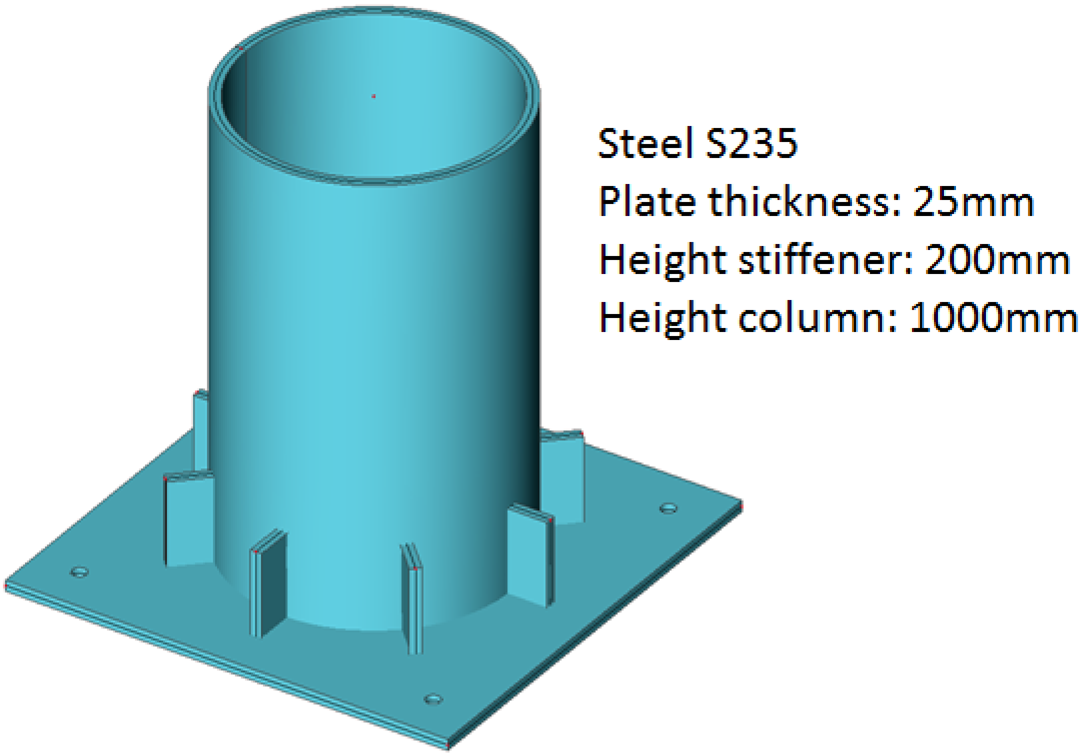
*Check the deformed mesh
Results > 2D Members > Displacement of nodes
Choose a Load case: Structure = Initial, Values = Deformed mesh
The beams are displayed in red, check if they deform along with the mesh of the plate.

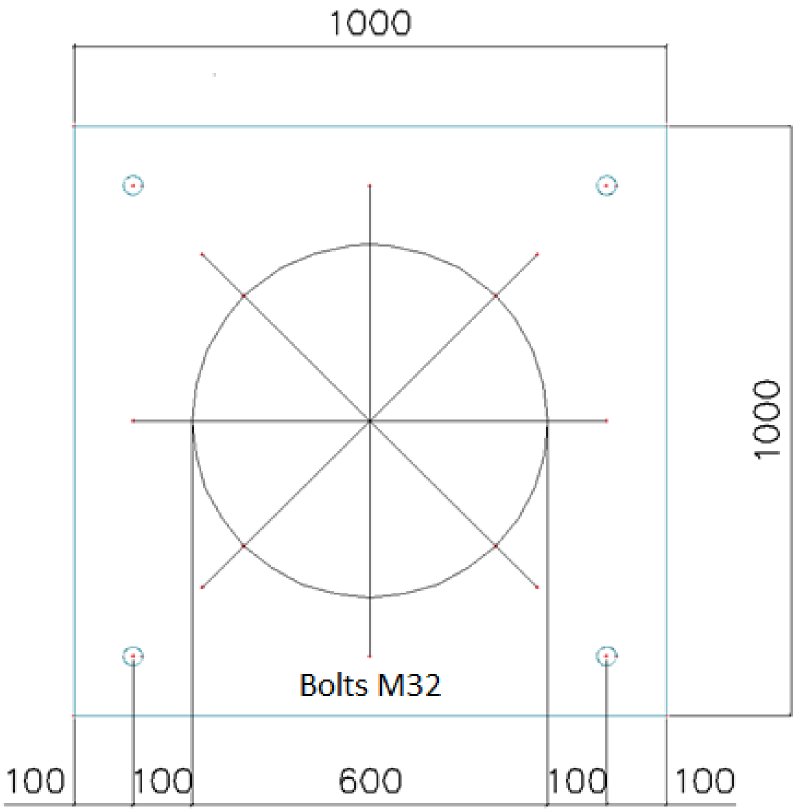


Example 22: Detailed study of a column base

1_Input of geometry

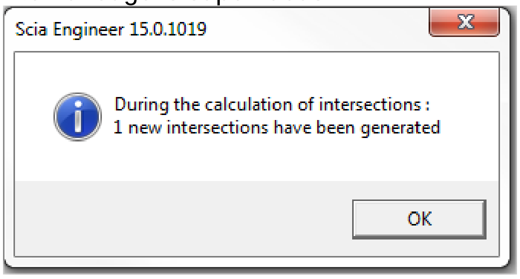
*Project data: General XYZ – Project level Advanced





*Input column base
Base plate: Structure > 2D Member > Plate
Column: Structure > 2D Member > Wall; New circle (centre – radius) with midpoint (0,5;0,5) and point at circle (@0,3;0)

Connect members/nodes  > Intersection column – base plate is generated automatically, an internal edge is superfluous



*Input bolt holes
Margin is neglected > diameter of the bolt holes = 32mm

-Input by means of a Line grid, see 'Tools' toolbar 

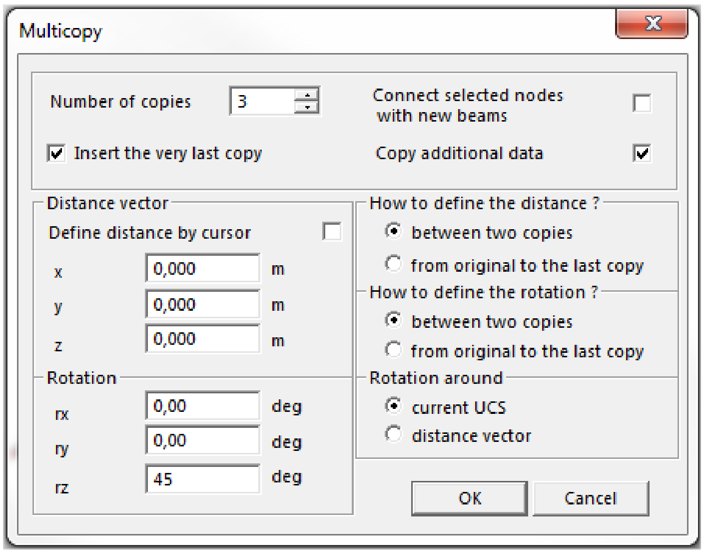
Snap to the dots of the Line grid by means of Cursor snap settings, see Command line toolbar  or via right mouse click in screen
-First bolt hole, via Structure > 2D Member > 2D member components > Opening; New circle (centre – radius) with point at circle (@0,016;0)

-Copy bolt holes, via 'Geometry manipulations' toolbar 

*Input stiffeners
-First fin: Structure > 2D Member > Wall; input of fin on line grid or at random position

-Move fin, via 'Geometry manipulations' toolbar ; Start point = midpoint of bottom side of fin, End point = midpoint of column

-Multicopy, via 'Geometry manipulations' toolbar 
Attention: Rotation around current UCS > Move UCS beforehand to the midpoint of the circle, via 'Tools' toolbar 



Connect members/nodes  > Intersections are generated automatically, an internal edge is superfluous

-Remove the part of the stiffeners at the inside of the column
Structure > 2D Member > 2D Member components > Cut-out

*Input supports
Main menu > Project > Functionality: Subsoil
Structure > Model data > Support
Base plate: Surface (elas. foundation); Default subsoil Subsoil 1
Bolt holes: Line on 2D member edge; all translations fixed

2_Actions after input

*Check structure data 

*Connect members/nodes  (Attention: connect the entire structure!)

3_Loads

*Load cases
LC 1: Self weight
LC 2: Normal force: –60 kN/m at the top edge of the column
LC 3: Moment: 20 kNm/m at the top edge of the column in the Y direction (lever arm = height of column = 1m)

*Load combinations
Linear – ULS: 1,00.LC 1 + 1,00.LC 2 + 1,00.LC 3

3_Finite elements mesh

*Global mesh refinement
Main menu > Calculation, Mesh > Mesh setup; size of the mesh elements = 0,025m

*Local mesh refinement around the bolt holes
Main menu > Calculation, Mesh > Local mesh refinement > Node mesh refinement; around midpoint of bolt holes, Radius = 0,050m en Ratio = 0,01

*Mesh generation

Main menu > Calculation, Mesh > Mesh generation, or ‘Project’ toolbar 

Graphical display: Set view parameters for all  > Structure > Mesh > Draw mesh
Verify that the inner parts of the stiffeners will not be taken into account for the calculation: no mesh is being generated on these parts

The elastic mesh in the mesh setup provides a fluent transition between mesh sizes.

4_Results

Results > 2D Members > Displacement of nodes
Choose a Load case: Structure = Initial, Values = Deformed mesh
Check if the structure is entirely connected

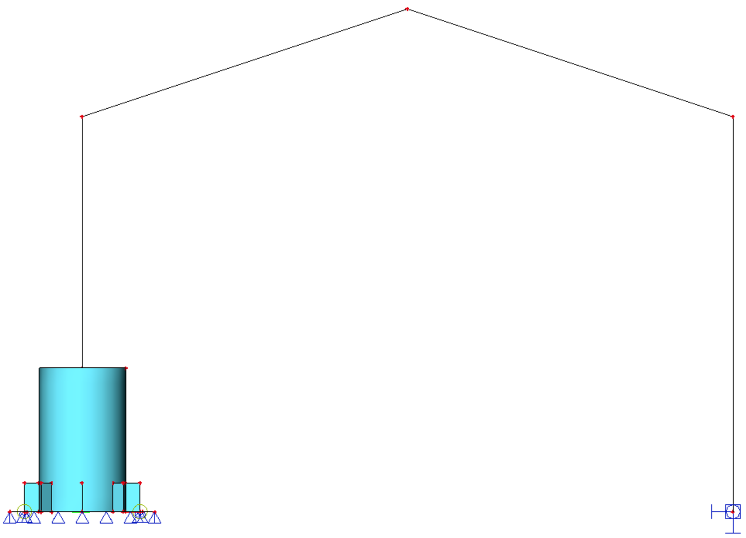
Results > 2D Members > Stresses
Look at the concentration of stresses around bolt holes and stiffeners

5_Link 2D (detail column base) – 1D (entire structure)

*Structure > 1D Member > Column; Add a 1D column with the same properties as the 2D column, insertion point = 0,5;0,5;1

*Transfer of the internal forces from the 1D structure to the 2D column base:
Structure > Model data > Line rigid arms; master node = insertion point of 1D column, slave edge = top edge of 2D column

A rigid arm is a very stiff 1D member which transfers all displacements from 1 master node to one or more other nodes, or to a (2D member) edge, without any change in the values of the displacements.

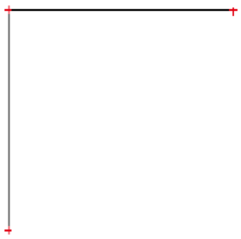


6_Extra

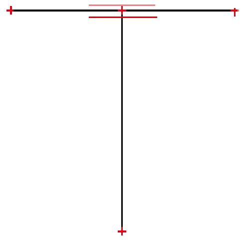
For advanced calculations, this analysis model can be expanded further on by means of e.g.
-Horizontal pressure only supports at the bolt holes
-Taking into account the tolerance of the bolt holes
-Stiffness parameters subsoil
-Pressure only subsoil
-Bevelling the stiffeners
-…

Annexes

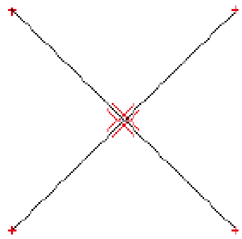
Annex 1: Connection of entities



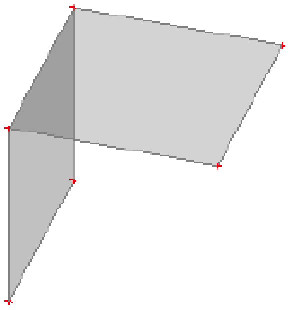
OK
Beams are connected



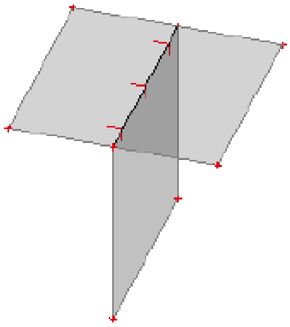
Structure menu
> Model data
> Connect members/ nodes



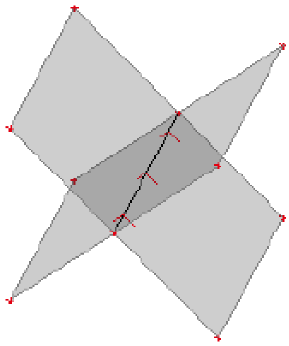
Structure menu
> Model data
> Cross-link



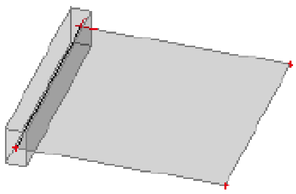
OK
Edges are connected



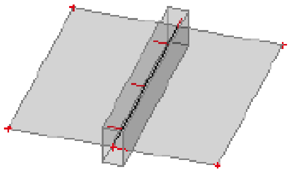
Structure menu
> Model data
> Connect members/ nodes



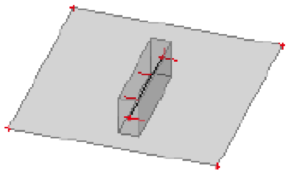
Structure menu
> Model data
> Connect members/ nodes



OK
Edge and beam are connected



*Connect members/ nodes: only nodes are connected
*Structure menu
> 2D Member
> 2D Member components
> Internal edge: entire beam connected

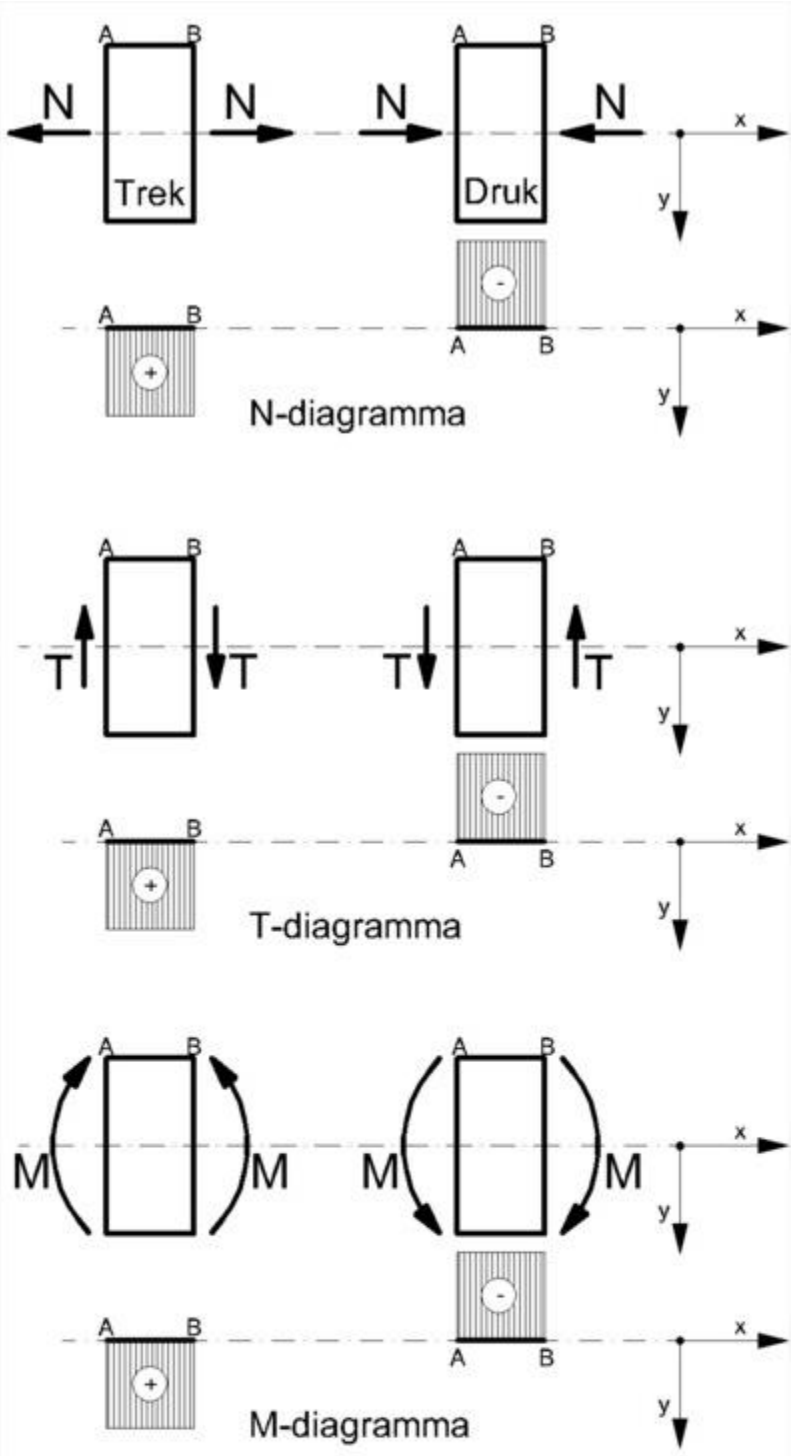


*Connect members/ nodes: only nodes are connected
*Structure menu
> 2D Member
> 2D Member components
> Internal edge: entire beam connected

Annex 2: Conventions for the results on 2D members

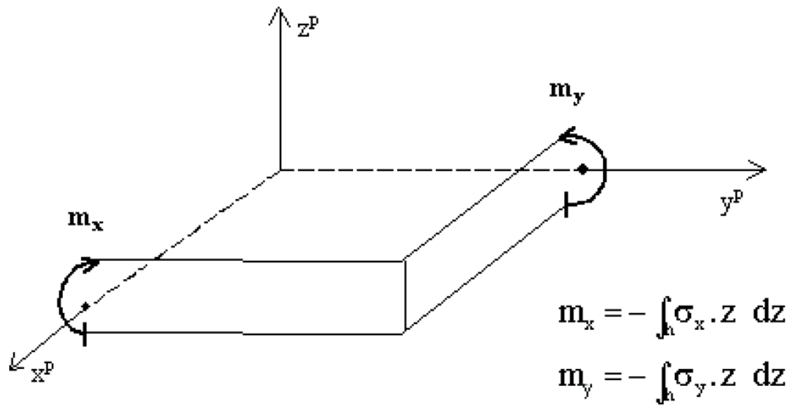
1_ Basic magnitudes = Characteristic values

Beams 1D

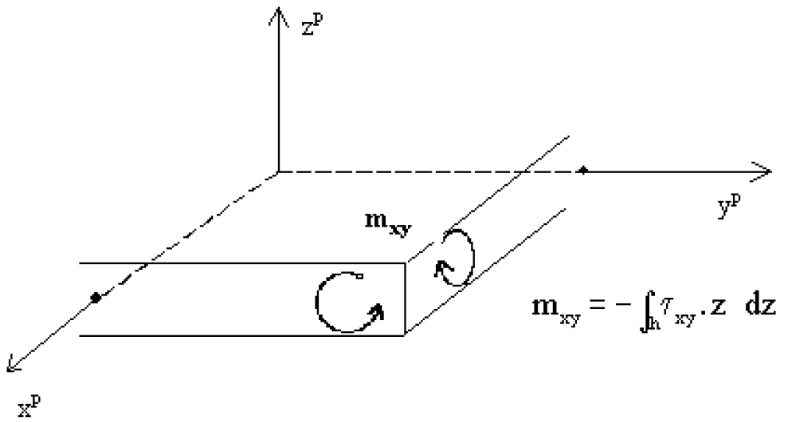


Bending (plates, shells)

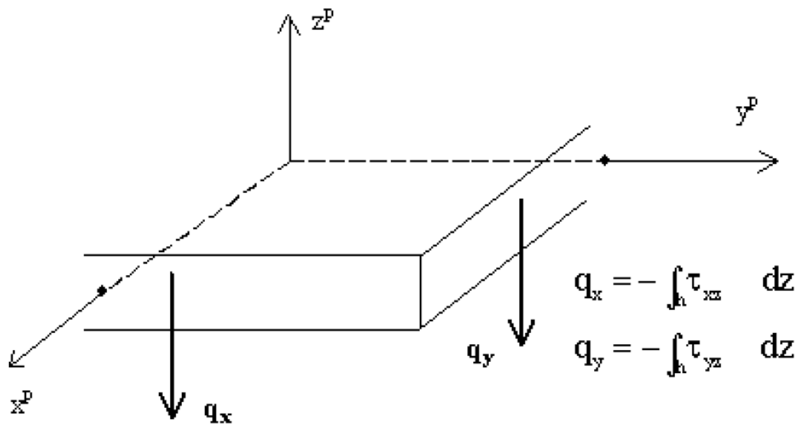
*Bending moments mx, my



*Torsion moment mxy

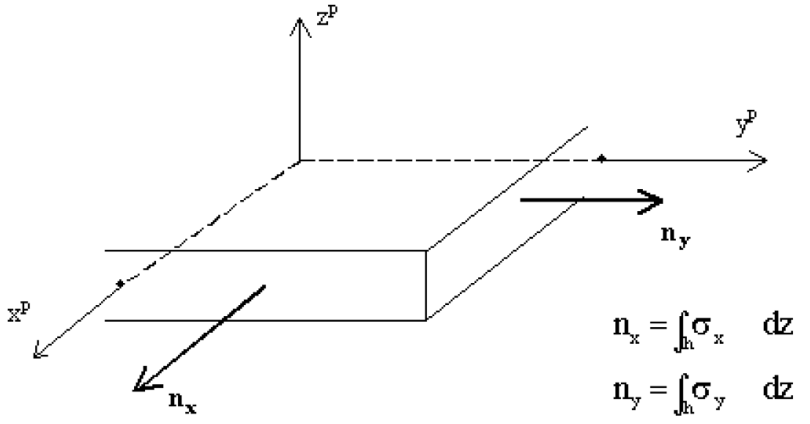


*Shear forces qx, qy (=vx, vy)

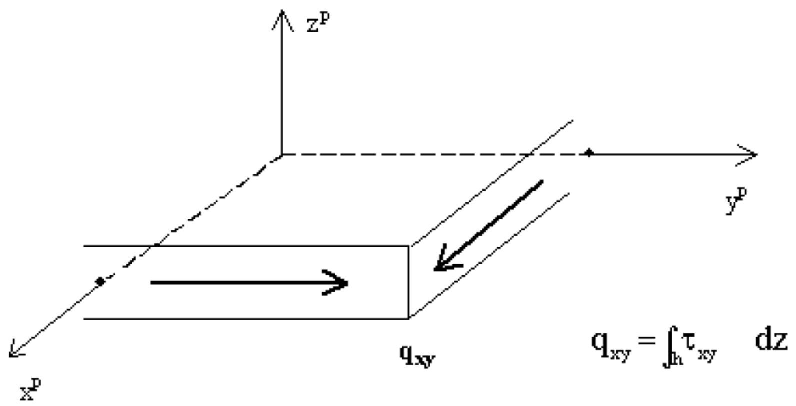


Membrane effects (walls, shells)

*Membrane forces nx, ny

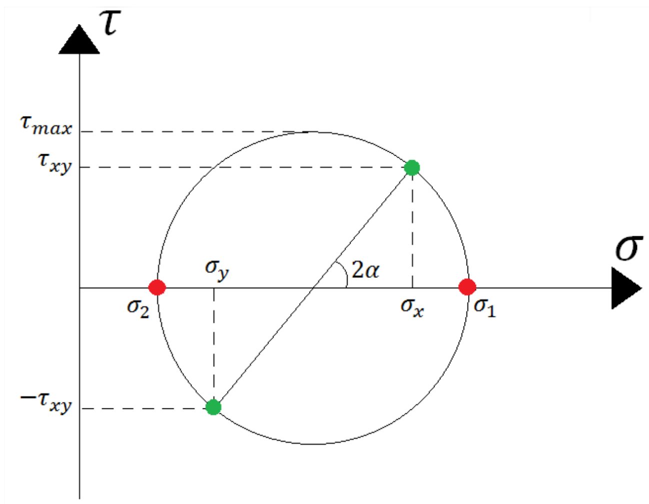


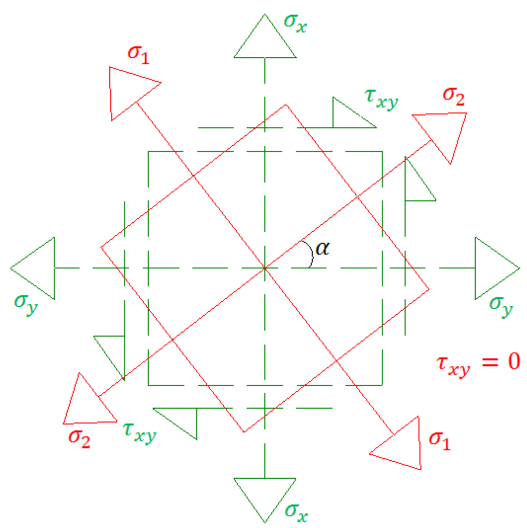
*Shear force qxy (=nxy)



2_Principal magnitudes

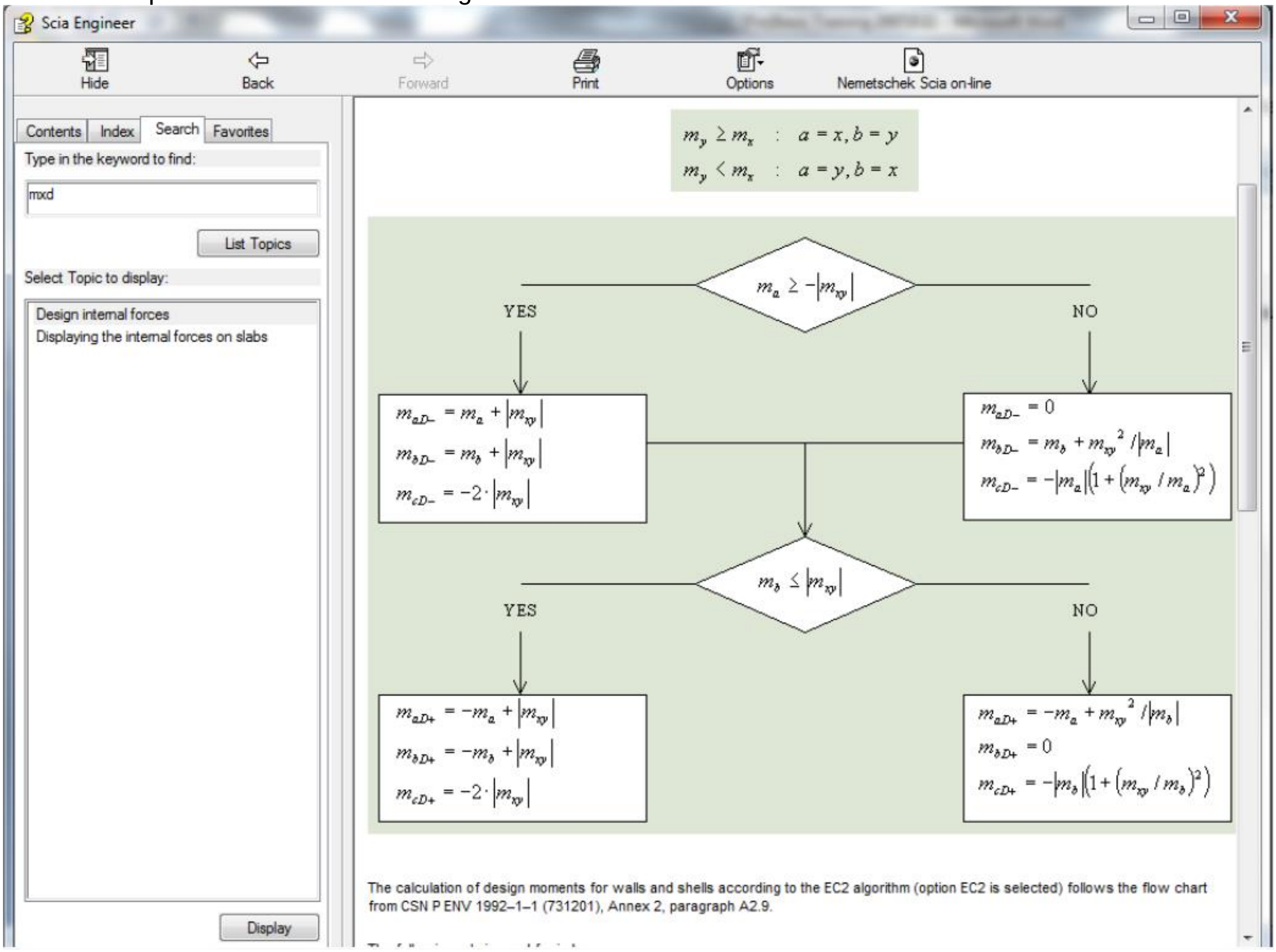
The principal magnitudes give the results according to the axes of the directions of the largest stresses (principal directions). These directions are defined with the help of the Mohr's circle.





3_Dimensional magnitudes = Design values

To derive the dimensional magnitudes from the basic magnitudes, formulas from the Eurocode are used.
See also Help > Contents > Reference guide for these formulas.



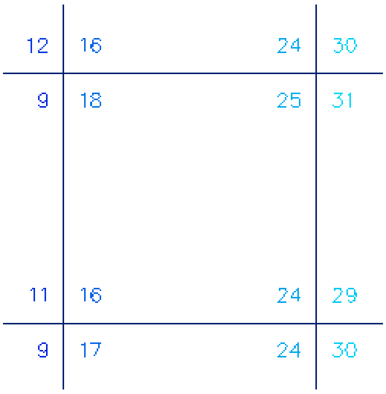
Annex 3: Results in mesh elements and mesh nodes → 4 Locations

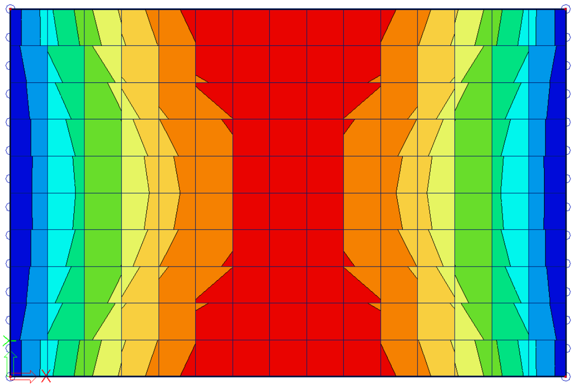
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always less accurate.

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 asking the results on 2D members, the option ‘Location’ in the Properties window gives the possibility to display these results in 4 ways.

1_ In nodes, no average

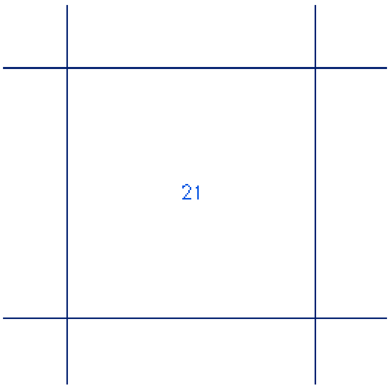
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.
This display of results therefore gives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

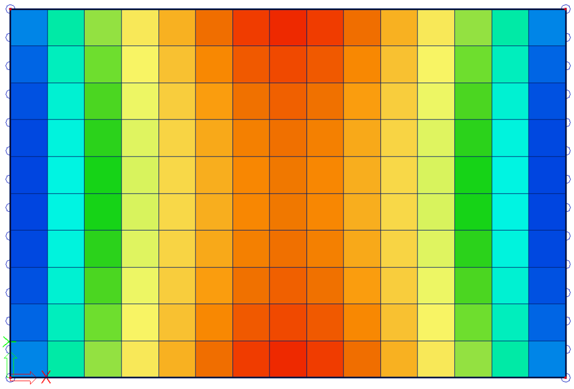




2_ In centres

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.



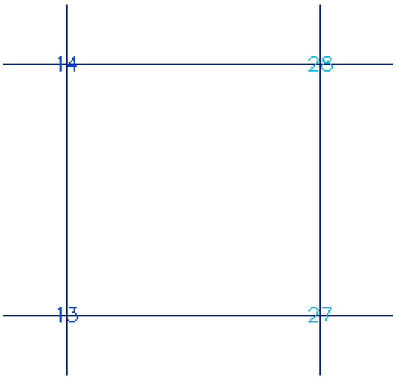


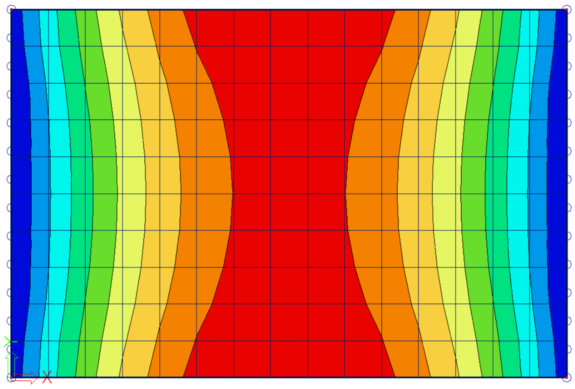
3_ In nodes, average

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.

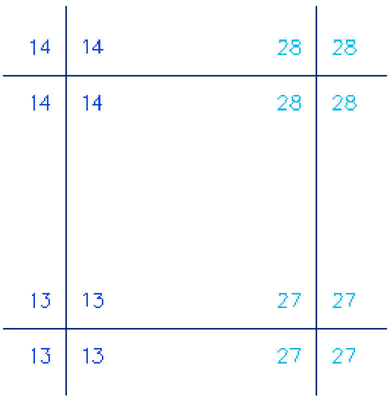
- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.

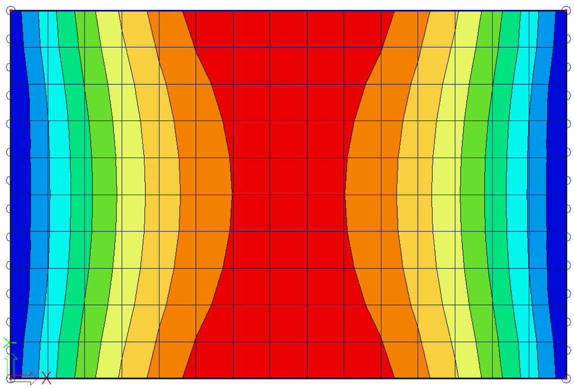




4_ In nodes, average on macro

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





Accuracy of the results

If the results according to the 4 locations differ a lot, then the results are inaccurate and the mesh has to be refined. A basic rule for a good size of the mesh elements, is to take 1 to 2 times the thickness of the plate.

Annex 4: Free loads

Definition

A free load differs from a ‘regular load’ by the fact that it is NOT attributed as an additional data to a specific 2D member. A free load can be created at an arbitrary position in space, and afterwards the user can specify to which element(s) the projection of this load is attributed to.

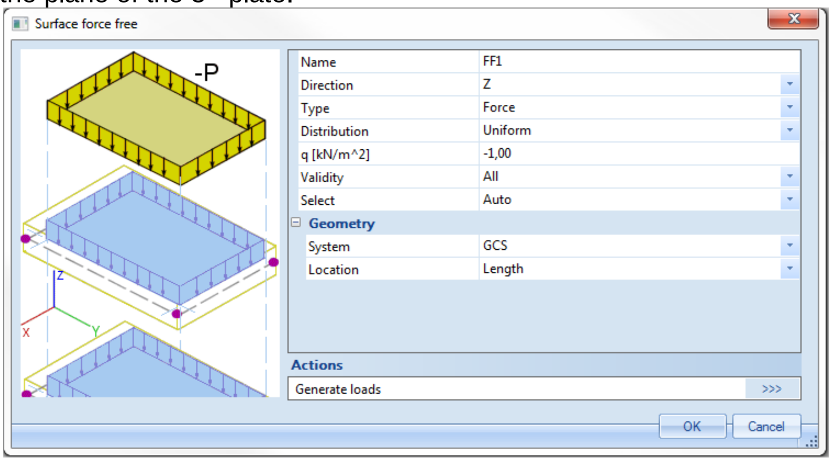
Attention: The geometry of a free load always has to be inputted in the XY plane of the current UCS. It is thus necessary to adapt the UCS in advance, and to assign the XY plane as the Active working plane.

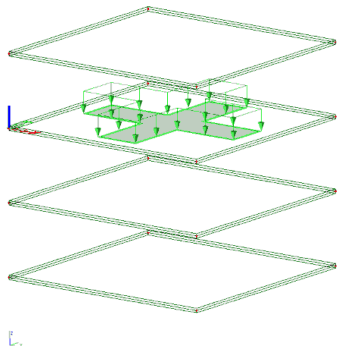
A free load *can* load all elements which are cut by the projection of the free load. Which elements will be actually loaded by the free load, depends on the parameters Select: Auto(matically), Select; and Validity: All, -Z, Z=0, +Z, From-to.
 Validity = -Z means: Only the elements situated *under* the free load (situated in the half-space defined by the negative Z direction of the UCS at input), can be loaded.
 Validity = +Z means: Only the elements situated *above* the free load (situated in the half-space defined by the positive Z direction of the UCS at input), can be loaded.

Example

An apartment building, where it is likely that the same load configuration acts on more than one storey floor.

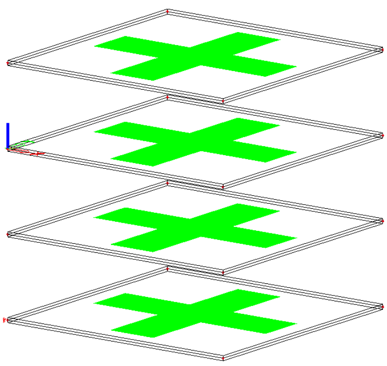
Let’s suppose: Four plates situated right above each other, and a free surface load inputted exactly IN the plane of the 3rd plate.





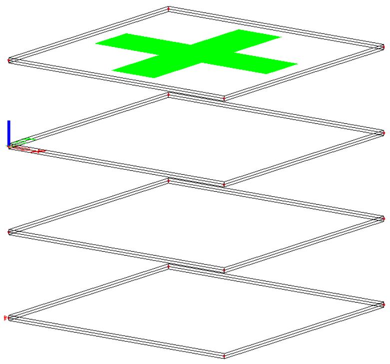
Main menu > Calculation, Mesh > Calculation; choose ‘Test of input data’
Main menu > Calculation, Mesh > 2D data viewer > Surface loads

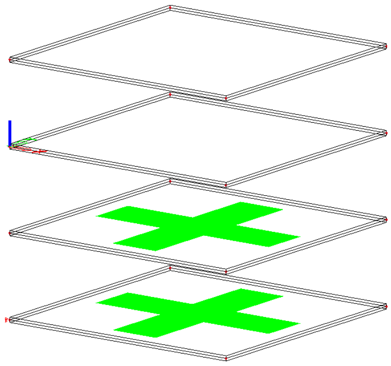
1) Select = Auto, Validity = All



2) Select = Auto, Validity = +Z / Validity = –Z

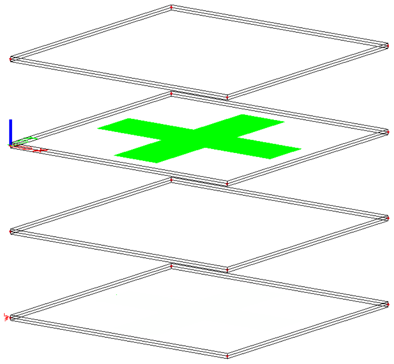
(Attention: The free surface load is placed exactly IN the plane of the 3rd plate.)





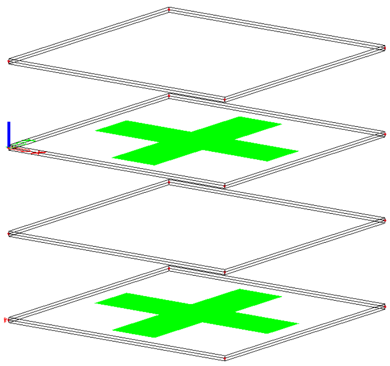
3) Select = Auto, Validity = Z=0

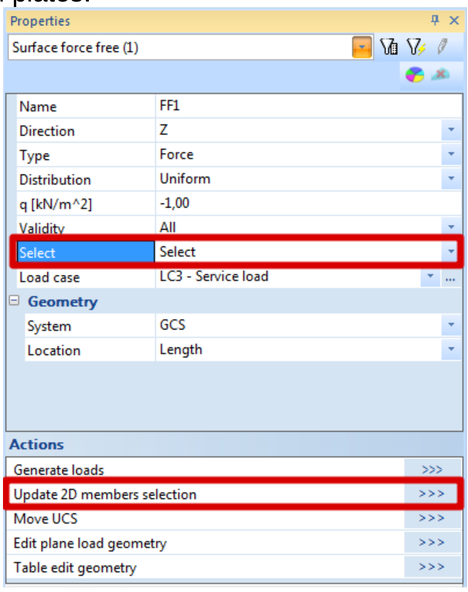
(Attention: The free surface load is placed exactly IN the plane of the 3rd plate.)



4) Select = Select, Validity = All

Actions > Update 2D members selection > Select the 1st and 3rd plate
Result: The load only acts on the manually selected plates.



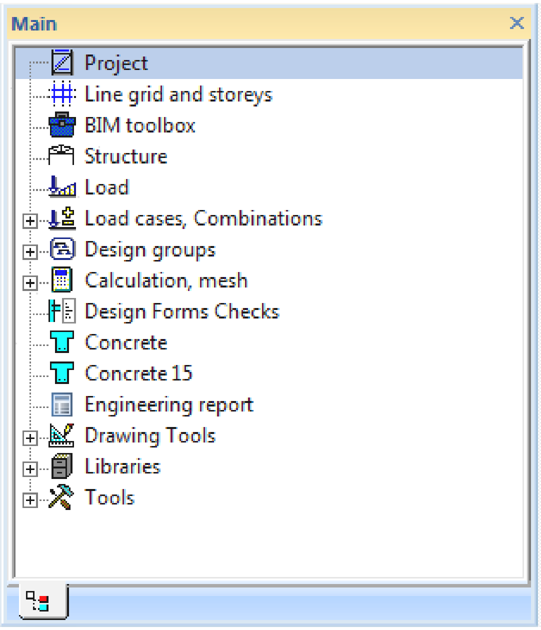


Attention when Select is put to Select, and Validity to e.g. +Z or -Z !

Annex 5: Overview of the icons in windows & toolbars

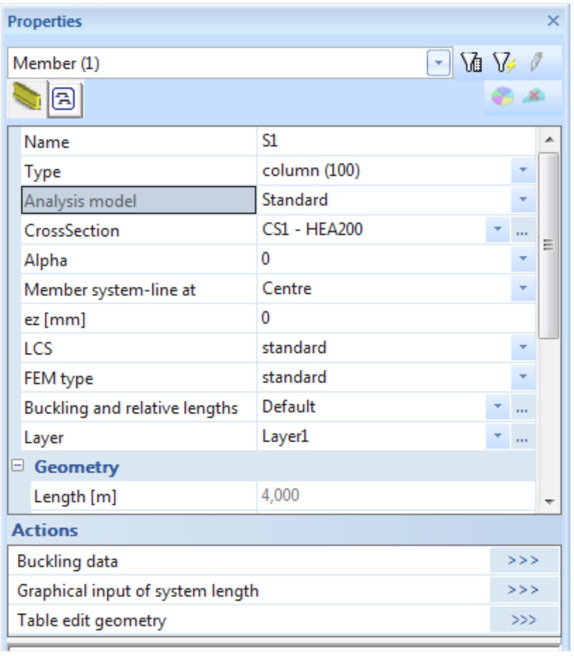
1_Main window

In the Main window one can find the links to the most used menus and actions. Some of those links are only activated when they can be effectively used in the project: e.g. the link to the Results menu is only shown after a calculation has been performed, and the Steel and/or Concrete menus are visible depending on the materials applied in the project.



2_Properties window

The Properties window gives information about selected objects and/or actions. Moreover it is possible to adjust the properties of each object directly via this menu. When several kinds of objects or actions are selected at the same time, it is possible to switch between their properties by means of the little arrow behind the object name. The number between brackets behind the object name represents the number of objects of this kind that is selected at this moment. If the number is larger than 1, only the corresponding properties are shown.



Select elements by more properties

Select elements by property

3_Menu bar

These 'written' menus group all actions by subject. A large number of these actions is also available in the Main menu and/or as icon in the toolbars.

4_Standard toolbars

The 'Activity' toolbar provides options for the visibility / invisibility of specific parts of the structure, which increase the ease of working and the readability of the project.

Activity toggle

Activity by layers

Activity by selection (Selected members On)

Activity by selection (Selected members Off)

Activity by working plane

Activity by clipping box

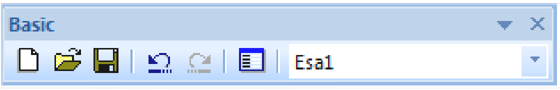
Activity by storey

Move activity by storey up

Move activity by storey down

Invert current activity

Draw inactive members



The 'Basic' toolbar contains a number of primary actions with regard to the current project and allows to modify the basic settings of the program (Setup Options).

New (Ctrl+N)

Open (Ctrl+O)

Save (Ctrl+S)

Undo

Redo

Setup

Esa1 Name of the opened *.esa file

The 'View' toolbar allows for the execution of a whole lot of simple view manipulations.

View in direction X

View in direction Y

View in direction Z

View in direction AXO

View perpendicular to working plane

Zoom in

Zoom out

Zoom by cut out

Zoom all

Zoom selection

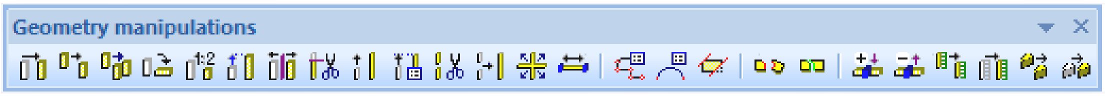
Perspective view

Undo view change

Redo view change

Generate structural model

Regenerate view



In the ‘Geometry manipulations’ toolbar one can find manipulations with basic entities (nodes, 1D members, 2D members), as well as with additional data.

Move

Copy

Multicopy

Rotate

Scale

Stretch

Mirror

Trim

Extend

Enlarge by defined length

Break in defined points

Join

Break in intersections

Reverse orientation

Polyline edit

Curves edit

Calculate member end-cuts

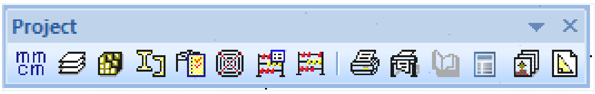
Divide surface by curve

Merges more surfaces into one

Connect members/nodes
Disconnect linked nodes
Copy additional data
Move additional data
Copy attributes
Move attributes

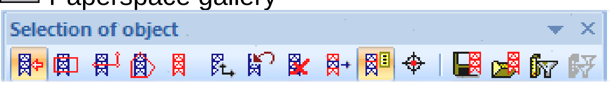
The ‘Modelling tools’ toolbar provides for manipulations with general solids.

Union of solids
Subtraction of solids
Intersection of solids
Division of solids
Generate vertexes
Clash check of solids
Move vertexes/points



The ‘Project’ toolbar collects various actions, from the definition of databases (layers, materials, cross-sections) for the project, to several options for the output.

Units
Layers
Materials
Cross-sections
Check structure data
Mesh generation
Calculation
Hidden calculation
Print data
Print picture
Document
Open Engineering report manager
Picture gallery
Paperspace gallery



The ‘Selection of object’ toolbar contains different possibilities to select a specific part of the structure. A selection can also be saved and loaded again later on.

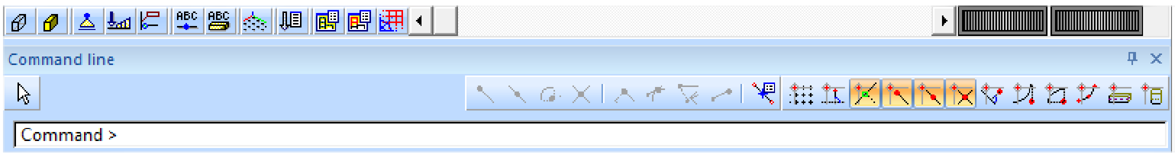
Selection by mouse
Selection by cut out
Selection by intersecting line
Selection by polygonal cut out
Select all
Selection by working plane
Previous selection
Cancel selection
Selection mode toggle (Select or Deselect)
Single selection mode toggle (All found or First found)
Visibility selection mode
Save selection
Load selection
Filter for selection on/off
Filter by service tree on/off

In the ‘Tools’ toolbar a number of clever means can be found for the input and graphical display of a structure.

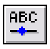
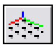
Setup UCS
Clipping box
Dot grid setting
Coordinates info

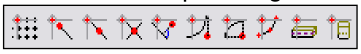
5_Command line toolbars

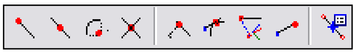
On the Command line itself, a number of commands for the operation of the program can be inputted. Also, during running actions, instructions on the next steps are shown.
Apart from that, quite a number of toolbars can be found here; some of them are only available during a certain action or in a specific menu.

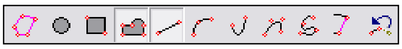


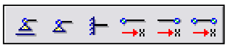
Show/Hide surfaces
Render geometry
Show/Hide supports

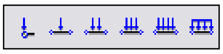
 Show/Hide loads
 Show/Hide other model data
 Show/Hide labels of nodes
 Show/Hide labels of members
 Show/Hide dot grid
 Set load case for display
 Fast adjustment of view parameters on whole model
 Fast adjustment of view parameters on selection

 Cursor snap settings
 Fast adjustment of cursor snap settings

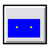
 Change insertion point, available during input of geometry

 Definition of new form, available during input of geometry

 Fast input of supports & hinges, available in Structure menu

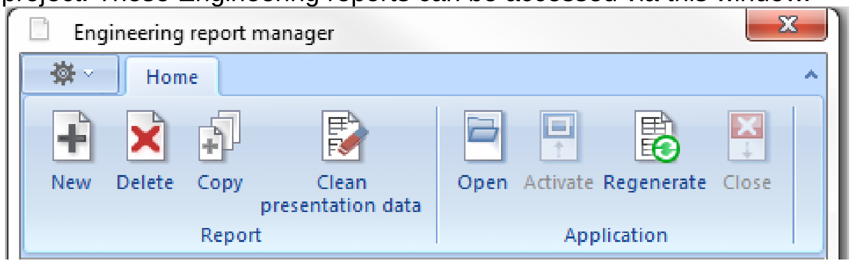
 Fast input of loads, available in Load menu

Fast display of results, available in Results menu

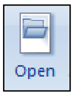
m Adjust Units of length
Plane XY Change Active working plane
Snap mode Adjust Cursor snap settings
Current UCS Adjust UCS (=User Co-ordinates System)
 Change Active code

6_Engineering report manager window

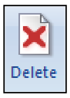
In the engineering report manager you can find an overview of all your engineering reports of the project. These Engineering reports can be accessed via this window.



Add new report

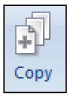


Open selected report in Engineering report application



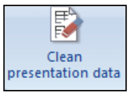
Delete selected report

Activate opened Engineering report editor

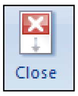


Copy selected report

Regenerate Engineering report editor

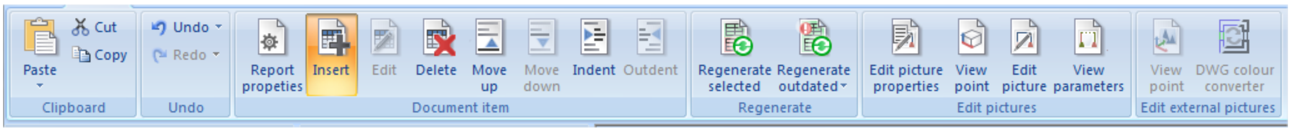


Clean presentation data for selected report

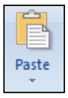


Close opened Engineering report application

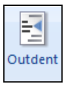
7_Engineering report home toolbar



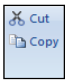
The Home toolbar contains manipulation tools to edit your Engineering report.



Paste items from clipboard



Report item outdent

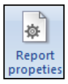


Cut/Copy items to clipboard

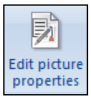
Regenerate selected report items

Undo/Redo an action

Regenerate all report items with outdated validity status

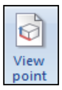


Show Engineering report properties

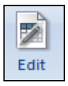


Edit picture properties

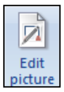
Insert new Engineering report item



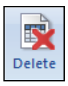
Edit picture view point



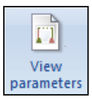
Edit selected report item



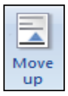
Edit picture in graphic editor



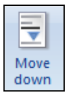
Delete report item



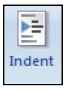
Show view parameters in editor



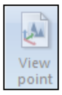
Move report item up



Move report item down



Report item indent

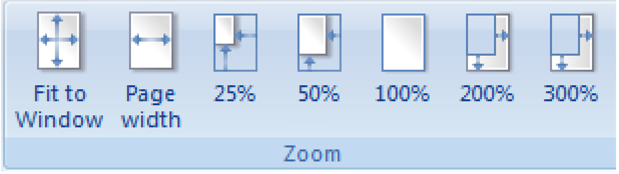


Edit external picture viewpoint

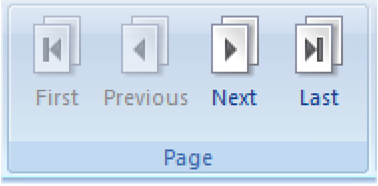
DWG colour converter

8_Engineering report View toolbar

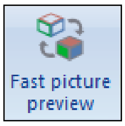
The view of the Engineering report can be zoomed by using the following buttons



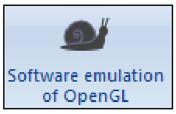
Moving through pages can be done by using the page buttons.



The following buttons are applied to influence the Engineering report.

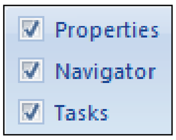


Fast preview of rendered pictures

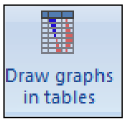


Rendering of pictures using software emulation of OpenGL

Fast table preview



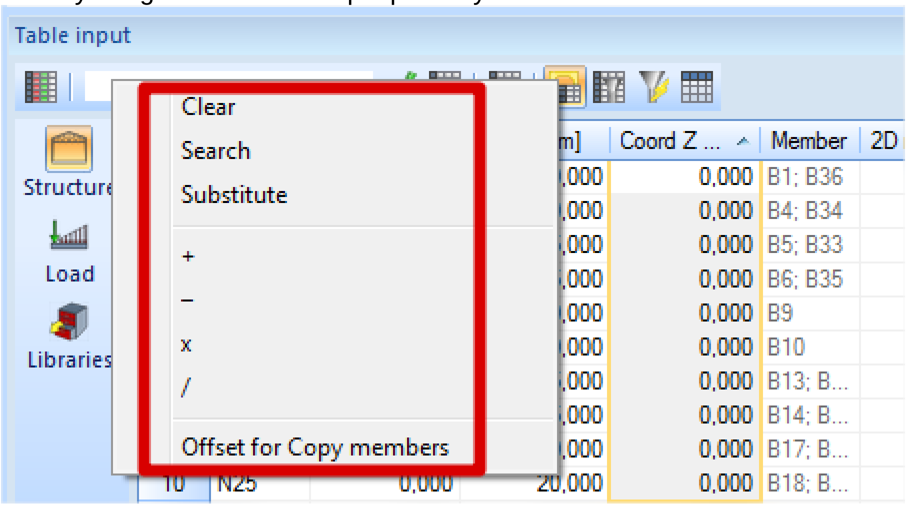
Show/hide properties, navigator, tasks.



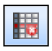
Draw graphs in tables

9_Table input toolbar

When you right click on the top inputbar you can see the different functionalities.

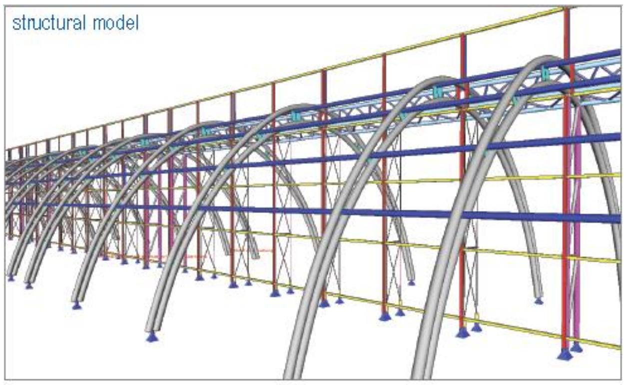


- Clear: Clears the top inputbar.
- Search: Performs a search for a certain element.
- +, -, x, /: These can be used to perform mathematical actions on a selection.
- Offset for Copy members: Performs a multicopy of elements.

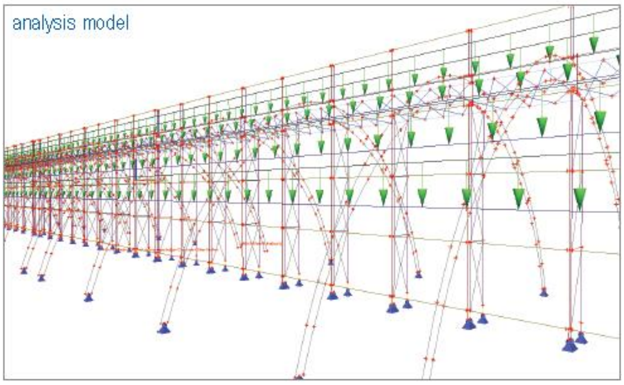
	Column selector		Activity in table
	Apply edit		Filter bar in table
	Copy row		Select by property in cell
	Delete row		Table to engineering report

Annex 6: Introduction to openBIM

Open BIM is a universal approach to collaborative design, realization and operation of buildings based on open standards and workflows. This means that Open BIM and even BIM in general are all about processes and not about software. Still, we need software to enable BIM. 3D modelling and adding information to these 3D models require dedicated software.
As far as the engineering market is concerned, the Nemetschek group offers several high performance solutions. One of them is **SCIA Engineer**.



Whereas for CAD software the additional data in a BIM model is often equally important as the geometrical data, the CAE model can do with far less. A general CAE model is built up out of centre lines (1D), mid planes (2D), cross sections, code compliant materials, supports... No need for textures, catalogue-ID's, unit prices...
SCIA Engineer, however, is unique in its kind as if offers you two (parallel) models in the same project. On the one hand, you have the analy-sis model dealing with all the information which is related to the analysis. On the other hand, you have the structural model at your disposal, which is dealing with all geometrical relations in the model.



CAD models primarily focus on the geometry. When importing them into a CAE software, there is little interest in irrelevant additional data. And when it comes down to importing or exporting the geometry, the **IFC format** is the best way to go.
All imported IFC models can be converted into the analysis models and fine-tuned for structural analysis purposes. We call this process **Structure2Analysis**. SCIA Engineer is packed with features which guide you through the whole conversion and make it a piece of cake. The structural model, however, remains as it is, allowing you to export it with or without changes.
Separate service called BIM Toolbox consists of all necessary functions for model conversion, align and clash check.

As IFC does not yet support analysis models, we have a number of proprietary links at your disposal.

- ETABS
- Steel Detailing Neutral File (SDNF)
- Prosteel
- STEPSTEEL
- DSTV
- general XML

Next to this we also link directly to some CAD-oriented software programs which have the analysis model.

- Allplan Engineering
- Tekla Structures
- Revit Structure